SIEMENS

COMOS

Automation
EI&C Operation

Operating Manual

06/2025 V 10.6.0

Introduction 1

Basic Engineering/IC Engineering 2

Function engineering in the layout 3

Electrical design 4

Object-based editing of connections in the database: 5

Using automatic labels 6

Drawing single line diagrams 7

Load estimation with single line diagrams 8

Object-based editing of cables and wires in the database 9

Object-based editing of terminal strips in the database 10

Object-based editing of potentials in the database 11

Using product data and manufacturer devices 12

Using a mounting catalog and hook-ups 13

Designing cabinets 14

Controlling and evaluating 15

Using templates and grayboxes 16

Legal information

Warning notice system

This manual contains notices you have to observe in order to ensure your personal safety, as well as to prevent damage to property. The notices referring to your personal safety are highlighted in the manual by a safety alert symbol, notices referring only to property damage have no safety alert symbol. These notices shown below are graded according to the degree of danger.

⚠ DANGER

indicates that death or severe personal injury **will** result if proper precautions are not taken.

⚠ WARNING

indicates that death or severe personal injury **may** result if proper precautions are not taken.

⚠ CAUTION

indicates that minor personal injury can result if proper precautions are not taken.

NOTICE

indicates that property damage can result if proper precautions are not taken.

If more than one degree of danger is present, the warning notice representing the highest degree of danger will be used. A notice warning of injury to persons with a safety alert symbol may also include a warning relating to property damage.

Qualified Personnel

The product/system described in this documentation may be operated only by **personnel qualified** for the specific task in accordance with the relevant documentation, in particular its warning notices and safety instructions. Qualified personnel are those who, based on their training and experience, are capable of identifying risks and avoiding potential hazards when working with these products/systems.

Proper use of Siemens products

Note the following:

⚠ WARNING

Siemens products may only be used for the applications described in the catalog and in the relevant technical documentation. If products and components from other manufacturers are used, these must be recommended or approved by Siemens. Proper transport, storage, installation, assembly, commissioning, operation and maintenance are required to ensure that the products operate safely and without any problems. The permissible ambient conditions must be complied with. The information in the relevant documentation must be observed.

Trademarks

All names identified by ® are registered trademarks of Siemens Aktiengesellschaft. The remaining trademarks in this publication may be trademarks whose use by third parties for their own purposes could violate the rights of the owner.

Disclaimer of Liability

We have reviewed the contents of this publication to ensure consistency with the hardware and software described. Since variance cannot be precluded entirely, we cannot guarantee full consistency. However, the information in this publication is reviewed regularly and any necessary corrections are included in subsequent editions.

Siemens Aktiengesellschaft
Digital Industries
Postfach 48 48
90026 NÜRNBERG
GERMANY

Ⓟ 05/2025 Subject to change

Copyright © Siemens 2012 - 2025.
All rights reserved

Table of contents

1 Introduction 13

1.1 Requirements for EI&C 13

1.2 Overview of the EI&C module 13

2 Basic Engineering/IC Engineering 15

2.1 Definition of standard LOOPs 15

2.2 Creating and placing positions 16
2.2.1 Creating a position 16
2.2.2 Information about the attributes of a position 16

2.3 Creating and designing functions 17
2.3.1 Supported standards 17
2.3.2 Using a function template 17
2.3.2.1 Creating a function template/function module 17
2.3.2.2 Structure of the generated function module 18
2.3.2.3 Opening a loop diagram or device sheet 19
2.3.2.4 Incomplete division into unit and location view 19
2.3.3 Manual functional design 20
2.3.3.1 Creating functions manually 20
2.3.3.2 Creating devices, channel requests and signals 20
2.3.3.3 Editing attributes of the signal 22
2.3.3.4 Requirements for marshalling 22

2.4 Edit and check interlocks 23
2.4.1 Entering the interlock data of the signal 23
2.4.2 Creating and assigning an interlock 23
2.4.3 Creating the logic of the interlocks 24

2.5 Location planning 25
2.5.1 Task 25
2.5.2 Creating basic structures for location engineering 26
2.5.3 Creating marshalling racks (field and control side) 26
2.5.4 Creating a field location 27
2.5.5 Creating a controller (PLC) 27
2.5.6 Determining bills of quantities by means of bulk processing 28
2.5.7 Creating I/O modules and I/O channels 29
2.5.8 Setting the location pointers of the devices 30
2.5.9 Setting the signal of an I/O channel 31
2.5.10 Creating field distributors and terminals 32
2.5.11 Creating a control cabinet and equipping it with basic objects 33

2.6 Implementing requests in the location view 34
2.6.1 Administration 34
2.6.2 Definition of "implementation" 34
2.6.3 Implement channel requests individually 35
2.6.4 Implement channel requests per query 35

2.7 Importing basic engineering data from Excel ... 36

2.8 Assigning signals ... 37
2.8.1 Open the "Signal/Device assignment manager" plugin ... 37
2.8.2 Specifying the base set ... 37
2.8.3 Checking the signal application ... 38
2.8.4 Assigning signals to devices ... 38
2.8.5 User interface reference ... 39

2.9 Creating marshalling/loops ... 41
2.9.1 Connecting and marshalling with the Marshalling Designer ... 41
2.9.1.1 Overview of the Marshalling Designer ... 41
2.9.1.2 Creating and opening the Marshalling Designer ... 41
2.9.1.3 Display of the marshalling designer ... 42
2.9.1.4 Placing field devices ... 42
2.9.1.5 Placing field distributors ... 43
2.9.1.6 Placing marshalling racks ... 43
2.9.1.7 Placing I/O modules (control) ... 44
2.9.1.8 Connecting objects on the Marshalling Designer ... 44
2.9.1.9 Working with several terminal strips and connection patterns ... 45
2.9.1.10 Carrying out marshalling on the Marshalling Designer ... 48
2.9.1.11 Recalculating wiring ... 49
2.9.2 Object-based connecting and marshalling ... 49
2.9.2.1 Connecting devices to the field distributor ... 49
2.9.2.2 Connecting the field distributor to the marshalling rack (field side) ... 50
2.9.2.3 Connecting the marshalling rack (control side) to I/O channels ... 50
2.9.2.4 Creating marshalling with automatic connection ... 51
2.9.3 Connecting and marshalling with the loop diagram ... 52
2.9.3.1 Signal-based search and classification search ... 52
2.9.3.2 Structure of the point diagram ... 52
2.9.3.3 Automatic Placing (AutoLoop) ... 54
2.9.3.4 Automatic Connecting (AutoConnect) ... 55
2.9.3.5 Up-to-dateness check and repeated evaluation ... 55
2.9.3.6 Connecting and marshalling with the loop diagram ... 56
2.9.4 Using the "Loop Document Manager" plugin ... 56
2.9.4.1 Purpose of the "Loop Document Manager" ... 56
2.9.4.2 Opening the "Loop Document Manager" plugin ... 57
2.9.4.3 Rules of up-to-dateness check ... 58
2.9.4.4 Checking the currentness of loops ... 59
2.9.4.5 Updating loop diagrams ... 59
2.9.5 More information on loops and position diagrams ... 61

2.10 Working with the "Brownfield Loop Wizard" plugin ... 61
2.10.1 Overview ... 61
2.10.2 Loading/creating import rules ... 61
2.10.3 Defining implementation rules ... 62

2.11 Using the Implementation manager ... 63
2.11.1 Purpose of the Implementation manager ... 63
2.11.2 Requirement of the Implementation manager ... 63
2.11.3 Refreshing the table contents ... 64
2.11.4 Implementing all requests automatically ... 65
2.11.5 Implementing selected requests automatically ... 65
2.11.6 Disconnecting implementations ... 66

2.11.7 Implementing requests manually ... 67
2.11.8 Reference Implementation manager ... 68

2.12 Using the Potential manager ... 69
2.12.1 Purpose of the Potential manager ... 69
2.12.2 Refreshing the table contents ... 70
2.12.3 Implementing potentials ... 71
2.12.4 Disconnecting potentials ... 71

2.13 Notes on engineering tasks for EI & C ... 72

2.14 Using the "Characteristics for connections" tool ... 72
2.14.1 Purpose of the "Characteristics for connections" plugin ... 72
2.14.2 Interface reference of the "Characteristics for connections" plugin ... 73
2.14.3 Setting connection information in bulk ... 75
2.14.4 Setting the wire number or connection label in bulk ... 76

3 Function engineering in the layout ... 79

3.1 Basics of working with layout documents ... 79
3.1.1 "Layout" definition ... 79
3.1.2 Definition of "engineering task" ... 79
3.1.3 Tasks in the layout ... 80

3.2 Overall workflow in the layout ... 81

3.3 Creating a layout ... 82

3.4 Placing objects ... 82

3.5 Assign subunits ... 83

3.6 Assigning areas ... 83

3.7 Selecting targets ... 84

3.8 Calculate task targets ... 86

3.9 Finding task targets ... 86

3.10 Working with polygons ... 87
3.10.1 Basics of polygons ... 87
3.10.2 Drawing polygons ... 88
3.10.3 Editing the polygon shape ... 89
3.10.4 Replacing a polygon shape ... 91
3.10.5 Polygon properties ... 91

3.11 User interface reference ... 92
3.11.1 User interface reference layout ... 92
3.11.2 Context menu in the layout ... 93

4 Electrical design ... 95

4.1 Creating or opening a circuit diagram ... 95

4.2 Controlling a circuit diagram ... 95
4.2.1 Controlling a construction representation ... 95
4.2.2 Controlling transformation tools ... 96
4.2.3 Creating and editing path texts ... 97

4.3 Display variants of a circuit diagram ... 98
4.3.1 Line display on the report ... 98

4.3.2 Status display on the report ... 99
4.3.3 Working layer display on the report ... 99
4.3.4 Revision display on the report ... 100
4.3.5 Display of references and device labels ... 100

4.4 Placing supplies and potentials ... 100
4.4.1 Placing supplies using a template ... 100
4.4.2 Placing power supplies from the Navigator ... 101
4.4.3 Using a supply with potential ... 102
4.4.4 Potential chains / Master-Slave potentials / Potential references ... 102
4.4.5 Assigning logical potential to a connection ... 103
4.4.6 Transferring potential ... 104

4.5 Placing components and devices ... 105
4.5.1 Placing stand-alone objects ... 105
4.5.2 Placing search text objects ... 106
4.5.3 Placing connection dependent objects ... 106
4.5.4 Placing and editing a black box ... 107
4.5.5 Pasting objects via the clipboard ... 108
4.5.6 Display of devices ... 109

4.6 Deleting elements of components or groupings ... 110

4.7 Deleting inconsistent elements ... 110

4.8 Editing object properties and hierarchies on the report ... 111
4.8.1 Using the "Assign object" tool ... 111
4.8.2 Using interactive text ... 113
4.8.2.1 Purpose of the interactive text ... 113
4.8.2.2 Definitions and requirements for interactive text ... 113
4.8.2.3 Notes on the use of interactive text ... 114
4.8.2.4 Exceptions in the case of interactive text ... 115

4.9 Placing attribute texts and working with editable text ... 116

4.10 Terminals, plugs and bridges on the construction display ... 116
4.10.1 Basic knowledge of terminals, plugs and bridges ... 116
4.10.2 Generating and editing bridges ... 117
4.10.3 Automatic display of the label with terminals and plugs ... 117
4.10.4 Controlling the display of the label ... 119
4.10.5 Using bridge objects ... 120

4.11 Connectors and connector information on the construction representation ... 121
4.11.1 Edit connector assignment ... 121
4.11.2 Memorize and apply connector assignment ... 122
4.11.3 Using the contact surface for relay and contactor ... 123
4.11.4 Output connector description of user-generated symbols ... 127

4.12 Connections on the construction display ... 127
4.12.1 Connection objects ... 127
4.12.2 Creating connections ... 130
4.12.3 Selecting the "Connection" tool ... 132
4.12.4 Function of the "Connection" tool ... 133
4.12.5 Type check during connection ... 134
4.12.6 Connection information ... 135
4.12.6.1 Entering connection information ... 135
4.12.6.2 Changing the display of the connection information for a report ... 136

4.12.6.3 Transfer connection information via implementation ... 136
4.12.6.4 Changing the display of connection information for individual connections ... 138
4.12.6.5 Resetting connection information ... 138
4.12.7 Connecting half-open connections ... 139
4.12.8 Merging half-open connections ... 140
4.12.9 Using consistency/inconsistency information ... 143
4.12.9.1 Display of consistency and inconsistency ... 143
4.12.9.2 Always show half-open connections as consistent ... 144
4.12.9.3 Collision check ... 145
4.12.10 Grounding ... 148
4.12.10.1 Setting grounding ... 148
4.12.11 Automatic Routing ... 148
4.12.11.1 Objective of automatic routing ... 148
4.12.11.2 Limits of automatic routing ... 149
4.12.11.3 Configuring routing ... 150
4.12.11.4 Rerouting ... 150
4.12.12 Options: Connection hooks ... 151
4.12.12.1 Purpose of connection hooks ... 151
4.12.12.2 Effect of the connection hooks on the engineering data ... 152
4.12.12.3 Specifying connection hooks manually ... 153
4.12.12.4 Changing connection hooks at a later time ... 153
4.12.13 Options: Change direction ... 154
4.12.14 Options: Graphical properties of connections ... 156
4.12.14.1 Lines in the electrical engineering reports ... 156
4.12.14.2 Surface reference of the "Graphical properties" window when using connection objects ... 157
4.12.14.3 Edit graphical properties of connections with connection object ... 157
4.12.14.4 User interface reference of the "Graphical properties" window when using purely graphical connection lines ... 158
4.12.14.5 Editing properties of graphical connections ... 159
4.12.15 Controlling synchronization of the report and database with the read/write mode ... 160
4.12.16 Changing the Read/Write mode ... 161
4.12.17 Potential information for writing connections ... 161
4.12.18 References and reference objects ... 161
4.12.18.1 Automatic references for physical potentials ... 161
4.12.18.2 Representation of automatic references ... 162
4.12.18.3 Use reference bracket/collected reference ... 165
4.12.18.4 Purpose of reference objects ... 167
4.12.18.5 Using reference objects ... 167
4.12.18.6 Connecting reference objects ... 168
4.12.18.7 Using reference objects via script ... 169

4.13 Cables and wires on the construction display ... 170
4.13.1 Using a cable or wire on the report ... 170
4.13.2 Creating new cable on the circuit diagram ... 172
4.13.3 Consistent and inconsistent cable states ... 173
4.13.4 Cable legend ... 176
4.13.5 Rotating cables ... 176
4.13.6 Assigning connection information ... 183
4.13.7 Disconnecting wires ... 183
4.13.8 Shielded cables ... 184
4.13.8.1 Creating a shielded cable ... 184
4.13.8.2 Representation of the shielded cable ... 185
4.13.8.3 Changing the representation of the shielded cable ... 186

4.13.8.4 Connecting a cable to the shield element ... 187
4.13.8.5 Assign the beginning and end of a shield ... 188

4.14 ANSI cables ... 188
4.14.1 Selecting the "ANSI cable" tool ... 188
4.14.2 Constructing ANSI cables ... 189

5 Object-based editing of connections in the database: ... 193

5.1 Definitions on the topic of "Connections" ... 193

5.2 Editing connections in the Navigator ... 194

5.3 Working with the "Connectors" tab and the Navigator ... 195

5.4 Working with the properties of two objects ... 196

5.5 Working with the extended attributes of connectors ... 198

5.6 Connecting half-open connections via query ... 199
5.6.1 Opening the "Bulk processing of half-open connections" tab ... 199
5.6.2 Looking for half-open connections via query ... 199
5.6.3 Connecting half-open connections via query ... 200
5.6.4 Reference of the "Bulk processing of half-open connections" tab ... 201

5.7 Using the "Connection attributes" plugin ... 202

6 Using automatic labels ... 205

6.1 Modifying the name/label ... 205

6.2 Using segments ... 207
6.2.1 Objective of segments ... 207
6.2.2 Creating and selecting a segment ... 208
6.2.3 Properties of the segment ... 210

7 Drawing single line diagrams ... 213

7.1 Creating a single line diagram ... 213

7.2 Construction on single line diagrams ... 213

8 Load estimation with single line diagrams ... 215

8.1 Objective of the load estimation ... 215

8.2 Individual calculation of an object ... 216
8.2.1 Conducting a single calculation ... 216
8.2.2 "Technical data" tab ... 216

8.3 Power estimate in the network ... 217

9 Object-based editing of cables and wires in the database ... 221

9.1 Structuring of cable structures ... 221

9.2 Creating a shield ... 221

9.3 Editing cables in the "Wires" tab ... 222
9.3.1 Opening the "Wires" tab ... 222
9.3.2 Working with the "Wires" tab for the cable ... 222
9.3.3 Working interactively with the "Wires" tab for the cable ... 223

9.4 Editing cables in the "Wire list" query ... 224

9.4.1 Creating a "Wire list" query ... 224
9.4.2 User interface reference ... 225
9.4.3 Using a "Wire list" query ... 226
9.5 Changing wire types ... 226
9.6 Working with cable drums ... 227
10 Object-based editing of terminal strips in the database ... 231
10.1 Design of terminal strip structures ... 231
10.2 Creating a terminal block ... 232
10.3 Editing terminal strips in the "Terminal strip" tab ... 232
10.3.1 Open the "Terminal strip" tab ... 232
10.3.2 Editing Cable and Connections on the "Terminal strip" tab ... 232
10.3.3 Editing connectors and auxiliary connectors in the tab ... 233
10.3.4 Editing bridges in the tab ... 234
10.3.5 Interactive editing of the "Terminal strip" tab ... 236
10.4 Editing terminal strips in the "Terminal strip editing" query ... 237
10.4.1 Creating a "Terminal strip editing" query ... 237
10.4.2 Using a "Terminal strip editing" query ... 237
10.5 Import terminal data ... 238
10.6 Using connection patterns ... 238
10.6.1 Purpose and requirement of the connection pattern ... 238
10.6.2 Principle of connection patterns ... 238
10.6.3 Basic rules of connector assignment ... 240
10.6.4 Assigning connection patterns using buttons ... 240
10.6.5 Assigning connection patterns with drag&drop ... 241
10.6.6 Copying connection patterns ... 242
10.6.7 Applying connection patterns to several terminal strips ... 242
10.6.8 Removing connection patterns ... 242
10.6.9 Using connection patterns in the Marshalling designer ... 243
11 Object-based editing of potentials in the database ... 245
11.1 Overview ... 245
11.2 Placing potentials on connectors ... 245
11.3 Collecting potentials by means of implementation ... 246
11.4 Assigning potentials ... 247
11.5 "Potential - assigned devices" plugin ... 247
12 Using product data and manufacturer devices ... 249
12.1 Definition and area of application ... 249
12.2 Installing the "Manufacturer device database" iDB add-on ... 250
12.3 Opening the properties of devices ... 250
12.4 Entering product data ... 251
12.5 Manufacturer device selection for a single component ... 252
12.5.1 Allocating a single device request to a manufacturer device ... 252
12.5.2 User interface reference of the "Manufacturer device search" dialog ... 252

12.5.3 Connecting catalogs ... 253

12.6 Manufacturer device selection for numerous components ... 253
12.6.1 Allocating multiple device requests to manufacturer devices ... 253
12.6.2 User interface of the "Device selection (product data)" dialog ... 255
12.6.3 Subsequently editing objects ... 255

12.7 Special components: creating manufacturer devices from requests ... 256
12.7.1 Creating a manufacturer device for a request ... 256

13 Using a mounting catalog and hook-ups ... 259

13.1 Creating constituent parts of the mounting catalog ... 259

13.2 Editing mounting accessories ... 261

13.3 Entering costs ... 262

13.4 Structure of the "HookUp" ... 262

14 Designing cabinets ... 265

14.1 Creating a cabinet design ... 265

14.2 Background information: Views ... 265

14.3 Mechanical design in the front view ... 266
14.3.1 Defining the point of origin ... 266
14.3.2 Snap mode on the cabinet design ... 267

14.4 Mechanical design in further views ... 268
14.4.1 Working with views ... 268
14.4.1.1 Preparing views ... 268
14.4.1.2 Placing a view ... 268
14.4.1.3 Symbol layout of the views ... 270
14.4.1.4 Changing view size manually ... 271
14.4.1.5 Adjusting or arranging view/slice automatically ... 272
14.4.1.6 Placing objects in a view ... 273
14.4.1.7 Placing a mounting plate in the rear view ... 274
14.4.1.8 Selecting and editing objects in the view ... 275
14.4.1.9 Synchronize view: Synchronizing via 3D ... 275
14.4.2 Basic functions in the cabinet design ... 276
14.4.3 Deleting components in the cabinet design ... 277
14.4.4 Using the mounting key ... 278
14.4.5 Placing and using rails ... 279
14.4.5.1 Equipping a mounting plate with rails ... 279
14.4.5.2 Distributing rails across multiple documents ... 279
14.4.5.3 Automatic distribution of the rail across multiple views ... 280
14.4.5.4 Representation of distributed views in 3D ... 281
14.4.5.5 Editing rails ... 282
14.4.6 Placing a cable channel manually ... 282
14.4.7 Linkpoints ... 284
14.4.8 Creating virtual channel and placing the channel part ... 284
14.4.9 Performing the collision check ... 286
14.4.10 Automatically placing components ... 286
14.4.11 Editing components graphically ... 287
14.4.12 Moving a component using the installation key ... 288
14.4.13 Showing and hiding components in the cabinet design ... 289

14.4.14 3D display ... 290
14.4.14.1 Entering Installation/3D data ... 291
14.4.14.2 Open 3D view ... 292
14.4.14.3 Moving the 3D view ... 293
14.4.14.4 Objective and scope of the collision check ... 293
14.4.14.5 Performing the collision check ... 294
14.4.14.6 Representation of the mounting area in 3D bodies ... 294
14.5 Pre-configuring and equipping components using coordinates ... 296
14.6 Using 3D assembly ... 297
14.7 Dimensioning components ... 299
14.7.1 Setting scale factor ... 299
14.7.2 Creating dimensionings ... 300
14.7.3 Aligning dimensionings ... 301
14.8 Display of the mounting area on reports ... 303
14.9 Drilling planning ... 303
14.9.1 Drill drawing on the cabinet design ... 303
14.9.2 Defining drill holes in the "Drilling data" tab ... 304
14.9.3 Example: Select devices for drilling ... 306
14.9.4 User interface reference for the "Drilling data" tab ... 311
14.9.5 Placing objects and drill holes on the drilling drawing ... 314
14.10 Cabinet wiring and routing ... 314
14.10.1 Automatic wiring in design diagrams ... 314
14.10.2 Types of cabinet wiring ... 315
14.10.3 Calling wiring ... 316
14.10.4 Using automatic wiring ... 316
14.10.5 Connections in the control cabinet diagram ... 318
14.10.6 Net Optimized Routing ... 320
14.10.7 Open air cabling ... 321
14.10.8 Length calculation of wiring ... 323
14.10.8.1 Coordinate system for 3D component calculations ... 323
14.10.8.2 Wire length calculation / routing via automatically generated connection points ... 324
14.10.8.3 Wire length calculation / Routing via manually specified connection points ... 326
14.10.8.4 Automatic offset calculation ... 331
14.11 Cabinet Equipment Assistant ... 331
14.11.1 Opening the Cabinet Equipment Assistant ... 331
14.11.2 Controlling call via script option ... 331
14.11.3 Using the Cabinet Equipment Assistant ... 332
14.11.4 Function overview of the Cabinet Equipment Wizard ... 333
14.11.4.1 "Document" field ... 333
14.11.4.2 "Object" field ... 333
14.11.4.3 "Edit" field ... 334
14.11.4.4 "Coordinates" field ... 334
14.11.4.5 "Options" field ... 335
14.11.4.6 "Properties" field ... 336
14.11.5 Actions ... 336
14.11.5.1 Inserting objects on the cabinet design ... 336
14.11.5.2 Placing objects on rail using the wizard ... 337
14.11.5.3 Aligning objects ... 338
14.11.5.4 Aligning an object using coordinates ... 338

14.12 Using the mounting specifications ... 339
15 Controlling and evaluating ... 343
15.1 Status control ... 343
15.2 Browsing in evaluating reports ... 343
15.3 Order lists ... 343
15.4 Parts lists ... 344
15.5 Terminal diagram ... 345
15.6 Cable lists ... 345
16 Using templates and grayboxes ... 347
16.1 Note on grayboxes ... 347
16.2 Template management with the "Template administration" and "Update assemblies" plugins ... 347
16.2.1 Overview of the template management ... 347
16.2.2 "Template administration" plugin ... 348
16.2.2.1 Assign rights to use the "Template administration" plugin ... 348
16.2.2.2 "Template management": Creating new templates ... 349
16.2.2.3 "Template management": Editing templates ... 350
16.2.2.4 "Template management": Saving and enabling changes ... 352
16.2.2.5 User interface reference "Template administration" plugin ... 353
16.2.3 "Update assemblies" plugin ... 354
16.2.3.1 User interface link for "Update assemblies" plugin ... 354
16.2.3.2 Configuring default selection for differences ... 355
16.2.3.3 Standard table "Y10M00N00031 Assembly updater property assignment" ... 356
16.2.3.4 Updating instances ... 357
16.3 Using subdocuments (placing documents) ... 359

Introduction

1

1.1 Requirements for EI&C

You have basic knowledge of working with COMOS in general and of handling reports.

You can find more information on this topic in the "Platform Operation" manual, keywords "Basic activities" and "Working with reports".

Requirement for Basic Engineering

- Engineering has taken place and the basic structures exist on the "Unit" tab.
- The A50 "EI&C" folder exists in the unit or subunit.

1.2 Overview of the EI&C module

Aim

You can use the EI&C module to perform the basic and detail engineering tasks involved in your EI&C engineering.

Overview of basic engineering in the EI&C module

EI&C basic engineering presupposes that FEED or PID engineering has already been performed and a unit structure has been created as a result. in basic engineering, the results of the previous engineering work are evaluated and amended, approval documents are created, cost estimates are refined and the project schedule is created.

The following are among the activities that take place in COMOS on the "Units" and "Locations" tabs:

- Definition of the equipment
- Specification of the measuring principles
- Specification of the requirements
- Acquisition of design data
- Specification of the location structure

Overview of the detail engineering in the EI&C module

In detail engineering the description of the concrete technical design of the constructible plant. Using the data from the basic engineering, planned engineering and construction processes can be accurately scheduled and the unit can be precisely calculated and specified for apparatuses, equipment, machines and processes.

Two methods are available in COMOS for this purpose in addition to the manual engineering:

- Working with the marshalling designer and AutoLoop
  The marshalling designer creates graphic connections and marshalling wires between the field and the control end. AutoLoop is used to automatically generate the loop diagram.
- Template-based planning:
  Template-based planning is used at technical function realizations that are standardized to a large extent. In the process you assign the templates in the form of documents or partial documents in the course of the detailed planning and enrich these with planning-specific details. Creation of the templates can be effected either in advance or parallel to the detail engineering.

In addition, there are a number of tools available for detail engineering to handle tasks such as the following:

- Sheet management
- Editing diagrams ("creating drawings")
- Component management
- Parts lists and evaluations
- Cabinet engineering

Overview of the mounting catalog in the EI&C module

The database contains a prepared mounting catalog which you can adapt individually. The mounting catalog consists of mounting objects, each with a hook-up containing the following information:

- Mounting drawing
- Required mounting accessories
- Cost objects
  Cost objects allow automated calculation of the quantity and cost of the necessary mounting accessories.
- Measuring points where the mounting object is used

Overview of the labeling systems in the EI&C module

Various labeling systems are available to you, for example:

- KKS (power plant classification system)
- Labeling in the style of DIN/IEC
- AKZ (unit labeling system)

The descriptions in this manual are based on labeling in the style of DIN/IEC. The quoted examples are based on COMOS projects that were created with the project structure "Default for process engineering DIN/IEC".

Basic Engineering/IC Engineering

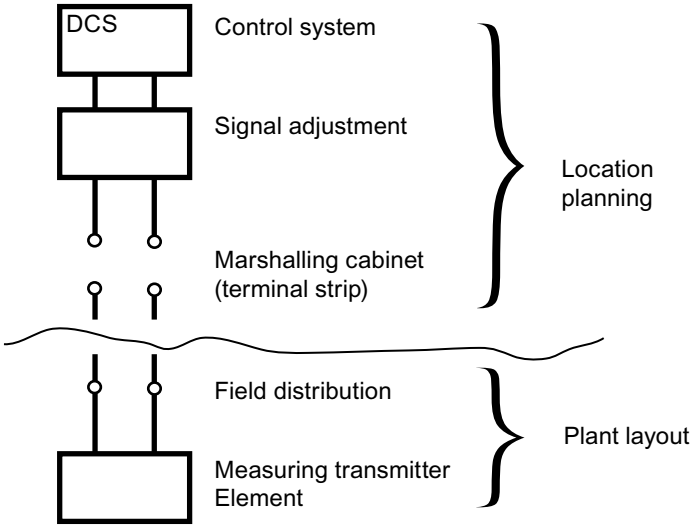
2

2.1 Definition of standard LOOPs

Assembly groups versus manual design

Two solutions can be selected in basic engineering:

- Semi-automated solution
  You use the Marshalling Designer here. The Marshalling Designer shows the following standard loop:



- Flexible solution
  In this method you choose the required objects for yourself from a library (that can also be expanded), connect these objects manually and also manually assign the required signal types.

This documentation describes the semi-automated solution. Here we will explain how you can quickly and securely plan the above loop with the help of the Marshalling Designer.

2.2 Creating and placing positions

2.2.1 Creating a position

Introduction

The position describes the I&C point. The position groups the measuring points and position locations according to their physical size.

Requirement

- The unit and subunit have been created.
- The "A50 EI&C" folder has been created.

Procedure

1. Select the "A50 EI&C" folder on the "Units" tab of the Navigator below the subunit.
2. Select the "New > General position" command in the context menu.

2.2.2 Information about the attributes of a position

Requirement

A position has been created in the "A50 EI&C" folder.

The "Attributes" tab of the position is open.

Linked attributes

Some of the position attributes are statically linked to corresponding attributes of the subunit. When a position is created, all the static links are refreshed once automatically. If the attributes of the subunit are later changed, the linked attributes at the position need to be updated manually.

The following tabs have linked attributes:

- Security requirements
- Ambient conditions

If the values of the linked attributes at the subunit and the position differ, the field has an orange background.

Updating linked attributes

If you want to create further links to automatic data import from the process planning, you can introduce these yourself. You can find more information in "Platform Operation", keyword "Refreshing static linkages".

2.3 Creating and designing functions

2.3.1 Supported standards

Requirement

The base project of the supplied database is used.

Function libraries

The following function libraries are prepared in the database:

- ANSI
- EN
- KKS

This manual describes the workflows using the "EN" library.

2.3.2 Using a function template

2.3.2.1 Creating a function template/function module

Requirement

The position has been created.

Definitions

- Function template/template of the function
  These two terms are used as synonyms in this manual. The term "function template" is preferred. The function template contains all the elements needed to create a function, including further structures.
- Function module
  A function module is derived from a function template and is created in the engineering data.

Creating a function template of a measurement function

1. Select the position.
2. Select one of the options of the "New > A10 Measurement function" position in the context menu.

Creating a function template of an actuating function

1. Select the position.
2. Select one of the options in the "New > A20 Actuating function" position in the context menu.

Result

The function module has been created.

2.3.2.2 Structure of the generated function module

Requirement

The function and its underlying objects have been generated from a function template.

Definition of "channel"

A channel is a defined input or output of a device, consisting of one or more connectors which all fulfill a common task. In this sense, the power connection of a device is a channel (consisting of plus and minus) and the signal connection is an additional channel.

In the Navigator, the channel or the channel request is displayed with the symbolic address instead of the name/label.

Structure of the function with substructures

The function is created with the following folders and objects:

- "A10 Basic engineering"
- "A20 Signal engineering"
- "A30 Detail engineering"

The objects are connected in such a way that the standard loop presented in the introduction is implemented. The function loop is presented in the "Loop diagram, detail display" document.

The request and manufacturer device data are shown on the "Device sheet" report. It lists all attributes of the tabs of the device request.

2.3.2.3 Opening a loop diagram or device sheet

Opening a loop diagram

1. Select the function in the Navigator.
2. Create a loop diagram using the context menu of the "New > Loop diagram" function.
3. Click the plus symbol to open the structure below the function.
4. Open the loop diagram.

Opening a device sheet

1. Select the measuring device in the Navigator.
2. Select the "EDA.001 Device sheet (measuring position)" command in the context menu.
   The attributes of the request and manufacturer device data are shown on the report.

2.3.2.4 Incomplete division into unit and location view

Requirement

The function and its underlying objects have been generated from a function template.

A common group as a technical boundary condition

For technical reasons: when an assembly is created, the connected objects must be located in a common branch.

A comparison of the objects created by an assembly with the standard model according to Definition of standard LOOPs (Page 15) has this result: Below the signal, objects are also created that belong to the "Locations" tab. This includes, for example, the input channel. Therefore, such objects are created with the request property.

You first define the functional requirements on the "Units" tab. You plan the concrete technical solution on the "Locations" tab:

The assembly is prepared as follows for this purpose:

- A common signal is placed on all objects in the function group. This signal is used to identity all the objects belonging together.
- The following applies to objects in the "01 Basic Engineering" folder and the associated objects in the "Locations" tab:
  The objects have the "Sheet zone" attribute on the "System data" tab. Using this attribute, the objects within the function group are differentiated into subgroups.

2.3.3 Manual functional design

2.3.3.1 Creating functions manually

Requirement

The position has been created.

No function template is used.

Procedure

1. On the "Units" tab in the Navigator, open the path to the position for which you want to create the function.
2. Select the position.
3. Select one of the options of the "New > A10 Measurement function" or "New > A20 Actuating function" position in the context menu.

Result

The function is created with the following empty folders:

- "A10 Basic engineering"
- "A20 Signal engineering"
- "A30 Detail engineering"

2.3.3.2 Creating devices, channel requests and signals

Requirement

The function has been created manually.

Creating requests

1. Select the "A10 Basic engineering" folder.
2. Select the "New > Device" command in the context menu of the folder.
3. Select at least one device request.
4. Select the "New > Terminal-, plug strips, ground" command in the context menu of the folder.
5. Select at least one terminal strip or a marshalling rack.
6. Create additional objects, if required.

Creating signals

1. Select the "A20 Signal engineering" folder.
2. Select the "New > Signals" command in the context menu of the folder.
3. Select at least one signal.
4. Select the "New" command in the signal's context menu.
5. Create a channel request.
6. Optional: Select the "New > Action block" command from the context menu of the signal. An action block is used within the scope of the logical (or "FDP") and can be created now in preparation.

Alternative creation of the signal

1. Open the properties of the function.
2. Select the "Signals" tab.
3. In an empty line in a list, enter the name of the signal in the "Signal description" column. The signal is created in the "A20 Signal engineering" folder.

Alternative creation of the channel request

The channel request of the controller does need to be created for the signal.

To create the channel request of the controller for the device request, follow these steps:

1. Select the "New > Channel request for control" command in the context menu of the "A10 Basic engineering" folder.
2. Create a channel request.

In this case, the signal must be manually assigned to the channel implementation later, because the signal is not specified by the owner.

Refer to Setting the signal of an I/O channel (Page 31).

Note

A channel request of the controller must be created at this point in time; otherwise, the subsequent automated mechanisms will not function. The channel request must be matched to the device channel request.

Result

The device requests, channel requests and signals have been created.

2.3.3.3 Editing attributes of the signal

Requirement

- A function is created in a position.
- At least one signal has been created below the function.

Editing attributes of signals at the function

1. Open the properties of the function.
2. Select the "Signals" tab.
   The created signals of the I&C point are listed in the table.
3. Specify the configurations for the signals in the table.
4. Click the "Apply" button.
   The configuration data is entered in the attributes of the signals.

Result

The attributes of the signals are current.

2.3.3.4 Requirements for marshalling

Requirement

The function has been created manually.

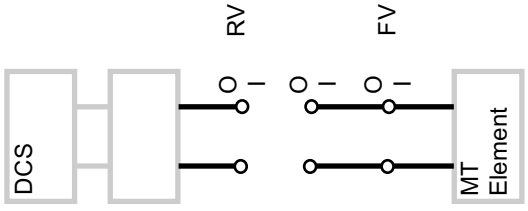
The basic structures have been created.

Connecting manually

When you create and connect objects manually, the input and output must not be reversed. This especially applies to terminals: here you also find an I-connection (I=Input) and an O-connection (O=Output).

If you exchange the terminals of the I-connector and the O-connector, neither the automatic placing (seeAutomatic Placing (AutoLoop) (Page 54)) nor the marshalling management (see Connecting and marshalling with the Marshalling Designer (Page 41)) will work correctly later.

The alignment of the terminals is as follows (O = Out; I = In):



Issuing a signal

A requirement for marshalling is that the device connections must have the same signal at the field end and control end.

2.4 Edit and check interlocks

2.4.1 Entering the interlock data of the signal

Requirement

At least one signal has been created in the "A20 Signal engineering" folder.

Entering the interlock data of the signal

1. Open the properties of the signal.
2. Select the "Signal data" tab.
3. Enter the required information in the "Interlocks" control group.
4. Click the "Apply" button.

2.4.2 Creating and assigning an interlock

Requirement

At least one signal has been created in the "A20 Signal engineering" folder.

The interlock data for the signals has been entered.

Creating the interlock

1. Select the signal.
2. Select the "New > Interlock" command from the context menu of the signal.
   Create an "Interlock" object for every interlock procedure.

Alternative creation of the interlock

1. Open the properties of the signal.
2. Select the "Signal data" tab.
3. In the "Interlocks" control group, enter the name of the interlock in an empty row of the "Label" column of the table.
   The interlock is automatically created below the signal in the Navigator on the "Units" tab.

Assigning an interlock

1. Open the properties of the signal.
2. Select the "Signal data" tab.
   The interlocks are listed in the table within the "Interlocks" control group.
3. In the "Units" tab in the Navigator, select the I&C point which you want to interlock with this signal.
4. Use drag&drop to move the I&C point into the "Actuator/Motor" column in the table row for the corresponding interlock.
   The "I&C" and "Unit" columns are automatically filled with the relevant information for the I&C point to be interlocked.
5. Specify the type of interlock in the "Function" column. You can find the description of the functions under "Function - Legend".
6. Select one or more of the options "On-site", "Control room", and "Fire service".
7. Click the "Apply" button.

Alternative assignment of the interlock

1. Open the properties of the interlock.
2. Select the "Interlock" tab.
3. Assign an object in the "Positioner/motor" field.
4. Click the "Apply" button.

Result

The interlocks have been created and assigned.

2.4.3 Creating the logic of the interlocks

Introduction

You need a logical operation of the interlocks when multiple interlocks are assigned to an I&C point.

Requirement

The function of the interlock has been specified.

Procedure

1. Open the properties of the signal.
2. Select the "Attributes > Signal data" tab.

3. Select the required logic for the interlock in the "Logic" list.
4. Enter a value in the "Signal linking assignment value" field in order to combine signals into groups.
5. Enter a note for the interlock in the "Note" field if required.
   This note will appear in the interlock list.
6. Click the "Apply" button.

Example

If you select the "AND" entry from the "Logic" list in the properties of several signals, the interlock is only triggered if the criteria for all signals of this group are met.

If you enter the value "1" in the "Signal linking assignment value" field, all signals with the value "1" in the "Signal linking assignment value" field will be assigned to a group and logically combined.

2.5 Location planning

2.5.1 Task

As part of the unit planning, elements were also created that belong to the location view according to the COMOS default model, see section "Incomplete division into unit and location view (Page 19)".
Thus the following tasks must be dealt with:

Task 1:

The field devices (measuring transmitters, sensors, valve elements) remain in the unit view and must be connected with the unit view via the field distributors. It is necessary to determine by means of a bill of quantities how many objects need to be wired up.

Task 2:

The control and signal processing requests must be connected with the location view via the implementation. This involves determining how many channels must be created in the location view. Then, the I&C objects are created in the location view and the request channels are implemented. Depending on the selected implementation mode, a reference is set to the device on the "Locations" tab or the objects are united and only still exist on the "Locations" tab.

Procedure

For example, the requests (for the measuring transmitter elements and the controller) are created in the plant design. This means that the "beginning" and the "end" of the loop have been determined at the start.

The following workflow is suggested in this documentation: First of all, the control system is determined and specified. After that, work continues in the direction of the measuring transmitters: The Ex-protection follows, then the marshalling cabinet, and finally the field distribution.

2.5.2 Creating basic structures for location engineering

Requirement

The "Locations" tab is open in the Navigator.

Procedure

To create the buildings, follow these steps:

1. Select the "New > Plant" command from the context menu of the project node.
2. Select the "New > Building" command from the context menu of the plant.
3. Select the "New > Floor, level" command from the context menu of the building.
4. Select the "New > Room, control center" command in the context menu of the floor.
5. Select the "New > Cabinet" command from the context menu of the room.

Result

The basic structures for location engineering are created.

2.5.3 Creating marshalling racks (field and control side)

Aim

You need two marshalling racks for your engineering, one for the field side and another for the control side. You use the marshalling connections between the two marshalling racks to join the field side with the control side.

Terminology

Special terminal strips will be created in the marshalling rack in a subsequent work step. These are also referred to as "marshalling racks" in COMOS. In this documentation, we use the term "marshalling rack terminal strips" to avoid confusion.

Requirement

The "Locations" tab is open in the Navigator.

The basic structures for location engineering are created.

Creating marshalling racks

1. Select the "New > MP (control bay)" command from the context menu of the "S001 Cabinet" object.
2. Select the "New > MP (field bay)" command from the context menu of the "S001 Cabinet" object.

Marshalling is conducted between the two objects, "MP (control bay)" and "MP (field bay)". You have to create the marshalling rack terminal strips below these two objects in a subsequent step.

Result

The marshalling racks (field and control side) have been created.

2.5.4 Creating a field location

Requirement

- The "Locations" tab is open in the Navigator.
- The basic structures for location engineering are created.
  See Creating basic structures for location engineering (Page 26).

Procedure

1. Select the "New > Field location" command from the context menu of the plant.
   The field location takes on the field distribution in a subsequent step.
2. Select the "New > Field distributor" command from the context menu of the field location.
3. Select the "New > Field location, measuring devices" command from the context menu of the field location.
4. Select the "New > Field location, actuating devices" command from the context menu of the field location.

2.5.5 Creating a controller (PLC)

Requirement

- The "Locations" tab is open in the Navigator.
- The basic structures for location engineering are created.
  See Creating basic structures for location engineering (Page 26).

Procedure

1. Select the "New > Station > Control (PLC) - general" command from the context menu of the "S001 Cabinet" object.
2. Select the "New > Rack" command from the context menu of the control.

2.5.6 Determining bills of quantities by means of bulk processing

Aim

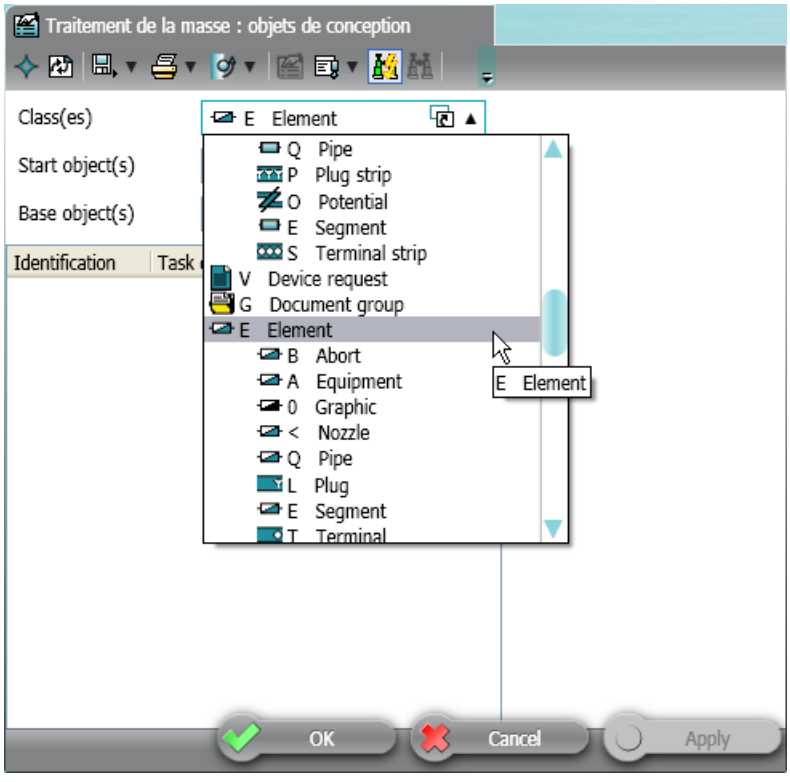
If the engineering follows the COMOS standard model, prepared queries are available at the following location:

<Subunit> > "A50 EI&C" > "0 Set implementation and location pointer"

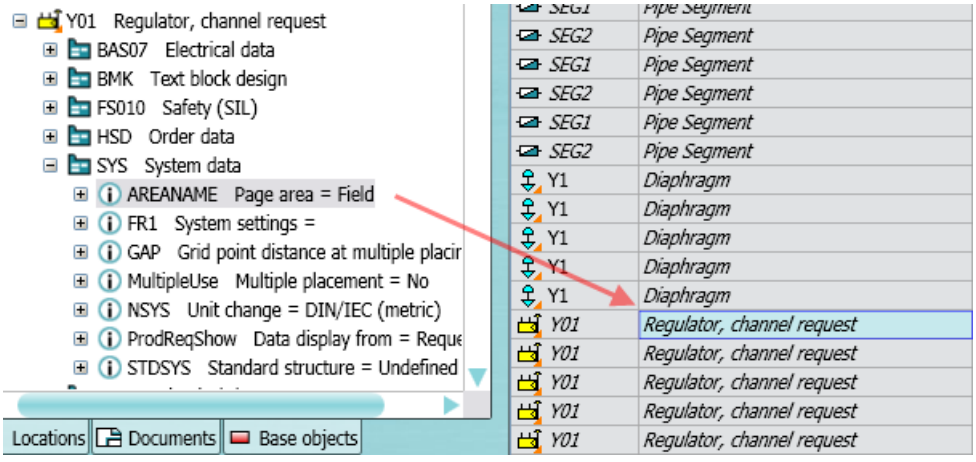
The following section presents a method for determining the quantity, if the prepared queries are not sufficient.

Procedure

1. In the menu bar, select "Extra > Bulk processing > Bulk processing for engineering objects > Bulk processing: Engineering objects".
2. Select the "Element" entry from the "Class(es)" list:
   All channel requests in the "A50 EI&C" folder have the "Element" class.



3. Drag the subunit from the "Units" tab into the "Start object" field.
4. Click "Search" in the menu bar.
   The time until the result is displayed increases depending on the amount of data scanned.
   You can limit the result further by using filters:
   - Select any input on the "Units" tab.
   - Search below the channel request for the "Y00T00001 System data" tab and the "Y00A00179 Sheet zone" attribute.
   - Drag&drop the attribute from the navigator into the header of the table.
     A new column is created.



You can filter the column.
For example: If, for example, you select the "Controller" filter text, you have filtered according to inputs and outputs.
You can see at the bottom left in the status bar of the bulk processing how many objects there are in the filtered list.

2.5.7 Creating I/O modules and I/O channels

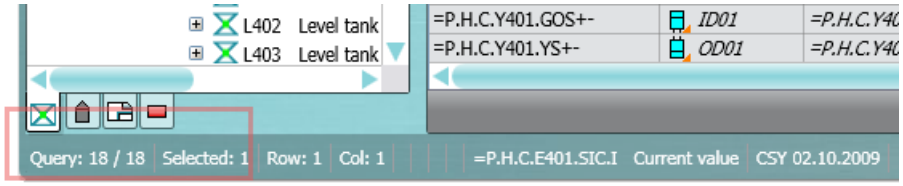
Requirement

The functions have been created and applied.

The control has been created.

Procedure

1. Determine the number of channel requests for the controller.
   The number of controller modules that are required depends on the number of inputs and outputs that have been created on the unit side.
   - On the "Units" tab open the query "<Subsystem> > "A50 EI&C" > "003 Set implementation and pointer" > "01 Implement PLC".
   - Make sure that the following start object is set: "A50 EI&C"
     The requirements are listed in the query. The quantity is displayed in the status bar.



2. On the "Locations" tab in the Navigator, open the path to the rack for your control.
   Example: "Plant > Building > Floor/Level > Room, control center > Cabinet > Station/ Control > Rack"
3. Select the required I/O modules under "New > Input/output" in the context menu of the "Mounting rack," object as follows:
   - I/O channels are assembled in I/O modules. There are I/O modules with different numbers of I/O channels.
   - The number of I/O channels and the type of desired I/O module determines the number of I/O modules to be created. The corresponding number of I/O channels is created automatically along with the I/O module.

Result

The required I/O channels have been created on the "Locations" tab in the Navigator.

In a subsequent step, each channel request from the "Units" tab will be assigned to an I/O channel in the "Locations" tab.

2.5.8 Setting the location pointers of the devices

Introduction

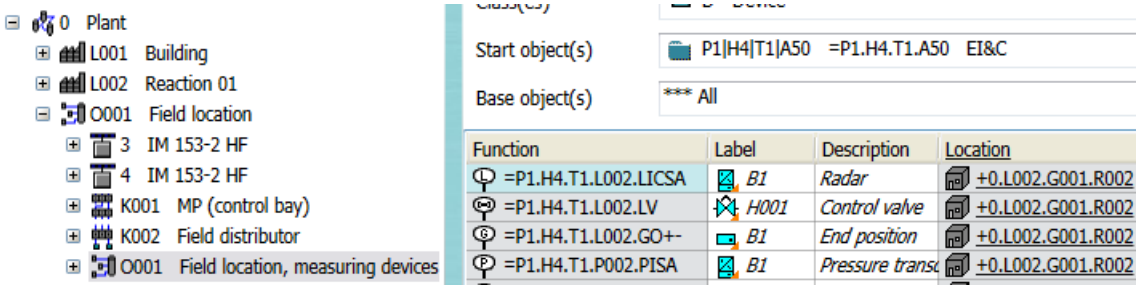
The devices are divided into two categories: You assign the sensors to "Field location, measuring devices". You assign the actuators to "Field location, actuating devices.

Requirement

- The device requirements in the "A50 EI&C" folder have been created.
- The "Field location, measuring devices" and "Field location, actuating devices" field locations have been created.

Procedure

1. Select the object "Set implementation and pointer" on the "Units" tab in the Navigator below the "A50 EI&C" folder.
2. Open the "Set location pointer" query below the "Set implementation and pointer" object.
3. Drag&drop the "A50 EI&C" folder from the Navigator into the "Start object(s)" field.
   The query lists all the device requests below the start object. In the COMOS standard model the device requests are in the "A10 Basic engineering" folders. The query shows whether the devices have a location pointer and, if so, where that pointer references.
4. On the "Locations" tab in the Navigator, open the path to the "Field location, measuring devices" and "Field location, actuating devices" objects below the field location.
5. To assign "Field location, measuring devices" to a sensor, drag&drop the "Field location, measuring devices" object from the Navigator to the "Location" column of the corresponding sensor in the query.



6. To assign "Field location, actuating devices" to an actuator, drag&drop the "Field location, actuating devices" object from the Navigator to the "Location" column of the corresponding actuator in the query.
7. Apply the changes and close the "Set location pointer" query.

Setting the location pointer for several devices at the same time

1. In the query, select all devices to which you want to assign the same location pointer.
2. Drag&drop to move the field location from the Navigator to the "Location" column of the query.

2.5.9 Setting the signal of an I/O channel

Introduction

In the COMOS standard model, the channel request is created at the signal. In this case, the signal is applied with the implementation.

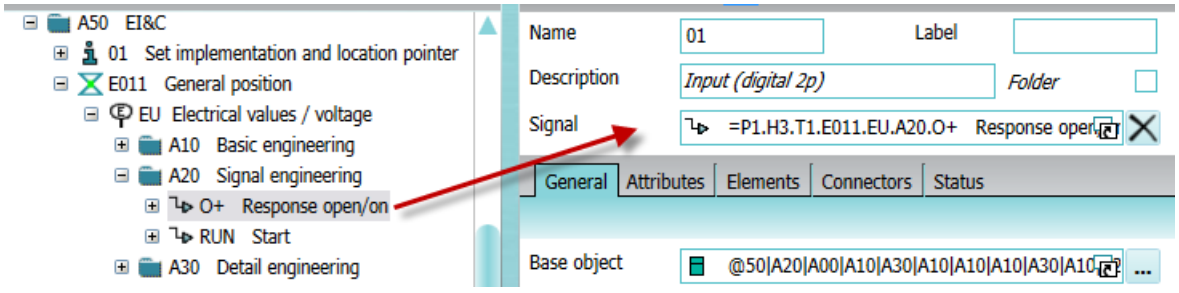
If a different structure of the engineering data was created, the signal must be assigned manually to the I/O channel.

Requirement

- The channel requests have each been implemented with an I/O channel.
- At least one signal has been created below the function.

Procedure

1. Select the "Navigate > Implementation" command from the context menu of the channel request.
   The I/O channel is shown on the "Locations" tab in the Navigator.
2. Open the properties of the I/O channel.
3. On the "Units" tab in the Navigator, open the path to the signal in the "A20 Signal engineering" folder.
4. Drag&drop the signal from the "Signal engineering" folder to the upper area of the I/O channel properties.



   The "Signal" field is inserted with the reference to the signal.
5. Click the "Apply" button.

Alternative procedure

Use the "Signal/Device assignment manager" plugin. See Open the "Signal/Device assignment manager" plugin (Page 37).

2.5.10 Creating field distributors and terminals

Requirement

- The location structure has been created.

Creating field terminals using field distributors

1. Determine the number of required terminals.
   - In the COMOS standard model, each measuring transducer uses two field terminals and the following applies: The number of field terminals equals the number of measuring transducers multiplied by 2.
   - Alternatively: Use a bulk processing for engineering objects.
2. On the "Locations" tab in the Navigator, open the path to the "Field distributor" object below the field location.
3. In the context menu of the "Field distributor" object, select under "New > Field distributor > <Required field distributor" as follows:
   - Field terminals are assembled in field distributors. There are field distributors with a varying number of field terminals.
   - The number of required field terminals and the type of desired field distributor determines the number of field distributors to be created. The respective number of field terminals is created automatically along with the field distributor.

2.5.11 Creating a control cabinet and equipping it with basic objects

Requirement

- The "Locations" tab is open in the Navigator.
- The basic structures for location engineering are created.

Procedure

1. Select the "New > Cabinet" command from the context menu of the "R001 Room, control center" object.
2. Select the "New > Cabinet construction > Cabinet system" command from the context menu of the "S001 Cabinet" object
   The following objects are created automatically below the cabinet system:
   - "Cabinet frame"
   - "Fastening parts"
   - "Power rail"
   - "Channels"
   - "Accessories"
3. Optional: Select further objects for mechanical design of the cabinet in the context menu.

Alternatively: Creating device requests at the "Cabinet" object

In the COMOS standard model, the device requests are created below the function in the "A10 Basic engineering" folder on the "Units" tab. To create the device requests below the cabinet as an alternative, follow these steps:

1. Select the "New > Rack" command from the context menu of the "S001 Cabinet" object
2. Select the required device requirements in the context menu of the "F001 Rack" object.

Result

The cabinet design has been created and equipped with basic objects.

Additional cabinet engineering takes place on a report.

2.6 Implementing requests in the location view

2.6.1 Administration

You can find more information on this topic in "EI&C Administration", keyword "Requests and implementations".

2.6.2 Definition of "implementation"

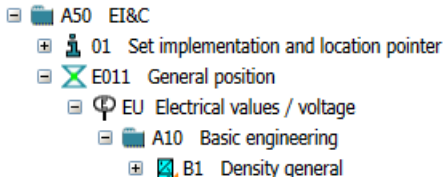
Most of the objects that had been created and edited up to this point had the "request" property. They are therefore not manufacturer devices. Manufacturer devices are devices that can be ordered from a manufacturer.

"Implementation" is now performed. This involves the assignment of another engineering object.

An "implementation" can be:

- A request (concrete) is implemented on a request (general).
  A very general engineering object is replaced by another engineering object that is somewhat more concrete. No product data is assigned at this time.
- A manufacturer device ("product") is implemented on a (concrete or general) request.
  A device request is replaced by a manufacturer device.
  See Using product data and manufacturer devices (Page 249).

Objects with the "Request" property can be recognized in the Navigator by the small red triangle in the lower right corner.



Request objects have the "Implementation" field in the properties on the "General" tab.

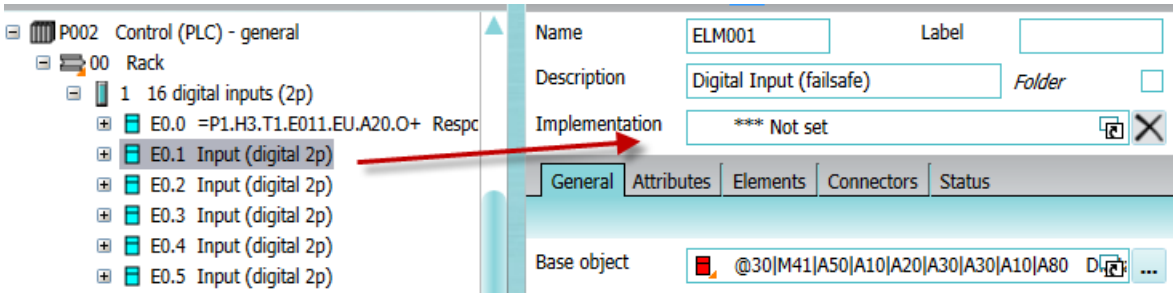
2.6.3 Implement channel requests individually

Requirement

- Channel requests have been created on the "Units" tab.
- I/O channels have been created on the "Locations" tab.

Procedure

1. On the "Units" tab in the Navigator, in the "A10 Basic Engineering" folder, open the properties of the channel request which you want to implement individually.
2. On the "Locations" tab in the Navigator, open the path to the I/O channel which you want to assign to the channel request.
3. Drag&drop to move the I/O channel from the Navigator to the "Implementation" field.



4. Apply the changes and close the channel request properties.

Result

I/O channels have been assigned to the channel requests.

2.6.4 Implement channel requests per query

Introduction

If you want to implement the I/O channels in bulk, perform the implementation per "Implement requests" query.

Requirement

Channel requests have been created on the "Units" tab.

I/O channels have been created on the "Locations" tab.

Setting the implementation using the "Implement requests" query

1. From the COMOS menu, select the "Extra > Query > Tools > Implement requests" command. The "Implement requests" query opens in the working area.
2. Select the "E Element" entry from the "Class" list.
3. In the Navigator, drag the "A50 EI&C" folder from the "Units" tab into the "Start object" field. The requests below the "A50 EI&C" folder are shown in the query.
4. Filter the list if needed.
5. On the "Locations" tab in the Navigator, open the path to the I/O channels which you want to assign to the inputs or outputs.
6. Use drag&drop to move the I/O channels from the Navigator to the "Implementation" column of the corresponding inputs or outputs.

Implementing several I/O channels at the same time

To implement several objects at the same time, proceed as follows:

1. Select the inputs or outputs in the query which you want to implement.
2. In the "Locations" tab in the Navigator, select the I/O channels which you want to assign to the inputs and outputs.
3. Use drag&drop to move the I/O channels from the Navigator to the "Implementation" column of the query.
4. Apply the changes and close the properties.

2.7 Importing basic engineering data from Excel

You can find more information on this topic in "Automation Interfaces", keyword "Generic Excel import".

2.8 Assigning signals

2.8.1 Open the "Signal/Device assignment manager" plugin

Procedure

1. In the menu bar, select the command "Plugins > Automation > Signal/Device assignment manager".
   The plugin opens. See User interface reference (Page 39).
2. Drag&drop the desired start object to the "Start object" field.
   Use, for example, the node "A50 EI&C". You can enter several objects as the start object.

Result

The evaluation starts automatically. All the found hardware signals and devices are listed. Hardware signals always have one object.

See: Specifying the base set (Page 37).

2.8.2 Specifying the base set

Requirement

- The "Signal/Device assignment manager" plugin is open. See also Open the "Signal/Device assignment manager" plugin (Page 37).

Procedure

1. Right-click in the head of a column.
2. Select the desired filter options.
3. Repeat steps 1 and 2 for each column in which you wish to set filters.

The number of displayed signals decreases or increases depending on the selection.

See also

User interface reference (Page 39)

2.8.3 Checking the signal application

Note
Reduced performance
The display of signal application can lead to reduced performance.

Procedure

To check the current use of a signal, click the "+" button in the details column of a signal:

Result

The following information is shown in the detail area:

- Positions below which the signal is located
- Devices to which this signal is assigned

2.8.4 Assigning signals to devices

Assigning an individual signal to a device

1. Select an entry in the Signals list.
2. Select an entry in the Device list.
3. Click the "Assign" button between the two lists.

Assigning an individual signal to a connector

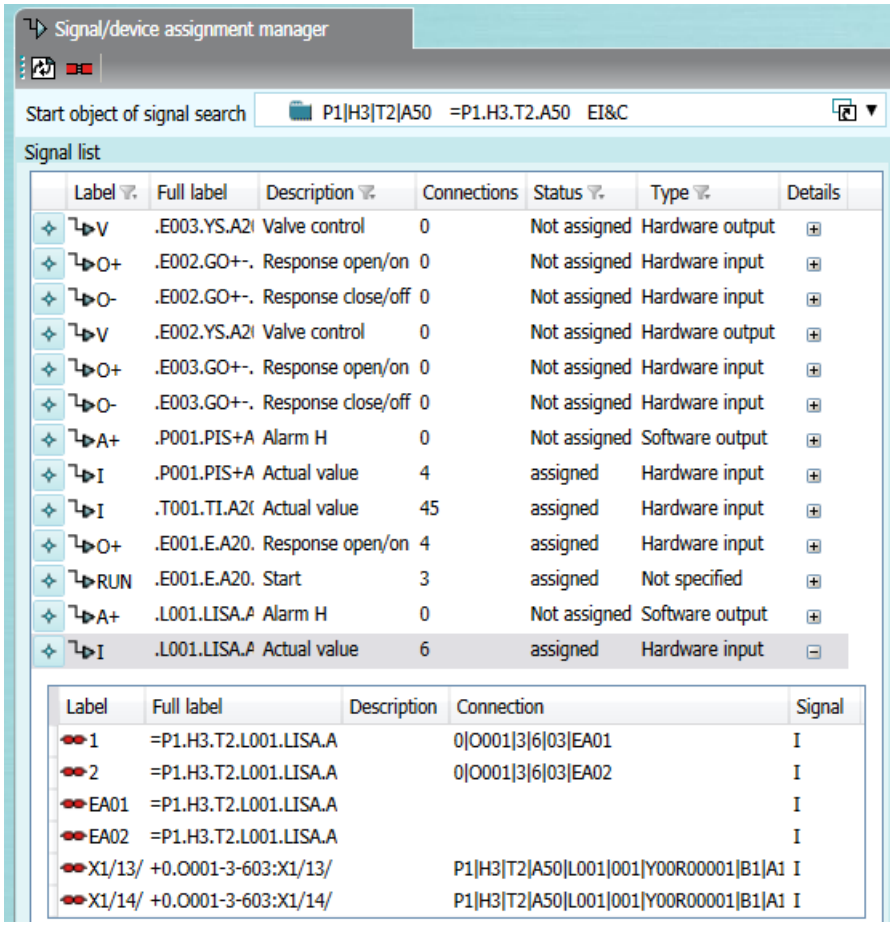
1. Select an entry in the Signals list.
2. Select an entry in the Device list.
3. Open the "Detail" area for the device.
4. Select a connector in the "Detail" area.
5. Click the "Assign" button between the two lists.

Assigning an individual signal to multiple devices

1. Select an entry in the Signals list.
2. Select multiple entries in the Devices list.
3. Click the "Assign" button between the two lists.

2.8.5 User interface reference

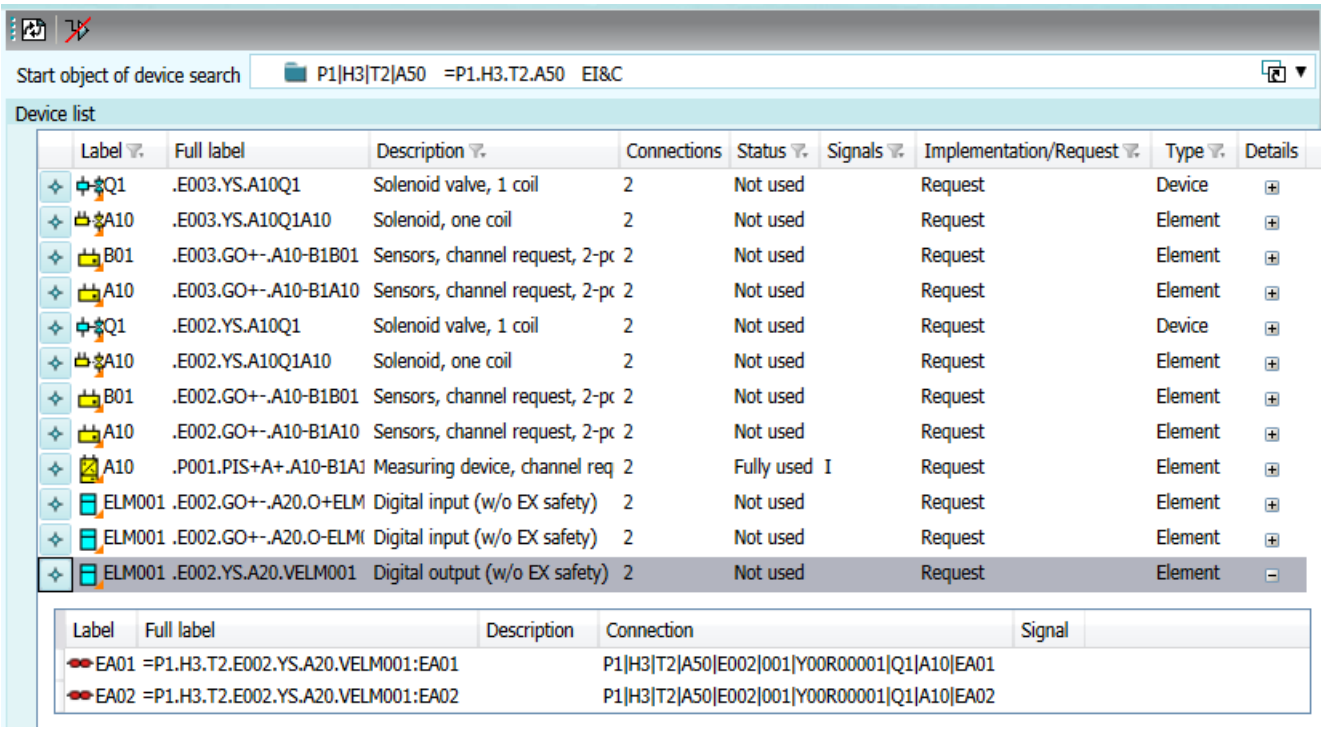
"Search for signals" area



You can use the button "Show single-line connectors" to activate and deactivate the search option.

