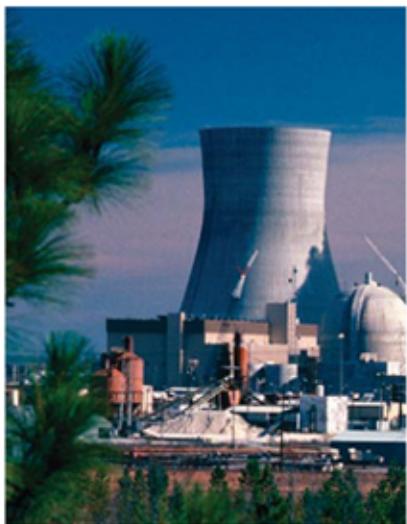


SmartPlant Electrical

Advanced User's Training Guide

Process, Power & Marine



 **INTERGRAPH**

Copyright © 1999-2010 Intergraph Corporation. All Rights Reserved.

Including software, file formats, and audiovisual displays; may be used pursuant to applicable software license agreement; contains confidential and proprietary information of Intergraph and/or third parties which is protected by copyright law, trade secret law, and international treaty, and may not be provided or otherwise made available without proper authorization.

Restricted Rights Legend

Use, duplication, or disclosure by the Government is subject to restrictions as set forth in subparagraph (c) of the *Contractor Rights in Technical Data* clause at DFARS 252.227-7013, subparagraph (b) of the *Rights in Computer Software or Computer Software Documentation* clause at DFARS 252.227-7014, subparagraphs (b)(1) and (2) of the *License* clause at DFARS 252.227-7015, or subparagraphs (c) (1) and (2) of *Commercial Computer Software--- Restricted Rights* at 48 CFR 52.227-19, as applicable.

Unpublished---rights reserved under the copyright laws of the United States.

Intergraph Corporation
Huntsville, Alabama 35894-0001

Warranties and Liabilities

All warranties given by Intergraph Corporation about equipment or software are set forth in your purchase contract, and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of such warranties. Intergraph believes the information in this publication is accurate as of its publication date.

The information and the software discussed in this document are subject to change without notice and are subject to applicable technical product descriptions. Intergraph Corporation is not responsible for any error that may appear in this document.

The software discussed in this document is furnished under a license and may be used or copied only in accordance with the terms of this license.

No responsibility is assumed by Intergraph for the use or reliability of software on equipment that is not supplied by Intergraph or its affiliated companies. THE USER OF THE SOFTWARE IS EXPECTED TO MAKE THE FINAL EVALUATION AS TO THE USEFULNESS OF THE SOFTWARE IN HIS OWN ENVIRONMENT.

Trademarks

Intergraph, the Intergraph logo, SmartSketch, SmartPlant, and INtools are registered trademarks of Intergraph Corporation. Microsoft and Windows are registered trademarks of Microsoft Corporation. MicroStation is a registered trademark of Bentley Systems, Inc. Other brands and product names are trademarks of their respective owners.

Training Perquisites

- An intermediate knowledge of electrical design.
- Familiarity with windows-based applications.
- A plant with default reference data.
- The following “Addition files for advanced training labs”:

For Exercise 1 – Creating a Schematic for Motor

[Typical1_Motor.sym](#)

[Typical2_Motor_with_Instrument.sym](#)

[Typical1_Instrument.sym](#)

[Typical1_ControlStation.sym](#)

[TransformerComplete.sym](#)

For Exercise 3 – Creating a Multi Tag Motor Schematic

[CD1sh1Motor3ph1.sym](#)

[HeaderOnDrawing_1.sym](#)

[HeaderTabularPage_1.sym](#)

[MultiTagTable_1.sym](#)

For import labs 1-8

[Spel_import.mdb](#)

Table of Contents

Preface.....	8
Working with the Data Model	9
Relationship Types in the Data Model	9
Working with the Data Dictionary Manager	12
Working with SmartPlant Engineering Manager	25
Plant Group Type and Hierarchy Templates	25
Creation of Site Plant and associate Application.....	26
Creation of Data Dictionary Template	27
Backup and Restore	30
Copy a Plant Structure	31
Running Engineering Manager in Debug Mode.....	33
SmartPlant Electrical Report Customization	34
Exercise 1 – Cable Schedule.....	34
Exercise 2 – United Properties	40
Exercise 3 – Sorting Data	41
Exercise 4 – Filtering Data	42
Exercise 5 – Displaying Plant Group Names in a Report.....	46
Exercise 6 – Grouping Data by Common Properties.....	47
Exercise 7 - Adding Sub-Class Tables for Data Collection	48
Exercise 8 – Motor Space Heater	51
Exercise 9 – Motor and Circuit Signals.....	52
Exercise 10 – Bill of Materials Report	55
Exercise 11 – All Feeder Load Summary (Enhanced) Report	58
Exercise 12 - Revisions and Document macros in Registered Report	62
Exercise 13 – Creating Report Cover Sheet	64
Exercise 14 – Creating Report Cover Sheet on the Same Worksheet	67
Macro Definition Guidelines.....	71
Macros Defined in Catalog Manager.....	71
Macros Defined in a CAD Application (SmartSketch, AutoCAD, or MicroStation) 71	71
Drawing Blocks and Electrical Items.....	73
Title Block Macro List (Templates)	74
Schematic Drawings.....	77
Exercise 1 – Creating a Schematic for a Motor	77
Exercise 2 – Creating a Schematic for a 3-Winding Transformer.....	82
Exercise 3 – Creating a Multi Tag Motor Schematic	84
Lab – Creating a New Template from Scratch	96

Lab – Schematics: Working with Blocks	101
Customizing Single Line Symbols	106
Combined macros in Smart Text	111
Customize Off Page Connector (OPC) symbols.....	115
Customize the reference label symbol in SLD.....	117
Show the PDB presentation as a “block” symbol	117
Ability to hide Feeder circuits from being shown in SLD’s	120
Moving circuits along the buses contour	121
Symbology.....	124
Assemblies in SLD.....	130
Showing dot connection on a bus in SLD	136
Show symbol macro short value in SLD	138
Comparing Documents.....	141
Cable Block diagrams (CBD).....	145
Create Equipment in CBD from typical on the fly	145
Cable Block Diagram for cables already pre-assigned	151
Create and connect control station and cabinet.....	153
Special Notes	154
Filtering the CBD on cable category.....	156
Reconnect cable in a different place along the line	157
Move macros.....	158
Scale a symbol	158
Allow you to scale symbol in CBD drawing, and keep the new scale	158
Print and Save option.....	160
Annotations and Redlining	160
Revision	160
Creating symbols for cable Block Diagrams	161
Move/Mirror/Rotate macro	166
Addition CBD items	168
Insert segment	168
Insert Move Vertex (Alt)	169
Identifying and Resolving Symbol Inconsistencies in an Existing CBD.....	169
Distinguishing Smart Data from Auxiliary Graphics - Changing the Style/Layer	171
Associate a Main item with a Project CBD	173
Wiring Diagram	175
Creating terminals and terminal strips in equipment	175
Connection type	180
Wiring project items	181
Batch cables sides’ termination	187
Wiring Shipped Reports	190

Table of Contents

Creating Wiring Symbols	192
SmartPlant Electrical Rule Manager.....	202
Creating Single component and relation rules.....	202
Rules that Handle Cable Insulation Voltages	206
Rules in Reference Data Explorer	208
Copy data with rule manager.....	210
Propagating data upstream	211
Propagating data downstream	214
Bus and its downstream disconnect electrical equipment	214
Propagating load data into circuit internals.....	215
Motor rated power to circuit breaker rated power	215
Copying data from loads to their upstream converting equipment	216
Fix Inconsistencies.....	216
SmartPlant Electrical Import Manager.....	217
Preparations: Database Connectivity	217
Import 1 – Motors List.....	219
Import 2 – Cable List.....	224
Import 3 – Cables \leftrightarrow Motors Link	230
Import 4 – Associate a Project Cable with Ref Cable	233
Import 5 – Power Distribution, Bus, Cells & Circuits.....	235
Import the Circuits	238
Associate the Feeder Circuits with the Bus	239
Import Cells	241
Associate the Cells with the Bus.....	242
Associate the Circuits with the Cells	245
Associate the Bus-Riser with the Coupler	246
Import 6 – Transformer	249
Import 7 – Manufacturer Select List.....	250
Import 8 – Heater Lookup Table	255
Import 9 – Import Motors List Using an Excel	258
Worksheet as Data Source	258
Import 10 – Import Motors List Using Query as Data Source	265
Import 11 - Import terminal Strip	270
SmartPlant Electrical Format Manager	283
Defining Project Naming Conventions.....	286
Define naming conventions using all plant group levels in item tag definition	286
Multiple Naming Conventions for Motors	287
Multiple Naming Conventions for Cables.....	291
Define Naming Conventions for Control Stations.....	297
Define Naming Conventions for Circuits and Cells	299
Define a Naming Convention for Signals	301
Define Naming Conventions for Instruments	303
Copy Naming Conventions.....	304

Copy Naming convention between item types	304
Copying naming conventions from unit 1 to unit 2	307
Copy Naming Conventions Across Plant Groups Belonging to The Same Site.....	308
Copy Naming Conventions Across Plant Groups Across Sites.....	309
Padding Zeroes in Item Tag Sequence Number.....	313
Changes log	317
Document Revision History	322

Preface

This training guide introduces concepts, procedures, and features of SmartPlant Electrical.

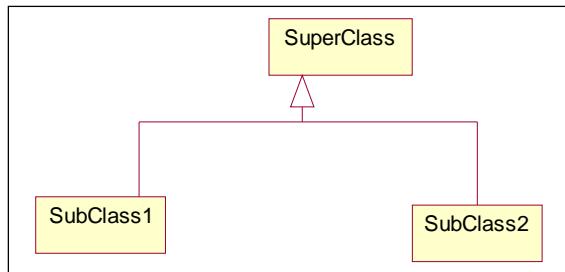
Send documentation comments or suggestions to PPMdoc@intergraph.com.

Working with the Data Model

Relationship Types in the Data Model

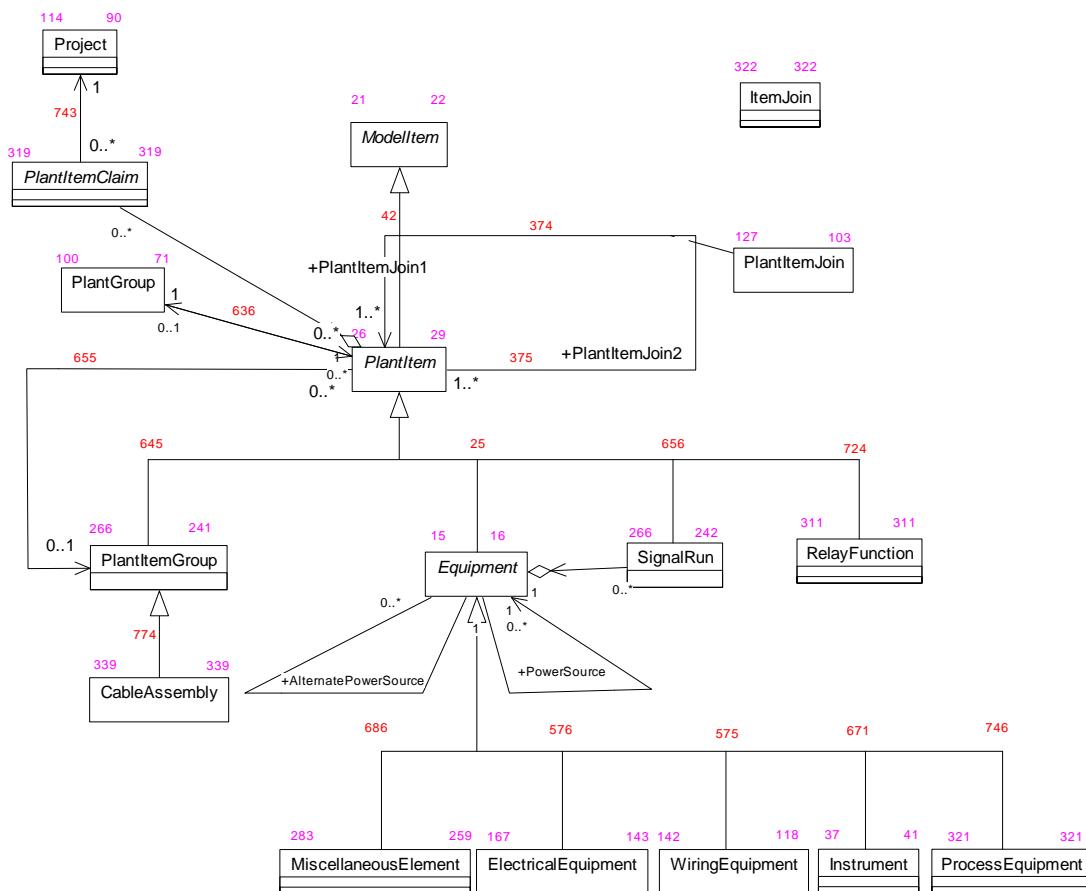
Generalization

Describes the hierarchy of SuperClass (parent) and SubClass (child)



SubClass1 and SubClass2 are children of the SuperClass

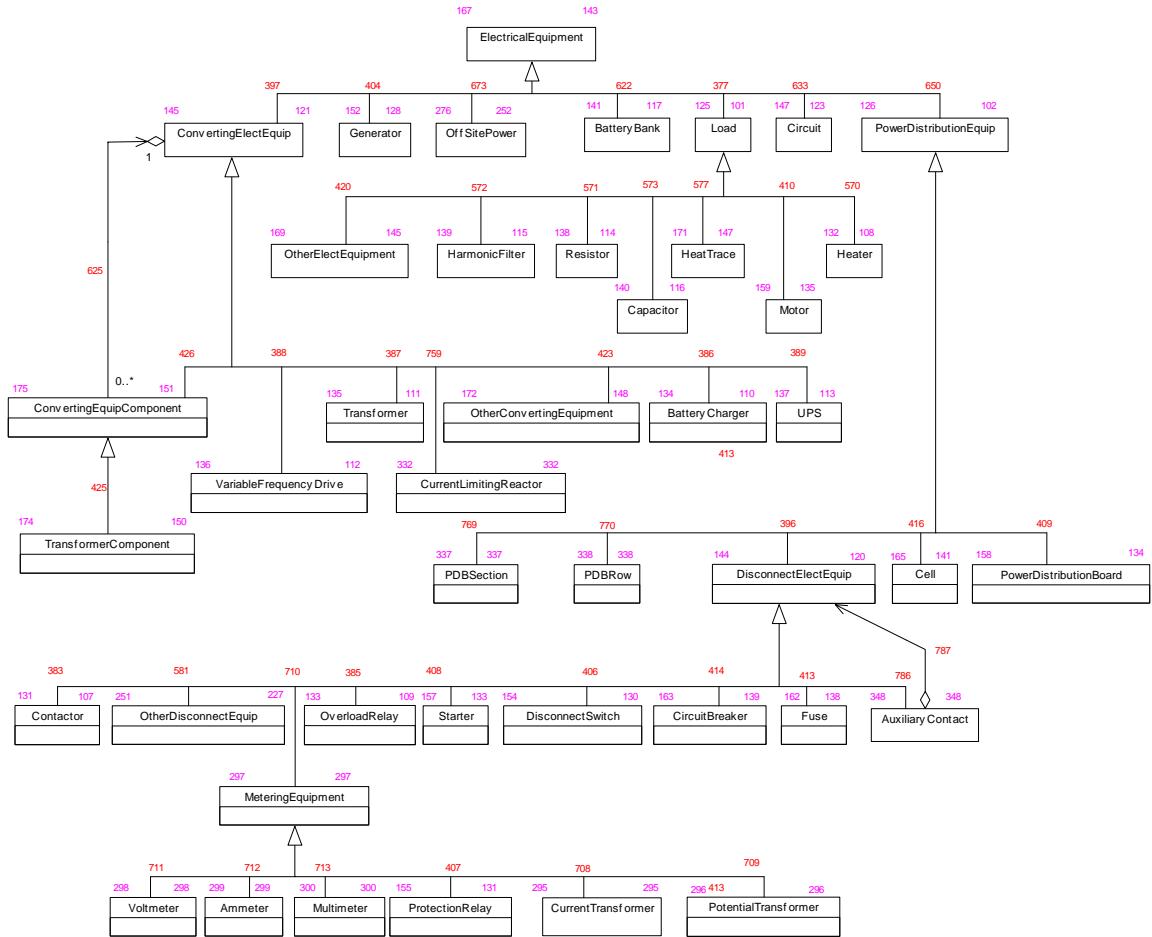
Hierarchy of Items in the Data Model



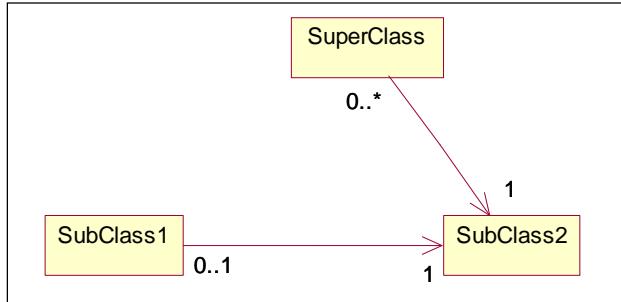
Every item is a ModelItem. PlantItem is a sub-class of ModelItem. Equipment is a sub-class of PlantItem. Equipment consists of sub-classes Miscellaneous Element, Electrical Equipment, Wiring Equipment and Instrument, and so on.

Note: The numbers appears next to each block is for internal use to produce the diagram.

Hierarchy of Electrical Equipment

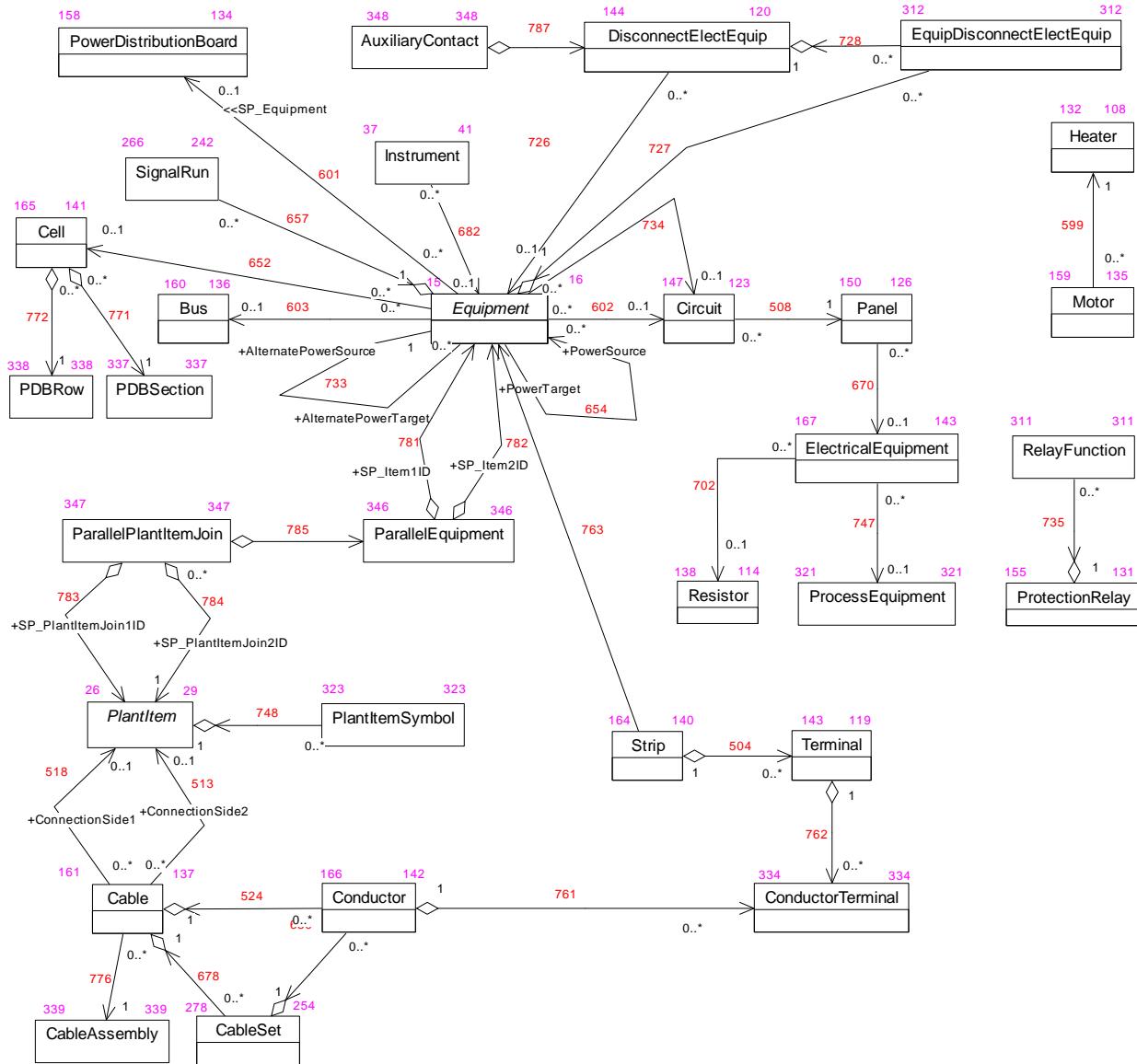


Associations - Relations Among Classes



For each SubClass2 there are maybe 0 or more items in the SuperClass. For Each SuperClass there must be one item in SubClass2.

Associations of Items in the Data Model



Equipment has a ‘many to one’ relationship with Circuit. This means that 0 or more Equipment items (fuse, contactor, circuit breaker, and so forth) can be related to 1 circuit.

Working with the Data Dictionary Manager

Data Dictionary Manager allows you to modify properties of the data model, including the database entries, select lists, and item types that form the underlying data structure.

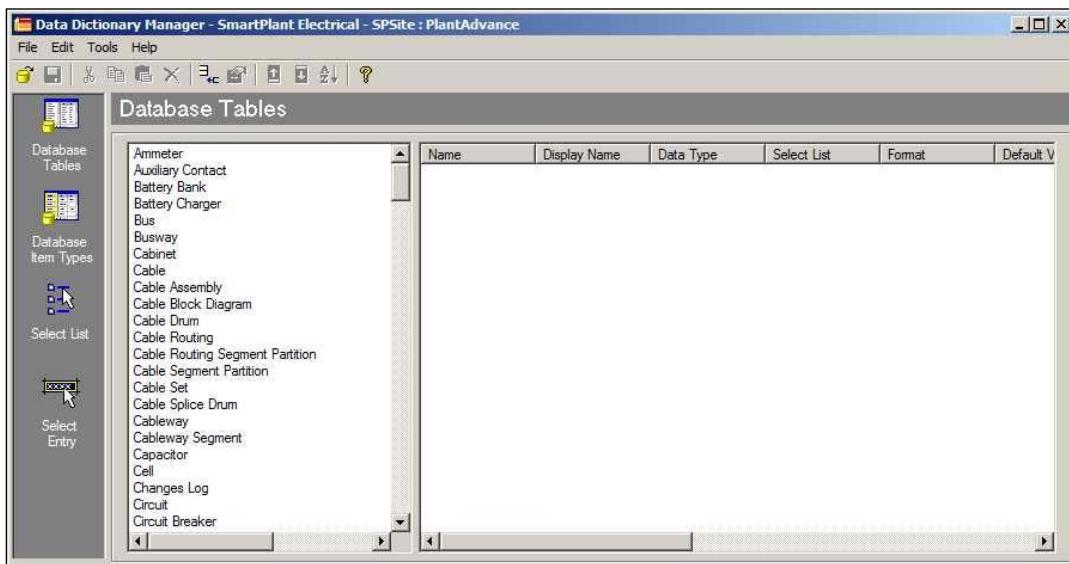
Data Dictionary Manager Tasks:

- Add and change properties for specific database tables
- Create select lists
- Add entries to a select list

Open Data Dictionary Manager

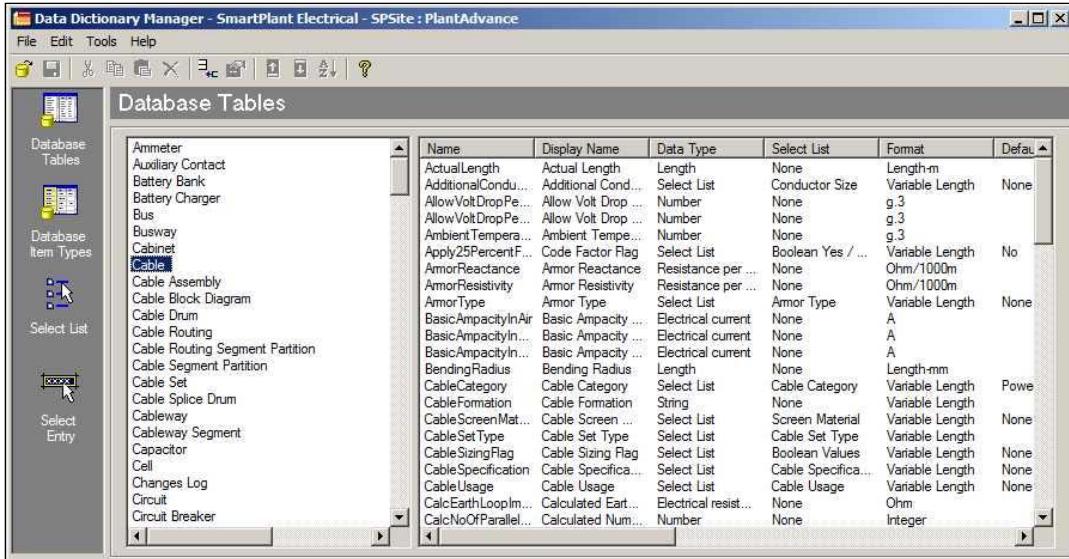
To open data dictionary manager do the following:

Click Windows Start button > All Programs > Intergraph SmartPlant Engineering Manager > Data Dictionary Manager.



Database Tables

For each table in the database, you can define properties or items that make up the table. These tables make up the data model, and their properties can be unique or common in several tables in the database.



Notes:

Each database table in the list refers to one of the following:

- A real engineering class, for example: Cable, Motor.
- A class that group engineering classes together, for example: WiringEquipment, ElectricalEquipment.
- A system class, for example: RefItemTypeStorage.

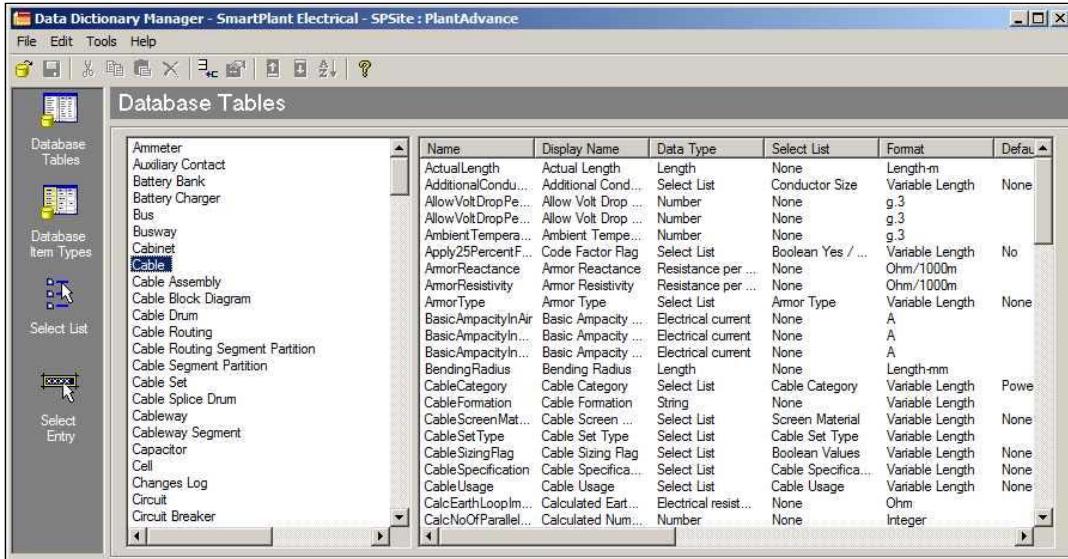
Properties can be used by multiple items, and there are two ways in which properties can be associated with multiple database tables:

- The ‘parent class’ shares its properties with the child class. In this case, every property of the parent class is visible to all of its child items. For example, a property added to **Load** is visible also to **Motor**, **Static Electrical Equipment** and **Miscellaneous Electrical Equipment**.
- Properties can be used by more than one class or database table, for example, by adding the same property to both the **Cable** and **Ref Cable** item types.

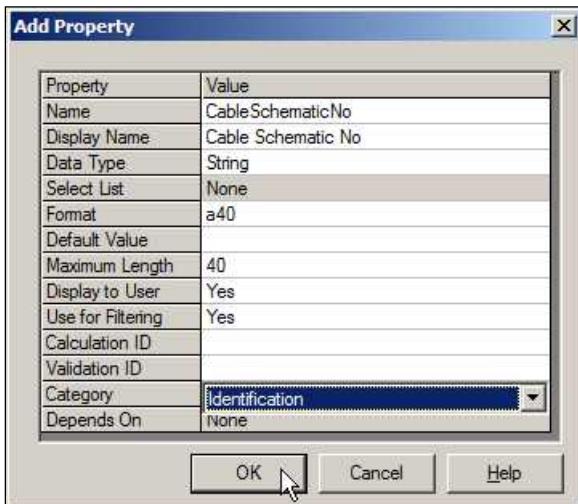
Add properties, and modify properties in Database Tables

Add Database Tables property

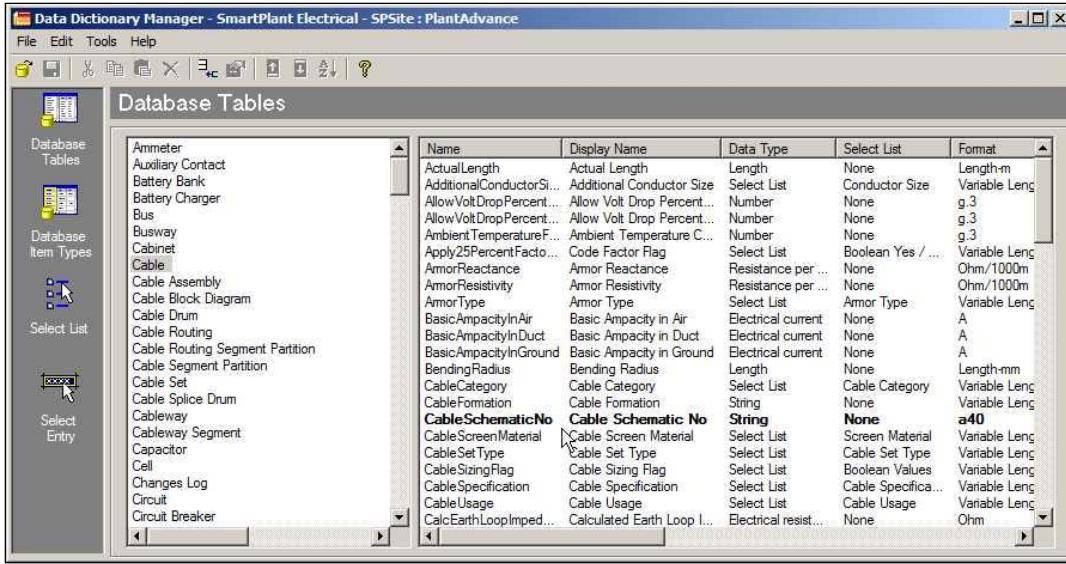
1. Open Data Dictionary, and select Database Tables.
2. From the table list select Cable.



- Right-click inside the right pane windows, and on the shortcut menu, click Add Property to open the Add property dialog box, where you can add the new property.

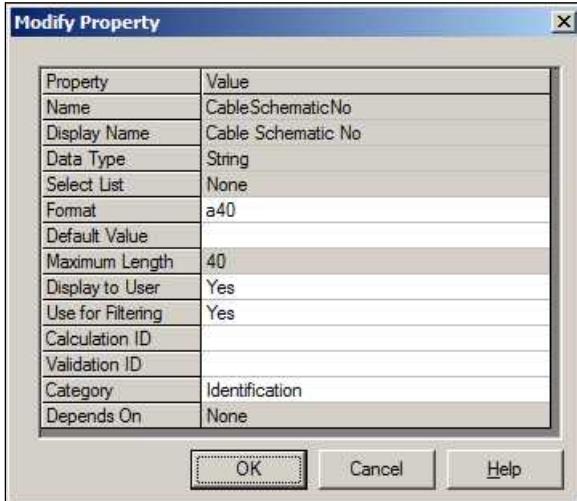


- Select OK to close the dialog box, and save the change by select the save icon.
After you save the change, the new property appear in bold fonts to distinguish from default properties (out of the box properties).



Modify Database Tables property

1. Select Database Tables.
2. From the tables list select Cable
3. Right-click on property you would like to modify, and on the shortcut menu click Properties to open the 'Modify Property' dialog box, where you can modify the property.



4. Close the dialog box and save the change.

Notes: Only value in white field can be modified.

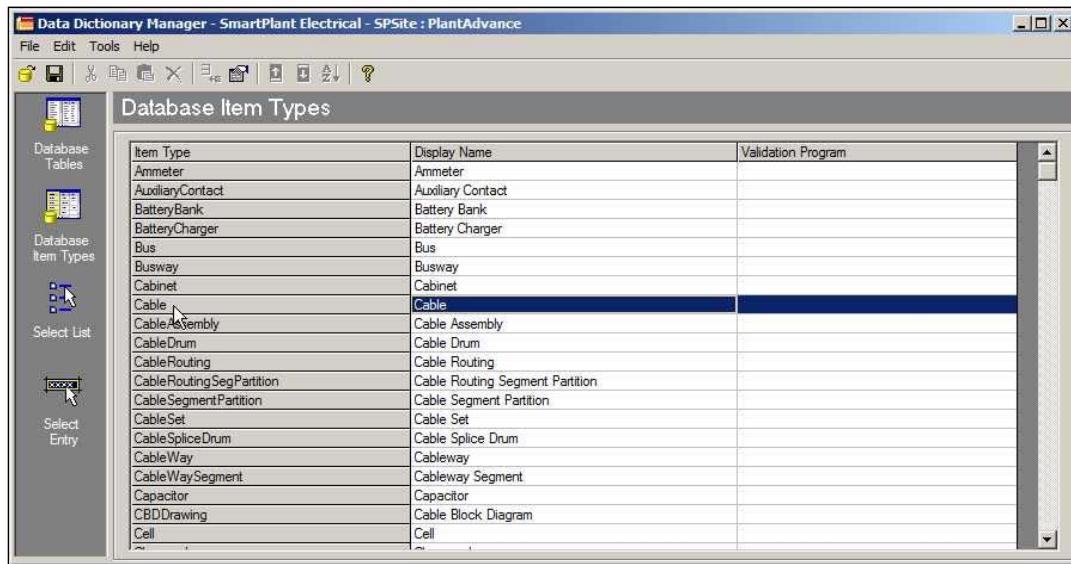
Property can be hidden by setting the **Display to User** flag to **No**, but it cannot be deleted from the database.

Database Item Types

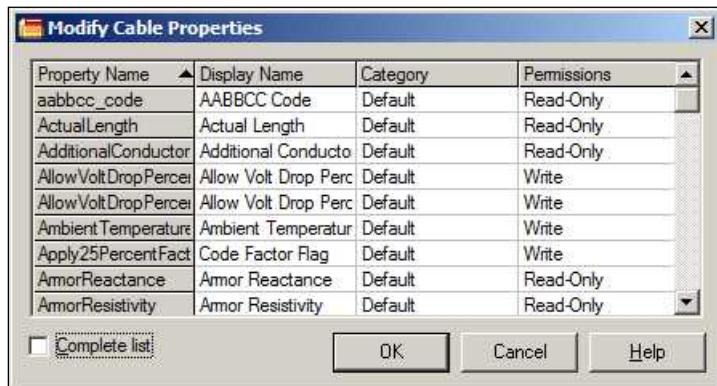
Using the **Database Item Types** option, you can edit display names, categories, and permissions.

To edit display name

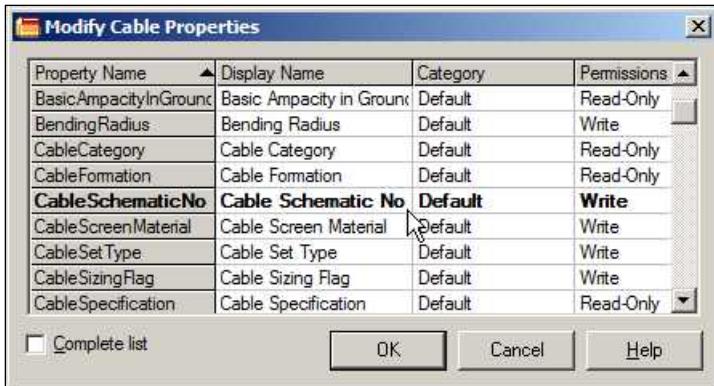
1. Select Database Item Types.
2. From the item type list select Cable.



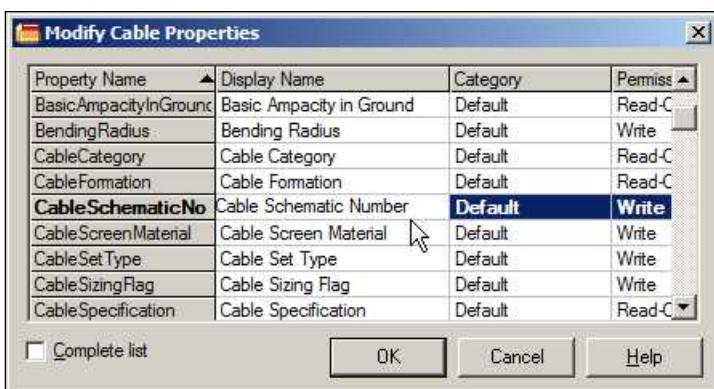
3. Right-click on Cable, and on the shortcut menu click Properties to open the 'Modify Cable Properties' dialog box, where you can modify the property.



4. Scroll down to the property you would like to edit its display name.



5. Edit the display name, and click OK to close the dialog box.



6. Save the change.

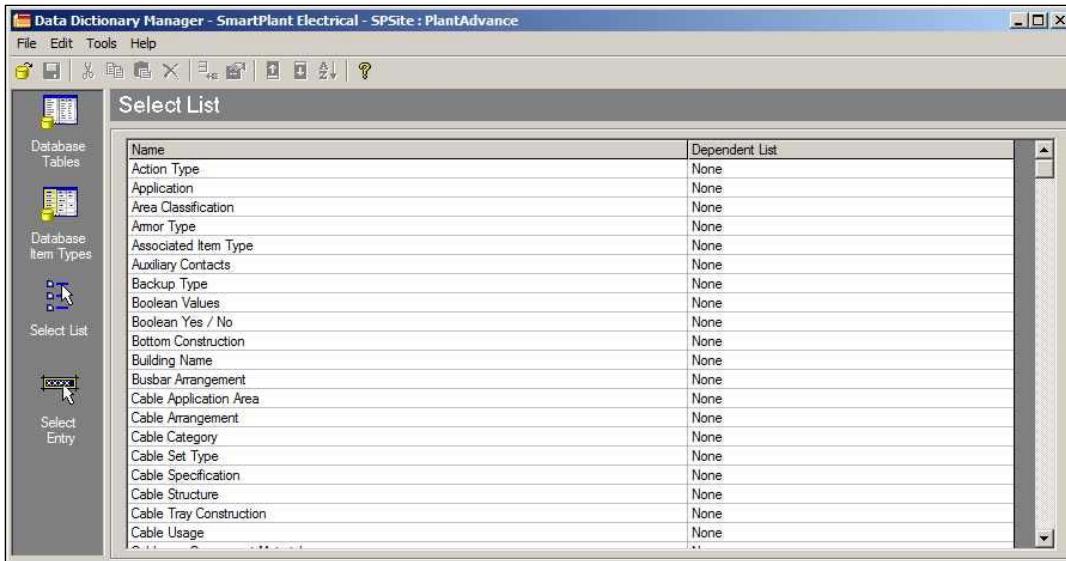
Select List

Select lists are simply groups of related values that narrow the specification of various aspects of the data model. Select lists allow users to select a value from a list for specific properties.

When you define properties in Data Dictionary Manager, you can specify whether you want users to select property values from a select list or type the value directly into the **Properties** window.

Some select lists have dependent lists, which provide further descriptions for items in the original select lists.

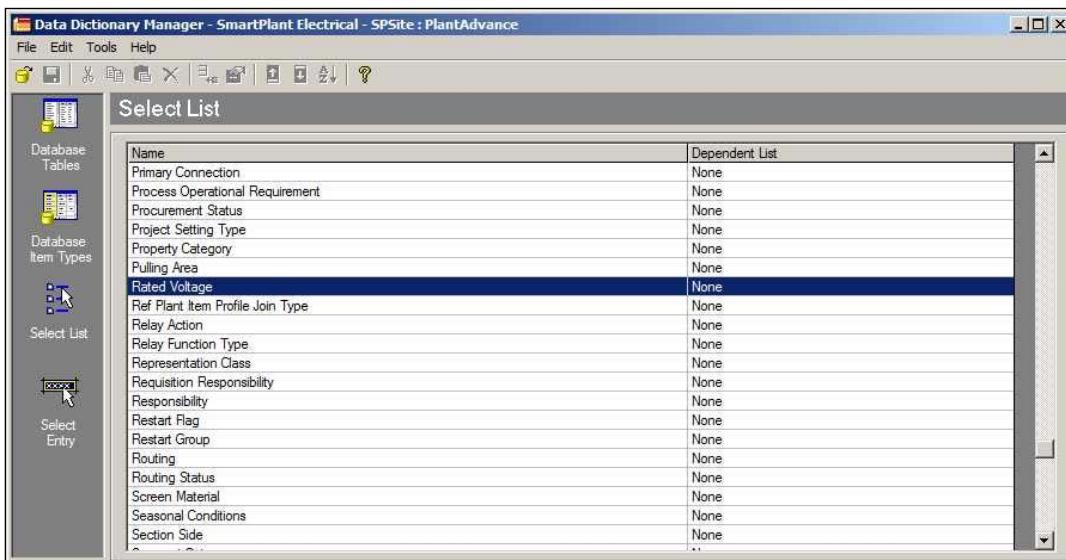
The range of values that you see in the **Properties** window is limited to values defined for a dependent list, if one is specified for the given property. For example, Electrical Equipment Type is dependent on Electrical Equipment Subclass, which is in turn dependent on Electrical Equipment Class. Because 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.

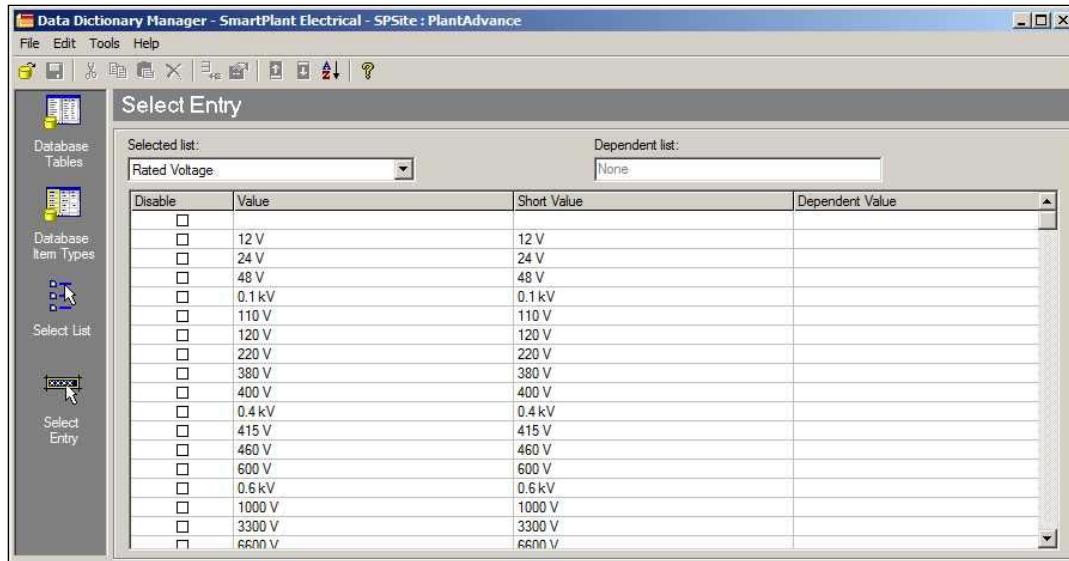


Select Entry

Use **Select Entry** on the **SmartPlant Electrical Data Dictionary Manager** dialog box to add values to select list.

You cannot delete existing items in a select list. To disable a value and prevent it from being viewable when this select list is used; click **X** in the **Disable** column to change the **X** to a check mark. A check mark in the **Disable** column indicates that the value is not displayed. Values defined here must be unique. The short value is optional. If a dependent list has been identified, the specific value from the dependent list is defined in the **Dependent Value** column. You can sort the values by clicking the column headings.



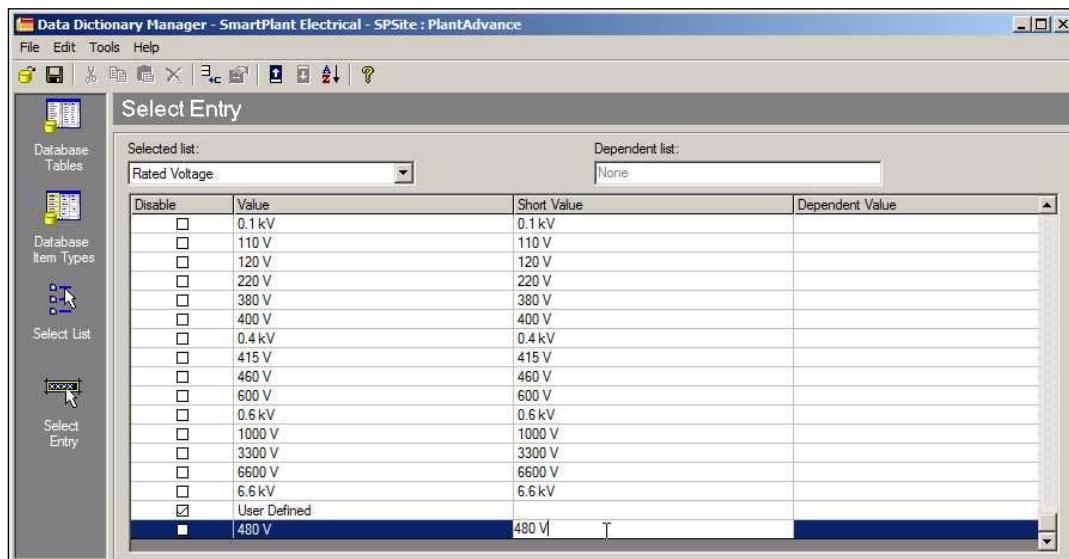


Notes: Select list entity cannot be deleted from the database.

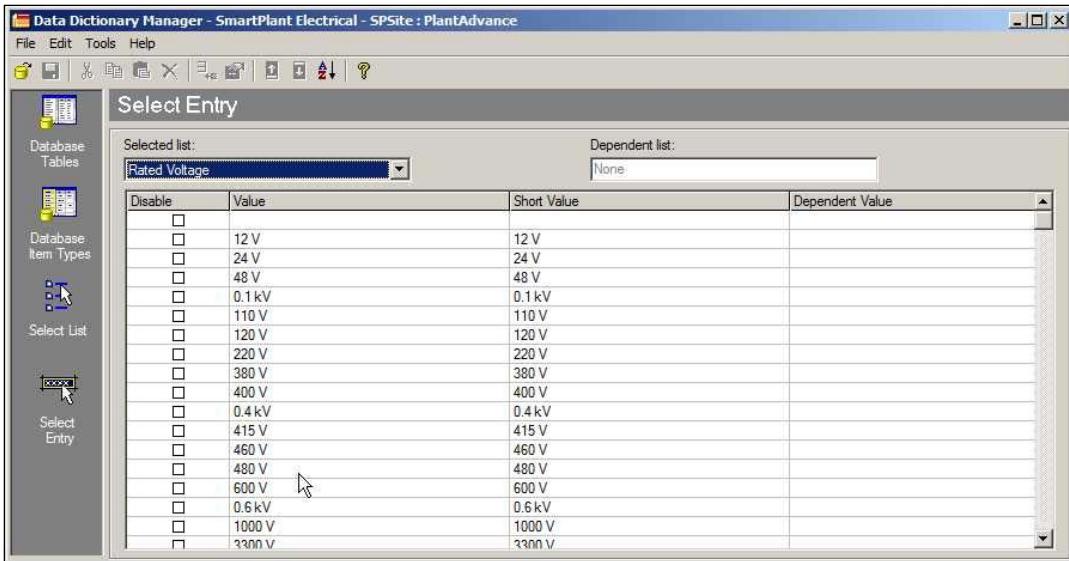
Select list entry can be hidden from user by check the disable check box, but it cannot be deleted from the database.

To add new entry to select list

1. Open the Rated Voltage select list.
2. Select the last row in the list (right after the default value, ‘user defined’).
3. Enter value and short value.

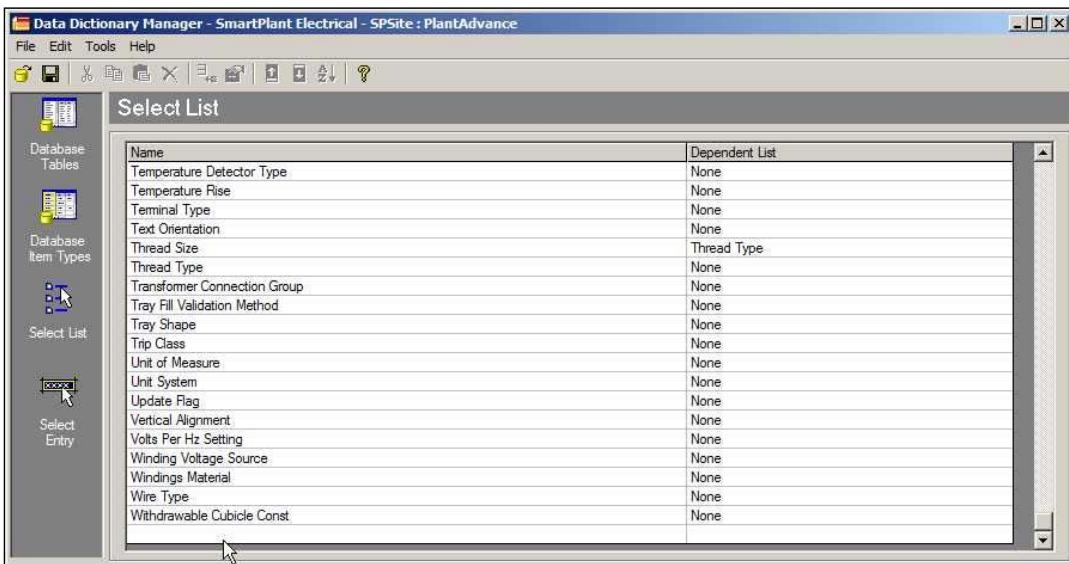


4. Save the change.

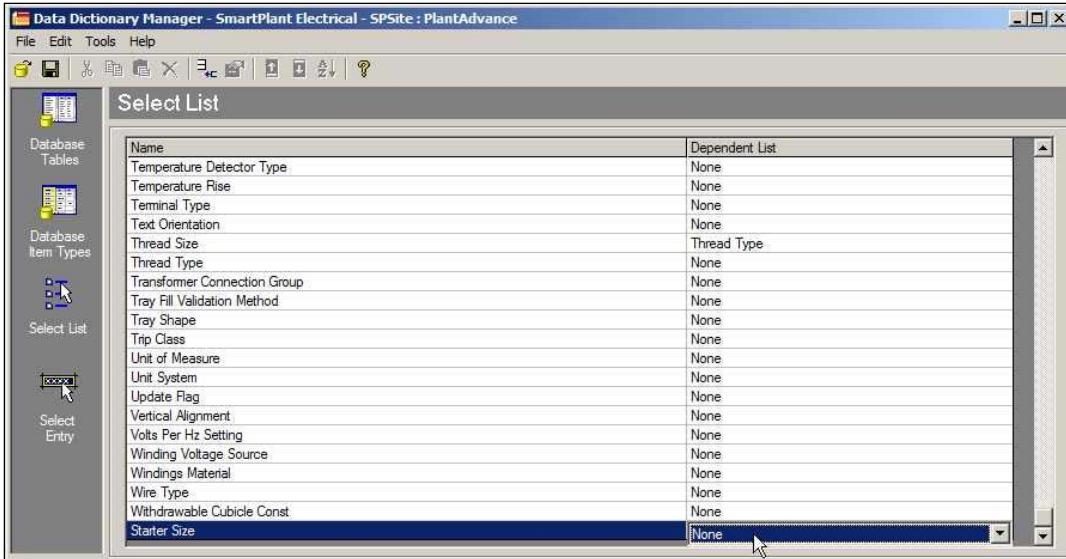


To add new select list

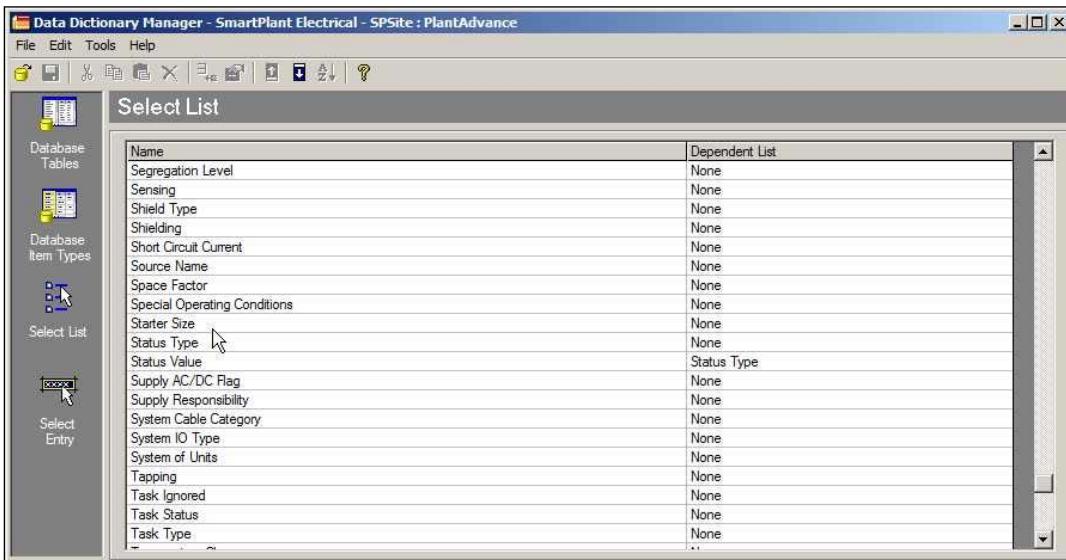
1. Open the select list window.
2. Select the last row in the list.



3. Type the name of the new select list, select dependent value (the default is 'None').

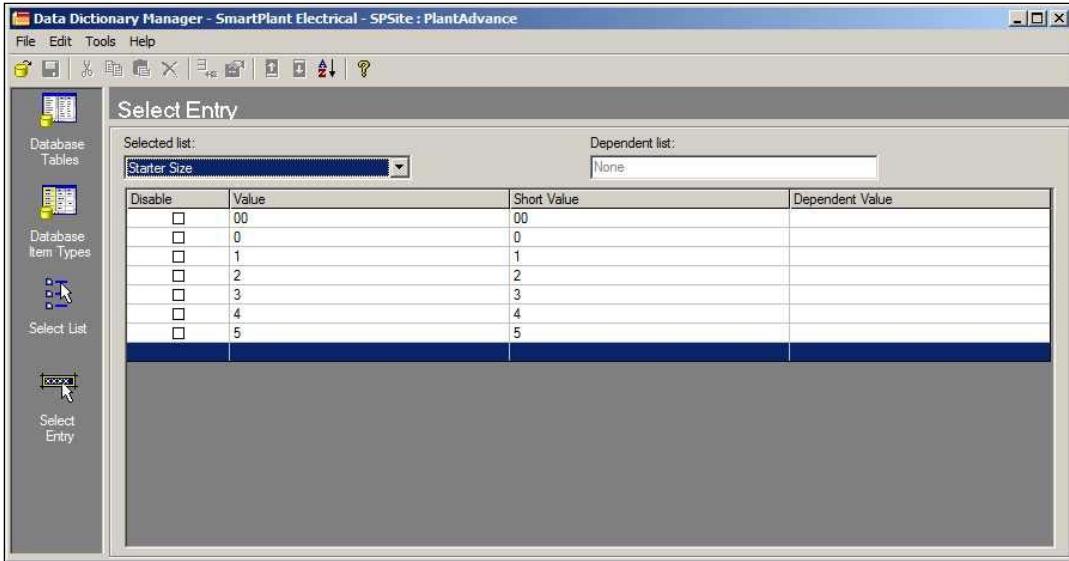


4. Save.



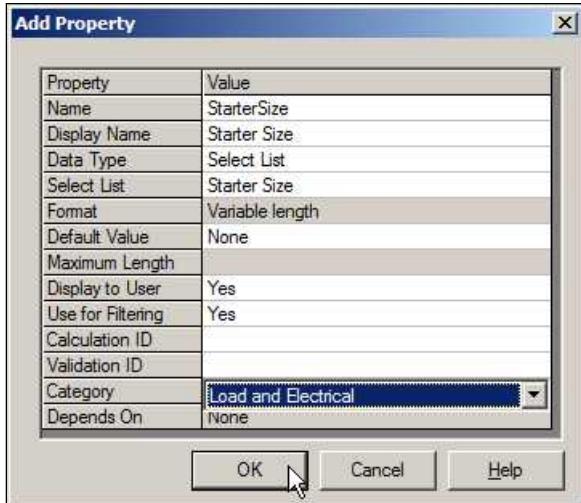
To add new entries to the new select list Item Creation

1. Select the select list 'Starter Size'.
2. From select entry window type new values and short value and save.



Data Dictionary Manager Lab

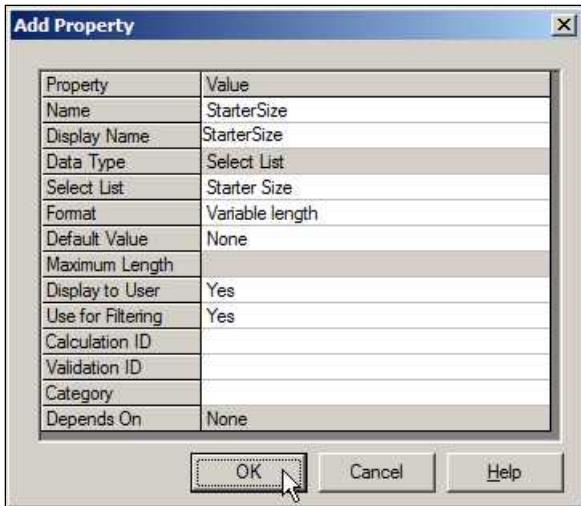
- 1.1. From Data Dictionary Manager add new property, ‘StarterSize’ to ‘Ref Starter’ table.
- 1.2 Enter the following values to StarterSize:



Save the change.

- 1.3. From Data Dictionary Manager add new property, StarterSize to ‘Starter’ table.
- 1.4. Enter the following values to StarterSize:

Notice: The property StarterSize already in the data model (created in step 1.1). When you select it from the list, it will populate the values automatically.



Save the change.

- 1.5. Open SmartPlant Electrical and create new typical starter in RDE (Reference Data Explorer) and name it, ‘ST Size1’.

Note: You create the new starter in the RDE under the folder:

Electrical Equipment->Power Distribution Equipment->Disconnect Electrical Equipment->Starters

- 1.6. Select the typical starter, ‘ST Size1’, and from property windows find the new property, ‘Starter Size’, and set its value to ‘1’.

- 1.7. Create new starter in EI (Electrical Index) and name it ‘ST-2’ , by copy the typical starter, ‘ST Size 1’ from RDE.

Note: You create the new starter in the EI under the folder:

Electrical Equipment->Power Distribution Equipment->Disconnect Electrical Equipment->Starters

- 1.8. Select the starter, ‘ST-2’, and from property windows verified its ‘Starter Size’ value set to ‘1’.

- 2.1. From Data Dictionary Manager add to Rated Voltage select list the value, ‘4.16 kV’.

- 2.2. Add to ‘Manufacturer’ select list the value, ‘**Global Electric**’.

- 2.3. Add to ‘Model’ select list the values, ‘AC100-001’, and ‘AC100-002’. Dependent value is the manufacturer, ‘Global Electric’. Save the changes.

Tip: Update select list can be done also from Smart Plant electrical, tools -> update select lists.

- 2.4. Reopen SmartPlant Electrical. Create in RDE, new typical off site power supply, and name it ‘Offsite Power Supply 4.16 kV’.

- 2.5. Select the offsite power supply, ‘Offsite Power Supply 4.16 kV’, and from property windows

set its rated voltage to ‘4.16 kV’, manufacturer and model to ‘Global Electric’ and ‘AC100-001’.

Working with SmartPlant Engineering Manager

SmartPlant Engineering Manager provides all the tools you need to effectively set up and manage your work with SmartPlant applications.

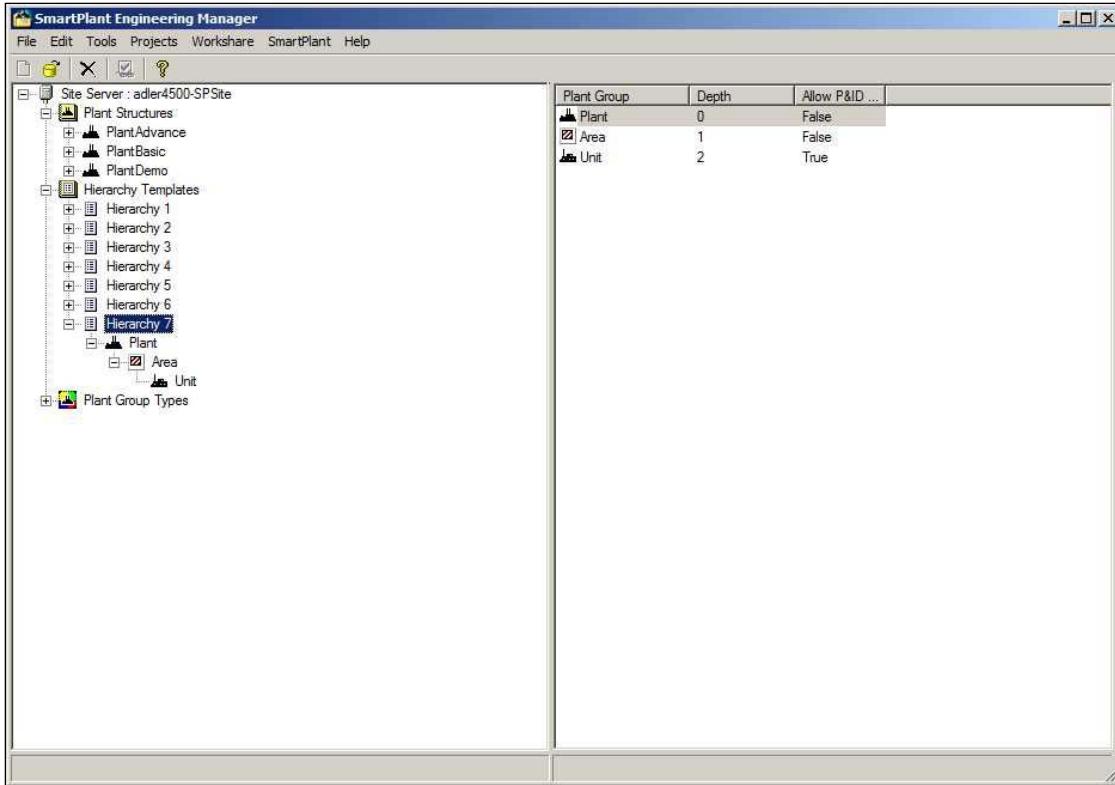
Supporting SmartPlant Electrical, SmartPlant P&ID, and SmartPlant Instrumentation, SmartPlant Engineering Manager manages the plant structures while the applications themselves are responsible for manipulating the actual data (including creating, deleting, modifying, launching, and so forth). SmartPlant Engineering Manager allows you to view not only the data related to the whole site but also data related to individual plants and projects. You can create and maintain SmartPlant Engineering sites, plant structures and projects, in addition to adding plant group types, modifying plant attributes, creating and modifying hierarchies, and associating SmartPlant applications.

Plant Group Type and Hierarchy Templates

Plant Group Types root contains the building blocks that make up plant structures. You use these plant group types to construct hierarchy templates that in turn used to create your plant structures.

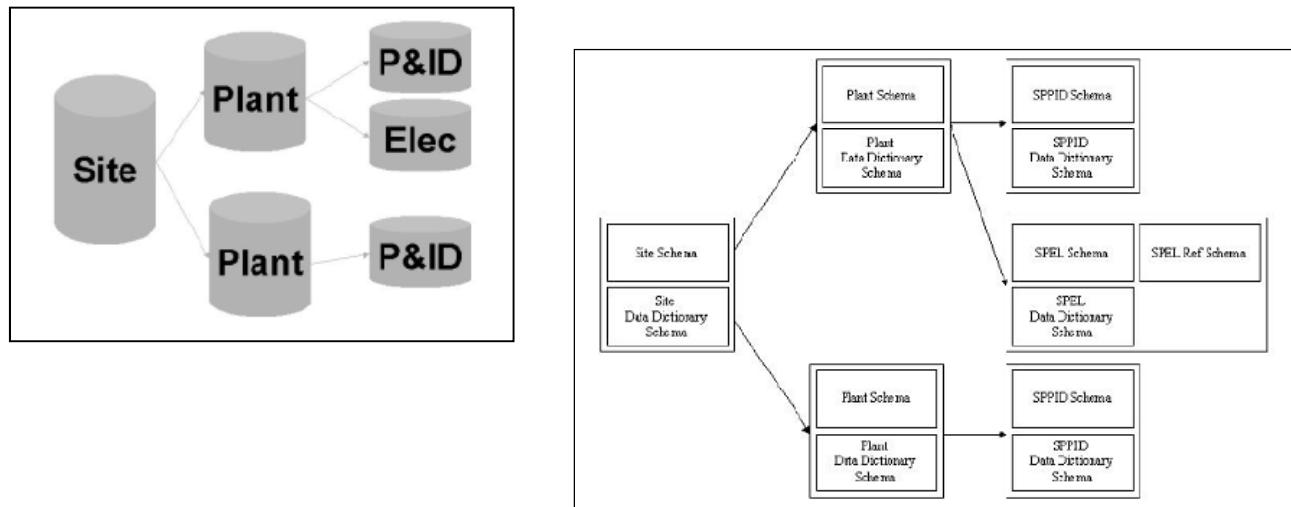
Hierarchy Templates root shows the various hierarchy templates that can be used to generate the structure of a plant. A hierarchy is made up of a set of two or more plant group types (formerly called hierarchy items) that are arranged in a tree structure.

Tip: When using SmartPlant Electrical as part of the SmartPlant Enterprise (SPE). It recommends checking with Intergraph what hierarchy to select before you creates the plant. As general, need to have the same hierarchy across the tools. The most common hierarchy in use is hierarchy template 7 (Plant/Area/Unit).



Creation of Site Plant and associate Application

SmartPlant Engineering Manager let you create the Site database schemas, the plant database schemas, and the associate application database schemas (SmartPlant Electrical application or/and SmartPlant P&ID application).



Creation of Data Dictionary Template

With SmartPlant Engineering Manager you can create data dictionary templates (DDT) that includes all the data in both the data dictionary (DD) and in the reference data explorer (RDE). You can use this feature to create new DDT at the end of your project work that will include all of your DD and RDE customization. And the next time you will start a new project (plant), you will be able to use the saved DDT (from your previous project) as a starting point and save you the time of doing the same customization again.

Roles and Access Rights

SmartPlant Engineering Manager contains two sets of rights: **site administrator rights**, and **plant structure rights**.

Site Administrator Rights

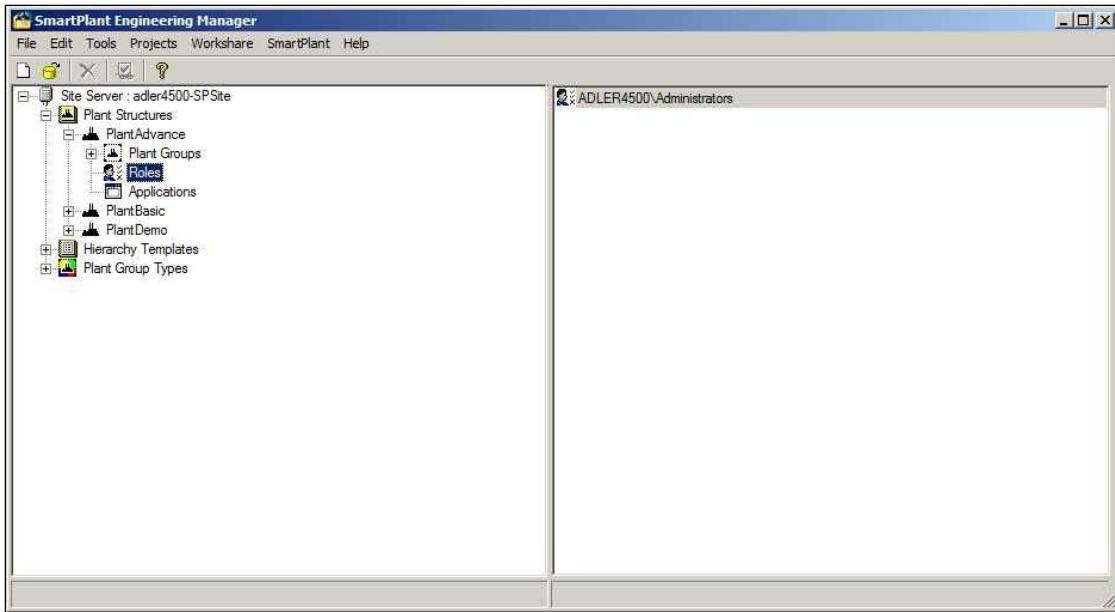
Site administrators, while not appearing in the **Roles** node in a plant structure, have a set of unique rights that cannot be granted to any other role. This group of users has privileges to create and modify plant structures, roles, hierarchies, and plant group types, to enable projects, and so forth.

Only site administrators see the **Hierarchy Templates** and **Plant Group Types** roots in the **Tree** view. Site administrators can see all plant structures and have full access to them.

 **Note:** Site administrator privileges do not extend by default to full privileges in the engineering applications. Additional site administrators can be added to the **Site Administrator** user group.

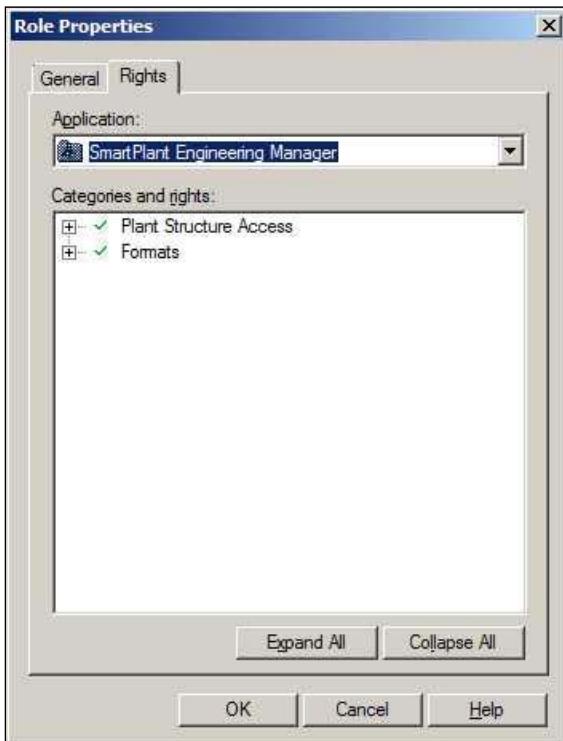
Plant Structure Rights

SmartPlant Engineering Manager incorporates user access as an integral product feature by using roles to define and maintain user privileges and right at the plant structure level. These rights are valid only within the plant structure with which the right is associated. To access the plant structure through SmartPlant Engineering Manager, the user must belong to a role that has at least read-only access to the plant structure.

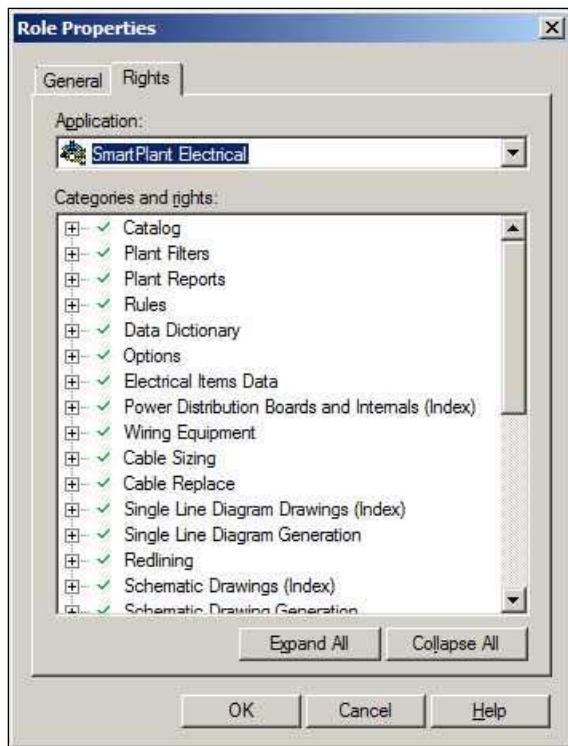


User access comprises two distinct sets of rights:

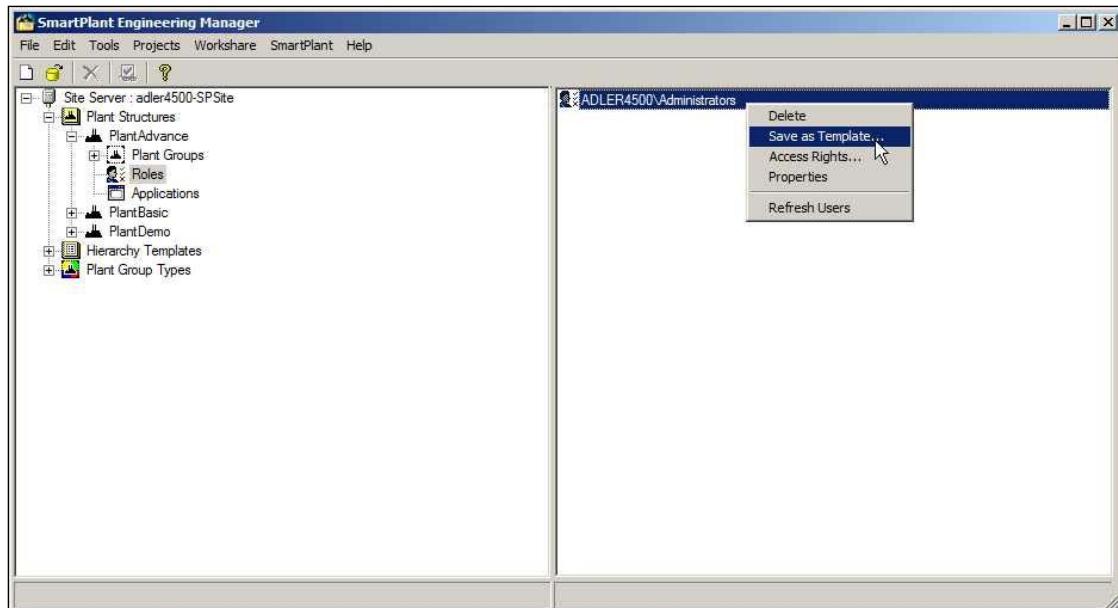
- Rights for the SmartPlant Engineering Manager Application



- Rights for Engineering Applications

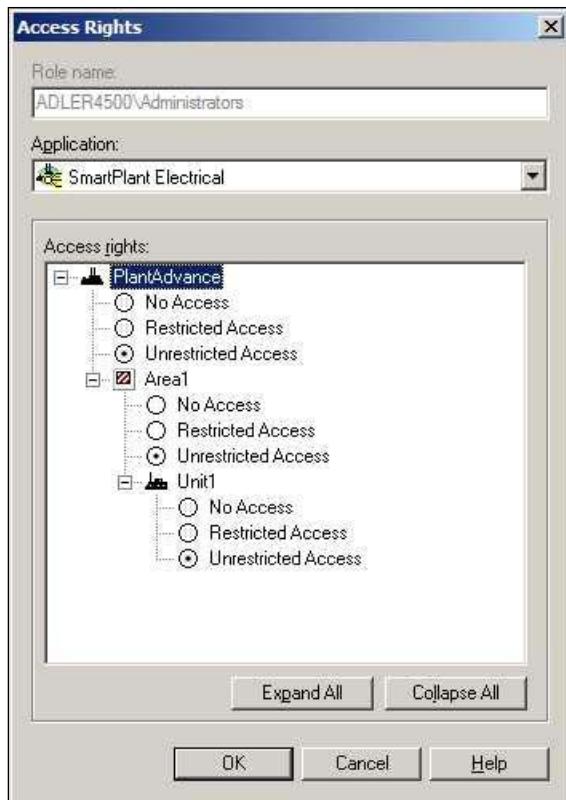


SmartPlant Engineering Manager provides role templates to help you easily create new roles for the engineering applications. You can create templates for specific roles and then use the template multiple times.



SmartPlant Electrical Hierarchy-Level Access Rights

After defining the roles for your project, you can set appropriate access rights for your groups of users on a particular plant hierarchy level. This way you can determine which users can access certain sections of your project. For example, if your hierarchy is defined as Plant/Area/Unit, you can select a group of users and grant them different access rights to various plants, areas, and units in your project.



Backup and Restore

The Backup and Restore functionality allows you to backup all data on a site and/or plant structure basis, including application/engineering data, for means of disaster recovery.

Using the zip.exe and unzip.exe executables installed with SmartPlant Engineering Manager, backups are bundled into a zip files that are named using the top level object name and type contained within the backup. These zip files contain various files depending on the type of backup and options selected. All database schemas in each backup instance are saved into a single file. For plant backups, files in the plant structure paths are bundled into a zip file inside the main plant backup zip file. The reference data files are archived into individual zip files and added to the main plant backup zip file.

A manifest document describing the data included in the backup is included in the backup zip file. This manifest document provides the input necessary to restore the data at a later time.

You can schedule backups using the scheduling functionality provided in the Backup wizard. This scheduling functionality uses the Windows Task Scheduling system to list, delete or modify

a backup schedule. Each backup schedule is stored in a control file in the \Engineering Manager\Schedules folder. The control file contents describe the type of backup and all other information needed in order to accomplish the scheduled backup.

Note: The steps require performing the back and the restore exists in SmartPlant Engineering Manager User's guide.

Copy a Plant Structure

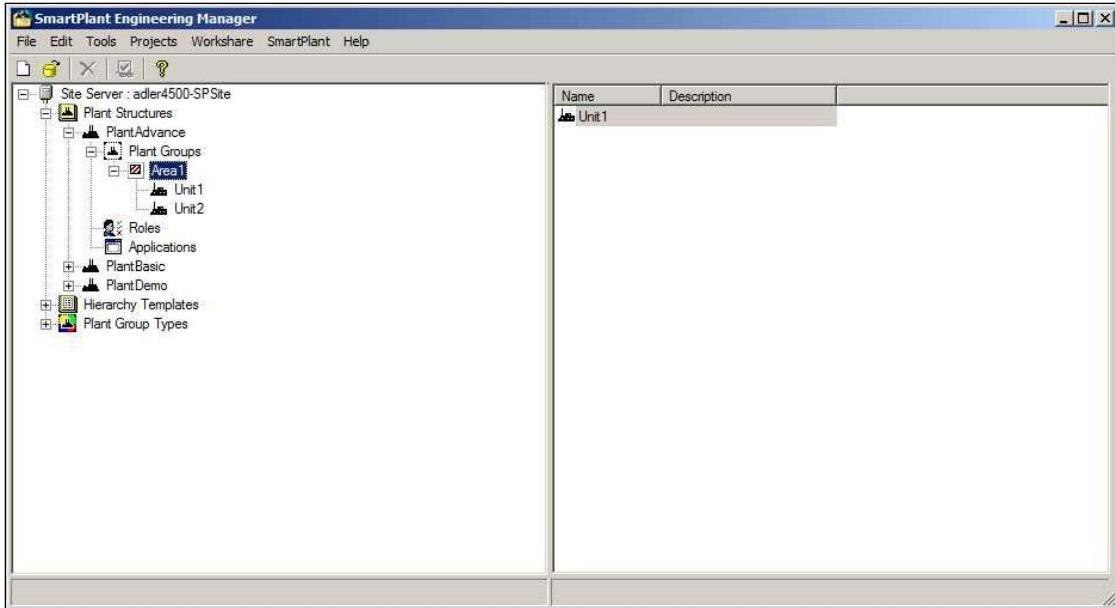
Copy plant structure allows you to make a copy of an existing plant structure and use this copy template to create new plant structures in the same or different site, and within or between different versions of Oracle and SQL server. The procedure has two stages

- **Save Plant Structure** - You use this command to save the plant structure you want to copy.
- **Load Plant Structure** - This wizard guides you through the process of loading and renaming the plant structure that you want to copy.

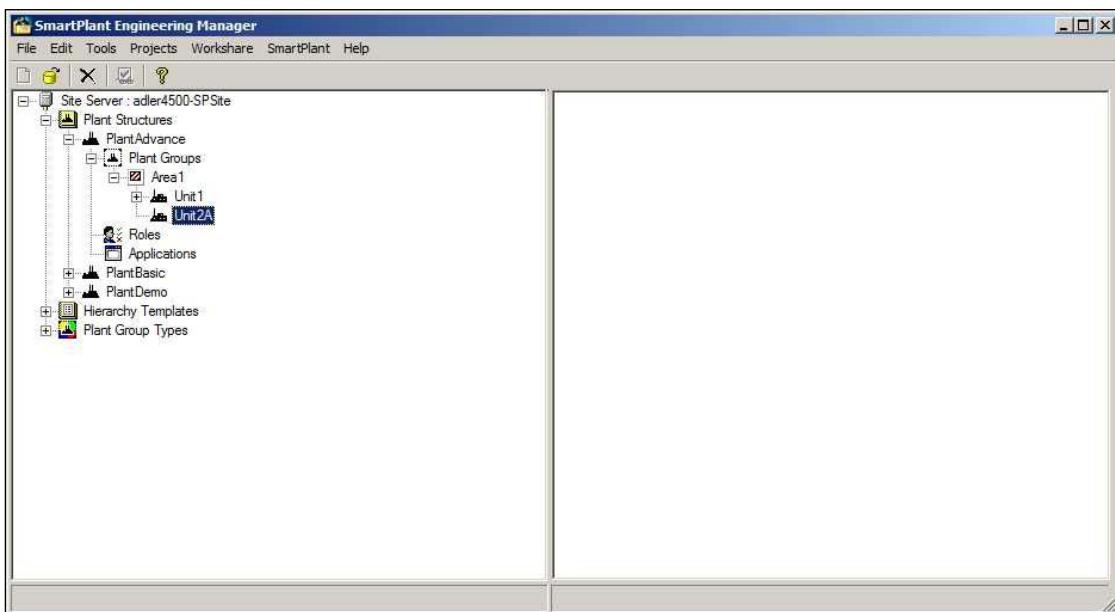
Change Plant Group Hierarchy Names

1. Expand the **Plant Structures** hierarchy, and locate the plant structure whose area or unit names you want to change.
2. Expand the plant structure and select the area or unit you want to change.

To demonstrate it, let's create new unit under 'Area 1' and name it Unit2 (unit code = 002).



3. Right-click on 'Unit 2' , and on the shortcut menu selects **Properties**.
4. On the **Properties** dialog box, change the name from Unit2 to Unit2A.



Note: The first time you log in to smart plant Electrical after change plant group name, you get prompt suggest you to run apply naming conventions to propagate the changes of the plant group name to the item tags. To propagate the changes, you select tools -> apply naming convention.

Let's change back the unit name from 'Unit2A' to 'Unit2'.

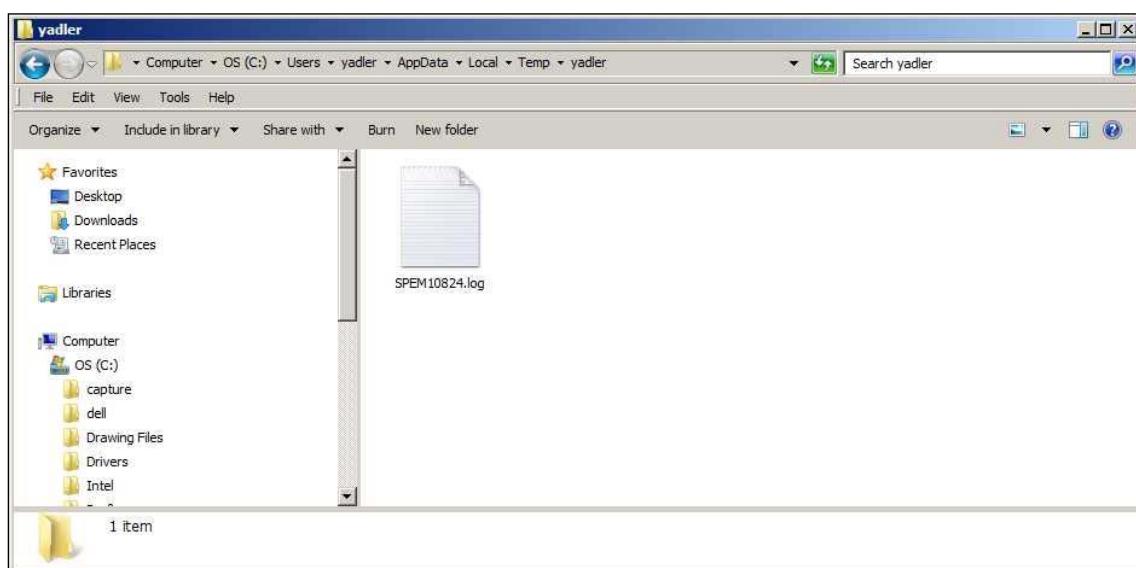
Running Engineering Manager in Debug Mode

Run Engineering Manager with the parameter **/DebugLevel=0**



It creates log file under the user temp directory.

The log file name is **SPEM<xxxxx>.log**.



SmartPlant Electrical Report Customization

This lab introduces various methods, tips and tricks that are used for report customization. It is important to mention at this point that this lab does not cover Visual Basic code which may be needed for complex reports.

Participants will be shown how to perform the following tasks:

- ❑ Adding ‘Tables’ for a report
- ❑ Using hidden areas of report for data manipulation
- ❑ Using Excel functions
- ❑ Special cases (cable from/to, space heater)
- ❑ United properties and SI values
- ❑ Grouping data by common properties
- ❑ Adding cover sheet to report

Exercise 1 – Cable Schedule

Example #1: Cable Schedule report

Overview

The cable schedule report includes two columns for ‘From’ & ‘To’ in columns E and F (under headers Side1 and Side 2).

The report also includes 2 calculated properties in columns G and H that bring again the ‘From’ & ‘To’ but with more information, if the ‘From’ or the ‘To’ is a circuit type.

The calculated properties use an excel IF statement, to check the object type of the ‘From’ or ‘To’. And if it finds the object is a circuit, it concatenates the names of the PDB, Cell & Circuit names.

A	B	C	D	E	F	G	H
1							
2							
3							
4							
5	Cable Name	Cable Spec.	Cable Formation	Actual Length	Equip. Cable Side 1	Equip. Cable Side 2	From(Side 1 Calculated) From(Side 2 Calculated)
6	CABLE-3	Power - NEC	3 x 10 AWG + 1 x 10 AWG	100.0 m	CKT-3	M-3	PDB-2/CELL-3/CKT-3 M-3
7							
8	CABLE-2	Power - NEC	3 x 10 AWG + 1 x 10 AWG	100.0 m	S-2	CKT-2	S-2 PDB-2/CELL-2/CKT-2
9							
10	CABLE-4	Power - NEC	3 x 10 AWG + 1 x 10 AWG	100.0 m			0 0
11							
12							
13							
14							

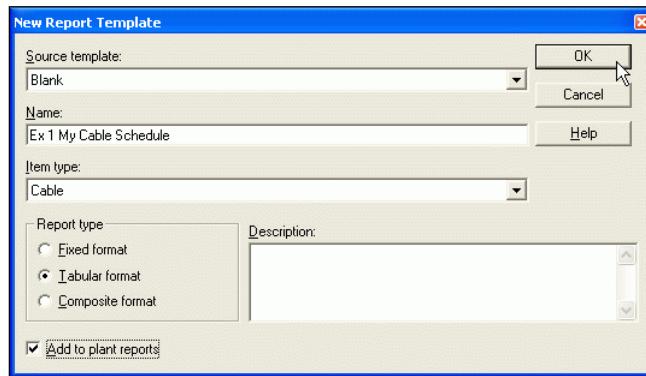
The report uses hidden columns (AA - AH) to stores addition information such as the type of the object, the PDB, cell and circuit names of the ‘From’ and ‘To’ sides.

At run-time, a custom function (IF statement) in the report is called. It checks the type of the objects and when it's a 'circuit', it concatenates the appropriate cells.

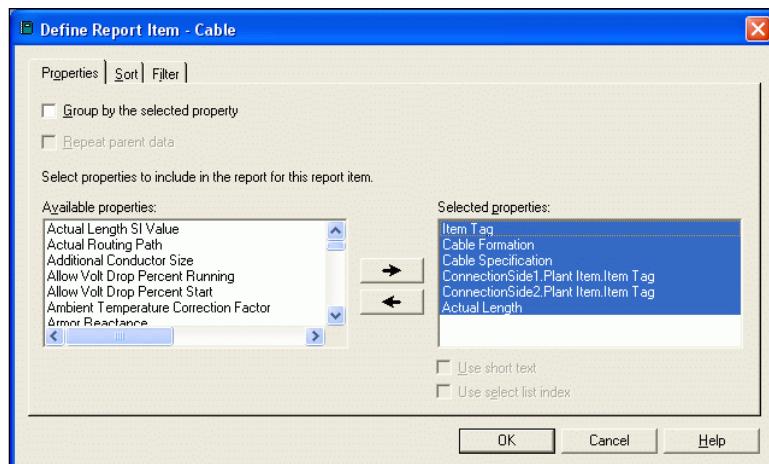
This exercise demonstrates the use of Excel functions in order to produce enhance report.

Steps to create the report

1. Create a new tabulated report based on a blank template.
 - Main item type = 'Cable'
 - Report name: Ex 1 My Cable Schedule



2. Select the following properties for cable item type:
 - Item Tag
 - Cable Formation
 - Cable Specification
 - ConnectionSide1.PlantItem.ItemTag
 - ConnectionSide2.PlantItem.ItemTag
 - Actual Length



3. Add 2 new tables below the **Cable** table:
 - Equipment ConnectionSide1 Cable

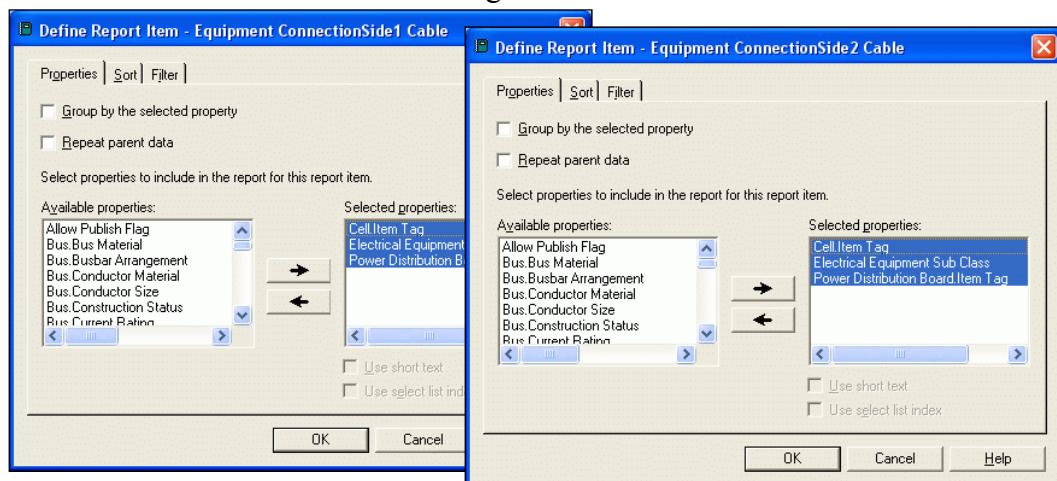
- Equipment ConnectionSide2 Cable

These relationships enable to retrieve information about the nature of connected equipment in side1 and side2 of the cable.

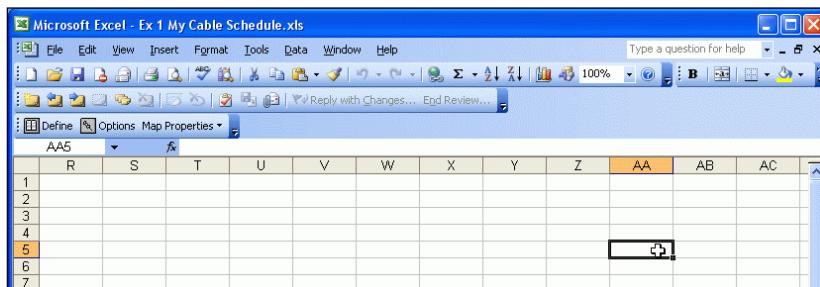


4. For each of these tables select the following properties:

- Cell.Item Tag
- Electrical Equipment Sub Class
- Power Distribution Board.Item Tag



5. Close the define report item dialog boxes and return to the excel page.
6. Create hidden area in the excel by scroll to the right until you get to cell AA5.



7. Create the following headers in cells AA5 to AH5.

W	X	Y	Z	AA	AB	AC	AD	AE	AF	AG	AH	AI	AJ
1													
2													
3													
4													
5	Type1	PDB1	Cell1	Side1	Type2	PDB2	Cell2	Side1					
6													
7													

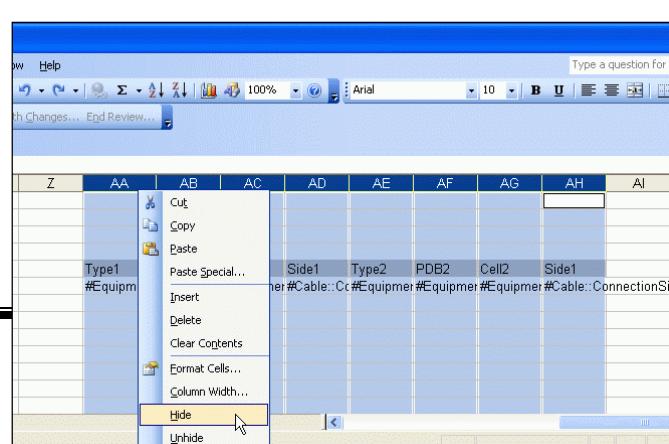
8. Use the information in the below **Mapping table** to map columns AA6 till AH6. And place the properties (macros) coming from the extra two tables, 'Equipment ConnectionSide1 Cable', and 'Equipment ConnectionSide1 Cable 2'. And from the cable table.

AA6	T	U	V	W	X	Y	Z	AA	AB	AC	AD	AE	
1													
2													
3													
4													
5													
6													
7													

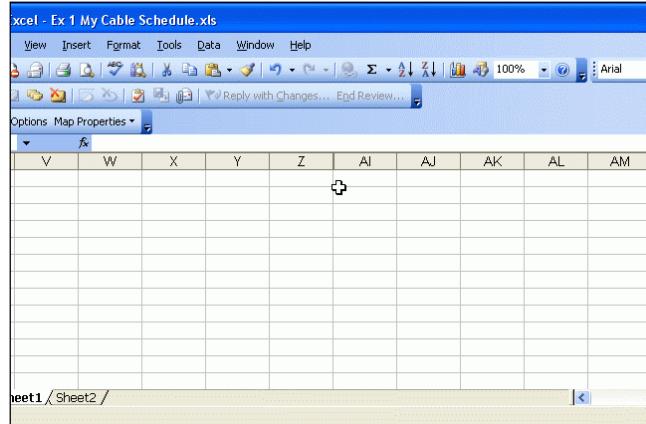
Mapping table

Column	Table	Property	Header
AA6	Equipment ConnectionSide1Cable	ElectricalEquipmentSubClass	Type1
AB6	Equipment ConnectionSide1 Cable	PowerDistributionBoard.ItemTag	PDB1
AC6	Equipment ConnectionSide1 Cable	Cell.Item Tag	Cell1
AD6	Cable	ConnectionSide1.Plant Item.Item Tag	Side1
AE6	Equipment ConnectionSide2 Cable	Electrical Equipment Sub Class	Type2
AF6	Equipment ConnectionSide2 Cable	Power Distribution Board.Item Tag	PDB2
AG6	Equipment ConnectionSide2 Cable	Cell.Item Tag	Cell2
AH6	Cable	ConnectionSide2.Plant Item.Item Tag	Side2

9. After finish work on the headers in cells AA5 to AH5, and on mapping the properties in cell AA6 to AH6. Hide the area from AA to AH by select the columns AA to AH, and select right click and hide.



10. Scroll to the left and create the headers for the report in cells A5 to H5



11. Use the information in the below **Mapping table** to map columns **A6 till H6**.

Mapping table			
Column	Table	Property	Header
A6	Cable	Item Tag	Cable Name
B6	Cable	Cable Specification	Cable Spec.
C6	Cable	Cable Formation	Cable Formation
D6	Cable	Actual Length	Actual Length
E6	Cable	ConnectionSide1.Plant Item.Item Tag	Equip. Cable Side 1
F6	Cable	ConnectionSide1.Plant Item.Item Tag	Equip. Cable Side 2
G6	Cable	= IF(AA6="Circuit",AB6 & "/" & AC6 & "/" & AD6,AD6)	From(Side 1 Calculated)
H6	Cable	= IF(AE6="Circuit",AF6 & "/" & AG6 & "/" & AH6,AH6)	From(Side 2 Calculated)

- 11.1. In columns G6 and G7 we use excel “IF” function, in order to calculate the PDB-Cell–Circuit names.

To enter the IF statement to cell G6 do the following:

1. Select cell G6

2. Type the following:

= IF(AA6="Circuit",AB6 & "/" & AC6 & "/" & AD6, AD6)

3. Type Enter.

Microsoft Excel - Ex 1 My Cable Schedule.xls						
	F	G	H	I	J	K
1						
2						
3						
4						
5						
6						
7						

5 **Equip. Cable Side 2 From(Side 1 Calculated) From(Side 2 Calculated)**

6 **#Cable::ConnectionSide1:#Cable::ConnectionSide1:#Cable::ConnectionSide2:Plant Item.Item Tag::ConnectionSi**

11.2. To enter the IF statement to cell H6 do the following:

1. Select cell H6

2. Type the following:

= IF(AE6="Circuit",AF6 & "/" & AG6 & "/" & AH6, AH6)

3. Type Enter.

Microsoft Excel - Ex 1 My Cable Schedule.xls						
	G	H	I	J	K	L
1						
2						
3						
4						
5						
6						
7						

5 **From(Side 1 Calculated) From(Side 2 Calculated)**

6 **#Cable::ConnectionSide1:#Cable::ConnectionSide2:Plant Item.Item Tag::ConnectionSi**

G6						
$\text{= IF(AA6="Circuit",AB6 & "/" & AC6 & "/" & AD6, AD6)}$						
A	B	C	D	E	F	G
1						
2						
3						
4						
5	Cable Name	Cable Spec.	Formation	Est. Length	Side1	Site2
6	#Cable::Item Tag:#Cable::Cable S#Cable::Cable #Cable::Estim#Cable::C #Cable::C #Cable::ConnectionSide1:#Cable::ConnectionSide2					

12. Save and close Excel.

13. To test the report, create in EI the following items:

1. Power Distribution Board, 'PDB-2' with bus 'BUS-2'.

Cell 'CELL-2 with incomer circuit, 'CKT-2', and cell 'CELL-3' with feeder circuit 'CKT-3'.

2. Offsite Power Supply,'S-2'.

3. Motor,'M-3'.

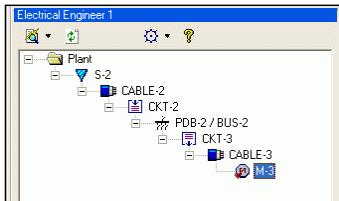
4. Power cables, 'CABLE-2','CABLE-3', and 'CABLE-4'.

Notes: 1. Use the typical cable, '3+1/C-10AWG -(Power / NEC)'.

2. Enter 100 m for the cable design length.

5. Connect in Electrical Engineer, CABLE-2 between the offsite power supply, and

the incomer circuit. And CABLE-3 between the feeder circuit and the motor.



14. In SmartPlant Electrical, select a range of power cables and run the report, or if you want to run the report for all cables, do not select any specific item or item type, and run the report.

15. After the report is generated, explore the results.

A	B	C	D	E	F	G	H
1							
2							
3							
4							
5	Cable Name	Cable Spec.	Cable Formation	Actual Length	Equip. Cable Side 1	Equip. Cable Side 2	From(Side 1 Calculated)
6	CABLE-3	Power - NEC	3 x 10 AWG + 1 x 10 AWG	100.0 m	CKT-3	M-3	PDB-2/CELL-3/CKT-3
7							M-3
8	CABLE-2	Power - NEC	3 x 10 AWG + 1 x 10 AWG	100.0 m	S-2	CKT-2	S-2
9							PDB-2/CELL-2/CKT-2
10	CABLE-4	Power - NEC	3 x 10 AWG + 1 x 10 AWG	100.0 m			0
11							0
12							
13							
14							
15							

This exercise provides a nice example for various features of the Excel report tool:

- Adding tables to retrieve information
- Using hidden areas of the report for data manipulation
- Using Excel functions for data manipulation

You may also notice that in cases a cable is not connected, zero appears. The IF function can be enhanced so that when the item type (cells AA6 & AE6) is blank, the resulting value will be populated with a space.

Exercise 2 – United Properties

Attributes that require engineering units in SmartPlant Electrical uses united data type of properties. This applies to most attributes with some exception like Motor Rated Power and Rated Voltage (which require specific values).

In practice, when such united attribute is used in SmartPlant Electrical, there are actually two properties inside the database. The one which is seen on the user interface is the one that uses the formats, with the unit of measure (alpha-numeric). The other property can be seen only from within the report tool holds the SI value.

It's pure numeric field which the application uses for conversion, based on the selected format.

An example:

1. Create new report selecting the previous report (Ex 1 My Cable Schedule) to be your source template. Name the new report, '**Ex 2 My Cable Schedule**'. Also check the option 'add to plant reports' check box.
2. Open the new report 'Ex 2 My Cable Schedule' in Edit mode.
3. Select the **Cable** table and click the **Define** button.
4. Scroll down till you see properties called '**Actual Length SI Value**'.
5. Select '**Actual Length SI Value**' and add it to the selected properties.
6. Map this property on the report.
7. Generate the report, making sure that you enter some design length values first.

