

Training manual

Copyright

Copyright (c) July 2013 IGE-XAO. All rights reserved. No part of this manual, or any portion of it, shall be reproduced, transcribed, saved or translated, under whatever form and by whatever means, without written authorization from IGE-XAO, 25 bld Victor Hugo, Immeuble Le Pythagore, BP 90312 31 773 COLOMIERS CEDEX FRANCE.

Table of Contents

A	INTRODUCTION	11
A.1.	THE WORKSPACE	12
A.2.	FOLDERS AND FILES	15
B	CREATING A NEW WORKSPACE	16
C	DRAWING THE FIRST PAGE OF A CIRCUIT DIAGRAM	18
C.1.	CREATING PAGE 1	18
C.2.	GRID	19
C.3.	DRAWING PAGE 1	20
C.4.	WORKING WITH A ZOOMED PART OF THE DRAWING	26
D	DRAWING THE SECOND PAGE OF A CIRCUIT DIAGRAM	27
D.1.	CREATING PAGE 2	27
D.2.	DRAWING PAGE 2	28
D.3.	COPYING EXISTING ELEMENTS ON PAGE 2	37
E	DRAWING CABLES	45
F	ADDITIONAL PROCESSING OF CIRCUIT DIAGRAMS	48
F.1.	USING CROSS-REFERENCE SYMBOLS	50
F.2.	USING INFO TEXT SYMBOLS	52
F.3.	PAGE INDEX	53
F.4.	TEXTS	55
G	HYPERLINKS	58
G.1.	MANAGING THE HYPERLINKS	58
H	REDLINING	59
I	PRINTING	61
I.1.	PRINT	61
J	DATABASE LISTS	63
K	GRAPHICAL LISTS	66
K.1.	GENERATING A GRAPHICAL LIST	66
K.2.	DEFINE PAGE TEXTS PERMANENTLY	68
K.3.	GENERATING ALL DESIRED GRAPHICAL LISTS IN ONE STEP	68
L	INSERTING OTHER DOCUMENTS	69
M	CREATING COMPONENTS	70
M.1.	CREATING A COMPONENT	70
M.2.	CHANGING EXISTING SYMBOLS	75
M.2.1.	Deleting Elements	75
M.2.2.	Adding Elements	76
M.2.3.	Adding Texts	76
M.2.4.	Moving Texts	78
M.3.	HANDLING GRAPHICS	81
M.4.	USING AVAILABLE COMPONENTS	83

M.5. INFORMATION ABOUT SPECIAL COMPONENTS AND THEIR CREATION	85
M.5.1. <i>Contacts</i>	85
M.5.2. <i>Terminals</i>	85
M.5.3. <i>Multi-Layer Terminals (standard)</i>	86
M.5.4. <i>Connectors</i>	87
M.5.5. <i>Cross-Reference Symbols</i>	88
M.5.6. <i>Info Text Symbols</i>	88
M.5.7. <i>Components with Auxiliary Contacts</i>	89
M.5.7.a. Symbols with cross reference as text	89
M.5.7.b. Symbols with graphical contact symbols in it	90
M.5.7.c. Examples of Components with Auxiliary Contacts	90
M.5.8. <i>PLC Components</i>	91
M.5.8.a. Examples of PLC Components	92
M.5.9. <i>Smart black box symbols</i>	95
M.6. CABLE SYMBOLS	96
M.6.1. <i>Creating User-Defined Cable Symbols</i>	96
M.6.2. <i>Creating User-Defined Cables</i>	97
M.6.3. <i>Settings for Cables</i>	99
M.7. INFORMATION ABOUT COMPONENT INSERTION	100
M.7.1.a. Relay Coils	100
M.7.1.b. Terminals	100
M.7.1.c. Assigning a Component Name	100
M.8. INFORMATION ABOUT SYMBOL DATABASES	101
M.8.1. <i>Creating a New Database</i>	101
M.8.2. <i>Directories and Names</i>	101
M.8.3. <i>Working with Symbol Folders</i>	101
M.8.4. <i>Working with Symbols in the Database</i>	101
M.8.5. <i>Favorites Symbol Folder</i>	102
M.8.6. <i>Symbol Database Connection to Modules</i>	102
N GROUPS	104
N.1. CREATING A GROUP	104
N.2. UNGROUPING SELECTED ELEMENTS	107
N.3. MOTOR TERMINALS AND SIMILAR COMPONENTS	107
N.4. CONNECTORS: USING SEVERAL PLUGS AND JACKS TOGETHER	108
O BASIC OPERATIONS WITH TEMPLATES AND STANDARD SHEETS	109
O.1. CREATING A STANDARD SHEET	110
O.2. CREATING A PAGE TEMPLATE	118
O.3. CREATING A WORKSPACE TEMPLATE	120
O.4. QUICKREFERENCE TEMPLATES AND STANDARD SHEETS	122
P TEMPLATES AND STANDARD SHEETS FOR GRAPHICAL LISTS	123
P.1. CREATING TEMPLATES FOR GRAPHICAL LISTS	123
P.2. FORMATTING TEMPLATES FOR GRAPHICAL LISTS	125
P.3. GENERATE COVER SHEETS FOR GRAPHICAL LISTS	132
Q EXTERNAL DATA TRANSFER	133
Q.1. DATA TRANSFER THROUGH DWG/DXF/DXB FORMAT	133
Q.2. PIXEL IMAGES TRANSFER	134

R	PROCESSING A WORKSPACE	135
R.1.	ADDITIONAL OPTIONS FOR CROSS-REFERENCES	135
R.2.	PLC FUNCTIONALITIES IN THE STANDARD LEVEL	136
R.2.1.	<i>Using the PLC database</i>	136
R.3.	ADDING AND DELETING PAGES	137
R.4.	COPY SINGLE PAGE IN THE SAME WORKSPACE	137
S	EASY EDITING IN DATABASE LISTS	138
T	CONTACT MIRROR AND TYPE DATABASE	139
T.1.	MANIPULATING THE TYPE DATABASE	141
T.2.	USING TYPES IN THE CIRCUIT DIAGRAMS	147
T.3.	AUXILIARY CONTACTS INSIDE ADD-ONS FOR COILS AND SUBTYPES	150
U	ADDITIONAL HINTS ABOUT COMPONENT MANAGEMENT IN THE TYPE DATABASE	152
U.1.	CABLE MANAGEMENT	152
U.2.	MANAGEMENT OF CONNECTORS	155
U.3.	CHANNEL DEFINITION FOR PLC I/Os	157
U.4.	BLACK BOXES	159
U.4.1.	<i>Assigning of types to intelligent black box symbols</i>	159
U.4.2.	<i>Creating a Black Box from Channel Information</i>	159
V	ADDITIONAL HINTS ABOUT THE TYPE DATABASE	161
V.1.	SEARCHING AND COPYING IN THE TYPE DATABASE	161
V.2.	COMPLETING COMPONENTS	162
V.3.	USING THE COMPONENT EXPLORER	164
V.4.	IMPORTING AND EXPORTING ARTICLE MASTER DATA	166
V.5.	MODIFYING EXISTING TYPES IN THE PROJECT	167
V.6.	DISPLAYING INFORMATION ON THE EQUIPMENT FROM THE TYPE DATABASE	167
V.7.	DEFINE WHICH TYPE DATABASE TO USE WITH A PROJECT	167
W	WIRE PROPERTIES	168
W.1.	WIRE DIRECTION	168
W.2.	CHANGING TARGETS	171
W.3.	DEFINING A LINK	173
W.4.	GRAPHICAL WIRE NUMBERING	174
W.5.	SIGNAL PROPERTIES	176
W.6.	WORKING WITHOUT SIGNAL PROPERTIES	177
W.7.	VIEW AND CHANGE WIRE ATTRIBUTES	178
W.8.	DEFINE THE POSITION OF THE WIRE TEXT	179
X	TERMINAL MATRIX	181
X.1.	GENERATING A TERMINAL MATRIX	181
X.2.	HANDLING MULTI-LAYER TERMINALS	183
X.3.	CREATING A TEMPLATE FOR A TERMINAL MATRIX	184
X.3.1.	<i>General Information</i>	184
X.3.2.	<i>Drawing Graphics Associated with Each Terminal – One Geometry for All Terminals</i>	190
X.3.3.	<i>Drawing Graphics Associated with Each Terminal – One Specific Geometry for Each Kind of Terminal</i>	192
X.4.	DRAW MORE THAN ONE TERMINAL STRIP PER PAGE	195

Y	CABLE PLAN WITH GRAPHICS	199
Y.1.	CREATING A CABLE PLAN WITH GRAPHICS	201
Y.2.	CREATING A TEMPLATE FOR CABLE PLANS WITH GRAPHICS	201
Y.3.	SHOWING NOT USED CABLE CORES IN CABLE PLAN	203
Y.4.	CABLE PLAN WITH SHIELD INFORMATION FOR CABLES	203
Z	WIRING LISTS	207
Z.1.	GRAPHICAL WIRING LIST	209
Z.1.1.	<i>Creating an Excel/Text File</i>	210
AA	FUNCTION AND LOCATION	215
BB	REVISION MANAGEMENT	220
CC	ADVANCED FUNCTION/LOCATION AND PRODUCT MANAGEMENT	221
CC.1.	DATABASE FOR FUNCTION/LOCATION AND PRODUCT MANAGEMENT	221
CC.2.	USING NESTED ASPECTS	229
DD	COMPLEX MODIFICATIONS OF THE DATABASE LISTS	233
DD.1.1.	<i>Product Editor</i>	233
DD.1.2.	<i>Terminal Editor</i>	233
DD.1.3.	<i>Cable Editor</i>	235
DD.1.4.	<i>Connector Editor</i>	235
DD.1.5.	<i>Signal Editor</i>	235
DD.1.6.	<i>Wire Editor</i>	235
DD.1.7.	<i>Function/Location Editor</i>	236
DD.1.8.	<i>Document Editor</i>	236
DD.1.9.	<i>REDLINING OBJECTS LIST</i>	237
EE	ADVANCED PROCESSING OF A PROJECT	238
EE.1.	NAVIGATION IN THE PROJECT	238
EE.2.	WORKING WITH AUTOMATICALLY GENERATED CONNECTIONS	239
EE.3.	ORTHOGONAL WIRING	240
EE.4.	ADVANCED FUNCTIONALITY FOR CABLES	243
EE.4.1.	<i>Define Cable Names Automatically</i>	243
EE.4.2.	<i>Display Cable Cores a Second Time</i>	243
EE.5.	ADVANCED FUNCTIONALITY FOR POTENTIAL NUMBERING	244
EE.6.	COPYING PAGES	246
EE.6.1.	<i>Copy Single Page in the Same Workspace</i>	246
EE.6.2.	<i>Copy Single Page in the Same Workspace or in Different Workspace</i>	246
EE.6.3.	<i>Copy Multiple Pages Between Different Workspaces</i>	248
EE.7.	CHANGING PAGE TEMPLATES	249
EE.8.	TRANSLATING A WORKSPACE	250
EE.8.1.	<i>Displaying Several Translations at the Same Time</i>	252
EE.8.2.	<i>Translation of Part Strings</i>	252
EE.8.3.	<i>Updating Translation Text in a Sheet</i>	252
EE.8.4.	<i>Looking up Texts in the Translation Database</i>	254
EE.8.5.	<i>Exchanging the Source Languages of a Workspace</i>	255
EE.9.	CHANGING THE FONTS AND ATTRIBUTES FOR ALL TEXTS IN THE WORKSPACE, SYMBOL LIBRARIES OR PAGE TEMPLATES	256
EE.10.	PLC FUNCTIONALITIES IN THE ADVANCED LEVEL	259
EE.11.	GENERATE GRAPHICAL LISTS IN ONE STEP	261

FF	CUSTOMIZING THE WORKSPACE/PAGE INFORMATION WINDOWS	262
GG	CUSTOMIZING THE WORKSPACE TREE	265
GG.1.	HIDE UNNECESSARY DATABASE AND GRAPHICAL LISTS	265
GG.2.	SORTING OF DOCUMENTS IN THE WORKSPACE TREE	266
HH	USER-DEFINED SQL-QUERIES	268
HH.1.1.	<i>Creating an SQL-Query for Adding a Database List in the Workspace Explorer</i>	268
HH.1.1.	<i>Definition of Graphical Formulas</i>	273
II	TERMINAL PLAN	275
II.1.	GENERATING A TERMINAL PLAN	275
II.2.	CREATING A TEMPLATE FOR A TERMINAL PLAN	276
II.2.1.	<i>Drawing Graphics Associated with Each Terminal</i>	279
II.2.2.	<i>Drawing Cables as a Grouping of Cable Cores</i>	279
II.2.3.	<i>Drawing Cables/Geometry on Both Sides of the Terminals</i>	282
II.2.4.	<i>Showing that Cable Cores are Used on Previous/Next Page</i>	283
II.2.5.	<i>Symbols for a Terminal Plan</i>	285
JJ	CONNECTOR MATRIX AND PLAN	286
JJ.1.	GENERATING A CONNECTOR MATRIX	286
JJ.2.	CREATING A TEMPLATE FOR A CONNECTOR MATRIX	287
JJ.3.	DRAW MORE THAN ONE CONNECTOR PER PAGE	288
JJ.4.	CONNECTOR PLAN	290
KK	TERMINAL ROW PICTURE PLAN	291
KK.1.	GENERATING A TERMINAL ROW PICTURE PLAN	291
LL	CABLE TERMINAL ROW PLAN	294
MM	PRODUCT ASSEMBLY LIST	295
MM.1.	GENERATING A TEMPLATE FOR THE PRODUCT ASSEMBLY LIST	297
NN	MULTICORES	301
NN.1.	USING MULTICORES	301
OO	INSERTING COMPONENTS NOT IN THE DRAWING VIA THE DATABASE EDITORS	303
OO.1.	EDITOR FOR COMPONENTS WITHOUT GRAPHICS	303
OO.2.	INSERTING SPARE OR GROUND TERMINALS VIA THE TERMINAL EDITOR	307
OO.2.1.	<i>Handling Cables Without Graphics</i>	308
OO.3.	INSERTING SYMBOLS FOR COMPONENTS WITHOUT GRAPHICS IN THE DIAGRAM (PICK LIST)	309
PP	AUTO DIAGRAM	310
PP.1.	INTRODUCTION	310
PP.2.	CREATING SYMBOLS (GROUPS)	310
PP.3.	PAGE TEMPLATES	313
PP.4.	EXCEL SPREADSHEET	314
PP.4.1.	<i>Project Data (ProjectInfo)</i>	314
PP.4.2.	<i>Page Data (PageInfo)</i>	315
PP.4.3.	<i>Select Symbol (Symbols)</i>	316
PP.4.4.	<i>Defining Alias Names</i>	317
PP.5.	AUTOMATIC GENERATION OF CIRCUIT DIAGRAMS	318
PP.5.1.	<i>The Autodiagram Command</i>	318

QQ	LISTS AND LABELS EDITOR	327
QQ.1.	CREATING LABEL TEMPLATES	327
QQ.1.1.	<i>Files for Label Templates</i>	331
QQ.1.2.	<i>Changing Label Templates</i>	331
QQ.1.3.	<i>Printing Labels</i>	331
QQ.2.	CREATING LIST TEMPLATES	332
QQ.2.1.	<i>Files for List Templates</i>	335
QQ.2.2.	<i>List Title (Text Object)</i>	335
QQ.2.3.	<i>Header Lines</i>	335
QQ.2.4.	<i>Group Header</i>	336
QQ.2.5.	<i>Group Footer Lines</i>	336
QQ.2.6.	<i>Data Lines</i>	336
QQ.2.7.	<i>Footer Line</i>	336
QQ.2.8.	<i>List Footer Line (Text Object)</i>	336
QQ.3.	EDITING LISTS	337
QQ.3.1.	<i>Changing Column Titles (Header Lines)</i>	337
QQ.3.2.	<i>Changing a List (Font Size, Fields Order, Adding or Deleting Fields)</i>	339
QQ.3.3.	<i>Changing the List Title</i>	340
QQ.3.4.	<i>List Title with Questions</i>	342
QQ.3.5.	<i>List Title with Generation Date</i>	343
QQ.3.6.	<i>Filtering in the Lists</i>	344
QQ.3.6.a.	List of Parts for a Particular Manufacturer	344
QQ.3.6.b.	List of Parts without Terminals	346
QQ.3.7.	<i>Calculations</i>	347
QQ.3.7.a.	Calculation of Order Costs	347
QQ.3.8.	<i>Line Numbering</i>	349
QQ.4.	DEFINING SQL-QUERIES FOR LISTS AND LABELS	350
QQ.4.1.	<i>Joining Two Lists</i>	350
QQ.4.1.a.	Joining Information from Cable List and Cable-Wires List	350
QQ.4.2.	<i>Filtering Double Records</i>	353
QQ.4.2.a.	List of Functions/Locations on Pages	354
QQ.4.3.	<i>Sorting a List</i>	354
QQ.4.3.a.	Order List	354
QQ.4.3.b.	Cable Length Addition	355
QQ.4.3.c.	Length Calculation for Cable Channels and Rails	357
QQ.4.4.	<i>Multiple Printing of Labels</i>	358
QQ.4.5.	<i>Labels for Component Names with Defined Content</i>	359
QQ.5.	USEFUL TABLES IN ACCESS DATABASE	360
QQ.6.	STATEMENT SUMMARY AND COMMANDS IN LISTS AND LABELS	361
QQ.6.1.	<i>SQL Statements</i>	361
QQ.6.1.a.	SELECT	361
QQ.6.1.b.	SELECT DISTINCT	361
QQ.6.1.c.	JOIN -clause	362
QQ.6.1.d.	ORDER BY	363
QQ.6.1.e.	WHERE/WHERE NOT	363
QQ.6.1.f.	UNION ALL	364
QQ.6.1.g.	Uppercase/ Lowercase	365
QQ.6.1.h.	Logical Operators	365
QQ.6.1.i.	Relational Operators	365
QQ.6.2.	<i>Commands in Lists and Labels</i>	365
QQ.6.2.a.	Conditions: If	365

QQ.6.2.b. Upper\$/Lower\$	366
QQ.6.2.c. Val	366
QQ.6.2.d. Fstr\$	366
QQ.6.2.e. AskString\$	366
QQ.6.2.f. Len, Left\$, Right\$	367
QQ.6.2.g. Relational Operators	367
RR CABINET LAYOUT	368
RR.1. DRAWING CABINET LAYOUTS	368
RR.1.1. <i>With the Cabinets Module</i>	368
RR.1.2. <i>Without the Cabinets Module</i>	377
RR.2. USING LAYERS	381
RR.3. DIMENSION	382
RR.4. COMPARISON BETWEEN CIRCUIT DIAGRAMS AND CABINETS	385
SS USEFUL TOOLS	388
SS.1. COMPRESSING PROJECTS	388
SS.2. REORGANIZE WORKSPACES	389
TT SETTINGS	390
TT.1. SYSTEM SETTINGS	390
TT.2. WORKSPACE PROPERTIES	391
TT.3. PROPERTIES FOR CIRCUIT DIAGRAMS	392
TT.4. PAGE PROPERTIES	393
TT.5. CUSTOMIZING THE INTERFACE	394
TT.5.1. <i>Quick Access Toolbar</i>	394
TT.5.2. <i>Define Hotkeys</i>	396
TT.5.3. <i>User Defined Categories</i>	396
UU COMMANDS EXPLORER	402
VV INTELLIGENT DRAWING LEGACY MODULE	406
VV.1. TOOLS FOR SCANNED DRAWINGS	407
VV.1.1. <i>Importing Multiple Scanned Drawings in One Step</i>	407
VV.1.2. <i>Covering Scanned Symbols with an (White) Area ("BkSymbol" Command)</i>	409
VV.2. TOOLS FOR MAKING DXF/DWG IMPORTED DRAWINGS MORE INTELLIGENT	410
VV.2.1. <i>Defining Patterns ("DefinePatterns" Command)</i>	410
VV.2.2. <i>Controlling and Changing Patterns, Assigning Texts ("ShowPatterns" Command)</i>	412
VV.2.3. <i>Recognizing Patterns ("RecognizeDrawingPatterns" and "RecognizeWorkspacePatterns" Commands)</i>	414
VV.2.4. <i>Displaying all Elements with Logic</i>	414

A INTRODUCTION

By following this training manual step by step you will gain experience in using *SEE Electrical*. If you have already worked with *SEE Electrical*, you can follow the examples step by step. You will learn the fundamental *SEE Electrical* functions.

The first chapters contain information about the features used further in this *Training Manual*. Chapters "Easy Editing in Database Lists" to "Function and Location" apply to *SEE Electrical Standard*, chapters "Complex Modifications of the Database Lists" to "List and Label Editor" apply only to *SEE Electrical Advanced*. Chapter "Cabinet Layout" requires *SEE Electrical Standard*, and some examples require the Cabinets module in addition.

Abbreviations used in this training manual:

CA	Select a category
CO	Select a command
M	Select from pull-down menu
+	Select an element with the cursor
#	Keyboard entry
>	Select a field in a window
<Input>	Type text or select element etc.
T	Click on tab in window
I	Select a toolbar icon

A.1. THE WORKSPACE

Most often a workspace in *SEE Electrical* contains circuit diagrams. Graphical lists are generated automatically using the diagram information, for example:

- ✓ List of Products
- ✓ List of Terminals
- ✓ List of PLC
- ✓ List of Wires
- ✓ List of Cables
- ✓ List of Documents
- ✓ etc.

Project data is used for the generation of the graphical List of Terminals (as well as terminal matrix in *Standard level*), List of Cables and List of Products.

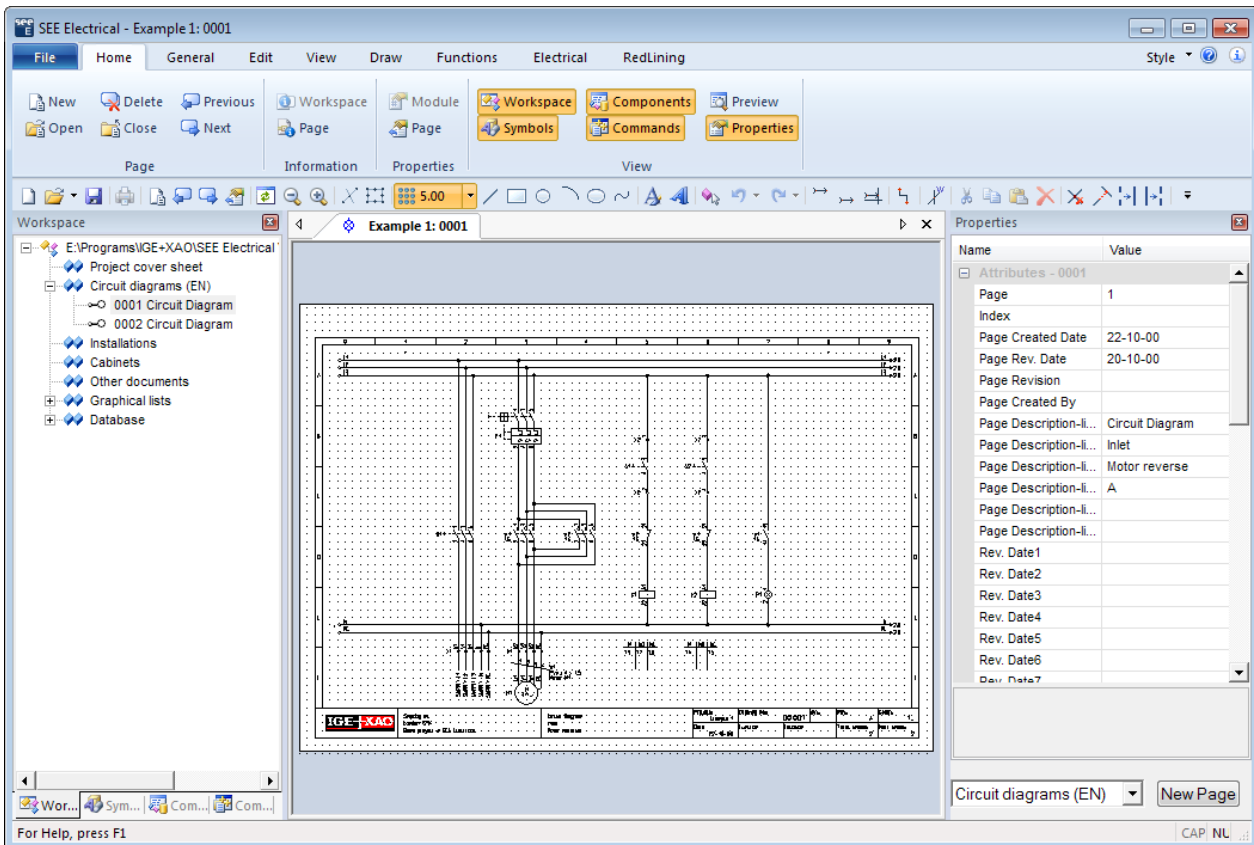
You can create drawings of cabinets or installation plans within a project.

SEE Electrical contains different modules that provide functions for drawing circuit diagrams, installations, or cabinets. The availability of the appropriate module allows you to create the examples.

Other documents can be added to the workspace in the **Other documents** area, for example *Word* files or *Excel* spreadsheets.

By default three areas appear on the screen when you launch *SEE Electrical*.

The Workspace/Symbols/Commands area is located on the left hand side in the *SEE Electrical* window. The drawing area is located in the centre of the *SEE Electrical* window. The **Properties** area is located on the right hand side in the *SEE Electrical* window.

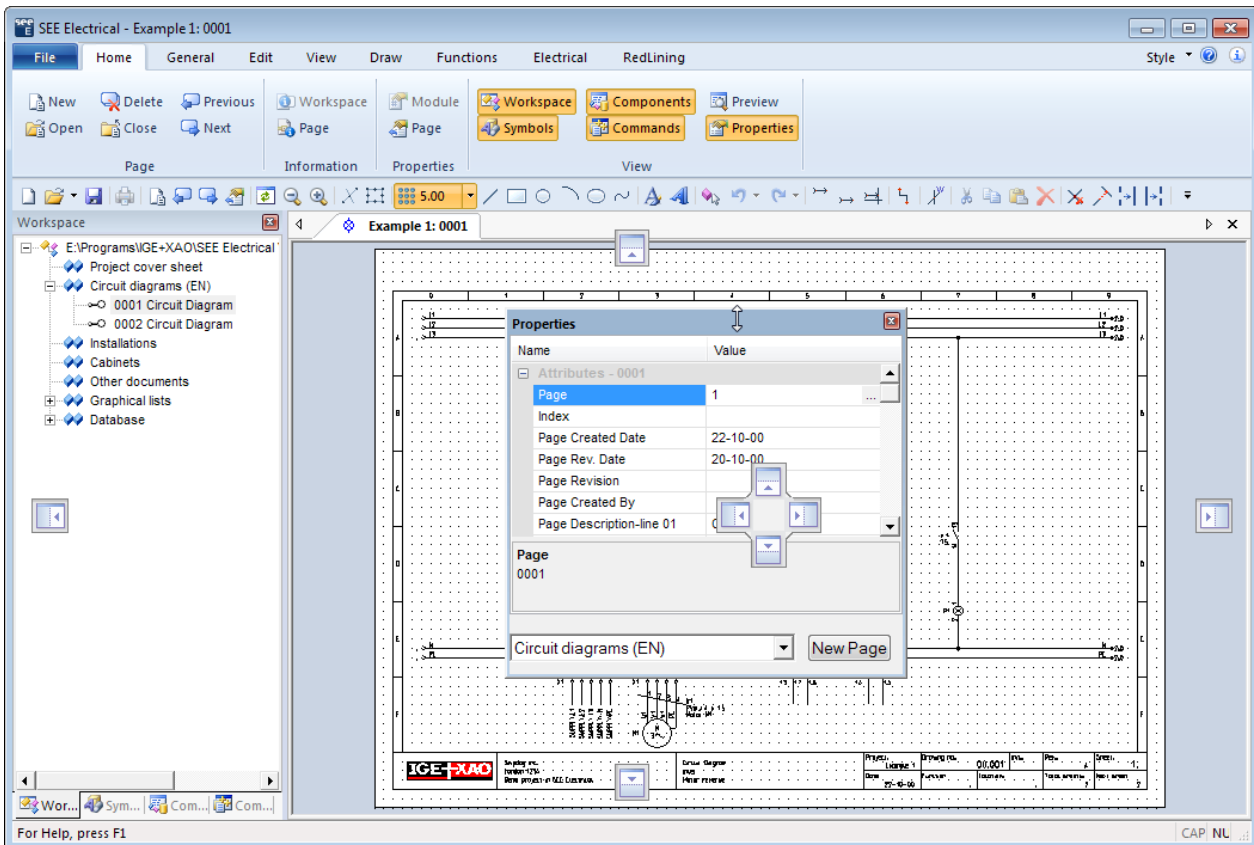


To display a list, go to the **Workspace** area, open Database lists and select the desired list. You can toggle between **Workspace**, **Symbols**, **Components** and **Commands**.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

You can change the position of the **Workspace/Symbols/Properties** and **Commands** explorers by dragging and dropping them at the desired location. Use the directional arrows which appear to drop the explorers at their new position.



To display a preview of the currently active page, go to the **Home** category and activate the **Preview** command. Within the **Preview** window that appears you have the possibility to zoom in and out the page. See **Working with a Zoomed Part of the Drawing** for more details.

A.2. FOLDERS AND FILES

SEE Electrical uses the following folders and files:

< SEE Electrical V7 folder>	The program files of <i>SEE Electrical</i> are saved in this folder.
...\PROJECTS	In this folder, you can find the workspace files of <i>SEE Electrical</i> delivered by default. Workspace files have the .SEP extension.
...\SYMBOLS	This folder contains the symbol databases in <i>SEE Electrical</i> . Symbol databases have the .SES extension. The TYPES.SES database, required in the <i>Standard</i> and <i>Advanced</i> levels is also stored here. Please note that the SYSTEM.SES library is required for internal purposes and must not be removed from this folder. If the Cabinet module is available, the IndexTable.SES is used to create index tables.
...\TEMPLATES	This folder contains workspace and page templates, templates for lists and labels. Fonts are saved here, too. SEP: Workspace templates TDW: Page templates CABLESNEW.MDB or CABLESNEWIEEE.MDB that contain settings for user defined cables. DAT: Fonts
...\TEMPLATES\LABEL_SETTINGS	SLS files (used for creating labels for different printer formats).

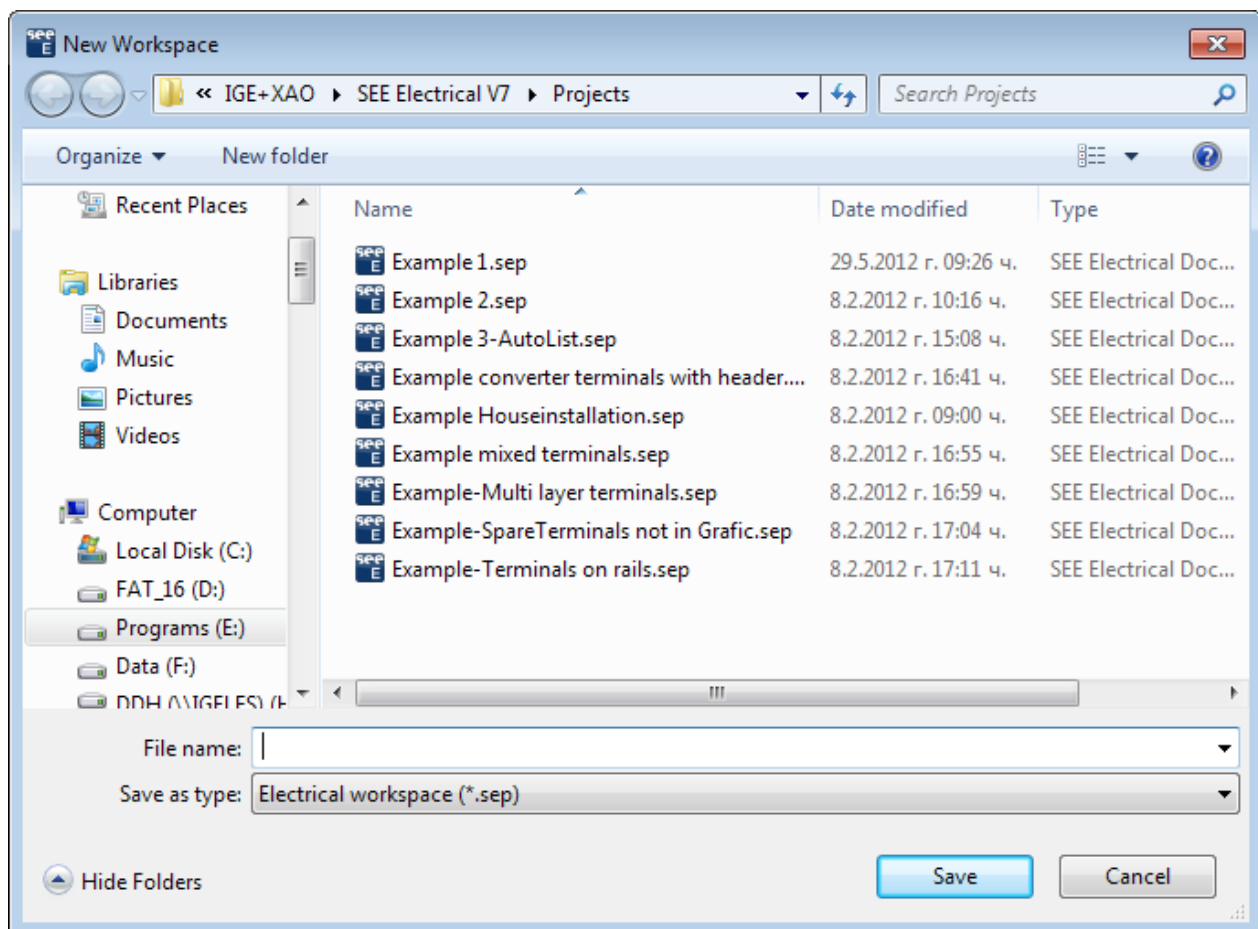
In addition, *SEE Electrical Advanced* uses the following files:

...\TEMPLATES	In this folder, you can find the TRANSLATIONNEW.MDB translation database used by <i>SEE Electrical Advanced</i> for translating the workspaces.
---------------	---

B CREATING A NEW WORKSPACE

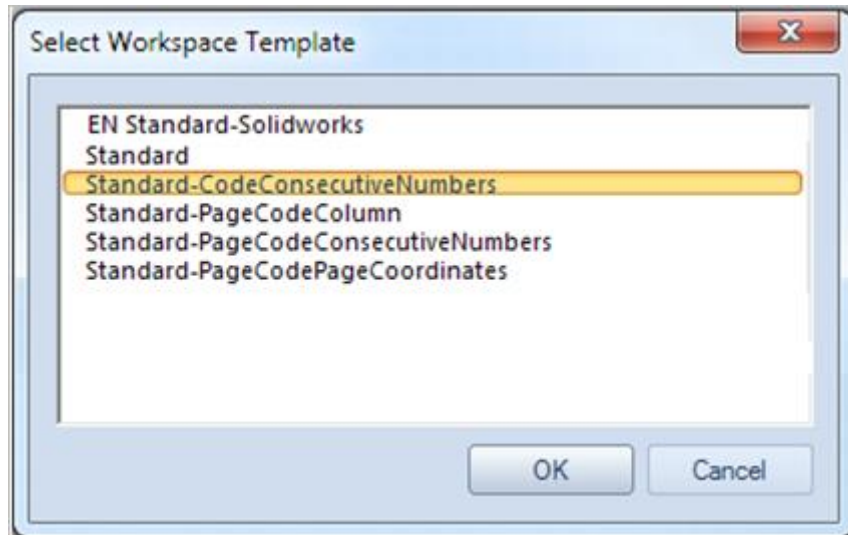
Exercise 2-1: Create a new workspace.

- 1.CA **File**
- 2.CO **New**



- 3.> File name
- 4.# My Workspace
You can type another workspace name.
- 5.> **Save**

The workspace is created. A list of available templates appears.



- 6.> <Template>
Select a workspace template.
A workspace template contains page templates which define, for example, the number of columns in the drawing, etc.
The *SEE Electrical* installation package contains templates. Choose the **Standard** template.
- 7.> Click **OK**.
The newly created workspace is open. The **Properties** pane visible on the left contains information about the workspace.
In the "*File-name*" field you can see the name and location of your workspace (<name>.SEP).
- 8.> Workspace Description-line 01
- 9.# Workspace Example
- 10.> Workspace Created Date
If the "**Workspace Created Date**" field is not visible, please scroll down.
The "**Workspace Created Date**" field is filled in automatically.
- 11.# Type in the desired date.
SEE Electrical shows the date in the **Workspace Created Date** line.
If you wish to use a different date format, you must change the date format in your *Windows* system settings. Depending on the operating system, different possibilities are available. When the date format of the current computer is different from the date format used in the workspace, and a conversion is not possible, the software will change the format for the date to a text format, to guarantee that the original date is to be seen.
- 12.> Workspace Created By
- 13.# Type in your name.
You can fill in additional information, if you wish.
The workspace information will be automatically inserted in the norm sheet of the circuit diagram, provided that your template contains the relevant text placeholders.

C DRAWING THE FIRST PAGE OF A CIRCUIT DIAGRAM

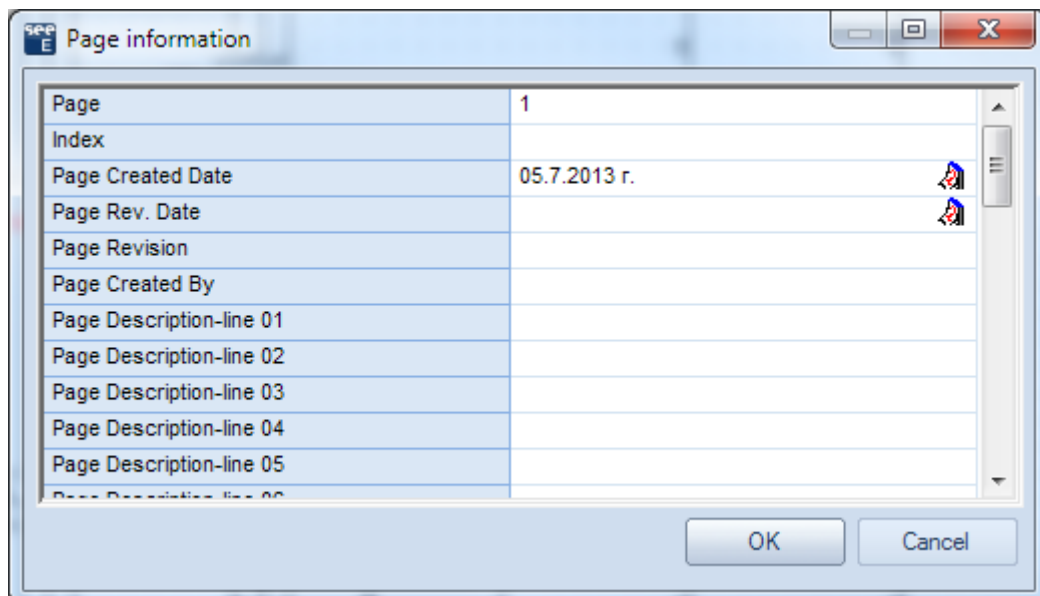
C.1. CREATING PAGE 1

Exercise 3-1: You will now create the first page of the project.

1.> Create page

Click the **New Page** button in the **Properties** pane containing the workspace information fields.

The **Page information** dialogue appears, allowing you to type in information about the new page.




2.> Page

Page number: "1" (automatically suggested).

By default, *SEE Electrical* offers you the first available page number in the respective module, in this case 1. You can modify it, if desired

3.> Page Created Date

SEE Electrical automatically inserts the current date. You can change the date by clicking the  field which appears in the "**Page Created Date**" field.

You can fill in different page information if desired.

4.> Page Description-line 01

5.# Motors

6.> **OK**

Close the dialogue box.

SEE Electrical opens a new circuit diagram page.


The page information will be automatically inserted in the page, if the corresponding text placeholders are available in the page template..

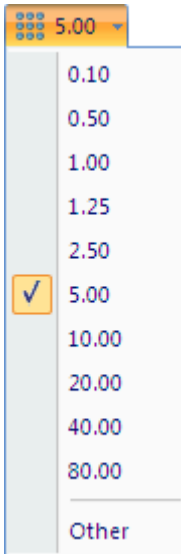
You can start drawing the circuit diagram.

C.2. GRID

Using a grid you can align precisely geometrical elements, texts and components.

You can toggle the visibility of the grid by clicking the  icon in the **Styles** panel.

After clicking , the list of the default grid sizes appears:



If you choose "Other", you can set your own grid size.

Hint:

*You can also define customised values to appear in the pull-down list of grid values. In order to do this, you must modify the **CAEGridSettings.xml** file from the \Templates installation directory. This file can contain no more than 10 grid values.*

C.3. DRAWING PAGE 1

Exercise 3-2: Insert the Power supply group.

- Click the **Symbols** tab in the left pane of the main *SEE Electrical* window to display the **Symbols Explorer**.

This pane contains the **Workspace**, **Symbols**, **Components** and **Commands Explorers**. If they are not visible, you can display them by activating the toolbars from the **View** panel of the **Home** category.

Exercise 3-3: Select the database that contains the symbols you wish to work with. In this case, select **Examples**.

- 1.T Activate the **Symbols** tab.
2. Double-click the *Examples* symbol database.
SEE Electrical opens the symbol database.
Various symbol folders are displayed.
3. Double-click the **Examples** symbol folder to open it.
All symbols are displayed in the **Symbols** area.
4. Click the **Power supply** group.

The symbol appears attached to the cursor.

5. "Drag" the symbol to the desired position in the drawing sheet.

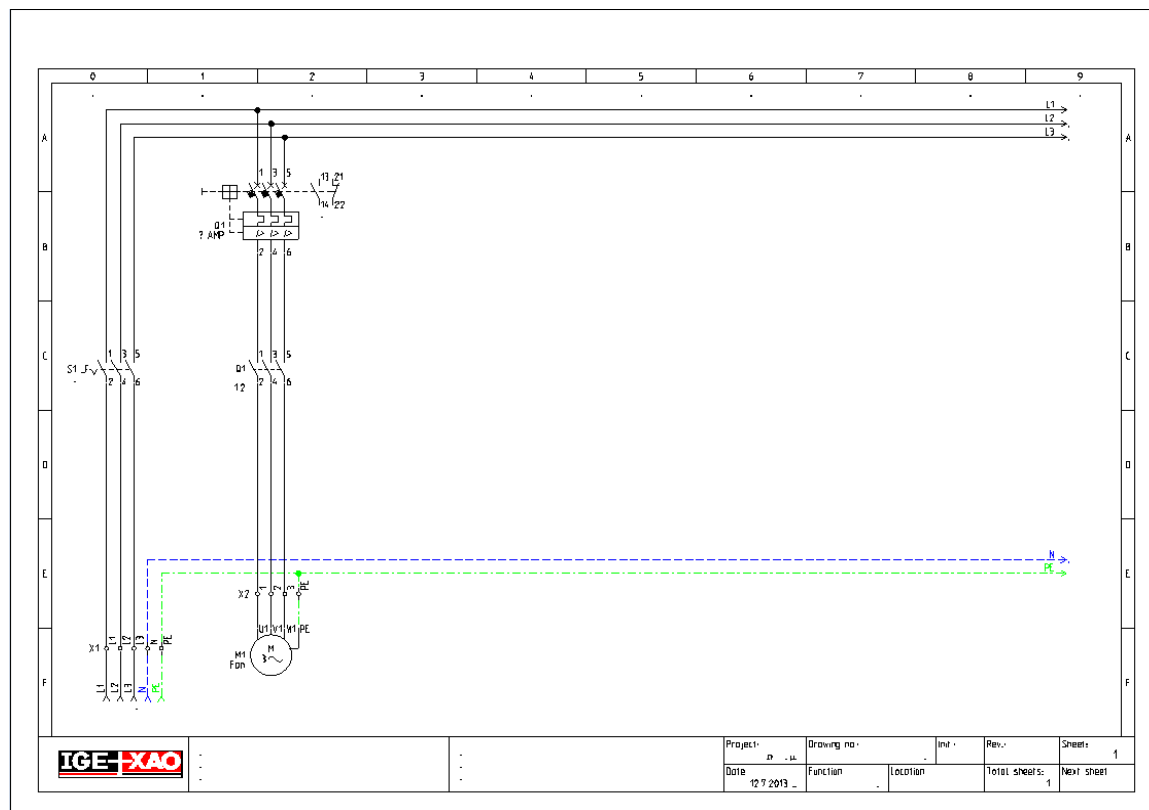


- 6.+ "Drop" the power supply in column 0.
- 7.> Product (-)
- 8.# X1
Enter the name of the terminal strip.
Do not change the terminal number.
- 9.> **OK**
The next terminals are automatically assigned to the terminal strip X1.
A dialogue box for the name of the potential appears.
- 10.> Product (-)
- 11.# L1
The name of the potential is suggested.
- 12.> **OK**
Click **OK** to accept the suggested name.
Use the same approach and accept the suggested names for the next four potentials.
Right-click to exit the insertion mode.

Exercise 3-4: Save the workspace.

- 1.CA **File**
- 2.CO **Save**

Exercise 3-5: Insert the Three-phase motor direct group.



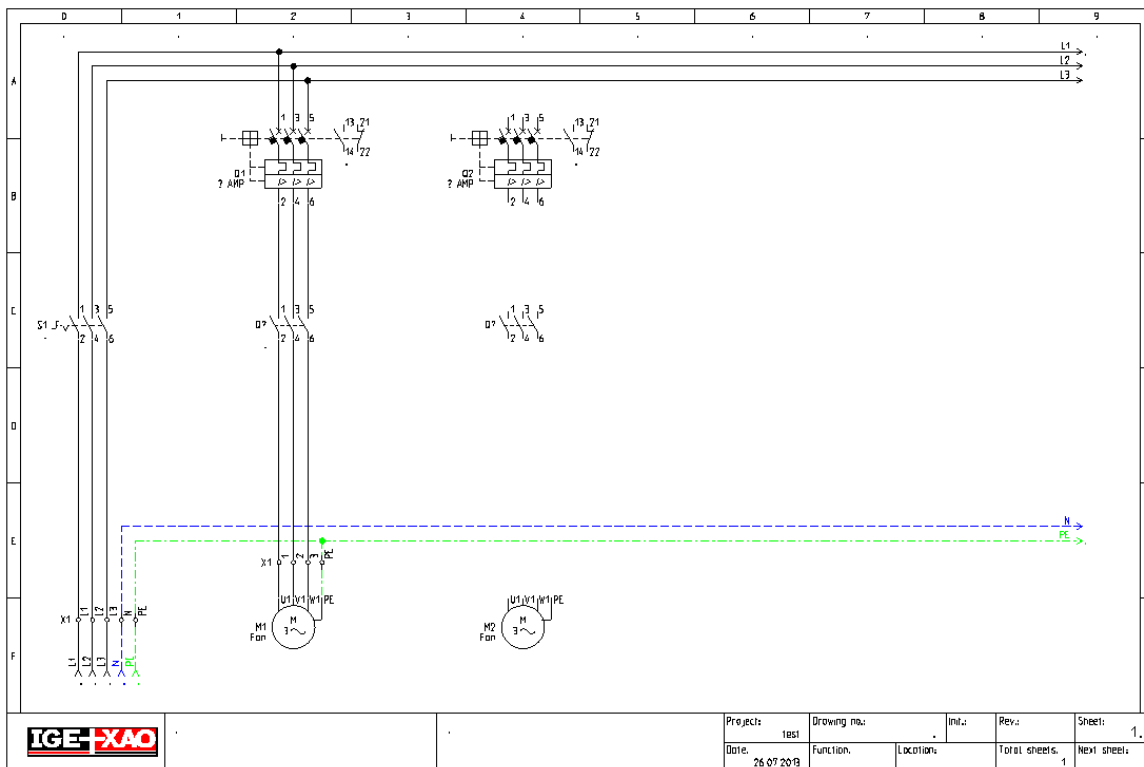
1. Move the cursor into the **Symbols** area.
2. Click the **Three-phase motor direct** group in the *Example* symbol folder.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

3. Drag the **Three-phase motor direct** group with the cursor to the drawing area.
- 4.+ Drop the group on the desired position in the sheet.
A dialogue box for the name of the terminal appears.
- 5.> Product (-)
6. # X2
The name X1 is suggested. Change it to X2.
- 7.> **OK**
The next terminals are automatically assigned to the terminal strip.
The dialogue box for the Main relay-contact NO appears.
- 8.> Product (-)
- 9.# Q?
You can enter the name of the Main relay-contact NO. But do not change it now, because you do not know it yet.
- 10.> **OK**
Right-click to exit the insertion mode.

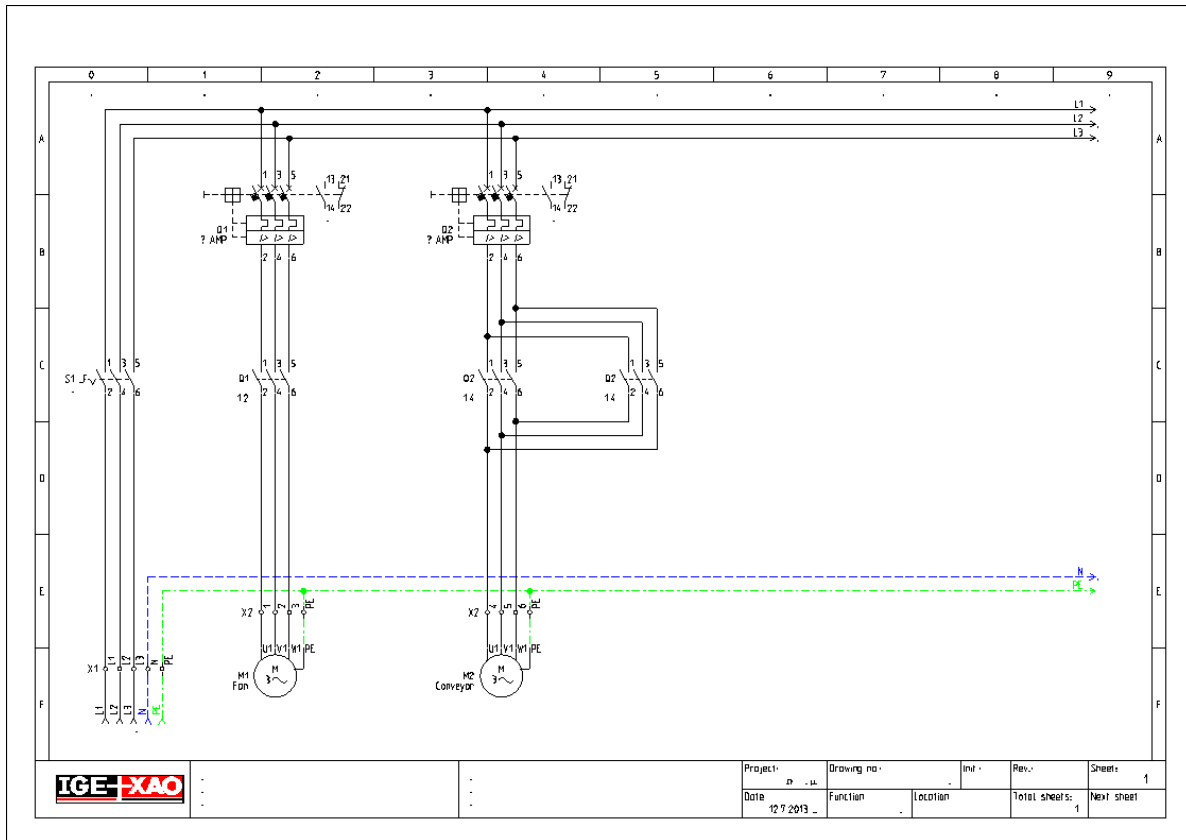
Exercise 3-6: Insert the symbols for Three-phase motor reverse.



1. Move the cursor into the **Symbols** area.
2. Open the **EN61346-2UK** symbol library.
3. Open the **Motors and Generators** symbol folder. Select the **Three-phase + PE** symbol and drag it with the cursor through the workspace.
- 4.+ Click in column 4 to drop the symbol there.
5. Double-click the **Relay-contacts, MAIN** symbol folder to open it, and select the "3-pole NO" contactor symbol.
6. Drag the symbol with the cursor to the desired place –in column 4 for this particular case. Click to drop the symbol.

- 7.> The **Component properties** dialogue box appears.
- 8.> Product (-)
- 9.# Q?
You can enter the name of the contactor. But do not change it now, because you do not know it yet.
- 10.> **OK**
Close the dialogue.
- 11.> Insert the second contactor at the desired position, proceeding in the same way.
The **Component Properties** dialogue box appears again. Leave the Q? value.
- 12.> Click OK
Close the dialogue.
13. Now open the **Terminals** symbol folder and select the **4 terminals 90° vertical** symbol.
Insert it at the desired position (in this case – in column 4) proceeding in the same way as described for the previous symbols.
A dialogue box for the name of the terminal appears.
- 14.> Product (-)
15. # X2
- 16.> Terminal Number
17. # 4
- 18.> Terminal Sorting
19. # 5
- 20.> **OK**
The next terminals are automatically assigned to the terminal strip.
21. Open the *Protective devices* symbol folder.
Select the "*3-pole trip breaker*" symbol, for example, and drag it to column 4. Drop the symbol at the desired position by clicking with the mouse.
22. Right-click to exit the insertion mode.
You have positioned your symbols.

Exercise 3-7: Draw the 3-pole connections.



1. Activate the **Electrical > Wire Connections > 3 Wires** command.
2. Click to select the starting point for the wire on the potential L1 vertically above the connections of the first terminal (X2:4).
3. Click to select the second point for the wire on the top connection of the first terminal (X2:4).
Three connections are drawn automatically: between the potential L1 and the first terminal (X2:4), between the potential L2 and the second terminal (X2:5), and between the potential L3 and the third terminal (X2:6).
They are automatically broken where the symbols are placed (for example, at the main relay-contact).
4. Now use the **Electrical > Wire Connections > Orthogonal Wiring** function to draw a 3-wire connection between the potentials L1, L2 and L3 and the motor.
Activate the **Electrical > Wire Connections > Orthogonal Wiring** command.
Click to select the starting point for the wire on the potential L1 vertically above the connection of the motor.
Click again to select the ending point on the first connection (U1) of the motor.
Four wires are automatically drawn to connect the motor to the potentials.
5. Now continue to draw multiple orthogonal wires proceeding as follows:
Click, for example, the connection between the potential L3 and the motor somewhere below the contactor. Then click the **connection 6** of the second contactor.
Similarly, click the **connection 1** of the same contactor and then click the connection between the potential L1 and the motor somewhere above the first contactor.
6. Right-click to exit the drawing mode.

Hints

1. While you are moving the cursor, a dynamic visualization of the connection and contact points appears, allowing you to see if the wires you are trying to draw are overlapping existing ones or are violating symbols. In such case, drawing is restricted since SEE Electrical automatically checks whether the desired connection is possible.
-


Exercise 3-8: Draw the missing wire between the potential PE and the terminal X2:7. Change the line style and the colour of the wire between the potential PE and the motor M2.

- 1.+ Use the **Electrical** ➤ **Wire Connections** ➤ **1 Wire** command, as already described, to draw the missing wire between the potential PE and the terminal X2:7.
- 2.+ Right-click the wire between the potential PE and the terminal X2:7 and select the **Properties** pop-up command.
The wire properties appear in the **Properties** pane in the right part of the main SEE Electrical window.
- 3.> Penstyle
- 4.> Select the line "Dash" style.
- 5.> Pencil colour
- 6.> Select green colour.
The colour and the line style of the wire are changed dynamically on the screen.
Repeat the same operation for the wire between the terminal X2:7 and the motor.
7. Select the motor M2. Its properties appear in the **Properties** pane. Fill in "PE" in the field for connection 03. Press the "Enter" key to validate.
8. Select the terminal X2:7 and, as described above, change its terminal number to PE via the **Properties** pane.

You have completed Page 1 of the circuit diagram.

Exercise 3-9: Save the workspace.

- 1.CA **File**
- 2.CO **Save**

Save frequently your workspaces. You can also click on the  icon.

C.4. WORKING WITH A ZOOMED PART OF THE DRAWING

It is often necessary to zoom parts of the drawing.

Exercise 3-10: Zooming the drawing through the **Preview** window.

Within the **Preview** window:

- 1.+ Click the first point of the rectangle outlining the area you want to zoom.
The rectangle is defined by two diagonally opposite points.
- 2.+ Click the second point of the rectangle opposite to the first one.
The selected area is zoomed.
You can move the zoomed area through the **Preview** window
- 3.+ Place the cursor on the grey rectangle that defines the zoomed area.
The cursor is displayed as a hand symbol.
- 4.+ Click and hold the left mouse button to move the rectangle.
To zoom out activate the **Zoom Original** command by pressing the F3 hot-key.

Exercise 3-11: Enlarge a part of the circuit diagram.

- 1.CA **View**
- 2.CO **Zoom Window (Zoom panel)**
- 3.+ Click the first point of the rectangle outlining the area you want to zoom.
The rectangle is defined by two diagonally opposite points.
- 4.+ Click the second point of the rectangle opposite to the first.
You can activate the **Zoom Window** command by pressing the F4 hot key.

Exercise 3-12: Switch back to the general view of the circuit diagram.

- 1.CA **View**
- 2.CO **Zoom Original (Zoom panel)**
You can see the whole drawing again.
You can activate the **Zoom Original** command by pressing the F3 hot key.

Exercise 3-13: Moving the zoomed area.

- 1.CA **View**
- 2.CO **Zoom Pan (Zoom panel)**
The cursor appears as a hand.
- 3.+ Click the left mouse button and move the cursor to the desired position.
To zoom out activate the **Zoom Original** command by pressing the F3 hot key

Hint:

*It is possible to zoom with a mouse wheel, pressing and holding down CTRL while scrolling the mouse wheel upwards (enlarge) or downwards (decrease).
If you have a mouse wheel, press and hold it, then you can move the currently zoomed part of the drawing.*

D DRAWING THE SECOND PAGE OF A CIRCUIT DIAGRAM

D.1. CREATING PAGE 2

Exercise 4-1: Create page 2 of the workspace.

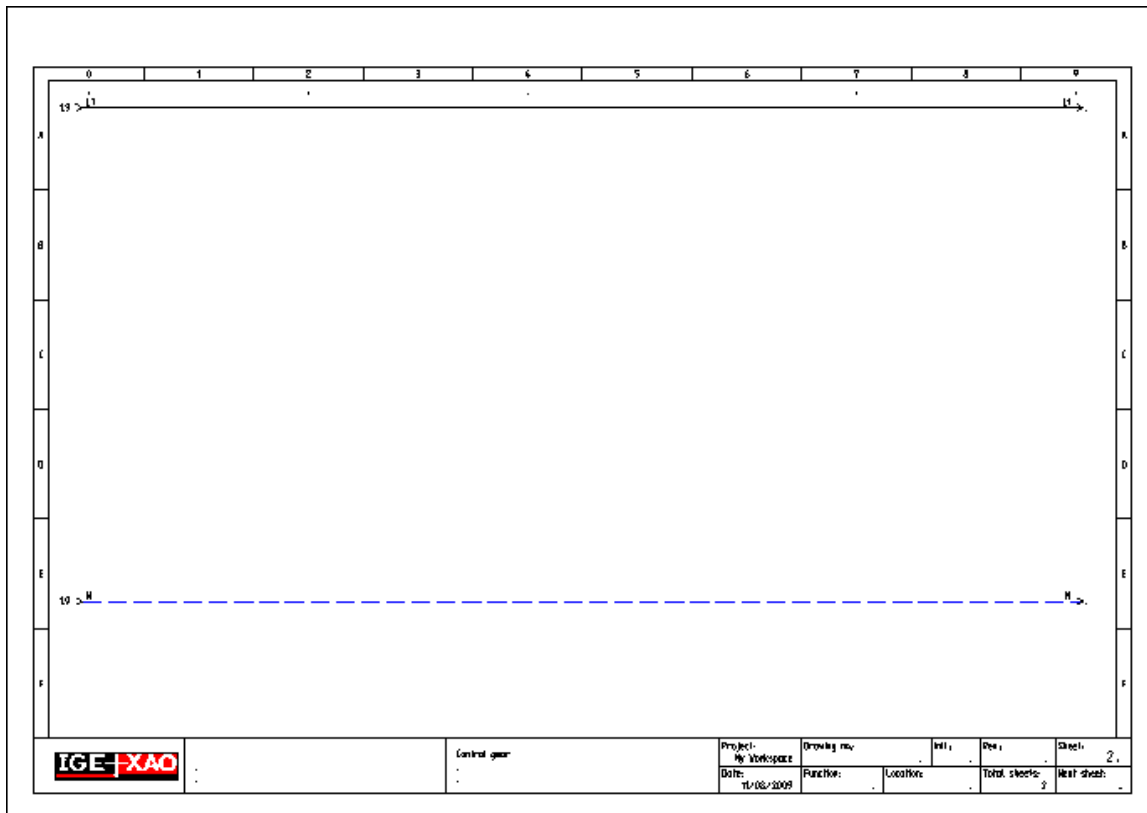
- 1.CA **Home**
- 2.CO **New** (**Page** panel)
If this function is not active, click the **Circuit diagrams** module in the Workspace Explorer area, then click **Home** > **Page** > **New** again.
The **Page information** dialogue box appears.
- 3.> Page description-line 01
- 4.# Control gear
- 5.> Page
Page number 2 is automatically suggested. Do not change it.
- 6.> Page Created Date
The current date is displayed in this line.
- 7.> Click **OK** to close the dialogue box.
SEE Electrical opens a new circuit diagram page.

Now, you can start drawing the second page of the circuit diagram.

D.2. DRAWING PAGE 2



In this chapter you will draw potentials L1 and N, some components and wires on page 2.

Exercise 4-2: Draw potential L1.



- 1.CA **Electrical**
- 2.CO **Top (Potential panel)**
- 3.> **Product (-)**
- 4.# **L1**
Type the name of the potential.
- 5.> **OK**
Close the dialogue.
Left to the potential, the cross-reference to the potential on page 1 appears automatically.

Exercise 4-3: Draw potential N. Change the line style before drawing: select "Dash" and blue colour.

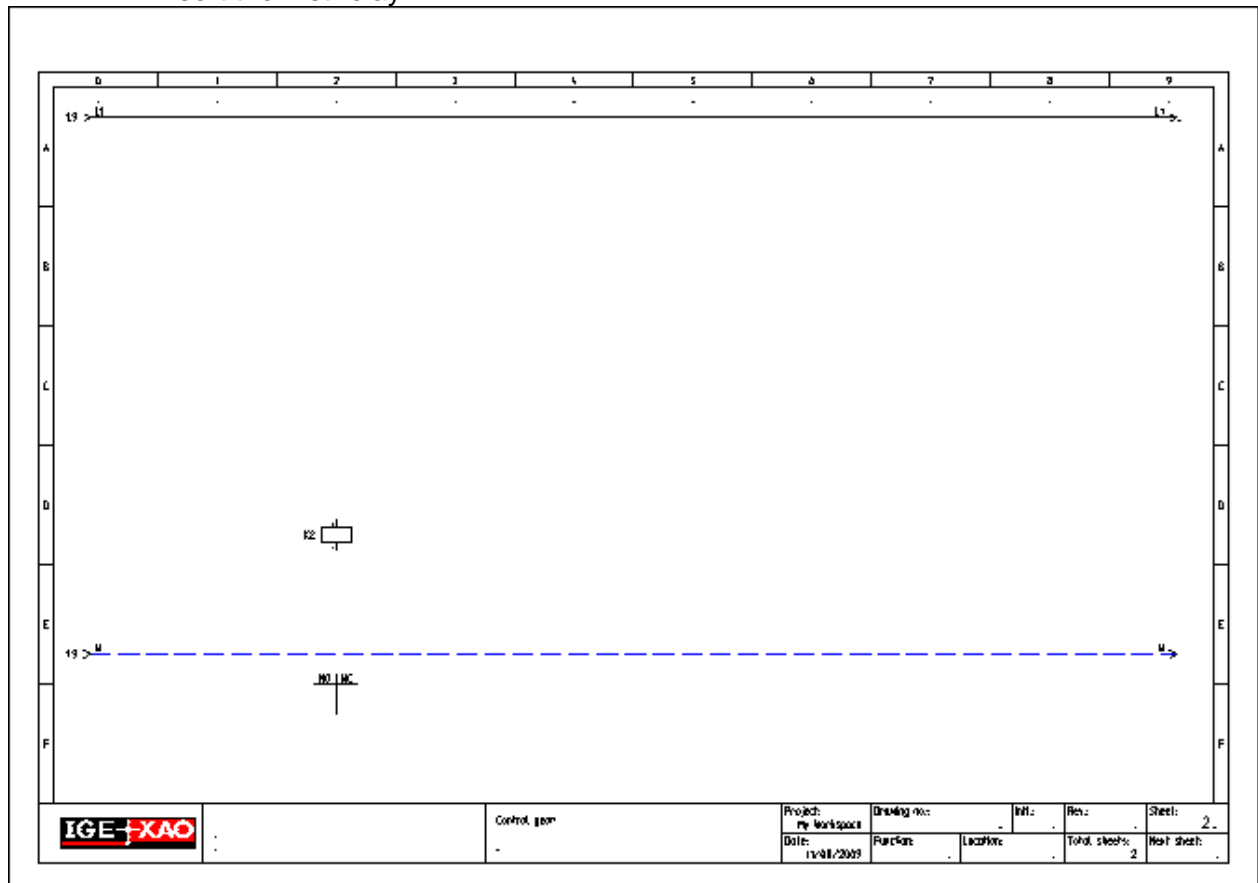
1. In the **Styles** panel of the **Draw** category, click the small arrow in the  icon. Select "Dash". Now you can draw with a dashed line.
2. In the **Styles** panel of the **Draw** category, click the small arrow in the  icon. Choose "Blue". Now you can draw with blue colour.



Exercise 4-4: Draw the potential N.

1. CA **Electrical**
2. CO **Bottom (Potential panel)**
3. > Product (-)
4. # N
5. > OK
The dialogue box closes.
Change the line style again - choose "Solid" line and black colour.
6. In the Style toolbar, change the line style - choose "Solid" line.
7. In the Style toolbar, change the colour - choose black.

Exercise 4-5: Insert the relay symbol.

- Insert the first relay.



- Select the "EN61346-2UK" symbol database.
1. Move the cursor into the **Symbols** area.
Double-click the "EN61346-2UK" symbol database or click the plus sign to the left of the symbol folder   **EN61346-2UK**.
The symbol database opens and the symbol folders are displayed.
 2. Double-click the **Relay coils** folder to open it.
If the desired symbol folder is not visible, scroll down the symbol tree to find it.

Training manual

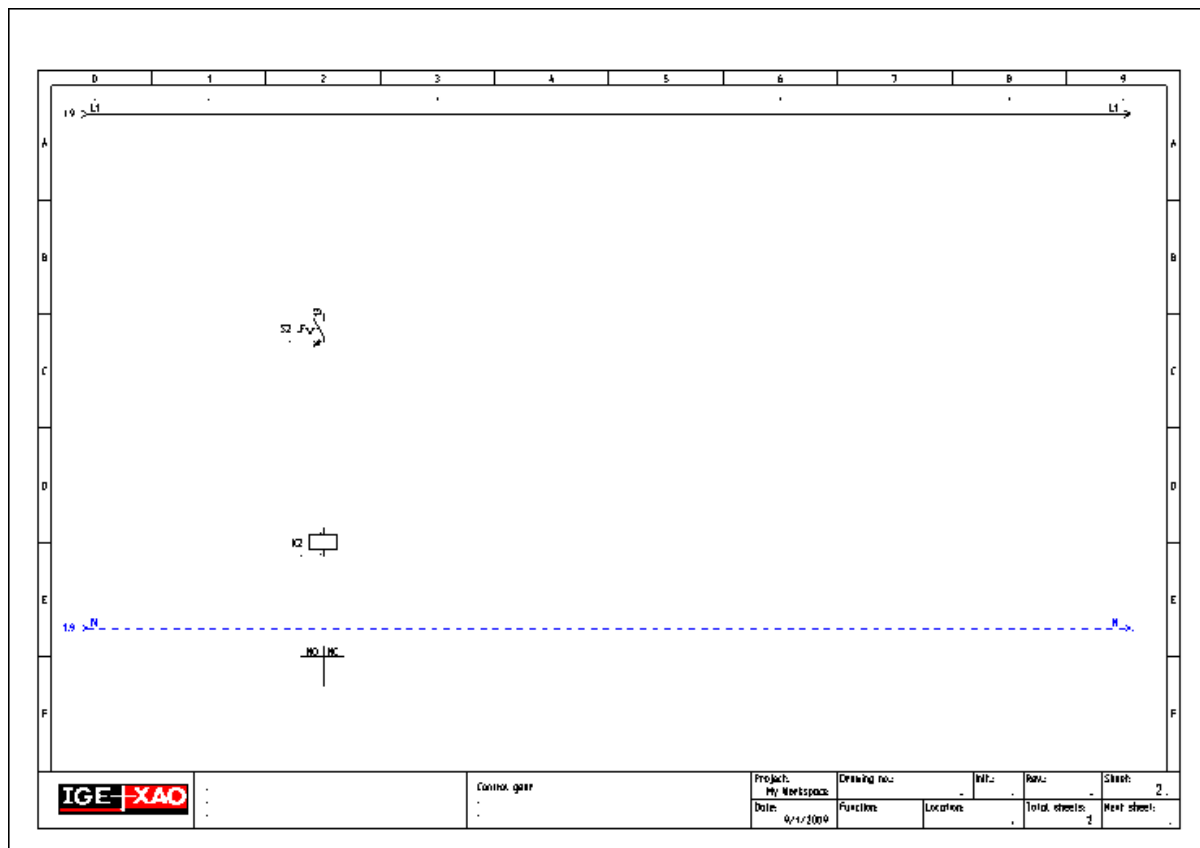
COPYRIGHT © 2013 IGE+XAO. All rights reserved

3. Select the 1-pole component.
4. Move the cursor to the drawing.
The symbol appears attached to the cursor.
- 5.+ "Drop" the symbol at the desired position in the drawing sheet, in this case column 2.
The contact cross appears under the relay.

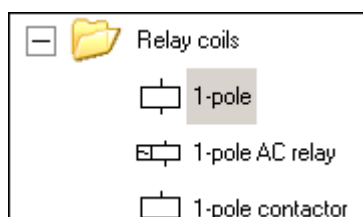
Hint:

You can move the contact cross to another position, if desired.

Exercise 4-6: Insert a one-pole Switchgear symbol.



1. Move the cursor to the Symbols area.
2. Close the "Relay coils" symbol folder by clicking the minus sign to the left of the symbol folder.



3. Double-click the "Switchgear, one pole" symbol folder to open it.
4. Click the **NO turn detent** component.
5. Move the cursor to the drawing area.
The symbol appears attached to the cursor.
- 6.+ "Drop" the symbol at the desired position in the drawing sheet, in this case column 2.
Double-click the symbol. The **Component Properties** dialogue appears.
- 7.> Connection 00
- 8.# 13
Type the contact number.
- 9.> Connection 01
- 10.# 14
Type the contact number.
- 11.> **OK**
Close the dialogue box.

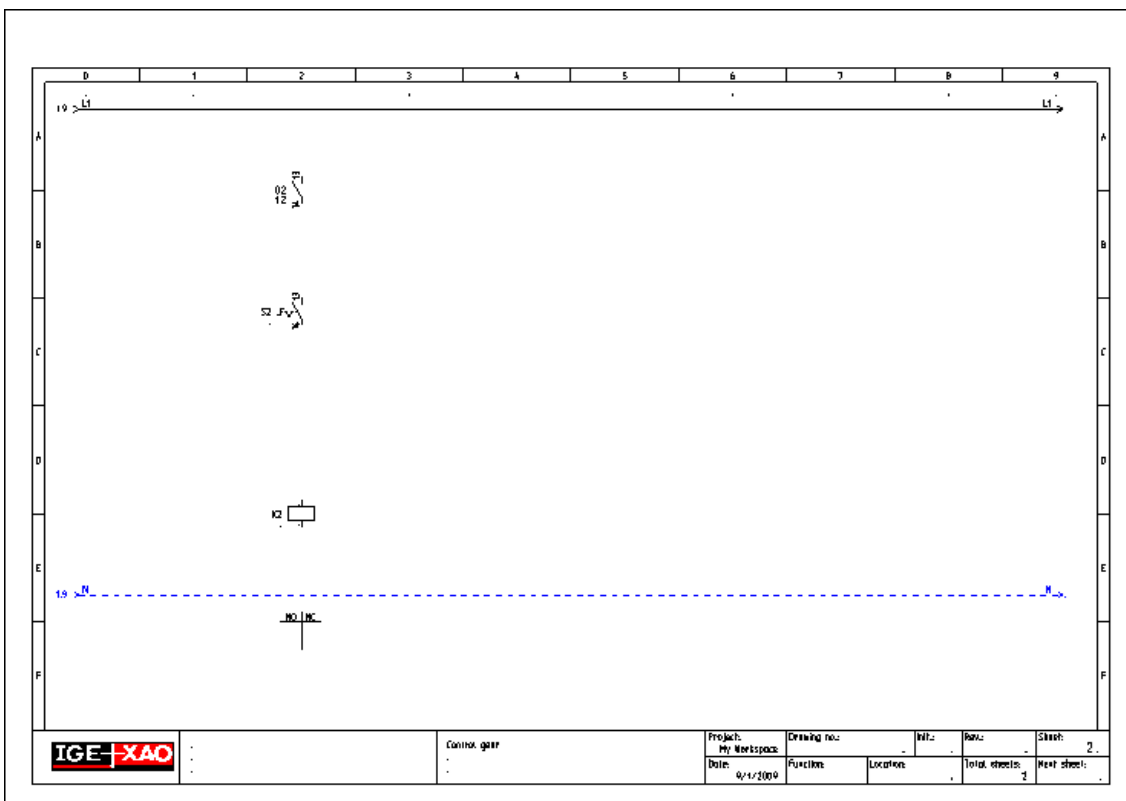
Hint

You can rotate a symbol by 90 degrees or more before inserting it.

- Press the "+" or "-" key on the numerical key board while the symbol is attached to the cursor.
The symbol is rotated 90 degrees clockwise or counter clockwise.


If you press the respective key once again, the symbol is rotated by another 90 degrees.
You can rotate the symbol with the help of the X or Z keyboard keys.

Exercise 4-7: Insert a Relay-contact NO symbol.



Training manual

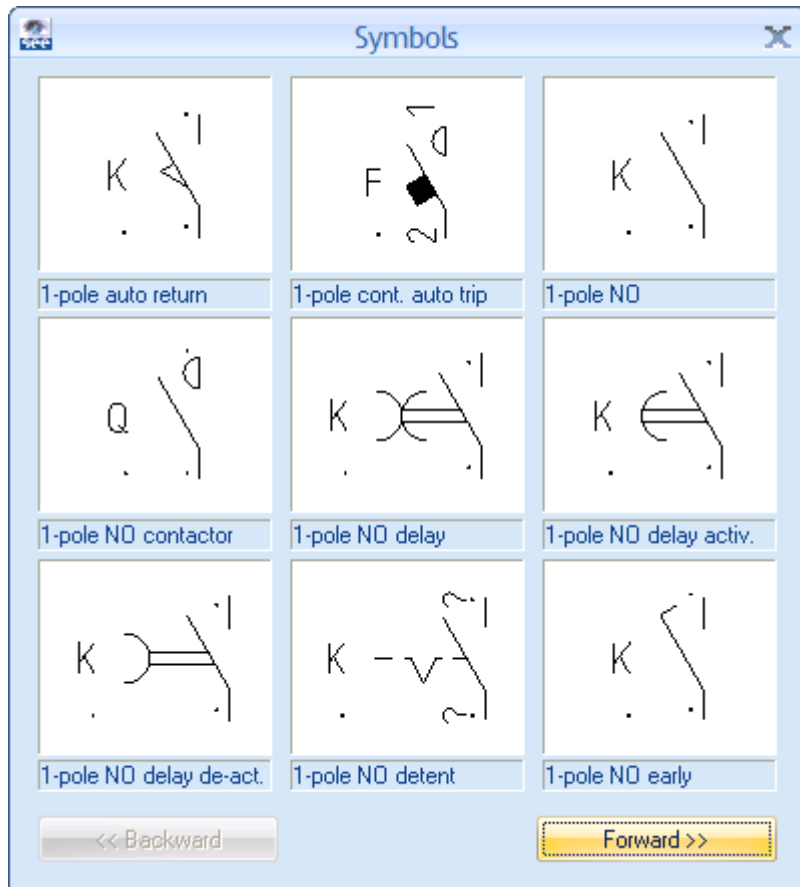
COPYRIGHT © 2013 IGE+XAO. All rights reserved

1. Move the cursor to the Symbols tree.
2. Close the "Switchgear one pole" symbol folder by clicking the minus sign left to the folder name.
3. Double-click the "Relay-contacts NO" symbol folder.
4. Click the "1-pole NO" symbol
5. Move the cursor to the drawing area.
The symbol appears attached to the cursor.
- 6.+ "Drop" the symbol at the desired position in the drawing sheet, in this case column 2.
- 7.> Product (-)
Type the name of the component which the contact must be assigned to.
- 8.# 1Q2
Click the  button in the "**Product (-)**" field.
The **Function Location Product** window appears listing the available contacts.
Choose the contact from this list.
- 9.> Connection 00
- 10.# 13
Type the contact number.
- 11.> Connection 01
- 12.# 14
Type the contact number.
- 13.> **OK**
Close the dialogue box.

Hint:

You can also select symbols by using the graphical overview.

- Right-click the symbol folder within the Symbols tree, where the symbol is located - in our example **Relay-contacts NO**.
- Select the **Graphical Overview** pop-up command.

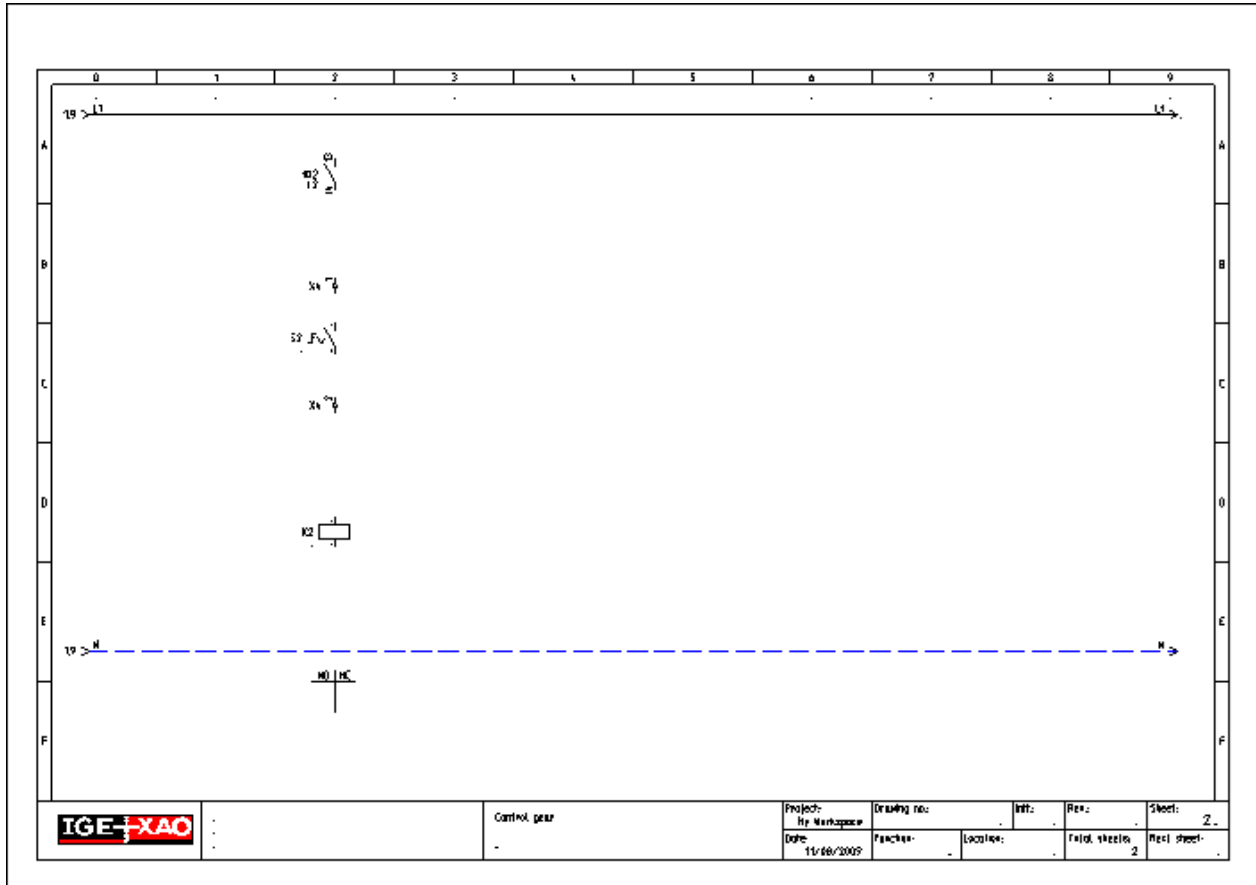


- Scroll forward or backward through the symbols by clicking the "<<**Backward**" and the "**Forward**>>" buttons.
- Click a symbol to select it.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

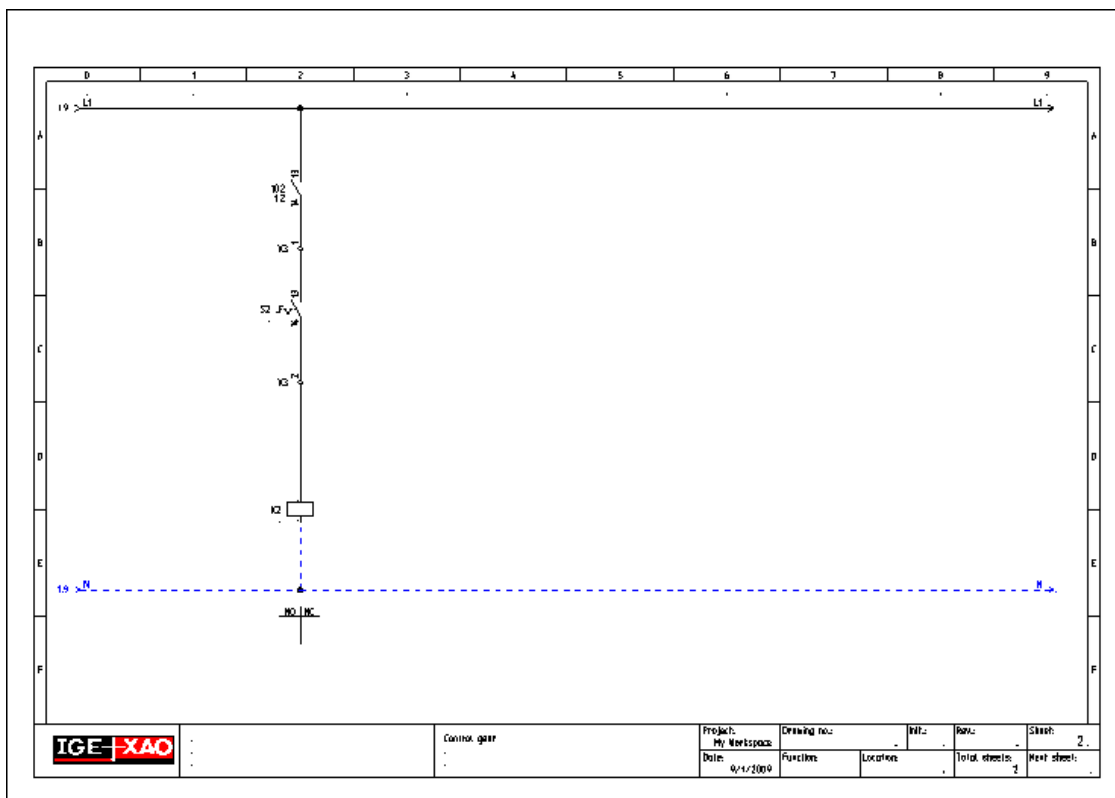
Exercise 4-8: Insert the terminals.



1. Move the cursor to the Symbols tree.
2. Close the "Relay-contacts NO" symbol folder by clicking the minus sign to the left of the folder name.
3. Double-click the "Terminals" symbol folder.
4. Click the **1 terminal 90° vertical** component.
5. Move the cursor to the drawing area.
The symbol appears attached to the cursor.
- 6.+ "Drop" the symbol at the desired position in the drawing sheet, in this case above the switch gear symbol in column 2.
The terminal name is asked.
- 7.> Product (-)
- 8.# X3
Type in the terminal name.
- 9.> Terminal number
- 10.# 1
Fill in terminal number 1.
- 11.> Terminal Index
- 12.# 1
The terminal index is used for sorting the terminals in the terminals list in order to insert terminals PE or N in the right place in the terminals list.
- 13.> Type
- 14.# 039061
Type in the type of the terminal.

- 15.> **OK**
Close the dialogue box.
- 16.+ Place the next terminal under the switchgear symbol in column 2.
The terminal name is asked.
- 17.> Product (-)
- 18.# X3
The component name, terminal number and index are suggested. Accept them.
- 19.> Type
- 20.# 039061
Fill in the type of the terminal.
- 21.> **OK**
The box closes.
Right-click to exit the insertion mode.

Exercise 4-9: Draw the wire.



- 1.CA **Electrical**
- 2.CO **1 Wire (Wires Connections panel)**
- 3.+ Select the starting point for the wire on the potential L1 above the symbols.
- 4.+ Select the second point for the wire on the potential N below the symbols.
The wire is drawn and automatically broken where the symbols are inserted.
5. Right-click to exit drawing mode.

Exercise 4-10: Change the line style and the colour of the wire between the relay coil and the potential N.

1.+ Right-click the wire and select the **Properties** pop-up command.

The wire properties appear in the **Properties** pane in the right part of the main *SEE Electrical* window.

2.> Penstyle

3.> Select the line "Dash" style.

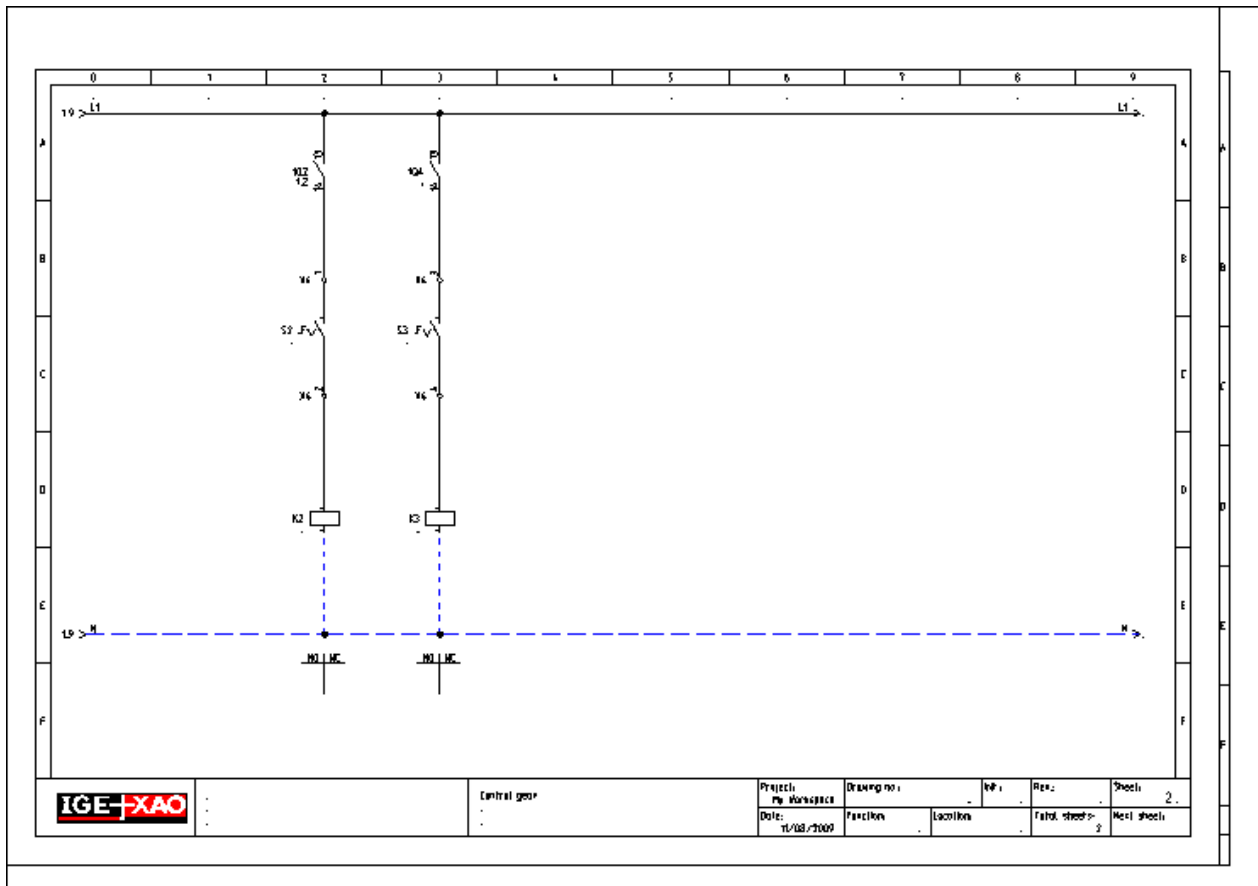
4.> Pencil colour

5.> Select blue colour.

The colour and the line style of the wire are changed dynamically on the screen.

D.3. COPYING EXISTING ELEMENTS ON PAGE 2

Exercise 4-11: Copy the drawn column.



1.+ The **Edit** ➤ **Select** ➤ **Normal** function is active in case the cursor appears as an arrow .

You can a single element or all of the symbols to be copied with a frame. The frame is defined as a rectangle by marking two opposite corners.

In this exercise you will use a frame.

Two ways exist for selecting part of the drawing by a frame:

If you wish to process only elements that are located entirely within the area, move the cursor from left to right:

The cursor graphic becomes: 

If you wish to process all objects that are even partly included in the area, move the cursor from the right to the left:

The cursor graphic becomes: 

Hint:

If you wish to select elements in drawings with a lot of content, it is possible that you cannot place the first corner point of the frame, since directly an element is selected.

The erroneous selection of an individual element can be avoided as follows:

- *Press the **W** key on the keyboard and keep it pressed, while you click the first corner point of the rectangle using the left mouse button.*
-

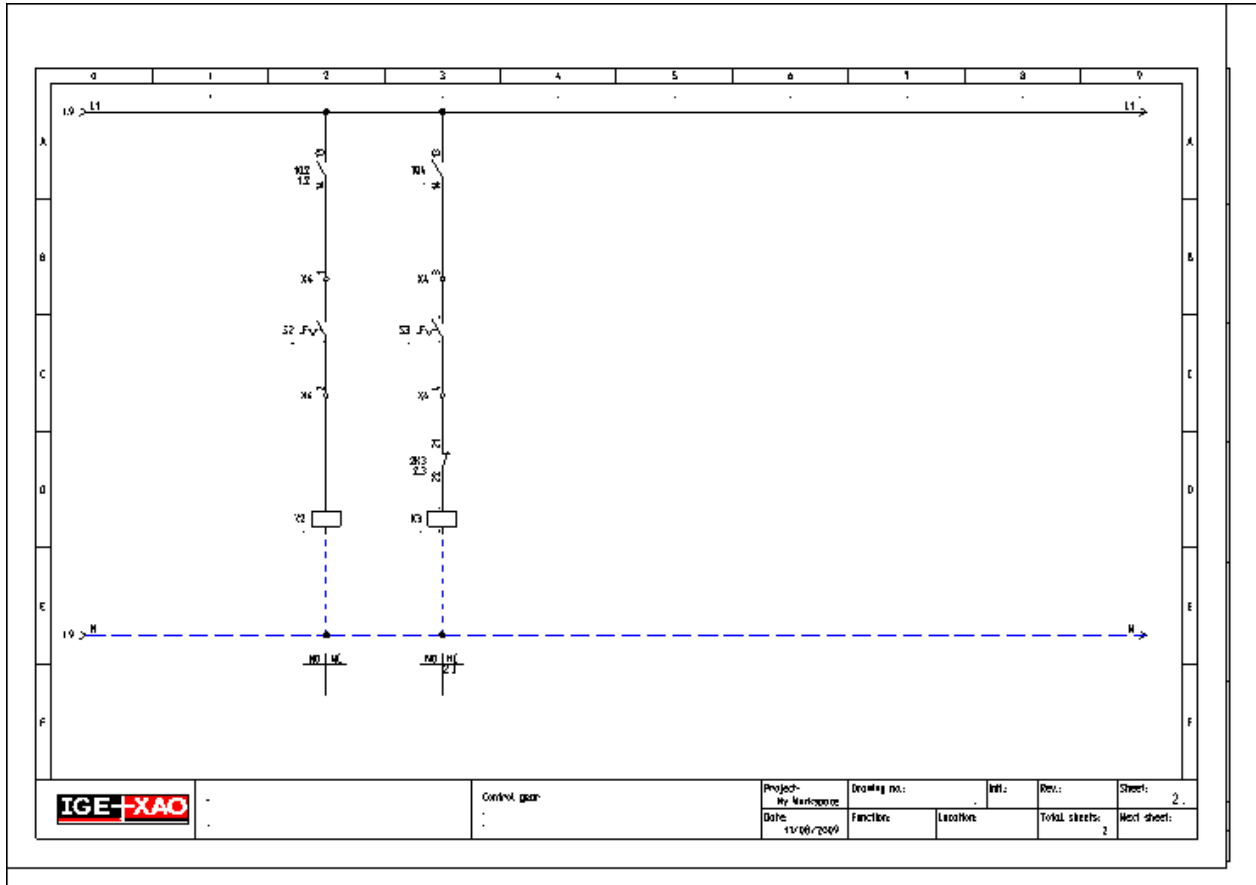
- 2.+ Press the left mouse button to define the first corner of the frame left above in the just now drawn column 2.
The symbols are selected, if they are located completely in the frame.
Hold down the left mouse button while dragging to define the second corner in the next step.
- 3.+ Select the second frame corner in the right bottom of the just now drawn column. All selected components and wires have been highlighted.
4. After you have selected the column, place the cursor near the top left node. This point has to be placed after copying. Press and hold down CTRL, press the left mouse button, and move the mouse. A copy of the column has been created and can be inserted in the desired position - in column 3. (If you do not press the CTRL key, the selected components are moved.)
- 5.+ Insert the copy in column 3 and release the CTRL key.
The sequence of the dialogue boxes for the components depends on the sequence, in which the components have been inserted or moved.
The terminal name is asked.
- 6.> Product (-)
X3 is suggested by default. Type in terminal number 3 and terminal index 3, if they do not appear automatically.
- 7.> **OK**
The name for the normally open contact is asked.
- 8.> Product (-)
Fill in the name of the contact NO.
- 9.# K4
- 10.> **OK**
Close the dialogue box.

Exercise 4-12: Deselect all the selected components.

You can unselect all components by clicking at a position on the page where no object is located.

- 1.CA **General**
- 2.CO **Deselect All (Select panel)**

Exercise 4-13: Insert the "1-pole NC" symbol. Place it on the existing wire in column 3.

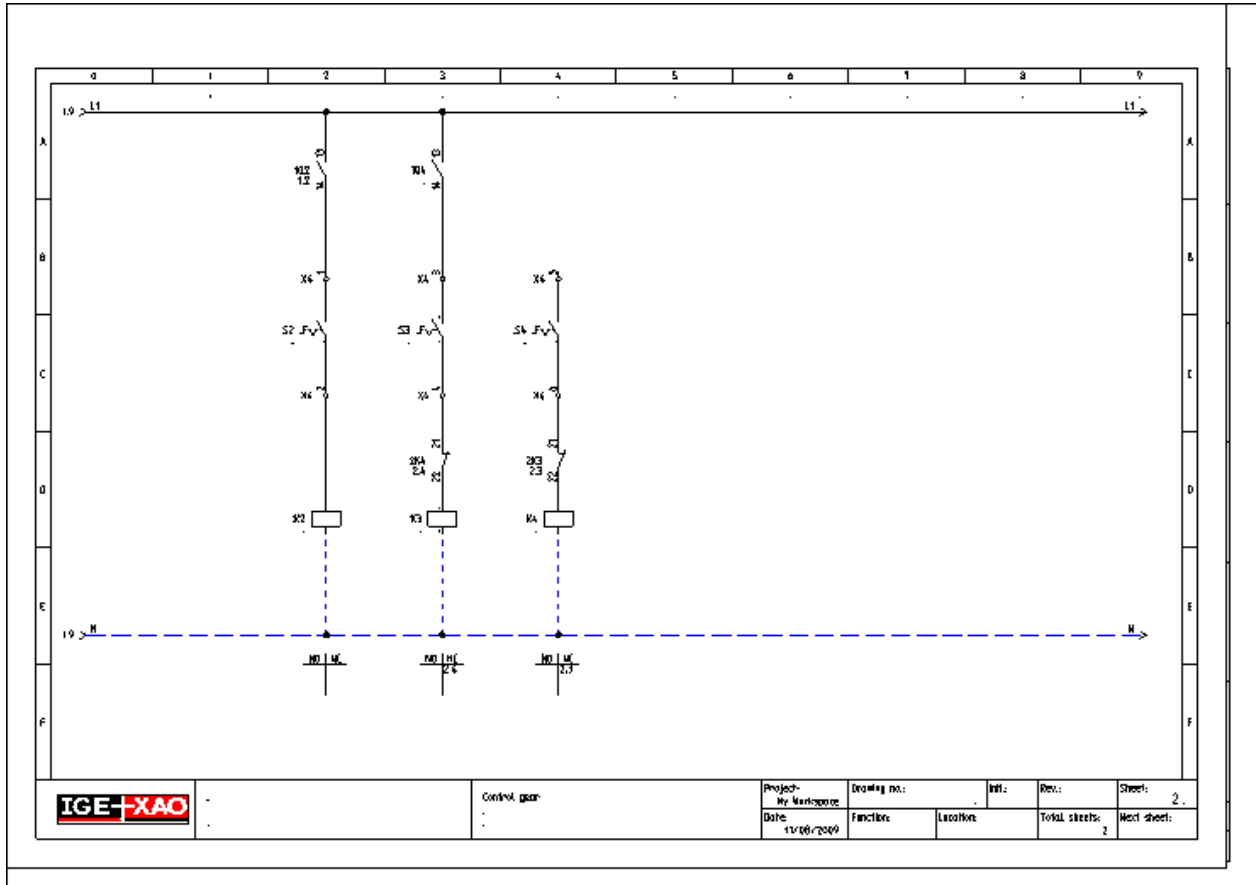


1. Move the cursor to the *Symbols* tree.
2. Double-click the "Relay-contacts NC" symbol folder to open it.
3. Click the "1-pole NC" symbol.
4. Move the cursor to the drawing area.
The symbol appears attached to the cursor.
- 5.+ "Drop" the symbol at the desired position in the drawing sheet, in this case above the relay coil in column 3.
The connection has been broken.
The dialogue box for the **1-pole NC relay contact** appears.
- 6.> Product (-)
- 7.# K4
- 8.> Connection 00
- 9.# 21
Insert contact number.
- 10.> Connection 01
- 11.# 22
Insert contact number.
- 12.> **OK**
13. Right-click to exit insertion mode.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

Exercise 4-14: Copy components and wires needed for column 4.



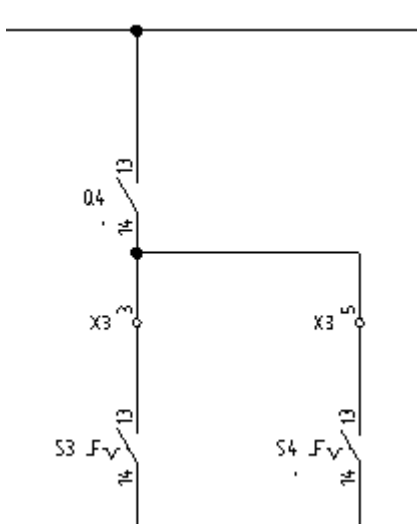
- 1.+ Define the first frame point so that the top terminal is located within the frame (in column 3).
- 2.+ Define the second frame point right below the group in column 3. All of the components within the frame are highlighted.
3. After you have selected the required components and wires, click onto the top connection of the relay coil. Press and hold down the CTRL key, press the left mouse button and move the mouse. "Drag" the copy of the group to the desired place in column 4.
- 4.+ "Drop" the copy and release the CTRL key.
The dialogue boxes for the terminals and the contact appear.
The sequence of the dialogue boxes for the components depends on the sequence in which the components have been inserted or moved.
The terminal name is asked.
- 5.> Product (-)
X3 is suggested by default.
Insert the terminal number 5.
Insert the same value for the terminal index.
For the **standard** and higher levels of **SEE Electrical**, at the end of the **Terminal number** line and **Terminal Index** line, the icon **+1** appears. This function allows always finding out the highest value for terminal number + 1 in the terminal strip, the same function is available for the terminal index.
- 6.> **OK**
The name for the **1-pole NC** contact is asked.

- 7.> Product (-) K4
Click the **Db** button in the "**Product (-)**" field.
The **Function Location Product** window appears listing the available relay coils.
- 8.> K3
Select the desired relay coil from the list.
- 9.> **OK**

Exercise 4-15: Deselect all of the selected components.

- 1.+ Click the left mouse button in an empty area of the window.

Exercise 4-16: Draw the missing wire.

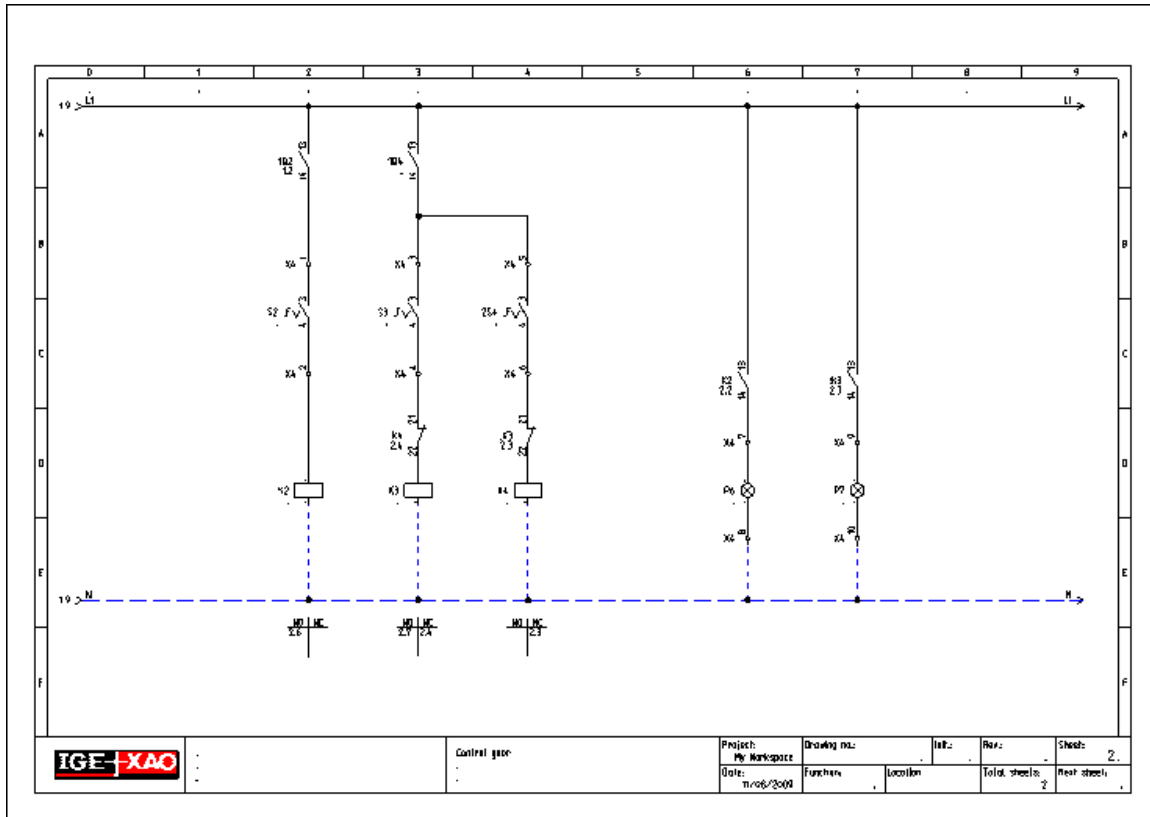


- 1.CA **Electrical**
- 2.CO **1 Wire (Wires Connections panel)**
- 3.+ Place the first point of the wire on the existing vertical connection.
- 4.+ Place the corner point of the wire.
- 5.+ Place the end point of the wire onto the terminal.
6. Right-click to exit wire insertion mode.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

Exercise 4-17: Draw the first missing column and then copy it to create the second missing column.



Insert again terminals, a normally open contact, and a lamp from the symbol database. You can find the lamp in the *Lamps* folder. Right-click to exit insertion mode.

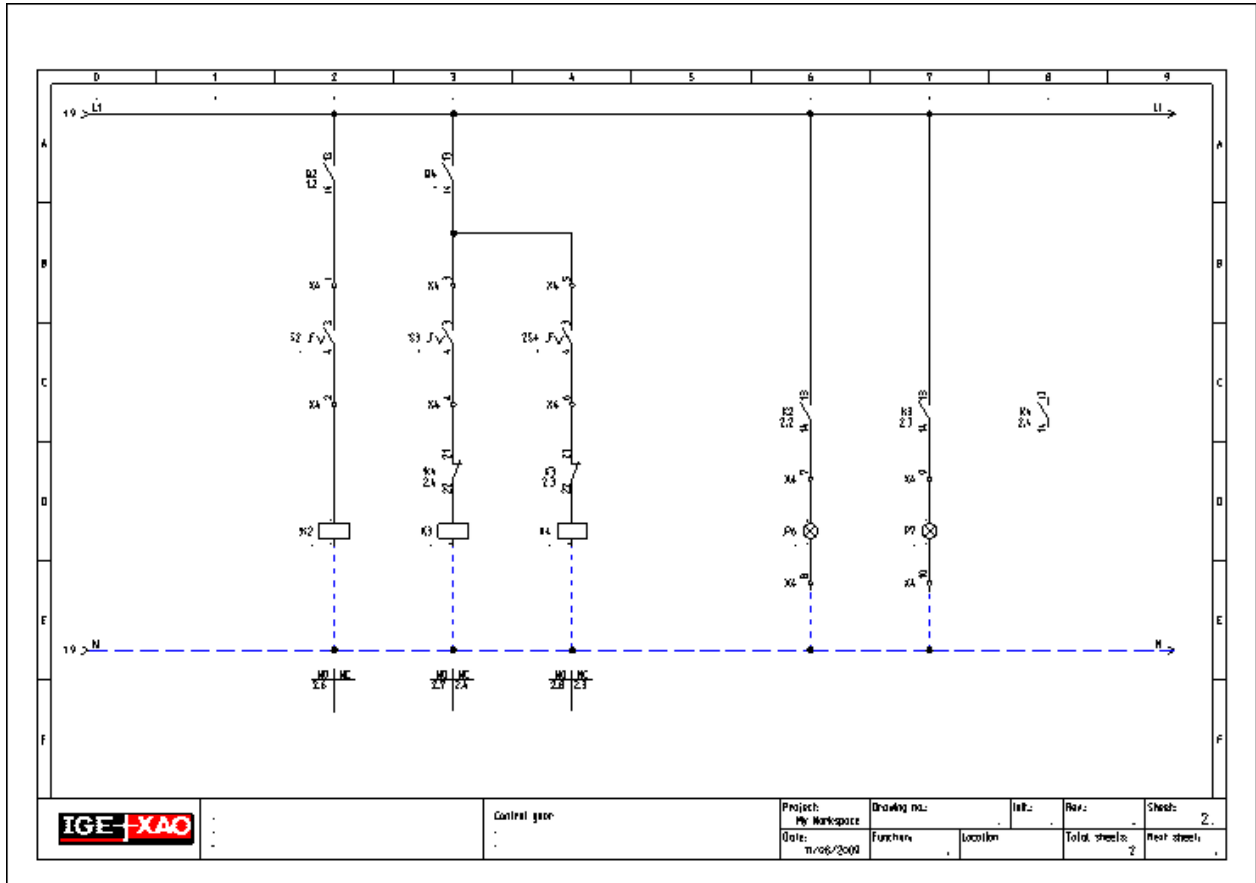
For the **1-pole NO relay-contact**, choose the component name K2 from the list of relay coils and type in the numbers 13 and 14. The terminals receive the suggested names and the type 036091.

Draw the wire by executing the **Electrical > Wire Connections > 1 Wire** command. Right-click to exit wire insertion mode.

- Select the wire between the terminal and the potential N to change its line style and colour.

You can copy the first new column to create the second by using a frame.

- Accept the suggested names for the terminals and fill in K3 as a name for the normally open contact.

Exercise 4-18: Copy the third normally open contact.


- 1.+ Click the contact that you want to copy. If the cursor does not appear as an arrow, click **General > Select > Normal** to activate the selection mode.
The contact has been selected.
2. Press and hold down the CTRL key, press the left mouse button and "drag" the contact to the desired position in column 8.
3. "Drop" the copy there.
The dialogue box for the contact name appears.
- 4.> Product (-)
- 5.> K4
Choose the name from the list in the **Function Location Product** window.
- 6.> OK
Close the dialogue box.
Deselect the contact.
7. Click anywhere within the empty area of the drawing.

Hint

You can insert multiple copies of the copied object.

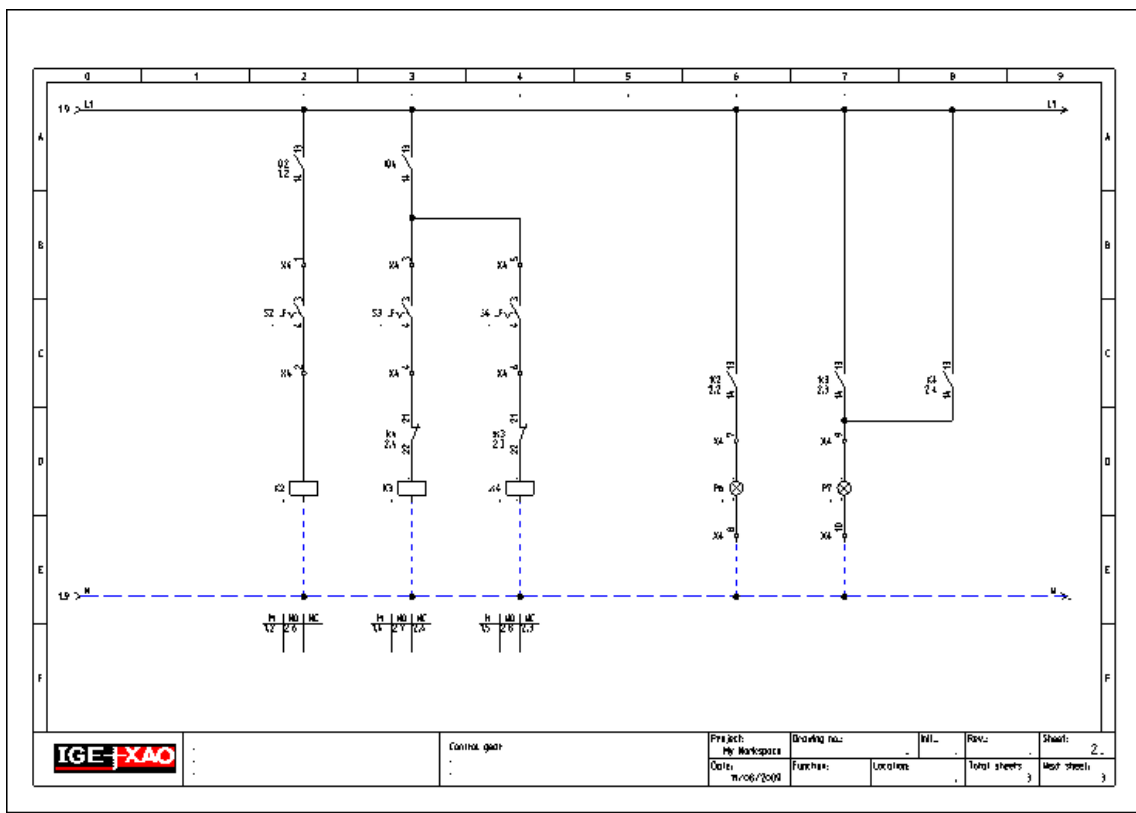
- Select the object you wish to copy.
- Execute the **Copy** pop-up command.
- Select the **Paste** pop-up command.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

- Press the number corresponding to the number of copies needed before you click to position the first copy of the object.
- Click to position the first copy at the desired distance from the original object. The next copy appears at the same distance, but in the next column. The same happens with the other copies. If you type 0, you can create more than nine copies and define the distance between them. Please note that this function is not available for the US version.


Exercise 4-19: Draw the missing wire.



- 1.CA **Electrical**
- 2.CO **1 Wire (Wires Connections panel)**
- 3.+ Select the starting point for the wire on the available wire between the NO contact K3 and the terminal.
- 4.+ Select the corner point for the wire.
- 5.+ Select the end point for the wire on the potential L1.
6. Right-click to exit the drawing mode.

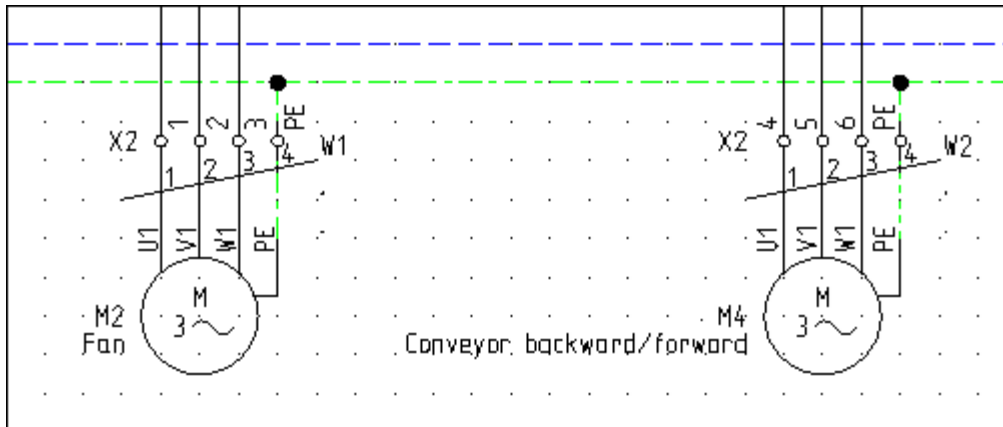
Exercise 4-21: Save the workspace.

- 1.CA **File**
- 2.CO **Save**

Save frequently your workspaces. You can also click on the  icon.

E DRAWING CABLES

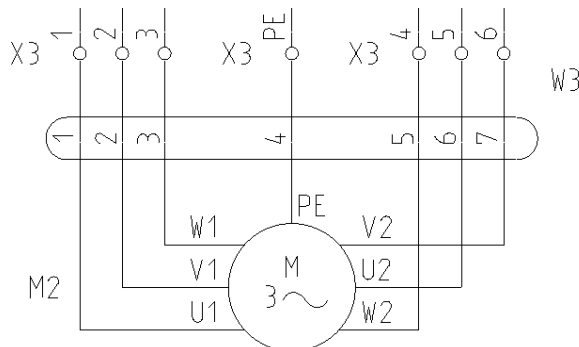
Exercise 5-1: Draw cables W1 and W2 on page 1.



- 1.CA **Electrical**
- 2.CO **Cable (Cable panel)**

Note

In case you have selected the appropriate setting in the **Cables** tab of the **Circuit Diagram Properties** window, when you execute the **Electrical > Cable > Cable** command, a dialogue appears allowing you to insert a cable with previously defined symbols for start, middle and end of the graphics. These user-defined cables are defined via the **Cable Setup** button in the **Cables** tab of the **Circuit Diagrams Properties** window. If you choose an user-defined cable from the dialogue, the cable will be inserted as previously defined. If you use user defined cable symbols, the middle symbol will be automatically enlarged to cover any gap between the wires.

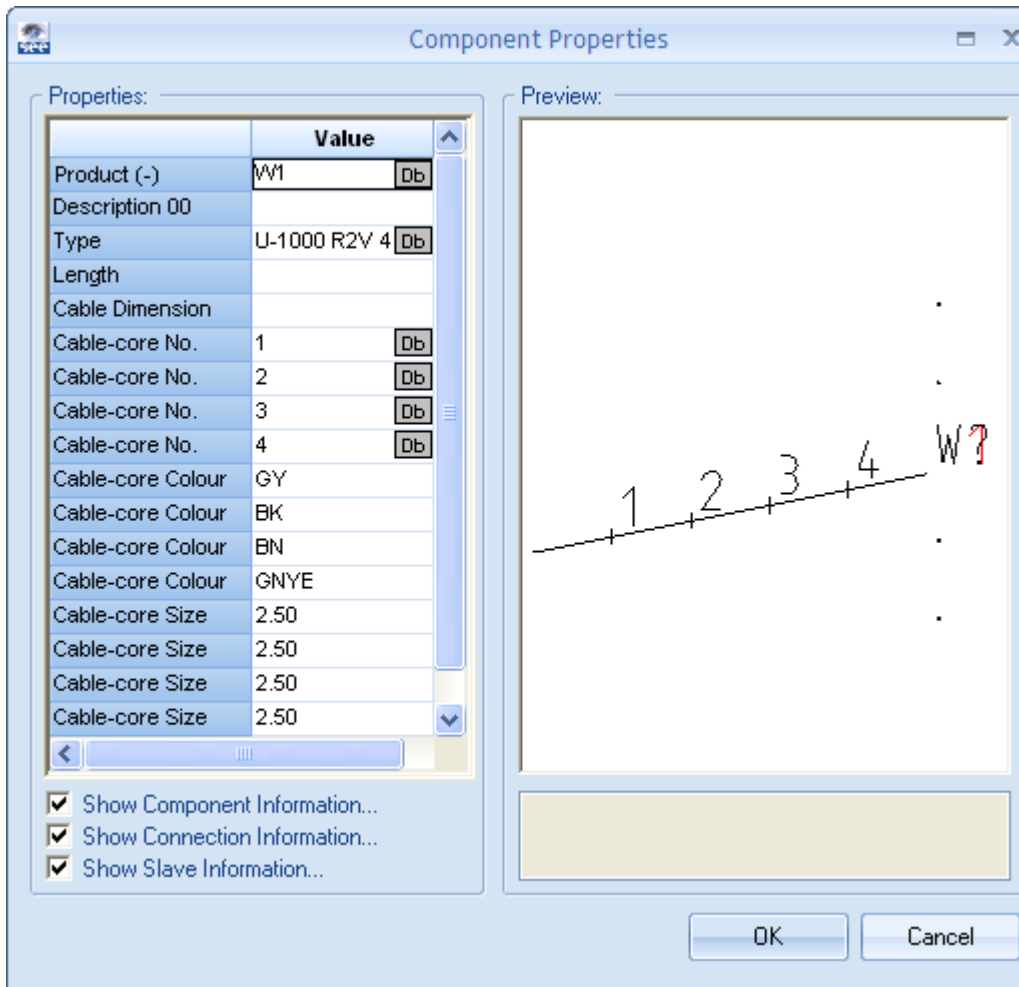


- 3.+ Select a starting point for the cable.
 - 4.+ Select the end point of the cable.
- The **Component Properties** dialogue for the cable appears.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

- 5.# Insert the cable information as desired.
Fill in the information in the dialogue box as shown below.



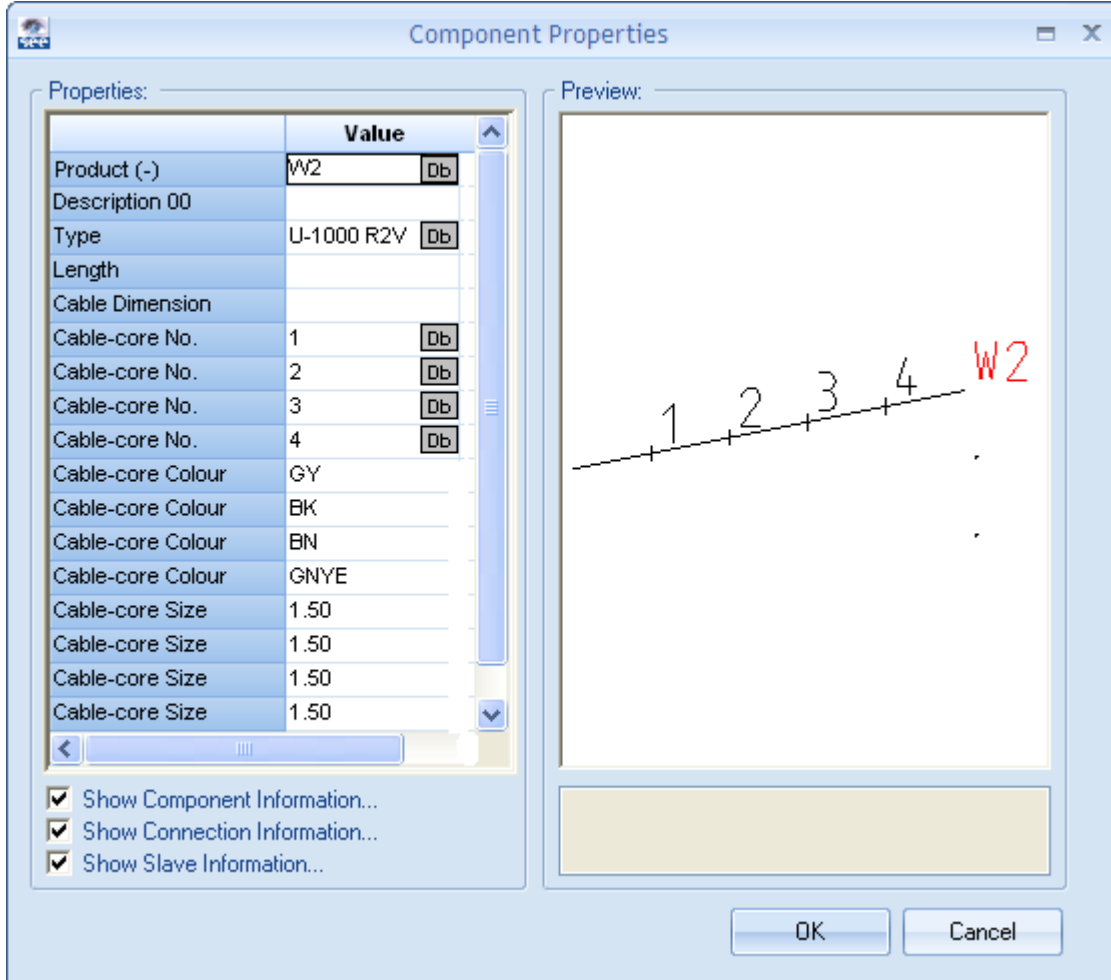
Properties:	Value
Product (-)	W1 Db
Description 00	
Type	U-1000 R2V 4 Db
Length	
Cable Dimension	
Cable-core No.	1 Db
Cable-core No.	2 Db
Cable-core No.	3 Db
Cable-core No.	4 Db
Cable-core Colour	GY
Cable-core Colour	BK
Cable-core Colour	BN
Cable-core Colour	GNYE
Cable-core Size	2.50
Cable-core Size	2.50
Cable-core Size	2.50
Cable-core Size	2.50

☒ Show Component Information...
☒ Show Connection Information...
☒ Show Slave Information...

OK Cancel

- 6.> Product (-)
7.# W1
8.> Type
9.# U-1000 R2V 4G2,5²
The cable core numbers and colours are filled in automatically according to the selected cable type.
Tick the check boxes in the **Show** column to define the data you wish to display on the drawing.
10.> **OK**
Draw the cable W2 by using the same approach.

Fill in the cable information as shown below and click **OK** to close the dialogue box.



Properties:	Value
Product (-)	W2 Db
Description 00	
Type	U-1000 R2V Db
Length	
Cable Dimension	
Cable-core No.	1 Db
Cable-core No.	2 Db
Cable-core No.	3 Db
Cable-core No.	4 Db
Cable-core Colour	GY
Cable-core Colour	BK
Cable-core Colour	BN
Cable-core Colour	GNYE
Cable-core Size	1.50
Cable-core Size	1.50
Cable-core Size	1.50
Cable-core Size	1.50

☒ Show Component Information...
☒ Show Connection Information...
☒ Show Slave Information...

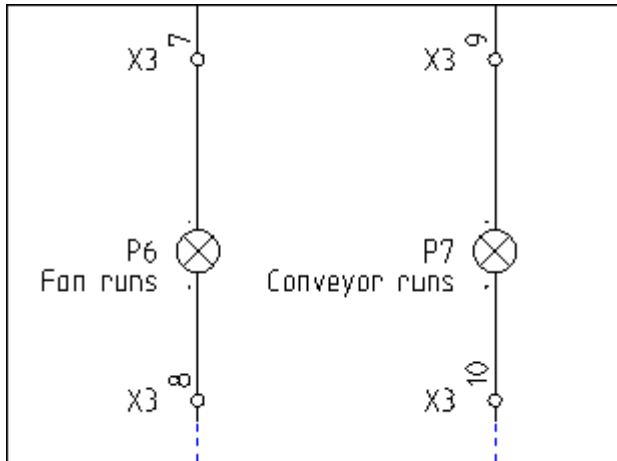
OK Cancel

Note

If you want to toggle on/off the visibility for all cable core texts, you can press the SHIFT key on your keyboard and then check/uncheck the visibility for the first cable-core text. The texts for the other cable cores are automatically switched on/off.


F ADDITIONAL PROCESSING OF CIRCUIT DIAGRAMS



Exercise 6-1: Insert description for the two lamps on page 2.




- 1.+ Double-click the left lamp.
The **Component Properties** dialogue appears where you can add a description text.
- 2.> Description
- 3.# Fan runs
- 4.> **OK**
Close the dialogue box.
- 5.+ Double-click the second lamp.
- 6.> Description
- 7.# Conveyor runs
- 8.> **OK**

Exercise 6-2: Name the contactors on page 1.

- Double-click page 1 in the **Workspace** pane to open it. 1. You can also click the  icon or press the Page Up key on the keyboard to go to the previous page.
- Double-click the first contactor on column 2.
Its **Component properties** dialogue box appears.

- 1.> Product (-)
- 2.# K2
Click the  button in the "**Product (-)**" field.
The **Function Location Product** window appears listing the available contacts.
Choose the K2 relay coil from this list.
- 3.> **OK**
Close the dialogue.
Double-click the second contactor on column 4.
Its **Component properties** dialogue box appears.
- 4.> Product (-)
- 5.# K3
Click the  button in the "**Product (-)**" field.

The **Function Location Product** window appears listing the available contacts.
Choose the K3 relay coil from this list.


