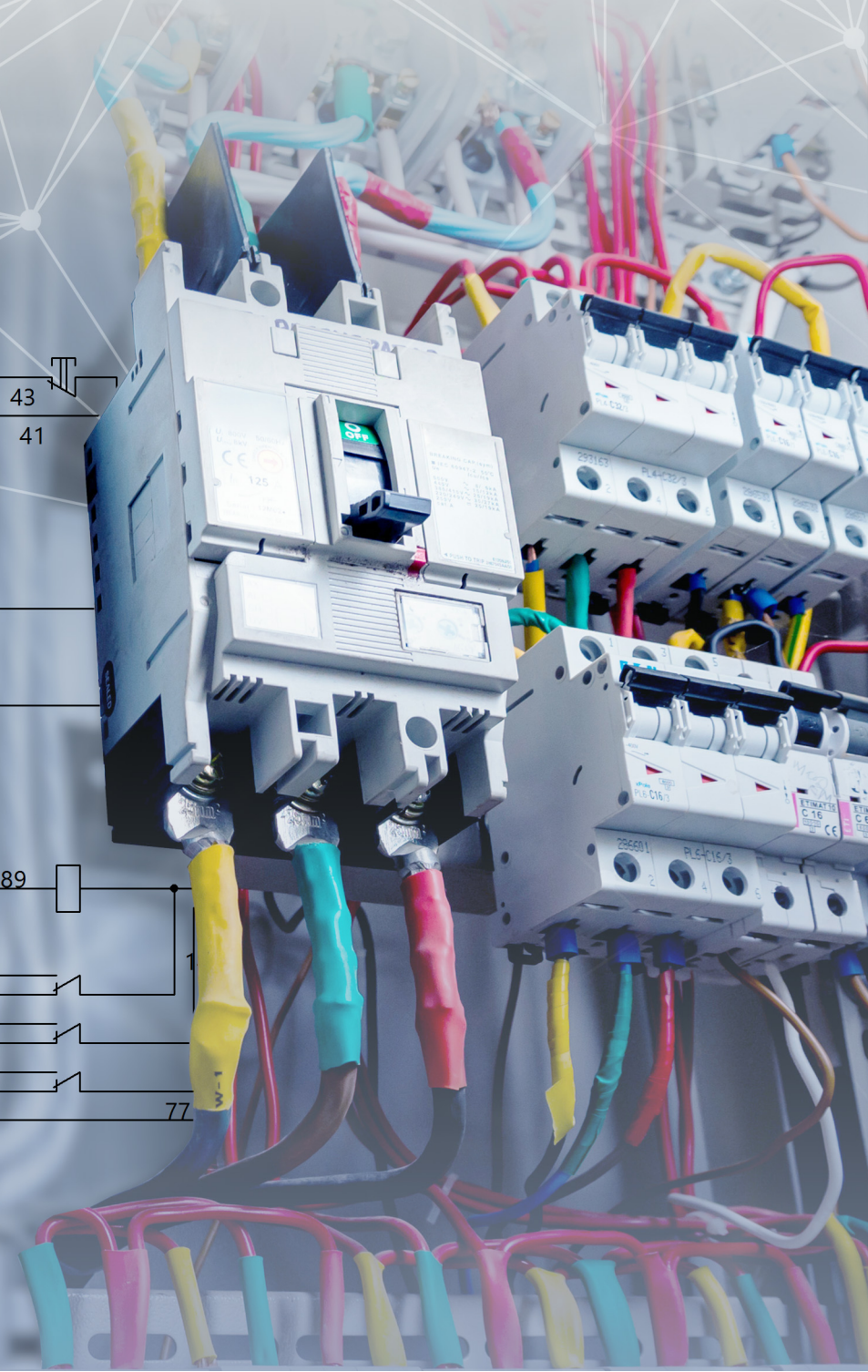
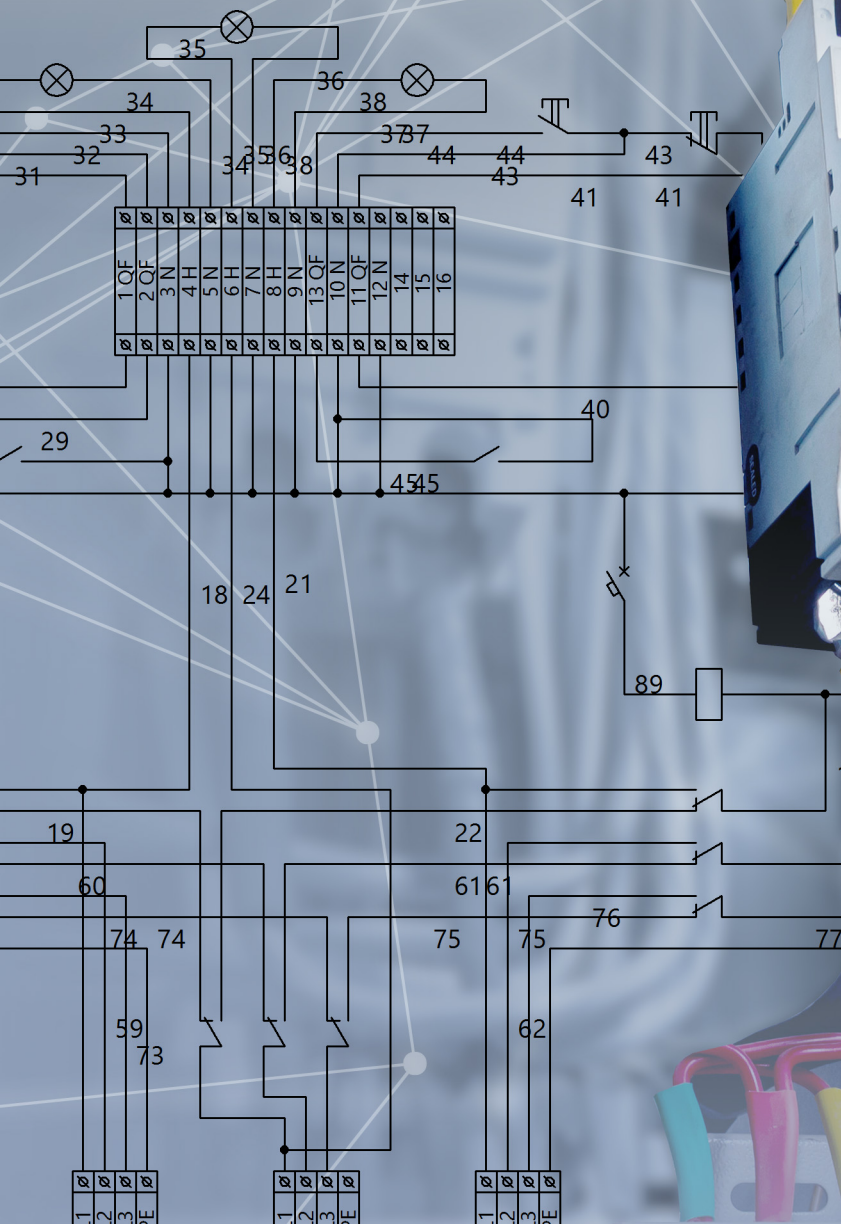


T-FLEX Electrical 17

USER MANUAL

ELECTRICAL SYSTEMS DESIGN



© Copyright 2022 Top Systems

All rights reserved. Any copying of this document in part or as a whole without a prior written permission obtained from "Top Systems" is expressly prohibited.

Top Systems assume no responsibility for any errors or omissions that may appear in this documentation. No claims are accepted for damages caused by using the information contained herein.

The information contained in this document is subject to change without notice.

T-FLEX CAD, T-FLEX Electrical are property of Top Systems Corp.

All other trademarks are the property of their respective companies.

Table of Contents

Introduction	6
Functionality of T-FLEX Electrical	6
Terms and abbreviations	7
Conventions	8
Creating electrical diagrams	10
Getting started	10
Components in diagram	11
Insert Component command	11
Component placing	16
Tag names	18
Creating compound components in context of diagram	22
Device	22
Connector	22
Terminal Block	24
Contactor	25
Chip	27
Linear array	28
Adding links with elements	29
"Unload Component" command	29
Plug and socket connection	30
Splitted representation of compound component	34
Moving and copying components	36
Connection points of component	37
Selection of similar components. "Replace Component" command	40
Component on different diagram types	41
Connection lines	42
"Connection Line" command	42
"Group Connection Line" command	46
"Bus Bar" command	48
"Break Connection Line" command	50
Designations on connection lines	51
Text	51
Screening and coaxiality	53
Twisted pair	55
Cable components in diagram	55
Wire	55
Cable	57
Harness	61
Symbols and other designations	62

Insert Title Block command.....	63
2D reports.....	64
List of elements.....	64
Wires table.....	65
Connection table.....	67
Electrical BOM.....	68
Diagram parameters.....	68
Check Diagram command.....	70
Insert Diagram command.....	71
Update Diagram command.....	72
Component Editor and parameters.....	73
General information about Component Editor.....	73
Base components.....	74
General information. Type tab.....	74
Parameters tab.....	76
Methods tab.....	78
Links tab.....	79
Parameters of components.....	81
Common parameters of components.....	81
Special parameters of base components.....	85
Special parameters of connectors.....	87
Special parameters of cable components.....	87
Special parameters of connection points.....	89
Creating a new parameter.....	90
Creating new prototypes.....	93
Base component prototype.....	93
Connector prototype.....	96
Device, contactor and terminal block prototypes.....	100
Wire and harness prototypes.....	102
Cable (cord) prototype.....	104
Connection point prototype.....	106
Save SE as Prototype command.....	107
Export and import of components.....	108
Update by Prototypes command.....	109
Default Metadata command.....	110
Creating components of electrical diagrams.....	111
Base component.....	111
Graphic.....	111
Creating pins.....	117
Creating component.....	120
Device.....	122

Graphic symbol.....	122
Create device	124
Connector.....	130
Create connector.....	130
Graphic symbol.....	133
Terminal block.....	139
Contactors.....	143
Chip	145
Symbol.....	146
Add components to configuration.....	147
Libraries of electrical diagram elements.....	149
Add components to library.....	150
Designing of electrical components in 3D.....	151
General approach for creation of 3D model by diagram.....	151
Prepare component 3D model	151
Component in 3D	153
Cable components.....	154
Cable components by diagram	154
Cable components creation.....	156
Wire.....	156
Cable.....	157
Harness.....	158
Common automenu options.....	160
Cable components manipulators.....	161
Harness layout.....	163
Unload harness to detail.....	164
Reports in 3D assembly.....	166

Introduction

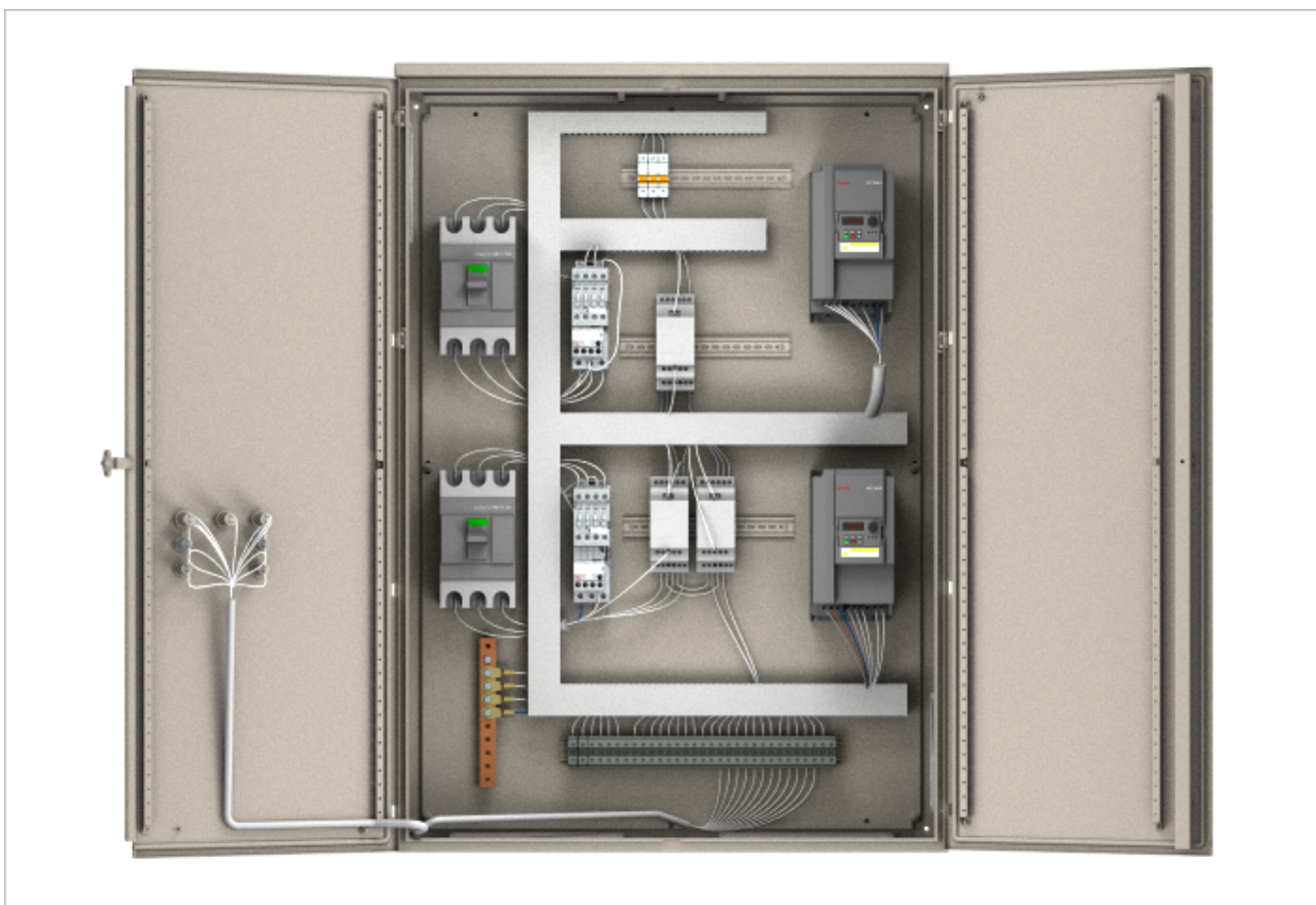
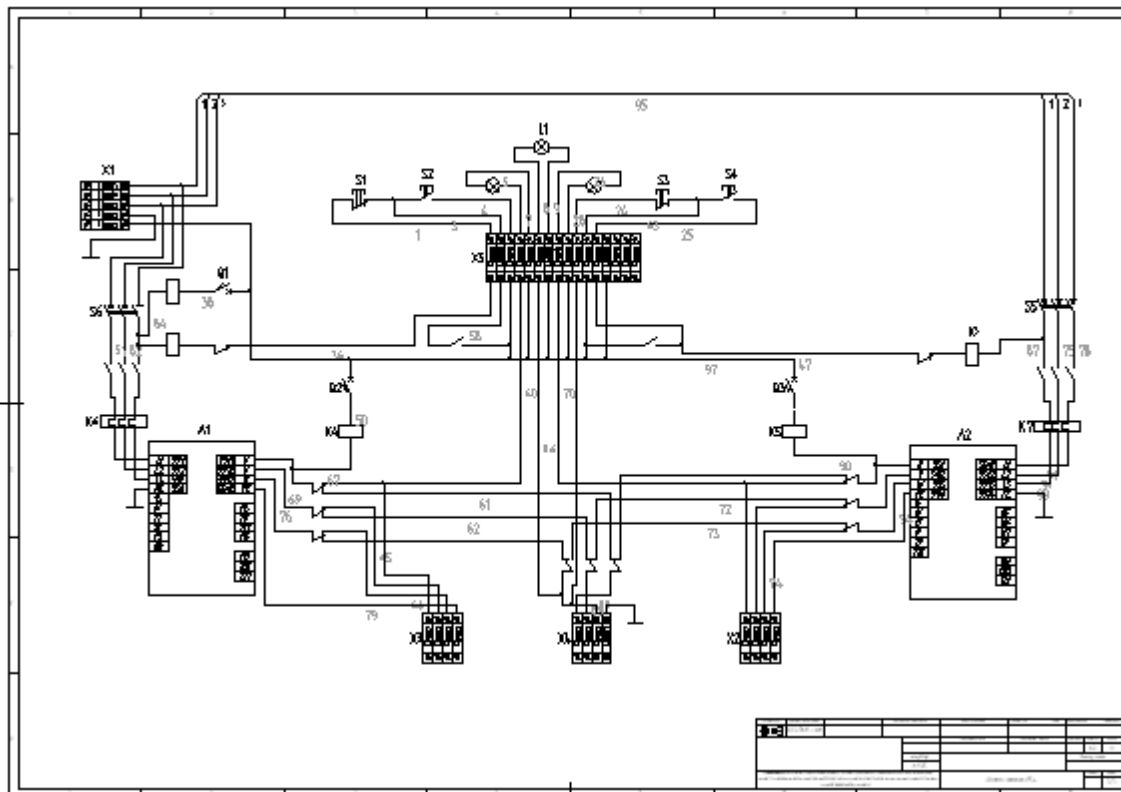
Functionality of T-FLEX Electrical

T-FLEX Electrical is a software product of T-FLEX PLM family used for designing of electrical components of various complexity. T-FLEX Electrical module is built in T-FLEX CAD interface. Integration with PLM system allows you to work in a common information space, without detaching from other software components of the family and changing the file format. Thus, the use of T-FLEX Electrical provides an opportunity for collaborative work of users within the same platform and a common information base in the tasks of designing complex electrical components.

A short list of capabilities and benefits of T-FLEX Electrical module:

- creation of electrical diagrams of different types;
- creation of 3D assemblies of electrical equipment associated with 2D diagrams;
- automatic and semi-automatic laying of cable components in 3D models;
- automatic creation of harness layout, preparation of assembly drawing with BOM;
- analysis of cable components geometry;
- automatic data collection into interactive product structure by diagram and by 3D model with capability to view data for design documents;
- automatic generation of documentation: list of elements, wires table, connection table, layout diagram;
- designing of several types of diagrams in one document with the possibility of interconnection;
- multiple-sheet diagrams creation;
- diagrams analysis for errors;
- using external diagrams and updating documents according to the initial diagram changes;
- extensive library of standard elements;
- library containing ready-made 3D models of various formats;
- creation of user's elements (connectors, devices, relays, etc.) and cable components of any type;
- creation of user's common database;
- exchange of libraries and components database between users;
- parallel several user's work on one project.

The following images show an electric diagram with a list of elements and a 3D model with cable components created on diagram basis.



Terms and abbreviations


Electrical component (element) is an element that includes schematic symbol, tag name, parameters, links with other elements.

Connection point (contact) is a special object that is used for connection of elements on the diagram.

Electrical components in T-FLEX Electrical are divided into types: Electrical Item, Device, Connector, Plug and Socket Connection, Relay, Terminal Block, Chip, Symbol.

Compound component is a component that consists of several elements, each of which has its own schematic symbol (for example, relay coil and contacts, terminals in terminal block). Compound components include device, connector, plug and socket connection, relay, terminal block, chip.

Creation of components in context of diagram - creation of a new component directly in the document of the diagram (the opposite case is the creation of components in a separate document with subsequent insertion into the diagram). The concept applies to compound components.

The term prototype in this documentation is used in two meanings. Prototypes are elements created in  **Product Editor** and representing information models of electrical components. In addition, according to the general terminology of T-FLEX CAD, prototypes are called document templates on Start page.

CC - cable components (incl. Wire, Cable, Harness).

CL - connection line.

CP - connection point.

EI - electrical item.

EMC - electromechanical connection.

FB - functional block.

GCL - group connection line.

PS - product structure.




SE - structural element. At T-FLEX Electrical, structural element is an information model of an electrical component. SE stores in itself the parameters of the component and a description of its links with other elements. The terms structural element and prototype describe the same object at different stages of work with the component. For more information on SE, see the chapter "General Information about Component Editor".



SS - schematic symbol.

TN - tag name.

Conventions


The following standard conventions are adopted in this document.

Designation Example in Text	What Means
<Enter>, <L>, <Esc>	The keys on the computer keyboard
[OK], [View]	Graphic button in the dialog window
	Mouse cursor
	Left mouse button click
	Right mouse button click

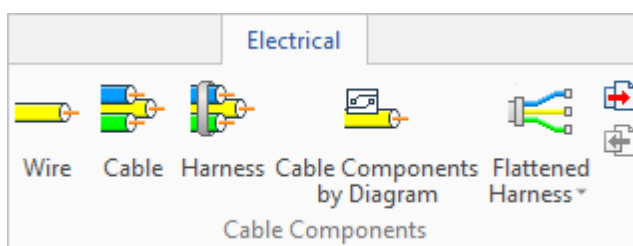
	Left mouse button double-click
	Icons on a toolbar, ribbon or automenu
O: Open Model	The command name. The combination of letters before the colon determines the sequence of keys on the keyboard to call the command from the status line.

You can invoke a command in the following ways: using keyboard, by icon on the toolbar, using textual command menu and by icon on the ribbon.

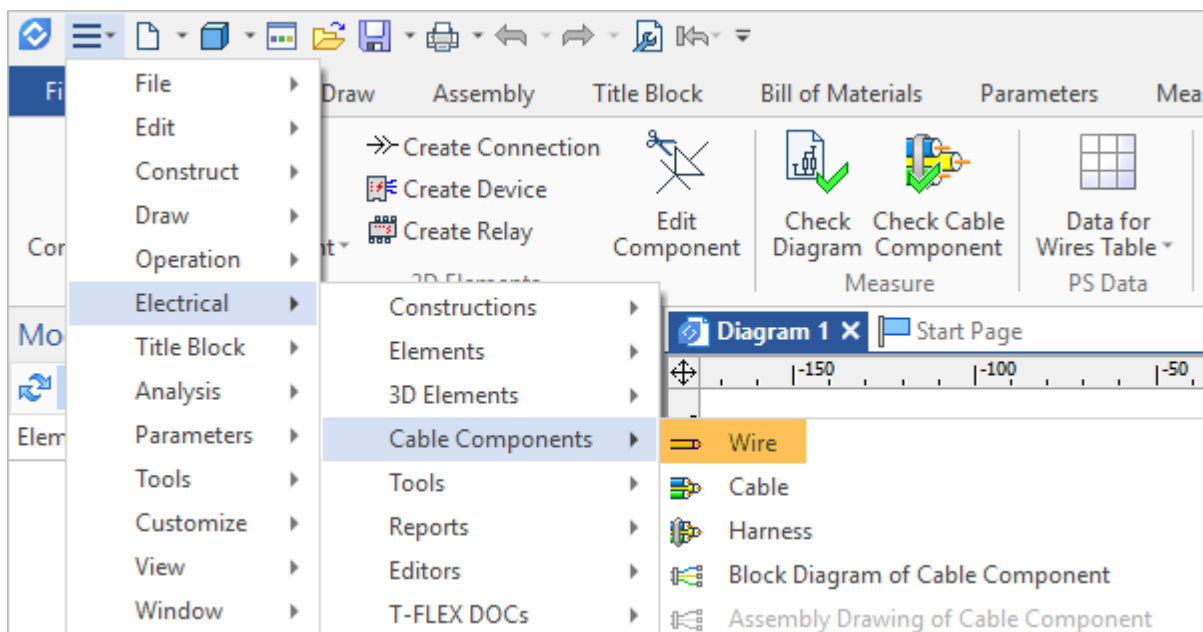
Ways to invoke the command are described in the tables. For example, the command **Wire** will appear in a table as follows:

Icon	Ribbon
	Electrical ? Cable Components ? Wire
Keyboard	Textual Menu
	Electrical > Cable Components > Wire

This means, the command can be invoked on the **Electrical** tab in **Cable Components** group.



The same command can be called via textual menu. To do this, open the groups **Electrical** and **Cable Components** in series.



Creating electrical diagrams

Getting started

You can create the diagram in a new document or add a special page for it in an existing document. To create a diagram in a new document, use the document prototype "Diagram" of Electrical group on Start page.

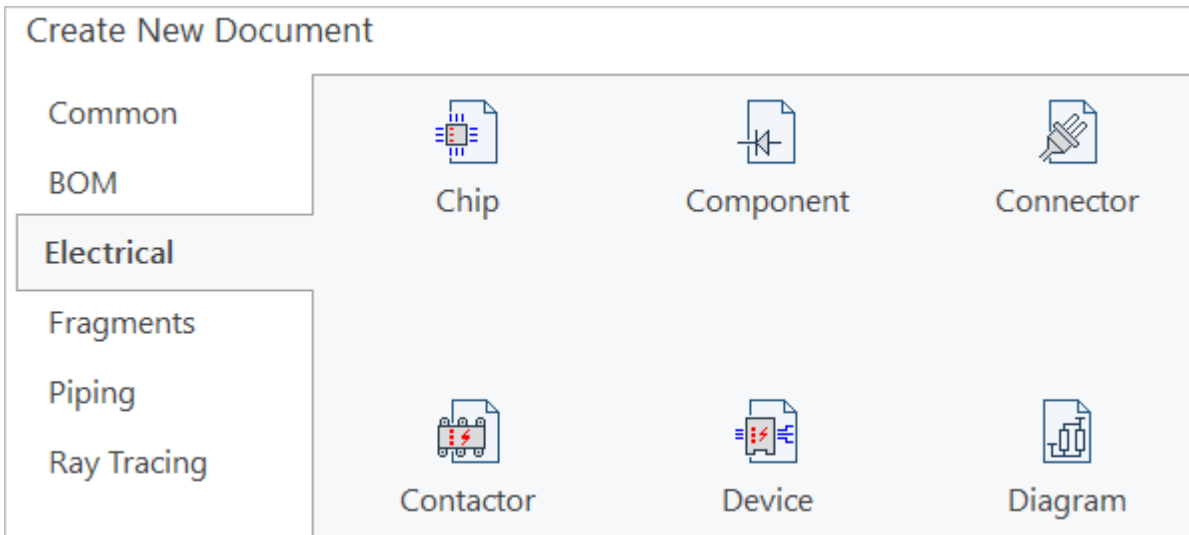
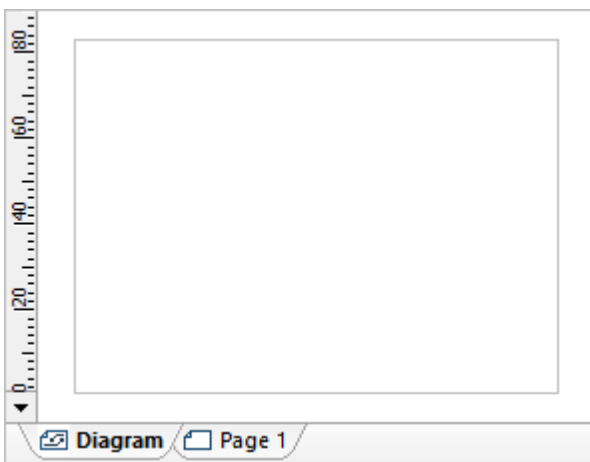



Diagram document has two pages. The first is the Diagram page, the second is the regular page. The regular page is used for preparation of design documentation by the diagram. The creation of the diagram is performing on the Diagram page.



To create a diagram in an existing document or create another Diagram page in it, use the command:

Icon	Ribbon
	Electrical ? Diagram ? New page
Keyboard	Textual menu
	Electrical > Constructions > New page

There is a grid on the Diagram page. The grid ensures the correct connection of the diagram elements with each other (for details, see "Create Schematic Symbol").

You can specify grid parameters, including **Grid point size**, in the commands of  **Grid** group on View panel.


The grid step can be changed using **Step** variable in **V: Variable Editor**.

Variables				
Name	Expression	Value	Com...	
Group:				
Step	2.5	2.5		

Components in diagram

Insert Component command

Use the following command to insert a component into the diagram:

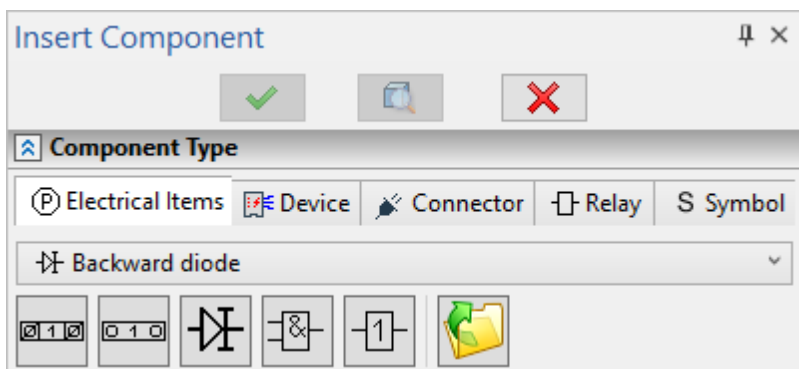
Icon	Ribbon
	Electrical ? Diagram ? Insert Component
Keyboard	Textual menu
	Electrical > Constructions > Insert Component

The command allows to select a component from the list and insert it in the diagram.

You can also insert component in the diagram from "Electrical Diagram Elements" library or Windows Explorer using Drag'n'drop mechanism. The exception is the elements of System folder, which are not intended to be added to the diagram. The program ignores attempts to insert them using Drag'n'drop.

The following tabs are available in **Component Type** section: **Electrical Items**, **Device**, **Connector**, **Relay**, **Symbol**.

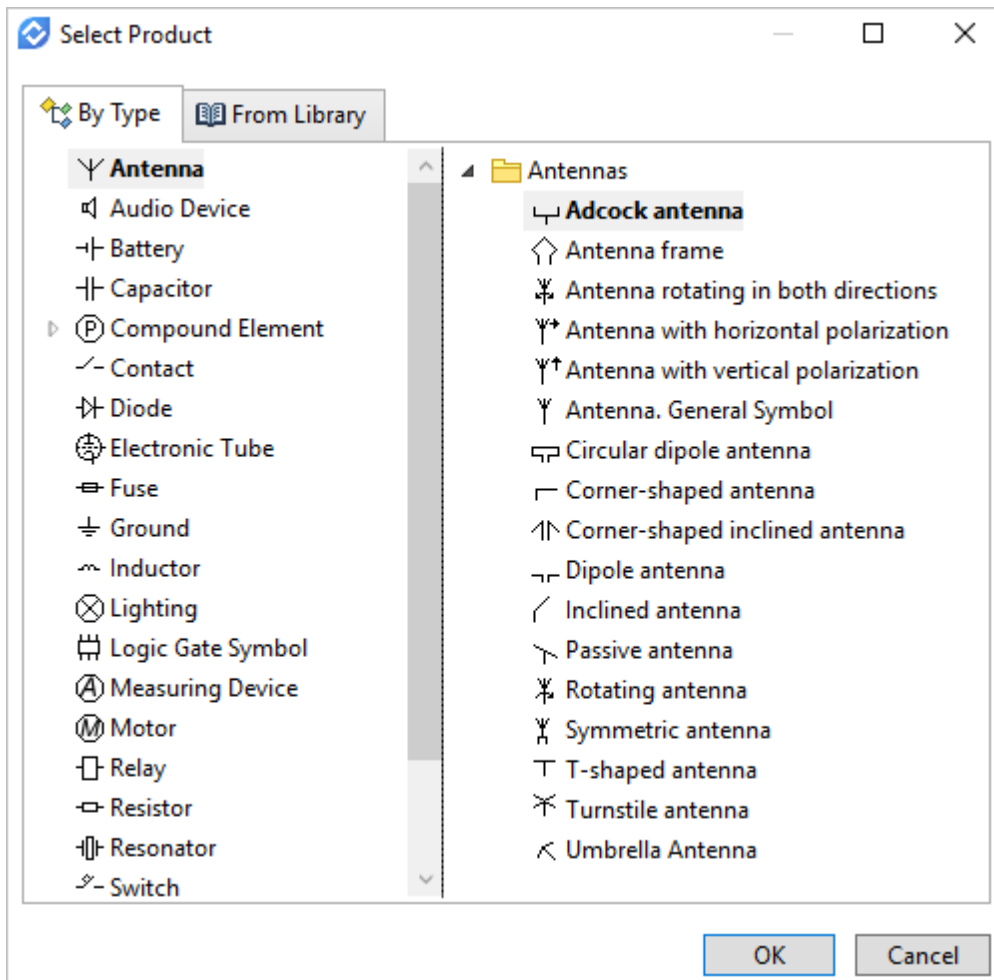
You can select a component from drop-down list on this tabs. The last selected components are displayed below as icons.



Select from dialog icon calls a dialog window with two tabs.

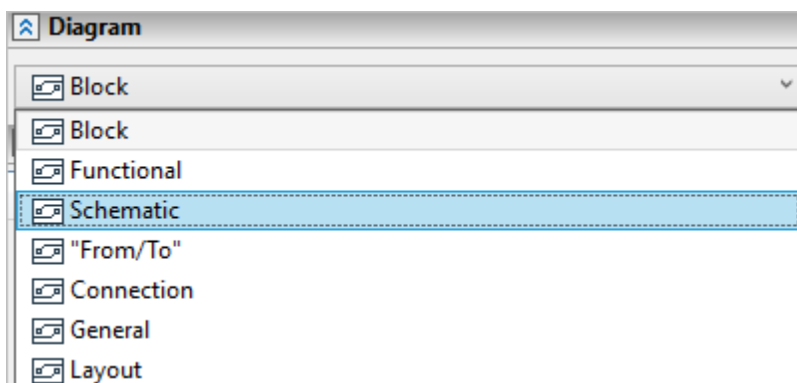
By Type tab displays all standard components from "Electrical Diagram Elements" library.

From Library tab displays all components from libraries (including user ones), connected to the corresponding configuration.

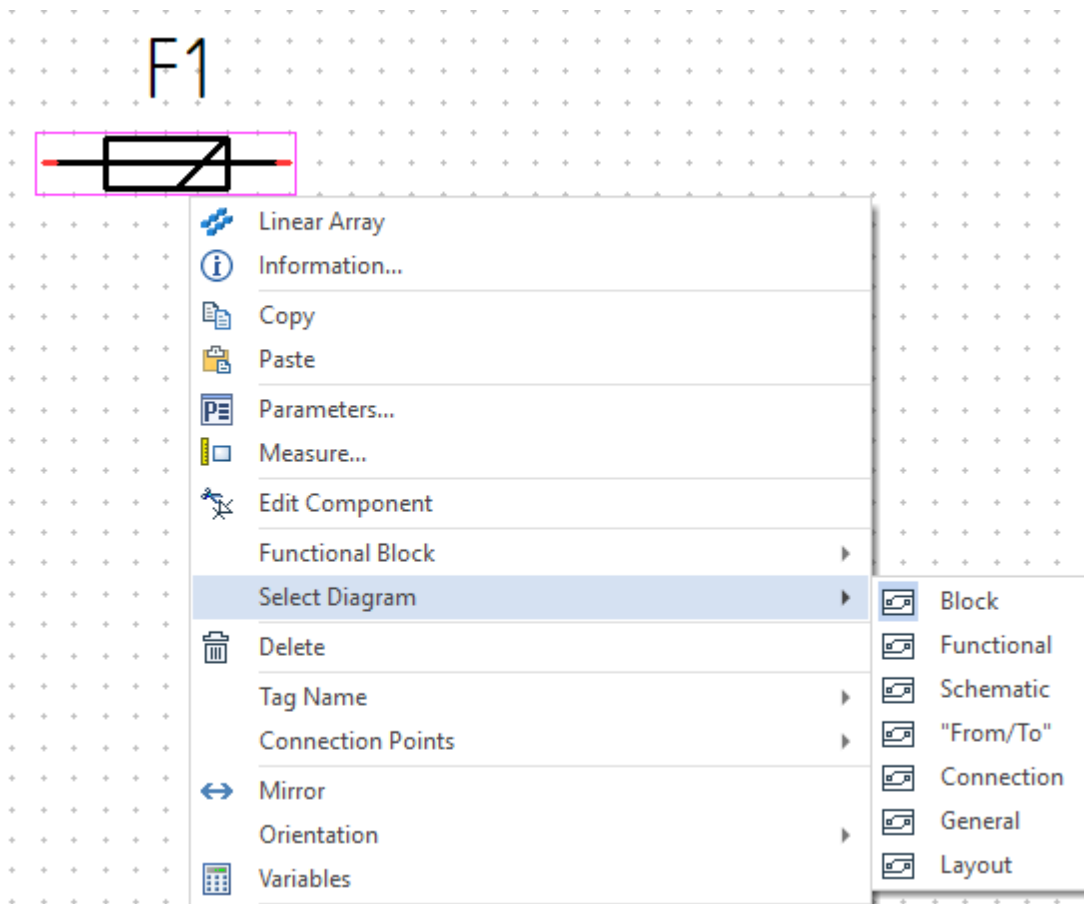


If the required component of diagram is missing in "Electrical Diagram Elements" library, you can create it by yourself. For more information, see "Creating components of electrical diagrams" section.



You can set diagram type of the component using the drop-down list in **Diagram** section.











After inserting the component, its diagram type can be changed via the context menu (**Select Diagram** - ...).




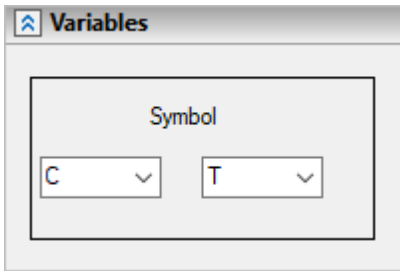
Components grouped by diagram type are presented in the model tree (Structural Elements folder, Electrical diagram group).


Parameters of the inserted component are displayed in **Parameters** section. To edit the value of parameter make the field editable by clicking on icon  (it will change to ). See chapter "Common Parameters of Components" for details on parameters.

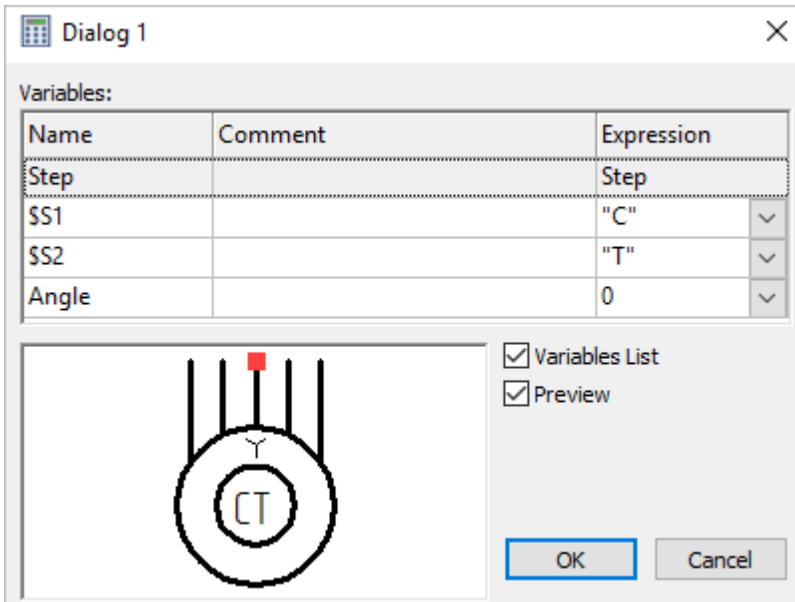
Parameters	
Parameter	Value
Annotation	
Data for PS from 3D	<input type="checkbox"/> 
Description	Slow fuse 
Family Name	F 
Group Name	Fuses 
Include in Reports	<input checked="" type="checkbox"/> 
Manufacturer	
Part No.	

Parameters window can be also called up via the component context menu.

Variables section is minimized by default. Opening it by  will give access to the control dialog, if such a dialog was created in the component document.




The control dialog can be also called up using  **Variables** command in the component context menu.





Variables List check box hides the dialog and displays a list of external variables of the fragment instead. If there is no control dialog in the component file, the check box will be unavailable.

Preview check box opens a window showing how the symbol of the product will change depending on the value of a variable.



If you re-select any component in the diagram, then  **Insert Component** command will start automatically (without **Component Type** section in the dialog).


The automenu of **Insert Component** command is described below.

The set of icons in the automenu depends on which element was selected in **Component Type** section.


Icon	Keyboard	Option	Comment
	<D>, <W>, <A>, <X> and others	Orientation	The option controls the rotation of the component in the diagram. Each rotation angle has a hot key that can be used to quickly perform an action. For more details, see the chapter "Create Schematic Symbol".
	<1>...<8>	Basic Position of Tag Name	The option controls default position of component TN on the diagram. Each position corresponds to a certain key on the keyboard. See the chapter "Tag Names" for details.

