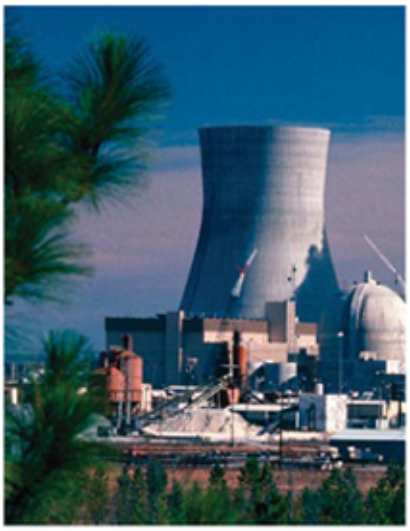
SmartPlant Electrical

Basic User's Training Guide

Process, Power & Marine


INTERGRAPH

Version 04.01.02.10 September 2007 DELE2-PE-200011A

Copyright

Restricted Rights Legend

Intergraph Corporation
Huntsville, Alabama 35894-0001

Warranties and Liabilities

Trademarks

Table of Contents

Preface

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

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

Introduction

Training Prerequisites

- An intermediate knowledge of electrical design
- Familiarity with windows-based applications
- A plant with default reference data
- Drawing symbol file CD1sh1Motor3ph1.sym

  Note: For example in chapter 11. (other symbol can be used instead)
- 'S-001 Demo SLD' drawing available

  Note: Available in Appendix A (last page of this book).

Scope

The purpose of this document is to educate the user on Smart Plant Electrical operational procedures and methods, features and functionalities.

The document contains the following sections:

- **Chapter 1** : Electrical Equipment
- **Chapter 2** : Tabular Editor
- **Chapter 3** : Power Distribution Equipment
- **Chapter 4** : Wiring Equipment
- **Chapter 5** : Propagating Default Data and Association
- **Chapter 6** : Cable sizing
- **Chapter 7** : Reports
- **Chapter 8** : Electrical Engineer
- **Chapter 9** : Single Line Diagram
- **Chapter 10** : Schematics
- **Chapter 11** : Associate External Document
- **Chapter 12** : Archive Document
- **Chapter 13** **:** Miscellaneouse Features
- **Appendix A** : 'S-001 Demo SLD' drawing.

Electricity and Electrical Engineering

Following is a brief synopsis of Electrical Engineering and Design and also some guiding principles that were employed in the creation of SmartPlant Electrical.

The Electrical Distribution Network

The electrical distribution network includes all the equipments and devices used to control and deliver electrical power safely and efficiently from the 'incoming' service to the ultimate end 'users' or 'consumers'.

Equipment

The distribution network consists of:

1.Sophisticated equipment that in some cases is designed and manufactured for particular network-specific (or plant design-specific) use.

2.Standard or 'typical' equipment that has the same characteristics from device to device or even from project to project. SmartPlant Electrical has the ability to handle both cases. One advantage the user will discover about SmartPlant Electrical is that the 'typical data' need only be entered once and then copied as needed to other similar devices. Both typical and copied items of equipment will have specific parameters defined in the course of the design and recorded on the project documents and drawings. The equipment includes (but is not limited to):

- Incoming utility feeders
- Generators
- Substations
- Switchgears
- Motor Control Centers
- Local Control Equipments
- Motors
- Heaters
- Converting Equipments
- Cables
- Low Voltage Distribution Panels
- Lighting

Electrical Engineering and Design

Involves the design of those electrical parameters and equipment that make up the distribution and utilization network. Part of the design is also involved with correctly assembling all the 'pieces' to form a cohesive and robust system (with calculated results) to fill the current needs of the client as well as providing for future expansion. This design is all handled in a framework of recognized Codes (National and Local), Standards, and Accepted Practices. Another important function of electrical design is to develop and record accurate calculations related to specific electrical and physical phenomena that the

engineer/designer uses daily. These design activities and the resulting design documents are ultimately used to:

- Provide safety and adherence to codes, client requirements, and so forth.
- Provide the contractor with direction and materials related needs during construction.
- Provide documentation to the particular facility's 'technical libraries' for future reference, and so forth.

Documentation

These engineering and design activities are presented to the facility and to the contractor in a variety of documents. Some of these documents will only be used during construction, however, some will form a part of the facility's technical library. Some of these documents will include (but not be limited to):

- Single Line Diagrams
- Motor schematic Diagrams
- Cable block diagrams
- Wiring diagrams
- Cable Schedules reports
- Cable Take off reports
- Cableway segment schedule reports
- Cableway component material take off reports
- Segment fill reports
- Drum composition reports
- Terminal strip schedule reports
- Material Take-offs and Reports
- Equipment List reports
- Electrical Load List reports

Workflow

SmartPlant Electrical does not force a particular workflow, and there are several ways to perform the required design. SmartPlant Electrical has been designed to create and enter data or entities in a series of stand-alone operations. These operations can be performed in sequences that suit the users needs. Some of the major issues with which SmartPlant Electrical development was concerned were:

1. The very real assumption that current contracts are short in the **Electrical Index** duration and that accurate and efficient tools are required to maintain the data involved.

2. The design process is never a 'one time through' operation and that changes do occur at any point during the life cycle of the design process.

3. SmartPlant Electrical users need a mechanism whereby they can set up 'default' information once and re-use it during the course of the project and in future projects as well.

Smart Plant Electrical Modules

EI – Electrical Index

The **Electrical Index** is the module where you create and store project-specific equipment and data. Logging into a SmartPlant Electrical project or plant will grant you accessing your tags and relations as per your granted access rights set by your Administrator. The **Electrical Index** will present the tags that belong to the entire plant or specifically the ones of the lowest plant group (for example, Unit), based on your settings.

To view all plant items, on the **View** menu, select the option **Show Items of All Plant Groups** (when selected, a check mark appears beside the option as shown), otherwise, to see only those items that belong to the current unit, deselect this option.

EE – Electrical Engineer

The **Electrical Engineer** is the module where graphical representations of Equipment and Associations are stored.

RDE – Reference Data Explorer

The **RDE** is the module where you create and store reference (or generic) equipment and data. This equipment and data can later be used as seed data to populate project-specific electrical equipment in the **Electrical Index**. The more that is invested in properly preparing the 'generic data' or 'defaults' the greater the savings will be during the course of the current project and on future projects.

Chapter 1

Electrical Equipment

Power Sources

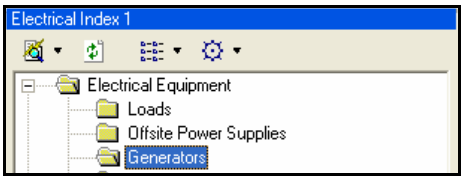
Power sources include generators, battery banks, and offsite power supplies.

Create a project Generator in Electrical Index (EI)

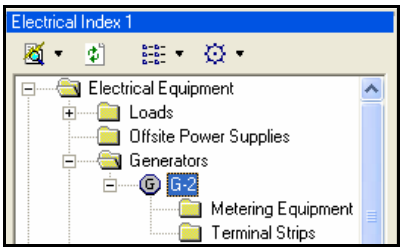
You create project items in **Electrical Index** (**EI**) window.

Use the following procedure to create a new project generator 'G-1'.

In the Electrical Index, click Electrical Equipment, Generators.

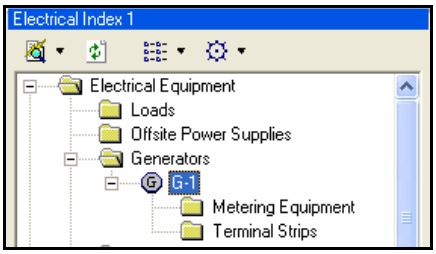


1.Right-click and on the shortcut menu, click **New Generator**.



2.Right-click, and from the shortcut menu, click **Rename**.

3.On the **Item Tag** dialog box, rename the generator to **G-1**.



Tip

To display the Item Tag dialog box each time you create a new item, click File, Preferences, and on the Preferences dialog box,general tab, under Open after creating or duplicating an item, check the Item Tag dialog box radio button.

Duplicate an Item

Use the following procedure to duplicate Generator G-1.

1.In the **Electrical Index**, select the Generator G-1

2.Right Click, and from the shortcut menu, select **Duplicate**.

3.On the **Item Tag** dialog box, type **G-2**.

Delete an Item

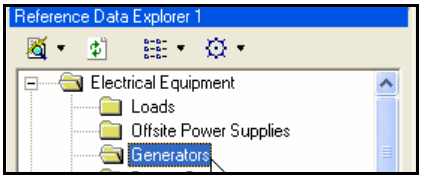
Use the following procedure to delete Generator G-2.

1.In the **Electrical Index**, select the Generator **G-2**.
2.Right-Click, and from the shortcut menu, select **Delete**.
3.On the **Delete Confirm** prompt, select **Yes**.

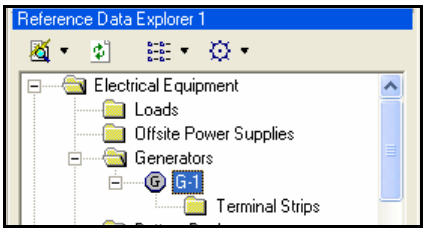
Create typical Generator in Reference Data Explorer (RDE)

You create typical items in **Reference Data Explorer** (**RDE**) window.
You use those typicals to create the project items in the **Electrical Index** (which contains the project specific items).

1.In the Reference Data Explorer, click Electrical Equipment, Generators.

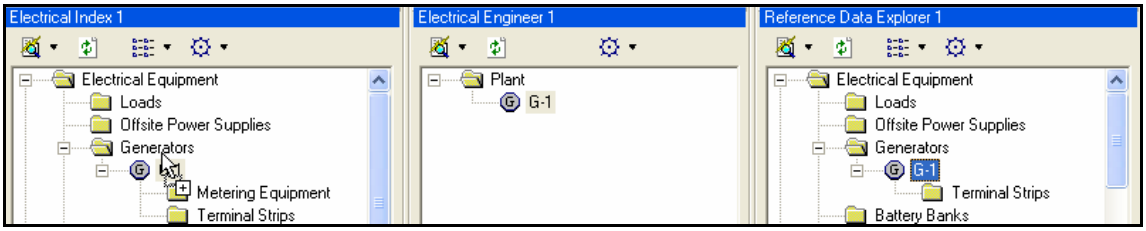


2.Right-click and from the shortcut menu, click **New Generator**.
3.Right-click, and from the shortcut menu, click **Rename**.
4.On the **Item Tag** dialog box, rename the generator to **G-1**.

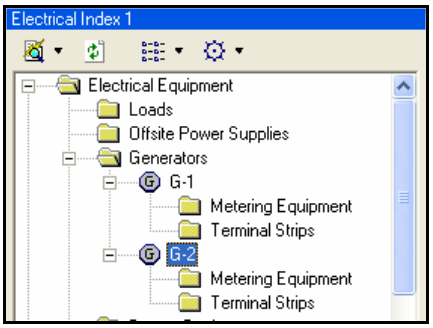


Create a project Generator in Electrical Index from Typical

1. In the **RDE**, select generator **G-1**.
2. Drag **G-1** onto the **Generators** folders in the **Electrical Index**.
3. A "⊞" sign appears when dragged cursor over the **Generators** folder in **Electrical Index**.



4.Release the mouse to create the generator.
5.Rename the generator to **G-2**.



Create an Offsite Power Supply

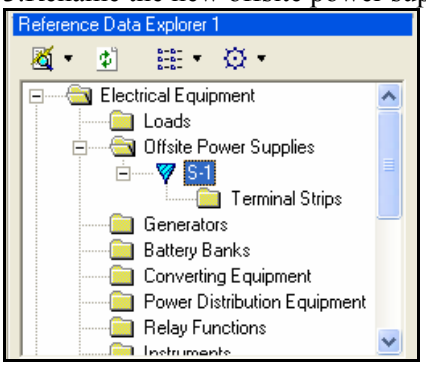
Create typical offsite power supply in RDE:

Use the following procedure to create a new typical Offsite Power Supply :

1.In RDE, select Electrical Equipment, Offsite Power Supplies

2.Right click on the Offsite Power Supply folder and select New Offsite Power Supply.

3.Rename the new offsite power supply and call it to '**S-1**'

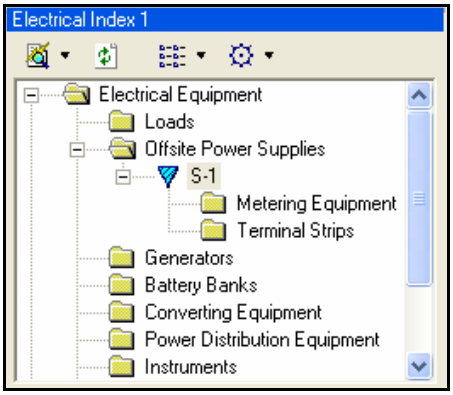


Create project Offsite Power Supply in EI:

Use the following procedure to create a new project Offsite Power Supply:

1.Select the typical Offsite Power Supply, 'S-1' from RDE, and copy it to EI.

2.Rename the new offsite power supply and call it '**S-1**'



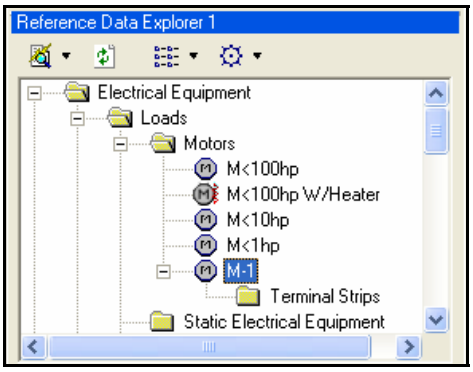
Loads

Loads include motors, heaters, heat traces, capacitors, harmonic filters, resistors, lighting fixtures, socket outlet, welding outlet, other electrical equipment.

Create Motor

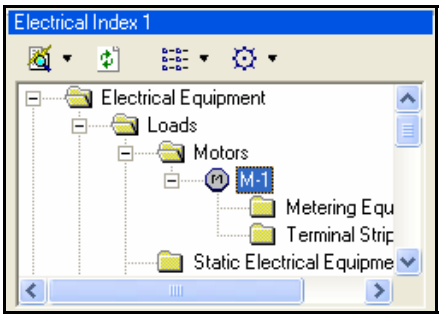
Create typical Motor in RDE:

Use the following procedure to create a new typical Motor:
1.In the RDE, select Electrical Equipment, Load, Motors.
2.Right click on the Motors folder and select New Motor.
3.Rename the new motor and call it '**M-1**'



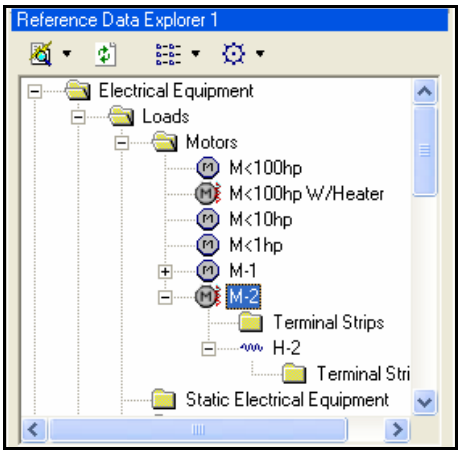
Create project Motor in EI:

Use the following procedure to create a new project Motor:
1.Select the typical motor 'M-1' from RDE and copy it to EI.
2.Rename the new motor and call it '**M-1**'.



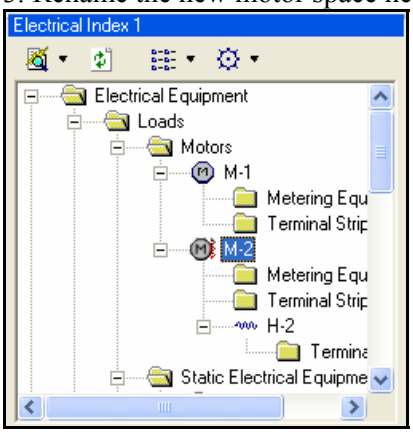
Create typical Motor with space heater in RDE:

Use the following procedure to create a new typical Motor with space heater:
1.In the RDE, select Electrical Equipment, Load, Motors.
2.Right click on the Motors folder and select New Motor with Heater.
3.Rename the new motor and call it 'M-2'
4. Rename the new motor space heater to 'H-2'



Create project Motor with space heater in EI:

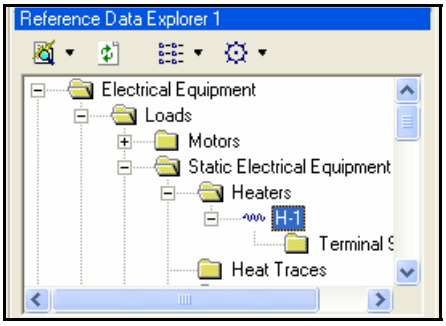
Use the following procedure to create a new project Motor with space heater:
1.Select the typical motor with space heater M-2' from RDE and copy it to EI.
2.Rename the new motor and call it '**M-2**'.
3. Rename the new motor space heater and and call it '**H-2**'



Create Heaters

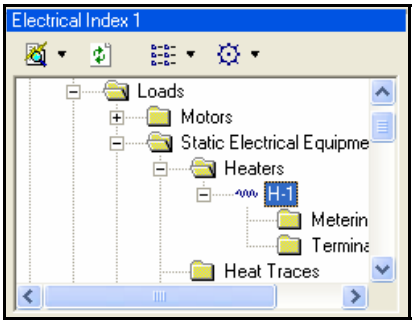
Create typical Heater in RDE:

Use the following procedure to create a new typical Heater:
1.In RDE, select Electrical Equipment, Load, Static Electrical Equipment,Heater.
2.Right click on the Heaters folder and select New Heater.
3.Rename the new heater and call it '**H-1**'



Create project Heater in EI:

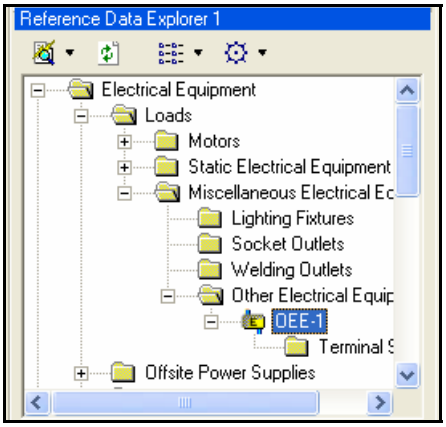
Use the following procedure to create a new project Heater:
1.Select the typical heater 'H-1' from RDE and copy it to EI.
2.Rename the new heater and call it '**H-1**'.



Create Other Electrical Equipment

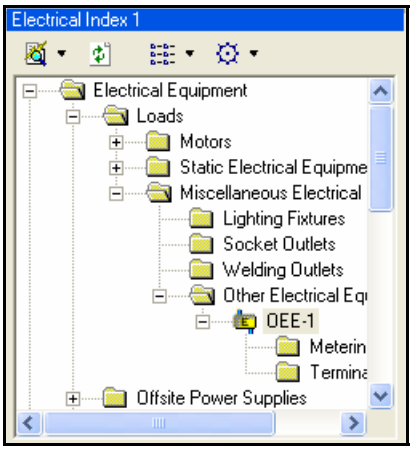
Create typical Other Electrical Equipment in RDE:

Use the following procedure to create a new typical Other Electrical Equipment:
1.In RDE, select Electrical Equipment, Load, Miscellaneous Electrical Equipment,Other Electrical Equipment
2.Right click on the Other Electrical Equipment folder and select New Other Electrical Equipment
3.Rename the new Other Electrical Equipment and call it '**OEE-1**'



Create project Other Electrical Equipment in EI:

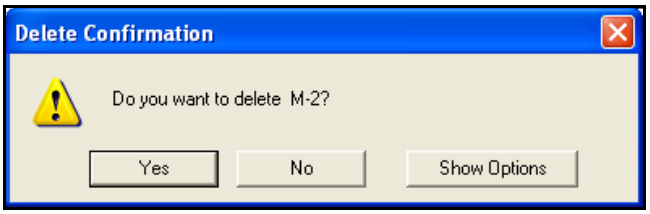
Use the following procedure to create a new project Other Electrical Equipment:
1.Select the typical Other Electrical Equipment, 'OEE-1' from RDE and copy it to EI.
2.Rename the new Other Electrical Equipment, and call it 'OEE-1'.



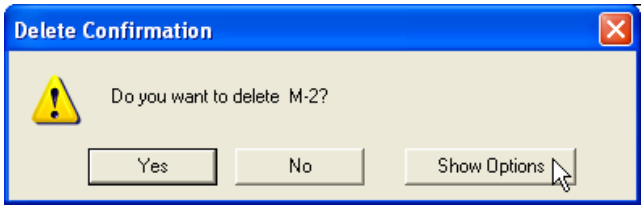
Delete Load with associated items

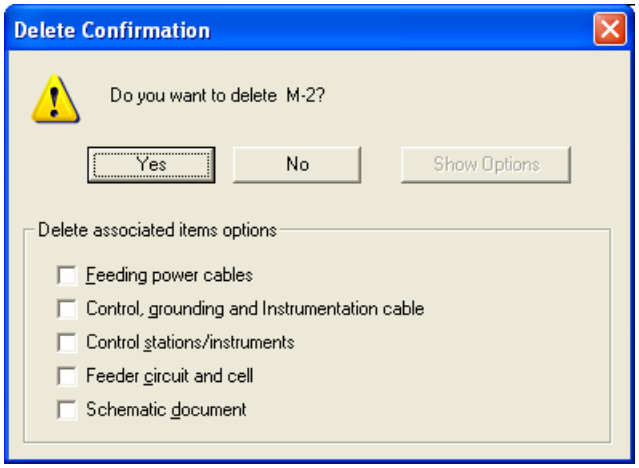
When deleting a load, the software lets you delete various items associated with the load.

Use the following procedure to delete project motor M-2 with associated items from EI:
1.From EI select the motor M-2.
2.Right click and select delete.
3.You will get the following delete confirmation window:

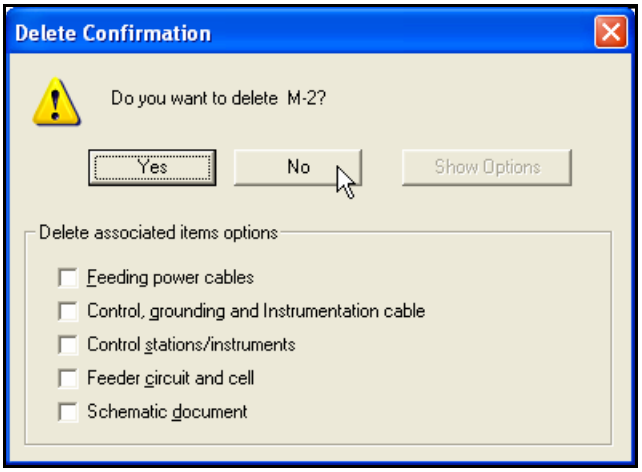


Selecting the show options button opens the delete associated items items box , where you can select the associated items you would like to delete with the load.

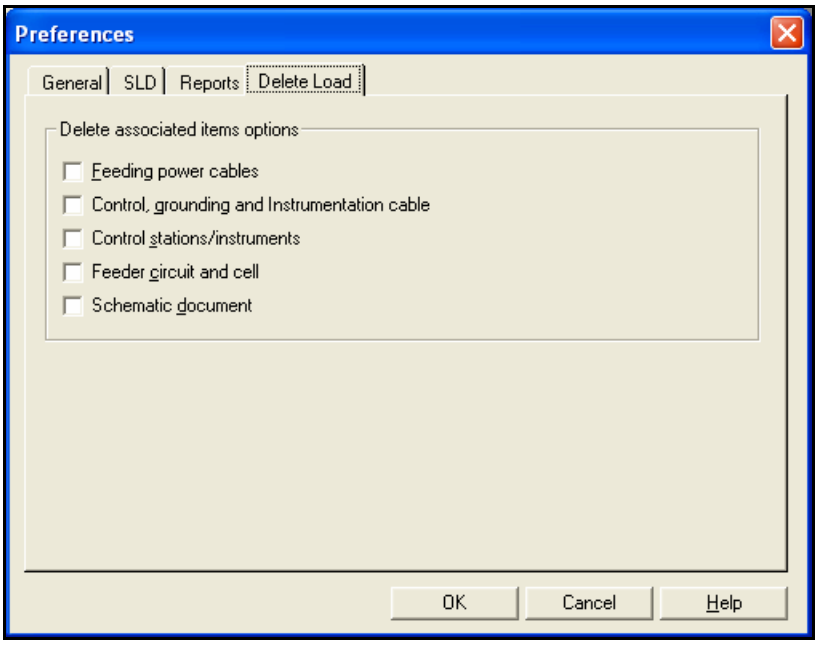




Select Yes to delete M-2 with any checked associated item.
Select No to cancel the deletion.



Note: You can set a preference that instructs the software the associated items to be deleted.

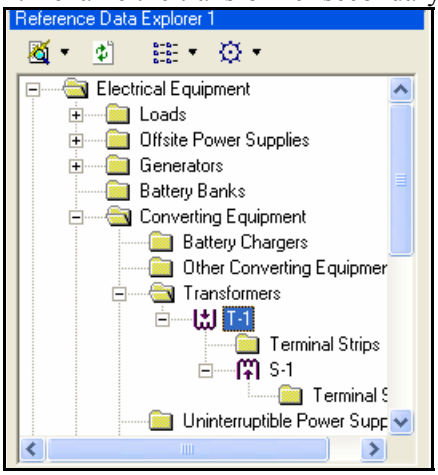


Converting Equipment

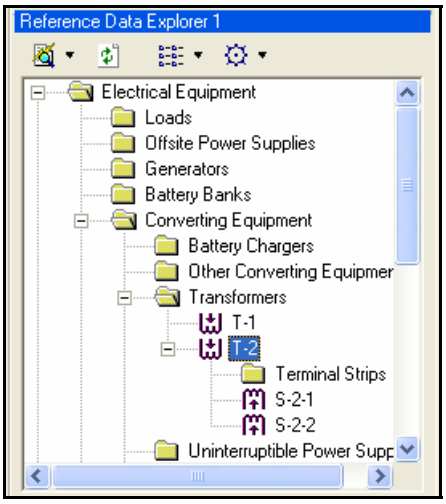
Create a Transformer

Create typical transformers in RDE:

Use the following procedure to create a new typical 2-Winding Transformer:
1.In RDE, select Electrical Equipment, Converting Equipment,Transformers
2.Right click on the Transformers folder and select New 2-Winding Transformer
3.Rename the new 2-Winding Transformer and call it 'T-1'
4. Rename the transformer secondary and call it 'S-1'

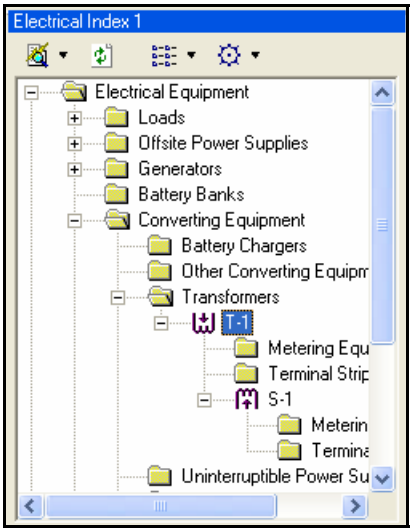


Use the following procedure to create a new typical 3-Winding Transformer:
1.In RDE, select Electrical Equipment, Converting Equipment,Transformers
2.Right click on the Transformers folder and select New 3-Winding Transformer
3.Rename the new 3-Winding Transformer and call it '**T-2**'
4. Rename the transformer secondaries and call them '**S-2-1**' and '**S-2-2**'

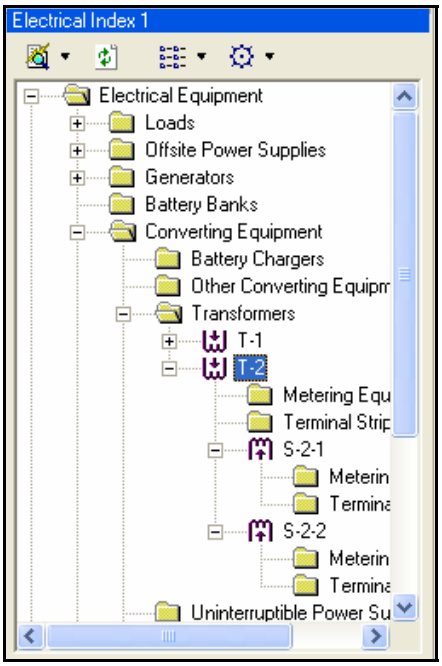


Create project Transformer in EI:

Use the following procedure to create a new project 2-Winding Transformer:
1.Select the typical 2-Winding Transformer, 'T-1' from RDE and copy it to EI.
2.Rename the new 2-Winding Transformer, and call it '**T-1**'.
3. Rename the transformer secondary and call it '**S-1**'



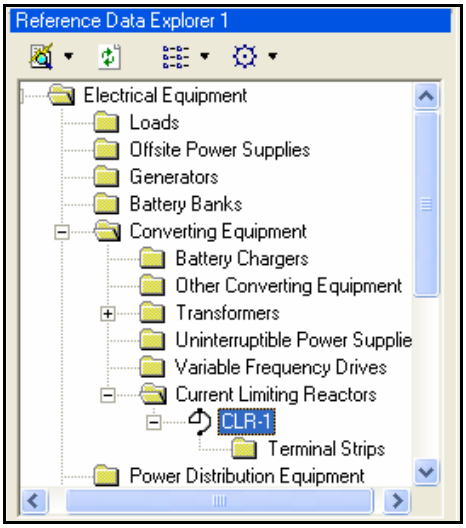
Use the following procedure to create a new project 3-Winding Transformer:
1.Select the typical 3-Winding Transformer, 'T-2' from RDE and copy it to EI.
2.Rename the new 3-Winding Transformer, and call it '**T-2**'.
3. Rename the transformer secondaries and call them '**S-2-1**' and '**S-2-2**'



Create a Current Limiting Reactor

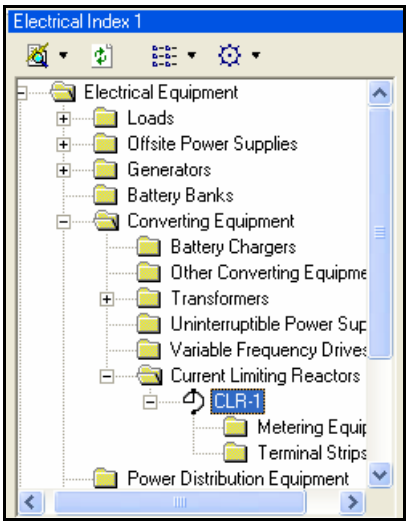
Create typical Current Limiting Reactor in RDE:

Use the following procedure to create a Current Limiting Reactor:
1.In RDE, select Electrical Equipment, Converting Equipment,Current Limiting Reactor
2.Right click on the Current Limiting Reactor folder and select New Current Limiting Reactor
3.Rename the new Current Limiting Reactor and call it '**CLR-1'**



Create project Current Limiting Reactor in EI:

Use the following procedure to create a new project Current Limiting Reactor:
1.Select the typical Current Limiting Reactor, 'CLR-1' from RDE and copy it to EI.
2.Rename the new Current Limiting Reactor, and call it '**CLR-1**'.



Common Features

 **Find** - Allows you to search for a specific item based on user-defined search criteria. The items that are available depend on the object from which you invoke the Find command.

 **Refresh** - Update the Electrical Index,Electrical Engineer,Reference Data Explorer, or Reference Electrical Engineer display. This feature is useful where multiple users are working on the same set of data.

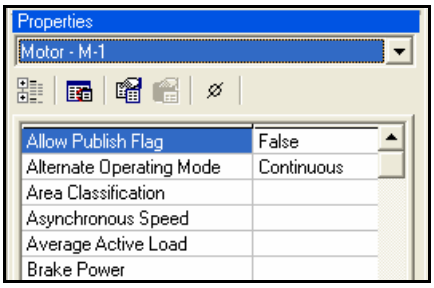
 **Views** - Allows you to toggle through different views of the items in the list view pane. The arrow beside the icon allows you to select a specific view.

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

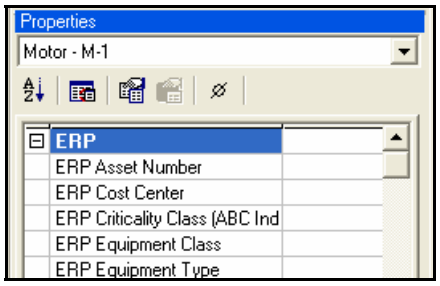
Properties Window (Property Grid)

The Properties window display is a two-column table that provides the common properties of the current item tag in the active window (Electrical Index, Electrical Engineer, Reference Data Explorer, Reference Electrical Engineer) or Tabular Editor.
You can display properties alphabetically or by categories by clicking the corresponding display mode button.

Click  to display alphabetically.



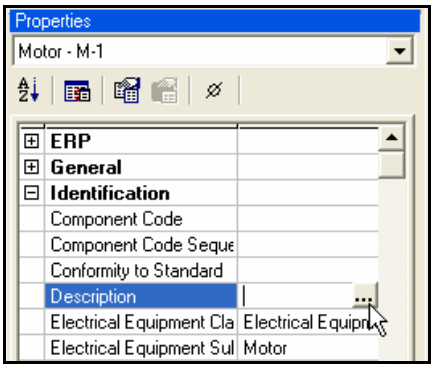
Click  to display by categories.

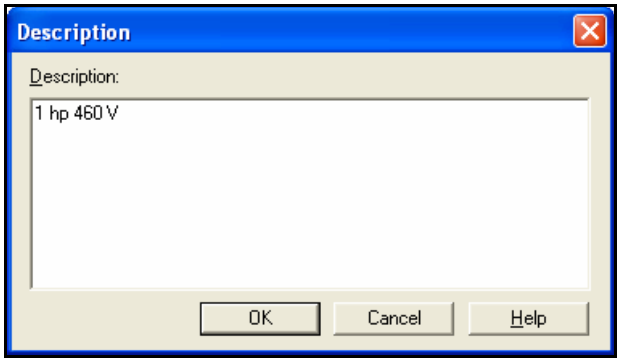


Data Types

Char data type

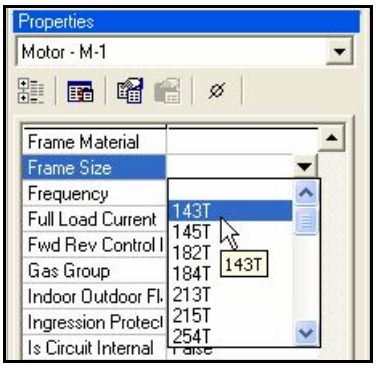
Character data type.

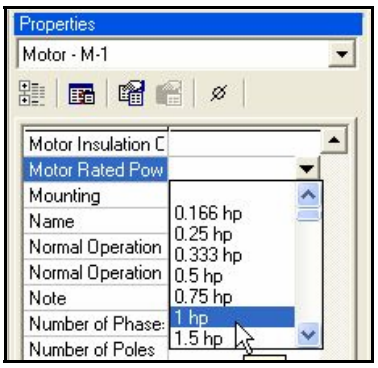


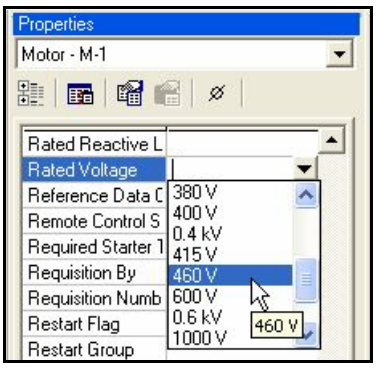


Select Lists data type

Select list data type.
You can hide values in a select list, but you can not delete them completely from the database.
You can rename values in a select list.
You can create additional values in a select list.





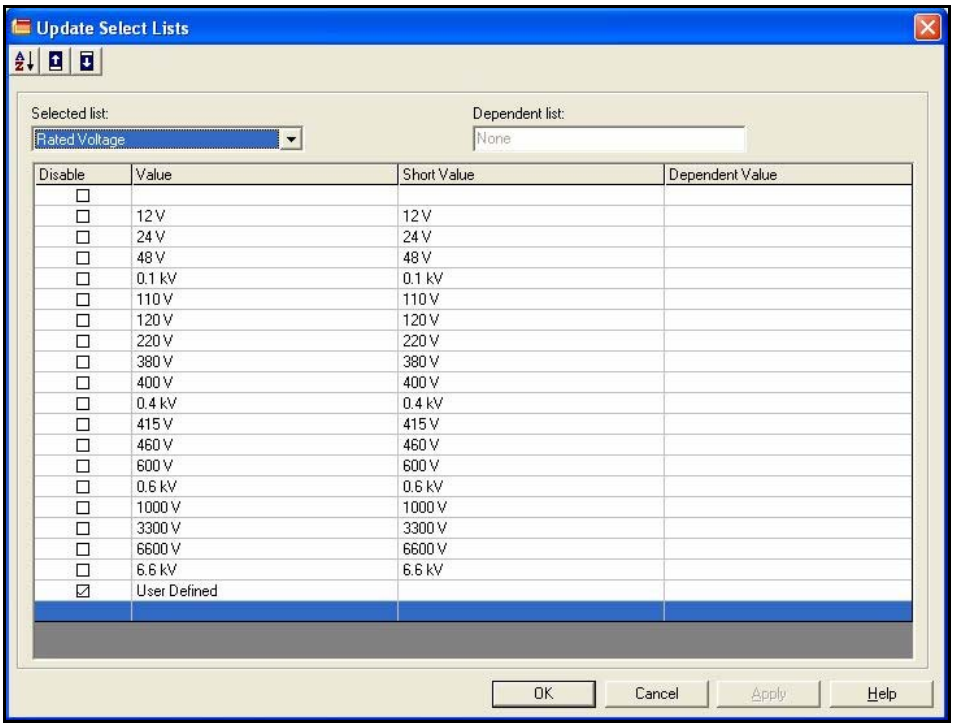


Adding values to select list

Notice that only users with access rights to the data dictionary can modify a select list.

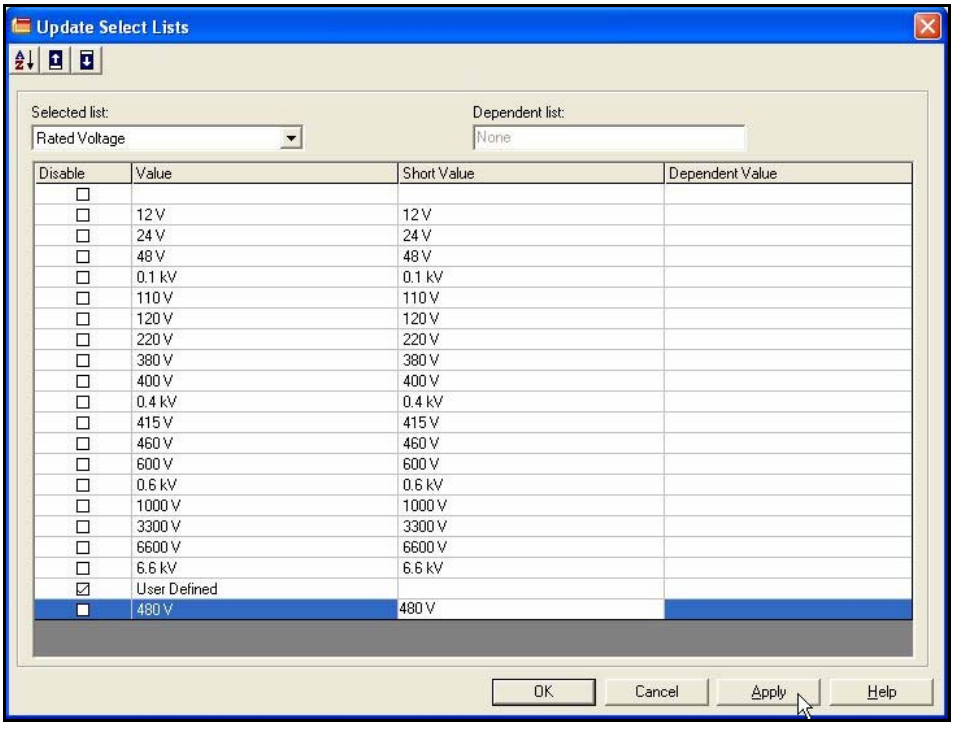
To add **480 V** and **4.16 kV** values to Rated Voltage Select list, select 'Update Select List' from Tools menu.

The Update Select Lists window opens where you can modify values or add new values.

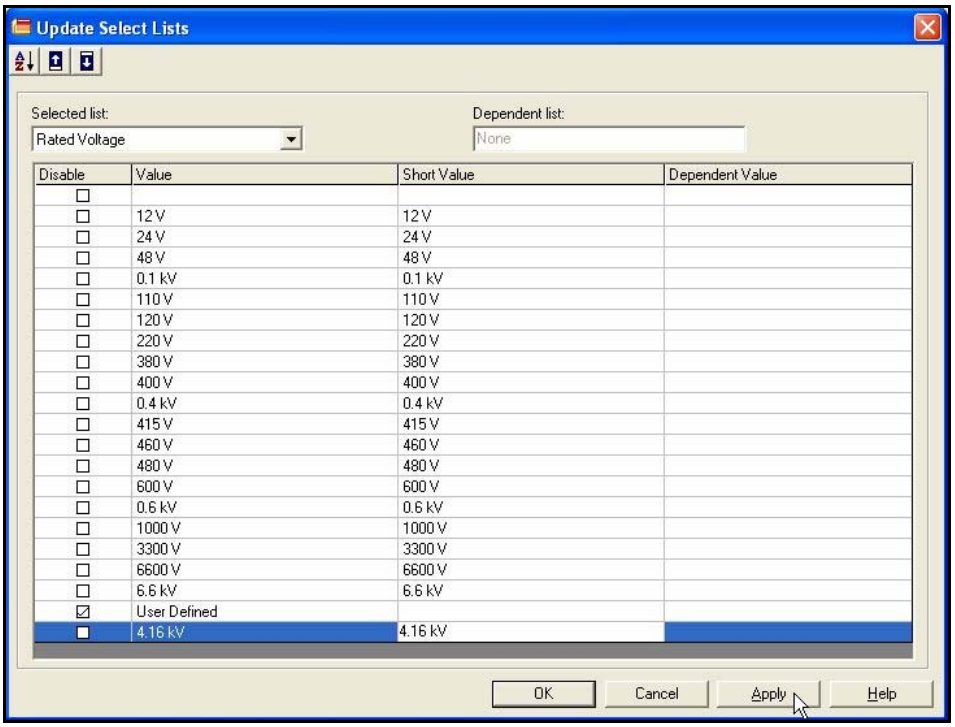


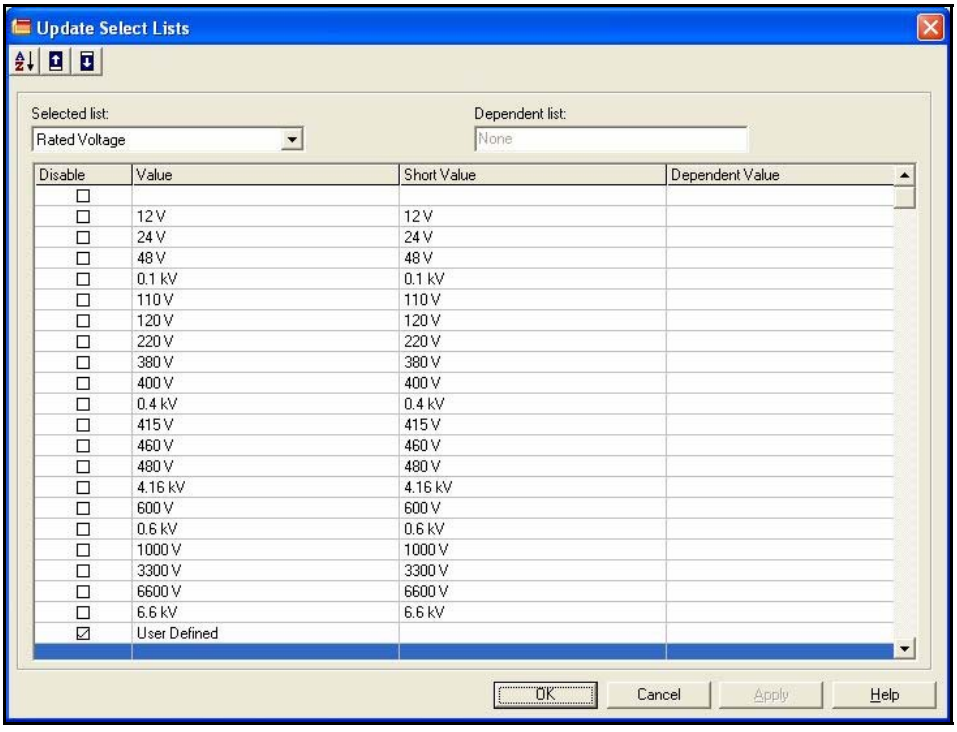
.

Type the new value **480 V** in the last row right after the disabled value called User Defined and select Apply



Do the same for the new value 4.16 kV

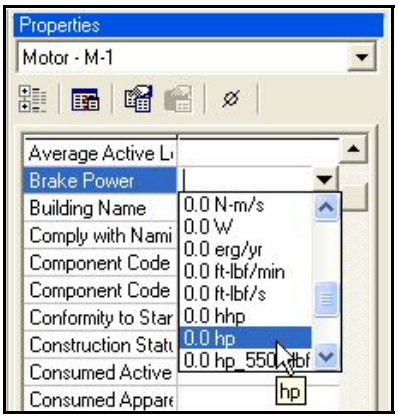


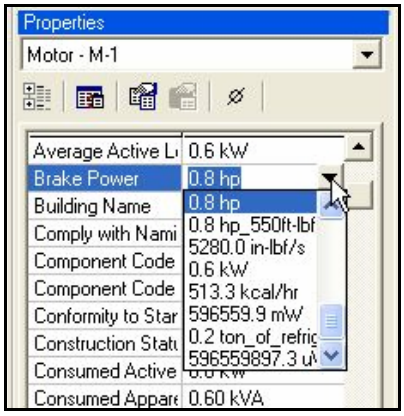


Select OK to close the Update Select Lists window.

Units of Measurement data type

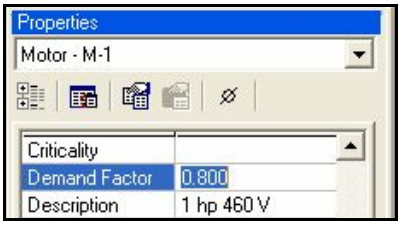
The 'Units of Measurement' select list is different from other select lists. The user cannot modify the values because they have associated conversions and calculations.
When the user selects one of these fields, such as 0.0 hp, 0.0 hp will appear in the window and the user can then type in the required value.
The default unit of measure for a particular property is set in the Data Dictionary.





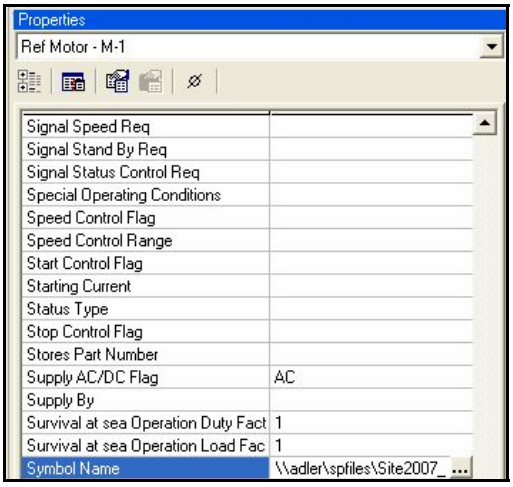
Calculated value data type

Displays a read-only value that the software calculates.



Refernce file path data type

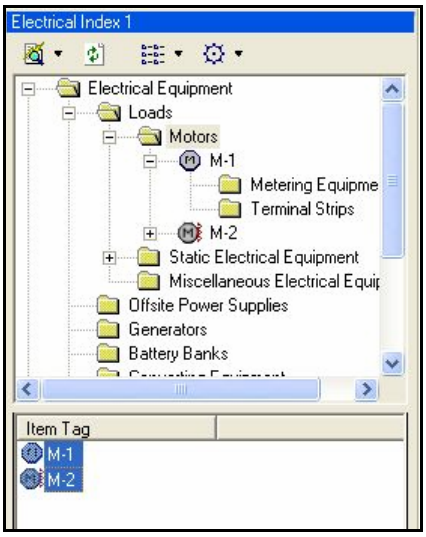
Allows you to set a reference file path.



Note: To define the symbol path for item in EI, you use the Associate Custom Symbol option.

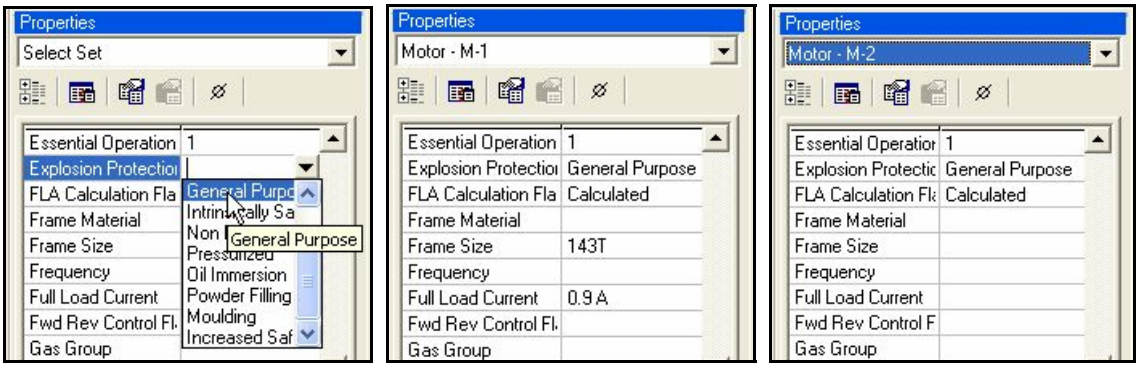
Select Set

Applies changes to sets of Items
Notice that select set is a temporary set and will not be saved after user moves to another action.
Select the Motors folder in the **Electrical Index** or the **Reference Data Explorer**. And from the bottom pane select the motors M-1, and M-2.



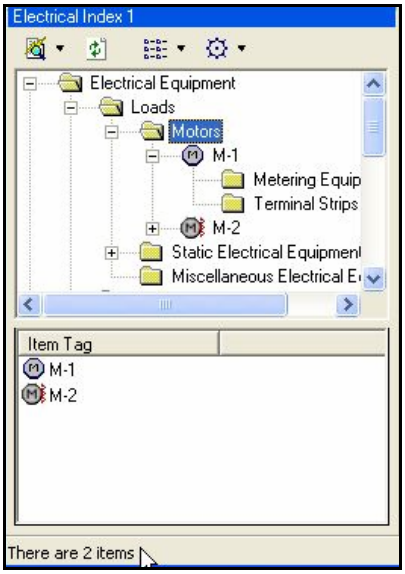
From Properties grid window pick the Select Set.
Note: all the motors that were picked in the bottom pane are also listed in the properties screen.

Whatever change is made to one property (with Select Set selected) it will apply to all items that are included in the Select Set.
When a select set is chosen, a value will appear only if all the items share the same value.



Item counter in the Electrical Index or the Reference Data Explorer

The counter of the selected items exists in the bottom left part of the SmartPlant Electrical window.The counter displays when highlighting a folder.



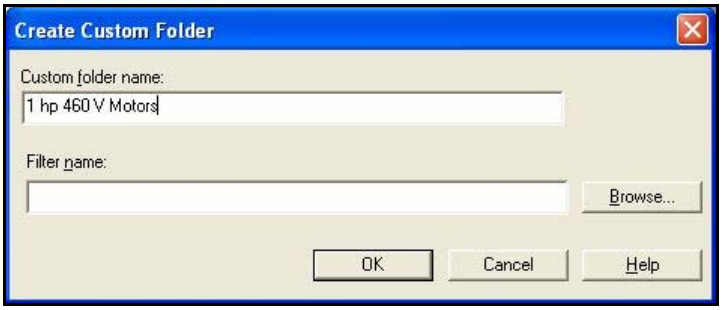
Create sub-folders

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

Create sub-folder for 1 hp 460 V project Motors in EI

1. From Electrical Index select Motor folder

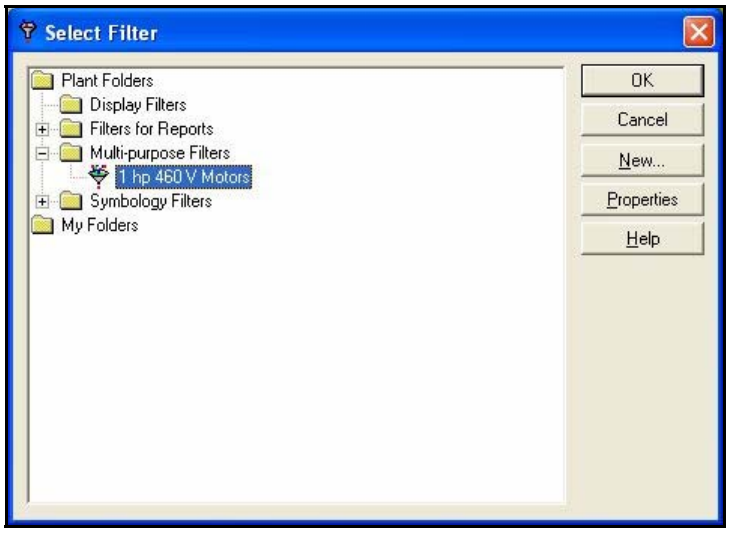
2. Right click and select New Custom Folder and name it '**1 hp 460 V Motors**'



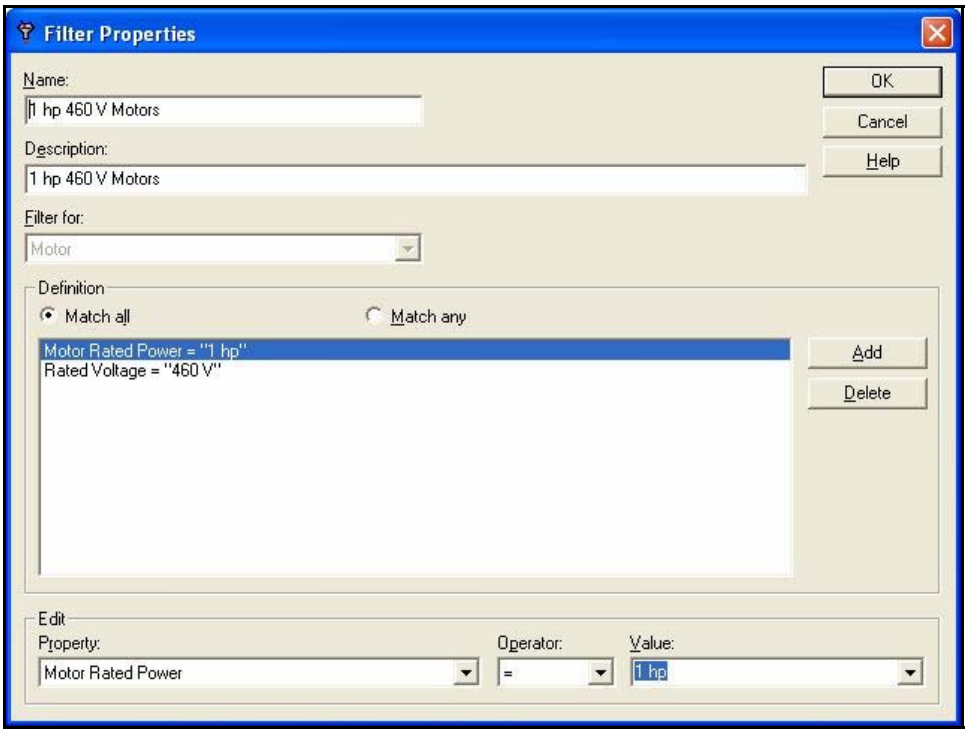
Select the Broser button to open select filter windows.
Select the Multi-purpose Filters folder and create new simple filter, name it **1 hp 460 V Motors**.

Notes: 1. Filter created under Plant folders available to all users, filter created under my folders available only to creator.

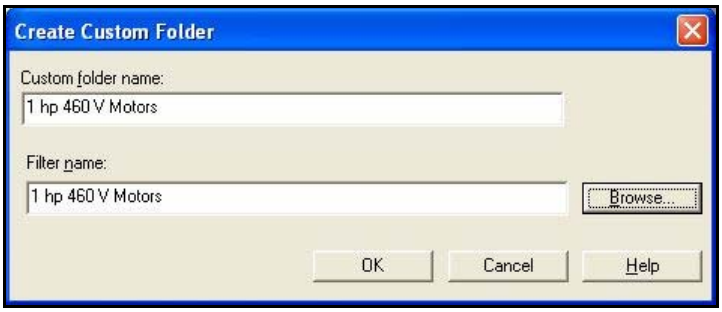
2. A compound filter consists of more than one simple filter. The simple filters are added to the compound filter by dragging the simple filter or by creating new simple filters under the compound filter in the filter hierarchy. Compound filters apply only to homogeneous item types.



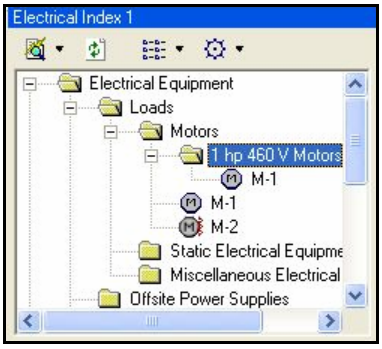
Set in the Filter Properties windows, Motor Rated Power = 1 hp, and Rated Voltage = 460 V
Set the match all/match any radio button to match all, to get only motors match both
1 hp and 460 V.



Select OK to close the Filter Properties window, and OK again to close the select Filter window
And return to create custom folder window.



The result in Electrical Index will be a new sub folder for the 1 hp 460 V motors.



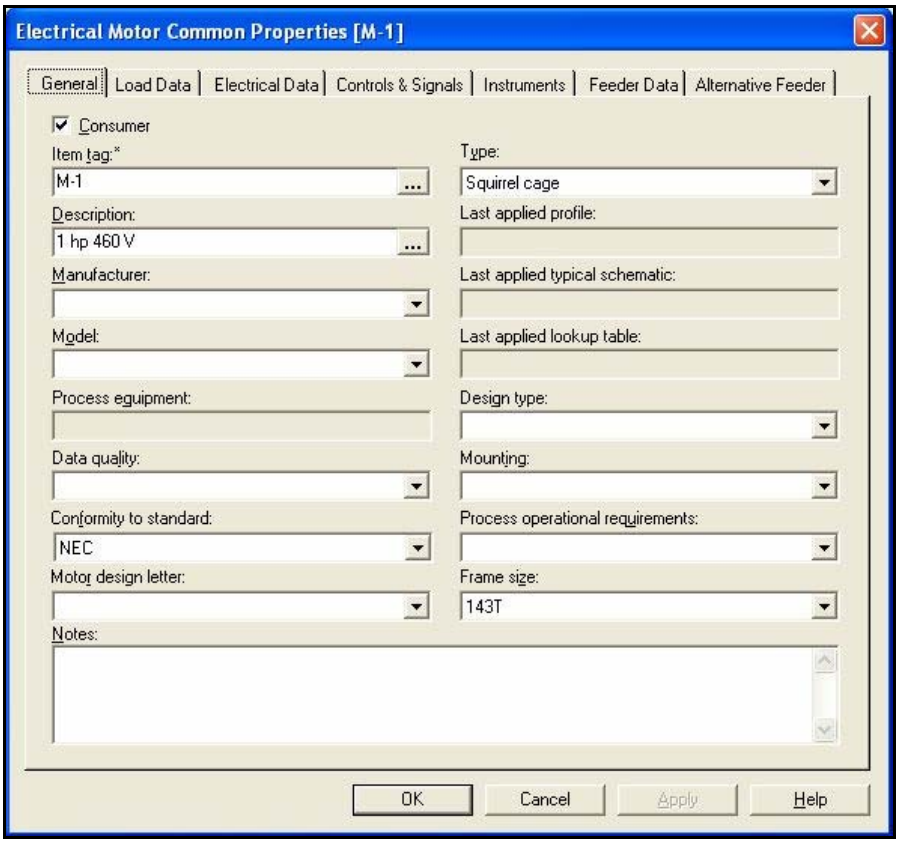
Electrical Loads (Consumers)

Entering engineering values using the Common Properties entry form.
Note: Rated power property, varies and depends on the type of electrical equipment. For all types of equipment it has units of measure type of property, while for Motors it is a select list.

Motor Form

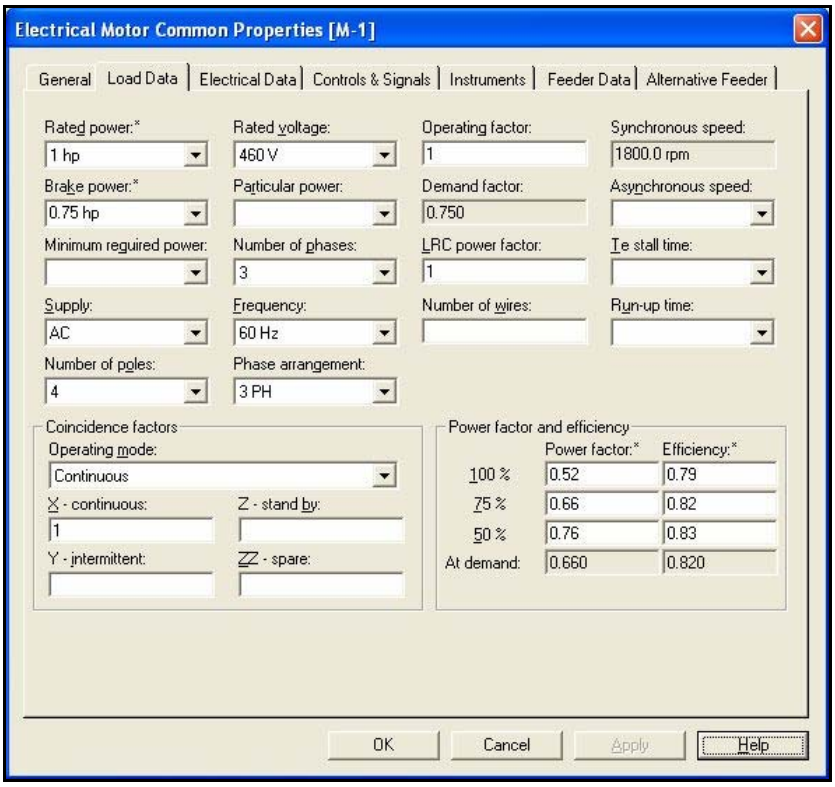
From Electrical Index, select motor M-1, Right-Click, and from the shortcut menu select **Common Properties.**
General tab

General motor properties identify and characterize the motor. An asterisk beside a property name indicates required data.



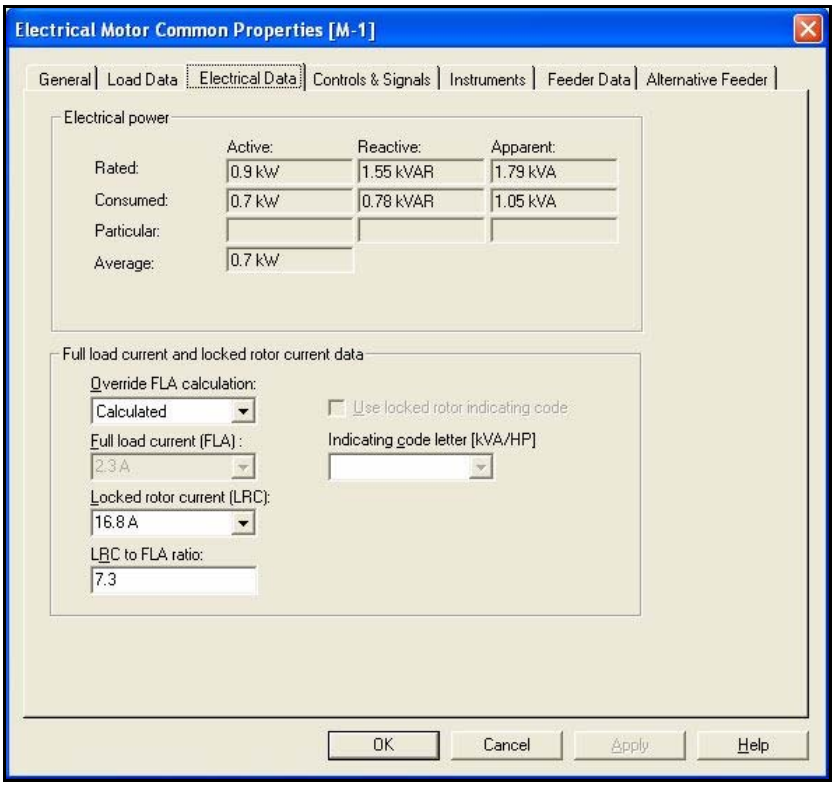
Load Data tab

Provides information about the mechanical power and usage requirements of the motor. You can find many of these quantities on the motor nameplate. An asterisk beside a property name indicates required data. To obtain the **Demand factor**, on the **Load Data** tab, you should fill in values for **Rated power** and **Brake power.** The software also validates that the brake power does not exceed the rated power value. Select a value for **Rated voltage** and let the system calculate the value for **Full load current** (this property appears on the **Electrical Data** tab). To obtain the synchronous speed, you should enter values for the **Frequency**, **Poles**, and **Supply** properties. The software validates that the asynchronous speed never exceeds the synchronous speed. Select an operating mode to enter the coincidence factor of that motor. To get the **Power factor** and **Efficiency** values at demand you need to enter the **Electrical Index** values at 50%, 75%, and 100% of operation. We will see later that there is a better way to populate these values using lookup tables.



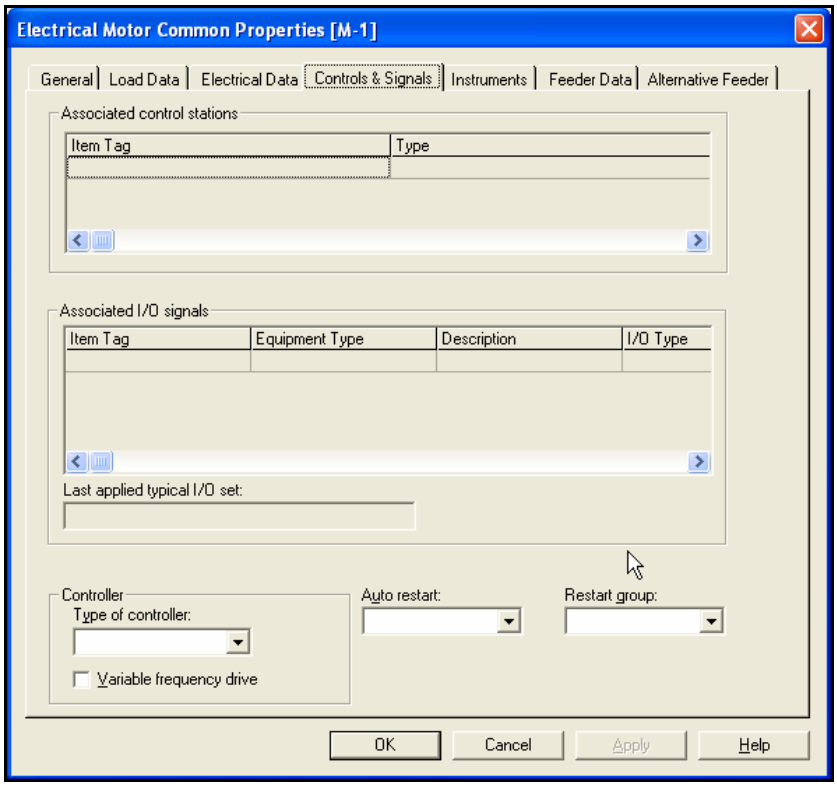
Electrical Data tab

Calculations behind these calculated fields.
Once you have entered all the previous values on the **Load Data** tab the software calculates all three vectors of the electrical consumption from the bus that feeds this motor.
The full load current can be the **Electrical Index** the calculated or entered manually, depending on the **Override FLA Calculation** flag. If it is set to **Calculate**, the values are calculated using the rated power, power factor at 100%, rated voltage, number of phases and according to whether the supply is AC or DC. Observe the relation between the FLA, LRC, and the **Electrical Index** ratio.



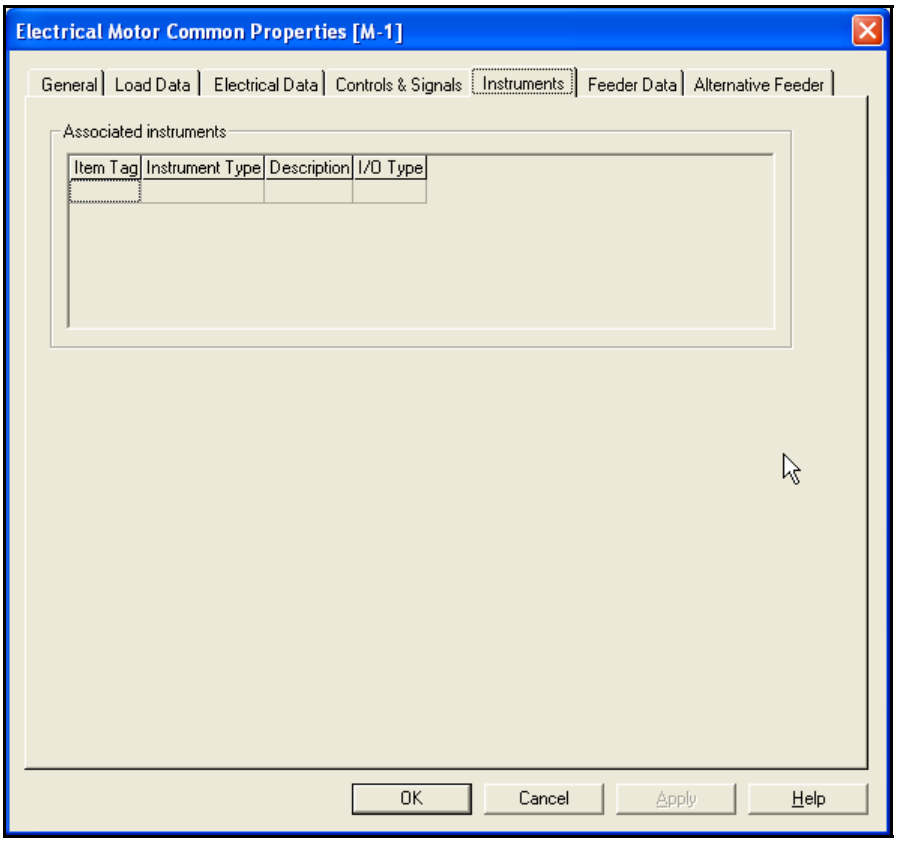
Controls & Signals tab

Provides optional information for control stations that control the motor operation, and controls and signals for the motor.



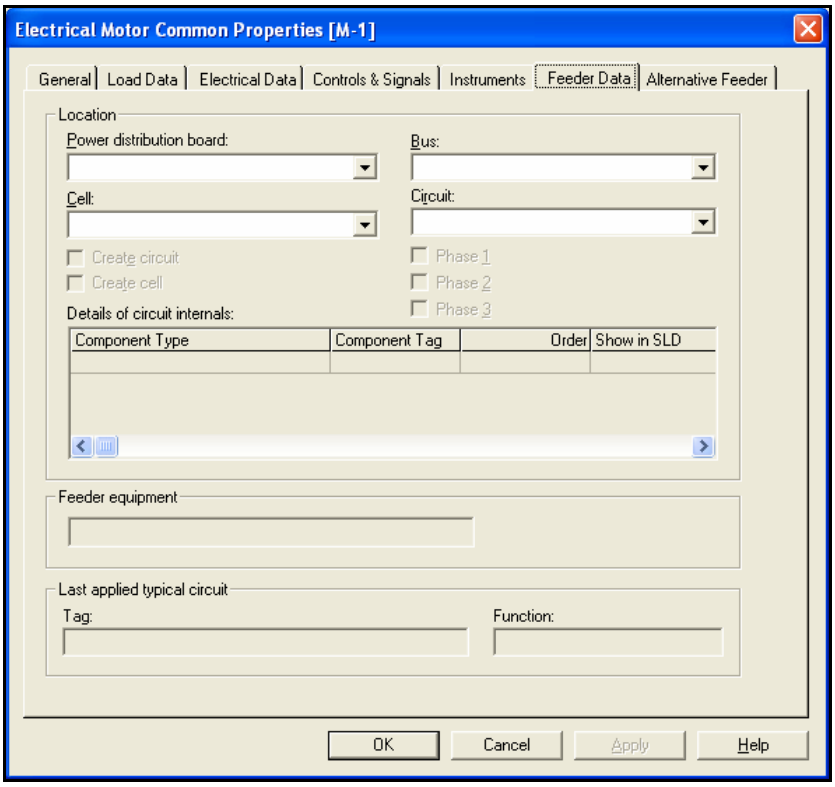
Instruments tab

Displays instruments that monitor and control the motor operation.



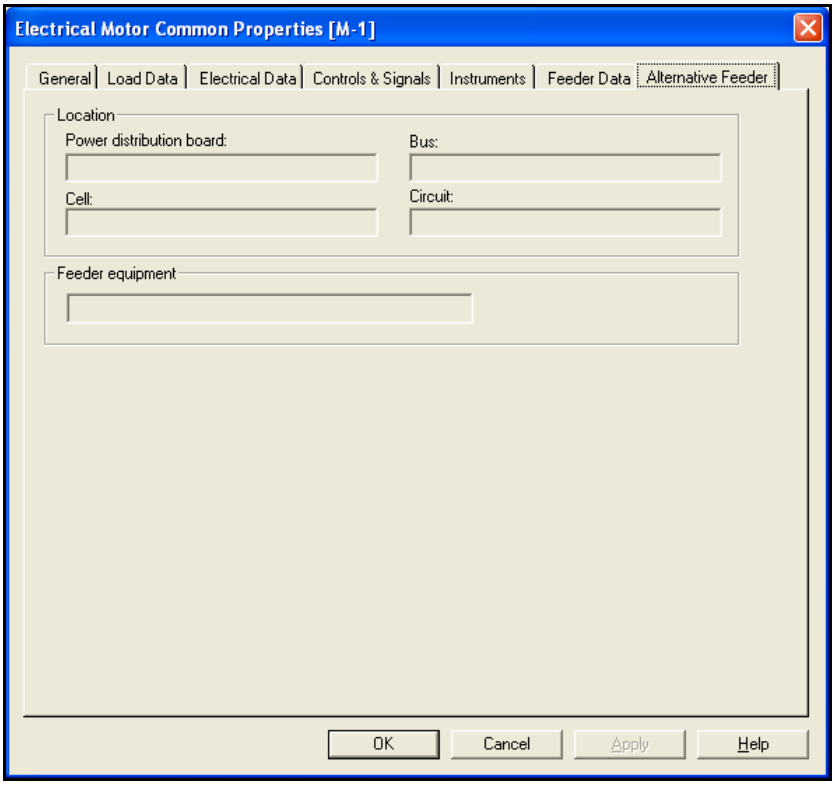
Feeder Data tab

Enables you to view and assign a feeder (provided an association to a feeder circuit does not yet exist). Once an association is made, the only way of dissociating a motor from its feeder circuit is in the Electrical Engineer or in Batch Load Association.
On this tab, you can:
Pre assign the Motor to a PDB and to a bus without actually creating feeder circuits.
Assign and Create feeder circuits on the fly, based on the last applied typical circuit.
Once the Motor is assigned to a Feeder circuit, you will see the actual components in that circuit shown in the Internals grid.
Feeder equipment, displays the name of the equipment other than feeders that the load is fed from. Such equipment can be a transformer's secondary, variable speed drive and so forth.



Alternate Feeder tab

Showing the second power source that may be feeding this motor, in case there is such:



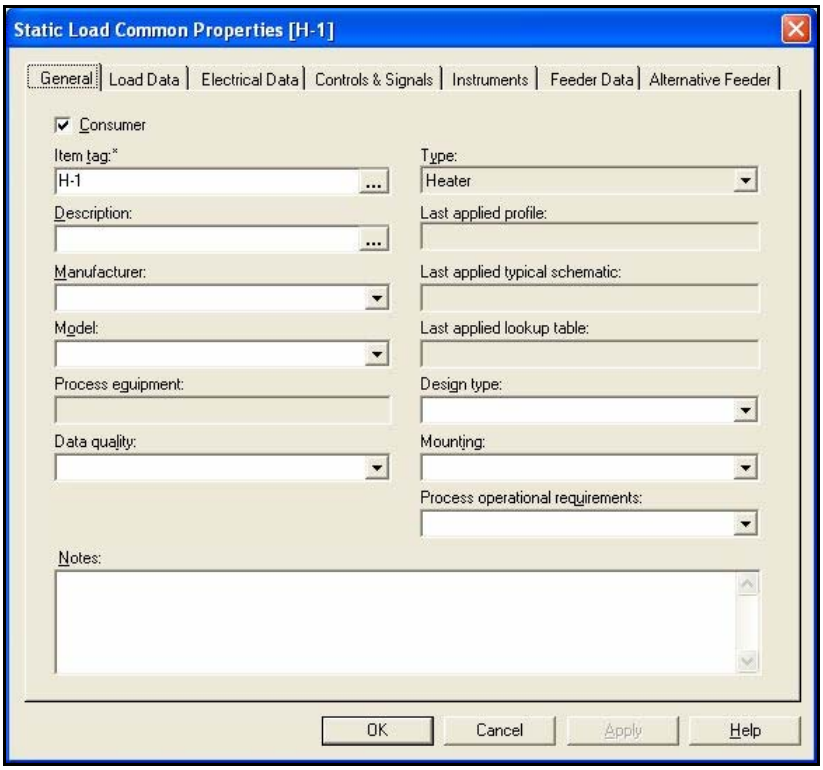
Static Loads, Heater Equipment Form

In this session you learn about all the various properties that may be edited or viewed for Static loads, using the forms and the **Electrical Index** behavior under various circumstances. Also, the corresponding editing functionality using property grid will be mentioned.
The function of the various tabs is very much the same as for the motor except for few minor differences due to the fact that static loads are not rotating, therefore properties such as brake power, LRC power factor, synchronous and asynchronous speeds, and so forth, do not apply.

Heater form

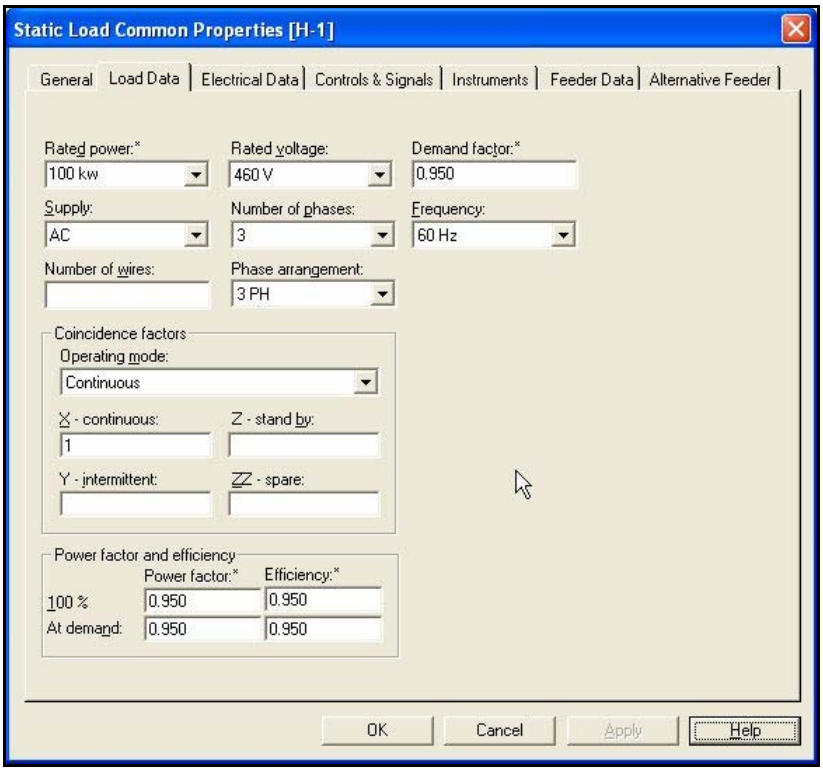
In the **Electrical Index**, select heater H-1, Right-Click, and from shortcut menu select **Common Properties**

General tab



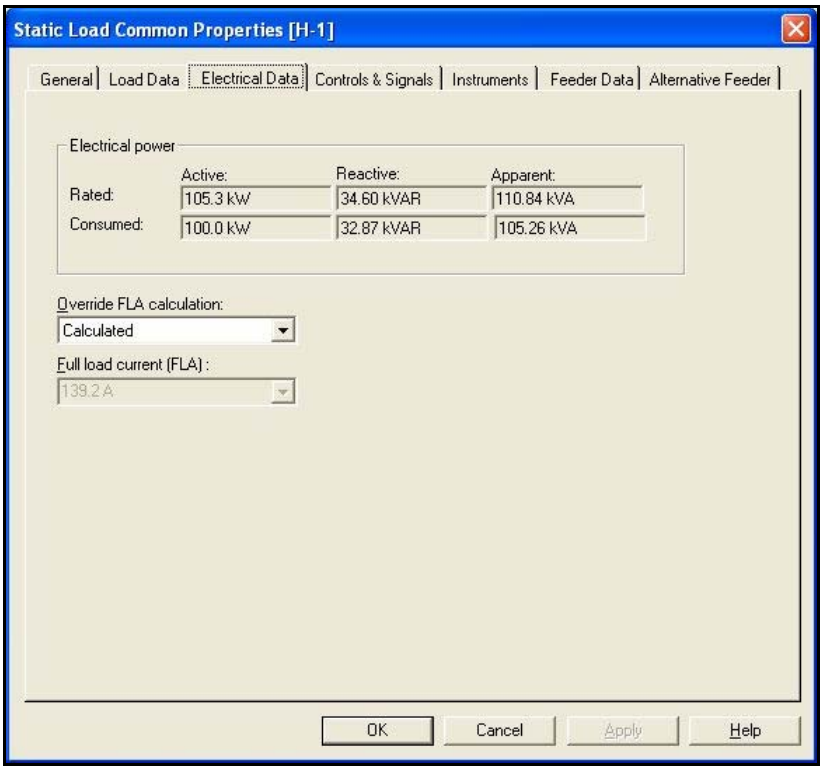
Load Data Tab

Calculated fields
Notes: Rated power for these types of loads is not a select list but rather a property with engineering units. Each type of static load has its own category of units of measure. For example, heaters have Power, whereas capacitors and harmonic filters have Reactive power, etc. The demand factor, power factor, and efficiency at demand are manual entries rather than calculated values.

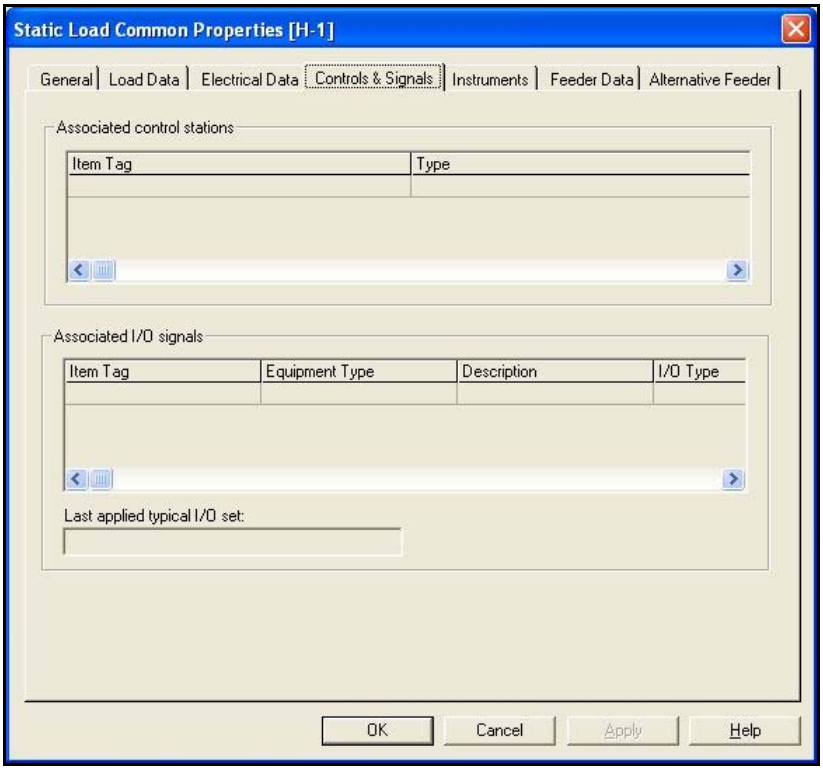


Electrical Data tab

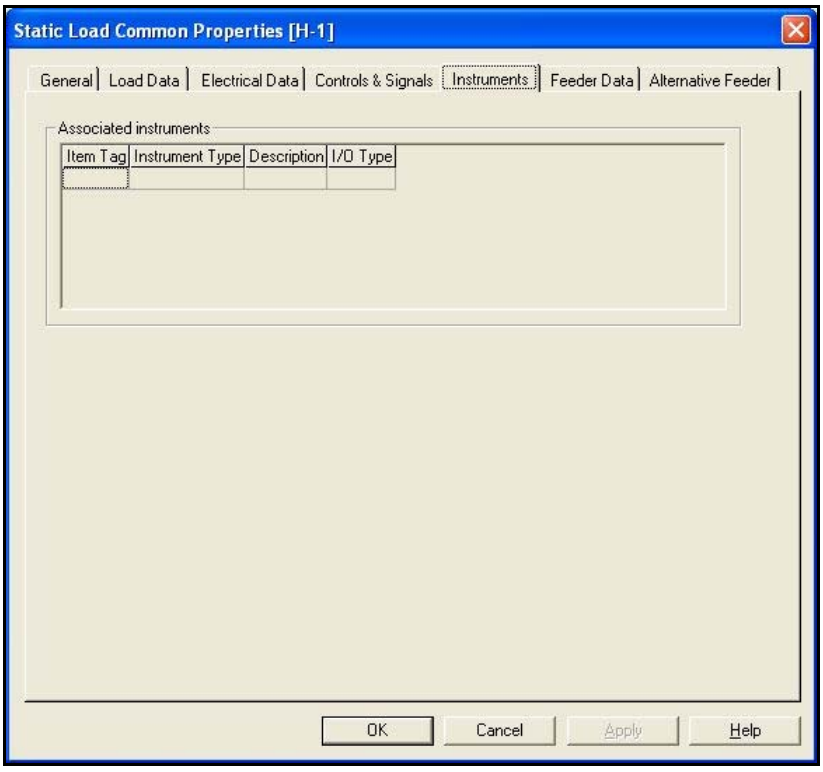
Calculations behind these calculated fields.
The calculated electrical properties are very similar to those for motors.



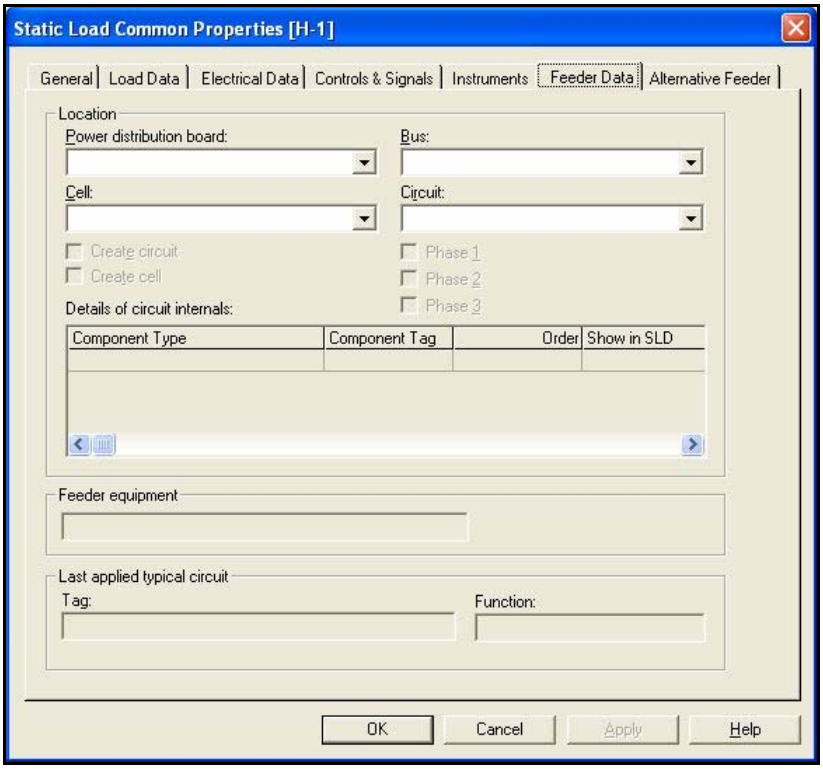
Control & Signal tab



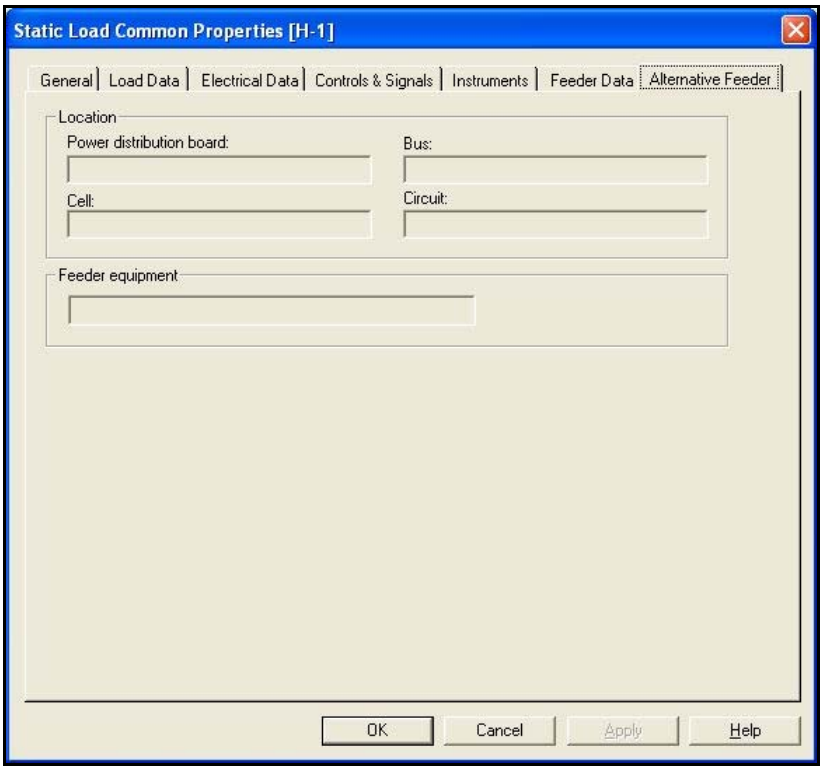
Instruments tab



Feeder Data tab

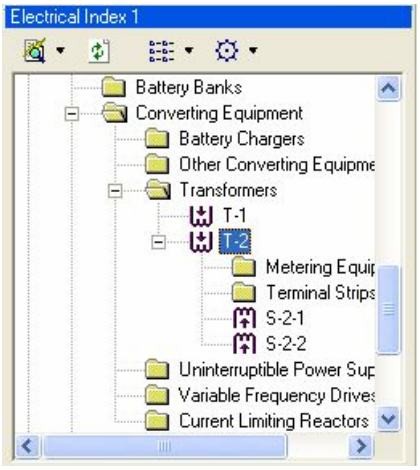


Alternative Feeder tab



Converting Equipment

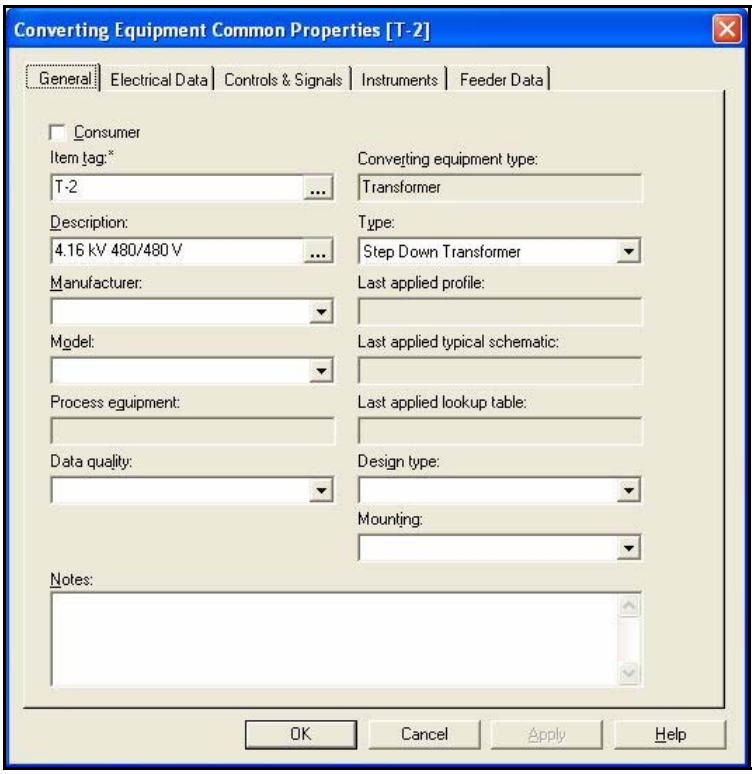
Transformer



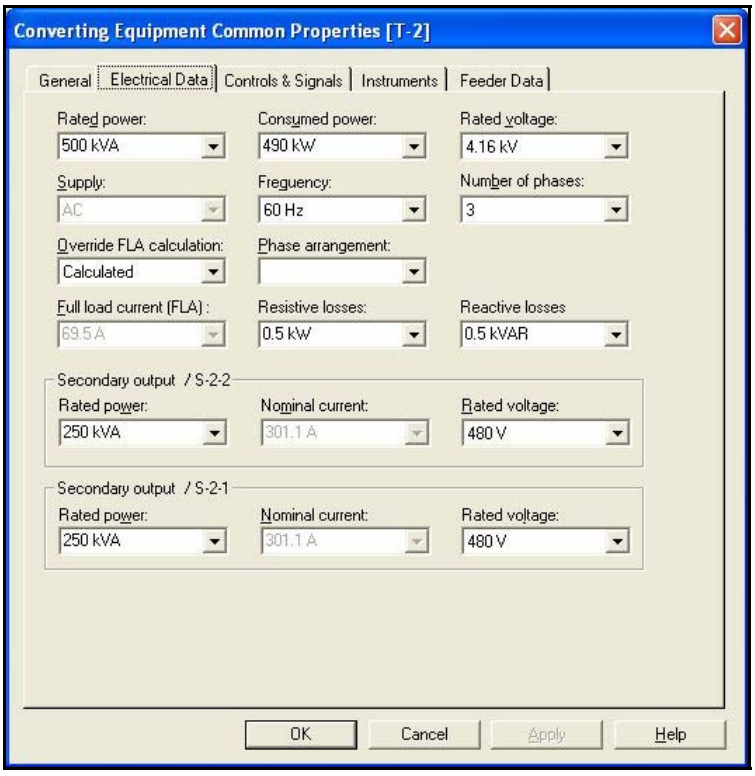
Transformer form

In the **Electrical Index**, select transformer T-2, Right-Click, and from shortcut menu select **Common Properties**

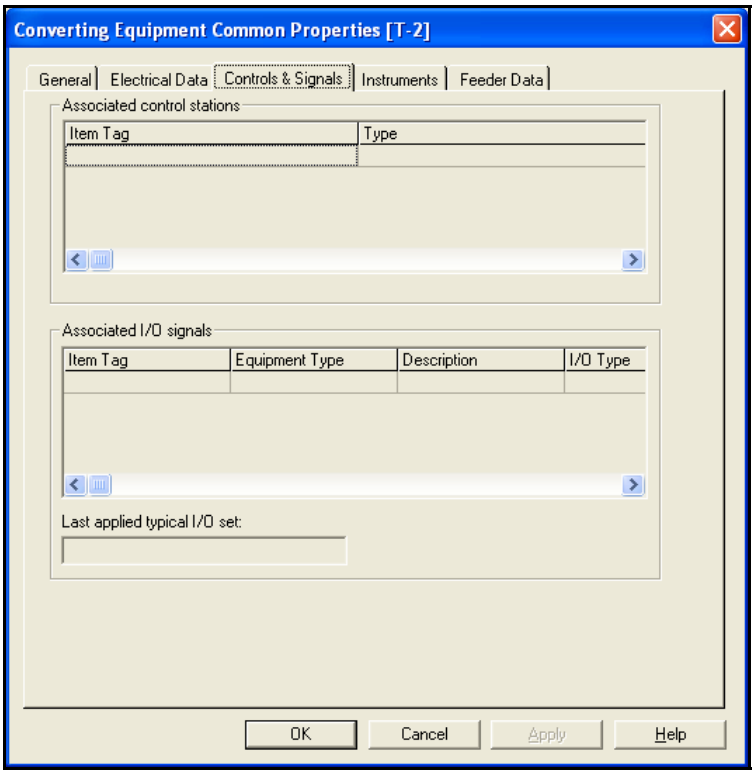
General tab



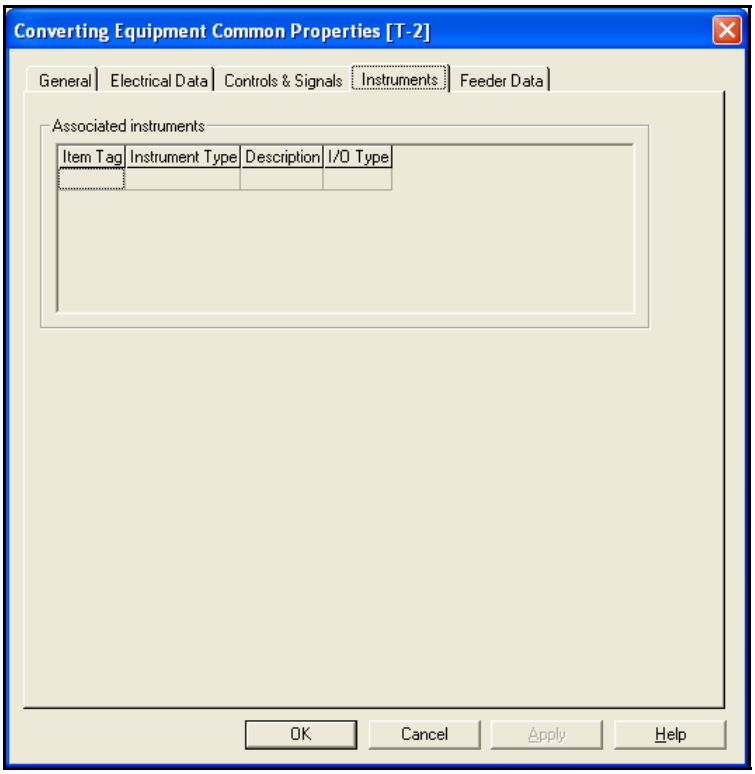
Electrical Data tab



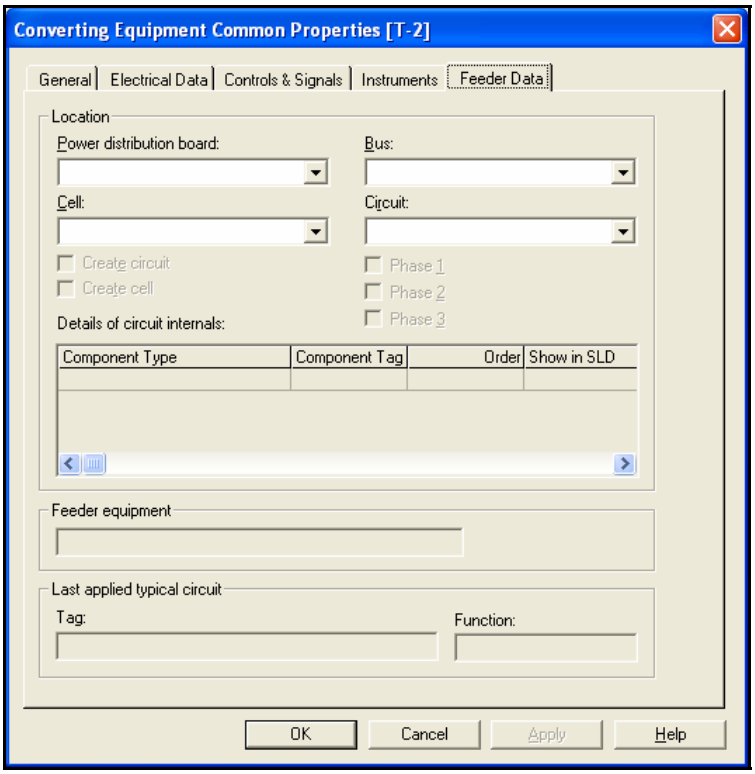
Control & Signal tab



Instrument tab

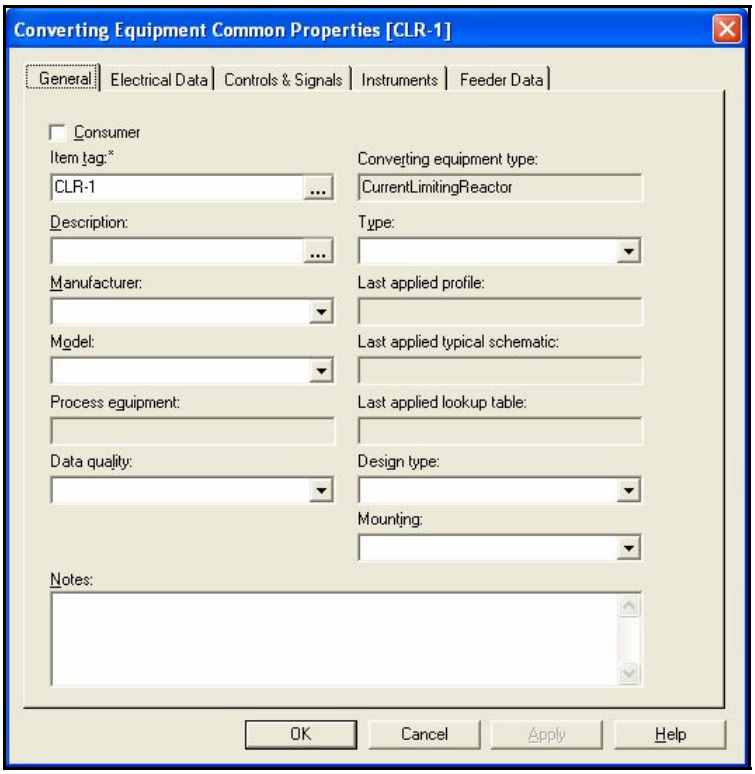


Feeder Data tab

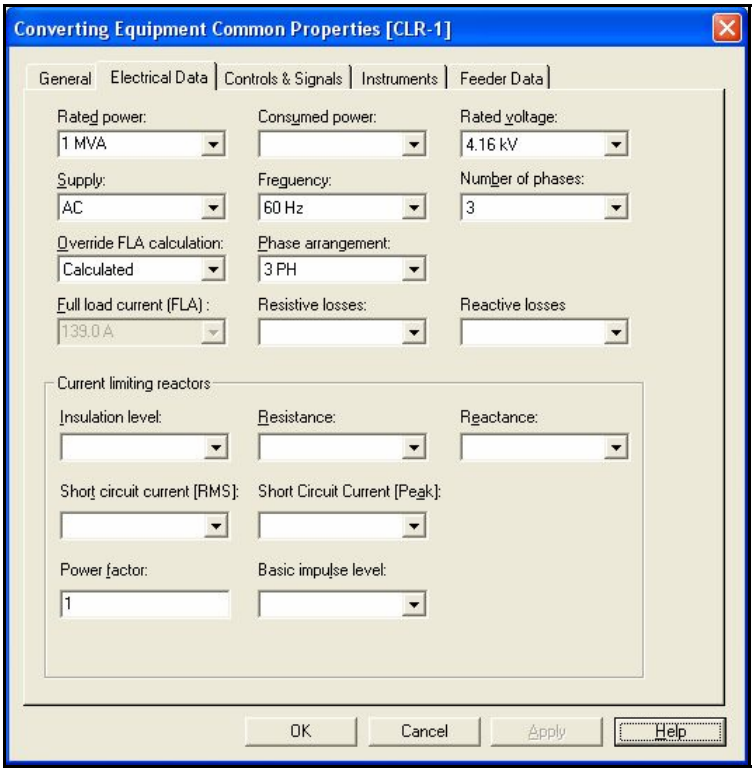


Current Limiting Reactor

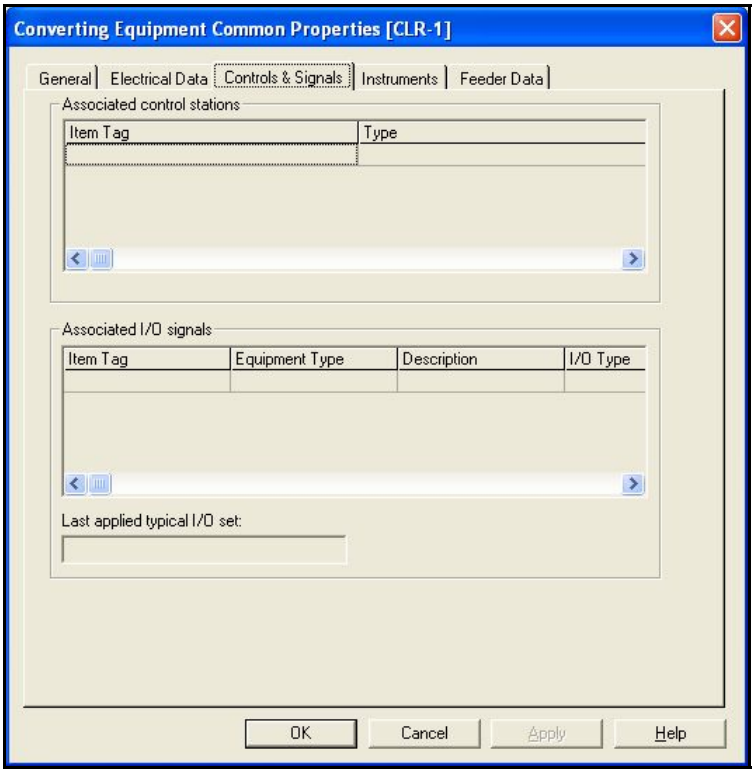
General tab.