- 6.> **OK**
Close the dialogue.
Double-click the third contactor on column 5.
Its **Component properties** dialogue box appears.
- 7.> Product (-)
- 8.# K4
Click the  button in the "**Product (-)**" field.
The **Function Location Product** window appears listing the available contacts.
Choose the K4 relay coil from this list.
- 9.> **OK**
Close the dialogue.
The cross references appear automatically under the contactors.

Hint

You can navigate between components and pages by clicking the cross-reference symbols. The corresponding page is open and the component is shown by a red pin.

Exercise 6-3: Look at the potentials on page 1. At the potentials L1 and N, cross-references to page 2 have been created.

Exercise 6-4: Switch to page 2. Look at the page. At the relay coils, cross-references to the contacts on page 1 have been created.

1. To switch to the next page, click on the icon  or press Page Down on the keyboard.

Exercise 6-5: Save the workspace.

- 1.CA **File**
- 2.CO **Save**

F.1. USING CROSS-REFERENCE SYMBOLS

If a wire cannot be drawn on the same page, symbols for cross-references are used. Cross-reference symbols are available in the *References* folder of the *EN61346-2UK* symbol database. Only two cross-references may receive one and the same name because a definite reference is required. The look of the cross-reference symbols is not important. The connection of two cross-reference symbols is possible via the component name.

Exercise 6-6: Insert two cross-references in your example project.

- Switch to page 1.
- Open one symbol library *EN61346-2UK*. Open the *References* symbol folder and choose "Reference (right)".
- Insert it and type the component name, for example V1.
- Switch to page 2. Select the "Reference (left)" symbol from the symbol libraries *EN61346-2UK*.
- Place it and type the name V1 again.

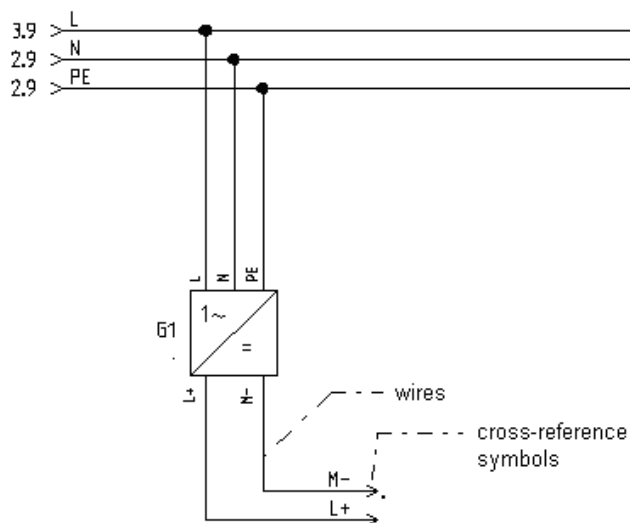
At both cross-references, a back-reference to the corresponding reference is created automatically.

You can generate the cross-reference symbols using the **Electrical > Wire connections > 1 Wire** command.

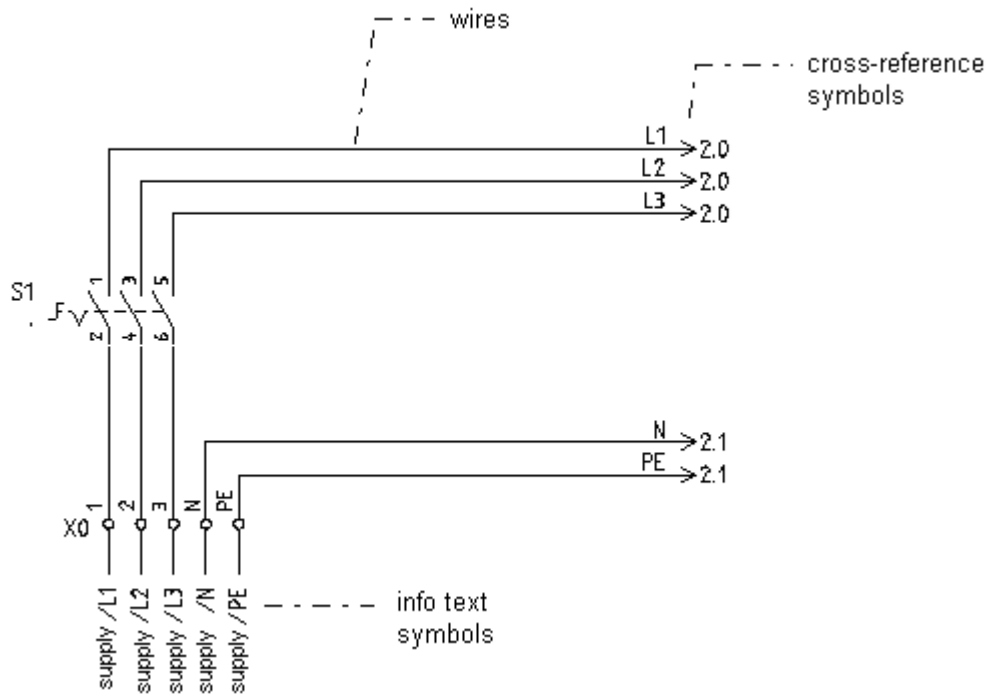
To insert cross-reference symbols while drawing a wire, double-click at the end of the wire. The cross-reference symbol appears automatically.

Exercise 6-7: A cross-reference can also be used for the Power supply, if you do not work with Wires list and Wiring list. If you do so, use potential lines for the power supply.

The potentials on the next pages are created via the functions for drawing a potential.



Or

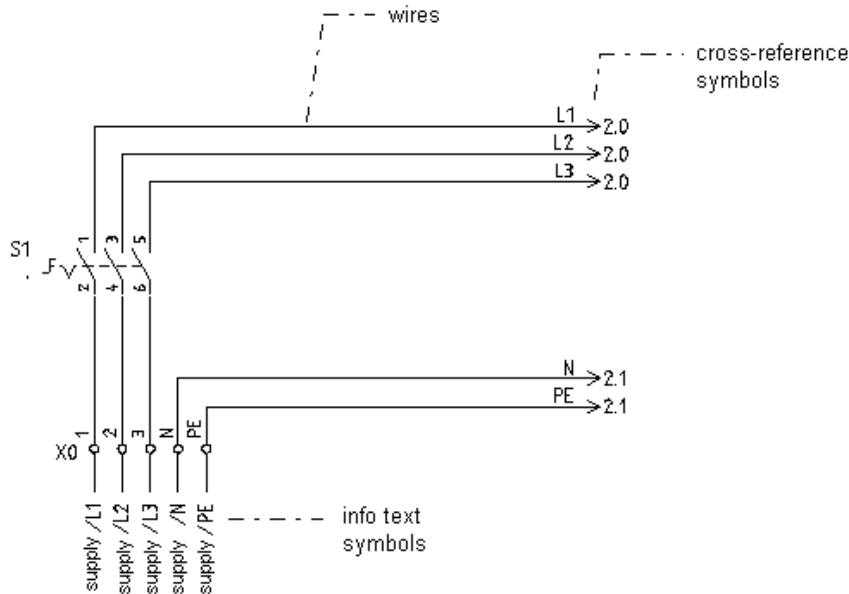


Exercise 6-9: Save the workspace.

- 1.CA **File**
- 2.CO **Save**

F.2. USING INFO TEXT SYMBOLS

If wires begin or end as shown below, i.e. if your responsibility breaks at the end of a wire, this wire has no target for *SEE Electrical*. The terminals of the terminal strip X0 have no target either.



However, if you insert an info text symbol (it consists of graphics, a connection point, and a component name) at the end of the wire, the wire, and consequently the terminal, has a target again. Info texts are available in the *EN61346-2UK* symbol library.

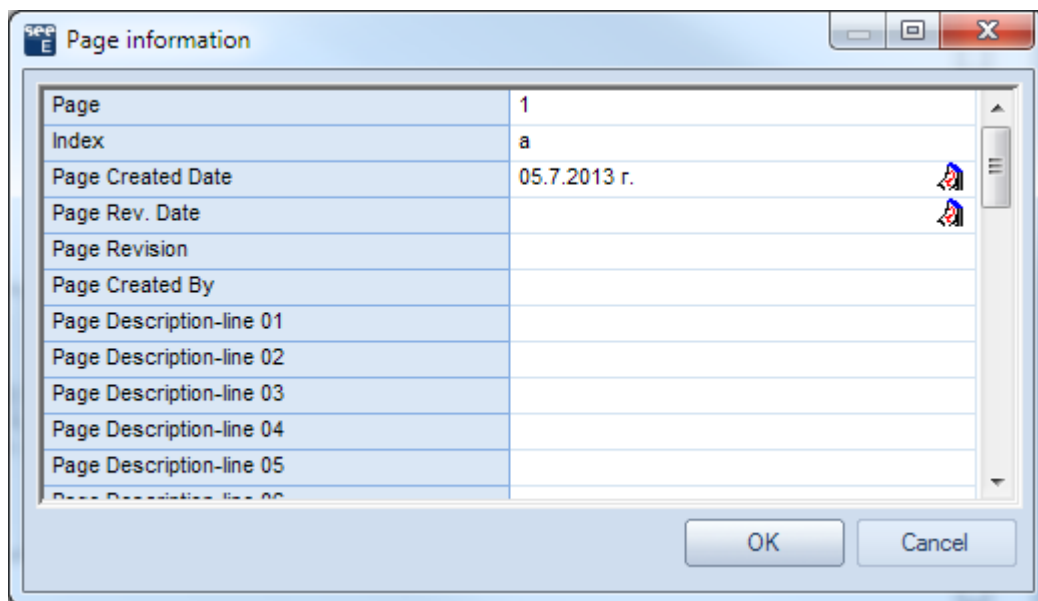
F.3. PAGE INDEX

If an installation has just been built, it is often necessary to add pages in order to have additional circuit parts in the project. If a page numbering for the component names is used, then the names of the components which are already installed must not be changed.

The page index allows inserting pages without changing the numbers of the existing ones.

Exercise 6-9: Insert page 1a in the workspace.

- Create a new page using the same approach as for the creation of page 2.



Page	1
Index	a
Page Created Date	05.7.2013 r.
Page Rev. Date	
Page Revision	
Page Created By	
Page Description-line 01	
Page Description-line 02	
Page Description-line 03	
Page Description-line 04	
Page Description-line 05	

- 1.> Page
Change the page number.
- 2.# 1
- 3.> Index
- 4.# a
Type in the page index.
- 5.> **OK**

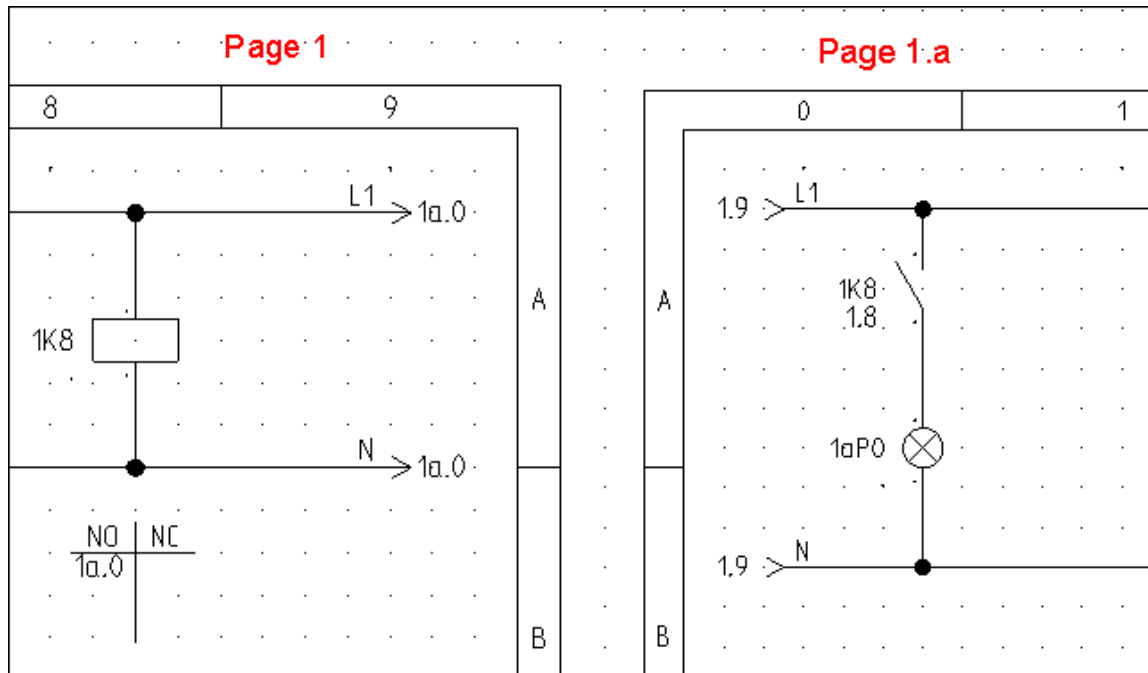
The page has been created.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

Hint:

Place a symbol of a lamp on page 1a. If the workspace is created using Page Number, it is automatically named 1aP0. When you draw the potential L1 on page 1a, the cross-references on pages 1 and 2 will be updated. The same happens to the cross references for coils and contacts. They will also be updated using the information of the page index.

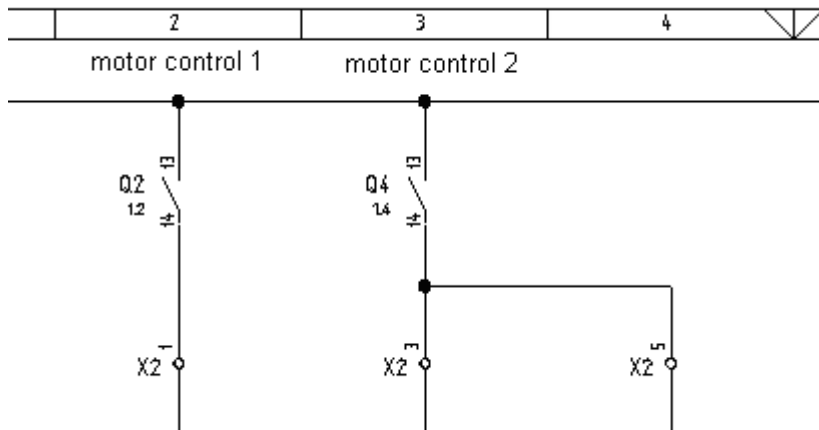


Exercise 6-10: Save the workspace.

- 1.CA **File**
- 2.CO **Save**

F.4. TEXTS




Exercise 6-11: You can insert comment texts in a page. Please insert the texts "Motor control 1" and "Motor control 2" in page 2.

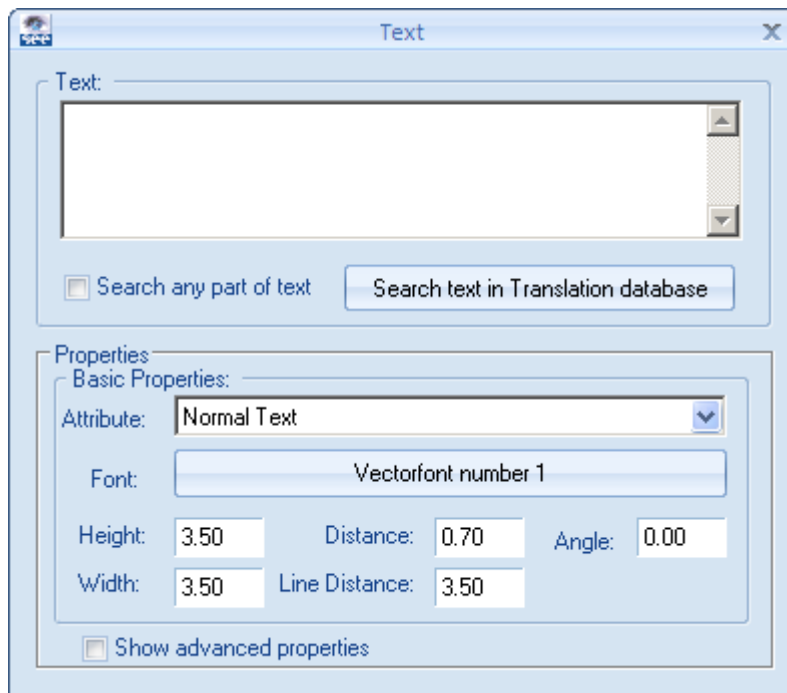


- 1.CA
- 2.CO

Draw

New Text (Elements panel)

You can click the  icon, too (the  icon allows you to create a new text, the  icon allows you to edit existing texts).

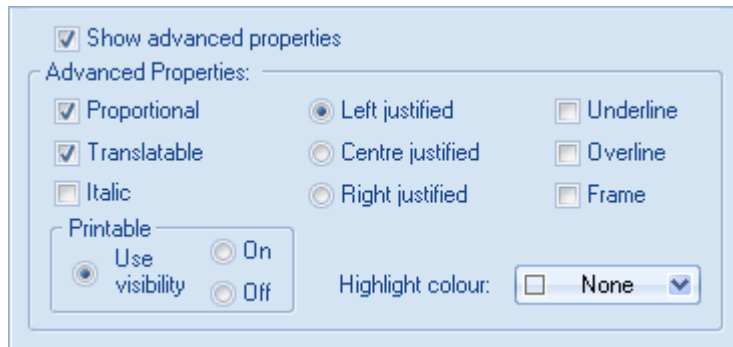



- 3.+ Move the cursor into the "**Text**" field.
- 4.# Motor control 1
Type in the text.

Training manual


COPYRIGHT © 2013 IGE+XAO. All rights reserved

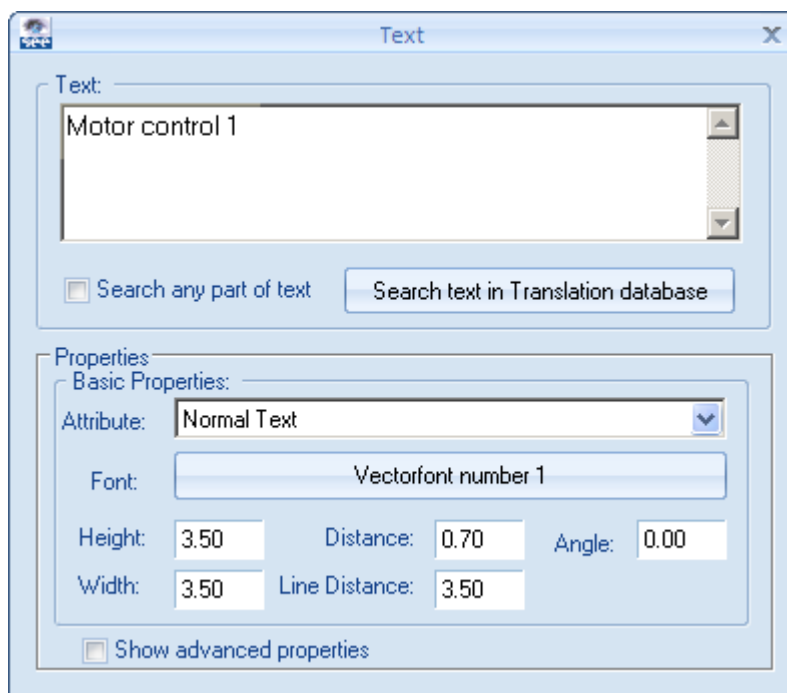
- 5.> Tick the "Show advanced properties" check box and select the desired text attributes, such as size, highlight colour and adjustment (Left justified, or Centre justified).




- 6.+ Insert the text in the drawing by clicking at the desired position. The **Text** dialogue box remains open.
- 7.+ Move the cursor into the "**Text**" field again.
8. Change the existing text or type in a new text, place the text in the drawing, etc.
- 9.> Click the  button to close the **Text** dialogue box.

Exercise 6-12: Change the text you just inserted.

- 1.CA **Edit**
- 2.CO **Edit Text** (**Text** panel)
- You can also click the  icon.
- 3.M Click the text you want to change.



- 4.+ Move the cursor into the "**Text**" field.

- 5.# <Text>
Change the text "Motor control 1" to "Motor 1".
The change can be seen directly in the drawing.
- 6.> If you want, tick the "**Show advanced properties**" check box and change the desired text attributes, such as size, highlight colour and adjustment (Left justified, or Centre justified).
- 7.+ Click the next text you want to change: "Motor control 2" to "Motor 2" for example.
The **Text** dialogue box remains open.
- 8.+ Move the cursor into the "**Text**" field again.
9. Change the existing text etc.
- 10.> Click the  button to close the **Text** dialogue box.

G **HYPERLINKS**

G.1. MANAGING THE HYPERLINKS

The commands give you the possibility to manage hyperlinks (web addresses or files) within the different pages of the project.

You can add a hyperlink to each graphical object present within the *SEE Electrical* drawings.

It is possible to view which objects have a hyperlink through the **Hyperlink** command in the **View** category.

If you add a hyperlink to a symbol, the link will be stored in the *Symbols* library together with the symbol.

All picture files (BMP, JPG, TIFF, etc.), PDF files or MS Office files can be linked to an object.

Activate the **Define** command and paste the link from your browser (if it is a web address) or the path to the file you want to link to the object into the field.

You can use the **Define** command in case you want to modify the hyperlink.

Click on the hyperlink and activate the **Open** command in order to load the link.

If you want to delete a hyperlink, click it and activate the **Delete** command.

H REDLINING

Redlining objects are graphical remarks inserted on the drawings. They can also be created in the Viewer software.

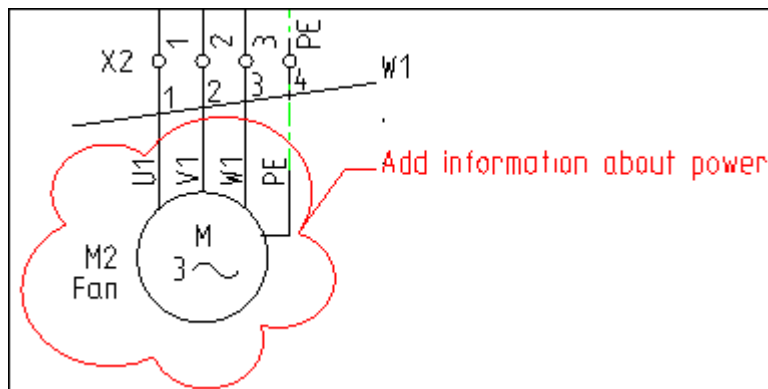
Redlining objects are stored separately in the database so that the original drawing is not changed or damaged.

All redlining objects are drawn in red and are saved in layer 1. They consist of a callout graphics and a callout text, which are processed in a single operation.

Exercise 7-1: Draw a callout around the first motor on page 1.

- Open page 1 of the workspace.

- 1.CA **Redlining**
- 2.CO **Cloud (Draw Callouts** panel)
- 3.+ Click the first point of the cloud outlining the area you want to mark.
The cloud is defined by two diagonally opposite points.
- 4.+ Click the second point of the cloud opposite to the first.
- 5.+ Click once again to mark the point where the callout text is to be positioned.
- 6.+ Type in the desired callout text, in our example "Add information about power".
- 7# Press CTRL+ENTER to start a new line of the text.
- 8.+ Click **OK** to close the window.
- 9.+ Click the left mouse button or press the Enter key to finish drawing
The redlining is displayed on the page:




Right-click to exit callout drawing mode.

Exercise 7-2: Edit and move the inserted callout.


Special commands are available to allow you to edit and move callouts. The conventional commands cannot be used in this case because the callouts are also available in the Viewer software.

- 1.CA **Redlining**
- 2.CO **Edit Text Callouts (Edit Callouts** panel)
- 3.+ Click the callout's text in the drawing to select it.
The **Text** window appears. You can edit the text.

- 4.+ Click the  button to apply the changes.
- 5.CA **Redlining**
- 6.CO **Select Callouts** (**Select Callouts** panel)
- 7.CO **Move Callouts** (**Edit Callouts** panel)
- 8.+ Click the callout you want to move.
- 9. Click on the new position of the callout.

Exercise 7-3: Select and delete the callout you just inserted.

Special commands are available to allow you to select and delete callouts. The conventional commands cannot be used in this case because the callouts are also available in the *Viewer* software.

- 1.CA **Redlining**
- 2.CO **Select Callouts** (**Select Callouts** panel)
-
-
- The cursor changes to .
- 3.+ Click the callout in the drawing to select it.
You can select several callouts by holding down the Shift or the CTRL keys during the selection (standard *Windows* procedure).
- 4.CA **Redlining**
- 5.CO **Delete Callouts** (**Select Callouts** panel)
-
- The callouts are deleted.
- Right-click to exit the callout deletion mode.

I PRINTING

I.1. PRINT

Exercise 8-1: After the project is completed, it can be printed.

- 1.CA **File**
- 2.CO **Print**
- 3.CO **Print Document**

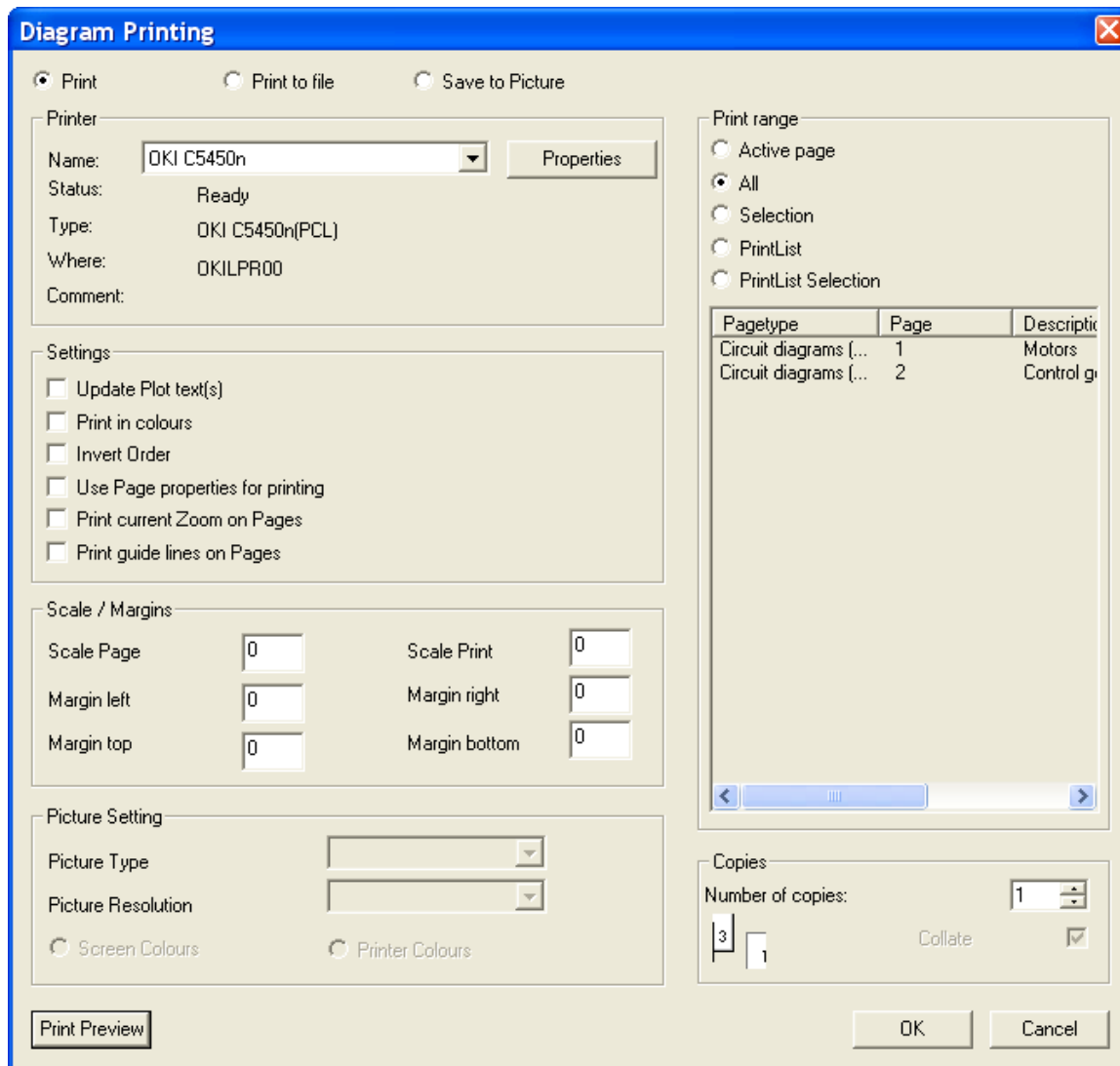


Diagram Printing

☒ Print
 ☐ Print to file
 ☐ Save to Picture

Printer:
 Name: OKI C5450n Properties
 Status: Ready
 Type: OKI C5450n(PCL)
 Where: OKILPR00
 Comment:

Settings:
☐ Update Plot text(s)
☐ Print in colours
☐ Invert Order
☐ Use Page properties for printing
☐ Print current Zoom on Pages
☐ Print guide lines on Pages

Scale / Margins:
 Scale Page: 0 Scale Print: 0
 Margin left: 0 Margin right: 0
 Margin top: 0 Margin bottom: 0

Picture Setting:
 Picture Type:
 Picture Resolution:
☐ Screen Colours ☐ Printer Colours

Print range:
☐ Active page
☒ All
☐ Selection
☐ PrintList
☐ PrintList Selection

Pagetype	Page	Description
Circuit diagrams (...)	1	Motors
Circuit diagrams (...)	2	Control g...

Copies:
 Number of copies: 1
 Collate ☒

Print Preview OK Cancel

A print preview is available for single pages of the project. This function can be accessed by clicking the **Print Preview** button in the **Print** dialogue or by executing the **File > Print > Print Preview** command.

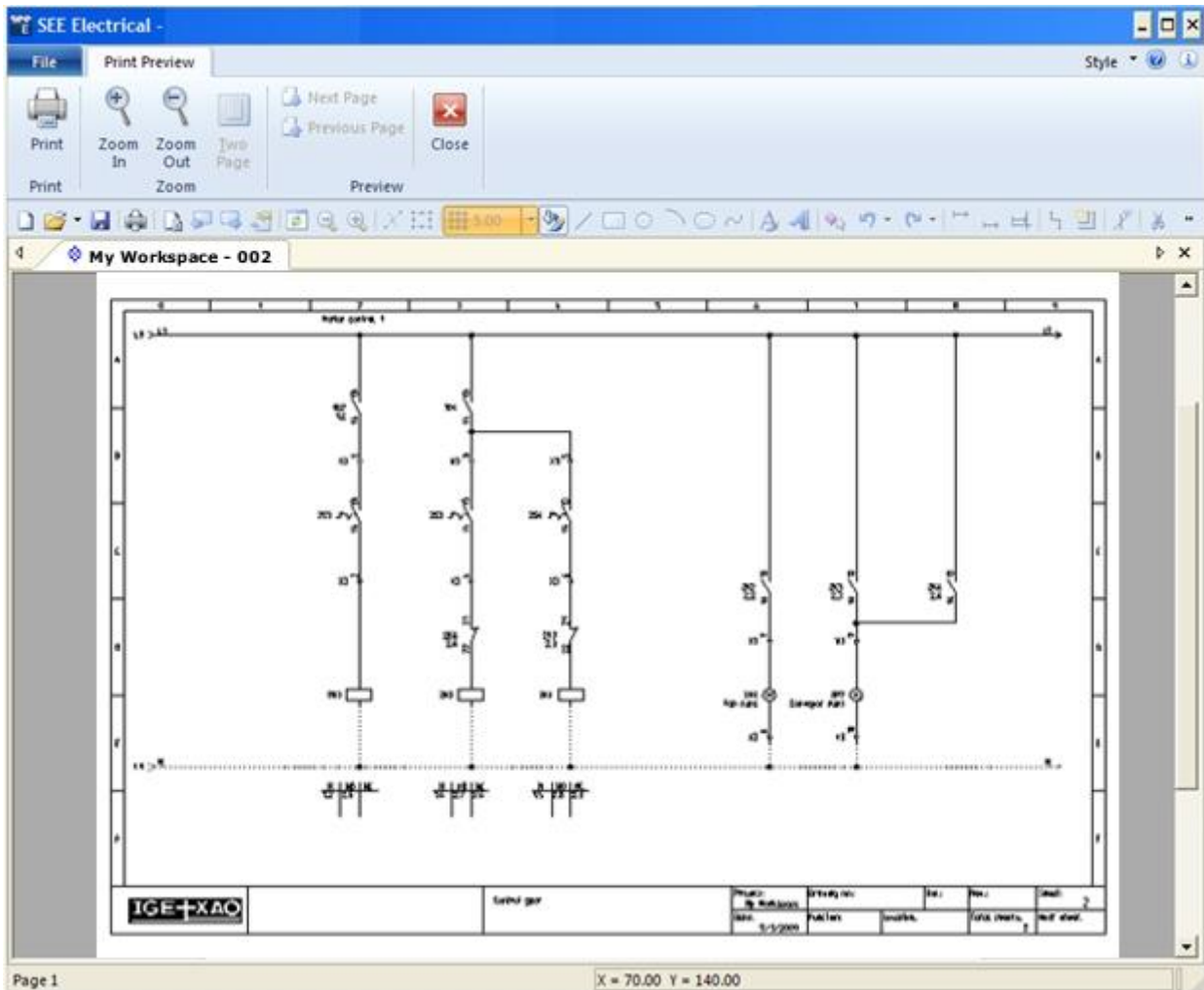
Training manual


COPYRIGHT © 2013 IGE+XAO. All rights reserved

Exercise 8-2: Print Preview

- 1.CA **File**
- 2.CO **Print ► Print Preview**

A print preview of the currently active page appears.



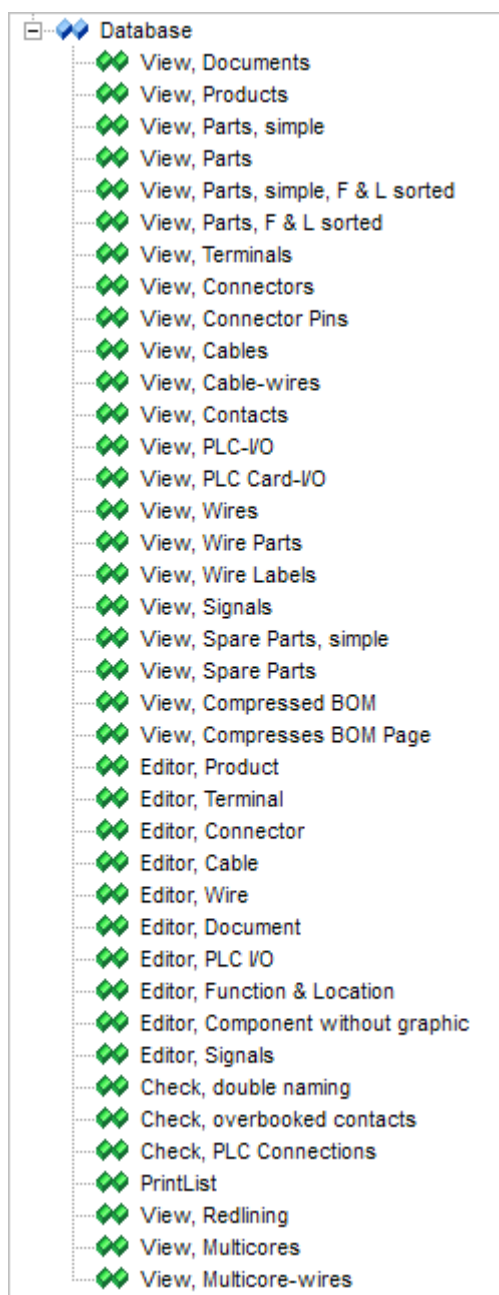
- 3.> Click the  icon to zoom the preview.
- 4.> Click the **Close** button to exit the print preview
or
Press the ESC key on the keyboard.

J DATABASE LISTS

Exercise 9-1: View the workspace database lists.

- Switch from the **Symbols** view to the **Workspace** view by clicking the **Workspace** tab.

The **Workspace** tree opens. According to the level (*basic*, *standard*, or *advanced*), different database lists are displayed.



- Select the desired list, for example **View, Products**.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

1. View, Products
Double-click the list to display its contents in the right pane:

DB My Workspace: View, Products								
	Kind of Document	Page Function (+)	Page	Index	Cell	Function (+)	Location (+)	Product (-)
1	Circuit diagrams (EN)		1		0			1S0
2	Circuit diagrams (EN)		1		2			1M2 Fan
3	Circuit diagrams (EN)		1		2			1Q2 ? AMP
4	Circuit diagrams (EN)		1		4			1F4
5	Circuit diagrams (EN)		1		4			1M4 Conveyor backward/forward
6	Circuit diagrams (EN)		2		2			2S2
7	Circuit diagrams (EN)		2		2			2K2

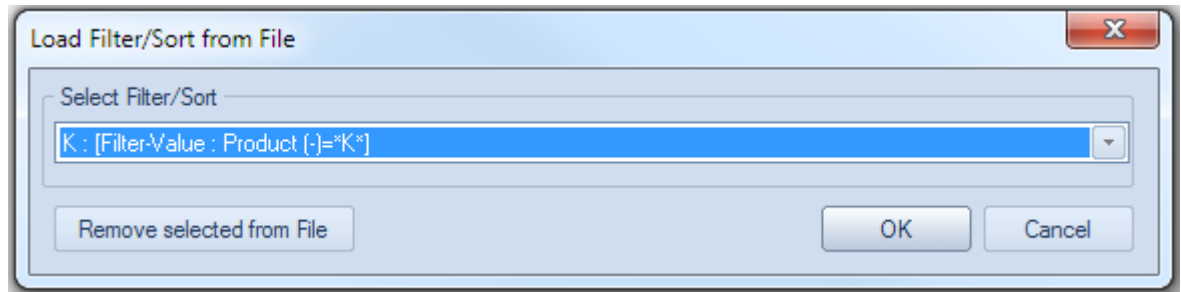
Exercise 9-2: You can sort or filter the information in the list.

- 1.> Select the "**Product**" column
Click the right mouse button.
- 2.M Select the **Sort descending on row** pop-up command.
The components will be sorted in descending order and the components on page 2 (i.e. with 2 as the first character in their names) will be displayed before the components on page 1.
- 3.> Select the "Product" column again.
Click the right mouse button.
- 4.M Click the **Set Filter On -> Product?** pop-up command.
- 5.# *K*
Type in the Filter-Value.
If you are searching for one specific component, you can type its complete name. You may use the wildcards characters ? and * for filtering according to any single character (?) or more characters (*).
If you press ENTER, only the records that satisfy the filter condition will be displayed.
- 6.> Select the "Product" column again.
Click the right mouse button.
- 7.M Select the **Remove Filter/Sort** pop-up command.
All of the records are displayed again.

Exercise 9-3: Creating, storing and loading filters and sorting.

- 1.> Select the "**Product**" column
Click the right mouse button.
- 2.M Click the **Set Filter On -> Product?** pop-up command.
- 3.# *K*
Type in the *Filter-Value*.
- 4.> Select the "**Product**" column again.
Click the right mouse button.
- 5.M Select the **Load Filter/Sort from File** pop-up command.

Select the desired filter/sorting fro the drop-down list in the window that appears:



6.> Click **OK**.

Exercise 9-4: You can save the lists in other files, for example in *MS-Excel* or *MS-Word* files.

1. Launch *MS-Excel* or *MS-Word*.

DB View, Products				
	Kind of Document	Page Function (=)	Page	Index
1	Circuit diagrams (EN)		1	
2	Circuit diagrams (EN)		1	
3	Circuit diagrams (EN)		1	
4	Circuit diagrams (EN)		1	
5	Circuit diagrams (EN)		1	

2..> Click the top left field in the database list as shown above.

The database list is selected.

3.# CTRL + C

The selected area has been copied.

4. Paste the copied data area within *MS-Excel* or *MS-Word* by clicking CTRL + V.

Exercise 9-5: Save the workspace.

- 1.CA **File**
- 2.CO **Save**

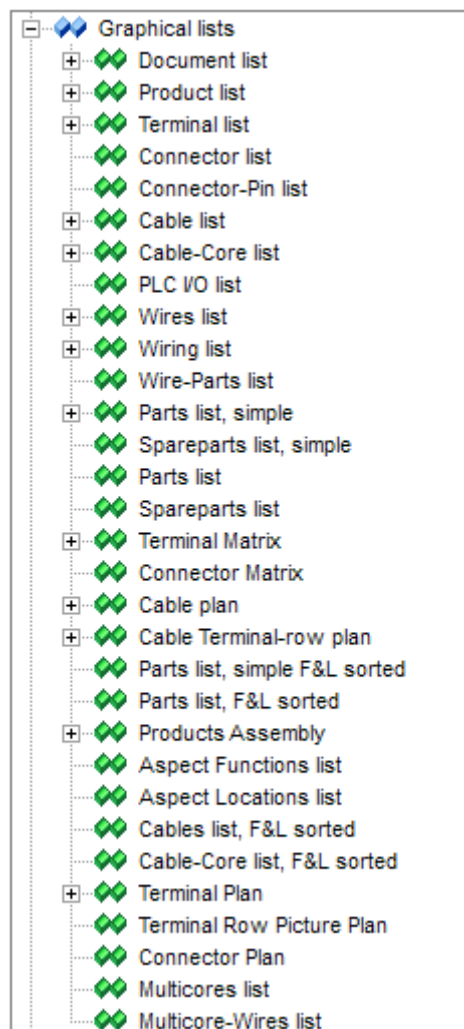
K GRAPHICAL LISTS

K.1. GENERATING A GRAPHICAL LIST

The graphical lists allow filling the project information in forms.

Exercise 10-1: Create a graphical product list for the workspace.

1. Expand the *Graphical Lists* node in the *Workspace* tree.



According to the level and the language version of the software, different **Graphical Lists** are available.

2. Select the **Products** graphical list and click the right mouse button.
- 3.CO **Generate**
The graphical list has been generated.

4. 0001
Click the plus + sign in front of **Products** in the Workspace Explorer and open the graphical list by double-clicking 0001.

Exercise 10-2: Generate other graphical lists using the same approach. Look at the graphical lists afterwards.

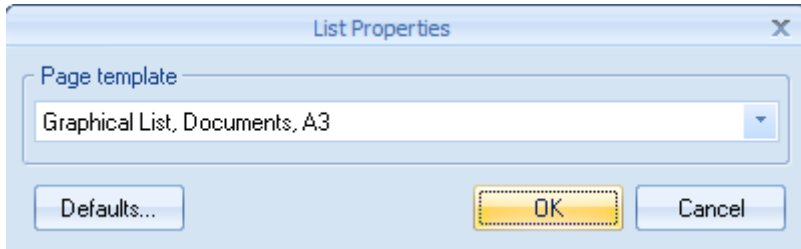
Terminal matrix and Wiring can be generated in *SEE Electrical standard* and *advanced*.
Terminal plan, Terminal row picture plan and Multicores can be generated only in *SEE Electrical advanced*.
Products Assembly can be generated only in *SEE Electrical advanced*.

Exercise 10-3: Save the workspace.

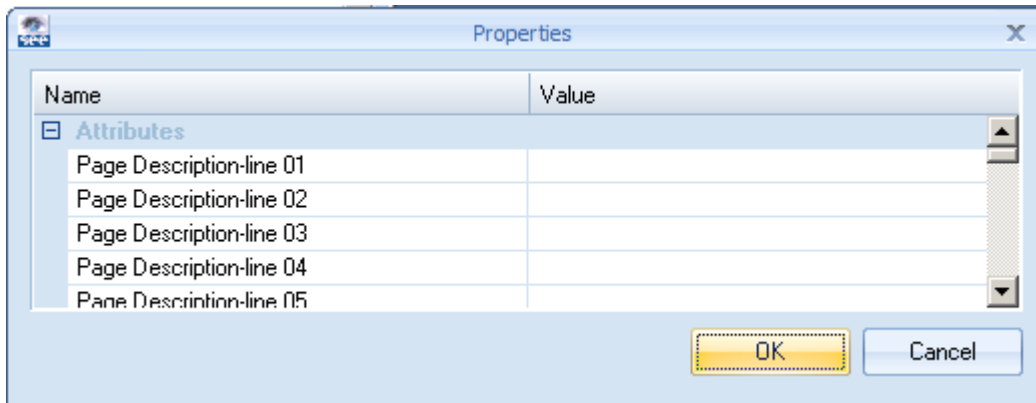
1.CA **File ► Save**

K.2. DEFINE PAGE TEXTS PERMANENTLY

The **Defaults** button is available in the **List Properties** dialogue for all lists.



You can define the texts that will appear in the text attributes of the defined template. The Terminal Matrix, Terminal Plan, Terminal Row Picture use the "Page Description 01" texts.



K.3. GENERATING ALL DESIRED GRAPHICAL LISTS IN ONE STEP

Within *SEE Electrical*, you can generate all desired graphical lists in a single operation.

If you possess the **Basic** or **Standard** levels of *SEE Electrical*, you need specific page templates in order to generate multiple graphical lists in one step.

If you possess the **Advanced** level of the software, a specific functionality is available allowing you to generate the lists.

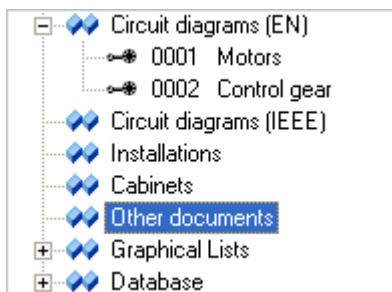
For further details on how to do this, see chapter **Creating Templates for Graphical Lists**.

L INSERTING OTHER DOCUMENTS

You can insert other documents into the workspace, such as *WinWord*-files or *Excel*-spreadsheets. The kind of these documents depends on the *Windows* programs installed on your computer and supporting the *ActiveX* technology. When you wish to view such a document, the respective program will be activated.

Exercise 11-1: Insert a document from another *Windows* application, which supports the *ActiveX* technology, as follows:

1. Select *Other documents* in the *Workspace* tree.



- 2.CA **Home**
- 3.CO **New (Page panel)**
SEE Electrical opens the **Page information** dialogue box.
- 4.> Page Description-line 01
- 5.# Other data
- 6.> Page
 Page number 1 is suggested automatically. Do not change it.
- 7.> Page Created Date
 The current date is suggested.
 You may fill in additional information, if desired.
- 8.> **OK**
 The dialogue box closes.
 The **Insert Object** dialogue box appears, allowing you to choose the other document to be inserted. The dialogue box contains the *Windows* applications installed on your computer which support the *ActiveX* technology.
- 9.> Choose one of the displayed applications.
- 10.> **OK**
 The dialogue box closes. The functions of the selected application are now available for editing the document.
11. Edit your document.

Exercise 11-2: Save the workspace.

- 1.CA **File**
- 2.CO **Save**

M CREATING COMPONENTS

Components for circuit diagrams must be created in accordance with explicit rules in order to identify the connections and to manage the components properly (for example, for a contact to be listed in the Products list).

M.1. CREATING A COMPONENT

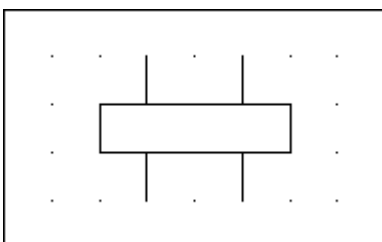
Using the example of the black box, we illustrate how components are created.

Hints:

1. *It is recommended to set a 5 mm grid for the drawing of components for circuit diagrams. You can draw new components using a grid of 5mm, 2,5 mm or less. Make sure that the created symbol fits into a 5 mm grid, i.e. the connections have to end at the grid points. You can set the grid via the Styles panel. A function is available in the **View** category allowing you to toggle the visibility of the grid.*
2. *The line width should be controlled. The line width for drawing connections must be identical with the line width for drawing wires. Most of the symbols in the included symbol libraries are drawn using a 0,25mm line width.*

Exercise 12-1: Draw the graphics for the component. You must use only the ordinary drawing functions, such as Draw Line, Rectangle, Circle etc., but not wires.

- Create a new page in your workspace.
- Construct the graphics.



- | | |
|------|---|
| 1.CA | Draw |
| 2.CO | Rectangle (Elements panel) |
| | Draw a rectangle with width 20 mm and height 5 mm. This is possible on the default grid of 5mm. The width and height of the rectangle are shown in the Status bar beneath the drawing area as you draw. |
| 3.+ | Click to define the left top corner point for the rectangle. |
| 4.+ | Click again to mark the right bottom corner point for the rectangle. |
| 5. | Right-click to quit the drawing mode. |
| | or |
| 3. | Click to define the left top corner point for the rectangle. |

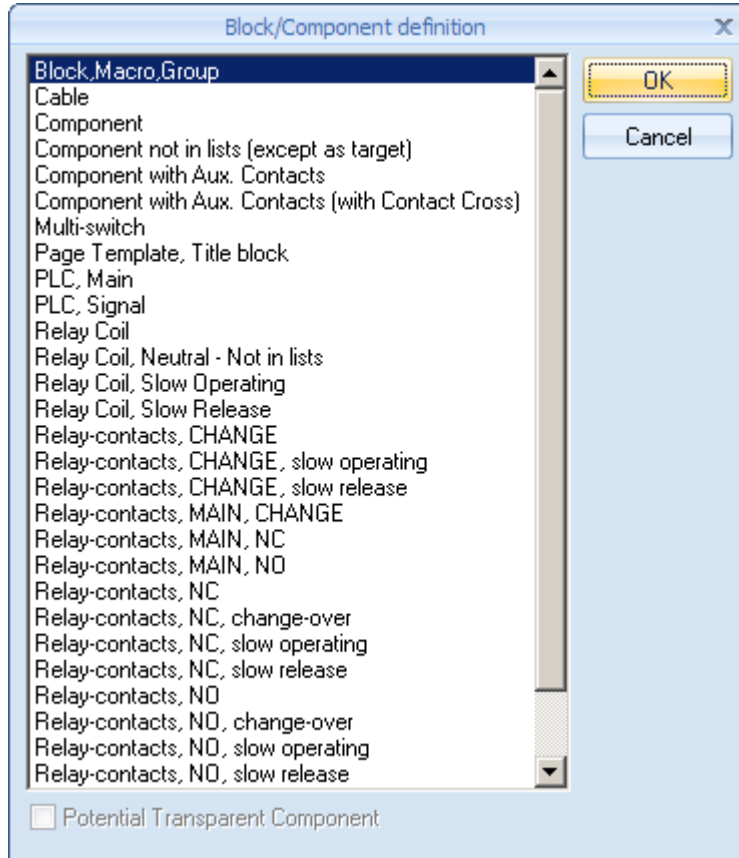
4. Keep the left mouse button pressed and move the cursor to the right bottom corner point of the rectangle.
8. Release the mouse button.
- 9.CA **Draw**
- 10.CO **Line (Elements panel)**
Draw two connection lines above and below the rectangle. These connection lines should be 5 mm long. The length of the lines is shown beneath the drawing area as you draw.
- 11.+ Click the starting point of the line.
- 12.+ Click the second line point.
13. Right-click to finish drawing the line, do not draw a polygon.
Afterwards, you can draw the next line.
- 14.+ Click the starting point.
- 15.+ Click the second point.
etc.
Right-click to finish drawing the last line, do not draw a polygon.
Right-click to exit the drawing mode.

Exercise 12-2: Define the symbol.

The graphics must be defined as a symbol.

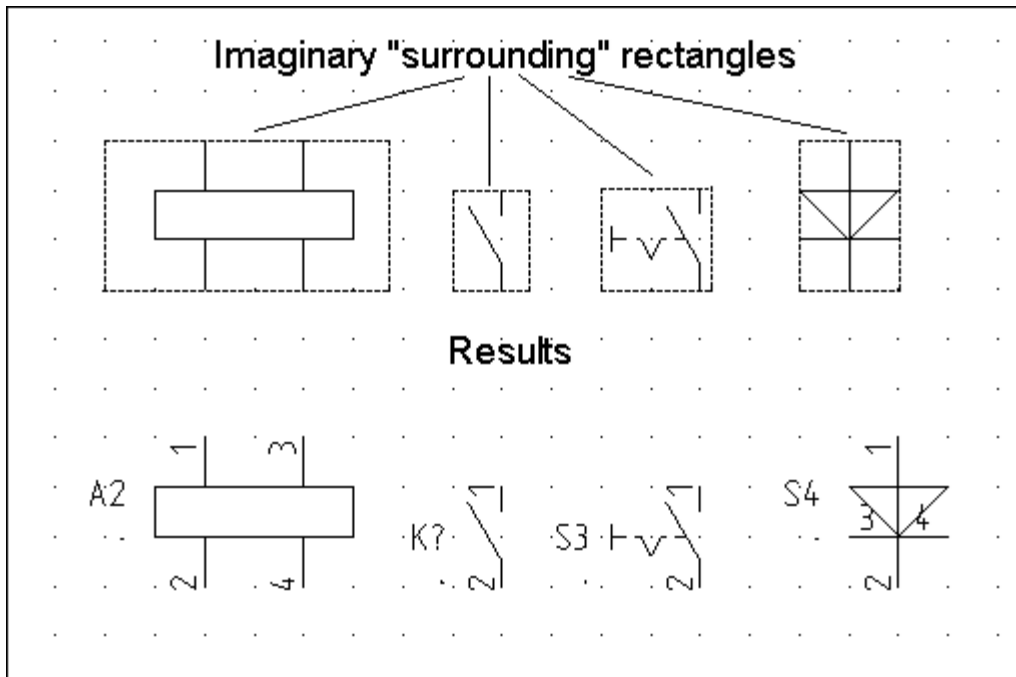
- 1.CA **General**
- 2.CO **Normal (Select panel)**
- 3.+ Click to define the first point of the frame.
The frame (rectangle) must include all the graphics of the symbol.
- 4.+ Click the second point of the frame.
5. Right-click.
- 6.M Select the **Block** pop-up command.
The **Block/Component definition** dialogue appears.

Choose the desired property for the symbol, for example Component.



- 8.> **OK**
The chosen Block/Component definition determines the kind of database list or graphical list where the symbol will be listed. A contact cross appears beneath **Coil** symbols, but not beneath **Component** symbols.
- 9.# A
Assign the letter code to the component name.
- 10.> **OK**
The graphics has been now completed with texts for the name, description, type and connections. The graphics and the texts have been grouped as a **Component** symbol. The letter code **A** for the component is used for the creation of its name, for example **3A8**.

Hints for the automatically located connections: Connection points are located automatically at the end of all the lines, which stick out of the symbol horizontally or vertically (they are surrounded by an imaginary rectangle).



At the diode, there are too many connections. In the following tasks, you will learn how to delete these connections.

If a symbol already has connections, no other connections will be automatically placed (otherwise the diode connections could never be removed). In this case you must, for example, copy the existing connections.

The order of placing the connection points corresponds to the order in which the lines have been created (drawing or copying).

The default settings for the automatically inserted texts are made in the symbol library SYSTEM.

Exercise 12-3: Save the symbol in the symbol library.

If the new symbol is used not only in the current workspace, but it must be available for future projects, you must save it in the symbol library.

If the new symbol is used only in the current workspace or it will be copied from the current page into another place if needed, it is not necessary to save it in the symbol library.

1. Activate the Symbols area.
If the **Workspace area** is visible, click the **Symbols** tab to activate the Symbols area.
2. MySymbols
Select the **MySymbols** database.
You can save only in this database or in a new database that you have created.
3. Right-click with the mouse.
- 4.M **New Folder**
- 5.# Relay coils
Type the name of the new symbol database.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

- 6.CA **Home**
- 7.CO **Component (Select panel)**
- 8.+ Select the new symbol.
- 9.+ Drag the symbol into the newly created folder in the symbol database. Hold down the left mouse button while you are dragging the symbol.

Important!!!!

Always drag the upper left connection of a symbol into the database. The symbol point that you hold while dragging into the database will be later the point that you place!



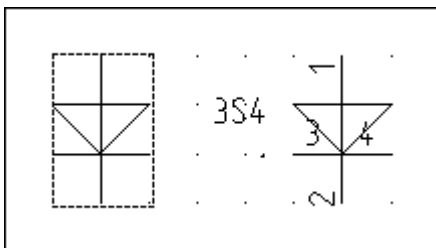
- 10.> Name
- 11.# Relay coil
- Type the symbol name and a symbol description.
- 12.> Click **OK**.
- The symbol has been saved in the symbol database.

M.2. CHANGING EXISTING SYMBOLS

M.2.1. DELETING ELEMENTS

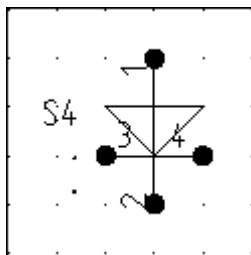
Exercise 12-4: Draw a diode. Create a symbol with "Component" properties using the diode. Use the known functions.

The diode will receive 4 connections because the end points of the crosslink finish at the imaginary "surrounding" rectangle, too.



Exercise 12-5: Change the diode symbol. Ungroup the symbol first.

- 1.+ Select the symbol.
2. Right click.
- 3.CO **Explode**
Execute the command from the pop-up menu.



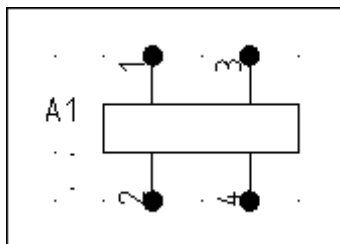
4. You can work now with the single component parts as if they have been created just now. Delete connections 3 and 4. (The connections consist always of the connection symbol and the connection text. When you select the text, the connection symbol is automatically selected too.)
5. Group the symbol again into a Component.
Save the symbol in the *MySymbols* symbol folder.

Make sure to always click the left upper connection when you drag the symbol into the database!

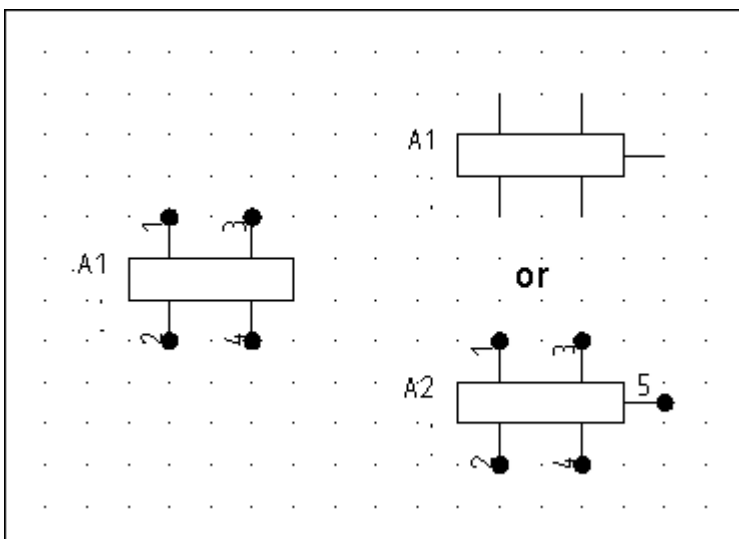
M.2.2. ADDING ELEMENTS

Exercise 12-6: Add one connection to the black box created at the beginning of this chapter.

- 1.+ Select the symbol.
(If the symbol is not available in the drawing, insert in from the symbol library.)
2. Right click.
- 3.CO **Explode**
Execute the command from the pop-up menu.



4. Draw a new line in the already known way.
5. You can either remove all the available connections or copy one available connection text.



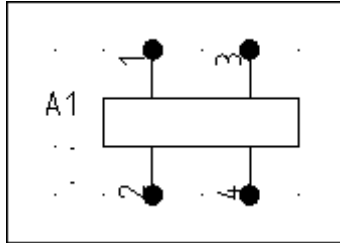
M.2.3. ADDING TEXTS

Exercise 12-7: The automatically inserted texts are often not enough. For example, for the black box you created first, you must have the description "Power supply" and the text 12V.

Add the texts to the black box that you created first.

- 1.+ Select the symbol.
(If the symbol is not available in the drawing, insert in from the symbol library.)
2. Right click.
- 3.CO **Explode**

Execute the command from the pop-up menu.

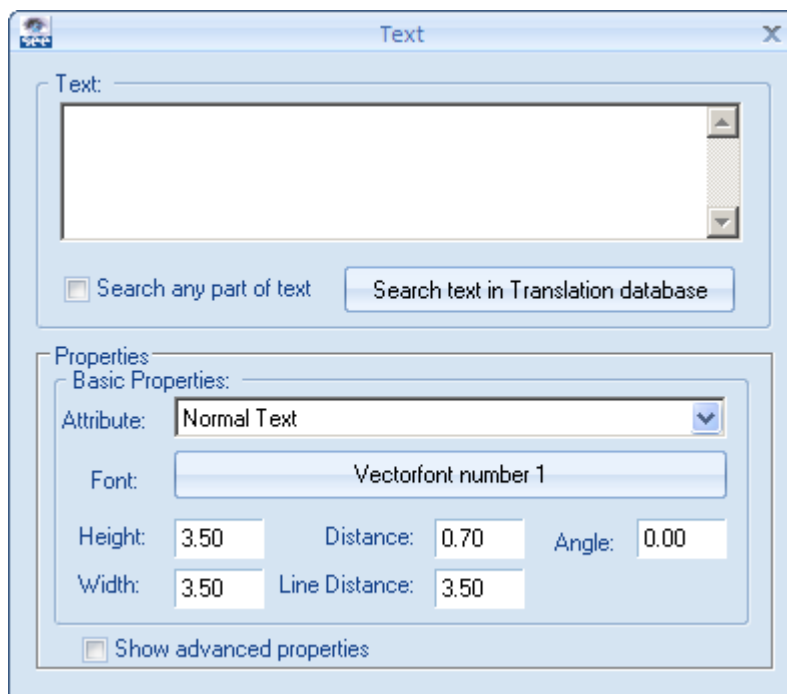


4.CA
5.CO

Draw

New Text (**Elements** panel)

New texts are required.



6.>

Attribute

Define the text attributes. Select the **Attribute** from the list.

In the list, you can find the following groups of attributes:

- ✓ *Workspace*
- ✓ *Content*
- ✓ *Function/Location*
- ✓ *Component*
- ✓ *Connection*
- and
- ✓ *Other.*

7.>

Component

Open the **Component** node in the list. Under Component, you can find the texts for product name, index, description and type, as well as additional texts called **Description** texts.

8.>

Description 01

Double-click the **Description 01** attribute under **Component**.

- 9.# 12V
Type the desired text in the input box.
- 10.> Right justified
The text has to be right justified.
- 11.+ Insert the text into the desired position in the drawing, left to the black box.
If the **Text** dialogue box is located over the place where you want to insert the text, you can move it. Left-click the blue bar of the dialogue box, hold the left mouse button and drag the box to another location. Drop at the desired location and release the left mouse button.
12. Group the symbol to a Component again.
13. Double-click the symbol.
You can change the Description 01 in the **Component properties** dialogue box.
14. Save the symbol in the MySymbols symbol folder.
Make sure to always click the left upper connection when you drag the symbol into the database!
15. Open the **View, Products** database list.
In the **Description 01** column, you will find your new text.
Select all the elements of the symbol again and group the elements into a **Component** symbol.

M.2.4. MOVING TEXTS

If a symbol is ungrouped, all the texts including connection texts can be easily moved using the **Drag and Drop** function.

Connection texts and corresponding connection symbols always compose a whole, so they are always moved together using **Drag and Drop**.

If a symbol is already used in a connection, it is not recommended to ungroup the symbol if you want to move the component name.

In this chapter, a function for relocating simple texts will be presented, even though they are part of a connection or a symbol.

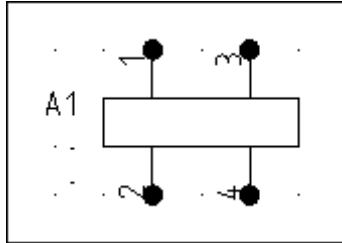
Attention:

Do not ungroup the connection between the connection symbol and the connection text using the **Explode** command!!!

Exercise 12-8: Move the component name and the connection texts. Ungroup the symbol before that.

- 1.+ Select the symbol.
(If the symbol is not available in the drawing, insert in from the symbol library.)
2. Right click.
- 3.CO **Explode**

Execute the command from the pop-up menu.



- 4.+ Move the component name.
To do this, use the **Drag and Drop** function. Click the text, hold the left mouse button and move the text to the desired location. Release the left mouse button.
- 5.+ Move a connection text.
- 6.CA **General**
- 7.CO **Single Element (Select panel)**
Connection symbol and connection text usually belong together. Use the **Single Element** function to select apart the connection text or the connection symbol.
- 8.+ Select a connection text.
- 9.M. Right click and select the **Move** pop-up command.
- 10.+ Pick the reference point for the movement on the text connection point.
- 11.+ Place the text in the desired position.
Proceed in the same way for other connection texts.
Then group the parts into a "*Component*" symbol again.

Exercise 12-9: Move the component name of the black box.

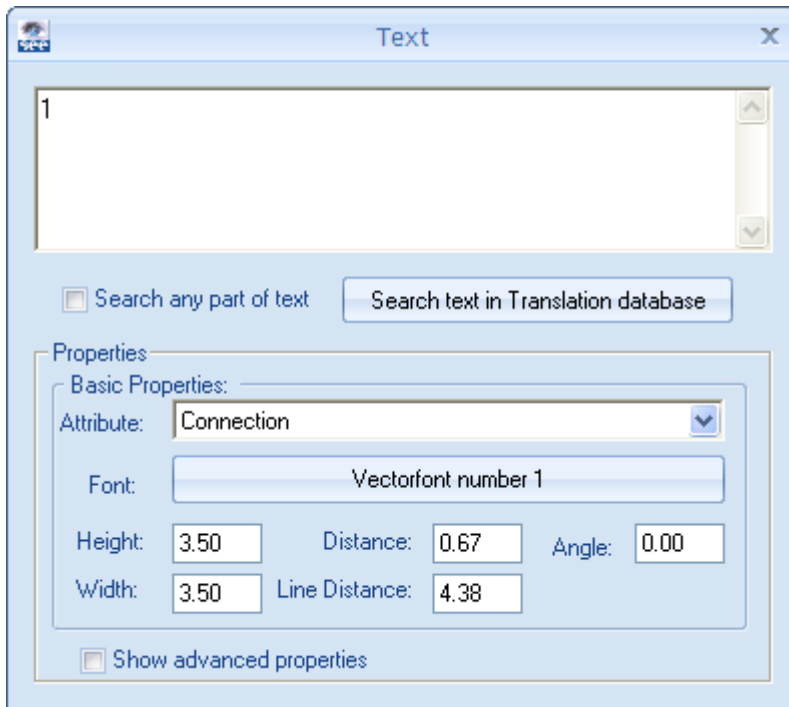
- Do not ungroup the symbol!

- 1.CA **General**
 - 2.CO **Single Element (Select panel)**
 - 3.+ Select the component name.
 - 4.M Right-click and select the **Move** pop-up command.
 - 5.+ Pick the reference point for the movement of the text.
 - 6.+ Place the text in the desired position.
- You can move all the texts without ungrouping the component.

Exercise 12-10: Rotate the texts for the connection numbers.

- Do not ungroup the symbol!

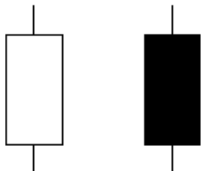
- 1.CA **Edit**
- 2.CO **Edit Text** (**Text** panel)
- 3.+ Select the text you want to rotate.



- 4.> Angle
- 5.# 0
Change the angle from 90 degree to 0 degree for example.
You can change more texts now, but please don't do so.
- 6.> Close the **Text** window.
- 7.CA **General**
- 8.CO **Single Element** (**Select** panel)
- 9.+ Select the next connection text you want to rotate.
- 10.+ Press CTRL on the keyboard and select a second text you want to rotate, etc.
- 11.CA **Edit**
- 12.CO **Edit Text** (**Text** panel)
The **Text** window is opened.
- 13.> Angle
- 14.> 0
The angle is changed for all texts selected.
- 15.> Close the **Text** window.

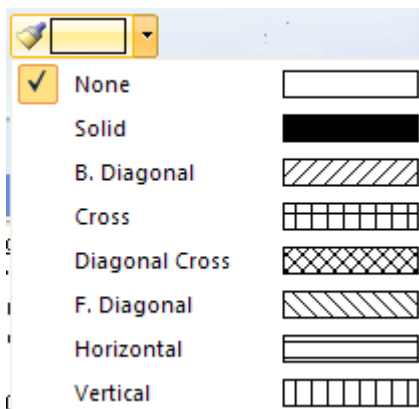
M.3. HANDLING GRAPHICS

Exercise 12-11: Draw a symbol for a coil.

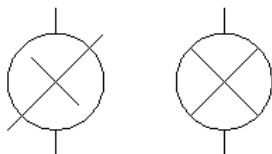


Fill in the rectangle. Use the **Draw > Elements > Fill/Hatch area** command.

Select the appropriate filling in the **Select brush** command (**Draw > Styles** panel). Circles and ellipses are automatically filled during the drawing, so the setting can be set up to "None".



Exercise 12-12: Create a signal lamp.

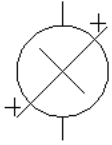


You cannot draw the lines on the grid so that they end at the circle. By using the **Edit > Change Element > Trim** command, you can truncate too long cutting lines. By using the **Edit > Change Element > Extend** command, you can extend too short lines to the cutting edge.

To trim the lines which are too long:

- 1.+ Click the cutting edge at the circle.
- 2.CO **Trim**
Select the function from the **Edit > Change Element** panel.

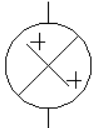
- 3.+ Click the element of the graphics that must be removed:



- 4.+ Click another element of the graphics that must be removed up to the cutting edge selected in the first step.
Right-click to exit the function.

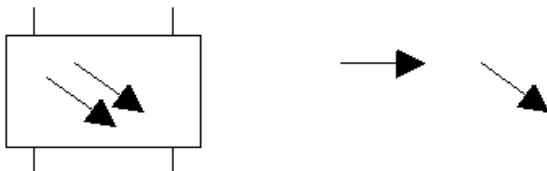
To extend the lines which are too short:

- 1.+ Click the cutting edge at the circle.
2.CO **Extend**
Select the function from the **Edit > Change element** panel.
3.+ Click the end of a line to be extended:



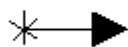

- 4.+ Click the end of another line that must be extended up to the cutting edge selected in the first step.
Right-click to exit the function.

Exercise 12-13: Draw the shown smoke detector.



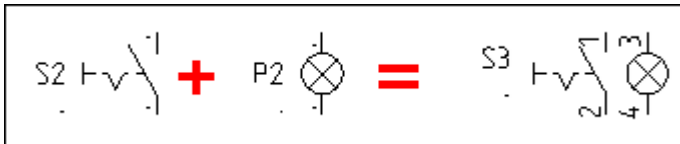
Draw the arrows at 0 degrees first. In this way, the arrowheads can be drawn symmetrically. Use the **Edit > Actions > Rotate** command to rotate the arrow afterwards.

To rotate the arrows:

- 1.+ Select the graphics of the arrow.
2.CO **Rotate**
Select the function from the **Edit > Actions** panel.
3.+ Fix the point which will be a rotation centre: 
4.+ Define the rotation axis: 
5.+ Move the cursor. The selected elements are rotated.
6.+ Place the selected elements at the desired rotation angle.

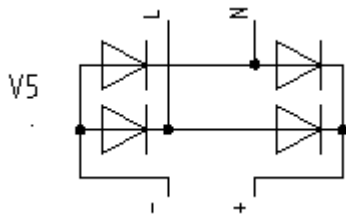
M.4. USING AVAILABLE COMPONENTS

Exercise 12-14: Create a new symbol for Lamp Switchgear pushbutton using the available NO push detent symbol from the Switchgear one pole symbol folder and the lamp symbol from the Lamps folder.



1. Insert the needed symbols from the symbol library.
Ungroup the first symbol.
- 2.+ Select the symbol.
3. Right click and select the Ungroup Selected pop-up command.
Ungroup the second symbol in the same way as described above.
Remove all the unnecessary elements or move the needed elements to an empty place on the screen.
Group the elements into the **Component** symbol **Lamp Switchgear pushbutton**.
Save the symbol in your symbol database.

Exercise 12-15: Create a Diode rectifier. Use the graphics of any diode symbol from the library as a basis.

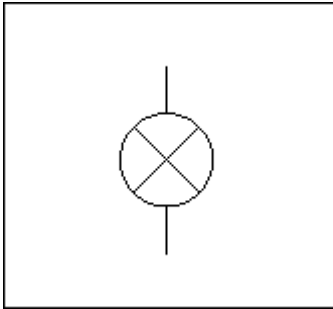


Attention!!!

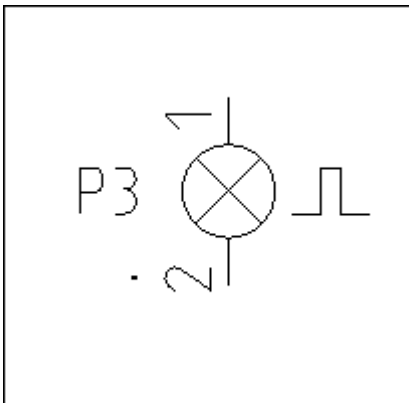
Draw the nodes as Graphics, for example as filled circles.

Exercise 12-16: If you need a component that can be easily created after deleting graphics elements from one available component, you can proceed as follows:

You need this component:



This component is available:



- Insert the available component.
- Right-click to activate the pop-up menu and click the **Select single element** command. Then select the element and delete it. (You can select several elements if you hold down the CTRL key.)

If the new symbol is used only in the current workspace, it is not necessary to save it in the symbol library.

M.5. INFORMATION ABOUT SPECIAL COMPONENTS AND THEIR CREATION

M.5.1. CONTACTS

Contacts require texts for component names, connection texts and a cross reference text.

If the contact has a description text placeholder, the description will be assigned from the coil, if it has been entered there.

In the case of change-over contacts, place the connections in the following order: root, normally open contact side, normally closed contact side.

SEE Electrical also differentiates between main contacts (main normally open contact, main normally closed contact) and auxiliary contacts (normally closed contact, normally open contact, change-over contact).

M.5.2. TERMINALS

Terminals require at least the following texts: component name, terminal number, terminal index, and at least one connection text.

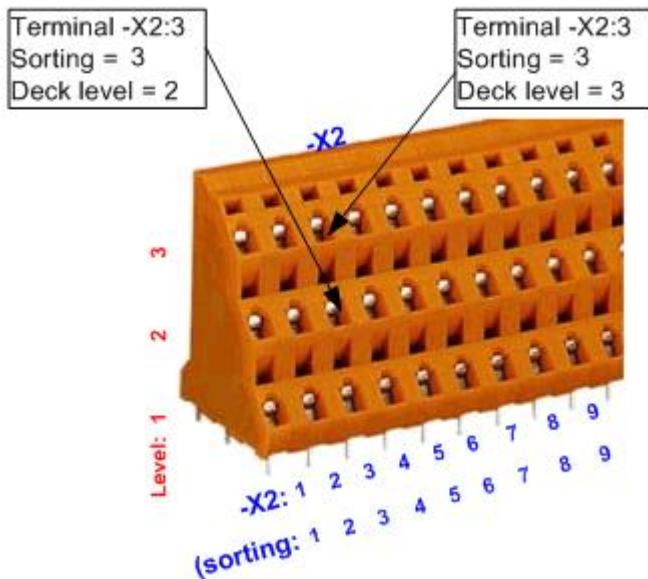
For a terminal number, you should type any number, ?, x or X, if the number has to be incremented. In this way you can obtain terminal numbers like L1, N, or PE.

If texts are placed automatically as you create a symbol in *SEE Electrical*, the connections are created automatically as described in the paragraph above, i.e. without texts. If you wish to have connection texts for diode terminals, you have to place the connection texts manually. Of course, texts should not be deleted in this case.

If you would like to display terminal strip name and terminal number in one text, such as X2:17, you have to use **Terminal name + number merged** the text attribute which you can find under the *Other* attributes node. Place in this case only the text with the **Terminal name + number merged** attribute into the graphics. Group the terminal symbol afterwards.

M.5.3. MULTI-LAYER TERMINALS (STANDARD)

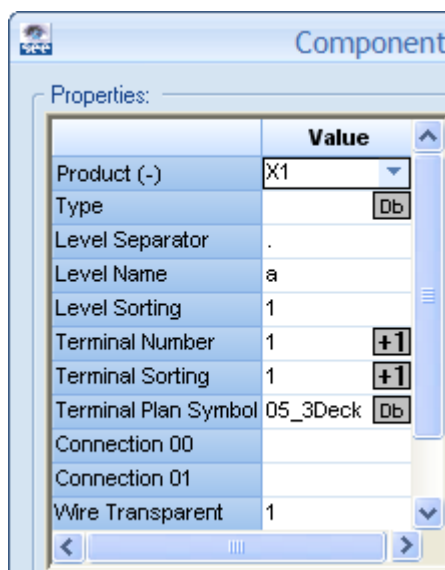
Multi-Layer terminals represent several terminals placed in the circuit diagram, with the same number and the same index, and grouped as a combined component (multi-layer terminal block). The single terminals represent the levels. They differ in the designation of the levels, and in the level number.



All elements of a multi-layer terminal have the same type.

In the List of parts (*standard* level), only one record exists. If you use the *Cabinet* module, only one symbol is available there.

The difference between multi-layer terminals and "normal" terminals is that the multi-layer terminals need additional text placeholders to identify the level number and the level name, as well as a level separator between the terminal number and the level name.

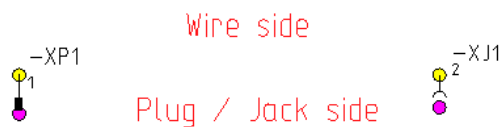


M.5.4. CONNECTORS

The connectors require specific texts. At the creation of the connector the "Product name", "Description", "Type", "Pin Name" and "Pin-Id (Sorting)" texts are generated automatically. "Pin Name" contains the pin name defined by the manufacturer. "Pin-Id (Sorting)" gives the sort order in which the pins are added automatically to the circuit diagram. If you want a different name than the one suggested automatically, change the "Product Name" in the **Component Properties** dialogue.

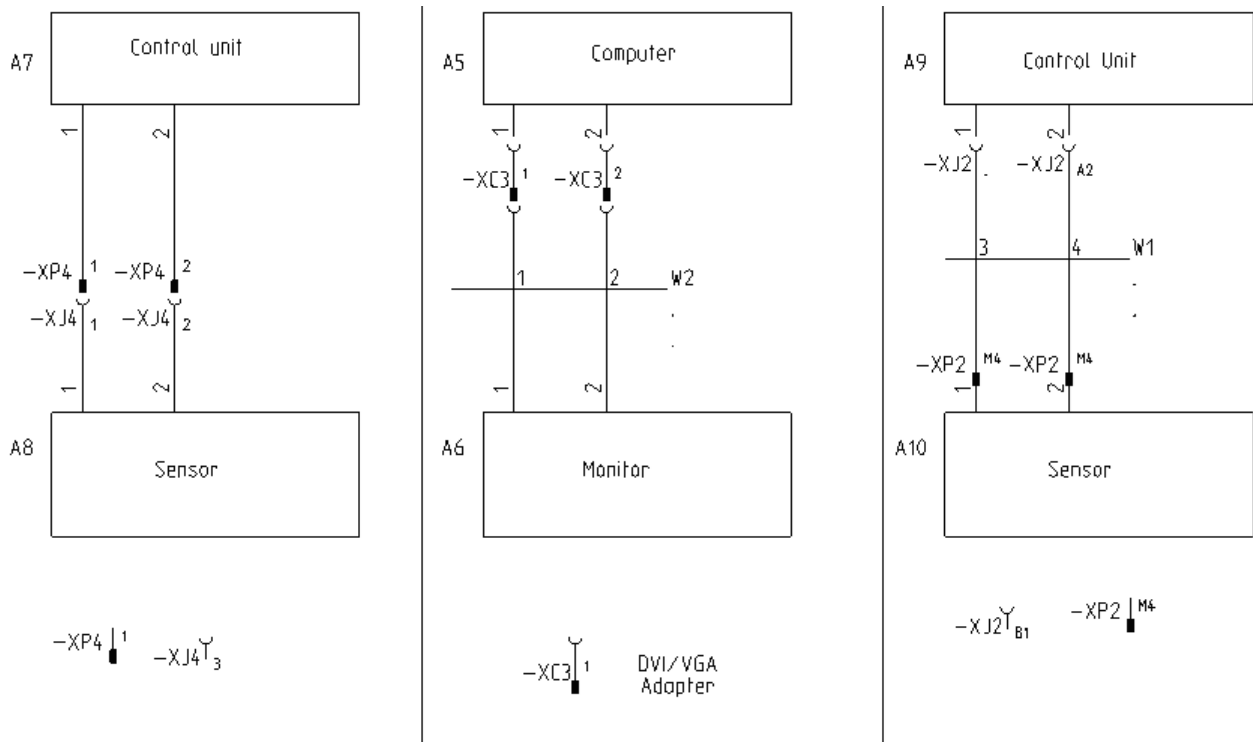
Plugs have a wire side and a plug/jack side. A plug is connected to a wire on its wire side and to a jack on the other side. The jack can be part of a component or of another connector.

Jacks have a wire side and a plug/jack side. A plug is connected to a wire on its wire side and to a plug on the other side. The plug can be part of a component or of another connector.



In *SEE Electrical* the wire side has to be represented by the first connection point and the plug side has to be represented by the second connection point.

No physical wire can be connected here. In case you draw a wire and connect it to the plug side, it will be used only to find the component/connector and write the information in the Connector-Pin list, but this wire will not be displayed in the wire list and the wire number/potential name will not be created. It is not possible to draw a cable in such situation.



Control the wire and plug sides before you store the symbol in the symbol library (**Electrical > View > Connection**). If you have to change the wire and plug connections, use the **Electrical > Connections > Swap** command.

M.5.5. CROSS-REFERENCE SYMBOLS

The way in which potentials are managed in the workspace defines how you have to handle your cross-reference symbols.

✓ Cross-reference symbols in workspaces with potential objects

Cross-reference symbols consist of the graphics and the texts for *Product*, *Code/Cell Reference* (**Text** dialogue ► *Attributes* ► *Components* node) and a connection symbol. If the connection symbol is inserted manually the field is empty. If the symbol is inserted automatically, the connection text is present in the field. The connection text has to be deleted, so that only the connection symbol remains. (Press the F6 hot key to activate the **Select Single Element** command, select the connection text and then delete it).

If several cross-references obtain the same name, you can define by an index which cross reference should be linked with the other one. There must always be only two cross-references with the same name and the same index because this is the only way to create a unique cross reference.

The text placeholder for the index requires the "*Product Index*" text property (it can be found in the "Component" area). The text should be placed manually before the cross-reference symbol is blocked.

If you rename a cross-reference which possesses an index, only the name of the cross-reference which possesses the same name and the same index, if present, will be changed.

✓ Cross-reference symbols in workspaces with potentials managed as wires

Cross-reference symbols consist of the graphics and the texts for *Product*, *Code/Cell Reference* (**Text** dialogue ► *Attributes* ► *Components* node) and a connection text.

Inside these symbols there is a connection text which determines whether a cross-reference symbol is recognized as source (connection text \$1) or as target (connection text \$0).

Cross-references can only be generated if a pair of cross-reference symbols with the same name is found, where one has the source attribute and the other the target attribute. A source symbol always finds the next target with the same name - on the next or one of the following pages. (This allows cross references to work even if a page is inserted or removed). A target symbols "looks back" to pages in front of the current page.

M.5.6. INFO TEXT SYMBOLS

Info text symbols allow you to manage targets for Terminals list, Cables list, Cable-cores List, Wires List, Multicores List (only for *advanced*). The designation of the "Info text" symbols is not automatically assigned/changed.

Info text symbols consist of the graphics, the *Product*, and the connection symbol. The connection text must be deleted so that only the connection symbol remains (Press the F6 function key to activate the **Select single element** command, select the connection text and then delete it).

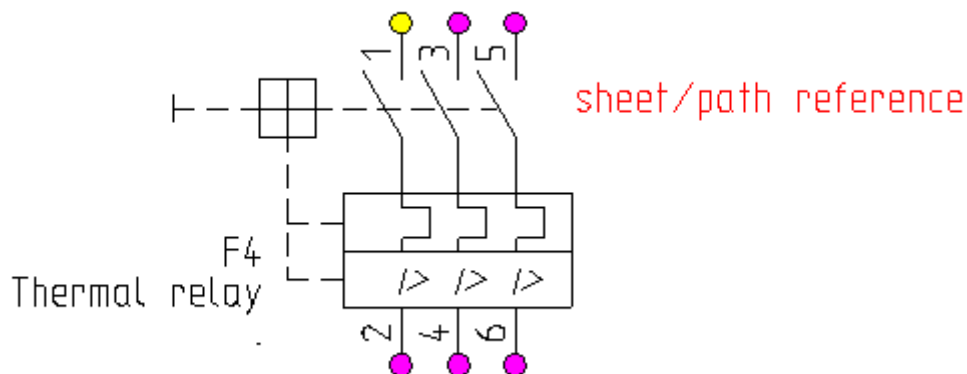
No characters (or blanks) should be entered for the component name when you create the symbol in order to prevent the automatic modification of the component's name. Therefore, when creating the symbol, do not fill in a component name, just click directly **OK**.

M.5.7. COMPONENTS WITH AUXILIARY CONTACTS

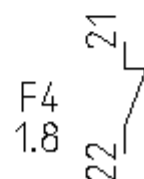
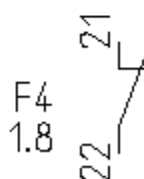
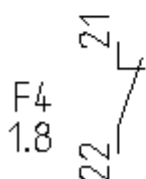
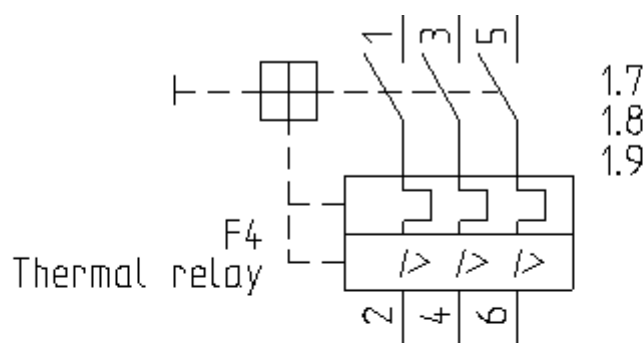
There are two ways of generating components with auxiliary contacts:

M.5.7.a. SYMBOLS WITH CROSS REFERENCE AS TEXT

A component with auxiliary contacts consists of the graphics and the texts for component name, connection texts, description 00, type etc. Additionally you need a text with the attribute "code/cell reference" (you can find it in the "Component" area of the text attributes). The single elements are integrated into a **Component with Auxiliary Contacts** using the **Edit > Actions > Block** command.



If you then position contacts with the same name as the master component, you get cross reference texts. For example:



The line spacing is taken from the line spacing defined in the text placeholder.

M.5.7.b. SYMBOLS WITH GRAPHICAL CONTACT SYMBOLS IN IT

A component with auxiliary contacts consists of the graphics and the following texts: component name, description, type, connection texts etc. Furthermore, the contacts available in the component are required. The single parts are grouped as a **Component with Auxiliary Contacts** by using the **Edit > Actions > Block** command.

M.5.7.c. EXAMPLES OF COMPONENTS WITH AUXILIARY CONTACTS

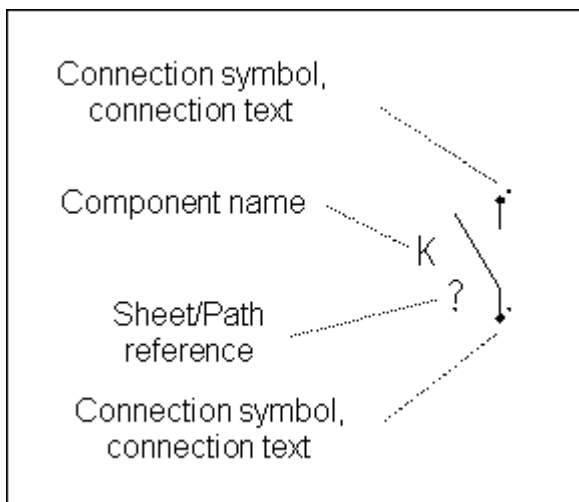
Contacts for the component with auxiliary contacts

If a cross-reference must go out of a component with auxiliary contacts to a contact, you must place contact symbols in the component.

First, create the graphics.



Create the texts. Pay attention that the texts receive the specified attributes.

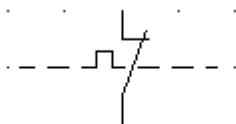


Move the connection text if necessary.

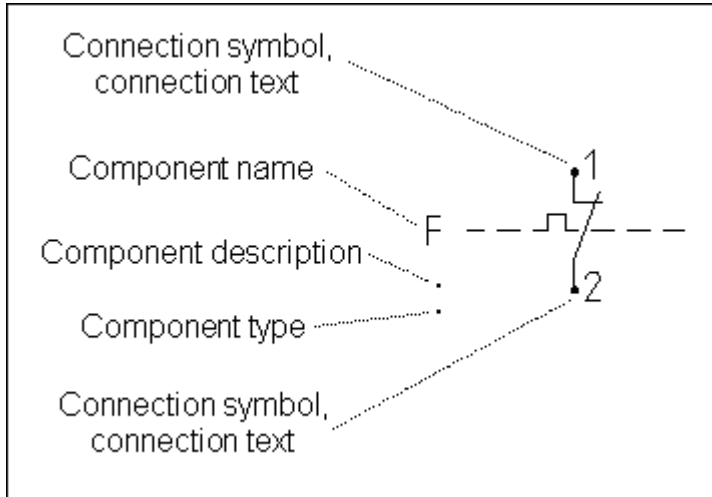
Select then all the elements of the symbol and group them into a Contact symbol, such as a normally closed contact, a normally open contact or a change-over contact.

Component with auxiliary contacts

Draw the graphics.

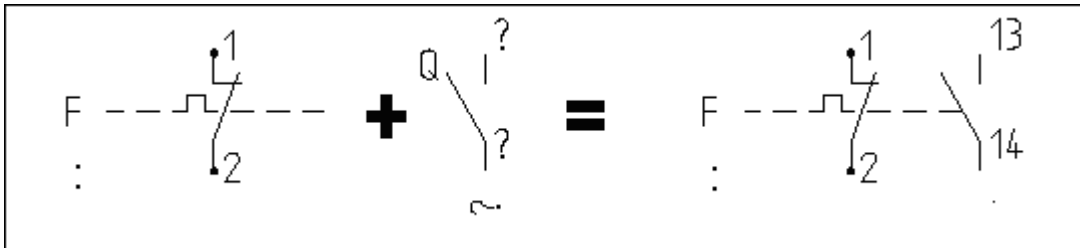


Create the texts. Pay attention that the texts receive the specified attributes.



Move the connection(s) text(s) if necessary.

Insert the required number of contacts. Insert the contact symbol without a text placeholder for the component name from the symbol database.



Select all the elements of the symbol and group them into a Component symbol with auxiliary contacts. *SEE Electrical* deletes the component names from the contact symbols. Save the symbol in the symbol database.

M.5.8. PLC COMPONENTS

For PLC-components, *SEE Electrical* differentiates between PLC-racks and PLC signals (inputs or outputs).

When the symbols are created properly, the PLC-racks receive cross-references to the inputs and outputs, and the inputs and outputs receive a back reference to the Rack.

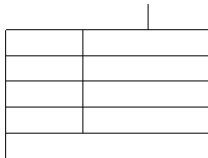
A single input/output always consists of the graphics and the following texts: component name, connection text(s), cross reference (optional, for the back reference to the Rack), Address, PLC comment (optional), description (optional). A PLC-signal can be created via the **Edit > Actions > Block** command.

A PLC-Rack consists of the component graphics, the texts for the component: component name, connection texts, comment and type and the number of PLC-signals (see above). You can integrate the single elements into a PLC-Rack using the **Edit > Actions > Block** command.

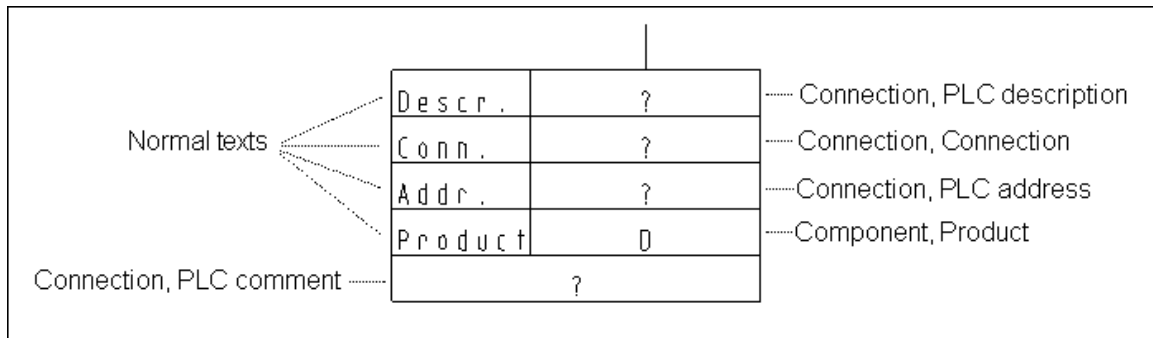
M.5.8.a. EXAMPLES OF PLC COMPONENTS

PLC-signals

- Draw the graphics.



- Create the texts. Make sure that the texts receive the specified attributes. All of the displayed texts including the texts for addresses and descriptions are necessary.



- Move the connection texts if necessary.
- Select all the parts of the symbol and create a PLC-signal symbol.
- Save the symbol in the symbol database.

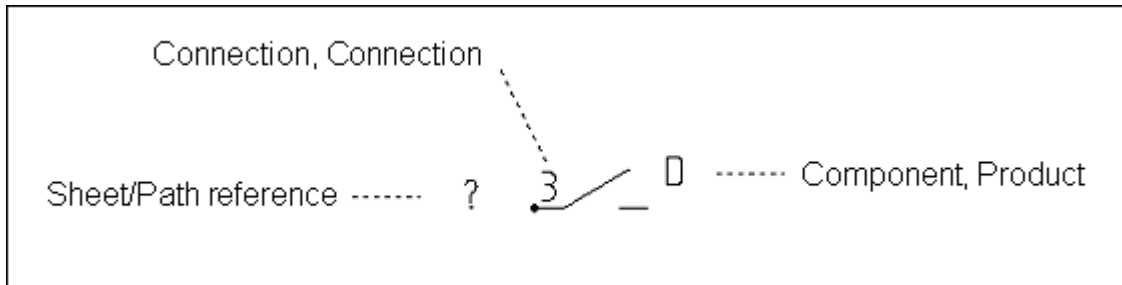
PLC-signals to be used in PLC-Rack

If a cross-reference must go out of a PLC-Rack to a PLC-signal, you should place symbols for PLC-signals in the Rack.

- Draw the graphics first.



- Create the texts. Make sure that the texts receive the specified attributes.

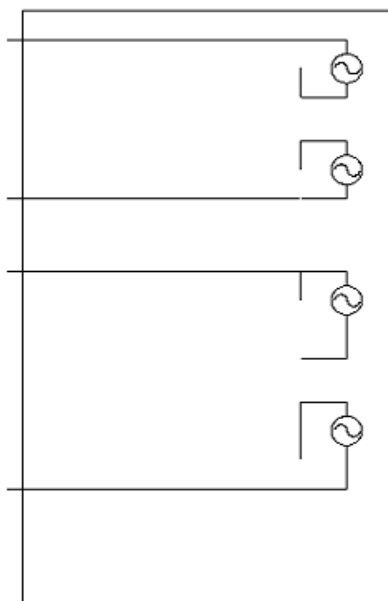


- Move the connection text if necessary.
- Select all the parts of the symbol and create a PLC-signal symbol.

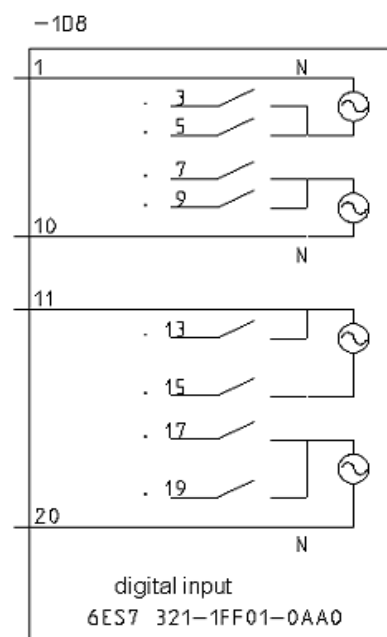
PLC-Rack

- Create the graphics first.

Graphics



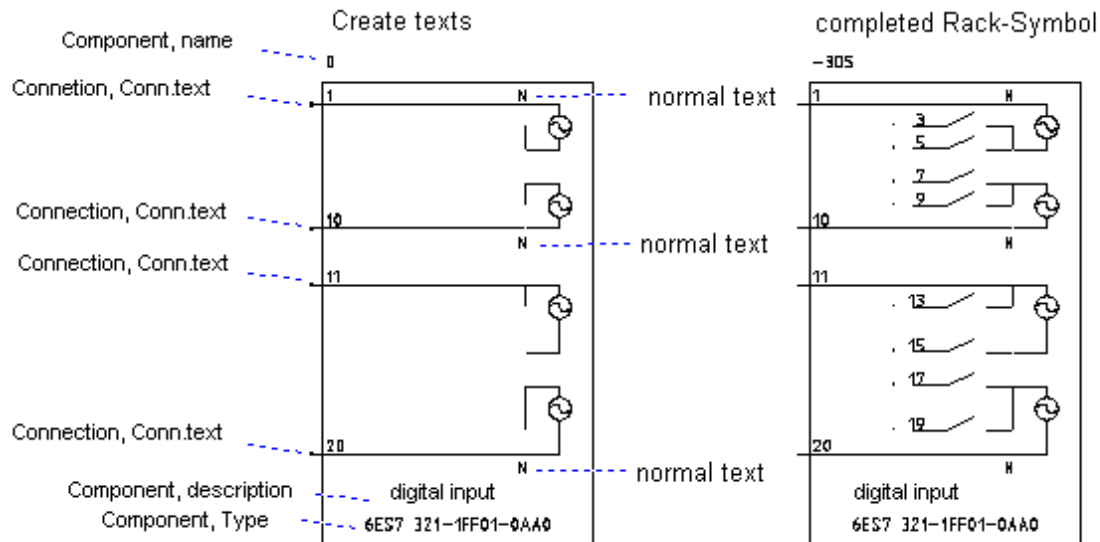
completed Rack-symbol



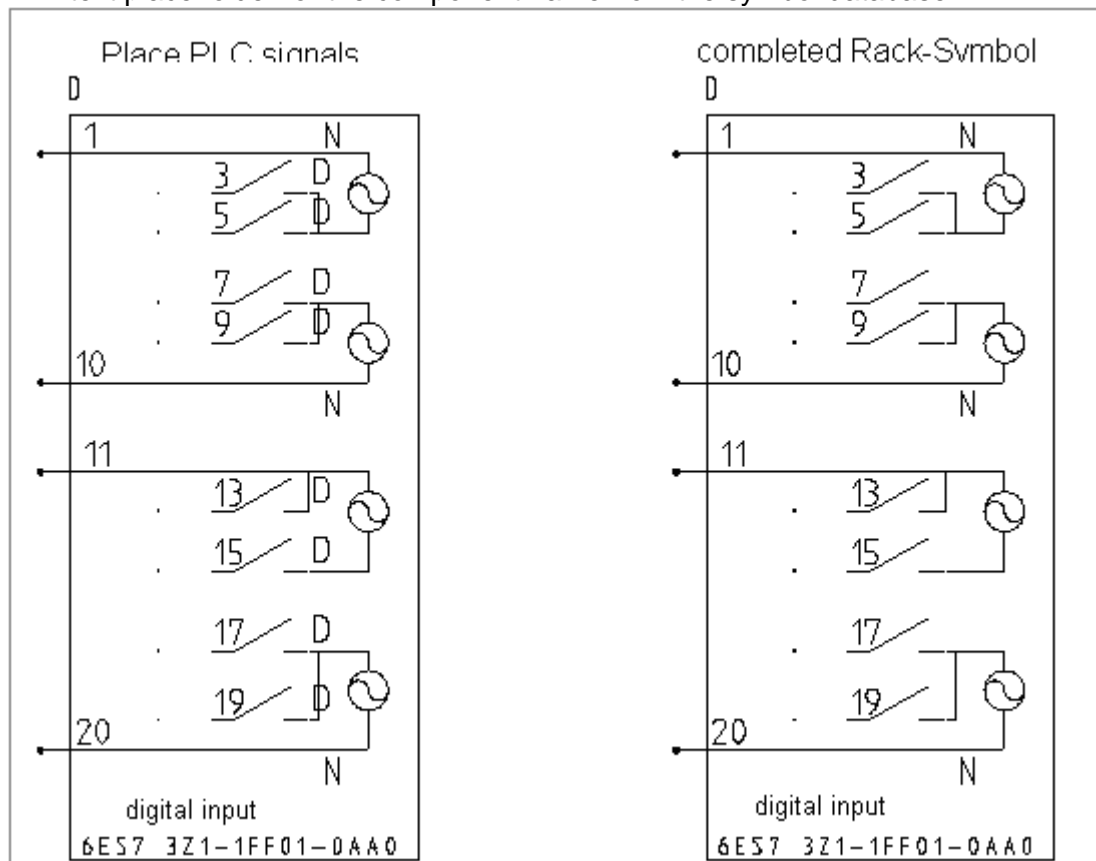
Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

- Create the texts. Make sure that the texts receive the specified attributes. At least one connection text is necessary. (If no connections are needed for the component, place the connection and define a space for the connection text).



- Move the connection text(s) if necessary.
- Place the required number of PLC-signal symbols. Insert the PLC-signal symbol without a text placeholder for the component name from the symbol database:



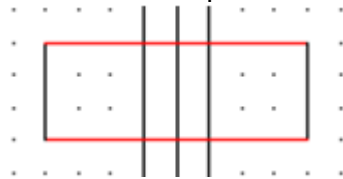
- Change the names of the PLC signals.
- Select all symbols and create a PLC-component symbol *SEE Electrical* deletes the component names from the PLC-signal symbols.
- Save the symbol in the symbol database.

M.5.9. SMART BLACK BOX SYMBOLS

The smart black box symbol creates connection points at the places where it connects with the wires.

1. To use smart black boxes draw a rectangle **Draw** ➤ **Elements** ➤ **Rectangle** around the wires.
2. Right-click it and select the **Block** pop-up command.
3. Choose the Smart black box symbol from the list.
4. Type in the code for the symbol.
5. Specify the lines that will automatically generate connection points.

The connection points are automatically created.



Hint:

*You can zoom in or out an intelligent black box symbol whose outline was generated as a rectangle, as follows: press the CTRL and SHIFT key on the keyboard at the same time and identify the rectangle while these keys are pressed. Only the rectangle is now selected and you can change its size when the default setting "**Trackers on the selected elements**" is active. Please make sure that you never change the size of an intelligent black box symbol when connections are already connected.*

M.6. CABLE SYMBOLS

(Advanced)

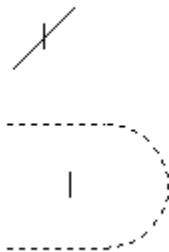
You can create user-defined cables and use them later when necessary.

M.6.1. CREATING USER-DEFINED CABLE SYMBOLS

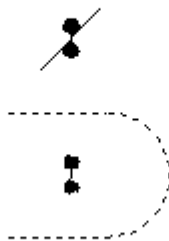
Cable symbols consist of the graphic and the text placeholders for component name, function, type, length, 2 connection symbols (without text), cable core number, cable core colour and cable core section.

The symbol must be created with the correct angle.

Two examples of graphics



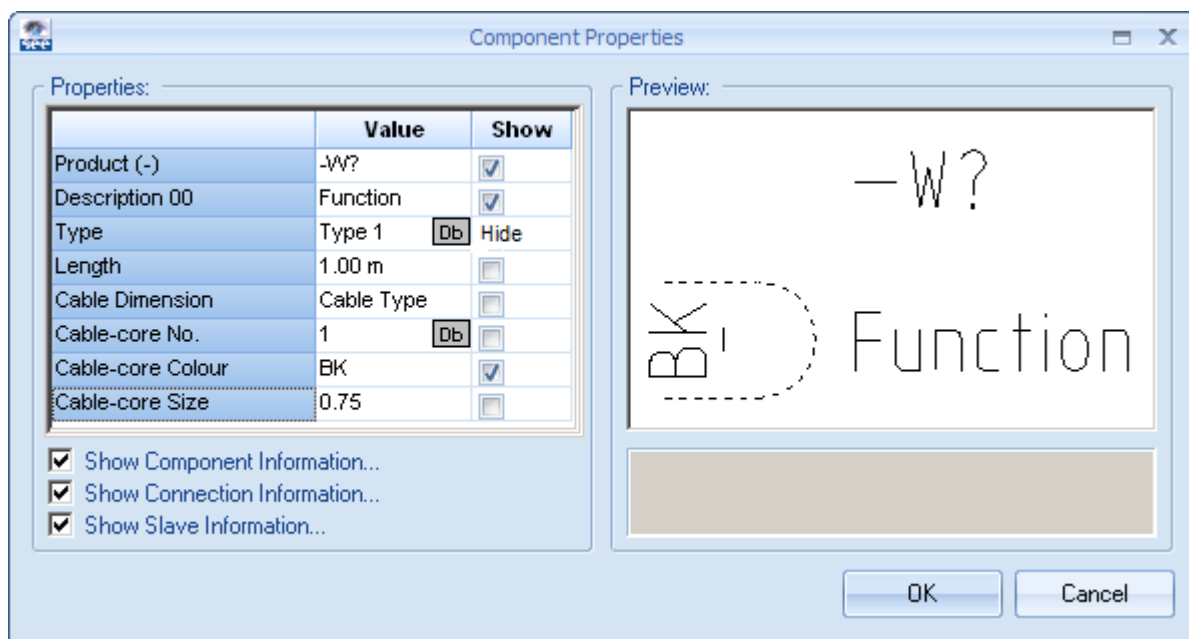
Connections



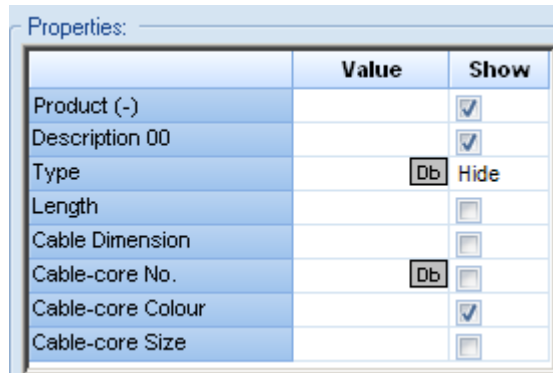
Text

—W?
Type 1
Function
1.00 m
Cable Type
BK
0.75

After grouping the elements into a cable symbol, you can define which texts will be visible via the check box in the "Show" column in the **Component Properties** window.



- Delete all texts.



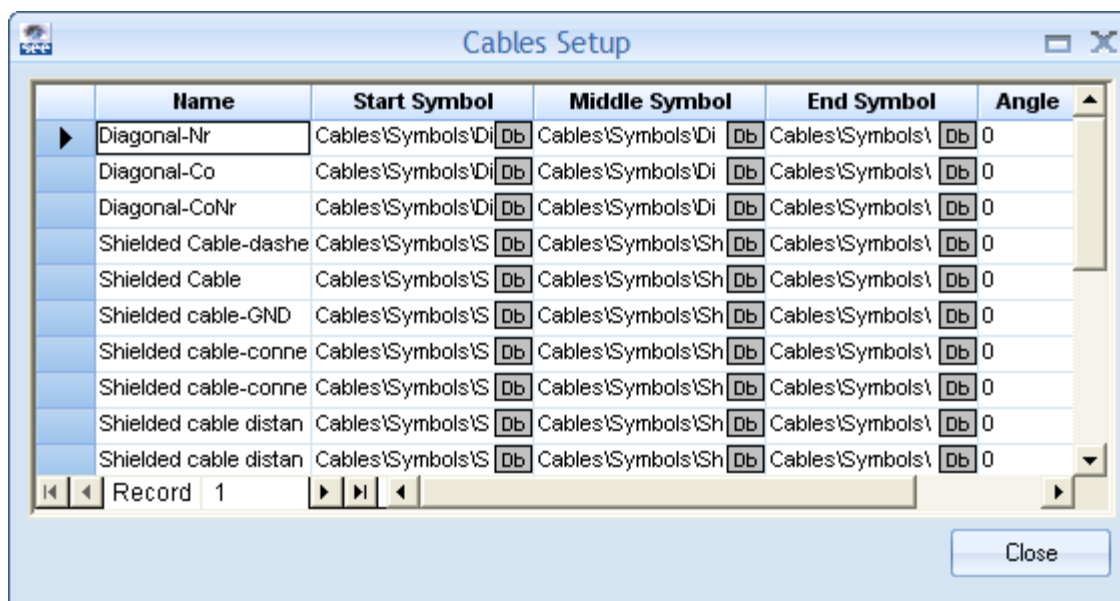
- Then move the cable symbol into the symbol library.

It is possible to insert the free texts 1 to 3 to the cable symbol. They also appear in the Cable list (database list and graphical list). These texts must be manually placed, if desired, before grouping the symbol.

M.6.2. CREATING USER-DEFINED CABLES

The functionality for creating your own cable definition and layout is accessible through the **Cables Setup...** button within the *Cables* tab of the **Circuit Diagrams Properties** dialogue. This dialogue is called by clicking the **Properties...** command from the pop-up menu that appears when you right-click the Circuit Diagrams module in the *Workspace Explorer*.


When you click the **Cables Setup...** button, the following window appears:



In the *Name* field, you have to specify a cable name by your choice.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

After you click the  button within the "**Start Symbol**" field, the **Symbol browser** opens, allowing you to select the additional symbol used at the first cable core. Its description will be displayed in this field.

In the same way, you choose the additional symbol to be used at the middle core and the additional one for the last core of the cable. Their names (descriptions) are displayed, respectively, in the "**Middle Symbol**" and "**End Symbol**" fields. The definition is saved in **CABLESNEW.MDB**. The file must be located in your *Templates* folder. The definitions are available for all workspaces.

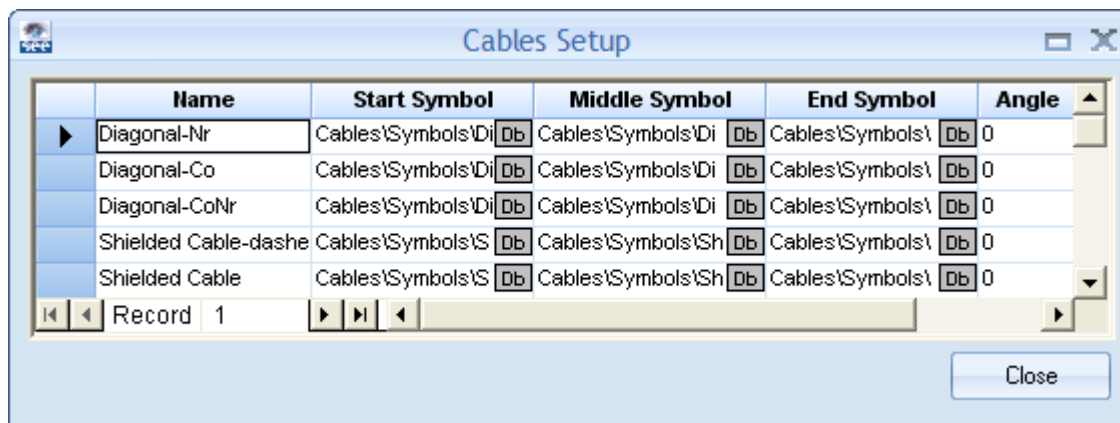
In the "**Angle**" field you have to specify the fixed rotation of your cable at the insertion. Because the symbols are used, the angle is fixed. If an angle of 0 degree is used, the cable cores are created from left to right. If an angle of 180 degree is used, the cores are created from right to left. The same is applied for 90 and 270 degrees.

Hints

- 1: For creating cable definitions, the various cable symbols must be present in the symbol database. Symbols must have all the necessary cable symbol properties assigned.
- 2: The symbols must be created at such angle as it is defined for use.
- 3: The defined cables are to be stored in the ...Template\CablesNew.mdb database. The cables from this database are available for all projects.

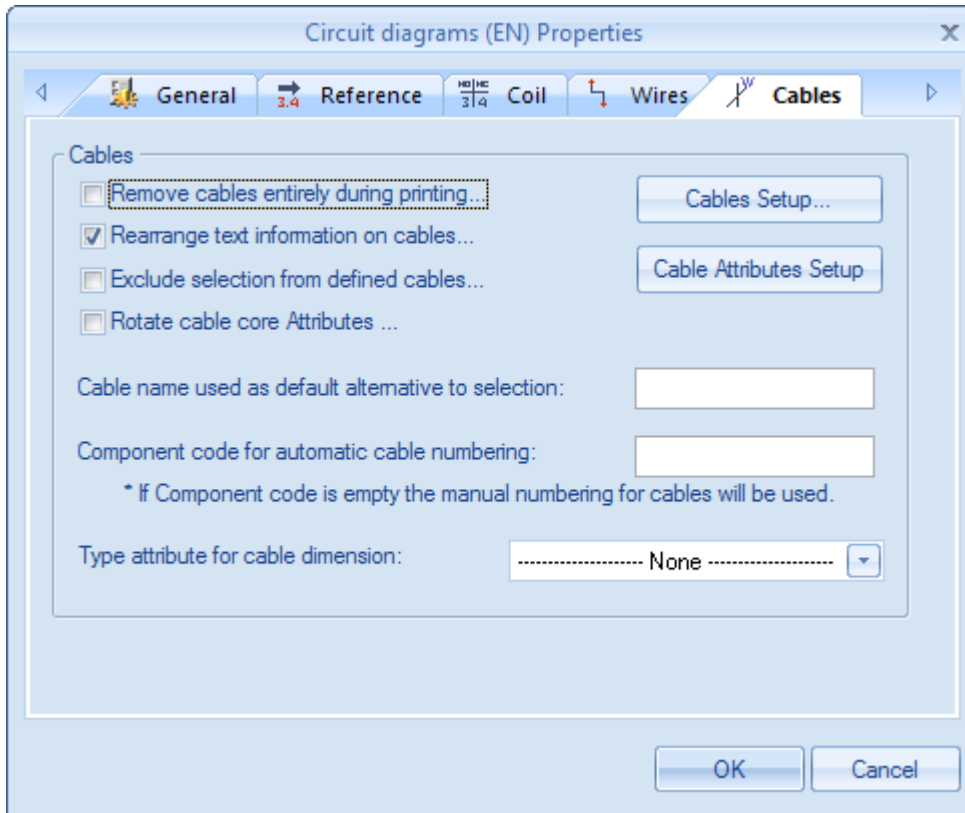
When you want to use or add a user-defined cable, a dialogue listing all pre-defined cables appears.

Example:



If no use-defined cable exists yet (possible list empty), no list will appear. In case you choose a user-defined cable, it will be inserted. Otherwise (if you cancel the dialogue), **SEE Electrical** will add a basic cable.

M.6.3. SETTINGS FOR CABLES



If you have created user-defined cables but you do not want to use them in a workspace, you can tick the "**Exclude selection from defined cables**" option.

If you have created user-defined cables and you want to use the same cable every time, you can type in the name of the desired cable in the "**Cable name used as default alternative for selection**" field:

Example:

Cable name used as default alternative to selection :	Shielded Cable
---	----------------

M.7. INFORMATION ABOUT COMPONENT INSERTION

Components can be inserted from the symbol database into the drawing by left-clicking with the mouse to select the desired component and dragging it into the drawing (the left mouse button should not be pressed). The component is attached to the mouse and can be placed in the drawing.

M.7.1.a. RELAY COILS

If a relay coil is placed, a contact cross appears beneath the component. The contact cross can be moved independently, if needed.

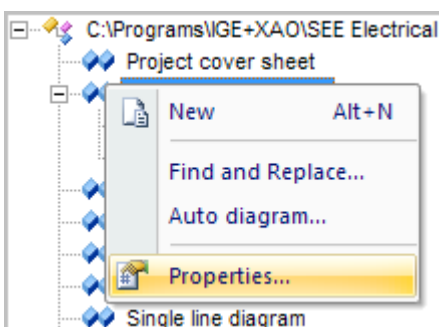
M.7.1.b. TERMINALS

Terminals are named automatically using incrementing numbers, after they have been inserted. Terminals are sorted using an assigned index. The index is usually defined automatically. You can change the index as you insert the terminal or afterwards in the Component Properties dialogue. For example, the index allows inserting PE terminals in the appropriate place.

M.7.1.c. ASSIGNING A COMPONENT NAME

You can define the method of assigning component names in the Circuit Diagrams Properties dialogue.

Right-click the Circuit Diagrams module in the Workspace Explorer and select the Properties pop-up command.



Within the Component numbering area, select the desired component numbering. For some symbols, such as contacts, it is possible to select the name of an available coil/relay or component with auxiliary contacts in the Product field. For terminals, you can select the name of an available terminal strip. For PLC inputs/outputs, you can choose the component name from the list of available PLC components.

M.8. INFORMATION ABOUT SYMBOL DATABASES

M.8.1. CREATING A NEW DATABASE


If you want to create a new empty symbol database, right-click in the empty space of the Symbols area and execute the **New Symbol Database** function in the pop-up menu.

M.8.2. DIRECTORIES AND NAMES

SEE Electrical stores the symbol databases in the directory defined in the **System settings** dialogue, accessible via the **File ► System settings** button, which can be executed only if no project is open.

A database file has the .SES extension.

You can create symbols and save them in your own symbol databases. It is not allowed to perform

changes in the supplied symbol databases marked with a  icon.

M.8.3. WORKING WITH SYMBOL FOLDERS

When the symbol database or a symbol folder is selected, you can right-click and choose the **New Folder** pop-up command in order to create a new subgroup for components.

Type in the name of the **New Folder**.

The new symbol folder is arranged in alphabetical order in the tree of symbol folders.

If you wish to **Delete Folder** or **Copy Folder**, select the symbol folder, right click, and choose the appropriate command from the pop-up menu. To rename a symbol folder, choose the **Properties...** pop-up command and type in its new name.

A warning message appears before deleting, you must confirm this process.

The symbol folder has been deleted including all symbols within it.

The renamed symbol folder can be relocated to another place within the database because the symbol folders are arranged alphabetically.

After copying a symbol folder, it can be pasted into another place by using the **Paste Folder** command from the pop-up menu.

M.8.4. WORKING WITH SYMBOLS IN THE DATABASE

- Select a symbol and right-click to choose the respective pop-up command in order to delete, rename, and copy the symbol.

A renamed symbol can be relocated to another place within the database because the symbols are arranged alphabetically.

After copying a symbol, it can be pasted into another symbol folder by using the **Paste Symbol** command from the pop-up menu.

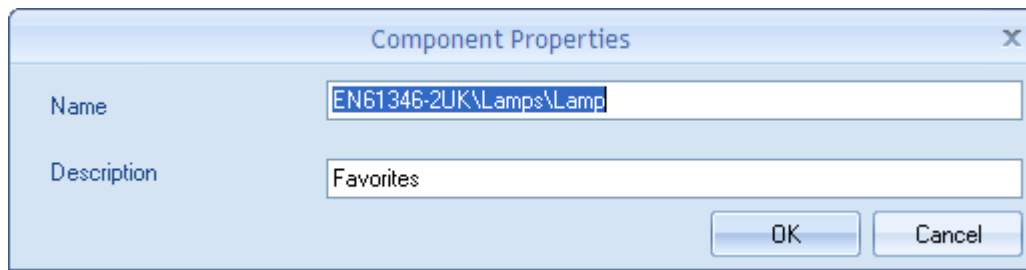
M.8.5. FAVORITES SYMBOL FOLDER

You can create your own, user-defined symbol folder, named "*Favorites*". By adding a symbol to this folder, you will create a shortcut to the referred symbol, whichever database it belongs to. Right-click the desired symbol and execute the **Add to Favorites** pop-up command. The selected symbol is added to the *Favorites* symbol folder.

You can add any symbols, selected according to your needs, to the *Favorites* symbol folder. **Favorites** are stored in the registry settings as a string list in the **General/Favorites** key.

The *Favorites* symbol folder is related to every module, e.g. *Circuit Diagram*, *Installation*, etc. It automatically expands, and clears, and collapses at filtering.

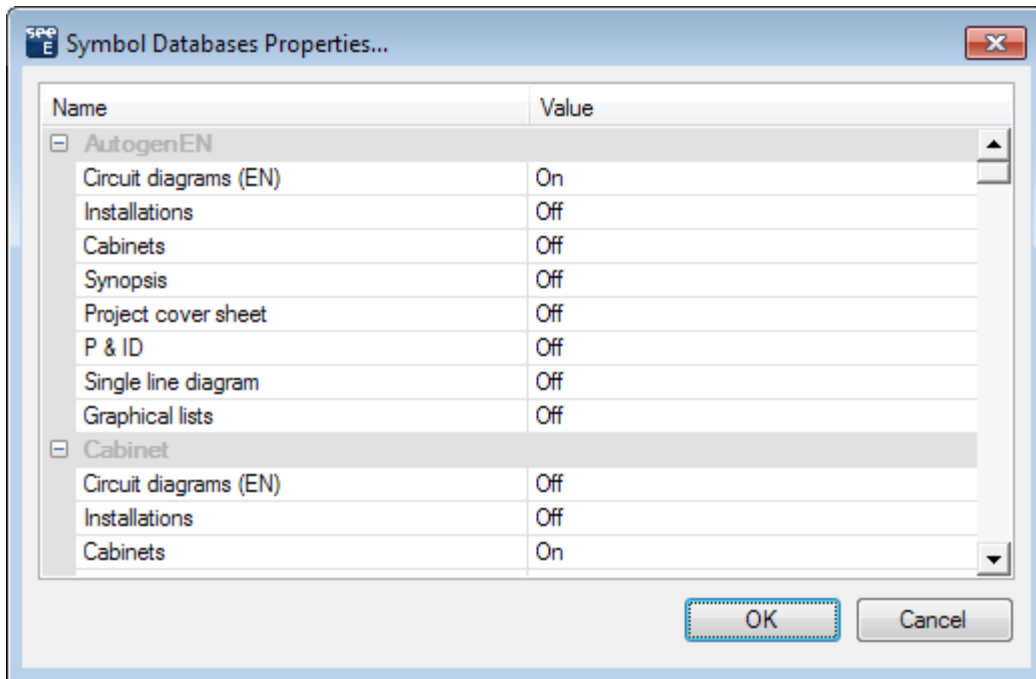
When you right click a symbol from this folder and execute the **Properties...** command, the favourite symbol properties – for the present case – will display the full link path.



M.8.6. SYMBOL DATABASE CONNECTION TO MODULES

Since the number of the symbol databases within *SEE Electrical* is considerable, the symbol databases are allocated to the respective modules before they are displayed. So, a particular symbol database is connected to the module used.

You can specify which databases to allocate. To do this, right-click within the **Symbol explorer** and select the **Properties...** pop-up command. In the window that appears, customize the visibility of the symbol databases (**ON** or **OFF**) and click **OK**.



This also means that the appearance of the **Symbol Explorer** (typically displayed in a more simplified and limited way) will vary depending on the module that you have activated, since all non-relevant symbol databases are left out.

N GROUPS

Several components including wires and potentials (such as Reverse circuit breaker, interlock) can be saved in the symbol database.

In this way you can save time working with repeating circuit groups.

Before saving the group in the symbol database, you can choose its performance.

- ✓ **Loose group**

The single symbols, wires and potentials of the group are available for single manipulation, that is, they can be moved, copied or deleted.

✓ **Component group**

The component group must remain grouped, i.e. it can be moved or deleted as a whole.

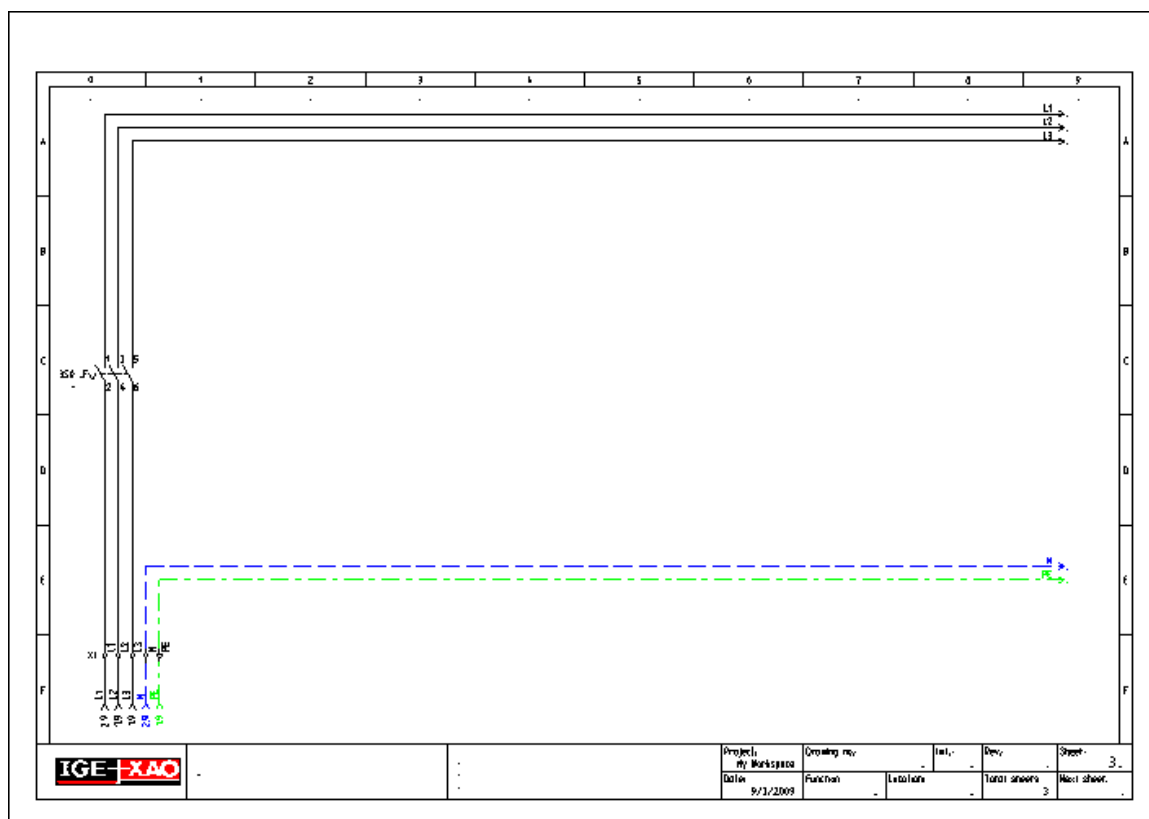
After ungrouping, the single symbols, wires and potentials in the group can become accessible again and can be used individually, if needed.

In both cases you can edit the component names and the potential names.

N.1. CREATING A GROUP

Using the *Power supply* example, we will illustrate how the two kinds of groups are created.