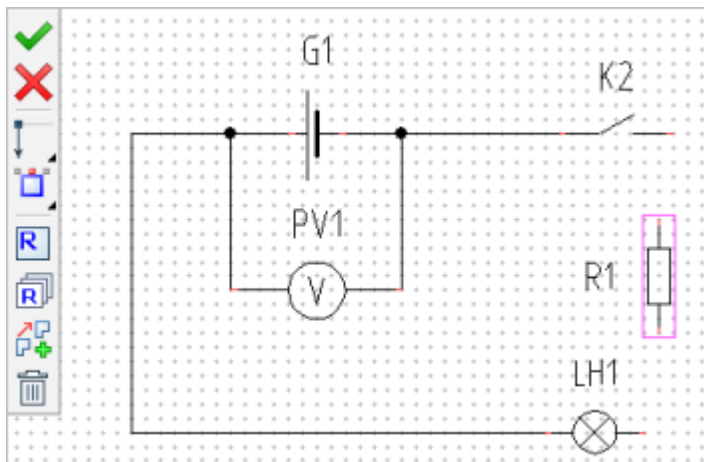
	<F>	Reverse	Symmetrically reflects the component in the diagram. For more details, see the chapter "Create Schematic Symbol".
	<N>	Type	Changes component type. For more details, see the chapter "Create Schematic Symbol".
	<V>	Form	Changes component form. For more details, see the chapter "Create Schematic Symbol".
	<R>	Repeat Insertion of Component	Adds the last inserted component to the diagram.
	<T>	Multiple Iterated Insertion of Component	Multiply adds the last inserted component to the diagram.
	<P>	Copy Component	Copies selected product with all its parameters. It is convenient to use the option instead of reinsertion commands if the parameters of the component have been changed in the diagram or if it was created in context of diagram.
		Delete Component	Deletes selected component from the diagram.
	<K>	Add Link with Element	Creates a link between elements of a compound component. For more details, see the chapter "Relay" in subsection "Creating Compound Components in Context of Diagram".
	<M>	Delete Link with Element	Deletes a link between elements of a compound component. For more details, see the chapter "Relay" in subsection "Creating Compound Components in Context of Diagram".
	<Y>	Splitted Schematic Symbol	Allows to place the elements of a compound component separately. Available for compound components created in separate documents (not in context of diagram).
	<O>	Move Whole Compound Element	Allows you to move a compound component as a single whole. For more details, see the chapter "Relay" in subsection "Creating Compound Components in Context of Diagram".

			Context of Diagram".
	<L>	Add Link to the Component on the Diagram	Creates a link between different representations of the same component. A link can be created between components that belong to different types of diagrams. For more details, see the chapter "Component on Different Diagram Types".
	<G>	Delete Connections	Deletes the connection between the component and the connection lines linked with it. The option is available only for components to which CL are connected.
	<U>	Create Relay	Calls Create Relay command dialog in the context of diagram. For more details, see the chapter "Create Relay Command".
	<U>	Create Device	Calls Create Device command dialog in the context of diagram. For more details, see the chapter "Create Device Command".
	<U>	Create Connector	Calls Create Connector command dialog in the context of diagram. For more details, see the chapter "Create Connector Command".
	<U>	Create Terminal Block	Calls Create Terminal Block command dialog in the context of diagram. For more details, see the chapter "Create Terminal Block Command".
	<U>	Create Chip	Calls Create Chip command dialog in the context of diagram. For more details, see the chapter "Create Chip Command".
	<U>	Edit Component	Calls up Create {component type} command dialog without the ability to edit the component structure.

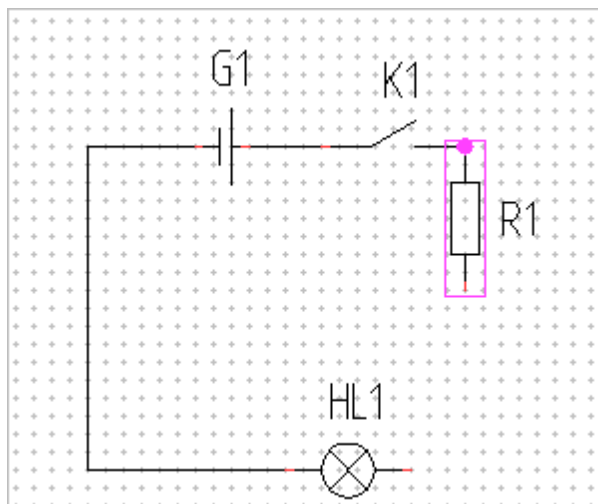
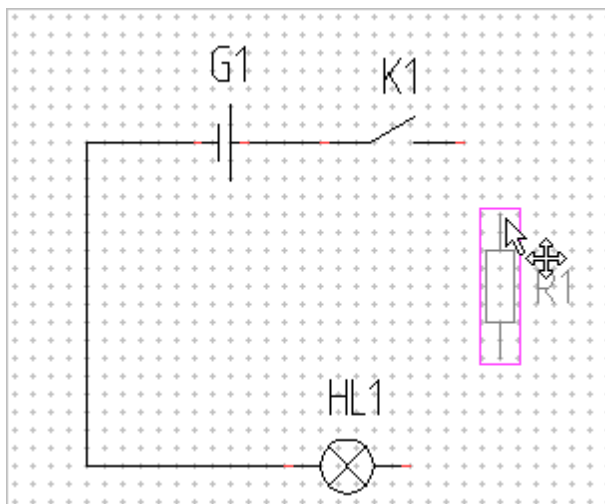
When selecting  a component in the diagram, a similar automenu is available. Some of the options described above are available in the context menu of diagram elements.

Component placing

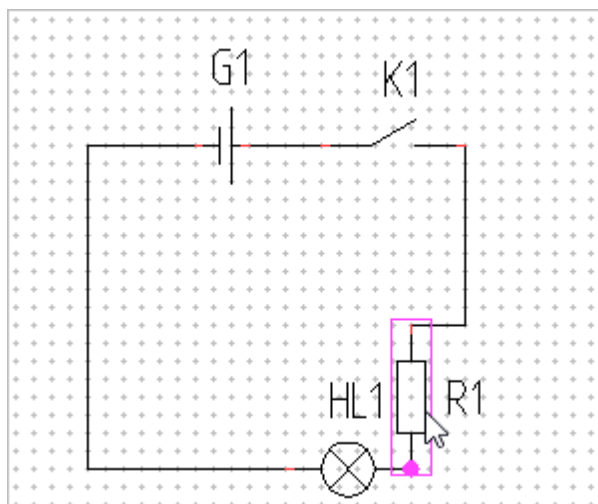
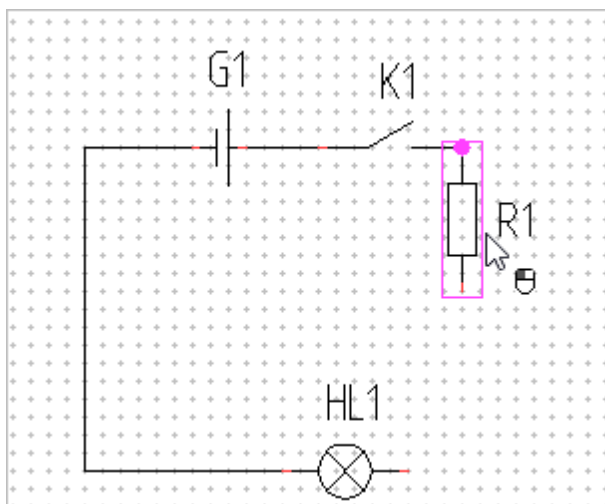
After selecting and setting up the component, it is necessary to place it on the diagram. The component is added to the diagram after clicking on the diagram page, but  **Insert Component** command remains active.



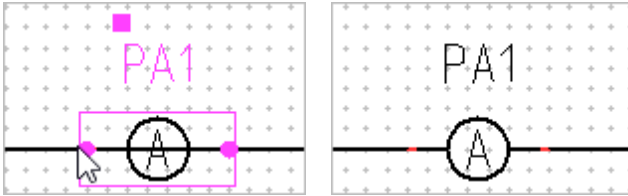
If a component is selected, it can be linked to another component without starting **Connection Line** command. To do this, when placing the component, it is necessary to combine one of its connection points with the CP of another component in the diagram. The connection between the components will be created; this will be indicated by a round pink marker.






In the same way, you can connect to the second connection point.





The component can be placed on an existing connection line. In this case, the CL will be automatically broken and connected to the connection points of the component placed on it.



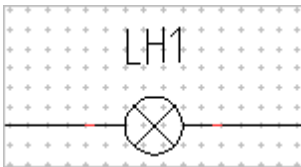
Placing a component on a diagram (snapping to a grid point) automatically saves all changes to a component made earlier in  **Insert Component** command. After placing the component, you can close the command with  or <Esc> - the changes will be saved.

Closing a command with  is required if you want to save changes made to the component after it is snapped to a grid point.


All components inserted into the diagram are added to the model tree in the Structural Elements folder. Together with them, their connection points  and the diagram types  assigned to them are added to this folder.

Tag names

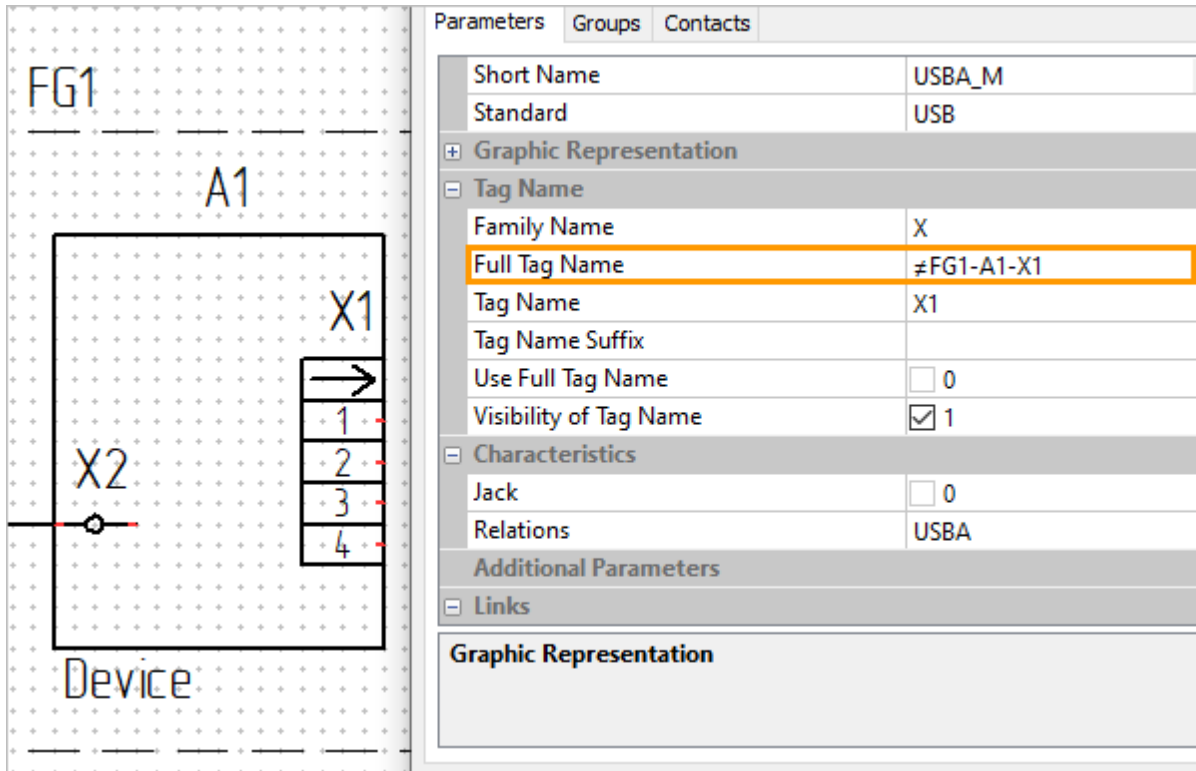
Tag names are intended for unambiguous recording in an abbreviated form of information about components on the diagram and in the reports.



After inserting the component into the diagram, tag name is automatically assigned to it. The default TN text consists of a prefix (for example, R) and an element serial number in the diagram. The TN prefix is set by default for all components. You can change the prefix of the TN and the TN of the component in its parameters.

Each component is created on one of the prototypes in  **Component Editor** and inherits the parameters of this prototype.

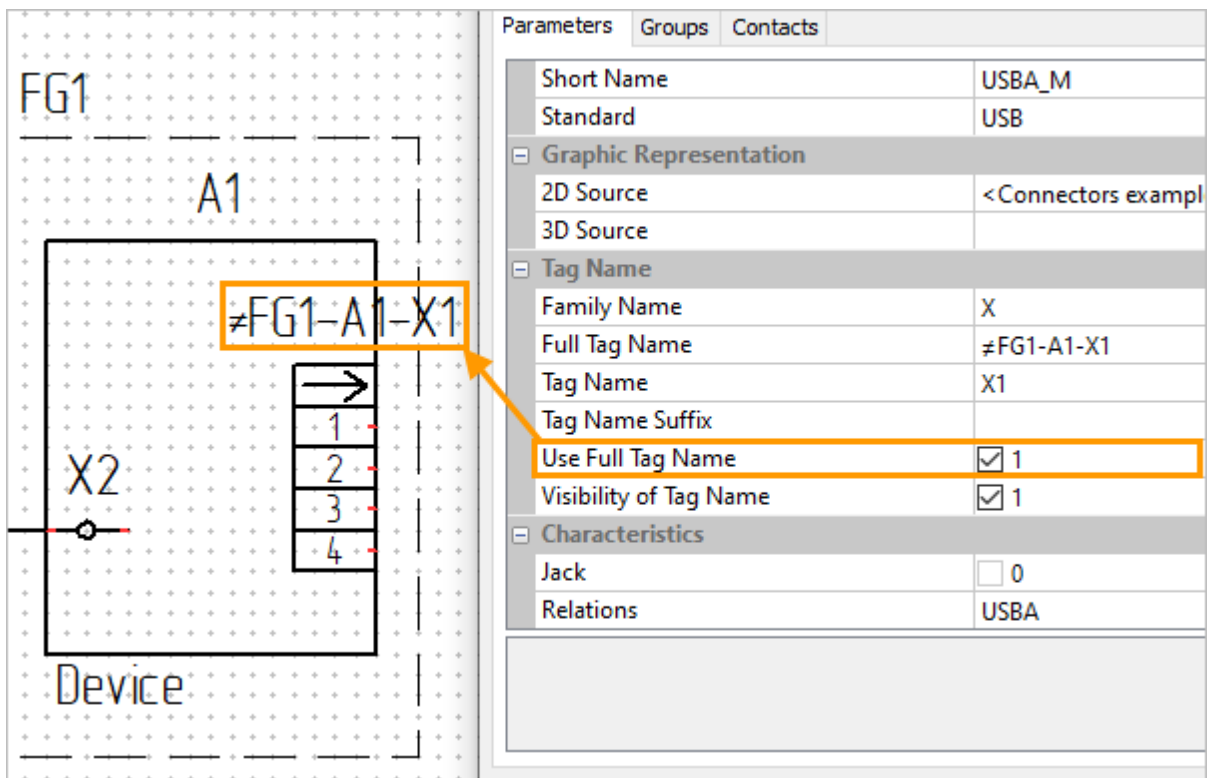
The **Full Tag Name** parameter is available for all components. The full TN consists of tag name of the element itself and tag name of "higher" products, that include it. The X1 connector in the screenshot below is part of A1 device, which is part of FG1 functional group. The entire chain is displayed in **Full Tag Name** parameter.



The diagram shows a component labeled 'Device' within a frame 'A1' of a functional group 'FG1'. The component has a connector 'X1' with four pins (1, 2, 3, 4) and another connector 'X2'. The parameter table on the right is as follows:

Parameters	
Short Name	USBA_M
Standard	USB
+ Graphic Representation	
- Tag Name	
Family Name	X
Full Tag Name	#FG1-A1-X1
Tag Name	X1
Tag Name Suffix	
Use Full Tag Name	<input type="checkbox"/> 0
Visibility of Tag Name	<input checked="" type="checkbox"/> 1
- Characteristics	
Jack	<input type="checkbox"/> 0
Relations	USBA
Additional Parameters	
- Links	
Graphic Representation	

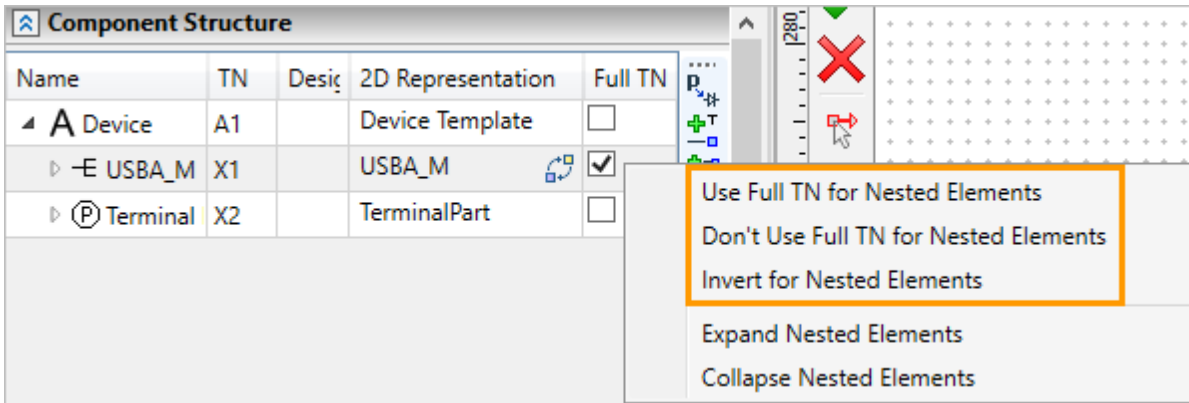
To display full tag name instead of "local" one on the diagram, you must check **Use Full Tag Name**.



The diagram now shows the full tag name '#FG1-A1-X1' next to connector 'X1'. The parameter table on the right is updated as follows:

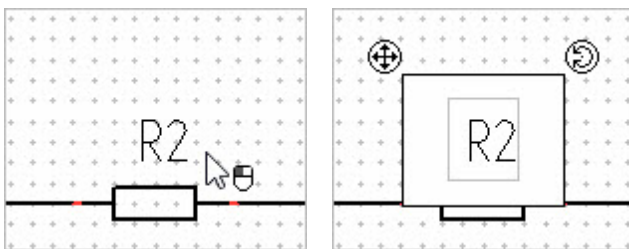
Parameters	
Short Name	USBA_M
Standard	USB
- Graphic Representation	
2D Source	<Connectors exampl
3D Source	
- Tag Name	
Family Name	X
Full Tag Name	#FG1-A1-X1
Tag Name	X1
Tag Name Suffix	
Use Full Tag Name	<input checked="" type="checkbox"/> 1
Visibility of Tag Name	<input checked="" type="checkbox"/> 1
- Characteristics	
Jack	<input type="checkbox"/> 0
Relations	USBA
Additional Parameters	
- Links	
Graphic Representation	


Thus, full tag name simplifies the identification of an element and allows tracing the entire "chain of occurrence" of a component. Quick access to **Use Full Tag Name** checkbox is provided in all component editing dialogs at all nesting levels (**Full TN** column).



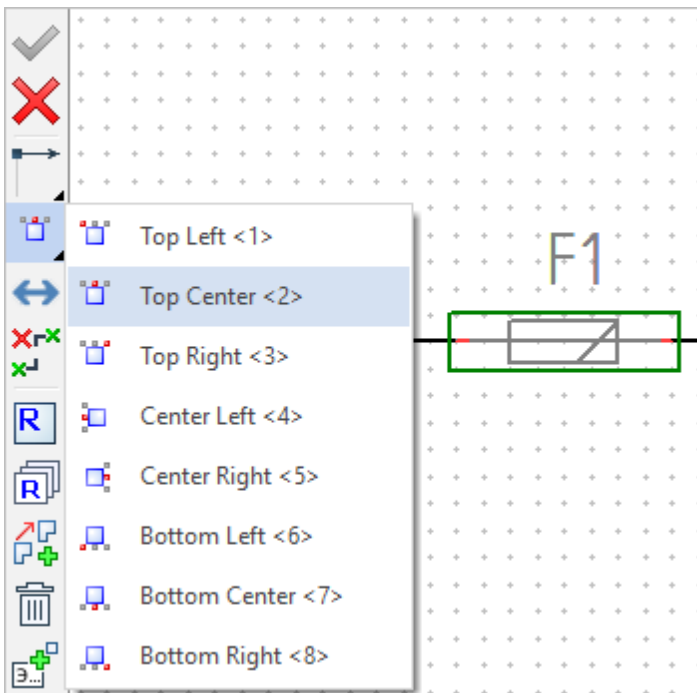
In addition, when you click on the RMB in the drop-down list, commands for group control of full tag names are available.

The tag name text can be changed directly on the diagram. It is edited in the same way as regular objects of the "Text" type.




The page number can be included in the tag names of all components in the diagram, including cable ones. To do this, check **Show page number** parameter in  **Diagram parameters** command. This option may be useful when working with multi-page diagrams.

The TN position is assigned by a special option in the automenu. The position assigned in this way is called basic.




The basic position can be also set via the context menu of the element (the option **Tag Name - Basic position**).

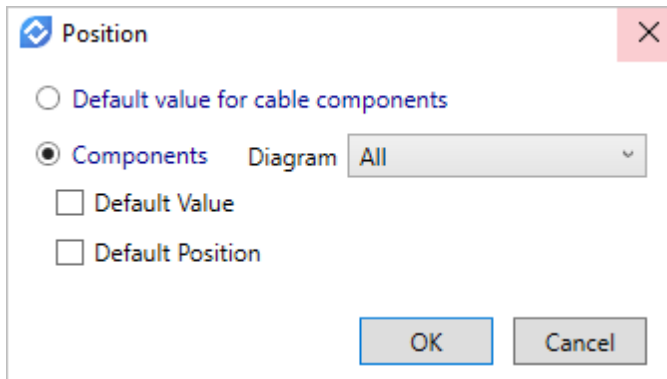
The TN can be placed at will by moving it with .

After adding components to the diagram, you may need to arrange the values and position of the TN.

To do this, use the command:

Icon	Ribbon
	Electrical ? Diagram ? Tag Name
Keyboard	Textual Menu
	Electrical > Constructions > Tag Name

After activation, a dialog box appears:

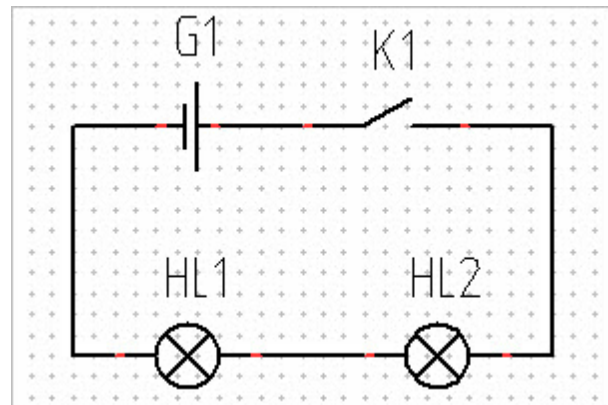
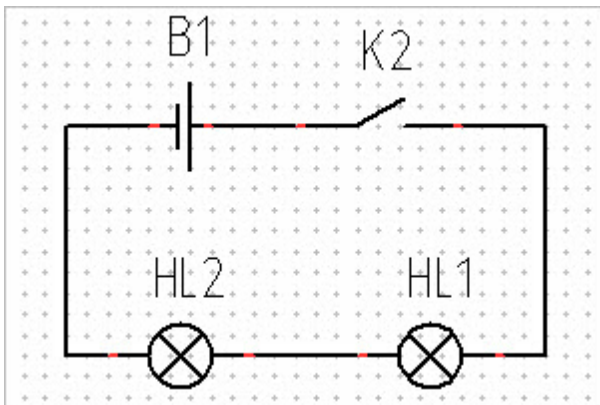


Default value for cable components option resets user values of CC tag names and set tag names for them in order.

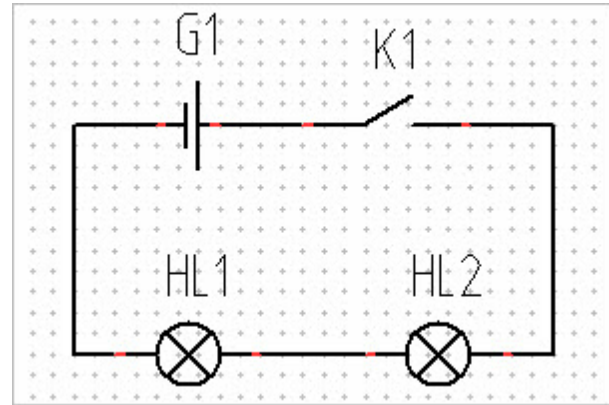
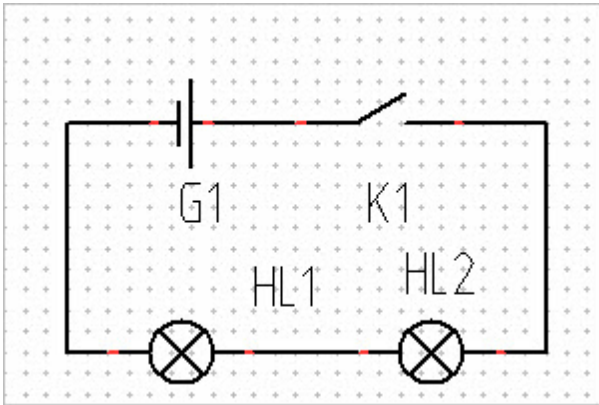
Diagram option allows to arrange the tag names of components with a specific diagram type. By default, **All** item is selected (i.e., the tag names will be arranged for all components in the document).

Default Value option allows you to set the standard numbering order for all components. Standard numbering of components: from top to bottom, from left to right.

The option also restores TN according to the TN prefix value. The default prefix value is the one specified in the corresponding line of component parameters. The figures below show a diagram before and after restoring default values.




Default Position option allows you to return the basic position for all tag names moved in the diagram. The figures below show a diagram before and after restoring the default position.



If you run the command without checking **Default Value** and **Default Position** options, the system will update the text and position of tag names that have not been changed "manually". All user TN changes will be saved. If the options are checked, the system will update all tag names, resetting user changes.


Default Value and **Default Position** options are also available in the context menu of single diagram elements.

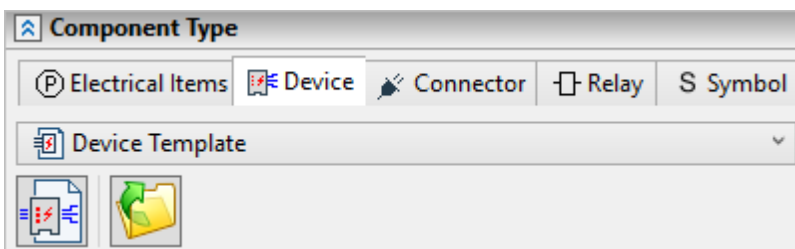
If necessary, you can disable the display of tag names. To do this, use options **Positional designation - Hide** in the context menu or **Visibility of Tag Name** in the component parameters.


You can also hide tag names of all components on the diagram. To do this, uncheck **Visibility of Tag Names** option in  **Diagram Parameters (Tag Names of Elements section)**.


Creating compound components in context of diagram

Device


To create a new device in the context of the diagram, select "Device Template" element in  **Insert Component** command.

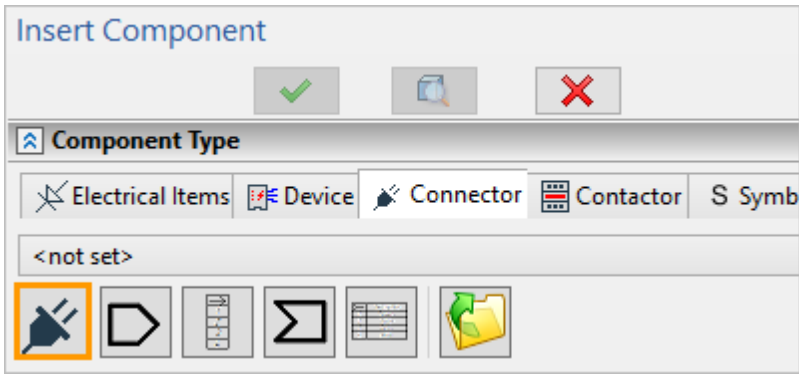


After placing an element on the diagram, the  **Create Device** dialog box opens. Here you can add connection points, connectors, terminals and terminal blocks to the device. Work with this window is described in detail in the chapter "Create device".

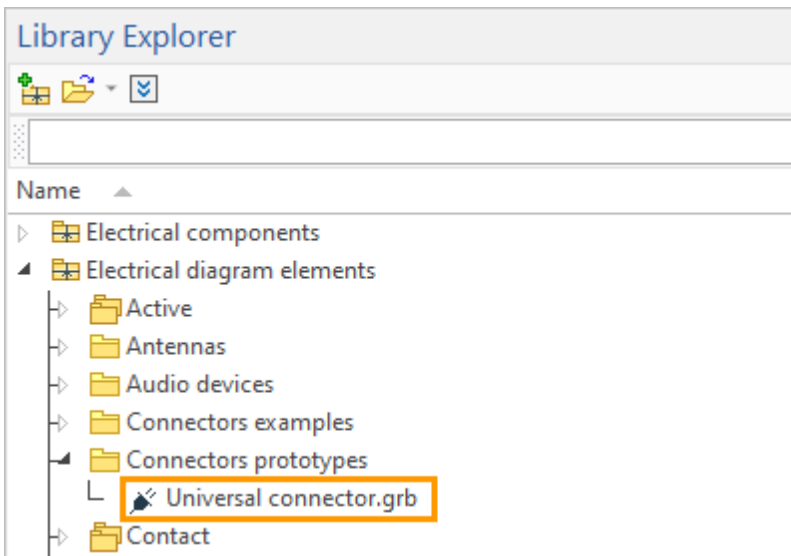
To return to the creation dialog, select the device and activate  **U: Edit Component** in the automenu. You can also call this command from the context menu of the component.

Connector

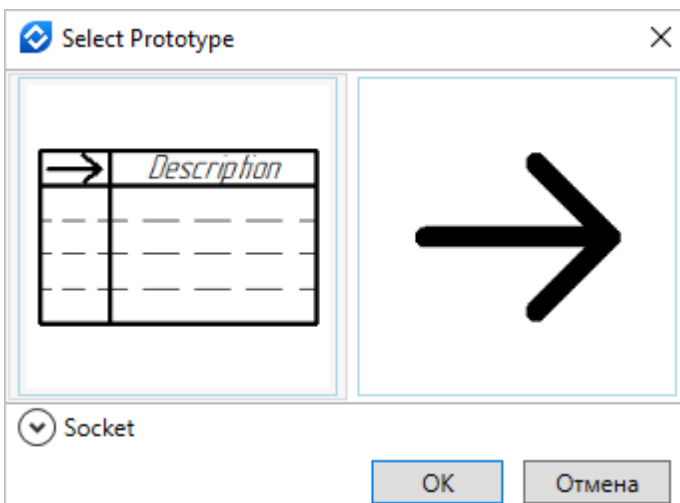
To create a new connector in context of diagram, select a corresponding template in  **Insert Component** command.




You can also insert this element from the library using drag'n'drop.

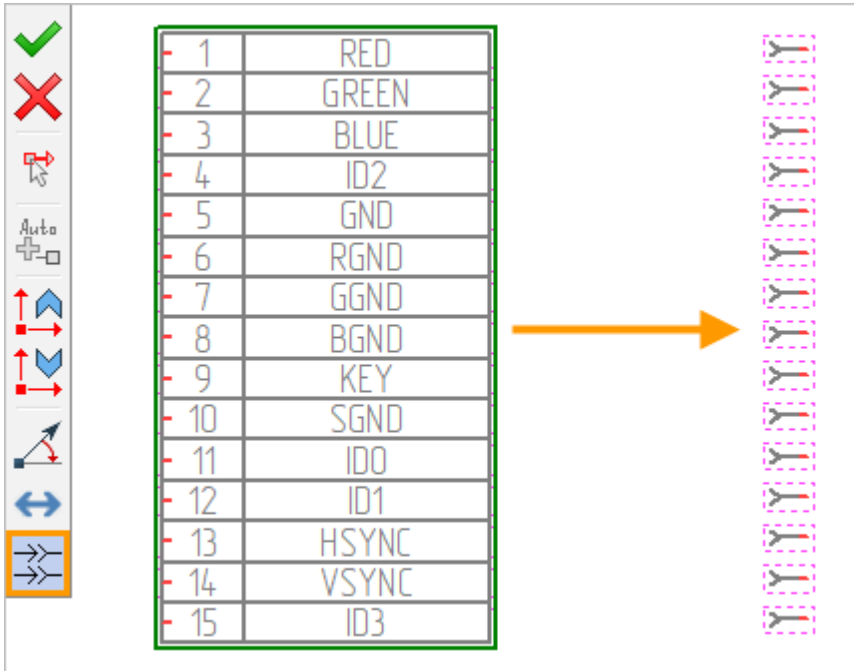


In the window that appears, you can choose between two graphical views of the connector - tabular and simplified. Select one of them and click [OK]. You can switch to separated-pins view later.




The dialog of  **Create Connector** command opens after placing the template on the diagram. Here you can add contacts to the connector. This dialog is described in detail in the chapter "Create Connector" command.


A button activating the separated-pins view is available in the automenu . To turn it off, just click on the button again.



When the connector composition is configured, click  in the automenu or at the top of the dialog.

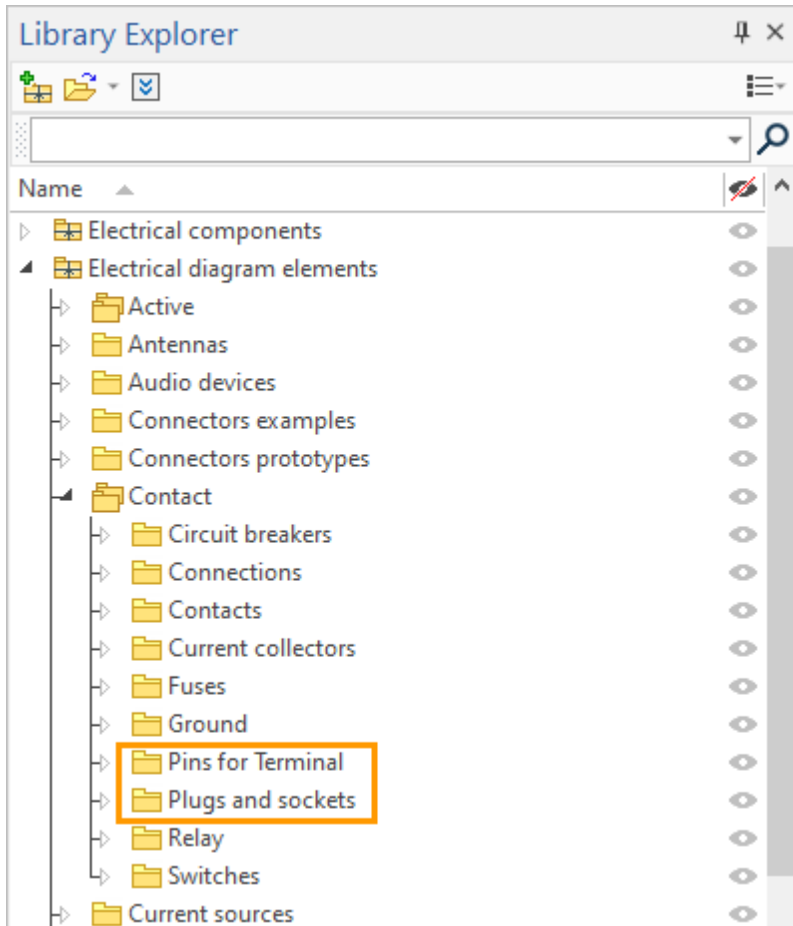
Select the connector and activate  **U: Edit Component** in automenu to return to the connector creation dialog. You can also call **Edit Component** command from the context menu of component.

Terminal Block


To create a new terminal block in context of diagram, select a terminal in  **Insert Component** command (component type - Electrical Item, subtype - Terminal).



You can also insert these element from the library using drag'n'drop.



After placing the terminal on the diagram,  **Create Terminal Block** option will appear in the automenu. Its activation will open a dialog for adding the remaining terminals.

The text inside the terminal symbol can be changed using  **Variables** command in the context menu of terminal.


Text field is used to enter a circuit designation (check box controls the visibility of the designation).


Text Length field is used to set the number of characters displayed in the terminal symbol (in fact, the parameter controls the length of the symbol).

Pin Position field is used to enter the terminal number (check box controls the visibility of the position).


You can create a terminal block in context of diagram if the terminals were inserted separately into it.

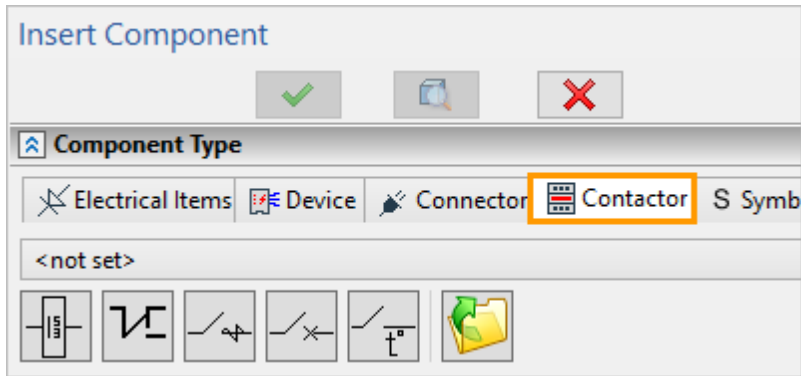
To do this, use  **Add Link With Element** command.

Another way to create a terminal block in context of diagram is to use  **Linear Array** command in the context menu of terminal.

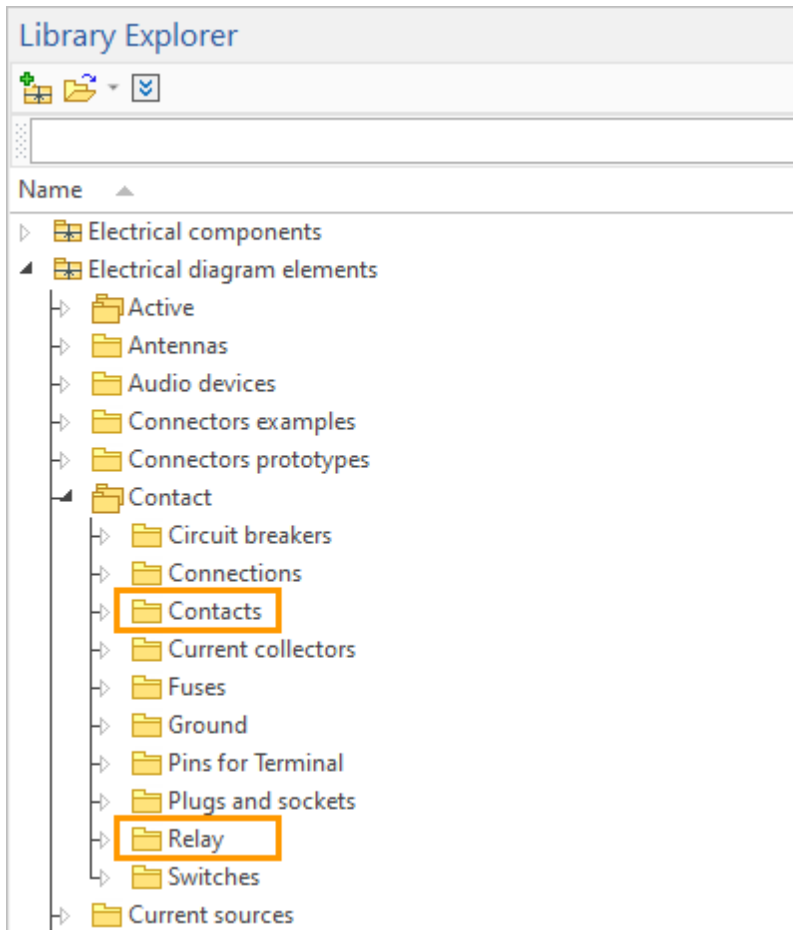
The terminals in terminal block created in context of diagram move individually by default. To move the terminal block as a whole, use automenu command  **Move Whole Compound Element**.


Contactor


To create a new contactor or relay in context of diagram, select a relay coil or a contact in  **Insert Component** command.





You can also insert these element from the library using drag'n'drop.




After placing the coil on the diagram,  **Create Contactor** option will appear in the automenu. Its activation will open a dialog for adding the contacts.


Select any element of contactor and activate  **U: Edit Component** in the automenu to return to contactor creation dialog. You can also call this command from the context menu of any contactor element.

You can create a contactor in context of diagram if its elements were inserted separately into it. To do this, use  **Add Link With Element** command.

Another way to create a contactor in context of diagram is to use  **Linear Array** command in the context menu of contact.