Cable Name	Cable Spec.	Cable Formation	Actual Length	Actual Length SI	Equip. Cable Side 1	Equip. Cable Side 2	From(Side 1 Calculated)	From(Side 2 Calculated)
CABLE-3	Power - NEC	3 x 10 AWG + 1 x 100.0 m		100	CKT-3	M-3	PDB-2/CELL-3/CKT-3	M-3
CABLE-2	Power - NEC	3 x 10 AWG + 1 x 100.0 m		100	S-2	CKT-2	S-2	PDB-2/CELL-2/CKT-2
CABLE-4	Power - NEC	3 x 10 AWG + 1 x 100.0 m		100			0	0

Note:

The value shown in the SI is in meters. Having the ability to use a standard unit of measure enables to perform all types of calculations without the need to separate the number and the unit of measure.

Exercise 3 – Sorting Data

In order to sort data in a deliverable report according to one or more defined properties, it is possible to use the Sorting mechanism of Define Report.

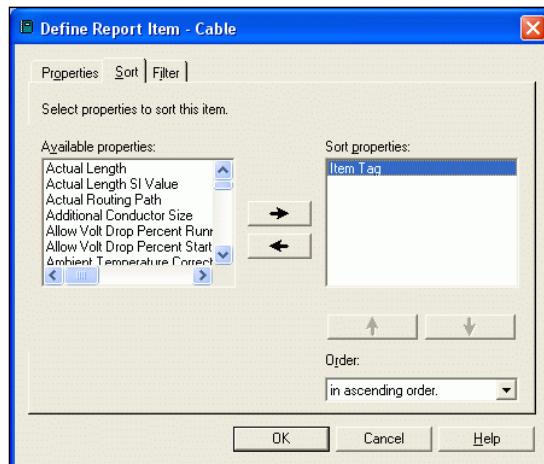
An example:

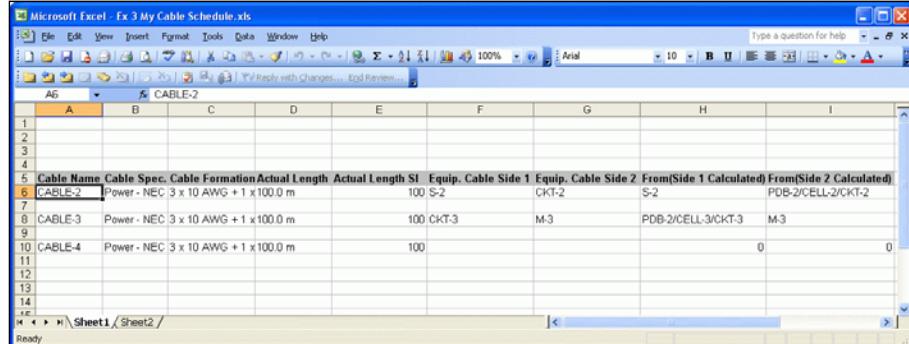
1. Create new report selecting the previous report (Ex 2 My Cable Schedule) to be your source template. Name the new report, '**Ex 3 My Cable Schedule**'. Also check the option 'add to plant reports' check box.

2. Open the report, ‘Ex 3 My Cable Schedule’ in ‘Edit’ mode.
3. Select the **Cable** table and click the **Define** button.
4. Click the **Sort** tab.
5. Select one or more properties for which you want to sort the report data.

 **Tip**

- Selected properties should be mapped on the report template.
6. Use the → and ← buttons to navigate the property into/from the ‘Sort properties’ field.
 7. Use the ↑ and ↓ buttons to define the order of the ‘Sort properties’ by which data is to be sorted.
 8. Under **Order**, select the desired sorting order: ascending or descending.
 9. Save the template and generate the report, making sure that you have retrieved the data sorted in the desired order.



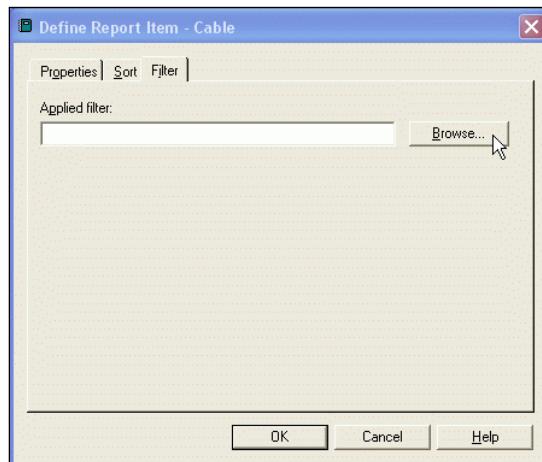


Cable Name	Cable Spec.	Cable Formation	Actual Length	Actual Length SI	Equip. Cable Side 1	Equip. Cable Side 2	From(Side 1 Calculated)	From(Side 2 Calculated)
CABLE-2	Power - NEC 3 x 10 AWG + 1 x 100.0 m		100	S-2	CKT-2	S-2	PDB-2/CELL-2/CKT-2	
CABLE-3	Power - NEC 3 x 10 AWG + 1 x 100.0 m		100	CKT-3	M-3	PDB-2/CELL-3/CKT-3	M-3	
CABLE-4	Power - NEC 3 x 10 AWG + 1 x 100.0 m		100			0	0	

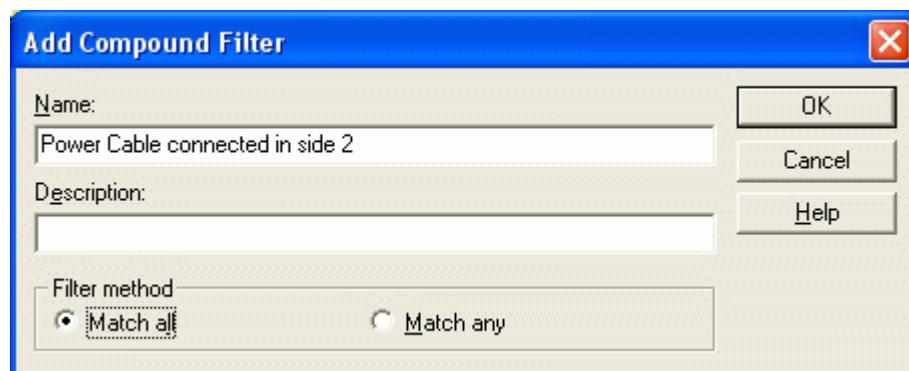
Exercise 4 – Filtering Data

In order to filter data in a deliverable report according to one or more defined properties, it is possible to use the Filter Manager.

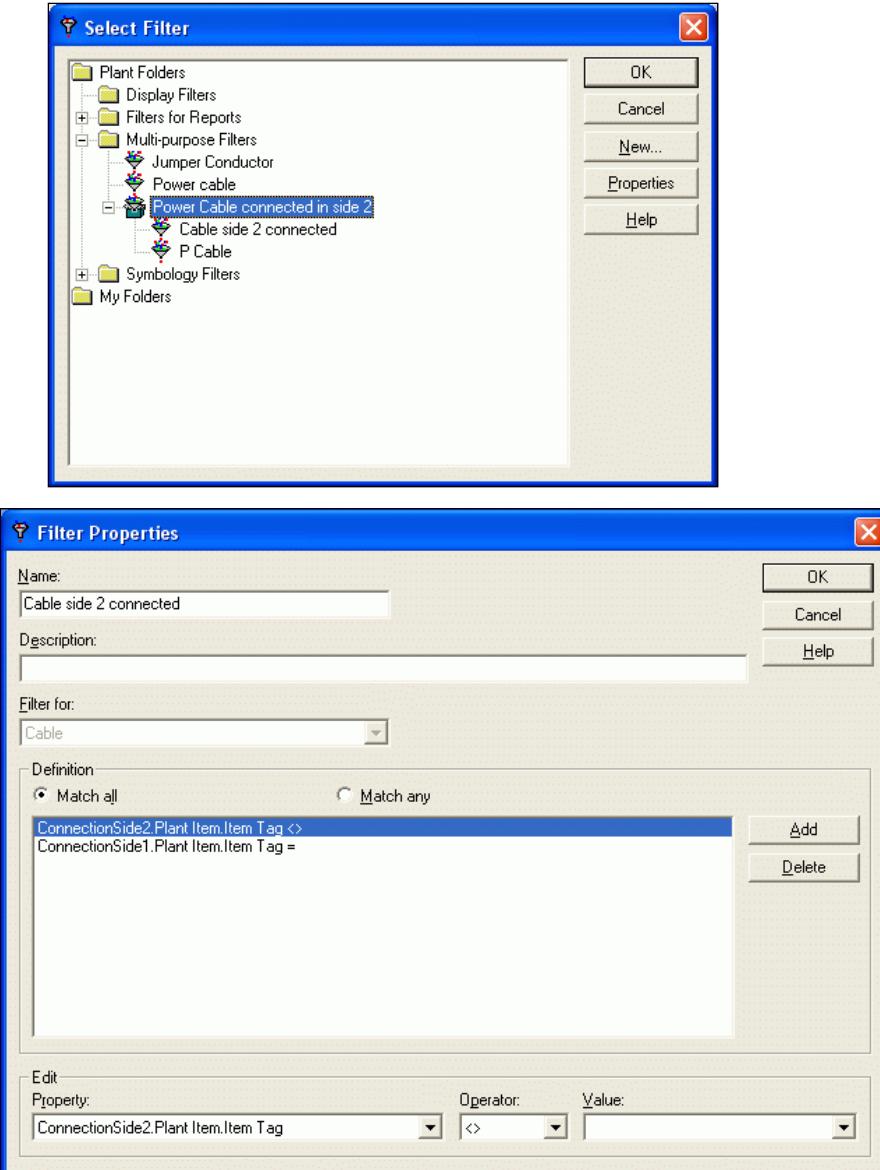
1. Create new report selecting the previous report (Ex 3 My Cable Schedule) to be your source template. Name the new report, '**Ex 4 My Cable Schedule**'. Also check the option 'add to plant reports' check box.
2. Open the report, '**Ex 4 My Cable Schedule**' in 'Edit' mode.
3. Select the **Cable** table and click the **Define** button.
4. Click the **Filter** tab.



5. Click **Browse** to open the Select Filter windows.
6. Create compound filter under 'multi-purpose filters' folder and name it 'Power Cable Connected in side 2'.



Create 2 simple filters in the compound filter.



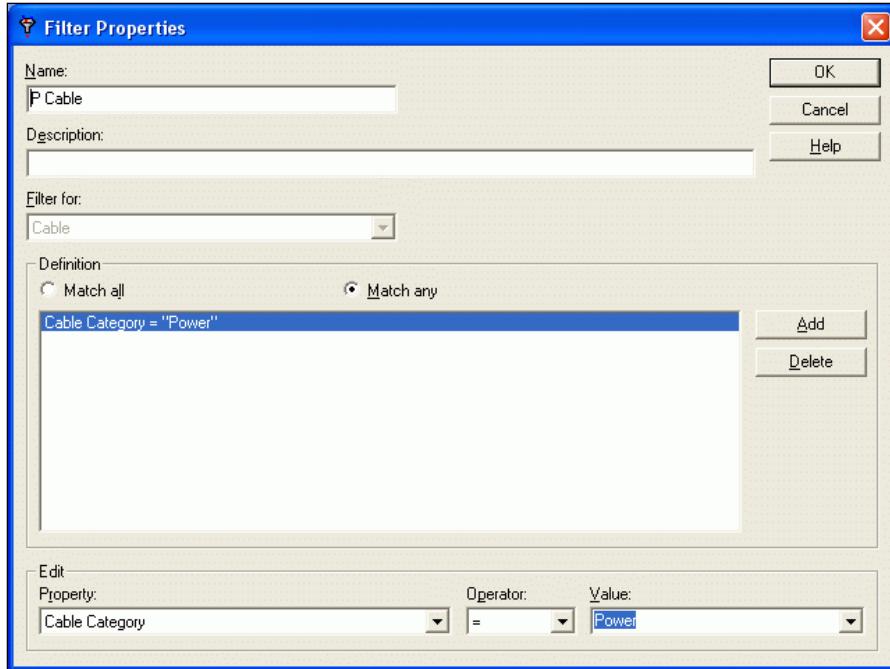
Note: The simple filter ‘cable side 2 connected’ uses the property connectionSide<X>.ItemTag (X = 1 stands for “from” side. And X = 2 stand for “to” side) to determine connection.

Using match all for the following 2 conditions

“**ConnectionSide2.PlantItemTag <>** “ (no value/data in the “Value” field)

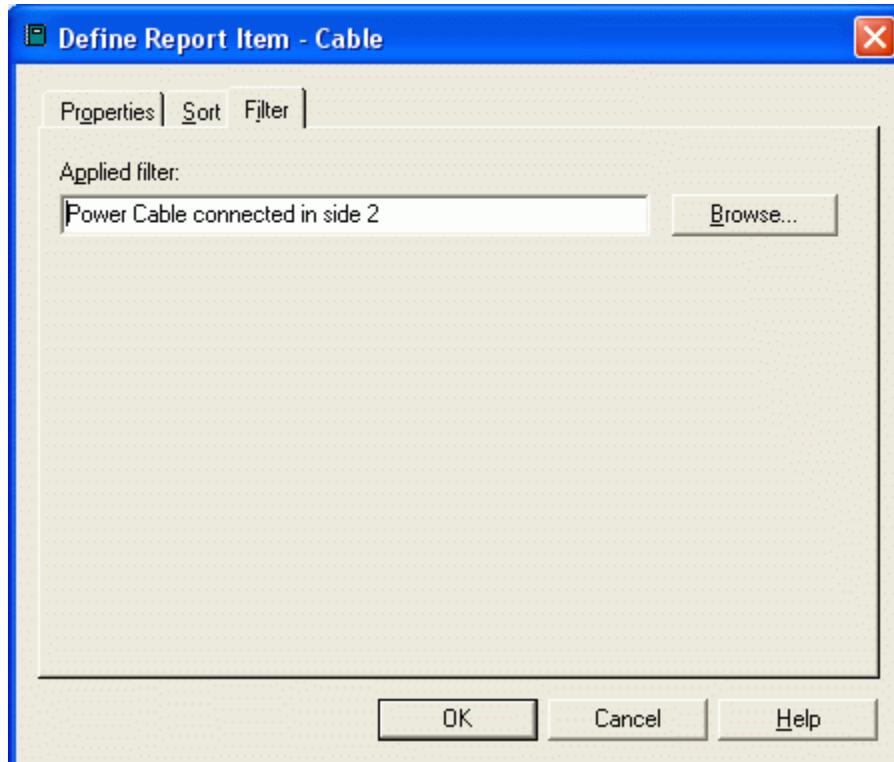
“**ConnectionSide1.PlantItemTag = “** (no value/data in the “Value” field)

Will make sure to get **Only** cables connected in side 2 (“to”) **AND** disconnected in side 1 (“from”).



7. After complete creating the 2 simple filters, return to the select filter select.

Make sure to select the compound filter, select OK to return the 'define report Item - cable' screen.



Select OK to return the 'define report contents' screen. And select close

to return the excel sheet. Save the change and close excel.

8. To test the filter. Disconnect cable ‘CABLE-3’ from circuit ‘CKT-3’.

Select the 3 cables (CABLE-2, CABLE-3, CABLE-4) and run the report.

You should get only power cables that connected to side 2 and disconnect from side1 (CABLE-3).

9. After complete the exercise, reconnect CABLE-3 to CKT-3.

Exercise 5 – Displaying Plant Group Names in a Report

The purpose of this exercise is to show how a report can include the plant group name of an item.

To start with, make sure that appropriate definitions were made to the item type for which you want to do this lab. In this example, the cable item type is associated with the unit.

1. Create new report selecting the previous report (Ex 4 My Cable Schedule) to be your source template. Name the new report, ‘**Ex 5 My Cable Schedule**’. Also check the option ‘add to plant reports’ check box.
2. Open the report, ‘**Ex 5 My Cable Schedule**’ in ‘Edit’ mode.
3. Select the **Cable** table and click the **Define** button.
4. In the properties tab add the **PlantGroup.Name** property to the selected properties.
5. In the filter tab delete the existing filter.
6. Select ok to return to the excel sheet and map the property to the report.

#Cable:Cable!Cable:Cable For #Cable:ActualLength#Cable:ActualLengthSI#Cable:PlantGroup#Cable:ConnectionSide1#Cable:ConnectionSide2#Cable:ConnectionSide1#Cable:ConnSide1#Cable:ConnSide2. The 'Plant Group' column is highlighted in yellow."/>

A	B	C	D	E	F	G	H	I
1								
2								
3								
4								
5	Cable Name	Cable Spec	Cable Formation	Actual Length	Actual Length SI	Plant Group	Equip. Cable Side 1	Equip. Cable Side 2
6	=Cable!Item">#Cable:Cable!Cable:Cable For #Cable:ActualLength#Cable:ActualLengthSI#Cable:PlantGroup#Cable:ConnectionSide1#Cable:ConnectionSide2#Cable:ConnectionSide1#Cable:ConnSide1#Cable:ConnSide2							
7								
8								
9								
10								
11								
12								
13								

7. Save the excel.
8. Run the report and test the result.

Cable Name	Cable Spec.	Cable Formation	Actual Length	SI	Plant Group	Equip. Cable Side 1	Equip. Cable Side 2	From(Side 1 Calculated)	From(Side
CABLE-2	Power - NEC	3 x 10 AWG + 1 x 100.0 m	100	SPPlantAdv	S-2	CKT-2	S-2	PDB-2/CCL	PDB-2/CCL
CABLE-3	Power - NEC	3 x 10 AWG + 1 x 100.0 m	100	SPPlantAdv	CKT-3	M-3	PDB-2/CCL-3/CKT-3	M-3	
CABLE-4	Power - NEC	3 x 10 AWG + 1 x 100.0 m	100	SPPlantAdv				0	

Tips:

- Based on the plant group type of the item type, the appropriate plant group name appears.
- Creating the report by selecting the previous report as a source template also keeps the filter.

Exercise 6 – Grouping Data by Common Properties

The report mechanism enables to group information with common properties on different sheet. This exercise explains how this is done.

Suppose we want to divide the list of cables that appear on the cable schedule by some criteria, for example, by cable formation.

This is done by placing the property (for which we want to divide the data) on the header section. You can do that by following this procedure:

- Create new report selecting the previous report (Ex 5 My Cable Schedule) to be your source template. Name the new report, '**Ex 6 My Cable Schedule**'. Also check the option 'add to plant reports' check box.
- Open the report, '**Ex 6 My Cable Schedule**' in 'Edit' mode.
- Map the **Cable Formation** property into the header area.

Tip

- The header area of the report is defined in the **Report Options** dialog box.



The screenshot shows an Excel spreadsheet titled "Microsoft Excel - Ex 6 My Cable Schedule.xls". The active sheet is "Sheet1". The data starts with a header row containing "Formation" and several calculated columns. Below this, there is a single row of data: "Formation: #Cable:Cable Formation:CableFormation#". The rest of the rows are empty.

	A	B	C	D	E	F	G	H	I
1									
2		Formation:	#Cable:Cable Formation:CableFormation#						
3									
4									
5		Cable Name	Cable Spec.	Actual Length	Actual Length SI	Plant Group	Equip. Cable Side 1	Equip. Cable Side 2	From(Side 1 Calculated) From(Side 2 Calculated)
6		#Cable: Item	#Cable:Cable	#Cable: Actual	#Cable: Plant	G#Cable: ConnectionSide1#Cable: ConnectionSide1#Cable: ConnectionSide2 Plan			
7									
8									
9									
10									
11									
12									
13									
14									
15									

4. Save and generate the report.

Depending on your data, Excel adds additional sheets where each sheet lists the cables that match specific cable formation.

Note that the second sheet will always be empty (this sheet is preserved to composite format type).

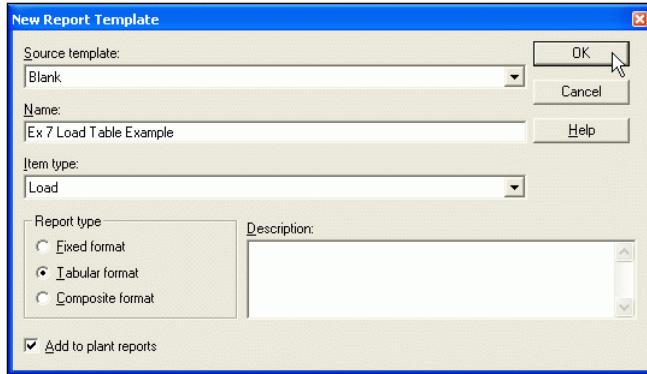
The screenshot shows an Excel spreadsheet titled "Microsoft Excel - Ex 6 My Cable Schedule.xls". The active sheet is "Sheet1". The data starts with a header row containing "Formation" and several calculated columns. Below this, there is a single row of data: "Formation: 3 x 10 AWG +1 x 10 AWG". The rest of the rows are empty.

	A	B	C	D	E	F	G	H	I
1									
2		Formation:	3 x 10 AWG +1 x 10 AWG						
3									
4									
5		Cable Name	Cable Spec.	Actual Length	Actual Length SI	Plant Group	Equip. Cable Side 1	Equip. Cable Side 2	From(Side 1 Calculated) From(Side 2 Calculated)
6		CABLE-2	Power - NEC	100.0 m	100	SPPlantAdv	S-2	CKT-2	S-2 PDB-2/CELL-2/CKT-2
7									
8		CABLE-3	Power - NEC	100.0 m	100	SPPlantAdv	CKT-3	M-3	PDB-2/CELL-3/CKT-3 M-3
9									
10		CABLE-4	Power - NEC	100.0 m	100	SPPlantAdv			0 0
11									
12									
13									
14									
15									

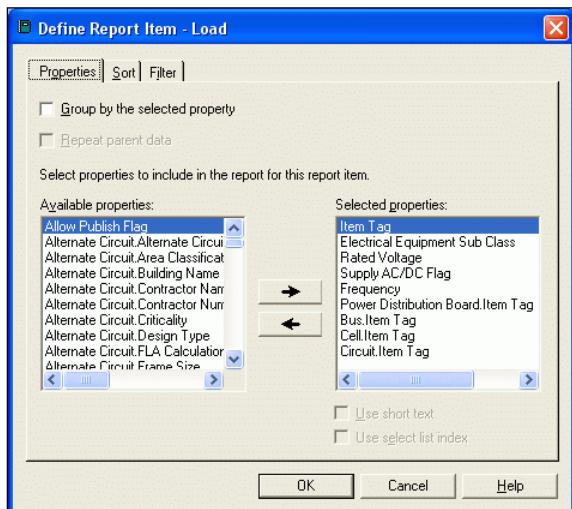
Exercise 7 - Adding Sub-Class Tables for Data Collection

This lab demonstrates a nice feature of the report mechanism. As mentioned earlier, the SmartPlant Electrical database is hierarchical. In many cases, a table has several child item types. An example of this is the **Load** table. In this example we shall create a report that retrieves information from it and selected child item types below it.

1. Create a new report based on the load table and name the report '**Ex 7 Load Table Example**'.



2. Select properties as shown.



3. Map these attributes on the report.

	A	B	C	D	E	F	G	H	I
1									
2									
3									
4									
5	Item Tag	Type	Rated Voltage	Supply	Frequency	PDB	Bus	Cell	Circuit
6	#Load::Item	#Load::Elec	#Load::Rated Vo	#Load::Supply	#Load::Frequ	#Load::Pow	#Load::Bus	#Load::Cel	#Load::Circ

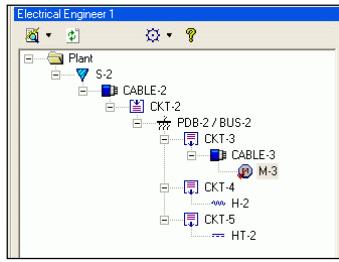
4. Save your report and run it.

	A	B	C	D	E	F	G	H	I
1									
2									
3									
4									
5	Item Tag	Type	Rated Voltage	Supply	Frequency	PDB	Bus	Cell	Circuit
6	H-2	Heater	460 V	AC	60 Hz	PDB-2	BUS-2	CELL-4	CKT-4
7									
8	M-3	Motor	460 V	AC	60 Hz	PDB-2	BUS-2	CELL-3	CKT-3
9									
10	M-2	Motor	460 V	AC	60 Hz				
11									
12	HT-2	Heat Trace	460 V	AC	60 Hz	PDB-2	BUS-2	CELL-5	CKT-5
13									
14									

Note: To get similar result. Add to BUS-2 new cell CELL-4 with feeder circuit CKT-4.

And another new cell CELL-5 with feeder circuit CKT-5.

Also create heater H-2 and heat trace HT-2 and connect them to circuit CKT-4 and CKT-5.

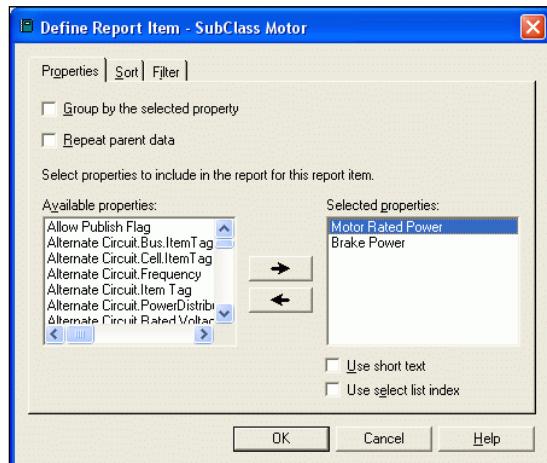


Suppose we want to include the rated power of the loads in this report.

Since each load item type has its own rated power (found in the child table), we cannot just add the rated power to the report as we did for the other properties.

In order to add these rated power properties and maybe others specific to the final item types, we need to add tables below load table (sub-class tables) as follows:

1. Re-open the report in ‘Edit’ mode.
2. Select the **Load** table.
3. Add the following tables:
 - SubClass Motor
 - SubClass Heater
 - SubClass Heat Trace
4. Select the **SubClass Motor** table and select Define to add the following properties:
 - Motor Rated Power
 - Brake Power



5. Select the **SubClass Heater** table and add the property **Heater Rated Power**.
6. Select the **SubClass Heat Trace** table and add the property **Heat Trace Rated Power**.

- Map these properties on the report.

Tip

- You can format the presentation to use border lines in order to enhance the presentation of the report.

A	B	C	D	E	F	G	H	I	J	K	L	M
1												
2												
3												
4												
5	Item Tag	Type	Rated Voltage	Supply	Frequency	PDB	Bus	Cell	Circuit	Motor	Heater	Heat Trace
6	#Load::Item	#Load::Elec	#Load::Rated Vo	#Load::Supply	#Load::Frequ	#Load::Pow	#Load::Bus	#Load::Cell	#Load::Circ	#SubClass Mo	#SubClass Mo	#SubClass H#SubClass Hea
7												
8												

- Save and re-generate the report.

You can observe that now we have the data coming from the final load item types.

A	B	C	D	E	F	G	H	I	J	K	L	M
1												
2												
3												
4												
5	Item Tag	Type	Rated Voltage	Supply	Frequency	PDB	Bus	Cell	Circuit	Motor	Heater	Heat Trace
6	H-2	Heater	460 V	AC	60 Hz	PDB-2	BUS-2	CELL-4	CKT-4			1 kW
7												
8	M-3	Motor	460 V	AC	60 Hz	PDB-2	BUS-2	CELL-3	CKT-3	1 hp	0.8 hp	
9												
10	M-2	Motor	460 V	AC	60 Hz					1 hp	0.8 hp	
11												
12	HT-2	Heat Trace	460 V	AC	60 Hz	PDB-2	BUS-2	CELL-5	CKT-5			1 kW
13												
14												

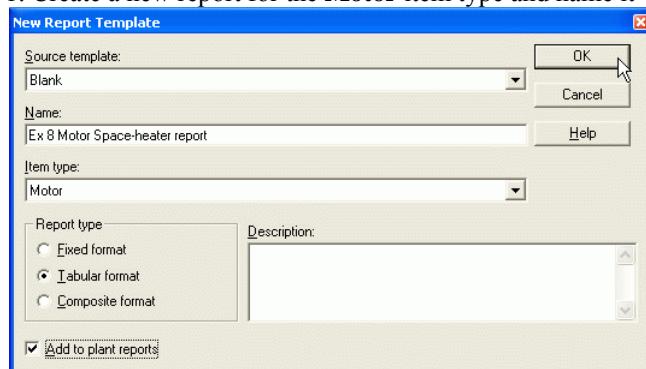
The shipped reports coming with SmartPlant Electrical installation include an example for this implementation ('PDB Load Summary' report).

Exercise 8 – Motor Space Heater

Motor can have a space heater to protect against condensation. A space heater is excluded from appearing as the rest of the heaters. Usually we want to show the space heater next to the motor it belongs to (as we do in the Electrical Index).

This lab demonstrates a simple report that shows the motor along with its heater.

- Create a new report for the **Motor** item type and name it 'Ex 8 Motor Space-heater report'.



While selecting the **Motor** table add a new table **Heater**.



2. For the motor select, 'Item Tag', 'Motor Rated Power', 'Rated Voltage', and 'Brake Power'.
3. For the heater, select the 'Item Tag', 'Heater Rated Power', and 'Rated Voltage'.
4. Map these properties on the report.

A	B	C	D	E	F	G	
1							
2							
3							
4	Motor			Heater			
5	Item Tag	Rated Power	Rated Voltage	Break Power	Item Tag	Rated Power	Rated Voltage
6	#Motor::Item Tag	#Motor::Rated Power	#Motor::Rated Voltage	#Motor::Break Power	#Heater::Item Tag	#Heater::Rated Power	#Heater::Rated Voltage
7							

5. Save the report and close Excel.
6. Add some motors with heaters.
7. Select some motors and run the report.
8. Check the report results.

A	B	C	D	E	F	G	
1							
2							
3							
4	Motor			Heater			
5	Item Tag	Rated Power	Rated Voltage	Break Power	Item Tag	Rated Power	Rated Voltage
6	M-4	1 hp	460 V	0.8 hp	H-3	100 W	120 V
7							
8	M-3	1 hp	460 V	0.8 hp			
9							
10	M-2	1 hp	460 V	0.8 hp			
11							
12							

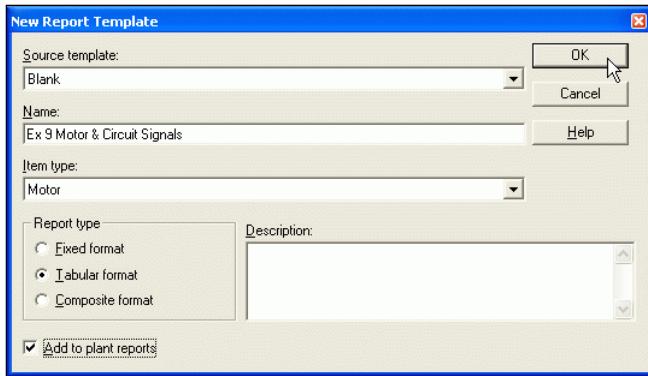
Note: To get similar result, add motor,'M-4' with space heater,'H-3'.

Exercise 9 – Motor and Circuit Signals

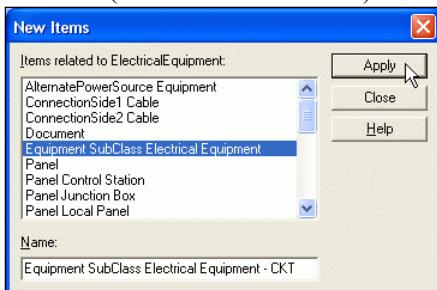
Motor as well as other electrical equipment items can have some associated signals that are used for transmitting various types of notifications from the field appliances to the control room. The circuit which feeds the motor also has signals that are related to the motor.

This lab demonstrates a simple report that shows the motor signals along with its circuit signal.

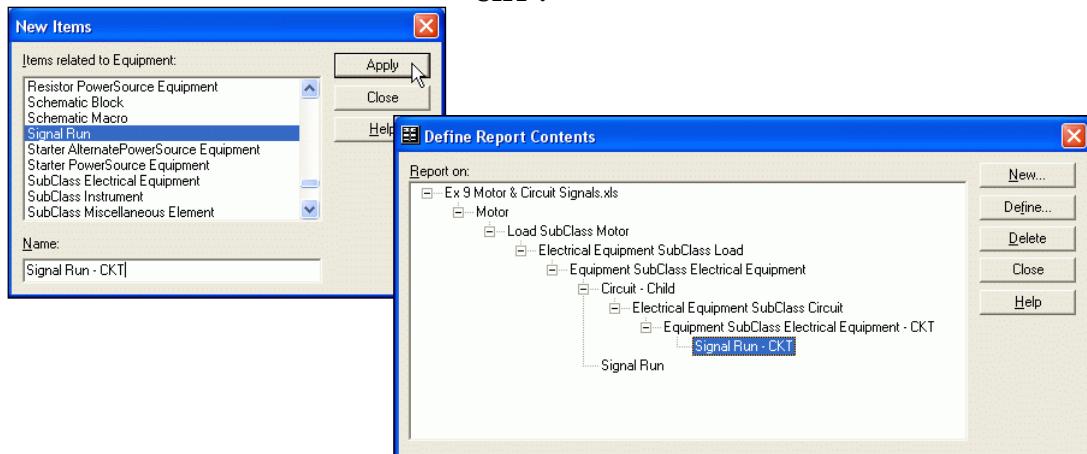
1. Create a new report for the **Motor** item type and name it '**Ex 9 Motor & Circuit Signals**'.



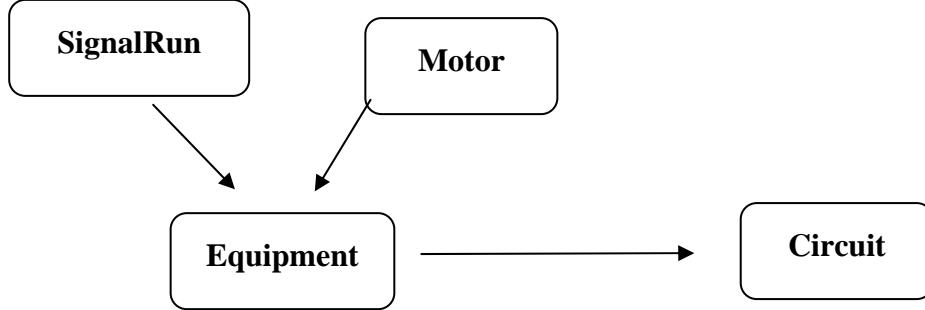
2. Add the following tables:
- From 'Motor' table add, '**Load SubClass Motor**'.
 - From 'Load SubClass Motor' add, '**Electrical Equipment SubClass Load**'.
 - From 'Electrical Equipment SubClass Load' add, '**Equipment SubClass Electrical Equipment**'.
 - From 'Equipment SubClass Electrical Equipment' add both, '**Circuit – Child**', and '**Signal Run**'.
 - From 'Circuit-Child' add, '**Electrical Equipment SubClass Circuit**'.
 - From 'Electrical Equipment SubClass Circuit' add, '**Equipment SubClass Electrical Equipment**' rename it (from the "Name:" field) to '**Equipment SubClass Electrical Equipment – CKT**', and select apply.



- g. From 'Equipment SubClass Electrical Equipment – CKT' add, '**Signal Run**' and rename it to '**Signal Run CKT**'.



This structure describes the class and relational connections between the motor, circuit and signal. The next figure shows the connections in general.



1. For the **Motor's 'Signal Run', and Circuit's 'Signal Run – CKT'** tables select the **Item Tag** property.
2. Map these properties on the report.

A	B	C	D
1			
2			
3			
Motor		Circuit	
5	Item Tag	Signal	Item Tag
8	#Motor::Item Tag#Signal Run::It#Circuit - Child:		#Signal Run - C
7			

3. Save the report and close Excel.
4. Add some motors and add signals to them.
5. Assign the motors to the circuit and add signals for the circuits.
6. Select several motors and run the report.
7. Check the report results.

A	B	C	D
1			
2			
3			
Motor		Circuit	
5	Item Tag	Signal	Item Tag
6	M-4		
7			
8	M-3	S-3	CKT-3
9			S-2
10	M-2		
11			
12			

Note: To get similar result, add signal 'S-2' to circuit 'CKT-3', and signal 'S-3' to motor 'M-3'.

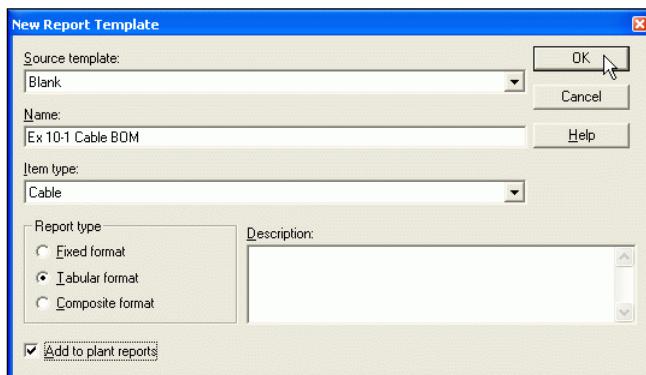
Exercise 10 – Bill of Materials Report

Creating a bill of material report uses a grouping option within the SmartPlant Electrical report generator. This option lets the administrator, count electrical items in many ways of filtering. The attributes that will be in use for this report are just those attributes that will classify the way of the filtering and counting. Each change in one of the attributes that creates the filter classification, will create a new line in the report.

Cables Bill of Material

The report counts the number of cables for a certain cable type.

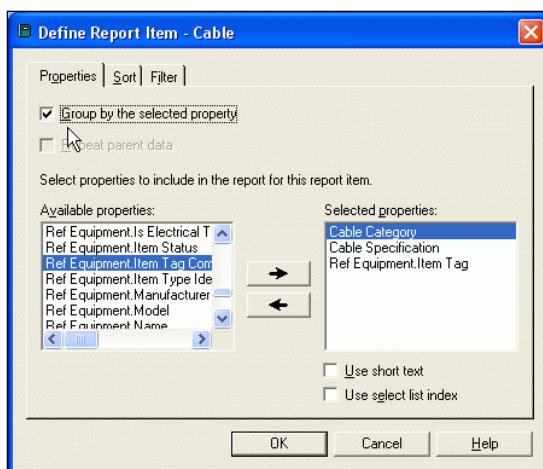
1. Create a new report for Cable item type and name it ‘Ex 10-1 Cable BOM’.



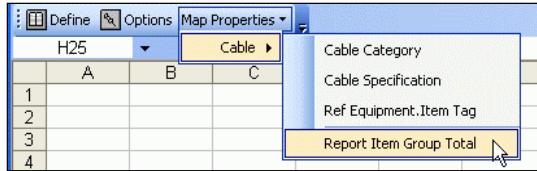
2. Add the following properties:

‘Cable Category’, ‘Cable Specification’, ‘Ref Equipment.Item Tag’.

3. In the ‘Define Report Item – Cable’ windows check the ‘Group by the selected property’.



Setting this check box will add a special property to map into the report, ‘Report Item Group Total’.



- Map all four of the properties on the report as shown.

	A	B	C	D	E
1	Cable Category:		#Cable::Cable Category::CableCategory#		
2					
3					
4					
5	Cable Spec.	Ref. Cable	Total		
6	#Cable::Cable Spec#Cable::Ref Eq#Cable::ReportItem Group Total::				
7					

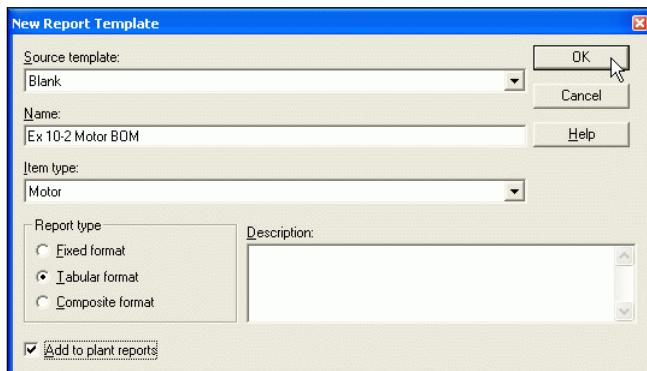
- Save the report and close Excel.
- Without selecting a specific item, run the report.
- Check the results.

Example:

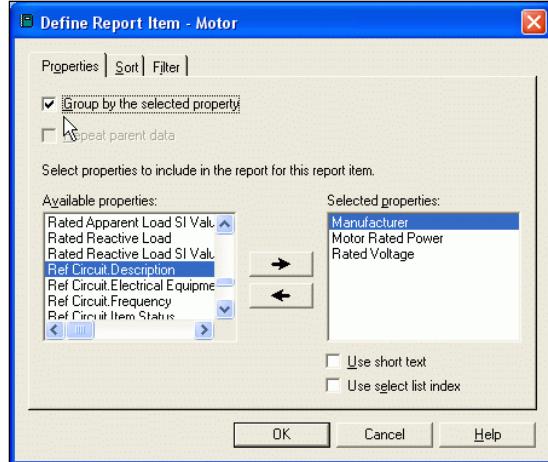
	A	B	C	D	E
1	Cable Category:	Power			
2					
3					
4					
5	Cable Spec.	Ref. Cable	Total		
6	Power - NEC	3+1/C-10AWG	3		
7					
8					

Motors Bill of Material

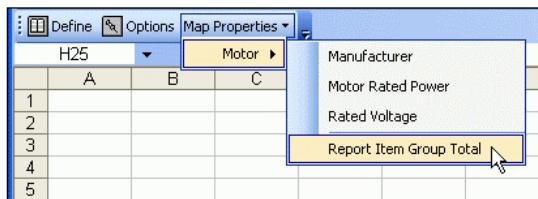
- Create a new report for Motor item type and name it '**Ex 10-2 Motor BOM**'.



- Add the following properties:
'Manufacturer', **'Motor Rated Power'**, **'Rated Voltage'**.
- In the 'Define Report Item – Motor' windows check the 'Group by the selected property'.



Setting this check box will add a special property to map into the report: ‘Report Item Group Total’.



4. Map all four of the properties on the report as shown.

	A	B	C	D	E
1					
2					
3					
4					
5	Manufacturer	Rated Power	Rated Voltage	Total	
6	#Motor::Manufa	#Motor::Motor R	#Motor::Rated Vo	#Motor::Report Item Group	
7					

5. Save the report and close Excel.
6. Without selecting a specific item, run the report.
7. Check the results.

Example:

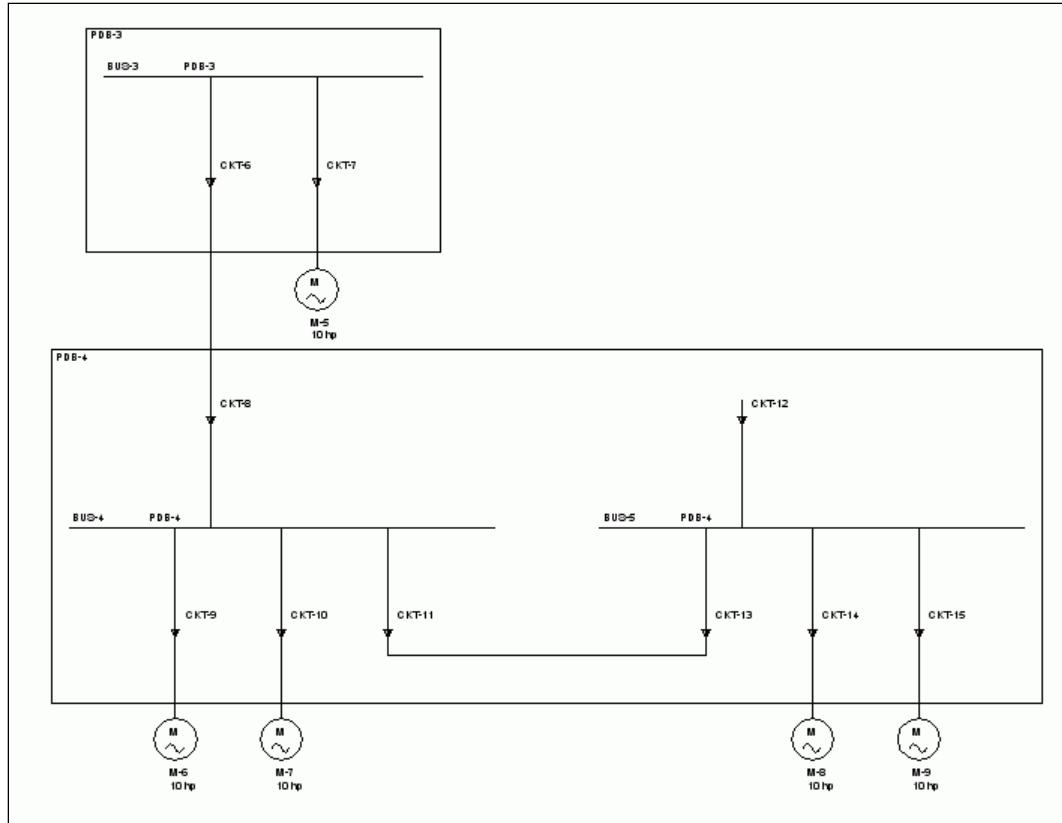
	A	B	C	D	E
1					
2					
3					
4					
5	Manufacturer	Rated Power	Rated Voltage	Total	
6		1 hp	460 V	2	
7					
8	Global Electric	1 hp	460 V	1	
9					
10					

Exercise 11 – 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** or the **Tabular Editor**. This report is based on either the consumed or rated power of the selected loads.

This lab summarizes the various variations of this report.

1. Create the following loads in SmartPlant Electrical (load names may vary according to the data you already have):
 - Add 5 motors (M-5 through M-9). For each motor, use:
 - Rated power = 10hp
 - Brake power = 8 hp
 - Rated voltage = 460 V
 - For motors M-5, M-6, M-8 & M-9 assign operating mode = ‘Continuous’, X Coincidence Factor = 1
 - For M-7 assign Operating mode = ‘Intermittent’, Y Coincidence Factor = 0.8
2. Apply a lookup table on these motors (use the default table that was shipped with SmartPlant Electrical, ‘Sample Lookup Tables for Motors’).
3. Add two power distribution boards and create the following associations (as mentioned previously, object names can vary depending on your existing data).



Tip

- Cables and circuit internals are not important for this exercise.

The following table shows the relative contribution of each load (already multiplied by corresponding coincidence factors):

Load Name	Operation Mode	kW	kVAr
M-5	Continuous	6.33	2.85
M-6	Continuous	6.33	2.85
M-7	Intermittent	5.06	2.28
M-8	Continuous	6.33	2.85
M-9	Continuous	6.33	2.85

- When done, verify the report results (manually sum each load's consumptions according to formulas) under the following conditions:

Case 1

Report preferences:

- Use each load coincidence factor

- Do not include coupled buses
- Selecting the upper level PDB (**PDB-3**), run the report.
- Report Values type = Consumed
- Compensated/Uncompensated = Compensated
- Verify results with ‘Batch load Associations’ with the ‘All feeders’ radio button selected.

Note that Bus-5 does not contribute to the totals.

Case 2

Report preferences:

- Use each load coincidence factor
- Include coupled buses connected by coupled buses only
- Selecting the upper level PDB (**PDB-3**), run the report.
- Report Values type = Consumed
- Compensated/Uncompensated = Compensated
- Verify results with ‘Batch load Associations’ with the ‘All feeders’ radio button selected.

Case 3

Report preferences:

- Use each load coincidence factor
- Include coupled buses connected by couplers and risers
- Selecting the lower level PDB (**PDB-4**), run the report.
- Report Values type = Consumed
- Compensated/Uncompensated = Compensated
- Verify results with ‘Batch load Associations’ for each of the buses.

Case 4

Report preferences:

- Use Bus PDB coincidence factor
- Include coupled buses connected by coupled buses only
- For Bus-3 and Bus-5 set X Coincidence factor =1
- For Bus-4 set:
 - X Coincidence factor =1 &
 - Y Coincidence factor =0.5
- Selecting the upper level PDB (PDB-3), run the report.
- Report Values type = Consumed
- Compensated/Uncompensated = Compensated

Case 5

Report preferences:

- Use each load coincidence factor
- Include coupled buses connected by coupled buses only
- Select Bus-5 and change the following properties as follows:
 - Drill Down Enable Flag = False
 - Consumed Active Load = 24 kW
 - Consumed Reactive Load = 6 kVAR
- Selecting the upper level PDB (PDB-3), run the report.
- Report Values type = Consumed
- Compensated/Uncompensated = Compensated
- Verify that you get same results with the ‘Batch Load Association’

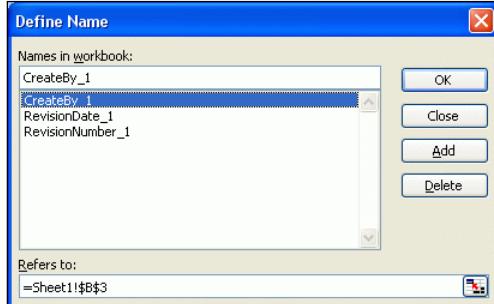
Exercise 12 - Revisions and Document macros in Registered Report

It's possible to show revision and document information on registered reports. This lab explains the technical aspect of adding such attributes to the report.

Create a new report (use 'Add to plant reports) for motors, selecting some properties. Map the selected properties on the report.

Define the Revision Macros

1. Select the cell for the revision macro and type-in the appropriate macro name in the **Name** box (see list of available macros below).
2. Use **Insert > Name > Define** to add new names/macros.



3. Save the Excel template.
4. Click **Actions > Register Report**.
5. Run the report.
6. In SmartPlant Electrical, right-click on the document, and on the shortcut menu, click **Document Properties**.
7. Add a new revision and click **OK**.

The software re-generates the report with the revision information.

The following revision macros 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 macro text. The lowest number that follows the underscore retrieves the latest revision. So, if, for

example, you want to include 4 revision levels in the document, you need to include the following macros:

- RevisionNumber_1
- ApproveBy_1
- RevisionDate_1
- RevisionNumber_2
- ApproveBy_2
- RevisionDate_2
- RevisionNumber_3
- ApproveBy_3
- RevisionDate_3
- RevisionNumber_4
- ApproveBy_4
- RevisionDate_4

Document Macros

You can also add macros for document information like document name, document description ...etc to the registered report.

The steps describe above for revision macros applicable for adding document macros.

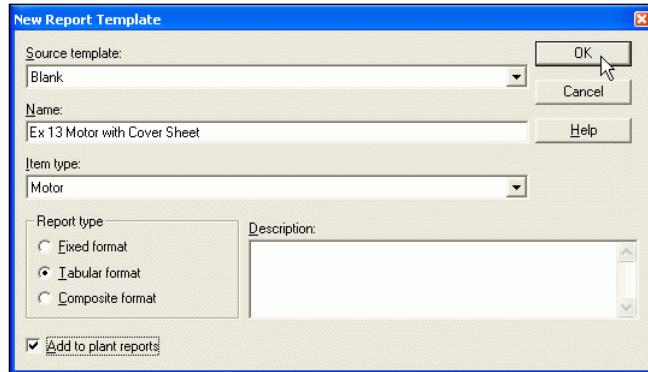
The syntax for the document macro is:

```
d_<property>
d_itemtag (document itemtag )
d_description (document description)
```

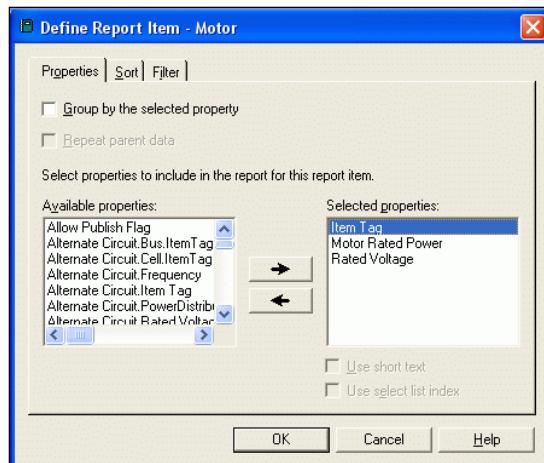
Exercise 13 – Creating Report Cover Sheet

A cover sheet for a standard report may sometimes be required in order to improve the design and the formal mode in which the report is being delivered. Report cover sheet can display some properties which are related to the report contents, general properties such as revision details or other plant related details.

1. Create a new plant tabular format report for motor item type, and name the report, ‘Ex 13 Motor with Cover Sheet’

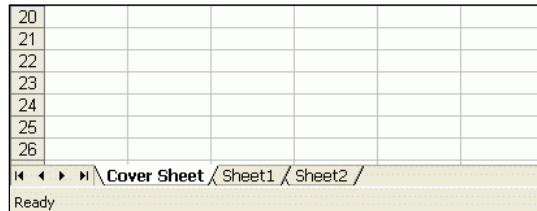


2. Define the following properties and map them as follows.

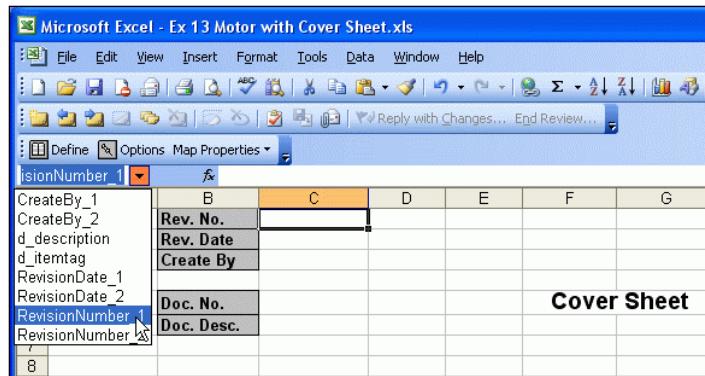


	A	B	C	D	E
1					
2					
3					
4					
5	Item Tag	Rated Power	Rated Voltage		
6	#Motor::Item	#Motor::Rated Power	#Motor::Rated Voltage	#	
7					
8					

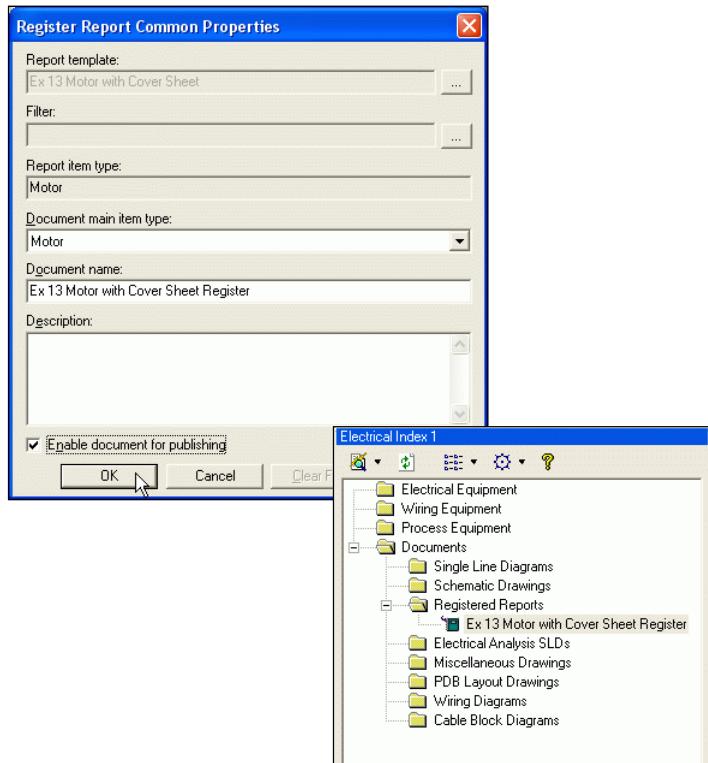
3. Add a new Worksheet to the Excel report: Insert→Worksheet menu, and rename the new worksheet to ‘Cover Sheet’. This is not a reserved name and the user can insert another name.



4. On the new sheet insert a title text box ‘Cover Sheet’.
5. You may insert revision and document macros.



6. Save the report.
7. Register the report.

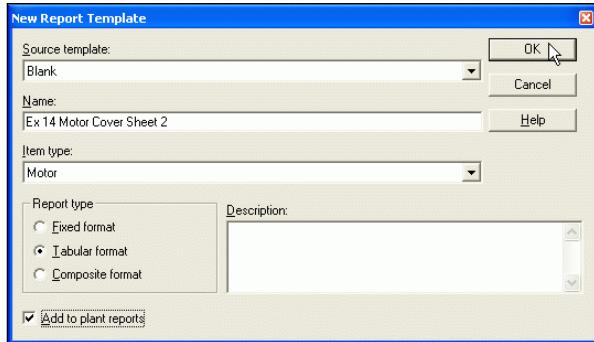


8. Run the registered report (document) and add revision to it.
 9. Check the report results.

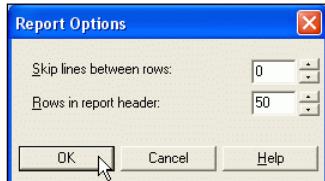
Exercise 14 – Creating Report Cover Sheet on the Same Worksheet

A second way to create ‘Cover Sheet’ for report can be done by using excel’s pagination option.

In the example shown below, we generate a “Data Sheet” type of report, using the tabular format.



In the report option select skip between lines = 0 and lines for the header = 50.

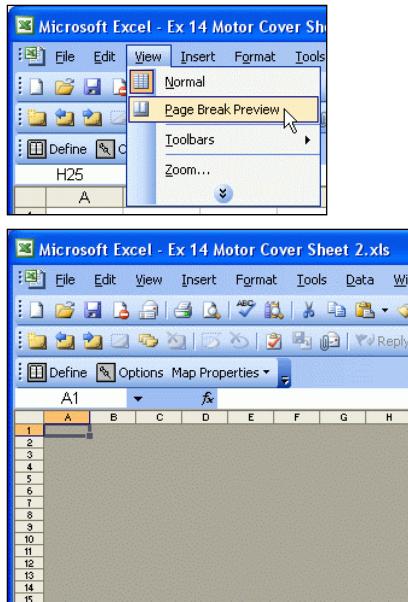


Select the following 3 properties for the report.

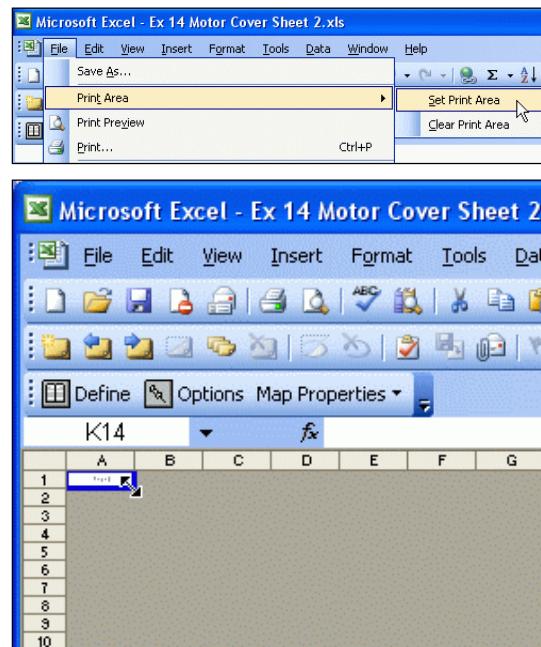


Close the define report windows and return to the excel.

To start the excel pagination, select from excel menu the option ‘Page Break Preview’.

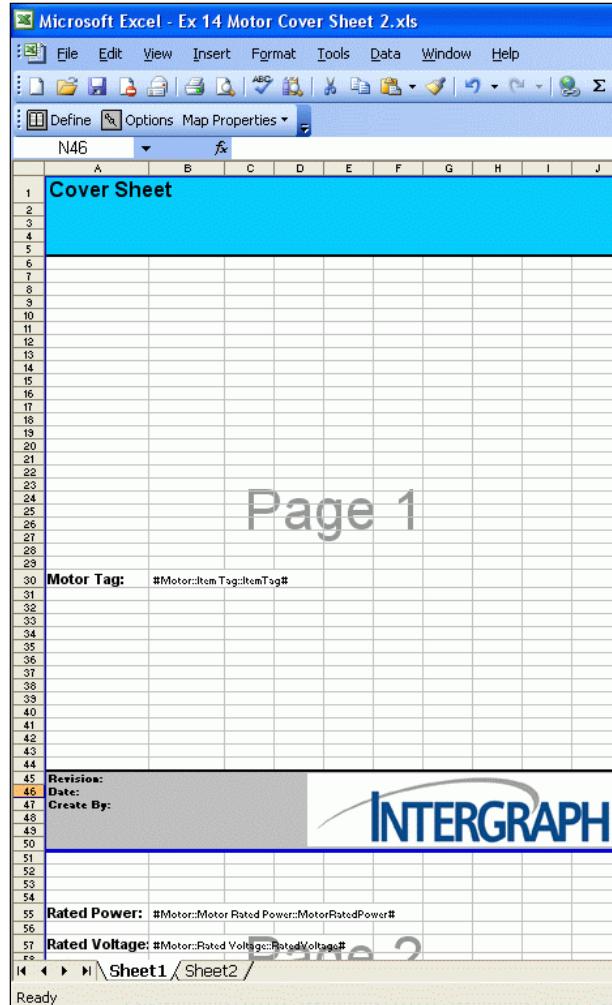


Select from file menu the option print area, and set print area.



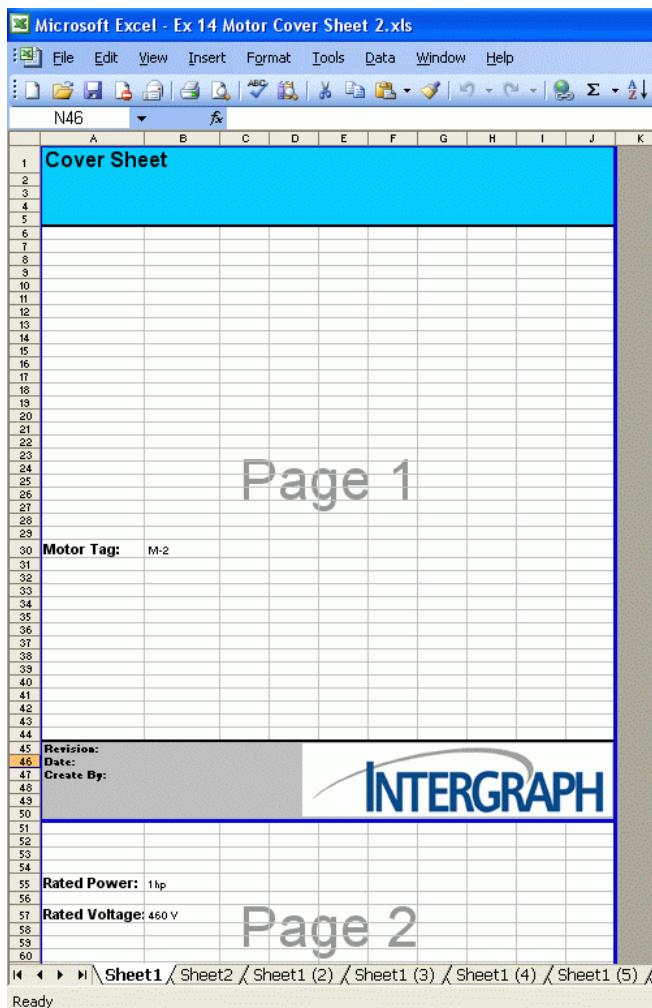
Resize the printing area, and set it to be 50 rows for the first page.

Add to the first page a cover sheet section, the header and the mapped **item tag** property. Also you can add title block section to the first page. To the second page add the other headers and their mapped properties.



Save the report.

Run the report and test the result.



Macro Definition Guidelines

You can generate schematics for the following item types:

- All loads
- All converting equipment
- All Disconnect equipment
- Circuits
- Buses
- Generators
- Battery banks
- Instruments

You place macros on each block to retrieve relevant information from the database.

Macros Defined in Catalog Manager

Macros are SmartText labels that you place in the drawing. These SmartText labels point to the property of a specific electrical item in SmartPlant Electrical and retrieve the relevant information from the database. Also, macros define the property labels and default values that will appear in the drawing if no data for the required property is found in the database.

Macros Defined in a CAD Application (SmartSketch, AutoCAD, or MicroStation)

In a CAD application, macros are text labels with a preceding ampersand (&). The macro text has to contain the SmartPlant Electrical item name and its required property name. Use the following syntax:

&ItemTypeName.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 a sequence number to ensure item uniqueness.

Macros for Panels

Use the following syntax to create macros for local panels or junction boxes that include circuits (use in cases where the circuit appears on the SLD)

&Circuit.Panel.Property.[Sequence]

Macros for Cables

Use the following syntax to create macros for cables:

Power Cables

&Cable.Property.[Sequence]

Control Cables

&ControlCable.Property.[Sequence]

Grounding Cables

&GroundingCable.Property.[Sequence]

Instrumentation Cables

&InstrumentationCable.Property.[Sequence]

Drawing Blocks and 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 Station

- Direct control station properties.
- Related control station cable properties. If data for more than one cable has to be retrieved, the macro name ends with a sequence number, for example '&ControlCable.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.

Title Block Macro List (Templates)

You specify macros in your title block by adding SmartText and defining the appropriate macro in the user properties. The following list summarizes the macros available for use in SmartPlant Electrical drawing title blocks.

Plant Hierarchy 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>.x, where x 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 has no sequence
&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 Properties Macros

Document property macros use the syntax Document.<property>. The following macros are available for specifying document properties.

Macro	Notes
&Document.Description	Document description
&Document.ItemTag	Document drawing number as defined in the Data Dictionary
&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.DocumentStatus	Document status as defined in the Properties window for a document
&Document.YYY	User-defined field name where YYY is any UDF defined in the Data Dictionary

Document Revision Macros

Document revision macros use the syntax Revision.<property>.x, where x is a number that specifies the sequence of the revision. This way, the title block can display the revisions in sequence. 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.ApproveBy.x	Revision approved by
&Revision.x.ZZZ	User-defined field name where ZZZ is any UDF 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	Description
<code>&Sheet.quantity</code>	Defines the total number of sheets in a multi-sheet schematic
<code>&Sheet.Number</code>	Defines the sequence number of a particular sheet
<code>&Sheet.Description</code>	Sheet description

Schematic Drawings

This section provides several examples on how the user should create the schematic drawing based on the information it contains. The key point of understanding how to create the schematic blocks is a good understanding of how SmartPlant Electrical searches the data when it solves the schematic drawing.

There are several mechanisms by which the engine searches for data:

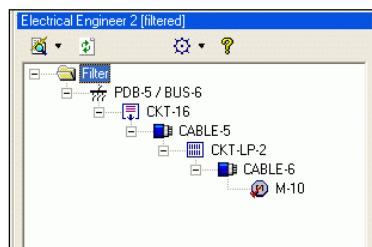
Superposition of blocks – These are associated with items (motors, circuits). The first degree item can be retrieved (if it's a motor, then cables can be found, if the block belongs to a circuit, it can retrieve the PDB/Bus + circuit internals).

Electrical relationships – With this method you can get most of the data related to the main item by introducing the right macros.

Exercise 1 – Creating a Schematic for a Motor

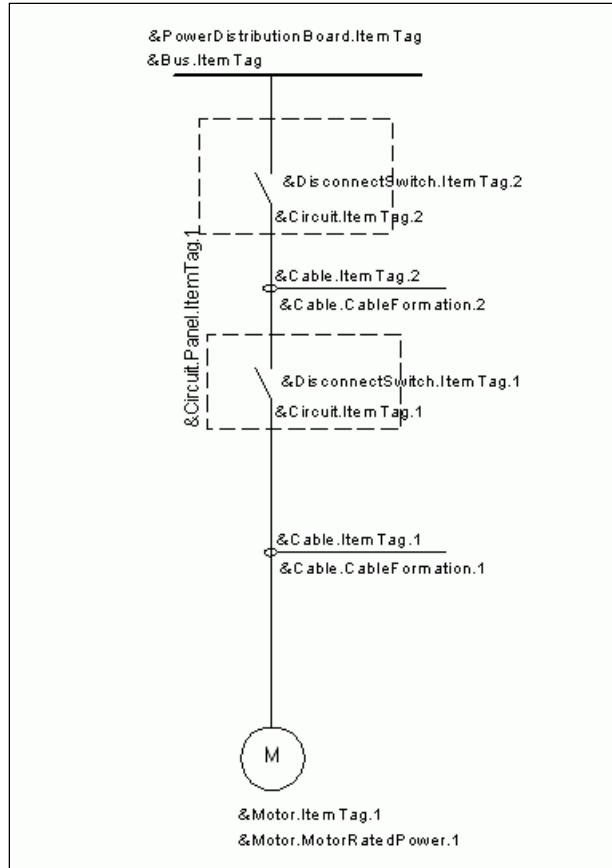
The first example in this lab involves a motor schematics that has two power cables and a disconnect switch in a local panel. Later you will add an instrument and a control cable and test the schematics with corresponding blocks.

1. To start with this lab, you should create the following data by manually adding items to the Index and making the right associations. Names can vary depending on your existing data.



Tips

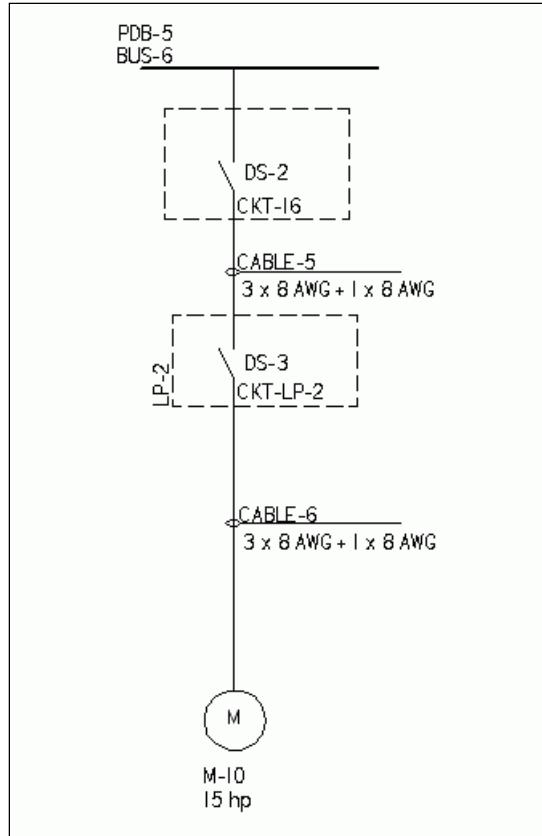
- CKT-LP-2 is a circuit inside a local panel (LP-2).
 - Both circuits (CKT-16 & CKT-LP-2) have a disconnect switch (DS-2 and DS-3).
2. In Reference Data Explorer (RDE), add new typical block, name it '**Typical1 Motor**',
Block Type = '**Motor**',
File name = '**Typical1_motor.sym**'.
 3. View and save the typical block.



4. In RDE, add a new typical schematic, name it '**Typical1 Motor**'. Add to it the '**A Wide.spe**' template,
and the typical block, '**Typical1 Motor**'.
5. Select the motor, '**M-10**' and generate schematic drawing.

 **Tip**

You may use apply option to apply the typical schematic 'Typical1 Motor' to motor M-10 prior generate the Schematic.

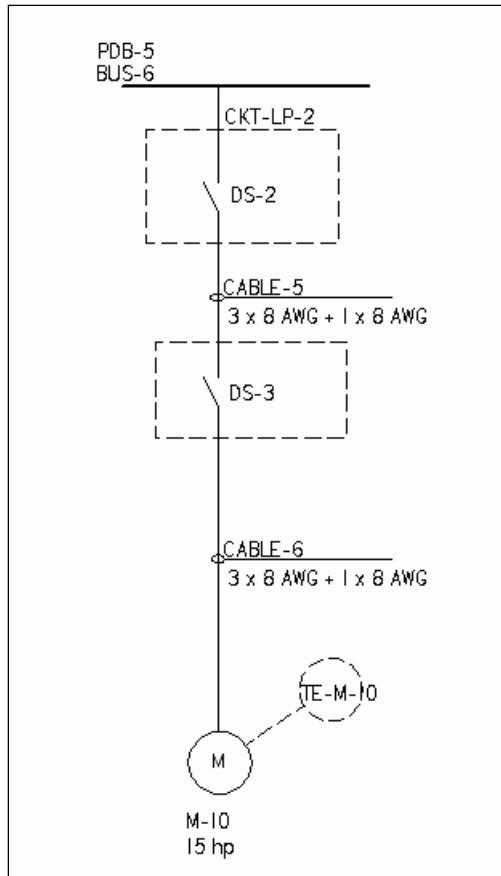


6. In Electrical Index (EI), add an instrument, name it '**TE-M-10**', and associate it with motor, '**M-10**'.

Tip

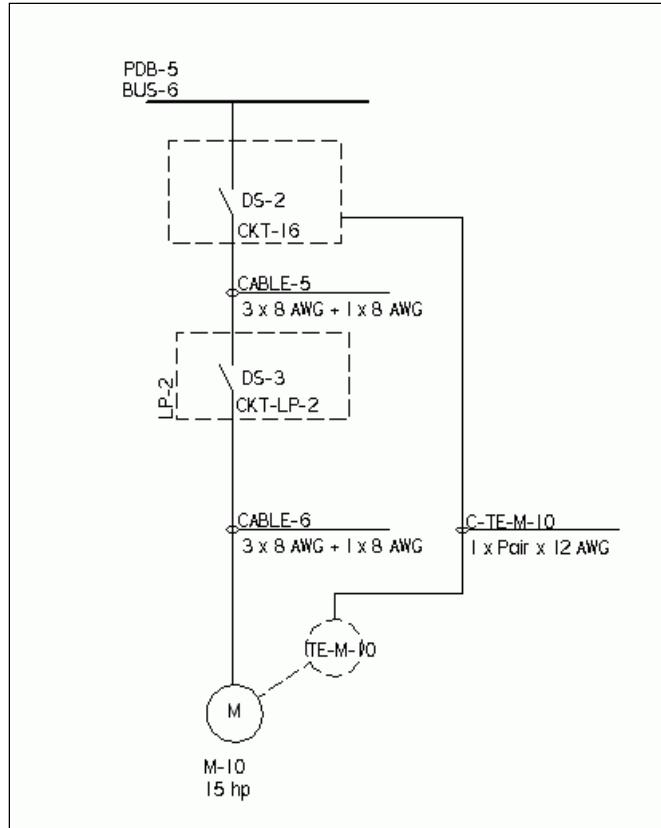
We want to show this instrument next to the motor.

7. In RDE, add a new typical block, name it '**Typical2 Motor with Instrument**', Block Type = '**Motor**', File name = '**Typical2_Motor_with_Instrument.sym**'.
8. In RDE, add a new typical schematic, name it '**Typical2_Motor_with_Instrument**'. Add to it the template '**A Wide.spe**' template, and the typical block, '**Typical2 Motor with Instrument**'.
9. Generate the schematic again using the new typical schematic. You should see the instrument (all the rest will remain intact).

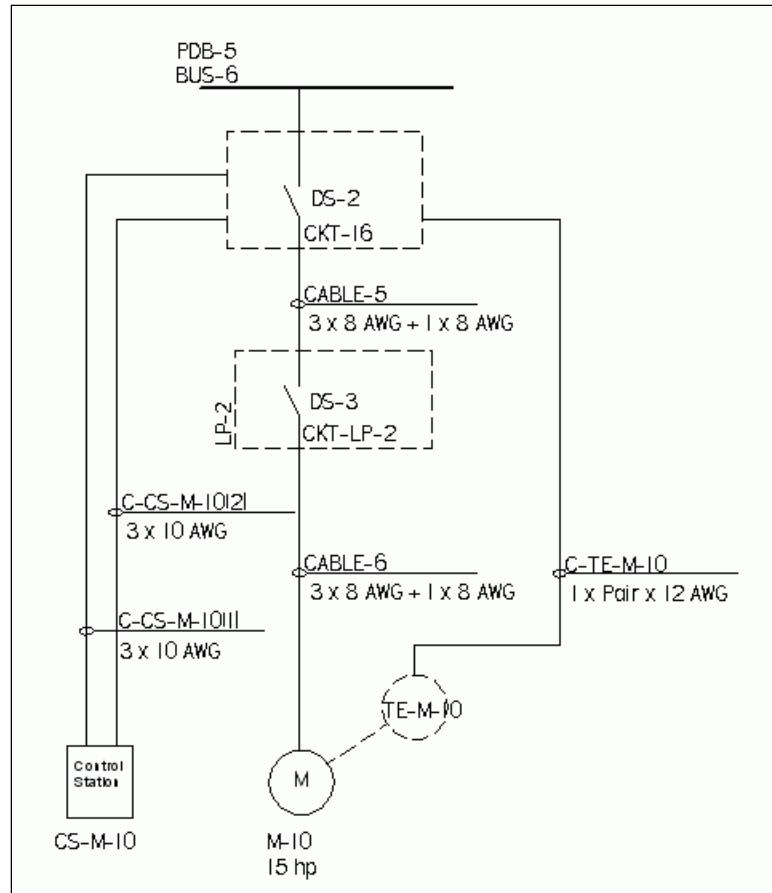


Suppose we have an instrumentation cable going from this instrument to the PDB circuit and we want to show it on the drawing.

10. In RDE, add new typical instrumentation cable, name it '**1 Pair 12 AWG**'.
 11. In EI, add new project Instrumentation cable, name it '**C-TE-M-10**', using the typical cable you created in previous step.
 12. Associate the cable to the instrument.
 13. In RDE, add a new typical block, name it '**Typical1 Instrument**',
Block type = '**Instrument**',
File name = '**Typical1_Instrument.sym**'.
 14. Add the new typical block, to the typical schematic, '**Typical1 Motor**'.
 15. Generate the schematic again, using the typical schematic, '**Typical1 Motor**'.
- You should see the instrument's cable.



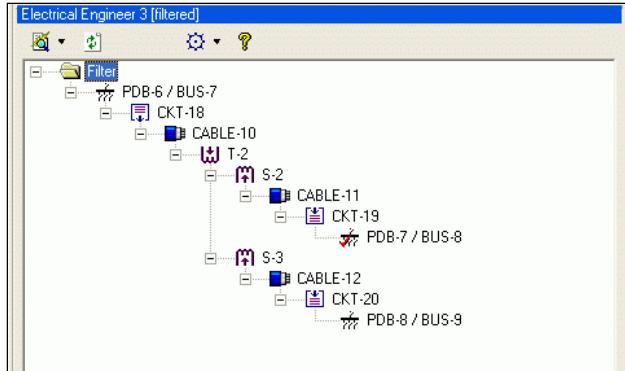
16. In EI, add a new control station, name it '**CS-M-10**', and associate it to the motor, '**M-10**'.
17. In EI, add two project control cables, name them, '**C-CS-M-10(1)**', and '**C-CS-M-10(2)**', using the typical control cable, '**3/C- 10 AWG - (Control)**', and associate them to the Control station.
Next step adds the control station block with the cables. You should define the 'sequence in group' for each of the cables.
18. In RDE, add new typical block, '**Typical1 ControlStation**',
Block Type = '**Control Station**',
File name = '**Typical1_ControlStation.sym**'.
19. Add the new typical block, to the typical schematic, '**Typical1 Motor**'.
20. Generate the schematic again. You should see the control station and cables.



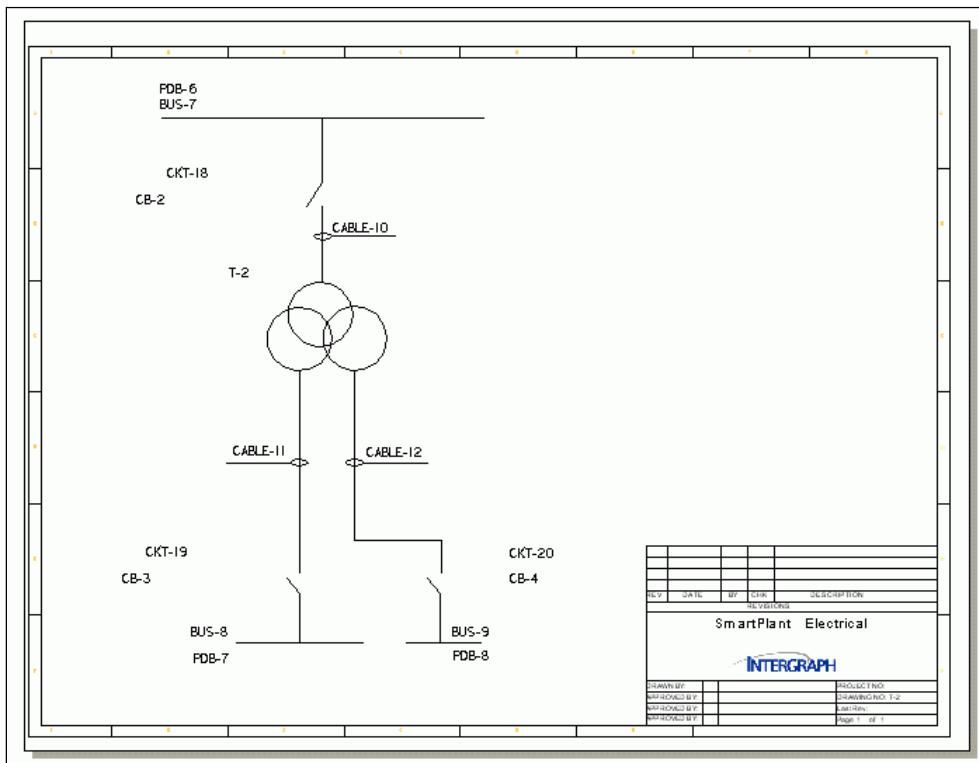
Exercise 2 – Creating a Schematic for a 3-Winding Transformer

The second example involves a 3-winding transformer as the main item. The schematic drawing demonstrates how to get the information on the transformer, up to the feeder that feeds it and the incomers that it feeds.

1. Create the following items:
 - One power distribution board, one feeder circuit with a circuit breaker.
 - Two power distribution boards with an incomer circuit, add a circuit breaker in each.
 - Three power cables.
 - A transformer with three windings.
2. Connect the items you have added according to the following screen:



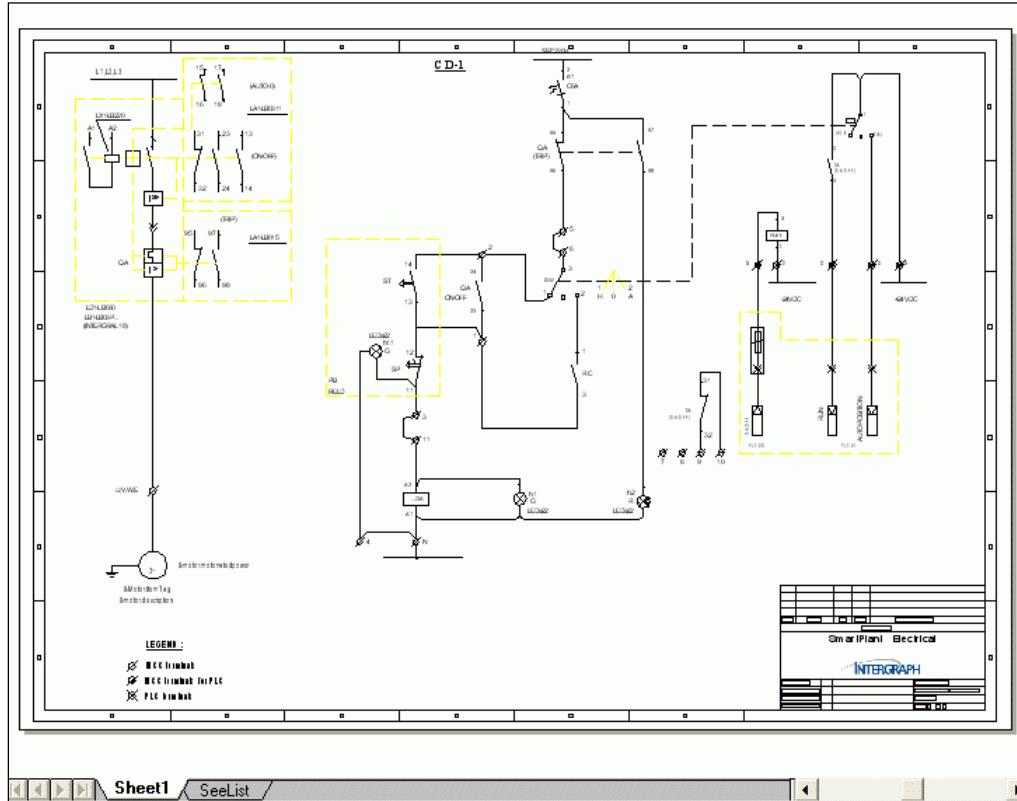
3. In the RDE, add a new typical block, name it '**Transformer Complete**'.
Block Type = '**Transformer**',
File name = '**Transformercomplete.sym**'.
4. In the RDE, add a new typical schematic, name it '**Transformer Complete**'.
Add to it the '**A Wide.spe**' template, and the typical block, **Transformer Complete**'.
5. View the typical schematic.
6. In EI, select the transformer and generate the schematics drawing.
7. Verify extraction of the data.

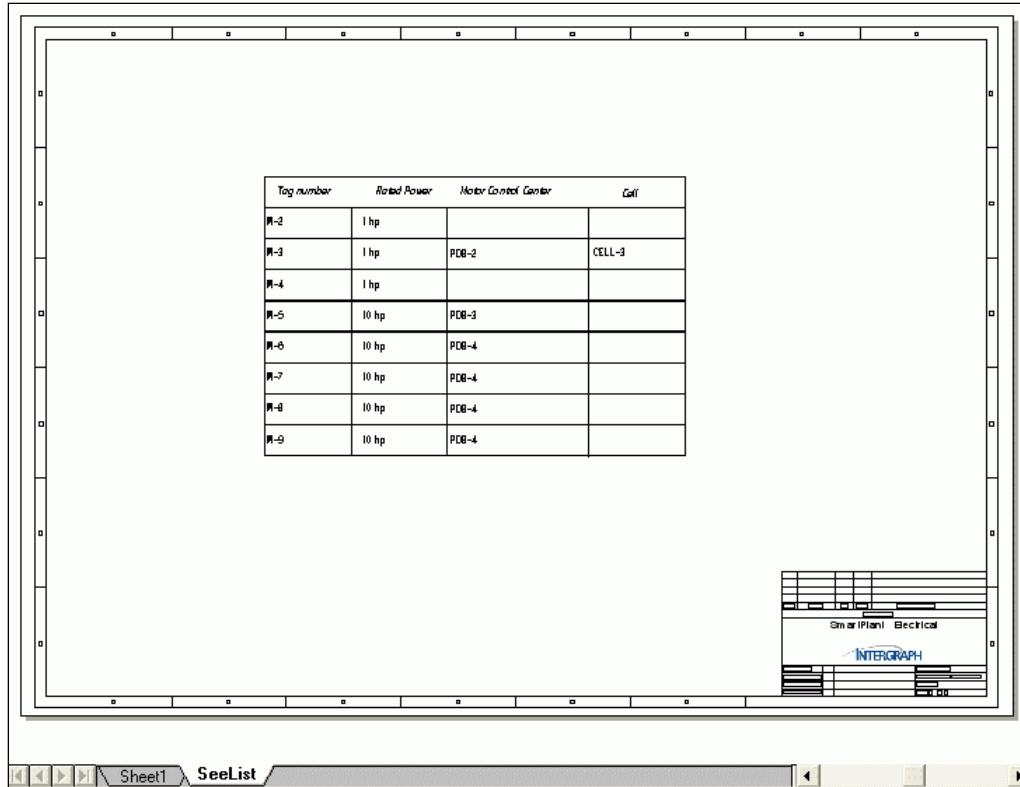


Exercise 3 – Creating a Multi Tag Motor Schematic

The purpose of this exercise is to understand how to create a multi tag schematics.

The first page of the motor schematic diagram that we create contains the graphical portion. The consecutive sheets contain table with data for the motors.





Note: It's possible to show the table also in the first page.

The following 8 steps demonstrate how to create and run the multi item schematic

Step 1: Create Multi-Tag Symbols

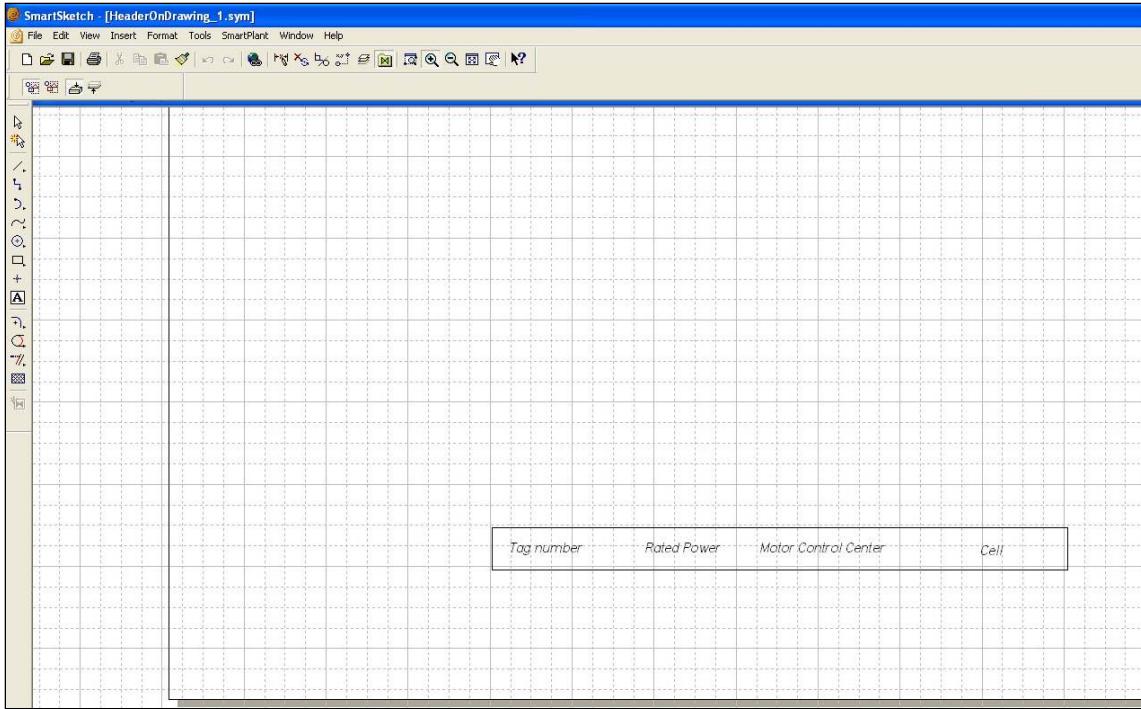
Creating 3 new symbols, using Smart Sketch or Catalog Manager.

Note: There is no need at this exercise to create the 3 symbols, as they should be available already for you in the electrical reference data files.

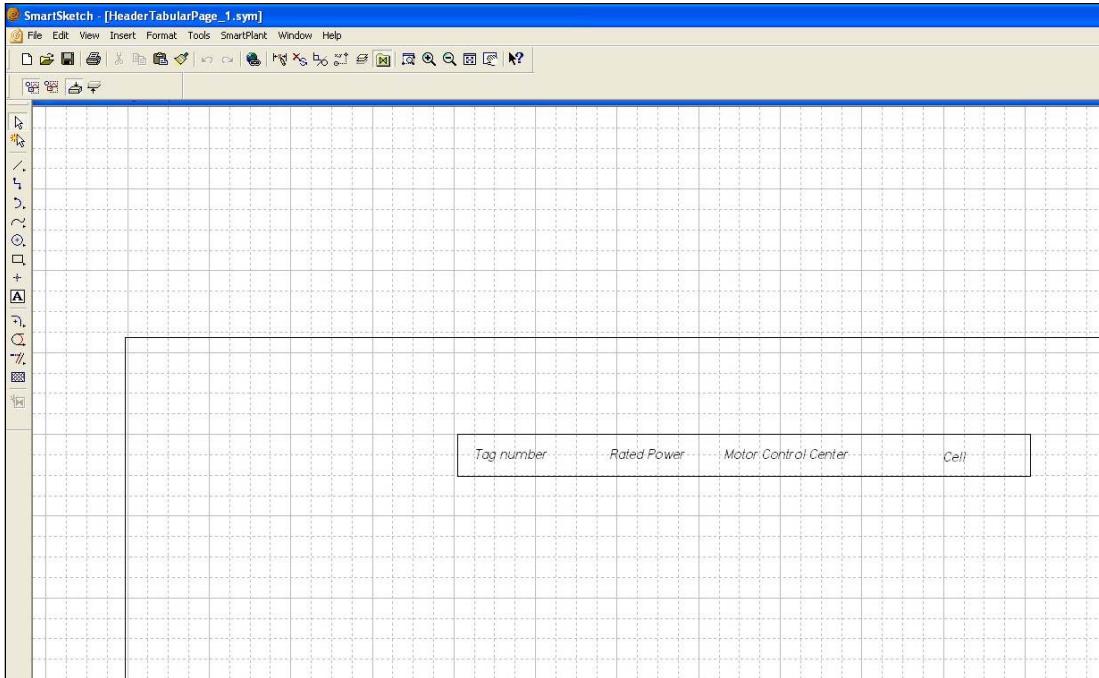
1. The first symbol is for the header of the first sheet.
2. The second symbol is for the header on consecutive sheets.
3. The third symbol is for the tabular data that contains the grid and the macros.

1. First Sheet Header Symbol

Schematic Drawings

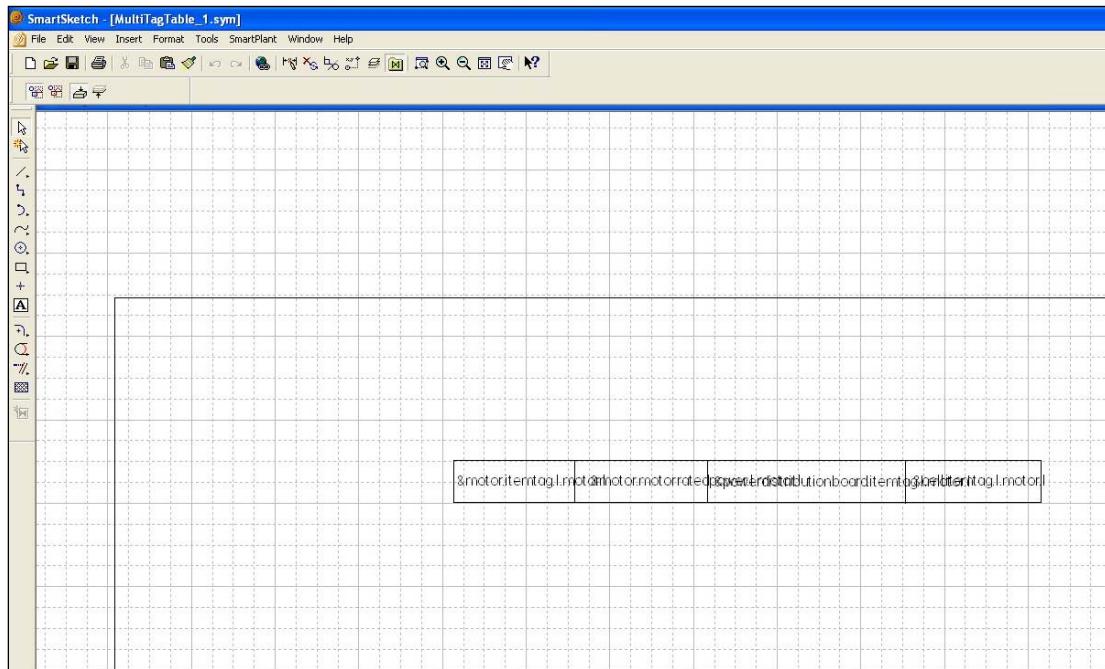


2. Consecutive Sheet Header Symbol



3. Tabular Data Symbol

The tabular data symbol will contain the grid and the macros.



This symbol will be positioned so that its origin will automatically be adjusted by the system.

Since the macros are not currently associated with a certain block type, the syntax of the macro is extended to express also the block type that we are referencing for the item type, as follows.

<ItemType>.<ItemAttribute>.<sequence number>.<BlockType>.<block sequence>

For example:

- &Cable.ActualLength.1.Motor.1
- &Cable.ActualLength.2.Motor.1
- &ControlStation.ItemTag.1.ControlStation.1
- &ControlStation.ItemTag.1.ControlStation.2

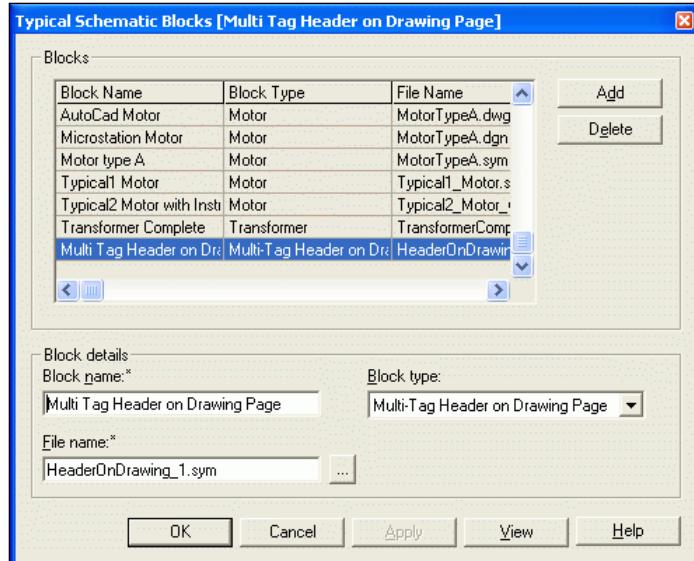
Important

- You must include the referenced <Block type> as one of the typical blocks; otherwise you will not be able to retrieve data for that item type.

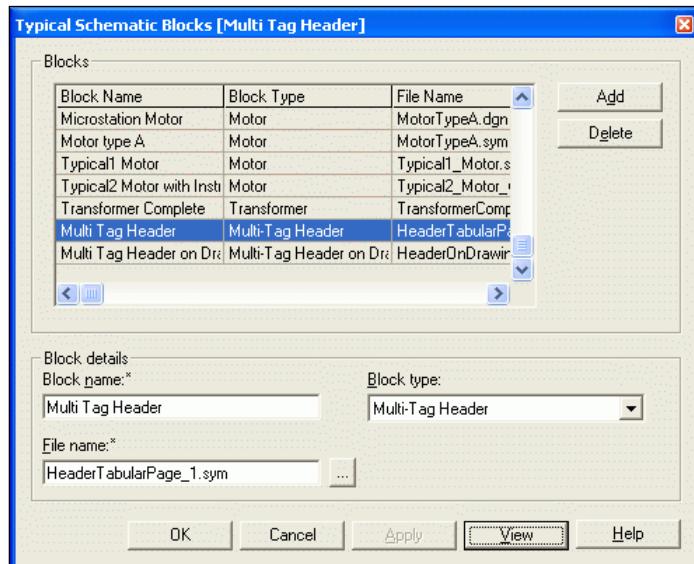
Step 2: Create Typical Schematic Blocks

In this step, we are going to create 3 new typical blocks and associate them with the 3 new symbols.

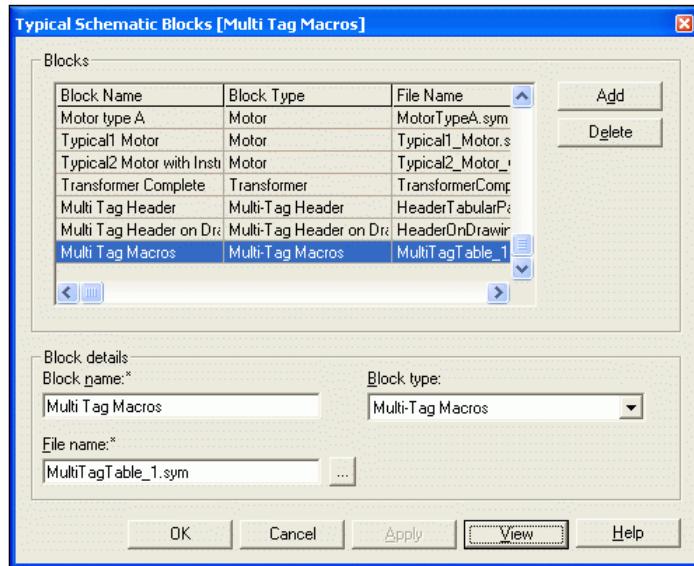
1. Associate the first block, '**Multi Tag Header on Drawing Page**', with the first sheet header symbol.



2. Associate the second block, '**Multi Tag Header**', with the consecutive sheet header symbol.

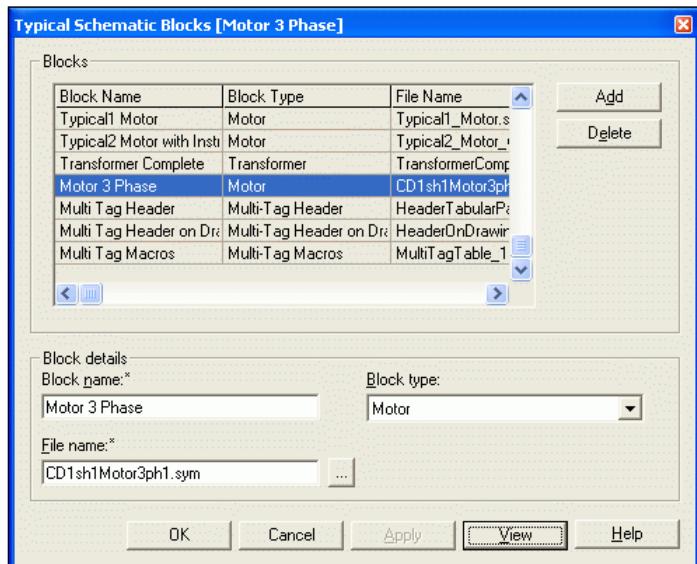


3. Associate the third block, '**Multi Tag Macros**', with the tabular data symbol.



Step 3: Create the motor Typical block

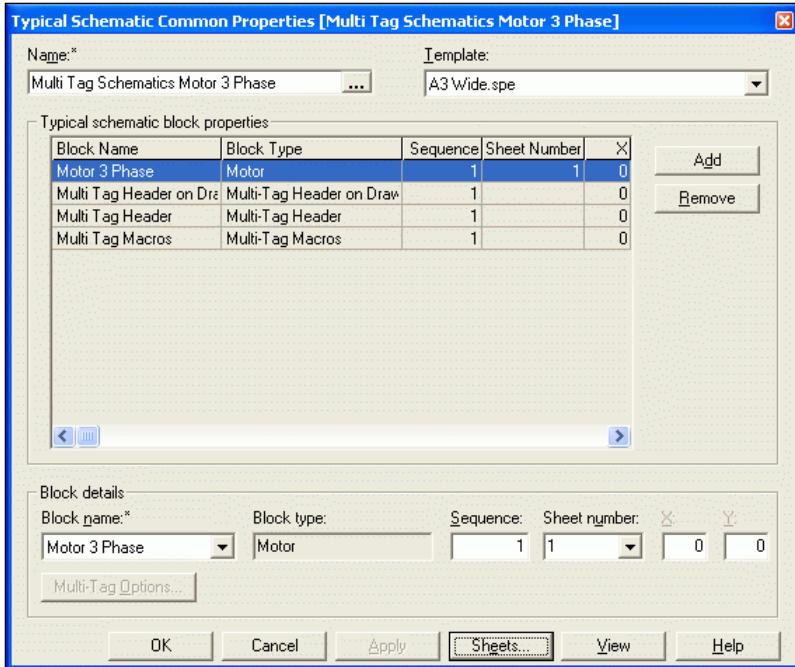
Create new typical block, '**Motor 3 Phase**'.