Exercise 13-1: Draw the required potentials, insert the symbols for terminals and switchgear. Use the known commands.





Exercise 13-2: Save the group for further usage in the symbol database.

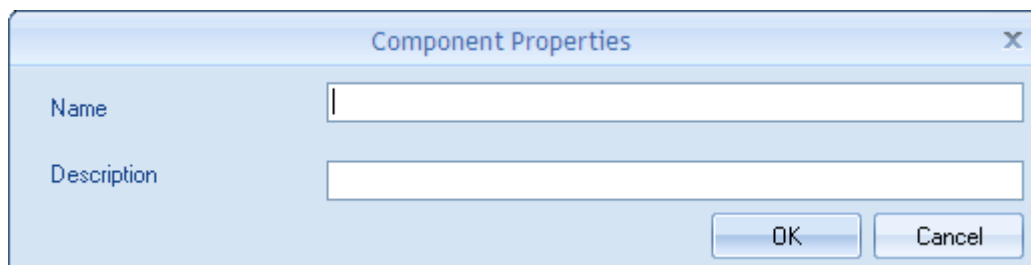
Before storing the group in the symbol database, decide what its performance will be: Single symbols, wires and potentials can be manipulated single again, i.e. they can be moved, copied or deleted (**Loose group**).

The group must be manipulated as a whole (**Component group**), i.e. it can only be moved or deleted as a group. (If desired, it can be ungrouped so that single symbols, wires and potentials in the group can become accessible and can be used individually again.)

In both cases you can edit the component names and the potential names at any time.

Exercise 13-2a: First, save the supply as a loose group.

1. Activate the Symbols Explorer.
2. MySymbols
Double-click the *MySymbols* database. You can save symbols only in this database or in a new one.
3. Right-click with the mouse.
- 4.CO **New Folder**
- 5.# Power supplies
Type in the name of the new symbol folder.
- 6.CA **General**
- 7.CO **Normal (Select panel)**
- 8.+ Select all the components of the power supply using a frame.
Two ways exist for selecting part of the drawing:
If you wish to process only elements that are located entirely within the area, move the cursor from left to right:
The cursor graphic becomes: 
If you wish to process all objects that are even partly included in the area, move the cursor from the right to the left:
The cursor graphic becomes: 
Select the components in the desired way.
- 9.+ Drag the selected symbols into the newly created "*Power supplies*" folder. Hold down the left mouse button while dragging the symbol.



- 10.> Name
- 11.# Power supply 1
Type the group's name.

12.>

OK

The group has been stored in the symbol database.

For example, if you delete page 3 in your project (using the **General** ➤ **Select** ➤ **All** and **Home** ➤ **Page** ➤ **Delete** commands), you can insert the group you have just stored.

Afterwards, the switchgear can be selected and deleted separately. The group can be manipulated as a whole only if all the elements are selected within a frame beforehand.

Exercise 13-2b: Now save the group as a component group.

Before saving, you must block the elements as a group.

1.CA

General

2.CO

Normal (Select panel)

3.+

Select the first point of the frame.

All of the elements have to be within the frame. You can use one of the two possible ways of selection, as described above.

4.+.

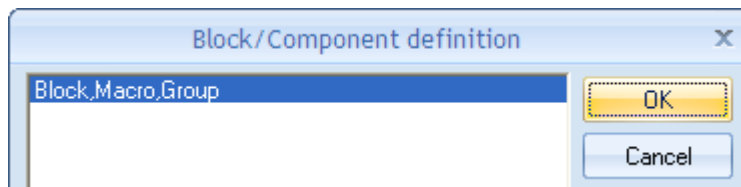
Select the second point of the frame.

5.

Right -click

6.CO.

Block



7.>

Block, Macro, Group

8.>

OK

The parts of the power supply are grouped.

Save the group.

8.

Activate the Symbols Explorer.

9.

MySymbols

Double-click the **MySymbols** database to open it. You can store symbols only in this database or in a new one.

10.CA

General

11.CO

Normal (Select panel)

12.+

Click to select the newly created group. All the components belonging to the group are selected simultaneously.

13.+

Drag the symbol group into the newly created "Power supplies" symbol folder. Hold down the left mouse button while dragging the symbol.



- 14.> Name
 15.# Power supply 2
 Type in the group's name.
 16.> **OK**
 The group has been stored in the symbol database.

Exercise 13-3: Ungroup the Power supply 2 group again.

Delete page 3 again (using the **General** ► **Select** ► **All** and **Home** ► **Page** ► **Delete** commands). Insert the **Power supply 2** group. Select the power supply. All of the elements can be selected only together. If you wish to select a single component, the group must be ungrouped first.

- 1.CA **General**
 2.CO **Normal (Select panel)**
 3.+ Select the group
 4.CA. **Edit**
 5.CO **Explode (Actions panel)**
 The single elements are accessible again (symbols, potentials and wires).
 Select the switchgear again. It is accessible now.

N.2. UNGROUPING SELECTED ELEMENTS

If a single symbol (such as a wire, potential) must be moved or deleted after inserting the group, the group has to be ungrouped. (This does not apply to loose groups.)

- Click to select the group.
- Execute the **Edit** ► **Actions** ► **Explode** command. The group has been ungrouped.

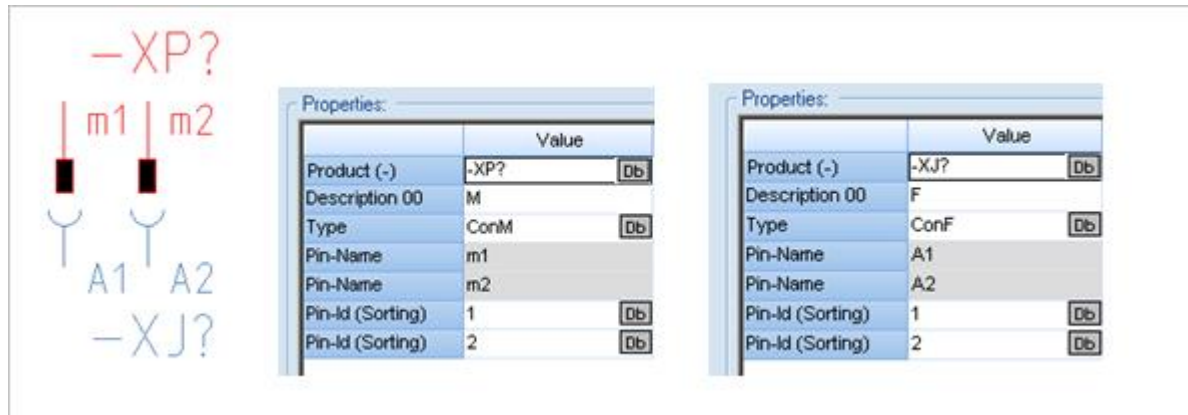
N.3. MOTOR TERMINALS AND SIMILAR COMPONENTS

If several terminals together are frequently needed, such as terminals for a motor, then those terminals must be defined as a group. For the second terminal and every next terminal, you can disable the view of the terminal strip using the **Edit Component** command.

N.4. CONNECTORS: USING SEVERAL PLUGS AND JACKS TOGETHER

If you group several connector symbols as a macro/group, a **Component Properties** dialogue appears listing all pins of this group that belong to the same connector.

Example: Two macro groups have been defined and stored together in the library – one contains the male and the other the female pins.



O BASIC OPERATIONS WITH TEMPLATES AND STANDARD SHEETS

There is a significant difference between templates for a workspace and templates for pages for a circuit diagram or for graphical lists in *SEE Electrical*.

Workspace templates

Workspace templates can be saved using the **File > Save as > Workspace template** command and they can later be chosen when you create a new workspace.

A workspace template consists of data about the component numbering, text size of cross-references, page template (norm sheet) for circuit diagrams, and templates for graphical lists. A workspace template may also contain a completed standard circuit diagram to be inserted if necessary, saved under another name and then changed as desired.

Templates for single pages for circuit diagrams (Page templates)

Page templates can be saved using the **File > Save as > Page template** command and they can later be chosen using the **File > Open > Page template** command at any time.

A page template can be also assigned to the workspace template, then it will be used automatically for creating a new workspace.

In addition to the norm sheet, a page template contains the properties of the current page, such as page size, position of the first top potential, position of the first bottom potential, grid etc.

Templates for graphical lists

You can create your own templates for graphical lists. In addition to the graphics of the norm page, you can define which records from a database list must be recorded into the graphical list. Before creating the graphical list, you can choose the template to be used. Select the respective graphical list in the **Workspace Explorer**, right-click and execute the **Properties** pop-up command. Select the new template and close the dialogue box by clicking **OK**. Now the new template will be used if you create the graphical list.

The template chosen for each graphical list is saved in the workspace template. The templates for graphical lists have to be set before storing the workspace template. They are used by default for creating the relevant graphical list.

0.1. CREATING A STANDARD SHEET

Standard sheets are created user-specific. This chapter will give you general information on how to create a standard sheet.

Exercise 14-1: Switch to an empty page of the workspace (there must be nothing in it except the standard sheet).

Delete the existing standard sheet using the **Edit > Select > All** and **Edit > Actions > Delete** commands. It is important that there is nothing on the current page.

You can disintegrate the existing standard sheet into its single parts later via the **Edit > Actions > Delete** command, right click and execute the **Explode** pop-up command, if you wish to edit it and create a new standard sheet.

Exercise 14-2: Change the properties of your new standard sheet as desired. Define the number of columns in the drawing, too.

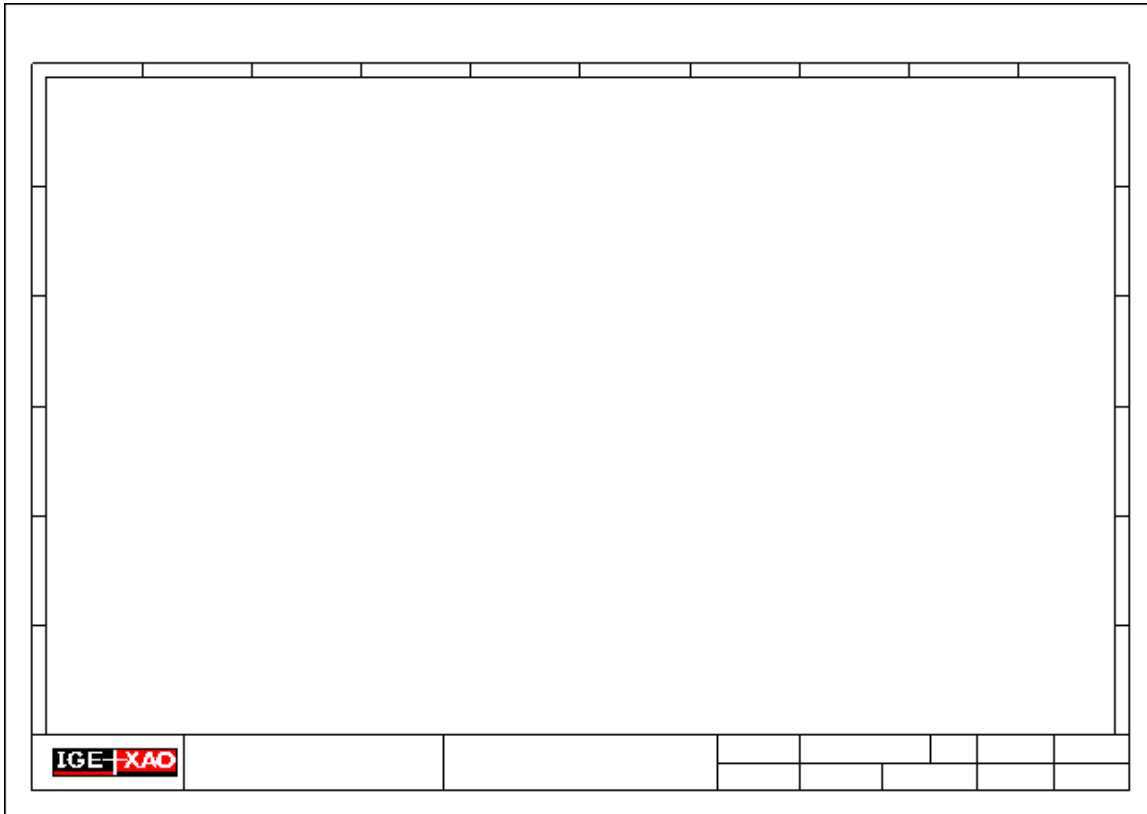
A standard sheet for circuit diagrams is usually drawn in the A3 or A4 format. For cabinet drawings you will need a standard sheet on a scale 1:1.

1. Select the page in the workspace tree that you are editing now.
2. Right click.
- 3.CO Select the **Properties** pop-up command.
The **Properties** window appears in the right pane of the main *SEE Electrical* window, displaying the properties of the current page.
- 4.> X-Extension of Page
Type in the new page size in the "**X-Extension of Page**" field.
- 5.# <new size>
- 6.> Y-Extension of drawing
Enter the new Y-extension of Page.
- 7.# <new size>
Look at the other settings in the dialogue box, position for the first top potential etc. Change other properties as desired. The position for the first top potential must be high enough, but within the section defined in the standard sheet. The definition of sections is described below.
- 8.> Press the Enter key to validate the new settings for each of the modified properties. The properties are changed dynamically.

Exercise 14-3: Construct the graphics for your standard sheet (use the commands from the **Draw** category to draw a line, rectangle, etc.). Draw the geometry for the columns, too.

The complete standard sheet or your Logo can be imported using the **File > Open > CADdy Classic Drawing(s)** or **AutoCAD DXF/DWG Drawing** commands. You can find information about importing DXF/DWG/DXB drawings in the relevant chapter of this training manual.

You can insert the company logo as a pixel image by using the **General ► Insert ► Picture** command. You can find information for inserting Bitmap Objects in the relevant chapter in this training manual.



Exercise 14-4: Insert the texts you need into the norm sheet, such as project name, page name etc.

There is a difference between unchangeable texts and texts customizable to the current project or page. You can fill in the data of the current project as texts into the standard sheet, using text placeholders. For example, you can write a ? sign there.

First, insert the unchangeable texts.

- 1.CA **Draw**
- 2.CO **New Text (Elements panel)**
- 3.> Attribute
The unchangeable texts must have a **"Normal Text"** attribute.
- 4.> Text
- 5.# <Type the text>
- 6.+ Insert the text

Enter the fixed texts and place them into the desired positions.

Project:
Sheet:
Page created by:
Page created date:

To insert a text:

Type the text in the input field, for example: "Project:". Go out of the **Text** dialogue box with the cursor and place the text into the drawing. Go into the input field of the **Text** dialogue box again and type in the next text, etc.

Then insert the texts for the column. These texts have the "Normal text" attribute:

	0	1	2
A	.	.	.

Next, insert the variable texts:

After you have inserted all of the fixed texts, place the changeable texts.

7.# ?

Type a question mark for the text placeholder.

8.> Attribute

Select the kind of the text placeholder.

The data from the **Workspace Information** window, which is available under the **Project** attributes node, belongs to the whole project and applies to all the pages.

The data from the **Page Information** window, which is available under the **Page** attributes, and it is specific for the current page.

9.> <Choose an attribute>

10.+ Insert the text

Choose the attribute for the changeable text, drag it into the drawing, and drop it on the desired location.

Project:	?
Sheet:	?
Page created by:	?
Page created date:	?

To choose an attribute:

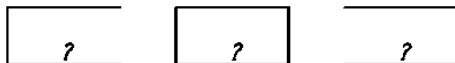
Choose the kind of text, for example **File-name** in the **Workspace** node of the attributes in the **Text** dialogue box. If you go out of the dialogue box, the question mark that you have entered in step 7 is attached to the cursor, and can be placed into the drawing. Go into the **Text** dialogue box again and select the next kind of text in the **Attribute** field, for example *Content / Page* or *Content / Page Created date*. Place the question mark again. After you have inserted all texts, close the **Text** dialogue box.

Exercise 14-5: Select all of the elements that belong to the standard sheet. Group the selected elements to a standard sheet.

- 1.CA **General**
- 2.CO **All (Select panel)**
3. Right-click with the mouse.
- 4.CO **Block**
- 5.> Select this function from the pop-up menu.
- 6.> Page Template, Title block.
- 6.> **OK**

You have now created the first standard sheet. To use it for the next pages, create a page template.

Exercise 14-6: Define texts to use in headers for columns and rows in a symbol.



Example for the vertical symbols

Each symbol is made by lines.


Make sure all three symbols are of the same size, for example 10 mm wide and 5 high.

Each symbol has to contain the text to use (Text attribute "Column name marker" or "Row name marker").

1. Drag the upper left point to library for all the symbols.

Lines are extended / shortened if necessary. The Y-size of a column symbol is not changed (the same is for the X-size of a row symbol).

Exercise 14-7: You must now define the section to use on the page. Open the **"Page Properties"** again.


- 1.CA **Home**
- 2.CO **Properties**
- 3.> Click the  button in the **"Page template sections"** field.

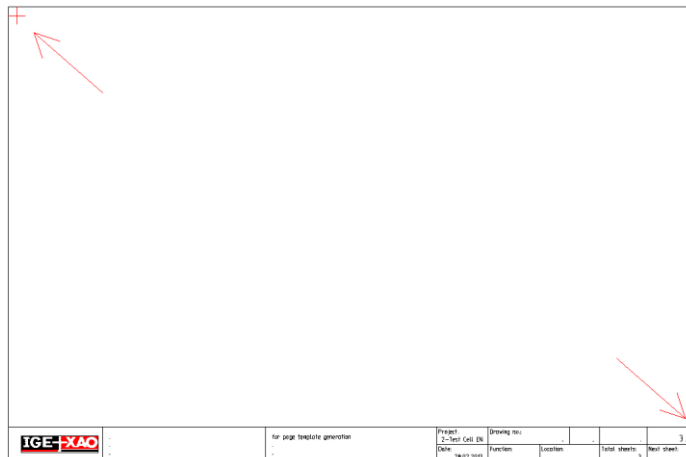


The **Define Sections** window appears. One section is always present by default.

Training manual

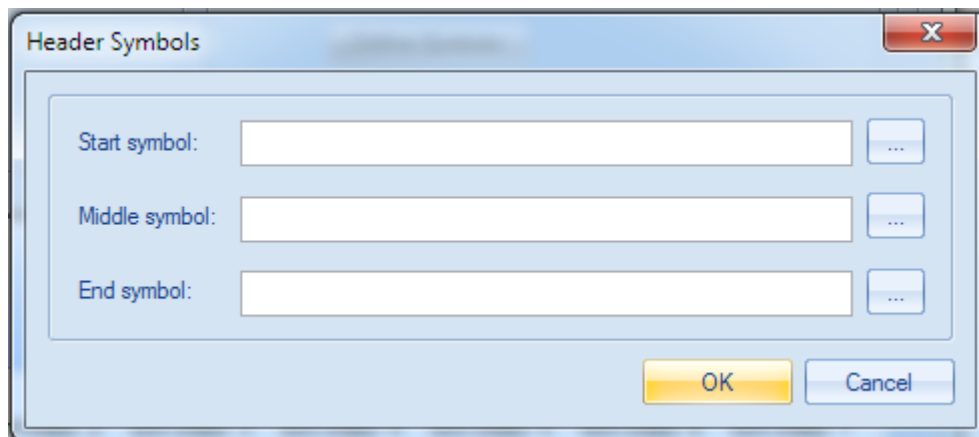
COPYRIGHT © 2013 IGE+XAO. All rights reserved

- 4.> Click the **Add** button to define another section.
- 5.> Insert a name for the section within the "**Name**" field.
- 6.> Click the  button to define the area to use for the section on the page.
The area is defined by clicking to diagonal opposite points (like you do when drawing a rectangle).




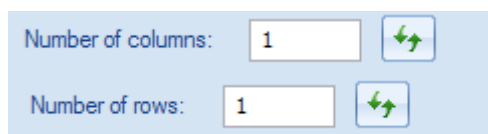
After the area is defined, the X- and Y-coordinate of its start point and its width and height are displayed in the "**Size**" field of the window.

- 7.> Click the **Define Symbols** button if you want to use symbols.
The **Header Symbols** window appears.

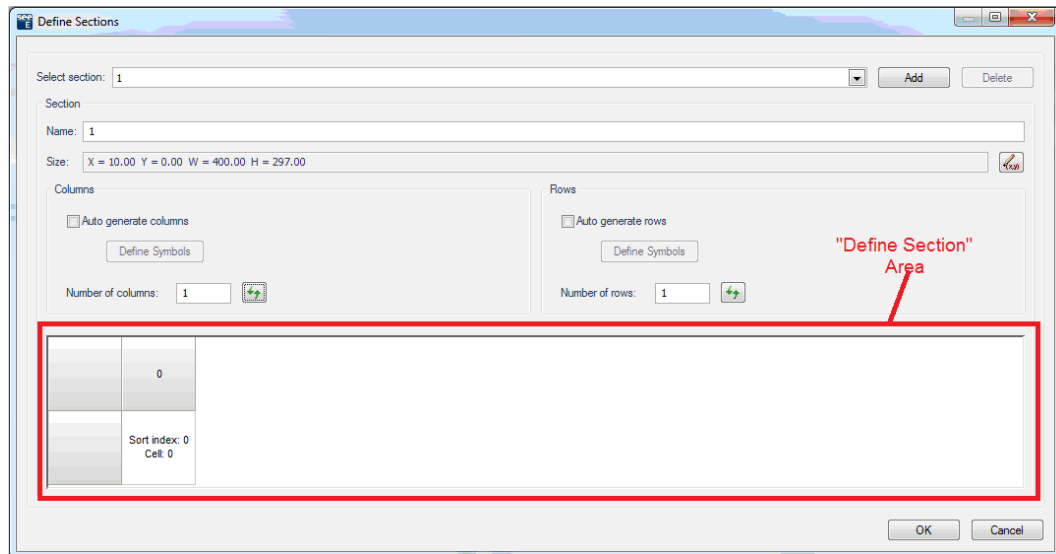


Three symbols are necessary, a start, middle and end symbol.

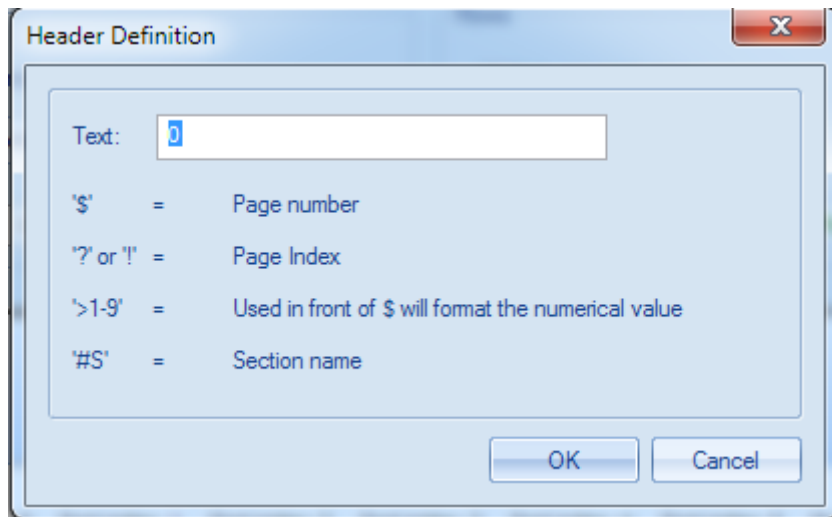
- 8.> Click the  button to browse for symbols.
- 9.> Enter 1 as a value in the "**Number of columns**" and "**Number of Rows**" fields.



- 10.> Click the **Refresh** button  to apply the changes to both
In **Define Section** area only one cell is visible now:



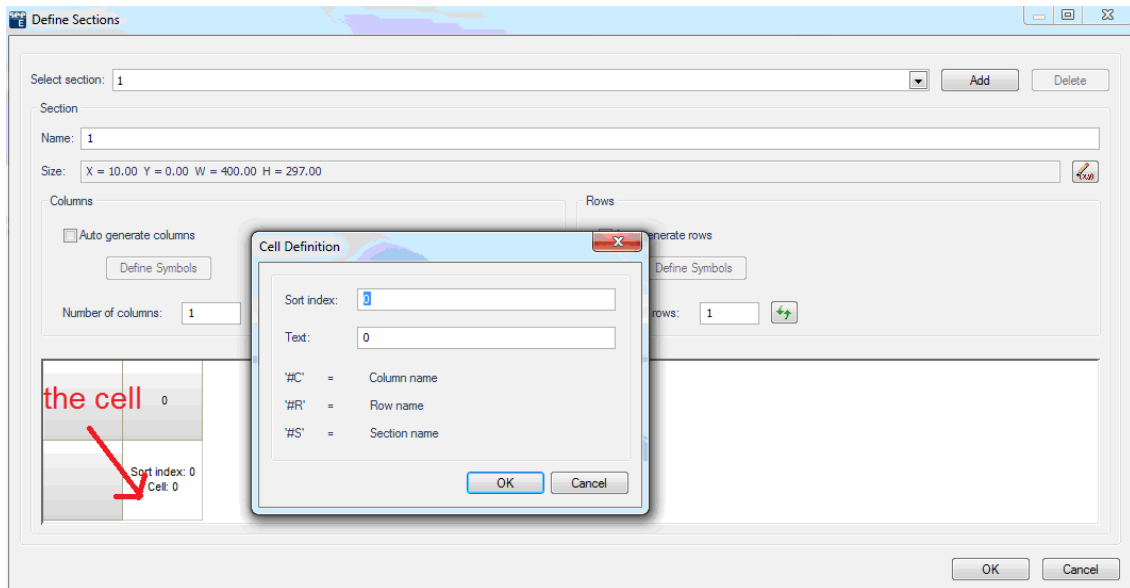
- 11.> Double-click the column header of the existing column to define the rules for the column.



- 12.> Enter 0 (zero) as the column name.
 13.> Click **OK**.
 14.> Define the rules for the row in the same way. (For example, add an A for the row name).
 15.> Double-click the cell to define its rules.

Training manual


COPYRIGHT © 2013 IGE+XAO. All rights reserved



- 16.> Insert 0 (zero) for the Sort index to sort from top to down and from left to right.
- 17.> The content of the text field is used if the cell (column/row) information will be used in the names of the components or cross references. If you want to use this value, you can define the cell information. If you do not want to use the cell information in the component names or cross references, leave the default value in the "**Text**" field.
18. Now define the number of columns and rows you want to have:

Example 1: only columns

Nothing is defined in the header of the rows. You will define only the "**Number of columns**" and will leave the "**Number of rows**" value to 1.

Use the **Refresh** button  to generate the number columns.

Example 2: columns and rows

For example 5 rows and 10 columns will be defined: it is important to create the rows first and then the columns if you want to sort your objects in the database lists from the top left corner of a sheet down first and then to the right like it is done in the default templates delivered.

- Define the rows.
- Use the **Refresh** button  to apply the value.
- Define the columns.
- Use the **Refresh** button  to apply the value.

The columns and rows are automatically made. The rules for name of first column and row are used for the automatically made ones.

19. **Please control the names of the columns and rows and the order and names of the cells!**

If you made a mistake, you can change the rule for each of the columns or rows, but it is easier to start with step 7 from this description again.)

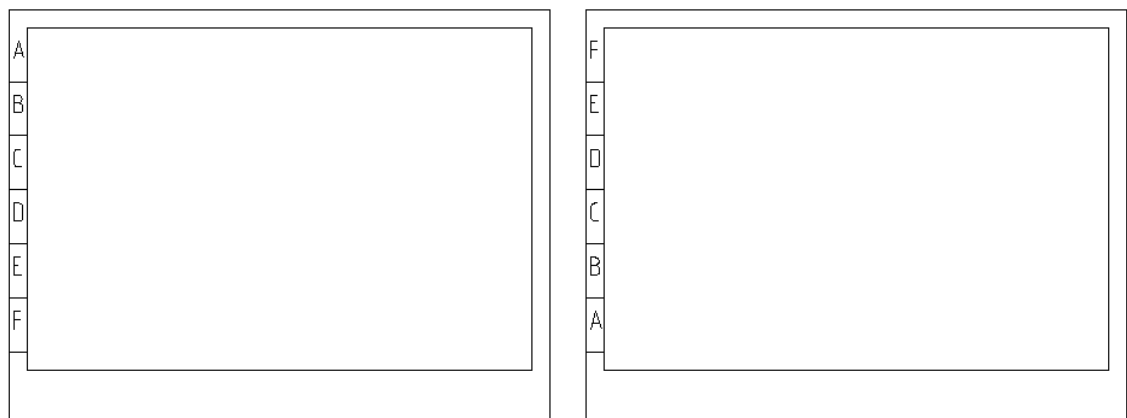
Example for columns only:

	0	1	2	3	4	5	6	7	8	9
	Sort index: 0 Cell: #R	Sort index: 1 Cell: #R	Sort index: 2 Cell: #R	Sort index: 3 Cell: #R	Sort index: 4 Cell: #R	Sort index: 5 Cell: #R	Sort index: 6 Cell: #R	Sort index: 7 Cell: #R	Sort index: 8 Cell: #R	Sort index: 9 Cell: #R

Example for columns and rows (= page coordinates)

	0	1	2	3	4	5	6	7	8	9
A	Sort index: 0 Cell: #C-#R	Sort index: 6 Cell: #C-#R	Sort index: 12 Cell: #C-#R	Sort index: 18 Cell: #C-#R	Sort index: 24 Cell: #C-#R	Sort index: 30 Cell: #C-#R	Sort index: 36 Cell: #C-#R	Sort index: 42 Cell: #C-#R	Sort index: 48 Cell: #C-#R	Sort index: 54 Cell: #C-#R
B	Sort index: 1 Cell: #C-#R	Sort index: 7 Cell: #C-#R	Sort index: 13 Cell: #C-#R	Sort index: 19 Cell: #C-#R	Sort index: 25 Cell: #C-#R	Sort index: 31 Cell: #C-#R	Sort index: 37 Cell: #C-#R	Sort index: 43 Cell: #C-#R	Sort index: 49 Cell: #C-#R	Sort index: 55 Cell: #C-#R
C	Sort index: 2 Cell: #C-#R	Sort index: 8 Cell: #C-#R	Sort index: 14 Cell: #C-#R	Sort index: 20 Cell: #C-#R	Sort index: 26 Cell: #C-#R	Sort index: 32 Cell: #C-#R	Sort index: 38 Cell: #C-#R	Sort index: 44 Cell: #C-#R	Sort index: 50 Cell: #C-#R	Sort index: 56 Cell: #C-#R
D	Sort index: 3 Cell: #C-#R	Sort index: 9 Cell: #C-#R	Sort index: 15 Cell: #C-#R	Sort index: 21 Cell: #C-#R	Sort index: 27 Cell: #C-#R	Sort index: 33 Cell: #C-#R	Sort index: 39 Cell: #C-#R	Sort index: 45 Cell: #C-#R	Sort index: 51 Cell: #C-#R	Sort index: 57 Cell: #C-#R
E	Sort index: 4 Cell: #C-#R	Sort index: 10 Cell: #C-#R	Sort index: 16 Cell: #C-#R	Sort index: 22 Cell: #C-#R	Sort index: 28 Cell: #C-#R	Sort index: 34 Cell: #C-#R	Sort index: 40 Cell: #C-#R	Sort index: 46 Cell: #C-#R	Sort index: 52 Cell: #C-#R	Sort index: 58 Cell: #C-#R
F	Sort index: 5 Cell: #C-#R	Sort index: 11 Cell: #C-#R	Sort index: 17 Cell: #C-#R	Sort index: 23 Cell: #C-#R	Sort index: 29 Cell: #C-#R	Sort index: 35 Cell: #C-#R	Sort index: 41 Cell: #C-#R	Sort index: 47 Cell: #C-#R	Sort index: 53 Cell: #C-#R	Sort index: 59 Cell: #C-#R

20. If you want to change the name of a column or row now, just double click its header and make your changes. Like this it for example is possible to have characters from F to A in the rows instead of A to F.



In this case it could be necessary to change the sort order too.

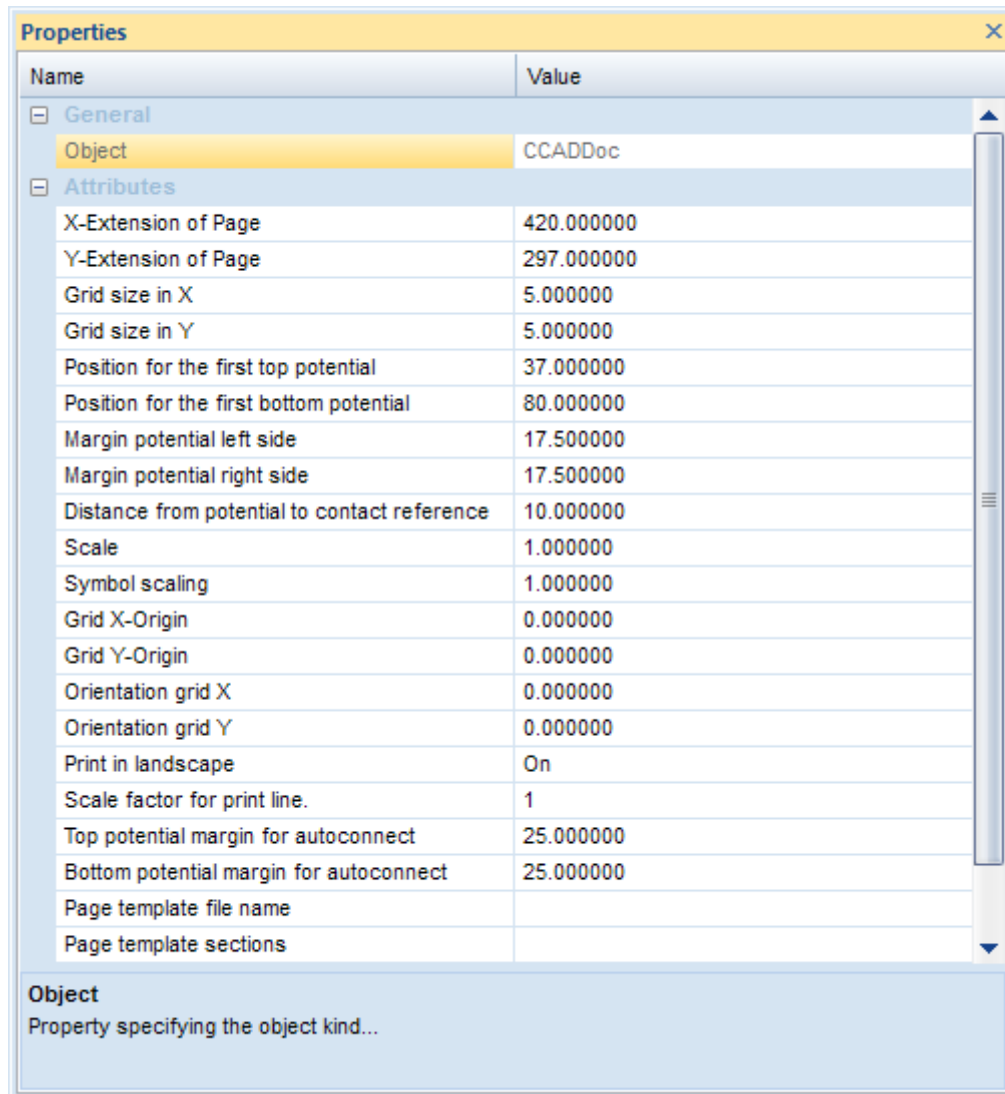
21. Click **OK** to close the window..
22. Now setup/control the other page properties as usual.
23. Keep in mind the positions for the first top and bottom potentials in relation to the section borders. The position for the first top potential is set relatively to the top border of section, the position for first bottom potential is set relatively to bottom border of section. The margins for the left and right potentials are also relative to the section borders.

0.2. CREATING A PAGE TEMPLATE

You can save the standard sheet created in the previous section as a page template. Before you save the page template, look at the current properties, because a page template consists of the standard sheet and the appropriate properties.

Exercise 14-8: Verify and change, in necessary, the properties for the page template.

1. In the Workspace tree, select the page that you are currently editing.
2. Right-click with the mouse.
- 3.CO Execute the **Properties** pop-up command.
The **Properties** window appears in the right pane of the main *SEE Electrical* window, displaying the properties of the current page.
Look at the properties that you can set. Change some properties, as desired.



Name	Value
General	
Object	CCADDoc
Attributes	
X-Extension of Page	420.000000
Y-Extension of Page	297.000000
Grid size in X	5.000000
Grid size in Y	5.000000
Position for the first top potential	37.000000
Position for the first bottom potential	80.000000
Margin potential left side	17.500000
Margin potential right side	17.500000
Distance from potential to contact reference	10.000000
Scale	1.000000
Symbol scaling	1.000000
Grid X-Origin	0.000000
Grid Y-Origin	0.000000
Orientation grid X	0.000000
Orientation grid Y	0.000000
Print in landscape	On
Scale factor for print line.	1
Top potential margin for autoconnect	25.000000
Bottom potential margin for autoconnect	25.000000
Page template file name	
Page template sections	
Object	
Property specifying the object kind...	

- 4.> Press the Enter key to validate the new settings for each of the modified properties.
The properties are changed dynamically.

Exercise 14-9: Save the page template.


1. In the Workspace tree, select the page that you are currently editing.
 - 2.CA **File**
 - 3.CO **Save as**
 - 4.CO **Page template**
 - 5.# <name>
 - 6.> **OK.**
- The page template has been saved. You can load it at any time or assign it as a page template to a project template, as described in the next chapter.

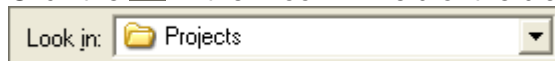
Exercise 14-10: Open the page template on another page.

1. <page>
In the Workspace tree, select the page that you are currently editing (page 1 or 2).
 - 2.CA **File**
 - 3.CO **Open**
 - 4.CO **Page template**
 - 5.> <Your template>
 - 6.> No
- The new page template has been loaded, the project data and page data have been inserted into the new standard sheet, and the existing circuit parts on the page have been kept.

O.3. CREATING A WORKSPACE TEMPLATE

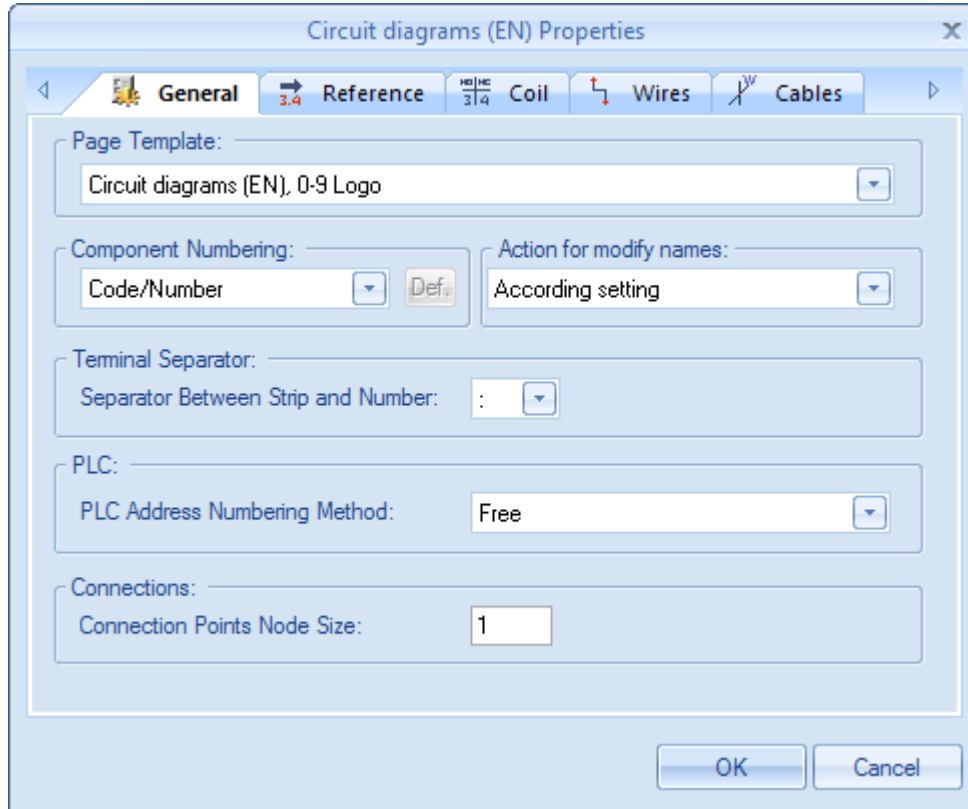
Exercise 14-11: Open an existing workspace template, change it and save it with another name.

- 1.CA **File**
- 2.CO **Open**
- 3.CO **Workspace**
- 4.> Click the  in the "Look in" field of the dialogue box



You can search projects in another directory. By default projects are saved in the...\\Projects directory, templates - in the...\\Templates directory.

- 5.> Select the...\\Templates directory under *SEE Electrical*.
- 6.> *SEE Electrical*
Select a workspace template. Its properties appear in the right pane of the **Open Workspace** dialogue.
- 7.> Open
The workspace template is opened.
8. Select **Circuit diagrams** in the Workspace tree.
- 9.CA **Home**
- 10.CO **Module (Properties panel)**



- 11.> Page Template
12. Select your page template.
Verify all of the properties in this window and the properties of the circuit diagrams.

- 13.> Select a method for naming the components in the workspace from the **Component Numbering** pull-down list within the **Circuit Diagrams Properties** dialogue box.
OK

Exercise 14-12: Save the new workspace template.

- 1.CA **File**
- 2.CO **Save as**
- 3.CO **Workspace template**
- 4.# <name of the template>
- 5.> **Save**

If you create a new workspace by using this template, the newly created page template applies to all new circuit diagram pages of the workspace. If you need to use different templates on different pages, it is possible to change the page template via the **File ► Open ► Page template** command, as described above.

Exercise 14-13: Close the workspace template to prevent undesired changes.

- 1. Click on the workspace name in the Workspace tree.
- 2. Right-click with the mouse.
- 3.CO **Close Workspace**

O.4. QUICKREFERENCE TEMPLATES AND STANDARD SHEETS

Workspace Template (-> **File** ► **Open** ► **Workspace**, Select the ...\\Templates directory, Select template <Name>.SEP)

Workspace Properties (-> <Name>.SEP in the **Workspace Explorer**, pop-up menu)
Function / Location settings
Revision

Circuit Diagram Properties (-> Circuit diagrams in the **Workspace Explorer**, pop-up menu)
Component Numbering, etc.
Select a page template.

Page Template (-> <Name>.TDW)

Create Standard sheet
Draw graphics
Block: Page Template, Title block

Page Properties (-> Page in the Workspace Explorer, pop-up menu)
Number of columns in page
Position of Potentials
Grid size

Save Page template (-> **File** ► **Save as** ► **Page Template**)

Cabinets Properties (-> Cabinets in the Workspace Explorer, pop-up menu)
Select page template, etc.

Page Template (-> <Name>.TDW)
See above

Graphical Lists – List of documents - Properties (-> List of Documents in the Workspace Explorer, pop-up menu)
Select page template, etc.

Page Template (-> <Name>.TDW)
See above

Graphical Lists - List of products – Properties (-> List of Products in the Workspace Explorer, pop-up menu)
etc.

P TEMPLATES AND STANDARD SHEETS FOR GRAPHICAL LISTS

P.1. CREATING TEMPLATES FOR GRAPHICAL LISTS

Exercise 15-12: Create templates for graphical lists.

1. Open the Graphical Lists area.
2. Select the graphical list you wish to create a template for.
3. Right-click with the mouse.
- 4.M Select the **Load Page Template** command from the pop-up menu.
 The newly created template can be loaded and changed. After you delete this template, it is possible to create a new one. (The template opens in its own window, i.e. it does not exist as a page in the Workspace Explorer).
 It was described in the **Creating a Standard Sheet** chapter how to delete an available standard sheet using the **Edit > Select > All** and **Edit > Actions > Delete** commands.
 Now, you must change the available standard sheet.
- 5CA **General**
- 6.CO **All (Select panel)**
7. Right-click
- 8.M **Explode** from the pop-up menu.
 After ungrouping the standard sheet, all the elements become accessible again.
9. You can add, erase and move lines, if needed.
10. You can also edit, erase and add texts.
 There is a difference between unchangeable texts and texts that have to be entered with data from the database lists.
 All of the texts in a template for graphical lists have the "Normal Text" attribute.
 Texts to be filled in with data from the database lists, must be presented always in the format #<number>, such as #120010 for page number. You can find a list of the available numbers for text placeholders at the end of this section or in the "*List Construction Set.SES*" symbol library.
11. A special text placeholder "#Lines 30 7.5" must be presented in the template in order to state the number of lines available in the template (for example 30) and the line-distance (for example 7.5).
 Enter the text exactly in the following format:
 #Lines <number of lines> <line-distance>
 The point is required as a decimal delimiter in the line-distance.
 After you have created the template, group it again.
- 11CA **General**
- 12.CO **All (Select panel)**
14. Right click
- 15.M Select **Block** from the pop-up menu.
- 16.> Page Template, Title block
- 17.> **OK**

Exercise 15-13: Save the template.

1. In the Workspace Explorer, select the current page under Graphical Lists.
- 2.CA **File**


Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

- 3.CO **Save as**
- 4.CO **Page Template**
- 5.# <name>
- 6.> **OK**

Exercise 15-14: Select a template for a graphical list as follows:

You can select the template for the graphical list in the **Properties** dialogue box.

1. In the Workspace Explorer, select the graphical list, which you would like to select a new template for.
2. Right click
- 3.CO **Properties**
- 4.> Select your new template.
Click the  icon to select the desired template. A pop-up list appears, including all of the templates for graphical lists.
Select the template.
- 5.> **OK**
The setting for the template has been saved within the workspace.
If you create a new workspace template, you can select new templates for graphical lists, too.

Examples for text placeholders from database lists:

You will find detailed information about all the lists in **The Graphical Lists** chapter of the software's Help.

List of products

Number	Description
120010	Page number
120020	Page Index
160020	Cell
140020	Function (=)
140050	Location (+)
160010	Product
160030	Description
160040	Type1
160041	Type2
160042	Type3
160043	Type4
160044	Type5
160045	Type6
160046	Type7
160047	Type8
160048	Type9
160049	Type10
165150	Free text 01
165151	Free text 02
12000170	Manufacturer

P.2. FORMATTING TEMPLATES FOR GRAPHICAL LISTS

The format of graphical lists (except for Terminal matrix, Terminal plan, Terminal row picture, Cable Terminal row plan, connector matrix, connector plan and Assembly list) can be customized by using: sorting, filtering, defining first page, page break after changing the defined values.

The control is realized via the page template using some keywords defined with the "#" sign:

```
#PageBegin 100
#PageBreak 140020, 140050
#Orderby 140020, 140050, 160010
```

<pre>#Lines 30 7.5 #WHERE [140020]="=A1" AND [120010]>10</pre>		
Function (=)	Location (+)	Product (-)
#140020	#140050	#160010

<pre>#PageBegin <page number></pre>	<p>The graphical list will start from the defined page number. Example (for List of Products): <pre>#PageBegin 100</pre> The first graphical List of Products always begins automatically from page number 100. <pre>#PageBegin ?</pre> Each time you generate the List of Products, a question about the start page number will appear. Using <pre>#PageBegin [<List ID>]</pre> it is possible to assign consecutive page numbers to all graphical lists. Example: First, the Document List is created. The page numbers of the Product List will be followed on seamlessly to the last page of the Document List. So the placeholder <pre># Page Begin [3011]</pre> is required in the Standard sheet symbol of the page template for the Product list. For the IDs of the various lists, see the chapter "Generate all desired graphical lists in one step" below.</p> <p>It is possible to define one page number and different indexes for pages that are generated for one kind of a graphical list. In this case, the page number will not be changed. The definition has to be made in the following way: <pre>#PageBegin <page number> <separator> <page index></pre> The page index has values in the range from A to Z. The numbering is executed in the same way as the numbering for the Excel columns. For example: <pre>#PageBegin 10 / A -> the result is pages 10.A, 10.B, 10.C.....</pre></p>
---	--

	<p>#PageBegin 10 / AA -> the result is pages 10.AA, 10.AB, 10.AC.....</p> <p>You have to make sure that the number of pages does not exceed the number of digits given for the index.</p> <p>Example: The X1, X2...X50 terminal strips are used in a workspace. The index that is used is A, B, C...</p> <p>After the creation of Index Z, the next terminal strip is generated on a page with index AA, but, in the project tree, the AA page is sorted after page A and the sorting will be the following:</p> <pre>A X1 AA X27 AB X28 B X2 etc.</pre>
#PageBreak <value>	<p>A new page will begin, if the defined value changes.</p> <p>Example (List of Products): #PageBreak 140020, 140050</p> <p>Each time the Function (140020) or Location (140050) changes, a page break is inserted and the list continues on a new page. The list can be sorted by the function in this way.</p>
#LineBreak <value>	<p>An empty line is inserted if the defined value changes.</p>
#ColumnBreak <values>	<p>The #ColumnBreak command allows you to switch to the next column if the given value changes (if a #PageBreak command exists and the value defined in there is the same defined in #Column Break, the page is changed).</p> <p>The same rules as for the #LineBreak command are valid.</p>
#Pos. <value>	<p>The placeholder #Pos. command enables you to get a line numbering (= consecutive numbers) in simple graphical lists like document list, product list, cable list etc. (line numbering is already possible in wiring list).</p> <p>The placeholder can be used together with a start value for the line number #Pos.[n], for example #Pos.[100] starts with value 100. If a start value for the line number is not found, the line number will start with value 1. The increment for the number always is 1.</p> <p>The line numbers cannot be used in terminal matrix, connector matrix, cable plan, cable terminal-row plan, products assembly, terminal plan, terminal row picture plan and connector plan lists.</p>

#OrderBy <values>	<p>The graphical list is sorted according to the defined values whose codes (#number) are stated after the keyword #OrderBy.</p> <p>Example (List of Products): #OrderBy 140020, 140050, 160010</p> <p>The List of Products is ordered, at first, by Function of the products in ascending order, then by Location of the products in ascending order, and then by component names in ascending order.</p> <p>You can sort the List of Products according to types #OrderBy 160040 or according to component names #OrderBy 160010.</p>
----------------------	---

	<p>Sorting in descending order is also possible: <code>#OrderBy 140050 DESC, 160010</code> The List of Products is ordered at first by Function of the products in descending order, then by component names in ascending order. In Document List <code>#OrderBy</code> is not available</p>
#Where [argument]	<p>Records in the list are filtered according to the defined value. Example (List of Products): <code>#Where [140050] = "+P1" AND [120010]>10</code> The List of Products will contain only components with Location "+P1" and placed on page number >10. Or <code>#Where [160040] = "M10" OR [160040] = "M20"</code> The List of Products will contain only components with types M10 and M20. Example (List of Documents): <code>#Where [160040] <> " not used "</code> The Product list contains all products without the ones with the type "not used". <code>#Where NOT [160040] IS NULL</code> or <code>#Where [160040] IS NOT NULL</code> or <code>#Where [160040] <> " "</code> The Product list contains all products without the ones with an empty type.</p> <p>Another example for use (Product list again): <code>#Where Instr ([160010]), "M") >0</code> The component list contains only components with the code "M" in their name. Next example for use (Product list again): <code>#Where NOT Instr ([160010]), "M") >0</code> The component list contains no components any more, that have the code "M" in their name. Example of use: Cable core list: The "Cable Type" is sometimes not entered (i. e. undefined, therefore you cannot filter on it, but this is necessary, since all entries are required, except for these with the value in the cable type: <code># WHERE (((If (IsNull ([160 200]), "", [160 200])) <> "Not Required"))</code> Using the If function you can determine whether a term is true or false. If the term is true, If returns a value, if the term is false, If returns another value. You define the values, which are returned by If. The If function syntax uses the following arguments: DESCRIPTION OF ARGUMENTS expr required: a term that you want to evaluate. Truepart required. Value or term that is returned if expr is "true". falsepart required. Value or term that is returned if expr is "wrong". Thus, IIF returns the value " ", if 160200 has no entry (and thus</p>

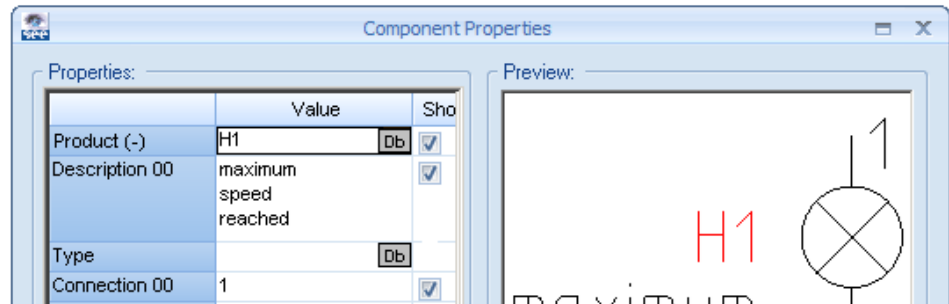
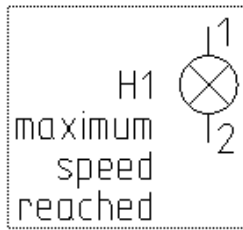
Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

	<p>we get an entry) and the value for the cable type, when 160 200 is an entry. Using Where you can give out only that, what is not equal to the term "Not Required".</p> <p>In Document list there are specific rules, refer to the specific list to get to know about them.</p> <p>In Terminal Matrix and Terminal Plan, Connector Matrix and Plan, Terminal Row Picture Plan, and Cable Plan, this attribute is not available.</p> <p>In other lists might exist several restrictions.</p>
#Columns <number of columns> <offset for next column>	<p>It is possible to output more than one column in multiple graphical lists that are filled line by line: Documents list, Product list, Part list, Cable list, Cable wire list, Terminal list, ... In Terminal Matrix and Terminal Plan, as well as in Terminal Row Picture, Cable Plan and Products Assembly this option is not available.</p>

List of Documents							
#Lines 30 7.5	#columns 2 150						
Function (=)	Location (+)	Sheet	Kind of Document	Description	Function (=)	Location (+)	Sheet
#180015	#180018	#120010#120020	#180010	#120100			

	<p>If two or more columns have to be generated, the page template must contain a text with "normal" attribute and the following content:</p> <p>#Columns <number of columns> <offset for next column></p> <p>For example, #columns 2 150 like in the example shown above.</p> <p>The second column needs to contain the geometry and the texts for the headline but no placeholder IDs such as #180015.</p> <p>The #ColumnBreak command allows you to switch to a new column if the defined criteria is executed (if #PageBreak exist, it has a higher priority).</p> <p>Syntax: #ColumnBreak <Value></p> <p>The rules are similar to the ones valid for the #PageBreak.</p> <p>Additionally, there are commands that allow you to define in which direction will be filled the columns.</p> <p>#RightThenDown: place the first entry in the first line, place the next entry in the first line of the next column</p> <p>#DownThenLeft (=default if nothing is defined): fill all lines in the first column, then start with line 1 in the second column</p>
Output multiline text in a single line in lists	<p>If, for example, the text "description 00" is to be split in three lines in a component to fit the space available in the circuit diagram, it is possible to display this text in one line in the product list.</p>



	<p>If, for example, the text "description 00" is to be split in three lines in a component to fit the space available in the circuit diagram, it is possible to display this text in one line in the product list.</p> <p>In the Terminal Matrix and Terminal Plan, as well as in the Terminal Row Picture, Cable Plan, Connector Matrix, Connector Plan and Products Assembly this option is not available.</p> <p>To output multiple lines in one single line, the page templates must be enhanced in the following way:</p> <p>The ID for the "Description 00" text is 160030.</p> <p>If in the template for the product list is present the #160030 \$SingleLine Placeholder, the text is displayed in one line</p> <p>maximum speed reached</p> <p>If no separator is defined, a blank one is used after each linebreak.</p> <p>If a separator is defined, it is used. If in the template for the product list is present the #160030 \$SingleLine [-] placeholder, the text is displayed with a – as a separator.</p> <p>Maximum-speed-reached.</p>
Use word wrap in an expression	<p>It is possible to use a line break inside a text added to a simple graphical list, to make the text fit a column. To use the wrapping the page templates of the simple graphical list be defined in the text that contains the ID. Syntax for wrapping is: %wt(<number of characters>). If no maximum number of characters is defined, then wrap is ignored.</p> <p>All other column format definitions are valid. The wrap text algorithm is applied to the text after all other formatting is made.</p> <p>Examples:</p> <p>1) if there is a column with ID #120100 (Page Description 1), the wrapping is defined like this:</p> <p>#120100 %wt(10)</p> <p>-> the wrap is made after 10 characters.</p> <p>2) if content of a column is made from several Ids (=a formula is used) like #fu[#120100][,][#120110], the wrap definition must be placed inside the information about the first ID to use: like this</p> <p>#fu[#120100 %wt(7)][,][#120110]</p> <p>The wrapping is applied on the entire formula text.</p>
Output multiple values in one field in lists	<p>In the simple graphical lists like Document List, Product List etc. it is possible to combine several values in one entry. For example for components with more than one type the product list can now be</p>

	<p>generated in a much nicer way.</p> <p>The combination of several values is done using the #FU command. This command uses the following syntax:</p> <p>#FU[#<1st ID>][Separator][#2nd ID][;][#3rd ID]</p> <p>Examples:</p> <ul style="list-style-type: none"> - Combine 2 types in the Product List: (Separator ;): #FU[#160040][;] [#160041] - Combine Page description 1 and 2 in the Document List (Separator space): #FU[#120100][] [#120110]
<p>The number of digits and the length of the text can be controlled.</p>	<p>Some formatting attributes have been introduced to allow the formatting of information displayed in the graphical lists (none of these formatting attributes can be combined with another one of these attributes).</p> <p>In the Terminal Matrix and Terminal Plan, as well as in the Terminal Row Picture, Cable Plan and Products Assembly this option is not available.</p> <p>Attention: only one of this formatting attributes can be used.</p> <p>Numerical values:</p> <p>Control the number of decimal places</p> <p>Add to your template a text placeholder with "Normal" text attribute that contains the command</p> <p>#<ID of text attribute> %.<n>f or #<ID of text attribute> %d</p> <p>Add this if three decimal places are necessary for the amount (ID 180040) in the part list. The result for the amount in the part list is 5.004.</p> <p>#180040 %d</p> <p>Add this no decimal places are necessary for the amount (ID 180040) in the part list. The result for the amount in the part list is 5</p> <p>For example:</p> <p>#180040 %.3f</p> <p>Add this if three decimal places are necessary for the amount (ID 180040) in the part list. The result for the amount in the part list is 5.</p> <p>Control the decimal places and add static text</p> <p>Add a text placeholder with "Normal" text attribute to your template that contains the command</p> <p>#<ID of text attribute> %<n>.f<text></p> <p>For example:</p> <p>#160101 %.2fmeter</p> <p>Add this if two decimal places are necessary for the length of the cable (ID 160101) in the cable list and the text "meter" must be inserted after the value. The result is 5.45 meter.</p> <p>Use full integer formatting</p> <p>Add to your template a text placeholder with "Normal" text attribute that contains the command</p> <p>#<ID of text attribute> %0<n>d</p> <p>For example:</p> <p>#120010 %03d</p> <p>Add this if three integer values are necessary for the length of</p>

	<p>the page number (ID 120010) in the Document list. If are present the pages 5, 11, 123 and 444, the result is the following: 005 or 011 or 123 OR 4444</p> <p>The #<ID of text attribute> %0<n> d command states that the 0 is added to the value as often as necessary to reach the minimum length of the string defined by <n>. Is a value exceeds the minimum length it does not change.</p> <p>Text strings</p> <p>Control the length</p> <p>Add to your template a text placeholder with the "Normal" text attribute that contains the command</p> <p>#<ID of text attribute> %<n>s</p> <p>For example</p> <p>#160030 %30s</p> <p>Add this if only 30 characters fit into the field available for the "Description 00" text (ID 160030) in the Product list. The text is truncated after 30 characters. If, for example, are used 50 characters, only 30 are displayed.</p>
--	--

Generate all desired graphical lists in one step

Within *SEE Electrical*, you are able to generate all desired graphical lists in just one click.

Using internal codes within the graphical lists, you can define a "chain reaction" at graphical lists generation. The internal codes also allow you to specify consecutive page numbering valid for *all* lists, e.g. the first document obtains page number 1, the next - 2 and so on, until completed. This enables more flexible handling of graphical lists.

The following attributes must be specified within the list:

#NextList List ID

Example :

#NextList 3100

There are no brackets [] allowed.

For this example, the next list generated will be 3100. This code must be inserted in the first list that you generate. The first list starts a "chain reaction", i.e. this code will automatically call and generate the next kind of graphical list (in this example, list type 3100).

The IDs for the types of graphical lists are as follows:

- 2000 - Other documents
- 3001 - Documents list
- 3011 - Products list
- 3020 - Terminals list
- 3025 – Connector list (only *Advanced*)
- 3026 Connector Pin list (only *Advanced*)
- 3030 - Cable list
- 3031 - Cable-core list
- 3050 – PLC I/O list
- 3060 - Wires list
- 3100 - Parts simple list
- 3101 – Parts list, Simple
- 3102 - Spareparts list (from *Standard*)
- 3103 - Spareparts list, Simple (from *Standard*)
- 3104 - Terminal matrix (from *Standard*)
- 3105 - Cable plan (from *Standard*)

- 3108 - Connector matrix
- 3106 - Cable terminal row plan (only *Advanced*)
- 3110 - Parts simple F&L sorted
- 3112 - Parts F&L sorted (only *Advanced*)
- 3220 - Products assembly (only *Advanced*)
- 3225 – Aspect Functions (only *Advanced*)
- 3226 – Aspect Locations (only *Advanced*)
- 3235 - Cables list F&L Sorted (only *Advanced*)
- 3236 - Cable core list F&L Sorted (only *Advanced*)
- 3270 - Terminal plan (only *Advanced*)
- 3271 - Terminal row picture (only *Advanced*)
- 3273 - Connector plan
- 3280 - Multicores list (only *Advanced*)
- 3285 - Multicores wires list (only *Advanced*)

PageBegin [List ID]

Example:

PageBegin [3000] –

The brackets [] must be used!

Page numbering of the current list continues the page numbering from the last page number generated in the list with ID 3000.

PageBegin [List ID] \$

The brackets [] must be used!

Example: [3000] \$ - Pages have the numbers in the function groups in list 3000: consecutive page numbering independent from the type of the list in which the data is inserted.

PageBegin <page number> (for ex. 20)

Example: 20 - The page numbering starts at page 20.

PageBegin <page number> % (for ex. 20%)

Example: 20% - Page numbering starts at page 20 for each function group.

P.3. GENERATE COVER SHEETS FOR GRAPHICAL LISTS

All graphical lists can generate a cover sheet if necessary.

This is done with the text #CoverSheet <"template name for cover sheet"> (text attribute "normal" text) inside the page template of the graphical list.

Example for document list:

#CoverSheet "Cover Document List".

The page template for the cover sheet *CoverDocumentlist.TDW* must be located in ...\\templates folder.

In the properties for the graphical list is defined the template to use for list generation and not the one to use for the cover sheet.

The cover sheet for the list is always loaded. Inside a coversheet no data source will be used. So the coversheet is a static drawing where only the default objects are shown and project attributes are read.

Q EXTERNAL DATA TRANSFER

Q.1. DATA TRANSFER THROUGH DWG/DXF/DXB FORMAT

Data transfer between various CAD-systems is possible through the *DWG/DXF/DXB* format. However, it is not possible to transfer data about electro-technical logic while transferring data through these formats.

Exercise 16-1: Import data into *SEE Electrical*.

Create a new empty page first.

- 1.CA **File**
- 2.CO **Open**
- 3.CO **AutoCAD DWG/DXF/DXB Drawing...**
- 5.> Files of Type
- 6.> AutoCAD DXF files (*.dxf)
Select the desired type of file.
- 7.> Look in
Select the folder where the file is located.
- 8.> Select the file.
- 9.> "Fit contents to Page"
The properties of the imported drawing have to be adjusted. The option must be ticked.
- 10.> Delete Page before import
The available drawing (i.e. the empty standard sheet) must be deleted. The option must be ticked.
- 11.> Open
The file has been imported and is displayed as a drawing.
You can edit it now as desired.




Exercise 16-2: Save data within *DXF* format.

1. Switch to any page in the workspace that you want to save within a *DXF*, *DWG* or *DXB* format.
- 2.CA **File**
- 3.CO **Save as**
- 4.CO **AutoCAD DWG/DXF/DXB Drawing...**
- 5.> Save as type
- 6.> DXF v2004 File format (*.dxf)
Select the desired type of file.
- 7.> File name
- 8.> <name>
Type the name of the file.
- 9.> Save in
- 10.> Select the folder where the file must be saved.
- 11.> Save
The file has been saved.
Using the same approach, you can save *DWG/DXF/DXB* files in all pages of your project by activating the **File > Save as > AutoCAD DWG/DXF/DXB Workspace...** command.

Q.2. PIXEL IMAGES TRANSFER

You can insert pixel images (JPG, BMP, etc.) in *SEE Electrical* drawings.

Exercise 16-3: You can import a logo as a Bitmap object.

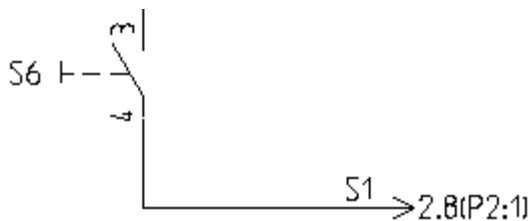
- 1.CA **General**
- 2.CO **Picture (Insert panel)**
The following formats are supported: *PCX*, *JPG* and *BMP*.
- 3.+ Select the first point of the frame, where the Picture is to be inserted. The frame is defined as a rectangle by two opposite corner points.
- 4.+ Select the second point of the frame for the Picture.
5. Select the desired file.
- 6.> Open
You have just inserted the Bitmap object into the drawing.
You can modify the Bitmap object.
7. Click the  icon to enable trackers on selected elements.

- 8.+ Left-click the bitmap object:
You can see the trackers on each corner point and between them.
- 9.+ Use the trackers to change the size of the bitmap object.
- 10.+ Click within the bitmap object, hold the left mouse button and move the object to the desired position.
11. Click the  icon again to switch the trackers off.
12. To rotate the bitmap select it and use the **Rotate** pop-up command to rotate it at 90 degrees.

R PROCESSING A WORKSPACE

(Standard)

R.1. ADDITIONAL OPTIONS FOR CROSS-REFERENCES

Exercise: Show targets for cross-references.



1. Select the cross-reference, which you want to show the target for.
2. Right-click to open the pop-up menu..
- 3.M. **Properties**
- 4.# Change the default setting for the **Show target** value from **Off** to **On**.
Type in the desired text.

R.2. PLC FUNCTIONALITIES IN THE STANDARD LEVEL

R.2.1. USING THE PLC DATABASE

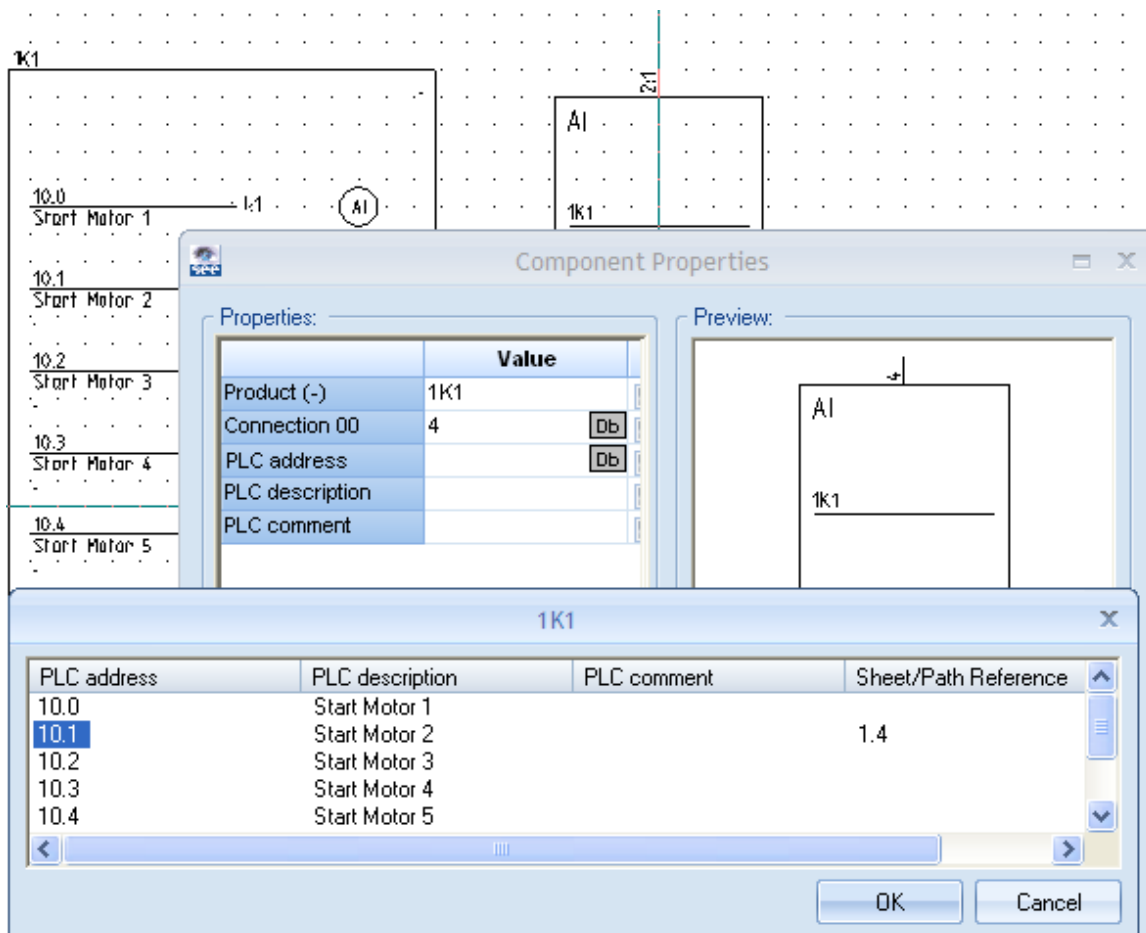
In *SEE Electrical Standard* level, there is a possibility to make sure you assign only those PLC addresses used in the rack, if there is one.

To do so proceed as follows: when giving the name and the PLC address to a PLC I/O, give the name first. After that you can click the **Db** button in the "Connection" line or in the "PLC address" line.

Connection 00	4	Db
PLC address	?	Db

After that a window appears, that shows all the PLC addresses defined in the rack. If PLC description or PLC comment is defined in the rack, you see the appropriate information.

An entry in the column "**Code/Cell reference**" shows, this I/O is already used.



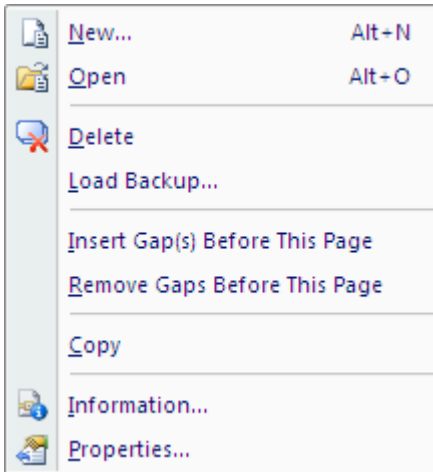
Review

If a Rack is already placed, it will be checked while inserting the inputs/outputs if the address is already available, and if the number of connections at the input/output complies with the number of the connections at the Rack input/output. An error message can appear.

In the Check PLC Connections database list, such errors are documented if you do not correct them.

R.3. ADDING AND DELETING PAGES

If you select a page in the **Workspace Explorer** and you right-click with the mouse, a pop-up menu appears, allowing you to add pages before the current page (**Insert Gap(s) Before This Page**) or remove empty pages before the current page (**Remove Gaps Before This Page**).



If you use a page based numbering, you can choose whether to change the component names or not.

The functions are available for each kind of plans, as well as for graphical lists. After generating terminal matrices, it is possible, for example, to start on page number 300.

R.4. COPY SINGLE PAGE IN THE SAME WORKSPACE

If you select a page in the **Workspace Explorer** and you right-click with the mouse, a pop-up menu appears, allowing you to copy the current page. You can later paste it in the active workspace. When the command is activated, the **Page information** dialogue appears. It contains all page information texts existing in the copied page. Assign the new page number, modify any page information and click **OK**.

Component names are either automatically adapted, or you are asked to confirm their names if the "Component Numbering" is set to "Free". References are automatically updated.

S EASY EDITING IN DATABASE LISTS

(standard)

In *SEE Electrical standard*, there are editors available in the *Database* lists. You can change data via these Editors. The changes are saved in the circuit diagrams.

Exercise 17-1: Make changes in the Product Editor.

1. Select the row of the record you want to change, for example the component Q1. The component information is displayed in the pane on the right.
2. In the right pane, select the row you want to change, for example Circuit breaker.

	Text Value
Product (-)	Q2
Description 00	Circuit breaker
Type	3VE1010 <input type="button" value="Db"/>
Connection	1
Connection	2
Connection	3
Connection	4
Connection	5
Connection	6

- 3.# Circuit breaker
Type in the desired text.
Click the next component you want to change.
Look at the page of the circuit diagram, where the changed components are located. The modifications are visible.
If you have changed the way of automatic numbering of the components or you want to rename all the components and the Automatic numbering has been selected (=> **Circuit diagram properties, General tab, Component numbering**), you can rename them by executing the **Renummer all components on all pages** command in the Product editor.

If you inserted or removed terminals, you can change the terminal number and sorting via the terminal editor executing the Renummer all shown terminals command found in the pop-up menu.

Through the pop-up menu, in the Contact list is available the **Renummer all contacts on all shown components** command, which allows you to renumber all the new contacts. The renumbering of the contacts can be made only if the contactors / relays / components with auxiliary contacts have a valid channel definition. This is necessary because the connection information is extracted from the channel definition even by renumbering. The renumbering starts on the first page, then continues page after page and on each page it begins from the upper left to the lower right corner.
If you use a filter for example for one page, the contacts of all contactors / relays / components with auxiliary contacts, located on the page, will be renumbered, i. e the master symbols are always decisive. If you filter on a single contactor / relay / component with auxiliary contacts, only the contacts on this will be renumbered.

T CONTACT MIRROR AND TYPE DATABASE

(Standard)

In *SEE Electrical standard*, you can choose to add cross references between a relay coil and its related contacts: using either a contact mirror or a contact cross.

Exercise 18-1: Check the circuit diagram properties for the project. A contact mirror instead of a contact cross must be used in the future. (You can save this setting in your workspace template.)

1. Click *Circuit diagrams* in the **Workspace Explorer**.
2. Right-click with the mouse
- 3.CO **Properties...**
The **Circuit Diagrams Properties** dialogue box appears.
- 4.T Coil
Select the **Coil** tab.
- 5.> Use contact mirror
Tick the option.
- 6.> **OK**

Exercise 18-2: Use contact mirror if the article information is not defined.

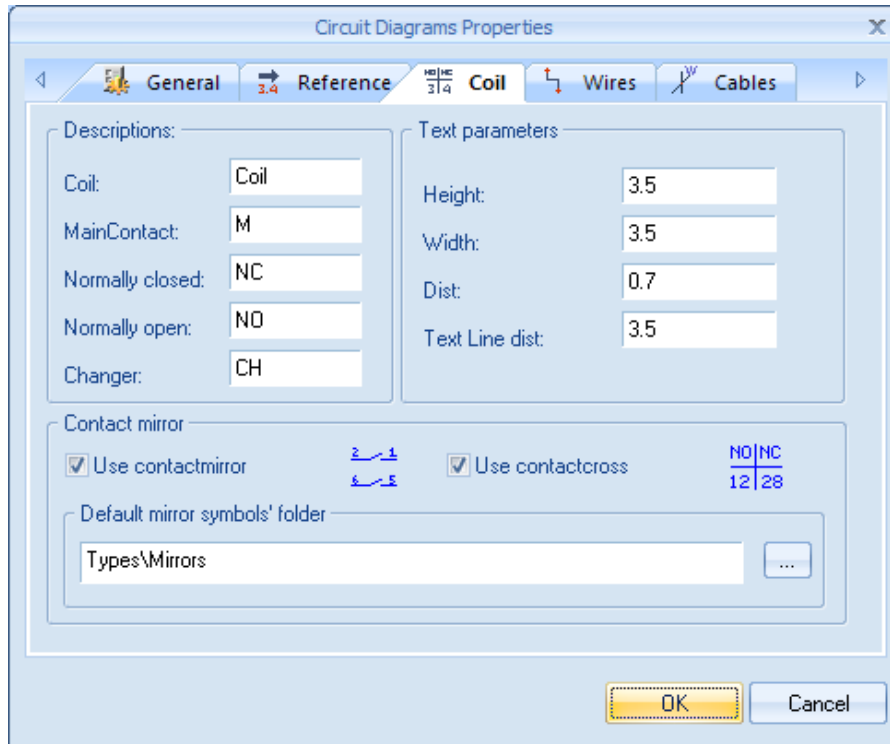
A symbol database with the symbol for the contact mirrors has to be defined in the **Coil** tab of the **Circuit Diagrams Properties** window. The first symbol in the symbols folder that has the same symbol ID as the already inserted contact is used for the building of the contact mirror. For example, if, in the symbols folder you have two symbols with ID for the main contact – one single pole and one three pole, the first one is used. Additionally you have to activate the "**Use contact mirror**" option.

1. Click *Circuit diagrams* in the **Workspace Explorer**.
2. Right-click
- 3.CO **Properties...**
The **Circuit Diagrams Properties** window appears.
- 4.T **Coil**
Select the **Coil** tab.
- 5.> Choose the *Default mirror* symbol folder to use.
- 6.> Choose the **Types** library and in there the **Mirrors** folder.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

6.> **OK**



The screenshot shows the 'Circuit Diagrams Properties' dialog box with the 'Coil' tab selected. The dialog is divided into several sections:

- Descriptions:**
 - Coil: Coil
 - MainContact: M
 - Normally closed: NC
 - Normally open: NO
 - Changer: CH
- Text parameters:**
 - Height: 3.5
 - Width: 3.5
 - Dist: 0.7
 - Text Line dist: 3.5
- Contact mirror:**
 - ☒ Use contactmirror (with a small diagram of a contact mirror)
 - ☒ Use contactcross (with a small diagram of a contact cross)
- Default mirror symbols' folder:**
 - Types\Mirrors

At the bottom of the dialog are 'OK' and 'Cancel' buttons.

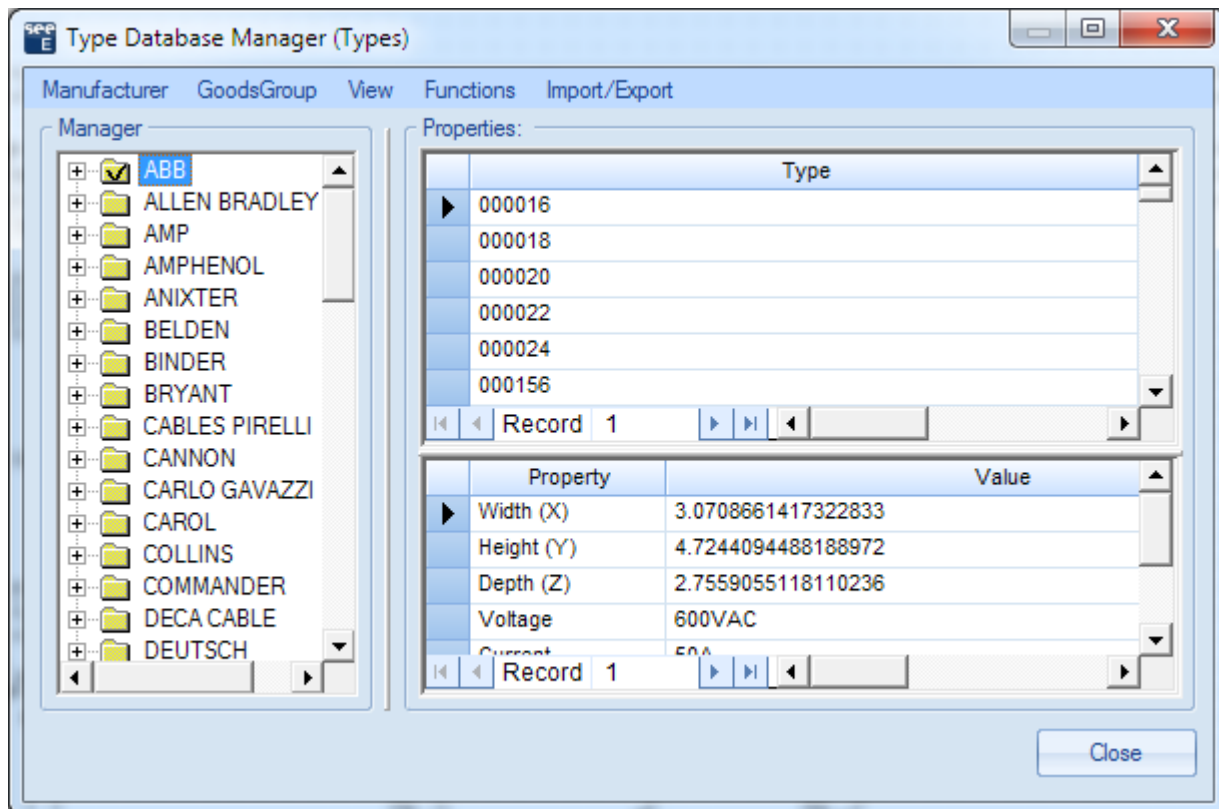
If you want to use article data with your coils, follow the rules defined in the next chapter.

T.1. MANIPULATING THE TYPE DATABASE

If you decide to use the article data on a coil, the contact mirror provides information about contacts available in a relay coil, and about contacts already used. The contact mirror can be created correctly only if it is known which contacts are available in the coil, i.e. what type the coil possesses. The way of constructing the contact mirror is defined in the so-called *Channel definition*. This can be performed in the Type database.

Exercise 18-3: Create a new Manufacturer.

- 1.CA **Functions**
- 2.CO **Database (Types panel)**



The **Type Database Manager** window appears.

This window is composed of three areas. You can move the borders of the areas and the borders of the columns in the right panes.

Adjust the **Type Database Manager** window as desired. Do the next steps as described.

Create the new manufacturer "Training".

- 3.M Select the **Manufacturer** menu.

- 4.M **Add**

- 5.# Training

Type in the name of the new manufacturer, i.e. **Training**.

- 6.> **OK**

The manufacturer has been created.

Select the newly created "Training" manufacturer in the left pane of the **Type Database Manager** window.


Exercise 18-4: Define a new type of a relay-coil.

Types must be unique, i.e. one type designation must appear only once.

1.M Click in the field under the **Type** column in the top right pane.


	Type	Description	Manufacturer	Goods Group
*			Training	

2.# Type 1
Enter the new type designation.

	Type
	Type 1
*	

3.# Relay coil AC
In the "**Description**" field, you can write the description of the new type, for example Relay coil AC.

4.> You can choose the desired manufacturer from the list by clicking within the *Manufacturer* column. The "*Training*" manufacturer is already available.

5.> Click within the *Goods Group* column. The  icon appears. If you click this icon, a list with the available goods groups opens. Select the desired goods group, in our case - "Auxiliary Contactors".
The goods groups are used for structuring the goods information. For example, in *SEE Electrical Standard* and *Advanced* levels, you can enter data about types for all kinds of components, i.e. switch gears, terminals, relay-coils etc. But, when you assign a type to a component, only the relevant type is to be shown. The goods groups enable this pre-selection.



6.> Expand the "*Training*" manufacturer.
The goods groups assigned to this manufacturer are shown, i.e. "Auxiliary Contactors".

7.> Expand the "Auxiliary Contactors" goods group.
The new type appears now in the Type Database Manager, too.

8.> Select the type you wish to assign information to, by clicking it in the Manager pane in the left part of the window.



In the bottom right pane of the **Type Database Manager** window, all the default properties of a type are displayed. (The default properties can be changed via the **Functions** ➤ **Settings** command, which opens the **Settings** window.)


Property	Value
Width (X)	
Height (Y)	
Description1	
Description2	
Price	
Order number	
Manufacturer	
Symbol name for	Db
Assign sub types	
Define Channels	
User setting 3	

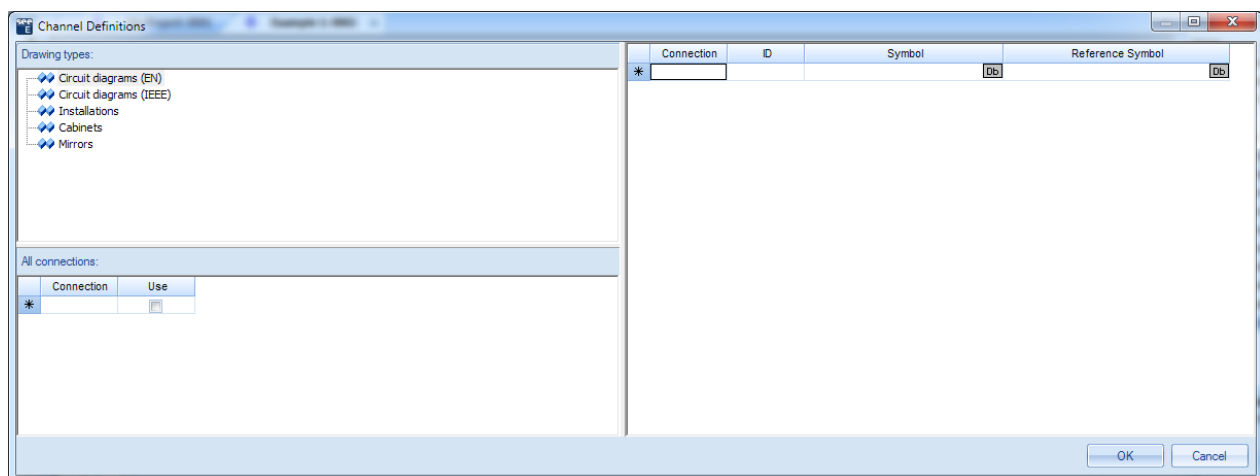
Record 1

- 9.> Click the *Value* column for the respective properties.
 Type in the following values:
Width: 100 (used in module Cabinets)
Height: 100 (used in module Cabinets)
Description 1: AC relay-coil
Order number: 123A

Exercise 18-5: Define channels for your relay coil type.

A relay-coil or a component with auxiliary contacts needs the information about the contact mirror that must be used. When a contact mirror must be shown under the relay-coil symbol, this property is essential. It is also necessary for a component with auxiliary contacts if an automatic contact numbering is to be performed.

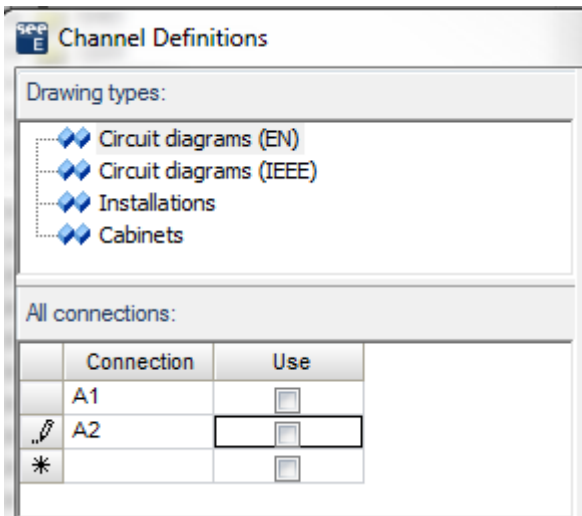
- 1.> Click the  icon in the **Value** column for the *Define channels* property.
 The **Channel Definitions** dialogue appears. Perform the channel definition.



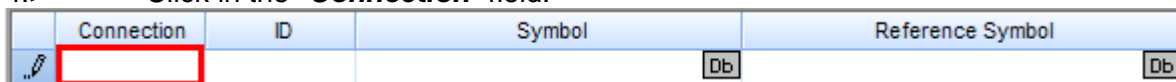
- 2.> Select the *Circuit Diagrams (EN)* drawign type.
 3.> In the **All connections** area, insert the connection texts A1 and A2

Training manual

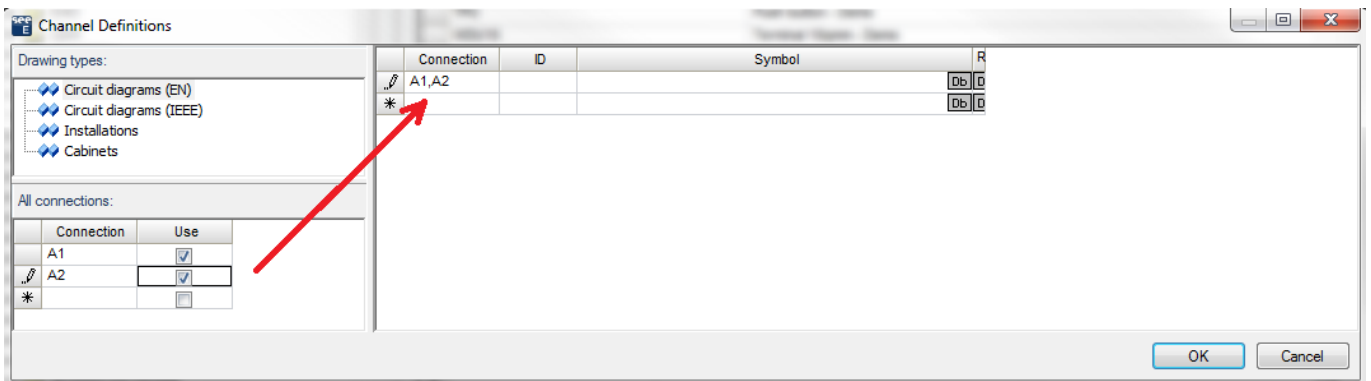
COPYRIGHT © 2013 IGE+XAO. All rights reserved



4.> Click in the "**Connection**" field:



5.> Check the "**Use**" check-boxes in the **All connections** area.




6.> ID

7.# Relay Coil

Select the *Relay Coil* ID from the pull-down list.

8.> Symbol

Click on the  button within this field. Select the desired symbol from the *Relay Coils* folder of the *EN61346-2UK* symbol library. The path to the selected symbol appears in the *Symbol* field.

9.> In the same way, in the **All connections** area, insert the connections: 1, 2, 3, 4, 5, and 6. Repeat steps 3 and 4.








10.> Click the "**Connection**" field of the second line.

11.> Check the "**Use**" check-boxes in the **All connections** area for the connection texts 1, 2, 3, 4, 5 and 6

12.> ID

13.# Relay-contacts, MAIN, NO

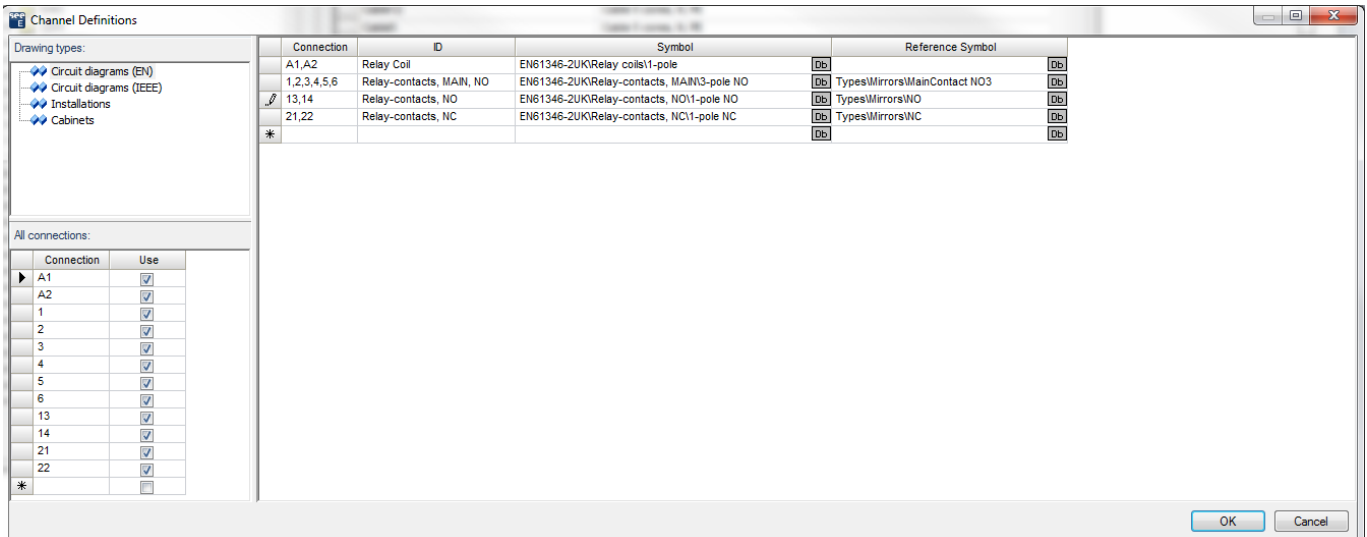
14.> Symbol

15. Click on the  button within this field. Select the **3-pole NO** symbol from the Relay-contacts folder, MAIN folder of the EN61346-2UK symbol library
- 16.> Reference Symbol
Click on the  button within this field.
- 17.> You can select the contact mirror symbol from the symbol databases again.
Open the *TYPES* database, and then open the *Mirrors* folder by double clicking on it and select the **Main Contact NO3** symbol.
18. Repeat steps 4 to 17 for the NO auxiliary contact.
- 19.# 13, 14
Type the contact numbers of the first NO contact.
- 20.# Relay-contacts, NO
Select the *Relay-contacts, NO* ID from the pull-down list.
- 21.> Symbol
Click on the  button within this field. Select the desired symbol from the Relay-contacts, NO folder of the EN61346-2UK symbol library.
Click on the  button within this field.
- 22.> Reference Symbol
Click on the  button within this field
You can select the contact mirror symbol from the symbol databases again.
Open the *TYPES* database, and then open the *Mirrors* folder by double clicking on it and select the *Contact NO* symbol.
- 23.> Connection
24. Repeat steps 4 to 17 for the NC auxiliary contact
- 14.# 21, 22
Type the contact numbers of the first NC contact.
- 15.# Relay-contacts, NC
Select the *Relay-contacts, NC* ID from the pull-down list.
- 16.> Symbol
Click on the  button within this field. Select the desired symbol from the Relay-contacts, NC folder of the EN61346-2UK symbol library.
- 17.> Reference Symbol
Click on the  button within this field once again.
- 18.> You can select the contact mirror symbol from the symbol databases again.
Open the *TYPES* database, and then open the *Mirrors* folder by double clicking on it and select the *Contact NC* symbol.
- 19.> **OK**

Training manual

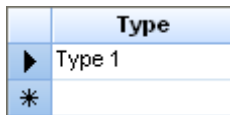
COPYRIGHT © 2013 IGE+XAO. All rights reserved

Finish the channel definition.



Exercise 18-6: Copy the type you have just created and copy a different channel information..

- 1.> Click the type in the type database
- 2.>



- 2.M **Functions**
- 3.M **Copy Selected Type**

A copy with the type name "*Copy of <type>*" appears, now it is "*Copy of Type 1*".

Now copy a new channel definition for this type.

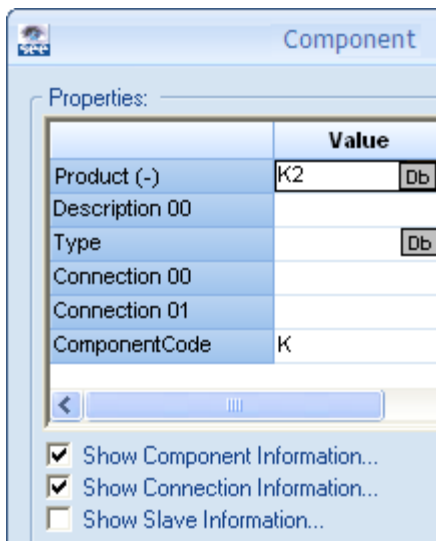
4. Make sure the type to add the channel definition to is selected.
 5. In the **Functions** menu of the **Type database manager** window choose the **Copy channel from other type** command.
 6. Select the type to copy the channel information from like you select a type normally. (Only one type can be selected here.)
 7. Terminate your selection. Doing this the channel information is copied .
- 4.> You can close the **Type Database Manager** window by clicking the **Close** button.

T.2. USING TYPES IN THE CIRCUIT DIAGRAMS


Use the newly defined type in the current circuit diagram.

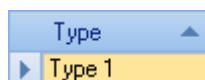
Exercise 18-7: Switch to page 2 of your training workspace, and define a type for the relay coil K2.


- 1.+ Double-click the relay-coil.



You can fill in the type in the Type field using the keyboard or you can select the type from the type database.


- 2.> Click .
- The **Type Database Browser** window opens.
- 3.> Open the Training manufacturer.
- 4.> Select the Type 1 by clicking the field on the left of it.



- 5.> Click the **Select** button.
The type is selected and appears in the right pane of the window.
- 6.> Close the **Type Database Browser** window by clicking on the .
- 7.> **OK**
Close the **Component Properties** dialogue box.
The contact mirror appears.

Exercise 18-8: Assign a type to coil K3.

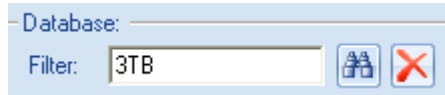
You can filter in the type database.

- 1.+ Double-click the K3 coil.
- 2.> Click  in the "**Type**" field.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

3.> Filter



4.# 3TB

Click the  icon to confirm.

All the types containing "3TB" in their name are displayed.

Please use types that provide a suitable number of contacts.

5.> Select a type by double-clicking in the field left to the Type, in this example 3TB46170BB4.

The type is selected.

6.> Close the **Type Database Browser** dialogue by clicking **OK**.

7.> **OK**

Close the **Component Properties** dialogue box.

Exercise 18-9: Filter the coil types in the Type Database according to the number of used contacts

1.+ Double-click the K4 coil.

2.> Click in the "Type" field

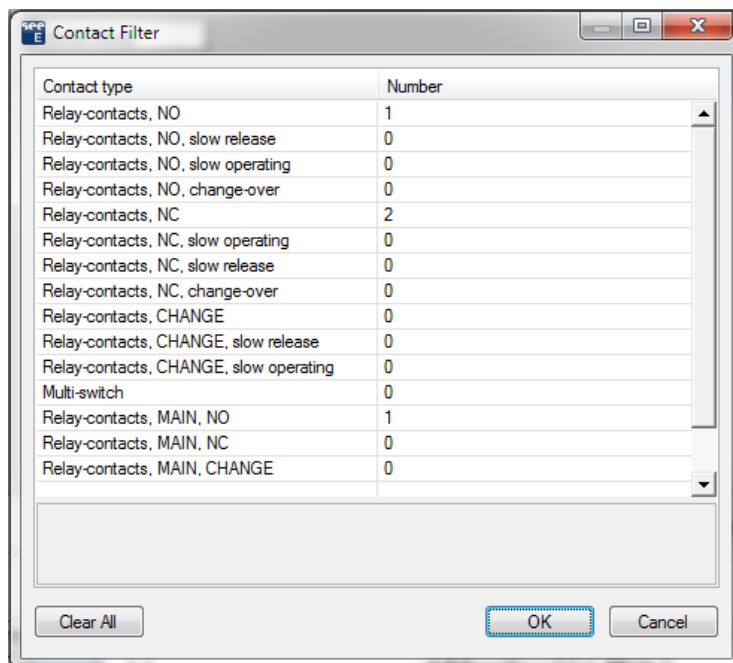
3. + Tick the "**Activate**" option in the **Contact Filter** area of the **Type Database Browser**.

After the filter is activated, the list will contain only the necessary number of contacts.




You can additionally add more contacts to the filter manually.

4.> Define the desired filter – the base of the filter is the number of contacts already present in the circuit diagram.

5.> To define more contacts, use the **Define** button..



Exercise 18-10: Assign types to the two motor protective switches on page 1.

- 1.+ Double click one of the motor circuit breakers.
The **Type Database Browser** window opens.
- 2.+ Click  in the **Type** field.
- 3.> Manufacturer..
- 4.> <Manufacturer> - Select as manufacturer **IGE+XAO**.
- 5.> Click in the field, where you want to group, e. g. **Goods group** and drag it with the mouse button pressed in the "**Drag a column header here to group by that column**" position. If necessary, drag another field in this position.
- 6.> Through the "**Collapse**" button you can close the folder of all grouping at once
Close the **Component Properties** dialogue box.
The contact mirror appears.
7. If you want to cancel a grouping, click on the appropriate field and drag it with the left mouse button pressed in the area of the column heading. Release the mouse button, when you can see red arrows.
8. <Type>
Select the **3VE1011** Type for the motor circuit breaker. You can find the type in the **Q** goods group.
Select the type through double click on the  area in front of the type.
9. Close the **Type Database** window with the  button.
10. OK.
Leave the **Component properties** window.

Exercise 18-11: If contactors or components with auxiliary contacts have a type with a channel definition, the contact numbering is performed automatically. If no free contact of the relevant type is available in the component, an error message appears.

- 1.+ Place a new contact, and assign it to an available contactor or a motor protective switch.

T.3. AUXILIARY CONTACTS INSIDE ADD-ONS FOR COILS AND SUBTYPES

Components do not always consist of only one element, they are often assembled from several elements or elements can be added to an article.

Fuses are typical examples of components assembled from several articles. They always consist of a socket, a fuse link, a screw cap and a fitting screw. In *SEE Electrical* these parts are called subtypes.

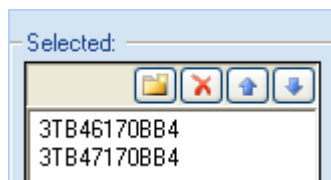
For a relay coil, an add-on is needed sometimes, but sometimes not.

Therefore, *SEE Electrical* offers two different approaches.

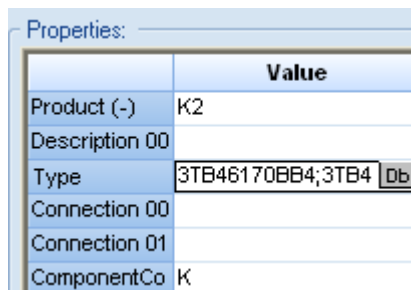
All of the articles must be stored in the type database.

Exercise 18-12: Assign an add-on for a relay coil.

- 1.+ Double-click the relay coil.
2. Go into the type database and select a second type.



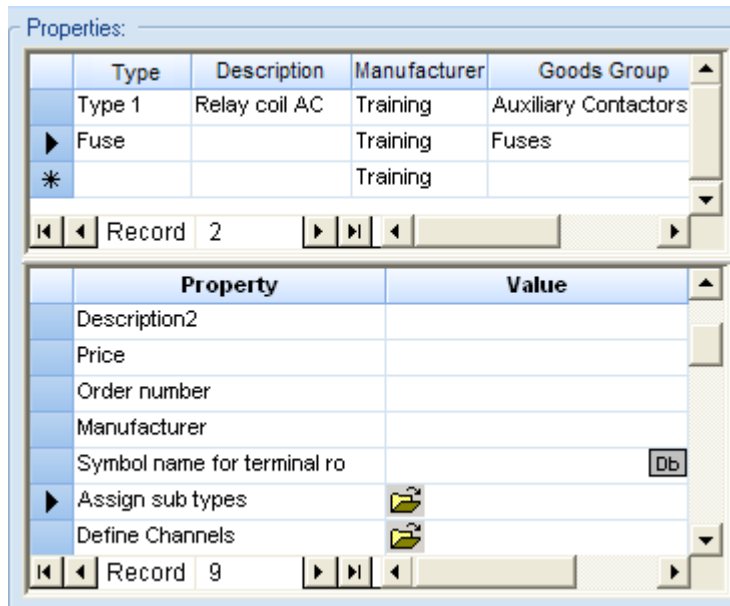
3. Close the **Type Database Browser** dialogue by clicking **OK**.



3. Both types are separated by a semicolon in the "Type" field of the **Component Properties** dialogue box.
 You can assign up to 10 types to a component.
 Look at the Products list. A column Type 02 with the second type is created. Look at the Parts list afterwards. The types remain on single lines there.
 Add-ons must be added to relay coils always using this approach.

Exercise 18-13: Define and use a type for a fuse.

- 1.+ Insert a fuse into your drawing. Double-click the fuse.
2. Go into the type database and create a type "Fuse".
The needed elements for the fuse are available. Otherwise, define them before you create the fuse type.
3. In the "**Property**" column, you can find the Assign sub types property.



4. Select Assign sub types.
5. You can choose all the types needed as elements for your fuse.
If several sockets or melt inserts are needed, choose the appropriate type several times.
6. Close the **Type Database Browser** dialogue by clicking **OK**.
Define then the next fuse type in the same way.
7. Assign the type to the fuse.
Look at the *Products* list. For the component "Fuse", only one type is recorded in the list of products, as well as in the Spare parts list.
In the Spareparts, simple list, each element of the fuse appears on a separate line.
Look in the *Parts* list. You only find the main type, too.
In the Parts, simple list the parts are on separate lines.

Once determined, the types for components with subtypes must not be changed again.

If a fuse sometimes has a splash guard and sometimes not, the type for the splash guard can be defined as a second type for the fuse.

If you need a 16A fuse now, but you need a 4A fuse later, define two types with different subtypes, etc.

U ADDITIONAL HINTS ABOUT COMPONENT MANAGEMENT IN THE TYPE DATABASE

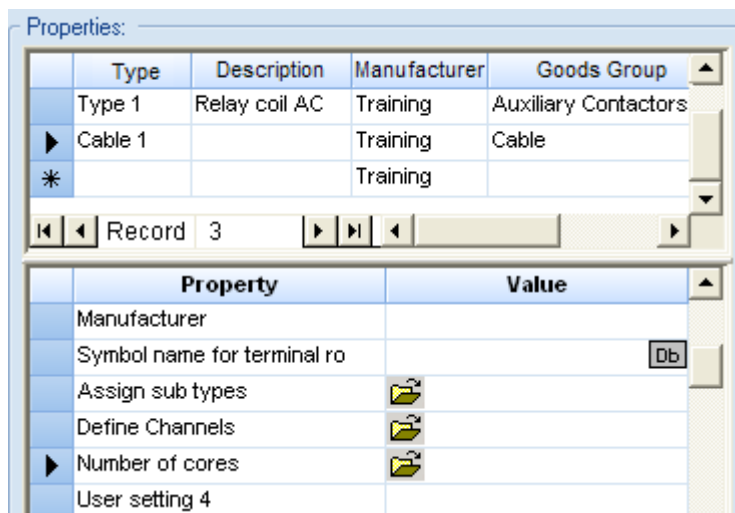
(Standard)

U.1. CABLE MANAGEMENT

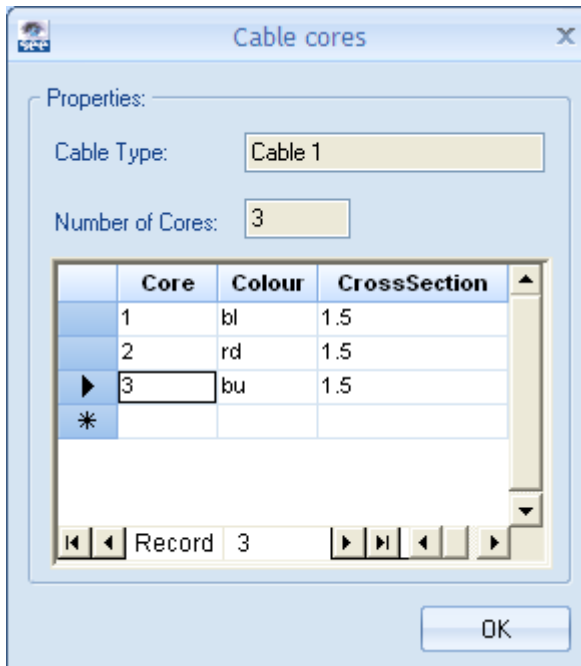
SEE Electrical can manage information about Cable-cores in the type database and Cable-cores used in a cable.

Exercise 19-1: Define a cable type.

1. Open the type database and create a cable type as usual.
2. In the **Property** pane, find the *Number of cores* attribute.



3. Click the  icon and define a colour and a cross-section for the cable-cores.



Properties:

Cable Type:

Number of Cores:

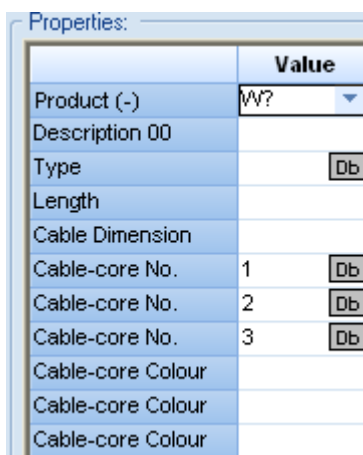
	Core	Colour	CrossSection
	1	bl	1.5
	2	rd	1.5
▶	3	bu	1.5
*			

Record 3

OK

4. Click **OK** to finish the definition.
5. Close the type database.


Exercise 19-2: Draw a new cable. Use the defined type.



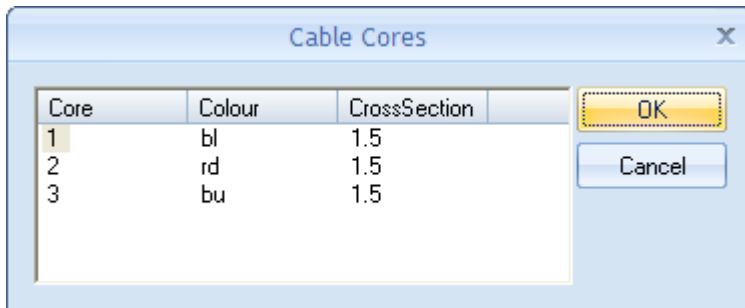
Properties:

	Value
Product (-)	WV?
Description 00	
Type	Db
Length	
Cable Dimension	
Cable-core No.	1 Db
Cable-core No.	2 Db
Cable-core No.	3 Db
Cable-core Colour	
Cable-core Colour	
Cable-core Colour	

Assign the newly defined type to the cable. The number and cross-section for the cable-cores are filled in automatically as defined in the type database.

If you wish to change the order of the cable-cores, delete all the core numbers, then click the  button in the first "Cable-core No." line.

A database opens, where all the free cores of the current cable are displayed. You can choose a core here.



Select the desired cable-core and click **OK**.

It appears in the first "Cable-core No." line. Proceed in the same way to assign the other cable-cores, too.

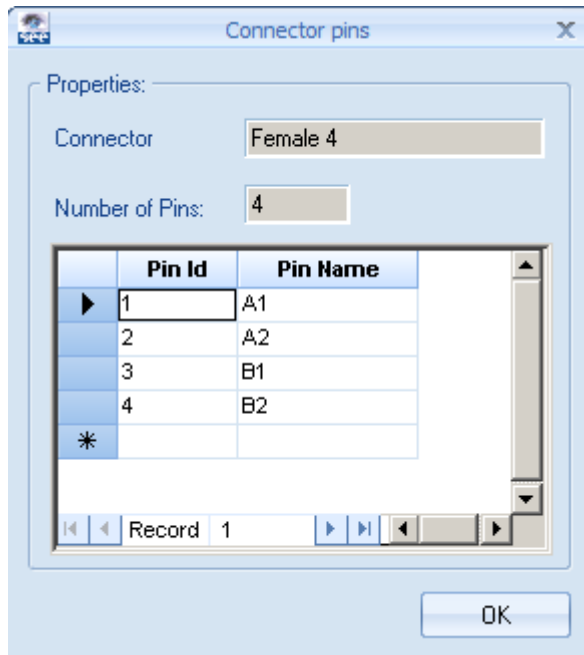
Exercise 19-3: Draw another cable with more cable-cores than available according to the assigned database type. You will receive an error message.

Look in the Cable editor to see which cable has too many cores.

1. Open the **Cable** editor.
2. Right click.
3. Select the **Check cables** pop-up command.

U.2. MANAGEMENT OF CONNECTORS

The "Pin Numbers" property allows you to define the pin numbers present in a connector.



If the pin numbers are defined correctly, *SEE Electrical* suggests automatically the pin name and controls the overbooking (if there is an overbooking, an error message appears).

The pins can get names such as 1, 2, 3... or 1A - 1Z, 1a - 1z, 2A – 2Z, etc.

The "Pin Id" has to be a numerical value. The sort order of the pins have to follow the sort order of the pins in the connector.

The pins have specific sort order in the connector determined by the manufacturer or by the way the pins have to be connected to the cable.

Start with the pin in the middle and go to the ones in the outer parts.


If the Pin Id has been determined in the correct order, the pin names are suggested in the right sort order at the insertion of the connector in the circuit diagram.

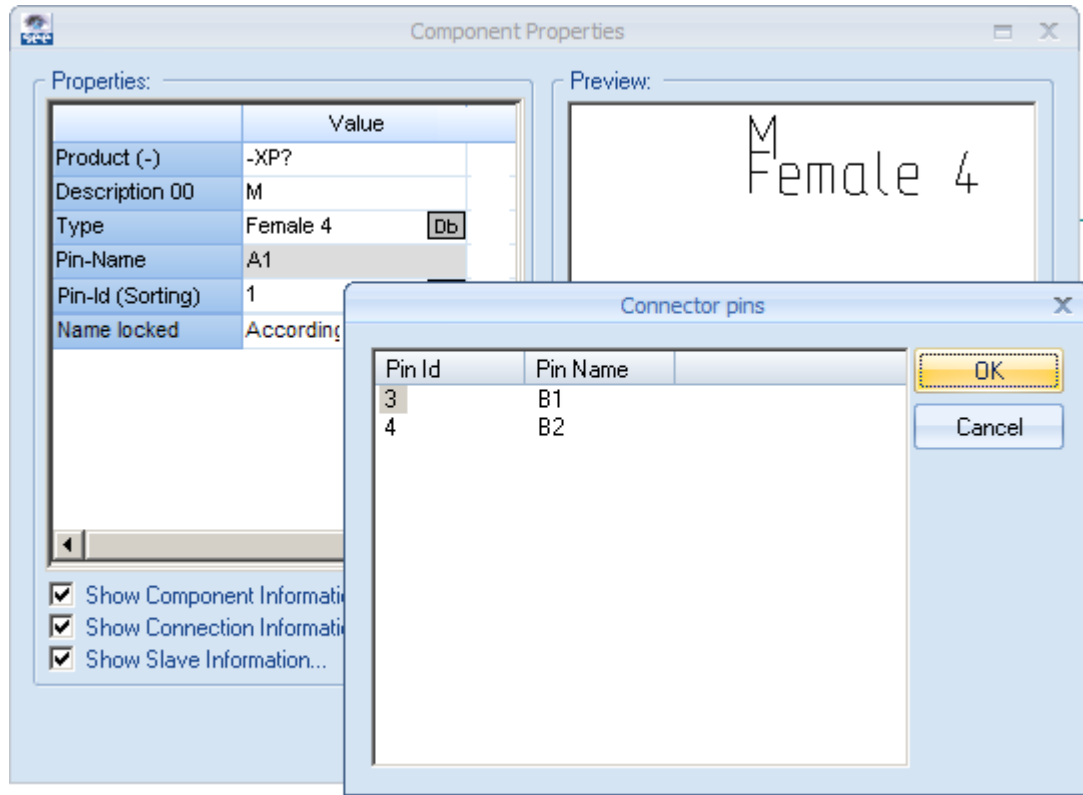
When the pin numbers are defined in the type, *SEE Electrical* controls that no pin is used twice and that the overbooking is impossible.

If the type and pin information has been assigned to a connector the pin can be changed.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

- Click the  button in the "**Pin – ID (Sorting)**" field.



The **Connector Pins** dialogue box displays all available pins.

U.3. CHANNEL DEFINITION FOR PLC I/Os

A PLC rack can be stored in the channel structure as follows:

You have a PLC rack with two external connections and four analogue I/O ports.

Port 1 has the connection points a, b, c;

Port 2 has the connection points d, e, f;

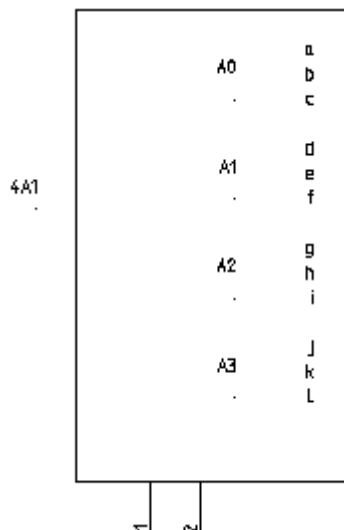
Port 3 has the connection points g, h, i;

Port 4 has the connection points j, k, l.

In this example you must add five channels to the channel definition.

Connection	ID
1,2	PLC, Main
a,b,c	PLC, Signal
d,e,f	PLC, Signal
g,h,i	PLC, Signal
j,k,l	PLC, Signal
*	

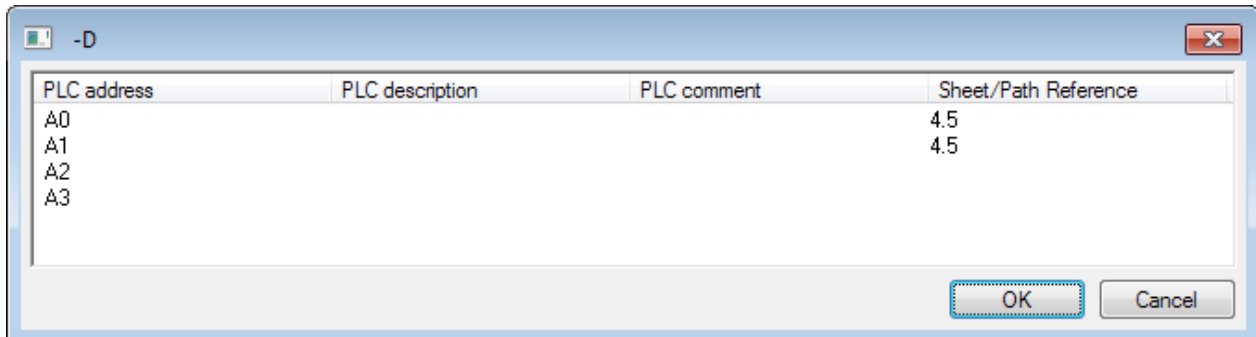
If you assign this type to a PLC Rack symbol, the connection texts are taken from the channel definition.



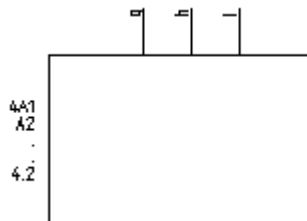
Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

Then you can insert the addresses.



The connection texts are displayed.



U.4. BLACK BOXES

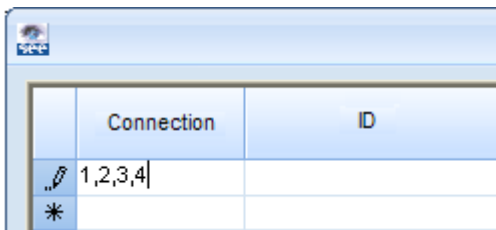
U.4.1. ASSIGNING OF TYPES TO INTELLIGENT BLACK BOX SYMBOLS

If a type is assigned to an intelligent black box, the connection texts are filled from the channel definition as usual. In this case you must draw the wires in the sort order the connection texts are defined in the channel definition to make sure the correct connection is attached to a wire. Please choose "intelligent black box" when defining the channel information in type database.

U.4.2. CREATING A BLACK BOX FROM CHANNEL INFORMATION

Symbols for Black boxes can be generated automatically if you define the necessary values in their channel definition.

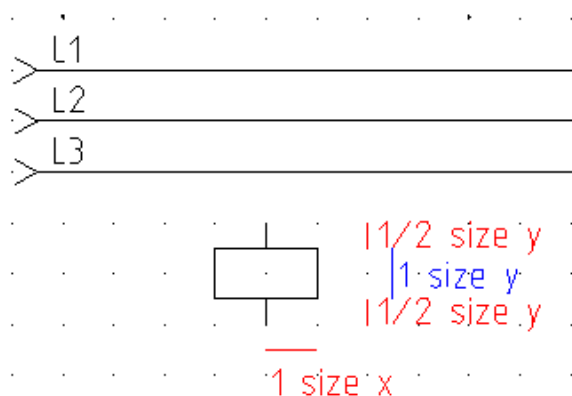
If you use this way of generating black boxes, a symbol of "component" type is generated and NOT an "intelligent black box".



Fill in the appropriate number of connection texts in the "**Connection**" field, separated by comma.

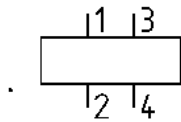
In the "**ID Circuit diagrams (EN)**" field, select the desired symbol type, for example "Component" or "Coil".

The size of the automatically created symbol depends on the default distance used for drawing potentials.



Position the newly created type via one of the commands **Functions** ➤ **Component** ➤ **Add**, **Functions** ➤ **Other** ➤ **Pick list** or by using the **Component Explorer**. A black box is automatically created.

The example above generates the following result:



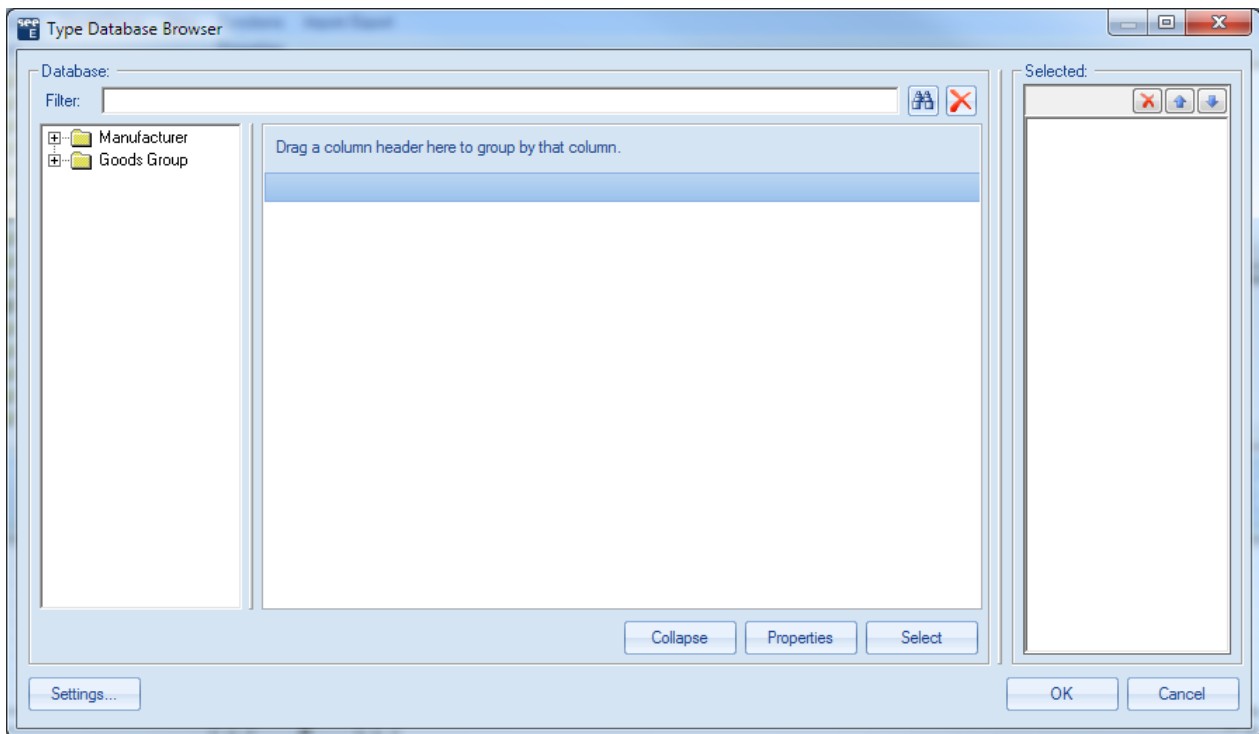
V ADDITIONAL HINTS ABOUT THE TYPE DATABASE

(Standard)

V.1. SEARCHING AND COPYING IN THE TYPE DATABASE

✓ Searching


In the **Type Database Manager** window, the **Functions** ➤ **Find Type** command is available. It opens the familiar **Type Database Browser** dialogue, allowing you to search for a type.



✓ Copying

In the **Type Database Manager** window, the **Functions** ➤ **Copy Selected Type** command is available. By generating a copy of the selected type, this command allows you to create new types quickly by modifying existing types.

V.2. COMPLETING COMPONENTS

If a component consists of several parts, you can use the **Functions** ➤ **Component** ➤ **Complete** command or activate the  icon to insert the parts that are not inserted yet.

Typical components consisting of several parts are:

Contactors: They consist of a coil and several contacts.

Some motor protective switches: They consist of the component itself and of contacts not located at the same place as the component.

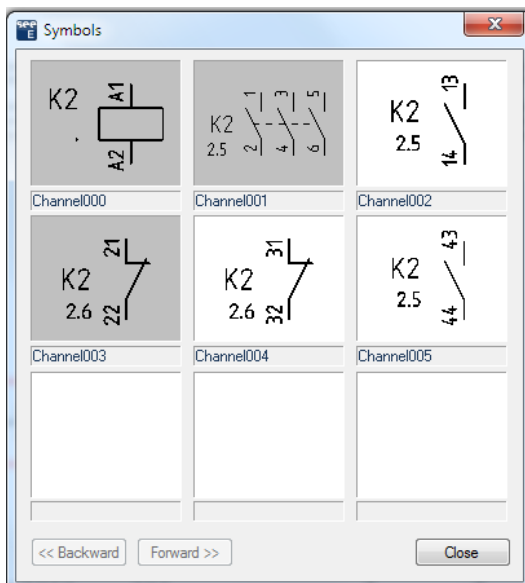
PLC-Racks: The Rack itself and several signals may be used.

Requirements:

In order to use this function, it is necessary to define in the Channel definition of the component and all its parts which symbol must be used in the *Circuit diagram* module.

This applies to all components in the workspace.

At least one part of the component must already be inserted in the drawing.



Channel definition:

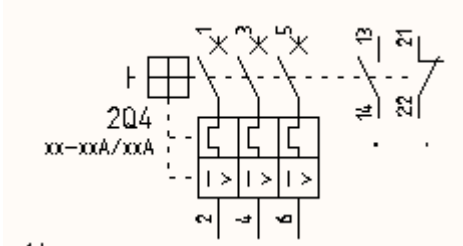
In the Type Database, the Circuit diagram symbol must be specified in the Channel definition.

Connection	ID	Symbol	Reference Symbol
A1,A2	Relay Coil	EN61346-2UK\Relay coils\1-pole 	
13,14	Relay-contacts, NO	EN61346-2UK\Relay-contacts, NO\1-pole NO 	Types\Mirrors\NO 
21,22	Relay-contacts, NC	EN61346-2UK\Relay-contacts, NC\1-pole NC 	Types\Mirrors\nc 
33,34	Relay-contacts, NO	EN61346-2UK\Relay-contacts, NO\1-pole NO 	Types\Mirrors\NO 
43,44	Relay-contacts, NO	EN61346-2UK\Relay-contacts, NO\1-pole NO 	Types\Mirrors\NO 

Components with auxiliary contacts/contactors/relays:

A component with auxiliary contacts (contactors/relay) doesn't consist only of one symbol in the Circuit diagram but dependent contacts (Slaves) are placed at other places different from the place where the main component (the Master) is located.