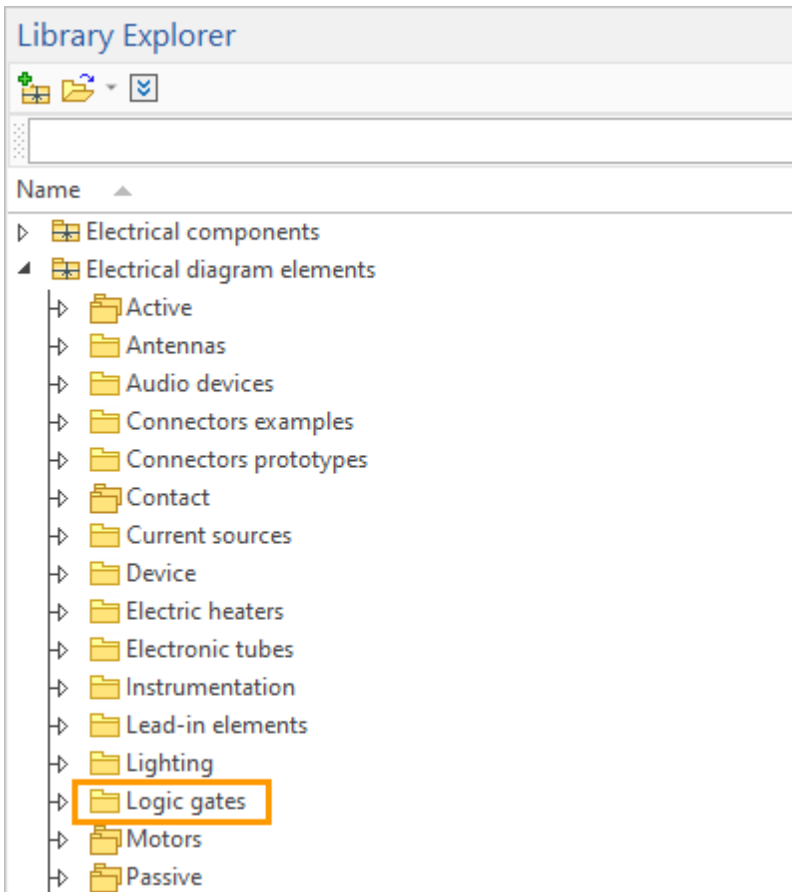
The elements of relay created in context of diagram move individually by default. To move the relay as a whole, use automenu command  **Move Whole Compound Element**.

Chip


To create a new chip in context of diagram, select a logic gate in  **Insert Component** command (element type - **Component**, subtype - **Logic Gate**).





You can also insert these element from the library using drag'n'drop.




After placing the logic gate on the diagram,  **Create Chip** option will appear in the automenu. Its activation will open a dialog for adding the remaining logic gates.

Select any logic gate of chip and activate  **U: Edit Component** in the automenu to return to chip creation dialog. You can also call this command from the context menu of any logic gate.

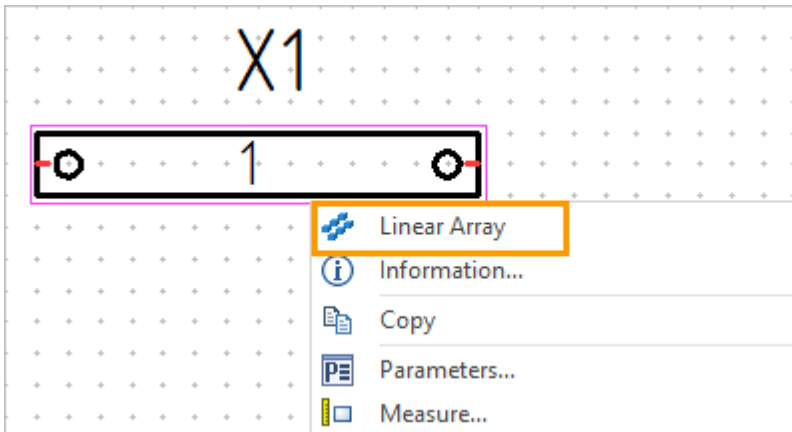
You can create a chip in context of diagram if the logic gates were inserted separately into it. To do this, use  **Add Link With Element** command.

Another way to create a contactor in context of diagram is to use  **Linear Array** command in the context menu of logic element.

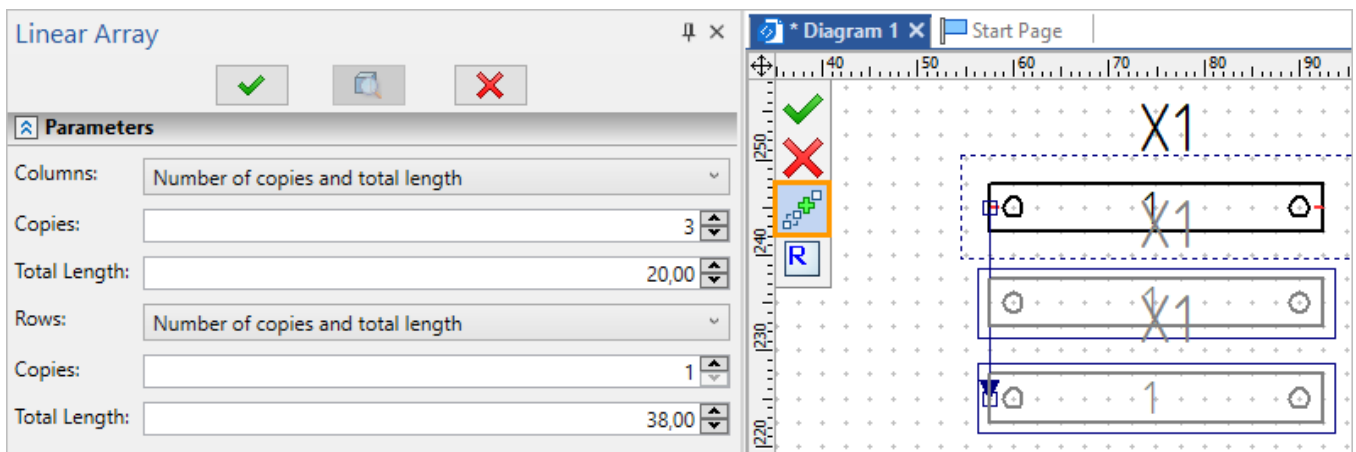
The logic gates in the chip created in context of diagram move individually by default. To move the chip as a whole, use automenu command  **Move Whole Compound Element**.

Linear array

Linear Array command is used to repeatedly insert an element into a diagram, as well as to create the products consisting of elements of the same type (terminal blocks, chips, contactors) in context of diagram. The command is launched from the context menu of a schematic element.



The following options are available in the command dialog.




Columns. The parameter controls the way to set the number of columns in the array of elements. The options are **Number of Copies and Total Length**, **Total Length and Step**, **Number of Copies and Step**. Depending on the option selected, certain fields will be shown below.

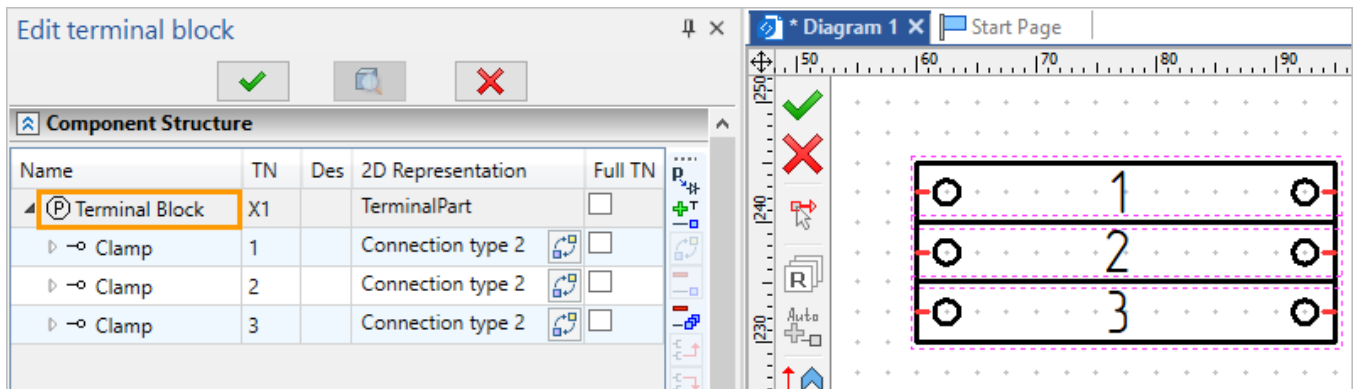
Copies. The field is used to set the number of copies in a column.


Total Length. The field sets the total length at which the columns of the array will be placed.

Step. The field sets the step through which the columns of the array will be placed.

Rows. The parameter controls the way of setting the number of rows (strings) in the array of elements. The options are the same as in the **Columns** parameter. Depending on the option selected, certain fields will be shown below. The functions of the fields is the same as for the fields under the **Columns** parameter.



The automenu option  **Unite into One Component** is used to automatically create a compound component from several elements of the same type (for example, a terminal block or chip). It is enabled by default. The head object of the compound component will be created automatically.

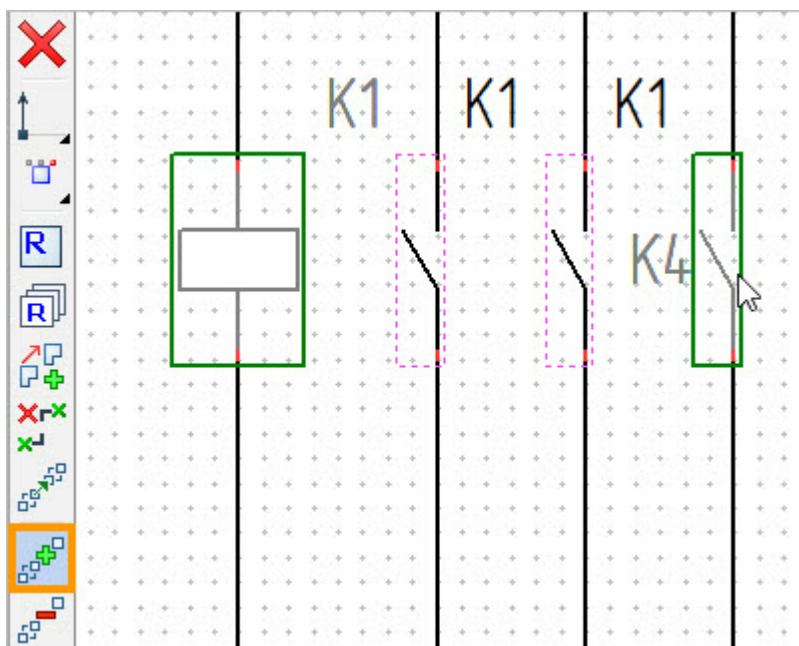


The button  in the automenu of **Linear Array** command allows you not to close the command after applying the changes.



Adding links with elements

It is possible to create a compound component from elements previously inserted into the diagram. For example, you can create a terminal block or connector in context of diagram and add them to a device, combine terminals in a terminal block, a relay coil and contacts in a contactor.


To do this, select an element in the diagram, activate  **Add Link with Element** option in the automenu, and point  the associated elements.



A dotted frame around elements indicates a link between them.

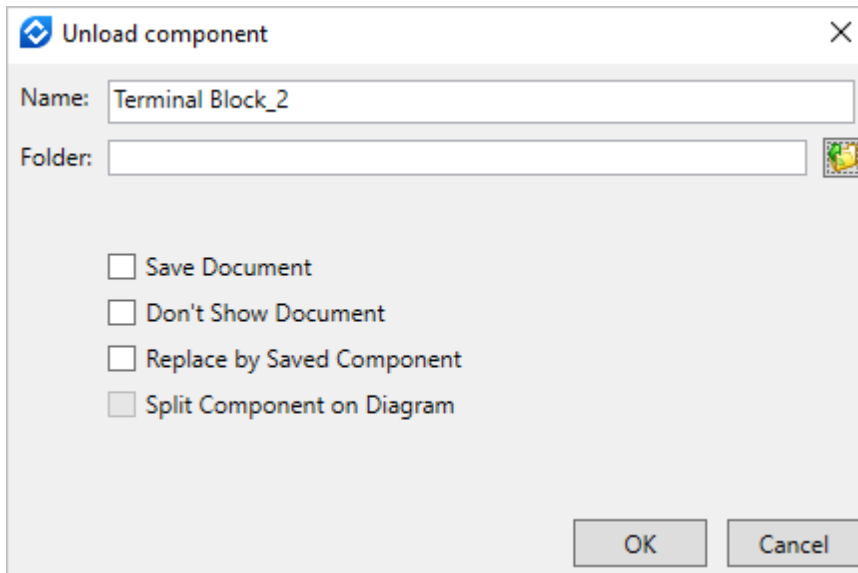
To remove a link between elements, use  **Delete Link with Element** automenu option. Using it is similar to using  **Add Link with Element** option.

"Unload Component" command

A component created in context of diagram can be saved in a separate document for later use as one of the diagram components. To do this, use  **Unload Component** command. It is available in the component context menu.

Unload Component is available for the components created in context of diagram.

After running **Unload Component**, a dialog appears:



Name field indicates the name of the document that will be created to save the component into it.

If **Save Document** parameter is checked and **Folder** for saving is specified, the document will be automatically saved to the specified directory after the command is completed.

Don't Show Document option allows you to disable the automatic opening of component document immediately after saving.

Replace by Saved Component option allows to replace the product created in context of diagram with the component inserted into the diagram as a fragment. After that, the changes in the component structure can be performed only in its file. Editing a component structure in context of diagram will be blocked.

Split Component on Diagram checkbox is available if **Replace by Saved Component** option is checked. It turns on splitted representation of saved component automatically.

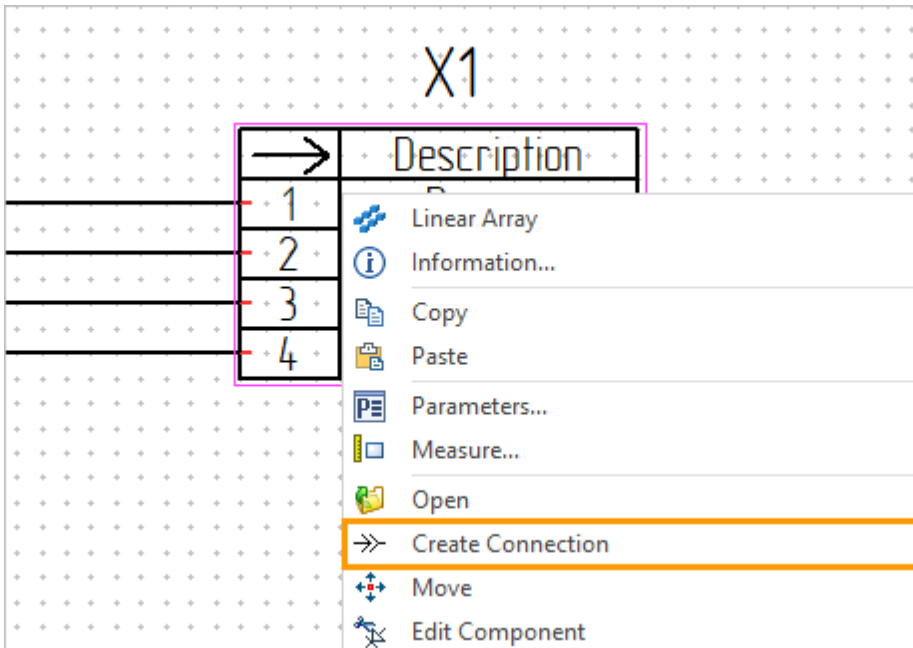
Plug and socket connection

In early versions of the module, a detachable connection (that is, the connection of two connectors - a plug and a socket) was a special diagram element that included both connectors from a pair, but having one graphic symbol. To create a detachable connection, a special prototype was used on **Start** page.

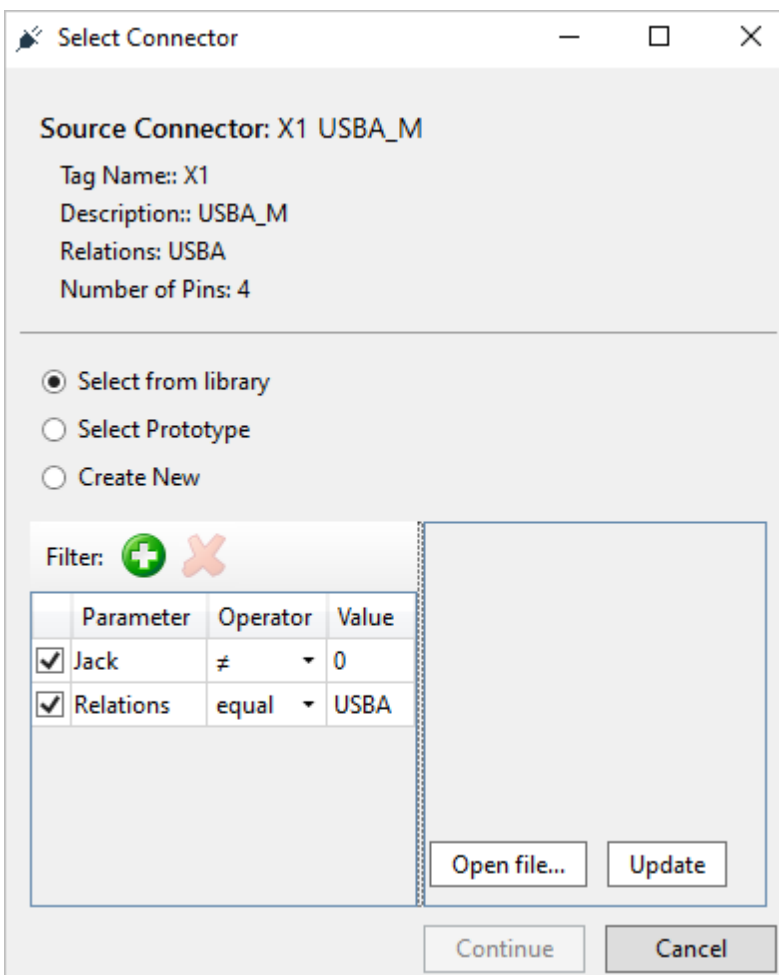
Then this principle was abolished. The connection of two connectors is now created in the diagram. There are three ways to create electromechanical connection: through the context menu of the connector, through the switching of the connection line and through the attachment of two connectors.

EMC creation via context menu of connector

The method is available for all views of connectors (tabular, pin-to-pin, simplified) and is used if a single connector is inserted into the diagram, to which you want to connect a counterpart. To connect, open the context menu of an existing connector and select **Create Connection** command.




After launch, a dialog for selecting a mating connector will open.

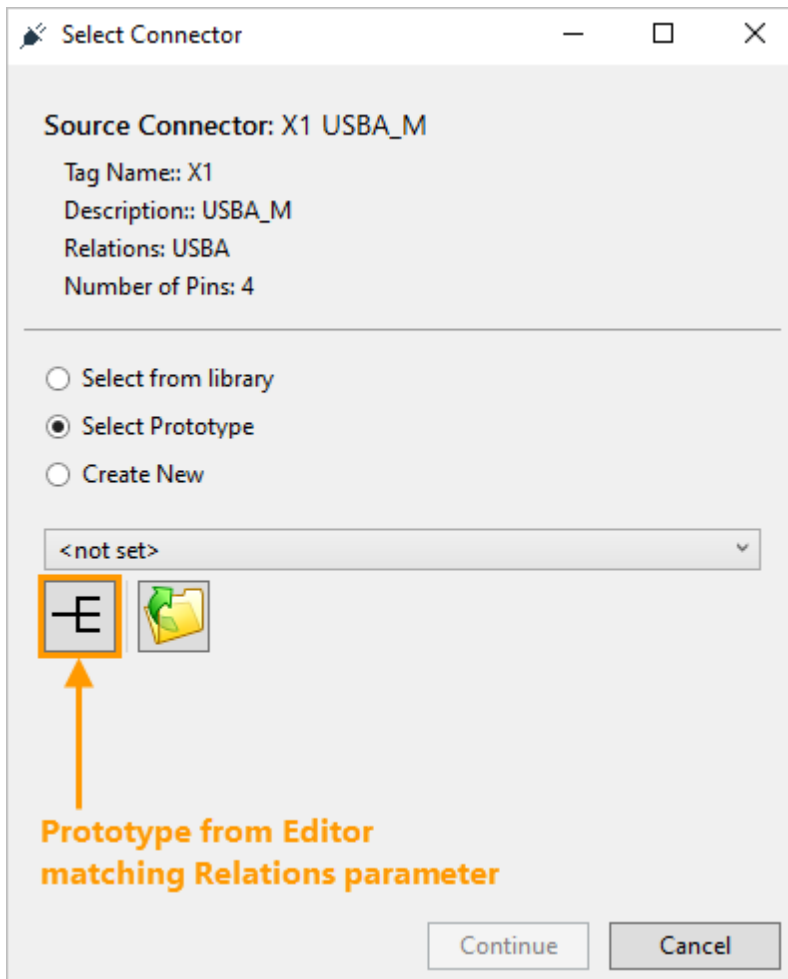


The top section shows the key parameters of the original connector. Below you need to select one of the options for creating a mating connector.

Select from library allows you to select a matching connector according to the specified filters from the switched connector library. For more information on selection, see here. [Open File] button is intended for direct selection of the connector file in Windows Explorer.

Select prototype allows you to select a connector prototype from **Component Editor** and create a

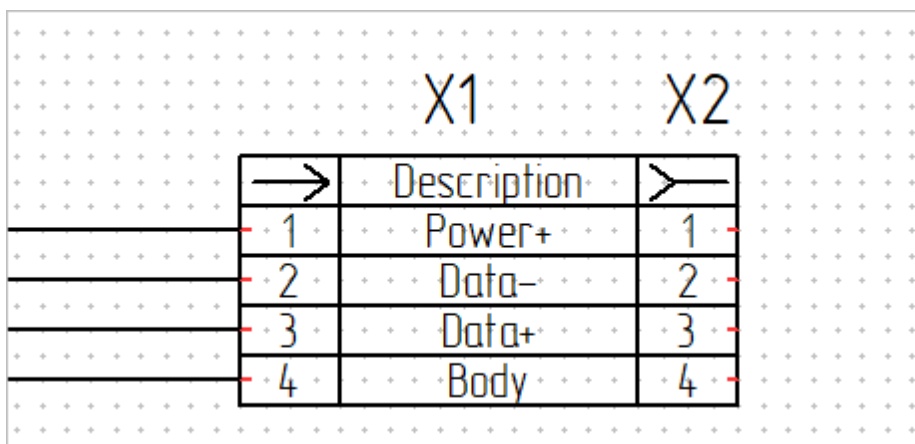
mating connector from it in the diagram context.  **Select from dialog** button opens a window with connector prototypes for direct selection. Next to it are icons of connector prototypes that match the original connector by **Relations** parameter (taking into account its "gender").



Create new allows you to create a new prototype for the mating connector. At this stage, the system proposes a **Connector** prototype in **Component Editor**, whose parameters have "neutral" values.

After specifying the settings in one of the options, click [Continue]. A window with the parameters of the mating connector will open, their values can be edited if necessary. Click [Done] to confirm the creation of the mate connector.

The mating connector is added to the diagram.

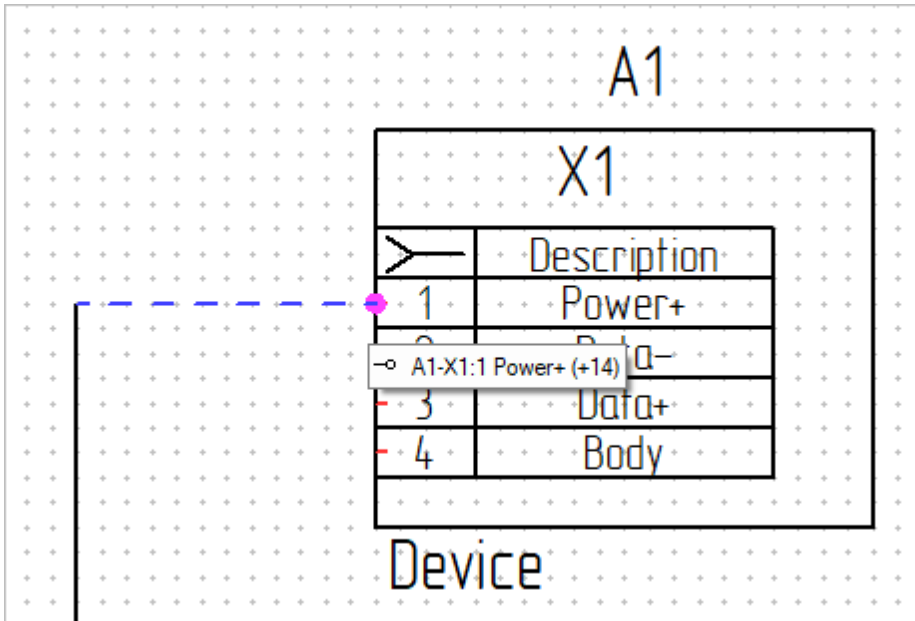


By default, it only displays a column with description of contacts. To configure the view of the connector on the diagram, use the command  **Variables** in its context menu. From the same menu

you can call its  Parameters dialog.

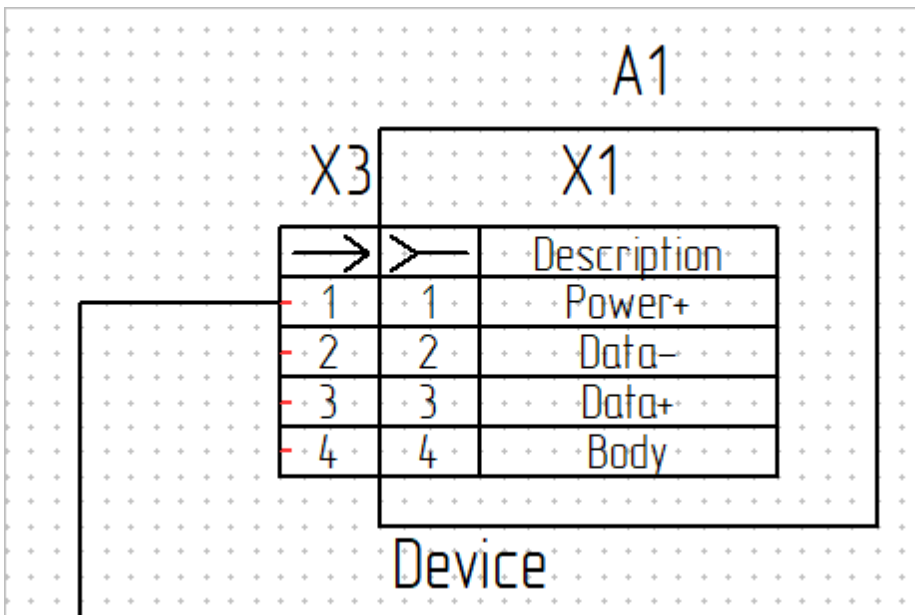
EMC creation by switching a connection line

The method is available for all connector representations within devices, as well as for free simplified connectors. To create a mating connector, connect the connection line to the pins of initial one.



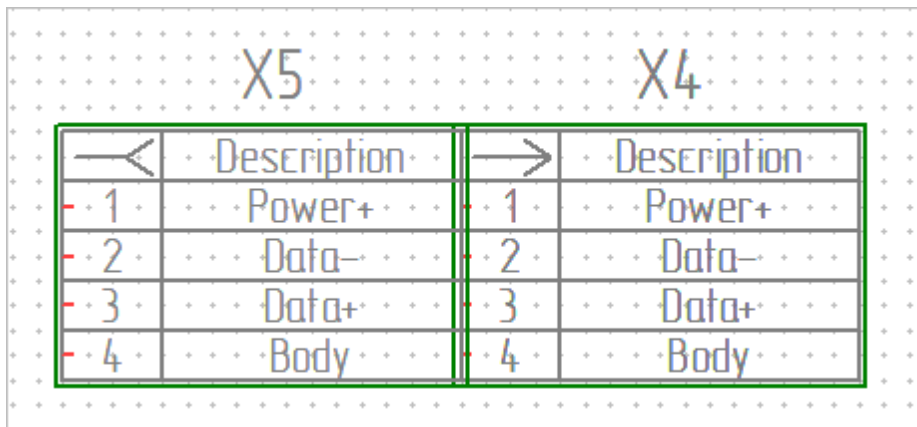
The dialog for creating a mating connector, described above, will open. The procedure for selecting a mating connector is the same.

This adds the mating connector to the diagram.

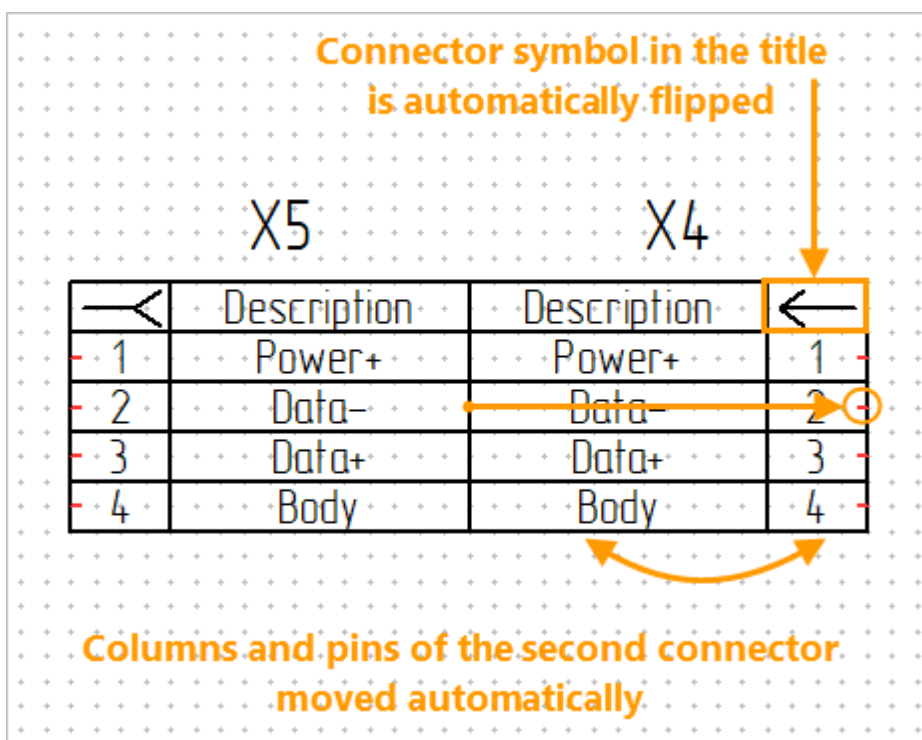


EMC creation by attaching two connectors

The method is available for all connector views and is used if you need to connect to each other two connectors that are already inserted into the diagram. To connect, place the symbols of connectors on the diagram close to each other. The position of the free contacts is irrelevant in this case.




At the time of connection, the connectors will be marked in green, after which the message "Mating pins of connectors found. Create electromechanical connection?" If the answer is positive, a detachable connection will be created on the diagram, if it is negative, the connectors will remain unconnected to each other.



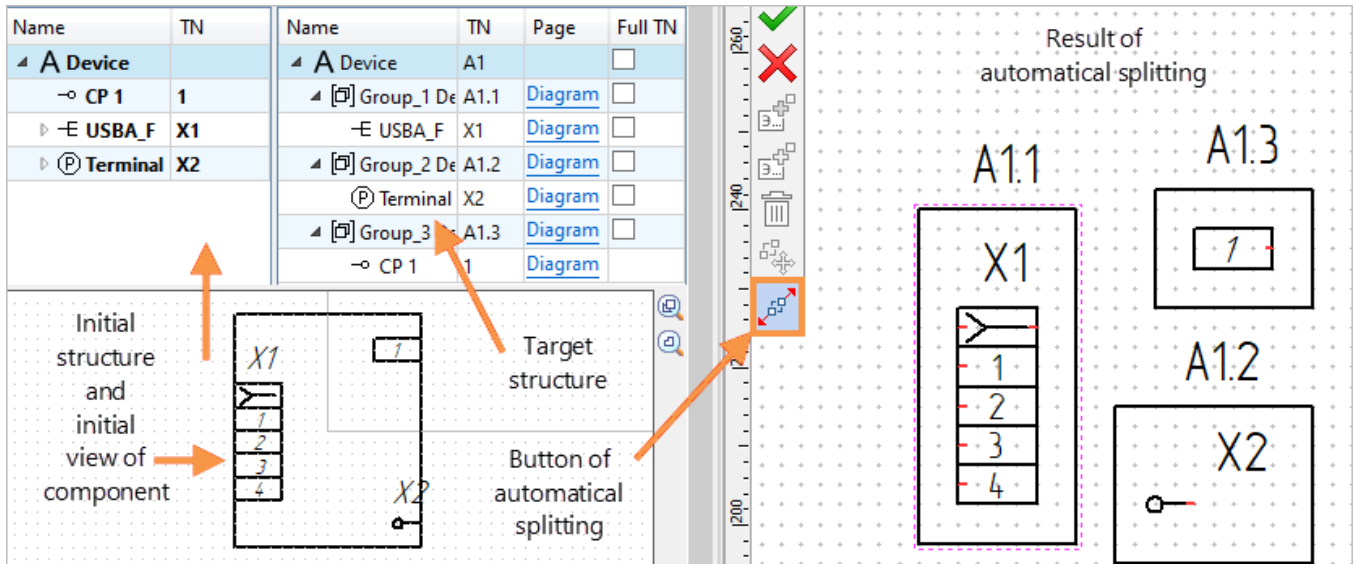
After creating the EMC, the columns of the connectors will be automatically rearranged, the connector symbol in the title of table will automatically rotate, and the connection points will be located "outside" the symbol of the connectors.

To disconnect the connectors from each other, call **Break EMC** command in the context menu of any of them. The position of the connectors will not change, but the link between them will be deleted and they can be moved around the diagram independently.

Splitted representation of compound component


It is possible to depict compound components in splitted way in T-FLEX Electrical. To do this, use  **Splitted Representation of Component** command. It can be launched from the automenu or from the context menu of the diagram element.



The dialog always displays the initial view and initial structure of splitted component.

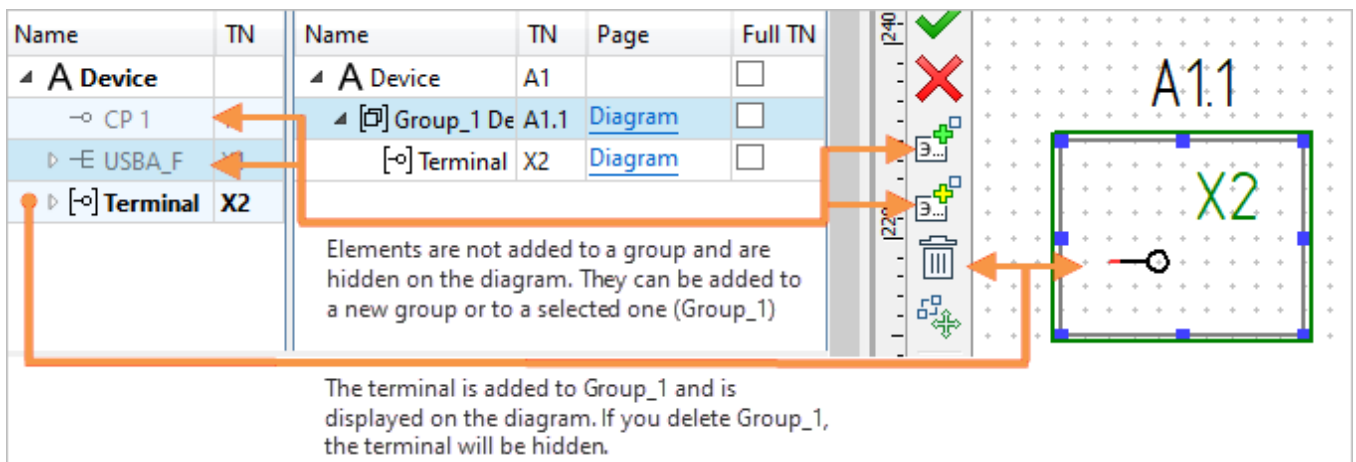


A **Group** is a part of a component that includes some of its elements.


Two splitting modes are available in the command: automatic and manual.

Automatic splitting  can be activated in the command's automenu, and there will be one element in each group. Thanks to this, each element of component can be placed in an arbitrary place on the diagram. You can manually change the result of automatic exploding. You can also not use auto-splitting at all and split the component into groups at your discretion.


When manually splitting the component, you can use option  **Add to New Group** (the group will be created automatically) or  **Add to Selected Group** (the element will be added to one of the existing groups; the group must be preselected). The element included in the group is marked in bold. For such an element, the commands for adding to groups become inactive.



Selection of element in one of the blocks marks it in the remaining blocks, as well as in the diagram.

Use  **Delete** automenu option to delete one or several groups. After that, the elements included in them will be available for adding to other groups.

Some elements of the component may not be added to groups at all. This will allow you to display the component on the diagram not completely, but partially.

The group can be moved to any other page in the diagram. There is a special button  for this in the automenu. It also allows you to move the group around the current page without closing the command. The name of the current **Page** where the group is located is displayed in the dialog.

Name	TN	Name	TN	Page	Full TN
▲ A Device		▲ A Device	A1		<input type="checkbox"/>
-○ CP 1	1	▲ [□] Group_1 De	A1.1	Diagram	<input type="checkbox"/>
▶ -E USBA_F	X1	[-○] Terminal	X2	Diagram	<input type="checkbox"/>
▶ [-○] Terminal I	X2				

The name of current page and the button for moving the group

The buttons for adjusting the group frame (only when splitting device and functional block)

The button for deactivation of splitted representation

The device and functional block are framed by default. As a result, when splitting, their groups will also be surrounded by frames, the size of which is calculated automatically.

If necessary, you can manually resize the frame using the point manipulators.

If a device group or FB group is selected during splitting, the buttons for contour control and are available in the automenu.

Default group designations consist of tag name of the initial component and the serial number of the group, separated by a symbol (for example, A1.1, A1.2, etc.).

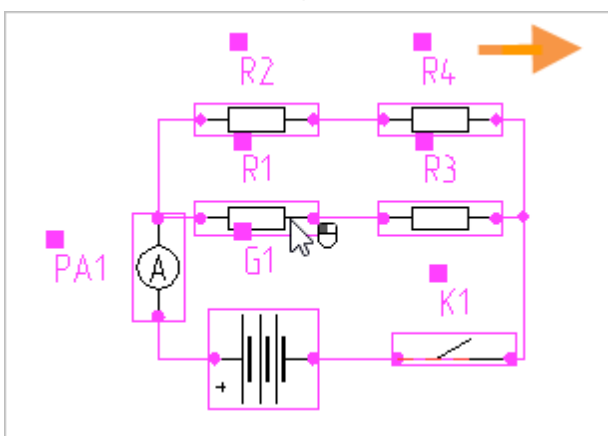
To hide the serial numbers of groups in the diagram (for all components), use **Visibility of Group Suffix** parameter in **Diagram Parameters** command. If it is unchecked, all groups in the diagram will be displayed only with tag names of initial component (A1).

To set the separator symbol in the group designation, use **Separator for Groups** parameter in **Diagram Parameters** command. This parameter allows you to designate groups as A1:1, A1-1, etc.

To return the component to its initial state and delete all the groups, use **Splitted Representation of Component** option in automenu again.

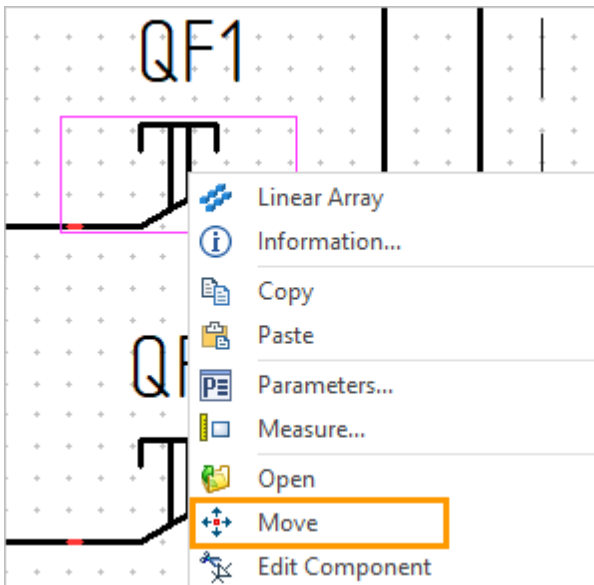
Moving and copying components

If you need to move all or several components on the diagram, select and move them with pressed . The selected components can be also moved using arrow keys on the keyboard.




To move component from one page to another, use **Move** command in the context menu of an element. After its launch, the component will disconnect from the connection lines and will follow

while you can switch between pages. Clicking in the page field will place the component at this point.