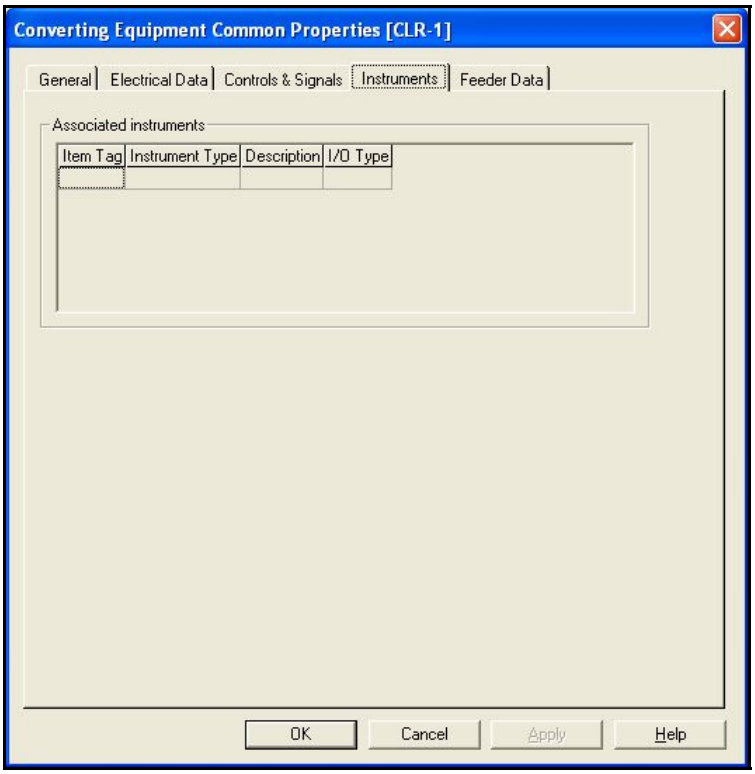
Electrical Data tab



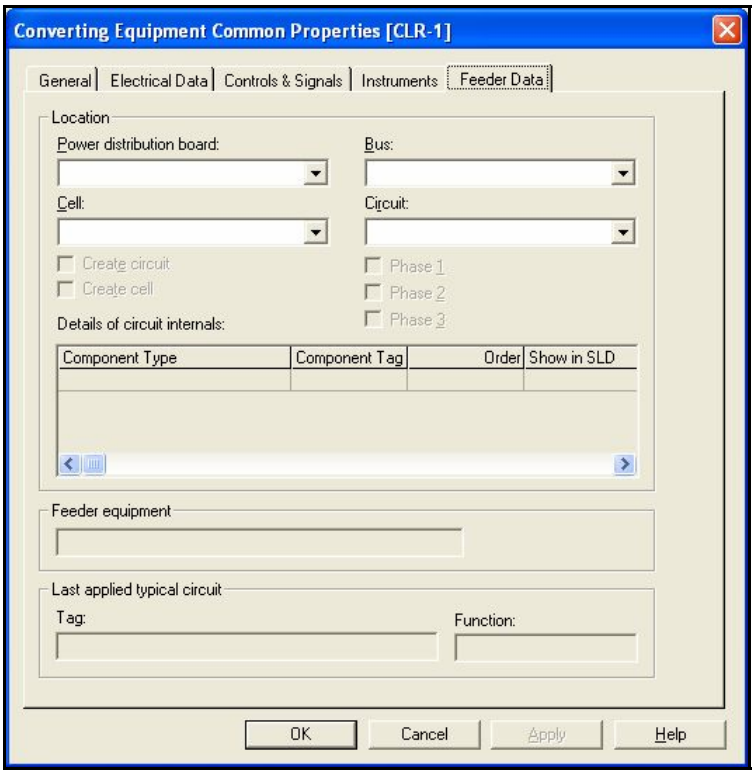
Controls & Signals tab



Instruments tab



Feeder Data tab



Lab 1

Typical Electrical Equipment

Create the following typical items in **Reference Data Explorer**:

Typical Generator

1.1. **'1 MVA 4.16 kV'**
Rated Apparent Power = 1 MVA
Rated Voltage = 4.16 kV
Supply = AC
Number of phases=3
Frequency = 60 Hz
Number of poles=4

Typical Offsite Power Supply

1.2. **'1 MVA 4.16 kV'**
Offsite Power Rated Power = 1 MVA
Rated Voltage = 4.16 kV
Supply = AC
Number of phases=3
Frequency = 60 Hz

Typical Motors

1.3. **'1 hp 460 V w/sh 100 W 120 V'**
Motor Rated power =1 hp
Rated Voltage = 460 V
Brake Power =0.75 hp
Supply=AC
Number of phases=3
Frequency=60Hz
Number of poles=4

Motor space heater name = '**100 W 120 V'**
Rated Power = 100 W
Rated Voltage = 120 V
Supply = AC
Number of phases = 1
Frequency = 60 Hz

1.4. **'10 hp 460 V'**
Motor Rated power =10 hp
Rated Voltage = 460 V
Brake Power =6.8 hp
Supply=AC
Number of phases=3
Frequency=60Hz
Number of poles=4

1.5. **'100 hp 460 V'**
Motor Rated power =100 hp
Brake Power =80 hp
Rated Voltage = 460 V
Supply=AC
Number of phases=3
Frequency=60Hz
Number of poles=4

Typical Heater

1.6. **'100 kW 480 V'**
Rated power =100 kW

Rated Voltage = 480 V
Supply=AC
Number of phases=3
Frequency=60Hz

Typical Transfortmer

1.7. **'1 MVA 4.16 kV 480V/480V'**

Rated power= 1 MVA
Rated voltage (of the primary) = 4.16 kV
Frequency=60Hz
Number of phases=3
Phase arrangment 3 PH

Secondary output '**S1-480V**'
Rated power = 500 kVA
Rated voltage = 480 V

Secondary output '**S2-480V'**
Rated power = 500 kVA
Rated voltage = 480 V

Typical Current Limiting Reactor

1.8. **'1 MVA 4.16 kV'**

Rated Power = 1 MVA
Rated Voltage = 4.16 kV
Supply = AC
Frequency = 60 Hz
Number of phases=3

Project Electrical Equipment

Create the following items in **Electrical Index:**

Project Generator

1.9. **'G-001'**

Copy typical Generator '1 MVA 4.16 kV' from RDE to EI to create project Generator, and name it 'G-001'
Set the following values for G-001:
i. Override FLA calculation = Calculated
ii. Power Factor = 0.95
iii. Efficiency = 0.9

Project Offsite Power Supply

1.10. **'S-001'**

Copy typical Offsite Power Supply '1 MVA 4.16 kV' from RDE to EI to create project Offsite Power Supply, and name it 'S-001'

Project Motors

1.11. 'M-100'

Copy typical motor '1 hp 460 V w/sh 100 W 120 V' from RDE to EI to create project motor, and name it '**M-100'**
Set the following values for motor M-100:
i. Override FLA calculation = Calculated
ii. Operation Mode=continuous (X=1.0)
Name the motor space heater '**SH-100'**
Set the following values for the motor space heater SH-100
i. Override FLA calculation = Calculated
ii. Operation Mode=continuous (X=1.0)

1.12**. 'M-101'**

Copy the typical motor **'**10 hp 460 V' from RDE to EI to create project motor, and name it '**M-101'**
Set the following values for motor M-101:
i. Override FLA calculation = Calculated
ii. Operation Mode=continuous (X=1.0)

1.13**. 'M-102'**
Copy the typical motor **'**100 hp 460 V' from RDE to EI to create project motor, and name it '**M-102'**
Set the following values for motor M-102:
i. Override FLA calculation = Calculated
ii. Operation Mode=continuous (X=1.0)

1.14. **'M-103'**
Duplicate motor M-101 in EI and name it '**M-103**'.

Project Heater

1.15. **'H-104'**
Copy the typical Heater '100 kW 480 V' from RDE to EI to create project heater, and name it '**H-104'**
Set the following values for heater H-104:
i. Override FLA calculation = Calculated
ii. Operation Mode=intermittent (Y = 0.5)
iii.Demand factor =1
iv.Power factor 100% = 0.9, At demand = 0.9
v. Efficiency 100% = 0.9, At demand = 0.9

Project Transformer

1.16. **'T-001'**
Copy typical Tansformer '1 MVA 4.16 kV 480V/480V' from RDE to EI to create project transformer, and name it '**T-001**'
Name and set the transformer secondery outputs
i. Secondary Output name = '**S-001-480V'**
ii. Secondary Output name = '**S-002-480V'**

Project Current Limiting Reactor

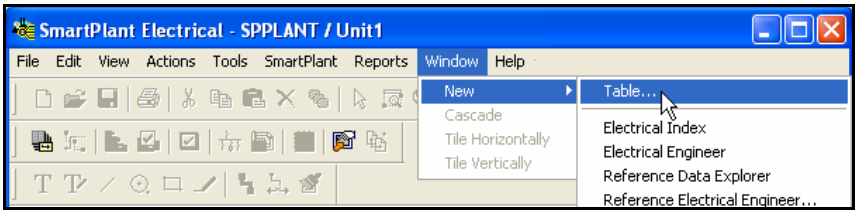
1.17. **'CLR-001'**
Copy typical Current Limiting Reactor '1 MVA 4.16 kV' from RDE to EI to create project Current Limiting Reactor, and name it '**CLR-001**'

Chapter 2

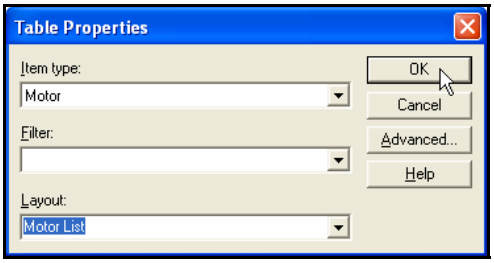
Tabular Editor

Create a Table

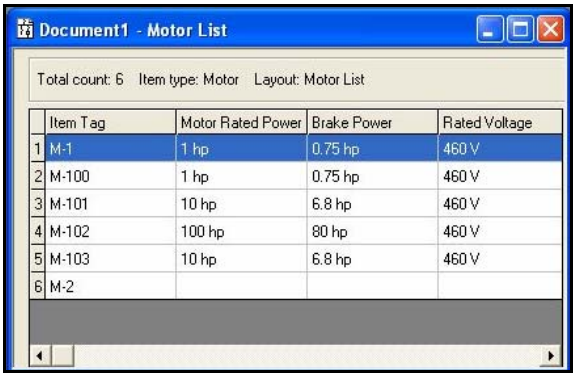
From Window -> New -> Select



Select an item type motor, layout motor list, and select OK to open the tabular editor document.



->



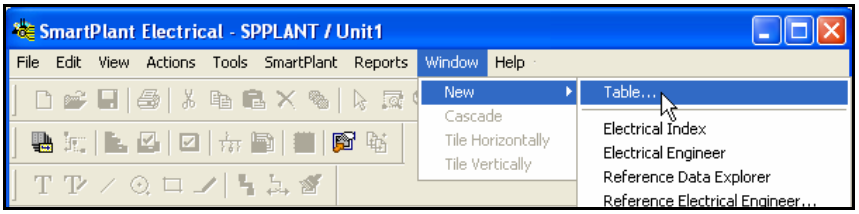
Notes:

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

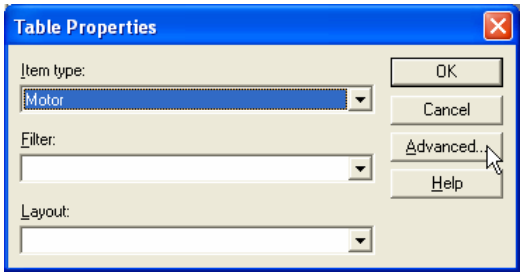
2. The flags show items of all plant group affect the tabular editor.

Create and use layout in tabular editor

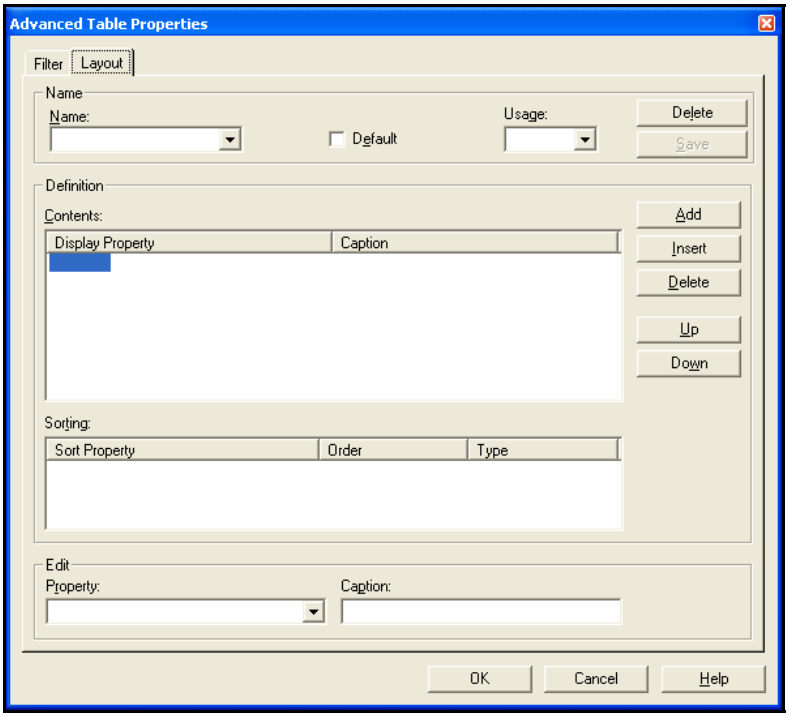
From Window menu select New, and Table



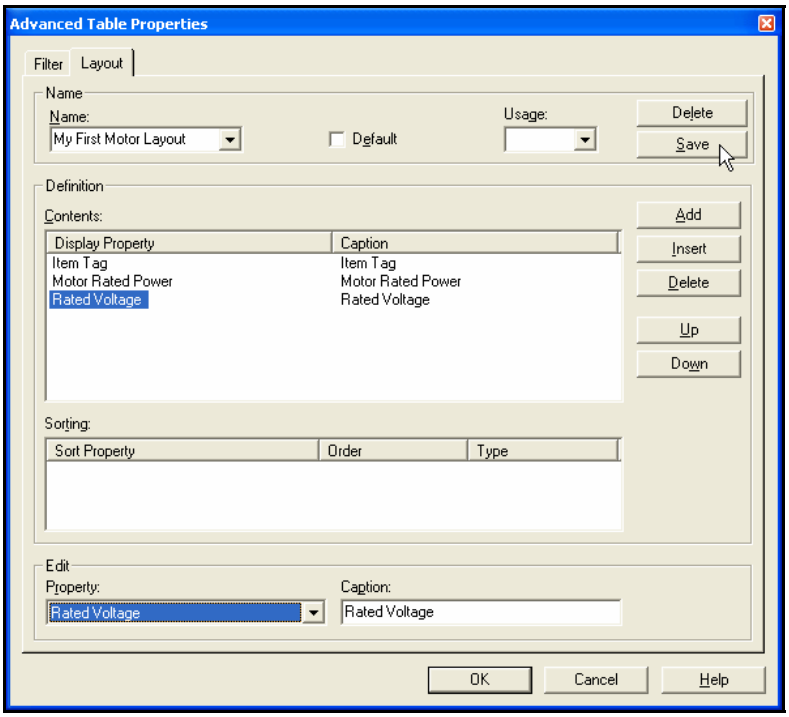
From Table Properties dialog box select item type motor, and select Advanced.



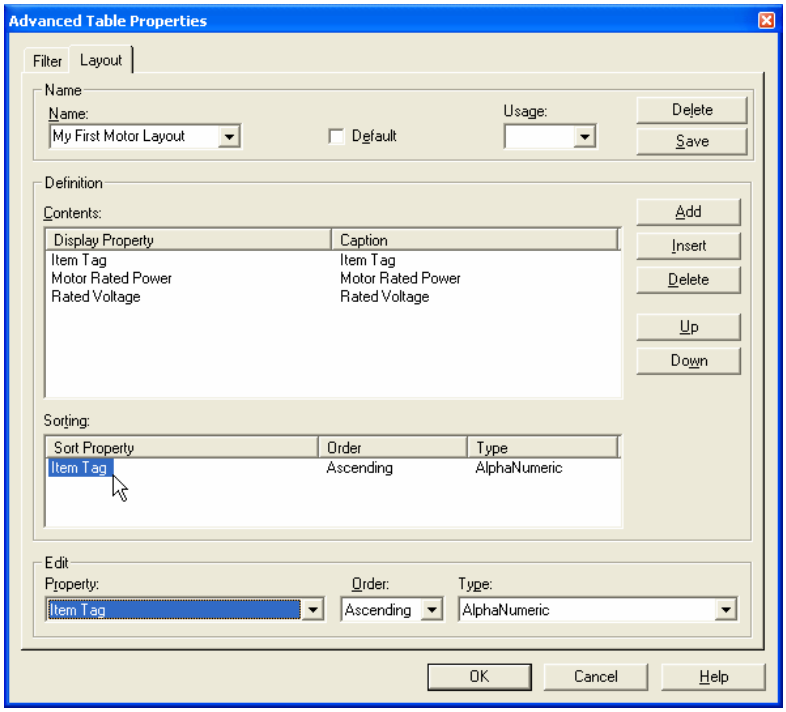
In the Advanced Table Properties dialog box, select layout tab.



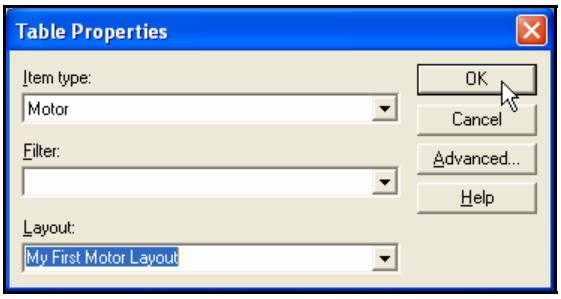
Name the new layout My First Motor Layout.
Add to the layout the properties:
Item Tag,
Motor Rated power,
Rated Voltage
Select Save.

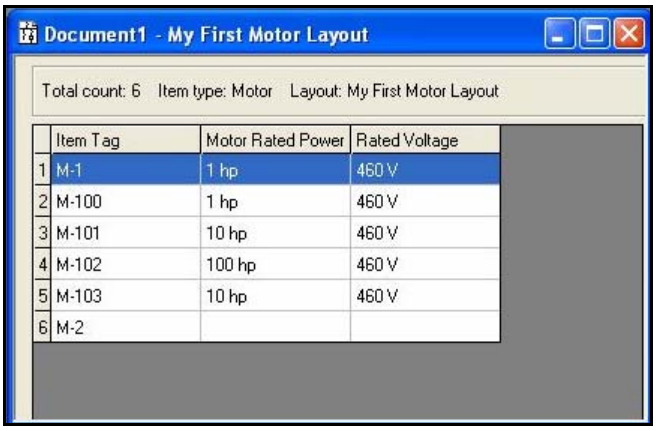


Lets also sort it according to the ItemTag



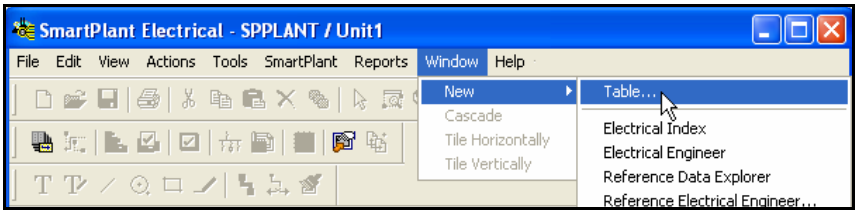
Save it and select OK to close the Advanced Table Properties.
In the Table Properties windows select the new layout, My First Motor Layout, from the layout drop down list and select OK close the Table properties windows and open the tablular editor document.



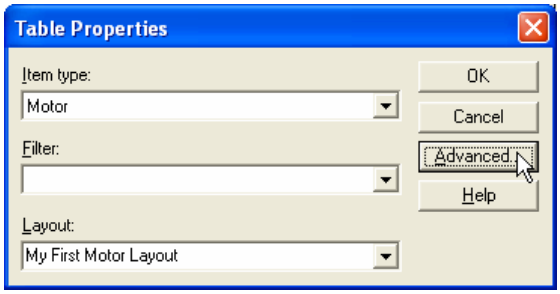


Layout usage

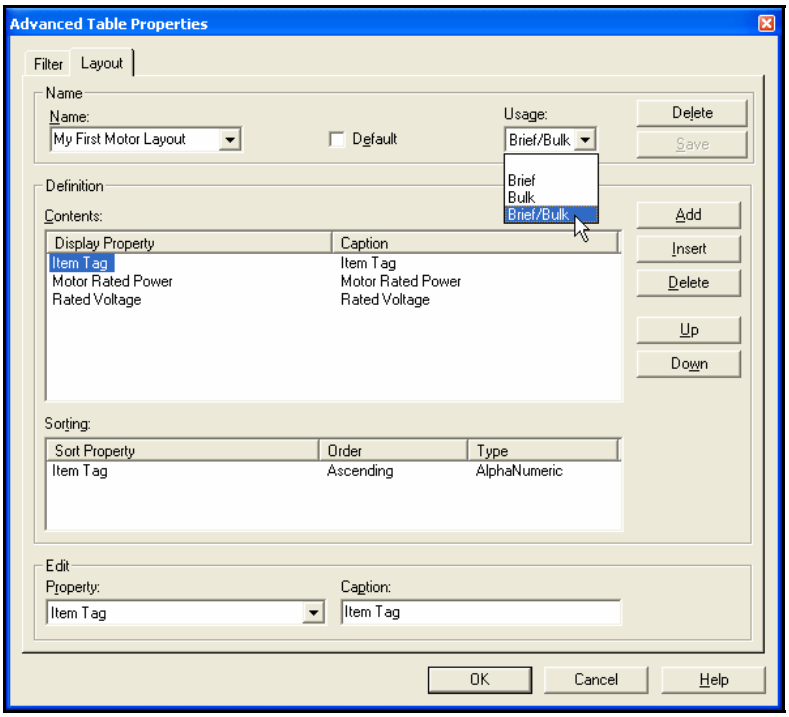
Lists the property display types associated with the **Properties** window. Options include a blank value, **Brief**, **Bulk**, and **Brief/Bulk**. The selected item type and display attributes define what appears in the **Properties** window when you select the **Show Brief Properties**, **Copy Bulk Properties**, or **Paste Bulk Properties** commands from the **Properties** window toolbar.
Lets select the usage of **My First Motor Layout** as **Brief/Bulk**.
From Window -> New -> Table



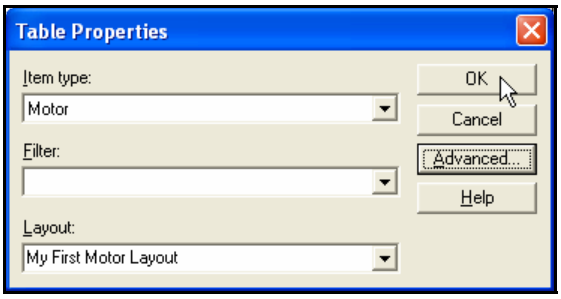
From Table Properties dialog box select item type motor, layout My First Motor Layout and select advanced.



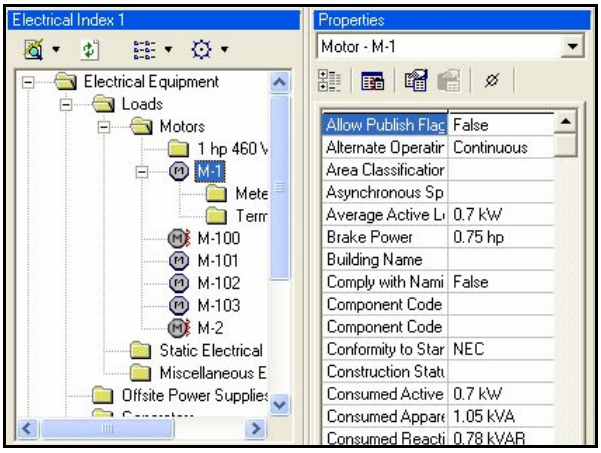
In layout tab, select from the usage drop down list the option Brief/Bulk, select save and select ok.



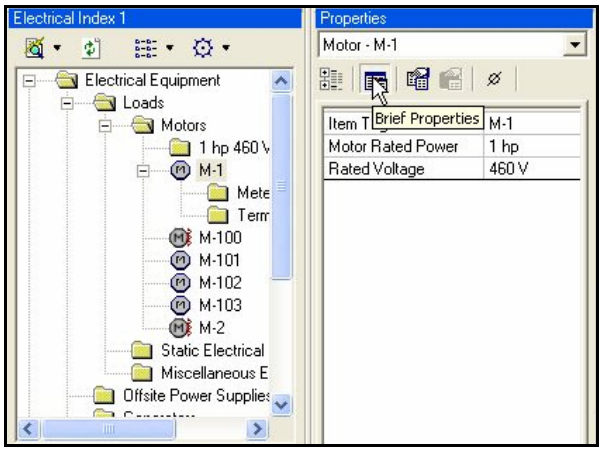
Select OK to close the Table Properties window.



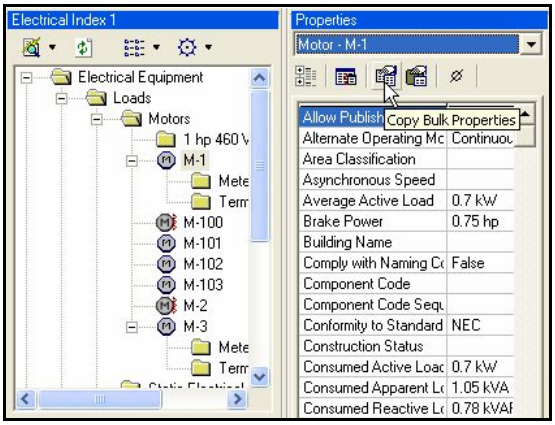
To use the show brief properties option select motor M-1 from Electrical Index.



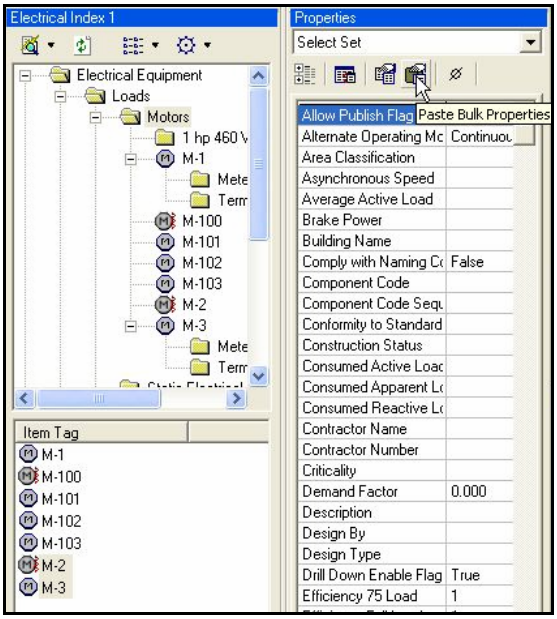
Select the Brief Properties button in property grid window.



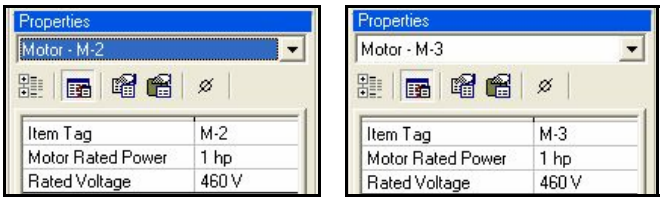
To use the copy and paste bulk properties option lets first create in Electrical Index new motor and name it **M-3**.
Select motor M-1 from Electrical Index, and in properties window select the copy bulk properties button.



To paste the bulk properties into selected tags select the motors M-2 and M-3 from Electrical Index (lower window).And in properties window select ‘Select Set’ and hit the ‘Paste Bulk Properties’ button.

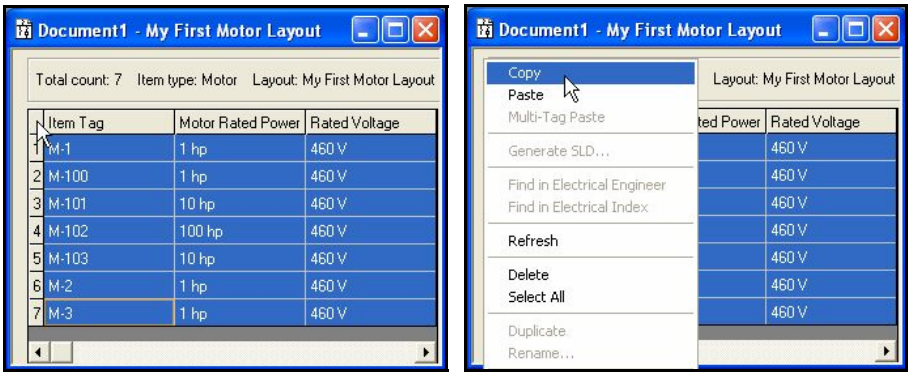


This will copy the values of the properties defined in the Bulk layout (Motor Rated Power and Rated Voltage) from M-1 to M-2 and M-3.

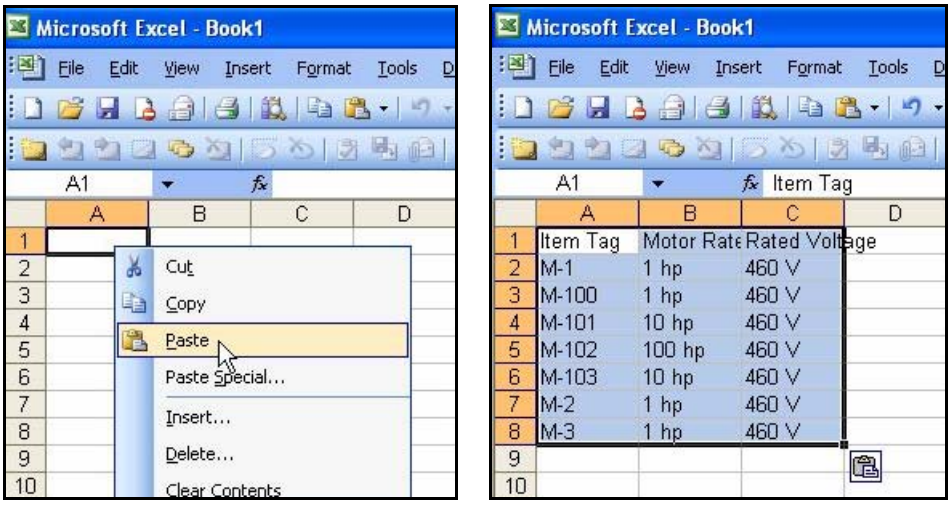


Using the Copy feature to copy a table to an external file

With the tablur editor open, using My First Motor Layout, select the upper left cell of a table. Select right click and copy.



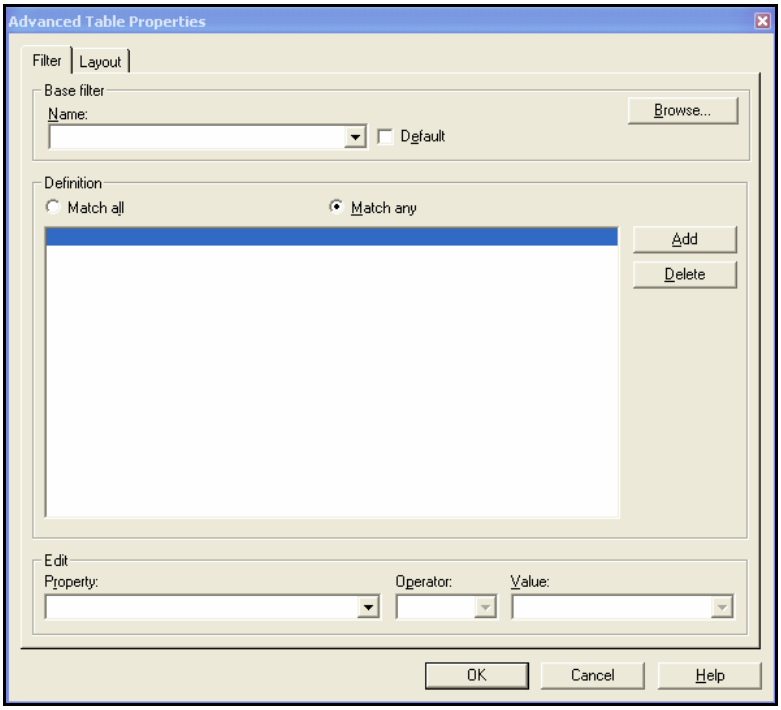
Open excel and select paste it into new spreadsheet.



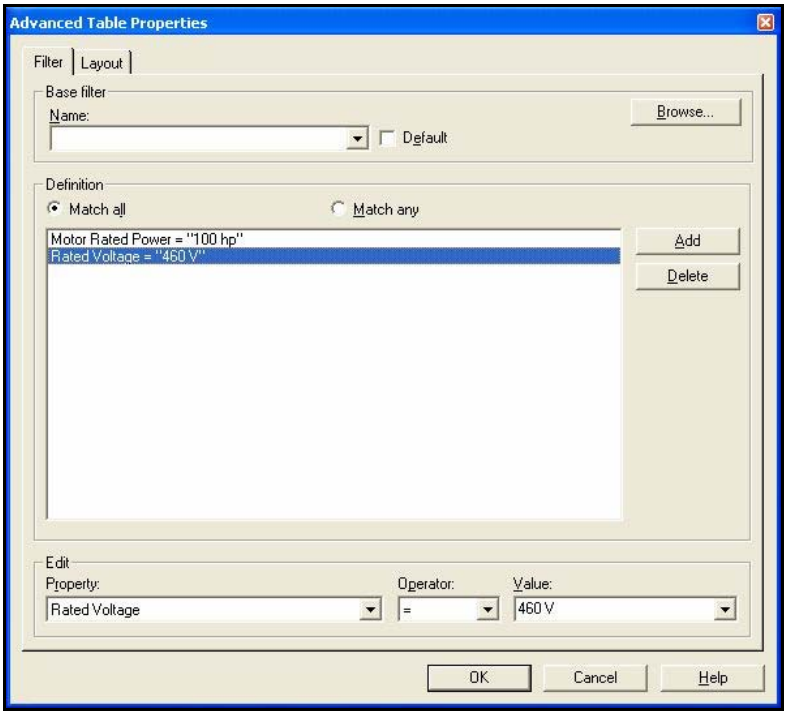
Using filters through the Tabular Editor

With the tablur editor open, using My First Motor Layout, right click and select table properties. From the **Table Properties** dialog box, select Advanced to open the

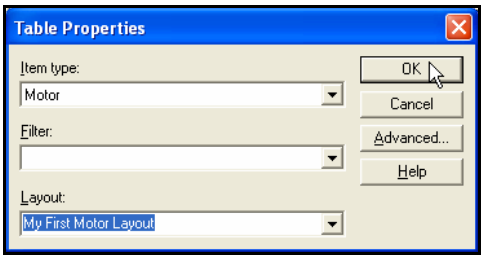
Advanced Table Properties.



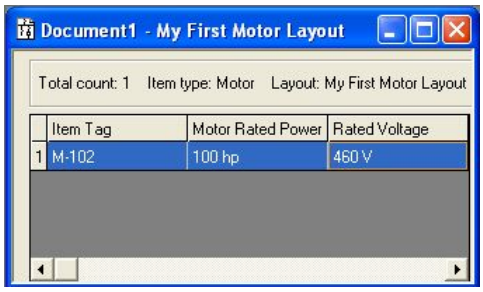
There are 2 types of filters you can create.
1."On the fly filter"
Filter you can use only for the current table.



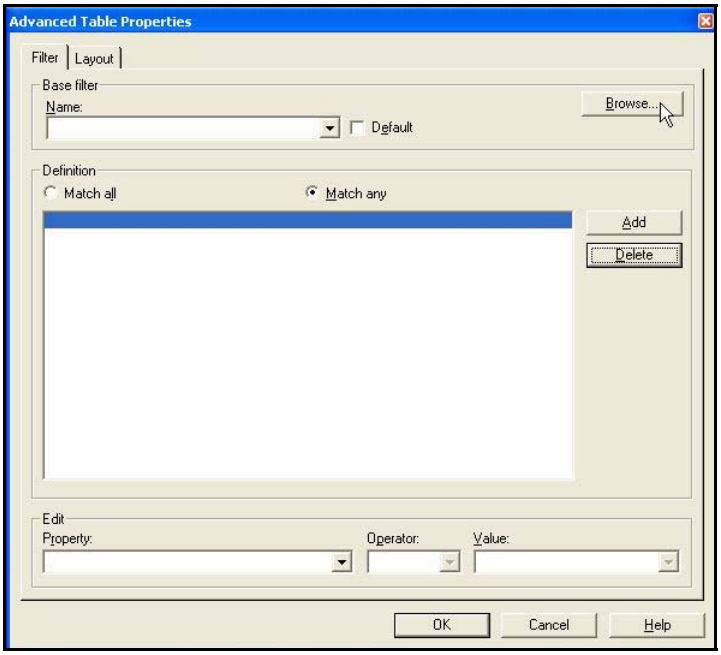
OK.



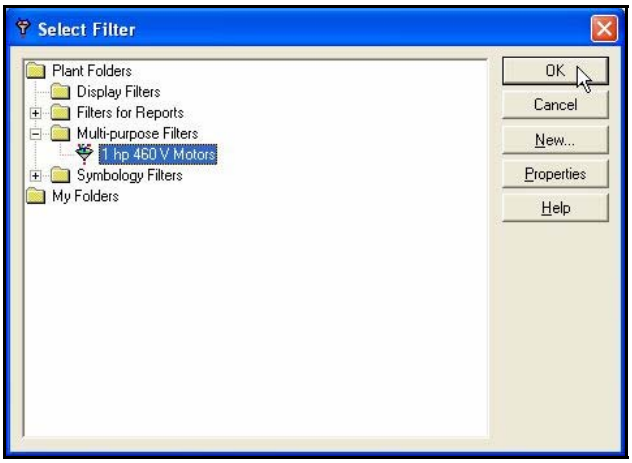
OK.

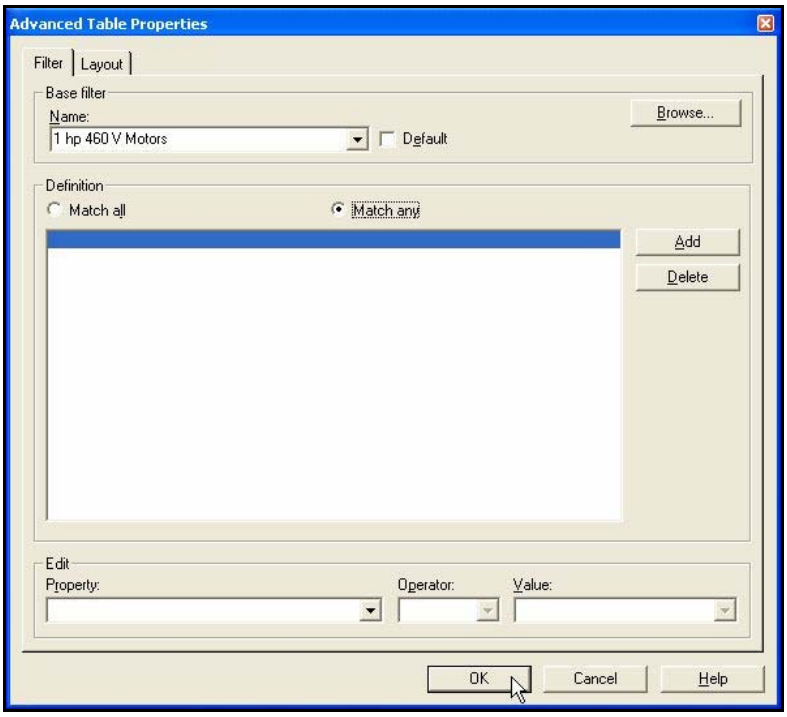


2. "Base Filter"
Filter that you can save in the database and use it as plant filter.
To create base filter, select the Browse button from Advanced Table Properties window.



In the Select filter dialog box select the 1 hp 460 V filter we created before.


Select Filter
Plant Folders
Display Filters
Filters for Reports
Multi-purpose Filters
1 hp 460 V Motors
Symbology Filters
My Folders
OK
Cancel
New...
Properties
Help


Advanced Table Properties
Filter
Layout
Base filter
Name:
1 hp 460 V Motors
Default
Browse...
Definition
Match all
Match any
Add
Delete
Edit
Property:
Operator:
Value:
OK
Cancel
Help

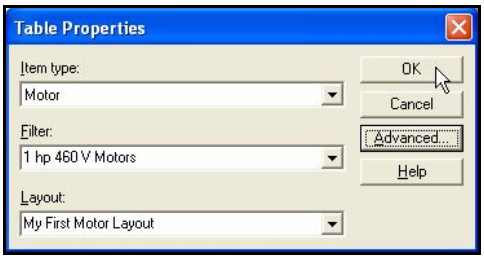
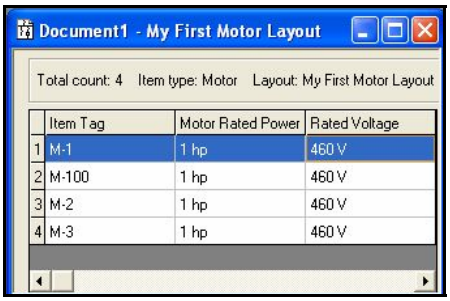


Table Properties
Item type:
Motor
Filter:
1 hp 460 V Motors
Layout:
My First Motor Layout
OK
Cancel
Advanced...
Help



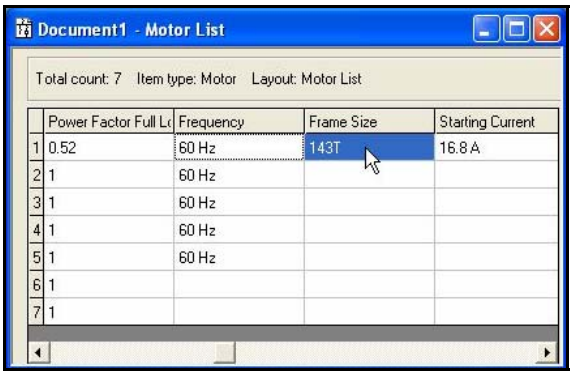
Using tabular editor and data maniulation

You can perform the following operations in the Tabular Editor:

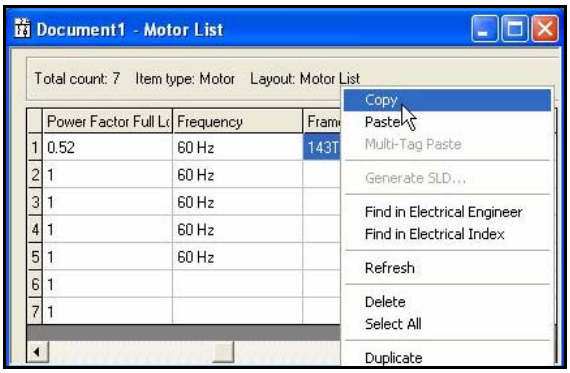
- Create new items
- Duplicate items
- Rename items
- Modify items
- Delete items
- Move a cell from one location to another like in Excel.
- Change the width of a cell by dragging the edge of the cell header and making the cell wider or narrower. When you save the settings, the software retains cell widths until the next time you open the layout.
- Copy and paste data from one item to another.
- Copy and paste data from one group of cells to another similar set of cells.
- Sort data.
- Filter data.
- Open the item common properties form
- Find items in Electrical Index and Electrical Engineer.

Copy and Paste

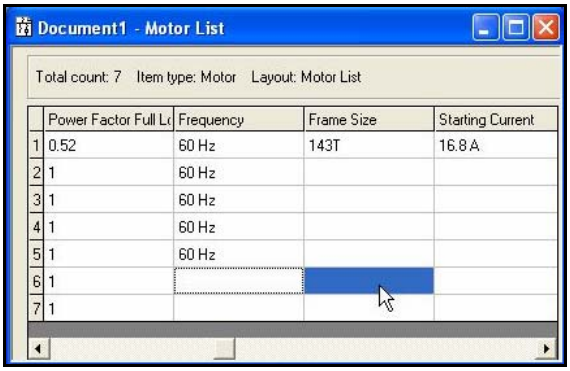
Open the tabular editor with motor as item type, and Motor List as layout.
Make sure that for motor M-1 you have Frequency = 60 Hz and the Frame Size = 134T.
Now, lets copy those 2 fields from M-1 to M-2. To do that, highlight both fields by marking it with left click.

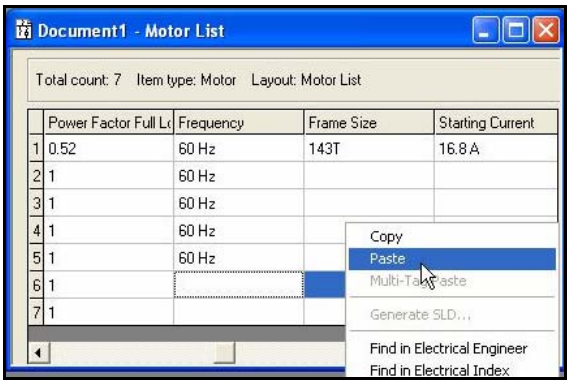


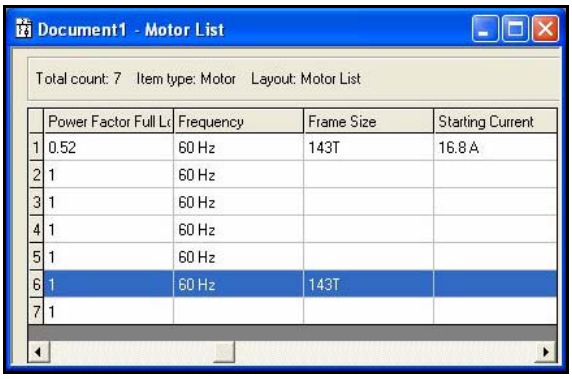
Then, right click and select copy, to copy it to the buffer.



To copy it from the buffer into M-2, mark the same fields for M-2 with left click.
And than right click and select paste.



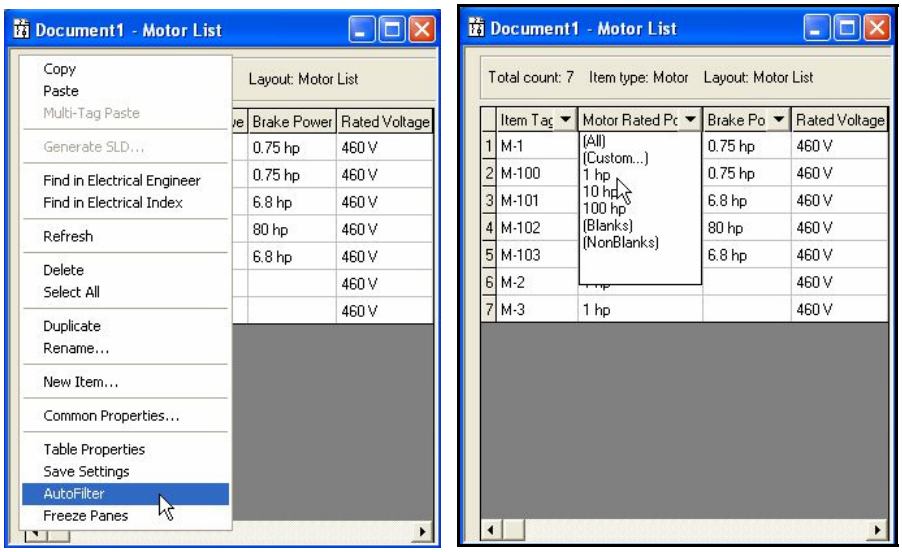


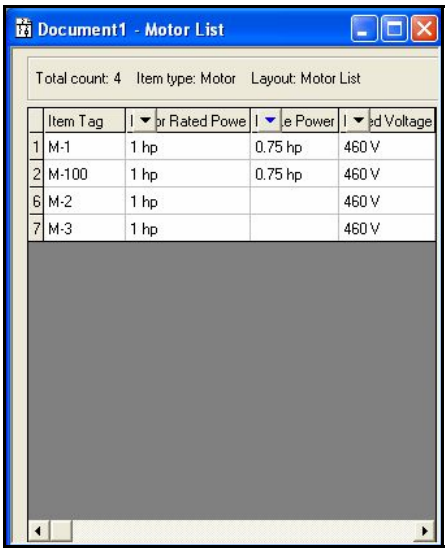


To copy a complete tag (without of course copying the value of the Item Tag property), you select the required tag by clicking on the left side sequence number of that tag, right-click to display the shortcut menu, click Copy, and then select another tag and click Paste on the shortcut menu.
To copy a complete record to a group of records, copy the required tag as before, then select the required target tags using the mouse + Ctrl key, and then right-click and paste.

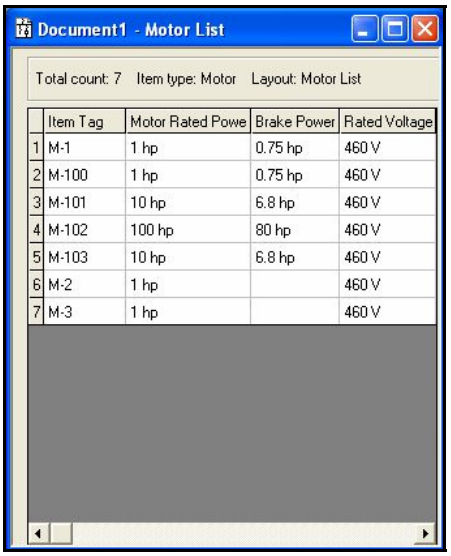
Auto Filter

Allows you to filter the tags according to various values that the selected property contains.
When you click the **Auto Filter** command, arrows appear at the top of each column. Click on the arrow at the top of the column that contains the data you want to filter on and choose the value from the list that appears. You can include more than one property in the auto filter of your table, creating a chained filter.



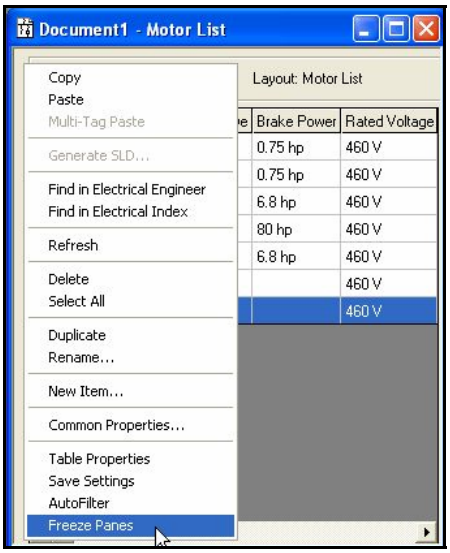


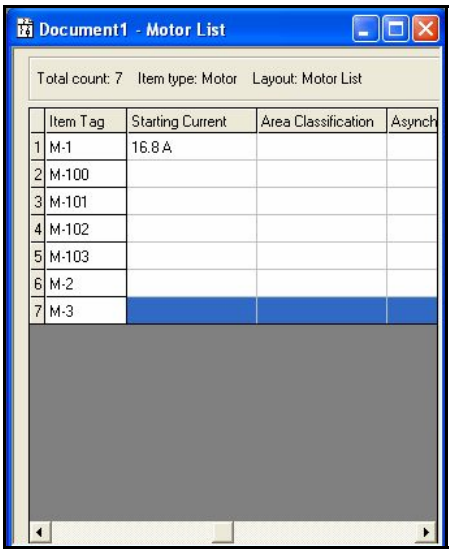
Click Auto Filter command again to remove the filter.



Freeze Panes

Allows you to scroll through the Tabular Editor while keeping the display of specified rows and columns constant. For instance, the first column can contain the item tag, but you want to always see the item tag as you scroll to the right end of the rows to view the supplier.





Lab 2

Create Table and view the data

2.1. Create new table with the following parameters and view the data.
Item type: **Motor**.
Layout name: **Motr Layout Lab 2**
Sort: **Item Tag**
Columns: **Item tag,Description,Motor Rated Power,Rated Voltage**
2.2. Add auto filter to the table you created, and filter on 1 hp motors or 10 hp motors,
And view the data.
Remove to auto filter for the next step.
2.3. From the table, add 3 new motors M-4,M-5 and M-6.
2.4. From the table, edit M-4 , 'Motor Rated Power' to 2 hp, and Rated Voltage to 460V.
Notes: 1. Double click on cell enables the cell for editing.
From the table, copy the complete record of motor M-4, to the group of motors M-5, and M-6.
2. Copy data has been executed only for properties that appear in the layout.

Modify layout and adding base filter

2.6. Modify the table layout by adding to it the properties, **Frequency,** and **Frame Size**.
Save the change and view the data.
2.7. Add to the layout 'Base Filter', and name the filter, '**2 hp 460 V Motors**'.
Note: The filter should bring 2 hp, 460 V motors.
Save the change and view the result.

Using freeze pane option

2.8. Use the freeze pane option in the table to freeze the **ItemTag**.
To view the result, scroll to the right and verify the ItemTag column freezes.

Chapter 3

Power Distribution Equipment

Power Distribution Boards (PDB)

PDB's and the related internal associated items play a major role in the design of an electrical distribution network. A PDB usually is a confined enclosure that contains the necessary electrical internal equipment that will accept and distribute power from power sources to loads or other secondary distribution PDB's. While all types of PDB's behave the same (SmartPlant Electrical wise) there are different names for these PDB's, based on what load they supply and what function they serve in the electrical network. The following terms are used in a mixed way in various regions and countries:

Switchgear, PDB that supplies power to other lower voltage PDB's such as MCC'S

Load Center is a PDB that distributes power to MCC's

MCC (Motor Control Center) is a PDB that supplies power mainly to motors.

Switchboard is usually a PDB that delivers power to small loads such as lighting fixtures and low voltage circuit breaker distribution boards

PDB's accept power (current under specific voltage levels) from power sources such as generators, UPS, battery banks, other PDB's, transformers, etc.

PDB's deliver power (current under specific voltage levels) to loads and to other PDB's.

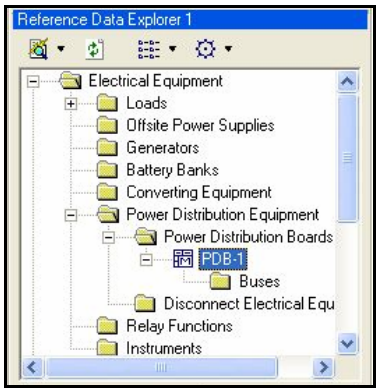
PDB's accept power through **incomer** circuit that are connected to internal physical metal buses (copper or other conducting material).

PDB's deliver power through internal **feeder** circuits that connect to the internal buses.
Internal buses of a PDB can supply power to each other by interconnecting circuits of 2 types.Either by coupler-riser to form a bus tie (for redundant buses where the 2 buses have the same voltage) or by simply feeder-incomer circuits (the feeder circuit of bus A supplies power to lower level voltage bus B, an auxiliary bus).

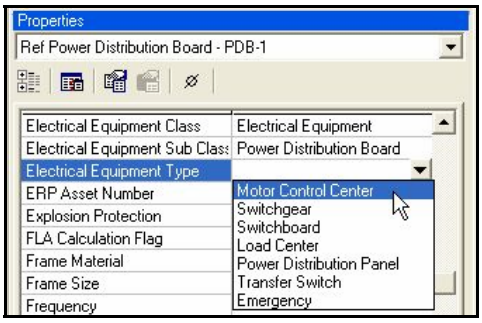
Create Typical PDB in RDE

Use the following procedure to create a new typical PDB:

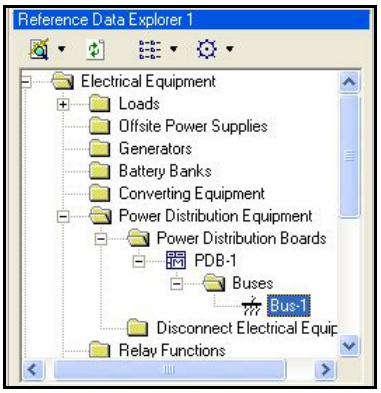
1. In RDE, select Electrical Equipment, Power Distribution Equipment, Power Distribution Board.
2. Right Click on Power Distribution Board folder and select New Power Distribution Board.
3. Rename the new Power Distribution Board and call it 'PDB-1'.



The type of the PDB can be defined from the Electrical Equipment Type select list.

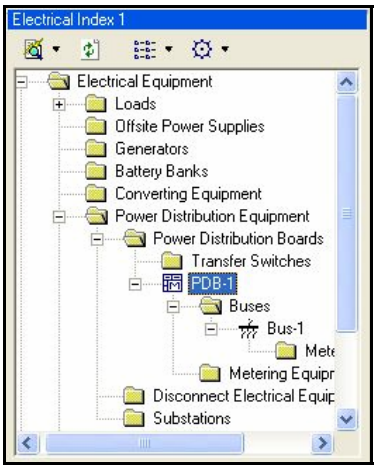


To create new Bus in the new PDB-1 do the following:
1. Select the Buses folder, right click and select New Bus
2. Rename the new bus and call it Bus-1

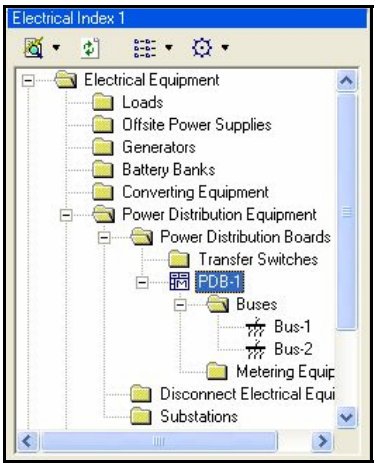


Create Project PDB in EI:

Use the following procedure to create new typical PDB in EI
1. From RDE select PDB-1 and copy it to EI, under Electrical Equipment, Power Distribution Equipment, Power Distribution Board.
2. Rename the new PDB to 'PDB-1'.

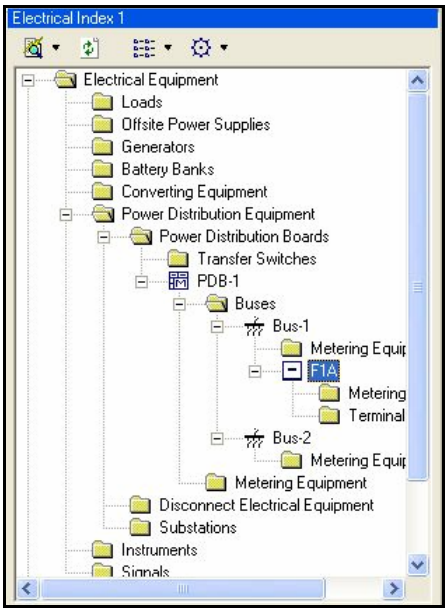


To add 2nd bus to the PDB do the following:
1. Select PDB-1, right click on the Buses folder and select New Bus.
2. Rename the new Bus and call it Bus-2.



Cell (bucket) – A removable compartment in a power distribution board that contains circuitry.

Use the following procedure to create project cell in EI:
1. Select the bus 'Bus-1', right click, select new Cell.
2. Rename the new cell and call it 'F1A'.



Circuits, and Circuit Internals

Circuits - A circuit, is a logic electrical object that contains electrical equipments.
We define 4 different circuit types in Smart Plant Electrical that you can create and use In PDB.

Incomer circuit A circuit that supplies power to a power distribution board.

Feeder circuit A circuit that distributes power from a distribution board to other equipment.

Coupler circuit A circuit that connects two buses in a power distribution board for the purpose of creating a bus tie. This circuit belongs to the bus that serves as the power source of the coupled bus in the PDB.

Bus Riser circuit A circuit that connects two buses in a power distribution board for the purpose of creating a bus tie. This circuit belongs to the bus that serves as the receiver of backup power from the feeder bus in the PDB.

Circuit Internals

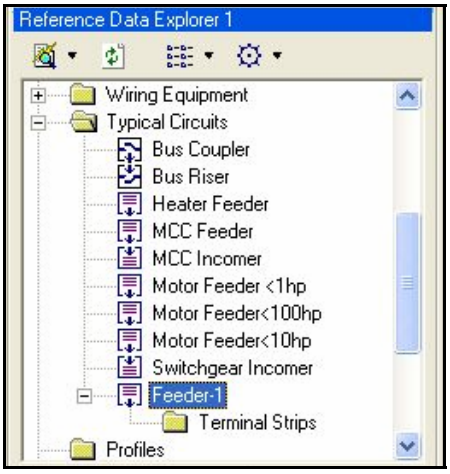
Circuit internals are electrical equipments that you can add to circuit.
The electrical equipments you can add are:

Battery Charger
Other Converting Equipment
2-winding transformer
UPS
VFD
Contactors
Overload Relay
Disconnect Switches

Circuit Breaker
Fuses
Starters
Other Disconnect Equipment
Current Transformer
Potential Transformer
Voltmeters
Ammeters
Multimeters
Protection Relays

Create typical circuit in RDE

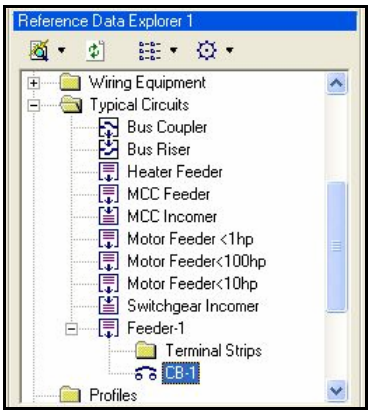
Use the following procedure to create new typical circuit:
1. In RDE, select Typical Circuit.
2. Right Click on typical circuit folder and select New Feeder Circuit
3. Rename the new typical circuit and call it 'Feeder-1'



Create typical circuit internals in RDE

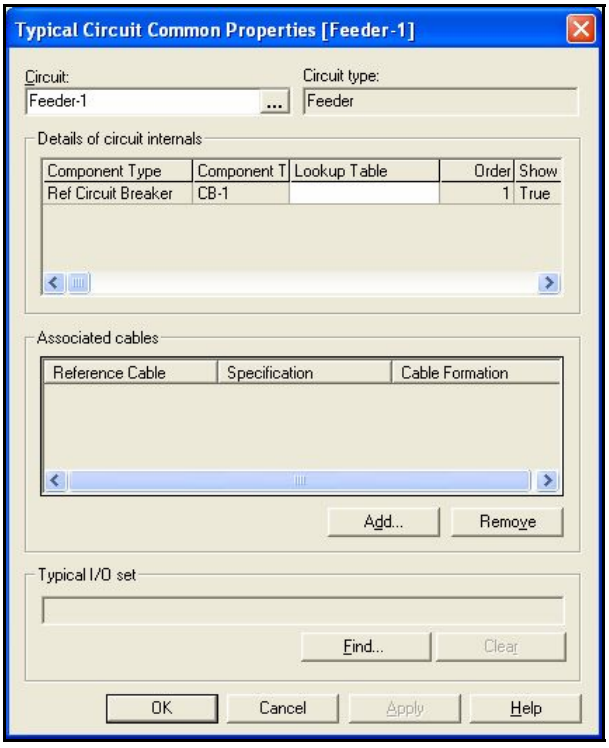
Use the following procedure to create typical circuit internal:

1. From RDE, select typical circuit 'Feeder-1'.
2. Right click, select New Circuit Breaker.
3. Rename the new circuit breaker and call it 'CB-1'



Typical Circuit Common Proprties window

To open typical circuit common properies, select the typical circuit, right click and Select common properties.



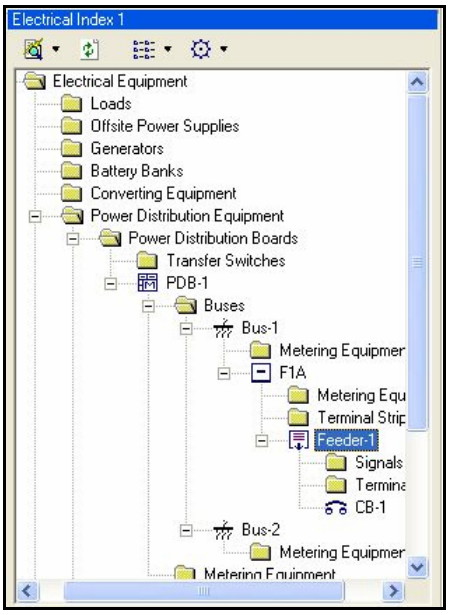
Associate cables section, allow you define typical Instrumentation, Control, or Grounding cable(s). And when you create project circuit, cable(s) will be created, and their 'To' side connected to the circuit. The 'From' side can be connected to a DCS/PLC.

Create PDB's circuit in EI

Use the following procedure to create circuit in a PDB:

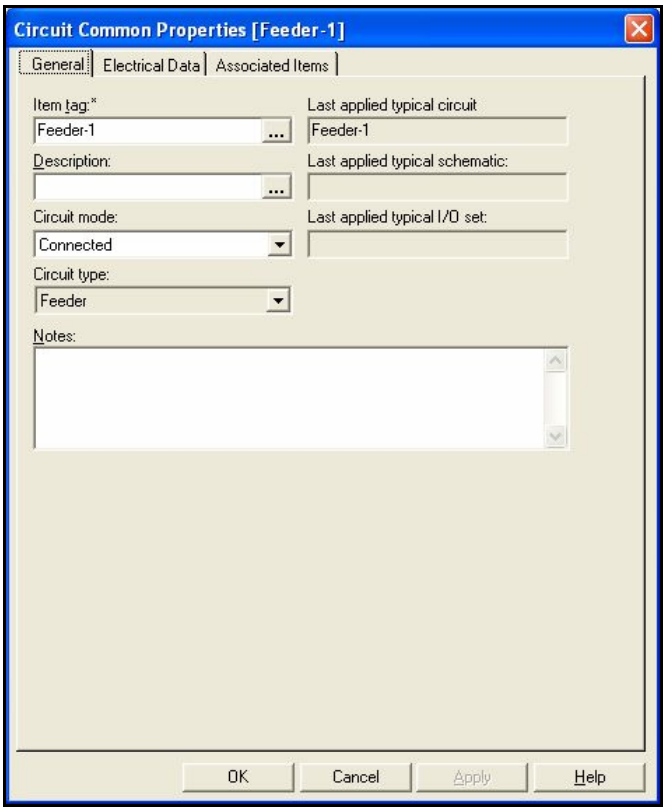
1.From RDE, select typical circuit 'Feeder-1', copy it to EI, under

the cell ‘F1A’ in PDB-1,Bus-1.
2.Name the new circuit ‘Feeder-1’



PDB’s circuit Common Proprties window

To open PDB’s circuit common properies, select the circuit, right click and select common properties



Transfer Switch

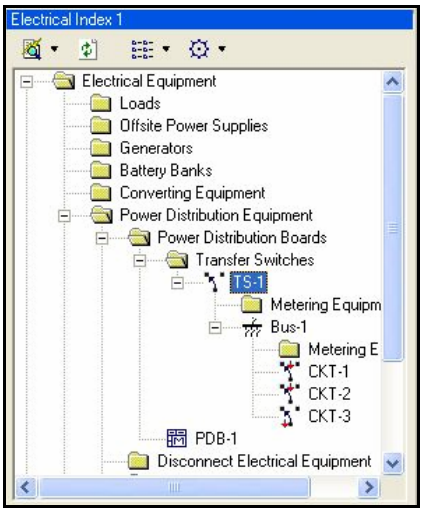
Transfer switches are two-way manual or automatic switches that are used to switch or transfer an essential load from the main power source to a backup power source.

Automatic Transfer Switches are an integral part of the power generation process, allowing smooth and immediate transfer of electrical current between multiple sources and the load. When the generator is operating, the transfer switch prevents dangerous feedback of current to the utility's system. It also ensures that different power sources are fully synchronized before the power is combined or loads are transferred, which is imperative for safe operation.
The transfer switch senses when utility power is interrupted, and starts up the generator if the utility power remains absent. In about five to ten seconds, when the generator is producing full power, the transfer switch disconnects the load from the utility and connects it to the generator, restoring electricity to the load. The transfer switch continues to monitor utility power, and when it is restored, switches the load from the generator back to the utility. Once the generator is disconnected, it goes through a cool-down routine and is automatically shut down.
These devices are standalone, usually housed in cabinets or mounted on walls, have 2 inputs to accept the mains and backup power source and one output, which connects to one of the switch inputs.
The transfer of the power from input 1 to input 2 can be done in the Electrical Index manually, switching the load from one to a second power source, or automatically, via a monitoring and protection circuitry that senses a loss in power in one of the sources and then switches to the alternate power source.

Create Transfer Switch

Use the following procedure to create Transfer Switch in EI

1. In EI, select Electrical Equipment, Power Distribution Equipment, Power Distribution Boards, Transfer Switches.
2. Right Click on Transfer Switches folder and select Transfer Switch
3. Rename the new Transfer Switch and call it 'TS-1'



Disconnect Electrical Equipment

Disconnect Electrical Equipment are the components that are used internally in Power Distribution Boards in buckets and circuits, or as stand-alone items. Individual disconnect electrical equipment that we create in **Reference Data Explorer** can be used in the project by dragging them to the appropriate **Electrical Index** folder or dragging them onto typical circuits.

Note: Power Distribution board internal circuit components are not shown in the individual folders.

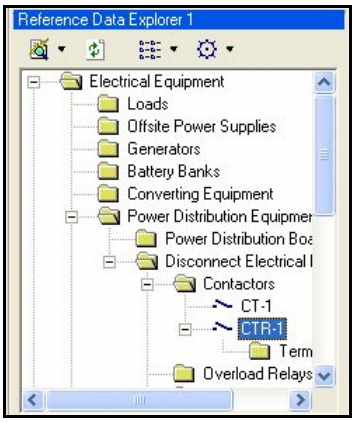
Disconnect Electrical Equipment include:

Contactors
Overload Relay
Disconnect
Circuit Breaker
Fuses
Starters
Other Disconnect Equipment

Typical Disconnect Electrical Equipment

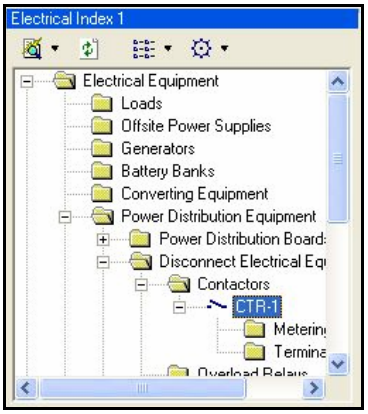
Use the following procedure to create new typical disconnect electrical equipment in RDE:

1.From RDE select, Electrical Equipment, Power Distribution Equipment, Disconnect Electrical Equipment,Contactors.
2.Right Click on the Contactors folder, select New Contactor.
3.Name the new contactor, 'CTR-1'.



Create stand alone project Disconnect Electrical Equipment.

Use the following procedure to create stand alone project disconnect electrical equipment:
1.From RDE select the contactor 'CTR-1' and copy it to EI, under Electrical Equipment, Power Distribution
Equipment, Disconnect Electrical Equipment,Contactors.
2.Name the new contactor 'CTR-1'



Metering Equipment

Metering Equipment uses for measuring electrical parameters such as voltages and currents. The software allows you to create voltmeters, ammeters, and multimeters. For measuring large voltages and currents, you can also create measuring transformers for these parameters. You create standard meters and measuring transformers in the Reference Data Explorer as standalone items or as part of reference circuits. In addition, you can create protection relays with various functions for protecting the circuit.

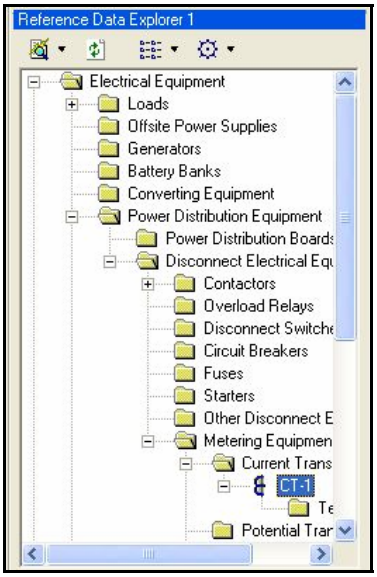
The metering equipment include:

Current Transformer
Potential Transformer
Voltmeters
Ammeters
Multimeters

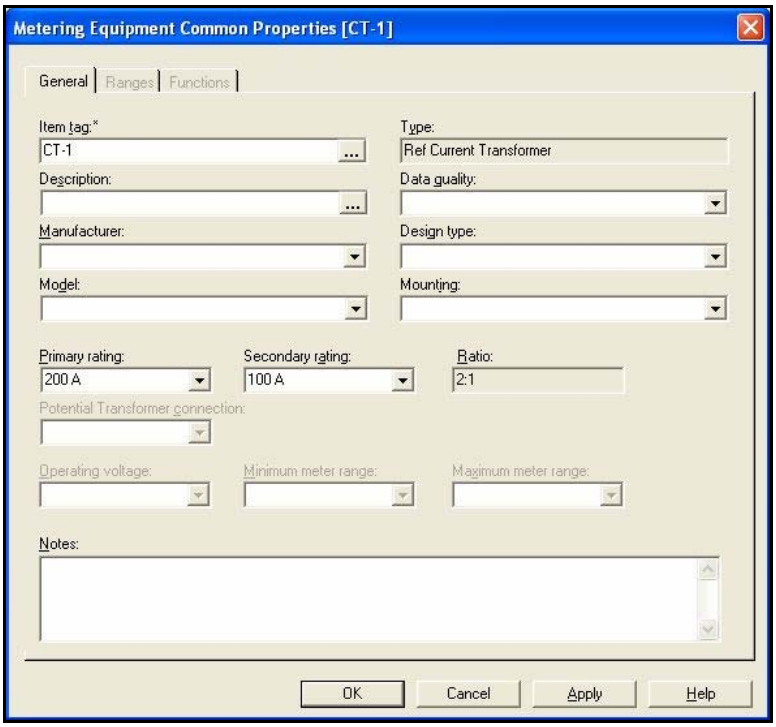
Protection Relays

Create typical stand alone metering equipment

Use the following procedure to create typcial stand alone CT:
1.From RDE, select Electrical Equipment, Power Distribution Equipment, Disconnect Electrical Equipment, Metering Equipment,Current Transformer
2.Right Click on the Current Transformer folder, select New Current Transformer.
3. Name the new current transformer 'CT-1'

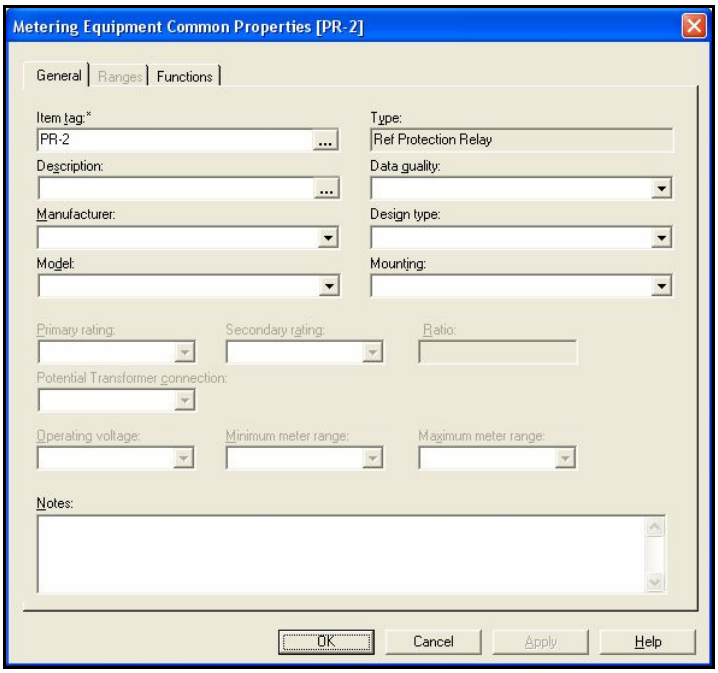


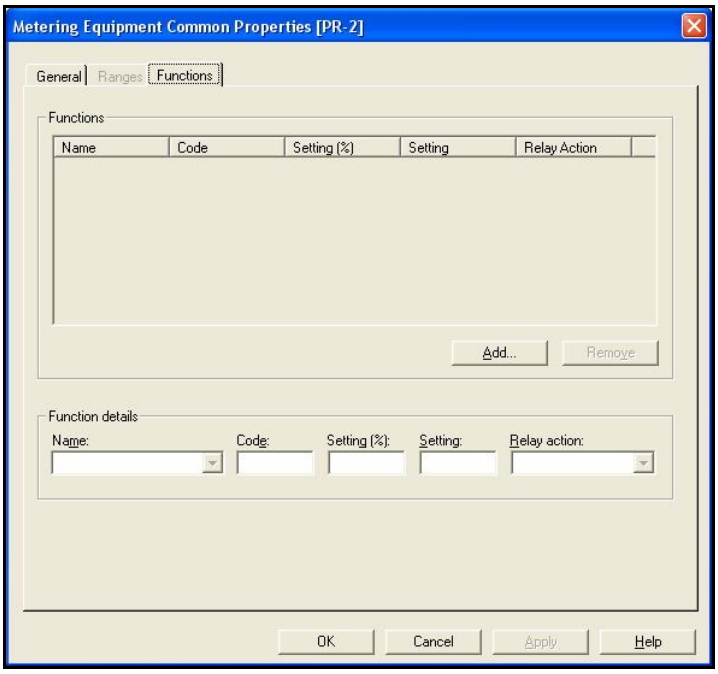
To open typcial current tansformer common properties, select the current transformer 'CT-1', right click and select Common properties.



Use the following procedure to create typcial stand alone Protection Relay:
1.From RDE, select Electrical Equipment, Power Distribution Equipment, Disconnect Electrical Equipment, Metering Equipment,Protection Relay.
2.Right Click on the Protection Relay folder, select New Protection Relay.
3. Name the new Protection Relay 'PR-2'

To open typcial protection relay common properties, select the protection relay 'PR-2', right click and select Common properties

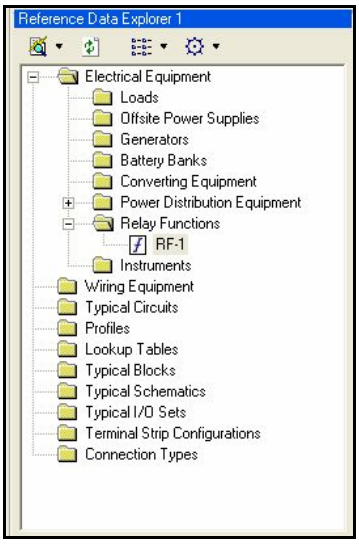


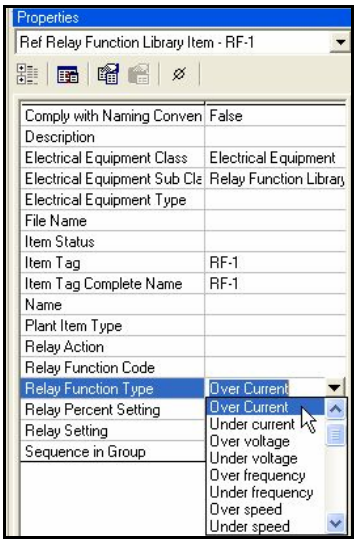


Create Relay function in RDE

Use the following procedure to create new typical relay function:

1. From RDE, select electrical equipment, relay function.
2.Right click on the relay function folder, select New Relay Function.
3. Name the new relay function ‘RF-1’

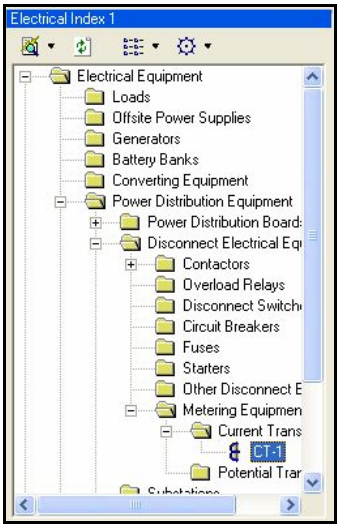




Create stand alone project Metering Eqipment

Use the following procedure to create stand alone project CT:

1.From RDE select the current transformer ‘CT-1’ and copy it to EI, under Electrical Equipment, Power Distribution Equipment, Disconnect Electrical Equipment, Metering Equipment, current transformer.
2.Name the new current transformer ‘CT-1’



Create project Metering Eqipment in EI.

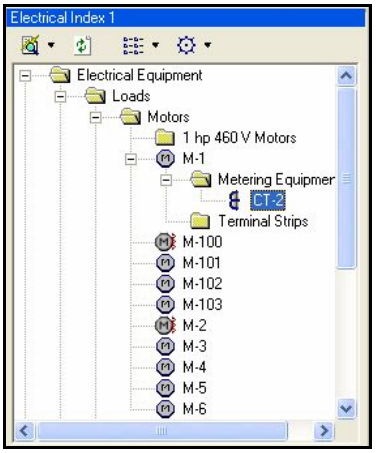
You can create project metering equipment under the following items

- **Loads**
- **Offsite Power Supplies**

- **Generators**
- **Battery Banks**
- **Converting Equipment**
- **Power Distribution Boards**
- **Transfer Switches**
- **Buses**
- **Cells**
- **Circuits**
- **Substations**

Use the following procedure to create a project CT:

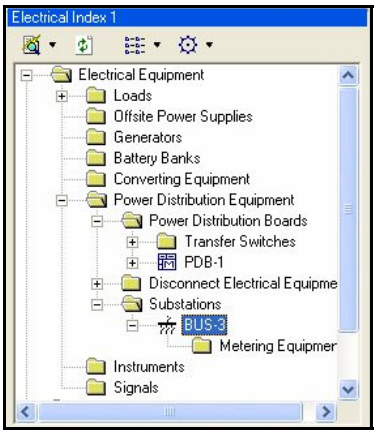
1.From RDE select the typical stand alone current transformer 'CT-1' and copy it to EI, under Electrical Equipment, Loads, Motors, M-1, Metering Equipment folder.
2.Name the new current transformer 'CT-2'



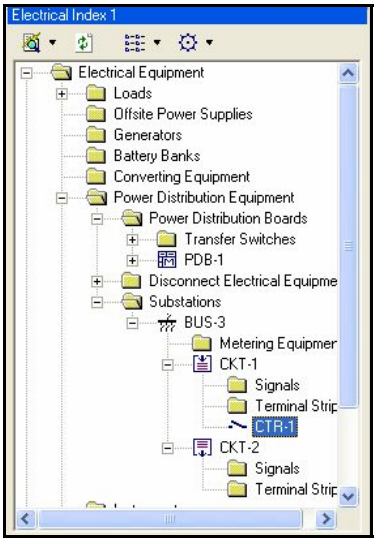
Substations

In most process plants the power to the process units passes through a high or medium voltage substation. This type of design is carried out by companies such as Areva, Alstom, ABB, Siemens, PowerLink, that in many of the cases are equipment manufacturers, designer and suppliers of power system generation and distribution. Smart Plant Electrical supports creating free standing buses that will simulate the actual situation in substations.

Use the following procedure to create a new substation in EI
1. In EI, select Electrical Equipment, Power Distribution Equipment, Substation.
2. Right Click on substations folder and select New Bus
3. Rename the new substation and call it 'Bus-3'



Stand-alone buses can be parent items of circuits and circuit internals, but not cells.

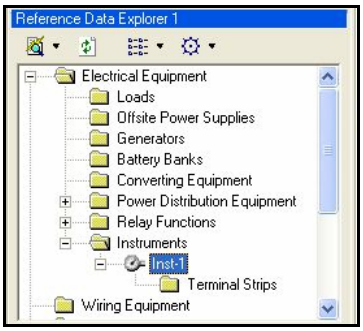


Instruments

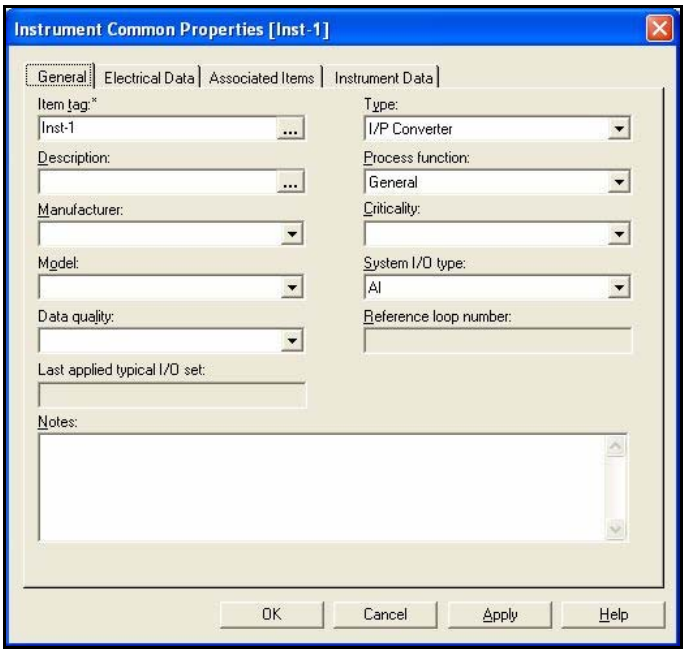
Instruments can be created in SmartPlant Electrical and associated with main equipment, such as motors or generators.

Use the following procedure to create a new typical instrument in RDE:

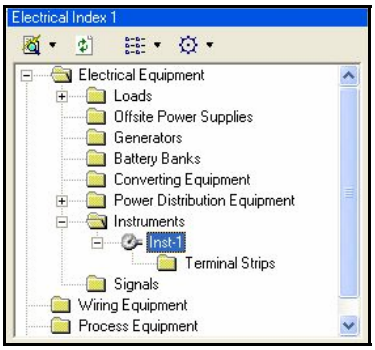
1. In RDE, select Electrical Equipment, Instrument.
2. Right Click on instruments folder and select New Instrument
3. Rename the new instrument and call it ‘Inst-1’



To open the instrument common properties, select instrument 'Inst-1', right click
And select common properties.

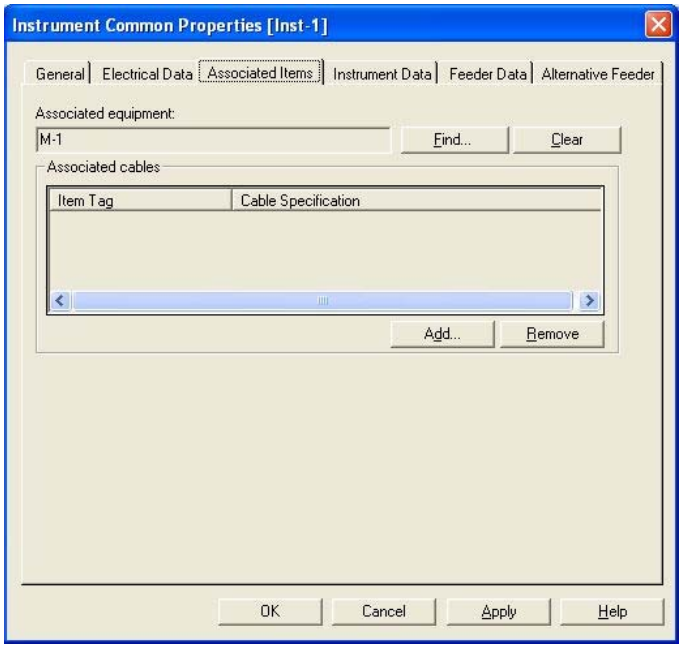


Use the following procedure to create a new project instrument in EI:
1.From RDE select the instrument 'Inst-1' and copy it to EI, under Electrical Equipment, Instrument
2.Name the new instrument 'Inst-1'

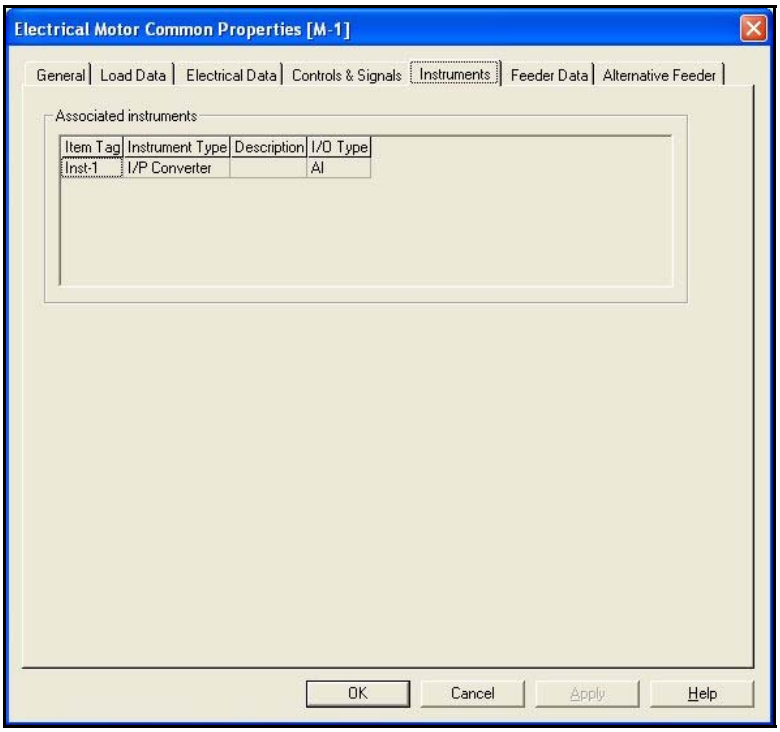


Use the following procedure to associate instrument with an equipment in EI:

1. Open the instrument common properties.
2. Select the associated items tab.
3. Select find button to open the find windows.
4. Select motor from the equipment type and select the find now button.
5. From the result list select the motor ‘M-1’ and select OK.