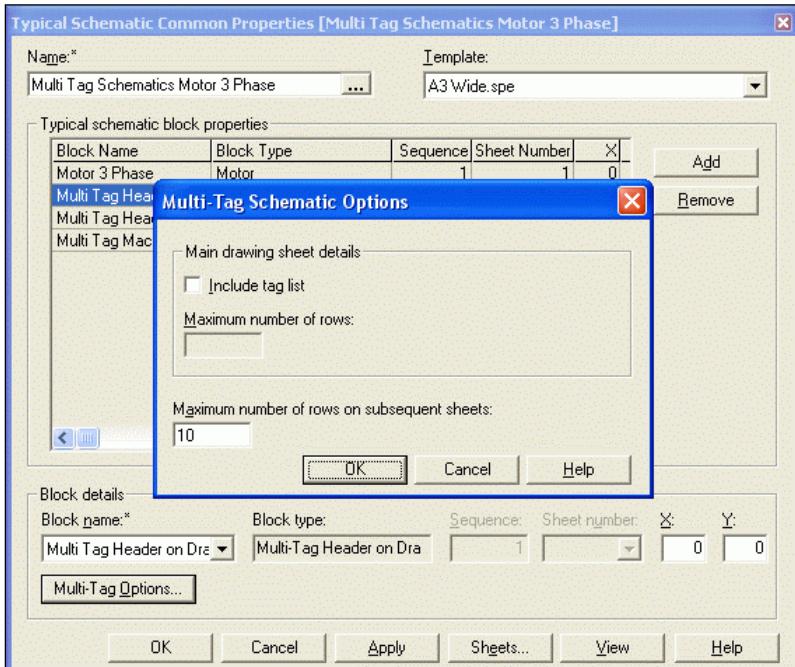
Step 4: Create a Multi Tag Typical Schematic

Create new typical schematic, '**Multi Tag Schematics Motor 3 Phase**'.

Schematic Drawings

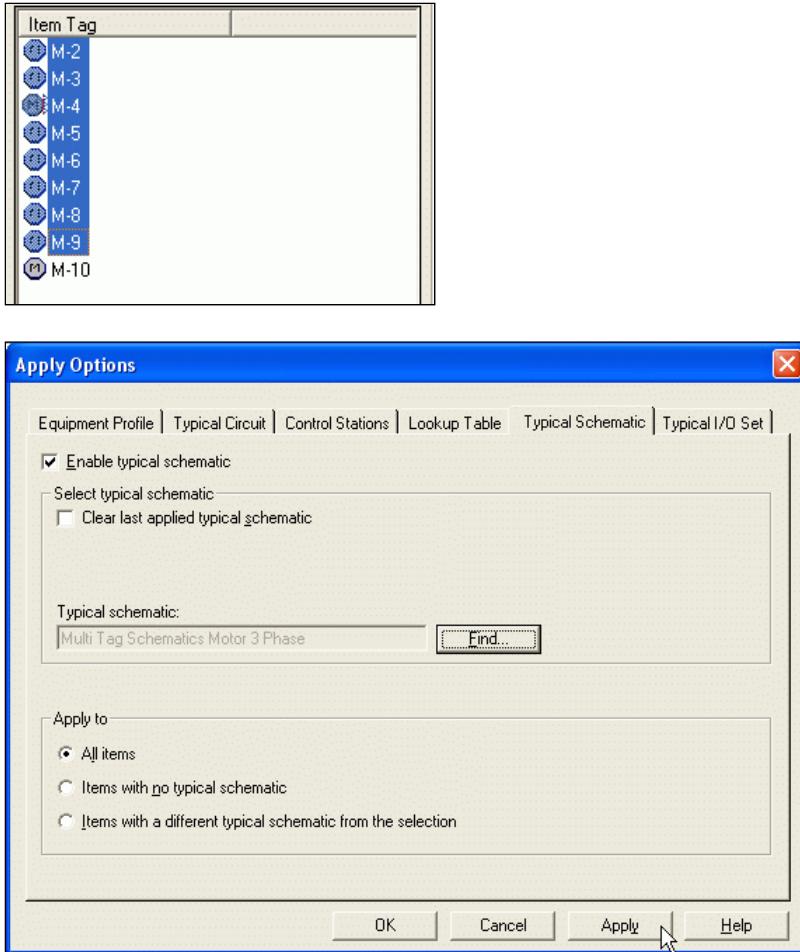


Select one of the Multi-Tags blocks to set its definitions.



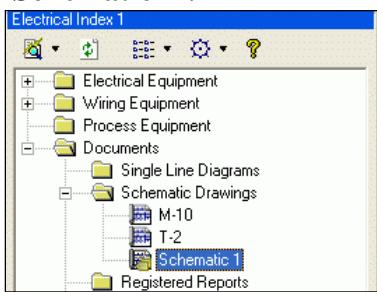
Step 5: Apply Typical Schematics on Project Motors

From electrical index select a set of motors that you want to collect on the same drawing and apply the typical schematics.



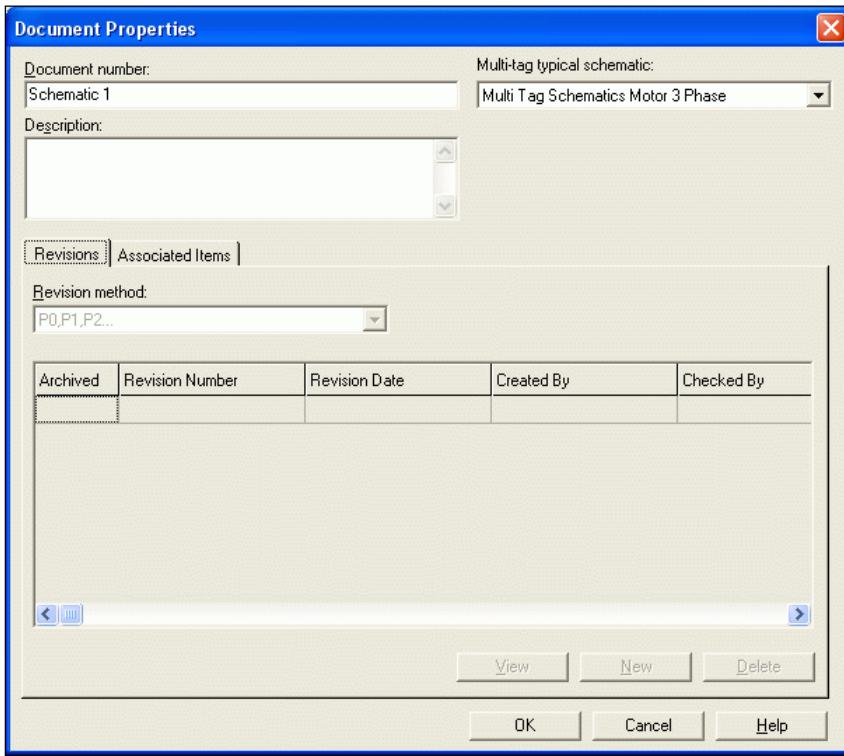
Step 6: Create a Multi Tag Schematic Drawing

In Electrical Index, Document, Schematic Drawing folder, creates a new multi-tag drawing, '**Schematic 1**'.



Step 7: Associate schematic drawing with multi-tag typical schematic

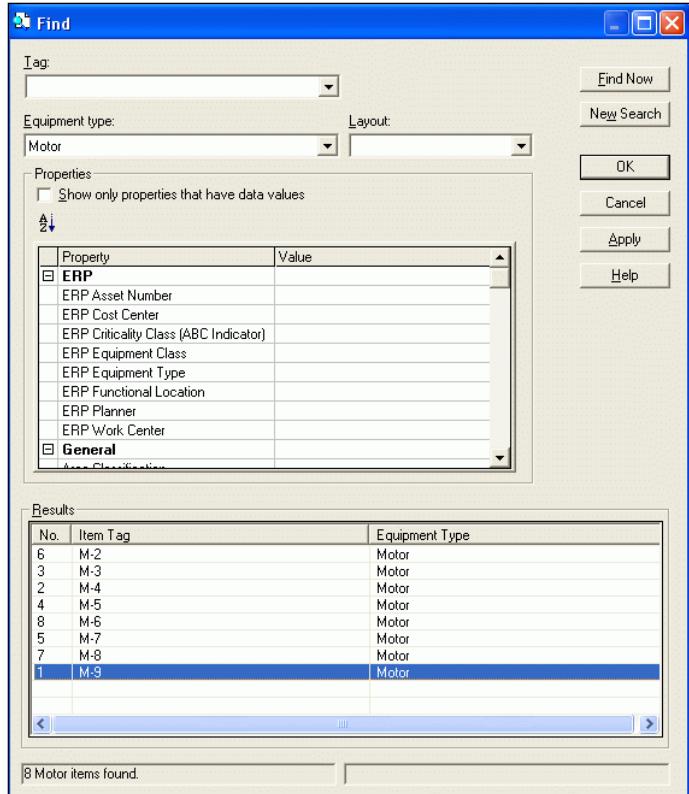
1. Open the Document Properties windows of the new document, '**Schematic 1**', and select the typical schematic, '**Multi Tag Schematics Motor 3 phase**', from the '**Multi-tag typical schematic:**' drop down list.



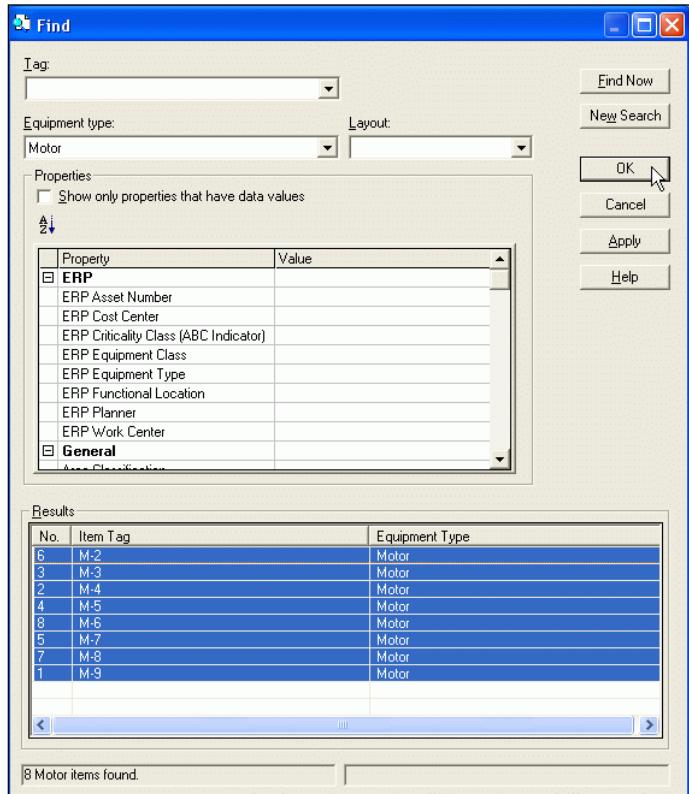
2. Click the 'Associated Items' tab.
3. In the item type select motor.
4. Select the 'associate' button.
5. In the find windows select find now, select the available motors (motor M-2 to M-9) and select OK.

Note: Since you selected in step 5 only motors M-2 till M-9. Only those 8 motors available for selection.

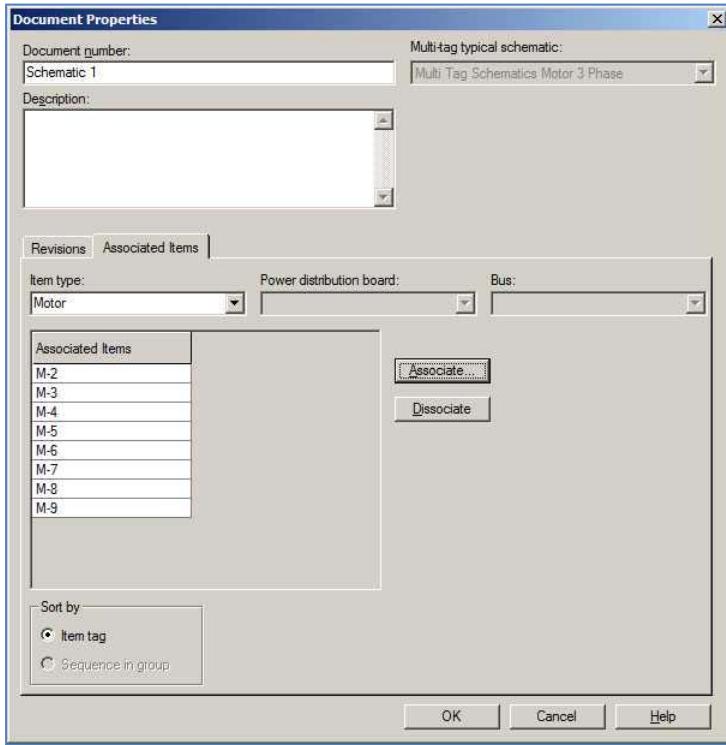
Schematic Drawings



Select all of the 8 motors and click OK.

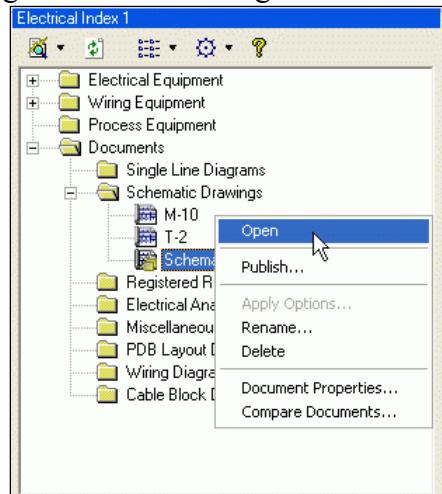


Click OK to close the document properties.

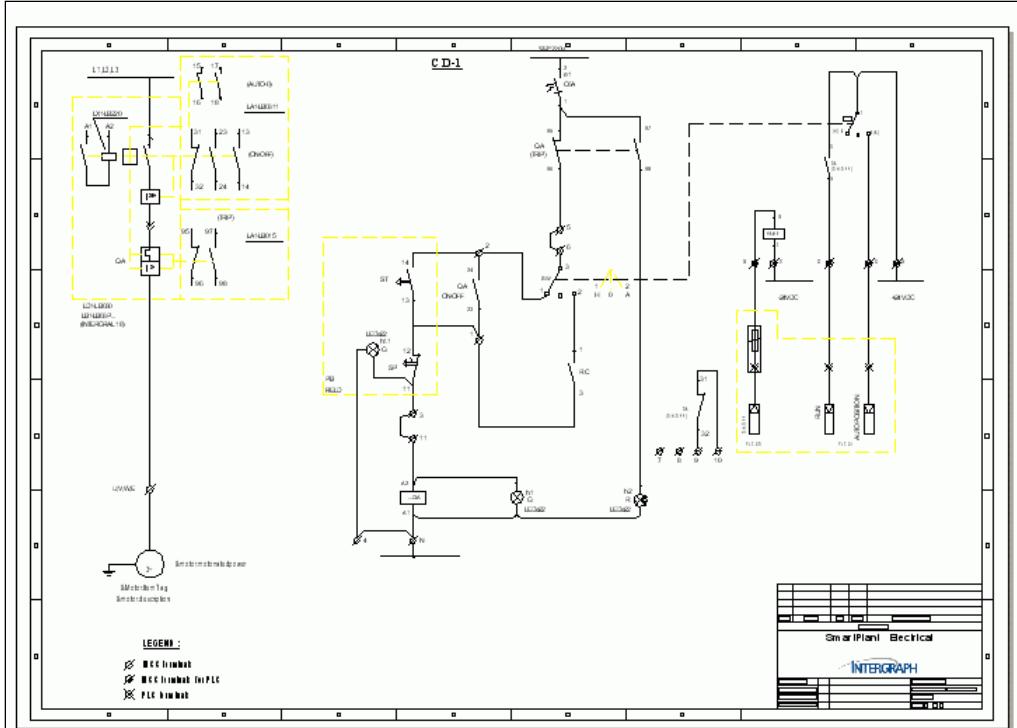


Step 8: Generate the multi-tag schematics

Either select the schematic document, 'Schematic 1', or select a single motor from the list to generate the drawing.



Schematic Drawings



Sheet1 SeeList

Tag number	Rated Power	Motor Control Center	Cell
M-2	1 hp		
M-3	1 hp	PDB-2	CELL-3
M-4	1 hp		
M-5	10 hp	PDB-3	
M-6	10 hp	PDB-4	
M-7	10 hp	PDB-4	
M-8	10 hp	PDB-4	
M-9	10 hp	PDB-4	

SmartPlant Electrical
INTERGRAPH

Sheet1 SeeList

Lab – Creating a New Template from Scratch

In this lab you will create from Catalog Manager, your own title block symbol file and use it to create a new template file.

1. Open Catalog Manager.
2. Connect to database and plant if it's not already done.
(**File > Open Database** and select your plant. this may take few seconds).
3. In the **Catalog Explorer** window select '**Schematic Blocks**' folder.
4. Right-click in an empty region of the lower pane.
5. On the shortcut menu, click **New Item**.
6. Select the **New item**.
7. Right-click the item, and on the shortcut menu, click **Open**.
8. To set the sheet size, click **File > Sheet Setup**.
9. Select the size that you want to use, for example, 'A Wide(11in x 8.5in)', and click **OK**.
10. Click **Fit** .

At this point, you are ready to draw the graphics, border, and title block along with its macros. You can also paste graphics directly from ready-made AutoCAD or MicroStation drawings. Such an operation may require you to set scale options in the **Tools > Options** dialog box.

For this exercise you will draw everything in the Catalog manager.

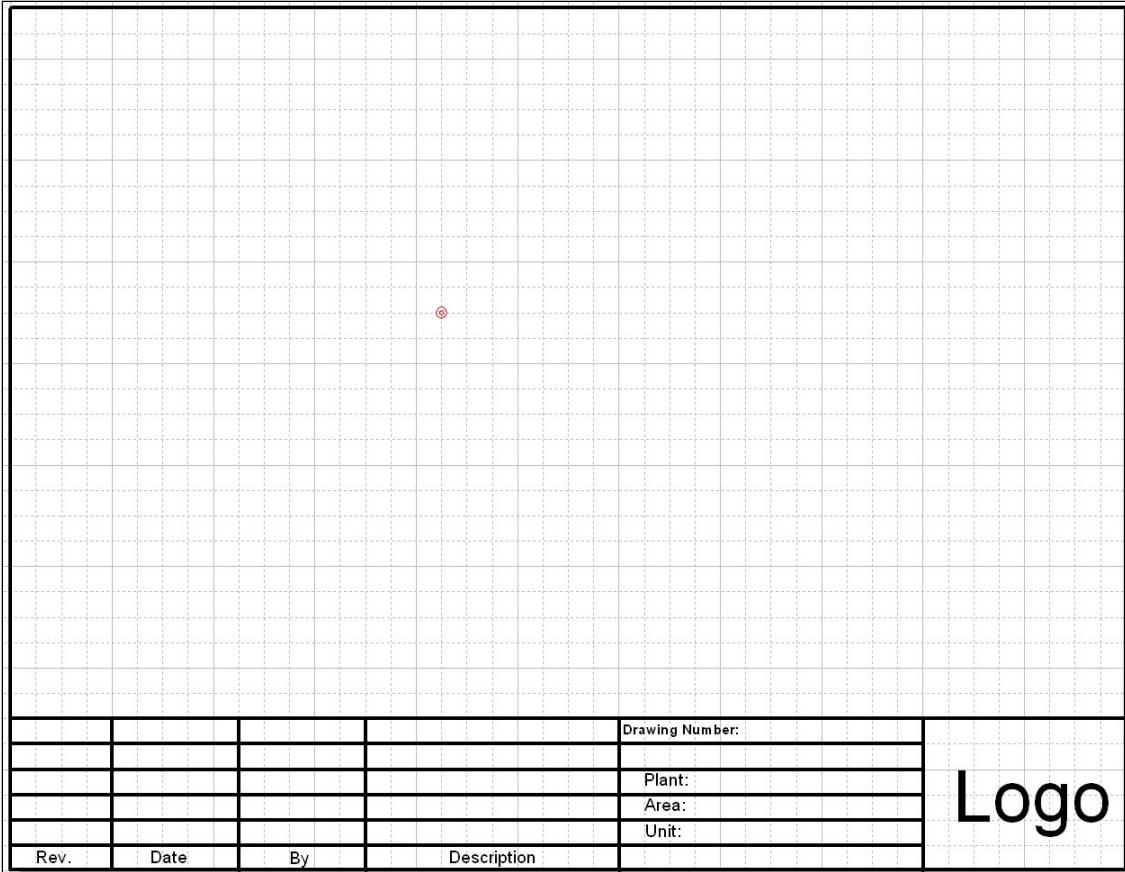
Here are some useful icons that will help you to get started. You can explore the Online Help to get more information for this and other toolbars.

Draw Toolbar

Provides one-click access to several common features of Catalog Manager. This toolbar allows you draw symbols and elements of symbols.

- ↖ [Select Tool](#)
- ↙ [Line Command](#)
- ↗ [Tangent Arc Command](#)
- ↔ [Curve Command](#)
- ⊕ [Circle by Center Point Command](#)
- [Rectangle Command](#)
- + [Point Command](#)
- Ⓐ [Text Box Command](#)
- ⌞ [Fillet Command](#)
- ⌞ [Trim Command](#)
- ⌞⌞ [Extend to Next Command](#)
- .FILL [Fill Command](#)
- ✳ [Create Symbol Command](#)

11. Draw the following title block.



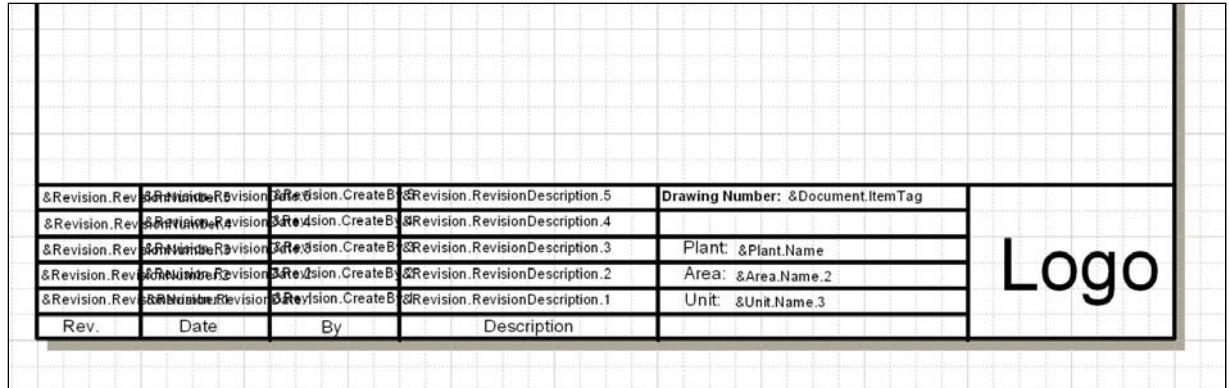
 **Notes**

When working in the view window (drawing area), you can switch off the grid display and snap from the **View** menu.

The next step is placing some macros. Title block macros are simple text labels in specific format preceded by ampersand ('&') character. The following lists some useful macros. The suffix 'x' should be replaced by the corresponding record number you want the data for where lower numbers correspond to the earliest revisions.

Macro	Description
&Revision.RevisionNumber.x	Revision number
&Revision.RevisionDate.x	Revision date
&Revision.CreateBy.x	Created by
&Revision.RevisionDescription.x	Revision description
&Document.ItemTag	Document/drawing number
&Document.Description	Document/drawing description
&Plant.Name	Plant name
&Area.Name.2	Area name
&Unit.Name.3	Unit name

After adding the macros, the title block appears as shown.



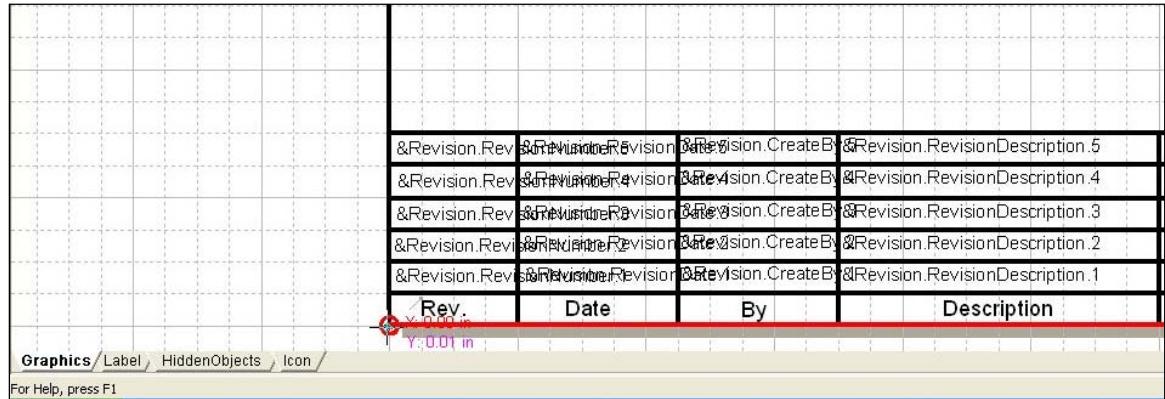
This concludes the drafting and designing of the title block symbol.

To save the symbol and have it referenced to the lower-left corner of the sheet, follow these steps:

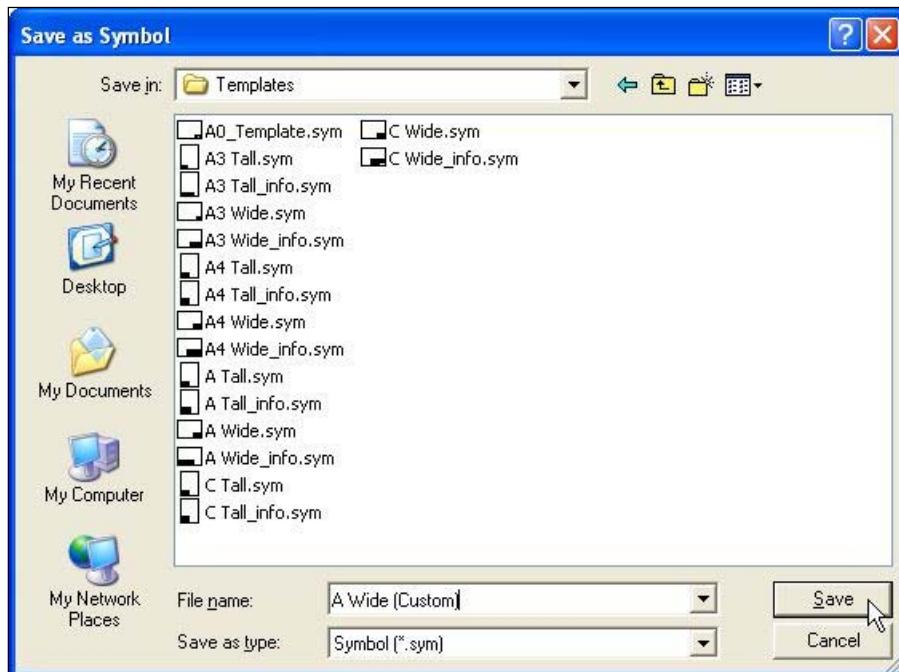
1. To be able to view the exact coordinates, on the **View** menu, turn off the **Grid Snap**.
2. Activate the ‘PinPoint’ toolbar (click **Tools > PinPoint**).
3. Press **Ctrl+A** to select all the objects on the drawing
4. Click the **Create Symbol** icon.
5. Click **Define PinPoint Origin** to display the ‘Define PinPoint Origin’ toolbar.



6. Type zeros in both the **X** and **Y** boxes. This positions the origin point in the bottom left corner of the sheet
7. Using the mouse, click the origin point on the lower-left corner ([0,0] coordinates).



8. In the **Save As Symbol** dialog box, in the **File Name** box type **A wide (custom)**.
9. Navigate to the **Templates** folder under the shared reference data folder and click **Save**.



10. To create the new Template file in SmartPlant Electrical, do the following:

1. Open SmartPlant Electrical.
2. Click **File > New > Template** to create new template document.
3. While the new template document is open, click **File > Sheet Setup** and set the sheet size to be '**A wide(11in x 8.5 in)**', and click **OK**.
4. Click **Edit > Insert > Symbol**, and select browse to the Templates folder

(shared), to select the ‘**A Wide (Custom).sym**’ symbol/title block, and click **Open**.

5. Click in the open document area to insert the symbol/title block into the template area.
6. Close the template, by click on the template document [X] button. In the save as dialog type ‘**A Wide (custom)**’, and select save.

At this point, the new template is ready for use.

Lab – Schematics: Working with Blocks

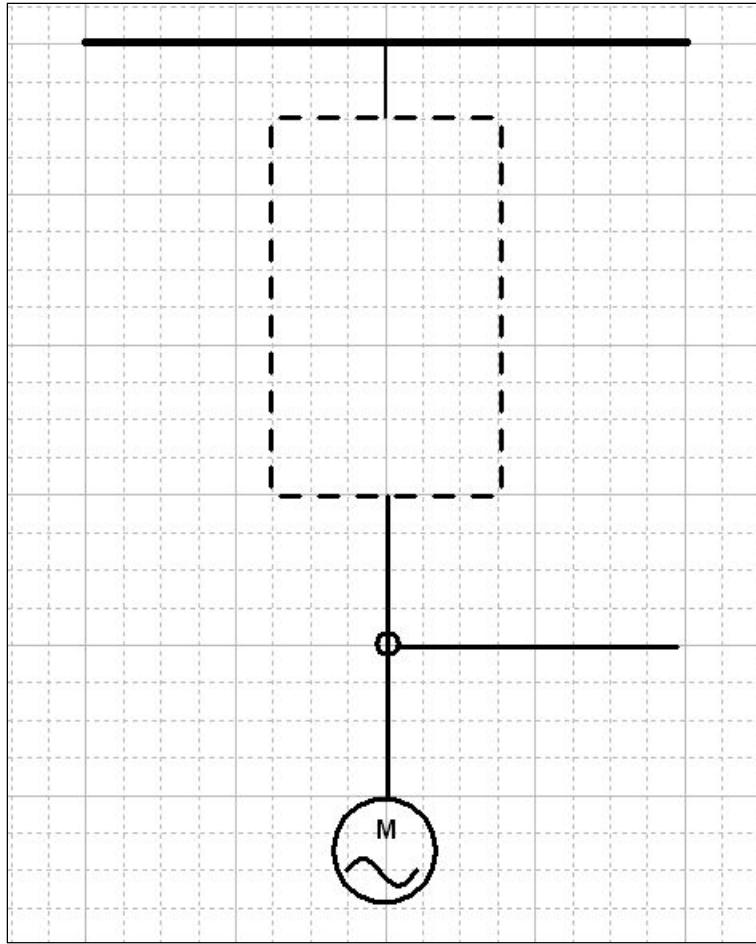
This lab demonstrates the technical aspect of creating a new schematic drawing and dividing it to symbols. It's possible to insert or paste the graphics from AutoCAD or MicroStation files.

This exercise only uses Catalog Manager.

In order to start your work, use the template symbol created in the template lab (earlier) or one of the existing template symbols that are shipped with SmartPlant Electrical.

1. Open Catalog Manager and connect to the plant database.
2. Click **File > Open**.
3. Navigate to the Templates folder and select the ‘A Wide (Custom).sym’ file.

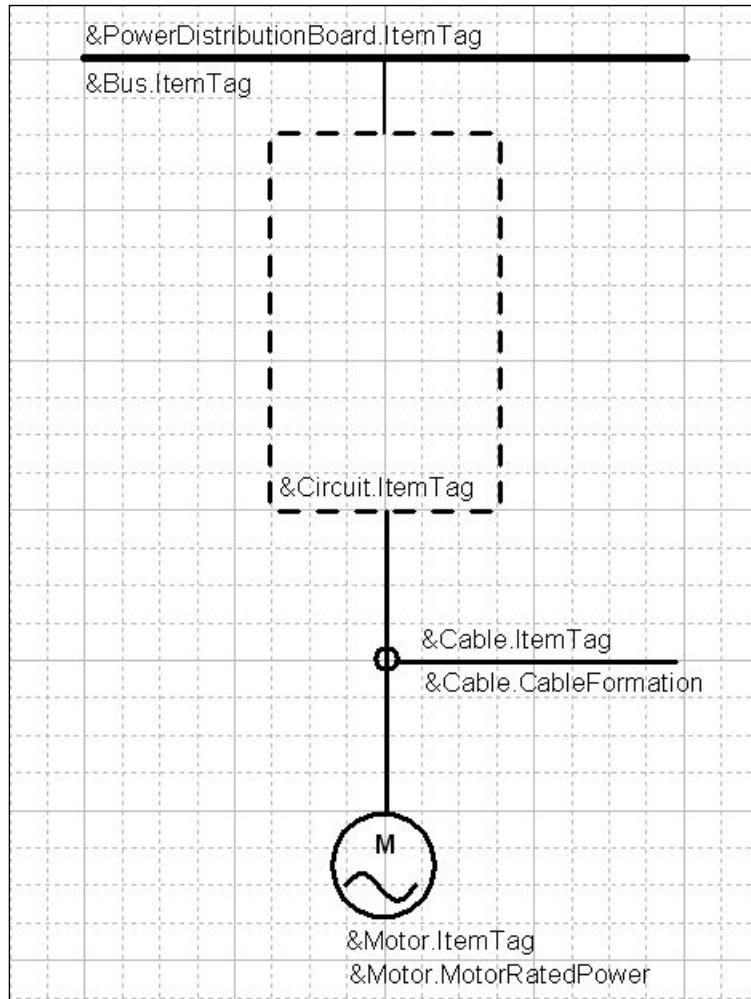
At this stage you are supposed to sketch a complete schematic drawing which will be divided into blocks. To shorten this stage, you will create a simplified drawing that represents a motor with a cable, a rectangle to represent the circuit, and a bus.



4. To change line style or width, right-click the object, and on the shortcut menu, click **Properties**.
5. Add the macros shown in the table.

Graphic Item	Macro	Position
Motor	&Motor.ItemTag	Below motor circle
Motor	&Motor.MotorRatedPower	Below motor circle
Cable	&Cable.ItemTag	Next to cable
Cable	&Cable.CableFormation	Next to cable
Circuit	&Circuit.ItemTag	Inside Circuit rectangle
Bus	&Bus.ItemTag	Above the bus
Power Distribution Board	&PowerDistributionBoard.ItemTag	Below the bus

6. Place the macros according to the following example.



You will now divide the complete sketch into two separate symbols, one for the motor and its cable and the second for the circuit and the PDB information.

Before you begin to create the two symbols, please save your changes to the open symbol file.

To create the two symbols do the following:

1. On the **View** menu, turn off the Grid **Snap**.
2. Activate the 'PinPoint' toolbar (click **Tools > PinPoint**).
3. Select the motor and its cable along with their macros.

Tip

- To select an element, click the **Select** button on the **Draw** toolbar, and select the element.

- You can select more than one element at a time by clicking the **Select** button, select the first element and then holding the **Shift** or **Ctrl** key as you click addition element you want to select. Alternatively, you can click the **Select** button and then drag the mouse to fence elements.

4. Click the **Create Symbol** icon.

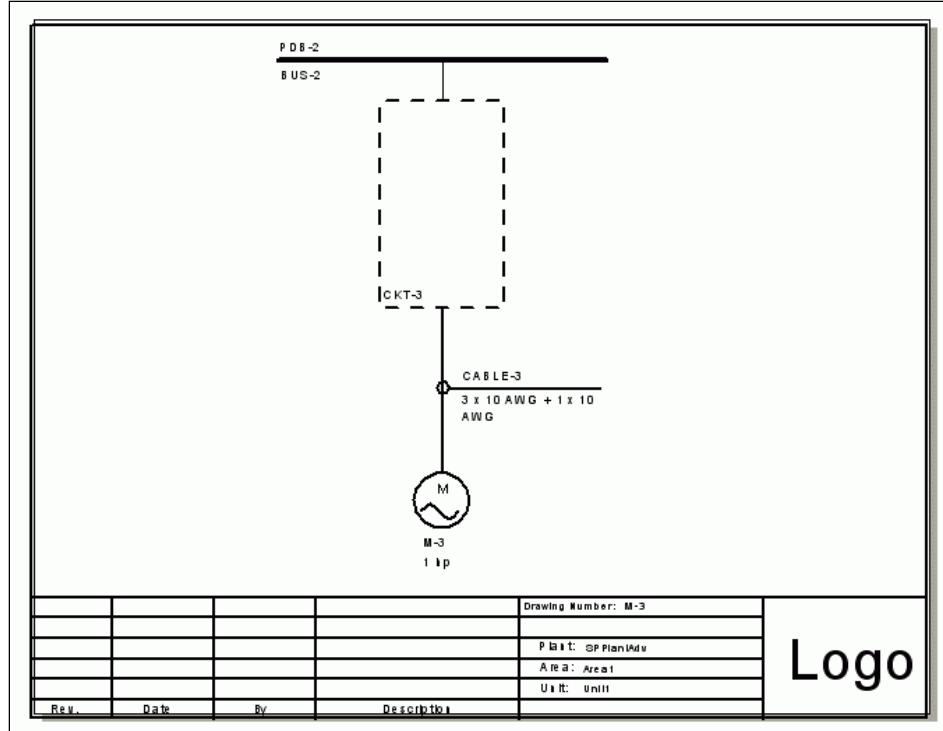
5. Click **Define PinPoint Origin**  to display the ‘Define PinPoint Origin’ toolbar.
6. Type zeros in both the **X** and **Y** boxes. This positions the origin point in the bottom left corner of the sheet.
7. Using the mouse, click the origin point on the lower-left corner ([0,0] coordinates).
8. In the **Save As Symbol** dialog box, fill in the symbol name ‘**Motor and cable**’ and save to the **Schematic Blocks** folder.
9. Repeat steps 3 – 8 for the circuit the bus and their macros, and name the 2nd symbol ‘**Circuit and PDB**’

This concludes the creation of the typical schematic symbols.

To test your work in SmartPlant Electrical, do the following:

1. In RDE create new typical block, and name it ‘**Motor and cable**’. Block type = motor. File name = ‘Motor and cable.sym’.
2. In RDE create new typical block, and name it ‘**Circuit and PDB**’. Block type = circuit. File name = ‘Circuit and PDB’.
3. In RDE create new typical schematic and name it ‘**Motor Schematic Lab**’. In the new typical schematic, add the ‘A Wide (custom)’ for template, and the blocks, ‘Motor and cable’, and ‘Circuit and PDB’ .
4. Using the new typical schematic, generate schematic for motor M-3.

Schematic Drawings



Customizing Single Line Symbols

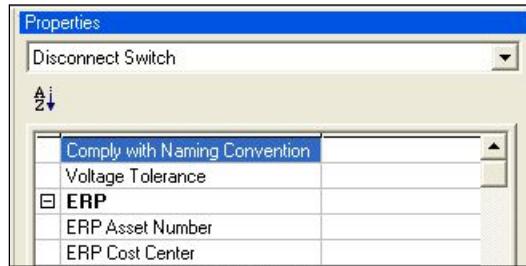
This lab demonstrates how to create or modify single line (SLD) symbol and work with SmartText. SmartText are special macros that the SLD mechanism recognizes.

The steps required to create a new SLD symbol are:

1. Adding a new item and settings the item type
2. Drawing the graphics
3. Placing connection point(s)
4. Placing labels (SmartText)

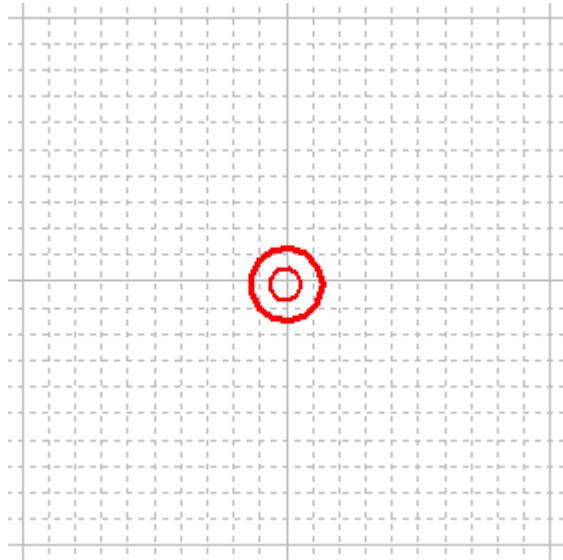
To add a new item:

1. Open Catalog Manager and connect to your plant.
2. Click the SLD folder in the Catalog Explorer.
3. In the **List** view, click an empty region, right-click, and on the shortcut menu, click **New Item**.
4. Select the 'New Item', right-click, and on the shortcut menu, click **Rename**.
5. Type a unique file name, for example **Disconnect switch-Fused**, and then press **Enter**.
6. Double-click the symbol to open the symbol (in the view window).
7. To define the item type of the symbol, in the **Properties** window, select 'Disconnect Switch', and then click **Save**.



8. To adjust the grid units, click **Tools > Options > View**.
9. In the **Grid** section, change **Grid spacing** to **0.1 in** and **Grid index** to **10**, and then click **OK**.

10. Zoom-in on the area where the red circle appears. The red circle is the origin point where you draw the symbol around. You are advised to re-position the red circle at the intersection of main grid lines as shown.



◆ Important

Some guidelines before you continue:

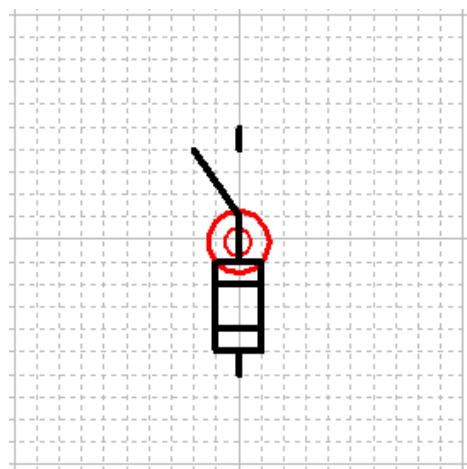
All graphics and constant text should be drawn on the Graphics layer.

The recommended size for single-line symbols is about 10mm or ~0.4" (measured in any direction around the object). You can diverge from this size but you need to make sure that the size does not cause objects to overlap or appear out of proportion in relation to other objects.

Make sure that the origin point is in the center-line of the object, otherwise the symbol will be shifted to the side when used by the program.

To draw the symbol, click the appropriate icon on the **Draw** toolbar.

You need to draw the following symbol.



Add connect points:

1. Make sure that the ‘Catalog Tools’ toolbar is displayed.

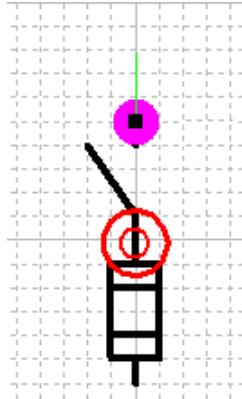
2. Click the **Place Point** icon . The cursor changes to a cross.

 **Tips**

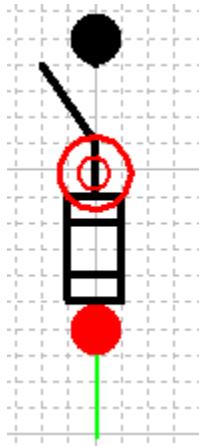
You need to add two connect points (Connection Point type). Each connect point is assigned with an index number according to the order it was added. Final objects like loads will usually get one connect point on the upper end of the graphics. If an item can be associated in between objects, the first index will be the upper (energizing side) and the second connect point is assigned to the lower (energized) side.

In addition, each connect point has a direction to tell the software in which direction to connect the connector. The direction is determined by a second click after placement of the connect point (Indicated by a green line when highlighting the connect point with the select tool). The first connect point should have the direction set to 90 degrees upwards and the second point set to 90 degrees downwards.

3. To add the first connect point click on the upper end of the symbol once and then click again above that point (at a 90 degree angle).



4. Add a second connect point, at an angle of 90 degrees downwards.

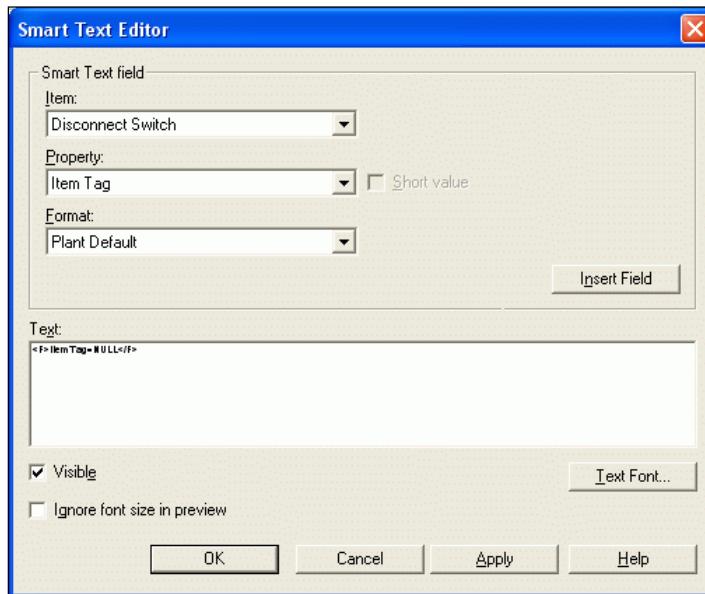


 **Tip**

5. To turn off the display of the connect point, click the **Show Points** icon  on the ‘Catalog Tools’ toolbar.

6. Add some labels (macros in the form of Smart Text) as follows:

- a. Click the ‘Smart Text Editor’ icon  on the ‘Catalog Tools’ toolbar.
- b. In the ‘Smart Test Field’ section select the following:
 - Item = ‘Disconnect Switch’
 - Property = ‘ItemTag’
 - Format = ‘Plant Default’
- c. Click **Insert Field**.

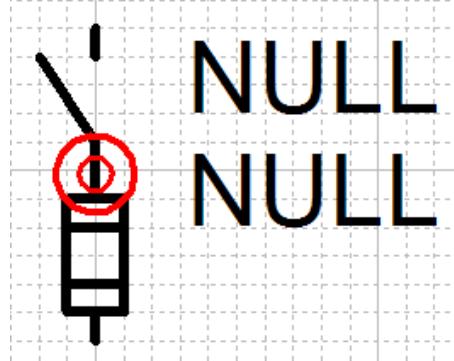


- d. Click **OK**.
 - e. Move the label to the right side of the symbol
 - f. Repeat the preceding steps for adding a label, but this time select **Current Rating** property.
7. Select the two properties, right-click, and on the shortcut menu, click **Properties**.
 8. In the properties dialog set:
 - Info tab: Unit spacing = World**
 - Paragraph tab: Font size = 0.1 in**

9. Click **OK**.

 **Notes**

-  Advanced users who work often with Catalog manager can create a specific text style from the **Format > Style** dialog box.
-  The graphics and the labels should appear as shown.



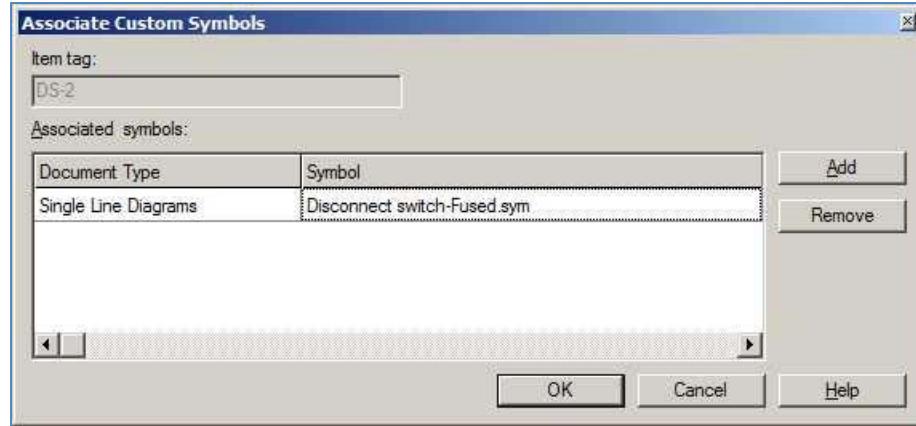
-  The labels that appear as 'NULL' will be replaced by the data at the time of generating the single line diagram.
-  Each symbol has an icon that inherits the contents of the graphic layer. You can observe the icon by clicking the Icon tab.



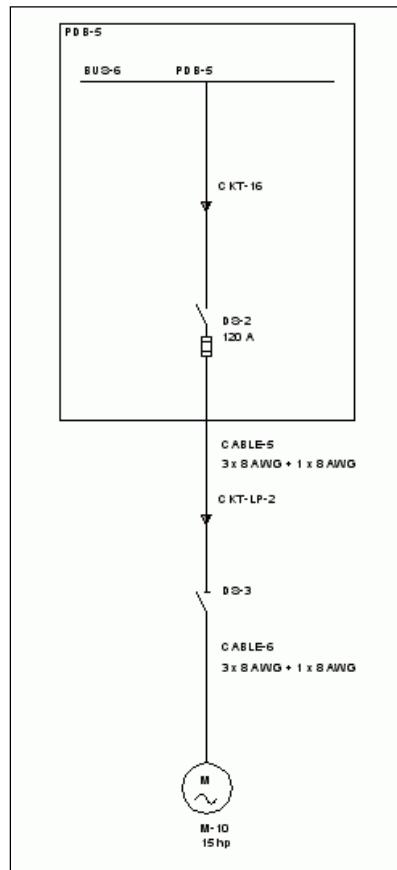
10. Save the symbol and close Catalog Manager.

To test your new symbol does the following:

1. Select the disconnect switch, 'DS-2' in PDB-5 (CKT-16).
2. From the property grid set its current rating to 120 A.
3. Select right click mouse on the disconnect switch, and from the short cut menu select Associate Custom Symbol. Select add, and from the document type select Single line diagram. Mouse click on the Symbol field reveal the ellipse button where you can browse and select you new symbol, **Disconnect switch-Fused.sym**.



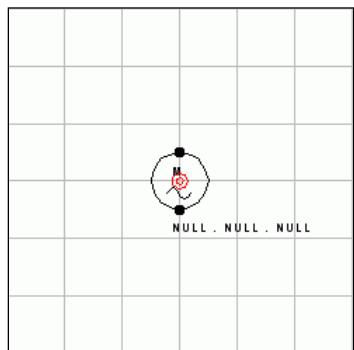
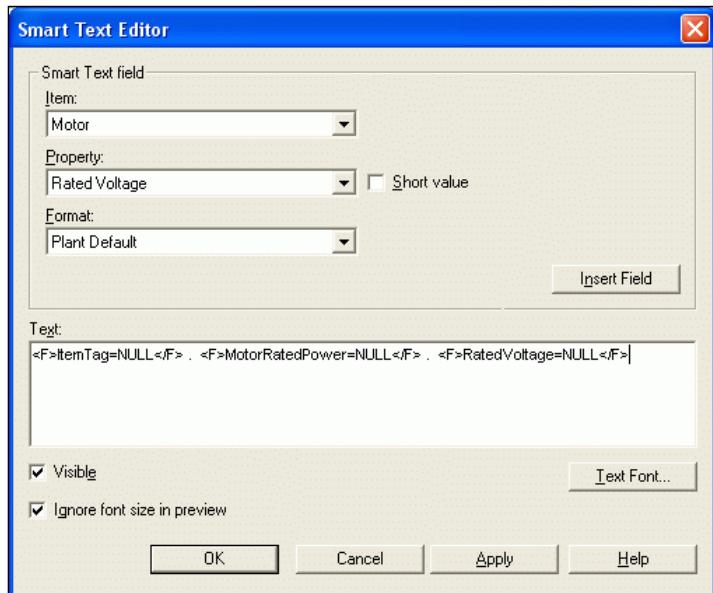
4. Generate single line diagram that includes the new symbol.

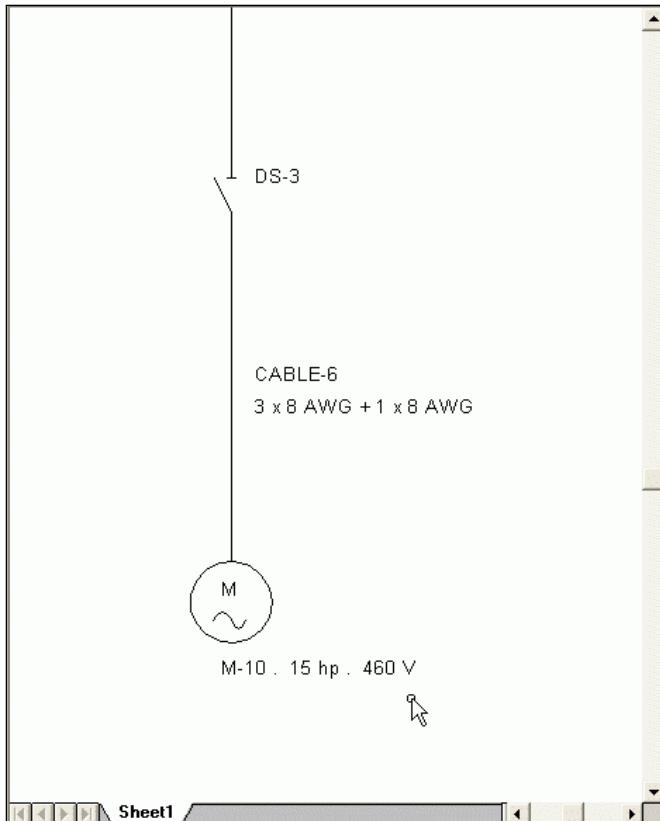


Combined macros in Smart Text

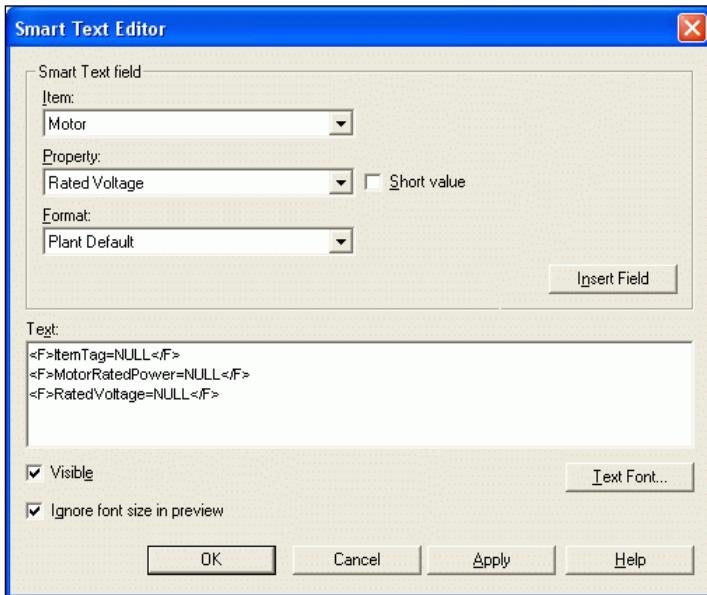
User is able to customize SLD symbols to contain a macro which is combined of few properties or combination of properties and text. The combined properties can be of any data type together.

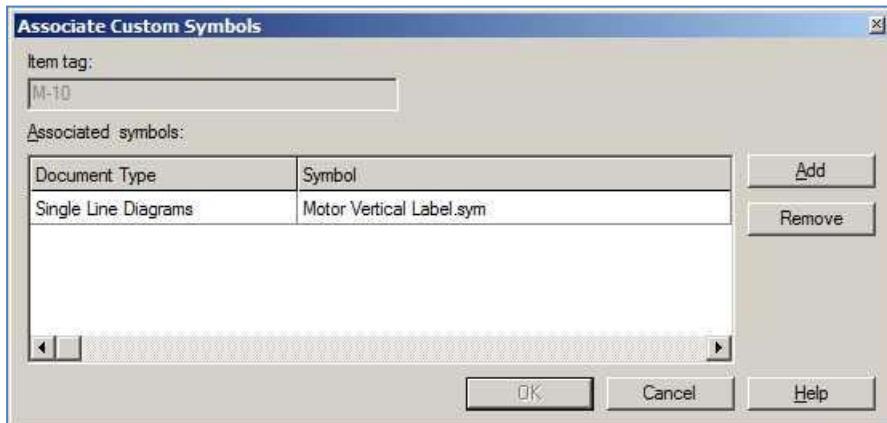
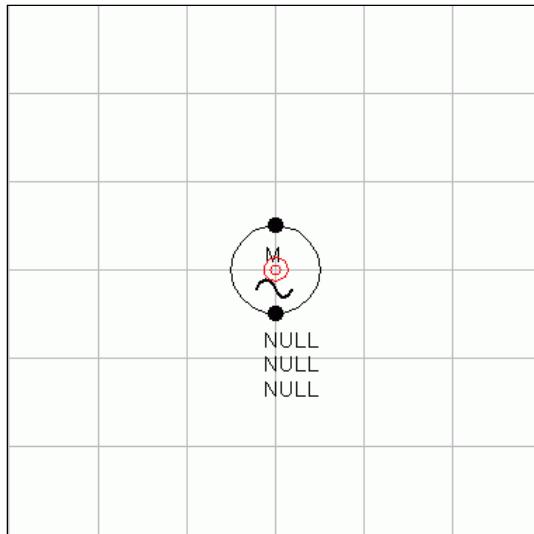
Properties can be combined horizontally like the following example:

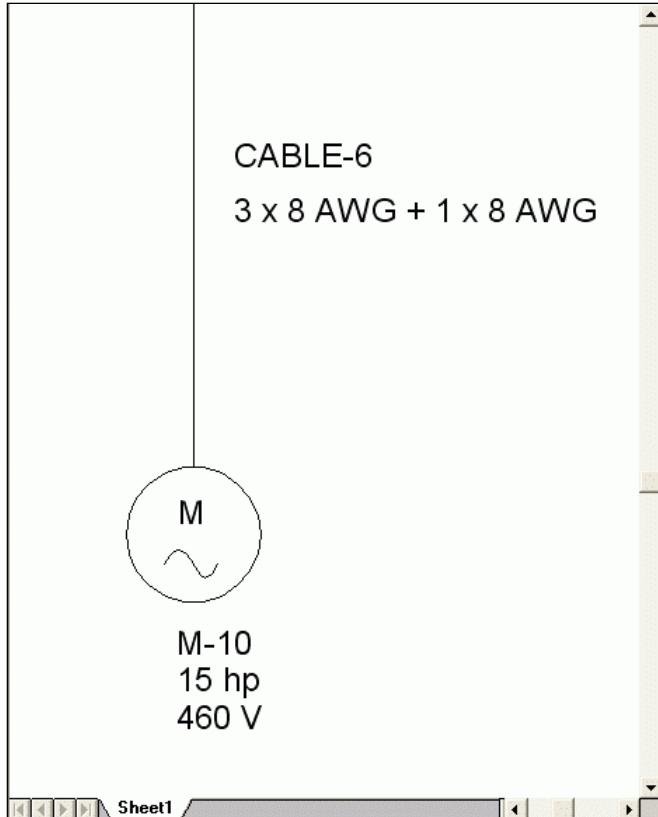




Properties can be combined vertically like the following example:

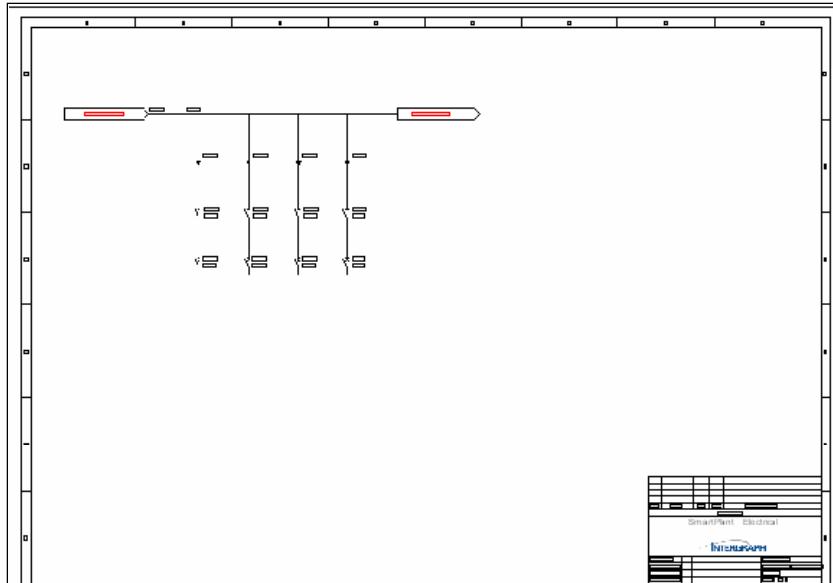






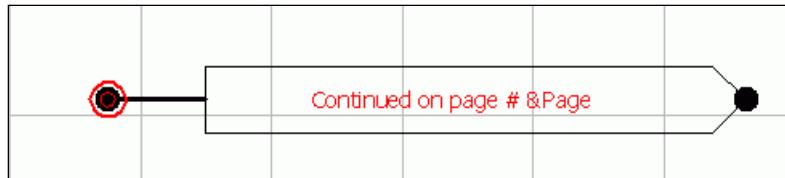
Customize Off Page Connector (OPC) symbols

When generating Power Distribution Board SLD in multi sheets. The bus comes with off page connector (OPC) symbol.



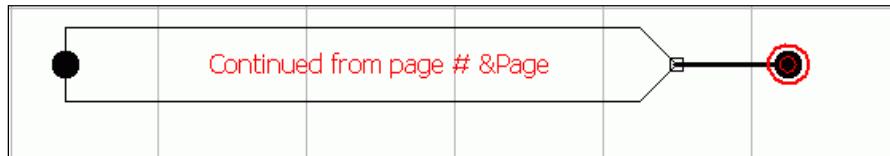
The OPC labels can be customized using the Catalogue Manager. The symbol names are:

OPC_next.sym



The plain text is “continued on page #”, the macro that contains the next page is “&Page”
The connection points are “auxiliary connection points with Index 1 on the left and Index 2 on the right. It is important that the origin circle is placed on the **left most** point

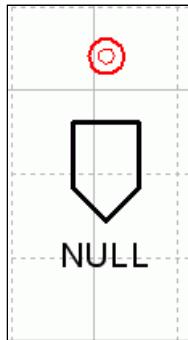
OPC_prev.sym



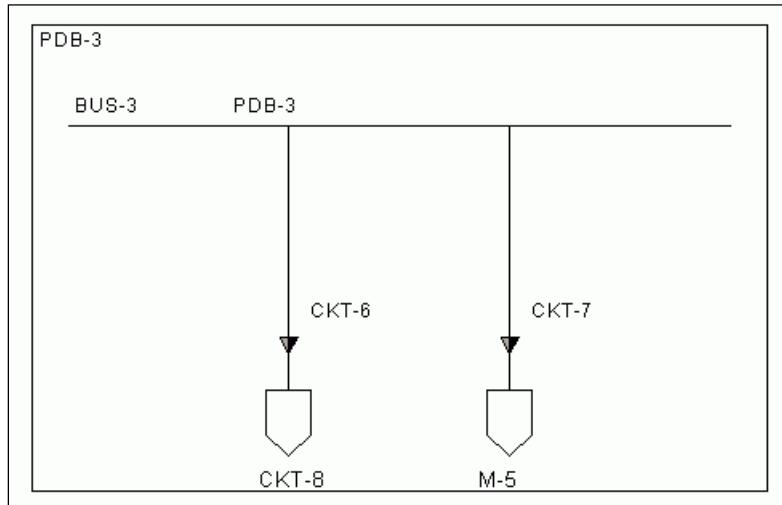
The plain text is “continued from page #”, the macro that contains the next page is “&Page”
The connection points are “auxiliary connection points with Index 1 on the left and Index 2 on the right. It is important that the origin circle is placed on the **right most** point

Customize the reference label symbol in SLD

Reference symbol in single line diagram uses the symbol, RefLabel.sym.



The symbol can be customized by the user as he needs.

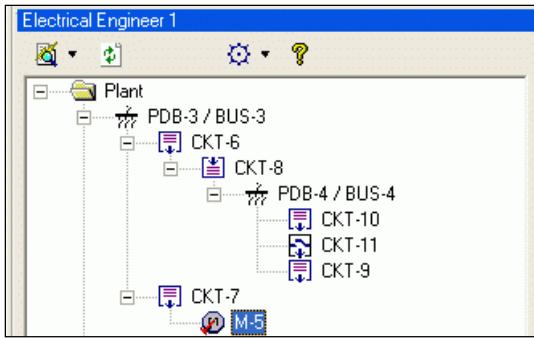


Show the PDB presentation as a “block” symbol

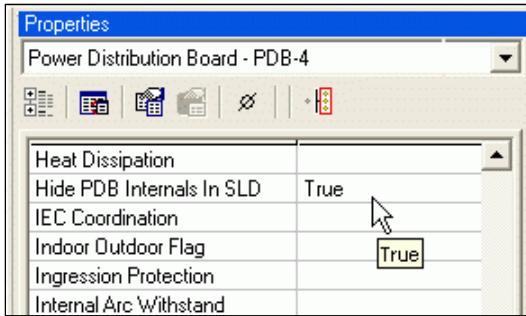
Single Line Diagram mechanism can hide specific PDB internals and display just a symbol for the whole PDB instead of its contents.

In order to use this option need to sign as TRUE the ‘Hide PDB Internals In SLD’ property on each PDB this hiding is required for.

Customizing Single Line Symbols



Set Hide PDB Internals In SLD to True for PDB-4.

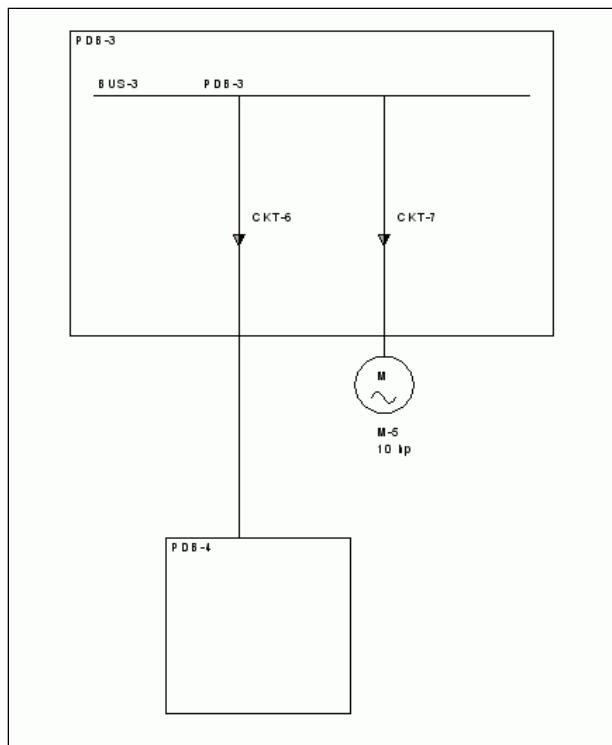
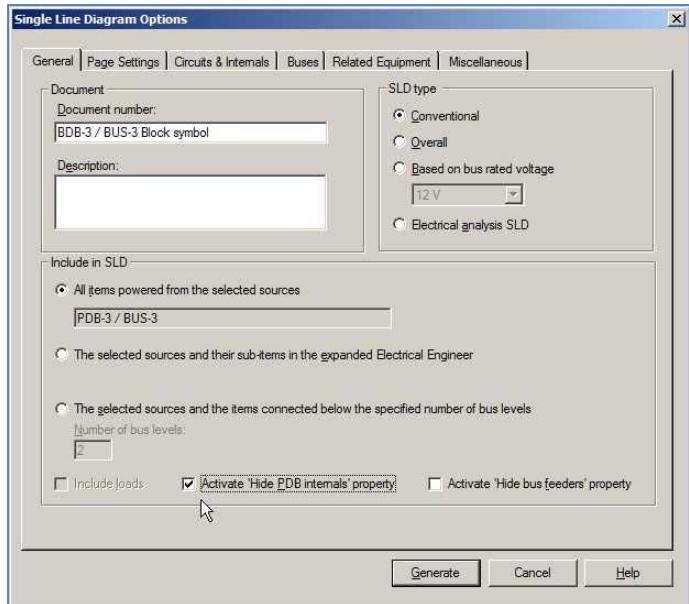


Generate SLD in EE start from PDB-3 / BUS-3



Note: Make sure you check the 'Activate 'Hide PDB internals'' property check box.

Customizing Single Line Symbols

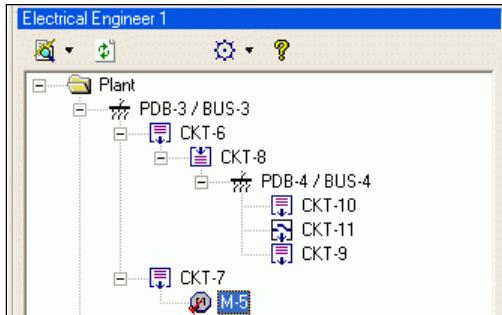


This ability is per drawing only and the user cannot set any preferences to handle it globally. Hiding PDB internals is enabled only if the SLD had been generated from the Electrical Engineer. Since for hidden PDB's, the software uses real symbol rather than just a frame, using symbology for such PDB's is available.

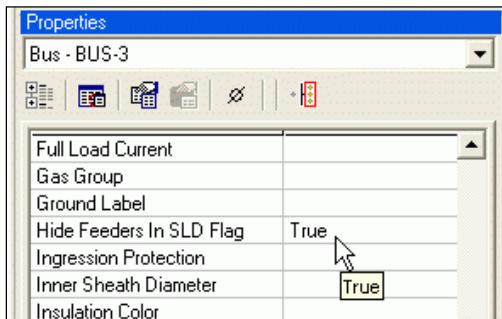
Ability to hide Feeder circuits from being shown is SLD's

The same mechanism for PDB can be done for a specific bus which will hide all of its feeder circuits and their internals. In order to use this option need to sign as TRUE the 'Hide Feeder In SLD Flag' property on each Bus this hiding is required for.

When generating an SLD document, need to sign the relevant check box on the drawing options dialog box.



Set Hide PDB Internals In SLD to True for BUS-3

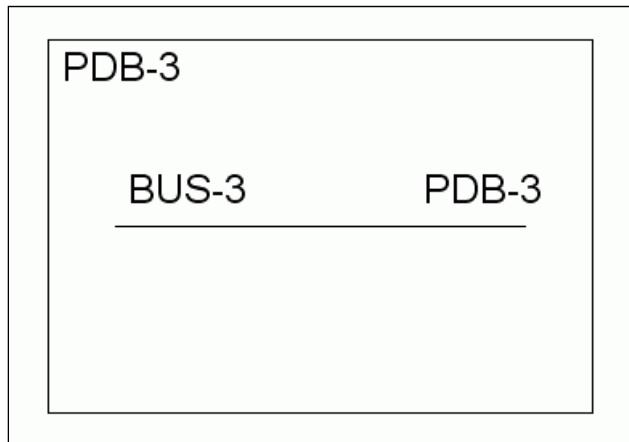
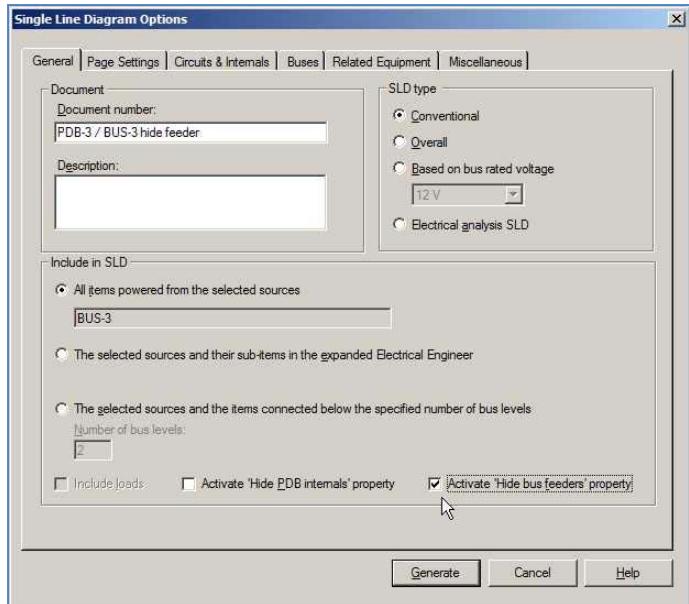


Generate SLD in EE start from PDB-3 / BUS-3



Note: Make sure you check the 'Activate 'Hide bus feeders' property check box.

Customizing Single Line Symbols

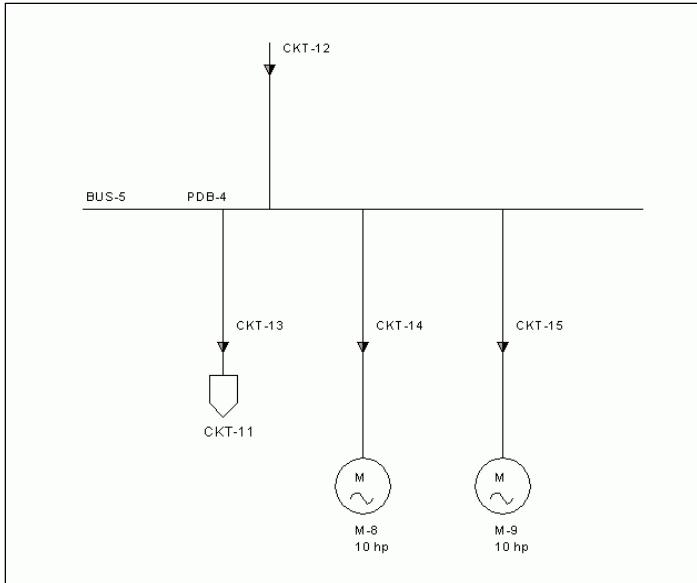


Moving circuits along the buses contour

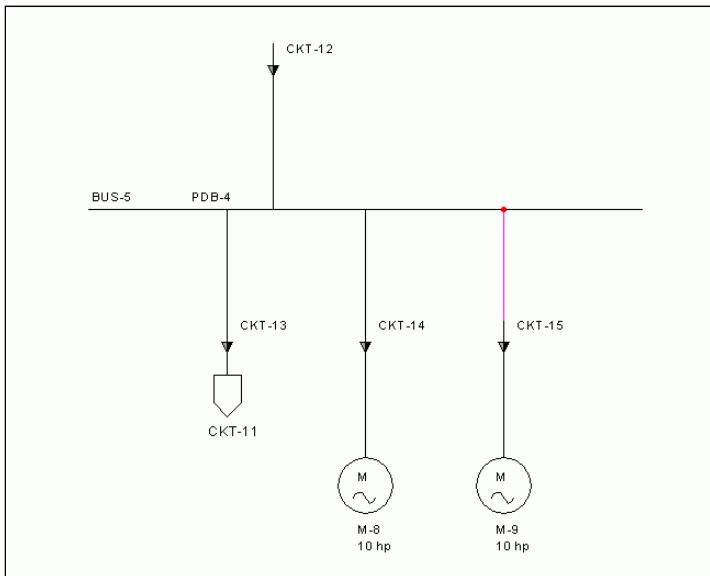
Generating Single Line Diagram document allows the user to change the connection point of any circuit on the bus and fit it to the positions of the various circuits and equipment on the drawing.

Generate SLD in EE start from PDB-4, BUS-5.

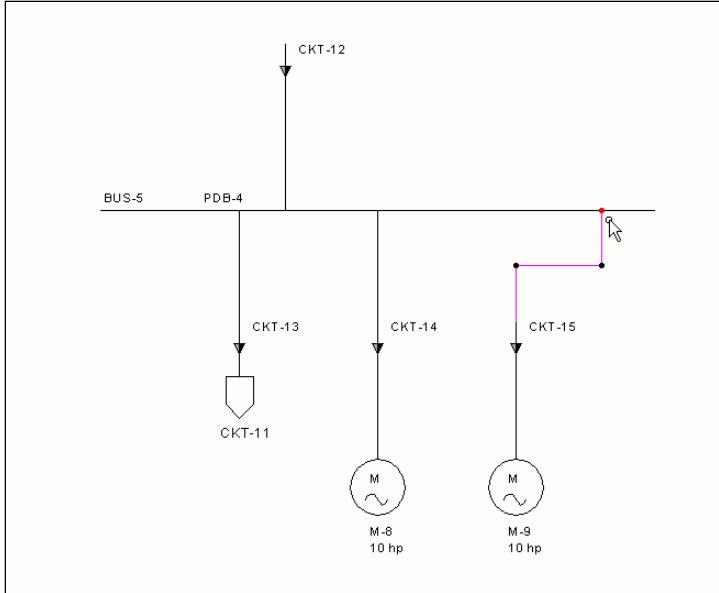
Customizing Single Line Symbols



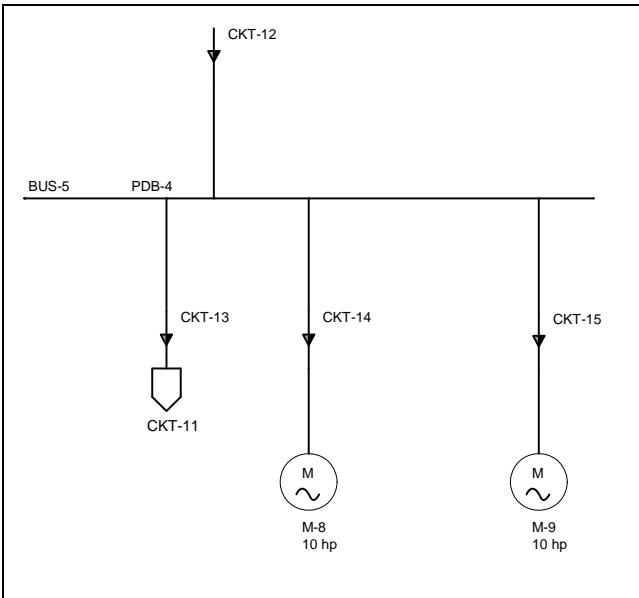
Select the connector above the circuit of which needs to be moved on the bus.



Once holding the connection point with the mouse. Drag and locate the connection point on any location on the bus.



After dragging the connection point, you can align the item using the align feature.



Once saving the document, the new location of the connection point is saved with the other document settings. As the document is deleted, the new location of the connection point returns back to default.

Symbology

Symbology provides graphical clarity to the drawing by differentiating among various items by their appearance. Symbology refers to the color, line width, shape, and style associated with items.

Symbology allows to:

- Define a symbol for a specific type of drawing (SLD, wiring or cable block diagram), for a specific item type, up to a level of a single or to a set of items.
- Associate a dedicated symbol to an item

The final presentation of a tag in a SLD (or any other drawing) will comply with the following rules.

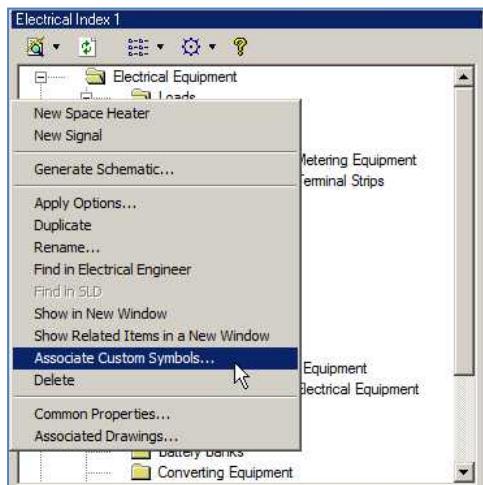
Priority 1 - The item has a custom symbol associated with it.

Priority 2 - If the item has no dedicated associated custom symbol, the system looks if there is any symbology defined and use it.

Priority 3 - If there is none of the 2 above, the system will use the default symbol (e.g. Motor.sym or Heater.sym).

Associating a custom symbol to an item.

Select the item in EI, and select right click and select ‘Associate custom symbols...’



From the ‘Associate custom symbols’ windows select the document type and the symbol.

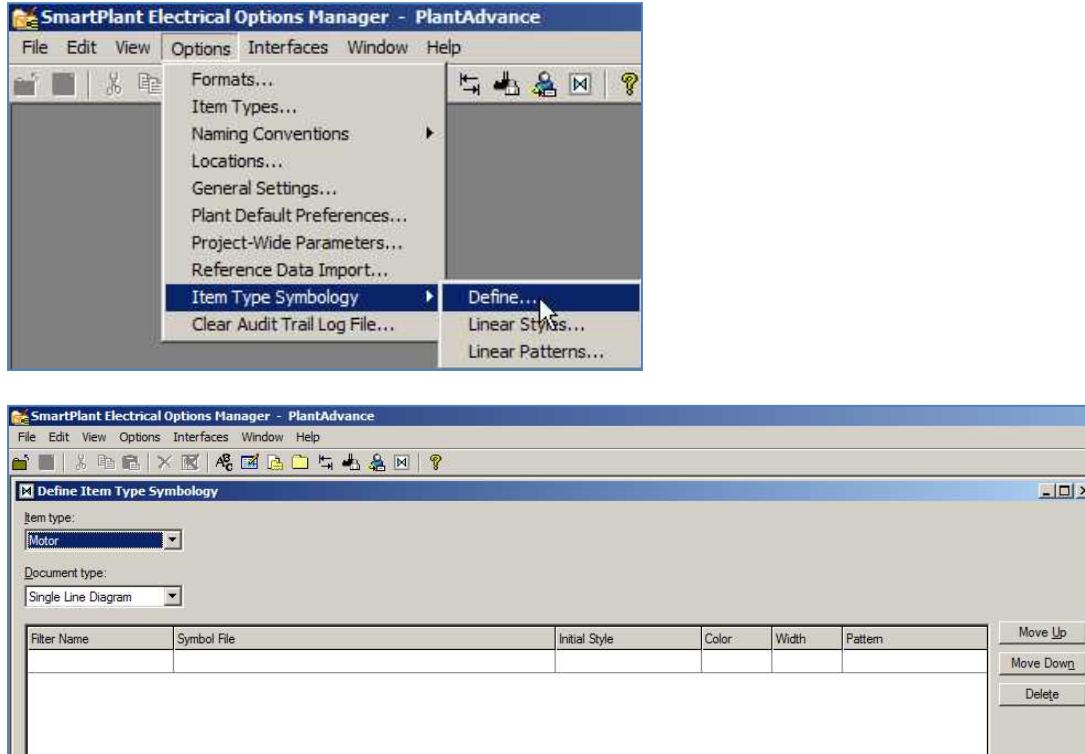


Notes: You can only associate SLD symbols of the same item type.
RDE item defines the SLD symbol name in the property grid.

Defining symbology to an item.

Symbology is defined and set in the Options Manager, and it defines the way an item will be presented in the various drawings (Single Line Diagram, Cable Block Diagram, Wiring Diagram, and PDB Layout).

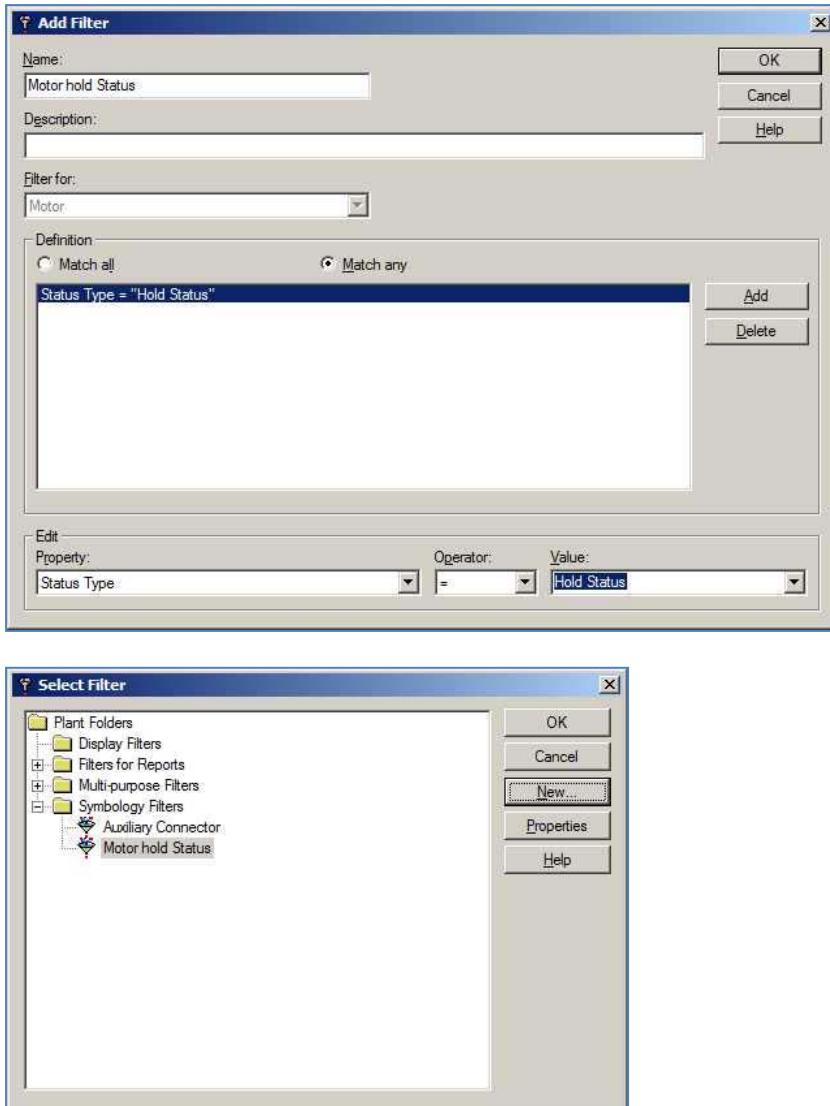
Let's open the Option Manager and select options->Item Type Symbology->Define



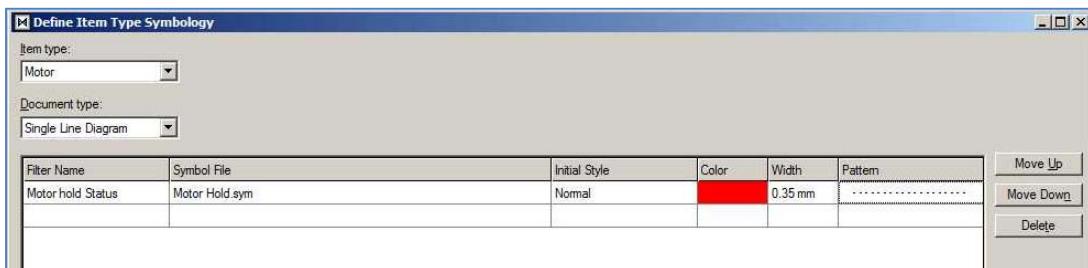
Select Motor as the Item type and Single Line Diagram as the document type.

Selecting the Filter name cell in the grid, create a filter that will present motors with Status type= "Hold status"

Symbology



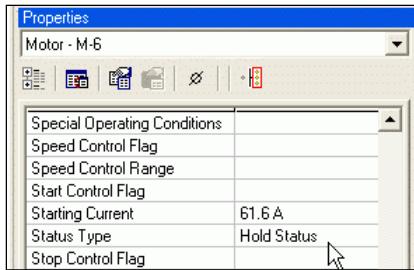
Select the “Motor Hold.sym” and set the Color to red and the pattern to dashed



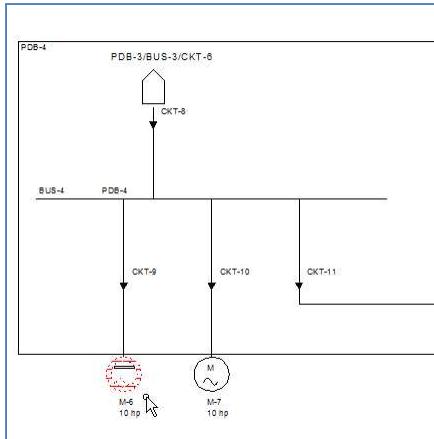
Select save to save the new symbology.

Reopen SmartPlant Electrical. Select motor M-6 and from property windows set its status type to ‘Hold Status’.

Symbology

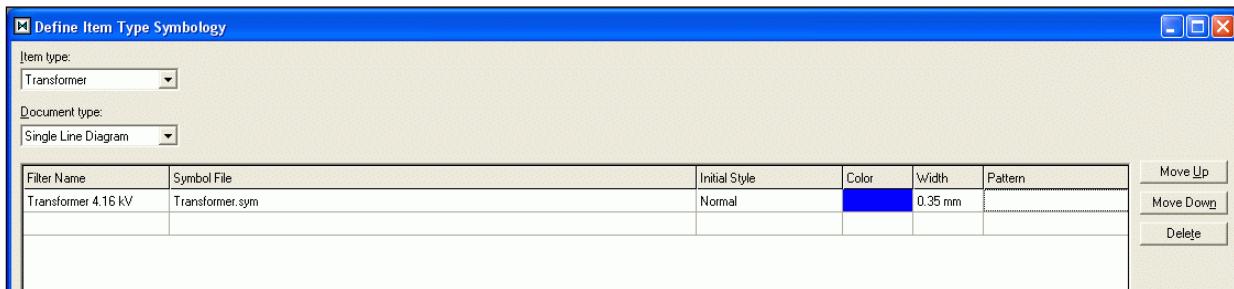


Generate SLD in EE start from PDB-4, BUS-4 and view the result.

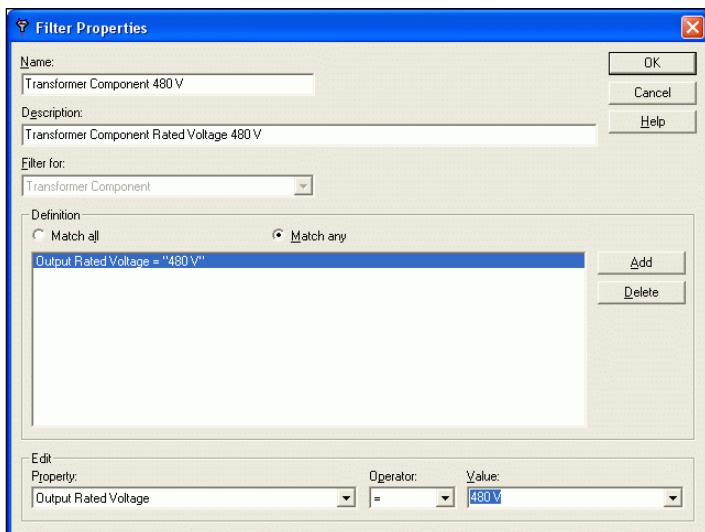
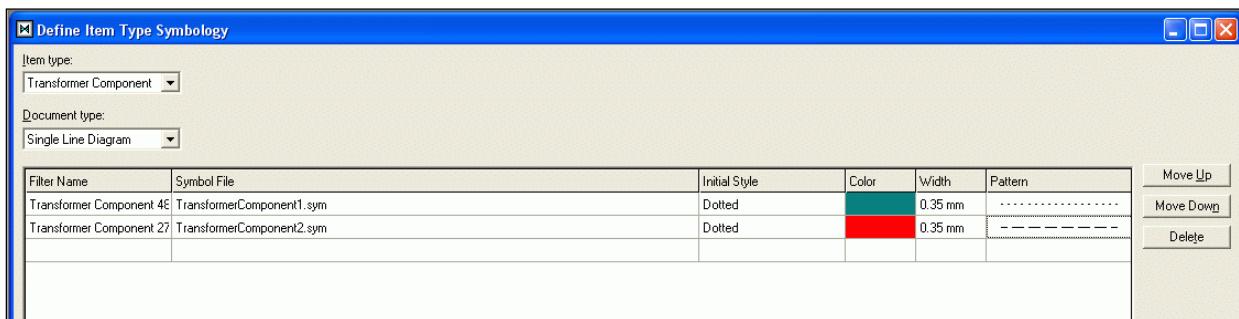
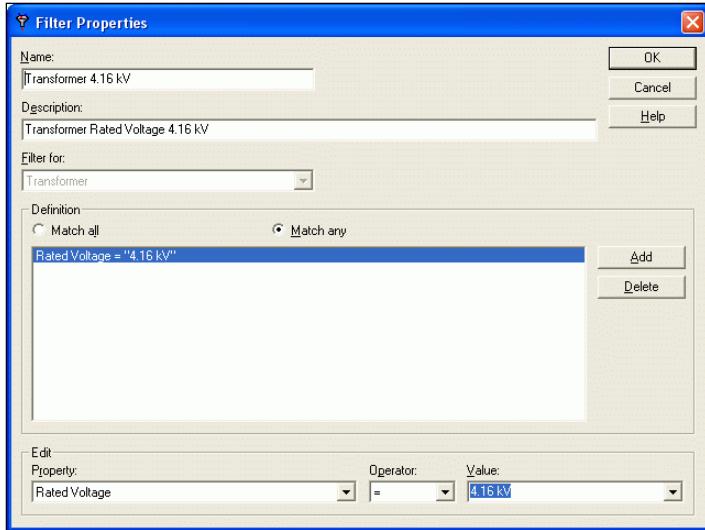


Transformer and transformer component

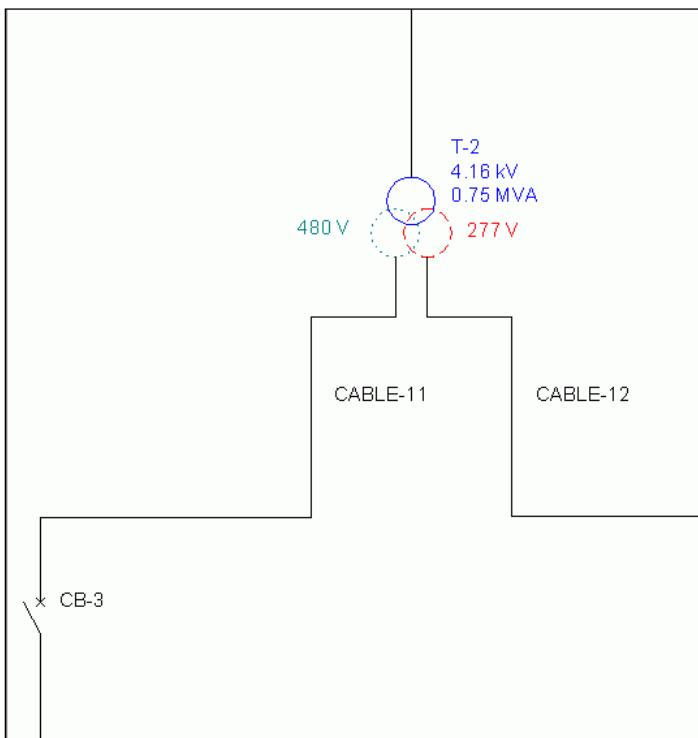
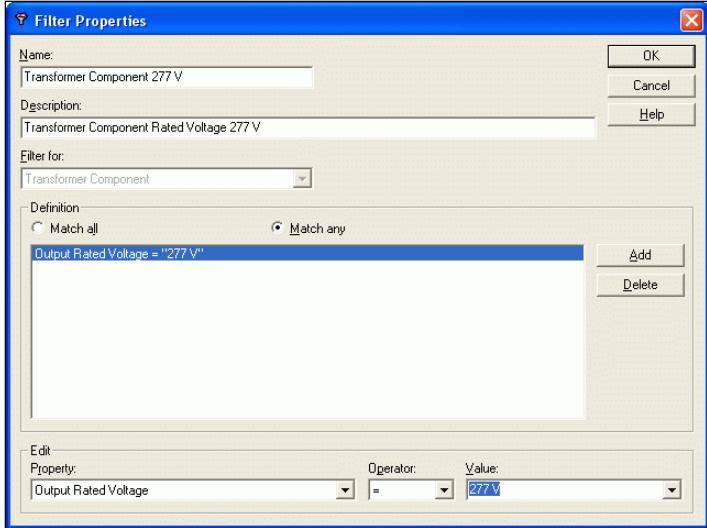
Another example is to define different symbol appearance for transformer and transformer components.



Symbology



Symbology



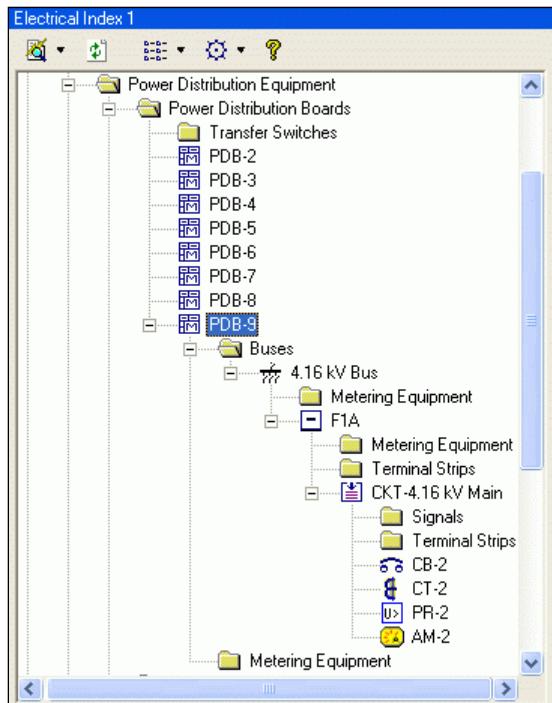
Assemblies in SLD

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. 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, or motors and their associated items. The assemblies symbol draw and create first in the Catalog Manager.

When creating an assembly or combining two assemblies or more, the items in the assemblies must have valid relations.

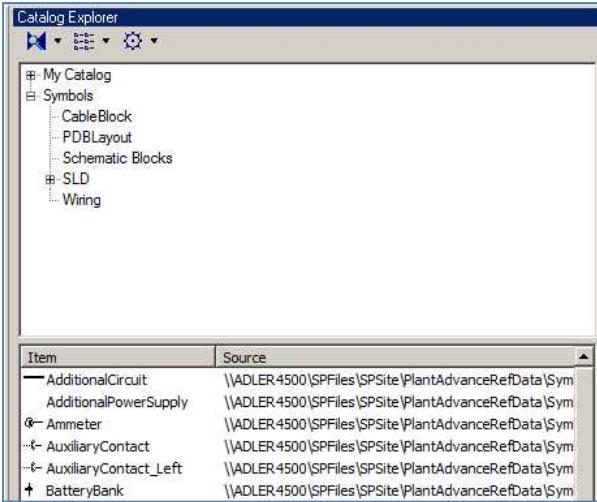
Let's create an assembly of a 4.16 kV volts switchgear incomer circuit that contains the following inclusive items:

Circuit
Circuit breaker
Current transformer
Protection relay
Ammeter

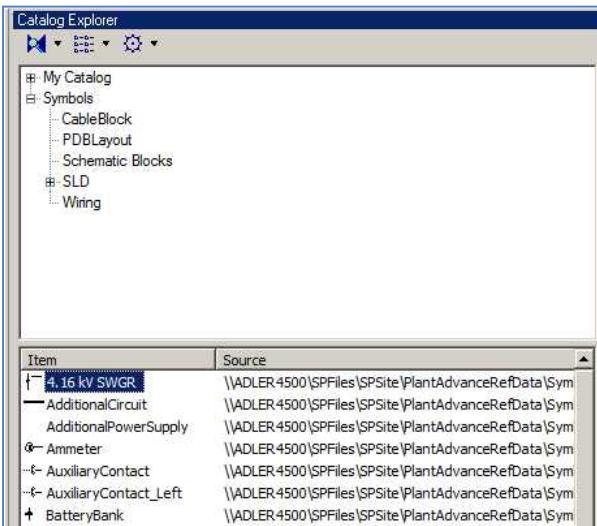


Open the catalog Manager and create the symbol assembly:

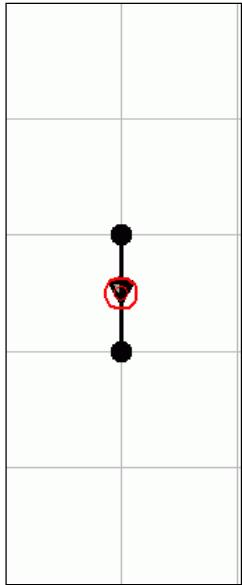
1. From Catalog Explorer select SLD



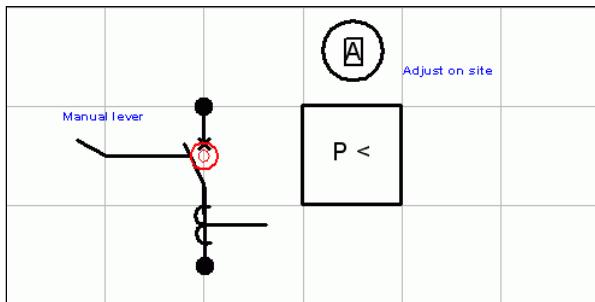
2. From catalog Explorer lower pane, right click on the symbol circuit and select clone.
In the bottom of the list find the symbol name ‘Clone of Circuit(1)’
Right click on the symbol and select rename. Rename the symbol to ‘4.16 kV SWGR’.



3. Right click on the new ‘4.16 kV SWGR’ and select open

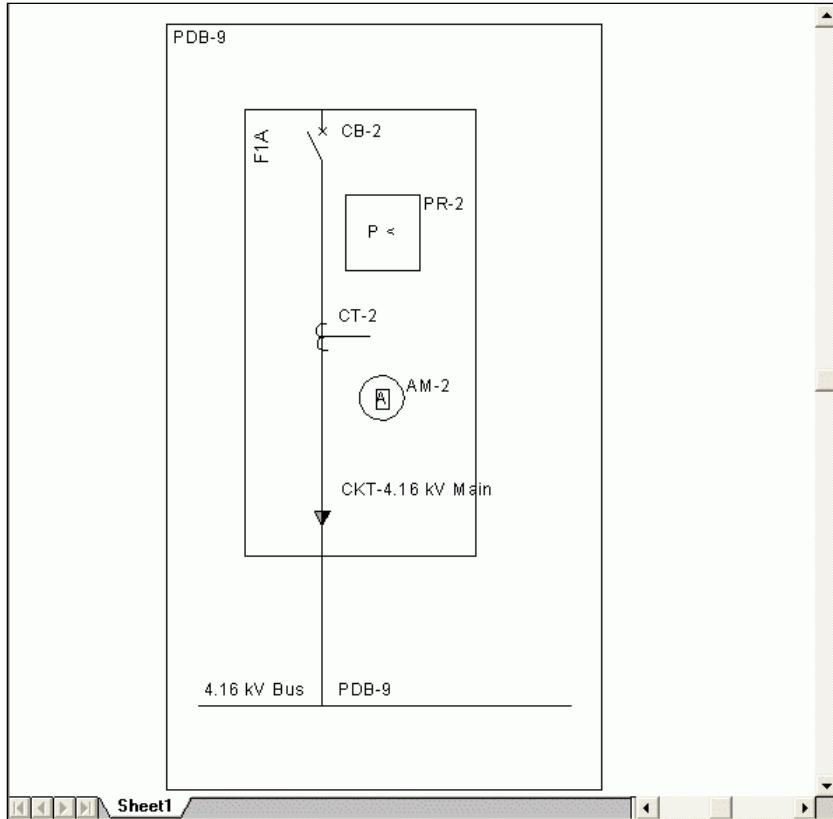


4. While the assembly symbol drawing is open, start and drag the individual items into the drawing and positioning, connecting all to one complete new symbol.
If you do not need the circuit item tag to appear? You may delete the circuit item tag macro.
Also add some text and lines as shown so it looks like:



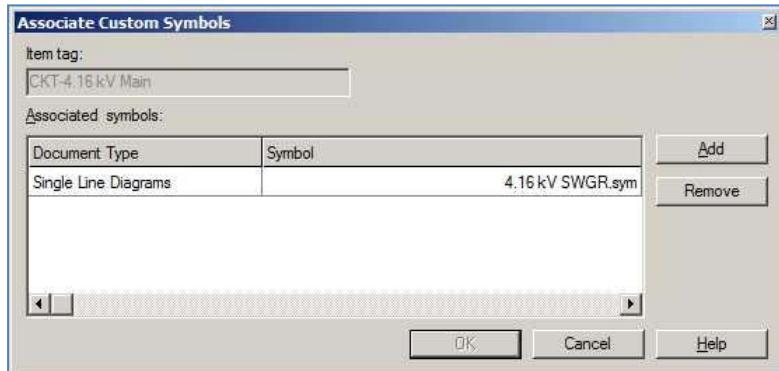
5. Save the symbol change.