The motor protective switch below has 6 connections (1, ..., 6) directly connected at the component and two dependent contacts (NO and NC) located and connected at another place in the Circuit diagram. There is only a cross-reference to the Slaves at the Master component and vice versa.

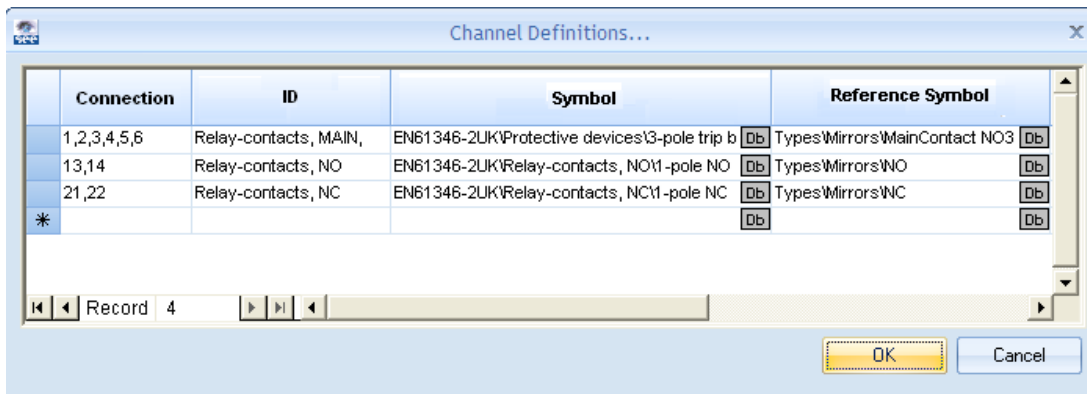


Therefore, you will find 3 lines in the channel definition:

A line for the main component (with connections 1,2,3,4,5,6)

A line for the NO contact (with connections 13,14)

A line for the NC contact (with connections 21,22).

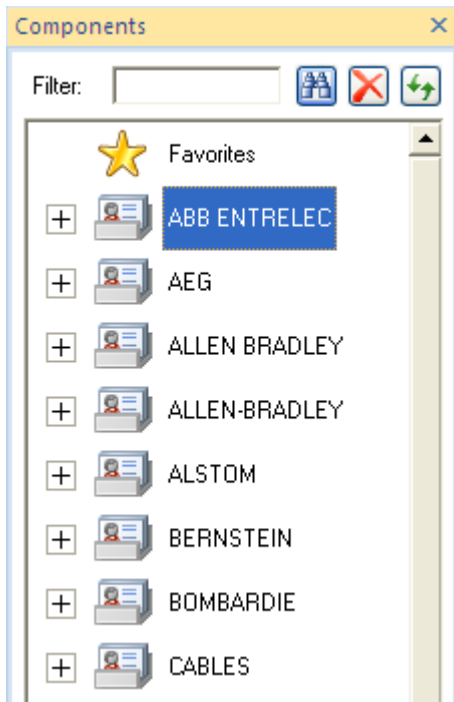


The channel definition for a contactor/relay is formed in a similar way.

V.3. USING THE COMPONENT EXPLORER

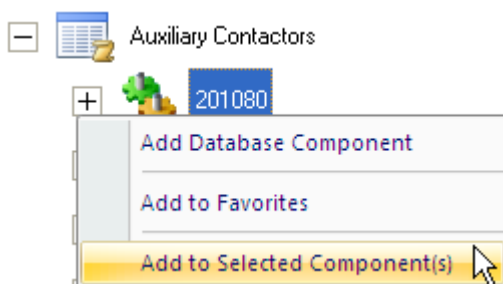
The **Component Explorer** allows you to handle components with type information.

To display the **Component Explorer**, select the **Home > View > Components** command.



By using this explorer, you can:

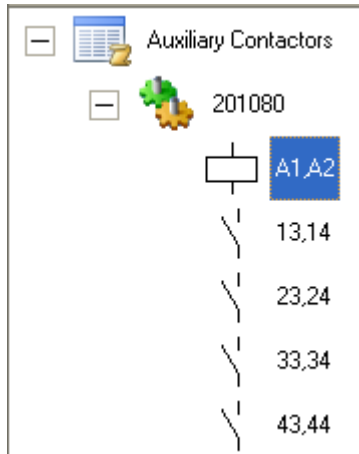
- ✓ Easily attach types to your components, as well as to multiple components in one step.
To do this, add one or several components, find the desired in the **Component Explorer** and right-click to activate the **Add to Selected Component** pop-up command.




- ✓ Use the Favorites folder for types often used
The handling of this Favorites folder is the same as the one in the Symbols Explorer.
- ✓ Use the **Add Database Component** pop-up command
If the channel information is properly defined, the **Add Database Component** pop-up command allows you to add the selected type into the drawing, as already known via the **Functions > Component > Add** command.

✓ Insert symbols directly from the **Component Explorer**

If the channel definition is correct, the symbols to be used in the Circuit diagram module are visible in the Component Explorer and can be "dragged" into the drawing.



Note

If you add a new type in the Type Database, you need to refresh the Component Explorer view to be able to access the new type in the Component Explorer. To refresh the view click the  icon.

V.4. IMPORTING AND EXPORTING ARTICLE MASTER DATA

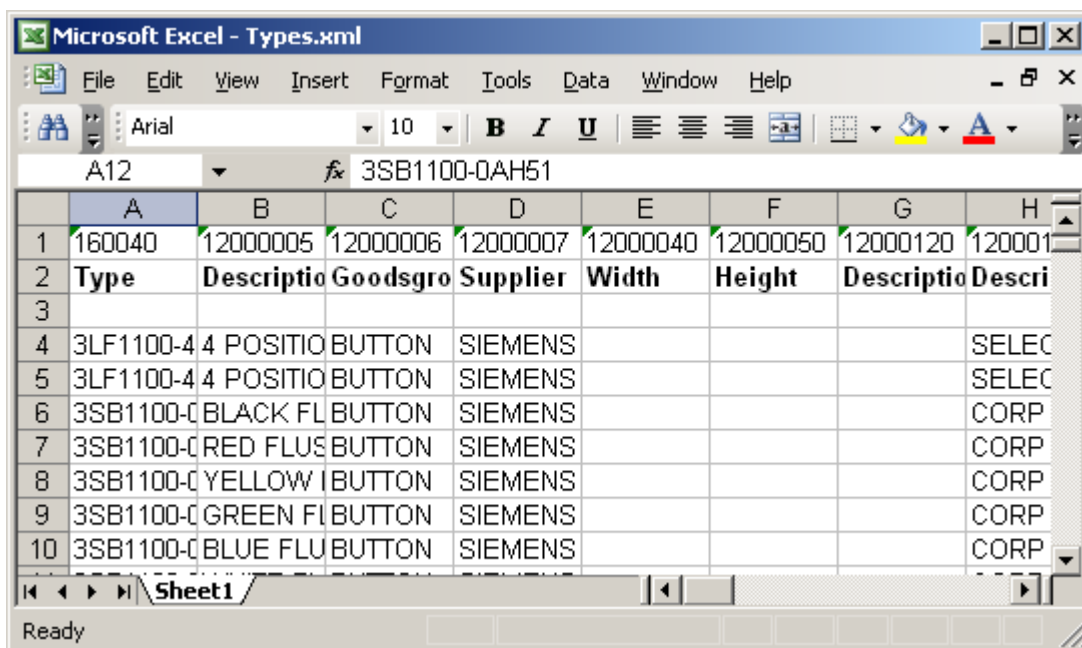
You can receive article master data from the component manufacturer or from another source and use it. *SEE Electrical* offers the possibility to import data with *ECAD* or *Excel* format in the type database.

You can find both functions in the **Type Database Manager** window, in the **Import/Export** menu.

Here, you are also given the opportunity to import types from a *XML* spreadsheet file, for example created with a *Microsoft Excel* application. However, a *XML* file may be generated from many other external programs. As well, exporting types to a *XML* spreadsheet is possible.

The *XML* spreadsheet represents an *ASCII* formatted text file presenting the data structured in rows and columns in a standard file structure, predefined with tags.

It is possible to open/edit/save *XML* Spreadsheets using a *Microsoft Excel* application. The .xml file looks very much like a standard *Excel* table, but it contains hidden control elements. That is why you can change only what is described below.



	A	B	C	D	E	F	G	H
1	160040	12000005	12000006	12000007	12000040	12000050	12000120	12000120
2	Type	Description	Goodsgro	Supplier	Width	Height	Description	Description
3								
4	3LF1100-4	4 POSITIO	BUTTON	SIEMENS				SELEC
5	3LF1100-4	4 POSITIO	BUTTON	SIEMENS				SELEC
6	3SB1100-C	BLACK FL	BUTTON	SIEMENS				CORP
7	3SB1100-C	RED FLU	BUTTON	SIEMENS				CORP
8	3SB1100-C	YELLOW I	BUTTON	SIEMENS				CORP
9	3SB1100-C	GREEN F	BUTTON	SIEMENS				CORP
10	3SB1100-C	BLUE FLU	BUTTON	SIEMENS				CORP

- ✓ The first line contains the ID from *SEE Electrical* for each column. You are not allowed to change this line.
- ✓ The second line contains the titles of the columns. You are not allowed to change this line.
- ✓ The following columns contain the information about the article in *SEE Electrical*.

Save the *Excel* file. Rename it, if desired, and keep in mind that you have to select the file type.

The contents and the structure of the *Excel* file can change, depending on the information contained in the exported *SEE Electrical* type database.

V.5. MODIFYING EXISTING TYPES IN THE PROJECT

It is possible that you may need to make changes to a type that you have already assigned to a symbol in your project.

These changes will not affect directly the current project, because the types used in the project are stored and cannot change automatically when the type database is changed.

For example, if you negotiated prices for your project a year ago, the information cannot be updated automatically.

To update the current type information, execute the **Functions > Types > Update** command.

V.6. DISPLAYING INFORMATION ON THE EQUIPMENT FROM THE TYPE DATABASE

You can display information from the type database on a piece of equipment inserted in the circuit diagram. In order to do this, you must use texts with the respective attributes during the symbol generation.

The necessary text attributes can be found in the *Type Properties* node in the **Text** dialogue.

V.7. DEFINE WHICH TYPE DATABASE TO USE WITH A PROJECT

(Standard and Advanced)

It is possible to use different type databases in *SEE Electrical*.

Because of this each company can name its type database according to their needs - *<MyCompanyTypes.SES>*.

If the type database carries the companies name, it might be much more convenient to include this file into the process of securing data.

Customers serving different clients of course will also benefit from this as they now can assign the clients type database to the client workspace(s).


If a type database different from the "TYPES.SES" is used, this is defined via a command. The **Set type database** command allows you to select the new database to use with the current workspace.


The definition is saved in the workspace (or workspace template).

Assign type database to multiple workspaces

The **Set MultiType DB** command allows to assign a type database to multiple workspaces or to all in one folder. It can only be executed, if no workspace is open.

First select the type database to use via the "**Select type database**" field.

The icon  allows you to select several workspaces.

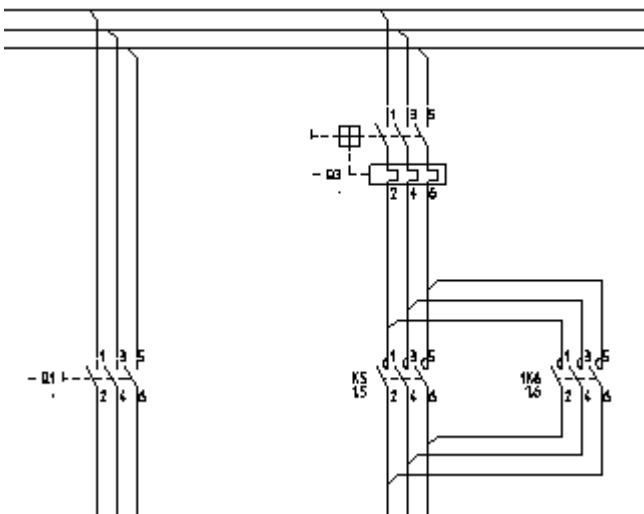
The icon  allows you to select folders, where all workspaces will be treated.
"Set type database" applies the type data base selected to all workspaces defined.

W WIRE PROPERTIES

(Standard)

W.1. WIRE DIRECTION

SEE Electrical *standard* offers you the possibility to use wire direction instead of the node.

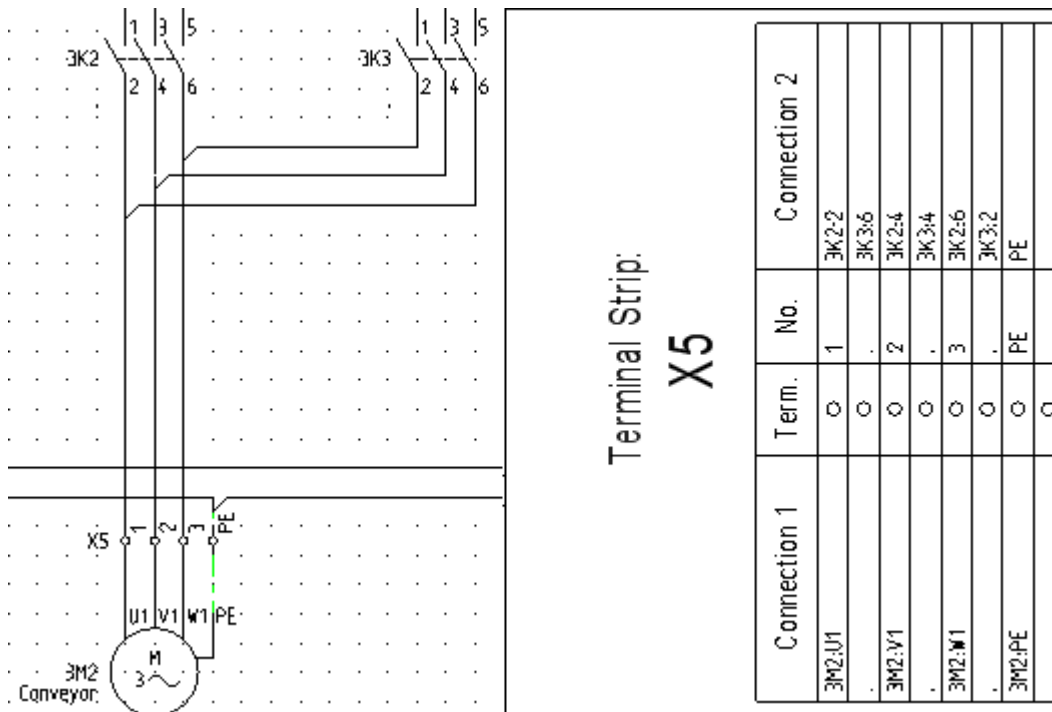


You can change the wire direction via the **Electrical** ➤ **Wires** ➤ **Direction** command.

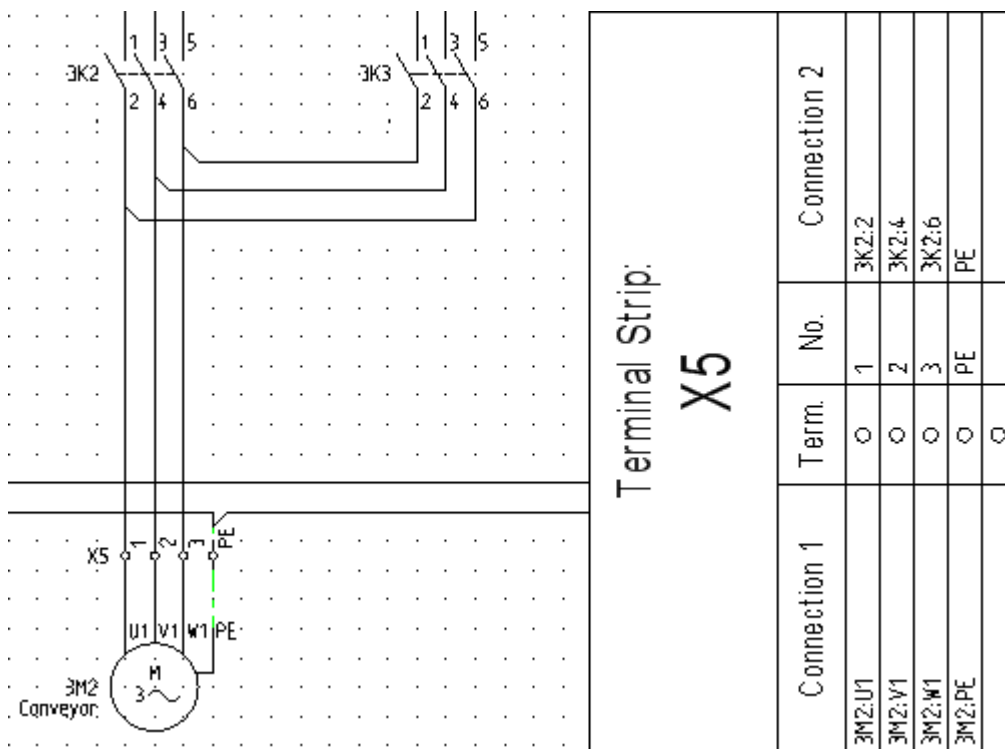
- Click on the node, the wire direction will be changed.
- Right-click *Circuit Diagrams* within the **Workspace Explorer**.
- Execute the **Properties** pop-up command and click the **Wires** tab.
- Tick the "**Show Wire directions**" option to display wire directions permanently.

By using wire directions, you can change the appearance of the terminal matrix and in the *advanced* level, the terminal plan as well.

Example 1:



Example 2:



Note :

*You can change the default wire directions via the following registry key:
[HKEY_CURRENT_USER\Software\CAE Development\SEE Electrical\Version
V7R2\1000\WireDirections] or [HKEY_CURRENT_USER\Software\CAE Development\SEE
Electrical\Version V7R2\1001\WireDirections] for circuit diagrams IEEE.*

The possible values are:

WireDirectionLeftRightDown: 15025, 15027, 15035 and 15037 (default EN)

WireDirectionLeftRightUp: 15026, 15028, 15036 and 15038 ((default EN))

WireDirectionLeftUpDown: 15022 ((default EN)), 15024, 15032 and 15034

WireDirectionRightUpDown: 15021 ((default EN)), 15023, 15031 and 15033

You have to be an administrator on the computer to be able to perform this operation.

W.2. CHANGING TARGETS

In the terminal matrix, internal and external targets must appear on one defined side.

Terminal Strip:

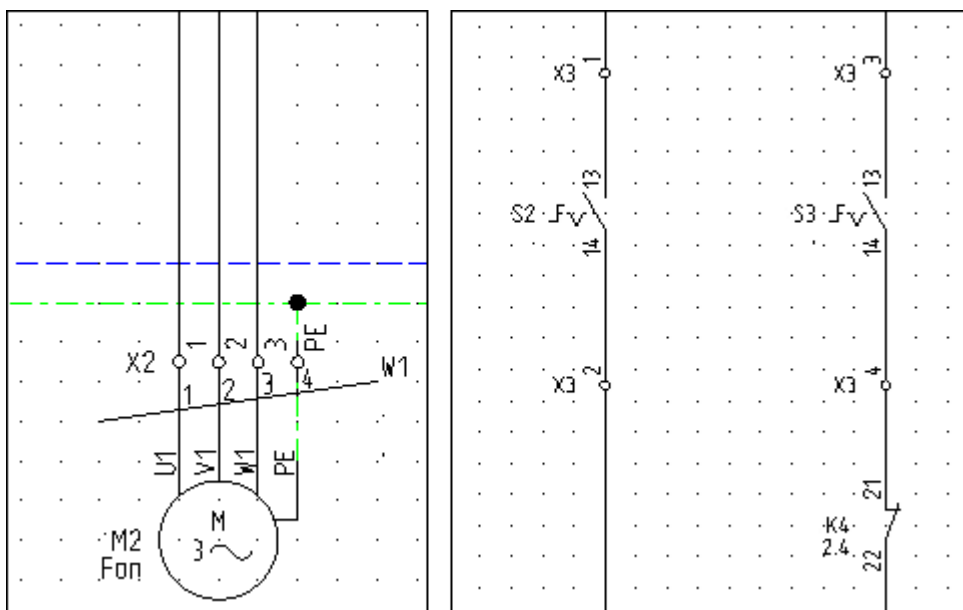
X3

Connection 1	Term.	No.	Connection 2
2S2:13	○	1	1Q2:14
2K2	○	2	2S2:14
2S3:13	⊗	3	1Q4:14
2K4:21	○	4	2S3:14
2S4:13	⊗	5	-
2K3:21	○	6	2S4:14
2P6	○	7	2K2:14
N	⊗	8	2P6
2P7	○	9	2K3:14
.	○	-	2K4:14
N	⊗	10	2P7
	○		

In the example above, internal and external targets are shown in colour either on the right or on the left side.

This comes from the circuit diagram as terminals find usually the internal targets at their upper connection, and the external targets at their bottom connection.

This can be useful in some cases but in others - not:



Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

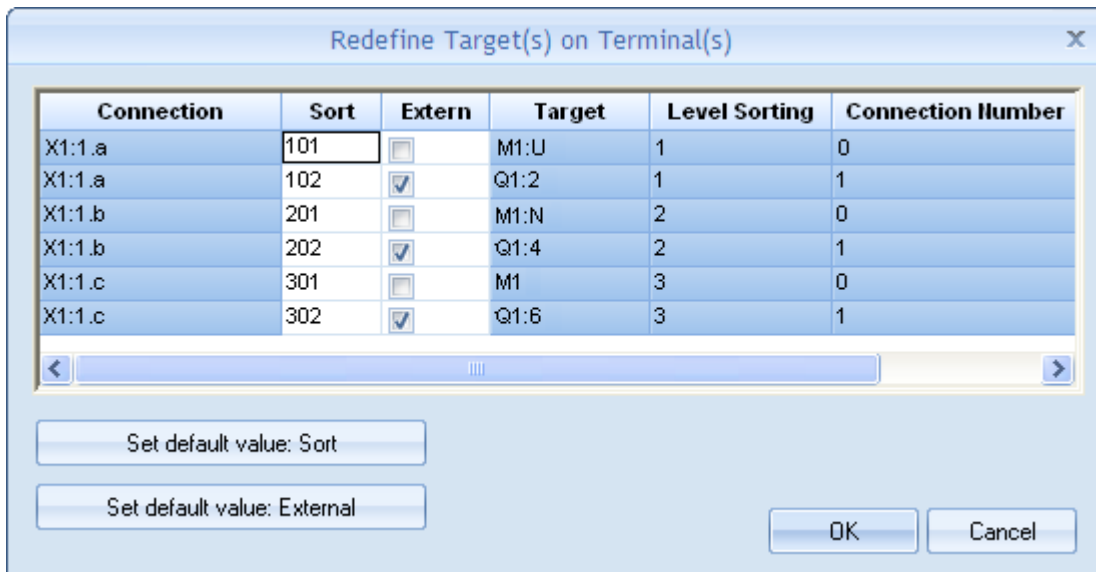
Exercise 21-1: Swap the internal and external target for both bottom terminals in the example displayed above on the right.

- 1..CA **Electrical**
- 2..CO **Connection (View panel)**
Each first connection within a component is marked in yellow.
The first connection of a terminal manages the internal target.
3. Select the terminals whose connections you wish to swap.
- 4..CA **Electrical**
- 5..CO **Swap (Connections panel)**
The connections of the highlighted terminals have been swapped.

Exercise 21-2: For terminals with more than 2 connections or Multi-Layer terminals, you can define more external/internal targets.

1. Select the terminals whose targets you wish to redefine.
- 2.CA **Electrical**
- 3.CO **Redefine target(s) on terminal(s) (Connections panel)**

The following dialogue appears:



Connection	Sort	Extern	Target	Level Sorting	Connection Number
X1:1.a	101	<input type="checkbox"/>	M1:U	1	0
X1:1.a	102	<input checked="" type="checkbox"/>	Q1:2	1	1
X1:1.b	201	<input type="checkbox"/>	M1:N	2	0
X1:1.b	202	<input checked="" type="checkbox"/>	Q1:4	2	1
X1:1.c	301	<input type="checkbox"/>	M1	3	0
X1:1.c	302	<input checked="" type="checkbox"/>	Q1:6	3	1

Set default value: Sort

Set default value: External

OK Cancel

- 4.> Use the check box within the "**Extern**" column to set the connections as external or not.
- 5.> Click **OK** to validate.
The connections of the selected terminals are redefined.

W.3. DEFINING A LINK

Links are indicated automatically from the *SEE Electrical standard* level. Two types of Links are created: Links identified via potentials and links identified via wires.

You can assign another type to the links, as well as the standard type. In this way you can differentiate between inlaying links, wire links etc.

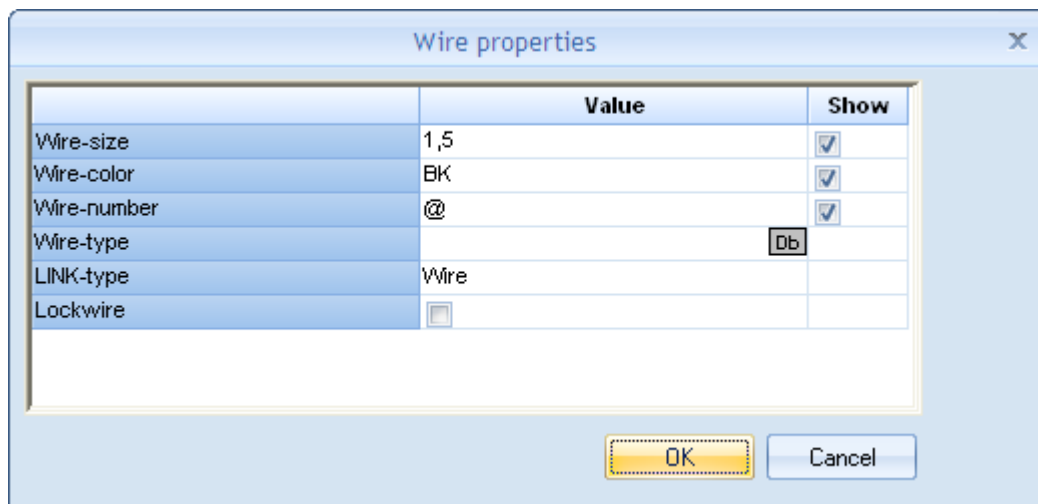
In the terminal matrix, the different link types can be displayed on various locations.

The following definitions are used:

- P Links identified via potentials
- 0 Links identified automatically via wires (main link type)
- 1 Link type 1, you can assign it instead of the Wire type, for example inlaying link
- 2 Link type 2
- etc. up to
- 10 Link type 10

Exercise 21-3: Switch to page 2 of your training workspace. Place a new terminal X2:11 beneath the terminal X2:1. Define the link type between the terminals X2:1 and X2:11.

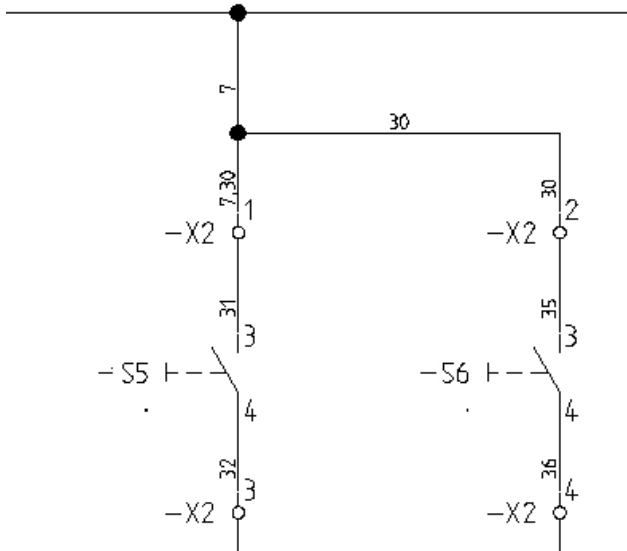
- 1..CA **Electrical**
- 2..CO **Properties (Wires panel)**
- 3.+ Identify the wire whose LINK-type you want to change.



- 4.> **LINK-Type**
The value "Wire" in the **LINK-type** line means that a link type is not defined. In the **training**, the LINK type "Wire" must represent inlaying links. Bridge type 1 would represent wire links.
- 5.> Bridge type 01
Select another link type.
- 6.> **OK**
The link type has been changed and you can identify the next wire or close the function.

W.4. GRAPHICAL WIRE NUMBERING

The graphical wire numbering is frequently requested. *SEE Electrical standard* provides this feature.



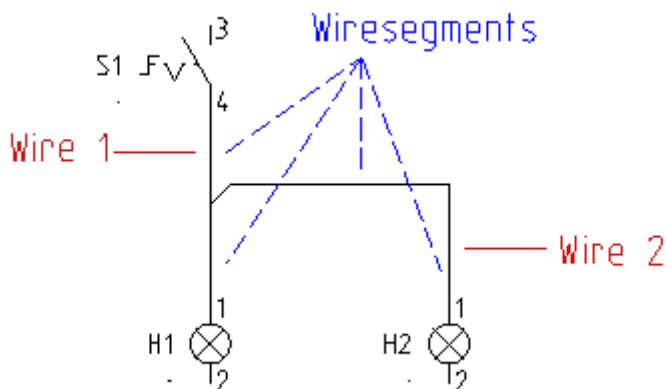
The functions for automatic numbering and removing wire numbers are available in the **Electrical** ➤ **Wires** ➤ **Numbers** subpanel.

It is important that the terms "net", "wire", "wire segment" are clearly defined in case signal types of wires are not used. In case you use signal types of wires you have to understand well the terms "signal type", "net", "wire", and "wire segment".

Signal type

Signal types can be: Power, Control, N, PE, Data bus, etc. For each signal type you can define attributes (colour, size, line width and visibility for wire texts).

You will find detailed information in the next chapter.



Net

In the example shown above is present one net that links the switch with the two lamps. A net can connect different connection points from different components. A net consists of different wires. If you use the signal type, different nets can have the same signal type.

Wire

The wire is the physical object that always links exactly two components. The net shown in the example above consists of two physical wires.

- One wire is connecting the S1 switch with the H1 lamp
- One wire is connecting the H1 lamp with the H2 lamp.

The wire directions in the net are very important.

Wire segment

Wire 1 contains two segments and Wire 2 has three segments. The segment between lamp H1 and the direction node is common for both wires.

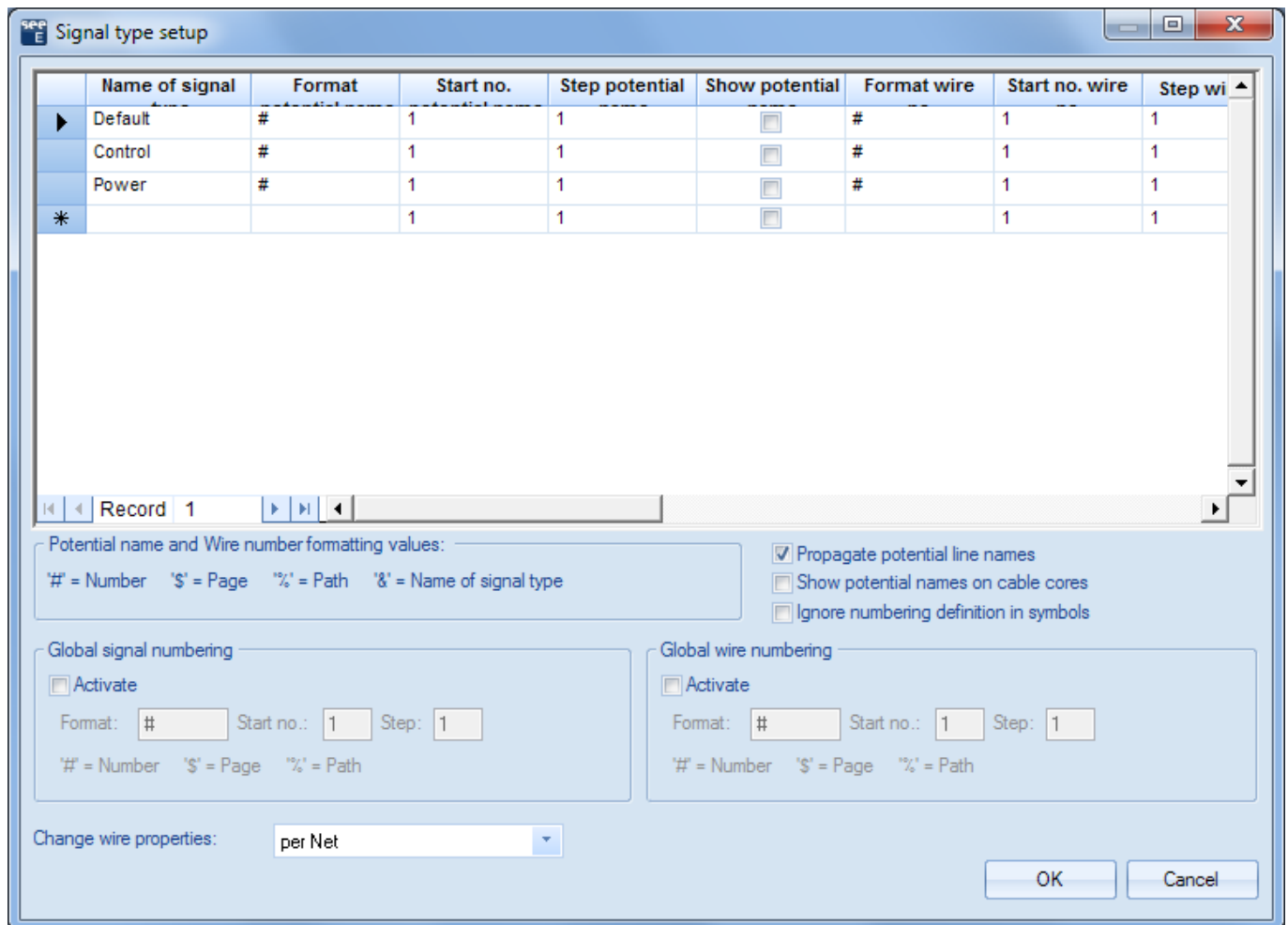
W.5. SIGNAL PROPERTIES

Signal properties

It is possible to assign *Signal* properties when drawing wires - for example, for control circuit or for power circuit. For each Signal type you can define different attributes such as wire colour or size.

First, it is necessary to activate the "**Signal types for wires**" option in the **Wires** tab of the **Circuit Diagram Properties** window, available after executing the **Properties** pop-up command for Circuit diagrams in the **Workspace Explorer**.

Afterwards, click the **Signal type setup** button to define the desired signal properties. For the *standard* level, only 4 Signal properties are possible.



Name of signal	Format	Start no.	Step potential	Show potential	Format wire	Start no. wire	Step wi
▶ Default	#	1	1	<input type="checkbox"/>	#	1	1
Control	#	1	1	<input type="checkbox"/>	#	1	1
Power	#	1	1	<input type="checkbox"/>	#	1	1
*		1	1	<input type="checkbox"/>		1	1

Record 1

Potential name and Wire number formatting values:
 '#' = Number '\$' = Page '%' = Path '&' = Name of signal type

☒ Propagate potential line names
☐ Show potential names on cable cores
☐ Ignore numbering definition in symbols

Global signal numbering
☐ Activate
 Format: # Start no.: 1 Step: 1
 '#' = Number '\$' = Page '%' = Path

Global wire numbering
☐ Activate
 Format: # Start no.: 1 Step: 1
 '#' = Number '\$' = Page '%' = Path

Change wire properties: per Net

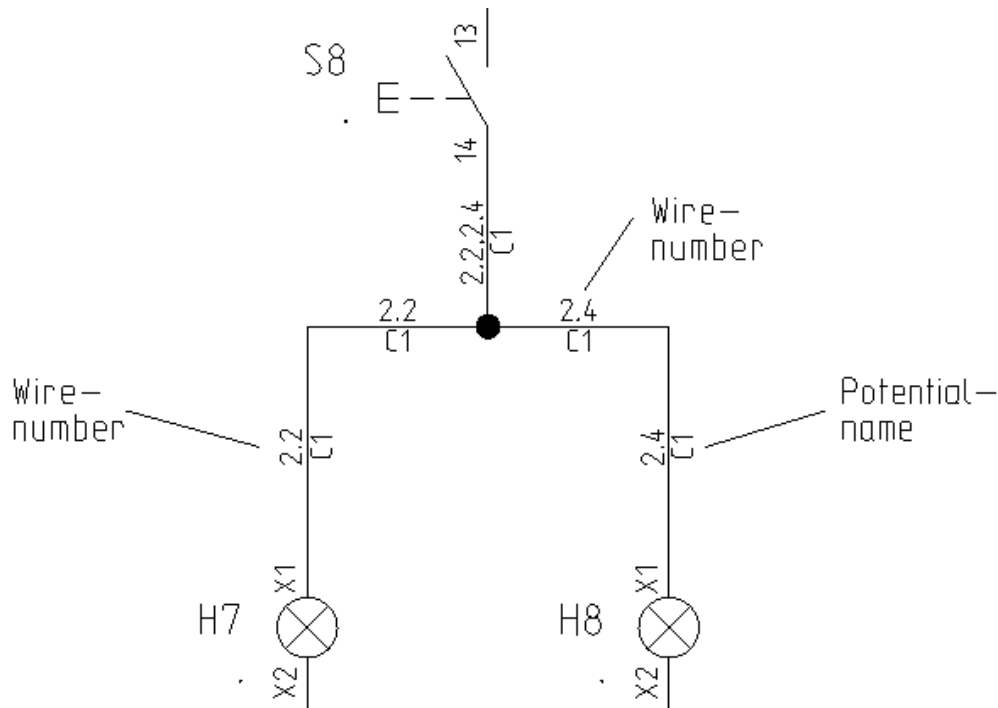
OK Cancel

Different options are available here. For more information, refer to the software's help.

If you work with signal properties, you can select the signal property from the Default menu in the **Electrical** category before you start drawing your potentials.

The properties, as you have specified them for the corresponding signal type, are displayed subsequently in the drawing.

Example:



You can use various ways to assign the desired values to the wires. It is your choice which one to use.

W.6. WORKING WITHOUT SIGNAL PROPERTIES

If you do not want to use signal properties, you can generate wire numbers and define the colour and size for your wires in case you need these values for your wire list.

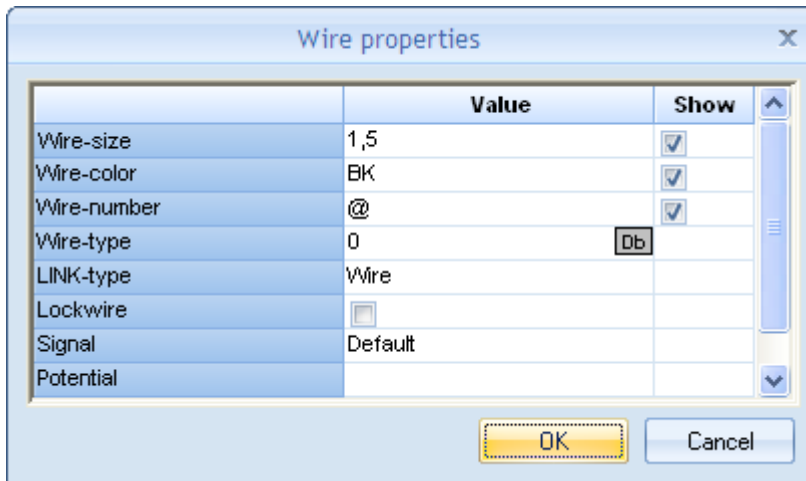
The "**Potential**" and "**Unique**" options in the **Wires** tab of the **Circuit Diagrams Properties** window define how the attributes are managed.

- Potential – all wires in one net have the same attributes such as colour, size and wire number.
- Unique – each wire can have its own attributes.

You can use various ways to assign the desired values to the wires. It is your choice which one to use.

W.7. VIEW AND CHANGE WIRE ATTRIBUTES

You can control the wire attributes defined for a specific wire. The following dialogue appear when you double click a wire:

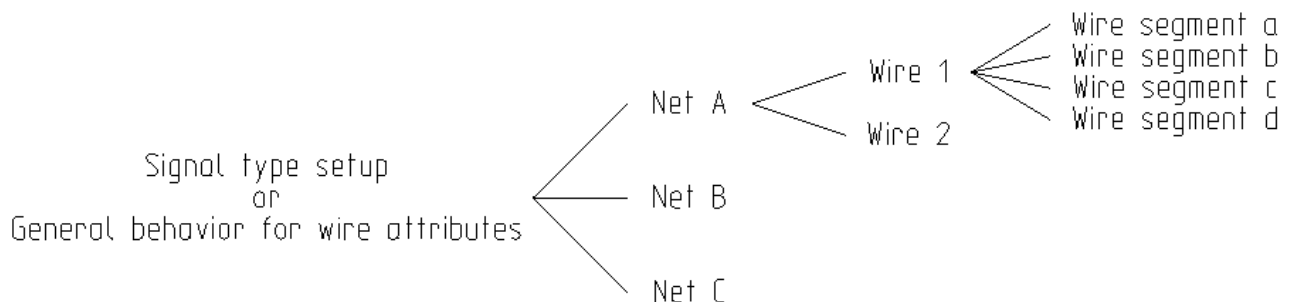


The information about the colour and the size of the wire is important if you work with Wires list. You can use various ways to assign the desired values to the wires. It is your choice which one to use.

In the **Properties** window of a wire you can switch on/off the visibility for the texts of the wire segments.

Segment Attributes	
Show wire number	On
Show potential	On
Show wire-size	On
Show wire-colour	On

The segment text will be displayed only in case a wire text is displayed from the **Circuit diagram Properties** and the **Wire properties**. If the visibility is switched off for a wire, the option for the wire segment will not have any importance. The visibility is determined in the following way:



W.8. DEFINE THE POSITION OF THE WIRE TEXT

It is possible to define the position of the text attributes for the wire number, colour and size. If you use Signal type of wire, it is possible to define additionally the attributes and the position of the potential name.

The positions of the wire properties have to be defined in a symbol. To define the settings for the horizontal and vertical wires you need to create two symbols – one for the horizontal wire and one for the vertical wire.

Each symbol has to contain one line which represents the wire and the texts for the wire number, the potential name, wire size and colour (the text have to have the text attributes (ID) for wire number, potential name, wire size and colour. The text properties such as text font, width and height, colour and angle will be used when you create the wire properties.

If there is no text representation in the symbol for one of the wire properties, the default settings will be used if the property will be displayed.

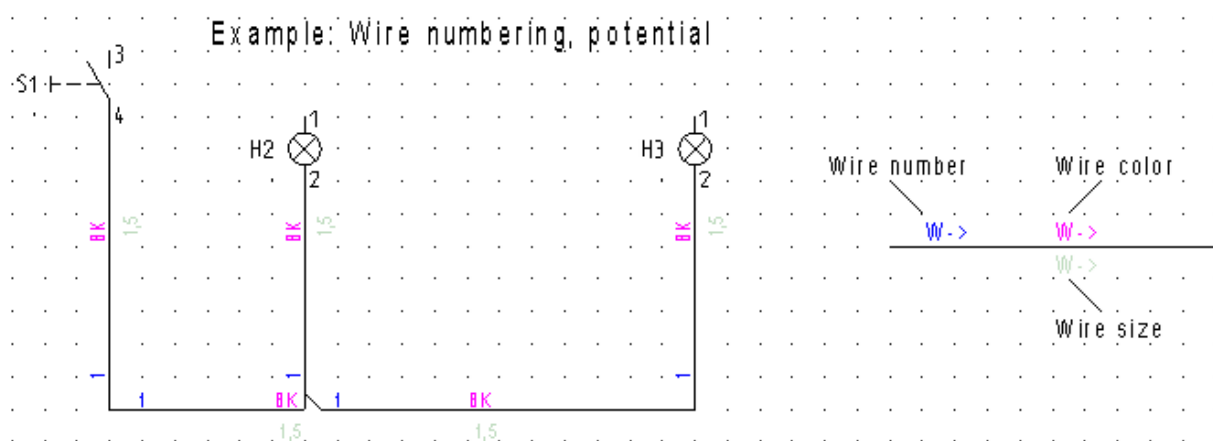
The position of the wire property on the wire will be taken from the position of the corresponding text in the symbol. The value of the text defines how will be treated the position in relation to the wire.

The following values are possible:

"W->" the position of the text will be calculated from the left extremity of the wire.

For example: the position of the text (dX, dY) in relation to the extremity of the line in the symbol is 5 mm. The "W->" value is defined in the text with the ID for the wire number.

When the wire number is created in the wire, it will be placed at 5 mm from the extremity of the wire regardless of the length of the wire.



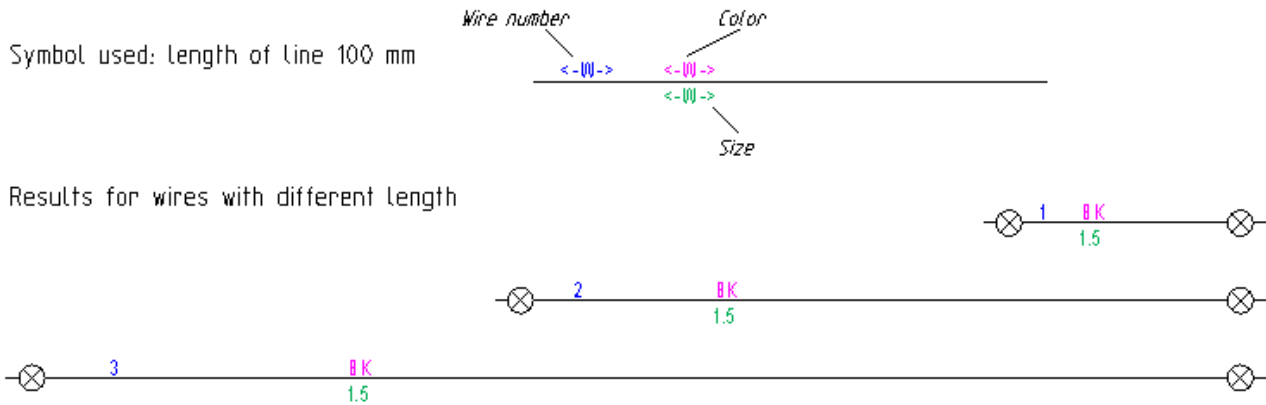
"<-W" the position of the text will be calculated from the right extremity of the wire. The alignment will be from the right extremity of the wire.

"<-W->" the position of the text will be calculated with the percent between the position of the wire property in relation with the wire.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

Example: wire numbering, potential



The symbol for the horizontal wire is supporting movable x- coordinate and the symbol for the vertical wire is working with movable y-coordinate.

Important: After you change a symbol, you have to restart SEE Electrical.

The symbols have to be defined in the registry of Windows in the following way:

<symbol database>\<symbol folder>\<symbol name>

The registry value for the horizontal wire symbol is ...\\1000\\WirePropSettings.

The registry value for the vertical wire symbol is ...\\1000\\WirePropSettingsV.

The setting is used for Circuit Diagrams EN and IEEE.

This way of defining the symbol properties is valid for all wire modes.

The registry settings

...\\Settings\\Text\\DefaultWirePropertiesFont

and

...\\Settings\\Text\\DefaultPotentialNameFont

X	TERMINAL MATRIX
---	-----------------

(standard)

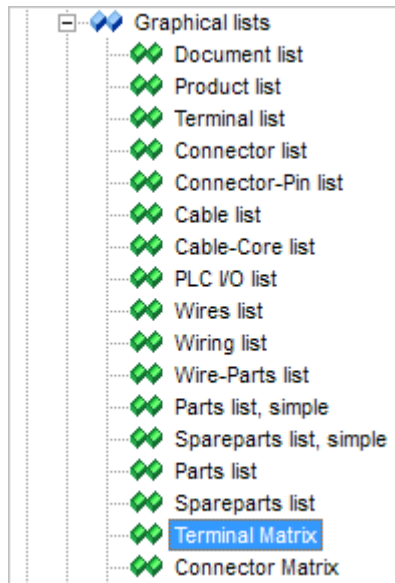
X.1. GENERATING A TERMINAL MATRIX

The terminal matrix facilitates the installation of terminal strips.

[illegible]

Exercise 22-1: Generate a terminal matrix for a terminal strip from the training project.

1. Select *Terminal matrix* from within the *Graphical Lists* in the Workspace tree. Expand the *Graphical Lists*.



2. Right-click with the mouse.
- 3.CO **Generate**
In the **Select Terminal Row** dialogue, choose the terminal strip(s) for which you wish to generate a matrix, for example X2 and X3.
- 4.> X2, X3
- 5.> OK
The Terminal matrices for the selected terminal strips have been created.

Exercise 22-2: Look at the terminal matrix. Each terminal strip appears on its own page.

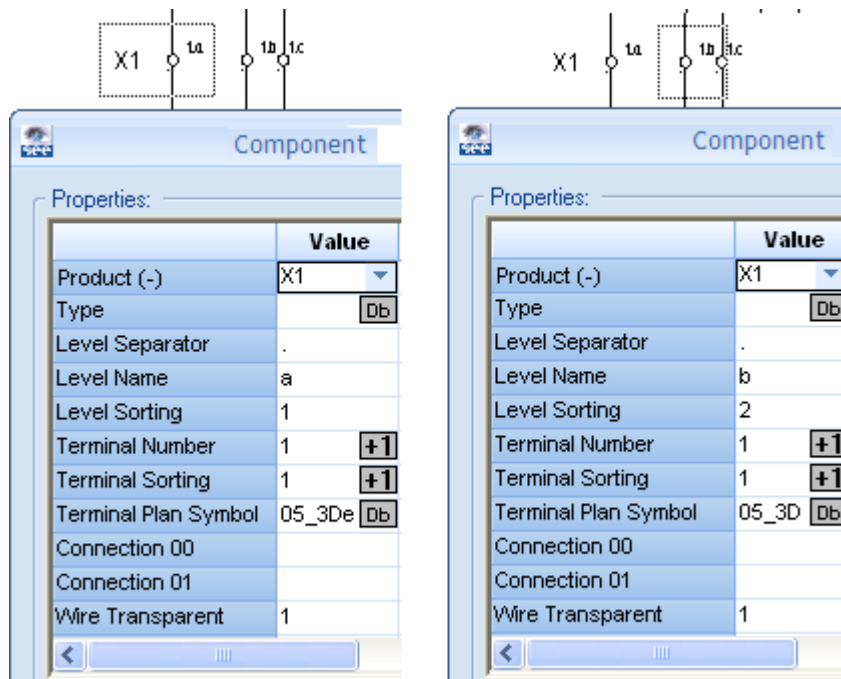
1. 0001
Double-click the page 1 beneath **Terminal Matrix** in the Workspace Explorer.
2. 0002
Double-click the page 2 beneath **Terminal Matrix** in the Workspace Explorer.

X.2. HANDLING MULTI-LAYER TERMINALS

You can handle multi-layer terminals in the *Circuit diagram* as follows:

- Place the multi-layer terminals.
- Enter the needed texts.

Single levels are connected together when the terminal name, terminal number, and terminal sorting are equal. (The terminal number can be also empty.)



- Select the answer "**Yes**" to the question if the components are to be combined.

In the terminal matrix, the terminal number, the level separator and the level name are entered together in the text placeholder for the terminal number.

X.3. CREATING A TEMPLATE FOR A TERMINAL MATRIX

Templates for terminal matrices are page templates with special properties.

X.3.1. GENERAL INFORMATION

Exercise 22-3: Load the current template for the terminal matrix and modify it as desired.

1. Graphical Lists
2. Terminal Matrix
Right-click
- 3.CO **Load Page Template**
The current template of the terminal matrix has been loaded.
- 4.CA **General > Select > All**
- 5.CA **Edit > Actions > Explode**
Now you can edit the template.

General approach for creating a terminal matrix:

- Create the graphics.
- Define text placeholders by choosing the **Draw > Elements > New Text** command.


You need the following text placeholders (available within *Attribute / Other node*):

Workspace name
Terminal Sheet
Date
Terminal Sheet Index
Function
Location
Terminal strip

Note	Cable type	Cable name			
Cabledesc. right	Cabletype right	Cablename right	.		
		9 cablelines			

Terminal Matrix

Terminal Strip:

Connection 1		Term.	No.	Connection 2	
Target left			Target right		
55 Lines					

Cable desc. Left	Cable type left	Cablename left	.		
		9 cablelines			

Position of the left target of the terminal (for example, Target left behind Connection1)

If you enter here "+P1", a potential appears as a target only at the first terminal, if several terminals are linked. If you enter another text, for example "Target left" as shown in the picture above, then each potential appears at each terminal for linked terminals too, as in the circuit diagram.

If you type "NoPot" as text, potentials or reference symbols don't appear as targets for terminals.

If the text placeholder for terminal number doesn't begin with a capital letter, a terminal number appears in each line where there is information about this terminal. If the text placeholder begins with a capital letter, the terminal number appears only in the first line.

Position of the right target of the terminal (for example, Target right behind Connection 2)

Number of lines available for terminals (for example 50 lines), the distance between the position text of the left target and the number of lines defines the distance between the third and next lines for terminals in the terminal matrix.

Sheet (and Sheet Index) where the terminal is located	Text placeholder for the function and location of the page	Column where the terminal is located	Terminal type	Terminal description
---	--	--------------------------------------	---------------	----------------------

In case you need reserve terminals, place the text "*Reserve terminals*" in the text placeholder for the left target. The text with an attribute *Spare-terminal* defines the target in the terminal matrix, if a reserve terminal is entered.

There is a feature for defining the index of the first and the last terminal in the page template. If the first available terminal in the Circuit diagram does not possess the index of the first terminal in the page template, reserve terminals are filled up to the first available index. After the last terminal, reserve terminals are filled up to the specified index.

To use this function, a text placeholder with the "normal text" property must be defined as follows:


```
#Spare +FirstIndex=<n> +LastIndex=<m> +N="<Reserve terminal> <Format  
Number>"
```

The text is inserted at the place where the terminal number for terminals in the Circuit diagram is located. No entry is made in the targets of the terminal left or right. (For more details, see chapter "*The Graphical Lists*" in the Help topics).

Cable desc. left	Cable type left	Cablename left					
		9 cablelines					

Terminal Matrix

Terminal Strip:

Connection 1	Term.	No.	Connection 2
Target left SS Lines		Number	Target right

Cablename left (if you type a dash "-", cable-name, cable-type and cable-description are not displayed)

Cable type left (Type1 – on the left) (if you type a dash "-", the cable-type is not displayed)

Cable-dimension left (if you enter an article number as a cable-type, you can type the text, such as NYM-J 5x1,5 in the cable-dimension)

Cable desc. left (on the left) (if you type a dash "-", the cable-description is not displayed)

Cable-core Number left, for example c- (on the left)

The texts with the attributes "Cable-core umber Left" (ID=180128) and "Cable-core Number Right" (ID=180131) are used to display a lot of information.

The first letter in the text allows you to define the information to be shown for a cable-core and the second letter allows you to set the text to be shown for a wire:

- displays nothing
- + displays the number (default)
- N displays the number (default)
- C displays the colour (if existing)
- c displays the colour (if existing) and the number (of the colour does not exist)
- E always displays the colour (also when number is not used)

If you need more information, you can position a text with the "Cable-core Number left" and "Cable-core Number right" attributes a second and a third time and thus display the information for signal type and signal name.

- Q displays information about the signal type
- P displays information about the potential name

Cable-core section left (on the left)

The cable-core section can be entered for cable-cores and/or wires, too. Two signs are needed here again. The combination Q- creates the section for cable-cores but not for wires, -Q creates the section for wires but not for cable-cores. For the Numbers of cable-lines left, the same rules apply as for the numbers of lines for terminals.

The following characters determine the appearance of the text:

- text is not shown
 - * text is always shown
 - + cable-core or wire square is shown only if cable-core or wire number is not empty.
- Example: -* -> cable core square is not shown, wire square is always shown.

Cable-name (right)

The same rules apply as for the cable texts left

Cable-type right

The same rules apply as for the cable texts left.

Cable-description right

The same rules apply as for the cable texts left.

Cable-core Number right

The same rules apply as for the cable texts left.

Cable-core section right

Numbers of cable-lines right, (the same rules apply as for the number of lines for terminals)

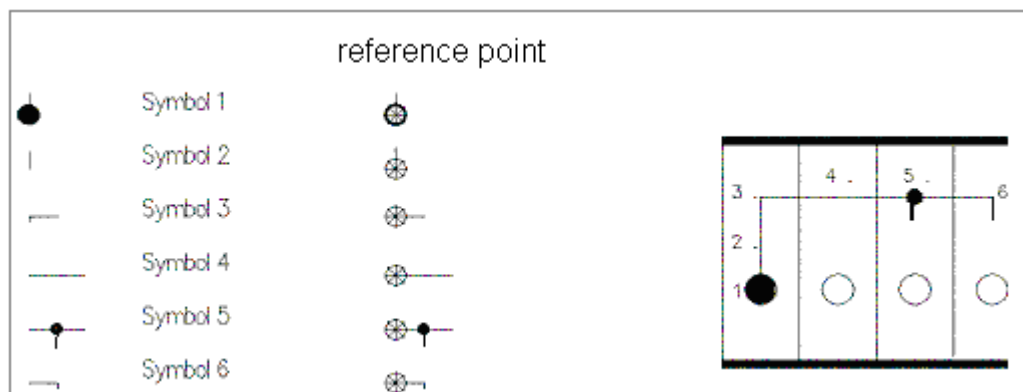
Additional texts can be entered manually.

Define the text placeholders for the bridges.

Target	No	Target
1M1/U	1	1K1/2
1M1/V	2	1K1/4
1M1/W	3	1K1/6
1M1/PE	4	PE
1M2/U	5	1K2/2
1M2/V	6	1K2/4
1M2/W	7	1K2/6
1M2/PE	8	PE
1H1/2	9	N
2M1/U	10	2K1/2
2M1/V	11	2K1/4
2M1/W	12	2K1/6
2M1/PE	13	PF

You need:

- ✓ one text placeholder (with the "LINK Type" text attribute);
- ✓ If the bridge type is not in between 1 to 10: if this text contains the letter P, then bridges identified via potentials are displayed; 0 => bridges identified via wires are displayed;
- ✓ 1 to 10: bridges of the appropriate LINK Type are shown)
- ✓ 6 symbols

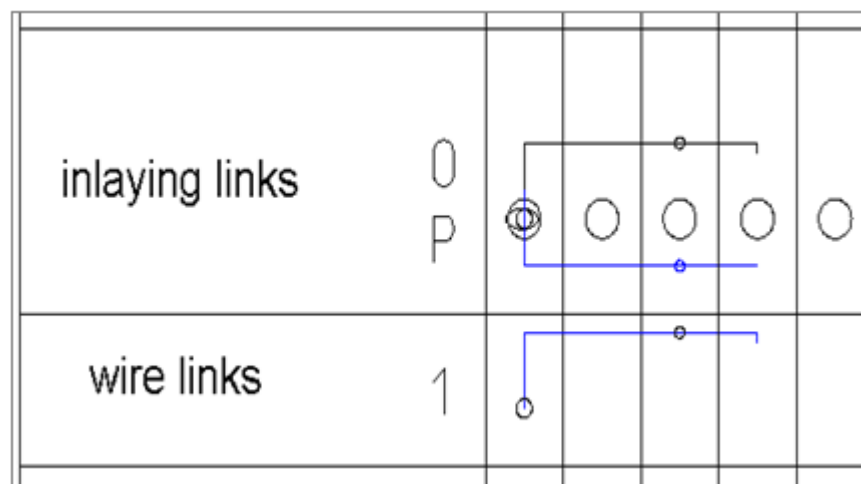


Create the symbols as follows:

1. Draw the graphics
2. Select the graphics of the symbol
3. Block to a symbol LINK Type 1 or LINK Type 2 or ... LINK Type 6
4. Drag the symbol into the symbol database
5. Delete the graphics used for the symbol
6. Pull the symbol out of the database and insert it.

After you have created and inserted all symbols:

1. Select 6 symbols and the text for the bridge
2. Block as Macro/Group
3. Move the symbol into the symbol database
4. Delete the graphics
5. Pull the symbol out of the database and insert it.



For each LINK Type, you must create a text placeholder (see the picture above - for example, bridge placeholders for P, 0 and 1). If a LINK Type is not defined in the template, then a LINK Type 0 is used.

If a LINK Type P is not placed, bridges identified via wires are not displayed. In this case, the target terminal is shown.

Here, you cannot use all of the settings options available via the template. For more details see chapter "Graphical lists" in the User Manual.

Exercise 22-3: Save the template.

1.CA **General > Select > All**

2.CA **Edit > Actions > Block**

Group the elements as Page Template, Title block symbol.

3.CO **File > Save as... > Page Template**

Define a name for the template and save it.

Set the new terminal matrix as a template in the properties of Terminal matrix.

X.3.2. DRAWING GRAPHICS ASSOCIATED WITH EACH TERMINAL – ONE GEOMETRY FOR ALL TERMINALS

This allows you to draw the geometry for all terminals belonging to the terminal strip.

[illegible]

Generating the page template:

1. Position all the elements of a terminal matrix or a terminal plan, as already described. Do not position the place holder for terminal number. (You can use an existing template and delete the geometry/texts that you do not need.)

		Path Sheet	
Note	Cable type	Cable name Cable
Cabledesc. right	Cablotype right	Cablename right	
		9 cablesines	

Terminal Matrix

Terminal Strip:

Connection 1
Targer left

No.
Term.
Connection 2
Targer right

Cable desc. Left	Cable type left	Cablenome left
		9 cablesines

Note	Cable type	Cable name Cable
------	------------	------------	-------------------

- Logo -

Project:
Date:

Drawing no.:
Function:

Location:

Rev.:
Page:

Intl.:

2. Generate a *Block/Macro/Group* symbol from all these elements (except for the terminal number, if there is one).

Generating the geometry for each single terminal:

- 3a. Draw the geometry for the first part of the terminal.

Group all the elements as a "*Terminal, Line-Block: start*" symbol. If the circle/ellipse for the bridge shall appear only once in a terminal, please add it to the "*Terminal, Line-Block: start*" symbol.

2H1:X1	○	13	X2:14 X2:13
2H3:X1	○	14	2S3:12
2H4:X1	○	15	X2:16 X2:15

- 3b. Draw the geometry for the last part of the terminal.

Group all the elements as a "*Terminal, Line-Block: end*" symbol.

- 3c. Draw the geometry for the middle part of the terminal (these elements will eventually need to be extended).

Group all the elements as a "Terminal, Line-Block: trim" symbol.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

- 3e. If you want, you can draw geometry for the terminal number. This part will contain the geometry for the bridge, in case the circle appears in each line or if a terminal occupies two or more lines.

Group all the elements as a symbol of the type "**Graphical symbol**".

2H1:X1	○	13	X2:14
.	○	.	X2:13
2H3:X1	○	14	2S3:12
2H4:X1	○	15	X2:16
.	○	.	X2:15


- 3f. Position the text for terminal number.
- 3g. Group the four symbols mentioned above together with the text for terminal number as a "*Block/Macro/Group*" symbol.
4. Group all the symbols (the macro/group for the standard sheet and the macro/group for single terminals) as a "*Page Template, Title block*" symbol.
5. Save the new page template.

X.3.3. DRAWING GRAPHICS ASSOCIATED WITH EACH TERMINAL – ONE SPECIFIC GEOMETRY FOR EACH KIND OF TERMINAL

Unlike the situation described above, in this case the graphics for the terminal is not taken from the page template, but from a symbol database.

Each terminal can be drawn with the help of a specific graphical symbol (stored in the symbol database), so you can use different symbols for the different types of terminals (for example, spare terminals, end and separator plates).

Terminal Strip: X2



Connection 1	Term.	No.	Connection 2
○ S1:3	○	1	L3
▽ S1:4	○	2	K2:21
○ S2:3	○	3	L3
○ S2:4	○	4	K1:21

The symbol for the terminal in the terminal plans has to be assigned to the single terminals, as described below in the part "Which symbols are used for the terminal?"

How to make a new template from an existing terminal plan template:

Remove all graphics and texts associated with the terminal symbol, except for the terminal number (this text marks the position of the first terminal symbol).

Add a normal text defining a default terminal symbol. The syntax is:

```
#LineSymbol="<symbol database name>\<folder name>\<symbol name>"
```

For example:

```
#LineSymbol="Graphical List\TerminalPlan\0Terminal"
```

Generating a terminal symbol for the graphical plan (to store in the symbol database)

The symbol needs to contain the graphics of the terminal symbol, as described above (graphical symbol (Id 180180)) and the following placeholder texts:

- Terminal number (Id 180112) – this text is obligatory
- Terminal connection text left (Id 180152) and right (Id 180153) – optional
- Terminal type (Id 180140) – optional
- Terminal description (Id 180142) – optional
- Terminal free texts 1, 2 and 3 (Ids 180180, 180181 and 180182) – optional

And the page reference texts:

- Terminal Page Function (=) (Id 180146), Terminal Page Location (+) (Id 180148), Terminal Sheet (Id 180115), Terminal Sheet Index (Id 180143), Terminal Cell (Id 180116) – optional

Note :

The size of the terminal symbol graphics has to fit the line distance given for the terminal plan (distance between the text for "Target left" and the text for "Number of lines for terminals"). The size and distance must be the same.

Where is the symbol placed in the terminal plan?

In the template, the position of the first line is marked by the "*Terminal number*" placeholder text (Id 180112).

If the terminal symbol has a normal text with the contents "#SybPos", this text is put on top of the placeholder text "Terminal number" (Id 180112) in the template (including the line offset). If no "#SybPos" text is found in the terminal symbol, the "Terminal number" placeholder text from the template is used to position the terminal symbol.

The search is faster if the "*Terminal number*" placeholder text in the symbol contains the "#SybPos" text.

Which symbols are used for the terminal?

The symbol name for the symbol used in Terminal Matrix, Terminal Plan and Terminal Row Picture Plan can be specified by the "*Terminal Plan Symbol*" component text in the circuit diagram or inside the *Type* database by the "*Terminal Plan Symbol*" type property (Id 12021300), in case you wish to use the same terminal symbol for all terminals of one type. Define the symbol name without specifying a symbol database and folder. The name of the symbol database and folder are given in the page template in the #LineSymbol text.

For spare terminals, the default terminal symbol is used.

Hint

In the *advanced* level you can define spare terminals via the Terminal Editor.

Inserting headers and endplates

You can insert headers and endplates, if necessary.

The default symbols for header and endplate are defined in the page template. Add to the #LineSymbol= placeholder an additional text with:

+P="<name of symbol for template>"

+H="<name for symbol for header>"

Example of an entry in the Terminal Plan template:

#LineSymbol="Graphical List\TerminalRowPicture\0Terminal" +H="0Header" +P="0Plate"

Plates and spare terminals from the "Component without graphics" list (*advanced*)

A plate is similar to a terminal and you can manage it as a terminal without graphic (manual component).

The symbol name of a plate can be specified in the same way as for a terminal – by using the "*Terminal Plan Symbol*" (Id 160450) or the type property (Id 12021300).

If the "+P" keyword is available in this text, then the header information ("H") from the next terminal is used in the Terminal Row Picture plan. Please note that if no header is specified, this means that the default header is used. If no symbol is defined, the default plate symbol is used instead of the default terminal symbol.

This means that "+P=0Plate" is the same as "+T=0Plate +P"

You can also define spare terminals with the Terminal Editor. If you want to add this to the plans, you have to use the "*Spare terminal text*" keyword. Replace the "*Spare*" keyword, in case you have used it to manage spare terminals up to now. If not, please refer to the text above about how to use the "*Spare*" keyword and the "*Spare terminal text*" keyword instead of this. The difference between the "*Spare*" and "*Spare terminal text*" keywords is that exactly the terminals defined in the **Terminal Editor** are used as spare terminals.

Terminal Matrix

Terminal Strip: **X1**

Connection 1	Term.	No.	Connection 2
SUPPLY 1/4	1	1/2	1/4
SUPPLY 1/4	2	1/4	1/4
SUPPLY 1/4	3	1/4	1/4
SUPPLY 1/4	4	1/4	1/4
SUPPLY 1/4	5	1/4	1/4
SUPPLY 1/4	6	1/4	1/4
SUPPLY 1/4	7	1/4	1/4
SUPPLY 1/4	8	1/4	1/4
SUPPLY 1/4	9	1/4	1/4
SUPPLY 1/4	10	1/4	1/4
SUPPLY 1/4	11	1/4	1/4
SUPPLY 1/4	12	1/4	1/4
SUPPLY 1/4	13	1/4	1/4
SUPPLY 1/4	14	1/4	1/4
SUPPLY 1/4	15	1/4	1/4
SUPPLY 1/4	16	1/4	1/4
SUPPLY 1/4	17	1/4	1/4
SUPPLY 1/4	18	1/4	1/4
SUPPLY 1/4	19	1/4	1/4
SUPPLY 1/4	20	1/4	1/4
SUPPLY 1/4	21	1/4	1/4
SUPPLY 1/4	22	1/4	1/4
SUPPLY 1/4	23	1/4	1/4
SUPPLY 1/4	24	1/4	1/4
SUPPLY 1/4	25	1/4	1/4
SUPPLY 1/4	26	1/4	1/4
SUPPLY 1/4	27	1/4	1/4
SUPPLY 1/4	28	1/4	1/4
SUPPLY 1/4	29	1/4	1/4
SUPPLY 1/4	30	1/4	1/4
SUPPLY 1/4	31	1/4	1/4
SUPPLY 1/4	32	1/4	1/4
SUPPLY 1/4	33	1/4	1/4
SUPPLY 1/4	34	1/4	1/4
SUPPLY 1/4	35	1/4	1/4
SUPPLY 1/4	36	1/4	1/4
SUPPLY 1/4	37	1/4	1/4
SUPPLY 1/4	38	1/4	1/4
SUPPLY 1/4	39	1/4	1/4
SUPPLY 1/4	40	1/4	1/4
SUPPLY 1/4	41	1/4	1/4
SUPPLY 1/4	42	1/4	1/4
SUPPLY 1/4	43	1/4	1/4
SUPPLY 1/4	44	1/4	1/4
SUPPLY 1/4	45	1/4	1/4
SUPPLY 1/4	46	1/4	1/4
SUPPLY 1/4	47	1/4	1/4
SUPPLY 1/4	48	1/4	1/4
SUPPLY 1/4	49	1/4	1/4
SUPPLY 1/4	50	1/4	1/4
SUPPLY 1/4	51	1/4	1/4
SUPPLY 1/4	52	1/4	1/4
SUPPLY 1/4	53	1/4	1/4
SUPPLY 1/4	54	1/4	1/4
SUPPLY 1/4	55	1/4	1/4
SUPPLY 1/4	56	1/4	1/4
SUPPLY 1/4	57	1/4	1/4
SUPPLY 1/4	58	1/4	1/4
SUPPLY 1/4	59	1/4	1/4
SUPPLY 1/4	60	1/4	1/4
SUPPLY 1/4	61	1/4	1/4
SUPPLY 1/4	62	1/4	1/4
SUPPLY 1/4	63	1/4	1/4
SUPPLY 1/4	64	1/4	1/4
SUPPLY 1/4	65	1/4	1/4
SUPPLY 1/4	66	1/4	1/4
SUPPLY 1/4	67	1/4	1/4
SUPPLY 1/4	68	1/4	1/4
SUPPLY 1/4	69	1/4	1/4
SUPPLY 1/4	70	1/4	1/4
SUPPLY 1/4	71	1/4	1/4
SUPPLY 1/4	72	1/4	1/4
SUPPLY 1/4	73	1/4	1/4
SUPPLY 1/4	74	1/4	1/4
SUPPLY 1/4	75	1/4	1/4
SUPPLY 1/4	76	1/4	1/4
SUPPLY 1/4	77	1/4	1/4
SUPPLY 1/4	78	1/4	1/4
SUPPLY 1/4	79	1/4	1/4
SUPPLY 1/4	80	1/4	1/4
SUPPLY 1/4	81	1/4	1/4
SUPPLY 1/4	82	1/4	1/4
SUPPLY 1/4	83	1/4	1/4
SUPPLY 1/4	84	1/4	1/4
SUPPLY 1/4	85	1/4	1/4
SUPPLY 1/4	86	1/4	1/4
SUP			


Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

Generating the Page template:

#PageBreak=0 #-Lines=8

<div style="margin-bottom: 10px;"> Path Sheet </div> <table border="1" style="width: 100%; border-collapse: collapse;"> <tr><td>Cable name</td></tr> <tr><td>cable name right</td></tr> <tr><td>8 cable lines right</td></tr> <tr><td> </td></tr> <tr><td> </td></tr> <tr><td> </td></tr> <tr><td> </td></tr> <tr><td> </td></tr> </table> <div style="margin-top: 10px;"> Terminal Strip: X? </div> <table border="1" style="width: 100%; border-collapse: collapse;"> <tr> <td>Connection1</td> <td>Term.</td> <td>No.</td> <td>Connection2</td> </tr> <tr> <td>Ziel links</td> <td></td> <td>Nummer</td> <td>Ziel rechts</td> </tr> <tr> <td>60 Zeilen</td> <td></td> <td></td> <td></td> </tr> </table> <table border="1" style="width: 100%; border-collapse: collapse;"> <tr><td>Cable name left</td></tr> <tr><td>9 cable lines</td></tr> <tr><td> </td></tr> <tr><td> </td></tr> <tr><td> </td></tr> <tr><td> </td></tr> <tr><td> </td></tr> <tr><td> </td></tr> <tr><td> </td></tr> </table> <div style="margin-top: 10px;"> Cable name </div>	Cable name	cable name right	8 cable lines right						Connection1	Term.	No.	Connection2	Ziel links		Nummer	Ziel rechts	60 Zeilen				Cable name left	9 cable lines								<div style="text-align: center; font-size: 2em; margin-top: 100px;">X?</div>
Cable name																														
cable name right																														
8 cable lines right																														
Connection1	Term.	No.	Connection2																											
Ziel links		Nummer	Ziel rechts																											
60 Zeilen																														
Cable name left																														
9 cable lines																														



Project
Drawing no :
Rev :
Int :

Date:

Function:

Location:

Page:

1. Insert the elements of a terminal matrix or a terminal plan as described. Do not insert the placeholders for the terminal strip and number. (You can use an existing template and delete the geometry/texts that you do not need).



2. Add a "Normal" text attribute to the content.

#PageBreak=0

or

#PageBreak=0 #Lines=8 (for example)

where #Lines=8 indicates that the terminal strip header will use the space for eight terminal lines (if you do not use any #Lines string to specify the terminal row header, the space will be calculated from the graphics in the template.)

The text "#Pagebreak" in the template can be used to control the generation of the new page.

#PageBreak=0

Multiple terminal strips are generated on one page like before, a new page is generated only when the previous page is already filled.

#PageBreak=1 or #PageBreak=160010 or #PageBreak="-"

are used to start a new page if the name of the terminal has been changed.

#PageBreak=140020 or #PageBreak="="

are used to start a new page if the function (=) has been changed.

#PageBreak=140050 or #PageBreak="+"

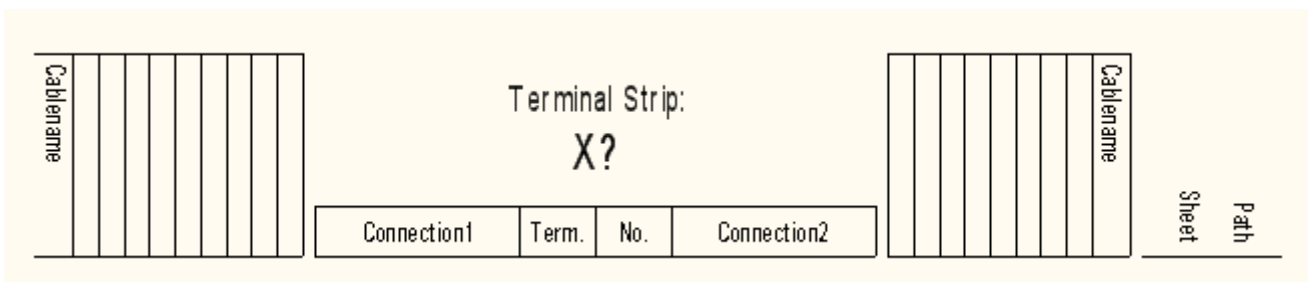
Are used to start a new page if the function (+) has been changed.

Additionally, you can define how to manage the function and location information in the terminal strip. If you add the text +DL0 to the #PageBreak command, the terminal strip name is always extended with the function and location information (even if the terminal strip has the same function and location information as the page where it is located).

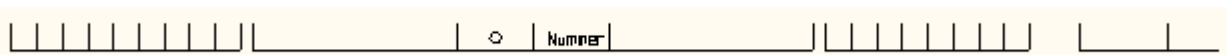
If you add the text +DL1 to the #PageBreak command, the terminal row name will contain only the function and location information in case they are different from the ones on the page.

3. Generate a *Page template* symbol from the inserted elements, except for the terminal number and terminal strip name.

1. The terminal row header part has to be created in the following way
All the geometry and texts that show information that is never changed have to be grouped as a "Graphical symbol". The graphical symbol and a text with the attribute "Terminal number" have to be grouped as "Macro"/"Group".



5. All the geometry and text that show information about the terminal that is never changed have to be grouped as a "Graphical symbol". The graphical symbol and a text with the attribute "Terminal number" have to be grouped as "Macro"/"Group".



6. Group all elements as a symbol of the type "Graphical symbol"

2H1:X1	⊙	13	X2:14
.	⊙	.	X2:13
2H3:X1	⊙	14	2S3:12
2H4:X1	⊙	15	X2:16
.	⊙	.	X2:15

7. Insert the text for the terminal number
8. Group the four symbols and the text for the terminal number as a "*Block/Macro/Group*" symbol.
9. Group all symbols (the macro/group for the standard sheet and the macro/group for the single terminals) as a "Page Template, Title Block" symbol.
10. Save the new page template.

Y CABLE PLAN WITH GRAPHICS

(standard)

The *Cable plan* with graphics enables a clearly-presented documentation of the cable cores and their targets.

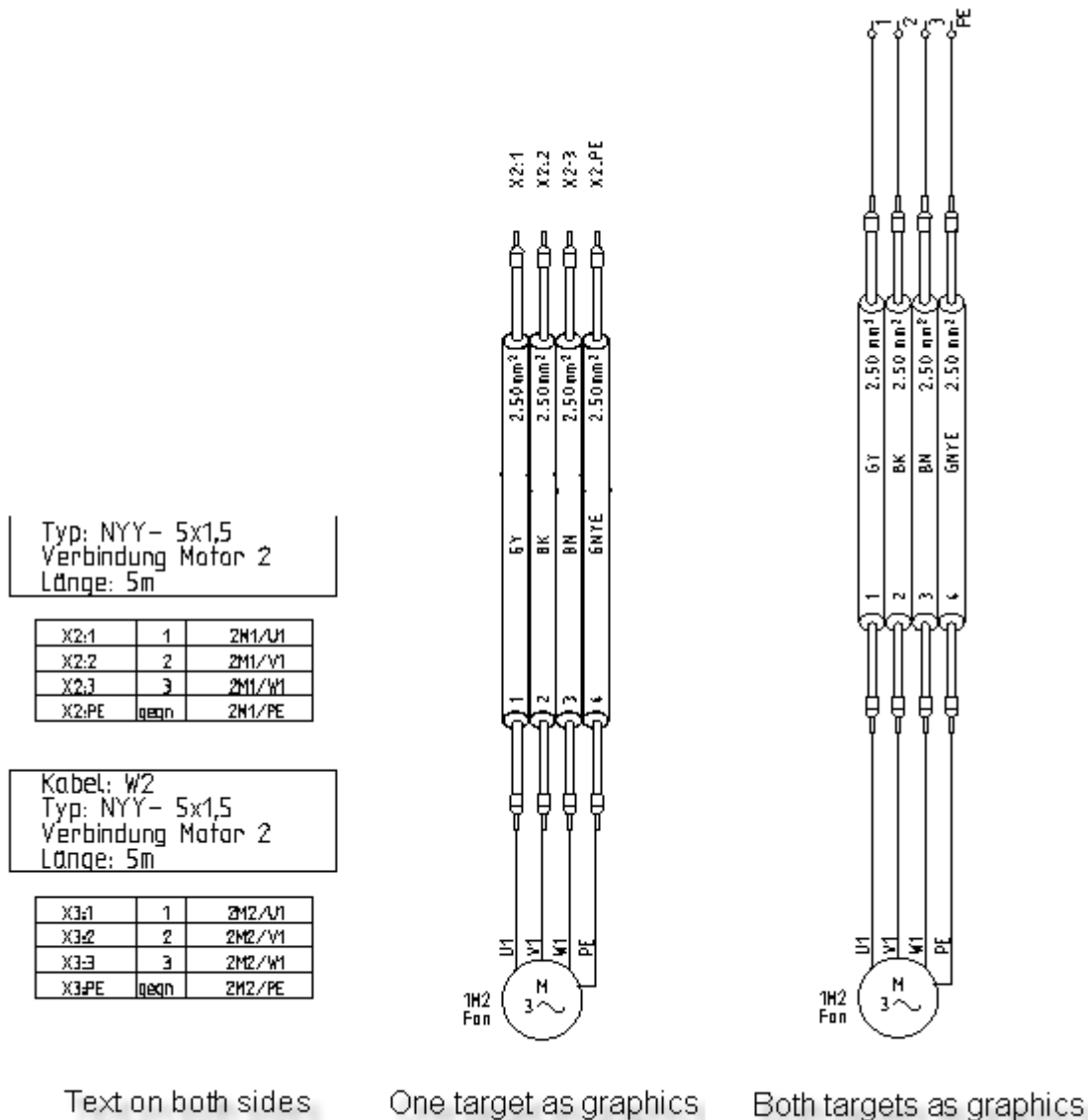
0	1	2	3	4	5	6	7	8	9	Page	Path																																
<table border="1" style="width: 100%; border-collapse: collapse;"> <tr> <td style="width: 33%;">Cable: W1</td> <td style="width: 33%;">Type: U-1000 R2V 4G2,5²</td> <td style="width: 33%;">Cable type:</td> </tr> <tr> <td>Remark:</td> <td>Length:</td> <td></td> </tr> </table>										Cable: W1	Type: U-1000 R2V 4G2,5 ²	Cable type:	Remark:	Length:																													
Cable: W1	Type: U-1000 R2V 4G2,5 ²	Cable type:																																									
Remark:	Length:																																										
<table style="width: 100%;"> <tr> <td style="width: 10%;">1M2-UI</td> <td style="width: 10%;">1</td> <td style="width: 10%;">DY</td> <td style="width: 10%;">2.50mm²</td> <td style="width: 10%;">1</td> <td style="width: 10%;">X2-1</td> <td style="width: 10%;">1</td> <td style="width: 10%;">2</td> </tr> <tr> <td>1M2-V1</td> <td>2</td> <td>BK</td> <td>2.50mm²</td> <td>2</td> <td>X2-2</td> <td>1</td> <td>2</td> </tr> <tr> <td>1M2-W1</td> <td>3</td> <td>BN</td> <td>2.50mm²</td> <td>3</td> <td>X2-3</td> <td>1</td> <td>2</td> </tr> <tr> <td>1M2-PE</td> <td>4</td> <td>DNYE</td> <td>2.50mm²</td> <td>4</td> <td>X2-PE</td> <td>1</td> <td>2</td> </tr> </table>										1M2-UI	1	DY	2.50mm ²	1	X2-1	1	2	1M2-V1	2	BK	2.50mm ²	2	X2-2	1	2	1M2-W1	3	BN	2.50mm ²	3	X2-3	1	2	1M2-PE	4	DNYE	2.50mm ²	4	X2-PE	1	2		
1M2-UI	1	DY	2.50mm ²	1	X2-1	1	2																																				
1M2-V1	2	BK	2.50mm ²	2	X2-2	1	2																																				
1M2-W1	3	BN	2.50mm ²	3	X2-3	1	2																																				
1M2-PE	4	DNYE	2.50mm ²	4	X2-PE	1	2																																				
<table border="1" style="width: 100%; border-collapse: collapse;"> <tr> <td style="width: 33%;">Cable: W2</td> <td style="width: 33%;">Type: U-1000 R2V 4G2,5²</td> <td style="width: 33%;">Cable type:</td> </tr> <tr> <td>Remark:</td> <td>Length:</td> <td></td> </tr> </table>										Cable: W2	Type: U-1000 R2V 4G2,5 ²	Cable type:	Remark:	Length:																													
Cable: W2	Type: U-1000 R2V 4G2,5 ²	Cable type:																																									
Remark:	Length:																																										
<table style="width: 100%;"> <tr> <td style="width: 10%;">1M4-UI</td> <td style="width: 10%;">1</td> <td style="width: 10%;">DY</td> <td style="width: 10%;">2.50mm²</td> <td style="width: 10%;">1</td> <td style="width: 10%;">X2-4</td> <td style="width: 10%;">1</td> <td style="width: 10%;">4</td> </tr> <tr> <td>1M4-V1</td> <td>2</td> <td>BK</td> <td>2.50mm²</td> <td>2</td> <td>X2-5</td> <td>1</td> <td>4</td> </tr> <tr> <td>1M4-W1</td> <td>3</td> <td>BN</td> <td>2.50mm²</td> <td>3</td> <td>X2-6</td> <td>1</td> <td>4</td> </tr> <tr> <td>1M4-PE</td> <td>4</td> <td>DNYE</td> <td>2.50mm²</td> <td>4</td> <td>X2-PE</td> <td>1</td> <td>4</td> </tr> </table>										1M4-UI	1	DY	2.50mm ²	1	X2-4	1	4	1M4-V1	2	BK	2.50mm ²	2	X2-5	1	4	1M4-W1	3	BN	2.50mm ²	3	X2-6	1	4	1M4-PE	4	DNYE	2.50mm ²	4	X2-PE	1	4		
1M4-UI	1	DY	2.50mm ²	1	X2-4	1	4																																				
1M4-V1	2	BK	2.50mm ²	2	X2-5	1	4																																				
1M4-W1	3	BN	2.50mm ²	3	X2-6	1	4																																				
1M4-PE	4	DNYE	2.50mm ²	4	X2-PE	1	4																																				
<div style="display: flex; justify-content: space-between; align-items: center;"> <div style="width: 30%;"> </div> <div style="width: 60%;"> <table border="1" style="width: 100%; border-collapse: collapse;"> <tr> <td style="width: 20%;">Project: My Workspace</td> <td style="width: 20%;">Drawing no:</td> <td style="width: 20%;">Unit:</td> <td style="width: 20%;">Rev:</td> <td style="width: 20%;">Sheet: 1</td> </tr> <tr> <td>Date: 25/09/2009</td> <td>Function:</td> <td>Location:</td> <td>Total sheets:</td> <td>Next sheet:</td> </tr> </table> </div> </div>										Project: My Workspace	Drawing no:	Unit:	Rev:	Sheet: 1	Date: 25/09/2009	Function:	Location:	Total sheets:	Next sheet:																								
Project: My Workspace	Drawing no:	Unit:	Rev:	Sheet: 1																																							
Date: 25/09/2009	Function:	Location:	Total sheets:	Next sheet:																																							

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

The targets of the cable-cores can be displayed as text information or graphics.

Examples:



Y.1. CREATING A CABLE PLAN WITH GRAPHICS

Exercise 23-1: Create the *Cable plan with graphics* for the training workspace.

1. Select **Cable plan** within the **Graphical lists** area in the Workspace tree.
2. Right-click with the mouse.
- 3.CO **Generate**
The *Cable plan with graphics* is generated.

Exercise 23-2: Viewing the *Cable plan*.

1. 0001
Select page 1 of the Cable plan with graphics by double-clicking on 0001 beneath *Cable plan* in the *Workspace tree*.

Y.2. CREATING A TEMPLATE FOR CABLE PLANS WITH GRAPHICS

The template consists of the following parts:

- ✓ Elements of the standard sheet -> Page Template, Title block symbol
- ✓ Texts for first page number and page break (optional) (PageBreak=, + or -)
- ✓ Header for Cable (Name, Type, etc.)
- ✓ Data about Cable core number
- ✓ Target left/ Target right
- ✓ Number of lines for Cable cores

Exercise 23-3: Create a template for Cable plan with graphics.

1. Draw the desired graphics and place the needed texts. Select All. Group the selected elements as a "Page Template, Title block" symbol.
2. Insert texts for the first page number and page break (optional) (PageBreak=, + or -)
3. Create the Header for the cable:
 - 3a. Group the graphics and the text with the "normal text" attribute as a "Graphical Symbol".
 - 3b. Insert the texts for the cables. These are texts with the attribute "Cable name", "Description", "Type", etc.
(Optional: A text of the form "Type: %s" provides the result, for instance Type: NYY 5x1,5.)
 - 3c. Group the "Graphical Symbol" and the texts for the cable as a "Block/Macro/Group" symbol.
4. Define the view of the data for the Cable core number.
 - 4a. Group the graphics and the text with the "normal text" attribute as a "Graphical Symbol".
 - 4b. Insert texts for:
Cable core number (+c (colour/number) or +C (only colour) or + (only number) or – (nothing);
Other cable attributes: colour, cross section, potential name, signal type of wire:

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

Position a text with the "cable core colour" or "cable core square" attributes. Type the code given below into this placeholder. The text is replaced by the associated text:

"S" – cable core square

"Q" – signal type of wire

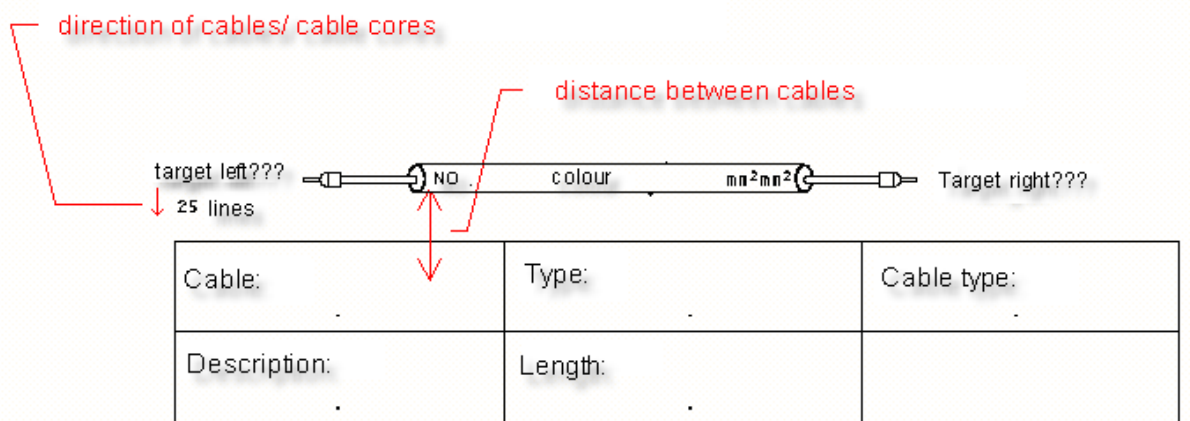
"P" – potential name for wire

Maximum five instances of the text for cable core colour and cable core square can be used, for example three times colour and two times square.

Function, Location, Sheet, Index, and Column where the Cable core is located (optional).

- 4c. Group the selected elements as a Block/Macro/Group symbol.
5. Define the targets left and right.
- 5a. You need texts with the attribute "Target left" (or "Target right").
If you type in +NoSymbol into the text for the target left, no target symbols will be displayed, only the component text is used as target.
- 5b. If Circuit diagram symbols must be placed, 2 routes and 1 text are required for the position of the symbol (look in the "List of Construction set" symbol library, "Cable plan with graphics" folder).
6. Define the distance between different cables on one page.
You can define the distance via the texts "Cable core" in the Cable core information, and "Cable name" in the Cable information.

You can define the direction, in which cable and cable cores appear in the template, by means of the distance between the texts "Target left" and "Number of lines" in the Cable core information.



7. Save the page template.
8. Assign the new page template in the Properties of the Cable plan.
Here, you cannot use all of the settings options available via the template. For more details, see chapter "Graphical lists" in the Help topics.

Y.3. SHOWING NOT USED CABLE CORES IN CABLE PLAN

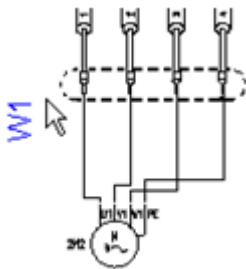
Spare cores can only be shown, if the cable got a type and the number of cores is defined inside this type.

If you want to show the spare cores, the template must be changed. Just add a text with attribute "normal" and content #Spare or #Spare=1.

If a text with content #Spare=0 is found, no spare cores are added to the cable plan.

Y.4. CABLE PLAN WITH SHIELD INFORMATION FOR CABLES

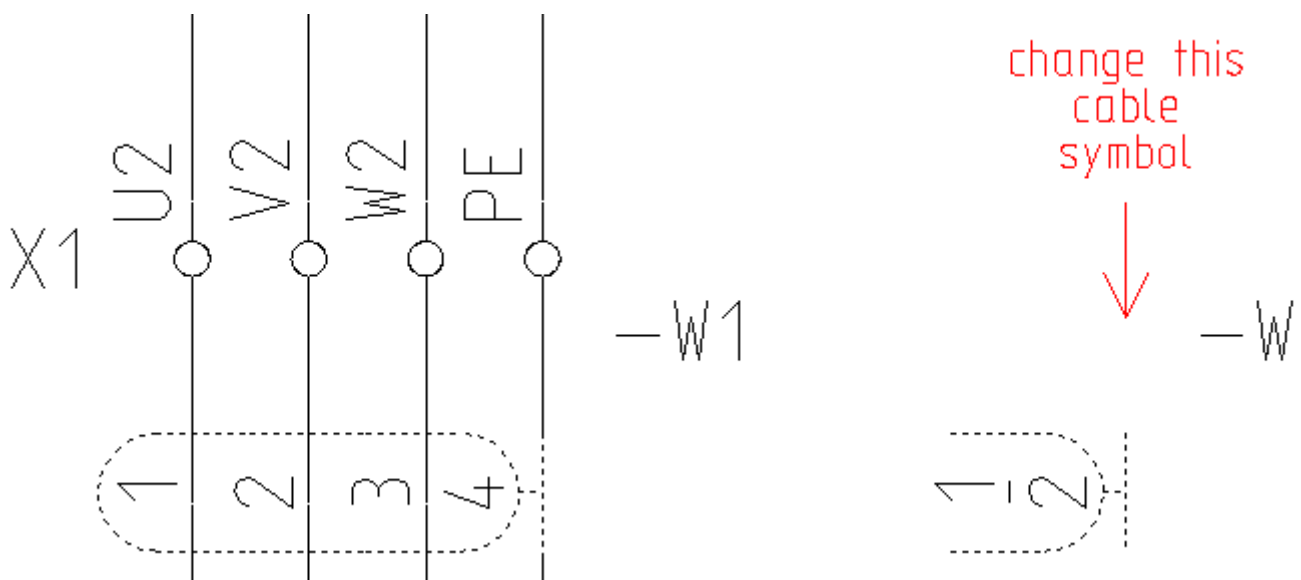
If you use user-created cable symbols for shields inside the circuit diagram, you may want to show these symbols in the graphical cable plan as well. This is possible with the following 3 steps:



1. Circuit diagram:

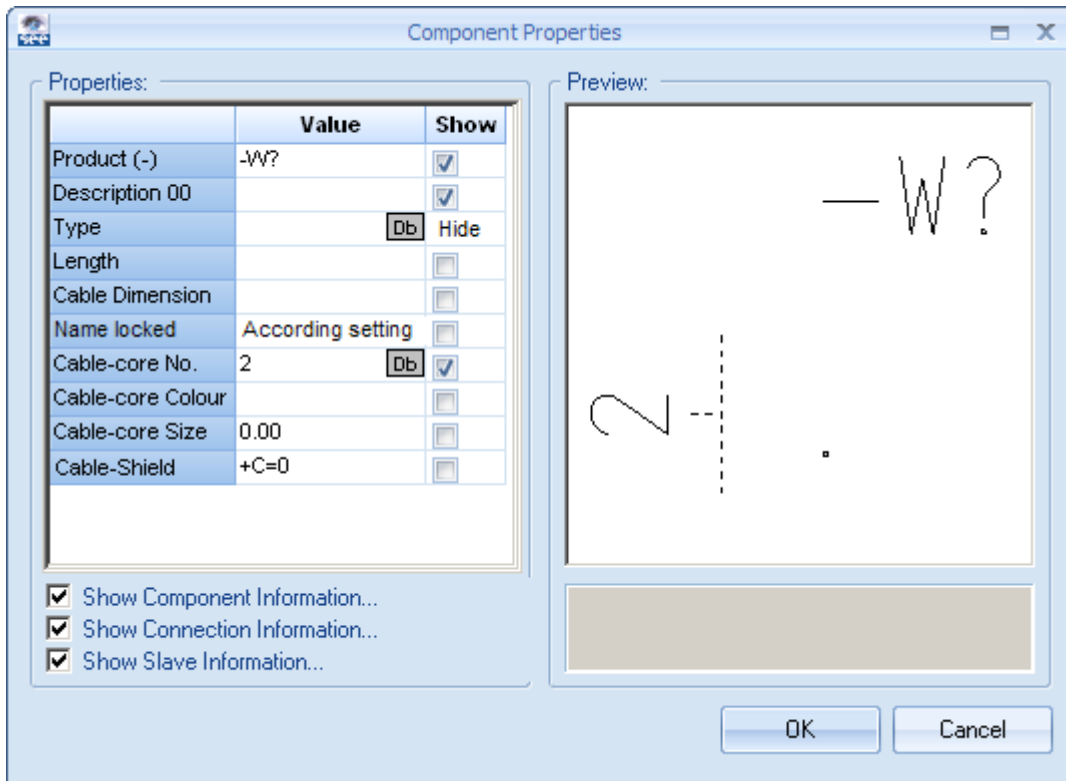
Each cable for the circuit diagram needs to contain one symbol with a component text with the "Cable-Shield" attribute. (This means, you have to add this text to your cable symbols and then redraw existing cables, if you want to use cables with shield in your cable plans.)

It is recommended that you change the cable symbol which contains the shield:



Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved



The component text "*Cable-Shield*" can contain the following keywords:

+S="Name of the symbol folder in the "Cables.SES" symbol library"

This name replaces the symbol group name in the template placeholder-text "*Cable-Shield*". It is used by all shields of this cable.

-S= Do not draw the shield in the cable plan. (The effect is the same as if this component text does not exist)

+C=<n>

where <n> gives the number of the core which is connected to the shield.

n=1: the first core is connected to the shield

n=2: the second core is connected to the shield

...

n=0: the last core is connected to the shield

2. Template for the graphical Cable plan

Add one or two placeholder texts "*Cable-Shield*" to the template of the graphical Cable plan.

These placeholders mark the position of the Cable-Shield (right and/or left).

The placeholder-text can contain the following keywords:

+S="Name of the symbol library\folder\symbol name"

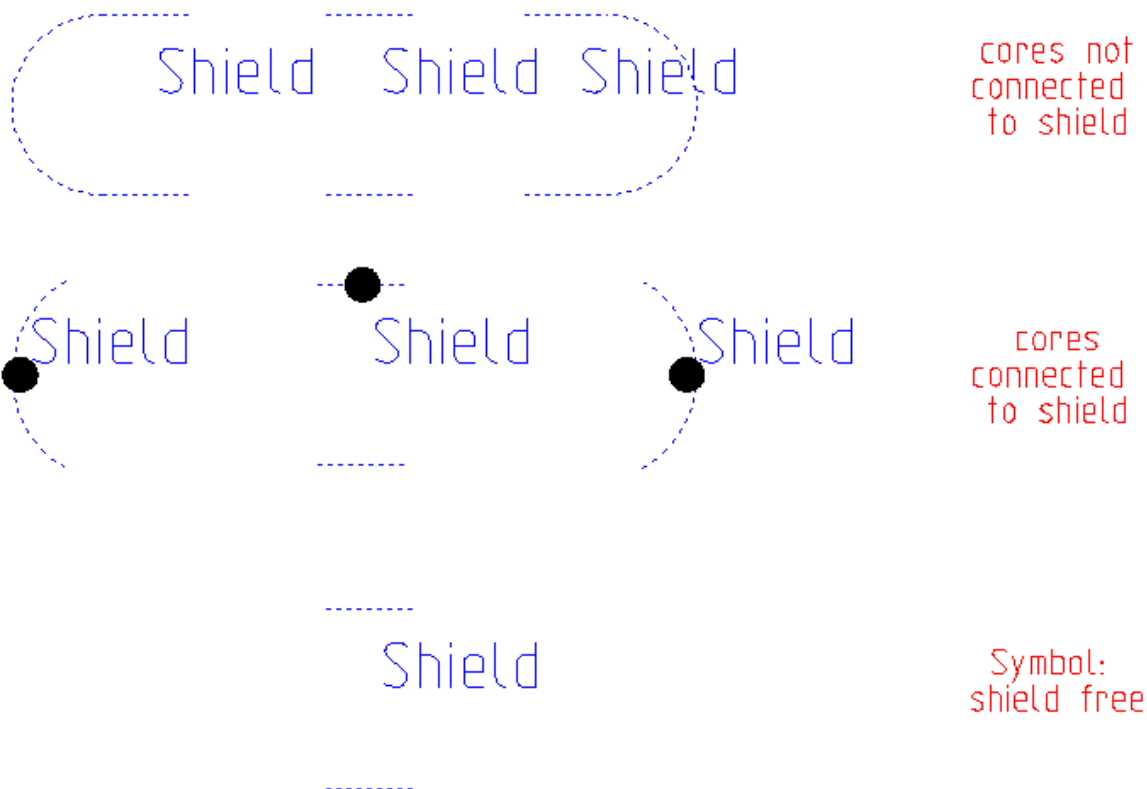
Default: +S="Cables\CablePlan\Shield"

If this placeholder exists, the shield symbols defined in the circuit diagram cable symbols are not used.

2. Generate cable-shield-symbols for the graphical Cable plan

A cable shield in the graphical *Cable plan* is constructed by symbols from a symbol group. Such a symbol group consists of 7 symbols. The default symbol group should be stored in the *CABLES.SES* library in the folder *CablePlan* like follows:

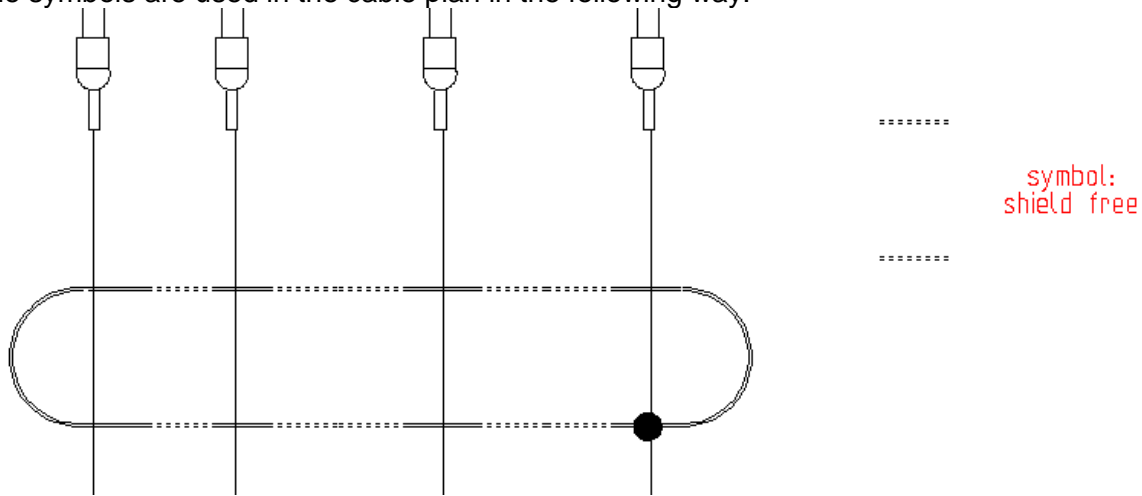
Symbol from the library	Description
Cables\CablePlan\Shield-Start	This is the start symbol if the first core is not connected to the shield.
Cables\CablePlan\Shield-StartC	This is the start symbol if the shield is connected to the first core.
Cables\CablePlan\Shield-Mid	This is the middle symbol if a core is not connected to the shield.
Cables\CablePlan\Shield-MidC	This is the middle symbol if the shield is connected to this core.
Cables\CablePlan\Shield-Free	If there is spacing between two cores in the cable plan, this symbol is used to make the shield look closed.
Cables\CablePlan\Shield-End	This is the end symbol if the last core is not connected to the shield.
Cables\CablePlan\Shield-EndC	This is the end symbol if the shield is connected to the last core.



Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

The symbols are used in the cable plan in the following way:



Each symbol consists of geometry and a placeholder-text "*Cable-Shield*", see the text "*shield*" in the picture below:



This placeholder text is necessary in order to place the symbol in the template on top of the text "*Cable-shield*". The text of the second symbol is positioned with a distance to the first one, that is given by the distance between the template-texts "*Target Left*" and "*Number of cable core lines*" (Id=180114). Each cable shield symbol has to fit into the space given by this distance.

Z WIRING LISTS

(standard)

The **WiringList** command allows you to generate a wiring list in *Excel* or *Text (ASCII)* format.

Term definitions

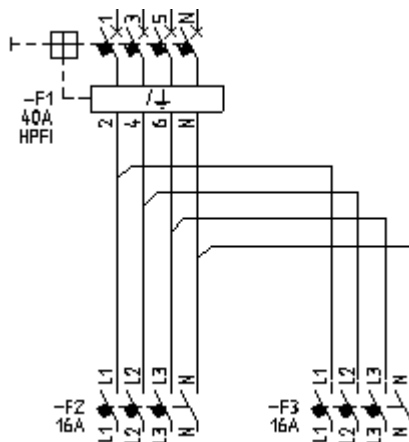
In the wiring list, the targets for each wire are inserted. The wiring list groups all targets which are on the same potential.

Each wire has properties: wire number, wire colour, wire section, wire type, signal and potential. If several wires are on the same potential, the wires are grouped with the same wire properties. In this case you can speak of a network.

If the "*Potentials first*" option is active, then in the listing of targets the potential is always first, if it exists.

The wire direction is taken into account when the wiring list is created, but no distinction is made between the start point and the end point of the wiring.

Example:



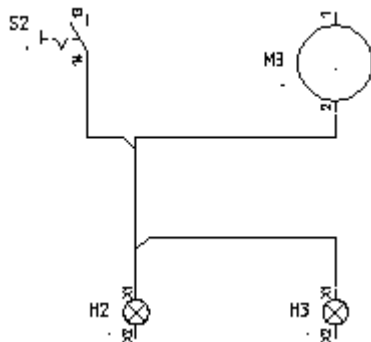
In the wiring list the elements of the first network can appear either in the order "-F1/2; -F2/L1; -F3/L1" or in the order "-F3/L1; -F2/L1; -F1/2".

In some cases two or more networks may appear, even though there is only one network.

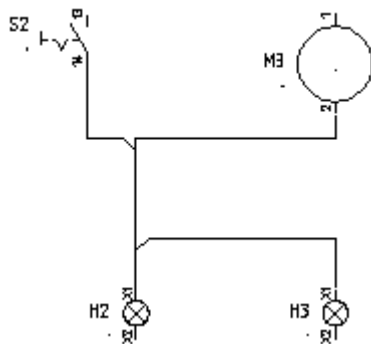
Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

Example:



The network "S2:14, H2:X1, H3:X1" appears - it cannot be continued from here. That is why the network "H2:X1, M3:2" appears in addition.



If you modify the wire direction, as shown above, you get: "S2:14, M3:2, H2:X1, H3:X1".

If you use wiring lists, you should not insert any symbols on the potentials, because the potential will appear twice as targets for the equipment.

Z.1. GRAPHICAL WIRING LIST

The graphical *Wiring list* is available in the *Graphical Lists* area of the **Workspace Explorer**.

The template contains only placeholders with the "Normal text" attribute.

For a wiring list specific criteria for sorting are valid, as described below. That is why the #OrderBy, #Where, #PageBreak and #LineBreak commands cannot be used here.

Besides the ordinary placeholder texts in the page template, the following special placeholder texts can be used:

ID	Description
#Lines 30 7.5	It specifies the maximum number of lines (30) and the distance (7.5) between them. This text can be located anywhere in the page. NOTE: You can change the number of lines and the line distance.
#180110 \$#	Consecutive number for line It is possible to add a prefix (X) or a postfix (z) to each number (\$#). If you want to do so, use the following syntax: #180110 x\$#z
#175150	Cross-section
#175151	Colour
#175152	Number (of the wire)
#175157	Signal type
#175158	Potential number
#180111	Combination of wire attributes If you want a combination of the wire attributes (number, size, colour, type, signal type or potential name) in one text, please proceed as follows: Combine the text placeholder #180111 in each combination with the placeholders: \$N = wire number, \$C = wire colour, \$S = wire size, \$T = wire type, \$P = signal type, \$Q = potential name Example: #180111 \$N \$C \$S If you want, you can use a separator (for example /). Example: #180111 \$N / \$C / \$S The attributes appear in the Wiring list in the sort order in which you define them in the template. You can add a blank between the different attributes, like shown in the example or a separator, for example a comma.
#180112	Represents the list of targets of each wire part Example: #1801112 #Sep= " / " #Len=170 #Cnt=0 #Sort="1NCS" If you use this placeholder, all targets are listed one after the other. Different parameters allow you to define the appearance. (All parameters must be in the definition.)
#Cnt=	Specifies the maximum number of targets in each line of the text (0 means ignore) You can either use #Cnt or #Len to control the length of the string.

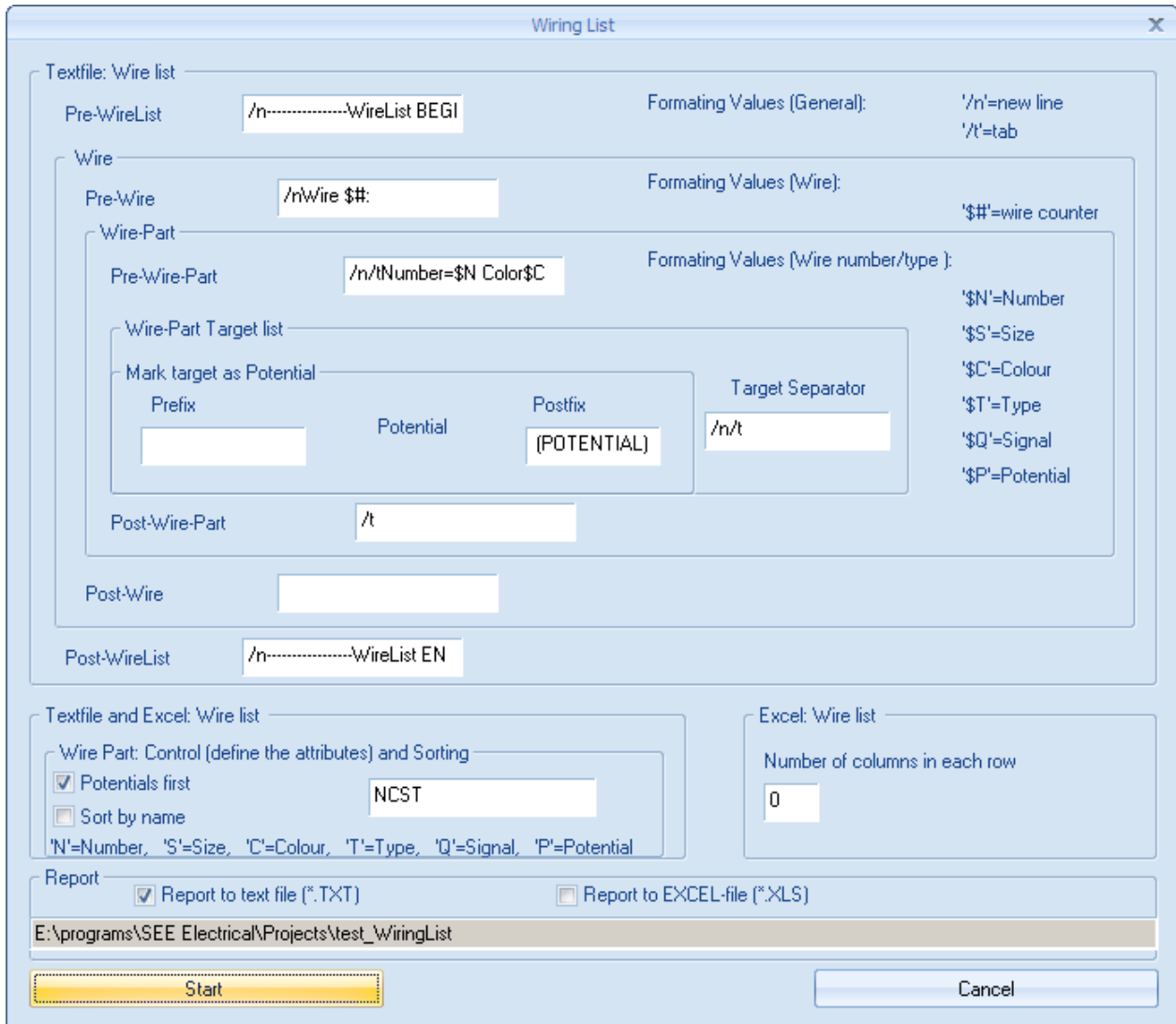
#Len=	Specifies the maximum length of each line of the text (0 means ignore) You can either use #Cnt or #Len to control the length of the string.								
#Sep=	Specifies a separator between the targets (for example /). The separator has to be inserted between inverted commas (" "). With this, you can also add blanks. Example: #Sep=" / "								
#Sort=	Controls wire part sorting. Example: #Sort="0NCS" or #Sort="1NCS" The number in that parameter specifies if potentials are set at the beginning of the wiring list or not and if the wiring direction is taken into account. The first character after "#Sort=" is "0", "1", "2", or "3" and it sets the sorting mode for targets: <table border="0"> <tr> <td>"0" disables "sort potentials first" by name" (ignore wire directions)</td> <td>+ disables "Sort targets"</td> </tr> <tr> <td>"1" enables "sort potentials first" by name" (ignore wire directions)</td> <td>+ disables "Sort targets (DEFAULT)"</td> </tr> <tr> <td>"2" disables "sort potentials first" by name" (ignore wire directions)</td> <td>+ enables "Sort targets"</td> </tr> <tr> <td>"3" enables "sort potentials first" by name" (ignore wire directions)</td> <td>+ enables "Sort targets"</td> </tr> </table>	"0" disables "sort potentials first" by name" (ignore wire directions)	+ disables "Sort targets"	"1" enables "sort potentials first" by name" (ignore wire directions)	+ disables "Sort targets (DEFAULT)"	"2" disables "sort potentials first" by name" (ignore wire directions)	+ enables "Sort targets"	"3" enables "sort potentials first" by name" (ignore wire directions)	+ enables "Sort targets"
"0" disables "sort potentials first" by name" (ignore wire directions)	+ disables "Sort targets"								
"1" enables "sort potentials first" by name" (ignore wire directions)	+ disables "Sort targets (DEFAULT)"								
"2" disables "sort potentials first" by name" (ignore wire directions)	+ enables "Sort targets"								
"3" enables "sort potentials first" by name" (ignore wire directions)	+ enables "Sort targets"								

Z.1.1. CREATING AN EXCEL/TEXT FILE

The files are created via the **WiringList** command in the **Commands** tab. The command is available only in case a circuit diagram sheet is currently open.

- Execute the command from the **Commands** explorer.

The following window appears:

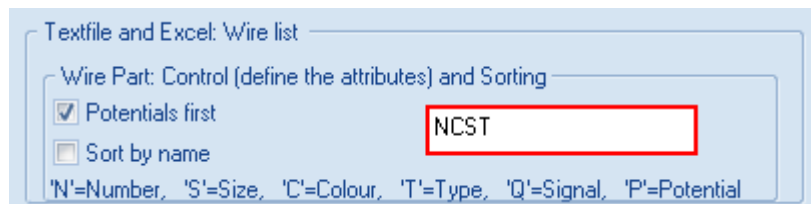


The screenshot shows the 'Wiring List' dialog box. It is divided into several sections:

- Textfile: Wire list**
 - Pre-WireList: /n-----WireList BEGI
 - Wire
 - Pre-Wire: /nWire \$#:
 - Wire-Part
 - Pre-Wire-Part: /n/tNumber=\$N Color\$C
 - Wire-Part Target list
 - Mark target as Potential
 - Prefix: (empty)
 - Potential: (empty)
 - Postfix: (POTENTIAL)
 - Target Separator: /n/t
 - Post-Wire-Part: /t
 - Post-Wire: (empty)
 - Post-WireList: /n-----WireList EN
- Formating Values (General):**
 - /n=new line
 - /t=tab
- Formating Values (Wire):**
 - \$#=wire counter
- Formating Values (Wire number/type):**
 - \$N=Number
 - \$S=Size
 - \$C=Colour
 - \$T=Type
 - \$Q=Signal
 - \$P=Potential

- Textfile and Excel: Wire list**
- Wire Part: Control (define the attributes) and Sorting
 - ☒ Potentials first (NCST)
 - ☐ Sort by name
 - 'N'=Number, 'S'=Size, 'C'=Colour, 'T'=Type, 'Q'=Signal, 'P'=Potential
- Report
 - ☒ Report to text file (*.TXT)
 - ☐ Report to EXCEL-file (*.XLS)
- Path: E:\programs\SEE Electrical\Projects\test_WiringList
- Excel: Wire list**
- Number of columns in each row: 0
- Buttons:** Start, Cancel

- Specify, in the area **Text file and Excel: Wire list**, which information is to be displayed and in what order it is to appear.



This close-up shows the 'Textfile and Excel: Wire list' section. The 'Wire Part: Control (define the attributes) and Sorting' group box contains:

- ☒ Potentials first (NCST)
- ☐ Sort by name
- 'N'=Number, 'S'=Size, 'C'=Colour, 'T'=Type, 'Q'=Signal, 'P'=Potential

In the example above, the information about wire number, wire colour, wire section and wire type will be displayed in the specified order.

- ✓ If you tick the **"Potentials first"** option, you can define if potentials should appear at the beginning of the wiring list or not.
- ✓ If you tick the **"Sort by name"** option, you can define whether the wires should be sorted by wire direction or by wire name.

Excel list

The information in an *Excel* file basically corresponds to the information in the graphical list. Column "A" shows a counter number, and the next columns display the wire properties, as defined in the **Text file and Excel: Wire list** area. The next column shows whether a signal is present. Then follow the columns with the targets.

	A	B	C	D	E	F	G
1	Counter	Wire-number	Wire-color	Wire-size	Wire-type	Potential	Target
2	001	L1	BK	1,5		L1	F1:1
3		L1	BK	1,5		L1	F2:1
4		L1	BK	1,5		L1	F3:1
5		L1	BK	1,5		L1	-Q1:1
6	002	L2	BK	1,5		L2	F1:3
7		L2	BK	1,5		L2	F2:3
8		L2	BK	1,5		L2	F3:3
9		L2	BK	1,5		L2	-Q1:3

By using the "**Number of columns in the line**" field, you can specify how many columns will be created for targets (=column "Potential", if it is present + columns for targets). If there are several destinations, when columns are available, a new line begins.

Excel: Wire list

Number of columns in each row

By default the *Excel* file is saved in the *VProjects* installation folder. It receives the name *<Workspace name.XLS>*.

Text file (ASCII file)

By default the *Excel* file is saved in the *VProjects* installation folder. It receives the name *<Workspace name_WiringList.TXT>*.

```

Example 1_WiringList.TXT - Notepad
File Edit Format View Help
-----wireList BEGIN -----
wire 001:
    Number=L1 ColorBK Size=1,5 Type=
        L1 (POTENTIAL)
        F1:1
    Number=L1 ColorBK Size=1,5 Type=
        L1 (POTENTIAL)
        F2:1
    Number=L1 ColorBK Size=1,5 Type=
        L1 (POTENTIAL)
        F3:1
    Number=L1 ColorBK Size=1,5 Type=
        L1 (POTENTIAL)
        -Q1:1

```

The following formatting options are available:

Pre-Wire list/Post-Wire list

Textfile: Wire list		Formating Values (General):	'/n'=new line '/t'=tab
Pre-WireList	/n-----WireList BEGI		

In this field, you have the possibility to define a title for the wiring list.

- ✓ /n defines the beginning of a new line
- ✓ /t defines if the values of this area should be inserted with a tabulation to the right

For the *Post-Wire* list the same rules apply. If a line is not desired, leave it empty.

Pre-Wire/Post-Wire

Wire		Formating Values (Wire):	'\$#'=wire counter
Pre-Wire	/nWire \$#:		

In this field, you have the possibility to specify a title for the wire network.

- ✓ /n defines the beginning of a new line
- ✓ /t defines if the values of this area should be inserted with a tabulation to the right
- ✓ \$# displays the order number.

For the *Post-Wire* the same rules apply. If a line is not desired, leave it empty.

Pre-Wire Part/Post-WirePart

Wire-Part	
Pre-Wire-Part	/n/tNumber=\$N Color\$C

In this field, you have the possibility to specify a title for a network part (= wires with the same wire properties).

- ✓ /n defines the beginning of a new line
- ✓ /t defines if the values of this area should be inserted with a tabulation to the right
- ✓ Various placeholders allow you to set the desired values: \$N (=wire number), \$C (= wire colour), \$S (=wire section), \$T (=type), \$Q (=signal) and \$P (=potential).

For the *Post-Wire-Part* the same rules apply. If a line is not desired, leave it empty.

Mark Target as Potential

In this field, you have the possibility to insert a prefix or a suffix which indicates potentials:

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

Example 1:

Mark target as Potential		
Prefix	Potential	Postfix
		(POTENTIAL)

Result:

L1 (POTENTIAL)
F1:1

Example 2:

Mark target as Potential		
Prefix	Potential	Postfix
(POTENTIAL)

Result:

(POTENTIAL L1)
F1:1

Target Separator

Target Separator
/n/t

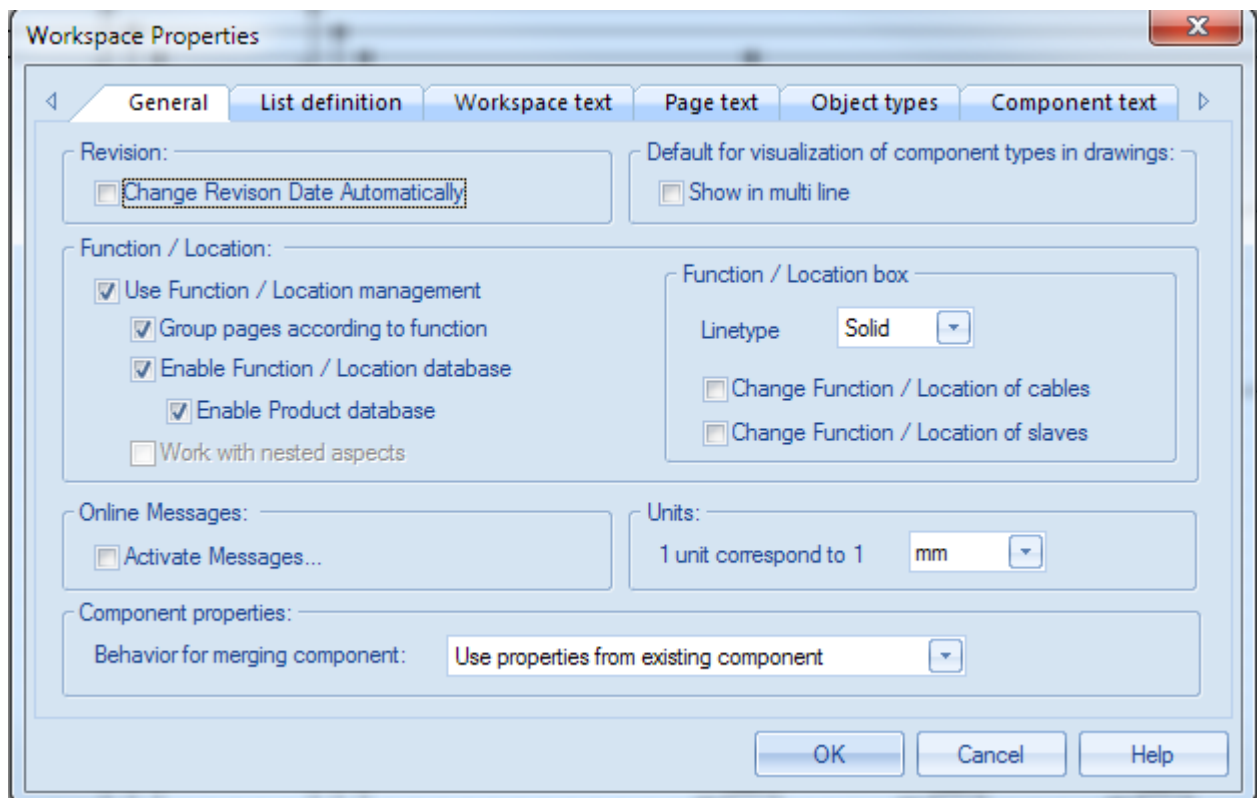
In this field, you have the possibility to define whether the targets will be displayed in a single line or not.

- ✓ /n defines the beginning of a new line
- ✓ /t defines if the values of this area should be inserted with a tabulation to the right
- ✓ If you type in a character as separator, for example a semi-colon, all the targets will appear on a single line.

AA FUNCTION AND LOCATION

(Standard)

In *SEE Electrical Standard* or *Advanced*, you can choose to work with function/location by activating the settings in the **Workspace Properties** window.



If you work with function and location, a component name consists of three parts: function (begins with =), location (begins with +) and component name (begins with -). In most cases the function and location are transferred from the legend of the standard sheet.

Exercise 25-1: Create a new workspace.