Column	Description
"Navigate" button	Use this button to navigate to the object in the Navigator.
"Label"	Relative designation (Relative label) to the start object of the list
"Fulllabel"	FullLabel of the object
"Description"	Description of the object
"Connectors"	Number of connectors. The number "0" indicates that a signal has not been assigned to a device.
"Status"	Status of the signal
"Type"	Type of signal
"Details"	Displays the connectors of the selected signal.

"Device search" area



You can delete the assignment of a device by using the button "Clear".

Column	Description
"Navigate" button	Use this button to navigate to the object in the Navigator.
"Label"	Relative designation (Relative label) to the start object of the list
"Fulllabel"	FullLabel of the object
"Description"	Description of the object
"Connectors"	Number of connectors. The number "0" indicates that a device does not have a signal.
"Status"	Status of the device
"Signals"	Indicates if signals are linked
"Implementation status"	• "Request" Lists devices for which the "Request" setting has been activated in the properties. • "Implementation" Lists devices for which the "Request" setting has been deactivated in the properties. All devices without the "Request" setting automatically count as manufacturer devices.
"Type"	Type of device
"Details"	Displays the connectors of the selected signal.

Filtering columns

You have the possibility of using predefined filters in a column. To filter a column click the button in the column header and select the desired options.

2.9 Creating marshalling/loops

2.9.1 Connecting and marshalling with the Marshalling Designer

2.9.1.1 Overview of the Marshalling Designer

Purpose

The Marshalling Designer helps you to connect your field devices with the location view (signal adaptation and control). The following applies in the COMOS standard model: The field devices were moved by the implementation of the "Units" tab into the folder "Field location, measuring devices" and "Field location, actuating devices".

After marshalling, the loop of a function can be created or displayed using a position diagram. The loop diagram can be created during the function engineering.

See Structure of the generated function module (Page 18).

2.9.1.2 Creating and opening the Marshalling Designer

Requirement

Location engineering has been performed in the "Locations" tab.

See Location planning (Page 25).

Procedure

1. Open the "Locations" tab.
2. Select "Plant > Building > Floor/Level > Room, control center > Cabinet"
3. Select the "Marshalling designer" command from the context menu of the cabinet.
   The object "Marshalling designer" is created below the cabinet.
4. To open the marshalling designer double-click the object.

2.9.1.3 Display of the marshalling designer

Zoom

If you want to zoom the display of the fields in the marshalling cabinet in or out, left-click in any field and move the mouse wheel in the desired direction.

Move all fields

To move all fields of the marshalling designer right-click on any field, keep the mouse button pressed and drag the fields to the desired position. For example, this provides the option of centering the view.

2.9.1.4 Placing field devices

Definition of "collector"

Two collectors are placed on the marshalling designer: the "Collector for measuring devices" and the "Collector for actuating devices". The field devices themselves do not appear on the marshalling designer. Instead, the collectors save the information regarding the field devices on the marshalling designer that should be taken into consideration.

Requirement

- Location engineering has been performed in the "Locations" tab.
  See Location planning (Page 25).
- On the "Locations" tab in the Navigator, the paths to the field devices below the "Field location, measuring devices" and "Field location, actuating devices" field locations are available.
- The requests are implemented in the location view.
  See Implementing requests in the location view (Page 34).
- The signals have been created.

Placing collectors

1. Click "Collector for actuating devices" in the "Node pallet" control group.
   The collector is displayed in the "Field (sensor)" section. The number of devices is evaluated directly. An object for the collector is created in the Navigator below the "Marshalling designer" field.
2. Click "Collector for measuring devices" in the "Node pallet" control group.
   The collector is displayed in the "Field (sensor)" section. The number of devices is evaluated directly. An object for the collector is created in the Navigator below the "Marshalling designer" field.
3. In order to move a collector, drag&drop the field to the desired position.

Assigning collectors to field devices

You can now drag your field devices into the collectors.

1. Select the collector field in the marshalling designer.
   The "Settings" area appears.
2. Select the desired tab in the Navigator.
   Multiple selection is possible.
3. Use drag&drop to move the devices from the "Settings" area into the field "Drag&Drop devices here".
4. Repeat this process for each additional collector field.

After you have dragged the devices into the "Settings" area, the number of assigned devices is displayed in the "Number of devices" row:

2.9.1.5 Placing field distributors

Requirement

- The field distributor has been created in the "Locations" tab.
  See also chapter .

Procedure

1. Select the field distributor.
2. Look for your field distributor with terminals below the "Field distributor" object.
3. Drag&drop to move the field distributor with terminals from the Navigator to the "Field distributor" area on the marshalling designer.

2.9.1.6 Placing marshalling racks

Requirement

- The marshalling cabinets have been created on the "Locations" tab in the Navigator.

Procedure

1. Find the "MP (field bay)" object below the cabinet.
2. Select a marshalling rack with terminals below the "MP (field bay)" object.
3. Use drag&drop to move the marshalling cabinet with terminals from the Navigator into the "Field side" area on the marshalling designer.
4. Select a marshalling rack with terminals below the "MP (control bay)" object.
5. Use drag&drop to move the marshalling cabinet with terminals from the Navigator into the "Control side" area on the marshalling designer.

2.9.1.7 Placing I/O modules (control)

Requirement

The I/O channels have been created on the "Locations" tab in the Navigator.

Procedure

1. Find the "Control (PLC) - general" object.
2. Select the input module with I/O channels below the rack.
3. Use drag&drop to move the input module with the I/O channel from the Navigator to the "Control" area on the Marshalling Designer.

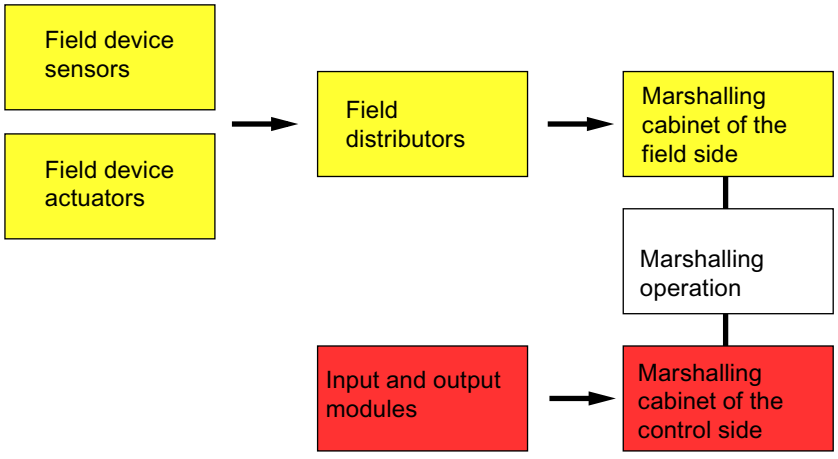
2.9.1.8 Connecting objects on the Marshalling Designer

Requirement

The following objects have been placed on the marshalling designer:

- Field devices
- Field distributors
- Marshalling racks (field and control side)
- I/O modules

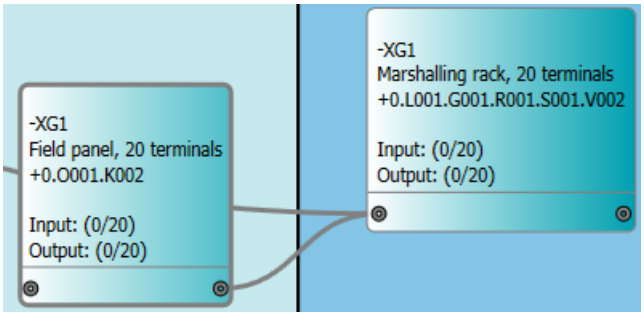
Sequence of connections



Procedure

1. Left-click the connection point of an object.

   If you move the mouse, you see a gray line leading away from the connection point. Connection points for inputs are located in bottom left-hand side on object. Connection points for outputs are located in bottom left-hand side on object.
2. Click with the mouse pointer on the suitable connection point of the object with which you wish to make the connection.
3. Make a right-click to conclude the connection.



4. Repeat the process as long as necessary until you have connected the field side (as it should be topologically) and the control side to one another.

2.9.1.9 Working with several terminal strips and connection patterns

Support of connection patterns in the Marshalling designer

The Marshalling designer evaluates the connection patterns for terminal strips. See Purpose and requirement of the connection pattern (Page 238).

Dividing the connection patterns by means of partial assignment

When a connection is set in the Marshalling designer, there is an automatic background search for connection patterns for the terminal strips. When a connection pattern is available, there is an automatic attempt to apply the connection pattern exactly to one terminal strip. If this is not possible, partial assignment is attempted. With partial assignment, a part of the connection pattern is first applied to one terminal strip and the rest of the connection pattern is applied to another terminal strip.

You can find more information on this topic in "EI&C Administration", keyword "Partial assignment".

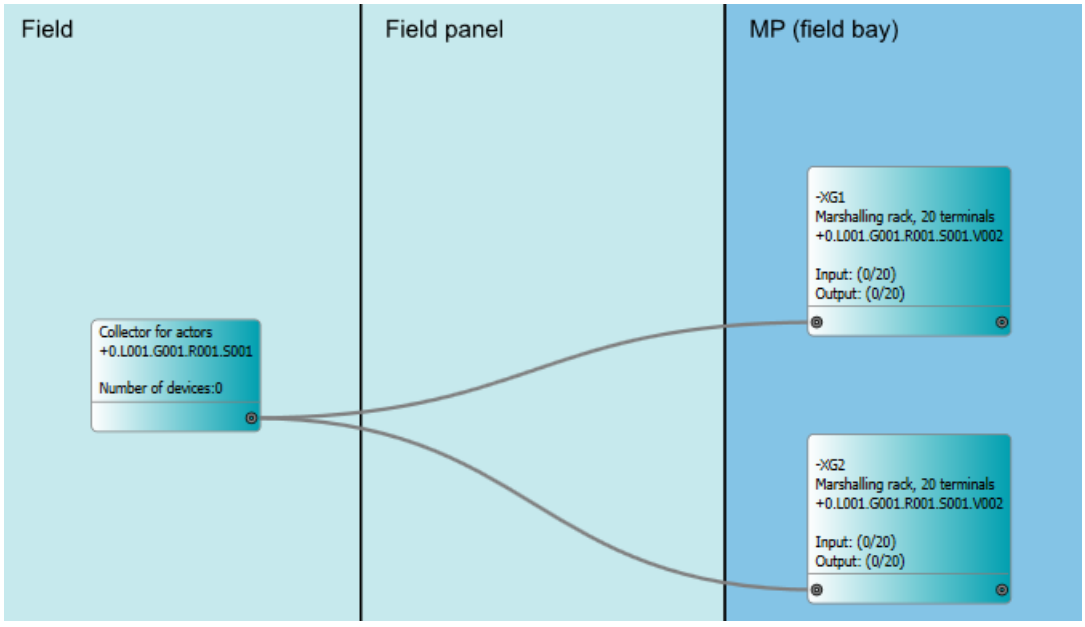
Example of using several, specialized terminal strips

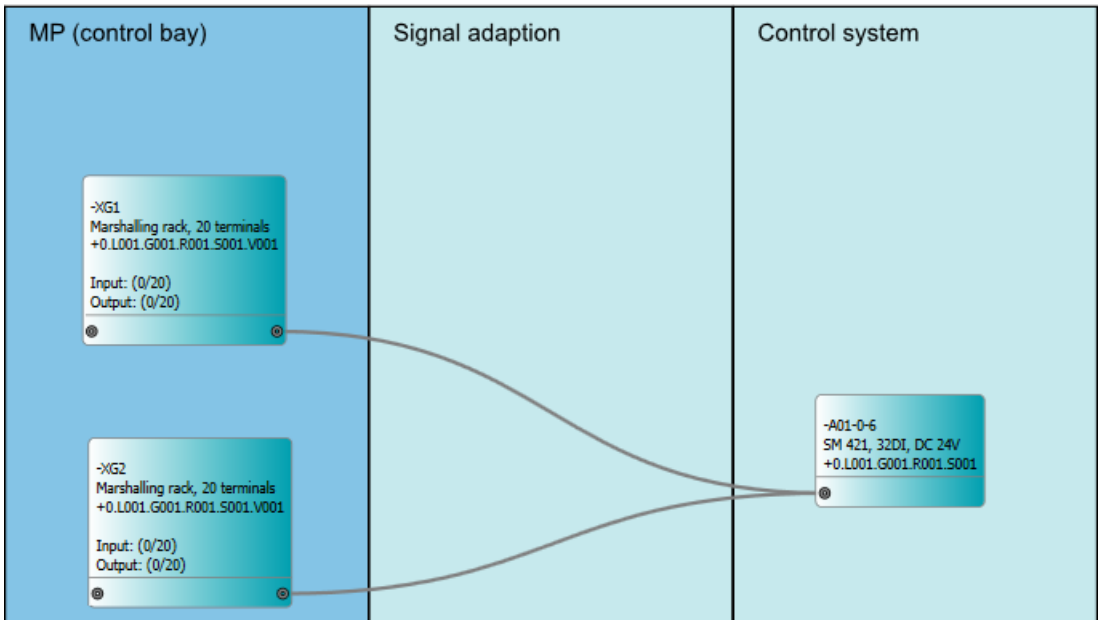
The example has the following structure:

- A collector for sensors is placed in the "Field" area.
- Two terminal strips that each take on a part of the connection pattern are placed in the MP "Field bay" area.

- Two terminal strips that apply the partial assignments of the connection pattern to an I/O module of the PLC are placed in the "MP (control bay)" area.
- An I/O module of a rack is placed in the "PLC" area.







Note

Implementing the example

1. Drag a connection from the collector for sensors to the left terminal strip.
2. Drag a dynamic connection from the first connection to the right terminal strip.
3. Follow the exact same procedure for the connection between the terminal strips in the "MP (control bay) area and the I/O module.

2.9.1.10 Carrying out marshalling on the Marshalling Designer

Requirement

- The objects have been placed on the Marshalling Designer and are connected.
- The Marshalling Designer is open.

Procedure

1. Click the "Create marshalling connections" button.

Displaying a loop

1. On the "Units" tab in the Navigator, open the path to the position of the M&C point at which you made the marshalling connections.
2. Select the "New > Loop diagram" command from the context menu of the position.
   The loop diagram is created.
3. Open the loop diagram.
   If marshalling was successful, the marshalling connections will be generated automatically and the loop will appear on the loop diagram.

2.9.1.11 Recalculating wiring

Procedure

You can update the wiring when working with the marshalling designer.

Click the "Recalculate wiring" button in the Marshalling Designer to do this.

Note

When working with several loop diagrams

During saving, the "Recalculate wiring" function corresponds to the report options of the loop diagram.

Recommendation:

Set the report options of the loop diagram to read, so that changes in the wiring are only applied by the marshalling designer and not from the loop diagrams. This prevents the wiring from a loop diagram from being overwritten in another way by the function "Update wiring".

2.9.2 Object-based connecting and marshalling

2.9.2.1 Connecting devices to the field distributor

Requirement

The devices have been created.

The field distributor has been created.

Procedure

1. In the "Locations" tab in the Navigator, open the path to the "K001 Field distributor".
2. You will find the field distributor with terminals below the field distributor. Open the properties of the field distributor with terminals.

3. On the "Locations" tab in the Navigator, open the paths to the field devices below the "Field location, sensor technology" and "Field location, actuator technology" field locations.
4. Use drag&drop to move the connectors of the channel requests of the field devices from the Navigator to the left "Device" column.
5. Select the "All in list" command from the menu.
   The connectors are inserted in the left "Device" column.
6. Apply the changes and close the field distributor properties.

2.9.2.2 Connecting the field distributor to the marshalling rack (field side)

Requirement

- The marshalling rack (field side) has been created.
- The field distributor has been created.
- The path to the field distributor is open on the "Locations" tab in the Navigator.

Procedure

1. Open the properties of the field distributor below the object "K001 Field distributor".
2. Select the "Terminal strip" tab.
3. In the "Locations" tab in the Navigator, open the path to the marshalling rack of the field side.
4. Use drag&drop to move the marshalling rack below the "F MP (field bay)" object from the Navigator to the left "Device" column.
5. Select the "Output > All in list" command from the menu.
   The terminals of the marshalling rack of the field side are inserted in the left "Device" column.
6. Apply the changes and close the field distributor properties.

2.9.2.3 Connecting the marshalling rack (control side) to I/O channels

Requirement

- The marshalling rack for the control side has been created.
- The I/O channels have been created.
- The path to the marshalling rack of the control side is open on the "Locations" tab in the Navigator.
- The path to the I/O channels is open in the "Locations" tab in the Navigator.

Procedure

1. Open the properties of the marshalling rack of the control side below the "C MP (control bay)" object.
2. Select the "Terminal strip" tab.
3. Use drag&drop to move the connectors of the I/O channels from the Navigator to the right "Device" column.
4. Select the "All in list" command from the menu.
5. Apply the changes and close the properties of the marshalling rack of the control side.

2.9.2.4 Creating marshalling with automatic connection

Introduction

The loop diagram has the ability to "Place automatically" and "Connect automatically". Marshalling and the loop are available for this when you first open it.

Requirement

- The devices are connected to the field distributor.
- The field distributor is connected to the marshalling rack of the field side.
- The marshalling rack of the control side is connected to the I/O channels.

Procedure

1. On the "Units" tab in the Navigator, open the path to the position for whose I&C point you performed the marshalling.
2. Select the "New > EFS.001 Loop diagram, detail display" command from the context menu of the position.
   The loop diagram is created.
3. Open the loop diagram.
   When the preliminary work has been successfully completed, the connections (marshalling) are generated automatically and the loop is shown on the loop diagram.

Result

Marshalling has been carried out. The loop is shown on the loop diagram.

2.9.3 Connecting and marshalling with the loop diagram

2.9.3.1 Signal-based search and classification search

Two alternatives for loops

Loops can be created by means of two different methods:

- Signal-based search
  Starting with a signal, the associated components are searched for with the help of signal tracking and placed on the loop.
- Classification search
  Requirement: The objects are classified with "LP".
  In this approach, the base data of the components is first classified and then the start objects of the loop are determined based on the classification.

Classifications in the database

You can find more information on this topic in "EI&C Administration", keyword "Controlling search methods for placeable objects".

2.9.3.2 Structure of the point diagram

Task

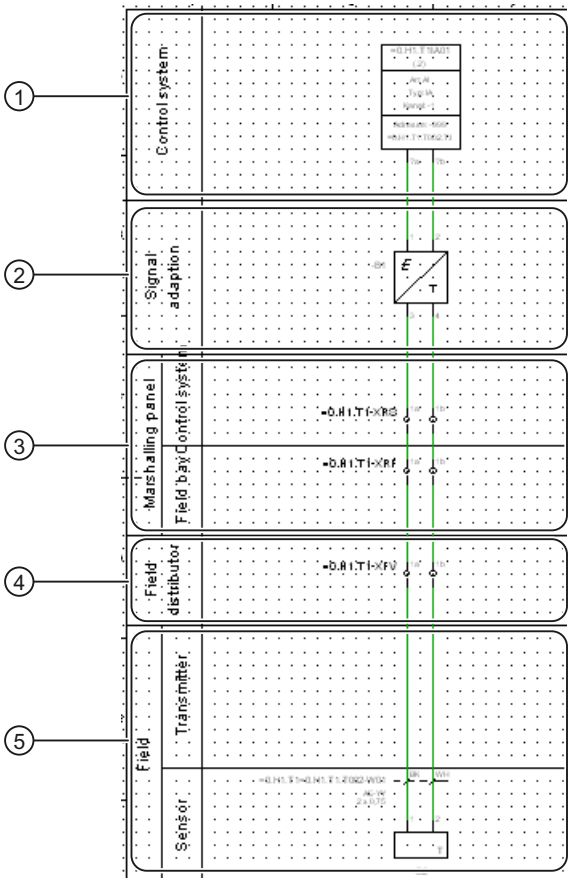
The loop of the I&C point is displayed on the loop diagram. The first time a loop diagram is opened, the objects of an I&C point are placed in the corresponding areas of the loop diagram via the "Place automatically" and "Connect automatically" automatic functions and the loop is automatically created.

Structure

The loop diagram is available in two versions:

- Loop diagram according to EN:
  The loop is displayed vertically on the loop diagram. The direction is from the bottom (field side) to the top (control side).
- Loop diagram according to ANSI:
  The loop is displayed horizontally on the loop diagram. The direction is from left (field side) to right (control side).

The following graphic shows the structure of the loop diagram according to EN:



① "Control" area
The I/O channels are located in the "Control" area.

② "Signal adjustment" area
The devices for the signal adjustment are found in the "Signal adaption" area.

③ "marshalling rack" area
The "marshalling rack" area is further divided into the "Control" and "Field" areas.
The terminals of the "control side" marshalling rack are found in the "Control" area.
The terminals of the "field side" marshalling rack are found in the "Field" area.

④ "Field distributor" area
The terminals of the field distributor are found in the "Field distributor" area.

⑤ "Field" area
The "Field" area is further divided into the "Transducer" and "Sensor" areas.
The measuring transmitters are found in the "Transducer" area.
The sensor devices are found in the "Sensor" area.

Defaults

The appearance and behavior of the loop diagram is controlled via the option script of the report template. You can find more information on this topic in "EI&C Administration", keyword "Position diagrams".

2.9.3.3 Automatic Placing (AutoLoop)

"Place automatically" automatic function

The "Place automatically" automatic function is performed the first time the loop diagram is opened. This function automatically places all objects below the position onto the loop diagram. Further, the terminals that are connected in the database are placed on the loop diagram.

Automatic page break

A multipage loop diagram will be created if necessary.

Subsequent changes

If you make changes at the objects below the position after you have opened the loop diagram for the first time, these changes will not automatically appear on the loop diagram. In order to show the changes on the loop diagram, select the "Options > Place automatically" command from the context menu of the loop diagram.

Start object for placing automatically

When the "Place automatically" function is called, COMOS makes a distinction depending on whether or not the objects are selected on the loop diagram:

- If no objects on the loop diagram are selected, the "Place automatically" function is called for the entire diagram:
  - COMOS searches for the owner of the loop diagram.
  - In the case of objects that are located parallel to the loop diagram or objects located below these objects, COMOS recursively checks across the joined connectors to determine the objects with which these start objects are associated.
  - A search for placeable objects is carried out below the signals.
  - All objects which have already been placed on the loop diagram are set as start objects for the search for new objects to be placed.
- If one or more objects on the loop diagram are selected, the "Place automatically" function is only called for the selected objects:
  COMOS searches only below objects selected on the loop diagram for new objects to be placed. The search is performed recursively via the joined connectors.

Controlling automatic placing

You can find more information on this topic in "EI&C Administration", keyword "AutoLoop".

2.9.3.4 Automatic Connecting (AutoConnect)

Applying connections

The "Connect automatically" automatic function is performed the first time the loop diagram is opened. All read connections are displayed on the loop diagram. COMOS reads from the database whether or not two objects are joined to one another via their connectors and displays this connection (blue connecting line).

Connect automatically

If the loop diagram has been correspondingly prepared, all objects which are graphically arranged below one another on the loop diagram are also joined on the loop diagram and in the database via their connectors.

Cables

If the cables in the database have already been laid and the loop diagram is correspondingly prepared, the cables are placed on the loop diagram.

Control automatic connection

You can find more information on this topic in "EI&C Administration", keyword "AutoConnect".

2.9.3.5 Up-to-dateness check and repeated evaluation

Up-to-dateness check

The following information is checked during the up-to-dateness check:

- Objects to be placed.
  If there is a difference from the objects currently placed in the loop diagram, the loop diagram is marked as outdated.
- Connections and cables.
  If inconsistent or missing connections or cables are detected, the loop diagram is marked as outdated.

Repeated evaluation

If a loop diagram is evaluated again, then:

- local changes in the symbols are retained.
- the segments are retained.
- the revision history is retained.

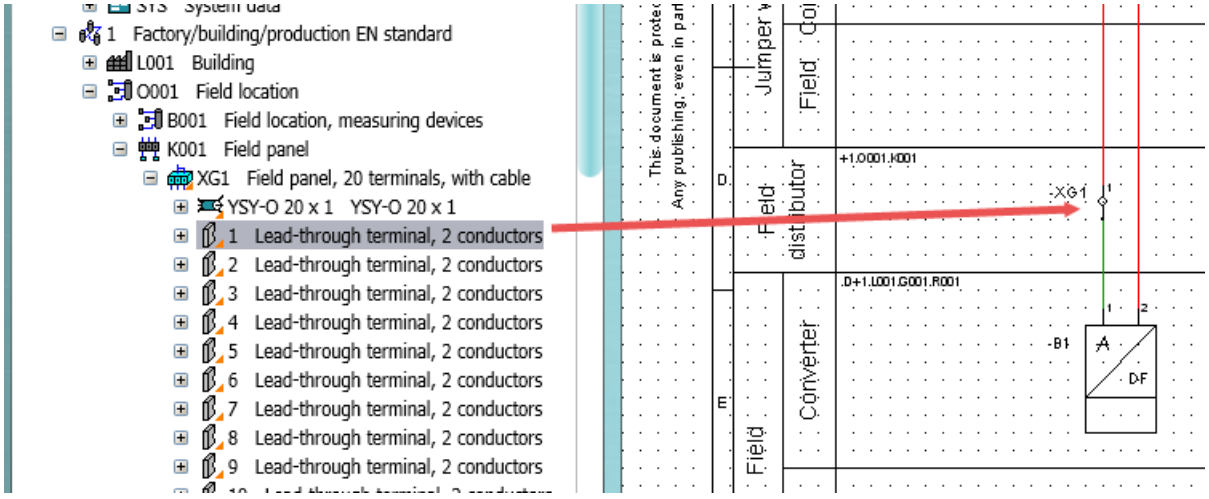
2.9.3.6 Connecting and marshalling with the loop diagram

Introduction

You can also carry out marshalling directly on the loop diagram.

Procedure

1. Open the loop diagram of the position for which you want to carry out the marshalling.
2. On the "Locations" tab in the Navigator, open the path to the field distributor.
3. Use drag&drop to move the terminals below the field distributor from the Navigator to the "Field distributor" area on the loop diagram.



4. Repeat these steps for all components you want to place on the loop diagram.
5. Open the context menu of the loop diagram without having selected an object.
6. Select the "Options > Connect automatically" command in the context menu.

Result

Marshalling has been carried out on the loop diagram.

2.9.4 Using the "Loop Document Manager" plugin

2.9.4.1 Purpose of the "Loop Document Manager"

The "Loop Document Manager" collects interactive reports. If there are loop diagrams among the interactive reports, the up-to-dateness of the loop diagrams is checked.

2.9.4.2 Opening the "Loop Document Manager" plugin

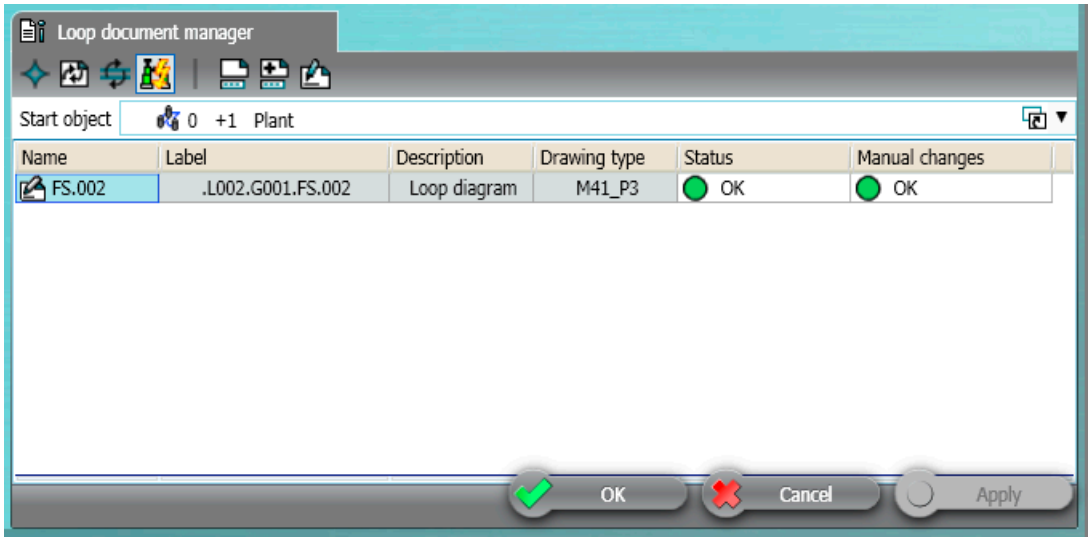
Requirement

The loop diagrams have been created.

Procedure

1. In the menu bar select the "Plugins > Automation > Loop Document Manager" command. The plugin is opened.
2. Drag an object from the Navigator into the "Start object" field.
   If the "Search at once" button is activated, the evaluation starts automatically.
3. If the "Search at once" button is deactivated, click the "Update" button in the toolbar of the plugin.

User interface



Button	Description
"Navigate"	Use this button to navigate to the loop diagram in the Navigator.
"Refresh - F5"	Use this button to start the evaluation of the start object. If the "Search at once" button is activated, the evaluation starts automatically.
"Refresh (selected)"	Use this button to refresh the selected subset of the loops.
"Search at once"	Use this button to control whether the evaluation is started immediately at a set start object in the plugin.
"Complete recalculation"	Use this button to reject all the manual changes and recalculate the loop diagram.

Button	Description
"Recalculate and retain manual changes"	With this button, you recalculate the loop diagram. All the manual changes are not overwritten.
"Open"	Use this button to open the selected loop diagram. A loop diagram has to be selected.

See also

Updating loop diagrams (Page 59)

2.9.4.3 Rules of up-to-dateness check

Search for placed objects by means of classification or labeling segments

In the first step, you identify which objects should be placed on the loop.

There are two possibilities for this identification:

- Identification of the objects by means of LP classification
  This is the faster method.
- Identification of the objects by assignment to a sheet area on the loop.
  This assignment takes place via the labeling segments.

You can find additional information on these two methods in the "EI&C Administration" manual, keyword "Controlling search methods for placeable objects".

Up-to-dateness check: Content changes

- First, all objects to be placed are calculated. If too many or too few objects have been placed, the loop must be generated again.
- All connections and cables are then checked. Inconsistent or missing connections or cables require a new generation of the loop.
- Then, changes to the potentials are checked.

Up-to-dateness check: Manual changes

The following actions are considered manual changes to the loop:

- Changing the position of an object placed by AutoLoop
- Changing the symbol of an object on the loop
- Manually inserted texts, lines, segments, etc.

The user decides whether the manual changes are retained or discarded.

Events that are not recorded

- Changes to the revision history

2.9.4.4 Checking the currentness of loops

Requirement

- The loop diagrams have been created.
- The "Loop Document Manager" plugin is opened. See also chapter Opening the "Loop Document Manager" plugin (Page 57).

Checking the currentness of loops

1. Select one of the following options:
   - Select the "Refresh - F5" command in the context menu to check all the loops.
   - In order to check a subset of the loops select the desired lines. Select the "Refresh (selected)" command in the context menu.
2. Check the "Status" column:
   - Green ("OK")
     Loop diagram is up-to-date.
   - Red ("Update")
     Loop diagram must be updated.
   - Gray ("not evaluated")
     The interactive report is not a loop diagram.
3. Check the "Manual changes" column:
   - Green ("OK")
     Loop diagram is up-to-date.
   - Yellow ("Refresh")
     The display is activated in the case of the following changes:
     Change in the position of devices and terminals (cables, shields and connections are not included)
     Change at the symbol of an object
     Inserting user-defined texts, lines or segments
   - Gray ("not evaluated")
     The interactive report is not a loop diagram.

2.9.4.5 Updating loop diagrams

You have two options to update the organizational chart:

- By means of the context menu of the organizational chart in the Navigator
- In the "Loop Document Manager" plugin.

Requirement

- The loop diagrams have been created.

Check organizational chart

1. Double-click the desired loop diagram in the list.
2. Check the contents of the loop diagram.

Discarding manual changes and recalculating the loop diagram

To completely update a loop diagram, follow these steps:

In the Navigator:

1. Select an organizational chart that you want to update in the Navigator.
2. Select the "Update loop > Complete recalculation" command in the context menu.

In the "Loop Document Manager":

1. Open the plugin: See also chapter Opening the "Loop Document Manager" plugin (Page 57).
2. Select a line with the entry "Update!".
3. Select the "Update loop > Complete recalculation" command in the context menu.

Update the loop diagram but retain manual changes

To update a loop diagram only partly, follow these steps:

In the Navigator:

1. Select an organizational chart that you want to update in the Navigator.
2. Select the "Update loop > Keep recalculation and manual changes" command in the context menu.

In the "Loop Document Manager":

1. Open the plugin: See also chapter Opening the "Loop Document Manager" plugin (Page 57).
2. Select a line with the entry "Update!".
3. Select the "Update loop > Keep recalculation and manual changes" command in the context menu.
   The following steps are executed in this case:
   - The movement of symbols on the loop is buffered.
   - Local editing of the symbols is buffered.
   - The loop is recalculated.
   - The movements and the local editing are assigned once again.
     You can find more information on this topic in the "EI&C Administration" manual, keywords "Loop diagrams and loops" and "Controlling search methods for placeable objects".

2.9.5 More information on loops and position diagrams

You can find more information on this topic in "EI&C Administration", keyword "Position diagrams and loops".

2.10 Working with the "Brownfield Loop Wizard" plugin

2.10.1 Overview

The "Brownfield Loop Wizard" plugin is an extension of the generic Excel import and facilitates the automatic implementation of devices, such as terminals, in existing planning on the basis of list information.

The aims are to import existing data on the basis of lists and predefined templates as well as to import supplier information.

You can find more information on this topic in "Automation Interfaces", keyword "Generic Excel import".

2.10.2 Loading/creating import rules

Requirement

- A configuration has been created in the "Generic Excel import" plugin.
  Alternatively, you create the configuration in the Brownfield Loop Wizard.
  You can find more information on this topic in "Automation Interfaces", keyword "Editing the data assignment configuration".

Procedure

1. Select the "Plugins > Automation > Brownfield Loop Wizard" command from the menu bar.
   The plugin opens. The "Load/create import rules" tab is displayed.
2. Select the desired Excel data.
   You can find more information on this topic in "Automation Interfaces", keyword "Selecting Excel data".
3. Set a start object.
   You can find more information on this topic in "Automation Interfaces", keyword "Defining import settings".
4. Load the desired configuration.
5. You have the option to change the rules of the loaded configuration. To do this, use the "Edit import rules" area.
   You can find more information on this topic in "Automation Interfaces", keyword "Working with rules".

2.10.3 Defining implementation rules

Introduction

Once you have loaded an Excel worksheet and a configuration is created, define the implementation rules. Each implementation rule is divided into the requirement and the implementation. To this purpose, you use the existing implementation rules to define a start object and limit the search results based on filter rules. The result of the search is a requirement object and an implementation object, which are interconnected via the implementation pointer.

Requirement

- The import rules are created.
  See Loading/creating import rules (Page 61).

Procedure

1. Select the "Implementation" tab in the "Brownfield Loop Wizard" plugin.
2. Click on the "Create implementation rule" button in the "Implementation rules" area.
3. Drag an import rule from the "Import rules" area as a start object and drop it into the "Rule for start object" field of the "Requirement" column.
4. If the start object imports multiple objects, for example, for creating a template, restrict the search results using the filter rules. Click on the "Create filter rule" button.
   The Filter rules area is displayed.
5. Make the required settings.
   The filter rules use the AND and OR operators.
   - If you use the search by name, description or labels, the search value is read from the Excel worksheet. Drag the corresponding column from the "Settings" field into the field for the search rule.
   - If you are using the search for a classification, select the appropriate information. They apply to all lines of the Excel worksheet.

   If the search finds more than one requirement object, the implementation process for the current row is aborted and an error message is generated.
6. Drag an import rule from the "Import rules" area as a start object and drop it into the "Rule for start object" field of the "Implementation" column.
   If the import rule have been created for one object only, no filter rules are necessary.
7. If you want to implement different types of objects, create additional implementation rules accordingly.
8. To save the current settings for the implementation, save your configuration.
9. Click the "Execute" button.

Result

The implementation pointers are set in the properties of the objects. If implementation objects are missing, they are created.

2.11 Using the Implementation manager

2.11.1 Purpose of the Implementation manager

Purpose

The Implementation manager expands the details area of an engineering task.

The Implementation manager makes it easier to implement manufacturer devices on requests (more precisely: on request objects). Requests are created as part of Basic Engineering. Requests are created as follows, for example:

- Automatically in function engineering
  See: Creating and designing functions (Page 17).
- Manually by the user
  See: Creating devices, channel requests and signals (Page 20).

Users have the following options:

- Implementing or removing requests automatically
- Implementing or removing requests manually

The Implementation manager is used in the detail area of various tasks.

Examples

Tasks that use the Implementation manager:

- "Implement PLC channels"
- "Implement safety-relevant PLC channels"

See also

Requirement of the Implementation manager (Page 63)

2.11.2 Requirement of the Implementation manager

- Purpose of the Implementation manager (Page 63)

Requirements

- The implementation manager was administered. See "EI&C Administration" > "Administering the implementation manager"
- A task is selected in the task user interface.
  You can find more information on this topic in "Modular Engineering", keyword "Selecting tasks in the properties window".
- The selected task uses the Implementation manager.
  You can find more information on this topic in "EI&C Administration", keyword "Preparing engineering tasks".

See also

Refreshing the table contents (Page 64)

2.11.3 Refreshing the table contents

Requirement

See chapter Requirement of the Implementation manager (Page 63).

Procedure

You first need to find the existing requests and manufacturer devices in the Implementation manager.

1. Optional: Select an entry in the standard table.
   Note: Administration determines whether additional entries are available in the standard table. You can find additional information on this topic in the "EI&C Administration" manual, keyword "Preparing engineering tasks".
2. Click the "Update" button to the left of the standard table:
   

Result

The following applies if nothing else is set:

- The query for the left table uses the following object as a reference: task, "Target object" table column.
- The query for the right table uses the following object as a reference: task, "Owner" table column.

See also

Implementing all requests automatically (Page 65)

Implementing selected requests automatically (Page 65)

Disconnecting implementations (Page 66)

Implementing requests manually (Page 67)

2.11.4 Implementing all requests automatically

Requirement

The table contents in the Implementation manager are up-to-date.

See chapter Refreshing the table contents (Page 64).

Procedure

1. In the toolbar, click on the arrow of the "Implement <...>" button:

2. Select the "Implement (all)" menu command.

Result

The free requests are automatically implemented on the free implementations in descending order. They are moved to the row of their implementation in the right-hand table.

Note

If there are not enough free implementations available, you can create new ones with the "New" button:



See also

Reference Implementation manager (Page 68)

2.11.5 Implementing selected requests automatically

Requirement

The table contents in the Implementation manager are up-to-date.

See chapter Refreshing the table contents (Page 64).

Procedure

1. Hold down the <Ctrl> key and select the requests to be implemented automatically in the left-hand table of the Implementation manager.
2. In the toolbar, click on the arrow of the "Implement <...>" button:

3. Select the "Implement (selected)" menu command.

Result

The selected requests are automatically implemented on the free implementations in descending order. They are moved to the row of their implementation in the right-hand table.

Note

If there are not enough free implementations available, you can create new ones with the "New" button:



See also

Reference Implementation manager (Page 68)

2.11.6 Disconnecting implementations

Requirement

The table contents in the Implementation manager are up-to-date.

See chapter Refreshing the table contents (Page 64).

Dissolving selected implementations

1. Hold down the <Ctrl> key and select the requests/implementations whose implementation you want to disconnect in the right-hand table of the Implementation manager.
2. In the toolbar of the Implementation manager, click on the arrow of the "Dissolve" <...> button.
3. Select the "Dissolve (selected)" menu command.

Dissolving all implementations

1. In the toolbar of the Implementation manager, click on the arrow of the "Dissolve" <...> button.
   You will see a menu.
2. Select the "Dissolve (all)" menu command.

See also

Reference Implementation manager (Page 68)

2.11.7 Implementing requests manually

Requirement

The table contents in the Implementation manager are up-to-date.

See chapter Refreshing the table contents (Page 64).

Procedure

Use drag&drop to drag a free request from the left-hand table to the row of the required implementation in the right-hand table.

If there are not enough free implementations available, you can create new ones with the "New" button:



Result

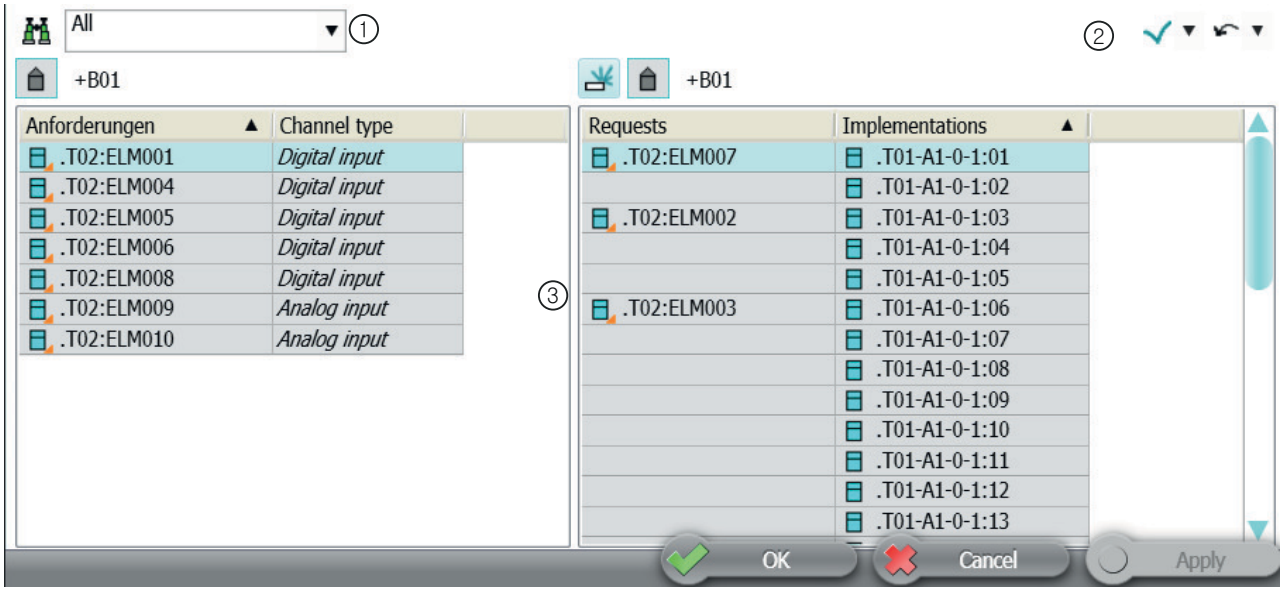
The request is implemented. It is moved to the row of its implementation in the right-hand table.

See also

Reference Implementation manager (Page 68)

2.11.8 Reference Implementation manager

Structure



1 Object filter: List for filtering the "Requests" and "Implementations" tables

2 Toolbar

3 Tables for implementing the channels

Toolbar

Icon	Tooltip	Description
	"Refresh"	Applies the selected object filters to the "Requests" and "Implementations" tables.
	"New"	Creates a new manufacturer device.
▾	"Implement (all)" Or "Implement (selected)"	Implements all requests or only the selected requests depending on which command was selected using the button with the arrow.
▾	"Dissolve (all)" Or "Dissolve (selected)"	Removes all implementations or only the selected implementations depending on which command was selected with the button with the arrow.

Table area

Depending on the task in which the Implementation manager is loaded, the table area has the following layout:

1	Left table	All free request objects
	Right table	All implementation objects and the implemented request objects
2	Left table	All free implementation objects
	Right table	All request objects and the implemented request objects

Note

This documentation describes the situation where the free requests are on the left and the implementations and implemented requests are on the right.

2.12 Using the Potential manager

2.12.1 Purpose of the Potential manager

Purpose

The Potential Manager is a version of the Implementation manager especially prepared for potentials. Target potentials are treated as a request and source potentials as a manufacturer device. A source potential can be the implementation of any number of target potentials.

The following user actions are possible:

- Implementing source potentials on target potentials
- Determining the order in which the function groups are connected to the source potential
- Changing the connection direction

See also

Refreshing the table contents (Page 70)

2.12.2 Refreshing the table contents

Requirement

- The objective is clear.
  See chapter Purpose of the Potential manager (Page 69).
- A task is selected in the task user interface
  You can find more information on this topic in the "Modular Engineering" manual, keyword "Selecting tasks in the properties window".
- The selected task uses the Potential manager.
  You can find more information on this topic in the "EI&C Administration" manual, keyword "Preparing engineering tasks". Keyword "Administrating the Potential manager"

Procedure

You first need to search for the existing target potentials and source potentials in the Potential manager.

1. Optional: Select one entry in each of the standard tables.
   Left filter: Filters the tables based on the functional classification of the potentials.
   Center filter: The filter values which are visible depend on the database.
   Right filter: The filter values which are visible depend on which filter value is set in the center filter
2. Click the "Update" button to the left of the standard table:

   
3. Optional: Select the "Hide occupied signals" button in the toolbar:

   You now only see the source potentials that have not been implemented.

Result

The free target potentials are visible in the left-hand table.

The source potentials and the first assigned source potential are listed in the table on the right.

In the "Connected" column, you will see the number of target potentials on which the source potential is implemented.

See also

Implementing potentials (Page 71)

2.12.3 Implementing potentials

Requirement

The table contents in the Potential manager are up-to-date.

See chapter Refreshing the table contents (Page 70).

Procedure

1. You have the following options:

Method	Procedure
Implementing manually	Use drag&drop to drag a target potential to a source potential. The potentials are implemented.
Implementing all potentials with the same name automatically	1. In the toolbar, click on the arrow of the "Implement <...>" button:  2. Select the "Implement (same names)" menu command. For every target potential that has not yet been implemented, there is a search for a source potential of the same name. The potentials are implemented.
Selected potentials	1. Select the required target potentials in the left-hand table. 2. In the toolbar, click on the arrow of the "Implement <...>" button:  3. Select the "Implement (selected)" menu command. There is a search for source potentials of the same name for the target potentials. The potentials are implemented.

2. Activate the task.
3. Execute the task.

See also

Disconnecting potentials (Page 71)

2.12.4 Disconnecting potentials

Requirement

The potentials are implemented.

See chapter Implementing potentials (Page 71).

Procedure

1. Select a source potential.
2. In the toolbar, click "Disconnect selected":

If the source potential is implemented on more than one target potential, only the implementation of the first target potential is disconnected.

2.13 Notes on engineering tasks for EI & C

The following engineering tasks are prepared for EI&C:

- "@20 > B20 > M41 > A10 > A10 Create sheet macro"
- "@20 > B20 > M41 > A10 > A20 Generate path macro"
- "@20 > B20 > M41 > A30 > A10 Control loop" (Create assembly) / "@20 > B20 > M41 > A30 > A10 Control loop" (Merge)

These engineering tasks use the "Template search" detail area.

You can find additional information on searching for templates in the "EI&C Administration" manual, keyword "Administer template search" and keyword "Administer target search".

2.14 Using the "Characteristics for connections" tool

2.14.1 Purpose of the "Characteristics for connections" plugin

With the "Characteristics for connections" tool, connectors or connection objects in a project below a selected node can be preset with either connection information or numbers in bulk.

Objective

The "Characteristics for connections" tool can be used for both projects with connection objects and projects without connection objects:

- In the case of projects without connection objects, connection information is written to the connectors. A cable object is not required for this.
- In the case of projects with connection objects, connection information is written to the existing connection objects.

This connection information is displayed on the report at the connection if the ShowConnectionInfo option is "true" in the report template or a connection information object was dragged onto the connection.

Condition

Opt for one of the two procedures:

- Setting connection information in bulk (Page 75)
- Setting the wire number or connection label in bulk (Page 76)

Delineation

The following connectors are not changed:

- Connectors with a set wire
- Connectors with a "@B" cable index
- Connectors not of the type "EE/I&C"

2.14.2 Interface reference of the "Characteristics for connections" plugin

Start object: Document or node from the location structure under which the connectors should be preset.

"Cross-section/color" tab

"Filter settings" control group:

- Placement
  - All: All connectors of all devices under the start node are taken into consideration.
  - Only placed: Only connectors of placed devices under the start node are taken into consideration.

Potential

- All: All connectors, regardless of their potential.
- Only with logical potential: Only connectors that are connected to a logical potential.
- All without logical potential: Only connectors that are not connected to a logical potential.
- "Set parameters for connected connectors" option: Is only visible for projects without connection objects.
  - selected: All existing connected or newly connected connections on the report receive the values from the "Color\cross-section" tab and the "Number" tab when executed.
  - deselected: All connected or half-open connections on the report receive the values from the "Color\cross-section" tab and the "Number" tab when executed.

"Parameters" control group:

- Cable type: All cable types of the "Y10M41N00007 Type information for wires" standard table
- Cross-section: All wire cross-sections of the "Y10M41N00005 Wire cross-section" standard table
- Color: All wire colors of the "Y10M41N00004 Wire color" standard table

- Reset
  - selected: Deletes all existing connection information on the connector
  - deselected: -
- Overwrite
  - deselected: Sets connection information only at connectors at which no connection information has been defined.
  - selected: Overwrites all existing connection information on the connector as well

"Execute" button: Applies all previously selected filter settings to the connection objects.

Please note: For purely graphical connections without connection objects, this deletes all settings on the "Number" tab from the connections.

"Number" tab

"Filter settings" control group:

- Placement
  - All: All connectors of all devices under the start node are taken into consideration.
  - Only placed: Only connectors of placed devices under the start node are taken into consideration.
  - "In documents increasing from start number": The connections are numbered independently in each document.
- Potential
  - All: All connectors, regardless of their potential.
  - Only with logical potential: All connectors that are connected to a logical potential.
  - All without logical potential: Only connectors that are not connected to a logical potential.
- Potential-dependent
  - deselected: The properties are set sequentially regardless of the potential.
  - selected: The properties are set sequentially depending on potential. This option is used for connection networks. On the "Number" tab, this option causes connected connectors to receive the same number.

"Parameters" control group:

- Start number: Defines the number with which numbering should start.
- Reset
  - selected: Deletes all existing line numbers or connection labels at the connectors.
  - deselected: -
- Overwrite
  - deselected: Only sets line numbers or connection labels at those connectors where line numbers or connection labels have not yet been defined.
  - selected: Also overwrites all existing line numbers or connection labels at the connectors.

"Execute" button: Applies all previously selected filter settings to the connection objects.

Please note: For purely graphical connections without connection objects, this deletes all settings on the "Cross-section/color" tab from the connections.

"Protocol" area

Shows a protocol of all executed actions. Entries of the "Error" or "Warning" type indicate that an action cannot be executed. You can view the source and position of corresponding objects.

2.14.3 Setting connection information in bulk

Introduction

Note

Note the following restrictions:

- If a wire is assigned to the connection, the specified connection information of the wire always has priority and cannot be overwritten with the "Characteristics for connections" tool.

Setting connection information

1. In the "Extra > Detail > Characteristics for connections" tool, select the "Cross-section/color" tab.
2. Drag&drop a node or document from the Navigator into the "Start object" field.
3. Set the filter settings for "Placement" and "Potential"
4. Set the parameters:
   - "Cable type"
   - "Cross-section"
   - "Color"
   - "Overwrite"
     Option disabled: The connection information is retained at objects which have already been edited.
     Option enabled: The connection information is overwritten at objects which have already been edited.
5. Click on the "Execute" button.
   The specified connection information is written to all connectors or connection objects below the selected start node.

Resetting connection information

1. Remove your selection in the parameters:
   - "Cable type"
   - "Cross-section"
   - "Color"
2. Select the "Reset" option
3. Click on the "Execute" button.
   The connection information is written to all connectors or connection objects below the selected start node.

See also

Interface reference of the "Characteristics for connections" plugin (Page 73)

2.14.4 Setting the wire number or connection label in bulk

Introduction

- The wire number is assigned in projects without connection objects.
  See also "Preparing wire numbers"
- The connection label is set as number in projects with connection objects.
  See also "Displaying connection information on the report"

Note

Note the following restrictions:

- Only numerical values can be used as connection labels of the connection objects in the "Characteristics for connections" tool.
- If a wire is assigned to the connection, the specified connection information of the wire always has priority and cannot be overwritten with the "Characteristics for connections" tool.

Setting the number

1. Select the "Number" tab in the "Extra > Detail > Characteristics for connections" tool.
2. Drag&drop a folder or document from the Navigator into the "Start object" field.
3. Select the filter settings for "Placement" and "Potential"

4. Set the parameters:
   - "Start number"
   - "Overwrite"
     Option deactivated: The number is retained for objects already edited.
     Option activated: The number is overwritten for objects already edited.
5. Click on the "Execute" button.
   The connectors are numbered in each document from top left to bottom right.

Resetting the number

1. Select the filter settings for "Placement" and "Potential"
2. Select the option:
   - "Reset"
3. Click on the "Execute" button.
   The numbers are deleted at all connectors or connection objects below the selected start node

See also

Interface reference of the "Characteristics for connections" plugin (Page 73)

Function engineering in the layout

3

3.1 Basics of working with layout documents

3.1.1 "Layout" definition

The layout is an interactive report in which the functional engineering of the unit is carried out.

You carry out the following tasks in the layout:

- You place the objects required for the unit.
- You arrange the objects according to the spatial and functional requirements.
- You structure the objects according to functional aspects in subunits and assign areas to the objects. To do this, you work with polygons.

See also

Overall workflow in the layout (Page 81)

Definition of "engineering task" (Page 79)

User interface reference (Page 92)

3.1.2 Definition of "engineering task"

Definition

Engineering tasks automate the creation and configuration of planning objects. They simplify the engineering.

Each task has an owner. This is the start point of the task.

Tasks execute the following actions:

- Setting references
- Moving engineering objects in the Navigator structure
- Copying in templates below the owner
- User-defined actions, for example starting complex calculations, and saving them in the owner and in the objects linked to it.

Administrators can adapt tasks to the company-specific requirements so that they provide you with optimum and flexible support in your projects. You can find more information on this topic in the "Platform Administration" manual, keyword "Preparing engineering tasks".

Task classes

Tasks are divided into task classes. Some classes are executed during the functional engineering in the layout, others during the detail engineering in the tasks user interface.

User interfaces for working with tasks

- Unit layout
- Tasks user interface

See also

Tasks in the layout (Page 80)

3.1.3 Tasks in the layout

Introduction

Some task classes are only used in the layout: There, users work with polygons, configure the tasks in this way and then execute them.

Available task classes

You work in the layout with the following task classes:

- "COMOS general"
  Structure tasks sort objects in the Navigator into the substructure of the unit. See chapter Definition of "engineering task" (Page 79).
- Areas
  Area tasks arrange objects into areas. Area tasks configure the relationships between the objects. During the detail engineering COMOS evaluates the relationships and then supports you in the configuration of the hardware and software of the objects.
  Example: The task "Assigned station" You determine on the layout which objects are controlled by the same PLC. During the implementation of the PLC channels in the detail engineering COMOS only offers the hardware channels of the devices lying under these function objects.

3.2 Overall workflow in the layout

Requirement

- You have experience of working with interactive reports.
  You can find more information on this topic in the "COMOS Getting Started" manual, keyword "Working with interactive reports" and in the "Platform Operation" manual, keyword "Working with interactive reports".
- You have experience of working with polygons.
  See also chapter Working with polygons (Page 87).

Procedure

1. Create the layout.
   See also chapter Creating a layout (Page 82).
2. Open the layout.
3. Place the required objects.
   See also chapter Placing objects (Page 82).
4. Create a functional assignment between the objects by placing them correspondingly on the layout.
5. Sort the objects into the substructure of the unit.
   See also chapter Assign subunits (Page 83).
6. Configure the relationships between the objects.
   See also chapter Assigning areas (Page 83).

Finding task targets

You can search for the following in the layout:

- Polygons of a specific area.
- Objects of an area to which the same target is assigned.

See also

Finding task targets (Page 86)

Creating a layout (Page 82)

3.3 Creating a layout

Requirement

- The project is open.
- The "Units" tab is visible in the Navigator.
- The unit has been created.

Procedure

1. Select the unit.
2. From the context menu select the "New > General > New Document" command.
   The properties of the document are opened.
3. Select the "Report" tab.
4. Click the "Set reference..." button next to the "Report template" field.
   The "Select report template" window opens.
5. Select the "Base" option.
6. Select the template "A30 > T > TL > TL_A10 > TL_A10 Plant layout, A3, landscape".
7. Confirm your entries.
   The "Select report template" window is closed.
   The text "Plant layout, A3, landscape" is shown in the "Description" field of the "General" tab.
8. Confirm your entries.

3.4 Placing objects

Possible methods

- Via the toolbar
- Drag&drop from the "Base objects" tab in the Navigator
- Drag&drop from the "Units" tab in the Navigator
- Using the context menu in the report, "Place template" command
  You can find more information on this topic in the "Platform Operation" manual, keyword "Placing objects by inserting a template".

Symbol bar

You can find more information on this topic in the "Platform Operation" manual, keyword "Placing objects via the symbol bar"

The objects are created under the unit layout in the Navigator.

Display of the symbols on the layout

In order to display the symbols of the objects on the unit layout the symbols have to be defined for this report. You can find additional information on this topic in the "EI&C Administration" manual, keyword "Preparing functional engineering in the layout".

3.5 Assign subunits

Aim

Assigning objects to a subunit.

Requirement

- The objects are placed in the unit layout.

Procedure

1. Click "Identify" in the standard toolbar: 
2. Click "Map task targets" in the report-specific toolbar: 
3. Select "Move" from the list of polygon types in the report-specific toolbar:
4. Draw a polygon that encloses the required objects.
   See chapter Drawing polygons (Page 88).
5. Select an object of the suitable type as the target.
   See chapter Selecting targets (Page 84).
6. Calculate the task targets.
   See also chapter Calculate task targets (Page 86).

Result

The objects that are enclosed by the polygon are moved under the object selected as the target in the Navigator.

3.6 Assigning areas

Aim

To configure the relationships between the objects by assigning them to areas.

Requirement

Objects are placed in the layout.

Procedure

1. Click "Identify" in the standard toolbar: 
2. Click "Map task targets" in the report-specific toolbar: 
3. Select the desired area in the report-specific toolbar in the list with the polygon types.
4. Draw a polygon that encloses the required objects.
   See also chapter Drawing polygons (Page 88).
5. Select a suitable object as the target.
   See also chapter Selecting targets (Page 84).
6. Calculate the task targets.
   See also chapter Calculate task targets (Page 86).

Result

The objects enclosed by the polygon contain a task of the corresponding task class. The task stores a reference to the target. In the detail engineering, you are supported as described above.

3.7 Selecting targets

Requirement

A polygon has been drawn in the unit layout.

Principle

How you proceed depends on whether the polygon encloses possible targets and, if so, how many:

Number of possible targets	Principle
None	You select a target in the "Select target" window.
One	The target is set as the default. If necessary, you can select a different target in the "Select target" window.
More than one	The targets are offered for selection in the "Select targets" window. If necessary, you can select a different target there.

In all situations, you can also create a new object and select it as the target.

Opening the "Select target" window

1. Select the polygon.
   The following control elements are shown in the report-specific toolbar:



2. Click "...".
   The "Select target" window opens.

Procedure when there is no possible target

1. Open the "Select target" window.
   You can see the entire unit tree.
2. Select the required object or create a new object and select it as the target.
3. Confirm your entries.

Procedure when there is a possible target

To retain the preset target, no user action is necessary.

To select a target other than the preset target, follow exactly the same procedure as when there is no possible target.

Procedure when there is more than one possible target

If there are several possible targets in the polygon, the "Select targets" window is opened automatically as soon as you have finished drawing the polygon. In the unit tree of the window, you see only the nodes in which possible targets are located. You cannot create new objects.

Proceed as follows:

1. Select the required object in the "Select target" window.
2. Confirm your entries.

To select an object located outside the polygon as the target, follow exactly the same procedure as when there is no possible target.

See also

Drawing polygons (Page 88)

3.8 Calculate task targets

Possibilities

- Calculate the task targets of all polygons at once.
  The time required for processing increases with larger unit layouts.
- Calculate selected task targets:
  - The task targets of one or more polygons
  - The task targets of selected objects

  The time required for processing is reduced accordingly.

All task targets at once

Click the button "Calculate task targets" in the report-specific toolbar: 

Selected task targets

1. Select the desired polygons or objects in the layout.
2. Select one of the following commands from the context menu:
   - "Calculate task targets (selected resources)"
   - "Calculate task targets for level<Number>"
   - "Calculate task targets (selected levels)"

3.9 Finding task targets

Aim

Search in the layout:

- For polygons of a specific area.
- For objects of an area to which the same target is assigned.

Procedure

1. Click "Identify" in the standard toolbar: 
2. Click "Search task targets" in the report-specific toolbar: 
   Two lists are shown in the toolbar.

3. Select the area that you want to search for in the left-hand list.
   Only the polygons of this type as well as the structure polygons are now visible on the layout.
   In the right-hand list, all the targets set for the polygon type you have selected will be offered.
4. Select a target.
   All the objects that have this object as the target are selected on the unit layout.

3.10 Working with polygons

3.10.1 Basics of polygons

Definition of a "polygon"

Polygons are symbols with which you assign a common target to several objects in the layout.

There are several types of polygon.

Structure polygons

Structure polygons sort the objects into the substructure of the unit.

The database contains polygons that sort the objects into the subunits.

Area polygons

Area polygons configure the relationships between the objects.

Example:

- The objects which monitor an HMI.
- The objects which control a PLC.

Principle

- You draw a polygon around all the objects that will have the same target. On the one hand this identifies the objects groups as being graphically related. On the other hand each object group located within the polygon is given an engineering task.
- You assign a target to the polygon and thus to the task as well. By calculating task targets, the objects are edited as specified by the task.

Example: You determine on the unit layout which objects are controlled by the same PLC. During the implementation of the PLC channels in the detail engineering COMOS only offers the device channels of these objects.

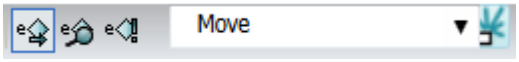
3.10.2 Drawing polygons

Requirement

- The layout is opened.
- Objects are placed in the layout.

Procedure

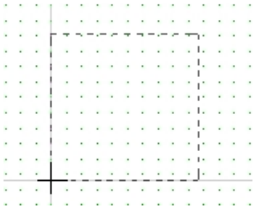
1. Click "Identify" in the standard toolbar:
2. Click "Map task targets" in the report-specific toolbar:
   The following control elements are shown in the toolbar:



3. Select a polygon type from the list.
4. Click "Create polygon":



   The initial polygon is attached to the mouse pointer:



5. Set the starting point by left-clicking at the desired position in the layout.
6. With additional left clicks, set the vertices of the polygon until it encloses the required objects.

   Note

   The points of origin of the objects must be located within the polygon.

7. Complete the polygon by left-clicking on its start point.

Result

- The symbol of the polygon is displayed on the layout.
- If there are several possible target objects located within the polygon, the "Select targets" window opens. If there is precisely one possible target within the polygon, it is set automatically as the target object.
  See also chapter Selecting targets (Page 84).

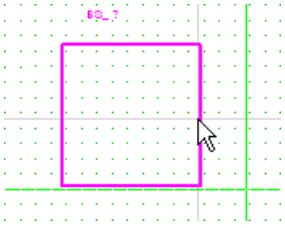
3.10.3 Editing the polygon shape

Requirement

- The layout is opened.
- A polygon has been drawn.

Moving polygons

1. Select the required polygon.
2. Hold down the <Ctrl> key.
3. Drag the polygon to the new position.

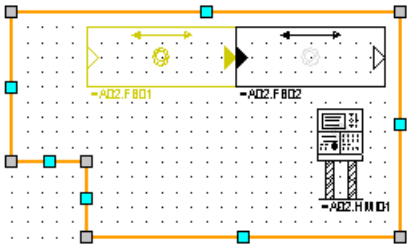
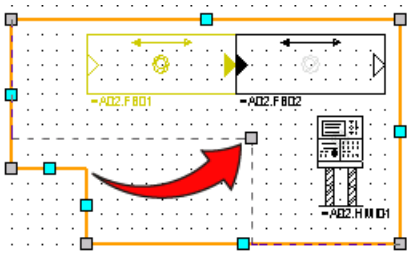
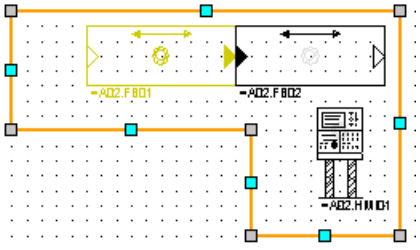


You will see a green and a gray cross. The green cross indicates the point of origin of the polygon movement. The gray cross is located at the tip of the mouse pointer. If the gray and the green crosses are congruent, the polygon is back at its initial position.