Now, if we generate the SLD just as is, with no special further management, setting the show metering equipment in the Drawing options will result in the items being shown in the SLD as per their individual symbol:

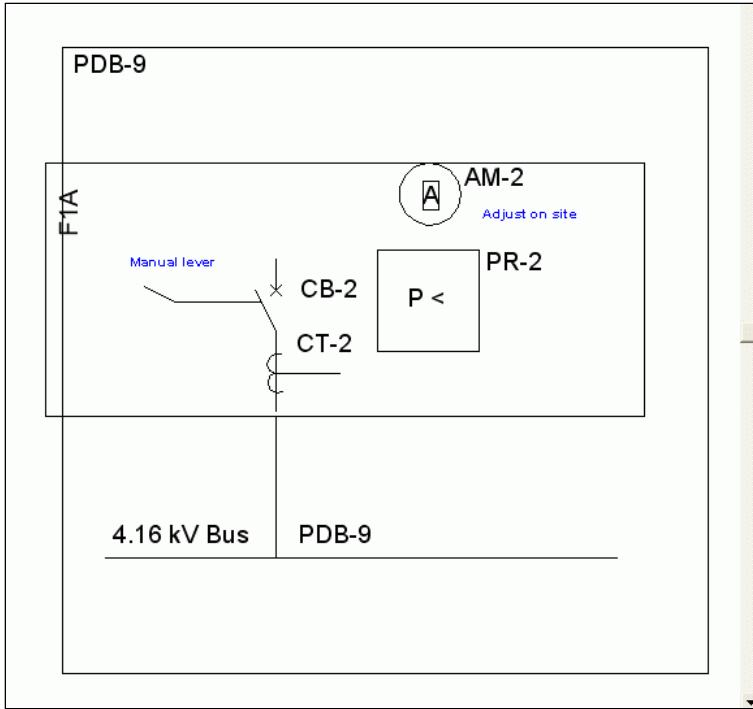


To make use of the special circuit symbol as an assembly, we need to associate the new symbol with The circuit using associate custom symbol option or by symbology.

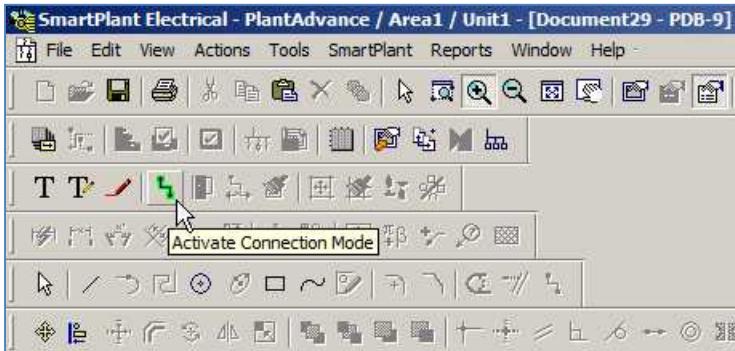
Let's associate the new symbol assembly with the incomer circuit using associate custom symbol option.

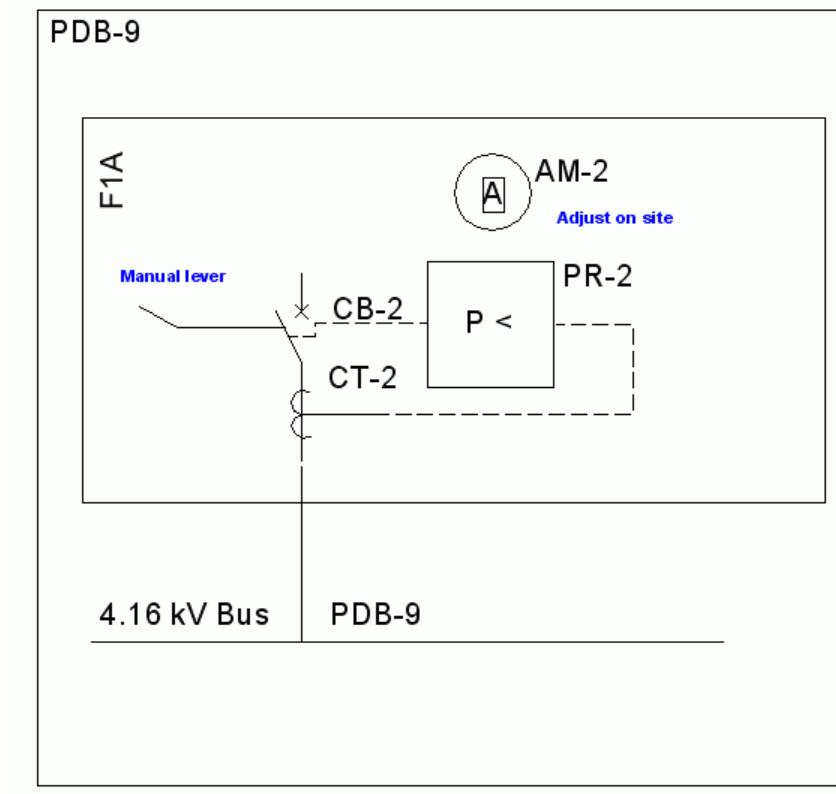


Generating the SLD once more



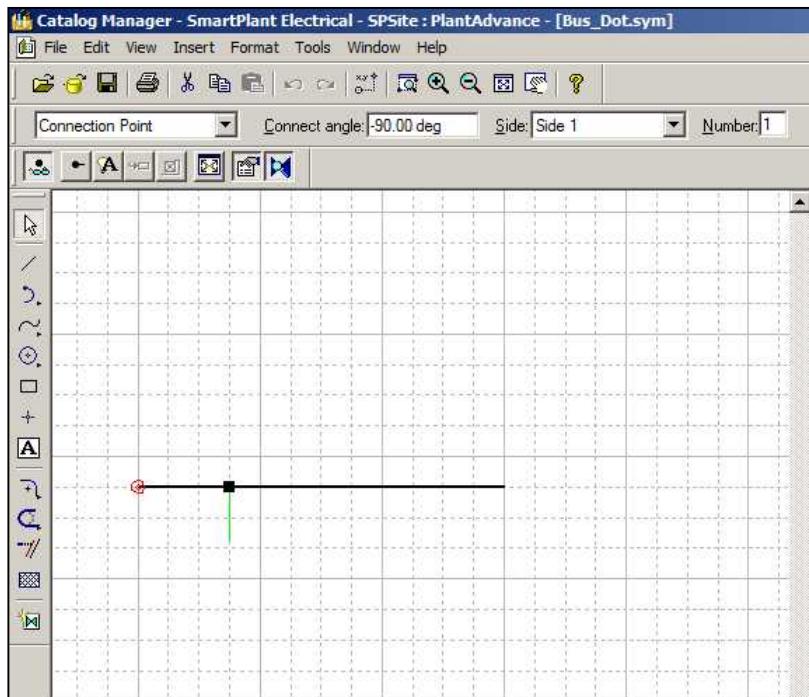
Perform the logical connections by select activate connection mode and make the connection.





Showing dot connection on a bus in SLD

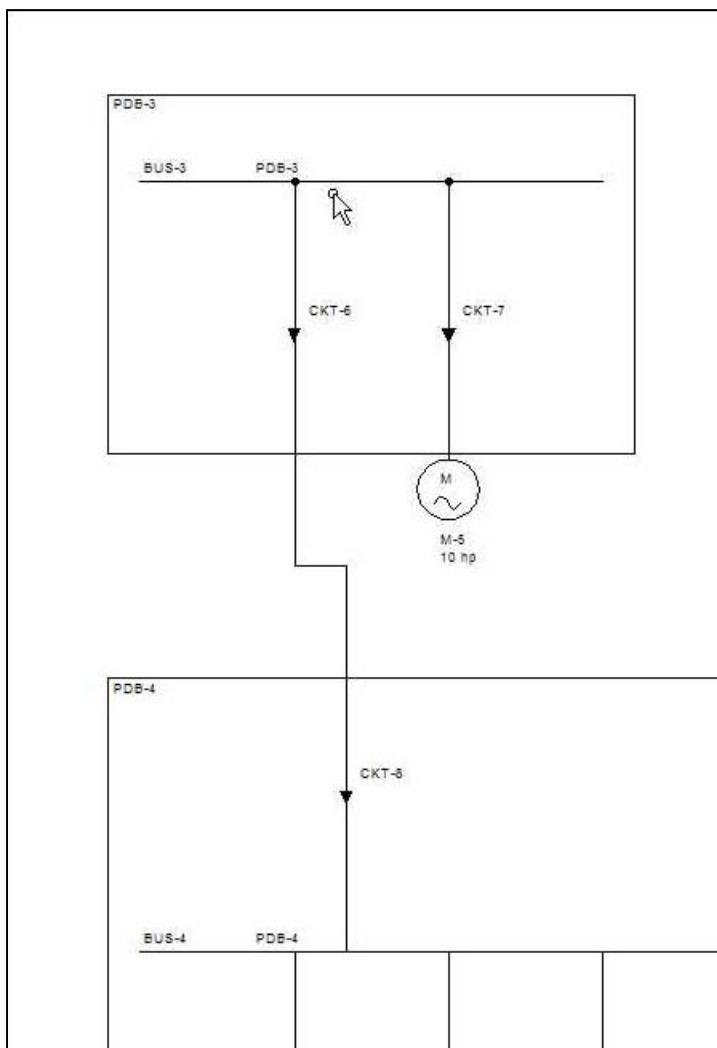
In order to have a dot when circuit is connected to bus in SLD need to add one connection point to a bus symbol.



Associate the new bus symbol (e.g. Bus_Dot.sym) with a bus and generate SLD.

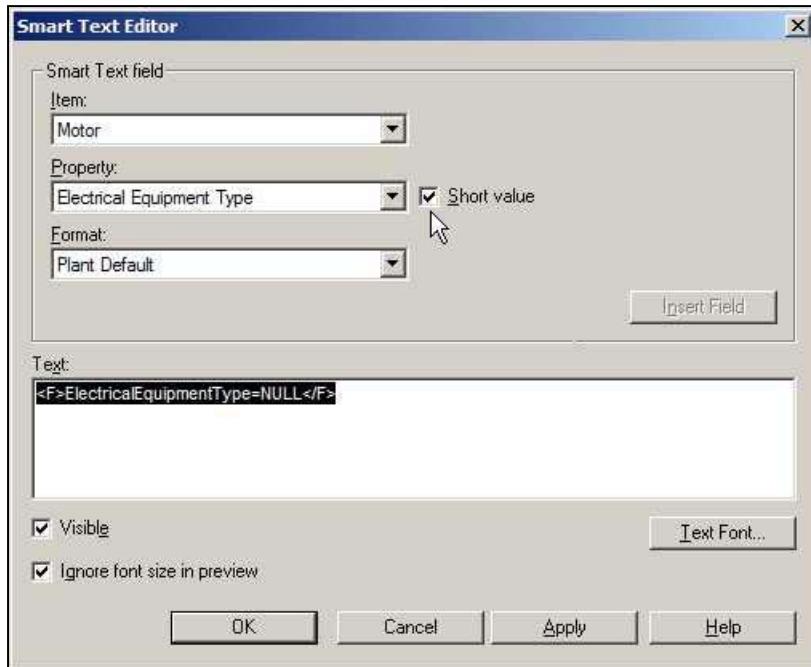
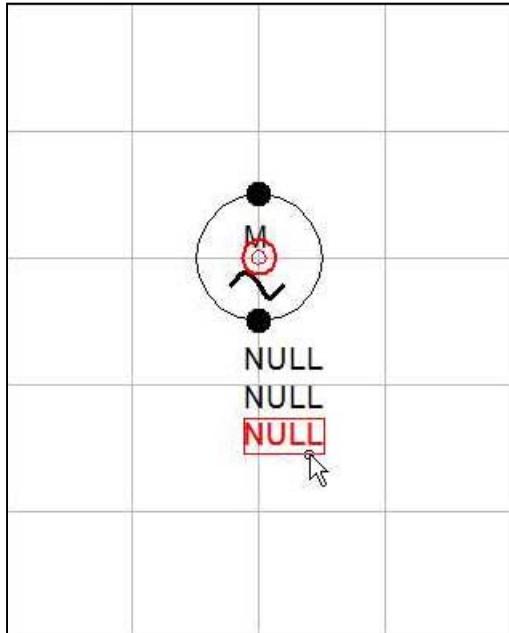


Showing dot connection on a bus in SLD



Show symbol macro short value in SLD

When using catalog manager to customize symbol. It is possible to show the select list short value.



Show symbol macro short value in SLD

Associate Custom Symbols

Item tag: M-5

Associated symbols:

Document Type	Symbol
Single Line Diagrams	Motor_ShortValue.sym

Add Remove OK Cancel Help

Update Select Lists

Selected list: Electrical Equipment Type

Dependent list: Electrical Equipment Sub Class

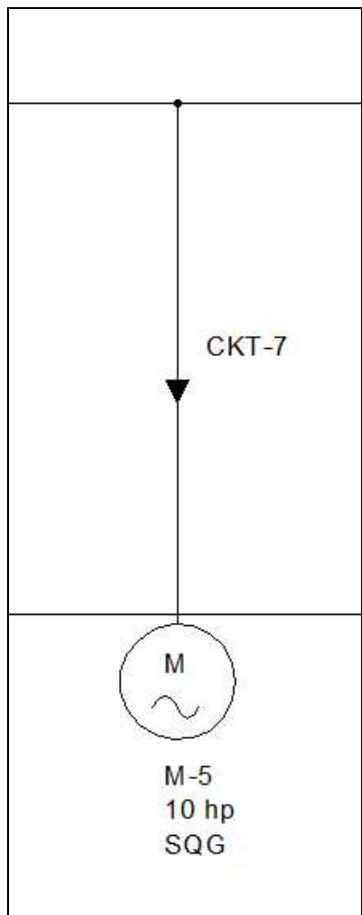
Disable	Value	Short Value	Dependent Value
<input type="checkbox"/>	Inductive Motor	Inductive Motor	Motor
<input checked="" type="checkbox"/>	Squirrel cage	SQG	Motor
<input type="checkbox"/>	Special Electrical Equipment	Special Electrical Equipment	Other Electrical Equipment
<input type="checkbox"/>	Lighting Fixtures	Lighting Fixtures	Other Electrical Equipment
<input type="checkbox"/>	Socket Outlets	Socket Outlets	Other Electrical Equipment
<input type="checkbox"/>	Welding Outlets	Welding Outlets	Other Electrical Equipment
<input type="checkbox"/>	Junction Box	Junction Box	Panel
<input type="checkbox"/>	Local Panel	Local Panel	Panel
<input type="checkbox"/>	Control Station	Control Station	Panel
<input type="checkbox"/>	Motor Control Center	Motor Control Center	Power Distribution Board
<input type="checkbox"/>	Power Distribution Panel	Power Distribution Panel	Power Distribution Board
<input type="checkbox"/>	Switchboard	Switchboard	Power Distribution Board
<input type="checkbox"/>	Transfer Switch	Transfer Switch	Power Distribution Board
<input type="checkbox"/>	Emergency	Emergency	Power Distribution Board
<input type="checkbox"/>	Load Center	Load Center	Power Distribution Board
<input type="checkbox"/>	Switchgear	Switchgear	Power Distribution Board
<input checked="" type="checkbox"/>	Feed-through	Feedthru	Terminal
<input type="checkbox"/>	Disconnect	Disconnect	Terminal
<input type="checkbox"/>	High Current	Hi Cur	Terminal
<input type="checkbox"/>	Ground	Gmd	Terminal
<input type="checkbox"/>	Fuse	Fuse	Terminal

OK Cancel Apply Help

Properties

Motor - M-5

Efficiency Operating	0.942
Electrical Equipment Class	Electrical Equipment
Electrical Equipment Sub Class	Motor
Electrical Equipment Type	Squirrel cage



Comparing Documents

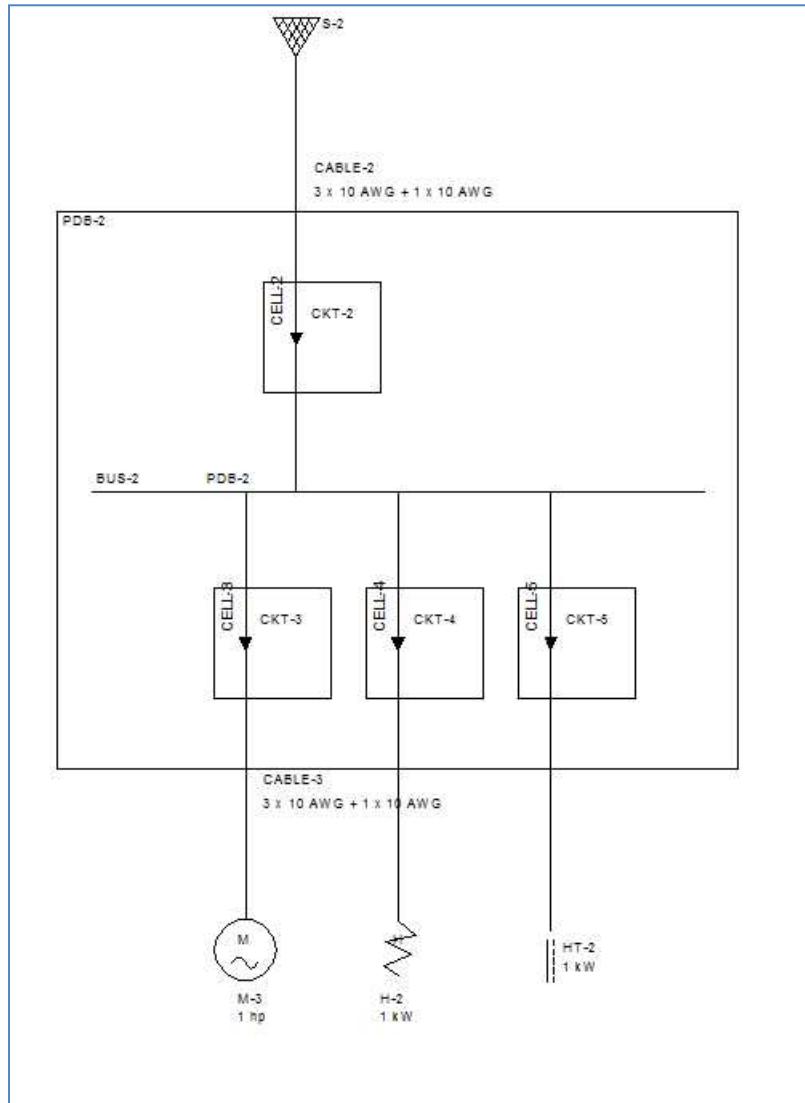
When SmartPlant Electrical works in archived documents enable.

The software archives a copy of the pertinent document revision when new revision is been added to the document. This allow us to compare between data in any of the archived revision and the current data. The compare feature works for Single Line Diagram, schematics, registered report, cable block diagram and wiring diagram.

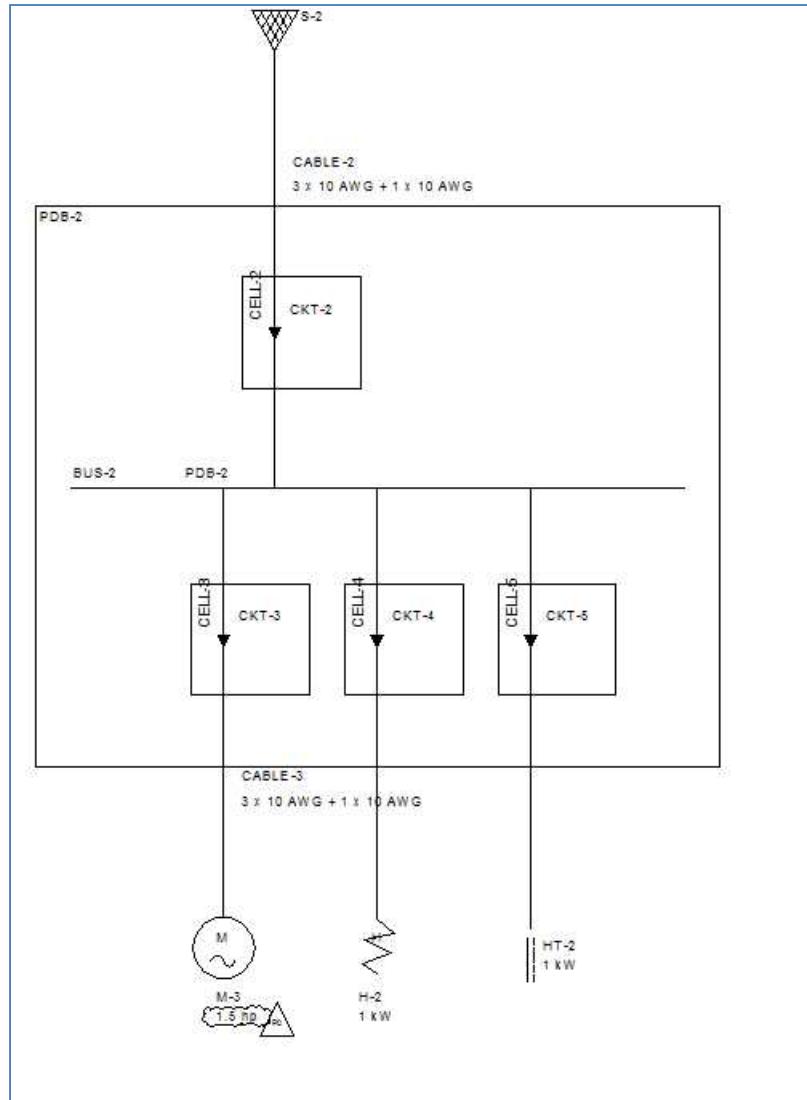
The following is step by step how to compare Single Line Diagram (SLD) document

In Options Manager, under general settings, set the Archive documents flag to Enable and select save.

1. Open Smart Plant Electrical, and generate a new SLD drawing.



2. Modify the drawing as you require.
3. Select the Save button to create and save a document for the drawing.
4. Select Document folder, Single Line Diagrams sub folder, and right click on the new document to open short cut menu where you can select Document Properties.
5. From the document properties windows, select PO for revision method, and select New to add new revision. Fill more details in the checked by, approved by, as you require and click OK to close the document properties windows.
7. Close the open drawing (file menu -> close).
8. Make a change to one or more value of property that appears in the SLD.
e.g. Change the Motor Rated Power
9. Select Document folder, Single Line Diagrams sub folder, and right click on the new document to open the short cut menu where you can select Compare Documents.
10. From the compare document windows, select the PO record and select the Compare button to compare between the current data with the archived document PO. The compare feature generates 2 types of documents:
 1. Single Line Diagram with the current data, that includes cloud symbol and triangle symbol (triangle with the compared revision inside), next to each property that its value has been changed (change between PO and the current data).



2.SLD document report in excel with list of item tags that have been changes
Including current value and previous value.

B	C	D	E	F	G	H	I	J
Date and time of Print:		24/03/11 0:29:7						
SLD Document Comparison S-2								
Plant/Project:		PlantAdvance						
Current Drawing Name:		S-2						
Compared revision: P0		Date: 3/24/2011		Created: yadler				
Tag	Item Type	Property	Current value	Previous Value	Change Qualification (U=update,N=New,D=Deleted)			
M-3	Motor	MotorRatedPower	1.5 hp	1 hp	U			
Added/Removed items from Document:		Reference ID						
Moved or relation changes:		Reference ID	Current relation	Previous relation				

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 only mode.

 **Notes**

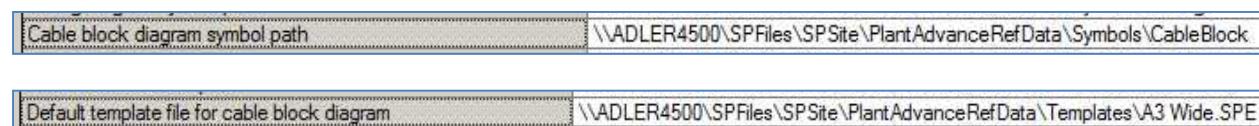
-  Same compare and view feature exists for registered report, schematics, cable block diagram, and wiring diagram.
-  For registered reports, the software opens two Excel files. One of the Excel files displays changed data with blue shading. The second file is a summary of all changes and it is called **Registered Comparison Report**. This report displays the previous and current data for each tag that has undergone a change. Note that you can compare 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.

Cable Block diagrams (CBD)

Cable Block Diagrams (CBDs) provide a graphical representation of the interconnection of all control, power, instrumentation, and grounding cables, and their related equipment such as power distribution boards (PDBs), 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**, **Electrical Engineer**, and **Reference Data Explorer** 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.

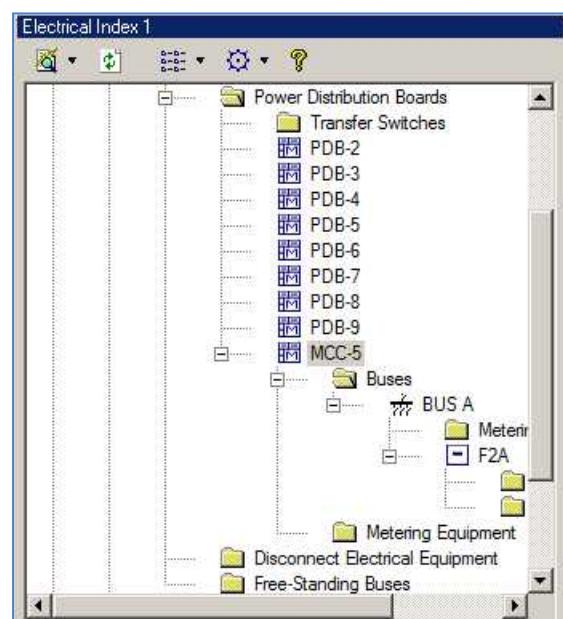
The path of the symbols for cable block diagrams is defined in Options Manager:



User can change the path to different location as he needs.

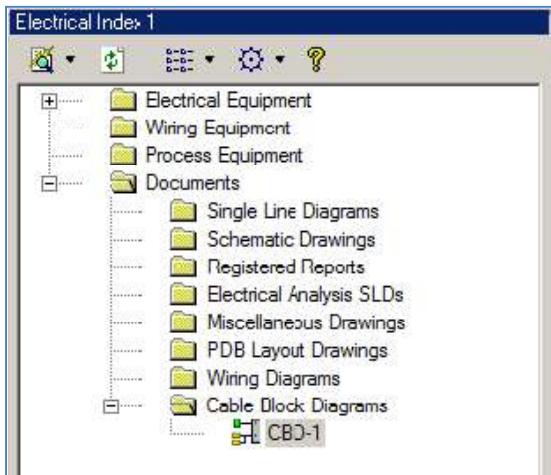
Create Equipment in CBD from typical on the fly

1. In EI create new PDB, and name it 'MCC-5'. Add bus and name it 'Bus A'. Add cell and name it 'FA2'.



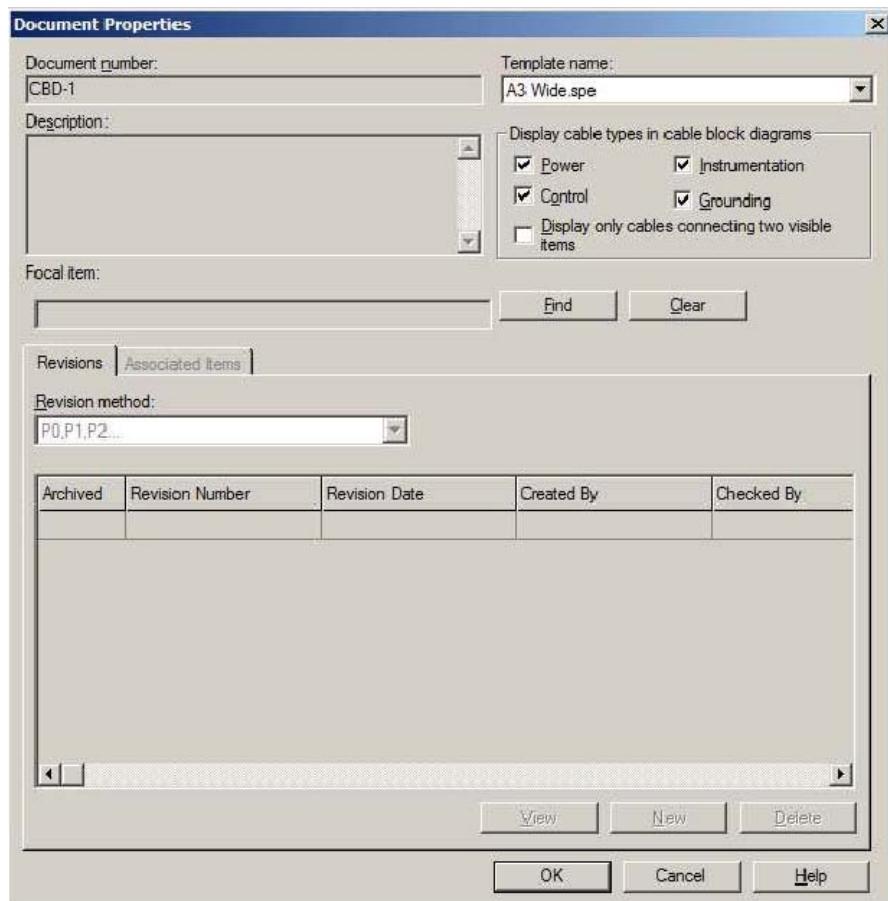
Cable Block diagrams (CBD)

2. In EI, select the cable block diagrams folder. Right click to create new cable block diagram and name it CBD-1.

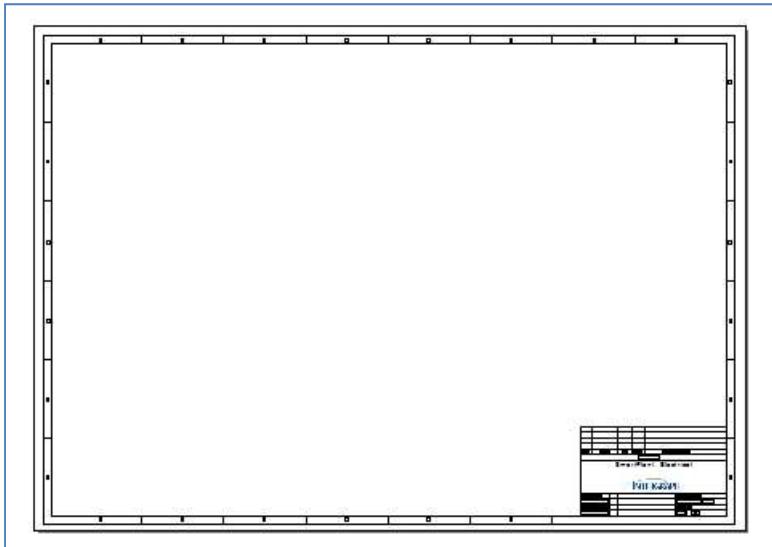


3. Select CBD-1, right click and select Document Properties.

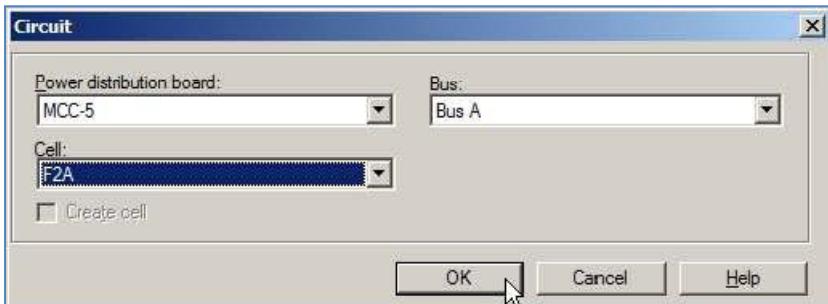
From the document properties screen you can select the template, and choose what cable category to show.



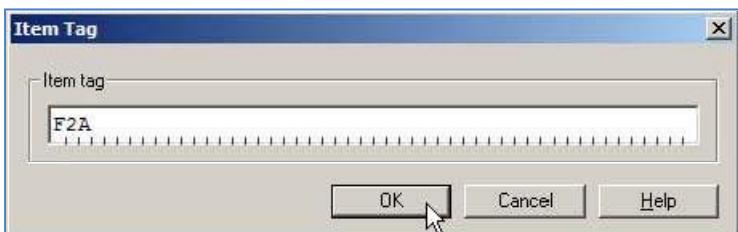
4. Select OK to close the document properties screen. Select CBD-1, right click and select open.

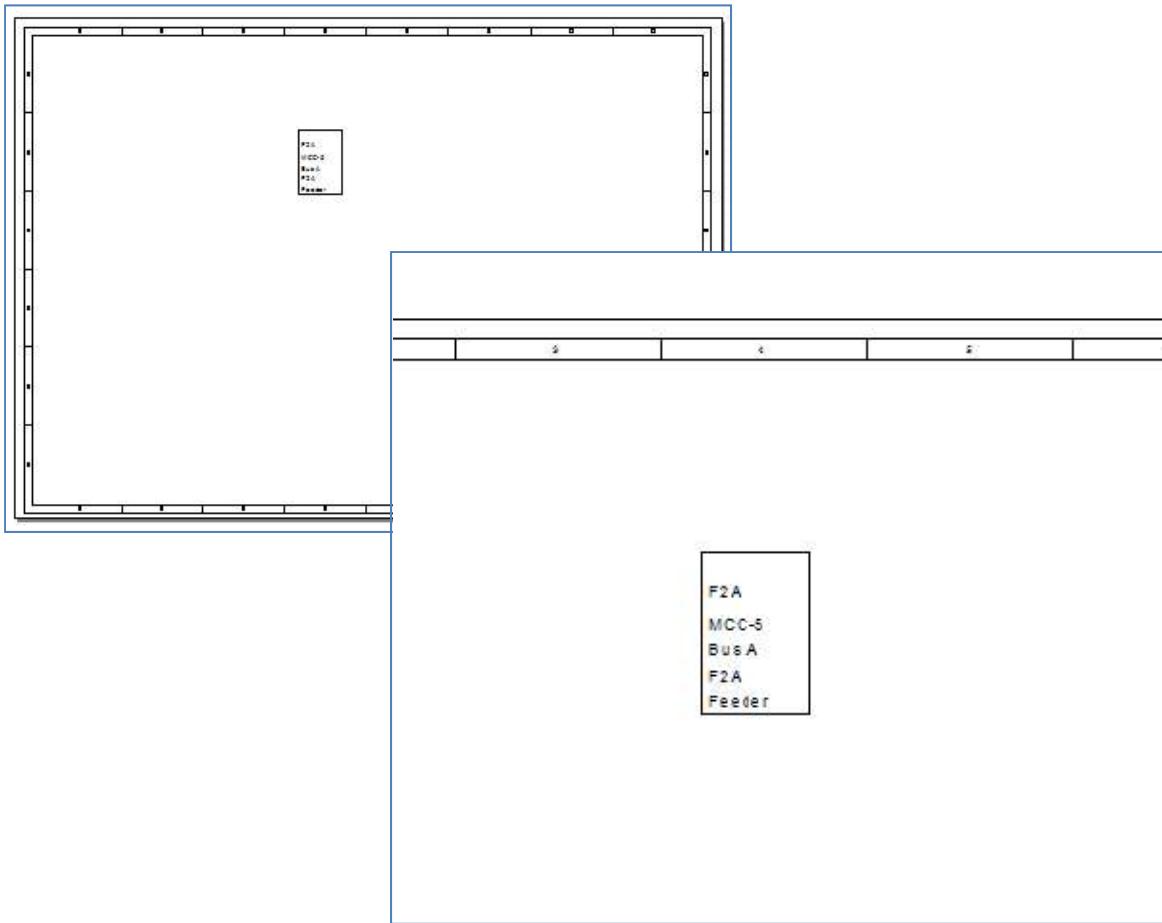


5. From RDE, drag the typical circuit 'Motor Feeder<10hp' into the open drawing. It will open circuit dialog box where you need to select what PDB, Bus and cell you want to associate the new project circuit. Select MCC-5, Bus A, and F2A and select OK.



In the circuit item tag, enter 'F2A' for the circuit name.

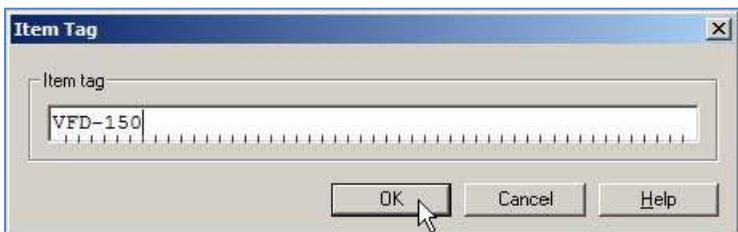


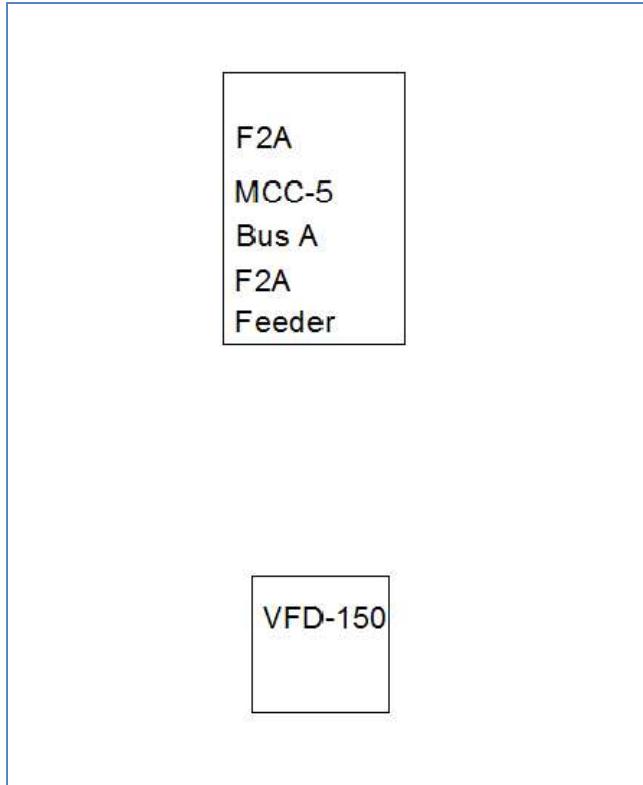


6. In RDE, create new typical VFD and name it VFD-1.

7. From RDE, drag VFD-1 into the drawing.

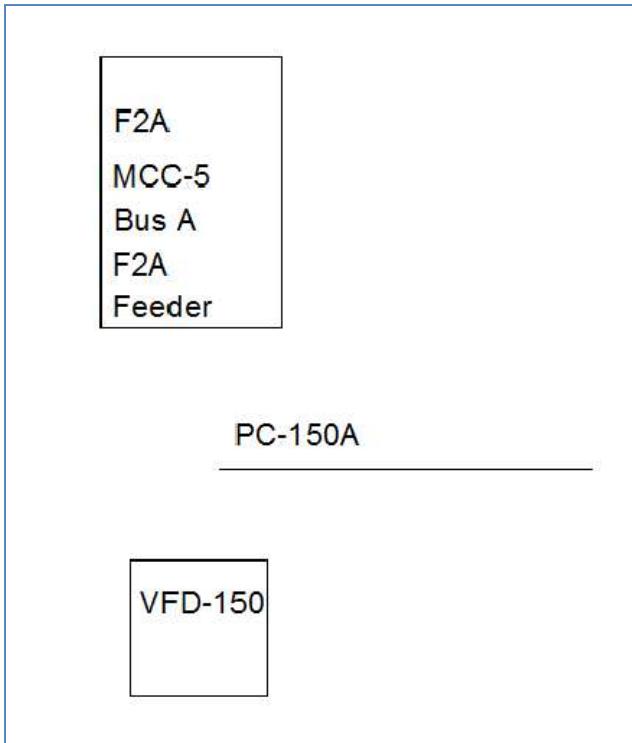
In the VFD item tag dialog box enter VFD-150.



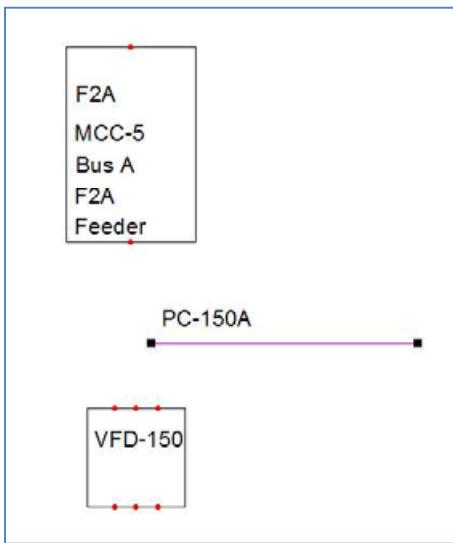


8. From RDE, drag the typical power cable, ‘3+1/C-12AWG -(Power / NEC)’ into the drawing.
In the cable item tag dialog box, enter PC-150A

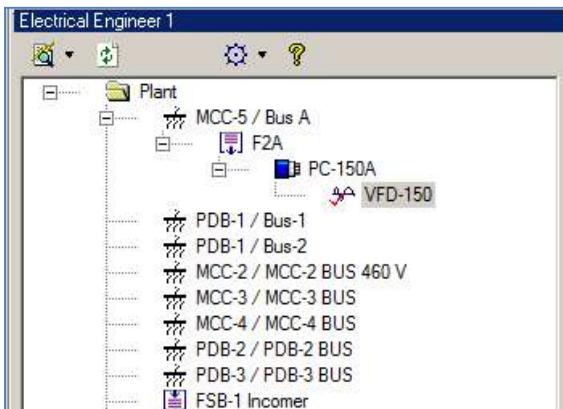
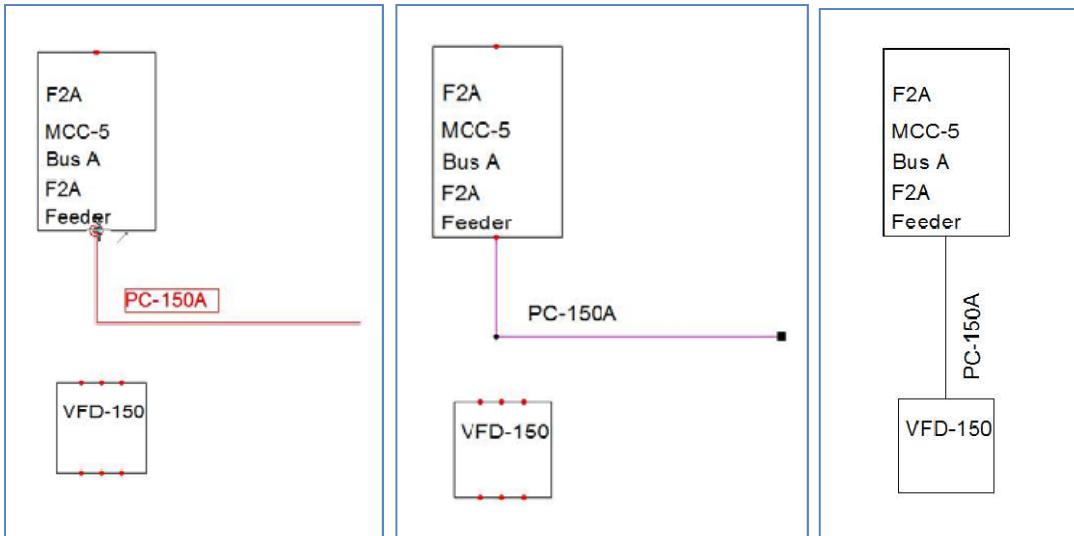




9. To connect the cable between the circuit and the VFD. Select the cable, to expose the connection point.



Select on one side of the cable, and drag it, to one of the feeder's connection point.
Repeat the same step to connect the other side of the cable.



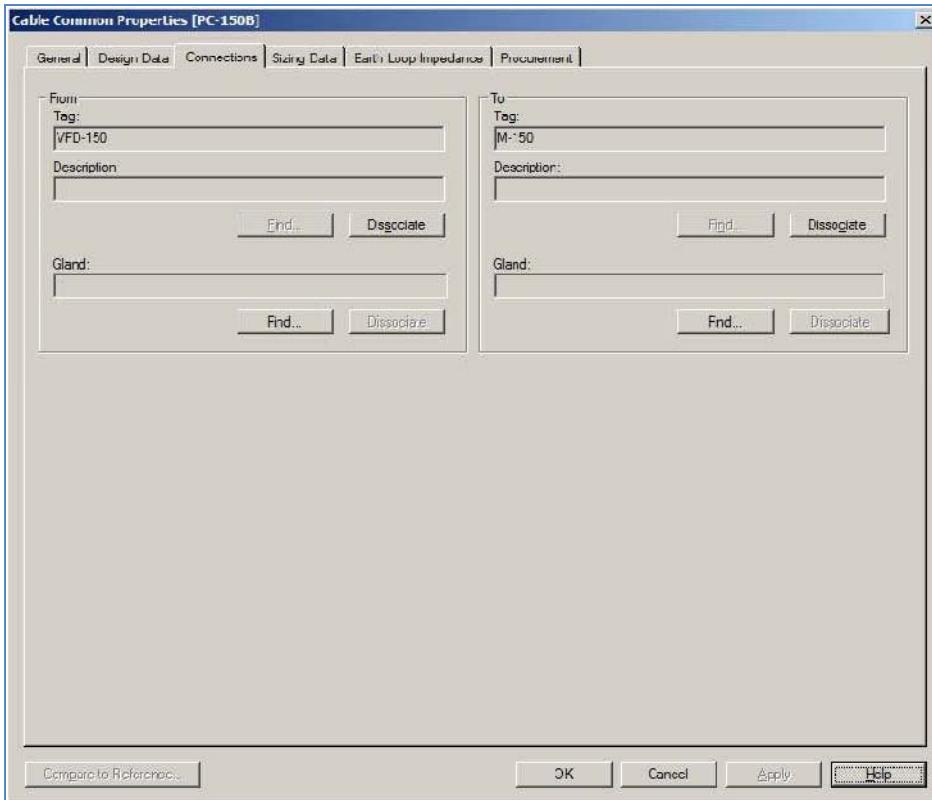
10. Save and close the document.

Note: As we saw in the above example, SmartPlant Electrical enables the user to locate a circuit as a legitimate item on a cable block diagram. Any circuit type can be dragged on the Cable Block Diagram and be connected with other equipment with cables. Each circuit can be dragged on the drawing if its Power Distribution Board is not on the drawing. On other hand, a PDB cannot be dragged on the drawing if one of its circuits is already located on the drawing. Placing circuit allow to have better granularity and ability to show only the cables that are related to a single load with its related equipment and the cables.

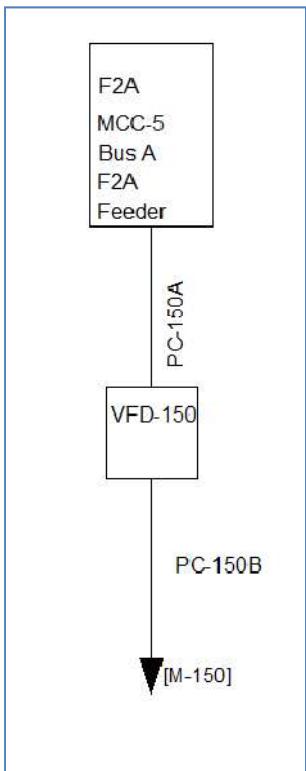
Cable Block Diagram for cables already pre-assigned

11. In EI, create new motor, 'M-150' using typical motor, 'M<10hp'.
12. In EI, create new power cable, 'PC-150B' using typical cable, '3+1/C-12AWG -(Power / NEC)'.
13. Open cable PC-150B and from connection tab make the connection from VFD-150 to motor M-150.

Cable Block diagrams (CBD)



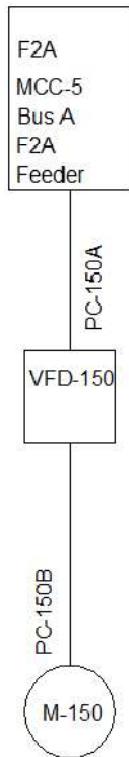
14. Open CBD-1.



The system shows the “reference to” equipment on cables that have connections at their open sides. (In the case the equipment at the open side is not in the drawing)

You may also select the macro and move it to a new place and save its position. Notice that when you select a macro, its cable gets highlighted- this would be useful in a drawing which has many cables and data and one may get confused which label belongs to what cable.

15. From EI, drag motor, M-150 into the open drawing.



16. Save and close the document.

Create and connect control station and cabinet.

17. Open CBD-1.

18. From RDE, drag to the open drawing the following typical items:

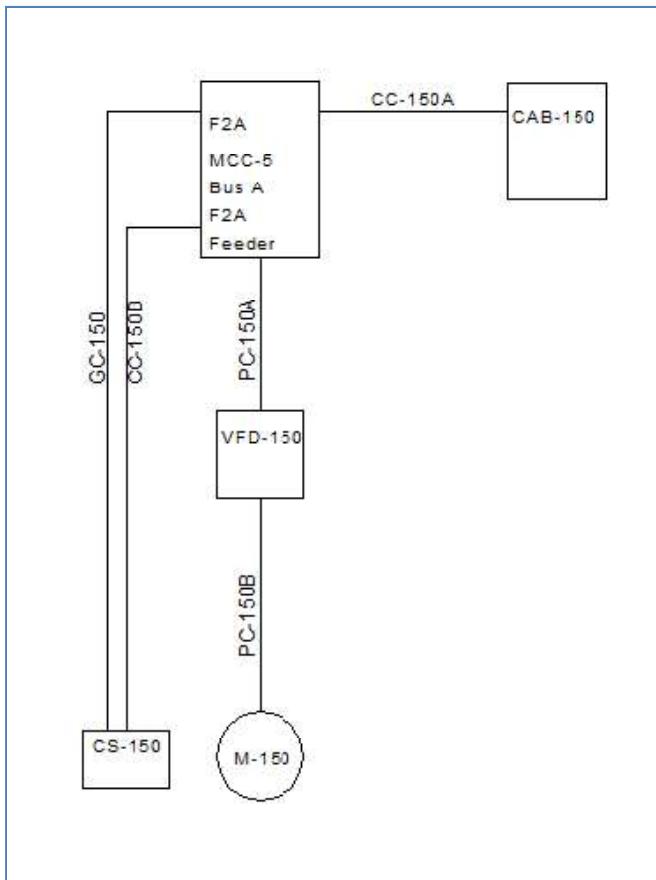
Typical Cabinet, ‘Cab-1’. **Note:** Typical Cab-1 need to be created first in RDE.

Typical Control Cable, ‘5/C- 2.5 mm²- (Control)’.

Typical Control Station, ‘Emergency Stop Station’.

Notice: The typical CS comes with one control cable, and one grounding cable.

19. Reposition, name and connect the new item in the drawing according to the following screen:



20. Save and close the document.

Special Notes

2 connection types used in CBD:

- '**Connection Point**' type, for power category cables.
- '**Non-Power Connection Point**' type, for all the other cable category types.

Selecting cables of one category will highlight only those connection types that match the cable type. Furthermore, each symbol has a definition of the connection **point side, 1 or 2** which coincides with the power flow of the point. Side 1 means the point at which power flows in, and side 2 corresponds to the point at which power flows out.

The system allows you to connect to each of the points, based on the SmartPlant Electrical rules for cable sides, depending on the cable category. Power cables have the convention of side 1 being the side at which power flows in (connect to the upstream) and side 2 corresponds to the side at which delivers power, exiting the cable.

For non power cables, there is no such limitation and each side of the cable can connect to any item type.

Connecting power category cables

The system has been designed smartly so that it identifies if the cable should be connected to side 1 of the cable or side 2, based on the item type, as the system behaves in the Electrical Engineer

Trying to violate or connect to an invalid point type will prompt a warning message or prevent you from connecting the cable against the rules

Only connection points will be illuminated when selecting power cables to connect

Connecting non power cables

From the Cable form - Connecting a non power cable from the form will follow the following rule:
Side 1 of the cable (the “from” side) will be automatically connected and presented in the CBD connected to the first available non power connection point type side 2

Side 2 of the cable (the “to” side) will be connected and presented in the CBD connected to the first available non power connection point type side 1

Trying to violate or connect to an invalid point type will prompt a warning message or prevent you from connecting the cable against the rules

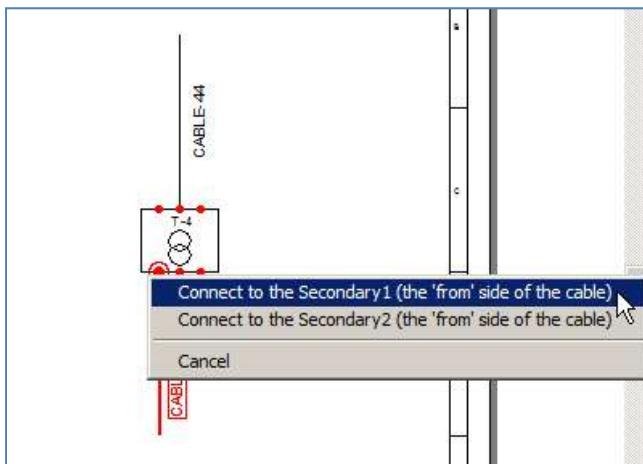
Only non power connection points will be illuminated when selecting non power cables to connect

Connecting the cables in the CBD - Connecting non power cables on the graphical environment will be done according to the selected connection point side. User selects the equipment he wants to connect the cable to, and depending on whether he selects a non power connection points side 1 or 2, the cable will be connected to the “to” or to the “from” side of the cable. Trying to connect the other side of the cable to a point type similar to the other side already connected will not be allowed and the system will warn you with messages say, “The ‘To’ side of cable ‘Cable-1’ is already connected to another equipment”.

Connecting to Transformer

Transformers are special type of items as they include also one or two related transformer components. Due to this special item type the software has been designed accordingly, to cope with this situation.

In the transformer symbol for CBD, connection point with *side 1* is interpreted by the system as the connections for the primary winding. Connection points defined as *side 2* are interpreted as being defined for the secondary and, if the case is a 3 winding transformer, since the system does not know to which of the secondary the user wants to connect the cable there will be a system prompt that the user needs to confirm:

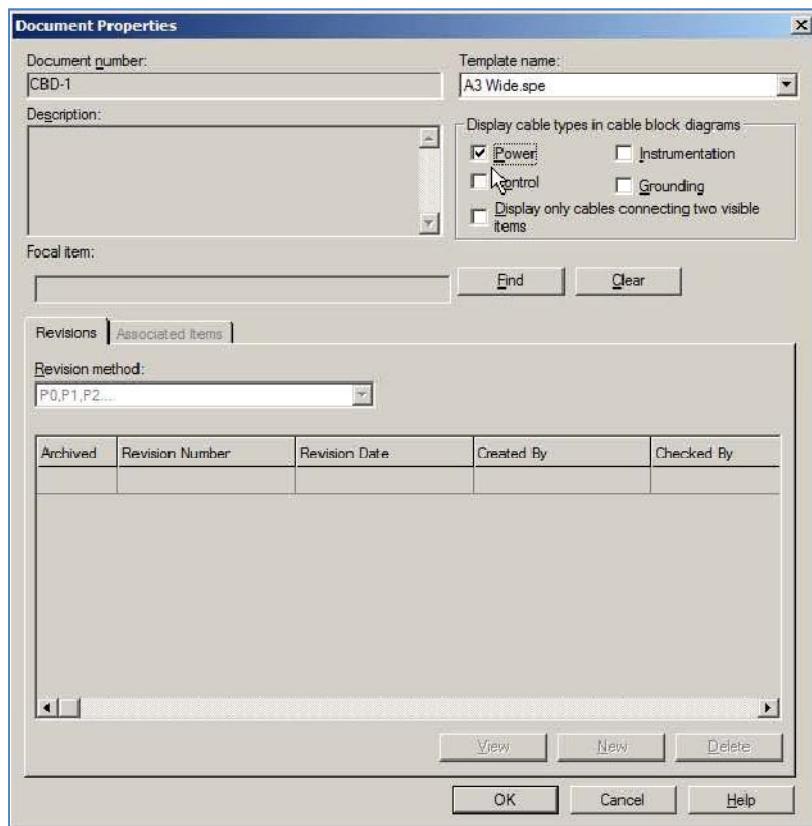


Filtering the CBD on cable category

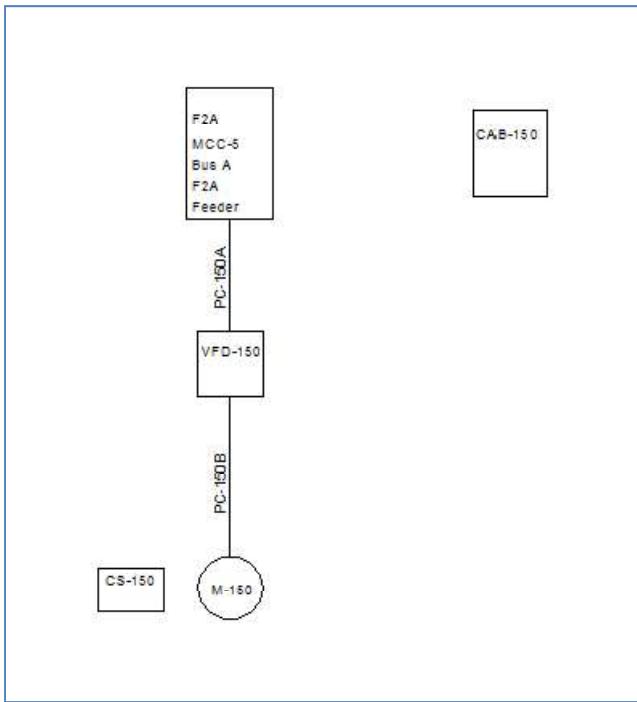
From the document properties, you can choose the cable types to be displayed in your cable block diagram. You can change the cable type at anytime by opening the **Properties** dialog box and selecting or clearing the relevant check boxes. Selecting a particular check box displays the cables of that type on your cable block diagram. The available options are:

- **Power**
- **Instrumentation**
- **Control**
- **Grounding**
- **Display only cables connecting two visible items** — With this option selected, cables of the selected cable types are displayed in the cable block diagram only if they connect between two items that are actually visible in the drawing; if only one of the items is visible, the cable is not displayed. Clear this check box to display all cables of the selected types.

1. Right click on cable block diagram CBD-1 and select document properties.
2. In the document properties windows, select only the ‘Power’ ,‘Display cable types in cable block diagram’ section.



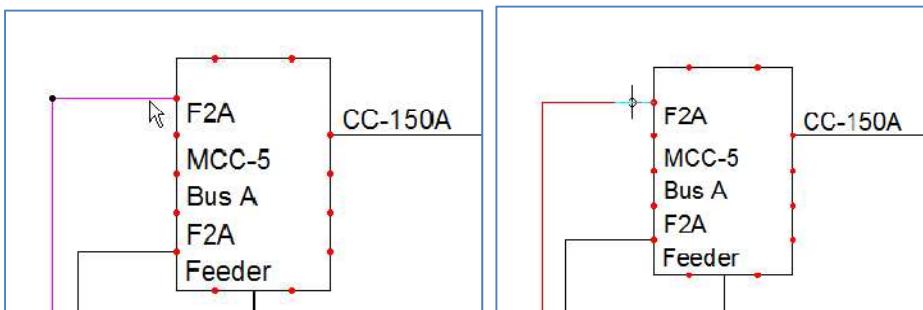
3. Open the cable block diagram CBD-1.

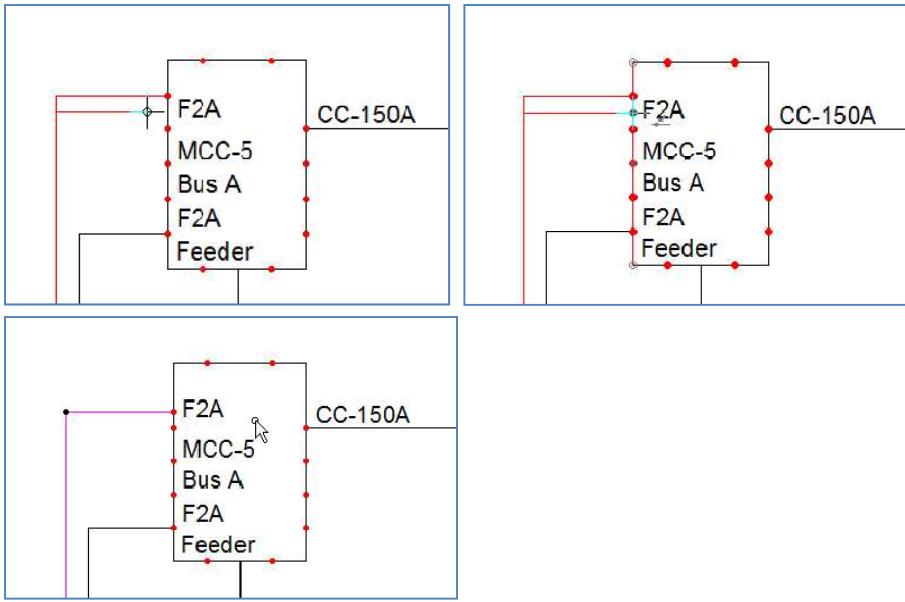


4. While the drawing open. Bring up the document properties, and change back the filter to the default (all the 4 cable categories are selected). Select OK to close the document properties.

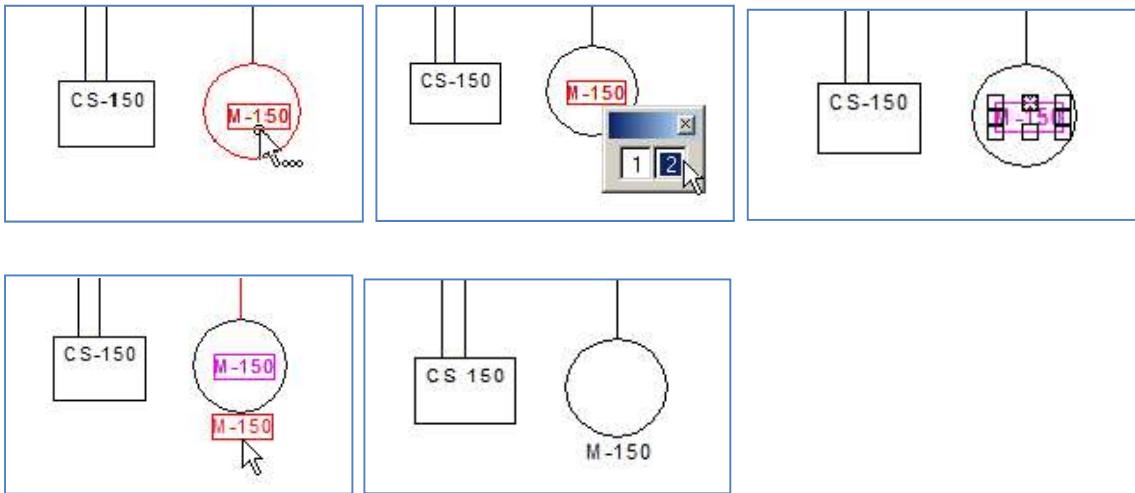
Reconnect cable in a different place along the line

Allow you to reconnect the cable in different place along the symbol line.



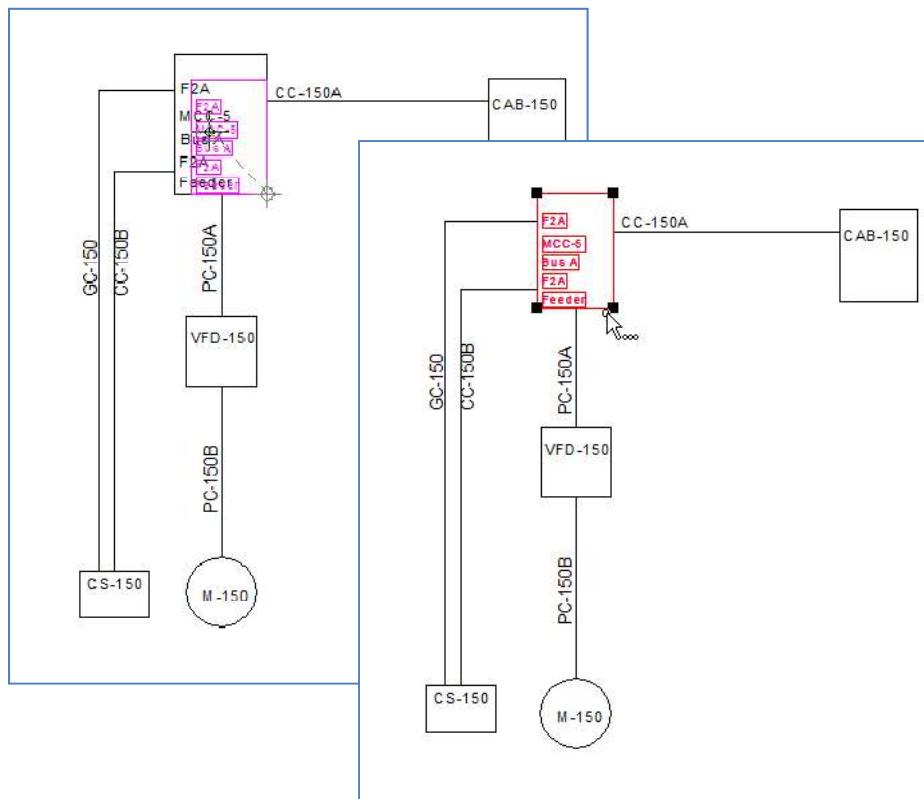
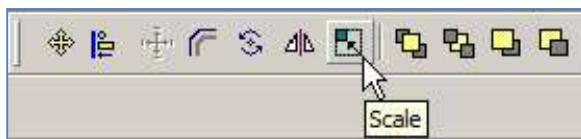
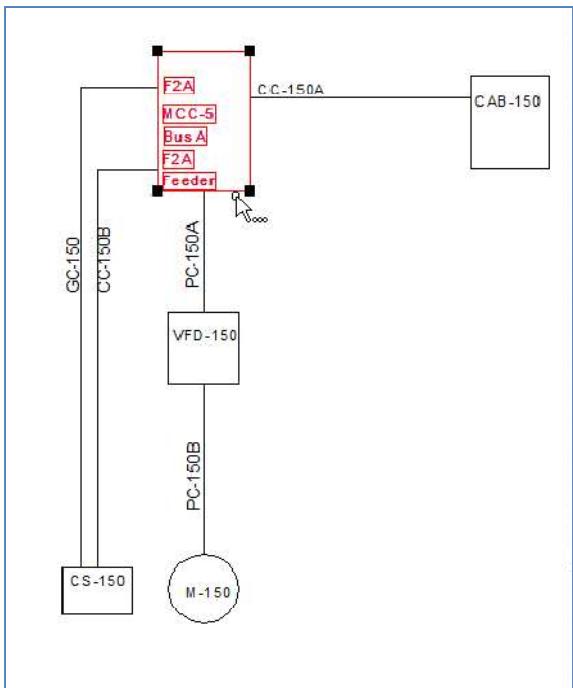


Move macros



Scale a symbol

Allow you to scale symbol in CBD drawing, and keep the new scale.



Print and Save option

User can print out CBD document.

User can save the document also as an external file.

The save as format could be one of the following:

*.SPE (can be views with SPEL or SSK)

*DWG

*DGN

*DXF

*DGN

Annotations and Redlining

You can mark up CBD document using text or symbols as annotations. The software saves annotations in the same layer as the graphical elements of the drawing.

Redlining appears in a separate drawing layer and you can hide and display redlining as you require. The software saves the drawing with the redlining items. When opening a drawing 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.

Revision

The software allow you to add revision to CBD document.

The software also allow you to use global revision for CBD documents and perform a number of revision activities for a group of selected documents.

Compare an Archived Document with a Current Document

The software allows you to archive cable block diagrams 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.

Creating symbols for cable Block Diagrams

The creation of CBD symbols is made in the Catalog Manager, as follows.

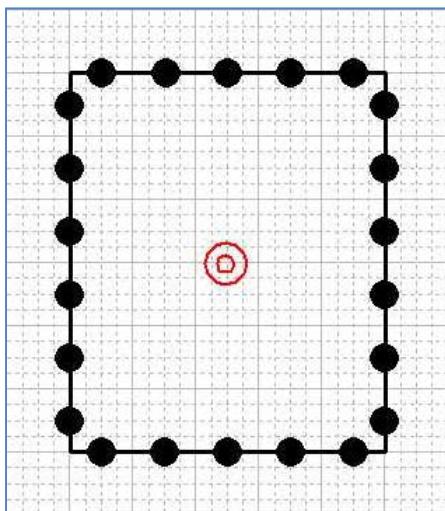
Like any symbol created with Catalog manager the process has 3 steps:

- The graphical representation
- The Connection points definition
- The properties definition (Smart text)

The graphical presentation of the CBD symbol is similar to the graphical presentation In SLD symbol.

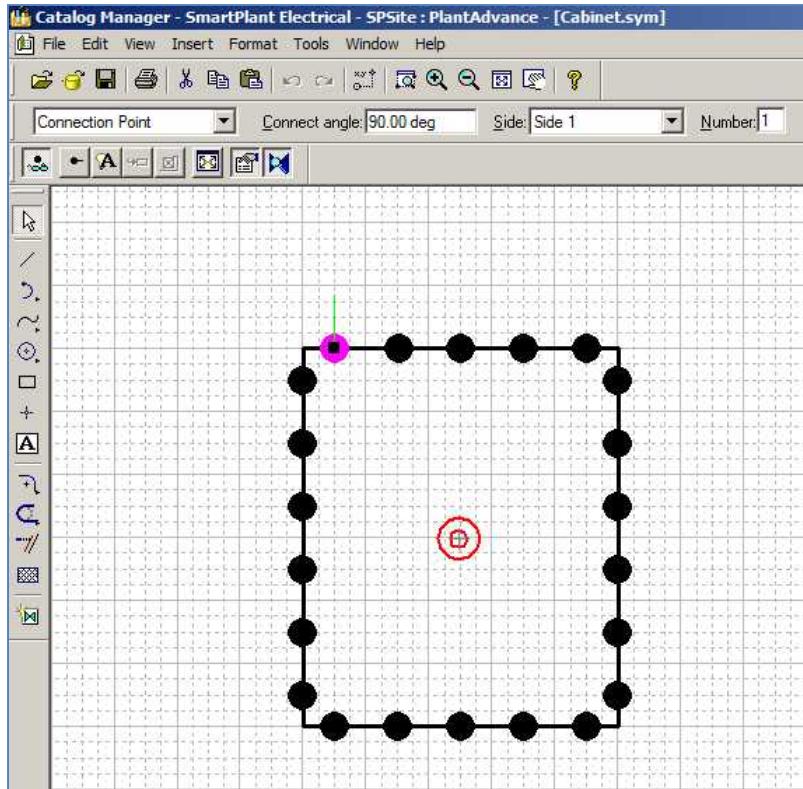
Defining the Connection point types

Cabinet.sym



Notes:

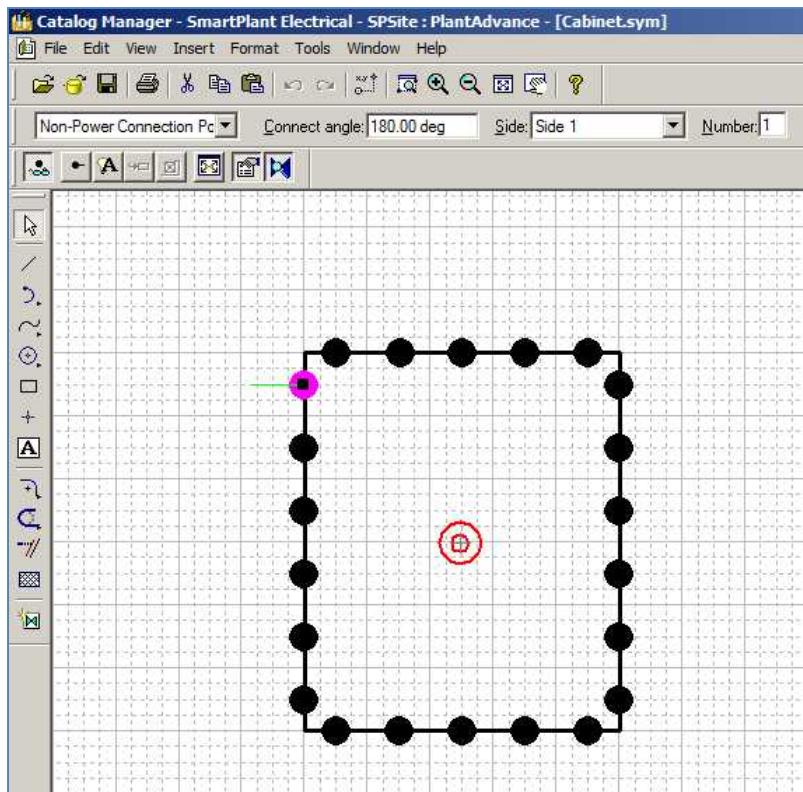
- 1- Connection points for CBD for power cable category should be of regular “connection point” types



The Connection side defines whether it is a point to which cables enter or exit and will connect their sides according to the convention used in SPEL:

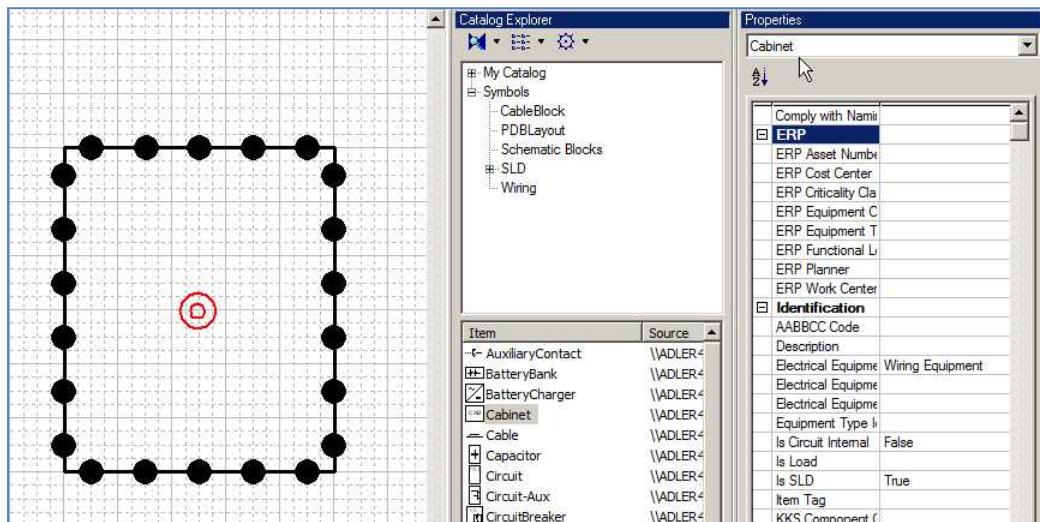
- **Connection point 2 of the symbol** is the point at which cables will be exiting, thus, convention is that Cable side 1 (the “*From*” side) connects to the Connection point side 2 of the symbol. Defining connection point side as 2 would mean that you expect that these are the points that the Feeder/outgoing cables will be connected (power flowing out of these points, incomers circuits of PDB)
 - **Connection points 1 of a symbol** is the point at which cables will enter, or power enters. Defining a connection point as side 1 would mean that it is an incoming connection point, for incoming cables. (incomers of a PDB, etc)
 - The Index number for that point type and side has its own sequencing from the other types and sides
- 2- Connection points to connect to non power category cables will be of “Non power point types:

Cable Block diagrams (CBD)

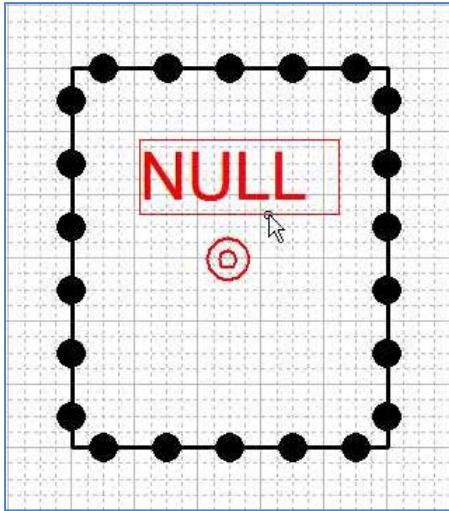


The same rules for the side apply here as well. Also, these points will be serving only non power category type of cables.

- 3- Symbols, created for CBD in the catalog manager should be associated with the proper item type. Unless this is so, the system will not allow associating this symbol to the relevant item tag.

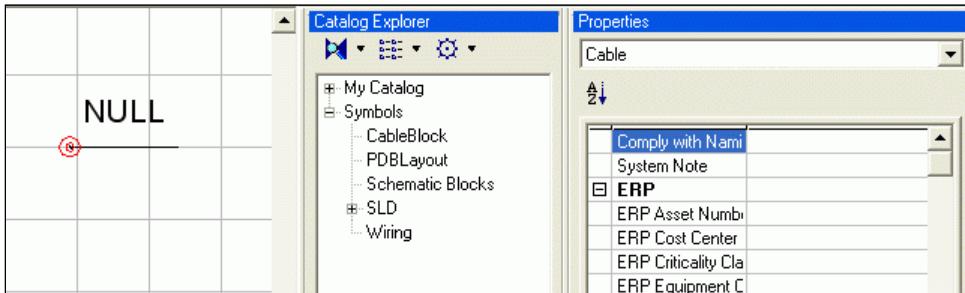


Adding the item tags as Smart text done similar to smart text in SLD symbol.



Customizing the cable, presentation along with their macros

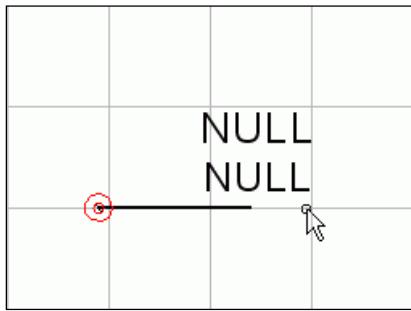
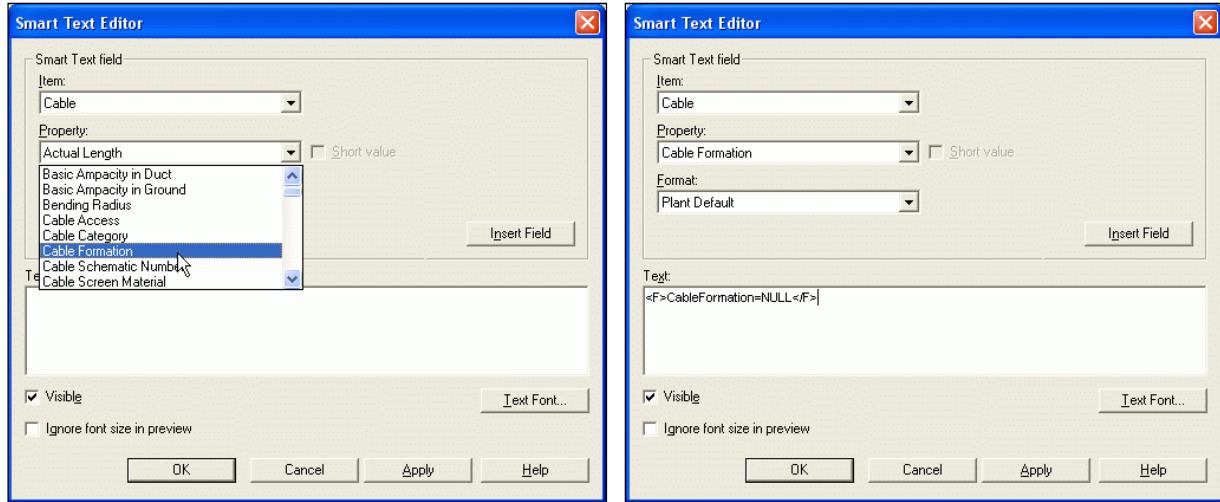
Cables are created in the Catalog Manager and graphically presented by a line that may have certain predefined style properties. There is very little control on the presentation of a cable in the CBD world except for the set of macros associated with it, and the style that can be defined in the symbology within the definitions of the Options Manager.



The macros that user wants to see on the cable are defined using the Smart text and the position is based on macros' distance from the origin point, as shown above.

Let's customize the cable symbol by clone the cable symbol and name it 'CableCustom'. Open the new 'CableCustom' symbol and add the macro 'Cable Formation', and position it below the 'Item Tag' macro.

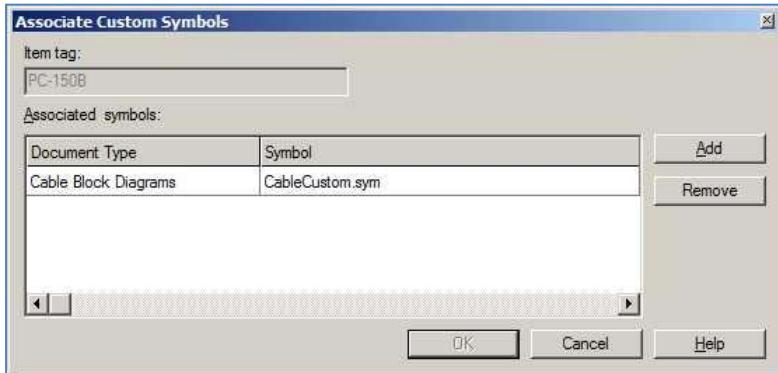
Cable Block diagrams (CBD)



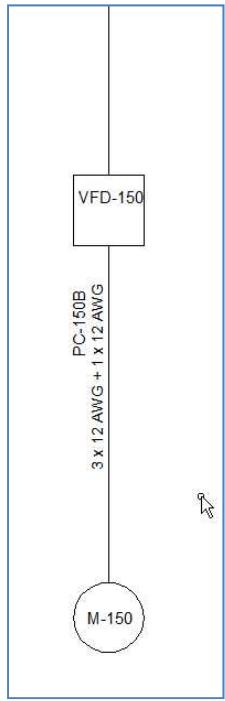
Save the change and close the symbol.

Note: The cable macros are placed relative to their connection point. For a manual connection in the Drawing, the macro will be relative to the first made association.
For an automatic generation, when cables have been assigned upfront, the origin is assumed to be the Side1 (hard coded).

Let's associate the new symbol with the power cable 'PC-150B' and open CBD-1



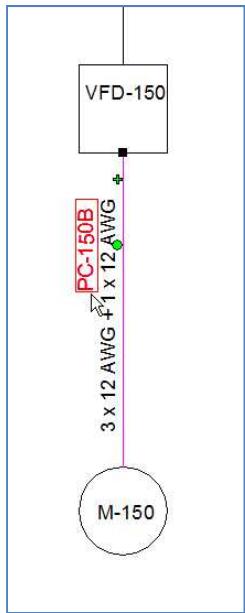
To see the new cable macro, create new CBD-2 and drag M-150 and VFD150 into the drawing.



Move/Mirror/Rotate macro

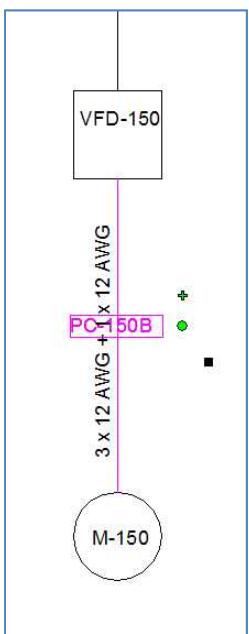
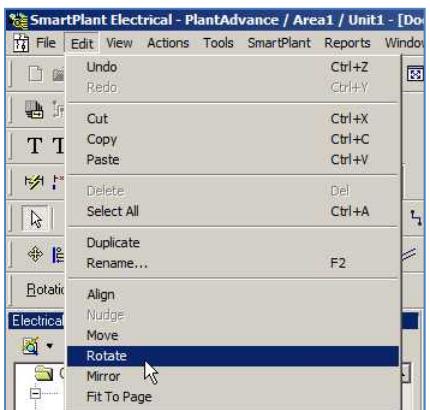
From the edit menu you can move, mirror or rotate a macro.

For example. To rotate the cable PC-150B item tag, select the macro



And from edit menu select rotate, and rotate it in 90 degree.

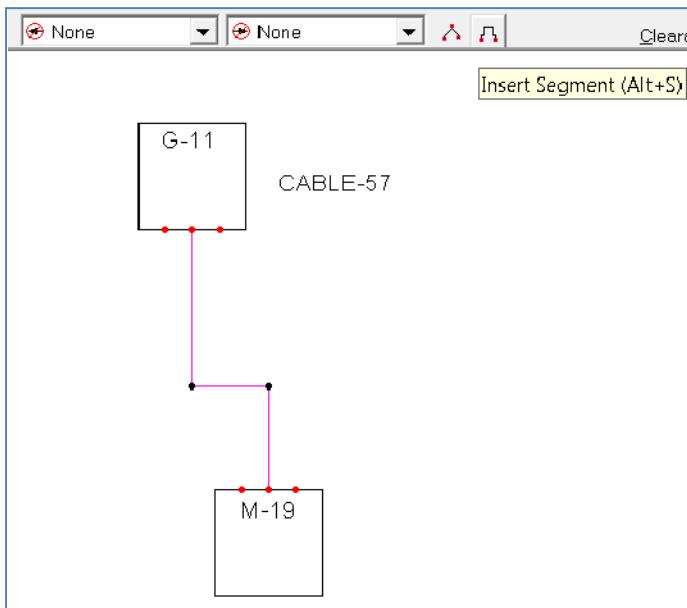
Cable Block diagrams (CBD)



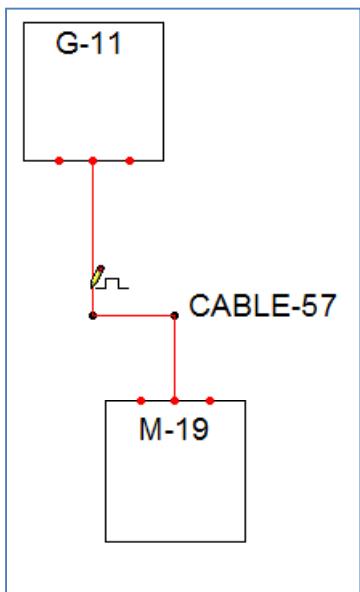
Add CBD items

Insert segment

Inset segment command allow you change the cable display in the CBD by dragging a part of the cable making 90° connections.

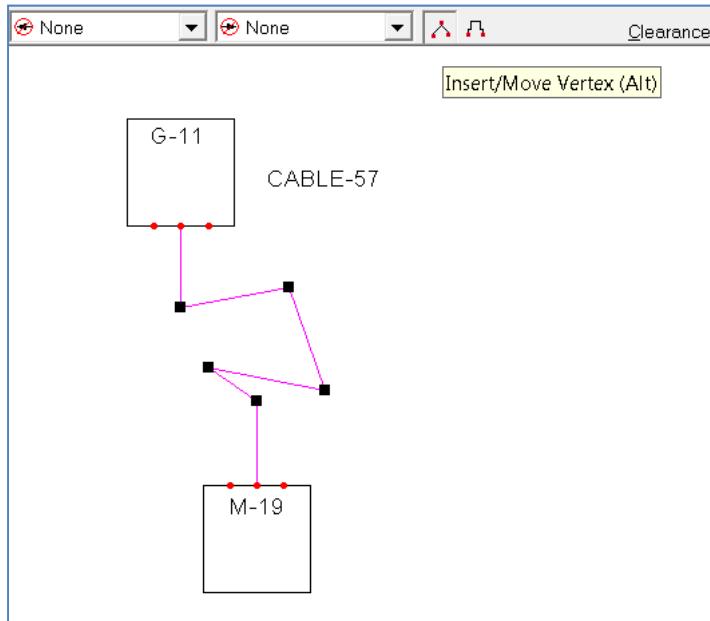


Moving the mouse over the cable displays a special cursor



Insert Move Vertex (Alt)

Insert Move Vertex (Alt) command Allow you change the cable angle.

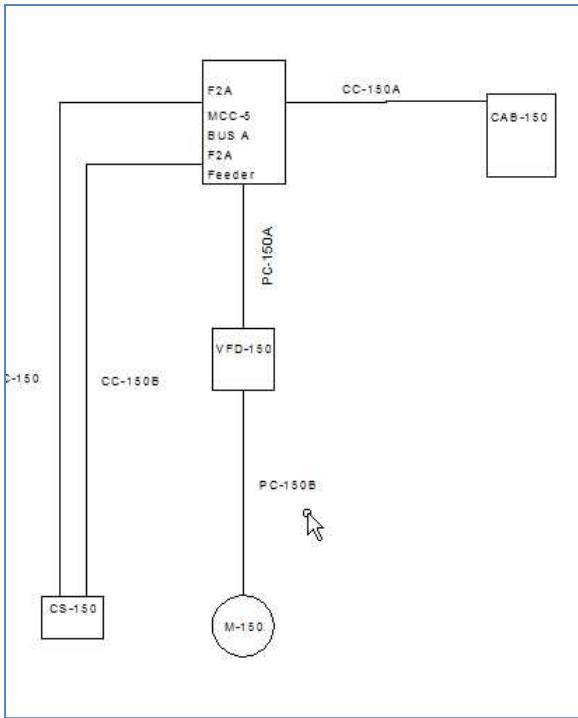


Identifying and Resolving Symbol Inconsistencies in an Existing CBD

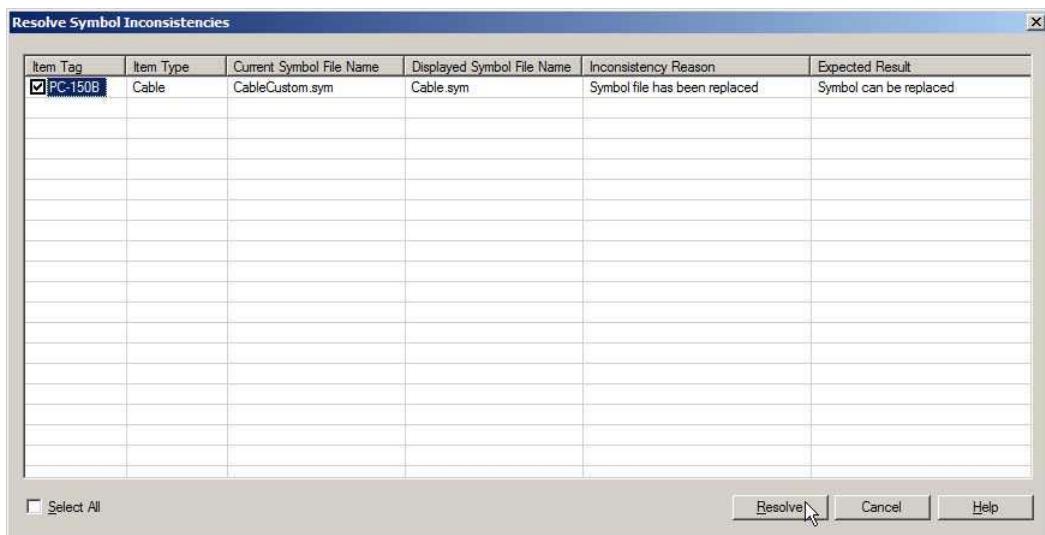
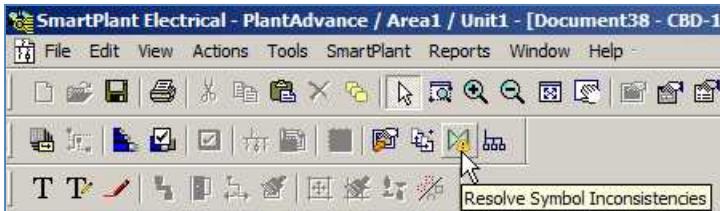
Since CBDs are saved in the database, the user needs to validate any changes that were made to the symbols in the existing documents. These symbol inconsistencies need to be identified and resolved so that the existing CBD drawings are kept up-to-date. Symbol inconsistency is caused by setting a custom symbol for any item on the drawing or by modifying the symbol itself.

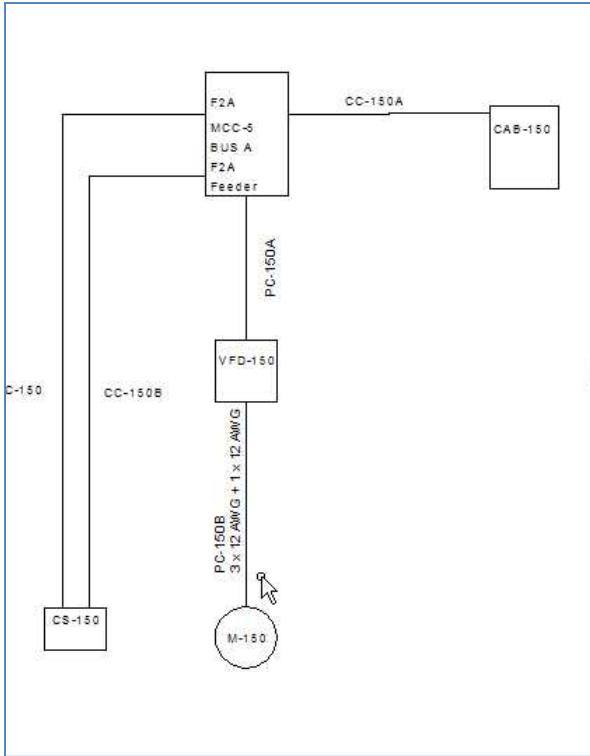
Example: Open CBD-1, which original symbols have been used.

Cable Block diagrams (CBD)



Cable **PC-150B** has been associated with new symbol. To identify the resolving the symbol inconsistencies select ‘Resolve symbol Inconsistencies’.



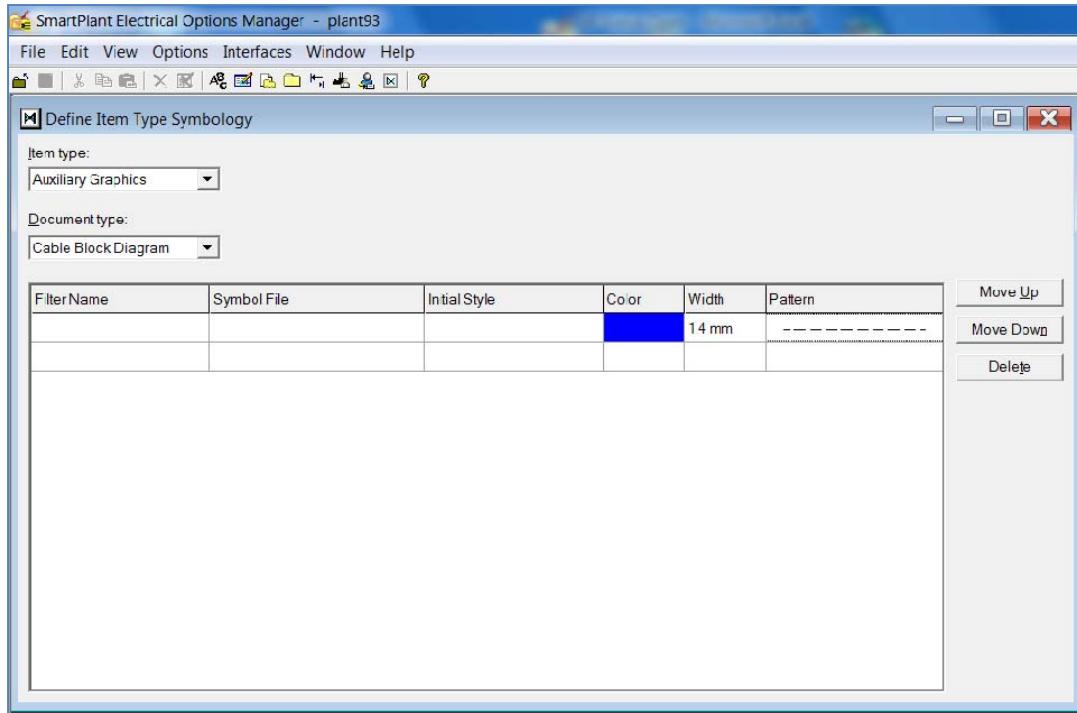


Distinguishing Smart Data from Auxiliary Graphics - Changing the Style/Layer

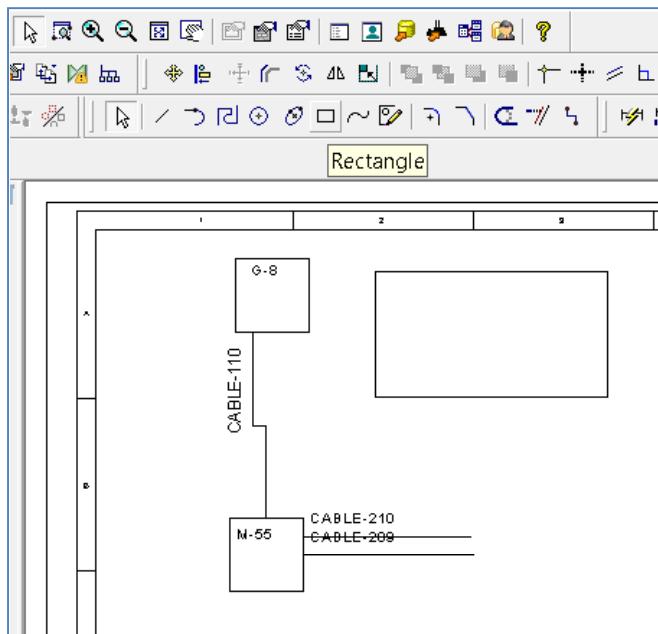
This feature provides an easy way to distinguish auxiliary (i.e., dumb) graphics from smart data. The user can add graphic items and text to a CBD and them display them on the fly in a different style and color.

The first step is to define a style In Options Manager for Item type Auxiliary Graphics and Document type Cable block Diagram.

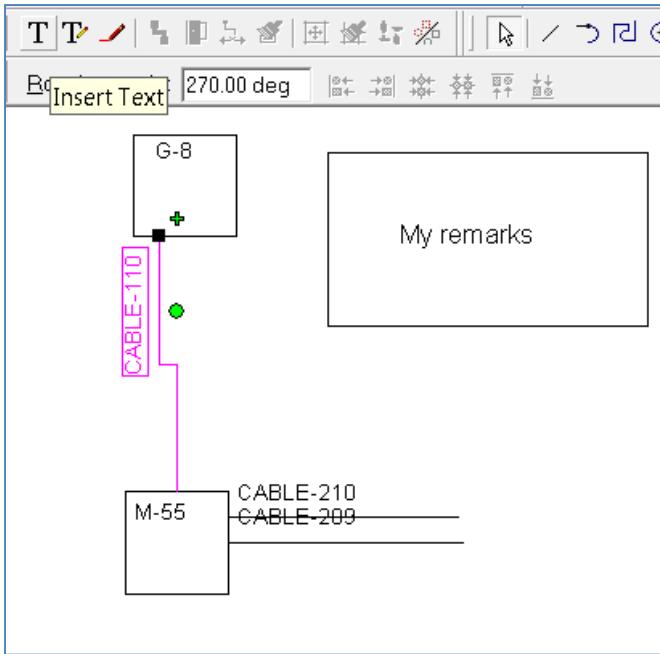
Cable Block diagrams (CBD)



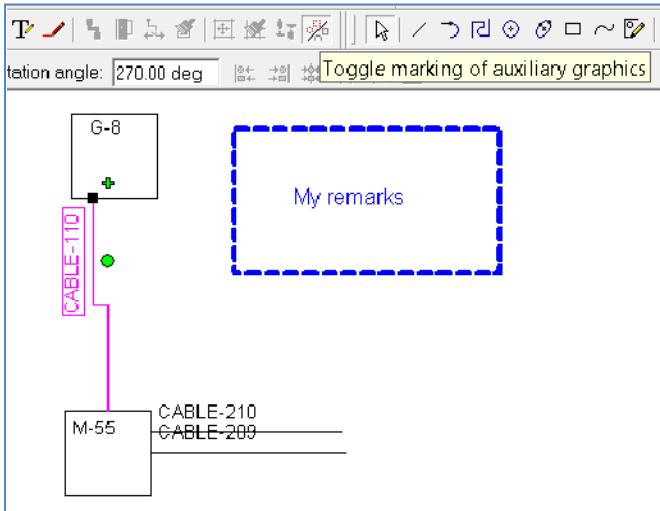
Insert into a CBD an auxiliary graphic, for example, a rectangle.
Click on the Toggle that marks auxiliary graphics – the item will get the style from the symbology.



Add a remark using **Insert text**:



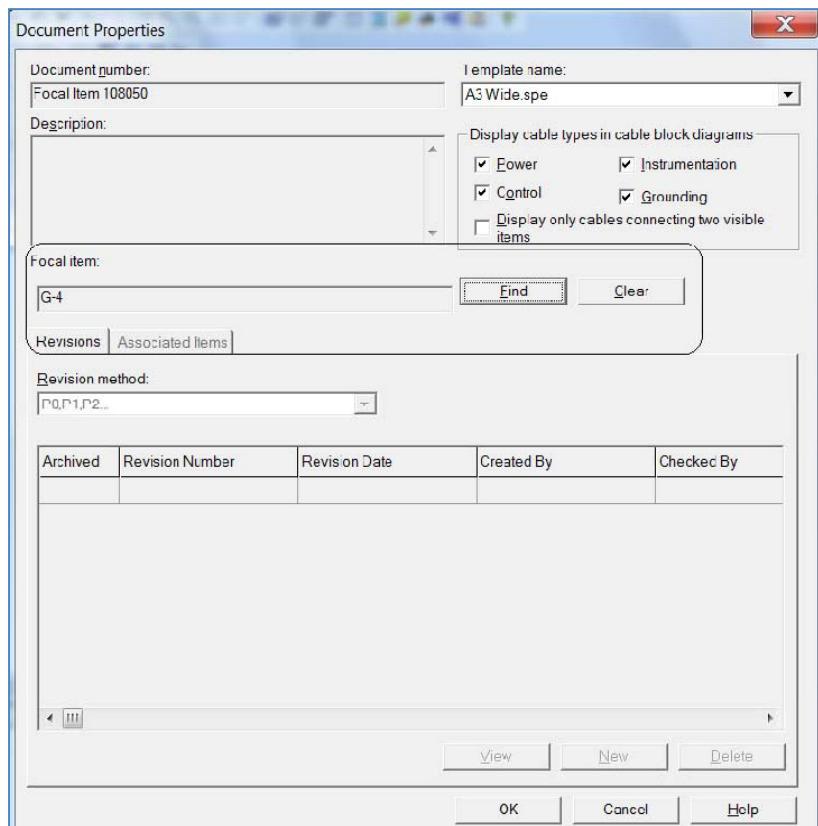
Click the **Toggle marking of auxiliary graphics** icon and the software will display the rectangle and its text using the special style.