The association between the instrument ‘Inst-1- and motor ‘M-1’ also can be seen from the moror ‘M-1’ common properties, instrument tab.

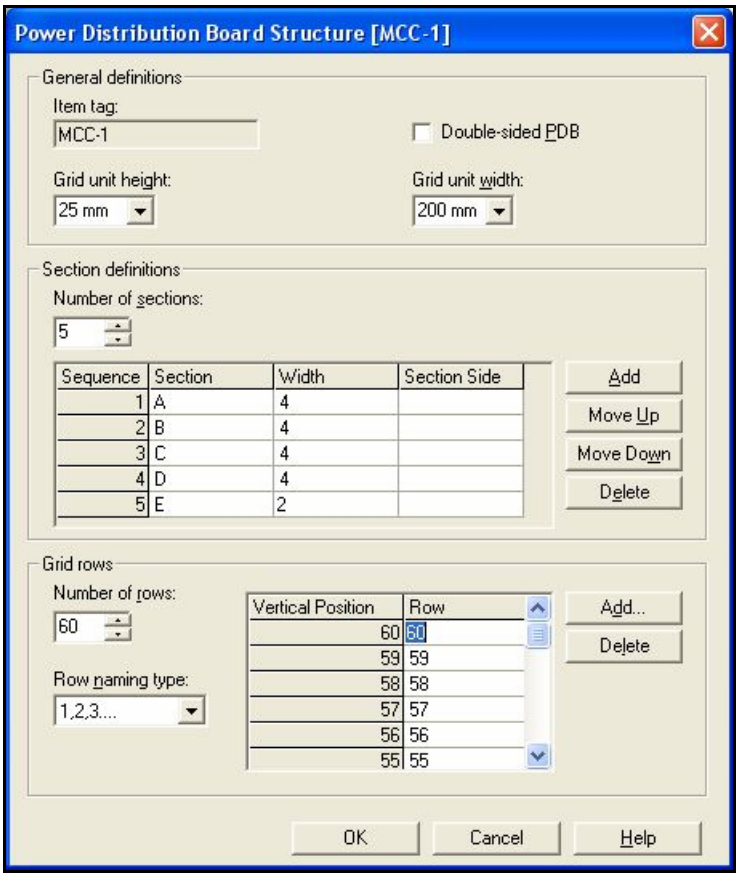


Design the PDB structure

SmartPlant Electrical provides you with tools for the design of power distribution boards. You can define a PDB structure when creating a PDB. Then, after creating a new PDB layout drawing, you drag a PDB to the drawing thus creating the structure (a grid) that was defined for that PDB. You can then place the cells in the drawing by dragging the cells from the Electrical Index to the appropriate location in the layout grid.

Create MCC and define its structure:

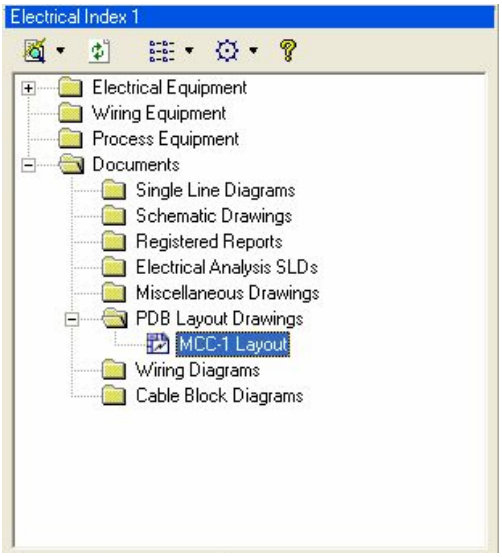
From EI create new MCC, name it MCC-1.
Select MCC-1, right click and select Design PDB Structure.
Enter the following data.



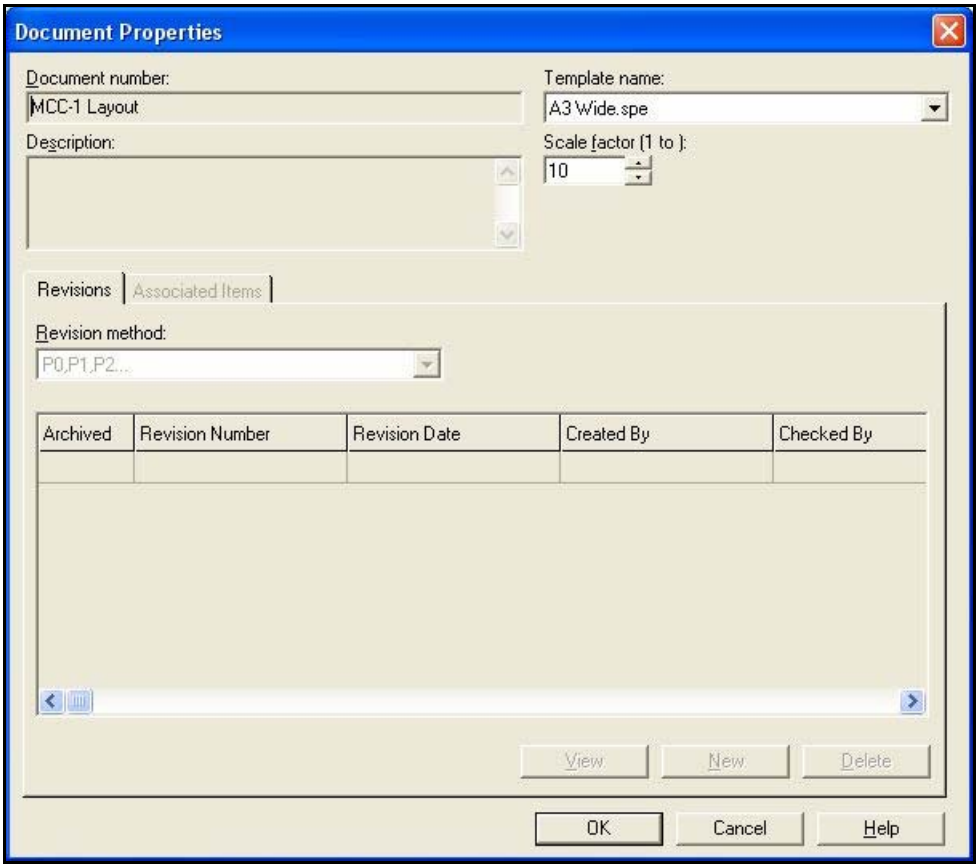
Click OK to confirm and close the form.

View the structure of MCC-1 in a drawing

Right click on Document, PDB Layout Drawings folder to create a new PDB layout drawing. Name it MCC-1 Layout.



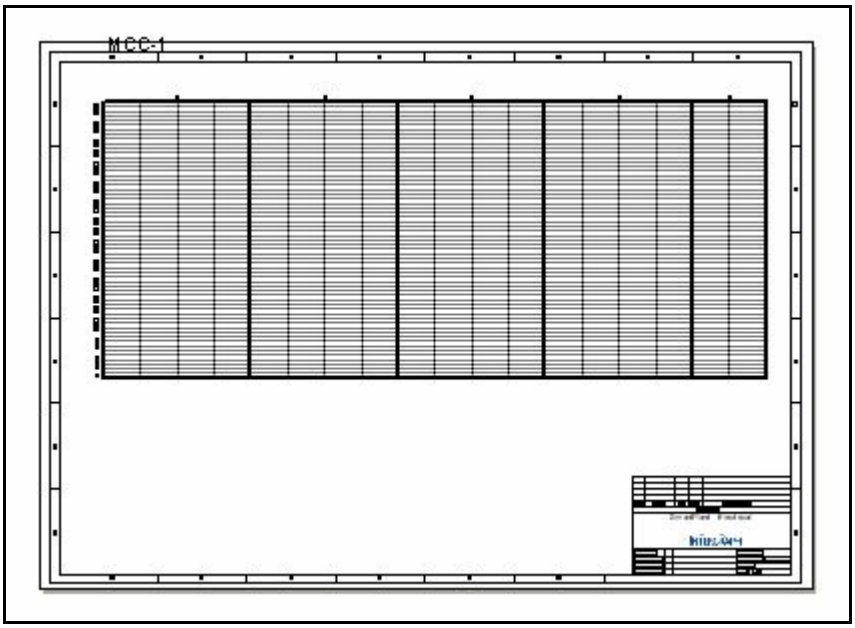
Right click on the drawing to open the Document properties dialog where you can define the template and the scale factor.



Click OK to confirm and close the form.

Right click on the MCC-1 Layout drawing, and select open to open the drawing.

Drag MCC-1 to the open drawing. Place the cursor on the left upper side of the drawing.



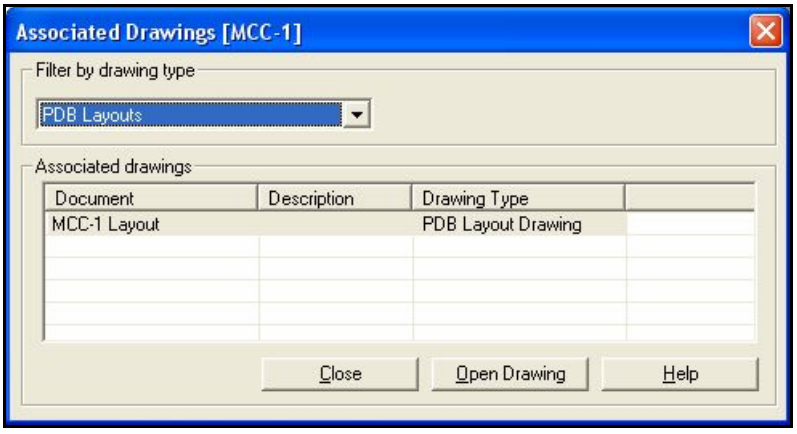
Save and close the drawing.

Viewing PDB layout drawings as related documents

Once the PDB layout has been created as a document it will appear as a associated document.

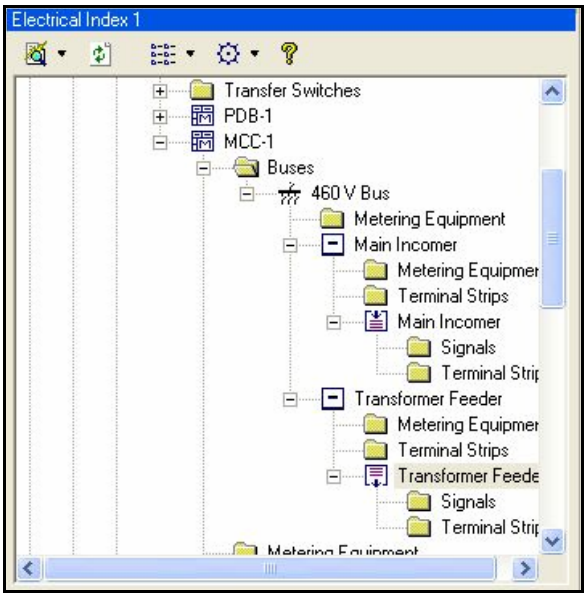
Right click on MCC-1, and select associated drawings.

Select either All or PDB Layouts:



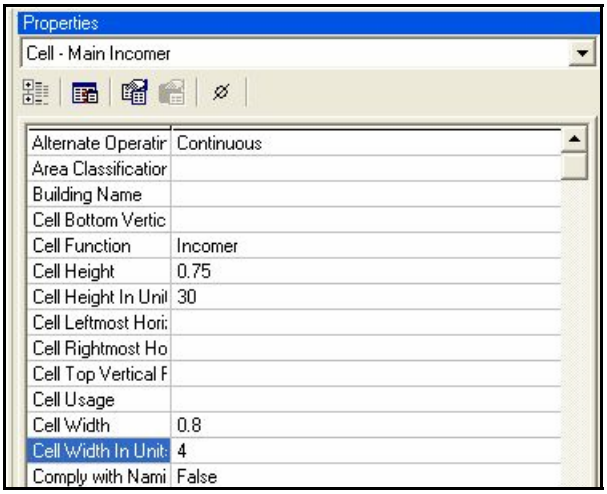
Creating cells

Lets us create one incomer cell and circuit, and one transformer cell and circuit



Let us define now the size of these cells, simply by defining the number of height units and width units each of them require (normally this is based on the connected equipment or vendor information of the circuit internals)

For the Main Incomer and the transformer we will assume 30 units' height and 4 units wide (half a section).

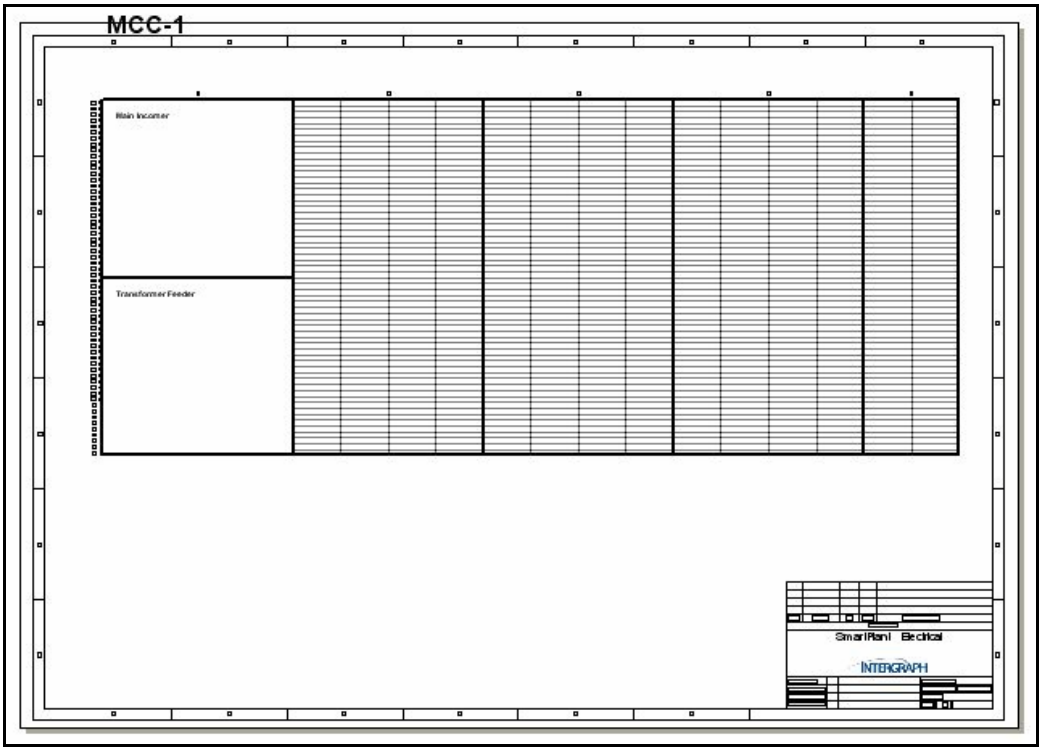


As you see, the cell height was calculated to be 0.75 meters (30 times 25mm; the values are in SI units) and the cell width was calculated to 0.8 meters (4 times 200mm)

Open the drawing to place and position these 2 cells.

First, zoom in to the leftmost upper part of the section A and allocate the Incomer cell as shown in the screen bellow. When dragging the cell allow some time until you see the cell attached to the dynamic

cursor and only then drop the cell. The upper-left most part of the cell is the positioning target. Place it to the starting point at the PDB structure .
In the same way place the Transformer Feeder cell at the lower part of section A.



Close the drawing and save the change.

Lab 3

Create the following typical items in RDE:

Typical Circuits

3.1. '**SWGR Incomer 4.16 kV**' (Rated Voltage = 4.16 kV)
'**F-1**' (Current Rating = 120 A, Rated Voltage = 4.16 kV)
'**DS-1**' (Current Rating = 120 A, Rated Voltage = 4.16 kV)
3.2. '**SWGR Feeder 4.16 kV**' (Rated Voltage = 4.16 kV)
'**DS-1**' (Current Rating = 120 A, Rated Voltage = 4.16 kV)
'**PR-1**' (Current Rating = 120 A, Rated Voltage = 4.16 kV)
3.3. '**MCC Incomer 120 V**' (Rated Voltage = 120 V)
'**F-1**' (Current Rating = 120 A, Rated Voltage = 120 V)
'**DS-1**' (Current Rating = 120 A, Rated Voltage = 120 V)
3.4. '**MCC Incomer 480 V**' (Rated Voltage = 480 V)
'**DS-1**' (Current Rating = 120 A, Rated Voltage = 480 V)
'**PR-1**' (Current Rating = 120 A, Rated Voltage = 480 V)
3.5. '**Motor Feeder 1 hp 460 V**'
'**F-1**' (Current Rating = 1 A, Rated Voltage = 480 V)
'**DS-1**' (Current Rating = 120 A, Rated Voltage = 480 V)

3.6. '**Motor Feeder 10 hp 460 V**'
'**F-1**' (Current Rating = 12 A, Rated Voltage = 480 V)
'**CB-1**' (Current Rating = 16 A, Rated Voltage = 480 V)
'**OL-1**' (Current Rating = 16 A,Rated Voltage = 480 V, Control Voltage = 24 V)
3.7. '**Motor Feeder 100 hp 460 V**' (Rated Voltage = 480 V)
'**DS-1**' (Current Rating = 120 A, Rated Voltage = 480 V)
'**CB-1**' (Current Rating = 16 A, Rated Voltage = 480 V)
3.8. '**Heater Feeder 100 W 120 V**' (Rated Voltage = 120 V)
'**CB-1**' (Current Rating = 0.9 A, Rated Voltage = 120 V)
3.9. '**Heater Feeder 100 kW 480 V**'
'**CB-1**' (Current Rating = 180 A,Rated Voltage = 480 V)
3.10. '**Bus Tie 480 V Coupler**'
'**DS-1**' (Current Rating = 120 A,Rated Voltage = 480 V)
3.11 '**Bus Tie 480 V Riser**'
'**DS-1**' (Current Rating = 120 A,Rated Voltage = 480 V)

Typical PDB's

3.12.'**SWGR 4.16 kV Bus**' (Electrical Equipment Type = Switchgear)
Bus Name = '**4.16 kV Bus**' (Rated Voltage = 4.16 kV)
3.13. '**MCC 120 V Bus A, 480 V Bus B/C**' (Electrical Equipment Type = Motor Control Center)
Bus 1 Name = '**120 V Bus A**' (Rated Voltage = 120 V)
Bus 2 Name = '**480 V Bus B**' (Rated Voltage = 480 V)
Bus 3 Name = '**480 V Bus C**' (Rated Voltage = 480 V)

Create the following project items in EI:

Project PDB's

3.14. a. '**SW-300**', by copy typcial PDB, 'SWGR 4.16 kV Bus'.
b. Add new Cell, '**F1A**', under the new PDB, 'SW-300', Bus, '4.16 kV Bus',
c. Create new incomer Circuit, '**SW-300 Incomer**', in the new Cell, 'F1A', by copying typical circuit,'SWGR Incomer 4.16 kV'.
d. Add new Cell, '**F2A**', under the new PDB, 'SW-300', Bus '4.16 kV Bus',
e. Create new feeder Circuit, '**SW-300 Feeder**', in the new Cell, 'F2A', by copying typical circuit,' SWGR Feeder 4.16 kV'.
f. Add new Cell, '**F2B**', under the new PDB, 'SW-300', Bus '4.16 kV Bus',
g.Create new feeder Circuit, '**SW-300 Feeder Spare**', in the new Cell, 'F2B', by copying typical circuit,' SWGR
Feeder 4.16 kV'.
3.15. a. '**MCC-201**', by copying typical PDB, 'MCC 120 V Bus A, 480 V Bus B/C'
b. Add new Cell, '**F1A**', in the new PDB, 'MCC-201', Bus, '120 V Bus A'.
c. Create new incomer Circuit, '**MCC-201 Incomer A**' , in the new Cell, 'F1A', by copying typical circuit, 'MCC Incomer 120 V'.
d. Add new Cell, '**F2A**', in the new PDB, 'MCC-201', Bus, '120 V Bus A'.
e. Create new feeder Circuit, '**SH-100 Feeder**', in the new cell, 'F2A', by copying the typical circuit,'Heater Feeder 100 W 120 V'.
f. Add new Cell, '**F1B**', in the new PDB, 'MCC-201', Bus, '480 V Bus B'.
g.Create new incomer Circuit, '**MCC-201 Incomer B**' , in the new Cell, 'F1B', by copying typical circuit, 'MCC Incomer 480 V'.
h. Add new Cell, '**F2B**', in the new PDB, 'MCC-201', Bus, '480 V Bus B'.
i. Create new feeder Circuit, '**M-100 Feeder**', in the new cell, 'F2B', by copying the typical circuit,' Motor Feeder 1 hp 460 V'.
j. Add new Cell, '**F2C**', in the new PDB, 'MCC-201', Bus, '480 V Bus B'.
k.Create new feeder Circuit, '**M-101 Feeder**', in the new cell, 'F2C', by copying the typical circuit,' Motor Feeder 10 hp 460 V'.
l. Add new Cell, '**F2D**', in the new PDB, 'MCC-201', Bus, '480 V Bus B'.

m. Add new feeder Circuit, '**Motor Spare Feeder**', in the new cell, 'F2D'.
n. Add new Cell, '**F2E**', in the new PDB, 'MCC-201', Bus, '480 V Bus B'.
o. Create new coupler Circuit, '**MCC-201 Coupler**', in the new cell, 'F2E', by copying the typical circuit,' Bus Tie 480 V Coupler'.
p. Add new Cell, '**F1G**', in the new PDB, 'MCC-201', Bus, '480 V Bus C'.
q. Create new incomer Circuit, '**MCC-201 Incomer C**' , in the new Cell, 'F1G', by copying typical circuit, 'MCC Incomer 480 V'.
r. Add new Cell, '**F2G**', in the new PDB, 'MCC-201', Bus, '480 V Bus C'.
s. Create new riser Circuit, '**MCC-201 Riser**', in the new cell, 'F2G', by copying the typical circuit, ' Bus Tie 480 V Riser.
t. Add new Cell, '**F2H**', in the new PDB, 'MCC-201', Bus, '480 V Bus C'.
u. Create new feeder Circuit, '**M-102 Feeder**', in the new cell, 'F2H', by copying the typical circuit, ' Motor Feeder 100 hp 460 V'.
v. Add new Cell, '**F2I**', in the new PDB, 'MCC-201', Bus, '480 V Bus C'.
w. Create new feeder Circuit, '**M-103 Feeder**', in the new cell, 'F2I', by copying the typical circuit,' Motor Feeder 10 hp 460 V'.
x. Add new Cell, '**F2J**', in the new PDB, 'MCC-201', Bus, '480 V Bus C'.
y. Create new feeder Circuit, '**H-104 Feeder**', in the new cell, 'F2J', by copying the typical circuit, 'Heater Feeder 100 kW 480 V'.

3.16. '**MCC-202**' by duplicating 'MCC-201'.

Rename all items in MCC-202 according to the One Line Diagram.

Transfer Switch

3.17. **'TS-001'**
'Bus Name = '**4.16 kV**'
Incomer 1 = '**TS-001 Incomer 1'**
Incomer 2 = '**TS-001 Incomer 2'**

.

Chapter 4

Wiring Equipment

SmartPlant Electrical has wiring capability, that allows you create wiring equipments, manage wiring entities, and create wiring documents. The software also provides wiring termination connection capability, and producing wiring connection diagrams, and cable block diagrams.

The software allows to do the following tasks:

Create cables,bus ways,cable drums,panels (cabinets,local panels,junction boxes),control stations.
Create i/o signals.
Create terminal strips for all types of applicable equipment.
Create terminals on terminal strips.
Manage both cable sides.
Manage both conductor sides.
Create wiring drawings and make connections.
Generate termination reports.
Create cable block diagrams.

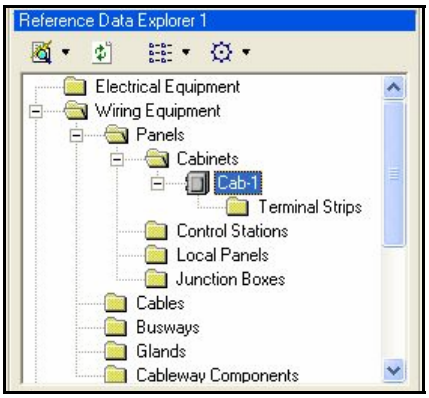
SmartPlant Electrical deals with equipment interconnections that are external to PDBs or main equipment item types. The software handles the internal circuit wiring in typical schematics. Cable terminations can reach and include equipment terminal strips.
After creating your wiring items and making the required terminations, you can generate cable wiring schedule and terminal strip connection reports.

Panels

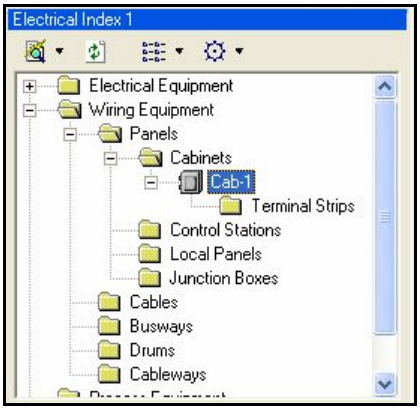
Cabinets

Serves as place holder for panels that will be retrieved as loads.
It has common properties form with 4 tabs, General, Electrical data, Feeder Data, and Alternative Feeder .
In the electrical tab you define the power consumption in Watts or Kilowatts and the system calculates the current as for any other load.
Cabinets can be connected to a bus under the feeder circuits as any other load.

Use the following procedure to create typical cabinet in RDE:
1. In RDE, select Wiring Equipment, Panels, Cabinets.
2. Right Click on Cabinets folder and select New Cabinet
3. Rename the new cabinet to ‘Cab-1’.



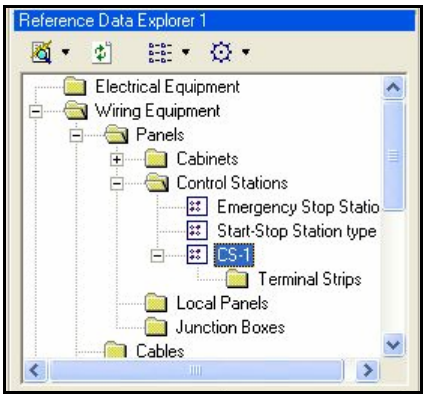
User the following procedure to create project cabinet in EI:
1. From RDE select cabinet, 'Cab-1', and copy it to EI, under Wiring Equipment, Panels, Cabinets folder.
2. Rename the new cabinet to 'Cab-1'.

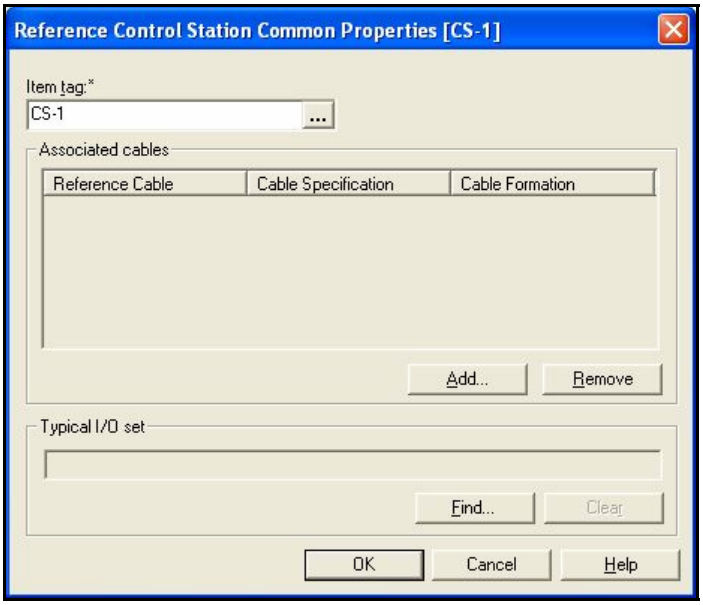


Control Stations

Local or remote panel that contains the control elements (start-stop push buttons) activated manually by the person operating the plant. You can create such control station with or without associated cables.
A control station can be associated to one or more cables (control, ground or instrumentation cable categories). Once a cable is associated to a control station, the control station becomes the "to" side reference of that cable.A control station can be associated to one load.
When the creation of the control station is a result of an automated process, the association is created as part of the automation process, both to the cable and to the load.
In case the control station is created manually, the user is responsible for these associations.

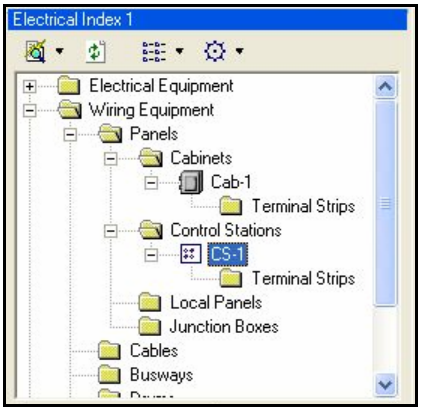
Use the following procedure to create typical control station in RDE:
1. In RDE, select Wiring Equipment, Panels, Control Stations.
2. Right Click on Control Stations folder and select New Control Station.
3. Rename the new control station to 'CS-1'

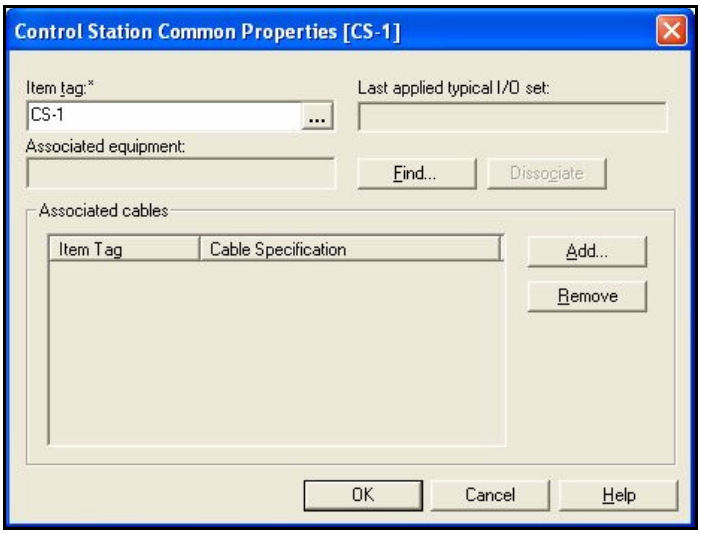




Use the following procedure to create project control station in EI:

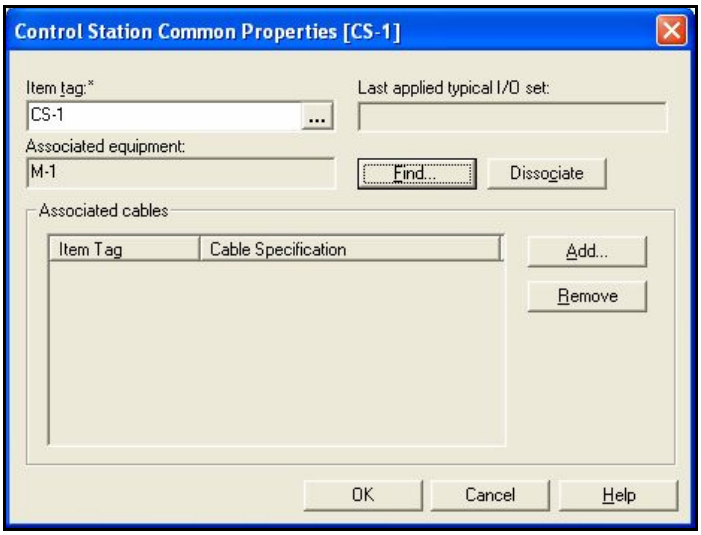
1. From RDE select control station, 'CS-1', and copy it to EI, under Wiring Equipment, Panels, Control Stations folder.
2. Rename the new control station to 'CS-1'.





Use the following procedure to associate equipment to control station in EI.

1.Open the control station common properties.
2.Select find and associate the motor 'M-1'



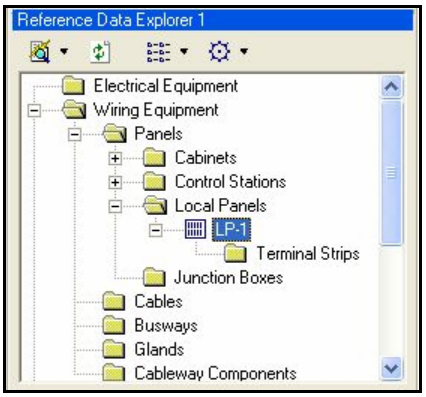
Local Panels and Junction Boxes

Local Panel

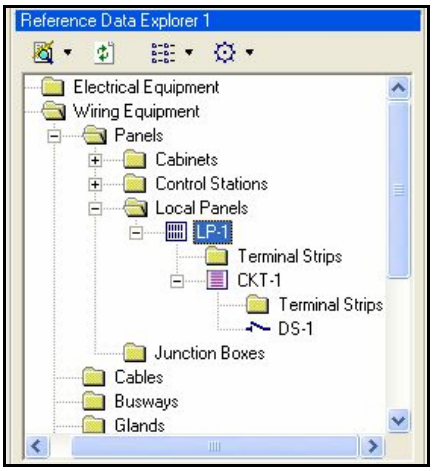
Cabinets containing circuit, equipment, and wiring items, used for example, to provide switching or isolation close to a motor.

Use the following procedure to create typical local panel in RDE:
1. In RDE, select Wiring Equipment, Panels, Local Panels.
2. Right Click on Local Panels folder and select New Local Panel.

3. Rename the new Local Panel to ‘LP-1’

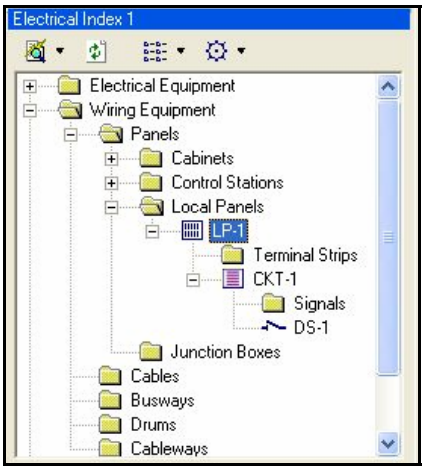


You can add circuit and circuit internals to the local panel.



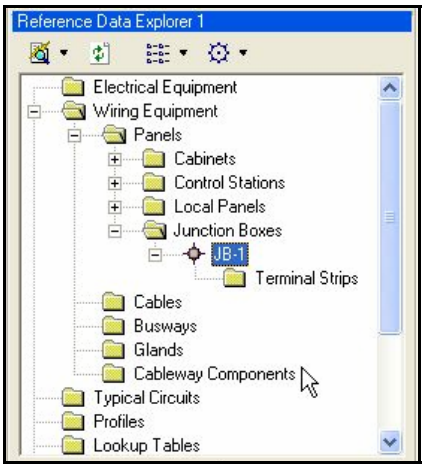
User the following procedure to create project local panel in EI:

1. From RDE select local panel, ‘LP-1’, and copy it to EI, under Wiring Equipment, Panels, Local Panels folder.
2. Rename the new local panel to ‘LP-1’

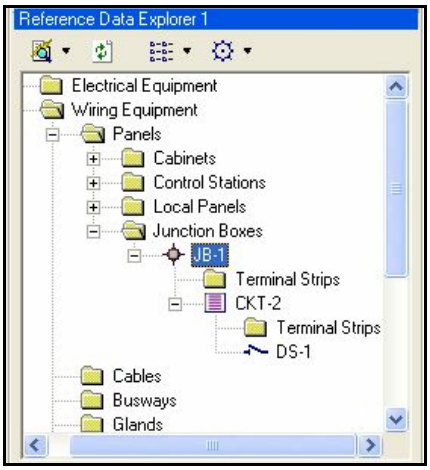


Junction Boxes
Cabinets used to connect wires that run between various items of electrical equipment.

Use the following procedure to create typical junction box in RDE:
1. In RDE, select Wiring Equipment, Panels, Junction Boxes.
2. Right Click on Junction Boxed folder and select New Junction Box.
3. Rename the new Junction Box to 'JB-1'.

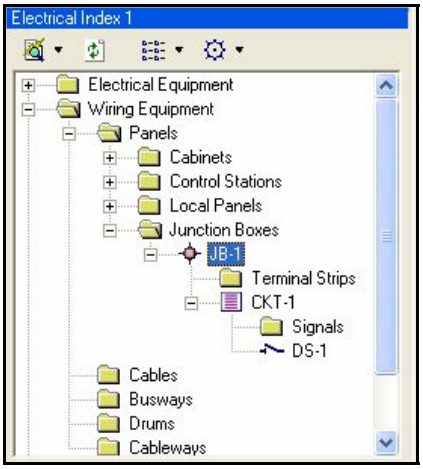


You can add circuit and circuit intetnals to junction box.



User the following procedure to create project junction box in EI:

1. From RDE select junction box, 'JB-1', and copy it to EI, under Wiring Equipment, Panels, Junction Boxes folder.
2. Rename the new junction box to 'JB-1'.

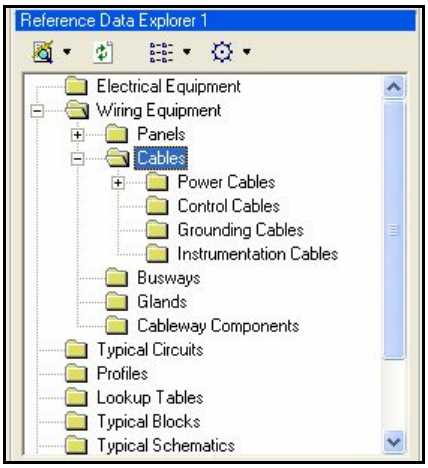


Note: In contrast to Power Distribution Boards (PDBs), local panels and junction boxes are much smaller in size and do not contain busbars.

Cables

Smart Plant Electrical specifies the following cable categories according to the purpose of the cable: Power, control, grounding, and instrumentation. The software supports compliance with the strict regulations governing cable definitions by using standard reference cables as a basis for all the cables that you create in your project. Reference cables contain technical cable data such as voltage rating, ampacity, resistivity, cable material, and data relating to the construction of the cable. For this reason, you must initially create all your reference cables in the **Reference Data Explorer**. The software allows you to organize your reference cables in families known as cable specifications. When you select a reference cable to use as a basis for your project cable, you must select the appropriate cable category and cable

specification for the reference cable. Also, when you size project cables, the software looks for the appropriate reference cable based on a particular cable specification and conductor arrangement.



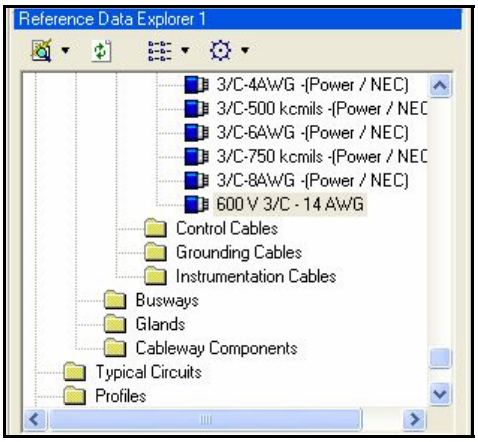
Power Cable Category

Cables of various sizes, construction and insulation, single or multi-conductor, designed to distribute primary power to various types of equipment .

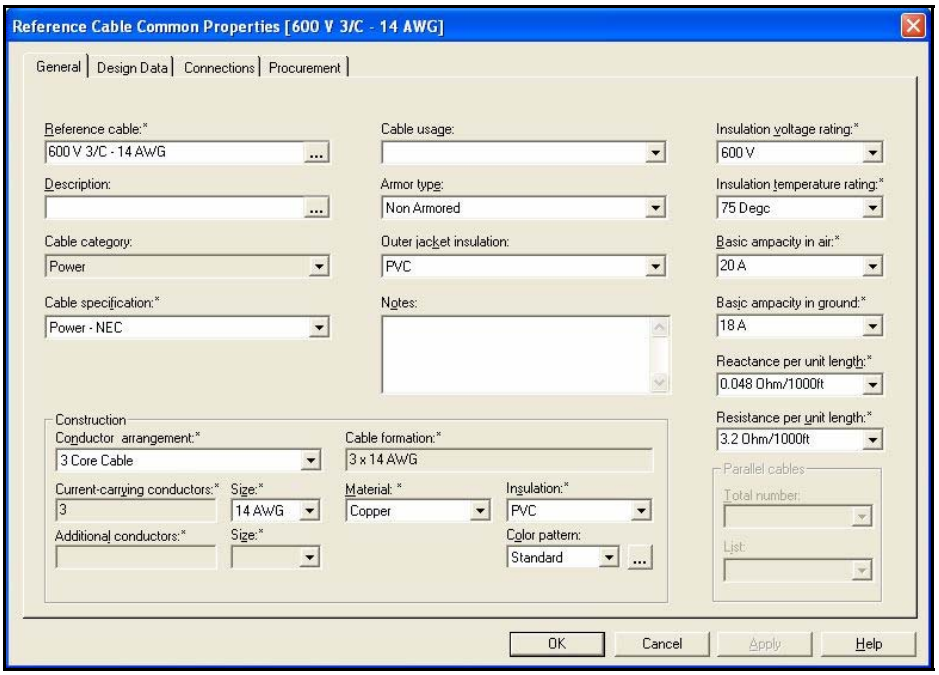
Reference Power Cable

Use the following procedure to create typcial power cable in RDE:

1. In RDE, select Wiring Equipment, Cables, Power Cables.
2. Right Click on Power Cables folder and select New .
3. Rename the new cable to '600 V 3/C - 14 AWG'.



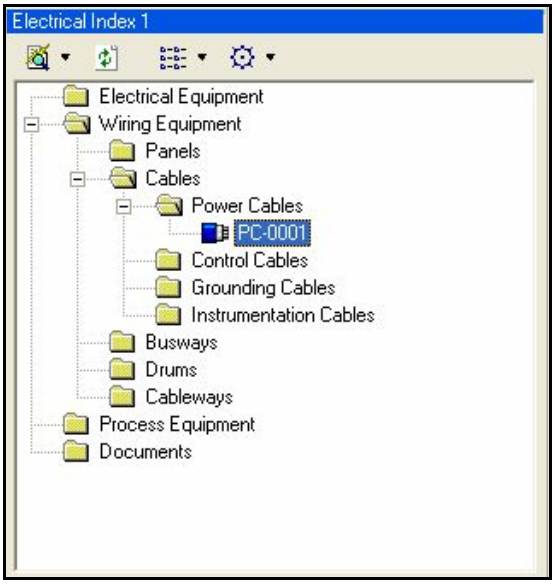
Edit the cable properties on the Reference Cable Common Properties dialog box according to the available catalog information, including conductor arrangement, size, material and electrical cable data.



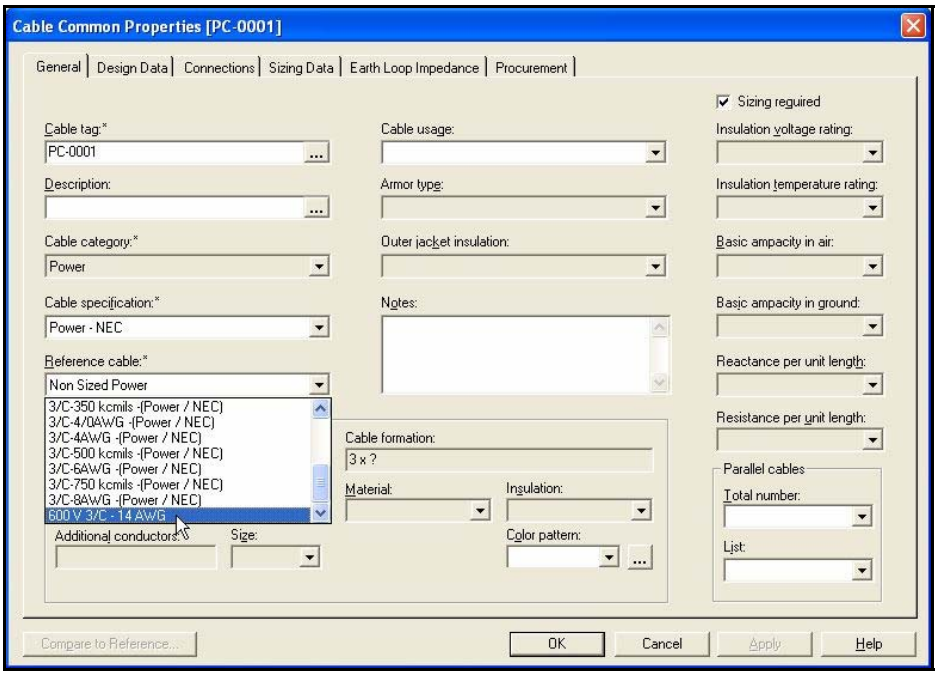
Project Power Cable

Use the following procedure to create project power cable in EI:

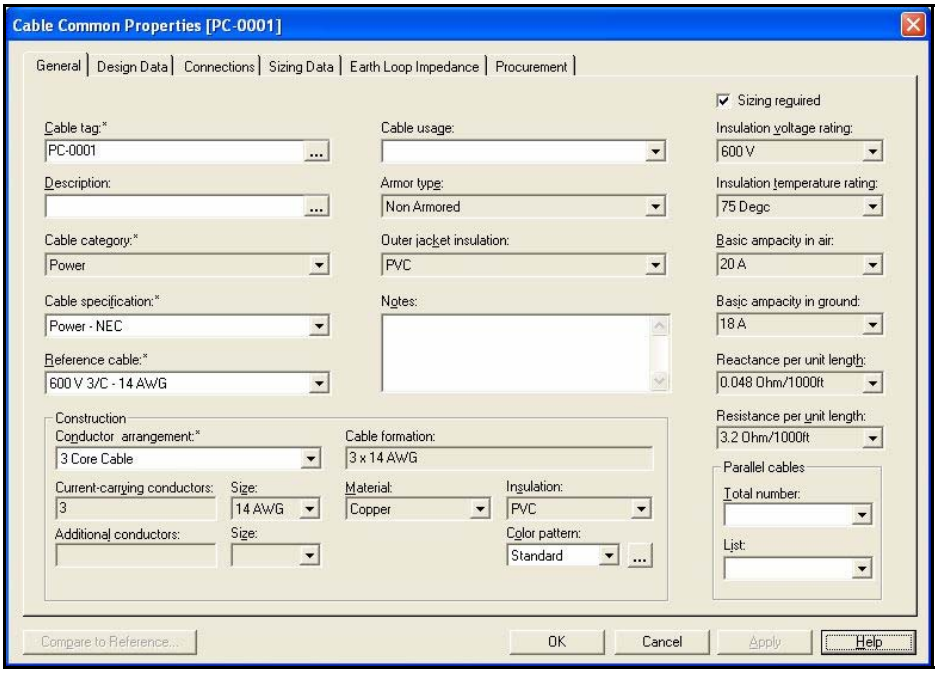
1. In EI, select Wiring Equipment, Cables, Power Cables.
2. Right Click on Power Cables folder and select New .
3. Rename the new cable to 'PC-0001'.



4. Open the cable common properties window, general tab, and select cable specification, Power- NEC, and conductor arrangement, 3 Core Cable.
   From the Reference cable select list pick a reference cable you would like to copy from.

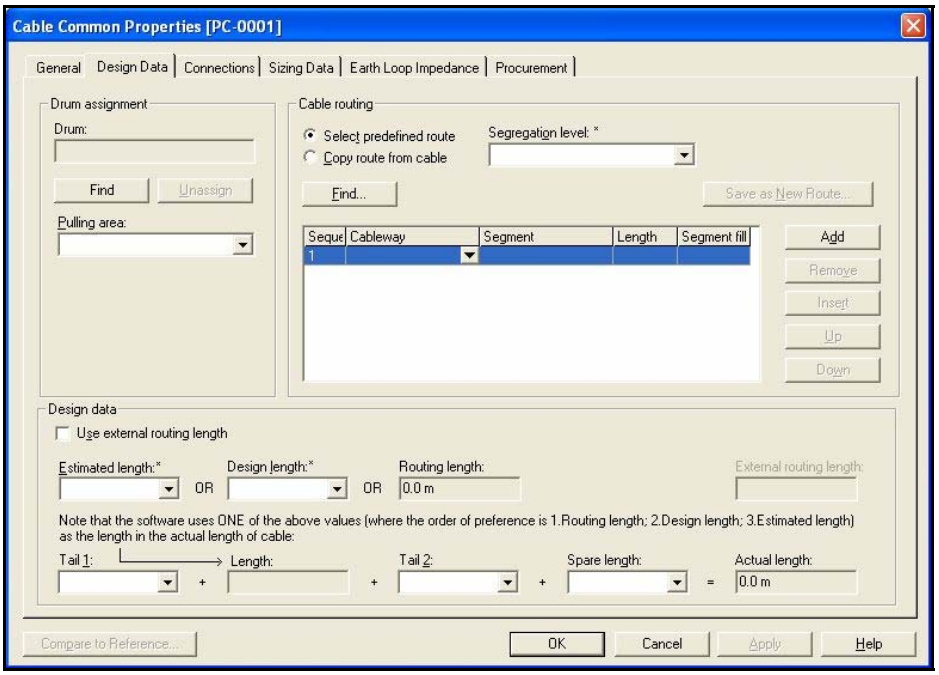


5.Selecting the reference cable 600 V 3/C – 14 AWG populate the values from the reference cable To the project cable.

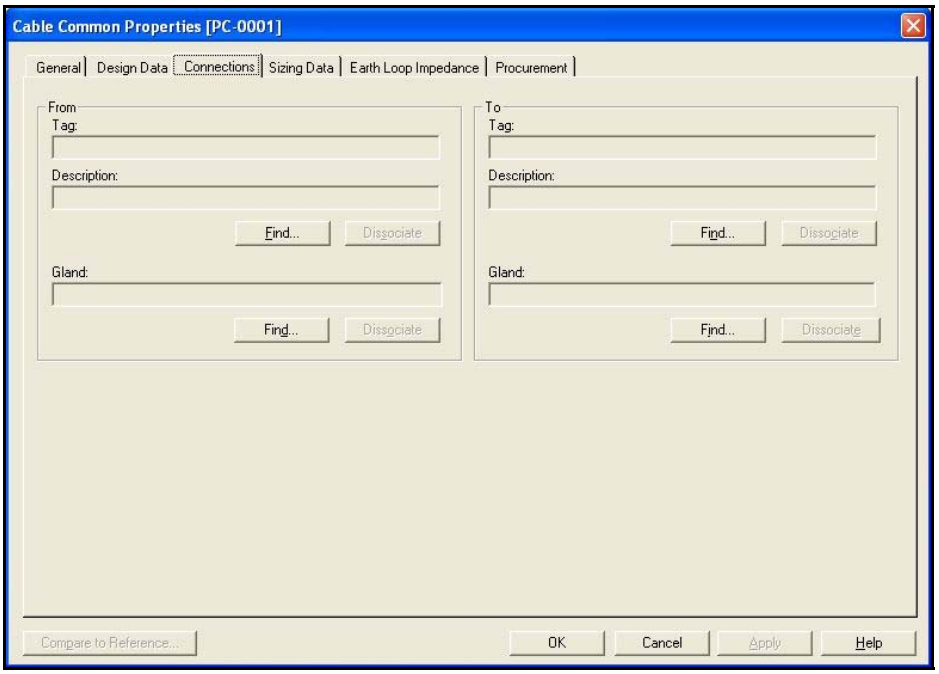


Note: 1. You can leave the default 'Non Sized Cable' for reference cable, fill in the values for cable construction manually, and select a reference cable later.
2.Asterisk indicates field required for cable sizing calculation.

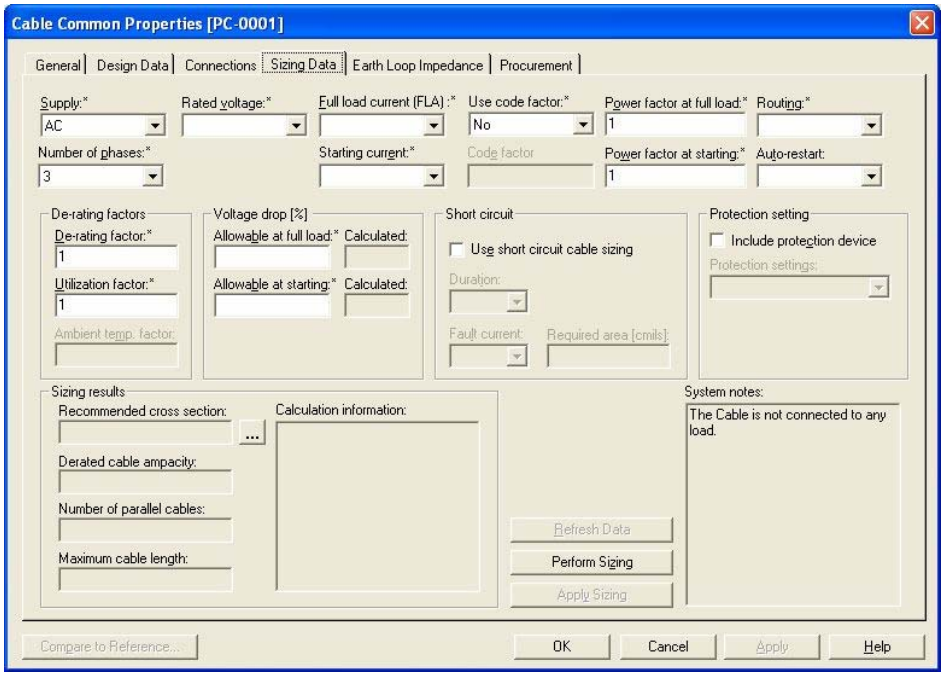
Design Data tab



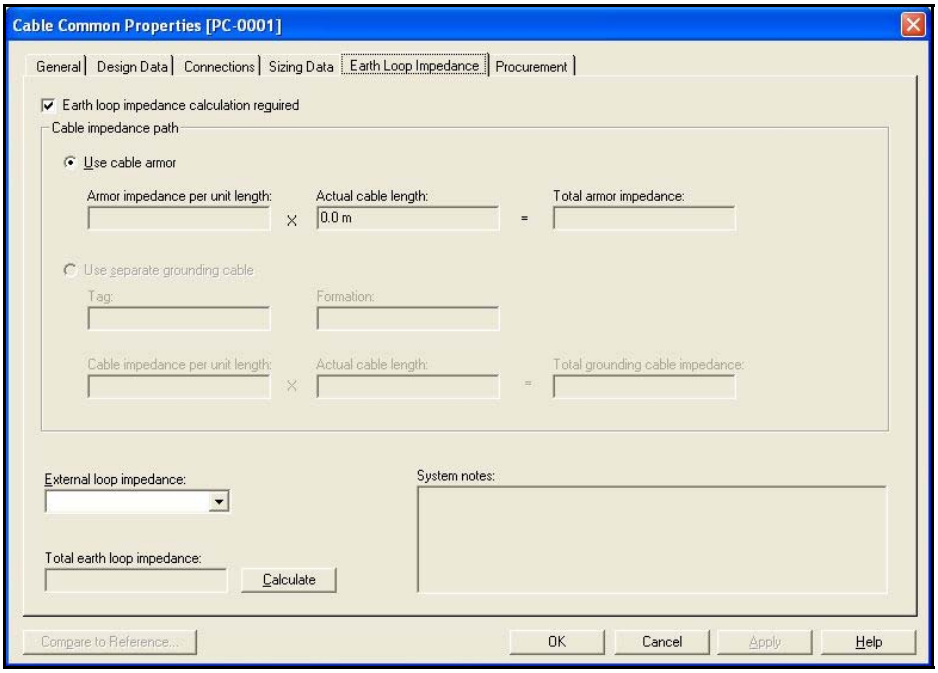
Connection tab



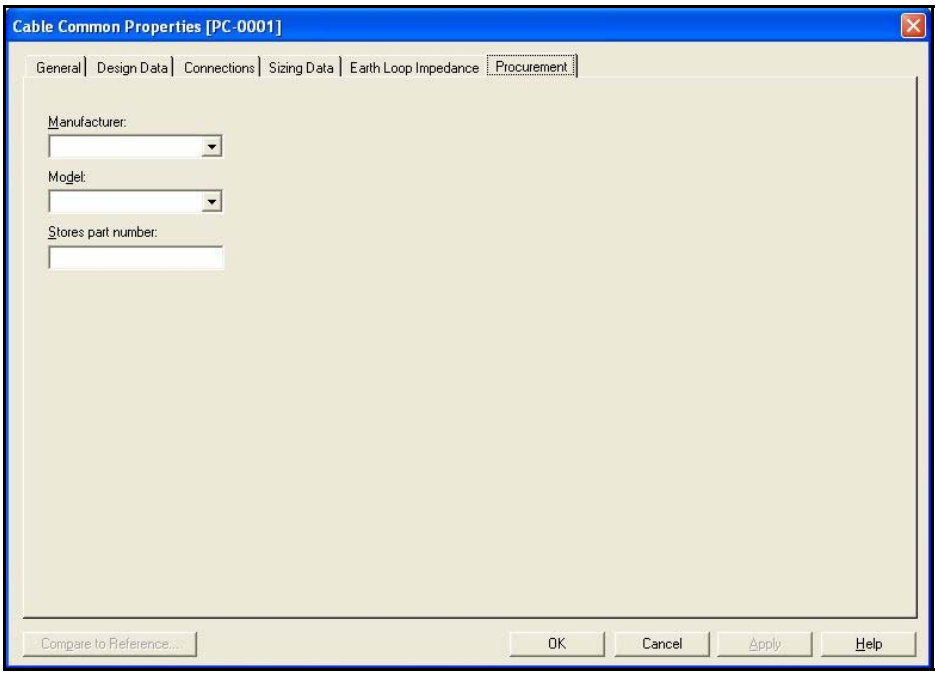
Sizing Data tab



Earth Loop Impedance tab



Procurement tab



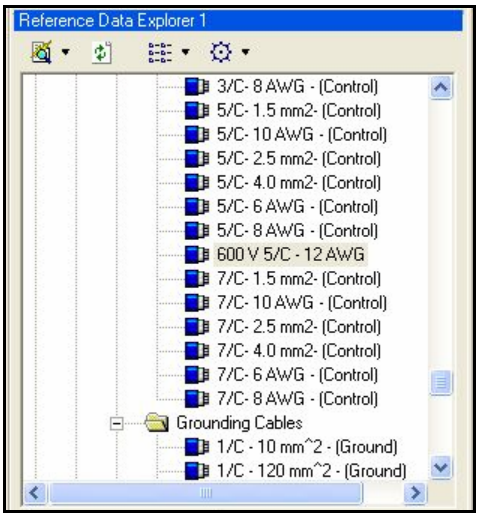
Control Cable Category

A multi-conductor cable made for operation in control or signal circuits. Cables in this category do not need sizing calculation.

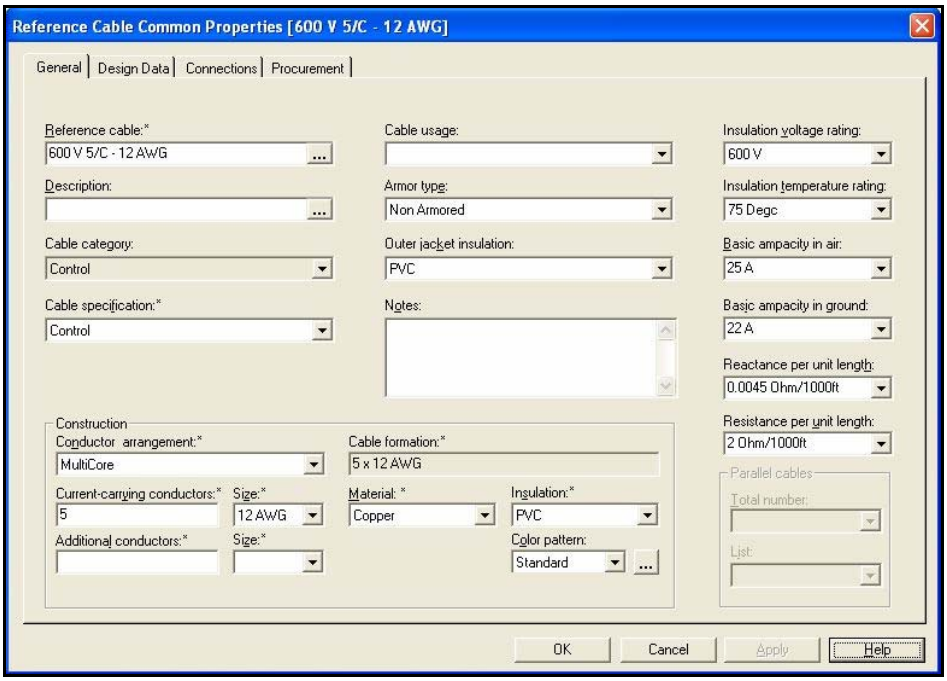
Reference Control Cable

Use the following procedure to create typcial control cable in RDE:

1. In RDE, select Wiring Equipment, Cables, Control Cables.
2. Right Click on Control Cables folder and select New .
3. Rename the new cable to '600 V 5/C - 12 AWG'.



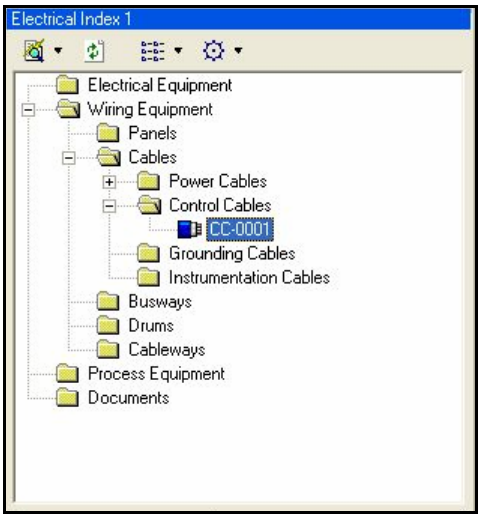
Edit the cable properties on the Reference Cable Common Properties dialog box according to the available catalog information, including conductor arrangement, size, material and electrical cable data.



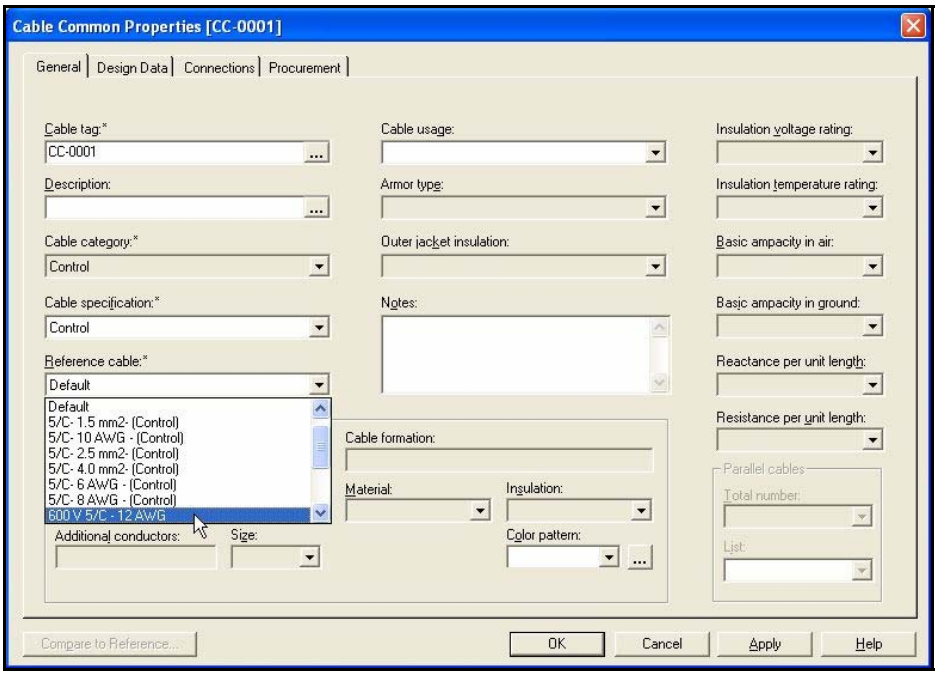
Project Control Cable

Use the following procedure to create project control cable in EI:

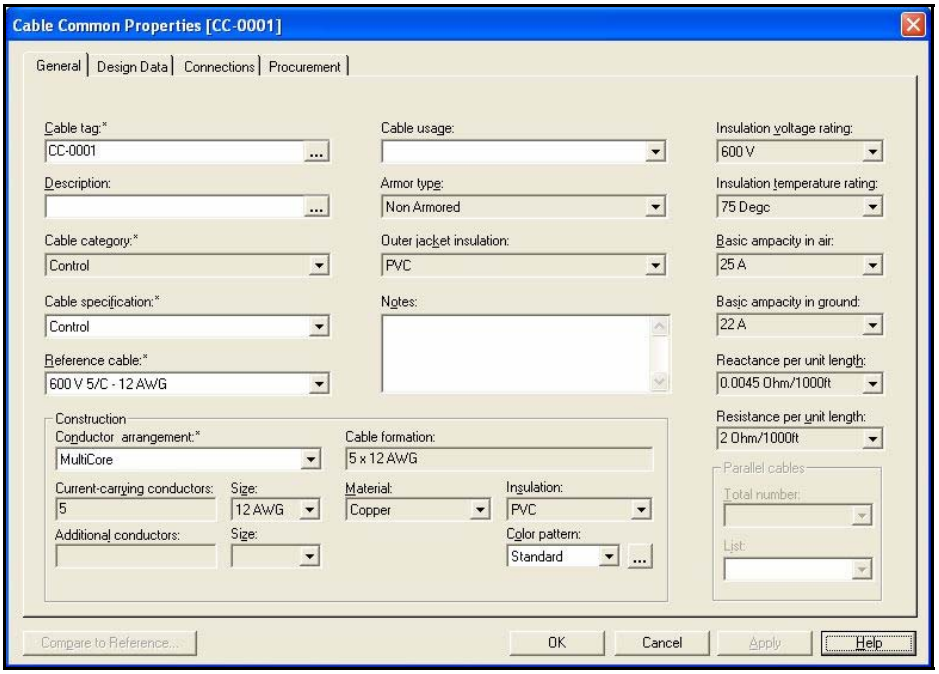
1. In EI, select Wiring Equipment, Cables, Control Cables.
2. Right Click on Control Cables folder and select New .
3. Rename the new cable to 'CC-0001'.



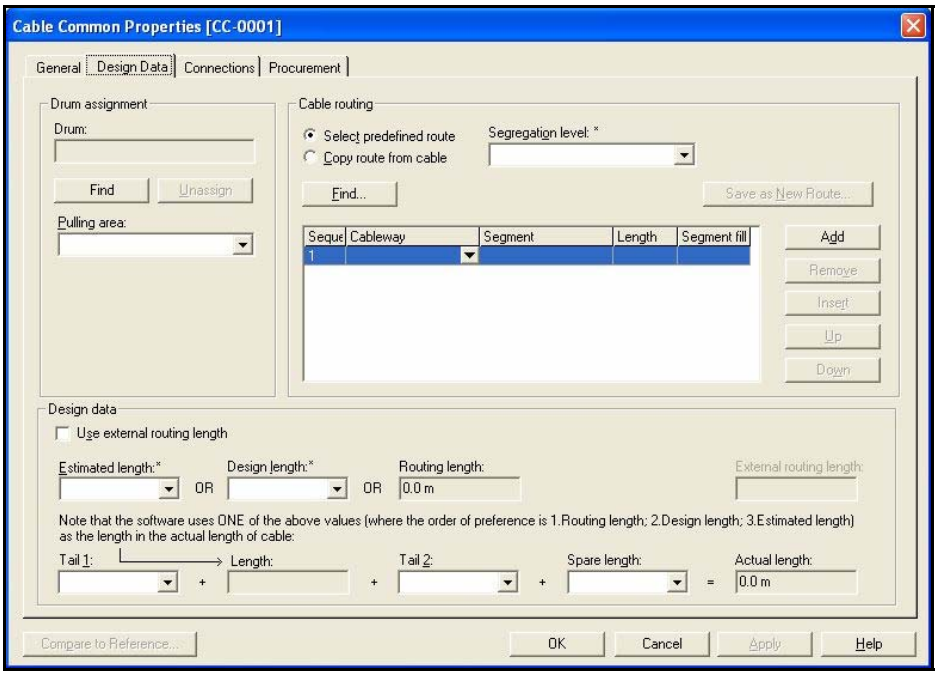
4. Open the cable common properties window, general tab, and select cable specification, Control, and conductor arrangement, MultiCore Cable.
   From the Reference cable select list pick a reference cable you would like to copy from.



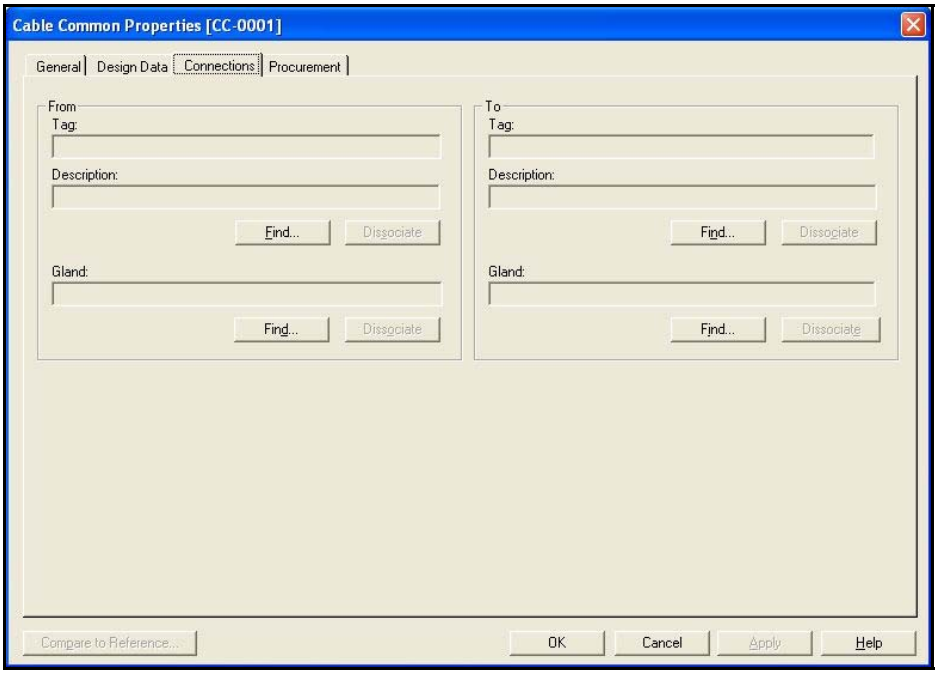
5.Selecting the reference cable 600 V 5/C – 12 AWG populate the values from the reference cable To the project cable.



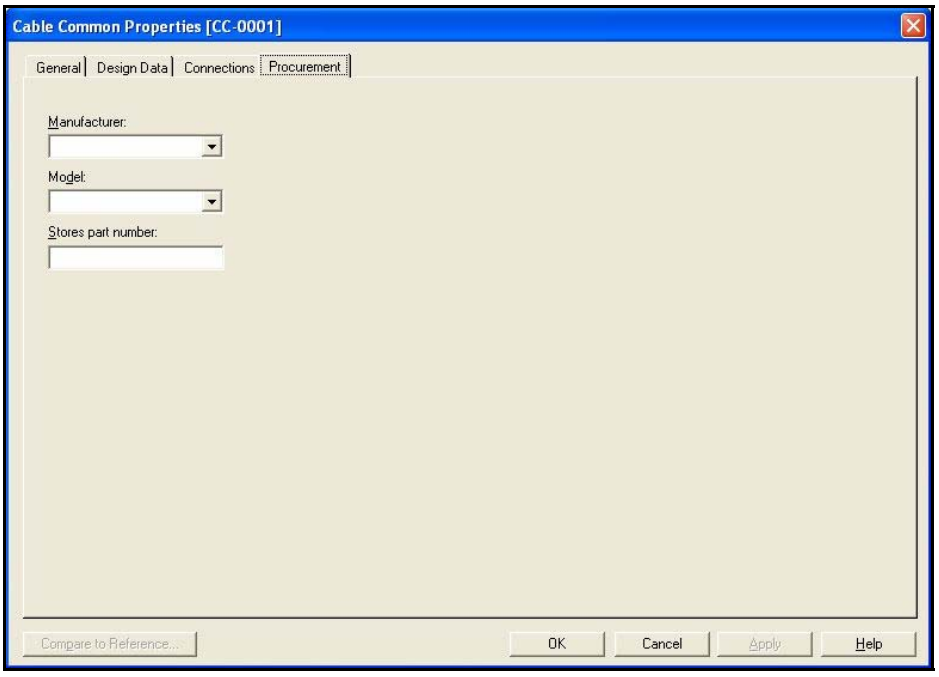
Design Data tab



Connections tab



Procurement tab



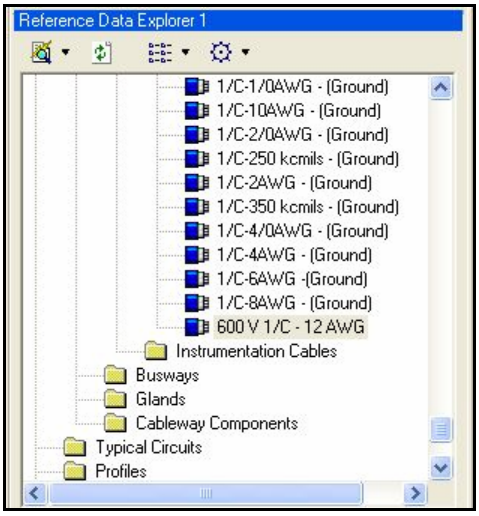
Grounding Cable Category

Grounding cables have only one conductor arrangement, which is the single conductor cable. The other parameters and behavior is similar to control cables.

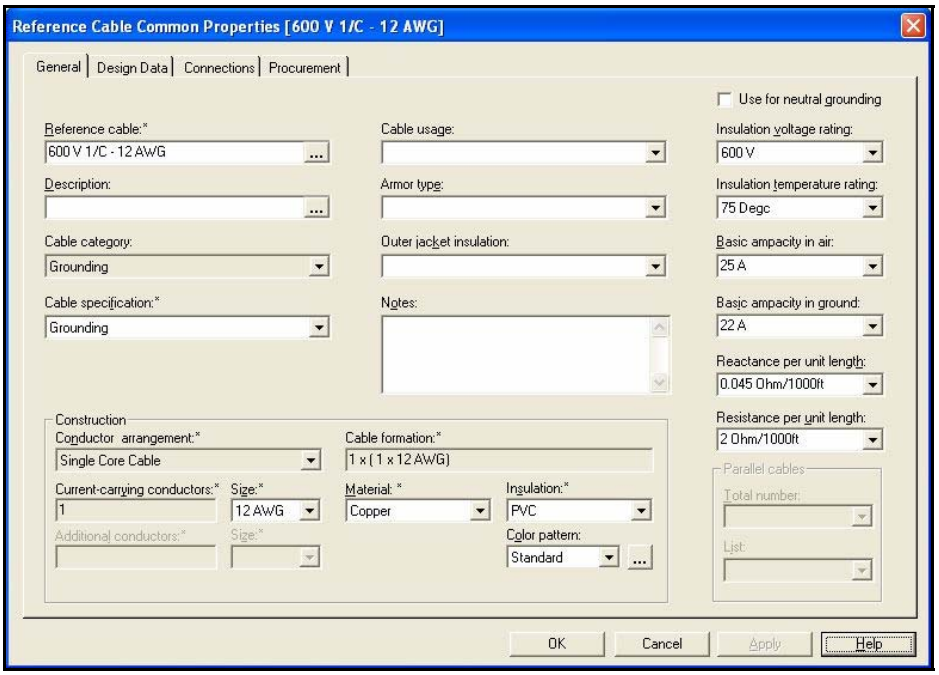
Reference Grounding Cable

Use the following procedure to create typcial grounding cable in RDE:

1. In RDE, select Wiring Equipment, Cables, Grounding Cables.
2. Right Click on Grounding Cables folder and select New .
3. Rename the new cable to '600 V 1/C - 12 AWG'.



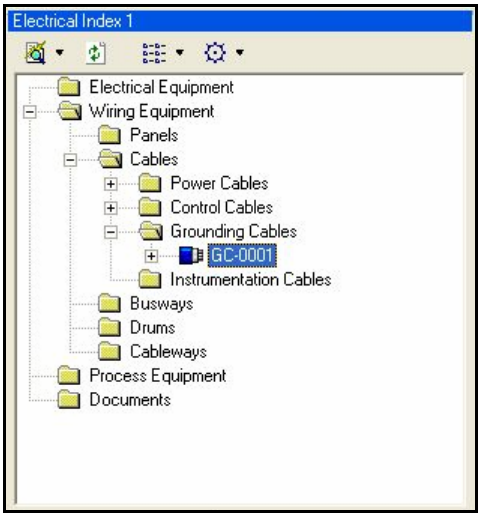
Edit the cable properties on the Reference Cable Common Properties dialog box according to the available catalog information, including size, material,insulation and electrical cable data.



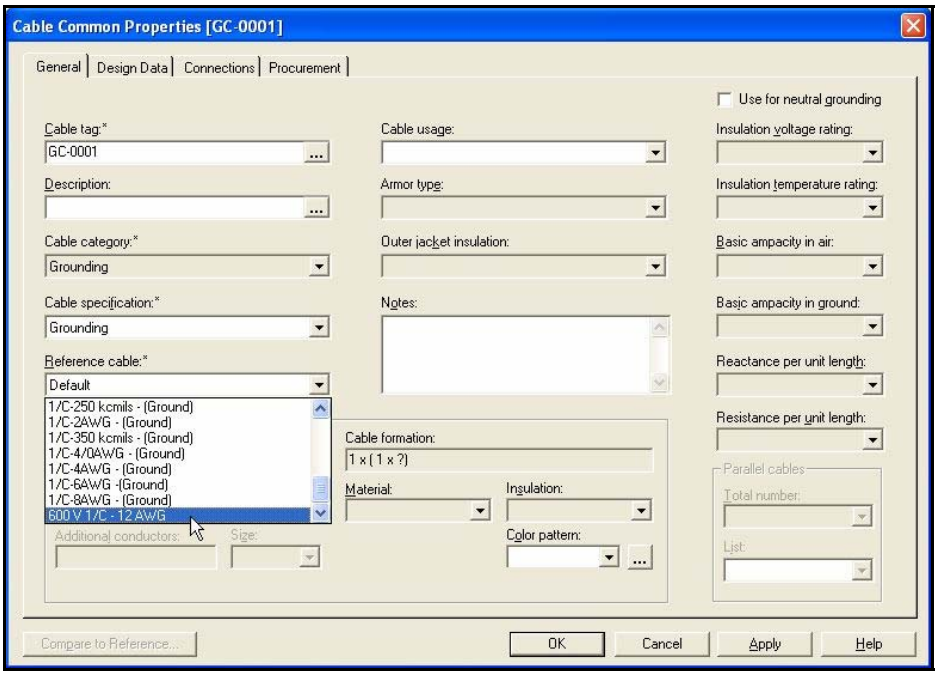
Project Grounding Cable

Use the following procedure to create project grounding cable in EI:

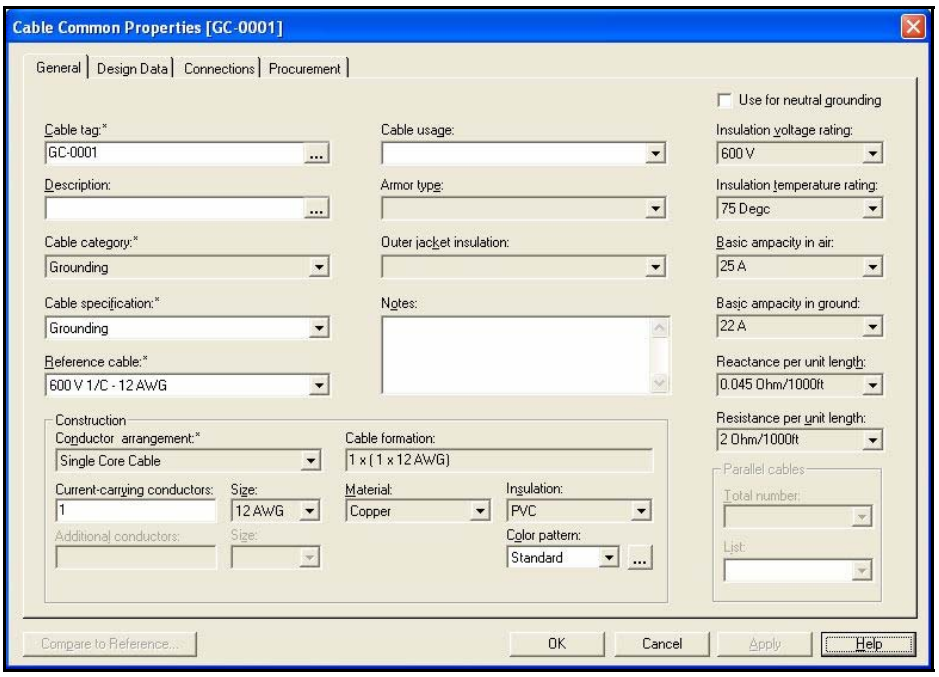
1. In EI, select Wiring Equipment, Cables, Grounding Cables.
2. Right Click on Grounding Cables folder and select New .
3. Rename the new cable to 'GC-0001'.



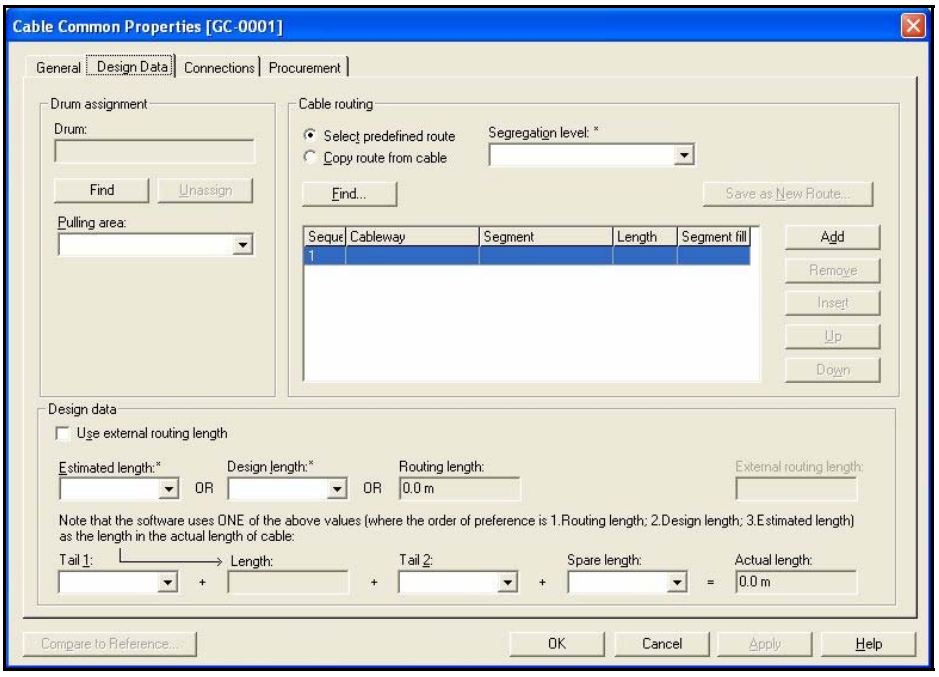
4. Open the cable common properties window, general tab, and select cable specification, Grounding. From the Reference cable select list pick a reference cable you would like to copy from.



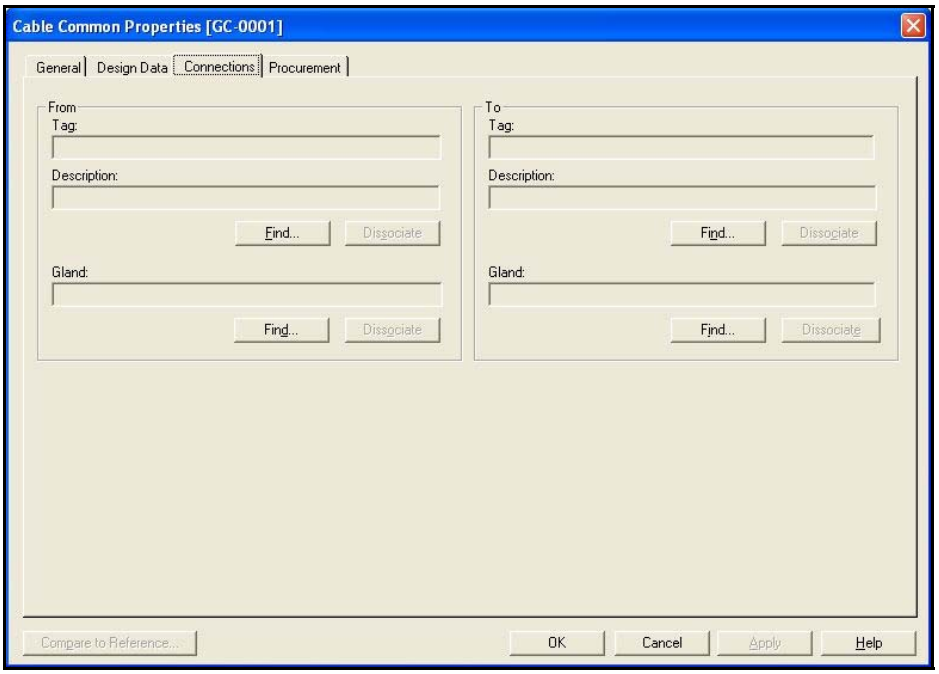
5.Selecting the reference cable 600 V 1/C – 12 AWG populate the values from the reference cable To the project cable.



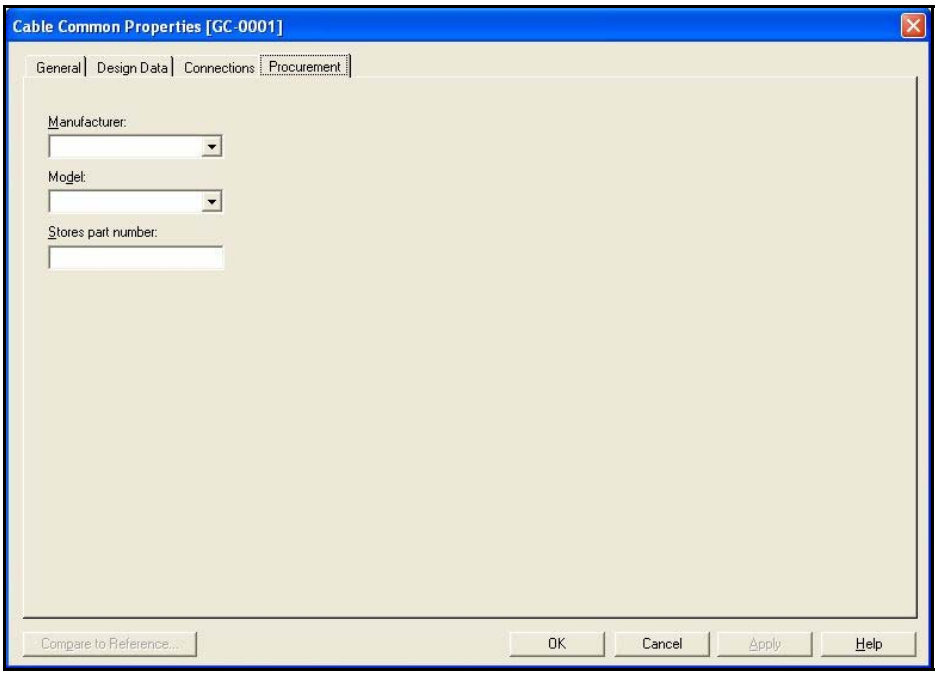
Design Data tab



Connections tab



Procurement tab



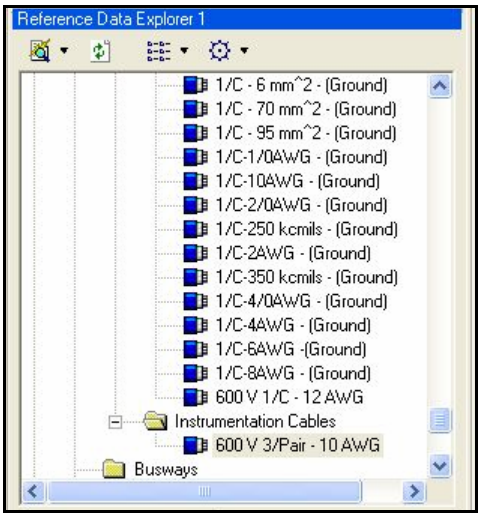
Instrumentation Cable Category

Cables composed of two or more pairs of twisted conductors.

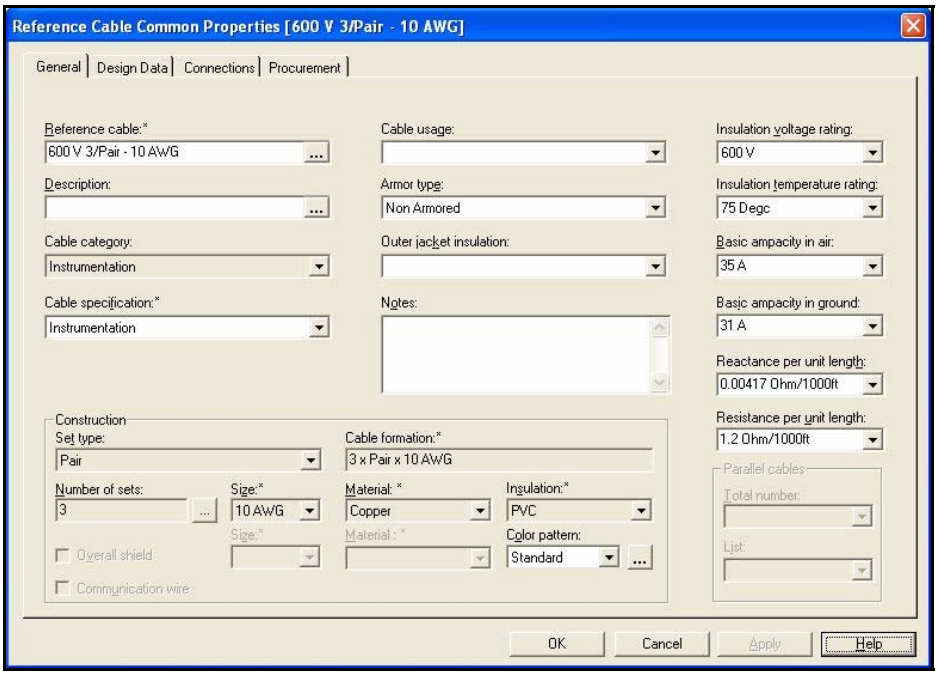
Reference Instrumentation Cable

Use the following procedure to create typcial instrumentation cable in RDE:

1. In RDE, select Wiring Equipment, Cables, Instrumentation Cables.
2. Right Click on Instrumentation Cables folder and select New .
3. Rename the new cable to '600 V 3/Pair - 10 AWG'



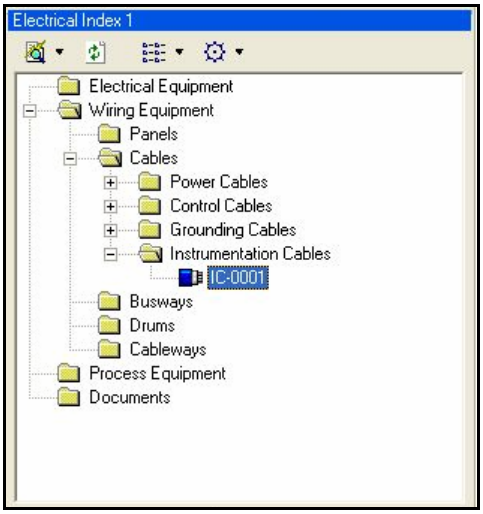
Edit the cable properties on the Reference Cable Common Properties dialog box according to the available catalog information, including set type, number of sets, size, material, insulation, and electrical cable data.



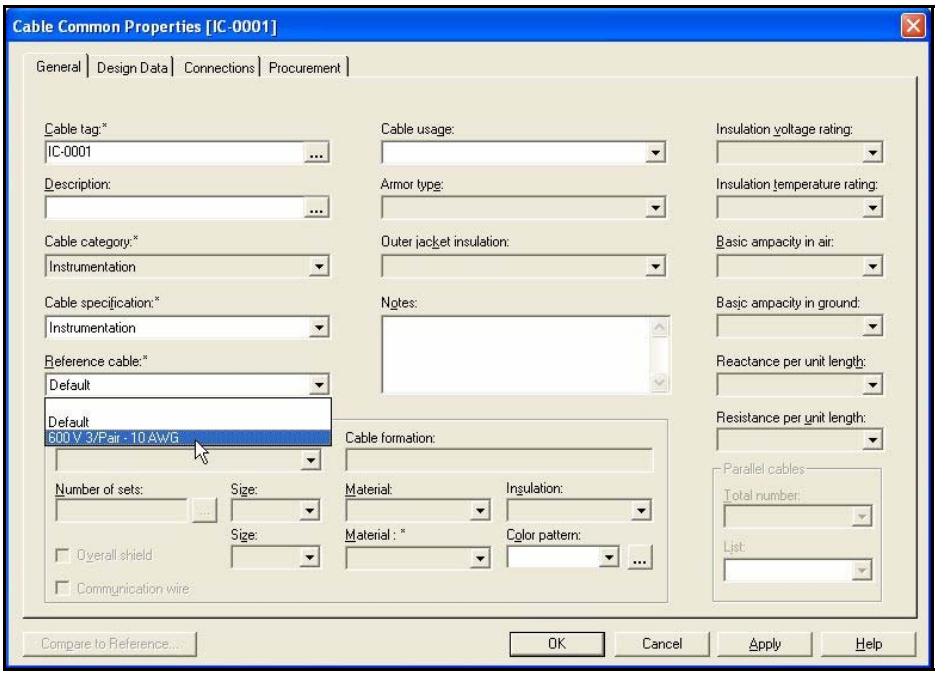
Project Instrumentation Cable

Use the following procedure to create project instrumentation cable in EI:

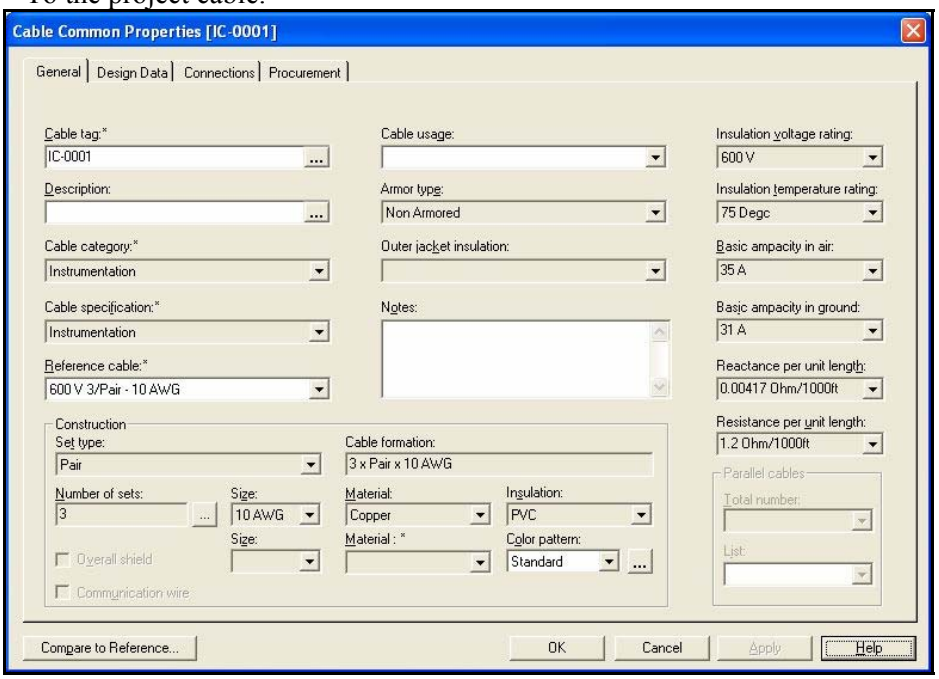
1. In EI, select Wiring Equipment, Cables, Instrumentation Cables.
2. Right Click on Instrumentation Cables folder and select New .
3. Rename the new cable to 'IC-0001'.



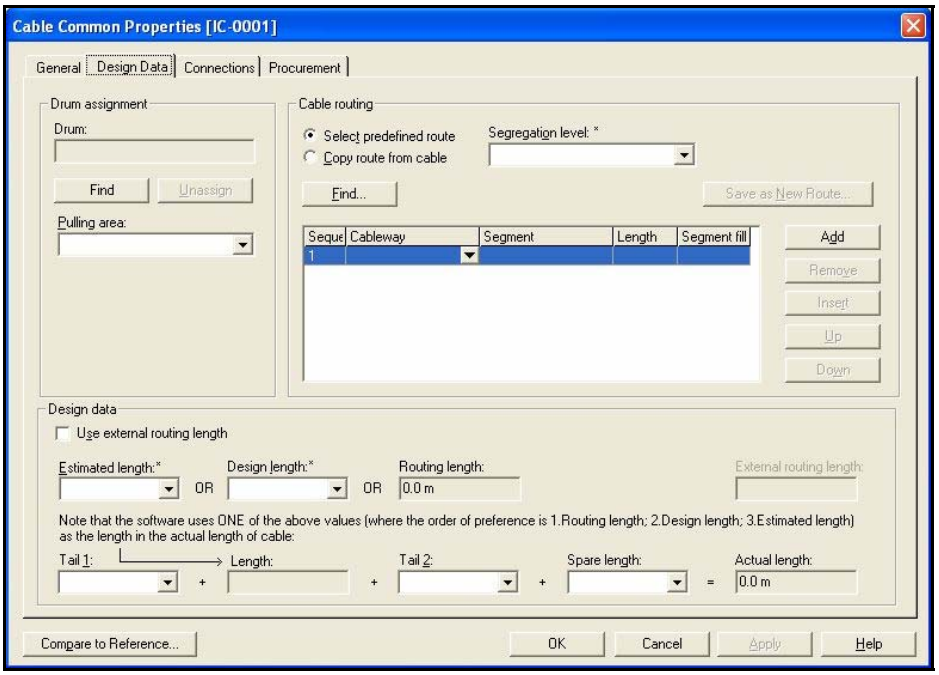
4.Open the cable common properties window, general tab, and select cable specification, Instrumentation.From the Reference cable select list pick a reference cable you would like to copy from.