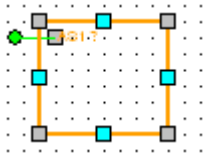
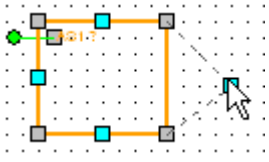
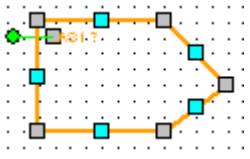
Adapting the polygon shape

You can adapt the shape and size of polygons at a later time. Proceed as follows:

1. Left-click twice on the polygon. Do not double-click.
   You will see the grab points:
   - The vertices have a gray grab point.
   - The mid points of the side lines have a turquoise grab point.
2. To change the length and shape of the lines, drag a gray grab point to a new position.
   The bordering lines are moved accordingly:

Initial situation	
Drag&drop movement	
Result	

3. To add a new vertex to the polygon, drag the turquoise grab point to a new position:

Initial situation	Drag&drop movement	Result
		

You can find more information on this topic in the "Platform Operation" manual, keyword "Editing graphical elements".

Transferring the polygon shape

You can transfer the shape of a polygon to another polygon. Proceed as follows:

1. Select a polygon.
2. Select the "Polygon shape > Memorize polygon shape" command in the context menu.
3. Select a different polygon.
4. Select the "Polygon shape > Use the memorized polygon shape" command in the context menu.

The second polygon has exactly the same polygon shape as the first polygon.

You can transfer polygon shapes across different levels. Example: Memorize the shape of a polygon for an HMI area and use it on a PLC area in the same report.

3.10.4 Replacing a polygon shape

Requirement

- The layout is opened.
- A polygon has been drawn.

Principle

If the subsequent editing of an existing polygon is too time-consuming using the grab points, draw a new polygon and adopt the properties of the existing polygon. The new polygon replaces the existing one.

Procedure

1. Select the polygon you want to replace.
2. Select the "Options > Redraw" command in the context menu.
   The existing polygon is grayed out. A new initial polygon is attached to the mouse pointer.
3. Draw a new polygon.
4. Delete the grayed out polygon.

3.10.5 Polygon properties

Polygon priority

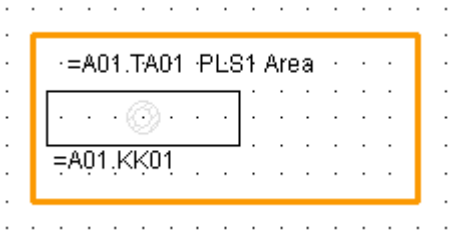
An object group can be assigned to several subunits using structure polygons.

The assignment to a "Subunit functions" polygon has higher priority than an assignment to a "Subunit general" polygon. The function group is located below the subunit functions.

How function groups belong to polygons

A function group belongs to a polygon in the following situations:

- When the point of origin of the function group symbol is located within the polygon:



- When the object group symbol lies within two overlapping polygons, the object group belongs to both polygons.
- If the object group symbol is located in two polygons and one encloses the other, the object group belongs to the inner polygon.

Report level

Polygons of the same type are saved in the same report level.

You can find more information on this topic in the "Platform Operation" manual, keyword "Layers in interactive reports".

3.11 User interface reference

3.11.1 User interface reference layout

Report-specific toolbar

Note

Showing/hiding

The additional buttons of the tools from the standard toolbar are superimposed on the report-specific toolbar.

To display the report-specific toolbar activate the "Identify" button in the standard toolbar:

Icon	Tool	Description
	"Map task targets"	Generates and configures an engineering task and defines the task target for one or more object groups.
	"Search task targets"	Allows you to search for object groups with a specific task target.
	"Calculate task targets"	Executes the tasks controlled by polygons and assigns the task targets to the object groups.

3.11.2 Context menu in the layout

Menu commands

In addition to the default commands, the context menu of the layout contains the following commands which are focus dependent:

Focus	Command	Description
Not set	"Calculate task targets"	Calculates the task targets for all polygons. As a result, it assigns the task target assigned to a polygon to all the objects within the polygon.
Object groups	"Calculate task targets (selected objects)"	Calculates the task targets for the selected object groups.
Polygons	"Calculate task targets (selected levels)"	Calculates the task targets for all polygons in the same level as the selected polygon.
Polygon	"Polygon shape > Remember polygon shape"	Places the shape of the selected polygon on the clipboard.
Other polygon in the same area	"Polygon shape > Use remembered polygon shape".	Applies the polygon shape from the clipboard to the selected polygon. The polygon shape of the selected polygon is converted into the memorized polygon shape.
No polygon in the focus	"Polygon shape > Create remembered polygon shape (original position)".	A new polygon is created from a cached polygon shape, whose position is according to the one of the original polygon.
No polygon in the focus	"Polygon shape > Create remembered polygon shape (under cursor)".	A polygon shape placed on the clipboard becomes a new polygon. The top left vertex of the new polygon is created below the mouse.

Additional information

You can find additional information on the default commands in the "Operation" manual, keyword "Context menu for interactive reports".

Electrical design

4

4.1 Creating or opening a circuit diagram

Requirement

The IC basic engineering has been performed.

Creating a circuit diagram

Circuit diagrams are included in the EI&C detail engineering. The circuit diagram is available at the sublocation or cabinet:

1. Select the
   "Locations" tab: "Plant > Building > Floor > Room > Cabinet"
   or
   "Locations" tab: "Location, two-level > Sublocation, without category"
2. Select "New > Circuit diagram IEC".

4.2 Controlling a circuit diagram

4.2.1 Controlling a construction representation

Requirement

A circuit diagram is opened.

Setting the scale

1. Click in the empty area of the report.
2. Select "Options > Scale".
   This command decreases or increases the representation of all the symbols on the report and adjusts the grid. The report itself and manually placed texts remain unchanged in size. All objects are shifted by changing the scale. If the scale is reduced, for example, all objects move upwards and to the left. This setting superimposes the according setting in the report template.

Set point of origin

1. Click in the empty area of the report.
2. Select "Options > Set point of origin".
   This command sets the relative point of origin of the report. When symbols or copy templates are placed, the placement is often relative to the point of origin of the report. Default for the point of origin: top left.

Controlling the representation of the connection hook

1. Click in the empty area of the report.
2. Select "Options > Connection hook".
   This command controls the representation of the connectors of two or more connections. A line is drawn on the connection point.

Controlling the representation of the direction arrow

1. Click in the empty area of the report.
2. Select "Options > Direction arrows visible".
   This command controls the display of arrows on the connection lines.

More information about default settings

The appearance and behavior of the circuit diagram is controlled by the option script of the report template. You can find additional information on this topic in the "EI&C Administration" manual, keyword "Controlling circuit diagrams".

4.2.2 Controlling transformation tools

Requirement

A report of the following drawing types is open:

- Circuit diagram
- Single line diagram
- Loop diagram
- Function diagram

Default setting in the report template

The default setting for the "Transform" menu is controlled in the report template. You can find additional information on this topic in the "EI&C Administration" manual, keyword "Default for control of transformation tool".

Controlling the transformation options

1. Click in the empty area of the report.
2. Select the "Options > Transform" command in the context menu.

   Note

   Limited function for circuit diagram, loop diagram and single line diagram

   The "Scale", "Distort" and "Mirror" options are not available in the reports of the type circuit diagram, loop diagram or single line diagram.

3. Activate or deactivate the options.
4. Optional: Select "Prevent all" to activate or deactivate all options in the "Transform" menu at the same time.

Effect

This function prevents the position or design of a symbol of a placed object from being changed. You can continue to edit the text of the symbol and connect the connectors.

The control of the transformation has an impact on the following tools:

- Report menu bar, "Transform" button
- Selecting an object on the report and editing the symbol graphic with the grabs.
- Context menu of an object, "Options > Mirror symbol <...>"
- Rotating an object with the space bar during insertion

Interaction between the default setting of the manual control

As soon as the first setting is made in the "Options > Transform" context menu, the default setting from the report template becomes invalid.

From this point forward, only the manual setting in the context menu of the report is evaluated.

4.2.3 Creating and editing path texts

Overview of path texts

Interactive reports of the EE/E&IC module often have an invisible checkerboard pattern. In this pattern, the columns are designated as "paths", the rows as "zones".

Creating path text

1. Create a text in a path on the interactive report, e.g. at the top border of the report.
2. Open the properties of the text.

3. Enter the text.
4. Activate the "Path text" option.
5. Click on the "OK" button to confirm your entries.
   A text frame is displayed.
6. Drag this frame to the required width. Make sure that the frame is not wider than the path itself. All objects that lie completely within the width of the frame obtain the path text.

Changing the appearance of path text

1. Select the "Properties" command in the context menu.
2. Activate the "Path text" option.
   The "Display" group is displayed.
3. Select one of the following options:
   - "Show texts unchanged"
     The text can extend beyond the text box if needed.
   - "Fit oversized texts"
     The size of the text font is adapted.
   - "Cut oversized texts"
     The text is cut off when needed and is only partially visible.
   - "Wrap too long texts"
     If necessary, additional text lines are generated.
   - "Adjust height automatically"

Result

The path text is only displayed for objects that did not have a description yet. An existing description is not overwritten.

You can find additional information on this topic in the "EI&C Administration" manual, keyword "Path texts".

4.3 Display variants of a circuit diagram

4.3.1 Line display on the report

The following explanations apply to reports with the options script entry `Application = "ELO"`.

The graphical properties of lines are edited like the graphical properties of connections. For more on this, see Options: Graphical properties of connections (Page 156).

You can find more information on this topic in the "Platform Operation" manual, keyword "Connections".

4.3.2 Status display on the report

Context menu in the Navigator, "Status display"

If the status display is activated in the Navigator, the following objects are colored accordingly on the report:

- Report objects
  These are placed objects.
- Potentials
- Cables or wires

Connections and purely graphical objects (objects that were drawn on the report and do not have an associated engineering object) are not detected by the status display.

Edit status

The following context menus are available if there is at least one placed engineering object (an RODevice) on the report in the current selection.

- Check status
  Select an object on the report.
  Select the "Check status" command in the context menu.
- Set status
  Select an object on the report.
  Select the "Set status" command in the context menu.

More information

You can find more information on this topic in the "Platform Operation" manual, keyword "Managing object status".

4.3.3 Working layer display on the report

Context menu in the Navigator, "Working layer display mode"

If you activate the working layer display in the Navigator, the objects are colored accordingly on the report.

More information

You can find more information on this topic in the "Platform Operation" manual, keyword "Working layer display mode".

4.3.4 Revision display on the report

Revision display of objects

Revision display is controlled via the project options and automatically activated when a revision is created for the circuit diagram.

More information

You can find more information on this topic in the "Platform Operation" manual, keywords "Visualization of the revision information" and "Display of revisioned reports".

4.3.5 Display of references and device labels

The display of references and device labels is controlled via the project properties.

See "EI&C Administration", keyword "Project properties".

4.4 Placing supplies and potentials

4.4.1 Placing supplies using a template

Requirement

A circuit diagram is opened.

Selecting the template for the supply

1. Click in the empty area of the report.
2. Select "Place template".
   The "Select document as template" window opens.
3. Enter the value "Units" in the top right field.
4. Select "Templates > @30 > M41 > B10 > A20 > A10 > A10 Power supply, 3-pole, with GRD".
5. Confirm with "OK".

The template will be positioned automatically. The engineering objects are created automatically.

4.4.2 Placing power supplies from the Navigator

Requirement

A circuit diagram is opened.

Selecting or creating power supplies

1. Select your potentials or supplies in the "Locations" tab.
2. Alternative in the COMOS standard model:
   – Select the subunit in which your circuit diagram is located.
   – Select "New > Potentials/power bars > L1/L2/L3".
3. Drag the potential/supply onto the report.

Selecting or creating terminals

1. Select your terminals in the "Locations" tab.
2. Alternative in the COMOS standard model:
   – Select the subunit in which your circuit diagram is located.
   – Select "New > Terminal-, plug strips, ground > Terminal strip, general".
   – Create terminals below the terminal strip.

Placing terminals

Select the required terminals with <CTRL> + mouse click and drag them onto the circuit diagram:

Connecting terminals with potentials

1. Select the "Connection" tool.
2. Select the desired number of connections.
3. Start the connection at the highest potential.

If a PE conductor is available, it is automatically detected. It is not necessary to first connect the electrically conductive potentials and then the PE conductor by itself.

Installing a fuse

1. Select a fuse in the "Locations" tab.
2. Alternative in the COMOS standard model:
   – Select the sublocation in which your circuit diagram is located.
   – Select "New > Devices > Protection devices > Fuse > <required fuse>".
3. Drag the fuse onto the connection, for example, between the terminals and the power rail.

The outer line turns yellow when a connectable position is found.

The cables are automatically disconnected, the connectors of the fuse are connected and additionally required engineering objects are created.

4.4.3 Using a supply with potential

If you use a supply with potential on a report, this has the following effects:

- The potentials have their own change marks in the revision management.
- All electrical components are connected to the potential by means of an electrical connection.
- The potentials on this report can be copied.
- The document analysis function, called via the "Check > Document" command in the context menu, counts the potentials.
- The potentials can be grouped on the report.

4.4.4 Potential chains / Master-Slave potentials / Potential references

There are two ways to organize potentials across reports:

- Chain
  Potentials are initially created as a chain. Thus all potentials are treated equally. The order of the chain is defined in the project parameters.
- Master / slave
  In this case all potentials relate to a master potential.

You can switch between these two options:

"Options" context menu ...

"Chain -> Master" / "Master->Chain"	Converts a chain into a master-slave relationship or back again as applicable. This option is only available for the master potential.
"Start of chain"	This potential is set as the start of the chain, the additional members of the chain are calculated automatically.
"Slave->Master"	Specifies this slave as the new master potential.

The two options determine where the "label" is initially output.

- Master: Label right
- Slave: Label left
- Chain: on both sides.

Differing from these defaults, the labels can be switched on or off at either side by means of the context menu:

"Options" context menu ...

"Label right"	Turns on the label at the right-hand end. The name is displayed if there is no label.
"Label left"	Turns on the label at the left-hand end. The name is displayed if there is no label.
"Invert arrow direction left"	Is the option active, the label on the left of the potential points to the inside. This shows the direction in which the components are supposed to be connected.
"Invert arrow direction right"	Is the option active, the label on the right of the potential points to the inside. This shows the direction in which the components are supposed to be connected.

"Options" context menu

Option	Description
"Connect in document order"	This option ensures that the potentials refer to one another according to the sheet order.
"Define order manually"	Use this option to open the "Form1" window, where you define the order of the potentials manually. Note: This option cannot be used with connection-dependent potentials.
"Sort in document order"	This option ensures that the potentials are sorted according to the order of the documents.

Sorting potential references at master/slave

The reference sequence is displayed unsorted at the master. If you change something and add a slave, for example, the references are sorted. The master context menu contains the "Options > Define order manually" and "Options > Sort in document order" commands for this purpose.

4.4.5 Assigning logical potential to a connection

Requirement

A circuit diagram is opened.

A supply or connection is drawn on the circuit diagram.

Note

Mixing of logical and physical potentials is not permitted.

Placing potentials on connections

To place a potential on a connection or supply, follow these steps:

1. In the toolbar of the report, activate the "Assign object" tool.
2. Use the "Assign object" tool to select the required connection on the report.
3. Drag&drop the potential from the Navigator to the connection in the report.

Disconnecting potentials on the report

After having dragged a potential to a connection on the report, the "Potential > Disconnect" command becomes available in the context menu. If you execute this command, the connection is maintained and the potential is removed from the connector in the database.

Multiple selection

You can select multiple connections in the report and assign a potential to all connections using the "Assign object" tool in the Navigator.

Potential vertical

In order to vertically display potential rails, select the potential rail on the report and then select the "Options > Vertical" command in the context menu. If the object is part of a group, all associated potential rails are shown vertically.

4.4.6 Transferring potential

Procedure

1. Select the source potential and then select "Options > Memorize" in the context menu.
   The engineering object associated with a potential is noted.
2. Select the target potential and then select "Options > Connect" in the context menu.
   The source potential is given a reference to the target potential.
   The target potential receives a reference to the source potential.
   The source potential is entered in the "implementation" field in the target potential.
   The target potential is entered in the "Request" field in the source potential.
   The target potential receives the name of the source potential.

This mode of operation is also allowed if the source potential already has an implementation to an additional source potential.

This mode of operation is also allowed for potentials within grayboxes.

Example

The source potential is a real potential and the target potential is a request potential.

Use the following base object for this:

"@30 > M41 > A50 > A30 > A10 > A40 > A40 Potential"

4.5 Placing components and devices

4.5.1 Placing stand-alone objects

Requirement

A circuit diagram is opened.

Placing stand-alone objects

Select one of the following options:

1. Placing components and devices from the Navigator
   - Select the object in the Navigator.
   - Use drag&drop to move it to the report.
2. Placing components and devices from the toolbar
   - Select the object in the toolbar.
   - Click on the report.
3. Placing components and devices using templates
   - Click in the empty area of the report.
   - Select "Place template".

Contact surface

See chapter Using the contact surface for relay and contactor (Page 123).

Result

The devices are placed on the report.

Placing manufacturer devices or requirements

It is not possible to place manufacturer devices or requirements in a report if the respective other one was already placed in the report.

4.5.2 Placing search text objects

Requirement

A circuit diagram is opened.

Placing a search text object

A search text object changes depending on other objects. There is no difference in use compared to stand-alone objects.

- Proceed as you would for placing a stand-alone object.
- Select the object.
- Select the "Settings > Search text" command in the context menu.

You can find more information on this topic in the "Platform Administration" manual, keyword "Using search text to define placeholder symbols".

Example

A search text object is placed on the report to serve as a template for a motor. "M1" is input for the search text, for example. As soon as the template is placed in the engineering project, for example, COMOS attempts to find a motor with the name "M1". The object that had been placed originally is replaced when a valid object is found.

It is also possible to enter a search text after a label.

Result

The devices are placed on the report.

4.5.3 Placing connection dependent objects

Requirement

A circuit diagram is opened.

Procedure

A connection dependent object changes depending on its connections.

Connection dependent objects change due to interdependency, for example, via the connectors. If you redefine a dependent object, this triggers a chain of dependencies. There is no difference in use compared to stand-alone objects.

Proceed as you would for placing a stand-alone object.

Connection dependent objects are displayed in blue on the interactive report.

Result

The devices are placed on the report.

4.5.4 Placing and editing a black box

Objective

A blackbox does not have a drawn symbol at the base object. Instead, a neutral symbol box is created on the report by a script. The symbol box features automatic contacts. Black boxes are used on reports to allow connection engineering to be performed before a concrete device is known.

You can find additional information on this topic in the "EI&C Administration" manual, keyword "Blackbox".

More complex placeholders are created with assemblies, referred to as grayboxes.

Placing a black box

Drag&drop can be used to create black boxes automatically in the "EI&C" module if:

- "Device" class, "Symbols" tab: No symbol exists for this drawing type.

If symbols exists for some drawing types and not for others, this object reacts like a black box on drawing types without symbols and as prepared beforehand within the symbol for other drawing types.

No preview of the objects is displayed during the drag&drop operation, since black boxes with automatic contacts do not have any symbols. Only a cross-hair is visible while the black box is being moved by drag&drop.

Automatically generated connectors

If you drag the base object onto the diagram, an engineering object with no connectors is generated.

The connectors are created when a connection is made. The connection can begin at any grid point of the border, but not at the corners. If you require additional connectors, you can drag the black box to make it larger until it covers more grid points.

- Names of automatically generated COMOS connectors:
  - Connector located on the left side:
    IX< No.>
  - Connector located on the right side:
    OX< No.>
  - Connector located above:
    IY< No.>
  - Connector located below:
    OY< No.>

Whereby <No.> is not a sequential counter, but is derived from the grid point. Example: OX10 is a connector on the right side of the black box that is located on the tenth grid point (without the grid points of the edge).

No COMOS connector is generated if a connection is open-ended.

Changing the representation of connectors

1. Select the black box.
2. Select the "Options > Edit symbol texts" command in the context menu.
3. Select the entry "eKP" in the "Symbol text label" list.
   The entry is only available if connector texts are visible.
   From a technical standpoint, this means that the settings of the connections were changed and the connections were therefore checked in.
4. Edit the fields in the "Font" group.
   The settings apply to the texts of all the connector points of this black box.

Features

You can open the properties of a connector that was generated automatically by right-clicking in the Navigator. If a label is entered there (as long as no implementation has been made yet), the label is also displayed at the black box.

Black box spanning sheets:

Black boxes may not span sheets.

4.5.5 Pasting objects via the clipboard

Purpose of the "Paste" context menu

The "Paste" context menu is available in interactive reports. This menu accesses the COMOS clipboard. Using the "Cut" or "Copy" context menu, objects are copied to the clipboard and then placed on any interactive report with the "Paste" command.

General use of the "Paste" context menu

You can find more information on this topic in the "Platform Operation" manual, keyword "Copying and cutting in interactive reports".

Special operation for the "EI&C" module

- Drawing type ET and FLUID: The "Paste" context menu places the engineering object parallel to the original.
- Exception: If the object is located next to the document and you copy across documents, the copy is not located parallel to the original but next to the new document.
- Drawing type FUP: The attribute "Y00A00192 Allow multiple placement" is evaluated additionally "Y00T00001.Y00A00192 Allow multiple placement".
  - The attribute is enabled:
    A second engineering object is created in the Navigator parallel to the first engineering object.
  - The attribute is disabled:
    The copy of the symbol generated in the report points to the same engineering object.
- Behavior when pasting copied connection objects:
  - If a connection line is copied and pasted again on the report together with its connected device, a new connection object is created in the database. The new connection object is given the same properties as the previously existing connection.
    Exception: The "Label" attribute in the properties of the connection object remains assigned at the original object and must be assigned manually for the new connection object.
- Behavior when pasting cut connection objects:
  - If a connected device with its connection line but no connection partner is cut and pasted again, this creates a half-open connection. This disconnects the wiring but preserves the properties of the connection object.
  - If only one connected device is cut out, the wiring is preserved. The resulting half-open connection at the connection partner shows a reference to the cut device.
    If a new half-open connection is then created at the cut device, a reference to the connection partner is displayed.
  - If only one connection is cut from the network, the wiring in the database is also disconnected.

4.5.6 Display of devices

Requirement

- A circuit diagram is opened.
- Devices have been placed.

Adapting the location of a device

To adapt the location of a device, double-click the object. The grab points appear. Use the grab points to rotate the object and move the texts.

Device label

The device label comprises the name of the device and the name of the owner.

To show or hide the device label for an object on the report, follow these steps:

1. Select the required object or several objects.
2. To show or hide the label, select the "Settings > Label visible" command in the context menu.

Graphical settings

1. Select a symbol and select "Graphical settings"
   This command is only available for objects which have a prepared additional symbol.
   Example: Terminals.
2. Select the graphical characteristics as desired:
   - Show/Hide an additional symbol
   - Rotate texts

4.6 Deleting elements of components or groupings

Structure of components

The following actions result in groupings on a report:

- Placing a component with associated elements.
- Manual grouping on the report by the user.

Deleting part of a group in the Navigator

If an object on the report is part of a group and is deleted in the Navigator, the following applies:

- The associated symbol is marked as inconsistent on the report. The symbol turns red.

You can delete the inconsistent elements from the report with a script. You can find additional information on this topic in the "EI&C Administration" manual, keyword "Deleting inconsistent elements".

4.7 Deleting inconsistent elements

Requirement

The inconsistent elements must be grouped on the report with the device.

Aim

Elements are an integral part of a device. If a user deletes an element placed on the report in the Navigator, the element is retained on the report, but it is displayed inconsistently.

All inconsistent elements, which belong to a device determined by a user, can be deleted on the report using script. If additional elements of the main object are placed, the remaining elements are moved up on the report to close any gaps.

Procedure

1. Open the object debugger.
2. Drag a device into field "A".
3. Drag the document that has elements to be deleted into field "B".
4. Use the following script:
```
Set GU = CreateObject('ComosRoUtilities.GlobalUtilities')
GU.GU_DeleteInconsistentGroupedReportObjects a, b, "M41_P2"
```
5. Activate the "Calculate" button.

4.8 Editing object properties and hierarchies on the report

4.8.1 Using the "Assign object" tool

Requirement

- A circuit diagram is opened.
- A device was placed on the circuit diagram.
- A further device was created in the navigator

Setting / exchanging device pointers

1. In the toolbar of the report, select the "Assign object" tool.
   The mouse pointer changes.
2. Select the device placed on the report.
   The selected object is colored.
3. Select the newly created device from the navigator.
4. Drag the device from the Navigator onto the object selected in the report.

Result

The reference to the database object was changed; this means the placed object references the newly created device in the navigator.

All connections and auxiliary connections already present on the originally placed device are transferred to the new device in the navigator.

Note

Behavior of connections and connection objects:

- The connection available at the original device is transferred to the new device. The associated connection object is moved from the original device to the new device.
- A collision takes place if a connection object is already available at the connection of the new device whose connection information deviates from the moved connection object. Resolve the collision manually in the detail window.
- If the connection of the new device is already connected to another connection, this connection is retained. If the same connection of the original device already has a connection to another connection, an auxiliary connection is created at the new device to keep the information of both the original and new devices.

Requirement

- A circuit diagram is opened.
- The hierarchy assignment is set in the options script of the report template (MoveObjectToLocation).
  You can find additional information on this topic in the "EI&C Administration" manual, keyword "Basic options for EE reports".

Setting the unit/location pointer or changing the hierarchy with the "Assign object" tool

1. In the toolbar of the report, select the "Assign object" tool.
   The mouse pointer changes.
2. Select an object on the report.
   The selected object is colored.
3. Select a unit or location in the Navigator.
4. Drag the object from the Navigator to the object selected in the report.

Result

- Option 1: Hierarchy assignment is activated
  The object selected on the report is moved below the new unit or location in the Navigator.
- Option 2: Hierarchy assignment is deactivated
  The unit or the location is set as pointer.

More information

You can find more information on this topic in the "Platform Operation" manual, keyword "Assigning objects".

4.8.2 Using interactive text

4.8.2.1 Purpose of the interactive text

Device labels as an image of the system and location structure

Device labels describe the location of an object in the data of the "Units" and "Locations" tabs.

Edit the device label on the report using interactive text. COMOS then changes the data structures until they correspond to the modified device labels.

Interactive text is available for the device labels of cables, potentials, and electronic components.

Every symbol text is checked if it is based on a changeable object property. If such a connection to the object data is found, the user entry is used to adapt the object data. Interactive text can result in the following, for example:

- A change of owner
- Generating a unit or location
- Moving an object
- Joining or separating potentials
- Changing names

4.8.2.2 Definitions and requirements for interactive text

Controlling the label using the project options

The device label is controlled as follows:

Properties of the project, "Options > Standard > Prefix and delimiter"

Interaction between the formal characters and interactive text

- "Prefix"
  The "Prefix" field determines the first character in the label depending on the object class or the object subclass.
  Example: If the prefix "=" is set for the "Unit" object class and if the top owner is a unit, the first character in the label is the equal sign.
  Prefixes must be entered in the properties of the project for the following object classes:
  - "Unit"
  - "Location"
- "Delimiter"
  The "Delimiter" field determines the character between two objects labels within the label depending on the object class or the object subclass.
  Note: If an object does not have an object label, the name is used.
  The properties of the project do not have to include delimiters. If no delimiters are entered, COMOS internally uses the dot as delimiter to separate the components of the label.

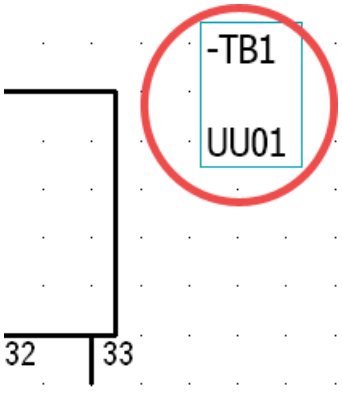
See also

Placing attribute texts and working with editable text (Page 116)

4.8.2.3 Notes on the use of interactive text

Working with interactive text

If you double-click a symbol text on an interactive report, a text box opens in which you can edit interactive text.



Basically the text after the delimiter is checked. Following applies:

Prefix unit (default =) and prefix location (default +): If text and data structure differ, the objects are generated or moved. If the unit as well as the location differ, one is generated and a reference is set at the other.

Prefix components (default -): If text and data structure differ, the objects are renamed or moved.

- Name, tag or a value can be changed directly by means of the report text.
- The user can trigger an owner change.
- A new unit or location can be generated. The separator for units and locations is a ".". If the dot is at the beginning, the search runs relative to the document or segment; otherwise, it is absolute.
  Examples:
  - "+L1.L2-F3 =>" Move the object, starting from the project root, to location "L1.L2" (might need to be generated) and rename it "F3".
  - "+.L1.L2-F3 =>" Move the object, relative to the document object, to location "L1.L2" (might need to be generated) and rename it "F3".
- Potentials with same labels are merged, otherwise they are removed from the string.
- Elements
  The following applies to the currently selected report object and all report objects from the same group:
  A new object is created if applicable. The elements are moved underneath the new object. The old object is deleted if it is not placed anywhere and no longer contains any elements.
- Interactive text is also available for device labeling texts if they have the "eB" label.
  With device labeling types, the label has to be divided by using "-". Even if labels are later returned as e.g. "-F1L1", you have to enter "-F1-L1".
  The entry is invalid if the device labeling type nesting depth is changed. Example: The current label is "-F1L1", "-F2-A1-L1" or even just "-L1" is entered. This action is canceled with a warning message.
  Only the object itself (lowest level) is moved to the new location; objects prior to this are copied as they might determine the names for subobjects.
  Example:
  - "-A1-B1-C1" is renamed to "-A2-B2-C2 =>"
  - "A1" and" B1" are copied and renamed to "A2" and "B2",
  - "C1" is later moved underneath "B2" and renamed to "C2".

4.8.2.4 Exceptions in the case of interactive text

Names with a dot

It is not possible to edit the device label by means of interactive text with objects of the Device system type if there is a dot in the device name.

For objects of the Device system type, it is not possible to enter a dot for the name using interactive text in the device label.

Background: The dot is used to structure potentials in interactive texts. For entry using interactive text, COMOS cannot differentiate between the name of a normal engineering object and a potential. For this reason, the dot is always interpreted as structural information for potentials.

4.9 Placing attribute texts and working with editable text

The properties of attributes can be displayed in a report.

If the attribute properties in the report are displayed and can be edited, then the following designation is used for them:

- Editable text
  You can find more information on this topic in the "P&ID Operation" manual, keyword "Editable texts on the report".
- InPlaceEdit

See also

Definitions and requirements for interactive text (Page 113)

4.10 Terminals, plugs and bridges on the construction display

4.10.1 Basic knowledge of terminals, plugs and bridges

System of connectors at terminals

You can find information about the connectors of the individual bridge types in the standard table "Y10 > M41 > A10 > Y10M41N00001 Bridge types" in the "Value 6" column.

Tests on connectors to be bridged

See also chapter Editing bridges in the tab (Page 234).

Checks for bridges in multideck terminals

Multideck terminals can only be bridged in a certain way.

The "Y00T0001.Y00A00186 Layer" attribute at the terminal levels of the multideck terminals is provided for this. If the levels differ, bridges cannot be made.

Editing bridges in the "Properties" tab

Bridges can also be edited as follows: Terminal strip properties, "Attributes > Strip" tab.

You can find additional information on this topic in chapter Editing bridges in the tab (Page 234).

Additional information

You can find additional information on this topic in the "EI&C Administration" manual, keyword "Base object for bridges".

4.10.2 Generating and editing bridges

Correlation between an internal connection and a bridge

An internal connection can be converted into a bridge.

Default case: Connecting terminals with an "internal connection"

1. Place two terminals on the report.
2. Connect both terminals with the connector tool.
   When you open the context menu of the connection between the two terminals, you can check the currently set bridge type:

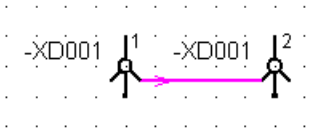
Switching between bridge types

1. Select the connection between the two terminals.
2. Select the "<current bridge type> > <desired type>" command in the context menu.

Switching "EC" and "ED" connectors to visible

Whether or not the additional "EC" and "ED" connectors are visible depends on the values of the respective bridge types in the "Bridge types" standard table.

Connectors ED visible:



Using bridges with connection objects

When you define a bridge type in the properties of the connection object, a bridge object is created. If you select the value "Not defined / Internal connection" from the "Bridge" list, any existing bridge object is deleted. If you change the bridge type, the old bridge object is deleted and a new bridge is created with the new type.

4.10.3 Automatic display of the label with terminals and plugs

Requirement

A report with the option "SymbolType = "M41_P2"" or "SymbolType = "M41_P8"" is open.

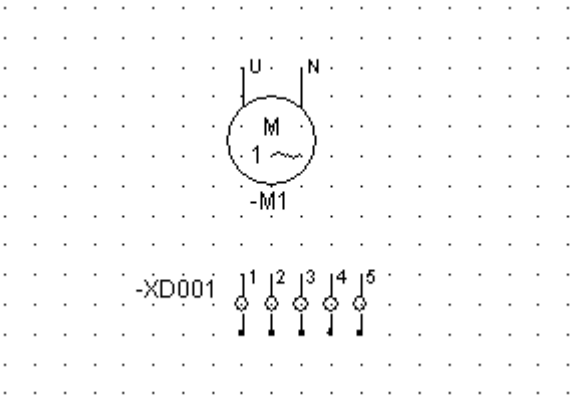
Conditions for an automatically shared label

A shared label is automatically displayed for terminals and plugs if:

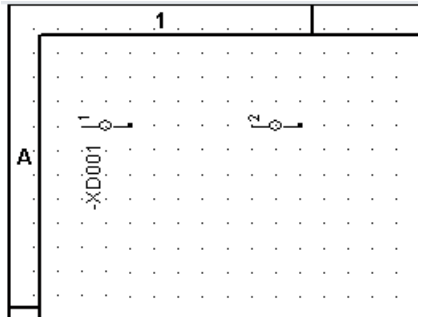
- The elements are located under a shared owner.
  This is the case if the terminals or plugs are part of the same strip.
- The terminals or plugs are located at the same height on the report.
- The terminals or plugs have the same angle on the report.
- The terminals or plugs have the same alignment (mirroring) on the report.
- The terminals or plugs are located next to each other on the report without interruption by a placed object.
  An interruption by a symbol is only permitted if the COMOS object associated with the symbol has been deleted in the Navigator.
  An interruption with graphical elements (line, circle) is permitted.

Display of the automatically shared label

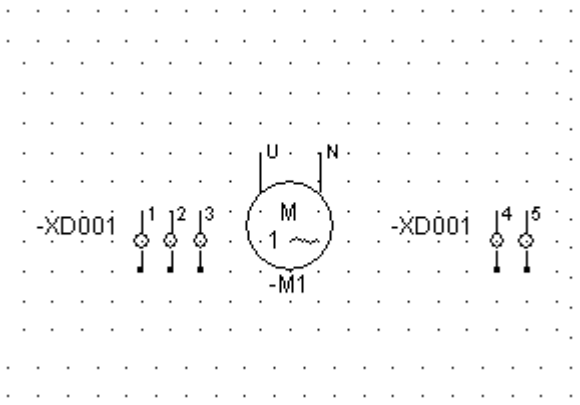
The label is displayed for all terminals or plugs of a shared display group at the left element:



This also applies to rotations and similar shared transformations:



If the terminals or plugs on the report are separated by the symbol of another engineering object, for example, the individual display groups each receive their own label:



See also

Controlling the display of the label (Page 119)

4.10.4 Controlling the display of the label

Refresh label

The report configuration determines when the automatic shared label is calculated. There are the following options:

- While you work in the report, COMOS automatically updates the display of the label.
  Requirement: The "RefreshDeviceLabel" option is activated in the report template. You can find additional information on this topic in the "EI&C Administration" manual, keyword "RefreshDeviceLabel".
- You update the display of the label manually.
  Select the command "Refresh device label" from the context menu of the report as required. The automatic shared label is not checked while you work in the report.

See also chapter Automatic display of the label with terminals and plugs (Page 117).

Controlling visibility of the label

1. Execute several single clicks:
   - The first single click selects the display group and marks an element in the display group as default. The default element is somewhat lighter in color.



   - The second and third single clicks select a terminal or a plug (a second single click is sufficient for the default element):



2. Select the "Settings > Label visible" command in the context menu.

4.10.5 Using bridge objects

Procedure

1. Connect two terminals on a circuit diagram.
2. Select an object in "@10 > A50 > A10 > A20 > A20 > A50 > A10!T00 Bridges".
3. Use drag&drop to drag the base object to the connection between the terminals.
   - The terminal connectors get a pointer to the bridge object.
   - Do not drag two bridge objects to one connection.

Additional information

You can find additional information on this topic in the "EI&C Administration" manual, keyword "Base object for bridges".

4.11 Connectors and connector information on the construction representation

4.11.1 Edit connector assignment

Definitions

COMOS connector: Connector object in the Navigator.

Symbol connector: Graphical connector on the report, derived from the symbol.

Edit connector assignment

This function replaces the functions "Switch connection descriptions", "Match connectors" and "Direction rotated".

The "Edit connector assignment" function changes the report and the connector objects in the Navigator. You have to have corresponding write rights.

1. Select an individual symbol on the circuit diagram.
2. Select the "Connector assignment > Edit" command in the context menu.
   The "Edit connector assignment" window opens.

   Note

   Do not edit bridges

   Note: You cannot edit the connector assignment of bridges. If you want to change the connector assignment, COMOS reports an error.

3. Compare the "Symbol" area with the "Assignment" group.
   In the "Symbol" area you see at which point of the symbol symbol connectors are defined.
   In the "Assignment" area you see a list of the symbol connectors and to which COMOS connector the symbol connector is assigned.

4. Switching labels and names:
   Deactivate the option "Switch only the label"
   - In order to assign a different COMOS connector to a symbol connector, select a different entry in the "Assignment" section of the list.
   - If the newly assigned COMOS connector was already assigned to another symbol connector, the new state of the symbol is inconsistent. Edit the other assignments until a consistent state has been reached.
   - If the COMOS connector was connected and a consistent state has been reached, the connections of the COMOS connectors are implemented. The graphical connection on the report remains unchanged, but the connections of the connector objects in the Navigator are changed.
5. Only switch label
   Activate the option "Switch only the label" The symbol script remains unchanged.
   In the "Assignment" group, the entries in the lists change.
   - In order to assign a different label to a symbol connector, select a different entry in the "Assignment" section of the list.
   - If the newly assigned label was already assigned to another symbol connector, the new state of the symbol is inconsistent. Edit the other assignments until a consistent state has been reached.
   - When a consistent state has been reached, the labels are implemented. To this purpose a new entry is assigned in the "Label" field to the connector objects in the Navigator.

The "Direction" of the connectors is not changed.

Editing multiple symbols simultaneously

When engineering objects have the same base object and symbol script, you can edit the representation of multiple objects simultaneously. To do so select multiple objects on the document and then proceed as you would for individual objects: Select the required command in the context menu and carry out the changes.

If you have already changed the representation and thus the symbol of an engineering object, you can use the commands "Memorize" and "Apply" to transfer the symbol script to one or more other object. You can find additional information on this topic in the "EI&C Administration" manual, keyword "Connector assignment in the symbol".

4.11.2 Memorize and apply connector assignment

Memorize and apply connector assignment

1. Select a symbol on the circuit diagram whose connector assignment you want to copy to other connectors.
2. Select "Connector assignment > Memorize" in the context menu.

3. Select an individual symbol or multiple symbols on the circuit diagram to which you want to apply the previously memorized connector assignment.
4. Select "Connector assignment > Apply" in the context menu.

Note

Multiselection only works for the selection of objects with the same connector assignment that is to be memorized. If the selected symbols have different connector assignments, the "Apply' function remains disabled in the context menu.

See also

Edit connector assignment (Page 121)

4.11.3 Using the contact surface for relay and contactor

Create contact surface

1. Create a relay or contactor in the Navigator:
   - In the context menu of the subunit, open "New > A10 Devices > A70 Relays/contactors".
   - Select an entry.
2. Create a new main contact or auxiliary contact:
   - Select the "New" command in the context menu of the relay or contactor.
   - Select an element.
     At least one device or element with the "Graphical type" attribute must be present in order for a contact surface to be displayed.
3. Drag the relay or the contactor onto the report.
   The contact surface is created together with the symbol of the component on the report.

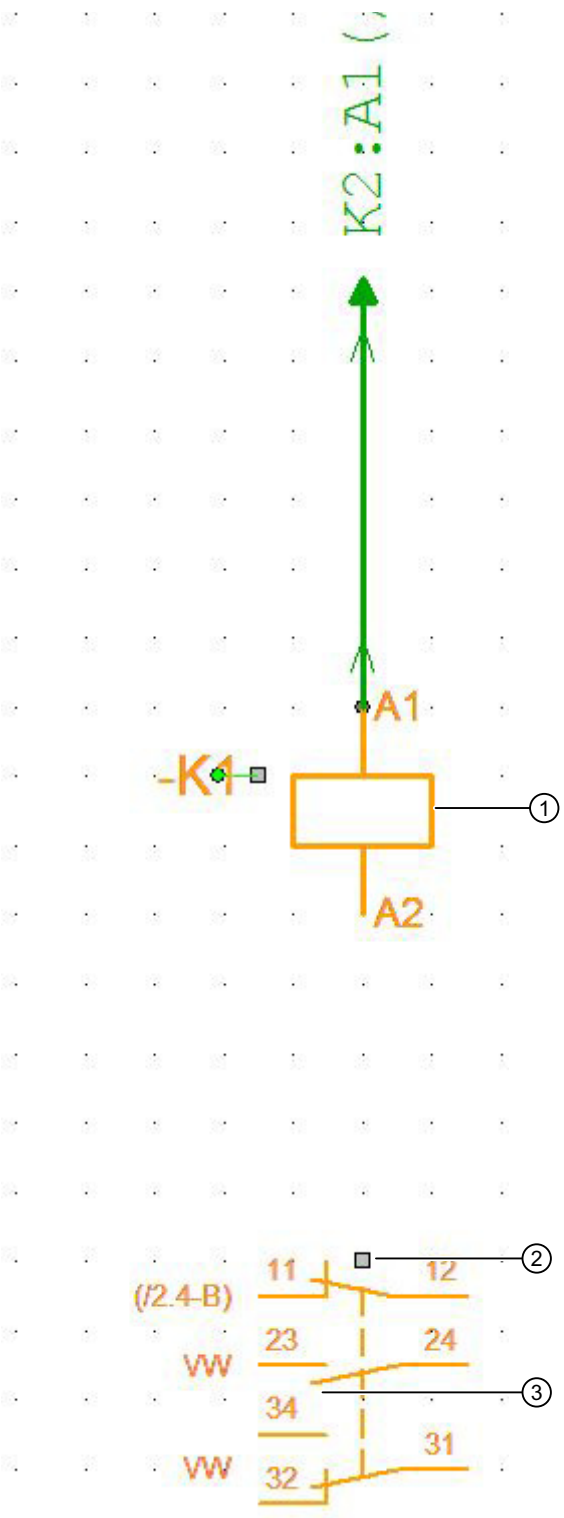
Moving contact surface with the component

1. Move the component.
   The contact surface is moved automatically.

Moving contact surface independently

If the properties of the project were prepared accordingly, the contact surface can be moved after the initial positioning.

1. To do this, click once on the relay or contactor object.
   The relay or contactor is selected together with the contact surface.
2. Click the contact surface once.
   The contact surface receives grab points, which include a rectangular point at the top end of the contact surface.


K2:A1
A1
-K1
1
A2
2
11
12
(/2.4-B)
23
24
VW
3
34
31
VW
32

①	Relay/contactor
②	Grab point of the contact surface
③	Contact surface of relay/contactor

Drag this grab point to the new position.

Set the sorting of the connections at the contact surface and the contact bridge

By default, all objects in the contact surface or a contact bridge are displayed on the circuit diagram sorted into groups:

- Devices with the "Graphical type" attribute
- Elements
  - Main contacts
  - Auxiliary contacts
  - Auxiliary switch blocks

The available connectors are sorted by label within these groups.

If you want to undo this predefined sorting, proceed as follows:

1. Open the properties of the contactor or relay.
2. Go to the "Attributes > System data" tab.
3. In the object settings, select "Suppress sorting."

Now the connectors are displayed sorted according to the order and nesting in the navigator.

Display

The numbers and letters have the following meaning:

(046.2)	Describes the sheet name and the path number and relates to the object that is connected at this connector. The zone number is not specified, so this form of notation states the column in which the object is to be found. In the example on the left, the object is thus on sheet 46, path 2.
15, 16, etc.	The symbolized connector. The number is derived from the designation of the connector below the NC contact or NO contact.
"VW"	An empty (unset) reference. The text that is displayed for an empty reference is specified in the project properties: "EE/I&C reference layout" tab, "Text for not solved references" field.

Action line

The action line on the contact surface is calculated and is not part of the symbols.

The display of the action line is in the style of EN 60617-2. You can turn off the action lines so that they are not drawn. You can find additional information on the display of the action line in the "EI&C Administration" manual, keyword "Contact surface for relays and contactors".

Additional information

You can find additional information on this topic in the "EI&C Administration" manual, keyword "Contact surface for relays and contactors".

4.11.4 Output connector description of user-generated symbols

Requirement

The object, such as a terminal, is placed on a circuit diagram.

Preparing the symbol of the base object

1. Open the symbol.
   The "Symbol Editor" tab opens.
2. Select the text parameter you want to output, for example, "%N I%".
3. Right-click on the text and select the "Properties" command in the context menu.
   The properties of the text parameter open.
4. Activate the "Label visible" option.
5. Click "OK" to apply the change and close the text parameter properties.

Entering the description of the engineering object

1. Navigate to the engineering object of the terminal.
2. Expand the object structure in the Navigator until you see connector "I".
3. Right-click on the connector and select the "Properties" command in the context menu.
   The properties of the connector open.
4. Enter the required description in the "Description" field.
5. Click "OK" to apply the change and close the connector properties.

4.12 Connections on the construction display

4.12.1 Connection objects

Electrotechnical connections are either represented purely graphically, or created as a connection object. The control is performed using a project option. The project option has been available since COMOS version 10.2.1 and enables electrotechnical connections to be created as a connection object. This means that a graphical or database connection can now be handled as an object.

This option is available for the circuit diagram and the wiring diagram.

Requirement

The connection mode "Create connection objects for connector types "E" and "I" (from 10.2 SP1)" has been set in the project options.

Connection objects

- If you draw a connection on the report, a new connection object is automatically created in the Navigator. This object contains all relevant information for connection.
- A connection object is also created even for half-open connections.
- If you work with reading connections, existing connections are represented as a connection line.
- When you change a connection from "writing" to "reading", the connection object in the Navigator is retained. The connection information from the database now has priority over the graphical information.

Definitions

- When two objects are connected, the connection object is created at the object defined as the "From" connection.
- If the connection is deleted "graphically with object" or in the Navigator, the connection object with its information is also deleted.
  When deleting an object the underlying connection information is also deleted.
  By deleting a connection graphically, the connection object and the data projected with it are preserved.
- If a connection is released graphically on the report at the owner of the connection object, the connection object is transferred to its connection partner.
- If an object is connected to another object, the connection information is transferred to the connection partner when the object with the definition "From" is deleted. In this way, data that have already been configured is retained for further processing.
- If, during rewiring, a new connection is made to a connection that already has a connection object with different information, a collision occurs that must be resolved manually in the detail window "Collision control for connection objects" (Page 146).
  A collision can be recognized by the orange color of the connection line on the report.
  Alternatively, collisions can also be resolved in bulk using the plugin "Collision control for connection objects" (Page 147).
- If values for the connection information are entered on the document, these are displayed to the created connection objects as default values.
- If a half-open connection is released graphically on the report at the owner of the connection object, the connection object is transferred to its connection partner. In addition, the original owner receives a new connection object, with the previously defined properties, except for the unique properties such as label and wire.

- If an object is placed on an existing connection line, the connection is divided into two new connection lines. Both connection lines receive their own connection object, with the original connection object remaining at the object defined with "From". The new connection object is given the same properties as the previously existing connection.
  Exceptions:
  - The "Label" attribute in the properties of the connection object remains assigned at the original object and must be assigned manually for the new connection object.
  - If another object is placed on an existing read connection, the connection lines are displayed inconsistently. The database information of the connection is not automatically adapted.
- If a connection line is copied and pasted again on the report together with its connected device, a new connection object is created in the database. The new connection object is given the same properties as the previously existing connection.
  Exception: The "Label" attribute in the properties of the connection object remains assigned at the original object and must be assigned manually for the new connection object.
- If a connected device with its connection line but no connection partner is cut and pasted again, this creates a half-open connection. This disconnects the wiring but preserves the properties of the connection object.
- If only one connected device is cut out, the wiring is preserved. The resulting half-open connection at the connection partner shows a reference to the cut device.
  If a new half-open connection is then created at the cut device, a reference to the connection partner is displayed.
- If only one connection is cut from the network, the wiring in the database is also disconnected.

More information

The "Migrate connections" tool allows you to transform connections in templates into connection objects and therefore to create a standardized, consistent project.

See also: Transforming graphical connection lines into connection objects

See also

Surface reference of the "Graphical properties" window when using connection objects (Page 157)

Edit graphical properties of connections with connection object (Page 157)

Object-based editing of connections in the database: (Page 193)

4.12.2 Creating connections

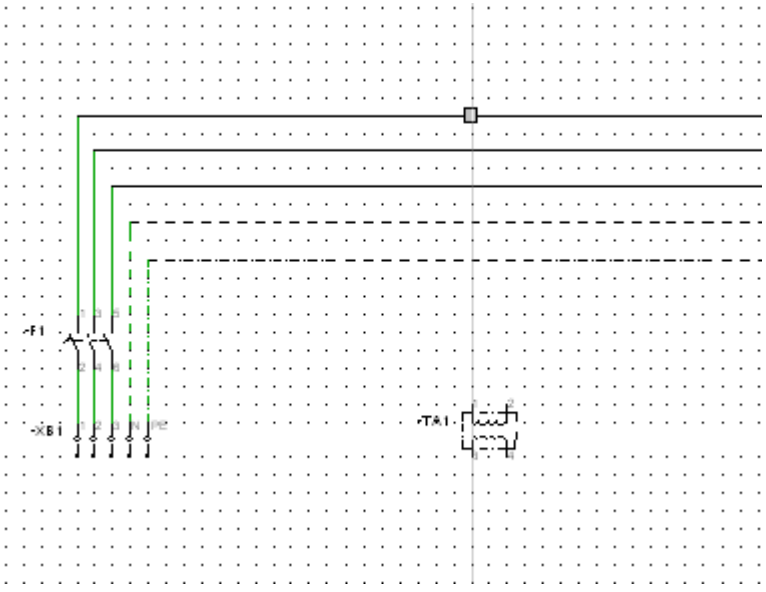
Requirement

- A circuit diagram is opened.
- The "Connection" tool is activated.

Crossing connections

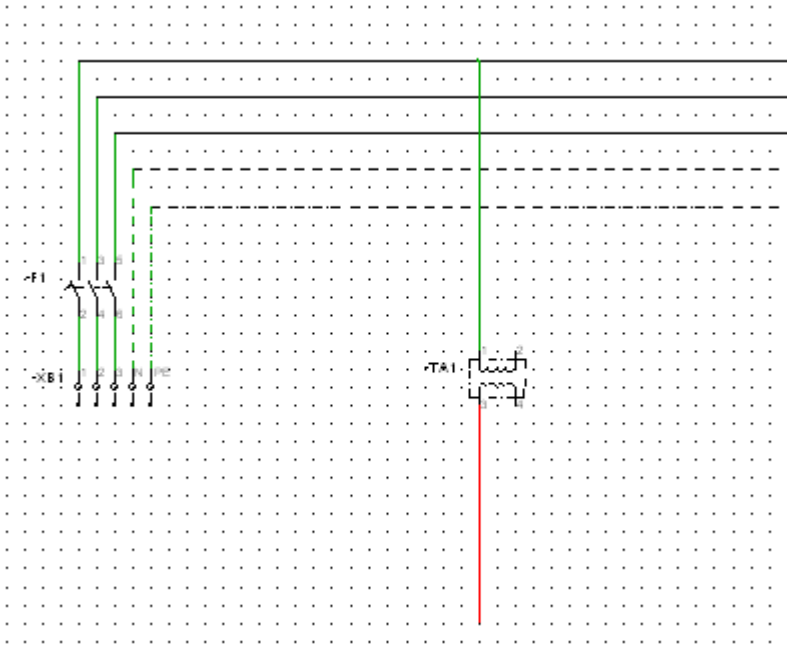
Connection lines can also be beyond components boundaries.

1. Drag a connection starting at the potential (the starting point of the connection is visible at the top as a gray box) across a transformer.
   The mouse pointer is shown as a cross below. Clearly visible: The line is still solid at the transformer because the connection is currently drawn across the component:



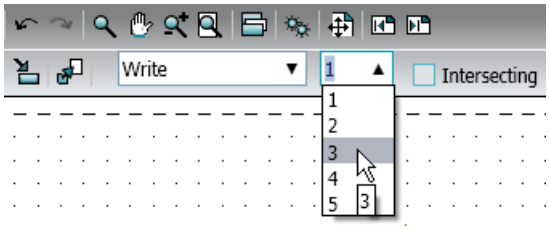
2. Left-click to confirm the next point.
   You can now set additional intermediate points and confirm each with the left mouse button.
3. You close the connection function with a right-click.

As a result, the connection is interrupted and the transformer is adjusted and connected. Important to note: The line across the transformer is now no longer solid because the component has automatically been modified at the connection:

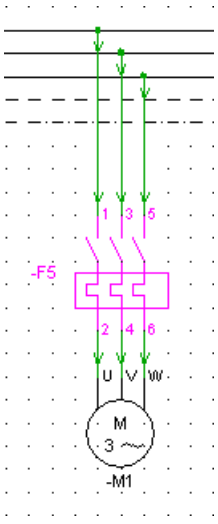

-F1
-XB1
-TA1

Multi-pole connections

1. Select the three-pole connection in the Connector tool:



2. Then, draw a connection from the supply via the protective switch up to the motor. The result is that the connections are interrupted, the components are adjusted and all connectors are joined correctly:



Result

1-wire connection

Terminals can be configured as a 1-wire terminal. In this case, only one component may be connected to the terminal connection. If you attempt to establish a second connection to the connector, the connection will be inconsistent. The analysis on the report outputs: "Error number 42: No connector at end point".

4.12.3 Selecting the "Connection" tool

Requirement

A circuit diagram is opened.

"Connection" tool

The "Connection" tool is located on the menu bar of the report. This tool is used to create connections between objects (components).



Result

The tool is selected. The mouse pointer changes into a cross-hair.

4.12.4 Function of the "Connection" tool

Requirement

- A circuit diagram is opened.
- The "Connection" tool is activated.

Functions

The "Connection" tool has the following functions:

- Magnetic connecting dots
  The "Connection" tool automatically links to the connector within a certain radius around the connecting dot of symbols. To switch off the function which automatically links a cable to connectors, hold the <Ctrl> key down while moving the mouse.
- Magnetic grid points
  The "Connection" tool links to a grid point automatically. To deactivate the function which automatically links a cable to grid points, hold the <Shift> key down while moving the mouse.
- The <Ctrl> key and the <Shift> key can be pressed simultaneously so that this automatic latching of the cursor to the grid can be completely deactivated.
- You can also drag connections beyond component boundaries. COMOS separates the connections and adapts the object in the connections.
- If you delete an object from the document, which is connected on both sides, the connections are closed.
- If you set an object on connections, the connections are separated. The object is adjusted to the connections.
- The connections are changed accordingly when you move connected components.
- Connection crosses on connecting lines
  Two other connections can also end at the same point when joining connectors. A connection crossing is displayed to distinguish such a case from a situation where a connection runs over another connection but in a purely visual form.

Automatic connections without the "Connection" tool

If two connectors overlay each other on a report, a connection object is created automatically and thus a database connection generated.

You can use the "ConnectDirectly" (Y00T00001_Y00A04701) "Connect directly" attribute on the "System data" tab to define for which documents connections are to be connected directly:

Value of the "Connect directly" attribute	Relevant document
0	Auto connect of two connections is disabled for all documents.
1	Auto connect is only enabled on circuit diagrams.
2	Auto connect is only enabled on wiring schematics.
3	Auto connect is enabled on circuit diagrams and on wiring schematics.

If you have selected "Read" mode for the connection, the components are not connected to each other directly.

See also

Connection objects (Page 127)

4.12.5 Type check during connection

Types and subtypes of connectors

Each connector has the following properties:

- Properties of the connector
- "Type" list
- "Subtype" list

Permitted combinations when connecting the types "ET/I&C" and "FLUID"

If connectors are connected on the report, the types and subtypes are checked automatically. You may only connect connectors of the same type. Each type has different subtypes. The combinations permitted for the subtypes depend on the type.

The following combinations are permitted for the "ET/I&C" type:

- The same type and the same subtype on both sides.
- The same type on either side and one of the two connectors has the subtype "None".

Permitted combinations when connecting the "Signal" type

You can find additional information on this topic in the manual "Logical Administration", keyword "Combinations permitted by data hierarchy".

Property not checked

The "I/O" property is not checked automatically when the connectors are connected.

Effect on the interactive report

Connectors with an invalid combination cannot be connected on the report.

If an inadmissible connection between two connectors is established through actions outside the report, the connection becomes inconsistent. See also chapter Display of consistency and inconsistency (Page 143).

4.12.6 Connection information

4.12.6.1 Entering connection information

Requirement

A circuit diagram is opened.

Entering connection information

1. Select the connection.
   The connection must be connected on at least one side.
2. Select the "Settings > Connection information" command in the context menu.
   If you work with connection objects, open the properties of the connection object.
3. Enter the required values for the attributes and confirm your entries.

Display connection info

There are two ways of displaying connection information.

Either via the base object:

1. Drag the following base object onto the connection:
   "@10 > A50 > A20 > B40 > A40 Connection information"
2. Select the placed connection information and confirm your entries.

Or via the icon in the menu bar:

1. Click the "Connection information" button in the toolbar of the report:
   
2. Select the connection on the report on which you want to display the connection information.

You can find additional information on the topic of "multi-pole connections" in the "EI&C Administration" manual, keyword "Connection information for parallel connections".

4.12.6.2 Changing the display of the connection information for a report

Requirement

- The IC detail engineering has been performed.
- The connection mode "Create connection objects for the connector types "E" and "I" mode (as of 10.2 SP1)" has been set (see Change connection mode).
- A circuit diagram was opened.

Procedure

1. Right-click the report to open the options in the context menu; there you select "Display connection information".
2. Select the information you want to display at all connection lines on the report and confirm with OK.

Fact

As soon as the first setting is made in the report, the default setting from the report template becomes invalid. From this point onward, the manual setting of the report is evaluated. The default settings of the report template can be restored by clicking "Reset".

see also

Changing the display of the connection information for individual connections. (Page 138)

4.12.6.3 Transfer connection information via implementation

Requirement

- The connection mode "Create connection objects for the connector types "E" and "I" mode" (as of 10.2 SP1) has been set.
- In the project settings, in the implementation area, the option "Object will not be replaced (by merge)" or "Object will be replaced (by merge)" was selected.
  Also see the section: " Settings control group"

Set implementation

1. Open the properties of a device.
2. Define the implementation pointer of the device via drag&drop in the "Implementation" field. Observe the following behavior of the engineering objects and their connections during the implementation.

Entry	Behavior of engineering objects	Behavior of connections and connection objects
"Object will not be replaced (by merge)"	The requirement objects in your engineering project are retained. An implementation object is assigned to the requirement objects by means of a reference and you can navigate between the two objects.	• The connection available at the requirement is transferred to the implementation. The corresponding connection object is moved from the requirement to the implementation. • A collision takes place if a connection object is already available at the connection of the implementation whose connection information deviates from the moved connection object. Resolve the collision manually in the detail window: Collision check (Page 145) • If the connection of the implementation object is already connected to another connection, this connection remains. If the same connection of the requirement object already has a connection to another connection, or has a connection object with connection information, an auxiliary connection is created for it on the implementation object to keep the information of both the requirement and implementation.
"Object will be replaced (by merge)"	The requirement objects in your engineering project are replaced by the implementation objects.	

Resetting implementation

1. Open the properties of the implemented object.
2. In the toolbar select the "Back to the original object" button.
   Prior to COMOS 10.6: Select the "Back to device request" button in the toolbar.
   You can find more information on this topic in "Platform Operation", keyword "Requirement, manufacturer device and base object change".
   Result: The implementation was reset. This means that the transferred connection information is located again at the requirement object and the original connection objects are located again at their engineering objects.

4.12.6.4 Changing the display of connection information for individual connections

Requirement

- The IC detail engineering has been performed.
- The connection mode "Create connection objects for connector types "E" and "I" (as of 10.2 SP1)" has been set (see Change connection mode).
- A circuit diagram was opened.
- Connections were created on the report.

Procedure

1. Select a connection line on the report.
2. Double-click on the connecting line to open the "Properties". The "Connection properties" tab opens.
3. Make the required settings and confirm with "Apply".
4. Click "Display connection information" to specify which information is to be displayed on this connection line on the report. By default the setting of the report template is applied.
5. Select the information you want to display on this connection line on the report and confirm with OK.
6. Close the tab with OK.

Fact

As soon as the first setting is made in the report, the default setting from the report template becomes invalid. From this point onward, the manual setting of the report is evaluated. The default settings of the report template can be restored by clicking "Reset".

4.12.6.5 Resetting connection information

You have two options to reset the connection information of a connection to the default values of the report.

Note

Note: Only the connection information that differs from the connection information of the report is displayed at the connection line.

This information is displayed in the footer of the report.

Requirement

You are familiar with Entering connection information (Page 135).

Resetting connection information at the connection

1. Select one or more connections.
   The connection must be connected on at least one side.
2. Select the "Settings > Reset connection information" command in the context menu.

4.12.7 Connecting half-open connections

You have several options to connect unconnected, half-open connections on a report or even across reports.

Requirement

- One or more circuit diagrams are open.
- Two half-open connections have been drawn, which means they are not connected at both ends on the report.

Memorizing and connecting half-open connection

1. Select a half-open connection on the report.
2. Select the "Connection > Memorize" command from the context menu.
3. Select another half-open connection on the same or a different report.
4. Select the "Connection > Connect with " command from the context menu.
   The associated connections are interconnected in the database and displayed as a reference at the half-open connections.
   Note: If one of the two half-open connections is already connected, an auxiliary connection is created in the database for the new connection.

Selecting and connecting half-open connections

1. Select a half-open connection on the report.
2. Select another half-open connection on the same or a different report.
3. Select the "Connection > Connect selected" command from the context menu.
   The associated connections are interconnected in the database and displayed as a reference at the half-open connections.
   Note: If one of the two half-open connections is already connected, the "Half-open connection" dialog opens.
   Select whether the existing connection should be disconnected and reconnected, or whether an auxiliary connection should be created in the database for the new connection.

Half-open connections at branches

If a branch has two or more half-open connections, it is permissible to reference different devices. To do this, you use the commands "Connection > Memorize" and "Connection > Connect" in sequence as usual.

Half-open connections to the same device

Different connectors of the same device may be connected to one another.

Application example: Graphical simplification of complex components whose symbol takes up a lot of space on the report. In this case it is recommended to work on connections with half-open connections You set the references between the half-open connections graphically as references via the context menu with the "Memorize" and "Connect with" commands.

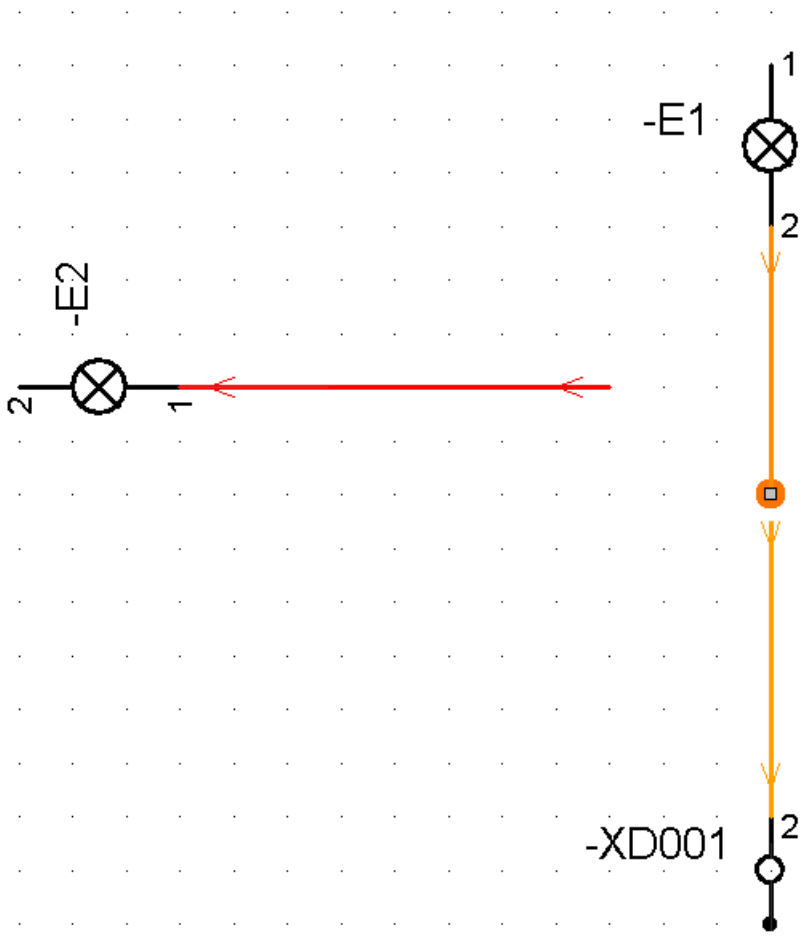
4.12.8 Merging half-open connections

Requirement

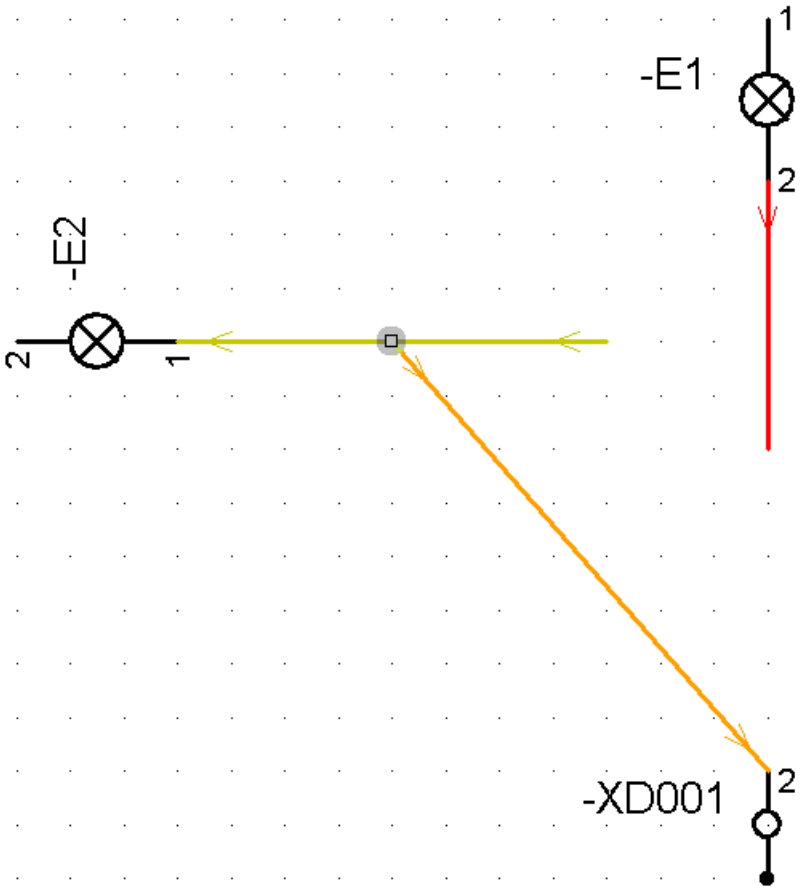
- A circuit diagram is opened.
- Two connections have been drawn and are not connected at both ends on the report.
- The two connections are located on the same report.
- Both half-open connections have a connection object in the Navigator.