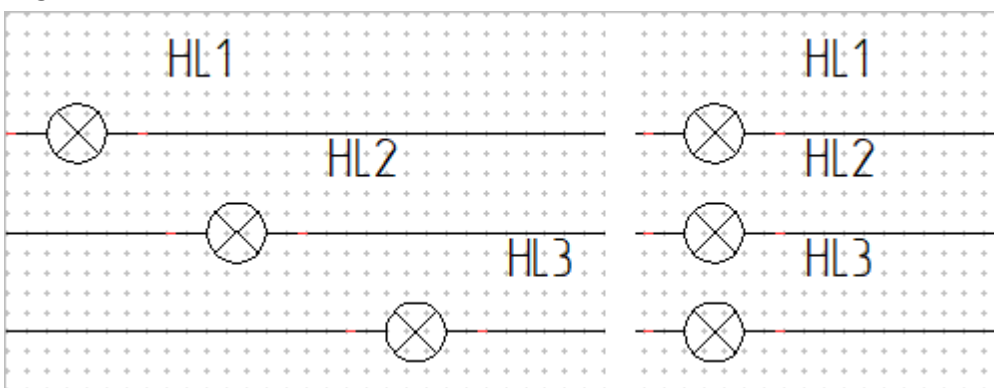
You can also copy all diagram or its elements using **Copy**, **Paste** commands from the context menu. In this case, the following tag names will be assigned to the copied elements.

If the diagram is copied on a standard 2D drawing page, a copy without structural elements will be created on it.

In the automenu of selected component, the option  **P: Copy Component** is available. The option is convenient to use if the parameters of component have been changed in the diagram and they need to be copied or if you need to insert into the diagram the component created in context of diagram again.

If you need to arrange the position of elements in the diagram, you can use **Align** command. The command will appear in the context menu if several elements of the diagram are selected.

4 alignment options are available: left, right, top, and bottom. The alignment is performed relatively the "virtual" line where the left (right, top, bottom) extreme connection point lies. The extreme point is selected from the connection points belonging to the selected products. For example, the alignment on the left result would look like this:

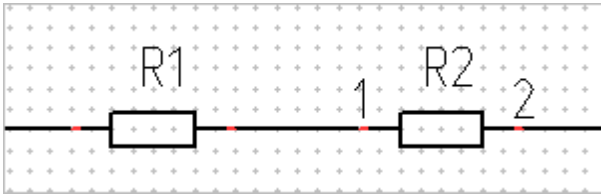



Connection points of component

Each connection point of the component has its own tag name and full tag name. The full tag name of connection point includes the tag name of component to which it belongs and own tag name of CP (for example, R1:1). The numeration order is set when the component is being created. At the stage of component creation, serial numbers are used by default as own tag names of connection points.

The wires table uses the full connection point tag name as the contact address.

Connection points - Show / Hide option is available in the context menu of components to display CP denotations. This option displays the serial number of CP in the component by default. The same function is performed by **Visibility of Connection Points** option in the parameters of diagram component.




The tag name and, accordingly, the full tag name of connection point can be changed by the user. It is convenient to do this in the command  **U: Edit Component**. The changes will be reflected in wires table.

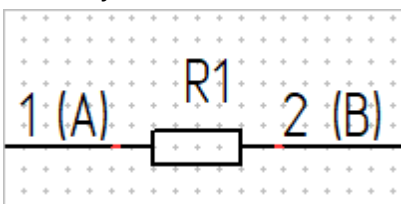
Component Structure

Name	TN	Des	2D Repr	3D Repr
Normally open contact de	S1		Normally	
Connection Point	1A		Fragment	
Connection Point	2		Fragment	

Parameters

Parameter	Value
For Reports	
Tag Name	
Full Tag Name	S1:1A
Tag Name	1A
Use Full Tag Name	<input type="checkbox"/>

If it is necessary to provide other clarifying information about connection point in the diagram in addition to the number, you should enter it in **Designation** parameter of CP. In  **U: Edit Component** command, the parameter displays its own **Designation** column in the dialog. To see the result on the diagram, you need to update the denotation of CP. To do this, use **Connection Points - Show** context menu option of component. The designation will be shown in brackets next to the number. If **Designation** parameter is filled in, its value will be indicated in brackets next to the tag name by default.



By default, **Tag Name** parameter is used to designate CP on the diagram. **Use CP Designations instead of System Values** checkbox allows you to use another CP parameter for this purpose - **Designation**. Previously, without this checkbox, it was impossible to use an arbitrary alphanumeric


combination to designate CP, since the CP Tag Name parameter supported only numbers, and the Designation parameter supported numbers and letters. Now you can enter arbitrary text into the CP Tag Name parameter, so the need for a checkbox has decreased. However, its function can still be used.

Name	TN	Des	2D Representation	Full TN
Pushbutton	QF1		Pushbutton	
Connection point 1	1	A1	Fragment Connector "CP1"	
Connection point 2	2	A2	Fragment Connector "CP2"	


Parameter	Value
Annotation	
Visibility of Annotation	<input type="checkbox"/>
Connection Points	
Format of CP Designations on Diagram	Pin (SpecDenotation)
Use CP Designations instead of Syst	<input checked="" type="checkbox"/>
Visibility of Connection Points	<input checked="" type="checkbox"/>
For Reports	
Data for PS from 3D	<input type="checkbox"/>
Description	Pushbutton


Show Lin...	Wire	Wire - Length	Wire - Unit	From - Connection Poi...	To - Connection Point
	Co...	*	m	GND1:1	GND1:1
	Co...	*	m	GND2:1	VD1:2
	Co...	*	m	GND2:1	#FG1-VD1:2
	Co...	*	m	GND2:1	#FG1-VD2:2
	Co...	*	m	QF1:A2	QF2:2
	Co...	*	m	QF1:A2	#A1-QF3:2

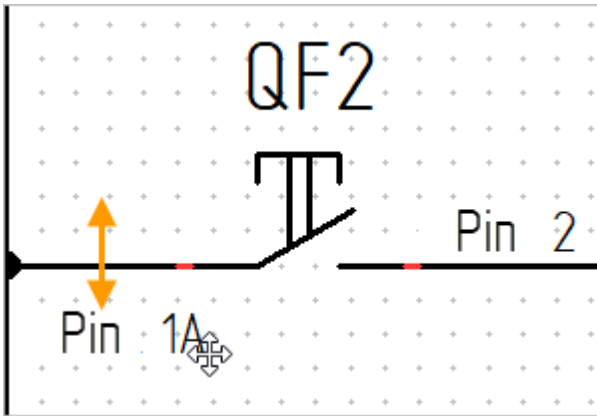
The denotations of the connection points in the diagram can be fully configured by the user. The wires table always displays the full tag name of connection point. By default, the diagram displays its own tag name. **Format of CP Designations on Diagram** field allows you to designate connection points in the diagram differently from the ones indicated in the wires table.

Parameter is available in component parameters and in  **Diagram Parameters** command. Thus, it is possible to configure CP denotations selectively or for all components at once. It takes to enter system synonyms of CP parameters in curly brackets {} and / or arbitrary text in this field. For example, a synonym of **Designation** CP parameter should be entered as {SpecDenotation}.


To get acquainted with the list of synonyms, run  **Measure** command in the context menu of CP.

The position of all CP denotations on the diagram relative to themselves can be controlled using  **Diagram Parameters** command (**Connection Points Denotations** settings group).

If you need to place the CP denotation at will, select it  and move to an arbitrary point in the diagram.

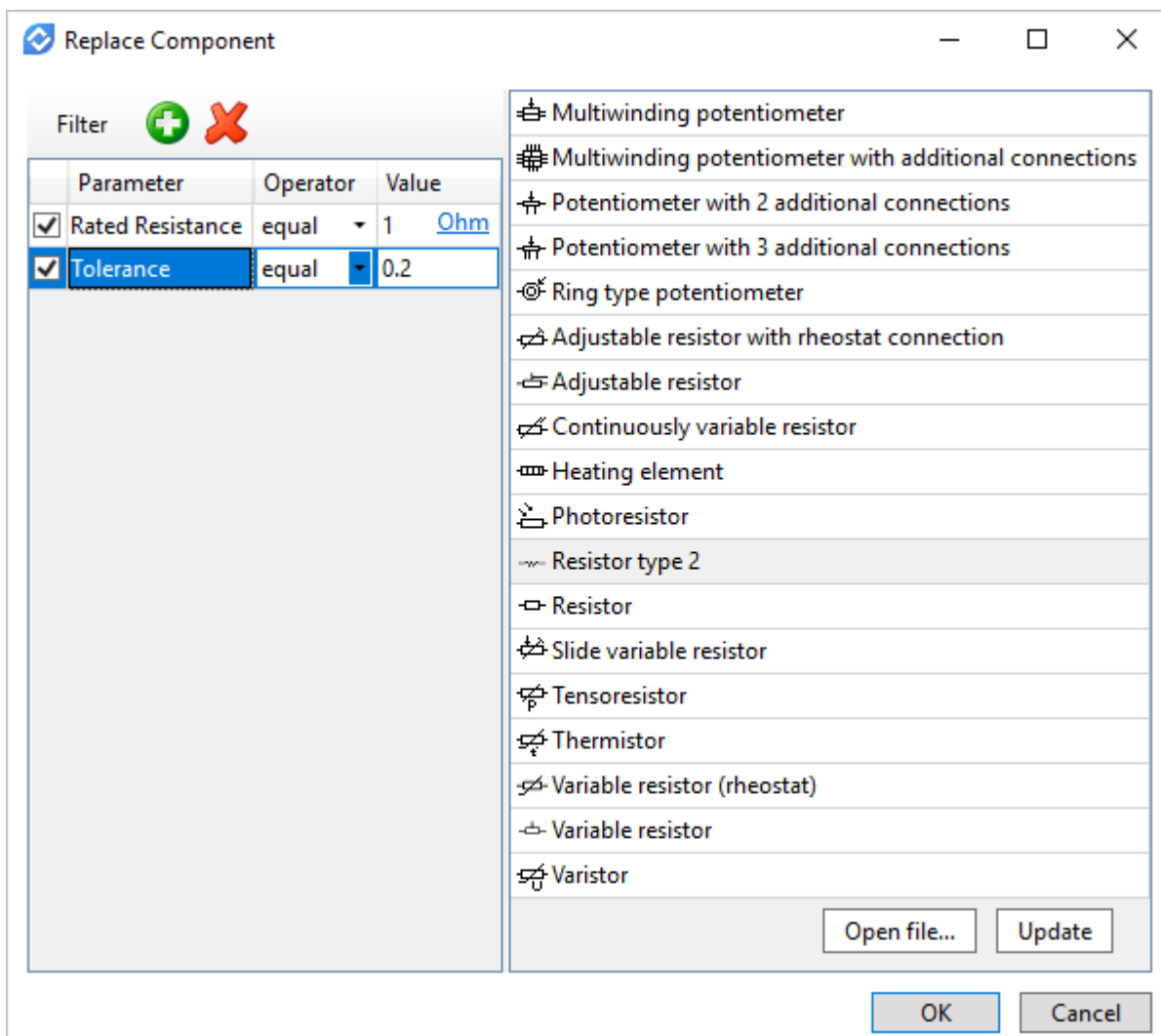


Selection of similar components. "Replace Component" command

 **Replace Component** command is used to replace the components (of **Electric Items** type) in the diagram. It allows you to select a component for replacement using the parameter filter. Direct replacement of one diagram component with another without parameter selection is also available.

Let us consider the option of replacing the resistor in the diagram.

After the command is launched, a dialog opens. To download the latest information about components, click [Update]. After that all library components created using the "Resistor" prototypes will be shown in the right section of dialog.







If new prototypes were created and (or) the parameters of existing prototypes were edited in



Component Editor, you need to [Update] the data in **Replace Component** command. Due to this, the changes made will be taken into account when selecting replacement components.

The left part of the dialog shows the parameters. The elements for replacement will be selected by its values. Initially, this list is empty for most elements, because only unique parameters of the selected electric item type are entered in this list by default (for details on the unique parameters, see "Component Editor and Structural Elements" chapter).

To add a filter (**Parameter**) by which elements will be selected, use . After selecting a parameter, make  on it to add to the list. To remove a parameter from the list, use .


The set of parameters in the drop-down list is taken from  **Component Editor**. If you need to select elements according to some parameter that is not in the list, you will need to add this parameter to all elements of the corresponding library. Then [Update] the parameters data in **Replace Component** command.

Operator column allows to specify how the components will be selected according to the specified parameter. If a component is selected according to one of the text parameters (say, **Name**), then it is convenient to set **Contains** operator for it. Equality / inequality operators can be set for selection by numerical parameters.

Enter the value of parameter by which the components will be selected in **Value** column.

If component selection by parameters is not required and it is enough to select a specific component to be replaced with, use [Open File] and select the file with the required electric item in the explorer.

Component on different diagram types

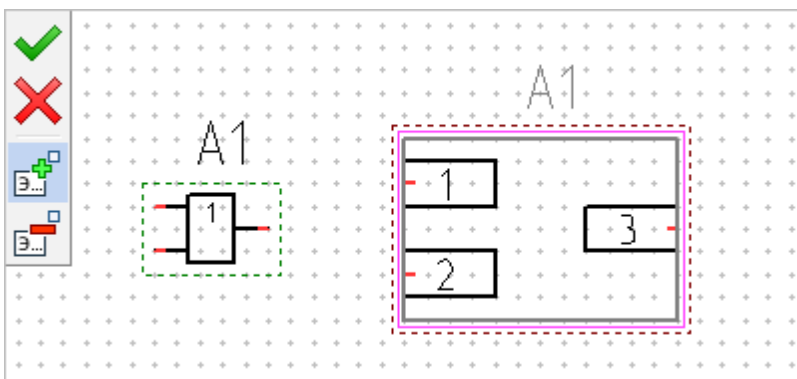
One component on different diagram types can be represented in different ways. To create a link between representations, use option  **Add Link to Component on Diagram**.

This option is available in the automenu of diagram components.

The link can be created only between components that belong to different types of diagrams.

To add a link, select a component, activate **Add Link to Component on Diagram** in the automenu and select the component with which the link will be created. A linked component can belong to the same page or to another (within the same document).

After selecting the components, the system will ask if it is necessary to synchronize the tag names for different representations of the selected component. The image below shows the result of synchronization.



After adding a link, you can see in the command dialog which elements represent the component on

which diagram.


Diagram	Component
Block	not set
Functional	Logic Gate Symbol_1
Schematic	not set
"From/To"	not set
Connection	Device_1
General	not set
Layout	not set



When you select a component in a dialog, the system activates the page on which it is placed.

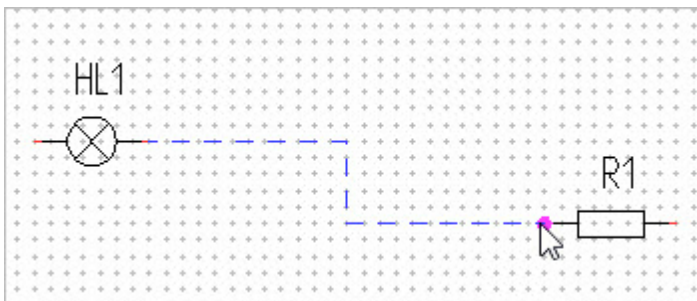
Connection lines

"Connection Line" command

To create line of electrical connection use a command:

Icon	Ribbon
	Electrical ? Diagram ? Connection Line
Keyboard	Textual menu
QW	Electrical > Constructions > Connection Line

Select  two connection points to connect two components. When pointing  at the connection point of the component, it will be marked with a pink marker. If CPs that you want to connect do not lie on the same vertical / horizontal line, the system will automatically create a kink on the CL.



You can specify **Type** and **Color** of the line in the dialog window of **Connection Line** command.

Parameters ↑ ×

Line

Type: Continuous ▼

Color: ▼


Parameters

Screened







Partial Screened

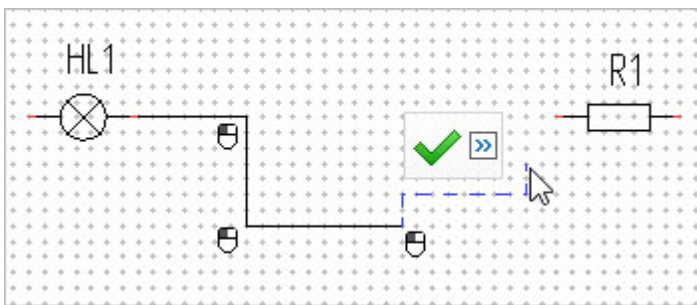
Coaxial

Twisted Pair

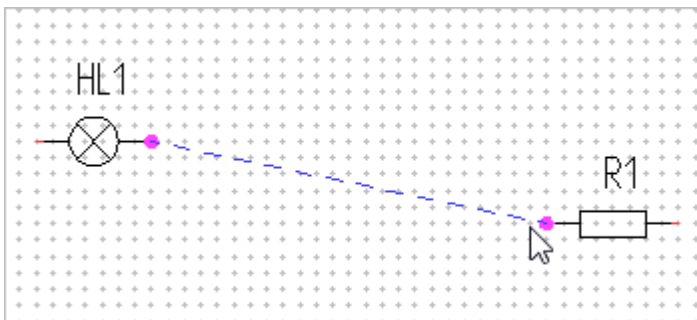
If you need to adjust the thickness of CL, go to  **ST: Document Parameters** command. **Thick Lines** parameter in **Lines** section (Diagram page) controls the thickness of CLs in the diagram.


You can add shield, coaxiality and twisted pair symbols to CL, as well as delete them in **Parameters** section of **Connection Line** command. For more information, see the chapter "Designations on Connection Lines".

A CL follows  under construction. You can  on the free point of the diagram during the construction process, after that two options are available. The first is to  on the dynamic panel to complete the construction of CL at a given point. The second option is to continue moving . In this case, the CL will continue to be drawn. Changing the direction of movement  will allow you to create a CL kink at the specified point of diagram.  after specifying the kink point cancels the creation of it in CL construction mode.

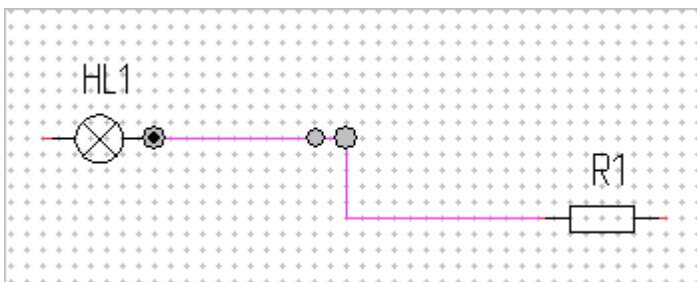





You can draw CL at any angle if necessary. To do this, hold down <Shift>.

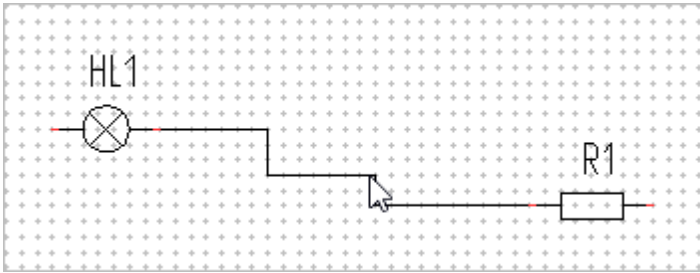



You can create CL without launching **Connection Line** command from Ribbon / text menu. It's enough to  at the first connection point, and **Connection Line** command will start automatically.

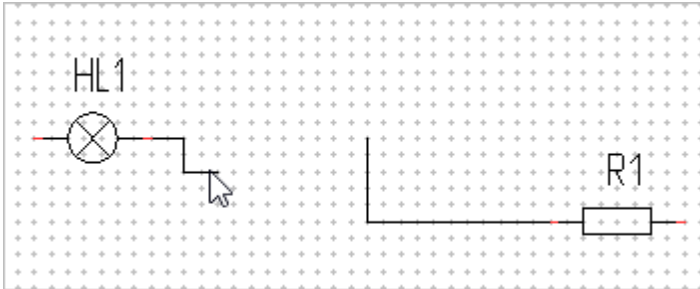
Point  at connection line to change it. Special markers for editing the line appear.




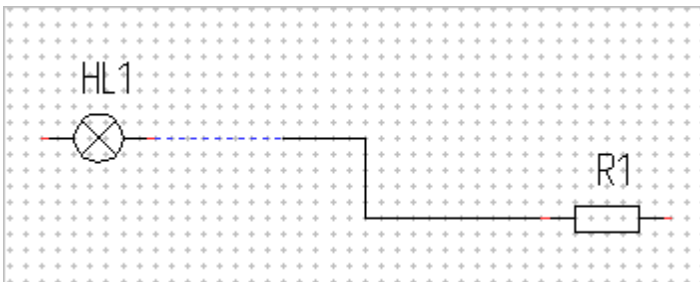
A large marker in the kink point allows to change the geometry of CL. To do this,  on it (the marker turns orange) and move . A connection line will follow , changing its length and geometry of the kinks.



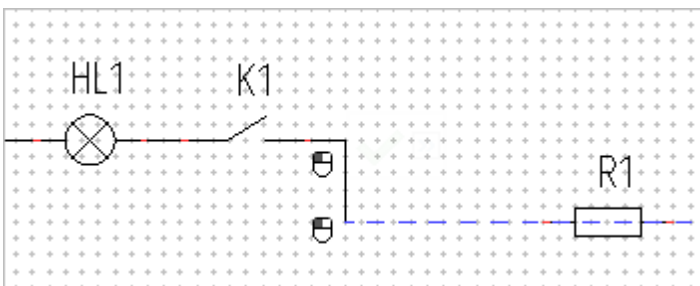
A small marker is used to break the connection line at a given point. After the break, one of the lines will follow , and it can be immediately connected to another CP.



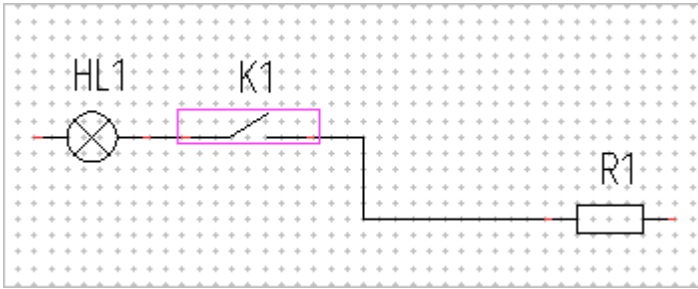
A marker with a black dot indicates that a connection has been created between the component CP and CL. At the same time the connection provides a fixation of CL end at this point.  on this marker allows you to disconnect the CL from the connection point on which it is displayed.








It is possible to draw one communication line through the connection points of several components without creating a separate CL for each pair of CP. In this case, you will need to control the position of the kinks manually. The breaks of CLs at the connection points of components will be created automatically.

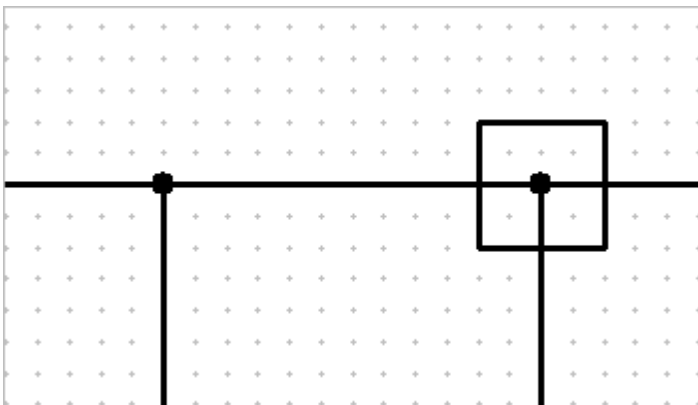





A reverse scenario is also possible - the placement of components on previously constructed connection lines. The breaks of CLs at connection points of components will be also created automatically.

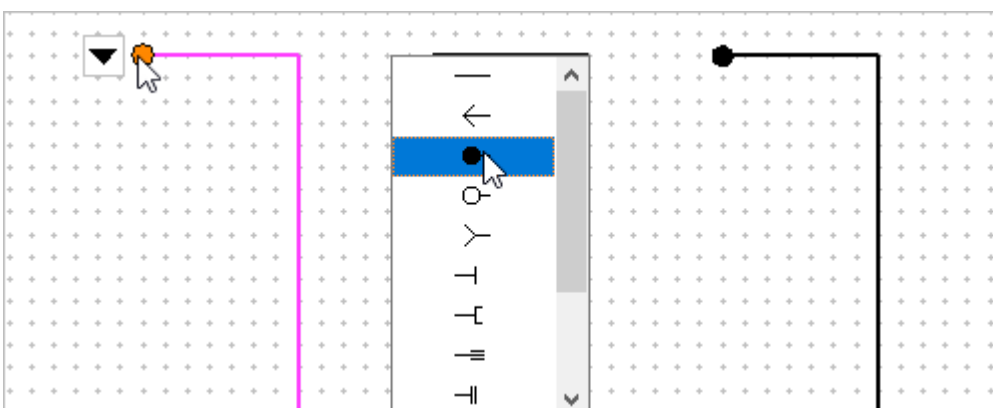



If the ends of connection line are not connected to CPs of the components, then CLs will move when editing. To fix the end of CL at one point, hold down <Ctrl>, and then  at the end point (it will be highlighted with an orange marker at that moment). If the point is fixed, a marker with a black dot will appear when you point  at it.

Connection lines may branch. The system automatically marks the branch points when connecting the CLs. If it is necessary to indicate a splice coupling at the branch point of CL, point  to this point and  on the marker that appears next to it .



It is possible to change the form of CL end. To do this, point  to the endpoint of CL. A marker  will appear next to it,  on it will open a list of available end forms. Select the appropriate form from this list.



When connecting products with connection, a new structural element  "Connection" is created. It contains information about connected components (see the folder "Structural Elements" in the model tree). The CL is a 2D representation of this structural element. The connection information will be used subsequently to create reports and set cable components.


An electric connection line does not carry by itself information about the leaders making this connection.

However, this information may be required in solving two tasks:

1. An electrical diagram of the product has been created, cable components are required to be set (to display information about them on the diagram and compile reports). The solution to this task is described in "Cable Components in Diagram".
2. An electrical diagram and a 3D model of the product have been created; cable components need to be set (to display information about them on the diagram, complete the 3D model and compile reports). The solution to this task is described in "Cable Components in 3D model".

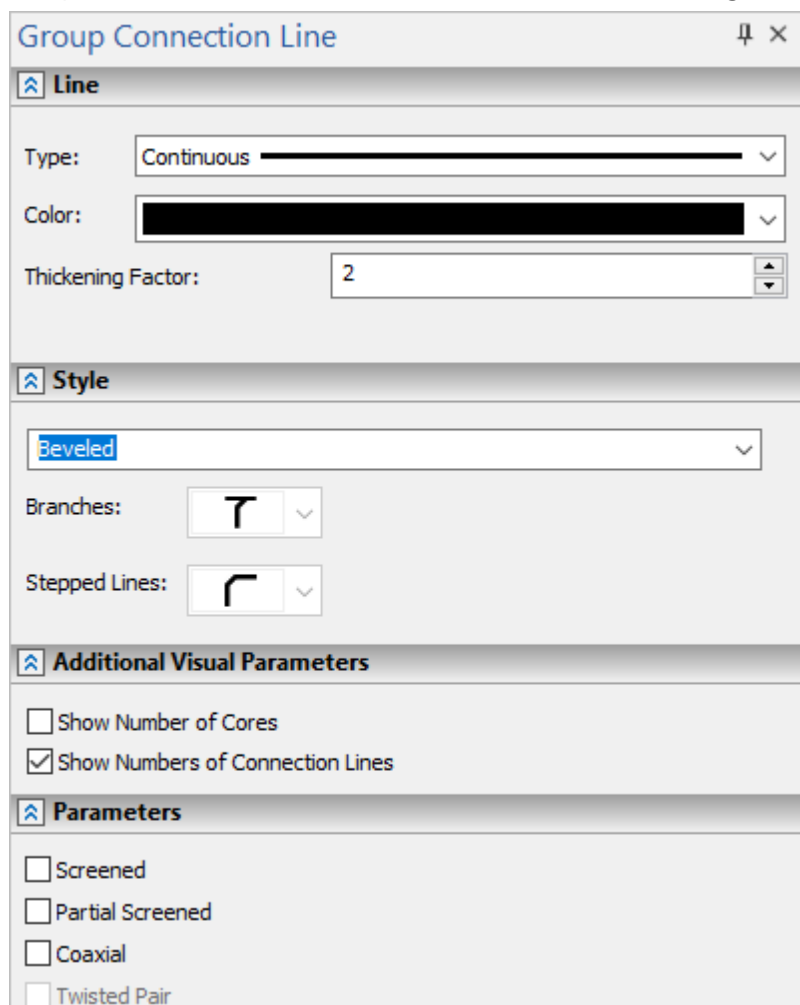
"Group Connection Line" command

Use the following command to create a group connection line:

Icon	Ribbon
	Electrical ? Diagram ? Group Connection Line
Keyboard	Textual menu
QR	Electrical > Constructions > Group Connection Line

A group connection line is constructed similarly to a simple CL. However, it has her own features.

You can set **Type** and **Color**, as well as **Thickening Factor** in **Line** section of GCL creation dialog. The last parameter controls the thickness ratio of GCL to regular CL.

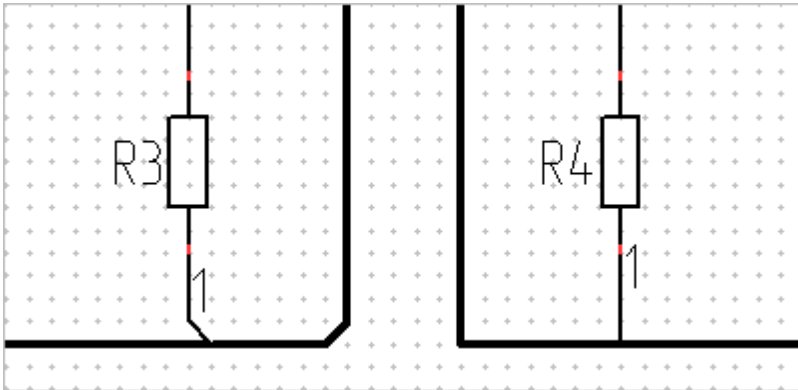


Style section is used to configure the geometry of branches and kinks of a GCL. A branching is a point on an GCL, where a regular CL is attached to it.

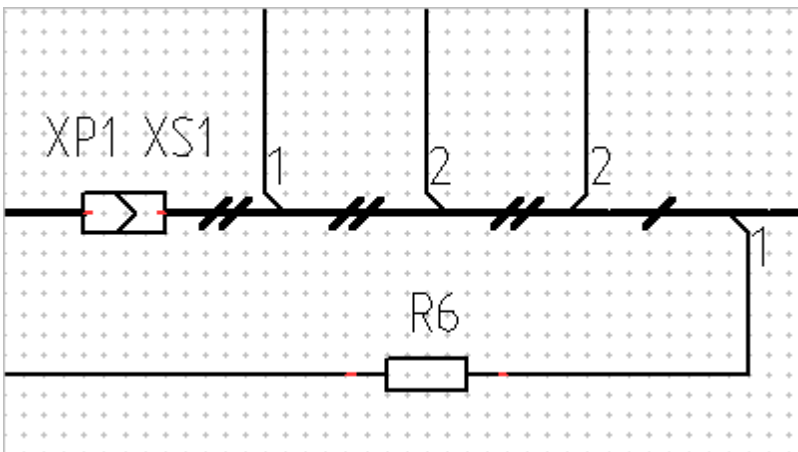
When connecting the regular and group connection lines, a bevel is formed on regular one. Its

presence or absence is controlled by **Style** parameter of group CL. Due to this, the geometry of all the branches related to the group CL is simultaneously edited.

Branches and **Stepped Lines** can be drawn with or without bevels. Depending on this, **Beveled** and **Single-line** styles are distinguished. A **Custom** style is also available. You can configure an arbitrary combination of branching and kinks geometry in it.

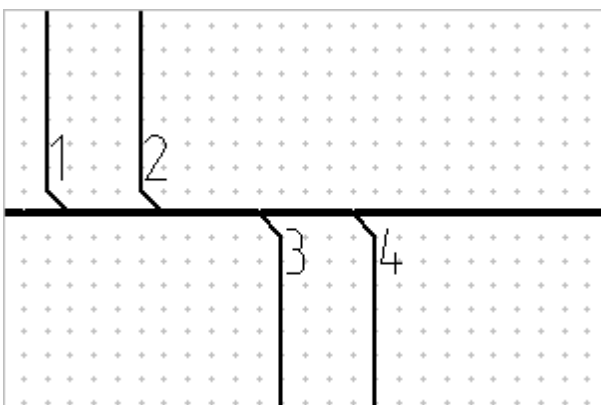


Show Number of Cores option is available in **Additional Visual Parameters** section. If it is checked, special "notches" will be applied to the group CL. They show how many regular CLs pass through the group CL section. A section here is a segment of a group CL between two branches or between a branch and a GCL endpoint.





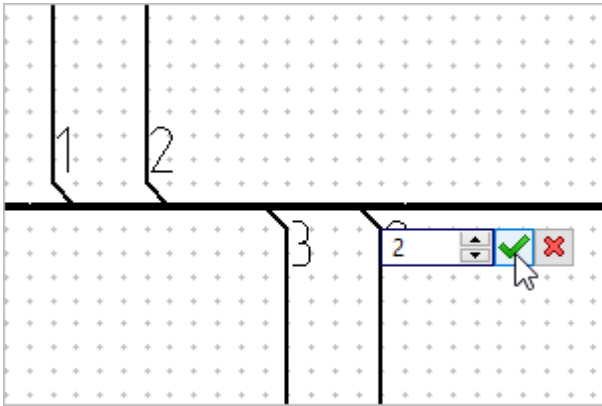
If GCL is directly connected to component, the system will assume that all the CLs included in it are connected to this component. In accordance with this, "notches" will be placed (see image above).


Each regular CL connected to the group one is numbered, getting its position "inside" of GCL.

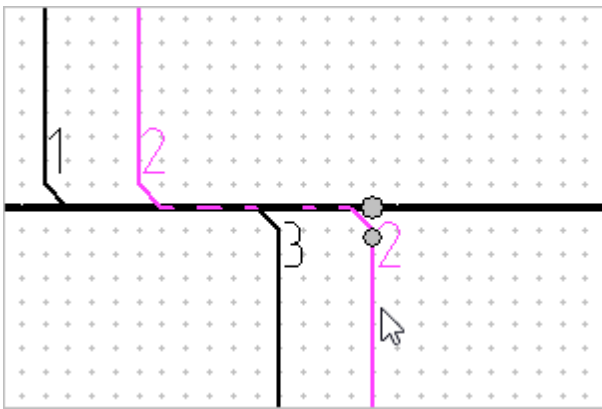


If several regular CL that are part of an GCL are one leader or have a common potential, you must

assign them the same "entry" number. To do this,  on the position number of CL, enter the required value in the field and confirm the action with .



In this example, connection lines 2 and 4 are combined into one (2), that is visible when you point  at it.




You can hide the position numbers of CLs using **Show Numbers of Connection Lines** option in **Additional Visual Parameters** section.

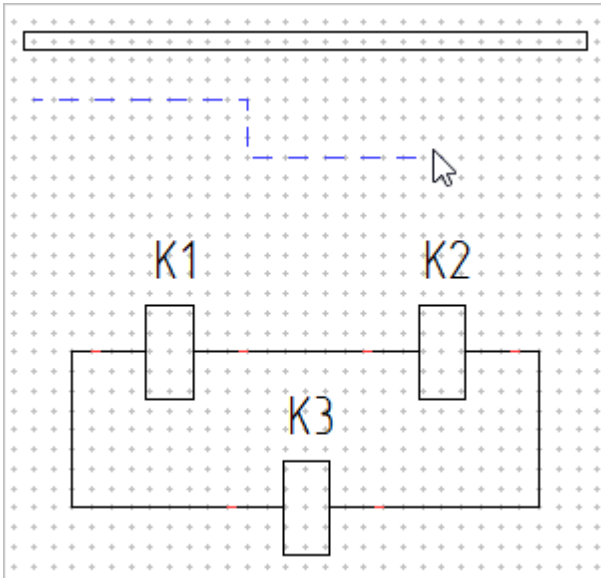
You can add shield, coaxiality and twisted-pair symbols to the GCL, as well as delete them in **Parameters** section. For more information, see the chapter "Designations on Connection Lines".

"Bus Bar" command

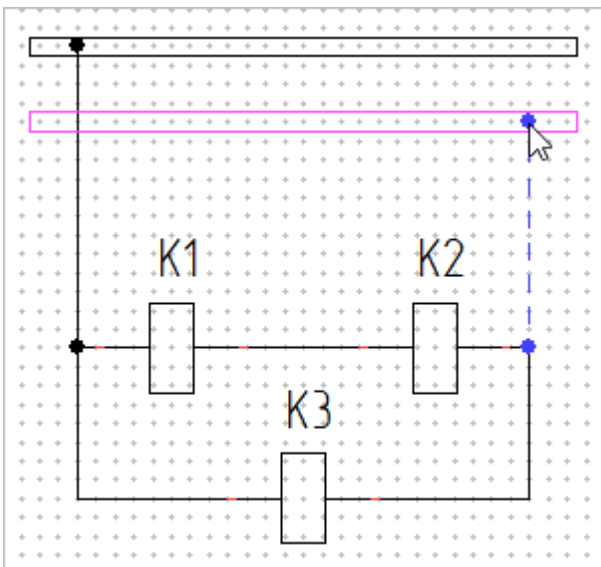
To draw a bus bar, use the command:

Icon	Ribbon
	Electrical ? Diagram ? Bus Bar
Keyboard	Textual menu
BB	Electrical > Constructions > Bus Bar

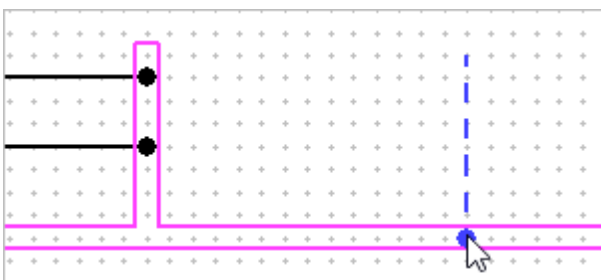
The construction of the bus bar is the same as the construction of the connection line. The only difference is that end points of bus bar must be located at the free points of the diagram.



To connect a diagram components to the bus bar, create a connection line between its connection point (or point on the CL) and the point inside the bus border.




A bus bar, as well as a connection line, may have a branch. To add it, build another bus (future branch) and connect it to the existing one.



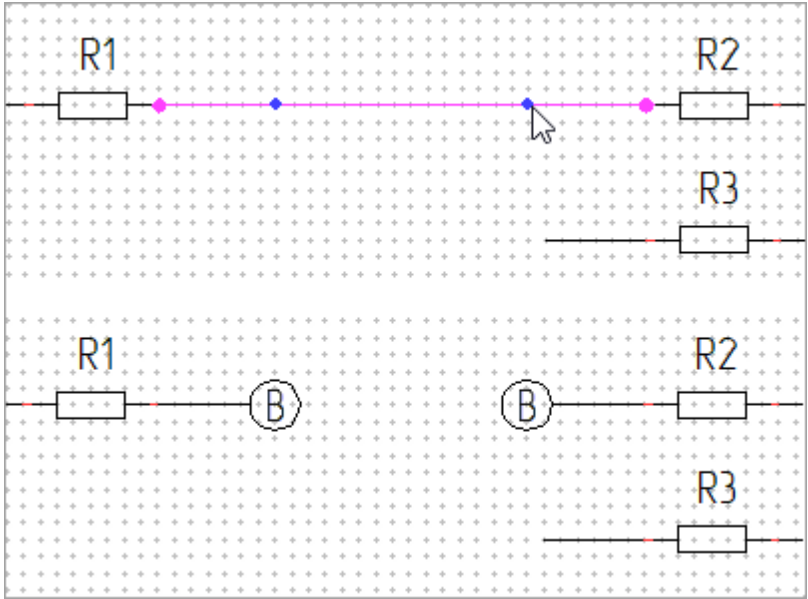
"Break Connection Line" command

Use the following command to create the break on connection line:

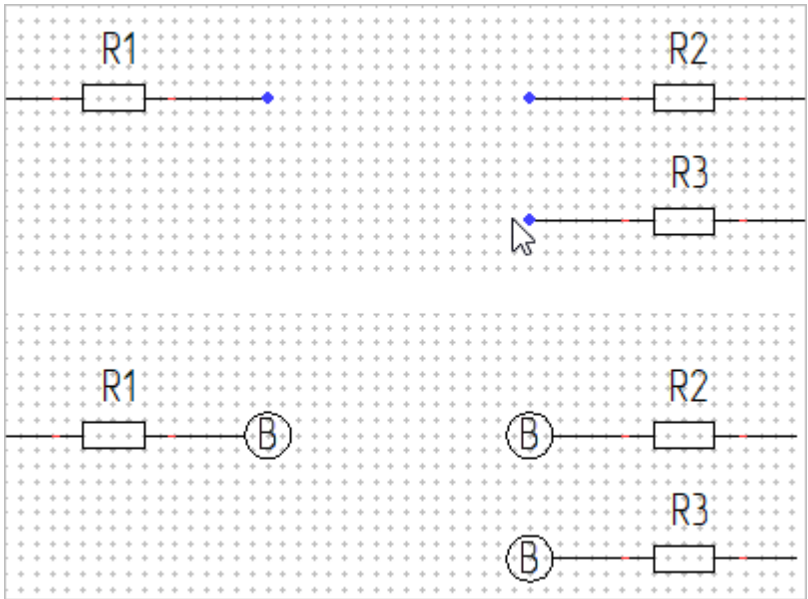
Icon	Ribbon
	Electrical ? Diagram ? Break Connection Line
Keyboard	Textual menu
QE	Electrical > Constructions > Break Connection Line

You must specify the points between which a break will be created after running the command.