Associate a Main item with a Project CBD

A Cable Block Diagram can be associated with a main item: electrical, consumer, power source, equipment (Also process) etc. This option is not mandatory. This main item can be associated with several CBDs. In the document properties of a CBD, there is a new group box **Focal item**. Use the **Find** button locate the main item.

Cable Block diagrams (CBD)



Wiring Diagram

Wiring diagram let you produce dynamic cable termination diagram.

The wiring diagram capability includes:

- Create terminal strips for all type of equipment, where applicable.
- Create, within the terminal strips, terminals
- Connect and terminate cable wires between equipments
- Manage both cable and conductor sides 1 and 2 (wires have 2 sides, each side can be labeled differently)
- Create wiring diagrams
- Creation of termination reports
- Retrieve connectivity data into schematics

Currently, the wiring diagram deal with interconnections between equipment, external to PDBs or the main Equipment. The cable termination reach up to and including the circuit strip.

Internal circuit wiring handled by typical schematic.

Symbols for Wiring Diagram

The wiring diagram symbol path and the default template file for wiring diagram are defined in Options Manager.

Wiring diagram symbol path	\ADLER4500\SPFiles\SPSite\PlantAdvanceRefData\Symbols\Wiring
Default template file for wiring	\ADLER4500\SPFiles\SPSite\PlantAdvanceRefData\Templates\A3 Wide.SPE

The system tries to present the wiring symbol of an item on a wiring diagram by first searching to see whether there is a custom symbol for the wiring of the object. In case there is not, the system will pick the symbol as defined in the symbology, for that specific item type, in case no symbol has been defined for the item type, it will select the default symbol as defined in the "wiring symbols" directory.

SPEL is shipped with default wiring symbols for terminal strip, terminal and first terminal. This will allow generating wiring diagrams out of the box without any further customization.

The minimum set of symbols required is: first terminal and terminal, cable, set and conductor (SPEL ship these as defaults).

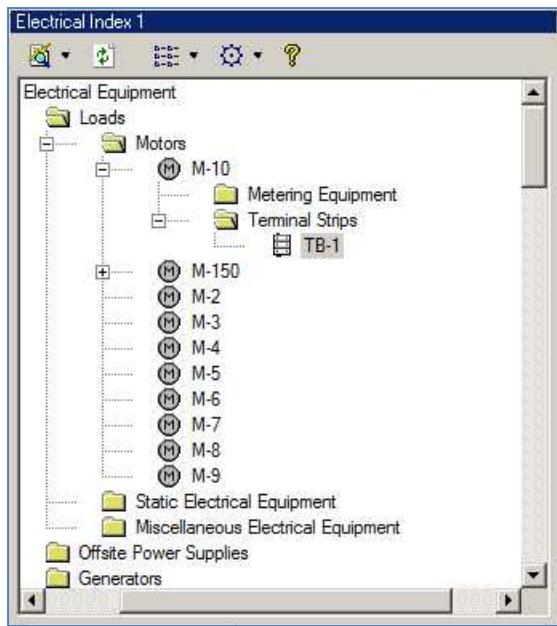
SPEL can terminate **only** those cables that have already been associated to equipment at their sides.

Creating terminals and terminal strips in equipment

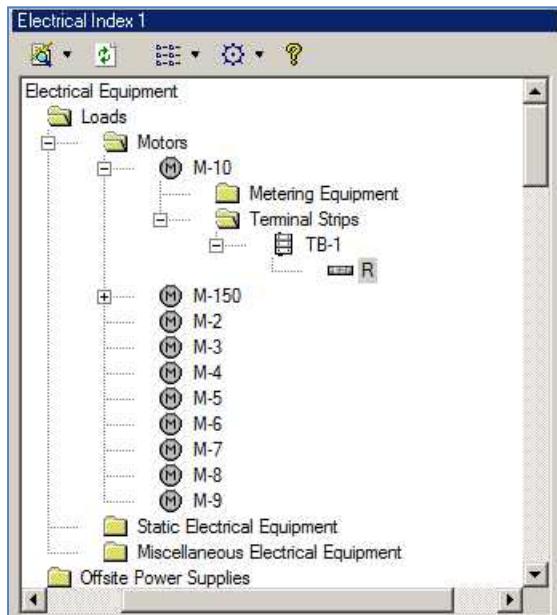
Individual terminals creation in Index

1. From EI, select the terminal strips folder of motor M-10
2. Right click to create new terminal strip, name it TB-1

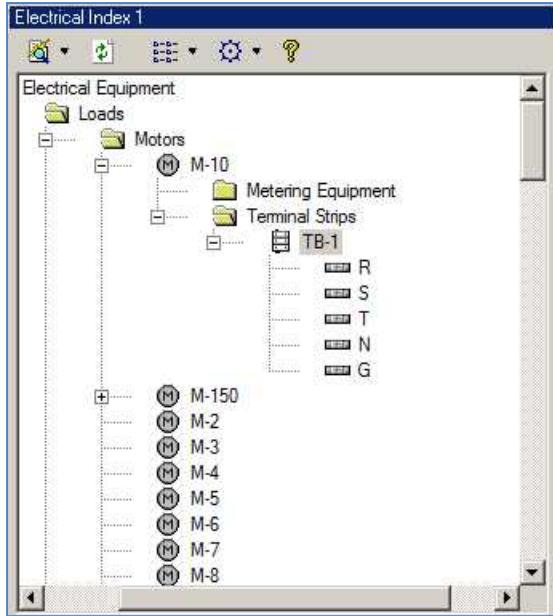
Wiring Diagram



3. Right click to create new terminal, name it R

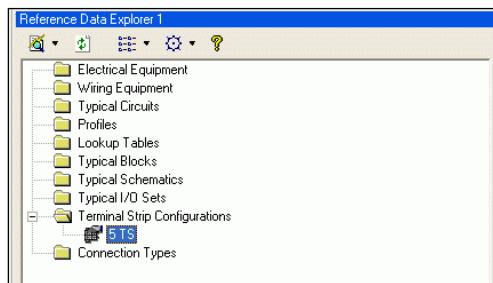
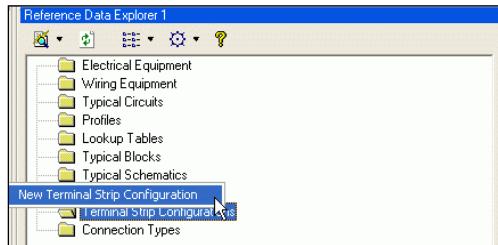


4. Add another 4 terminals.



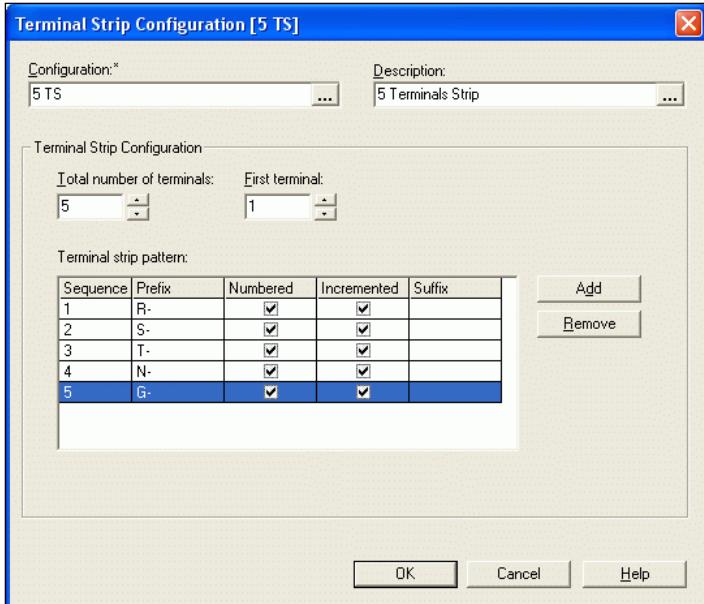
Using typical strip configuration in RDE

1. Create new terminal strip configuration, '5 TS' in RDE.

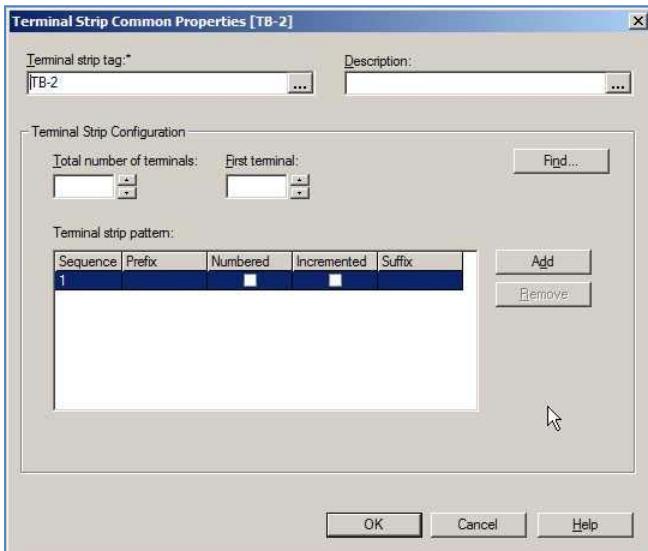


2. Right click on the new configuration, select common properties, and type in the following configuration.

Wiring Diagram

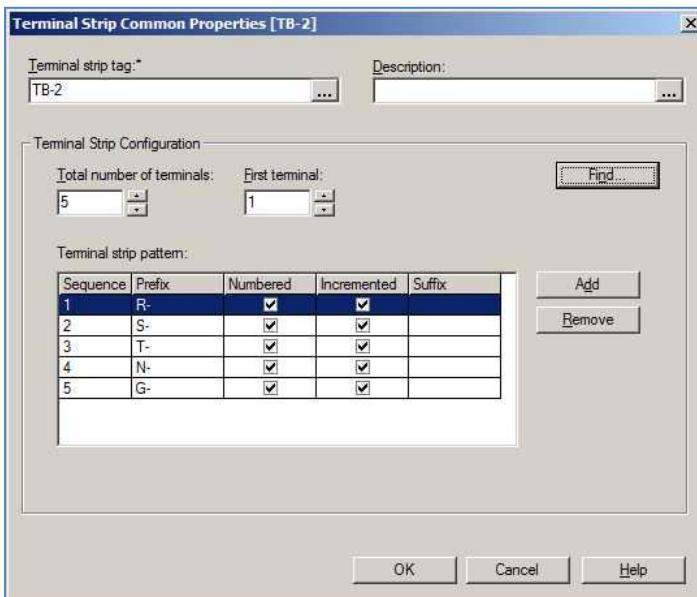
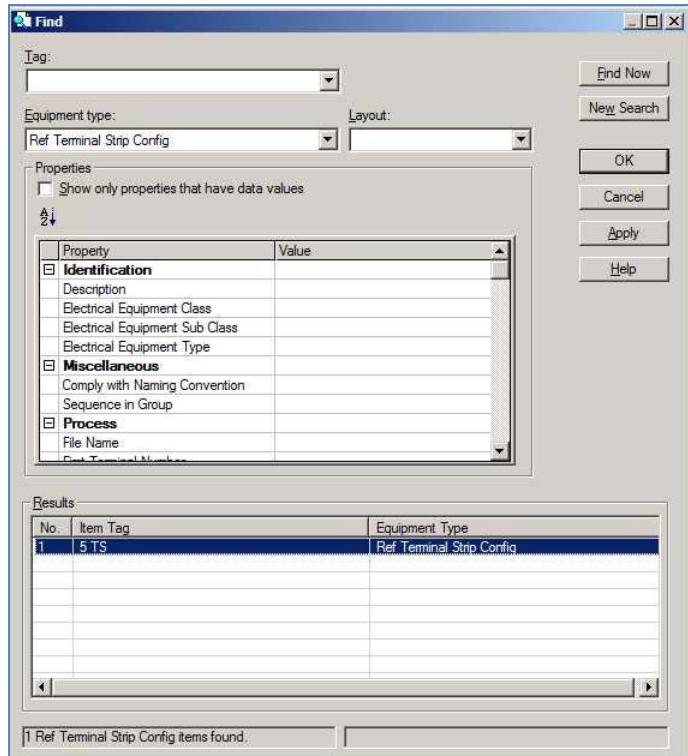


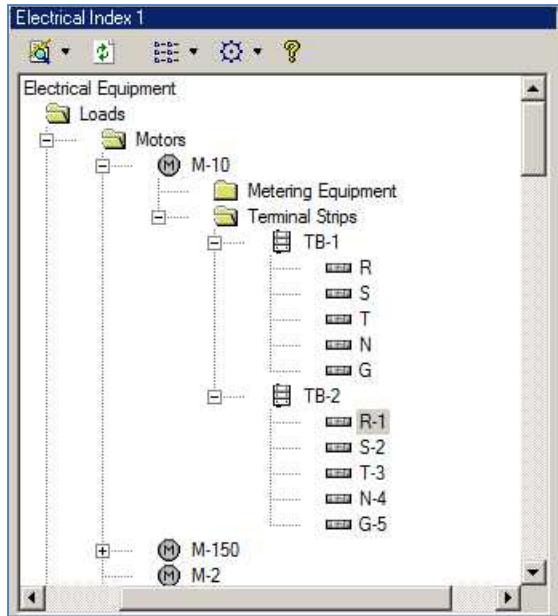
3. Create addition terminal strip for M-10 and name it TB-2
4. Right click on TB-2 and select common properties.



5. Select the find button in the common properties screen.
6. Find the new strip configuration we defined in the RDE and select it.

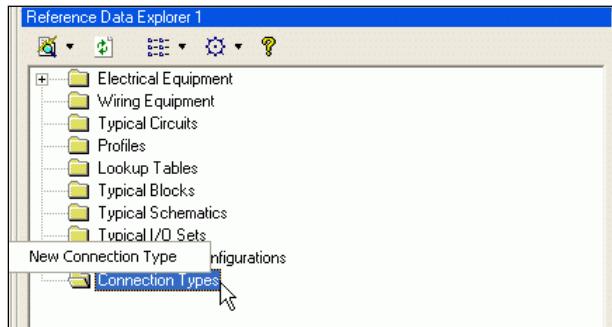
Wiring Diagram



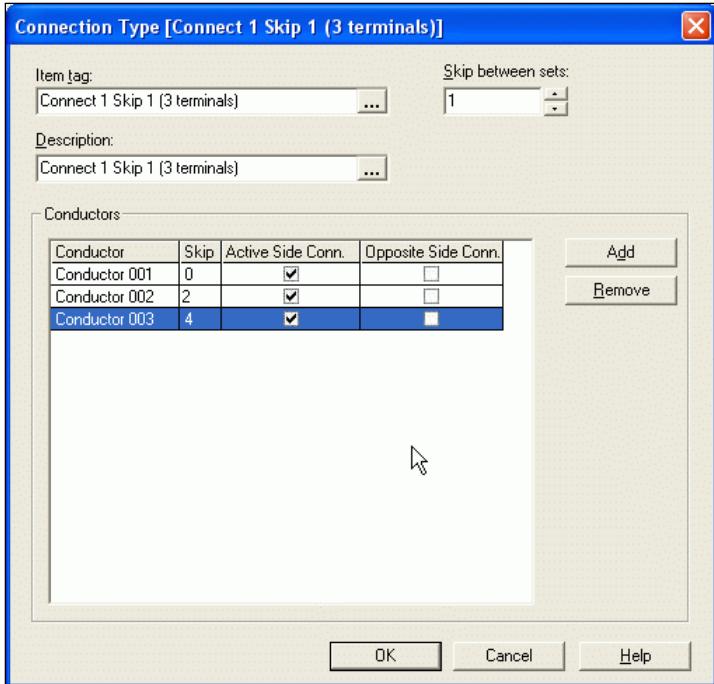


Connection type

The next RDE item that is needed for wiring is “**connection type**”, and it relates to how the conductors/cores should connect to the terminals upon trying to connect a cable to a terminal strip:



Create connection type for configuration of connecting one terminal and skip one terminal



We will be using this connection type later, when we will perform the actual connection.

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.

Active Side Conn The actual side of the terminal on which the user is landing the cable.

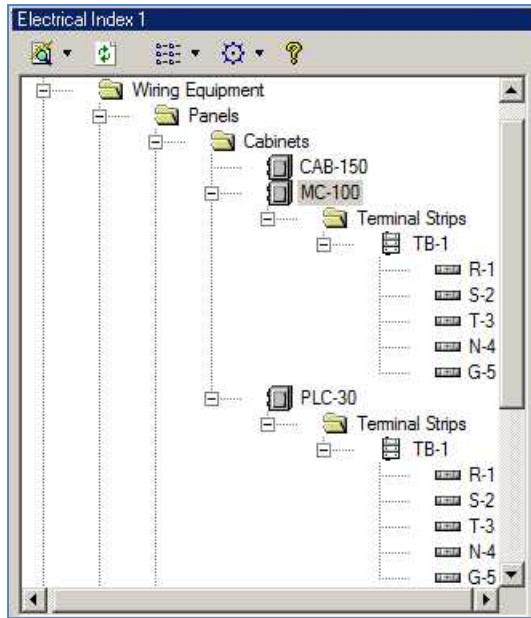
Opposite Side Conn The opposite side of the terminal on which the user is landing the cable.

Note: There are no typical fully wired objects in RDE. No ability to define a completely wired assembly.

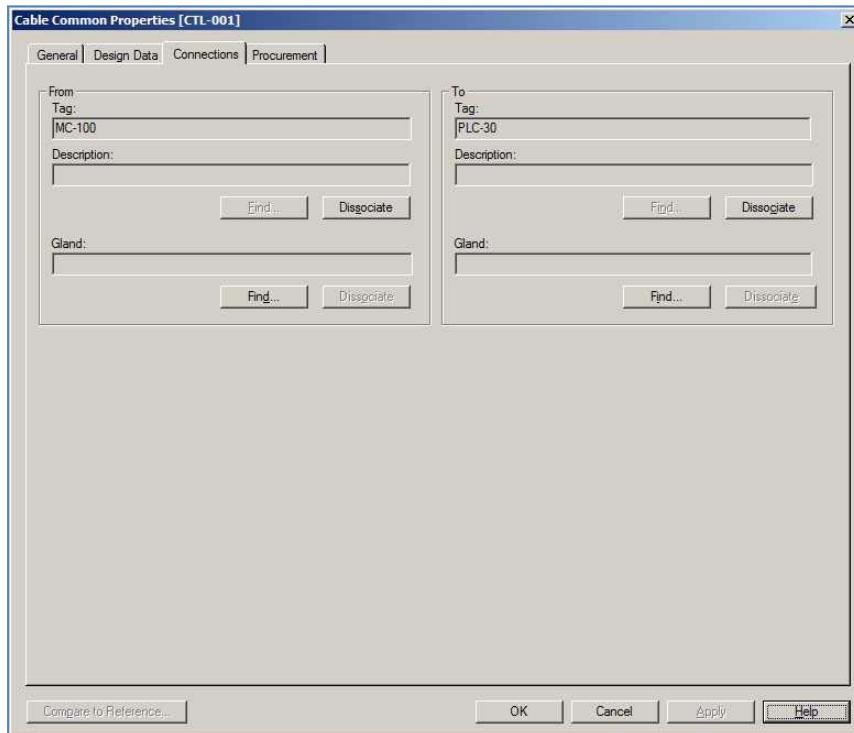
Wiring project items

1. Create 2 cabinets, 'MC-100', and 'PLC-30' in EI.
2. Create strip, 'TB-1' for each of the cabinet, using the typical strip '5 TS'.

Wiring Diagram

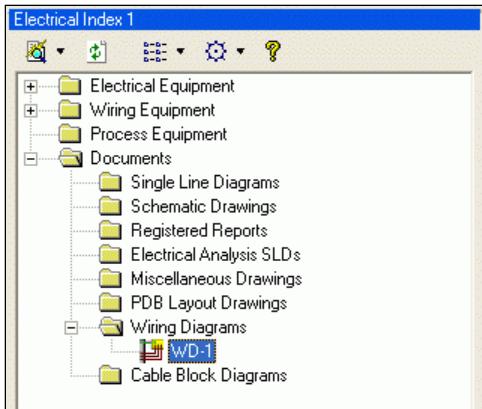


3. Create control cable, 'CTL-001' using the typical control cable '3/C- 10 AWG - (Control)'.
4. Open the common properties windows for cable CTL-001 and from the connection tab make a Connection From 'MC-100' to 'PLC-30'.

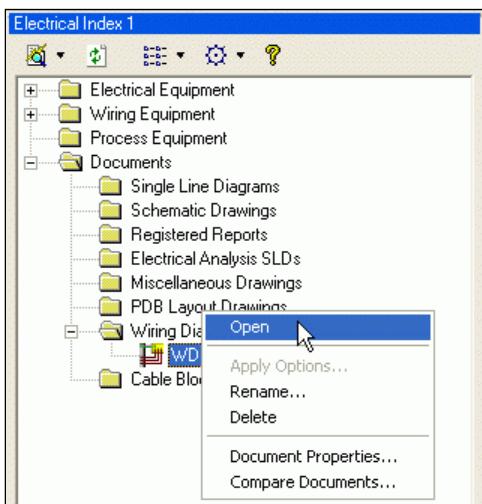


5. Create new wiring diagram (WD) and name it 'WD-1'.

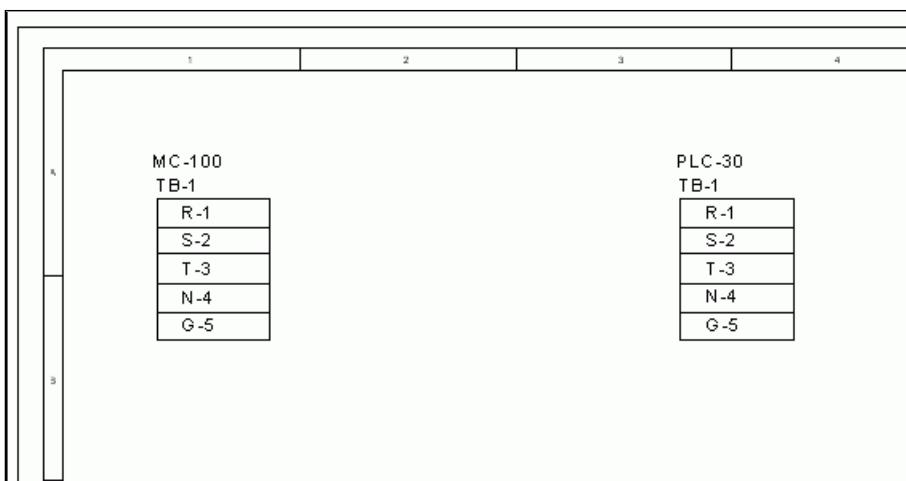
Wiring Diagram



5. Right click on WD-1 and select open.

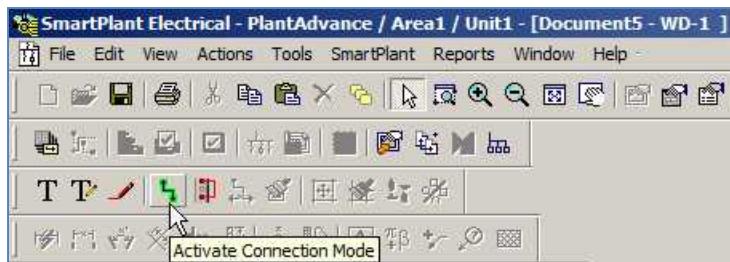


6. Drag the MC-100 and PLC-30 into the drawing.

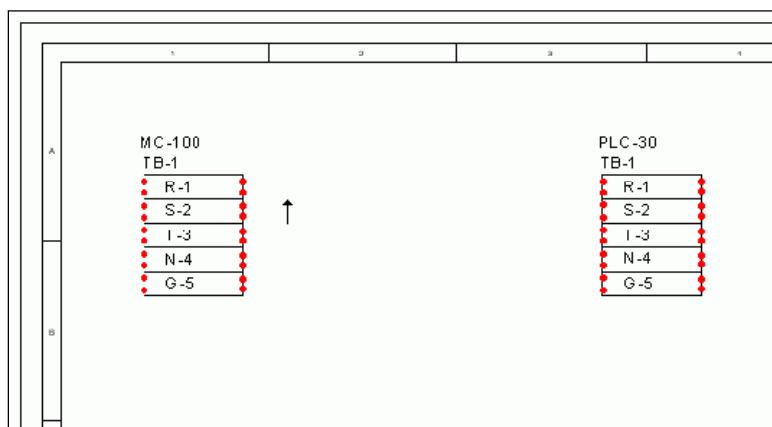
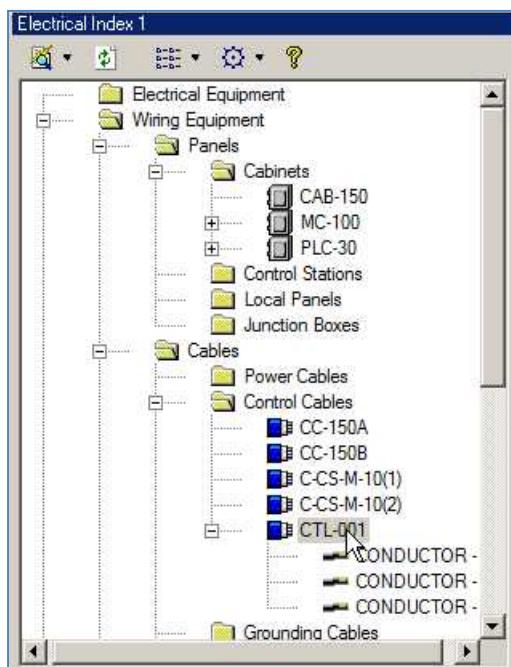


7. Select Activate Connection Mode.

Wiring Diagram

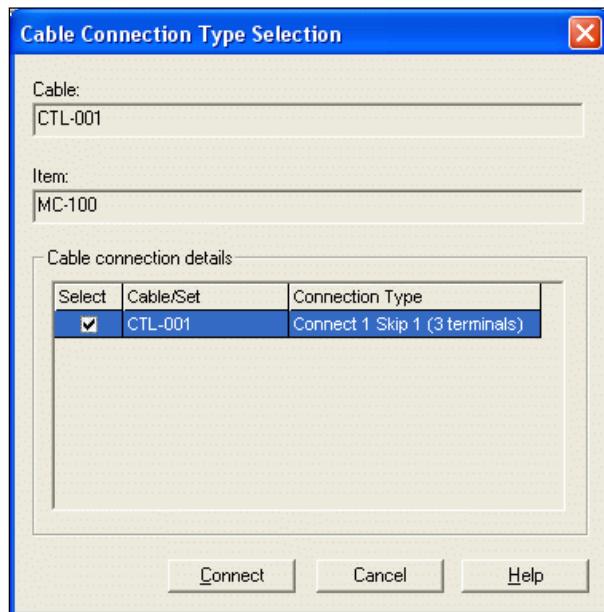
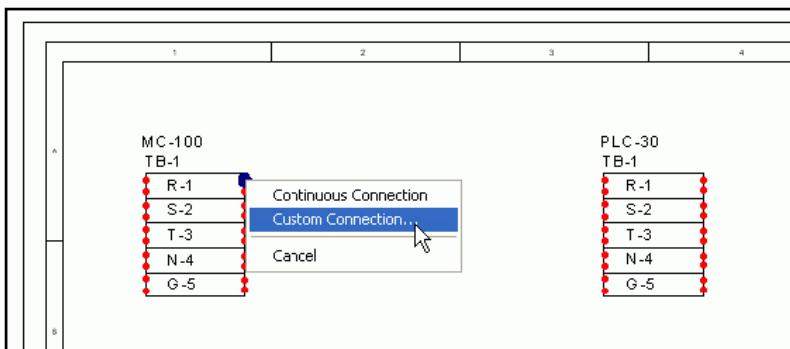


8. Select the cable that connects this equipment and move your cursor over the wiring diagram.

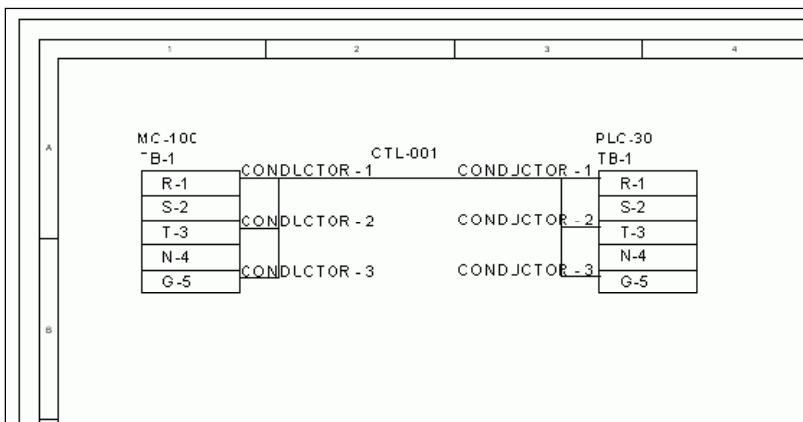


7. Select the first terminal of the left equipment strip, at its connection point at the right side and click on the blue circle.

Wiring Diagram

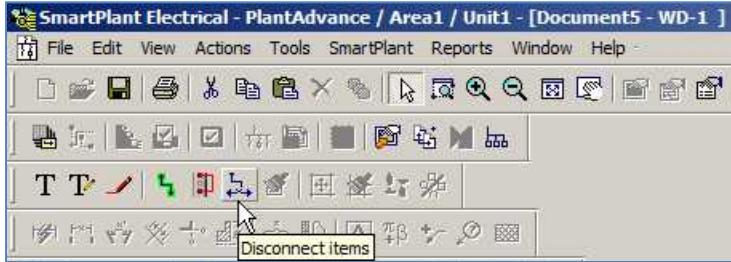


After connecting on the first terminal, drag the cursor with the cable over to the next equipment on the right side and land it on the first terminal. Select the “Custom connection” again and finish the connection.



Wiring Diagram

To move a wire/set from one terminal to another you need to disconnect it and reconnect (right click on the wire or on the set or on the cable and select “disconnect”)

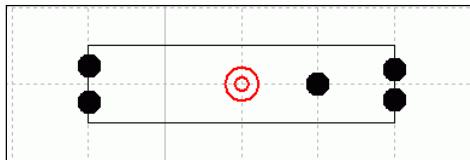


8. Save the document and close it.

Note: The number of conductors that one can connect to a terminal depends on the number of connection points available on that terminal (as defined in the terminal symbol in catalog manager)

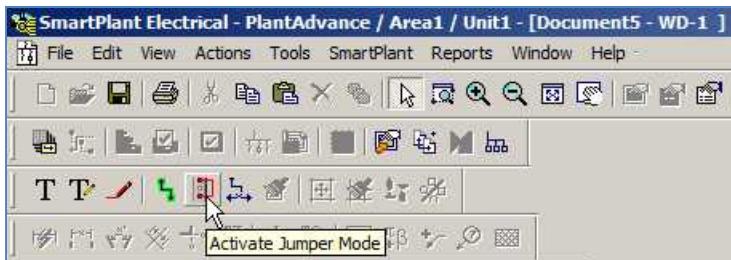
To connect 2 conductors to a terminal one needs to define the terminals having 2 connection points at each side, like this:

Each terminal also has addition connection point to connect a jumper.



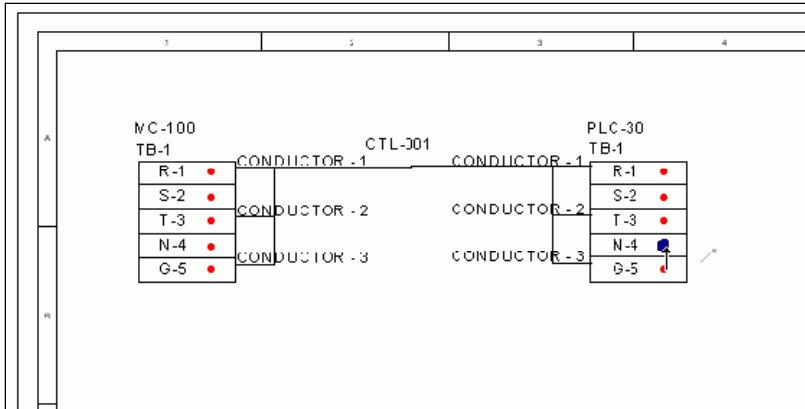
To connect jumper in PLC-30, TB-1 between terminals N-4 and G-5 open the wiring diagram WD-1.

1. Open the document and select activate jumper mode

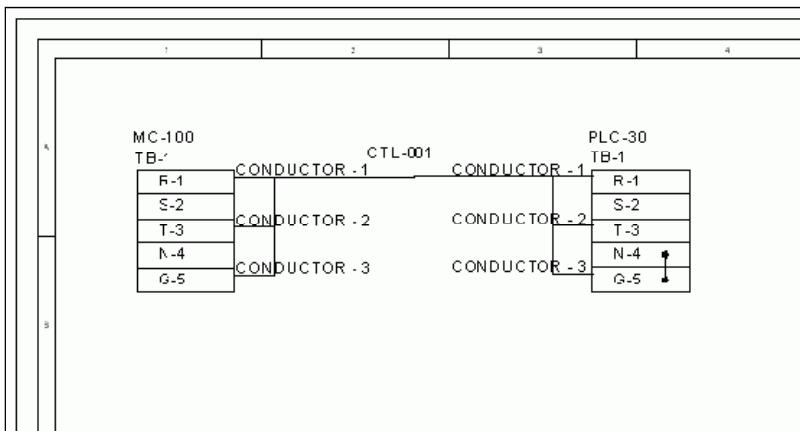


2. Select the first jumper connection point.

Wiring Diagram



3. Select the second jumper connection point and finish the connection.



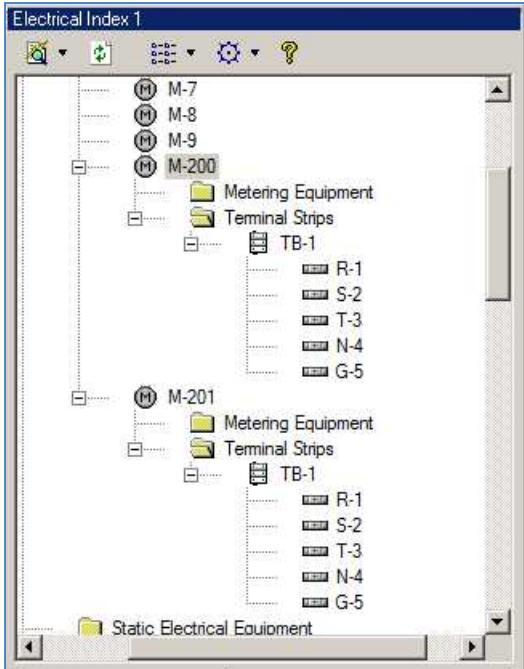
4. Save the document and close.

Batch cables sides' termination

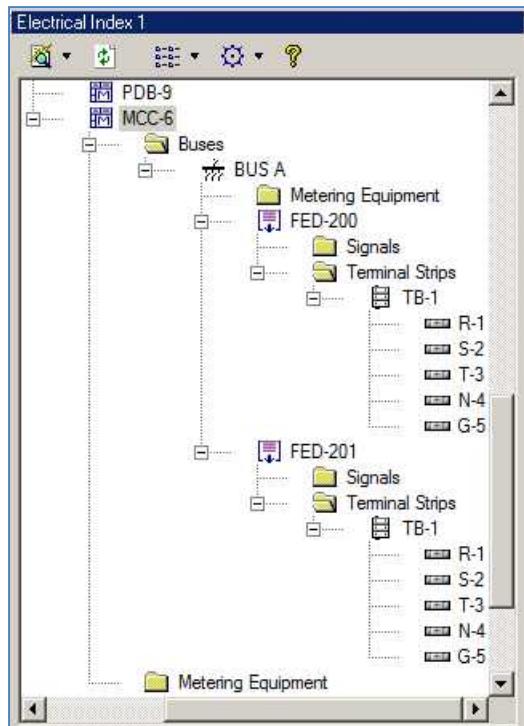
This feature supports performing a wiring operation on a set of cables that share the same pattern of wiring; a good example for that would be the ability to terminate all the control cables associated to 10 hp motors. Make sure that you have few motors sharing the same connectivity.

- 1.Create in EI motor, 'M-200', with terminal strip, 'TB-1' using the typical strip '5 TS'.
- 2.Create in EI motor, 'M-201' by duplicate motor 'M-200'.

Wiring Diagram



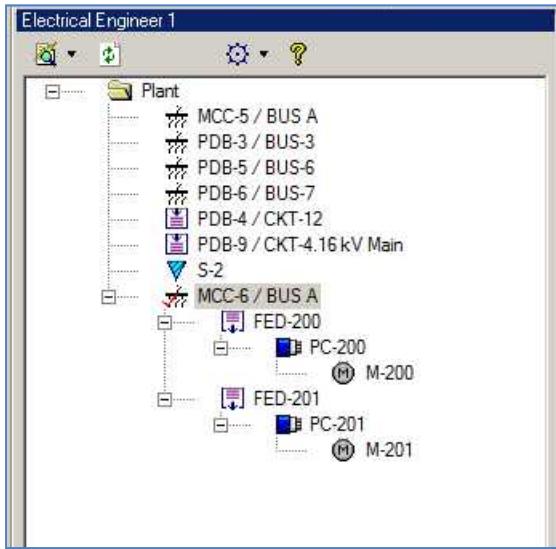
3. Create in EI Power Distribution Board, 'MCC-6' with bus, 'BUS A'.
4. Create in BUS A feeder circuit, 'FED-200' with strip 'TB-1' using the typical strip '5 TS'.
5. Create in BUS A feeder circuit, 'FED-201' by duplicate 'FED-200'.



6. Create in EI power cable, 'PC-200' using the typical power cable '3/C-12AWG -(Power / NEC)'.
7. Create in EI power cable, 'PC-201' by duplicate 'PC-200'.

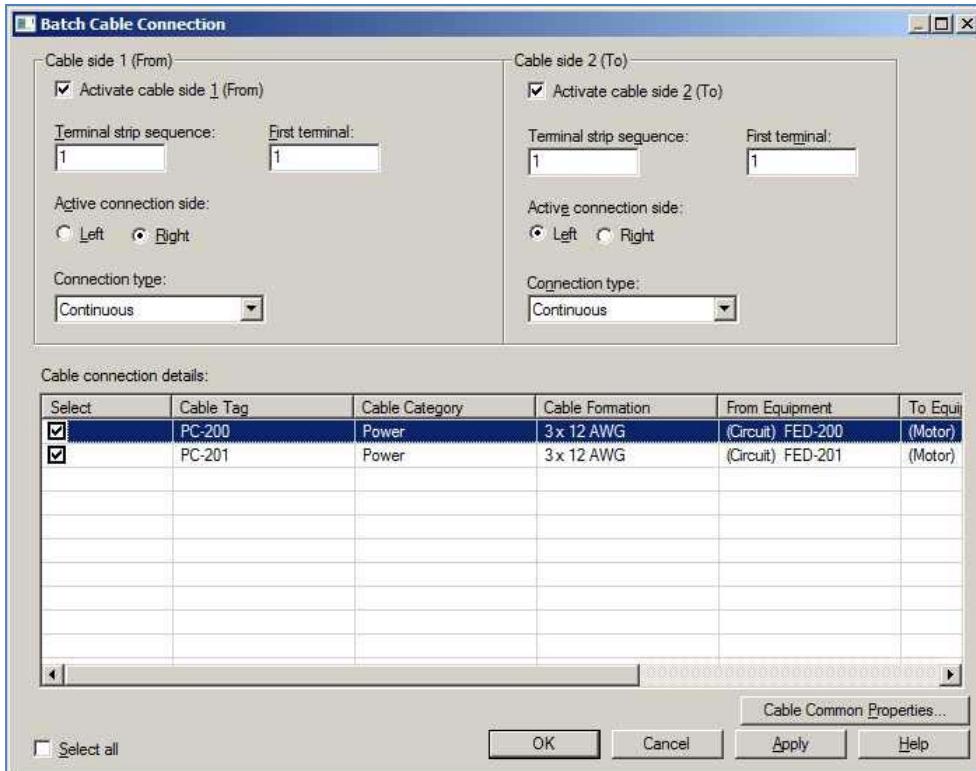
Wiring Diagram

8. Make the following connection in EE for cables 'PC-200' and 'PC-201'.



9. Select from EI cables 'PC-200' and 'PC-201'.

10. From Actions menu select > cables > 'Batch Cable Connection', to open the 'Batch Cable Connection' UI where you can make the batch connection.



Note: Be sure to select the right cable active connection sides otherwise the conductors will be shown terminated on the wrong side.

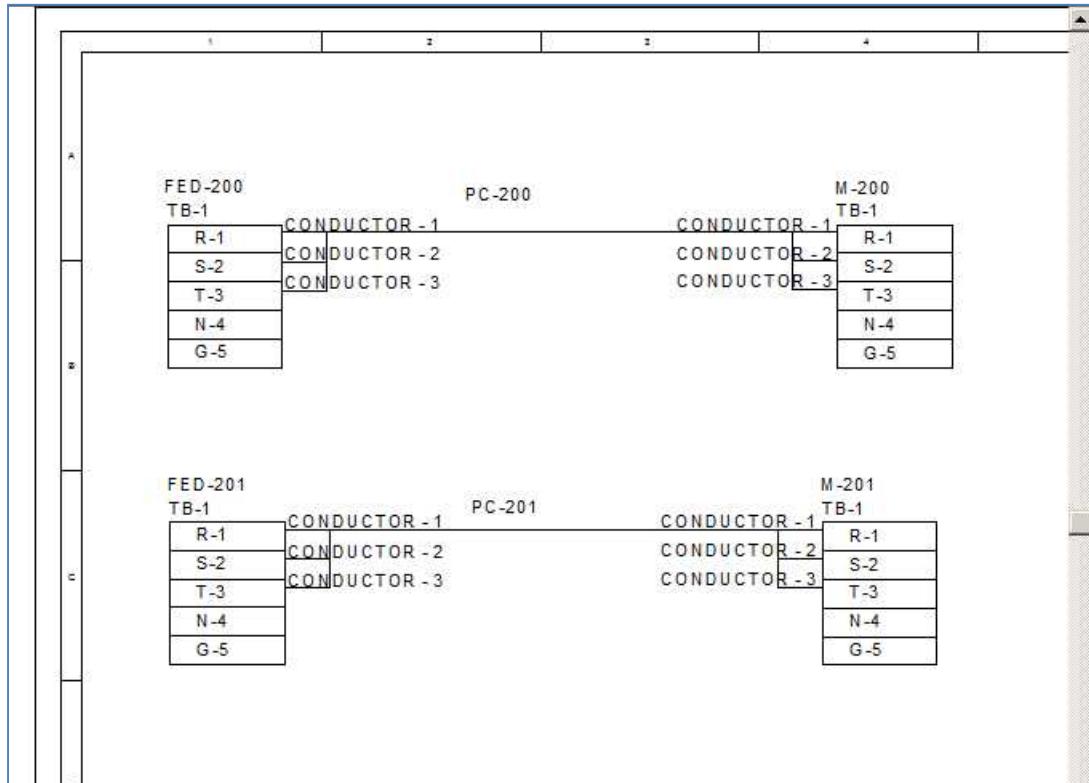
Wiring Diagram

Select OK to make the connection.



11.Create new wiring diagram, WD-2.

12.Open the wiring diagram WD-2, and drag to the open drawing circuits FED-200,FED-201, and motors, M-200,M-201.

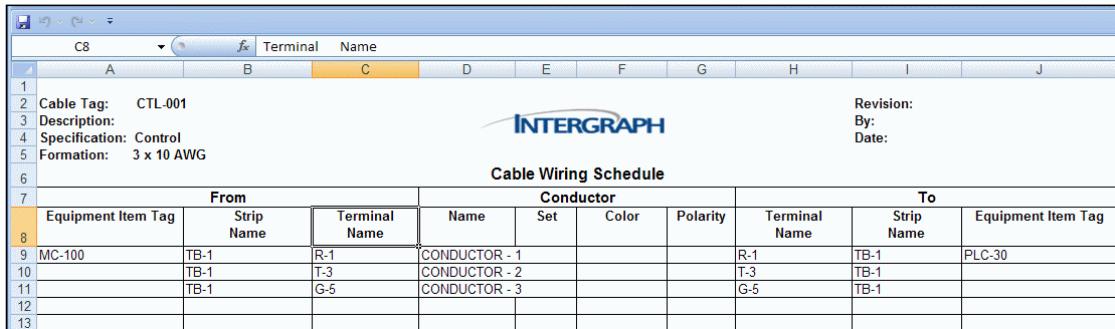


Wiring Shipped Reports

Cable wiring schedule

This report is based on the cable item type and retrieves and lists the selected cables with their conductors showing to which strip and terminal they are connected.

Select cable CTL-001 and run the report.



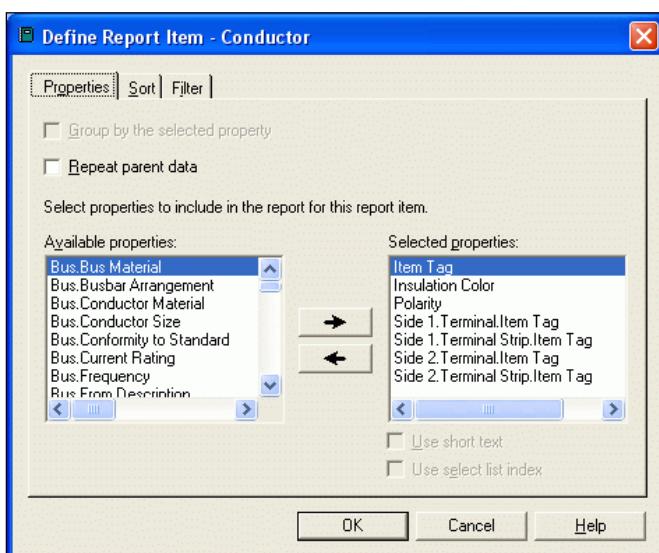
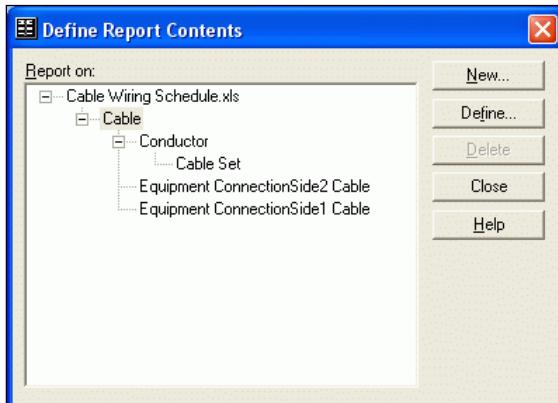
Cable Tag: CTL-001
Description:
Specification: Control
Formation: 3 x 10 AWG

INTERGRAPH

Cable Wiring Schedule

From			Conductor				To		
Equipment Item Tag	Strip Name	Terminal Name	Name	Set	Color	Polarity	Terminal Name	Strip Name	Equipment Item Tag
MC-100	TB-1	R-1	CONDUCTOR - 1				R-1	TB-1	PLC-30
	TB-1	T-3	CONDUCTOR - 2				T-3	TB-1	
	TB-1	G-5	CONDUCTOR - 3				G-5	TB-1	

The following definitions and tables were used to get the “**conductor to terminal**” information:



As you can see, among the attributions of the conductor you can find the connectivity data of both sides of the conductor.

Terminal strips schedule

Report that based on the terminal strip you select in the Electrical Index.

Don't select specific item type and run the report.

The screenshot shows a Microsoft Excel spreadsheet titled "Terminal Strip Schedule.xls". The data is organized into several sections:

- Header:** PDB: MCC-6, Bus: BUS A, Equipment: M-200, Type: Motor, Description:.
- Revision Information:** Revision: [empty], By: [empty], Date: [empty].
- Section Headers:** Terminal Strip Schedule.
- Table Headers:** Cable, Set, Wire, Polarity, Color, Terminal Strip, Terminal.
- Data Rows:** PC-200, CONDUCTOR - 1, TB-1, R-1; PC-200, CONDUCTOR - 2, TB-1, S-2; PC-200, CONDUCTOR - 3, TB-1, T-3; [empty], [empty], TB-1, N-4; [empty], [empty], TB-1, G-5.

Cable	Set	Wire	Polarity	Color	Terminal Strip	Terminal
PC-200		CONDUCTOR - 1			TB-1	R-1
PC-200		CONDUCTOR - 2			TB-1	S-2
PC-200		CONDUCTOR - 3			TB-1	T-3
					TB-1	N-4
					TB-1	G-5

Terminal strips schedule- Panel

Report that based on the panel you select in the Electrical Index.

Don't select specific item type and run the report.

The screenshot shows a Microsoft Excel spreadsheet titled "Terminal Strip Schedule - Panel". The data is organized into several sections:

- Header:** Panel: MC-100, Type: Cabinet.
- Revision Information:** Revision: [empty], By: [empty], Date: [empty].
- Section Headers:** Terminal Strip Schedule - Panel.
- Table Headers:** Cable, Set, Wire, Polarity, Color, Terminal Strip, Terminal.
- Data Rows:** CTL-001, CONDUCTOR - 1, TB-1, R-1; CTL-001, CONDUCTOR - 2, TB-1, S-2; CTL-001, CONDUCTOR - 3, TB-1, T-3; [empty], [empty], TB-1, N-4; [empty], [empty], TB-1, G-5.

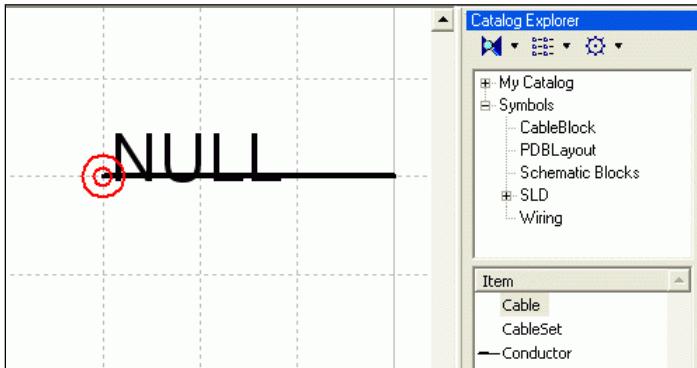
Cable	Set	Wire	Polarity	Color	Terminal Strip	Terminal
CTL-001		CONDUCTOR - 1			TB-1	R-1
CTL-001		CONDUCTOR - 2			TB-1	S-2
CTL-001		CONDUCTOR - 3			TB-1	T-3
					TB-1	N-4
					TB-1	G-5

Creating Wiring Symbols

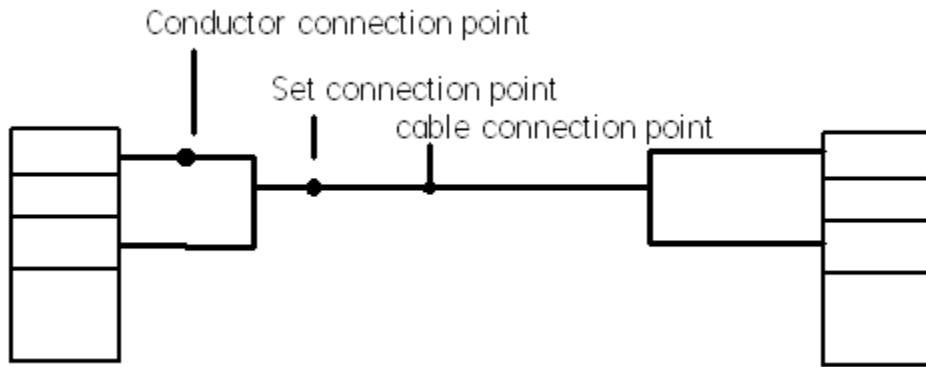
As mentioned, the system provides default symbols for the wiring for cables, conductors, terminals and strips.

Wiring Diagram

Cable default shipped symbol

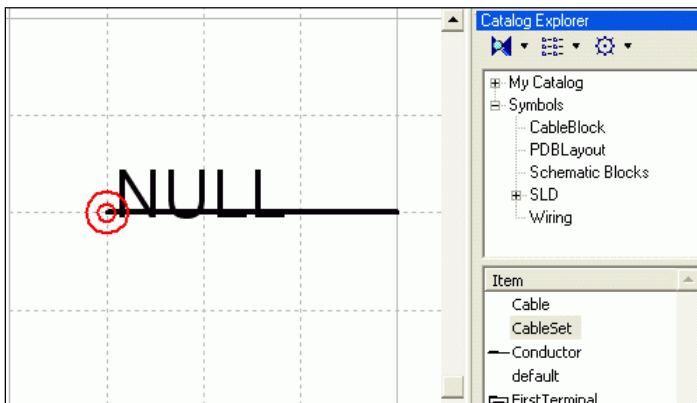


The macro is positioned along the cable, related to the left side origin point, either above, or below
The macro is referenced to the origin point that represents the connection of the cable to the set or to the conductor, as shown in the following illustration:



The cable connects to the cable set, the cable set to the conductor, and the conductor to the terminal, at both cable sides. Macros are then referenced to the relevant connection points.

Cable Set default shipped symbol

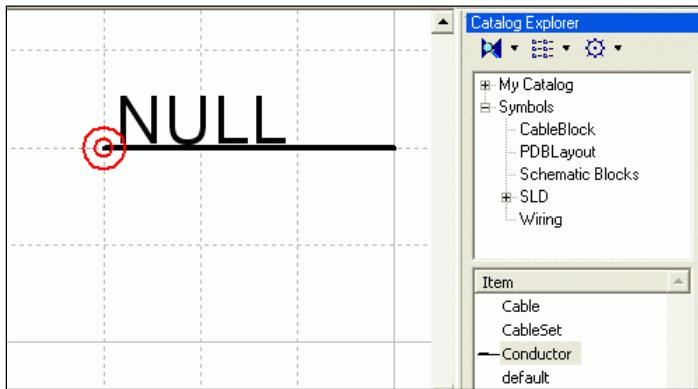


The macro is positioned along the cable set, related to the left side origin point, either above, or below

Wiring Diagram

Conductor default shipped symbol

As mentioned above



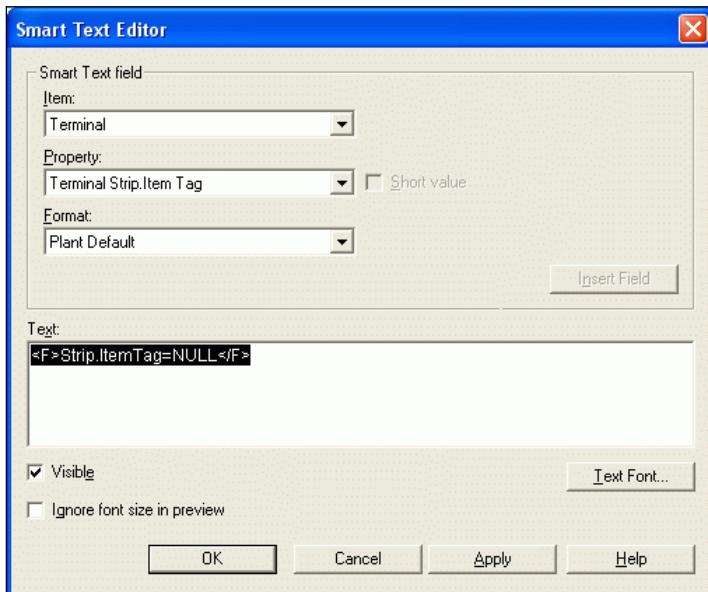
Terminal and first terminal default symbols

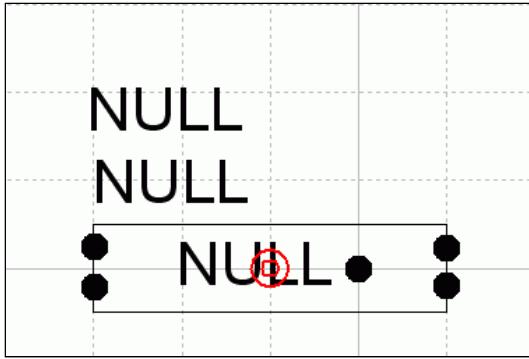
When there is no symbol of terminal or strip associated with the equipment terminated the system uses default shipped symbols (or ones that user customized). It will use the **first terminal and terminal** set of symbols as described below.

The first terminal differs from regular terminal by the fact that we usually want to see the equipment name and strip name above the first terminal.

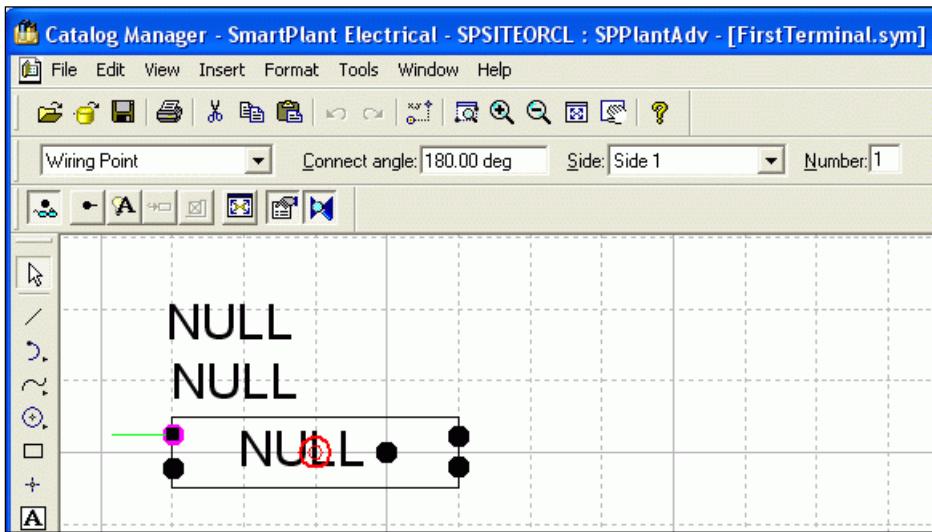
First terminal shipped symbol

The first terminal is a **hard coded identified symbol**, contains the terminal graphics, its terminal number macro and, above it, the strip name, and above it the equipment name.





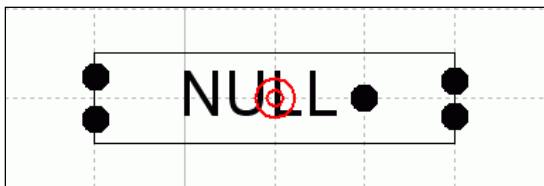
A wiring connection point is needed wherever user wants to attach a conductor. Each wiring connection point has connection angle, a side 1 or side 2 index and, within the side a sequence index (2nd wiring connection on side 1 of the terminal):



Note: Although usually you would need a filter to identify first terminal, in this case SPEL hard coded it and can be used out of the box.

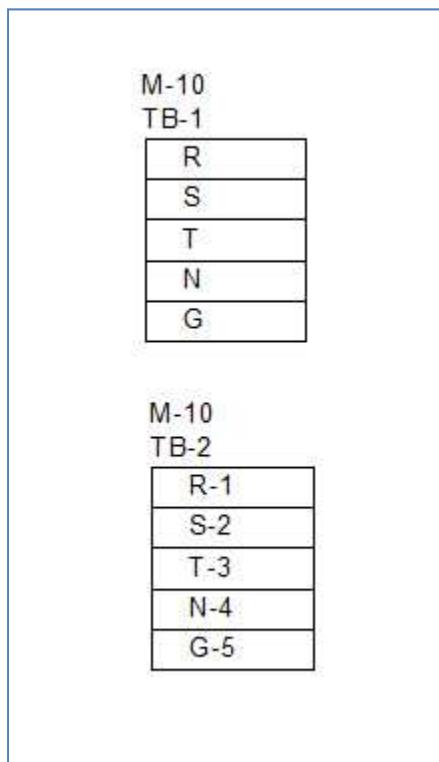
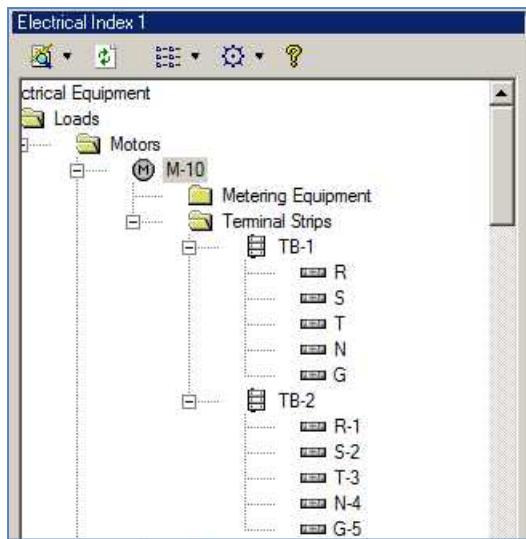
Terminal default shipped symbol

This is a regular terminal that contains only the graphics and the macro for the terminal number



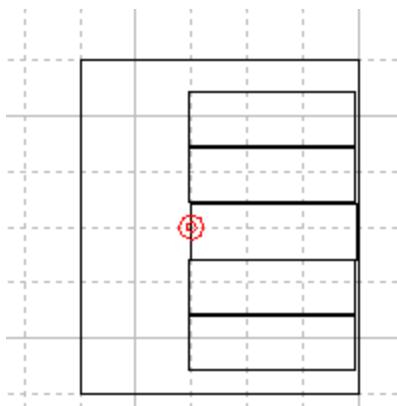
Motor strip with 5 terminals will look as the following:

Wiring Diagram



Creating more complex symbols for wiring objects (assemblies of wiring objects)

Suppose we want to have the following presentation for a local panel with a terminal strip of 5 terminals:



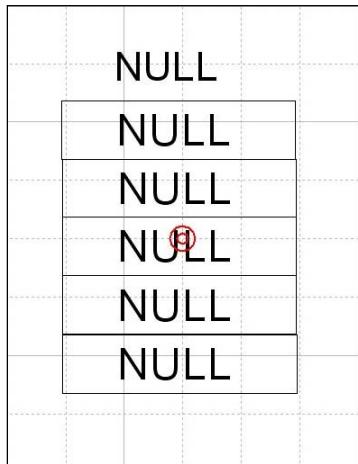
For this we will have to create an assembly, composed of the following individual symbols, that, when embedded properly one into the other will provide the complete assembly:

A symbol for a panel that contains a symbol for a strip, embedded within the panel.
5 symbols representing 5 regular terminals, embedded in the strip

Here is the workflow creating this assembly:

Step 1: Create a strip of terminals

Create a new symbol, of strip item type, and drag into it 5 times the regular terminals, as shown:



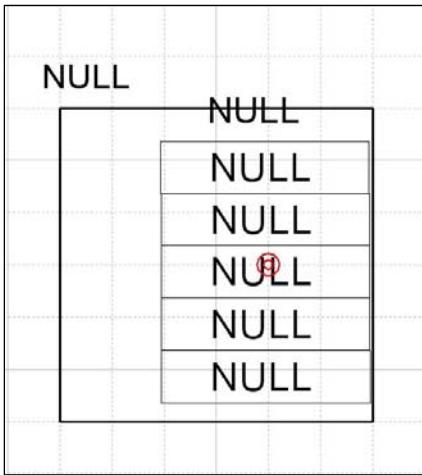
Note: make sure you select the strip item type before saving the symbol. Each terminal that you will drag will have an index number, corresponding to the sequence of the terminal in the strip

When creating assemblies you don't have to use the first terminal since each terminal stands for its own.

Step 2: Creating the local panel

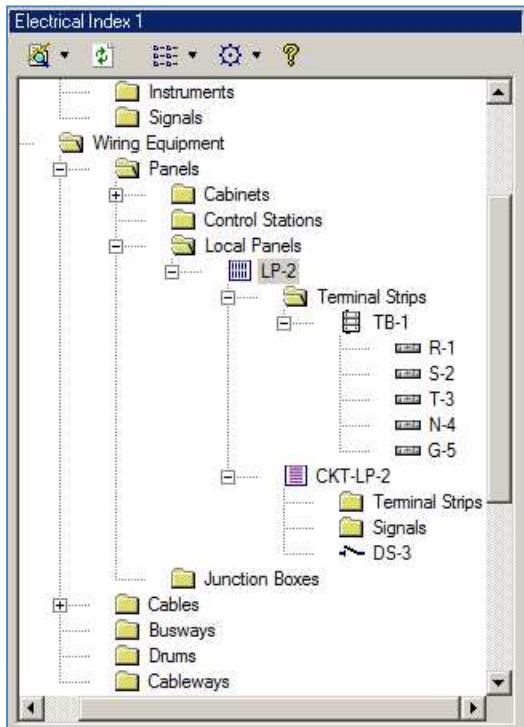
Wiring Diagram

Create a new symbol of a local panel and embed the strip into it.

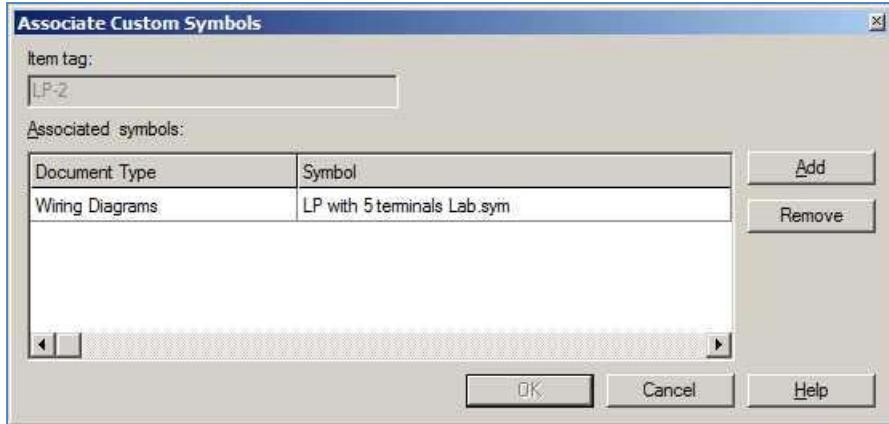


Save the local panel symbol.

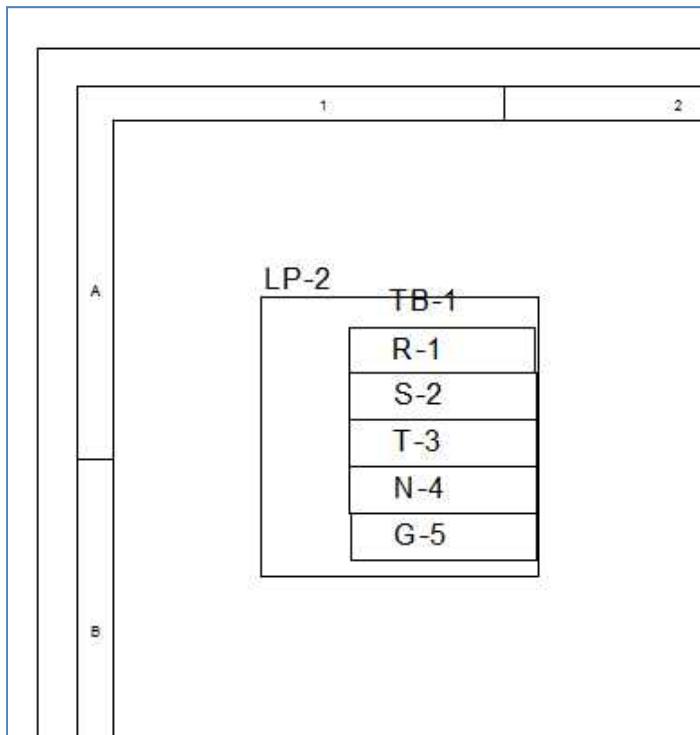
Select the local panel LP-2 and create new terminal strip TB-1 using the terminal strip configuration '5 TS'.



Associate the local panel with the new symbol created in step 2:



Create a new wiring diagram, WD-3 and drag the local panel into it.

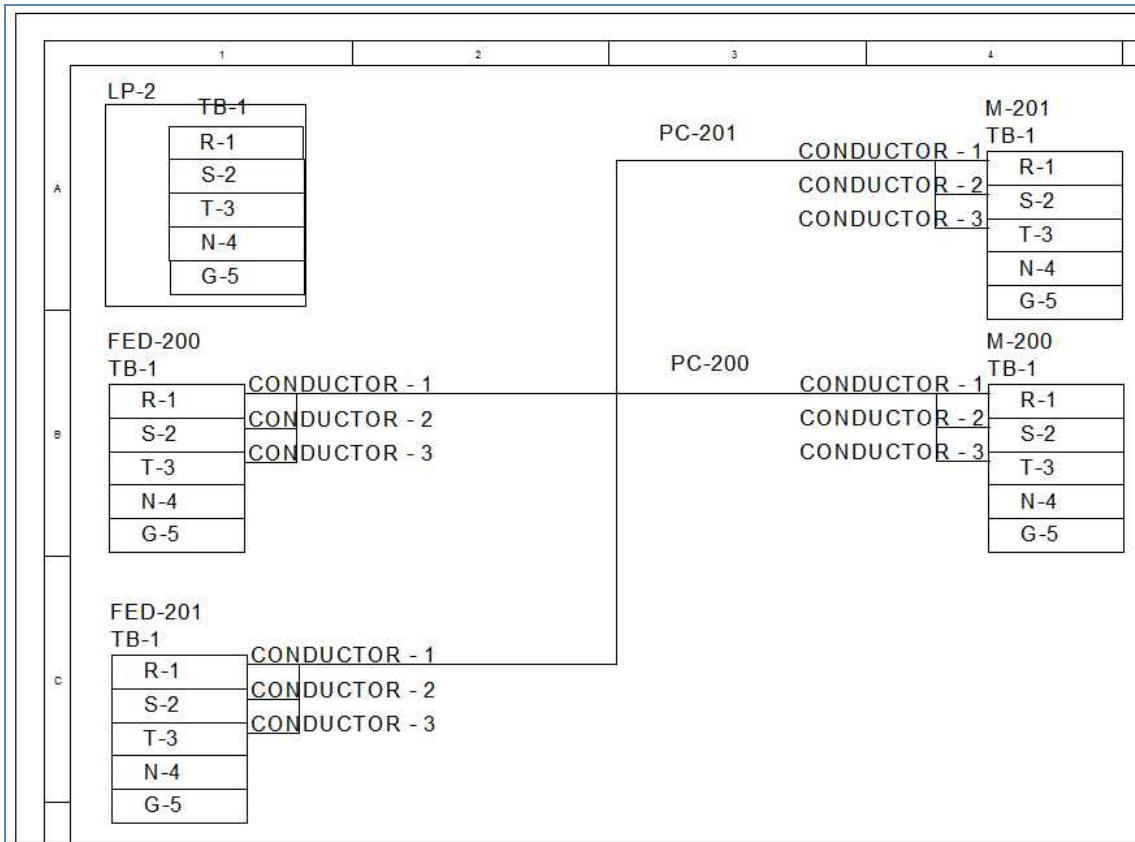


Gapping

When creating wiring diagrams or cable block diagrams the cables might cross each other. SmartPlant Electrical allows gapping such cables.

Generate a wiring diagram that has cables that cross each other.

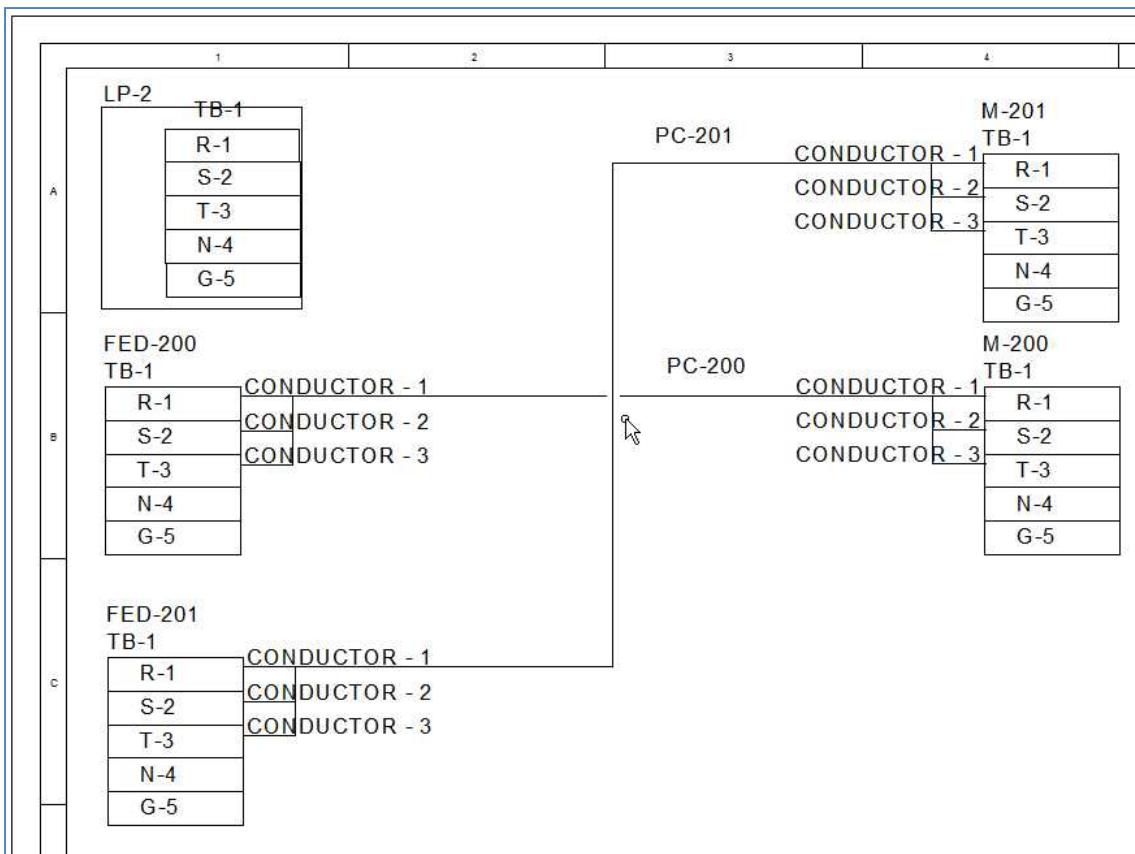
Wiring Diagram



Select Gap Now from tools menu.



Wiring Diagram



SmartPlant Electrical Rule Manager

SmartPlant Electrical Rule Manager is an environment used for creating, manipulating and managing sets of consistency rules that apply whenever a user associates two items or modifies properties of a certain item tag in SmartPlant Electrical. These rules determine how items act and interact within a project. Usually, you create a set of rules at the beginning of a project. You can modify any rule at any stage of the project life-cycle when project requirements dictate a change.

A rule is a combination of various consistency criteria that you can specify for rule components. A rule component can be either an item type or a filter created in Filter Manager. The following types of rules are available:

- Relation rules - rules in which consistency criteria apply to two specific components.
- Single component rules - rules in which consistency criteria apply to one component only.

Rules are used in SmartPlant Electrical for the following reasons:

- To ensure that consistent relationships are created between items when you create a new item relationship.
- To ensure that consistent relationships are created when you modify item properties for related items.
- To check the consistency of item properties or property values.
- To copy property values from one item to another, allowing the propagation of data from one item to its related/connected item.

Creating Single component and relation rules

Options Manager Settings

Open options manager and select general settings.

Set the following consistency rule settings:

Consistency rules: **Enable**

Consistency rules for batch operations: **Continue current operation**

Consistency rules	Enable
Consistency rules for batch operations	Cancel current operation

Save the change

Open Rule Manager, and create the following rules:

1. Rule Name: '**Start Current >= FLA**' (under 'single component rules' folders).

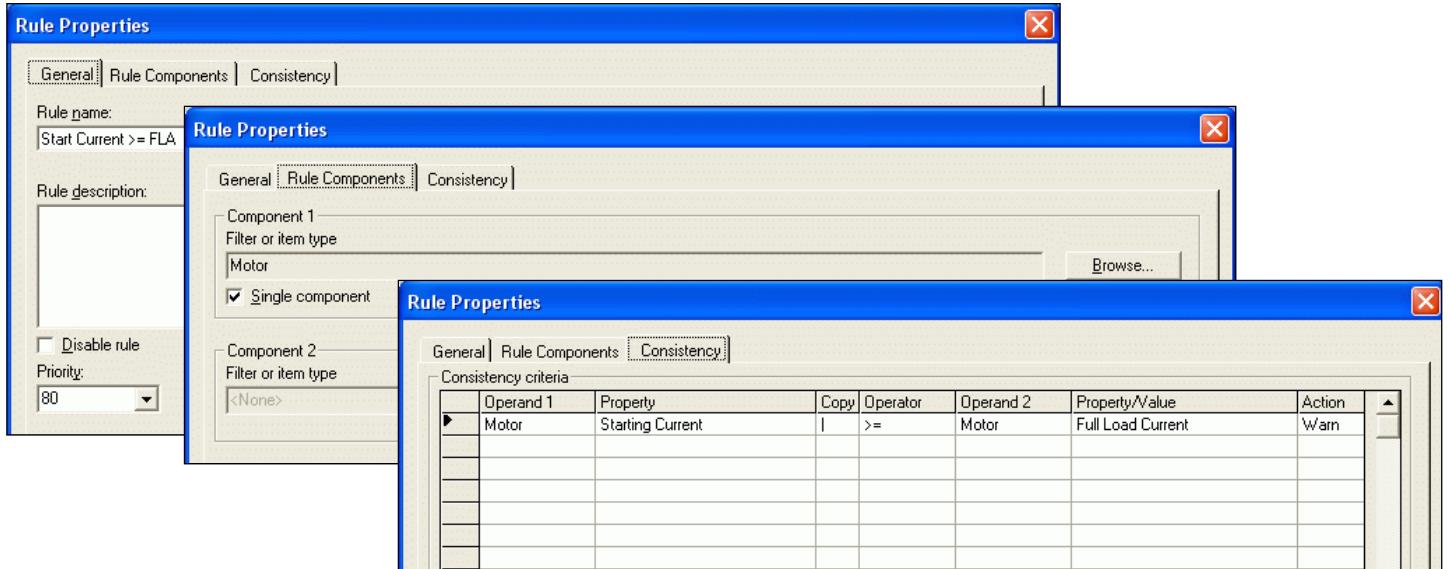
Rule Type: **Single Component**

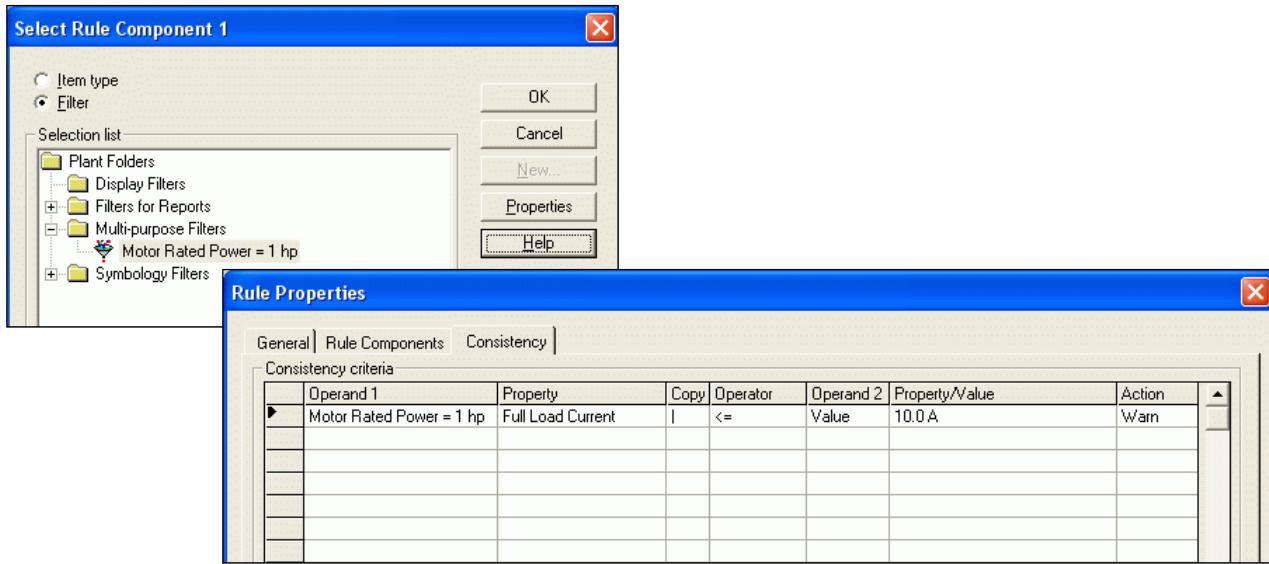
Item type: **Motor**

Consistency Criteria: **Starting Current >= Full Load Current**

Action: **Warn**

Priority: **80**





3. Rule Name: 10 hp motor rated voltage 120/460 V

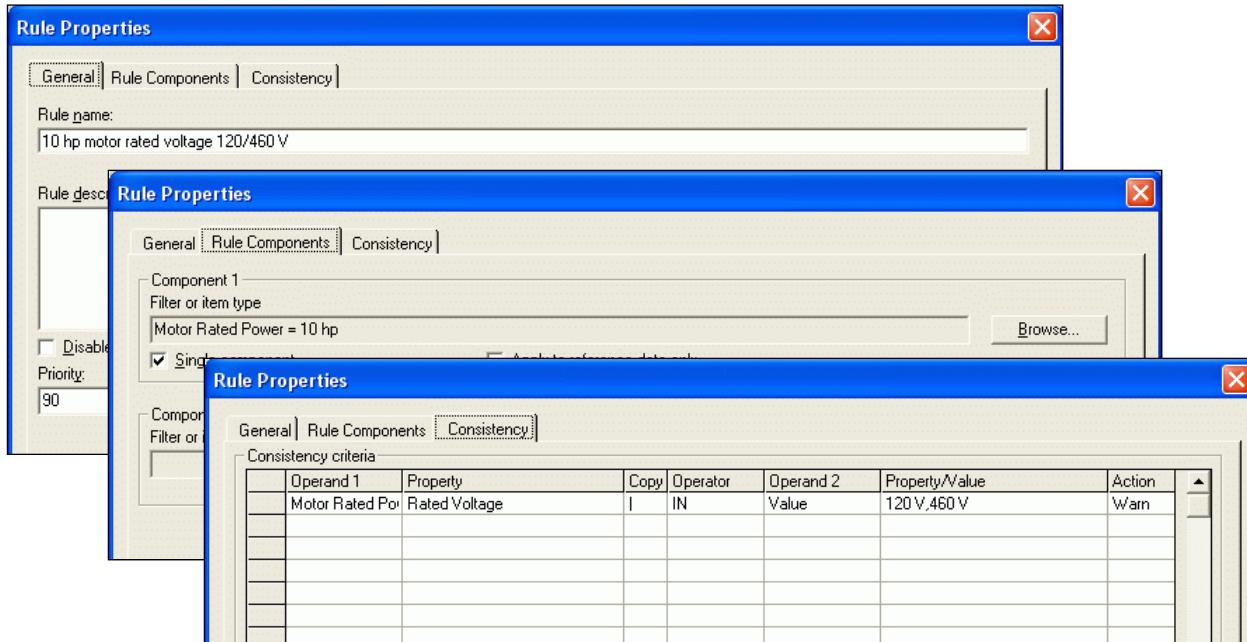
Rule Type: **Single Component**

Filter: **Motor Rated Power = 10 hp**

Consistency Criteria: **Rated voltage IN 120V, 460 V**

Action: **Warn**

Priority: **90**



Important:

Do not place a space after the separator.

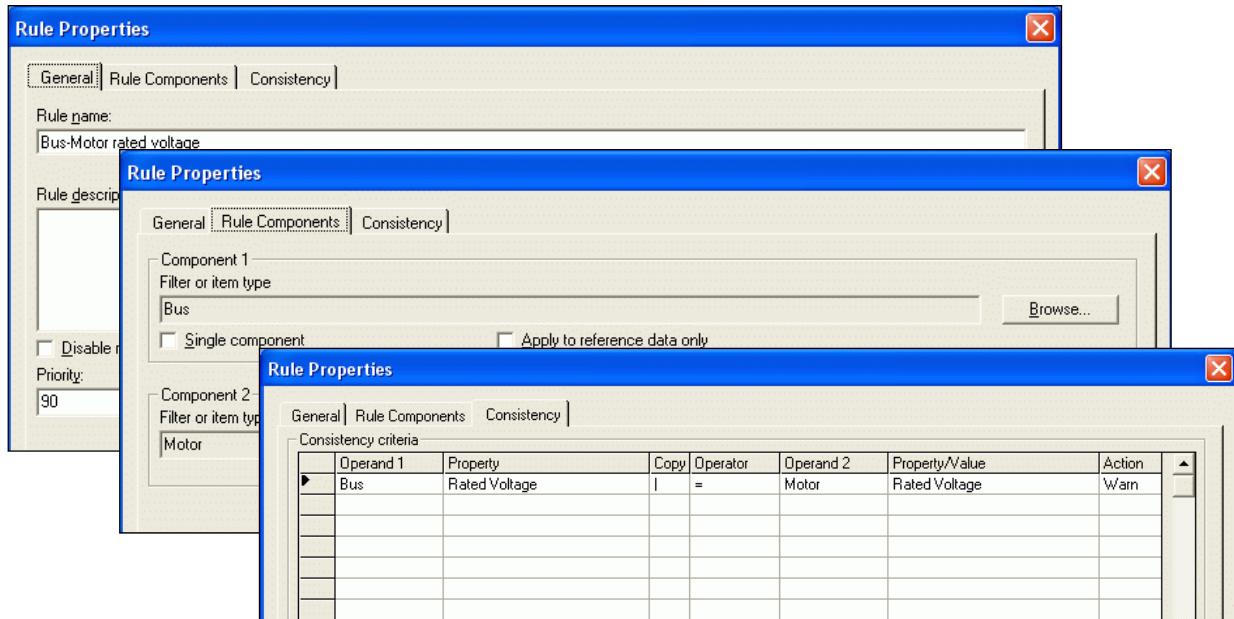
4. Rule Name: Bus-Motor rated voltage (create in relation rule folder).

Rule Type: **Relation Rule**

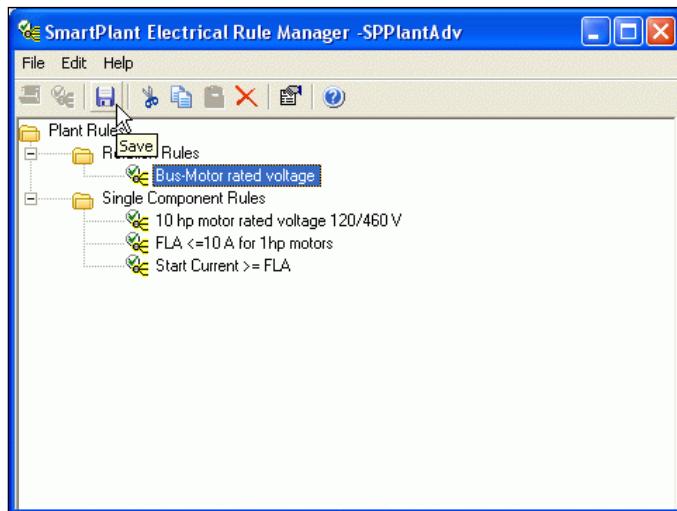
Component 1: Bus, Component 2: Motor

Consistency Criteria: Bus.Rated Voltage = Motor.Rated Voltage

Action: **Warn**, Priority: **90**



Save the rules.



Open Smart Plant Electrical and check rules 1-4 under different cases:

Case 1 - Single mode

Edit the data of one motor at a time and observe the result when it contradicts the rule.

Case 2 - Batch operation (continue current operation)

Create a typical motor in RDE that contradicts one or more rules. Use the new typical motor in a new motor profile, and apply the profile in a batch operation on 2 or more new motors.

Case 3 - Batch operation (cancel current operation)

In options manager, select general settings, and change the consistency rules for batch operations, from **continue current operation**, to **cancel current operation**, and save the change. Reopen SmartPlant Electrical and repeat case 2.

Rules that Handle Cable Insulation Voltages

Rule Manager should be able to handle various validations pertaining voltage related properties, such as Insulation Voltage Rating of a cable, the rated voltage, associated items to cables, converting equipment voltages (transformer's secondary voltage) validations, and so forth.

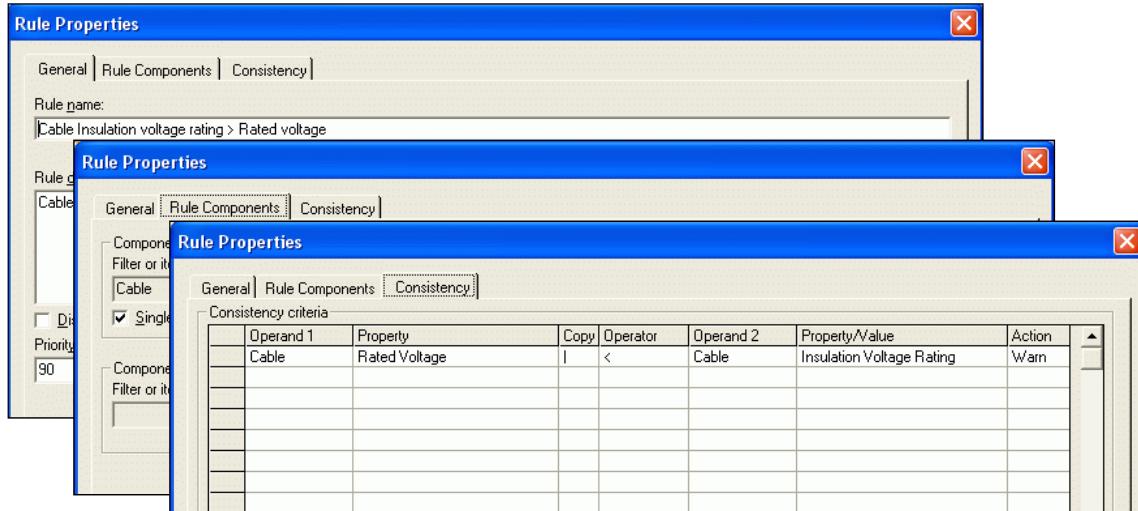
In order to perform these validations, new rules should be added in the Rule Manager. Here are few examples:

Case 1

The following rule will handle the following validations:

Prevent/warn when the cable sizing selects a reference cable that has its insulation voltage rating a value lower than the operating voltage. Assigning or replacing a reference cable to a project cable that would result in rated voltage that exceeds the insulation voltage rating of the cable.

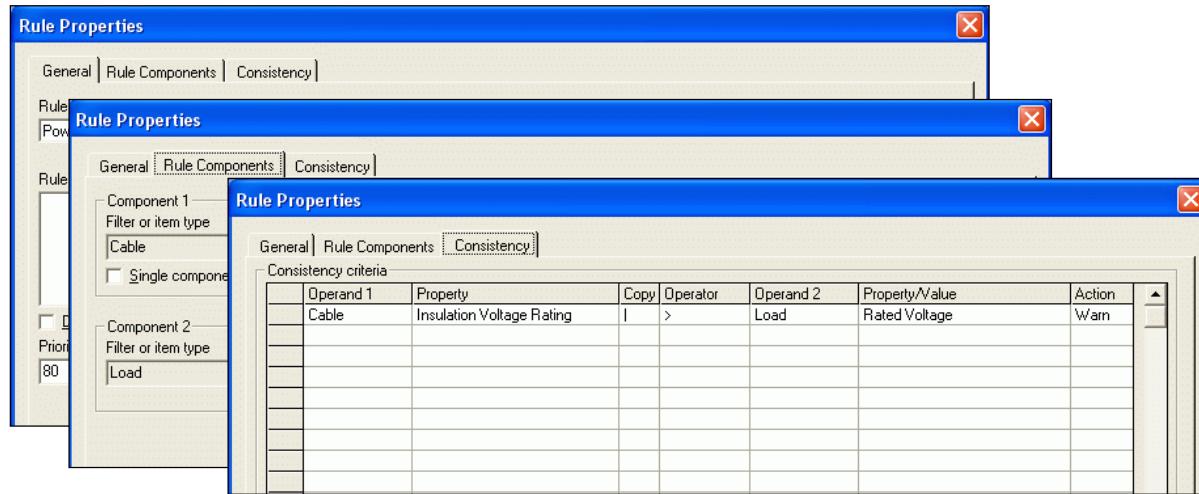
Create a single component rule for Cable item type as follows:



The program also considers insulation voltage ratings in the form of '6/10 kV' and it will validate the rated voltage vs. line-line voltage (second value).

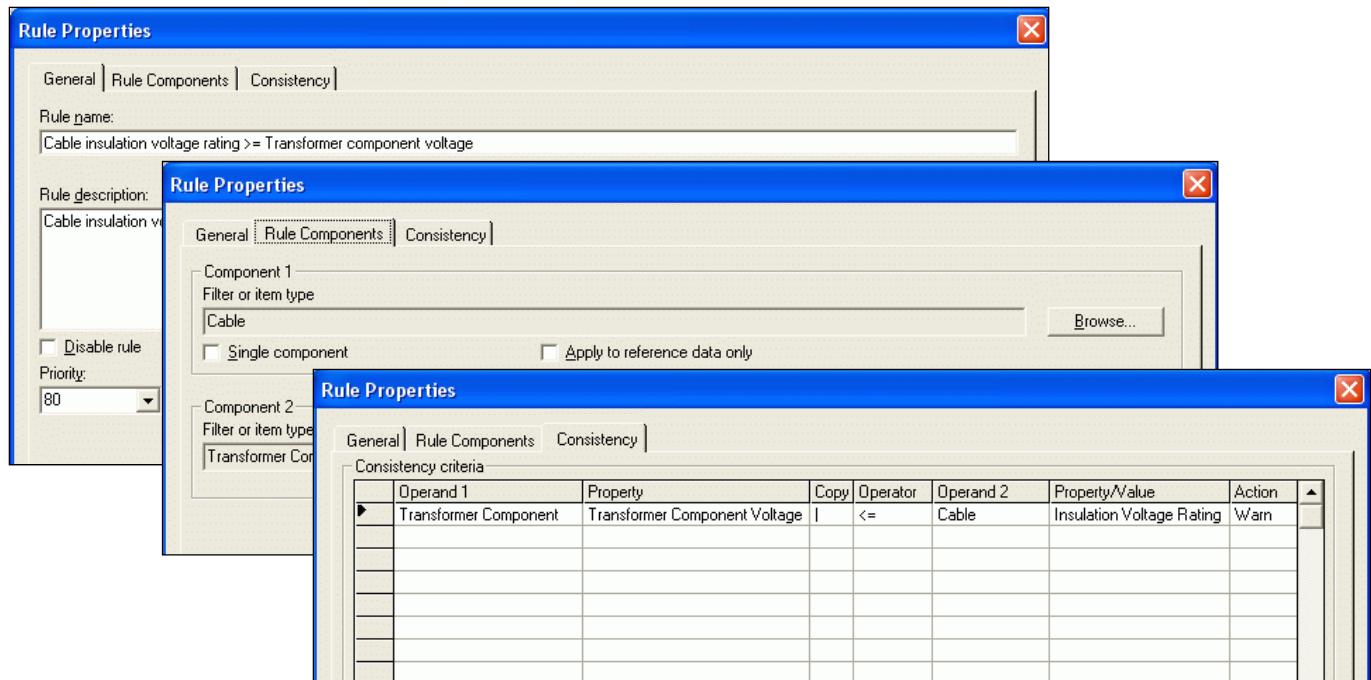
Case 2

Prevent/warn an association of a power cable to a load with rated voltage that exceeds the insulation voltage rating. In this example, the rule that should be created will be based on cable and load item types, as follows:



Similar rules can be created to validate the voltages between other item types and cables.

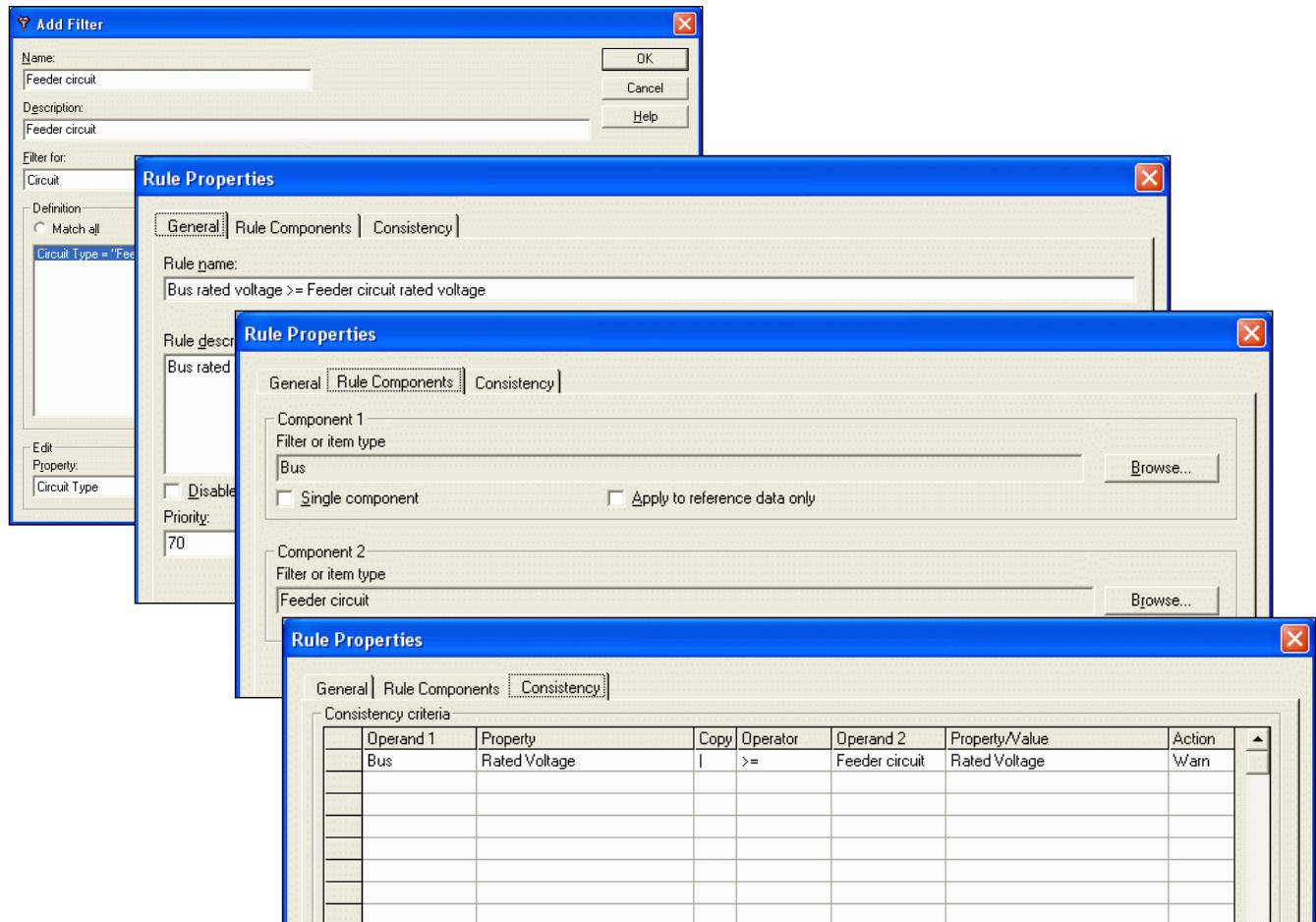
Case 3 Validating the output voltage of transformer's secondary and cable:



Case 4

Validating that bus rated voltage is higher or equal than the circuit's rated voltage.

Note: You should first create a filter for feeder circuit.



Other Cases

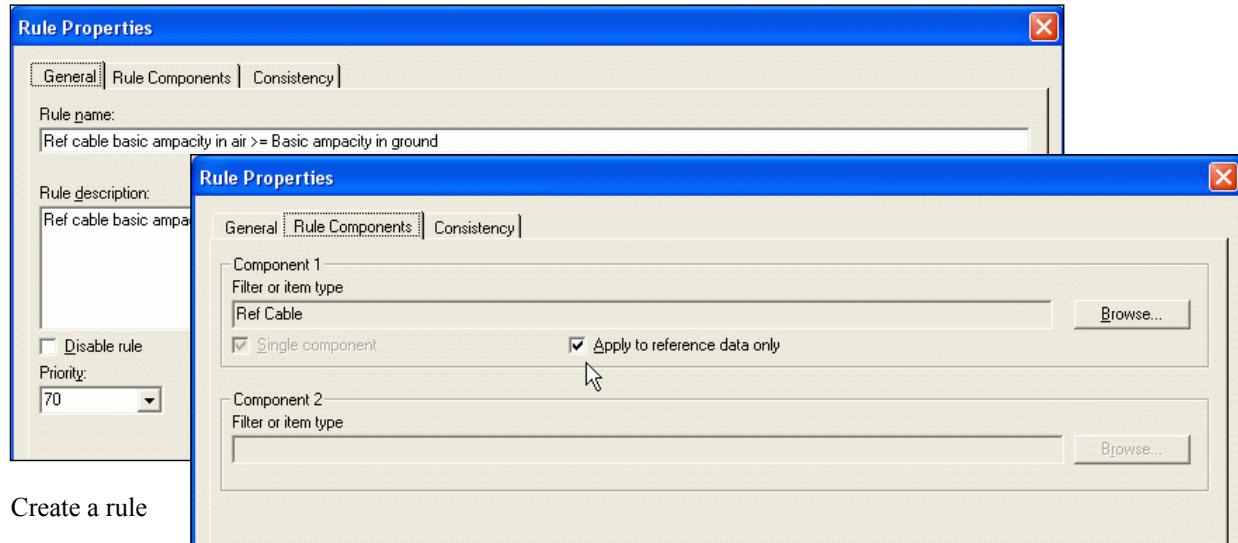
You can create additional rules to cover:

Cable to feeder circuit association to validate cable voltage rated equals circuit rated voltage.

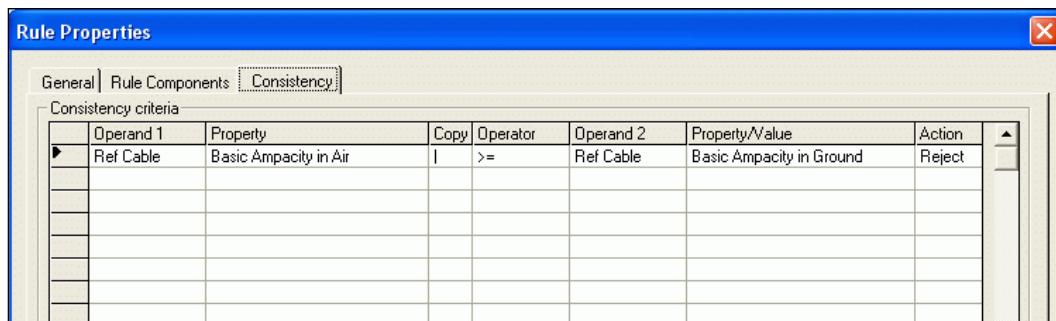
Cable to feeder circuit association to validate cable insulation voltage rating higher than circuit rated voltage.