Manually connecting half-open connections

1. Drag the open end point of the first connection to the open end point of the second connection.
2. If the connection touches another half-open connection at the end, the connection and the connection point is colored dark orange:



   Once the action is complete, the two connections become one connection.
3. If the connection touches another half-open connection somewhere in the internal area, the connection turns light green and the connection dot turns gray:



Once the action is complete, there are two connections on the report connected by a connection dot.

Connecting half-open connections using wire assignment

When you place an existing wire on the two ends of a half-open connection, they are automatically connected.

This connection remains intact even when you remove the placed wire again.

If you want to disconnect the connection again, select "Disconnect connection" in the context menu of the connection.

Note:

As long as the attribute "Y00T00001.Y00A00192" "Allow multiple placement" at cable and wire has the value "1" it is automatically recognized that two connections have the same wire information and they are connected to each other again.

If you want to prevent the automatic connection, set the value to "0" at the attribute "Y00T00001.Y00A00192" "Allow multiple placement" respectively.

Half-open connections in an assembly (graybox)

If an assembly with half-open connections is placed, the following applies:

A connection from the main document cannot be connected to the end of the placed half-open connection of the assembly. The connection becomes inconsistent.

4.12.9 Using consistency/inconsistency information

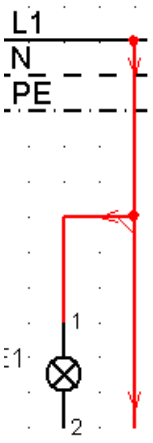
4.12.9.1 Display of consistency and inconsistency

Requirement

- A circuit diagram is opened.
- A connection is drawn and not connected at both ends on the report.

Inconsistency display with open or half-open connections

Connections open at one end or two ends are not allowed. They are marked as "inconsistent" and displayed in red:



Exceptions:

- Reference
  If open connections have a reference, then they are considered "Consistent" in the report.
- Connection manually marked as consistent

Inconsistency display in case of inadmissible types or subtypes

If an inadmissible connection between two connectors is established through actions outside the report, the connection is marked as "inconsistent" and displayed in red.

The inconsistency display takes precedence over other graphical changes to the connection. See also chapter Lines in the electrical engineering reports (Page 156).

See also

Always show half-open connections as consistent (Page 144)

4.12.9.2 Always show half-open connections as consistent

Requirement

- A circuit diagram is opened.
- A connection is drawn and not connected at both ends on the report.

Always show half-open connections as consistent

If assembly templates are created, for example, a half-open connection may be intentional and therefore logically consistent. In this case, the inconsistency display for this connection can be deactivated manually.

1. Select the connection.
2. Select the "Connection > Always show as consistent" command in the context menu. A check mark appears in front of the "Always show as consistent" command.

To activate the inconsistency display once again, proceed as follows:

1. Select the connection.
2. Select the "Connection > Always show as consistent" command in the context menu. The check mark in front of the "Always show as consistent" command disappears.

The connection end object

If the database is prepared accordingly, the following applies:

- If the option "Connection > Always show as consistent" is activated, a connection end object is displayed at the end.
- A connection can still be connected to the open end of a connection despite the connection end object.

Connecting a half-open connection

If a half-open connection is connected, the option "Always show as consistent" is reset.

If the connection becomes inconsistent again, for example, if the component or the reference is deleted, the following applies: The option "Always show as consistent" must be set once again manually.

4.12.9.3 Collision check

Collision control for connection objects

Background

When connecting connections that already have connection information (such as the closing of two half-open connections), you can decide which information the new connection should retain. Some collisions are resolved automatically in COMOS using stored rules; some collisions cannot be resolved automatically. In this case, both items of information are compared, and you are informed that you must make a decision.

When a corresponding collision occurs, the information of the source connection is prioritized initially. The connection line is colored orange in the document and a message is displayed in the detail section of the user interface. You can edit the collision directly or edit it at a later point in time in bulk processing.

See also chapter Resolve collisions in the "Collision check for connection objects" plugin (Page 147).

Note

Saving per session

Note that the user interface only saves unresolved collisions at connections per COMOS session in the list view. However, the status of the connection is retained for further sessions. This allows a test run by using the "Collision check for connection objects" window at any time in the future.

Check

A collision check of connected objects takes place in the following cases:

- Moving grab points
  You move a grab point from one object to another. The connection to the old object is deleted and a new connection to the new object is created.
- "Memorize" and "Connect with"
  This is an alternative way to create a new connection. Connecting lines and reference brackets are included.
- Close connection
  Example: If three objects are connected with each other and you delete the middle object, the connection between the two remaining objects is closed. If there are connection objects on both sides, a collision occurs. If the connection object belongs to the deleted object, no collision occurs.

Rules of the collision check

If values were already set by the system, the resulting collision is automatically resolved:

The connection object saves values that were set either using bulk processing of the attributes or manually in the user interface. If this connection object is connected to a connection object without values, the manually set values are applied.

Note: The connection properties of the document are not transferred to the connection object but are merely displayed there.
Because the collision check only compares values saved at the connection object, these default values do not trigger a collision.

Properties of the connection object locked

As long as there are unresolved collisions, the properties of the connected objects listed are locked for editing.

Use the "Collision check for connection objects" detail window

When a collision occurs when connecting two objects on the circuit diagram, the "Collision check for connection objects" window opens automatically in the detail area of the Navigator. The window displays all collisions of the active COMOS session. You can resolve collisions directly and decide which values are to be applied for the connection.

Resolving collisions for connection objects

1. Expand the displayed collisions:
   You see a table with all different properties of the two objects that require a decision by the user and the name of the associated connector in the header.
2. Click the following button to display all associated objects to which you can navigate: 
3. Only select values in the table that you want to apply for the connection. If multiple values are different, you can either select each value individually or select all values of an object with "Select all".
4. Click "Resolve collision".
   Note: The "Resolve collision" button remains grayed out until you have selected a value for each property.

Note

Note: As long as there are unresolved collisions, the properties of the connected objects listed are locked for editing.

Result

The collision was resolved, and a consistent connection was created.

Reference

To work in a larger view and to view all collisions within a project, click the "Extended" button.

The "Collision check for connection objects" plugin opens in the working area of COMOS. Here can set a start object and thereby limit the quantity.

See also

Resolve collisions in the "Collision check for connection objects" plugin (Page 147)

Resolve collisions in the "Collision check for connection objects" plugin

The "Collision check for connection objects" plugin offers you an overview of all connection collisions of your project below a selected starting node. Collisions can be resolved individually or in bulk.

Resolving collisions

1. Open the plugin: "Collision check for connection objects"
   You have two options for opening the plugin:
   Via the menu: Plugins > Automation > "Collision check for connection objects"
   In the detail window below the Navigator area:
   See also Use the "Collision check for connection objects" detail window (Page 146).
2. Use Drag&Drop to set a node from your location or unit structure as start object below which the connection objects are checked for collisions.
   All collisions are displayed.
3. Expand the collision to compare the connection objects involved and their properties:
   Right-click on the row > Context menu > Hierarchy level > Expand.
4. Check the fields highlighted in orange: Orange means that these properties are different, and that no consistent connection can be created.
   The following navigation options are available:
   - To the object in the Navigator
   - To the connection in the circuit diagram

   Note

   Note: As long as there are unresolved collisions, the properties of the connected objects listed are locked for editing.
5. Decide for all displayed collisions which values and properties are to be applied for the connection:
   You can select either a single value or an entire row.
   Right-click the selected value to open the context menu.
   Click "Apply all values" - the entire row is applied for the connection object.
   Or: Click "Apply this value" - only this value is applied for the connection object.
   As soon as a collision is successfully resolved, it disappears from the current view.
6. Click "Apply" to save your changes.

Reference

You can customize the view of the "Collision check for connection objects" plugin.

To do so, navigate to the query “@99|A90|Y00R00070 Hardcoding object" in your base project SO1, and edit the entry "A00248".

4.12.10 Grounding

4.12.10.1 Setting grounding

We distinguish between protective grounding and functional grounding.

You can specify grounding at the connector of an engineering object or its base object.

If you make no change, the default setting is protective grounding.

The grounding of a cable is defined using the subtype of the connector and can be set at the connector or the connection line on the circuit diagram.

Setting the grounding in the Navigator

1. In your engineering project, select a connector in the Navigator or navigate to its base object.
2. Open the properties of the connector.
3. "EE/EI&C" must be selected in the "Type" list.
4. In the "Subtype" list, select protective grounding or functional grounding.
5. Confirm your selection with "Apply".

Setting the grounding via the context menu

1. Select a connection on the circuit diagram.
2. Right-click the connection line to open the context menu.
3. Select the grounding you want to use: "Settings > Grounding system > ...".

Result

The connection line on the circuit diagram is shown according to the selected grounding:

Functional grounding: Connection line is solid.

Protective grounding: Connection line is dash-dotted line.

4.12.11 Automatic Routing

4.12.11.1 Objective of automatic routing

Requirement

A report has been opened.

Activating automatic routing

Select "Options > Route automatically" in the context menu of the report.

Basic properties of automatic routing

When you connect a new object on the report or move it, automatic routing is performed. The following rules apply:

- Connecting components without overlapping lines.
- Connecting components without crossing other components or labels.
- While moving components all connected lines and their dynamic connectors are recalculated and maintain their connection.
- When drawing a connection with more than two points every default segment is routed separately to keep the points specified by the user.
- If connections are drawn on a function diagram across pages, half-open connections are created instead of routing.

4.12.11.2 Limits of automatic routing

Small gaps between grid points

- If you have selected very small gaps between the start and target of a connection, routing may not be performed.
- If the distance between start and target is smaller than or equal to the current grid, routing is not performed for this connection.
- If a component connector is created just one grid point away from a barrier, routing is not performed.

Connector is not located on the grid

Since automatic routing is restricted to a certain number of grid points and is also based on the grid, it may be the case that component connectors do not lie on the grid.

- Routing is carried out on the current grid as long as it has not been selected greater than 4.
- If routing fails on the grid currently selected, this grid is halved internally for calculation if the connectors lie on a grid that can be divided by 2.

Please note that the response time of the program may be slowed down by these calculations.

Representation in the absence of the routing option

If routing is not possible, the connection line is split and displayed as a half-open, reading connection.

4.12.11.3 Configuring routing

You have the option to influence the behavior of individual objects with automatic routing of connection lines.

Excluding an object from routing

On the documents for electrotechnical design (circuit diagram, function diagram and single line diagram), there is an attribute that controls how obstacles are handled.

Make the following settings:

- (Y00T00001.Y00A07410).Value == 0:
  The object is detected as an obstacle during routing and the connection line is routed around the object symbol.
- (Y00T00001.Y00A07410).Value == 1:
  The object is not detected as an obstacle during routing and the connection line is routed through the object symbol.

4.12.11.4 Rerouting

Requirement

- A circuit diagram is opened.
- Connections are drawn.

Procedure

1. Select the connection.
2. Select the "Options > Reroute" command in the context menu.
3. The connection is redrawn.

Rules for rerouting

- The system searches for the shortest connection.
- Obstacles are bypassed.
- Cables placed on the connection are moved as well.
  See also chapter Using a cable or wire on the report (Page 170).

The same rules that apply to automatic routing also apply in this case. See chapter Objective of automatic routing (Page 148).

4.12.12 Options: Connection hooks

4.12.12.1 Purpose of connection hooks

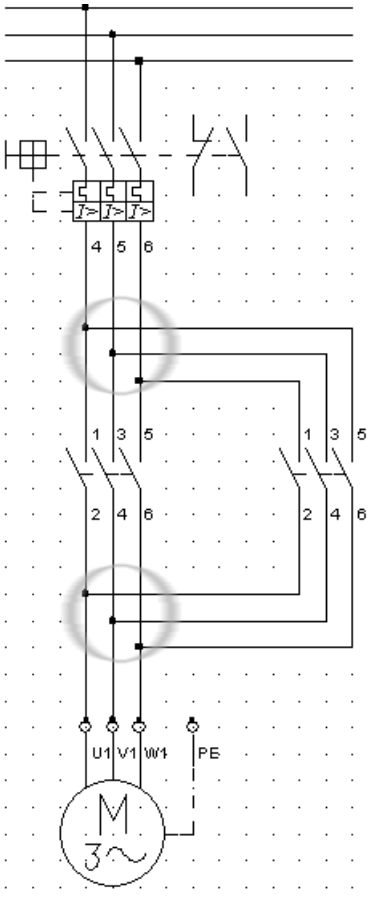
Overview

In actual plant construction, however, it is only permissible to establish connections at connectors. Connections cannot be placed everywhere on a cable.

The connection hooks on the report show the components at which a connection ends.

Example

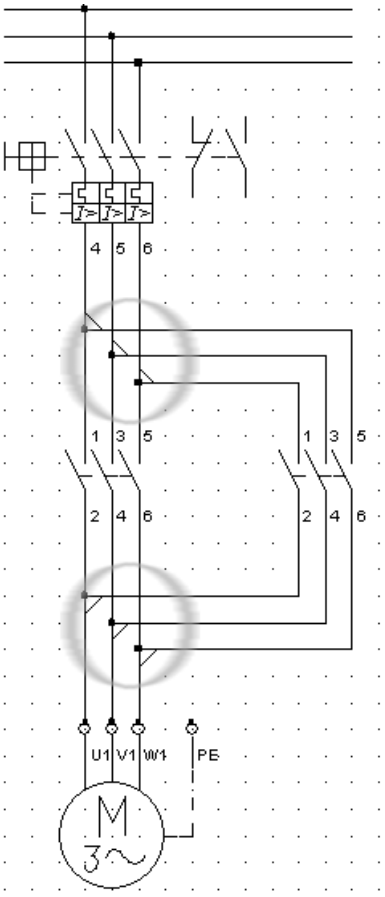
A simple form of motor switching with safety switches has been set up in the following illustration. The connections of the separate safety switch at the bottom of the illustration do not indicate which components the switch is wired to:



A connection hook is attached to the contact points of the connections to display the missing information:



This allows you to see that the circuit breaker is wired at the bottom to the terminal strip and at the top with the motor protecting switch:



4.12.12.2 Effect of the connection hooks on the engineering data

Overview

The report only tells you which component a connection has been wired to. COMOS goes beyond drawing management; it also manages the connectors using the engineering data. The reports and the engineering data are reconciled automatically.

Thus if an object is wired to a specific component, actual concrete connectors are joined to it in the engineering data. Therefore, wiring is not restricted to the report but also affects the engineering data.

See also the section titled "Controlling synchronization of the report and database with the read/write mode (Page 160)".

4.12.12.3 Specifying connection hooks manually

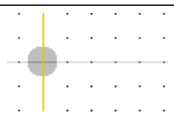
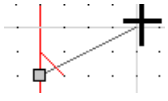
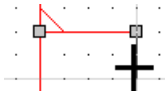
Requirement

- A circuit diagram is opened.
- The "Connection hook" option is activated in the construction representation.

Controlling the direction of the connection hook

If the starting point of a connection is a link, the first two input points determine the direction of the connection hook. If the link is at the end of the connection, the last two points determine the direction.

1. Select the Connector tool.
2. Draw a vertical line on the report. For this example, the connection does not necessarily have to be closed. Conclude the connection by right-clicking.
3. Set the next three points in the following order:

Point 1 on the line	
Point 2 to the outside, somewhat higher	
Point 3 vertically downwards	

Conclude by right-clicking.

In COMOS, logical potentials have connection hooks, but physical potentials do not.

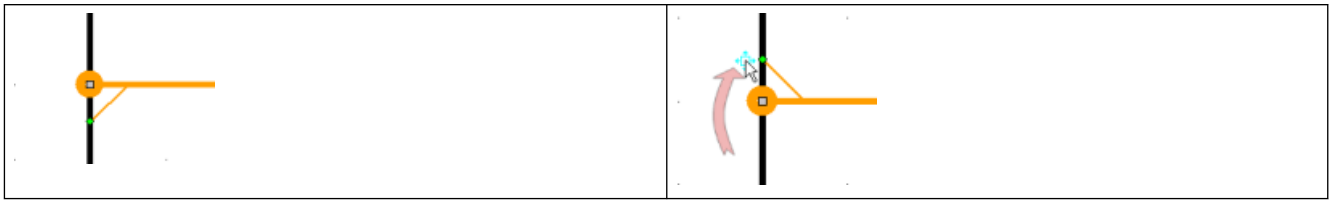
Controlling connection hooks via the options script

Specifications for connection hooks can be set in a report. You can find more information on this topic in the "Platform Administration" manual, keyword "ConnectionHookMode".

4.12.12.4 Changing connection hooks at a later time

Procedure

1. Single-click twice on the connection so the grab points become visible.
2. Enlarge the image until you can clearly see the connection hooks.
3. Slide the round grab point of the connection hook to the other side of the connection point.



Result

The direction is changed. The connector pin-outs in the engineering data are also modified accordingly.

4.12.13 Options: Change direction

Requirement

A circuit diagram is opened.

Displaying direction arrows for the connection direction

1. Right-click on an empty location in the report.
2. Select the "Options > Direction arrows visible" command in the context menu.

Changing the connection direction of an individual connection

1. Select the connection for which you want to change the direction.
2. Open the context menu and select the "Options > Change direction" command.

Correlation between the connection direction and the connector attribute "Direction"

The connection direction is displayed graphically and stored in the connectors of the engineering data.

The "Direction" system attribute which can assume the states "From" and "To" exists at the connectors.

To edit or create the system attribute, the following options are available:

- Base data, the properties window of the connector
- Engineering data, the properties window of the connector

See also chapter Working with the extended attributes of connectors (Page 198).

The following section assumes that the reader is familiar with "working with the extended attributes of connectors".

Pre-assigning the "Direction" system attribute in the base data

1. Switch to the base data and open the structure in the Navigator until you see the connector objects below the component.
2. Open the properties window of a connector.
3. Click the "Extended" button.
4. If the connector has not been edited, the "Direction" item is not in the "System" list. Click on "New attribute". A new line appears in the "User-defined" list.
5. Double-click in the "Name" field and enter the value "Direction".
6. Set the "From" or "To" value.
7. Click "Apply".
   COMOS recognizes the name as a system attribute and moves the entry to the "System" list.

When you create a device based on this base object, the connection there is pre-assigned accordingly. This default is taken into account by COMOS if the user action is consistent.

Consistent cases are when:

- A pre-assigned "From" connector is connected to a pre-assigned "To" connector.
- A pre-assigned "From" connector or "To" connector is connected to a connector that is not pre-assigned.
- The two connections are not preset.

In all other cases, COMOS attempts to create a consistent state, regardless of the default values. The default setting in the "Direction" system attribute may also be deleted and reset by COMOS if needed. If the inherited default setting at the engineering object is deleted once, the default setting in the base data no longer has an effect on this connector in the engineering data. COMOS sets the states "From" and "To" for this connector as needed.

Controlling the "Direction" system attribute in bulk processing

Connectors that are connected in the engineering data have the "checked in" inheritance mode. Such connectors can be controlled using bulk processing.

1. Select the "Extra > Bulk processing > Bulk processing for connectors" command in the COMOS menu.
2. Set the following settings for connectors in the bulk processing:
   - "Search technique: Recursive"
   - "Inheritance mode: Own (checked in)"
   - "Collection type: All"
3. Select the desired connectors in the "Name" column.
   The "Extended" button appears in the right area of the window.
4. Click the "Extended" button and edit the "Direction" system attribute.

COMOS attempts to create a consistent state starting with the inputs. When possible, COMOS sets the connection direction based on the settings in the bulk processing. The connecting direction drawn on the reports is updated automatically.

Checking the "Direction" system attribute using an evaluation

1. Click on the unit and select the "00 Evaluating reports" folder.
2. Select the "Evaluating reports" folder and then the "New > Cable lists > Cable occupation list" command.

The cable occupation list contains the "Connection from" and "Connection to" columns. These columns evaluate the "Direction" system attribute. All connectors in the "Connection from" column are "From" connectors and vice versa. You can check this by opening the properties in a field of the "Connection from" or "Connection to" column using the context menu.

Controlling connections via the options script

You can make settings for the appearance of connections in a report. You can find additional information on this topic in the "EI&C Administration" manual, keywords "DrawDirectionArrow", "PreferredConnectionDirection".

4.12.14 Options: Graphical properties of connections

4.12.14.1 Lines in the electrical engineering reports

Requirement

The following explanations apply to reports with the options script entry `Application = "ELO"`.

Preliminary knowledge

The basic technique for configuration of line types is described in the "Platform Administration" manual, keyword "Configuring line types".

Display

The display of connections is handled in the following order:

1. Inconsistent connections are always displayed in red.
2. Line type of the bridge
3. Administrator default: Line type of the connector
   You can find more information on this topic in the "EI&C Administration" manual, keyword "Administering connection types and line types".
4. Default value of the continuous connection

5. User selection: Line type in the context menu of the report
   See chapter Editing properties of graphical connections (Page 159).
6. Drawing type specific lines
   – "Bubble line"

   You can find more information on this topic in the "EI&C Administration" manual, keyword "Templates for line types".

4.12.14.2 Surface reference of the "Graphical properties" window when using connection objects

Requirement

The connection mode "Create connection objects for connector types "E" and "I" (from 10.2 SP1)" has been set in the project options.

"Line" control group

The following table describes the control elements of the "Line" control group in the "Graphical properties" window:

Control element	Description
"Width" list	Select the width of the line from this list.
"mm" list	Select the unit for the width of the line from this list.
"Line type" list	Select the type of the line from this list, for example, "Continuous line, round end caps".
"Layer" list	Select the layer on which the line is to be created from this list. You can find more information on this topic in the "Platform Operation" manual, keyword "Assigning layers".

See also

Connection objects (Page 127)

4.12.14.3 Edit graphical properties of connections with connection object

Requirement

- The connection mode "Create connection objects for connector types "E" and "I" (from 10.2 SP1)" has been set in the project options.
- A circuit diagram is opened.
- Connections have been drawn on the circuit diagram.

Procedure

1. On the report, click on the connection whose graphical properties you want to edit.
   The connection is highlighted in color.
2. Right-click on the connection.
3. Select the "Options > Graphical properties" command in the context menu.
   The "Graphical properties" window opens. See Surface reference of the "Graphical properties" window when using connection objects (Page 157)
4. Select a line type from the "Line type" list.
   The entries in the standard table "Y10|M00|A10|Y10M00N00002" are displayed.
5. Click "OK" to save the changes.

See also

Connection objects (Page 127)

4.12.14.4 User interface reference of the "Graphical properties" window when using purely graphical connection lines

"Line" control group

The following table describes the control elements of the "Line" control group in the "Graphical properties" window:

Control element	Description
"Width" list	Select the width of the line from this list.
"mm" list	Select the unit for the width of the line from this list.
"Line type" list	Select the type of the line from this list, for example, "Continuous line, round end caps".
"Layer" list	Select the layer on which the line is to be created from this list. You can find more information on this topic in the "Platform Operation" manual, keyword "Assigning layers".
"Label" field	You can assign a line label in this field. You can find additional information on this topic in the "EI&C Administration" manual, keyword "Options for labels ("Header.Class")".

"Geometry" control group

The following table describes the fields of the "Geometry" control group in the "Graphical properties" window:

Field	Description
"Angle"	If a positive angle (e.g. 45) is entered, the selected line is rotated counter-clockwise. If a minus sign precedes the angle, the line is rotated clockwise.
"Scaling factor X"	Scales a horizontal line in the X direction.
"Scaling factor Y"	Scales a vertical line in the Y direction.

4.12.14.5 Editing properties of graphical connections

Requirement

- A circuit diagram is opened.
- The circuit diagram shows purely graphical connections.

Procedure

1. On the report, click on the connection whose graphical properties you want to edit.
   The connection is highlighted in color.
2. Right-click on the connection.
3. Select the "Options > Graphical properties" command in the context menu.
   The "Graphical properties" window opens.
   Select a line type from the "Line type" list.
   The supplements to the selection list "Y10 > M41 > A10 > Y10M41N000018" are displayed at the end of the list.
4. Click "OK" to save the changes.

Templates for line types are created with the aid of standard tables.

You can find more information on this topic in the "EI&C Administration" manual, keyword "Templates for line types".

Editing the graphical properties of potentials

To edit the graphical properties of potentials, select a potential on the report. Proceed as described above.

4.12.15 Controlling synchronization of the report and database with the read/write mode

Overview

The connections that were dragged onto the report reflect the connection of the connectors in the database but are not identical with them from a technical point of view. It is thus possible to have report connections without their corresponding counterparts in the engineering data and vice versa.

Read/Write mode controls the extent to which the report connections and the joining of the connectors in the engineering data are reconciled.

Read/Write mode

If you select the "Connection" tool, the Read/Write mode list appears.

Mode	Description	Color display
"Read"	The connections must have been set in the database. The connection line on the report searches for the corresponding connection in the database. The connection line is consistent if the connection is found. With this option the connection lines in the report have no effect on the database.	• Consistent connection: blue • Inconsistent connection or no connection in the database: red
"Read/Write"	See "Read" mode. If no connection is found, the connection line generates the connection between the objects in the database. With this option the connection lines write into the database if no connections had been set in the database before.	• Consistent connection or no connection in the database: black • Inconsistent connection (the connection in the database is connected with another object): red
"Write"	A connection is generated between the engineering objects if no connection is found in the database. If a connection is already entered at the object connectors, this connection is deleted and overwritten. With this option the database information is always overwritten by the diagram connection lines.	Color: green

Differences

As explained above, the technical difference between report connections and connections involving connectors can lead to the following differences, for example:

Difference	Description
Connectors are joined in the engineering data, but there are no associated report connections	This is usually no problem. If only some of the connector cross-references are available in the form of connections on the report, an inconsistency can occur.
Report connections without associated connected connectors	In this case the report connections are marked in red (inconsistent).

4.12.16 Changing the Read/Write mode

Procedure

1. Select the connection.
2. Select the "Settings" command in the context menu, followed by the required mode.

Result

The mode is changed for the corresponding connection. The color of the connection changes accordingly.

4.12.17 Potential information for writing connections

Overview

The potential information of the writing connection overwrites that of the connector in the engineering data. The potential pointer is reset.

Example

A connector with potential information which refers to the potential exists in the engineering data.

Place the connector on a report at a writing connection. The writing connection also has a potential that differs from the potential at the connector.

4.12.18 References and reference objects

4.12.18.1 Automatic references for physical potentials

Requirement

- A circuit diagram is opened.
- A physical potential has been created in the Navigator and already placed in another report.

Creating/using references for a potential

Drag a physical potential that you are already using in a report onto a second report. A page reference automatically appears at the beginning or end of the potential.

4.12.18.2 Representation of automatic references

Requirement

- A circuit diagram is opened.
- A connection is drawn and not connected at both ends on the report.

The representation of references is controlled using the project properties. You can find additional information on this topic in the "EI&C Administration" manual, keyword ""Reference display" control group".

Purpose

References allow the graphical interruption of a connection. This connection may then be continued in the same report as well as in another report.

Fundamentally there are two types of cross-references:

- 1 to 1 references
  Example: Component connections
- 1 to n references
  Example: Potentials.
  Potentials often range across several pages. In this case, there are many references to an individual object (to the potential).

Connections open at one end

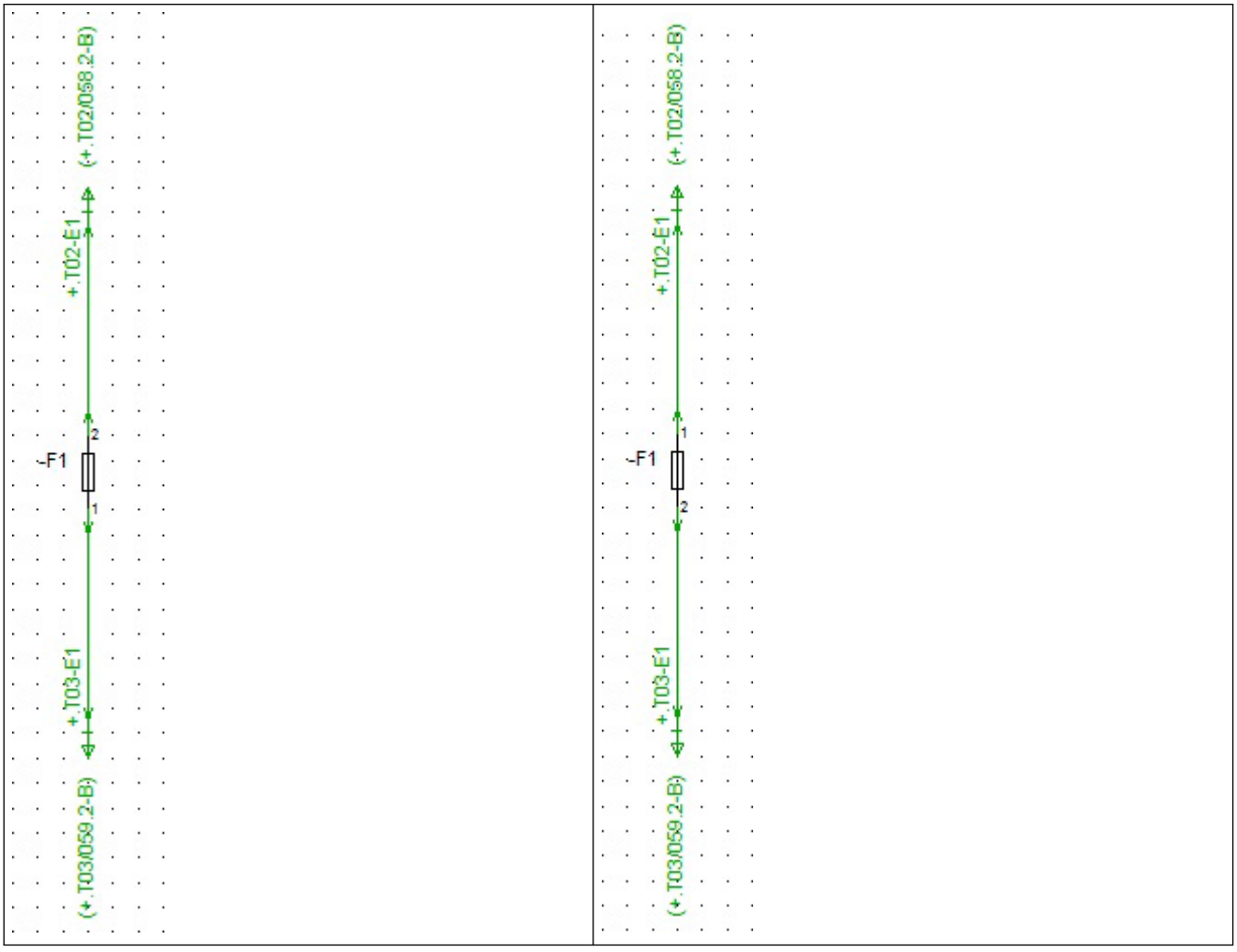
A component connection open at one end with cross-reference is represented as follows:



Direction of references

It makes no difference in terms of the consistency of the data which document cross-reference is drawn at the top or the bottom. The only thing that matters is that the document cross-references on the report in their totality are consistent with the links in the database.

That is why both of the following drawings are valid:

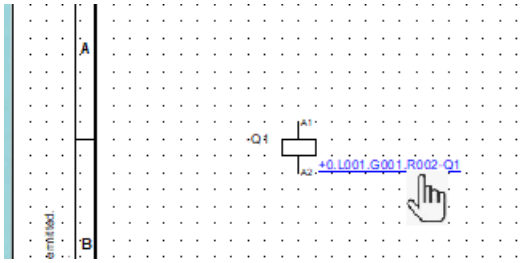


COMOS default

The order of the cross-references on the report results from the device labeling. The predecessor is at the input, the successor at the output. In the example above you will find the specification "(2.1) -M3". In this example "2" stands for the sheet and "1" for the path, i.e. the column. M3 is the owner.

Smart links

References can be navigable. You can find more information on this topic in "EI&C Administration", keyword "Preparing smart links".



Move your mouse cursor over a reference.

The reference is displayed underlined in blue. In addition, the mouse cursor changes to a hand icon.

Clicking on the reference takes you directly to the target object, which is highlighted in pink and zoomed in on.

If the target object is on a different document, this document is automatically opened and brought to the foreground.

Creating references for reports with multiple sheets

When creating a reference, COMOS takes into account that a report may have multiple sheets.

Example for four reports: Of these four reports the first three are single sheets and the fourth one has two sheets. You now set a reference from the first report to the second sheet of the fourth report. "4.2.1" is displayed in the reference.

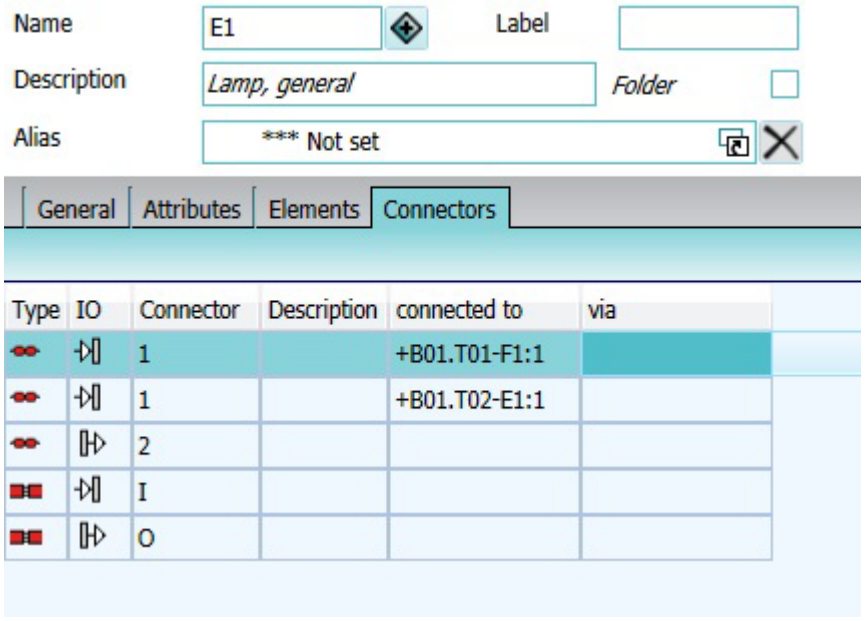
Bridge references

By default, references to half-open bridges are labeled in such a way that the reference is built according to the placement information.

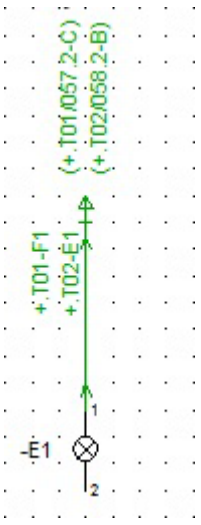
Connectors with multiple connections (multiple line references)

For connectors with multiple connections, you do not need to draw all connections for references. Cross-references in COMOS have multiple lines.

Example of an auxiliary connector: The following light also has a joined auxiliary connector at connector "1":



If you draw a connection for the cross-reference at connector "1" on the report, both counterpart connectors are displayed automatically:



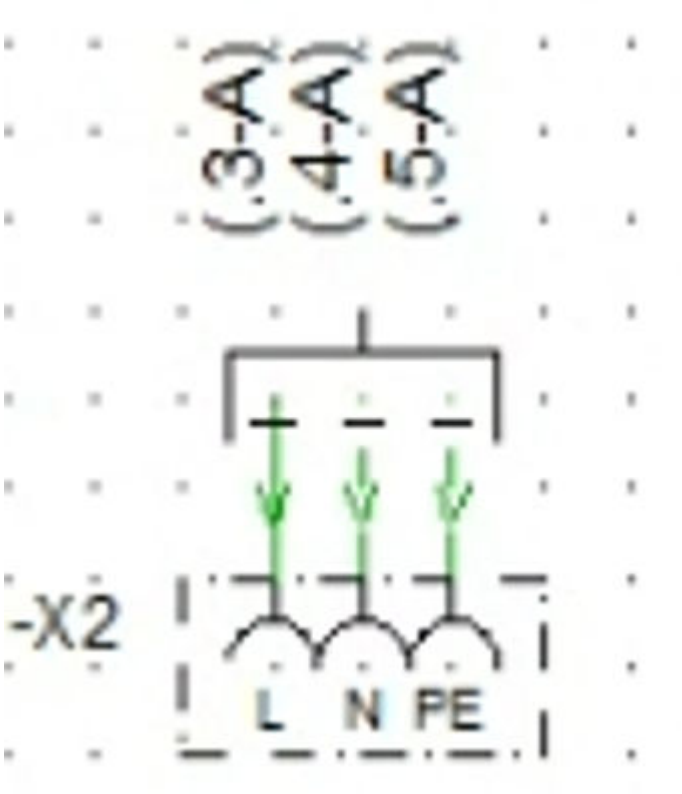
4.12.18.3 Use reference bracket/collected reference

Requirement

- A circuit diagram is opened.
- Multiple connections are drawn and only connected at one end on the report.

Reference bracket/collected references

If multiple open connections point to the same target, you can set a reference bracket:



If the objects are located in different sheet areas, the abbreviations of the sheet areas are displayed on the opposite side of the reference bracket.

Creating a collected reference

1. Select several open ends with a reference.
2. Select the "Options > Create reference bracket" command in the context menu.

Setting a collected reference

1. Select the "Connection > Memorize" command in the context menu.
2. Select the "Connection > Connect with" command from the context menu.
   The connection direction runs from the memorized connection to the current connection. The memorized connector receives the "From" property, the current connector the "To" property.

Deleting a collected reference

1. Select the collected reference.
2. Select "Delete" from the context menu.

4.12.18.4 Purpose of reference objects

Reference objects enable you to prepare half-open connections in your templates and manage them later.

This makes it easier to track connection partners across multiple circuit diagrams and in case of a large number of circuit diagrams and references.

4.12.18.5 Using reference objects

Requirement

- The reference object with the functional classification O4E4 (iDB) or RO (cDB) must be available as a base object.

Placing reference objects

1. In the Navigator, go to the"@10|A50|A20|A30|A30 reference object" base object in the Base objects tab.
2. Drag&drop the "@10|A50|A20|A30|A30 reference object" base object to the circuit diagram. An engineering object is created in the Navigator parallel to the project.

Note

Reference objects can be placed up to two times. This includes referencing by means of implementation pointer.

When the maximum number of placements is reached, the drag&drop function of this object is automatically disabled and all future connections between reference objects will become inconsistent.

Information on the reference object

- Device label (Unit (%N DevUnit%), Location (%N DevLocation%) Name (%N DevName%)
- Potential information from the referenced connection

  Note

  To display the potential information at the reference object, the "Allow potential tracking" option must be enabled in the project properties of your project.

Attributes for display

You have two options for activating the attributes:

- In the context menu of the reference object "Graphical properties > ..."
- In the system properties of the reference object under the Attributes tab

Navigating between reference objects

You can navigate between placed reference objects in the circuit diagrams with the arrow buttons in the menu bar.

4.12.18.6 Connecting reference objects

You have several options for connecting reference objects.

If a reference object was connected, it references to the database object of the connector at the other end of the half-open connection.

Requirement

A circuit diagram was opened.

Connection by placing the reference object twice

See chapter Using reference objects (Page 167).

Drag an engineering object that has already been generated from the Navigator to the circuit diagram.

A connection is automatically created as soon as a reference object references to a second connector in the database. To do so, at least one document on which a reference object was placed must be open.

Connection by drawing on the diagram

See chapter Using reference objects (Page 167).

You can connect a placed reference object to a connector of a different object on the circuit diagram by using the connection tool in the toolbar of the circuit diagram.

Connection by merging two objects

- Using object debugger
- Using engineering tasks (keyword: `Merge, MoveOrMerge)`

You can find more information on this topic in the "Modular Engineering" manual, keyword "Reference of the predefined task functions (basic types)".

Connection by implementing two reference objects

1. In the Navigator, open the properties of the reference object in the context menu.
2. Drag&drop the second reference object to the "Implementation" field in the "General" tab.

Note: If an already placed reference object was implemented with a different reference object, the maximum number of placements has been reached.

If the implementation is removed again, the connection of the reference objects is separated.

Connection by automatically setting the implementation pointer

To use the implementation pointer automatically, use the "Memorize" and "Connect" function; it is located in the context menu of the reference object.

See chapter Connecting half-open connections (Page 139).

Connection by script

See chapter Using reference objects via script (Page 169).

Connection by bulk processing

To connect reference objects in bulk, use the "Bulk processing of half-open connections" plugin.

See chapter Connecting half-open connections via query (Page 199).

4.12.18.7 Using reference objects via script

Example script for creating a reference object

```
Set GC = CreateObject("ComosROUtilities.GlobalCastings")
 Set comosDoc = A
 Set referenceDBObject = B

 Set IReport = comosDoc.Report()
 Set ReportDocument = IReport.ReportDocument

 Set ro = ReportDocument.CreateReportObject(40,28,40,28)

 Set roReference = CreateObject("Comos.WSP.WspRoReference")
 Set ro.XObj = RoReference

 Set roDev = GC.GC_GetIRoDevice(ro)
 Set roDev.Device = referenceDBObject

 ro.EvalDone = False
 ro.EvalDone = True
```

Example script for calling the referenced connector of a placement

```
Set DocObjOfReferenceObject = A
 Set expectedconnector = B

 Set referencedConnector = DocObjOfReferenceObject.Reference2

 Output expectedConnector.SystemFullName()
 Output referencedConnector.SystemFullName()
 Output cstr(referencedConnector Is expectedConnector)

 Result:
 B01|T02|F3|EA02
 B01|T02|F3|EA02
 True
```

4.13 Cables and wires on the construction display

4.13.1 Using a cable or wire on the report

Requirement

- A circuit diagram is opened.
- Connections are available.

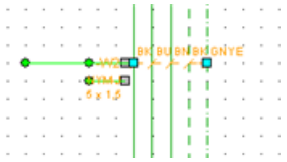
Creating a cable object

1. Select a subunit.
2. Select the "New > 14 Cable/lines" command in the context menu.
3. Select a cable.
   Do not use a cable of the type "xxxx cable, unspecified". These entries are templates for creating new cable objects and are not intended to be placed on the report.

Placing a cable

1. Select the cable object (not a wire object).
2. Place the cable object.
   The cable is displayed on the diagram in the form of a dotted line.

3. Single-click twice to activate the grab points.
4. Using the grab points, drag out the cable symbol until it is wide enough to include all connections that are to belong to the cable:



Moving cables automatically with the object

1. Activate the entry "Options > Route automatically" in the context menu of the report.
2. Move a component or the connection.
   The cable is moved automatically with the connection.
3. The following applies to cable symbols on a connection with several connection lines:
   – The basis for the movement is the intersection of the cable with the first connection line. For this reason, a cable symbol cannot be moved correctly automatically if a connection with several connection lines bypasses an obstacle on different sides at the same time.

Dragging connections across cables

1. Drag a connection line across a cable symbol.
   The cable is assigned to the connection.

Exceptions

The use of a cable on both sides of a connection connected by links does not work correctly in the following case:

- Connect semi-open connections with "Connection > Memorize" and "Connection > Connect". A link is created.
- Assign a cable to one side of the link.
  The wire information is set.
- Assign the same cable to the other side of the link.
  The wire information already set is found and displayed correctly in the report.
- Remove the cable from the connection at only one side of the link.
  The wire information at the connector is deleted.
- Refresh document
- The second assignment of the cable is found on the other side of the link.
  The wires are applied once again because the wire information was deleted. The wire assignment may be different from the initial state.

See also

Rerouting (Page 150)

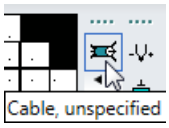
4.13.2 Creating new cable on the circuit diagram

Requirement

- A circuit diagram is opened.

Procedure

1. Select the symbol of the cable from the toolbar:

   The cable is created on the circuit diagram and in the Navigator.
2. Select the cable on the circuit diagram.
3. Select the "New > Wire" command in the context menu.
4. Place the wire on the circuit diagram.
   The wire is created on the circuit diagram and in the Navigator.
   The label is displayed to identify the cable to which the wire belongs.
5. Place the wire at the desired position.
   Wire symbols that are not assigned are not deleted automatically on the circuit diagram. If you place the cable symbol subsequently, symbols that are already placed are not moved onto the associated cable symbol.
6. To create more wires, repeat the previous steps.

Result

When you place the cable, to connect a motor for example, the new wires are automatically connected where the cable intersects the connection line. The properties of the wires are updated in the Navigator.

Note

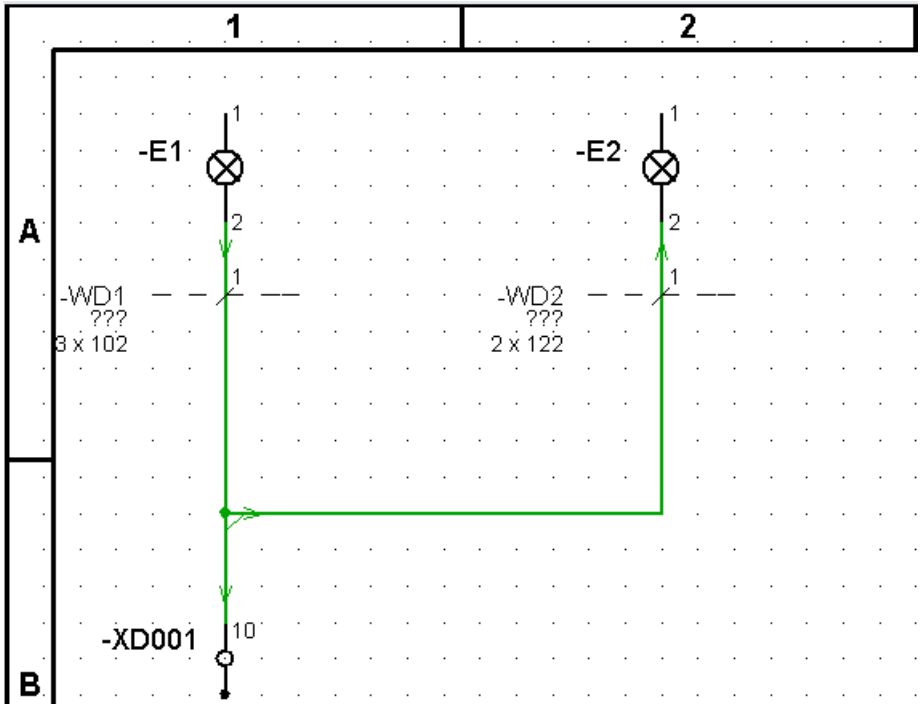
Graphic wire information is not updated

When you place wires from the Navigator on the circuit diagram, the connection information is automatically updated. Wires generated on the circuit diagram are only a graphic information and are not automatically updated.

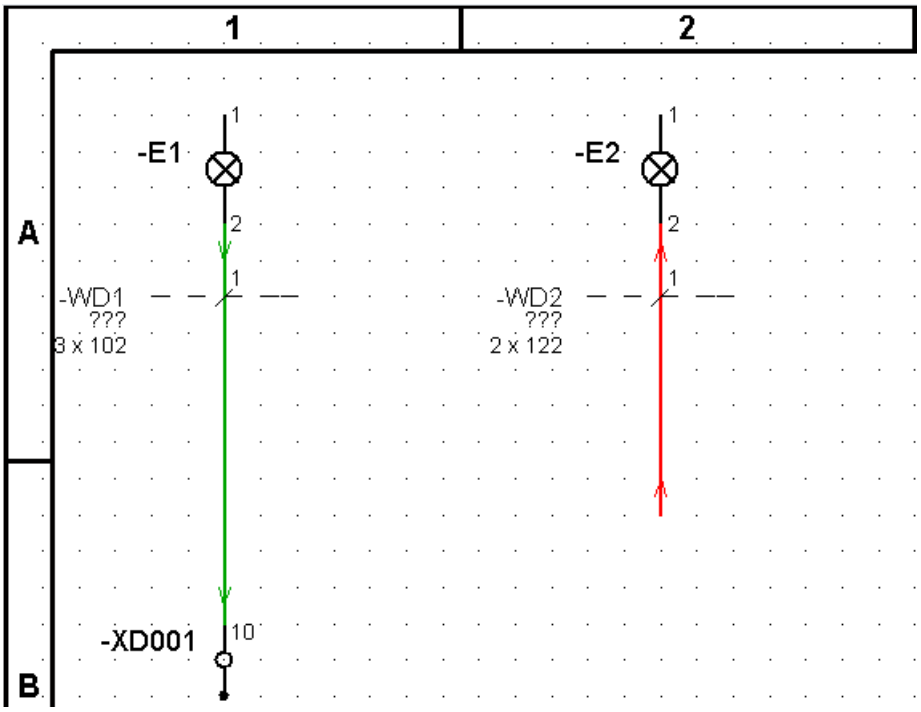
4.13.3 Consistent and inconsistent cable states

Consistent states

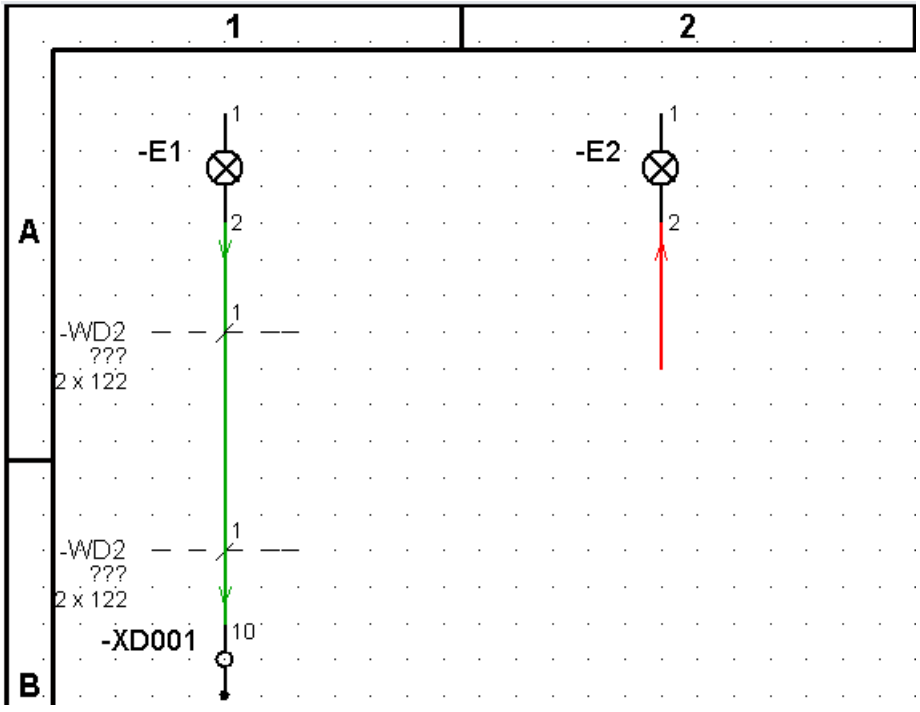
- In the case of branching connections, it is permissible to assign a separate cable or wire to each branch:



- It is permissible to set a cable or a wire for semi-open connections:

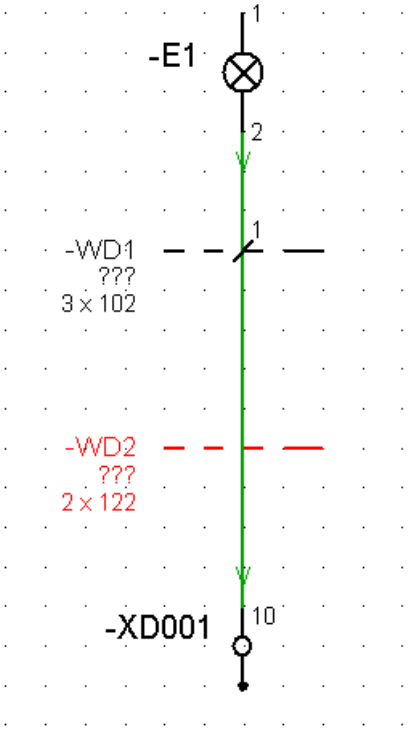


- It is permissible to set a cable or wire twice on the same connection.



Inconsistent states (automatic rejection of the assignment)

- It is not permissible to assign two wires to one connection.
  This inconsistent state also affects semi-open connections and connections via links.
  If a connection on the report is in contact with two placed cable objects, the following applies:
  - The first item of cable information is applied.
  - Any subsequent cable information is rejected. The corresponding cable or wire is labeled as inconsistent (red).
  - The connection is labeled as consistent.



Editing inconsistent states

- Option 1: Drag the inconsistent cable from the connection.
- Option 2: Drag the consistent cable from the connection.
  In this case, the inconsistent cable remains inconsistent. As soon as the inconsistent cable moves slightly, a new intersection is created with the connection and the cable becomes consistent.

Wire assignments

If you delete an object from a chain of three and one of the two connections has a wire assignment, the connection is merged through the removal of the middle device. In order to avoid incorrect reassignment of the wires the connection is switched to "reading" when a cable is applied.

4.13.4 Cable legend

Requirement

- A circuit diagram is opened.
- Connections are available.
- The cable has been placed.

Legend

-W2	Name of the cable object
NYM-J	Label of the cable object
5x1.5	Cable cross-section (number of wires x wire cross-section)

The wires are numbered sequentially. This number is also entered on the diagram at the intersection of the cable and the connection:



If wires have already been set in the database, this information is retained as far as possible, but the remaining wires are set during positioning on the diagram if only part of the information has been set. In this way, the wires are allocated in the correct order.

4.13.5 Rotating cables

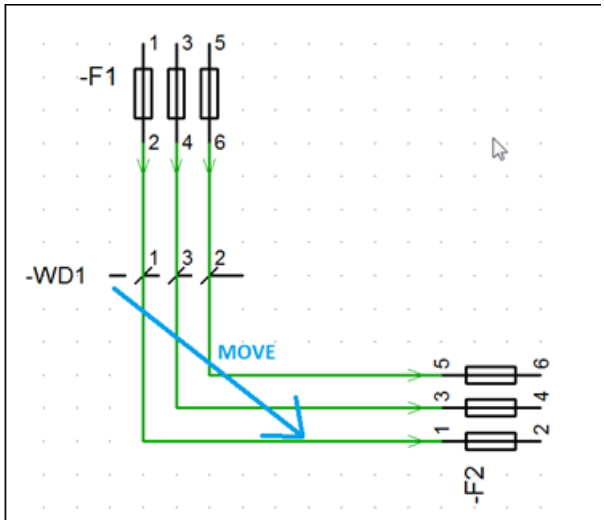
Rules

When a cable placed on a report is moved, a rotation is only possible if you press the space bar. If you have placed the cable on the correct connection element, the cable rotates automatically. Otherwise, use the rotation point before you move the cable. That can lead to a resetting of already existing configurations.

Example

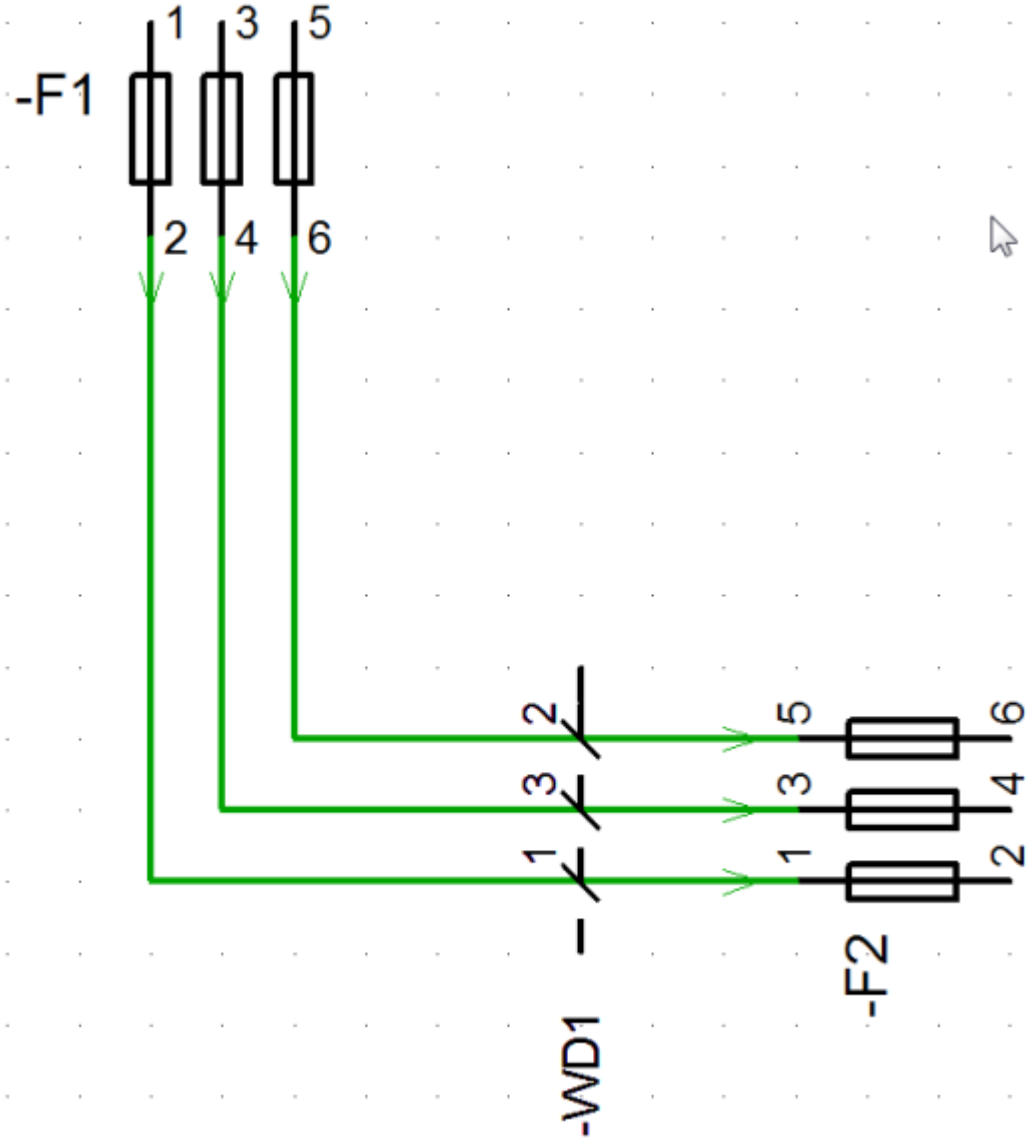
- Cable angle: 0
- Wire: 1, 2, 3

- Segment angle: 270



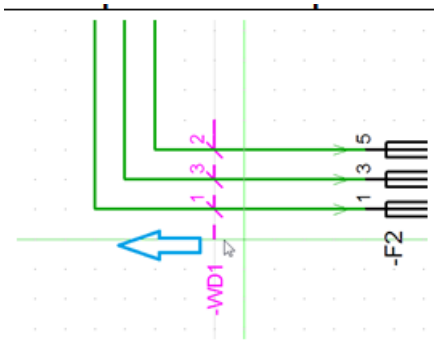
- Cable angle: 90

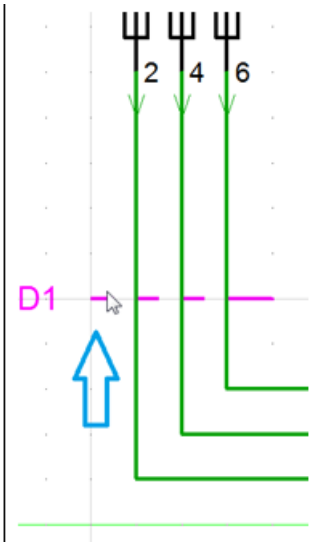
- Wire: 1, 3, 2
- Segment angle: 0

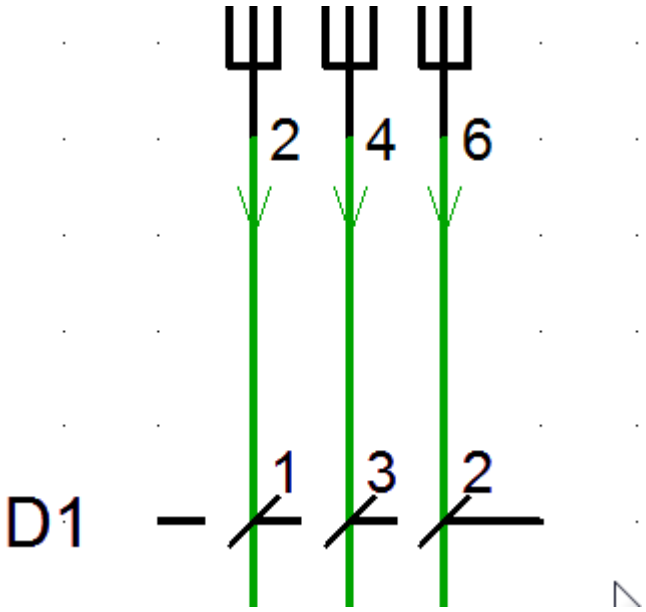


If you move the cable and the angle changes, the cable symbols are hidden until the moving process has been completed.

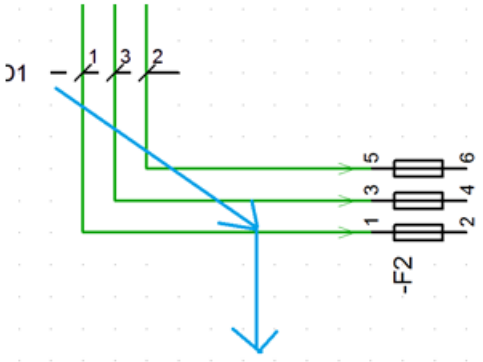
Example

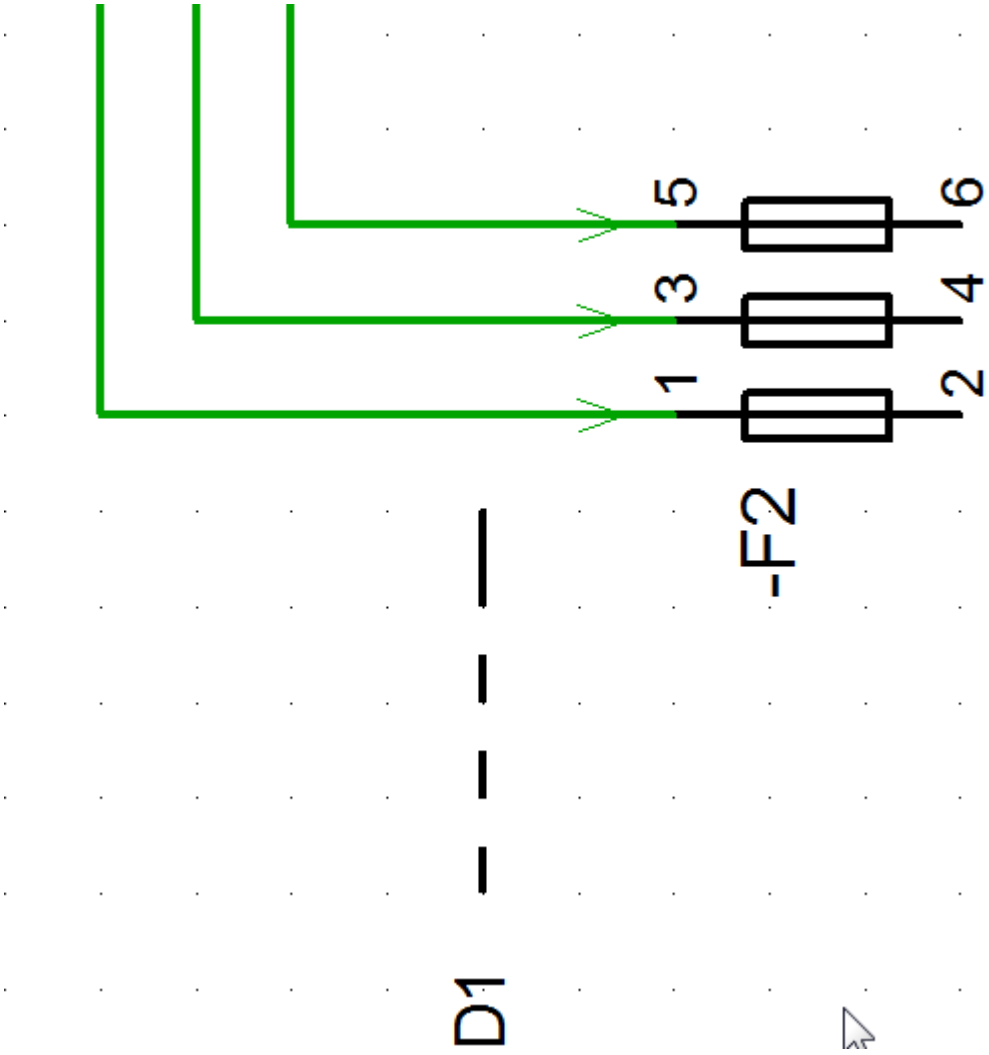




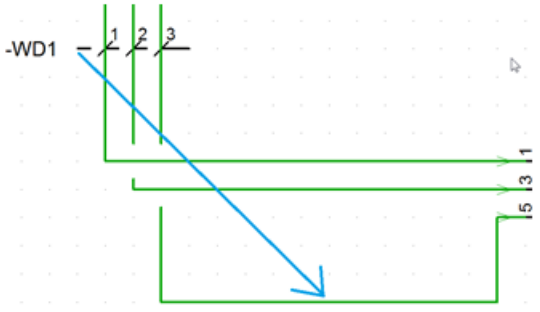


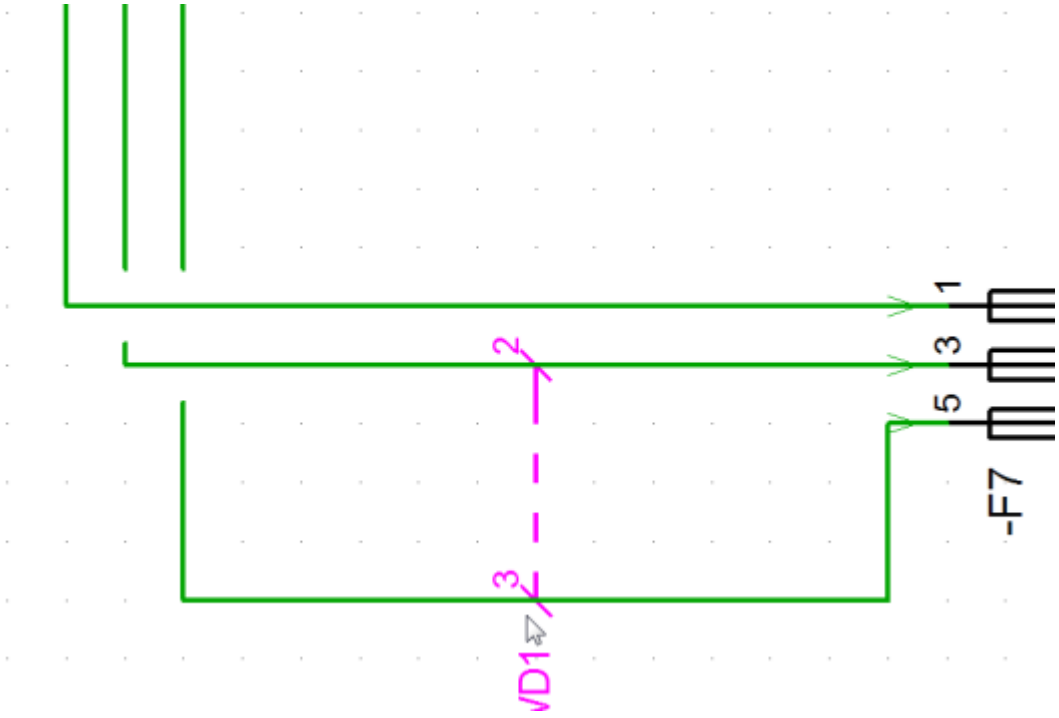
If you move the cable without rotation, the cable loses the configuration of the wire if you did not apply it to the connection.