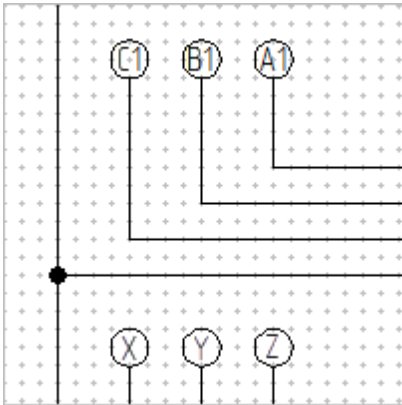
Points can be located on the same connection line. In this case, the connection between the components will be kept after creating a break.





Break points can be also created at the ends of different connection lines, while the CLs can be located on different pages. In this case, a connection will be created between the components closest to the end points.

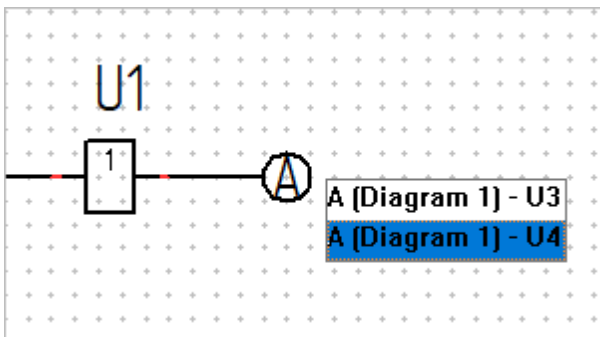



Break designations are named alphabetically. If all letters of the alphabet are used, the gaps will be called A1 ... Z1, then A2 ... Z2, etc.




Break designation can be changed manually. To do this, select **Edit Break Designation** in the context menu. A window will appear, you can enter any text in it.

To quickly find the mate of the break, point  at its designation. A marker  will appear, it opens the list of mate break points. The designation of mate point, the name of the page on which it is located and the component closest to it are indicated.



 on one of the points will open a page with it, while the point will be displayed large in the center of the screen.



There will be more than one variant in the list if more than 2 points were selected when creating the break.

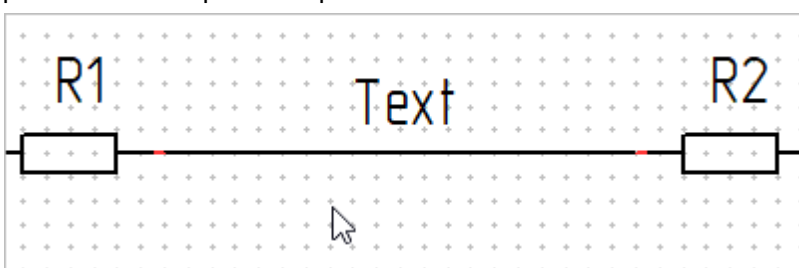
To remove a connection line break,  on its designation and select **Delete Break** in the context menu. In this case, the connection between components previously connected through the CL with a break will be also deleted.

Designations on connection lines

Text

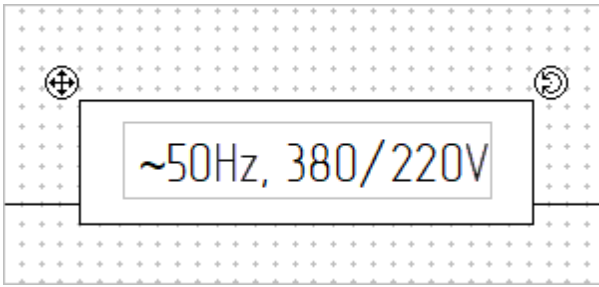
To add text to a connection line, use **Add Text** command in its context menu.

When the command is called up, an inscription appears on the connection line, it follows .  will place it in the specified place.

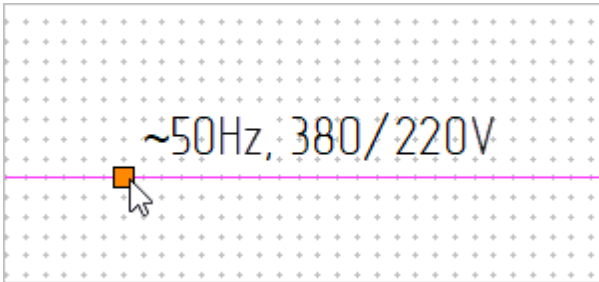



The inscription contains the word text by default. Inscription editing is performed in the same way as

regular texts (TE: Text).




A special marker is used to move the text, it appears when you point at connection line.

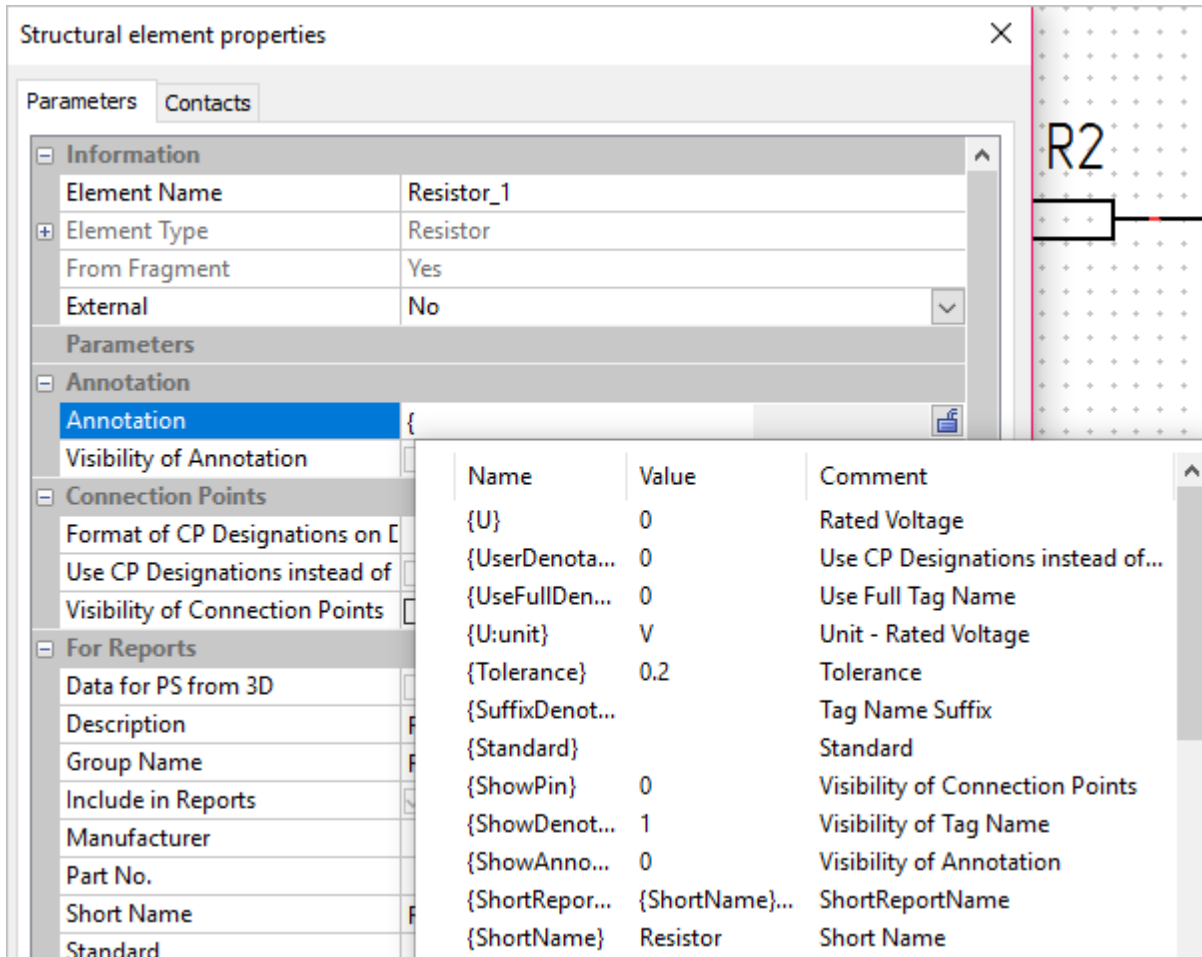


It is necessary to  on this marker and select **Delete Selected Text** in the context menu to delete text from a connection line. If you want to delete all texts from the connection line, select **Delete All Texts** in its context menu.

If a cable component is assigned to the connection line (see "Cable Components in Diagram"), then parameter values of cable component can be written on it.

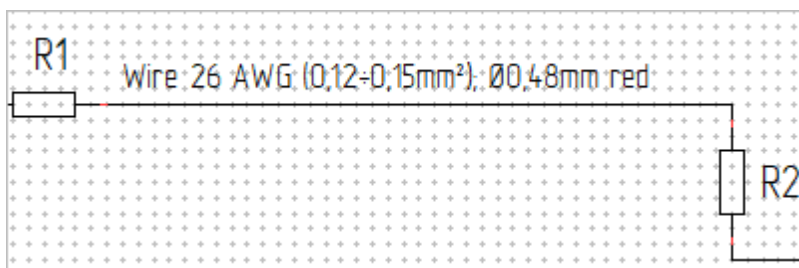
To do this, find this cable component in the list of structural elements and run  **Parameters** command from its context menu.

Then enter { symbol in an empty field with the value of **Annotation** parameter. A drop-down list with synonyms of the parameters will open. The length and width of the columns in this list can be changed for convenience. Select the parameter in **Comment** column that should be indicated on the CL. Its synonym is entered in the field. The text that is added to the CL will be displayed in the gray block next to the input field. The width of the gray block can be also changed manually.





It is possible to enter any combination of parameter synonyms and arbitrary text into the field of **Annotation** parameter.

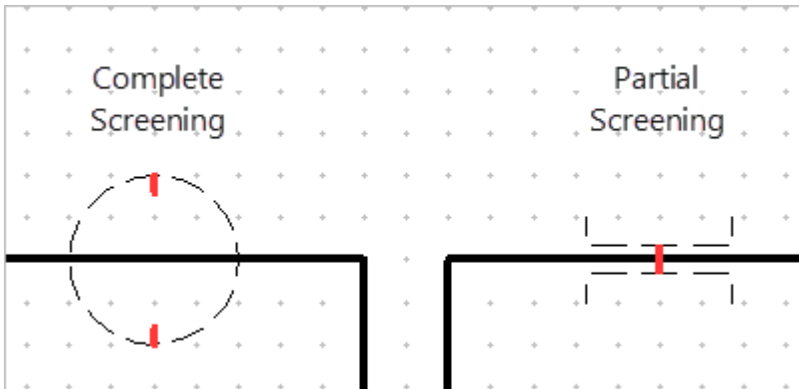
You need to check **Visibility of Annotation** parameter to get the annotation displayed on the CL. Changes should be saved with [OK].







Screening and coaxiality

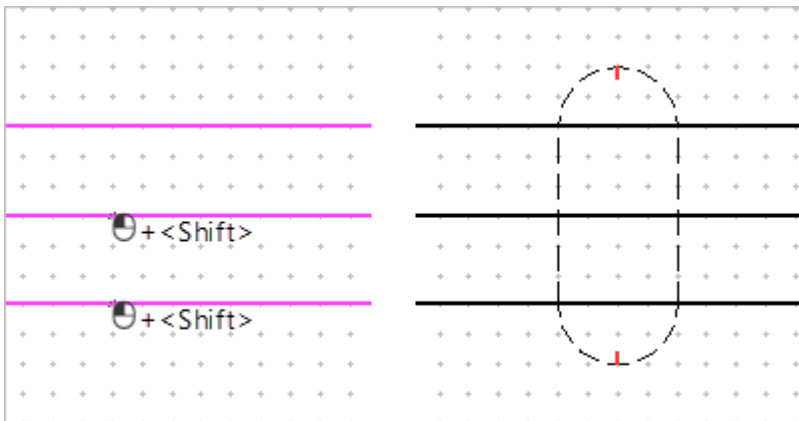
To add a screening or coaxiality symbols, select the appropriate command in the context menu of the connection line. The symbol will follow .  will place it in the specified place.

When adding screening, you can check **Partial** in the dialog to change the symbol:




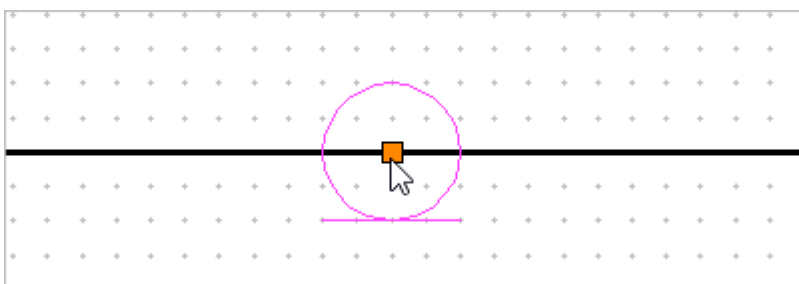
When coaxiality symbol is added to a screened connection line, all symbols are replaced by combined ones . **Autosubstitution of Designations** parameter in  **Document Parameters** (**Lines** section, **Connection Lines** group) allows to turn off autochange of symbols.

If you need to apply one screening symbol to several connection lines, enable  **S: Connection Lines Selection** automenu option in **Add Screening** command, then select  additional connection lines while holding <Shift>.




The second way is to select several connection lines with a frame from right to left and select **Add Screening** command in the context menu. The result will be the same as in the image above.

To move the symbol,  on a special marker in its center!. The symbol will follow the cursor.

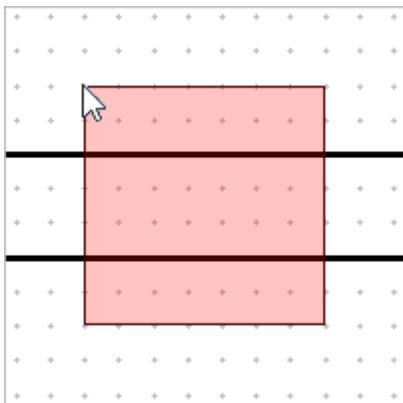


To remove screening and coaxiality symbols, use **Delete** option, that is available in the context menu of the central marker.


The coaxiality and screening symbol sizes are configured in  **ST: Document Parameters** command (**Lines** section, **Connection Lines** group).

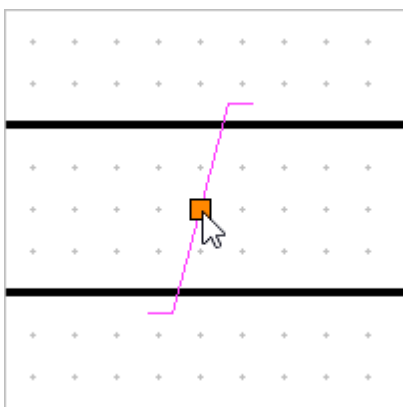
Twisted pair

To add a twisted pair symbol, select two connection lines with a frame (from right to left).



Then select **Add Twisted Pair** option in the context menu. The option is available in the context menu only when selecting two connection lines.

It is necessary to  on a special marker in the center of the symbol to move it. The symbol will follow the cursor.



To remove twisted pair symbol, use **Delete** option, which is available in the context menu of the central marker.

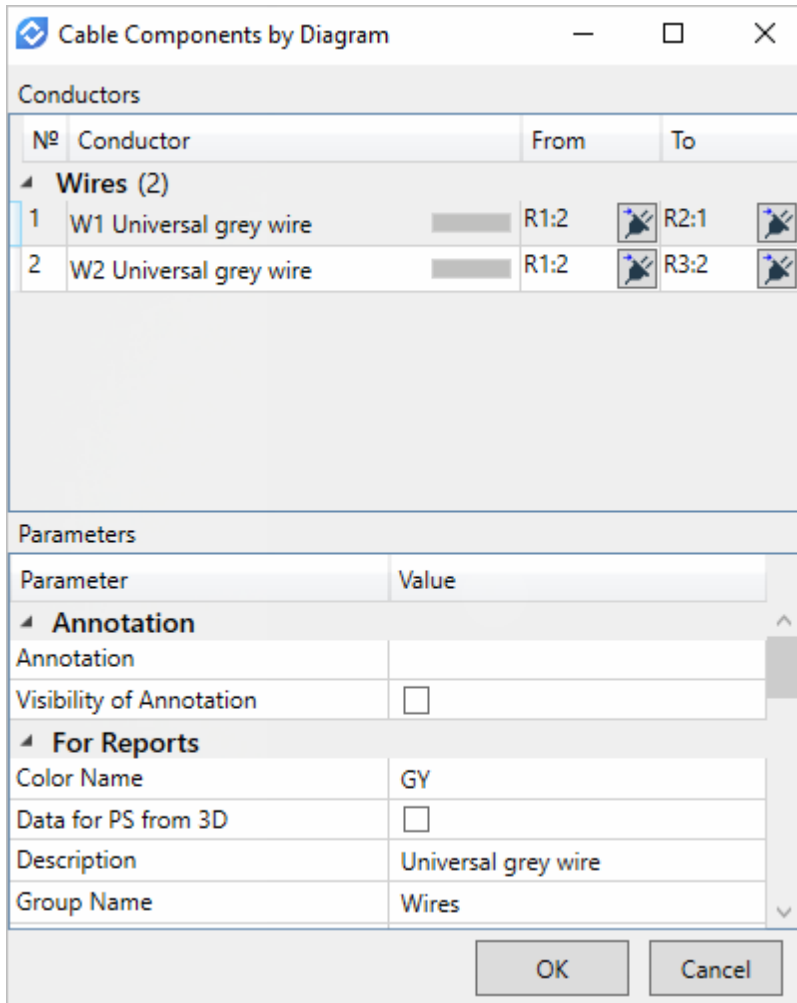
Cable components in diagram

Wire

To identify that component contacts are connected through a wire, call the context menu of connection line and select **Cable Component** -  **Wire**.

A wire can only be assigned to a regular CL; this command is not available for a group CL.

After starting the command, a dialog with a list of conductors appears. The number of positions in the list is determined automatically; if a CL with branches was selected, then there will be more than one conductor in the list.



The system assigns a universal grey wire by default, but you can select other types of wires. To do this, click on the name of the wire in the dialog and select the type from the drop-down list. The wire set in the list is inherited from **Component Editor**.

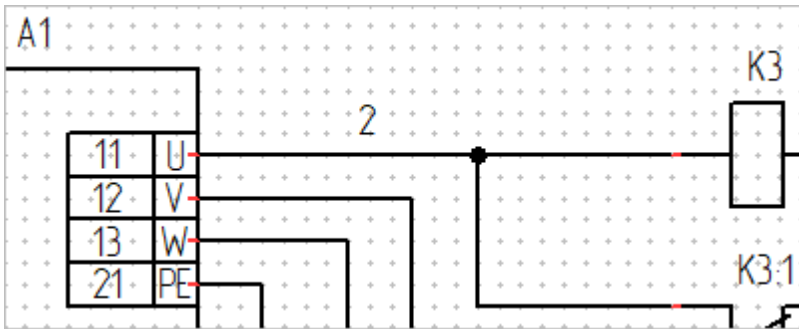
If the required type of wire is not in the drop-down list, you can create it by yourself. See the chapter "Step 1: Creating a Prototype - Wire" for more on this.

From and **To** columns are filled automatically. To configure addressing manually, open the drop-down list in the cell with the contact number and select the required contact.

Terminal ends can be installed on the contacts. To do this, use the icon that opens the list of available ends. The installation of terminal ends on connector pins is locked.

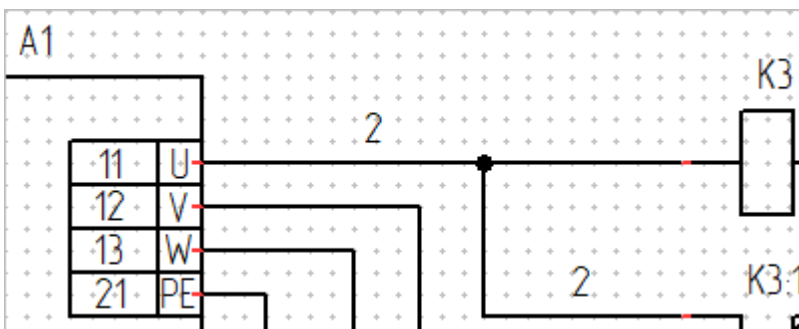
Parameters block of the dialog shows the parameters of the selected wire. The default values are inherited from **Component Editor**.

After pressing [OK], the tag name of the wire (its serial number in the diagram) is displayed on the connection line.



✎ **Wire group** will appear in **Structural Elements** folder of the model tree. All wires, including those included in the cables and harnesses, will be added there. The assigned wire will be also added to **Cable Components** folder.

Sometimes you need to duplicate the tag name of the wire on the connection line. To do this, use **Add Tag Name of Cable Component** command in CL context menu.




To make changes in the assigned wire, use  **Edit Wire** command in the context menu of connection line.

The values of the CC parameters (for example, name, color, etc.) can be written on the connection line. For more information, see "Text" chapter.

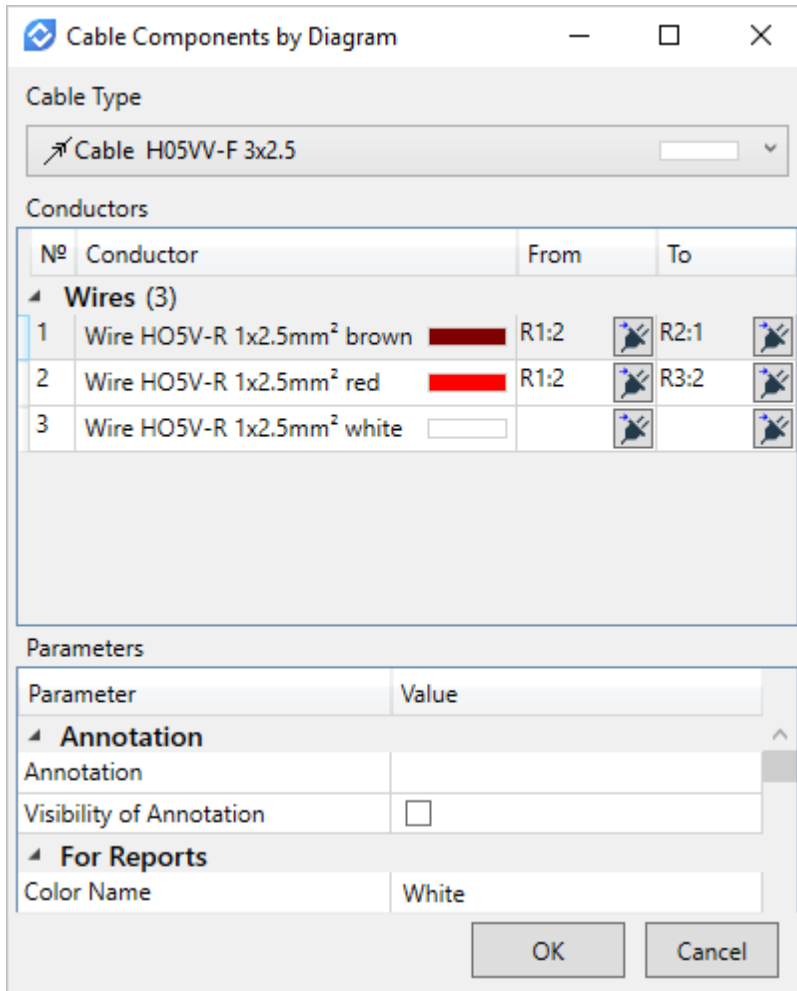
To remove the correspondence between CL and the wire, find the required wire in **Cable Components** folder and run **Delete** command in its context menu.


Cable

To identify that component contacts are connected through a cable, call the context menu of connection line and select **Cable Component** -  **Cable**.

A cable can be assigned to a regular CL and group CL.


After starting the command, a dialog with an empty list of conductors appears. To fill it, select **Cable Type** from the drop-down list at the top of the dialog.





The number and types of conductors in the cable are inherited from the prototype created in  **Component Editor**. It is not possible to change them in the cable assignment dialog.

If the required type of cable is not in the drop-down list, you can create it by yourself. See the chapter "Step 1: Creating a Prototype - Wire" for more on this.

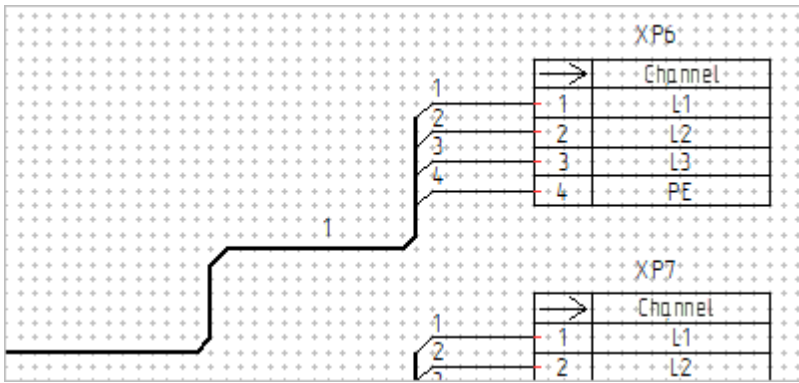
From and **To** columns are filled automatically. To configure addressing manually, open the drop-down list in the cell with the contact number and select the required contact.

If you want to swap "From" and "To" within the same pair of contacts, change one of them manually, and then  on any other wire in the list. The second contact will be recorded in the response cell.

Terminal ends can be installed on the contacts. To do this, use the icon  that opens the list of available ends. The installation of terminal ends on connector pins is locked.

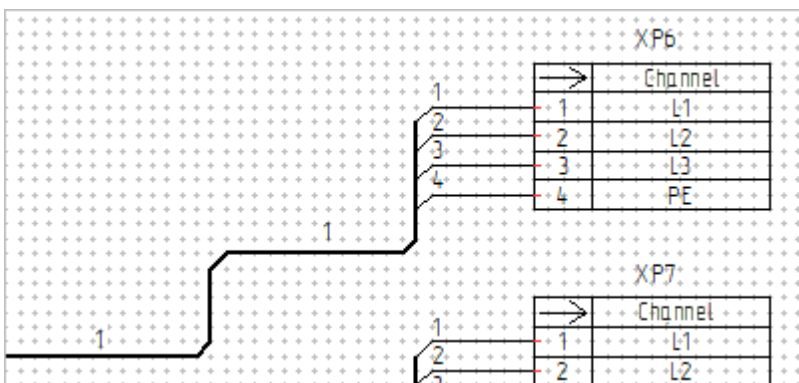
Parameters block of the dialog shows the parameters of the cable. The default values are inherited from  **Component Editor**.

After pressing [OK], the tag name of the cable (its serial number in the diagram) is displayed on the connection line.



✈ **Cable** and ✈ **Wire** groups will appear in **Structural Elements** folder of the model tree. The cables and included wires will be added there. The assigned cable will be also added to **Cable Components** folder.

Add Tag Name of Cable Component command is also available for the cable (for details, see "Wire" chapter).

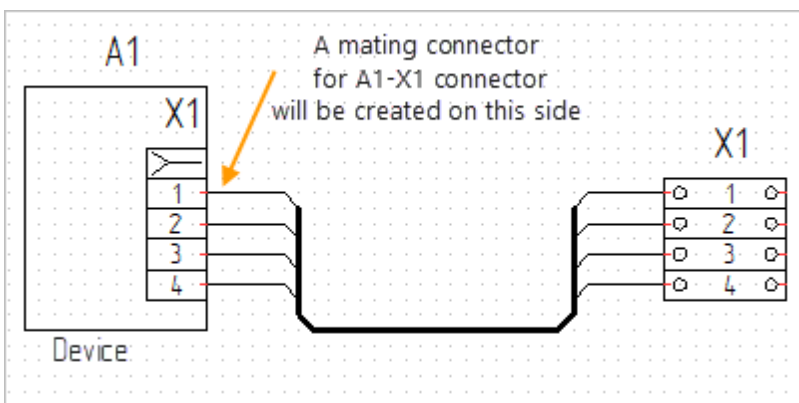



To make changes in the cable, use  **Edit Cable** command in the context menu of connection line.

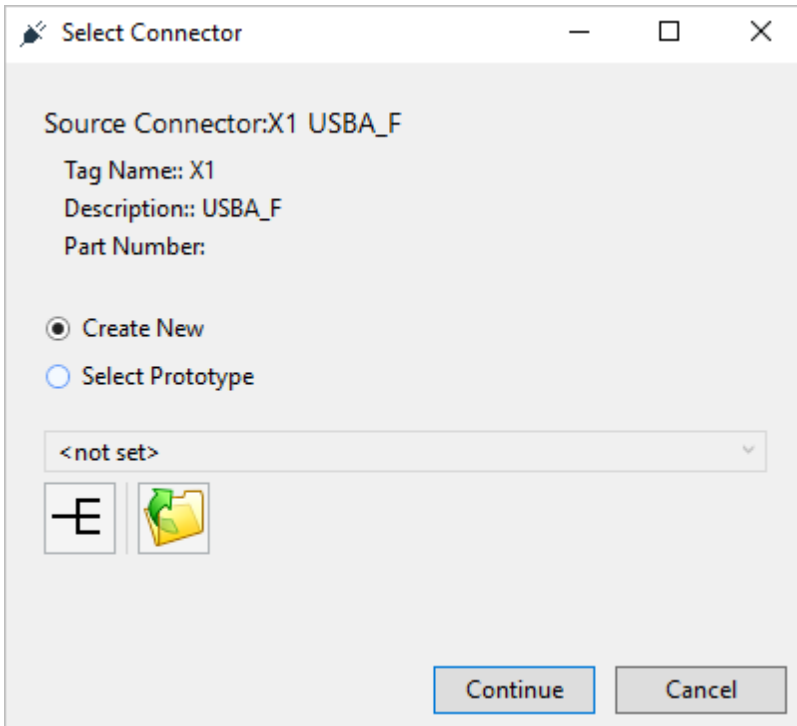
The values of the CC parameters (for example, name, color, etc.) can be written on the connection line. For more information, see "Text" chapter.

To remove the correspondence between CL and the cable, find the required cable in **Cable Components** folder and run **Delete** command in its context menu.

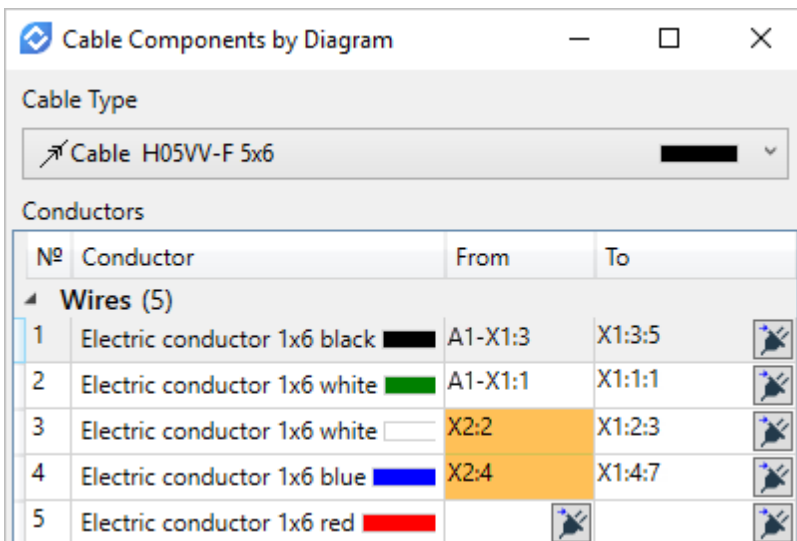
If CL is connected to the connector included in device, the system will offer to create a mating connector on the side of the cable when assigning a cable to this CL.



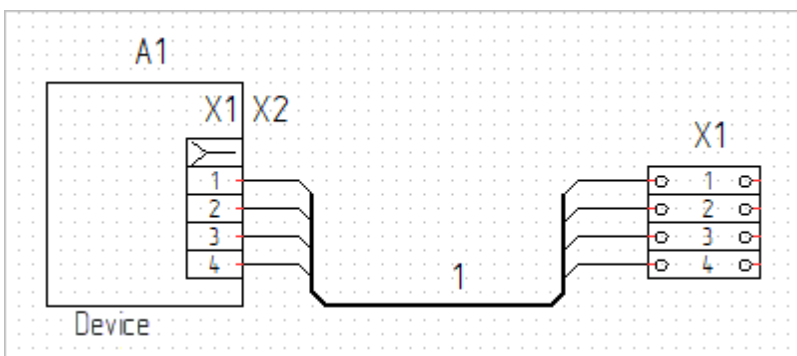
A dialog for adding a mating connector appears immediately after selecting **Cable Type**. It proposes to create a new connector with a custom set of parameters (**Create New**) or select one of connectors prototypes previously created in  **Component Editor (Select Prototype)**, and then click [Continue].



After that, a window with parameters of the connector opens. The values of parameters can be changed if necessary. [Done] will return to the dialog with the structure of cable. **From** and **To** columns will be filled with internal addresses by default (for more information about them, see "Wires Table" chapter). You can change them to external ones manually. The internal addresses are highlighted in the illustration below.



After the assignment of cable, the tag name of the new connector will appear on the diagram.



The structural element of new connector will be added to \ominus Connectors group, $\omin�$ Electromechanical Connection SE will be also created (see **Structural Elements** folder in 3D model tree).

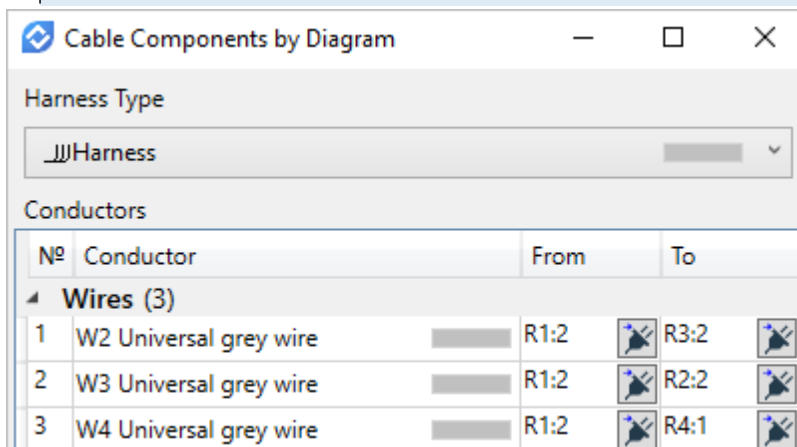
Harness

To identify that component contacts are connected through a harness, call the context menu of connection line and select **Cable Component -  Harness**.



A harness can be assigned to a regular CL and group CL.

After starting the command, a dialog with an empty list of conductors appears. To fill it, select **Harness Type** from the drop-down list at the top of the dialog.


If the required type of harness is not in the drop-down list, you can create it by yourself. See the chapter "Step 1: Creating a Prototype - Wire" for more on this.





The number of wires in the harness is determined automatically.

The system assigns a universal gray wire by default, but you can select other types of wires. To do this,  on the name of the wire in the dialog and select the type from the drop-down list. The wire set in the list is inherited from  **Component Editor**.

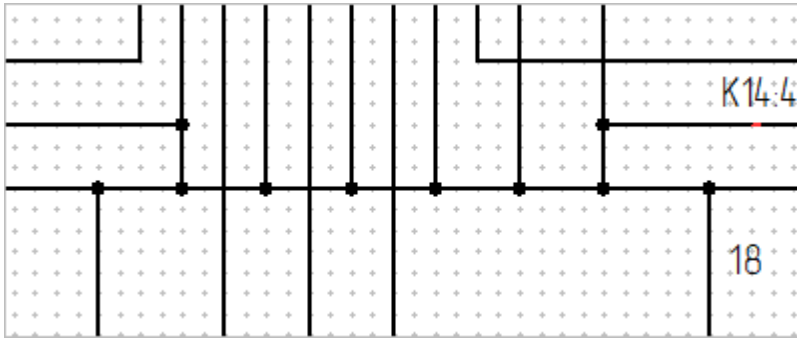
From and **To** columns are filled automatically. To configure addressing manually, open the drop-down list in the cell with the contact number and select the required contact.

If you want to swap "From" and "To" within the same pair of contacts, change one of them manually, and then  on any other wire in the list. The second contact will be recorded in the response cell.

Terminal ends can be installed on the contacts. To do this, use the icon  that opens the list of available ends. The installation of terminal ends on connector pins is locked.

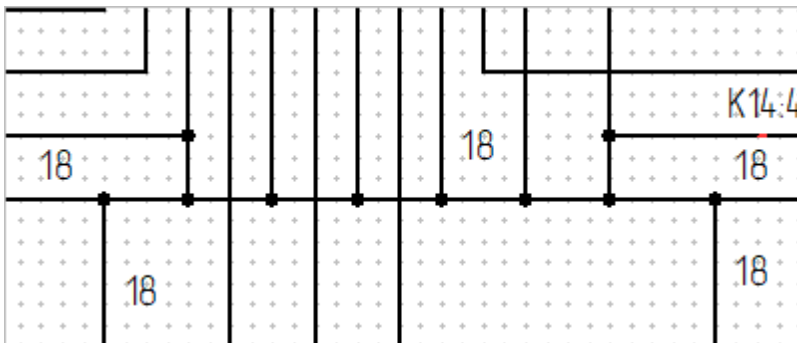
Parameters block of the dialog shows the parameters of the harness. The default values are inherited from  **Component Editor**.


After pressing [OK], the tag name of the harness (its serial number in the diagram) is displayed on the connection line.



↗ **Harness** and ↗ **Wire** groups will appear in **Structural Elements** folder of the model tree. The harnesses and included wires will be added there. The assigned harness will be also added to **Cable Components** folder.

Add Tag Name of Cable Component command is also available for the harness (for details, see "Wire" chapter).




To make changes in the cable, use  **Edit Harness** command in the context menu of connection line. The values of the CC parameters (for example, name, color, etc.) can be written on the connection line. For more information, see "Text" chapter.

To remove the correspondence between CL and the harness, find the required harness in **Cable Components** folder and run **Delete** command in its context menu.

If CL is connected to the connector included in device, the system will offer to create a mating connector on the side of the harness when assigning a harness to this CL. For details, see "Cable" chapter.

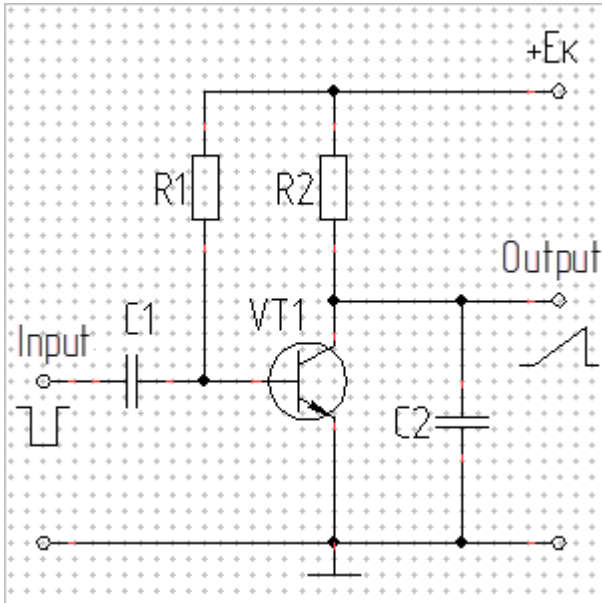
Symbols and other designations

Some commands of **Draw** tab of the Ribbon panel are available on the Diagram page. You can place a text, a table, perform a sketch, etc. on the diagram. They will be added to the page with the diagram as graphic objects, not as elements of the diagram. In some cases, it's convenient to turn off

snap to the grid using  **QG: Grid** on **View** panel when using these tools on a page with a diagram.

It should be considered that simultaneous moving the elements of the diagram and drawing elements is not possible. Drawing elements can be moved only separately from diagram elements. In this regard, it is recommended to add drawing elements to the page with drawn-up diagram, and not to the Diagram page. In this case, all the commands of **Draw** tab will be available on the regular page.

Symbols designating electrical pulse can be added to the diagram. To do this, use the command **Insert Component - Symbol** or drag'n'drop elements from the library "Electrical Diagram Elements - Pulses".




The symbols are located at the "Input" and "Output" terminals in the image above.