Rules in Reference Data Explorer

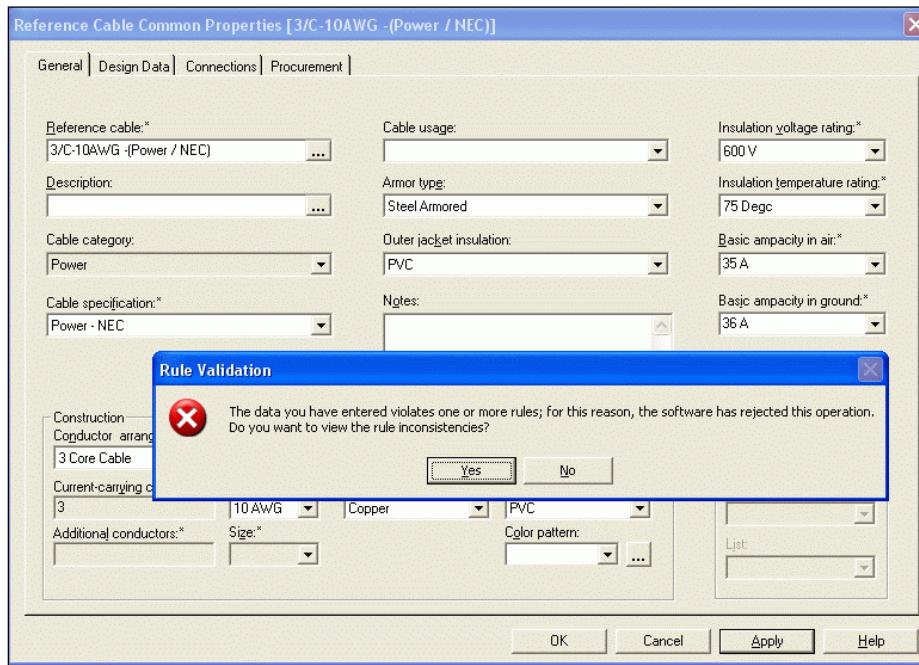
You may define rules on reference items by setting the 'Apply to Reference data only' flag.



for reference cables to validate that basic ampacity in air is not less than basic ampacity in ground:



Save the rule and test it in SmartPlant Electrical.



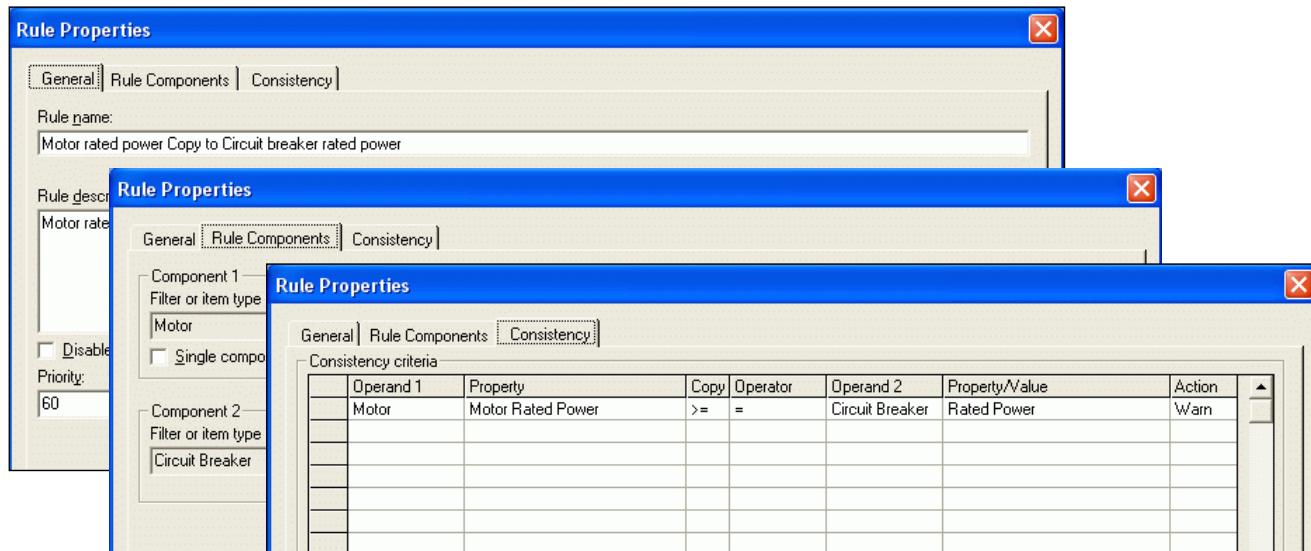
Copy data with rule manager

The following copy functions are available:

None
> Copy Item1 to Item2 if Null
< Copy Item2 to Item1 if Null
<> Copy Bidirectional if Null
>= Copy Item1 to Item2
<= Copy Item2 to Item1
<=> Copy Bidirectional

Copy Function	Description
None	= No copy action across properties.
> Copy Item 1 to Item 2 if Null	The copy operation is from Item 1 -> Item 2 only if value in property of Item 2 is NULL, otherwise there is no copy.
> Copy Item 2 to Item 1 if Null	The copy operation is from Item 2 -> Item 1 only if value in property of Item 1 is NULL, otherwise there is no copy.
<> Copy Bidirectional if Null	The copy operation is either way, from Item 1 -> Item 2, or vice versa, depending on the item its property was changed, but only if the property of the item to be copied to be NULL before the copy.
>= Copy Item 1 to Item 2	The copy operation is from Item 1 -> Item 2.
<= Copy Item 2 to Item 1	The copy operation is from Item 2 -> Item 1.
<=> Copy Bidirectional	The copy operation is either way, from Item 1 -> Item 2, or vice versa, depending on the item its property was changed.

The copy operation, as specified, can only be associated with the operator being set to equal ('='), as appears in the following example (all others operator expressions cannot result in copy data):



In the above example, we tell the system to copy Motor's Motor Rated Power value, to Circuit Breaker's Rated Power value (forcing this direction to be equal).

The “action” column value specifies to the system what should be the reaction of the system in case user is trying to cause an inconsistency.

In the above example, trying to change the circuit breaker data from what has been copied from its related motor will violate the consistency and the system will warn the user.

Selecting the “Reject” action will result in not accepting any inconsistency and will never result in an inconsistency. Trying to violate the rule will always cause a correction or a rejection.

It is important to understand that the copy feature, as defined, basically will try to force the values of the relevant properties to follow the result of the rule (single directional or bi-directional).

Note: The rule will be implemented ONLY IN CASE THAT PRIOR TO THE VALUE CHANGES, THE VALUES OF THE PROPERTIES IN THE RULE WERE THE SAME.

In case of the values of the properties prior to the change were dissimilar, the system will assume that user intentionally decided to have the inconsistency, and will not try to force the copy, but instead, will warn the user of the inconsistency and present him with the possible suggestions to correct the inconsistency.

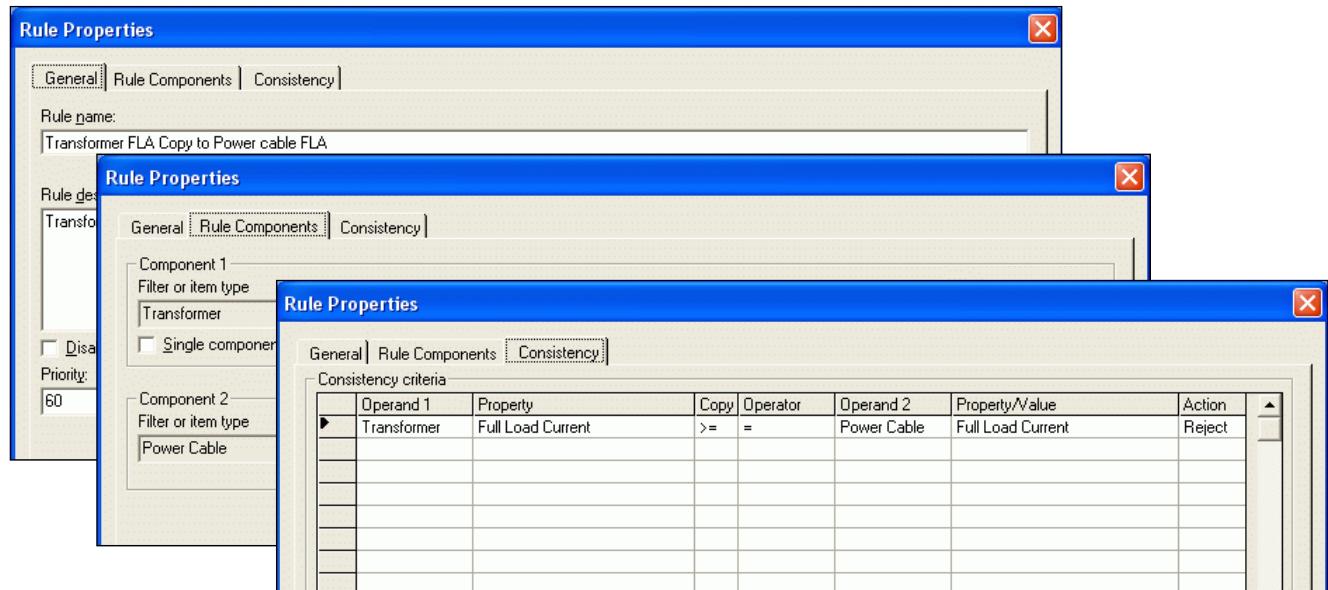
The following examples will demonstrate how we can use rules to copy data.

Propagating data upstream

Transformer and its upstream connected power cable

The rule is to make sure that transformer full load current and its upstream connected power cable full load current are the same.

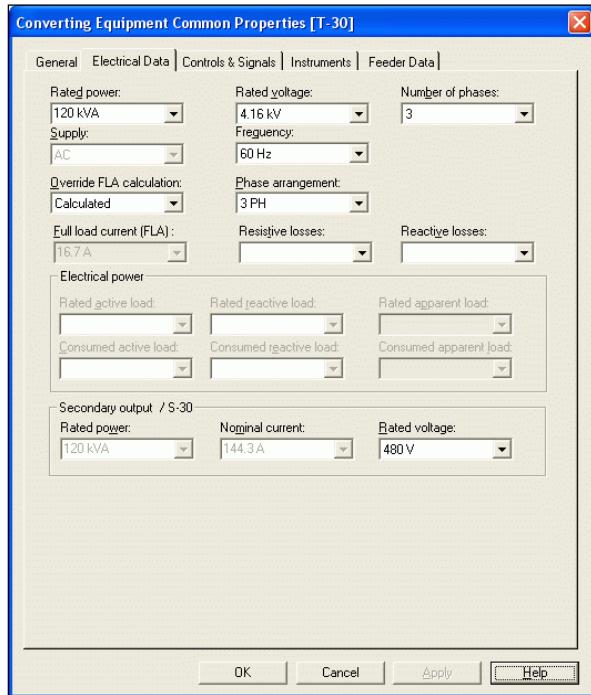
Create and save the following rule.



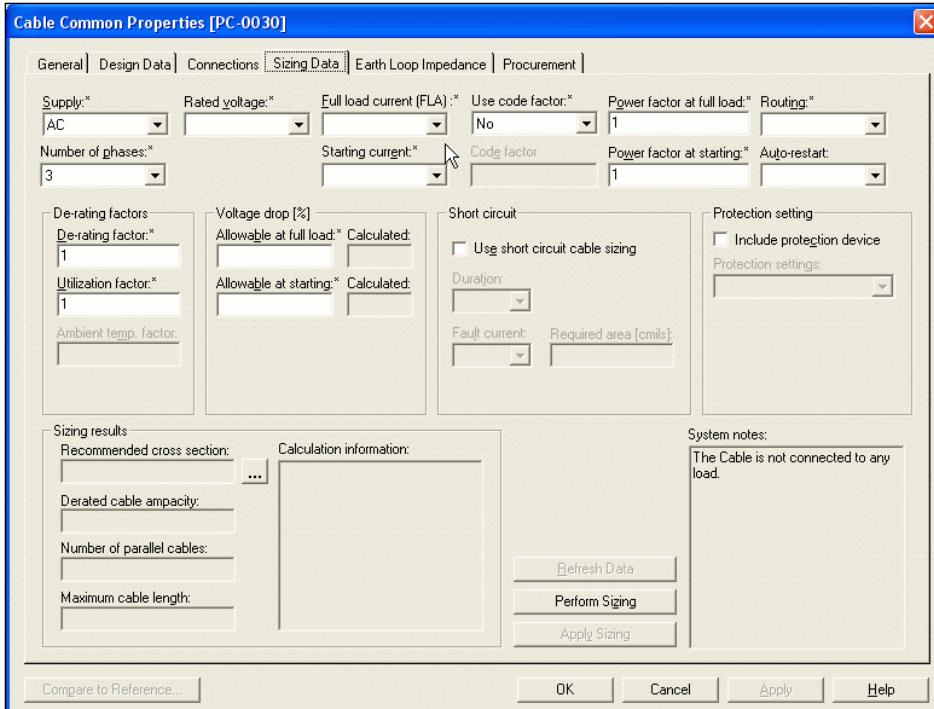
Note: Operand 2 is filter for power cable.

Reopen SmartPlant Electrical and create the following items to test the rule.

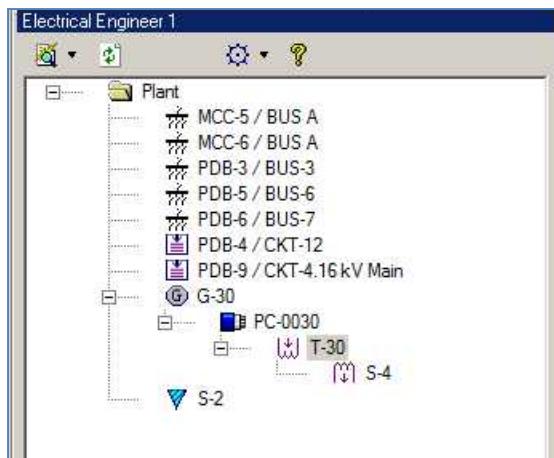
1. Create new generator, G-30, transformer, T-30 (2-winding transformer), and power cable PC-0030 (user typical cable '3+1/C-1/0AWG -(Power / NEC)').
2. Connect cable PC-0030 to generator, G-30.
3. Enter the following values to the transformer, T-30.

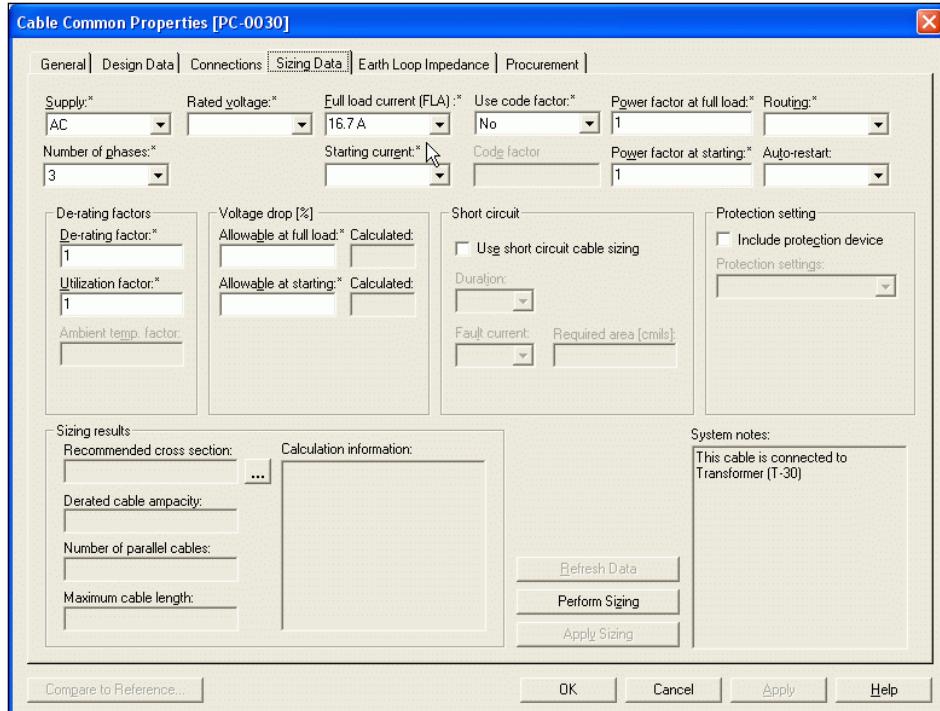


4. Before the cable is connected to the transformer, its full load current is NULL.



- Associate transformer T-30 primary, to cable, PC-0030 in EE. The association will propagate the transformer, FLA into the cable, FLA.





Change the cable FLA to other value will not be saved (it let you display the change but reopen the cable common properties to refresh the screen display the value before the change).

Propagating data downstream

Bus and its downstream disconnect electrical equipment

The rule is to copy the bus rated voltage into the disconnect electrical equipment rated voltage. Create and save the following rule.

Operand 1	Property	Copy	Operator	Operand 2	Property/Value	Action
Bus	Rated Voltage	>=	=	Disconnect Electrical Equipment	Rated Voltage	Warn

To test the rule, reopen SPEL, set the rated voltage to 460 V for PDB-10, BUS A. Verify that all the circuit F2A internals got the rated voltage populated with 460 V automatically from the rule.

Propagating load data into circuit internals

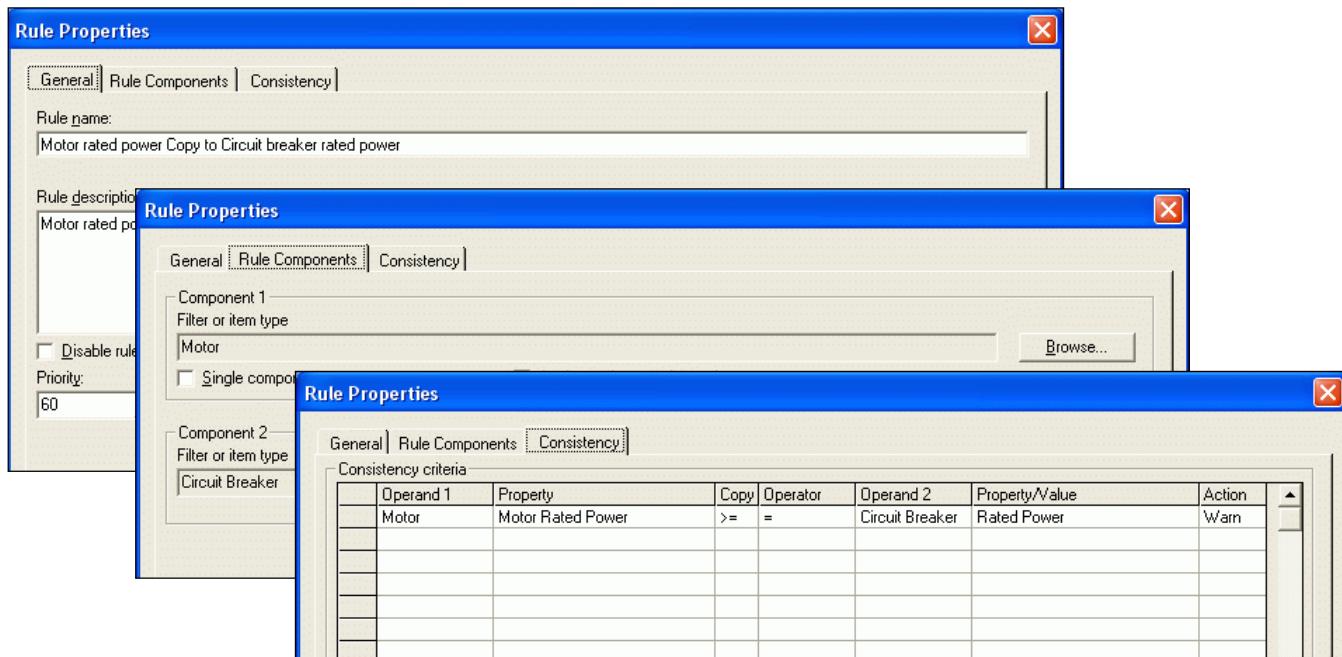
The rule manager, let us validate the connected equipment to the circuit internals. It allows propagate data from connected equipment to their feeder protection and switching equipment, and also validate whether there is an inconsistency between the size of the load and its protection device.

This feature and functionality supports 2 main issues:

1. Propagating load data upstream to circuit internal, disconnect and protection equipment.
 2. Using the load data properties as key fields populates the circuit internals with matching look up table's data. This operation will result in actually "selecting" the suitable circuit breaker or fuse for connected load.

Motor rated power to circuit breaker rated power

Set rule to validate and copy motor rated power to its circuit breaker rated power.

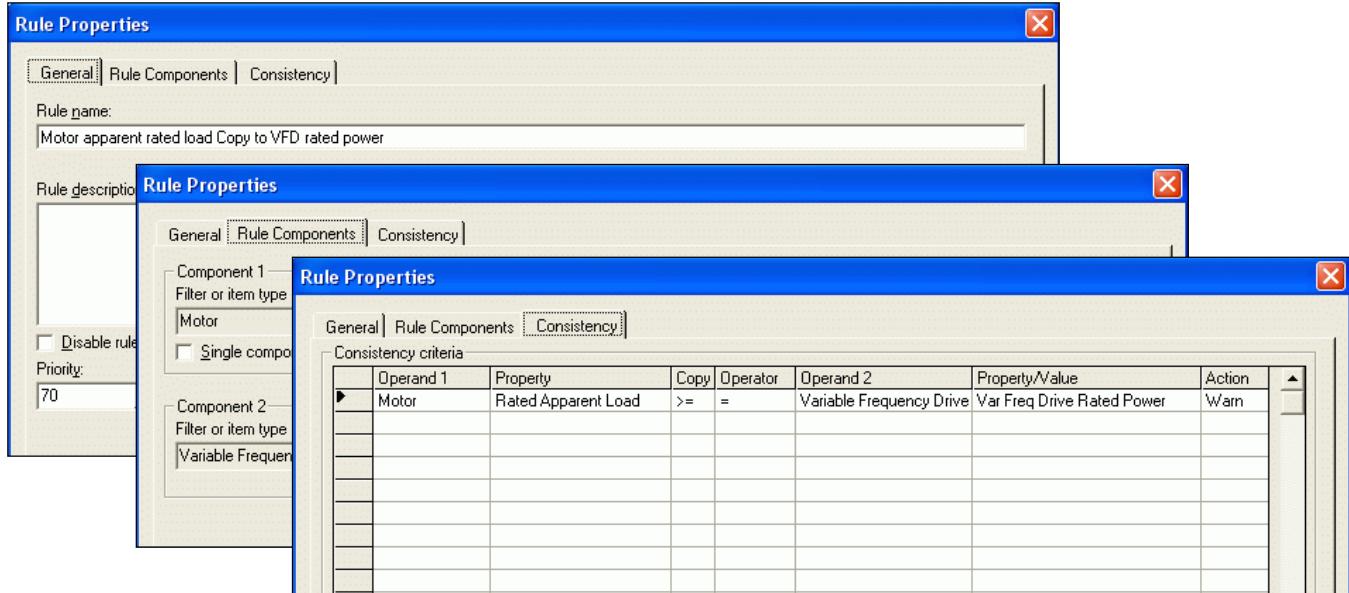


Save the rule.

To test the rule, reopen SPEL. Create circuit breaker, 'CB-200' in feeder circuit, 'FED-200' (MCC-6). Set the motor rated power of motor, 'M-200' to 10 hp. Verify that 'CB-200' got the rated power populated with 10 hp automatically from the rule.

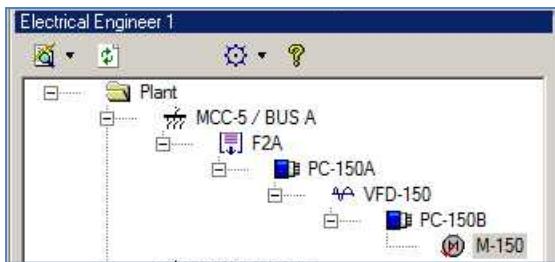
Copying data from loads to their upstream converting equipment

Set a rule to validate and copy the motor apparent rated load to VFD rated power.



Save the rule.

To test the rule, reopen SPEL. Select motor M-150 and verify it contains value for its ‘Rated Apparent Load’. Verify that ‘VFD-150’ got the ‘Var Freq Drive Rated Power’ populated with the same value automatically from the rule.



Fix Inconsistencies

This option checks for rule violations, performs any copy rule operations, fixes any inconsistencies that are found, and displays 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 Actions > **Fix Inconsistencies**.
 2. To display the log file as an .xls file, on the **Fix Inconsistencies** 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.

SmartPlant Electrical Import Manager

This lab will introduce participants to SmartPlant Electrical Import Manager, describe its features and benefits, demonstrate SmartPlant Electrical Import main options and give attendees the opportunity to get some ‘hands-on’ time with SmartPlant Electrical Import Manager.

Participants will be introduced to how to Import the following:

- Defining connectivity to external file source
- Items (Motors, Cables)
- Link between items (Cables <> Motors)
- Importing power distribution board, bus, cells & circuits of various types
- Transformer item type import
- Select List import
- Look-Up Table
- Using Excel as a source
- Using query as a data source

The purpose of this import exercise is to demonstrate the import module capabilities and train the user how this module can be used in order to migrate information from external sources.

Preparations: Database Connectivity

Pre-requisites: Microsoft Access file used for this exercise ‘**Spel_import.mdb**’

Place this file in a new directory on your local hard drive.

e.g. D:\Program Files\SmartPlant\Electrical\ImportDemo\ .

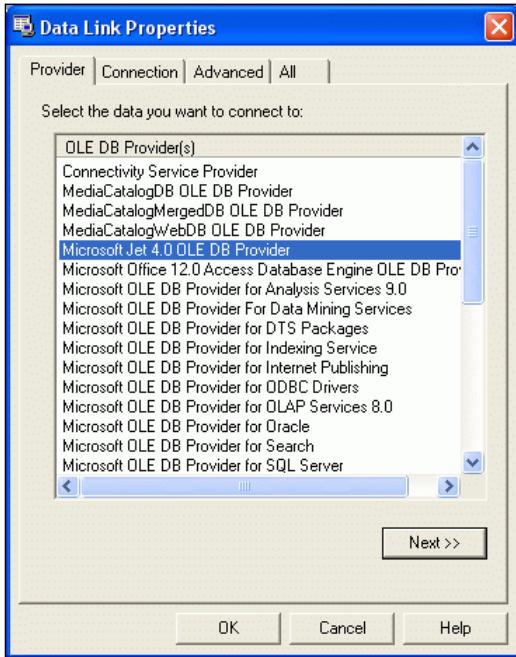
1. Run Import Manager.
2. Click **File > New > Import Project** to open the save as windows.
3. Browse to the folder you would like to save your new import project xml file.

e.g. D:\Program Files\SmartPlant\Electrical\ImportDemo\

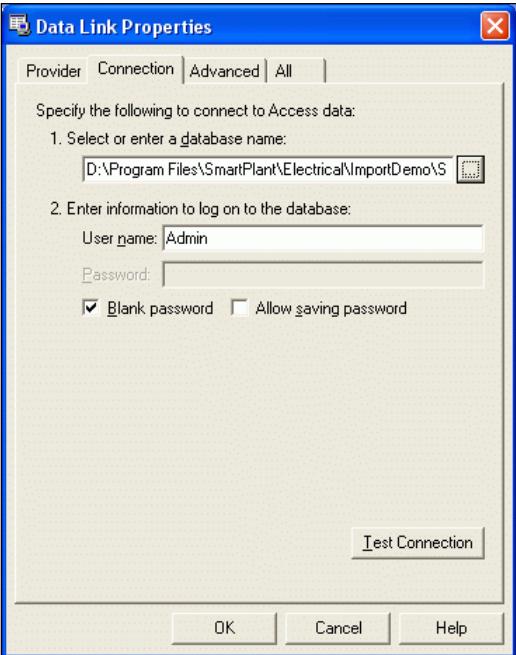
Name the new import project ‘**Spel_Import_Access_Labs1-8.xml**’ and select save.

4. Click **Tools > Options**.
5. On the **Data Source** tab, click **New**.

6. On the **Data Link Properties** dialog box, select **Microsoft Jet 4 OLE DB Provider**.



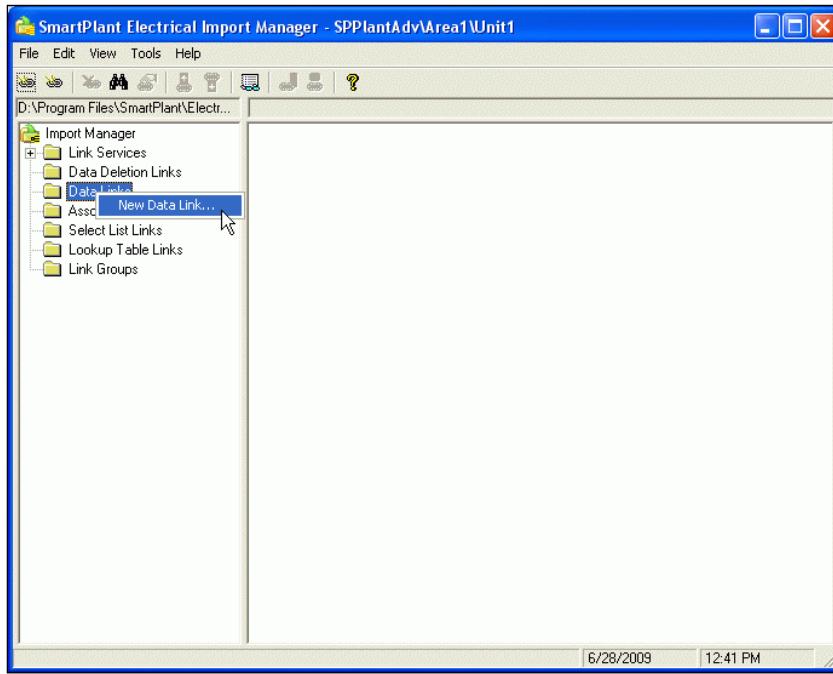
7. Click **Next**.
8. Click the **Connection** tab.
9. Select the MS Access file, '**Spel_import.mdb**' and click **Test Connection** to verify that you have a connection to the database.



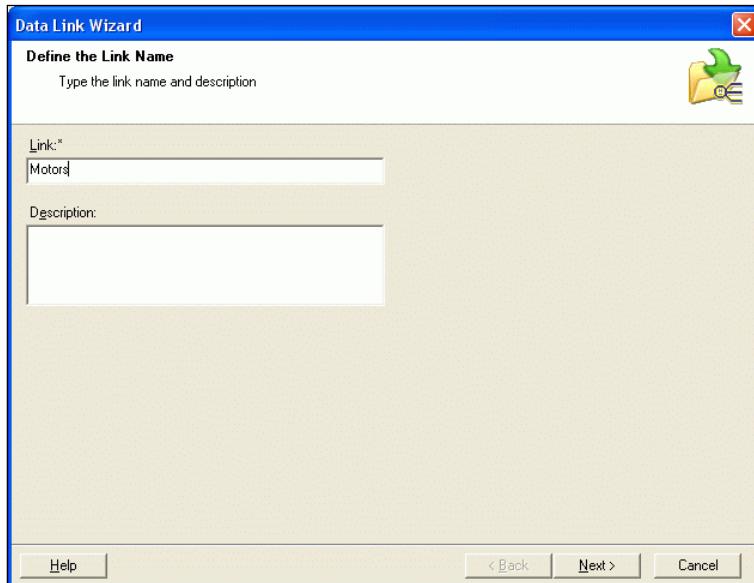
At this point you are ready to start defining the links for the import.

Import 1 – Motors List

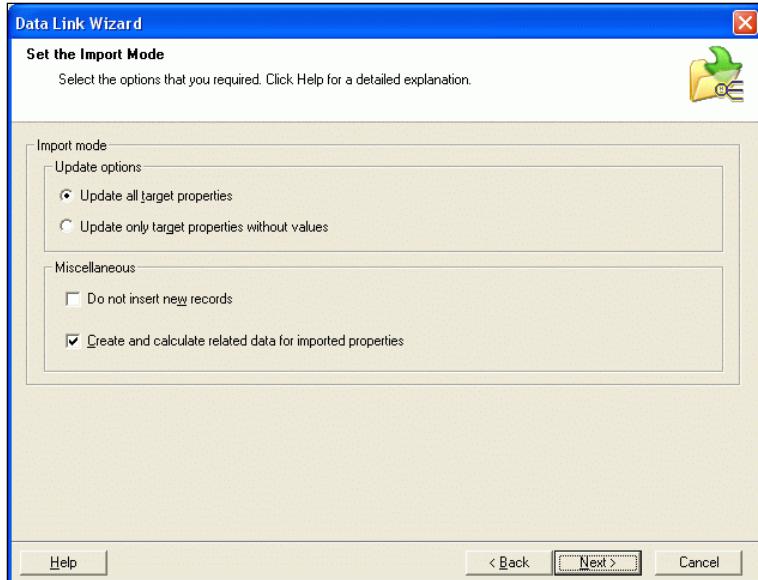
1. Run the Import Manager.
2. Right-click the **Data Links** folder, and on the shortcut menu, click **New Data Link**.



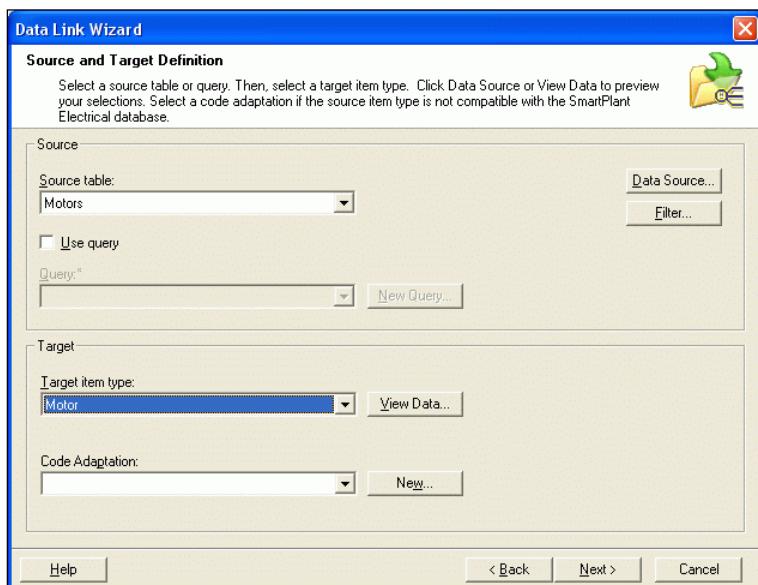
3. In the **Link** box, type **Motors** and click **Next**.



Except the default settings in the ‘Set the Import Mode’ screen and click Next.



4. In the **Source table** list, select **Motors**.
5. In the **Target item type** list, select **Motor**.



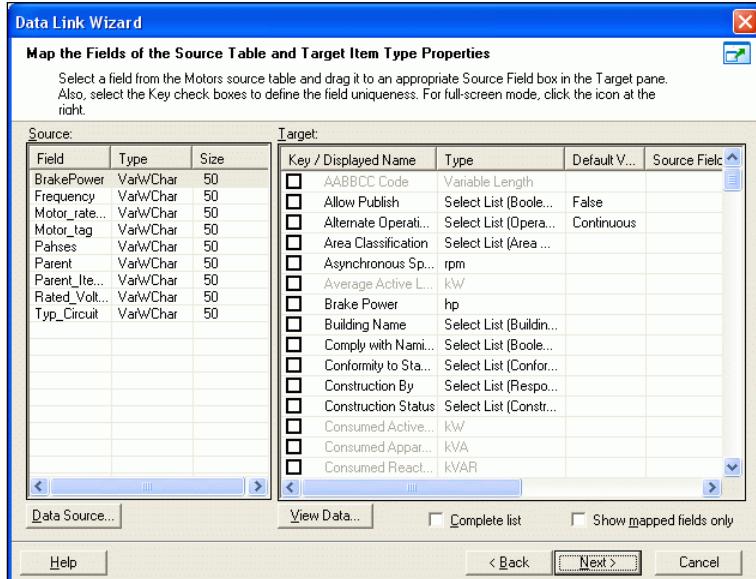
 **Tip:** Click **Data Source** to see the list of motors you are going to import.

Data Source - Motors				
Z	A	Y	X	
Motor_tag	Motor_rated_power	Parent	Parent_item_class	Rated_Volt
M-001	75 hp	M-001-P	Cable	380 V
M-002	10 hp	M-002-P	Cable	380 V
M-003	75 hp	M-003-P	Cable	380 V
M-004	75 hp	M-004-P	Cable	380 V
M-005	50 hp	M-005-P	Cable	380 V

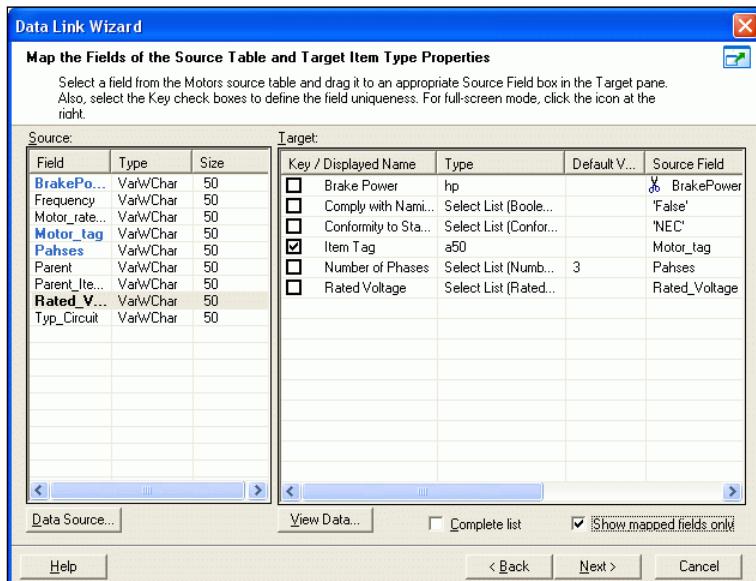
Total Records: 5

Target	Source
Brake Power	BrakePower
Comply with Naming Conventions	'False'
Conformity with Standard	'NEC'
Item Tag	Motor_Tag
Motor Rated Power	Motor_Rated_Power
Number of Phases	Phases
Rated Voltage	Rated_Voltage

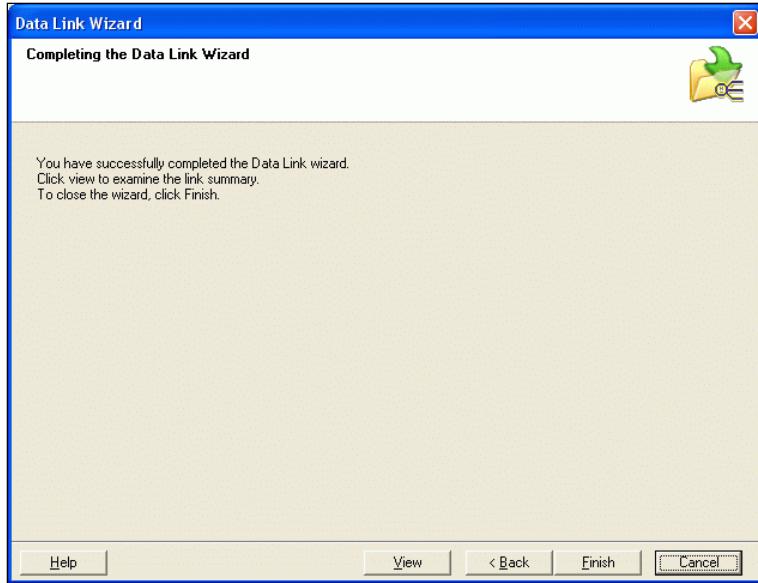
6. Make the mapping, by drag and drop the source field from the source section to the match field in the target section.



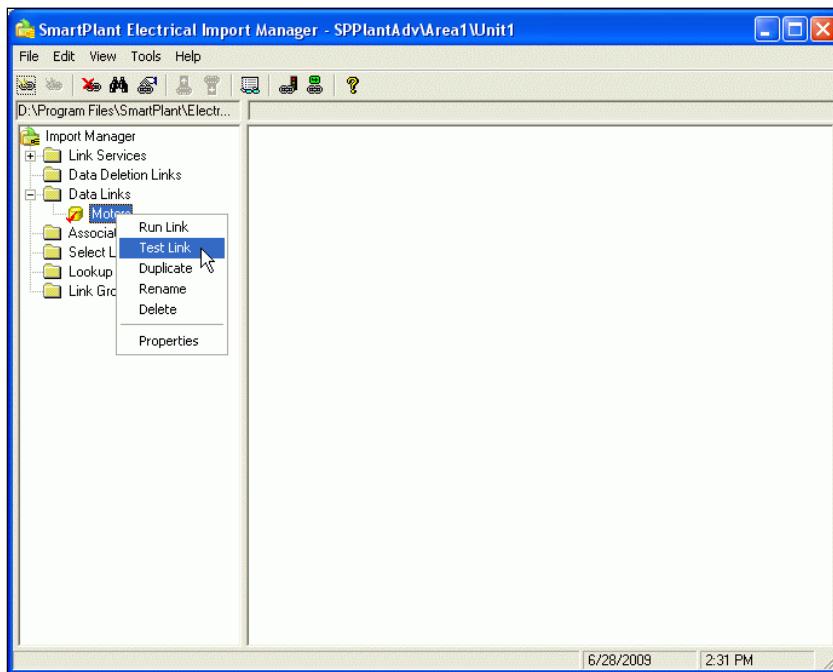
When done, you should see the following mapping.



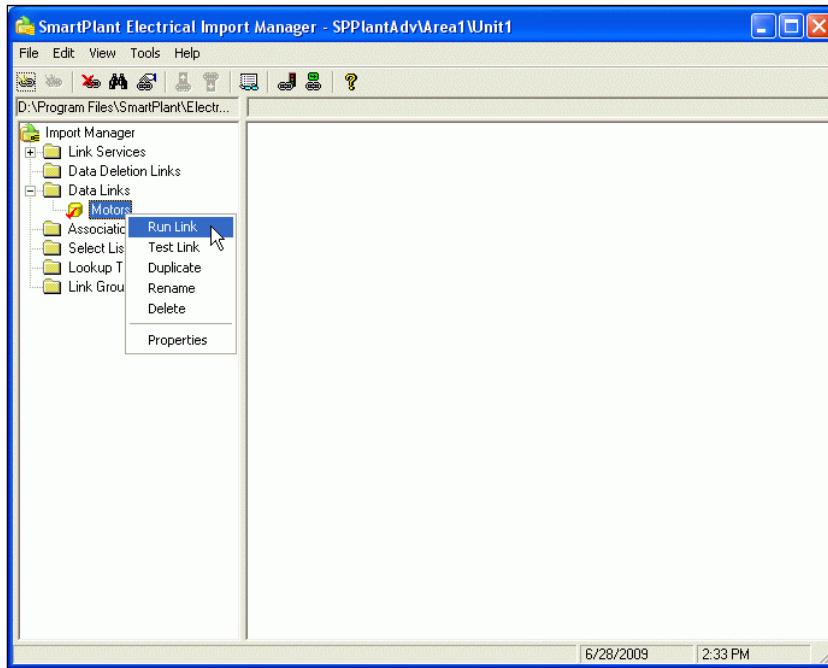
7. Select next and Finish.



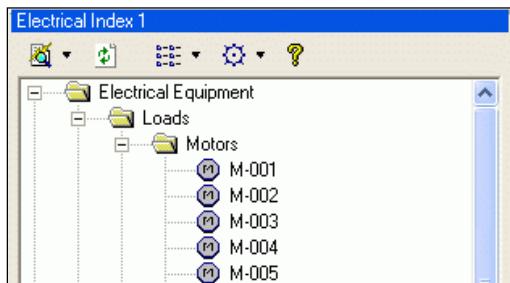
Right-click on the new link, and on the shortcut menu, clicks **Test Link**.



8. Right-click on the new link, and on the shortcut menu, clicks **Run Link**.

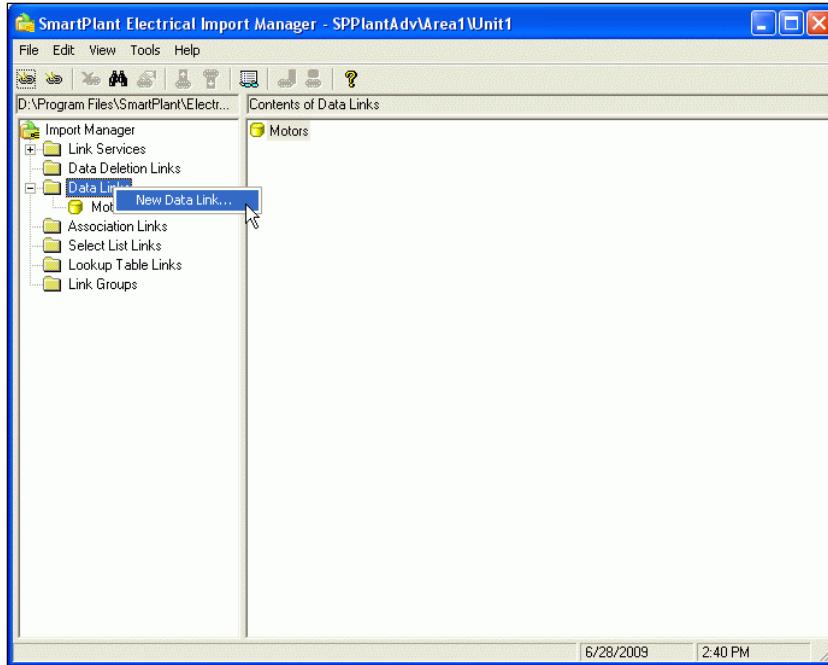


9. Open SmartPlant Electrical and test the result.

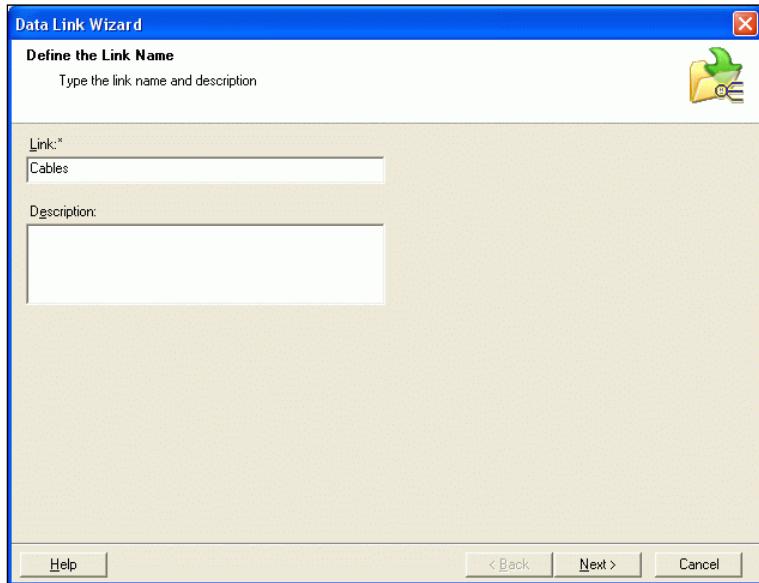


Import 2 – Cable List

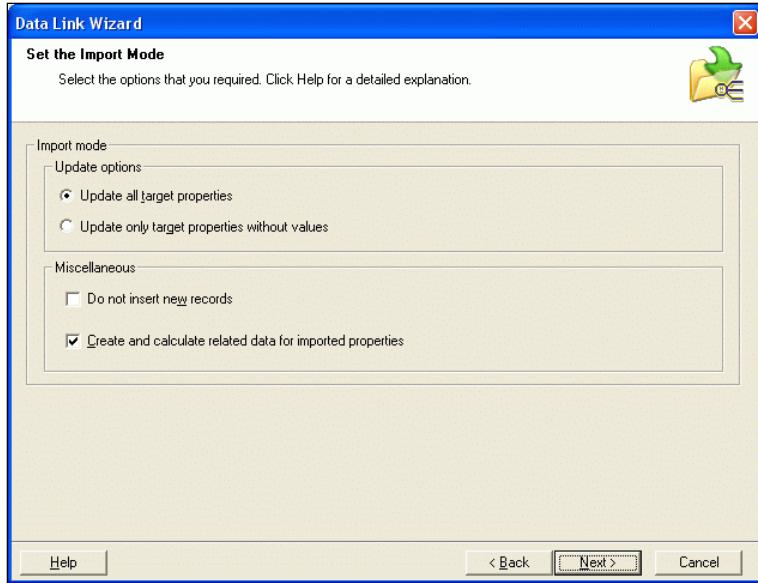
1. Run the Import Manager.
2. Right-click the **Data Links** folder, and on the shortcut menu, click **New Data Link**.



3. In the **Link** box, type **Cables** and click **Next**.

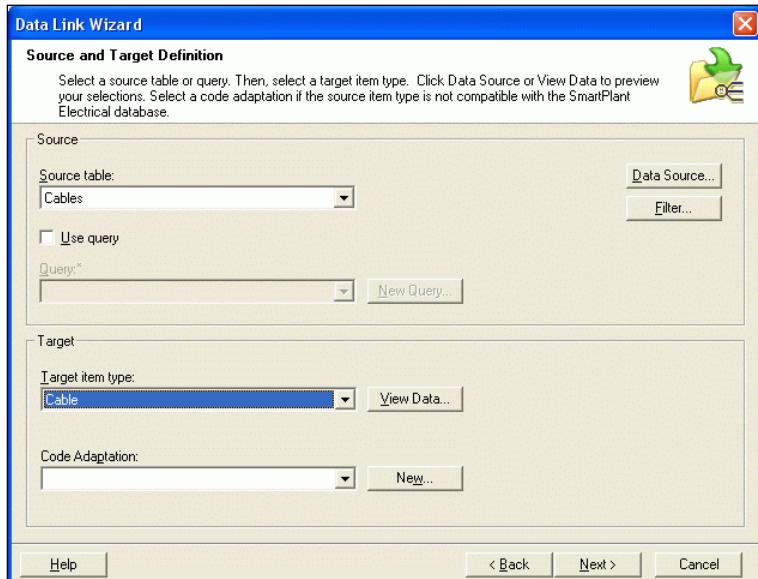


Except the default settings in the ‘Set the Import Mode’ screen and click Next.



4. In the **Source table** list, select **Cables**.

5. In the **Target item type** list, select **Cable**.



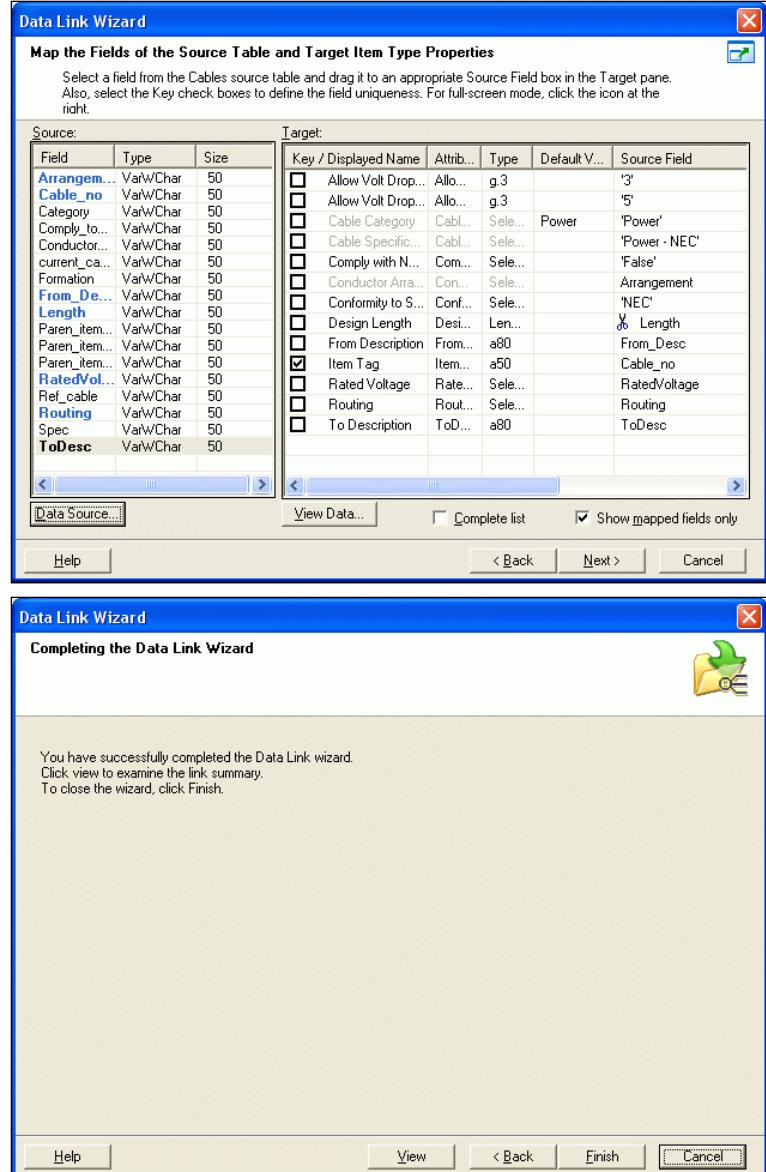
Tip

Click **Data Source** to see the list of cables you are going to import.

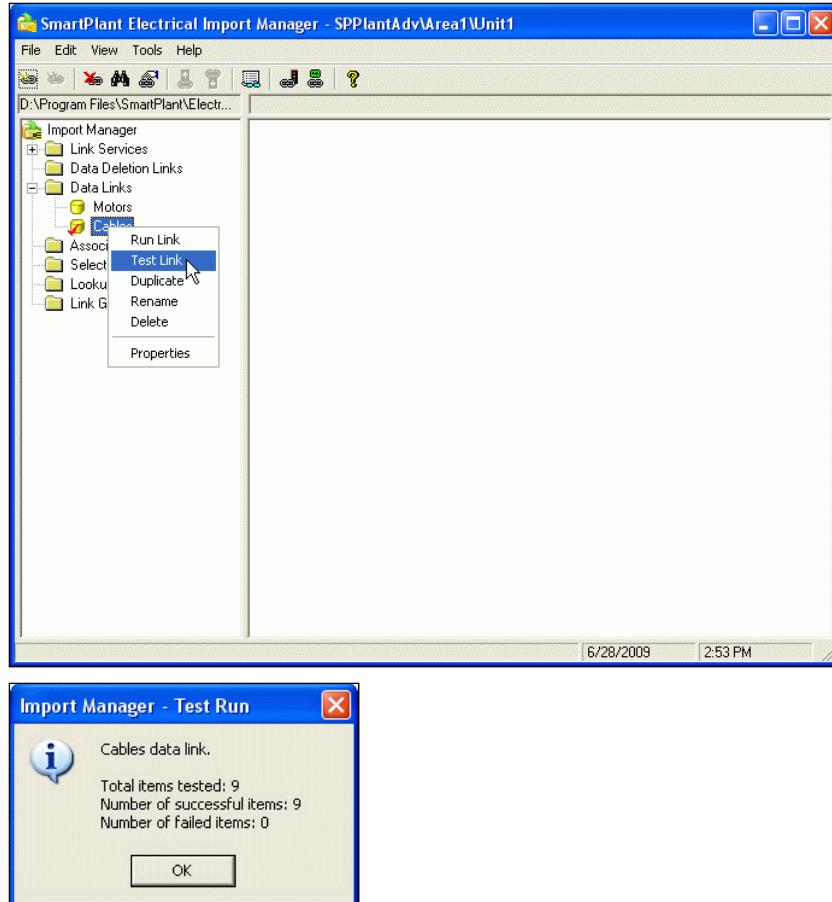
Data Source - Cables				
Cable_no	Paren_item_tag	Paren_item_tag1	Paren_item_class	Comply_to
M-001-P	CKT-M-001		Feeder	False
M-002-P	CKT-M-002		Feeder	False
M-003-P	CKT-M-003		Feeder	False
M-004-P	CKT-M-004		Feeder	False
M-005-P	VFD-08		VFD	False
M-005-P/2	CKT-M-005		Feeder	False
P-T-001	G-001(Import)		Generator	False
P-T001-X	T-001	X	Transformer Component	False
P-T001-Y	T-001	Y	Transformer Component	False

6. Drag and drop the fields from left to right as shown below (or type text strings).
7. Click **Next** to continue.
8. On the last page of the wizard, click **Finish**.

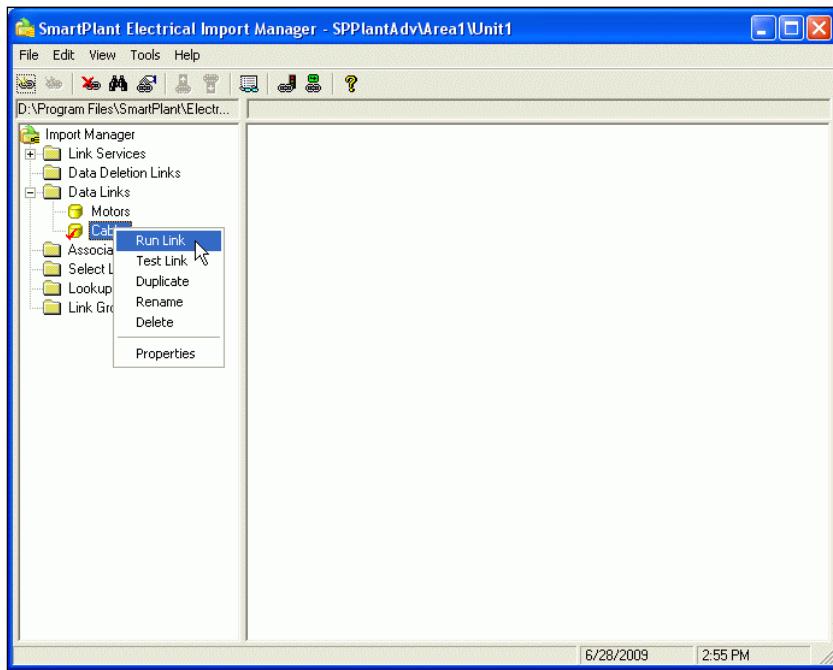
Target	Source
Allow Volt Drop Percent Running	'3'
Allow Volt Drop Percent Start	'5'
Cable Category	'Power'
Cable Specification	'Power - NEC'
Comply with Naming Conventions	'False'
Conductor Arrangement	Arrangement
Conformity to Standard	'NEC'
Design Length	Length
From Description	From_Desc
Item Tag	Cable_No
Rated Voltage	Rated_Voltage
Routing	Routing
To Description	ToDesc



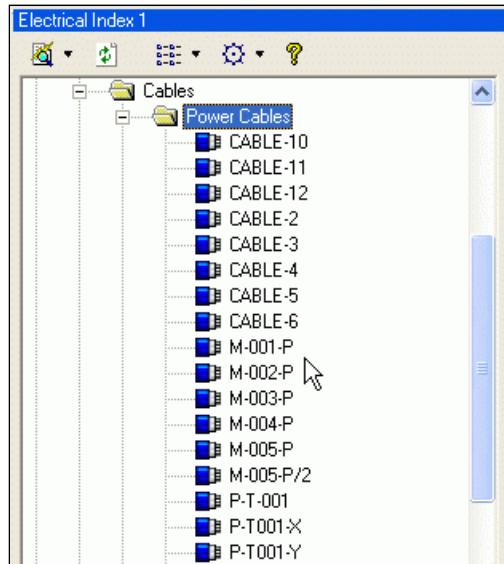
- Right-click the link, and on the shortcut menu, click **Test Link**.



10. Right-click the link, and on the shortcut menu, click **Run Link**.

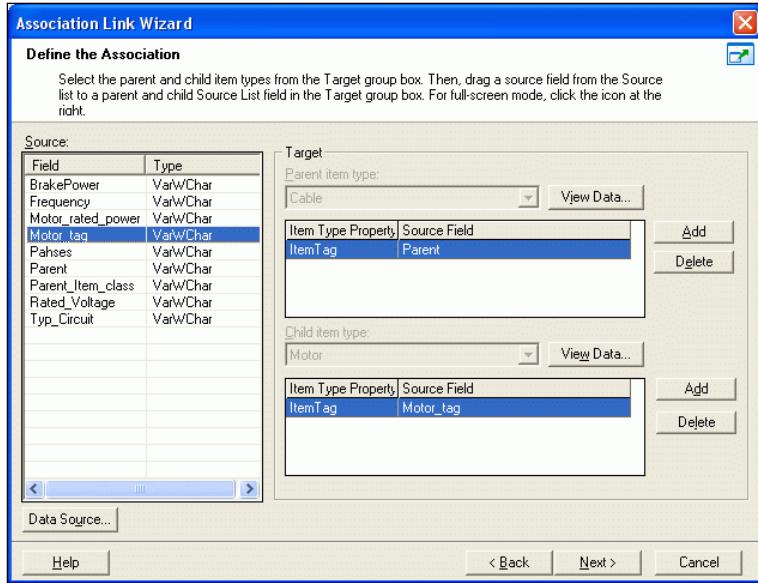


11. Test the result in SmartPlant Electrical.

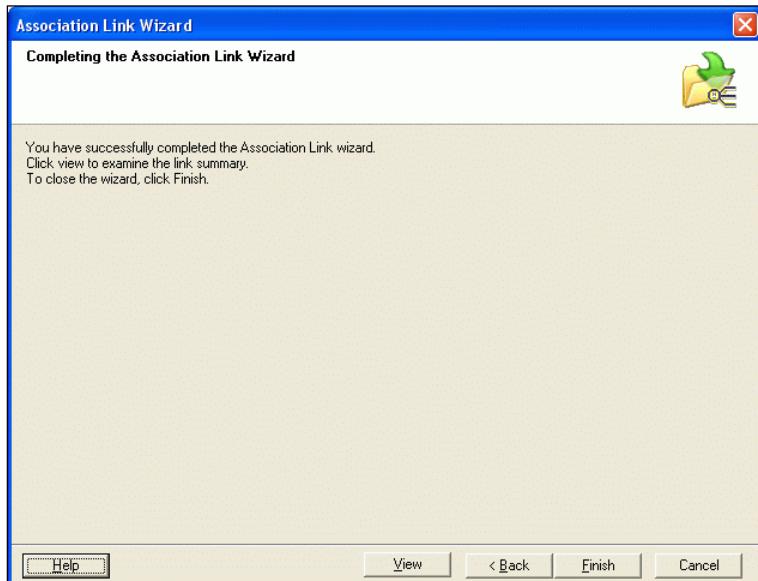


Import 3 – Cables <> Motors Link

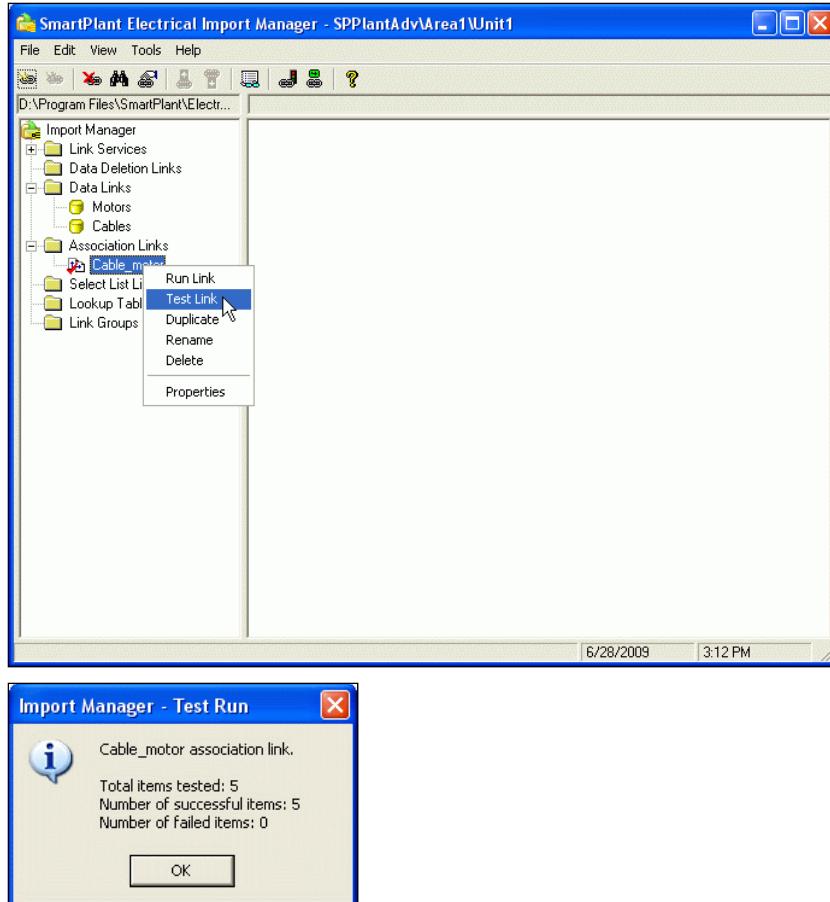
1. Run the Import Manager.
2. Right-click the **Association Links** folder, and on the shortcut menu, click **New Association Link**.
3. In the **Link** box, type **Cable_motor** and click **Next**.
4. In the **Source table** list, select **Motors**.
5. Click **Next**.
6. Target Parent Item Type is ‘Cable’ with ‘Item Tag’ as the main ‘Parent’ field. Target Child Item Type is ‘Motor’ with ‘Item Tag’ as the main field, which is ‘Motor_Tag.’ Both ‘Parent’ and ‘Motor_Tag’ fields should be dragged from the fields list to the Source on the left side (to the right target side).



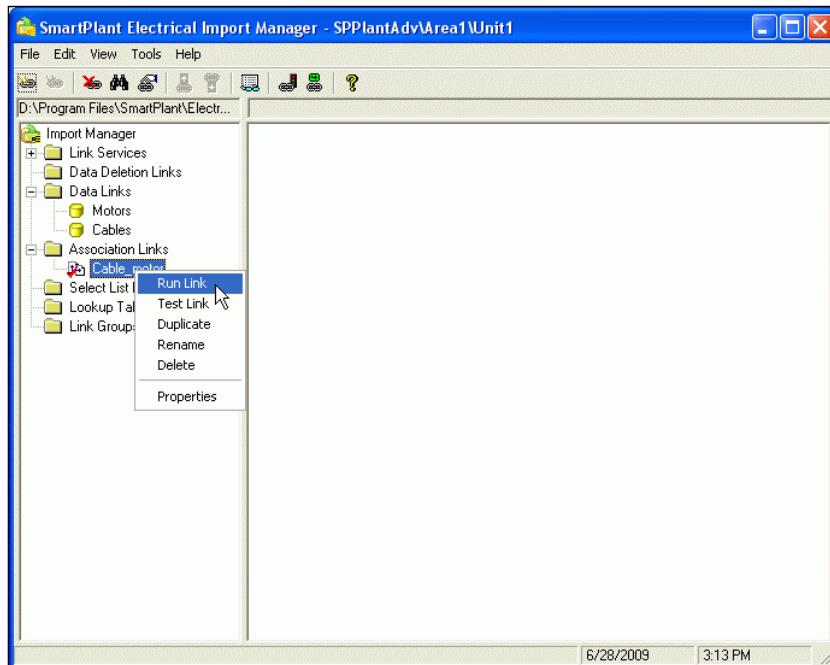
7. Click **Next** to continue.
8. On the last page of the wizard, click **Finish**.



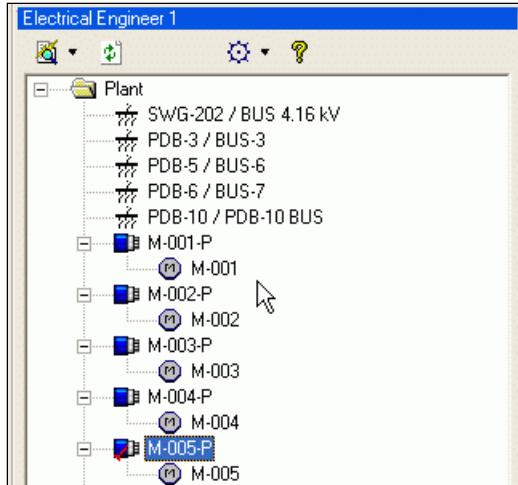
9. Right-click the link, and on the shortcut menu, click **Test Link**.



10. Right-click the link, and on the shortcut menu, click **Run Link**.



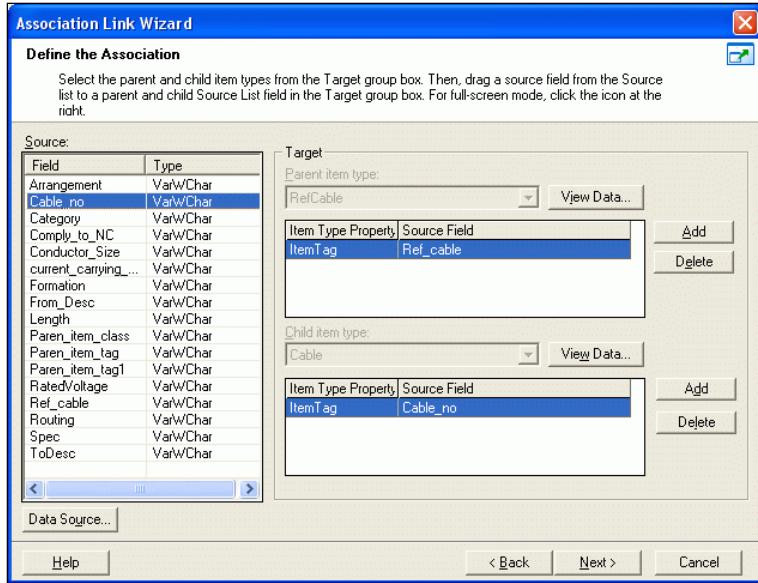
11. Test the result in SmartPlant Electrical.



Import 4 – Associate a Project Cable with Ref Cable

This lab will link two of the cables that were imported with reference cables. Once this operation is completed, electrical properties and conductors will match those of the ref cables.

1. Run the Import Manager.
2. Right-click the **Association Links** folder, and on the shortcut menu, click **New Association Link**.
3. In the **Link** box, type **RefCable to Cable** and click **Next**.
4. In the **Source table** list, select **Cables**.
5. Click **Next**.
6. Target Parent Item Type is ‘RefCable’ with ‘Item Tag’ as the main ‘Parent’ field. Parent source field = ‘Ref_Cable’. Target Child Item Type is ‘Cable’ with ‘Item Tag’ as the main field, Child source field = ‘Cable_no’.



7. Click **Next** to continue.
8. On the last page of the wizard, click **Finish**.
9. Right-click the link, and on the shortcut menu, click **Test Link**.

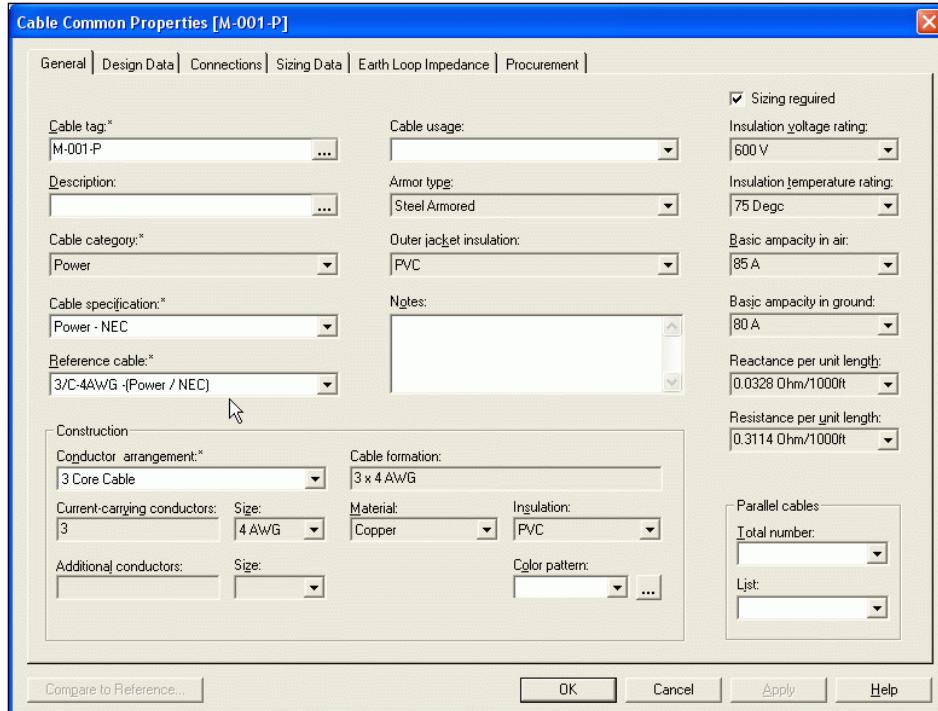


10. Right-click the link, and on the shortcut menu, click **Run Link**.

11. Test the result in SmartPlant Electrical.

Tip

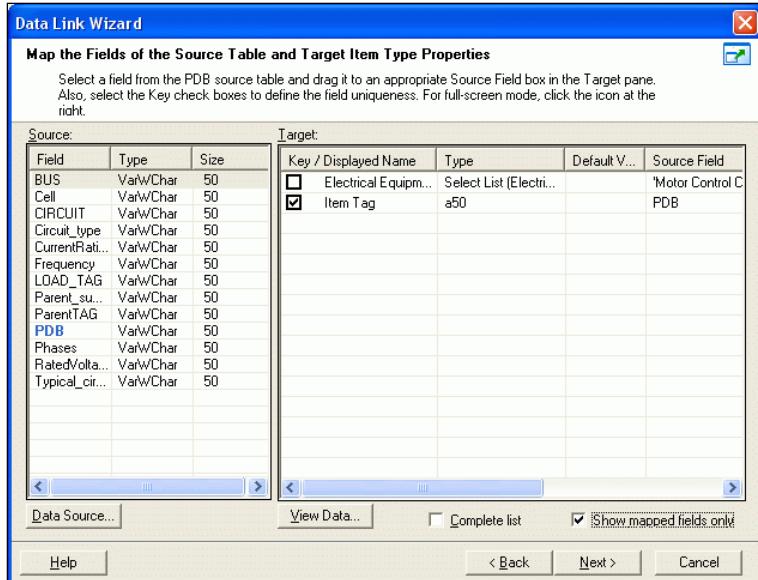
Cables M-001-P & M-002-P should be associated with 3 conductors, 4AWG cable. Note that electrical properties of the reference cable were copied to these cables.



Import 5 – Power Distribution, Bus, Cells & Circuits

1. Run the Import Manager.
 2. Right-click the **Data Links** folder, and on the shortcut menu, click **New Data Link**.
 3. In the **Link** box, type ‘**PDB**’ and click **Next**.
Except the default settings in the ‘Set the Import Mode’ screen and click **Next**.
 4. In the **Source table** list, select **PDB**.
-  **Tip**
5. Click **Data Source** to see the list of PDBs you are going to import.
 5. In the **Target item type** list, select **PowerDistributionBoard**.
 6. Link the following properties.

Target	Source
Item Tag	PBD
Electrical Equipment Type	'Motor Control Center'



7. Test and run the link.
8. Check the results in SmartPlant Electrical.

Next, we will import the bus.

Since we shall need to know which bus belongs to which PBD, the PBD name will be imported into a temporary property (Name). Both properties should be marked as ‘Key’ since they define the unique PBD-Bus.

1. Run the Import Manager.
2. Right-click the **Data Links** folder, and on the shortcut menu, click **New Data Link**.
3. In the **Link** box, type ‘Bus’ and click **Next**.

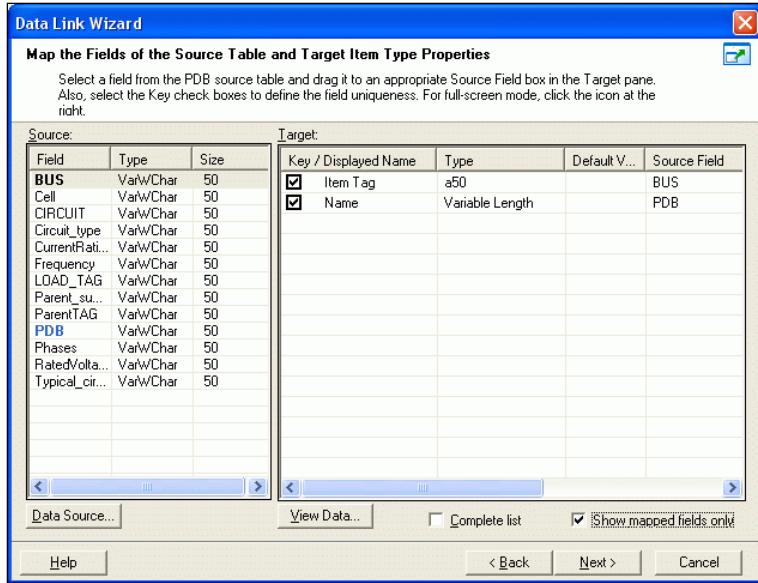
Except the default settings in the ‘Set the Import Mode’ screen and click **Next**.

4. In the **Source table** list, select **PDB**.

Tip

6. Click **Data Source** to see the list of PDBs you are going to import.
5. In the **Target item type** list, select **Bus** and click **Next**.
6. Link the following properties.

Target	Source
Item Tag (key)	Bus
Name (key)	PBD



Test and run the link.

Note: At this time the bus has been imported but you cannot see the bus in SPEL before you associate it with the PDB.

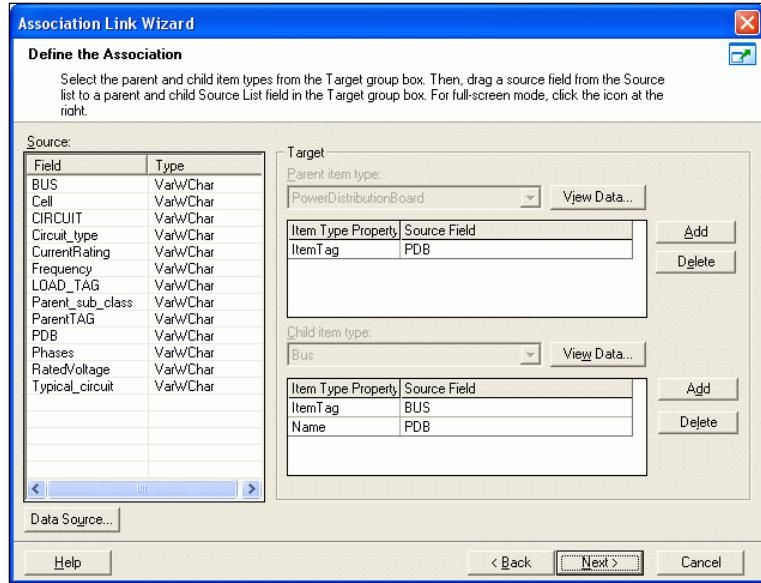
To associate the Bus with the PDB create a new association link:

1. **New Association Link.**
2. Enter '**PDB**' → Source Table is '**PBD**' → **Next**.

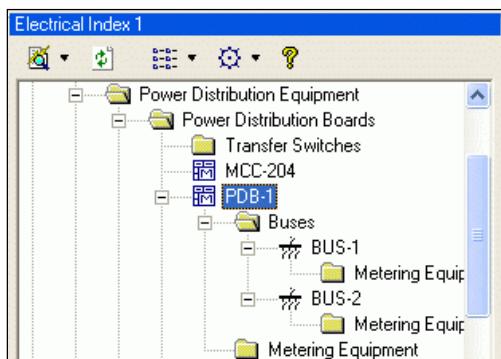
Target Parent Item Type is 'PowrDistributionBoard' with 'Item Tag' as the main 'Parent' field. Parent source field = 'PDB'.

Target Child Item Type is 'Bus' with 'Item Tag' as the main field. Child source field = 'BUS'.

'Name' as the main field, Child source field = 'PDB'



3. Next → Finish.
4. Test and run the link.
5. Check the results in SmartPlant Electrical.



Next stage is to import the circuits, cells and their relationships. If you look at the source data (PDB table) you will see:

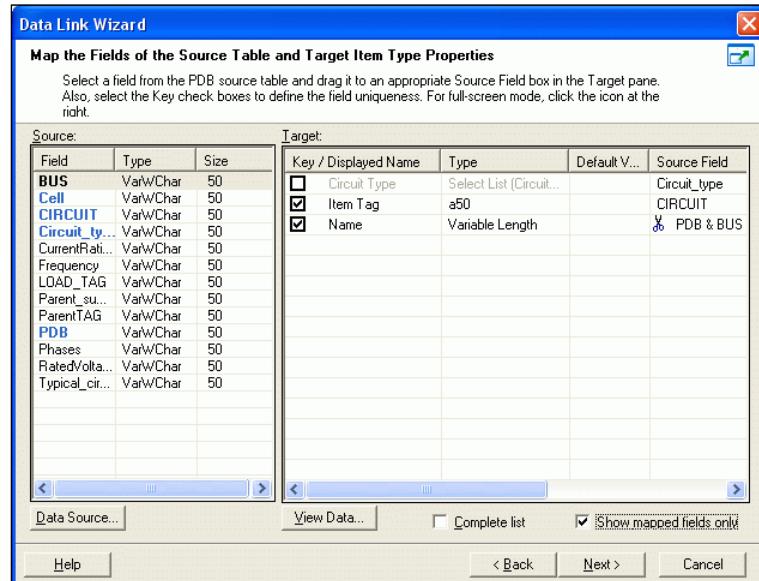
The feeders in the source do not have cells.

Only the incomers, bus-riser & coupler have cells.

Import the Circuits

1. Run the Import Manager → Data Links → New Data Link.
2. Enter ‘Circuits’ → Next → Next → Source Table is ‘PBD’
3. Target Table is ‘Circuit’ → Next.
4. Link the following properties:

Target	Source
Item Tag	Circuit (key)
Name	PDB & BUS & Cell (key)
Circuit Type	Circuit_type

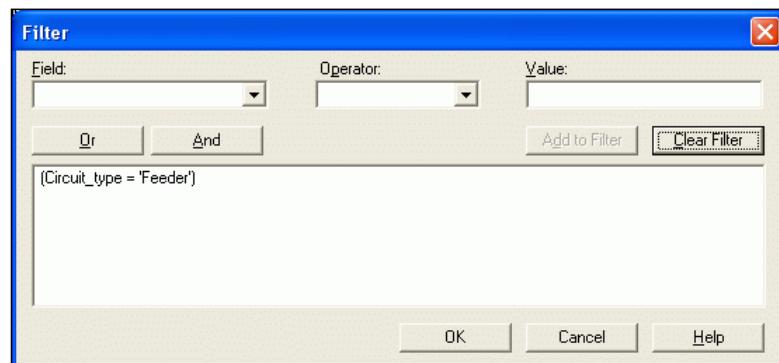


Note that the Name property holds a string that identifies the PDB, the Bus & cell where the circuits are located.

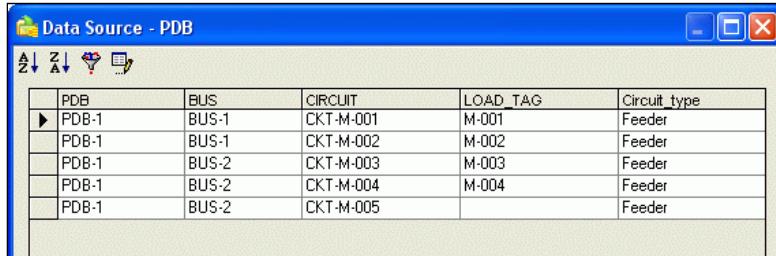
5. **Next → Finish**
6. Test and Run the link.

Associate the Feeder Circuits with the Bus

1. Create a new association link: → Association Links → New Association Link
2. Enter ‘**Feeders to Bus**’ → Source Table is ‘PBD’. Press the ‘Filter’ button in the Source group box and create the following filter:
Field = ‘Circuit_type’, Operator=‘=’, Value=‘Feeder’. Press ‘Add to Filter’.



Click **Data source** to verify that the filter works.



PDB	BUS	CIRCUIT	LOAD_TAG	Circuit_type
PDB-1	BUS-1	CKT-M-001	M-001	Feeder
PDB-1	BUS-1	CKT-M-002	M-002	Feeder
PDB-1	BUS-2	CKT-M-003	M-003	Feeder
PDB-1	BUS-2	CKT-M-004	M-004	Feeder
PDB-1	BUS-2	CKT-M-005		Feeder

3. Click **OK** → **Next**.

Notes

Target Parent Item Type is ‘Bus’

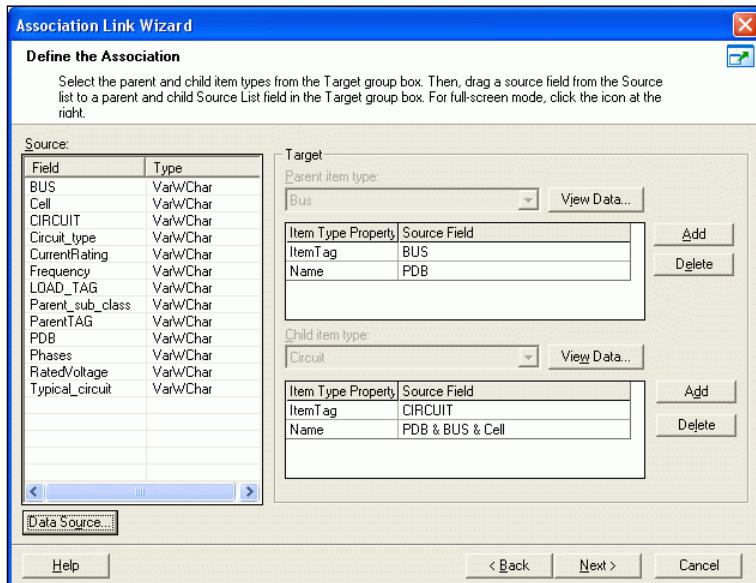
Target parent item type property = ‘Item Tag’, Parent source field=‘Bus’

Target parent item type property = ‘Name’, parent source field = ‘PDB’

Target Child Item Type is ‘Circuit’

Target child item type property = ‘Item Tag’, child source field=‘Circuit’

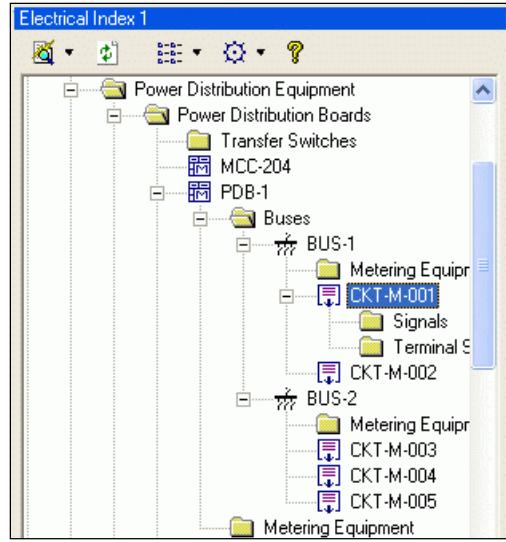
Target child item type property = ‘Name’, child source field = ‘PDB & Bus & Cell’.



4. Click **Next** → **Finish**.

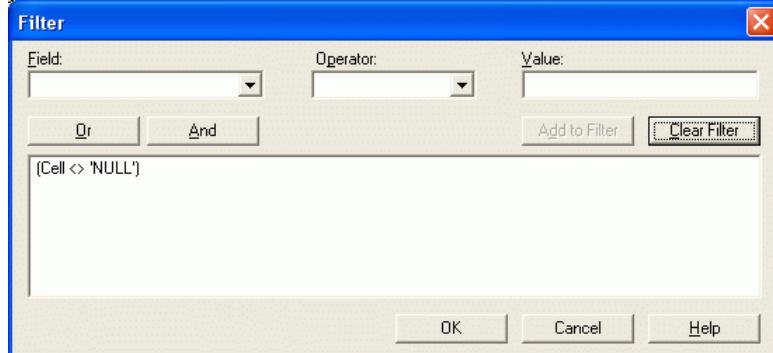
5. Test and Run the link.

6. Check the results in SmartPlant Electrical.



Import Cells

1. Create a new data link as follows: Link name ‘Cells’ → Next → Next.
2. Source Table is ‘PDB’. Click **Filter** in the **Source** group box and create the filter (**Cell <> 'NULL'**) → OK.



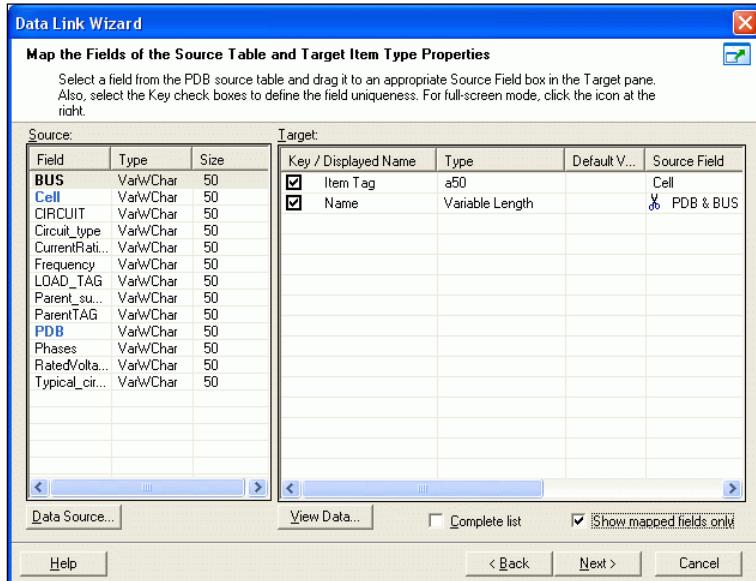
3. Click **Data source** to verify that the filter works.

PDB	BUS	CIRCUIT	LOAD_TAG	Circuit_type	Cell
PDB-1	BUS-1	Bus-1 Incomer		Incimer	[Bus-1] Incimer
PDB-1	BUS-1	Bus-1 Coupler		Coupler	[Coupler]
PDB-1	BUS-2	Bus-2 Incomer		Incimer	[Bus-2] Incimer
PDB-1	BUS-2	Bus-Riser		Bus Riser	[Bus-Riser]

4. Target table is ‘Cell’ → Next.

5. Link the following properties:

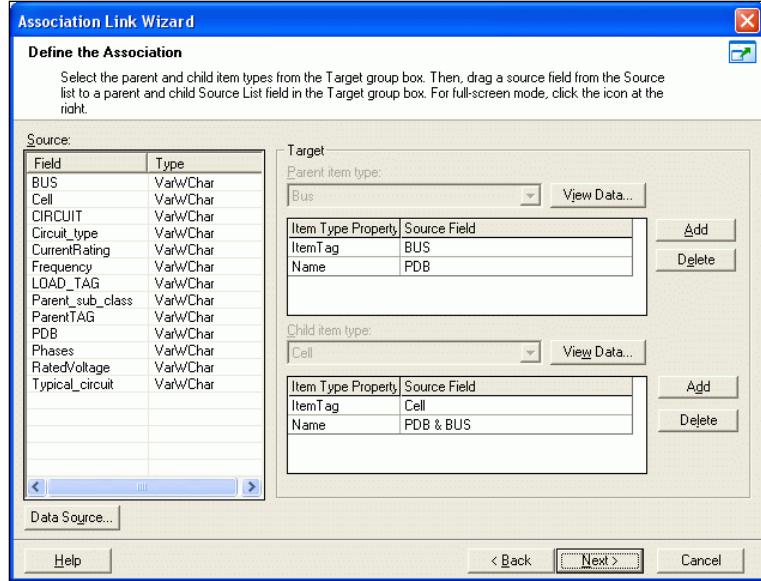
Target	Source
Item Tag	Cell (key)
Name	PDB & BUS (key)



6. **Next → Finish.**
7. Test and Run the link.

Associate the Cells with the Bus

1. Create a new association link: **Association Links → New Association Link.**
2. Enter '**Bus to Cells**' → Source Table is '**PBD**' → **Next**.
3. Target Parent Item Type is '**Bus**'.
4. Target parent item type property = '**Item Tag**', Parent source field = '**Bus**'.
5. Target parent item type property = '**Name**', parent source field = '**PDB**'.
6. Target Child Item Type is '**Cell**'.
7. Target child item type property = '**Item Tag**', child source field = '**Cell**'
8. Target child item type property = '**Name**', child source field = '**PDB & Bus**'

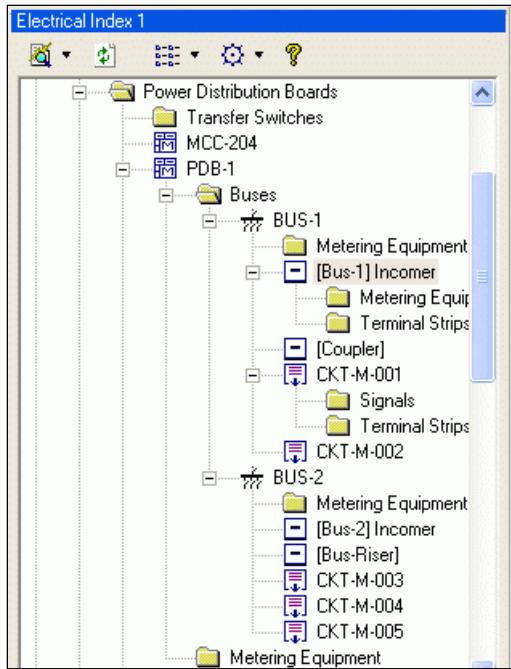


9. Next → Finish.

10. Test and Run the link.



11. Check in SmartPlant Electrical that the new cells are associated with the bus.



Associate the Circuits with the Cells

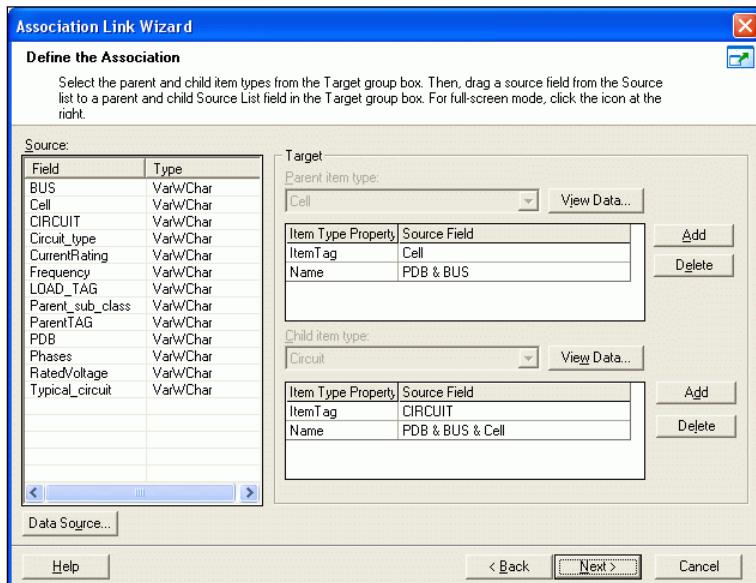
1. Create a new association link: **Association Links → New Association Link.**
2. Enter '**Cells to Circuits**' → Source Table is 'PBD'. Press the 'Filter' button in the Source group box, and create the filter (**Cell <> 'NULL'**) → **OK** → **Next**.

Target Parent Item Type is 'Cell'

- a. Target parent item type property = 'Item Tag', Parent source field='Cell'
- b. Target parent item type property = 'Name', parent source field = 'PDB & BUS'

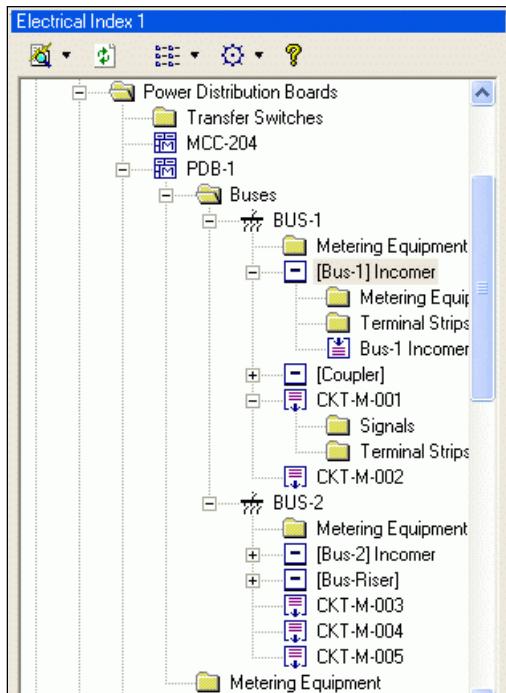
Target Child Item Type is 'Circuit'

- a. Target child item type property = 'Item Tag', child source field='CIRCUIT '
- b. Target child item type property = 'Name', child source field = 'PDB & BUS & CELL'



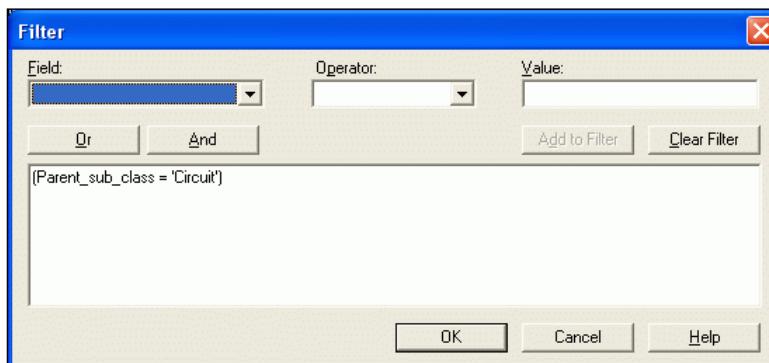
3. **Next → Finish.**

4. Test and Run the link.
5. Check in SmartPlant Electrical that the circuits are associated with the cells.



Associate the Bus-Riser with the Coupler

1. Create a new association link:
Association Links → New Association Link.
2. Enter '**Coupler to bus-riser**' → Source Table is '**PBD**'.
3. Click **Filter** in the **Source** group box and create the filter **Parent_sub_class = 'Circuit'**



View data source to validate that the filter returns one row

Data Source - PDB								
PDB	BUS	CIRCUIT	LOAD_TAG	Circuit_type	Cell	Typical	ParentTAG	Parent_s
PDB-1	BUS-2	Bus-Riser		Bus Riser	[Bus-Riser]		Bus-1 Coupler	Circuit

- Click **Next** to open the define the association windows.

Target parent Item Type is ‘Circuit’

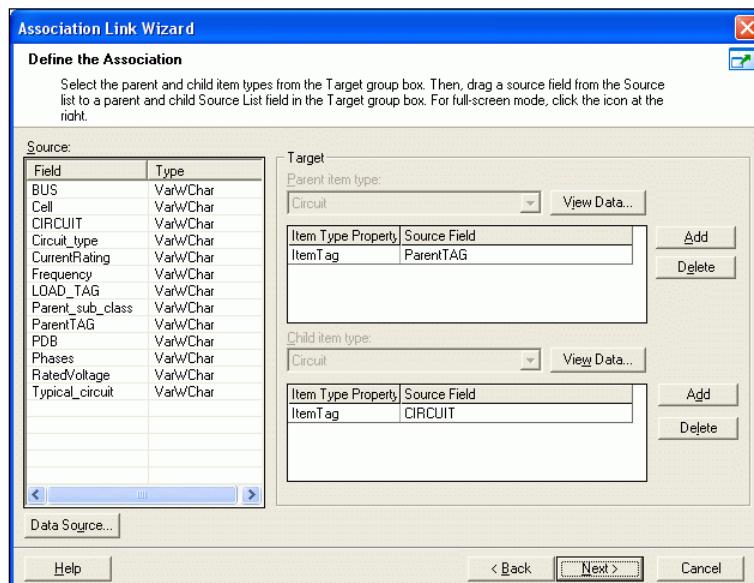
Target parent item type property = ‘Item Tag’, parent source field=‘ParentTAG’
(this property holds the name of the coupler circuit)

Target child item type is ‘Circuit’

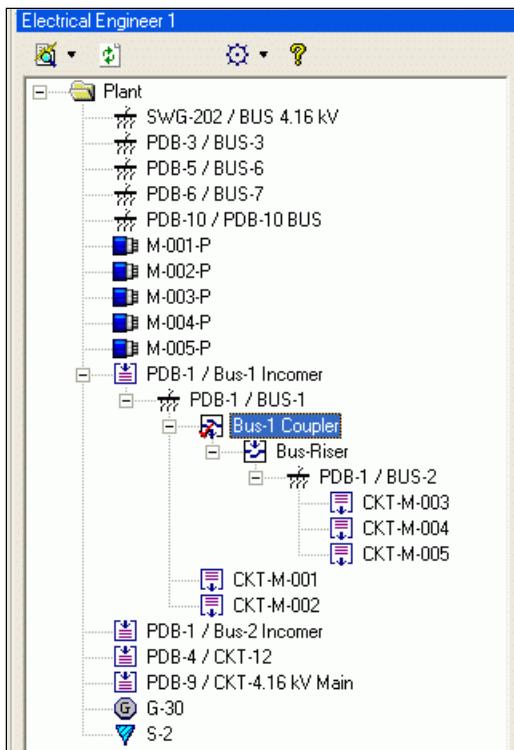
Target child item type property = ‘Item Tag’, child source field=‘CIRCUIT’

Note

As long as these names are unique the association will work.



- Next → Finish.**
- Test and Run the link.
- Check in SmartPlant Electrical (Electrical Engineer) that the bus-riser is associated with the coupler.



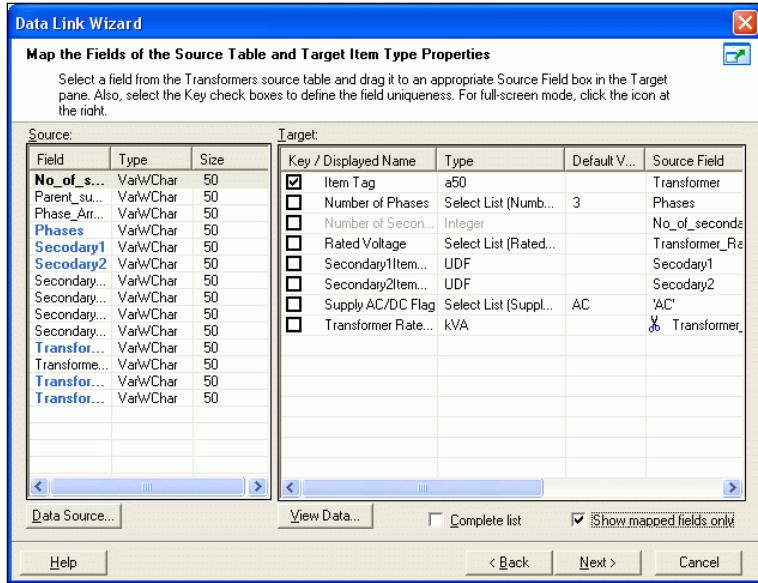
This concludes the import of the power distribution board and its components.