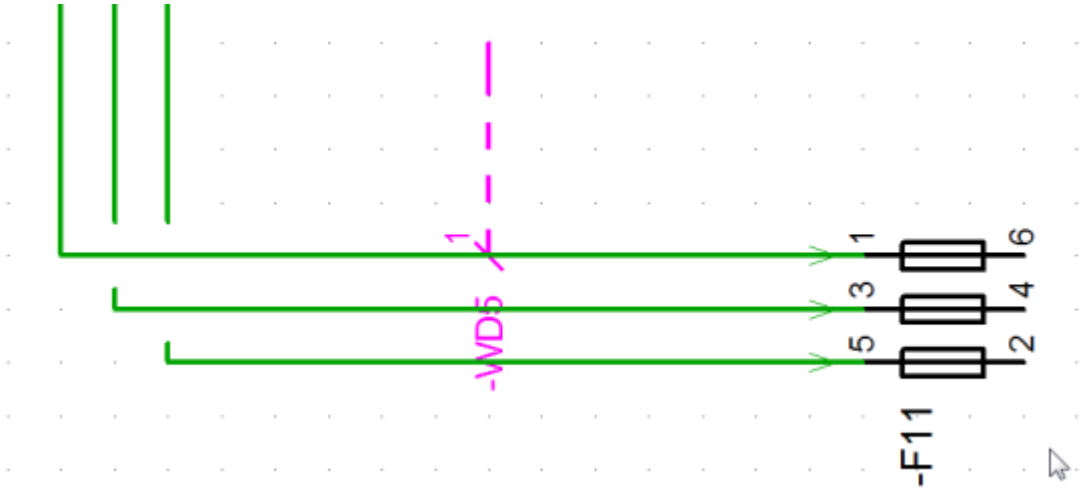
If the distance of the connection to the new position is longer than the cable length, the cable is not lengthened, in order that all connections are retained.





The cable only rotates in one direction. In this case:

- Segment angle: 0
- Cable angle: 90



4.13.6 Assigning connection information

Requirement

The function is only offered if the selected connection, or at least one connection in the case of a multiple selection, meets the following requirements:

- The connection is not a potential.
- The connection is also consistent in the database and is not displayed in red on the report.
- The connection is not assigned to a cable.

If wire information already exists at the connectors and a cable object has been assigned to the connections, the information of the cable object is evaluated and displayed.

Procedure

1. Select the required connection.
2. Right-click on the selected connection.
3. Select the "Settings > Connection information..." command in the context menu.
   - If you do not use any connected objects, the "Connection information" window is opened.
   - If you use connected objects, open the properties of the selected object.
4. Enter the required information.
5. Click "OK" to save your entries.

User interface reference window "Connection information"

Control element	Description
"Cable type" list	You select the information regarding the material structure and properties of the cable from this list. This information is managed in the form of a key.
"Cross section" list	You select the cross section of the cable in mm^2 from this list.
"Color" list	You select a permissible color according to its IEC code from this list; this is the color in which the connection will be displayed on the report.
"Number" field	You assign a label for unique lines in this field.

4.13.7 Disconnecting wires

Requirement

- A circuit diagram is opened.
- A connection has been drawn.
- A cable object is placed on the connection.

Procedure

Drag the cable symbol on the report away from the connection.

Result

The wires are dissolved in the engineering data and are no longer displayed on the report.

4.13.8 Shielded cables

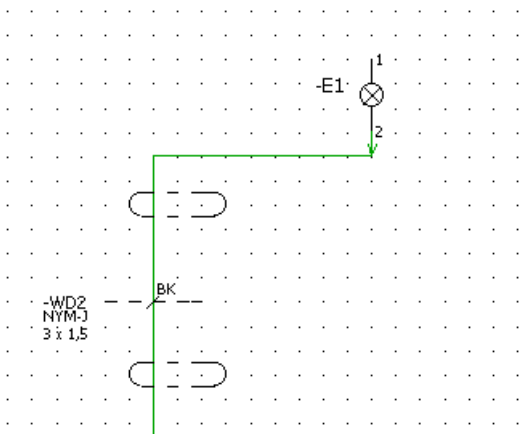
4.13.8.1 Creating a shielded cable

Requirement

- A circuit diagram is opened.
- A connection has been drawn.
- A cable object is placed on the connection.

Creating freely placeable shields ("SH shield")

1. Mark the cable (not the connection) on the report.
2. Select the "New > SH shield" command in the context menu.
   The "SH shield" command creates a shield object in the Navigator. This means that a connection can be connected to this type of shield on the report.
3. Place the shield next to the cable symbol on the connection:



   You can place several shield symbols using the same approach.

If you place an "SH shield" object on the cable symbol, the context menu "Settings" is no longer available. Instead, use the command "Shielded representation" to equip the cable symbol with a shield symbol.

If you still want to equip the cable symbol with an "SH shield", the following applies:

If you single-click on the symbol several times, you select the components of the symbol group one after the other. In this way, you can select the cable symbol only to get the "Settings" context menu, for example.

Create shield symbol for the cable symbol ("Shielded representation")

1. Mark the cable (not the connection) on the report.
2. Select the "Settings > Shielded representation" command in the context menu.
   The "Shielded representation" command does not create an object. This means that a connection cannot be connected to this type of shield on the report.

Settings for the cable symbol

The settings in the engineering object are transferred to the "System" tab in the "Graphic settings" options group.

4.13.8.2 Representation of the shielded cable

Requirement

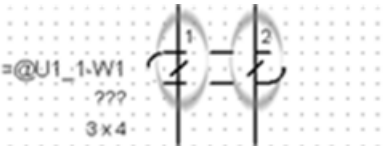
- A circuit diagram is opened.
- A connection has been drawn.
- A cable object is placed on the connection.
- A shield is placed.

Display and labeling

A shielded cable is displayed as a flat oval in the report. The wires are automatically assigned to the connections when you drag the shielded cable over a connection and the label for the wires is written next to the connection. The wires are displayed in the database together with the assigned connections.

Example

In the example, wire label "1" appears next to the left-hand connection and "2" next to the right-hand connection.



Legend

The following table describes the labels:

Label	Description
-W7	Designation of the cable, where the minus sign is the separator taken from the project options. This designation can be generated automatically.
(N)YM-J	Subtype of the cable. This information is taken from the "parent object".
10x1.5	Number of wires and cross-section. This information is taken from the "Name" field of the object.

4.13.8.3 Changing the representation of the shielded cable

Requirement

- A circuit diagram is opened.
- A connection has been drawn.
- A cable object is placed on the connection.

Changing the representation

1. Click on the shielded cable twice.
   The grab points appear.



2. Change size
   To widen the shield, left-click on the grab point, hold the mouse button down, and drag the point to a new position.
3. Moving descriptions
   To move a designation text, left-click on the rectangular grab point, hold the mouse button down, and drag the point to a new position.
4. Rotating the shielding
   To rotate the shielding, click on the round grab point at the far left, hold the mouse button down, and drag the grab point to a new position.

Moving or editing shields

If shield objects were placed on the report, these are initially grouped with the cable. A group of this kind can be moved, deleted, etc., as desired.

To delete or move an individual shield object, follow these steps:

- Click on the shield several times in succession.
  The single-clicks run through the group until only the shield is marked.
- Select the shield on the report. Select the "Grouping > Cancel" command in the context menu.
  All shields can be moved individually as you require.
  If you select the "Grouping > Restore" command in the context menu, all shields will be connected to the cable symbol once more.

4.13.8.4 Connecting a cable to the shield element

Requirement

- A cable object with shield element has been created in the Navigator.
- A circuit diagram is opened.
- A connection has been drawn.
- The shield element is placed on the circuit diagram. The "Shielded display" of the line object does not provide this functionality.

Connecting a cable to the shield element

Shields have one graphic connector to the left and right respectively to which you can join a connector.

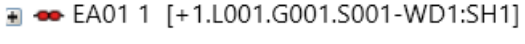
Use the "Connection" tool to connect the shield to a device, such as a PE terminal.

Note

Note: A maximum of two connections can be connected to one shield element. In this case, these connections are automatically connected to each other.

Result

- No connection is created under the Engineering object of the shield in the Navigator.
- On the "Wire" tab of the cable object, the connection of the component is displayed on the shield element.
- In the Navigator, the shield information is displayed at the connection of the device:



4.13.8.5 Assign the beginning and end of a shield

Requirement

- A circuit diagram is opened.
- A connection has been drawn.
- A cable object is placed on the connection.
- A shield has been created for the cable object.

Procedure

If the connections are so far apart on the chart that they cannot all be covered by the shield, open the shield. Proceed in the same way as if the connections were on different reports.

1. Select the individual shield, not the group.
2. If necessary, cancel the grouping by selecting the "Grouping > Cancel" command in the context menu.
3. Select the "Settings > Start of shield" command in the context menu.
4. Place a second shield.
5. Select the "Settings > End of shield" command in the context menu.

Result

The shield's rounded edges disappear from the report. This enables you to tell from the report whether the shielded cable is complete or whether it encompasses other connections at a different position.

4.14 ANSI cables

4.14.1 Selecting the "ANSI cable" tool

Requirement

- A circuit diagram is opened.
- The tool is available for this circuit diagram.

"Connection" tool

The "ANSI cable" tool is located on the menu bar of the report. This tool is used to draw a connection between objects (components) in the style of the ANSI standard.



Result

The tool is selected. The mouse pointer changes into a cross-hair.

4.14.2 Constructing ANSI cables

Requirement

- A circuit diagram is opened.
  See chapter Creating or opening a circuit diagram (Page 95).
- The tool is selected.
  See chapter Selecting the "ANSI cable" tool (Page 188).

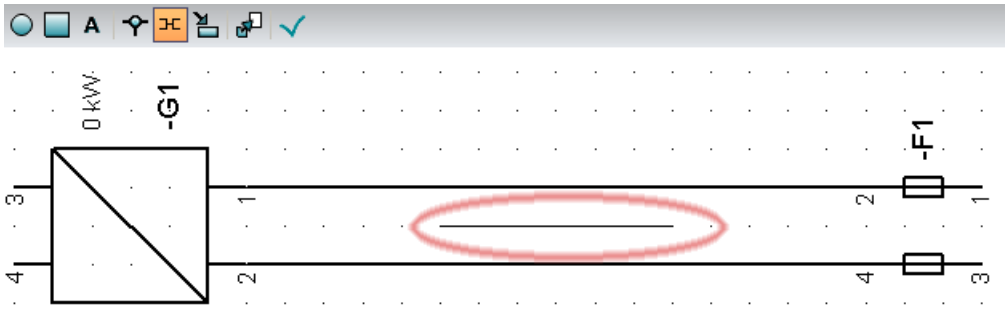
Introduction

This tool allows multiple connection lines to be grouped together graphically. As such, all connection lines remain uninterrupted. Please note that this functionality does not work with oblique connections.

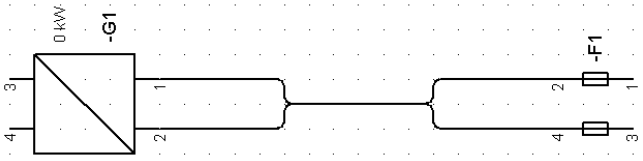
Operation

1. Click on the "ANSI cables" icon.
2. Draw a horizontal or vertical line with two mouse-clicks. This line fulfills two functions:
   – The starting point of the line is the point through which the grouped cable runs.
   – The horizontal or vertical length of the line determines the lengths of the grouped area.
3. After the second mouse-click, which determined the end of the line, the mouse cursor changes into a marking pointer. Draw a selection frame or mark the lines with <Shift> + mouse-click.
4. Confirm with Return.

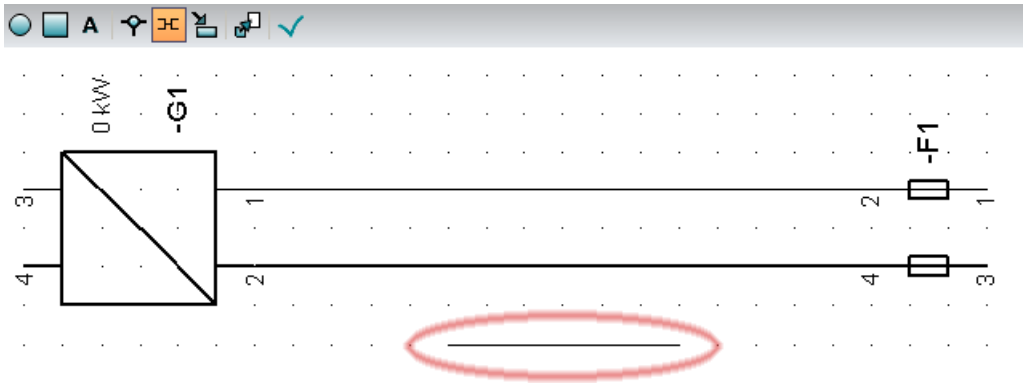
If the grouped area is to run in the middle of the connections, then the line must run through the middle:



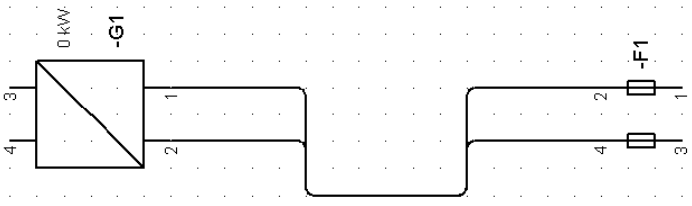
Result:



If the line is drawn at another point, the grouped area also runs along there:

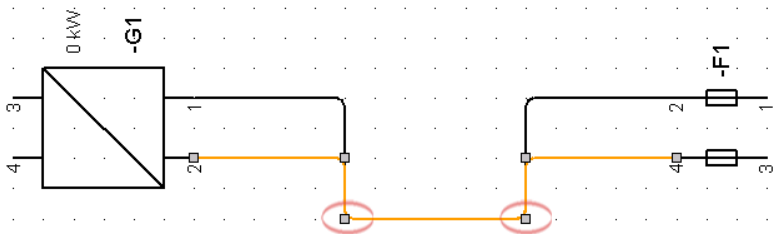


Result:



Editing grouped cables

Since all connections remain in uninterrupted, you can simply switch a line to manipulation mode with two separate mouse-clicks. Grab points appear at the corner points:



The bended line can be placed again wherever you wish with the aid of the grab points.

Object-based editing of connections in the database: 5

5.1 Definitions on the topic of "Connections"

Definition of "connection"

Connections are the logical object links that have been set in the database or on the report. You can check the existing connections in the properties of an object in the "Connectors" tab. This "Connections" tab is always visible for base objects For engineering objects, the "Connections" tab is displayed if necessary.

If the project option for using connection objects is activated, the logical operation between two objects in the database is also created as an object.

Definition of "connector"

Connectors always have one of the following statuses:

- "Input"
- "Output"
- "Neutral"

Connections without a status do not exist.

Connection direction

The following applies to all sequences of actions presented below for connecting two connectors: Where possible, the extended attribute "Direction" is also set accordingly in the connectors.

1-wire terminal

A terminal that can only be connected to one component.

See also

Connection objects (Page 127)

Connections on the construction display (Page 127)

5.2 Editing connections in the Navigator

Joining connectors

1. In the Navigator, select two devices which you want to connect to one another.
2. Expand the object structure in the Navigator until you see the connectors.
3. Select the connection of the device which is to serve as a source connector.
4. Drag&Drop the connector to the required target connector of the second device.

Result

The connectors are joined to each other. A connection object is created below the collector object of the source connector. The connection is entered in the properties of the connection object.

This procedure can only be used to select items individually.

Note

Using connection objects

In order to work with connection objects, the project option "Create connection objects for the connector types "E" and "I" (as of 10.2 SP1)" in the category "EI&C" must be selected.

Disconnecting a connection in the Navigator

If a connection has been set between two connectors using the Navigator or the "Connectors" tab, you can detach the connection in the Navigator.

To detach a connection in the Navigator, follow these steps:

1. Right-click on one of the joined connectors.
2. Select the "Dissolve connection" command in the context menu.

Result

The connections are detached. The connection object below the collector object is retained. The information of the connection object remains in the properties of the selected connection. When a wire is placed, the wire information is retained.

Assigning connection information to a signal or potential in the Navigator

1. In the Navigator, select the connector you want to join.
2. Drag&Drop the connector to the required signal or potential.
   The connector is joined to the required object. The pointer is set automatically. Alternatively, you can Drag&Drop the required object to the connector.

Result

- The signal or potential information is assigned to the connector.
- The signal or potential information is displayed at the connector in the Navigator.

Display in the Navigator

If the connectors in the Navigator are not located directly below the owner, the full label ("FullLabel") is displayed instead of the label ("Label").

Navigating between connectors and connection objects

In the context menu, you can select whether you want to navigate to the associated connector, connection object, cable or wire.

5.3 Working with the "Connectors" tab and the Navigator

Overview

If you do not want to work exclusively via the Navigator, you can instead work using the properties of one of the two objects to be connected.

Procedure

1. Open the properties of one of the required objects.
2. Select the "Connectors" tab.
3. Select the second object in the Navigator.
4. Expand the object structure in the Navigator until you see the connectors.
5. Drag the required connector (e.g.: "EA01") from the Navigator to the corresponding connector of the "Connectors" tab using drag&drop.

Result

The objects are connected to each other. A connection object is created at the connector of the source object.

Multiple selection

The multiple selection feature enables you to connect several objects simultaneously.

If you want to join several connectors simultaneously, you have two options:

- Select several connectors in the Navigator only and drag&drop the object with which you want to establish a connection to the "Connectors" tab. The free connectors are allocated in the same order in which they are sorted on the tab.
- Select several objects both in the Navigator and on the "Connectors" tab. Drag the selected objects from the navigator onto the tab. The selected connectors are allocated in the relevant order.

A corresponding warning message is put out if the number of the relevant selected connectors does not match. If you decide to carry out the action nonetheless, the connectors are allocated in order until the smaller of the two groups selected no longer has any free connectors.

Sorting

If you use the "Connectors" tab in conjunction with the Navigator, the "Sorting" function is available. You can use the symbols to control which connectors are to be displayed on the tab. You can also sort the table columns.

Disconnecting connector cross-references

1. Open the properties of the relevant object.
2. Select the "Connectors" tab.
3. Select one of the following options:
   – If the connector has not been set via an interactive report, but via the Navigator or the "Connectors" tab, right-click on an entry in the "Connected with" column. Select the "Connection > Dissolve connection" command in the context menu.
   – If you have already set the wires, select the "Disconnect (retain wires)" command. The cable information is retained.
4. Click "OK" to save the changes and close the "Connectors" tab.

Result

The connector cross-references are detached. The connection object is deleted in the Navigator.

5.4 Working with the properties of two objects

Overview

If you open the properties of two objects simultaneously, you can join connectors without using the Navigator at all.

Procedure

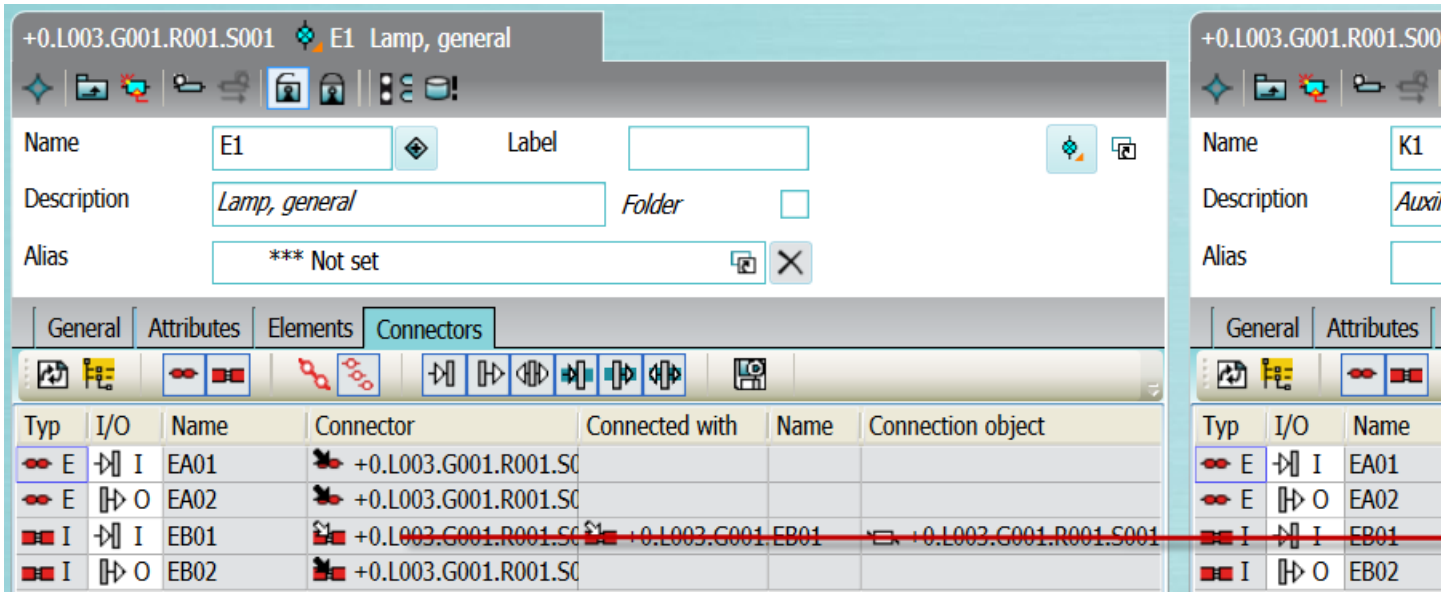
The properties of an object are closed by default as soon as you open the properties of a second object.

To open the properties of two objects simultaneously and thus to join the connectors, follow these steps:

1. Open the properties of one of the required objects.
2. Move the mouse cursor to the title of the properties.
   The "Reuse tab/window" or "Leave tab/window open" button appears.

Based on an unfixed condition, proceed as follows:

1. Click the "Reuse tab/window" button.
   The text changes from "Reuse tab/window" to "Leave tab/window open".
   The button becomes orange.
   The properties of the object are fixed open and are not closed.
2. Select the "Connectors" tab.
3. Open the properties of the second object.
4. Use the cross-shaped move cursor to drag the properties of the second object next to the properties of the first object.
   You can find more information on this topic in the "Platform Operation" manual, keyword "Move cursor".
5. Select the "Connectors" tab.
6. Drag&drop one or more connectors from the "Connectors" tab of one object to the same tab of the other object.



Result

The connection is created between the objects. The connection object is created below the collector object of the source object.

Working with two monitors

If you are working with two monitors and open the properties of two objects simultaneously, you can filter according to inputs, for example, in the properties of the first object and then filter according to outputs in the properties of the second object.

5.5 Working with the extended attributes of connectors

Opening the extended attributes of an individual connector

1. Select an object with a connector in the Navigator.
2. Open the structure below it in the Navigator.
   You see the connector objects.
3. Select the "Properties" command from the context menu of the connector object.
4. In the subsequent dialog of the contact point, click "Extended".
   The "Attributes" dialog opens.

Creating an attribute for an individual connector

1. Click the "New attribute" button in the "Attributes" dialog.
   A new line appears in the "User-defined" area.
2. Enter a value.
   - Enter a fixed value
     Click in the "Name" field and enter a name.
     Double-click in the "Value" field and enter a value.
   - Enter a list value
     Click in the "Name" field and enter a name.
     Click in the "Set reference" field and select a standard table.
     Double-click in the "Value" field and select an entry from the list.
3. Activate the "Synchronization" option if necessary.
   For information on this, refer to the next section, "Synchronizing extended attributes".

Synchronizing extended attributes

1. Click the "New attribute" button at the top left.
2. Set the values as described in the preceding section.
3. Activate the "Synchronize" option.

If the connector is connected, the opposite connector takes on the extended attribute and the value. The wiring direction plays no role here. If synchronized extended attributes have been created at both connectors, all are synchronized.

The connection is rejected if the following properties are all present:

- There are extended attributes with identical names at both connectors
- The attributes with the same name have different values
- At least one of these attributes has the "Synchronized" property

Additional information

You can find more information on this topic in the "Platform Administration" manual, keyword "Preparing extended attributes of connectors".

5.6 Connecting half-open connections via query

5.6.1 Opening the "Bulk processing of half-open connections" tab

In the menu bar, select the command "Plugins > Automation > Bulk processing of half-open connections".

See also

Looking for half-open connections via query (Page 199)

Reference of the "Bulk processing of half-open connections" tab (Page 201)

5.6.2 Looking for half-open connections via query

Requirement

- Half-open connections are drawn.
  See chapter Creating connections (Page 130).
- The "Bulk processing of half-open connections" tab is opened.
  See chapter Opening the "Bulk processing of half-open connections" tab (Page 199).

Procedure

1. Select an entry in the "Search technique" list.
   - "One level only"
     Lists the half-open connections of the objects set in the "Source" and "Target" fields.
   - "Recursive"
     Lists the half-open connections of all objects set in the "Source" and "Target" fields or at a lower hierarchical level.
2. Select an entry in the "Connector type" list.
   Evaluates the following property: Properties window of connectors, "Type" list.
   - Specific entry is selected (selection is half-open connections only): All displayed half-open connections in the "Source" and "Target" lists have the same connector type. This means that each half-open connection in the "Source" list can be connected to each half-open connection in the "Target" list.
   - "All" entry is selected: The displayed half-open connections in the "Source" and "Target" lists do not have the same connector type. The respective connector type is entered in the "Connector type" column. The list of permitted connector type combinations determines which half-open connection in the "Source" list can be connected to which half-open connection in the "Target" list.
3. Drag a unit, a location or an engineering object into the "Source" field.
4. Drag a unit, a location or an engineering object into the "Target" field.

Result

The half-open connections that belong to matching connectors are displayed in the "Source" and "Target" lists.

See also

Type check during connection (Page 134)

Connecting half-open connections via query (Page 200)

Reference of the "Bulk processing of half-open connections" tab (Page 201)

5.6.3 Connecting half-open connections via query

Requirement

The half-open connections are displayed in the "Bulk processing of half-open connections" tab.

See chapter Looking for half-open connections via query (Page 199).

Procedure

1. Drag an entry from the "Label" column of the "Source" list to an entry in the "Assigned connector" column of the "Target" list.
   The cell turns green when you select the correct field as target.
2. Optional: Click on "Update".
   Updates the tab. This also hides connections that have been edited and are therefore no longer half-open.
3. Repeat the steps for the next half-open connection.

Effect

- The connectors are connected in the "Bulk processing of half-open connections" tab.
- The connectors are connected in the Navigator.
- The connectors are connected in the report by means of references.

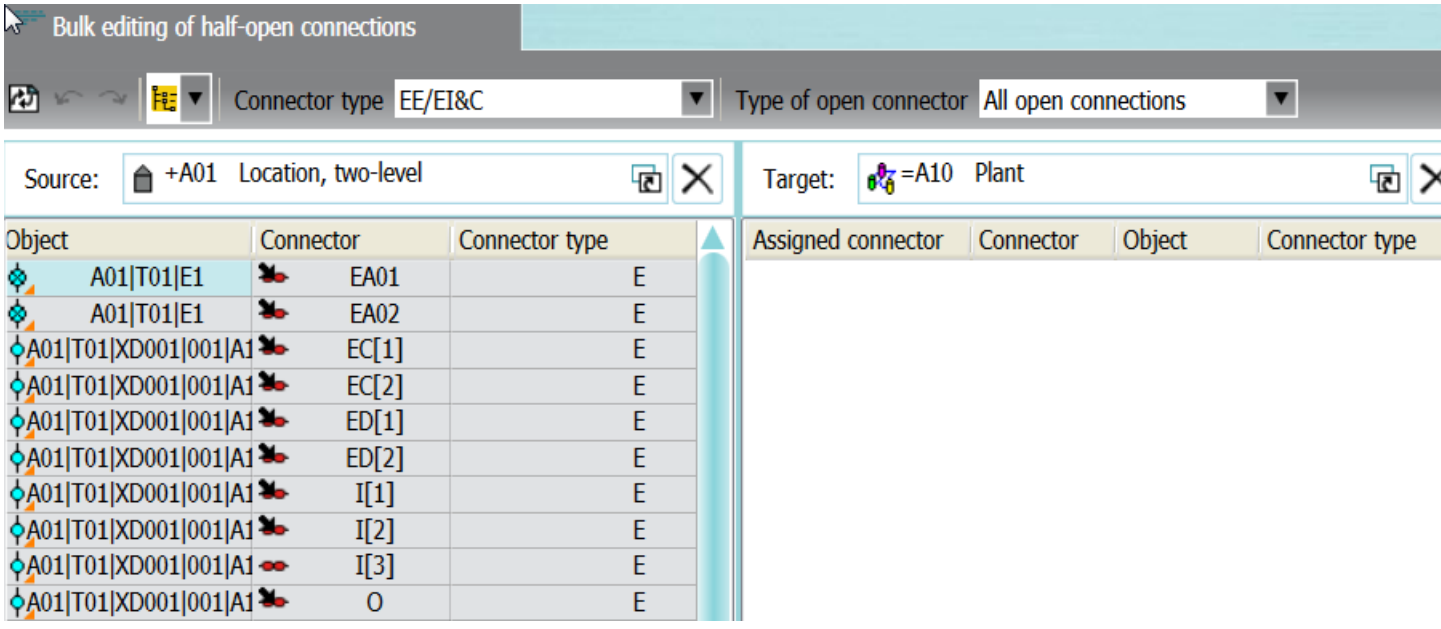
See also

Type check during connection (Page 134)

Reference of the "Bulk processing of half-open connections" tab (Page 201)

5.6.4 Reference of the "Bulk processing of half-open connections" tab

Reference



Control element	Description
"Update" button	Updates the tab. This also hides connections that have been edited and are therefore no longer half-open.
"Search technique" list	• "One level only" Lists the half-open connections of the objects set in the "Source" and "Target" fields. • "Recursive" Lists the half-open connections of all objects set in the "Source" and "Target" fields or at a lower hierarchical level.
"Connector type" list	Evaluates the following property: Properties window of connectors, "Type" list. The connector type selected here is also entered in the "Connector type" columns.
"Type of open connector" list	Select the connectors you want to display: • All open connections • Half-open connections • Reference objects
"Source"/"Target" field	Result depends on the "Search technique" list.
"Source" list: "Object" column	Owner of the connector.
"Source" list: "Connector" column	Label of the connector.
"Source" list: "Connector type" column	Connector type of the connector.
"Target" list: "Assigned connector" column	Assigned connector from the "Source" list.
"Target" list: "Assigned connector" column	Assigned owner from the "Source" list.
"Target" list: "Connector" column	Owner of the connector.
"Target" list: "Object" column	Label of the connector.
"Target" list: "Connector type" column	Connector type of the connector.

See also

Opening the "Bulk processing of half-open connections" tab (Page 199)

5.7 Using the "Connection attributes" plugin

Objective

Note

Query depending on project option

The "Connection attributes" query varies depending on the "Connection mode" project option in the category "Automation > EI&C". If the "Create connection objects for the connector types "E" and "I" mode (as of 10.2. SP1)" is selected, the query evaluates the connection objects.

The "Connection attributes" query collects all connections below a start node to edit connection attributes in bulk.

Opening a "Connection attributes" query

Select the command "Plugins > Automation > Connection attributes" from the menu bar.

The query depends on the setting of the "Connection mode" project option in the category "Automation > EI&C" "Connections" control group.

Using the "Connection attributes" query for graphical connections

1. Control the query using the icon buttons.
   You can find more information on this topic in the "Platform Administration" manual, keyword "Query for connectors".
2. Set an object in the "Start object(s)" field.
3. Edit the wiring information.
   - "Wire end handling From"
     The extended attribute "WireEnd" is set in the properties of the connector.
     For more on this, see Working with the extended attributes of connectors (Page 198).
     Examples for wire end handling are:
     Information on dismantling the connection, information on the use of plugs.
   - "From"
     RelativLabel of the source connector.
   - "Connection type"
     The extended attribute "ConnectionClass" is set in the properties of the connector.
     For more on this, see Working with the extended attributes of connectors (Page 198).
   - "Type info", "Cross section" and "Color"
     For more on this, see Assigning connection information (Page 183).
   - "Cross section"
   - "Color"
   - "Length"
     Indicates the length of a wire connection. This value is then only calculated if cabinet wiring has taken place.
   - "Wire end handling To"
   - "To"
     RelativLabel of the target connector.

Swapping connector direction

1. Select a field in the "From" column or the "To" column.
2. Select the "Connection" command from the context menu.
   - The extended attribute "Direction" is set in the properties of the connector.
   - The value in the "I/O" list remains unchanged.

Using the "Connection attributes" query for connection objects

1. Control the query using the icon buttons.
   You can find more information on this topic in the "Platform Administration" manual, keyword "Query for connectors".
2. Set an object in the "Start object(s)" field.
3. Edit the connection attributes of the connection objects in the displayed columns.

Base object of the query

The base object of the query differs depending on whether you want to edit connection objects or graphical connections.

Base object of the query for connection objects: @20|A70|Y20|M41|D50

At this base object, you can edit the pre-defined columns of the query or show and hide them.

Base object of the query for graphical connections: @20|A70|Y20|M41|B30

Using automatic labels

6

6.1 Modifying the name/label

Requirement

The IC detail engineering has been performed.

Objective

The "Modification of Name/Label" dialog is used to automatically recognize the regularities of "Label" and to edit them in numeric order.

Open the dialog

Select the COMOS menu command "Extra > Detail > Modification of Name/Label".

Setting the number of objects to be processed

1. Use drag&drop to move the object from the Navigator to the "Start object" field under which you want to search for engineering objects or documents.
2. Select one of the following options:
   - If you are searching for engineering objects, activate the "Engineering objects" option.
   - If you are searching for documents, activate the "Documents" option.
3. Click the "Search" button.
   The search results are listed in a table.

Setting a processing rule

1. Use drag&drop to move the object from the results list to the "Name" field before which a gap in the numerical order is to be inserted or closed.
   The name mask for this object is read out. The corresponding information is read out from the "System" tab and entered in the "Name" and "Label" fields.
2. To display all objects which have the same name/label mask, click the "Search" button.
   The list of results is filtered. All objects with the same name/label mask are listed in the table.

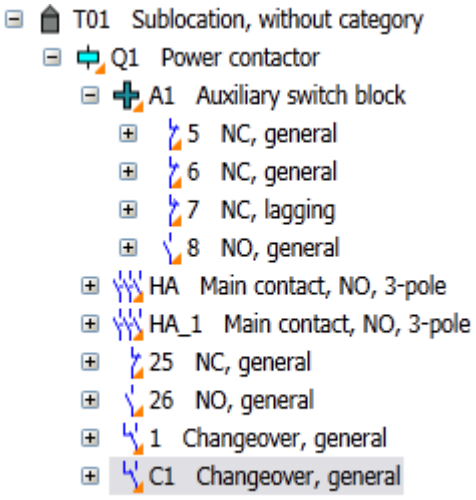
3. Select one of the following options:
   - If you want to modify the name of an object in the results list, activate the "Name" option.
   - If you want to modify the label of an object in the results list, activate the "Label" option.
     You can also activate both options at the same time.

   You can now also use the following options in the context menus of the objects in the results list:
   - "Renumber"
   - "Close gap (before object)"
   - "Insert gap (before object)"
     Instead of the context menu command you can also use the buttons in the menu bar.
4. Right-click the desired object in the results list.
5. Select the required command in the context menu.
6. Make the required settings, if applicable.
7. Click "OK" to confirm the changes.

Label number for auxiliary contacts

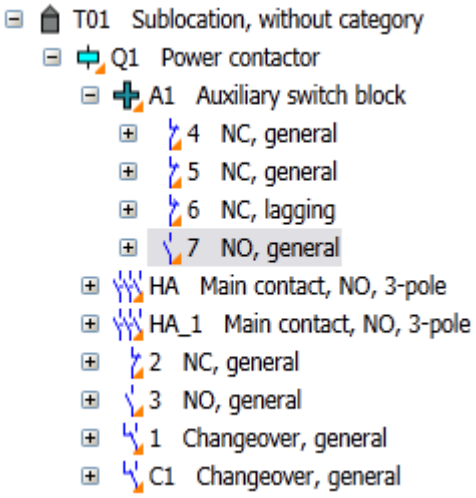
Example:

You have created a power contactor with an auxiliary switch block and the required auxiliary contacts.



To change the label of the existing auxiliary contacts, select the "Rename structure" command in the context menu of the power contactor.

All contacts with a numeric label or an empty label are renamed in descending order after the last existing label value. Contacts with alphanumeric labels are not renamed. Auxiliary contacts are not renamed.



You can find additional information on this topic in the "EI&C Administration" manual, keyword "Contact surface for relays and contactors".

6.2 Using segments

6.2.1 Objective of segments

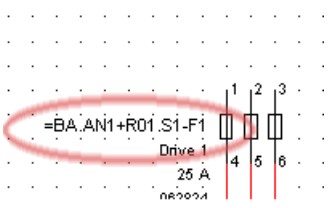
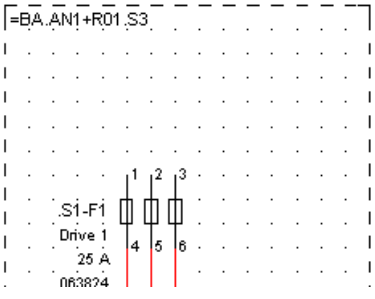
Terminology

The terms "segment" and "label segment" are used as synonyms.

Objective

Label segments are rectangular areas on circuit diagrams that group objects belonging to one location to enable clearer labeling. This abbreviation of texts at the devices also covers lines (class: "Device", subclass: "Line").

If the labeling of the segment matches the labeling of the objects within it, the labeling of the objects is displayed in abbreviated form:

without segment	with segment
	

See also

Creating and selecting a segment (Page 208)

6.2.2 Creating and selecting a segment

Requirement

A circuit diagram is opened.

See chapter Purpose of the interactive text (Page 113).

Placing identification frames

1. Select the "Identification frame" tool in the menu.
   The label segment is displayed.
2. Click a position of the report.
3. Drag out the rectangle to the required size.

A simple identification frame does not produce an engineering object.

Selecting a label segment

If a selection frame is dragged out with the mouse (multiple selection), identification frames are not selected.

Click to select an identification frame. When pressing <Ctrl>+ left mouse button to select individual identification frames, the sequence decides whether an identification frame can be selected.

Selecting the size of the segment

1. Perform two individual clicks on the identification frame.
2. Use the grab points to change the size of the identification frame.

Note

You can use the "Settings > Left continuation" or "Settings > Right continuation" command in the context menu to change the appearance of the identification frame.

Selecting the content of the identification frame

1. Make sure that nothing is selected in the report.
2. Press and hold down the "Ctrl" key.
3. Click on the identification frame.

Calculation of the device label (BMK)

The device label is located in the top left corner of the identification frame by default.

A reference object is used to calculate the device label. If the reference object is a manufacturer device, the device label of the manufacturer device is always used and not the device label of the requirement object.

See also: Purpose of device labels.

Default behavior: The first object placed is always used as the reference object.

Use one of the two options to specify the device label for an identification frame:

Option 1:

- Drag&drop the available objects from the plan or the Navigator into the identification frame.
  Result: The label corresponds to the position of the placed object in the location or plant structure.
- Place new objects from the symbol bar in the identification frame.
  Result: The label corresponds to the position in the location or plant structure connected to the current circuit diagram.

Note

Note: The first object placed is used at the reference object and highlighted in yellow for the selection of the segment.

Option 2:

1. Open the "Properties" window from the context menu of the identification frame or by double-clicking.
2. Drag an object from the "Locations" or "Plants" tab into the "reference object" field.
3. Confirm your entries.

Note

Note: As soon as a reference object has been placed manually, the object placed first will no longer be highlighted in yellow, because it is no longer used as a reference object.

The device label of the identification frame is calculated based on the position of the placed reference object.

This selection overwrites the previously placed object in the placement sequence.

If the reference object placed in the properties is deleted, it reverts to the default setting. The first object placed on the chart is used as the reference object.

See also

Creating or opening a circuit diagram (Page 95)

Properties of the segment (Page 210)

6.2.3 Properties of the segment

Requirement

- A circuit diagram is opened.
  See chapter Creating or opening a circuit diagram (Page 95).
- An identification frame is placed.
  See chapter Creating and selecting a segment (Page 208).

Settings on the report

To display the properties of a identification frame, follow these steps:

1. Select the identification frame.
2. Open the "Properties".

Label is valid for	
This control group is required in connection with abbreviated labels and has no significance for automatic placement.	
Unit	Abbreviates the label of the device to the "Unit label" part of the name.
Location	Abbreviates the label of the device to the "Location label" part of the name.
Label	Displays the abbreviated parts of the name as text on the identification frame.
Multiline output	This option creates a new line break for the text on the identification frame.
Shared amount	On the identification frame, the lowest common denominator of all the objects in the identification frame is calculated as the unit/location and only this shared amount is indicated. The device labeling becomes longer accordingly. Objects with functional classification "A680 " are not taken into account when calculating the shared amount. For example, placeholders of grey boxes.

Sheet zone	
This control group is required in connection with automatic placement and has no significance for the abbreviated label.	
Name	The name is used as a target during automatic placement. The "Name" list contains a number of prepared names that are typically used in the E&IC field. The name of an identification frame and therefore sheet zone name must be unique for each report.

Base object	
(reference field)	A base object of the identification frame can be set here. If the field remains empty or the base object cannot be found, COMOS uses only the basic functions of an identification frame.

Reference object	
(reference field)	A reference object of the identification frame can be set here. If the field remains empty, COMOS uses the first automatically found object in the identification frame on the report as the reference object.

You can find additional information on this topic in the "EI&C Administration" manual, keyword "Identification frame".

Drawing single line diagrams

7

7.1 Creating a single line diagram

Requirement

The IC basic engineering has been performed.

Creating a single line diagram

Single line diagrams are part of the EI&C detail engineering. In the COMOS standard model, the single line diagram is in the "Locations" tab at the sublocation:

1. Select the sublocation.
   In the COMOS standard model, the following alternatives are available:
   - Site > Building > Floor > Sublocation, unspecified
     or
   - Location > Sublocation, w/o folder
2. Select "New > FA.xxx Circuit diagram, single-line display"

7.2 Construction on single line diagrams

Requirement

A single line diagram is opened.

Single line symbols

By default, the libraries stored in COMOS have symbols for circuit diagrams and single line diagrams. You can place the same objects on a circuit diagram and on a single line diagram.

Selecting or creating power supplies

1. Select your potentials or supplies in the "Locations" tab.
2. Alternative in the COMOS standard model:
   - Select the subunit in which your circuit diagram is located.
   - Select "New > A30 Potentials/power bars > A20 L1/L2/L3".
3. Drag the potential/supply onto the report.

Selecting or creating fuses

1. Select your three-pole fuse on the "Locations" tab.
2. Alternative in the COMOS standard model:
   - Select the subunit in which your circuit diagram is located.
   - Select "New > A10 Devices > A40 Protection devices > A10 Fuse > A30 Fuse, 3-pole".
3. Drag the fuse onto the report.

Selecting or creating devices

Create the following devices as an example:

1. "Generator", nominal power 50 KW
2. Two three-phase motors, each with nominal voltage of 400 volts
3. Transformer, nominal voltage 400 volts and output voltage 230 volts
4. Heat source, nominal voltage 230 volts

Result

The single line diagram has been created.

Load estimation with single line diagrams

8

8.1 Objective of the load estimation

Aim

You can calculate the performance of single devices and the load dimensioning of networks.

The basis for this calculation is a single line diagram, in which the objects that are involved in the calculation are displayed and connected with one another.

The following options are available:

- Performance calculation single device
- Load calculation network
  The entire load that is drawn by all connected consumers is calculated for potential rails, transformers, and power supplies. The consumers must be connected to the potential rails with the attribute "Assigned energy bus".
- Interconnected subnetworks
  Depending on the switch setting, this function calculates the overall loads from networks which are interconnected with one another.
- Network checking
  Validity check for all objects that are connected in the network.

Supported device classes

- Generators
- Motors
- Consumers
- Transformers
- Power rails (potentials)

The use of single line diagrams is useful, but not mandatory.

8.2 Individual calculation of an object

8.2.1 Conducting a single calculation

Procedure

1. Open the properties of the required device.
2. Switch to the "Attributes > Technical data" tab.
3. Enter the required information in the fields of the "Input calculation values" control group.
4. Click the "!" button to calculate any missing data.

Result

The results of the calculation are displayed in the fields of the "Calculation" control group.

8.2.2 "Technical data" tab

You perform the single calculation for estimating a load on the "Technical data" tab. The design of the tab is described in the following using the sample of a motor. In the case of simpler types of consumer, such as lights or air-conditioning systems, fewer control elements will be displayed on the "Technical data" tab.

"Input calculation values" control group

The following table describes the control elements of the "Input calculation values" control group on the "Technical data" tab:

Control element	Description
"Product type" list	You select the product type from this list. Examples of product types: • "Motor" • "Generator" • "Consumer"
"Current type (input)" list	You select the current type from this list. Examples of current types: • "Direct current" • "Alternating current" • "Rotary current"
"Nominal voltage" list	You select the nominal voltage from this list. This is the producer description for the current.
"Nominal current" field	You enter a value for the nominal current in this field. This is the customer description for the current.
"Nominal power" field	You enter a value for the nominal power in this field. This is the manufacturer's specification of the nominal power.

Control element	Description
"Output current" field	You enter a value for the output current in this field. This is the actually by the machine required current.
"Output power" field	Via this field you can enter the value for the output voltage. This is the by the machine actually required performance.
"Power factor" field	The relationship between the amount of active power P to the apparent power S is identified as the power factor. The power factor can be between 0 and 1.
"Efficiency" field	You enter a value for the efficiency in this field. This value is a producer description.
"Runtime factor" field	You enter a value for the runtime factor in this field. The runtime factor states the average runtime of the machine. This means that the machine will run 24 hours a day if you enter 100%, for example.

"Calculation" control group

The following table describes the fields of the "Calculation" control group on the "Technical data" tab:

"Consumption" column	"Nominal values" column	"Output values" column	"Runtime values" column
"Current" field	Calculates the current consumed by the machine.	Calculates the actual current consumption of the machine.	Calculates the average current consumption of the machine.
"Power" field	Calculates the power consumed by the machine.	Calculates the power consumption of the machine.	Calculates the power consumption of the machine.
"Apparent power" field	Calculates the apparent power consumed by the machine.	Calculates the actual apparent power consumption of the machine.	Calculates the average apparent power consumption of the machine.
"Reactive power" field	Calculates the reactive power consumed by the machine.	Calculates the actual reactive power consumption of the machine.	Calculate the average reactive power consumption of the machine.

8.3 Power estimate in the network

Objective

You can perform a load estimation for the entire network. The load estimation is shown below, using the example of a two-level network which consists of a generator, three consumers (one heater and two motors), two potentials and one transformer.

Requirement

- A supply ("potential"), generator and motor are placed on a single line diagram and connected with each other.
- Values for nominal voltage, nominal power and nominal current are entered for the generator and the motor.

- The infeed ("potential") for the generator and the motor is assigned as energy bus using the attribute of the same name.
- The single line diagram is open.

Calculating the load at the potential

To engineer the network, follow these steps:

1. Open the properties of the potential.
2. Select the "Technical data" tab in the properties.
3. To start the calculation for the potential, click the "!" button.
   The screen displays how much current the generator is feeding into the potential and the load which is actually being consumed at the potential.

Simulating overload

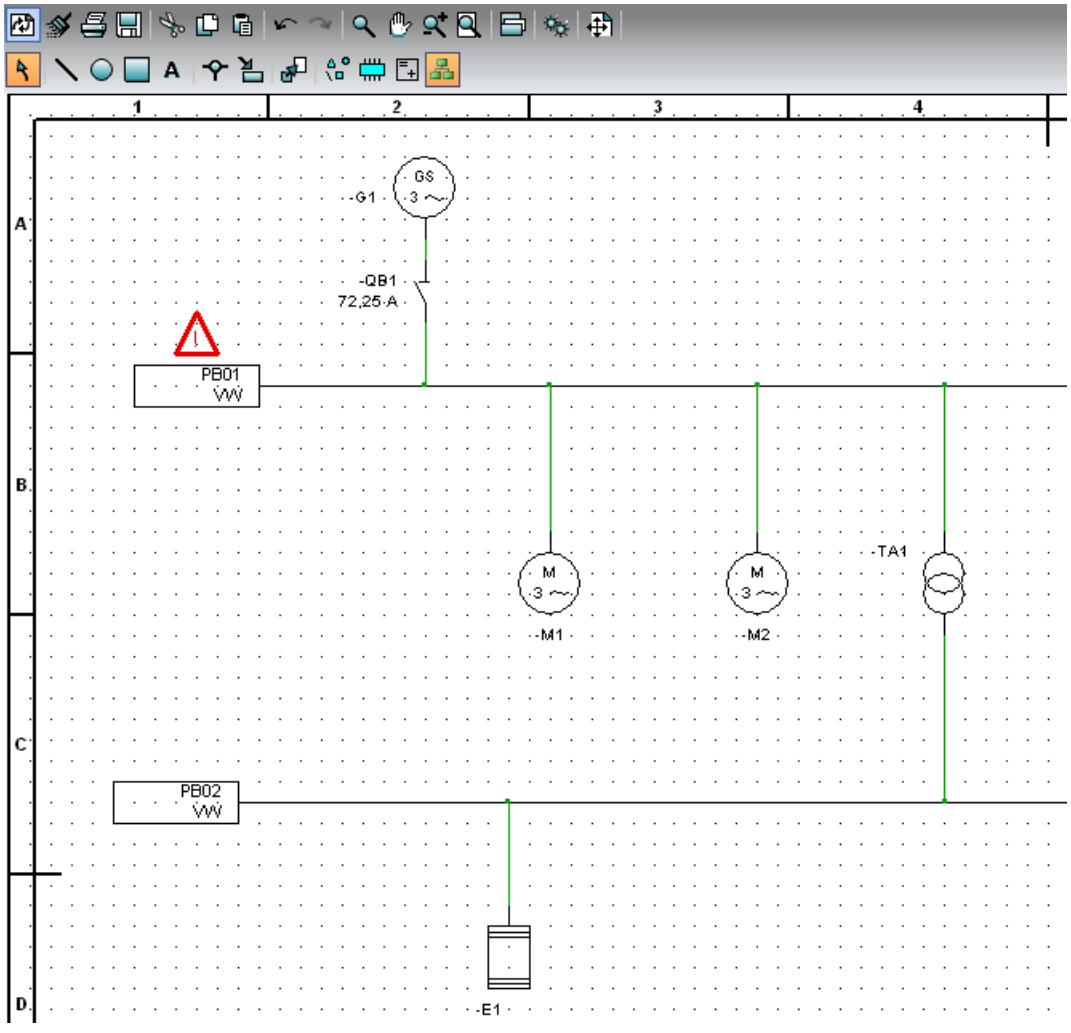
1. Equip the network with a second motor.
2. Open the properties of the potential.
3. Click on the "Technical data" tab.
4. To start the calculation, click the "!" button.
   You now see that the amount of received current is getting closer to the amount of released current. At the same time, a yellow warning signal is displayed on the single line diagram. The warning signal always appears if the released current uses more than 90% of the incoming current or incoming power.
5. Place the following objects:
   – a transformer
   – a heating source
   – another potential
6. Connect the heating source to the second potential.
7. Connect the second potential to the first via the transformer.
8. Open the properties of the second potential.
9. Select the "Technical data" tab.
10. To start the calculation, click the "!" button.
    You will see that the amount of current is greater than the amount of received current.
11. Open the properties of the first potential.
12. Select the "Technical data" tab.
13. To start the calculation, click the "!" button.

Result

You see that both the dedicated consumers and the derived ones are connected to the potential. This means that the values for the released current and power exceed the values for the incoming current and power. The potential is overloaded.

At the same time, a red signal is automatically displayed on the potential in the single line diagram.

Example



Object-based editing of cables and wires in the database

9

9.1 Structuring of cable structures

Requirement

A cable has been created in the Navigator.



Structuring of the cable structure

The structures for cables are structured as follows:

- There is single object that represents the cable (as a rule, the cable itself has no connectors).
- There is an object located hierarchically below each wire.
- Optional: One or more objects are located hierarchically below the shielding.
- Optional: There is one or more hierarchically nested objects for the stranding.

Overview of object-based editing options for cables

- "Wires" tab
  The "Wires" tab is available at the object of the cable.
  This tab provides an overview of the connectors of wires that belong to the cable. The "From" and "To" column indicate the connected objects.
- "Wire list" query
  You can create the "Wire list" query below the cable. The query provides an overview of the connectors of wires that belong to the cable. The "From" and "To" column indicate the connected objects.

You can also edit the cables and wires on the report. For more on this, see Cables and wires on the construction display (Page 170).

9.2 Creating a shield

Requirement

A cable has been created.

Procedure

1. Select the cable in the Navigator.
2. Select the command "New > Shield" in the context menu.

9.3 Editing cables in the "Wires" tab

9.3.1 Opening the "Wires" tab

Requirement

- The IC detail engineering has been performed.
- A cable has been created.

Procedure

1. Select the cable in the Navigator.
2. Select the "Properties" command in the context menu.
3. Open the "Attributes > Wires" tab.

There is an entry (a row) for each wire and shield. Two connectors are displayed per row: One connector for the source ("From") and one connector for the target ("To").

9.3.2 Working with the "Wires" tab for the cable

General

This tab is only available for cables.

Dissolving a cable pointer

1. Select one or more lines.
2. Select the command "Remove reference" from the context menu in the "Cable" column.

Swapping wires

1. Select two lines.
2. Select "Swap" from the context menu of any column.
   Invalid assignments are rejected by COMOS. Undefined categories are valid as well as the same categories. The "OnCableSet" event is not deleted when wires are swapped.

Dissolving a wire

1. Select the desired line.
2. Select "Dissolve wire" from the context menu of any column.
   The "From" and "To" cells are empty for this line. The connection is still available in the engineering data.

Dissolving a connection

1. Select the desired line in a column.
2. Select "Disconnect connection" in the context menu.
   The "From" and "To" cells are empty for this line, the connection is dissolved, the wire at the connectors is removed.

Changing the connection direction

1. Select the desired line in the ""From" or "To"" column.
2. Select "Change connection direction" in the context menu.
   The "From" and "To" cells are swapped. Where possible, the extended attribute "Direction" is also set accordingly in the connectors.

Creating a connection to another wire

1. Select the desired line in the ""From" or "To"" column.
2. Select "Cut" from the context menu.
3. Select the desired line in the ""From" or "To"" column.
4. In the context menu, select "Paste".

9.3.3 Working interactively with the "Wires" tab for the cable

Editing wires using drag&drop

To edit the "Wires" tab using objects from the Navigator, follow these steps:

1. Select the target number in the "Wires" tab.
2. Select the same number of objects in the Navigator.
3. Drag&drop the objects from the Navigator into the "Wires" tab.

4. A context menu appears with the following entries:
   - All in list
   - All connected
   - All not connected
   - "Input"
   - "Output"
   - "Neutral", possibly
5. Make a selection in the context menu.

Invalid assignments are rejected by COMOS.

Result

A connection object is created.

9.4 Editing cables in the "Wire list" query

9.4.1 Creating a "Wire list" query

Requirement

- The IC detail engineering has been performed.
- A cable has been created.

Creating and opening a query

1. Select the cable in the Navigator.
2. Select the command "New > Wire list" in the context menu.
   The "Wire list" query can be created only once below a terminal strip. Once the query is created, the "New > Wire list" command is no longer available.
3. To open the query, double-click on the object in the Navigator.

You can find information on administration in the "E&IC Administration" manual, keyword "Base objects of the queries".

9.4.2 User interface reference

"Wire list" query

Control element	Description
"Start object" field	Start object of the search.
"Location" column	If an object from the "Locations" tab is set as the start object in the Navigator, this column shows the location.
"Unit" column	If an object from the "Units" tab is set as the start object in the Navigator, this column displays the unit.
"Cable" column	Displays the cable containing the wires.
"Stranding" column	When stranding has been created, this column shows the stranding.
"From" column	Takes the extended "Direction" attribute into consideration. The default setting for this column is the connector with the property "Output". The connector is not located below the cable, but rather below the collector object for the connection object of the device. Example: The connector is located under the collector object for connection objects of a terminal strip.
"Wire" column	The set wire.
"To" column	Takes the extended "Direction" attribute into consideration. The default setting for this column is the connector with the property "Input". The connector is not located below the cable, but rather below the collector object for the connection object of the device. Example: The connector is located under the collector object for connection objects of a terminal strip.

Context menu of the query

See chapter Working with the "Wires" tab for the cable (Page 222).

9.4.3 Using a "Wire list" query

Set start object

1. Drag&Drop an object or a selection of several objects from the Navigator into the "Start object" field.
   Permitted:
   - Cable
   - Unit
   - Location
   - Project

   Starting with the start object, the cables below are listed with their wires.
   Cables below cables are not possible.

Editing wires

1. Select one or several rows.
   There is an entry (a row) for each wire and each shield object. Two connectors are displayed per row: A connector to the inside (connection inside) and a connector to the outside (connection outside).
2. Proceed as described here: See chapter Working with the "Wires" tab for the cable (Page 222).
   The commands are only available in the context menu in the "Wire list" query.

Interactively editing cables in the query

1. Select one or several rows.
   There is an entry (a row) for each wire and each shield object. Two connectors are displayed per row: A connector to the inside (connection inside) and a connector to the outside (connection outside).
2. Select one or more objects in the Navigator.
3. Proceed as described here: See chapter Working interactively with the "Wires" tab for the cable (Page 223).
   The commands are only available in the context menu in the "Wire list" query.

9.5 Changing wire types

Overview

From a database point of view, the information on the cable is not part of the logical chain between the devices. The information on the cable is managed by means of a reference. This means that you can change the cable type at any time without having to adapt devices or connectors.

Procedure

1. Open the properties of the cable in the engineering project.
2. Select the "General" tab.
3. Select the "Base data" tab in the Navigator.
4. Drag&drop the base object of the required wire type into the "Base object" field.

Result

The wires are reassigned from top to bottom. If possible, the information is retained. If the new cable type has fewer wires than were previously assigned, then information will be lost.

9.6 Working with cable drums

Requirement

- Cables have been created.
- At least the attributes: "Cable type" and "Cable cross-section" are entered on the tab "Attributes > Technical data".

Procedure

1. Navigate to the base object of the cable drum: "@10 > A50 > A20 > B40 > A60"
2. Drag&drop the base object on the desired object in the "Locations" tab in the Navigator. For example onto the partial location.
3. Open the properties of the cable drum.
4. Enter the data of the cable drum in the "Cable drum design" control group.
5. Click the "Calculate" button.

6. Drag a cable from the Navigator into the "assigned cable".
   The fields of the "Cable specification" control group are completed automatically.
7. Confirm your entries.
8. Open the properties of the assigned cable.
9. Open the "Attributes > Technical data" tab.
10. Drag&drop the cable drum from the Navigator to the "Assigned cable drum" field in the "Cable data" control group.
11. Enter a value for all the cables in the "Base length (input)" field.
12. Confirm your entries.

Result

The added values of the assigned cables are displayed in the properties of the cable drum in the "Technical data" tab. All the assigned cables are listed under the "Utilization" control group. The total length of the cables is calculated and is displayed in the "Cable drum utilization" control group.

The assigned cables are displayed in the Navigator below the cable drum.

Query "Cable drum assignment"

Base object: "@20 > A70 > Y20 > M41 > D30 Cable drum assignment"

Use this query to assign a cable drum to the cables.

The "Cable type" and "Cable cross-section" properties must be entered in the properties of the cables. The properties of the cables are write-protected and cannot be edited in the query.

One of the following conditions must be fulfilled so that you can assign the cable drum:

- The cable drum is empty.
- If a cable is already assigned to the cable drum, the following conditions must be met:
  - The cable type of the assigned cable and of the new cable agree
  - The cable cross-section of the assigned cable and of the new cable agree
  - The maximum weight of the cable drum is not exceeded
  - The maximum cable length of the cable drum is not exceeded
    If the remaining length is negative, the maximum length is probably not entered at the cable drum and a calculation is not possible.

Changes in the properties of the cables, the cable drum or their assignment to each other result in automatic recalculation of the attribute values at the cable drum.

Query "Cable list at drum (EI&C)"

Base object: "@20 > A70 > Y20 > M41 > D40 Cable list at drum (EI&C)"

Sorts the cables by their length. In order to create the query below the cable drum on the "Locations" drum, drag&drop the base object onto the cable drum. Start the search subsequently.

Editing the cable list in Excel

If you want to edit the query "Cable list at drum (EI&C)" in Excel, the cable drum has to be created in the Navigator in a location structure.

1. Use the COMOS default functionality for the export.
2. To edit the cable list deactivate the write protection in Excel.
3. Edit solely the "Assigned cable drum" column.
4. Select the base object "@20 > A90 > M41 > A10 Excel import for cable drum".
5. Create the object in the location structure.

6. Open the object.
7. Select the desired Excel file.
8. Define a folder for the log file in the script.
   Example: "LogFilePath = "c:\temp\"
9. Select the lines that you want to import in the preview.
10. Click the "###" button in the menu bar of the symbol script.

The following actions are executed during importing:

- Creating and assigning new cable drum objects
- Changing cable drums that are already assigned
- Deleting the assignment to a cable drum
- Checking the assignment for validity
- Listing of all the executed actions in a log file

Object-based editing of terminal strips in the database

10

10.1 Design of terminal strip structures

Requirement

A terminal strip has been created in the Navigator.



Design of terminal strip structure

The structures for terminal strips are designed as follows:

- There is a single object that represents the terminal strip. The terminal strip in the COMOS standard model does not have separate connectors.
- There is a component located hierarchically underneath each terminal strip which represents the installation device. The installation device in the COMOS standard model does not have separate connectors.
  This component is placed on the report.
- As a rule, the terminal contacts are located below the installation device.

The three levels can only be used together. It makes no sense to create a terminal strip without an installation device; nor does it make sense to use an installation device without a terminal strip or terminal contacts.

Overview of object-based editing options for terminal strips

- "Terminal strip" tab
  The "Terminal strip" tab is available at the object of the terminal strip.
  This tab provides an overview of the connectors of the installation devices or terminals that belong to the terminal strip. See chapter Editing terminal strips in the "Terminal strip" tab (Page 232).
- "Terminal strip editing" query
  You can create the "Terminal strip editing" query below the terminal strip. This query provides an overview of the connectors of the installation devices or terminals that belong to the terminal strip. See chapter Editing terminal strips in the "Terminal strip editing" query (Page 237).

You also have the option of generating bridges via a report.

Multideck terminals

Multideck terminals can only be connected in a certain way.

10.2 Creating a terminal block

Requirement

A terminal strip has been created.

Procedure

1. Click on the terminal strip in the Navigator and select the "New" command in the context menu.
2. Select a terminal block or a terminal installation device.

10.3 Editing terminal strips in the "Terminal strip" tab

10.3.1 Open the "Terminal strip" tab

Requirement

- The IC detail engineering has been performed.
- A terminal strip has been created.