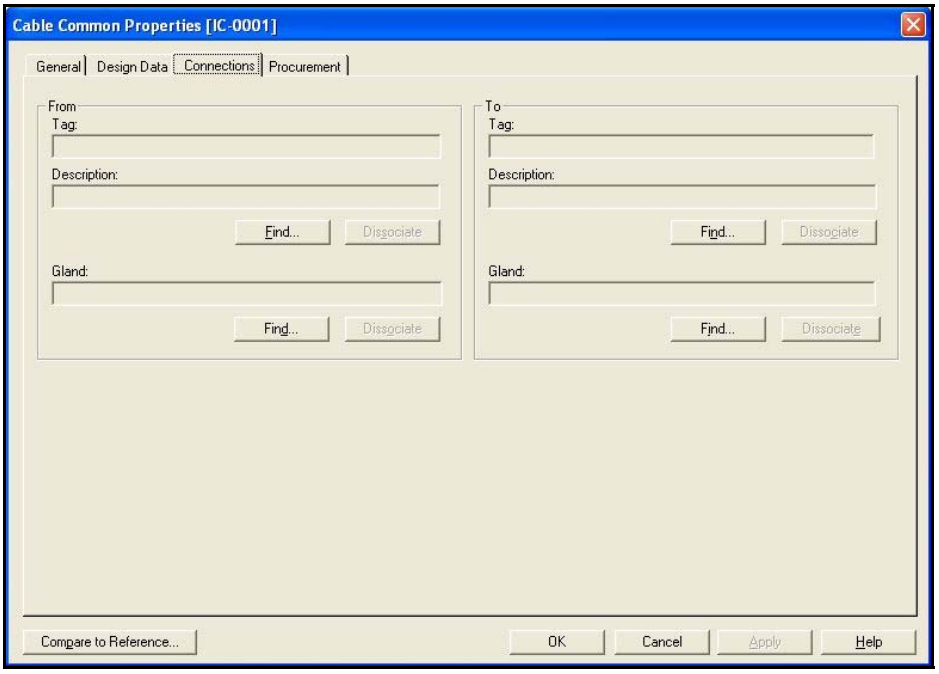
5.Selecting the reference cable 600 V 3/Pair – 10 AWG populate the values from the reference cable To the project cable.



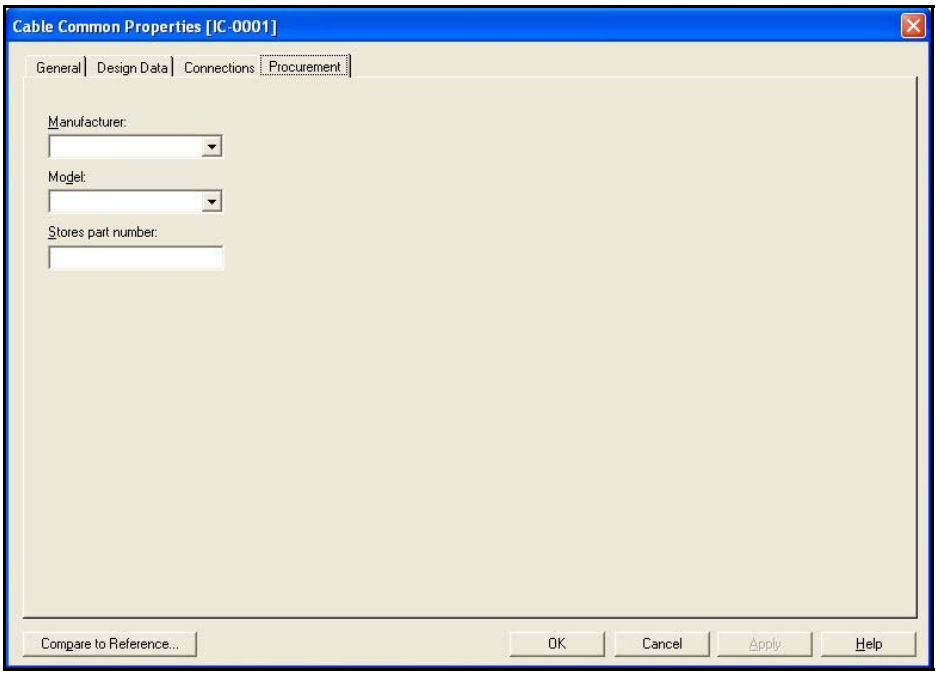
Design Data tab



Connections tab

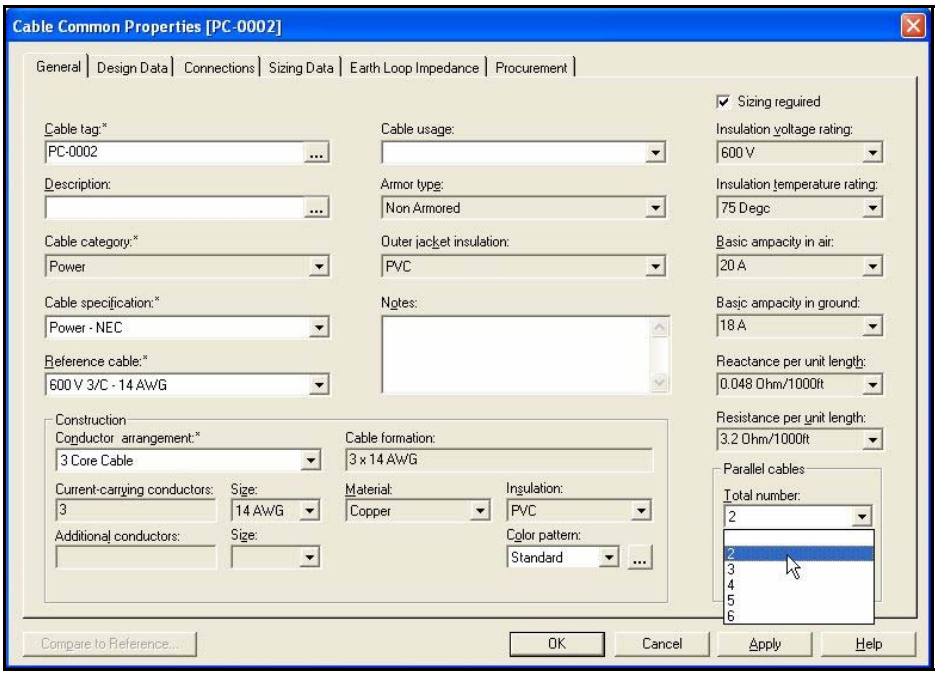


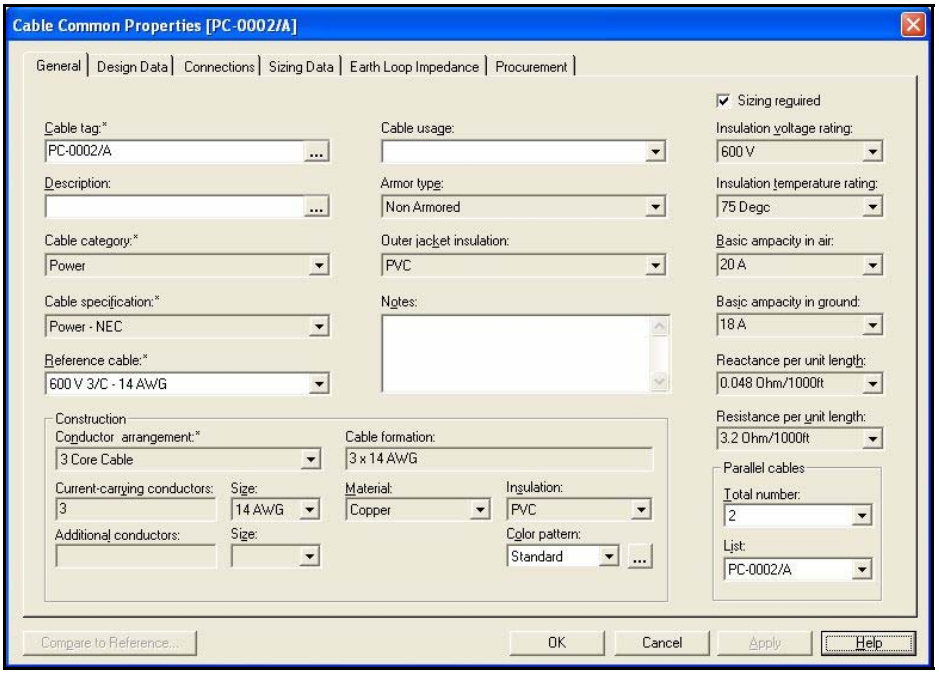
Procurement tab

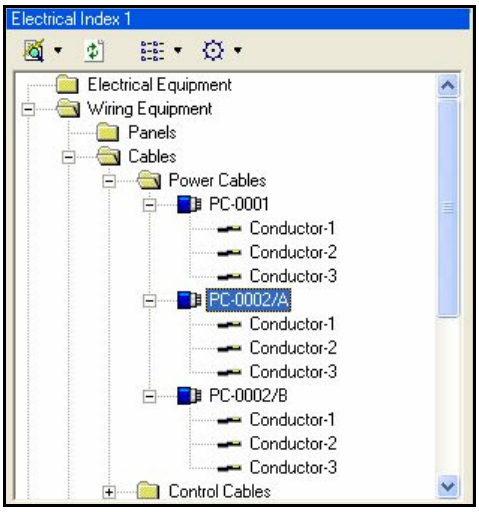


Parallel Cables

Two or more cables used to share the current in heavily loaded power cable which permits the use of smaller conductors.
In Smart plant Electrical parallel cables are groups of power cables that are created as a result of the cable sizing, added, or converted into parallels cables, and as such they behave as a group for all senses and operations.
The number of parallel cables (or cores per phase) is indicated in the UI.


Cable Common Properties [PC-0002]
General
Design Data
Connections
Sizing Data
Earth Loop Impedance
Procurement
Sizing required
Cable tag:*
PC-0002
Cable usage:
Insulation voltage rating:
600 V
Description:
Armor type:
Non Armored
Insulation temperature rating:
75 Degc
Cable category:*
Power
Outer jacket insulation:
PVC
Basic ampacity in air:
20 A
Cable specification:*
Power - NEC
Notes:
Basic ampacity in ground:
18 A
Reference cable:*
600 V 3/C - 14 AWG
Reactance per unit length:
0.048 Ohm/1000ft
Construction
Conductor arrangement:*
3 Core Cable
Cable formation:
3 x 14 AWG
Resistance per unit length:
3.2 Ohm/1000ft
Parallel cables
Current-carrying conductors:
3
Size:
14 AWG
Material:
Copper
Insulation:
PVC
Total number:
2
Additional conductors:
Size:
Color pattern:
Standard
2
3
4
5
6
Compare to Reference...
OK
Cancel
Apply
Help


Cable Common Properties [PC-0002/A]
General
Design Data
Connections
Sizing Data
Earth Loop Impedance
Procurement
Sizing required
Cable tag:*
PC-0002/A
Cable usage:
Insulation voltage rating:
600 V
Description:
Armor type:
Non Armored
Insulation temperature rating:
75 Degc
Cable category:*
Power
Outer jacket insulation:
PVC
Basic ampacity in air:
20 A
Cable specification:*
Power - NEC
Notes:
Basic ampacity in ground:
18 A
Reference cable:*
600 V 3/C - 14 AWG
Reactance per unit length:
0.048 Ohm/1000ft
Construction
Conductor arrangement:*
3 Core Cable
Cable formation:
3 x 14 AWG
Resistance per unit length:
3.2 Ohm/1000ft
Parallel cables
Current-carrying conductors:
3
Size:
14 AWG
Material:
Copper
Insulation:
PVC
Total number:
2
Additional conductors:
Size:
Color pattern:
Standard
List:
PC-0002/A
Compare to Reference...
OK
Cancel
Apply
Help

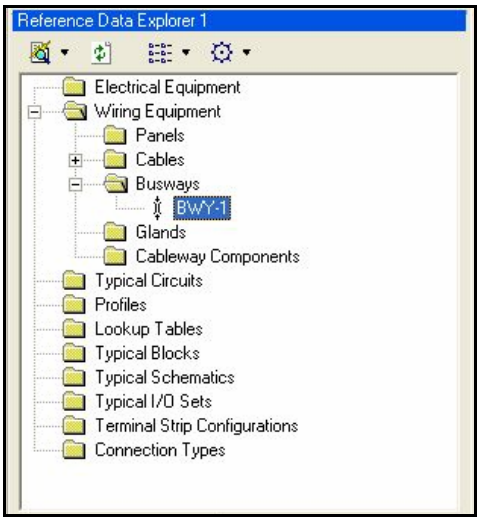


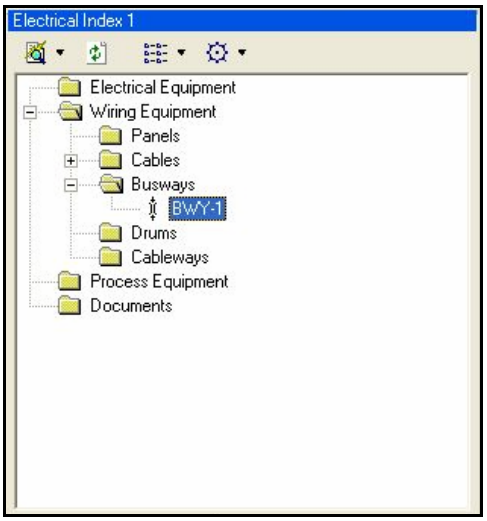
Change the cable back to none parallel cable by select none in the parallel cable total number, and hit apply.

Busways

A busway is a bus bar or other metallic bar, like in the PDB, used to connect between two item tags, on cases that regular cables cannot do the job due to the high required current or for other reason.

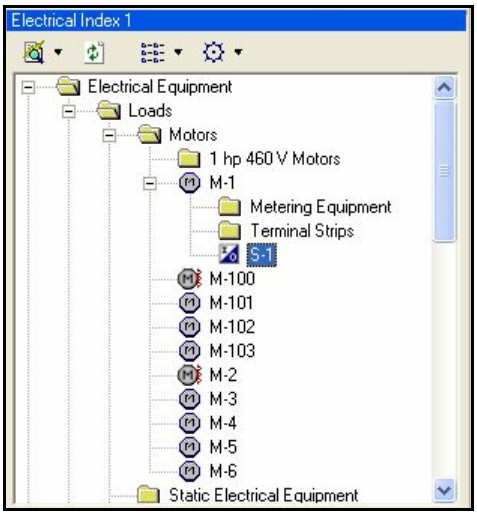
A bus way functionality is similar to the one of the power cable without the sizing capability and with no common properties form.





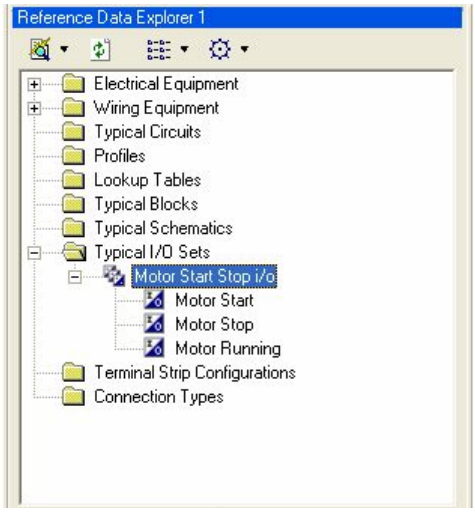
Signals

The system allows creation of i/o signals. Creating a i/o signal under a specific item is enabled for loads, generators,battery banks,converting equipments, circuits, stand alone disconnect electrical equipments,instruments and constrol stations.

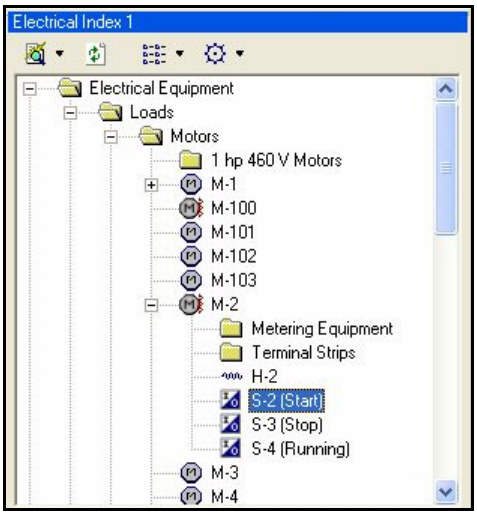


I/O Sets

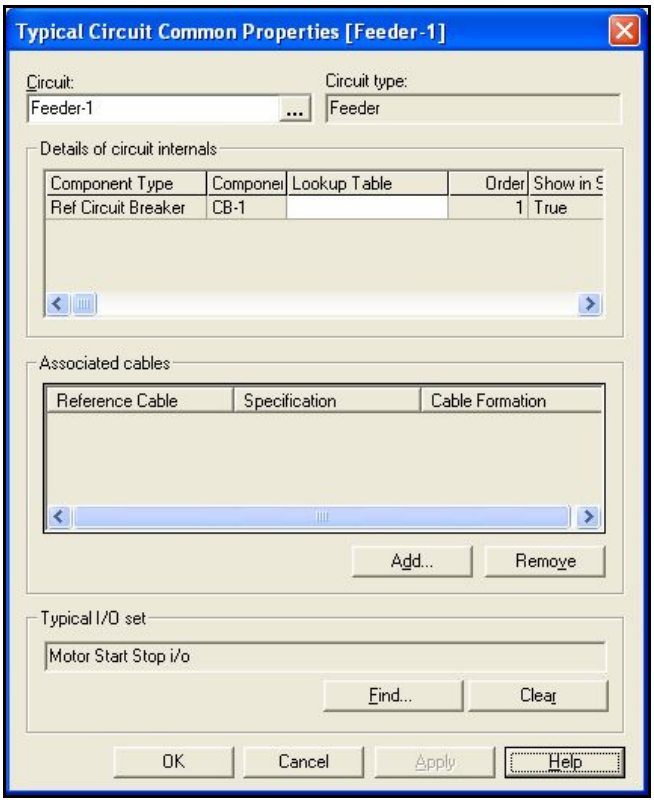
Set of one or more typcial i/o signals.
You create the i/o set in RDE.



You can use it for any project project items with i/o signal enabled, by drag and drop it from The RDE to EI.



You may also use it in conjunction with typical control station or typcial circuit.



Lab 4

Create the following typical wiring items in RDE:

Note: See page 122 for additional cable info

Reference Power Cable
4.1. **'5kV 3 + 1/C – 750 kcmil'**
4.2. **'600 V 3 + 1/C - 350 kcmil'**
4.3. **'600 V 3 + 1/C - 14 AWG' Note:** For SH-100
4.4. **'600 V 3 + 1/C – 12 AWG' Note:** For M-100,M-101, and 103
4.5. **'600 V 3 + 1/C – 1/0 AWG' Note:** For M-102
4.6. **'600 V 3 + 1/C – 2/0 AWG' Note:** For H-100

Reference Control Cable
4.7. **'600 V 5/C – 14 AWG'**

Reference Grounding Cable
4.8. **'600 V 1/C – 14 AWG'**

Reference Instrumentation Cable
4.9. **'600 V 3/Pair – 14 AWG'**

Typical i/o Set
4.10. **'Motor St Sp Rn I/O'**

Reference Control Station
4.11. **'Motor St St Rn Control Sation'**
Associate to control station 'Motor St Sp Rn Control Sation', the reference control cable, '600 V 5/C – 14 AWG', and reference grounding cable, '600 V 1/C – 14 AWG'.
Associate to control station, 'Motor St Sp Rn Control Sation', the typical i/o set, 'Motor St Sp Rn I/O'

Reference Local Panel
4.12. **'LP Type A'**
Add to the local panel, circuit **LP Type A CKT-1** with new disconnect switch, '**DS-1'**

Reference Junction Box
4.13. **'JB Type A'**
Add to the junction circuit '**JB Type A CKT-1'** with new disconnect switch, '**DS-1'**

Reference Instrument
4.14. **'RTD Surface Element'**
Associate to the instrument, reference instrumentation cable '600 V 3/Pair – 14 AWG' , and grounding cable '600 V 1/C – 14 AWG'

Create the following project wiring items in EI:

Project Power Cable
4.15. **'PC-TS-001-1'** **Note:** Use typical '5kV 3 + 1/C – 750 kcmil'
4.16. **'PC-TS-001-2'** **Note:** Use typical '5kV 3 + 1/C – 750 kcmil'
4.17. **'PC-SW-300-1'** **Note:** Use typical '5kV 3 + 1/C – 750 kcmil'
4.18. **'PC-T-001-1'** **Note:** Use typical '5kV 3 + 1/C – 750 kcmil'
4.19. **'PC-MCC-201-1'** **Note:** Use typical '600 V 3 + 1/C - 350 kcmil'
4.20. **'PC-MCC-202-1'** **Note:** Use typcial '600 V 3 + 1/C - 350 kcmil'
4.21. **'PC-SH-100-1'** **Note: U**se typical '600 V 3 + 1/C - 14 AWG'
4.22. **'PC-M-100-1'** **Note:** Use typical '600 V 3 + 1/C – 12 AWG'
4.23. **'PC-M-101-1'** **Note:** Use typical '600 V 3 + 1/C – 12 AWG'
4.24. **'PC-M-102-1'** **Note:** Use typical **'**600 V 3 + 1/C – 1/0 AWG'
4.25. **'PC-M-103-1'** **Note: U**se typical '600 V 3 + 1/C – 12 AWG'
4.26. **'PC-H-104-1** **Note: U**se typical '600 V 3 + 1/C – 2/0 AWG'

Project Control Station

4.27. '**CS-M-100-1**' (using typical control station 'Motor St Sp Rn Control Sation')
Name the project control cable 'CC-M-100-1'
Name the project grounding cable 'GC-M-100-1'
Name the signals to St-M-100-1,Sp-M-100-1 and Rn-M-100-1

Project Local Panel

4.28. '**LP-001**' (using typical local panel 'LP Type A')

Project Junction Box

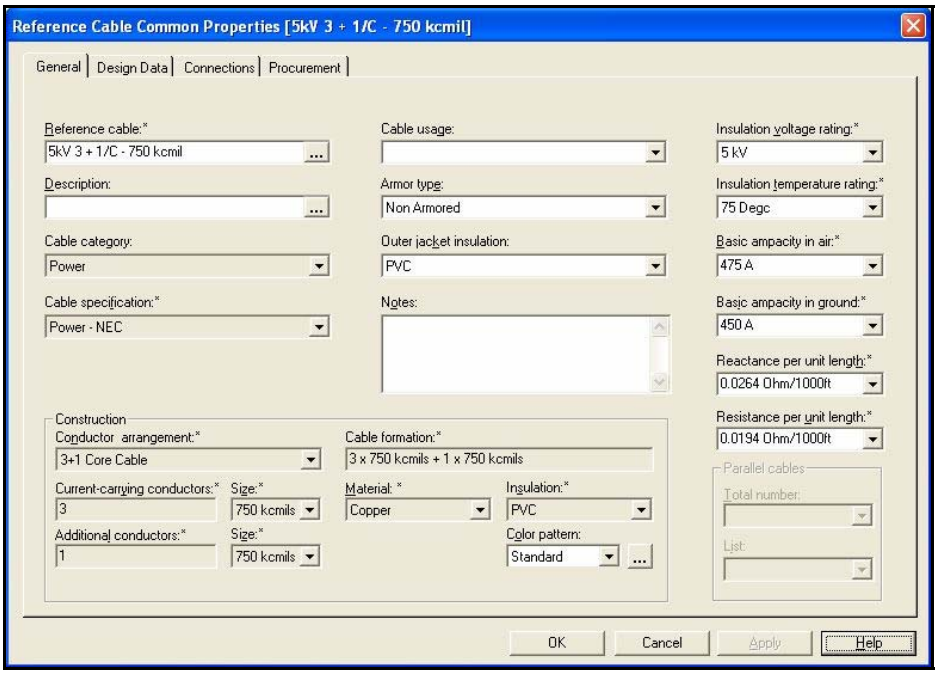
4.29. '**JB-001**' (using typical junction box 'JB Type A')

Project Instrument

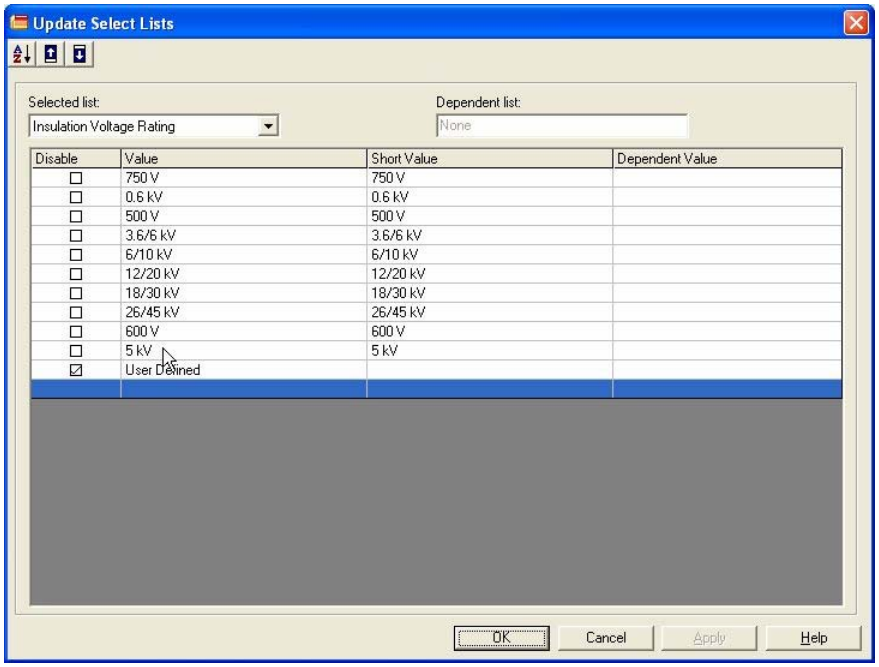
4.30. **'INST-100-1'** (using typical instrument 'RTD Surface Instrument')
Name the project instrumentation cable 'IC-INST-100-1'
Name the project grounding cable 'GC-INST-100-1'

Addition information on cables for the lab

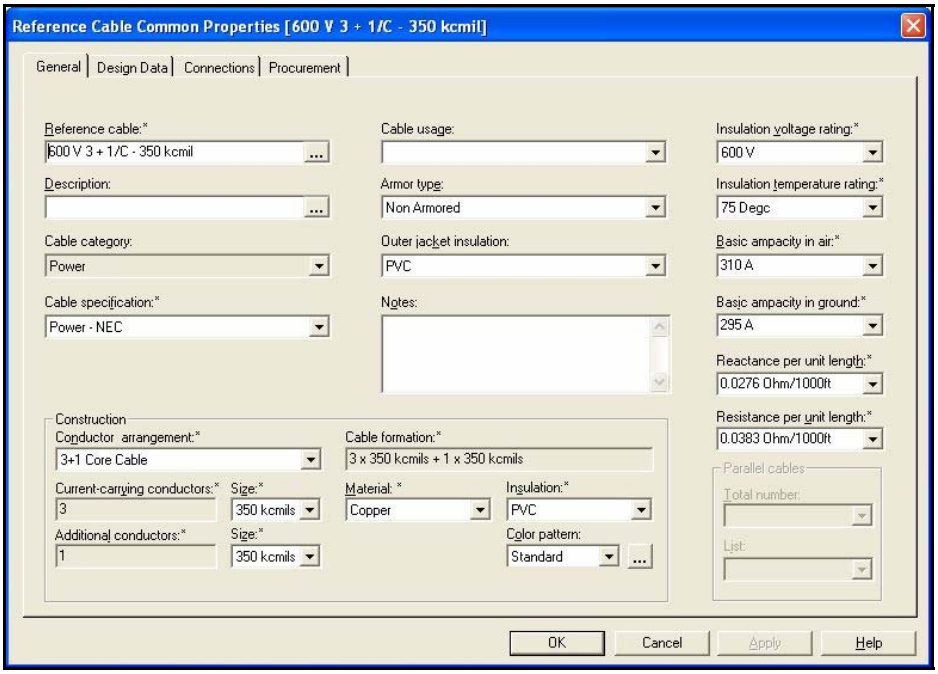
Typical Power Cable '5kV 3 + 1/C – 750 kcmil'



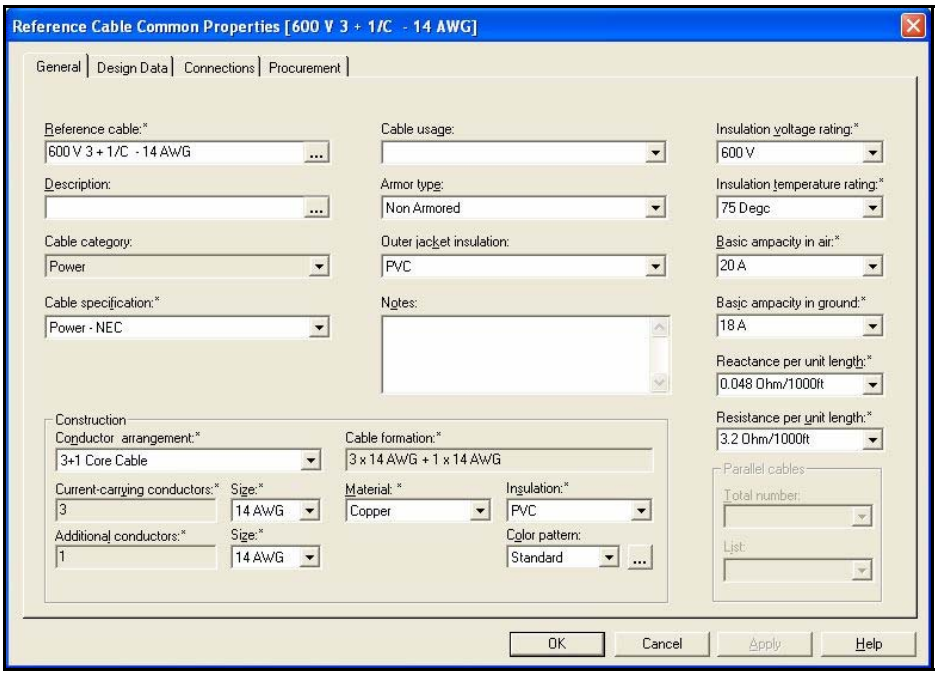
New insulation voltage rating 5 kV



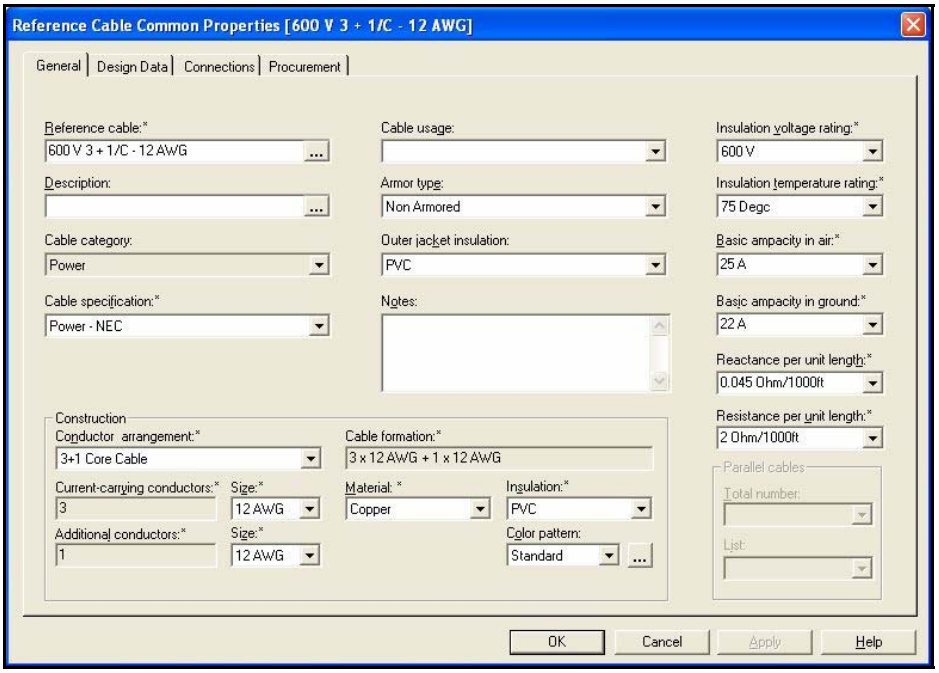
Typical power cable '600 V 3 + 1/C - 350 kcmil'



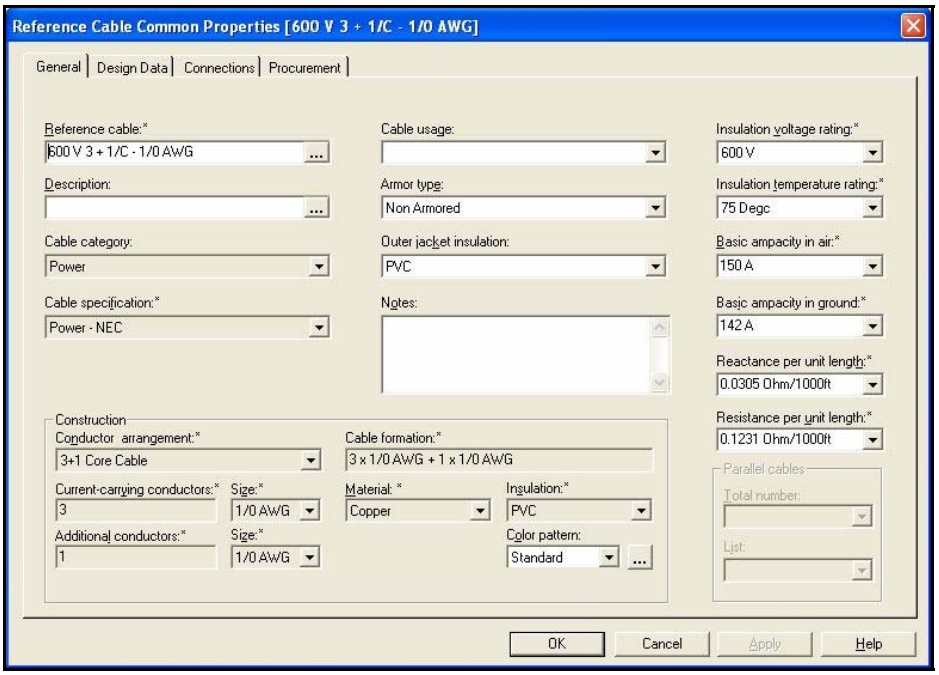
Typical power cable '600 V 3 + 1/C - 14 AWG'



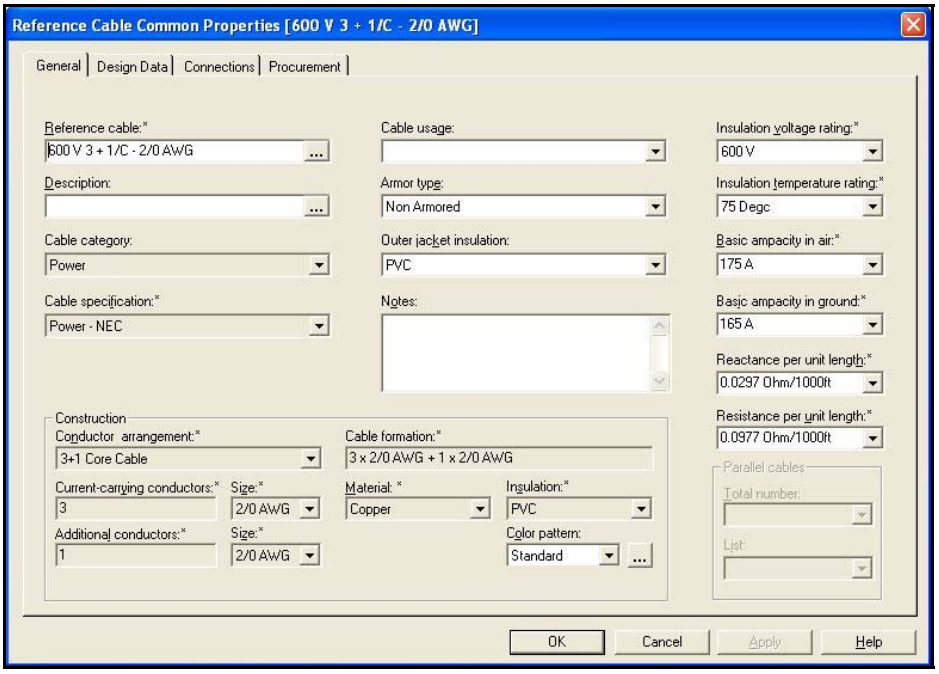
Typical power cable '600 V 3 + 1/C - 12 AWG'



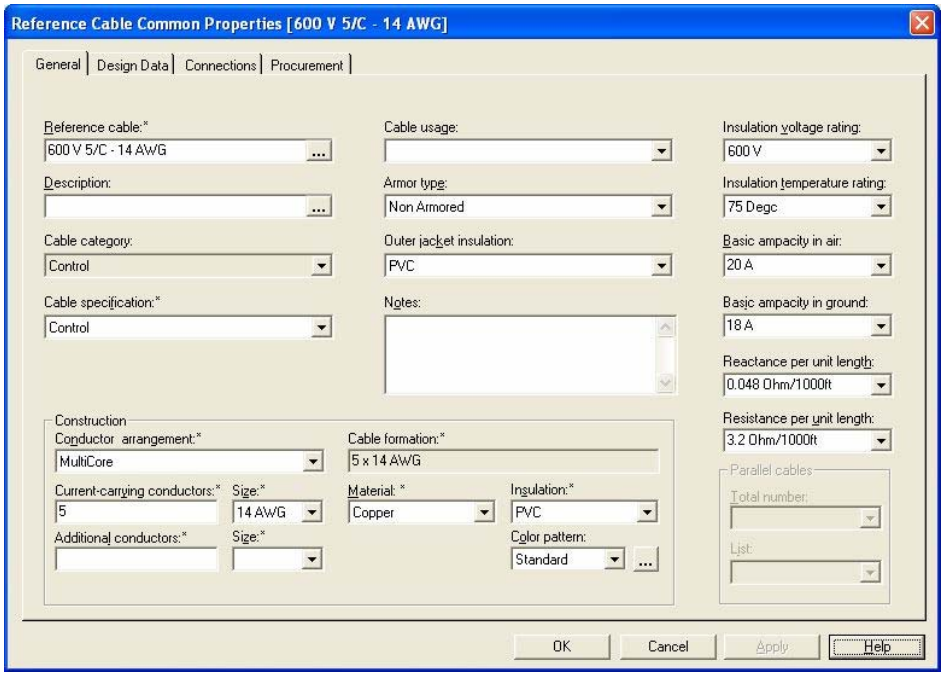
Typical power cable '600 V 3 + 1/C - 1/0 AWG'



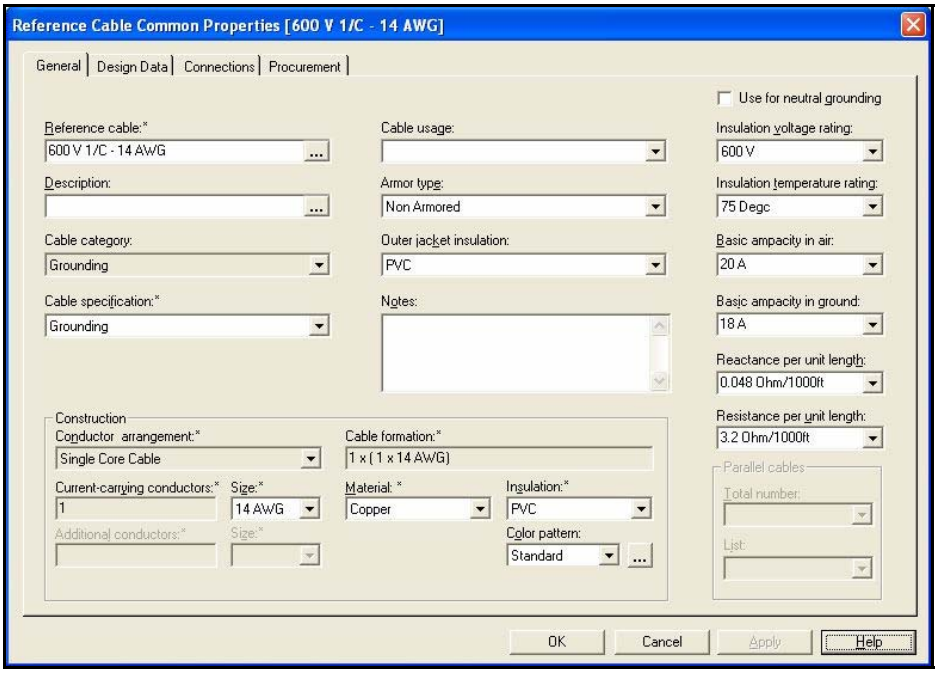
Typical power cable '600 V 3 + 1/C - 2/0 AWG'



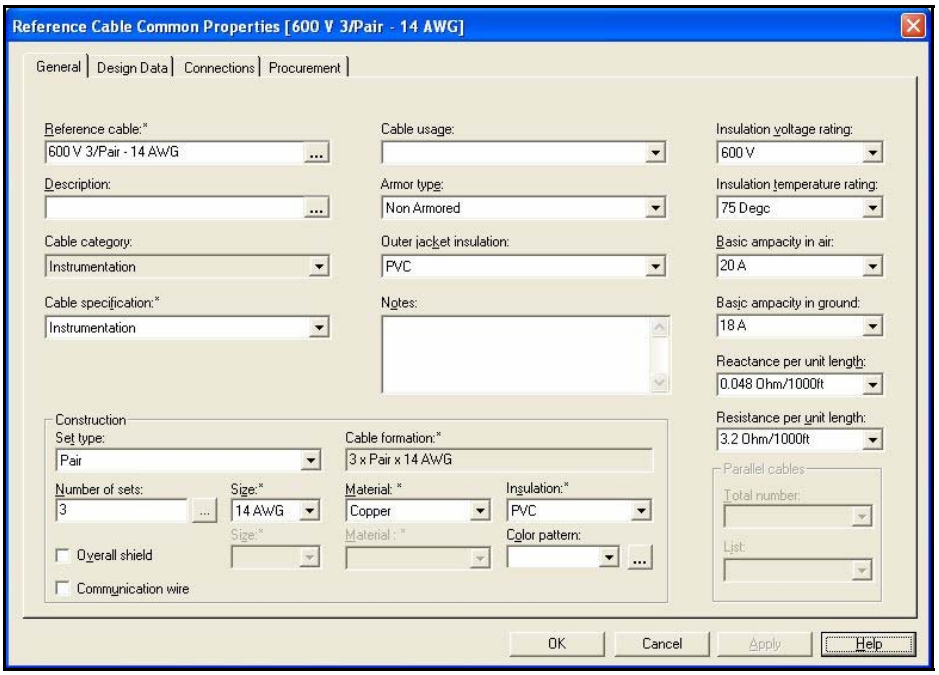
Typical control cable '600 V 5/C – 14 AWG'



Typical grounding cable '600 V 1/C – 14 AWG'



Typical instrumentation cable '600 V 3/Pair – 14 AWG'


Reference Cable Common Properties [600 V 3/Pair - 14 AWG]
General
Design Data
Connections
Procurement
Reference cable:*
600 V 3/Pair - 14 AWG
Description:
Cable category:
Instrumentation
Cable specification:*
Instrumentation
Cable usage:
Armor type:
Non Armored
Outer jacket insulation:
PVC
Notes:
Insulation voltage rating:
600 V
Insulation temperature rating:
75 Degc
Basic ampacity in air:
20 A
Basic ampacity in ground:
18 A
Reactance per unit length:
0.048 Ohm/1000ft
Resistance per unit length:
3.2 Ohm/1000ft
Construction
Set type:
Pair
Cable formation:*
3 x Pair x 14 AWG
Number of sets:
3
Size:*
14 AWG
Material: *
Copper
Insulation:*
PVC
Overall shield
Communication wire
Color pattern:
Parallel cables
Total number:
List:
OK
Cancel
Apply
Help

Chapter 5

Propagating Default Data and Associations

This part deals with advanced procedures and operations that will save you time, enable the reuse of default engineering data and minimize the errors due to manual data entry.

Apply Options

The **Apply Options** allows creating and populating project items in Electrical Index with massive default equipment in batch operations.

The following apply options are available:

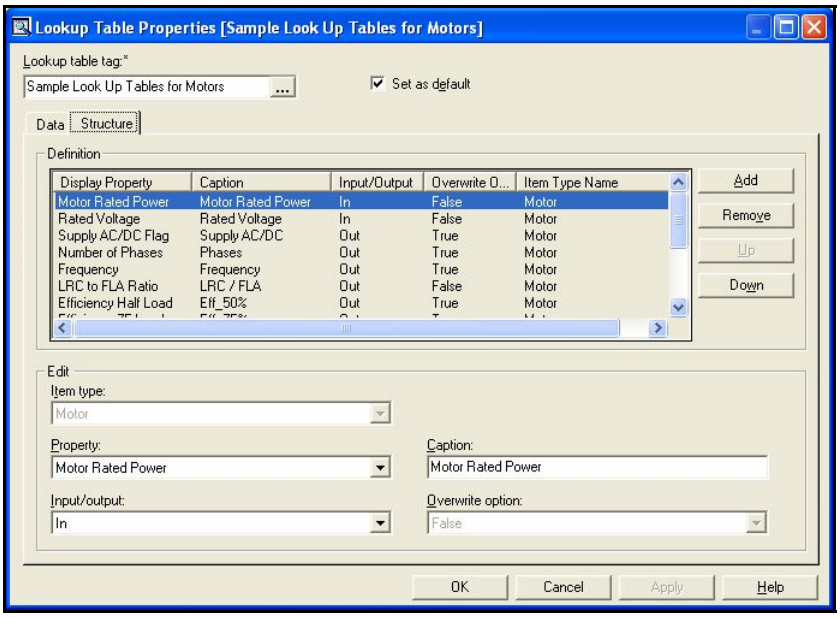
Equipment Profile

Typical Circuit
Control Station
Lookup Tables
Typical Schematic
Typical I/O Set

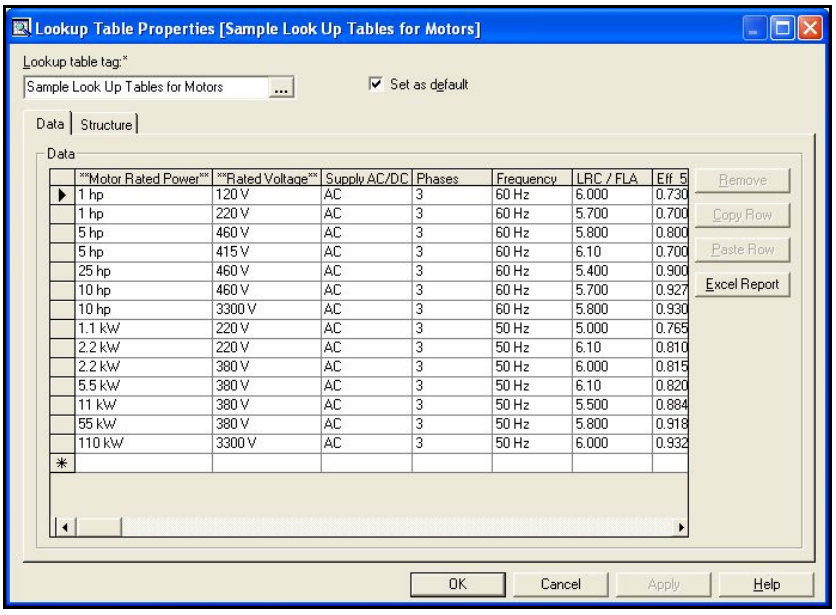
When applying any of these options, you need to select the enable check box found in the Apply option windows. Only one is allowed at a time (checking the flag on one tab resets any previously checked flag).You can apply set of items, by selecting set of items from **Electrical Index,** lower pane window and making multiple selections, or in the Tabular Editor by highlighting a subset of items from the table.

Apply Lookup Tables

Lookup Table (LUT) – Reference table you create in RDE, containing structure of key properties ,in. And none key properties, out (result).



And **data,** values for the key and none key properties.



Function LUT on project item in Electrical Index, by using the apply LUT, used to search for exact match between all key properties value, in LUT, and in project items. And according to the search result, to update the none key properties value of the project item with the LUT none key properties value.

Create Lookup Table

From the Reference Data Explorer select Lookup Tables hierarchy folder, motors folder (or other type).
Right-click to create new lookup table, name it '**3 Phase Motor LUT**'.
Select the '3 Phase Motor LUT', right-click and select common properties.

Note: You can have more than one Lookup Table for each equipment item type. The default look up table check box is set to the default Lookup Table for that equipment item type.

Select the Structure tab.
In the edit section you select the properties that you would like to include in your '3 Phase Motor LUT'.
From the Property list, find the property that contains the info you need.
In caption, enter the name of the heading you would like to see.
From Input/Output, select Input to specify a key property, or select Output to specify none key property.

Note: It is very important that you pay particular attention to the Input/Output option.
When this look-up table is applied to an item, the program looks for an exact match of all the properties that are specified as key fields.

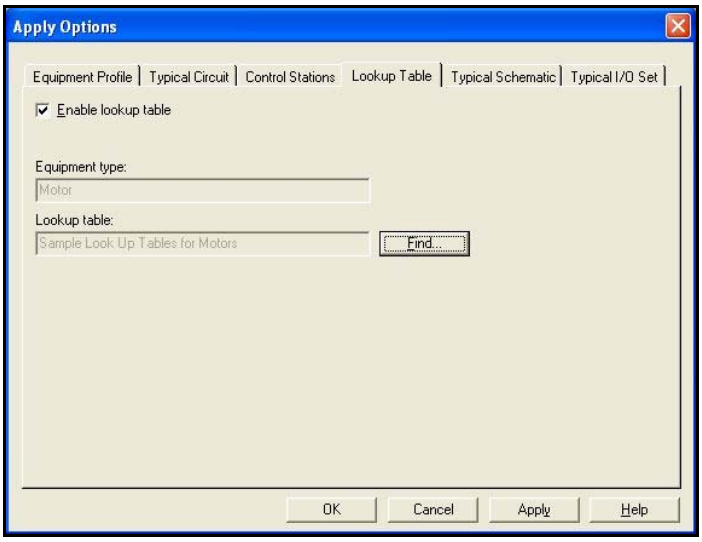
Set True value in Overwrite option to overwrite whatever value is already in the property.
Setting False will only write to empty properties.
Set properties and click the Add button. The definition window reflects the properties you have selected.
When you have selected several properties, you can arrange the Electrical Index order by using the Up and Down buttons. The Remove button removes the highlighted property.
Select the Data tab.
Select the arrow to the right of the property and select the appropriate values.
When you finish Click OK.

Apply Lookup Tables

Applying look up tables may result in overwriting project data. One should be aware of this. The right timing on whether and when to commit this operation is an engineering decision and would be based on your project state and your workflow.
You should make sure that the item that have been selected for this operation have the key fields populated with data that will be recognized by your lookup table, otherwise, data will not be populated.

1. In the Electrical Index, expand the folder hierarchy by clicking the + icons or by double-clicking the folders.
2. Navigate to the folder for the desired equipment type and do one of the following:
    - Expand the folder and select an item from the list.
    - Click the folder, and in the list view pane, hold Ctrl while you select more than one item.
3. Click Tools > Apply Options.
4. On the Apply Options dialog box, click the lookup table tab.
5. Select Enable lookup table.

6. Beside the lookup table box, click Find.
7. On the Find dialog box, click Find Now to display the available lookup tables.
8. From the list of lookup tables, highlight the lookup table that you want to apply.
9. Click OK.



Apply Typical Circuit

The purpose of this feature is to have a powerful way to create new circuits and internals or to replace existing circuits or internals.

Apply a Typical Circuit to an Item

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

2.Navigate to the folder for the equipment type that you require and do one of the following:

- Expand the folder and select an item from the list.
- Click the folder, and in the list view pane, hold Ctrl while you select more than one item.

Tips

You can apply a typical circuit to all loads, all converting equipment items, and circuits.

You can also select more than one item by selecting a subset of item tags in the Tabular Editor.

3.Click Tools > Apply Options.

4.On the Apply Options dialog box, click the Typical Circuit tab.

5.Select Enable typical circuit.

6. Beside the Typical circuit box, click Find.

7.On the Find dialog box, click Find Now to display the available typical circuits.

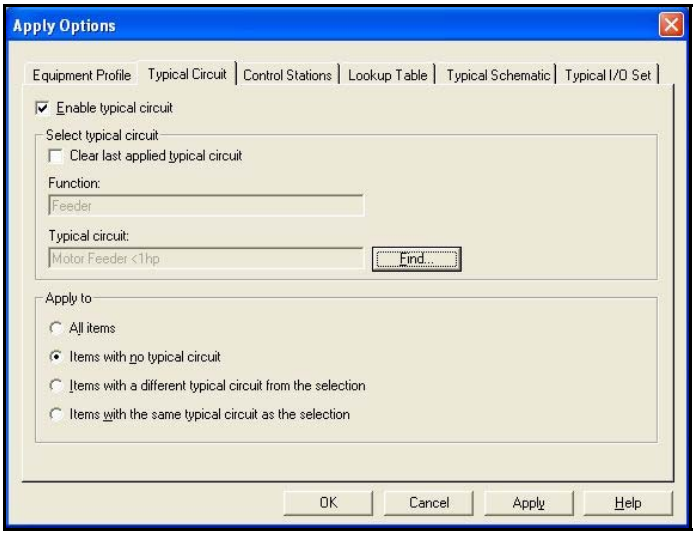
8. From the list of typical circuits, highlight the typical circuit that you want to assign to the equipment.

9. Click OK.

10.Under Apply to, select one of the following options according to the rule the software uses to apply the typical circuit:

- **All items** – Applies the typical circuit to all the selected items.
- **Items with no typical circuit** - Applies the typical circuit only to items without an existing typical circuit. Use this option to add a typical circuit only to new items among the items that you selected.
- **Items with a different typical circuit from the selection** - Changes the typical circuit circuit of the selected items to the current typical circuit. Use this option to add a typical circuit to items with a modified typical circuit definition.
- **Items with the same typical circuit as the selection** - Updates an existing typical circuit for the selected items.

11.Click Apply.



Notes:

- Applying typical circuit on item will not create the internal components if the item is not yet associated with a circuit. In this case, only a reference to the typical circuit is maintained.
  You can verify which typical circuit is assigned to an item by opening the "Feeder Data" tab of the item.
- Circuit internals (also, typical circuit internals) must have a unique sequence (order) property, which determine the electrical index power path in the Electrical Index.
  Applying a typical circuit on an item must preserve the electrical relations (if existed before) between the last internal component in the circuit and any other "external" object that was associated before (usually this would be a cable).
- Applying typical circuits is available upon selecting single or multiple items.
- Clear last applied typical circuit make it possible to clear pre associated typical circuit.
- The actual project items are created as per the typical circuit (you need to associate the load with a PDB – Bus and Cell in order to create them)

Guidelines and rules:

1.When assigning typical circuits, the system checks and prompts in case of discrepancy between the reference typical of the load and the circuit we are trying to associate it to. In case of a batch, the user is the decision-maker and he has the option to choose the scenario. No prompt will be issued in the batch apply. The "Apply" will govern and not the circuit.(still, the rules will govern- in case there are some motors that were not applied a "non all items has been successfully committed" message will be issued)

2.Associating a motor to a circuit that has no internals will result in creating the actual internals if there is a reference in the motor. If not, it will not generate anything. Internals will be created once the profile is applied on the load. The form of the load/motor will be updated to show this typical circuit.

3.Trying to associate a motor with a typical circuit on a circuit (by the drag and drop or applying a profile) that has different type of typical defined within it, will be preceded with a message: "Load and

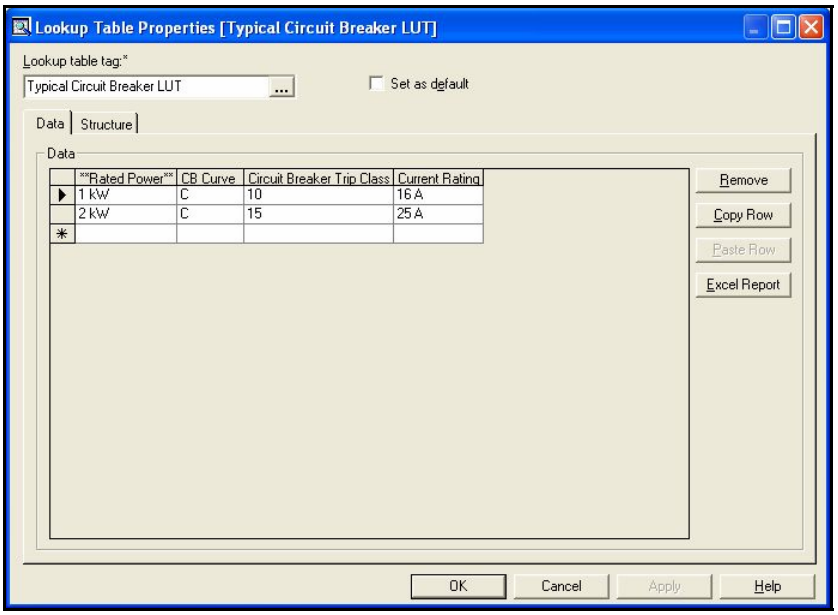
circuit internals are different- do you wish do override circuit internals? “Yes” or “Cancel” will tell the system how to proceed. If yes, the system will replace, as in the apply profile the internals of that circuit.

Reapplying a Typical Circuit
Typical circuits will replace the existing internal circuits and the Electrical Index components with the Reference Data Explorer components that constitute the typical circuit. Any existing electrical relations and association will be kept and preserved (so the Electrical Engineer will not be affected).
It will be up to the designer to make sure that what he is doing is right.

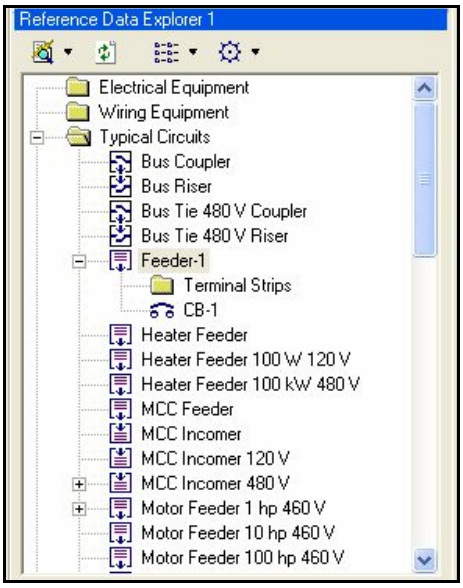
Associating and Applying Lookup tables to circuit internals

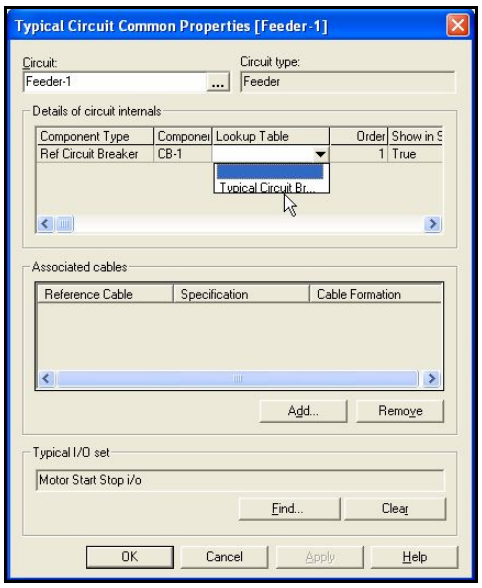
You can associate lookup tables with circuit internals, both at the RDE and in the EI.
This option gives you a great advantage in populating reference data to project protection and disconnect equipment. The system allows you to associate LUT with reference circuit internal via the typical circuits as follows:

1.Create circuit breaker lookup table



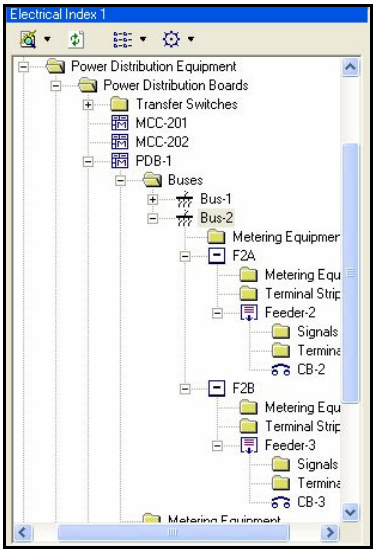
2.Associate the CB LUT to typcial circuit that has CB.

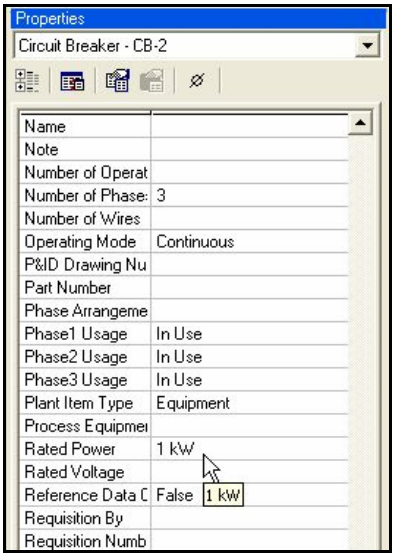




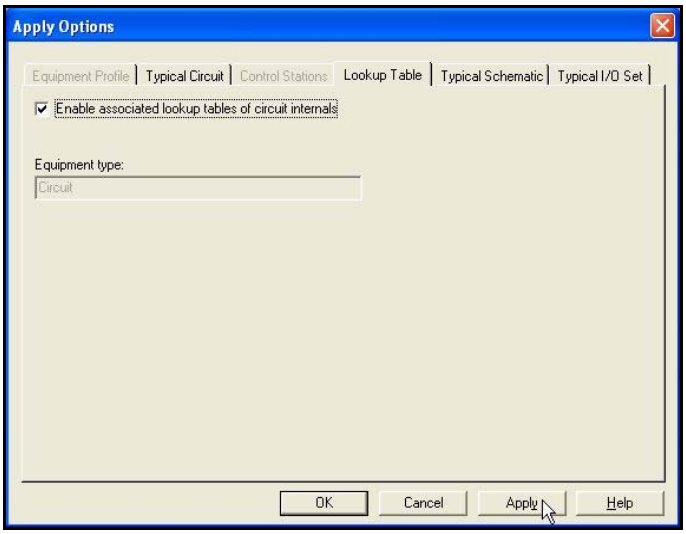
3.Create 2 project feeder circuits using the typcial circuit Feeder-1.
Set data in the property grid for the circuit breaker to match the keyfiled of the LUT (1 kW,2kW).

Note: You may also associate manually any lookup table to circuit internal, not via the typcial circuit.

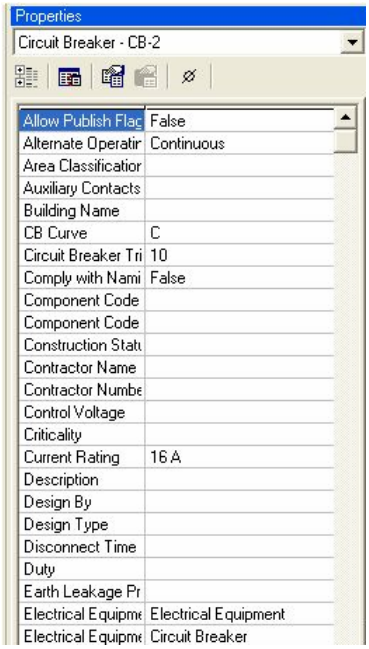




4.Propagate the lookup table data by select the circuits and select Apply.



Note: To select multiple item use the lower pane or tabular editor.



Apply Control StationTypical Circuit

Applying typical control station on loads will add control station, cables (instrumentation,control and grounding), and associate the cables and the load to the control station. Or replace existing control station, and cables, including their associations.

Apply a Typical control Station to an Item

1. In the Electrical Index, expand the folder hierarchy by clicking the + icons or by double-clicking the folders.
2. Navigate to the folder for the desired equipment type and do one of the following:
    - Expand the folder and select an item from the list.

- Click the folder, and in the list view pane, hold Ctrl while you select more than one item.

  Tip

  You can also select more than one item by selecting a subset of item tags in the Tabular Editor.

3. Click Tools > Apply Options.
4. On the Apply Options dialog box, click the Control Station tab.
5. Select Enable control station .
6. Click Add.
7. On the Find dialog box, click Find Now to display the available control station.
8. From the list of control stations, while holding Ctrl, highlight each control station that you want to associate with the item.
9. Click OK.
10. Under Apply to, click All items.

    Tip

    If required, you can click Items with no control station to add control stations only to items that do not have any existing control stations.

11. Under Action, click Add or Replace to determine how the software deals with control stations that have the same name as existing control stations. This option applies only if you click All items.
12. Click Apply.

Notes:

- You can identify to which load the control station belongs to and what are the cables that are associated to it by opening the control station property dialog.

- You can view the control station associated with an item of electrical equipment by selecting Common Properties on the shortcut menu for the item, and on the dialog box that opens, selecting the Control & Signals tab.

- Reapplying control station, add control station, or replace existing control stations according to the selection of the 'Replace control stations' check box.
  Checking the replace control stations check box, deletes the existing control stations and associated cables, and replaces them with new control station and associated cables.
  Any associations and relations of these items will remain as before, the To and From relations of the cables will be transferred to the new control stations and the cables. Any additional control station cable will have its To side associated to the control station and its From side will not be assigned.

- You can apply a typical control station to a load, converting equipment item, disconnect electrical equipment item, a generator, and a battery bank.

- If you select a reference control station that has one or more associated control/instrumentation/grounding cables, the software creates these cables in the control station that you add to the project item.
- If you apply control station to one or more items that are connected to circuits, the software associates the control station cables (only instrumentation and control) with the circuits automatically when you select the appropriate Options Manager setting. If you do not use the setting, you need to make the association manually.

- If you modify the components of a reference control station that already has a reference from an item, the software does not update the change in the reference until you next apply a profile to the item.

Apply Typical Schematic

You use apply typical schematic when you need to specify a typical schematic or to modify the associated typical schematics of a group of items.You can associate typical schematics to the following items:

loads, converting equipment, circuits, disconnect electrical equipment, generators, battery banks and buses.

Apply a Typical Schematic to an Item

1.In the Electrical Index, expand the folder hierarchy Electrical Equipment > Loads by clicking the + icons or by double-clicking the folders.
2.Select the Motors, Static Electrical Equipment, or Other Electrical Equipment folder and do one of the following:
   - Expand the folder and select an item from the list.
   - Click the folder, and in the list view pane, hold Ctrl while you select more than one item.
3.Click Tools > Apply Options.
4.On the Apply Options dialog box, click the Typical Schematic tab.
5.Select Enable typical schematic.
6.Beside the Typical schematic box, click Find.
7.On the Find dialog box, click Find Now to display the available typical schematics.
8.From the list of typical schematics, highlight the typical schematic that you want to assign to the item.
9.Click OK.
10.Under Apply to, select one of the following options according to the rule the software uses to Apply the typical schematic:

All items
Applies the typical schematic to all the selected items.

Items with no typical schematic
Applies the typical schematic only to items without an existing typical scheamtic.
Use this option to add a typical schematic only to new items among the items that you selected.

Items with a different typical schematic from the selection
Changes the typical schematic of the selected items to the current typical circuit.
Use this option to add a typical schematic to items with a modified typical schematic definition.
11.Click Apply.

Notes:

- If you modify the components of a typical schematic that already has a reference from an item, the software does not update the change in the reference until you next apply a typical schematic or a profile to the item.
- You can assign a new typical schematic or change an assigned typical schematic for an item at any time.

Apply Typical I/O Set

You apply typical i/o set on item to create in a batch operation set of signals.
You can apply the i/o set on the following items:
Loads,Generators,Battery Banks,Converting Equipment,Circuit,Disconnect Electrical Equipment

Apply a Set of Signals to an Item

1. In the Electrical Index, expand the folder hierarchy by clicking the + icons or by double-clicking the folders.
2. Navigate to the folder for the desired equipment type and do one of the following:
   - Expand the folder and select an item from the list.
   - Click the folder, and in the list view pane, hold Ctrl while you select more than one item.

   Tip
   You can also select more than one item by selecting a subset of item tags in the Tabular Editor.
3. Click Tools > Apply Options.
4. On the Apply Options dialog box, click the Typical I/O Sets tab.
5. Select Enable typical I/O sets.
6. Beside the Typical I/O sets box, click Find.
7. On the Find dialog box, click Find Now to display the available set of signals.
8. From the list of signal sets, highlight the set of signals that you want to assign to the item.
9. Click OK.
10. Under Apply to, select one of the following options according to the rule the software uses to apply the set of signals :

    All items
    Applies the set of signals to all the selected items.

    Items with no typical I/O sets
    Applies the set of signals only to items without an existing set of signals.
    Use this option to add a set of signals only to new items among the items that you selected.

    Items with a different typical I/O set from the selection
    Changes the set of signals of the selected items to the current set of signals.
    Use this option to add a set of signals to items with a modified typical I/O set definition.
11. Click Apply.

    Notes:
    - If you modify the components of a set of signals that already has a reference from an item, the software does not update the change in the reference until you next apply a set of signals or a profile to the item.
    - You can assign a new set of signals or change an assigned set of signals for an item at any time.

Apply Profile

The Apply profile maps the reference profile item properties, to the project item and creates additional items, as specified in the Profile.
If we use motor item type, then for a new motor, it will create new items, and when reapplying a profile on an existing motor, the operation will result in replacing the previously applied and or associated items with the items specified in the new applied profile.
The right timing on whether and when to commit this operation is an engineering decision and would be based on your project state and your workflow.

Apply a Profile to an Item

1. In the **Electrical Index**, expand the folder hierarchy by clicking the + icons or by double-clicking the folders.
2. Navigate to the folder for the desired equipment type and do one of the following:
   - Expand the folder and select an item from the list.
   - Click the folder, and in the list view pane, hold **Ctrl** while you select more than one item.

   Tip

   You can also select more than one item by selecting a subset of item tags in the Tabular Editor.
3. Click Tools > Apply Options.
4. On the Apply Options dialog box, click the Equipment Profile tab.
5. Select Enable equipment profile.

   Tip

   The default profile appears in the Create item based on profile box. You can select a different profile by performing the following steps.
6. Beside Create item based on profile, click Find.
7. On the Find dialog box, click Find Now to display the available profiles.
8. From the list of profile, highlight the profile that you want to apply.
9. Click OK.
10. Under Apply to, select one of the following options according to the rule the software uses to Apply the profile:

    All items

    Applies the profile to all the selected items.

    Items with no profile

    Applies the profile only to items without an existing profile.

    Use this option to add a profile only to new items among the items that you selected.

    Items with a different profile from the selection

    Changes the profile of the selected items to the current profile.

    Use this option to add a profile to items with a modified profile definition.

    Items with the same profile as the selection

    Updates an existing profile for the selected items.

11. Select the **Populate empty item properties** check box to overwrite null data values for target item properties with values that exist in the profile.

Clear the check box to leave the properties empty. (In all cases, the software does not overwrite existing values.)

12. Click Apply.

Notes:

- Applying a profile is a useful way of creating new items and populating large amounts of data in your project. When you apply the profile, the software automatically applies any definitions for typical circuits, control stations, typical I/O sets, and lookup tables that you define on the other tabs of the apply **Options** dialog box.
- After you apply a profile for which the software creates new items and relations successfully, the operation may still partially fail, for example if certain values of updated data conflict with a rule. In such cases, the software displays an error message and records the problem in the log file.

- When you apply a profile, the software replaces all formerly associated items with the items specified in the current profile including any values that you have modified. The software assigns new item tags according to the naming convention.
- If you apply a profile that contains a reference item to one or more project items, the software populates the properties of those project items with values copied from the reference item.
- If you modify the components of a typical circuit that already has a reference from an item, the software does not update the change in the reference until you next apply a typical circuit or a profile to the item.
- When an item has associated cables, and you connect the item to a circuit by applying a profile with a typical circuit, if there is more than one cable, the software associates the free ends of all the cables to the circuit.
- If you apply a profile for a reference motor with a heater to a project motor without a heater, the software adds a heater to the project motor.
- If you apply a profile for a transformer, the software applies the profile only if the reference and project transformers have the same number of windings and where the Number of Secondaries property for the project transformer has a value of 1 or 2. If no value appears for this property, you must type one manually in accordance with the number of secondary windings in the transformer.
- If the item that you are applying a profile to is connected to a feeder circuit, the system replaces the circuit so that it matches the applied typical circuit as specified in the profile.
- If the item is connected to feeder equipment other than a circuit (such as a variable frequency drive, a transformer, or a battery bank), the software does not apply the profile.
- When you apply a profile to an item, the software can overwrite existing data. Since this process is not reversible, you should proceed with great care.

Assigning Loads to PDB

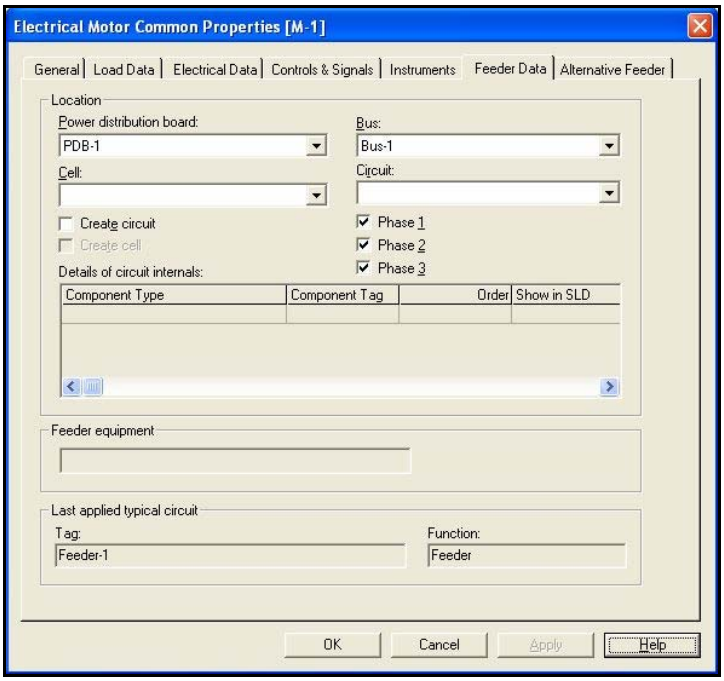
All electrical consumers and distribution equipment (PDB's) need power source. The power source must match the requirements of the supplied equipment, in voltage levels, current consumption and by providing suitable protection and control circuits.

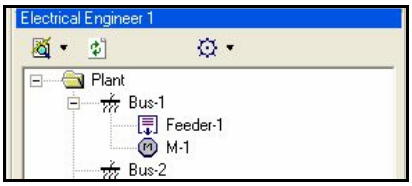
- A load can be assigned and connected to one circuit only.
- A circuit may feed one or more loads or pieces of equipments.
- You can assign the load to a feeder circuit from more than one place:
  1. From the load common properties window.
  2. By drag & drop the load from EI to EE.
  3. By moving the load within the Engineer.
  4. Through the Batch Load Assignment.

From the load common properties window

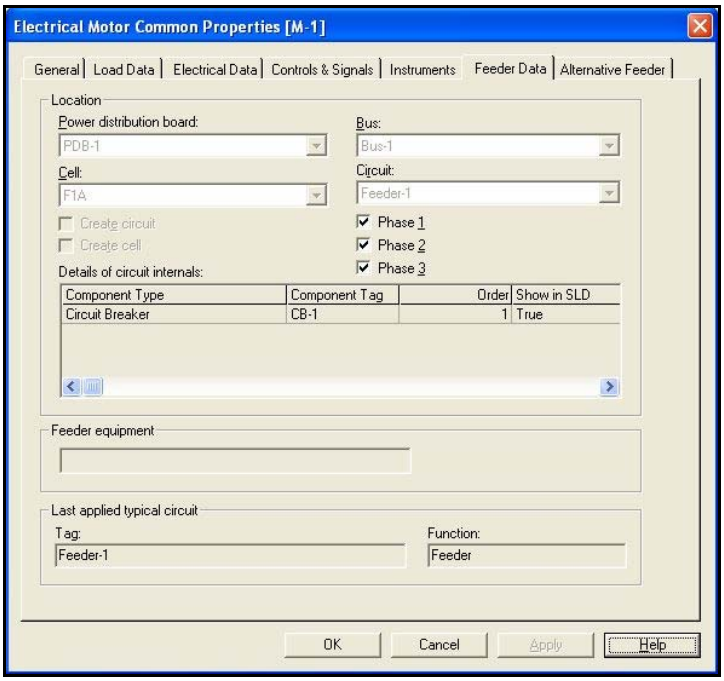
You can assign the load to PDB and Bus from the load common properties, Feeder Data tab.
There are three possbilities to assign the load from the load common properties:

1.Pre-assign a load to a PDB and its bus without creating and connecting the load to a feeder circuit.



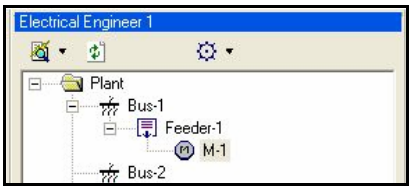


2. You can assign the load to an existing spare or other feeder circuit by selecting the circuit from the appropriate list on the load common properties windows.



Note: Once you associate the load with a circuit, the feeder data becomes read-only.

Further changes are possible only through the **Electrical Engineer**.

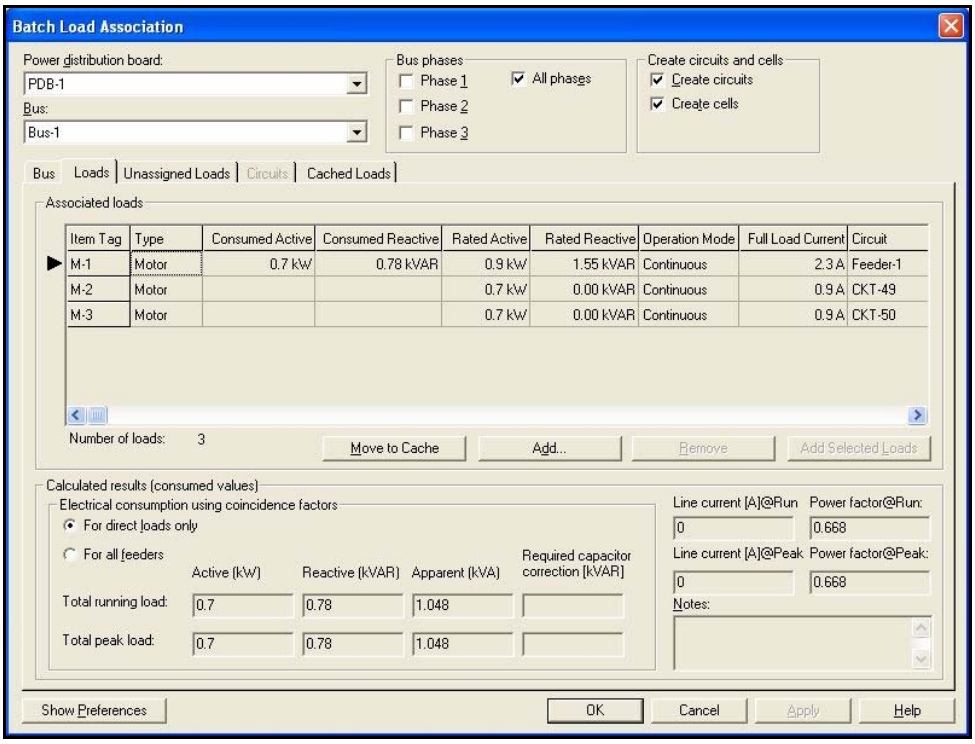


3. You can create a new feeder circuit on the fly by selecting the **Create circuit** check box, (you can also select the **Create cell** check box if required.).
   This option automatically creates a new feeder circuit under the bus, populates the circuit with the components specified in **Last applied typical circuit**, and connects the load to that circuit. (If a load has an associated set of items such as a cable, a junction box, and so forth, the uppermost associated item is connected to the circuit in the Electrical Engineer.)

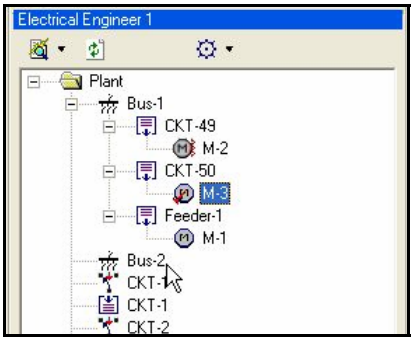
Clicking **Apply** or **OK** commits the operation. Clicking **Cancel** reverts to the previous setting.

Batch Load Association

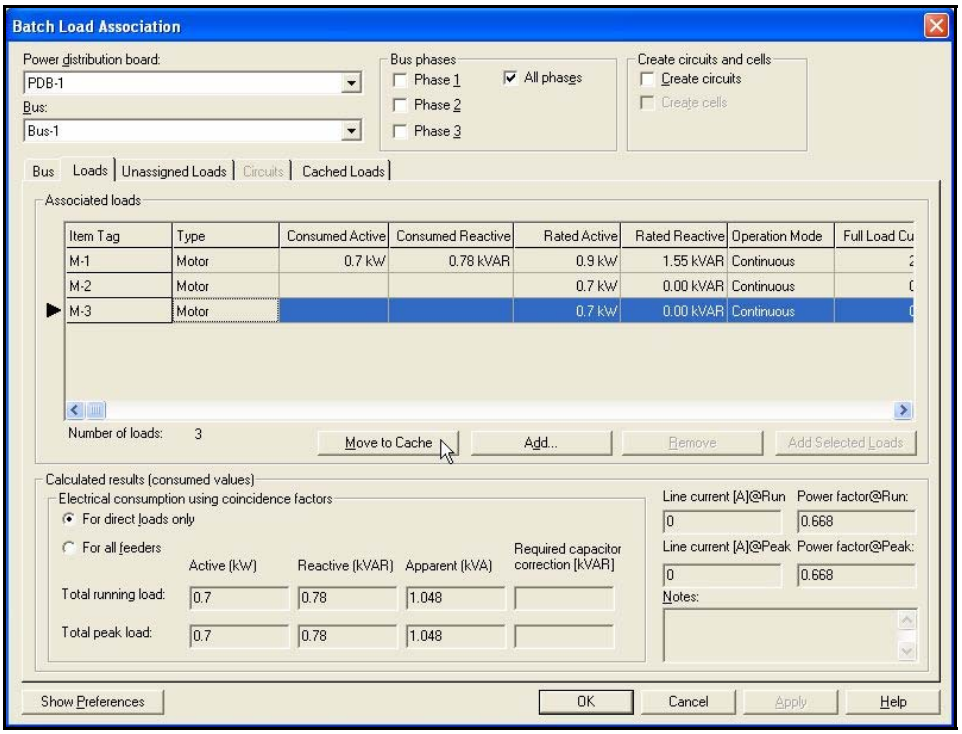
It is common practice to associate loads with a PDB not one by one, but rather using a batch operation based on some common properties. For example, all the motors of Area 1 should be fed by MCC-100, bus A. The user needs a way of assigning loads by selecting a set of loads and associating all the tags to the same PDB and bus by means of a single command.Since this operation provides the total installed load values, it gives the user an ability to perform some iterations on the assignments without actually assigning the loads to final circuits.As with the forms, the user can perform a pre-assignment first by selecting a PDB and bus only, thereby adding the loads but not creating circuits. Alternatively, the user can create circuits on the fly automatically by selecting the **Create circuits** / **Create cell** check boxes.To select a set of loads you select a set of tags using the Tabular Editor and then click **Add Selected Loads**. The calculated values shown are the total calculated electrical quantities of all the assigned loads.



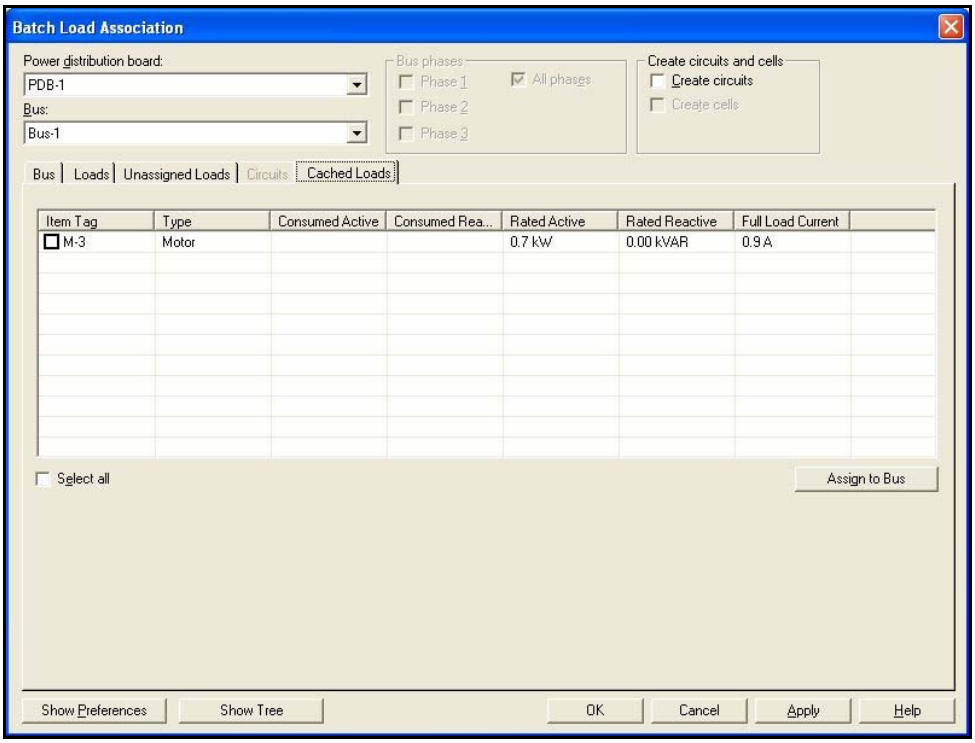
The load balancing will be done by using the move to cache functionality.
One of the biggest challenges in electrical design is to perform load balancing. This is an operation that is done at the early stages of the project, moving loads from one bus to another, watching the total bus load, until buses are evenly loaded.
SmartPlant Electrical supports this operation in the Batch load association form.
It is assumed that this operation is done in early stages of the project at which no feeder cables have been yet created between the loads and the feeder circuits and profiles have not yet been applied and populated and created massive data sets.
Suppose we want to move M3 from PDB-1, Bus-1 to Bus-2



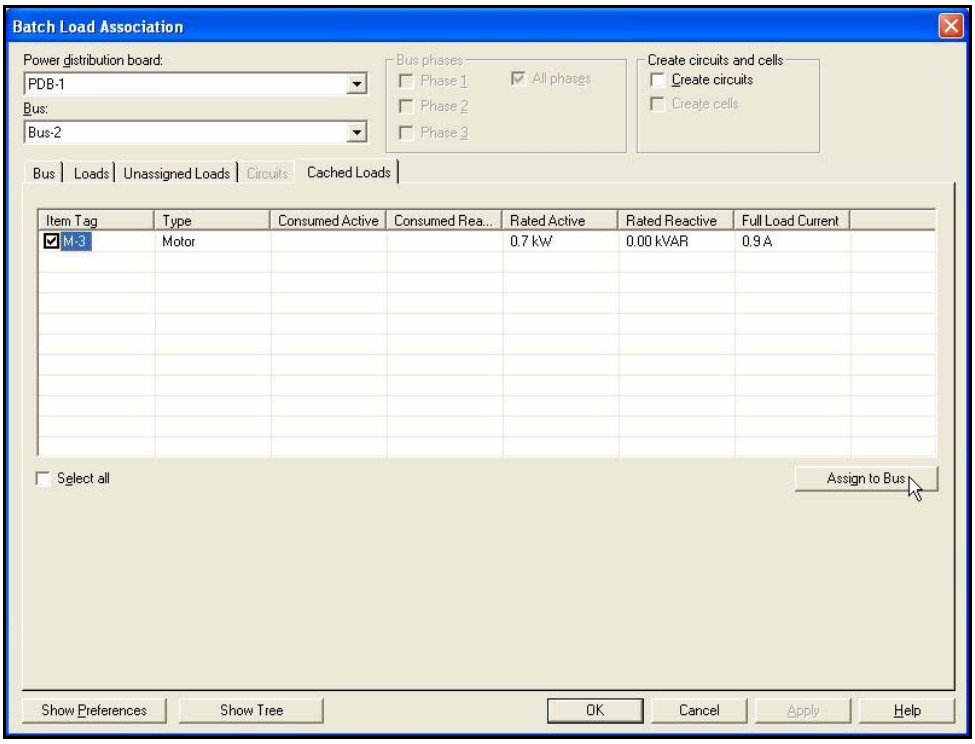
In the batch load assignment, we select the loads and clicking the **Move to Cache** button, which will move the load to the last new tab named **Cached Loads**:

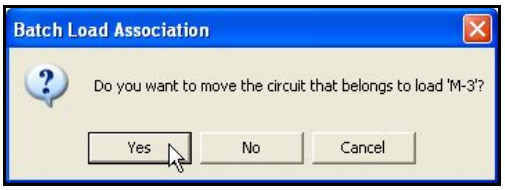


The load will appear in the **Cached Loads** tab:



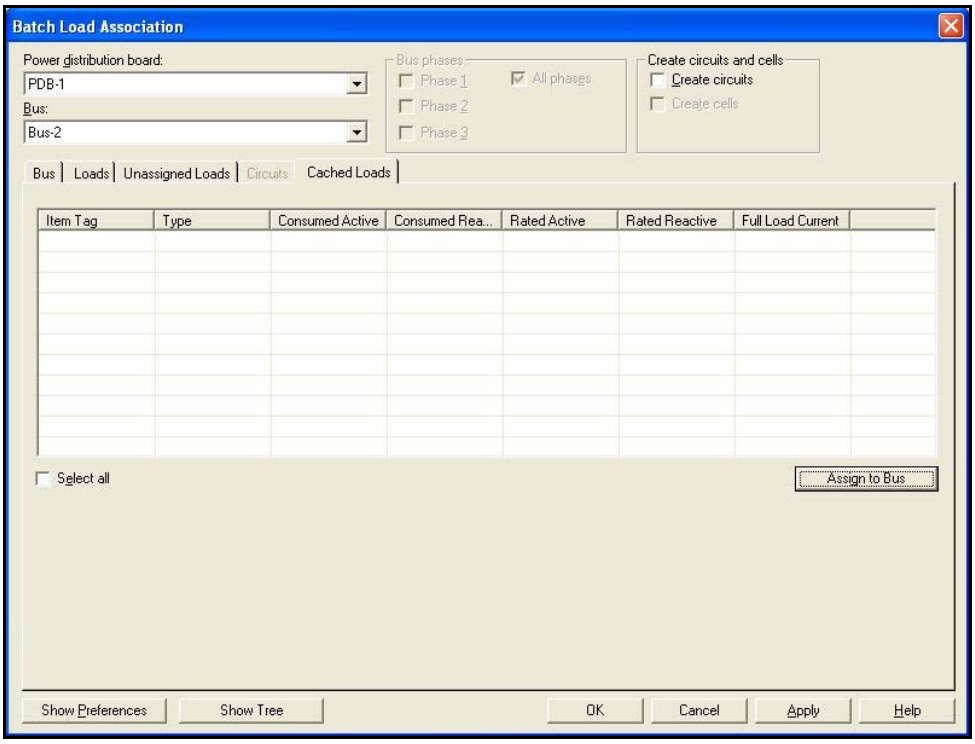
We will click on the Apply to save the operation.
Select the PDB and bus where we want to move the load, and select the ‘Assign to Bus’.

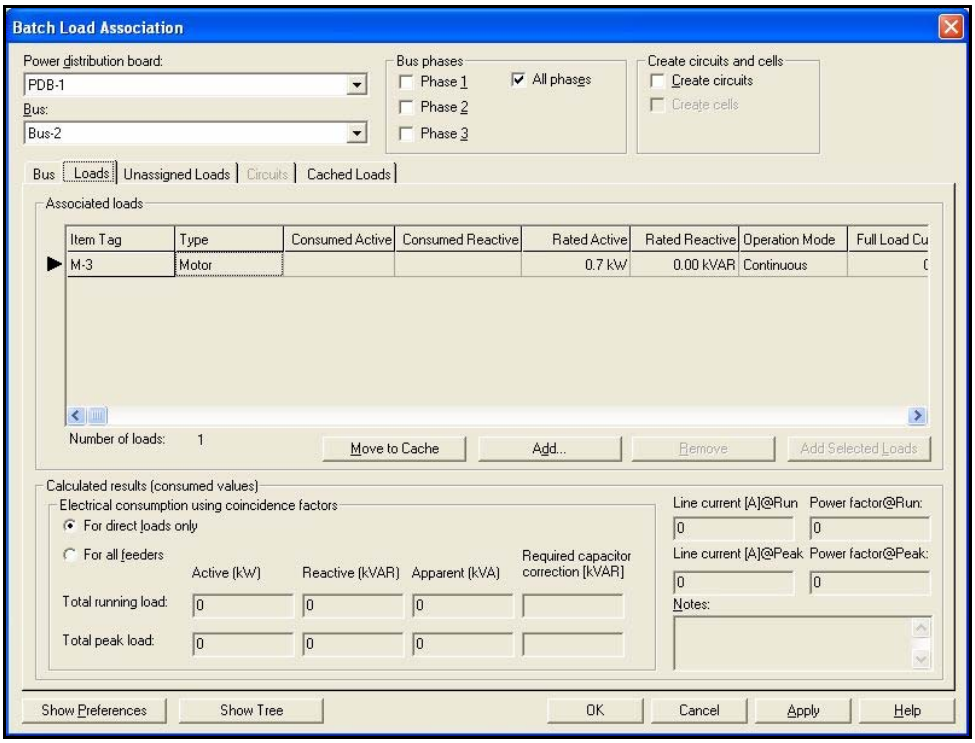


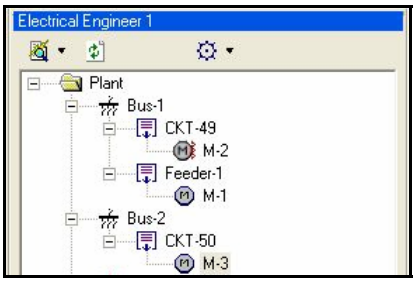


Select Yes to move the circuit.

Note: Select Yes, move the load with the upstream connected items (feeder,cable,etc.).


Batch Load Association
Power distribution board:
PDB-1
Bus:
Bus-2
Bus phases
Phase 1
Phase 2
Phase 3
All phases
Create circuits and cells
Create circuits
Create cells
Bus
Loads
Unassigned Loads
Circuits
Cached Loads
Item Tag
Type
Consumed Active
Consumed Rea...
Rated Active
Rated Reactive
Full Load Current
Select all
Assign to Bus
Show Preferences
Show Tree
OK
Cancel
Apply
Help


Batch Load Association
Power distribution board:
PDB-1
Bus:
Bus-2
Bus phases
Phase 1
Phase 2
Phase 3
All phases
Create circuits and cells
Create circuits
Create cells
Bus
Loads
Unassigned Loads
Circuits
Cached Loads
Associated loads
Item Tag
Type
Consumed Active
Consumed Reactive
Rated Active
Rated Reactive
Operation Mode
Full Load Cu
M-3
Motor
0.7 kW
0.00 kVAR
Continuous
Number of loads: 1
Move to Cache
Add...
Remove
Add Selected Loads
Calculated results (consumed values)
Electrical consumption using coincidence factors
For direct loads only
For all feeders
Active (kW)
Reactive (kVAR)
Apparent (kVA)
Required capacitor correction [kVAR]
Total running load:
Total peak load:
Line current [A]@Run
Power factor@Run:
Line current [A]@Peak
Power factor@Peak:
Notes:
Show Preferences
Show Tree
OK
Cancel
Apply
Help

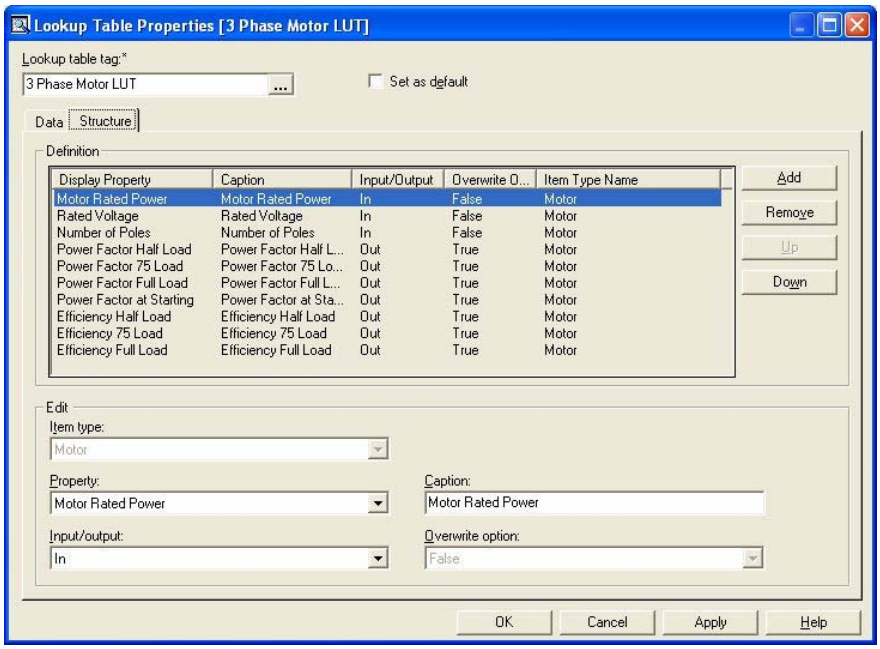


Note: Until loads have been moved and assigned, they will remain in the Cached loads Tab of **each and every PDB,** until assigned to a bus.