Import 6 – Transformer

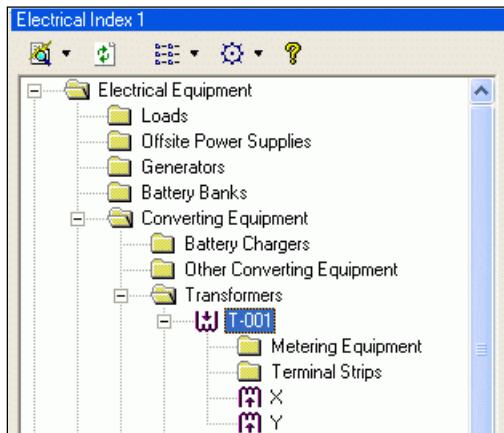
Transformer is a unique item type in SmartPlant Electrical due to the transformer's secondary windings. In order to import a transformer, it's mandatory to specify in the data link file the number of secondary windings and the secondary's item tags. If additional information is required to be imported to the secondary windings, it is necessary to make an additional import session for TransformerComponent item type (this lab does not cover this import).

1. Run the Import Manager → **Data Links** → **New Data Link**.
2. Enter '**Transformer**' → **Next** → **Next** → Source Table is ' Transformers'.
3. Target Table is ' Transformers' → **Next**.
4. Link the following properties.

Target	Source
Item Tag (key)	Transformer
Number of phases	Phases
Number of secondaries	No_of_secondaries
Rated Voltage	Transformer_rated_voltage
Secondary1ItemTag	Secondary1
Secondary2ItemTag	Secondary2
Supply AC/DC Flag	'AC'
Transformer Rated Power	Transformer_Rated_Power

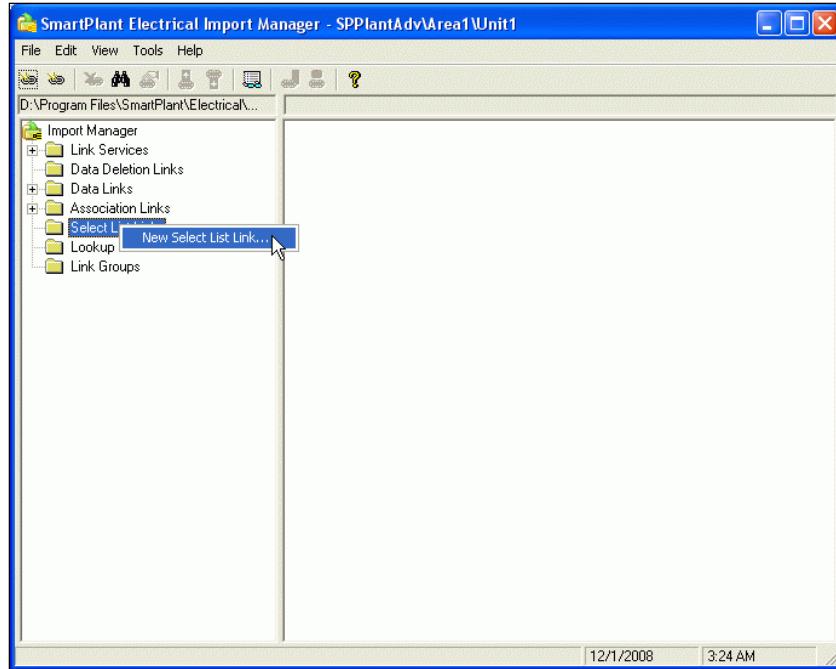


5. **Next → Finish.**
6. Run the link.
7. Check in SmartPlant Electrical that the new transformer and its secondary windings are available in the Index and in the Engineer.



Import 7 – Manufacturer Select List

1. Run the Import Manager → **Select List Links** → **New Select List Link**.



2. Enter 'MFR' → Source Table is 'mfr_model' → Target Table is 'Manufacturer' → 'Data Source' is available to see the MFR_Model data that will be imported.

Select List Link Wizard

Source and Target Definition

Select a source table or query then select a target select list. Click Data Source or View Data to preview your selections.

Source

Source table:

Use query

Target

Target select list:

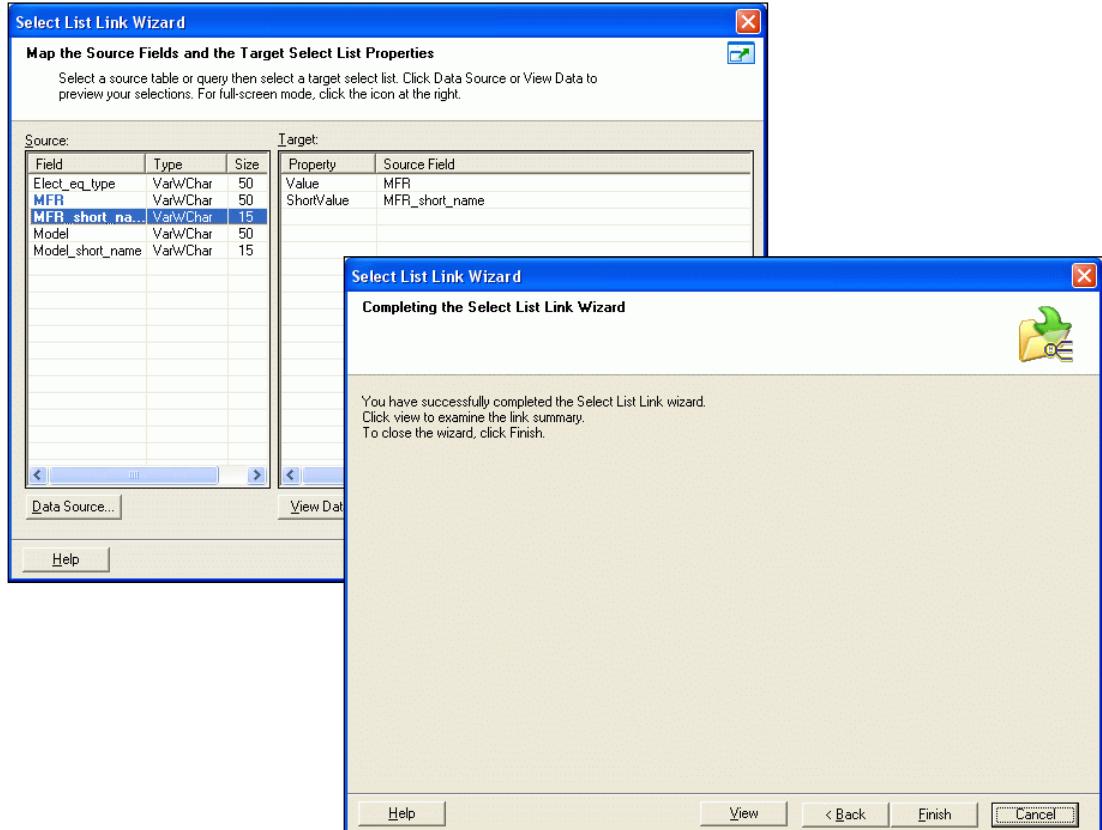
Data Source - mfr_model

MFR	MFR_short_name	Model	Model_short_name	Elect_eq_type
Manufacturer1	MFR1	Model1	M1	Motor
Manufacturer1	MFR1	Model2	M2	Motor

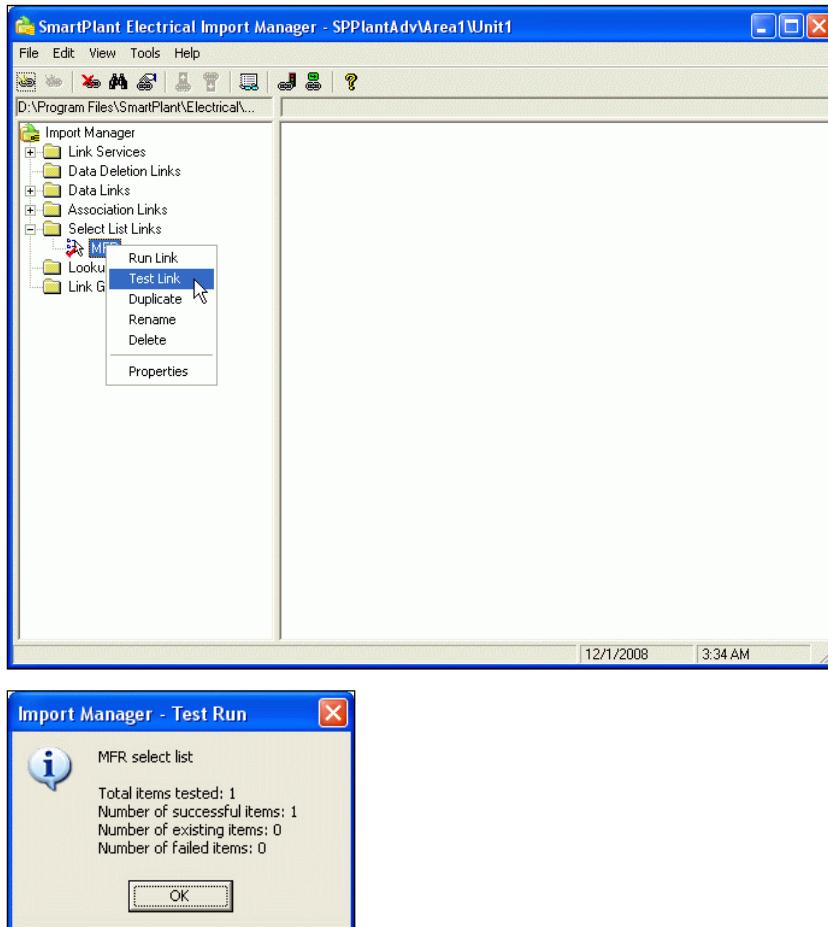
Total Records: 2

3. Drag and drop the fields from Left to Right as shown below (or enter fixed text strings) → Next → Finish.

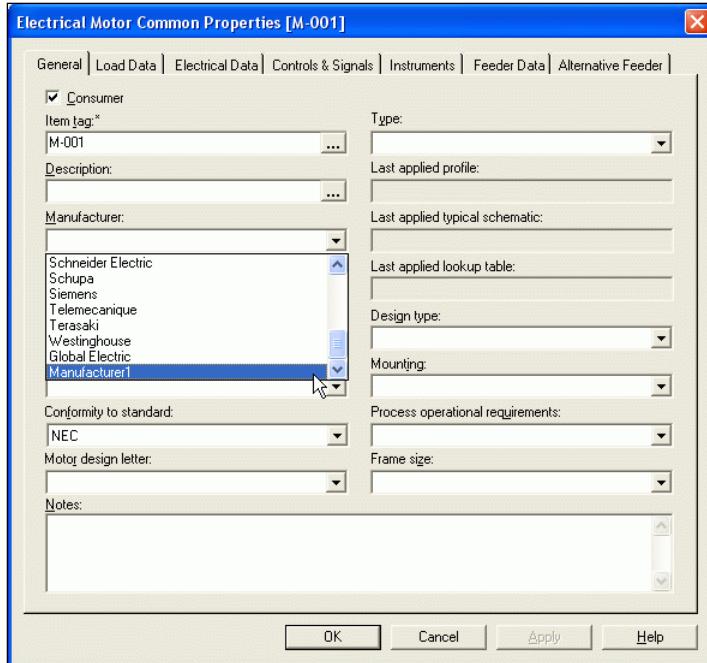
Target	Source
MFR	Value
MFR_Short_Name	ShortValue



4. Right-click the link, and on the shortcut menu, click **Test Link**.



5. Right-click the link, and on the shortcut menu, click **Run Link**.
6. Run SmartPlant Electrical → Electrical Index → Electrical Equipment → Loads → Motors → Select any Motor → Right Mouse to 'Common Properties' → Manufacturer Select-List will show the new value.

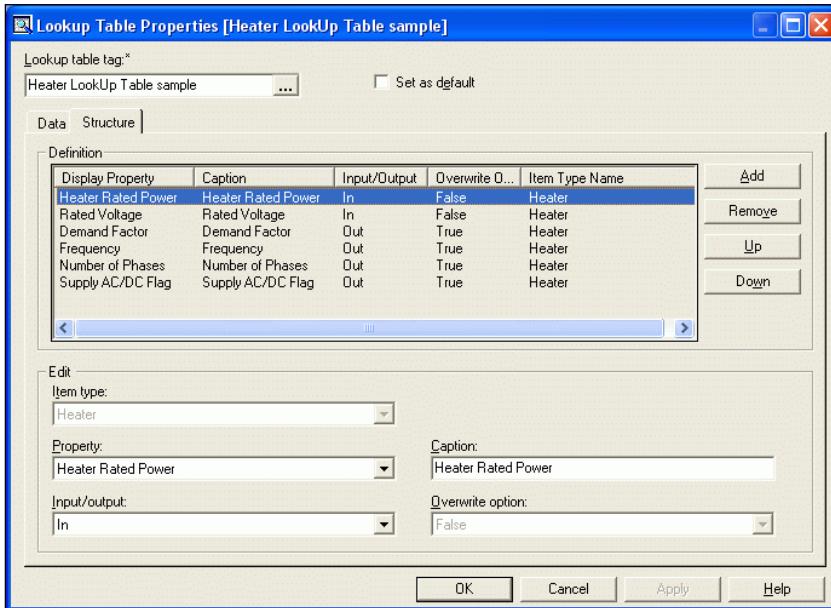


- **Note:** You may want to continue and import the models. A second import link is required. Prior to that, the import manager must be closed and re-opened to refresh the new entries (of the manufacturer list).

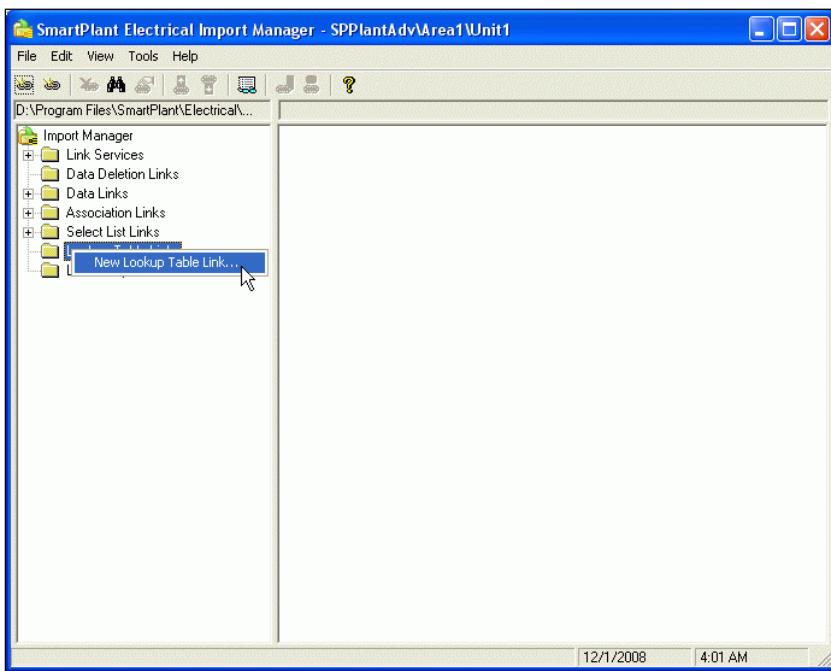
Import 8 – Heater Lookup Table

First step before importing data into a lookup table is to define this table inside SmartPlant Electrical. Create a new lookup table for heaters as follows:

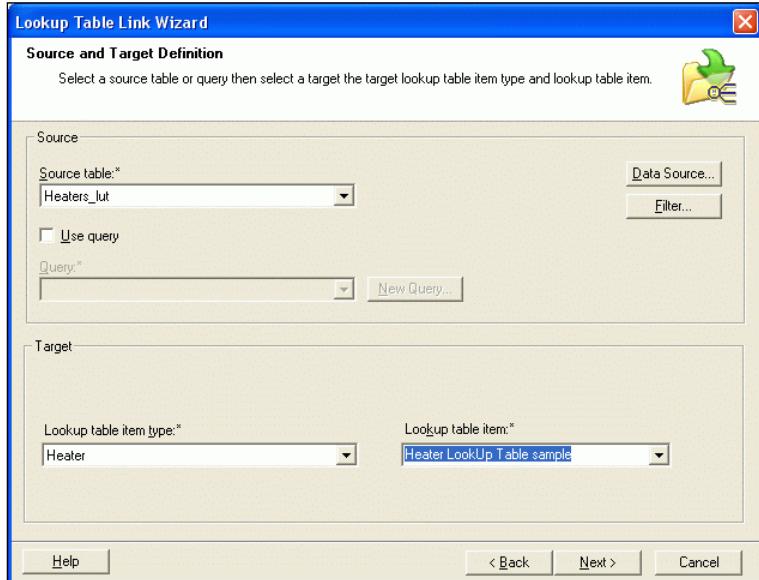
1. Lookup table name: ‘**Heater LookUp Table sample**’.
2. Define the following properties as shown in the screen shot below:



3. Close the dialog to save your changes.
4. Run the Import Manager → Lookup Table Links → New Lookup Table Link



5. Enter '**Heaters**' → Source Table is 'Heaters_Lut' → Target Lookup Table Item Type is 'Heater' and Target Lookup Table item is 'Heater LookUp Table sample' → 'Data Source' to view the data that will be imported

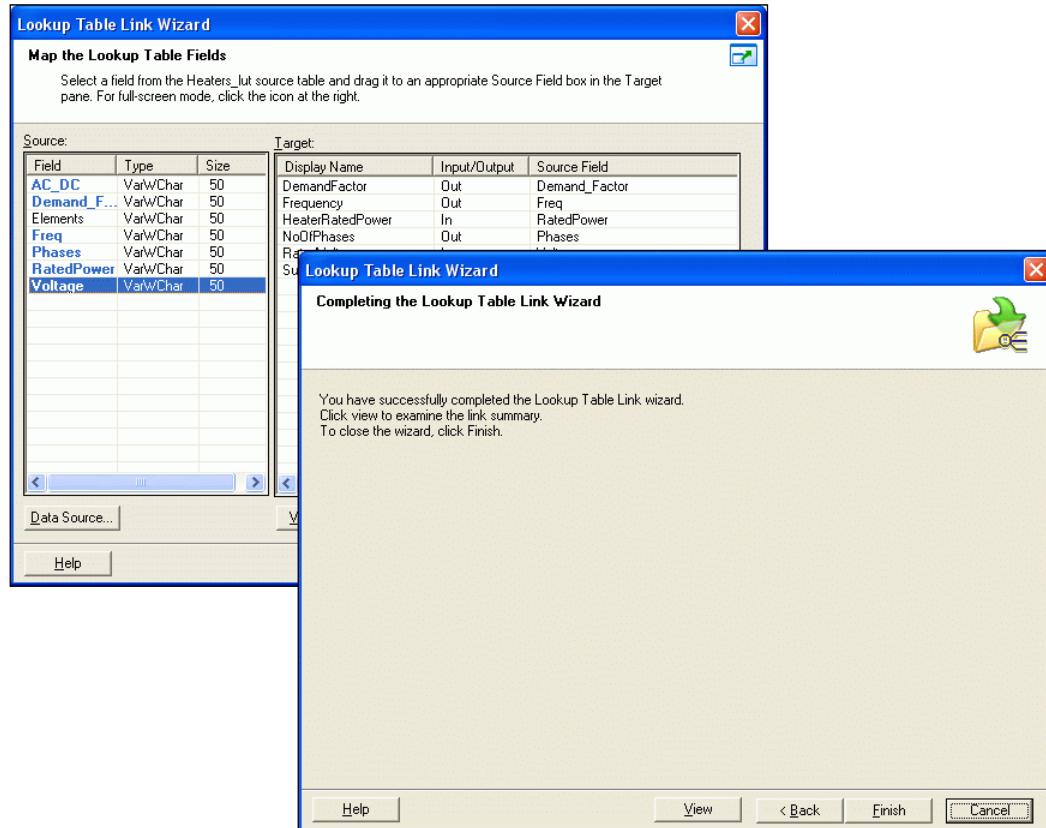


6. Click **Data Source** to see the data that will be imported.

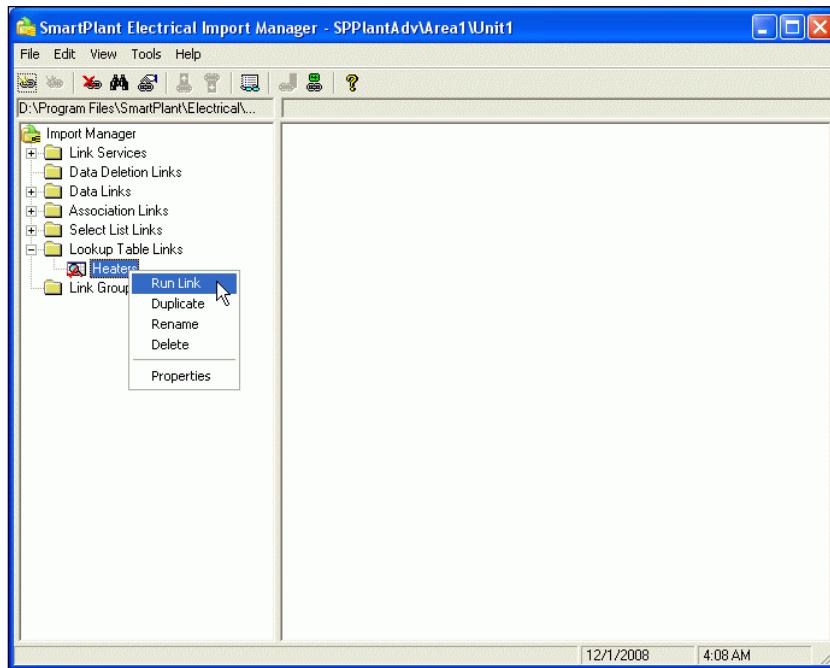
Data Source - Heaters_Lut					
	RatedPower	Voltage	Elements	Phases	AC_DC
▶	10 kW	220 V	2	1	AC
	30 kW	380 V	2	3	AC

7. **Next** → Drag the fields from left to right as shown below → **Next** → **Finish**.

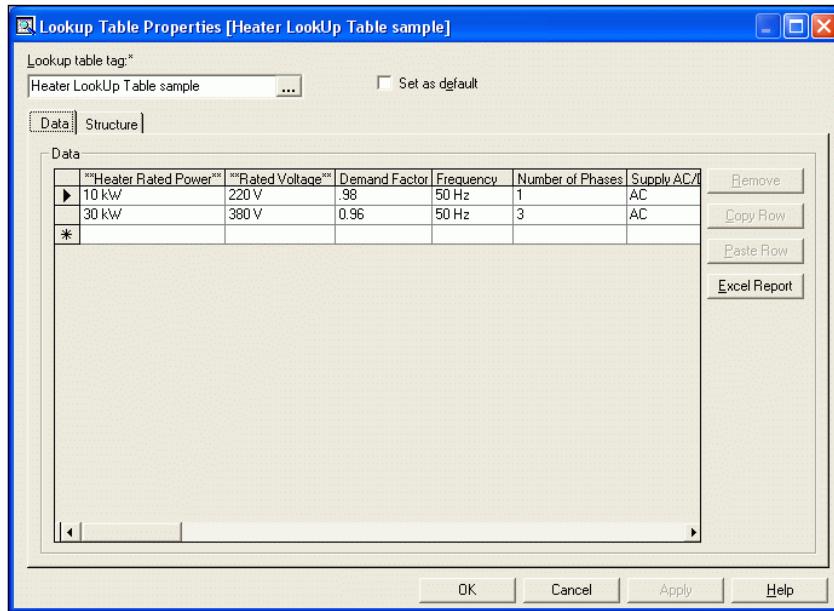
Target	Source
Demand Factor	Demand_Factor
Frequency	Freq
Heater Rated Power	RatedPower
Number of Phases	Phases
Rated Voltage	Voltage
Supply AC/DC Flag	AC_DC



8. Right-click the link, and on the shortcut menu, click **Run Link**.



9. Run Smart Plant Electrical and view the result.



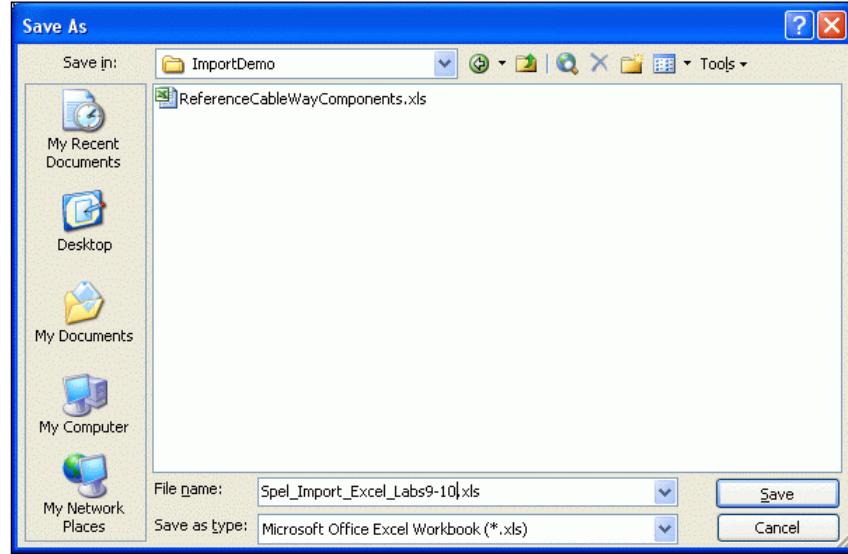
Import 9 – Import Motors List Using an Excel Worksheet as Data Source

1. Define table in new excel worksheet.

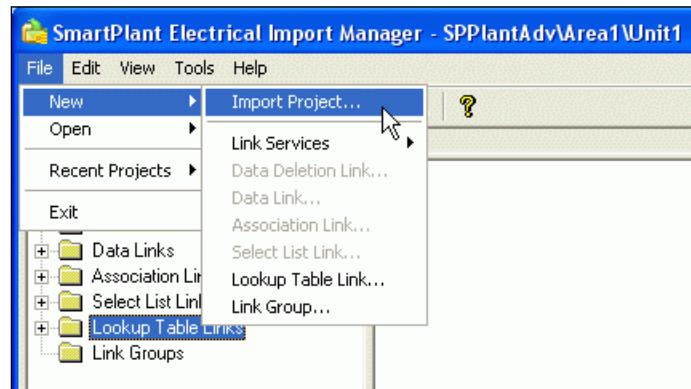
The table data from the top screenshot:

	motor_item	rated_power	break_power	rated_voltage
1	M-901	1 hp	0.8 hp	460 v
2	M-902	2 hp	1.8 hp	460 v
3	M-903	5 hp	4.5 hp	460 v
4				
5				

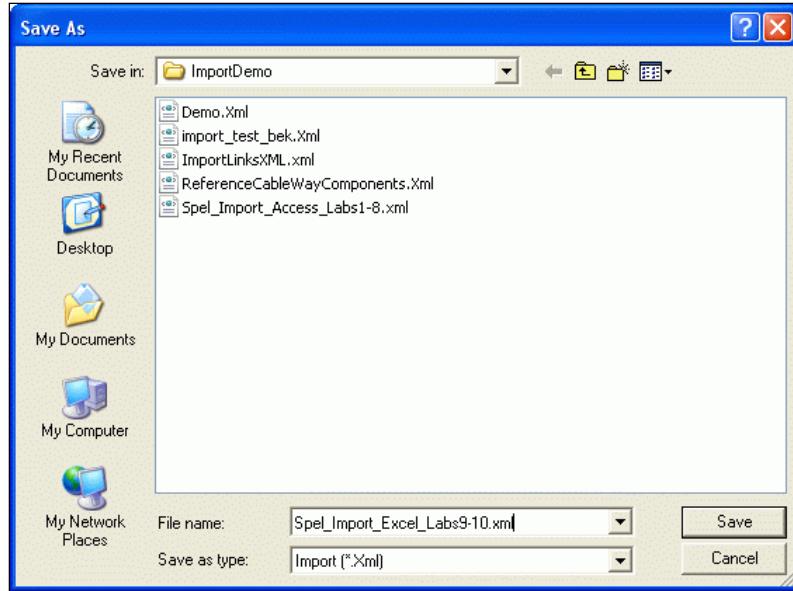
Save the excel worksheet and name it ‘Spel_Import_Excel_Labs9-10’.



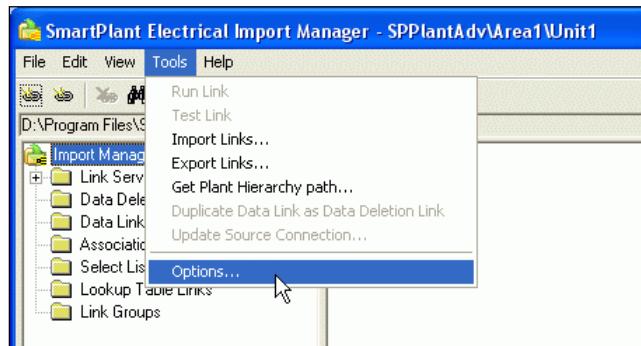
2. In import manager create new import project.



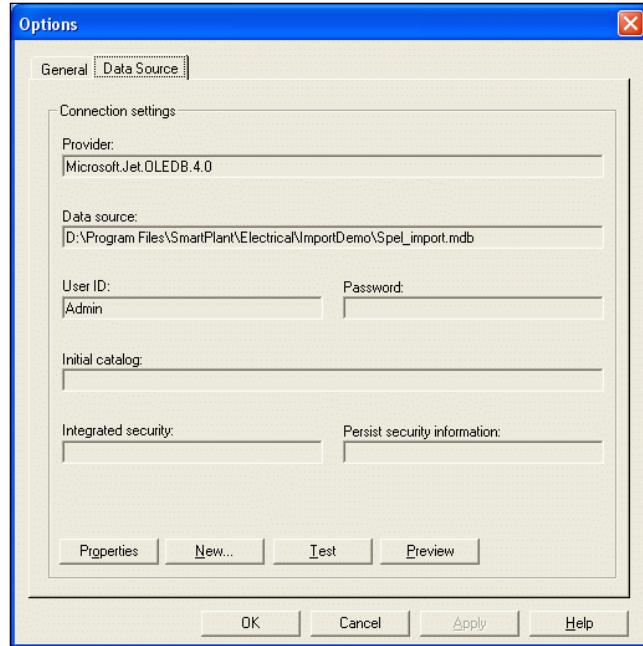
Name the new import project ‘Spel_Import_Excel_Labs9-10’



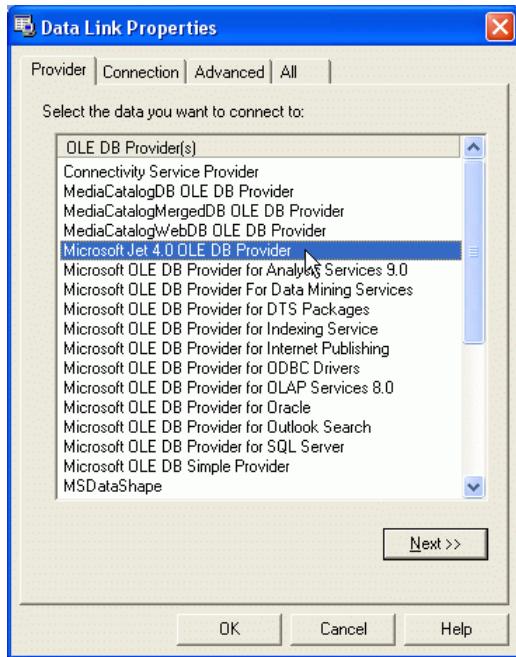
Select options from tools menu.



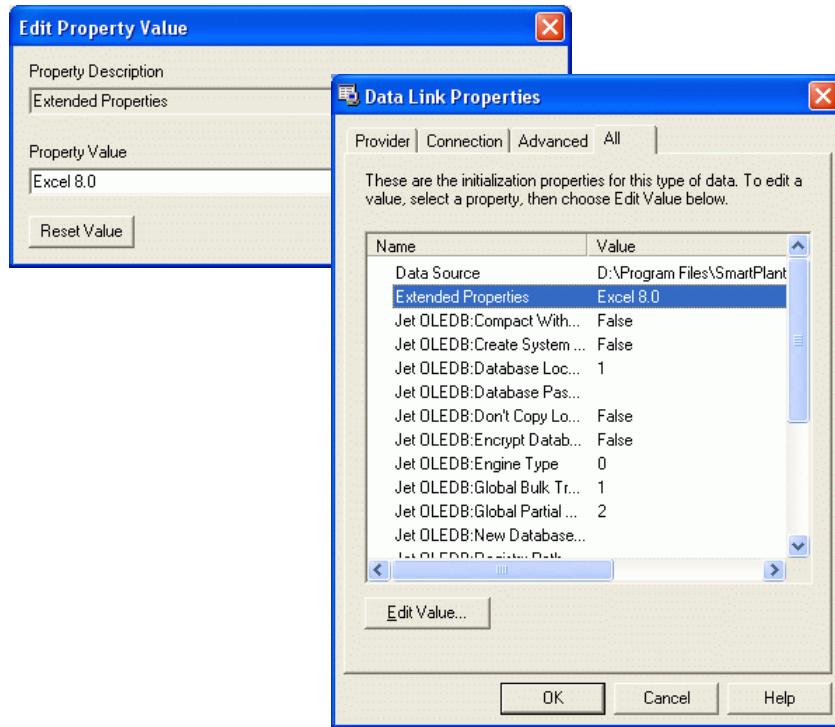
In the options windows select data source tab.



Select properties button and in the provider tab make sure the Microsoft Jet 4.0 OLE DB Provider is selected.



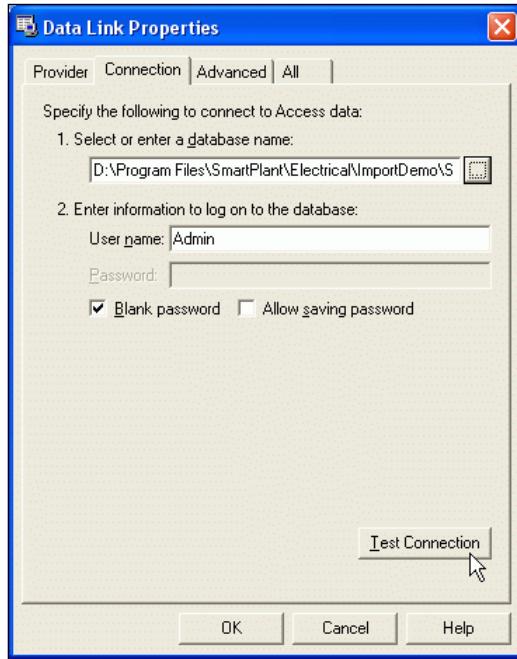
Select the all tab and modify the extended properties value and type in Excel 8.0



Select the connections tab, and click on the ellipse.



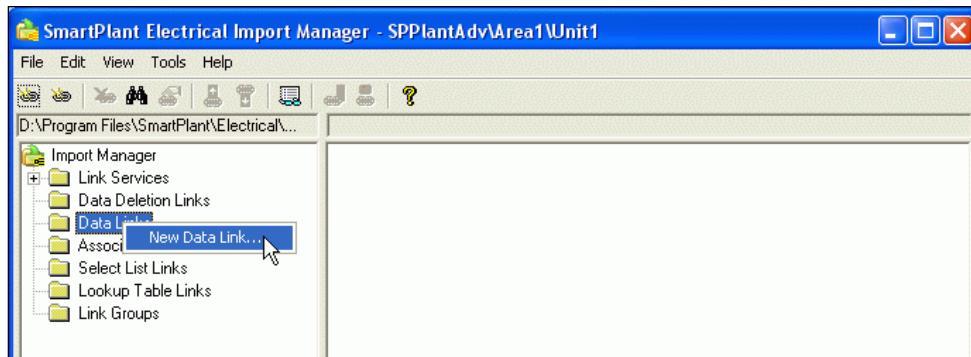
In the select access database, change the ‘file of type’ drop down list from mdb to all files and pick the ‘Spel_Import_Excel_Labs9-10.xls’.



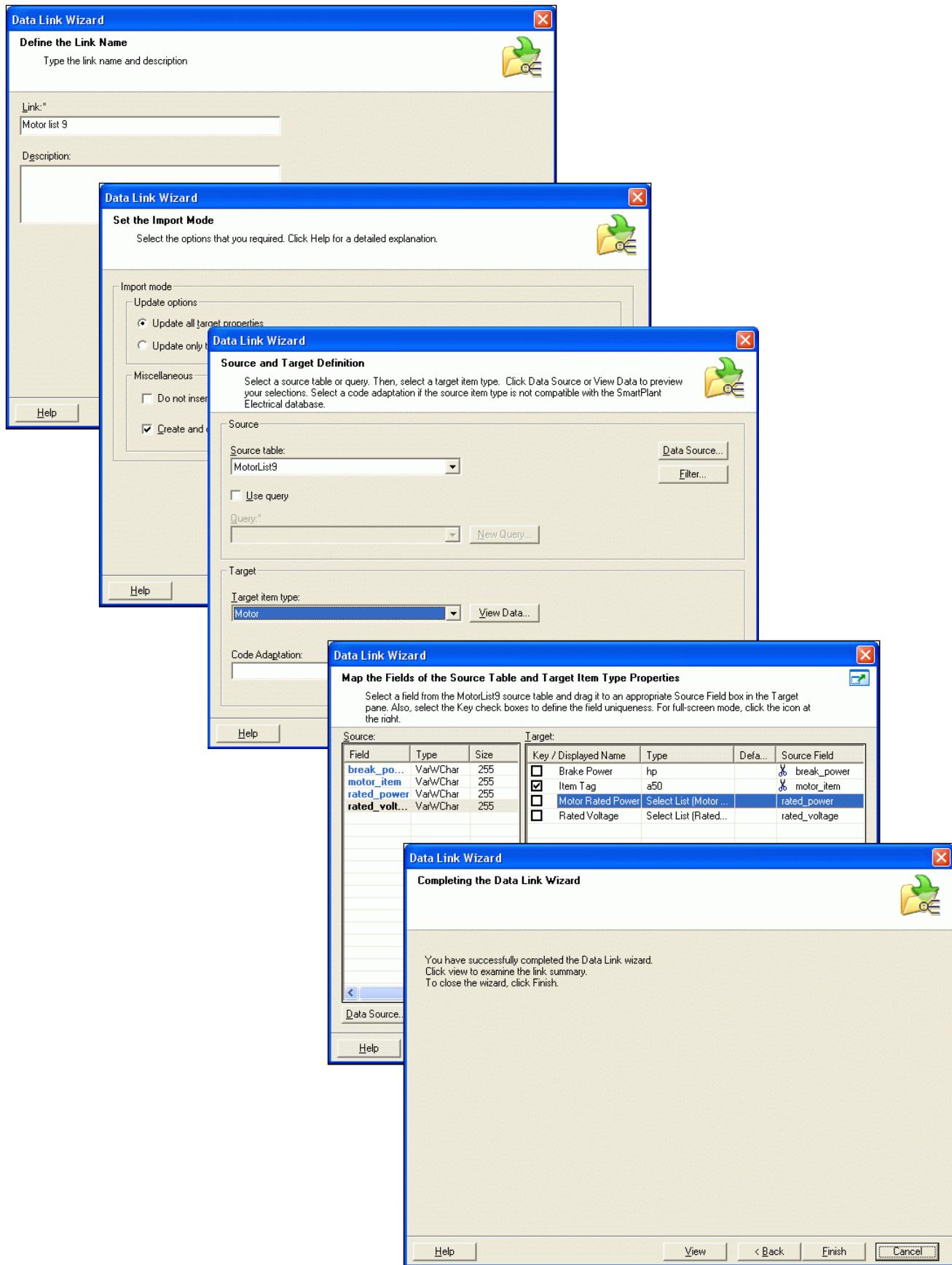
Test the connection



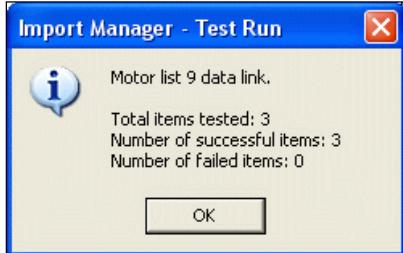
3. Create new motor data link using the new excel worksheet we setup.



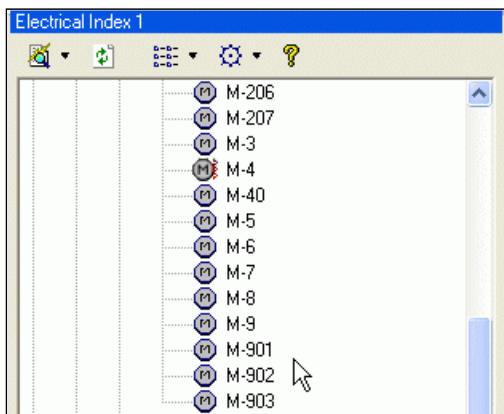
SmartPlant Electrical Import Manager



Test and run the link.



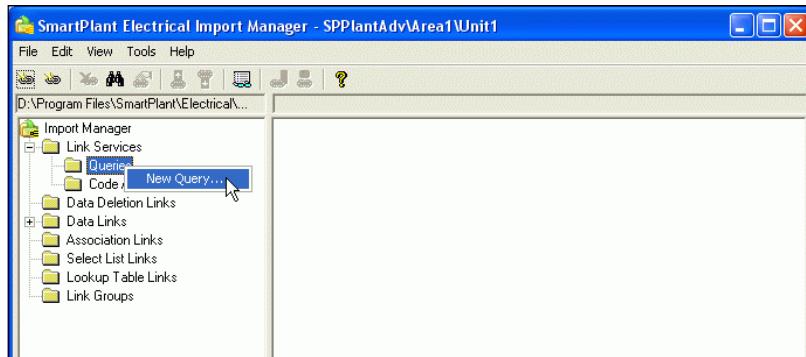
Open spel and test the result.



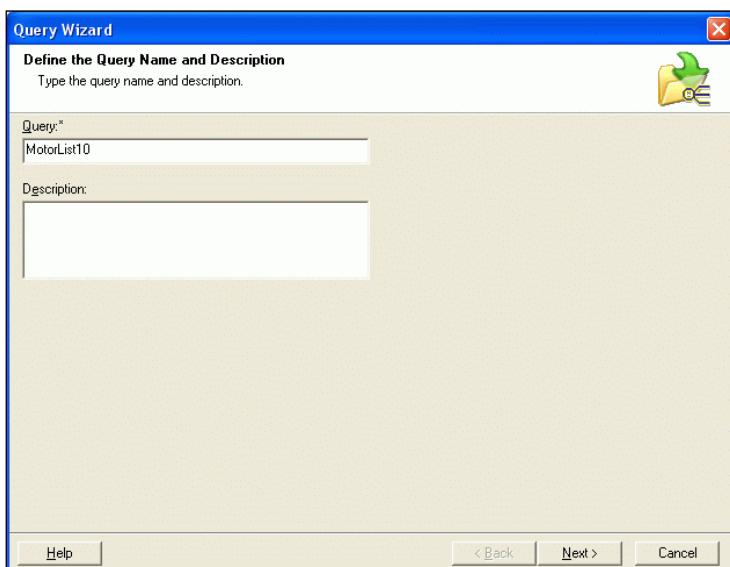
Import 10 – Import Motors List Using Query as Data Source

1. Open the excel worksheet we used in the previous lab and define the following 2 tables (MotorList10a and MotorList10b).

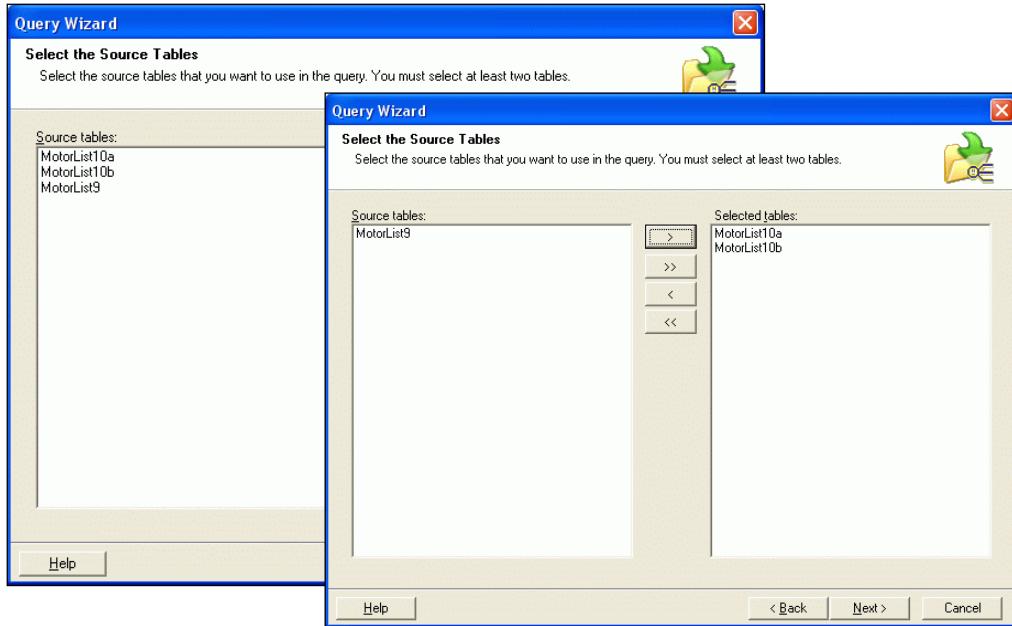
	A	B	C	D	E
1	motor_item	rated_power	break_power	rated_voltage	
2	M-901	1 hp	0.8 hp	460 v	
3	M-902	2 hp	1.8 hp	460 v	
4	M-903	5 hp	4.5 hp	460 v	
5					
6					
7					
8	MotorList10a				
9	motor_item	rated_power	break_power	rated_voltage	
10	M-904	1 hp	0.8 hp	460 v	
11	M-905	2 hp	1.8 hp	460 v	
12	M-906	5 hp	4.5 hp	460 v	
13					
14	MotorList10b				
15	motor_item	description	frequency	operation_mode	
16	M-904	Squirrel cage	60 Hz	Continuous	
17	M-905	Squirrel cage	60 Hz	Continuous	
18	M-906	Squirrel cage	60 Hz	Continuous	
19					



2. Create new query.



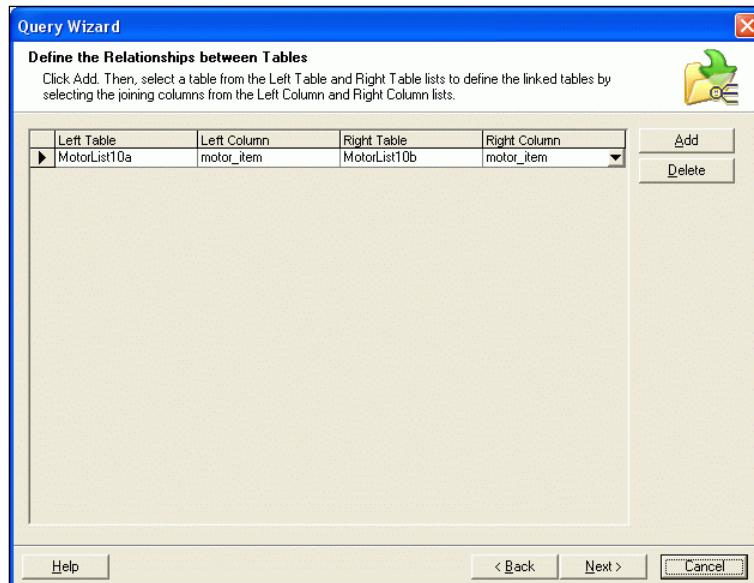
3. Select tables to include in the query.



Tip

More than two tables can be selected for the query.

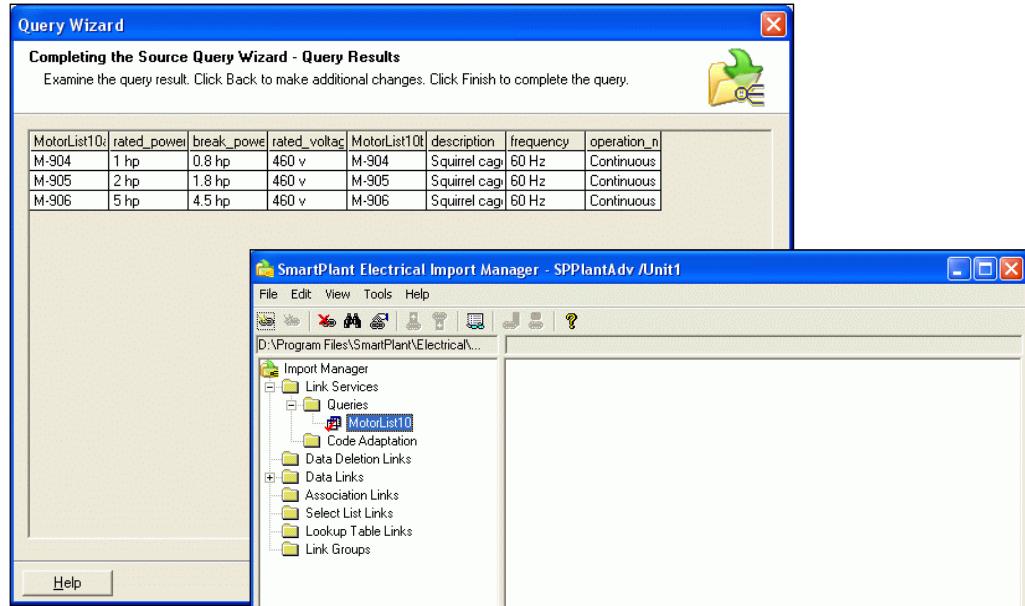
4. Define the matching properties in the query table.



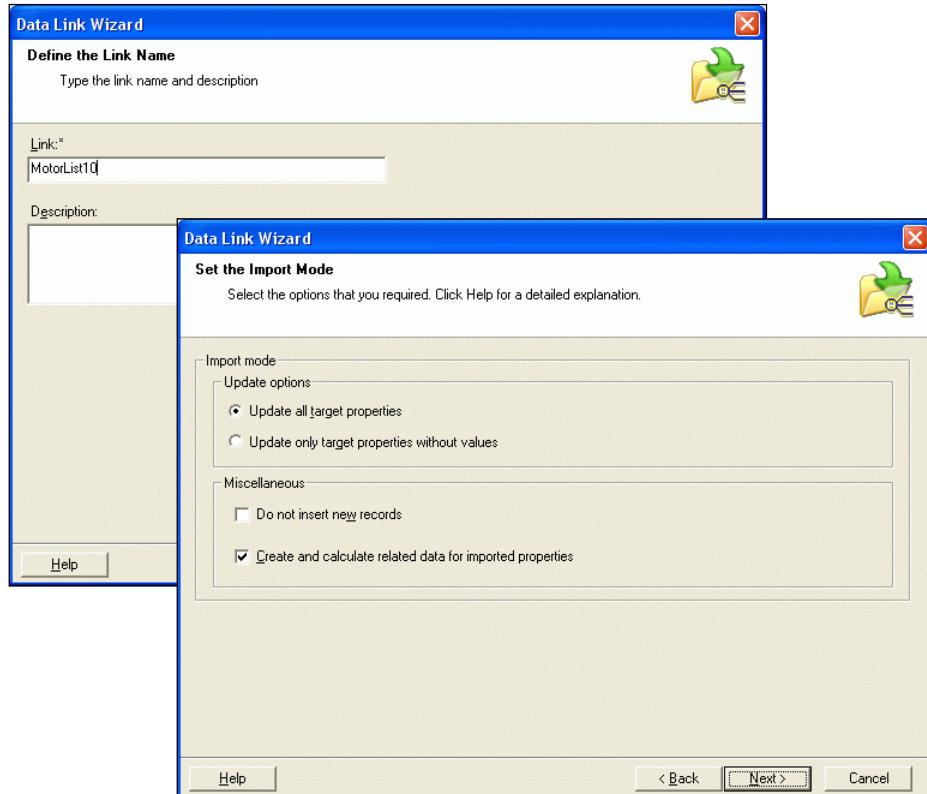
Tip

If more than two tables are defined on this query, additional relationships can be added.

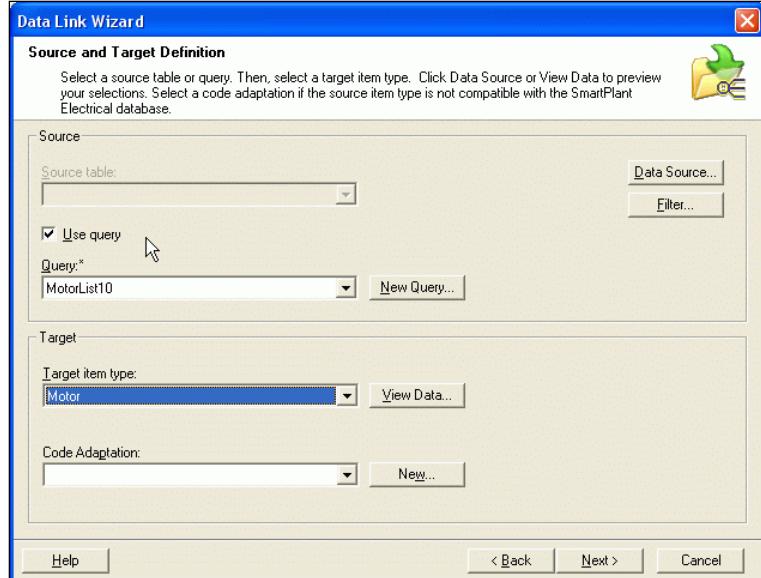
5. View the query results and finish the query.



6. Create a new data link.



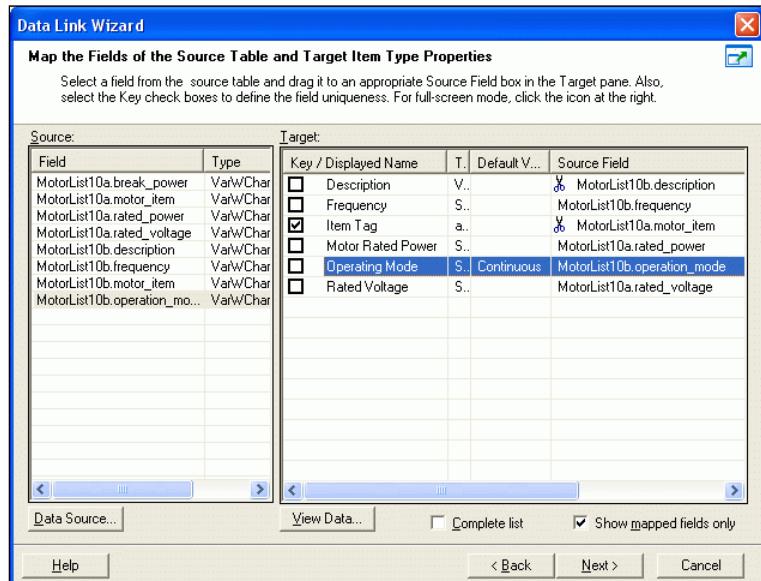
7. In the **Source and Target Definition** part of the wizard, select the **Use query** check box and select the required query.



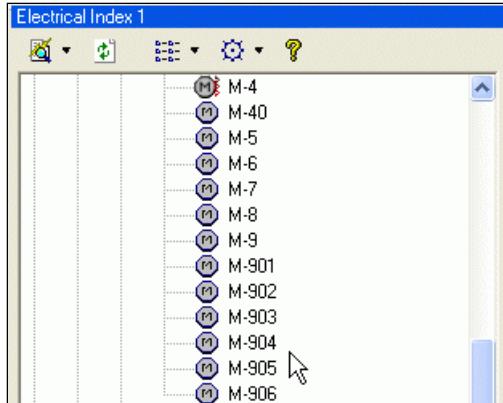
💡 Tip

A new query can be created from within the wizard in the same way by using the **New Query** command button.

8. Define the mapping.



9. Finish the link test and run.



Tip

A query can be used in each of the link types.

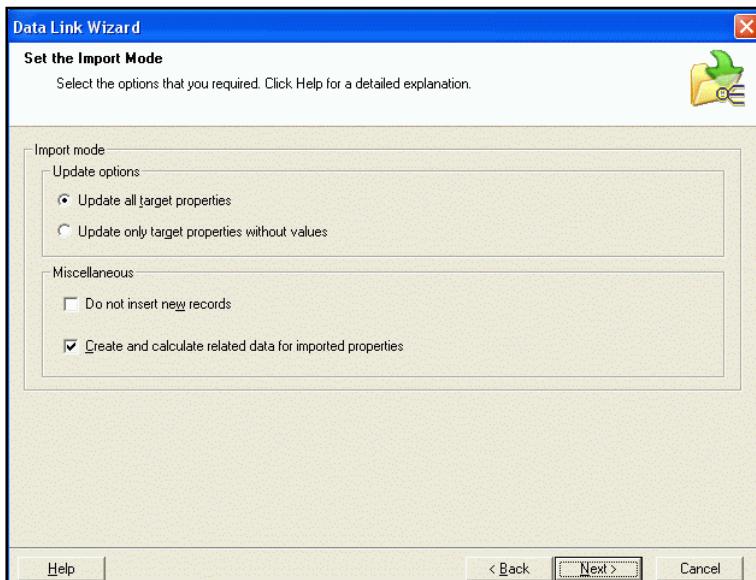
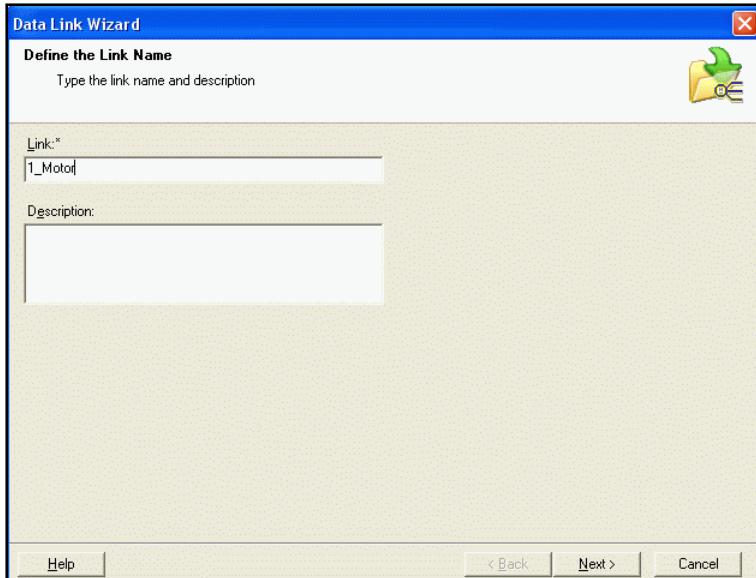
Import 11 - Import terminal Strip

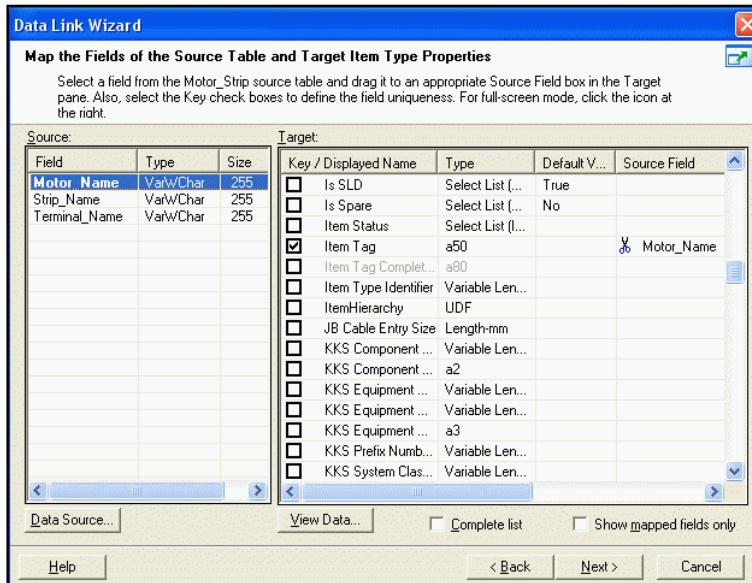
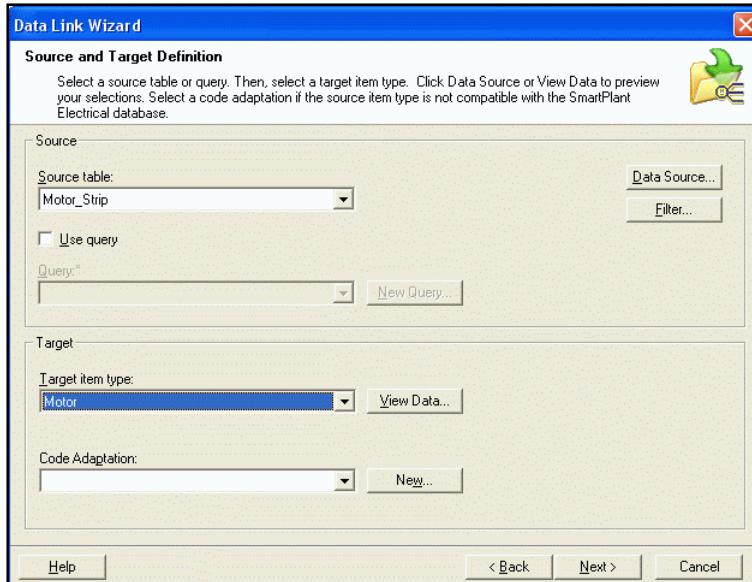
Import Manager enables importing terminal strips and terminals, and associates them to an item.

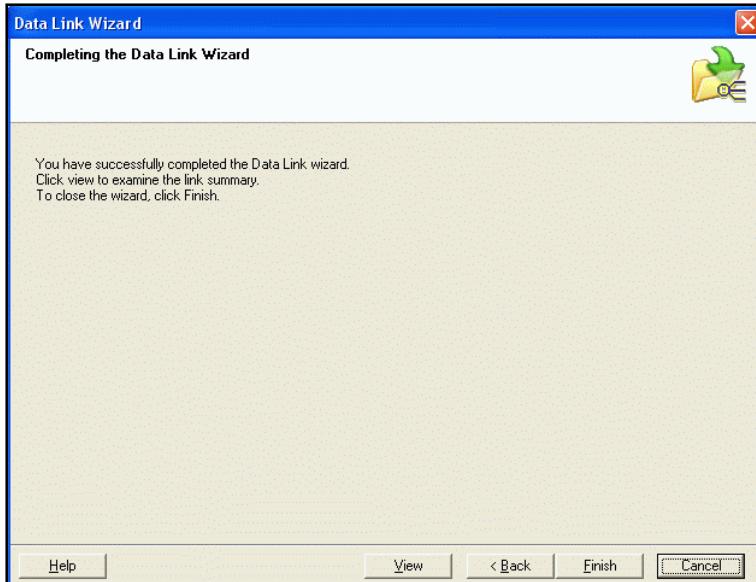
- i. Create new excel file and name it Spel_Import_Excel_Lab11.xls
- ii. Add the following data to the excel file and specify the range as table, 'Motor_Strip'.

	A	B	C	D
1	Motor_Name	Strip_Name	Terminal_Name	
2	U1-MOTOR- 2	Terminal_Strip1	A1	
3	U1-MOTOR- 2	Terminal_Strip1	A2	
4	U1-MOTOR- 2	Terminal_Strip1	A3	
5	U1-MOTOR- 2	Terminal_Strip1	A4	
6				

- iii. Configure the import manager to use the new excel source.
- iv. Create and run the following links:
 1. Import the Motor

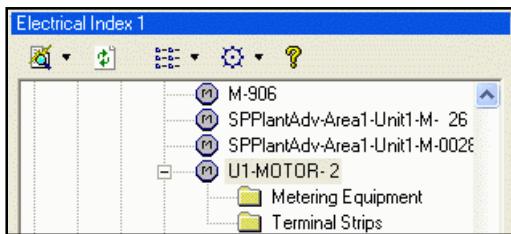




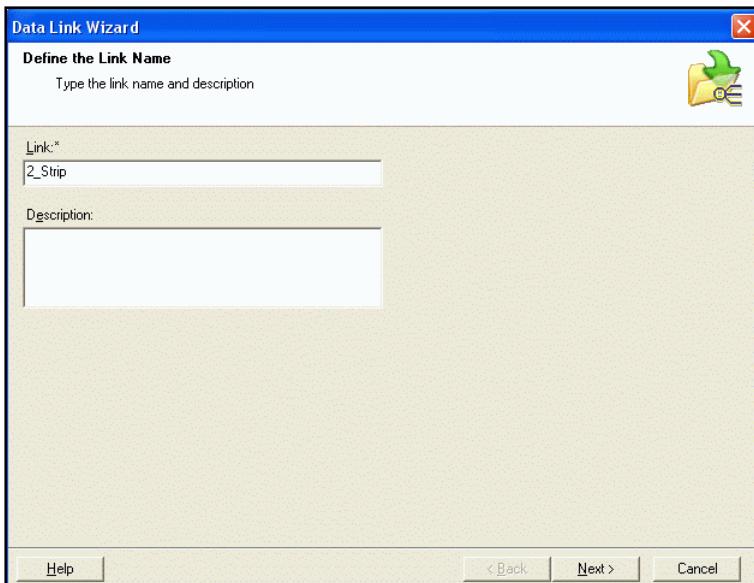


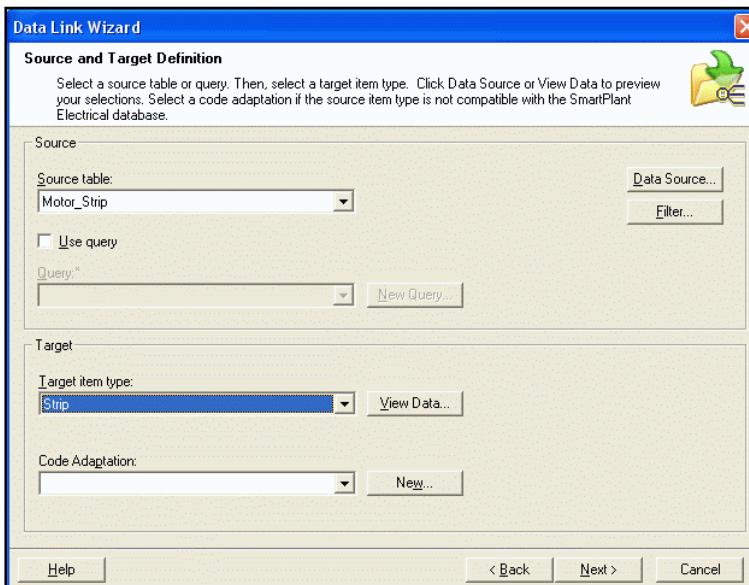
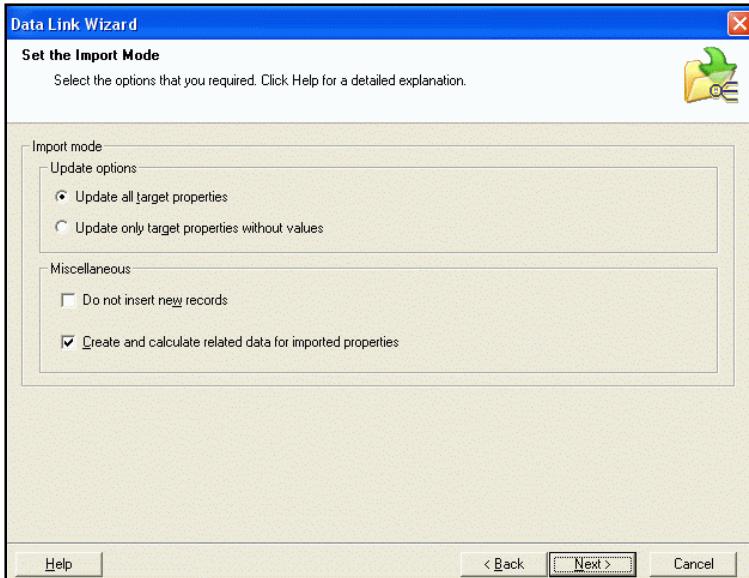
Test and run the link.

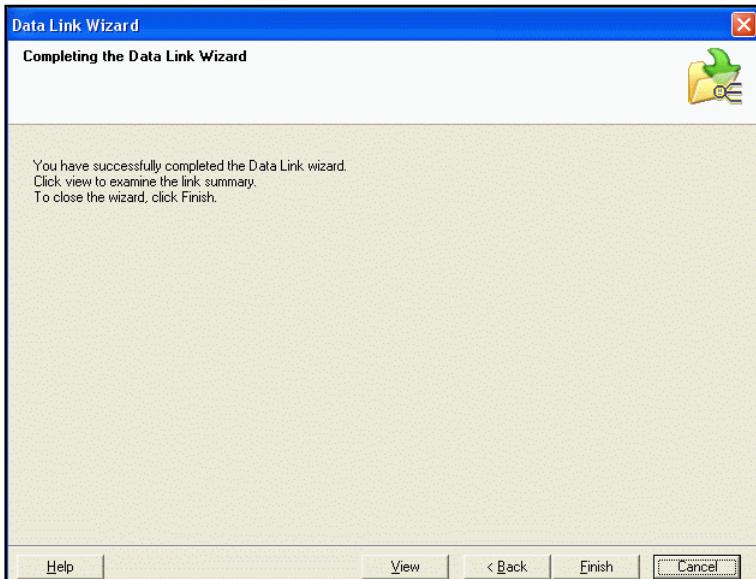
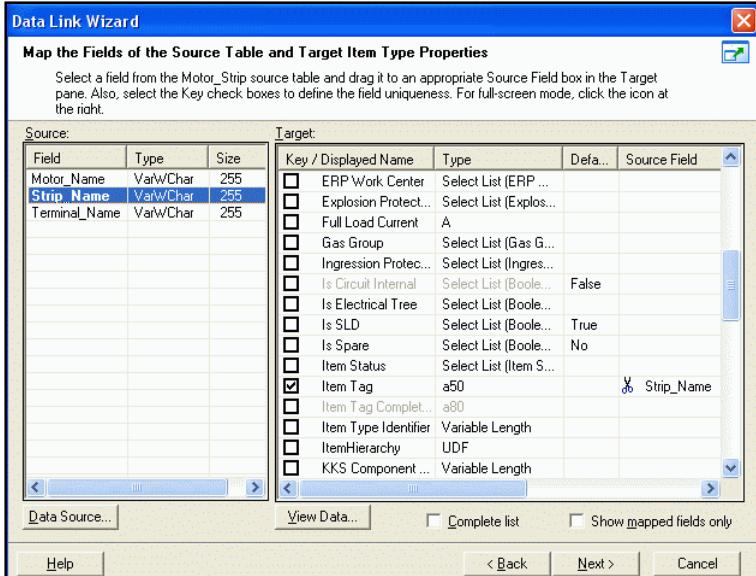
The result is:



2. Import the strip

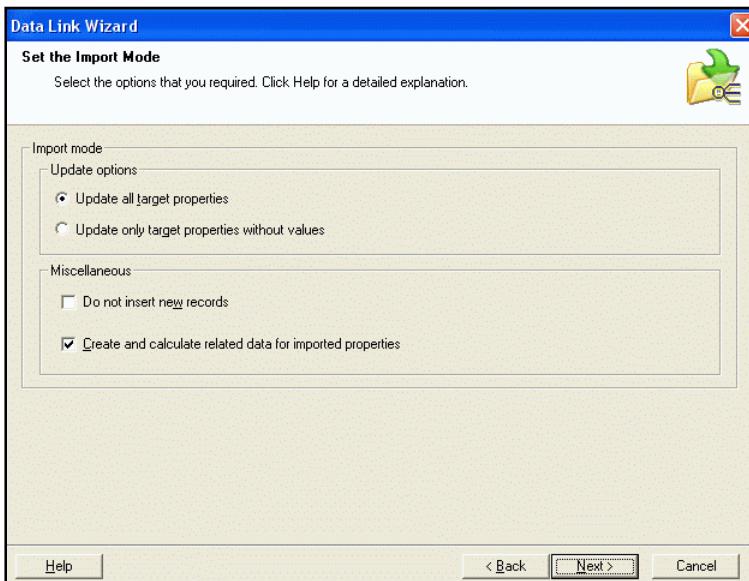
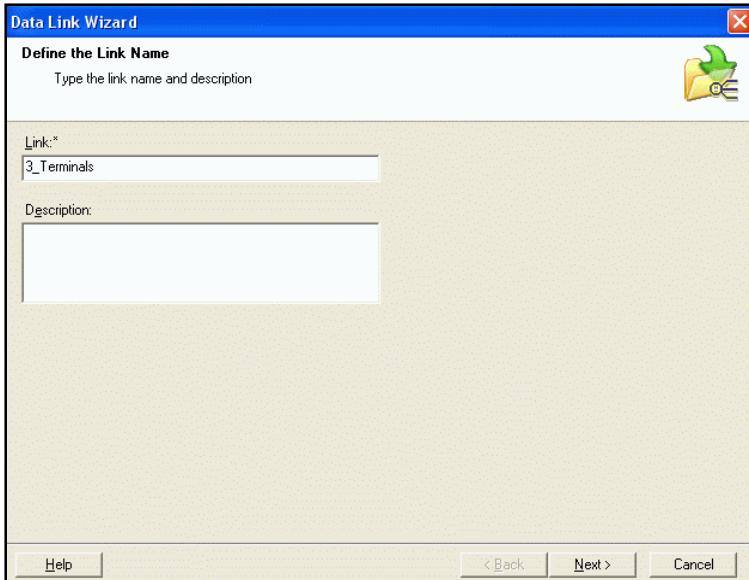


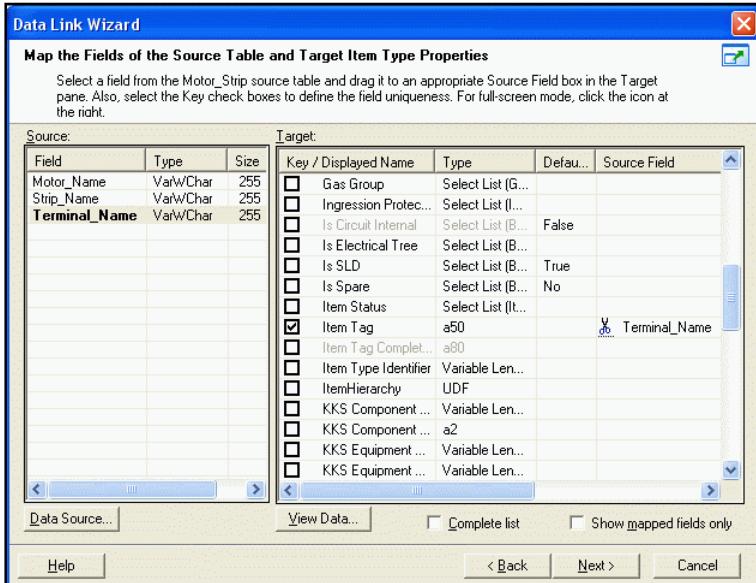
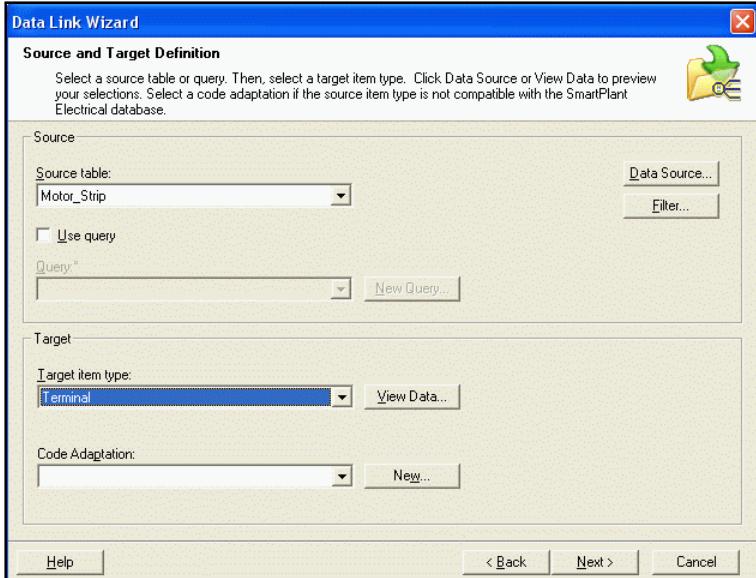


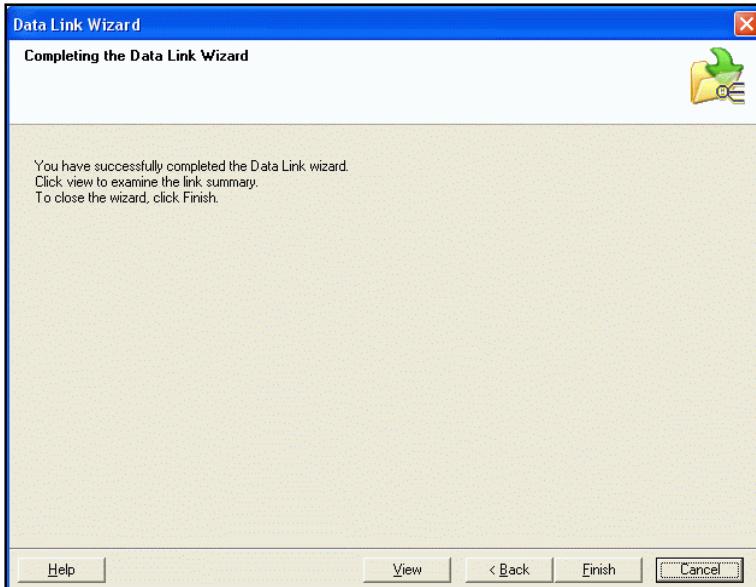


Test and run the link.

3. Import the strip terminals

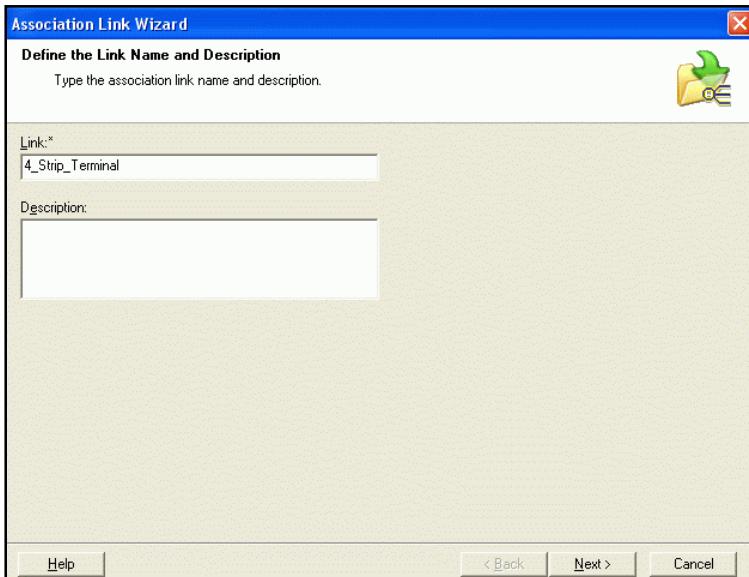


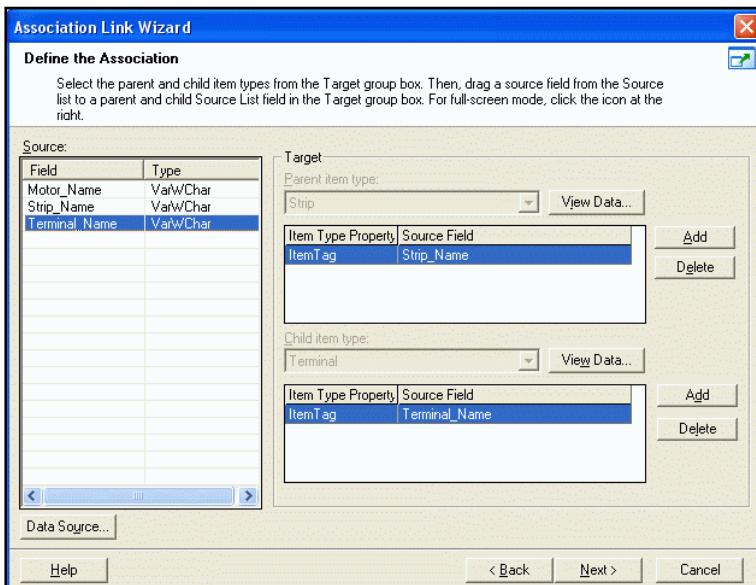
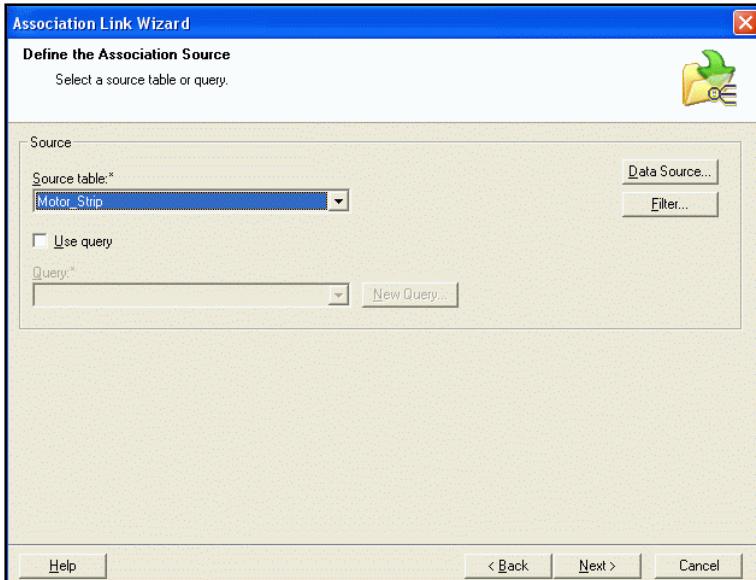


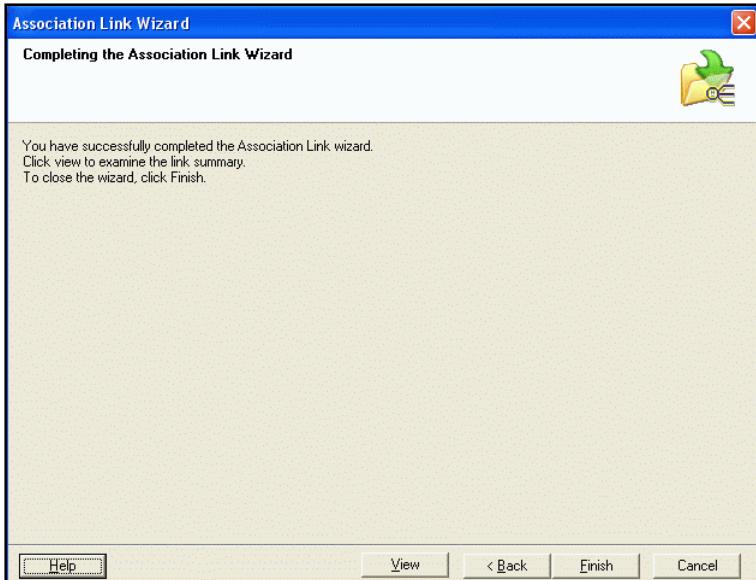


Test and run the link.

4. Associate terminal to strip

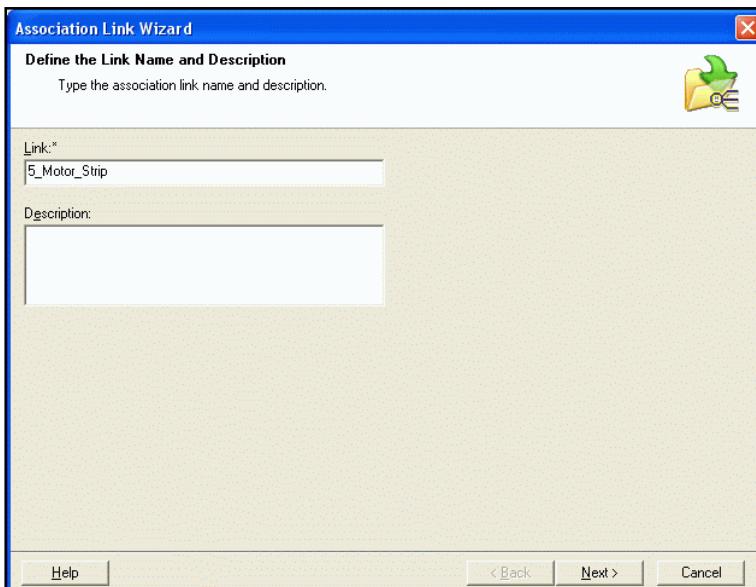


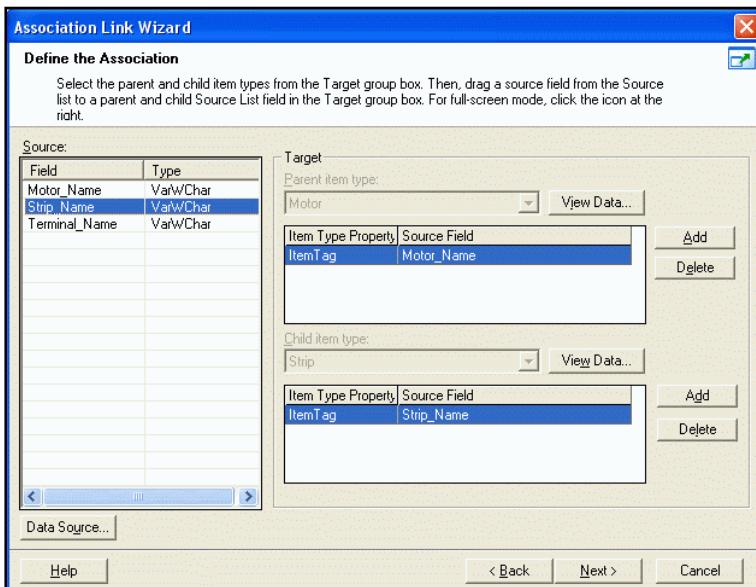
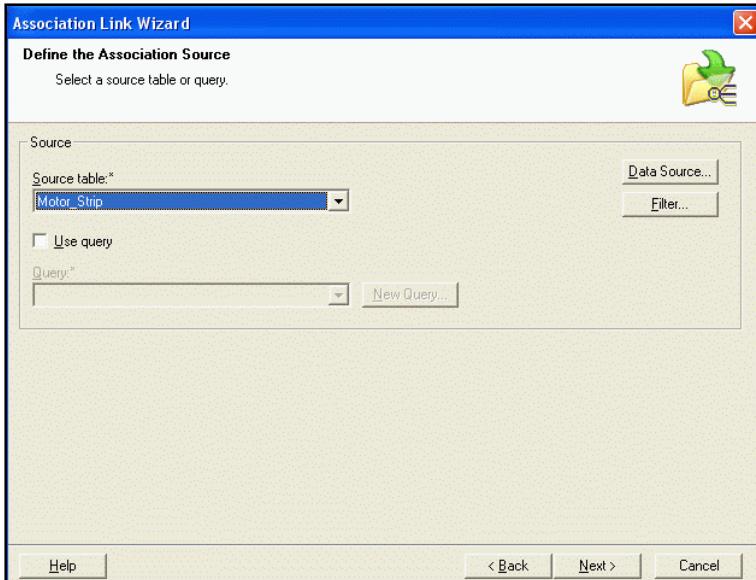


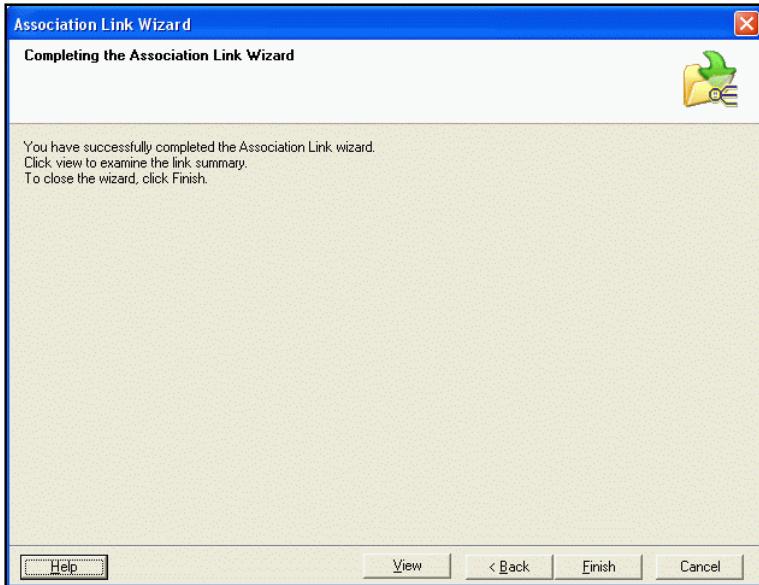


Test and import the link.

5. Associate the strip to motor

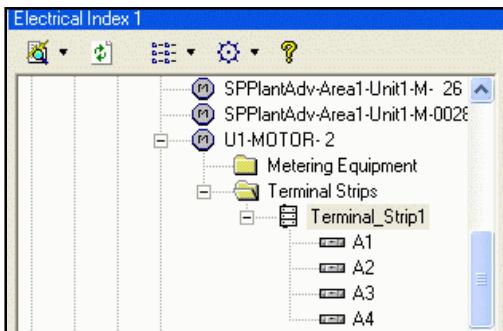






Test and run the link.

The result in SmartPlant Electrical is:

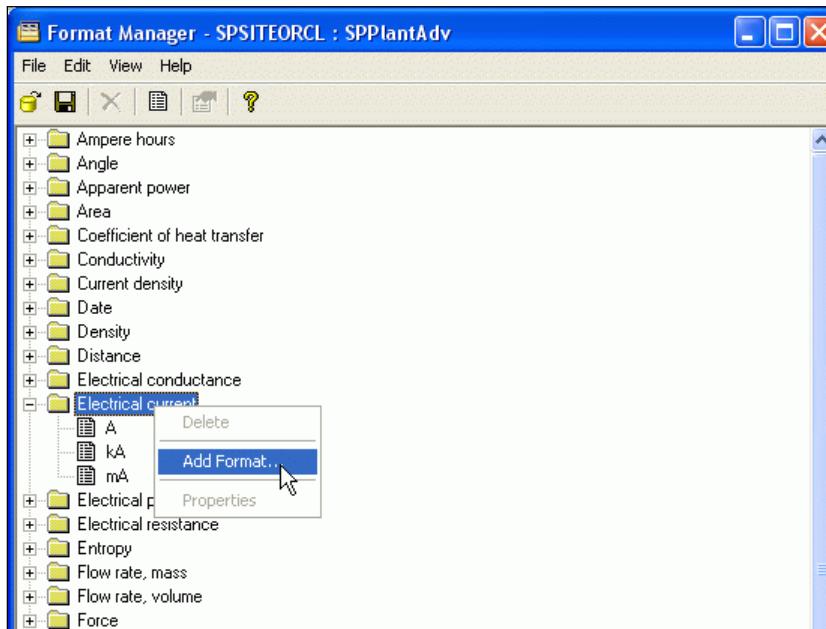


SmartPlant Electrical Format Manager

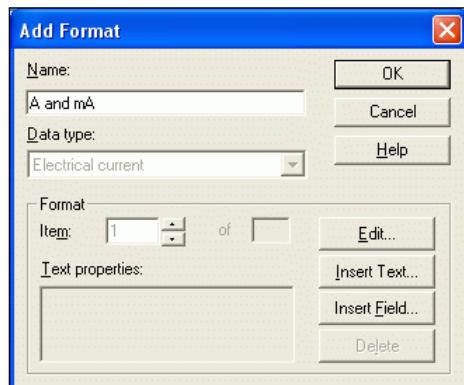
Format manager defines the characteristics and format for labels, report data, and formatted properties. The purpose of this lab is to demonstrate how you can customize the format and presentation of a property that uses 'Electrical current' unit of measure.

You will create a new 'compound' format that displays electrical current as a combination of A and mA.

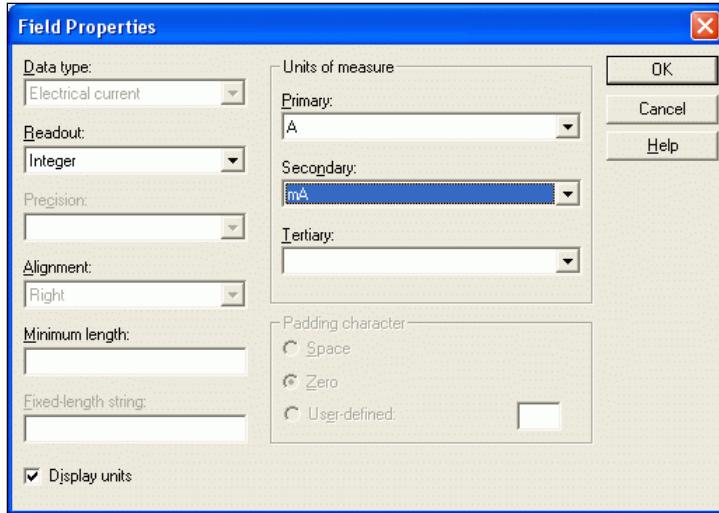
1. Run Format Manager
2. Add a new format for Electrical current.



3. In the **Name** box type a name (for example: 'A and mA').



4. Click **Insert Field** to open the Field Properties.
5. Define the entries as shown in the picture



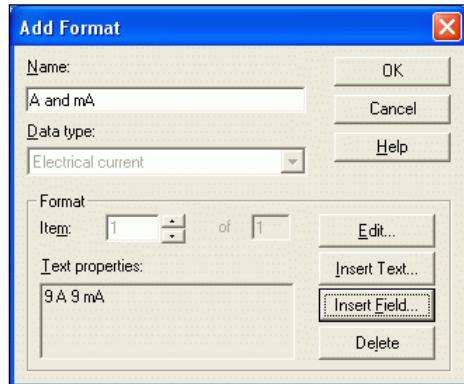
- In the **Unit of measure** section, **Primary**, **Secondary** & **Tertiary** fields define the first, second and third unit of measurement that the software uses in the format respectively.

Tip

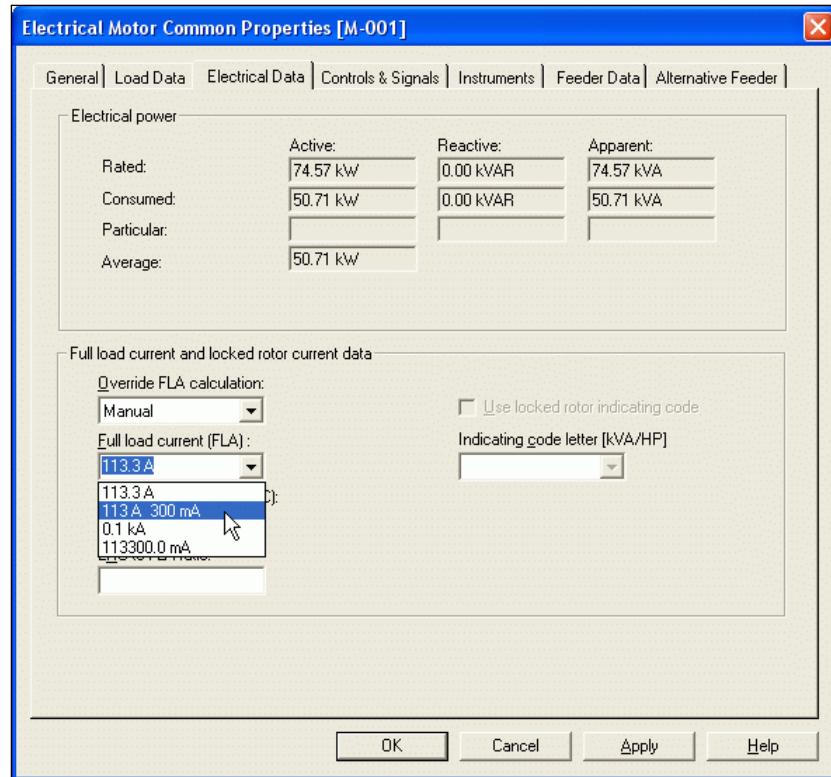
The readout is 'Integer' so you do not see decimal points.

- Click **OK**.

The **New Format** dialog box appears as shown.



- Click **OK**.
- Click **Save**.
- To test the new format close and re-open SmartPlant Electrical in order for the application to read the modified format file.



Defining Project Naming Conventions

Defining NCs (Naming Conventions) in SmartPlant Electrical is a procedure that should be performed at the first stages of the project, before items are created, although SmartPlant Electrical allows you to do it later.

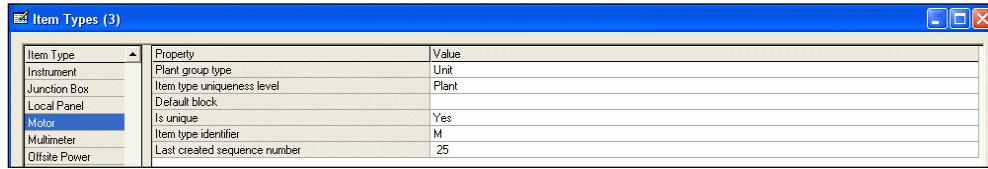
The whole process is performed in the Options Manager, after all item types have been defined for their plant group type and uniqueness.

Plant group type — specifies the plant group type to which the software assigns the item type. For example, if you assign **Plant** as the highest plant group type for motors, whenever you create a motor in SmartPlant Electrical, it will belong to the entire plant. If you intend to register your plant with SmartPlant Foundation, you must make sure that you assign the appropriate item types to the same plant group type as in the tool you want to interface using SmartPlant Foundation. For example, in SmartPlant Instrumentation, cabinets and panels are always assigned to the highest plant group type.

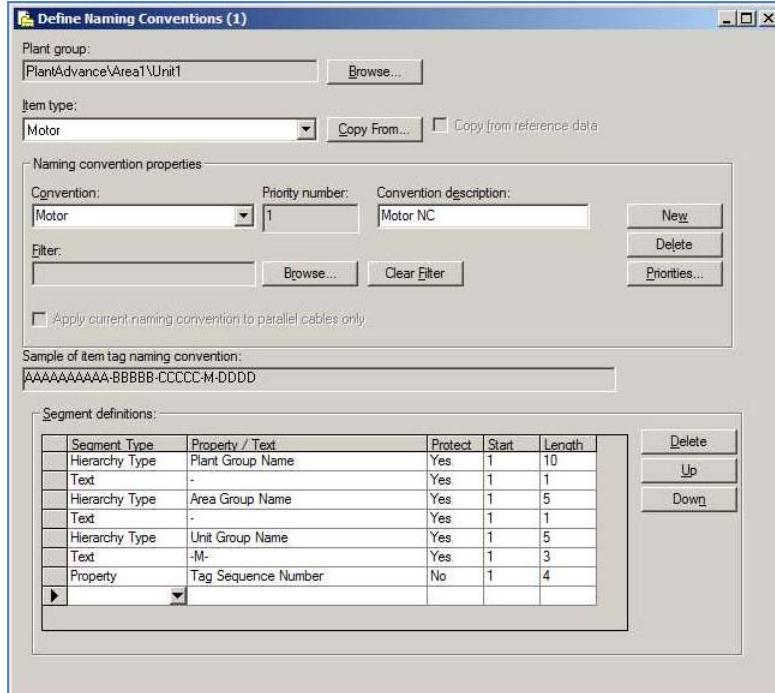
The following labs will demonstrate the way NC can be applied for different item types:

Define naming conventions using all plant group levels in item tag definition

For motor item type specify the plant group type to be unit and the item type uniqueness level to be plant.



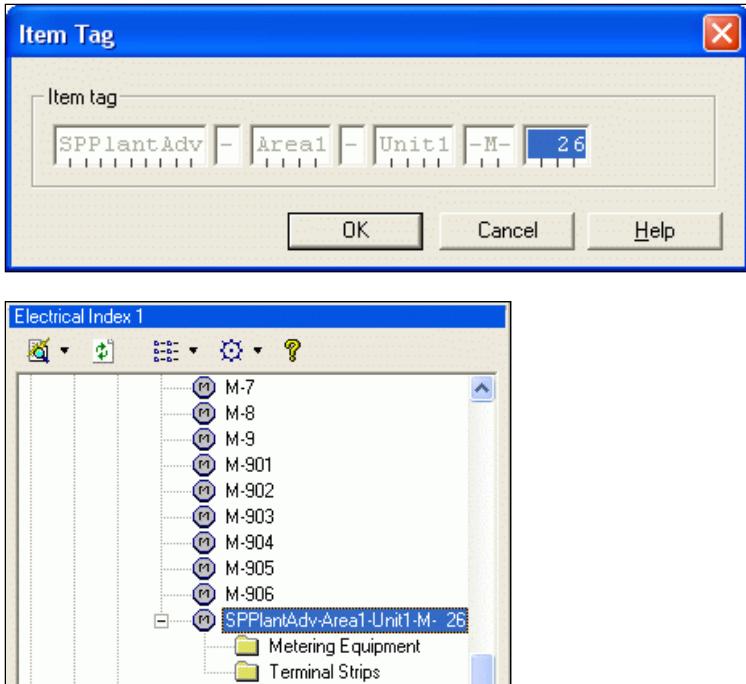
Define the following motor naming convention:



Defining Project Naming Conventions

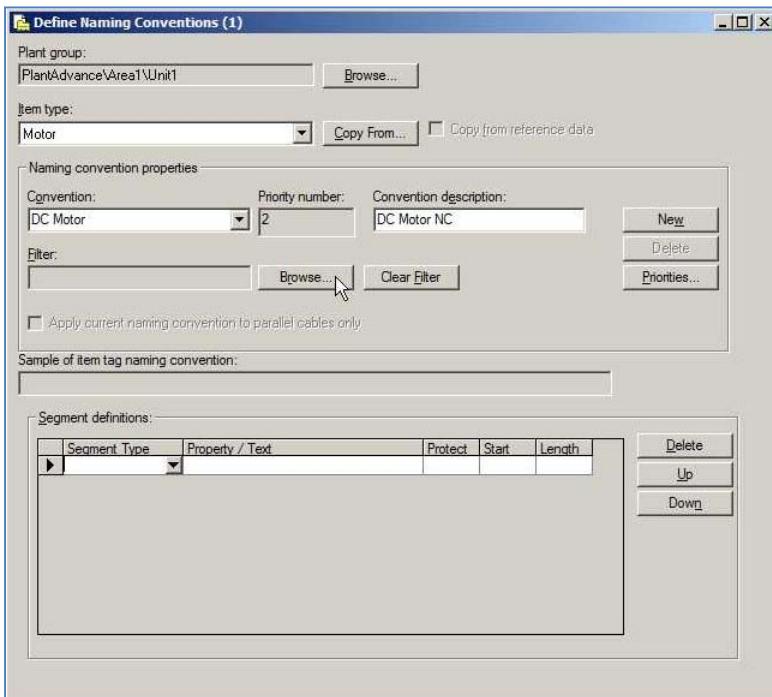
Save the change and reopen SmartPlant Electrical.

Create new motor.