Procedure

1. Select the terminal strip in the Navigator.
2. Select the "Properties" command in the context menu.
3. Open the "Terminal strip" tab.

10.3.2 Editing Cable and Connections on the "Terminal strip" tab

Requirement

- Connections have been created.
  See also chapter Editing connections in the Navigator (Page 194) and chapter Working interactively with the "Wires" tab for the cable (Page 223).

Editing a connection

1. Select one or more lines in the "Terminal strip" tab.
   Each line represents a terminal.
2. Open the context menu in the "Connected with" column.

Select one of the following options:

Entry in context menu	Description
"Delete connection"	Dissolves the connection and deletes the connection information, including the wire information.
"Dissolve connection (keep connection object at terminal)"	Dissolves the connection. The connection object is retained at the terminal.
"Dissolve connection (keep connection object at partner)"	Dissolves the connection. The connection object is retained at the partner.
"Memorize (keep connection object at terminal)"	Only the information with whom there is a connection is transferred.
"Memorize (keep connection object at partner)"	Connection object is transferred along with all information stored there.
Connect	Creates a connection

Laying cable or stranding

1. In the navigator, select a cable or stranding with wires that you want to place on the terminal strip.
2. Drag-and-drop the cable or stranding to the "Over" field for a terminal.
   The context menu opens and shows you all the wires, shields and transposition of your selected cable.
3. Select individual wires or shields, "all free" or "all free" (except shields).
   Note: If you have not used shields, the option does not appear in the context menu.

Editing a wire

1. Select one or more lines in the "Terminal strip" tab.
2. Open the context menu in the "Over" column.

Select one of the following options:

Entry	Description
"Dissolve wire"	Dissolves the wire.
"Cut"	Copies the wire to the clipboard.
"Connection information..."	Connection information is a simple, alternative technique to wires. A connection can only have either a wire or connection information.

10.3.3 Editing connectors and auxiliary connectors in the tab

Input and output - exchanging connection

Select the "Exchange I/O connections" command of the desired line in the "Terminal" column of the "Terminal strip" tab in the context menu.

The command swaps the input and output or inside and outside. For this, the left and right columns for "Connected with" and "Connected over" are swapped within a row. The action is only performed if the result is logical.

Where possible, the extended attribute "Direction" is also set accordingly in the connectors.

Creating auxiliary connectors

1. Select exactly one line in the "Terminal" column of the "Terminal strip" tab.
2. Select "Create auxiliary connectors" from the context menu.

Deleting free auxiliary connectors

1. Select any line in the "Terminal" column of the "Terminal strip" tab.
2. Select "Delete free auxiliary connectors" from the context menu.

Displaying auxiliary connectors

If you have created auxiliary connectors, you can use the context menu to determine if all auxiliary connectors should be displayed or only connected auxiliary connectors.

10.3.4 Editing bridges in the tab

Introduction

Bridges connect terminals, so you can only set them within a terminal strip. Between the various terminal strips there are only connections, not bridges.

Tests on connectors to be bridged

Terminal connectors are first checked for the "IO" attribute. Here "I" can be entered for "Input" and "O" for "Output".

Note

In the "Terminal strip" tab, "I" is entered in the "Connectors" column on the left and "O" in the "Connectors" column on the right.

The test of the "IO" attribute is independent of the name of the connector. This means I1, I2 can be used as well as IA/IB1 or IA/IB2 . This enables correct assignment of the distributor terminals as well.

In a neutral case, i.e. terminal.IO = "N", the decision regarding "input" and "output" is based on the name "I" or "O" or "I1", "I2", "O1". The name is checked up to the parenthesis. As a result, for example, I1 and I1[1] are bridged, but not I1 and I2. This enables correct assignment of the auxiliary connectors as well.

If an assignment still cannot be made, the connector is listed in the error list.

Creating bridges

1. Open the properties of the terminal strip.
2. Select the "Terminal strip" tab.
3. Select the terminals which you want to connect via a bridge.
   Two bridge categories are managed for each terminal: an internal bridge and an external bridge.
4. Select one of the following options:
   - If you want to create an internal bridge, right-click in one of the three unnamed columns in front of the "Terminal" column and select the "New bridge (internal)" command in the context menu, followed by the required bridge type.
   - If you want to create an external bridge, right-click in one of the three unnamed columns after the "Terminal" column and select the "New bridge (external)" command in the context menu, followed by the required bridge type.

A connection object is created. The bridge is created below the object.

Selecting a bridge

You have the following options for selecting the bridge:

- Click on an element of the bridge and select the "Select bridge" command from the context menu.
- Double-click on an element of the bridge.

A selected bridge is displayed in violet. Calling the function again deselects the bridge.

Expanding a bridge

1. Select the bridge you want to expand.
2. Click on a line in the "Terminal strip" tab that is not yet in the bridge.
3. Select "Paste to bridge" from the context menu.

Reducing or dissolving bridges

1. Select an element of the bridge you want to edit.
2. To remove the selected element from the bridge, select the "Remove from bridge" command from the context menu.
3. In order to dissolve the bridge completely, select "Unbridge" from the context menu.

Grouping bridges

1. Select the first bridge.
2. Select additional bridges using the context menu or "Ctrl"+double-click.
3. Select "Add up bridges" in the context menu.

A single bridge containing all bridge connectors is created out of the bridges of the same type. All the bridges to be grouped must be selected.

Changing the bridge type

1. Select an element of the bridge you want to edit.
2. Select "Change bridge type" in the context menu.

This option is only available if you right-click precisely on a bridge that had been created. The user must distinguish between internal and external bridges.

Bridge information

1. Select an element of the bridge you want to edit.
2. In the context menu, select "Bridge information".

Coloring bridges

You can define colors for bridges.

1. Open the "Y10 > M41 > A10 > Y10M41N00001 Bridge types" standard table.
2. Enter a value in the "Value 2" column that represents the color.
   You can find all available values and the corresponding colors in the "Y10 > M00 > A10 > Y10M41N00004 RGB colors" standard table.

10.3.5 Interactive editing of the "Terminal strip" tab

Setting connections using drag&drop

1. Select the target number in the "Terminal strip" tab.
2. Select the same number of objects in the Navigator.
3. Drag&Drop the objects from the Navigator into the "Terminal strip" tab.
   A context menu appears with the entries "Input", "Output" and possibly "Neutral".
4. From the context menu, select whether you want to set the input or output.

Invalid assignments are rejected by COMOS.

10.4 Editing terminal strips in the "Terminal strip editing" query

10.4.1 Creating a "Terminal strip editing" query

Requirement

- The IC detail engineering has been performed.
- A terminal strip has been created.

Procedure

1. Select the terminal strip in the Navigator.
2. Select the "New > Terminal strip editing" command in the context menu.
   The "Terminal strip editing" query can be created only once below a terminal strip. Once the query is created, the "New > Terminal strip editing" command is no longer available.
3. Double-click the query.

10.4.2 Using a "Terminal strip editing" query

Structure of a "Terminal strip editing" query

There is an entry (a row) for each terminal. Two connectors are displayed per row: A connector to the inside (connection inside) and a connector to the outside (connection outside).

Auxiliary circuit connections can also be displayed. Bridges are not evaluated.

Comparison of the "Terminal strip" tab and the "Terminal strip editing" query

The "Terminal strip" tab contains special terminal strip functions in both the context menu and toolbar.

In the "Terminal strip editing" query, terminal strip functions are only available in the context menu.

Editing bridges

For more on this, see chapter Editing bridges in the tab (Page 234).

Editing wires and connections

For more on this, see chapter Editing Cable and Connections on the "Terminal strip" tab (Page 232).

Editing auxiliary connectors

For more on this, see chapter Editing connectors and auxiliary connectors in the tab (Page 233).

Interactive editing of a "Terminal strip editing" query

For more on this, see chapter Interactive editing of the "Terminal strip" tab (Page 236).

10.5 Import terminal data

"Generic Excel import" interface

Terminal data is imported using the "Generic Excel import" interface.

See "Automation Interfaces", keyword "Generic Excel import".

10.6 Using connection patterns

10.6.1 Purpose and requirement of the connection pattern

Connection patterns specify the way in which device connectors may be connected with terminal strips.

A device connection consists of several COMOS connectors. The number, sequence and connector subtypes of the COMOS connectors must be specified.

Requirement

You can find more information on this topic in "EI&C Administration", keyword "Administering connection patterns".

10.6.2 Principle of connection patterns

Assignment rules by types and subtypes

Before two COMOS connectors are connected, there is always a check to ensure that the connector subtypes match. See also Type check during connection (Page 134).

This mechanism is also effective if a device is connected with a terminal strip: The connections of the device connector are applied and checked one after the other to see whether the subtype of the terminal matches and whether the connector of the terminal has not been connected yet.

If the terminal currently being checked is invalid, the next terminal is checked. If a valid terminal is found, COMOS attempts to connect the next connection of the device connector with the terminal strip.

Because each connection of the device connector is checked individually, inconsistencies may occur.

If a terminal is connected with the "+" connector at the I connector, for example, it is technically possible without the connection pattern for the terminal to be connected with the "-" connector on the output side of the O connector.

Additional assignment rules with connection pattern

A connection pattern creates a check of COMOS connectors together.

Example pattern:

Connector label	EA1	EA2	EA3	EA4
Connector subtype	+	-	N	#

Abbreviation of subtypes:

- Wire: +, -, A, B, C, D
- Normal: STD
- Neutral: N
- Protection: PE
- Bridge: @B
- Shield: SH
- Reserve terminal: #
  Reserve terminals are not connected.
- Any: *

If a connection pattern is used, connection of the device connector is only possible if the same pattern is available on the opposite side (with the terminal strip).

The following sequence applies here:

- Device connector to terminal strip connection
  - First, an exact assignment is sought. The source pattern and the target pattern in this step are completely identical.
  - A partial assignment is sought if an exact assignment is not possible. The target pattern is more comprehensive than the source pattern in this case. The source pattern fits into the target pattern without changes, but there are still some free COMOS connectors in the target pattern.
- Terminal to terminal connection
  - An exact assignment is sought. The source pattern and the target pattern in this step are completely identical.

10.6.3 Basic rules of connector assignment

- A connection pattern is only assigned completely. If fewer terminals are free than are required for the connection pattern, the terminals will remain unassigned.
- The assignment of a connection pattern overwrites existing connection patterns. An existing connection pattern will be removed, if necessary, to process the assignment of a new connection pattern.
- The "Pattern description" is entered in the "Assigned pattern" column.
- An entry is set in the "Subtype" column according to the connection pattern.

10.6.4 Assigning connection patterns using buttons

Requirement

- The "Connection pattern definition" tab is open.
- The templates of the connection patterns have been prepared.
- Terminal blocks have been created at the terminal strip.

Assigning a connection pattern for the terminal range

1. Enter the terminal range to be edited in the "Terminal range from no." and "to no." fields.
   If one of the two fields has a red border, an invalid value has been entered.
   In this case, the last value recognized as valid is used in this field.
2. Select a row in the "Patterns" table.
   Make sure that you only select one row.
   All rows with a green background are selected. Click the row again to remove it from the selection.
3. Click the "Assign pattern" button.
   Starting with the first terminal in the terminal range, all terminals are assigned in sequence according to the connection pattern.
   Any existing selections in the "Terminal strip" table are ignored.

Assigning several connection patterns for a terminal range

1. Enter the terminal range to be edited in the "Terminal range from no." and "to no." fields.
   If one of the two fields has a red border, an invalid value has been entered.
   In this case, the last value recognized as valid is used in this field.
2. Select several rows in the "Patterns" table.
   All rows with a green background are selected.
   The connection patterns are selected with the first click and deselected with another click.
3. Click the "Assign pattern" button.
   Starting with the first terminal in the terminal range, all terminals are assigned in sequence according to the different connection patterns.
   Any existing selections in the "Terminal strip" table are ignored.

10.6.5 Assigning connection patterns with drag&drop

Requirement

- The "Connection pattern definition" tab is open.
- The templates of the connection patterns have been prepared.
- Terminal blocks have been created at the terminal strip.

Assigning a connection pattern with drag&drop

1. Optional: Select several subsequent rows in the "Terminal strip" table.
2. Select a row in the "Patterns" table.
   Make sure that you only select one row.
   All rows with a green background are selected. Click the row again to remove it from the selection.
3. Drag&drop the row from the "Patterns" table into the "Terminal strip" table.
   - If there is no multiple selection in the "Terminal strip" table, a row is selected during the drag&drop process. The selection of the row moves when you move your mouse.
     The connection pattern is applied once starting at this row.
   - If there is a multiple selection in the "Terminal strip" table, the following applies:
     If the multiple selection is larger than the connection pattern, the connection pattern is applied as often as possible within the selection.
     If the multiple selection is smaller than the connection pattern, the connection pattern is applied once and extends from the range of selection.

Assigning multiple connection patterns with drag&drop

1. Optional: Select several subsequent rows in the "Terminal strip" table.
2. Select several rows in the "Patterns" table.
   The connection patterns are selected with the first click and deselected with another click in the "Pattern" column.
3. Proceed as described in the "Assigning a connection pattern with drag&drop" section.

10.6.6 Copying connection patterns

"Pattern" XML file

The assignment of connection patterns is saved as an XML file below the terminal strip. The default name is "Pattern". You can rename the XML file. Only one XML file is recognized below each terminal strip.

Copying the "Pattern" XML file

1. Copy the "Pattern" XML file.
2. Paste this file into another terminal strip.
3. If this additional terminal strip has a sufficient number of terminals, the connection patterns are transferred.

10.6.7 Applying connection patterns to several terminal strips

Procedure

1. Open the "Connection pattern definition" tab as follows:
   COMOS menu, "Plugins > Automation > Connection pattern definition"
2. Set the start object.
3. Drag&drop a terminal strip from the Navigator or from the list into the "Terminal strip (default)" field.
   The terminals in the default terminal strip are displayed in the list.
4. Edit the connection patterns as described in Assigning connection patterns with drag&drop (Page 241).
5. Select one or more entries in the list.
6. Click the "Assign" button.

10.6.8 Removing connection patterns

1. Open the properties of the terminal strip.
2. Open the "Connection pattern definition" tab.
3. Click on the "Assigned pattern" column in the left table.
4. Select "Remove all mappings" in the context menu.
   All connection patterns in the left table are removed.

10.6.9 Using connection patterns in the Marshalling designer

The Marshalling designer automatically evaluates connection patterns.

The operation of the Marshalling designer remains unchanged. For more on this, see Working with several terminal strips and connection patterns (Page 45).

Object-based editing of potentials in the database 11

11.1 Overview

COMOS distinguishes between two different types of potentials:

- Physical potentials (rails)
- Logical potentials (potential information)

The following part of this chapter deals with logical potentials, hereafter referred to as "potential".

Logical potentials and physical potentials are not mixed. Logical potentials cannot be created below physical potentials, for example.

11.2 Placing potentials on connectors

Requirement

- The devices have been created.
- A logical potential has been created as an engineering object.

Placing a logical potential on a connector

1. In the Navigator, open the structure until you can see the required device and the logical potential.
2. Drag&drop the logical potential to the device.
   The potential is displayed at the connector of the device in the Navigator.
3. Open the properties of the device.
4. Select the "Connectors" tab.
5. Right-click on any table column and then select the "Columns > Log. potential" command in the context menu.
   The potential is displayed in the new column.

Name syntax

The name of the potential that has been set is input on the "Connectors" tab in the "Log. potential" column. Another character can be input before the name, often a minus sign. This is the prefix, i.e. the separator that was input in the project options.

An asterisk can appear at the end of the name. The asterisk only appears at the start object, which is the object where the potential has its origin. This enables you to tell whether the information concerning the potential was simply transferred (inherited via the connector) or whether it was defined here.

Disconnecting a potential

First execute the "Columns > Log. potential" command on the "Connectors" tab in the properties of the device. The potential is displayed in the new column.

You can only remove the potential again in the "Log. potentials" column. Use the "Disconnect" command in the context menu to do this.

Change direction

The change of direction also works for connections that are joined to logical potentials. You can change the direction at the connector of the component. In addition, the system checks whether there is an array connector, for which the direction then also changes.

Result

The potential is assigned to the connector. Connections to this connector also recognize the potential.

11.3 Collecting potentials by means of implementation

Procedure

1. In the base project in the Navigator, prepare a base object for a logical potential on the "Base objects" tab. To do this, activate the "Request" option in the "Implement requests" control group on the "System settings" tab.
2. Save your settings.
3. Change to the engineering project.
4. Create this base object on the report.
   A potential request is created on the "Units" tab in the Navigator.
5. Create a potential on the "Units" tab in the Navigator.
   This potential serves as the owner.
6. To implement the owner potential on the potential request, open the properties of the potential request.
7. Drag&drop the owner potential from the Navigator into the "Implementation" field.

Result

The request potentials are collected underneath the owner potential. Matching links are generated on the engineering reports at the potentials.

Dissolving an implementation

If you dissolve an implementation, the corresponding potential is removed from the unit.

11.4 Assigning potentials

Requirement

Two components point to one another with references.

Procedure

1. Drag&drop a potential to one of the components.
2. Select one of the following options:
   - If you are assigning the potential for the first time, it will be assigned to both components. Multiple placing is now possible for the potential.
   - If you are reassigning the potential, the "Assign potential" window will open.
   - If you want to disconnect the existing chain and assign the potential to the selected component only, select "No". The former potential is retained for all other components.
   - If you want to assign the new potential to the whole chain, select "Yes".

Result

The potential is assigned accordingly.

If you are not assigning a new potential to a component, but are reassigning the potential itself, the "Assign potential" window will open in the case of potentials which have been placed multiple times.

11.5 "Potential - assigned devices" plugin

Introduction

The "Potential - assigned devices" plugin enables you to display all devices connected to a potential without activating potential tracking in the project options.

User interface

Button	Description	Function
	Refresh	Updates the displayed device list.
	Export data	Enables the export of the created device list.
	Scan devices	Starts the scan for all connected devices of the specified potential and calculates the potential tracking.
	Stop	Cancels the scan.

Using the plugin

You have two options for using the plugin:

In the tab "Potential - assigned devices":

1. In the Navigator, open the properties of the potential in the context menu.
2. Click on the tab "Potential - assigned devices".

In the plugin in the menu:

1. Open the plugin: "Plugins > Automation > Potential - assigned devices".
2. Set a node or potential as the start object using drag-and-drop.
   Note: Documents cannot be selected as start objects.
3. Click on "Scan devices".

Result

You receive a list of all assigned devices grouped according to potentials.

From the results list, you can right-click to navigate to the object, its base object, or its position.

Using product data and manufacturer devices 12

12.1 Definition and area of application

Definition of product data

Product data constitutes the minimum amount of attributes that are required to order a product. The product that can be ordered is also referred to as manufacturer device. Product data usually includes the following properties:

- Attribute values
- Connectors
- Elements
- Accessories
- Symbols

In the course of engineering, a certain subset of all user inputs is used to select or describe a manufacturer device (or "real device") that can be ordered.

Overview of the work procedure

This is the work procedure from the templates to the manufacturer device that can be ordered:

1. A number of the attributes in the base data are labelled as product data-relevant.
2. You have the option of entering a template in the base data.
3. The attributes relating to the product data are filled in by the user in the engineering data.
   SeeEntering product data (Page 251).
4. You can use the product data entered in the engineering project to search within the available manufacturer devices and then select the required device from the list of results. The following versions are available to assign product data:
   - Set implementation
   - Set base object pointer
   - Transfer values from a catalog
     SeeManufacturer device selection for a single component (Page 252).

No manufacturer devices for special production items are available in COMOS. In such cases, use the product data entered in the engineering project to generate a manufacturer device.

12.2 Installing the "Manufacturer device database" iDB add-on

Objective

There is an add-on package for COMOS iDB that provides additional manufacturer devices as objects. The add-on package is referred to as the "E&IC manufacturer device database". The terms "Device Layer" or "Manufacturer Device Content" are also used in some cases. The add-on package contains a subset of the following device groups:

- SITOP power supply
- SINAMICS Frequency Converter
- SIMOCODE
- SENTRON
- SIRIUS

Licensing and use

You need a license to place the objects of the "E&IC manufacturer device database" on a report.

You can find additional information on this topic in the "Manufacturer device database guide".

12.3 Opening the properties of devices

Requirement

The IC detail engineering has been performed.

Edit properties

If you have placed components, you can immediately specify the required attributes for these objects:

1. Select the object in the Navigator and select "Properties" from the context menu, or double-click the object on the report.
2. Select the "Attributes > Technical data" tab.
3. Enter your values.

Other manufacturer devices are offered here later, depending on these values.

12.4 Entering product data

Requirement

- The attributes that are relevant to product data have been prepared.
- Definition and application area for product data are familiar.
  See chapter Definition and area of application (Page 249).

Procedure

1. Create engineering objects.
   The attributes that are relevant to product data can be identified by the additional, grayed-out field. If templates have been entered for the request at the base object, they are displayed in the additional field.
   In the following picture, "T1" has been entered as the default at the base object.



2. Edit the attribute during the course of the engineering process in the usual way.
   Depending on the specification at the base object, the input field turns orange when it deviates from the specification:



   The subsequent procedure to be followed depends on which type of implementation you choose.

Example

1. Select a sublocation.
2. Select the "A70 Protection devices" folder.
3. Select the "New > A10 Protection devices > A10 Fuse > A30 Fuse, 3-pin" command in the context menu.
4. Open the properties of the fuse.
5. Open the "Attributes > Order data" tab
   The "Manufacturer" attribute is relevant to the product data.

12.5 Manufacturer device selection for a single component

12.5.1 Allocating a single device request to a manufacturer device

Procedure

1. Open the properties of the device request to which you want to assign a manufacturer device.
2. Select the "General" tab.
3. To set a reference to a manufacturer device, click the "..." button next to the "Base object" field. The "Look up manufacturer device for" window opens. Which entries are visible is controlled by the following attributes:
   "Attributes > Order data" tab, "Available catalogs" control group. The attributes in the "Available catalogs" control group may only be edited in the base data.
   For administration, see Connecting catalogs (Page 253).
4. Select the required manufacturer device in the lower area of the window.
5. Click "OK" to confirm your selection and close the "Look up manufacturer device for" window.

12.5.2 User interface reference of the "Manufacturer device search" dialog

User interface element

User interface element	Description
"Update" button 	Via this button you can update the display of the manufacturer devices in the lower table, if e.g. you selected a new selection criteria.
"Show relevant attributes only" button 	If you click on the arrow next to the button, you can select if all attributes of the currently selected device request or only the most important relevant attributes of the selected device request should be displayed in the upper table.
"Attributes" column	In this column the attributes are displayed which you allocated to the device request.
"Requests" column	This column displays the attribute values which you assigned to the device request via the "Technical data" and "Order data" tabs.
"Selection" column	If you click on a line in the column you can select a sorting criterion for the display of the manufacturer devices in the lower table.
"Display" column	When activated, this option displays a column for this attribute in the lower area.
"Manufacturer device" column	The name of the found manufacturer devices is displayed in the column.
"Description" column	A short description of the manufacturer device is displayed in the column.
"Manufacturer" column	The manufacturer of the manufacturer device is displayed in this column.
"Supplier" column	The supplier of the manufacturer device is displayed in this column.

12.5.3 Connecting catalogs

Catalogs are connected in two steps:

1. All required manufacturer catalogs are integrated in the COMOS base data.
   You can find additional information on this topic under "EI&C Administration", keyword "Administering the manufacturer catalogs".
2. A subset is selected from the list of available manufacturer catalogs which is to be available at a specific base object.
   You can find additional information on this topic under "EI&C Administration", keyword "Preparing available catalogs for manufacturer devices".

12.6 Manufacturer device selection for numerous components

Requirement

- You have created device requests.
- Manufacturer devices are available in the database.

Searching for manufacturer devices

There are two different ways of searching for manufacturer devices:

- Default: XML based mode
  If you select the XML based mode, all manufacturer devices that fit your request are imported to a temporary XML string. This increases the speed when searching for manufacturer devices.
- Alternatively: Database-based mode
  Database-based mode switches to the "Manufacturer device selection" dialog.

12.6.1 Allocating multiple device requests to manufacturer devices

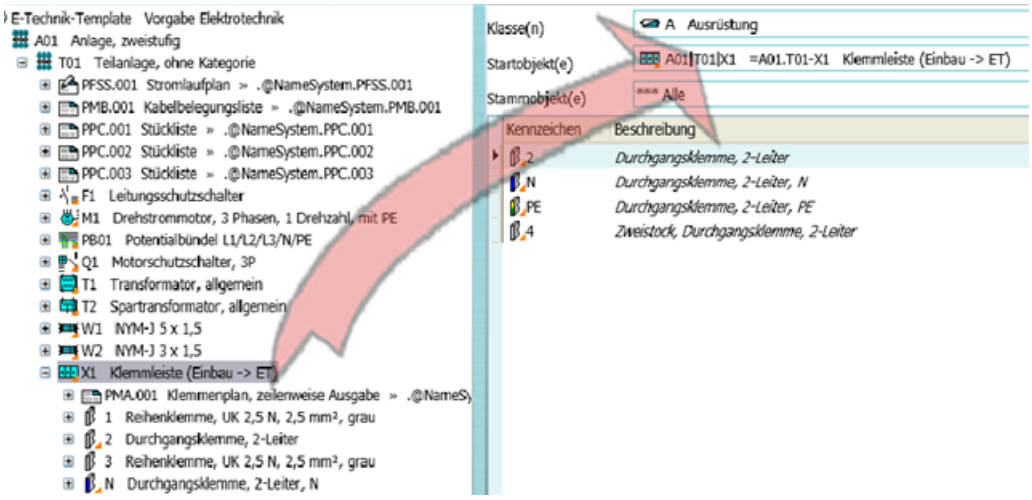
Requirement

XML based mode is activated.

Procedure

1. In the COMOS menu bar, select the following command:
   "Extra > Detail > Device selection (product data)".
   The "Device selection (product data)" window opens.
2. Select the required class, for example "A Equipment", from the "Class(es)" list.

3. Drag&drop the required device from the Navigator into the "Start object(s)" field.
   The upper area of the window displays a list of the device requests which are located below the device that you have defined as the start object.



4. Click on the [PD?] button.
   Additional columns are displayed. A separate column is displayed for each attribute relevant to product data. In the case of engineering objects which do not have an entry for this product data-relevant attribute (or which do not have the attribute at all), the column remains blank at this point. The inherited values from the base object may be displayed. The subsequent steps consider all visible columns. If only some of the product data-relevant attributes should be considered, you need to hide some of the columns again.
5. Optional: Hide PD columns
   To do this, select a column and click [Del].
   The attribute is retained in the engineering data. The column (= attribute) is hidden and not considered any longer in the next steps.
6. From the list of results, select the device requests to which you want to assign manufacturer devices.
   In the lower area of the window the manufacturer devices are selected that fit to the selected device requests.
7. In the lower area of the window select the manufacturer device which you want to assign to the selected device request.
8. Click on the "Set base object" button.

No hits

There can be three reasons for no base objects being found:

- No matching manufacturer devices had been created in or imported to the base data yet.
- The engineering objects selected in the upper window differ too much. COMOS tries to find base objects that suit all selected engineering objects. Reduce the number of selected engineering objects and repeat the procedure.
- The search considers attributes that are not available in the manufacturer devices. Reduce the number of the attributes to be considered by deleting some of the columns in the upper window. Then repeat the search.

Result

The base object pointer is newly set for all selected device requests. The device requests are not labeled as a request anymore.

12.6.2 User interface of the "Device selection (product data)" dialog

Buttons

Button	Tooltip	Description
PD? ▼	"Create columns with product data attributes"	When you click on the arrow next to the button another button is displayed, via which you can select for which product data attributes columns should be displayed in the upper window area (e.g. "Create all", "Create with values only", "Create with own values only").
▼	"Request"	When you click on the arrow next to the button another button is displayed via which you can select if only requests, only manufacturer devices or requests and manufacturer devices should be displayed in the upper area of the window.
	"Execute"	Searches the products for all selected objects and displays them in the lower area of the window.
	"Set base object"	Via this button you can allocate a manufacturer device to a device request which you selected in the upper area of the window.

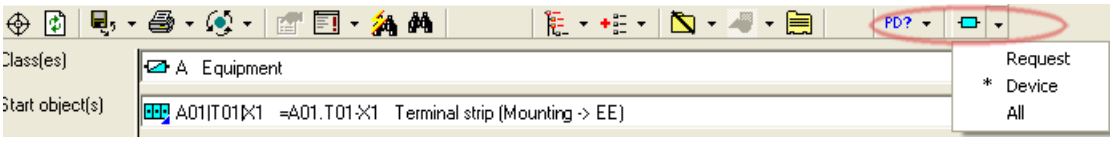
You can find more information on this topic in the "Platform Administration" manual, keyword "Standard queries".

12.6.3 Subsequently editing objects

Procedure

If you want to subsequently change the assignment of a manufacturer device, you need to select a different setting in the "Device selection (product data)" dialog window.

1. Open the dialog window and set a start object.
2. Now in the icon bar click on the icon on the far right (next to the PD icon) on the small arrow:



Select the "Device" option from the small menu that drops down.

Result

All components for which a manufacturer device has been implemented are displayed, and all components without implementations are hidden.

With this set of hits, you can work as described in Allocating multiple device requests to manufacturer devices (Page 253).

12.7 Special components: creating manufacturer devices from requests

12.7.1 Creating a manufacturer device for a request

Introduction

If the engineering objects are one-off or special production items, the base project does not contain any manufacturer devices from which you can make a selection. In such cases, use the entries in the engineering project to create a manufacturer device in the base project. The details of the previous engineering object and the new manufacturer device then match. This enables you to verify that the previous entries are valid and relevant for orders.

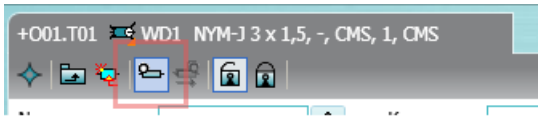
Thus there is feedback from the engineering project to the base project.

Requirement

- You have the "Product data" function right.
- The object has one of the following classes:
  - "Accessories"
  - "Element"
  - "Device"
- You have entered all relevant information in the object properties. The product-relevant data on the "Order data" tab has been completed. If one of the requirements is not met, contact your administrator.

Procedure

1. Open the properties of the object.
2. Click the "Create base object 'Product data'" button on the menu bar of the properties.



Result

- A new base object is created below the previous base object. The project is retained. If the original base object came from the base project, the new base object is also created there.
- This new base object becomes the new basis of the engineering object. The entries that were made for the engineering object are transferred to this new base object as templates.
- The manufacturer device base object created above is assigned to the engineering object. The "Implementation" field is hidden.
- The assignment of a manufacturer device causes the orange background function to take effect. See also chapter Entering product data (Page 251).

Using a mounting catalog and hook-ups 13

13.1 Creating constituent parts of the mounting catalog

Overview of objects

You work with the following objects in the mounting planning:

- Mounting object
- Mounting accessories
- Cost objects

Requirement

- The location structure has been created.
- Functions have been created.

Creating the "Hook-up" folder

1. Select the "New > @Templates Templates" command in the "Units" tab in the context menu of the project node.
   The "Copy templates" node is created in the Navigator. Various folders with copy templates are displayed in the detail area.
2. Select the "M41 EI&C" folder.
3. Select the "A10 Hook-up" folder below the "M41 EI&C" folder.
4. Drag&drop the "A10 Hook-Up" folder into the Navigator onto the node "@Template copy templates".
   Examples are already prepared in this folder.

Creating a mounting object

1. Create a link object
   - Select the "A10 Basic Engineering" folder below the function.
   - Use the context menu to select: "New > Mounting > Mounting Link object":
2. Create a mounting object using the link object
   - Open the properties of the created object "Mounting Link object".
   - Select the "Attributes > Hook-up" tab.
   - Select the desired hook-up in the Navigator in the"A10 Hook-up" folder.
     For example "A20 Pressure transducer > P01_000 P-MU. el.".
   - Drag&drop the hook-into the "Hook-up" field on the "Attributes > Hook-up" tab.
   - Confirm your entries.

Example with drawn hook up

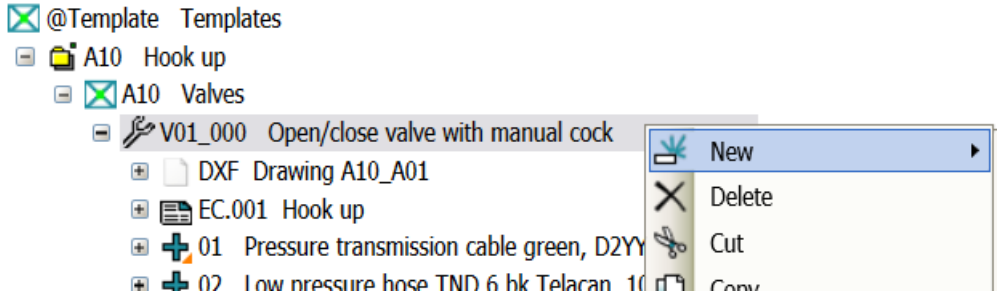
An example with a drawn hook up is prepared in the example library:

"A10 Hook-Up > A10 Valves > V01_000 Open_Close_Valve_with_Manual_Cock"

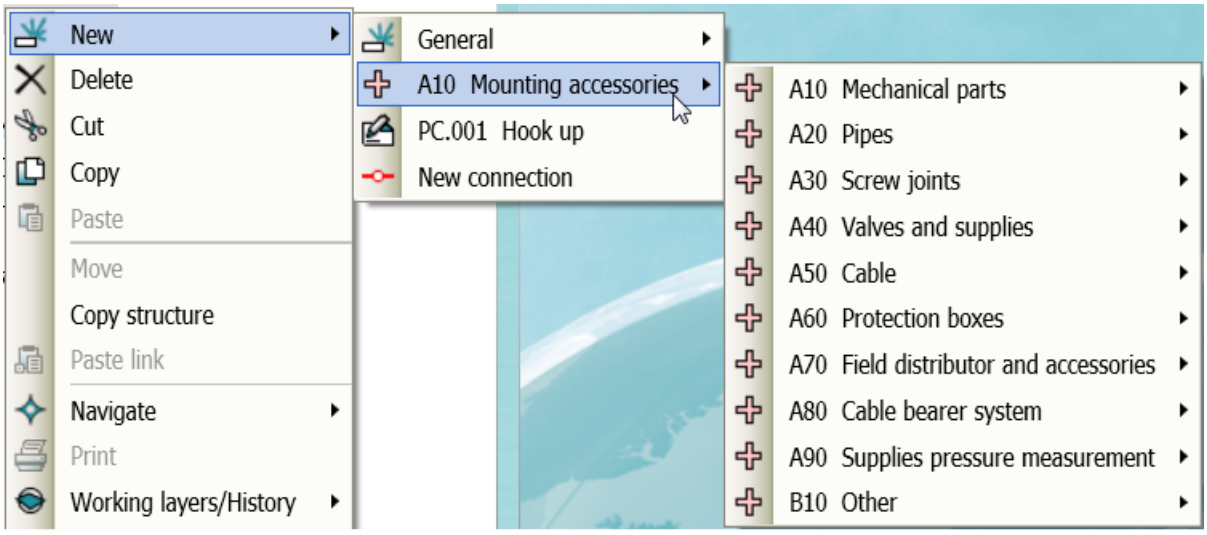
Creating additional mounting accessories

Mounting objects consist of several constituent parts, the so-called mounting accessories. In the prepared example objects, the mounting accessories are created automatically below the mounting object.

1. Select the mounting object (not the link object).



2. Use the context menu to select: "New > Mounting accessories":



Cost objects

Mounting accessory objects each have a cost object. In the prepared example objects, the cost object is created automatically below the mounting accessory object.

13.2 Editing mounting accessories

Requirement

The mounting accessory has been created.

Procedure

1. Open the path to the mounting accessory and open the properties.
2. Select the "Attributes > Order data" tab.
3. Enter the desired information.
4. Apply the changes and close the properties.

Result

The order data of the mounting accessory has been specified.

13.3 Entering costs

Introduction

You use the "Example costs" object to define the material and mounting costs of each individual mounting accessory.

Requirement

The mounting accessory has been created.

Procedure

1. Open the path to the mounting accessory for which you want to specify the costs.
2. Open the properties of the "Example costs" object located below the mounting accessory.
3. Select the "Attributes > Costs" tab.
4. Enter the costs for the mounting accessory in the "Data" control group.
5. Apply the changes and close the properties of the expense object.

Result

The costs of the mounting accessory have been specified.

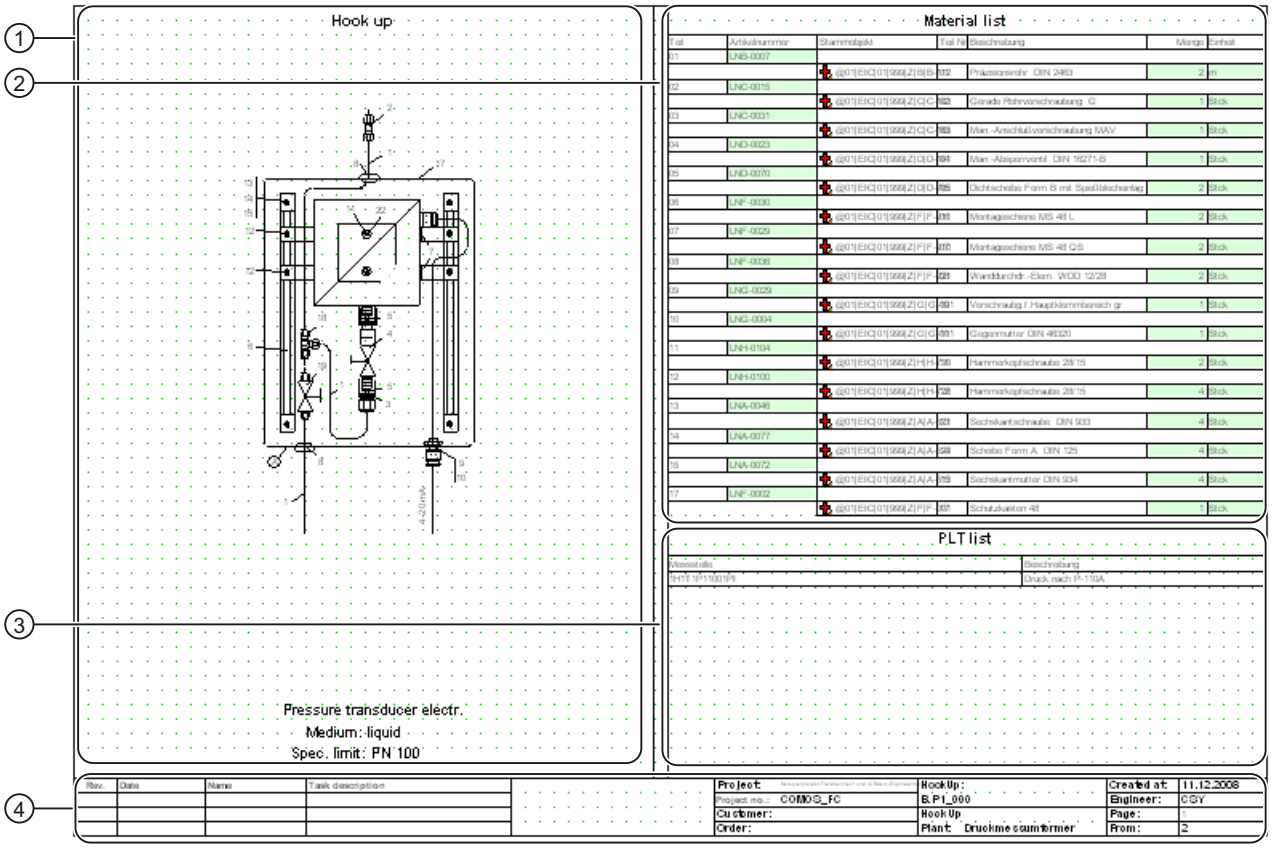
13.4 Structure of the "HookUp"

Introduction

The mounting drawing and the material required for mounting are displayed on the "HookUp", which also shows the measuring points where the mounting object is used. The "HookUp" is automatically created below each mounting object.

Structure of the "HookUp"

The following figure shows how the "HookUp" is structured:



① "HookUp" area

This area of the "HookUp" contains the drawing. The symbol of the "HOOKUP" drawing type is displayed here for the associated mounting object. The numbers in the hook-up drawing correspond to the numbering in the list on the right and the numbers of all mounting accessories in the Navigator.

② "Material list" area

The mounting accessories for the mounting object are listed here.

③ "Measuring location list" area

The measuring locations where this mounting object and, consequently, this "HookUp" is used are listed here.

④ Title block

The title block displays the information which you have specified in the project data.

Designing cabinets

14

14.1 Creating a cabinet design

Requirement

- A cabinet has been created.
  See chapter Creating a control cabinet and equipping it with basic objects (Page 33).
- The standard model for cabinet engineering is clear.

Creating a cabinet design

1. Click on the "Locations" tab in the Navigator.
2. Standard model: Open "W01 Site > L001 Building > R001 Room, control center"
3. Right-click on the "S001 Cabinet" object and select "New > LU.??? Cabinet design".

The corresponding command is no longer available in the context menu once the cabinet design has been created.

Note

COMOS contains simple engineering examples, which you open and use for the initial steps. You can find these examples by right-clicking on the cabinet design in the Navigator and selecting the "New > D Cabinet engineering" command in the context menu.

14.2 Background information: Views

Definition of "views"

Cabinet designs are two-dimensional but are intended to display three-dimensional objects. You work with "views" for this.

View: A special, rectangular point of view on the items in the diagram. Views are also called "sections".

A view allows you to see objects directly from the front, directly from the left and so on. Just as a cube has six sides, there is a maximum of six different views.

Views overview

The following views are available on the right side of the design diagram:

Symbol	Tooltip
	"Front view"
	"Side view, left"
	"Side view, right"
	"Rear view"
	"Plan view, from top"
	"Plan view, from bottom"

It is not possible to freely select a point of view for the engineering and to look at the engineering from "diagonal left top", for example. Such a "free view" is only possible in the 3D module.

Views are projections

Technically speaking, a view is a "projection". This means that it is first necessary to exactly calculate what you will see in a view, since objects can be hidden. This calculation takes some time. For that reason no views are placed initially on a design diagram.

You decide by yourself if you need a front view, a top view, etc. When engineering a simple control cabinet, two views are often sufficient (a front and a side view). The first step when working with design diagrams is, therefore, to always place at least two views.

Reference

You can find additional information on this topic in the "EI&C Administration" manual, keyword "Symbol variants for views".

14.3 Mechanical design in the front view

14.3.1 Defining the point of origin

You have two options to define the point of origin on the design diagram of a control cabinet:

- On the report template of the design diagram. You can find more information in the manual "EI&C Administration > Preparing cabinet engineering > Report template".
- In your engineering project on the current cabinet design:
  1. Open the desired document.
  2. Right-click on a point in the document and select the entry "Set point of origin" in the context menu.
  3. Place the green circle at the position where you want to have your point of origin.

When you have set a point of origin, you can use it to calculate coordinates. You can find more information in the chapter "Cabinet Equipment Assistant".

See also

Cabinet Equipment Assistant (Page 331)

14.3.2 Snap mode on the cabinet design

Requirement

- A cabinet has been created
- A cabinet design with the report template "A30.L.LU.LU_A10.LU_A20 snap point dimensioning" has been set up.
  You can find more information on this topic in the "Platform Administration" manual, keyword ""A30 Templates acc. IEC 61355" document group".

Activate the snap mode function

1. Open a cabinet design of the "snapping design" type
2. Activate the "Activate snap mode function" checkbox in the menu bar of the report.
3. Determine the snap precision. This specifies the radius around the selected object in which the search for snap points is to be conducted.
4. Confirm your selection with "Accept".
5. Select one or more objects in the report.

Result

According to the selected snap precision, a preview shows the points or lines a selected object will be affixed to when you move the object and release the left mouse button.

Note

Note: The snap mode is only available for the front view onto a cabinet diagram.

Options

- When pressing the <SHIFT> key, the selected object is affixed to the grid points.
- Press and hold the <i> key while dragging an element onto the rail to briefly deactivate the snap mode function.
- With multiple selection, the preview displayed refers to the group object which is located under the mouse pointer when being moved.

14.4 Mechanical design in further views

14.4.1 Working with views

14.4.1.1 Preparing views

You can find more information about this topic under "EI&C Administration",

Keyword "Views" and keyword "Managing multiple placement of rails".

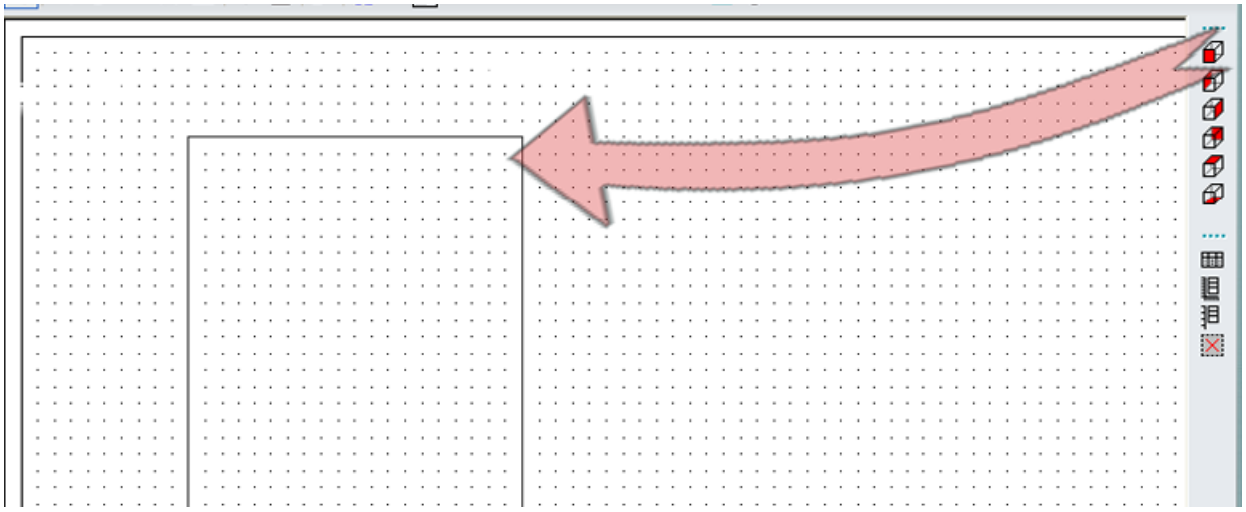
14.4.1.2 Placing a view

Requirement

The cabinet design has been created.

Placing views

1. Open a cabinet design.
   On the right side of the cabinet design, you will find six symbols for the views.
2. Click on a symbol and drag it onto the cabinet design.
   A frame appears after you have dragged the symbol onto the cabinet design.



The frame is a defined view and specifies the area in which work is conducted. Please note that no object is allowed to protrude over the view frame.

Placing multiple views

All six views may be placed on the report. However, this will increase the calculation time for the report.

Note

A certain view, for example the "front view", may only be placed once on a report. For technical reasons, the program does not prevent a user from placing a particular view on a report multiple times, but the design diagram does not function properly if this is the case.

Automatic placing

If multiple views are placed, then all of them are always updated. Thus, if you place a component in the front view, COMOS uses information such as the dimensions of this component and the other components already placed, etc., to calculate an automatic placement for the other views and displays it immediately. Sometimes the automatic placement will not fit your expectations. Then you have to manually mark the component in the relevant views and move it to the corresponding valid position.

Placing views retroactively (Place objects (2 1/2 D))

If the views already contain objects and you place another view, this new view will be empty at first.

1. Select the view.
2. Open the context menu and select the "View/Slice > Place objects (2 1/2 D)" command.

14.4.1.3 Symbol layout of the views

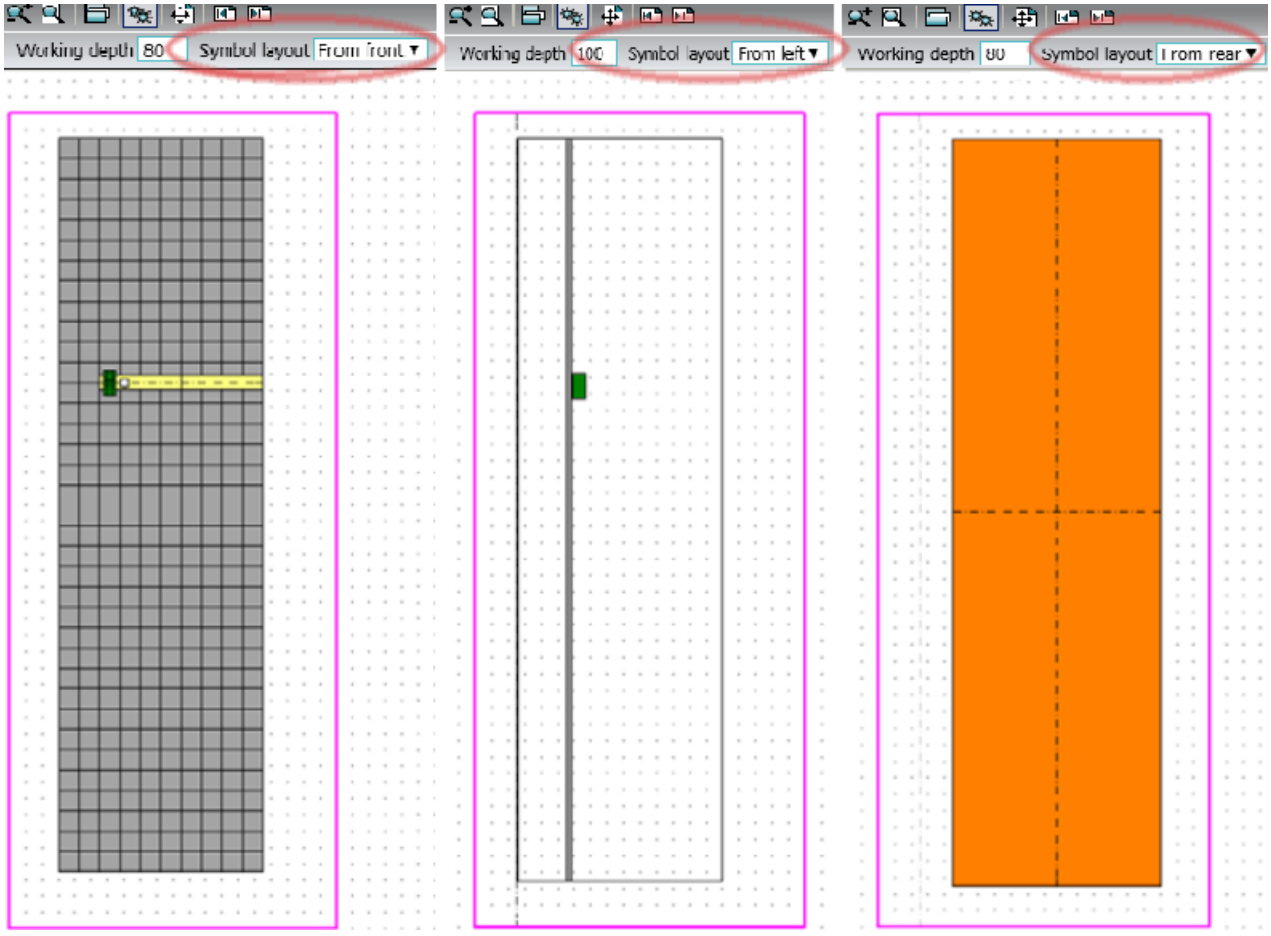
Requirement

The cabinet design has been created and is open.

Definition: Symbol layout of the views

Each view portrays a certain symbol layout. The symbol layout defines how the mounting elements you want to place on the design diagram should be installed.

If, for example, you have placed a front view on the design diagram, the default setting of the symbol layout is "From front". If you have placed a left view on the design diagram, the default setting of the symbol layout is "From left". If you have placed a rear view on the cabinet design, the standard setting of the symbol layout is "From rear", etc.



Changing the symbol layout

You may need to change the symbol layout of a view. For more on this, see Placing a mounting plate in the rear view (Page 274).

To change the symbol layout of a view, follow these steps:

1. Select the view for which you want to change the symbol layout.
2. Select a new symbol layout from the "Symbol layout" list.
3. Click the "Apply" button to save your selection.

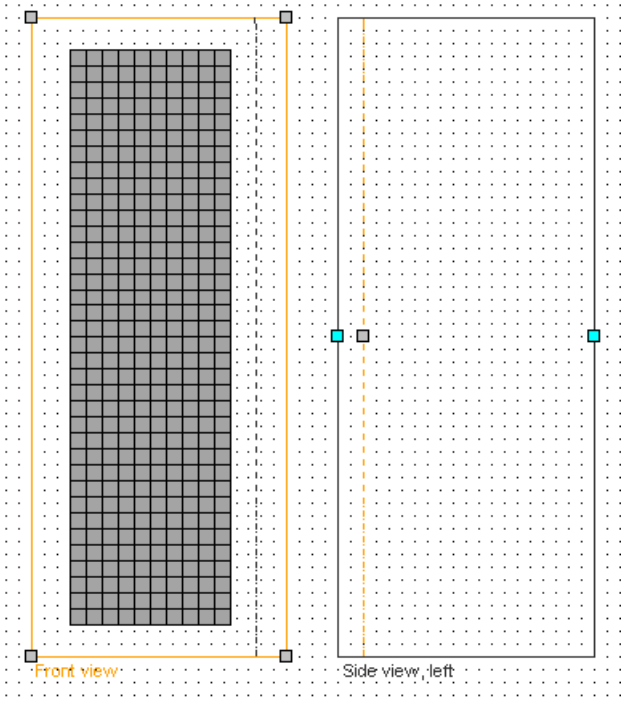
14.4.1.4 Changing view size manually

Requirement

The cabinet design has been created and is open.

Change the size of views retroactively

When you select a view, the view receives grab points:



You can use these grab points to change the views. The gray grab points of the front view, which are located on the left side of the report, can be used to change the size of the view. The blue grab points of the side view, which are located on the right side of the report, can be used to move the view lines. The view lines display the left and front section of the display cube.

It is permissible to change the size of a view retroactively. This approach is useful if a placed object extends beyond the edge of a view.

For rails, there is the possibility of automatic distribution to additional views. See section Placing a mounting plate in the rear view (Page 274).

14.4.1.5 Adjusting or arranging view/slice automatically

Requirement

The cabinet design has been created and is open.

Options for automatic size adjustment

- Context menu in margin of view: "Views/Slices > Synchronize views"
  Adapts the other views to the size of the selected view.
- Context menu in margin of view: "Views/Slices > Adjust view to required object space"
  Adapts this view to the space requirement of objects placed in this view.
- Context menu of the report: "Views/Slices > Adjust all views to maximum required space"
  Initially determines the space requirement of the placed objects for each view. Then, the largest space requirement of all views is determined and all views are extended or reduced to this space requirement.

Options for automatic position adjustment

- Context menu of the report: "Views/Slices > Realign views"
  Moves the views to the same height and reduces the distance.

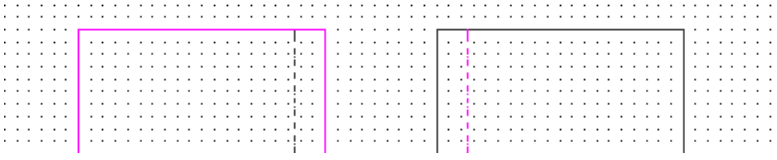
14.4.1.6 Placing objects in a view

Requirement

- The cabinet design has been created and is open.
- The views have been placed.

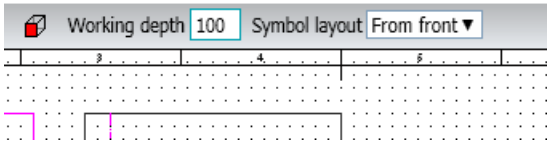
Specifying the working depth in a view

The position of an object in an area is defined by three coordinates. However, if you place an object in the front view, only two of the three coordinates will be set: the width and height coordinates. The missing coordinate is the working depth. This coordinate is automatically assigned to each object that is placed in the front view. The currently set working depth of the front view is displayed in the side views by a vertical dashed line:



The following options are available for specifying the working depth:

- Report menu, "Working depth" field
  When a view is selected, the "Working depth" field is displayed on the symbol bar.
  If you enter another value here and click on the exclamation mark, the associated dashed line moves.



- Context menu of the view, "View/Slice > Set working depth" command.

If the working depth of the front view is set to 250 mm, for example, and you place an object on the front view, this object will be placed on the left side view 250 mm deep into the area.

The working depth is managed as an attribute in the properties of the view objects: "Attributes > Settings" tab, "Working depth" attribute.

Placing the cabinet frame

To place the cabinet frame in one of the views, follow these steps:

1. Click on the plus sign (+) in the Navigator next to the "Cabinet system" element.
   The element "Cabinet frame" is displayed below the cabinet system.
2. Drag the "Cabinet frame" element onto the cabinet design in the front view.
   After having placed the cabinet frame in the front view, the cabinet frame in the other view is automatically placed on the height of the working depth of the views.

Placing mounting elements

The component elements that you need in order to structure the control cabinet can be found in the context menu of the cabinet frame.

You also find the "Fastening parts" node below the cabinet system in the Navigator. A mounting plate and mounting parts which you can use to mount the control cabinet can be found in the context menu of the node.

To structure the control cabinet, follow these steps:

1. First create the component elements which are located in the context menu of the cabinet frame.
2. Right-click on the "Fastening parts" node and create the object for the mounting plate.
3. Drag the component elements as well as the mounting plate on after each other into the cabinet frame on the cabinet design.

14.4.1.7 Placing a mounting plate in the rear view

Requirement

- A mounting plate is placed on a cabinet design in the front and the rear view.

Defining the symbol layout for double-sided mounting

In order to correctly mount parts on the back of the mounting plate, you first have to change the symbol layout of the mounting plate. The symbol layout shows how the mounting elements you want to place on the mounting plate should be installed. See also chapter "Changing view size manually (Page 271)".

If you select a rear view for the mounting plate, the default setting for the symbol layout is "From rear". With double-sided mounting of a mounting plate, however, the symbol layout "From front" is correct for both sides of the mounting plate. This displays also the mounting elements on the back of the mounting plate from the front.

To change the symbol layout, follow these steps:

1. Select the rear view that you placed on the cabinet design.
2. Select the "From front" setting from the "Symbol layout" list.
3. Click the "Apply" button to confirm the selection.

14.4.1.8 Selecting and editing objects in the view

Selecting objects

Context menu of the view, "View/Slice > Select all objects" command:

You use this command to select all the objects placed in the selected view.

14.4.1.9 Synchronize view: Synchronizing via 3D

Requirement

- A cabinet has been created.
- Several cabinet documents (cabinet design, mounting diagrams) have been created under the cabinet.

Synchronizing via 3D

1. Open the report "LU.001, Cabinet design".
2. Mark a view on the cabinet design.
3. Open the context menu and select "View/Slice > Synchronize with 3D".

The rails you placed on the front and back of the mounting plate via the mounting diagrams are now displayed on the mounting plate of the cabinet design.

14.4.2 Basic functions in the cabinet design

Requirement

- The views have been placed.
- Cabinet components have been placed.

Selecting dependent objects

You can move objects which are located on the rail and the rail itself at the same time.

1. Select a rail on the cabinet design.
2. Open the context menu and select "Options > Select dependent objects".
3. Move the rail.
   The objects on the rail will be moved automatically.

You can also use the "Select dependent objects" function when you want to move a mounting plate and also want the objects on the mounting plate to be moved automatically. In this case, select the mounting plate and then the "Select dependent objects" option in the context menu.

Hiding

If objects are selected in a view, you can hide them via the "Options > Hide" context menu. They remain visible in all other views.

If a view on the report has at least one hidden object, then the view object offers the "View/ Slice > Show all objects" command in the context menu.

Construction mode

Click on the report and select "Options > Construction" in the context menu.

With the "Construction" option you can activate the extended placement points. In construction mode, lines and dimensions not only snap on the grid, but also snap on the capture points of the objects.

Identify surfaces

Report button, "Identify surfaces" standard table

- Yes
  If a component is placed in the view, the following applies:
  - If a component was moved into the view with drag&drop, the component located below the mouse pointer during the drag&drop process is highlighted.
  - If the component to be placed is dropped, it is placed in the currently highlighted component. The working depth plays no role here.
- Mounting key
  See chapter Editing components graphically (Page 287).
- No
  If a component is placed in the view, the following applies:
  - The working depth of the view is evaluated
  - The component found at the height of the working depth is used as placing target.

14.4.3 Deleting components in the cabinet design

Requirement

- The views have been placed.
- Cabinet components have been placed.

Deleting a placed object from a view

1. Select the object which you want to delete.
2. Open the context menu and select "Delete > In this view".

Deleting a placed object from all views

1. Select the object which you want to delete.
2. Open the context menu and select "Delete > In all views".

Deleting a placed object from all views and 3D

1. Select the object which you want to delete.
2. Open the context menu and select "Delete > In all views and in 3D".
3. Alternative with only one view: Select "Delete > In this view and in 3D".

The command "Delete > In all views and in 3D" may trigger an additional function under certain circumstances. Where:

- The component to be deleted is placed multiple times
- The option "Synchronize" is activated in the rail editor

The following applies:

1. Select the object which you want to delete.
2. Open the context menu and select "Delete > In all views and in 3D".
   The "Synchronize" window opens.
3. In the "Synchronize" column, select those reports from which the component is to be deleted as well.
4. Confirm your selection.

Deleting engineering objects and placements

1. Select the object which you want to delete.
2. Open the context menu and select "Delete > Object".
   The placement is deleted in the report.
   The engineering object is deleted in the Navigator.

14.4.4 Using the mounting key

Objective

The mounting key specifies which components may be placed on which other components during cabinet engineering.

Entering mounting keys

The possible combinations are set and are entered on the following tab:

- "Mounting key" tab
  You can find additional information on this topic in the "EI&C Administration" manual, keyword "Installation/3D data" or "Mounting key".

Evaluation of the mounting key during placing test

A component is placed on another object if the following applies: a surface of the component to be placed touches a surface of the receiving object.

If the mounting key specifications are violated during placing, the following applies:

- The relevant component changes to red.
- The placing itself is not prevented.

If the specifications on the mounting key are observed during placing, the component changes to yellow.

14.4.5 Placing and using rails

14.4.5.1 Equipping a mounting plate with rails

Requirement

- A cabinet design has been created.
  See chapter Creating a cabinet design (Page 265).
- A mounting plate is placed on the cabinet design.

Procedure

1. In the Navigator, open the structure below the "Cabinet system" object.
2. Select the object "Fastening parts" in the Navigator.
3. In the context menu, select the "C-rail, general" command.
   After having selected the component part it is displayed in the Navigator.
4. Select the rail in the Navigator and drag it onto the mounting plate.
   If you place the rail on the mounting plate, the following applies:
   - The working depth is automatically adjusted to the selected object so that you do not have to set the working depth manually.
   - The mounting key is checked.

Placing additional mounting components

Place additional mounting components using the same method.

14.4.5.2 Distributing rails across multiple documents

Objective

Create an additional view automatically if the rail is too long to display it completely in one view. This function is referred to as distributed view and is only available for rails.

Requirement

- A cabinet design and another design diagram have been created.
  See chapter Creating a cabinet design (Page 265).
- A rail has been placed.
  See chapter Equipping a mounting plate with rails (Page 279).

Procedure

1. Open the cabinet design.
2. Select the rail.
3. Select the "Copy" command in the context menu of the rail.
4. Open another design diagram.
5. Select the view.
6. Select the "View/Slice > Place objects from the clipboard, retain view" command in the context menu of the view.
   The distributed representation is only generated if the placement is made with this context command. If the rail is too long to be displayed completely in the view on the second design diagram, the "Distributed placing" dialog opens.
7. Acknowledge the "Distributed placing" dialog.
   The rail is displayed if this is possible in the current view. The rail is locked for editing in the distributed view.
8. Optional: To display the continuation, scroll to the next page in the report as follows: Click the "Page" button pointing to the right at the top of the report in the icon bar.
9. Optional: To release the rail in the distributed view, select the "Options > Release" command in the context menu of the rail.

You can find additional information on this topic in the "EI&C Administration" manual, keyword "Multiple placement of rails".

Alternative

If you want to place a rail using the Navigator more than once, use the context menu command "Place the objects selected in the tree, keep view".

Result

The rail reaches all the way to the margin in the view. The rail is displayed with a tear-off edge at the margin. A second page was automatically generated in the second design diagram and a view was created on the second page. The rail is continued in the view on the second page.

The following context menus are not available in the distributed view:

- "Place objects (21/2 D)"
- "Adjust view to required object space"
- "Wiring"

14.4.5.3 Automatic distribution of the rail across multiple views

Requirement

The base object of the view is prepared for automatic distribution.

Technical procedure

If a rail is placed in a view, an automatic check of the space requirements is performed. If the rail is larger than the view, the following tasks are performed automatically:

- The view in which the rail is to be placed serves as a template for additional views.
- The check determines how many views of this size are needed to display the rail, including the placing devices.
- The corresponding number of new pages are created in the document.
- A view is generated on each page based on a template.
- The rail is divided between the views. The representation of the rail shows a breaking edge at the border of the view.

Boundary conditions

- The view can be extended in one direction only. It is not permissible for a rail to be so large that the view needs to be expanded in two or three directions during the placement.
- If a rail has been distributed over multiple views, the number of views in this view chain can no longer be changed. It is not permissible to enlarge the rail later.

14.4.5.4 Representation of distributed views in 3D

Technical procedure

When a distributed view is created, this has the following effect on the display in 3D:

- The first view is automatically located at the beginning of the 3D display.
- The additional views are added in series.
  This structure is called a view chain.
- Distributed views generated automatically are automatically synchronized with 3D. The distributed views of the rail are positioned in 3D in such a way that there are no gaps in the view chain.
- If the objects in the view are edited at a later point in time, the user must manually select the "Synchronize with 3D" command from the context menu. This synchronizes all views of the view chain with 3D.

14.4.5.5 Editing rails

Context menu for a rail

The following options are available in a rail context menu especially to edit the rail:

- "Options > Recalculate rail"
  The rail is updated when changes have been made in the rail editor or in the Navigator. The "Recalculate rail" command also applies changes which were made using a query. For example, with a query you can set the attribute "Y00T00011.Y00A04298 Gaps" and, in doing so, set the distance between components on the rail. The attribute is hidden by default.