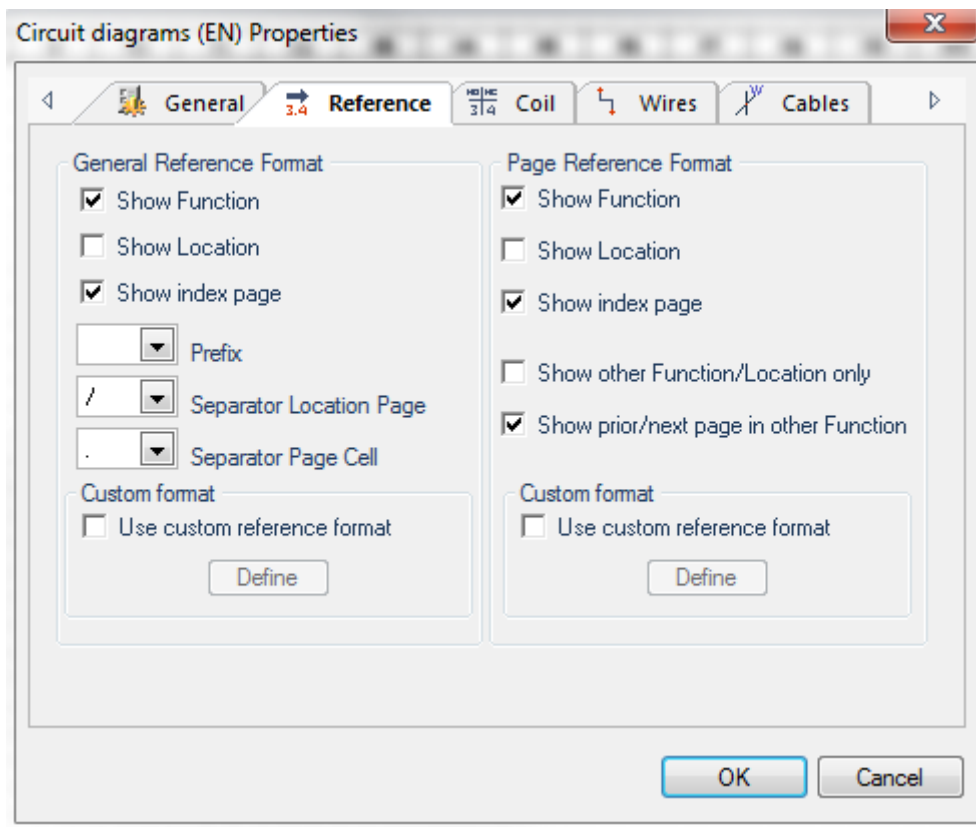
- 1.CA **File**
- 2.CO **New**
- 3.> File name
Type in a name for the new workspace.
- 4.> **Save**
- 5.> Choose the workspace template.
- 6.> Click **OK** to confirm.

Exercise 25-2: Change the properties of the workspace. You are going to work with function and location in your new workspace.

1. Click the workspace name in the **Workspace Explorer**.
2. Right-click with the mouse
- 3.CO **Properties**
The **Workspace Properties** window appears.
- 4.> Use **Function/Location** management
Tick the option. The "**Group pages according to function**" option is automatically activated. Leave it ticked.
- 5.> Choose the linetype to use for the function/location box.
- 6.> **OK**

If you have chosen to "**Group pages according to function**", you must change the settings for the generation of cross references as in this case you might have duplicated page numbers so the function must be included in cross reference too.

7. Click **Circuit diagrams** in the **Workspace Explorer**.
8. Right-click with the mouse.
9. Open the **References** tab.



10. Enable the "**Show Function**" option.
11. **OK**

Exercise 25-3: Create page 1 of the workspace and enter a function/location in the Page information dialogue box. The values appear also in the standard sheet of page 1.

1. Click *Circuit diagrams* in the Workspace Explorer.
2. Right-click with the mouse
- 3.CO **New**
Type the function and location data in the **Page information** dialogue box.
- 4.> Function
- 5.# A1
- 6.> Location
- 7.# 01
- 8.> **OK**

Create pages 2 and 3 in the same way. Enter function/location for page 2, too.

Give a different identification for page 3.

You can see in the workspace tree that the workspace has been restructured according to the function and location.

Exercise 25-4: Copy parts of the diagram of the already constructed training workspace into the new workspace.

1. Select the part of the circuit diagram you wish to copy.
2. Press CTRL + C.
3. Switch to the page in the new workspace where you wish to paste the copy.
4. Press CTRL+ V. Paste the copy.
When you work with function and location, all the component names obtain a dash "-".

Exercise 25-5: If several components must have a function/ location different from the page to which they belong, they could be assigned to another function/location using the Function/Location Box.

- 1.CA **Electrical**
- 2.CO **Function/Location Box (Function/Location panel)**
- 3.+ Fix the first point of the rectangle of the box.
- 4.+ Fix the second point of the rectangle of the box.
- 5.# B1
Fill in the new value in the *Function* field.
- 6.# 02
Fill in the new value in the *Location* field.
- 7.> **OK**
The input box closes.
- 8.> **Yes**

Confirm the renaming of all the components inside the Function/Location box.

The new identifications have been assigned to all components located within the drawn rectangle.

In the "Workspace properties" there is a setting that allows you to define if the function/location boxes will change the names of the cable cores or slaves found inside the box too.

Exercise 25-6: If the name of the Function/Location box has to be changed, proceed as follows:

- 1.+ Double-click the Function/Location box whose texts you want to change
2. Type in the correct names in the corresponding fields.
- 3.> **OK**
The input box closes.
- 4.> **Yes**
Confirm the renaming of all the components inside the Function/Location box.
All the components inside the Function/Location box are renamed.

The size of a function / location box can be changed with the help of the trackers. The size of the texts belonging to the box is not changed.

If, after a modification, a component is not located inside the box any more, it will show its function / location texts. If a component is found inside box after modification that has not been inside before, it will also show its function/location texts.

Please, disable trackers directly when you do not need them.

Exercise 25-7: If some separate components must have another function/location, they can be assigned to the function/location as follows:

- 1.+ Double-click the component whose function and location you wish to change.
2. Fill in the correct values in the appropriate fields.
- 3.> **OK**
Close the dialogue box.

Exercise 25-8: View the Products list and the Documents list.

1. Open the Graphical Lists area in the Workspace Explorer.
2. Select Documents.
- 3.M Right-click and execute the **Generate** pop-up command.
4. Double-click the 0001 page to open it.
5. Repeat the same procedure for the Products list.
The components are displayed with their different functions/ locations.

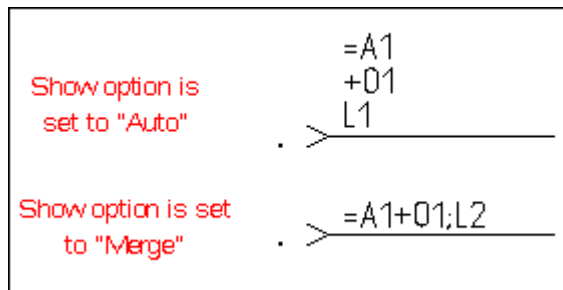
Hint

If you use function/location, it is possible to set the **Show** option for function/location in three levels by editing the component: **Auto** (the function/location is displayed only if it does not correspond to the values set for the page), **Hide** (they are never shown), **Show** (they are always shown). Furthermore, it is possible to set the **Merge** option to show the function, location and product name in one line.

Properties:		
	Value	Show
Function (=)	=A1	Auto
Location (+)	+01	Hide
Product (-)	-K6	Show
Connection 00	1	Auto
Connection 01	2	Merge

The **Merge** setting is helpful especially for potentials when their Function/Location is different from the page where they are located.

As you can see in the illustration below, setting the **Show** option to **Merge** allows displaying the texts in one line. In this way, potentials can be placed as usual with a distance of 5 mm between each other.



BB REVISION MANAGEMENT

(Standard)

The revision management system allows you to update automatically any page on which data has been modified including cases when the change is a result of an automatic *SEE Electrical* functionality. *SEE Electrical* automatically inserts the new date on the corresponding pages.

To activate this function, tick the "**Change Revision Date Automatically**" option within the **General** tab of the **Workspace Properties** window.

This feature is particularly useful in the case of relay coils and their contacts, typically represented on more than one page.

For example, you move one contact in page 12 (with the relay coil in page 7) and save the workspace afterwards. *SEE Electrical* will automatically change the revision date on page 12 as well as on page 7.

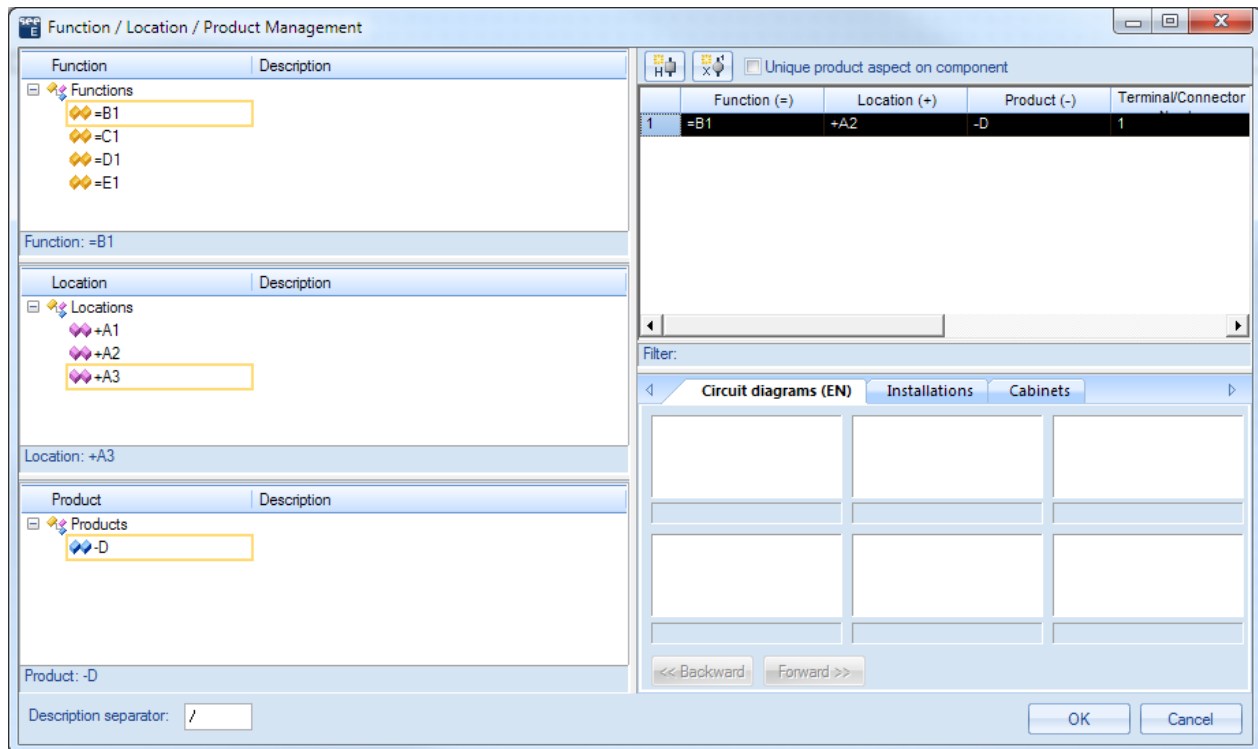
CC ADVANCED FUNCTION/LOCATION AND PRODUCT MANAGEMENT

(Advanced)

CC.1.DATABASE FOR FUNCTION/LOCATION AND PRODUCT MANAGEMENT

The **Aspects** command from the **Commands** tab allows you to define and manage functions and locations as well as the product database for the current workspace.

This command allows you to define the functions/locations and the product database for the whole workspace before you start drawing or to add them later. You can also add some descriptions, as well as display the pages and/or components which belong to a function/location.



Notes

1. When you launch the command for the first time, in case some functions/locations already exist in the workspace, they will be displayed. Otherwise, the window appears empty.
2. If you use the **Aspects** command from the **Commands** explorer without activating the settings for function/location management in the Workspace Properties window, the "Use function/location management" and "Group pages according to function/location" options will be automatically activated. In case you do now wish to group the pages according to their function/location, you need to deactivate this option manually afterwards.

Training manual

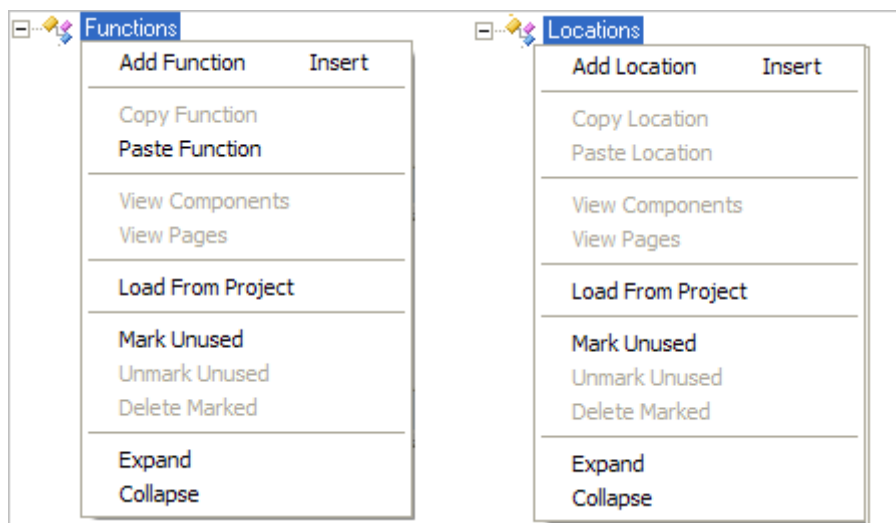
COPYRIGHT © 2013 IGE+XAO. All rights reserved

Function / Location:

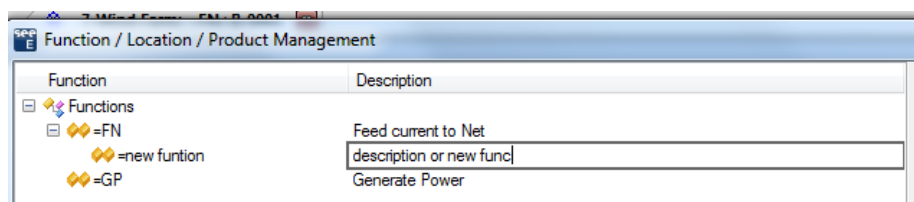
- ☒ Use Function / Location management
- ☒ Group pages according to function
- ☒ Enable Function / Location database
- ☒ Enable Product database

To add a Function\Location:

1. Right-click to display the pop-up menu for **Functions** or **Locations**.



2. Select the **Add Function** or the **Add Location** pop-up commands to create new functions/locations.
You can create functions/locations hierarchically, by selecting an existing function/location and executing the **Add Function** or **Add Location** pop-up commands.
3. Type in a name and a description (optionally) directly in the field that has been created.



To modify a Function\Location:

1. Select the function or location you wish to modify.
2. Type in the new values in the appropriate field.

To delete a Function\Location:

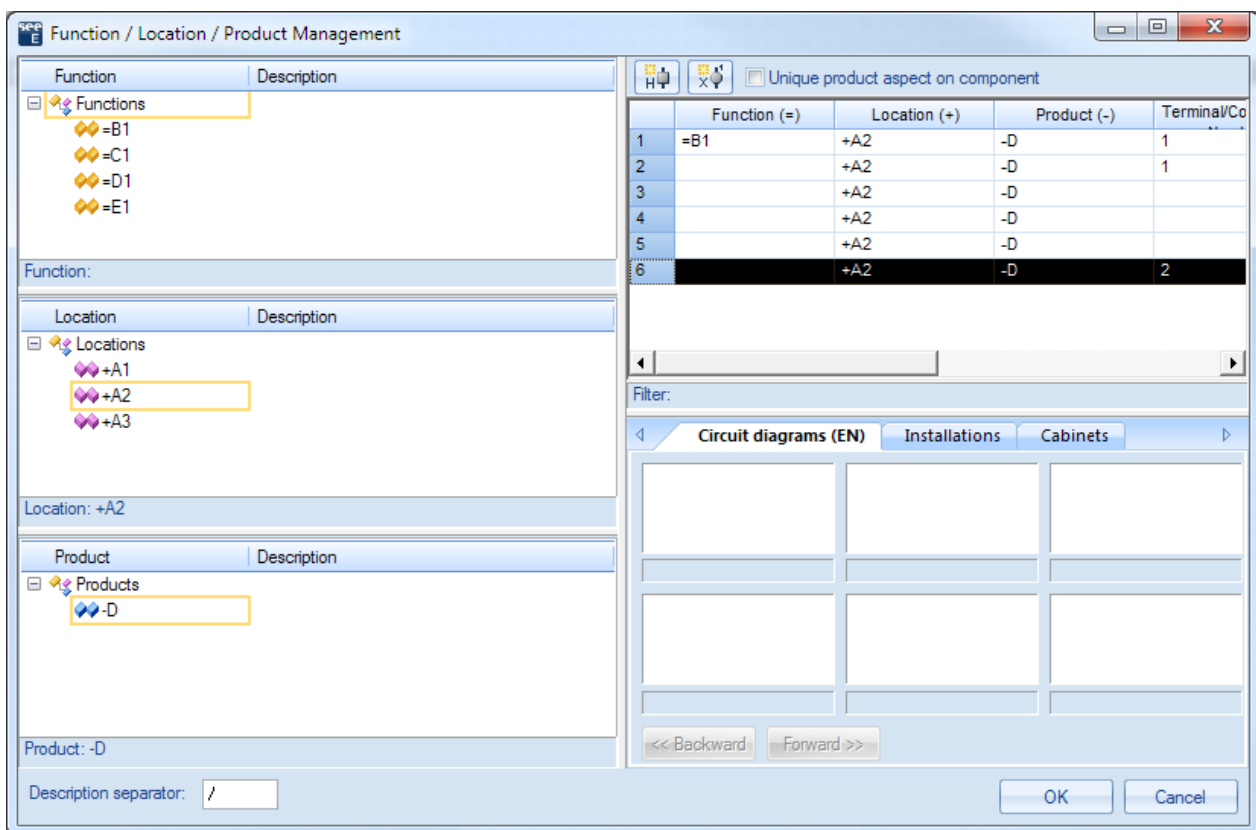
1. Select the function or location you wish to delete.
2. Right-click and select the **Delete Function** or **Delete Location** pop-up command.
3. Click **Yes** to confirm the deletion.

Note

If there are pages or components belonging to this Function/Location, a warning message appears and the deletion is cancelled.

To display components belonging to a Function\Location:

1. Select a Function or a Location.
2. Right-click and select the **View Components** pop-up command.
The components belonging to the respective function/location appear in the right pane of the window.



If you double-click a component, *SEE Electrical* opens the page where it is located.

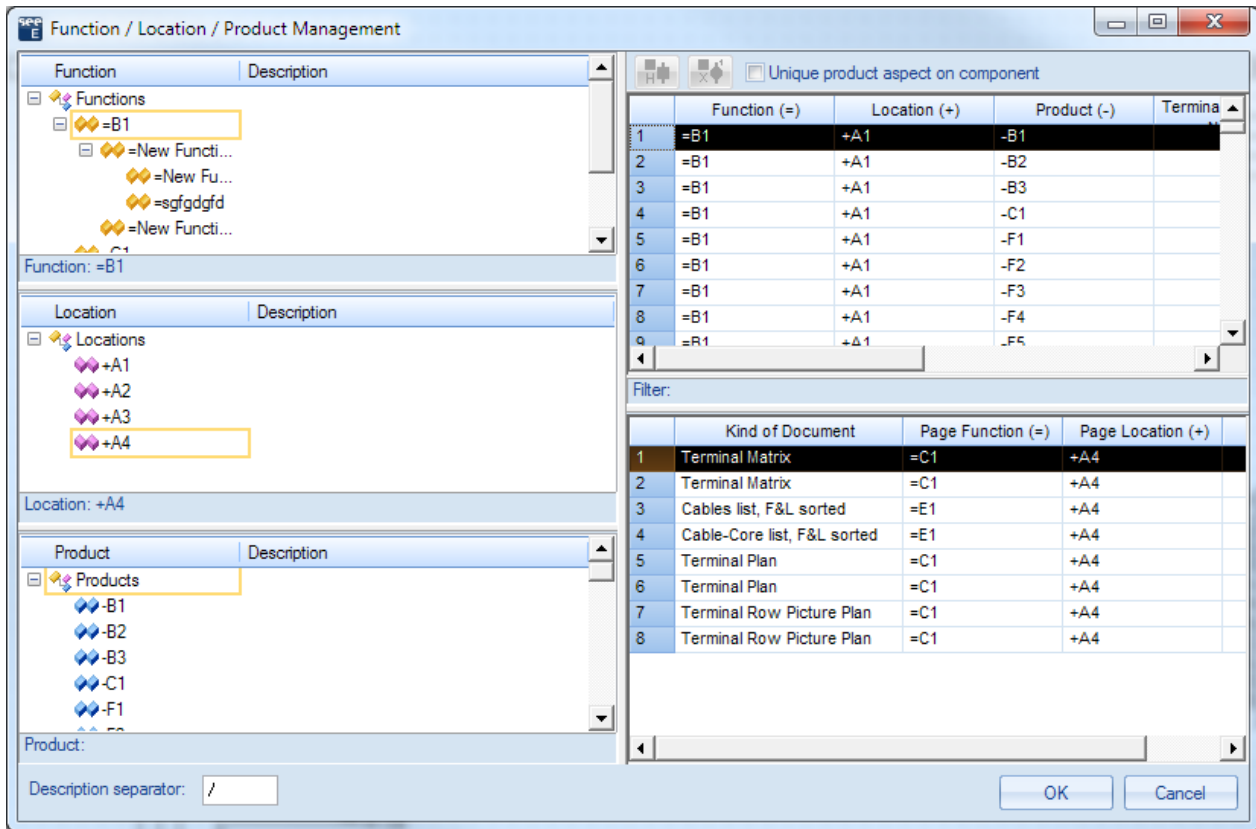
To display pages belonging to a Function\Location:

1. Select a Function or a Location.
2. Right-click and select the **View Pages** pop-up command.

The pages belonging to the respective function/location appear in the right pane of the window.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved



If you double-click on a page, *SEE Electrical* opens it.

To load Function/Location information from project:

If the Function/Location information from the project is not read properly when you open the Function / Location Manager for the first time, you can use this command to recover the missing data.

To collapse / expand the Function/Location tree:

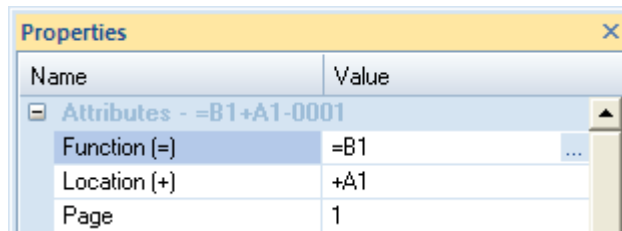
If you click a function/location and you execute the respective pop-up command, you can expand/collapse the tree structure.

To use the Function/Location Database in a circuit diagram:

In case the "**Enable Function/Location database**" option is active in the **Workspace Properties** window, you can access the Function/Location Management window and select the desired function/location for the pages and components in your workspace.

To select a function/location for a page:

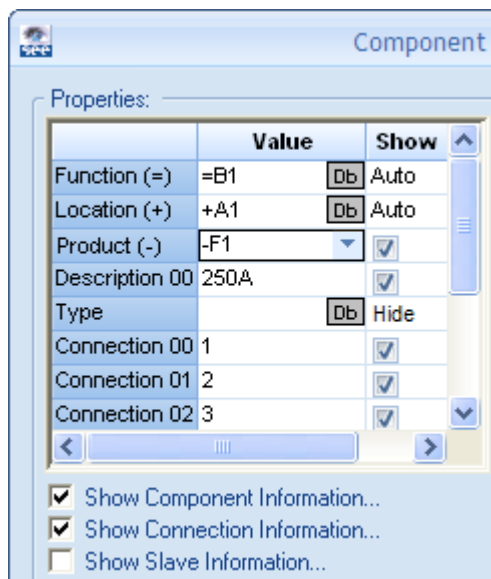
1. Execute the **Home** ➤ **Information** ➤ **Page** command and click the  button in the "**Function**" field.



The **Function/Location Management** window appears, allowing you to select a function and/or a location for the page.

To select a function/location for a component:

1. Double-click the component to open the **Component Properties** window, for example:



2. Click the button in the "Function" field.
The **Function/Location Management** window appears, allowing you to select a function and/or a location for the component.

To Export parts of the Aspects tree

You can export parts of the Aspects tree into an XML file and later import then into a new (or empty) project.

1. Select the node you want to export.
Only the information from the selected node and the structure below this node will be exported. If you want to export information for the function and location, you have to export to 2 XML files.
2. Select the **Export To XML** pop-up command.
3. Define the name of the XML file.

To import function/location information into the Aspects tree

You can import function/location information into the Aspects tree from an XML file.

1. Select the node you want to import the information into.
2. Right-click and select the **Import from XML** pop-up command.
3. Decide if you want to overwrite the existing information.

Note

The functions and locations can also be managed via the Function/Location database editor.

To add products:



If you have enabled the Product database in the **Workspace Properties** window ➤ **General**/tab, the Products area is available in the **Function/Location/Product Manager**.

1. Right-click within the **Products** area
2. Select the **Add Product** pop-up command.

You can add information about the new product in the new entry for a Product that appears. Within this line it is possible to insert a product's name and description.

If you define a component code, you can use the **Renumber** command to renumber the components later on, if necessary

To add components

The **Add Component** command is also available in the right pane of the window - . You can also assign a type to the component in the right pane. **Add Terminal/Connector** (Hotkey SHIFT + - >)- adds a terminal/connector to the selected product. This command is also available in the right pane of the window - .

After you execute the command, a window appears where you can define the number and sorting of the terminal/connector. Define how many terminals or connector pins will be generated in the "**Amount**" field. In "**Offset**" field, you can define the spacing between the values in "Terminal/Connector sorting".

To copy, paste and delete products:

Use the available pop-up commands to execute the different actions.

How to avoid inserting components with the same product aspect

The "**Unique Product aspect on component**" The functionality is available in the right part of the window only in case a circuit diagram page has been opened. It is used to avoid inserting components with the same product aspect twice.

You can use the **Mark unused** command to find which products are not used.

In case you want to use one component at two different places, disable the setting and place the component a second time.

If the functionality is enabled and a product is already present in a workspace, the **OK** button is greyed out in case you select this product (-) aspect again.

How to use the Product Aspect

When working with the Product aspect, component (and cable) names have to be assigned with the help of the *Function/Location/Product manager*. (the component attributes "*Description00*" and "*Type*" are filled from the "*Component*" definition here.) The function and location currently chosen in the *Function/Location/Product manager* when making a component in the "product" area are used in the component name.

After a component is created, a type can be added from type database.

You can also add terminals and connectors with the appropriate command.

If components are defined in the *Function/Location/Product manager* and the type used has a proper channel definition (connection texts and symbol to use in circuit diagram are defined), you can double-click the graphical symbol in the lower right area of the window and place it directly into an open diagram.

If a component contains more than one channel, all available channels are displayed in this section. The channels that have been already used are greyed out. On mouse over they show the page and cell where the symbol is located. In this case you can navigate to this page by double-clicking in case the window has been opened via the **Aspects** command.

How to change component names, when working with product database:

If you want to change the name of a component, you should select the name in the **Components** area, select the correct aspects for the component and apply them with the **Change aspect** command. (If the "**Unique product aspect on component**" option is enabled, no component with the chosen product aspect can exist. If it does, you have to delete it first.) If you use the **Change aspects** command, all appearances of the component in workspace (for example linked slaves etc.) also change their names.

If you just select another name for a component that has several appearances in the workspace, the names of the other appearances are not changed.

If you want to attach a slave to another master, you have to select the new name.

Types with channel definition - the Symbol Explorer section

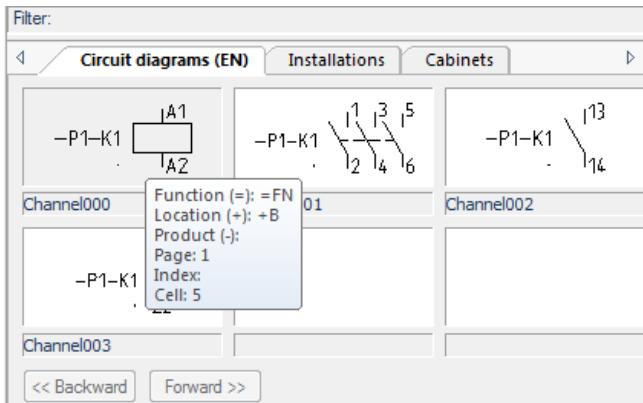
If a channel definition is properly defined for a type used on a component, in the lower right part of the aspects manager window are displayed the symbols defined in the channel.

You can double click a symbol in this symbol explorer section and place it in the current drawing.

After a symbol is placed, it is greyed-out and the page and cell it is found in are displayed on mouse over.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved



By double clicking on an existing (= greyed-out) symbol in the symbol explorer section, you will be navigated to the page where this part of the component is used.

If symbols for different modules are defined in the channel, all are found in the tab referring to the module.

CC.2. USING NESTED ASPECTS

(Advanced)

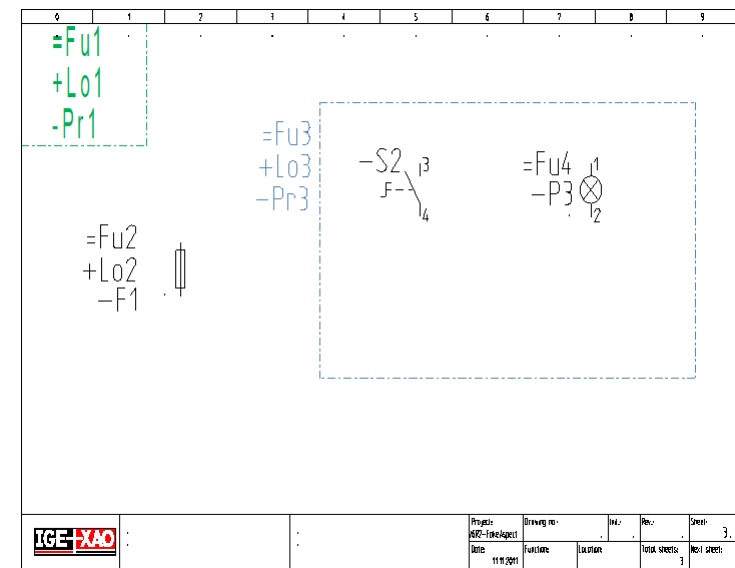
An option within the *Workspace Properties* window offers you support of relative and absolute aspects (nested aspects). The option can be enabled if no pages exist in the workspace.

First Example for the use of nested aspects

The aspects from the page are added to the aspect found on each object. The box transfers its aspects to the components it contains.

Object	Reference Designation
Page	=Fu1 / +Lo1 / -Pr1
Fuse	=Fu1=Fu2 / +Lo1+Lo2 / -Pr1-F1
Box	=Fu1=Fu3 / +Lo1+Lo3 / -Pr1-Pr3
Switch	=Fu1=Fu2 / +Lo1+Lo3 / -Pr1-Pr3-S2
Lamp	=Fu1=Fu3=Fu4 / +Lo1+Lo3 / -Pr1-Pr3-P3

In this example all aspects seen at the components or the box are sub-aspects of the aspects defined for the page - they are relative aspects.



2nd Example for the use of nested aspects

Only one aspect is used here, but the rule is valid for all three of them. A workspace contains the following structure for the function:

- = Fu1 (main function)
 - =Fu2 (sub function in Fu1)
 - =Fu3 (sub function in Fu1)
 - = Fu 4 (another main function)
- The page has assigned function =Fu1

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

The fuse -F1 is relatively referenced, this means that it is found in the sub-function =Fu2 of the main function =Fu1.

The box and the switch are also relatively referenced.

Absolute references show a ">" in front:

The fuse -F2 is placed in the main function =Fu4 which is not a sub -unction of =Fu1. So the fuse -F2 is absolutely referenced, and that is why it has the > sign in front of the function text.

The lamp is also absolutely referenced to function =Fu4.

Object Reference Designation

Page =Fu1 / +Lo1 / -Pr1

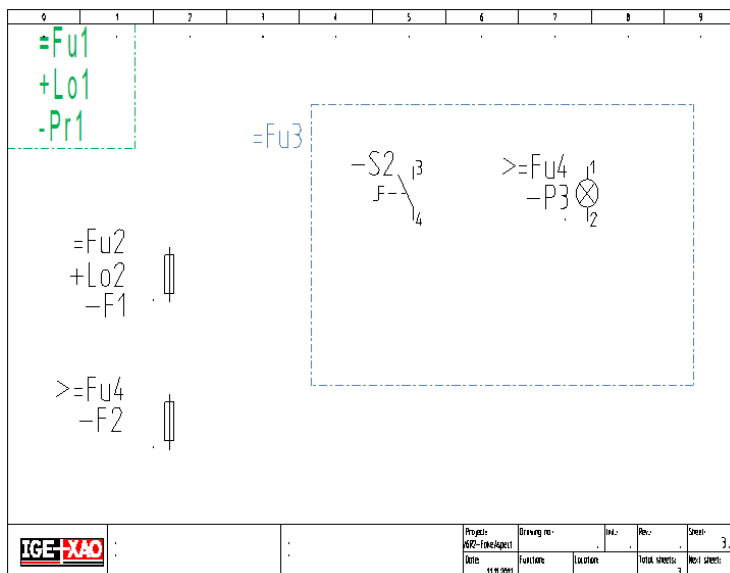
Fuse -F1 =Fu1=Fu2 / +Lo1+Lo2 / -Pr1-F1

Fuse -F2 =Fu4 / +Lo1 / -Pr1-F2

Box =Fu1=Fu3 / +Lo1 / -Pr1

Switch =Fu1=Fu3 / +Lo1 / -Pr1-S2

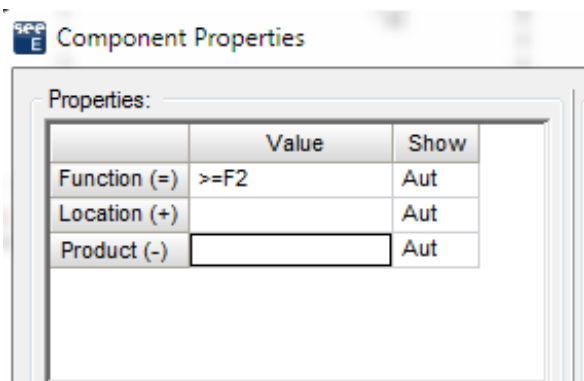
Lamp =Fu4 / +Lo1 / -Pr1-P3



How to define absolute aspects on components

First Case: The Aspect database is not used

Add the ">" sign in front of the aspect you want to define an absolute value for.



Second Case: The Aspect database is used

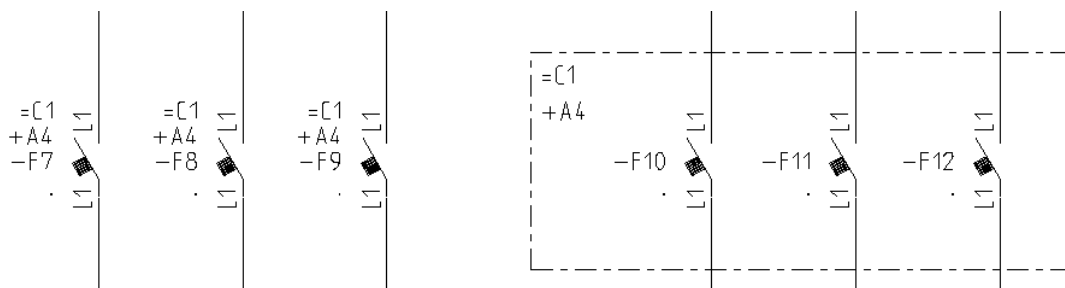
The aspects have to be defined in the **Aspects Manager**.

If the component not found inside a function / location box and an aspect is chosen that is not a child of the one defined on the current page, the > is created in front of this aspect on the component

If the component is found inside a function / location box and an aspect is chosen that is not a child of the one defined for the function/location box, the > is created in front of this aspect on the component.

How to define relative aspects for function / location boxes.

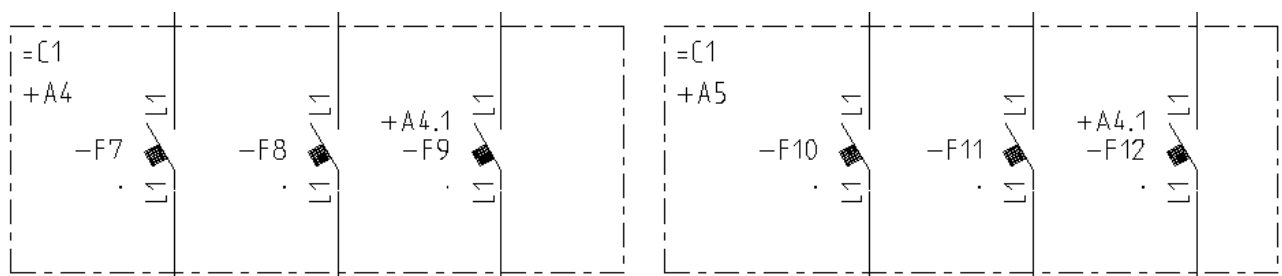
If you use a function / location box to hide the aspects already defined for components you have to select **No** when the "**Rename all components inside this function/location box?**" question appears.



If the answer is **No**, the aspects for the components are not changed - only the view of the texts is updated.

If the answer is **Yes**, the aspects from the box are added to the ones already present on the components. The $-F7$ component will be $=C1=C1+A4+A4-F7$

If the aspect defined on a box is changed and changes are applied to the components inside the box, only the affected part is changed.



The location aspect for the $-F7$ component is $+A4$, the location aspect for the $-F9$ component is $+A4+A4.1$. After location is changed from $+A4$ to $+A5$, the location aspect for the $-F7$ component is $+A5$, the one for $-F9$ is $+A5+A4.1$ so only the affected part is change.

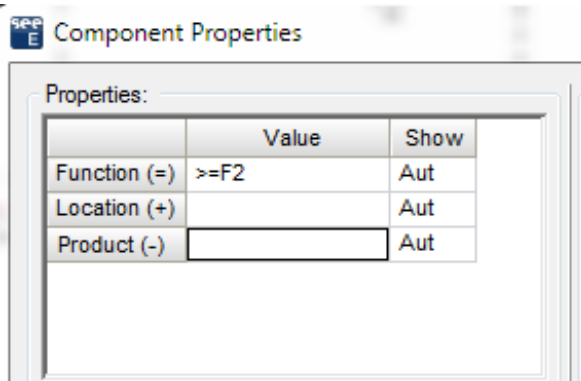
Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

How to define absolute aspects for function / location boxes

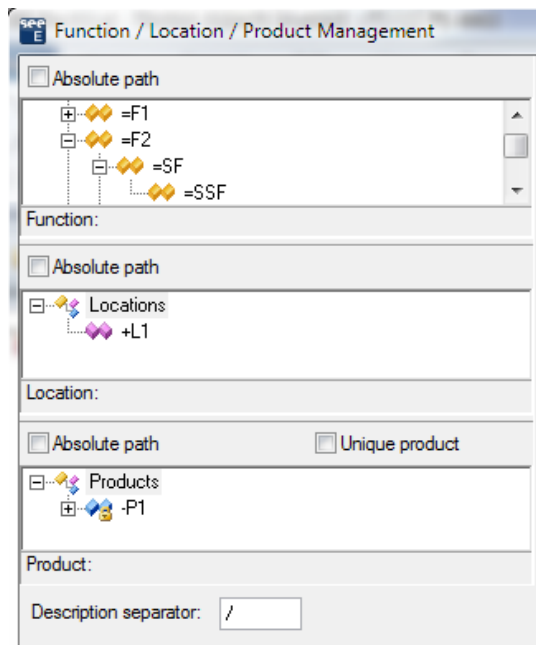
First Case: The Aspect database is not used

Add the ">" sign in front of the aspect you want to define an absolute value for.



Second Case: The Aspect database is used

The aspects have to be defined in the **Aspects Manager**. When choosing the properties for a component or function/location box, the "**Absolute path**" option is available and can be activated for the absolute aspects.



DD COMPLEX MODIFICATIONS OF THE DATABASE LISTS

(Advanced)

DD.1.1. PRODUCT EDITOR

You can edit one or several records. If a record is selected, you can change everything within this record in the right pane of the editor. If several records are selected in the list by pressing CTRL or SHIFT, you can change only description, type, and function/location, if needed. The changes apply to all of the selected components.

The **Renummer all components on all pages** function is available as a pop-up command and allows you to rename all the components on all the pages, as long as the user-defined Component Numbering setting is not activated (from *standard* level).

DD.1.2. TERMINAL EDITOR

You can edit one or more records.

The **Renummer all components on all pages** and **Renummer all shown terminals** (=filtered) functions are available as pop-up commands.

Edit multiple records:

The sequence of defining the sorting criteria is important. The criteria are shown in the headline of the window.

- a) Change the terminal names for the whole terminal strip:
 - Select all terminals in the terminal strip. Change the terminal names.
- b) Renummer all terminals in the terminal strip
 - Select all terminals in the terminal strip.

In the "**Terminal number**" field, enter the following: 1>1 (start from terminal number 1, increment 1) or

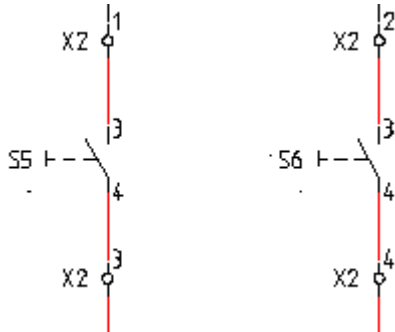
1>10 (start from terminal number 1, the next terminal has an increment 10, therefore it is 11)

This method can also be used for terminal index.

Training manual

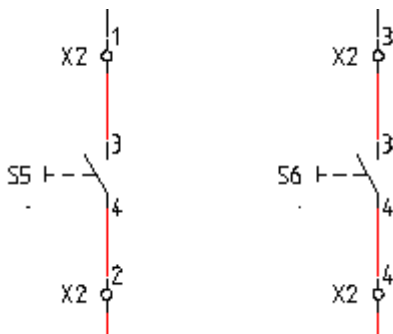
COPYRIGHT © 2013 IGE+XAO. All rights reserved

- c) Renumber the terminals in a terminal strip as follows: all the terminals on page 1 in the top line first, then all the terminals below (starting in column1) etc.



The terminal editor sorts directly so renumbering in this way is possible. Select the desired terminal strip in the "**Terminal strip**" column. Right-click with the mouse and set the filter on. Right-click and select **Renumber all shown terminals** from the pop-up menu.

- d) Renumber the terminals in a terminal strip not in the available sequence but all terminals in column 1 of page 1 first, then the terminals in column 2 etc.



- Select the desired Terminal strip in the "**Terminal strip**" column.
 - Right click and set the filter on.
 - Select the "**Page**" column.
 - Right-click and sort the column in ascending order.
 - Select the column X – it is sorted in ascending order, and then the Y column is sorted in descending order. Right-click again and click **Renumber all shown terminals** from the pop-up menu.
- e) Renumber all terminals, except terminals with number PE, etc.
- Sort by terminal strip, then by terminal numbers, select all the terminals except these with PE number etc. and change as described under b).
- f) Change the terminal type
- Select all the desired terminals and enter a type.
- g) Renumber combined terminals in the **Terminal Editor** (*advanced level*)
- In case you have multi-layer terminals in your circuit diagram, make sure to use the ">=" formula when you rename them in order to avoid omitting numbers during the renumbering.
- For example: 1>=1.

When you rename multi-layer terminals, where there is more than one terminal with the same name, number and sorting, a message appears, asking you whether to rename all components. Click the **Rename All** button to make sure all single levels of multi-layer terminals remain combined.

DD.1.3. CABLE EDITOR

Multiple records can be edited at the same time. By selecting the **Check cables** pop-up command, you can check out overfilled cable-cores.

Cables can be automatically renumbered if a component code is defined for them in the **Cables** tab of the **Circuit Diagram Properties** window and if an automatism for renumbering of components is enabled.

The renumbering is possible in the "Editor, cable" database list via the **Renumber all cables on all pages** command.

DD.1.4. CONNECTOR EDITOR

The "Editor, Connector" allows you to change the information for the connectors and their pins. Multiple records can be edited at the same time.

In the editor you can change the name and the type of the connectors as well as the pin-ID. The name of the pin cannot be changed here, it is defined in the channel definition of the type. If a type is assigned to a connector and the pin ID is changed, the pin name is updated directly from the type information.

Pin Ids or pin names can be changed only if one pin is selected in the editor.

DD.1.5. SIGNAL EDITOR

Multiple records can be edited at the same time.

DD.1.6. WIRE EDITOR

Multiple records can be edited at the same time.

It is possible to switch the visibility for the wire texts in the Wire Editor. Depends on the settings in the wire properties (**Circuit Diagram Properties** ► **Wires** tab) if the change will be executed per wire and not per net.

If you select more than one entry, the visibility checkbox for the texts is dimmed, because in this case the different wires can have different options for the visibility.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

Potential	*
Show wire number	<input checked="" type="checkbox"/>
Show potential	<input checked="" type="checkbox"/>
Show wire-size	<input checked="" type="checkbox"/>
Show wire-colour	<input checked="" type="checkbox"/>

To switch the visibility in this case just click once to turn the visibility on for all the selected wires. If you want to turn it off, click a second time. To confirm the change, click elsewhere in the editor.

DD.1.7. FUNCTION/LOCATION EDITOR

Multiple records can be edited at the same time. The position of the page in the Workspace Explorer can be adjusted.

DD.1.8. DOCUMENT EDITOR

You can select, within this editor, multiple documents from a project and delete them in a single operation by using the **Delete Selected...** pop-up command.

In the "Editor, Document" you have the possibility to change the page template for the selected pages.

	Kind of Document	Page Function (=)	Page Location (+)	Product (-)	Page	Index	Page Created Date	Page
1	Circuit diagrams (EN)				1		22-10-00	20-10
2	Circuit diagrams (EN)				2		22-10-00	20-10
3	Docu				1		03.04.2013	
4	Produ				1		03.04.2013	
5	Termin				1		03.04.2013	
6	Termin				2		03.04.2013	
7	Termin				3		03.04.2013	
8	Cable				1		03.04.2013	
9	Cable-				1		03.04.2013	
10	Wires				1		03.04.2013	
11	Wires				2		03.04.2013	
12	Wires				3		03.04.2013	
13	Wiring				1		03.04.2013	
14	Wiring				2		03.04.2013	
15	Parts				1		03.04.2013	
16	Termin				1		03.04.2013	
17	Termin				2		03.04.2013	
18	Termin				3		03.04.2013	

- Select the pages you want to change the template for.
- Right-click and select the **Change page template** pop-up command.

If you have selected only one document, a message appears asking you if you want to clear the entire page, or if you want to change only the template.

If you have selected multiple documents, a message appears asking you if you want to clear the drawing (or change the template) only on the active page, or on all of the selected pages.

If you work with IEEE circuit diagrams, make sure that the number and the position of the sections are equal in the old and in the new template.

If you select different kind of documents, a message appears informing you that it is not possible to change the template unless selecting documents of the same kind.

DD.1.9. REDLINING OBJECTS LIST

Callouts are shown in the "View, Redlining" list with the text they contain and the page where they are found.

Because of this, callouts can be used to keep an overview about the revisions made in a workspace. The "View, redlining" list gives you the possibility to navigate to the redlining in the page via the pop-up menu.

If the database list is already open and you need to refresh it, please close and reopen it.

If the database list "View, Redlining" is opened, there is a question, if the existing "Redlining list" shall be opened or if a new one must be created.

This database list can then be inserted into a page using the **List2drw** command or in the **Advanced** level, a user defined graphical list can be created. (#Where Export_4212)

EE ADVANCED PROCESSING OF A PROJECT

(Advanced)

EE.1. NAVIGATION IN THE PROJECT

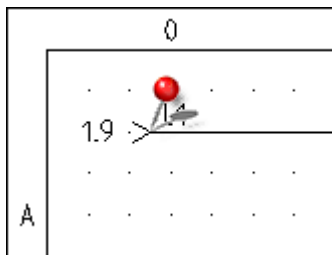
Two ways of navigation exist in *SEE Electrical*:

✓ Navigation via cross-references:

This method of navigation allows you to go from one page to another with the help of the cross-references between different components, for example potentials.

Double-click the cross-reference symbol on the potential L1 on page 1 of your training workspace.

SEE Electrical automatically opens page 2, where the source reference of the potential L1 is marked with a red pin.



✓ Navigation via the database editors

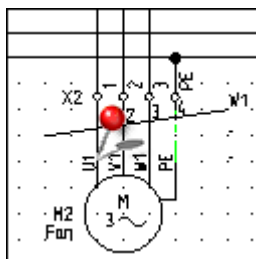
This method of navigation allows you to go from a component in the database editor to the page where it is inserted.

Open a database editor, for example the Product Editor.

Select a component, for example the motor 1M2.

Right-click and execute the **Go to the Page** pop-up command.

SEE Electrical automatically opens the page, where the component is located and it is marked with a red pin.



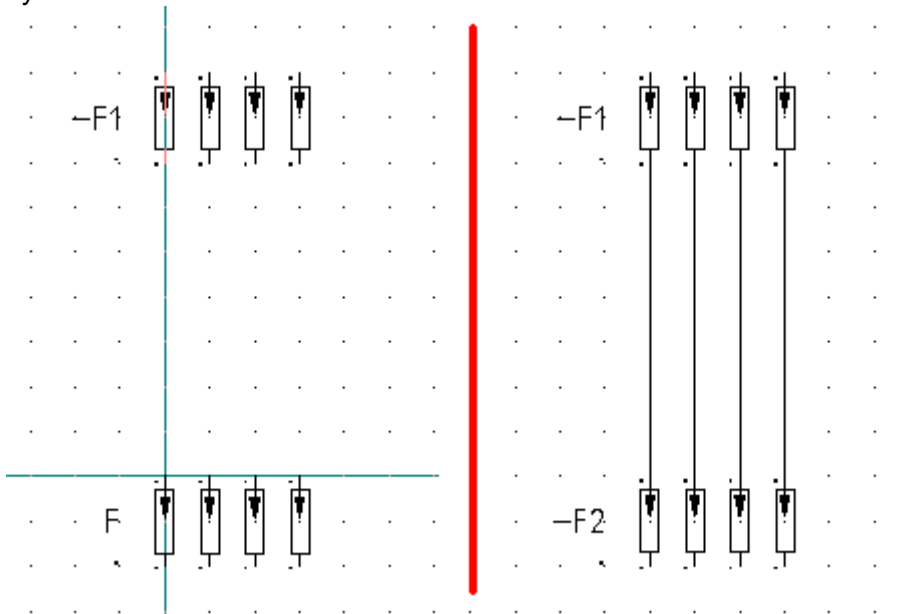
Navigation is implemented for some graphical lists such as Products list, Terminal list, PLC I/O list, Cable list, Products Assembly.

Tick the product name and select **GoTo** pop-up command.

EE.2. WORKING WITH AUTOMATICALLY GENERATED CONNECTIONS

When adding symbols you can view the relevant wires by means of the **Electrical ► Wire Connections ► Auto Connection** command.

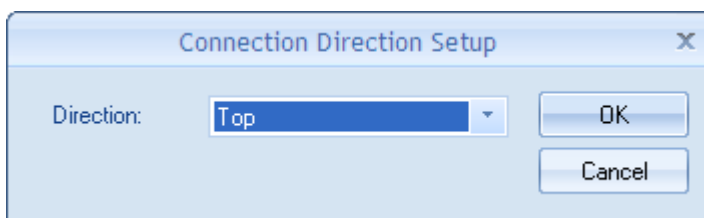
If enabled, the command automatically adds vertical/horizontal wires when new symbols are added. The example below shows screen shots (of the graphical representation) before and after adding a symbol:



When using the automatic wiring tool, you are enabled to define the route direction of the connection pin by the **Electrical ► Connections ► Direction** command.

By analyzing the shape of a given symbol, *SEE Electrical* in most cases calculates the possible directions for connecting the symbol (up, down, right, left). However, the shape of some special user-defined symbols may cause *SEE Electrical* to fail the calculation. In such case, *SEE Electrical* looks for the basic definition of directions set by the user.

The command is used to set this parameter as a default one. So, after activating it, specify the connection direction:

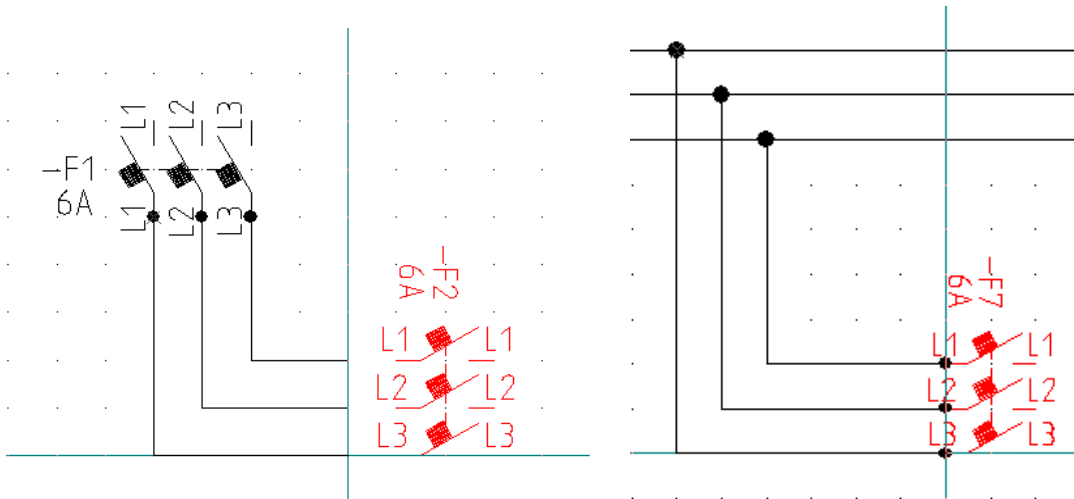


Important!

The command is only active and available when using the automatic wiring tool.

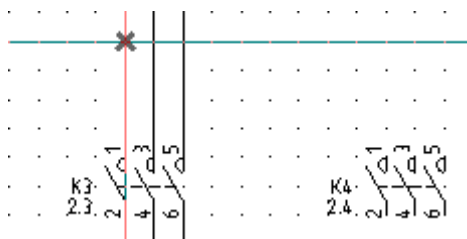
EE.3. ORTHOGONAL WIRING

The **Electrical** ➤ **Wire connections** ➤ **Orthogonal Wiring** command allows you to draw multi-mode wiring. You are allowed, optionally, to draw direct orthogonal wiring.

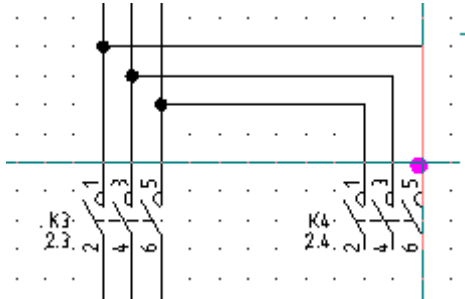


Exercise 29-1: Draw an orthogonal wiring. To do this, open page 1 of your workspace and delete all wires between the symbols which represent the motor reverse protection (columns 4 and 5). Proceed as follows:

- 1.CA **Electrical**
- 3.CO **Orthogonal Wiring (Wire Connections panel)**
- 3.+ Position the first point of the connection on the first top potential.
A wiring must always begin on an existing wire or on a component.
- 4.+ Move the cursor downwards to the motor.
- 5.+ Place the ending point of the wire on the connection point U1 of the motor.
Now continue to draw the missing wires between the second main contactor in the motor reverse protection. The **Orthogonal Wiring** command is still active.
- 6.+ Place the first point of the wire in the left vertical wire above the first main contactor.



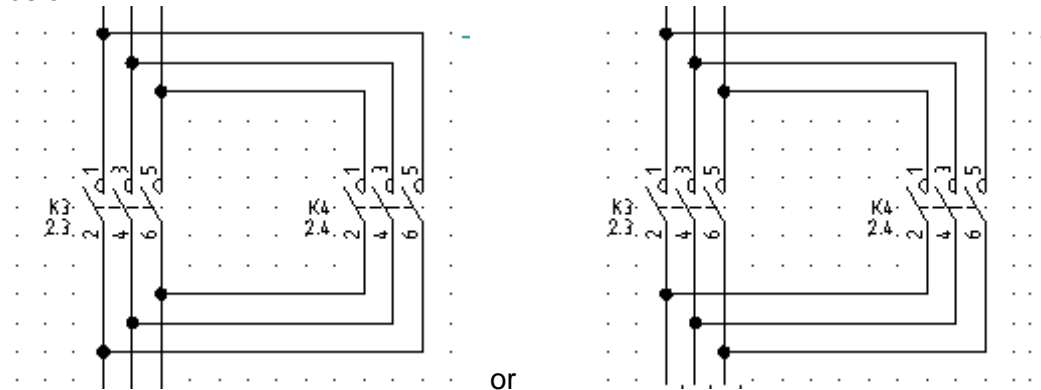
- 7.+ Move the cursor to the right until you reach the connection 5 of the second contactor.



- 8.+ Move the cursor downwards and place a corner point under connection 6 of the second contactor. Then move the cursor horizontally until you reach the left vertical connection between the first contactor and the motor.

- 9.+ Click to position the ending point.

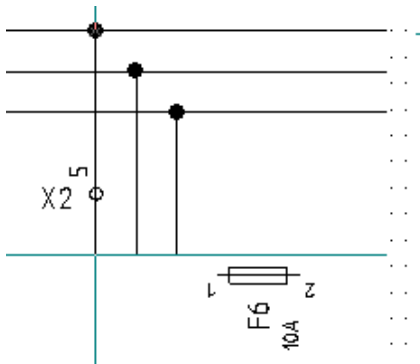
According to the cursor position, you will get the result shown on one of the two pictures below:



- 10.> Right-click to exit the orthogonal wiring mode.

Hints

1. By pressing the Tab key, you can toggle between horizontal and vertical drawing, provided that this is technically possible in the current situation.
2. By using the Shift key, you can switch on or off the orthogonal creation of wires, if this is not appropriate in the current situation.

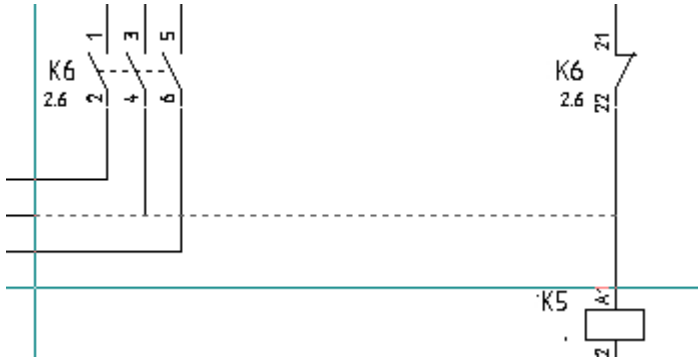


Training manual

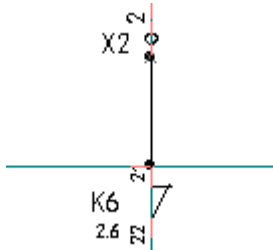
COPYRIGHT © 2013 IGE+XAO. All rights reserved

3. While drawing, SEE Electrical checks if the respective wire is possible. If this is not the case, the wire representation appears dashed.

4.



4. While you are drawing using this functionality, the connection points of the components are also visible.



EE.4. ADVANCED FUNCTIONALITY FOR CABLES

EE.4.1. DEFINE CABLE NAMES AUTOMATICALLY

It is possible to define a component code for the cables. If you have not defined a component code in the "**Component code for automatic cable numbering**" field (**Circuit Diagrams Properties** window ➤ **Cables** tab), the cable names have to be assigned manually.

If you have defined a component code, the "Component Numbering" method (**Circuit Diagrams Properties** window ➤ **General** tab) is applied for the cables.

EE.4.2. DISPLAY CABLE CORES A SECOND TIME

Select the cable you want to display for a second time, for example, after a cross reference. The **Copy Ghost** command allows you to make a copy of an existing cable core. With the **Paste** command the copy can be placed elsewhere in the diagram. The copy is only "ghost" object. It cannot be edited. If you modify the original cable core, the changes are forwarded to the ghost copy. If the original cable core is deleted, a "?" appears on the copy of the cable core. It has to be deleted manually.


EE.5. ADVANCED FUNCTIONALITY FOR POTENTIAL NUMBERING

With the **Define Signal Number** (**Electrical** category) command you can define the way of numbering of the signals for each connection. This is possible if you use the Signal type property.

If two components connected to one wire define different rules for the numbering, one is taken by random.

Make sure you do not set up a rule for the wire numbering that generates duplicated numbers.

How to define a definition:

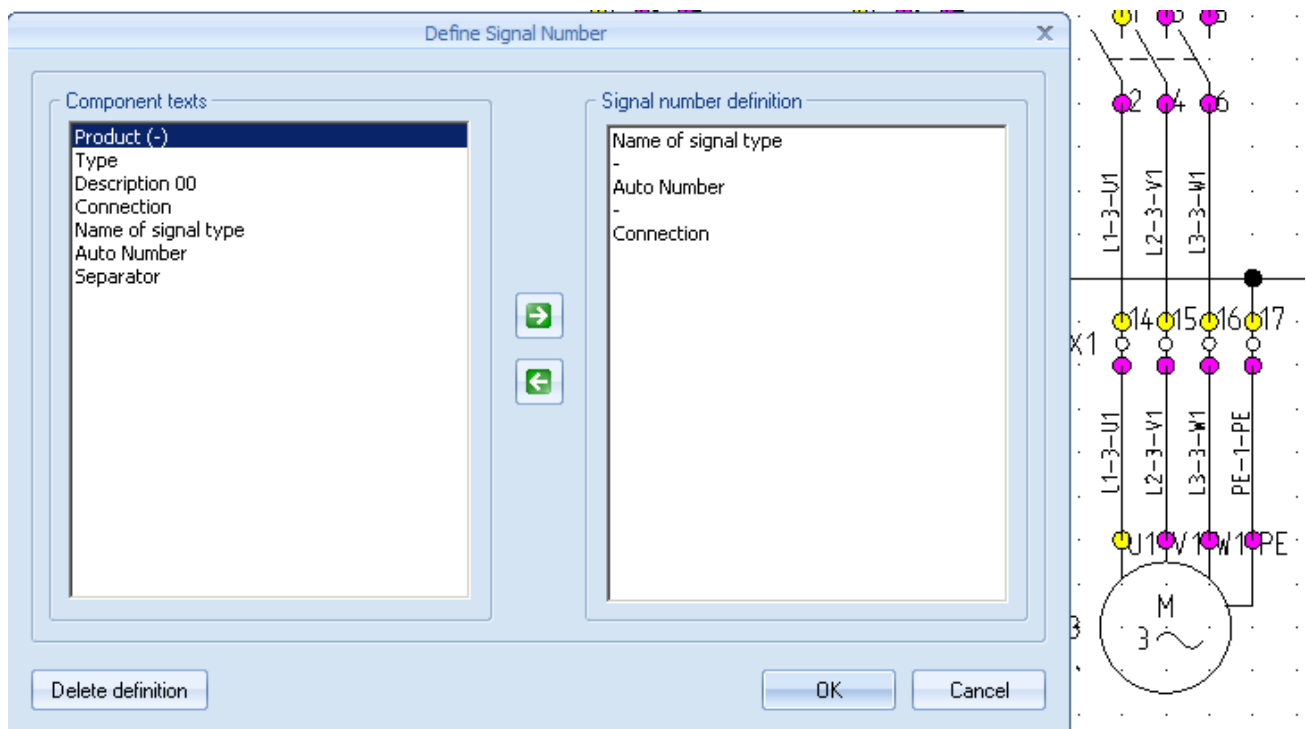
Select an entry from the **Components texts** area and move it to the **Signal number definition** area, using the  button.

If you want to remove an entry from the **Signal number definition**, use the  button.

How to remove a definition:

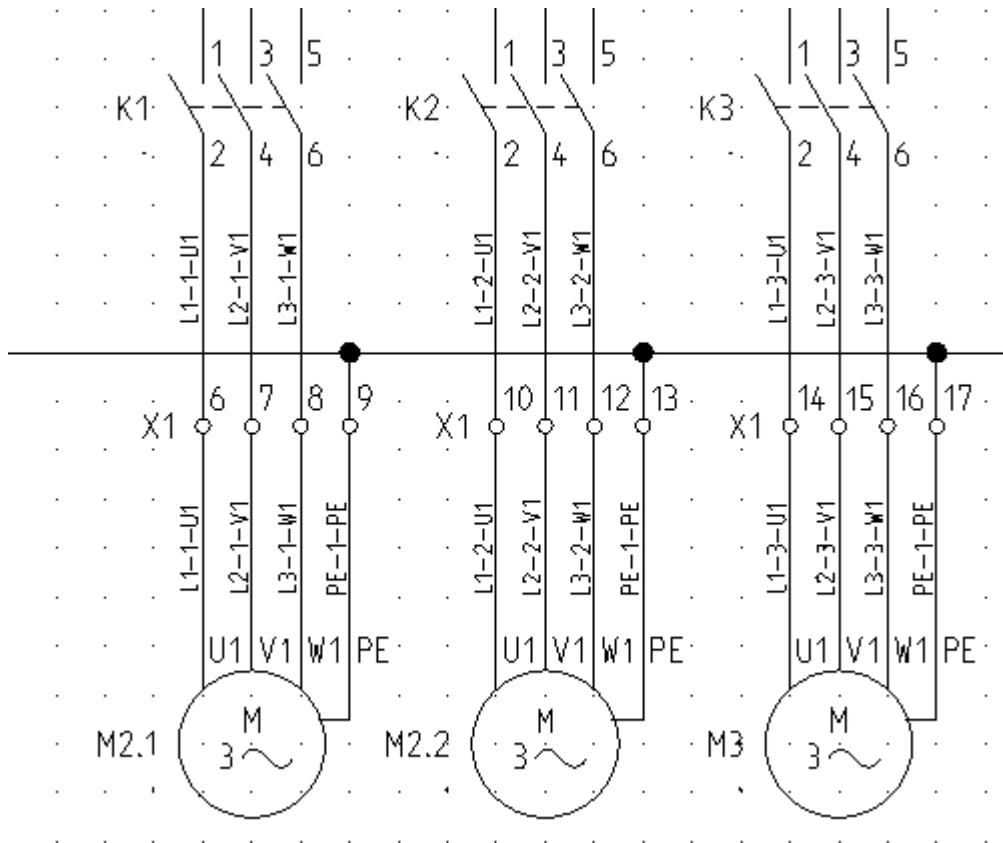
Use the **Delete definition** button.

Example of a rule:



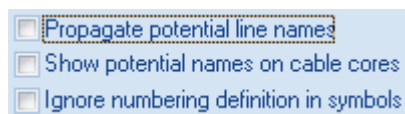
The separator can contain any text string, for example "-" or a text such as "Wire number".

Result:



The rule is stored with the symbol you defined it for.

If the "**Ignore numbering definition in symbols**" option inside the **Signal setup** dialogue (**Circuit diagrams Properties** window ► **Wires** tab), the numbering definition from the connection is ignored and is used the definition from the signal type.



EE.6. COPYING PAGES

EE.6.1. COPY SINGLE PAGE IN THE SAME WORKSPACE

If you select a page in the **Workspace Explorer** and you right-click with the mouse, a pop-up menu appears, allowing you to copy the current page. You can later paste it in the active workspace. When the command is activated, the **Page information** dialogue appears. It contains all page information texts existing in the copied page. Assign the new page number, modify any page information and click **OK**.

Component names are either automatically adapted, or you are asked to confirm their names if the "Component Numbering" is set to "Free". References are automatically updated.

The functionality is already available in the **Standard** level.

EE.6.2. COPY SINGLE PAGE IN THE SAME WORKSPACE OR IN DIFFERENT WORKSPACE

In the **Workspace Explorer**, you can copy and/or move pages from one workspace to another using **Drag & Drop**. You can also do this within one and the same workspace.

Hint

To create a copy of the page and keep the existing one, press the "CTRL" key while moving the page.

The copying of pages between workspaces is only possible, provided that the following rules are kept:

- ✓ Both workspaces must be open.
- ✓ The module from which the page is copied and the module to which it is copied must be of the same type.
- ✓ For pages in the Circuit diagrams module, if Function and Location management is enabled in the source workspace:
 - ✓ Function and Location management must also be enabled in the destination workspace;
 - or
 - ✓ If Function and Location are not enabled in the destination workspace, the page that is copied must have empty values for Function and Location.
- ✓ If the source Function is different than the destination Function, the copied page takes the value of the destination Function.

The copied pages are exact copies of the original page. This means:

- ✓ If you copy a page within the same workspace, you have to change the names of copied terminals (in the diagram or via the **Terminal Editor**), as they are duplicated to the ones on the original page.

If you copy the page to a different workspace, this may not be necessary, since the terminals may not exist there before copying.

- ✓ If you copy a page within the same workspace, you have to change the names of copied cables (in the diagram or via the **Cable Editor**), as they are duplicated to the ones on the original page.

If you copy the page to a different workspace, this may not be necessary, since the cables may not exist there before copying.

If you exchange the name of the cable on the copied page, only the cores of this cable receive the new cable name, as there is no link between the objects on the original page and their copies in the same workspace.

- ✓ For contacts and PLC I/Os, if the master is on the copied page, too, the link is still valid and the name of the master is inherited by the slave.

If the master is on a different page, you have to assign the correct name. We recommend that you display the **Component Properties** dialogue and make sure the right name is assigned.

- ✓ The names of components are treated as defined by the selected component numbering method. This means, if "Free" is chosen for component numbering, you have to change the names, otherwise they are changed automatically.

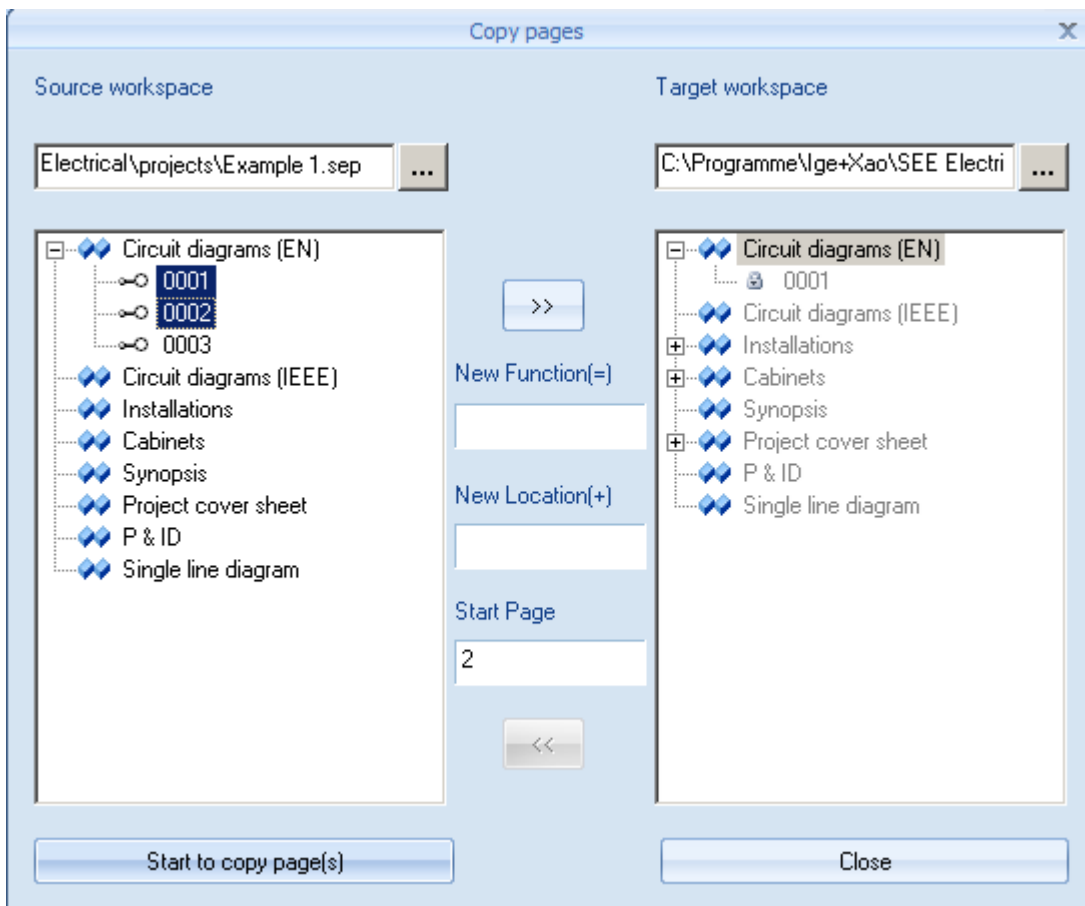
Note :


A message box appears informing you that only saved information will be copied. It also shows you the number the page will receive after copying. You can later change that number, if desired.


EE.6.3. COPY MULTIPLE PAGES BETWEEN DIFFERENT WORKSPACES

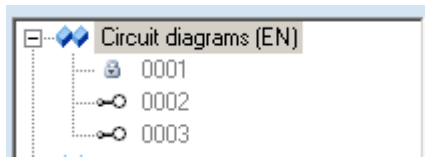
In the **Commands** explorer, the **CopyP** command allows you to copy pages from one workspace to another.


The target workspace has to be closed before executing the command.



1. Execute the command.
2. Define the source workspace.
3. Define the target workspace. The pages that exist in the target workspace are marked with a lock  0001.
4. Insert a value in the "**Start Page**" field and, if necessary, information in the "**New Function (=)**" and "**New Location (+)**" fields.
You have to make sure that the pages you want to copy do not exist in the target workspace. In this case, an error message appears.
The selected pages will be inserted in a temporary target workspace, the number of the first new page will be the one you defined, the numbers of the other new pages will be consecutive to the one you defined for the first one.
5. Open the list of pages in the source workspace and mark the pages you want to copy.

6. With the  button move the selected page to the target workspace. .



If you make a mistake, you can mark the pages in the temporary target workspace and remove them by pressing the  button.

7. If you want to copy more pages, insert the new *Start page 2* again, select the pages to be copied and move them to the target workspace..
8. Use the **Start to copy page(s)** button to copy the pages to the target workspace.
9. Close the **Copy pages** window and open the target workspace.

If in the pages exist terminal strips or cables, their names will be changed. For example, from X1 and W2 to X1#0001 or W2#0001 to avoid duplicate names. You have to rename them manually or in the database editors.

The names of the slaves that are on the same page as their master are changed according the change of the name of the master. If the slaves are located on a different page, you have to control their names.

EE.7. CHANGING PAGE TEMPLATES

It is possible to change page templates for all circuit diagrams at the same time or for a single page: The **Functions** ➤ **Change Page template on all pages...** command allows changing the page template in all of the circuit diagrams in a single operation.

In the Document Editor you can select the pages for which you want to change the template and execute the **Change Page template on all pages** pop-up command.

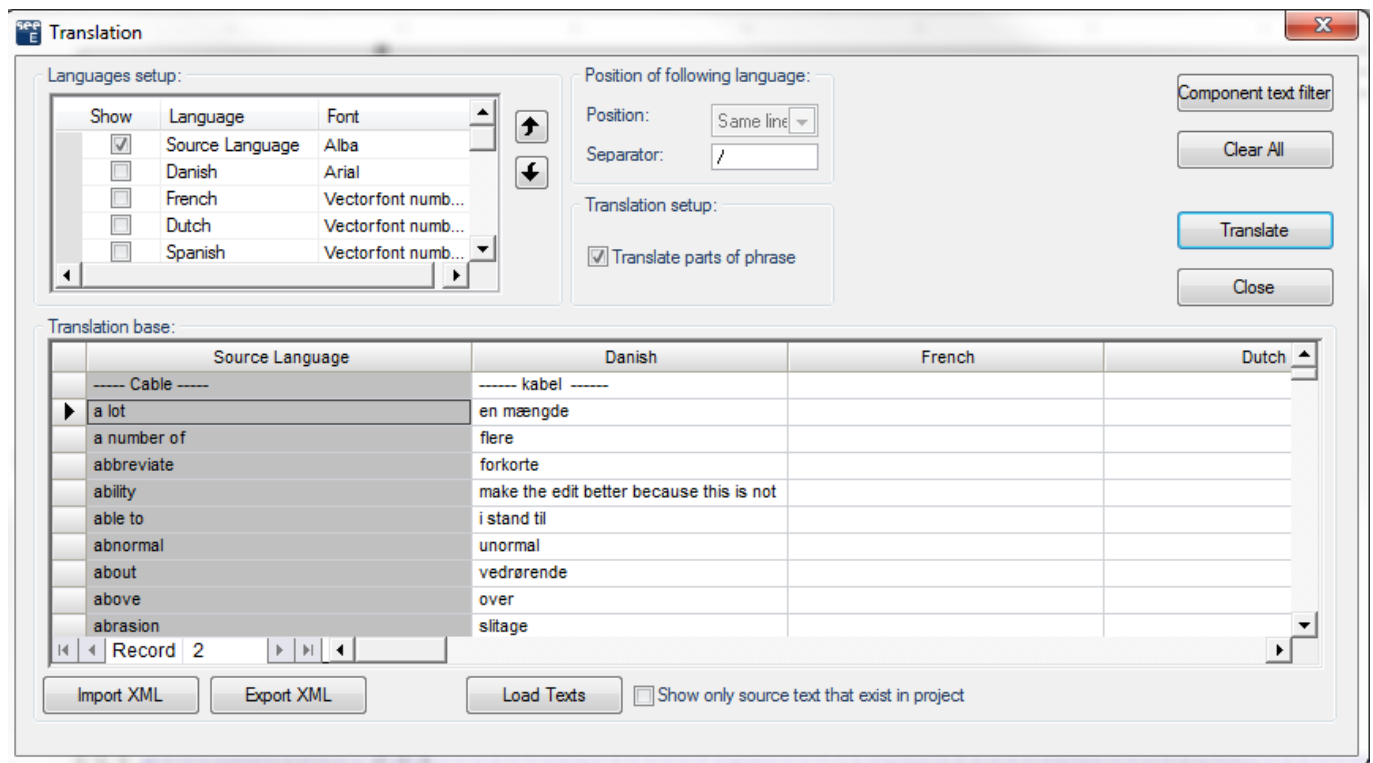
The **File** ➤ **Open** ➤ **Page Template** command allows changing the page template for the current page. You can choose whether other objects to remain on the page or not.

In both cases, the page templates settings have an effect on the components, i.e. the component names may be customized.

EE.8. TRANSLATING A WORKSPACE

(Advanced)


The **Functions** ➤ **Other** ➤ **Translate** command allows you to translate all texts contained in the circuit diagrams at the same time (consequently – all texts in the existing graphical lists, too).

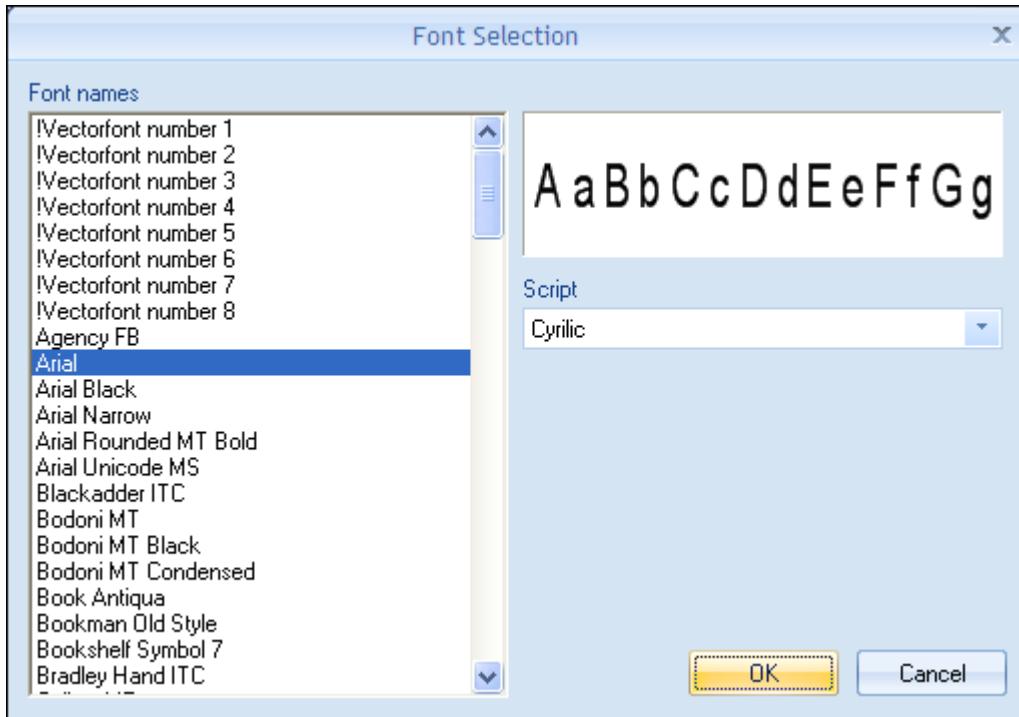


The source texts and the translated texts are stored in the *TRANSLATIONNEW.MDB* file (in the ... \SYMBOLS installation folder). The translation database can also be edited using *MS Access* or you can export it to XML, send the XML file to a translation office and reimport the XML file again.

You can also add your own texts to the translation database.

- ✓ If you wish to create a new line:
 - Select an existing line, right-click and choose the **Add New line** pop-up command.
- ✓ If you wish to delete entries:
 - Select the entry you wish to delete, right-click and execute the **Delete line** pop-up command. Please note that it is only possible to delete single lines.
- ✓ If you wish to create a second line in the entry:
 - Press CTRL and ENTER and continue with the input on the second line in the entry.

- ✓ To display a translation with a different font proceed as follows: after clicking the  button in the "**Font**" column for a language, you can select the specific font for your translations. If you are translating to languages that are not inside the range of the codepage you normally use, here you can change the settings for the font. In Western Europe, the script is Western, for Russian you have to use the Cyrillic script. Scripts are only available for Windows fonts, for example for "Arial".



The **Load texts** button allows you to load texts from the current workspace into the translation database.

The **Show only source texts that exist in project** button allows you to concentrate on the texts currently necessary.

Clicking on the **Component text filter** button opens a window in which you can set the texts to be translated (or not) selecting them by their IDs.

Attention!

The **Edit > Text > Edit Text** command shows texts always in the source language. If the translatable text must be changed, change the source text if it is incorrect, or change the text in the target language in the translation database. Then run the translation process again.

EE.8.1. DISPLAYING SEVERAL TRANSLATIONS AT THE SAME TIME

(all levels and Viewer)

It is possible:

- ✓ To show different translations in one line/in different lines:
If you wish to display them in one line, you can choose a separator in the respective field.
Languages can be moved to the desired position in the *Languages setup* area of the **Translation** window or via the **View** ► **Other** ► **Language** command.

EE.8.2. TRANSLATION OF PART STRINGS

It is possible to translate part strings. The software searches the part strings in the following way:
First is searched the whole text to be translated. If a match has been found, the text is translated.
If the text is not found in the translation database, the software searches for a part string of the text.
The software searches first for the part string that has the largest number of characters. If this part string is not found in the translation database, the next one in the text is taken into consideration, etc.

Examples:

<u>Source text</u>	<u>French translation</u>
Motor	Moteur
Motor control	Contrôle du moteur

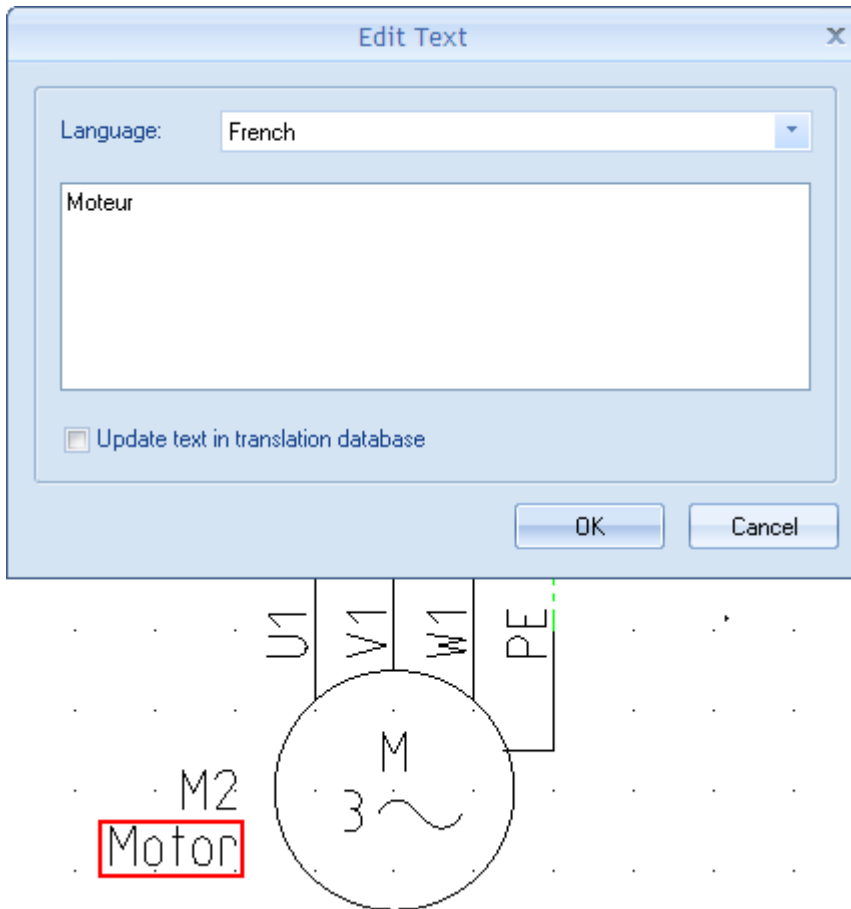
<u>Diagram: Source text</u>	<u>French translation</u>
Motor 1	Moteur 1
Motor 2	Moteur 2
Motor control	Contrôle du moteur
Motor control circuit	Contrôle du moteur circuit

EE.8.3. UPDATING TRANSLATION TEXT IN A SHEET

(Advanced)

Generally, when you modify an already translated text, you can only change the text in the source language. In the example below the text "Motor" has been translated from English to French – Moteur.

The **Functions** ► **Other** ► **Update Translated Text** command allows you to directly edit the translation text.



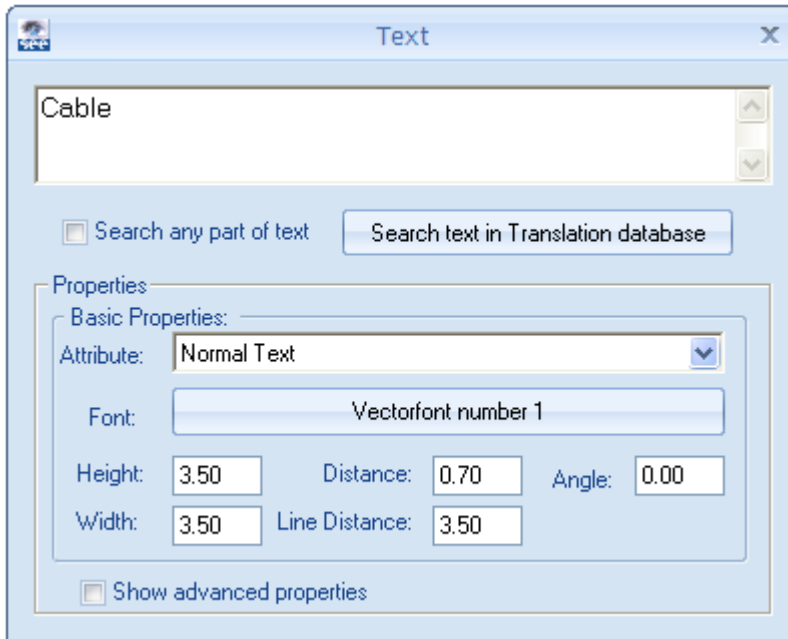
Exercise 29-2: Update a translated text in your workspace.

- 1.CO **Functions ► Other ► Update Translated Text**
2. Select the text whose translation you wish to update.
3. Select the language from the "**Language**" pull-down list in the dialogue that appears.
4. Fill in the new text.
An additional option allows you to modify or add a text into the translation database.
- 5.> **OK**
The modified translation is inserted into the translation database.

EE.8.4. LOOKING UP TEXTS IN THE TRANSLATION DATABASE

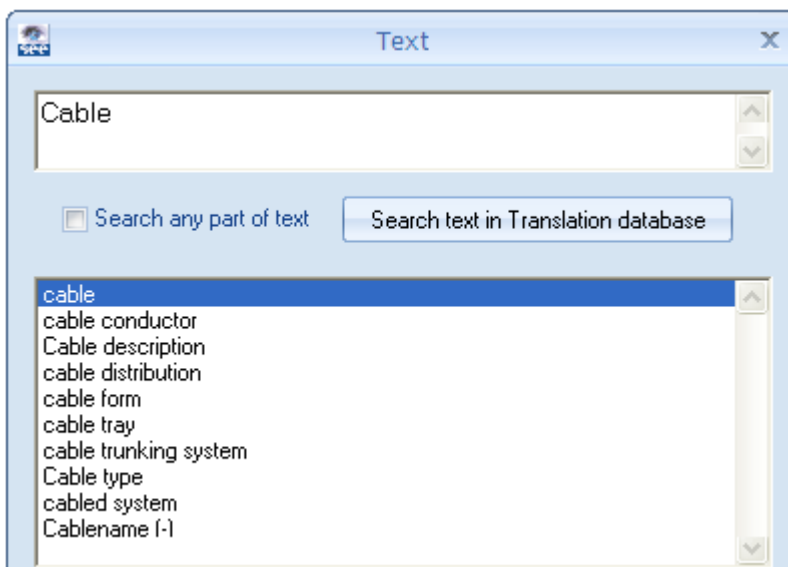
If you use a text frequently, it must be written at all the places in the same way.

When you use the **Draw > Elements > New** or **Edit > Text > Edit Text** commands, you can look up in the translation database while entering the text if the text is available and how it was written exactly. In this way you can avoid different spellings and reduce errors.



Type the text and click the **Search text in Translation database** button. All the texts from the translation database ("**Source language**" column) beginning with this text are displayed and can be selected by double-clicking with the mouse.

If you activate **Search any part of text**, the whole phrase is searched in the translation database. For the text "Cable", the "cable channel" term will also be found.



EE.8.5. EXCHANGING THE SOURCE LANGUAGES OF A WORKSPACE

(Advanced)

Through the **TransText2SourceText** and **TransText2SourceTextWsp** commands, available in the **Commands** pane you can transfer one existing translation language as a source language in a single page or a workspace.

The **SwapSourceLanguageInTranslationDatabase** command gives you the possibility to define as a source language each language available in the current translation database (*TranslationNew.MDB* in the *Templates* folder of your *SEE Electrical* installation).

If the new source language contains empty or duplicated entries, there will be a loss of data, because the entry in the source language column cannot be empty; if one term is found twice in the new source language, the second entry will overwrite the first one or it will be refused.

Make sure to avoid both situations before you activate the command.

Within the translation database you can manage twenty one languages including the source language.

Exercise 29-1: Swap the languages in the translation database.





Make a backup copy of your existing translation database (*TranslationNew.MDB* in the *Templates* folder of your *SEE Electrical* installation).

- 1.CO **SwapSourceLanguageInTranslationDatabase**
2. Select the language that shall be the new source language in the "**Select language to be Source Language**" field.
3. > Type the language name of the new source language.
 Make sure to use a name not already used as a language column. If you have to use an already existing name, it will be used only in case it does not contain entries. If it contains entries, an error message will appear and you will have to delete the entries before you could use the language.
 If you use the name of an existing column, a new column ("**Language New**") will be generated in order to keep the amount of twenty one languages in the language database. In case you have duplicated or empty entries in the new source language, a list of these entries appears. A *SwapTranslationSourceLanguage.log* file is created in the *..\Templates* folder that contains the list of entries.


EE.9. CHANGING THE FONTS AND ATTRIBUTES FOR ALL TEXTS IN THE WORKSPACE, SYMBOL LIBRARIES OR PAGE TEMPLATES





You can change the fonts in the workspaces, page templates and symbol libraries. All tools are located in the **Commands** pane of the software. It is possible to change the fonts for a single project or for several project, symbol libraries and templates.

Exercise 29-3: Finding and changing all fonts used in a single workspace.




- 1.CO **FontToolReadSingleProject**
2. Define the project to scan for fonts - the current project is suggested by default.
3. Press the  button to choose another project.
- 4.> Define the XML file to write the font settings into in the "**Set XML file to export text**" field.
5. Press the  button to open and load the desired folder and then type name of XML file.
6. Close the folder window with the help of the **Open** button.
If you select an existing XML file, a question appears asking you if you want to add the results of the new scan to this XML file or if you want to start again. If you add the new results to the existing file, you can benefit from the mappings already done and you will be sure that you use the same definitions for all your data.
7. Press the **Scan project for texts** button to start the scanning process.
- 8.CO **FontToolMapFont**
9. Choose the first XML file with the results from the scanning of your data.
10. Change the settings for "**New Font Name**", "**New Font Height**", "**New Font Width**" and "**New Font Distance**" for each of the different fonts and the combinations of text attributes.
In case you need to use a different code page than the default one, it is possible to choose the appropriate script when defining the font. The script you defined appears in the "**New Font Script**" field but cannot be changed
11. Save your changes with the help of the **Set font mapping** button.
12. Close the window with **Cancel**.
- 13.CO **FontToolChangeSingleProject**
14. Define the project to replace the fonts in - the current project is suggested by default.
15. Press the  button to choose another project.
16. Choose the XML file to take the mapping information from in the "**Set XML file to export text**" field.
17. Press the  button to open and load the desired folder and then select the XML file.
18. Close the folder window with the help of the **Open** button.
19. Press the **Replace projects text** button to start the replacing process.

Exercise 29-4: Finding and changing all fonts used in several workspaces.




- 1.CO **FontToolReadProjects**
2. Press the  button within the **Projects** area to define the projects to scan for fonts.
3. Press the  button if you want to remove a project from the list.
- 4.> Define the XML file to write the font settings into in the "**Set XML file to export text**" field.

5. Press the  button to open and load the desired folder and then type name of XML file.
6. Close the folder window with the help of the **Open** button.
If you select an existing *XML* file, a question appears asking you if you want to add the results of the new scan to this *XML* file or if you want to start again. If you add the new results to the existing file, you can benefit from the mappings already done and you will be sure that you use the same definitions for all your data.
7. Press the **Scan projects for texts** button to start the scanning process.
8. CO **FontToolMapFont**
9. Choose the first *XML* file with the results from the scanning of your data.
10. Change the settings for "**New Font Name**", "**New Font Height**", "**New Font Width**" and "**New Font Distance**" for each of the different fonts and the combinations of text attributes.
In case you need to use a different code page than the default one, it is possible to choose the appropriate script when defining the font. The script you defined appears in the "*New Font Script*" field but cannot be changed
11. Save your changes with the help of the **Set font mapping** button.
12. Close the window with **Cancel**.
13. CO **FontToolChangeProjects**
14. Press the  button within the **Projects** area to define the projects to scan for fonts.
15. Press the  button if you want to remove a project from the list.
16. Choose the *XML* file to take the mapping information from in the "**Set XML file to export text**" field.
17. Press the  button to open and load the desired folder and then select the *XML* file.
18. Close the folder window with the help of the **Open** button.
19. Press the **Replace projects text** button to start the replacing process.







Exercise 29-5: Finding and changing all fonts used in a single or in several libraries.

- 1.CO **FontToolReadSymbol**
2. Press the  button within the **Symbols** area to define the libraries to scan for fonts.
3. Press the  button if you want to remove a library from the list.
- 4.> Define the *XML* file to write the font settings into in the "**Set XML file to export text**" field.
5. Press the  button to open and load the desired folder and then type name of *XML* file.
6. Close the folder window with the help of the **Open** button.
If you select an existing *XML* file, a question appears asking you if you want to add the results of the new scan to this *XML* file or if you want to start again. If you add the new results to the existing file, you can benefit from the mappings already done and you will be sure that you use the same definitions for all your data.
7. Press the **Scan symbols text** button to start the scanning process.
8. CO **FontToolMapFont**
9. Choose the first *XML* file with the results from the scanning of your data.
10. Change the settings for "**New Font Name**", "**New Font Height**", "**New Font Width**" and "**New Font Distance**" for each of the different fonts and the combinations of text attributes.

In case you need to use a different code page than the default one, it is possible to choose the appropriate script when defining the font. The script you defined appears in the "New Font Script" field but cannot be changed

11. Save your changes with the help of the **Set font mapping** button.
12. Close the window with **Cancel**.
13. CO **FontToolChangeSymbol**
14. Press the  button within the **Library** area to define the projects to scan for fonts.
15. Press the  button if you want to remove a library from the list.
16. Choose the XML file to take the mapping information from in the "**Set XML file to export text**" field.
17. Press the  button to open and load the desired folder and then select the XML file.
18. Close the folder window with the help of the **Open** button.
19. Press the **Replace symbols text** button to start the replacing process.

Exercise 29-6: Finding and changing all fonts used in a single or in several templates.

- 1.CO **FontToolReadTemplate**
2. Press the  button within the **Templates** area to define the templates to scan for fonts.
3. Press the  button if you want to remove a template from the list.
- 4.> Define the XML file to write the font settings into in the "**Set XML file to export text**" field.
5. Press the  button to open and load the desired folder and then type name of XML file.
6. Close the folder window with the help of the **Open** button.
If you select an existing XML file, a question appears asking you if you want to add the results of the new scan to this XML file or if you want to start again. If you add the new results to the existing file, you can benefit from the mappings already done and you will be sure that you use the same definitions for all your data.
7. Press the **Scan templates text** button to start the scanning process
8. CO **FontToolMapFont**
9. Choose the first XML file with the results from the scanning of your data.
10. Change the settings for "**New Font Name**", "**New Font Height**", "**New Font Width**" and "**New Font Distance**" for each of the different fonts and the combinations of text attributes.
In case you need to use a different code page than the default one, it is possible to choose the appropriate script when defining the font. The script you defined appears in the "New Font Script" field but cannot be changed
11. Save your changes with the help of the **Set font mapping** button.
12. Close the window with **Cancel**.
13. CO **FontToolChangeTemplate**
14. Press the  button within the **Templates** area to define the templates to scan for fonts.
15. Press the  button if you want to remove a template from the list.
16. Choose the XML file to take the mapping information from in the "**Set XML file to export text**" field.
17. Press the  button to open and load the desired folder and then select the XML file.
18. Close the folder window with the help of the **Open** button.
19. Press the **Replace templates text** button to start the replacing process.

EE.10. PLC FUNCTIONALITIES IN THE ADVANCED LEVEL

Automatic numbering of addresses

If you use *SEE Electrical advanced*, you can set the PLC Address Numbering Method in the **Circuit Diagrams Properties** (Decimal, Octal, Hexadecimal (i.e. lower case letters in addresses, for example E0.a) or HEXADECIMAL (i.e. upper case letters in addresses, for example E0.A)). If you enter a value for an address (for example, in the Rack), the next values based upon it are automatically numbered.

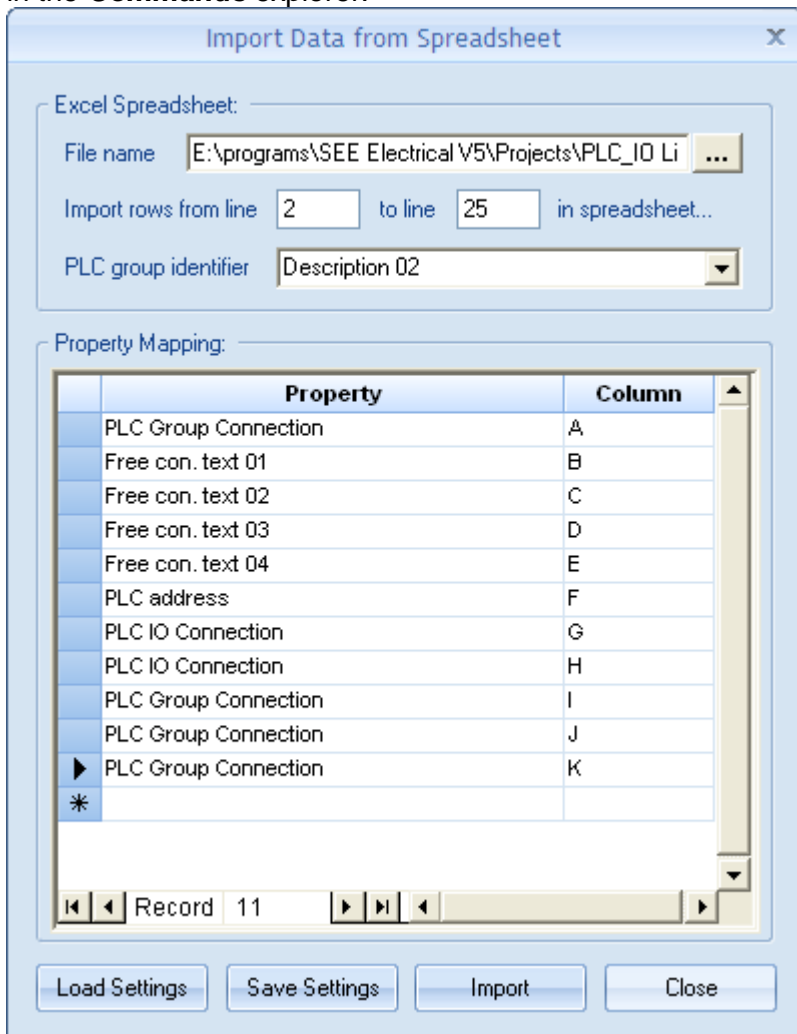
So if you need to position a lot of addresses on the page, position the first one, give the good name of the component and the address. Then copy the next ones. The addresses are automatically numbered according to the chosen PLC Address Numbering Method.

Import the PLC allocation list

There is a feature in *SEE Electrical* for importing a PLC allocation list if it is available as an Excel file. PLC allocation lists usually do not contain information about the component name of the PLC in the Circuit diagram. Instead of this, the PLC components are identified via module names. For this reason, the module names are taken into consideration while importing the allocation list.

Import the Excel List

The import of the Excel list in the workspace can be done by executing the PLCImportExcel function in the **Commands** explorer.



Property	Column
PLC Group Connection	A
Free con. text 01	B
Free con. text 02	C
Free con. text 03	D
Free con. text 04	E
PLC address	F
PLC IO Connection	G
PLC IO Connection	H
PLC Group Connection	I
PLC Group Connection	J
▶ PLC Group Connection	K
*	

Define the text property for the group identifier of the PLC. This text property must be available in the list of the importable text properties.

Symbols

The PLC-Rack symbols and PLC-connections symbols must contain all the texts that are imported, i.e. PLC-Rack symbols and PLC-connections symbols must contain in particular the "PLC group identifier" and the "PLC-address" connections. For the import of the allocation list, it must be chosen which text should receive the "PLC group identifier" (for example, Free text 01 or Free text 20 or Description). It is important to decide which text is possible and you have to insert this text in your PLC Rack and PLC connections.

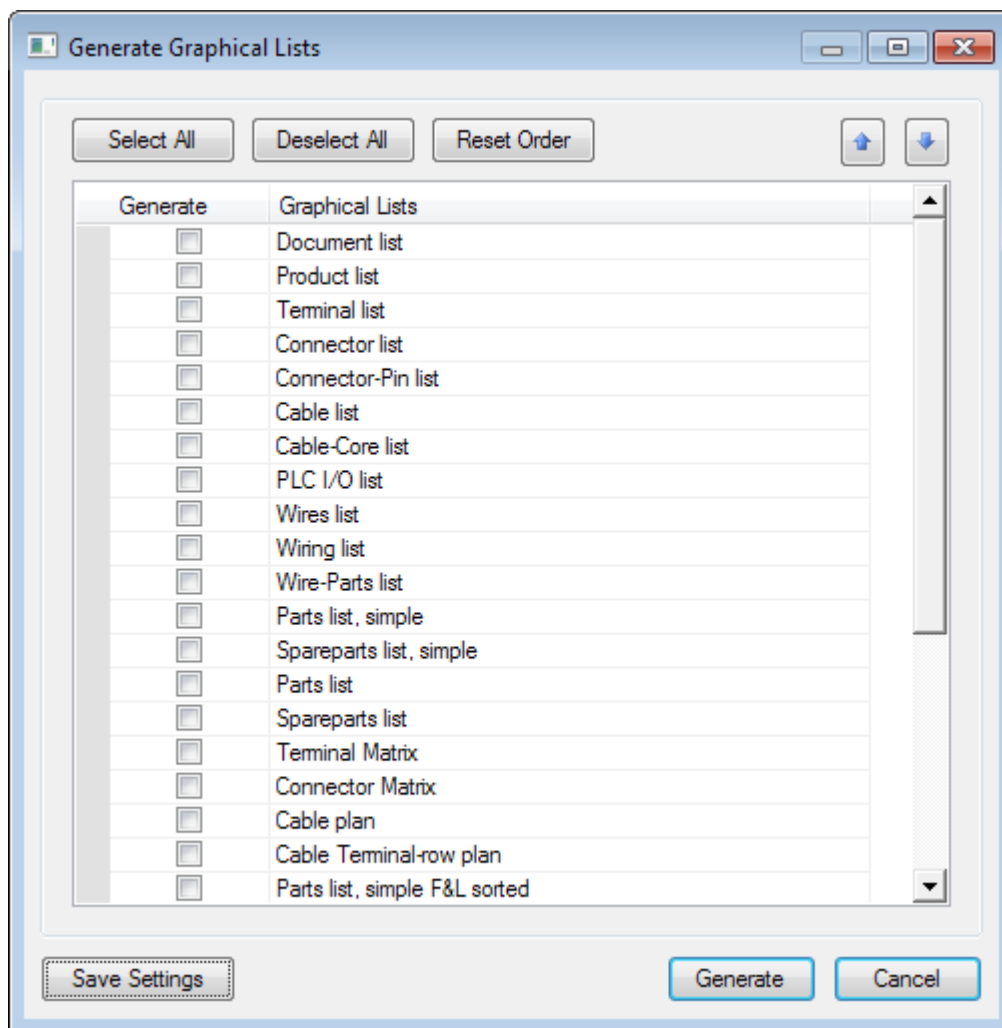
Assign the information to the symbols

Double-click the **PLCImportAssign** command from the **Commands** explorer to execute it. Afterwards identify the symbol that you want to assign information to.


EE.11. GENERATE GRAPHICAL LISTS IN ONE STEP

Advanced

The *Graphical lists* can be generated in one step using the **Generate** command in the *Graphical lists* node.



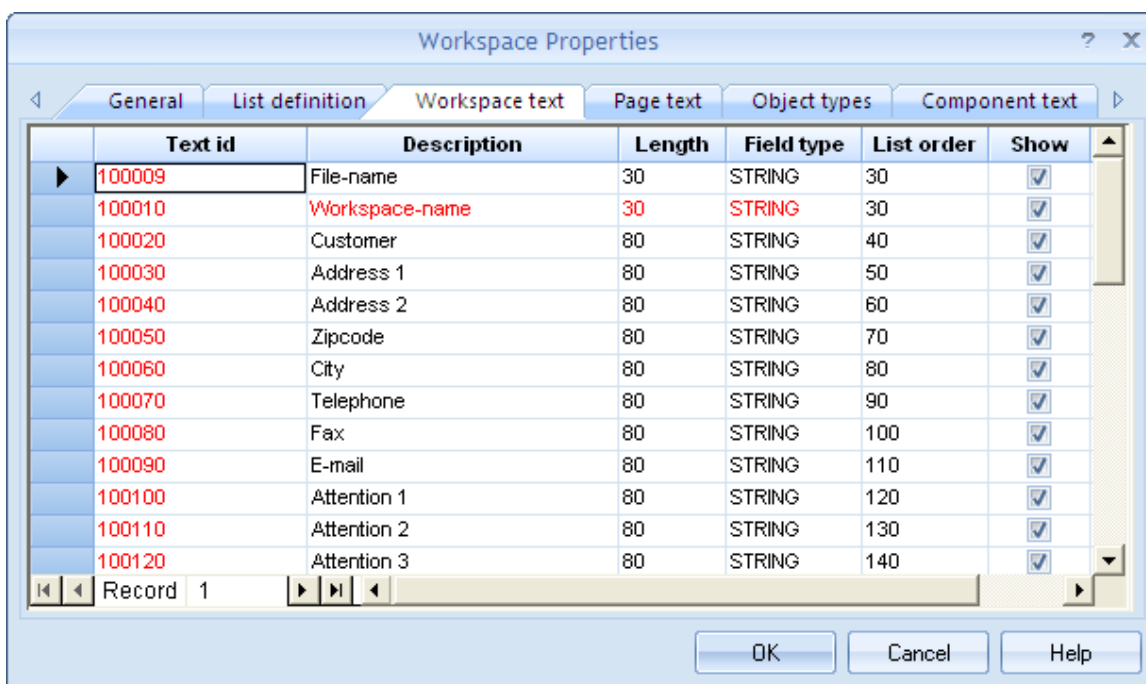
Exercise 29-7: Generate the graphical lists with the help of the **Generate** pop-up command:

- 1.+ In the **Workspace Explorer**, right-click the *Graphical lists* node.
- 2.CO **Generate**
- 3.> Select the lists you want to generate.
- 4.> The  button allows you to define the order in which the lists are generated. You can make sure that the document list is generated as last one so it will contain all others.
- 5.> You can use the **Save Settings** button to save the settings you just defined for using them again.
- 6.> **Generate**

FF CUSTOMIZING THE WORKSPACE/PAGE INFORMATION WINDOWS

You can change the texts in the Workspace information and Page information dialogue boxes. For example, Project-description line 3 could become Commission number. You can change the sequence of displaying the texts in the windows, hide records, or add your own records.

Right-click the workspace name in the **Workspace Explorer** and select the **Properties** pop-up command. You can find the options for changing texts in the **Workspace text** tab or the **Page text** tab.



Records highlighted in red can be changed.

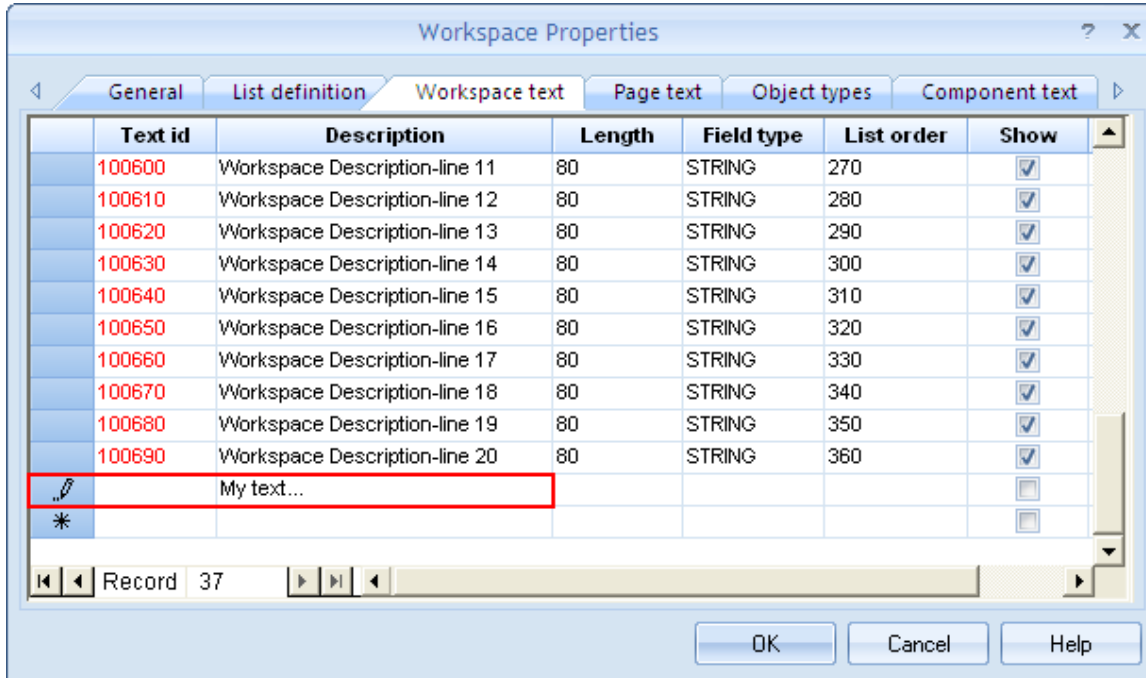
The IDs for user-defined Workspace texts must be within the range 102000 to 110000, and the page texts –within the range 122000 to 130000.

The example below illustrates how a user-defined workspace text is created. Similar procedures are applicable for the creation of page texts and component texts.

Example:

1. Open the **Workspace Properties** window in the already described way and click the **Workspace text** tab.
2. Scroll down to the very bottom of the window and click in the empty Description field.
3. Type in your text there, for example: "My text..."

A new empty line is automatically inserted:

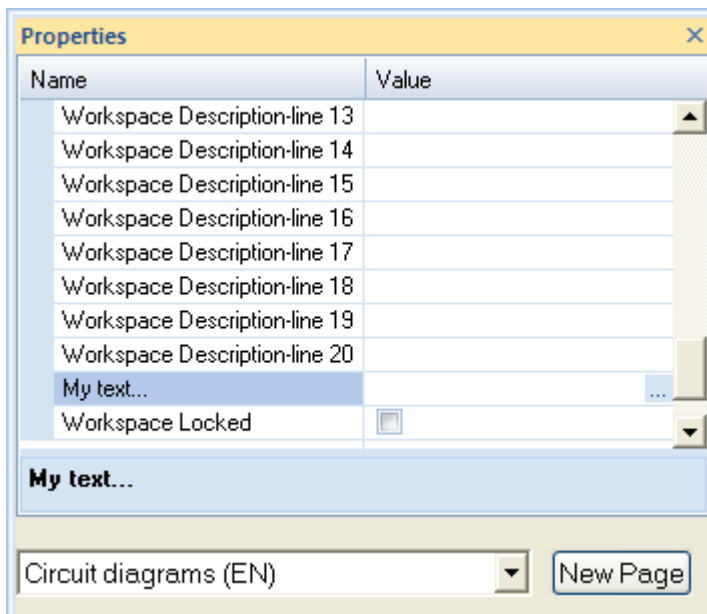


4. Fill in the other fields as desired, tick the check box in the "Show" column and click **OK** to apply your new text.

A message appears informing you that you have to close the workspace and open it again in order to activate the new setting(s).

5. Close the workspace and re-open it.

The new text is displayed in the corresponding **Workspace information** pane:



Hints

1. If you click **OK** without entering the **Text id** field, a message appears, informing you that your text ID must be within a certain range.
 2. To delete any user-defined entry, select it and press the **Delete** key on the keyboard.
-

You can also hide not needed Database lists and Graphical lists.

You can find the customizing options for the workspace tree within the **Workspace Properties** dialogue box by clicking the **List definition** tab.

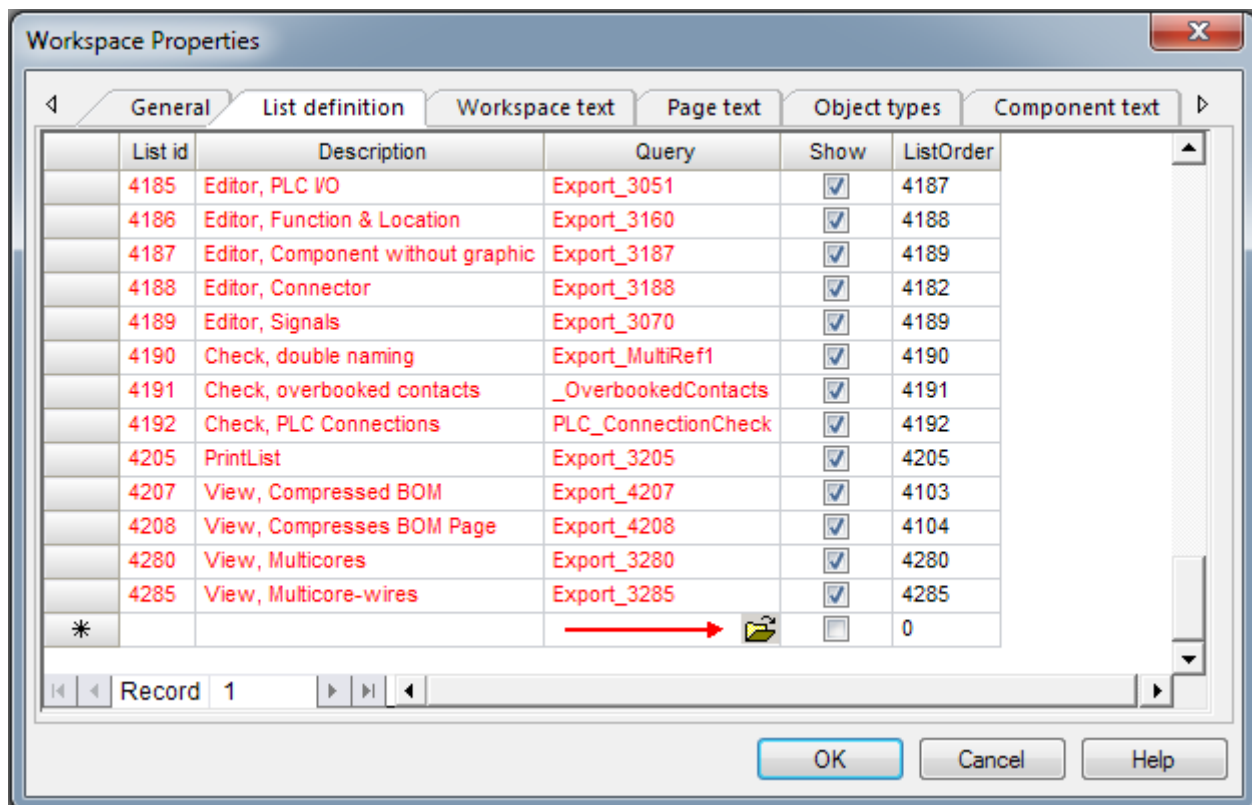
GG CUSTOMIZING THE WORKSPACE TREE

(Advanced)

GG.1. HIDE UNNECESSARY DATABASE AND GRAPHICAL LISTS

It is possible to create your own database and graphical lists, and thus your own SQL queries. (However, it is not possible to insert the "Terminal Matrix", "Terminal Row Picture", "Terminal Row Picture Plan" or "Cable Plan" graphical lists for a second time into the project tree. Also database editors are not user-defined customizable.)

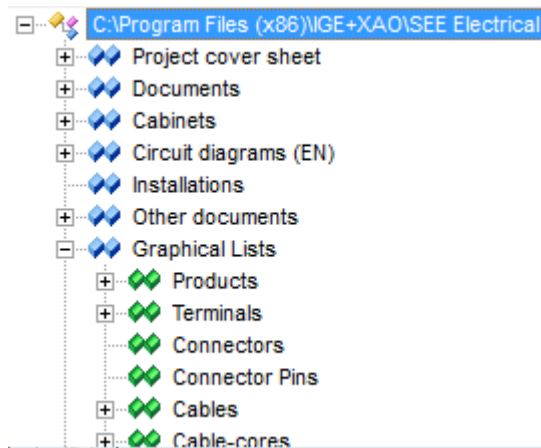
In the **List definition** tab of the **Workspace Properties** window, you can find the options for customizing.



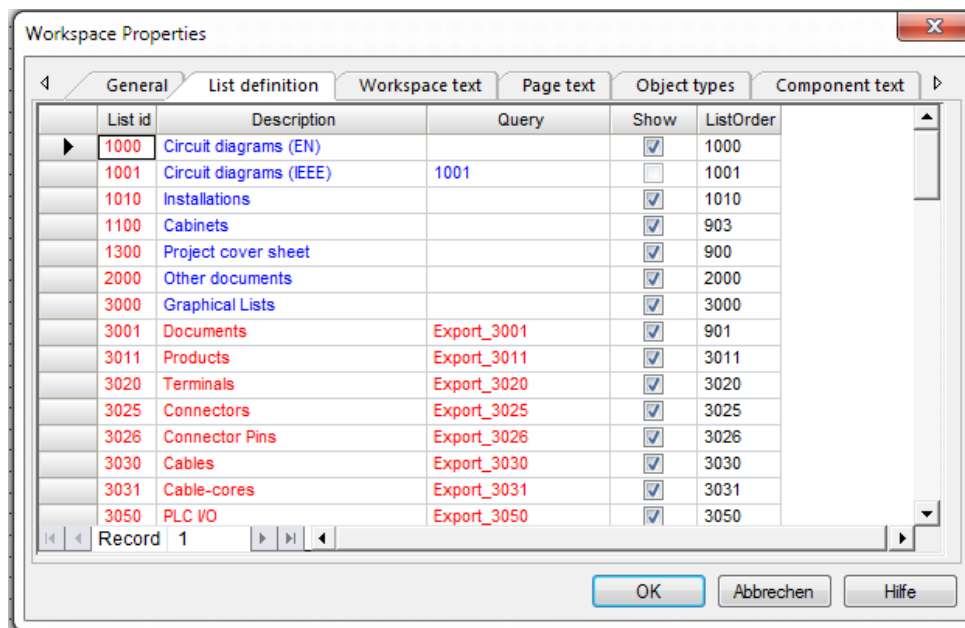
User defined database lists must contain a number from the range 4501 to 4999. The numbers for custom graphical lists must be between 3501 and 3999.

GG.2. SORTING OF DOCUMENTS IN THE WORKSPACE TREE

You can change the order of appearance of the lists in the workspace tree. For example, you can order the workspace tree in the following way:



To sort the lists as desired, change the list order number in the respective field:



For example give 1 for the Project cover sheet, 2 for the Document list, 3 for the cabinets and 4 for the Circuit Diagrams.

The native sort orders for the grafical lists are values between 3000 and 3999.

The database lists can be sorted using values between 4000 and 4999.

The sort order given here is the one used in the **Print** command when all pages are printed one after the other.

The "*View, Documents*" and "*Editor, Documents*" database lists show the documents as ordered in the workspace tree.

The order of the documents part of the *Graphical lists* depends on the sorting order within the **List Definition** tab, e.g if the order is the default one (3001) or a customized order is used. If the default order has not been changed, for compatibility reasons, the *Documents list* is always inserted in the top of the lists and you can still use the **FirstContentLine** command.

If you use a customized order, the *Documents list* is sorted according to the workspace sort order. If you want to keep the *Documents list* in first place in the *Graphical lists* and it has to be in first place also in the *Documents* node, you could change the sort order to 3002.

HH USER-DEFINED SQL-QUERIES


(Advanced)

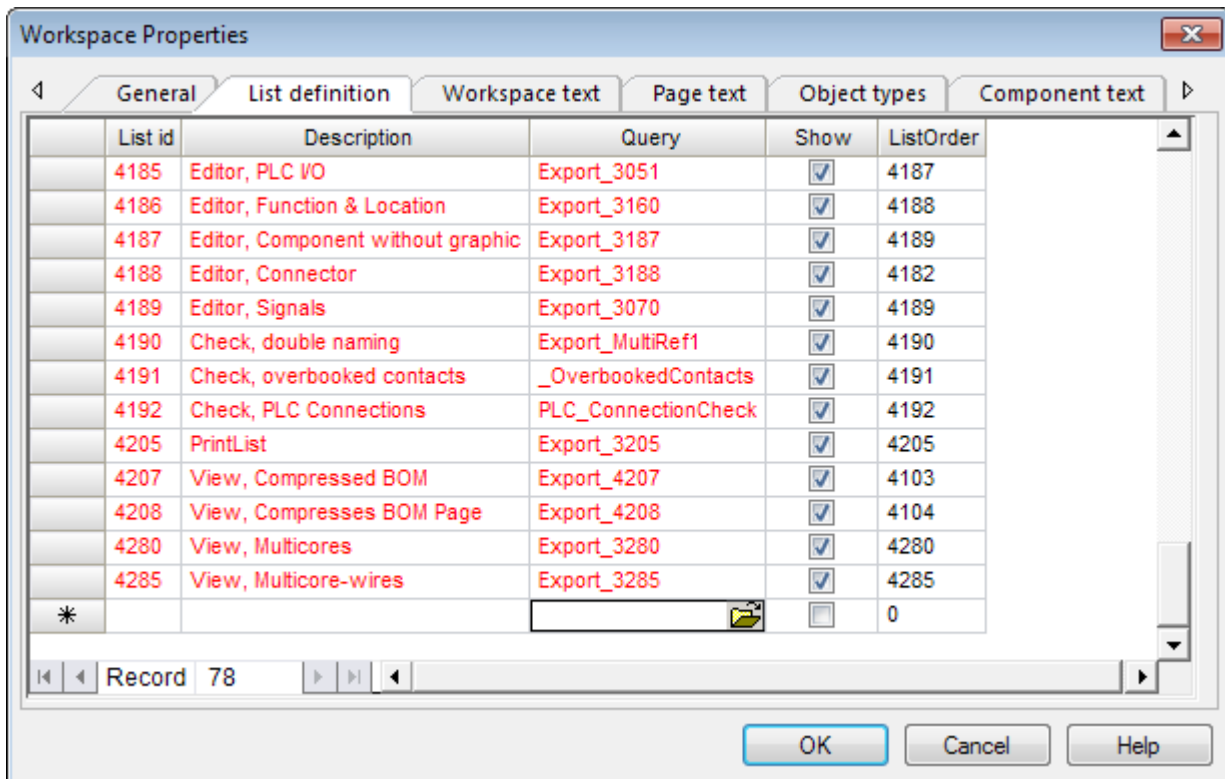
SQL is embedded in the *SEE Electrical* database to allow you to create any kind of form, including summarization of different parameters.

You have the possibility to create your own Database lists and Graphical lists, and herewith your own SQL-queries. (However, it is not possible to add the "Terminal matrix", "Terminal plan", "Terminal plan (multi level)" or "Cable plan" into the Workspace Explorer again. Database editors are not customizable either.)

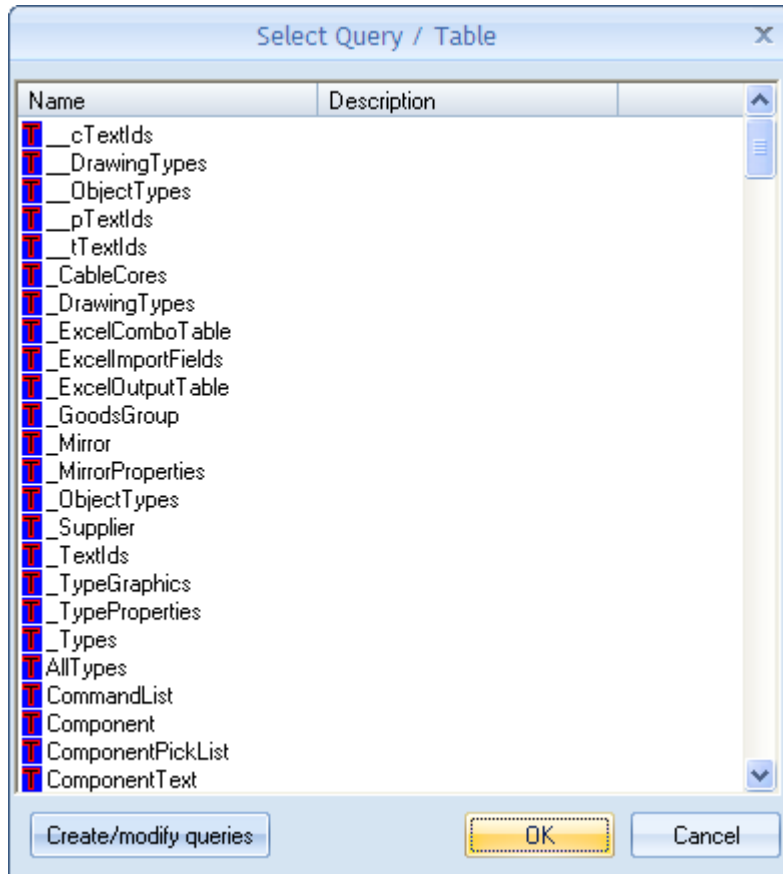
SQL-queries are created or modified by means of the SQL Builder.