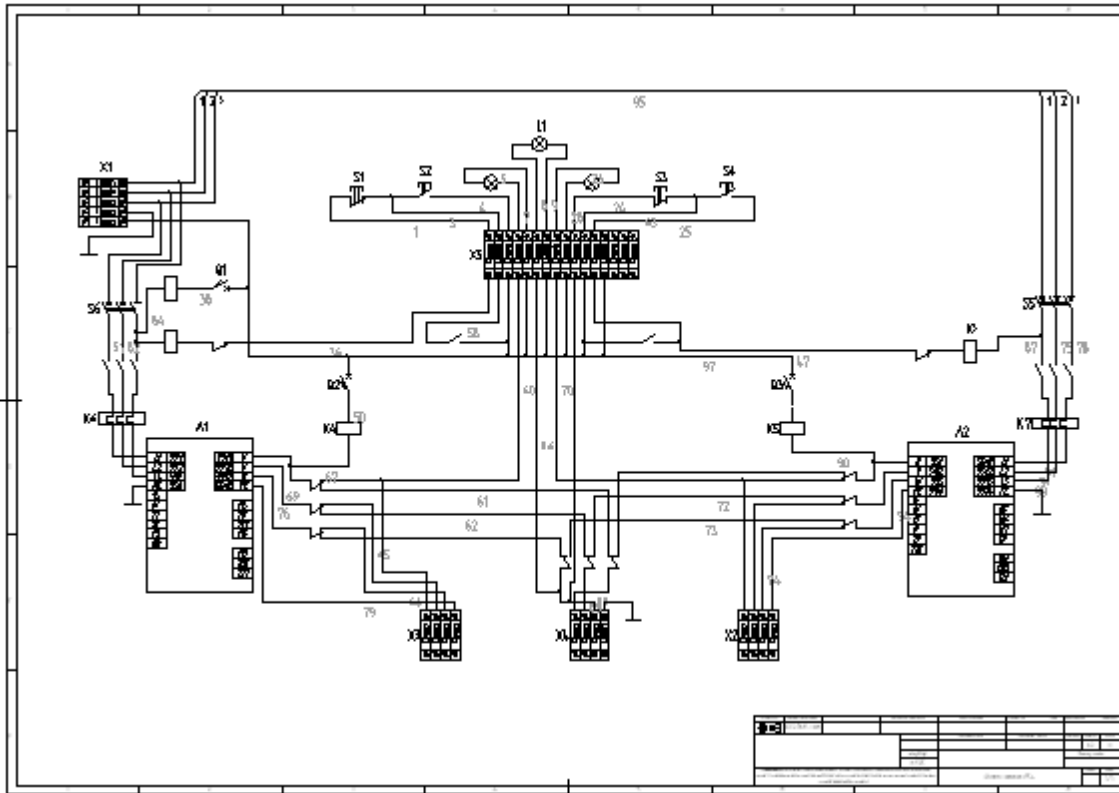
Symbols are elements of a diagram, but they are not represented in reports; moving restrictions (like to drawing elements) do not apply to them.

Insert Title Block command





Use following command to insert title block:

Icon	Ribbon
	Electrical ? Diagram ? Insert Title Block
Keyboard	Textual Menu
	-

The command duplicates the diagram on a regular page, suggesting to choose a template of title block from the list before that. After selecting a template, a dialog opens to fill in the title block and additional columns. Pressing on [OK] will open the regular page with processed diagram.



After executing **Insert Title Block** command, you can change the diagram can be made both on a regular page and on the "Diagram" one. The changes will be synchronized between pages automatically.

By default, the Electrical tab commands cannot be started on a page with processed diagram. To enter editing mode and run these commands, it is necessary to  on any diagram element on a regular page and select  **Activate Drawing View** in context menu or in dynamic panel. When the editing is finished, it is necessary to  again on any diagram element and select  **Close Drawing View** in the context menu.

After executing **Insert Title Block** command, a link between Diagram page and a regular one is created. As long as the link exists, the command will not be available on the Diagram page (since it's not possible to title block to one page twice).

If you want to replace the title block template, you can use one of two methods. The first one is to delete a regular page with processed diagram. The link between the pages will be also deleted, and **Insert Title Block** command will be available again. The second way is to replace the title block template in "Assembly Structure" window.

If a document contains several pages of diagram type and each of them must be processed, run **Insert Title Block** command on each of them, selecting the corresponding templates of title blocks.

2D reports

List of elements

An "Electrical Elements" product structure is created in each new "Diagram" document by default. To open **Product Structure** window with a completed list of elements, use the command:

Icon	Ribbon
	Electrical ? Data for PS ? Data for List of Elements
Keyboard	Textual Menu
-	-

If **Product Structure** window is open, select "Electrical Elements" PS type and click **Update**.

The list shows all electrical components, with the exception of cable ones and their complementary parts.

Product Structure			
Electrical elements 1			
Show Linked Object	Tag Name	Description	Quantity
A A1 Device	A1	Device	1
A A2 Device	A2	Device	1
-o X4 Clamp	X4	Clamp	1
-o X3 Clamp	X3	Clamp	1
-o X2 Clamp	X2	Clamp	1
✓- S6 Three poles disconne...	S6	Three poles disconne...	1
-o X12 Clamp	X12	Clamp	1
-o X6 Clamp	X6	Clamp	1

Select a component in the list and run **Enlarge** option in the context menu to find it in the diagram from the list of elements. A page with the selected component will automatically open, while it will be displayed large in the center of the screen.

Use **Representation** parameter to group components in **Product Structure** window. If selected **Diagram**, the components will be grouped by the types of diagrams that were assigned to them. If selected **Elements**, the components will be grouped into functional groups (resistors, diodes, etc.).

Run **Create Report based on Product Structure** to create "List of elements" report. Two templates are available in the command dialog: for placement on a separate page and on the current one.

Click [Create] to complete the report creation. If the components in the document belong to different types of diagrams, the system will ask a diagram type you want to create a report for.

Wires table

To open **Product Structure** window with a completed wires table, use the command:

Icon	Ribbon
	Electrical ? Data for PS ? Data for Wires Table
Keyboard	Textual menu
	Electrical > Reports > Data for Wires Table

Component contacts are presented in pairs in wires table. Connections between them are created by means of communication lines.

If nothing is connected to the component (that is, none of its contacts is connected to the contacts of other components), then this component will not be included in the connection table.

Product Structure

List of Wires 2 By wire owner

Show Li...	Wire Description	Wire	Wire Length	From Connectio...	To Connectio
-> C. ☰	Wire HO5V-R 1x2.5mm ² blue	* W2	☰ 0.564	m * R1:2	* R2:2
-> C. ☰	Universal grey wire	* W3	☰ 0.7402	m * R1:2	* R4:1
-> C. ☰	Wire 26 AWG (0,12±0,15mm ²); ...	* W1	☰ 0.3	m * R1:2	* R3:2

The connected pins of components are shown in the columns **From Connection Point** and **To Connection Point**.

"Connection" structural elements created after connecting the CL to the pins are shown in **Linked Object** column.

Wire Description column will be filled if cable components are assigned to the CLs.

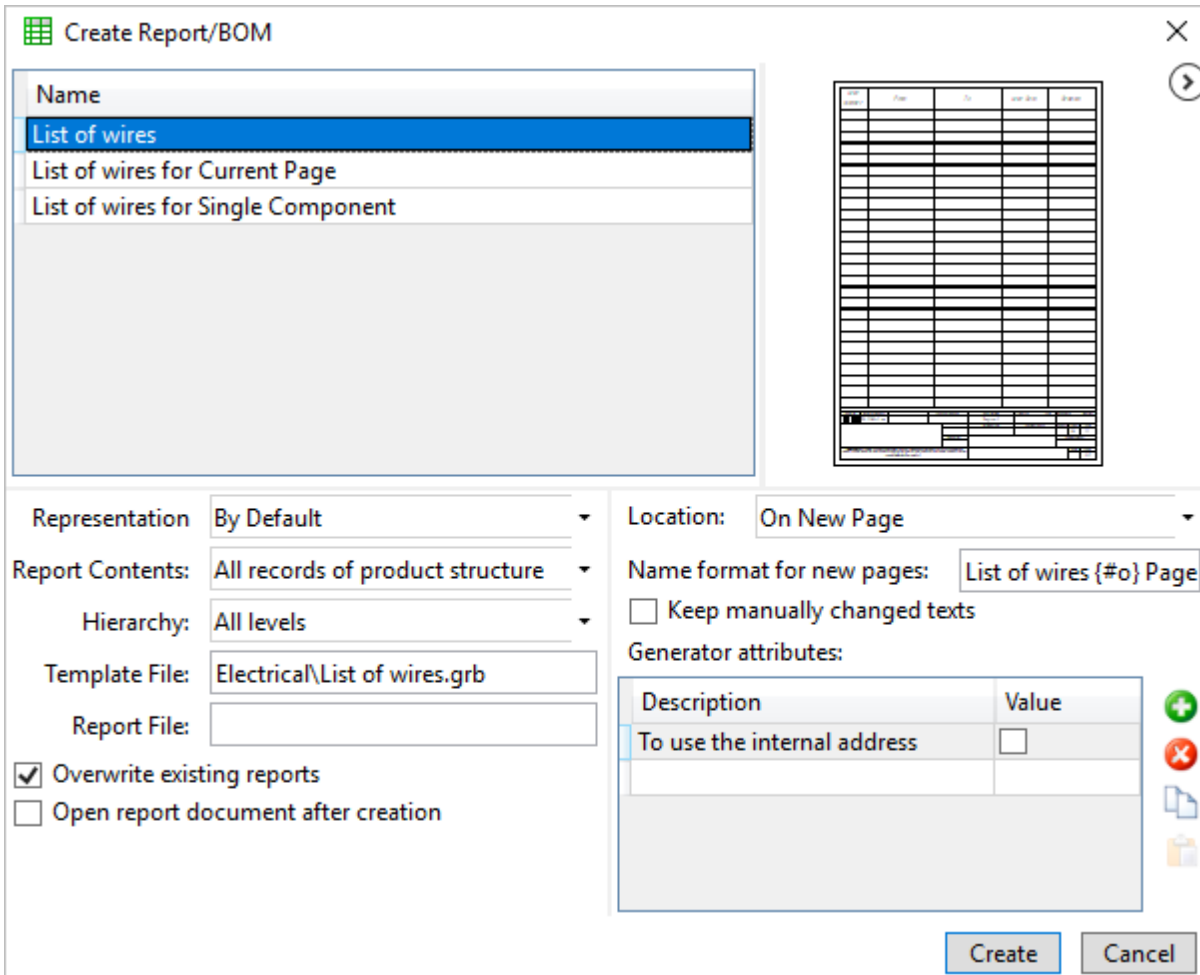
Wire Length column is filled with default values. The exact values of wires length will be obtained after associating the diagram with the 3D model (for more details see the chapter "Reports in 3D assembly").

The columns **Internal From CP** and **Internal To CP** are also added to the Wires Table. They display the connectors pins "externally" connected to the device connectors. See illustration below (the symbols of "external" connectors are not displayed, there are only their tag names on the diagram).

Connection Point	Internal From CP	Internal To CP	To Connection Point
* A1-X1:1	* X1:1	* X2:1	* A2-X1:1
* A1-X1:2	* X1:2	* X2:2	* A2-X1:2
* A1-X1:3	* X1:3	* X2:3	* A2-X1:3
* A1-X1:4	* X1:4	* X2:4	* A2-X1:4

Use **Representation** parameter to group wires in **Product Structure** window. If **By Wire Owner** selected, the cable components will be grouped by type: first, all the harnesses with the CC included in them will be listed, then all the cables with the wires included in them, then the individual wires. If **By Diagram** selected, the wires will be grouped by diagram type. The diagram type of CC is determined by the components that are switched through this CC.

To create "Connection Table" report, use **Create Report based on Product Structure** command. Several connection table templates are available in the command dialog.



Representation parameter controls the grouping in reports in the same way as in Product structure window (see the description above). The difference is revealed when choosing **Diagram** representation: only wires will be indicated in the report; cables and harnesses that they are included into will not be shown.


Each connection table template can have its own set of **Generator Attributes**, they allow to additionally configure reports.

To Use the Internal Address. The data from columns **Internal From CP** and **Internal To CP** will be entered into the columns **Origin** and **Destination** of the report.

Click [Create] to complete the report creation. If the components in the document belong to different types of diagrams, the system will ask a diagram type you want to create a report for.

Connection table


To open **Product Structure** window with a completed connection table, use the command:

Icon	Ribbon
	Electrical ? Data for PS ? Data for Connection Table
Keyboard	Textual menu
	Electrical > Reports > Data for Connection Table


The connection table shows all components added to the diagram. Tag names of components are specified in **Owner** column.

Each row of the connection table contains 2 component contacts (columns **Output 1** and **Output 2**). Tag names of cable components connected to the corresponding contacts are displayed in columns **Wire 1** and **Wire 2**.

Contact Type column is filled in only for electrical items of "Contact" type, the SE parameter "Type" is displayed in it.


To create a connection table report, use  **Create Report Based on Product Structure** command. Two templates are available in command dialog, they are intended for create reports by entire diagram.

To complete the report creation, click [Create]. If the components in the document belong to different types of diagrams, the system will ask a diagram type you want to create a report for.

To create a connection table for individual diagram component, use the context menu option  **Create Connection Table <component TN>**. Such a connection table will not be created if nothing is connected to the component.

Electrical BOM

An "Electrical BOM" product structure is created in each new "Diagram" document by default. To open **Product Structure** window with a completed electrical BOM, use the command:

Icon	Ribbon
	Electrical ? Data for PS ? Data for Electrical BOM
Keyboard	Textual Menu
-	-

If **Product Structure** window is open, select "Electrical BOM" PS type and click **Update**.


The BOM shows all electrical components, including cable ones.

To create an electrical BOM report, use  **Create Report Based on Product Structure** command. The same report templates are available in the command dialog as for PS "General BOM".

To complete the report creation, click [Create]. If the components in the document belong to different types of diagrams, the system will ask a diagram type you want to create a report for.

Diagram parameters



Use next command to configure tag names, designations of connection points and communication lines.

Icon	Ribbon
	Electrical ? Diagram ? Diagram Parameters
Keyboard	Textual menu
	Electrical > Tools > Diagram Parameters

The left block displays diagram types associated with electrical components in the document. Font settings are available for each diagram type (separately for tag names, designations of connection points and connection lines).


The right part displays the parameters of selected diagram. Parameters values may be different for different diagram types.

Parameter	Function
Tag Names of Elements section	

Visibility of Tag Names	Controls tag names displaying of all diagram components.
Visibility of Group Suffix	Controls the visibility of serial numbers in groups designations ( Splitted Representation command).
Group Separator	Sets the symbol separating tag name and serial number of the group in group designation ( Splitted Representation command).
Show Page Number	Adds page number to the tag names of all components in the diagram.
Connection Points Denotations section	
Visibility	Controls CP displaying of all components in the diagram.
Use CP Designations instead of System Values	Replaces system CP designations with custom ones.
Format of CP Designations on Diagram	The field is intended to enter a "rule" setting the designations to all CPs in the diagram.
Transversal Alignment	Sets the position of CP designations relative to CP themselves in the transverse direction (* - see below).
Longitudinal Offset	Specifies the offset of CP designations in the longitudinal direction *.
Transversal Offset	Specifies the offset of CP designations in the transverse direction *.
*: longitudinality and transverseness are determined relative to an imaginary line parallel to the X axis when the component is rotated by 0 degrees. It is assumed that when the diagram component is rotated, the imaginary line rotates with it and the transverse and longitudinal relative to this line are preserved.	
Designations on Connection Lines section	
Visibility	Controls displaying of designations on all connection lines of diagram.
Alignment	Sets the position of CL designations relative to CLs themselves.
Offset	Sets the distance between CL designations and CL themselves.

Check Diagram command

Use following command to check diagram:

Icon	Ribbon
	Electrical ? Measure ? Check Diagram
Keyboard	Textual menu
-	-

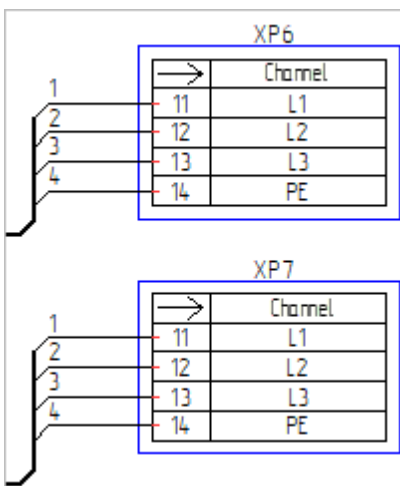
After its launch, a dialog opens. A list of available checks is presented there.


Check the test into the list to perform it. A list of diagram elements satisfying the verification condition will be displayed below, and the elements themselves will be highlighted on the diagram.

Elements without 2D Representation. The check finds components that contain only a 3D view and a structural element. For more information about such elements, see "Create 3D Component" command.

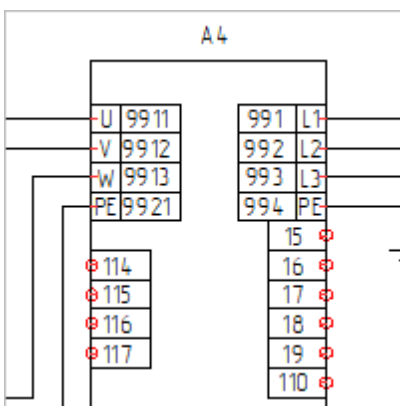
Elements without 3D Representation. The check finds the components of the diagram that are not associated with the 3D model and selects them with a frame on the diagram. For more details, see the description of "Component in 3D" command.

Connectors without Counterparts. The test finds connectors that are not connected to other connectors and selects them with a frame in the diagram.



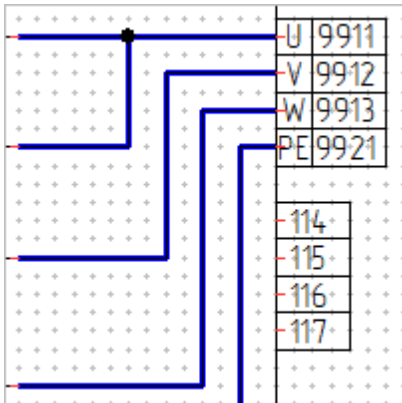
Library Elements with Changed Parameters. The check finds the components whose parameter values have been manually changed in the diagram (i.e. their parameter values differ from those specified in  **Component Editor**) and selects them with a frame in the diagram.

Vacant Connection Points. The check finds the connection points the communication lines are not connected to, and selects them with round markers in the diagram.



Vacant Ends of Connection Lines. The check finds the ends of connection lines that are not connected to anything, and selects them with round markers in the diagram.

Connection Lines Without Assigned Cable Components. The check finds connection lines for which cable components are not assigned, and marks them with color. Marking visibility depends on the scaling of the diagram page.



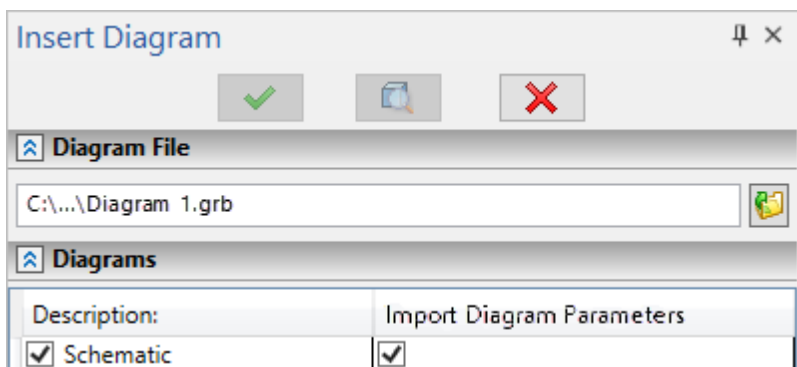
Wires Duplication in Connection. The test finds the conductors switching the same pairs of connection points, and selects these connection points with round markers in the diagram.


Insert Diagram command

Use following command to insert the diagram into different document:

Icon	Ribbon
	Electrical ? Diagram ? Insert Diagram
Keyboard	Textual menu
-	Electrical > Constructions > Insert Diagram


After running the command a dialog will open. It takes to select the diagram file in it and then check diagram types that must be inserted.



Checking **Import Diagram Parameters** allows you to insert a diagram into the target document saving the settings made in  **Diagram Parameters** command.

After confirming the insert, the diagram will be added to the document on a new page.


A diagram inserted from another file may be modified in the current file.

You can also update the inserted diagram by transferring changes from the original document to the current file. To do this, find the diagram type in the model tree ("Structural Elements" folder,  "Electric Diagram" group) and select **Update Diagram from Source** option in its context menu.


It is not recommended to update the diagram if important changes were made in it after

insertion into the document, as these changes will be lost.

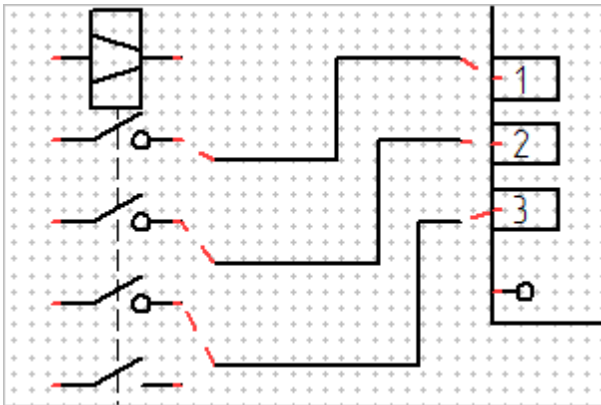
Remove Link to the Diagram option is available in the same context menu, it deletes the link between the original diagram and its copy in the current document.

A link to the source file with the diagram can be found in **Assembly Structure** window (<Alt + 7>).  **Group by Links** option must be active to display a diagram in it.


Update Diagram command

As a result of some actions, the connection lines in the diagram cannot be automatically regenerated. This is possible, in particular, when inserting a diagram from external file, when changing the grid step, or when splitting the component into parts using  **Splitted Representation** command.

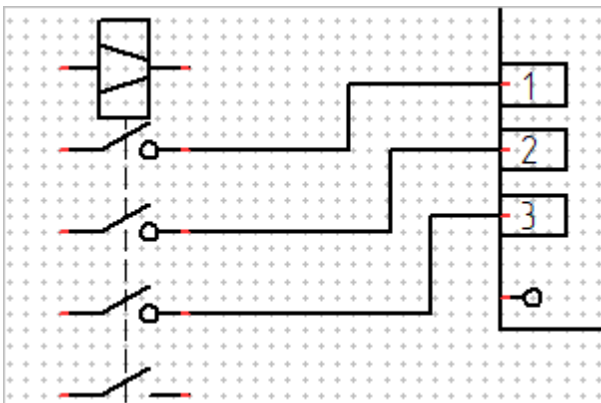
In this case, the connection lines are displayed in red dotted line.



Use the following command to restore connection lines:

Icon	Ribbon
	Electrical ? Diagram ? Update diagram
Keyboard	Textual menu
	Electrical > Tools > Update diagram

The restoring result will look like this:




Also, **Update Diagram** command can be useful in cases not related to connection lines, for example, for updating tag names of some components, etc.

Component Editor and parameters


General information about Component Editor

To open **Component Editor** window use the command

Icon	Ribbon
	Electrical ? Prototypes ? Component Editor
Keyboard	Textual Menu
	Electrical > Editors > Component Editor

Component Editor contains data on all components that are created in T-FLEX Electrical and can be used in the development of diagrams. The data is stored in the form of information models - prototypes.

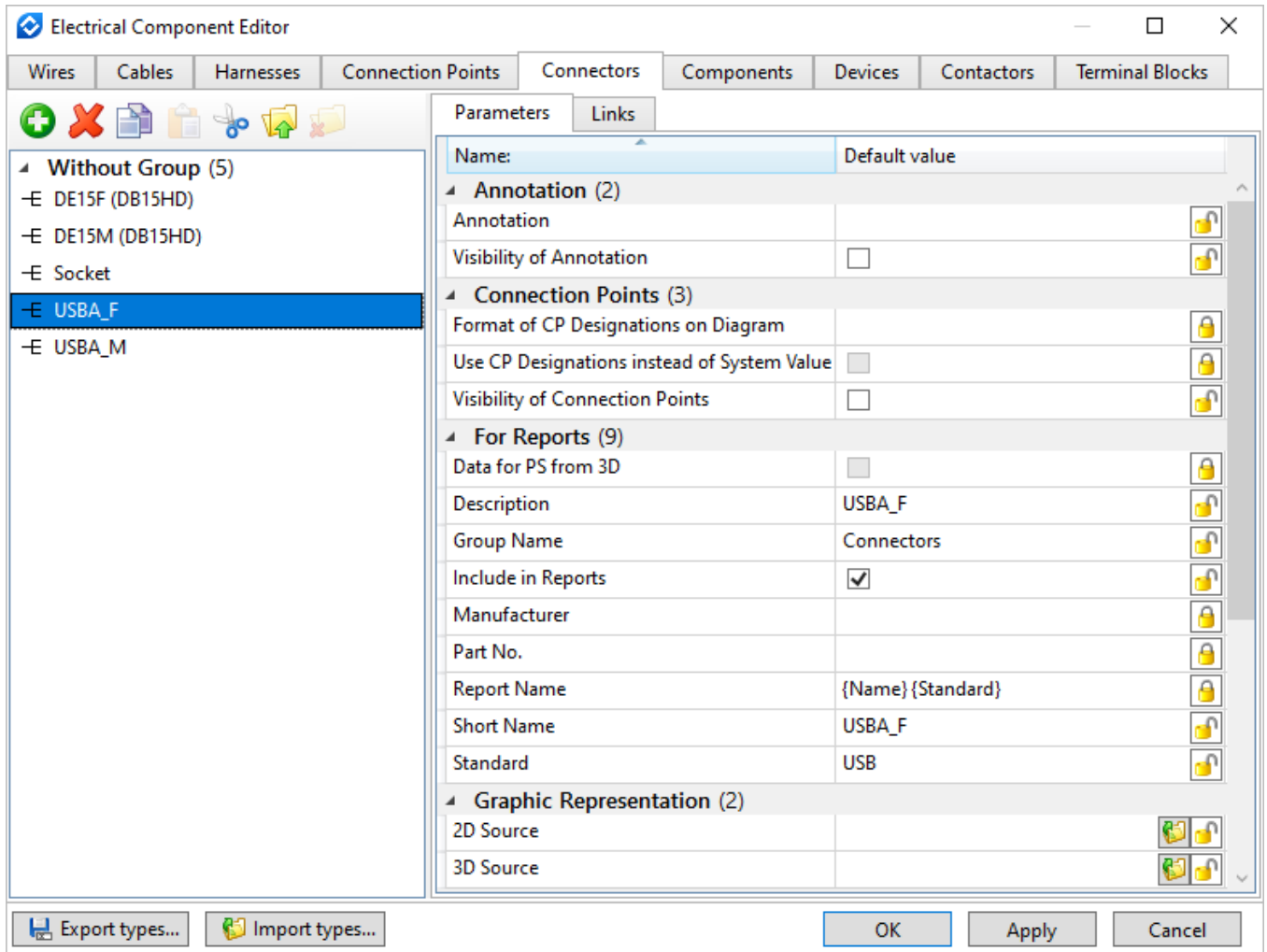
Each diagram element consists of a graphic symbol (image), a structural element (component information model), and connection points. The **Component Editor** is a "repository" of structural elements from which they are loaded into a specific diagram element.

All data from **Component Editor** is contained in electric.xml file. In the delivery state, this file is stored in Program Files\T-FLEX CAD 17 (16)\Program, the user version is stored in Users\{UserName}\AppData\Local\Top Systems\T-FLEX CAD 3D 17 (16)\Eng. To return **Component Editor** to the delivery state, use  **Default Metadata** command.


Components are divided into 9 types in **Component Editor**: **Basic Components** (in earlier versions - Electrical Items), **Devices**, **Connectors**, **Terminal Blocks**, **Contactors**, **Connection Points**, **Wires**, **Cables**, **Harnesses**. Each of the types is presented on a separate tab. Products of different types differ in a set of parameters.

Connection Points are not independent components and are inserted into the diagram only as part of other elements.

Prototypes of electrical components belonging to the selected group are displayed on the left side of the window. The properties of the selected element are displayed on the right.




The exception is **Base Components**. Their representation in **Component Editor** is described in detail here.

The main properties of components are presented on Parameters tab and are described in Common Parameters of Components chapter. Information from the tab is also displayed in  Parameters window, called from the context menu of diagram element.

Some product types also have a **Links** tab, described here.

Users can share the content of **Component Editor**. For more information, see the chapter Export and Import of Components.

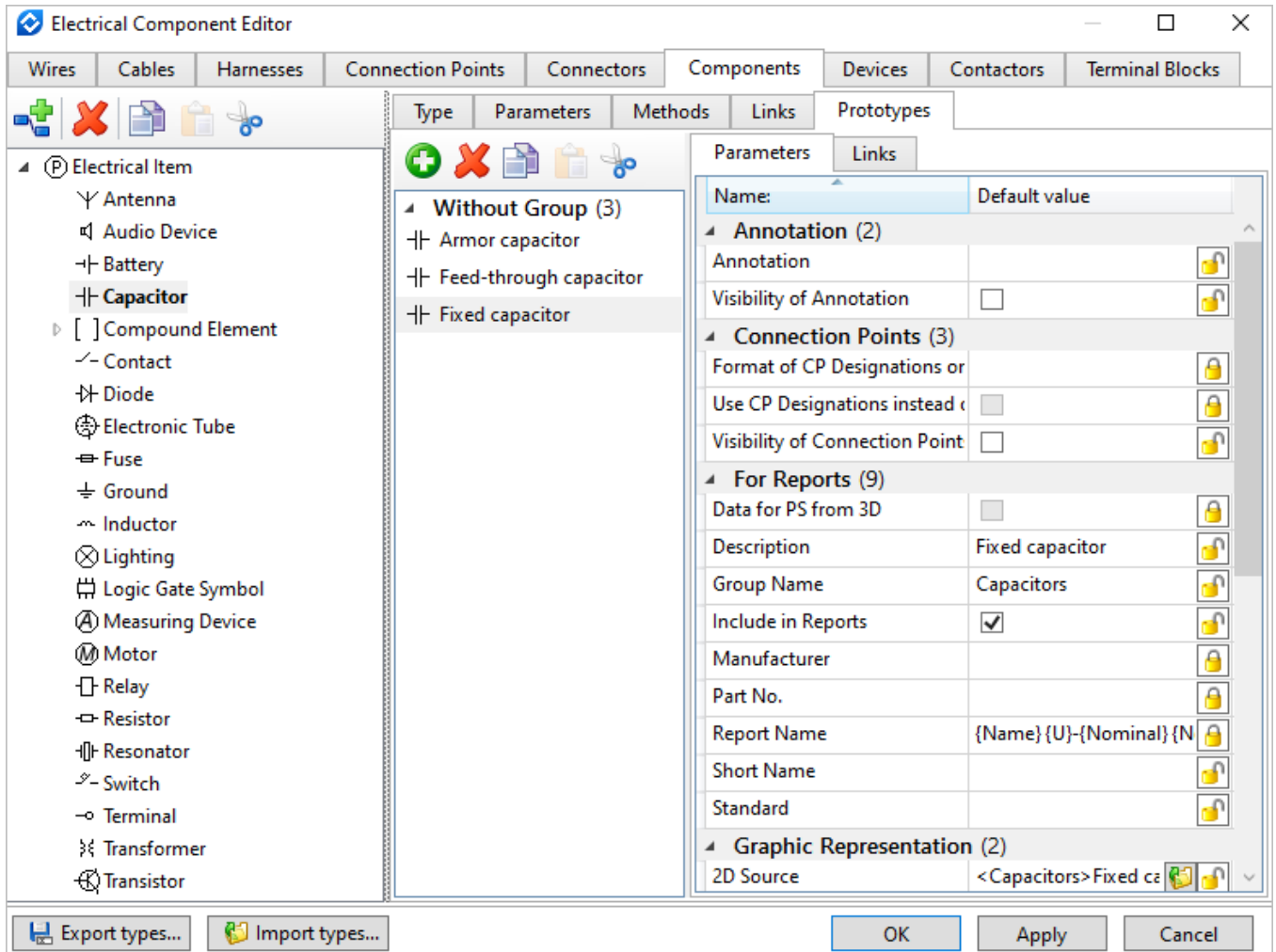
After making changes in **Component Editor** and saving them, the changes will affect only the newly created elements (in the context of the diagram or in a separate document). To transfer changes to previously created documents, use  **Update by Prototypes** command.


Base components

General information. Type tab




A **Base Component** (or just a **Component**) is a base element type in T-FLEX Electrical. Both single-component diagram elements (for example, grounding, diode) and compound ones (contactor, connector, etc.) are created on its basis. Therefore, more options are available on **Components** tab in **Component Editor** than on others.


The base component is the only component type for which grouping by child types is available. A list of child types is displayed on the left side of **Component Editor** window. The user can edit this list.



 **New Child Type** button allows you to add a new type to the base component tree. When creating a new child type, it is taken into account which object is selected in the tree. If **Basic Component** line is selected, a new child type will be created at the level below (that is, at the same level as diode, resistor, contact, etc.). If one of the child types is selected, its subtype will be created (as is done, for example, in **Compound Element**).


 **Delete Selected Object** button removes the selected child type from the tree.

 **Copy**,  **Cut**,  **Paste** buttons let you copy, cut and paste the selected child type, accordingly. Basic properties of child types are edited on **Type** tab.

Type	Parameters	Methods	Links	Prototypes
Name:	<input type="text" value="Antenna"/>			
Synonym:	<input type="text" value="Antenna"/>			
Editor:	<input type="text"/>			
Comment:	<input type="text"/>			
	<input type="checkbox"/> Abstract <input type="checkbox"/> Enumeration <input type="checkbox"/> Hide elements on diagram			
Icon				<input type="button" value="Select Icon Path"/> <input type="button" value="Load Icon from File"/> <input type="button" value="Delete Icon"/>

Name. The field specifies the name of the type.

Synonym. The field specifies a synonym for the type name (its system name).

Editor. You can specify the path to auxiliary grb-file intended for editing the parameters of the element in the field. In this case, the dialog for editing the variables of the fragment created from this file will be displayed instead of  **Parameters** window. Variables and parameters are synchronized by name.

Comment. The field contains a description of the type (arbitrary information about it).

Abstract. If the flag is activated, elements of this type cannot be created. The type will only serve to transfer common characteristics (parameters, links) to inherited types.

Hide elements in the diagram. If the flag is set, auxiliary objects will be hidden on SE graph.

Icon. You can  **Select Icon Path**,  **Load Icon from File** and  **Delete Icon** in the section.

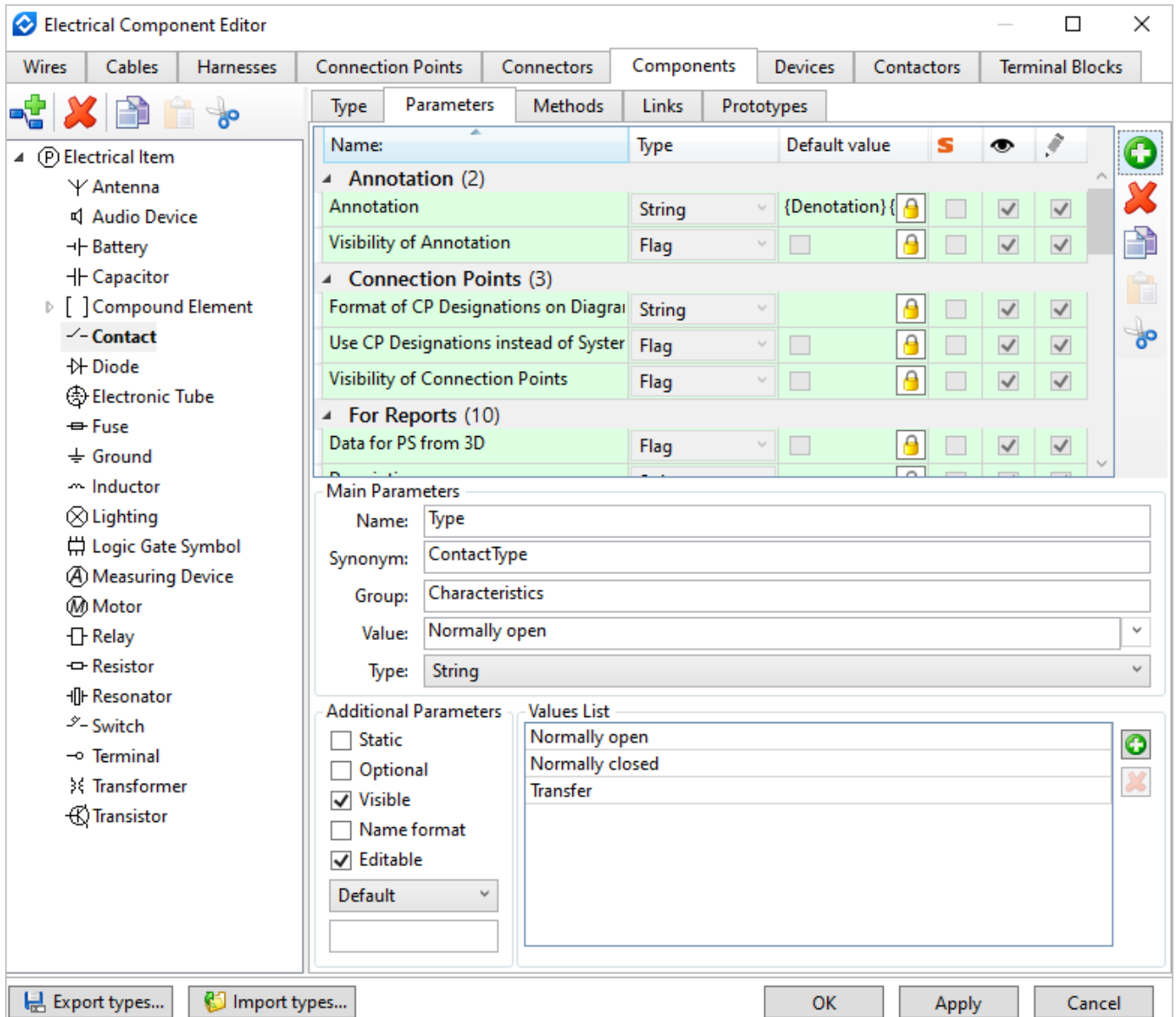
Basic component prototypes are presented on **Prototypes** tab in the middle of the window.


Parameters and **Links** tabs on **Components** tab are presented twice - for child types and prototypes.


Parameters tab

This chapter describes how to work with parameters of child types of base components. Parameters of prototypes (including base components) are described here.




You can edit the parameters set of selected child type on **Parameters** tab relating to the child base component types.



Parameters inherited from **Basic Component** type are marked in green. The values of these parameters can be edited (for this you need to click on the icon ) , but parameters themselves cannot be deleted. Some child types (eg antennas, motors) have only such parameters. However, others (for example, diodes, capacitors, contacts) have additional parameters marked in white. These parameters can be edited and deleted.

 **Add Parameter** button is used to create a new parameter for the selected child type. All manually created parameters are highlighted in white. After creating a parameter, you need to set its properties at the bottom of the window.

 **Delete Parameter** button deletes the selected parameter.



 **Copy**,  **Cut**,  **Paste** buttons allow you to copy, cut and paste the selected parameter, correspondingly.

Name field is used to specify the name of parameter.

Synonym field sets the system name of parameter by which it will be possible to refer to it when solving some tasks.


Group field specifies the name of the group to which the created parameter will be assigned. If the name of one of existing groups is entered (for example, **Characteristics** or **For Reports**), the

parameter will be added to it. If a new group name is entered, a group with this name will be created and this parameter will be added to it.


Value field sets the parameter value. A parameter can have a predefined set of values that should be entered in the **Values List** section.  **New Value** and  **Delete Value** buttons to the right of the list allow correspondingly adding a value to the list and removing the selected value from it. The list of values is displayed when you click on the arrow button to the right of the **Value** field.

The parameter value can be formed from the values of other parameters using synonyms. See here for details.


Type field is used to set the type of parameter value. The following options are available: **Flag** (the parameter can take the values Yes and No), **Integer**, **Numeric** (units can be assigned to the parameter), **String** (for text parameters), **Icon** (the parameter will be associated with a graphic image), **Enumeration** (an "advanced" variant of values list in which other parameters are used as values).

By default, parameters of child types are passed to their associated prototypes. **Static** flag defines that the parameter belongs to child type and must not be "passed" to prototypes created on its basis (that is, for example, resistor type has a parameter, but variable resistor and potentiometer prototypes do not have it). This flag is also displayed in a separate column  in **Component Editor**.

Optional flag specifies that parameter is an additional one. More explicitly, its function is revealed when creating a parameter through the modal parameters window (for more details, see here).