You can find additional information on the context menu in the chapter Basic functions in the cabinet design (Page 276).

14.4.6 Placing a cable channel manually

Requirement

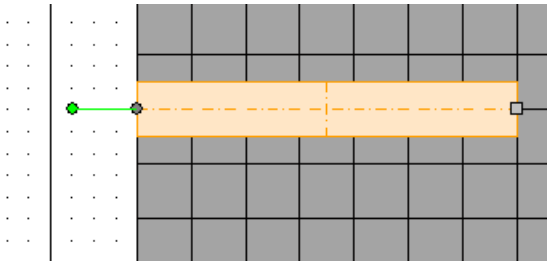
The cabinet object has been created in the engineering data.

Creating a channel

1. Select the cabinet object.
2. Select the object "04 Channels".
3. Select a channel with the "New" command in the context menu.

Placing a channel

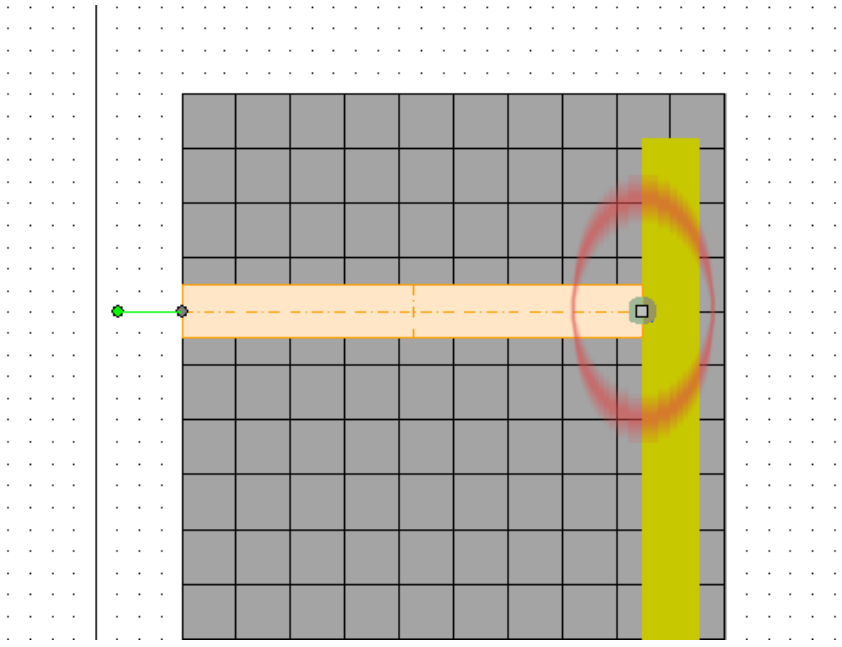
1. Place a cable channel on the report in the same way as a rail.
2. Enlarge the view until you can clearly see the cable channel.
3. Double click twice on the cable channel.
   The cable channel switches to edit mode and now has grab points:



Extending and connecting a channel

You can use the grab points to rotate and drag the cable conduit as you wish. This also applies to the mounting rail, which you have already placed on the design diagram.

After placing a channel fittingly on the edge of another channel, the other channel is displayed in yellow:



The two channels will then automatically be linked in the engineering data.

If the cable conduits have been connected correctly as described above, they will later be considered for auto routing.

Creating virtual channels

Virtual channels are a part of routing. Here, distances up to 1mm are automatically corrected and form connections. For more on this, see Linkpoints (Page 284).

Specifying key and attributes of the cable conduit

The key is a control value at the cable conduit or duct that is calculated based on the set cable attributes for permissible voltages. The attributes are also used for routing to prevent different voltages from being combined in the same conduit. If no attributes are defined, all voltages are considered as permissible and any cable can then be routed through the cable conduit.

1. Open the properties of a duct.
2. Open the "Attributes > Technical data > Cabling" tab.
3. Define the permissible attributes for your duct/cable conduit.
   The key of the channel is calculated automatically based on the set attributes.
4. Open the properties of a created cable.
5. Open the "Attributes > Technical data > Cabling" tab.
6. Define the permissible attributes for your cable.
   The key of the cable is calculated automatically based on the set attributes.

Checking cable conduits

COMOS enables you to determine whether all cable conduits on the report are connected to each other correctly.

To perform the check, follow these steps:

1. Select the view in which you placed the cable conduits.
2. Open the context menu and select "View/Slice > Check cable conduits".

14.4.7 Linkpoints

Objective

Wiring within the cabinet takes place with channels, virtual channels and channel parts.

In particular, the following tasks are supported by the use of channel parts:

- Direct connection between two points on channels (= shortest spatial route). Here, the rule that routing can only be performed through channels is circumvented.
- Routing between two views
- Open air wiring

Terminology

- Channel, undefined entry points:
  A channel for which an entry point can be at any location. This allows you to form open shafts, supports, clips and Lütze combs.
- Channel, defined entry points:
  A channel for which an entry can be only made via entry points specified in the symbol. This allows slot shafts, closed pipes, etc. to be formed.
- Duct:
  Ducts are not placed; they are used for managing the routing. Ducts manage the following information: Cable type, occupied to (= filling level), max. filling level.
  A channel with no duct is not permitted.

14.4.8 Creating virtual channel and placing the channel part

Requirement

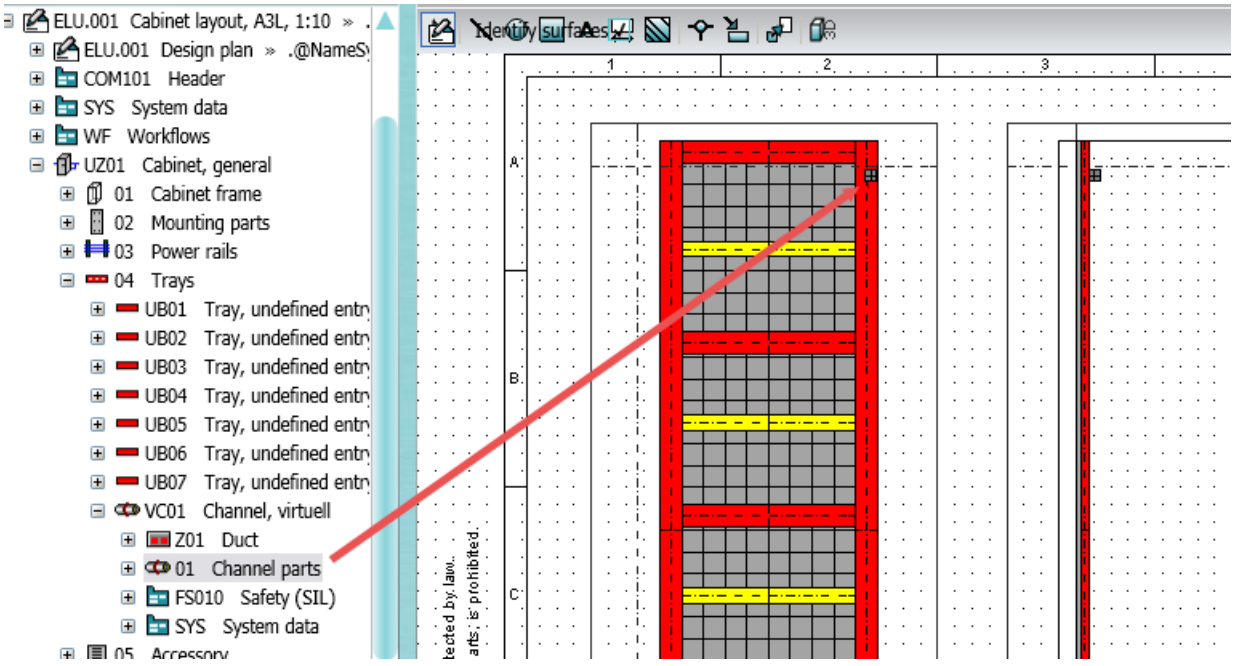
The cabinet object has been created in the engineering data.

Creating virtual channels

1. Select the cabinet object.
2. Select the object "04 Channels".
3. Select the "New > 04 Channel, virtual" command in the context menu.

Placing a linkpoint

1. Select the virtual channel and open the next level in the Navigator.
   The "01 Linkpoint" object is below the virtual channel.
2. Use Drag&Drop to move the "01 Linkpoint" object to the cabinet design in a view.



The first linkpoint must be connected to a channel. To do this, drag the linkpoint from the Navigator and drop it directly on a channel.

Note

Connection of linkpoints using the "Connection" tool is not supported.

Check that the virtual channel has a connection to a channel:

1. Select the virtual channel on the cabinet design.
2. Select the "Navigate > Connections" command from the context menu.

Check during the placement that a connector has been created below the linkpoint. To do this, follow these steps:

1. Zoom in on the display until the linkpoint can be very clearly seen on the report.
2. In the Navigator, open the structure below the linkpoint.
3. Drag the linkpoint to the edge of the channel.

Placing additional linkpoints

1. Select the virtual channel, for example, "VC01 Channel, virtual".
2. Select the "New > 02 Linkpoint" command from the context menu as often as needed.
3. Use Drag&Drop to move the linkpoints to the cabinet design in a view.
   The following rules apply:
   – The first and last linkpoints must be connected to a channel.
   – The linkpoints must be placed on a common report.

14.4.9 Performing the collision check

Requirement

- The "3D view" window opens and / or a cabinet diagram opens.

Procedure

1. Select one of the following options:
   – Right-click in the empty area of the 3D view. Select "Extras > Check > Collision check" in the context menu.
   – Right-click on the empty area of the cabinet diagram. Select "Views/Slices > Collision check" in the context menu.

Result

- The following status display is activated in the Navigator: "Status display > 16 3D collision (EE mounting diagram)" You can find more information on this topic in the "Platform Operation" manual, keyword "Using the status display".
- The objects are colored in the 3D view or the control cabinet plan.

14.4.10 Automatically placing components

Requirement

- The cabinet design is open.
- Additional components are prepared in the Navigator on the "Locations" tab.

Objective

The editable space in the view is detected based on objects already placed and distributed. Place the objects manually on the mounting plate or on the mounting rail.

Procedure

1. Select one or several similar engineering objects on the "Locations" tab in the Navigator. The engineering objects are not placed on the cabinet diagram.
2. Select two similar components in the view. These components are called limiting components.
3. In the context menu of the objects selected in the view, select "Options > Automatically place in tree selected object(s)".
   The objects selected in the Navigator are placed and distributed between the limiting components.
   The working depth of the limiting components is applied to the placed objects.

14.4.11 Editing components graphically

Requirement

- The cabinet design is open.
- Components are placed in the cabinet view.
- All components are aligned parallel to the axes: the rotation is 0°, 90°, 180° or 270°.
- The objects have the following property:
  - Device class CableChannel

Align automatically

1. Select several components on the cabinet design.
2. Select the command "Options > Align automatically" in the context menu of the selected objects.
   - The alignment is determined within the selected components. All subsequent adjustments only take place within this alignment.
   - The components outside the space are determined among the selected components. These components are called limiting components.
   - The limiting components remain unchanged.
   - All other components are distributed evenly in alignment within the area of the limiting components.

Extending

1. You may place a rail or a cable channel.
2. Place a second component with the following specifications:
   - The two components do not touch each other.
   - The second component points in the direction of the first component with the grabs.

3. Select the second component.
4. Select the "Options > Extend (Alignment)" command from the context menu of the second component.
   The second component is extended in the direction of the grab point to the first component. The components touch each other. If the two components have the ability to form connectors, they connect with each other.
5. Select the "Options > Extend (Opposite direction)" command from the context menu of the second component.
   The second component is extended in the opposite direction of the grab point to the first component.
   The components touch each other. If the two components have the ability to form connectors, they connect with each other.

Maximize

1. Place a cable channel and a rail. These components are called limiting components.
2. Place a third component with the following specifications:
   - The third component does not touch the limiting components.
   - The third component points in the direction of a limiting component with the grabs.
3. Select the third component.
4. Select the "Options > Maximize" command from the context menu of the third component.
   The third component is extended in both directions until the two limiting components are touched.
   If all components have the ability to form connectors, they connect with each other.

See also

Basic functions in the cabinet design (Page 276)

14.4.12 Moving a component using the installation key

Requirement

- The cabinet design is open.
- Components are placed in the cabinet view.
- The components have installation keys.

Moving a component over an identified surface

1. Select a component that you want to move in the cabinet design.
2. Select a component in the cabinet design on which you want to place the component.
   The "Identify surface" list appears in the report bar.

3. Select the entry:
   – Mounting key
   – See chapter Basic functions in the cabinet design (Page 276) for information on the "Yes" and "No" entries.
4. Click "Run" (green check mark) in the report bar.

14.4.13 Showing and hiding components in the cabinet design

You have two options to control the visibility of elements on the control cabinet diagram:

- In the view of the diagram
- By activating or deactivating the different graphical layers of the control cabinet diagram.

Requirement

- The cabinet design is open.
- Components were placed in the cabinet view.

Setting the visibility of objects in the view

1. Select a view on the cabinet design.
2. Open the context menu of the view with a right-click.
3. Select the "View/Slice > Visibility of objects" option.
   The "Visibility of objects" window opens and shows you a list of all objects that were placed on the cabinet design.
4. In the "Visible" column, select the checkboxes of the objects that you want to display. All disabled objects are hidden in the design.

Setting the visibility of objects via graphical layers

Multiple graphical layers were predefined for the control cabinet diagram and can be used for each object on the circuit diagram with the "Installation/3D data" tab. You can find the layer in the base objects under the following path "@99 > A80 > M41_P5"

1. Open the properties of one or more objects on the diagram and define the assignment to the layers in the "Graphical layers" control group in the "Installation/3D data" tab.
   - You can find more information on this topic in "EI&C Administration", keyword ""Installation-/3D data" tab for design diagrams".
   - See also Basic knowledge of report layers, Assigning layers

   Note

   Note as of COMOS Version 10.4.2: The following graphical layers must be used analogously to the options in the "Installation/3D data" tab to ensure an identical representation with regard to previous COMOS versions:
   - Mounting distance
   - Mounting footprint
   - Guide lines
   - Grid lines

2. Click the "Layers" button on the toolbar and edit the desired settings.

   Note

   Note:

   It is not possible to deactivate the lines / graphics which specify the frame of objects, such as frames or the borders of a view.

   Do not place "Frame" or "Frames of the view" on Layer 249 "-". Instead, leave them on the standard layer and hide this layer in the settings of the report layers.

3. To apply your settings immediately, activate the "Apply immediately" option.
   The selected settings only apply to the current session.
   To save your settings, click "Save" and confirm your settings.

14.4.14 3D display

Aim

The "3D" function is used only as preview of the 3D model in COMOS and to calculate collisions.

14.4.14.1 Entering Installation/3D data

Entering dimensions

1. Open the object properties.
2. Switch to the "Attributes > Installation/3D data" tab.
3. If the entry "Cabinet frame" or "Cube" was selected in the "3D type" list, the following applies:
   Fill in the "Cubical dimensions" fields in the "Installation data" control group.
   - "Width" attribute
   - "Height" attribute
   - "Depth" attribute
4. If the entry "Cylinder" or "Cylinder segment" was selected in the "3D type" list, the following applies:
   Fill in the "Cylindrical dimensions" fields in the "Installation data" control group.
   - "Diameter" attribute
   - "Length" attribute
   - Attribute "Angle (cyl. Segment)"
     The "Angle (cyl. segment)" attribute defines the angle until which the segment is displayed as filled. The start value 0 degrees is calculated from the top and the segment filled in the clockwise direction.

Entering the mounting distance

1. Open the object properties.
2. Switch to the "Attributes > Installation/3D data" tab.
3. In the "Installation data" control group, fill in the fields in the "Mounting distance (mm) to" area.

Entering the mounting area

1. Open the object properties.
2. Switch to the "Attributes > Installation/3D data" tab.
3. Fill in the "Areas for mounting (mm) from" fields in the "Installation data" control group.
   The distance from the dimensions of the body is entered for this purpose.
   Example: If the value -50 is entered for a plate in the "Left" field, the mounting area to the left is 50 mm smaller than the dimension of the component.
   The value is always entered in mm, which means it is independent of the "Unit system" project option.

Specify the rotation angle

1. Open the object properties.
2. Switch to the "Attributes > Installation/3D data" tab.
3. Enter the rotation angle in the "Mounting" control group.
   Note: The specified rotation angle only indicates the pre-defined rotation with which a component is placed and can be changed later in the circuit diagram.
   Therefore, always compare the specified angle in the "Element of rail" area of the "Cabinet Equipment Assistant".

Changing the connection direction

1. Open the object properties.
2. Switch to the "Attributes > Installation/3D data" tab.
3. Specify the connection in the "Cabinet wiring" control group and select the desired connection direction from the "Connection direction on upper connections" list or from the "Connection direction on lower connections" list.
   Also note the specified search range to avoid routing errors.

14.4.14.2 Open 3D view

Requirement

- A control cabinet has been created and equipped.
  See also chapter Designing cabinets (Page 265).
- The properties of the objects used in the cabinet construction are entered in the tab "Attributes > Assembly/3D data".
  See also chapter Entering Installation/3D data (Page 291).
- A cabinet design has been created.
  See also chapter Creating a cabinet design (Page 265).

Procedure

1. Open the cabinet design.
2. Click the "3D" button on the standard toolbar.

Result

The "3D view" window opens.

14.4.14.3 Moving the 3D view

Keyboard shortcuts

1. Click <Alt> and the left mouse button.
   The shape of the cursor changes.
2. Keep the buttons pressed and move the mouse.

Keyboard shortcuts

1. Click <Ctrl><Alt> and the left mouse button.
   The shape of the cursor changes.
2. Keep the buttons pressed and move the mouse backwards or forwards.

14.4.14.4 Objective and scope of the collision check

Aim

The 3D collision check determines which of the objects placed in the 3D display violate the placing rules.

Objects that do not violate the placing rules are automatically considered to be placed correctly.

Scope of the collision check

The 3D collision check takes into consideration the following information:

1. Dimension and position (Bounding Box)
2. Mounting key
3. 3D types according to the attribute
   The following 3D types are not included in the 3D collision check:
   - Cabinet frame
   - Objects whose names start with "@" ("@SECTION", for example)
   - Class "Device", subclass "Pipe" (DetailClassPipe or constant "Q")
4. Mounting distance
5. Mounting area

The sequence of the items above is also the sequence (priority) in the collision check.

On entering this information, seeEntering Installation/3D data (Page 291).

14.4.14.5 Performing the collision check

Requirement

- The "3D view" window opens and / or a cabinet diagram opens.

Procedure

1. Select one of the following options:
   - Right-click in the empty area of the 3D view. Select "Extras > Check > Collision check" in the context menu.
   - Right-click on the empty area of the cabinet diagram. Select "Views/Slices > Collision check" in the context menu.

Result

- The following status display is activated in the Navigator: "Status display > 16 3D collision (EE mounting diagram)" You can find more information on this topic in the "Platform Operation" manual, keyword "Using the status display".
- The objects are colored in the 3D view or the control cabinet plan.

14.4.14.6 Representation of the mounting area in 3D bodies

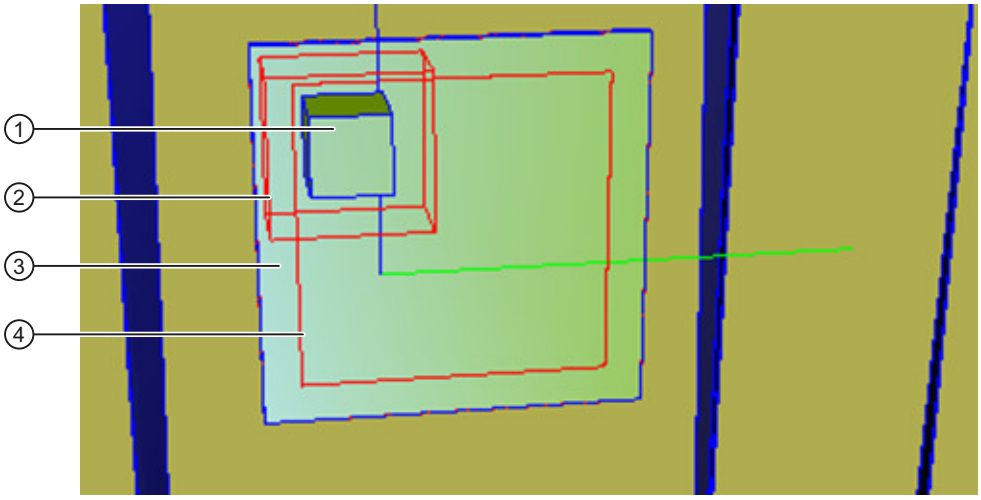
Basic rules for 3D display

- Display of the mounting area for cube-shaped objects
  3D types: EloFlexQuader
  The display is similar to the 2D output with the following exception:
  The border lines are not displayed as dotted lines, but shown as a rectangle using the "obstruction" setting (default: red).
  The mounting area is calculated separately for each side of the cube.
- Display of the mounting area for cylinder-shaped objects
  3D types: EloCylinder, EloFlexCylinderSeg
  The mounting area is projected onto the outside of the body because it would otherwise not be visible for a solid cylinder.

Example

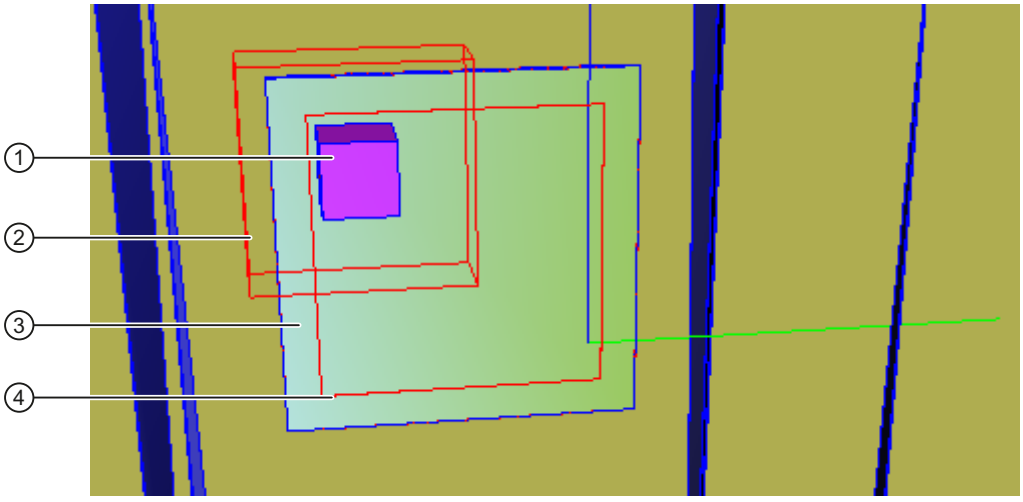
The mounting area is displayed together with the collision information in the 3D display.

The following example shows a correctly placed component:



①	Correctly placed component.
②	Mounting distance of the component (2).
③	Mounting plate
④	Mounting area of the mounting plate (4).

The following example shows how the values for the mounting distance together with the dimensions result in a collision warning:



①	Component with collision warning (here: color purple).
②	Mounting distance of the component (2).
③	Mounting plate
④	Mounting area of the mounting plate (4).

The component is placed within the mounting area, but the mounting distance of the component exceeds the margin of the mounting plate.

14.5 Pre-configuring and equipping components using coordinates

Using 3D coordinates, you have the option to perform automatic placement of components on other components at predefined positions.

Requirement

A control cabinet diagram with front view has been created.

"Configuration" tab

The coordinates X/Y/Z are defined in the "Configuration" tab.

1. In the Navigator, open the properties of the component that you want to equip with other components,
   For example, an assembly plate or rail.
2. Select the "Configuration" tab.
3. Define the coordinates at which the desired component should be mounted later.
   The coordinates are calculated relative to the higher-level/current objects, whereby Y is the offset in the up/down direction (positive = up) and Z is in the depth direction (positive = down).
   Note: All coordinates must be specified. An empty field is not a permissible input; use the value "0" instead.
4. In the "A°" field, enter an initial rotation angle with which the selected component is to be placed.
5. In the "Assembly" group, use drag&drop to drag a component to the field that you want to add to your assembly plate or terminal strip.
6. Add all components as described in step 4.
7. Confirm your entries.
8. Using drag&drop, place your pre-configured component from the Navigator on the circuit diagram.

Result

All elements that were added in the "Configuration" tab are placed together with the component as assembly.

If you transform the object, for example by using rotation, all associated components move with it.

Options

- You can also equip objects already placed on the circuit diagram with components at a later point in time. Always update the document afterwards to have the changes displayed.
- Components can also be placed outside of an object,
  for example limit switches outside the terminal strip. To do this, use a negative value as coordinate.

Reference

Always check your entries using the 3D view.

See also chapter: 3D display (Page 290)

14.6 Using 3D assembly

Analogous to the use of "grayboxes", you have the option of using so-called 3D Assemblies for control cabinets and equipping these placeholders as an engineering task or by means of modular engineering.

Requirement

The principle of grayboxes and the use of engineering tasks are well known.

Components

- "3D Assembly" base object
  - Is used in the design document.
  - Functional classification: "D330"
- "3D Assembly Point of Origin" object
  - Is positioned in the copy template and used to define the insertion position of the template objects in 3D.
  - Functional classification: "D340"
- Engineering task: @20|B20|M41|A10|A30 "Generate 3D Assembly"
  - Copies the objects from the template and positions them in the documents.

Creating and positioning 3D Assembly

1. Right-click on a control cabinet in the Navigator to open its context menu.
2. Select "New > B20 3D Assembly"

3. Activate the option "Preview 3D Assembly"
   - In the properties of the "3D Assembly" object: "Attributes > Installation/3D data > Preview 3D Assembly"
   - In the properties of the circuit diagram, valid for all 3D Assemblies positioned: "Attributes > System data > Preview 3D Assembly"
   - In the context menu of the circuit diagram, as soon as at least one 3D Assembly has been positioned.
4. Position the "3D Assembly" object on the front view of the circuit diagram using drag&drop. A blue frame and a blue cross represent the 3D Assembly in the front view in the circuit diagram. In other views the cross is displayed in gray.
   Once a template is assigned, a preview of the contents of this template is displayed. This can be enabled/disabled per 3D Assembly or per document. The preview display depends on the execution status of the associated engineering task.

Note

The "3D Assembly" object behaves like other COMOS objects in the control cabinet. It can be moved or transformed using the Control Cabinet Equipment Wizard and can be snapped onto other equipment or positioned at a defined mounting depth.

Creating and positioning Point of Origin of the 3D Assembly

Every control cabinet diagram that is to be used as a copy template must have a Point of Origin

1. Right-click on a circuit diagram in the Navigator to open its context menu.
2. Select "New > A20 Point of Origin of 3D Assembly"
   Note: The "Point of Origin of 3D Assembly" object must always be below the circuit diagram in the Navigator and parallel to the view objects.
3. Position the "Point of Origin of 3D Assembly" object on the front view of the circuit diagram using drag&drop. Positioning in other views is not supported.

Note

The "Point of Origin of 3D Assembly" object behaves like other COMOS objects in the control cabinet. It can be moved or transformed using the Control Cabinet Equipment Wizard and can be snapped onto other equipment or positioned at a defined mounting depth.

Using engineering task (eBlock)

Each "3D Assembly" object has an engineering task "Create 3D Assembly".

This engineering task has the 3D Assembly as the reference object and the template - a page macro - as the target.

It can be executed using the context menu of the "3D Assembly" in the control cabinet diagram or linked to an action trigger in the "Modular concept diagram" and executed in the modular configurator.

14.7 Dimensioning components

14.7.1 Setting scale factor

Requirement

The design diagram is created.

See chapter Creating a cabinet design (Page 265).

Scale factor options

The scale factor is saved at the following locations:

- At the report
- At the view objects

Note

Note that the report and the view objects must have the same scale.

Scale factor of the construction representation

You can change the scale factor of the construction representation. For more on this, see Controlling a construction representation (Page 95).

Scale factor of the views

NOTICE
Scale factor of the view objects The view objects themselves are also capable of setting a scale. Do not change the default setting; otherwise, the calculation times will increase significantly. Consistent work is only possible when the scale on the view object corresponds to the default setting.

See also

Creating dimensionings (Page 300)

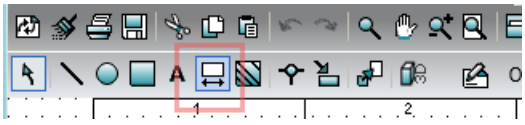
14.7.2 Creating dimensionings

Requirement

- The views have been placed.
  See chapter Placing a view (Page 268).
- Cabinet components have been placed.
- The scale factor is set.
  See chapter Setting scale factor (Page 299).

Activating the "Dimension" tool

1. Activate construction mode.
2. Click the "Dimension" button on the toolbar.



Select one of the options.

Option	Description
"Normal"	Dimension for two points.
"Continuous"	You use this option for continuous dimensioning from a particular point, at any height you wish.
"Baseline"	You use this option for dimensioning a base line starting from a particular point.
"Chain"	When this option is activated, a maximum of 100 construction steps are supported.

Creating a single dimension ("normal")

1. Left-click the first point.
2. Left-click the second point.
3. Left-click the point that determines the height of the dimension line.
4. Right-click to close dimensioning mode.

Creating continuous dimensioning

1. Left-click the first point.
2. Left-click the next point that determines the first distance and first height of the dimension line.
3. Keep left-clicking to set the next distance and next height of the dimension lines.
4. Right-click to close dimensioning mode.

Creating chain dimensioning

1. Left-click the first point.
2. Left-click the second point.
3. Left-click the point that determines the height of the dimension line.
4. Left-click the next point that determines the next distance.
   The height of the dimension line is the same for all increments of the chain dimensioning.
5. Right-click to close dimensioning mode.

If the anchor point of a dimensioning (an object at which a dimensioning is added) is deleted in a chain dimensioning, the two dimensionings are grouped together.

Chain dimensioning using an initial measuring value

An initial measuring value can be entered for chain dimensioning.

In this way, you can manually transfer a value from one dimension chain to another dimension chain. The value is static. If the transfer value of the first dimension chain changes, you need to manually update the initial measuring value of the following chain dimensioning.

To enter an initial measuring value, follow these steps:

1. Single-click on the dimensioning.
2. Select the "Properties" command in the context menu.
3. Enter the value in the "Initial measuring value" field.

See also

Aligning dimensionings (Page 301)

14.7.3 Aligning dimensionings

Requirement

The dimensionings are created.

See chapter Creating dimensionings (Page 300).

Alignment options

The following options are available for dimensionings:

- Move
- Center
- Height adjustment

Move dimensioning text

1. Single-click twice on the dimensioning text.
   Upon the second single-click, a small box appears under the dimensioning text.
2. Move the boxes on the dimension line.

Center dimensioning text

1. Single-click on the dimensioning text.
2. Select "Options > Center dimension text" in the context menu.

Changing the height of the dimension line retroactively

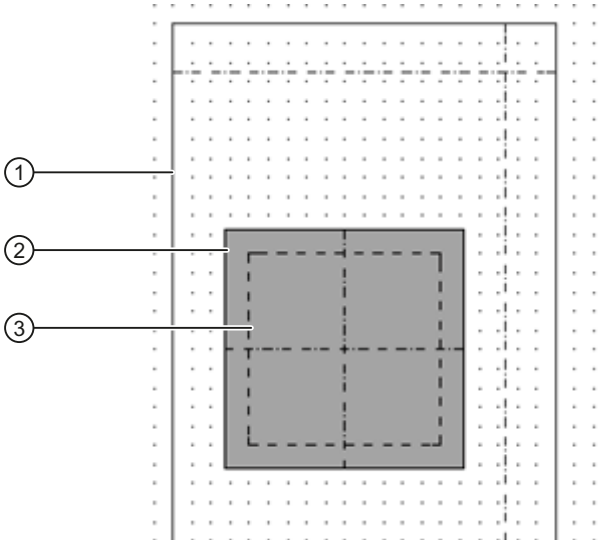
1. Single-click twice on the dimensioning text.
   Upon the second single-click, a small box appears under the dimensioning text.
2. Use drag&drop to change the distance of the box.
3. To transfer the new height of the dimension line to further dimensions of chain dimensioning or continuous dimensioning, follow these steps:
   - Single-click on the dimensioning.
   - Select the "Options > Rearrange chain dimension" command in the context menu.

Rearrange dimensioning

1. Select a dimensioning.
2. Select the "Options > Rearrange chain dimension" command in the context menu.
   - If only one dimensioning is selected, all other dimensionings in the chain are aligned to this dimensioning.
   - Dimensioning texts are centered.
   - If holes were created in the chain due to manual deletion of dimensionings, the chain is split up into several chains. Only the first chain will retain the set initial measuring value in this case.

14.8 Display of the mounting area on reports

The mounting area of a mounting plate is displayed below.



①	View in a cabinet report.
②	Mounting plate
③	Mounting area of the mounting plate (2).

14.9 Drilling planning

14.9.1 Drill drawing on the cabinet design

You can display the drill holes of placed components on the mounting plate in your cabinet design.

Requirement

A cabinet design and the mounting components have been created.

Creating drill holes

You have two options to define drill holes:

Option 1: In the attributes of the mounting plate or the attributes of the cable conduit:

See chapter: Defining drill holes in the "Drilling data" tab (Page 304)

Option 2: Manually placing a single drill hole from the Navigator:

1. Create a construction object in the Navigator underneath your cabinet: "@30|M41|A50|A30|A10|A70|A10!A61 construction objects"
2. Below it, create a drill hole in the Navigator: "New > B10 Drill hole/Slot"
3. Place the drill hole, for example, on the mounting plate, a rail or a cable channel in which you need it.

Showing drill holes on the cabinet design

1. Open the "Layers" window in the toolbar of the cabinet design.
2. Show the layer "42 Drilling".
3. Save your settings and confirm with "OK".

Note

Note: If you use objects described under option 2 as drill holes and want to hide them, they must have the functional classification "O3".

Afterwards, they can be shown and hidden with the "42 Drilling" layer.

14.9.2 Defining drill holes in the "Drilling data" tab

Requirement

The mounting components have been created.

Using the "Drilling data" tab

1. Standard model: Select "1 Site > L001 Building > G001 Floor > R001 Room > S001 Cabinet > UZ01 Cabinet, general > 02 Mounting parts > U2 C-rail, general".
2. Open the properties of an object which can only be attached by drilling holes in the mounting plate.
3. Select the "Drilling data" tab. Channels also have the "Drilling data" tab.
4. Specify the properties of the drill holes. You can specify three rows of drill holes.
5. Click the "Generate drill holes" button.

Selecting between static and dynamic drilling data

All objects which produce drill holes on the mounting plate must contain drilling data, in which the coordinates relative to the point of origin have been created.

Drilling data (static)

You use static drilling data for objects for which you want to define each hole separately, for example, for mounting plates. The list expands automatically when you enter the data.

Drilling data (dynamic)

You use dynamic drilling data for objects for which you want to define a regular hole distance, for example, for rails.

To define the hole distance, specify a starting position and a gap. The coordinates of the other holes are then calculated using this data.

Optional: Enter a width for oblong holes (slots).

Specifying the drill area

You have the possibility to specify locked areas in the dynamic drilling data, such as the drilling depth dependent on the placed device.

The "Relevant" setting describes the quantity of objects that require a corresponding drill hole.

Select from the "Relevant" list which objects is to be considered during the drilling.

You have the following options:

- No
- Object only
- Direct carrier only
- Object & direct carrier
- Object & all carriers
- All carriers without object

For more information on the result of the drill holes in front and side view, see: Example: Select devices for drilling (Page 306)

Using a graphical drilling scheme

An object can also contain a corresponding graphic as a drill scheme, which means you will not have to create the drill holes. Save the graphic in the "Graphic" field in the "Attributes > Installation/3D data" tab.

If you want to export the drilling data, however, you will still need to specify the coordinates of the drill holes.

Initially define the position of the drill holes using coordinates.

1. In the "Mounting" group, drag&drop the object of a drill hole into the field that you want to add to your busbar or cable channel.
2. Define the coordinates at which the desired drill hole should be set later.
   The coordinates are calculated relative to the higher-level/current objects, whereby Y is the offset in the up/down direction (positive = up) and Z is in the depth direction (positive = down).
   Note: All coordinates must be specified. An empty field is not a permissible input; use the value "0" instead.
3. Add all drill holes as described in step 2.
4. Confirm your entries.

Note

Note:

- The drill hole objects must have the functional classification "O3".
- The position of the drill holes cannot be subsequently changed in the Cabinet Equipment Assistant.

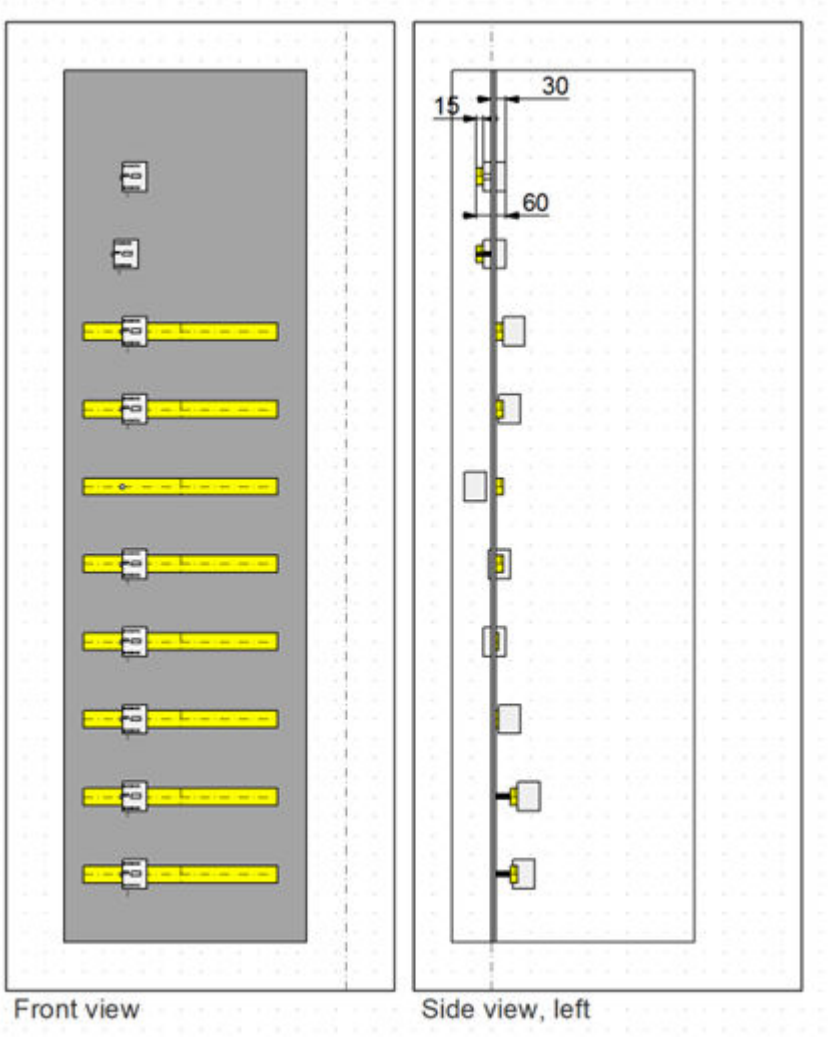
Result

When you place objects on the drilling drawing, the drill holes will appear.

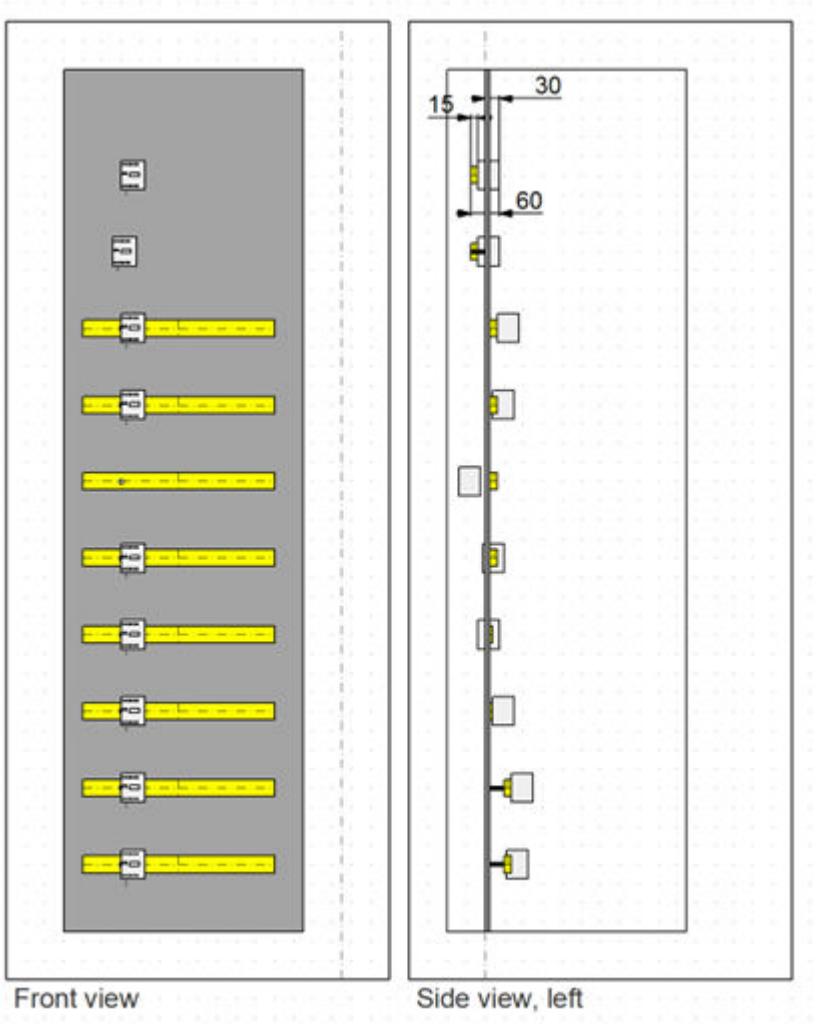
14.9.3 Example: Select devices for drilling

The "Relevant" attribute on the "Drilling data" tab influences the behavior of the drill holes in relation to the underlying carrier objects such as rail and mounting plate

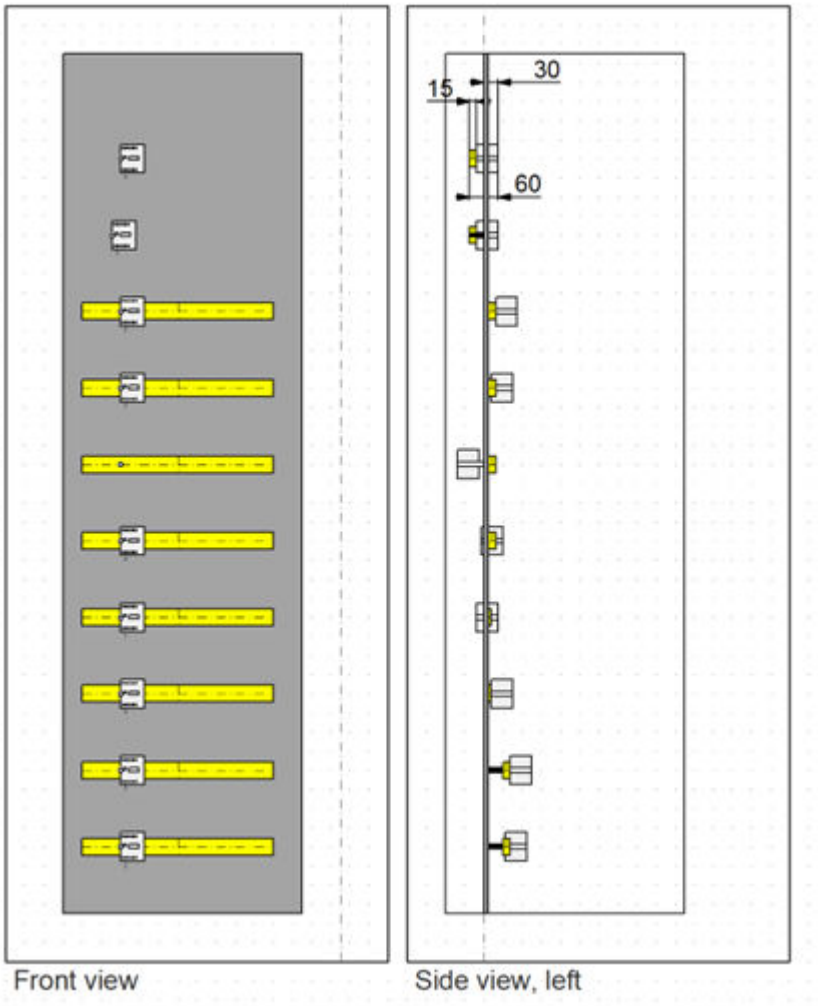
Example: Attribute relevant "Object only"



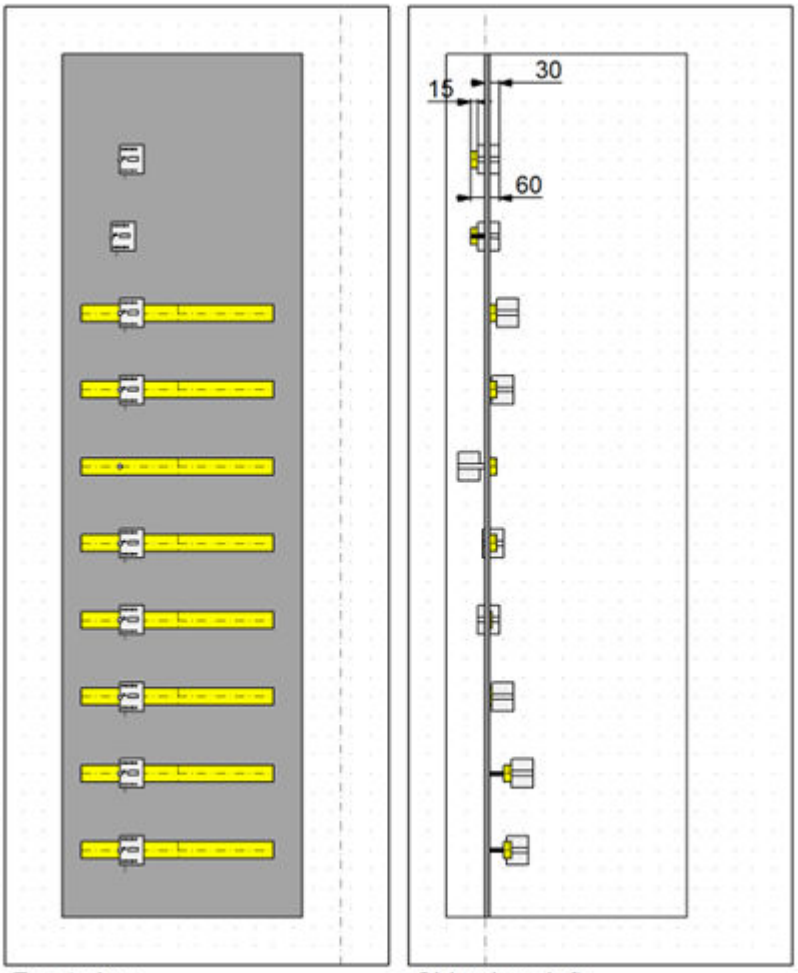
Example: Attribute relevant "Direct carrier only"



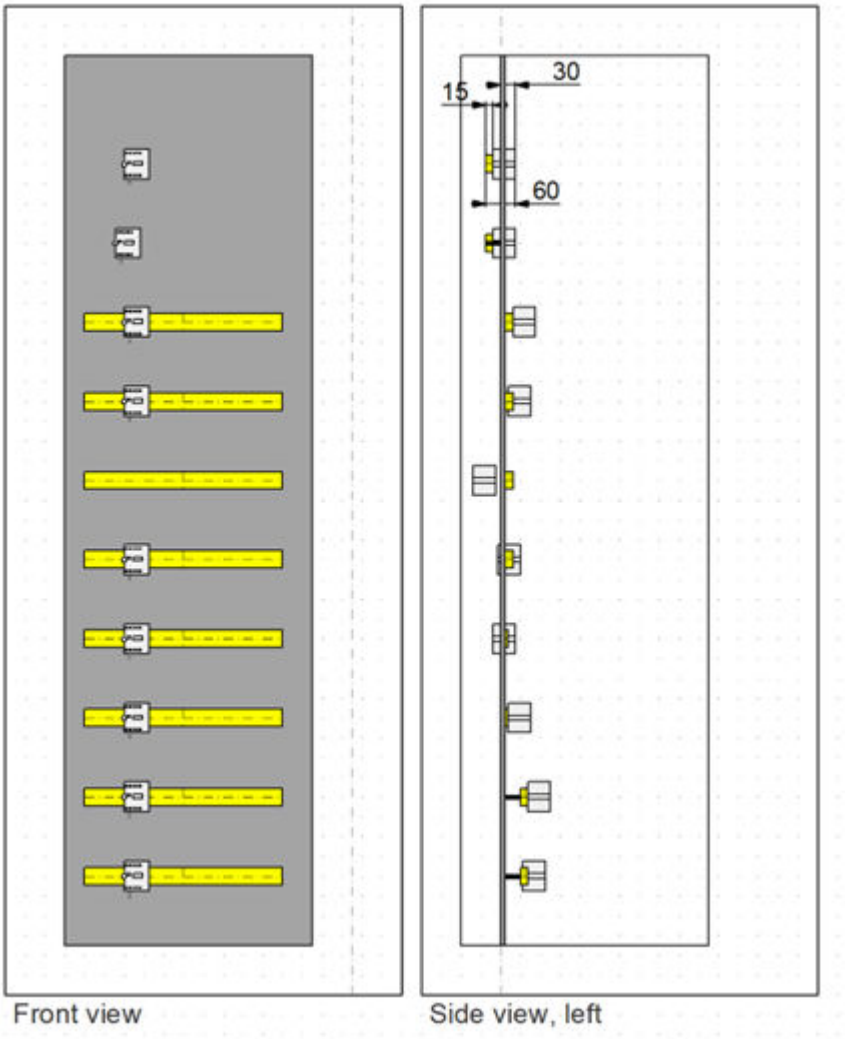
Example: Attribute relevant "Object & direct carrier"



Example: Attribute relevant "Object & all carriers"



Example: Attribute relevant "All carriers without object"



See also

Defining drill holes in the "Drilling data" tab (Page 304)

14.9.4 User interface reference for the "Drilling data" tab

The actual data of drill holes is entered on this tab. The coordinates of the drill hole are specified relative to the point of origin. The tab displays object data, along with the dynamic or static drilling data of an object.

"Object data (display)" control group

Control element	Description
"Width" field	This field displays the width of the object in millimeters. You edit the width of the object in the "Installation/3D data" tab.
"Height" field	This field displays the gap between the object and the edge of the drawing in millimeters. The object is placed at the point of origin. Example: The object has a height of 1800 mm. The "Height" field displays the value "+- 900". You edit the height of the object on the "Installation/3D data" tab.
"Angle" field	This field displays the angle of rotation, if the object has been rotated. The angle is calculated automatically when the object is rotated.
"Generate drill holes" button	Generates the corresponding drill holes based on the specified values

"Drilling data (static)" control group

Control element	Description
"Drill hole number" column	This column displays a consecutive number.
"Diameter" column	This field displays the diameter of the drill hole. Default: Diameter in millimeters.
"Center X" column	This field displays the center of the drill hole on the X axis.
"Center Y" column	This field displays the center of the drill hole on the Y axis.
"Model" column	Select the configuration of the hole from this list. For example: Thread, metric.
"Mounting distance" column	
"Relevant" column	Defines the drilling depth depending on objects and restricted areas. You have the following selection possibilities: • No • Object only • Direct beam only • Object & direct beam • Object & all beams • All beams without object
"Use static data" option	If this option is activated, the entries in the "Drilling data (static)" table will be used to calculate the drill holes.

"Drilling data (dynamic)" control group per row

Control element	Description
"Center X" field	This field displays the center of the drill hole on the X axis. Note: Enter a value in the "Center X" or the "Center Y" field.
"Center Y" field	This field displays the center of the drill hole on the Y axis. Note: Enter a value in the "Center X" or the "Center Y" field.
"Diameter" field	This field displays the diameter of the drill hole. Default: Diameter in millimeters.
"mm" list	Select the desired unit from this list.
"Hole spacing" field	This field displays the gap between the individual drill holes. Default: Distance in millimeters.
"Width (for elongated hole)" field	This field displays the width for elongated holes (slots).
"Configuration" list	Select the configuration of the hole from this list. For example: "Thread, metric".
"Mounting distance" field	Specifies the mounting distance.
"Relevance" list	Defines the drilling depth depending on objects and restricted areas. You have the following options for selection: • No • Object only • Direct beam only • Object & direct beam • Object & all beams • All beams without object

"Mounting" control group

Control element	Description
"Show accessory position"	
"Key"	Calculated automatically from the coordinates of the added objects.
Coordinate "X"	Offset on the X axis
Coordinate "Y"	Offset on the Y axis
Coordinate "Z"	Offset on the Z axis
"Installation component"	Drag&drop the object of a drill hole into the field that you want to add to your rail or cable channel.
"Protocol" field	Displays feedback, such as when overlaps would be created between the hole and cutouts.

14.9.5 Placing objects and drill holes on the drilling drawing

Requirement

All objects which produce drill holes on the mounting plate contain corresponding information on the "Drilling data" tab.

You have created the drilling plan underneath the mounting plate.

Procedure

1. Select the "Locations" tab in the Navigator.
2. Open the drilling plan.
3. Place a front view on the drilling plan.
4. Select the mounting plate in the Navigator.
5. Right-click on the front view which has been placed on the drilling plan.
6. Select the "View/Slice > Place the objects selected in the tree, keep view" command in the context menu.
7. Save the changes.

Result

All objects, which have drilling data and are directly placed on the mounting plate, are created.

You can now take dimensions or place other clips or holes.

14.10 Cabinet wiring and routing

14.10.1 Automatic wiring in design diagrams

Requirement

The affected objects must be connected correctly in the engineering data beforehand. This is e.g. the case if a circuit diagram was created in the preliminary work.

ET objects are correctly connected when the connectors fit. In many cases, COMOS will try to prevent incorrect connectors being connected at all.

Performing automatic wiring

1. Open a design diagram.
2. Right-click on the document to select "View/Slice > Cable routing" in the context menu. The shortest route is calculated in the view for the cable channels between all connected connectors of the type ET/MSR.

Cable routing works for the following views with the "Symbol layout From front" setting:

- XZ (front)
- XZ (back)
- XY (top)

The cables are also set in the cable route segments used for wiring. In other words: In the engineering data, the cable is entered in the corresponding attribute of the route.

14.10.2 Types of cabinet wiring

Cabinet wiring is started at the view object. There are two types of cabinet wiring:

- With a closed channel system
- With an open channel system

Closed channel system

To check whether a system is closed, click the "View/Slice > Check cable conduit" command in the context menu of a selected view.

If this is the case, wiring is only performed through ducts. These can also be defined via virtual channels for the wiring to work in a closed system. These virtual channels connect individual networks via defined points.

The individual networks can be distributed across multiple views. You must ensure that the cable routing is synchronized across all views; for this to work, a link object must be defined with a unique reference.

The following attributes control whether wiring may be performed through a duct:

- Channel type: You can only wire connections that are of the same type as the duct.
- Allocated to: The code calculates a rough estimate of the available space, where the following applies:
  The attribute "Y00A03525 Maximum filling degree" is checked on the "Technical specifications" tab.
- "Key": Control value composed of set cable options of a duct or cable conduit. It must match the key on the cable.

Open channel system

In an open channel system the connections are wired directly via virtual channels. The virtual channel contains the linkpoint. To interconnect them directly place them on the cabinet design next to each other.

See also

Net Optimized Routing (Page 320)

Open air cabling (Page 321)

14.10.3 Calling wiring

Cabinet wiring is called in the design diagram. There are various types of routing.

They are controlled by the "CabinetRoutingMode" script option and set on the report template

Procedure

1. Select a view in the design diagram.

- Click the "View/Slice > Check cable conduit" command in the context menu.
- Click the "View/Slice > Check cable conduit" command in the context menu.

Result

Wiring is drawn.

14.10.4 Using automatic wiring

Requirement

- The cabinet object has been created in the engineering data.
- The virtual channel has been created.
- A linkpoint has been placed.

Automatic routing through channels and linkpoints

The first and last linkpoint must be located on a cable channel. For more on this, see Creating virtual channel and placing the channel part (Page 284).

The linkpoints are included in the automatic routing as follows:

- If a linkpoint is connected with a cable tray by means of a connector, wiring takes place by means of the connected cable channel. This applies even if the direct connection to the linkpoint is shorter.
- Then, the linkpoints that are not connected to a cable channel are checked. These linkpoints are used if they result in a shorter spatial connection. If the shortest connection uses channels, then the linkpoints remain unused.

- The linkpoints are sorted by name and used in this order. The default sorting in the Navigator corresponds to the order of the names.
- The linkpoints are used if they were placed "From front" in the symbol layout.

General routing is performed taking into consideration the specified cable types and the allocation and depending on the defined ducts.

To connect objects in the cabinet using free channels and linkpoints, follow these steps:

1. Right-click in the free area of the cabinet design outside the views
2. Select this context menu command: "Views/Slices > Cable routing"

The "Views/Slices > Cable routing" command can also be used for a single view.

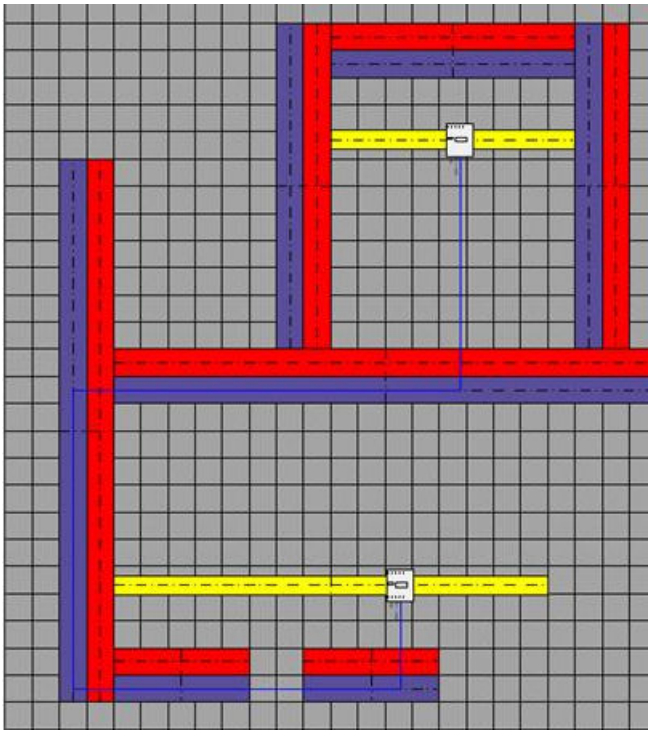
Skipping conduits during routing

You can exclude individual conduits from the routing and skip them. For example, to distinguish between high voltage and low voltage conduits.

1. Open the properties of a cable conduit.
2. Define the following attributes in the "Control cabinet wiring" group:
   Search distance at start point: Defines the point as of which the found cable conduits are to be skipped.
   (Base object in the base project: "@40|A20|Y00|A10|A10|Y00T00011|A01|B01|Y00T00011|Y00A05889 Search distance at start point")
   Search distance at stop point: Defines the point as of which routing is to continue as usual in conduits of the matching type.
   (Base object in the base project: "@40|A20|Y00|A10|A10|Y00T00011|A01|B01|Y00T00011|Y00A05890 Search distance at stop point")

Result

The cabling runs through the channels and virtual channels.



14.10.5 Connections in the control cabinet diagram

Since COMOS version 10.2.1, electrotechnical connections can be created as connection objects.

As of COMOS 10.4.1, this connection object can also be used when planning on the circuit diagram.

Requirement

The connection mode "Create connection objects for the connection types "E" and "I" mode (as of 10.2 SP1)" has been set in the project options.

Connection objects

The connection object in the circuit diagram is not created manually, but uses the connection object from a circuit diagram and results from creating connections using routing.

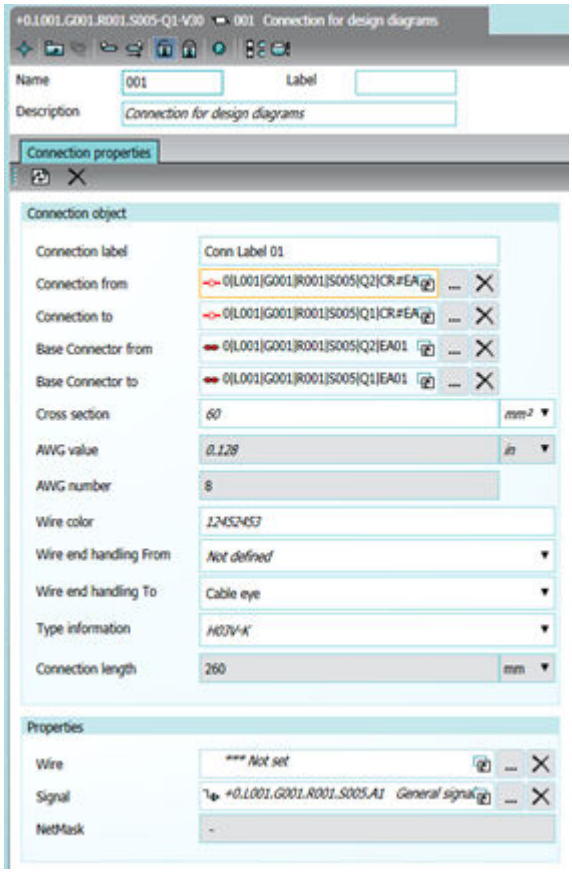
It is influenced by the "CabinetRoutingMode" script option of the report template.

Additional information is available in the "EI&C Administration" manual in the chapter "Report template of the control cabinet diagram"

Cabinet wiring is called in the design diagram. There are various types of routing:

- "Cabinet Routing Mode = 0":
  The connections in the control cabinet are still based on the original connection in the circuit diagram.
- "Cabinet Routing Mode = 1" or "Cabinet Routing Mode = 2":
  New CR connections are created in the circuit diagram and a connection is created between them as a connection object.
  For connections that correspond to an already configured connection on the circuit diagram, the information at the connection object of the circuit diagram is also written to the connection object of the control cabinet diagram as soon as the routing has been executed.
  Note: If information on the connection object on the circuit diagram is subsequently changed, the information in the circuit diagram is not automatically updated. In case of doubt, run the routing again.

Properties of the connection object in the control cabinet plan



Result

The connection object is created under one of the devices in the V30 folder.

14.10.6 Net Optimized Routing

The shortest cable length of a connection can be calculated with Net Optimized Routing . Entanglements are also possible here. The devices do not have to be directly connected to each other at their connectors. Instead, they only need to be grouped using attributes, for example, because they are at the same logical potential.

Requirement

- CabinetRoutingMode 1 or 2 was set on the report template.
  Additional information is available in the "EI&C Administration" manual in the chapter "Report template of the control cabinet diagram"
- The "Cabinet Routing" connection type is available.
- Engineering in the circuit diagram is complete and all information has been defined on the connection object.

Attribute for grouping connectors

The information about which connectors are grouped is defined with the "Cabinet Routing NetMask" connector attribute. This corresponds functionally to the information of the attribute "Connected To".

Every connection with a valid "CRNetMask is considered and connected during Net Optimized Routing.

Note that all connectors must have a defined attribute.

1. Open the extended connector attributes at the connector of your device.
2. Set a value, e.g. L1*, at the "CRNetMask" attribute.
   This value is then automatically set as a prefix to the name of the connector.
   Note that the name may not contain any special characters.

Query "NetMask"

1. Open the query:
   Right-click on the design diagram "New > B10 Query: Cabinet Routing Netmask".
2. Drag&drop your control cabinet from the "Locations" tab in the Navigator into the "Start object" field.
   All connectors in your control cabinet and the defined connector attributes are listed.
3. Check your entries.
4. Open a design diagram.
5. Right-click in the free area of the control cabinet diagram outside the view.
6. In the context menu, select the command: "Views/Slices > Cable routing".

Result

All connections are re-routed.

14.10.7 Open air cabling

In control cabinet wiring, connections are usually routed through cable ducts.

You also have the option of direct connections outside the cable ducts.

There are two available options:

- By means of linkpoints and virtual channels
- Through direct connection of the devices

Open air wiring by means of linkpoints

If the channel parts are not placed on a channel, open air cabling is permitted.

1. Place a linkpoint in the view without the linkpoint part being placed on a channel.
   The channel part is not connected to a channel by means of a connector. The channel part is called the open air channel part.
2. The following options are available:
   – Connect the open air channel part with a channel part on a channel.
   The open air channel part is now the starting point or end point of open air cabling.
   See also chapter Creating virtual channel and placing the channel part (Page 284).
   – Connect the components directly with an open air channel part.
3. Place additional open air channel parts.
4. The following options are available for cabling:
   – Use automatic routing.
   The open air channel parts that are not connected are used during automatic routing.
   If an open air channel part is closer than the next channel, routing into the open air channel part takes place.
   – Connect the open air channel parts manually.

The connections of the open air cabling are taken into consideration in the length calculation of the wiring.

The open air cabling is permitted in the following cases:

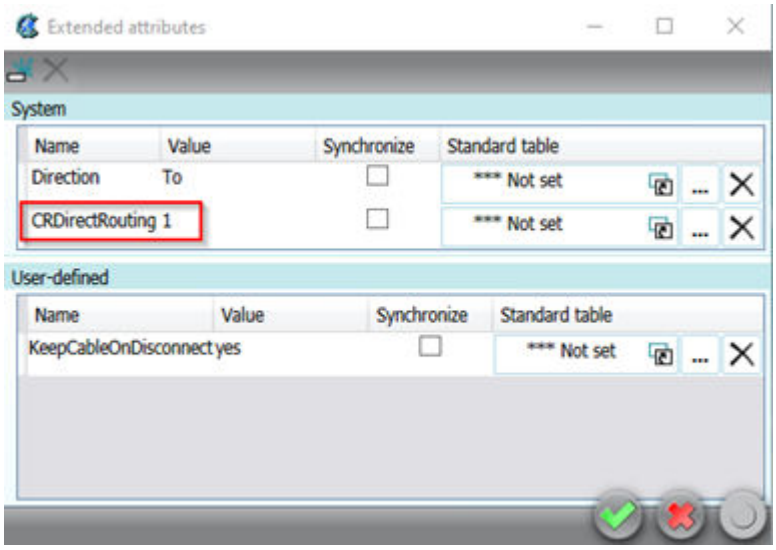
- The open air channel parts are located in the same view
- The open air channel parts are located in different views
- The open air channel parts are located in different documents

Open air wiring through direct connection of devices

Option 1:

The "Direct Routing Mode" can be activated individually at each connector.