Lab 5

Create look up table

5.1. **'3 Phase Motor LUT'**



Motor Rated Power	Rated Voltage	Number of Poles	Power Factor Half Load	Power Factor 75 Load	Power Factor Full Load	Power Factor at Starting	Efficiency Half Load	Efficiency 75 Load	Efficiency Full Load
1 hp	460 V	4	0.52	0.66	0.76	0.60	0.79	0.82	0.83
10 hp	460 V	4	0.68	0.79	0.84	0.50	0.90	0.90	0.90
100 hp	460 V	4	0.80	0.85	0.87	0.30	0.95	0.95	0.95

Apply the look up table '3 Phase Motor LUT' on motors M-100,M-101,M-102,M-103

5.2. Associate motor M-100 with control station CS-M-100-1 (from control station common properties).
5.3. Associate motor M-100 with instrument INST-100-1 (from instrument common properties).

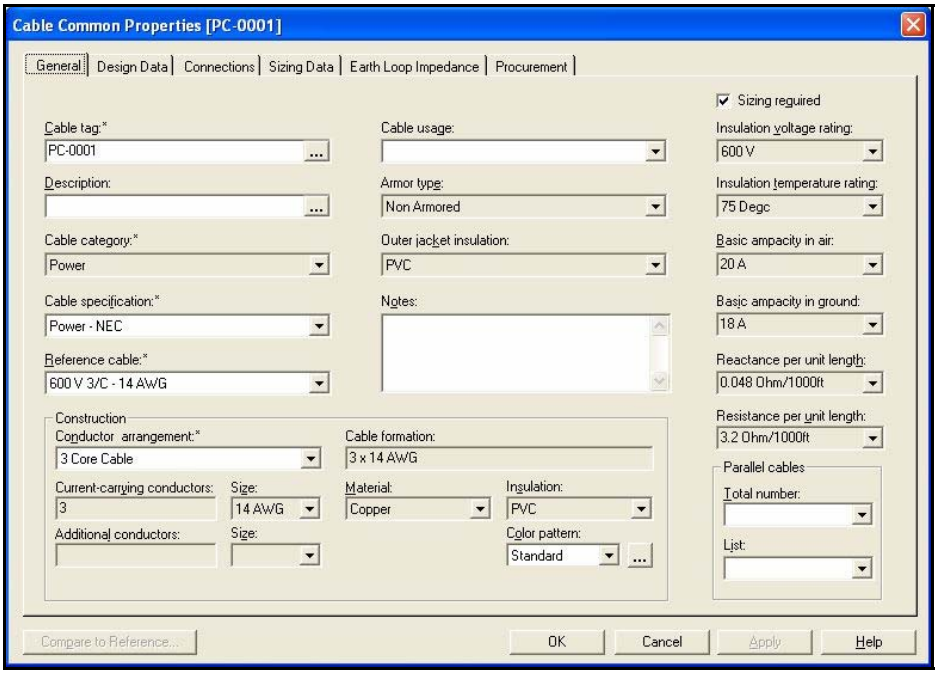
Chapter 6

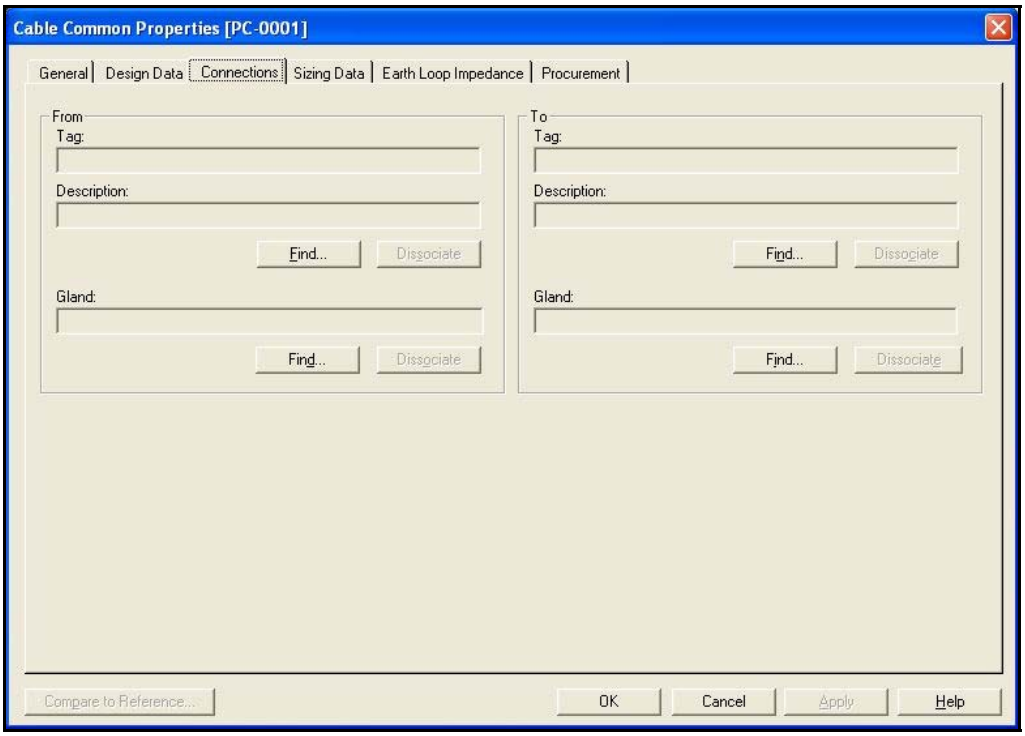
Cable Sizing

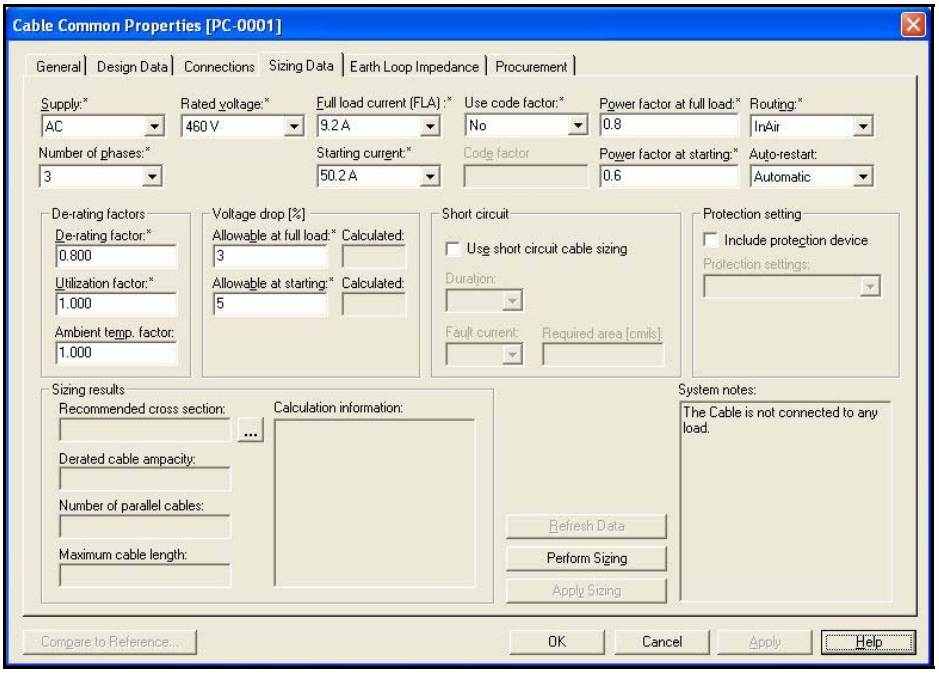
Sizing power cable

Cables can be sized as stand-alone cables (not associated to loads or other objects). In this case, the user needs to provide sufficient electrical characteristics in order to perform the sizing calculation.
The same applies if the cable(s) are associated to loads.
It is important to understand that the cable sizing procedure is iterative. There are several procedures involved, but not always all these procedures are executed.

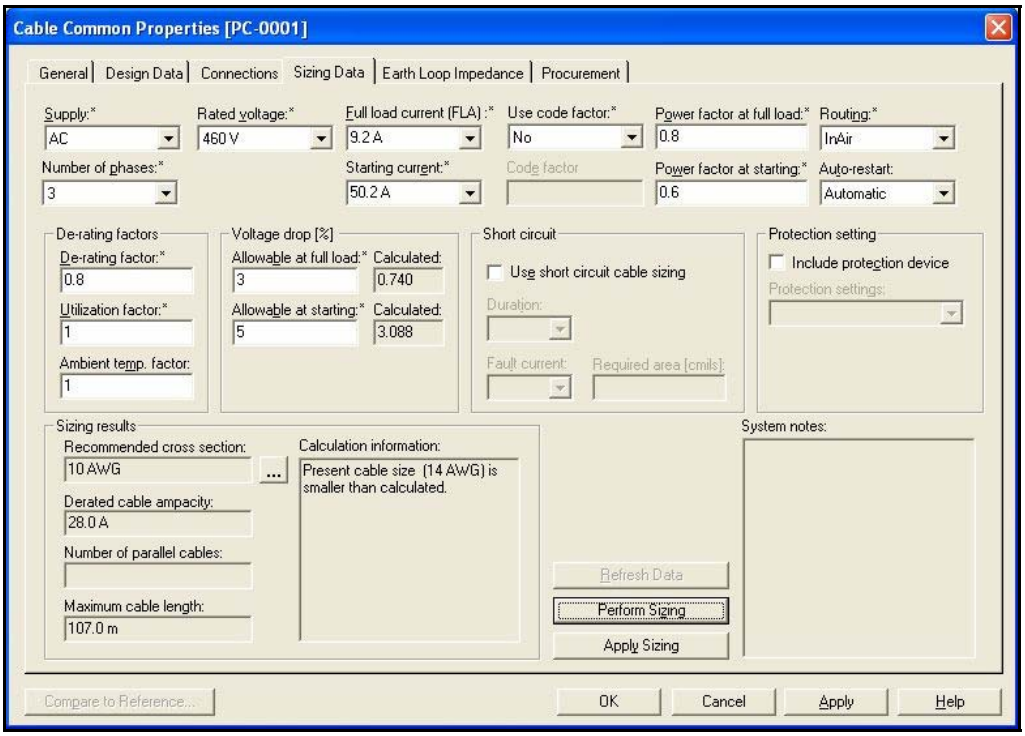
Select the project cable PC-0001 and enter the following values:



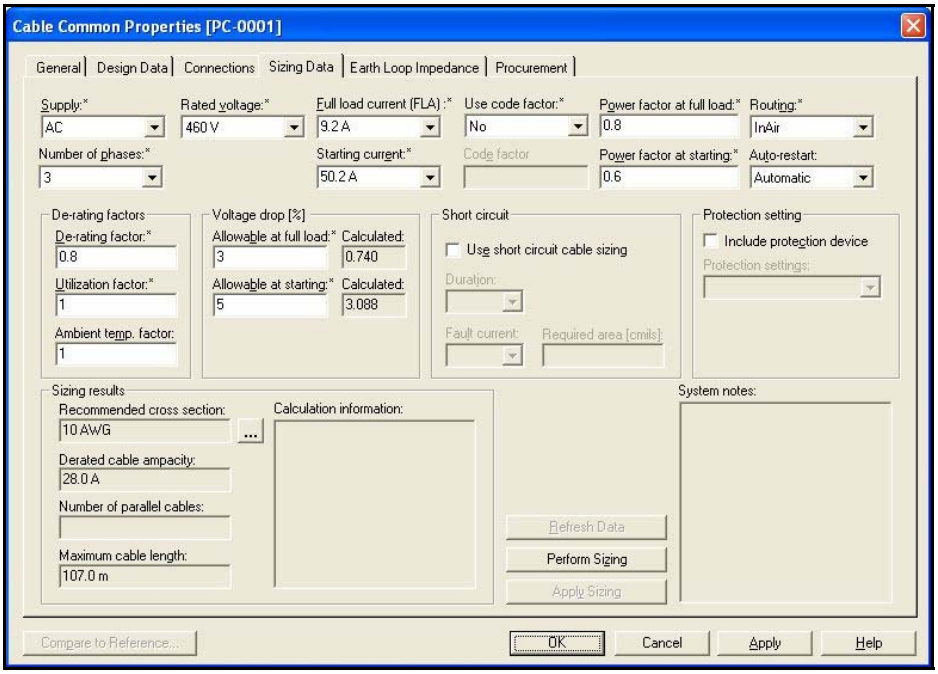


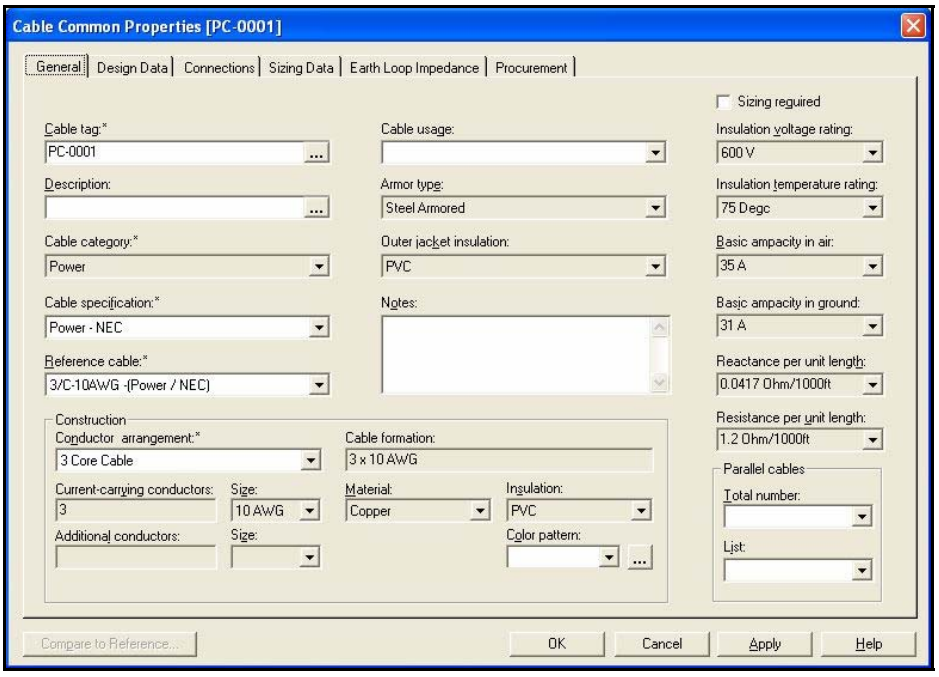


To size the cable, select the perform sizing button.



The present cable size (14 AWG) is smaller than calculated and the software recommended Cross section cable of 10 AWG.
Select the Apply sizing to use the recommended cable size.





The following procedures are done for motors or loads in general when sizing a cable.

Ampacity Sizing

In this procedure the software looks for a cable that satisfies the de-rated ampacity. If the software can not find a cable that can be used, it will try to assign parallel cables (starting from a minimum cable size defined in the options manager and up to a maximum number of allowable cables, also set in the option manager). Once the software finds the cable that matches the required de-rated ampacity, it will proceed to the voltage drop running validation.

De-rating factors

The basic ampacity of the cable applies to standard conditions that the manufacturer specifies. Under actual operating conditions, such as a higher environmental temperature, or a location where several cables are routed close to one another, the ampacity changes. The derating and utilization factors take into account the operating conditions of the cable.

The software let you enter 3 separate values for the de-rating factor:

- **Derating factor** – Value between 0 and 1 which the software uses to calculate the cable derated ampacity under actual operating conditions. The software multiplies the basic ampacity by the derating factor to obtain the derated ampacity. A value of less than 1 means that the derated ampacity is lower than the basic ampacity, and this will result in a requirement for a larger size cable.

- **Utilization factor** – Value between 0 and 1 to represent an additional factor that derates the cable ampacity. If you do not need to use this property, set the value to 1. The software multiplies the basic ampacity by this factor to obtain the derated ampacity.

- **Ambient temp. factor** – Value between 0 and 2 to represent a temperature factor that affects the derated cable ampacity. For the software to apply this factor to the calculation, in the Options

Manager, you must first set the value of the **Apply Temperature Correction Factor to Cable Sizing** property to **Yes**; setting the value of the property to **No** is the equivalent of using a value of 1 for the temperature factor.

The overall derating factor the software is calculated is :
Df= (De-rating factor **X** Utilization factor **X** Ambient temperature factor)

Voltage Drop (VD) at Running (VDN)

In this validation, the software validates the cable voltage drop percentage during normal full load current running conditions satisfies the maximum allowable cable voltage drop that is set by the user.

Voltage Drop at Starting Conditions (VDS)

In this validation, the software validates the cable voltage drop percentage during start of the motor satisfies the maximum allowable cable starting voltage drop that is set by the user.

Important: For loads other than motor (static), only the first and second procedure is carried out. This means that the system does not require the following properties "Power factor at starting", "Starting Current" and "Voltage drop [%] Allowable at starting".
The sizing program will recommend the required cable size from the family of cables that you Specify under cable specification and with the value that you select under conductor arrangment.

Important: The Reference Data Explorer catalog cables will be the source for the sizing and cable selection. It is important to make sure your cable catalog is full of all the standard cable sizes and configurations. If a specific size and arrangement is missing from the catalog, the sizing may find a different cable, which has a higher cross section.

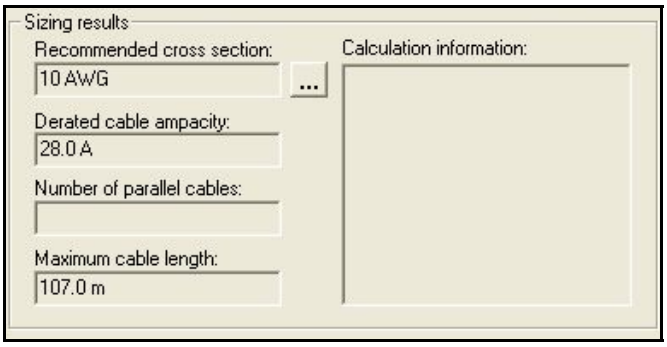
Note: If there isn't any cable found with the specified construction or electrically suitable, the software needs to notify this to the user. The message should say clearly that the sizing of this cable was not successful and a proper cable was not found to satisfy the requirements.
When the cable is associated to a load (the "To" side is associated with the load), the cable should inherit electrical properties from the load. The automatic data transfer is valid for loads only. For other cables association this information needs to be entered manually.

Ambient Temperature Correction Factor

Ambient temperature affects the ampacity of the cable, meaning that in higher ambient temperatures, the cable will be able to carry less current than it is specified in its basic ampacity. The temperature effect can be calculated or taken from tables (NEC tables give these figures).
The program lets you enter the ambient temperature derating factor value manually.
The ability of this factor to affect is set at the Options Manager, general settings.

Calculating Maximum Allowable Power Cable Length

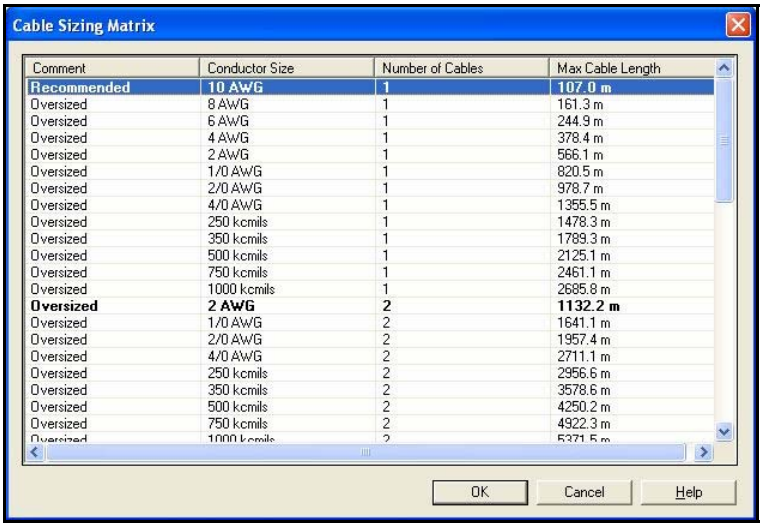
As part of the sizing procedure, the system calculates and shows the maximum permissible cable length for that particular cable type that meets the voltage drop specifications.



Viewing All Possible Combinations and Acceptable Cable Sizes in Sizing Procedure

The system provides the user a window to identify all the recommended/acceptable cables that may be used for a specific data set of electrical parameters. The user will be able to select any of the recommended or acceptable cable configurations of the Electrical Index the single or parallel cables.

Click the  browse button adjacent the recommended/acceptable cross section gives:



This gives the user the whole range of possibilities to select from, the Electrical Index the a single cable or an assembly of parallel cables of different sizes. Selecting a cable from this list will replace the current cable with the selected one.

Fault Short Circuit Cable Sizing

Set 'True' in property grid for 'Enable Short Circuit Sizing', will include an additional step in the sizing algorithm- to match the fault short circuit current values imposed by the network to the cable short circuit capabilities. The algorithm is based on the following formula:

$$\left(\frac{I}{A}\right)^2 t = K \times \log_{10} \frac{T_2 + \beta}{T_1 + \beta}$$

I = Short Circuit current [Amperes]. This is the maximum permissible (allowable) short circuit current that the conductor can withstand.

A = Conductor area [Circular-mils]. See appendix at the end of this document for additional information.
t = Time of short-circuit [Seconds]. This value is determined by the time it take the protective devices to disconnect the power. Some standards use fractions of the frequency. Sometimes, this time is assumed to be 5 or 10 seconds.

K coefficient = constant

Material	K
Copper	0.0297
Aluminum	0.0125

β coefficient = constant. Depends on the conductor material as follows:

Material	β
Copper	234.5
Aluminum	228

T_2 = Maximum allowable short-circuit temperature in Degrees Celsius. This property is the temperature at which the cable-insulation starts to damage. These temperatures have been established for various materials of insulation as follows:

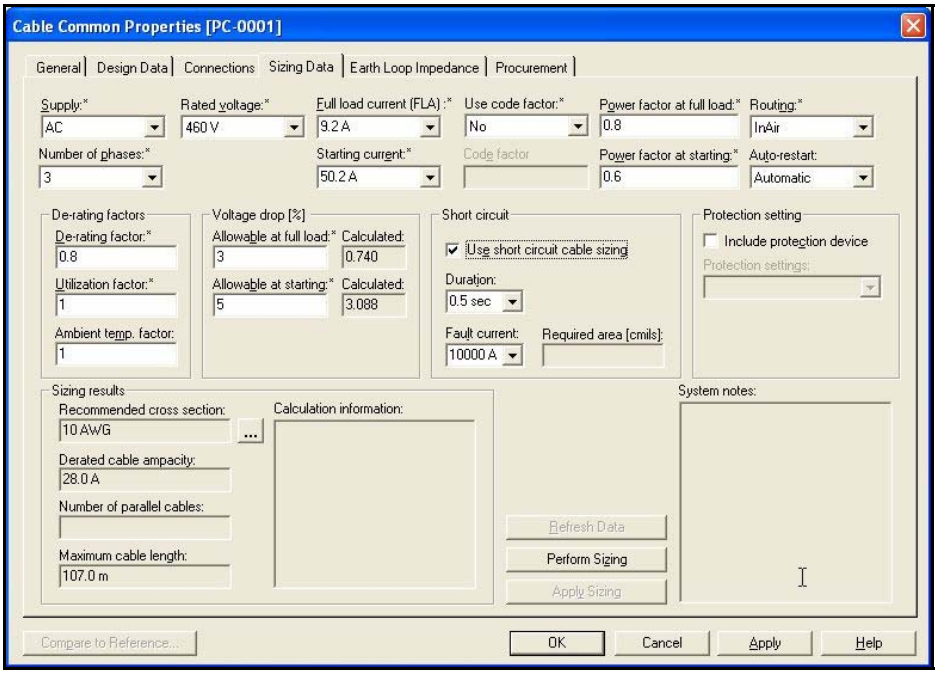
Material	**Temperature T2**
	[°C]
PVC	150
XLPE	250
Silicon Rubber	350

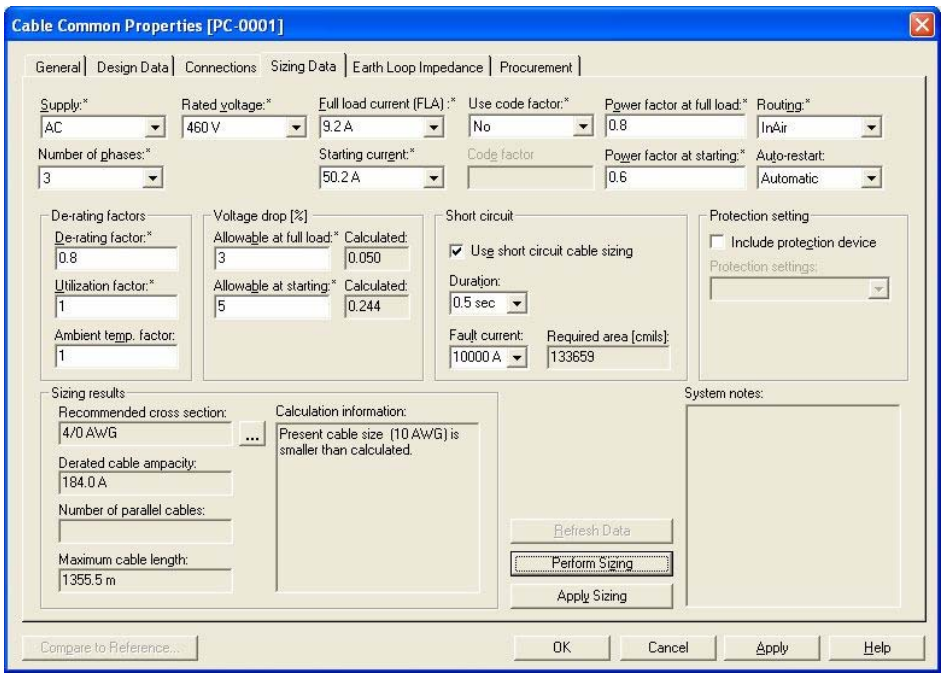
T_1 = Maximum operating Temperature in Degrees Celsius. Values are given below:

Material	**Temperature T1**
	[°C]
PVC	75
XLPE	90

Silicon Rubber	90

The above listed tables are defined at the **Reference Data Explorer** under the Conductor new special look-up tables.
In the **Cable** form there is a new set of properties, related to this topic:





Short circuit duration time is specified by the user (can be defined in data Dictionary).
The required minimum cross section area in cmils will be calculated. The software will recommend a cable with a cmils cross section equal or larger than this minimum calculated.
Project cables and conductors will be automatically updated with the short circuit data of the conductors look up tables. Creating project cables, by dragging from the Reference Data Explorer or in Index or through profiles will apply automatically conductor look up table data, if exists in the project conductors and populate the above mentioned properties with look up data
These properties are read/write enabled in Index, but user shall be instructed that the short circuit sizing procedure will use data from look up tables and will not look at the project conductor property value any time the system will perform the short sizing calculation, the look up table data and values will be used.
Changes made in project cables conductor of the above properties data will NOT set the "sizing required flag." In case look up data is missing the system will issue a message upon trying to perform a short circuit sizing

Properties Copied from Loads

The following properties that are copied from the load (static or motor) to the cable may affect the size of the cable:
Supply (AC/ DC)
Number of phases
Rated Voltage
Full Load Current
Starting Current (for motors only).
Use code factor (for motors only).
Power Factor at full load.
Power Factor at Starting (for motor only).
Auto Restart (flag)

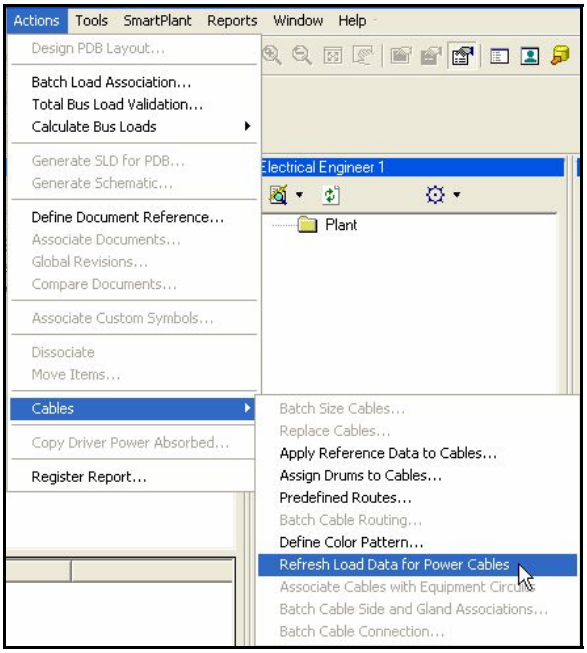
Required Data for Sizing and Selection

For successful cable sizing, the following information is required:
Conductor Arrangement
Estimated Length and/or Design Length – the software needs at least one of them. If both are defined, the software will use the "Design Length" which is the most accurate length.
Supply (AC or DC)
Number of Phases (applies only if AC supply)
Rated Voltage
Use code factor/code factor (valid for motors only).
Power Factor at full load ($0 < value < 1$)
Power Factor at Starting ($0 < value < 1$), (only for motors= LRC Power Factor)
Routing (InAir or UnderGround)
Full Load Current
Starting Current (valid for motors only for voltage drop at starting validation)
Auto Restart flag (currently not in use by the algorithm)
Voltage drop [%] Allowable at Running
Voltage drop [%] Allowable at Starting (for motors only)
De-Rating Factor ($0< value < =1$), default value set at the data dictionary, modified per each cable
Utilization Factor ($0< value < =1$), default value = 1
If not all the information that should apply is provided, the software should notify the user to fill in the missing information. For example, "Enter a value for 'Code factor." Message is displayed when this property is missing (need to verify all required properties).

Refresh Load - Changes in Electrical Data of Load

If a load has an already sized cable and the load's electrical properties are changed, the cable gets a notification that the load's electrical properties changed. In this case, the "Refresh Data" command button of the cable should become enabled.
The electrical properties that should trigger this message are those that affect the sizing of the cable and are copied from the load.
Note: In this case, if the "Sizing" tab of the cable is opened, it will say that the load's electrical properties have changed which means that the user needs to verify the sizing results.

You may also perform this operation in batch, for all the power cables that the Electrical Index loads have been updated by using the following command:



Power Systems

DC Systems

SP- Electrical cable sizing support DC power systems.
DC (Direct current) systems are not commonly used in power industry. These systems will be used in indoor machines usually. DC power requires special motors and usually used in machines where very high precision is required.

AC Systems
AC Single Phase Systems
This type of cable sizing is supported by SP-Electrical.
AC stands for Alternating Current. The cable sizing supports single phase and 3 phase systems.

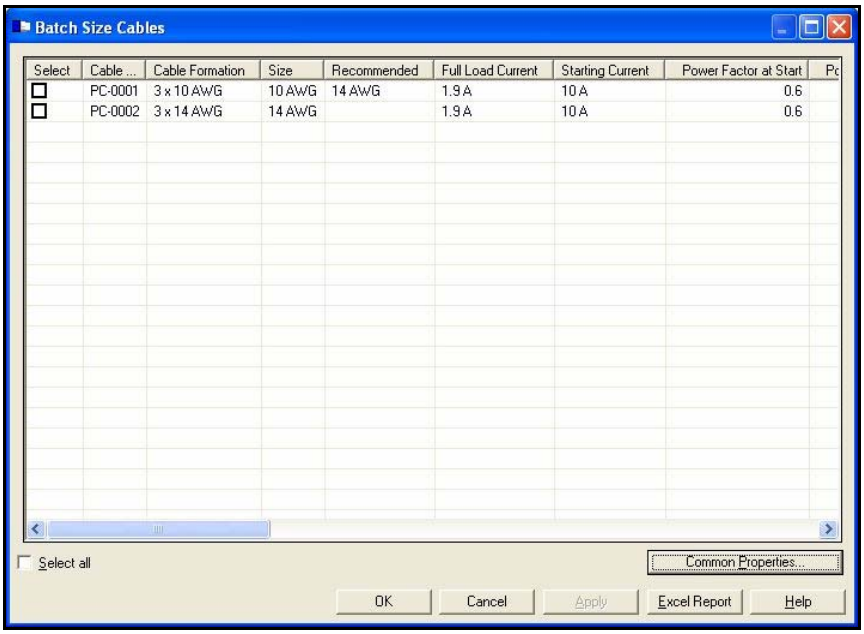
AC Two Phase Systems
We do not support cable sizing of ac other than 1 or 3 phases.

Whenever the user attempts to size a cable that the number of phases is other than 1or 3 the system will issue a message (when trying to perform the sizing):
" Number of phases must be the Electrical Index the 1 or 3, please correct ".

AC Three Phase Systems
This type of cable sizing is supported by SP-Electrical.
Three-phase systems are the most commonly used power systems in the industry.

Batch Sizing of Cables

In order to perform batch sizing: Select multiple cables in the **Electrical Index** or tabular editor -> Actions -> Cables -> Batch size cables.



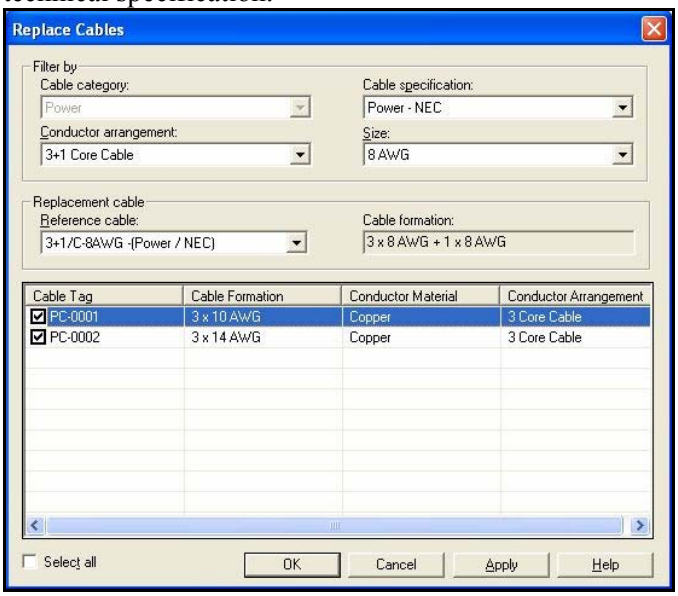
The dialog collects all the selected cables and performs the sizing. If everything goes OK and the sizing passes without any missing data, the recommended sizes are populated in the grid.

If there is missing data that prevents the system from sizing the cable(s), the "Comment" column states there is missing data. If data is missing, all that you need to do is right-click the cable record to open the Cable Common Properties form and add the missing data.
To apply sizing, check the select check box for each cable you need to perform the sizing, and hit Apply Sizing .
The batch sizing process is usually necessary upon a change in cable design parameters or change in loads, where sizing of individual cables would be time consuming.

Replacing Cables

Replacing project cables is possible in two ways: manual replacing cable and batch replace.

The rules that should apply are as follows:
When a cable is replaced by another catalog cable, it should get the cable characteristics of the newer catalog cable: voltage rating, temperature ratings, ampacity, reactance and resistance, formation, sizes, materials etc.
Replacing a project cable by another should retain project data of the original cable:
Name and service, Design data: Procurement, from/to, associations to objects, and sizing results.
Any other cable property defined by the project that does not pertain to the catalog cable construction or technical specification.

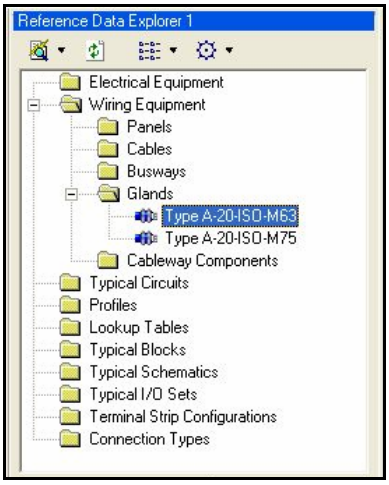


In case there are Parallel cables, all these cables are replaced as per the selected cable
Cable construction (number of conductors, size, formation) is adjusted according to the new selected cable.

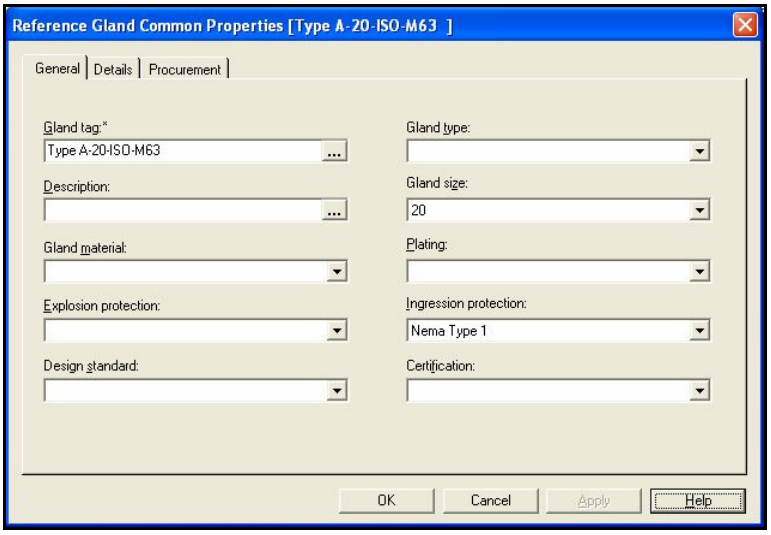
Note: After replacing the cables, you should perform a cable sizing operation.

Glands

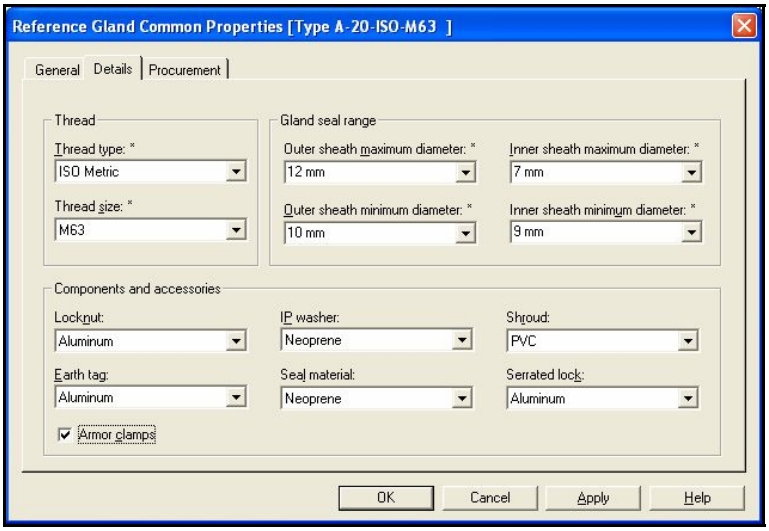
Reference cable glands can be created in the **Reference Data Explorer**:



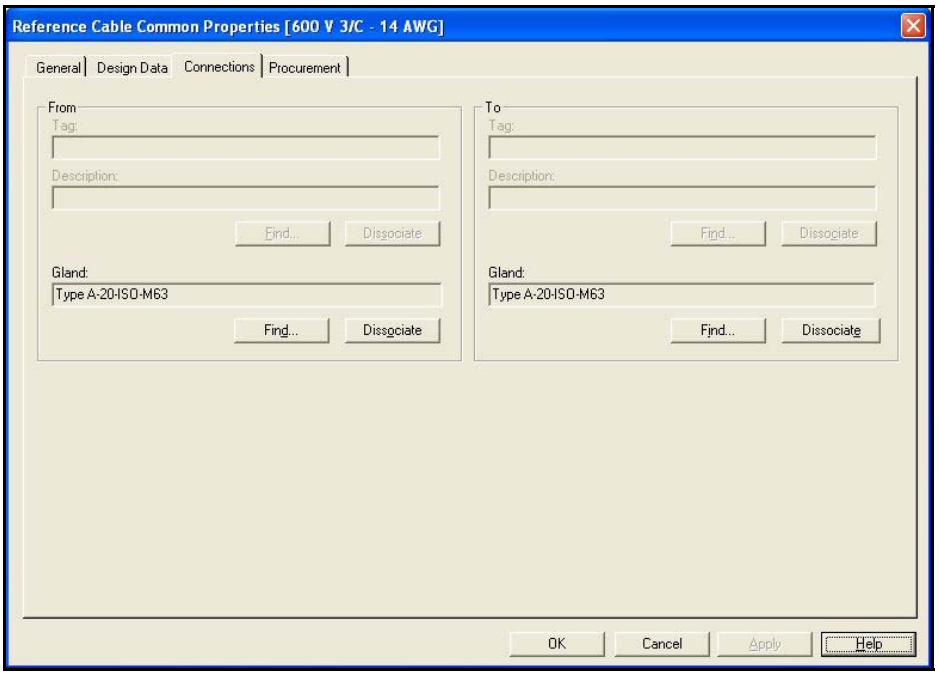
With the use of the following UI specify the major gland data:



While the General tab defines its certification and type basic properties, the Details Tab specifies the more detailed dimensions.



While the inner and outer dimensions provide information on the size of cables that it can accommodate, the Armor clamp flag, when checked means that this gland is suitable for armored cable (has metal armor) You may associate reference glands to reference cables in the **Reference Data Explorer** from the Connections tab:



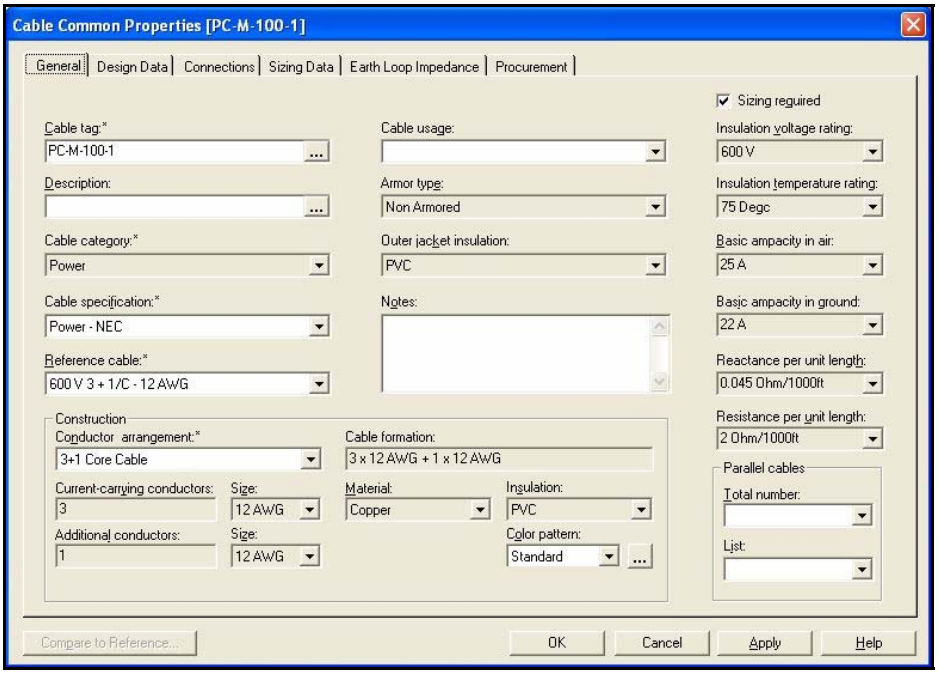
This will allow creating the project cables already fit with some default glands.
Draging a ref cable or performing apply option that involves cables will create project glands.
Project Cables can be associated with reference glands on a one to one basis and in batch.

Lab 6

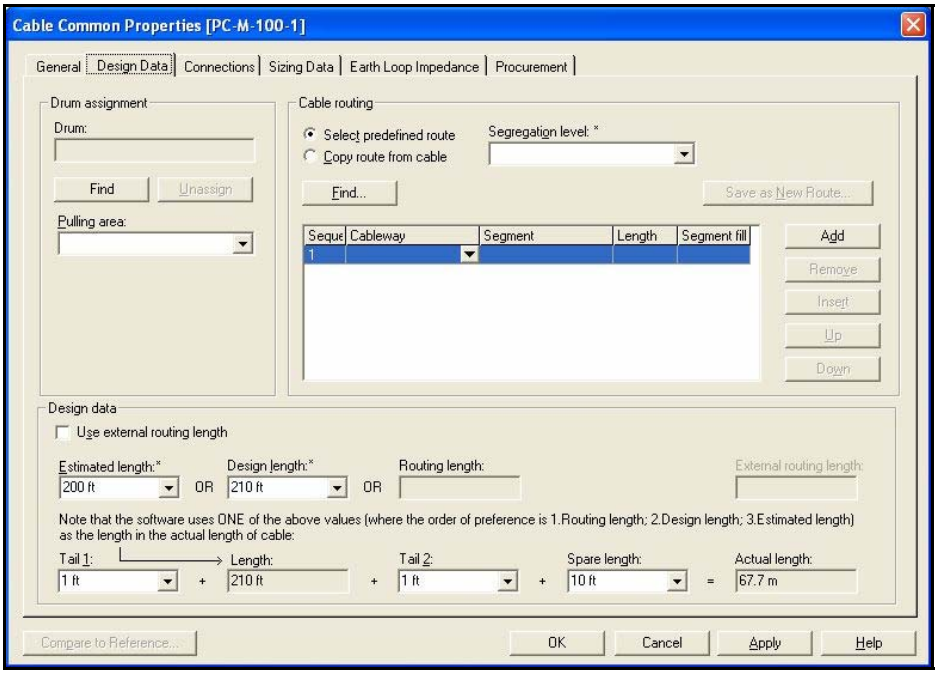
6.1. Size the project power cable '**PC-M-100-1'**

Enter the following data:

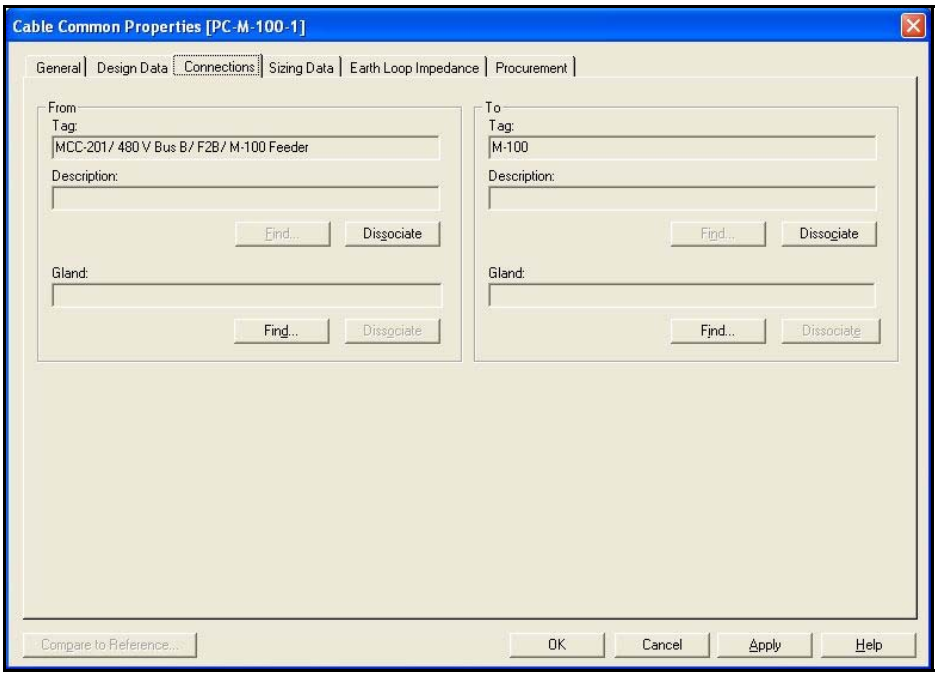
General tab



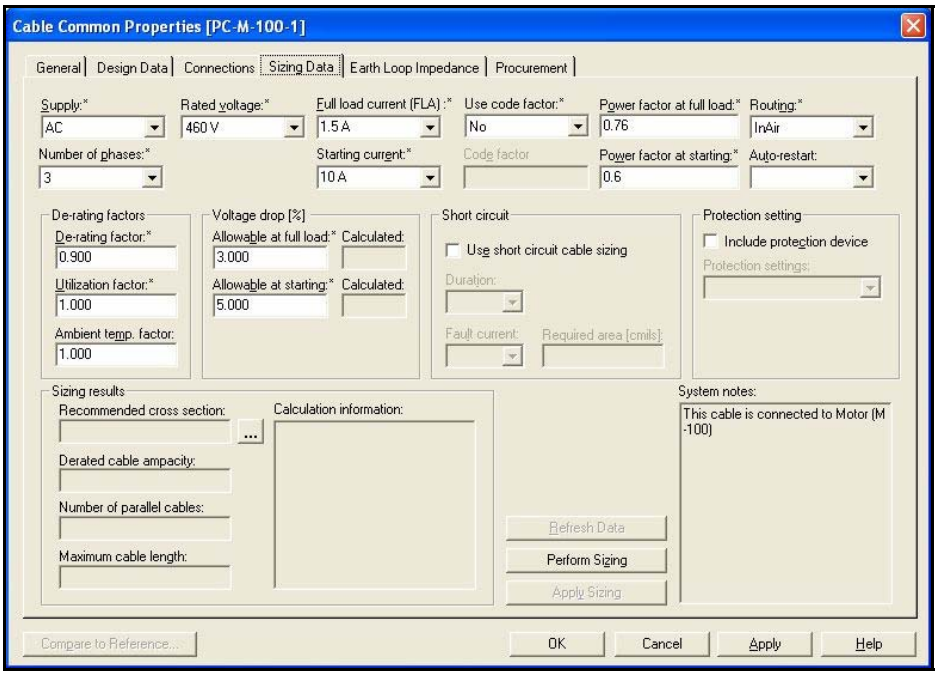
Design Data tab



Connections tab



Sizing Data tab



6.2. Change the design length to 700 ft and size again.

Chapter 7

Reports

Reporting is the process of retrieving information from the database and displaying the information as formatted output. At any time during the design creation process, you can create a report. Each report consists of a Microsoft Excel workbook and a report definition, which describes the data to collect and how to organize the data in the workbook.
Each report you create is based on an item type. This item type serves as the starting point for collecting data for your report. Examples of item types include motors, cables, control stations, and buses. Several default report templates already exist; however, reports are fully customizable. You can create your own reports that contain the information you want to see in a format you choose.

Report Writer

The SmartPlant Electrical report writer lets you quickly and easily create and modify reports. The report writer runs with Microsoft Excel and therefore allows all formatting to be done with Excel formatting commands: for example, drawing lines, sorting column data, and inserting objects. When you edit the SmartPlant Electrical report, you use an add-on 

toolbar to define and map properties in the database to a specific column in the Excel file. This toolbar also lets you filter the database information, sort the information, and format the column header information.
You open the report writer from the reports menu, which let you create new reports, modify existing reports, delete reports, and run the reports, plant report, or user-level reports (my report).
Plant reports are saved in the server directory (Option Manager specifies this default directory)
My reports are saved locally on the user machine (~\Profiles\<username>\My Reports\Output).

Creating New Report Templates

To create a new report template, click Reports > New.

The Report dialog box lets you create a new report from by selecting the Blank value for the Source template. You can also select any existing report to use as a template or starting point for a new report of that item type. All reports stored for the project and the reports stored for the current user are displayed in this list.

You can define three report types for a new report: Fixed format, Tabular format, and Composite format.

Fixed format. For a report that is defined as fixed format, each item that you select for the report is printed on a separate sheet with no repetition of data. The fixed format type is similar to the instrumentation spec. sheet reports.

Tabular format. A tabular report prints all of the items selected on a single sheet. Most of the SmartPlant Electrical delivered reports are defined as tabular format.

Composite format. A composite report is a combination of fixed format and tabular format. The first sheet is fixed format and contains the data common to the items on the subsequent sheets. The subsequent sheets have the specific item data. For example, a pump vendor report would have the vendor's name and information on the first sheet, and the listing of the pumps from that vendor on the second sheet.
If you click Add to project reports, the report is stored in the project reports location defined in Options Manager. If you do not click this checkbox, the report is stored in the user report directory. By default, the user report directory is located in your user name profile directory.

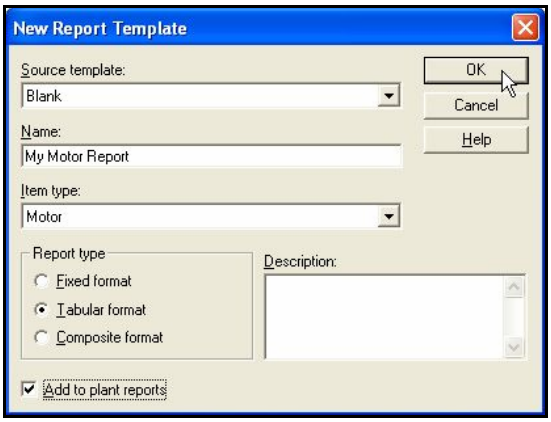
SmartPlant Reports Toolbar

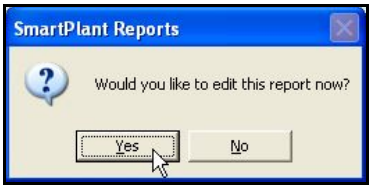
After you create or modify the report, Microsoft Excel is opened, and the SmartPlant Reports toolbar is displayed in the worksheet.

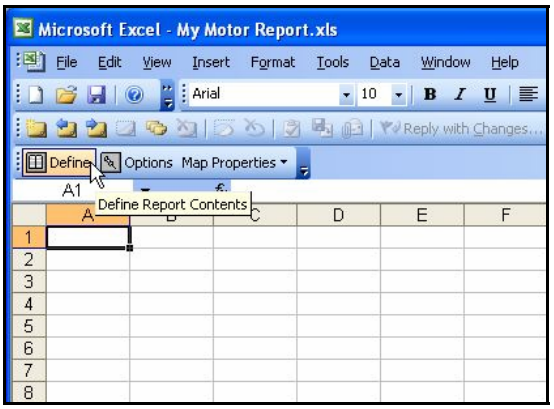
The toolbar lets you define attributes that are available to the report and define the sorting criteria for this data. The toolbar also lets you filter the data for a specific value or range of values. You define the report header information here as well as map the defined attributes to specific cells.

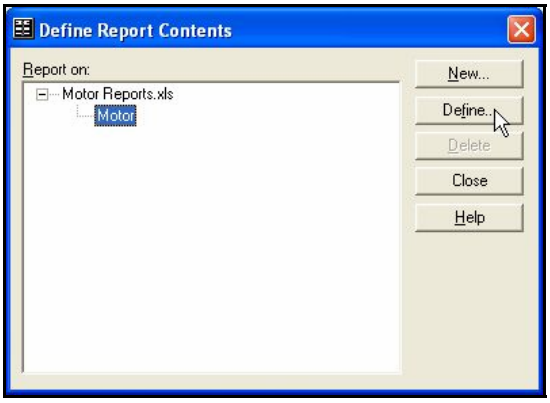
When you click Define on the SmartPlant Reports toolbar, a dialog box is displayed, allowing you to create, edit, and remove items in the report tree view.
When you click Define on the Define Report Contents dialog box, a list of attributes is displayed for that report item. From this dialog box, click Sort to sort by one or by multiple attributes in ascending or descending order. The Filter tab allows discrimination on a certain value or values for the property.
Items and Properties that can be used in the Reports
Defining items and the properties that you wish to see in your report is procedure that requires a knowledge in the data model of Smart Plant Electrical.
There can be reports that are very easy to retrieve the required data and there are such that are more complex. The available list of items and related properties available is based on the data model associations, and you may need to select a single item type or use a chain of item types- all depending on your needs. There is more than one way of accessing items and data, and it depends from which item type.

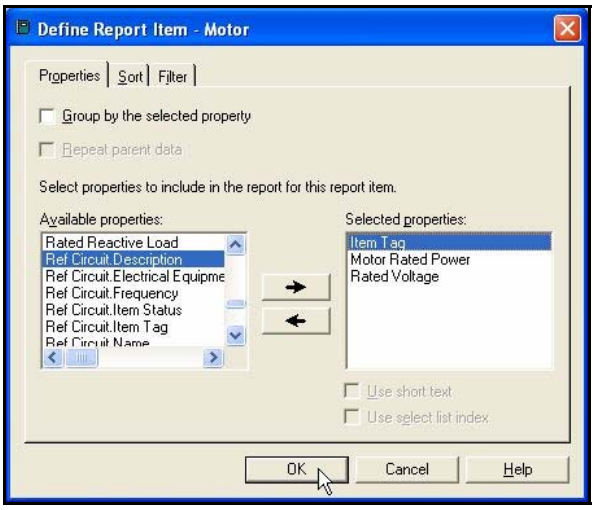
To create a simple report that retrieves properties from motor item type do the following:

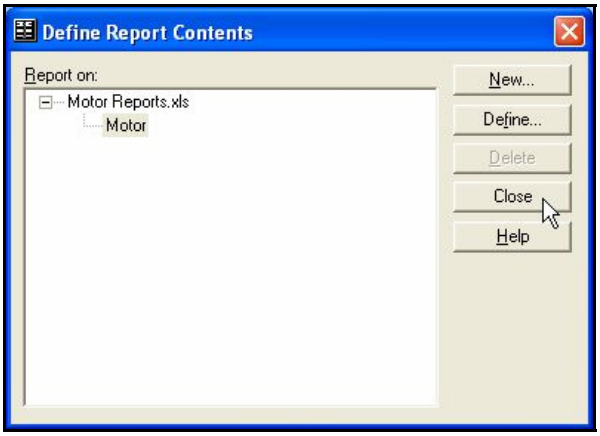

New Report Template
Source template:
Blank
Name:
My Motor Report
Item type:
Motor
Report type
Fixed format
Tabular format
Composite format
Description:
OK
Cancel
Help
Add to plant reports

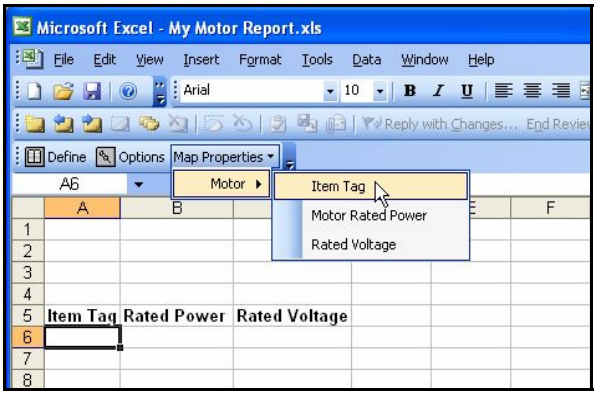

SmartPlant Reports
Would you like to edit this report now?
Yes
No

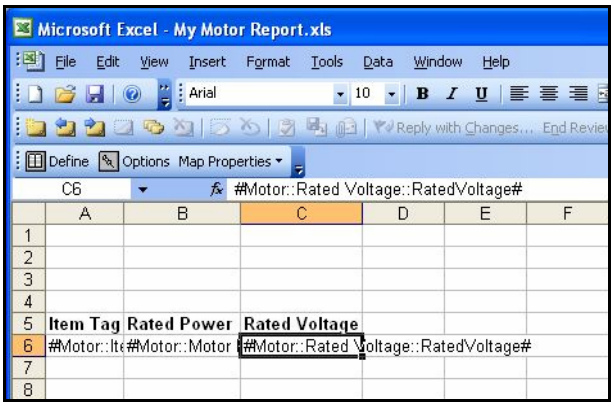

Microsoft Excel - My Motor Report.xls
File Edit View Insert Format Tools Data Window Help
Arial 10
Reply with Changes...
Define Options Map Properties
A1
Define Report Contents
A B C D E F
1 2 3 4 5 6 7 8

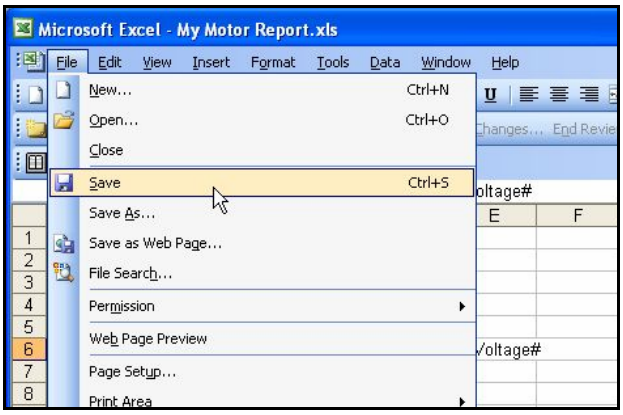

Define Report Contents
Report on:
Motor Reports.xls
Motor
New...
Define...
Delete
Close
Help


Define Report Item - Motor
Properties
Sort
Filter
Group by the selected property
Repeat parent data
Select properties to include in the report for this report item.
Available properties:
Rated Reactive Load
Ref Circuit.Description
Ref Circuit.Electrical Equipme
Ref Circuit.Frequency
Ref Circuit.Item Status
Ref Circuit.Item Tag
Selected properties:
Item Tag
Motor Rated Power
Rated Voltage
Use short text
Use select list index
OK
Cancel
Help

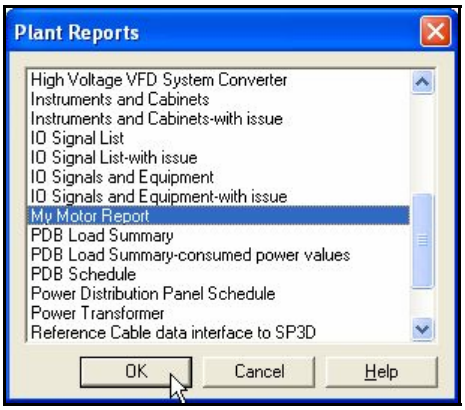

Define Report Contents
Report on:
Motor Reports.xls
Motor
New...
Define...
Delete
Close
Help

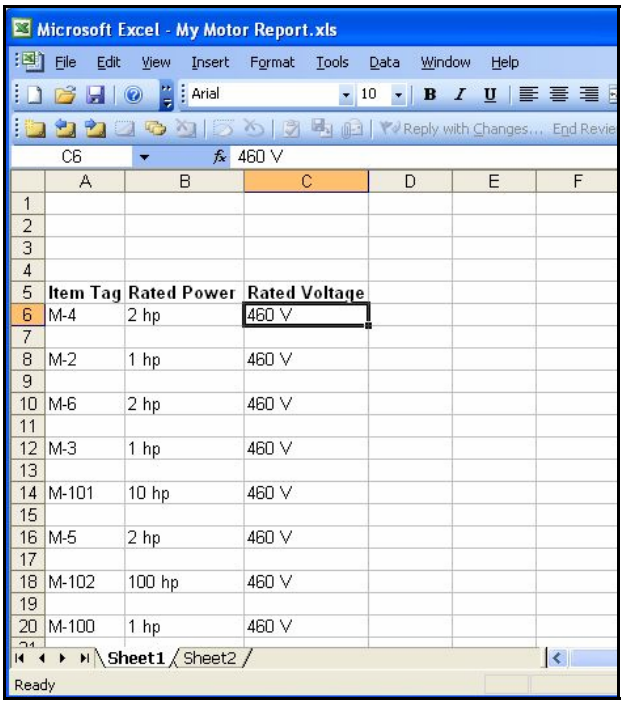

Microsoft Excel - My Motor Report.xls
File Edit View Insert Format Tools Data Window Help
Arial 10
Define Options Map Properties
A6
Motor
Item Tag
Motor Rated Power
Rated Voltage
Item Tag Rated Power Rated Voltage





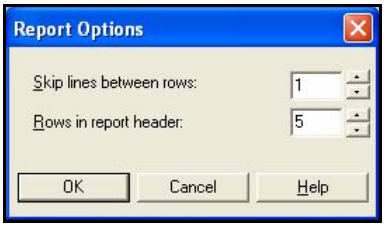
To run the report:





Report Options

Options on the SmartPlant Reports toolbar define the report header and basic information in the body of the report. The Report Options dialog box applies only to tabular and composite reports because the fixed report prints each set of data on a different sheet.



Skip lines between rows defines the number of lines between each printed row of data. Rows in report header defines the basic report information. Project name, unit name, date, report title, and company logos are examples of information that goes in a report header. This information is printed on each page of the report. Also, if any of the information mapped in the header changes, the software prints a new page with the header information. Because a page break occurs if the value for an attribute changes in the report header, the basic report information (attributes) should not be mapped in any rows in the report header. Otherwise, every time a value changes in the header, the information prints on a new page. In the example above, the mapping of attributes would begin in row 6 or after.

You may cause the header to be displayed on each page by File-→Page setup and selected the Sheet tab to "row to repeat at top"

Map Attributes

After you define attributes, they are available to be mapped to specific columns and rows in the spreadsheet. Use Map Attributes on the SmartPlant Reports toolbar to place attributes. Point to the cell to insert the data, and select the item from the Map Attributes list.

Editing Report Templates

To edit an existing report, click Reports > Edit. In the Edit Report Template dialog box, you can change the title and open the template in Excel for editing.

Deleting Reports

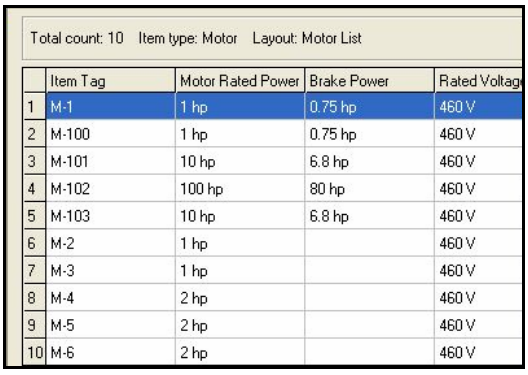
To delete existing reports, click Reports > Delete. Depending on how project permissions are set, you can delete a single report or multiple reports.

Running Reports

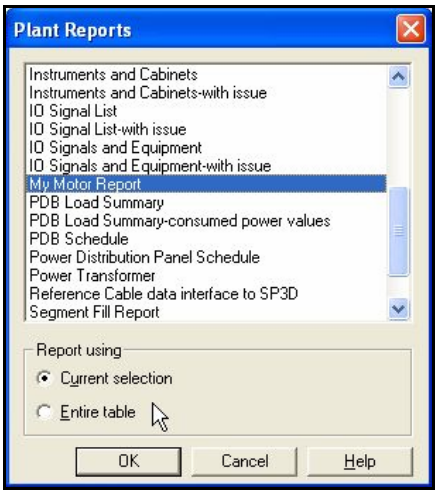
Reports >Plant report or My Report, and select the report you want to run.

Run Reports and Specify Items Through Tabular Editor

You can use the tabular editor to filter data when running the report.



	Item Tag	Motor Rated Power	Brake Power	Rated Voltag
1	M-1	1 hp	0.75 hp	460 V
2	M-100	1 hp	0.75 hp	460 V
3	M-101	10 hp	6.8 hp	460 V
4	M-102	100 hp	80 hp	460 V
5	M-103	10 hp	6.8 hp	460 V
6	M-2	1 hp		460 V
7	M-3	1 hp		460 V
8	M-4	2 hp		460 V
9	M-5	2 hp		460 V
10	M-6	2 hp		460 V



Current selection - Produces a report containing the items currently selected in your current table. This option is not available if no table items are selected.

Entire table - Produces a report containing the contents of the Tabular Editor.
There is no need that the opened table will have properties matching the properties of the report. One property is enough.

Shipped Reports

SmartPlant Electrical provides you with a number of predefined reports (more complex reports that use embedded VB code to fetch and organize data) that user can customize or use as reference to create his own standard. The following list is a partial list out of the ones actually shipped:

I/O Signal List – with issue

I/O Signal List is a report that is intended for publishing signal data to SmartPlant Instrumentation. The report includes all I/O signals defined in the units of the current plant. This report provides the following information:

·I/O signal tags
·Description
·I/O types
·Host
·Card
·Channel
·Address
·CS tag
·Reference instrument loop

I/O Signal and Equipment – with issue

I/O Signal and Equipment - with issue is a report that is intended for publishing signal and equipment data, or just equipment data to the integrated environment. The report includes all I/O signals and equipment defined in the units of the current plant. The report provides the following information:

·I/O signal tags
·Description
·Equipment tags
·Item type
·I/O type
·Host
·Card
·Channel
·Address
·CS tag
·Reference instrument loop

Instrument and Cabinet Load List – with issue

Instrument and Cabinet Load List is a report that is intended for publishing instruments and cabinets that have been retrieved from SmartPlant Instrumentation as loads to the integrated environment. The report actually displays data for all electrical equipment types; however, when you use it for publishing data to the integrated environment, the software automatically selects data for instruments and cabinets only; you should not use a filter to do so. This report provides the following information:

·Instrument / cabinet tags
·Tag types
·Description
·Instrument rated power
·Cabinet rated power

Electrical Load List

Electrical Load List is a report that you generate based on the load item type. The software retrieves the load tags with the relevant load tag data, such a rated power, brake power for motors, as well as values for efficiency, power factor, full load current, and so forth.

PDB Load Summary

PDB Load Summary is a report that you generate based on a load item type. The purpose of this report is to provide a list of loads that are associated with power distribution boards, to sum the electrical kW, kVAR values of these loads and, based on the individual coincidence factors, calculate normal and peak consumptions that enable you to estimate the desired capacities that a particular PDB has to deliver.

PDB Load Summary – Consument Power Values

The PDB Load Summary - Consumed Power values report displays consumed power values instead of rated power values. The rest of the report is identical to the PDB Load Summary report.

All Feeder Load Summary (Enhanced) Report

The All Feeder Load Summary (Enhanced) report is a summary of the loads that are connected to the buses of a power distribution board that you select in the **Electrical Index** or the **Electrical Engineer**. This report is based on either the consumed or rated power of the selected loads.
The All Feeder Load Summary (Enhanced) report provides a list of loads that are associated with power distribution boards, to sum the electrical kW, kVAR values of these loads and, based on the coincidence factors of the relevant buses or the individual loads, calculate normal and peak consumptions that enable you to estimate the desired capacities that a particular PDB has to deliver. The enhanced report sums the loads that are connected to the downstream buses. This ability to look for downstream loads that are fed from the selected bus is sometimes called "drill down" or "roll up".

Cable Take-Off

Cable Take-Off is a report that provides a summary of all the existing project cable types, including the unsized cables.
The report specifics are as follows:
▪ Provides technical details about the cable types.
▪ Calculates the total estimated and design lengths of each cable type.
▪ Calculates the lengths of single core conductor configuration cables based on the number of conductors of each cable.
▪ rovides estimated cable lengths both in meters and feet.
▪ Allows you to expand each cable type to view the individual cable tags.

Cable Schedule – with issue

Cable Schedule - with issue is a report that lists all the existing project cables for publishing to the integrated environment.
This report provides the following information:

▪ Cable tag names
▪ Cable types
▪ Estimated and design cable lengths
▪ Store part numbers
▪ Cable origin and destination sides (the **To** and **From** cable sides). The **To** and **From** information ▪ includes the tag of the electrical item that the cable is associated with. If one of the cable sides is assigned to a PDB, the report displays the path of the circuit tag: PDB/ bus/(cell)/ circuit.

Cable Sizing Summary

Cable Sizing Summary is a report that lists all the cable sizing input and output data for a specified power cable. You are required to create a filter for this report to display power cables only. This report is useful as a quick reference for your sizing considerations and criteria at any time during the life cycle of your project. This report provides the following information:

▪ Cable tag name.
▪ Cable origin and destination sides (the **To** and **From** cable sides). The **To** and **From** information ▪ ▪ ▪ includes the tag of the electrical item that the cable is assigned to.
▪ Input values such as load voltage, full load current, number of phases, and so forth.
▪ Sizing results such as reference cable, conductor size, calculated voltage drop, and so forth.

Cable Wiring Schedule

Cable Wiring Schedule is a report that provides a list of selected cables and their conductors showing to which terminal strips and terminals the cables are connected. The report is based on the item type of the cables for which you want to generate a report.

Terminal Strip Schedule Reports

SmartPlant Electrical provides users three terminal strip schedule reports. These reports show wiring information from the perspective of the wiring equipment that you select in the **Electrical Index**. The

connection between the terminals and the conductors is based on the Conductor-Terminal table. The following reports are available:

- Terminal Strip Schedule - Panel - A report based on the panels you select in the **Electrical Index**.
- Terminal Strip Schedule - PDB - A report based on the power distribution boards you select in the Electrical Index.

Drum Composition

Displays the cable assignments for each drum in your plant.

Segment Width Estimate

You can use the Segment Width Estimate report for calculating the tray widths and number of layers needed so that you can select appropriate reference cableway components.

Cableway Component Material Take-Off

Displays the quantity of items required for each type of reference cableway component when ordering items from the manufacturer.

Cableway Segment Schedule

Displays the list of cables in each cableway segment that makes up your cable routing.

Segment Fill Report

Displays the fill status and tray loading for each segment and indicates whether or not the segment is overfilled.

PDB Schedule

PDB Schedule is a report that you generate based on the circuit item type. The report specifics are as follows:

- The report lists, for all the power distribution boards in the current plant, all the circuits, loads, circuit internal components with the technical details of each circuit.
- The report includes the associated schematic drawings.
  The report uses a predefined filter to display the associated schematic drawings.

Power Distribution Panel Schedule

PDB Schedule is a report that is intended for single phase loads connected to distribution panels. The report specifics are as follows:

- The report lists, for all the power distribution boards in the current plant, all the circuits, each with its protection device type and current rating (the software supports a single protection device per circuit) and the loads associated with each circuit.
- The report sorts the circuits by phase and then by circuit (all the circuits for phase 1, then phase 2, and then phase 3).
- The report calculates the following: Full load current for each phase, Total load current of the bus, Total kW per phase, Total kW per bus.
- This report is not suitable for PDBs that include 3-phase loads.

Lab 7

7.1. **Create custom report 'Power Cable List'**
Source Template: **Blank**
Report Name: **Power Cable List**
Item type: **Cable**
Report type**:** **Tabular format (plant report)**
Attributes: '**Item Tag'**, '**Cable formation'**, '**Cable Specification'**, '**Cable Category'**,'**Ref Equipment.Item Tag'.**
Filter: **Power cables** (filter to retrieve power cables only).

7.2. Save and close the report.
7.3. Run the report and check the result.
7.4. Close the report.
7.5 Run the report using the tabular editor and explore the 2 options you have when running report with tabular editor.
7.6. Close the report.
7.7. Open and modify the report, **'Power Cable List',** to add sort.
Sort: **Item Tag** (accending order).
7.8. Save the change.
7.9. Run the report and check the result.
7.10. Close the report.

Chapter 8

Electrical Engineer

Electrical Engineer manages the electrical associations among equipment in the plant.The Electrical Engineer, combined with the Electrical Index as a source, provides for:

- Navigation through the electrical network.
- Connection between the power distribution equipment of the network infrastructure.
- Modification of relations between the power equipment (electrical flow of energy).

The Electrical Engineer shows any electrical entity that has at least one electrical relation.
In the root of the tree view, the Electrical Engineer shows entities which have electrical children but not parents.

The Electrical Engineer allows you to generate and display single line diagrams (SLDs) that show the electrical network distribution powered from an item selected in the Electrical Engineer.

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

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

- The Electrical Engineer displays all electrical items that have at least one electrical association.
- Certain items appear automatically in the Electrical Engineer after you create them in the Electrical Index. These items appear in the Electrical Engineer even if they do not have any electrical association. These items are offsite power supplies,generators,buses, circuits, battery banks, battery chargers,transformers,UPSs, and current limiting reactors.
- The Electrical Engineer automatically displays power cables that the software creates when you create a load or when you activate the Apply Option command to automatically associate that load with a feeder.
- For a multiple cable assembly, you can hide or display the individual cables by right-clicking the icon, and on the shortcut menu, clicking Show Multiple Cables. The software indicates the display toggle state by a check mark.
- Items that have no hierarchical parents appear at the root of the Electrical Engineer provided that they feed other items.

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

- You can open more than one instance of the Electrical Engineer.

Electrical association between equipment in electrical engineer, can be done by drag and drop operations from electrical index items to electrical enginer, or drag and drop operations in the electrical engineer, follow certain rules.

In addition to the above, the Electrical Engineer provides the following functionality from the right-click menu:

- **Expand All Sub-Items**
- **Show Only**
- **Show in a new window**
- **Show related items in a new window**
- **Rename**
- **Dissociate an Item from its Feeding Item**
- **Select Alternative Power Source**
- **Common Properties**
- **Add cable**
- **Apply Optiopns**
- **Find Other Parents Items**
- **Find Item in Electrical Index**
- **New SLD**
- **Open SLD**
- **Find in SLD**
- **Associate custom Symbol**
- **Total Bus Load Validation**

Working with the Reference Electrical Engineer

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

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

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

Lab 8

Connection of the electrical equipments created in the previous labs to complete
The on line diagram, 'S-001 Demo SLD'.

Note: Appendix A, includes the on line diagram,'S-001 Demo SLD'.

8.1. Connect cable PC-TS-001-1 to offsite power supply S-001.
8.2. Connect cable PC-TS-001-2 to generator G-001.
8.3. Connect transfer switch, circuit TS-001 Incomer 1 to cable PC-TS-001-1
8.4. Connect transfer switch, circuit TS-001 Incomer 2 to cable PC-TS-001-2
8.5. Connect current limiting reactor CLR-001 to transfer switch, circuit TS-001 Feeder.
8.6. Connect cable PC-SW-300-1 to current limiting reactor CLR-001.
8.7. Connect switchgear SW-300, circuit SW-300 Incomer to cable PC-SW-300-1
8.8. Connect cable PC-T-001-1 to swithgear SW-300, circuit SW-300 Feeder.
8.9. Connect transformer T-001 primery to cable PC-T-001-1.
8.10. Connect cable PC-MCC-201-1 to transformer T-001, secondery S-001-480 V.
8.11. Connect cable PC-MCC-202-1 to transformer T-001, secondery S-002-480 V.
8.12. Connect MCC-201 Incomer B to cable PC-MCC-201-1.
8.13. Connect MCC-202 Incomer B to cable PC-MCC-202-1.
8.14. Connect cable PC-SH-100-1 to MCC, 'MCC-201', Bus, '120 V Bus A', Circuit, 'SH-100 Feeder'.
8.15. Connect space heater 'SH-100' to cable 'PC-SH-100-1'.
8.16. Connect cable PC-M-100-1 to MCC, 'MCC-201', Bus, '480 V Bus B', Circuit, 'M-100 Feeder'.
8.17. Connect motor 'M-100' to cable 'PC-M-100-1'.
8.18. Connect cable PC-101-1 to 'MCC-201', Bus, '480 V Bus B', Circuit, 'M-101 Feeder'.
8.19. Connect motor 'M-101' to cable 'PC-M-101-1'.
8.20. Connect MCC, MCC-201, Bus, '480 V Bus C', Circuit, MCC-201 Riser to
MCC, MCC-201, Bus, '480 V Bus B', Circuit, MCC-201 Coupler.
8.21. Connect cable 'PC-M-102-1 to MCC, 'MCC-201', Bus, '480 V Bus C', Circuit, 'M-102 Feeder'.
8.22. Connect motor 'M-102' to cable, 'PC-M-102-1'.
8.23. Connect cable 'PC-M-103-1' to MCC, 'MCC-201', Bus, '480 V Bus C', Circuit, 'M-103 Feeder'.
8.24. Connect motor, 'M-103' to cable, 'PC-M-103-1'.
8.25. Connect cable 'PC-H-104-1 to MCC, 'MCC-201', Bus, '480 V Bus C', Circuit, 'H-104 Feeder'.
8.26. Connect heater, 'H-104' to cable, 'PC-H-104-1'.
8.27. Connect MCC, MCC-202, Bus, '480 V Bus C', Circuit, MCC-202 Riser to
MCC, MCC-202, Bus, '480 V Bus B', Circuit, MCC-202 Coupler.

Chapter 9

Single Line Diagram

SmartPlant Electrical can generate the following single line diagram types:

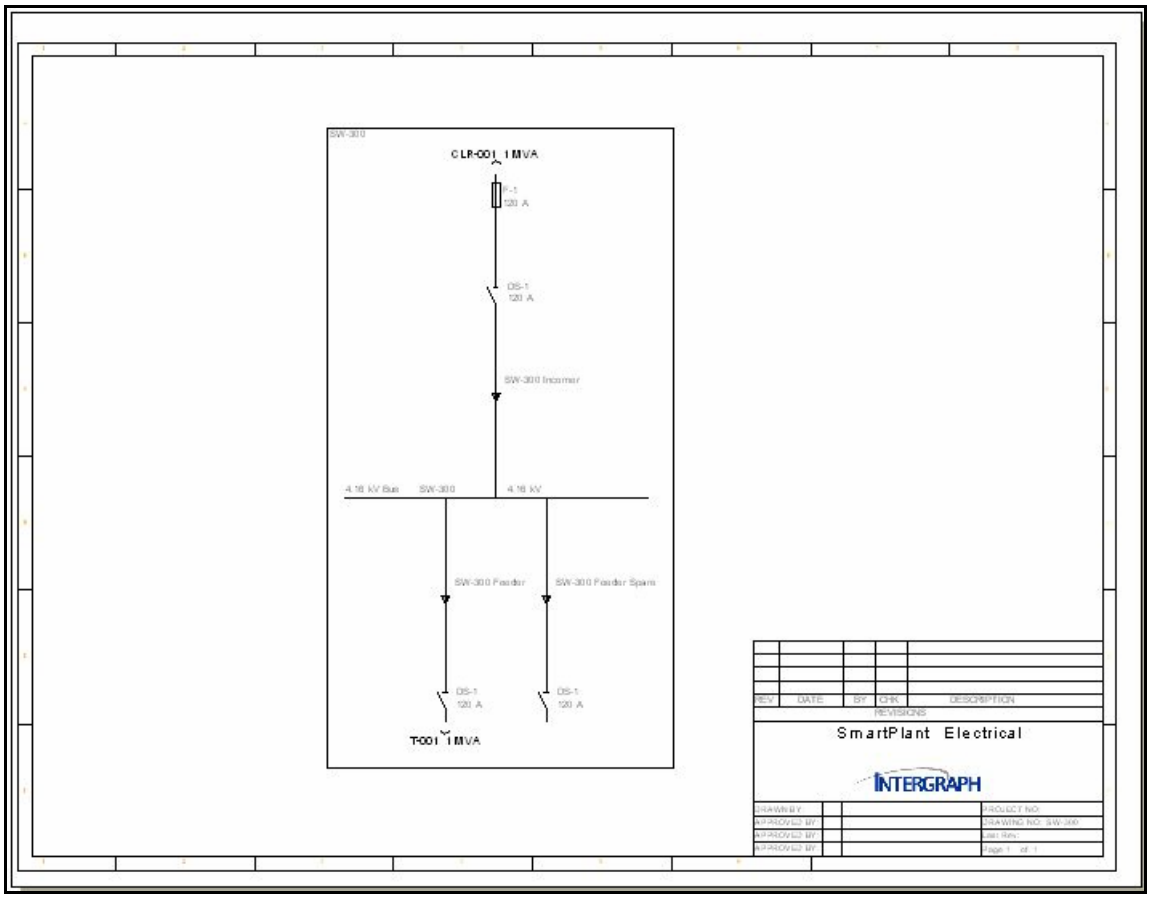
1.A power distribution board-based single line diagram (automatically type SLD)
2. An Electrical Engineer-based single line diagram (automatically type SLD)
3. A blank single line diagram template (Manual type SLD)

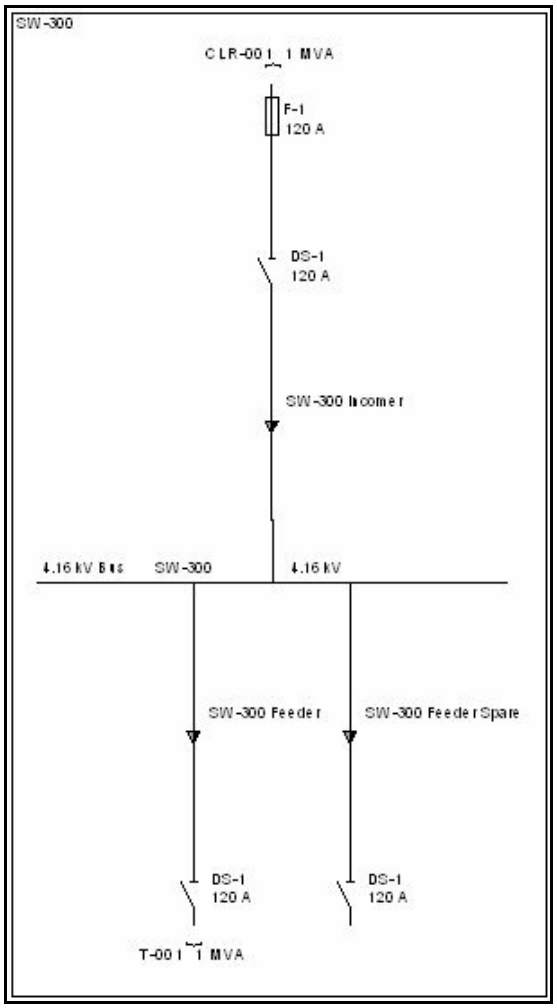
A power distribution board-based single line diagram

Is a document that includes a power distribution board general arrangement drawing and a power distribution board (PDB) schedule. The software generates a single line diagram for a selected power distribution board that contains all the PDB buses, circuits, circuit components, and their inter-connections (bus couplings).

To generate power distribution board single line diagram, do the following steps:

1.From Electrical Index, select PDB, 'SW-300'.
2.Right click on 'SW-300' and select '**Generate SLD'.**
3. In the Single Line Diagram Options, type 'SW-300' for document number, and set your preferences.
4. Select **Generate** button to start generate the SLD.


SW-300
CLR-001 1 MVA
F-1
120 A
DS-1
120 A
SW-300 Incomer
4.16 kV Bus SW-300
4.16 kV
SW-300 Feeder
SW-300 Feeder Spare
DS-1
120 A
DS-1
120 A
T-001 1 MVA
REV
DATE
BY
CHK
DESCRIPTION
REVISIONS
SmartPlant Electrical
INTERGRAPH
DRAWN BY
APPROVED BY
APPROVED BY
APPROVED BY
PROJECT NO.
DRAWING NO. SW-300
Last Rev.
Page 1 of 1



5.To close the document, select file -> close or, select the 'X', close active window.
Closing the document save the SLD in the Electrical Index, Document, Single Line Diagrams folder.

An Electrical Engineer-based single line diagram

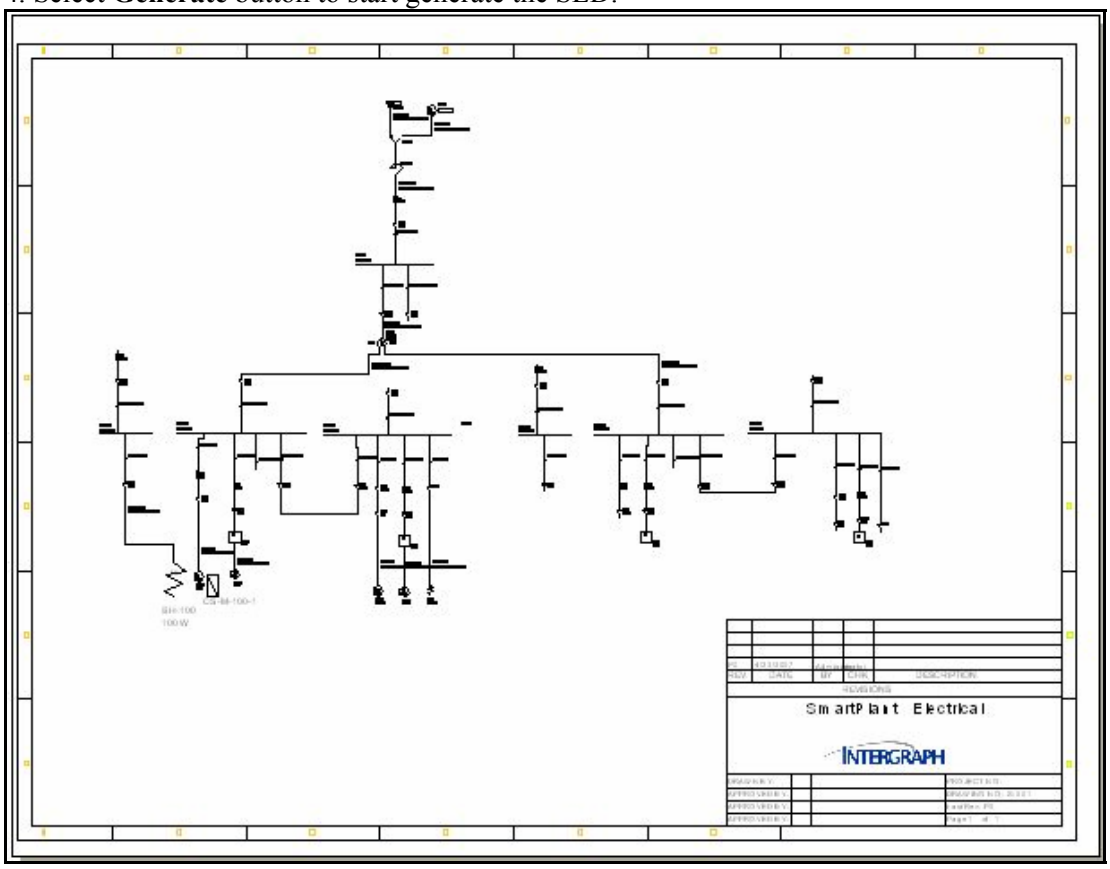
Is a main or key single line diagram of the electrical network for a selected set of items created according to the generation options that you predefine.

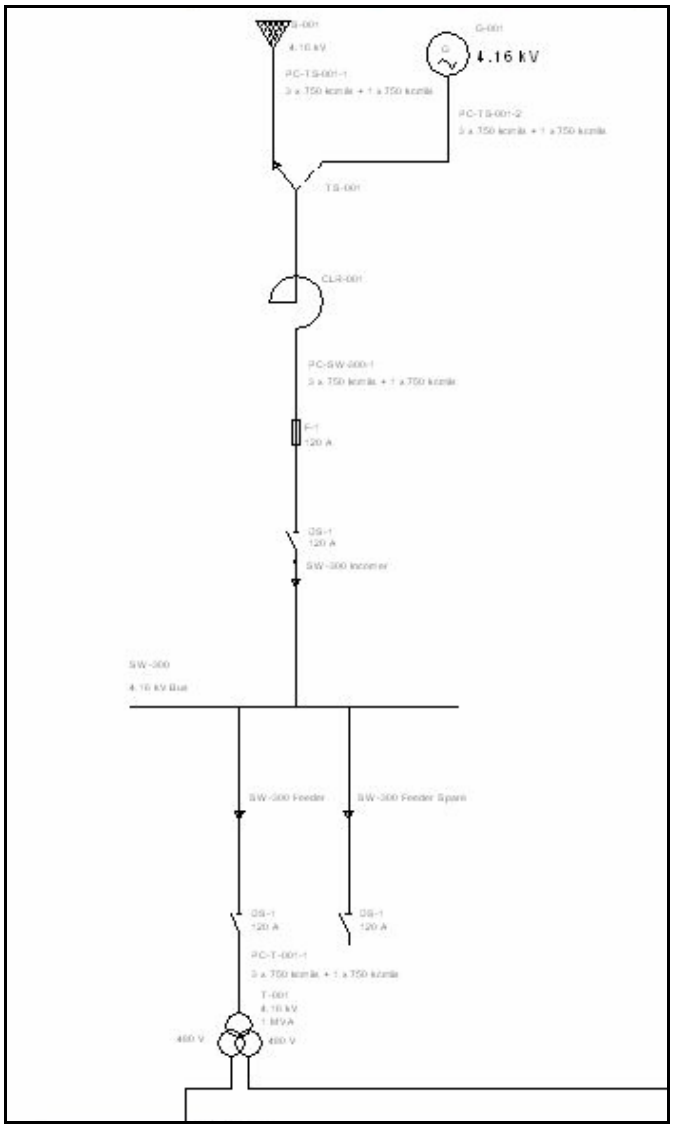
To generate Electrical Engineer-based single line diagram, do the following steps:

1.From Electrical Engineer, select , offsite power supply, 'S-001', Generator, 'G-001', MCC-201', Bus, '120 V Bus A', and 'MCC-202', Bus, '120 V Bus A'.
Note: Multile items selection done with the 'Ctrl' key.

2.Right click, and select, '**New SLD**'.
3. In the Single Line Diagram Options, type 'S-001' for document number, and set your preferences.

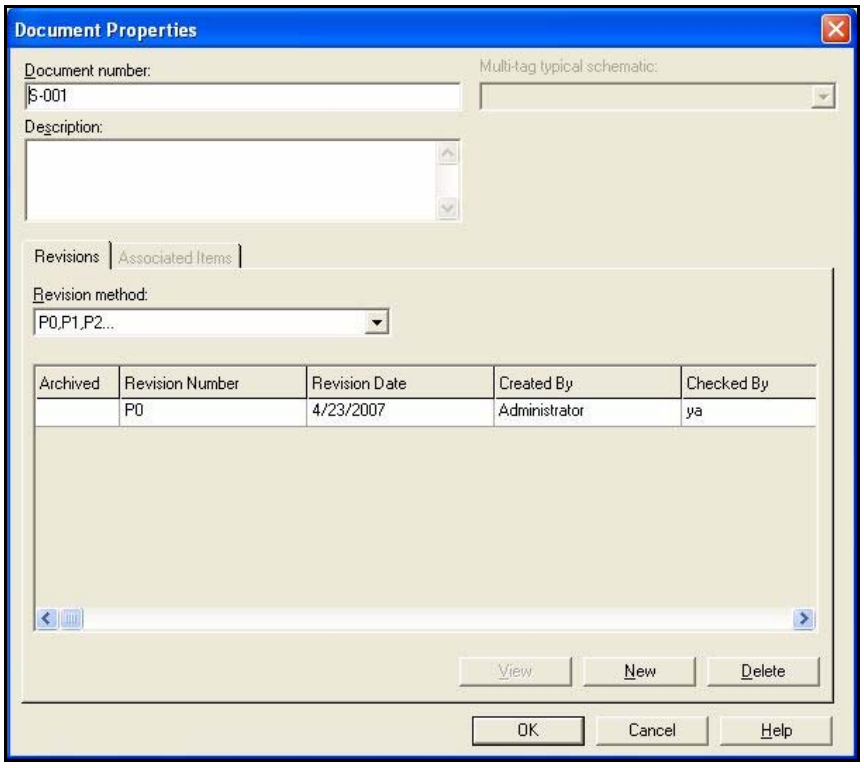
4. Select **Generate** button to start generate the SLD.





5.To close the document, select file -> close or, select the 'X', close active window.

Closing the document save the SLD in the Electrical Index, Document, Single Line Diagrams folder.

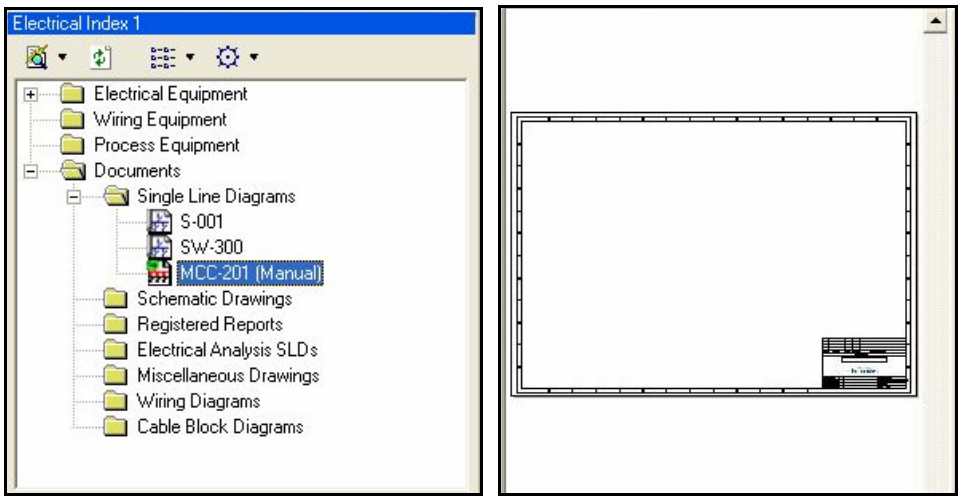


A blank single line diagram template (manual SLD)

Allows you perform a drag-and-drop operation from the Electrical Index on those items that you want to appear in your single line diagram. These items include buses, offsite power supplies, generators, battery banks, converting equipment not contained within PDBs, and standalone disconnect electrical equipment. After positioning the items on the SLD template, the software will complete the drawing according to the generation options that you defined.