HH.1.1. CREATING AN SQL-QUERY FOR ADDING A DATABASE LIST IN THE WORKSPACE EXPLORER

For accessing the SQL Builder, right-click first on the workspace name in the *Workspace Explorer* and select the **Properties** pop-up command to open the *Workspace Properties* window. Then click the  icon in an empty Query field within the **List definition** tab:



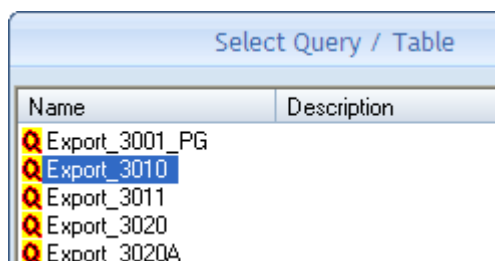
The following dialogue appears:



When you click the **Create/modify queries** button, the SQL Builder opens. Here, after clicking the **Select table / query** button, the dialogue you can see above opens again. Now you can define your own query.

Example of creating a list with a total amount of prices:

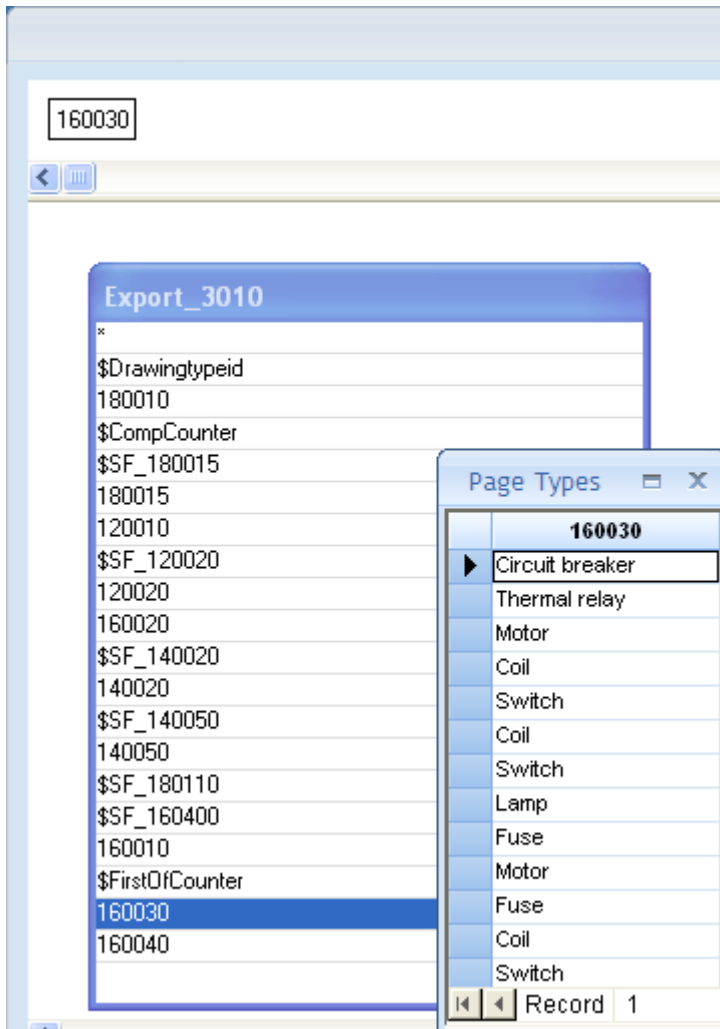
Select, within the **Select Query / Table** window, the **Export_3010** query, for example, and click **OK**:



Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

From the *Export_3010* table that appears, select 160030 which is the ID of Description 00, and click the **Preview / Test Query** button in the SQL Builder. A table is opened that contains the "Description 00" column from the Products database list with all entries from your project:

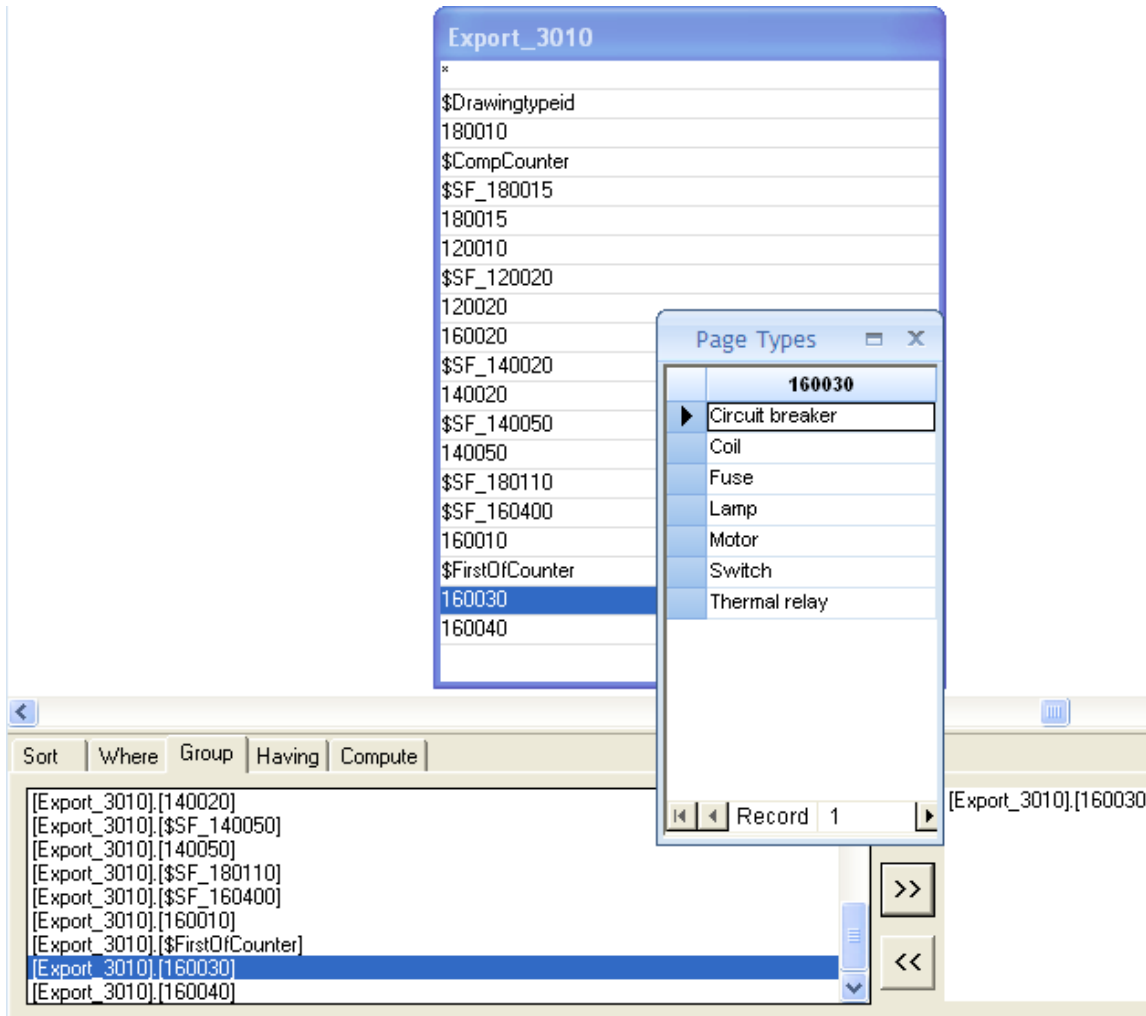


Close the **Page Types** dialogue.

Now, group the entries first, i.e. create a column containing all entries only once.

For this purpose, within the **Group** tab, select [Export_3010].[160030] and move it to the right bottom pane using the corresponding directional button.

Then click the **Preview / Test query** button. The **Page Types** dialogue appears again but now containing all the entries grouped – no multiple entries are displayed:



The next step is to calculate the total number of each entry (item) which will be displayed in another column next to the respective item entry to show what the existing quantity of that is.

So, close the **Page Types** dialogue and click the **Compute** tab.

Within the "**Function**" field, select Count from the scroll-down list that opens.

Then click within the "**Column**" field and select [Export_3010].[160030] from the pull-down list.

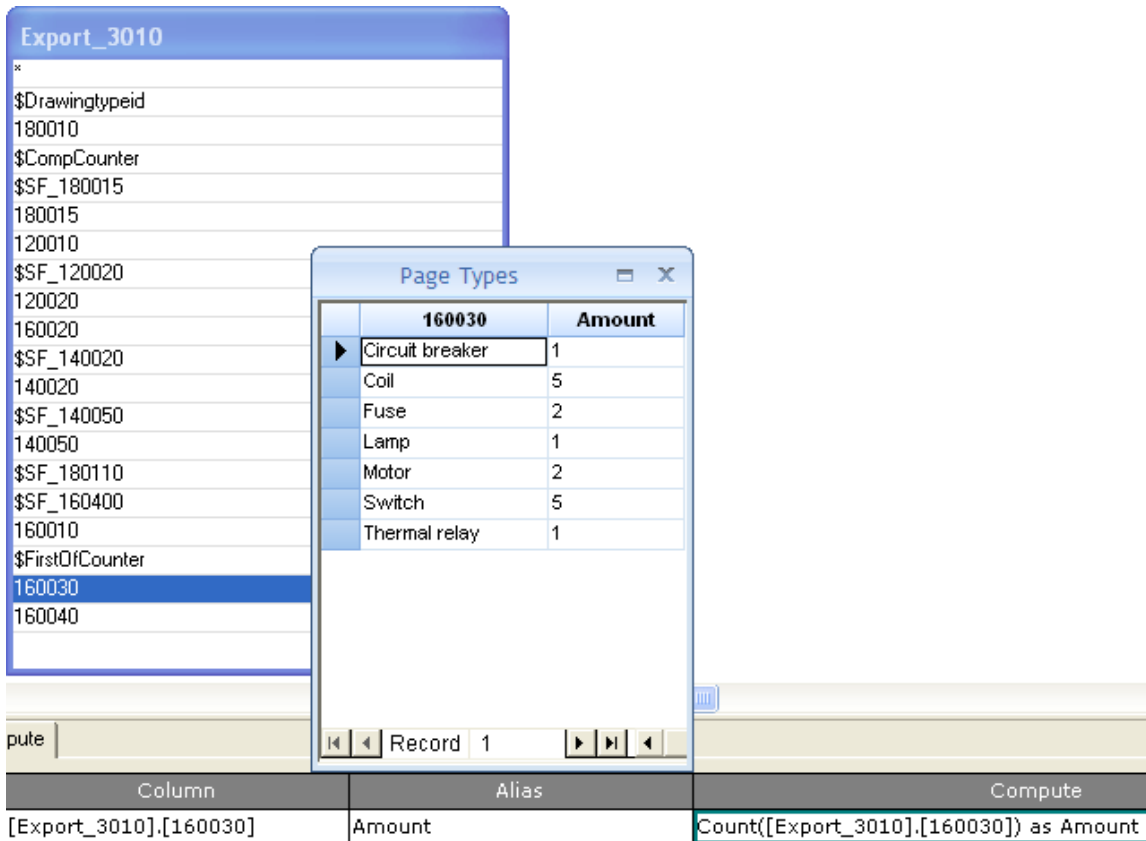
In the **Alias** field, type in, ex. Amount for the type.

Select afterwards the "**Compute**" field being already filled in, and then click the **Preview / Test Query** button.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

The **Page Types** dialogue again appears but now showing also the number of each kind of item, i.e. the amount of each grouped entry. As well, this is the way in which the respective database list will look within SEE Electrical:



The screenshot shows the 'Export_3010' list on the left, the 'Page Types' dialog in the center, and the SQL query editor at the bottom.

Export_3010 List:

- \$Drawingtypeid
- 180010
- \$CompCounter
- \$SF_180015
- 180015
- 120010
- \$SF_120020
- 120020
- 160020
- \$SF_140020
- 140020
- \$SF_140050
- 140050
- \$SF_180110
- \$SF_160400
- 160010
- \$FirstOfCounter
- 160030
- 160040

Page Types Dialog:

160030	Amount
Circuit breaker	1
Coil	5
Fuse	2
Lamp	1
Motor	2
Switch	5
Thermal relay	1

SQL Query Editor:

Column	Alias	Compute
[Export_3010].[160030]	Amount	Count([Export_3010].[160030]) as Amount

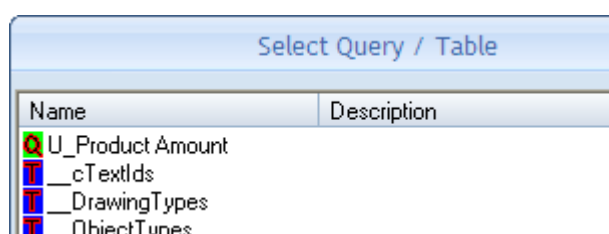
Close the **Page Types** dialogue.

Hint

If you want to view the string of your query, click the **Show SQL string** button: a small message dialogue appears displaying your SQL-query string. Click **OK** to close it.

- Click the **Save query** button to save your new defined SQL-query, give it a relevant name, ex. **Product Amount**, and exit the SQL Builder.

Your query is added to the list in the **Select Query / Table** dialogue window:



The 'Select Query / Table' dialog shows a list of queries and tables:

Name	Description
U_Product Amount	
__cTextIds	
__DrawingTypes	
__ObjectTypes	

- Define now a new **List Definition** (Database list) in the **Workspace Properties** window and choose for query the one just created.

4285	View, Multicore-wires	Export_3285	<input checked="" type="checkbox"/>	0
4501	Product Amount by Description	U_Product Amount	<input type="checkbox"/>	0

- Tick the check box in the "**Show**" column and click **OK** to save the changes.
- Close the workspace. When you re-open it, the new database list will be displayed in the **Workspace Explorer**.

User-defined Database Lists must receive an ID number within the range 4501 –4999. The number for user-defined Graphical lists must be within the range from 3501 to 3999.

In this new database list created with the user-defined query described above, all Descriptions 00 concerning the various items are listed and what is more, all products with the same description are counted and their total number is displayed for each description:

Workspace

Editor, Cable

Editor, Wire

Editor, Document

Editor, PLC I/O

Editor, Function & Location

Editor, Component without graphic

Check, double naming

Check, overlooked contacts

Check, PLC Connections

PrintList

View, Multicores

View, Multicore-wires

Product Amount by Description

DB

Example 1: Product Amount by Description

	Description 00	Amount
1	Circuit breaker	1
2	Coil	5
3	Fuse	2
4	Lamp	1
5	Motor	2
6	Switch	5
7	Thermal relay	1

HH.1.1. DEFINITION OF GRAPHICAL FORMULAS

Within the graphical lists, the query is inserted as a normal text containing the following syntax:
 #QUERY [<name of query>] #WHERE [<argument>]="<content>".

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

For example: #QUERY [U_APd] #WHERE [APno]="1"

B: Power electrical equipment

Description of equipment	Power (KW)
#QUERY [U_APd] #WHERE [APno]="1"	#QUERY [U_APP] #WHERE [APno]="1"
#QUERY [U_APd] #WHERE [APno]="2"	#QUERY [U_APP] #WHERE [APno]="2"
#QUERY [U_APd] #WHERE [APno]="3"	#QUERY [U_APP] #WHERE [APno]="3"
#QUERY [U_APd] #WHERE [APno]="4"	#QUERY [U_APP] #WHERE [APno]="4"
#QUERY [U_APd] #WHERE [APno]="5"	#QUERY [U_APP] #WHERE [APno]="5"
#QUERY [U_APd] #WHERE [APno]="6"	#QUERY [U_APP] #WHERE [APno]="6"
#QUERY [U_APd] #WHERE [APno]="7"	#QUERY [U_APP] #WHERE [APno]="7"
#QUERY [U_APd] #WHERE [APno]="8"	#QUERY [U_APP] #WHERE [APno]="8"
#QUERY [U_APd] #WHERE [APno]="9"	#QUERY [U_APP] #WHERE [APno]="9"
#QUERY [U_APd] #WHERE [APno]="10"	#QUERY [U_APP] #WHERE [APno]="10"

Summary Power: electrical equipment

#QUERY [U__APtotPW]

Maximum 10 pieces of equipment can be listed in the list in this example. The index for the components listed in this list has to be unique, as we want to see each piece of electrical equipment in one line.

This template generates the following result:

B: Power electrical equipment

Description	Power (KW)
cooker	7.50
Boiler	4.00
washing machine	2.50
air conditioner	1.50

Summary Power: electrical equipment

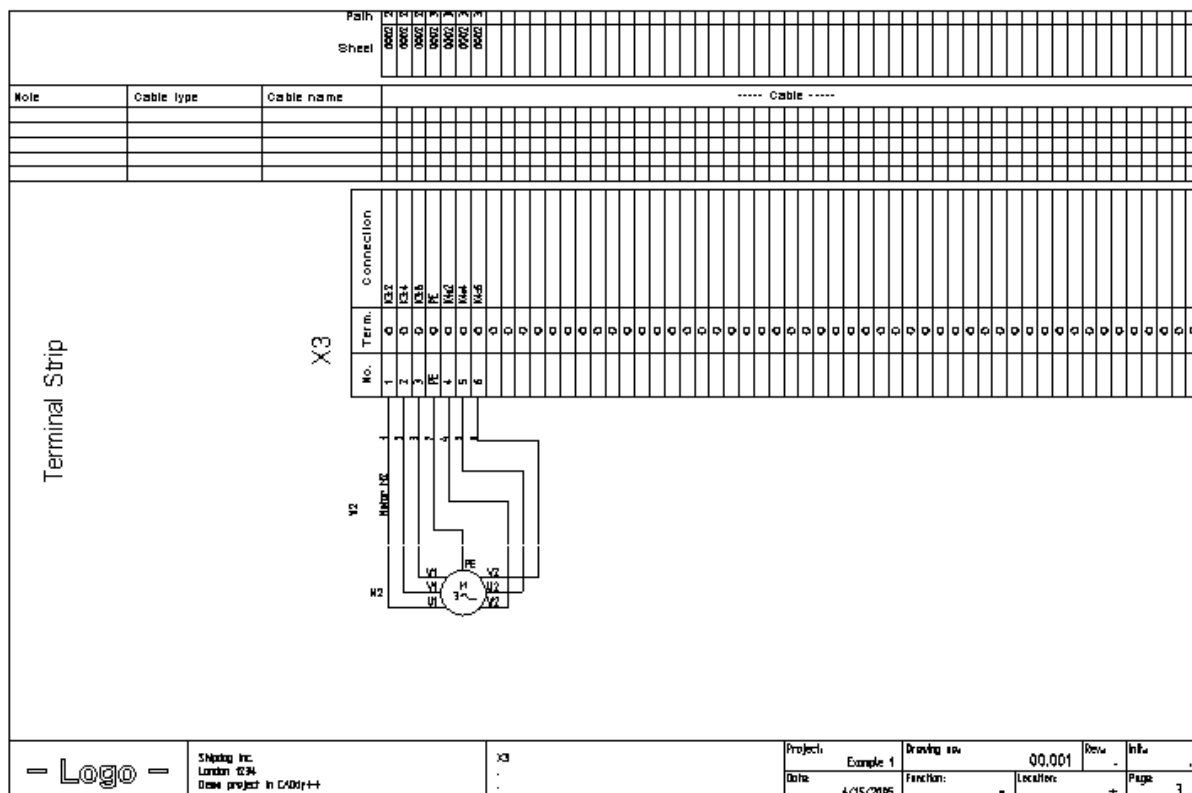
15.50 KW

II TERMINAL PLAN

(Advanced)

II.1. GENERATING A TERMINAL PLAN

In terminal plans with graphics, the first target is assigned to each terminal as a symbol; the second target is assigned to the terminal as a text.



Exercise 32-1: Generate a terminal plan for the training project.

1. Select *Terminal plan* in the **Graphical lists** area in the **Workspace Explorer**.
2. Right-click with the mouse.
- 3.CO **Generate**
The **Select Terminal Row** dialogue appears.
- 4.> Choose the terminal strip(s) for which you wish to generate a terminal plan.
By default all terminal strips are selected.
- 5.> Click **OK**.
The terminal plan has been created.

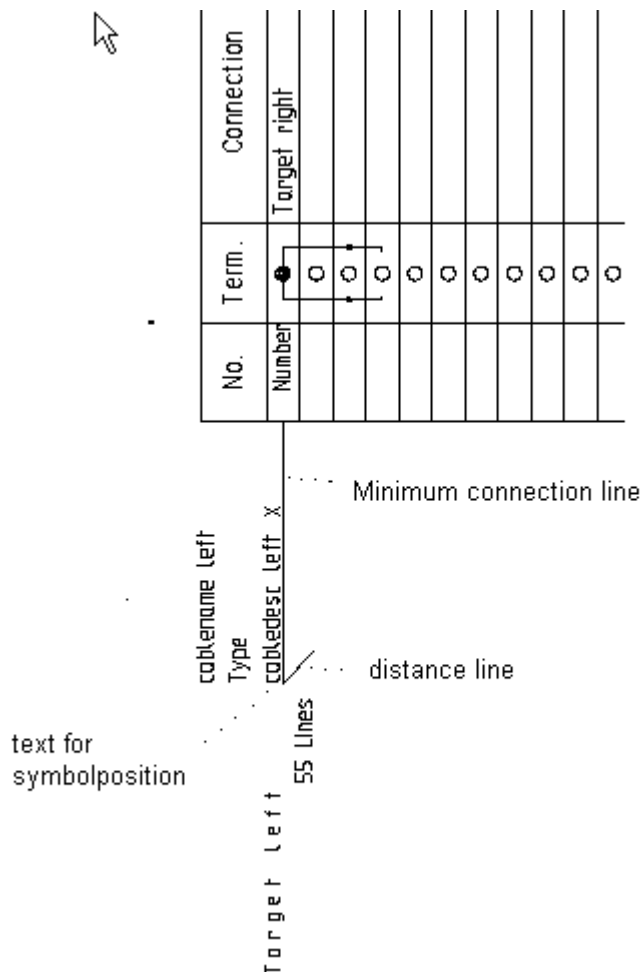
Exercise 32-2: View the terminal plan. Each terminal strip is displayed on a separate page.

1. 0001
Select page 1 of the terminal plan by double-clicking on 0001 under *Terminal plan* in the **Workspace Explorer**.
2. 0002
Select page 2 of the terminal plan by double-clicking on 0002 under *Terminal plan* in the **Workspace Explorer**.

II.2. CREATING A TEMPLATE FOR A TERMINAL PLAN

Forms for terminal plans represent page templates with special properties. In general, they are created as forms for terminal matrices.

Two "routes" and one "text" are needed in addition, in order to define where the symbols must be placed and how the wires are to be represented.



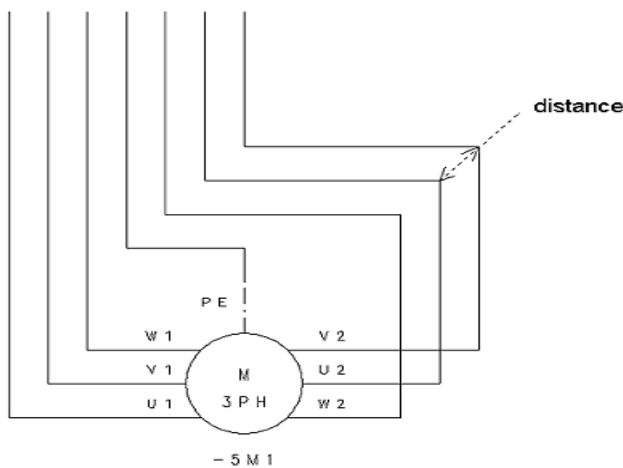
1. First route: Minimum connection route

The route must display a connection between the place where the text of the first terminal will appear and the place where the component symbol will appear.

This route will be extended to the first connection of the component symbol while generating the terminal plan, if necessary.

Only vertical or horizontal lines are allowed.

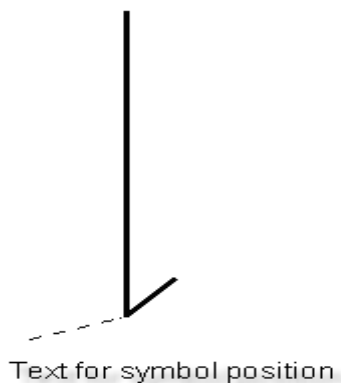
2. Second route: The route defines the distance between the terminal row and the last remote object.



If several connection lines exist, this route defines the distance between the connection lines; to be more precise: the difference between the x and y values defines the distances between the connection extension lines.

3. The text

The text defines where the reference point of the component is placed.



The "routes" and the "text" have special identifications. Insert the Routing symbol from the **List construction set** symbol database into the Terminal plan and change it, if you want to create a completely new Terminal plan.

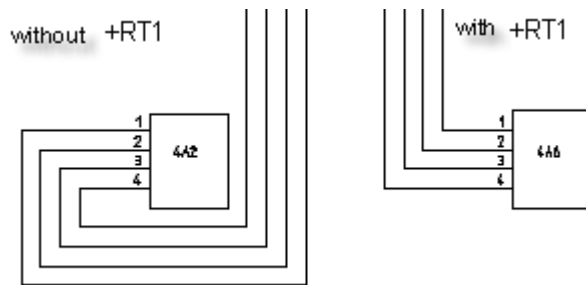
Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

Routing

If the connection of a symbol is not located on the top side of the symbol, the connection lines are drawn on the right side round to the symbol.

If you insert the +RT1 text in the text placeholder for the left target, the connection lines are drawn directly to the symbol connections.



Symbol scaling

+SF, +SX and/or +SY in the text placeholder for the left target defines the Symbol scaling in the Cable plan.

+SF defines a scaling factor in X and Y direction.

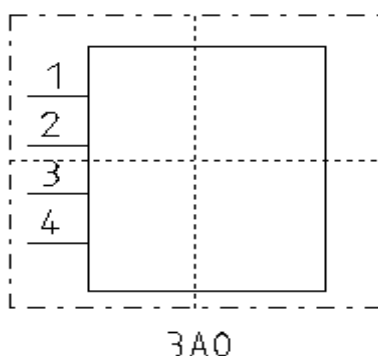
Example: +SF0.8 makes all the symbols smaller by the factor 0.8.

+SX or +SY define limits for the component in X/Y direction. *Example:* +SX100: If the extension of a symbol in X direction is greater than 100 mm, then the scaling factor for this special component is defined so that its extension in X direction is less than or equal \leq 100 mm. +SY applies adequately.

Explicit component text insertion

The *Function*, *Location*, *Product*, *Description* and *Type* component texts can be displayed at places different from those where they are located within the symbol in the *Circuit diagram*.

The texts are located outside a rectangle that frames the symbol.



You must define the position of the texts in the Form for the *Cable plan* as follows:

- Insert a text with a Function, Location, Component name, Description and/or Type attribute.

If you do not work with Function/Location, you do not need to insert the placeholders. If the Function must be shown at the place where the text is located at the symbol in the Circuit diagram, do not insert this text, etc.

If the text placeholder for the component name contains the =+- signs, the Function and Location, as well as the Component name appear consecutively in this text.

- Enter one of the following definitions in the placeholder for the left target of the terminals:
 - ✓ +MTX0: place the text on the right to the component
 - ✓ +MTY0: place the text beneath the component
- Group the component texts and the text for the left target as a "Macro/Group" symbol. You cannot use here all of the settings options available in the template. Refer to the "Graphical lists" chapter in the Help files for more details.
- Display cable core information and colour, potential name or signal type
 The texts with the "Cable-core Number Left" (Id=180128) and "Cable-core Number Right" (Id=180131) attributes are used to display a lot of information.
 The first letter in the text allows you to decide which information is shown for a cable-core and the second letter defines which text is shown for the wire:
 - shows nothing
 - + shows number (default)
 - N shows number (default)
 - C shows colour (if colour exists)
 - c shows colour (if colour exists) and number (if colour does not exist)
 - E shows always colour (also with number not used)

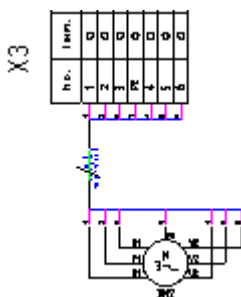
If you need more information, you can position a text with the "Cable-core Number Left" and "Cable-core Number Right" attributes a second and a third time and display the information for signal type or potential name with their help.

- Q generates information about the signal type
- P generates information about the potential name

II.2.1. DRAWING GRAPHICS ASSOCIATED WITH EACH TERMINAL

As described in the chapter "Creating a Template for the Terminal Matrix", it is possible to use a symbol for each terminal instead of a grid for the whole page.

II.2.2. DRAWING CABLES AS A GROUPING OF CABLE CORES



Training manual

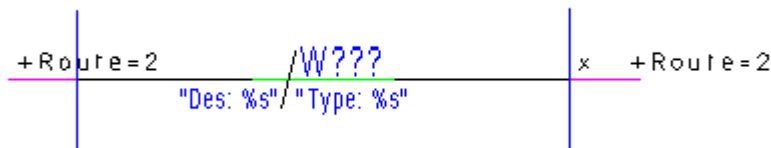
COPYRIGHT © 2013 IGE+XAO. All rights reserved

The groupings of cable cores are generated because, instead of texts for the cable information, definitions for a cable group are added into the page template for the *Terminal Plan*. The text for the number of lines available for cables has to be defined additionally to this cable group.

Generating the page template:

Generate all objects necessary for the terminal plan as already described. (You can load an existing page template and delete the texts for the cable information from this template.

- Block all this objects as "Block/Macro/Group".
- Generate the cable group as follows:



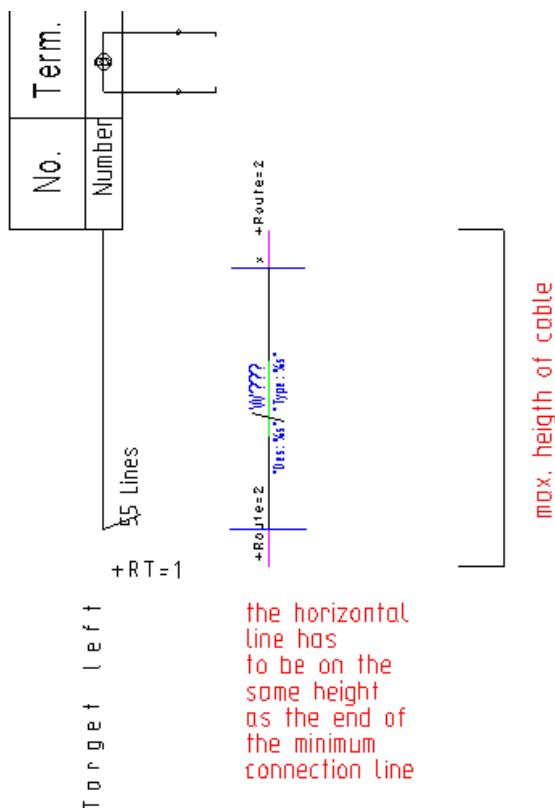
(The symbols are shown rotated by 90 degrees.)

The cable symbol (ID=1600) consists of 3 blocks:

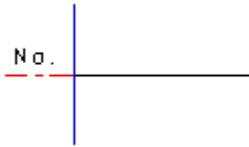
- ✓ the routing information from cable to target
- ✓ the cable
- ✓ the routing information from cable to terminals
- ✓

After you create all 3 parts, block them as "Cable".

The length of the three parts together has to be equal to the distance between the geometry of the terminal plan and the position of the text placeholder "Graphical target left". Additionally the horizontal line in the cable has to be at the same height as the end of the line for the minimum distance.



Part 1: Generate routing-information from cable to the target:



The part for the routing-information from the cable to the target consists of:

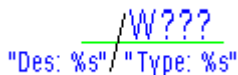
- ✓ the text placeholder for cable core number left (ID 180128), for example. No.
- ✓ the text placeholder for cable core square (ID 180138) (optional)
- ✓ the minimum connection line, (black horizontal line, ID 180125)

As in the terminal plan, the line shows a connection between the positions, where later the text for the first cable core is found, to the position, where the component symbol is found. Eventually the line is extended to the position, where the first contact point of the component is found, when generating the plans.

- ✓ line for cable core number (lilac, horizontal line, ID 1 (shown dash dotted here))
- ✓ vertical line, used to connect all cable cores, ID 1, shown blue here. This line crosses the other two lines in their common endpoint.

- Block all these objects as "Block/Macro/Group".

Part 2: Generate cable:



The part for the cable contains:

- ✓ The geometry for the cable.
Generate the geometry like you normally do. You can add normal texts if you want. Block all as "Graphical Symbol".
- ✓ Text placeholder for cable name, for example W??? (ID 180110)
- ✓ Text placeholder for "Type" (ID 180140)
Example: Type %s" where the placeholder "%s" is replaced by the cable type.
- ✓ Text placeholder for "Description" (ID 180142)
Example: Des: %s" where the placeholder "%s" is replaced by the cable description.
- ✓ Text placeholder for "cable type" (ID 180144)
Example: Dim: %s" where the placeholder "%s" is replaced by the text "cable-type".
- ✓ Text placeholder for "Length" (ID 180145)
Example: L= %s" where the placeholder "%s" is replaced by the length

- Block all these objects as "Block/Macro/Group".

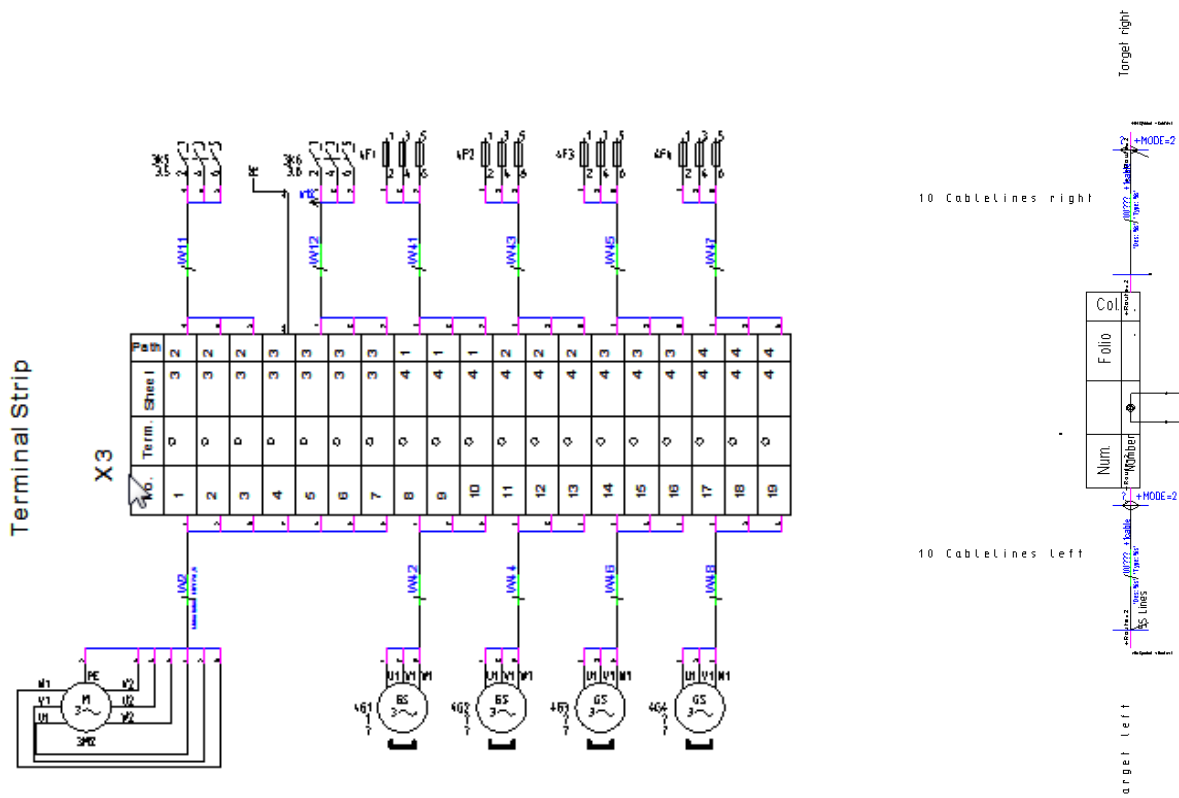
Part 3: Generate routing-information from cable to terminal



This block consists of the same elements as the cable core left mirrored.

- Block all these objects as "*Block/Macro/Group*".
- Select the blocks "Routing-information from cable to target", "Cable" and "Routing-information from cable to terminal".
- Block all these objects as "*Block/Macro/Group*".
- Position the number of lines on top of the minimum connection line.
- Block all the objects as "*Page Template, Title block*".

II.2.3. DRAWING CABLES/GEOMETRY ON BOTH SIDES OF THE TERMINALS





If you want to add the graphical representation to both sides of the terminal, you can insert the same objects used to insert the representation on the left side, just use "Target right " instead of " Target left".

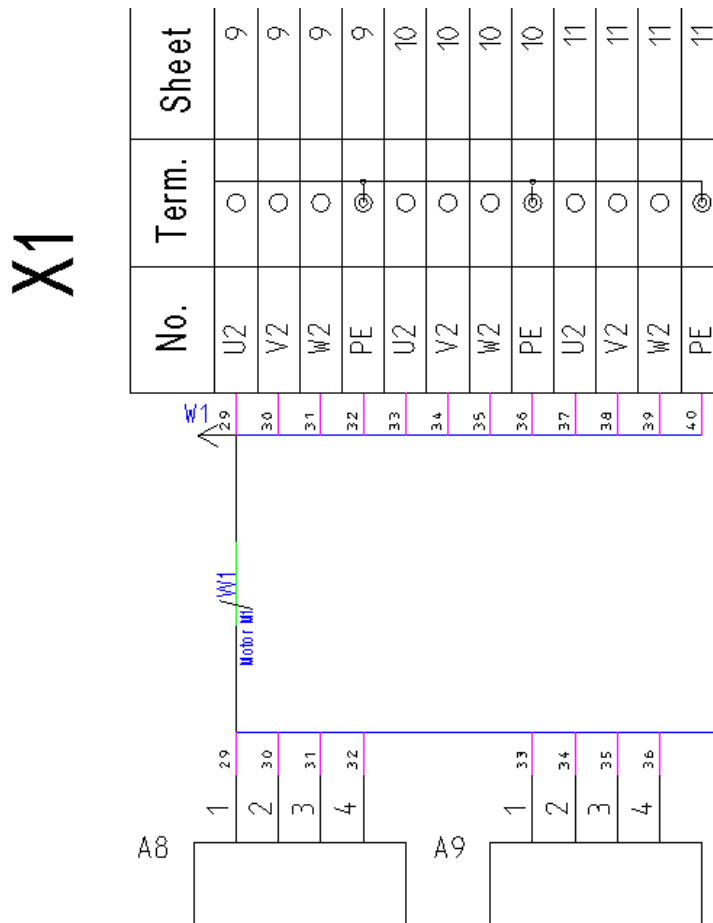
Of course, you also have to use " cable core number right" instead of "cable core number left" etc for the other necessary texts. One text is orientated in the direction of the connection point of the terminal and the other is orientated in the direction of the connection point of the target side.

II.2.4. SHOWING THAT CABLE CORES ARE USED ON PREVIOUS/NEXT PAGE

The *Terminal Plan* can generate information showing whether cable cores of a cable are used on

previous  or next  page.


Example:



In the page template you have to add two macro/groups to each placeholder for left or right cable:

- ✓ group for cable reference symbol to previous page;
- ✓ group for cable reference symbol to next page.

Each cable reference group contains the following elements:

- ✓ text with attribute "cable reference LAST" or "cable reference NEXT" (depends on which cable reference group you are preparing)
- ✓ geometry which represents the cross reference, for example  .

With the text with the attribute "cable reference LAST" or "cable reference NEXT" you can control the position on which the cross-reference symbol appears. Use the following keywords to do so:

- ✓ +Mode=0 (and other values): the Cable reference-symbol is not drawn
- ✓ +Mode=1: the reference-symbol is moved to the end of the grouping-line of the target-routing;
If the collecting-line of the target-routing gets an offset, then both cable reference symbols get the same offset.
- ✓ +Mode=2: the reference-symbol -symbol is moved to the end of grouping-line of the terminal-routing;
If the collection-line of the terminal-routing gets an offset then both cable reference symbols set the same offset.
- ✓ +Mode=3: the Cable reference symbol is not moved (relative to the cable name).

In the template the following flags control the behaviour:

If the cable-name left (id=180127) and cable-name right (id=180130) in the template each contain the substring "+1Cable", then the cable name is shown only once at each terminal row (each graphical cable has only one core-grouping).

Cable-core-routing:

The macro/group for the left cable routing contains 2 core-groupings, which each contain a "Cable-core Number Left" (id=180128) (together with 3 lines building the "cross").

If the "Cable-core Number Left" contains the text "+Route=1" or "+Route=2", the cable-core routing style is changed as follows:

- ✓ +Route=1 (Default): each core-grouping line automatically gets an offset (this is safer, because lines do not appear that easily on top of each other).
- ✓ +Route=2: If this setting is found, the core-grouping line gets an offset only if it intersects with another core-grouping line.

The symbol for the right cable routing works in the same way, but uses "Cable-core Number Right" (id=180131).

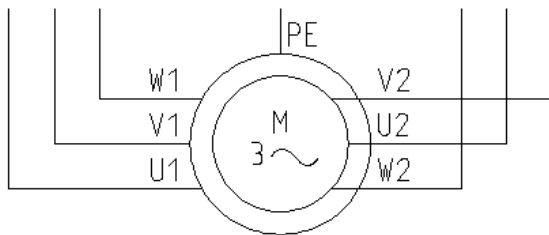
Note :

If a cable is on the left and right side of the terminal-row (only one page), the Cable-reference-symbols may not be correct.

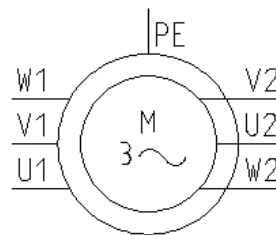
II.2.5. SYMBOLS FOR A TERMINAL PLAN

At the terminal plan generation, the component symbols used in the circuit diagram are placed. Drawing connecting lines in the terminal plan is better possible if the connections of the components are stuck out upward:

This way:



Not this way:



Suppress symbols:

If you add the +NoSymbol text to the left target, no symbols are drawn, only the component text is used as target.


JJ CONNECTOR MATRIX AND PLAN

(Advanced)

JJ.1. GENERATING A CONNECTOR MATRIX

The connector matrix facilitates the installation of connectors.

Connector Matrix


Connector:  XJ1

Type: Conn7F female connector

Description	Connection	Pin-M	Pin	Plugged to
M01/8		1	A	XJ1/1
M01/4		2	B	XJ1/2
M01/1		3	C	XJ1/3
M01/E		4	D	XJ1/4
M01/2		5	E	XJ1/5
M01/0		6	F	XJ1/6
M01/2		7	G	XJ1/7

Cable name left: 2 cablelines

Cable name


Connector:  XP1

Type: Conn7M male connector

Description	Connection	Pin-M	Pin	Plugged to
M02		1	1	X01/A
M04		2	2	X01/B
M06		3	3	X01/C
M08		4	4	X01/D
M10		5	5	X01/E
M12		6	6	X01/F
M14		7	7	X01/G

Cable name left: 2 cablelines

Cable name



Bhishop Inc.
London 1234
Demo project in SEE Electrical

XJ1, XP1

Project: Example 1-V7demo
Date: 31.01.2012

Drawing no.: 00.001
Function: Location: Page: 1.

Exercise: Generate a connector matrix.

1. Select *Connector matrix* from within the *Graphical Lists* in the Workspace tree. Expand the *Graphical Lists*.
2. Right-click with the mouse.
- 3.CO **Generate**
In the **Select Terminal Row** dialogue, choose the connector(s) for which you wish to generate a matrix, for example X2 and X3.
- 4.> X2, X3
- 5.> OK
The Connector matrices have been created.

JJ.2. CREATING A TEMPLATE FOR A CONNECTOR MATRIX

Templates for connector matrices are page templates with special properties.

A *Connector Matrix* provides similar possibilities, which you know from *Terminal Matrix*. That is why, the template creation follows similar rules, too.

Differences in the text place holders between Terminal Matrix and Connector Matrix (or Terminal Row Picture and Connector Plan):

	Terminal Matrix and Terminal Row Picture	Connector Matrix and Connector Plan
Header: (ID=180110)	Terminal strip	Connector
Row:	Terminal	Pin
ID=180112	Terminal number	Connector Pin
ID=180137	Terminal index	Pin-ID

In a Connector Matrix or a Connector Plan, cables are entered only on the left side of a pin because the plug on its second side will be always plugged into another connector. By default, the attached connector is always shown on the right side of a pin, so cables are not necessary here.

Bridges are not required in the Connector Matrix or in the Connector plan, so there are no placeholders in the page template.


More information about this you can find in the Manual.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

JJ.3. DRAW MORE THAN ONE CONNECTOR PER PAGE

Connector Matrix										Connector Matrix									
Connector: XJ1										Connector: XP1									
Type: Conn7F female connector										Type: Conn7M male connector									
Description:										Description:									
Plugged to										Plugged to									
Pin-M										Pin-M									
Pin										Pin									
Path										Path									
Sheet										Sheet									
MED1										MED1									
MED2										MED2									
MED3										MED3									
MED4										MED4									
MED5										MED5									
MED6										MED6									
MED7										MED7									
MED8										MED8									
MED9										MED9									
MED10										MED10									
MED11										MED11									
MED12										MED12									
MED13										MED13									
MED14										MED14									
MED15										MED15									
MED16										MED16									
MED17										MED17									
MED18										MED18									
MED19										MED19									
MED20										MED20									
MED21										MED21									
MED22										MED22									
MED23										MED23									
MED24										MED24									
MED25										MED25									
MED26										MED26									
MED27										MED27									
MED28										MED28									
MED29										MED29									
MED30										MED30									
MED31										MED31									
MED32										MED32									
MED33										MED33									
MED34										MED34									
MED35										MED35									
MED36										MED36									
MED37										MED37									
MED38										MED38									
MED39										MED39									
MED40										MED40									
MED41										MED41									
MED42										MED42									
MED43										MED43									
MED44										MED44									
MED45										MED45									
MED46										MED46									
MED47										MED47									
MED48										MED48									
MED49										MED49									
MED50										MED50									
MED51										MED51									
MED52										MED52									
MED53										MED53									
MED54										MED54									
MED55										MED55									
MED56										MED56									
MED57										MED57									
MED58										MED58									
MED59										MED59									
MED60										MED60									
MED61										MED61									
MED62										MED62									
MED63										MED63									
MED64										MED64									
MED65										MED65									
MED66										MED66									
MED67										MED67									
MED68										MED68									
MED69										MED69									
MED70										MED70									
MED71										MED71									
MED72										MED72									
MED73										MED73									
MED74										MED74									
MED75										MED75									
MED76										MED76									
MED77										MED77									
MED78										MED78									
MED79										MED79									
MED80										MED80									
MED81										MED81									
MED82										MED82									
MED83										MED83									
MED84										MED84									
MED85										MED85									
MED86										MED86									
MED87										MED87									
MED88										MED88									
MED89										MED89									
MED90										MED90									
MED91										MED91									
MED92										MED92									
MED93										MED93									
MED94										MED94									
MED95										MED95									
MED96										MED96									
MED97										MED97									
MED98										MED98									
MED99										MED99									
MED100										MED100									



Bhishop Inc.
London 1234
Demo project in BEE Electrical

XJ1, XP1

Project: 1-Example 1-Vtdemo

Date: 06.02.2012

Drawing no.: 00.001

Function: Location:


Rev.: 1

Page: 1

Generating the Page template:

#PageBreak=0 #-Lines=8

Connector Matrix										Connector Matrix									
Connector: XJ1										Connector: XP1									
Type: Conn7F female connector										Type: Conn7M male connector									
Description:										Description:									
Plugged to										Plugged to									
Pin-M										Pin-M									
Pin										Pin									
Path										Path									
Sheet										Sheet									
MED1										MED1									
MED2										MED2									
MED3										MED3									
MED4										MED4									
MED5										MED5									
MED6										MED6									
MED7										MED7									
MED8										MED8									
MED9										MED9									
MED10										MED10									
MED11										MED11									
MED12										MED12									
MED13										MED13									
MED14										MED14									
MED15										MED15									
MED16										MED16									
MED17										MED17									
MED18										MED18									
MED19										MED19									
MED20										MED20									
MED21										MED21									
MED22										MED22									
MED23										MED23									
MED24										MED24									
MED25										MED25									
MED26										MED26									
MED27										MED27									
MED28										MED28									
MED29										MED29									
MED30										MED30									
MED31										MED31									
MED32										MED32									
MED33										MED33									
MED34										MED34									
MED35										MED35									
MED36										MED36									
MED37										MED37									
MED38										MED38									
MED39										MED39									
MED40										MED40									
MED41										MED41									
MED42										MED42									
MED43										MED43									
MED44										MED44									
MED45										MED45									
MED46										MED46									
MED47										MED47									
MED48										MED48									
MED49										MED49									
MED50										MED50									
MED51										MED51									
MED52										MED52									
MED53										MED53									
MED54										MED54									
MED55										MED55									
MED56										MED56									
MED57										MED57									
MED58										MED58									
MED59										MED59									
MED60										MED60									
MED61										MED61									
MED62										MED62									
MED63										MED63									
MED64										MED64									
MED65										MED65									
MED66										MED66									
MED67										MED67									
MED68										MED68									
MED69										MED69									
MED70										MED70									
MED71										MED71									
MED72										MED72									
MED73										MED73									
MED74										MED74									
MED75										MED75									
MED76										MED76									
MED77										MED77									
MED78										MED78									
MED79										MED79									
MED80										MED80									
MED81										MED81									
MED82										MED82									
MED83										MED83									
MED84										MED84									
MED85										MED85									
MED86										MED86									
MED87										MED87									
MED88										MED88									
MED89										MED89									
MED90										MED90									
MED91										MED91									
MED92										MED92									
MED93										MED93									
MED94										MED94									
MED95										MED95									
MED96										MED96									
MED97										MED97									
MED98										MED98									
MED99										MED99									
MED100										MED100									



Bhishop Inc.
London 1234
Demo project in BEE Electrical

XJ1, XP1

Project: 1-Example 1-Vtdemo

Date: 06.02.2012

Drawing no.: 00.001

Function: Location:

Rev.: 1

Page: 1

1. Group the geometry and texts from the title block as "Page Template/Title Block" symbol.
2. Generate all geometry and texts that show information that belongs to the Connector.
3. Place a text with the attribute "Connector".
4. Generate all geometry and texts that show information that belongs to the pin. The geometry and texts that are not changed have to be grouped as "Graphical symbol".
5. Place a text with the attribute "Pin number".
6. The graphical symbol and the text with the attribute "Pin number" have to be grouped as "Macro/Group".
7. . Add a "Normal" text attribute to the content.

#PageBreak=0

or

#PageBreak=0 #Lines=8 (for example)

where #Lines=8 indicates that multiple connectors will be generated and #Lines=8 indicates that the "connector header" uses the space given for 8 terminal-lines (if no #Lines is used to specify the connector header, the necessary space is calculated from the graphic found in the template)

The text "#Pagebreak" in the template can be used to control the generation of the new page.

#PageBreak=0

Multiple connectors are generated on one page like before, a new page is generated only if the page is full.

#PageBreak=1 or #PageBreak=160010 or #PageBreak="-"

are used to start a new page if the name of the connector has been changed.

#PageBreak=140020 or #PageBreak="="

are used to start a new page if the function (=) has been changed.

#PageBreak=140050 or #PageBreak="+"

Are used to start a new page if the function (+) has been changed.

You can define how is managed the function and location information in the connector's name.

The connector's name is always extended with the function and location information if you add the text +DL0 to the #PageBreak command (even if the connector has the same function and location information then the page it is found on).

If you add the text +DL1 to the #PageBreak command then the connector's name contains only the function and location information in case they are different from the ones found on the page.

8. Group all symbols (the macro/group for the standard sheet and the macro/group for the single terminals) as a "Page Template, Title Block" symbol.
9. Save the new page template.

JJ.4. CONNECTOR PLAN

(Advanced)

Connector Plan


Connector: **XJ1**

Type: Conn7F female connector

Description:

Pin-M	Pin	Plugged to
1	A	XP1A
2	B	XP1B
3	C	XP1C
4	D	XP1D
5	E	XP1E
6	F	XP1F
7	G	XP1G

Path Sheet: - 000107, - 000107, - 000107, - 000107, - 000107, - 000107, - 000107




Connector: **XP1**


Type: Conn7M male connector

Description:

Pin-M	Pin	Plugged to
1	1	XJ1A
2	2	XJ1B
3	3	XJ1C
4	4	XJ1D
5	5	XJ1E
6	6	XJ1F
7	7	XJ1G

Path Sheet: - 000107, - 000107, - 000107, - 000107, - 000107, - 000107, - 000107





Shredded int.
London 1234
Bene project in SEE ELECTRICAL

XJ1, XP1

Project: 1-Example 1-1/2diana
Date: 01/07/2012

Drawing no. 00 001
Function: LOCATION: Pages 1

The *Connector plan* is created following the same rules as the *Terminal Row Picture*, but of course, it shows connectors and their pins.
More information about creating of templates you can find in the Manual.

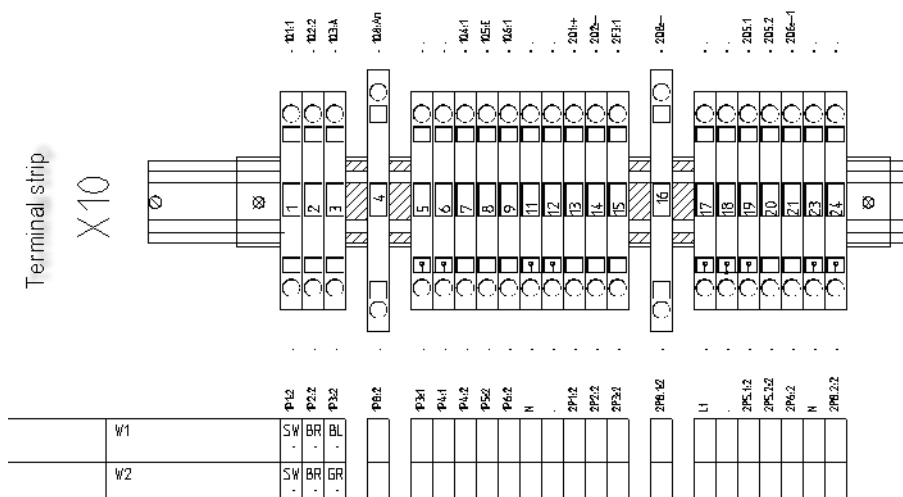
KK TERMINAL ROW PICTURE PLAN

(Advanced)

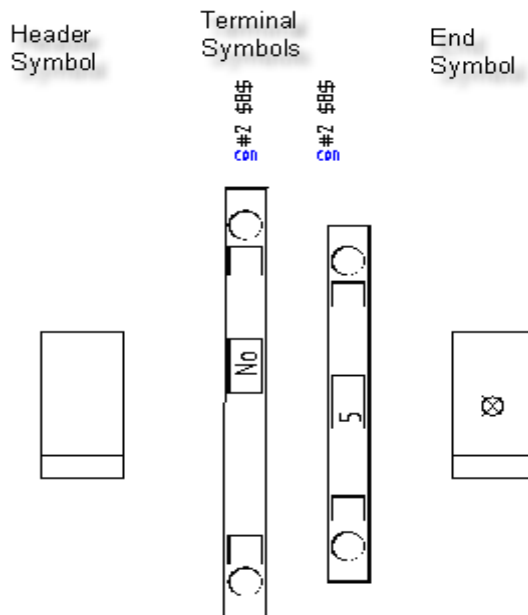
KK.1. GENERATING A TERMINAL ROW PICTURE PLAN

The *Terminal Row Picture Plan* enables you to insert a specific symbol for each terminal. For example, if single terminals in a terminal strip are mixed with multi-layer terminals, you can see directly which these are. In addition, a diode terminal can be displayed in a different way than a Switching terminal, etc.

Example:



It is possible, by means of a Header, to show additional information in front of the terminal as the one shown above the cover plate of the terminal strip. The End symbol accomplishes the same.



You can specify in three ways which symbol must be used for displaying the terminal:

Via the terminal type:

In the *Type database*, you can assign the symbol by selecting the "Symbol name for terminal plan (multi level)" property or select the symbol by means of the **Symbol** browser.

You can also determine, in addition to the terminal symbol, which Header symbol and which End symbol to be used in the Terminal plan (multi level). The definition must be formed as follows:
+T="<Terminal Symbol Name > +H="<Header Symbol Name>" +E="<End Symbol Name>".

If you do not wish to place a "Header Symbol", you can type +H0 behind the name of the terminal symbol for the *Terminal plan* (multi level), therefore +T="<Name>" +H0.

If you wish to give, before each terminal, the "Header Symbol" indicated in the Symbol for the *Terminal plan* (multi level), type +H1, therefore +T="<Name>" +H1.

If you wish to give, behind each terminal, the "End symbol" indicated in the Symbol for the *Terminal row picture*, type +E, therefore +T="<Name>" +E.

Via the terminal symbol in the Circuit diagram:

You can define a symbol by the "Symbol name terminal plan" text. The symbol is searched in the TERMINALROWPICTURE symbol library of the "Terminalplan" symbol folder. If it must be searched in another location, please specify <Symbol database>\<Folder>\<Symbol name>.

This symbol is also used if another symbol is assigned by the type.

You can also determine, in addition to the terminal symbol, which "Header Symbol" and which "End symbol" must be used in the Terminal plan (multi level). The definition must be formed as follows:
+T="<Terminal symbol Name> +H="<Header Symbol Name>" +E="< End Symbol Name >". (If you wish to insert Header and End Symbols here, you must always indicate the Symbol for the Terminal Row Picture plan, too!)

If you do not wish to place a Header Symbol, you can type +H0 behind the name of the terminal symbol for the Terminal Row Picture plan, therefore +T="<Name>" +H0.

If you wish to give, in front of each terminal, the Header Symbol indicated in the Symbol for the Terminal Row Picture plan, type +H1, therefore +T="<Name>" +H1.

If you wish to give, behind each terminal, the End Symbol indicated in the Symbol for the *Terminal Row Picture* plan, type +E, therefore +T="<Name>" +E.

If you want to assign, behind the last symbol of a terminal strip, the "*End Symbol*" defined in the Symbol for the *Terminal Row Picture* plan, and if the symbol for the *Terminal Row Picture* plan is given in the Type, then fill in here, in the Circuit diagram, only an +E for this symbol.

In this way, you can assign the symbols for the Terminal row picture to the terminal types and then set the end plate flexible via +E at the symbol, or accomplish varying definitions.

Default Symbol

If a symbol name is found neither in the terminal symbol nor in the type, the "0Terminal" symbol from the *TERMINALROWPICTURE* Symbol database is inserted.

Exercise 33-1: Generate the Terminal Row Picture Plan for the training workspace.

1. Select *Terminal Row Picture Plan* in the **Graphical lists** area of the **Workspace Explorer**.
2. Right-click with the mouse.
- 3.CO **Generate**
The **Select Terminal Row** dialogue appears.
- 4.> Choose the terminal strip(s) for which you wish to generate a terminal row picture plan.
By default all terminal strips are selected.
- 5.> Click **OK**.
The *Terminal Row Picture Plan* is generated.

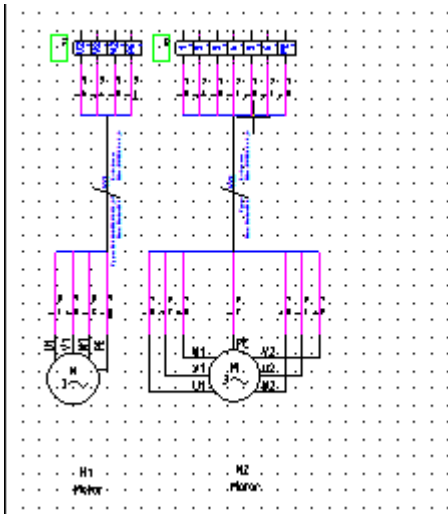
Exercise 33-2: View the Terminal Row Picture Plans. A separate sheet is created for each terminal strip.

1. 0001
Select sheet 1 of the *Terminal Row Picture Plan* by double-clicking on 0001 beneath **Terminal Plan (multi level)** in the *Workspace* tree.
2. 0002
Select sheet 2 of the Terminal Row Picture Plan by double-clicking on 0002 beneath **Terminal Row Picture Plan** in the **Workspace Explorer**.

LL CABLE TERMINAL ROW PLAN

(Advanced)

The *Cable Terminal Row Plan* is a plan which allows you to generate various views of the cables and the terminals in the circuit diagram, according to the selected template.



For further details on how to generate a page template for the *Cable Terminal Row Plan*, please consult the "Graphical Lists" chapter of the Help files.

MM PRODUCT ASSEMBLY LIST

(Advanced)

This list gives a complete overview of the entire component by collecting all master and slaves information and presenting all information in one symbol.

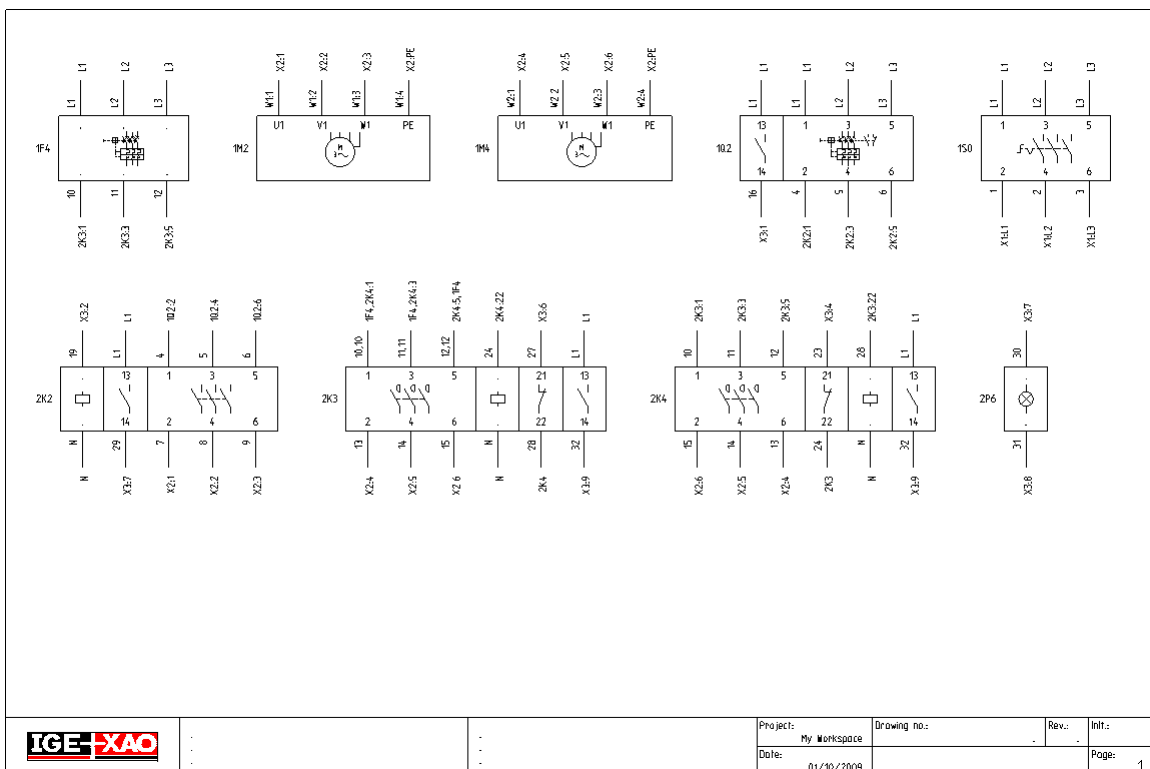
The *Product Assembly* list is generated using the same symbols for the components as used in the circuit diagrams. A rectangle is created automatically around the symbol.

For the components which consist of master and slave symbols, like relays for example, the master and slave symbols are grouped together in a common rectangle. The sorting order inside this group depends on the sort order set in the channel definition for the assigned type. Items not present in the diagram are also shown but with empty wiring information.

If no channel information exists, the individual items are arranged in the order in which they are inserted into the diagram.

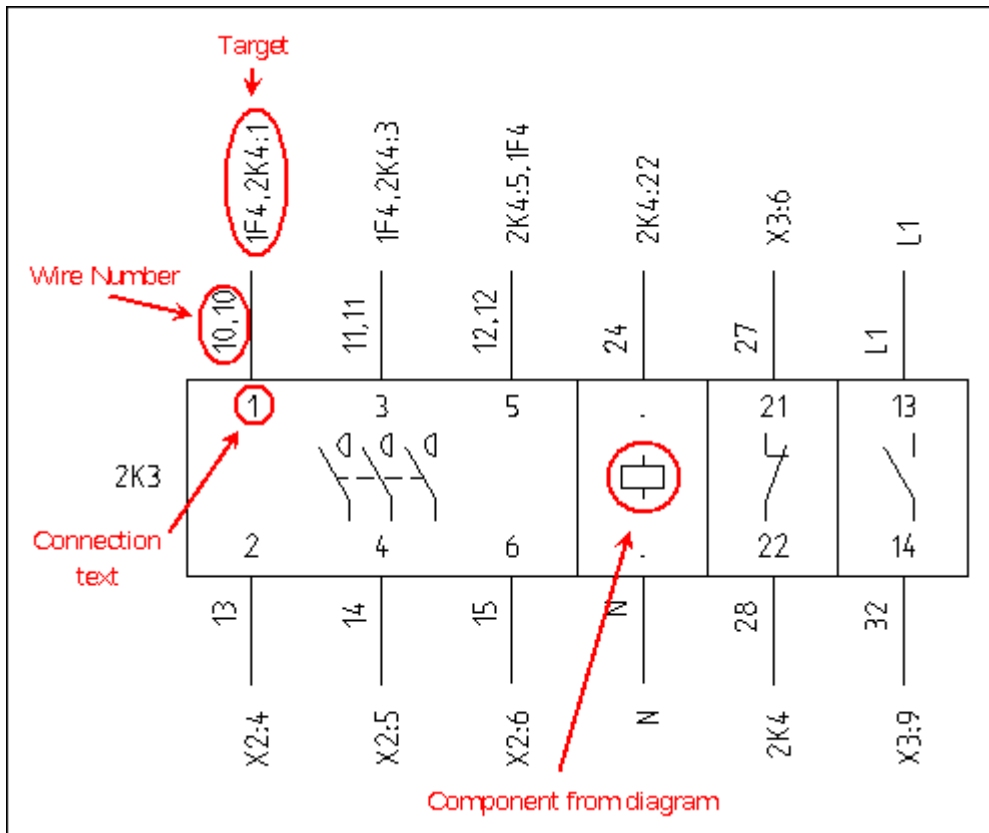
If wire numbers are defined in the circuit diagram, they will appear on the wires in the assembly list as well.

Example:



Training manual

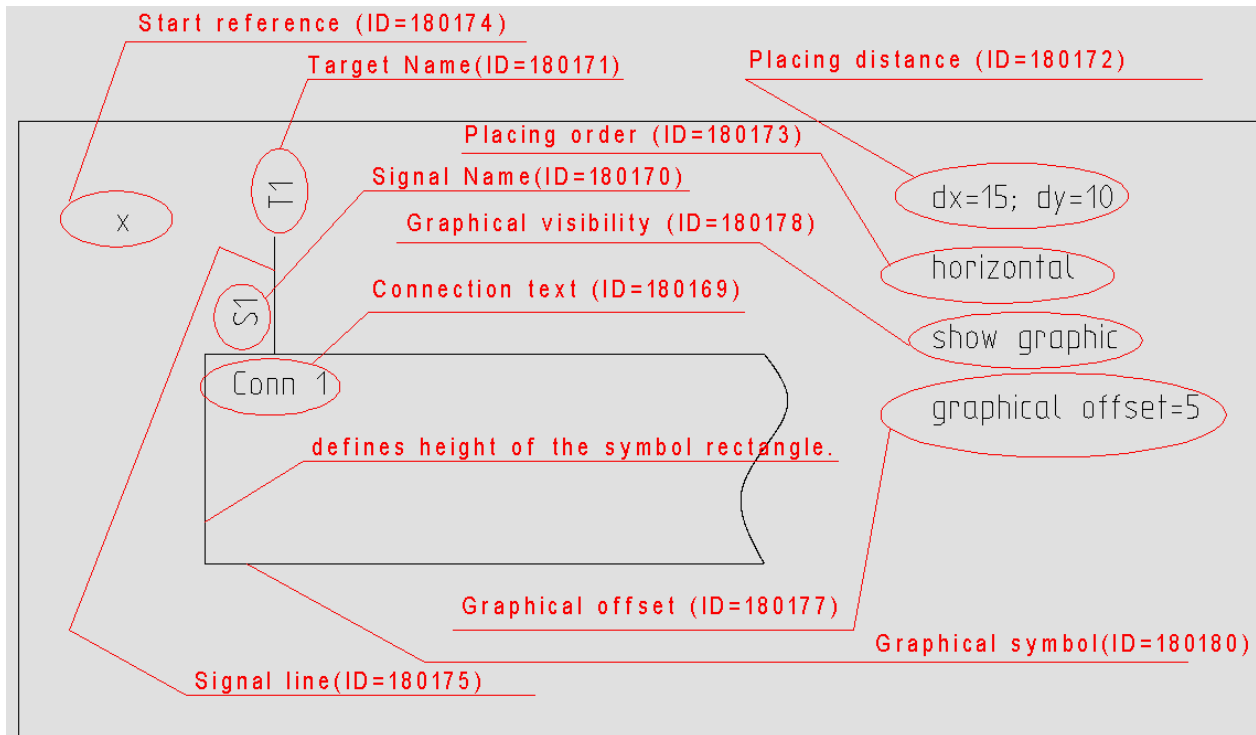
COPYRIGHT © 2013 IGE+XAO. All rights reserved



The page template controls the distance, the orientation and the layout of the generated graphical list.

MM.1. GENERATING A TEMPLATE FOR THE PRODUCT ASSEMBLY LIST

Besides the ordinary placeholder texts in the page template, the following special placeholder texts are necessary:



You find all the different text attributes in the "**Other**" area in the text attributes list.

- Add a text with the attribute "Start reference" (ID 180174), for example "X". This text defines the start point for inserting the component symbols. It also controls the area used for placing the components.
"ex=420; ey=50" or "endx=420; endy=50" defines the (absolute) x and y value of the right bottom edge.
"dx=410; dy=205" defines the area to the reference point of the "Start reference" text.

Note :

*You get exactly the same component if the right / bottom edge is on the wrong side of the "start reference" text (no rectangle inside the drawing boundaries). If no right edge for the placing is defined in the template, the right bottom edge is calculated from the "X- and Y- Extension of Page" properties: right side of drawing area = (max. x-coordinate of drawing) - 2*0.03* (max. x-coordinate of drawing) and bottom side of drawing area = (min. y-coordinate of drawing) * 0.101*

- Add a text with the attribute "Placing distance" (ID 180172), for example dx=15; dy=10. This text defines the distance between two components in x- and y- direction.

- Add a text with the attribute "*Placing order*" (ID 180173), for example "*horizontal*". This text defines the order of placing of the components. The key words can be "*horizontal*" or "*vertical*". If it is "*horizontal*", the placing of components starts from left and continues to the right, until the row is full. Then the placing continues in the next row. "*Horizontal*" is the default value.

If "*vertical*" is chosen, the placing of components starts on top and goes to the bottom until the column is full. Then the positioning continues in the next column.

- Add a text with the attribute "*Graphical visibility*" (ID=180178), for example "*show graphic*".

This text determines if the graphical symbols are shown inside every component or not. In order for the graphical symbol to be displayed, the content of this placeholder has to be "*show graphic*". If this is not the case, the graphical symbol will not be shown.

- Add a text with the attribute "*Graphical offset*" (ID=180177), for example "*graphical offset=5*".

This text defines the offset from the boundary rectangle to the graphical symbol. The software looks for the "=" sign and takes the value after it to generate the offset.

- Add texts to control the position of the component name, description and type

If no definitions for component name, function and location are made, the texts appear at their default positions.

- Add a text with attribute "product (-)" to the block for the component inside the template if you want to define the position of the component name or the look of the text yourselves (the attribute is found in the *Component* section in the **Text Attributes**).

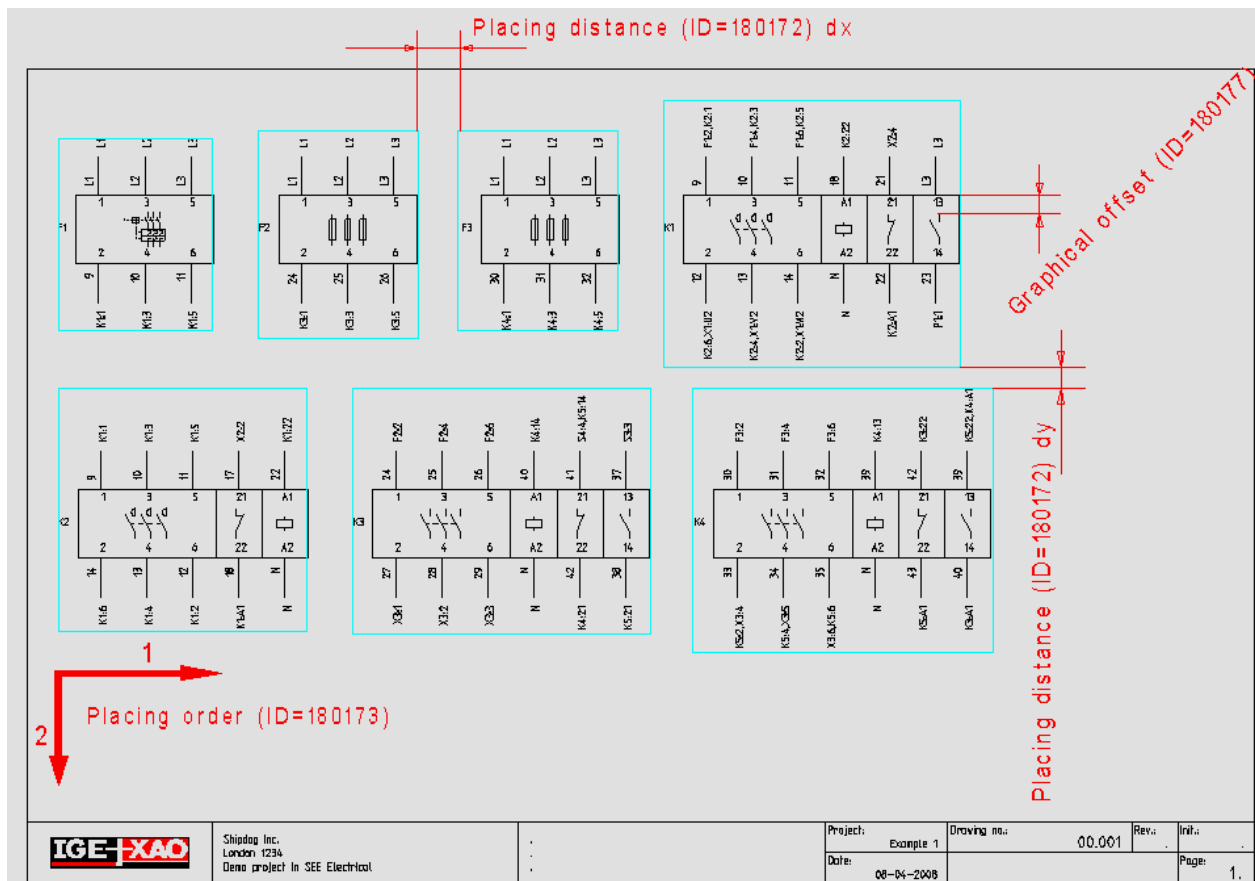
You have to define the position of the component name if you want to define description, type and / or function / location.

- Add a text with attribute "description 00" to the block for the component inside the template (the attribute is found in the *Component* section in the **Text Attributes**)
- Add a text with attribute "type" (the attribute is found in the *Component* section in the **Text Attributes**). If more than one type is defined in the component, all types will be shown like type1; type2, etc.
- Add texts with attributes "function" and "location" (attribute is found in the *Function & Location* section in the **Text Attributes**)
- Add a text with the attribute "*Connection text*" (ID=180169), for example "*conn 1*". This text defines the attributes for the connection point name and its position inside the symbol.
- Add a text with the attribute "*Signal Name*" (ID=180170), for example "*S1*".

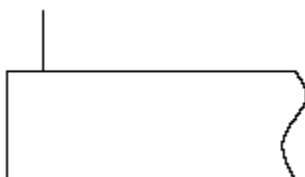
This text defines the attributes for the signal name and its position inside the symbol. It can be used to add more information about the wire. The value of the text defines the information which is shown. You can position it max 6 times in the template and use the following codes:

- ✓ "+N" --> Cable-core or wire number (default flag)
- ✓ "+C" --> Cable-core or wire colour
- ✓ "+c" --> Cable-core or wire colour and number if colour is empty
- ✓ "+S" --> Cable-core or wire size
- ✓ "+Q" --> Cable-core or wire signal
- ✓ "+P" --> Cable-core or wire potential

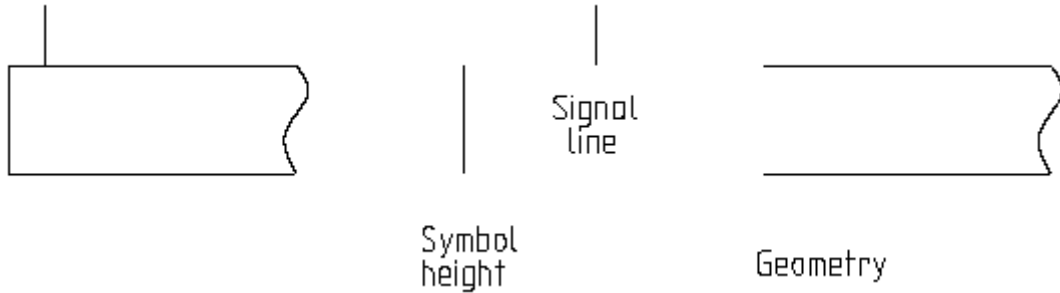
- Add a text with the attribute "Target Name" (ID=180171), for example "T1". This text defines the attributes for the target name and its position inside the symbol.



Graphical symbol



The graphical symbol contains 3 parts:



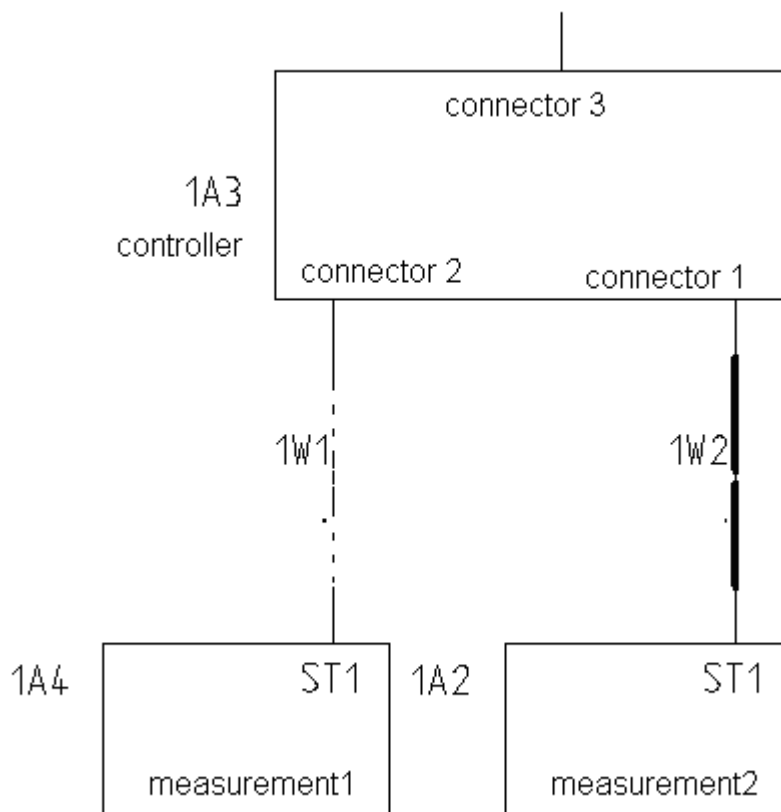
- Draw a line (ID 1) and block it as "*Signal line*".
The line has to be strictly vertical. This line defines the length of the connection lines. (If in the parent group of this line there are other elements, they will be placed in the symbol, too.)
- Draw a line (ID 1) and block it as "*Symbol height*".
The line has to be strictly vertical. The line defines the height of the symbol.
- Draw geometry by using the **Draw > Elements > Line** command (part of a rectangle).
This geometry is used to generate the rectangle for the components.
- Block the 3 parts as "*Block, Macro, Group*".

NN MULTICORES

(Advanced)

NN.1.USING MULTICORES

Multicores are used for drawing a wire that must contain multiple cables, for example, if you have to go from one circuit board to another circuit board upon connectors.



Multicores can be displayed in the special database lists Multicores and Multicore-wires

Exercise 36-1: Define a wire as a multicore wire.

- 1.CA **Electrical**
- 2.CO **Multicore (Multicore panel)**
- 3.+ Identify the wire you wish to define as a multicore.
The multicore receives a name.
You can define another wire as a multicore. Do not do this now. Right-click to exit the multicore drawing mode.

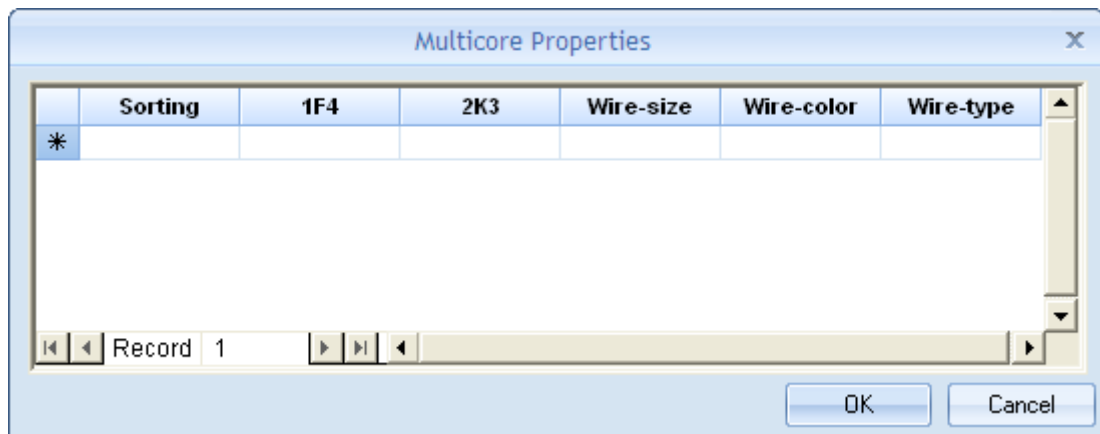
Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

4. Double-click the text of the Multicore name. You can enter a description, type and length of the multicores. You will find the corresponding data in the list of multicores afterwards.
5. You can change the pen style and pen width, if you wish. In this way, you can highlight the multicores graphically. Use the functions already known.

Exercise 36-2: Assign wires to the multicores.

- 1.CA **Electrical**
- 2.CO **Properties (Multicore panel)**
- 3.+ Select the multicores that you wish to assign single wires to and edit the wires.
The following dialogue box appears:



- 4.> Click in the line to add wires.
Define the wires sorting, type in the connection of the first component, the connection of the second component, wire-colour, wire-size and wire-type.
To delete a wire, select the line and press the **Delete** key.
- 5.> After you have defined all the wires in the multicore, click **OK** to close the dialogue box.

Exercise 36-3: View the Multicores and Multicores-wires lists within the Graphical lists.

1. Select **Multicores** in the **Graphical lists** area of the Workspace Explorer.
2. Right-click with the mouse
- 3.CO **Generate**
The Multicores graphical list is generated.
Double-click the page 001 in the *Workspace Explorer* to open it.

00 INSERTING COMPONENTS NOT IN THE DRAWING VIA THE DATABASE EDITORS

(Advanced)

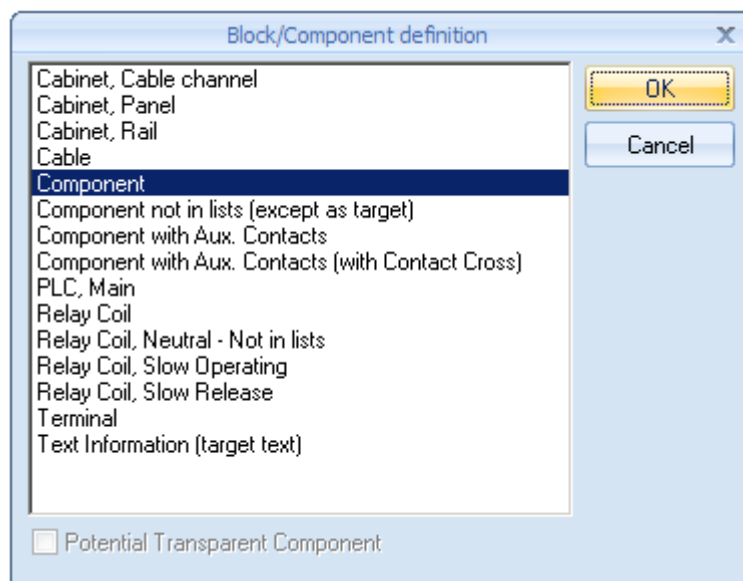
You can insert components for the Part list or spare terminals via the database editors. It is not necessary any more to insert them into the diagrams. You can also import such components without graphic with the help of an Excel file.

00.1. EDITOR FOR COMPONENTS WITHOUT GRAPHICS

This Editor contains information about components without graphics in the workspace (for example, spare terminals, separation and end plates for terminals, inserted via the Terminal Editor; a box for shipping of the designed machine, added via the Component Without Graphic Editor, etc.). It allows you to insert manual components that do not have any graphical representation and are not inserted in the circuit diagram pages.

Quick guide: Inserting components without graphics

1. Open the Component without graphic Editor.
2. Right click and select the **Add new component** command.
The following dialogue appears:



3. Select the desired type for the element without graphics and click **OK**.
The element is inserted in the database list, even if it does not exist in the circuit diagram.
You can modify its properties in the right pane of the Editor.

Hint

If you right-click on a row containing a component and you execute the **Add new component** command, the new component will be a copy of the existing one. To create a different kind of component, right-click the headline of the Component Without Graphic Editor.

In the right pane of the Editor you can:

- ✓ assign a name (Function (=), Location (+), and / or Product (-)) to a component without graphic
Make sure the "Amount" value is 1, otherwise you generate duplicated components.
If you want to create a component, spare terminal or cable with a name already existing in the diagrams, the entry will not be created.
- ✓ add an amount to a component without graphic (for example add 1000 labels)
Make sure not to define a component name in this case, otherwise you generate duplicated components.
- ✓ add new component properties to the components without graphic

4. Select a row on the left pane of the window.
5. In the right pane, open the pull-down list of properties in the last row.

	Text Value
Product (-)	
Description 00	
Type	
Amount	1
Description 01	
Description 01	
Description 02	
Description 03	
Description 04	
Description 05	
Description 06	
Description 07	
Description 08	
Description 09	
Description 10	
Description 11	
Description 12	

6. Type a text value for the property and press Enter.
A new row appears automatically, allowing you to add more component properties.

Notes :

1. You can also add terminals without graphic using this Editor. For further details about creating terminals, see also **Error! Reference source not found.**
2. You can also add cables without graphic using this Editor. For further details about creating cables, see also **Error! Reference source not found.**

To delete a component not in graphic:

1. Select the element.
2. Right-click and select the **Delete Selected** command.
3. Click **OK** to confirm.

Notes:

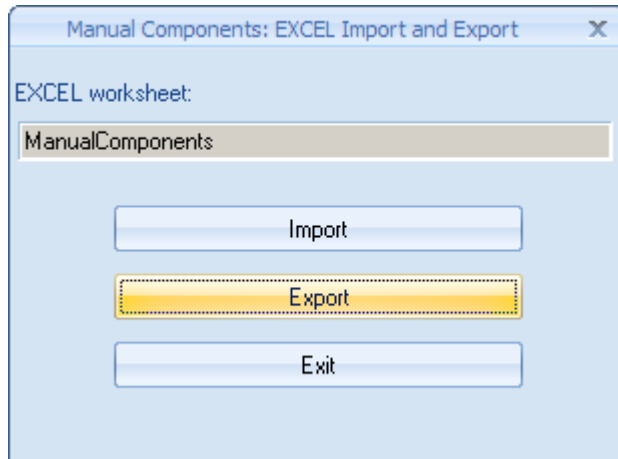
1. *If you wish to insert these components in the circuit diagram, you can do that through the **Pick List** command in the **Functions** category. Make sure you have defined a database type and a graphic for the element in the channel definition of the type. If in the channel definition no symbol is defined but connection texts and a size for the symbol are set, a black box is generated. If nothing is defined, a rectangle with one connection point is generated when the component is inserted.*
2. *When you position a component without graphic into a diagram, it receives a name in accordance with the IEC 60757 norm, depending on its type (set during its creation) or the type assigned to it from the Type Database. If no specific type is defined (the symbol is simply created as "Component") and no type is assigned from the Type Database, then the name from the Editor is used (the Product (-) field). When you insert a component into a circuit diagram page, it disappears from the Component Without Graphic list.*
3. *If a cable is inserted via the Picklist, you have to follow the rules for drawing cables. If you want to insert cable cores at different places, you can take the first cable core from the Picklist, but after that you have to draw the other cable cores via the **Electrical > Cable > Cable** command, as the cable disappears from the Component Without Graphic list, as soon as the first core is inserted.*

Quick guide: Importing/Exporting manual components from an Excel file

A pop-up command in the Components without Graphic Editor allows you to import/export Excel files, containing information about components without graphics (manual components).

1. Right click in the **Components without Graphic Editor** and activate the **Excel - Import/Export** command.

The following dialogue appears:



2. Click the **Import** or the **Export** button and select the file to process.
3. Click **Exit** to quit the command.
Close and reopen the database editor for the modifications to appear.

Hint:

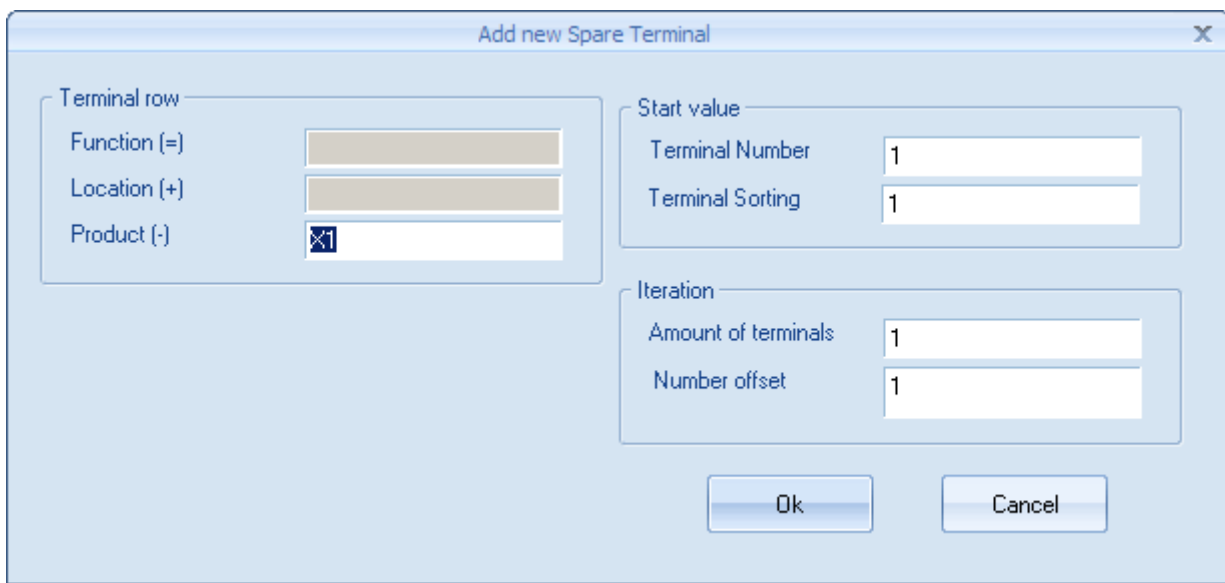
The Excel file used needs a very specific format. We recommend that you create some components without graphic of the desired type and export the list to Excel. Then you can use this file as a master file, put data into it and import it. The first row of the Excel sheet always contains a headline (descriptions of the columns) and can be empty. The second row contains the IDs for identification of the columns. Do not change this row. The third row contains the definitions for the components.

00.2. INSERTING SPARE OR GROUND TERMINALS VIA THE TERMINAL EDITOR

A function allows you to insert spare/ground terminals in the database.

Quick guide "Insert spare terminals":

1. Right click a line in the Terminal Editor list.
 2. Select the **Add new component** command.
- The following dialogue appears:



3. This window allows you to easily define a lot of spare terminals in a single operation.
4. Add the name of the terminal strip to which the spare terminals will belong. (Function (=), Location (+) and Product (-)).
4. Define the first terminal number and sorting in the "Terminal number" and "Terminal sorting" fields.
With "Number offset" you define the way, in which terminal number (if numerical value is given) and terminal sorting are to be increased.
Please keep in mind that a terminal is uniquely specified by a terminal strip, a terminal number and a terminal sorting. If a terminal already exists, it is not generated in here. It is not possible to define the single levels of multilevel terminals here.
5. Type in the desired values and click **OK**.
The spare terminals are inserted in the database list. They also are added to the graphical terminal lists at that position defined by their terminal sorting.

For further details about importing data from an *Excel* file, see also **Error! Reference source not found..**

Note:

You can insert these terminals in the Circuit diagram as well, by using the **Pick List** command from the **Functions** category.

OO.2.1. HANDLING CABLES WITHOUT GRAPHICS

Quick guide: Inserting cables without graphics:

1. Right click a line in the Cable Editor list.
2. Select the **Add new component** pop-up command.
3. Type in the desired values into the input fields on the right pane.

If you want to add a name for the cable, the name must be unique. If you do not define a name, you can define an amount of cables.

The amount of cable cores is not managed in the case of a cable without graphics. This means that when you insert at least one core in the circuit diagram, the cable disappears from the list of components without graphics. The same happens if you rename an existing cable in the diagram with the name of a cable existing only in the Editor.

Excel import:

When cables are imported from an *Excel* file, and a cable with the same name already exists, the new imported cable receives an extension to its number, separated by "#". For example, if a cable W1 already exists, the "W1" cable imported gets the new name "-W1#001" (if a second cable "W1" is imported, it gets the name "W1#002").

Behaviour in the Pick List:

Let us assume that W1 and W2 are cables without graphics in the *Pick List*.

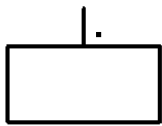
If you select W1 from the Pick List and change the name to W2, the cable W1 is taken from the Pick List when you select it and the cable W2 is taken from the Pick List, because it is not found in the diagram.

You will never find cables without graphics in the List of Cable-cores, because the cores of these cables do not have targets.

00.3. INSERTING SYMBOLS FOR COMPONENTS WITHOUT GRAPHICS IN THE DIAGRAM (PICK LIST)

If you have defined "Components without graphic" (see *Database lists, Editor, Components without graphic*), you can position the elements via the pick list.

If a symbol for the circuit diagram has been assigned to the type in the channel definition, the given symbol is inserted. If no symbol has been assigned, *SEE Electrical* automatically creates a symbol with one connection point.



When you position a component without graphics into the diagram, it receives the name you have defined in the Editor. If you have not defined a name in the Editor, it gets an automatic name at the insertion. As soon as you position it in the diagram, it disappears from the *Component without graphic List*.

If a cable is inserted via the *Picklist*, you have to follow the rules for drawing cables. If you want to insert cable cores at different places, you can take the first cable core from the Picklist, but after that you have to draw the other cable cores via the **Electrical > Cable > Cable** command, as the cable disappears from the *Component Without Graphic list*, as soon as the first core is inserted.

PP AUTO DIAGRAM

(Advanced)

PP.1. INTRODUCTION

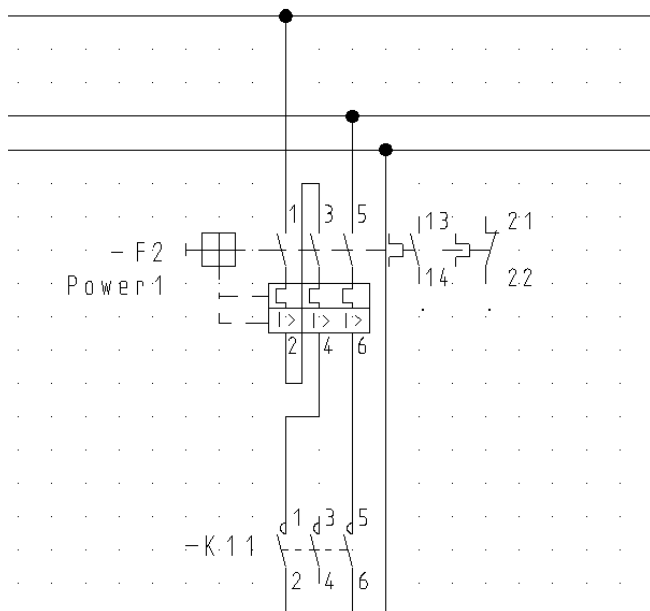
The automatic generation of circuit diagrams is executed by using special symbols (groups) and a *Microsoft Excel* spreadsheet.

To use the Autodiagram function, you need to have knowledge about *Microsoft Excel* and about working with *SEE Electrical*, especially about creating symbols and page templates.

PP.2. CREATING SYMBOLS (GROUPS)

You can create symbols easily as they include regular components, their texts and wires and two additional text placeholders to indicate the start-point and the end-point of the group.

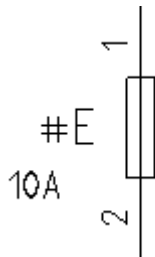
1. Create the groups as usual by using components and wires or insert an existing group, ungroup it, if it is a fixed group (if the elements have been grouped using **Edit ► Block, Macro/ Group**).



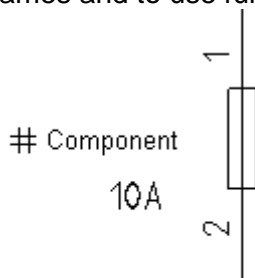
Component name, description and type must be replaced with texts from the Excel spreadsheet by executing the Autodiagram function for the automatic generation of circuit diagrams. Replace the texts at the component with texts from text placeholders.

You can define placeholder texts in one of the following ways:

2. Specify the column from the *Excel* spreadsheet where the right identification is included (if the column E includes the component name, then a placeholder with format #E is required instead of the component name F2).

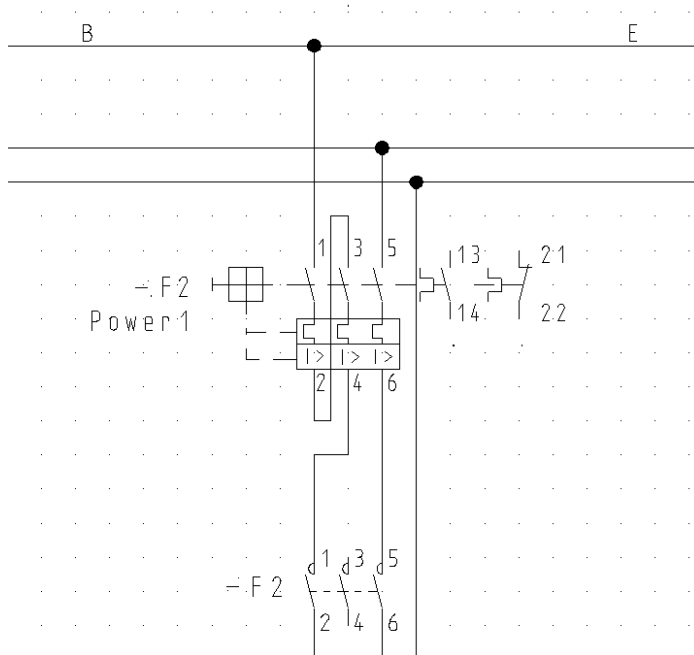


3. Specify an Alias name
 Alias names allow entering descriptive names instead of the column names, such as #Component instead of #E. You can assign the names of the *Excel* columns to the alias-names in the Alias sheet within the *Excel* spreadsheet.
 By using alias-names, the *Excel* columns can be changed without changing the texts in the components groups. In this case, the Alias-sheet must be adapted.
 The use of alias-names facilitates the implementation of text placeholders. However, it is not compulsory to use alias-names.
 If you work with alias-names, you have to choose free identification for the component names and to use function/location.



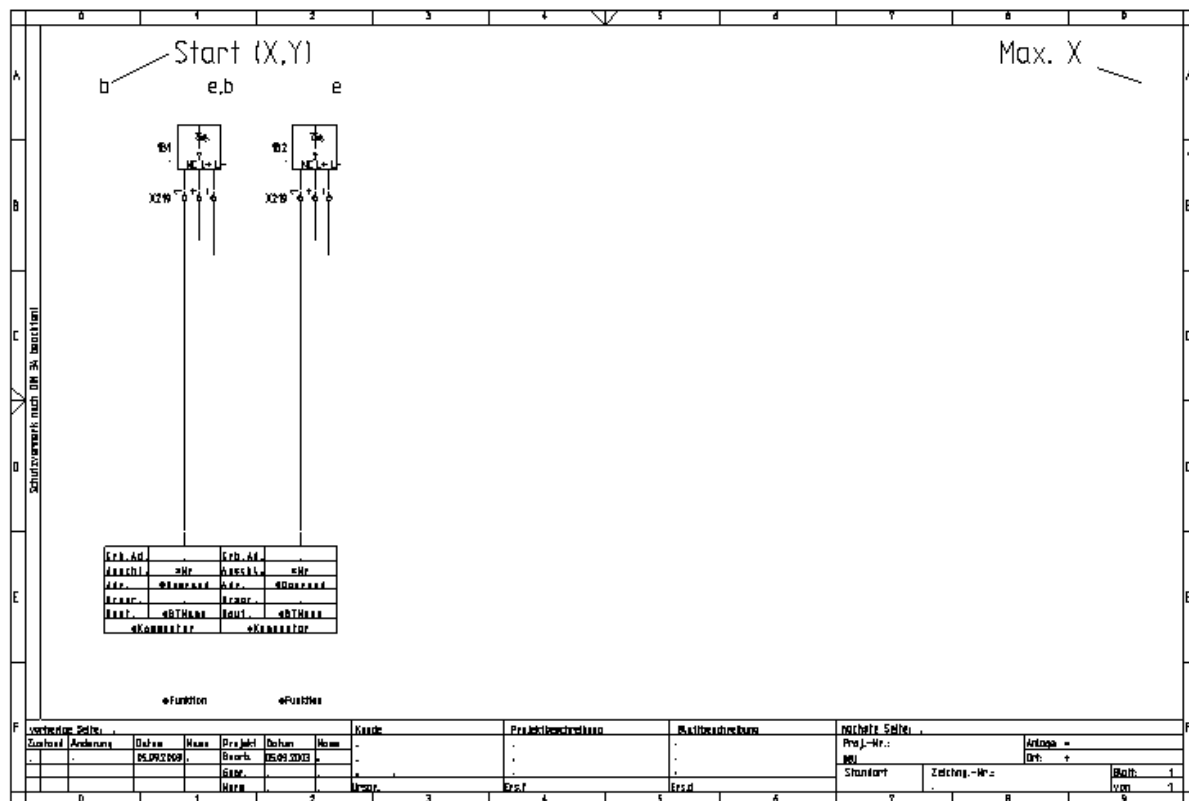
4. Each group needs a marker for the start-point and a marker for the end-point. Over the end marker of the first group, the start marker of the next group is placed, etc. The groups follow each other in a chain. The programme can evaluate whether a group fits into one page. If not, a new page is automatically created.

5. Use the marker B (Text > Attribute > Other: "**Symbol Start Marker**") and E (Text > Attribute > Other: "Symbol End Marker"). Place appropriate texts (select the **Draw > Elements > New text** command – you will find the text attributes in the "**Attribute**" field under **Other**).



6. Create a group (Select the objects and execute the **Edit > Actions > Block > Block,Macro,Group** command). If the group includes relay coils, do not select the contact mirrors or the contact crosses within the frame.
7. Save the groups in the symbol database which you use only for the automatic generation of Circuit diagrams (select the group or the single objects and drag it into the symbol database – drag the point of the group where the letter B is placed). All of the groups that could be combined with each other must be saved in the same folder of the symbol database.

The circuit diagram is constructed as follows:



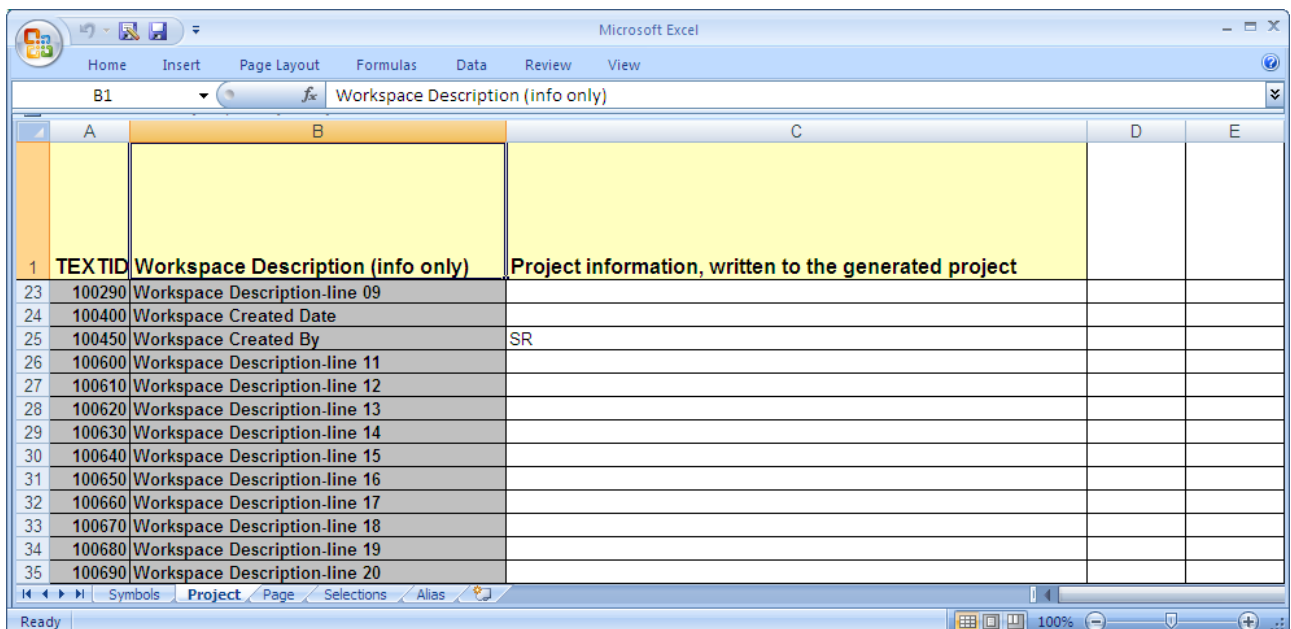
PP.4. EXCEL SPREADSHEET

The *Excel*-spreadsheet includes multiple sheets with different functions. You can change the names of the sheets. The illustration is made by using the included *Excel* spreadsheet *PLC.XLS*.

The *Excel* columns are indicated with letters and the rows – with numbers. Fields are indicated as a combination of the corresponding column name and row name.

PP.4.1. PROJECT DATA (PROJECTINFO)

This area contains information entered in the **Project information** window. If the norm sheet contains appropriate text placeholders, then the texts are displayed in the circuit diagrams as well.

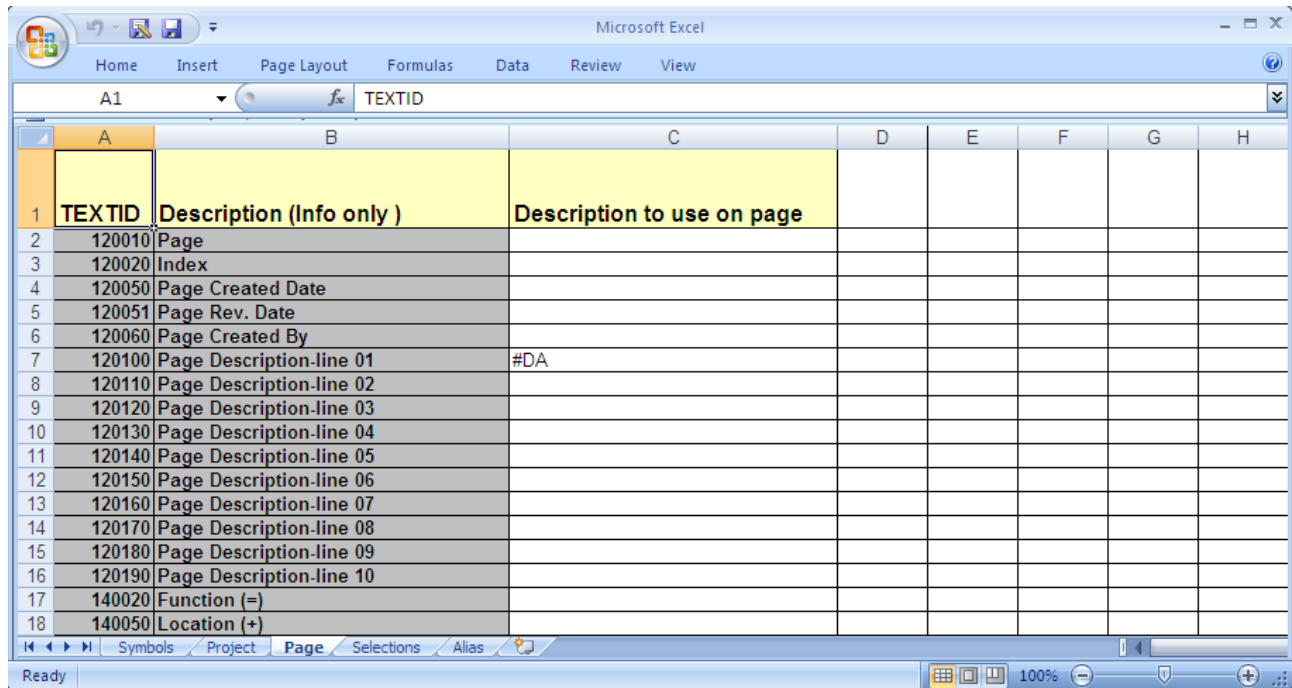


	A	B	C	D	E
1	TEXTID	Workspace Description (info only)	Project information, written to the generated project		
23	100290	Workspace Description-line 09			
24	100400	Workspace Created Date			
25	100450	Workspace Created By	SR		
26	100600	Workspace Description-line 11			
27	100610	Workspace Description-line 12			
28	100620	Workspace Description-line 13			
29	100630	Workspace Description-line 14			
30	100640	Workspace Description-line 15			
31	100650	Workspace Description-line 16			
32	100660	Workspace Description-line 17			
33	100670	Workspace Description-line 18			
34	100680	Workspace Description-line 19			
35	100690	Workspace Description-line 20			

Do not change the "**TEXTID**" column in any case.

PP.4.2. PAGE DATA (PAGEINFO)

This area contains information entered in the **Page information** window for each page. If the applied norm sheet contains appropriate text placeholders, then the texts are displayed in the circuit diagrams, too.



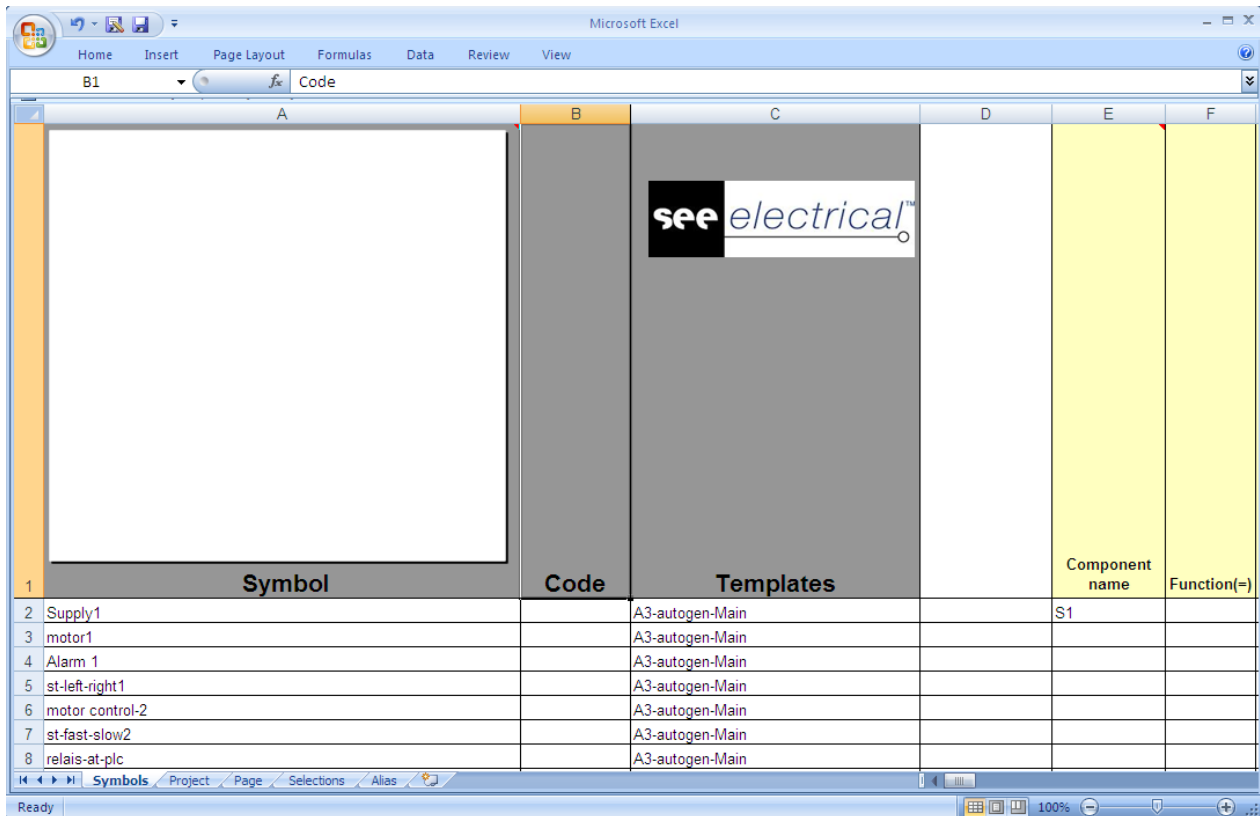
	A	B	C	D	E	F	G	H
1	TEXTID	Description (Info only)	Description to use on page					
2	120010	Page						
3	120020	Index						
4	120050	Page Created Date						
5	120051	Page Rev. Date						
6	120060	Page Created By						
7	120100	Page Description-line 01	#DA					
8	120110	Page Description-line 02						
9	120120	Page Description-line 03						
10	120130	Page Description-line 04						
11	120140	Page Description-line 05						
12	120150	Page Description-line 06						
13	120160	Page Description-line 07						
14	120170	Page Description-line 08						
15	120180	Page Description-line 09						
16	120190	Page Description-line 10						
17	140020	Function (=)						
18	140050	Location (+)						

Do not change the "**TEXTID**" column in any case.

PP.4.3. SELECT SYMBOL (SYMBOLS)

Specify the groups to be used in the current project.

Specify also whether you want to insert special texts, for example, in the component names and also a description or a type. You can do this if appropriate text placeholders exist in the group (see topic "Create symbols" above in this chapter).



	Symbol	Code	Templates	Component name	Function(=)
1					
2	Supply1		A3-autogen-Main	S1	
3	motor1		A3-autogen-Main		
4	Alarm 1		A3-autogen-Main		
5	st-left-right1		A3-autogen-Main		
6	motor control-2		A3-autogen-Main		
7	st-fast-slow2		A3-autogen-Main		
8	relais-at-plc		A3-autogen-Main		

Add the groups to be used and delete the unnecessary lines.

Column A (Symbolname):

The column includes the names of the existing symbols. This name must be available in the *SEE electrical* symbol database.

Column B (New page):

If you type the letter "s", a new page will be created while inserting the group indicated in this line regardless of whether the new group fits into the current page or not.

Column C (Page template):

Define the page template to use, if a new page is required.

Next columns:

Here, you can define texts for the components. The texts will be inserted by using the automatic generation function if corresponding text placeholders are available in the groups.

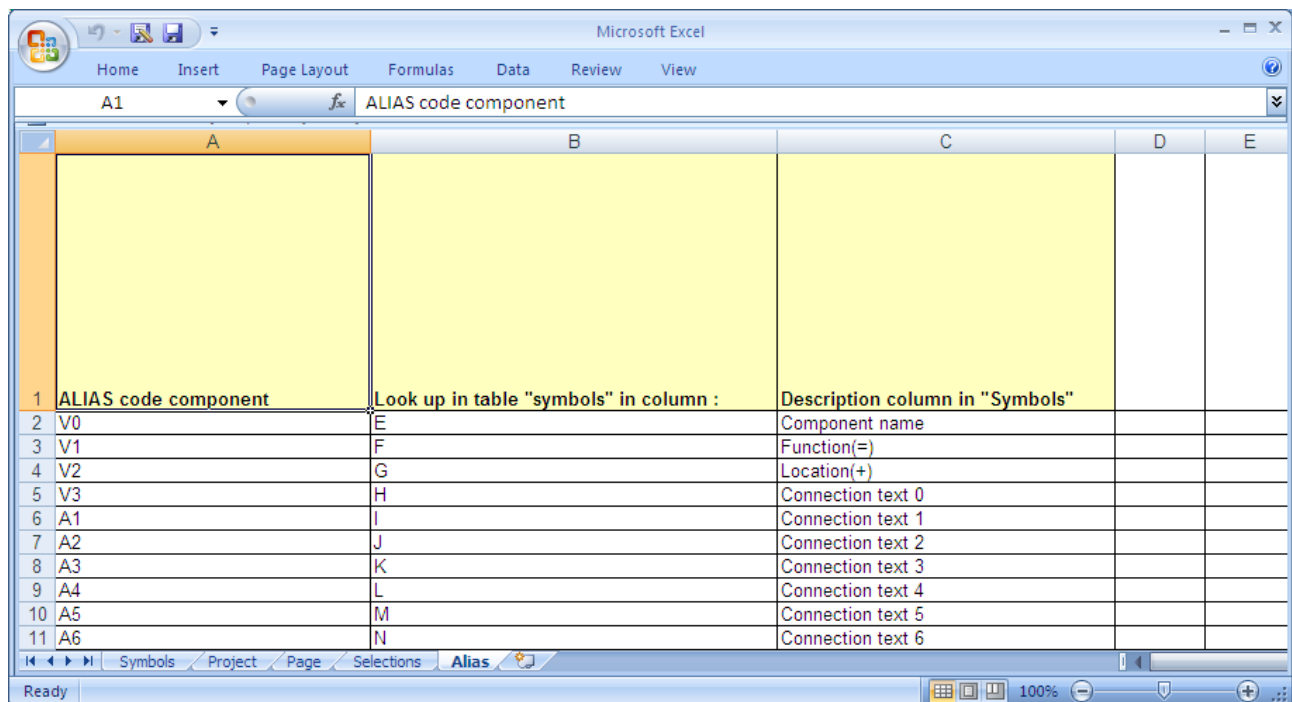
PP.4.4. DEFINING ALIAS NAMES

The groups contain text placeholders, for example for component names etc. that will be filled in with data from the corresponding columns of the *Excel*-sheet Symbols. This text placeholders must be defined as <column identification>, i.e. E, F, etc. You can remember E difficult for the component identification.

Alias names allow entering descriptive texts instead of the column names, for example "*Component Name*" instead of "E", etc. The assignment of the internal column names of the Excel-program to the descriptive alias names is executed in the alias table.

Working with alias names is useful since all of the components can be provided with text placeholders with the Component Name format. It will be easy to change the assignment of the internal column names to the descriptive alias names. If you use text placeholders of the <Column identifier> format in the groups, you can change the assignment only by changing the symbols.

Using alias names facilitates the use of text placeholders, too. Using alias names is not compulsory.



	A	B	C	D	E
1	ALIAS code component	Look up in table "symbols" in column :	Description column in "Symbols"		
2	V0	E	Component name		
3	V1	F	Function(=)		
4	V2	G	Location(+)		
5	V3	H	Connection text 0		
6	A1	I	Connection text 1		
7	A2	J	Connection text 2		
8	A3	K	Connection text 3		
9	A4	L	Connection text 4		
10	A5	M	Connection text 5		
11	A6	N	Connection text 6		

PP.5. AUTOMATIC GENERATION OF CIRCUIT DIAGRAMS

Requirements

Create a new project, for example AUTOMATIC.

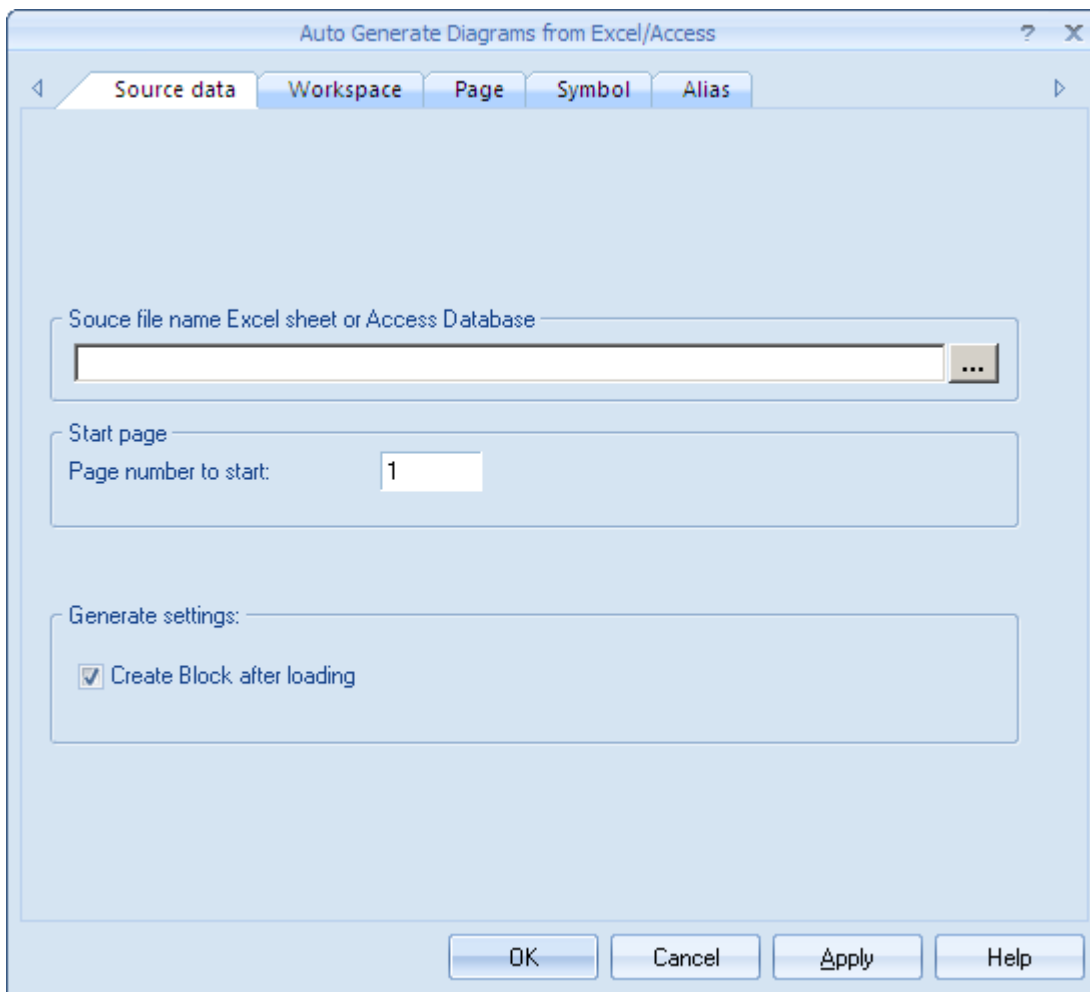
If you use an existing project, it will be overwritten, i.e. all existing data will be lost!

Hint:

You can save an empty workspace with a page 1000 as a workspace template.

PP.5.1. THE AUTODIAGRAM COMMAND

If the project is created and the page 1000 is active and visible, launch the **Auto diagram** command. You will find the command in the pop-up menu available for the module "Circuit Diagrams".



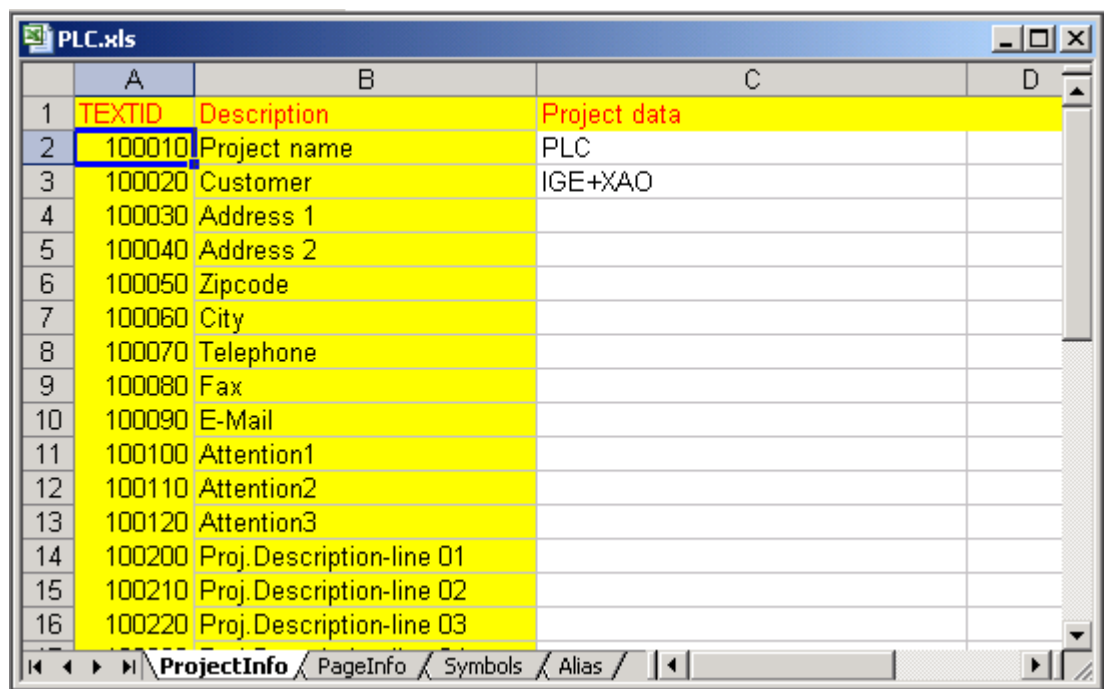
- Enter the required settings (find more details below).
- Click the **OK** button to start the automatic generation of the circuit diagram.