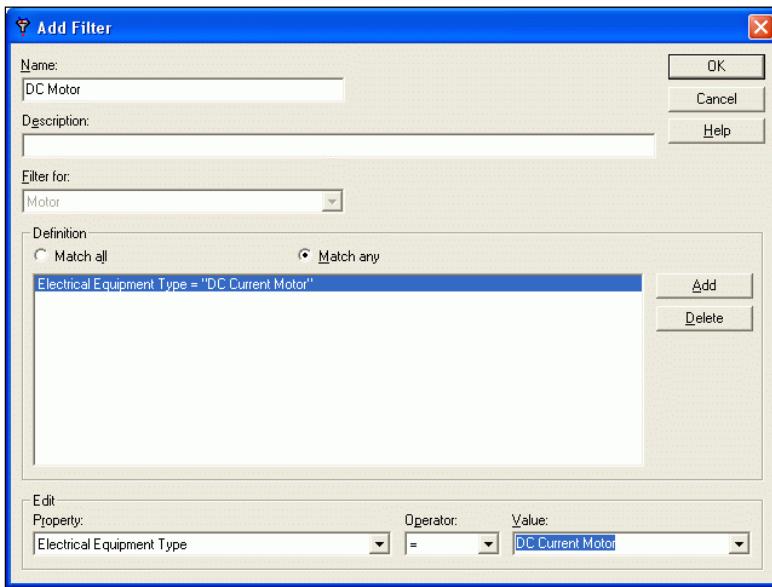
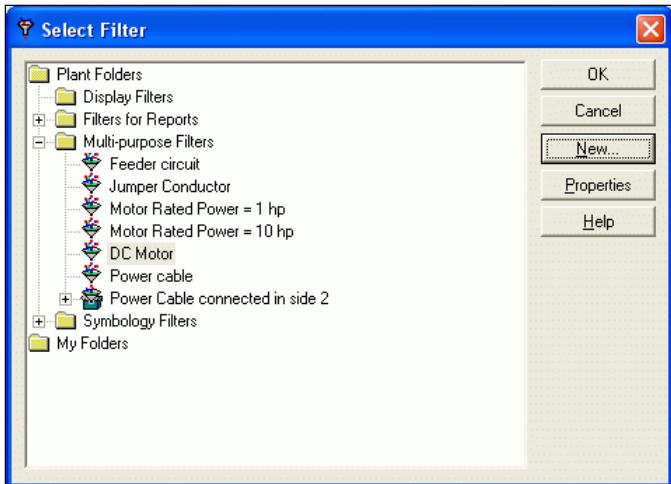
Multiple Naming Conventions for Motors

Define a second naming convention for motor item type, based on motor type. For this, we shall create a special filter, say for DC Motor:

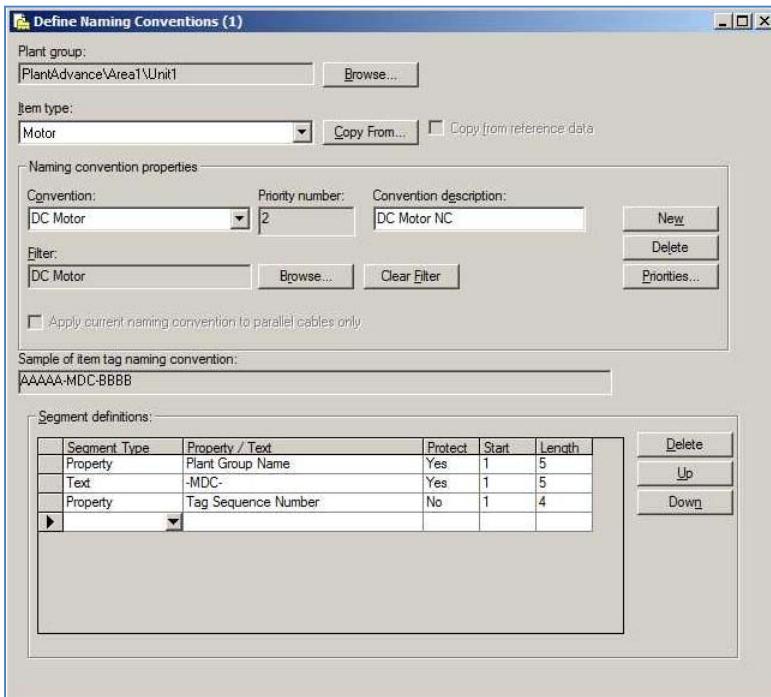
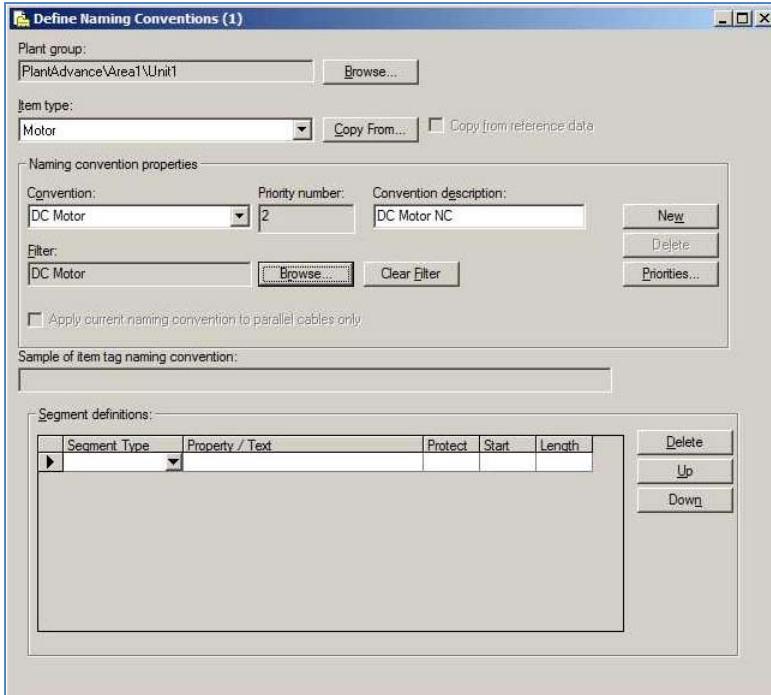


Defining Project Naming Conventions

Select the browse button and add filter for DC Motors.

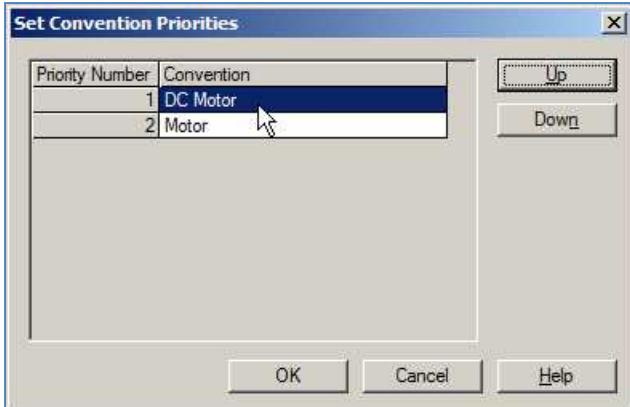


Defining Project Naming Conventions

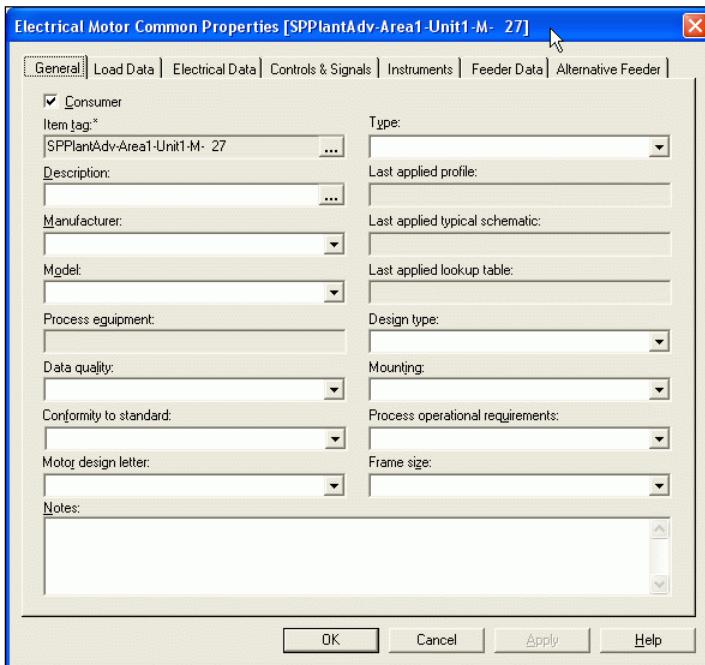


It is important to make sure that priorities are fixed so that the most selective NC will have higher priority, otherwise the system will not be able to detect it properly.

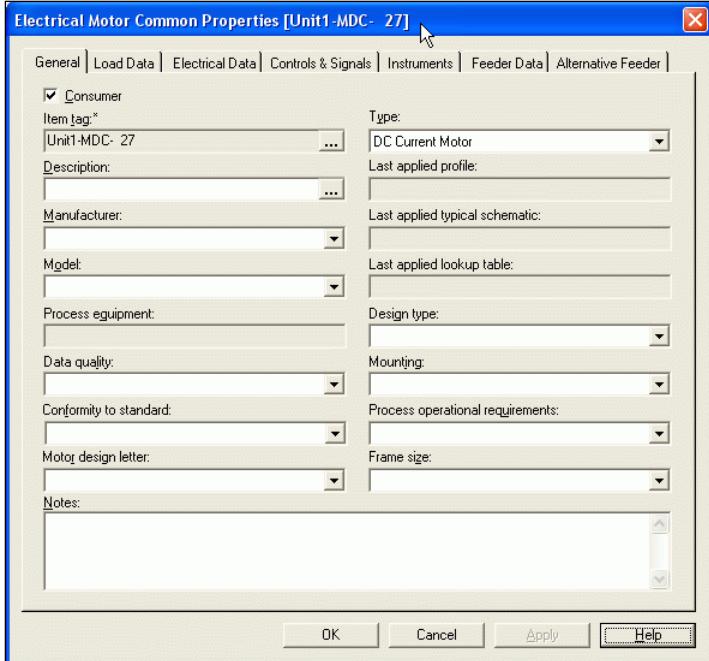
Defining Project Naming Conventions



Reopen SPEL and create a motor in electrical index.



Change the type of the motor to dc current motor and select apply. The item tag changes to comply with the DC motor naming convention.



Multiple Naming Conventions for Cables

Defining multiple naming conventions for Cables and including the connected equipment tag in the NC.

The following lab will describes how we can define different NC for different cable categories.

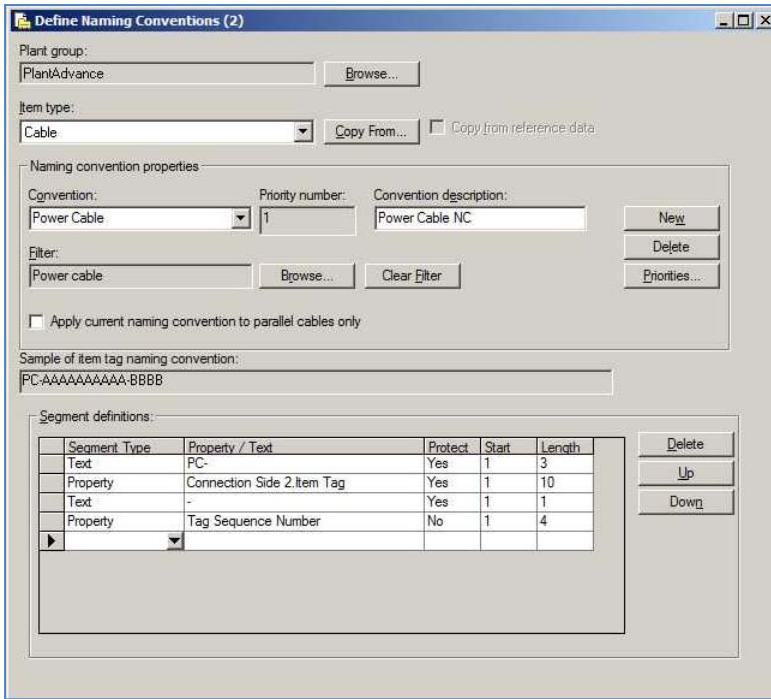
For cable item type specify the plant group type to be plant and the item type uniqueness level to be plant.

Item Type	Property	Value
Bus	Plant group type	Plant
Busway	Item type uniqueness level	Plant
Cabinet	Default block	
Cable	Is unique	Yes
Cable Drum	Item type identifier	CABLE
Cableway	Last created sequence number	34

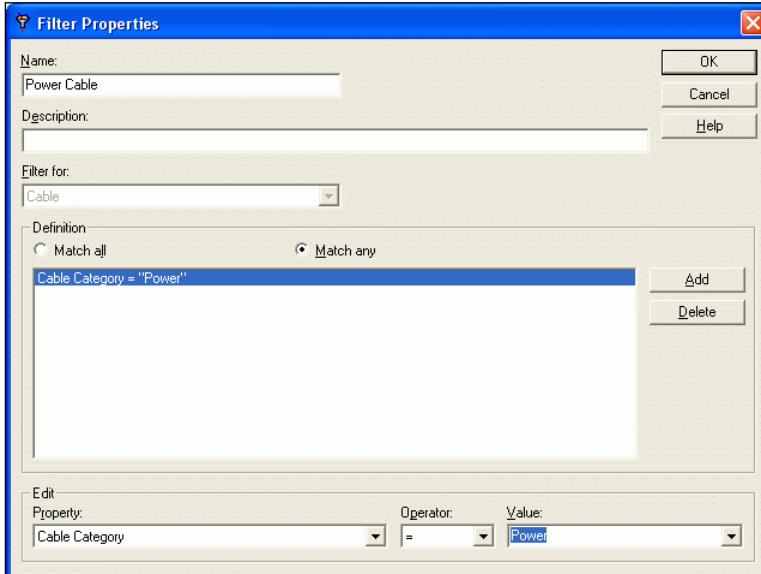
Create naming convention for cable using filter base on the cable category property.

Power Cable

Defining Project Naming Conventions

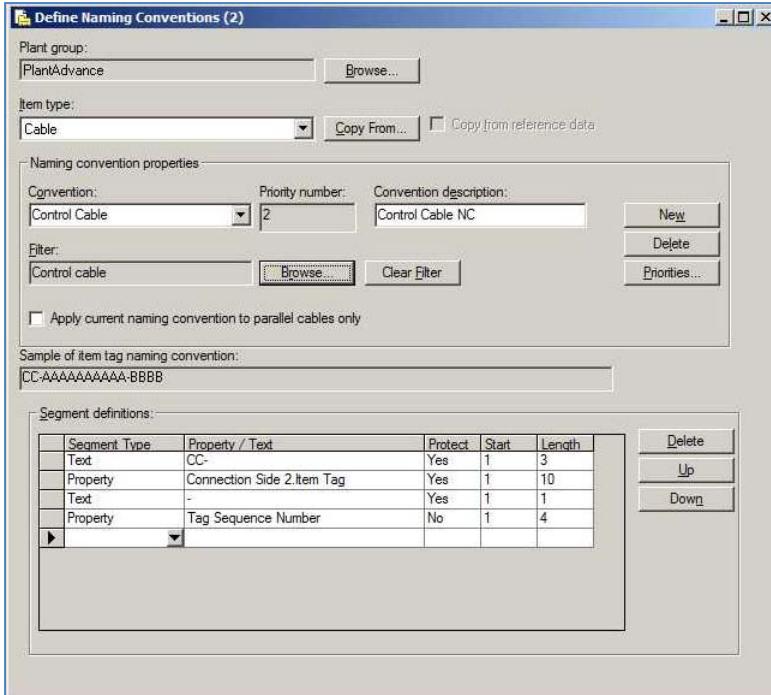


The filter for power cable will be:

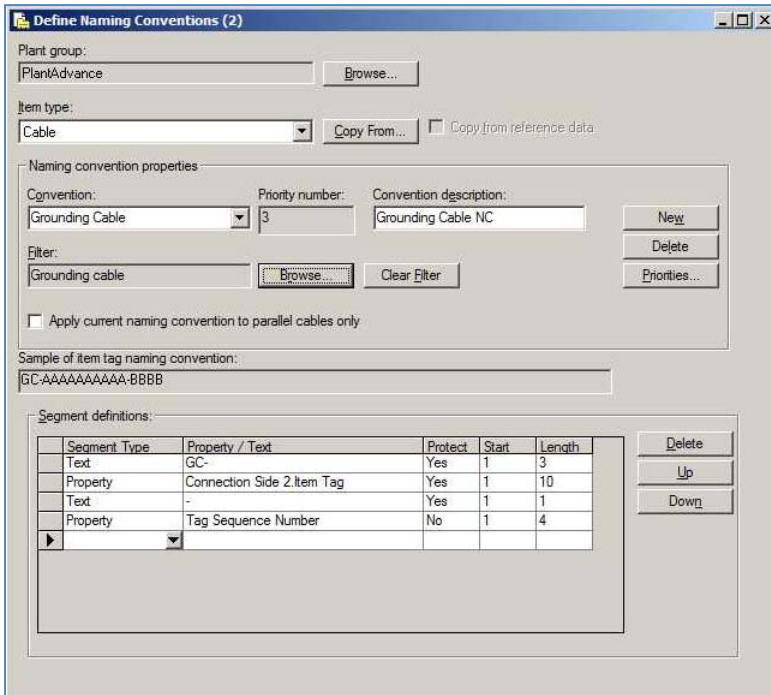


Control Cable

Defining Project Naming Conventions

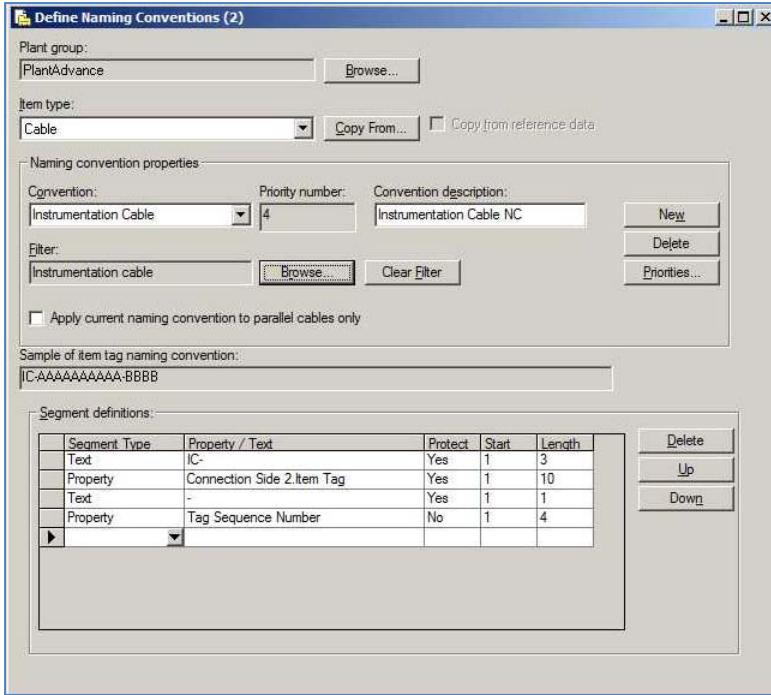


Grounding cable



Instrumentation cable

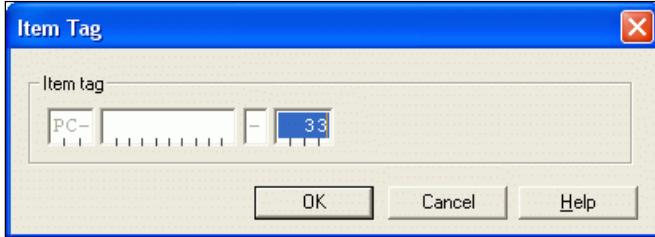
Defining Project Naming Conventions



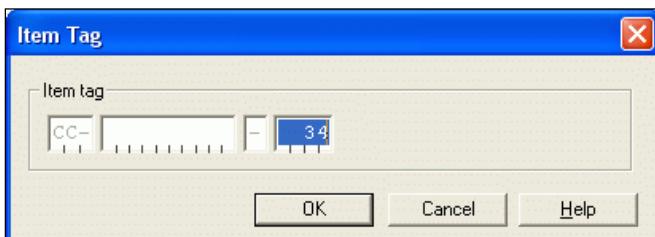
Priorities are given automatically by the system. There is no need for to change them, unless there is a special need for that.

After save the naming convention change. Reopen SPEL and test the new cable naming conventions.

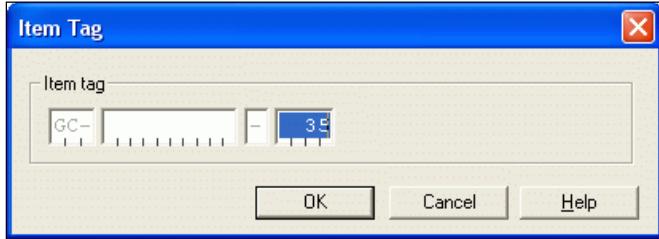
Power Cable



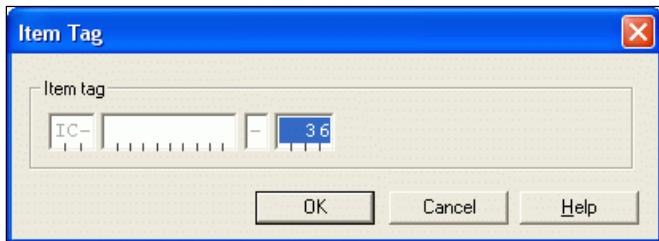
Control Cable



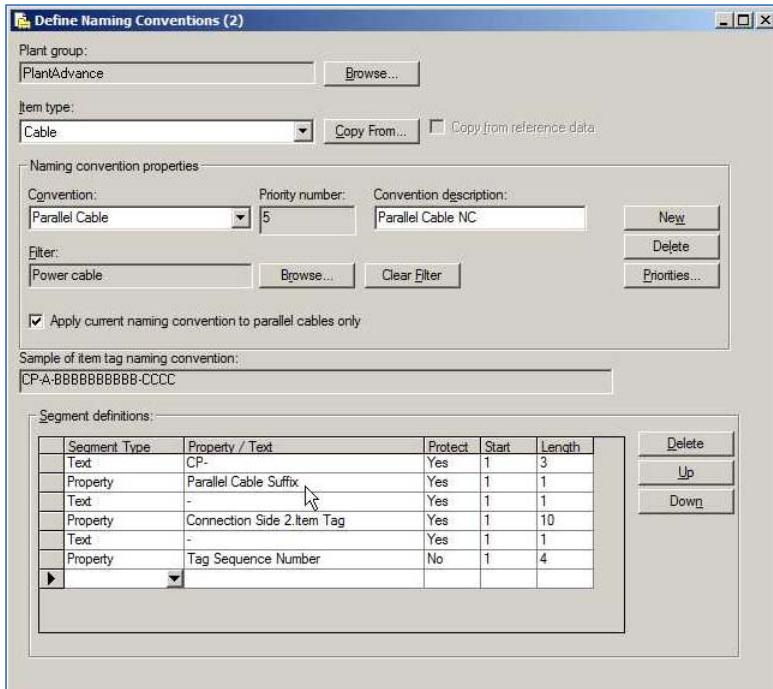
Grounding Cable



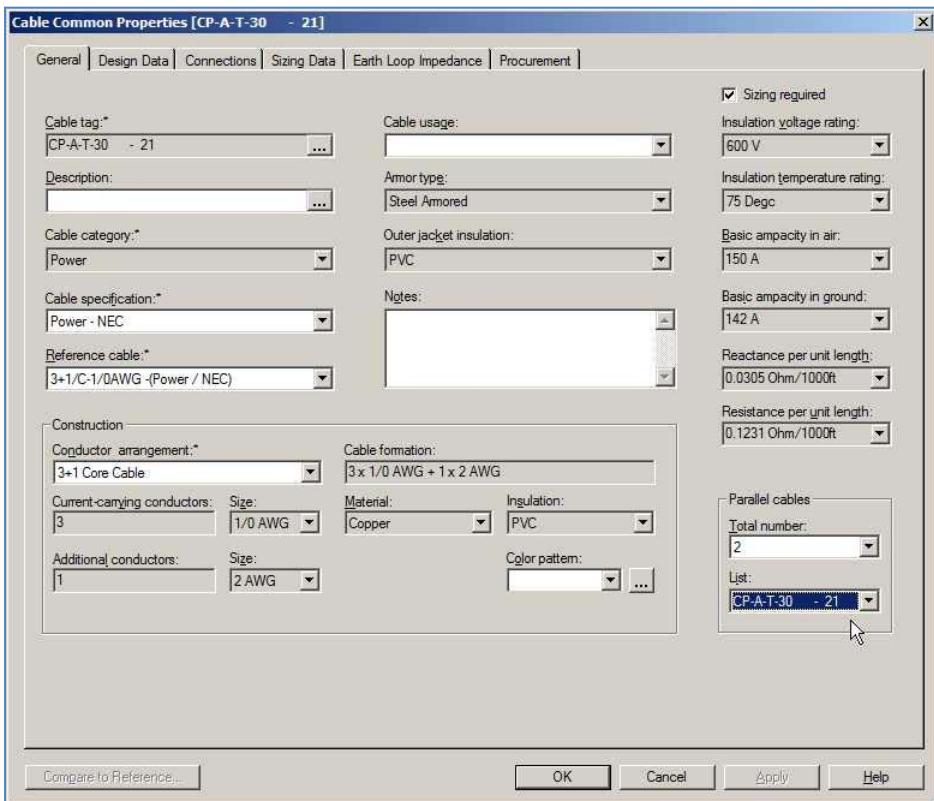
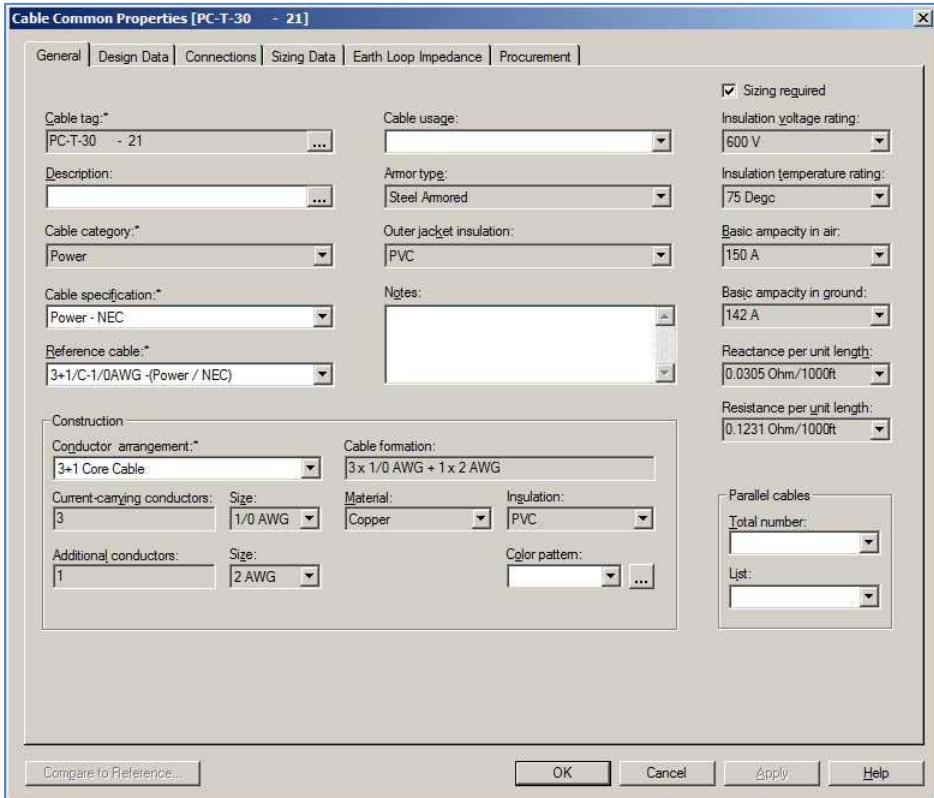
Instrumentation Cable



Define Parallel Cable Naming Convention

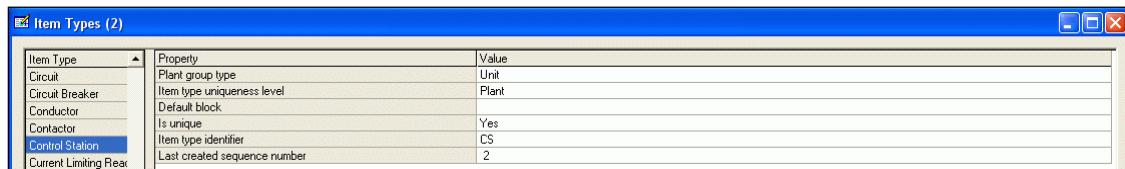


Defining Project Naming Conventions



Define Naming Conventions for Control Stations

For control station item type specify the plant group type to be unit and the item type uniqueness level to be plant.

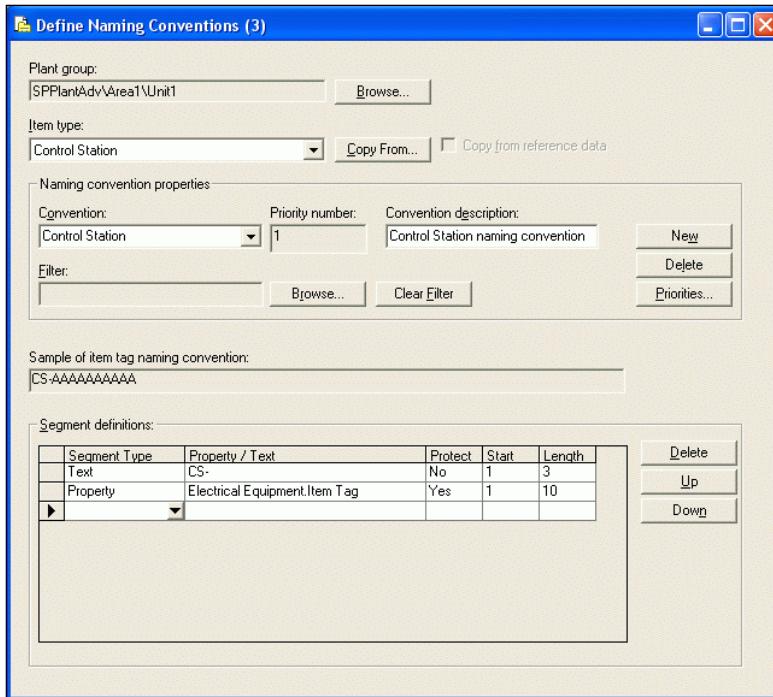


The screenshot shows a Windows-style dialog box titled "Item Types (2)". On the left is a vertical list of item types: Item Type, Circuit, Circuit Breaker, Conductor, Contactor, Control Station, and Current Limiting Reactor. The "Control Station" item is selected and highlighted with a blue background. To the right of the list is a table with two columns: "Property" and "Value". The table contains the following data:

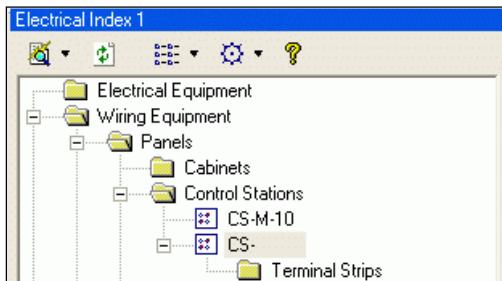
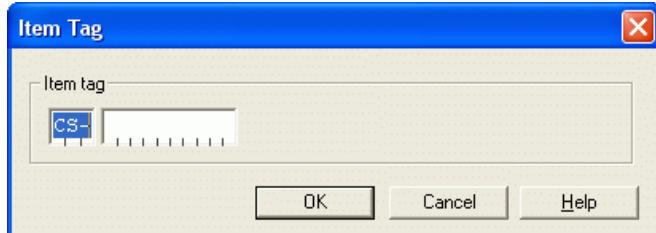
Property	Value
Plant group type	Unit
Item type uniqueness level	Plant
Default block	
Is unique	Yes
Item type identifier	CS
Last created sequence number	2

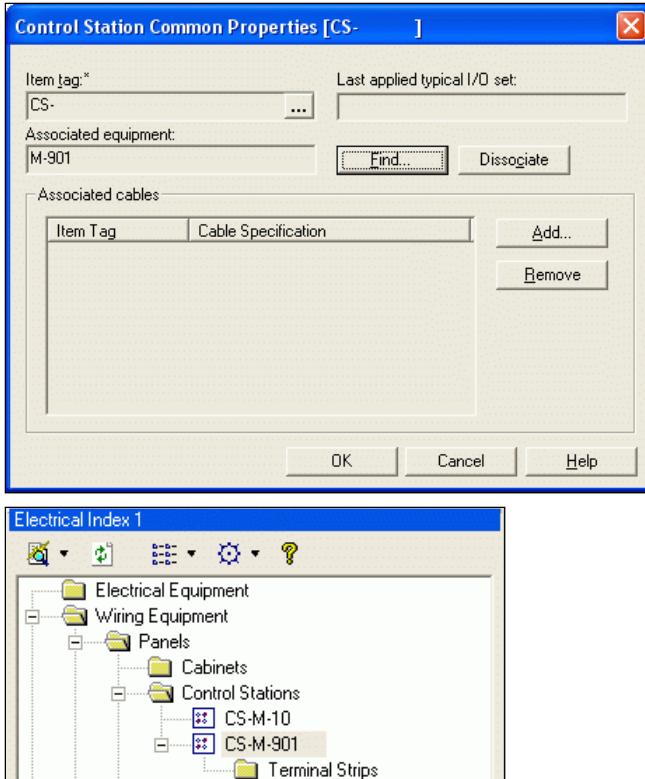
Defining Project Naming Conventions

The following naming convention for control station includes the associated equipment.



After save the change, reopen SPEL, create new control station in electrical index. And associate it with motor.





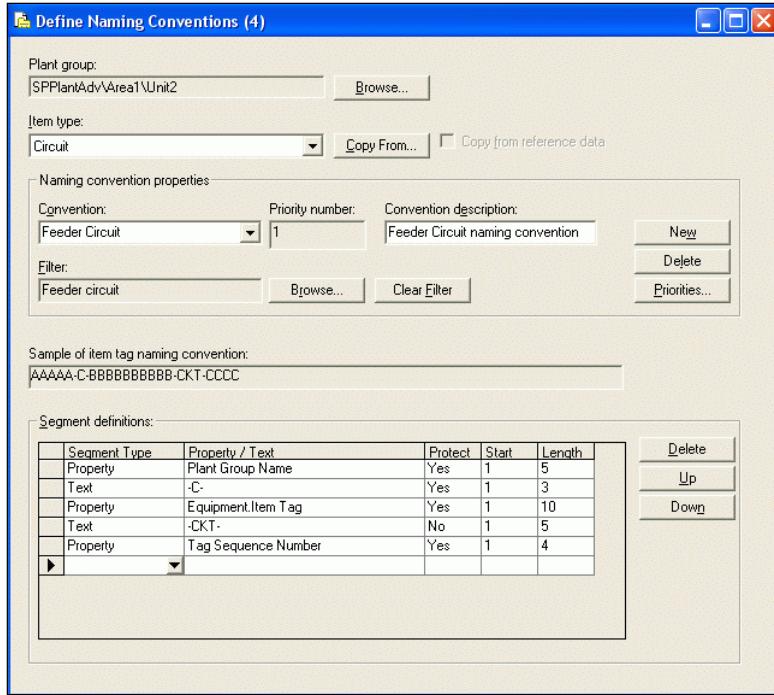
Define Naming Conventions for Circuits and Cells

It is possible to include the load's item tag in the circuits and cell's naming convention.

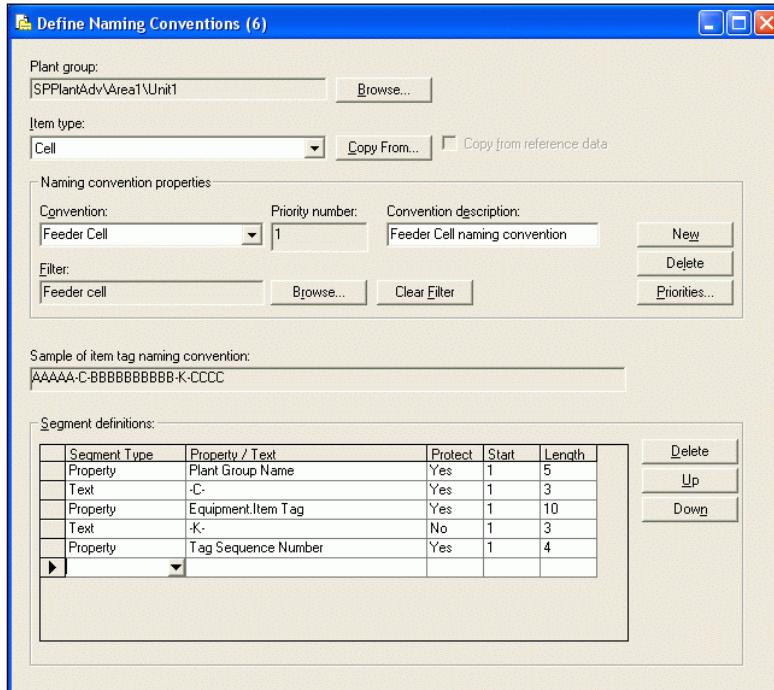
An example is shown below:

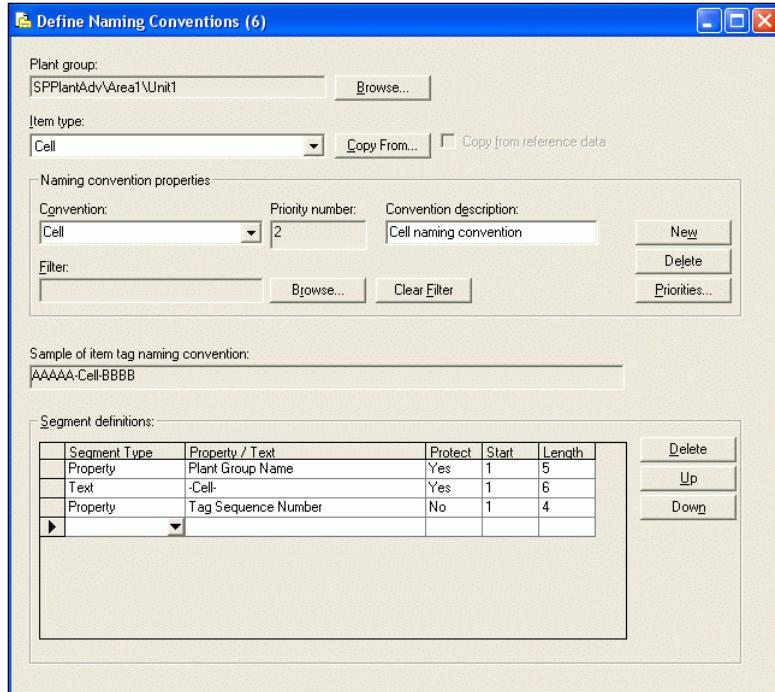
1. For circuit item type set plant group type to be unit, and item type uniqueness level to be parent.
2. For cell item type set plan group type to be unit, and item type uniqueness level to be power distribution board.
3. Create a naming convention for feeder circuits, based on a filter for feeder circuits:

Defining Project Naming Conventions

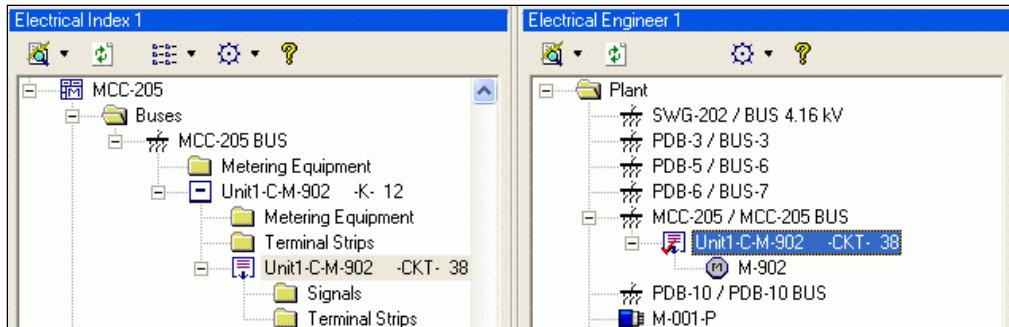


4. For the cell, create two naming conventions, one for feeder cells and second for default cells. Feeder cells naming convention should get the higher priority.





Associating the cell and circuit to a load, will be properly presented in the EI and EE.



Multiple loads assigned to a cell/circuit – last load governs the cell name.

Note

Make sure that there is a default naming convention defined for cells that are not of feeder type. Failing to define this results in the cell created with Comply with NC='False', which will require the user to manually switch it to 'True' later, after you associate it to a feeder circuit.

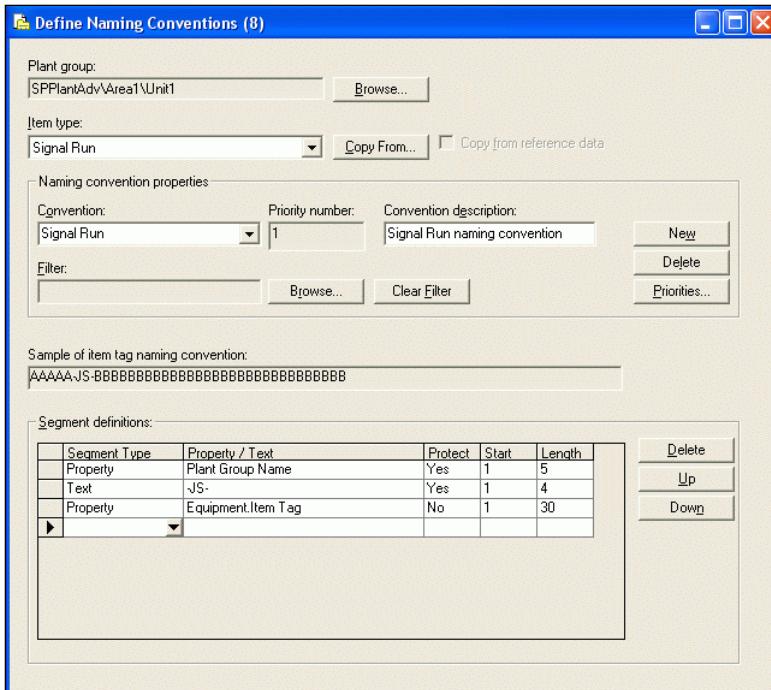
Define a Naming Convention for Signals

Use Equipment.ItemTag property in the Signal-Run naming convention in order to include main equipment item tag.

Typical examples:

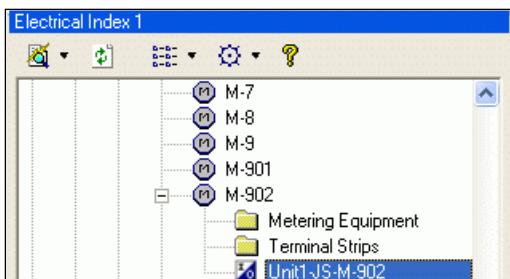
For signal run item type specify the plant group type to be unit, and item type uniqueness level to be plant. Include the name of the equipment be part of the signal item tag. The naming convention would look as follows:

Defining Project Naming Conventions

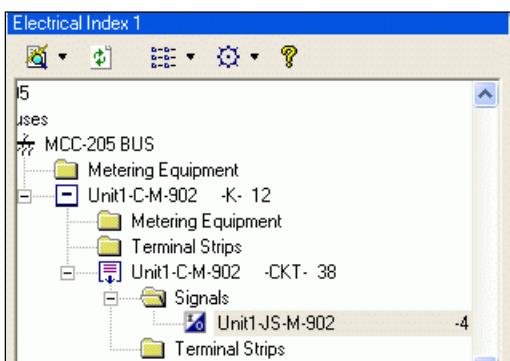


Save the change and reopen SPEL.

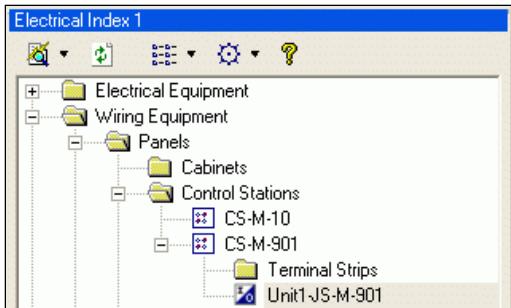
The signal can be created under the load.



More typical is to create the signal under the feeder circuit.



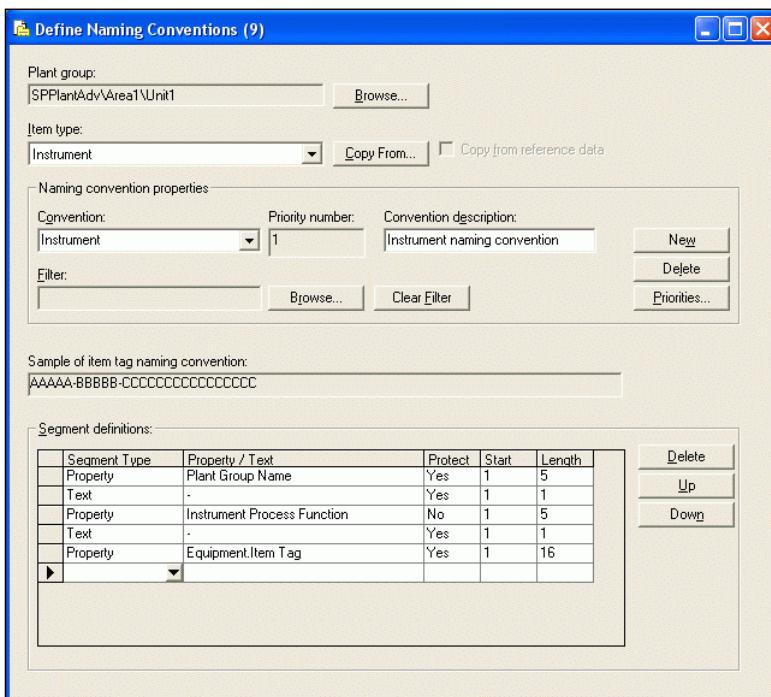
Or to create the signal under the control station.



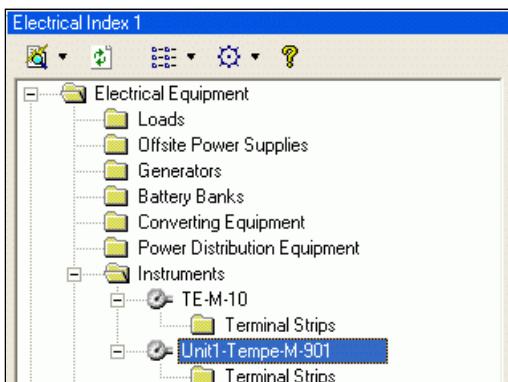
Define Naming Conventions for Instruments

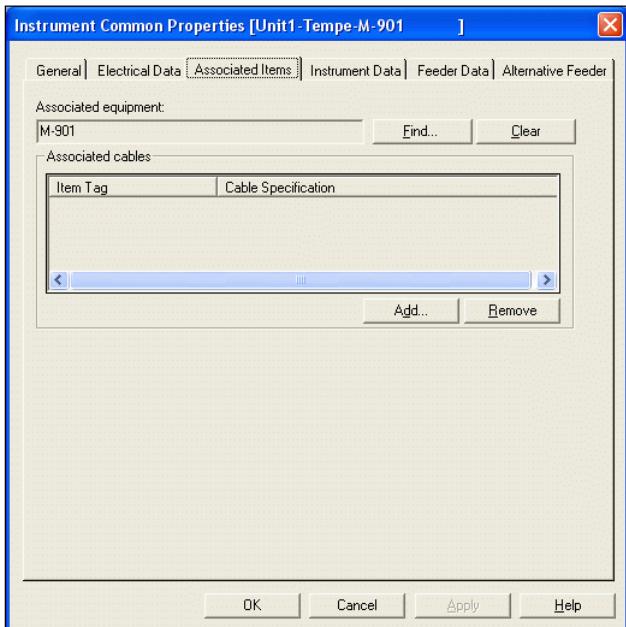
You can include the main equipment that the instrument is associated to appear in its item tag.

To demonstrate it let's specify the plant group type to be unit, and item type uniqueness level to be plant. And let's create the following naming convention.



This naming convention will result in the following tagging of instruments in the project:

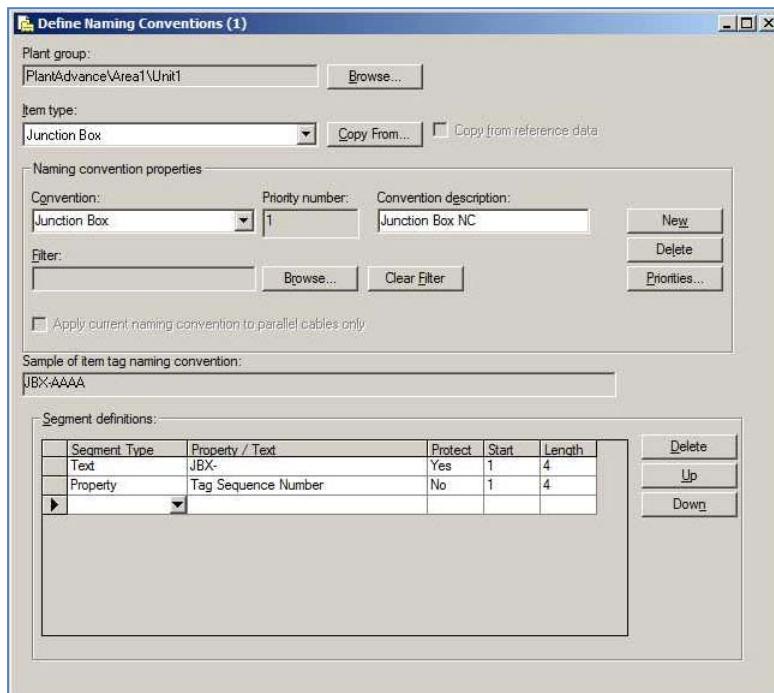




Copy Naming Conventions

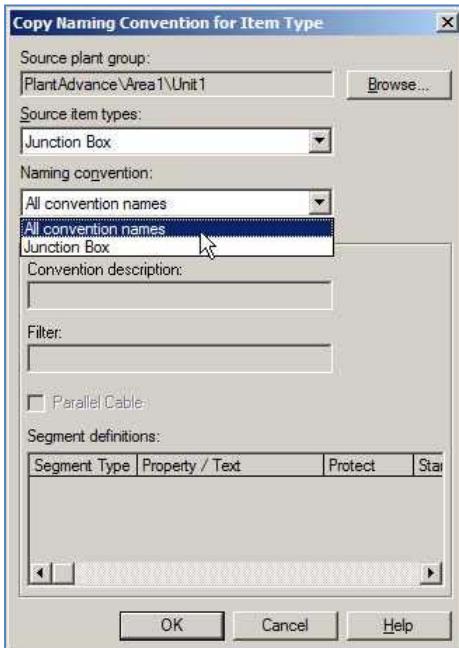
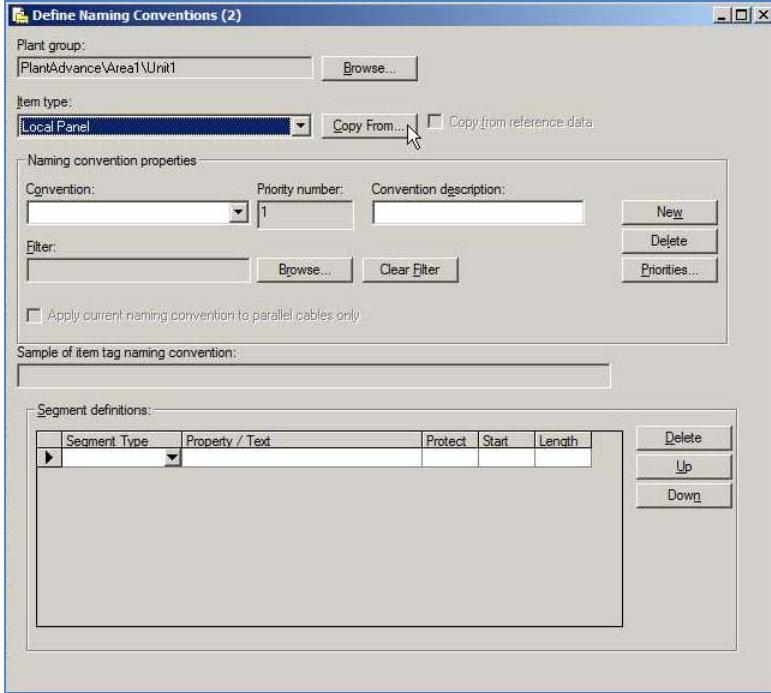
Copy Naming convention between item types

Allows you to copy naming convention from one item type to another. For example, let's say we defined naming convention for Junction Box in unit 1.



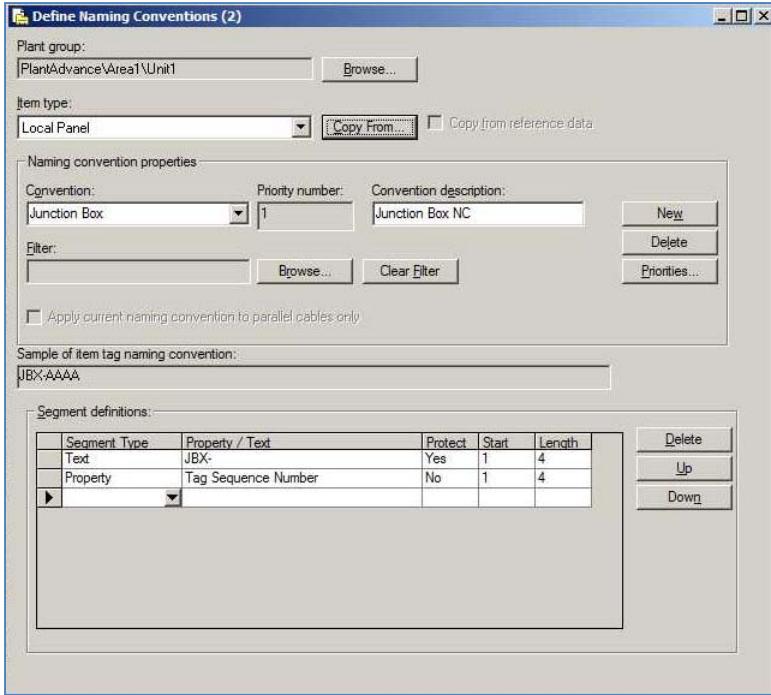
Defining Project Naming Conventions

We can use it and copy the same naming convention to local panel item type and save us the time of redefine the same naming convention.

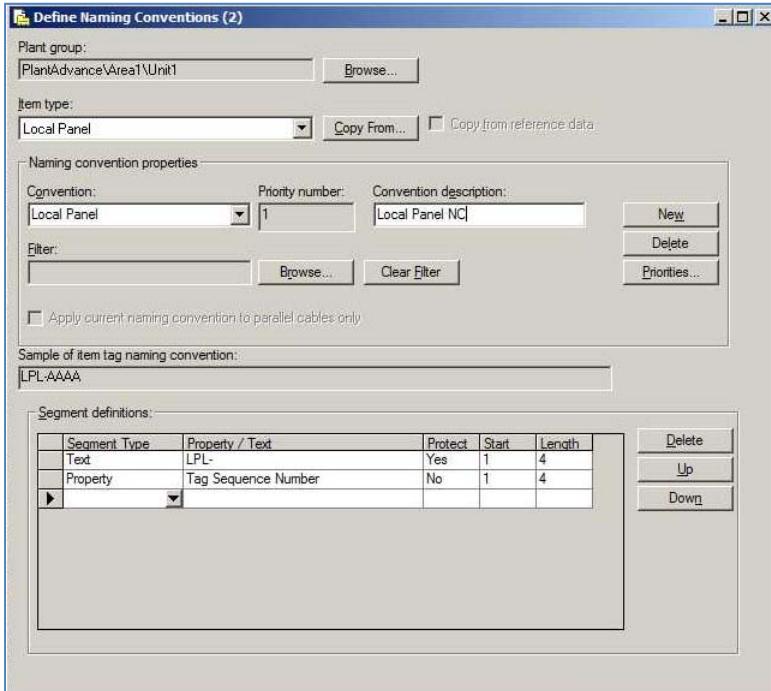


Notice: You can select all naming convention to be copied, or specific naming convention.

Defining Project Naming Conventions



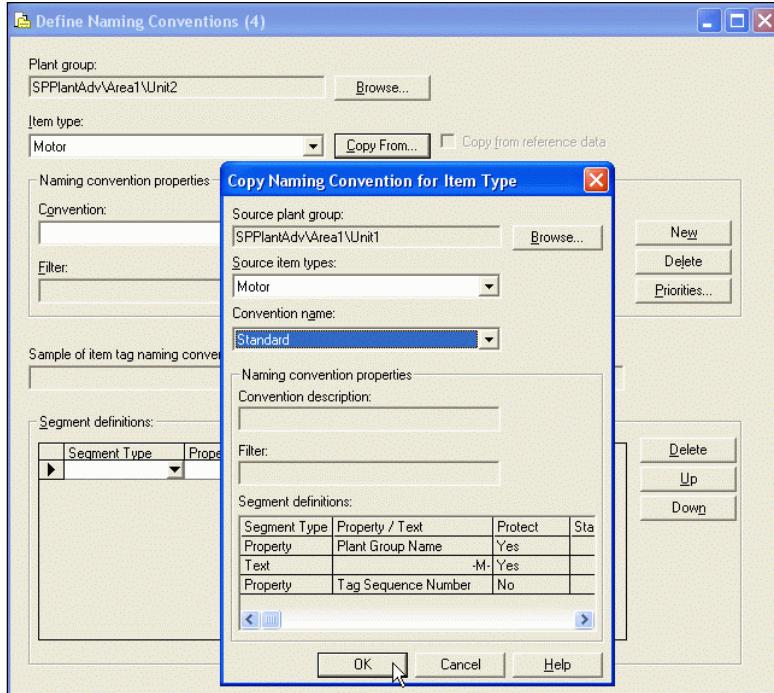
You can modify the text in the first segment to reflect local panel.



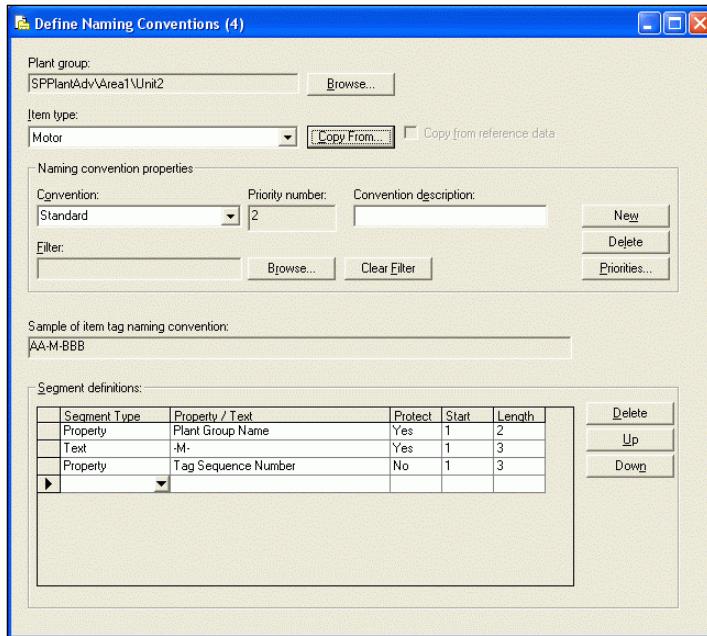
Copying naming conventions from unit 1 to unit 2

Assuming naming conventions have been defined for Unit1 and we want to make use of that into Unit2:

1. Log into the new Unit2 by browsing to unit2 in the Select Plant Group window.
2. Select the item type for which you want to copy the naming convention.
3. Click on the ‘Copy from’ button and select the Source plant group as Unit 1.
4. Select from Source item types the item that you want to copy from:



5. Click **OK** copies the ‘Standard’ motor’s naming convention from unit 1 to unit 2.



Notes:

- In case of multiple NC, all of them will be copied.
- This option gives you also the ability to copy NC of one item type to another item type across units.

Copy Naming Conventions Across Plant Groups Belonging to The Same Site

To copy a complete set of NC from Unit 1 to Unit 2, or from any two different plant groups belonging to the **same site**, for all the item types that have NC definitions:

1. Click **Options > Naming Conventions > Copy for Plant Group**.
2. Browse to and then select the source Unit1.
3. Browse to and then select the target Unit2.

Click **OK** to complete the copying the whole set of NC from Unit1 to Unit2.

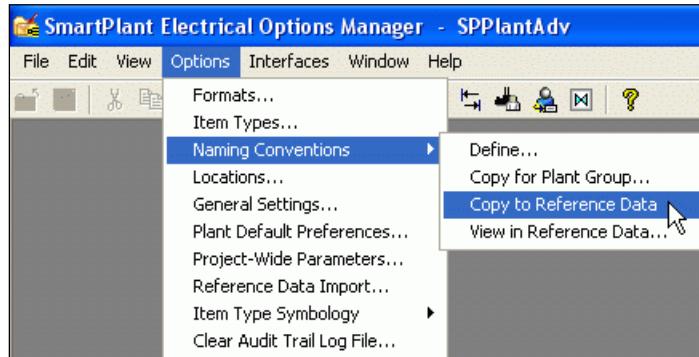


Copy Naming Conventions Across Plant Groups Across Sites

This feature enables to copy NC across plant groups' independent from their location.

The idea behind this process is that users can copy the defined NC as a reference NC template as part of the seed plant, and later, when creating a new plant, use the seed plant reference NC as a source for the NC of the new plant.

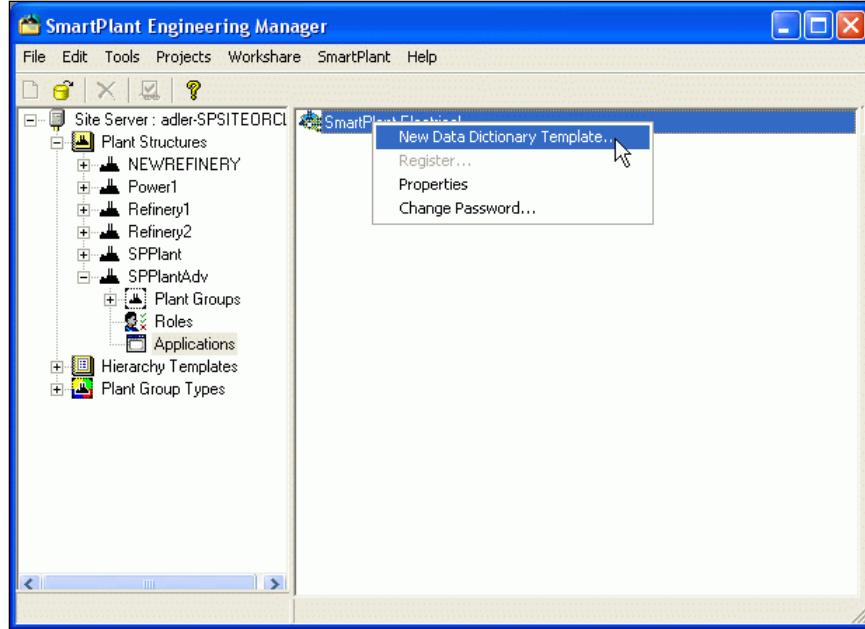
1. Save your current plant NC definitions.



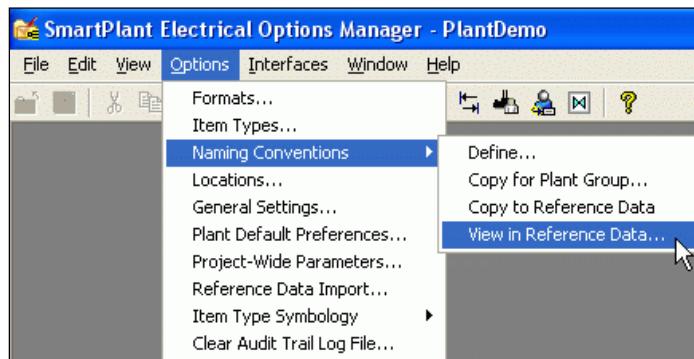
2. This operation may overwrite any previous saved reference NC.



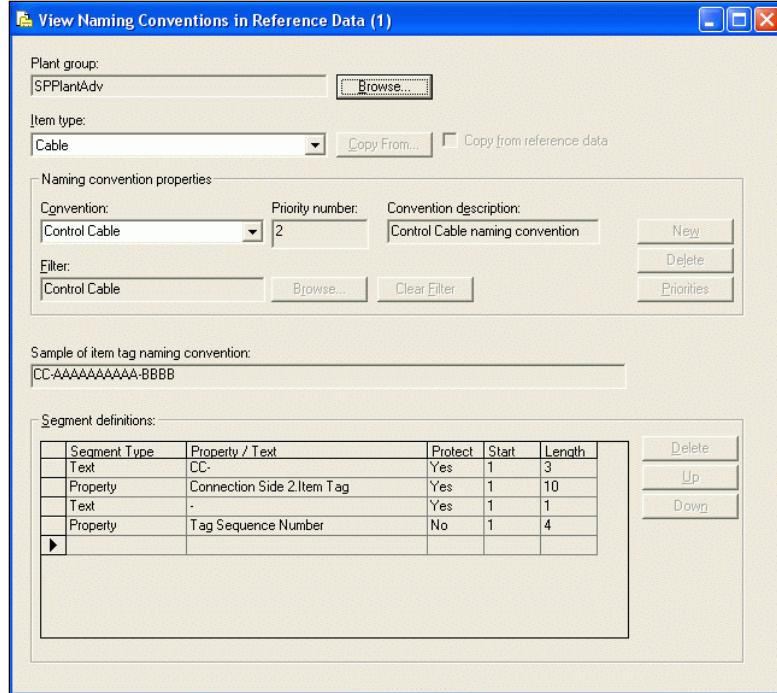
3. From SmartPlant Engineering Manager create new data dictionary template.



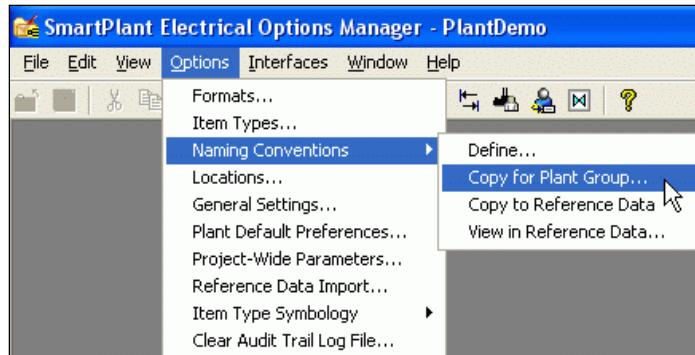
4. From SmartPlant Engineering Manager create new plant and during associate application use the saved data dictionary template from step 3 (it includes the naming convention information you would like to copy).
5. In the new plant, Options Manager let you the chance before copying the reference NC, to view it:



This opens the NC definitions screen in a view only mode, allowing you to browse to all reference Units (the Unit names are shown as saved, just as a reference).

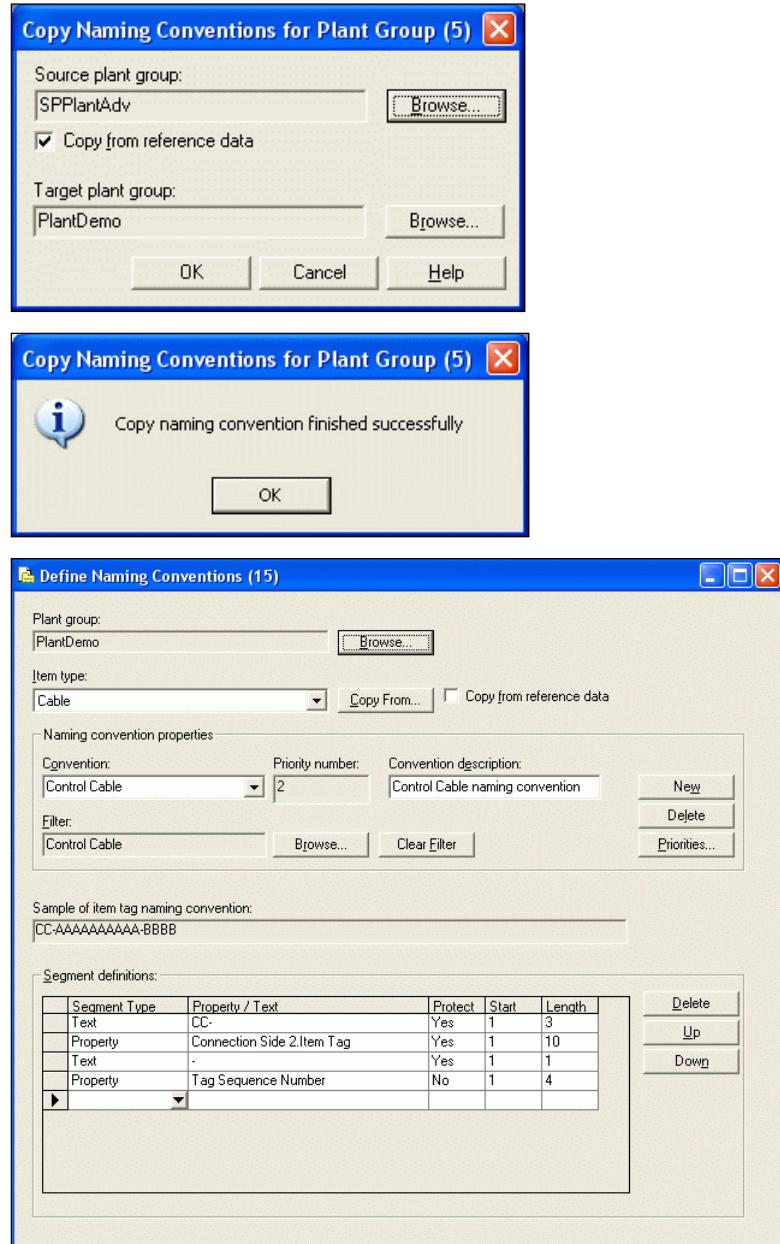


6. To copy the reference NC to your new plant use the copy naming convention for plant group option:



Defining Project Naming Conventions

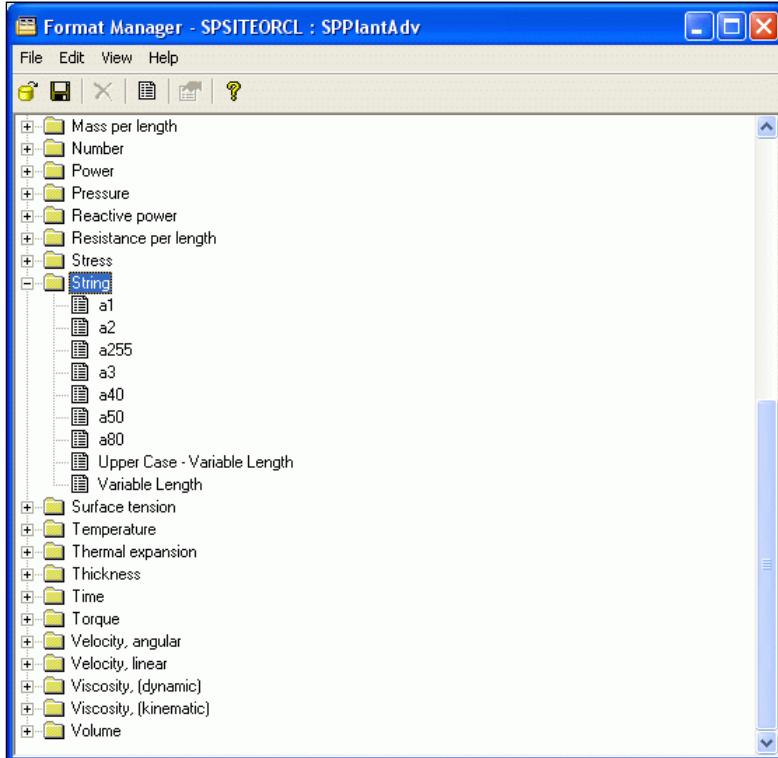
For the source select the **Copy from reference data only** check box, and browse to select the source plant group. For the target browser to select the target plant group and click ok to start the copy.



Padding Zeros in Item Tag Sequence Number

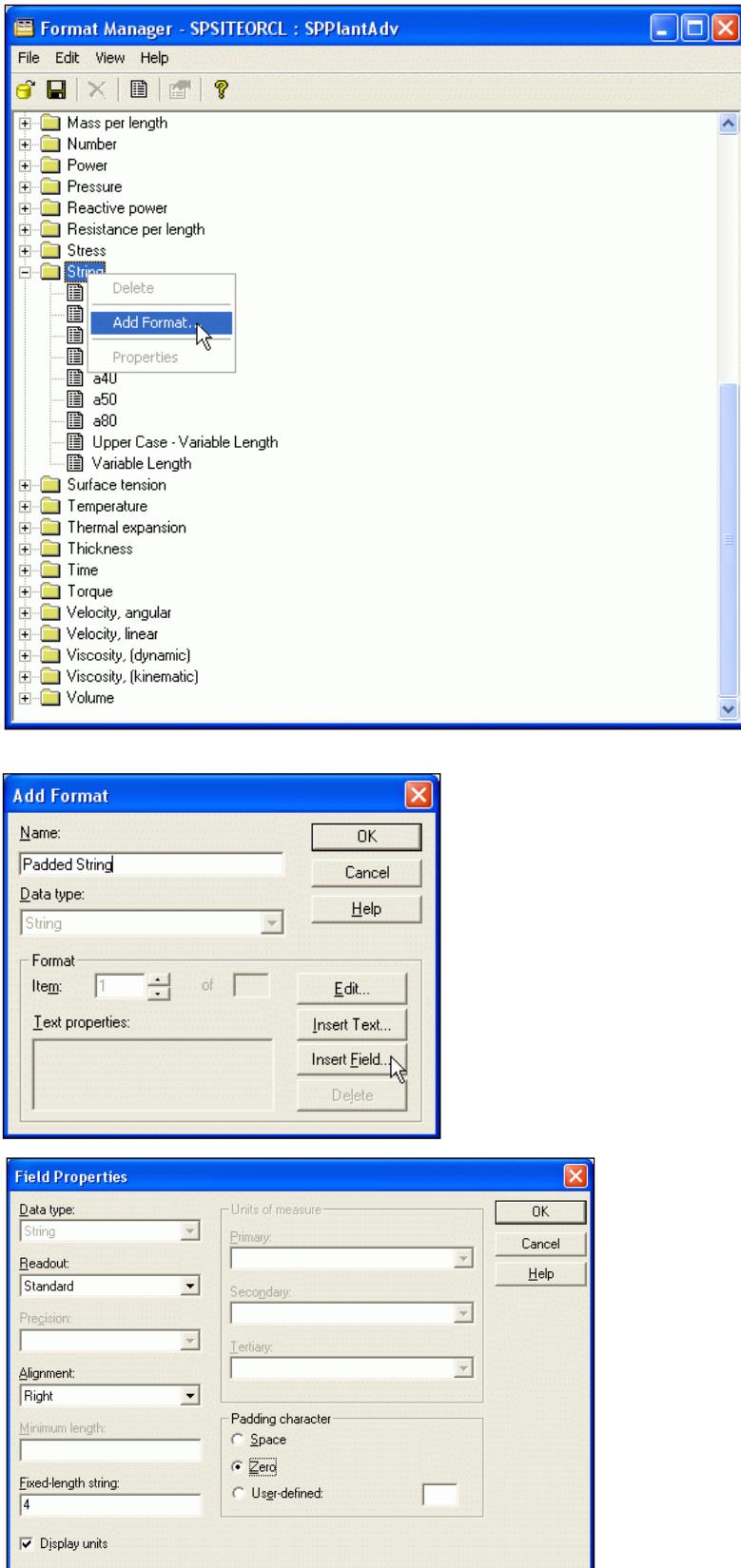
It is possible to have zeroes filled in the item tag sequence number rather than spaces or blanks. This is done by create new format in format manager with padding zeros as follows:

1. Open the Format Manager.

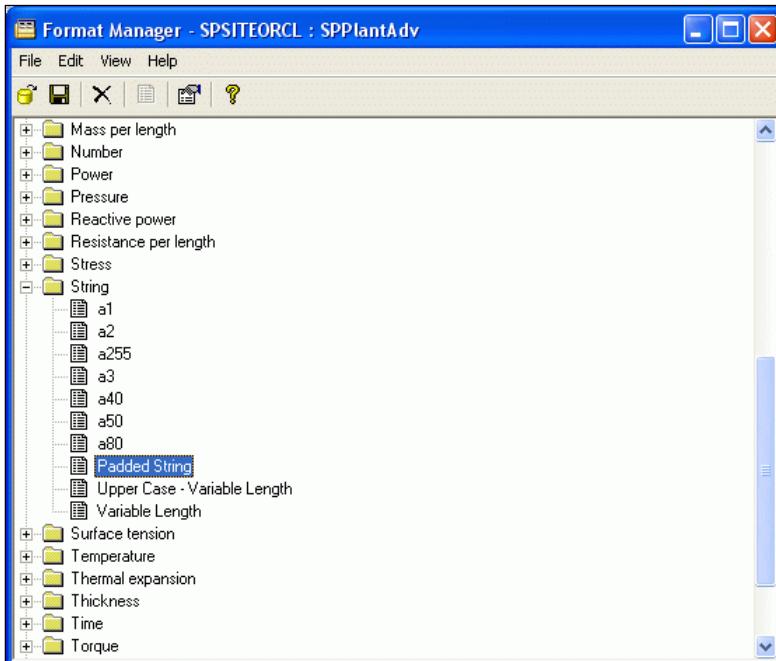
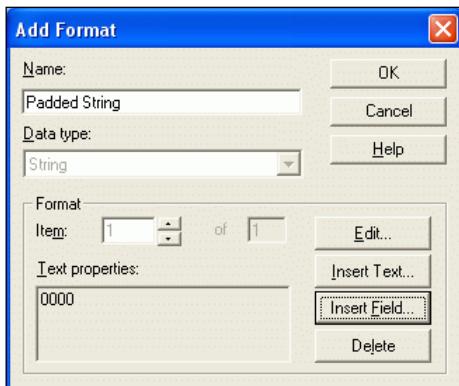


2. Add a new String data type format as shown and save the change.

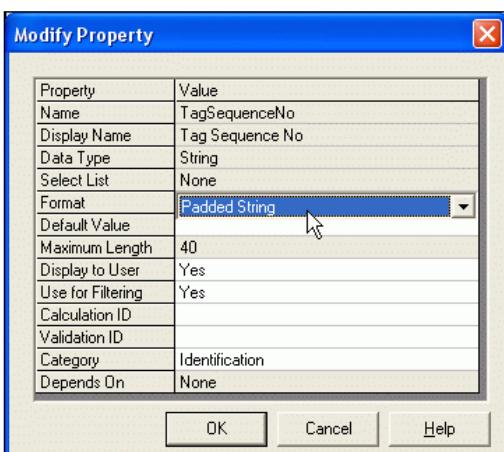
Defining Project Naming Conventions



Defining Project Naming Conventions

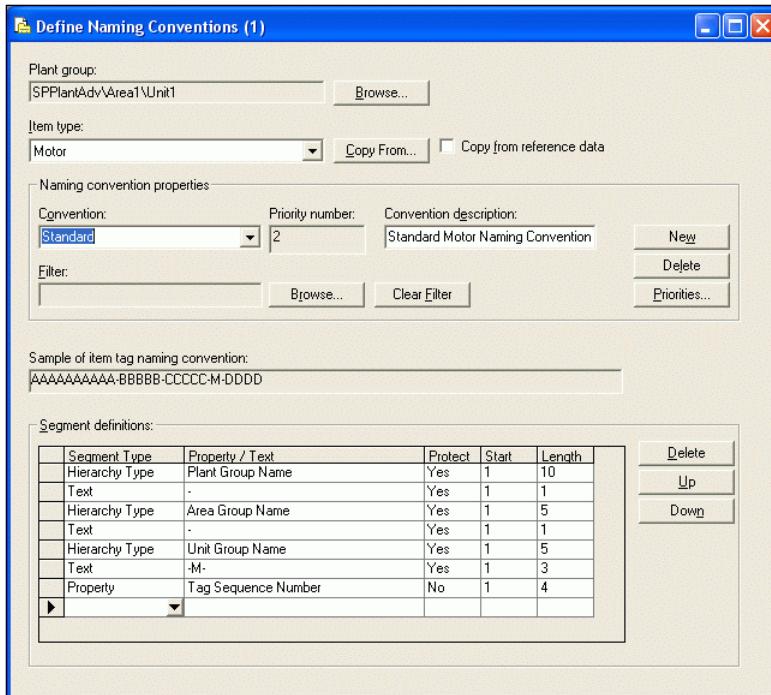


3. Open the Data Dictionary Manager and select Database Tables.
4. Select the Plant item table and double click on the Tag Sequence number property.
5. Change the format from Upper Case – Variable Length to Padded String and save the change.

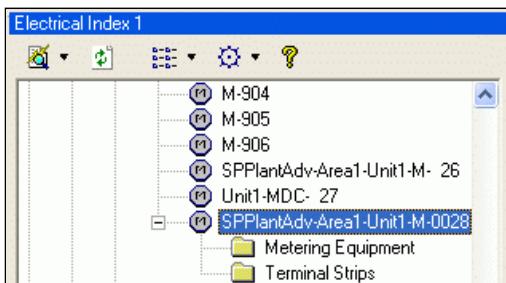
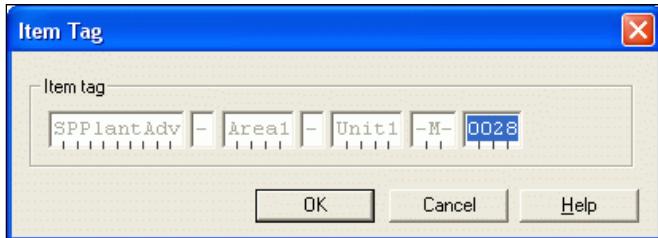


Defining Project Naming Conventions

6. Create naming convention that includes tag sequence number.



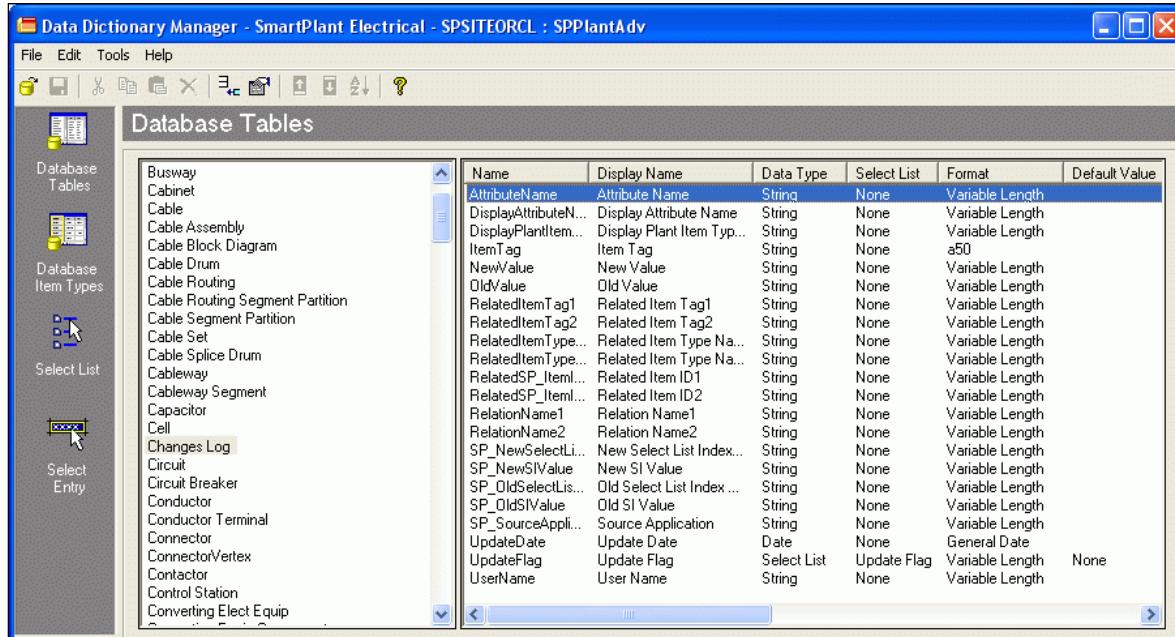
7. With this done, all the spaces that are left blank in the Tag Sequence number segment will be padded with zeroes, as shown in the following screen shot:



Note: Rename the tag will still pad the zeroes.

Changes log

A table exists in SmartPlant Electrical database, named ‘Changes log’ with the following properties:

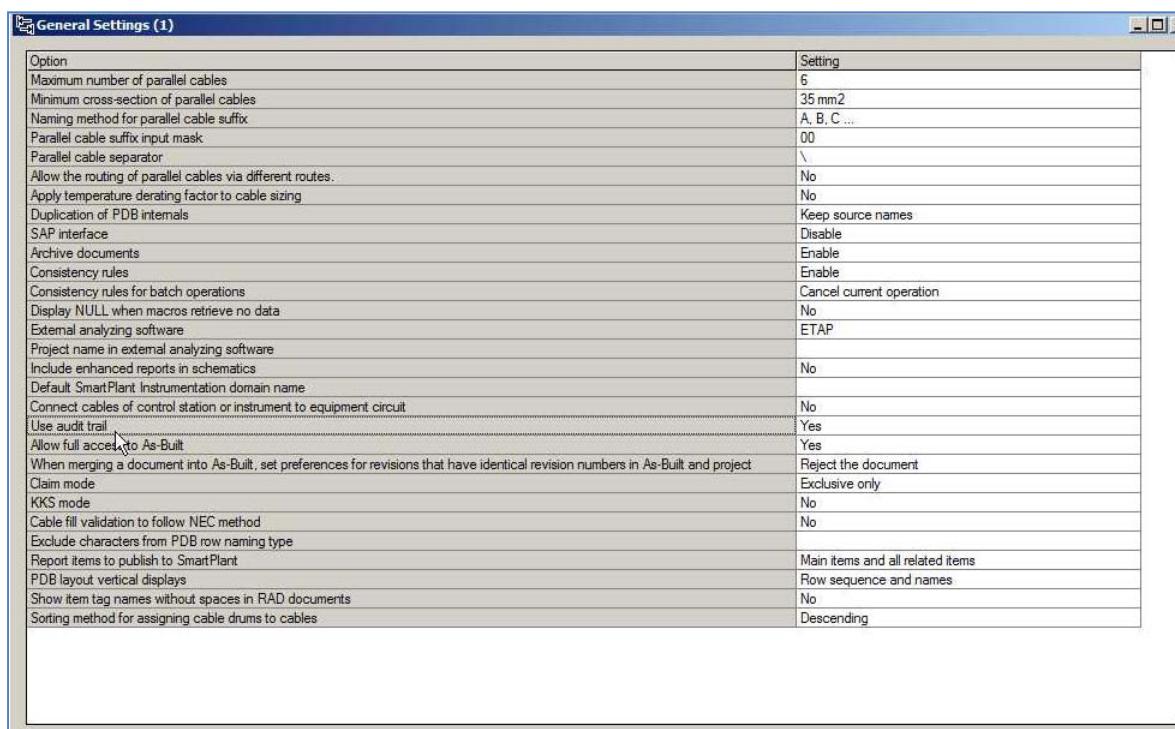


The screenshot shows the Data Dictionary Manager interface. On the left, there's a tree view of database tables and item types. The 'Changes Log' table is selected. On the right, a detailed table view shows its properties:

Name	Display Name	Data Type	Select List	Format	Default Value
AttributeName	Attribute Name	String	None	Variable Length	
DisplayAttributeName	Display Attribute Name	String	None	Variable Length	
DisplayPlantItem...	Display Plant Item Typ...	String	None	Variable Length	
ItemTag	Item Tag	String	None	a50	
NewValue	New Value	String	None	Variable Length	
OldValue	Old Value	String	None	Variable Length	
RelatedItemTag1	Related Item Tag1	String	None	Variable Length	
RelatedItemTag2	Related Item Tag2	String	None	Variable Length	
RelatedItemType...	Related Item Type Na...	String	None	Variable Length	
RelatedItemType...	Related Item Type Na...	String	None	Variable Length	
RelatedSP_Iteml...	Related Item ID1	String	None	Variable Length	
RelatedSP_Iteml...	Related Item ID2	String	None	Variable Length	
RelationName1	Relation Name1	String	None	Variable Length	
RelationName2	Relation Name2	String	None	Variable Length	
SP_NewSelectLi...	New Select List Index...	String	None	Variable Length	
SP_NewSIValue	New SI Value	String	None	Variable Length	
SP_OldSelectLi...	Old Select List Index ...	String	None	Variable Length	
SP_OldSIValue	Old SI Value	String	None	Variable Length	
SP_SourceAppli...	Source Application	String	None	Variable Length	
UpdateDate	Update Date	Date	None	General Date	
UpdateFlag	Update Flag	Select List	Update Flag	Variable Length	None
UserName	User Name	String	None	Variable Length	

This table and properties stores the user changes and with tabular editor or reports user can view the changes.

1.In the Options Manager, make sure to set the ‘Use audit trail’ flag to ‘Yes’

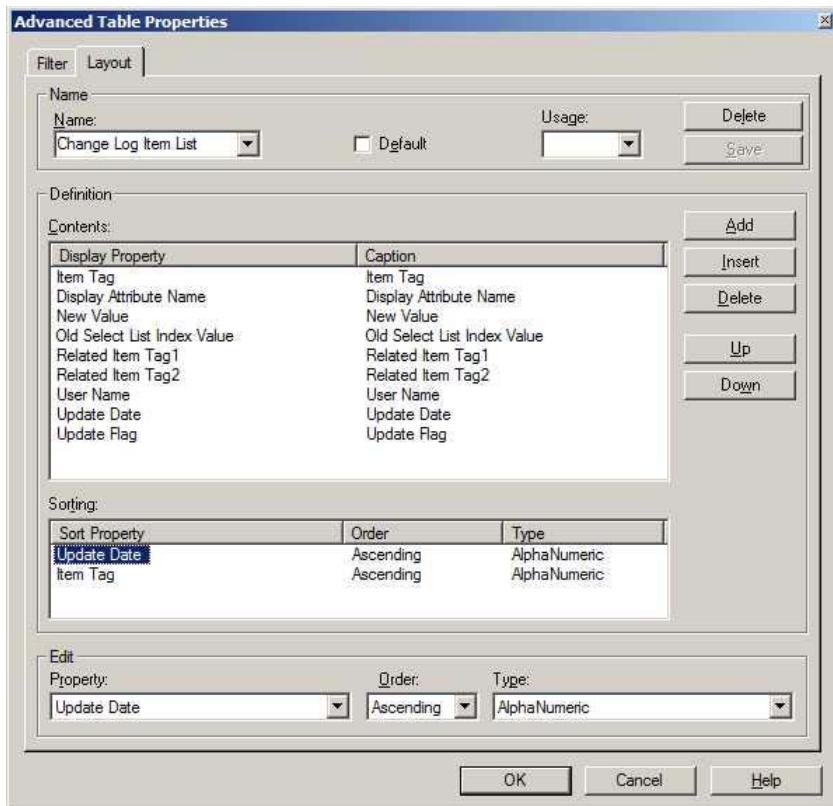


The screenshot shows the General Settings dialog box with a list of options and their current settings:

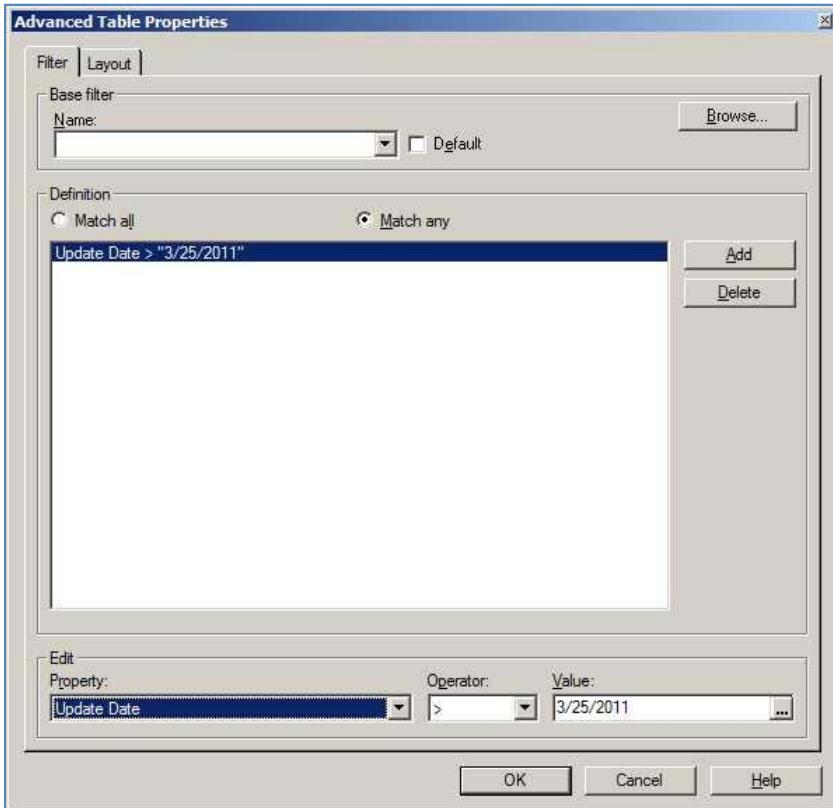
Option	Setting
Maximum number of parallel cables	6
Minimum cross-section of parallel cables	35 mm ²
Naming method for parallel cable suffix	A, B, C ...
Parallel cable suffix input mask	00
Parallel cable separator	\
Allow the routing of parallel cables via different routes.	No
Apply temperature derating factor to cable sizing	No
Duplication of PDB internals	Keep source names
SAP interface	Disable
Archive documents	Enable
Consistency rules	Enable
Consistency rules for batch operations	Cancel current operation
Display NULL when macros retrieve no data	No
External analyzing software	ETAP
Project name in external analyzing software	
Include enhanced reports in schematics	No
Default SmartPlant Instrumentation domain name	
Connect cables of control station or instrument to equipment circuit	No
Use audit trail	Yes
Allow full access to As-Built	Yes
When merging a document into As-Built, set preferences for revisions that have identical revision numbers in As-Built and project	Reject the document
Claim mode	Exclusive only
KKS mode	No
Cable fill validation to follow NEC method	No
Exclude characters from PDB row naming type	Main items and all related items
Report items to publish to SmartPlant	Row sequence and names
PDB layout vertical displays	No
Show item tag names without spaces in RAD documents	Descending
Sorting method for assigning cable drums to cables	

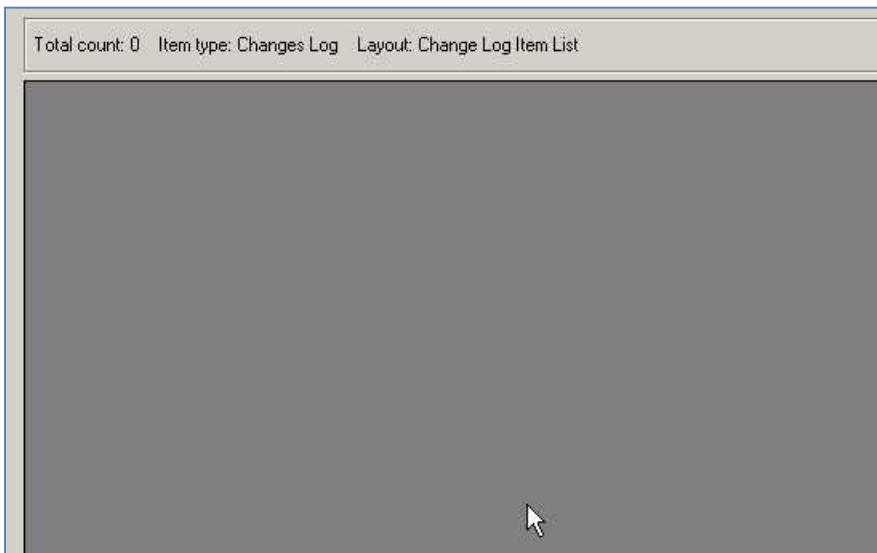
Defining Project Naming Conventions

2. Using Tabular editor, create new layout for the changes log table with the following properties.

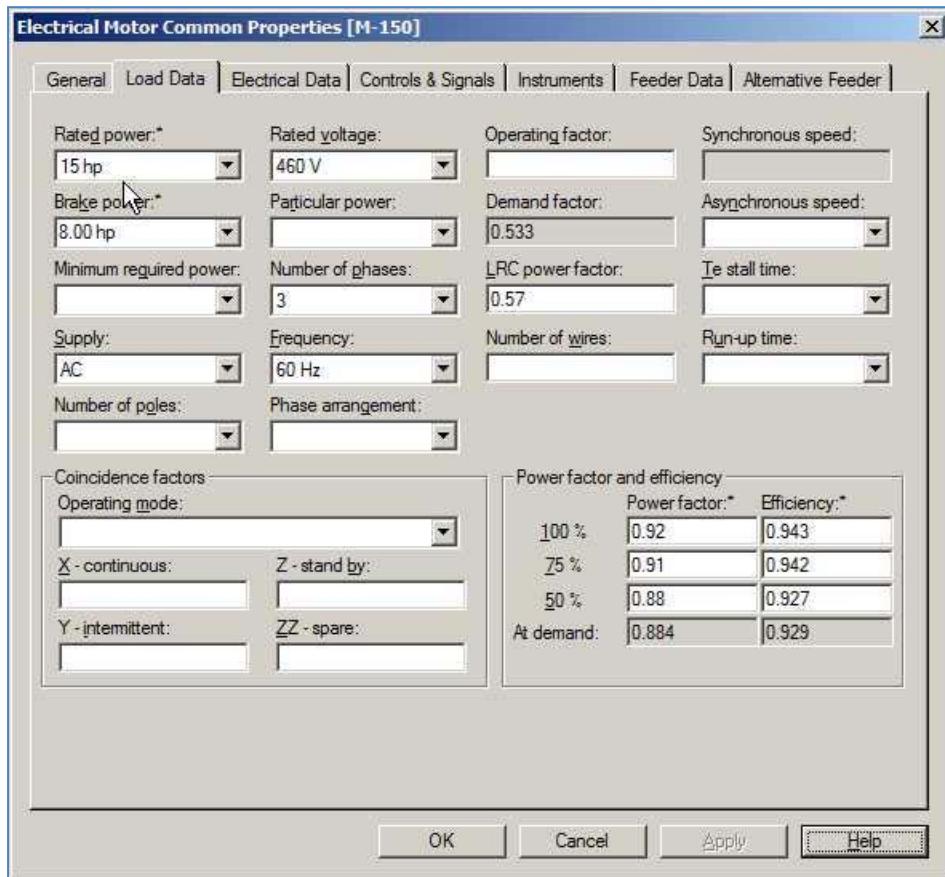


3. Add filter to show only changes occurred after certain date.





Lets do change in the motor rated power for motor, M-150 and see the change in the change log table.



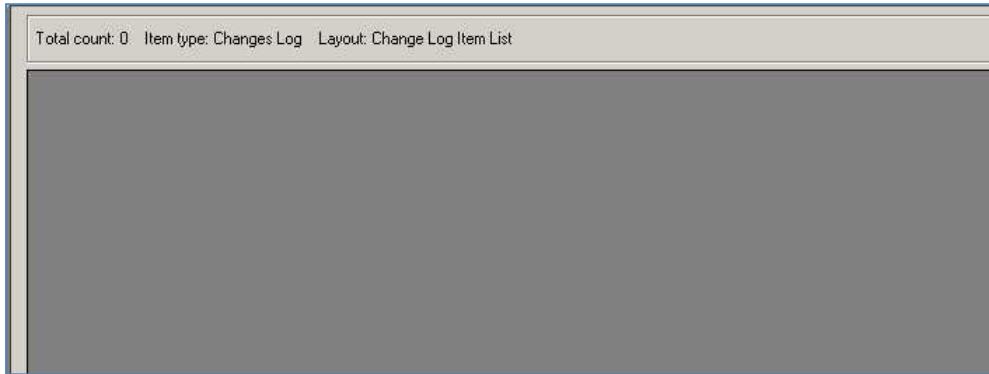
Total count: 15 Item type: Changes Log Layout: Change Log Item List									
	Item Tag	Display Attribute Name	New Value	Old Se	Relat	Related	User Name	Update Date	Update Flag
1	M-150	Consumed Reactive Lc	3.40 kVAR				yadler	3/25/2011 10:26:33 AM	Update
2	M-150	Demand Factor	0.533				yadler	3/25/2011 10:26:33 AM	Update
3	M-150	Consumed Active Load	6.42 kW				yadler	3/25/2011 10:26:33 AM	Update
4	M-150	Rated Active Load	11.86 kW				yadler	3/25/2011 10:26:33 AM	Update
5	M-150	Rated Reactive Load	5.05 kVAR				yadler	3/25/2011 10:26:33 AM	Update
6	M-150	Consumed Apparent Lc	7.26 kVA				yadler	3/25/2011 10:26:33 AM	Update
7	M-150	Full Load Current	16.2 A				yadler	3/25/2011 10:26:33 AM	Update
8	M-150	Motor Rated Power	15 hp				yadler	3/25/2011 10:26:33 AM	Update
9	M-150	Efficiency Operating	0.929				yadler	3/25/2011 10:26:33 AM	Update
10	M-150	Power Factor Operating	0.884				yadler	3/25/2011 10:26:33 AM	Update
11	PC-150B	Comply with Naming Cc	True				yadler	3/25/2011 10:26:35 AM	Update
12	PC-M-150 - 14	Item Tag	PC-M-150 -				yadler	3/25/2011 10:26:35 AM	Update
13	PC-M-150 - 14	Item Tag Complete Nar	PC-M-150 -				yadler	3/25/2011 10:26:35 AM	Update
14	M-150	Rated Apparent Load	12.89 kVA				yadler	3/25/2011 10:26:37 AM	Update
15	VFD-150	Var Freq Drive Rated P	12.89 kVA				yadler	3/25/2011 10:26:37 AM	Update

Clearing the changes log table

When the audit trail flag is set to Yes, the changes log table is being increased rapidly and need to be cleared from time to time.

To clear the table, open options manager.

From the Options menu select the ‘Clear Audit Trail Log File’ and the ‘Clear Audit Trail’ dialog box is opened where you can clear the table.



Document Revision History

- Rev. 0. Initial Release. SPEL 02.00.00.00
- Rev. 1. SPEL 03.00.00.32
- Rev. 2. SPEL 03.05.00.26
- Rev 3. SPEL 03.05.04.08
- Rev 4. SPEL 04.00.01.07
- Rev 5 SPEL 04.01.02.10
- Rev 6 SPEL 04.01.02.10
- Rev 7 SPEL 04.01.03.07
- Rev 8 SPEL 05.00.00.18
- Rev 9 SPEL 06.00.01.06 (JUN-29-2009)
- Rev 10 SPEL 06.00.02.34 (FEB-27-2010)
- Rev 11 SPEL 06.00.03.0234