To create new manual SLD do the following steps:

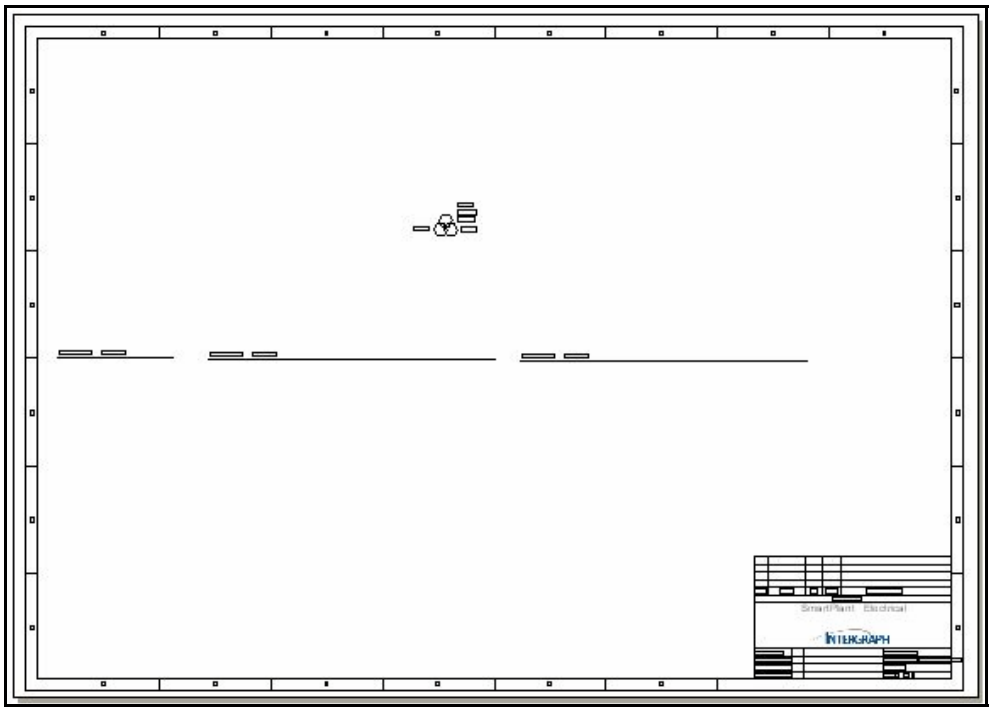
1.In Electrical Index, document folder, select Single Line Diagram folder and create new document, name it MCC-201 (Manual)



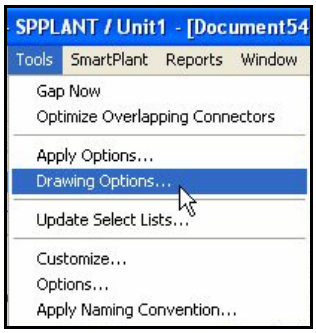
2.Drag to the new template the following items:

MCC-201 busses, '120 V Bus A', '480 V Bus B', '480 V Bus C'

Transformer, 'T-001'

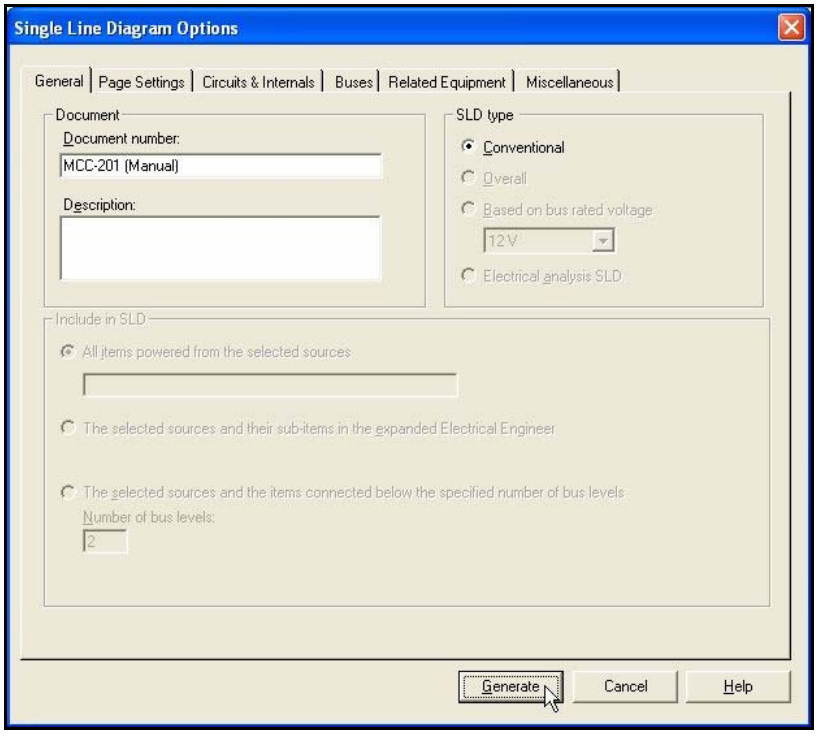


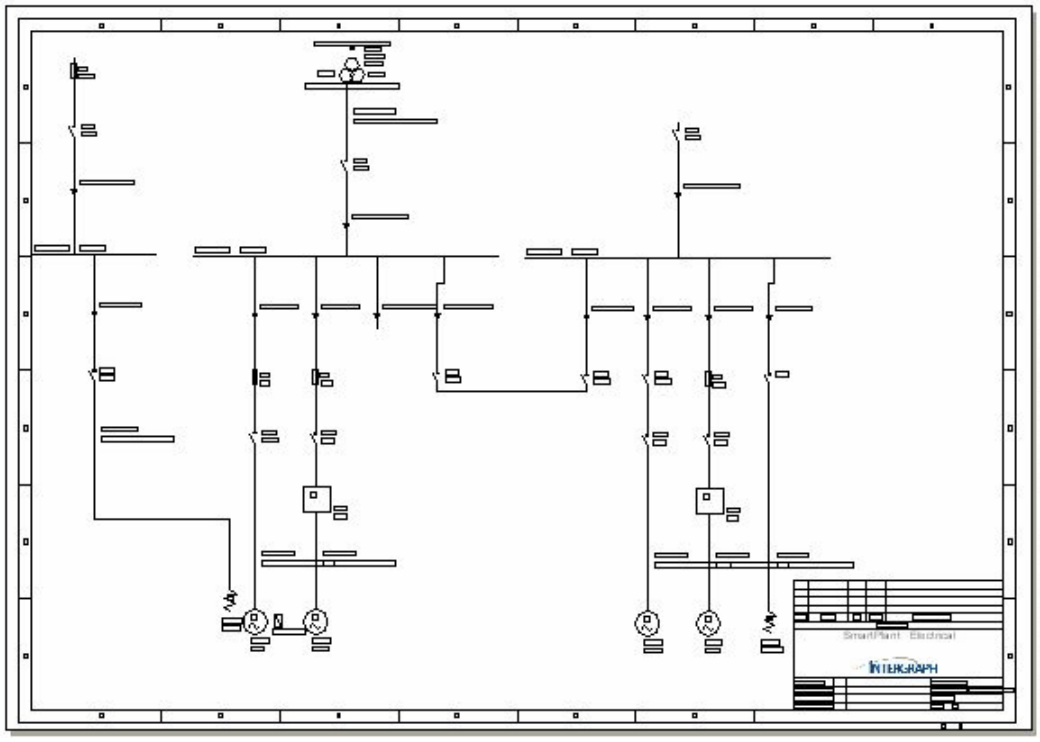
3.Select from Tools -> Drawing Options



4.Set the preferences you need in the Single Line Diagram Options window.

To start running the SLD select the generate button.





5.To close the document, select file -> close or, select the 'X', close active window.

Closing the document save the SLD.

Notes: (1) At any time user may add or remove items from being shown in SLD.
And next time user will generate the SLD, the deleted items will not be included, and he added one will be included.
(2) In automatically generated SLD, user can not drag and drop items.
(3) Compare option is possible only between the same type of SLD's.

SLD symbols managment

When generating an SLD, the software searches for a symbol for each electrical item that is included in the SLD source. SmartPlant Electrical supplies you with a set of default symbols that you can use in your drawings, or you can create your own custom symbols in Catalog Manager. After creating your symbols and defining them in the Options Manager, the software then uses the SLD symbols according to the following rules:

1. SmartPlant Electrical checks whether the items included in the SLD source are associated with symbol files.Wherever such an association exists, the software uses the associated symbol file to represent the electrical item in the single line diagram. You associate an item with a symbol file using the associate custom Symbols command from the Actions menu for an item that you select in the Electrical Index.

2.If the item is not associated with a custom symbol, Smart Plant Electrical checks if there is any symbology defined and use it.

3. If the item is not associated with custom symbol, and there is no symbology defined. Smart Plant Electrical uses a symbol with a name that matches the item type name of the electrical item to be included in the single line diagram. This symbol resides in the folder defined in the Default SLD Symbol Path location in Options Manager.

Managing SLD

The software allows you perform various actions with an open single line diagram. Apart from regular actions in a drawing such as zooming, selecting, saving, and so forth.

Moving items
You can select an item and drag it to another place in the drawing.

Modifying item properties
When selecting an electrical item in SLD document, the item also selected in the property grid window, where you can modify the item properties.

Modifying symbol color, width, and pattern
You can change symbol color, width and pattern from option manager, define item type symbology.
Note: Advanced symbol customization, and creating new symbol done by catalog manager.

Annotations and Redlining
You can mark up single line diagram 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. When opening single line diagram 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.

Saving an SLD as an External File
You can save the SLD from the file menu as external file.

Printing an SLD

While the SLD document is open, you can print the document from the file menu.

Lab 9

9.1. Generate automatic SLD type, from Electrical Engineer. Start from the offsite power supply, 'S-001', and the generator, 'G-001', to get the "Demo SLD", 'S-001 Demo SLD'. Name the document, '**S-001 Demo SLD**'.
Save the document to the database and also as external file.
Name the external file, '**S-001 Demo SLD Extrnl.spe**'.

9.2. Generate automatic SLD from Electrical Engineer. Start from the offsite power supply, 'S-001', and generator, 'G-001', but change in the Single Line Diagram Options window, the SLD type, from conventional, to overall. Name the new SLD document, **'S-001 Demo SLD Overall**'. Save the document to the database and also as external file. Name the External file, '**S-001 Demo SLD Overall Extrnl.spe**'.

9.3. Generate overall type SLD from Electrical Engineer. Start from PDB,'SW-300',Bus, '4.16 kV Bus', Circuit, 'SW-300 Incomer' Name the new SLD document, '**SW-300 Incomer Overall**'.
Save the document to the database and also as external file. Name the external file, '**SW-300 Incomer Overall Extrnl.spe**'.

Chapter 10

Schematics

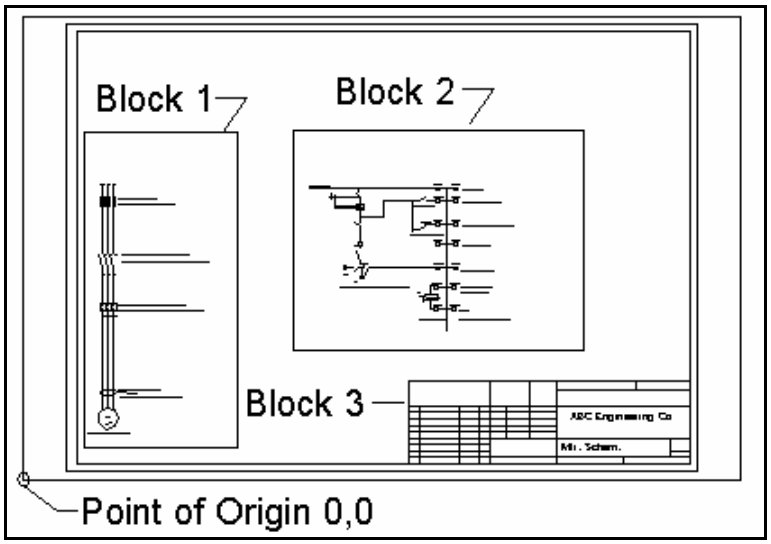
A schematic is a graphical representation of wiring and motor control systems. You create a schematic by assembling it from typical blocks.
A block is a graphical representation of an electrical item. Blocks are parts of a complete drawing. To create a block, start by creating an entire drawing in Catalog Manager or any CAD application such as SmartSketch, AutoCAD, or MicroStation. Once the complete drawing is ready, you select a part of the drawing and turn it into a block.
You can generate schematics for the following item types:

- All loads
- All converting equipment (except CLR).
- All disconnect equipment (except metering equipment).
- Circuits
- Buses
- Generators
- Battery banks

The software can generate a schematic by retrieving data for power related items from one block or several blocks. If your generated schematic drawings contain the same information (that is, the information repeats itself in identical sections of different schematic drawings), you should consider creating a schematic block for the repeated parts so that this single block can be used by several other schematic drawings. Note that you can create schematic drawings that include several sheets. Multi-sheet schematics are mostly used when you need to display complex electrical equipment for which a single sheet is not enough to show all the relevant wiring details of the equipment.
After creating a block, you define the macros for that block. A macro is the smart text that you attach to the block graphic. The software retrieves this smart text from the database and attaches it to the block, thus making it part of the block file. You can create your macros in Catalog Manager, SmartSketch, AutoCAD, or MicroStation.
Also, you must attach your blocks to templates. A template contains the definition for the desired title block, border, and page size. You attach your blocks to templates in Catalog Manager, SmartSketch, or any other CAD application.
Before generating a schematic drawing, you create a typical block and a typical schematic that you will attach to a load for which you want to generate a schematic drawing.

Typical blocks

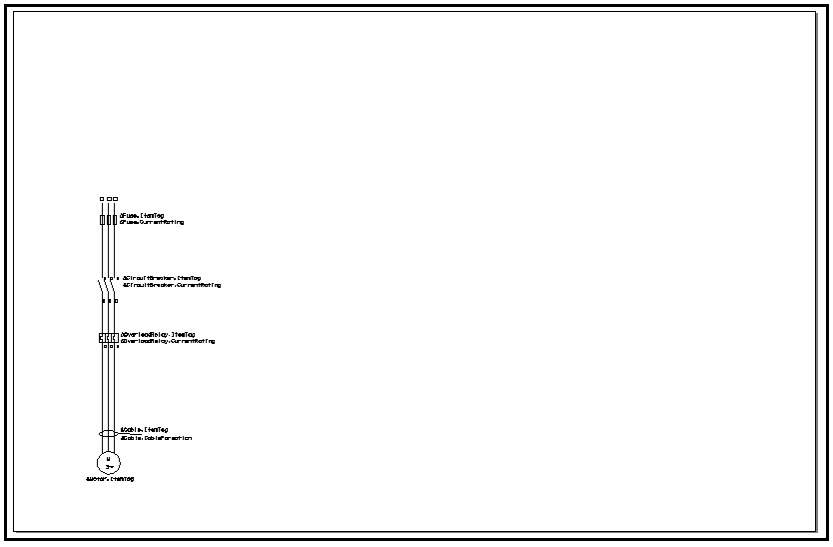
Typical block associated with drawing file (block or symbol) and it's the basic element for Typical schematic. There are several ways to create a block/symbol. Lets look at the following drawing.



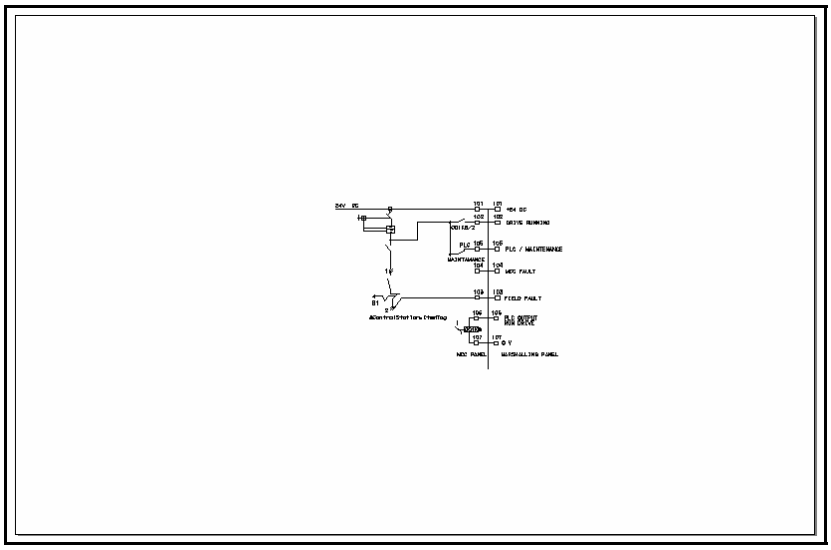
You will notice that there are 2 groups of symbols called Block 1 and Block 2. The border makes Block 3. Assume, for the sake of this discussion, that this was a representation of how you wanted your schematics to appear, except with a variation on block 1, which we will call Block 1A.

The easiest way to create your symbols would be to make 4 copies of this drawing (one for each block that you want to make a symbol from – Blocks 1, 1A, 2, 3). Delete everything else from the particular drawings except the graphics that you want to see on that symbol.

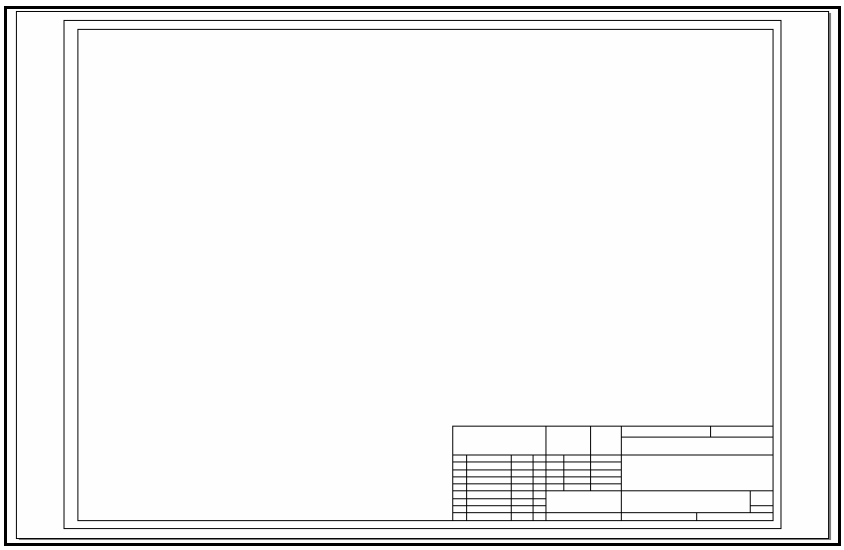
So Symbol #1 (a drawing of just the graphics needed for Typical Block #1) may look like this:



Symbol 1A may be some variation on this set of graphics. Symbol 2 may look like this



Of course, you will make a template out of the Block #3, which may look like this:

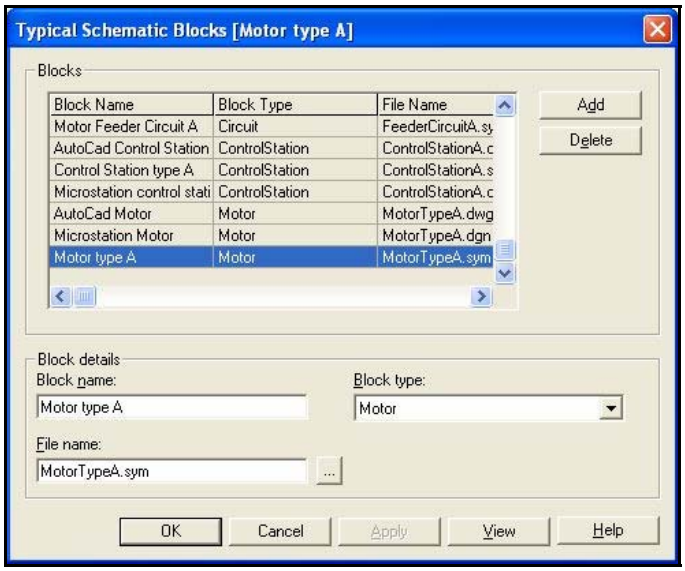


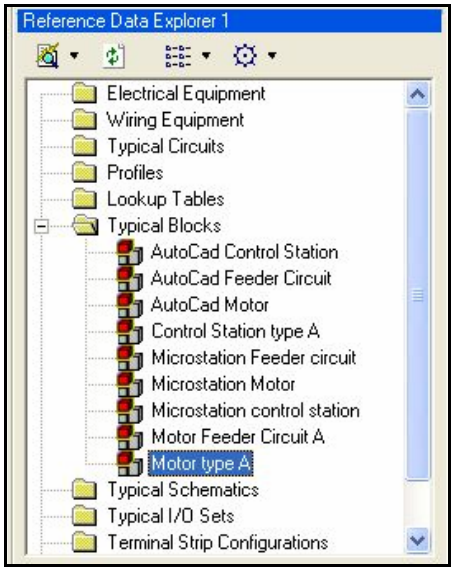
Notice the relationship between the placement of these Blocks and the 'Point of Origin'. This is how the software places the symbols or blocks in the correct physical relationship to each other and to the sheet.
Once the symbol files are prepared, you can save them in whatever format you are working. Then all that is left is to add your macros and create a Typical Block from that Symbol File. The above is true for all types of drafting packages.

Create a Typical Block

1.From RDE select Typical Block folder > right click -> New Typical Block
2.Give it a Name
3.Open Common Properties of the new block.
4.Select a block type.
5.Browse to the drawing file and select the symbol/block you need.
6.Select Apply or OK.

You can view the block at any time by clicking the **View** button.
If you want to create another block while you are in this window, just select add and create another one.

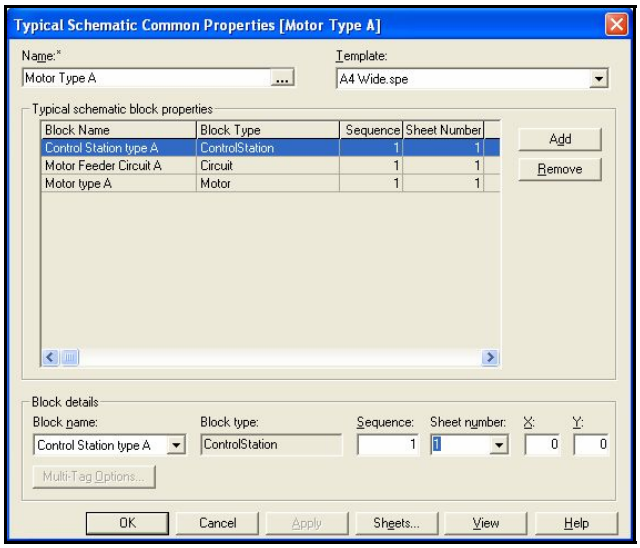


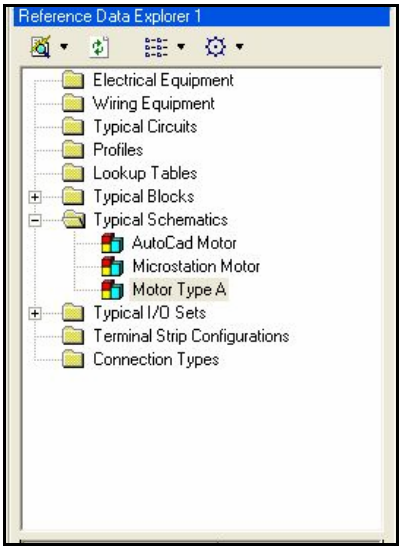


Create a Typical Schematic

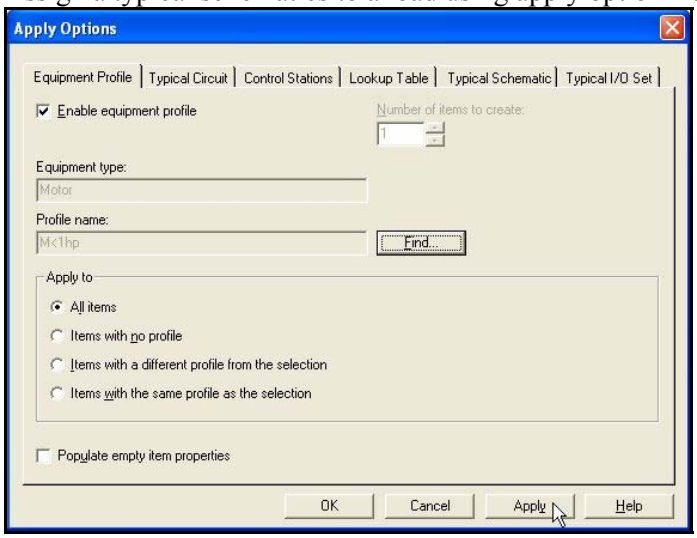
1.In the **Reference Data Explorer**, right-click the **Typical Schematics** folder, and then click **New**.
2.Type a name of up to 80 characters for the typical schematic.
3.Right-click the schematic and click **Common Properties** to open the **Typical Schematic** dialog box.
4.Select template.
5.Select **Add** to add a typical block (you can 1 or more blocks).
6.To finish select Apply or OK.

You can view the typical schematics at any time by clicking the **View** button.

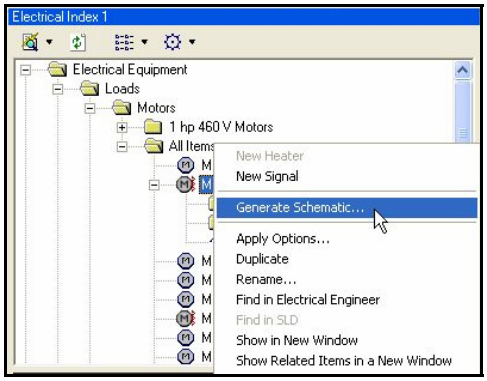


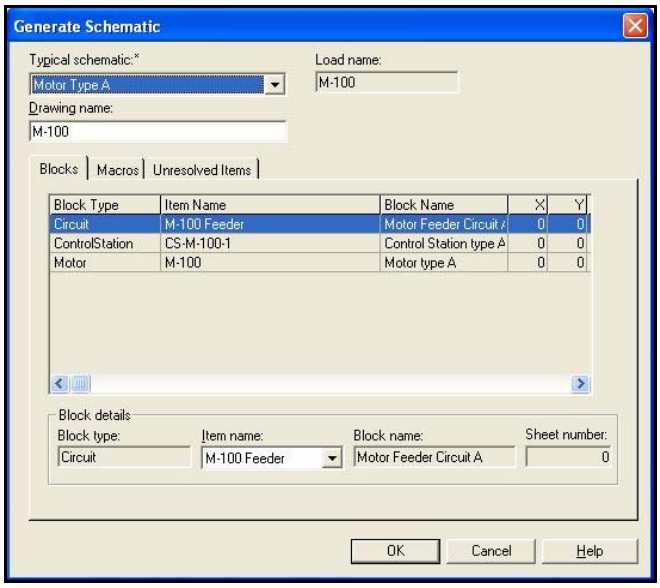


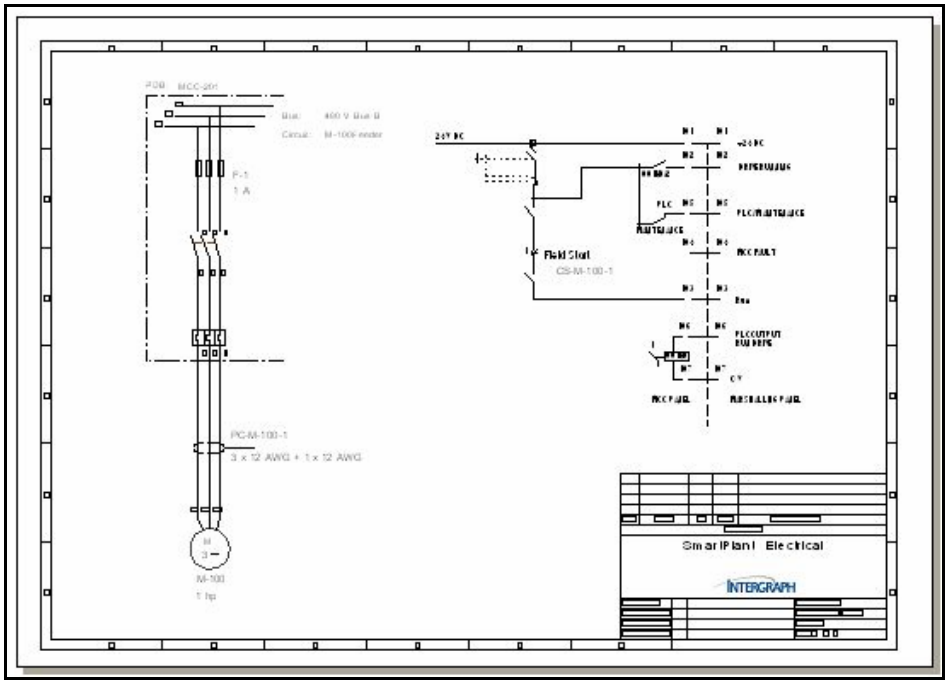
When this typical schematic is used to create a schematic of a particular motor, the macros will be replaced with specific data from that particular motor.
Create a Schematic Drawing in SmartPlant Electrical
Creation of a drawing block file (sym, dwg or dgn), using the **Electrical Index** the integrated SmartPlant Electrical Catalog Manager or by using any of the following independent software: SmartSketch, Autocad or MicroStation. In the block file, the user will draw its graphics (as expected to appear with the **Electrical Index** correct displacement from page origin) and in addition set of required Smart Text labels or "macros", later on, to be replaced by SmartPlant Electrical system with relevant data from the SmartPlant Electrical database, according to the attached object.
Create a new typical block item in the **Reference Data Explorer** .
Edit the new typical block item properties, to attach it with a relevant drawing block file (one or more).
Create a new typical schematics item in the **Reference Data Explorer**.
Edit the new typical schematics properties to attach it with a relevant the **Reference Data Explorer** typical block item (one or more).
Set each typical block sequence (in case two or more blocks of same item type are used that typical schematic e.g. two Control Station blocks etc).
Assign a typical schematics to a load using apply option from **Electrical Index**.



Generate the schematics from **Electrical Index**.



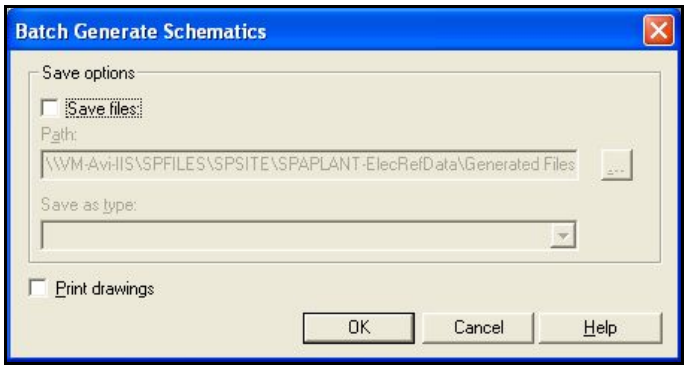




Generating Schematics in Batch Mode

You can generate your schematics in batch by simply selecting a number of motors the **Electrical Index** the in the tabular editor or in the lower pane of the **Electrical Index**:

The following dialog box appears:



You can save the generated schematics as external CAD files in any commercially available CAD format and have them printed on the fly.

Macros

You place macros on each block (or symbol) to retrieve relevant information from the database. The macro name describes where (what table/attribute) it will find the information.If you place a Macro called 'Motor.ItemTag' in a symbol or block, then create a Schematic (or SLD that uses that symbol), the program will replace the macro with the 'Data' in the database from the table 'Motor', attribute 'ItemTag'. This particular Macro happens to be the attribute that contains the name of the motor. So, when this Macro is placed in block (or symbol), and a Schematic (or SLD) is created for a particular motor, the program gets the name of the motor and places it in the drawing. The formatting of the text put in the drawing is the same as the formatting of the Macro.

Macro Syntax

The macro text has to contain the SmartPlant Electrical item name, its required property name, and a sequence number, according to the following syntax:
&[ItemTypeName].[Property].[Sequence]
For example, to specify a name (item tag) of a motor, the text label has to be:
&Motor.ItemTag.1
If a particular block contains several items of the same item type (that is, several cables, control stations, and so on), you have to use the sequence number to ensure item uniqueness.

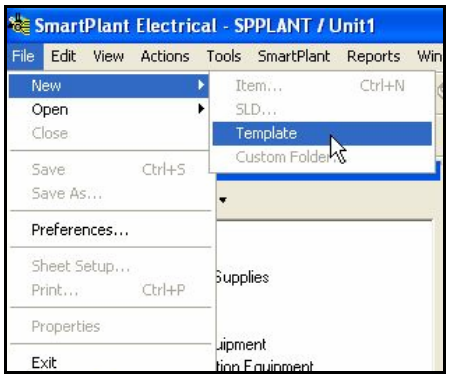
Annotations and Redlining

- You can mark up schematic drawings 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. If a drawing has more than one sheet, you can create redlining and hide or display it independently for each sheet. Therefore, while working in redlining mode, you cannot access any

items in the main drawing layer. The software saves the drawing with the redlining items. When opening schematic that contains redlining items, the software automatically displays the drawing in redlining mode. You can switch back to regular mode and modify the drawing as needed. The software saves the existing redlining items regardless of whether you open the drawing in redlining mode or not. You can add text, symbols, lines, circles, rectangles, and watermarks as redlining.

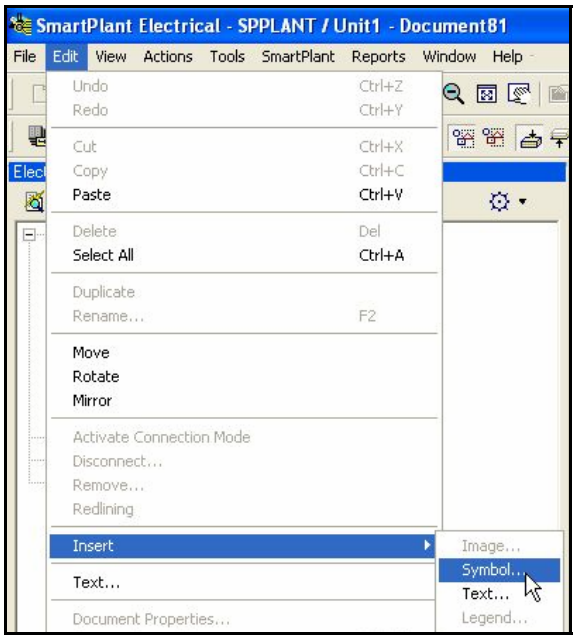
Create Templates (template for Schematic and SLD)

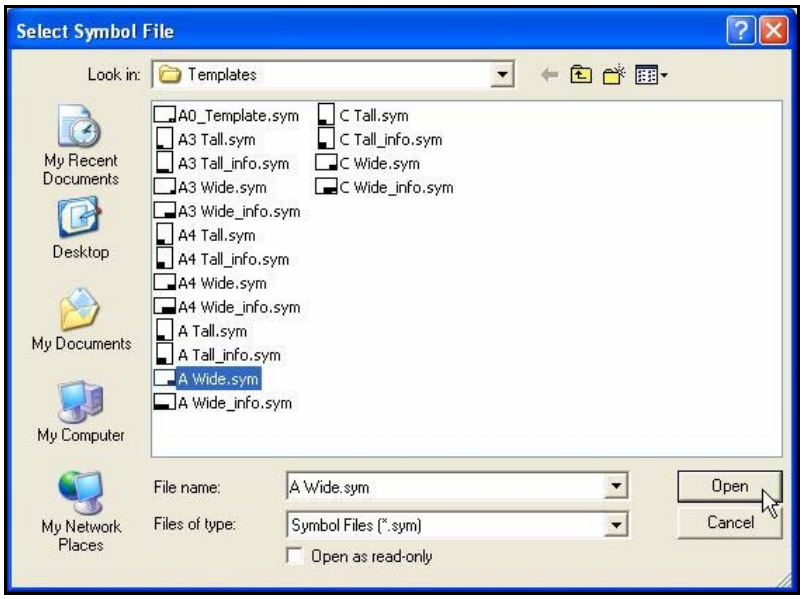
1. From File menu select **New** -> **Template**.

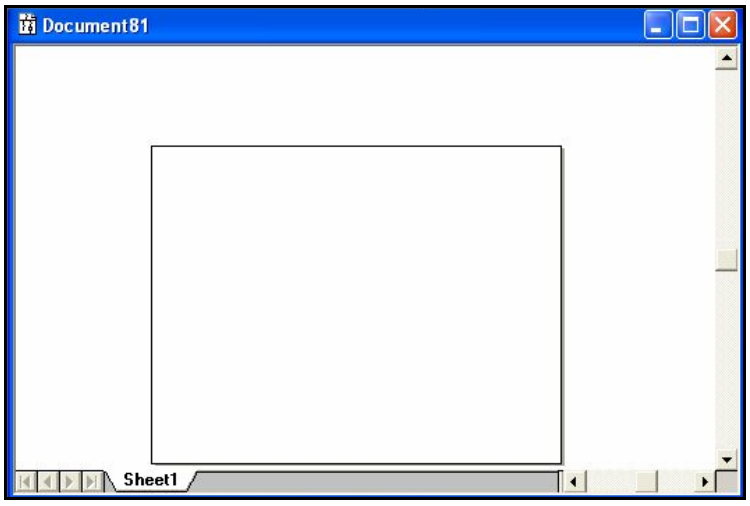


2. From **File** menu select **Sheet Setup** and select the required page size.

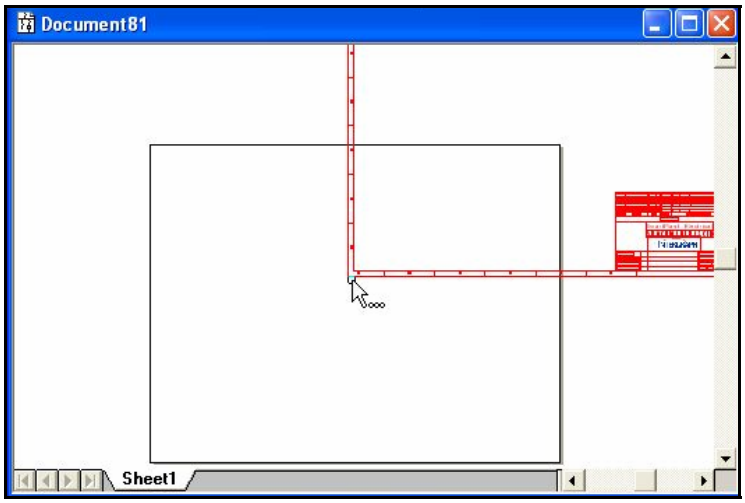
3.From Edit menu select Insert Symbol, and browse to your template folder to select a symbol template and select open.


SmartPlant Electrical - SPPLANT / Unit1 - Document81
File
Edit
View
Actions
Tools
SmartPlant
Reports
Window
Help
Undo Ctrl+Z
Redo Ctrl+Y
Cut Ctrl+X
Copy Ctrl+C
Paste Ctrl+V
Delete Del
Select All Ctrl+A
Duplicate
Rename... F2
Move
Rotate
Mirror
Activate Connection Mode
Disconnect...
Remove...
Redlining
Insert
Text...
Document Properties...
Image...
Symbol...
Text...
Legend...

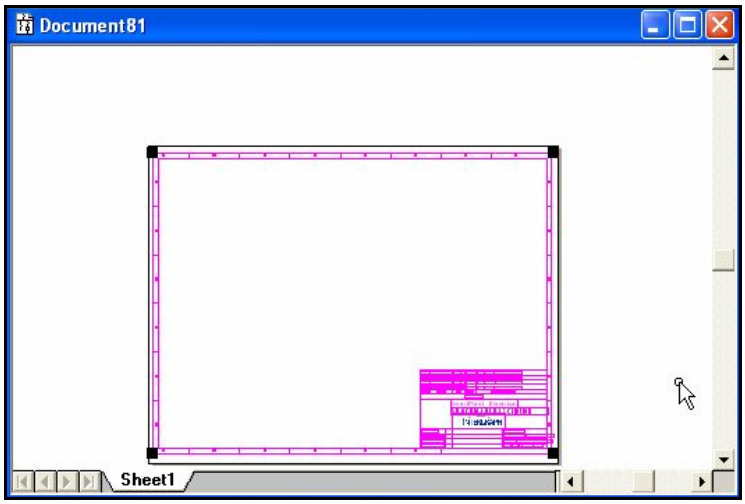

Select Symbol File
Look in: Templates
My Recent Documents
Desktop
My Documents
My Computer
My Network Places
A0_Template.sym
A3 Tall.sym
A3 Tall_info.sym
A3 Wide.sym
A3 Wide_info.sym
A4 Tall.sym
A4 Tall_info.sym
A4 Wide.sym
A4 Wide_info.sym
A Tall.sym
A Tall_info.sym
A Wide.sym
A Wide_info.sym
C Tall.sym
C Tall_info.sym
C Wide.sym
C Wide_info.sym
File name: A Wide.sym
Files of type: Symbol Files (*.sym)
Open as read-only
Open
Cancel



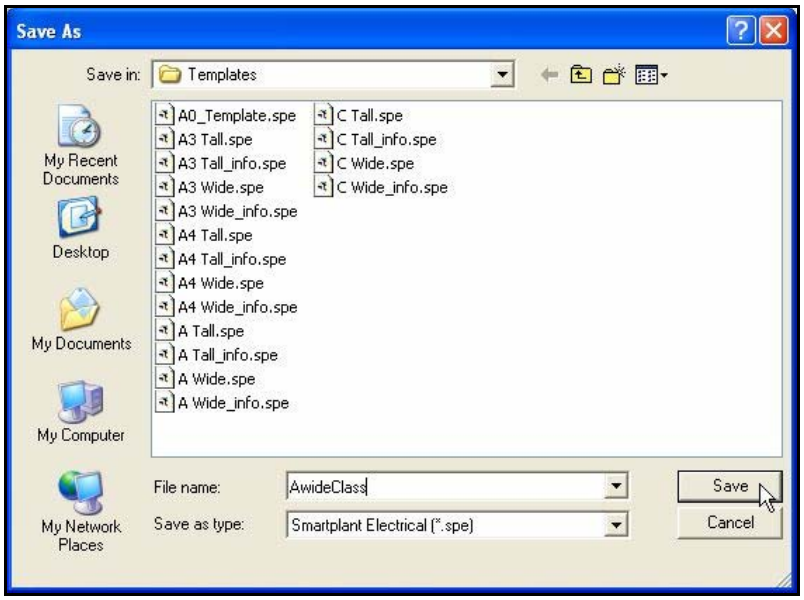
4.Click on the document area to inset the symbol.



5. Drag the symbol to match the template area.



6. Select file -> save and name the template AwideClass.spe



Lab 10

Create new typical schematic and generate motor schematic

10.1. Create 3 typical blocks:
Block 1
Typical block name: **MTR_A**
Blcok Type:**Motor**
File name: **MotorTypeA.sym**

Block 2
Typical block name: **FDR_A**
Blcok Type:**Circuit**
File name: **FeederCircuitA.sym**

Block 3
Typical block name: **CS_A**
Blcok Type: **Control Station**
File name: **ControlStationA.sym**

10.2. Create a Typical Schematic

NAME: **MTR_SCM_A**
Template: **AWide.spe**
Blocks: **MTR_A , FDR_A,** and **CS_A**

10.3. Generate schematic

Generate schematic for motor, 'M-1' using typical schematic, 'MTR_SCM_A'.
View the result and save the document.

Notes: 1. Apply the typical schematic,'MTR_SCM_A' to motor, M-1 before you generate the schematic .
2. Save the document also as external file (name it '**M-1.spe**').

Chapter 11

Associating external Documents

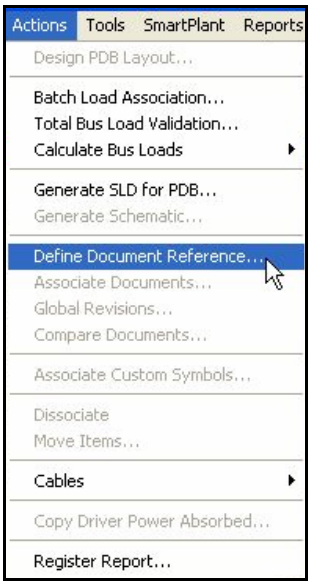
Associate documents

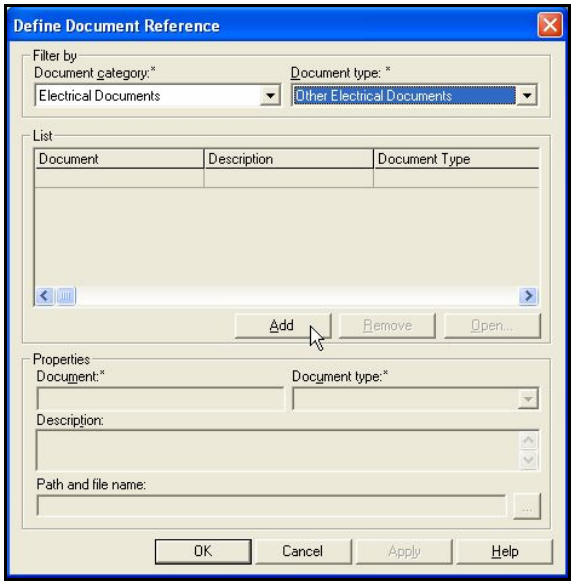
You define a document reference by adding a reference to the list of document references. You can then associate this reference with an external file. The software adds this association by storing the file path in the database.

After you define external document references, you select one or more electrical items and associate them with external documents. You can associate an electrical item with a document reference or with an external file that is linked to the document reference. The software allows you to associate single or multiple electrical items with the same external document. Also, you can associate the same external file with multiple items in the software.

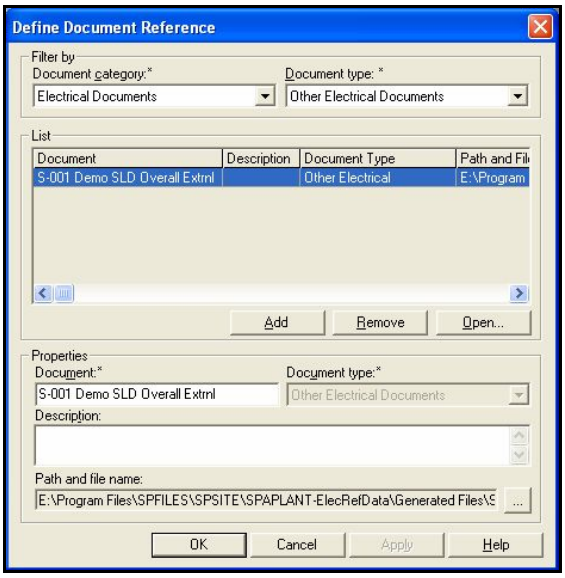
Define a Document Reference

Lets define the document reference for the external file, 'S-001 Demo SLD Overall Extrnl.spe'.

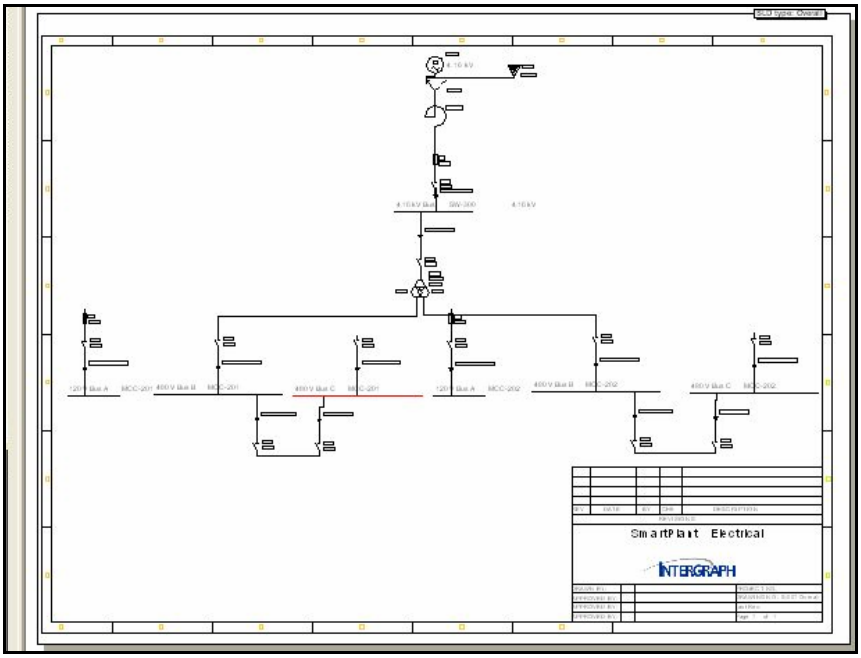




Name the document 'S-001 Demo SLD Overall Extrnl'.

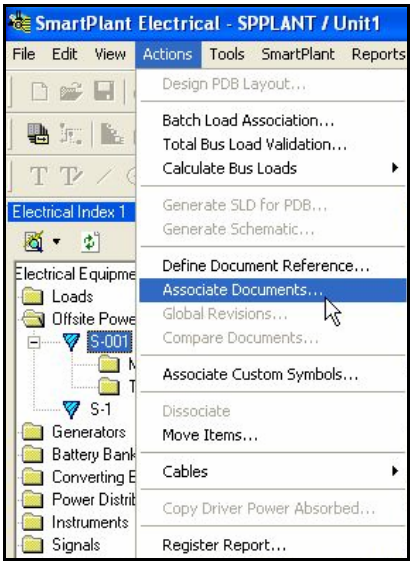


Select open to view the document.

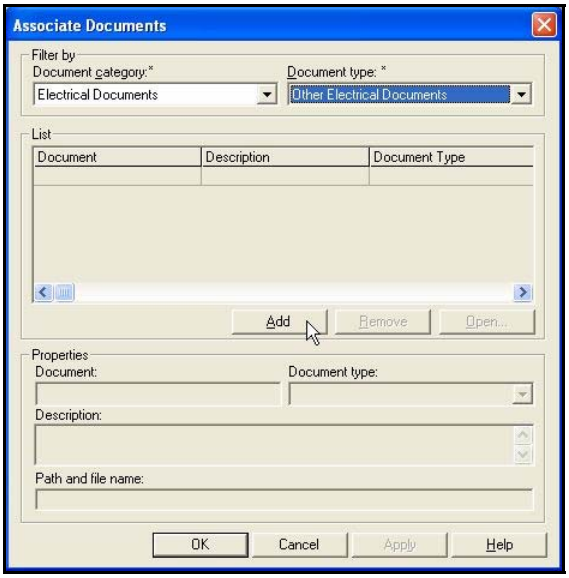


Associate a Document

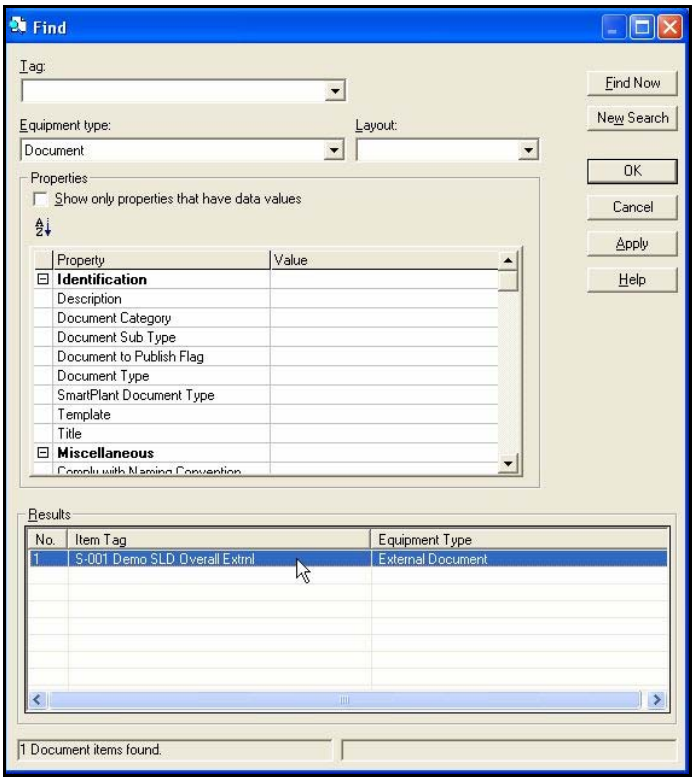
Lets now associate the reference document with defined, with the offsite power supply, 'S-001'.
Select the offsite power supply, 'S-001', and select actions, associate documents.



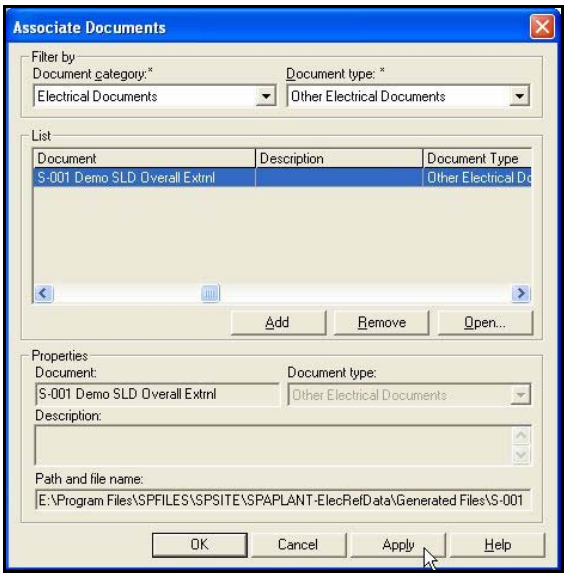
Select add.



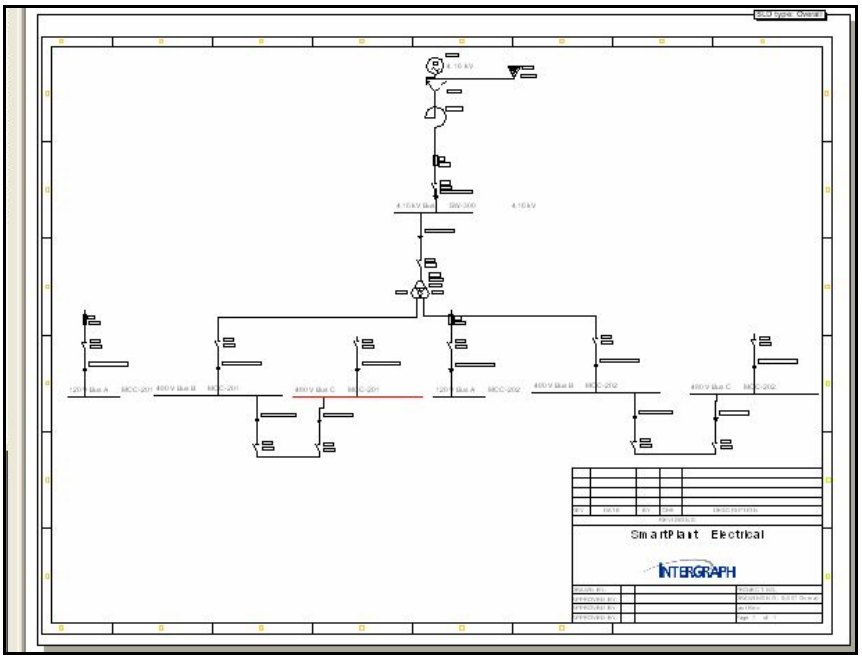
Find and select the document you want to associate.



Select apply.

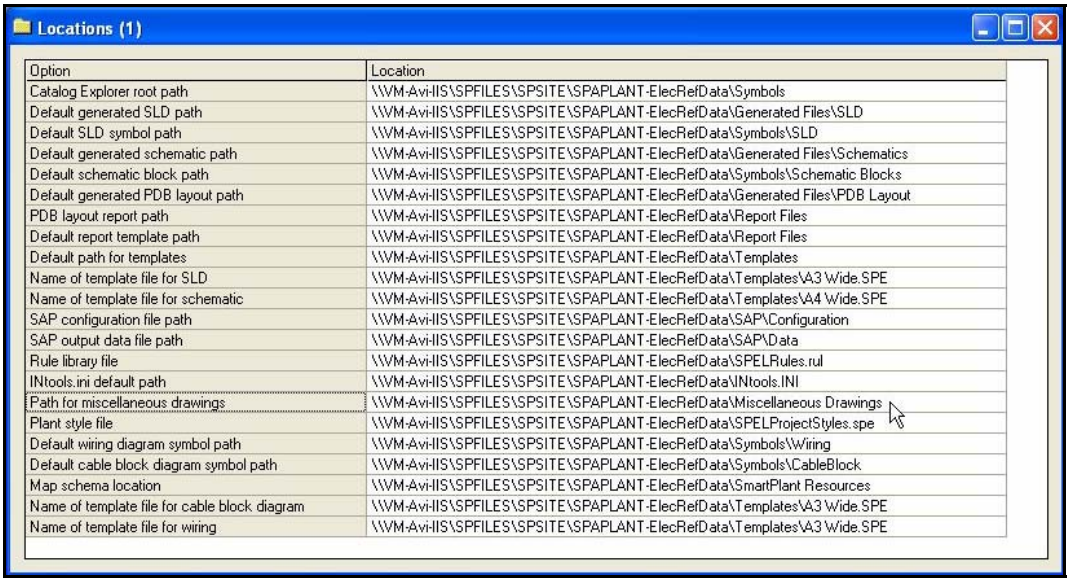


To view the associated document, select the offsite power supply, 'S-001' in electrical index. Select the action menu, select associate document. Select the appropriate document category and document
Type. In the associate document window, select from the list, the associated document, 'S-001 Demo SLD Overall External', and select open.



Miscellaneous Drawings

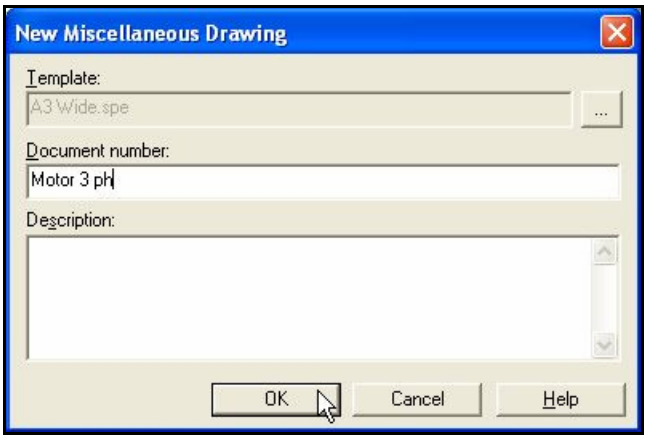
Miscellaneous drawings allows you to create none typical drawings using CAD and save them as SPEL documents, uses the SPEL revision management.
The miscellaneous drawing path defined in the option manager

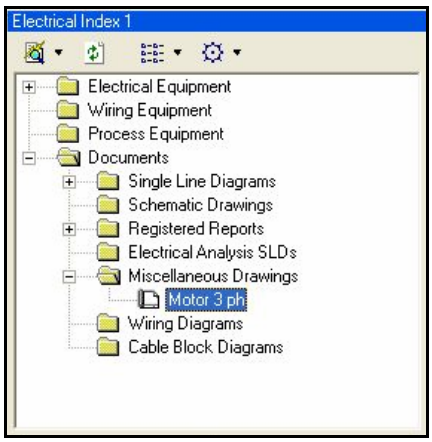


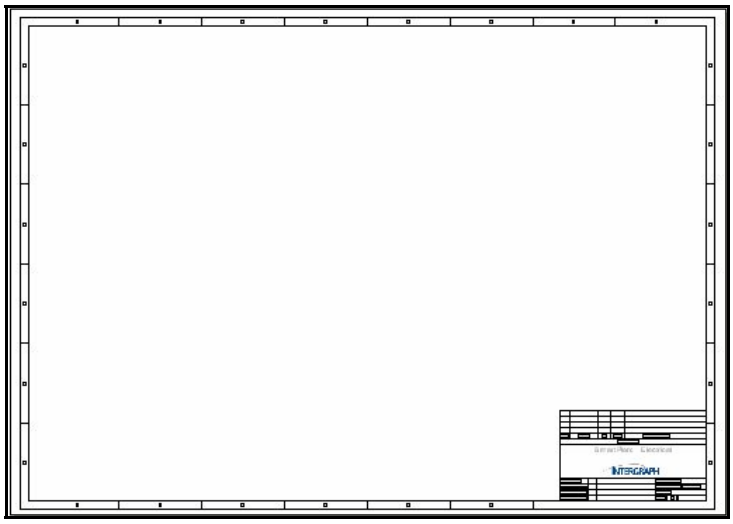
Option	Location
Catalog Explorer root path	\\VM-Avi-IIS\SPFILES\SPSITE\SPAPLANT-ElecRefData\Symbols
Default generated SLD path	\\VM-Avi-IIS\SPFILES\SPSITE\SPAPLANT-ElecRefData\Generated Files\SLD
Default SLD symbol path	\\VM-Avi-IIS\SPFILES\SPSITE\SPAPLANT-ElecRefData\Symbols\SLD
Default generated schematic path	\\VM-Avi-IIS\SPFILES\SPSITE\SPAPLANT-ElecRefData\Generated Files\Schematics
Default schematic block path	\\VM-Avi-IIS\SPFILES\SPSITE\SPAPLANT-ElecRefData\Symbols\Schematic Blocks
Default generated PDB layout path	\\VM-Avi-IIS\SPFILES\SPSITE\SPAPLANT-ElecRefData\Generated Files\PDB Layout
PDB layout report path	\\VM-Avi-IIS\SPFILES\SPSITE\SPAPLANT-ElecRefData\Report Files
Default report template path	\\VM-Avi-IIS\SPFILES\SPSITE\SPAPLANT-ElecRefData\Report Files
Default path for templates	\\VM-Avi-IIS\SPFILES\SPSITE\SPAPLANT-ElecRefData\Templates
Name of template file for SLD	\\VM-Avi-IIS\SPFILES\SPSITE\SPAPLANT-ElecRefData\Templates\A3 Wide.SPE
Name of template file for schematic	\\VM-Avi-IIS\SPFILES\SPSITE\SPAPLANT-ElecRefData\Templates\A4 Wide.SPE
SAP configuration file path	\\VM-Avi-IIS\SPFILES\SPSITE\SPAPLANT-ElecRefData\SAP\Configuration
SAP output data file path	\\VM-Avi-IIS\SPFILES\SPSITE\SPAPLANT-ElecRefData\SAP\Data
Rule library file	\\VM-Avi-IIS\SPFILES\SPSITE\SPAPLANT-ElecRefData\SPELRules.rul
INtools.ini default path	\\VM-Avi-IIS\SPFILES\SPSITE\SPAPLANT-ElecRefData\INtools.INI
Path for miscellaneous drawings	\\VM-Avi-IIS\SPFILES\SPSITE\SPAPLANT-ElecRefData\Miscellaneous Drawings
Plant style file	\\VM-Avi-IIS\SPFILES\SPSITE\SPAPLANT-ElecRefData\SPELProjectStyles.spe
Default wiring diagram symbol path	\\VM-Avi-IIS\SPFILES\SPSITE\SPAPLANT-ElecRefData\Symbols\Wiring
Default cable block diagram symbol path	\\VM-Avi-IIS\SPFILES\SPSITE\SPAPLANT-ElecRefData\Symbols\CableBlock
Map schema location	\\VM-Avi-IIS\SPFILES\SPSITE\SPAPLANT-ElecRefData\SmartPlant Resources
Name of template file for cable block diagram	\\VM-Avi-IIS\SPFILES\SPSITE\SPAPLANT-ElecRefData\Templates\A3 Wide.SPE
Name of template file for wiring	\\VM-Avi-IIS\SPFILES\SPSITE\SPAPLANT-ElecRefData\Templates\A3 Wide.SPE

To create new miscellaneous drawings do the following steps:

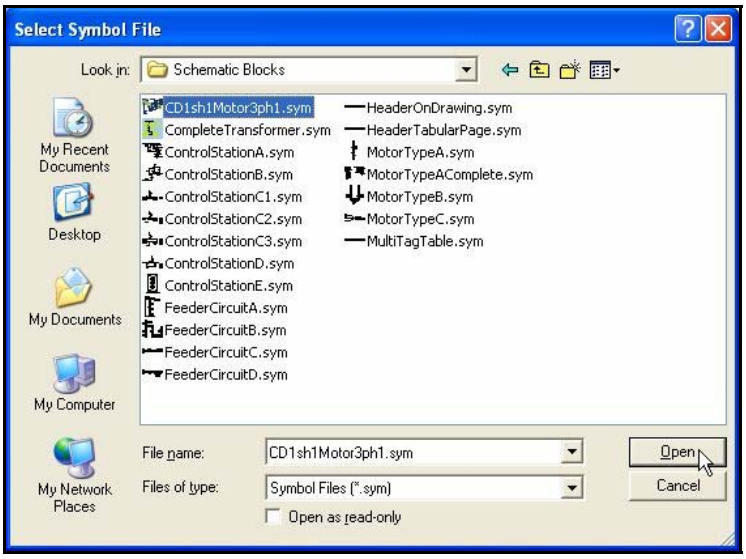
1.Select the Miscellaneous Drawings folder, right click, and select New Miscellaneous Drawings.
2.Select template and enter document number and select OK.



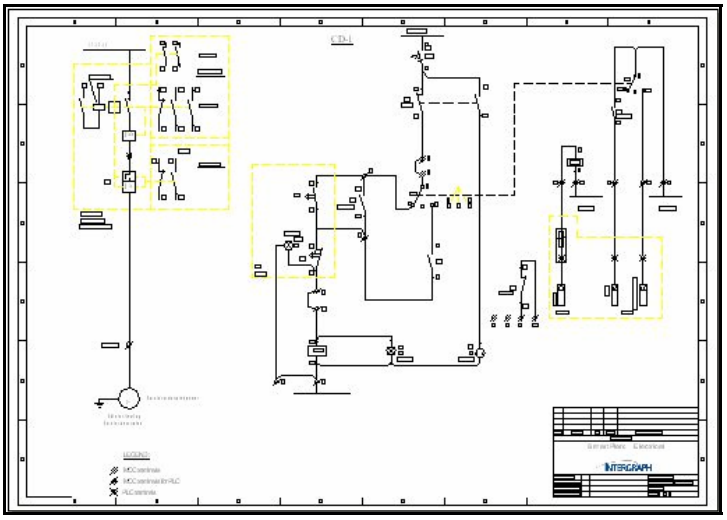




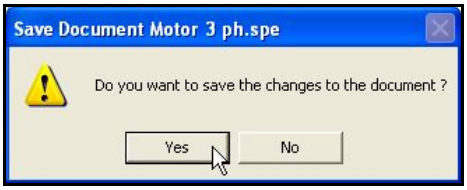
3.From edit menu, select insert, symbol, and select the symbol, 'CD1sh1Motor3ph1.sym'.



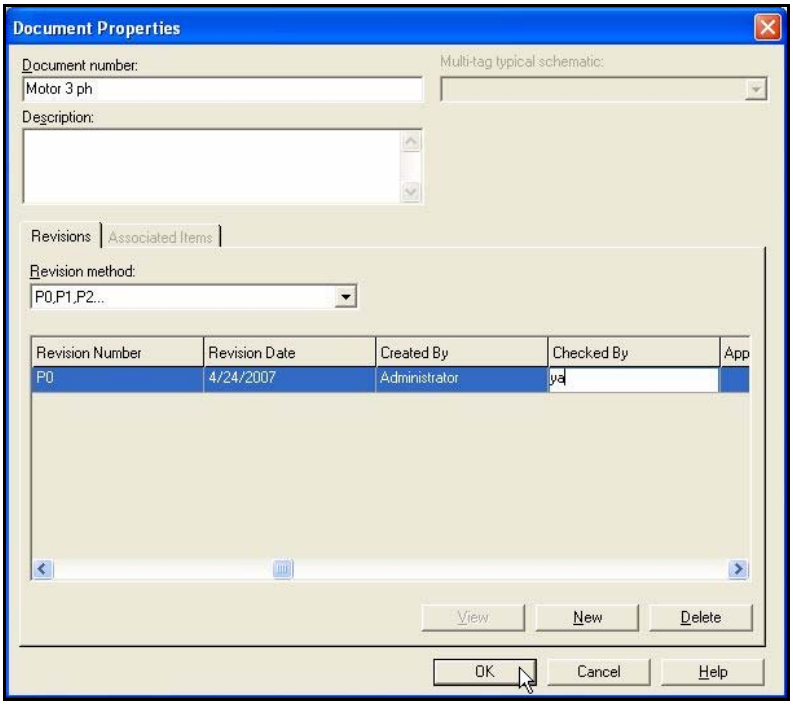
4.Select open and click on the template area to insert the symbol. Rearrange the symbol to match the template area.



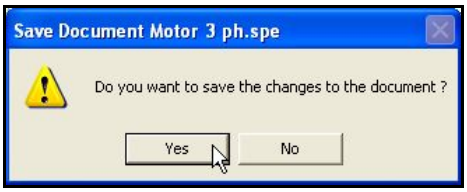
5.Close the document and save the change.



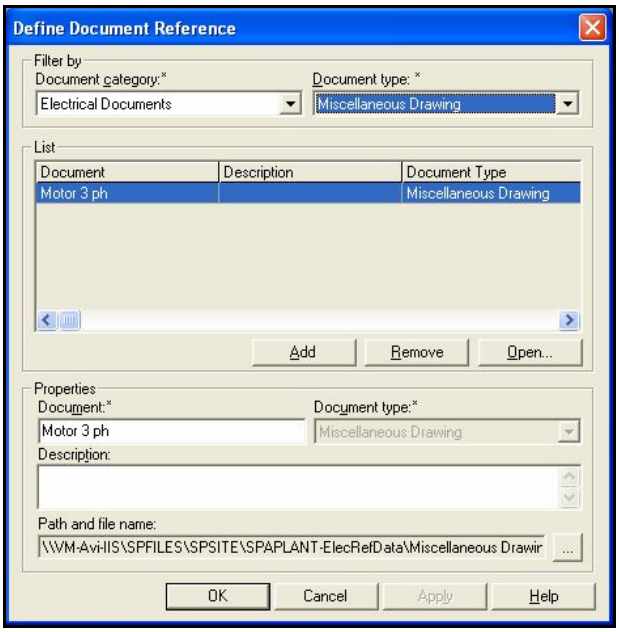
7. To add first revision, P0 to the document, open the document, and while the docment is open, select the document, and select document properties. Select revision method, and select new.
Add additonal details if need (e.g. checked By), and select OK.



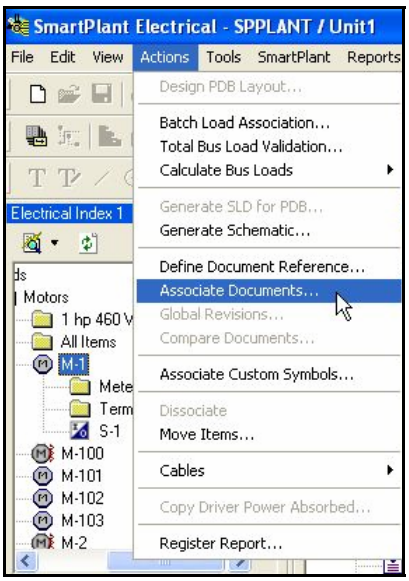
8. Close the document and save the change.

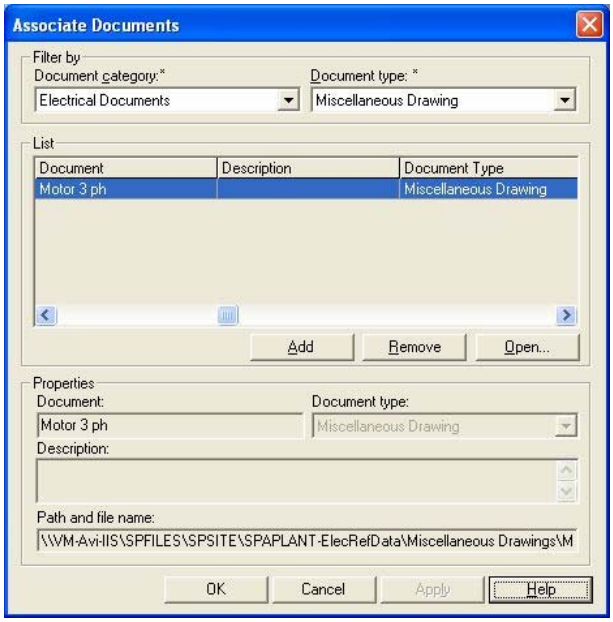


Notice: Miscellaneous drawings appears as one of the document reference.



And it can be associated to any item.





Lab 11

Define and associate Document

11.1 Define new document under document category = electrical documents, and document type = other Electrical Documents.
Name the document 'SW-300 Incomer Overall', using the external file, 'SW-300 Incomer Overall Extrnl.spe'.

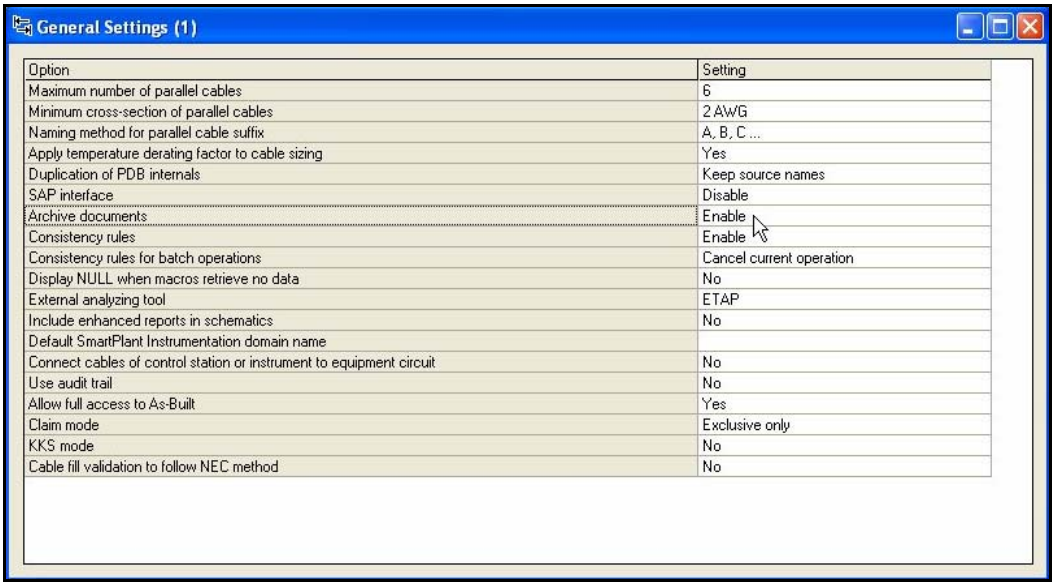
11.2 Associate the document, 'SW-300 Incomer Overall', with both that PDB, 'SW-300', and its Bus, '4.16 kV Bus'.

Chapter 12

Archiving Documents

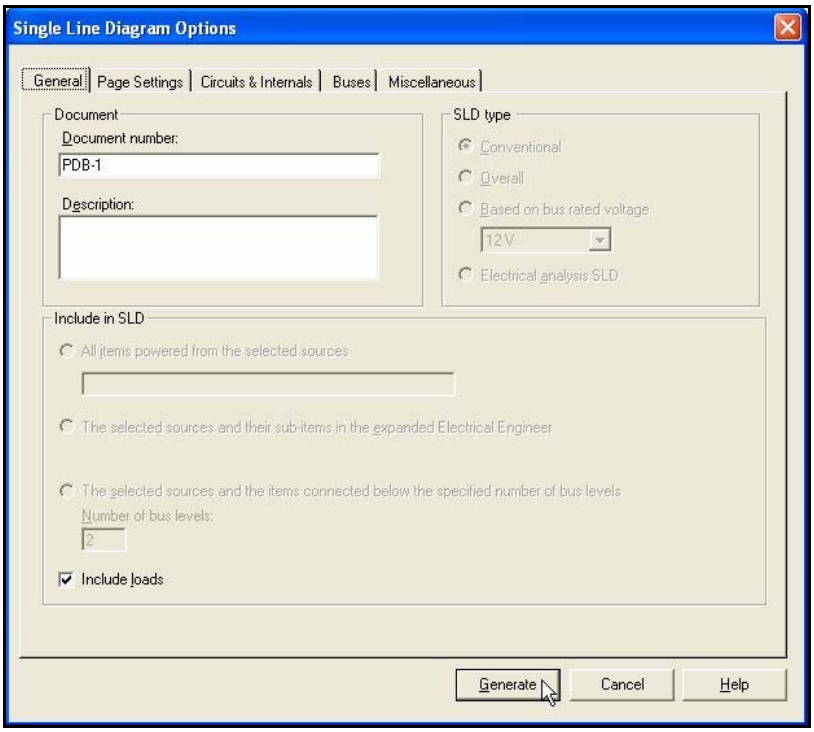
Smart Plant Electrical let you revise, archive and compare documents.

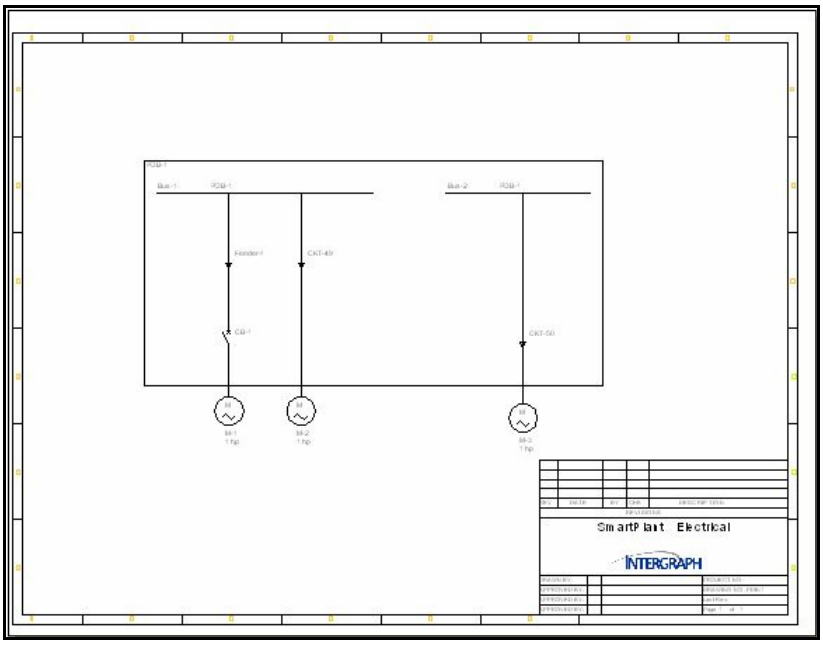
1. From option manager verify the archive documents is enabled.



Option	Setting
Maximum number of parallel cables	6
Minimum cross-section of parallel cables	2 AWG
Naming method for parallel cable suffix	A, B, C ...
Apply temperature derating factor to cable sizing	Yes
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 tool	ETAP
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	No
Allow full access to As-Built	Yes
Claim mode	Exclusive only
KKS mode	No
Cable fill validation to follow NEC method	No

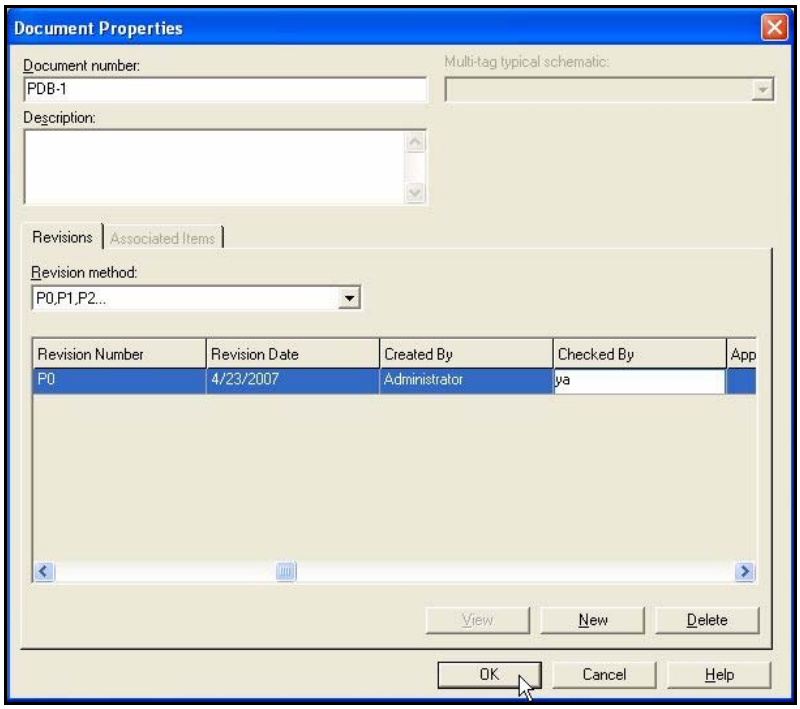
2. Save the change, and reopen Smart Plant Electrical.
3. Generate a new SLD for PDB, and name it, ‘PDB-1’, (automatic SLD from electrical index).



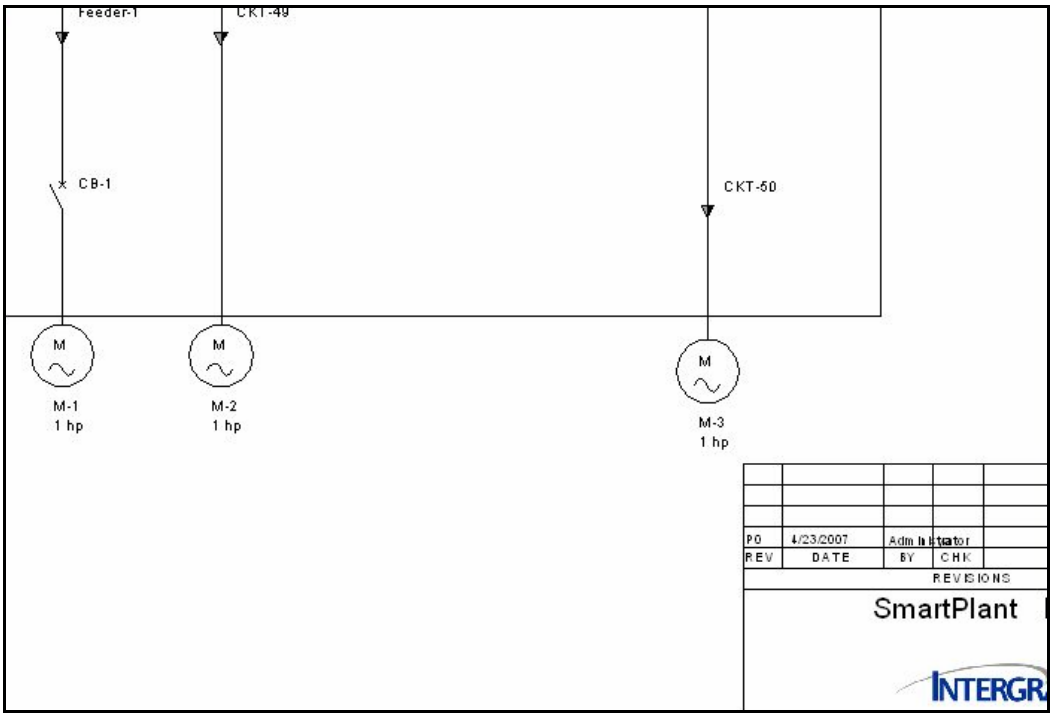


4. Close the document and save it.

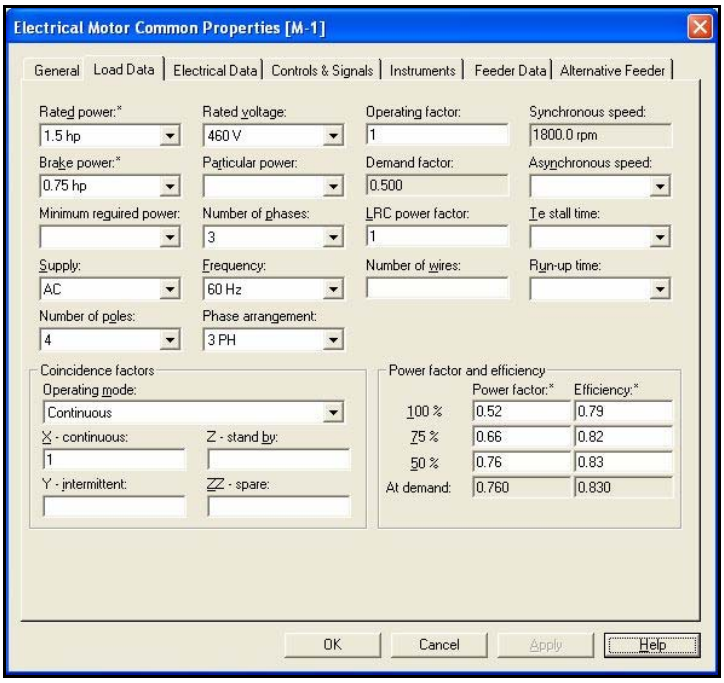
5. Open the document, 'PDB-1', (from electrical index, documents folder, single line diagrams folder). While the SLD document is open, select the document, 'PDB-1', (from electrical index, documents folder, single line diagrams folder.), select right click, and select, 'document properties'. In the document properties window, select new to add new revision.


Document Properties
Document number:
PDB-1
Multi-tag typical schematic:
Description:
Revisions
Associated Items
Revision method:
P0,P1,P2...
Revision Number
Revision Date
Created By
Checked By
App
P0
4/23/2007
Administrator
ya
View
New
Delete
OK
Cancel
Help

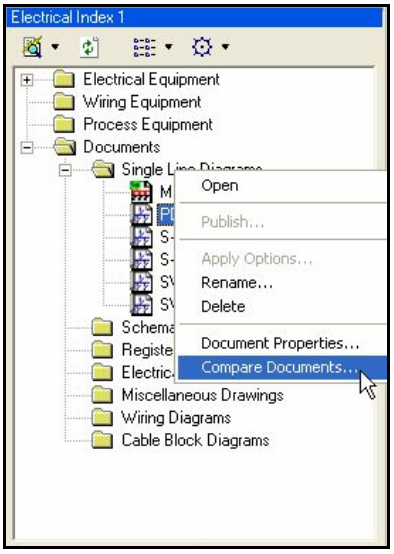

Document Properties
Changes saved successfully to database.
OK

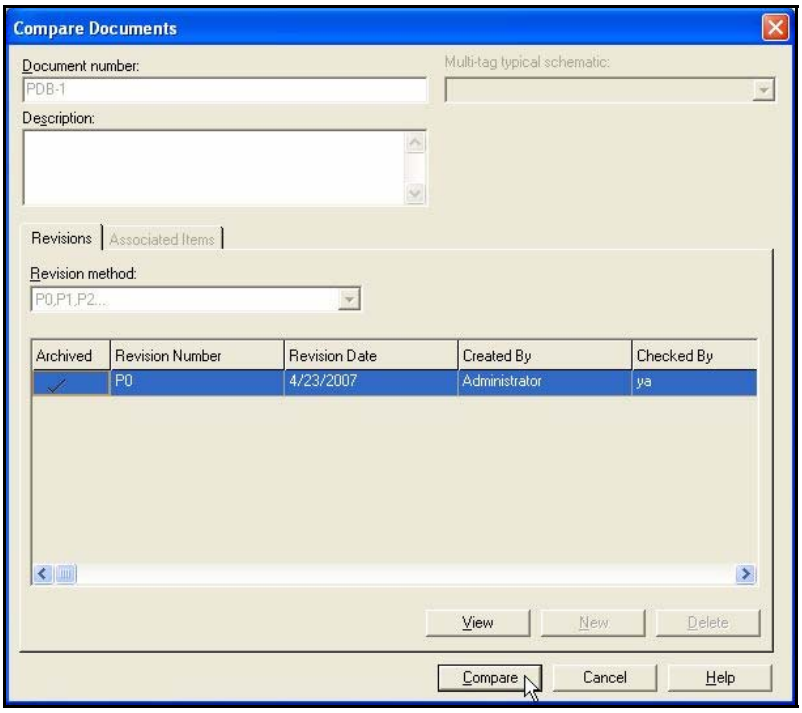

Feeder-1
CKT-49
CB-1
CKT-50
M
M
M
M-1
1 hp
M-2
1 hp
M-3
1 hp
P0
4/23/2007
REV
DATE
BY
CHK
REVISIONS
SmartPlant

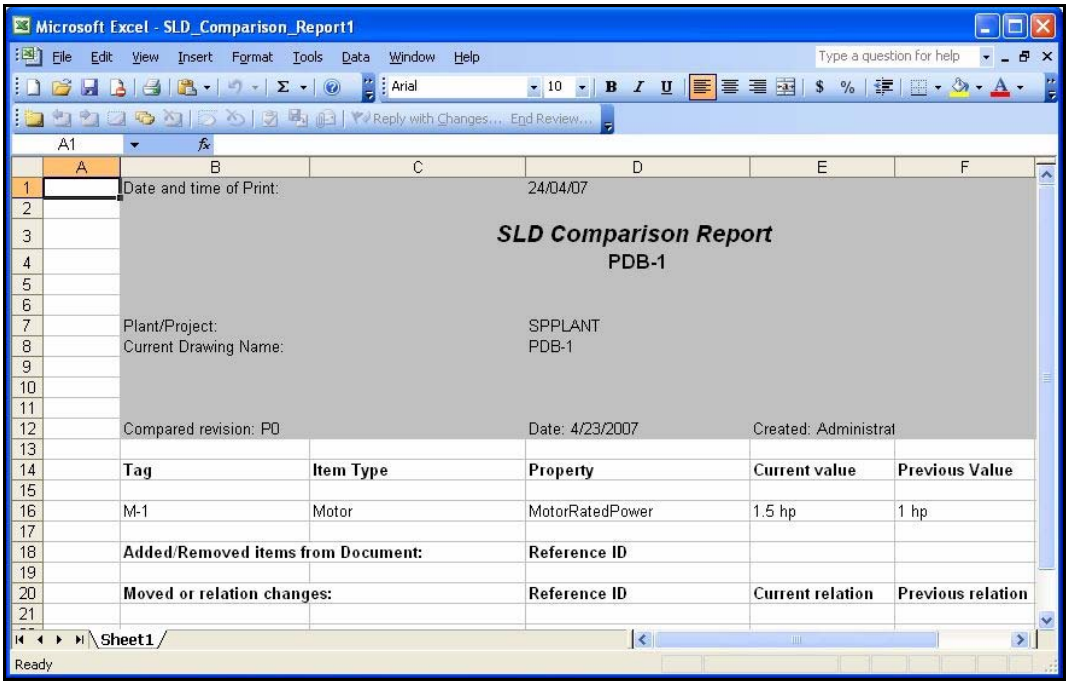
6. Close the document.
7. Lets change the motor rated power of motor, ‘M-1’ from 1 hp to 1.5 hp.

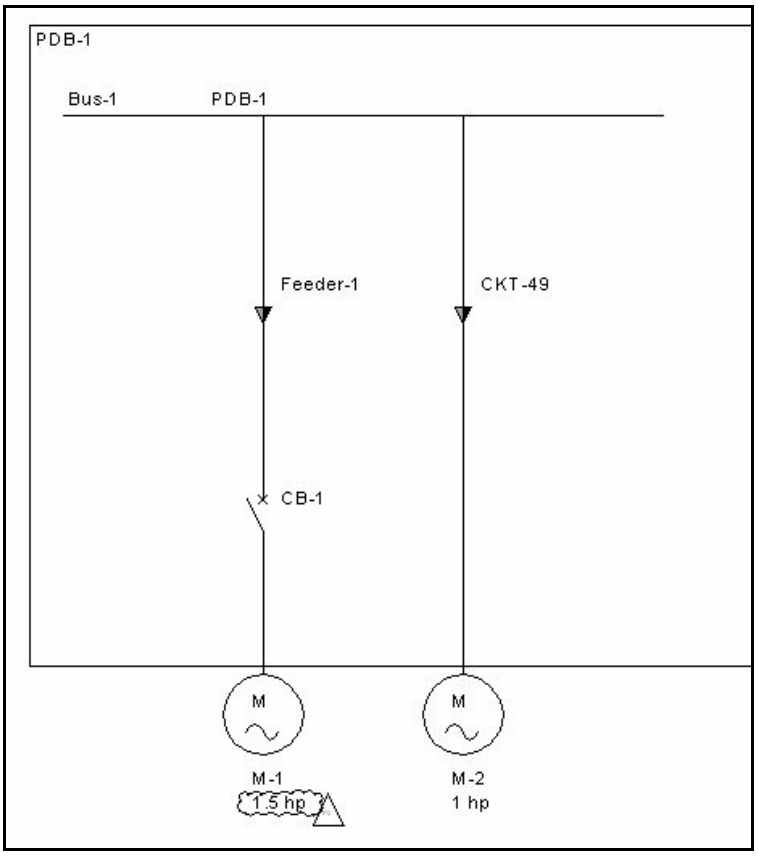


8. To run compare document for SLD document, ‘PDB-1’. Select the document, and select right click, and select ‘compare document’.




Compare Documents
Document number:
PDB-1
Multi-tag typical schematic:
Description:
Revisions
Associated Items
Revision method:
P0,P1,P2...
Archived
Revision Number
Revision Date
Created By
Checked By
P0
4/23/2007
Administrator
ya
View
New
Delete
Compare
Cancel
Help


Microsoft Excel - SLD_Comparison_Report1
File Edit View Insert Format Tools Data Window Help
Type a question for help
Arial 10
Reply with Changes... End Review...
A1
Date and time of Print: 24/04/07
SLD Comparison Report
PDB-1
Plant/Project: SPPLANT
Current Drawing Name: PDB-1
Compared revision: P0 Date: 4/23/2007 Created: Administrat
Tag Item Type Property Current value Previous Value
M-1 Motor MotorRatedPower 1.5 hp 1 hp
Added/Removed items from Document: Reference ID
Moved or relation changes: Reference ID Current relation Previous relation
Sheet1
Ready

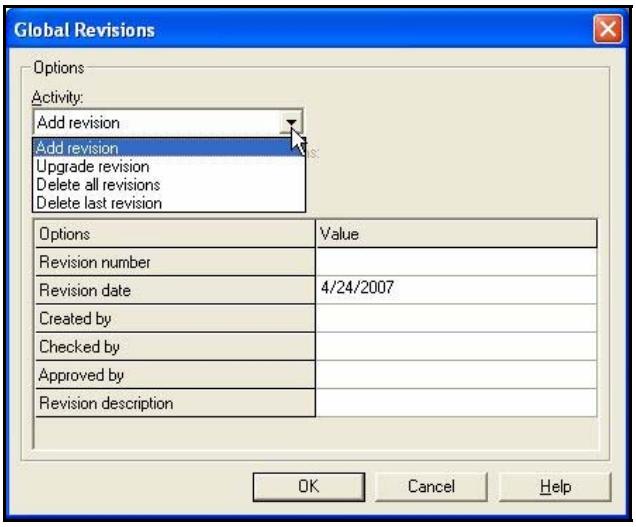


Note: Same workflow of archive document and compare works for schematics document and register report.

Global revisions

The ability to assign a specified Global revision to a set of documents is a powerful and important feature often requested by those who need to issue deliverables packages all at once, all in same or in a specified revision number.

There are several options that can be used in conjunction with this feature, and these can be seen in the Global Revision Dialog. To open the global revisions dialog, select 1 or more documents you would like to revise, and select Global Revision from action menu.



Add Revision add new revision to all selected documents. The system will add a new revision (P0, P1 type; A, B, C type; 1, 2, 3 type, or any "other" type). The system will identify the requested method and if needed will modify the method window accordingly.

Upgrade revision upgrade the selected set of documents to next higher revision level to each document. This opens a selection window from which user selects the method of revisions for documents that have no revision method yet defined or the revision method is "other"

Delete last revision delete only the last revision of the selected set of documents.

Delete All revisions delete all document revisions.

Project Reports (register report)

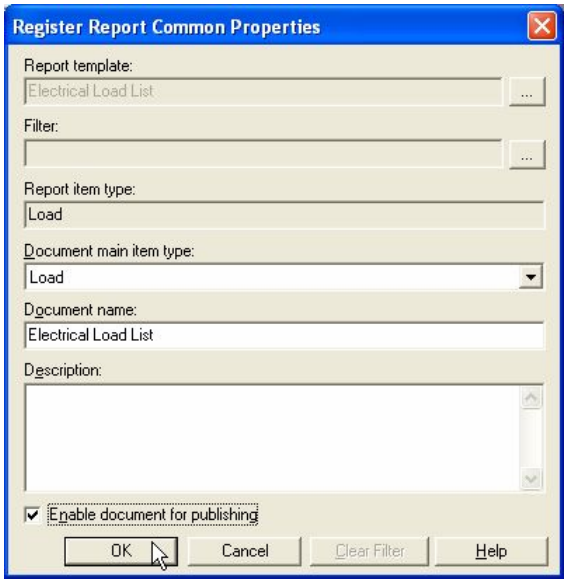
Project reports (plant reports), that were created as templates can be saved as project deliverables, that have revision level, and can be archived for future viewing and compare.

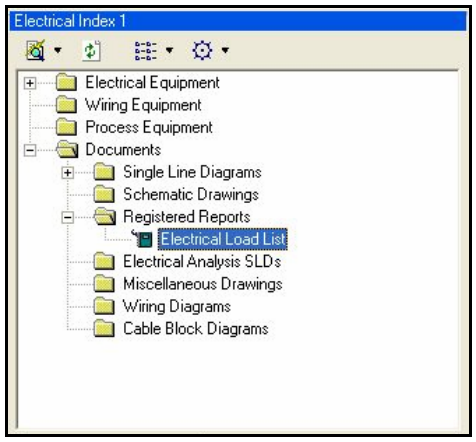
The idea is to take the Excel templates, that were created in the classic report engine as plant report and use them as the source for generating or declaring project reports. The procedure involves what is called Report Registration- meaning, to take an existing Excel template report and to transform it into a Project Document.