The circuit diagrams will be generated.
The settings are stored within *Windows* and are available for the next execution of the function.

Source data tab

Enter first the sheet of the *Excel*-file and the fields where the data for your project come from.

The included *Excel*-file *PLC.XLS*, sheet **Project data** contains example data:



	A	B	C	D
1	TEXTID	Description	Project data	
2	100010	Project name	PLC	
3	100020	Customer	IGE+XAO	
4	100030	Address 1		
5	100040	Address 2		
6	100050	Zipcode		
7	100060	City		
8	100070	Telephone		
9	100080	Fax		
10	100090	E-Mail		
11	100100	Attention1		
12	100110	Attention2		
13	100120	Attention3		
14	100200	Proj. Description-line 01		
15	100210	Proj. Description-line 02		
16	100220	Proj. Description-line 03		

Source file name (*Excel/Access*)

Source file name Excel sheet or Access Database

Enter here the name of the *Excel*- or *Access*-file, that comprises the defaults about the automatic generation of circuit diagrams.

First page

Start page

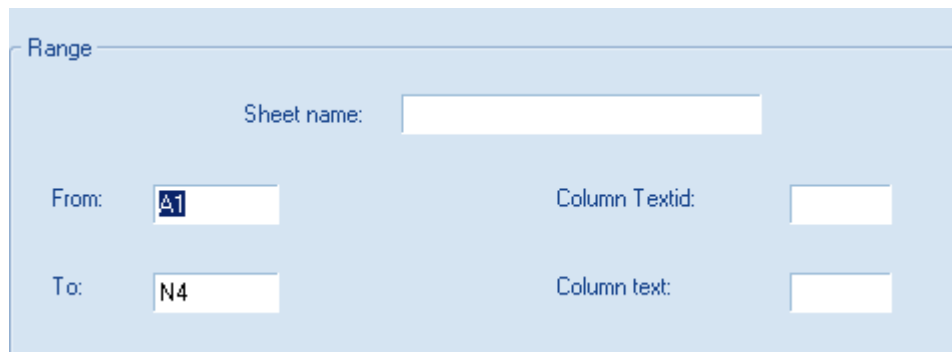
Page number to start:

You specify here from which page the automatic project engineering must start.

Attention:

No circuit diagrams must be present after the defined start page number in this field – in such case, they will be deleted.

Workspace information tab



From:

The first field from the Excel-sheet that contains data about the project.

To:

The last field from the Excel-sheet that contains project information.

Sheet name:

For example: *Project data*, name of the Excel-sheet to be used; if you wish to change the sheet names of your Excel-table, you must then type valid names.

Column TextID:

Type the letter of that column of the Excel-sheet containing the TextID data.

Column Text:

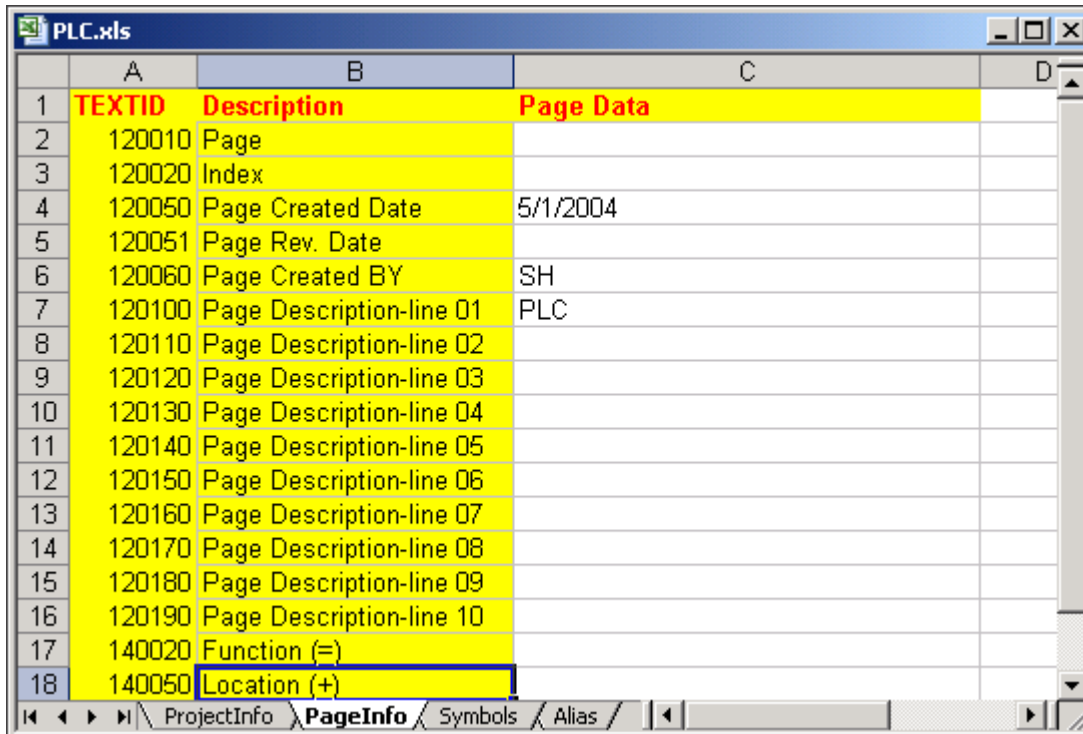
Type the letter of the column of the Excel-sheet that contains the entries about the workspace information.

Page information tab

- Enter first the sheet of the *Excel*-file and the fields on this sheet where the page data come from.

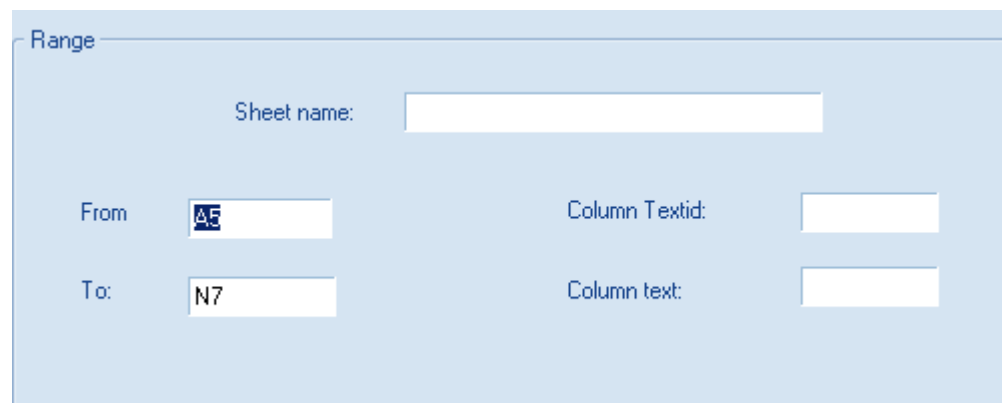
All the pages of the project receive the same information.

Example data are illustrated in the Excel-file PLC.XLS, sheet Page Data:



	A	B	C	D
1	TEXTID	Description	Page Data	
2	120010	Page		
3	120020	Index		
4	120050	Page Created Date	5/1/2004	
5	120051	Page Rev. Date		
6	120060	Page Created BY	SH	
7	120100	Page Description-line 01	PLC	
8	120110	Page Description-line 02		
9	120120	Page Description-line 03		
10	120130	Page Description-line 04		
11	120140	Page Description-line 05		
12	120150	Page Description-line 06		
13	120160	Page Description-line 07		
14	120170	Page Description-line 08		
15	120180	Page Description-line 09		
16	120190	Page Description-line 10		
17	140020	Function (=)		
18	140050	Location (+)		

Entries in the **Page information** tab:



Range

Sheet name:

From:

To:

Column Textid:

Column text:

From:

Type in the first field of the *Excel*-sheet that contains data for page information.

To:

The last field from the *Excel*-sheet that contains data for page information.

Sheet name:

For example: Page Data, name of the *Excel*-sheet; if you wish to change the sheet names in your *Excel*-table, you must type valid names.

Column TextID:

Enter the letter of the column in the *Excel*-table that contains the TextID-records.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

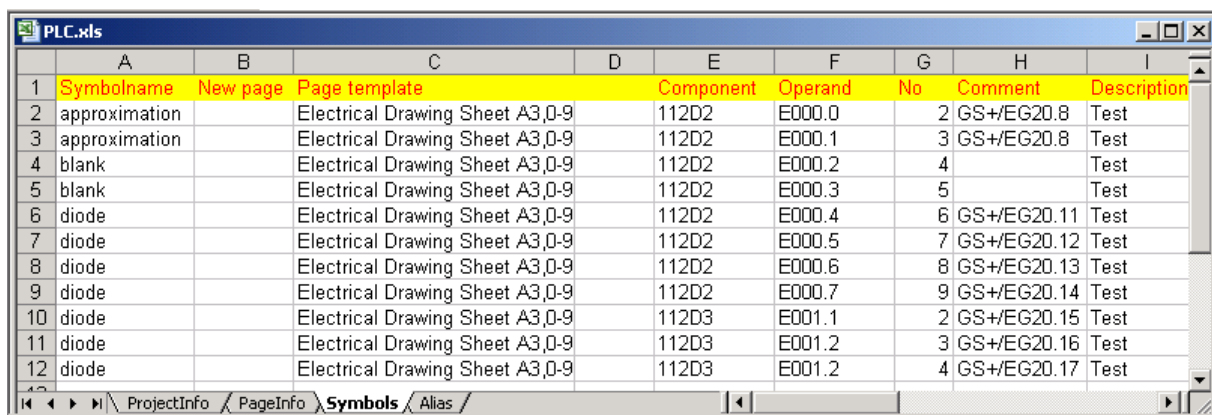
Column Text:

Enter the letter of the column in the *Excel*-table that contains the records about Page information.

Symbol data range

Type the sheet of the *Excel*-file and the fields on this sheet where the data come from about the groups to be placed in.

Input data are illustrated in the *Excel*-file *PLC.XLS*, sheet **Symbols**:



	A	B	C	D	E	F	G	H	I
1	Symbolname	New page	Page template	Component	Operand	No	Comment	Description	
2	approximation		Electrical Drawing Sheet A3,0-9	112D2	E000.0	2	GS+/EG20.8	Test	
3	approximation		Electrical Drawing Sheet A3,0-9	112D2	E000.1	3	GS+/EG20.8	Test	
4	blank		Electrical Drawing Sheet A3,0-9	112D2	E000.2	4		Test	
5	blank		Electrical Drawing Sheet A3,0-9	112D2	E000.3	5		Test	
6	diode		Electrical Drawing Sheet A3,0-9	112D2	E000.4	6	GS+/EG20.11	Test	
7	diode		Electrical Drawing Sheet A3,0-9	112D2	E000.5	7	GS+/EG20.12	Test	
8	diode		Electrical Drawing Sheet A3,0-9	112D2	E000.6	8	GS+/EG20.13	Test	
9	diode		Electrical Drawing Sheet A3,0-9	112D2	E000.7	9	GS+/EG20.14	Test	
10	diode		Electrical Drawing Sheet A3,0-9	112D3	E001.1	2	GS+/EG20.15	Test	
11	diode		Electrical Drawing Sheet A3,0-9	112D3	E001.2	3	GS+/EG20.16	Test	
12	diode		Electrical Drawing Sheet A3,0-9	112D3	E001.2	4	GS+/EG20.17	Test	

Symbol information tab entries:

Range

Column Page template:

X

Sheet name:

Column module name:

B

From:

A9000

To:

Column code:

L

Symbol database/folder

Database:

AutogenEN.ses

Folder:

E:\programs\SEE Electrical\Symbols

Module positions:

☒ Use module begin and end markers

Start X:

50

Start Y:

250

Max X:

405

☐ Use X and Y position

Column for X position

Column for Y position

From:

Type in the first field of the *Excel*-sheet that contains data about symbols.

To:

The last field in the *Excel*-sheet that contains data about symbols. You could specify a wider range than the just now defined one, in order to avoid changes while adding component groups.

Sheet name:

For example: Symbols, name of the *Excel*-sheet; if you wish to change the sheet names in your *Excel*-table, you must type valid names here.

Column Page template

Enter here the letter of the column of the *Excel*-table that contains the name of the page template that is to be used if needed.

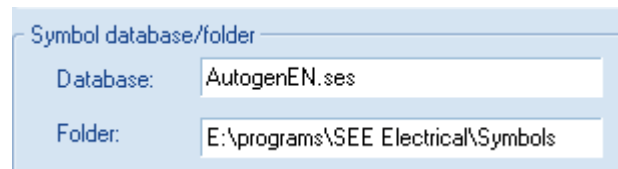
Column module name:

Determine the letter of the column in the *Excel*-table that contains the symbol names.

Column code:

Enter the column letter in the *Excel*-table that contains "s" for page break.

Symbol database/folder area



Symbol database/folder

Database: AutogenEN.ses

Folder: E:\programs\SEE Electrical\Symbols

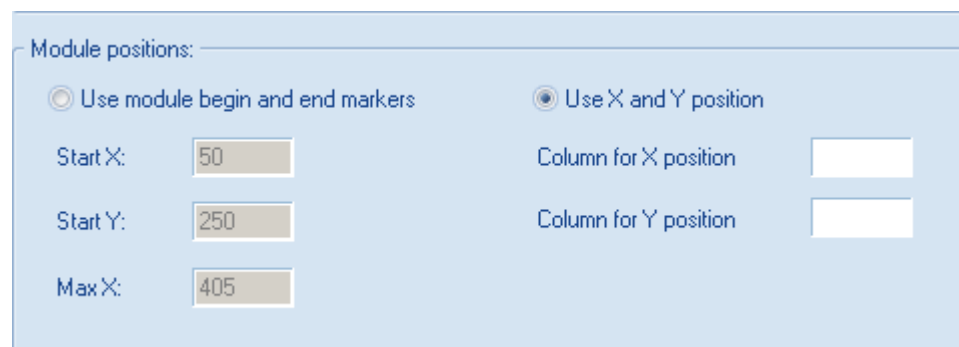
Database:

Name of the symbol database that contains the groups for the automatic generation of circuit diagrams.

Folder:

Folder in the symbol database that contains all the groups needed for the automatic generation of the circuit diagrams.

Module positions area



Module positions:

☐ Use module begin and end markers
 ☒ Use X and Y position

Start X: 50
 Column for X position:

Start Y: 250
 Column for Y position:

Max X: 405

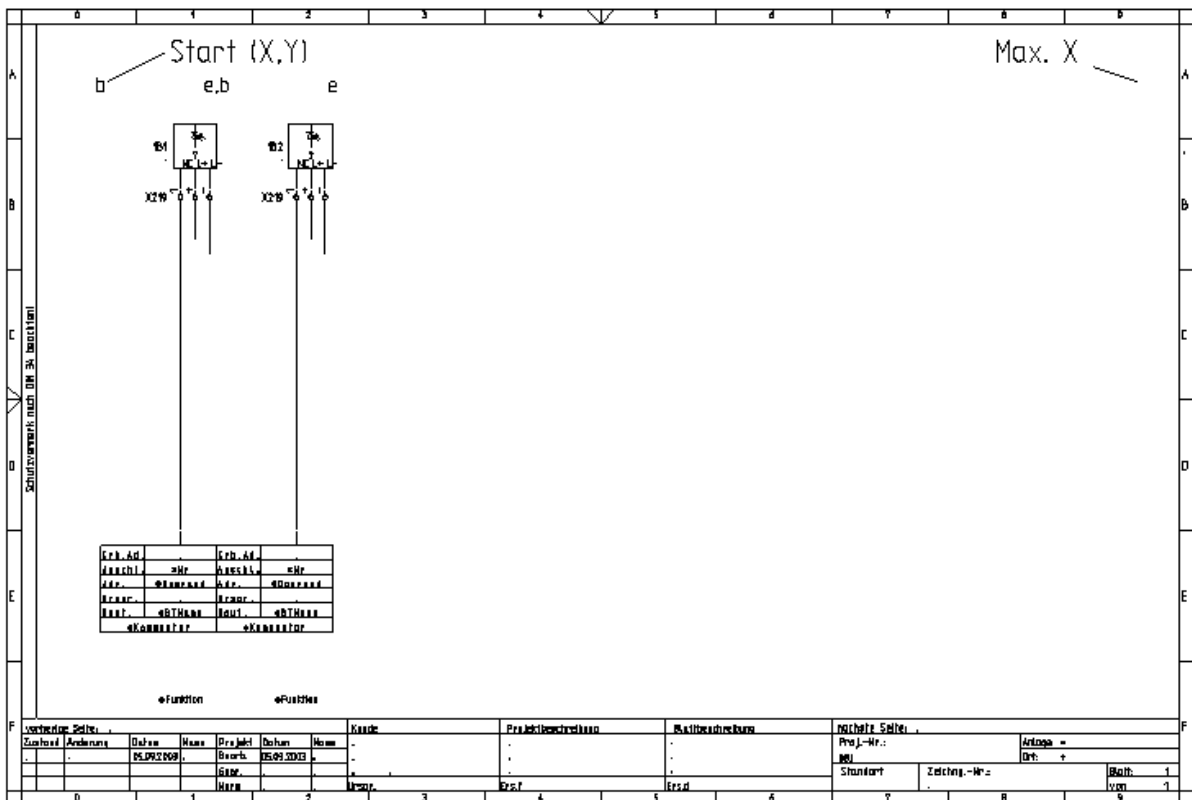
Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

Activate "**Use module begin and end markers**" to determine the coordinates X and Y where the first group must be placed.

The **Max.X** field specifies the maximum area available in the page template. The possibility to place another group in one page depends on your end marker, if it can be placed on a position less than or equal to the value of Max. X. Otherwise, a new page is created.

By selecting the "**Use X and Y position**" option, you can specify the coordinates for the symbol positions from two columns in the *Excel*-list.



You can define the page template in the *Excel*-spreadsheet. The values in the Module positions area must comport with the used page template.

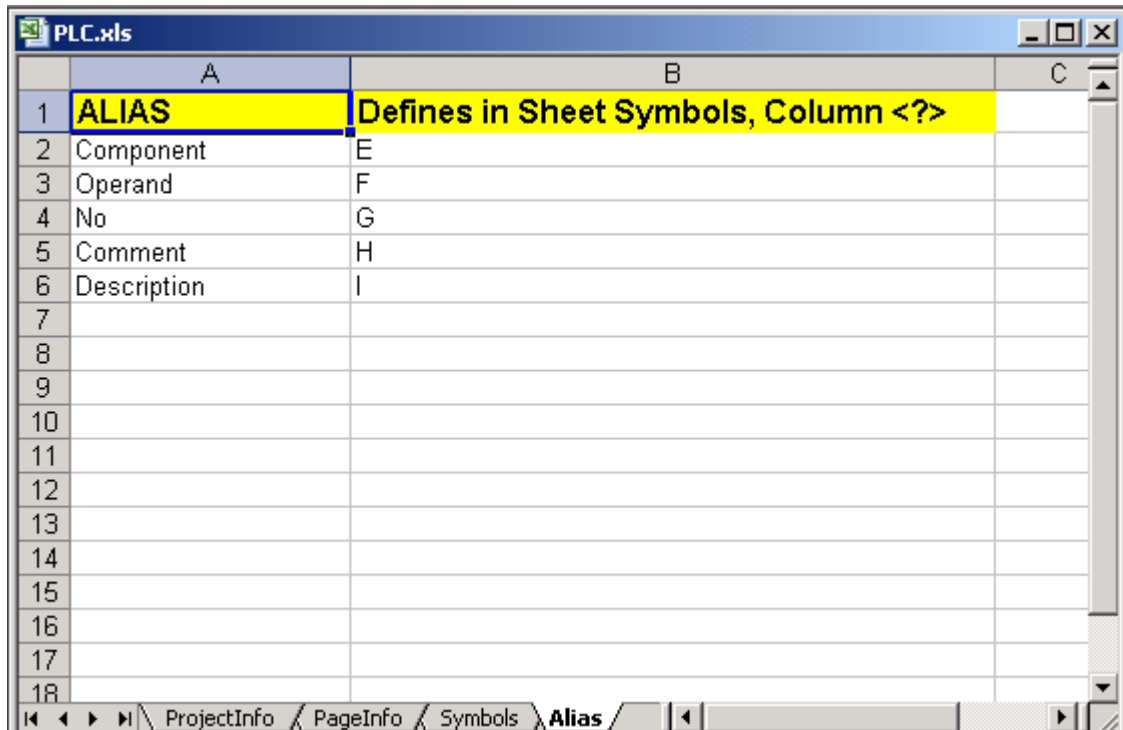
The number of groups that can be placed in one page depends on the size of the page template and on the size of the groups that you must place, furthermore also, if there is an "s" record in the *Excel*-sheet Symbols that causes a page to break or not.

Alias data range

Use alias names for the text reservation fields, define the sheet of the *Excel*-file and the fields on this sheet where the data must come from.

Using alias names is not compulsory.

Data entries are illustrated in the example *Excel*-file *PLC.XLS*, sheet *Alias*:



	A	B	C
1	ALIAS	Defines in Sheet Symbols, Column <?>	
2	Component	E	
3	Operand	F	
4	No	G	
5	Comment	H	
6	Description	I	
7			
8			
9			
10			
11			
12			
13			
14			
15			
16			
17			
18			

Alias information tab:

If something is defined here, it is assumed that you use alias names!

Range, if used

Sheet name:

From
 Column Alias name:

To:
 Column for Alias lookup column:

From:

The first field of the *Excel*-sheet that contains relevant information about alias names.

To:

The last field in the *Excel*-sheet that contains relevant information about alias names.

Sheet name

For example: *Alias*, name of the *Excel*-sheet; if you wish to change the sheet names of your *Excel*-table, you must enter valid names here.

Column alias name:

Enter the letter of the column in the *Excel*-table that contains the alias name (for example #Component Name).

Column for Alias lookup column:

Enter here the letter of the column in the *Excel*-table that specifies in which columns of the *Excel*-spreadsheet the texts for the components must be searched (#T for alias name #Component Name in our example).

QQ LISTS AND LABELS EDITOR

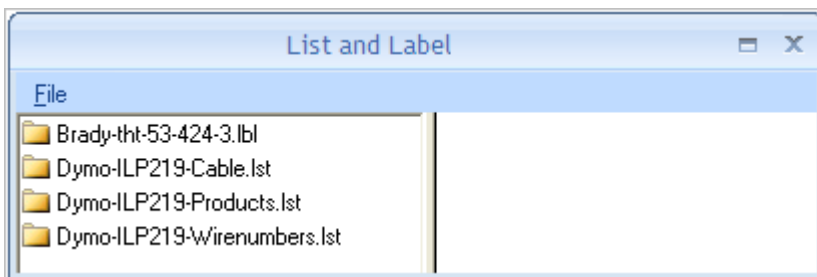
(Advanced)

QQ.1. CREATING LABEL TEMPLATES

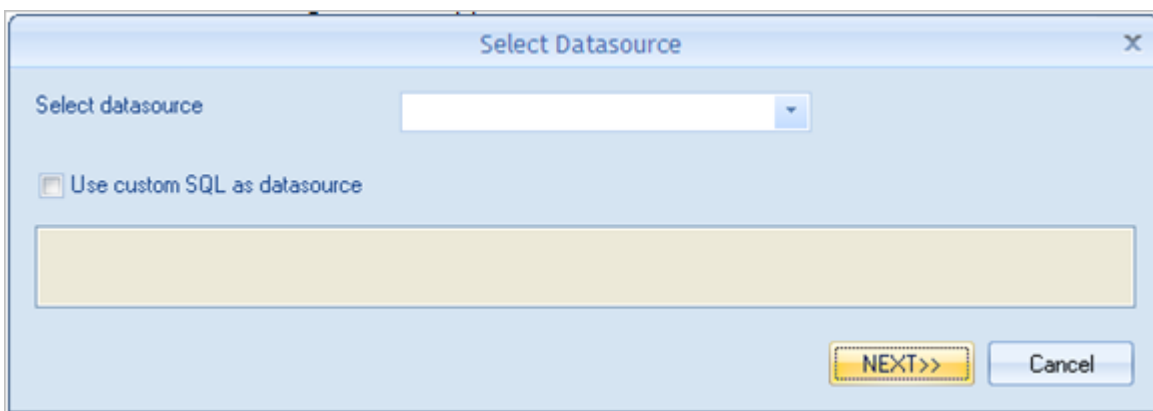
Exercise 39-1: Create a template for labels.

Only one label can be edited. When you print this label, it can be multiplied according to the specified number of lines and columns in the page.

- 1.CA **File**
- 2.CO **List and Label**



- 3.M **File**
Click the **File** menu in the **List and Label** window.
- 4.M **New Label**
- 5.# <name>
Type the name of the new template.
- 6.M **Save**
The following window appears:



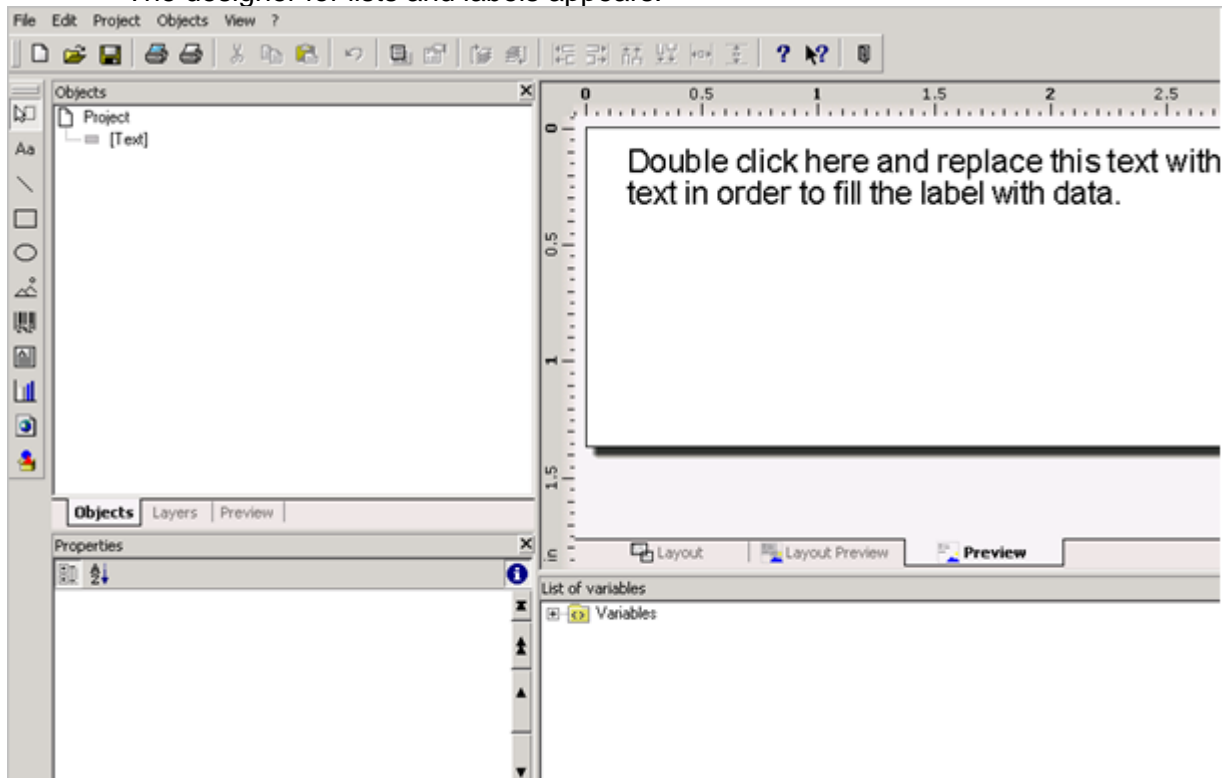
- 7.> Select data source
Select the database, from those included in the *SEE Electrical* database lists, which contains the fields you wish to be recorder in the label. Only one list can be selected.
- 8.> View, Products
- 9.> Next

Training manual


COPYRIGHT © 2013 IGE+XAO. All rights reserved

- 10.> Next
Click **Next** to close the **Project Wizard**.
11. Select the printer and other print options.
- 12.> Next
Close the dialogue box by clicking **Next**.
- 13.> Select the template for your label that you wish to print.
If the desired template for your label is not in the list, you may select **User-defined** and enter the size of the label manually.
Size of the often used labels can be saved in the CMBTL901.INF file in the folder of *SEE Electrical* (you will find the description of this file structure in the User Manual).
By the "**Only show templates for current page settings**" option you can specify whether all templates have to be displayed or not.
- 14.> Next
Close the dialogue box by clicking **Next**.
15. The dimensions of the label are displayed.
Now you can make changes as needed. (You have to correct the respective record in the CMBTL901.INF. file). Adjust the format of your label if you have chosen **User-defined**.
- 16.> Select the desired Print order.
- 17.> Next
Close the dialogue box by clicking **Next**.
- 18.> Next
Close the dialogue box by clicking **Next**.
- 19.> In most cases, it is not necessary to generate a title line for the labels. Uncheck the "Add title line" option.
20. Done!
Close the window.

The designer for lists and labels appears:

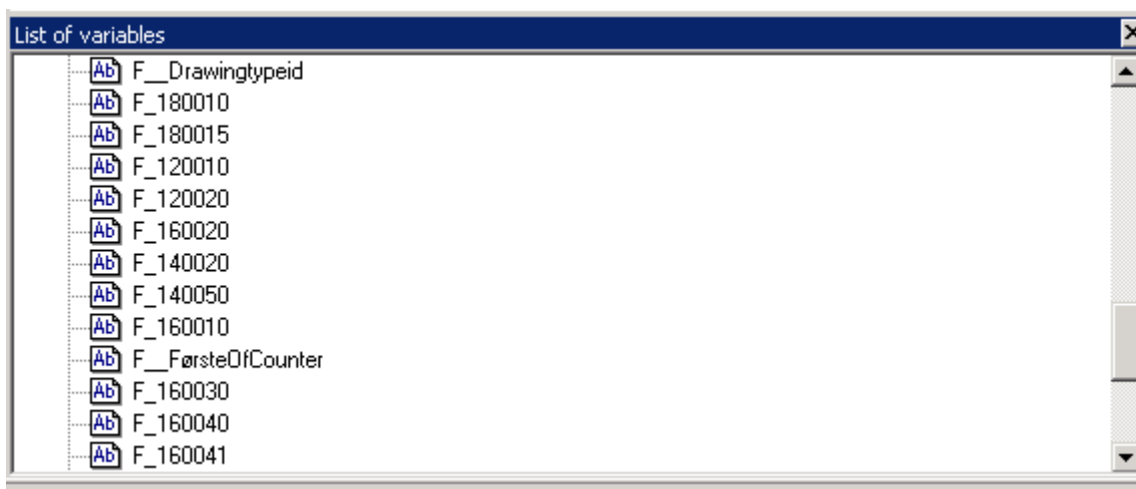


Select the fields to be printed on your label.

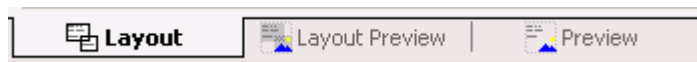
Open the variables list by clicking on the plus sign :



The displayed variables depend on the chosen database list.



- 21.> Click the desired variable.
You can find information about the list of variables in chapter "*Graphical lists*" in the User Manual where each list is described.
- 22.+ Drag the variable into the label area above.
You can define the text area using two opposite points as for a rectangle.
You can move this window or change its size. When defining the size of the window, consider the max. length of the text to be printed.
If **Layout** is active,

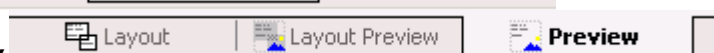


you will see the code number of the text field (for example 160010 for the component name).

If **Layout Preview**



or **Preview**

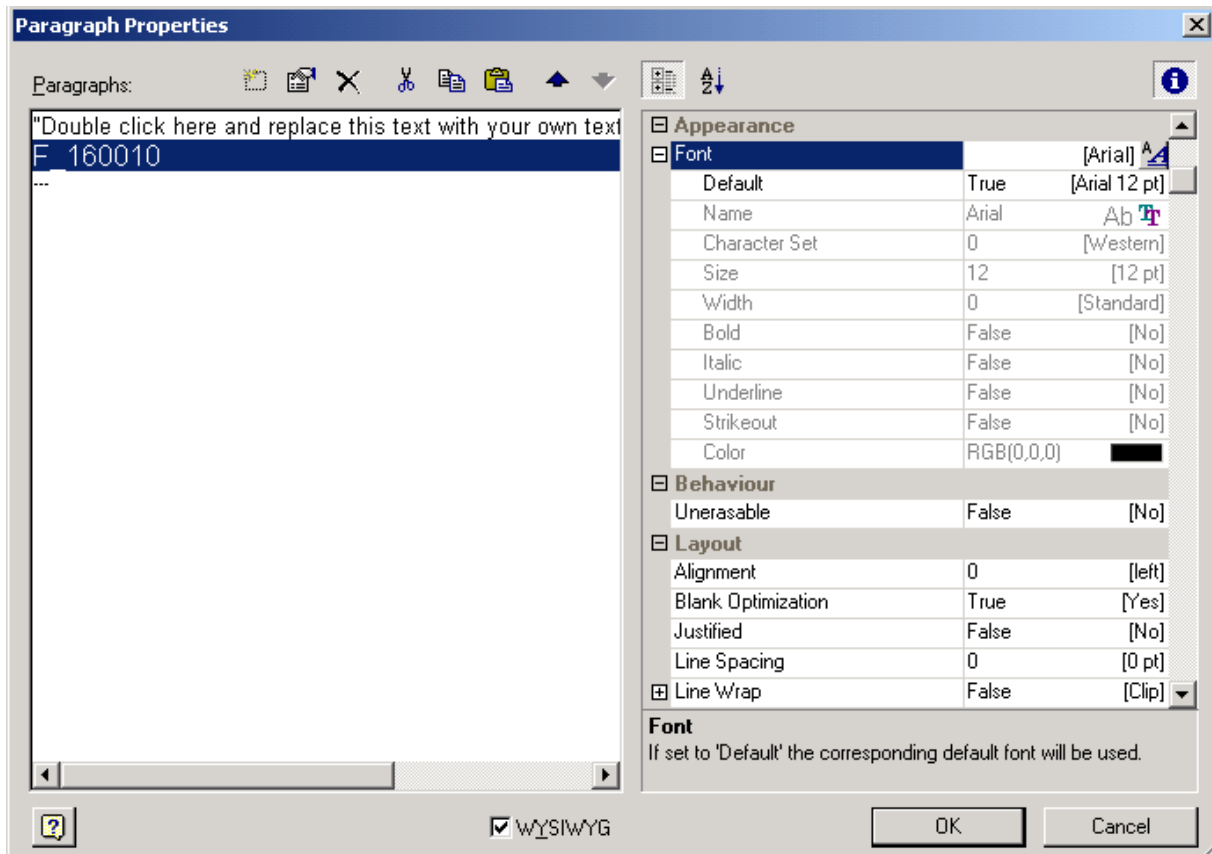


is active, you will see the text description of the field, for example "Component name".

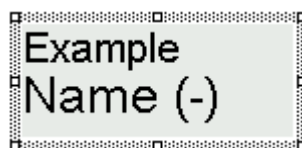
Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

23. Double click the text to define attributes and alignment of the text, for example – centred.
The following window appears:



- 24.> **OK**
Click **OK** to close the window.
25. Choose and place several fields.
Did you place all the fields?
You have another possibility instead of working with blank characters: you can change the size of the available text area via drag points.



- 26.M **File**
27.M **Save**
28.M **File**
29.M **Exit**
Finish the label template designing.
The template can be used for creating labels.

QQ.1.1. FILES FOR LABEL TEMPLATES

The label template always contains 5 CDS, LBL, LBP, LBV and ~LBL files. You must back up these files. If you want to create a new label template using an existing template, copy these 5 files in *Windows Explorer*, a "Copy of *" is created.

Then, you can rename the label template in the start window of the **List and Labels Designer**.

QQ.1.2. CHANGING LABEL TEMPLATES

If changes of the *Label Layout* are required after the label has been created (for example, the result after printing does not match the label or it is not positioned correctly), you can change your label template.

- Open the List and Label Designer, click your label and select Design from the pop-up menu.
- Click Next to reconfirm the chosen database. The List and *Label Designer* appears.
- Choose **Project** ➤ **Page setup** and then, in the **Layout** window, select the **Page setup** tab to change the position and the size of the label. Click the **Printer Selection** tab to change the printer, too.

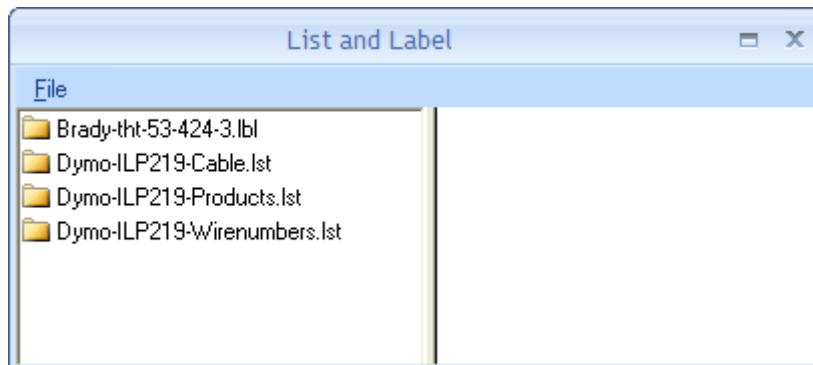
QQ.1.3. PRINTING LABELS

- Open the start window of the *List and Label Designer* and double-click the template. Please, make sure that the print options are set correctly.
- Select **Start** position to specify where to start printing (line 2, label 3 left). This setting allows continuing after a paper break. Click **Select**, and then click on the field to start. Then, print the labels.

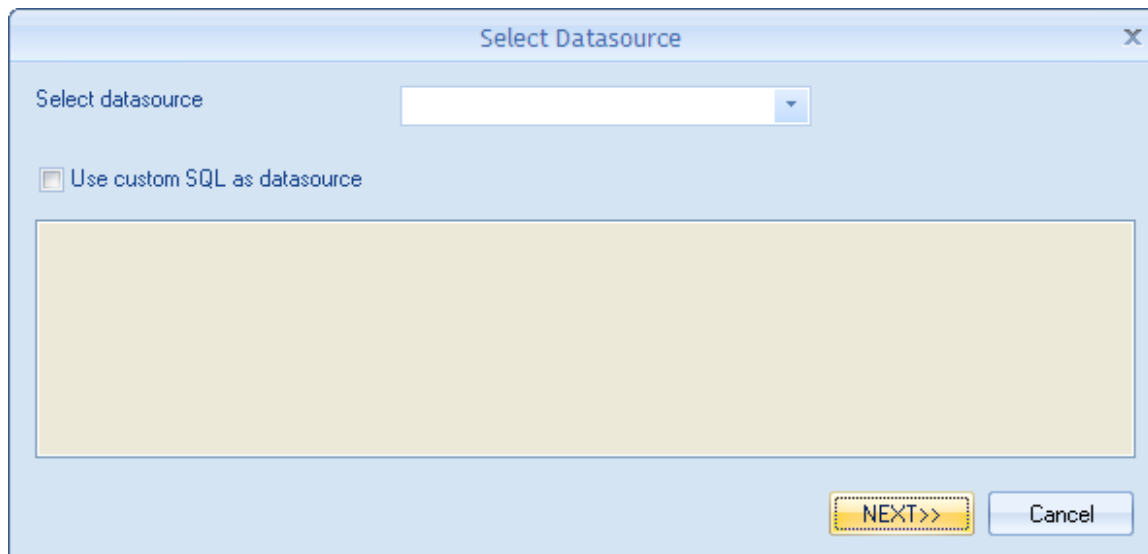
QQ.2. CREATING LIST TEMPLATES

Exercise 39-2: Create a list template.

- 1.CA **File**
- 2.CO **List and Label**

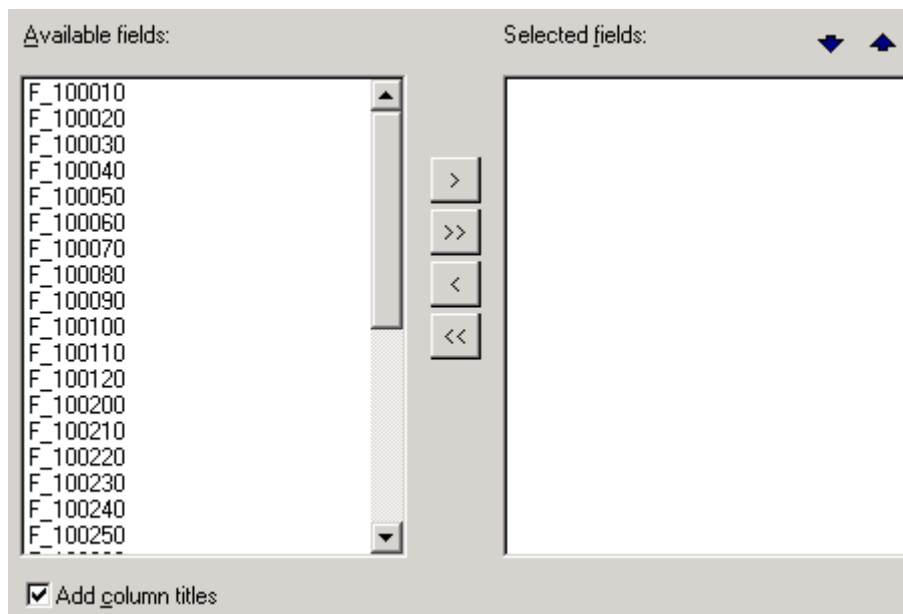


- 3.M **File**
Select the **File** menu in the **List and Label** window.
- 4.M **New List**
- 5.# <name>
Type the name of the new template.
- 6.M **Save**
The following window appears:








- 7.> Select data source
Select the database, from those included in the *SEE Electrical* database lists, which contains the fields you wish to be recorder in the label. You can select only one list.
- 8.> Products
- 9.> Next
- 10.> Next
Press **Next** to close the **Project Wizard** window.

11. You can specify whether to use one printer for all pages or to use different printers for first page/following pages.
- 12.> Next
After you have made your choice, click **Next** to close the dialogue box.
- 13.> Specify the printer and the other printing options.
- 14.> Next
Close the dialogue box by clicking **Next**.
15. You can define page numbering and add title.
If you want to add a title, specify if it is to be printed on the first page or on all pages.
- 16.> Next
Close the dialogue box by clicking **Next**.
- 17.> Select Zebra pattern for the table to print the lines between rows and columns. You can choose the Create summary option. The summary displays the number of records only on the last page or on all the pages. Specify whether to create summary on all the pages or on the last page only.
- 18.> Next
Close the dialogue box by clicking **Next**.
- 19.> Choose the fields for the list:



(The fields in the list depend on the chosen database list, example: Product). You will also find common fields in the list, such as project name, project editor etc. You do not have to select these common fields but place them in the header line. Choose to **Add column titles** or not before you select the fields.

Click then the desired fields and click on the  icon to move them into the **Selected fields** area.

You can remove fields from the selection by clicking on the left arrow icon . You can select multiple fields and move them to the selection by clicking on the right arrow icon . The icon  moves all the fields to the selection and the icon  removes all the fields from the selection.

Using the  arrows, you can arrange the selected fields as desired.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

You can find the variable list in the chapter **Graphical lists** of the **User Manual**, where each list is described.

If the fields have been selected in the desired order, click "Done!" to close the dialogue box.

The list template is created.

In the next steps, you will find hints about changing the created list.

Did you place all the fields?

- 20.M **File**
- 21.M **Save**
- 22.M **File**
- 23.M **Exit**

Finish creating the template.

The template can be used for generating lists.

List templates include :

- ✓ *List title (text object)*
- ✓ *Header lines (in tables)*
- ✓ *Group Header (in tables)*
- ✓ *Data lines (in tables)*
- ✓ *Group Footer lines (in tables)*
- ✓ *Footer lines (in tables)*
- ✓ *List Footer lines (in tables)*

List of parts						→ Title
						¶
Project:123456	→	→	→	→	→	Headerline-1¶
(blank)	→	→	→	→	→	Headerline-2¶
Type	→	Number	→	Article-description	→	supplier → Headerline-3¶
VTL5032	→	3	→	frequency-changer-11-37-kW	→	Danfoss → Data-line¶
VTL5008	→	1	→	frequency-changer-0,75-7,5-kW	→	Danfoss
	→		→		→	Data-line¶
RS/K10	→	1	→	Switch-2St.,1S	→	MOELLER → Data-line¶
						¶
Page-1	→	→	→	→	→	Footer-line¶

Example of a list without groups:

List of parts			Title
Project: 123456			Header line 1
(blank)			Header line 2
Supplier: Danfoss			Group Header 1
Type	Number	Article description	Group Header 2
VTL5032	3	frequency changer 11-37 kW	Data line
VTL5008	1	frequency changer 0,75-7,5 kW	Data line
Supplier: MOELLER			Group Header 1
Type	Number	Article description	Group Header 2
RS/K10	1	Switch 2St.,1S	Data line
<u>Page 1</u>			Title

QQ.2.1. FILES FOR LIST TEMPLATES

The list template contains 5 CDS, LST, LSP, LSP and ~LST files. You must back up these files. If you would like to create a new list template by using an available one, copy the 5 files using the Explorer; a "copy of ..." is created.

Then, you can rename the list template in the start window of the *List and Label Designer*.

QQ.2.2. LIST TITLE (TEXT OBJECT)

List titles are created automatically if you have activated the "**Add title**" option in the *Project Wizard*. The list titles can consist of more than one line.

The list titles cannot contain parts of *SEE Electrical* projects (such as project name, project created date, editor, etc.)

QQ.2.3. HEADER LINES

Header lines in a table are used to define column titles, if you are not working with a group structure. Header lines can contain parts of *SEE Electrical* projects (such as project name, project created date, editor, etc.).

QQ.2.4. GROUP HEADER

Groups are used for structuring the list, for example, if all the products of a manufacturer must be printed consecutively, and the name of the manufacturer has to appear as a group header. You must define a condition for the group change in the header, i.e. if data in the specified field(s) change, a new group starts. You can choose whether to continue on a new page and show the group header again or not.

The group header may consist of more than one line, for example, a title and column titles.

QQ.2.5. GROUP FOOTER LINES

If you structure the list using groups, the group footer line can contain the article total price or the article total length. Besides, it can contain only texts.

QQ.2.6. DATA LINES

Data lines contain fields that must be entered in the list.

The available fields depend on the chosen database list of *SEE Electrical*. However, the available fields vary in the database lists too. For example, the "Free Text 01" field appears in the list of products only if at least one component in the current project contains a text with such attribute. If a desired field is not available for a data line, you must choose another database list or a combination of two database lists (see below the definition of SQL-queries), or you must use a project that contains the needed fields.

QQ.2.7. FOOTER LINE

If you do not work with groups, the footer line can include the total price of all the products or their total length. Besides, it can contain only texts.

QQ.2.8. LIST FOOTER LINE (TEXT OBJECT)

Footer lines are generated automatically if you have activated the page numbering option in the *Project Wizard*. Footer lines can contain more than one line.

QQ.3. EDITING LISTS

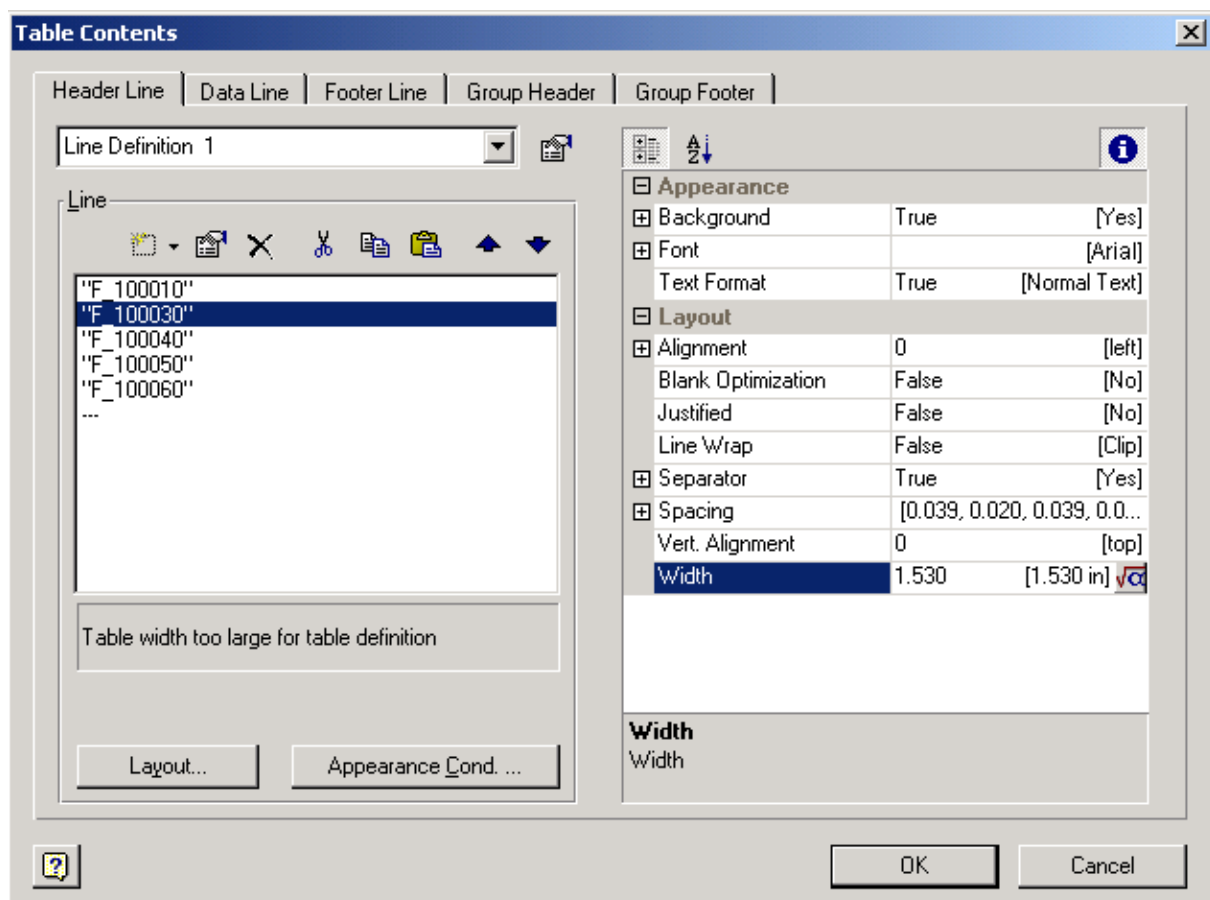
Exercise 39-3: Change the available template.

- 1.CA **File**
 - 2.CO **List and Label**
 - 3.> Select the template to change
 4. Right-click with the mouse. The pop-up menu appears.
 - 5.CO **Design**
 - 6.> Next
- Changes are to be made in the **Layout Preview** area, as you see here, not only the field number as in the **Layout** area, but also the text content of the field. You can activate the **Layout Preview** tab in the bottom window's border.



QQ.3.1. CHANGING COLUMN TITLES (HEADER LINES)

1. If you edit a list template, you can change column titles by double-clicking it.

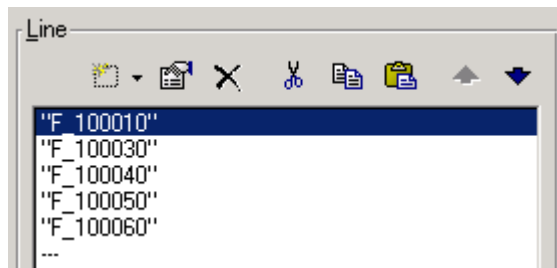


Training manual

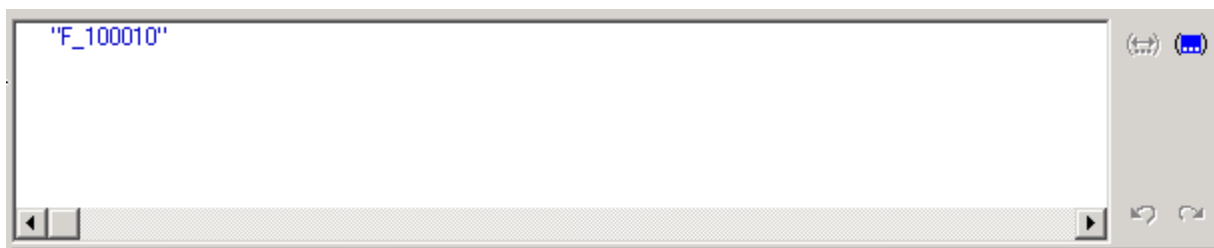
COPYRIGHT © 2013 IGE+XAO. All rights reserved

2. In the **Table contents** window ➤ **Header Line** tab window, you can change the Layout of the text or the Font in the right window area, i.e. the size and the name of the font.

To change the column title, double-click on it, for example "F_100010", if you want to enter a header line, for example "Project name".



3. If you double-click any field, the following new window appears:



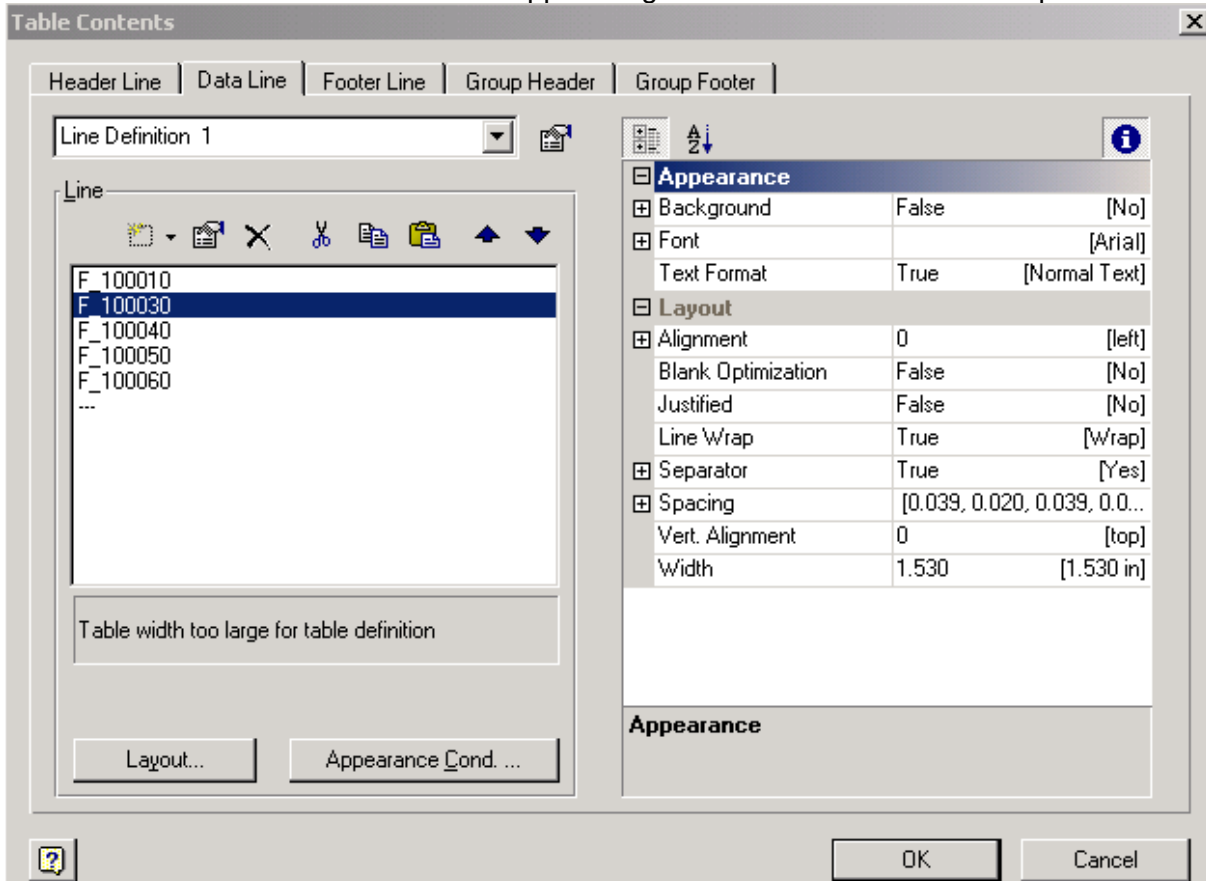
4. Double-click again and type the desired text. The text must be enclosed within double quotation marks ".

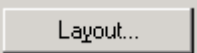


5. Close the dialogue box by clicking **OK**.
6. Change all header lines of the columns in this way and then click **OK** to close the **Table Contents** window.


QQ.3.2. CHANGING A LIST (FONT SIZE, FIELDS ORDER, ADDING OR DELETING FIELDS)

1. If the list template is open, double-click one of the fields in the table.
The Table contents window appears again but the **Data Line** tab is open now.

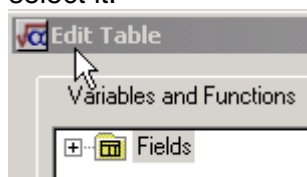


Click  to change the layout of a table line (Print margins, the font preference of the text). Besides, you can change the **Font** and the **Layout** in the right window's area.

Click the  or  arrows to move line down or to move line up.

After a field selection, click the  icon to delete the selected field. After you have closed this window by clicking **OK**, you must double-click the column header line of the table and delete the corresponding record.

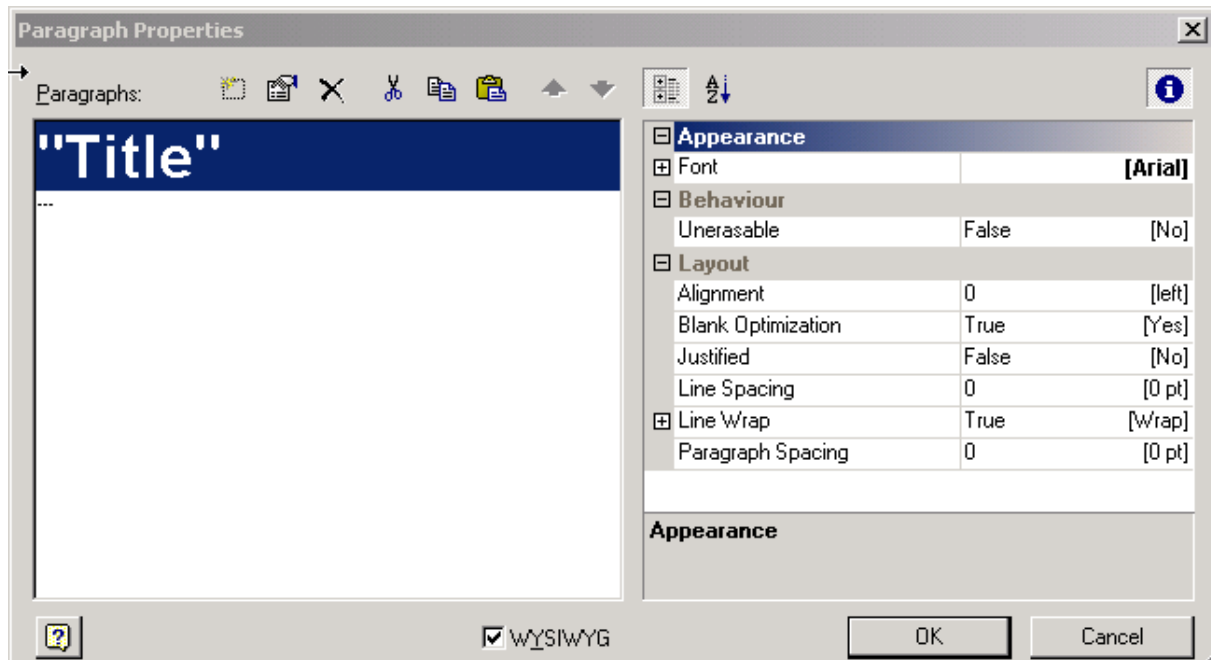
Click on the  icon to insert a field. The **Edit Table** window appears. You can double-click the **Fields** folder in the upper left pane and double-click the desired field code to select it.




Click **OK** to close the **Edit Table** window. You must afterwards double-click the column header line in the table, insert the corresponding record and put it in the right place.

QQ.3.3. CHANGING THE LIST TITLE

1. You can edit the list title by double-clicking on it.
2. The following window appears:

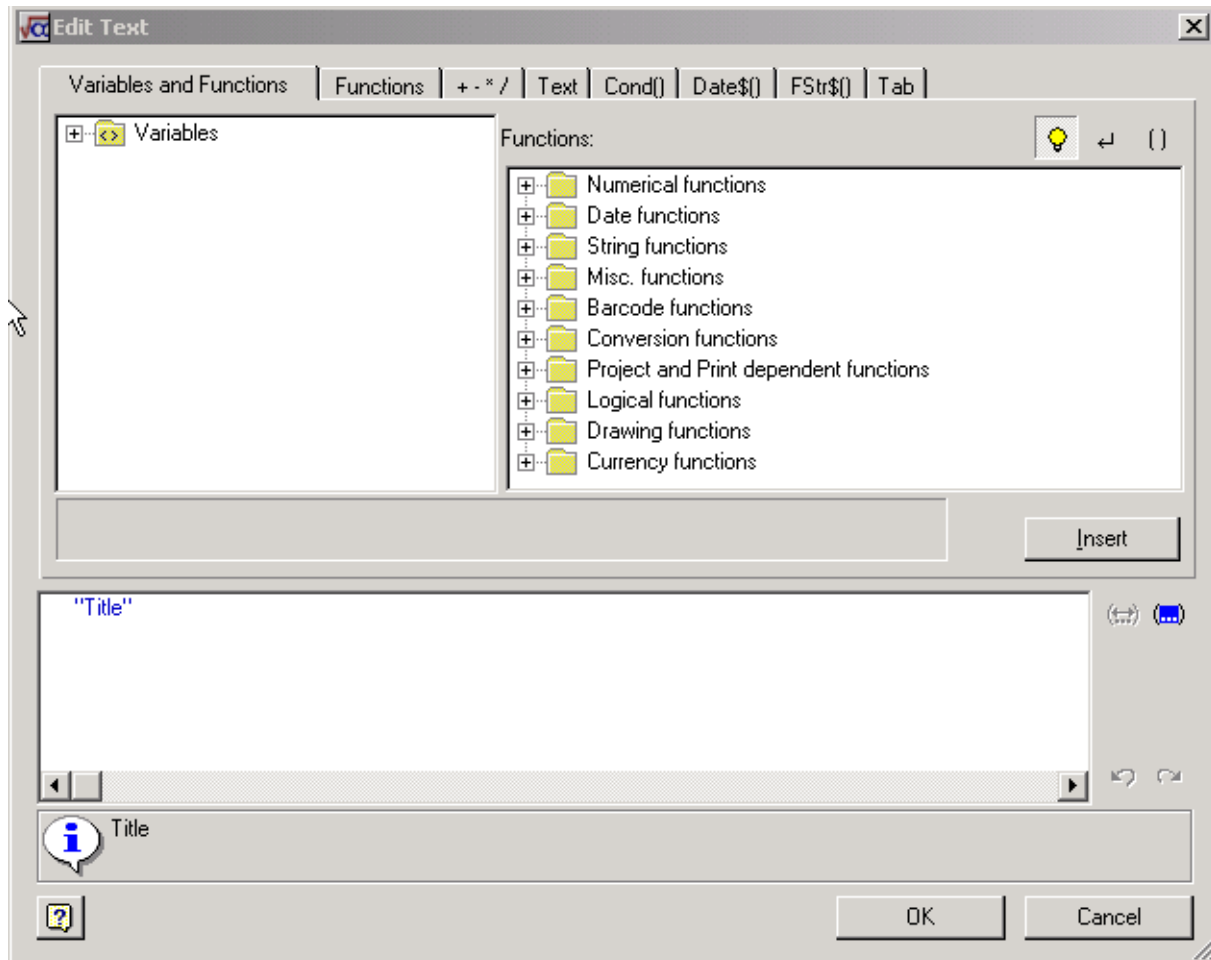


You can change the alignment of the list to appear centred, make the font **Bold** or Underline, change the type and size of the font (see properties under **Font** or **Layout** in the right area of the window).

You can delete the list title by clicking on the  icon.

You can insert an additional list title by clicking on the .

4. If you wish to change the text in the list title, double-click the text again –in the above example, double-click Title. The following window appears:

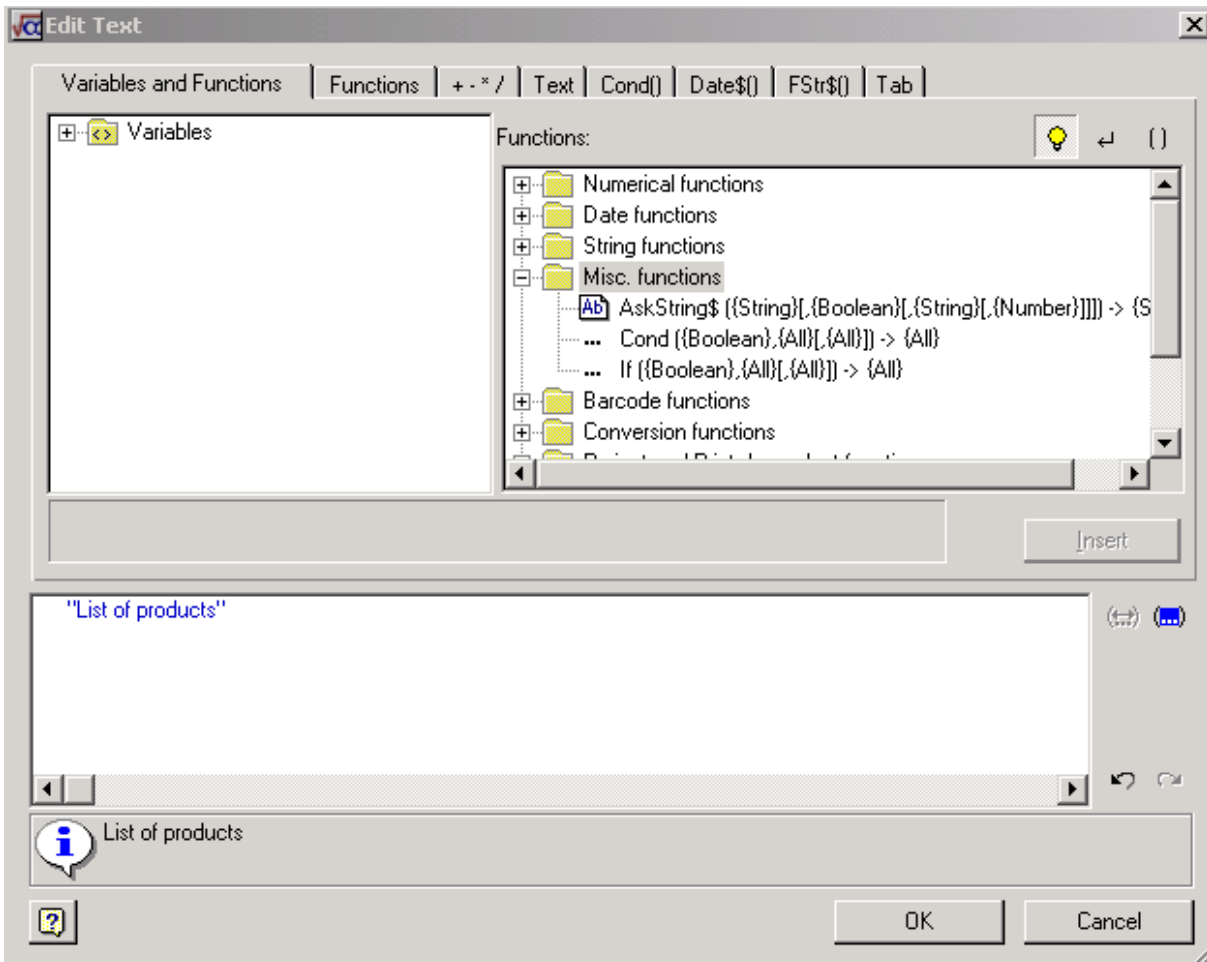


Change the text by double-clicking on it again. The texts must be enclosed within double quotation marks, for example "List of products".

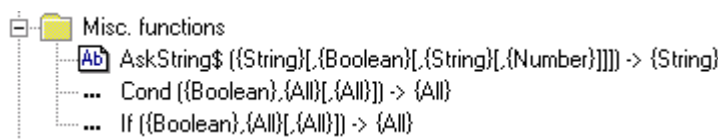
3. Click **OK** to close the window.
4. Close the **Paragraph Properties** window by clicking **OK**.

QQ.3.4. LIST TITLE WITH QUESTIONS

1. If the window shown in the following illustration is open, you can define questions to appear while creating the list. (If the window shown in the following illustration is not open, double-click the list title and the Paragraph Properties window will appear. Double-click the list title again.)



2. Define a question about the project name.
Click behind the text that must be displayed in the header line, e.g. "*List of products*".
Open then the *Miscellaneous functions* folder in the right window pane by double-clicking it.
Select "AskString\$ {...}" again by double-clicking it.



3. The variable is displayed behind the text.

"List of products" + AskString\$ (...)

The variable "AskString {...}" allows you to create a question by entering user-defined texts.

You must enter the arguments, too.

Move the cursor to the place between the next two characters of the variable: Type then how the variable must be named, for example, you can type "project name". The text must be enclosed within double quotation marks ".

```
"List of products" + AskString$ ("Projectname" ...)
```

Move the cursor forward behind ("Projectname",

```
"List of products" + AskString$ ("Projectname" ...)
```



Choose ".F." from the automatically displayed pop-up menu to specify that the dialogue with the question will appear only once, at the beginning of the list.

Move the cursor forward to the position ("Projectname",.F., . Enter the value to be suggested automatically, for example "Name", as the entry of the project name is expected. The text must be within double quotation marks ".

Give the max. character number for the text. We suppose 200 characters for the project name. Move the cursor to the position between the characters,).

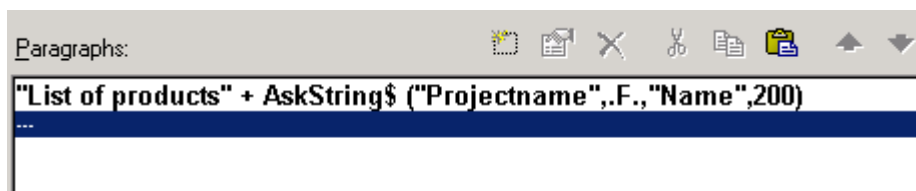
```
"List of products" + AskString$ ("Projectname" ,.F., "Name",200)
```

Now, the variable for the question about project name (or another user-defined text) before generation of the list has been defined.


4. Click **OK** to finish editing the first header line.

QQ.3.5. LIST TITLE WITH GENERATION DATE

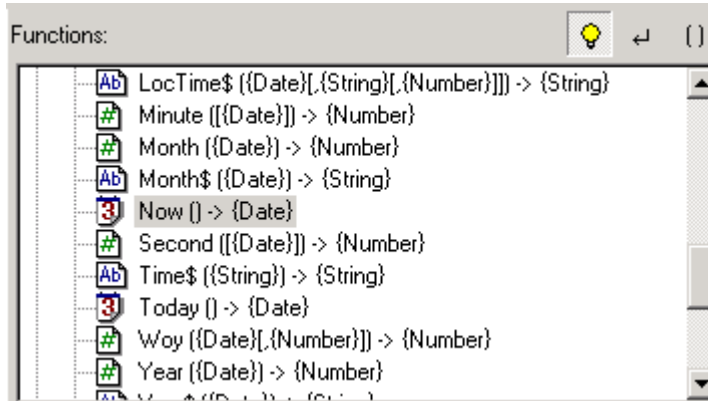
- Enter the date of generation in the title of the list.
- Double-click the title.
- Double-click onto ... beneath the line of the list title "Products + AskString\$("Projectname",.F., "Name",200).



You will go into the window again where you can edit the texts. For example, type the text "created:" on the keyboard in the text area.

- Click then behind the text (position the cursor behind the text).
 - Open the "Date functions" folder in the "**Functions**" area by clicking the plus sign .
 - Scroll the suggested variables until you see "Now() ->{Date}".
- This variable shows the current day.

- Double-click on it.



The function is transferred to the text line.

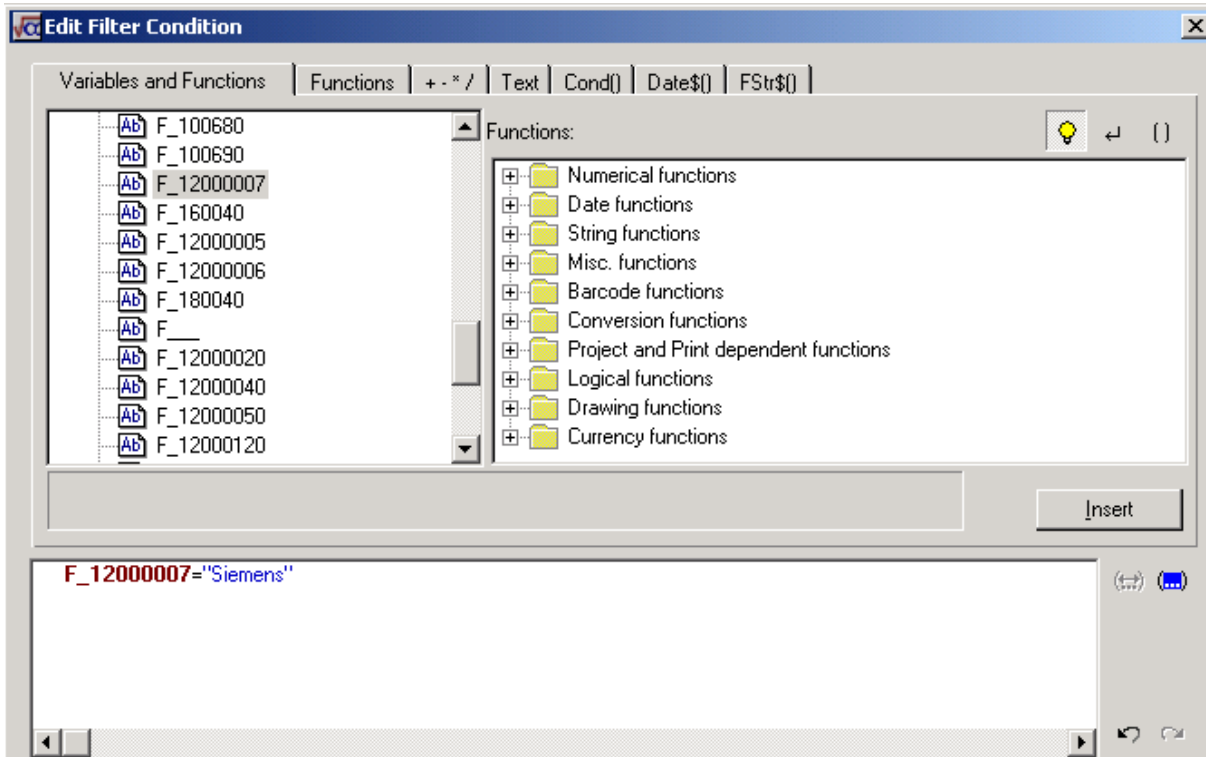


QQ.3.6. FILTERING IN THE LISTS

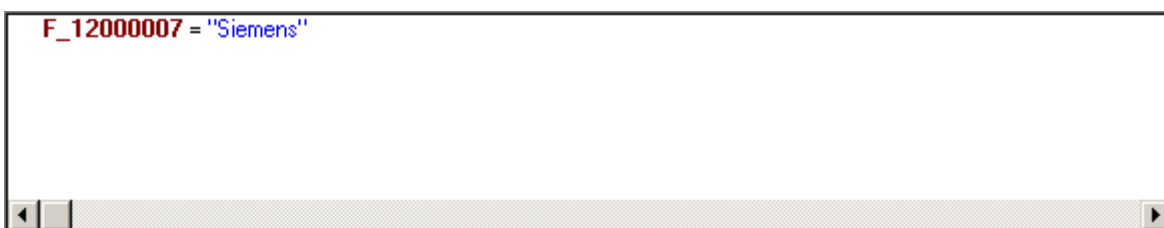
It is possible to define multiple filters. By means of some examples, the definition of frequently used criteria will be explained.

QQ.3.6.a. LIST OF PARTS FOR A PARTICULAR MANUFACTURER

1. If the fields of the list template have been defined (List of parts), define then the filter using the **Project** ➤ **Filter**.
2. Double-click the *Fields* folder in the left window's area to open it and double-click the field code you like to define a filter for. In the example, this is the manufacturer and the code is F_12000007.



3. The field code is displayed in the bottom window's pane.
Type here the manufacturer's name for the filter, behind a "=" sign. The name must be between double quotation marks ". Uppercase and lowercase are taken into consideration.



You can type the expression "Ask String ...", too (see below), for example.
`F_12000007 = AskString ("Manufacturer,.F.,\"SIEMENS\",200)`

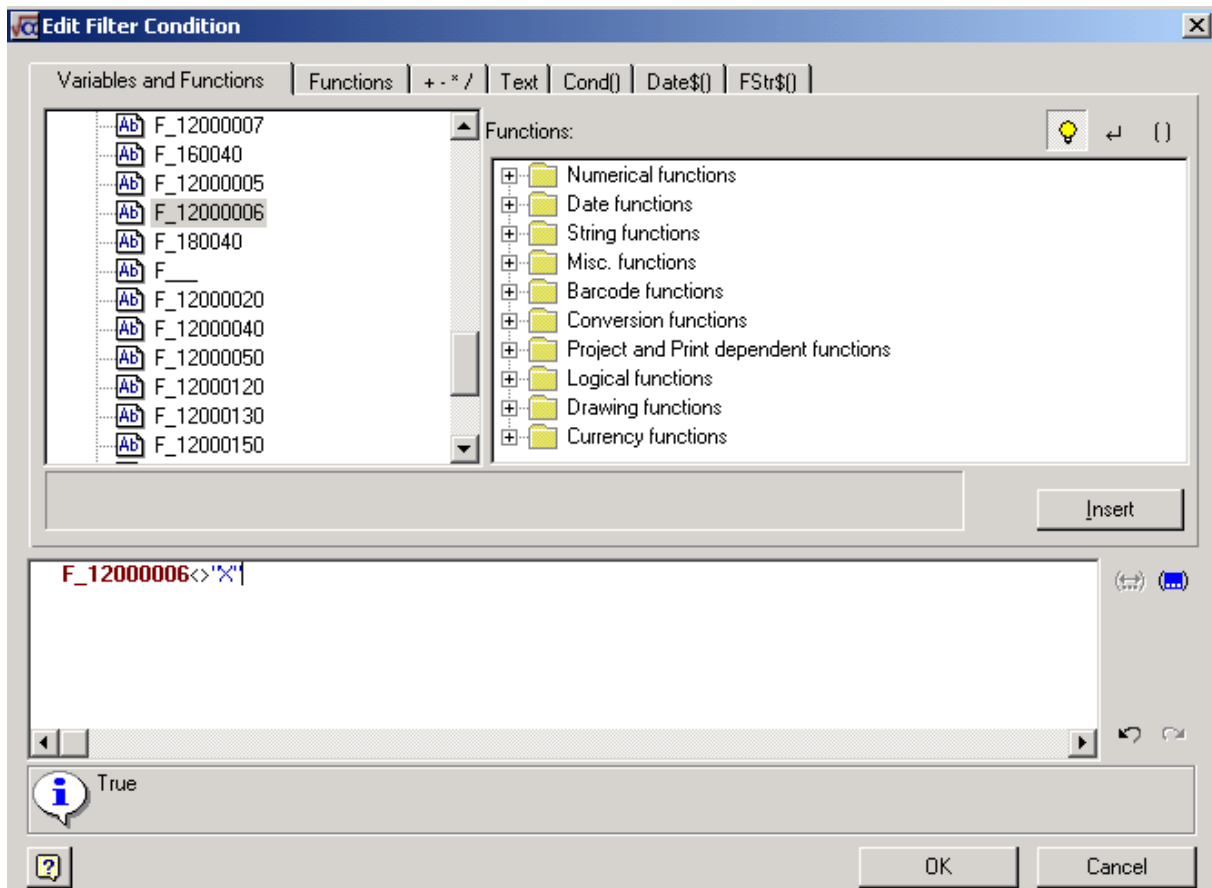
The "Ask String\$..." executes a question about the manufacturer.
 You will find an illustration about the "Ask String\$..." expression in the chapter "*List title with questions*" above how to enter the project name manually in the list.
 If you wish to be sure that the manufacturer will be found after using uppercase or lowercase, you can use the command "Upper\$...":
`F_12000007 = Upper$(AskString$("manufacturer,.F.,\"SIEMENS\",200))`
 The expression "Upper\$..." converts all the letters into uppercase.

4. Close the dialogue box by clicking OK and save the list template.

QQ.3.6.b. LIST OF PARTS WITHOUT TERMINALS

Condition: All the terminal strips belong to the same article group, for example to X.

1. If the fields for the list template are determined (for List of parts, simple), define the filter using **Project** menu ➤ **Filter**.
2. Double-click the "**Fields**" folder in the left upper area and double-click the field code you wish to define a filter for. In the example for the article group this is the code F_12000006. The field code is transferred into the bottom area of the window.
2. Type the expression
3. F_12000006 <> "X"
in the bottom area of the window.

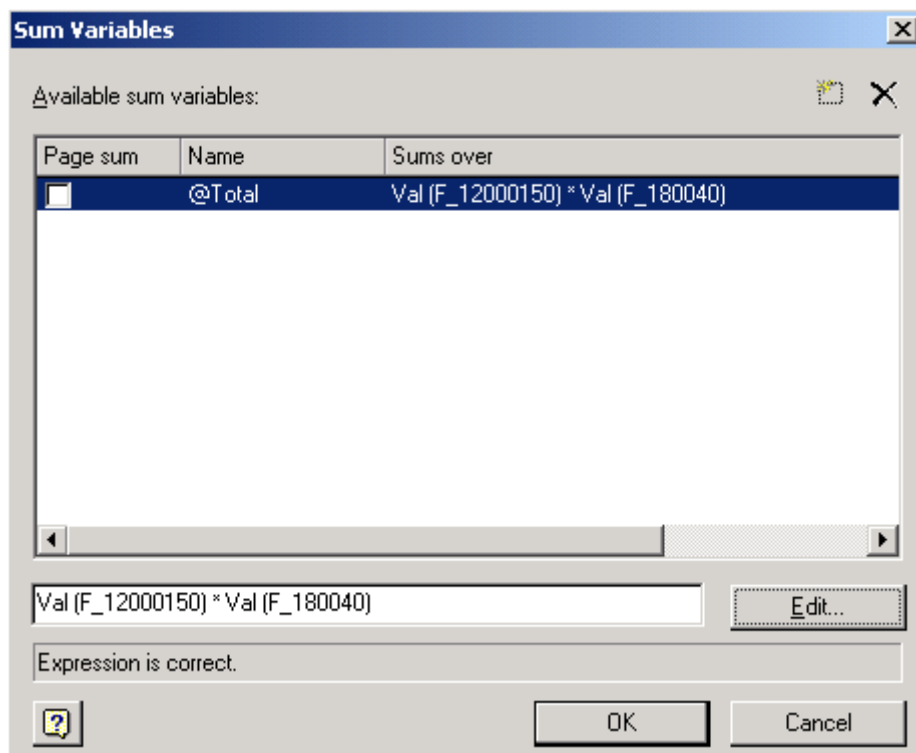


QQ.3.7. CALCULATIONS

To execute calculations, for example about costs of an order, it is necessary to define sum variables. You can define sum variables by selecting **Project** menu ➤ **Sum variables**.

QQ.3.7.a. CALCULATION OF ORDER COSTS

1. Define sum variables
Calculate the total price of the articles of a type first. The total price is calculated as follows:
Unit price (code 12000150) x **Number** (code 180040).
The sum variable "Total" is defined as follows:



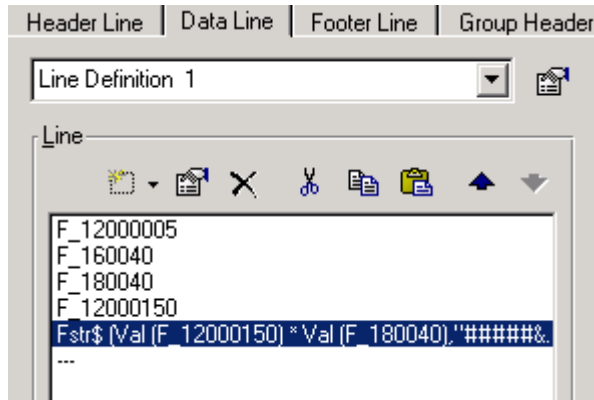
As the unit price and number are transferred from the *SEE Electrical* project as text (=character string), the contents of the field is converted into a number using the **Val** command.

The option "**Page sum**" must not be activated, as otherwise the prices in one page will be accumulated. However, the total price of all the articles in the list will not be calculated.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

2. Edit data line
If the sum variable is defined, the table can be edited. The data line includes the record **Unit price x Number** in addition to the data fields.

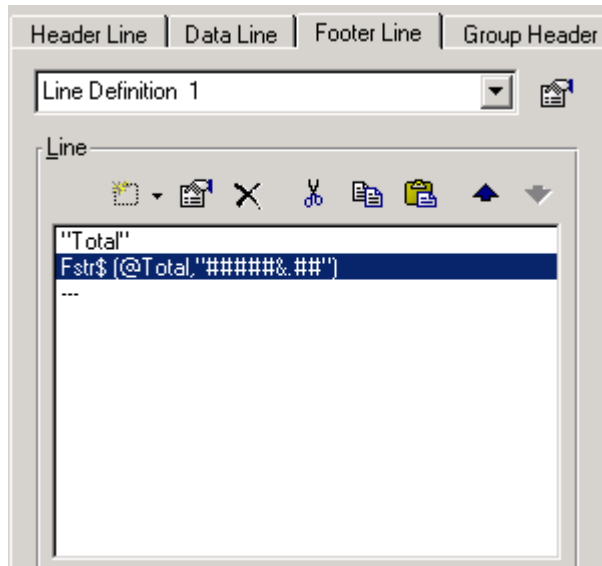


The result of the calculation **Unit price x Number** is a number. This number must be formatted (2 decimal places) by using the command:

Fstr\$(<value>,"#####&.&#").
in the form

Fstr\$(Val(F_12000150)*Val(F_180040),'#####&.&#');

3. Edit Footer line
The total for the whole project is recorded in the footer line:

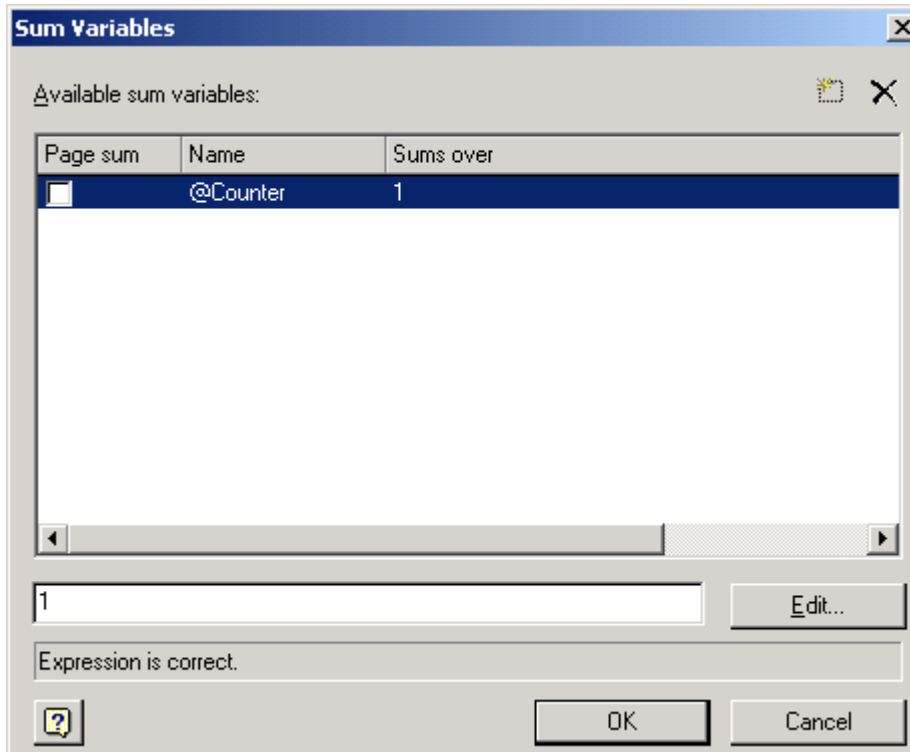


The number is formatted by using the command: **Fstr\$(<value>,"#####&.&#").**

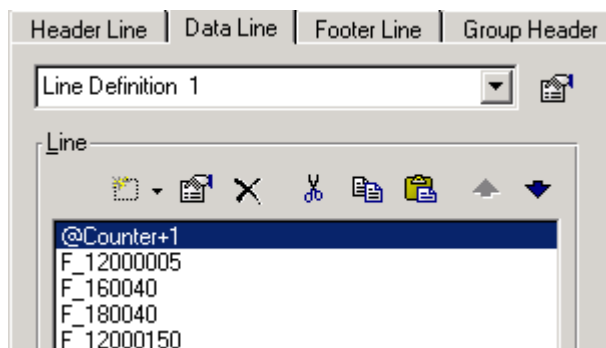
QQ.3.8. LINE NUMBERING

If you wish to print a current number per line, each line number must be calculated.
 Define a sum variable for the line numbers counter as described in the chapter "*Calculations*" using "Project" -> Sum variables.

1. Define sum variable



2. Edit data line
 After you define the sum variable, the table can be edited. The data line includes another record **<counter>+1** in addition to the data lines.

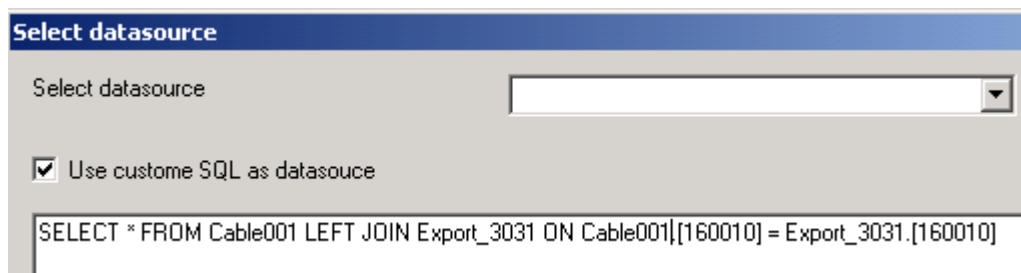


QQ.4. DEFINING SQL-QUERIES FOR LISTS AND LABELS

SQL-queries allow generating lists that go beyond the simple ordering, filtering and grouping fields from a database list.

QQ.4.1. JOINING TWO LISTS

It is possible to join data from two different database lists into one common list using SQL-queries in the **List and Labels** area.



QQ.4.1.a. JOINING INFORMATION FROM CABLE LIST AND CABLE-WIRES LIST

1. Define a SQL-query

```
SELECT * FROM Cable001 LEFT JOIN Export_3031 ON Cable001.[160010] = Export_3031.[160010]
```

Example for SQL-query:

SELECT

*

LEFT JOIN

ON <value1>

= <value2>

The SELECT statement selects data from a table.

Syntax of the statement:

SELECT <fields> FROM <table in Access database>

All of the fields are selected.

The second table is joined to the first table.

It defines which values in the two tables have to match in order to join data. In the example above, it is the component name (code 160010). The values must be defined in the form <table name>.[field name].

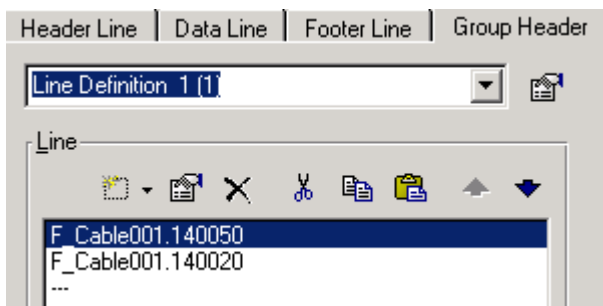
2. Defining Group header
 Data within a list can be structured by inserting group headers.

Example:

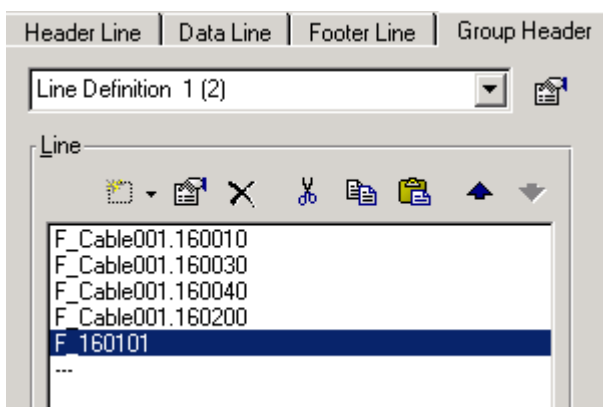
The cable data is displayed in a group header area above the cable-wires data.

The definition of the group header is made in 3 lines:

Cables					List Header
Cable: W1 Type: NYY 5x1,5qmm Length: 20m					Group header
Nr.	Colour	Size	Target 1	Target 2	Group header
1	sw	1,5	X1:L1	1M1:U	Data line
2	sw	1,5	X1:L2	1M1:V	Data line
3	sw	1,5	X1:L3	1M1:W	Data line
5	gn/ge	1,5	X1:PE	1M1:PE	Data line
Cable: W2 Type: NYY 5x1,5qmm Length: 10m					Group header
Nr.	Colour	Size	Target 1	Target 2	Group header
1	sw	1,5	X2:L1	2M1:U	Data line
2	sw	1,5	X2:L2	2M1:V	Data line
3	sw	1,5	X2:L3	2M1:W	Data line
5	gn/ge	1,5	X2:PE	2M1:PE	Data line
Page 1					Footer



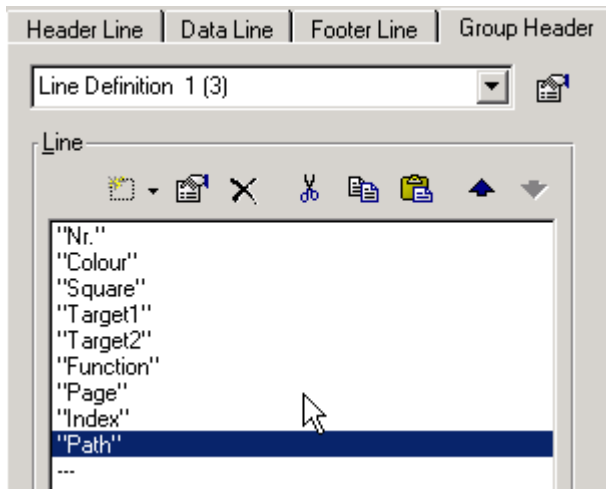
Cable Function - location



Cable name, type, length ...

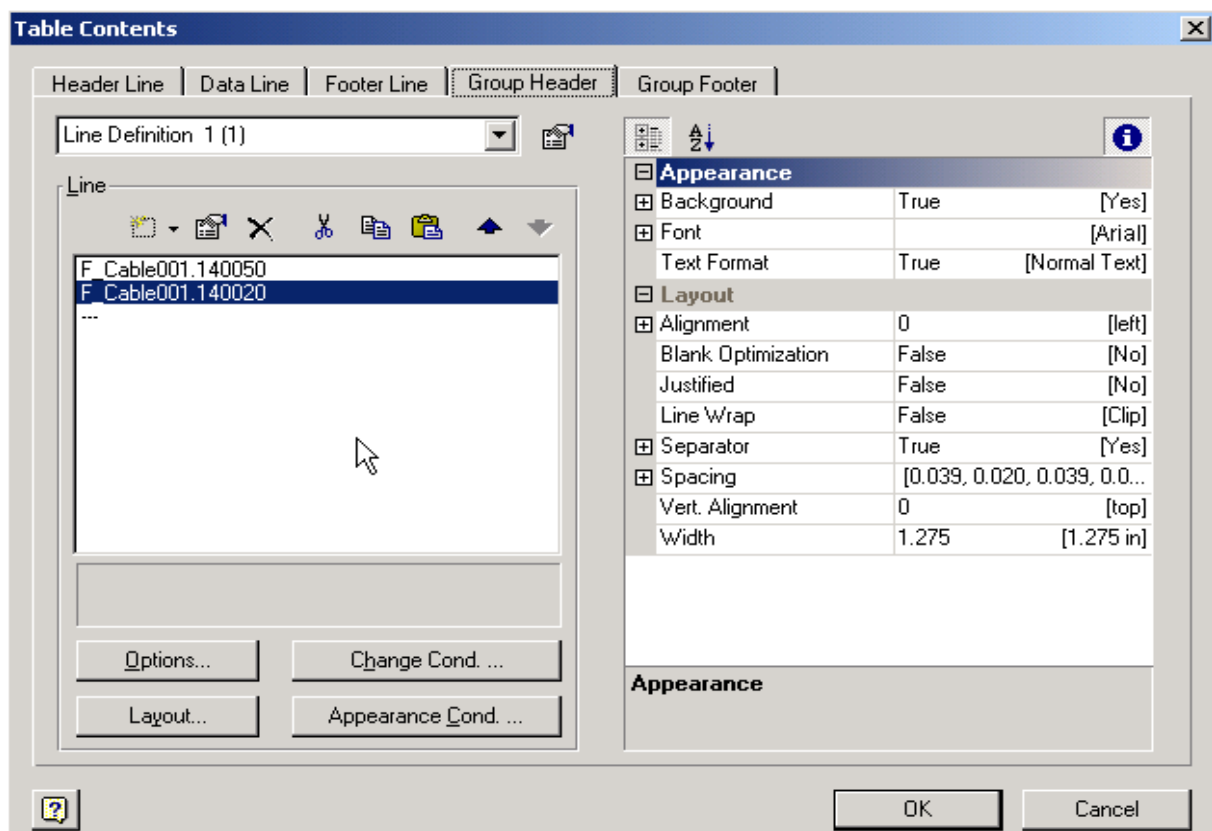
Training manual


COPYRIGHT © 2013 IGE+XAO. All rights reserved

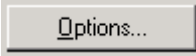


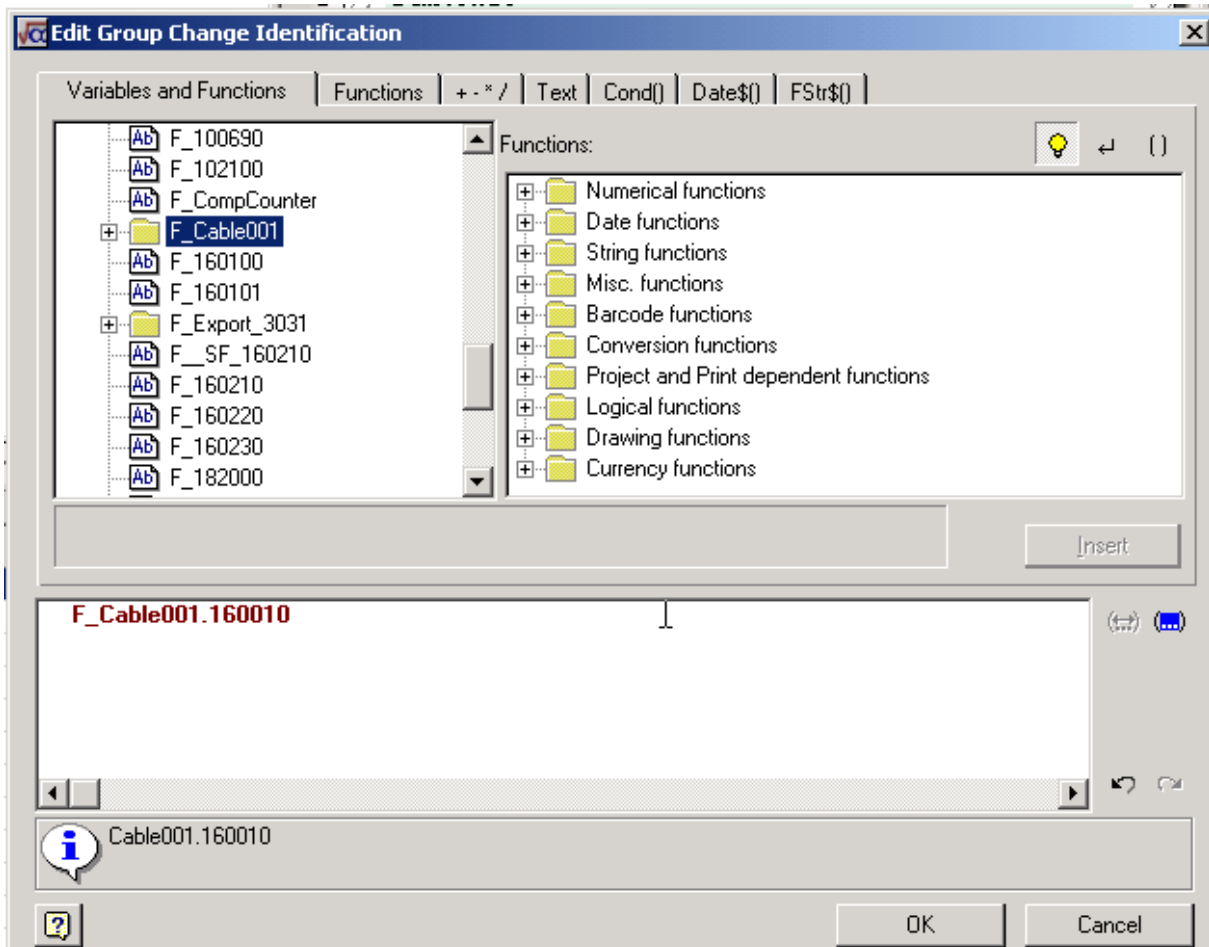
Column titles

- 2.a Insert a page break after each group
It is essential in some cases to continue on a new page after fulfilling a specific condition, although the previous page has not been finished yet. If you work with a group header, you can define the condition for your list, too.



Select  to type the condition.

Select  and define page break after group changing:



The group must change when the cable name changes (field code 160010) that comes from table "Cable001".

QQ.4.2. FILTERING DOUBLE RECORDS

You can filter double records using the SQL-statement SELECT with the predicate DISTINCT.

Example: A list including functions and locations on the pages must be created.

Each combination must be displayed only once.

QQ.4.2.a. LIST OF FUNCTIONS/LOCATIONS ON PAGES

1. Define a SQL-query
 SELECT DISTINCT Export_3001.[180015], Export_3001.[180018] FROM Export_3001

SELECT DISTINCT	The SELECT statement selects data from the table. Each combination of the selected data is displayed only once by using the predicate DISTINCT. Syntax of the statement: SELECT <fields> FROM <Table of Access database> As the fields here do not have names, enclose the field code in square brackets "[" and "]".
-----------------	---
2. Place fields for function and location.

QQ.4.3. SORTING A LIST

To sort lists in a specific way, use SQL-statements.

Examples:

*If you want to receive an order list from the part list, the list has to be sorted by manufacturer.
If the lengths of equal cables must be added, the list must be sorted by cable type before addition.*

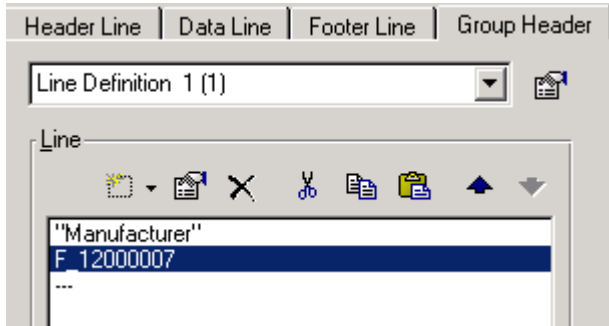
QQ.4.3.a. ORDER LIST

If you want to receive an order list from the part list, the list must be sorted by manufacturer/manufacturer. Then the articles must be grouped beneath the manufacturer. It might be necessary to break the page, if the manufacturer/manufacturer changes.


1. Define a SQL-query
 SELECT * FROM Export_3100 ORDER BY Export_3100.[12000007]


SELECT	The SELECT Statement selects data from a table. <u>Statement syntax:</u> SELECT <fields> FROM <table of Access database>
ORDER BY	This clause enables sorting of lists. <u>Syntax:</u> ORDER BY<value> The value must be defined under the following form: <table name>.<[field name]>

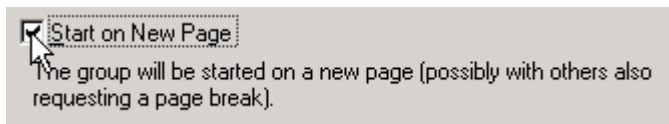
2. Define a Group header



2a. Define group change identification

Select , the group change identification is **F_12000007**.

Select then  and specify to start on a new page, if the group changes.



QQ.4.3.b. CABLE LENGTH ADDITION

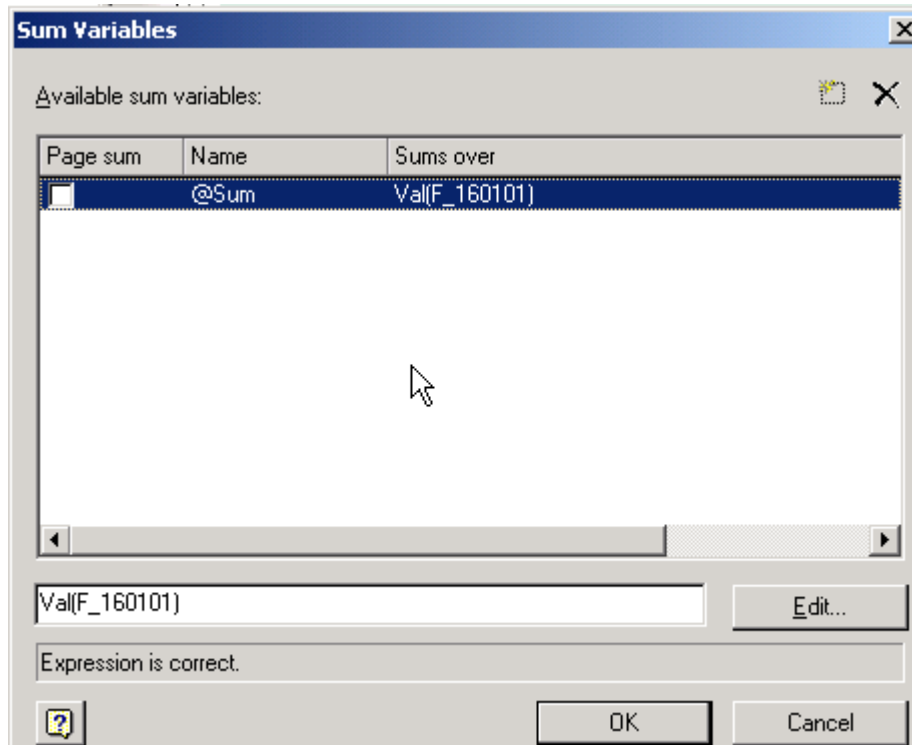
Add the length of cables of equal types.

1. Define a SQL-query
 SELECT * FROM Cable001 ORDER BY Cable001 [160040]
SQL-query:

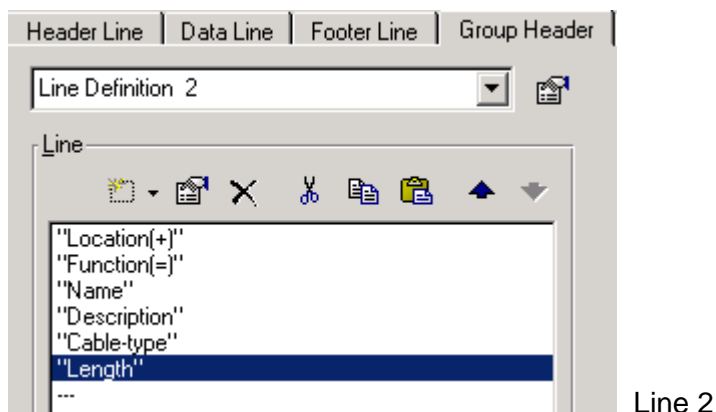
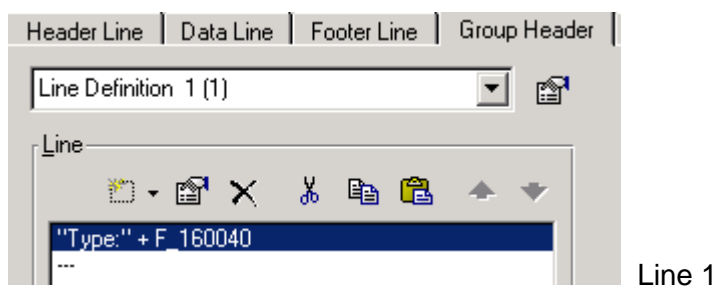
SELECT	The SELECT Statement selects data from a table. <u>Syntax:</u> SELECT <fields> FROM <table in Access database>
ORDER BY	To sum the length of cables of equal types, sort the list by cable type first. Use the ORDER BY clause. To sort then the cables by name, specify the name as second parameter. <u>Syntax:</u> ORDER BY <value> The value must be defined under the following form <table name>.<[field name]>. Multiple sorting criteria must be separated with commas ", ".

2. Define Sum variables
 To calculate the cable length, define a sum variable.
 You can define sum variables using **Project menu > Sum variables**

Example: Calculate cable length

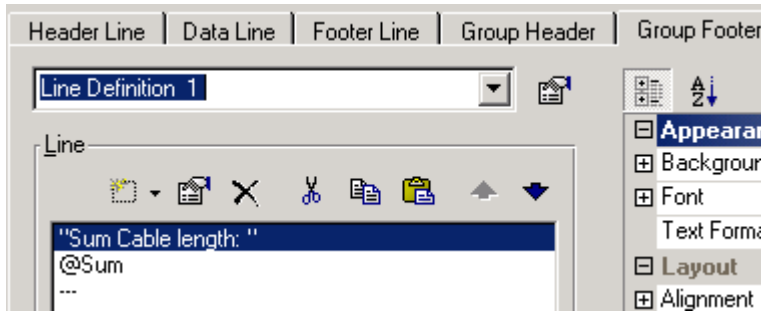


3. Specify a Group header



4. Data line must include the cable length.
5. Define a Footer line

Calculation of cable length for multiple cable types is performed in the Group Footer.



6. Define the group change identification:



QQ.4.3.c. LENGTH CALCULATION FOR CABLE CHANNELS AND RAILS

- 1: Define a SQL-query

```
SELECT * FROM Export_3010A, Export_3011 WHERE
Export_3010A.Name=Export_3011.[160010] ORDER BY Export_3010A.ObjectType,
Export_3010A.Name
```

Clauses SQL-query:

SELECT

The SELECT Statement selects data from a table.

Syntax:

SELECT <fields> FROM <table in Access database>

Multiple tables are separated with commas ",".

WHERE

WHERE <value1> = <value2>

You can define which values from the two tables must be equal, in order to calculate data. In this example, the values are component name in the NAME field of the Export_3010A table and component name in the 160010 field of the Export_3010A table.

The values must be defined under the following form:

<table name>.[field code].

ORDER BY

To perform printing of the cable channels first and rails after them, the sorting by object types is first necessary. Use the ORDER BY clause. Then, sorting by type must be done before sorting by component name.

Syntax: ORDER BY<value>

The value must be defined under the following form:

<table name>.[field code].

Multiple sorting criteria must be separated with commas ",".

2. Define sum variables. Compare the "Cable length addition" example and follow the steps there.
3. Define group header. It contains the record:
 If(Contains(F_ObjectType,"19102"),"Cable channel", "Rail")+ ": "+F_160040+" "+
 F_12000005



The expression defines to print the "Cable channel" text for objects of type 19102 only, and otherwise – the text "Rail".

Syntax

```
If (Contains (<field>,"value"),
<"Result, if condition is complied with">,
<"Result, if condition is not complied with (can be omitted)">)
  <Field 1> + ": " + <Field 2>
```

This is also a possibility to join different fields in one line.

4. The data line must contain the length; compare the "Cable length addition" example.
5. Define a Footer Line as in the "Cable length addition" example.
6. Define group change identification as in the "Cable length addition" example but the change must be executed by changing the object type, i.e. the condition is F_ObjectType

QQ.4.4. MULTIPLE PRINTING OF LABELS

If you need to print labels multiple times, print all the labels first and then once again, or print the labels twice over.

1. Print multiple labels (List of products)
 SELECT * FROM Export_3010 UNION ALL SELECT * FROM Export_3010
 i.e. the SELECT-statement is repeated after UNION ALL.
The statement:
 SELECT * FROM Export_3010 UNION ALL SELECT * FROM Export_3010 UNION ALL
 SELECT * FROM Export_3010
 executes three times repeated printing.
2. Print multiple labels serially (List of products)
 SELECT * FROM Export_3010 ORDER BY Export_3010.[160010] UNION ALL SELECT *
 FROM Export_3010 ORDER BY Export_3010.[160010]
 The same labels appear directly one after another, for example
 S1 S1 F1 F1
 Q1 Q1 M1 M1

3. For Labels for terminals (List of spare parts), it is necessary to create a link to the List of products in order to perform filtering on the components in the circuit diagram.
SELECT * FROM Export_3101 LEFT JOIN Export_3010 ON
Export_3101.[160010]=Export_3010.[160010] ORDER BY Export_3101.[160010] UNION
ALL SELECT * FROM Export_3010 LEFT JOIN Export_3010 ON
Export_3101.[160010]=Export_3010.[160010] ORDER BY Export_30101.[160010]

QQ.4.5. LABELS FOR COMPONENT NAMES WITH DEFINED CONTENT

It is possible to print labels only for defined component name content as follows:

1. Only records with defined content
SELECT * FROM Export_3010
WHERE (Instr (Export_3010.[160010],"K") OR Instr (Export_3010.[160010],"S"))
2. Only records without defined content
SELECT * FROM Export_3010
WHERE NOT (Instr (Export_3010.[160010],"M") OR Instr (Export_3010.[160010],"Q"))

QQ.5. USEFUL TABLES IN ACCESS DATABASE

AllTypesDISTINCT	All types in the project (without types, incl. subtypes) –no data about article's information registry –without number
Bom_Explode2	All types in the project (without types, incl. subtypes) –no data about article's registry of information –numbers.
Bom01	List of cables
Cable001	Component names and types
ComponentTypes	Component names and types
ComponentTypesExploded	List of documents
Export_3001	List of products with components from Circuit diagrams and Cabinets (channel, top hat rail, etc .)
Export_3010	List of products with type information
Export_3011	Terminal names and types as well as article's registry of information
Export_3020	List of terminals without type information
Export_3020A	List of terminals with type information
Export_3020B	All terminal types – article's registry of information –without number
Export_3020D	Connector List
Export_3025	Connector Pin List
Export_3025	List of cables
Export_3030	
Export_3030A	Only cable types
Export_3030B	Only different cables types
Export_3030C	List of cable-wires
Export_3031	List of contacts
Export_3040	List of PLC
Export_3050	List of wires
Export_3060	List of potentials
Export_3070	List of parts
Export_3100	List of spare parts
Export_3101	List of parts, simple
Export_3102	
Export_3103	List of spareparts, simple
Export_3104_Terminals	Combination of list of terminals and list of wires
Export_3180	List of products without components from Cabinets and less columns
Export_3181	Terminal list with x-y-coordinates and type
Export_3182	List of cable-wires with x-y-coordinates
Export_3225	List of Aspect Functions
Export_3226	List of Aspect Locations
Export_3280	Multicores
Export_3285	Multicores-wires
Export_Multiref	List of products without Cabinets with type Id
TypeInformation	Type Information

QQ.6. STATEMENT SUMMARY AND COMMANDS IN LISTS AND LABELS

QQ.6.1. SQL STATEMENTS

QQ.6.1.a. SELECT

The SELECT statement selects data from a database.

Syntax:

SELECT <fields> FROM <table in Access database>

Fields

The fields might be explicitly defined under the form:

<table name, containing the field>.<field name>

If you know the "right" field name, you can type it directly behind the dot operator. The field codes must be placed between square brackets "[" and "]".

Multiple fields can be separated with commas ",".

Select all of the fields in the table using an asterisk (*).

Examples:

SELECT ComponentTypes.Device, ComponentTypes.Location

or

SELECT Export_3001.[180015], Export_3001.[180018]

or

*SELECT **

Table in Access database

You can type the name of one or more tables where the data comes from. Multiple tables are separated with commas ",".

Example.

*SELECT * FROM Export_3010A, Export_3011*

Combine information with the help of SELECT:

Example for generation of combined result:.

```
SELECT Export_3010.[140050]+Export_3010.[140030]+
Export_3010.[160010] AS NameCombined,
Export_3011.[Manufacturer]+ Export_3011.[Goodsgroup]
AS Articledata
FROM Export_3010 LEFT JOIN Export_3011
```

QQ.6.1.b. SELECT DISTINCT

The SELECT statement with the DISTINCT predicate allows you to avoid double records.

Syntax:

SELECT DISTINCT <fields> FROM <table in Access database>

Example:

```
SELECT DISTINCT Export_3001.[180015], Export_3001.[180018]
FROM Export_3001
```

QQ.6.1.c. JOIN –CLAUSE

Databases usually contain a lot of tables. JOIN-clause is used for joining tables. It enables you to list data that cannot be found in one table.

Example: The list of cables contains information about the length of the cables but the list of cable-cores does not contain such information. If a list must contain information about cables length and cable-cores, it must be composed by matching data coming from both lists.

You can achieve this by joining the records of both lists. You must define a SQL-statement, for example, with a WHERE or ON clause.

Syntax of WHERE clause:

WHERE <Condition for selection>

Condition for selection

<Value1> <relational operator> <Value2>

Values : Specify which values from both tables must match in order to join the data. For example, this can be the component name (field code 160010). The values must be defined in the following format: <table name>.[field name].

Relational operators: =, <, > are allowed

Example:

```
SELECT *
FROM Export_3010A, Export_3011
WHERE Export_3010A.Name=Export_3011.[160010]
```

Syntax of ON clause:

<table 1 in Access database> LEFT JOIN <table 2 in Access database> ON <Condition for assignment>

Condition for assignment

<Value1> = <Value2>

It defines which values from both tables must match in order to join the data. In this example, it is the component name (field code 160010). The values must be defined in the <table name>.[field name] format.

Example:

```
SELECT *
FROM Cable001
LEFT JOIN Export_3031
ON Cable001.[160010] = Export_3031.[160010]
```

The ON clause may include several fields. The fields must be linked by using logical operators:

Example:

```
SELECT *
FROM Export_3103
LEFT JOIN Export_3010
ON
    Export_3103.[140050]=Export_3010.[140050]
AND
    Export_3103.[160010]=Export_3010.[160010]
```

QQ.6.1.d. ORDER BY

This clause allows sorting the list.

Syntax:

ORDER BY <value>

Value

The value must be defined in the following format: <table name>.<[field name]>.

Example:

```
SELECT * FROM Export_3100 ORDER BY Export_3100.[12000007]
```

Multiple sorting criteria are separated with commas ",".

QQ.6.1.e. WHERE/WHERE NOT

The WHERE clause can be used for joining two tables. WHERE and WHERE NOT can also be used for filtering.

Join two tables

See JOIN clause

Filtering

Some examples for using the WHERE clause:

Perform filtering of articles for one particular manufacturer from the parts list Syntax:

WHERE <value1> <relational operator> <value2>

Value1

The values must be defined in the <table name>.[field name] format.

Relational operators:

=, <, > allowed

Logical operators:

and, or, not

Value2

The values must be defined under a "<text>" form.

Example:

```
SELECT * FROM Export_3100
WHERE Export_3100.[12000007]="Siemens"
```

Print labels only for components with specified name value

Syntax:

WHERE (Instr (<Field>,"value")

Field

The field from the project database must be defined in the following format: <table name>.[field name].

Value

Enter text for the field value.

Logical operators:

and, or, and not

Examples:

1. Only records with defined values (component name "K" or "S") must be selected

```
SELECT * FROM Export_3010
```

```
WHERE (Instr (Export_3010.[160010],"K") OR Instr (Export_3010.[160010],"S") )
```

2. Select only records with the defined values (component name without "M" or "Q")

```
SELECT * FROM Export_3010
```

```
WHERE NOT (Instr (Export_3010.[160010],"M") OR Instr (Export_3010.[160010],"Q") )
```

QQ.6.1.f. UNION ALL

UNION ALL can be used to execute a query in the same list twice or multiple times. (For example, it can be used for multiple printing of labels).

Syntax:

```
SELECT <selection> UNION ALL SELECT < selection>
```

Example:

```
SELECT * FROM Export_3010 UNION ALL SELECT * FROM Export_3010  
(double printing)
```

or

```
SELECT * FROM Export_3010 UNION ALL SELECT * FROM Export_3010  
UNION ALL SELECT * FROM Export_3010  
(triple printing)
```

VAL

VAL converts a text into a number.

Syntax:

Val (<value>)

Example:

```
SELECT * FROM Export_3030A WHERE (Val(Export_3030A.[160200]) > Val("4.00")) and  
(Val(Export_3030A.[160200]) < Val("8.00"))
```

QQ.6.1.g. UPPERCASE/ LOWERCASE

It is not important for the SQL-query whether upper case or lowercase is used in the *SEE Electrical* database.

Example:

```
SELECT * FROM Export_3100 WHERE Export_3100.[12000005]="COIL"
and
```

```
SELECT * FROM Export_3100 WHERE Export_3100.[12000005]="Coil"
```

provide the same result, i.e. all records containing the "Coil" text are provided whether written in uppercase or lowercase.

QQ.6.1.h. LOGICAL OPERATORS

The operators AND, OR, and NOT are available.

QQ.6.1.i. RELATIONAL OPERATORS

The operators =, <, >, <> are available. InStr expression is available, too.

InStr

InSTR checks whether the string in the defined field contains a pattern string.

Syntax: InStr (<field>,"<pattern string>")

Example: Component names must contain the letter "M"

```
WHERE (InStr (Export_3010.[160010],"M"))
```

QQ.6.2. COMMANDS IN LISTS AND LABELS
QQ.6.2.a. CONDITIONS: IF

Syntax

```
If (Contains (<field>,"value"),
<"result, if the condition is satisfied">,
<"result, if the condition is not satisfied (can be omitted)">)
```

Example: Print labels (in one line, it doesn't match more than 4 characters), a page break can occur, but only when an empty character is found.

```
(F_160010)>3,Left$ (F_160010,3)+" "+Right$ (F_160010,4),F_160010)
```

If the component name is > 3 characters long, it is split into 3 characters on the left and 4 characters on the right with one space between. If the component name is <3 it remains whole.

QQ.6.2.b. UPPER\$/LOWER\$

Upper\$ (....) = "<VALUE>"

This command converts the letters case into upper so that it is possible afterwards to be compared with the value after "Upper\$ ()" (this value must always be written in uppercase).

The LOWER command changes the letters case into lower.

QQ.6.2.c. VAL

Val(<field>)

VAL transforms a text into a number. The fields in *SEE Electrical* databases contain generally numbers as texts.

QQ.6.2.d. FSTR\$

Fstr\$(<value>,"####.##")

This command allows formatting numbers.

QQ.6.2.e. ASKSTRING\$

AskString (<variable name>,<when does the question dialog appear>,<suggested value>,<max. number of characters>)

The variable "AskString {...}" allows you to execute a dialogue with a text question and enter user-defined texts while creating the list.

Arguments:

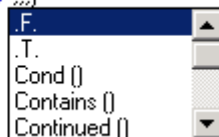
<variable name>

This text will be shown when the question appears. The text must be placed between double quotation marks " .

```
'Bauteilliste' + AskString$ ('Projektname' [...])
```

< When does the question dialog appear >

```
"List of products" + AskString$ ("Projectname" [...])
```



By selecting .F., you specify that the question dialogue is executed only once – at the beginning of the list.

<suggested value>

Enter the value to be automatically suggested. The text must be placed between double quotation marks ".

<max. number of characters>

Enter the max. number of characters expected for your text.

QQ.6.2.f. LEN, LEFT\$, RIGHT\$

Len gives the length of the strings.

Right\$ or Left\$ cut the right or the left characters from a string.

The following expression divides the strings into 2 parts if the length of strings exceeds a determined length. Using this, you can, for example, split the component name and print on two lines although a line break is not possible within one "word". Line break is possible only where a Blank character is.

If (Len(F_160010)>3,Left\$ (F_160010,3)+" "+Right\$ (F_160010,4), F_160010)

QQ.6.2.g. RELATIONAL OPERATORS

The operators =, <, >, <> are available. The expression "Contains" is available, too.

Example for Contains:

If(Contains(F_ObjectType,"19102"),"Cable channel","Rail")

This expression defines: for object type 19102 print the text "Cable channel", otherwise print the text "Rail".

RR CABINET LAYOUT

You can construct a *Cabinet Layout* as a part of the circuit diagram. It is possible to use symbols there, too.

Functions for dimensioning are provided with all levels of *SEE Electrical*.

RR.1.DRAWING CABINET LAYOUTS

Cabinet layouts can be created 1:1 or using a scale.

Drawing in 1:1 is convenient because you do not have to change the dimensioning scale; symbols for components are drawn directly 1:1 and they can be used again afterwards. The font size must be adapted. You need a page template with a standard sheet, where the cabinet layout fits 1:1. Change the dimensions in **Page Properties**.

Working with a scale is convenient because you can use the A3 standard sheet later. The symbols must be created in another page using the 1:1 scaling. For symbols insertion, set the scaling factor. The dimensioning scale must be changed. Font size can be maintained. Choose, in **Page Properties**, whether to work with a scale or not.