1. Open the "Expanded properties" of a connector.
2. Set the value of the attribute "CRDirectRouting" to "1" to allow a direct connection of the connectors during routing.
3. Optional: Set the value of the attribute "CRDirectRouting" to "0" to prevent a direct connection of the connectors during routing.



Option 2:

When using more than one connector, the direct connection can also be defined on the device.

1. Select the "Installation/3D" tab in the properties of a device that you want to connect directly.
2. Set the attributes "Direct Routing for Top Connectors" and "Direct Routing for Bottom Connectors" to "yes" or "no" if you do not want them to be taken into account.

Note

Note:

- If you use the attributes "Direct Routing for Top Connectors" or "Direct Routing for Bottom Connectors", the CRDirectRouting attribute is not set.
- If the extended attribute "CRDirectRouting" has been set on a connector, this overrides the setting on the device.
- Direct routing is used only if it is set on both connectors and both devices or their connections are placed in the same view.

14.10.8 Length calculation of wiring

14.10.8.1 Coordinate system for 3D component calculations

Point of origin in 3D body

The point of origin in the 3D body is influenced by the following setting:

Component properties, "Assembly/3D data" tab, "Position of origin" group

The following settings must be present in the COMOS standard:

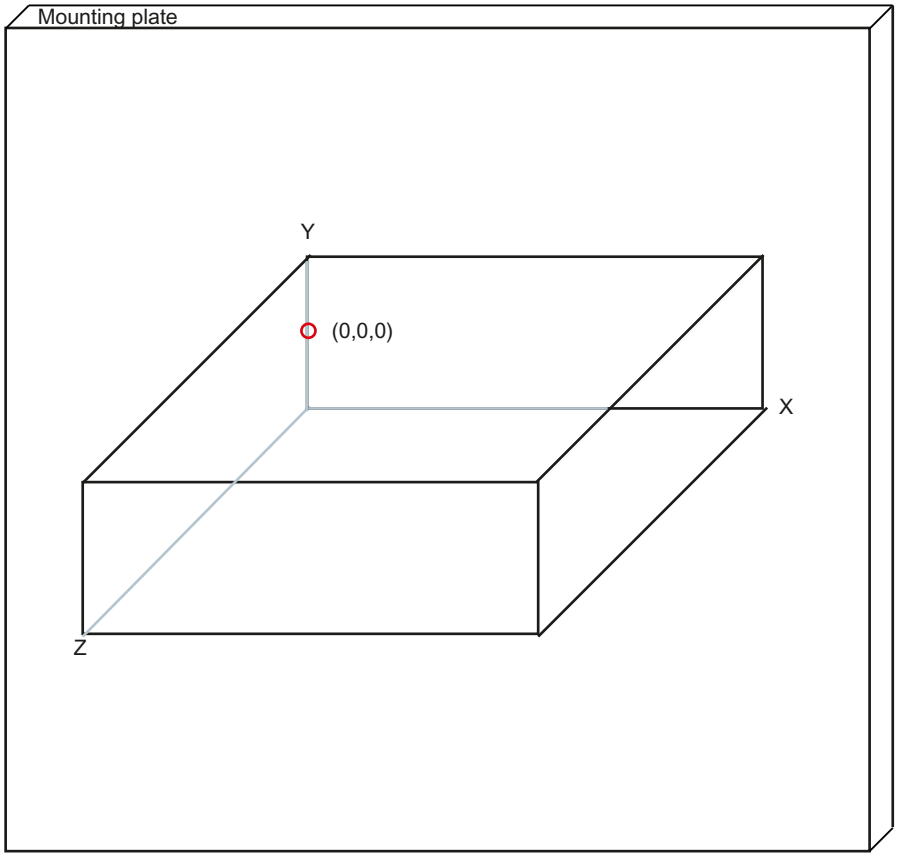
- Left: Switched on
- Back: Switched on
- All other options are deactivated.

A deviation from these defaults results in an incorrect coordinate calculation in cabinet construction.

Taking into consideration the defaults listed above, the following point of origin results for the calculation of coordinates in components:

- X = 0 (according to the "Left" option)
- Y / 2 (half component height)
- X = 0 (according to the "Back" option)

The point of origin for an example body is entered in the graphic below:



Types of coordinate calculation

The point of origin in 3D bodies is the same for both types of coordinate calculation:

- Routing via automatically generated connectors
  See also chapter Wire length calculation / routing via automatically generated connection points (Page 324).
- Routing via manually specified connection coordinates
  See also chapter Wire length calculation / Routing via manually specified connection points (Page 326).

14.10.8.2 Wire length calculation / routing via automatically generated connection points

Requirement

- The following attribute is deactivated:
  Properties, "Assembly/3D data" tab, "Cabinet wiring" control group. "Routing via connector coordinates" option
- The 3D coordinates of the components have been entered.

Routing by means of connector coordinates is deactivated

This calculation type is the default setting.

Because the design diagrams are true to scale, the drawn connections are used to determine required lengths. To do this, the connection lines are incorporated into the middle of placed channels.

Half the depth of a component is used for the Z depth.

If connectors are drawn in the diagrams, their position in the space is used. In this case, the wiring also takes the relevant lateral view into account when determining the Z depth.

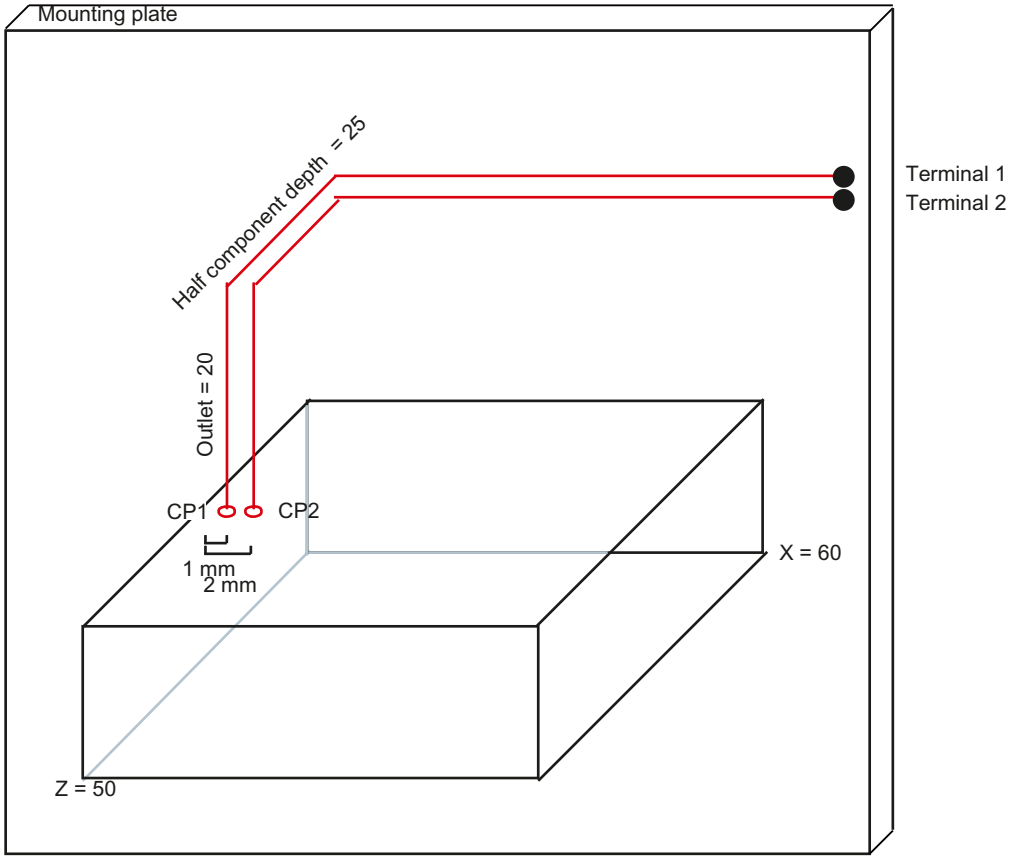
The following points apply:

- Calculating the length of a connection:
  Length plus half the depth of a component.
- Creating connectors
  Starting at the left edge of a component at distances of 1 mm.
- Connector direction/connection direction
  The connector direction is the direction in which the search for the entry point to the next cable channel is conducted, starting at the connector.
  The connector direction is determined based on the connector distance from the outer edge of the component.

Example of a wiring via automatic connection points

The following component is wired as an example in the graphic below:

- Width (X) = 60 mm
- Height (Y) = 20 mm
- Depth (Z) = 50 mm
- Position of terminals: X = 70 mm
- Position of terminals: 20 mm above top edge of component
- Position of EA1 and EA2: assigned automatically by position



- Length of cable routing for EA1 to terminal:
  Outlet 20 mm + half component depth 25 mm + (70 mm - 1 mm) = 114 mm
- Length of cable routing for EA2 to terminal:
  Outlet 20 mm + half component depth 25 mm + (70 mm - 2 mm) = 113 mm

14.10.8.3 Wire length calculation / Routing via manually specified connection points

Requirement

- The following attribute is activated:
  Properties, "Assembly/3D data" tab, "Cabinet wiring" control group. "Routing via connector coordinates" option
- The 3D coordinates of the components have been entered.
- The 3D connector coordinates are entered at the objects.

Routing via connector coordinates is activated

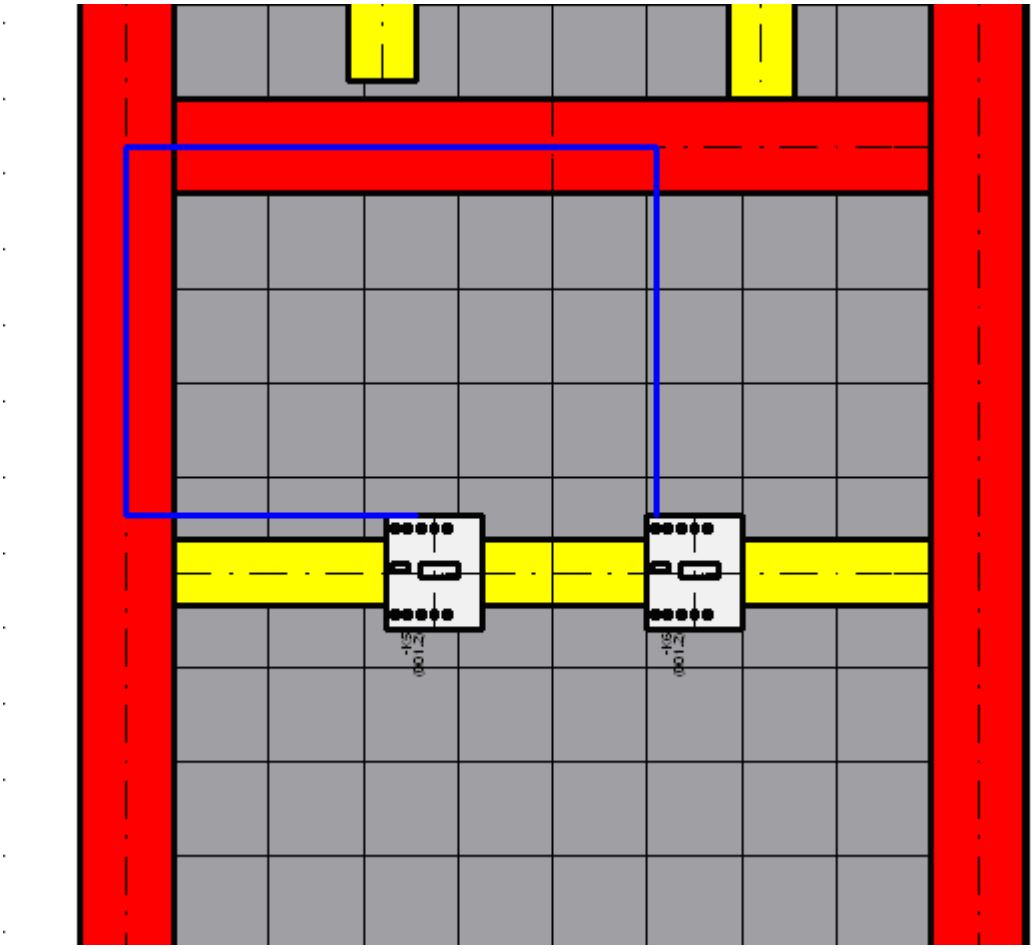
- Overview:
  - Considered Z depth: controlled by means of connector coordinates.
  - Position of the connectors: controlled by means of connector coordinates.
  - Connection direction: controlled by an extended attribute.
- The connector coordinates are managed by means of extended connector attributes.
  When the "Routing via connector coordinates" option is activated, all attributes mentioned below must be present. Without coordinate information, report connectors are not created and the routing is not conducted.
  The following attributes are no longer taken into consideration:
  - "Installation-/3D data" tab: "Connector top"
  - "Installation-/3D data" tab: "Connector bottom"

  All dimensions in mm. All dimensions relate to the point of origin of the component.
  The extended attributes are:
  - "XCoord"
  - "YCoord"
  - "ZCoord"
- The connector direction is managed by means of an extended attribute.
  The extended attribute is:
  - "'InletDirection"
    The following values are possible:
    -1: Automatic calculation (distance to component edge). This option corresponds to the behavior when "Routing via connector coordinates" is deactivated.
    0: Connection from top
    1: Connection from bottom
    2: Connection from left
    3: Connection from right
    The connector direction is viewed locally to the component. A rotation influences the connector direction.

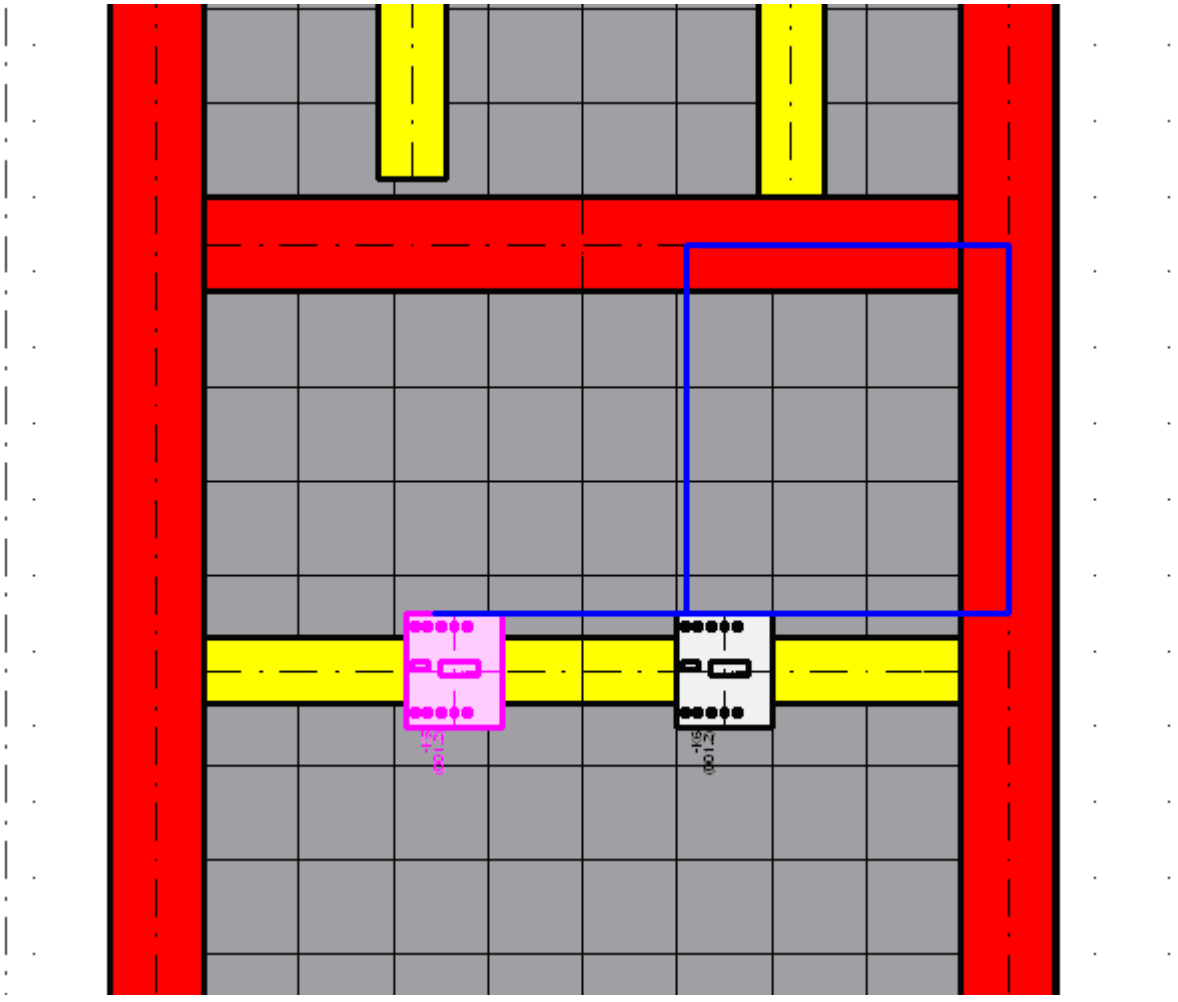
See chapter Working with the extended attributes of connectors (Page 198) for information on extended attributes.

Example of the connector direction

In the following figure, the value "2" (connection from left) has been entered for the connectors of the extended attribute "InletDirection" on the left component:



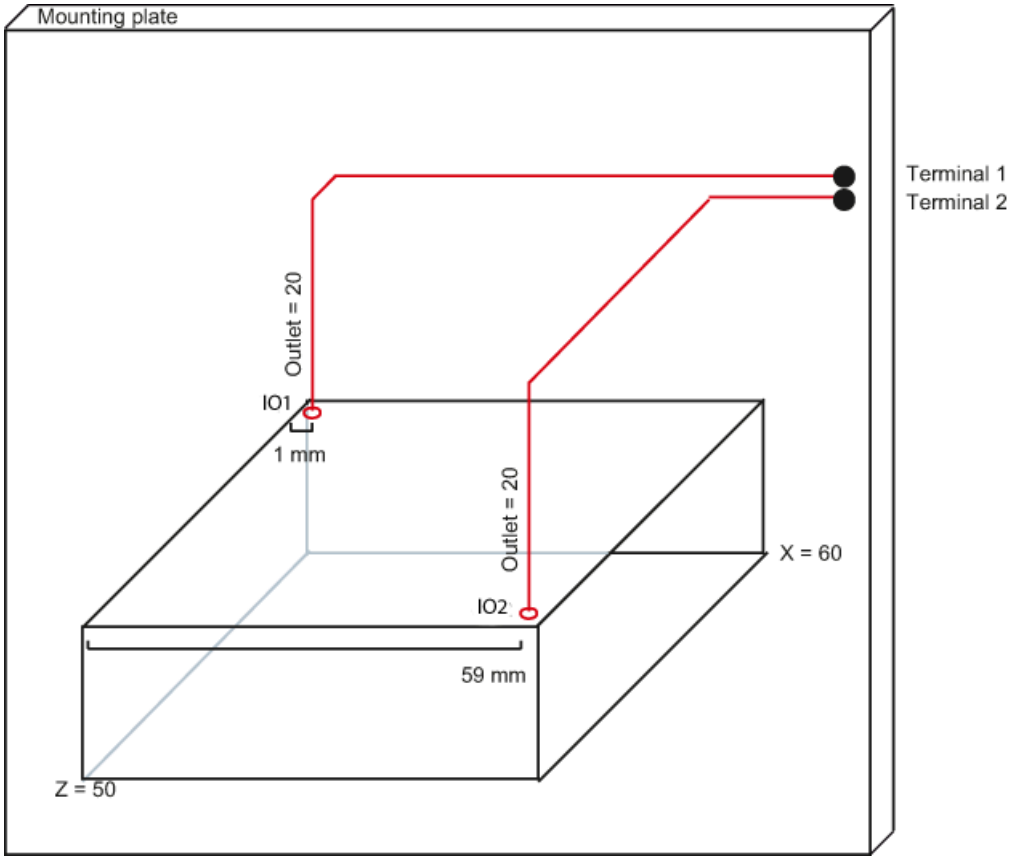
In the following figure, the value "3" (connection from right) has been entered for the connectors of the extended attribute "InletDirection" on the left component:



Example for wiring by means of connector coordinates

The following component is wired as an example in the graphic below:

- Width (X) = 60 mm
- Height (Y) = 20 mm
- Depth (Z) = 50 mm
- Position of terminals: X = 70 mm
- Position of terminals: 20 mm above top edge of component
- Position of EA1 and EA2: entered by the user



- Length of cable routing for EA1 to terminal:
  Outlet 20 mm + Z coordinate of connector 1 mm + (70 mm - 1 mm) = 90 mm
- Length of cable routing for EA2 to terminal:
  Outlet 20 mm + Z coordinate of connector 49 mm + (70 mm - 59 mm) = 80 mm

Activating routing in opposite direction

You can overwrite the preset routing direction on the cabinet design at each placed object. This means routing can take place in both directions.

1. Navigate to the object.
2. Open the attributes of the object.
3. In the "Control cabinet wiring" group, activate the option "Permit search in opposite direction".
4. Connect the connectors of two objects on the cabinet design.

5. Right-click the connection to open the context menu.
6. Select "View/Slice > Cable routing"
   The connection is rerouted.
   Note: When you have activated routing in the opposite direction, the connection line routes graphically over your object. Technically, this means that the cables in the cabinet are routed behind your component. The additional cable length required is taken into account automatically based on the dimensions of your component.

14.10.8.4 Automatic offset calculation

A depth offset between the component and cable channel is taken into consideration. The depth offset is managed by means of an extended attribute.

The extended attribute is:

- "DistanceFromChannel"
  The attribute is created automatically during the routing.

This means that the calculated cable length is much closer to the length that is actually required.

The automatic offset calculation is independent of the "Routing via connector coordinates" option.

14.11 Cabinet Equipment Assistant

14.11.1 Opening the Cabinet Equipment Assistant

The "Cabinet Equipment Assistant" offers functions for easy cabinet engineering.

Opening the plugin

"Plugins > Automation > Cabinet Equipment Assistant"

The "Cabinet Equipment Assistant" tab is called in the Navigator.

See also

Using the Cabinet Equipment Assistant (Page 332)

14.11.2 Controlling call via script option

The Cabinet Equipment Assistant can be opened automatically by opening a cabinet design.

Procedure

The script option can be set at the template for all drawings.

1. Select "Report template > Open report template" from the context menu of the diagram.
   The report template opens in the working area. Change to it.
2. Select "Options" from the context menu of the report template.
3. Set the script option `AutoStartToolWindow = TRUE`
   If the option is not available, add it manually.
   The Cabinet Equipment Assistant starts automatically when you open the drawing.

See also

Using the Cabinet Equipment Assistant (Page 332)

14.11.3 Using the Cabinet Equipment Assistant

Requirement

The Cabinet Equipment Assistant is open.

See chapter Opening the Cabinet Equipment Assistant (Page 331).

Procedure

1. Open a cabinet design.
2. Mark an item of equipment on the cabinet design.
3. The edit fields of the Cabinet Equipment Assistant become active.
4. Edit the properties of the component that you selected on the cabinet design.
   See chapter Function overview of the Cabinet Equipment Wizard (Page 333).
5. Optional: Synchronize dependent documents
   - Synchronize all dependent documents:
     Click "Synchronize dependent documents with 3D"
     See also chapter "Document" field (Page 333).
   - Alignment using coordinates.
     See also chapter: "Coordinates" field (Page 334)
   - Synchronization of the rail currently being edited with the currently open document:
     Click "Synchronize with 3D".
     See also chapter "Options" field (Page 335).

Note

You cannot reset changes to the position of objects on the control cabinet diagram using the "Back" button if you have made these changes in the control cabinet Equipment Wizard.

See also

Actions (Page 336)

14.11.4 Function overview of the Cabinet Equipment Wizard

14.11.4.1 "Document" field

Icon	Tooltip	Description
	Synchronize dependent documents with 3D	This function saves the changes in all views and all affected documents. The 3D view is also synchronized. If it is important for editing of 3D data to be available simultaneously on all affected documents, it is necessary to run the "Synchronize dependent documents with 3D" function before closing the document currently being edited. Background: If an object is placed multiple times, changing the position of the object on the current report (or in the 3D display) also affects the position on other reports. Example: Simultaneous use of mounting diagram and design diagram. The "Synchronize dependent documents with 3D" option transfers all changes to all other reports on which the component is also placed. The reports do not need to be open. This applies both to manual editing of the object on the report and to changes carried out via the Cabinet Equipment Assistant. This function is also useful for design diagrams: If multiple design diagrams are open, not all design diagrams are always updated automatically at the same time. This function forces updating to take place.
		The field displays the cabinet design on which an object is selected.
	Cable Routing - complete	Routes the cables in the entire cabinet design according to specifications.

14.11.4.2 "Object" field

Icon	Tooltip	Description
	"Previous object on rail"	Selects the object one position to the left of the currently selected object.
		The box shows the object selected in the cabinet design.
	"Next object on rail"	Selects the object one position to the right of the currently selected object.
	"List of placed objects"	Opens the "Element of rail" tab.
	"Maximum available space exceeded".	Selects objects that were placed next to the rail due to lack of space.

14.11.4.3 "Edit" field

The functions offered change depending on the selected object.

Icon	Tooltip	Description
	"Move to front"	Moves the selected object to the beginning of the rail.
	"Move to back"	Moves the selected object to the end of the rail.
	"Remove from rail"	Removes the selected object from the rail.
 	"Rotate"	Rotates the selected object.
	"Move to left"	Moves the object one position to the left.
	"Move to right"	Moves the object one position to the right.
	"Move up"	Moves the object one position up.
	"Move down"	Moves the object one position down.
	"Insert at the end"	Attaches the selected object to the rail. See also Inserting objects on the cabinet design (Page 336).

Functions

- Multiselection

14.11.4.4 "Coordinates" field

Coordinate mode	Coordinate origin	Option	Coordinates
2D Abs (2D absolute)	Point of origin top left	Two-dimensional alignment of a placed object.	X/Y*
2D RV (2D relative to view)	Point of origin of view	Two-dimensional alignment of a placed object relative to the currently selected view.	X/Y*
2D RSO (2D relative to subordinate object)	Point of origin of the subordinate object	Two-dimensional alignment of a placed object relative to the subordinate mounting plate.	X/Y*
2D RTO (2D relative to Point of Origin)	Manually defined point of origin.	Two-dimensional alignment of a placed object relative to the point of origin of the design diagram.	X/Y*

Coordinate mode	Coordinate origin	Option	Coordinates
3D Abs (3D absolute)	Point of origin top left	Three-dimensional alignment of a placed object.	X/Y/Z*
3D RV (3D relative to view)	Point of origin of view	Three-dimensional alignment of a placed object relative to the currently selected view.	X/Y/Z*
3D RSO (3D relative to subordinate object)	Point of origin of the subordinate object	Three-dimensional alignment of a placed object relative to the subordinate mounting plate.	X/Y/Z*

*Also note the displayed tooltip.

Note

Note: The coordinates of an object can only be specified when using a mounting plate. The coordinates are grayed out for objects placed on rails.

See also

Aligning an object using coordinates (Page 338)

Defining the point of origin (Page 266)

14.11.4.5 "Options" field

Checkbox "Synchronize with 3D"

Status	Effect
Checkbox selected	The data is synchronized with the 3D view in the background. If required, report objects are created and the objects are placed in the view.
Checkbox cleared	The data is not synchronized with the 3D view. The objects therefore remain on the diagram even if it is currently not visible in the 2D view. If non-synchronized changes exist, the "Synchronization with 3D required" switch appears and some of the control elements are disabled.

"Transform dependent objects" checkbox

Status	Effect
Checkbox selected	Objects placed on a DIN rail, mounting plate, etc. can be moved without having to select them separately, e.g. by selecting the rail. It is enough to simply select the rail, for example.
Checkbox cleared	Objects placed on a DIN rail, mounting plate, etc. can be moved by separately selecting them in the context menu: "Options" > "Select dependent object".

Checkbox "Own grid"

Select this checkbox to define a grid. The grid specified here will be used for actions from the Cabinet Equipment Assistant. It overwrites the pattern defined in the diagram, but only for subsequent actions. The "Grid" box rounds off values automatically to two decimal places.

14.11.4.6 "Properties" field

The functions offered change depending on the selected object.

Icon	Tooltip	Description
Box	"Length"	Box for inputting the length for a selected object.
	"Apply"	Checking in the information at the object.
	"Extend (opposite direction)"	Extends the selected object to the left up to the next object.
	"Extend (alignment)"	Extends the selected object to the right up to the next object.
	"Maximize"	Extends the selected object in both directions up to the next objects.

See also

Aligning objects (Page 338)

14.11.5 Actions

14.11.5.1 Inserting objects on the cabinet design

To insert objects from the Navigator in the cabinet design using the Cabinet Equipment Assistant, follow these steps:

1. Select the object to be placed in the Navigator.
2. Select the insertion location (object) on the cabinet design.
3. Click "Options > Attach".
   The object to be placed is attached at the insertion location.

Note

You can also place objects with multiselection. If you place more objects at the insertion location than there is space available, the icon "Object > Maximum available space exceeded" is displayed. All excessive objects are selected by clicking "Maximum available space exceeded".

14.11.5.2 Placing objects on rail using the wizard

Open the rail wizard with "Object > List of placed objects". The "Element of rail" tab opens.

It includes a list of all objects. Gaps in front of or in between objects are also displayed as objects.

Selected objects are displayed in gray. Objects placed next to the rail are displayed in red.

Creating new objects

1. Select the rail in the cabinet design or in the Navigator.
2. Open the "Element of rail" tab.
3. Use drag&drop to move an object from the Navigator to the list.
   The object appears in the list and is placed at the front of the rail.

Editing objects

1. Select the rail in the cabinet design or in the Navigator.
2. Open the "Element of rail" tab.
3. Select an object.
   Note: A gap is also displayed as an object and can be selected.
4. Edit the objects using the commands in the right margin of the tab.
   See also "Edit" field (Page 334).

Inserting objects

1. Select the rail in the cabinet design or in the Navigator.
2. Open the "Element of rail" tab.
3. Select an object and drag it to the list of objects.
   Note: A gap is also displayed as an object.
   Option 1: Drag the device in the list between two objects.
   A black line appears and the object is inserted at this location.
   All subsequent objects under the black line are moved by the width of the inserted device.
   Option 2:Drag the device in the list onto a gap object.
   The object is inserted in this place in the gap and fills it completely or partly.
   All subsequent objects are only moved by the amount that the width of the inserted device exceeds the dimensions of the gap.

14.11.5.3 Aligning objects

Adjusting the length

1. Select the object on the cabinet design.
2. Enter the value in the "Length" box.
3. Save your entries by clicking "Apply".

Aligning at other objects

To dock an object on the drawing to another object, follow these steps:

1. Select the object on the cabinet design.
2. Extend the object in the required direction.
   The object is extended and ends flush with the next object.

See also

Aligning an object using coordinates (Page 338)

14.11.5.4 Aligning an object using coordinates

You can display the 2D or 3D coordinates of a placed object and move the object precisely by changing the coordinates.

Requirement

A cabinet design with mounting plate has been created and objects were placed on it.

Moving placed objects using coordinates

1. Select a placed object on the mounting plate.
   Note: Only individual objects that are not placed on a rail can be moved using coordinates.
2. Select the coordinate mode in the "Coordinates" group of the Cabinet Equipment Assistant.
3. Enter the X/Y or X/Y/Z coordinates to which you want to move the selected object. Note the displayed tooltips and dependencies to other objects on the cabinet design. Depending on the reference point to which you want to move the object, the coordinates can also have negative values.
4. Confirm your entries.

See also

"Coordinates" field (Page 334)

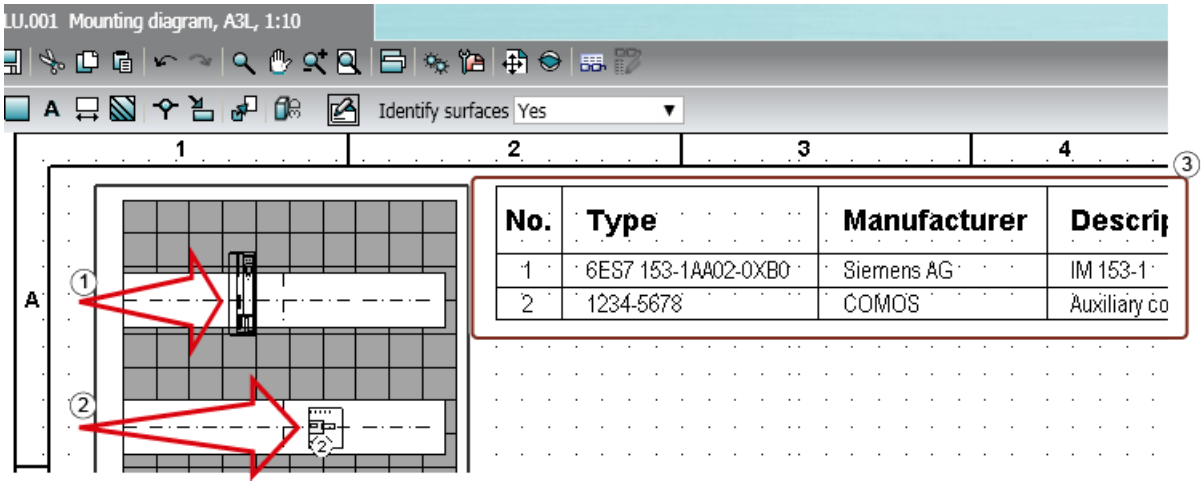
14.12 Using the mounting specifications

How the mounting specifications work

The mounting specifications are implemented in COMOS using the following components:

- Report with the "Design" drawing type
  The "Mounting diagram" is prepared accordingly in the standard DB
- Mounting-relevant information at the components, especially the "Article number" field
- "Mounting legend" symbol
- "Text via legend" option. This option replaces the label with the legend entry
- At the component: "Legend entry" field

If these components are prepared according to the COMOS standard, the following view results:



①	Component with legend entry. There is a "1" in the "Legend entry" field in the properties of the component.
②	Component with legend entry. In this case, the legend entry replaces the label and therefore appears at the placed component. There is a "2" in the "Legend entry" field in the properties of the component.
③	Legend table. Mounting-relevant information is taken from the properties of the components and listed here.

Mounting specifications in the cDB

1. Create the following report: "ELU | Mounting diagram".
2. Edit the mounting diagram.
   See chapter Mechanical design in the front view (Page 266).

3. Optional: Display the legend entry at the placed component
   - Open the document properties and the tab "SYS | System data".
   - Enable the "LEGENDE | Text via legend" option.
   - Example: See Position (2) in the graphic.
4. If not yet present on the mounting diagram: Place the "Mounting legend" symbol.
   The "Mounting legend" symbol creates an empty table.
5. Optional: Control the "Mounting legend" symbol.
   - Open the properties of the "Mounting legend" symbol and the "System data" tab.
   - "Allocation legend" option: Select an entry in the list.
   - "Output legend" option: Select an entry in the list.
6. Select a placeable engineering object, for example the "Auxiliary contactor" engineering object.
7. Open the properties and the tab "HSD | Order data".
8. Edit the "M001 | Article number" field and other manufacturer data.
   Save your entries.
9. Update the mounting diagram.
   - The table is updated in the mounting legend.
     The following mounting legend is prepared in the standard DB: Type, manufacturer, description, quantity.
   - Placed components: Tab "GD | Assembly/3D data": The "Legend | Legend entry" field is set by COMOS.
     Special case for deleted components: When a component is deleted from the mounting diagram, the last valid legend entry is retained in the properties of the component. With unplaced components, the "Legend entry" field therefore has no meaning.
   - When enabled: A circle and the legend entry appear at the placed component (in this case: auxiliary contactor).

Mounting specifications in the iDB

1. Create the following report: "A10 | Mounting diagram".
   The following report has already been prepared accordingly in the iDB:
   "Plant > Building > Floor > Room > Cabinet": "Mounting diagram"
2. Edit the mounting diagram.
   See chapter Mechanical design in the front view (Page 266).
3. Optional: Display the legend entry at the placed component
   - Open the document properties and the tab "`Y00T00001` | System data".
   - Enable the "`Y00A00236` | Text via legend" option.
   - Example: See Position (2) in the graphic.
4. "Mounting legend" symbol: The symbol currently needs to be prepared by the customer administrator.

5. Select a placeable engineering object, for example the "Auxiliary contactor" engineering object.
6. Open the properties and the tab "Y00T00025 | Order data".
7. Edit the "Y00A00302 | Article number" field and other manufacturer data.
   Save your entries.
8. Update the mounting diagram.
   - The table is updated in the mounting legend.
   - Placed components: Tab "Y00T00011 | Assembly/3D data": The "Y00A00236 | Legend entry" field is set by COMOS.
   - When enabled: A circle and the legend entry appear at the placed component (in this case: auxiliary contactor).

Controlling and evaluating 15

15.1 Status control

The working state of objects can be displayed with the "Status display" command. For information on status display in the Navigator see: "COMOS Administration" manual, keyword "Status management".

15.2 Browsing in evaluating reports

The "Browse" button can be made available for evaluating reports. You can find more information on this topic in the "Platform Administration" manual, keyword "EnablePaging".

15.3 Order lists

Order lists

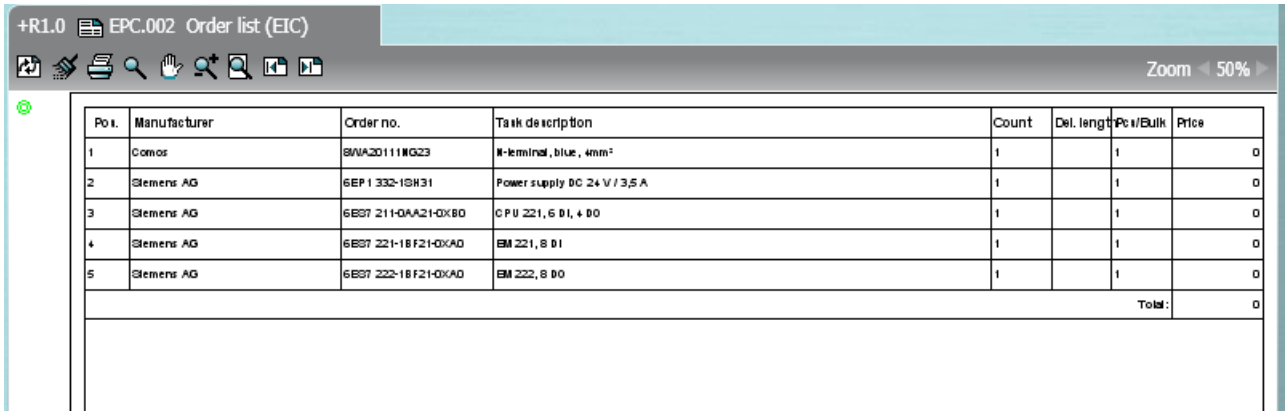
All the same article numbers are grouped together in an order list.

The order list behaves like the material list, but it only considers the manufacturer devices, i.e. the "Request" option must be disabled.

Columns:

- Position
- Number
- Order number
- Order text
- Manufacturer

- Price
- Total



Pos.	Manufacturer	Order no.	Task description	Count	Del. length	Pcs/Bulk	Price
1	Comos	8WA2011 1NG23	N-terminal, blue, 4mm²	1		1	0
2	Siemens AG	6EP1 332-1SH31	Power supply DC 24 V / 3,5 A	1		1	0
3	Siemens AG	6ES7 211-0AA21-0XB0	CPU 221, 6 DI, 4 DO	1		1	0
4	Siemens AG	6ES7 221-1BF21-0XA0	EM 221, 8 DI	1		1	0
5	Siemens AG	6ES7 222-1BF21-0XA0	EM 222, 8 DO	1		1	0
						Total:	0

15.4 Parts lists

Requirement

The bills of materials only contain objects which meet the following criteria:

- The objects have a direct relationship to their owners. References are not evaluated.
- Only objects which are located within the unit structure or location structure are evaluated on the "Units" and "Locations" tabs in the Navigator.
- The engineering object has a base object.
- The engineering object is not of the "Potential" or "Cable" type.
- The engineering object features the "Attributes > Installation/3D data" tab in the properties.
- The engineering object features the "Attributes > Order data" tab in the properties.
- The "Order relevant" option is activated on the "Attributes > Order data" tab in the properties of the engineering object.
- The "Article number" field is completed on the "Attributes > Order data" tab in the properties of the engineering object.

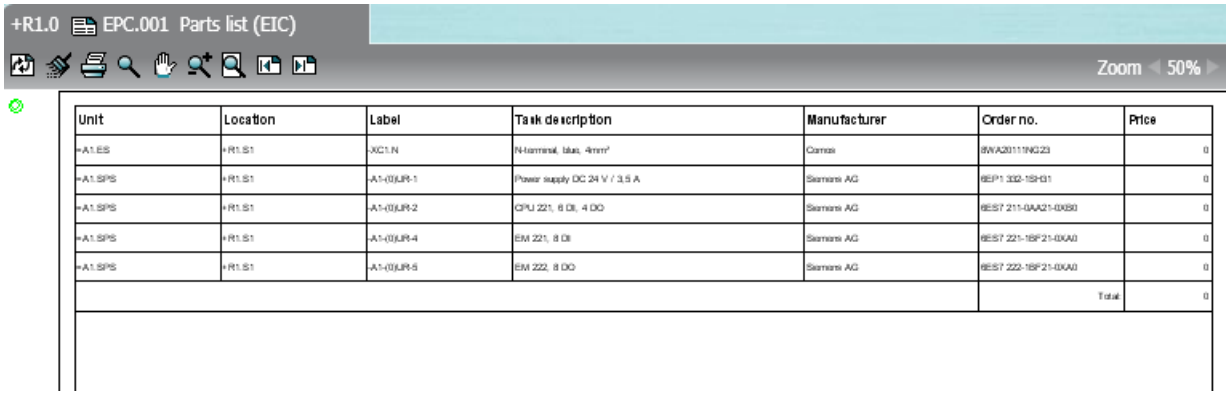
Parts lists

A list of objects specifies all articles individually. Terminals and plugs are sorted by type and summarized below a strip. Accessories are sorted below the devices, identified by "+" in the "Pos" column.

Columns:

- Position
- Unit
- Location

- Name
- Description
- Manufacturer
- Order number
- Device description
- Price



Unit	Location	Label	Task description	Manufacturer	Order no.	Price
+A1.ES	+R1.S1	-XC1.N	N-terminal, blue, 4mm²	Comos	8WA2011NG23	0
+A1.SPS	+R1.S1	-A1-(0)UR-1	Power supply DC 24 V / 3.5 A	Siemens AG	6EP1 332-1SH31	0
+A1.SPS	+R1.S1	-A1-(0)UR-2	CPU 221, 6 DI, 4 DO	Siemens AG	6ES7 211-0AA21-0XB0	0
+A1.SPS	+R1.S1	-A1-(0)UR-4	EM 221, 8 DI	Siemens AG	6ES7 221-1BF21-0XA0	0
+A1.SPS	+R1.S1	-A1-(0)UR-5	EM 222, 8 DO	Siemens AG	6ES7 222-1BF21-0XA0	0
					Total	0

15.5 Terminal diagram

Output formats for terminal diagrams

Terminal diagrams can be output in two formats:

- Line-by-line output
  All terminals are listed, line by line.
- Matrix output
  All cables are listed in a matrix.

15.6 Cable lists

Output formats for cable lists

Cable lists can be output in two formats:

- Cable assignment list
  Lists the contact points of the connected devices for every cable and wire.
- Cable list of objects
  Lists the total length of the cables to be ordered, grouped according to cable type and cross section.

Using templates and grayboxes 16

16.1 Note on grayboxes

You can find additional information on EI&C specific information about grayboxes in the "EI&C Administration" manual, keyword "Preparing grayboxes".

Due to the similarity of functions, you can also find more information on this topic in the "P&ID Operation" manual, keyword "P&ID functions for graybox".

16.2 Template management with the "Template administration" and "Update assemblies" plugins

16.2.1 Overview of the template management

Definition: Templates

If a group of logically or physically related objects can be reused, this group can be made available as a "Template". A template can be made available as an object group or as a report (document in document).

- Basic knowledge of templates: See "Platform Operation", keyword "Basic knowledge of templates and usages".
- Using templates: See "Platform Operation", keyword "Placing objects by inserting a template".
- Administration of templates: See "Platform Administration", keyword "Administration of templates".

Purpose of the template administration of templates

COMOS offers you the possibility to manage templates and to transfer changes to the templates into the existing planning structures. This means that you have the possibility to use new template versions or also corrections for already modified applications without having to delete instance-specific adaptations, such as attribute values or subsequently configured supplements, in the usages.

Three tools are available for this objective:

- "Template administration"
- "Update assemblies"
- "GroupTemplate" Report option

Note

Note: The uses of the templates must have been copied into the project using engineering tasks in order to use the "Template administration" and "Update assemblies" plugins.

"Template administration" plugin

A corresponding preparation of the templates to be used is necessary for reliable functioning of the update. A dedicated release process via the independent "Template administration" plugin is the basic requirement for using the "Update assemblies" plugin.

"Update assemblies" plugin

The "Update assemblies" plugin offers you an overview of the utilization of the templates including the original template version.

Template changes can be implemented both structurally as well as document-based. Object information and graphical changes on interactive reports are retained in the usages.

- You can visualize deviations from the original template. You can reject deviations or apply them in the updated versions. Any contradictory changes are displayed as collisions and you have the possibility to work through the collisions in detail.
- Use a preconfiguration for the matching of specific object classes and the handling of differences.
- In order to use the updates a version management of the templates is required. To do so, prepare the database accordingly.

Interaction with the "GroupTemplate" report option

When you place a template on a report, you get an object group in the basic state. The report object generated is not grouped if the following command has been entered in the options script of the report template of the template: GroupTemplate = False.

Similar function: "GroupGraybox". Due to the similarity of functions, you can read about using "GroupTemplate" in the "GroupGraybox" chapter. You can find more information on this topic in the "P&ID Operation" manual, keyword "P&ID functions for graybox".

16.2.2 "Template administration" plugin

16.2.2.1 Assign rights to use the "Template administration" plugin

Purpose

By default, you need administrator rights to work with the "Template administration" plugin.

The following instructions describe how to enable users without the "Administrator" meta right to use the "Template management" plugin.

Procedure

1. Select the "Administrator > System > User management" command in the menu bar.
   The "User management" window opens.
2. Select the "Groups" tab.
3. Create a new group called "TemplateEditors".
4. Add the desired COMOS users on the "Members" tab.
   Confirm your entries.
   Note: The "Rights" tab remains blank for the "TemplateEditors" group. COMOS only expects the existence of the group "TemplateEditors" without checking any properties for this group.

You can find more information on this topic in the "Platform Administration" manual, keyword ""Groups" tab: Administration of groups and roles"".

Recommendation: Limitations of default rights

Note

Recommendation on the restriction of standard rights

If you use the "Template administration" plugin, Siemens recommends that you remove all other options for editing templates from users.

Proceed as follows to do this:

- Revoke the rights to edit the associated master data.
- In the engineering project, revoke all object rights except "Read" for the "@Template" branch.

You can find more information on this topic in the "Platform Administration" manual, keyword ""Groups" tab: Administration of groups and roles"".

16.2.2.2 "Template management": Creating new templates

Requirement

- You have the rights to open the "Template administration" plugin. See Assign rights to use the "Template administration" plugin (Page 348).
- The rights groups have been created. You can find more information on this topic in the "Platform Administration" manual
- The template objects and the engineering task "Create assembly" have the attribute "Y00A05563 Template version" on the "Y00T00005 COMOS System" tab.
  The "Template version" attribute is read out and displayed in the version history of the plugin.

Procedure

1. Select the command "Plugins > Automation > Templates > Template administration" in the menu bar.
   Alternatively, you can also open "Template administration" via the context menu of a template node.
   The "Template administration" plugin opens.
   See User interface reference "Template administration" plugin (Page 353).
   If the @Template object already exists, the objects lying below the node are displayed here.
2. From the list select whether you wish to use the template on the "Units" tab or the "Locations" tab in the Navigator.
3. To change the base object for the template object or the template folder click the "Options" button in the menu bar of the plugin.
   The "Options" window opens.
4. Carry out the desired changes.
5. Confirm your entries.
   The "Options" window is closed.
6. Select one of the following options:
   - To create a new folder for template objects click the "Create template folder" button in the menu bar of the plugin.
   - To create a new template object click the "Create template" button in the menu bar of the plugin.
7. Enter a name and a description.
8. Confirm your entries.

Result

A new template has been created in the selected structure.

16.2.2.3 "Template management": Editing templates

Requirement

- You have the rights to open the "Template administration" plugin.
  See Assign rights to use the "Template administration" plugin (Page 348).
- The template objects and the engineering task "Create assembly" have the attribute "Y00A05563 Template version" on the "Y00T00005 COMOS System" tab.

Note

Checked-out templates are blocked

As long as a template is checked out, it is locked for editing in this working layer for other users in the "TemplateEditors" group. Users with administrator rights can still edit this template. To remove the lock for users of the "TemplateEditors" group, enable the Edit Mode and check in or undo the changes.

Procedure with option 1:

1. Select the command "Plugins > Automation > Templates > Template administration" in the menu bar.
2. From the list select whether you wish to use the @Template node on the "Units" tab or the "Locations" tab in the Navigator.
3. Select the template object that you want to edit.
4. Click the "Edit template" button in the menu bar of the plugin.
5. Carry out the desired changes.
6. Confirm your entries.

Procedure with option 2:

1. In the Navigator, open the context menu of a template object in the template area.
2. Select "Templates > Template administration".
   The "Template administration" plugin is opened and the "Tab" and "Current template object" fields are pre-filled.
3. Select the template object that you want to edit.
4. Click the "Edit template" button in the menu bar of the plugin.
5. Carry out the desired changes.
6. Confirm your entries.

Result

- The version history of the object shows when and by whom the template object was changed.
- The symbol of the template object changes and indicates that several versions are available.

16.2.2.4 "Template management": Saving and enabling changes

Requirement

- You have created a new template or edited an existing template.
  - See "Template management": Creating new templates (Page 349).
  - See "Template management": Editing templates (Page 350).
- Editing mode is activated.

Saving without check in of changes

If you have changed template objects and do not want to check in these changes yet, click the "Save template changes" button in the menu bar of the plugin.

Note

Templates are locked

When you edit a template or do not check in changes to the template, this template is locked in this working layer for other users. You can only edit this template further with Administrator rights.

Undoing changes

If a version of the template object already exists and you want to discard the current changes, click the "Reset all template changes" button in the plugin menu bar.

Check in changes

1. Click the "Check in template changes" button in the plugin menu bar.
   A window opens.
2. Enter a comment.
3. Confirm your entries.
   If there are instances that use an older version of the copy template, update the instances before checking in.

Result

- The changed template has been checked in.
- The version history has been updated.
- In the background, the JSON files were created or updated for all documents that belong to the changed template. These JSON files are required to display a preview.

16.2.2.5 User interface reference "Template administration" plugin

Menu bar

Icon	Tooltip	Description
	"Create template"	Creates a new template object below the selected node.
	"Create template folder"	Creates a new template folder below the selected node.
	"Edit template"	Activates edit mode for the selected template.
	"Check in template change"	Saves the changes to the template and releases them.
	"Save template change"	Saves the changes to the template.
	"Reset all template changes"	Undoes the changes since the last check-in.
	"Delete"	Deletes the selected object and the underlying nodes.
	"Options"	Opens the "Options" window. There you can define the base objects for the template and the template folder.

"Templates" control group

Control element	Description
"Tab" list	Select the tab into navigator to which the templates are saved from this list.
"Current template object" field	Shows the currently selected template object.
Field	Shows the name of the selected template object.
 "Change" button	Activates the edit mode for the name.
"Template object" field	Displays the description of the selected template object.
 "Change" button	Activates the edit mode for the description.
Template node with underlying structure	• Double-click the objects to open their properties. • Double-click the documents to open them.

"Version history" control group

- Table shows a history of all versions of a template that have been created.
  It contains information on the version number, date of creation, author, uses and current status of a version.
- "Close" button
  Closes the "Template administration" plugin.

See also

"Template management": Creating new templates (Page 349)

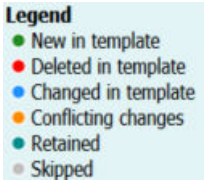
16.2.3 "Update assemblies" plugin

16.2.3.1 User interface link for "Update assemblies" plugin

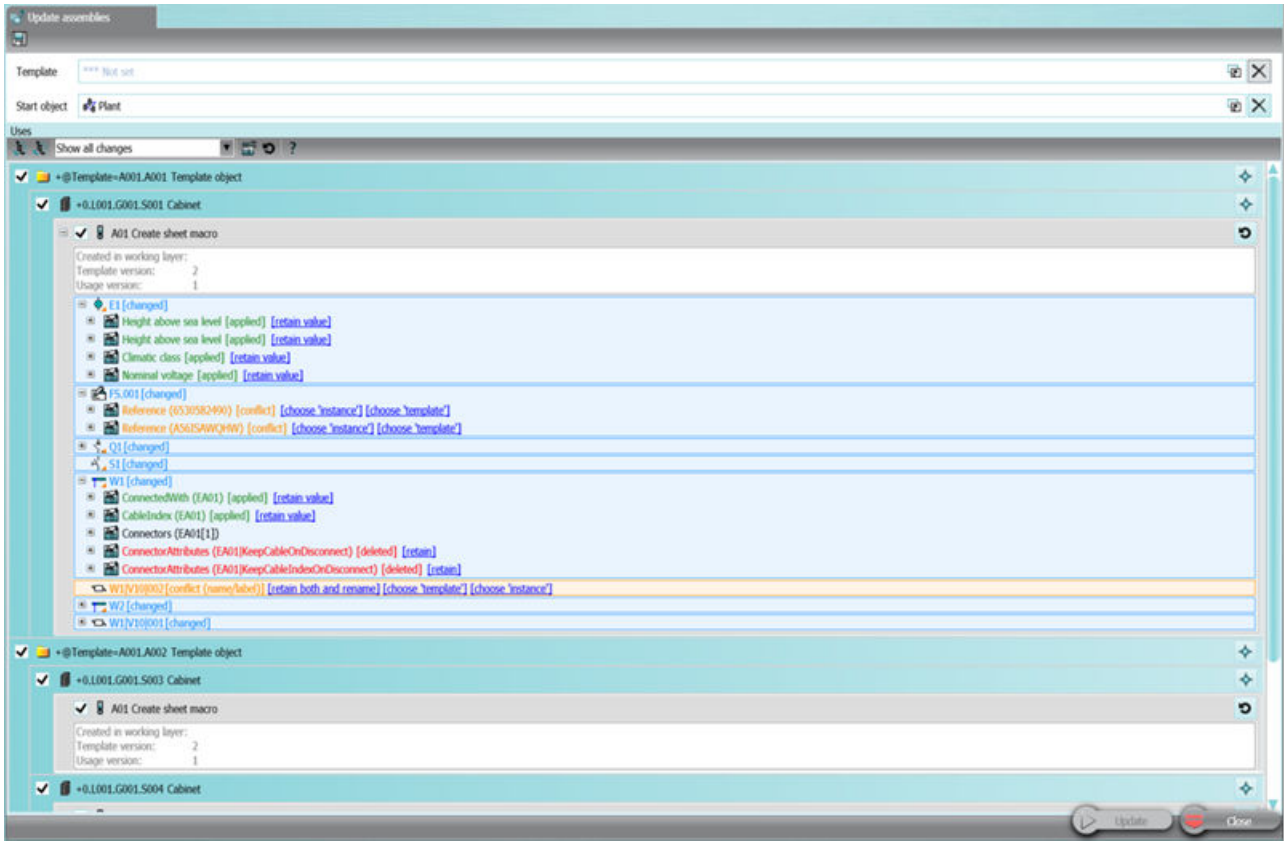
	Saves the current selection
"Template" field	All instances of a template are displayed in the "Uses" area.
"Start object" field	All instances below the one or more start objects are grouped by template in the "Uses" area if no template has been set in the "Template" field.

"Uses" area

Tools

Icon	Tooltip	Description
	Expand all	Expands all layers in the nested tree structure
	Collapse all	Closes all layers in the nested tree structure
"Changes" list	• Show all changes • Show all changes that require action	• Displays all changes in an instance, grouped by template. • Filters the changes in the structure according to entries for which a decision must be made.
	Configuring default selection for differences	Opens the object and property rules in a new window. See Configuring default selection for differences (Page 355)
	Restore default selection for all differences	Restores all differences for which a decision must be made according to the default selection. The restore can be performed for all instances or just one.
	Help	Opens a legend for the color-coding of changes: 
	Navigating	Navigates to the corresponding folder or object in the Navigator.

Overview



16.2.3.2 Configuring default selection for differences

Requirement

- A template that is used in your engineering project has been changed.
- The "Update Assemblies" plugin was opened.
- Instances in which the template is used are displayed.

Procedure

1. Click on the "Configure default selections for differences" icon in the menu bar. A new window opens.
2. Define the object and property rules, how conflicts between the changes in a template and the instance are going to be handled. By default, both changes are applied if there is no conflict; if there is a conflict, the changes in the template have priority.
3. Confirm your entries.

Result

The standard procedure for handling differences between template and instance is saved and not every conflict displayed in the list needs to be resolved manually.

These rules are stored in a standard table. See Standard table "Y10M00N00031 Assembly updater property assignment" (Page 356)

16.2.3.3 Standard table "Y10M00N00031 Assembly updater property assignment"

The "Update assemblies" plugin first checks whether a classification has been stored for the object in the standard table. If the plugin does not find a classification, it uses the default definition.

Node

"Standard tables > Y10 > M00 > A20 > Y10M00N00031 Assembly updater property assignment"

User-defined attributes for the matching

- Name (abbreviation: @N)
- Label (abbreviation: @L)
- Description (abbreviation: @D)
- CDevice (abbreviation: @C)
- MappingTableXML (abbreviation: @M)

The abbreviations refer to the VSUI Notation.

Structure

Column	Description
"Name"	You enter a name according to the following scheme in this column: A+ 3 numbers incremented in steps of 10. For example: "A010" Default: "A010"
"Description"	Enter the description of the object whose attributes are evaluated in this column. For example: "Action" Default: "Default"
"Value 1"	Enter the hierarchical or functional classification of the base object in this column. User classifications are permitted. For example: "C1: M00.A010.A040" Defaults: "Default"
"Value 2"	In this column enter the attributes of the object that are to be matched For example: "@N;@L;@D;Y00T00001.Y00A03136" Default: "@N;@L;@D;Y00T00093.Y00A02222;Y00T00093.Y00A02224;SIE0006.SIE0060;SIE0006.COM0002;Y00T00025.Y00A00253;BMK.BMK32_FUE;BMK.BMK32_DET"

16.2.3.4 Updating instances

Initial situation

A template that is used in your engineering project has been changed.

See:"Template management": Editing templates (Page 350)

Aim

The instances derived from the template should incorporate the changes. Important data for the configuration from the old objects can be retained or overwritten in the engineering project depending on the requirement.

You can first carry out the resulting changes in a working layer to better check the effects and, if necessary, influence them by changing the default setting for changes down to the single attribute.

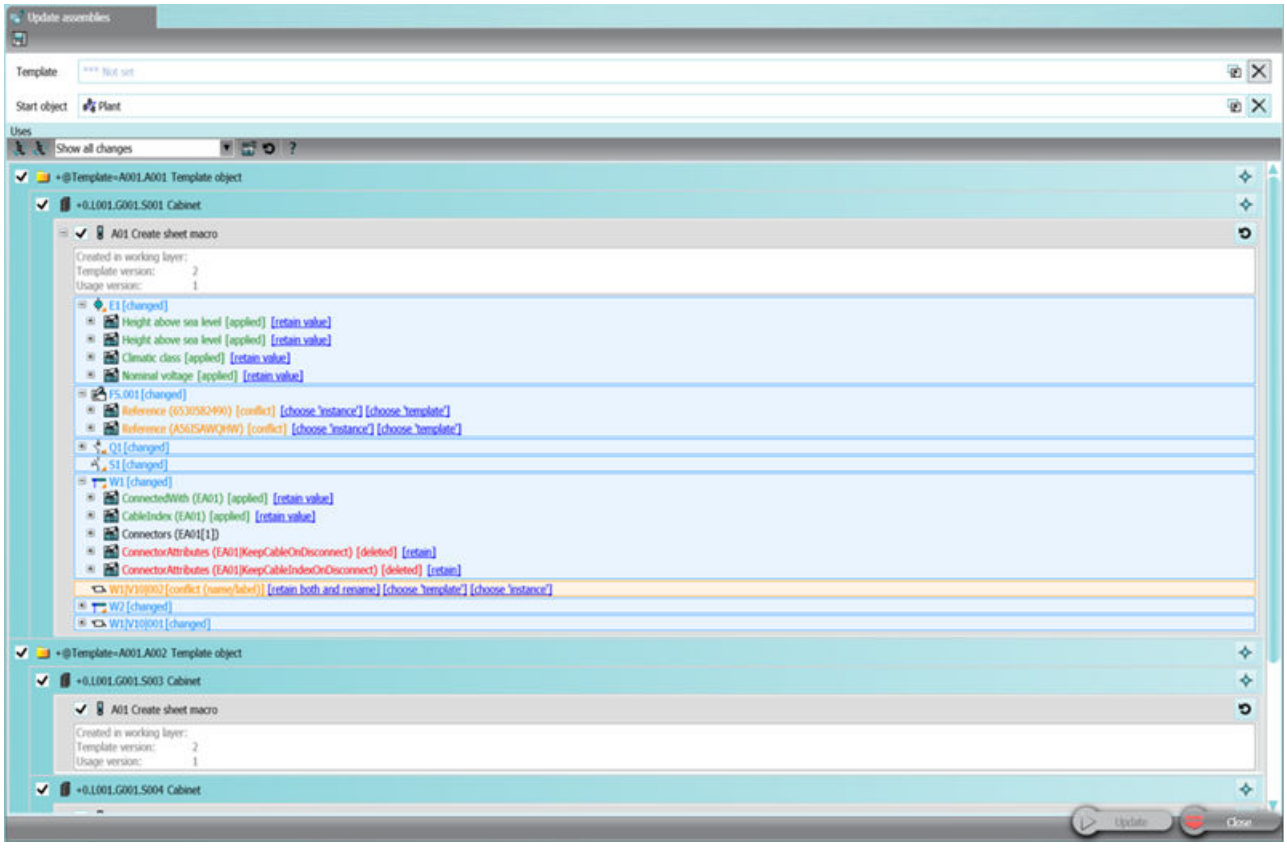
Requirements

- You have read rights in the "@Template" node.
- You have rights for changing the objects in the engineering project.

Procedure

1. Select the "Plugins > Automation > Templates > Update assemblies" command in the menu bar.
   Alternatively, you can also open the "Update assemblies" plug-in via the context menu in the Navigator:
   "Templates > Update assemblies"
2. In the "@Template" node, navigate to the template object - the object under which the template is located.
3. Set the desired start object under which the search for the usages is to be carried out.
   You can also select multiple objects using multiple selection in the Navigator and drag them to the start object field.
   All instances below the one or more start objects are grouped by template in the "Uses" area if no template has been set in the "Template" field.
   In the displayed tree structure you will find:
   - The template
   - The engineering task that created the instance with working layer and version number
   - Instance with all changes and paths to the usages

4. Optional: Drag the template object into the "Template" field.
   The current version of the template and a list of all usages of the selected template are displayed in the "Template version" row.
   Changes that are relevant for the updating of the instances are evaluated and displayed automatically.
   The owner of the engineering task who generated the instance is displayed for each instance. Information such as the working layer in which the instance was generated and the version of the instance are also displayed. All differences of the instance compared to the original version (current - 1) of the template are listed below the instance.
   - Attribute with description
   - Attribute values with unit and range between minimum and maximum value.
   - Attributes with links or linked standard tables
   - References
   - Objects report



Note

Note: Changes to objects that are placed on a report but do not have any placement information in the database are not listed.

5. Select one of the following options from the list:
   - To display all changes select the entry "Show all changes".
   - To display only those changes through which conflicts are triggered select the entry "Show changes which require action".
6. Select one of the following options:
   - Select the differences that should be reconciled for each individual instance. Here you have the option of not applying changes to the template in the usage / instance. Conflicts that have arisen must be resolved manually if you have not predefined a default behavior in the options.
   - To undo the selection for an instance click the "Reapply default selection for all differences in this instance" button.
     Existing changes are evaluated again. You can correct the selection.
     To undo the selection for all instances click the "Reapply default selections for all differences" button in the menu bar of the plugin.
   - You can pre-select how conflicts are handled in the "Configure default selections for differences" menu bar. See "Configure default selections for differences" (Page 355)
7. Confirm your entries.

Note

"Update" stays grayed out:

If the "Update" button stays grayed out despite all conflicts being resolved, there may be additional conflicts in subordinate layers.

Use the filter settings to locate these conflicts and resolve them.

16.3 Using subdocuments (placing documents)

Requirement

- An interactive report with the same drawing type is prepared as a template.
- The circuit diagram is opened.

Placing a report on a report (connected sub-documents)

1. Use drag&drop to move the report out of the Navigator into the circuit diagram.
   The content of the template is shown. In addition, there is a control triangle at the top left.

You can find more information on this topic in the "Platform Operation" manual, keyword "Placing a report on a report".

Break apart placed document

1. Select the control triangle.
2. In the context menu, select "Break apart placed document".
   The control triangle disappears. The grouped contents of the placed report are located in the report. The placed report is deleted.

You can find more information on this topic in the "Platform Operation" manual, keyword "Breaking apart a placed report".