To register the plant report, 'Electrical Load List', do the following:

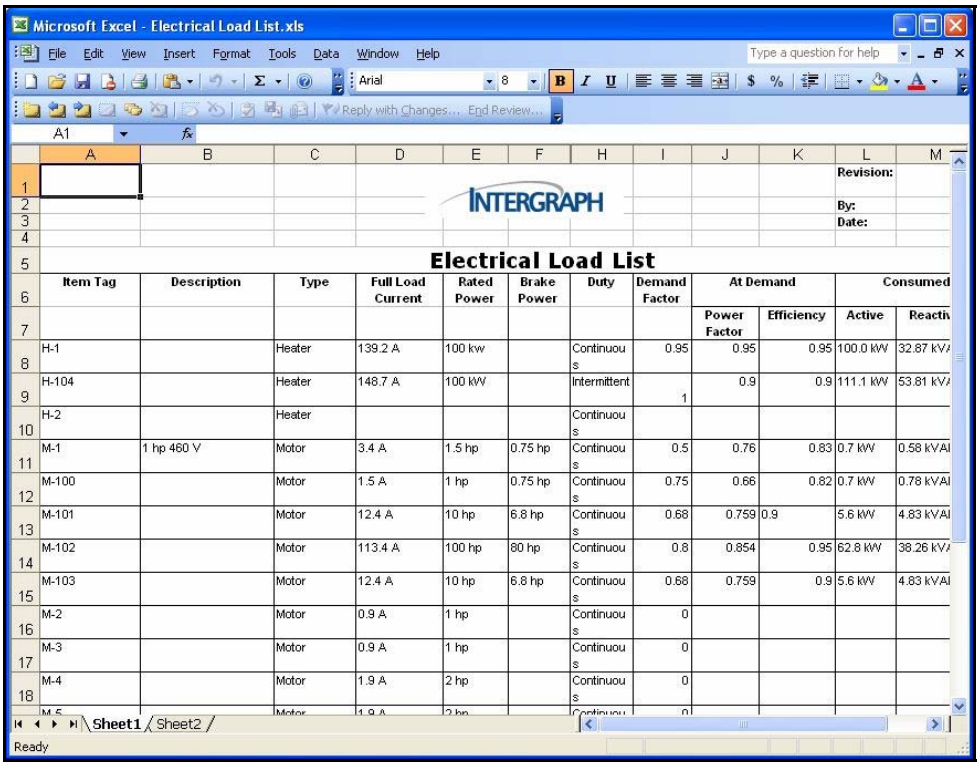
1.Select Register Report from the Actions menu.

2.From the register report common properties window, select the 'Electrical Load List', for the report template. Name the document, 'Electrical Load List'. Select OK to generate the document.

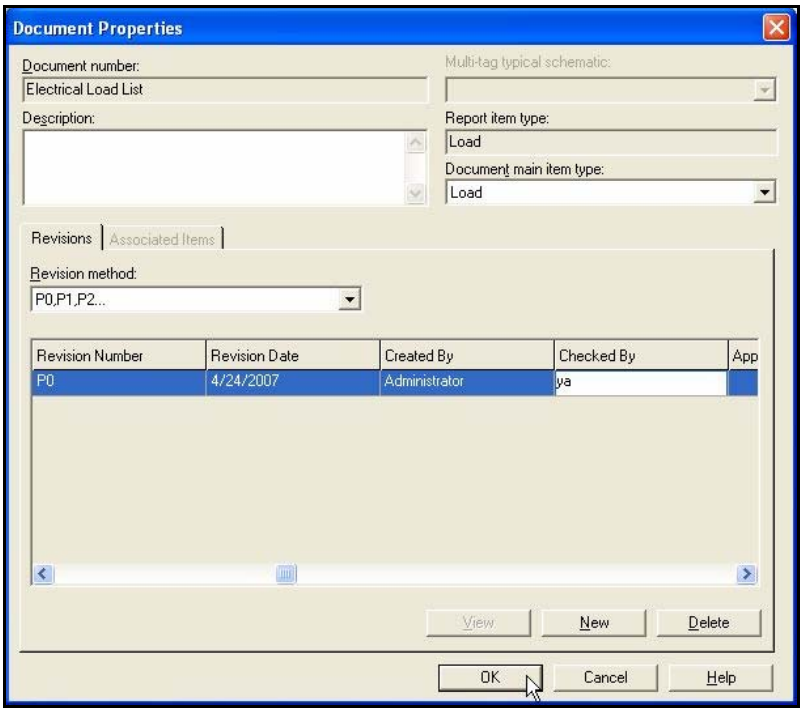


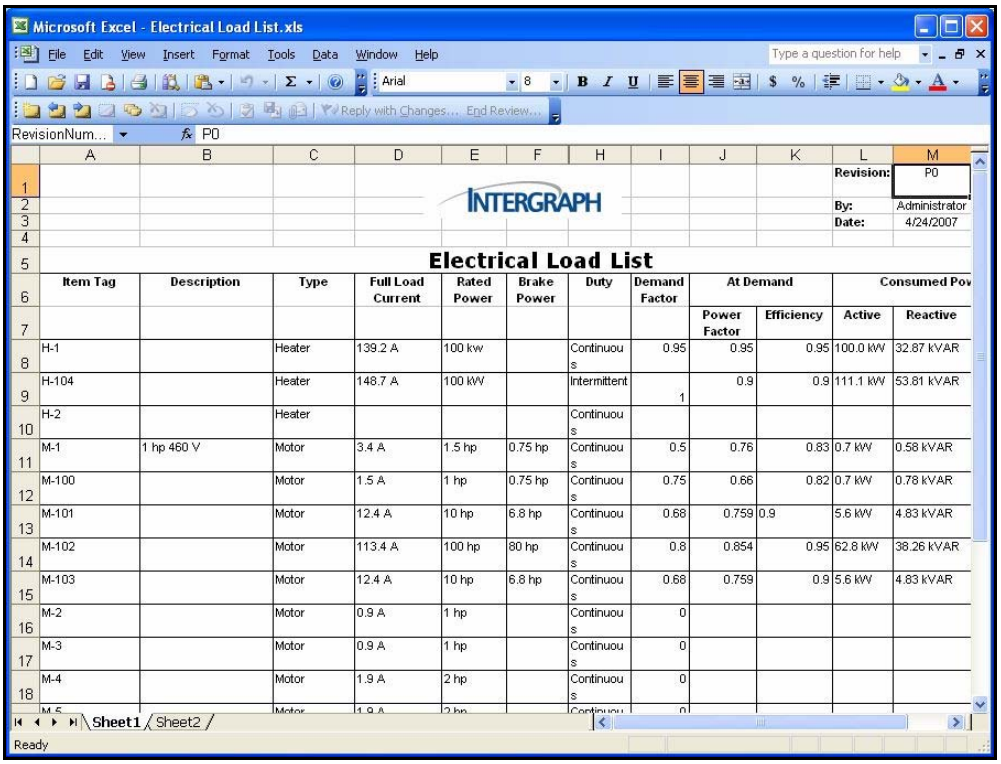


1. To revise the register report, select it from the documents folder, registered reports folder. Right click and select open.



4. While the document is open, select the document from electrical index, documents folder, registered reports folder, right click and select document properties.
5. In the document properties window select new.





Lab 12

Compare register report

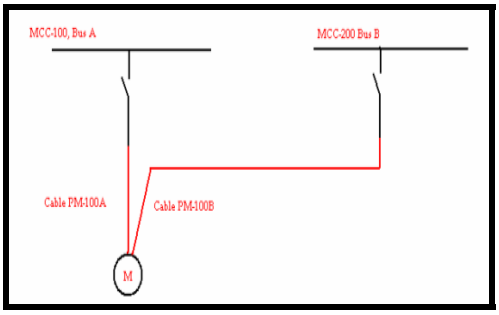
12.1. Register the plant report, ‘Electrical Load List’. Name the document, ‘Electrical Load List’.
12.2. Add first revision (P0) to the registered report document, ‘Electrical Load List’.
12.3. Change the motor rated power of motor, ‘M-1’ to 2 hp.
12.4. Run the compare document option for the registered report document ‘Electrical Load List’, and check the result.

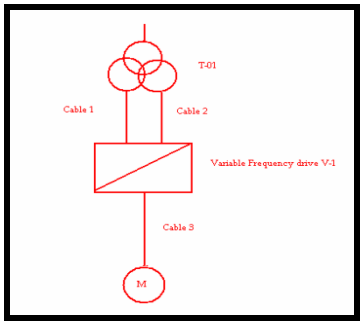
Chapter 13

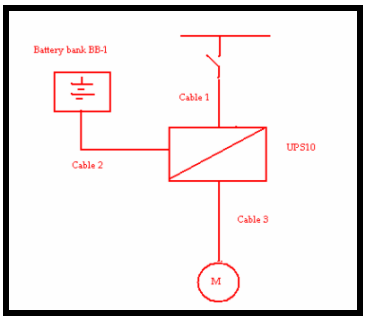
Miscellaneous Features

Dual Power Source Equipment

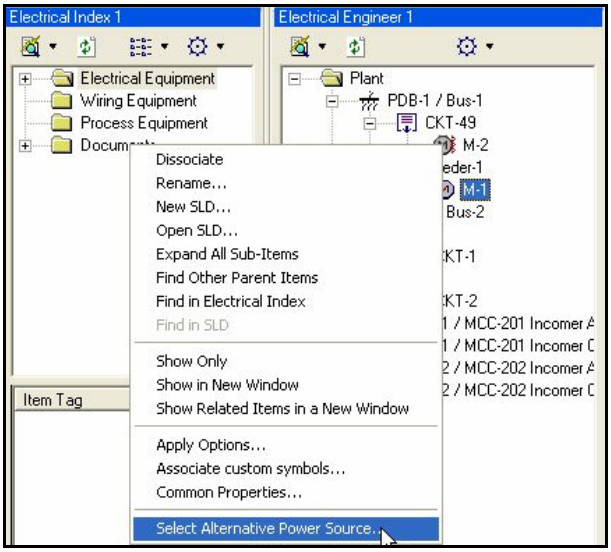
This feature is implemented to answer the requirement of equipment that have 2 power sources, a Primary and an Alternate, as follows:







The user needs to define a second or alternate power source for an electrical equipment by selecting the equipment in the electrical engineer, that has one main power source already defined:



This command will invoke a second Electrical engineer instance, from which user will select the alternate power source.
The following conditions should be met for this operation to be successful:
The equipment is not already connected to an alternate feeder
The destination power source is a valid parent (power cable, feeder circuit, etc).

Power Dual Source Exercise

In this exercise we shall learn and accomplish the following:

▪ How to connect a variable frequency drive to a main power source, coming from a VFD feeder circuit.

▪ How to connect a second power source supplying the VFD from a power transformer.

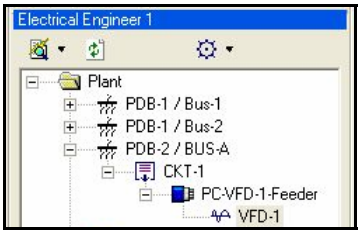
▪ Generate a single line diagram for this configuration.

▪ How to dissociate the second power source.

▪ Switching and modifying the second power source.

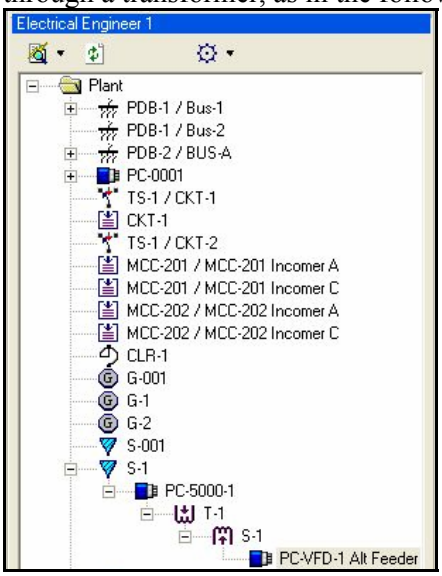
Step 1 - Connect the VFD to the Main Power Source

Create VFD-1 and drag it to under a feeder circuit via a power cable

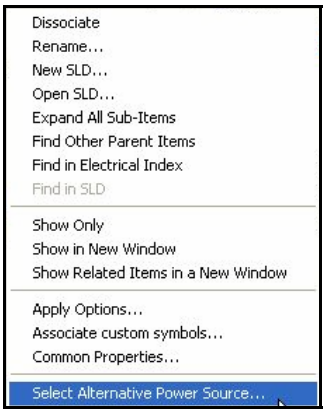


Step 2 - Connect a Second, Alternate Power Source for VFD-1

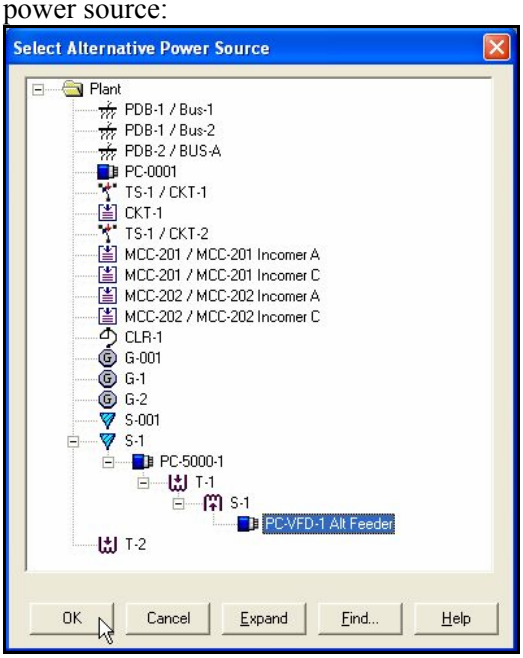
Let prepare a second power source for the VFD-1 by creating a new feeder branch from another bus through a transformer, as in the following picture:



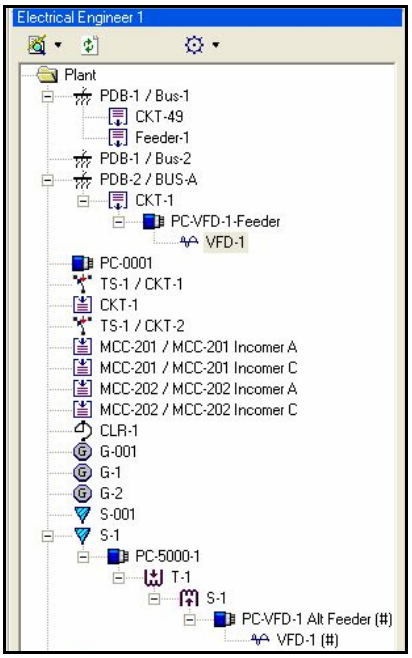
Right-click on VFD-1 and select “Select Alternative Power source..” :



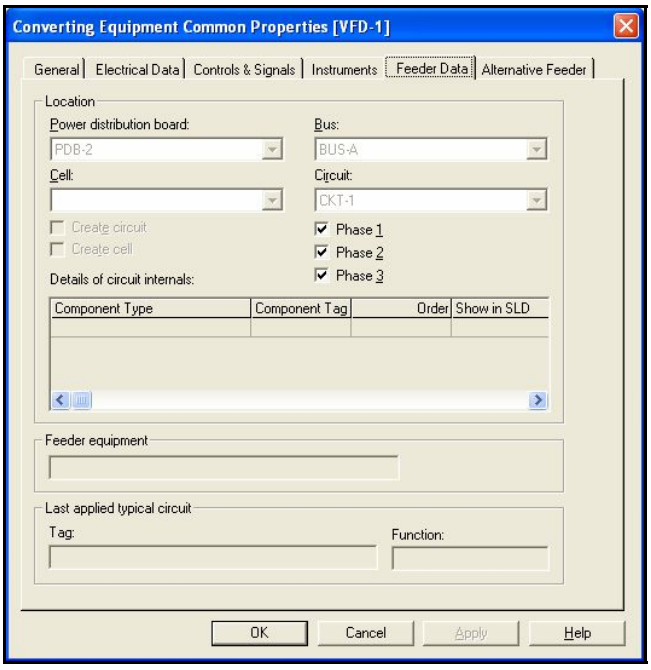
This will open a second instance of the Electrical Engineer that we shall use to navigate to the second power source:



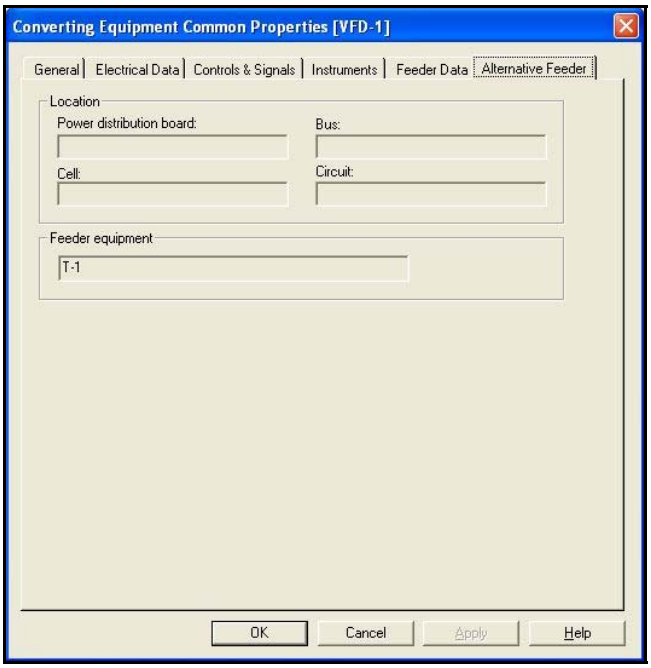
Select 'PC-VFD-1 Alt Feeder' source from the 'Select Alternative Power Source' window, and select OK. The result will be as follows:



The "(#)" sign indicates that this is an alternative power source branch
If we look at the VFD common properties, Feeder Data tab:

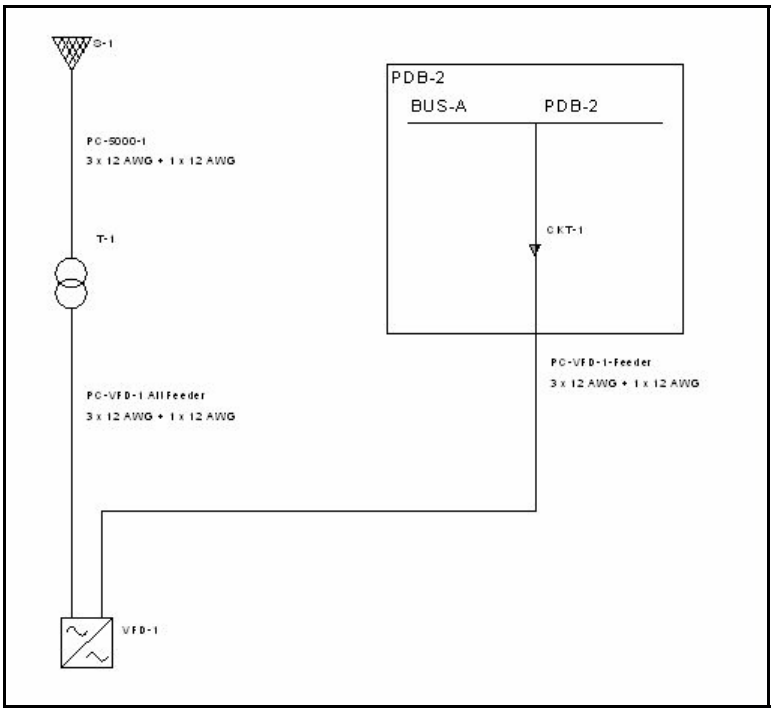


And in the Alternative Feeder tab:



Step 3 - Generate an SLD for This Configuration

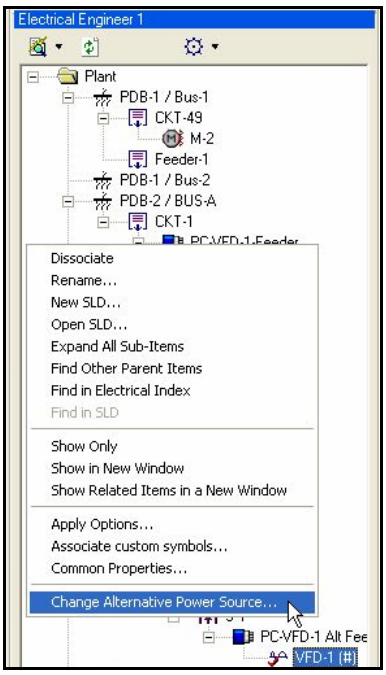
For the dual power source equipment to show properly, a new symbol must exist for the item type, named as the original but with Alt added to the name. In our case it would be '**VariableFrequencyDriveAlt.sym**'.

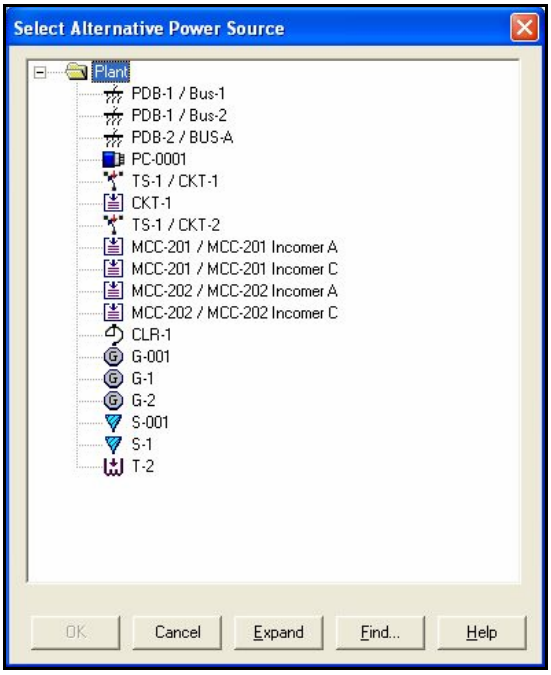


Step 4 - Change the Alternate Source

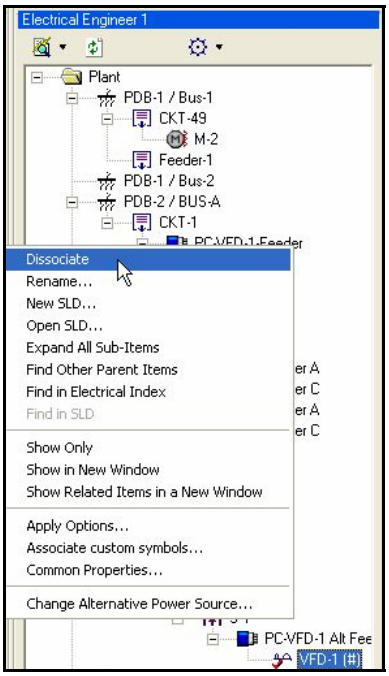
In the electrical engineer, select the VFD-1 Feeder cable, right click, and select change alternative power

Source. This will open once more the select alternative power source screen, and allow you to navigate to an alternative feeder.





Step 5 - Dissociate the Alternate Power Source
Dissociation of the alternate power source is possible only at the final element itself, in our case, it is the VFD:



Rules
The following rules will apply:

Case A - Equipment has both primary and alternative power source and alternative is connected via a cable or a longer multi item branch

The instance that connected to Primary power source can be dragged or dissociated as usual. All operations on the items with regards to his feeder will be assumed as operations on its primary power source.

To edit/change the alternative power source will be as follows:
The instance of "equipment" that is connected to an alternative power source can only be dissociated in the engineer (the system will identify the alternative power source from the primary). User will have to select the item in the engineer and dissociate. The only place along the chain of the feeder that can be broken (dissociation taking place) by the user is the equipment itself and not any of its branch items.
The "Cable" (or any item in the feeder chain), connecting between alternative power source and "equipment" cannot be dissociated or dragged.

For re-connecting of the branch to another alternative power source the user will have to use the response Engineer window again.

Case B - Equipment is connected to alternative power source only, no primary is connected.
"Equipment" and its "Cables or branch items" (if exists) are connected to Alternative Power Source only.
All actions that are not using the "select alternative power source" command will be interpreted by the system as regular operations on the primary power source, so, Dragging "equipment" to any other branch will add a main power source, leaving alternative power source untouched.
Dragging or dissociating of "Cable" in an alternative power source branch is not allowed. The "Cable" can be re-connected using response window.

Case C - Equipment has no power source at all, just connected to a feeder cable (after apply for example)
"Equipment" doesn't have any feeding equipment but has a "parent Cable or something else".
"Equipment" can be connected to alternate power source using response window, "Cable" can be dragged to any valid power source creating a new main power source for the "Motor"

Delete an Item that Feeds an Alternative Power Source
The system does not allow you to delete an item that is in the path of the alternative power source feeding electrical equipment, only after the electrical equipment has been dissociated from its immediate upstream item that feeds it.

Dissociate Alternative Power Sources
The dissociation of the alternative power source will be also from the engineer, by the use of the "Dissociate" command.
Dissociation will be possible only at the final element and not along any of its feeder items..
Dissociating primary source will not affect its alternative and vice versa.

Rules in Operations that Involve Circuits
All the operations and activities that deal with circuits on equipment will be acting on the main feeder and not on the alternative circuit.

Associating an item to a alternative source
The alternative source will not be required to match the last applied typical circuit of the item.

Applying typical circuit
Applying typical circuits on a dual input connected item will affect only its main feeder circuit, not the alternative circuit.

Single line diagrams
For the purpose of proper handling in the SLD, the dual input equipment would need to have a special symbol that will have 3 connection points, to handle both the 2 inputs (loads, instruments and cabinets) and 2 inputs + one output for the converting equipment types.

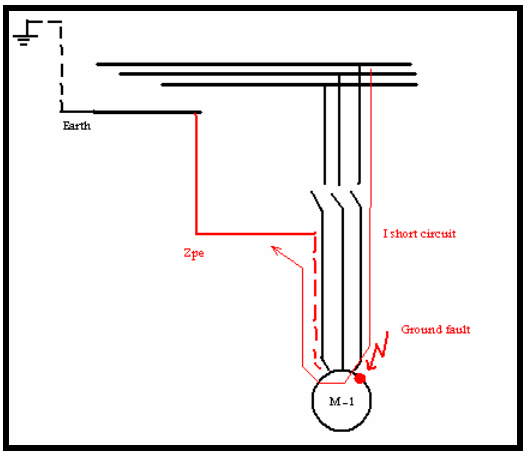
The 3rd connection point of the symbol will be dedicated for the Alternative Power source
The SLD algorithm will position the dual input item so that its main feeder point will be positioned first, and the system will run the connector from its alternate feeder to the alternative connection point at the equipment.

Load Summary reports
Since the "All feeder report " load summary may count the load twice in case the load is fed from 2 different user will have to disconnect the circuit by setting the Circuit mode parameter to disconnect.
In case user doesn't do that, the load summary will count the load twice.
The drill down report will assume that circuit is connected unless CircuitMode=Disconnect

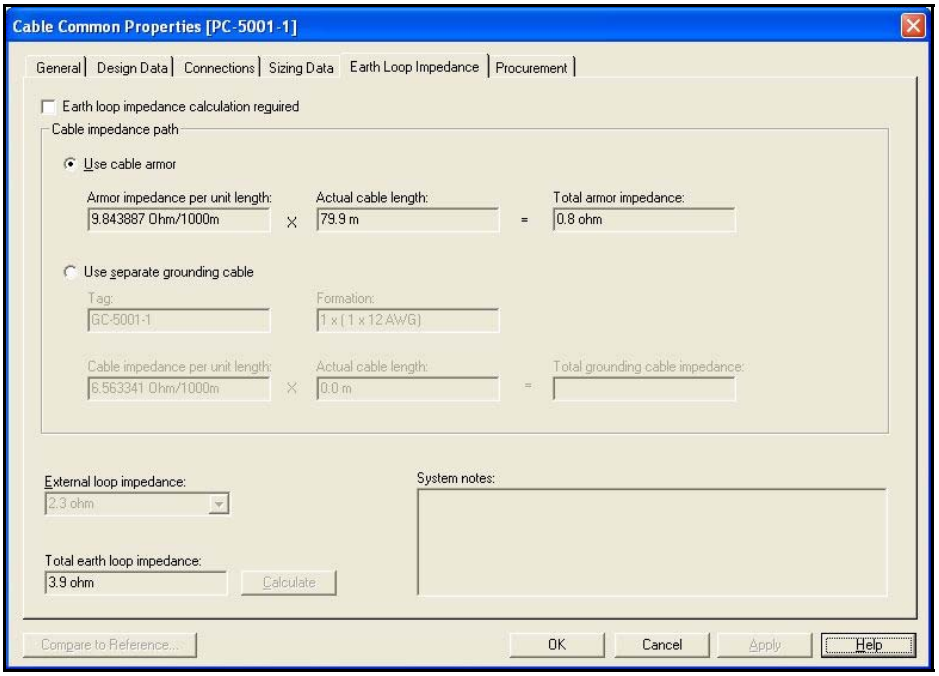
Batch load assignment
The batch load assignment, when operated in Normal mode (not drill down), will not count the load in case the circuit is feeding the load as alternative power source. The load will be accounted for in batch load assignment only for circuits that feed the loads as primary power sources.
The All Feeder mode will take the load into account in both cases, provided the alternative power source feeder circuit is not in disconnected mode.

Earth Loop Impedance Calculations in Cable Sizing

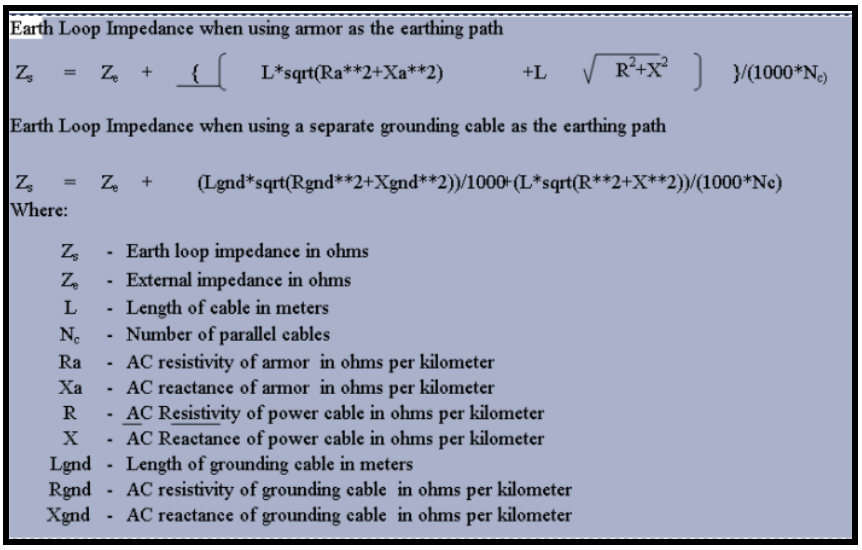
The Earth loop impedance is defined as the total resistance seen by the source of current when clearing a faulty current to the earth. The earth loop impedance is an important safety factor since it determines what will be the highest possible potential at the point of the fault to earth. IEC regulations restrict this voltage to be no higher than 50 volts, therefore, EPC are requested to calculate this value on a per cable basis and make sure that the protective device for the circuit, fuse or circuit breaker protection values and settings are such that nor time nor earth potential will exceed codes, and will be able to clear the faulty current in a predetermined time.
The calculation is done on a per cable basis and it includes both the cable armor earth impedance (in case the armor is the earthing path) and any external to the cable impedance that is present from the power source through the final load and back to earthing point.
The following illustration shows how a short circuit current flows through a earth fault:



In order to calculate the earth loop impedance user will select the Earth Loop Impedance tab of the power cable Common Properties:



The calculation is based on the following formula:



The calculation is based on the earth path formed by the cable armor metal sheet, or, of a separate grounding cable associated to the "to" side of the load.
Any change in the cable grounding cable or properties

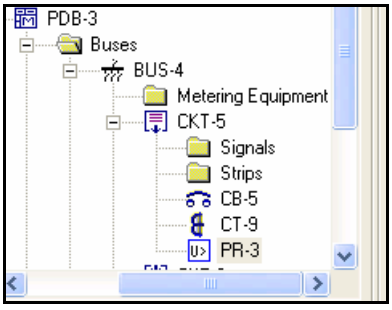
Working with Metering and Protection Relay Symbols in the SLD

Metering and protection relay like Ammeter,Voltmeter,Multimeter, and Protection relay can be add to SLD and become database object. You would be able also to define the logical relations between themselves and any other piece of equipment in the single line diagram, and saving it to the database.

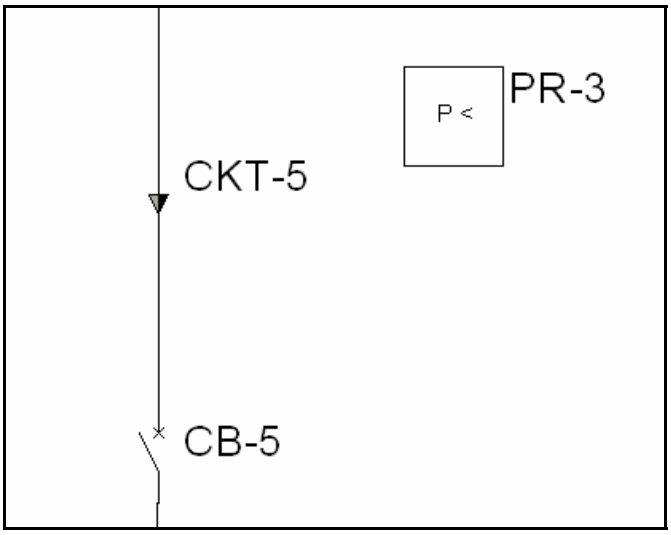
Lets create a circuit that has:

- Circuit breaker
- Current transformer
- Protection relay

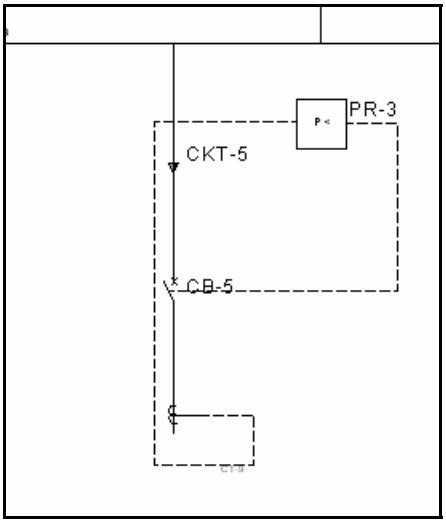
And have them automatically shown in SLD, and we will just reposition them and interconnect the logical lines, performing the required relations:



Each of these circuit internals should be associated with a symbol that has been designed so there is at least one "auxiliary" connection point



Activate the connection mode to light the red auxiliary connection points and connect them as needed:



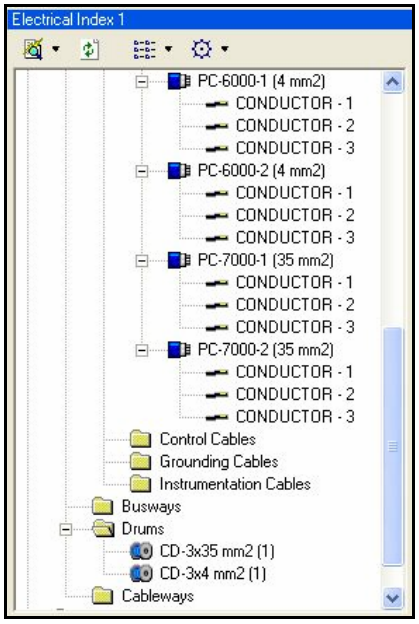
To disconnect and remove a relation simply right click on the relation line---→Disconnect.
Saving the drawing will save it with the set of auxiliary relations.
Opening a new SLD drawing with the same item selected will show the metering equipment and the relations.

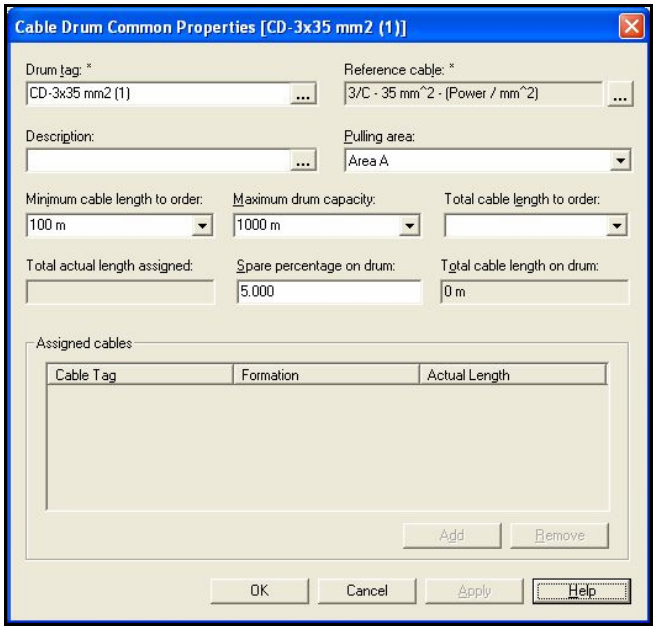
Note: You may use the auxiliary connection points and lines for any other purpose of showing any relation across items. Each relation must have its own 2 auxiliary connection points. You may not connect 2 relations to the same connection point.

Cable Management System

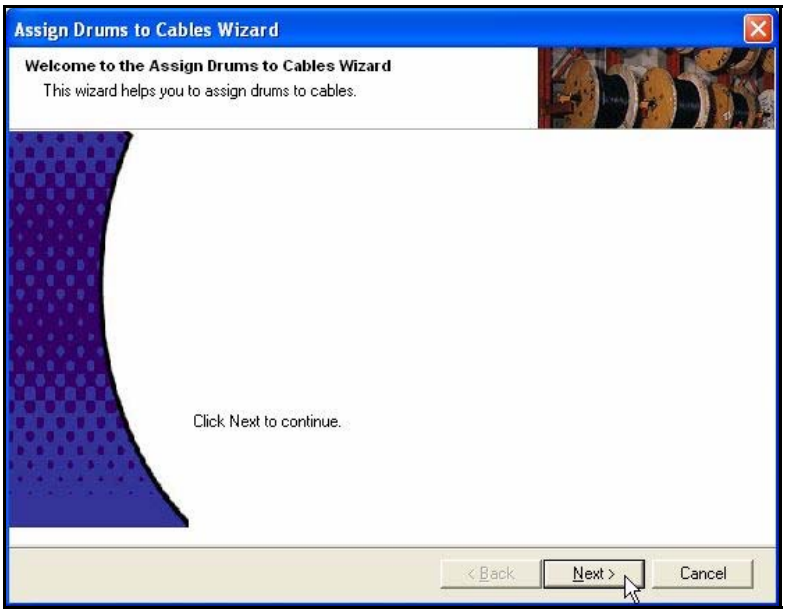
Automating Cable to Drum Assignment

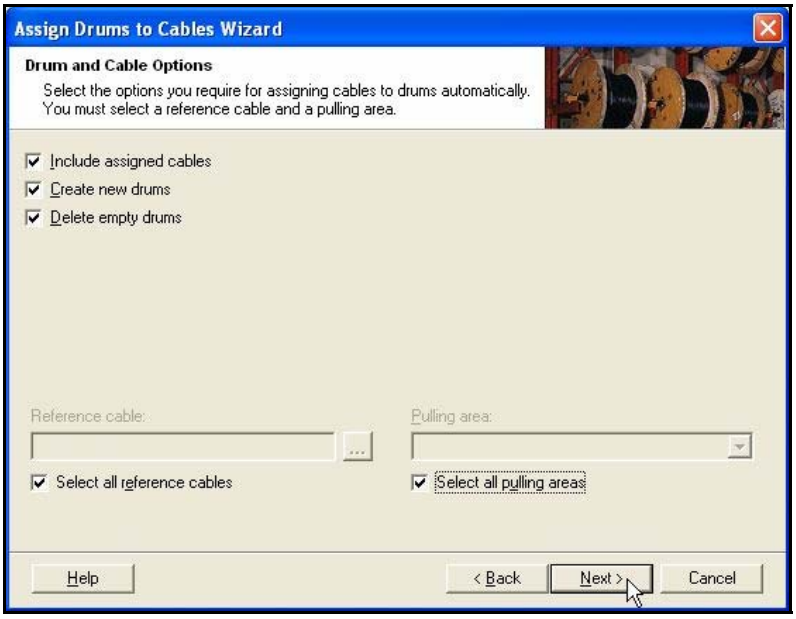
Lets create 2 drums, one for 3 x 4mm2 and second for 3 x 35mm2.
And also lets create 2 power cables, 3 x 4 mm2, and 2 power cables, 35 x mm2.





At this point in time we shall do an automatic cable to drum assignment
This procedure can be automated using the following User Interface:





Include assigned cables means that we wish to do a complete revamp in the design and start the assignment from scratch. This will result in first dissociation of the assigned cables from the drums.

Create new drums will create new drums in case the operation fills up all existing drums and still cables are left to be assigned, otherwise, these cables will be left untouched waiting for new drums to be created manually.

Delete empty drums in case the auto assignment shuffles results in empty drums, they will be deleted

Select all reference cables will handle all the cables of all types in the project, otherwise, only for the selected type of cable

Select all pulling areas will perform the operation for all pulling areas, otherwise, only for the selected one.

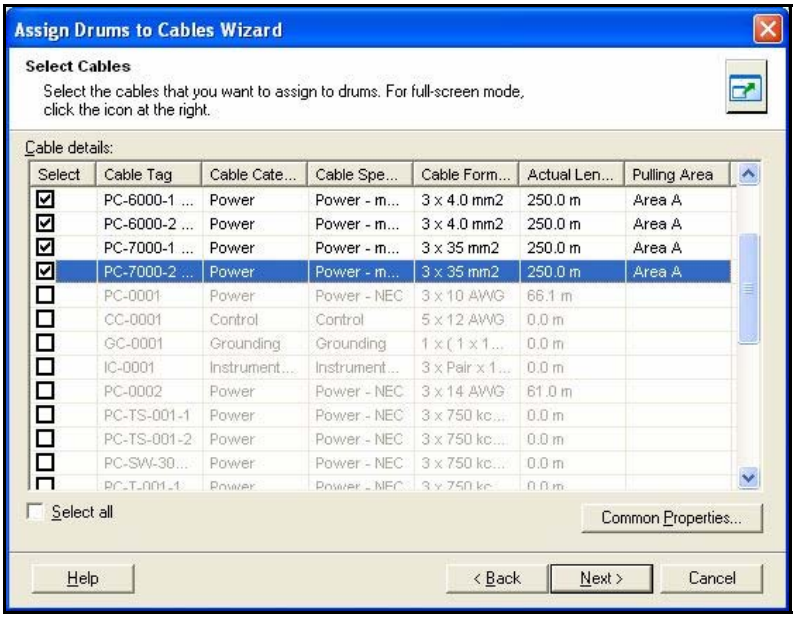
The algorithm works as follows:

Based on the defined cable set (all reference cables, include assigned cables), the system will organize them in a descending actual length order, from the longest cable down to the shortest, per cable type and per pulling area set, as defined in the check boxes options.

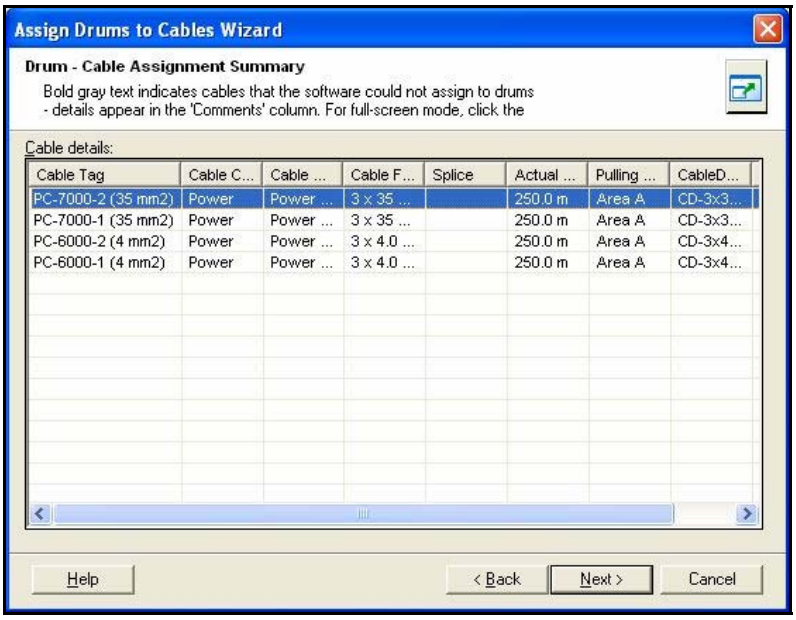
The system will also set the same order to the existing drums of the same reference cable, starting from the drum that has the longest free cable available, to the shortest (free cable on drum would mean the maximum cable on drum figure – total cable assigned to the drum, including spare).

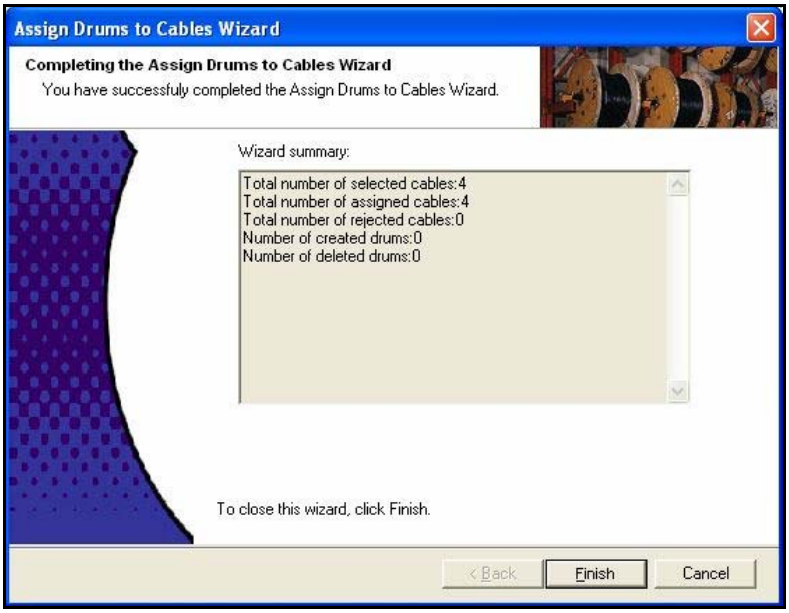
Then, the system will start, per pulling area and per cable type assigning the longest cable to the drum which has the longest free spare (provided it doesn't exceed the maximum cable on drum figure)

Following the selection of set of cables the Wizard will find the cables and sort them:

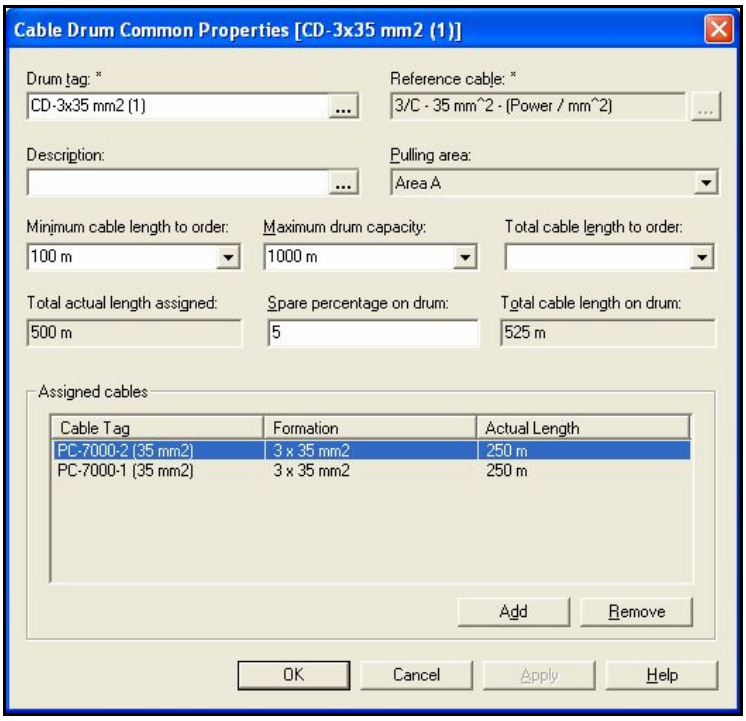


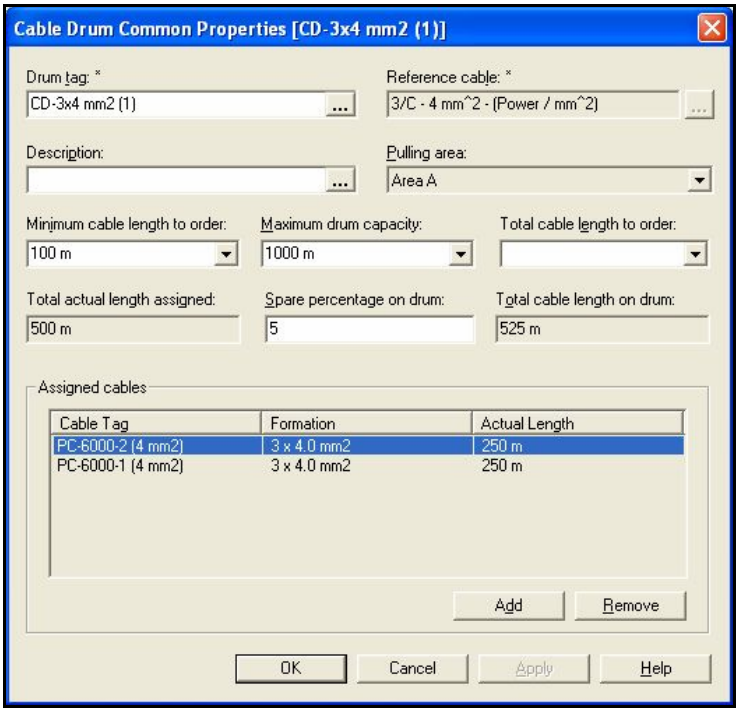
Note: Cables with no actual length or with no pulling areas cannot be selected and will not be processed.
If data is missing for a cable do a right-click → Common Properties to edit the cable data.
Selecting the cables and clicking the Next command:





Cables have been assigned to corresponding drums, from the longest to the shortest. The system needs to accommodate and take into account that the actual cable on the drum must include the spare.





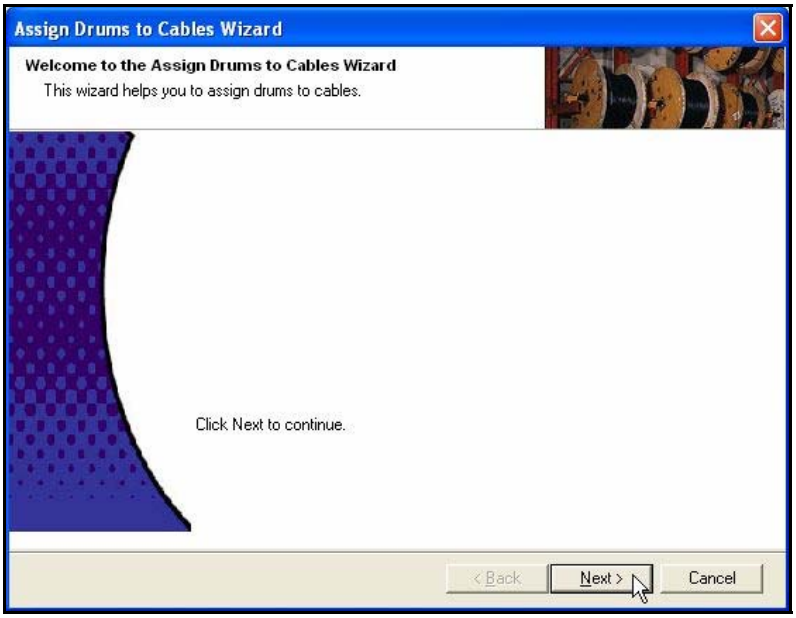
Cable splices in drums

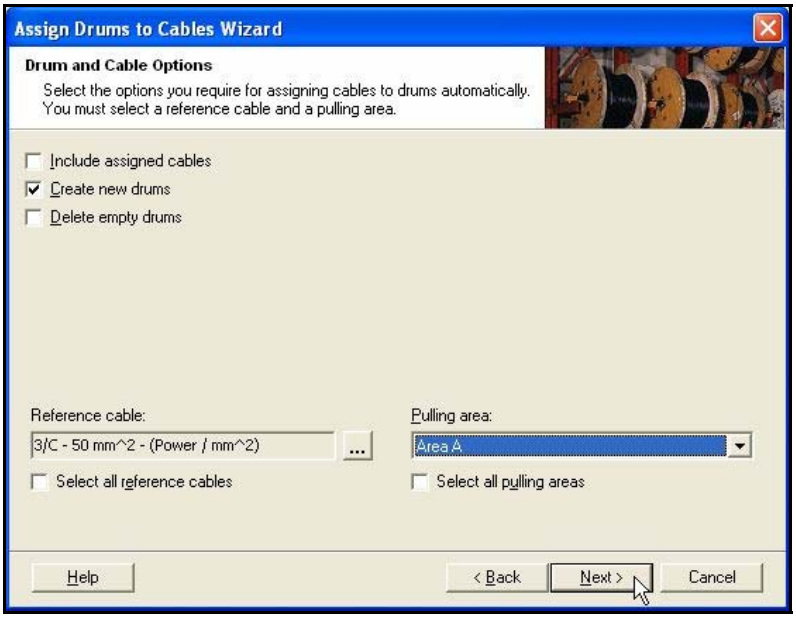
Cable splices in drums allow you to manage the cable's drumming with length more than maximum drum capacity. There are often situations in which cable lengths are very long, exceeding the standard maximum cable allowable on drum. The solution that would be normal practice would be to order these cables as cut-lengths, and using splices to connect the cut lengths.

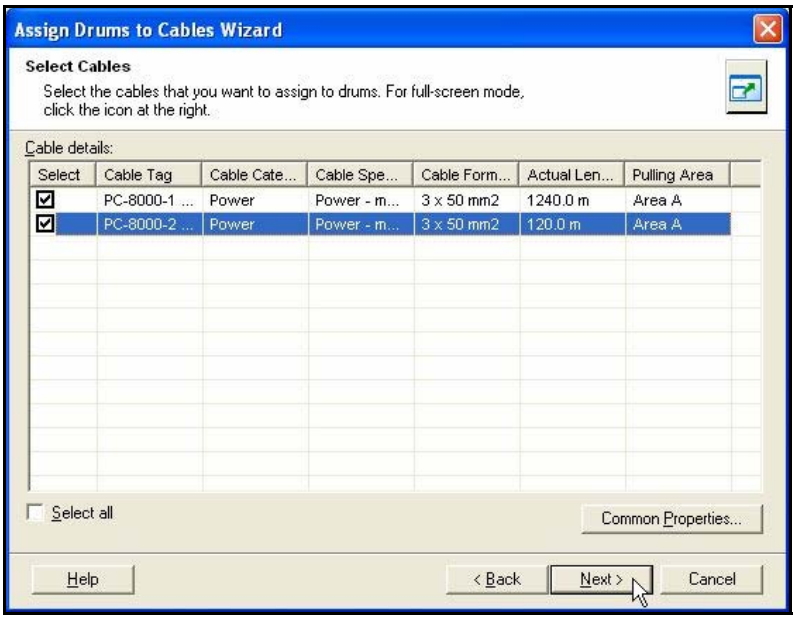
Smart Plant Electrical cable splices in drums automating the procedure of defining the number of splices based on the drum maximum capacity per cable and cut virtually the cable.

To demonstrate how this works let us use the 3 x 50 mm2 cable and set the RDE capacity on drum to be 400 meters.
Lets pick 2 project cables of same cable formation and assign them to the same pulling area.
Set one of the cables to have the length of 1240 meters and the other cable have 120 meters.
With these 2 cables selected in Index:

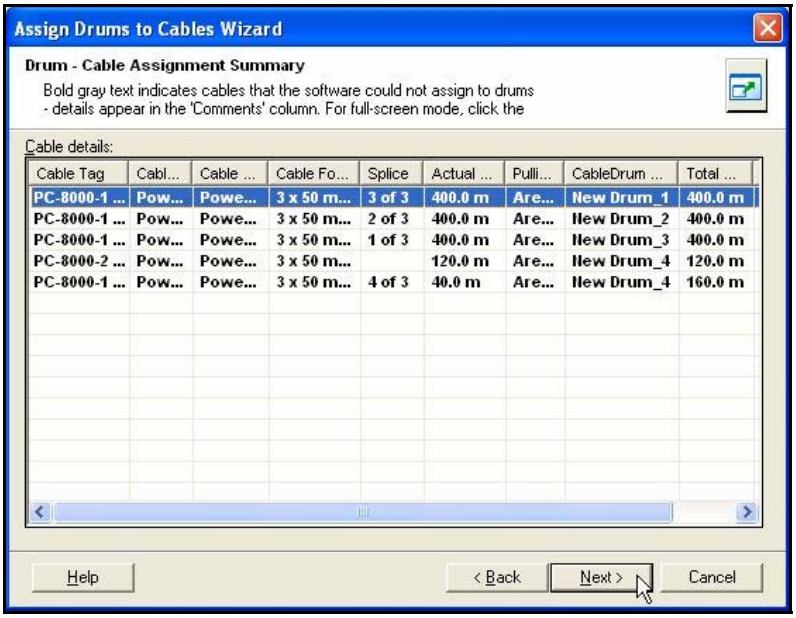
Once the wizard opens, enter the data as shown:



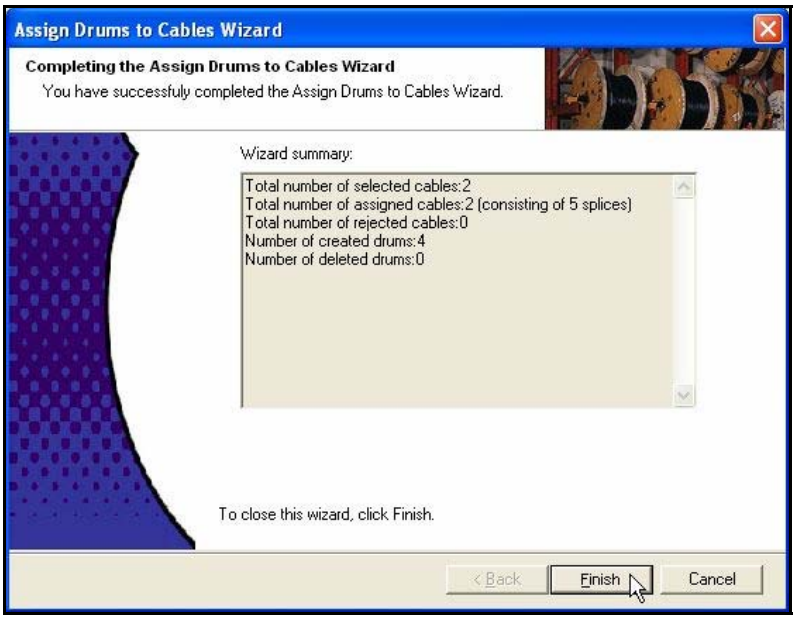




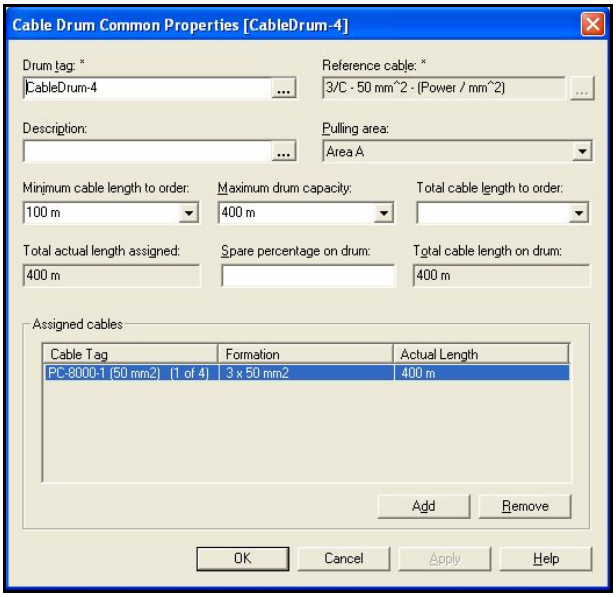
Next screen will show how this long cable was split into "virtual cuts":

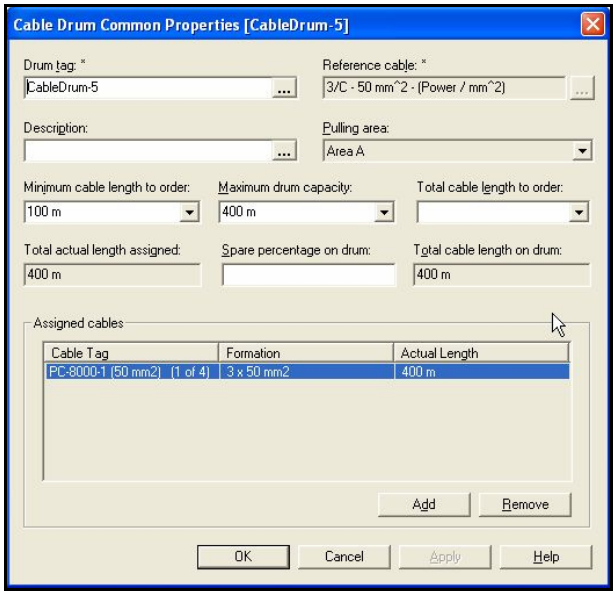


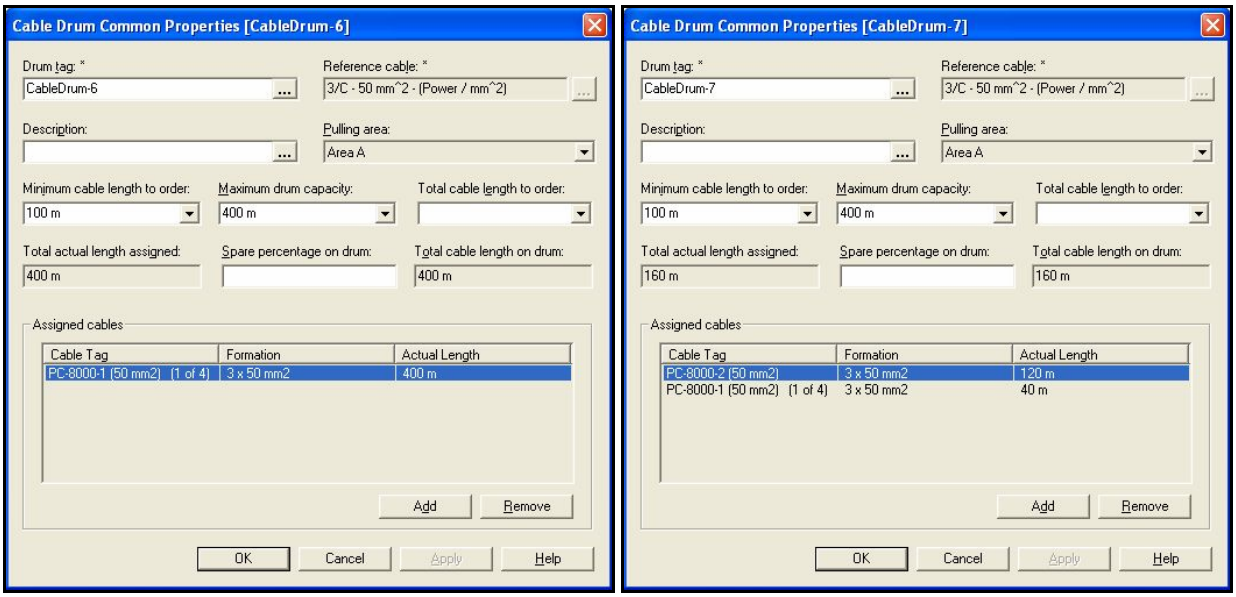
The 1240 meters long cable has 3 splices, each on a different drum, with the shortest one sharing the same drum with another short cable that could fit in one drum.



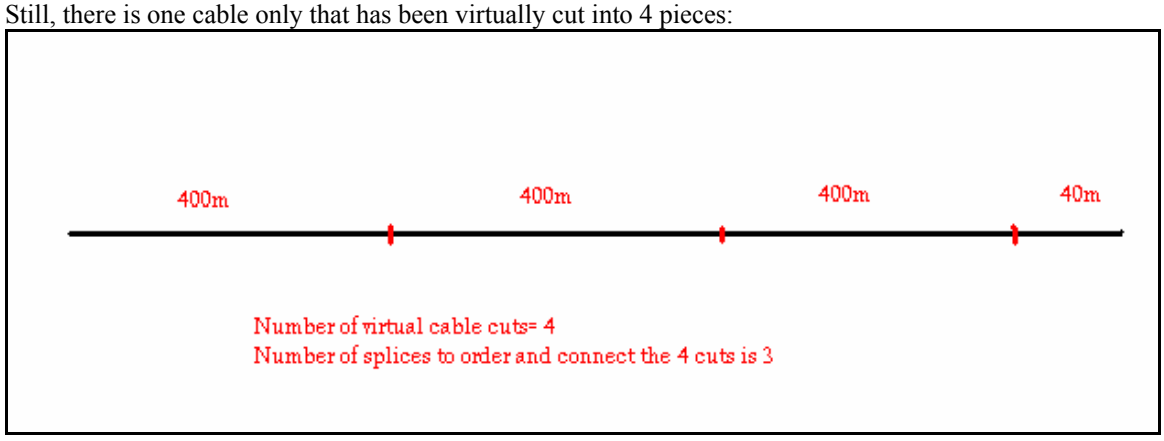
Looking at the drums common properties forms:





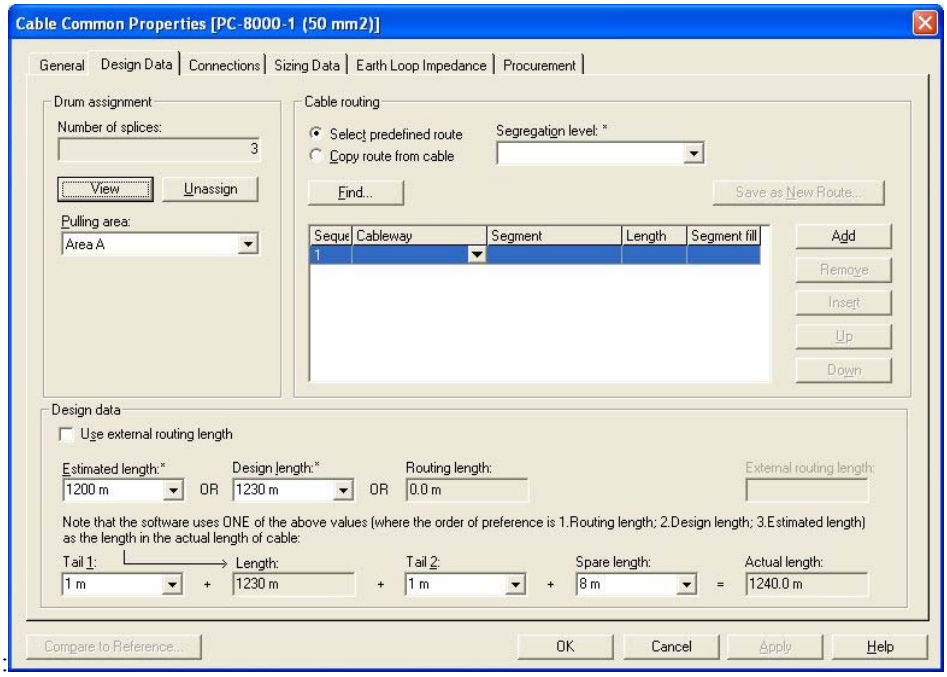


Still, there is one cable only that has been virtually cut into 4 pieces:

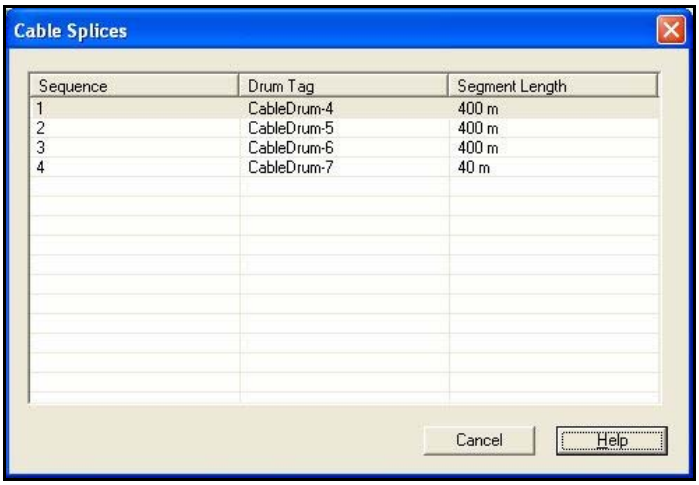


Cable splices cannot be removed from the drum and need to be accessed via the cable common properties.

To see this in the cable common properties select the long cable , the one with the 1240m



We can clearly see that we have this cable "cut" into 4 pieces. Clicking on the View will show.



Note: The number of splices is the number of points at which the cable is cut. It is NOT the number of pieces of cables that will exist.

Un-assigning the cable from the drum will eliminate the splices and the virtual cable cuts.
User may customize the cable schedule report to include the information about the cable being spliced by using the new property "Number of Splices"
The drum composition report will show each of the cuts on its drum.

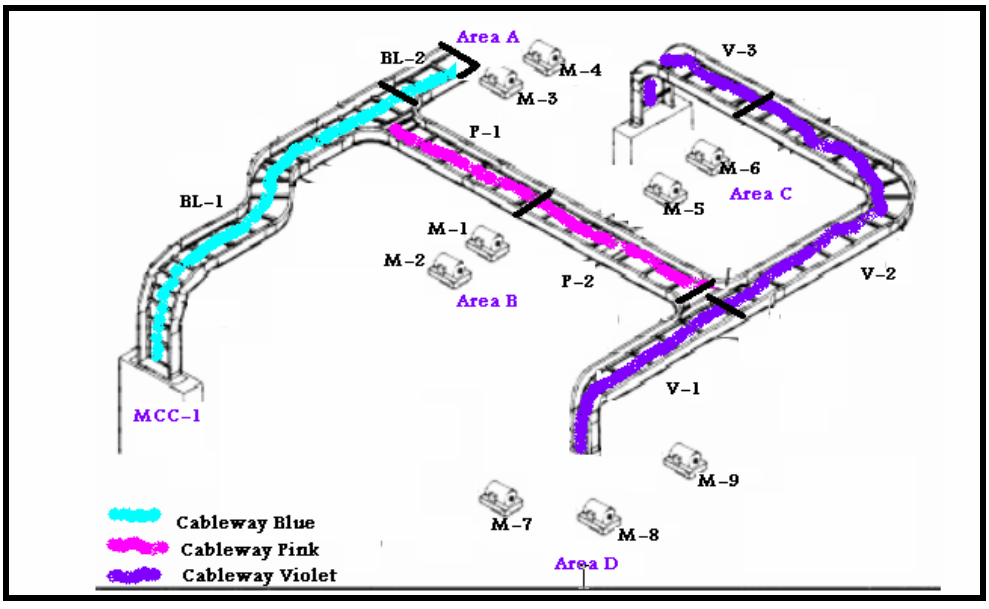
Cable Routing

The cable management supports the following activity:

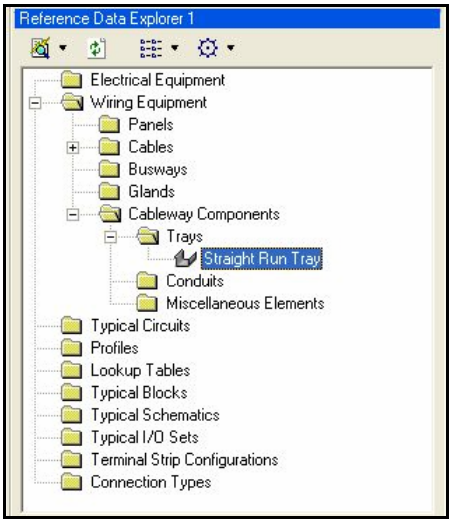
- Creation of a library of reference items for Trays, Conduits and Miscellaneous cableway components
- Assignment of Project segments to reference cableway components
- Defining Predefined cable routes
- Connectivity of segments in between themselves
- Assignment of cables to cableways using the following methods:
- Manual segment to cable assignment
- Copying predefined routes to cables
- Copying of one cable route to another cable
- Batch cable routing using manual, copy route and copy from other cable route
- Segregation level definitions and validations
- Segment fill validation.
- Segment weight validation.

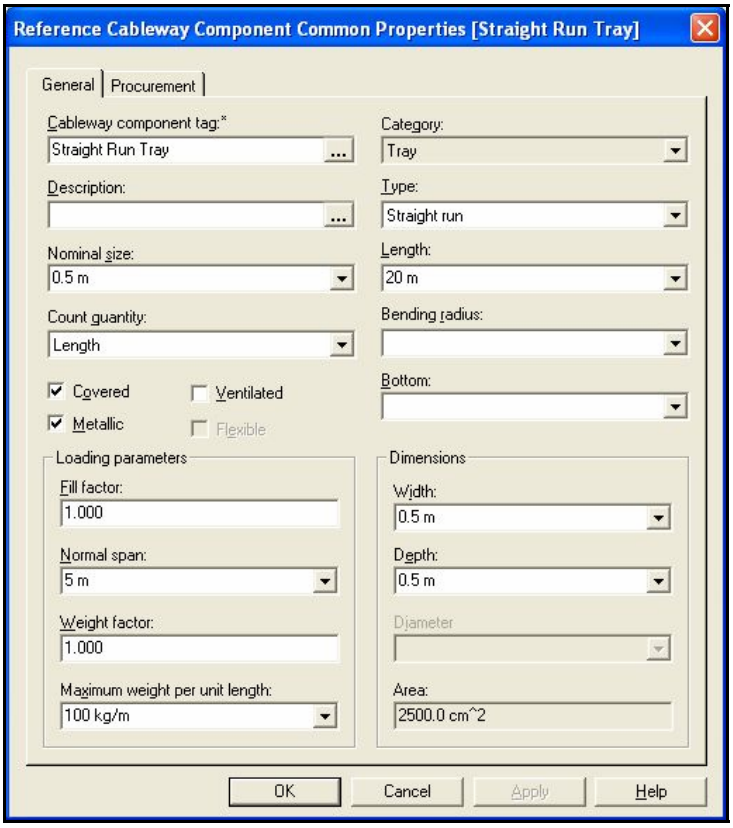
Cable Routing Exercise

We will create 3 cableways, with the Electrical Index segments, specify tray types, defines predefined routes and perform cable routing, as per the following equipment layout:

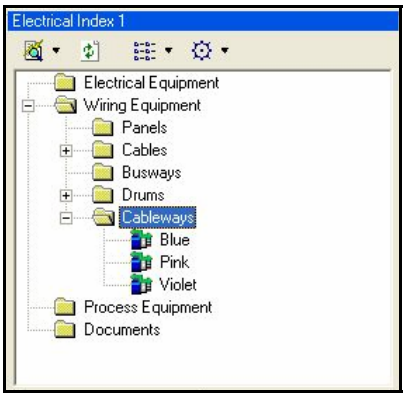


Step 0 - Create typical tray
Create typical tray in the RDE so we can use it for project tray.

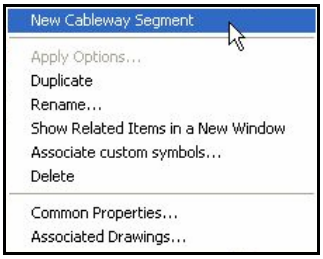


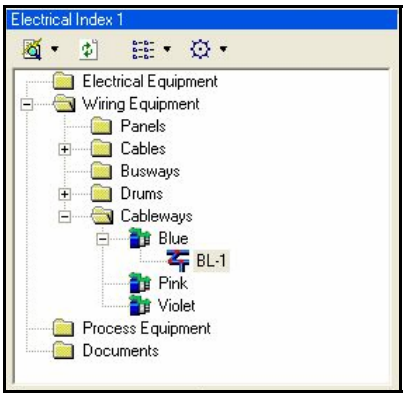


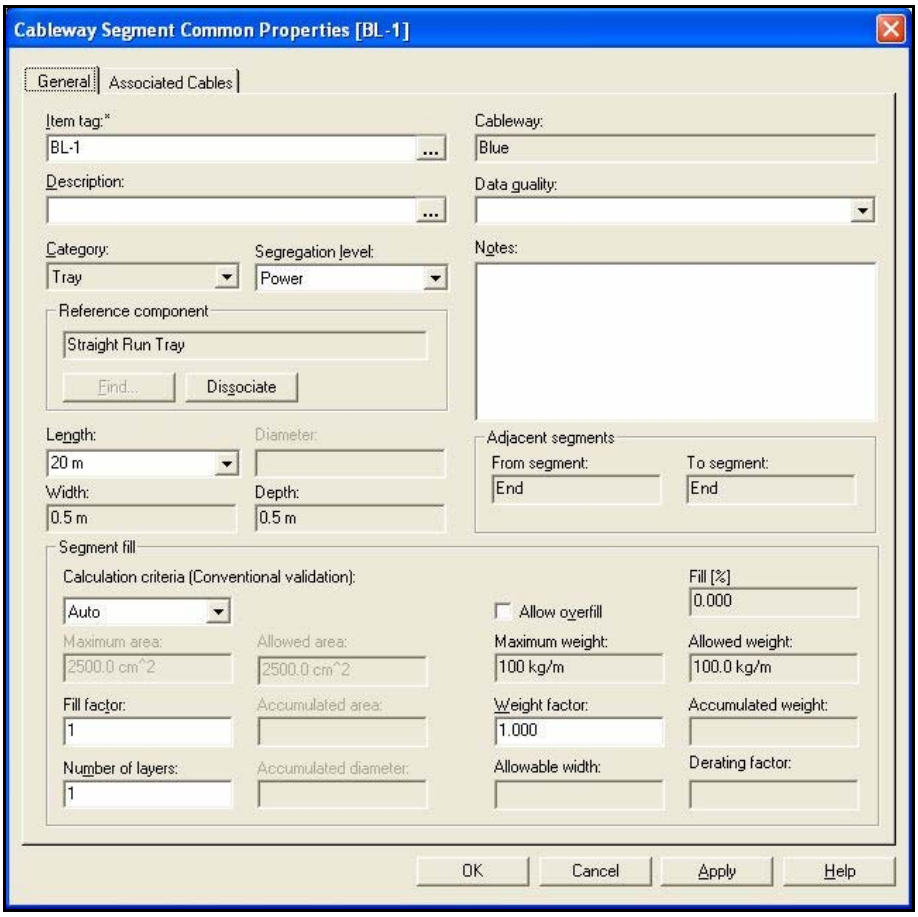
Step 1 - Create cableways infrastructure
In the Index, select Wiring →Cableways and create:
Cableway "Blue", "Pink" and "Violet"



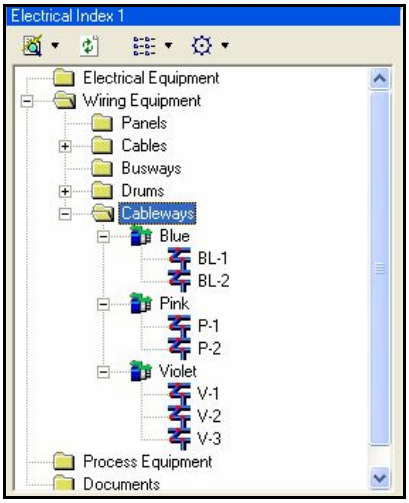
Right-click on the "Blue" cableway to create segments "BL-1" and "BL-2", as follows:





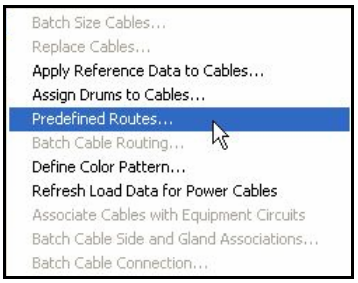


In the same way create the entire segments within the 3 cableways:

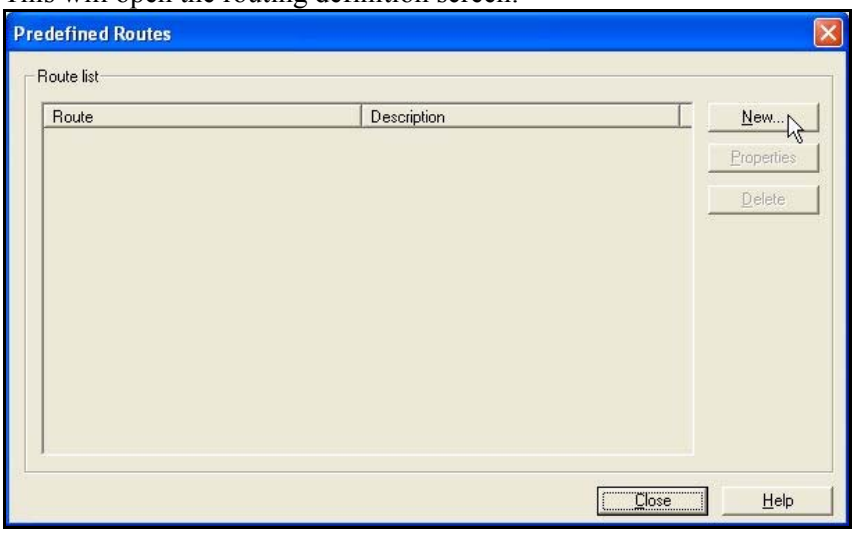


Step 2 - Create Predefined Routes

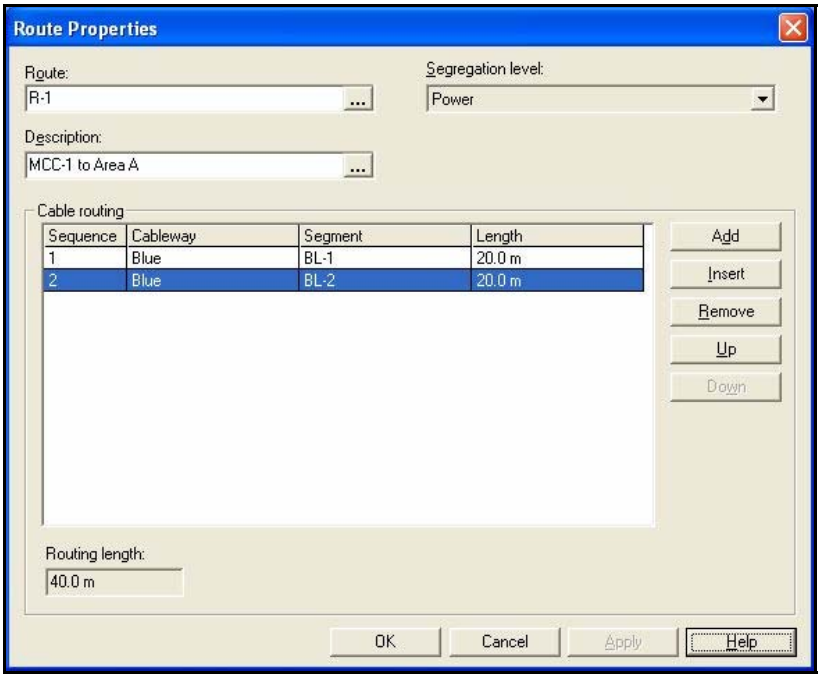
Since we have 4 areas in which equipment is located, let us create 4 predefined routings, which we shall use later to route the cables to the motors.

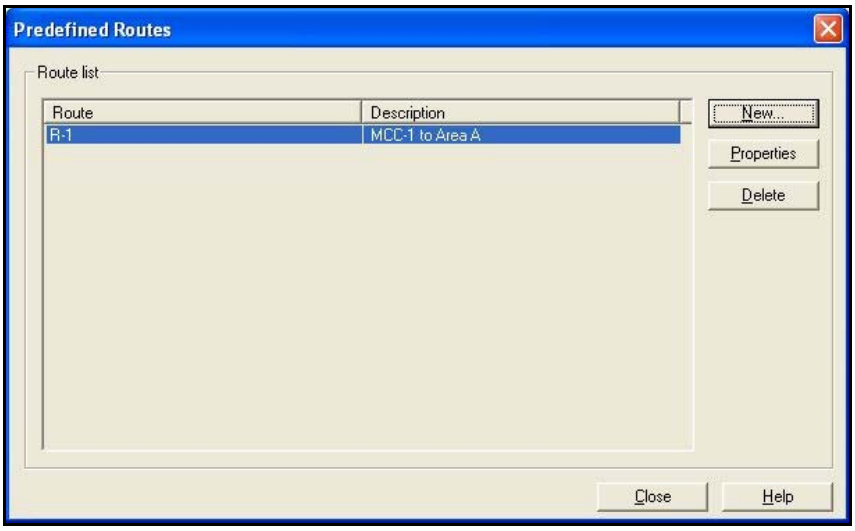


This will open the routing definition screen:

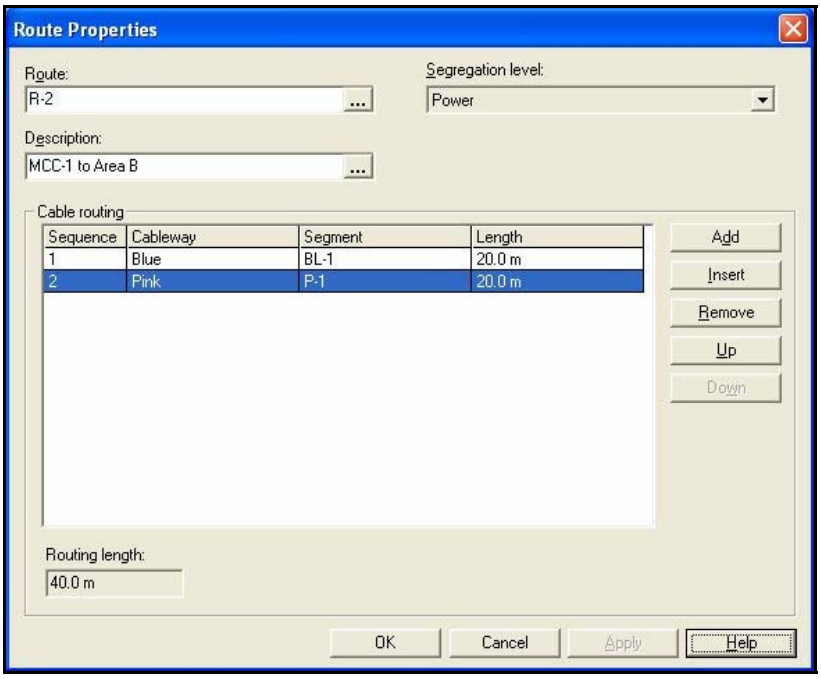


Click **New** to create a new predefined route, R-1, from MCC-1 to Area A (for motors M-3 and M-4):

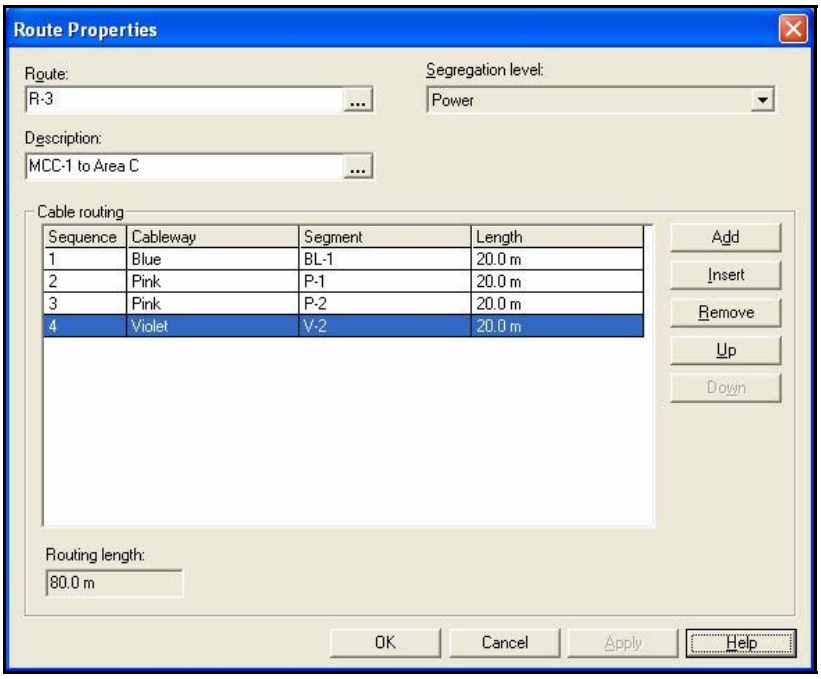




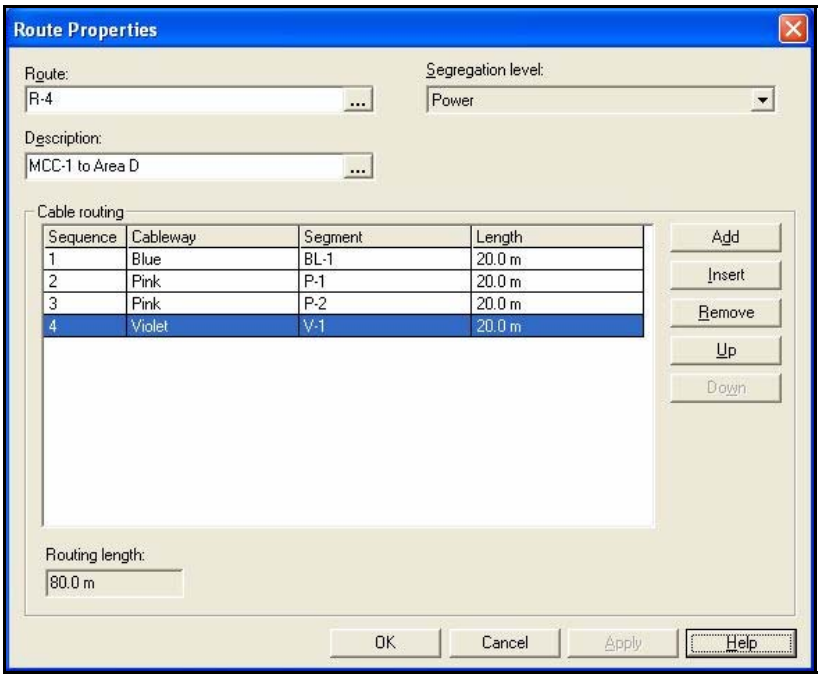
Repeat this procedure to create now the other routes, as follows:
R-2, for MCC-1 to Area B



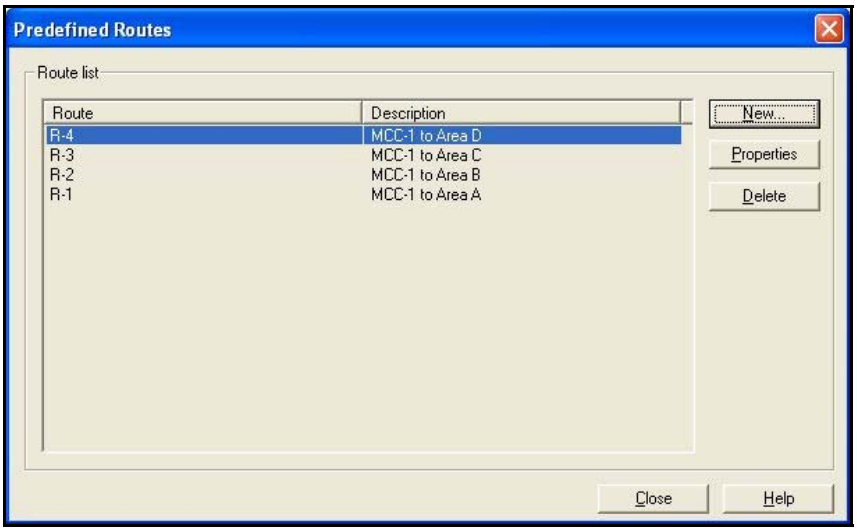
R-3, for MCC-1 to Area C



R-4, for MCC-1 to Area D



At this stage we should have the 4 predefined routing paths:



Step 3 – Define the Cable Routing

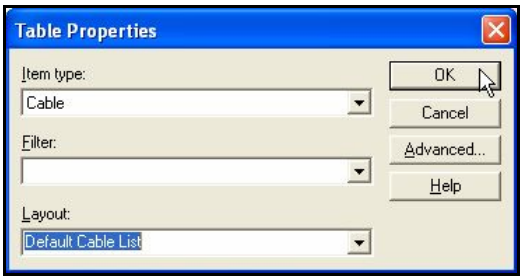
We should have in our project, already available, feeder cables for all the motors, as follows:
For M-1, cable PC-M-1-1-Feeder
For M-2, cable PC-M-1-1-Feeder
For M-3, cable PC-M-3-1-Feeder
For M-4, cable PC-M-4-1-Feeder
For M-5, cable PC-M-5-1-Feeder
For M-6, cable PC-M-6-1-Feeder
For M-7, cable PC-M-7-1-Feeder
For M-8, cable PC-M-8-1-Feeder
For M-9, cable PC-M-9-1-Feeder

According to the equipment layout, the cables should be routed as following:

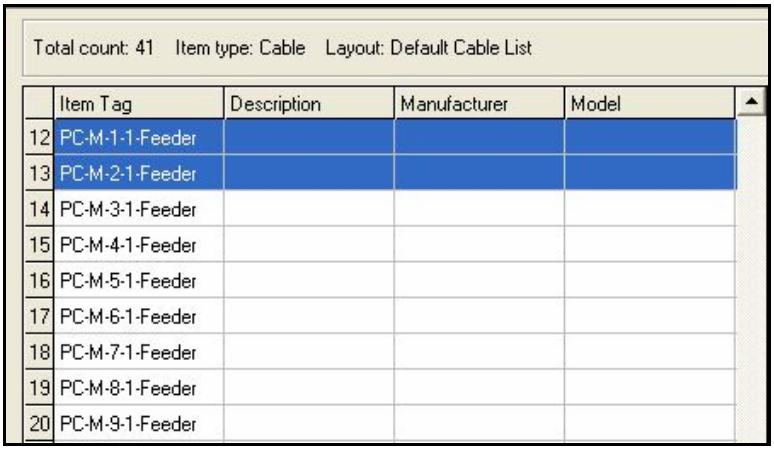
Cable	**Route**
PC-M-1-1-Feeder	R-2
PC-M-2-1-Feeder	R-2
PC-M-3-1-Feeder	R-1
PC-M-4-1-Feeder	R-1
PC-M-5-1-Feeder	R-3
PC-M-6-1-Feeder	R-3
PC-M-7-1-Feeder	R-4
PC-M-8-1-Feeder	R-4

PC-M-9-1-Feeder	R-4

Open the Tabular Editor, and select “Default cable List” layout

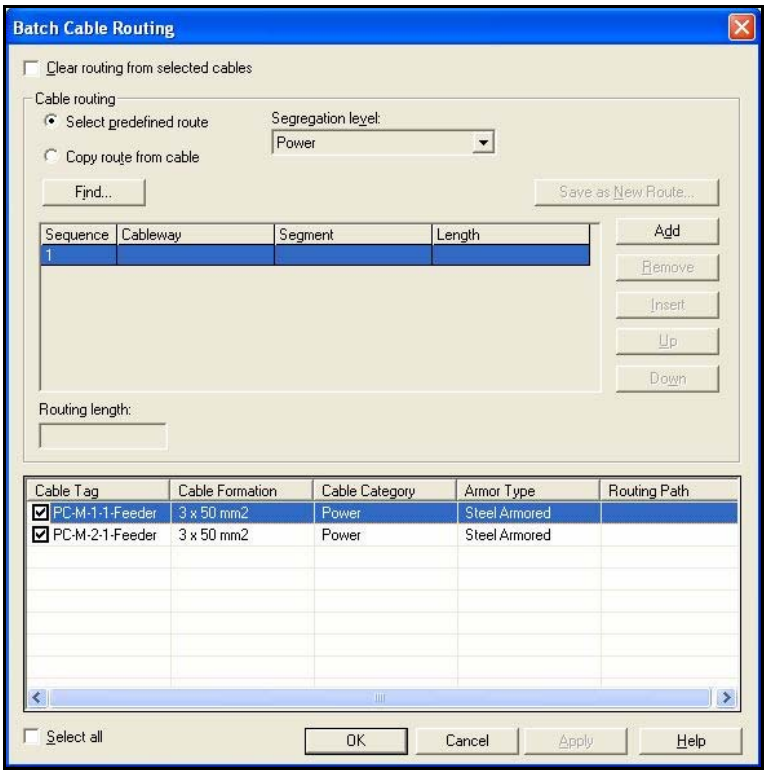


And click **OK**.
From the open table, select cables ‘PC-M-1-1-Feeder’, and ‘PC-M-2-1-Feeder’

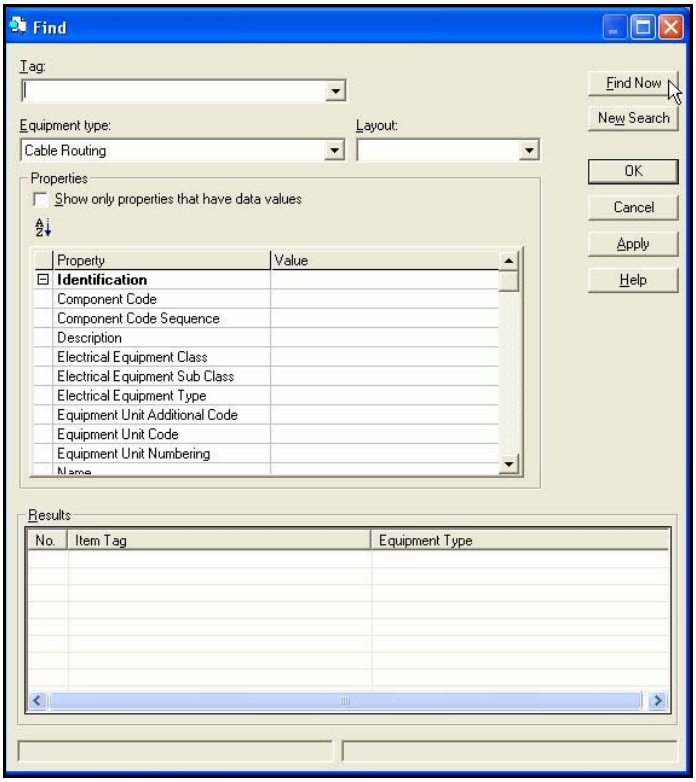


Click Actions → Cables →Batch Cable Routing:

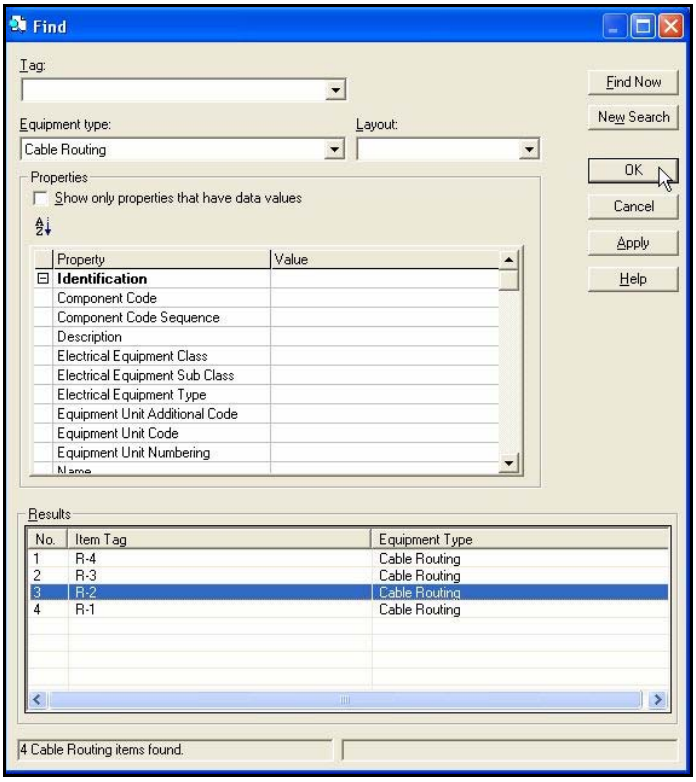
The **Batch Cable Routing** dialog box opens with the 2 selected cables shown in the grid:



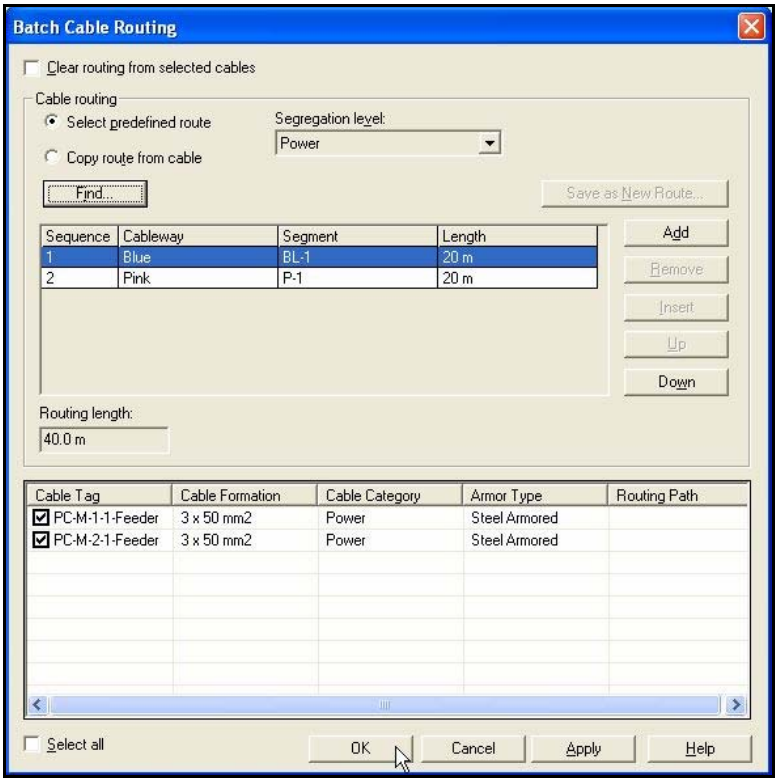
Select the **Select predefined route** radio button and then click **Find**:
This operation opens the Find dialog box, and allow you to search for a predefined route.