Visible flag determines if parameter is hidden in parameters list of selected diagram element. This flag is also displayed in a separate column  in **Component Editor**.

Name Format flag defines that parameter value can be formed from the values of other parameters (such as **Annotation** and **Report Name**). See here for details.

Editable flag determines if parameter value can be changed in the **diagram**. This flag is also displayed in a separate column  in **Component Editor**.

Editor drop-down list is intended for linking the parameter with some CAD editors, in particular:

Default - there is no connection to the editor,

RGB - the parameter sets the color from the system color library (for example, **Sheath Color** for cable components),

File Path - a link to the file is specified through the parameter (for example, **3D Source** at connectors),

Material - the parameter sets the material from the system library of materials (for example, **Sheath Material** for cable components).

The field under the drop-down list is intended for setting a link to the editor.

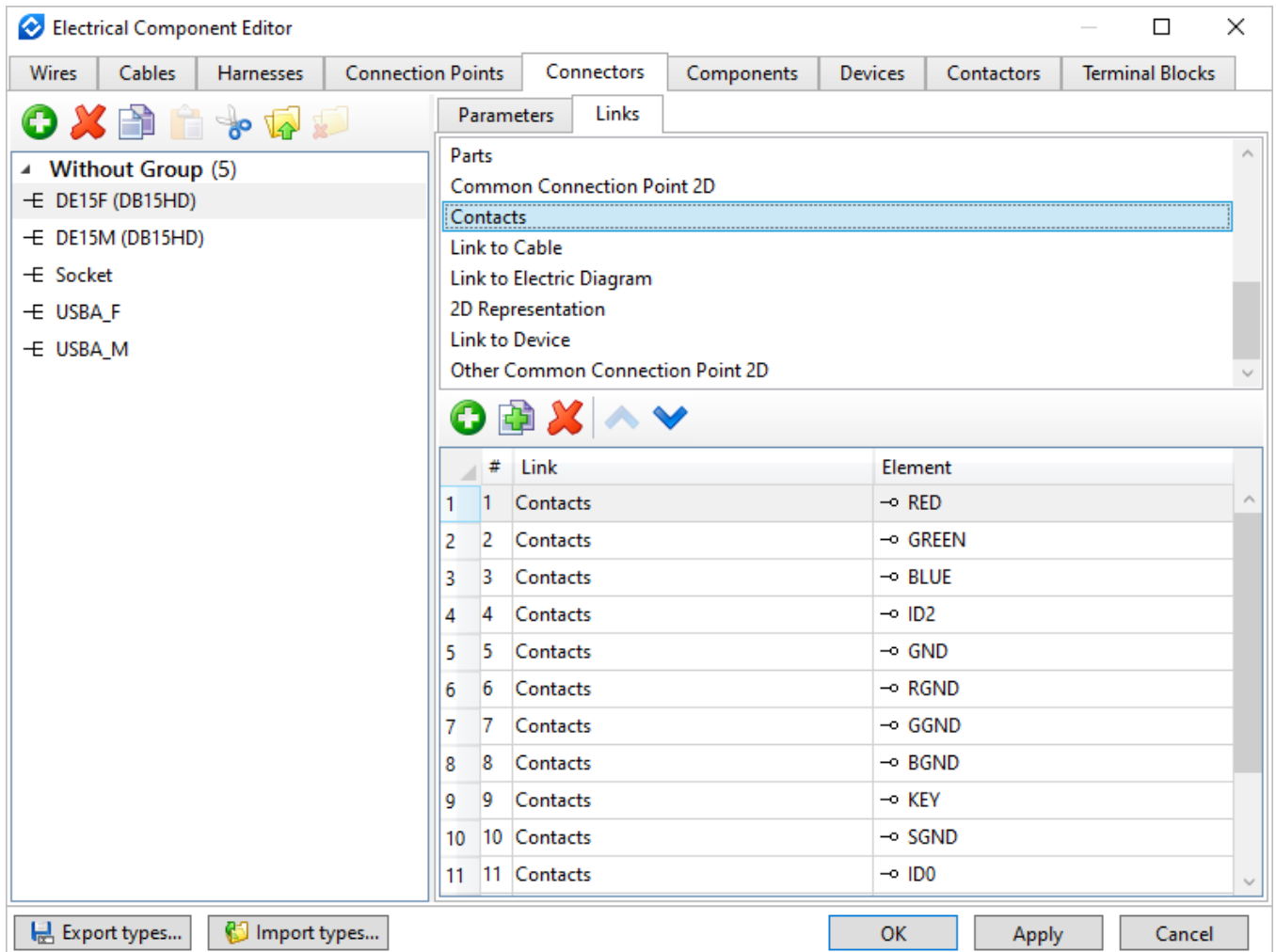
Methods tab

The settings on this tab are related to system ones.

A method is an action that can be performed on an object. Methods refer to the basic functionality of structural elements. A method can be either a macro or a reference to a "function" in a user library (file with a .dll extension).

Links tab

Links tab is available for all types of components, except for **Connection Points** and **Wires** (these elements are located at the "lower" level of the component structure and cannot have links to "lower-level" parts). You can set links of the selected child type or prototype with other objects on this tab.



Most of the links are system ones and do not need to be configured by the user.


The following links are subject to customization:

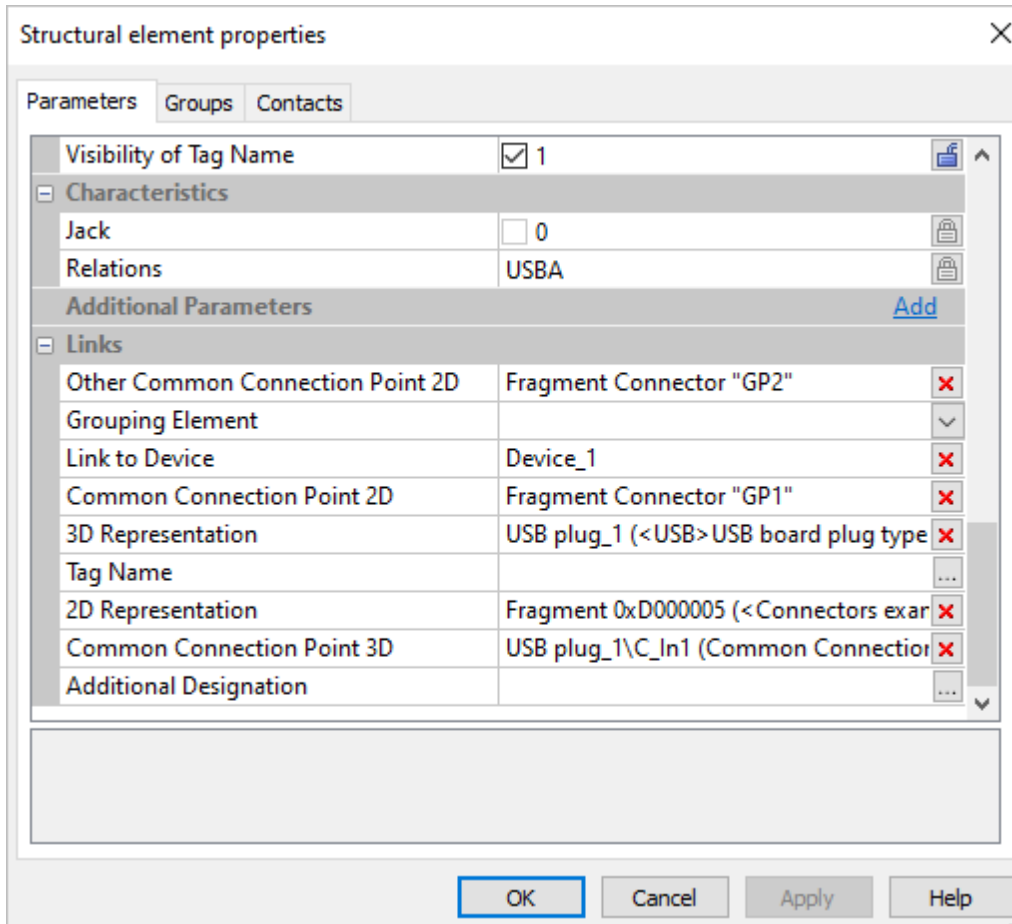
Contacts when creating prototypes of **Connectors** (see details here) and **Components** (see details here),

Connectors (Input) when prototyping **Cables** (see details here),

Connectors (output) when prototyping **Cables**,

Cores when prototyping **Cables**.

The list also displays links that are filled in the diagram. They are presented in the lower part of  **Parameters** dialog called from the context menu of diagram element.



Grouping Element. This field specifies the system name of the functional group that the component belongs to. If the group includes a compound component, this connection will be filled only at the top-level structural element (for example, the connection will be for the contactor structural element, but the contacts and coils included in it will not).

Link to Device. It is presented only for connectors and terminal blocks. The field indicates the system name of the device, if it includes a terminal block or connector.

Common Connection Point 2D. Available for connectors only. For a simplified connector, the field displays 2D connector of common connection point (cable side). For pin-to-pin connector, the field displays a hidden 2D connector of common connection point.

3D Representation. The field displays the object in 3D scene with which the selected diagram element is associated. For components, it can be a solid or a 3D fragment, for a common connection point, it can be a 3D connector (LCS). See here for more details.

Tag Name. System parameter, currently not used.


2D representation. The field contains a link to a 2D fragment with image of the selected diagram element.


Common Connection Point 3D. Available for connectors only. The field contains the system name of the LCS, which is used as a common connection point for the connector in 3D scene.



Additional designation. System parameter, currently not used.

Other Common Connection Point 2D. Available for connectors only. For a simplified connector, the field indicates the 2D connector of the common connection point (from the mounting side). A hidden 2D connector of common point on the mounting side is indicated for the pin-to-pin connector.

To add a link, select the child type or prototype in **Component Editor**, open **Links** tab, select link type in the list (for example, **Contacts**) and click  **Add Linked Object** button.

 **Copy Selected** button allows you to copy the selected linked item and paste it to the end of the list.

 **Delete Element** button removes the selected linked item from the list.


 **Move Up** and  **Move Down** buttons allow you to change the ordinal number of selected linked element by moving it up and down the list.

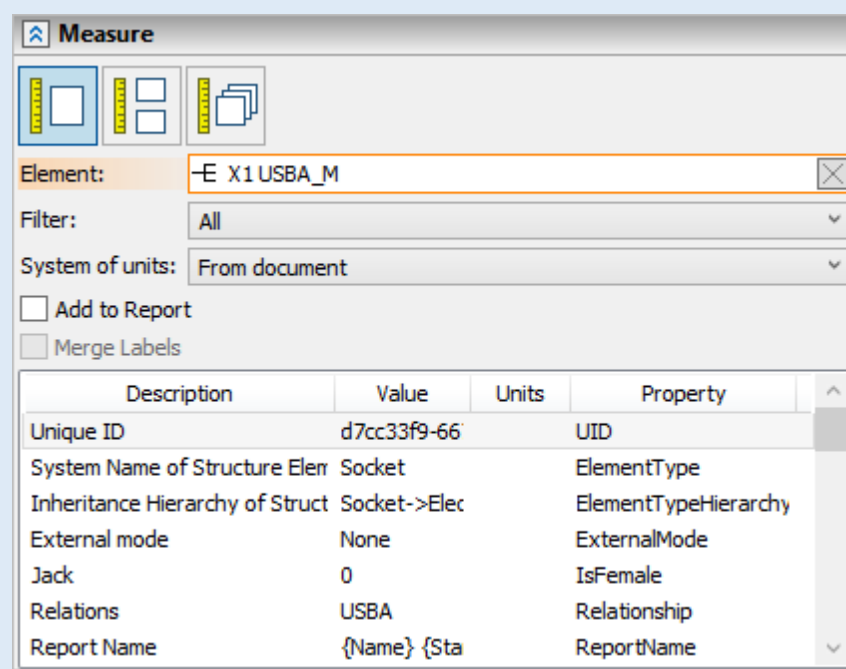
Parameters of components

Common parameters of components

The parameters of electrical components are divided into groups depending on the functions they perform.

Most of the parameters are common to all types of components. Unique parameters related to a specific component type are presented in **Characteristics** group. Such parameters are described in separate chapters of the current subsection. The parameters from general groups (**Annotation**, **Graphic Representation**, etc.) are described below.

Next to some parameters, their system synonyms are indicated (see below for more details), since these synonyms can be used when setting the values of other parameters. It is convenient to look at the complete list of synonyms for a specific element in  **Measure** command (called from the element's context menu).

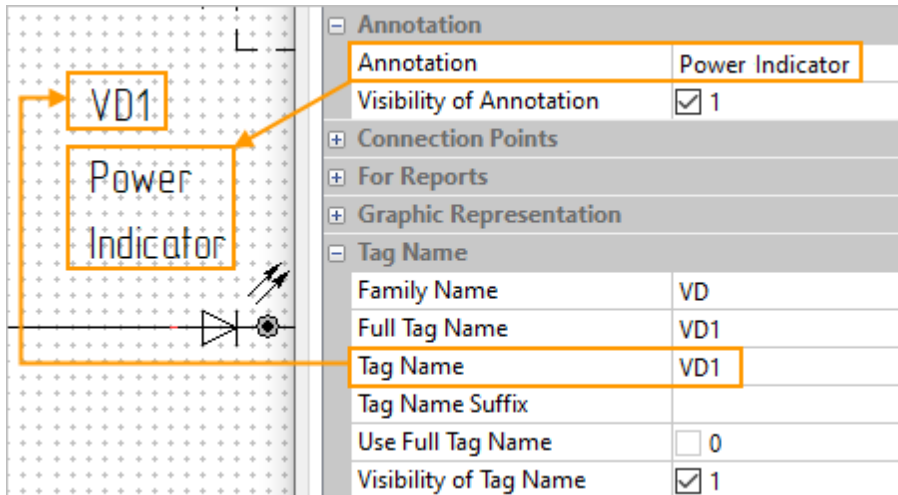


Description	Value	Units	Property
Unique ID	d7cc33f9-66		UID
System Name of Structure Eler	Socket		ElementType
Inheritance Hierarchy of Struct	Socket->Elec		ElementTypeHierarchy
External mode	None		ExternalMode
Jack	0		IsFemale
Relations	USBA		Relationship
Report Name	{Name} {Sta		ReportName

Annotation Group

Annotation {Annotation} allows you to display component information on the diagram in an additional text box. The field supports the input of synonyms, see below for details.

Visibility of Annotation controls the visibility of the element annotation on the diagram.



Connection Points group

Format of CP Designations on Diagram. The wires table always displays the full tag name of the connection point. By default, the diagram displays its own tag name. **Format of CP Designations on Diagram** field allows to designate connection points on the diagram differently from the way they are indicated in the wires table. See here for more details.

Use CP Designations instead of System Values. Tag Name parameter of CP is used to designate it on the diagram. The checkbox allows you to use another CP parameter for this purpose - **Designation**. See here for more details.

Visibility of Connection Points controls the tag name visibility of component connection points on the diagram.

For Reports Group

Data for PS from 3D. If the flag is set, the reports will use the data of a 3D fragment, linked with the structural element in **Component in 3D** command. By default, reports display the information about structural elements contained in diagram components.

Product Structure	
Description	Part No.
{SDescription}	{SDrawing}
Photo-diode	
Varicap diode	
Thermistor	
Varistor	
Push-button switch (not stay put)	

**Data for PS
is taken from
diagram element**

Product Structure	
Description	Part No.
{SDescription}	{SDrawing}
Push button switch for flashlight 17 mm	
Photo-diode	
Varicap diode	
Thermistor	
Varistor	
Push-button switch (not stay put)	

**Data for PS
is taken from
3D fragment**

Group Name is a name that is used as a name of group when combining components of this type in the product structure.

Group name is also used when constructing a common name.

		<u>Diodes</u>	
D1	Photo-diode	Group Name	1
D2	Varicap diode		1
		<u>Resistors</u>	
R1	Thermistor 0.5-1 Ohm ±20%		1
R2	Varistor 0.5-1 Ohm ±20%		1

Include in Reports. If the flag is checked, information about the component will be displayed in the reports (list of elements, wires table, etc.).

Manufacturer {Manufacturer} - parameter for specifying the manufacturer of the component.

Description {Name} - a parameter displaying the name of component. The field supports the input of synonyms.

Part No. is a parameter for specifying an alphanumeric product code (for example, ABCD.123456.789).

Report Name - full name of component in the report. By default, the report includes information about the component taken from this field.

The field supports the input of synonyms, that is, the value of this parameter can be formed from the values of other parameters. As an example, we'll consider a component such as a resistor.

Report Name parameter has the following default value: {Name} {Power} - {Nominal} {Nominal: unit} ± {Tolerance: percent}% {Standard}.

The values in curly braces are synonyms for other resistor parameters, they are specified in **Synonym**

field for base components In particular:

{Name} - **Description**,

{Power} - **Rated Power**,

{Nominal} - **Rated Resistance**,

{Nominal: unit} - units of **Rated Resistance** parameter,

{Tolerance: percent} - **Tolerance** in percents,

{Standard} - **Standard**.

As a result, the resistor is displayed in the report with the following name: Resistor 0.5-1 Ohm \pm 20%.

Short Name {ShortName} - short name of the component. The field supports the input of synonyms.

Short Report Name {ShortReportName} specifies a unique part of component name, which is displayed in the report when grouped additionally by the common name. The field supports the input of synonyms.

Standard {Standard} - a parameter that includes the standard designation by which the component is manufactured. The standard is also used into a common name.

Graphic Representation Group

2D Source contains a link to 2D fragment associated with selected structural element. This parameter is filled in **Component Editor** only for basic components, for them the link looks like <Library name> File name.grb. For other components, the parameter is filled in after inserting it into the diagram and can contain both a link to the library (relative path) and a full link to the file (absolute path).

3D Source is a link to 3D fragment that contains a 3D representation of a diagram element. By default, this parameter is blank. It is necessary to fill it out when designing cable products in 3D, see details here.

Tag Name Group

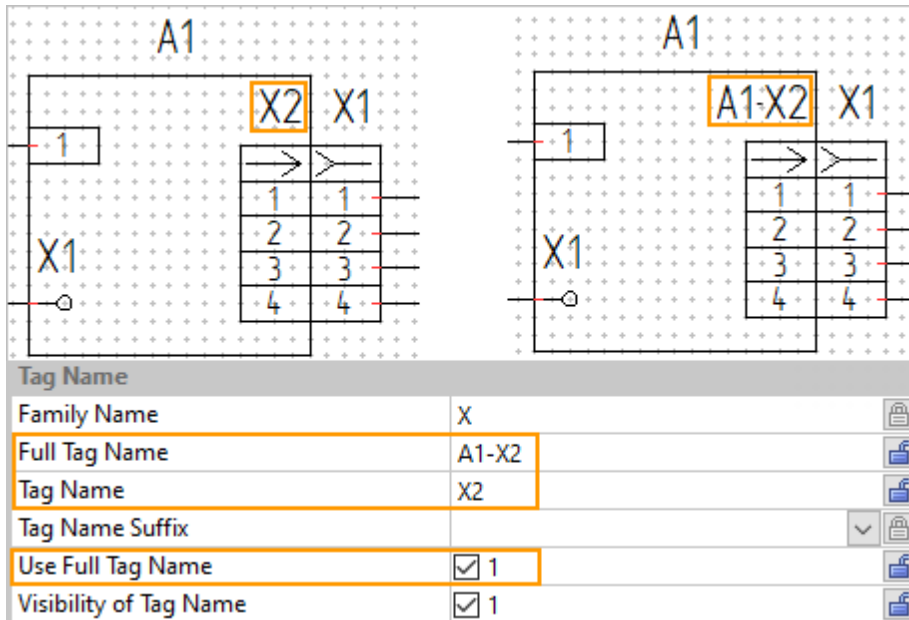
Full Tag Name {FullDenotation} contains the complete component designation consisting of the proprietary tag name and the tag name of components in which it is included. See here for details.

Tag Name {Denotation} contains its own component designation, usually consisting of a prefix and a serial number on the diagram. See here for details.

Tag Name Prefix is an alphanumeric combination (by default - alphabetic), which determines the component's belonging to a particular component group (resistors, devices, terminal blocks, etc.). See here for details.

Tag Name Suffix is an alphanumeric combination (not filled in by default) that defines a special feature of the component (for example, its digital "nature", measuring or protective function).

Use Full Tag Name replaces the element's own tag name with its complete tag name in the diagram. See here for details.



Visibility of Tag Name controls the visibility of element tag name on the diagram (it means own tag name of the element).

Special parameters of base components

The special parameters of base components, contactors and terminal blocks are collected in **Characteristics** group, with the exception of **Use as Whole** parameter of compound components (belongs to **Tag Name** group). Components with different child types (**Antennas**, **Resistors**, **Diodes**, etc.) may have a different set of characteristics. The column **Child Types** indicates which types the parameter belongs to.

Parameter	Synonym	Function	Child Type
Rated Power	{Power}	Sets rated power of component (in W All by default).	
Rated Voltage	{U}	Sets rated voltage of component (in All V by default).	
Rated Current	{I}	Sets rated current of component (in All A by default).	
Maximal Frequency	{NominalMax}	Sets maximal frequency of audio device (in Hz by default).	Audio device
Minimal Frequency	{NominalMin}	Sets minimal frequency of audio device (in Hz by default).	Audio device
Inverse Voltage	{InverseU}	Sets inverse voltage of diode (in V by default).	Diode
Inverse Current	{InverseI}	Sets inverse current of diode (in A by default).	Diode
Tolerance	{Tolerance}	Sets tolerance of unique parameter of component (as a decimal).	Inductor, Capacitor, Resistor, Resonator
Rated Inductance	{Nominal}	Sets rated inductance of inductor (in μ H by default).	Inductor
Rated Capacitance	{Nominal}	Sets rated capacitance of capacitor (in μ F by default).	Capacitor

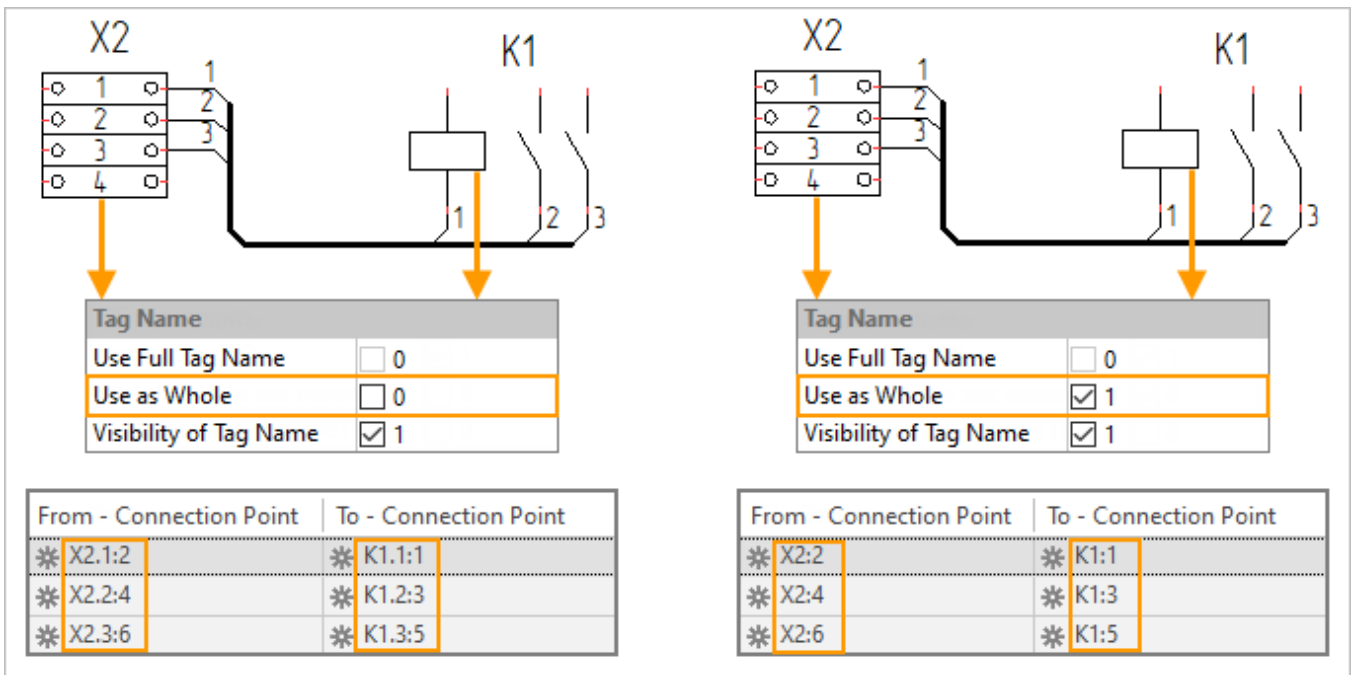
Type	{ContactType}	Sets contact type. Options Normally Open, Normally Closed, Transfer are available.	Contact
Rated Resistance	{Nominal}	Sets rated resistance of resistor (in Ohm by default).	Resistor
Rated Frequency	{Nominal}	Sets rated frequency of resonator (in kHz by default).	Resonator
Use as Whole		The flag controls the full tag name of connection points of terminal blocks and contactors. It excludes the tag name of "intermediate" elements of component (terminals in the terminal block, contacts in the relay). Thus, only the tag name of the grouping element (terminal block or contactor) and the tag name of connection points are displayed in the full tag name of CPs. In functional groups, the flag affects the way of displaying of FGs in reports.	Compound element, Contactor, Terminal block

Units of special parameters can be used in parameter values (for example, **Report Name** or **Short Report Name**). To do this, enter in the field:

{Nominal: unit} - unit of component parameter with {Nominal} synonym.

{Tolerance: percent} - **Tolerance** parameter in percent.

Use as whole controls the full tag name of connection points of terminal blocks and contactors, excluding the tag name of "intermediate" elements of component (terminals in the terminal block, contacts in the contactor). Thus, only the tag name of grouping component (terminal block or contactor) and the tag name of connection points are saved in the contact tag name. The changes are displayed in the connection table.



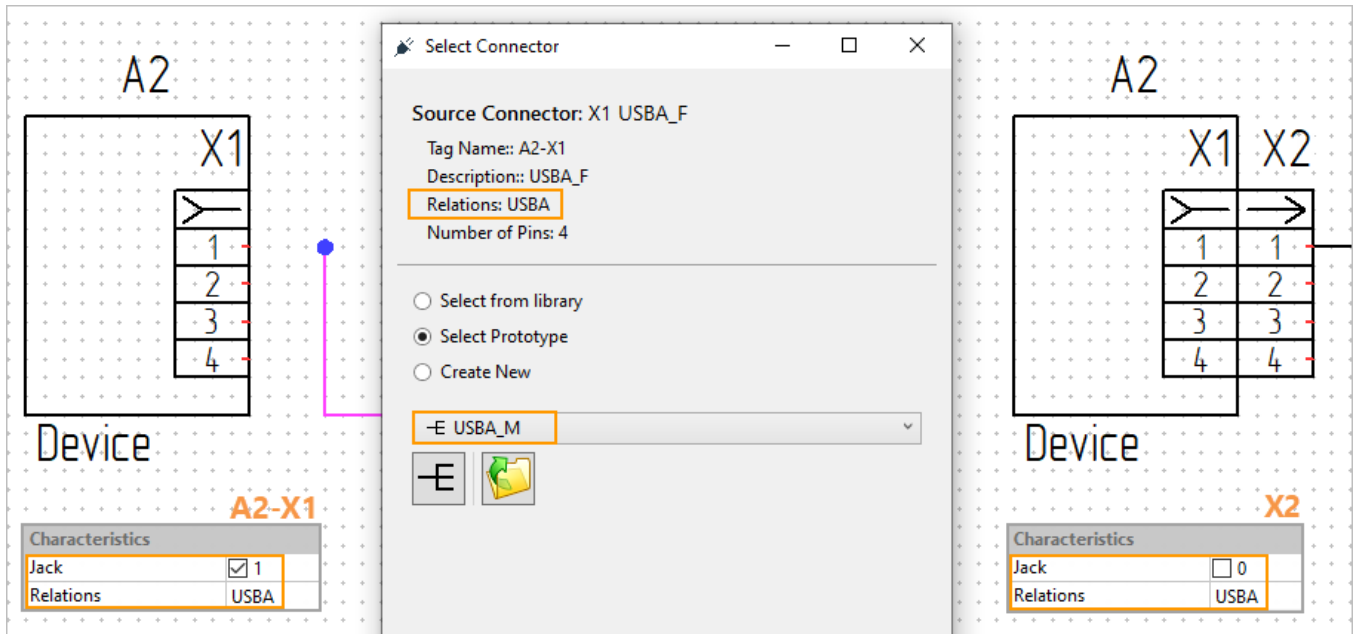
The checkbox also affects the display of functional groups in reports.

Special parameters of connectors

Special connector parameters are collected in **Characteristics** group.

Socket checkbox determines whether the connector is a socket or a plug.

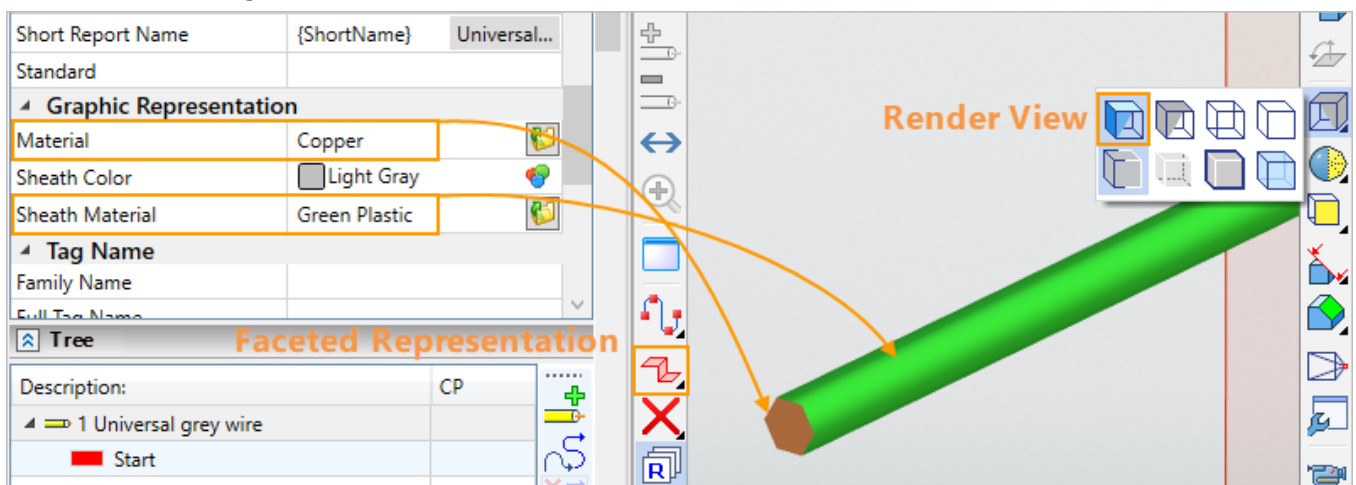
Relations field is for identifying the connectors that can be connected to each other. If the same value is entered for a female connector and a male one in **Relations** field, the system will offer the user one of them when creating a detachable connection in **Select Prototype** mode.



Special parameters of cable components

Special parameters of cable components are presented in groups **Graphic Representation**, **For Reports** and **Characteristics**.

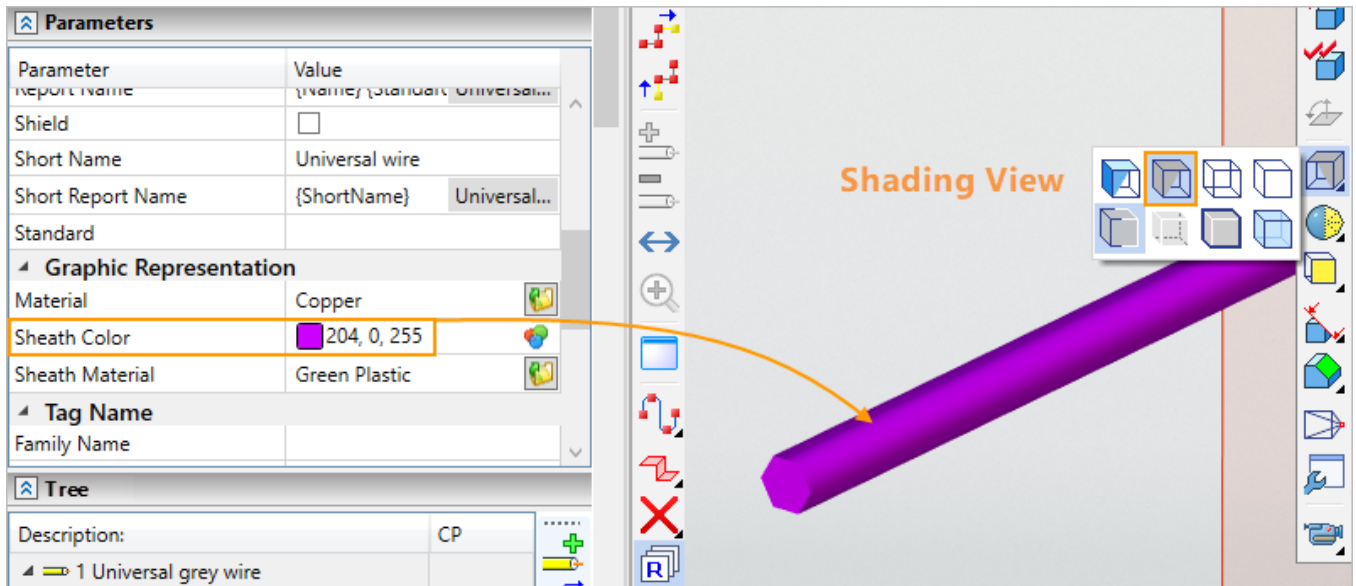
Material {Material} allows you to select the material of stripped wire from the material library. Cable component is rendered in 3D scene with the selected wire material only in **Faceted Representation** mode in CC editing command and **Render** mode on **View** panel. See here for details.



Sheath Material {CoatingMaterial} allows you to select the material of the CC sheath (insulation) from the material library. CC is drawn in a 3D scene with a selected sheath material if **Render** mode is enabled on the **View** panel.

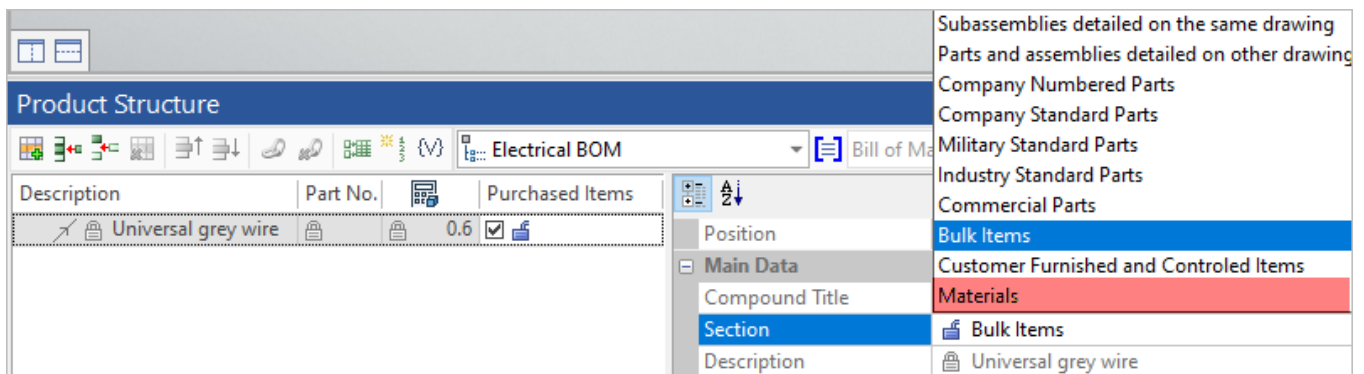
Sheath Color {Color} specifies the color from the color library with which the CC will be drawn in the

3D scene. The skin color determines the appearance of the CC in the scene if Tint mode is enabled in the View panel.



Color Name {ColorName} defines the color of the CC to display in reports. The parameter is reserved for future implementation and is not yet used by the system.

Purchased Item determines whether the cable component is purchased. If yes, it will be taken into account in the list of purchased products (a special report, that can be implemented upon request). Attention: the product, that should appear on the list of purchased items, cannot belong to "Materials" section. If a cable component needs to be included in this report, you should change its section in the parameters of PS record.




Shield determines if the CC is shielded. The parameter is reserved for future implementation and is not yet used by the system.

Diameter {Diameter} defines the diameter of the cable component with which it is drawn in 3D scene. The default unit is millimeters.

Length {Length} sets the length of the CC (in meters by default), which will be displayed in the reports. If the length of CC is not specified, the PS will contain the value calculated automatically from the 3D model. If the design of CC is carried out in 3D, it is advisable to set the length only for cables and cords.

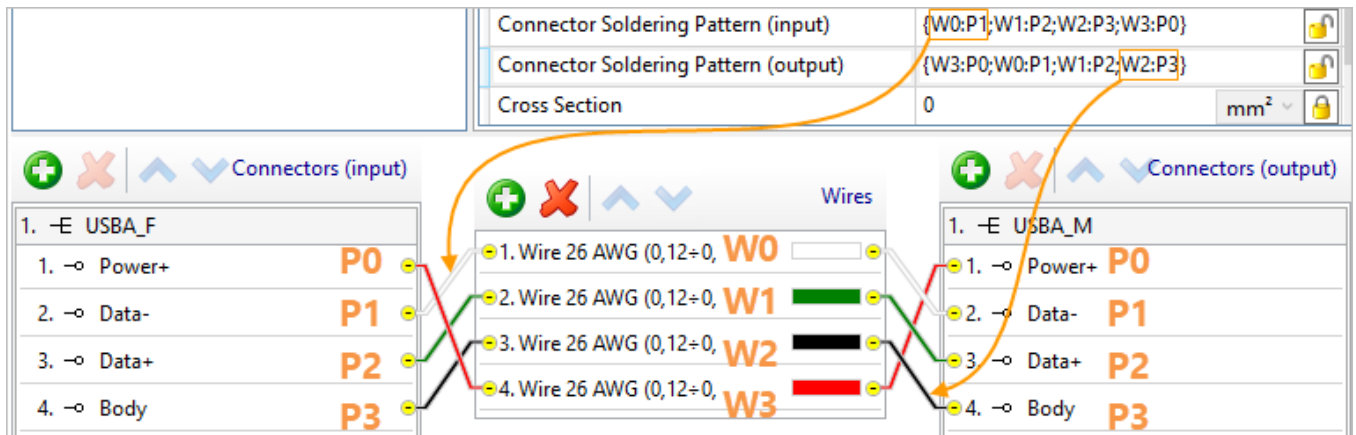
Linear Density sets the linear density of the CC, which is used to calculate its mass (as the product of linear density and length).

Minimal Bending Radius specifies the limiting radius with which the CC can be bent without violating its structure and / or increasing the current load at the bending site. Control of this parameter is carried out in  **Check Cable Component** command.

Comment {Description} specifies additional information about the CI.

Cross Section defines the area of the CC section (by default in mm²). The parameter was created for the selection of conductors by cross-section of contact (it is reserved for future implementation and is not yet used by the system).

Connector Soldering Pattern (Input) stores information about the order of connecting the wires of the cord to the contacts of connectors at the input. W0, W1, etc. are the designations for wires (wire 0, wire 1). P0, P1, etc. are the designations of connector contacts (pin 0, pin 1). If several connectors are connected to the cord on one side, their pinout will be shown in the field separated by commas. The field is filled in automatically when making changes in the connection window, so the user needs to set the cord pinout only in this window. The information from the field is used by system to create connections correctly.



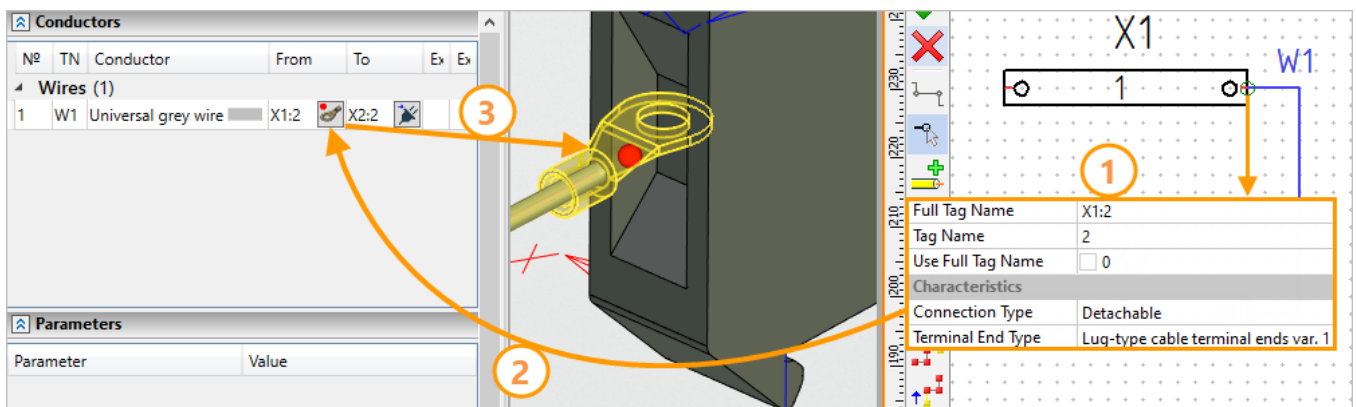
Connector Soldering Pattern (Output) stores information about the order of connecting the wires of the cord to the contacts of the connectors at the output.