RR.1.1. WITH THE CABINETS MODULE

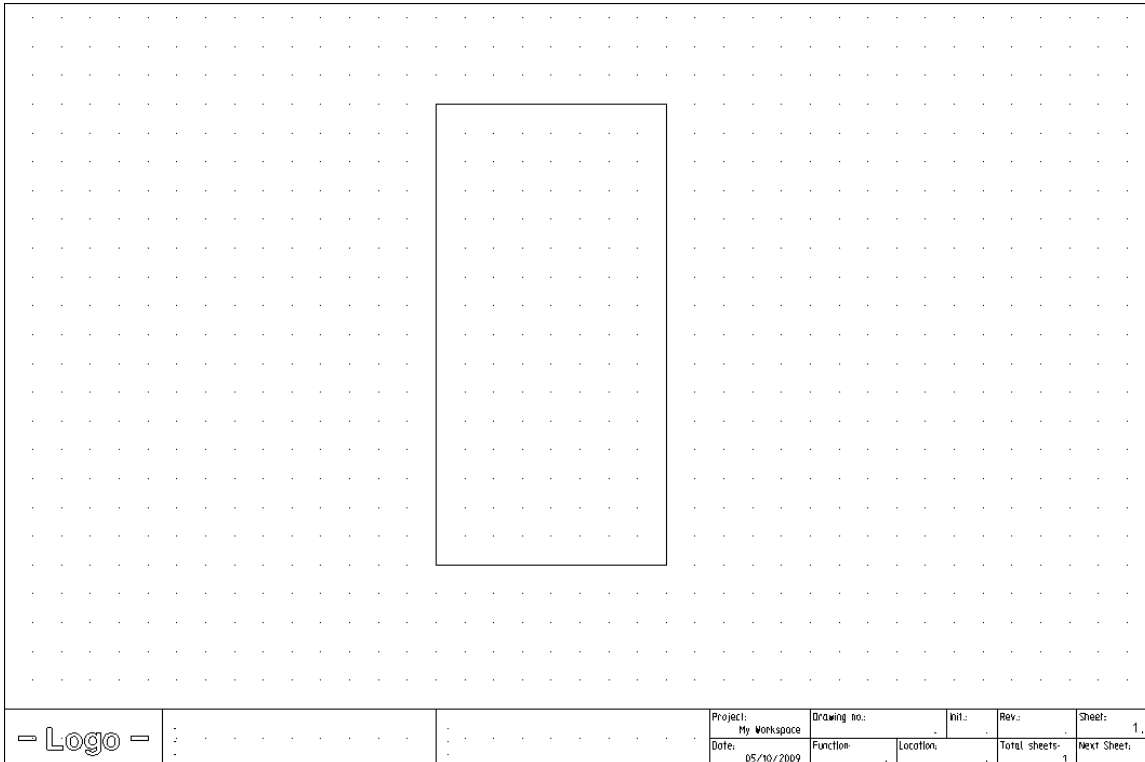
(Standard)

The following exercises illustrate the creation of a cabinet scaled 1:10.

Exercise 40-1: Create a new page within the Cabinets module and load an appropriate page template.

- Create the new page in the known way.
- Select the *Cabinets* module in the **Workspace Explorer**.

The default page template for Cabinets loads. The scale for drawing and the symbol scaling are 1:10 by default.

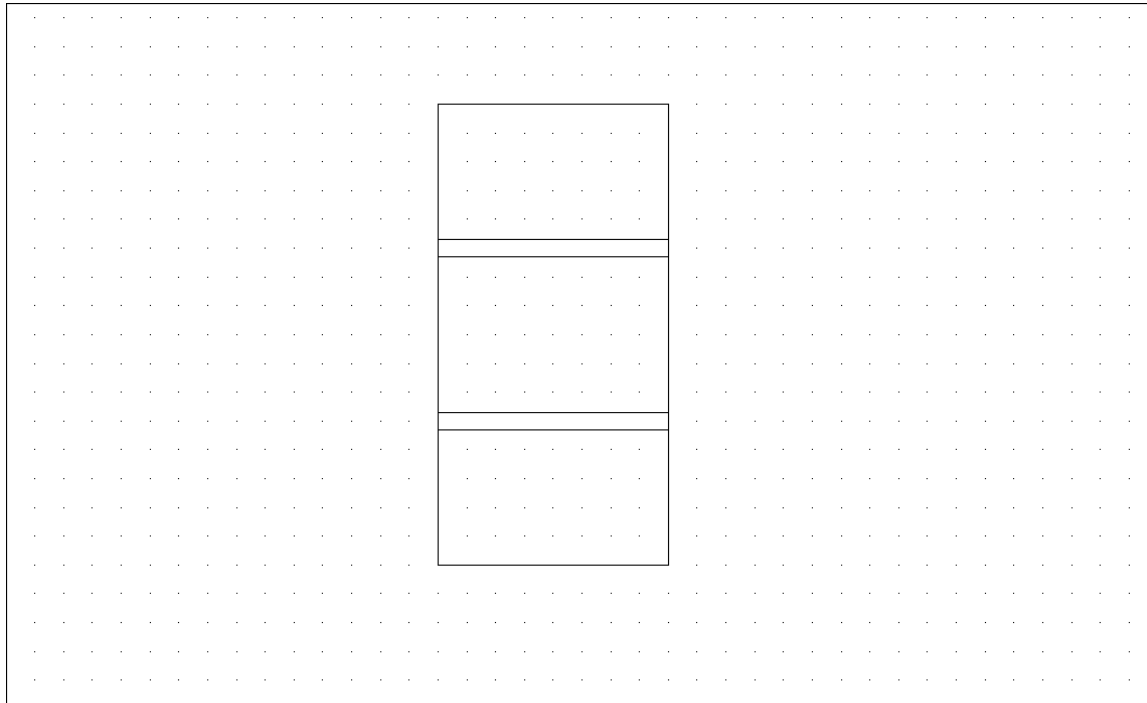
Exercise 40-2: Draw a panel 800 x 1600 mm.


In the Status bar (to the right under the drawing area), the dimensions are shown while drawing lines and rectangles. Set a grid size (at least 10x10 mm).

You can draw a Panel.

- 1.CA **Cabinet**
- 2.CO **Panel (Elements panel)**
- 3.+ Select the starting point of the panel.
- 4.# Press the space bar.
- 5.> dX
- 6.# 800
- 7.> dY
- 8.# 1600

Exercise 40-3: Draw a Cable/Wire Channel with a width of 60 mm and a length of 800 mm.



- Logo -			Project: My Workspace	Drawing no.: .	Init.: .	Rev.: .	Sheet: 1.
			Date: 05/10/2009	Function: .	Location: .	Total sheets: 1	Next Sheet: .

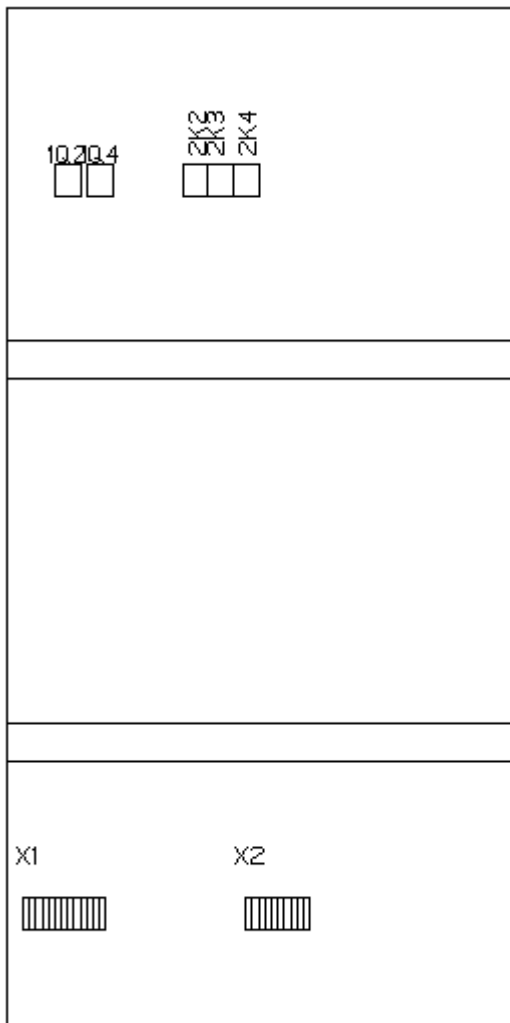
- 1.CA **Cabinet**
- 2.CO **Cable/Wire Channel (Elements panel)**
- 3.> Width
- 4.# 60
- 5.> Length
- 6.# 800
- 7.> **Insert Channel**
- 8.+ Place the cable channel in the desired place.
9. Place the cable channel again in another place.

Through the "+" or "-" keys in the numeric block of the keyboard you can rotate the cable channels (or other elements) before placing them if you did not enter the correct angle when defining the width and length.

Exercise 40-4: Insert components using the pick list that contains all the components from the circuit diagram.

When you insert a component, it is deleted from the pick list. The component name appears automatically next to the symbol.

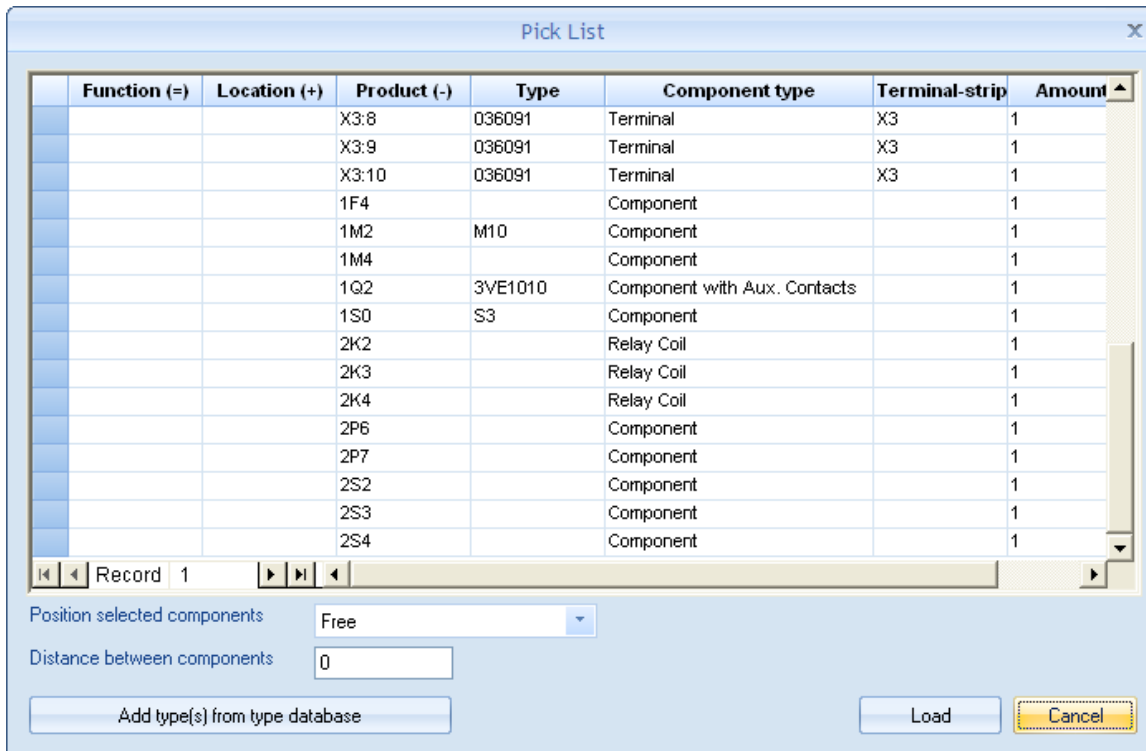
When you delete a component from the cabinet drawing, it appears in the list again.



Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

- 1.CA **Functions**
- 2.CO **Pick List (Other panel)**



Function (-)	Location (+)	Product (-)	Type	Component type	Terminal-strip	Amount
		X3:8	036091	Terminal	X3	1
		X3:9	036091	Terminal	X3	1
		X3:10	036091	Terminal	X3	1
		1F4		Component		1
		1M2	M10	Component		1
		1M4		Component		1
		1Q2	3VE1010	Component with Aux. Contacts		1
		1S0	S3	Component		1
		2K2		Relay Coil		1
		2K3		Relay Coil		1
		2K4		Relay Coil		1
		2P6		Component		1
		2P7		Component		1
		2S2		Component		1
		2S3		Component		1
		2S4		Component		1

Record 1

Position selected components: Free

Distance between components: 0

Add type(s) from type database Load Cancel

All of the components in the circuit diagram are displayed in this list.

- 3.> Double-click a component to select it.
 You can select several components using the standard *Windows* procedure for multiple selection. Click **Load** to insert the selected components into the cabinet drawing.
 In the "**Position selected components**" field, you can choose between **free**, **horizontally** or **vertically** aligned for the selected components to be inserted on the page.
 You can specify the distance between components. Afterwards, you can place the first component.
- 4.+ Place the first component.
 Place the other components.

If you place a lot of devices horizontal or vertical - especially terminals - you might like to hide the texts of the symbols.

Pressing the h key while inserting them hides the component name.

Pressing H hides all component names and connection texts. The option is useful especially for terminals that are made by rectangles only.

Exercise 40-5: The relation between devices in Circuit diagrams and Cabinets is possible if you assign a type.

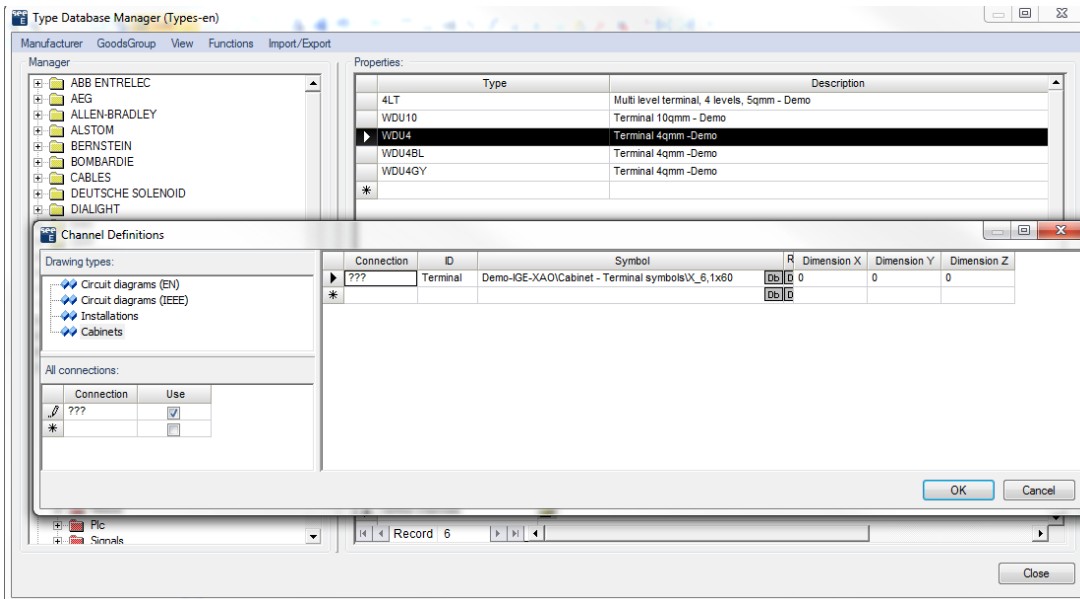
In the type database, you can set the "*Width*" and "*Height*" properties of the rectangle for the cabinet layout.

If width and height are not specified for a type, a rectangle identified with a yellow diagonal cross appears. This way you are warned that the size of this component is not real.

If you want a detailed view of your construction symbols, you can design a graphic in scale 1:1 or you can import from the *DXF-DWG* format. The symbols for *Cabinets* must satisfy the same criteria as the symbols for *Circuit diagrams*, i.e. a symbol for relay coil must be assigned to the category of relay coils. You can assign symbols to a type in the type database in channel definition as shown in exercise below..

Exercise 40-6: Create a terminal symbol for the cabinet.

1. Create a new page within Cabinets.
Create symbols on the new page.
2. Type the value 1 for scale and Symbol scaling as the symbols must be created with a scale 1:1.
3. Create a rectangle with a size 52x6.
4. Group the graphics into a Terminal symbol.
5. Select **Edit > Text > Edit Text** and change the size of the text for the component name to 35mm. As the symbol is minimized by the factor 10 after the insertion, it applies to texts too.
6. Move the text of the component name one position above the terminals (use the **Select single element** and **Move** commands)
7. Double click the symbol and switch off the visibility of all texts.
The symbols for terminals are so small that the texts will overlap.
8. Insert the symbol into the MySymbols symbol database in the Cabinet folder. Type the Training-terminal name.
9. Go to the type database. Create a new terminal type, for example UK10-Training, assign the Symbol name for Cabinets property to it. In the Value column, enter the following:
<Symbol database>\< Folder >\<Symbol name>, in our example MySymbols\Cabinet \Training- terminal
10. If you have already placed terminals in the cabinet, delete them.
11. Assign the type UK10-Training to the terminals.
12. Place the terminals again.
The symbol is now used.
13. For a terminal of a terminal strip, you can set to view the terminal strip name by double-clicking the terminal and activating the "Show" option for the component name.



Exercise 40-7: Use several cabinet symbols in one component.

It is possible to insert cabinet symbols for all types of components in case more than one type of component has been added. In the channel definition, in the "**Cabinet symbol**" field, you have to define a symbol for the cabinet or the size of the rectangle or circle to be generated.

The sizes are defined in the following way:

33x30 (for a rectangle)

D30 (for a diameter)

or

R15 (for a radius).

If subtypes have been used, the symbols attached to them in the channel definition are also used.

If a component has multiple types and, in the channel definition, a cabinet symbol or a dimension has been defined for more than one for these types, all the symbols/rectangles for the component can be inserted in a cabinet. In this case you can select only one component and insert the single parts. There is no general rule how the other types have to be inserted in relation to the first one. If you select multiple parts and use horizontal or vertical automatic placing, the automatism is interrupted when you use a component with multiple symbols. You have to insert all the symbols for this component and after this for the next component, to define where to start the automatism again. Then the automatism continues.

Example

- if a coil consists of the coil itself and two add-ons, one to be mounted on left and the other to be mounted on right side, you have to insert first the add-on for the left side, then the coil and after that the add-on for the right side.

- if a coil consists of the coil itself and an add-on to be inserted on top of the coil, you must insert the coil first and then the add-on.

If you have to insert the first type of a component via the picklist and skip the others, the component is not present in the pick list any more (because the pick list contains only symbols inserted in the circuit diagram and not in the cabinet). Then you have to use the **Complete database component** command to complete the component.

If symbols for the cabinet are used, it is really important they have as much connection points as defined in the first line in the channel (where is defined the cabinet symbol).

If you have to define a cabinet symbol, for example, in the first line in the channel definition and also in the second one, the one defined in the first line needs to contain as much connection points, as defined in the first line, and the cabinet symbol attached to the second line needs to contain as much connection points, as defined in the second line etc.

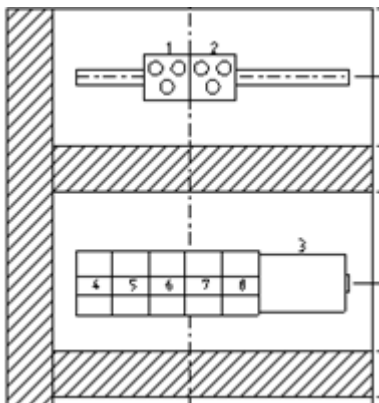
Hiding component names in case multiple symbols are used for a component

If you press the **H** key on the keyboard while placing the symbols, the component name is hidden for all the following symbols. If you want to display the name for the next symbol again, press the **H** key again.

It is possible to switch on or off the displaying of the name in the **Component Properties**.

Exercise 40-8: By means of a section, delete again all components in your circuit diagram.

Exercise 40-9: Draw the rails in your cabinet.



- | | |
|-----|--|
| 1.K | Cabinet |
| 2.M | Rail (in the Elements filed) |
| 3.> | Width |
| 4.# | 35 |
| 5.> | Length |
| 6.# | 700 |
| 7.> | OK |
| 8.+ | Place the rail. |
| 9.+ | Place the rail a second time and, if necessary, a third time. |
| 10. | Through the Pick list command, place components on the rails. |

Exercise 40-10: Delete the components from the rails and move components from one rail to another.

Components are fixed on the rails, you cannot select them directly.

1. Right click.

The context menu appears.

- 2.M Select a single element in the component group.
- 3.+ Click the component to be selected.
- 4.+ Press the **CTRL** button if you want to select more components.
5. Delete the components.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

If you want to select more components, you can do this by the means of a section, as you do the following:

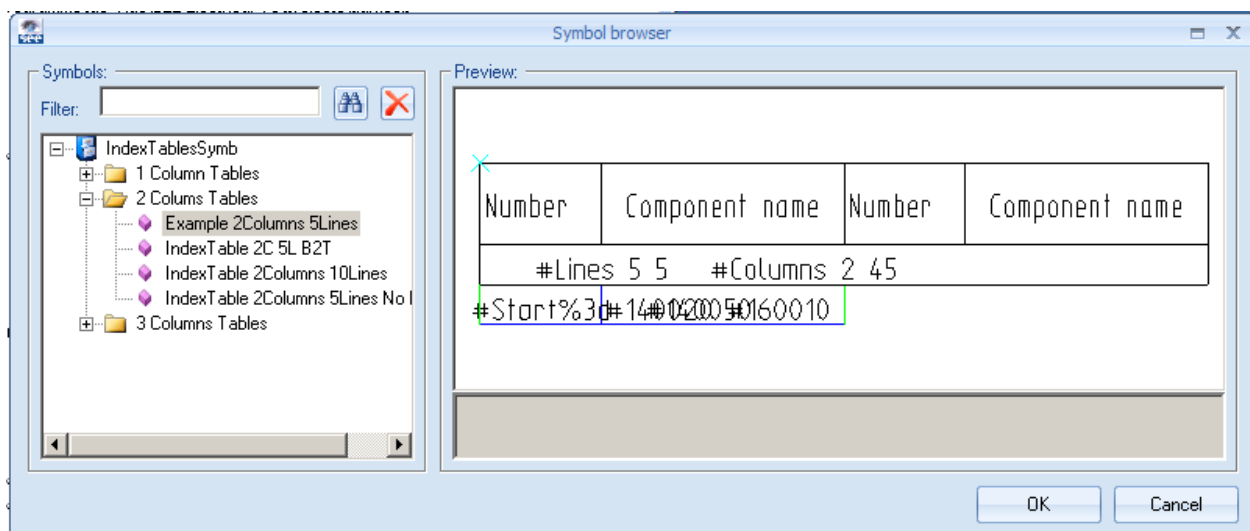
- 1.# Simultaneously, press the SHIFT and CTRL buttons on the keyboard.
- 2.+ Define the first point of the section.
- 3.+ Define the second point of the section.
4. Through the **Move selected elements** pop-up command, now you can move the components on another rail.

Exercise 40-11: Create index.

It is possible to insert an index table into the cabinet drawing.

The index is created with the help of user defined templates. The templates are located in the "IndexTableSymb.ses" symbol library by default.

If you activate the command on a page that has components on it, *SEE Electrical* offers all templates that are present:



Make your choice in the Symbol browser and click **OK**.

The Index table is generated and you have to insert it with the cursor.

If the number of components exceeds the number of the places available in the template, an error message appears. Delete the index and choose another template.

The index can be deleted like any other object.

If an index is already present in the drawing, the **Index components** command updates the existing index.

The component name is replaced by the ordinal number from the index. The component name is still present in the component, but it is automatically hidden.

In the "**Cabinet Properties**" there is a setting that allows you to define if each terminal is shown in the index table or only one entry for the whole strip.

Creation of templates for indexes:

The template is created using geometry and specific texts.

- #Lines <Number of lines> <Distance between lines>

This value sets number of lines and distance between lines, as well as Top to Bottom, or Bottom to Top direction of the drawing (negative distance -> Bottom to Top).

The text has to have the attribute "Normal". It can be placed at any position in the template.

Examples: #Lines 30 4 or #Lines 30 -4

- #Columns <Number of columns> <Distance between columns>

This value is only necessary, if you want to use more than one column.

The text has to have the attribute "Normal". It can be placed at any position in the template.

Examples: #Columns 2 36

- #Start <formatting attribute for ordinal number>

The text has to have the attribute "Normal". The %2d formatting attribute formats the ordinal number with minimum two digits, for ex. 01, 02, ..., 10, ..., 99, 100, ..

- Add texts with the text attribute "Normal" and with the component IDs you want to be displayed.

#160010 component name or terminal name/separator/number

#140020 component function

#140050 component location

- Add geometry.

You have to put the lines for the single cells that contain the index information on layers 450, 451 and 452.

The lines for the header must not be on these layers.

Header: geometry not
on layer 450, 451 or 452

Number	Name
#Lines 10 4	

Cell: geometry on
layer 450, 451 or 452

#Start%.2d#1400#140050#160010

Block the geometry and the text as a symbol of "Cabinet, Index component table" type. Store the symbol in the "IndexTableSymb.ses" library.

Exercise 40-12: Save your workspace.

RR.1.2. WITHOUT THE CABINETS MODULE

You will create the cabinet scaled 1:1.

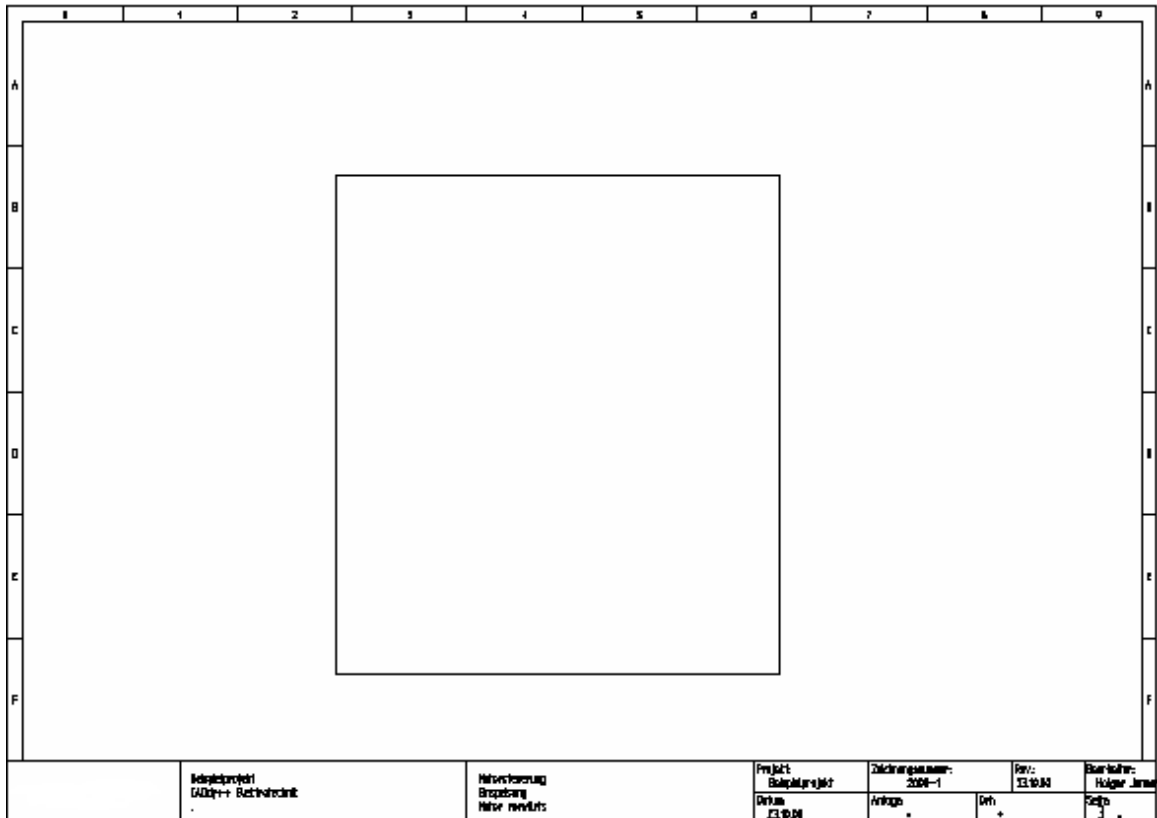
Exercise 40-13: Create a new page and load an appropriate page template.

1. Create the new page in the known way.
2. <Page>
Choose the new page in the Workspace.
Load the page template A3x5. It represents a page of A3 format enlarged by the factor 5.
This page allows you to create a Cabinet with the maximum size of 1500 x 1250 mm.
- 2.CA **File**
- 3.CO **Open**
- 4.CA **Page template**
- 5.> <Your template>, for example A3x5
The setting for scale and symbol are found in the page properties.
- 6.> **OK**
The page template has been loaded. The workspace information and the page information have been saved in the standard sheet.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

Exercise 40-14: Draw a panel 800 x 900 mm.



In the status bar (right beneath the drawing area), the dimensions are displayed as you are drawing the lines and rectangles.

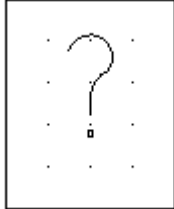
Choose a larger grid through the Toolbars (at least 10x10 mm.)

- 1.CA **Draw**
- 2.CO **Rectangle (Elements panel)**
- 3.+ Select the first point of the rectangle.
- 4.+ Select the second point of the rectangle.
Look at the dimensions in the status bar.

Exercise 40-15: Draw a rectangle 40 x 50 mm.

- 1.CA **Draw**
- 2.CO **Rectangle (Elements panel)**
- 3.+ Select the first point of the rectangle.
- 4.+ Select the second point of the rectangle.
Look at the dimensions in the status bar.

Exercise 40-16: Place one text for the component name.



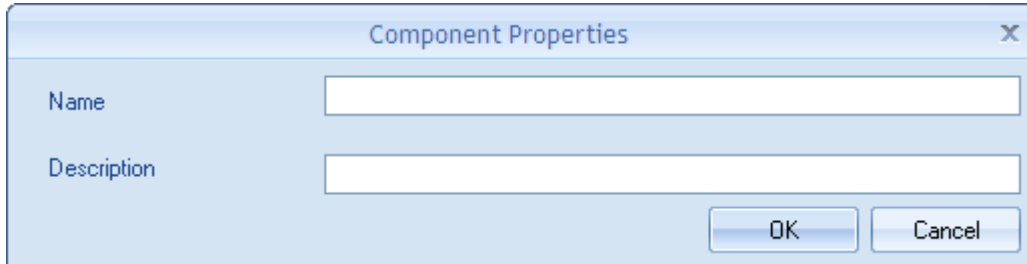
- 1.CA **Draw**
- 2.CO **New Text (Elements panel)**
- 3.> Height
Change the height for the text because the component and the text will be reduced when printing later. Text with height 3.5 would be too small.
- 4.# 35
- 5.> Centre justified
The text will be centred.
- 3.# ?
Type in a question mark "?" for the text.
- 4.+ Place the text within the component rectangle.

Exercise 40-17: Group the rectangle and the text into a symbol and save in the symbol database.

- 1.CA **General.**
- 2.CO **Normal (Select panel)**
- 3.+ Select the first point of the frame. The rectangle and the text must be entirely inside the frame.
- 4.+ Select the second point of the frame.
- 5. Right-click with the mouse
- 6.CO **Block**
- 7.> Block/Macro/Group
Select the symbol to be created.
The elements are integrated into a group.
Save the group in a new folder of the *MySymbols* symbol database.
- 8. Click the Symbols tab.
- 9. MySymbols
Double-click *MySymbols* to open it.
- 10. Right-click with the mouse.
- 11.CO **New Folder**
- 12.# Symbols for cabinets
Enter the name of the new Symbol folder.
- 13.+ Select all parts of the symbol using a frame.
All elements must be within the frame.
Select the first point of the frame.
- 14.+ Select the second point of the frame.
- 15.+ Drag the symbol into a Symbols library for cabinet symbols. Hold the left mouse button pressed while dragging.

Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

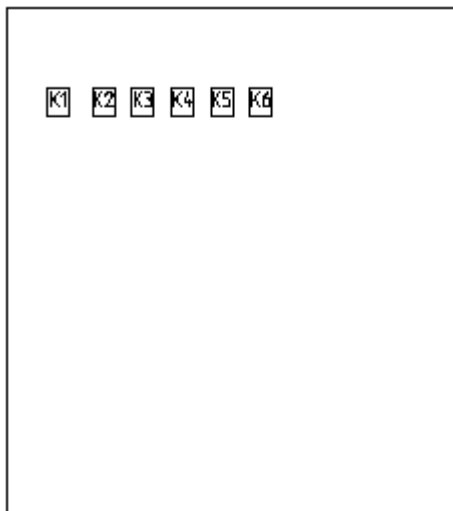


The image shows a 'Component Properties' dialog box with a title bar containing a close button (X). Inside the dialog, there are two text input fields: 'Name' and 'Description'. At the bottom right, there are two buttons: 'OK' and 'Cancel'.

- 16.> Name
- 17.# Power supply 2:
Enter the name of the group.
- 18.> **OK**
The group is stored in the symbol database.

Exercise 40-18: Insert the component in multiple places in the panel.

For example:

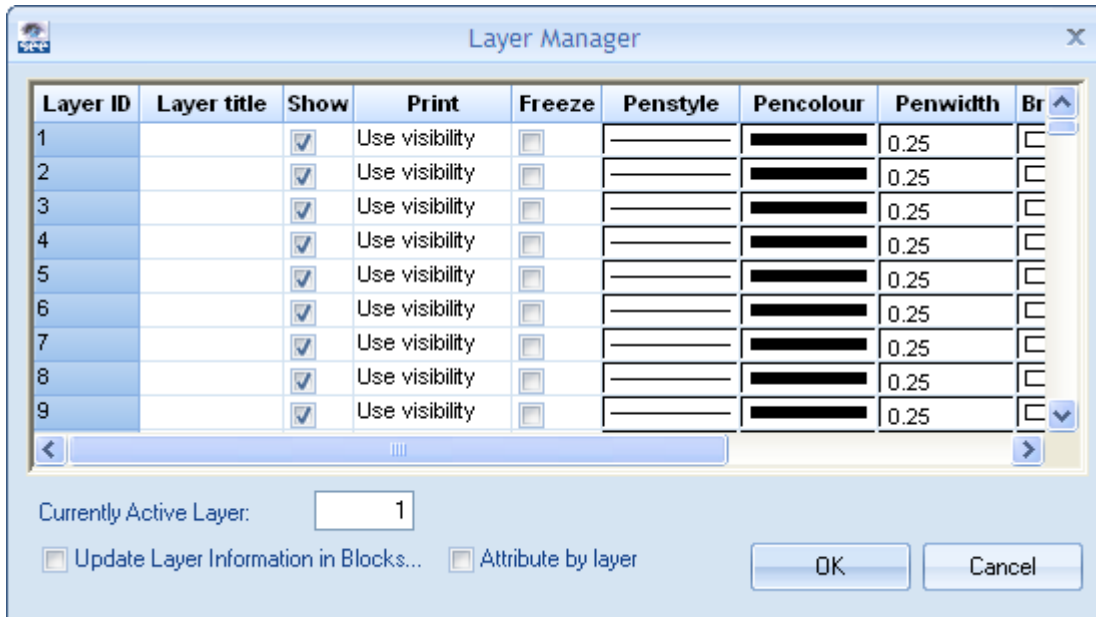


- 1.+ Insert the component from the symbol database into several places on the page.
- 2. Finish the placement.
- 3.+ Double-click the text and change it.
- 4.> **OK**
Change the next text.
- etc.

RR.2. USING LAYERS

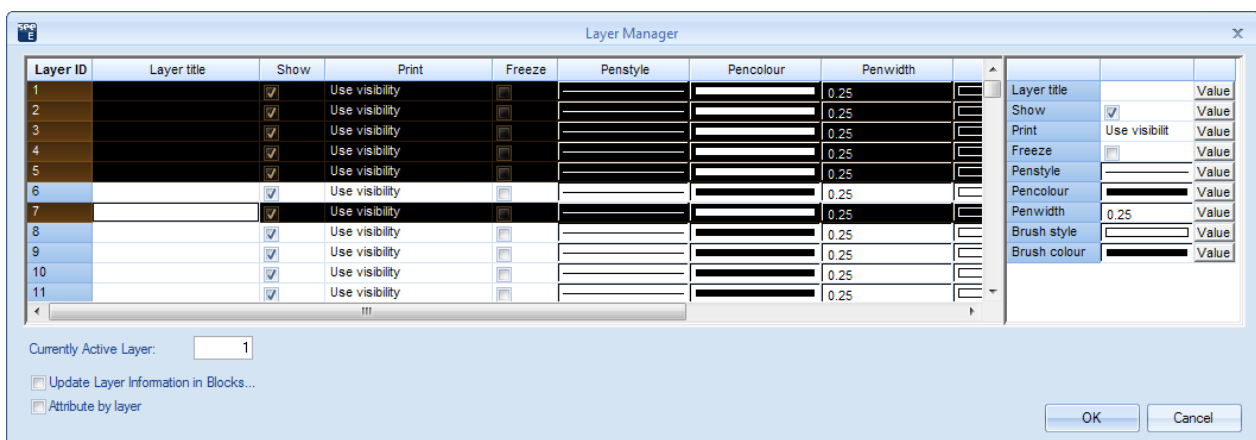
SEE Electrical provides 512 layers for structuring drawings. You can use layers, for example, for hiding some data while printing.

You can change the settings for layers by clicking on the  icon.



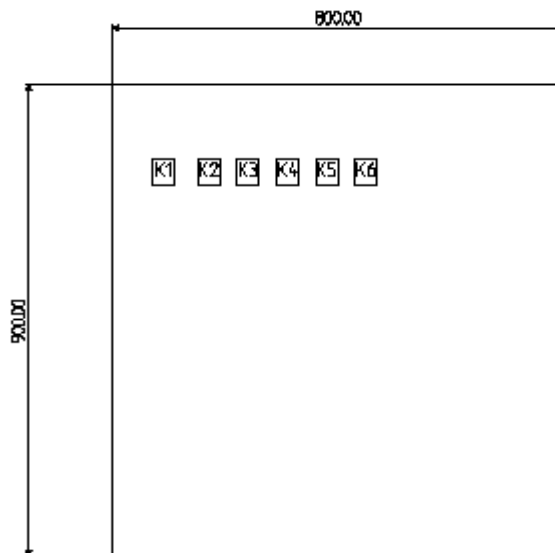
It is possible to select multiple layers and change their attributes in one step. The attributes can be changed for selected layers (by using the SHIFT or CTRL keys) or for all layers (by clicking on the "**Layer ID**" field).

When multiple layers are selected, an editor appears in the right area of the **Layer Manager** window. It allows managing of the layer attributes such as visibility, color, line type etc.

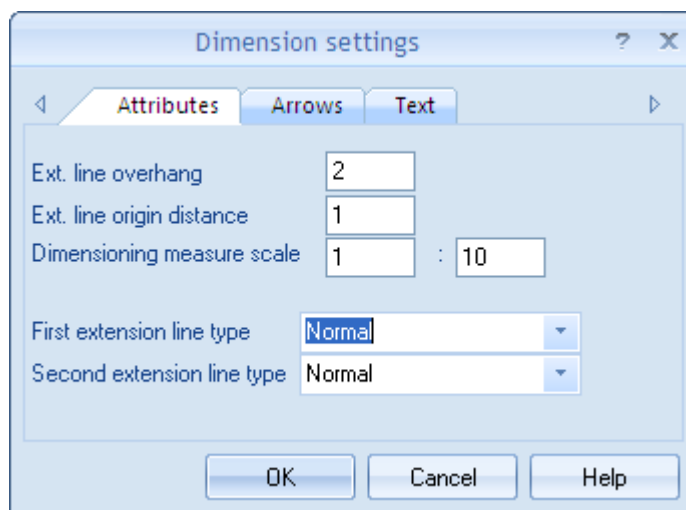


RR.3.DIMENSION

Exercise 40-19: Define dimensions for the panel. Adjust the settings for the dimensions.

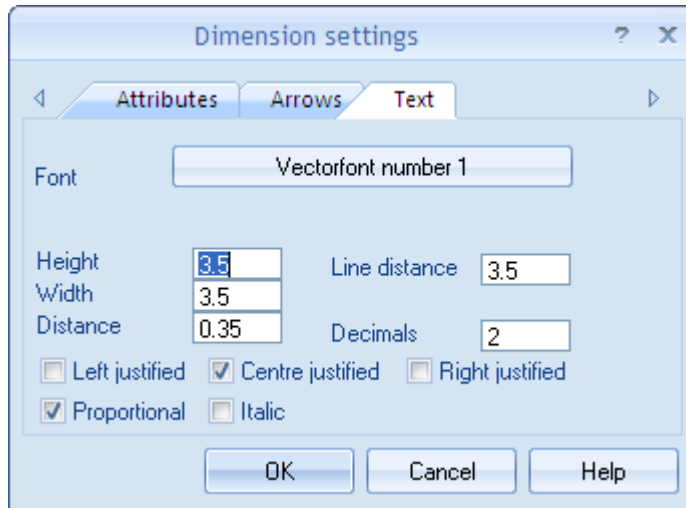


- 1.CA **Draw**
- 2.CO **Dimension**
- 3.CO 



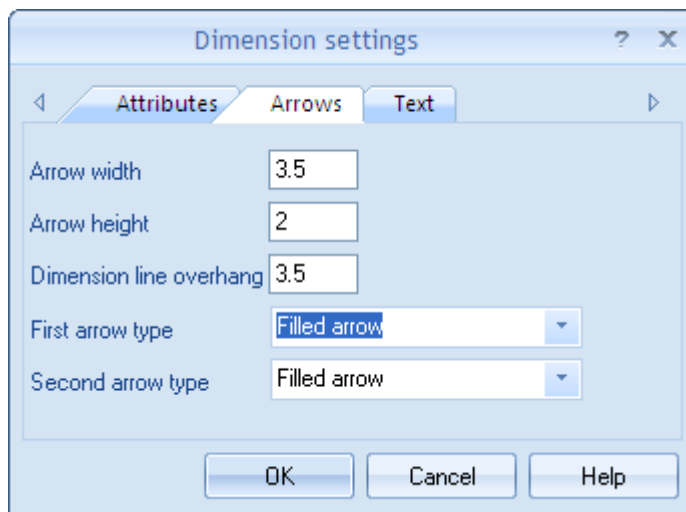
- 4.T **Text**

Click the **Text** tab.



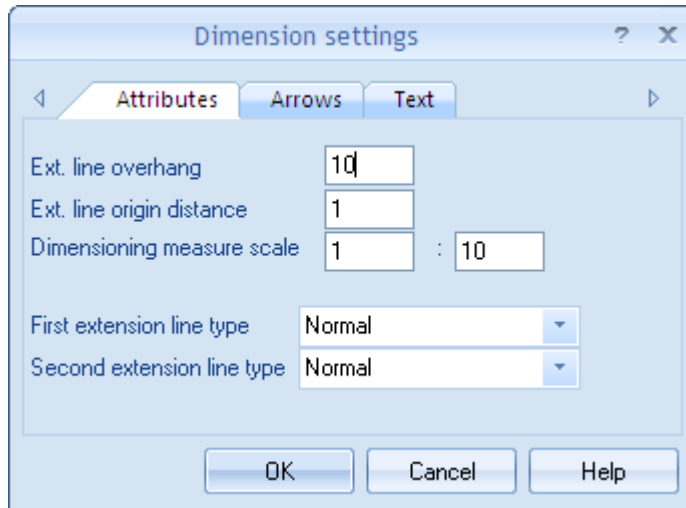
5.> Height
6.# 35
7.T Arrows

Click the **Arrows** tab.



8.> Arrow width
9.# 10
10.> Arrow height
11.# 10
12.> Dimension line overhang
13.# 15
14.T Attributes

Switch to the **Attributes** window.



- 15.> Extension line overhang
- 16.# 10
- 17.> **OK**
- The settings have been set.
- 18.CA **Draw**
- 19.CO **Dimension**
- 20.CO **Btw. 2 Lines**
- 21.+ Identify the left border line of the panel.
- 22.+ Place the dimension line by clicking with the mouse.
- 23.+ Identify the right border line of the panel.
- 24.+ Right-click to exit the drawing mode
- 21.+ Identify the top border line of the panel.
- 22.+ Place the dimension line by clicking with the mouse.
- 23.+ Identify the right border line of the panel.

Exercise 40-20: Save the workspace.

- 1.CA **File**
- 2.CO **Save**

RR.4.COMPARISON BETWEEN CIRCUIT DIAGRAMS AND CABINETS

(Cabinets module)

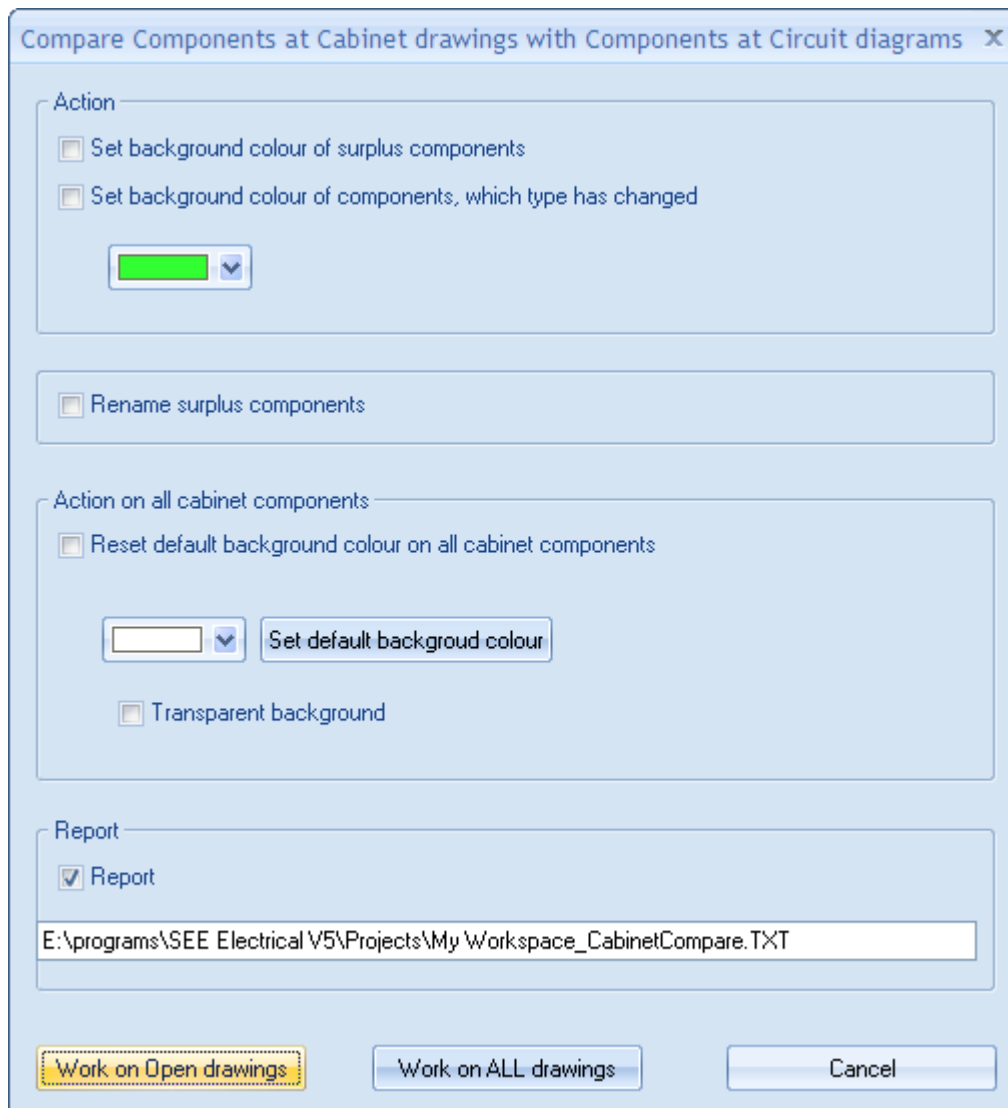
If you have deleted components in the circuit diagram, they must also be deleted from the cabinet drawing. If types have been changed in the circuit diagram, they must respectively be changed in the symbols from the cabinet drawings.

If an existing cabinet drawing is added from one project to another project, the equipments which exist in the circuit diagram must be assigned to the equipments in the cabinet.

Exercise 40-21: Delete a piece of equipment from the circuit diagram, which you have already inserted in the cabinet drawing.

Open the page in the Cabinets module, where the deleted equipment has been inserted.

- 1.CA **Cabinet**
- 2.CO **Components Comparison (Functions panel)**



The dialog box is titled "Compare Components at Cabinet drawings with Components at Circuit diagrams". It contains several sections for configuring the comparison process:

- Action:**
 - ☐ Set background colour of surplus components
 - ☐ Set background colour of components, which type has changed
 - A color selection box showing a green color.
- ☐ Rename surplus components
- Action on all cabinet components:**
 - ☐ Reset default background colour on all cabinet components
 - A color selection box showing a white color.
 - A button labeled "Set default background colour".
 - ☐ Transparent background
- Report:**
 - ☒ Report
 - A text field containing the path: "E:\programs\SEE Electrical V5\Projects\My Workspace_CabinetCompare.TXT"

At the bottom, there are three buttons: "Work on Open drawings" (highlighted with a yellow border), "Work on ALL drawings", and "Cancel".

- 3.> Set background colour of surplus components
- 4.> Work on Open drawings
Click the **Work on Open drawings** or the **Work on All drawings** button to define the pages which will be processed. The differences are displayed and the corresponding components appear selected.
- 5.#
If you wish to delete all the marked components, press the **Del** key on the keyboard. Do not click on the drawing before that, because this will cancel the selection on all the surplus components.
The **Undo/Redo** command is available only in case you work on the open drawings.
Please consider this before executing the **Cabinet > Functions > Components Comparison** command on your workspace.
If you do now wish to delete the surplus components, but to check them, you can reset their background colour. To do this, tick the "**Reset background colour on all cabinet components**" option in the **Components Comparison** dialogue. Click again the **Work on Open drawings** or the **Work on All drawings** buttons, as desired.

Exercise 40-22: In the circuit diagram, change the type for a component which has already been inserted in the cabinet drawing. Open the page in the Cabinets module where the component is inserted.

- 1.CA **Cabinet**
- 2.CO **Components Comparison (Functions panel)**
- 3.> Set background colour of components which type has changed
- 4.> Work on Open drawings
Click the **Work on Open drawings** or the **Work on All drawings** button to define the pages which will be processed. The differences are displayed and the corresponding components appear selected.
- 5.#
If you wish to delete all the marked components, press the **Del** key on the keyboard. Do not click on the drawing before that, because this will cancel the selection on all the surplus components.
The **Undo/Redo** command is available only in case you work on the open drawings.
Please consider this before executing the **Functions > Components Comparison** command on your workspace.
If you do now wish to delete the marked components, but to check them, you can reset their background colour. To do this, tick the "**Reset background colour on all cabinet components**" option in the **Components Comparison** dialogue. Click again the **Work on Open drawings** or the **Work on All drawings** buttons, as desired.
- 6.CA **Functions**
- 7.CO **Types**
- 8.CO **Clear Old Properties**
You must now delete the information about the modifications of the component types. Once a component type has been changed in the circuit diagram, the component is marked internally. If you have already checked the components with modified types, you need to delete the internal marking, using the **Clear Old Properties** command. After its execution, a report file appears in the *Projects* directory with the name <Project name>_TYPE and the TXT extension. In our example, the created file is *My Workspace_Type.TXT*.

Exercise 40-23: Create another project, similar to your training workspace. Copy the cabinet drawing into this project from your training workspace.

- 1.CA **Cabinet**
- 2.CO **Components Comparison** (**Functions** panel)
- 3.> Rename surplus components
- 4.> Work on Open drawings
Click the **Work on Open drawings** or the **Work on All drawings** button to define the pages which will be processed.
Components that exist both in the *Circuit diagrams* and the *Cabinet* with the same database type, but with different names, are updated.

SS USEFUL TOOLS

SS.1. COMPRESSING PROJECTS

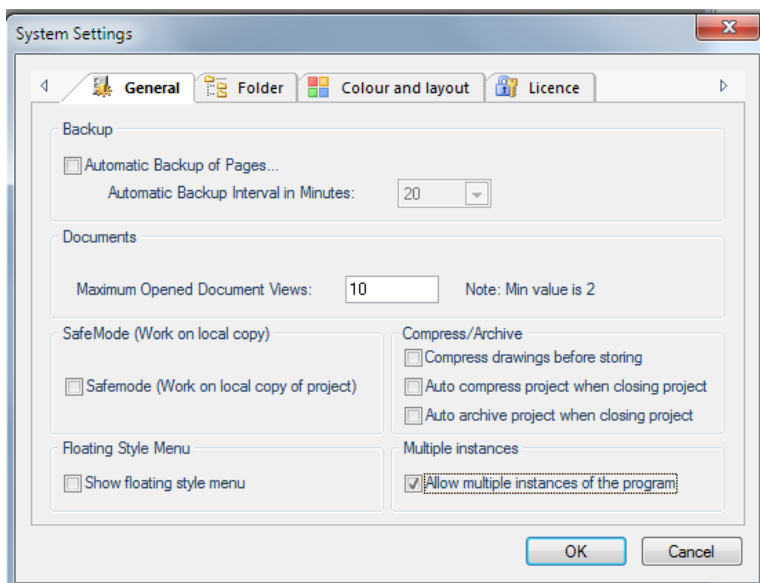
Projects in *SEE Electrical* are *MS Access* database files.

Because of the specific *MS Access* properties, the objects inserted in the database remain there even if they are deleted. They are not really removed, unless the database is compressed. You should do this exercise periodically to avoid that your projects become extremely large and slow to manipulate.

Exercise 41-1: Compress your workspace.

1. Close all open workspaces.
The command allowing you to compress a workspace is only accessible when all workspaces are closed.
- 2.CA **File**
- 3.Co **Compress...**
A dialogue appears, containing all existing workspaces.
- 4.> <Project name>
Select the project you wish to compress.
- 5.> Open
The selected project is compressed.

It is possible now to compress workspaces each time they are closed. This functionality is activated via the "**Auto compress project when closing project**" option in the **System Settings** window.



SS.2. REORGANIZE WORKSPACES

All data in *SEE Electrical* is saved as *Windows Compound Files* within a *Microsoft Access*® database. Occasionally data within the database is out of synchronization with the content of the drawings.

The "**Regenerate Database**" functionality refreshes the contents of the *Microsoft Access*® database.

1. Open a page from the workspace
2. **File ► Regenerate Database**
3. A dialogue box appears asking you if you want to update the database from the pages and then store your project or if you want to update the pages from the database.

Yes: Cross-references are updated automatically from the database. Entries in the database lists are created automatically, too. If you establish that errors have occurred, i.e. automatically created entries do not correspond to the information in the drawing, the database has to be regenerated. The workspace will be automatically saved afterwards.

No: If you have changed something via a database editor and your modifications do not appear automatically in the drawing, then the drawings must be updated from the database.

Cancel: Aborts the process.

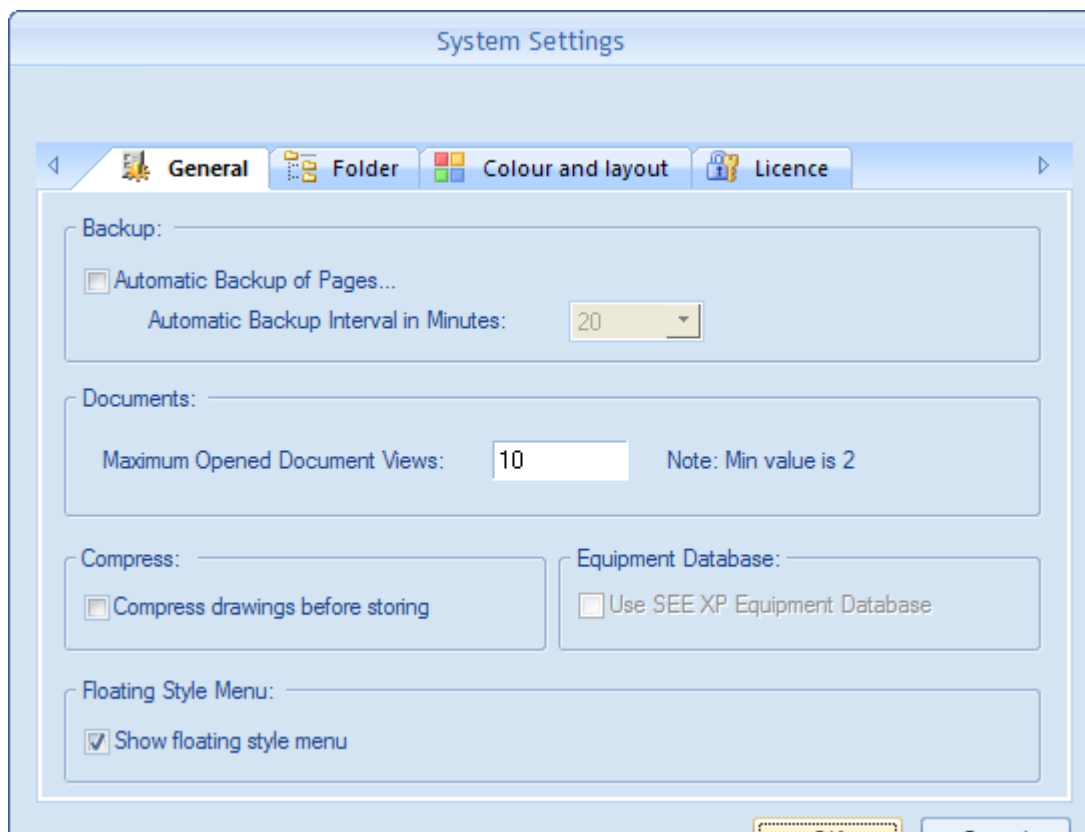
TT SETTINGS

TT.1. SYSTEM SETTINGS

The system settings apply to the system, i.e. to all projects and drawings. The system settings are stored in the *Windows* registry.

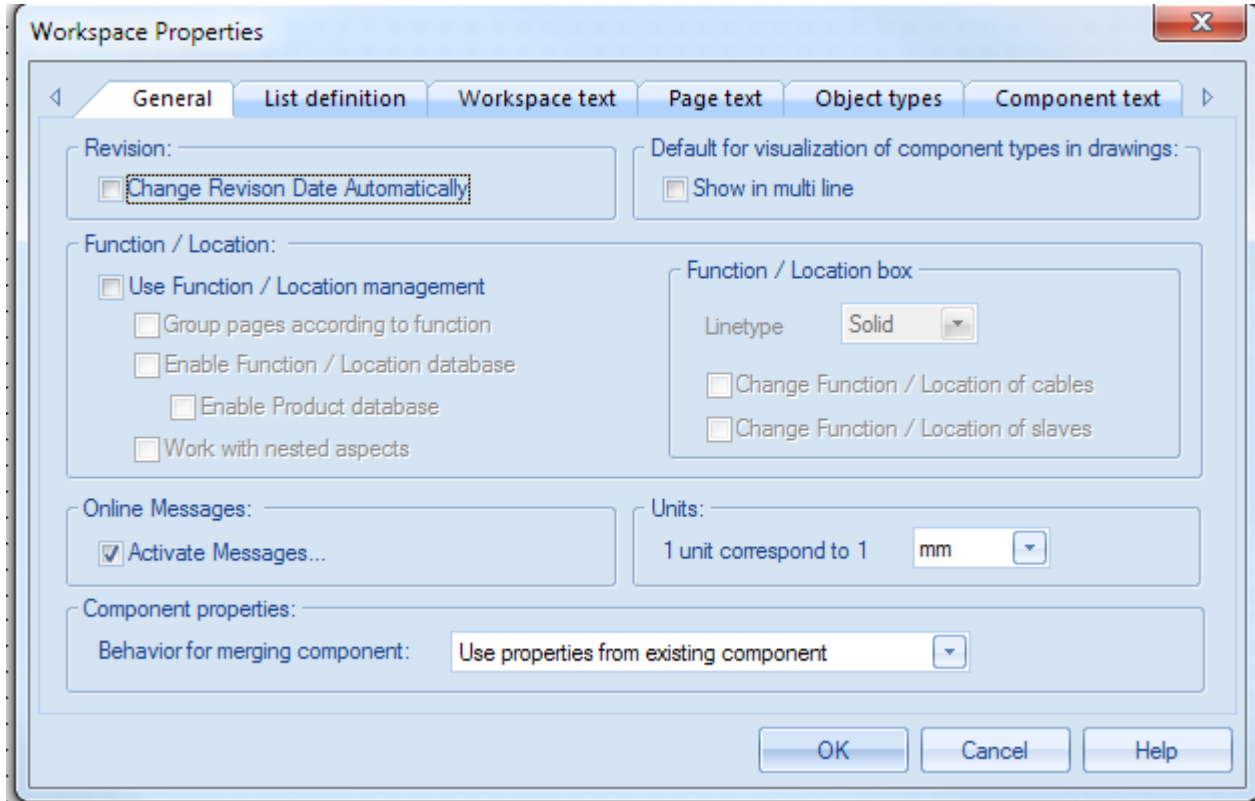
The **File** ➤ **System Settings** function is available only when no project is open.

Choose, in the **System Settings** window, whether to make automatic Backup of Pages or not and set the maximum number of opened document views.



Click the **Folder** tab to define the folders where to store your templates, projects and symbols (for example, they could be stored in the network). Use the settings in the **Colour and Layout** tab for the colours for the background, for the cursor, etc.

TT.2. WORKSPACE PROPERTIES



The **Workspace Properties** window is available for all levels.

The settings in the **General** tab regarding the Function/Location Management are available for the **standard** level and are described in the "Function and Location" chapter.

The "**Enable Function/Location database**" and "**Enable Product database**" options are available in **advanced** level and are described in the **AA. Advanced Function/Location and Product Management** chapter.

The other tabs are available for **advanced** level and are described in the "Customizing the Workspace/Page Information Windows" chapter.

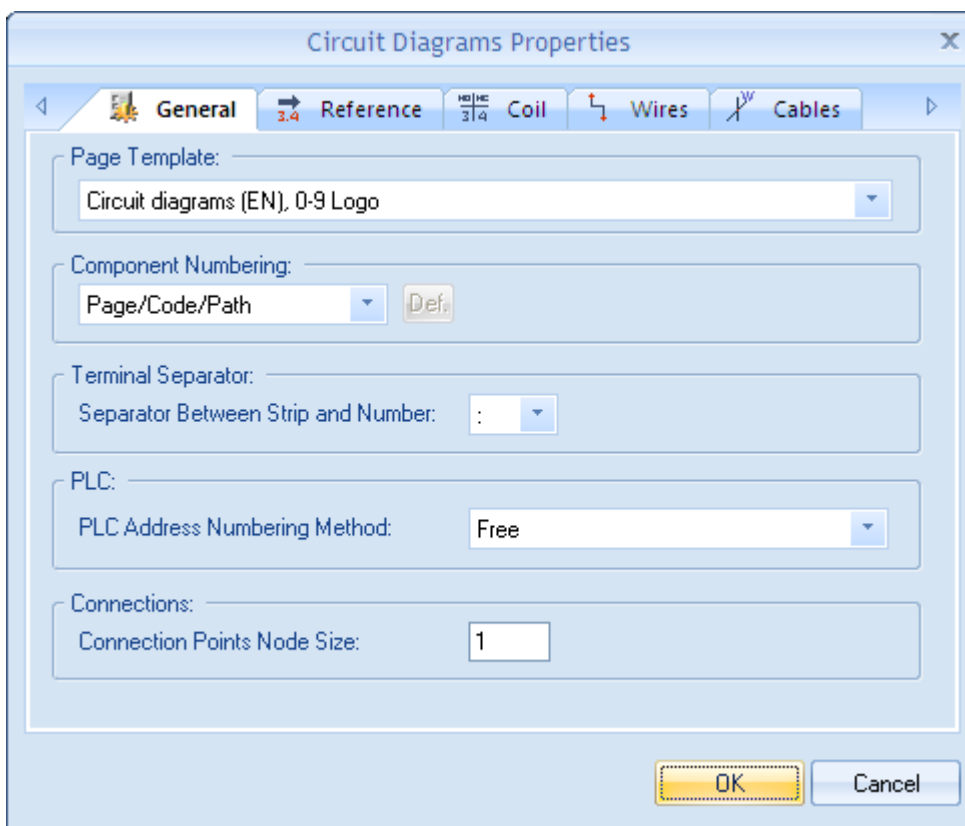
TT.3. PROPERTIES FOR CIRCUIT DIAGRAMS

The circuit diagram properties apply to the whole project. They are defined when you create the workspace template. The properties are loaded when you create a project by using the workspace template, and they are saved then within the project.

You can select the **Properties** function as follows:

Select *Circuit diagrams* in the **Workspace Explorer**, then select **Home > Properties > Module** or

Select *Circuit diagrams* in the **Workspace Explorer**, then right-click with the mouse and select **Properties**.



Here you can choose the page template, the component numbering method and the text parameters for the cross-references creation.

Under *SEE Electrical standard*, you can set to display cross-references for relay coils in contact cross or contact mirror, and whether to show wire numbers in the drawing or not. You can choose which format for the cross reference will be used.

The circuit diagram properties can be changed at any time, for example, if another page template must be used for some projects. The page template will be used then for creating a new page. It is not recommended to change the setting of the component numbering during the project processing because the components numbered before making the change keep their old numbers unless they are renumbered. The modification of the "Component numbering" property must be made before drawing the circuit diagrams.

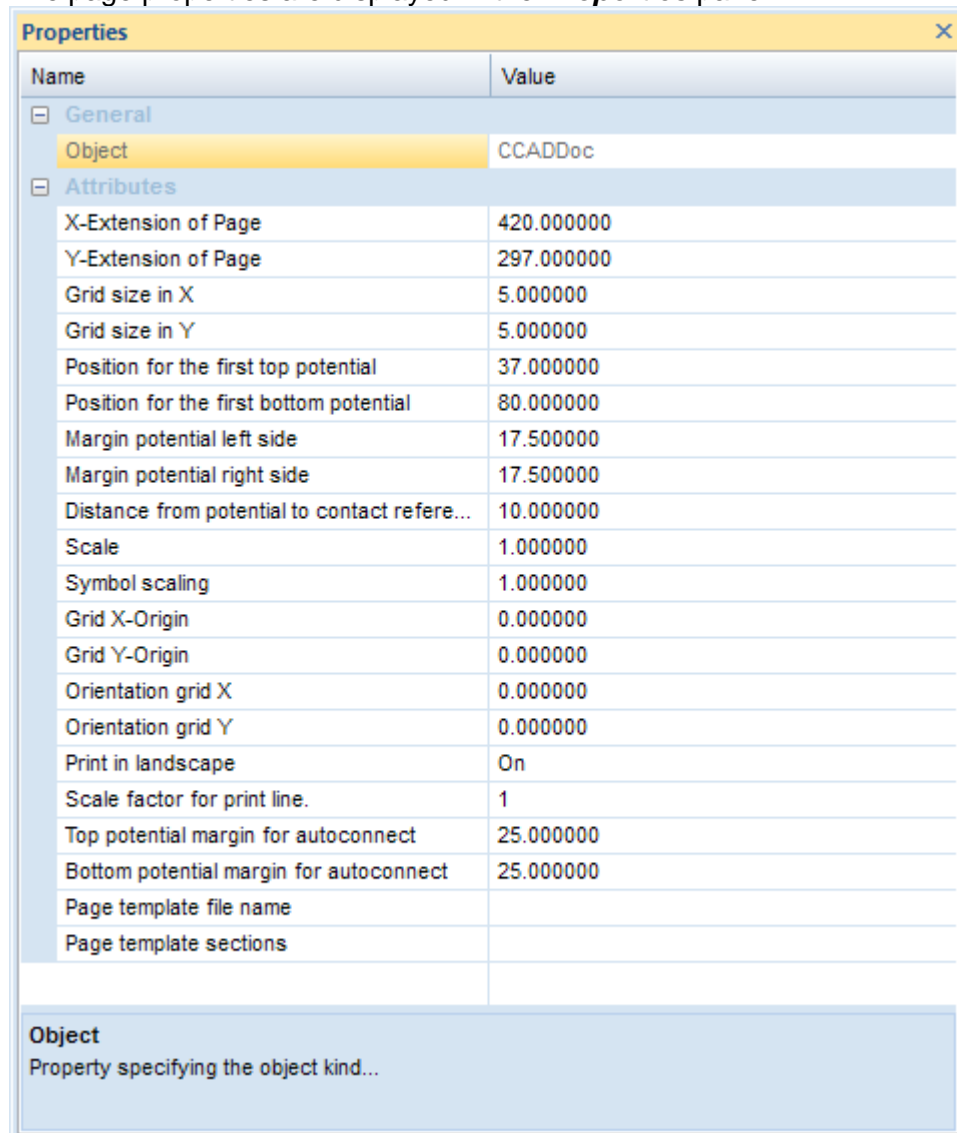
TT.4. PAGE PROPERTIES

The page properties apply to a page of the project. The page properties are specified when you create a page template. When you create a new page, the page properties are loaded from the page template and then they are stored for this page within the project.

You can select the **Page properties** function in two ways:

- Select a page under Circuit diagrams in the Workspace Explorer, and then select **Home** ➤ **Properties** ➤ **Page**.
- Select a page under Circuit diagrams in the **Workspace Explorer**, right-click with the mouse and select **Properties....**

The page properties are displayed in the **Properties** pane.



Name	Value
General	
Object	CCADDoc
Attributes	
X-Extension of Page	420.000000
Y-Extension of Page	297.000000
Grid size in X	5.000000
Grid size in Y	5.000000
Position for the first top potential	37.000000
Position for the first bottom potential	80.000000
Margin potential left side	17.500000
Margin potential right side	17.500000
Distance from potential to contact refere...	10.000000
Scale	1.000000
Symbol scaling	1.000000
Grid X-Origin	0.000000
Grid Y-Origin	0.000000
Orientation grid X	0.000000
Orientation grid Y	0.000000
Print in landscape	On
Scale factor for print line.	1
Top potential margin for autoconnect	25.000000
Bottom potential margin for autoconnect	25.000000
Page template file name	
Page template sections	
Object	
Property specifying the object kind...	

Here you can specify the number of columns in the current page and the dimensions of the drawing. It is possible to specify different numbers of columns for the different pages. The dimensions of the drawing can also be different on the different pages. Of course, projects are usually constructed using the same structure.

You can set the column number in the first column (0 or 1), the margin of the left column and the margin of the right column, the positions of potentials, etc.

The grid is also defined here.

You can specify the symbol scaling, too. When you insert a symbol from the symbol database, it is scaled with the defined factor.

You can also define the page template sections.

TT.5. CUSTOMIZING THE INTERFACE

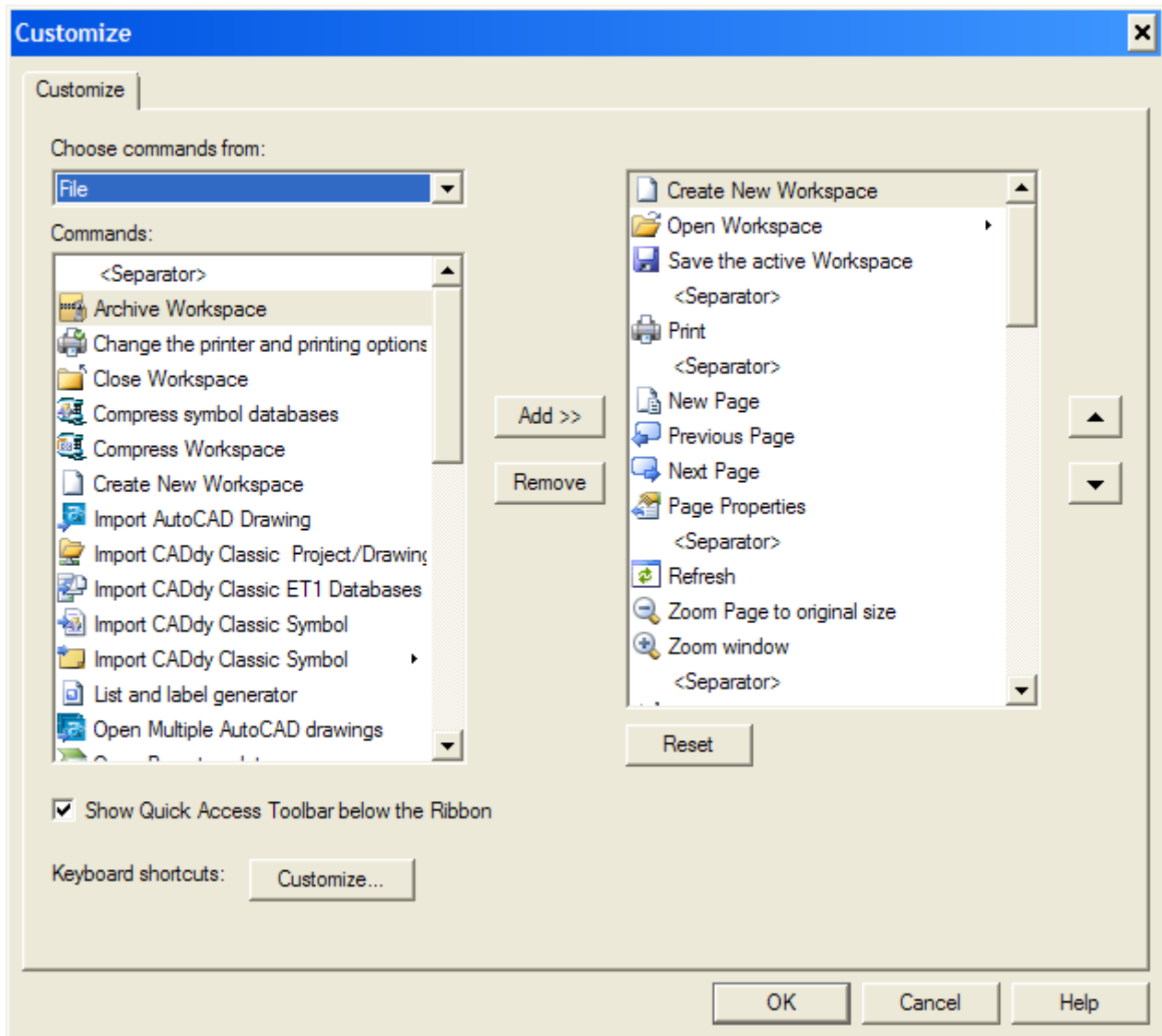
SEE Electrical allows you to customize some of the functionalities of the interface such as the application look, the commands in the **Quick Access Toolbar**, etc.

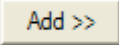
TT.5.1. QUICK ACCESS TOOLBAR

Exercise 42-1: Add the icon for the **Draw Circle** command.

- 1.> Right-click in the **Categories** area
- 2.CO Select the **Customize Quick Access Toolbar** pop-up command

The following window appears:



- 3.> Choose the **Draw** category from the "**Choose commands from**" pull-down list.
The available commands appear in the **Commands** area
- 4.> Select the **Draw Circle** command.
- 6.+ Click the  button.
The command is moved to the right area of the window.
- 7.> Click **OK**.
The command is now available in the Quick Access Toolbar.

TT.5.2. DEFINE HOTKEYS

Exercise 42-2: Assign a hotkey to the command **Insert Bitmap**.

The **Customize** window that allows you to customize the Quick Access Toolbar should be open.

- 1.> Activate the **Customize** button.

The **Customize Keyboard** dialogue window appears

- 2.> In the **Categories** pane, select the menu that contains the function you want to assign a keyboard shortcut to.

- 3.> Select in the **Commands** pane the command to which you would like to assign a shortcut or whose shortcut you wish to change.

If a command already has a shortcut, it is displayed in the **Current keys** field.

- 4.> In the **Specify key sequence for** field select the area for which you define the key combination, e.g. for the cover sheet area or the wiring diagram etc.


- 5.> **New Shortcut** key

- 6.# <Shortcut key>

Press the desired key or shortcut on the keyboard. Letters can be combined with the keys SHIFT, CTRL or ALT.

- 7.> Assign

If the shortcut key is already used elsewhere, the

Assign button remains greyed: . In such case, you must select another key combination.

- 8.> Close

TT.5.3. USER DEFINED CATEGORIES

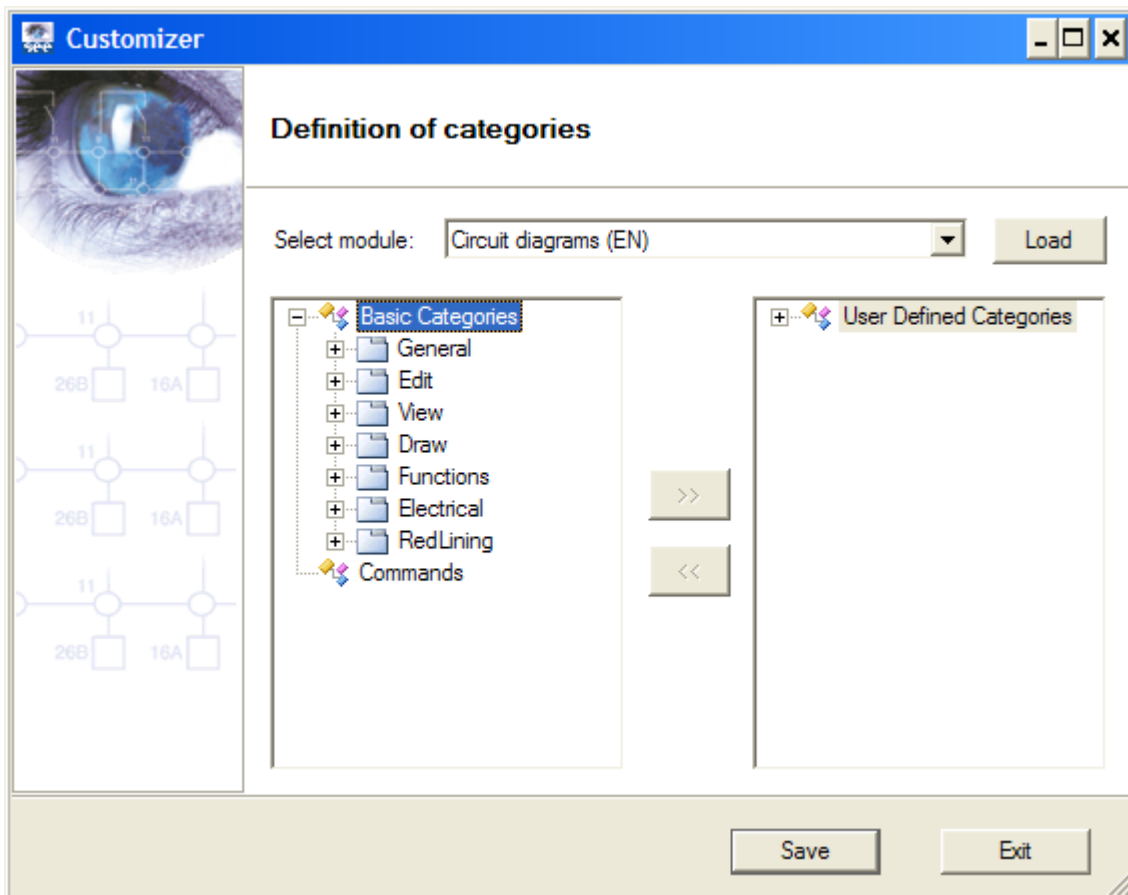
The customization is made with the *Customizer.exe* which is delivered with your *SEE Electrical* installation. The file is found in the *SEE Electrical* folder.

The customization can be executed in **Basic** level, but the commands in *SEE Electrical* will be available in case you have the right licence level.

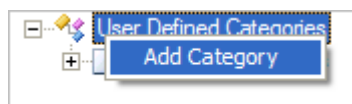
Exercise 42-2: Create a user defined category and panel.

SEE Electrical must be closed before executing the procedure!

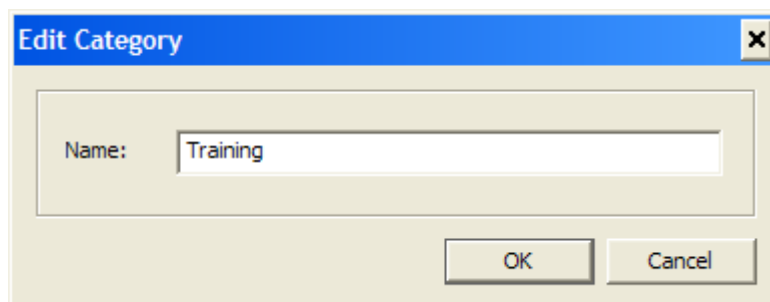
- 1.+ Open the *Customizer.exe*.
- 2.> **Next**
- 3.> Select the Circuit Diagrams (EN) or (IEEE) category from the pull-down list.
- 4.> Click the **Load** button.



- 5.> Right-click on the "User Defined Categories" node and select the **Add Category** pop-up command.,



- 6.> Type in the name of the new category – "Training" and click **OK**.

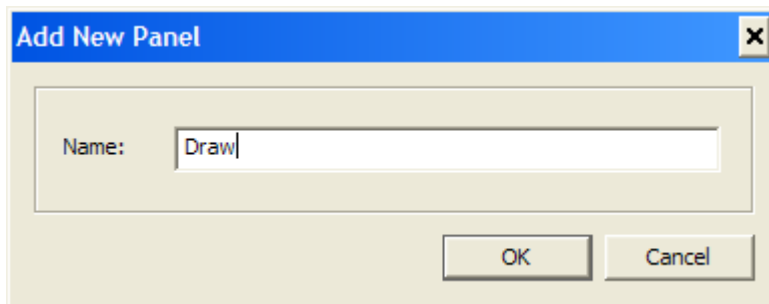


7. To add a panel to your category, right-click it and select the **Add Panel** pop-up command.

Training manual


COPYRIGHT © 2013 IGE+XAO. All rights reserved

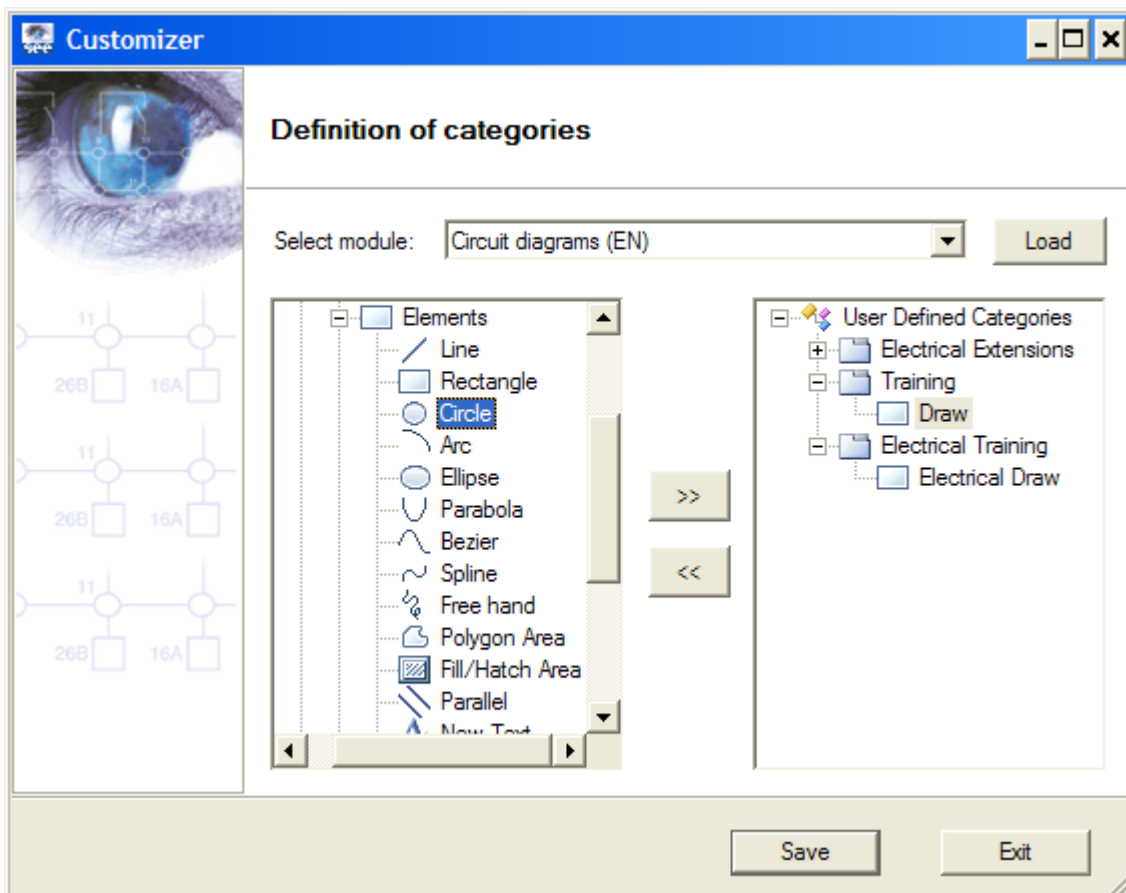
8. Type in the name of the new panel, in this case "Draw".




9. Repeat the steps to create an "Electrical Training" category and an "Electrical Draw" panel in it.

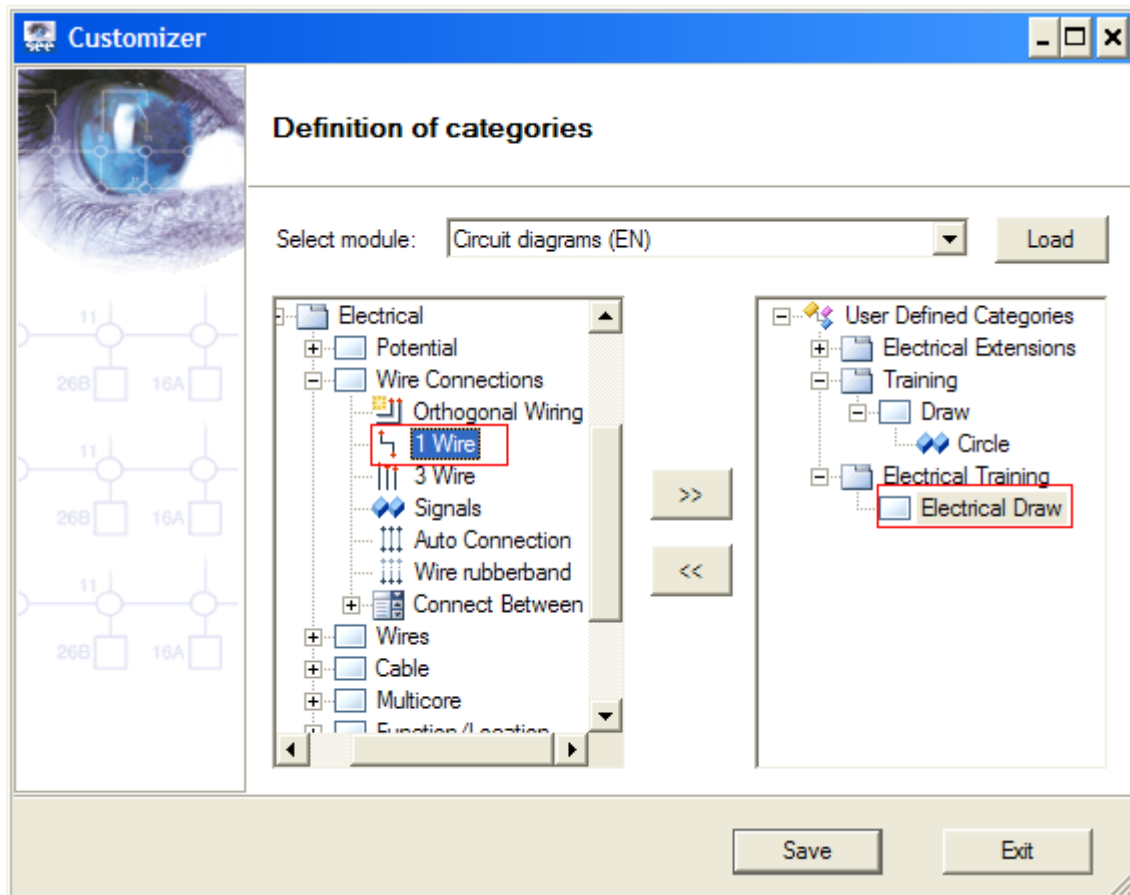
Exercise 42-4: Add commands to the user defined categories.

- 1.> In the **Customizer** window, select the **Draw** panel of your **Training** category.
- 2.> In the left pane of the window, explode the "**Basic Commands**" node and select the **Draw** ► **Elements sub-node**.
- 3.> Select the **Circle** command.
- 4.> Click the  button.



The command is added to the user defined **Draw** panel.

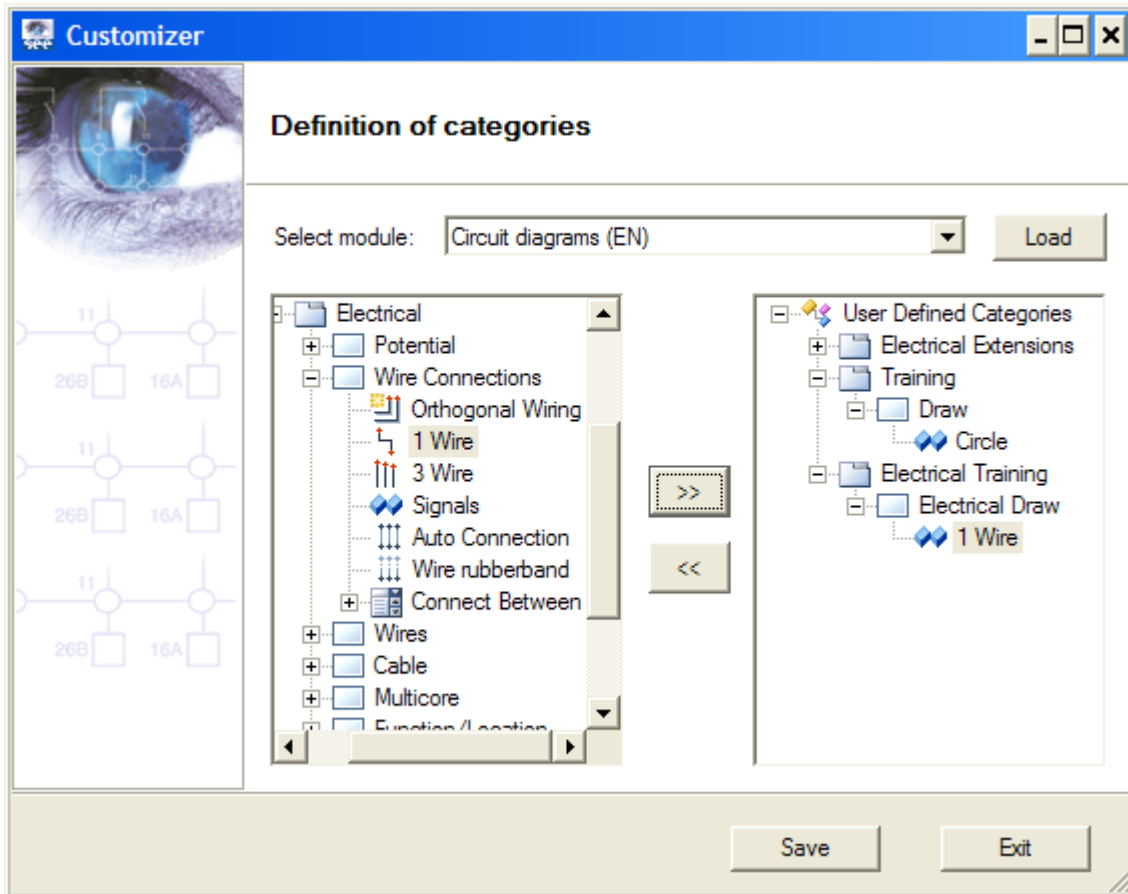
- 5.> In the **Customizer** window, select the **Electrical Draw** panel of your **Electrical Training** category.
- 6.> In the left pane of the window, explode the "Basic Commands" node and select the *Electrical* ► *Wire Connections* sub-node.
- 7.> Select the **1 Wire** command.
- 8.> Click the  button.



Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

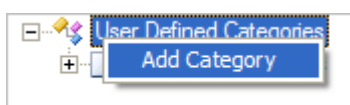
The command is added to the user defined **Electrical Draw** panel.



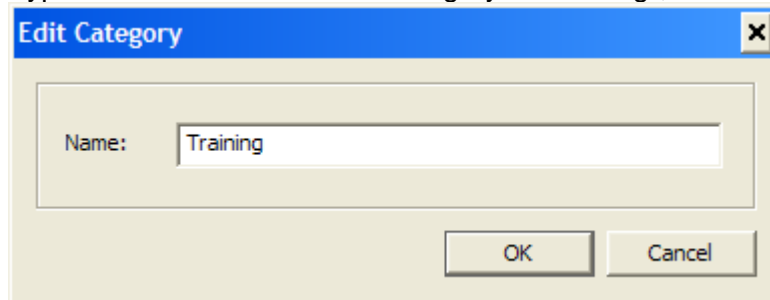
If you want to remove a command, select it and click the << button.


Exercise 42-5: Add commands from the **Commands** explorer to the user defined categories.

- 1.+ Open the *Customizer.exe*.
- 2.> Define the basic settings. All necessary files are stored by default in the *SEE Electrical* folder.
- 3.> **Next**
- 4.> Select the **General** category from the pull-down list.
- 5.> Click the **Load** button.
- 6.> Right-click on the "User Defined Categories" node and select the **Add Category** pop-up command.,



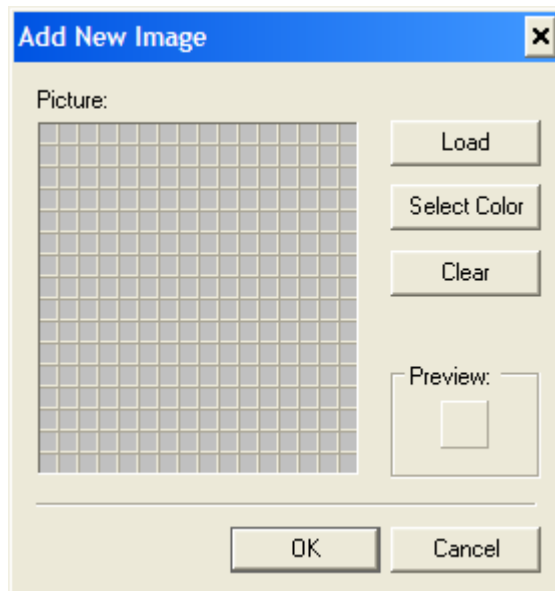
- 7.> Type in the name of the new category – "Training", and click **OK**.



10. To add a panel to your category, right-click it and select the **Add Panel** pop-up command.
 11. Type in the name of the new panel, in this case "Tools".
 13.> Explode the "Commands" node in the left part of the window.
 14.> Scroll down to find the **ETINFO** command and select it.
 15.> Select the Tools panel that you created.
 16.> Click the  button.
 The **ETINFO** command is added to the user defined category.
 17.> Click the **Save** button.
 Open *SEE Electrical*.

Exercise 42-6: Assign/Change an icon for a command.

- 1.> In the **Customizer** window, select the **ETINFO** command in your **Training** category.
 2.> Right-click and select the **Set Image** pop-up command.
 3.> **New**
 A new icon must be created.

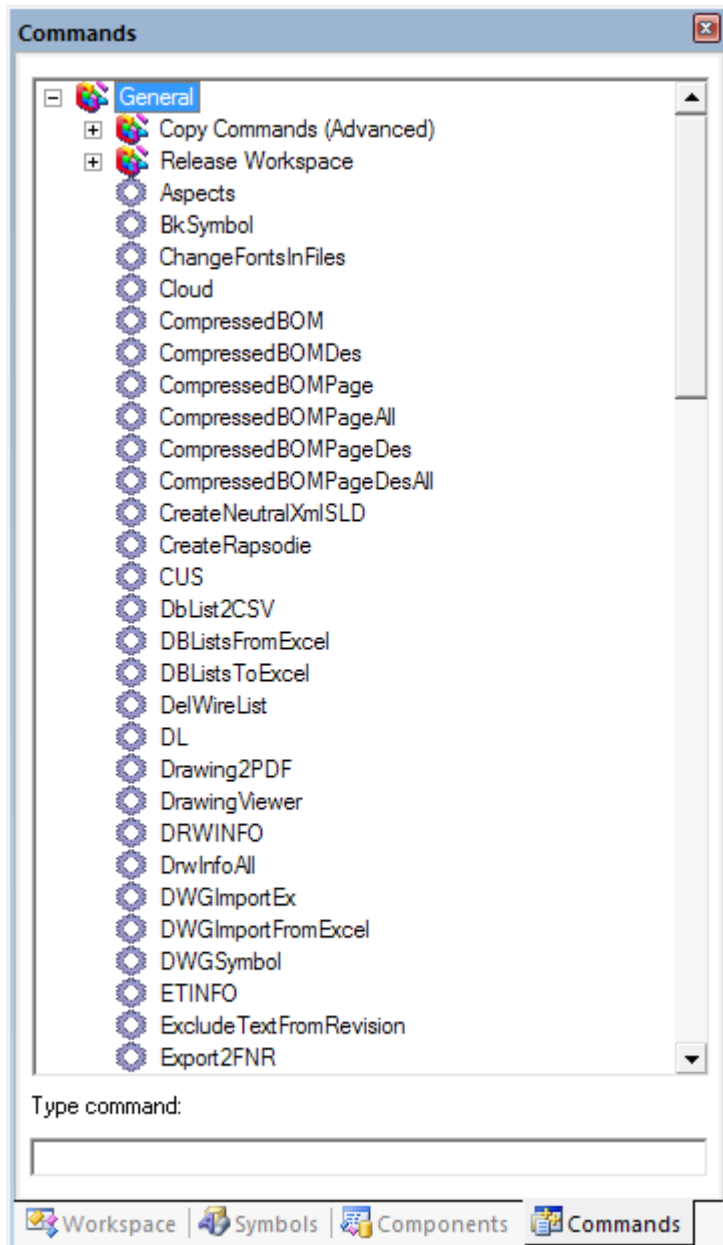


- 6.+ Construct a new icon. To change the default drawing colour, click the **Select colour** button.
 7.> **OK**
 Click **OK** to finish the construction of the new icon.
 7.> **Save.**
 Save the changes you made in the *Customizer*.

UU COMMANDS EXPLORER

The **Commands** explorer can be used to access commands provided with *SEE Electrical* or defined by the user.

In case it is not visible, you can display it by executing the **Home > View > Commands** command.



The commands are displayed alphabetically. You can type in directly the name of the command in the "**Type command:**" field at the bottom of the pane and press ENTER to execute the desired function.

It is possible also to group the commands. When you right-click on a command in the *Commands explorer* a pop-up menu appears. This pop-up menu contains functionalities to generate a group or delete one or all existing group(s). The grouping according to your needs helps to find commands quicker.

The commands can be copied and pasted to a group, they can be cut from a group and pasted to another group or they can be removed from a group, but they cannot be removed from the root level of the commands explorer. This means that you will have the commands you grouped two times: once in the root of the *Commands Explorer* and once in the selected group.

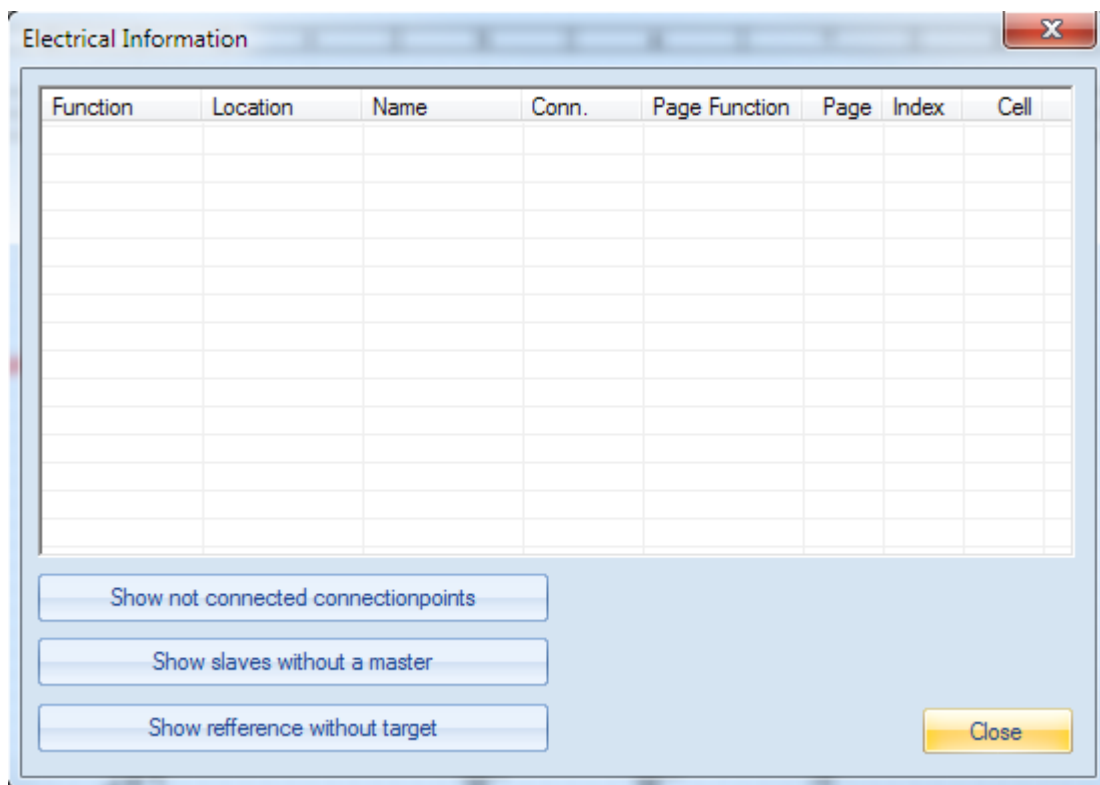
Electrical information about not connected connection points/ Information about contacts without component/ contactor

This function enables you to check if the connection points of all components in the workspace are connected.

In addition, the contacts not assigned to a contactor or to a component with auxiliary contacts can be displayed.

Exercise 43-1: Call a function from the **Command Bar**.

- 1.+ Click with the cursor within the **Command Bar**.
- 2.> ETINFO
Double-click the command in the list to execute it.



- 3.> Show not connected connection points
Click the button corresponding to the desired function, such as "Show not connected connection points" in the example above.

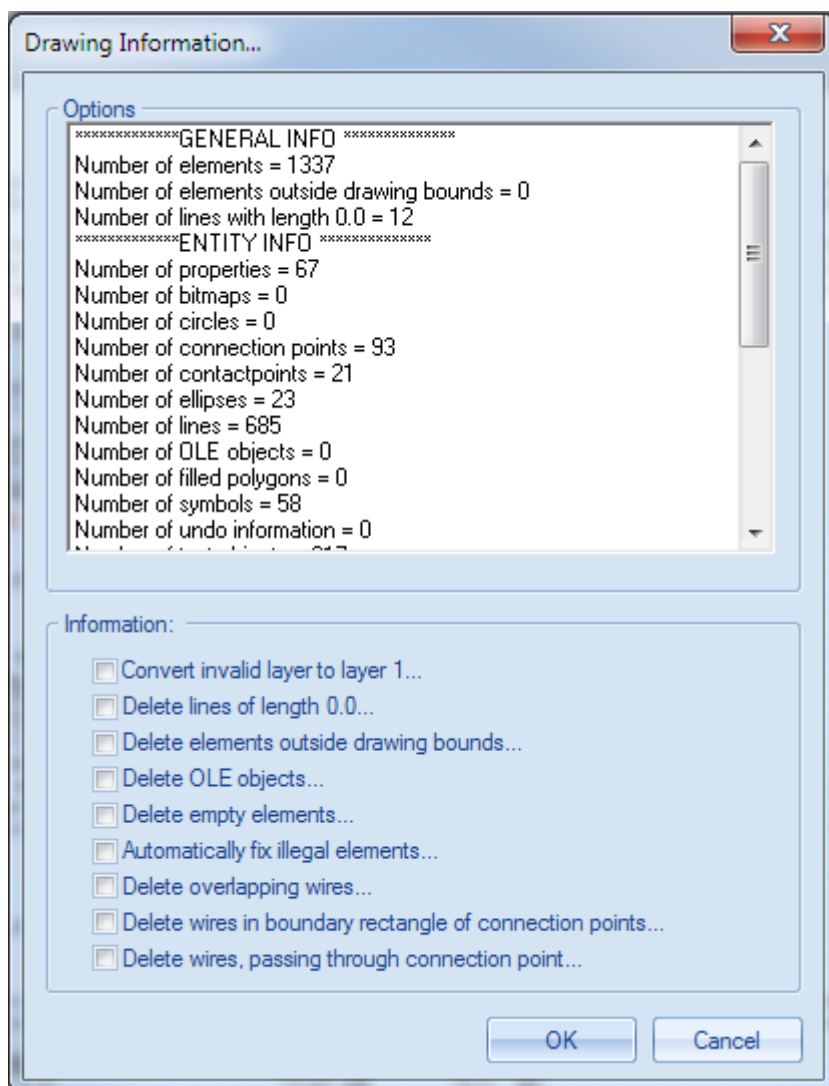
Training manual

COPYRIGHT © 2013 IGE+XAO. All rights reserved

4. Click **OK** to close the dialogue.
Delete elements outside drawing bounds
It can occur that an element is inserted outside the current drawing area (for example, by mistake while dragging). Then, you will see that nothing is shown about a component in the *Products list*.

Exercise 43-2: Call the function from the **Command Bar**.

- 1.+ Click with the cursor within the **Command Bar**.
- 2.# **DRWINFO**
Press ENTER to confirm the command



- 3.> Delete elements outside drawing bounds.
- 4.> **OK**

Exercise 43-3: Create groups of commands in the **Commands** explorer

- 1.> Right-click on a command.
- 2.> Select the **New Group** pop-up command.
- 3.> Give a name to the group, for example, Training and click **OK**.
The group is created.
- 5.> Copy the desired commands by right-clicking and selecting the **Copy Command** or **Cut Command** pop-up commands.
- 6.> Paste the commands in the group with the help of the pop-up menu.

VV INTELLIGENT DRAWING LEGACY MODULE

The *Intelligent Drawing Legacy* module offers you two tools that help to optimize the workflow of data imported into *SEE Electrical*.

Tools for scanned drawings:

These tools allow you to:

- ✓ import a lot of scanned drawings in one step;
- ✓ place a white background directly when a symbol is inserted.

These tools are available for all levels of the software.

Tools for making DXF/DWG imported drawings more intelligent (recognize patterns):

These tools allow you to:

- ✓ define patterns;
- ✓ recognize patterns.

These tools are available for the *Standard* and *Advanced* levels.

Tools for scanned and DXF/DWG imported drawings:

These tools are available for the *Advanced* level.

VV.1. TOOLS FOR SCANNED DRAWINGS

VV.1.1. IMPORTING MULTIPLE SCANNED DRAWINGS IN ONE STEP

Advanced

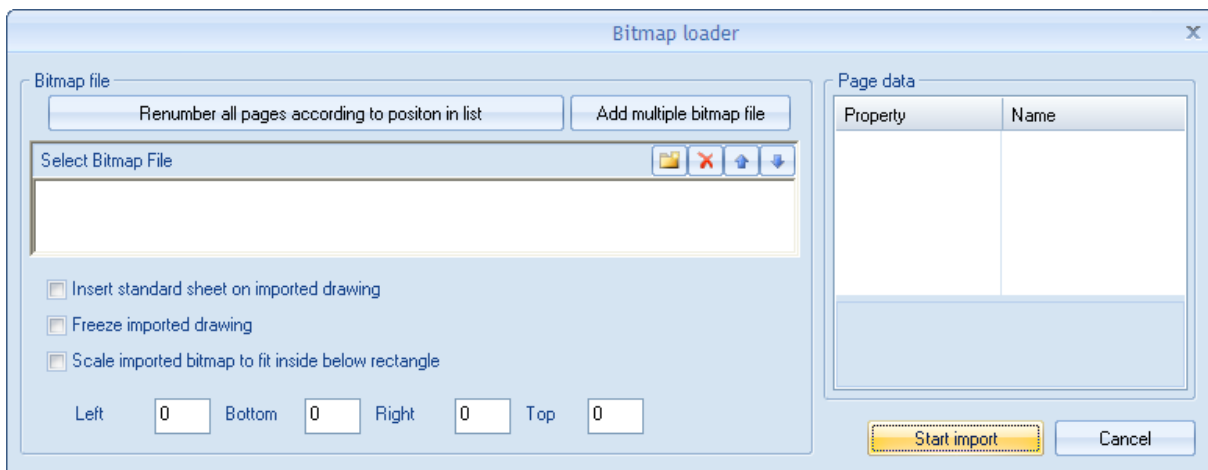
The *Scan* module allows you to import several images (of *.TIF and *.JPG format) simultaneously. The **SCANIN** command is available in the **Command** pane. It allows you to import different types of images and insert them as part of a workspace. With the help of this function you can insert a scanned image as a different page in the workspace.


Note:

You can use other images, often old, and insert them as part of the workspace.

Exercise 44-1: Insert scanned drawings in your project.

1. Select the **SCANIN** command from the list of commands in *SEE Electrical*. The **Bitmap loader** window appears:



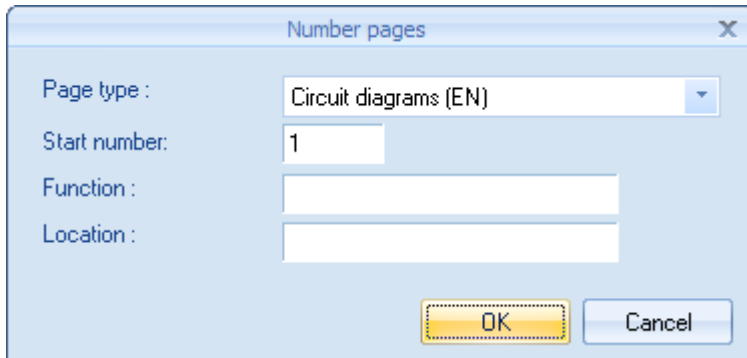
2. Click the **Add multiple bitmap file** button and select the files you want to insert in the workspace.
3. Once you have added the images, you can change the order in which they are listed via the manipulation buttons: .

Note:

The listing order is important since it allows you to define the order in which the images will be inserted in the workspace. That is to say, the first image can be inserted as page 1, the second as page 2, etc. You can also "drag and drop" the file to modify the insertion order.

4. When the order is defined, click the **Renumber all pages according to position in list** button so that *SEE Electrical* can insert a consecutive number for each scanned image.

The following dialog box appears:



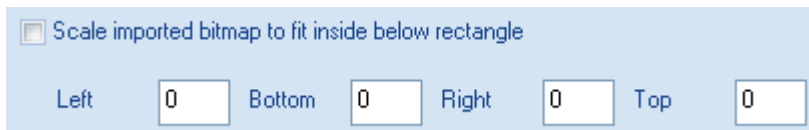
5. Type in the necessary information and click **OK**.

Each scanned image can receive different settings, if you select it (it is highlighted in blue) and you modify the information in the right pane of the **Bitmap loader** window.

6. Tick the **"Insert standard sheet on imported drawing"** option to insert a default template (set as property for each type of sheet) during the image insertion.



7. Tick the **"Scale imported bitmap to fit inside below rectangle"** option and define the coordinates of the rectangle:



8. Click the **Import** button to start the import.

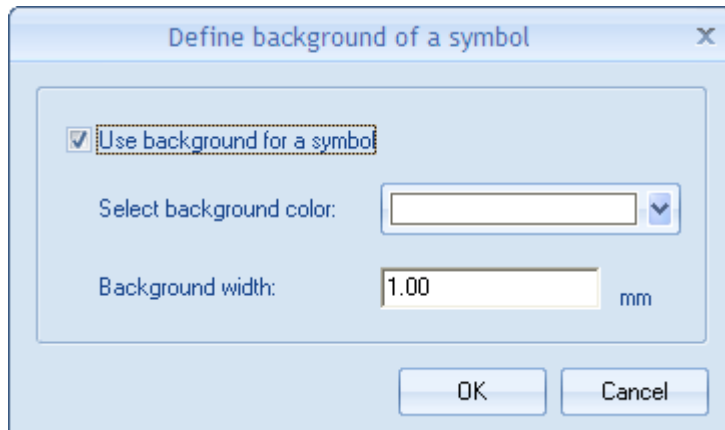
Attention:

The contents of the current sheet will be deleted during the import!!! The sheet will temporarily receive the imported files.

VV.1.2.COVERING SCANNED SYMBOLS WITH AN (WHITE) AREA ("BKSYMBOL" COMMAND)

The **BkSymbol** command allows you to define if a background is to be inserted together with a symbol. This is really helpful when scanned drawings need to be processed.

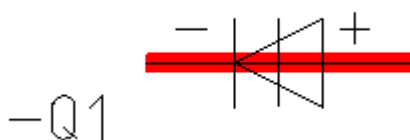
1. Execute the command and define the appropriate settings.



The **"Use background for a symbol"** checkbox allows you to enable the functionality when you process drawings with scanned information and disable it when you process drawings created directly in *SEE Electrical*.

By default the background colour is set to white, but you can define a different colour within the **"Select background colour"** pull-down list.

The **"Background width"** option defines the width of the filling area for the background.



2. Click **OK** to validate the selected settings.

VV.2. TOOLS FOR MAKING DXF/DWG IMPORTED DRAWINGS MORE INTELLIGENT

VV.2.1. DEFINING PATTERNS ("DEFINEPATTERNS" COMMAND)

The command allows you to exchange patterns found in the diagram with symbols that are made following the rules for components, terminals, etc. Potentials and standard sheets can also be recognized.

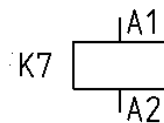
In this way, for example, drawings imported by *DXF/DWG* import can be converted to good working *SEE Electrical* circuit diagrams that produce terminal plans, etc. This means that drawings which do not contain any logic can be changed into ones with electrical logic.

You need the specific *PATTERNS.SES* symbol library. Please get in contact, if it is missing.

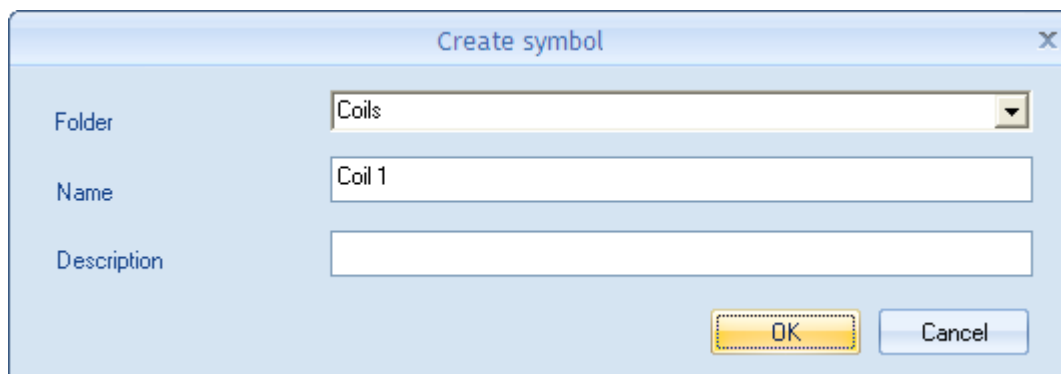
The symbols can be added to existing folders only, so if you need new ones, create them before you try to define a pattern.

Exercise 44-2: Define patterns in the project.

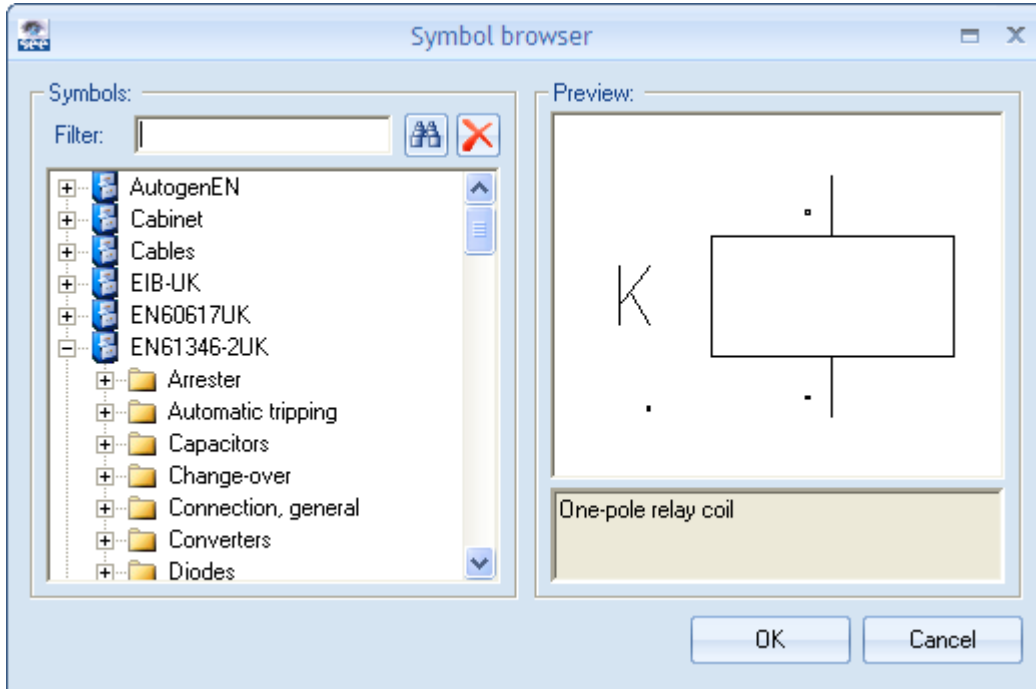
1. Draw the geometry for the first pattern.



2. The geometry and texts have to be single elements, not symbols or macro/groups.
3. Select the geometry/texts that shall be added to the pattern.
4. Execute the **DefinePatterns** command.
5. Define the insertion point (this will be used later as insertion point for the symbol).
6. Specify the folder and the name under which the symbol will be stored in the *PATTERNS.SES* library and click **OK**.



6. Assign the *SEE Electrical* symbol that shall replace the pattern later.



7. Select the desired symbol and click **OK** to confirm.
8. Define all the patterns used in this way, including potentials.

To recognize potentials, *SEE Electrical* symbols have to be prepared inside a symbol library. This means you have to draw a potential and drag it into a folder of a symbol library. Make sure you use the same insertion point which you define later for the pattern.

Potential lines

You can also store potential lines as patterns. Before you do so, store a *SEE Electrical* potential as a symbol in one of your symbol libraries.

Standard sheet

If you want a standard sheet to be recognized, you need to store it as pattern, too. Before you do so, store a *SEE Electrical* standard sheet as a symbol in one of your symbol libraries.

Normally a standard sheet cannot be stored in a symbol library. To store it, proceed as follows:

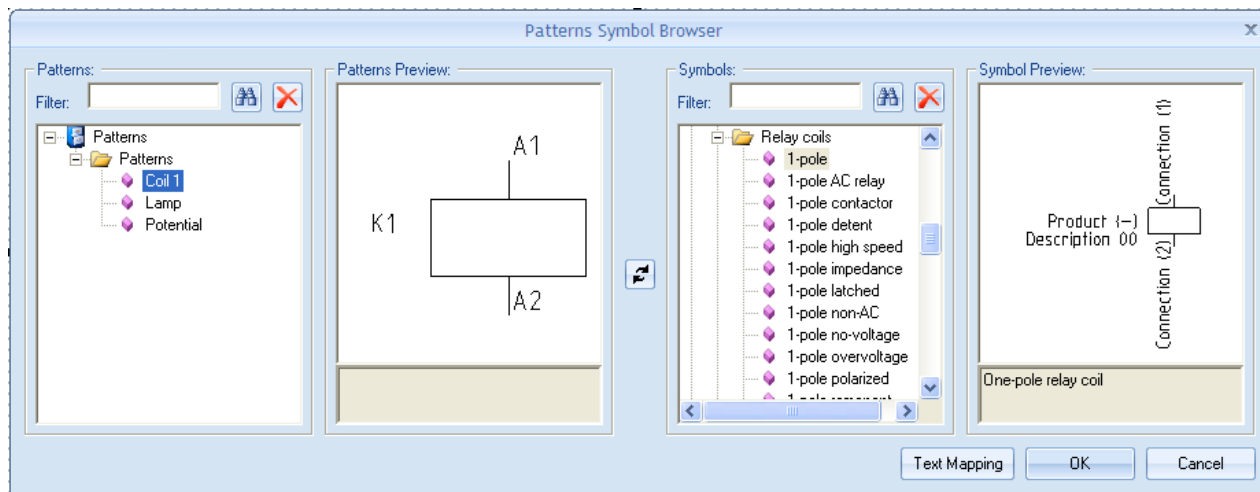
- Take the "symbol origin" symbol from the "symbol origin" folder of your "System" symbol library. Place the symbol at the left lower corner of the standard sheet.
- Select the standard sheet and the "symbol origin" symbol.
- Click on to the "symbol origin" symbol and drag both to the folder of the symbol library where you want to store the standard sheet.

VV.2.2.CONTROLING AND CHANGING PATTERNS, ASSIGNING TEXTS ("SHOWPATTERNS" COMMAND)

The **ShowPatterns** command allows you to control and change the assignment of patterns to *SEE Electrical* symbols. It also makes it possible for the user to specify which text shall contain which information.

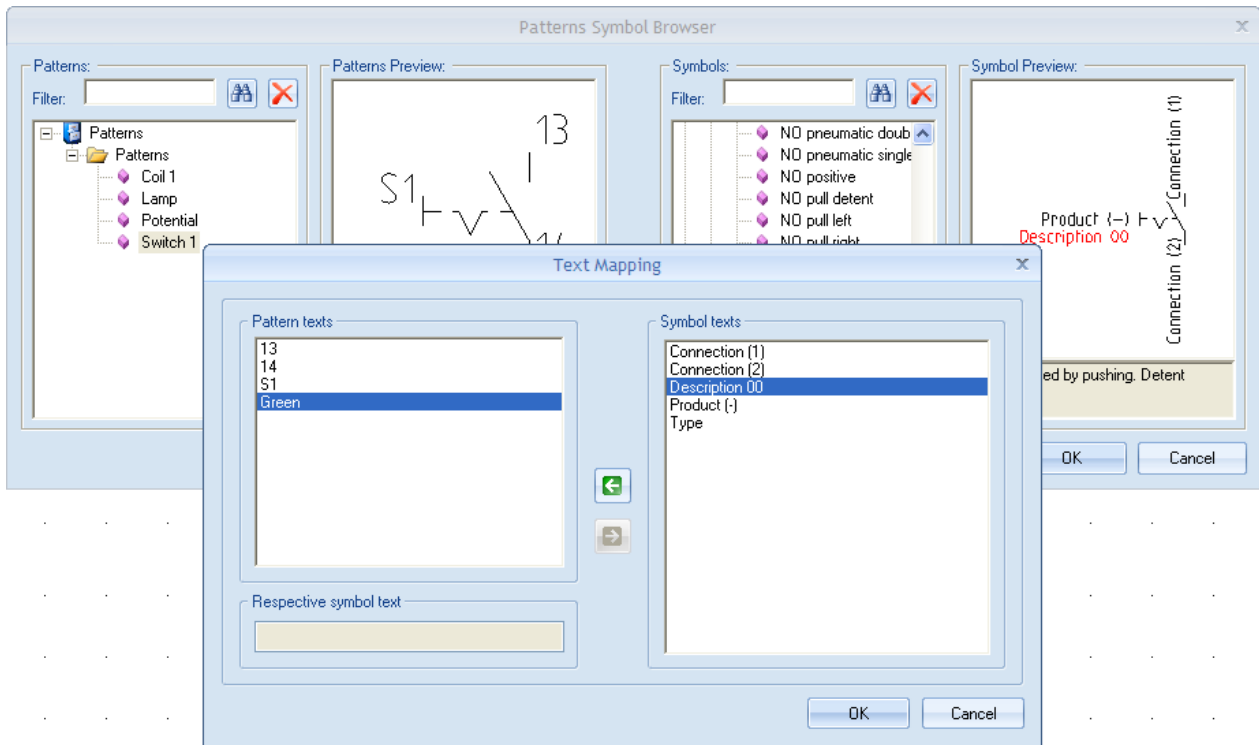
Exercise 44-3: Control the assignment of patterns to *SEE Electrical* symbols.

1. Execute the command.
The following window appears:




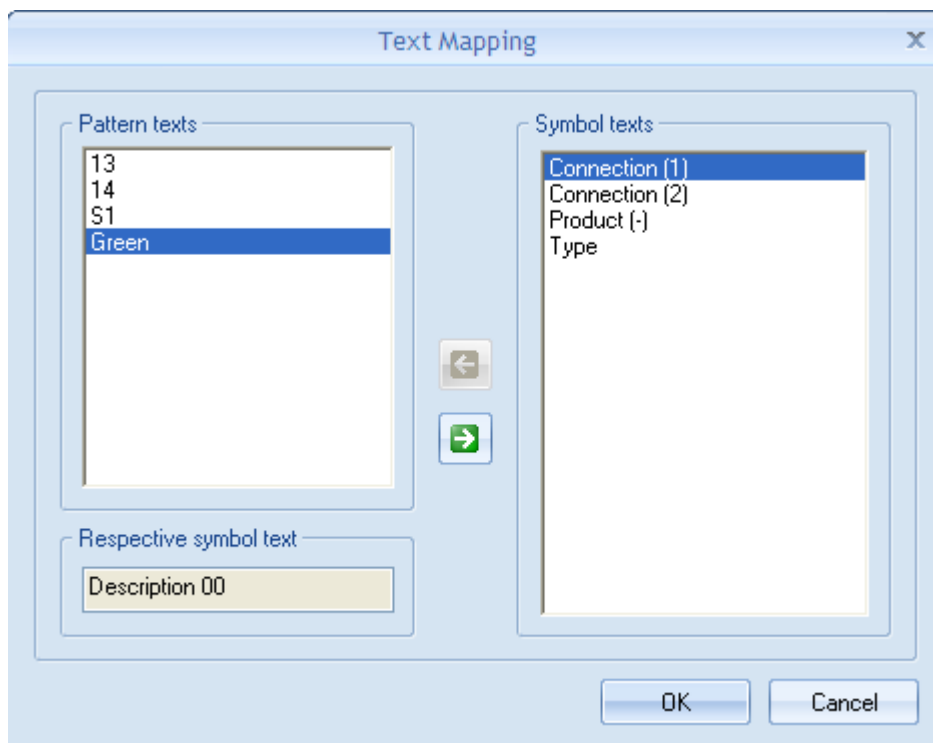
The  button exchanges a symbol you have chosen in the *SEE Electrical* symbol libraries with the one assigned to the pattern (the one you see in **Pattern Preview** pane).

Via the **Text Mapping** button you can assign a text from the pattern to a text in the *SEE Electrical* symbol. For instance, in the example shown below you can define that the text "Green" found in a position not commonly used by *SEE Electrical* is used later as a description. All texts found in a similar position at a pattern similar to the switch will be recognized as "Description".



2. Choose a pattern text and a symbol text.

Via the  button you can assign them to each other. The symbol text is taken from the list on the right side. The assigned text is displayed in the **"Respective symbol text"** field.



3. After you have assigned the texts, click **OK** to close the window.

VV.2.3.RECOGNIZING PATTERNS ("RECOGNIZEDDRAWINGPATTERNS" AND "RECOGNIZEWORKSPACEPATTERNS" COMMANDS)

Exercise 44-4: Replace the patterns in the drawing and/or the workspace with the previously defined *SEE Electrical* symbols.

1. Load the page with the drawing in which you wish to recognize the patterns.
2. Execute the **RecognizeDrawingPatterns** command.
All the patterns found in the *PATTERNS.SES* symbol library are searched in the drawing and replaced by the symbols assigned.
3. If you want to recognize all patterns found in your workspace, execute the **RecognizeWorkspacePatterns** command instead.

VV.2.4.DISPLAYING ALL ELEMENTS WITH LOGIC

By using the **MICAW** command, you can display all elements with electrical logic on the current sheet (symbols, potentials, wires and standard sheets).

Exercise 44-5: Display all elements with electrical logic in the current sheet.

1. Double-click the command in the **Commands** explorer to execute it.
All logical elements are marked in green.
12. Click anywhere in the drawing to display the elements without the green marking.