Click **Find Now** to find all the routes that match the specified segregation level:

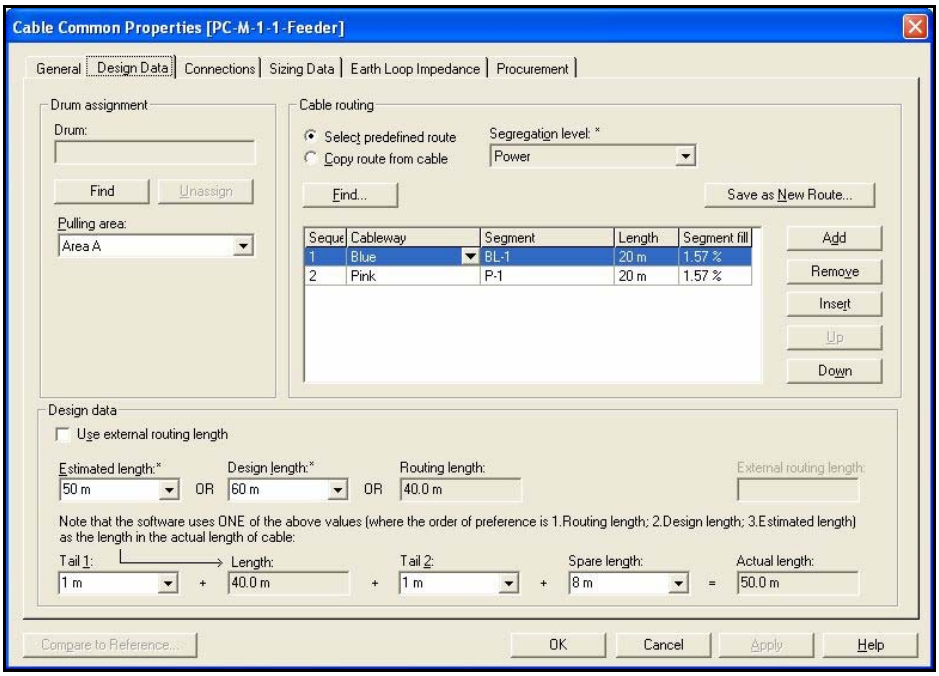


Select R-2 and click **OK**. Observe that the routing data of R-2 has been copied to the Cable routing grid area:



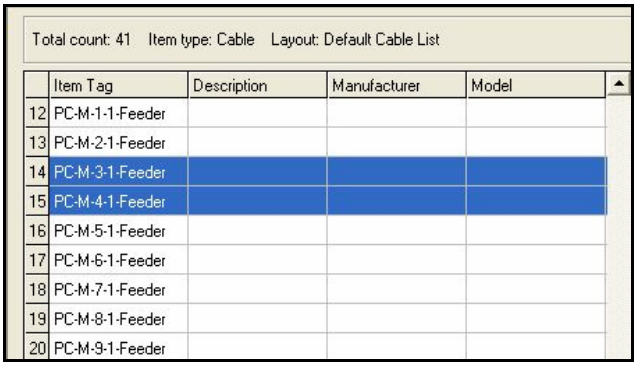
The following message is displayed:

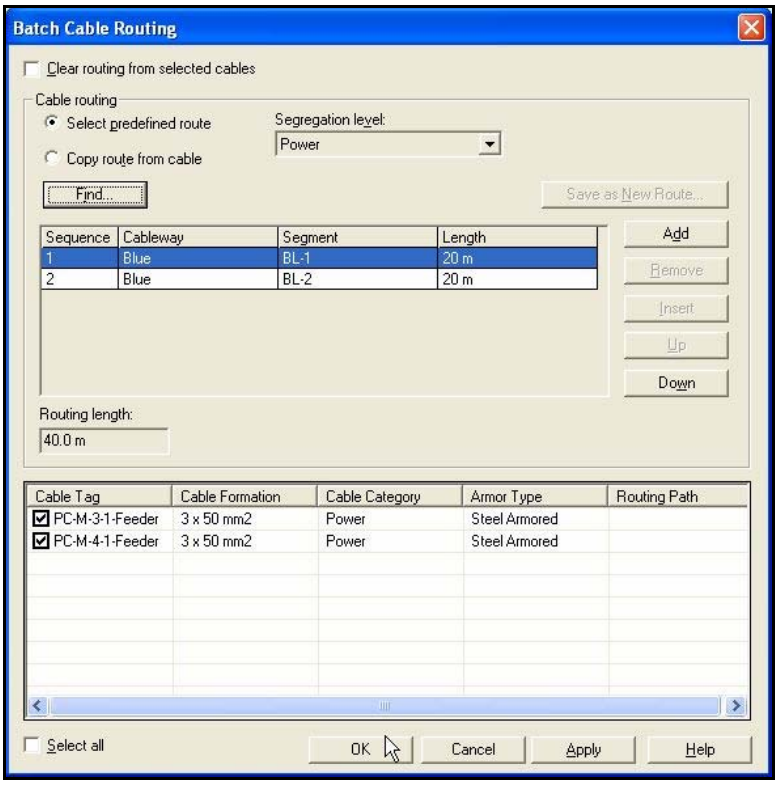
To make sure the cables have been routed, let us look at the cable common properties of 'PC-M-1-1-Feeder':



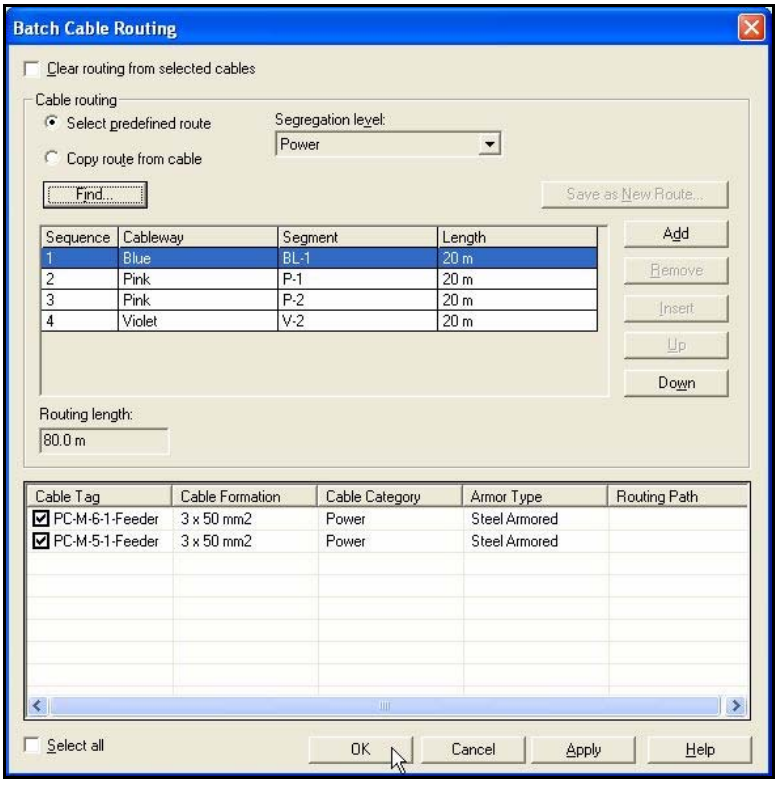
Notice that the cable actual length equals to the route:

Let us repeat the operation for the rest of the cables, as per the above table:
M-3 Feeder and M-4 Feeder to R-1:

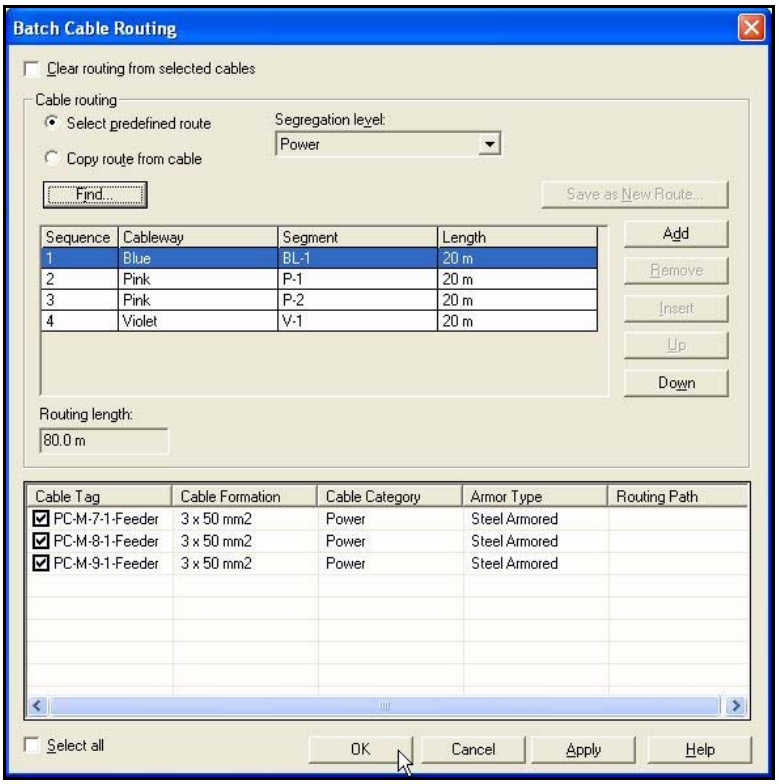




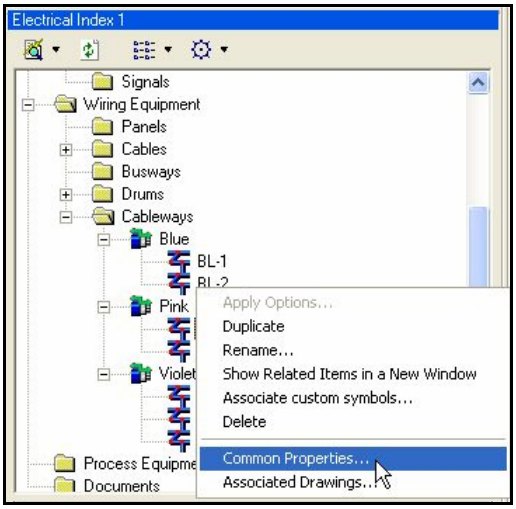
Click **OK** to route the cables.
M-5 and M-6 to R-3:



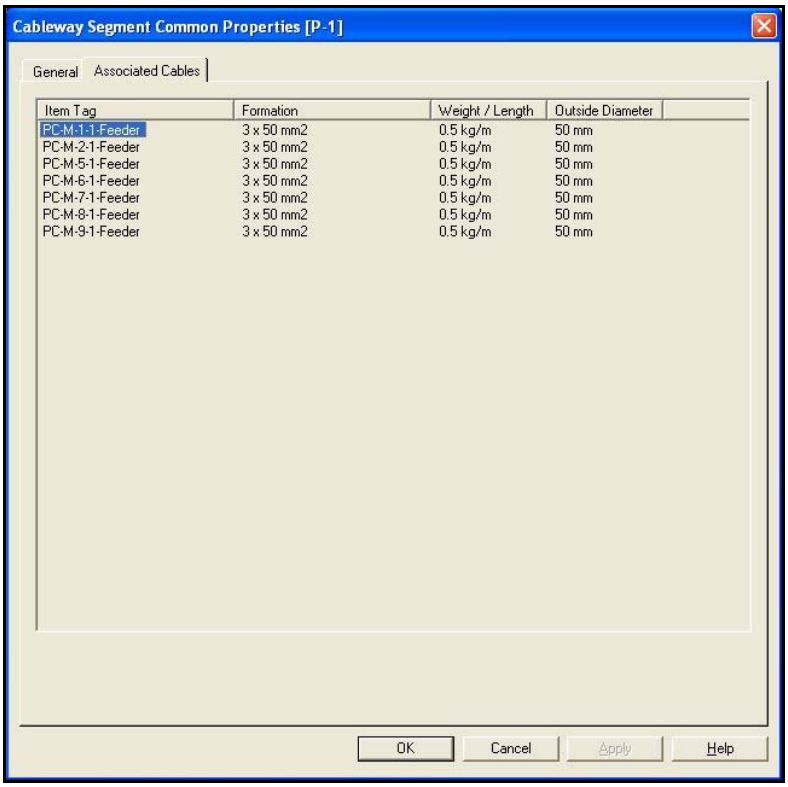
M-7 Feeder, M-8 Feeder and M-9 Feeder to R-4



In the **Cableways** folder, select segment P-1 and right-click **Common Properties**:



Select the **Associated Cables** tab and see that all the cables that have been routed via this segment:

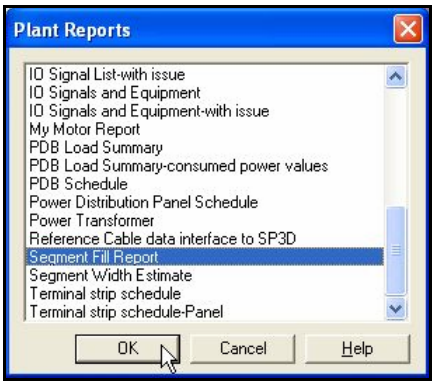


Step 4 - Generate a Segment Fill Report

Lets generate the segment fill report and see the loading of the segments.

1.In the **Electrical Index**, select the **Cable Ways** folder:
2.Click **Reports -> Plant Reports**.

3.Select the Segment Fill Report:



Segment Fill Report.xls

INTERGRAPH

Date:

Segment Fill Report

Cableway	Segment	Type	Size	Allowable area	Accumulated area	Accumulated outside dimeters	Number of layers	Allowable load	Accumulated load	Criteria	Overfill
Blue	BL-1	Tray	0.5	2500.0 cm^2	176.4 cm^2	450.0 mm	1	100.0 kg/m	4.5 kg/m	By Area	Ok
	BL-2	Tray	0.5	2500.0 cm^2	39.2 cm^2	100.0 mm	1	100.0 kg/m	1.0 kg/m	By Area	Ok
Pink	P-1	Tray	0.5	2500.0 cm^2	137.2 cm^2	350.0 mm	1	100.0 kg/m	3.5 kg/m	By Area	Ok
	P-2	Tray	0.5	2500.0 cm^2	98.0 cm^2	250.0 mm	1	100.0 kg/m	2.5 kg/m	By Area	Ok
Violet	V-1	Tray	0.5	2500.0 cm^2	58.8 cm^2	150.0 mm	1	100.0 kg/m	1.5 kg/m	By Area	Ok
	V-2	Tray	0.5	2500.0 cm^2	39.2 cm^2	100.0 mm	1	100.0 kg/m	1.0 kg/m	By Area	Ok
	V-3	Tray	0.5	2500.0 cm^2			1	100.0 kg/m		By Area	Ok

Sheet1 Sheet2

Cable Routing Validation Rules

The following validations are carried out while performing cable routing:
Segregation level of the cables and route must match.
Segment fill validation bases on the selection in the calculation criteria select list.

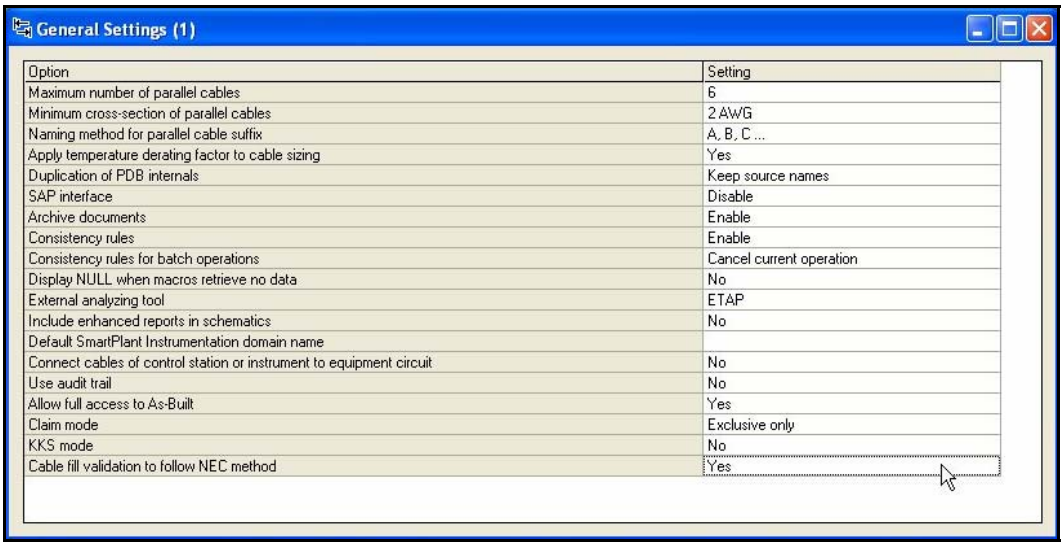
Auto This is the SPEL default. It bases on whether the tray is covered . If covered, the validation is based on the total available **area of the cross section of the tray**
If uncovered, the validation is based on the total available **width of the tray multiplied by the number of allowed layers.**

Area The validation bases on the total available **area of the cross section of the tray**

Width The validation bases on the total available **width of the tray multiplied by the number of allowed layers.**

NEC Cable Tray and conduit fill validation

Smart Plant Electrical allows you to change the tray fill and conduit validation criteria and follow NEC method.
To switch to NEC validation method, you should set in the option manager, in the General settings, the flag, 'Cable fill validation to follow NEC method' to Yes.



This will replicate the validation currently implemented in SP3D into SPEL and is based on NEC tray fill code and values in NEC tables.

Setting this parameter will be for the whole project and override any rule of the covered or uncovered, by area or width. The tray fill validation rules will be strictly according to the algorithm that follows the NEC 2005 Handbook.

The NEC rules for tray fill validation are based on data taken from Article 392.9 and 392.10 and the related tables that appear in these sections.

Basically, the rules are based on the following primary parameters and the various combinations of them:

- The voltage rating the cable is carrying (currently this is set to the insulation rated voltage). NEC divides the voltage to two groups, 2000v and below OR above
- Single core or multi core cables arrangement. The code doesn't address a mixture of these 2 categories of cables; therefore SPEL doesn't support such a mixture either.
- Is there a mixture of power and non power carrying cables (instrumentation, control, grounding)
- Type of tray on which cables are laid; whether it is ladder, Ventilated trough, Solid Bottom, Ventilated channel, solid channel
- Conductor sizes
- Tray dimensions

For each of the possible combinations of the above mentioned parameters NEC tabulates one of the following criteria:

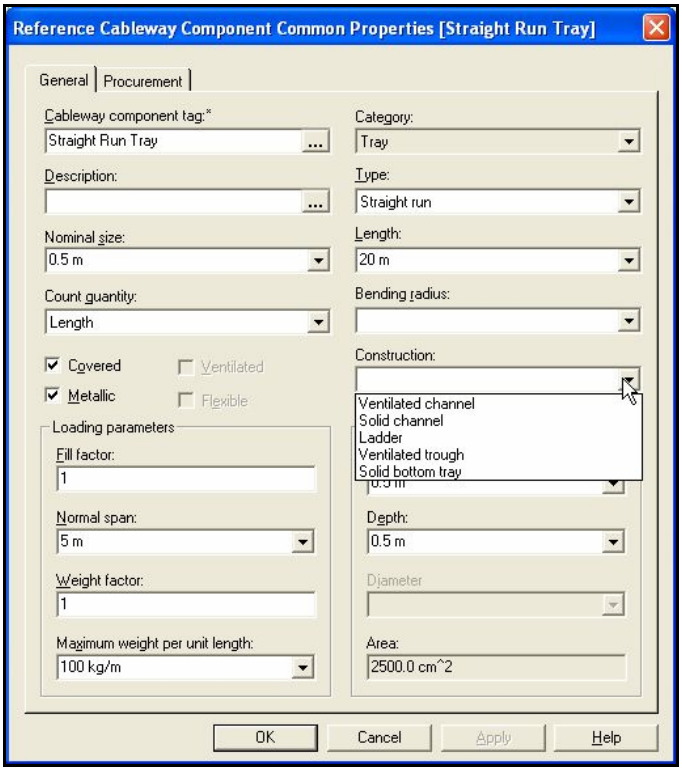
- The allowable number of layers OR
- The allowable Fill Area OR
- The allowable tray width percent

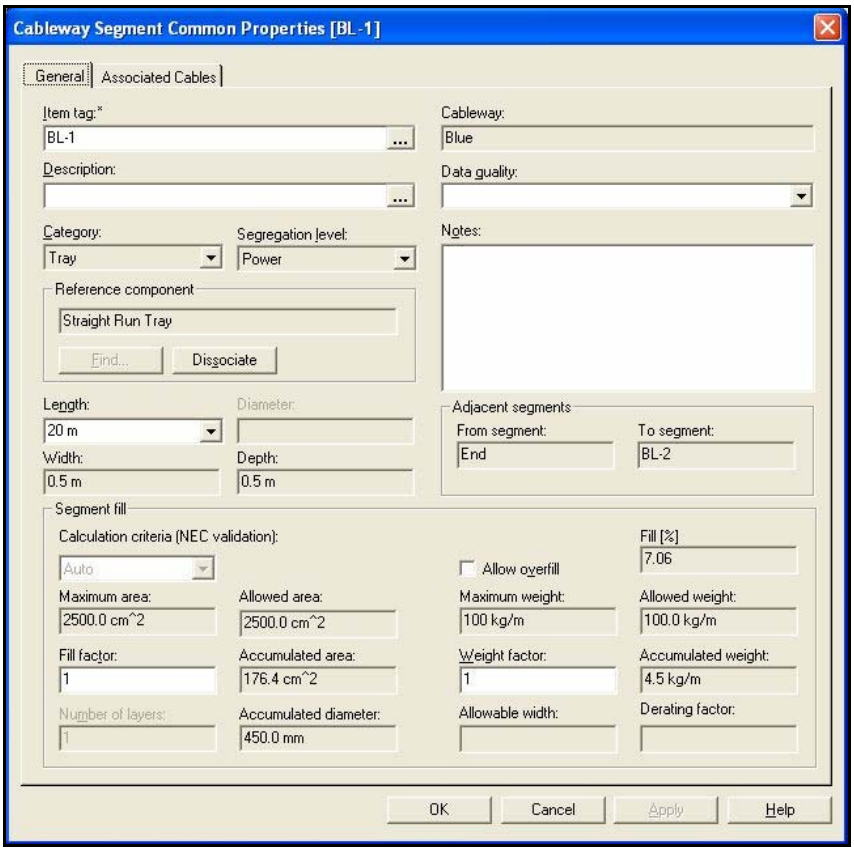
All the limiting numbers of the allowable width and cross sectional areas are in fixed hard coded in SPEL program.

Note: NEC fill %fill calculation occurs only if the segment is associated to a reference raceway component.

The demonstration of the NEC tray fill validation will be done by following the algorithm branches, each branch at a time, and for each, showing that SPEL complies with NEC 2005 code.

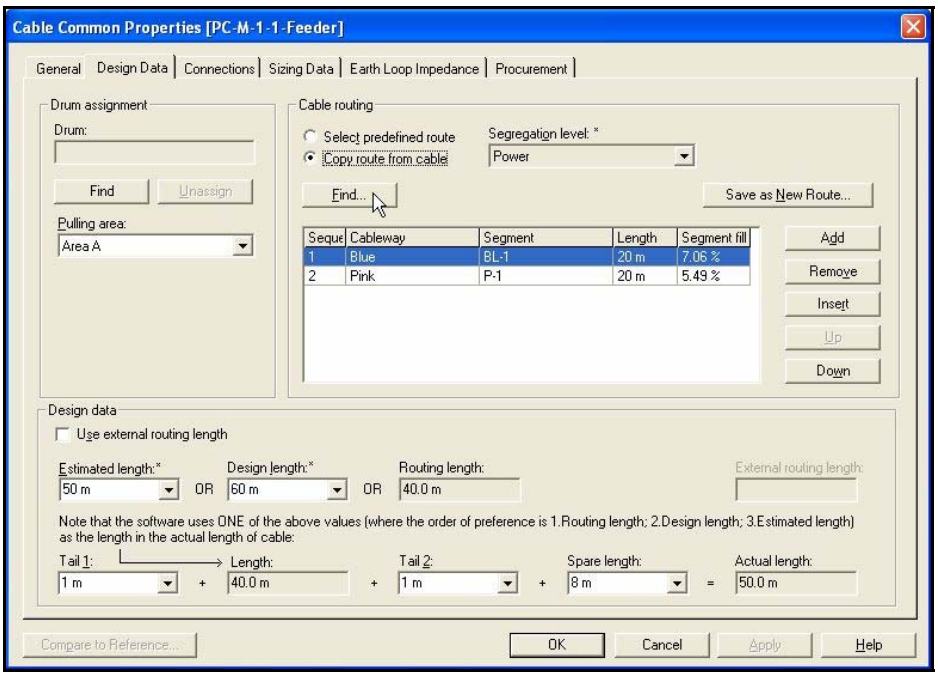
Whenever the Options Manager is set to NEC fill validation the raceway trays components Common Properties is slightly modified as shown:





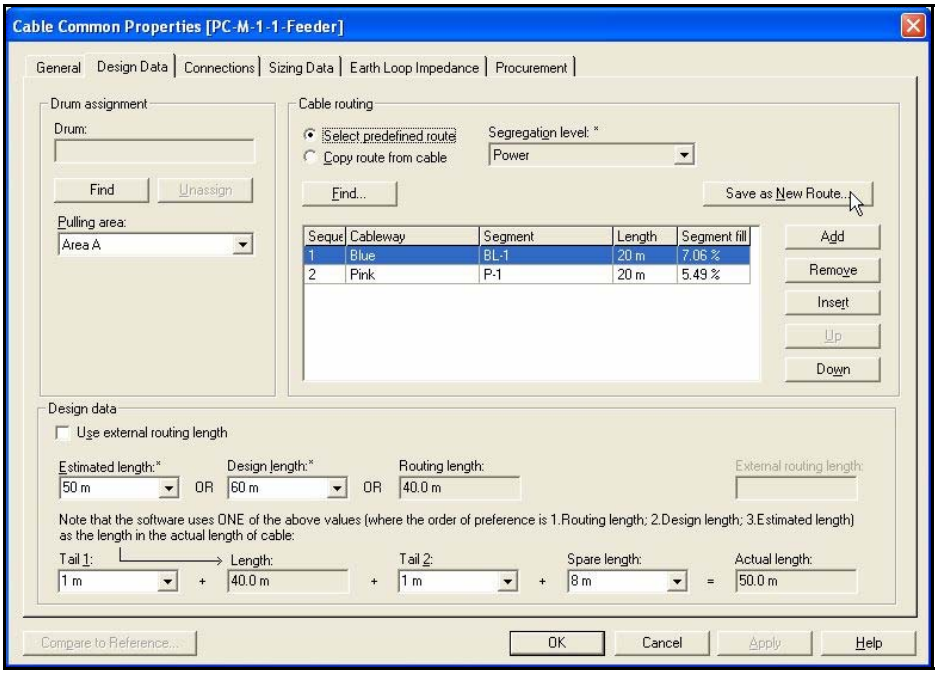
Copy Route from One Cable to Another

Smart Plant Electrical lets you copy route from one cable to another.



Save Routing of a Cable as a Predefined Route

Smart Plant Electrical lets you save a cable route as predefine roye,
Where you can use it later to route another cable.



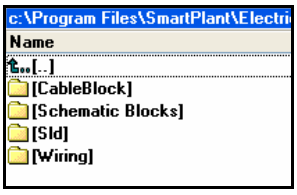
Cable Block diagrams

Cable block diagrams are important drawings that accompany the project and the end user. This document gives the user the overall picture of the general overview of the system interconnection policy as well as the details of which cables go to where. For the end user, the Owner Operator, this document is a reference for existing cabling and how to handle future new additions of cables.
With this feature users will be able to associate cables to equipment, to generate and document, on a graphical environment and layer, the interconnectivity of number of equipment by simple drag and drop of equipment operations and requesting the system to complete the interconnectivity

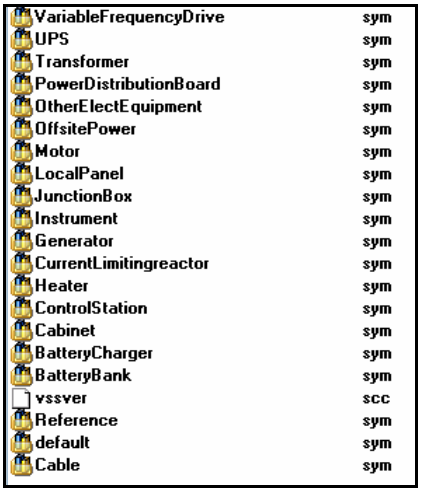
The path of the symbols for cable block diagrams is defined in Options Manager:

Default Cable Block Diagram Symbol Path	\\Mstein-t41p\SmartPlant\Electrical Reference Data\Symbols\CableBlock

This is the new shipped directory for the various types of symbols:

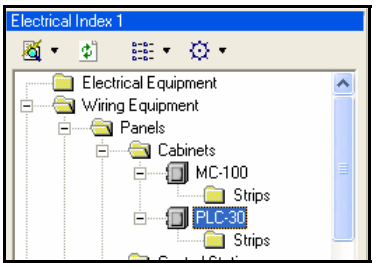


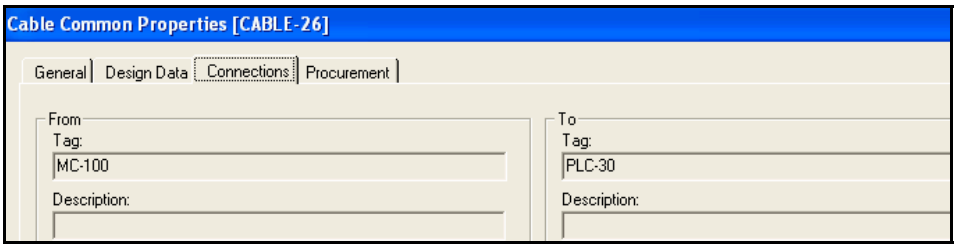
Within the CableBlock directory you will find the default shipped symbols.



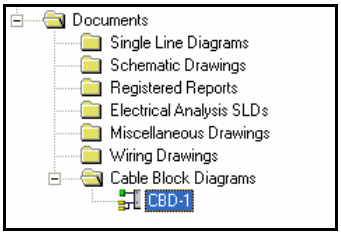
Note:

Refer to the Catalog manager about how to customize and create your own symbols for Cable Block diagrams.
We will first show the behavior of items that have cables already pre-assigned.
Create a control cable and associate its sides to both a Marshalling cabinet MC-100 and a PLC cabinet PLC-3

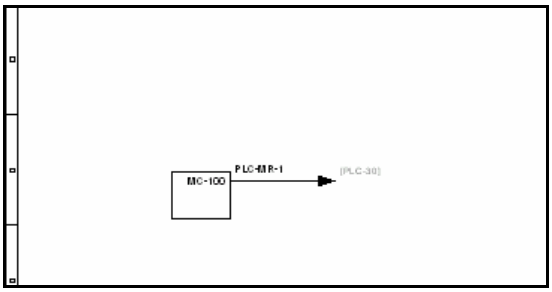




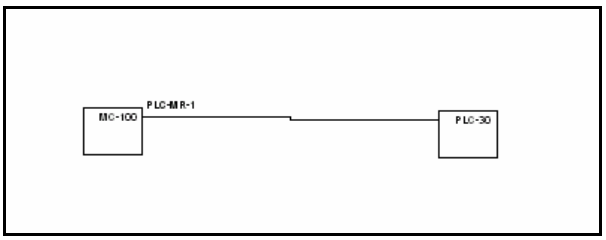
Open a new CBD



Right click to open the drawing and drag into it MC-100:

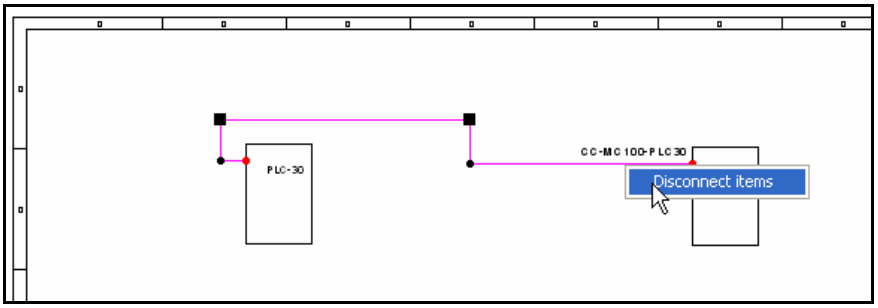


Observe that since a cable is already assigned the cabinet will be showing the relation.
Dragging the second cabinet will complete the interconnection diagram.

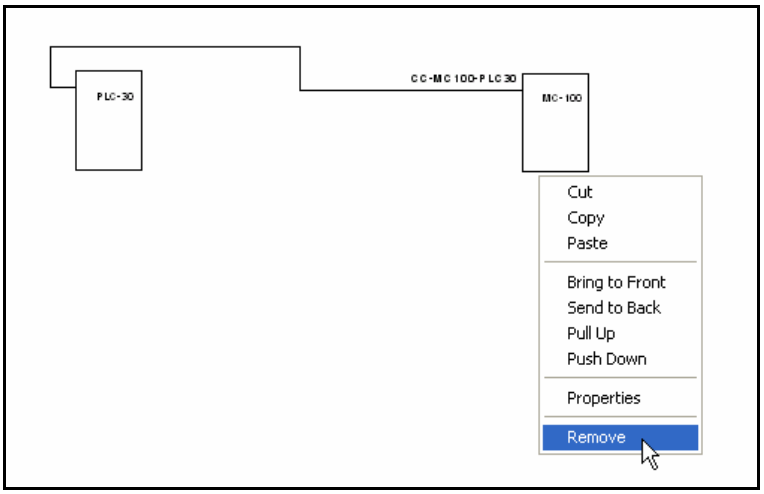


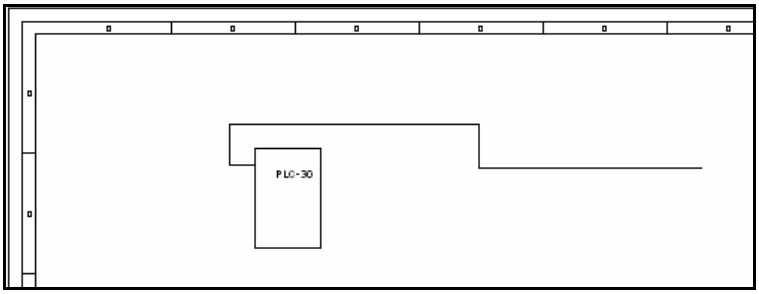
You may select and move the connectors (cables) to better position them. Saving the document will also save the new position.

You may disconnect the cable from the equipment by selecting the cable and right click:



Confirming this command results in disconnecting the equipment from both sides.
To remove equipment from being shown in the CBD:





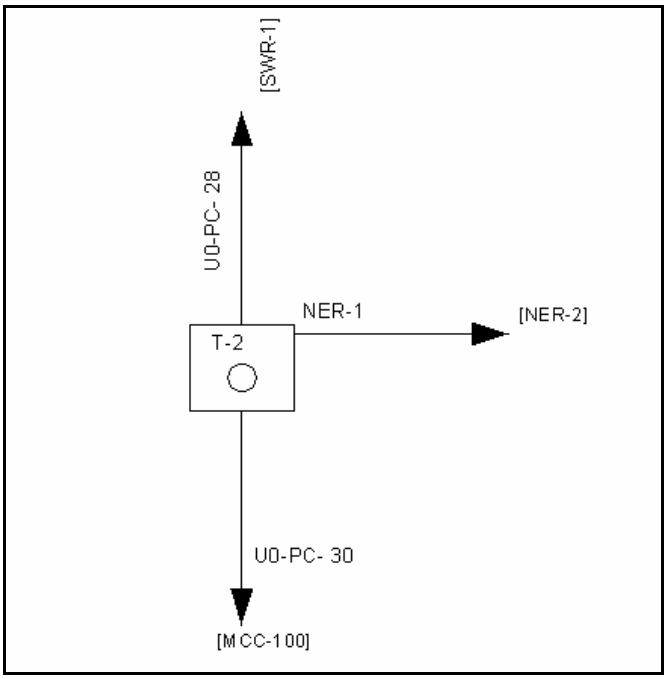
The system shows the "reference to" equipment on cables that have connections at their open sides. (in the case the equipment at the open side is not in the drawing)

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.

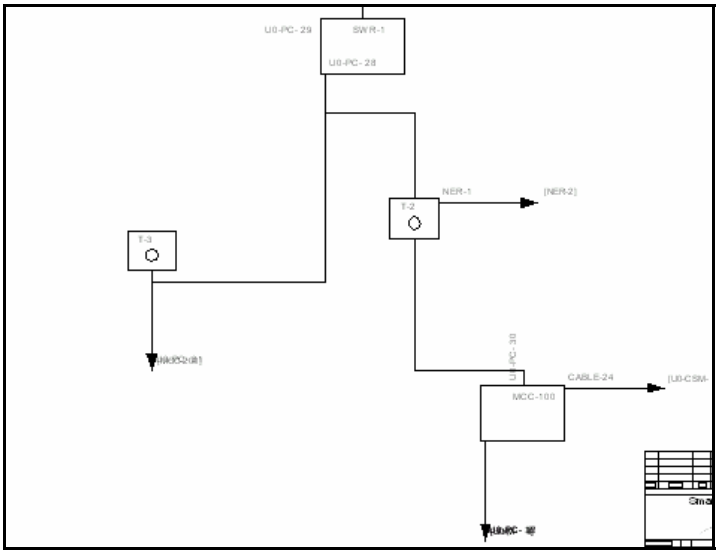
Showing power cables in CBD

Drag a transformer that has connections on the primary and secondary and also a power feeder that connects a motor with a circuit to see how these behave.

First, let us take the transformer and drag it to the drawing:



The system is designed so that the cable that connects to the primary gets connected to the upper side of the transformer symbol at the connection point defined as “Side2” (to side off the cable) pointing upward, and the transformer component gets connected to the side1 of the cable, pointing downwards.
When we now drag the PDB to the drawing, the cables will connect to the boundary of the PDB (there are no circuits granularity).



Some tips with regards to the points to which cables connect:

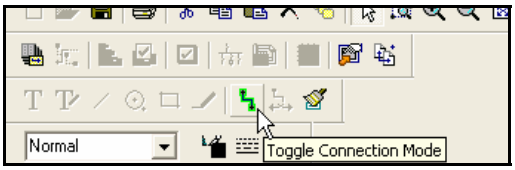
As you will see later, when learning about creating symbols for CBD's, there are 2 types of connection points that the system uses to connect cables to equipment. Standard "connection points" (used for power cable connection) and Non Power connection points (used for non power cables connections).
Each connection point has definition whether the point will accept the "from" side of the cable (side2) or the "to" side of the cable.
In the symbols that SPEL ships as defaults, the from side has been defined in the right side of the symbol and the cables connected to that side will be point out to the right.

Connecting cables to equipment in the drawing

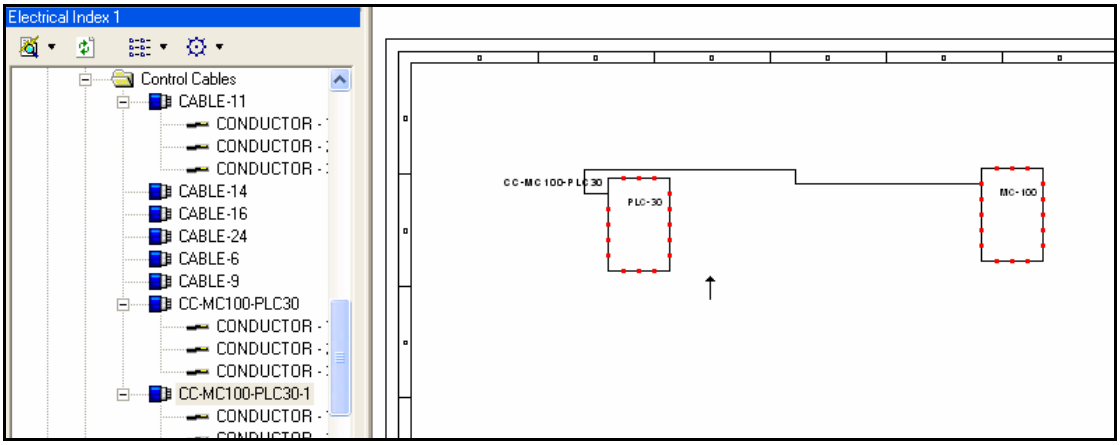
User can associate cable sides to equipment directly from the drawing environment.

With the same drawing open we will now connect a new control cable in between the 2 cabinets.

Create a new control cable. Select it and from the Menu---Edit-----Activate connection mode OR, "Toggle" the icon from the tool bar:

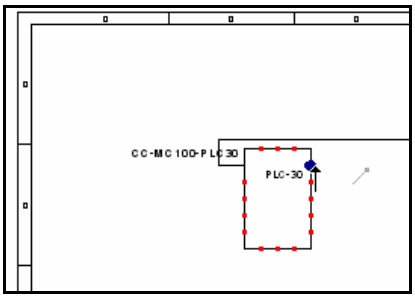


With the cable selected, move your cursor to the drawing and observe how the connection points of the equipment have turned highlighted.



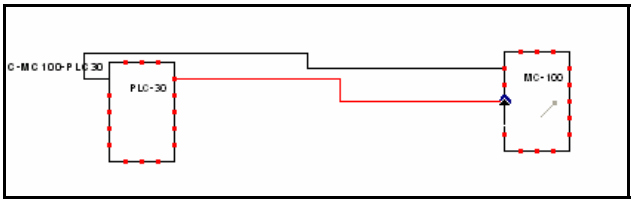
Select a valid connection point in the equipment you want to pre assign the cable to (selecting a connection point turns the red point to blue circle) :

Remember that the system is now connecting the cable to its "to" and "from" sides according to the selected connection points. It is recommended that you know the configuration of the connection points around the item symbol, where are the "to/from" sides, and where are the "power/non power" connections.

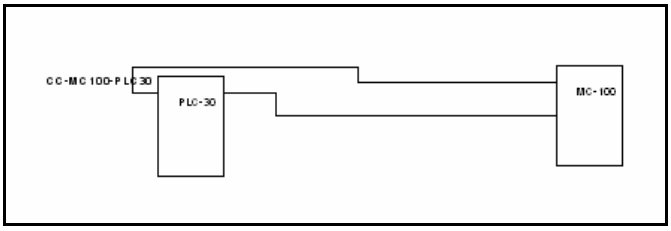


Click to select the side to connect:

Move the cursor to the second equipment and select a connection point:



This will result in completing the assignment of both cable sides



Note: There are 2 type of connection types used for CBD:

- Standard regular Connection point type, for power category cables
- Non power connection point type, for all the other cable category type

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 SPEL 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.

Cables that connect to PDB circuits

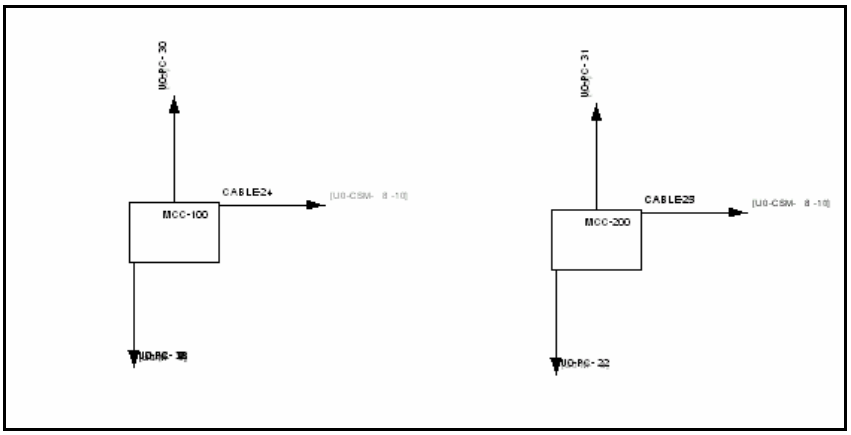
In CBD we do not show internal items of the PDB such as circuits. Since cables connect to circuits and not to PDB, when associating cables sides to PDB in the cable block diagrams, the user will then have to

associate the cables to the specific circuit using either the cable Common properties or the electrical engineer.

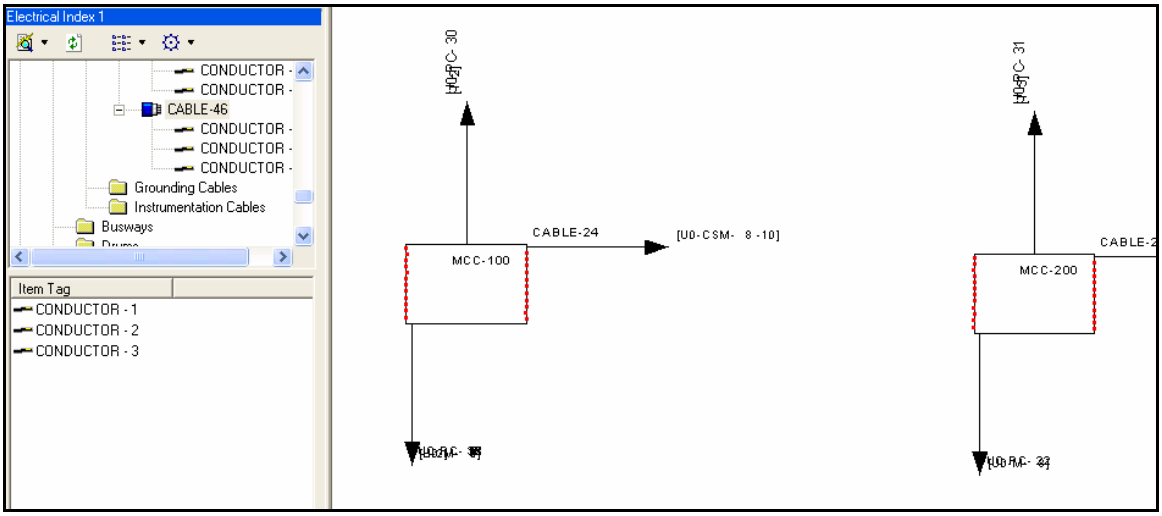
Let us connect a new control cable between two PDB circuit, a very common scenario.

Duplicate the control cable that we used in previous example.

Drag two PDB's to the drawing and zoom in:

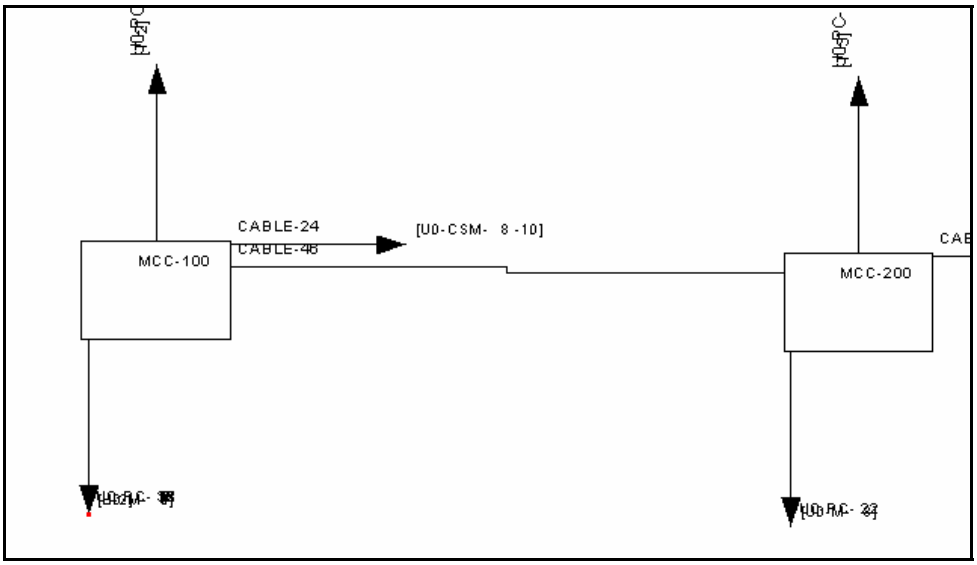


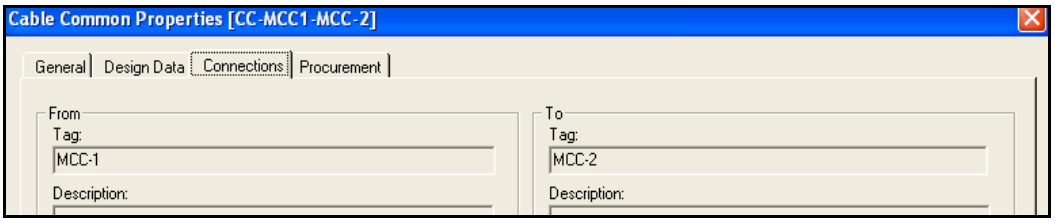
Select connection mode and the control cable to connect:



Select the cable side you want to connect and perform the connection at the other side of the cable (zoom in so you can identify the connection points):

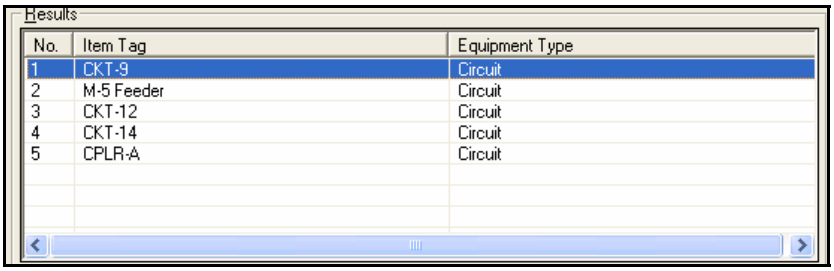
Select now this cable to see the assignment:



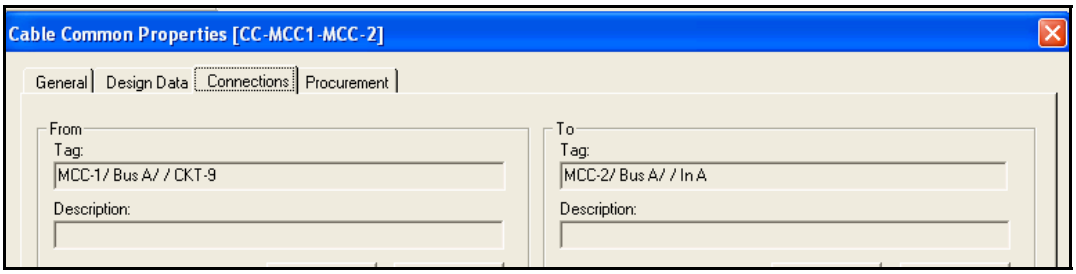


As you can see, the cable sides have been associated to the PDB only.

Clicking on the Find button and searching for the possible circuit will filter out and show only circuits available in the specific PDB:



After this association the complete path to the circuits will be shown:



Special notes:

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 similar to the following:

Wiring connection in SPEL

SPEL has the ability to produce wiring schematics based on typical templates, embedded with macros that retrieve data from the data base. Although this is fine for the graphical part, since there is no real termination data (cable, wires and terminals that connect to), there is a lack to produce dynamic cable termination reports as wiring diagrams.
The new functionality will provide for a graphical environment to support basic wiring and termination capabilities that will enable:

- Create terminal strips for all types of equipment, where applicable
- Create, within the terminal strips, terminals
- Connect and terminate cable wires between equipment
- 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

At this stage we shall deal with interconnections ***between equipment***, external to PDBs or the main equipment. (So, the internal circuit wiring will still be handled by the typical schematics, and the cable terminations will reach up to and including the circuit strips. No internal cabinet or equipment wiring is planned)

Workflow of the demonstration

The symbols for the special symbols used to present wiring are stored in the path defined in the Options Manager:

Default Wiring Diagram Symbol Path	\\Mstein-t41p\SmartPlant\Electrical Reference Data\Symbols\Wiring

This directory will contain the various symbols that the wiring engine expects to get.

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.

Note: 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.

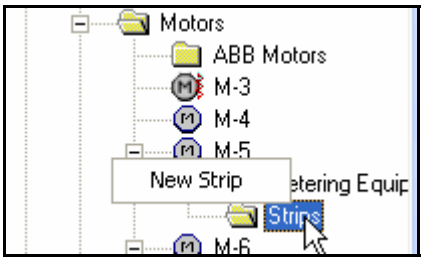
Creating terminals and terminal strips in equipment

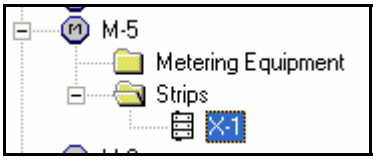
There are 2 ways to create terminals in SPEL.

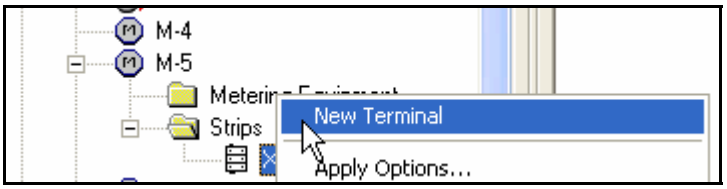
- Using typical RDE strip configurations
- Individual terminal creation

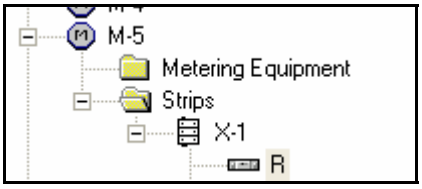
Individual terminals creation in Index

Let us create a terminal strip with 5 terminals on a motor

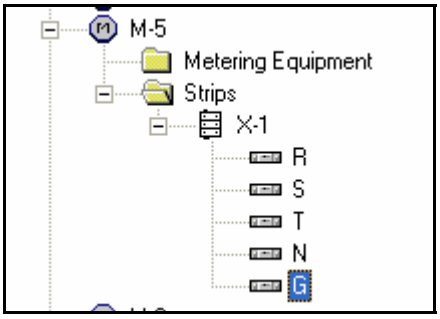






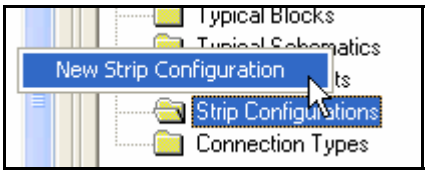


With the use of duplicate:

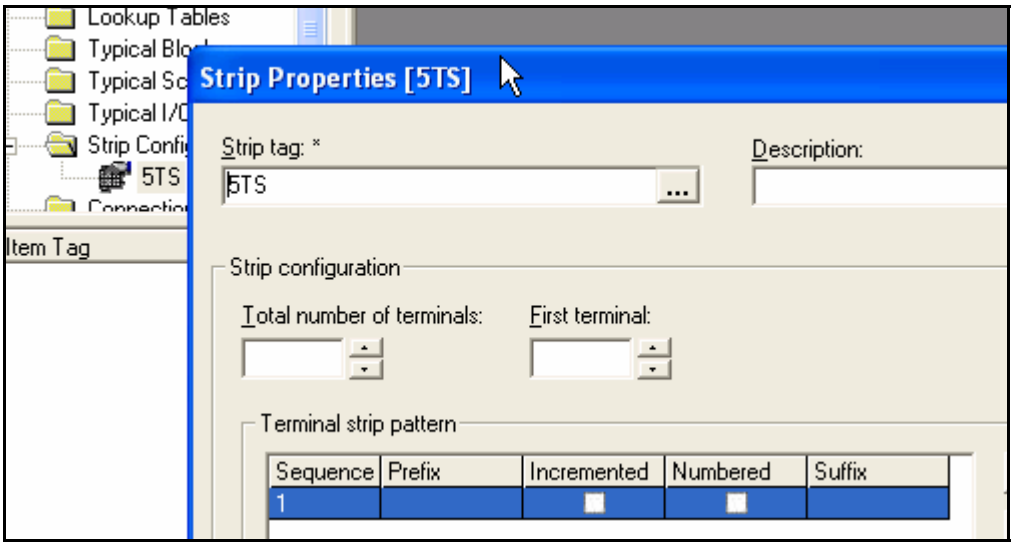


Using RDE typical templates

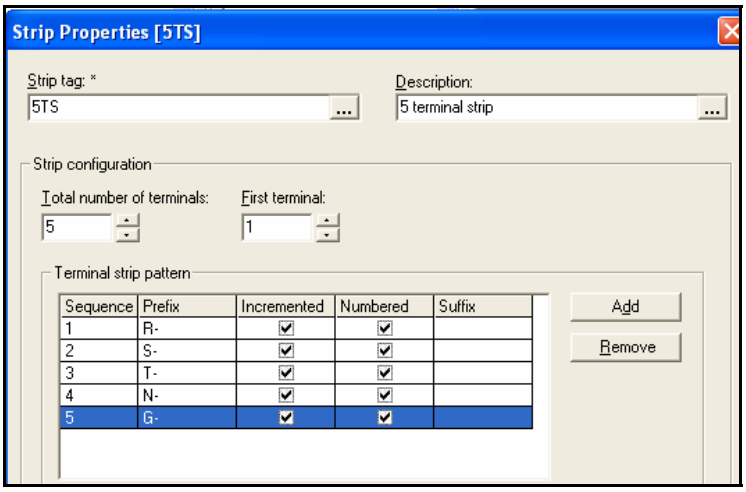
Define in RDE a typical wiring configuration



Let us create the same 5 terminals as a typical configuration.
Open the common properties of the configuration

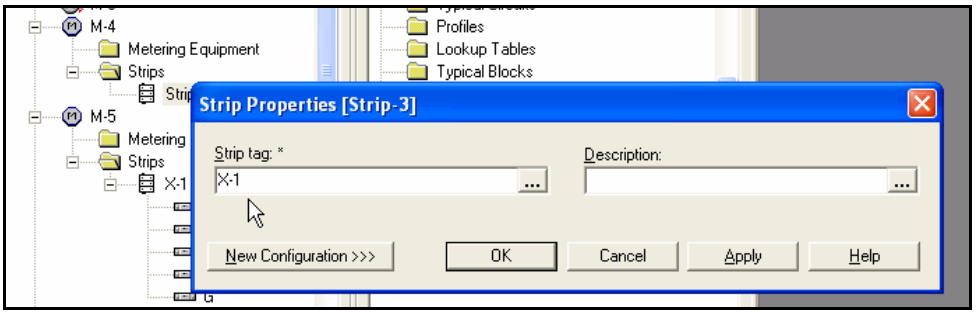


And enter data as follows:

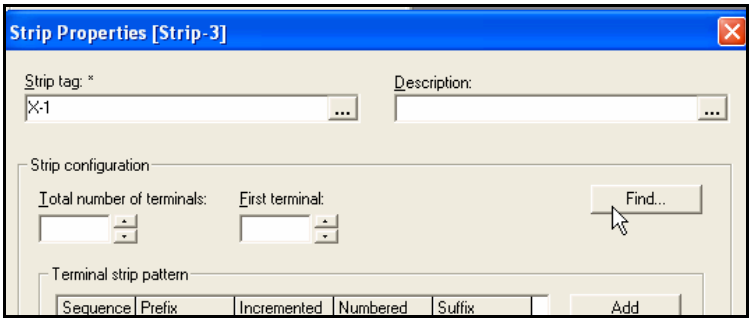


This is a typical terminal strip configuration and now we can use it to create project terminals.

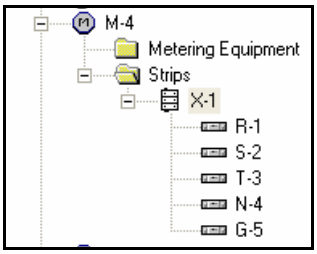
Create a new terminal strip under a motor:



And select the "Strip Configuration" button to initiate a Find for a reference configuration in the RDE:

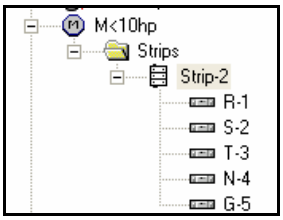


Select the 5TS and confirm



Note: You could have created a new configuration on the fly without saving it to the RDE

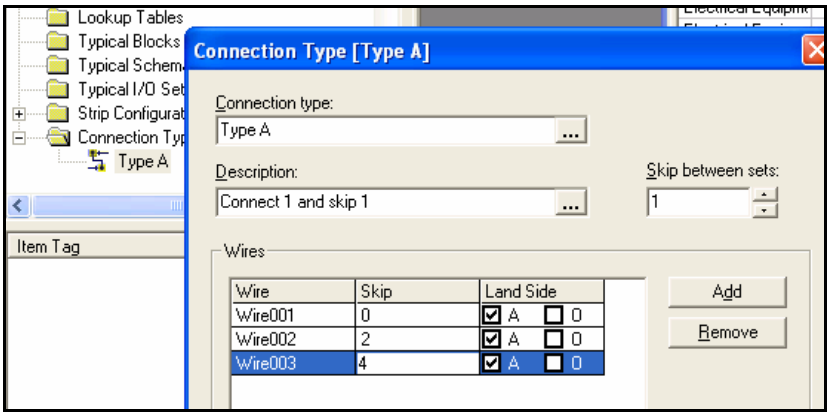
Another way of using the RDE is via reference main items associated to terminal strips:



And then simply drag it to the Index creating new items with terminals

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:

Let us create a Connection type that will result in connecting one and skip one terminal



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

Skip = Number of terminals that a wire should skip and connect to, terminals counted from the starting terminal user specifies/points when landing the cable on the strip. Skip 2 in the above configuration, for example means that 2nd wire will be connected to the 3rd terminal (based on user starting to connect the cable from first terminal of the strip)

Land side= The side of the terminal that the wire should connect to. Kind of left and right side of the terminal;

A=Active meaning is the actual side of the terminal on which the user is landing the cable.
O=Opposite side of the terminal on which the user is landing the cable.

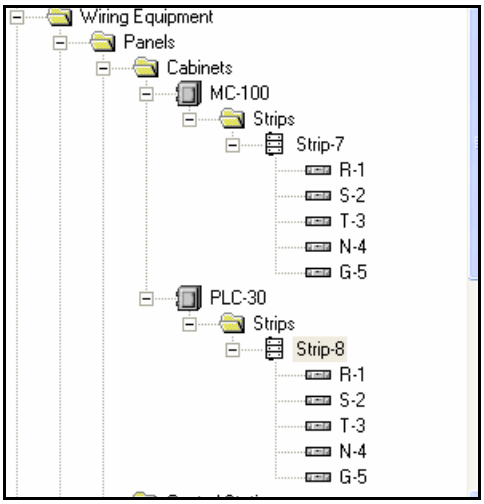
Note: There are no typical fully wired objects in RDE. No ability to define a completely wired assembly.

Wiring project items

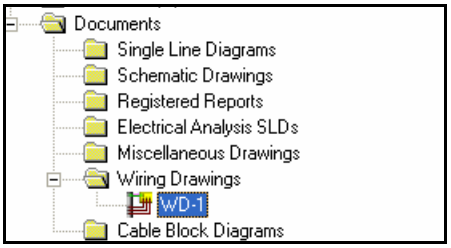
Since we can wire and terminate only those cables that have been already associated before we start this exercise we should make sure we have the objects we want to wire inter connected by cables.

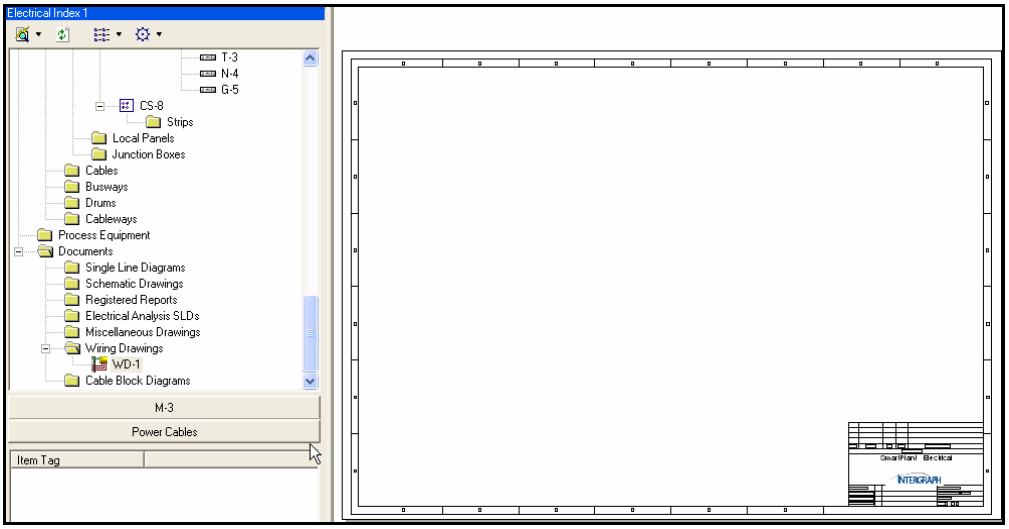
We will use the PLC-30 and MC-100 that we connected in previous session in the CBD

For these 2 items create strips:

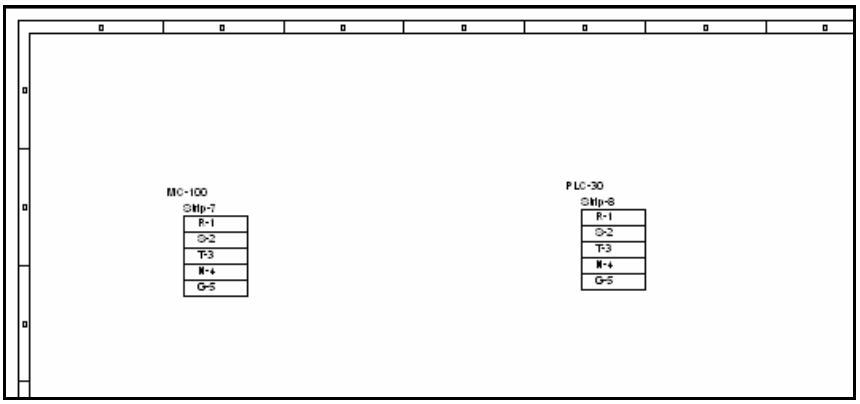


Open a new wiring diagram



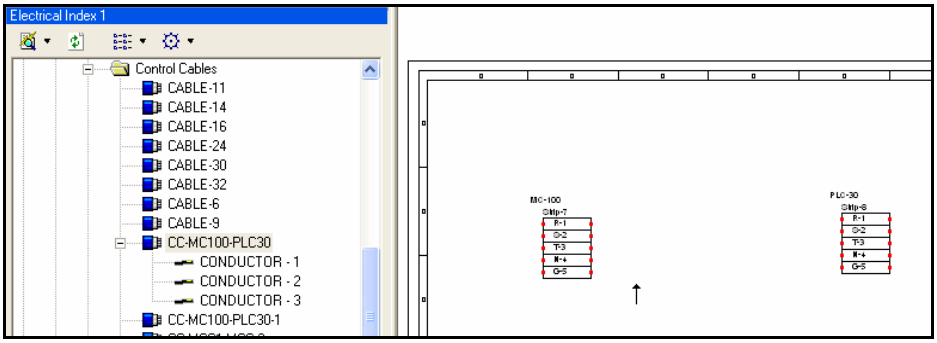


Drag the PLC-10 and MC-100 into the drawing

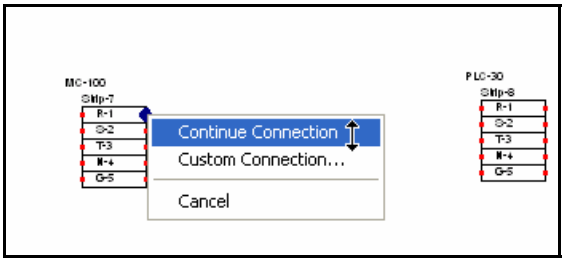


Toggle to the connection mode:

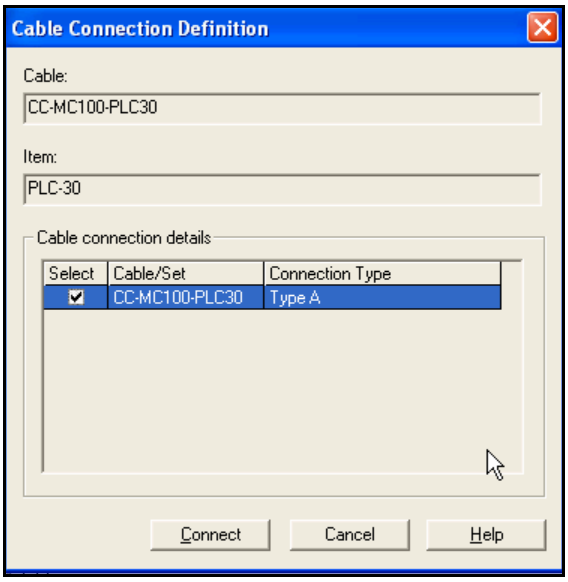
Select the cable that connects this equipment and move your cursor over the wiring diagram.



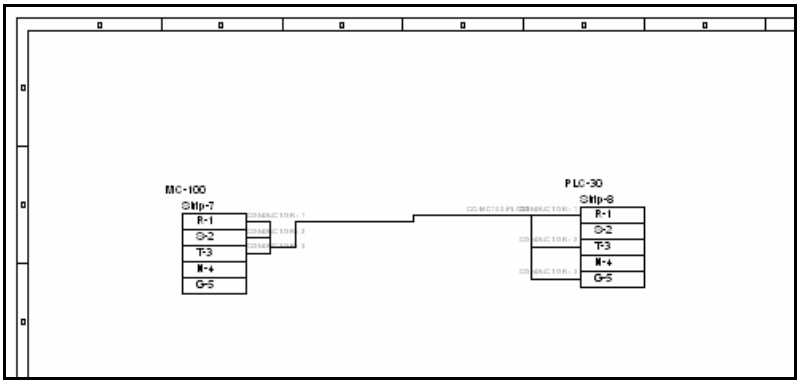
Select the first terminal of the left equipment strip, at its connection point at the right side and click on the blue circle.



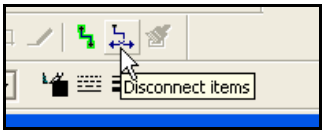
After connecting on the first terminal, drag the cursor with the cable over to the next equipment on the right side and land it on the first terminal. Select the “Custom connection”.



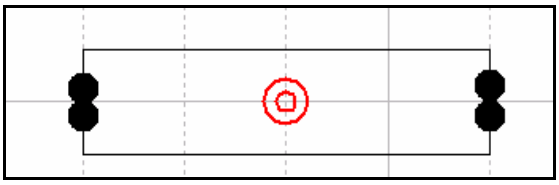
Click on the "Connect" button



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")



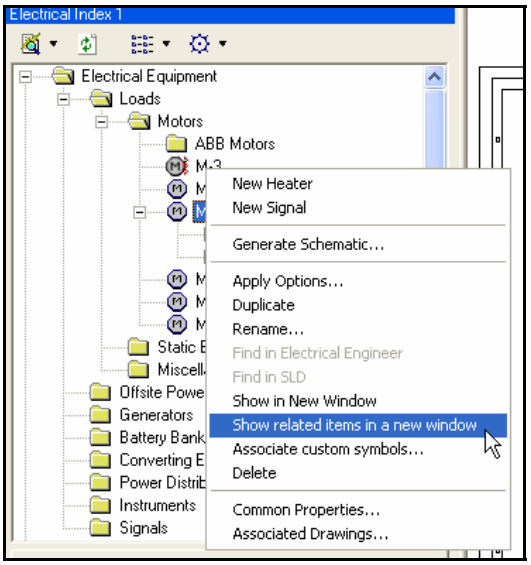
Note: The number of conductors that one can connect to a terminal depends on the number of connection points available on that terminal (as defined in the terminal symbol in catalog manager)
To connect 2 conductors to a terminal one needs to define the terminals having 2 connection points at each side, like this:



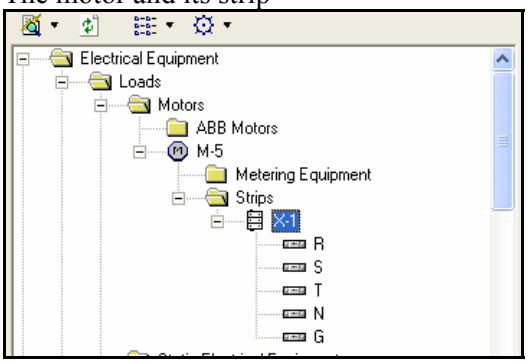
Viewing and grouping of related items

Many times we have to search for those items that are associated to each other, such as the motor, its control station and feeder circuits SPEL for wiring purposes.
In order to make our work more effective SPEL provides a new feature that allows viewing the items in a grouped way, saving us the time to search for them.

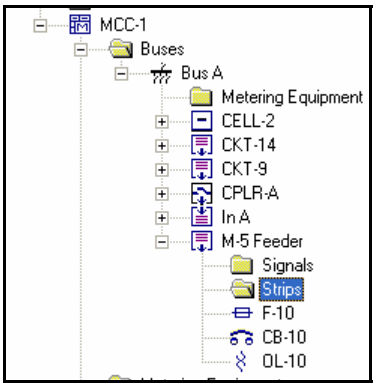
Let us select a motor and see how we can view on the fly its related items:



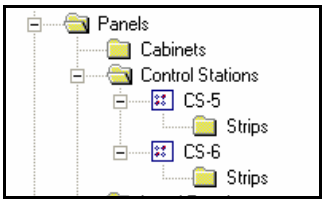
The result will be:
The motor and its strip



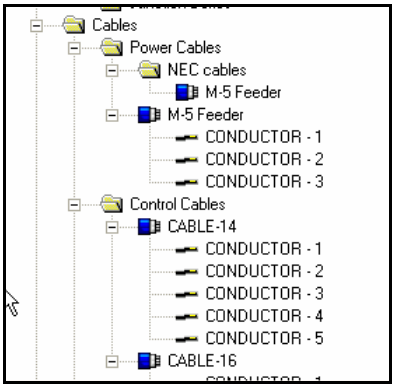
The feeder circuit



The control stations:



The feeder and control cables:



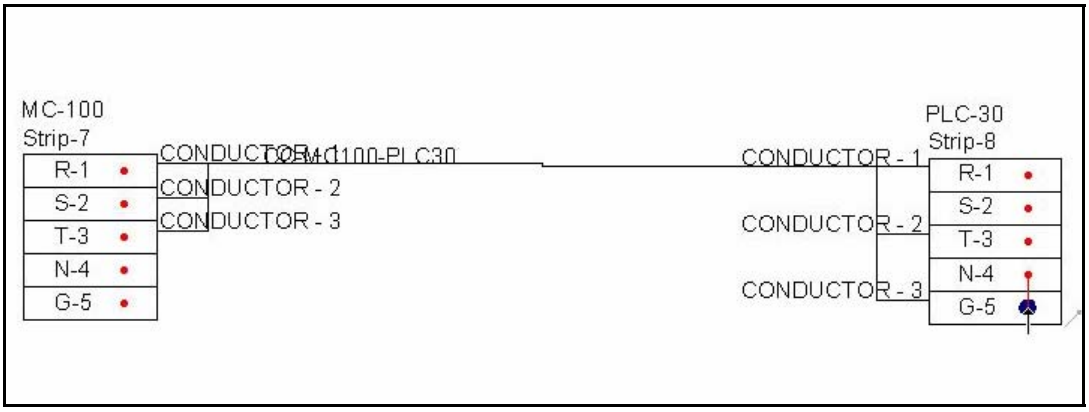
Users will now simply drag the items into the drawing.

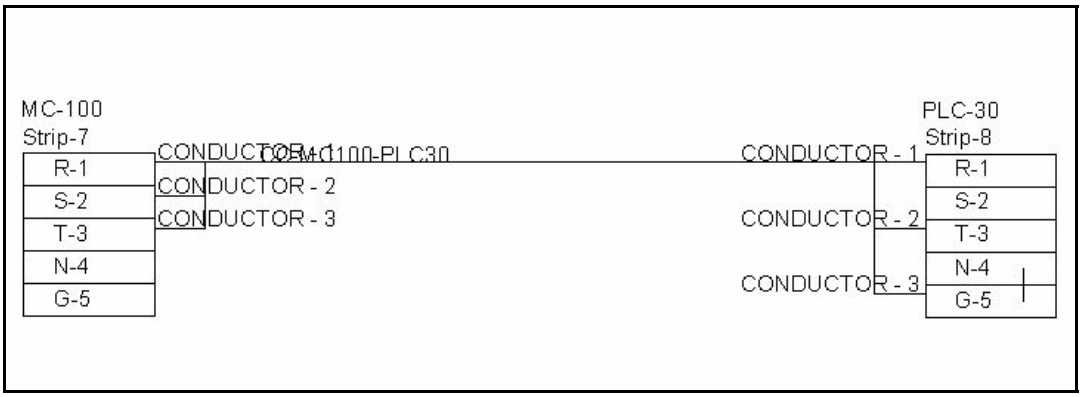
Jumpers

You can connect jumpers on terminal strips in a wiring diagram.

To add jumport, select activate jumper mode button.

Select the terminals you want to add the jumper

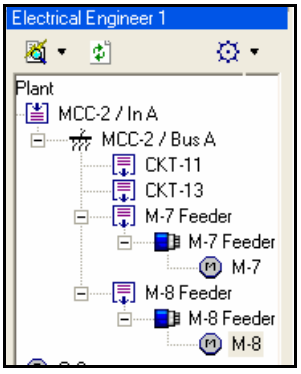




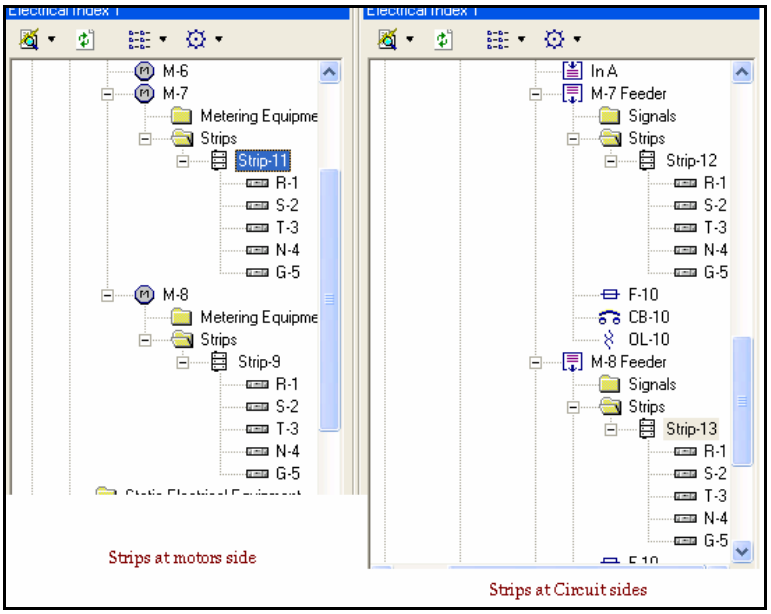
Batch cables sides' termination

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 10hp motors.

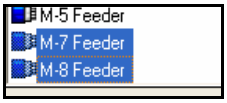
Make sure that you have few motors sharing the same connectivity. For this, you may create the same type of strips of 2 sets of motors connected to feeder circuits and cables connecting them to the circuits.



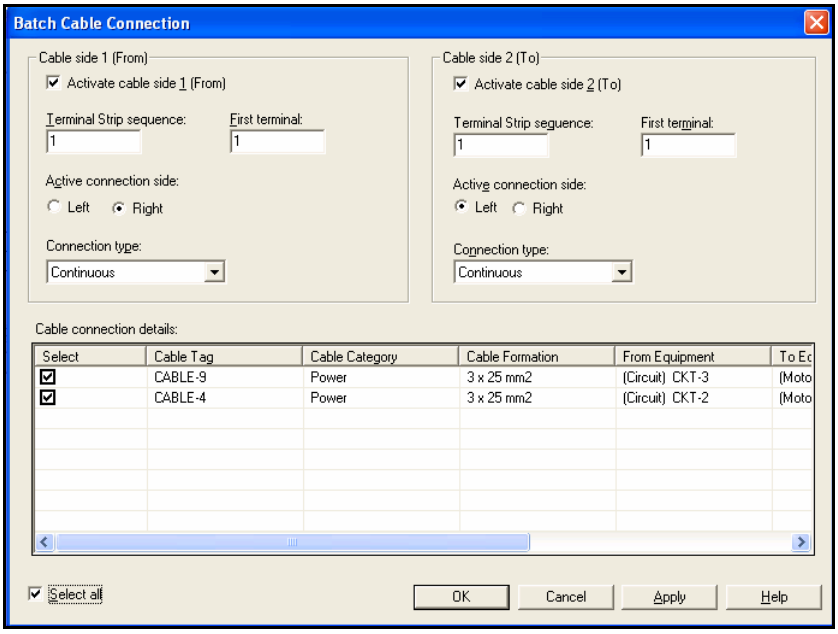
And their relevant strips at both sides:



Select the feeder cables of these 2 motors:

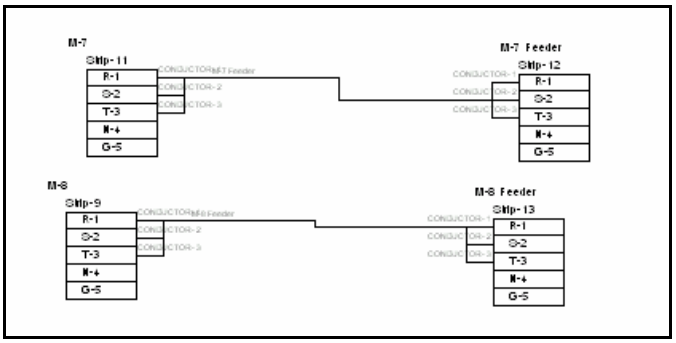


And from Menu--→Actions-→cables select the "Batch cable connection" to open the following UI:



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

Let us now see the actual connection made by the system. Either open a new wiring drawing or use the previous one.
Drag the 2 motors and the 2 corresponding circuits to the wiring drawing and see that they have been connected according to the specified strips and connection patterns:



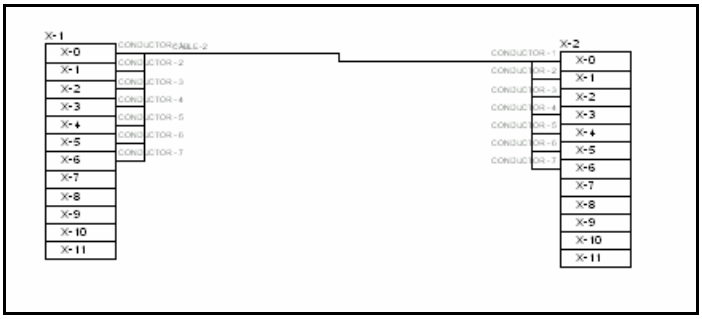
This is what you would to perform the wiring of a complete set of motors with their control stations and circuits and interface wiring to instrumentation.

Termination reports
Shipping reports

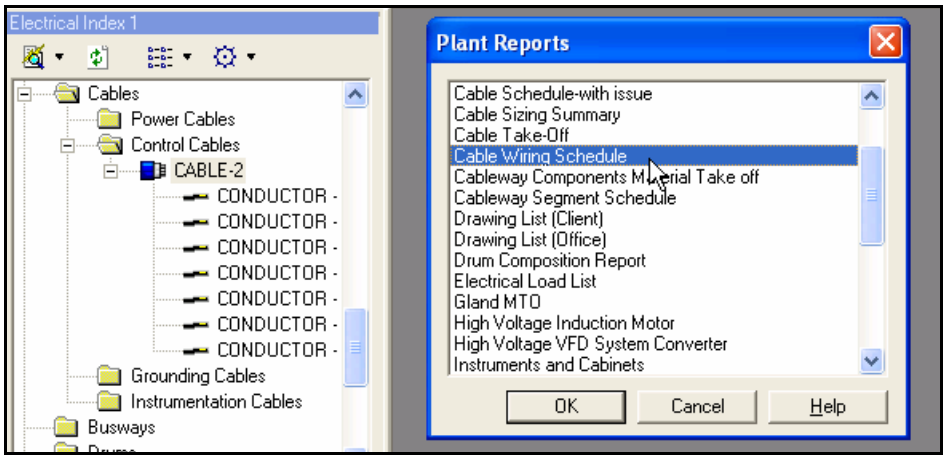
2 sample reports are shipped with SPEL: **"Cable Wiring schedule"** report template and **"Terminal strip connection"** report template

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.
For a cable that is connected between 2 strips of equipment



Selecting the cable



and running the report will generate the following:

Cable Tag: CABLE-2
Description:
Specification: Control
Formation: 7 x 1.5 mm2

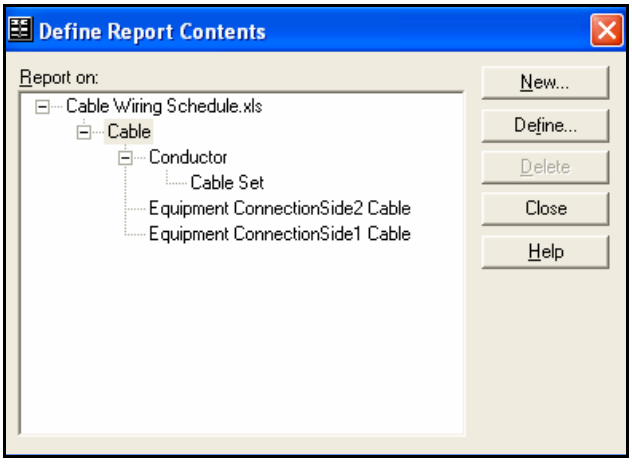
INTERGRAPH

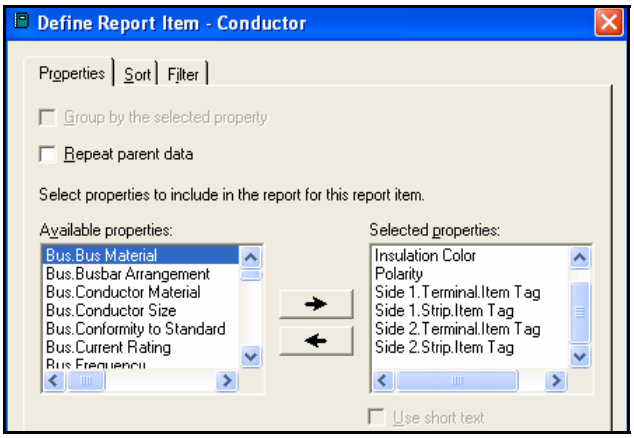
Revision:
By:
Date:

Cable Wiring Schedule

	From			Conductor				To		
	Equipment Item Tag	Strip Name	Terminal Name	Name	Set	Color	Polarity	Terminal Name	Strip Name	Equipment Item
9	Cab-2	X-1	X-0	CONDUCTOR - 1		Black	+ Positive	X-0	X-2	Cab-3
10		X-1	X-1	CONDUCTOR - 2		White	- Negative	X-1	X-2	
11		X-1	X-2	CONDUCTOR - 3		Blue	- Negative	X-2	X-2	
12		X-1	X-3	CONDUCTOR - 4		Yellow	Shield	X-3	X-2	
13		X-1	X-4	CONDUCTOR - 5		Yellow	None	X-4	X-2	
14		X-1	X-5	CONDUCTOR - 6		Red	+ Positive	X-5	X-2	
15		X-1	X-6	CONDUCTOR - 7			- Negative	X-6	X-2	

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.

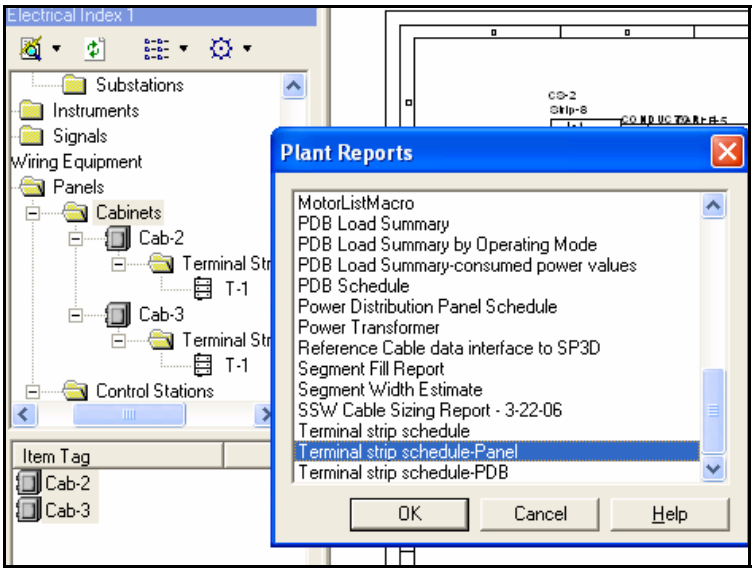
Terminal strip connections report

SPEL provides 3 shipped reports supporting identification of terminal strips connectivity data:

Terminal strips schedule- Equipment
Terminal strips schedule- Panel

These reports provide the wiring information from the connected equipment perspective.
It lists, per equipment, all the terminal strips and tabulates their wiring information.

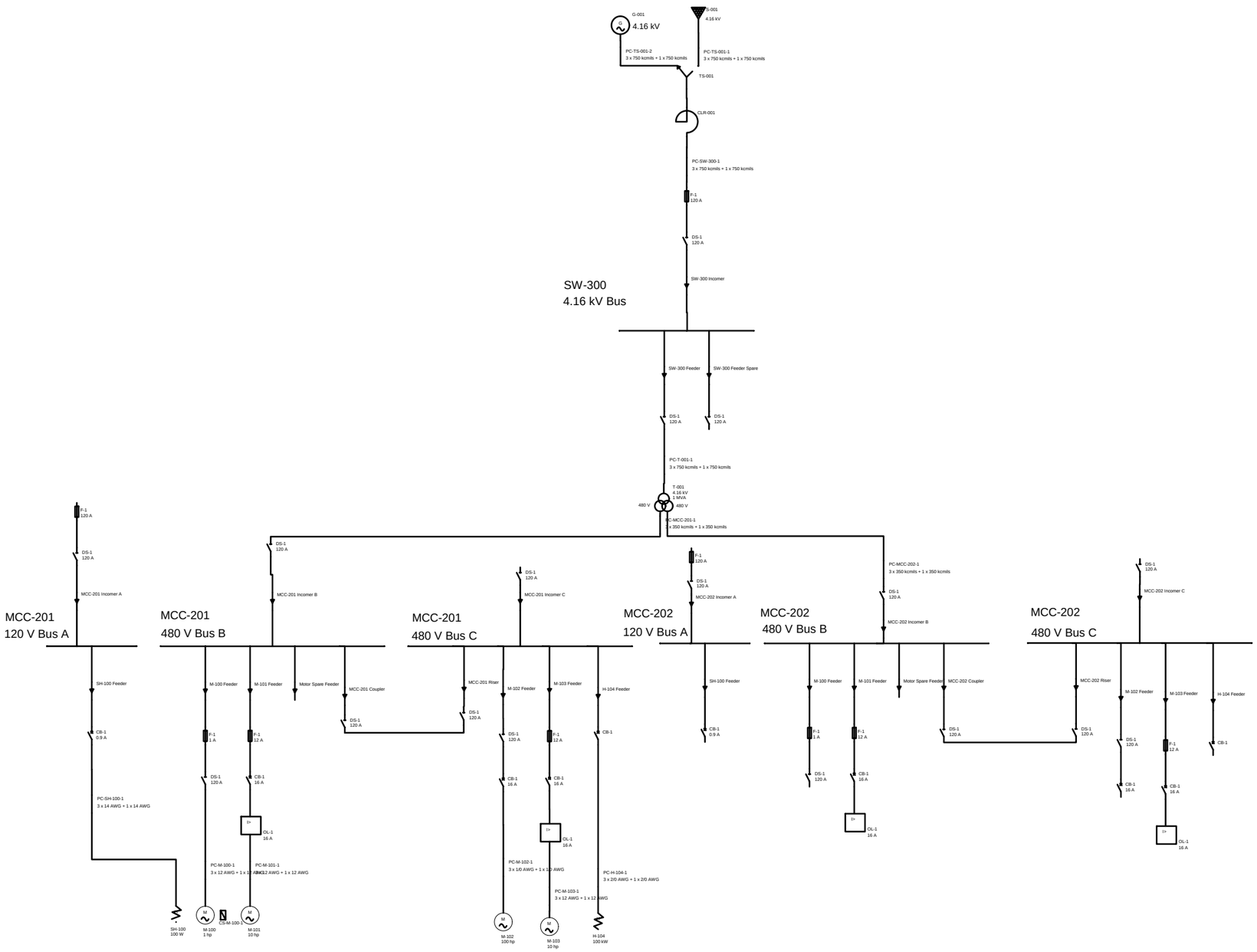
So, selecting the equipment, power distribution board or just terminal strips (cabinets in our above example):



Note: Additional conductors connected to the same terminal will be presented as additional lines with the same terminal number, one row per conductor connected.

Document Revision History

- Rev. 0. Initial Release of SmartPlant Electrical version 1.0.0.0. Basic Training Doc. #DELE2-TB-100001A (6/18/02)
- Rev. 1 match to release 1.0.3.8 (8/06/02)
- Rev. 2 match to release 1.0.4.5
- Rev. 3 match to release 01.05.00.08 (4/14/03)
- Rev. 4 structure change and match to release 02.00.00.00 (09/03/03)
- Rev. 5 Updates and enhancements to match version 03.00 (August 2004)
- Rev.6 Updated for version 03.05.00.26 (September 2005)
- Rev. 7 Updated for version 03.05.00.26 (March 2005)
- Rev. 8 version 03.05.00.26 (July-20-2006)
- Rev. 9 Version 04.00.01.07 (April-24-2007)
- Rev. 10 Version 04.01.02.10 (September-10-2007)



REV	DATE	BY	CHK	DESCRIPTION
P0	4/23/2007	Administrator		

REVISIONS

SmartPlant Electrical

INTERGRAPH

DRAWN BY:		PROJECT NO:
APPROVED BY:		DRAWING NO: S-001 Demo SLD
APPROVED BY:		Last Rev: P0
APPROVED BY:		Page1 of 1