Special parameters of connection points

Special connection point parameters are presented in **For Reports** and **Characteristics** groups.

Designation {SpecDenotation} specifies an alphanumeric combination that can be used to indicate connection points in the diagram and in the wires table. See here for details.

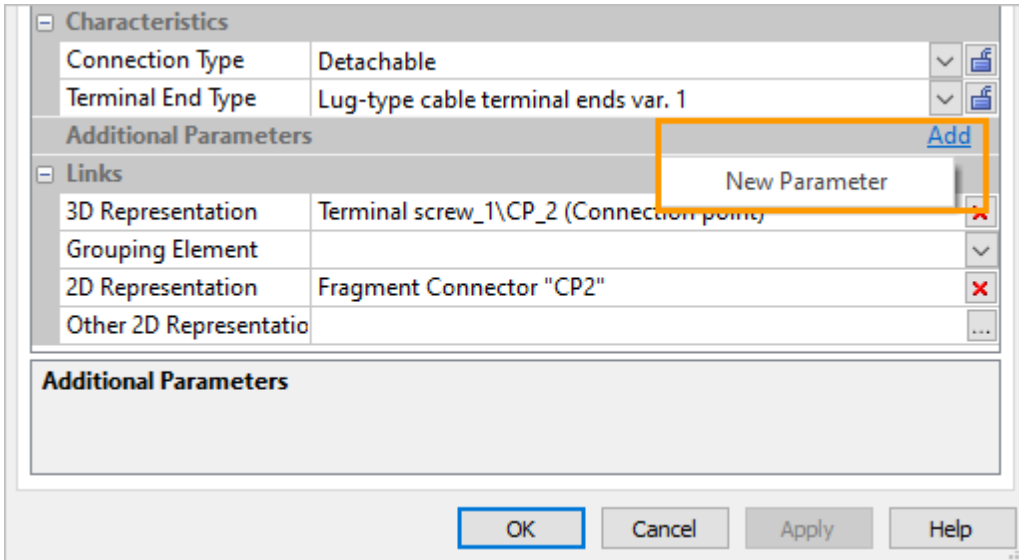
Terminal End Type allows you to assign a terminal end to a selected contact. Information about the terminal end will be displayed in the reports, and also taken into account when creating cable components, including creation in 3D scene. If the terminal end is specified in CP parameters (step 1), then its transfer to cable component (step 2) and to 3D scene (step 3) will be performed automatically.



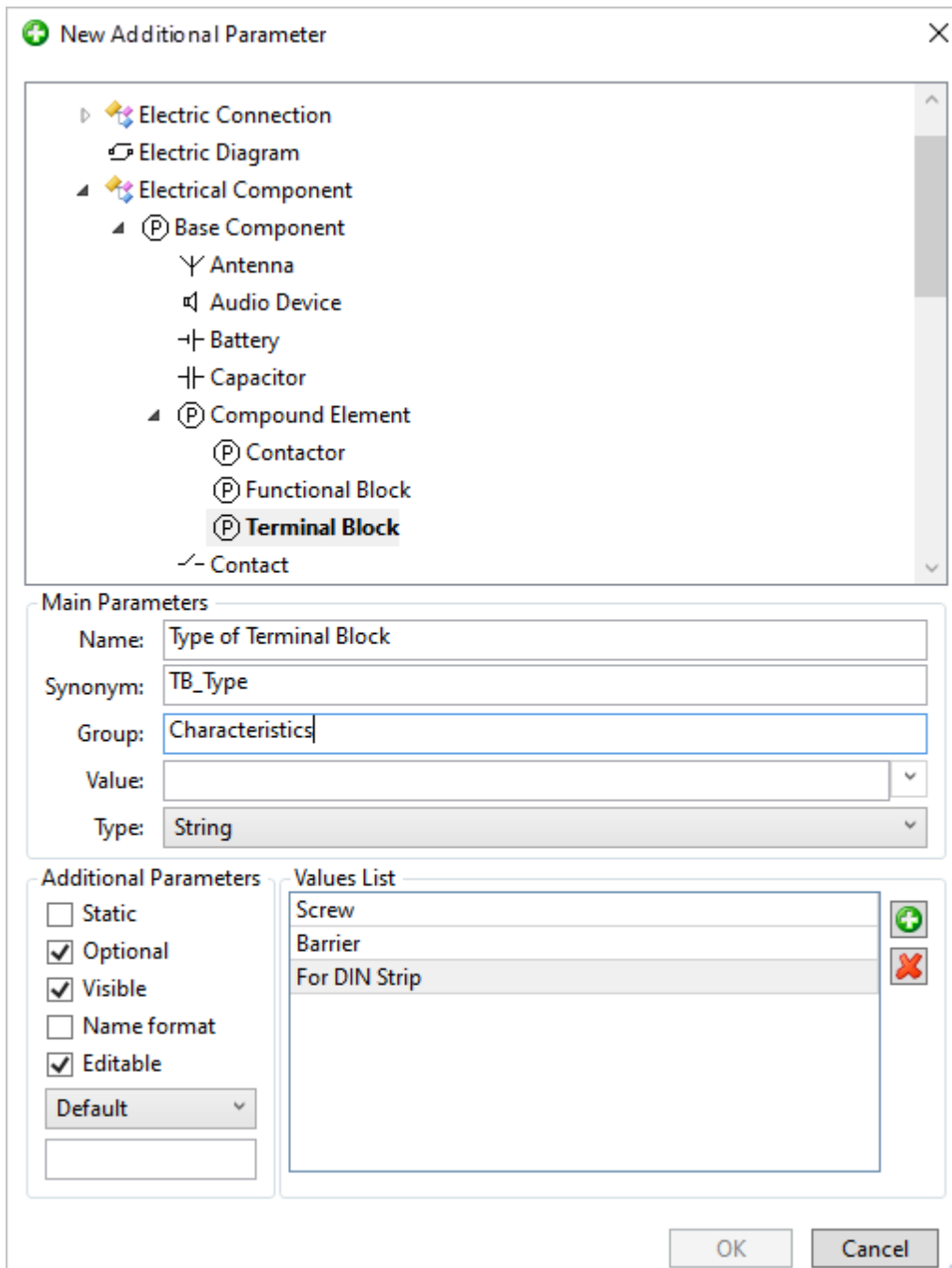
Connection Type is used to indicate the type of connection in the selected contact of component. Available options are **Soldering**, **Welding**, **Molding**, **Detachable**. The information can be displayed in wires table (this function can be implemented by request).

Creating a new parameter

To add a new parameter to the component parameters, call **Parameters** window from the context menu of the diagram element. Click **Add** and then **New parameter** in **Additional Parameters** section.



A dialog for creating a new parameter will open. At the top, Electrical component tree is displayed. The component type the previously selected element in the diagram belongs to will be selected. To create an additional parameter for a component of another type, you need to call this dialog from its parameters.

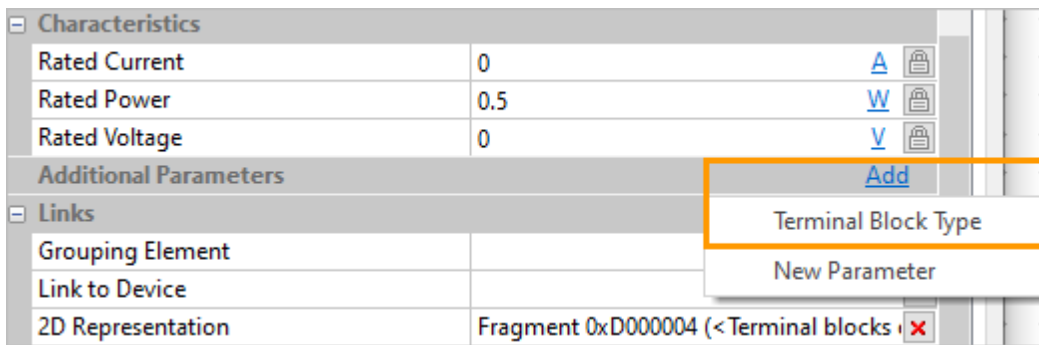



The functions of the parameters in **Main Parameters**, **Additional Parameters** and **Values List** are described here.

After specifying the properties of new parameter, click [OK]. The new parameter will be displayed in **Additional Parameters** block in **Parameters** window, called from the context menu (for ease of identification). It is displayed in the group specified in **New Additional Parameter** dialog in **Edit Component** command.

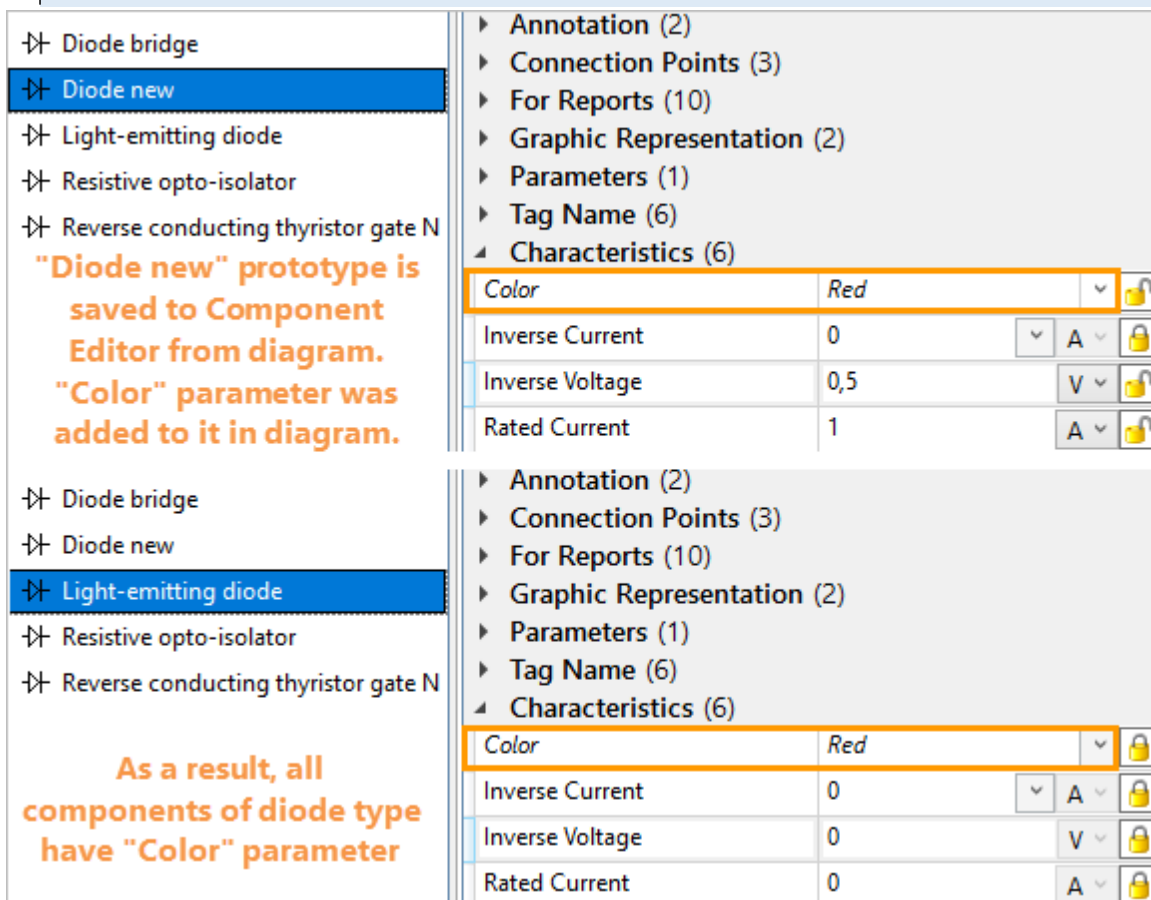
Dialog of Edit Component command		Parameters window (contextual menu)	
<div style="border: 1px solid orange; padding: 2px;"> Characteristics </div>		<div style="border: 1px solid orange; padding: 2px;"> Characteristics </div>	
Rated Current	0	Rated Current	0
Rated Power	0,5	Rated Power	0.5
Rated Voltage	0	Rated Voltage	0
<div style="border: 1px solid orange; padding: 2px;"> Terminal Block Type </div>		<div style="border: 1px solid orange; padding: 2px;"> Additional Parameters </div>	
Terminal Block Type	For DIN Strip	Terminal Block Type	For DIN Strip

The created parameter can be added to the parameters of other components of the same type. To do this, click **Add** in **Additional Parameters** section, and then select the previously created parameter from the list.



The created parameter can be transferred to the prototypes of **Component Editor** so that new components of this type are created with the new parameter. To do this, select  **Save SE as Prototype in Component Editor** in the context menu of diagram element. **Component Editor** dialog will open, you need to specify a name for the new prototype there. The manually added parameter will be marked in italics.


The set of parameters is common for all components of the same type (or a child type, if we are talking about the basic component), therefore it is impossible to add a parameter to only one prototype. Saving a prototype in **Component Editor** with a new parameter will result in this parameter appearing for all prototypes belonging to the same type.



The new parameter of basic component can be added in one more way - on **Parameters** tab in **Component Editor**. See here for more details.

To delete an additional parameter from the selected element, open  **Parameters** window and click

✗ in the line with it.



If you want a new parameter to appear in existing library elements, use  **Update by Prototype** command.

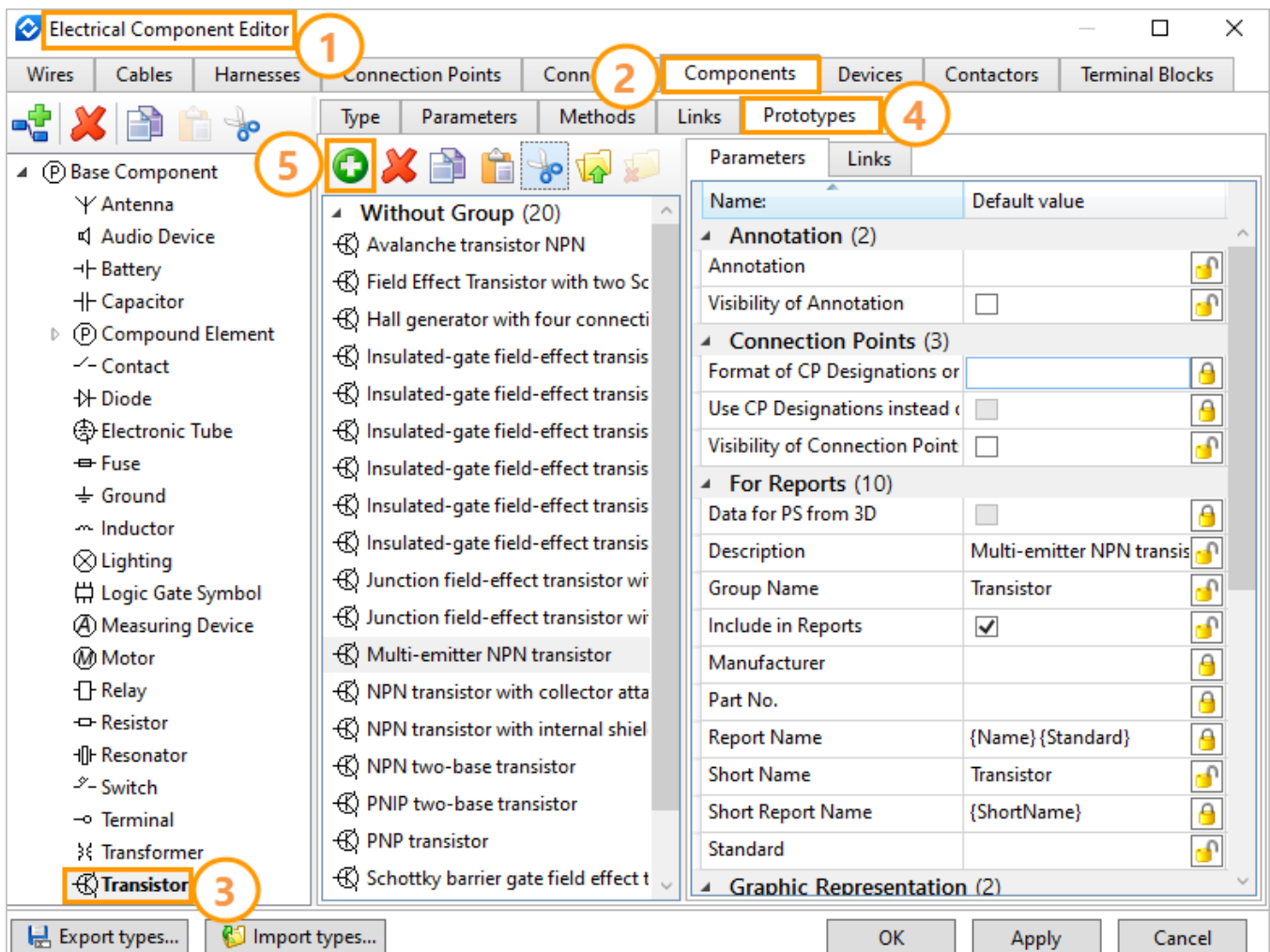
Creating new prototypes

Base component prototype

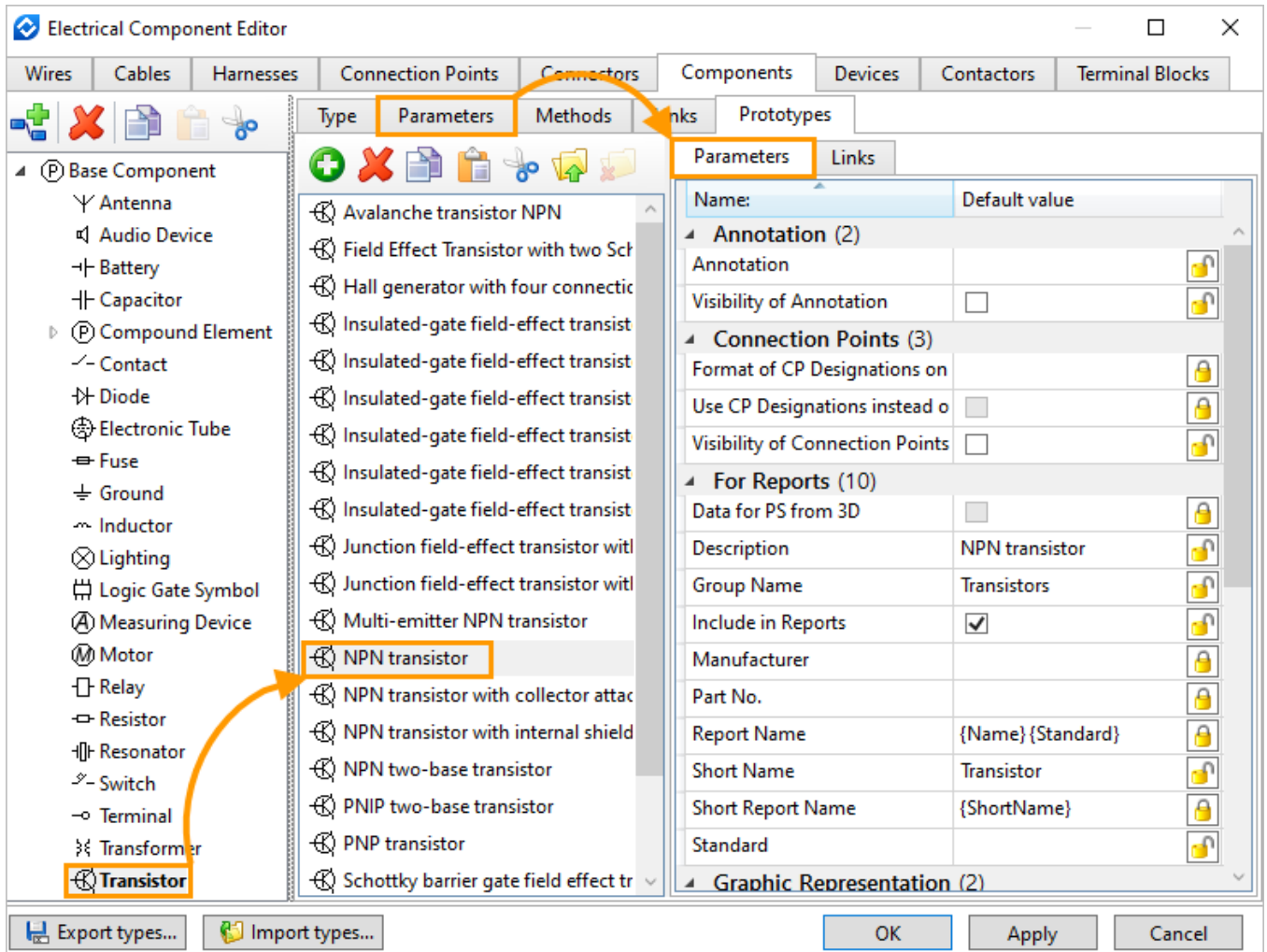
Let's create a new prototype "n-p-n transistor" of **Basic component** type.

The finished component can be found in the library "Electrical diagram elements / Active / Transistors".

Run  **Component Editor** command (1) and go to **Components** tab (2). Let's choose **Transistors** child type (3). Open **Prototypes** tab (4) and click on  **New Element** button (5).



A new prototype will be added to the list of prototypes, you need to specify its name ("NPN transistor"). The new prototype inherits all parameters and connections from **Transistor** child type.



By default, parameter fields are protected from being changed, as indicated by the icon . If you click on the icon, it will change to , and the fields will become editable.

For a new component, be sure to fill in:

Description. The field participates in the compilation of **Report Name**, which contains the element name for list of elements and BOM. **Report Name** will be filled in automatically, since it uses a synonym for **Description** parameter.

Group Name. The field must be filled in so that the component is correctly grouped in the list of elements.

Include in Reports. The flag must be activated in order for a component with this prototype to be displayed in the list of elements, wires table, etc.


2D Source. Here we will enter an absolute or relative link to the file with diagram element in which we use this prototype. The file name can be thought up in advance. An absolute link looks like C:\Folder name\...\Filename.grb, relative or link to the library - <Name of the folder in the library>File name.grb.

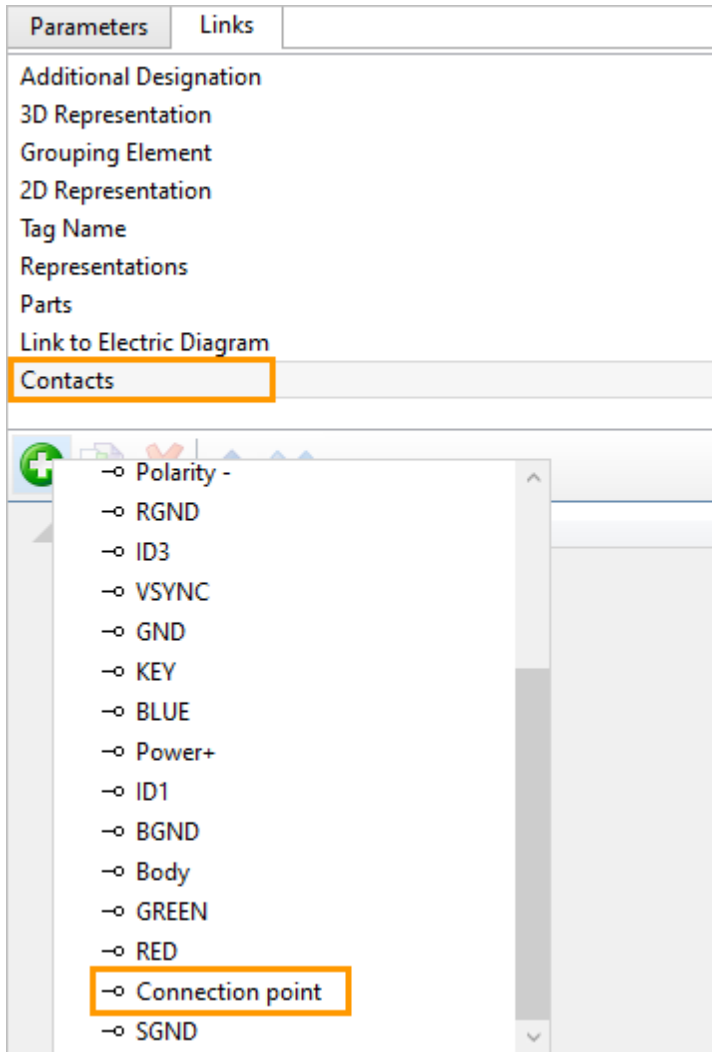
Family Name. The important parameter that ensures the correct alphanumeric identification of the component on the diagram. Usually given by letter or letters.


Visibility of Tag Name. The checkbox must be activated in order for the component to be displayed in the diagram with its tag name.

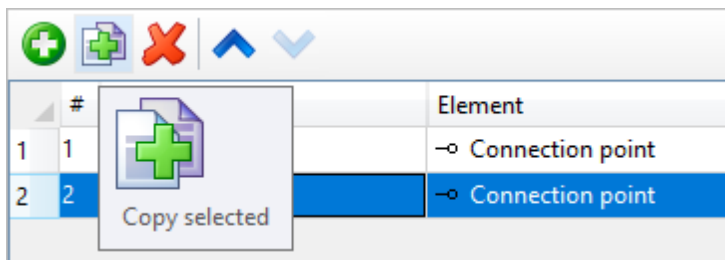
For Reports (10)		
Data for PS from 3D	<input type="checkbox"/>	
Description	NPN transistor	
Group Name	Transistors	
Include in Reports	<input checked="" type="checkbox"/>	
Manufacturer		
Part No.		
Report Name	{Name}{Standard}	
Short Name	Transistor	
Short Report Name	{ShortName}	
Standard		
Graphic Representation (2)		
2D Source	<Transistors>NPN transistor	
3D Source		
Parameters (1)		
Tag Name (6)		
Family Name	VT	
Full Tag Name		
Tag Name		
Tag Name Suffix		
Use Full Tag Name	<input type="checkbox"/>	
Visibility of Tag Name	<input checked="" type="checkbox"/>	


These and other parameters are described in more detail here and here.

You need to specify only **Contacts** link on **Links** tab for components, that is, to define the number and type of connection points of component. Our transistor should have 3 connection points. We will activate **Contacts** line and add the first CP using the button , selecting the required CP type from the list and double-clicking on it.



The remaining two CPs will be of the same type, so it is convenient to add them using  **Copy Selected** option.



If a connection point of the required type is not in the drop-down list, create a new prototype on **Connection Points** tab in  **Component Editor**. See here for more details.



The component prototype is ready, it remains to save the changes in **Component Editor** by clicking [OK].

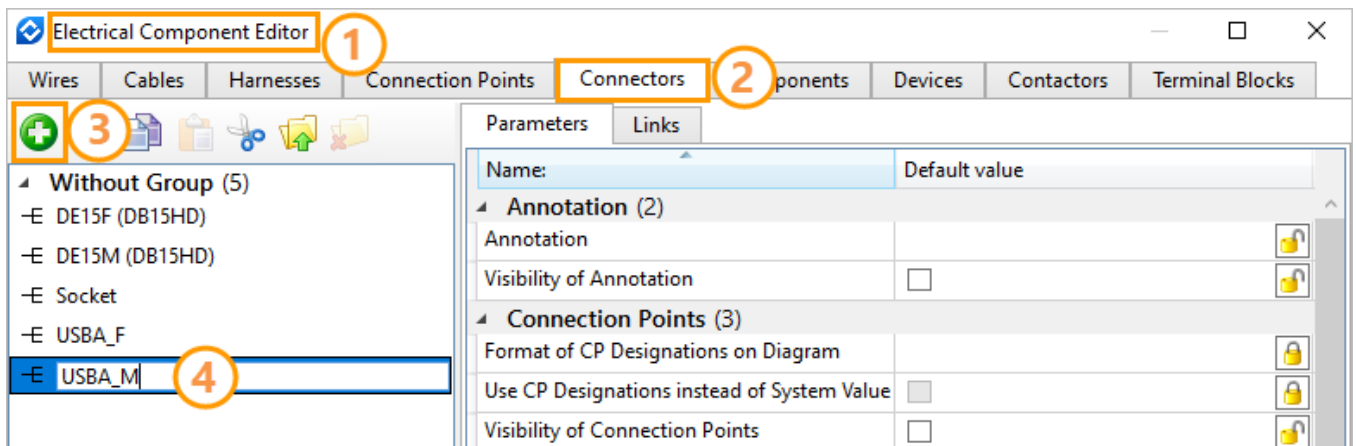
Continued: Creating the base component



Connector prototype

Let's create a new prototype of **Connector** type "USBA_M".

The finished component can be found in the library "Electrical Diagram Elements/Connectors Examples".

Run  **Component Editor** command (1) and go to **Connectors** tab (2). Click on  **New Element** button (3). A new prototype (4) will be added to the list of prototypes, you need to specify its name ("USBA_M").



By default, parameter fields are protected from being changed, as indicated by the icon . If you click on the icon, it will change to , and the fields will become editable.

For a new connector, be sure to fill in:

Description. The field participates in the compilation of **Report Name**, which contains the element name for list of elements and BOM. **Report Name** will be filled in automatically, since it uses a synonym for **Description** parameter.

Group Name. The field must be filled in so that the component is correctly grouped in the list of elements.

Include in Reports. The flag must be activated in order for a component with this prototype to be displayed in the list of elements, wires table, etc.

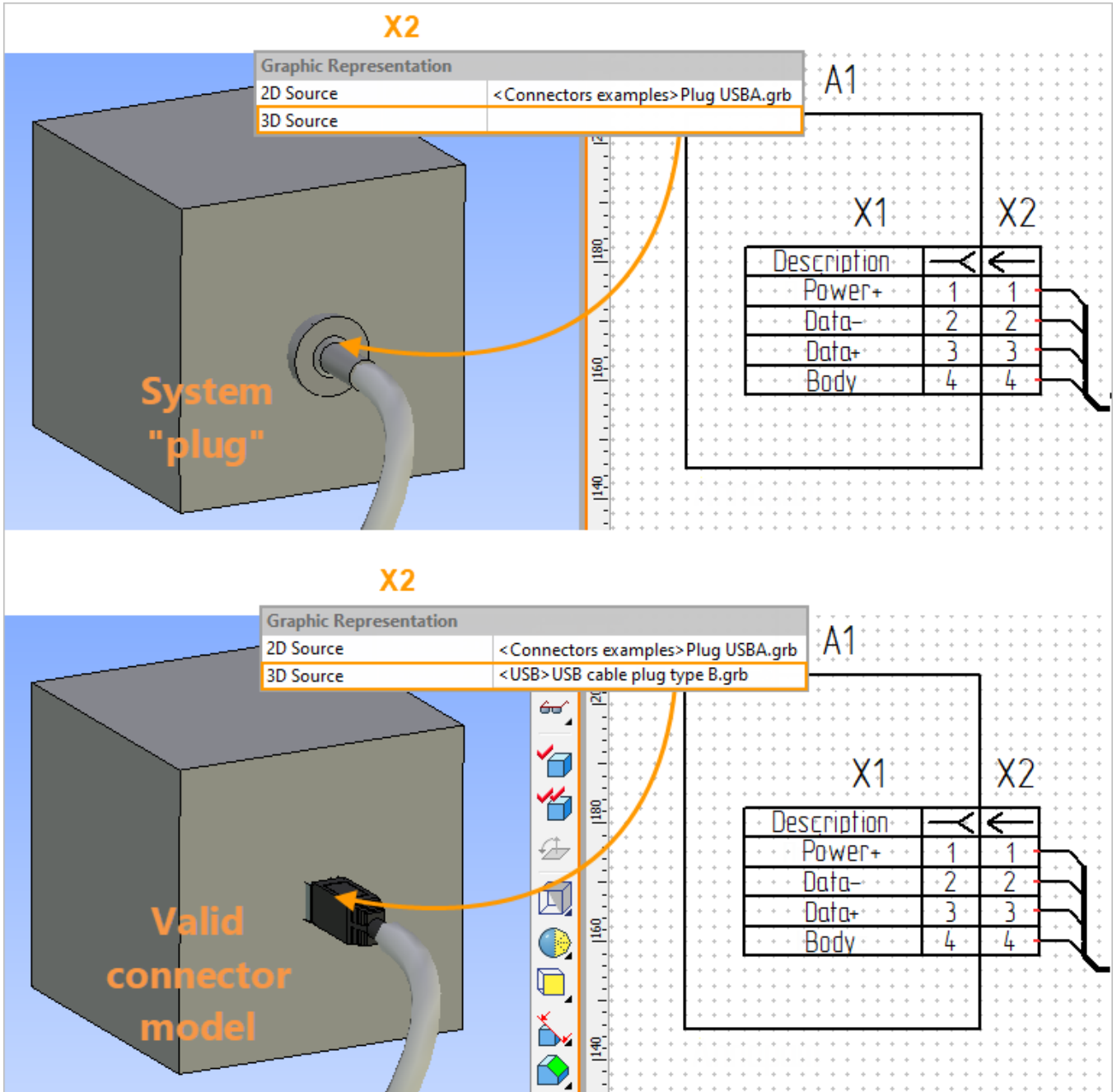
Family Name. The important parameter that ensures the correct alphanumeric identification of the component on the diagram. Usually given by letter or letters.

Visibility of Tag Name. The checkbox must be activated in order for the component to be displayed in the diagram with its tag name.

Jack. The checkbox determines the type of connector (jack or plug). It is necessary to set the required state of the flag.

Optional parameters with useful functions:


3D Source. The parameter allows you to correctly draw connectors on cable products in 3D scene, so filling in the field is important in those connector prototypes that will be used in cable components. Otherwise, the system "plug" will be used instead of 3D model of the connector. You must enter a link to the file with the 3D model of the connector in the field. An absolute link looks like C:\Folder name\... \Filename.grb, relative or link to the library - <Name of the folder in the library> File name.grb.

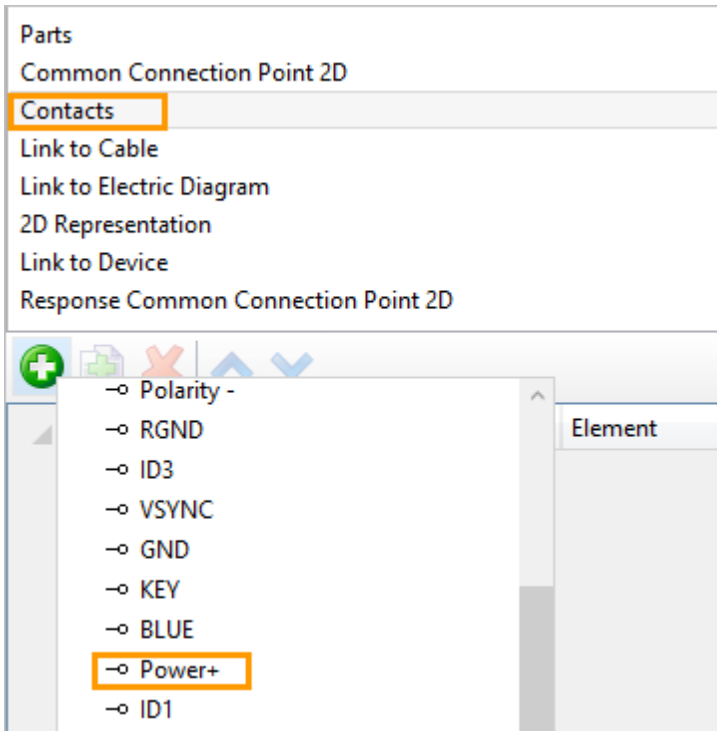


Relations. This parameter provides automatic selection of paired connectors when creating electromechanical connections without spending time searching for a suitable connector in the prototype list. We know that USBA_M connector will connect to USBA_F connector that already exists in **Component Editor**. Therefore, enter the same value in **Relationship** field for USBA_F and USBA_M connectors.

▾ For Reports (10)	
Data for PS from 3D	<input type="checkbox"/>
Description	USBA_M
Group Name	Connectors
Include in Reports	<input checked="" type="checkbox"/>
Manufacturer	
Part No.	
Report Name	{Name}{Standard}
Short Name	
Short Report Name	{ShortName}
Standard	
▾ Graphic Representation (2)	
2D Source	
3D Source	<USB>USB board plug type B.grb
▸ Parameters (1)	
▾ Tag Name (6)	
Family Name	X
Full Tag Name	
Tag Name	
Tag Name Suffix	
Use Full Tag Name	<input type="checkbox"/>
Visibility of Tag Name	<input checked="" type="checkbox"/>
▾ Characteristics (2)	
Jack	<input type="checkbox"/>
Relations	USBA

These and other parameters are described in more detail [here](#) and [here](#).

You need to specify only **Contacts** link on **Links** tab for connectors, that is, define the number and type of component connection points. Our connector should have 4 pins. We will activate **Contacts** line and add the first CP using the button , selecting the required CP type from the list and double-clicking on it.



The remaining three CPs will be added in the same way.

Common Connection Point 2D			
Contacts			
Link to Cable			
Link to Electric Diagram			
#	Link	Element	
1	Contacts	-o Power+	
2	Contacts	-o Data-	
3	Contacts	-o Data+	
4	Contacts	-o Body	

If a connection point of the required type is not in the drop-down list, create a new prototype on **Connection Points** tab in **Component Editor**. See here for more details.

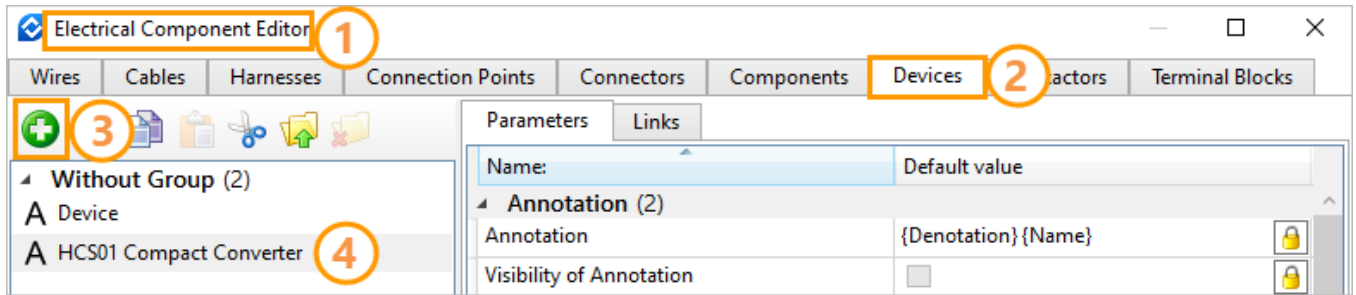
The connector prototype is ready, it remains to save the changes in **Component Editor** by clicking [OK].

Continued: creating a connector.

Device, contactor and terminal block prototypes

Let's create a new prototype of **Device** type "HCS01 Compact Converter". Contactor and terminal strip prototypes are created in the same way.

run **Component Editor** command (1) and go to **Device** tab (2). Click on **New Element** button (3). A new prototype is added to the list of prototypes (4), you need to specify its name ("HCS01 Compact Converter").



By default, parameter fields are protected from being changed, as indicated by the icon . If you click on the icon, it will change to , and the fields will become editable.

For a new device, be sure to fill in:

Description. The field participates in the compilation of **Report Name**, which contains the element name for list of elements and BOM. **Report Name** will be filled in automatically, since it uses a synonym for **Description** parameter.

Group Name. The field must be filled in so that the component is correctly grouped in the list of elements.

Include in Reports. The flag must be activated in order for a component with this prototype to be displayed in the list of elements, wires table, etc.

Family Name. The important parameter that ensures the correct alphanumeric identification of the component on the diagram. Usually given by letter or letters.

Visibility of Tag Name. The checkbox must be activated in order for the component to be displayed in the diagram with its tag name.

For Reports (10)	
Data for PS from 3D	<input type="checkbox"/>
Description	HCS01 Compact Converter
Group Name	Devices
Include in Reports	<input checked="" type="checkbox"/>
Manufacturer	
Part No.	
Report Name	{Name} {Standard}
Short Name	
Short Report Name	{ShortName}
Standard	
Graphic Representation (2)	
Parameters (1)	
Tag Name (6)	
Family Name	UZ
Full Tag Name	
Tag Name	
Tag Name Suffix	
Use Full Tag Name	<input type="checkbox"/>
Visibility of Tag Name	<input checked="" type="checkbox"/>

These and other parameters are described in more detail here.



The prototype of device is ready, it remains to save the changes in **Component Editor** by pressing [OK].

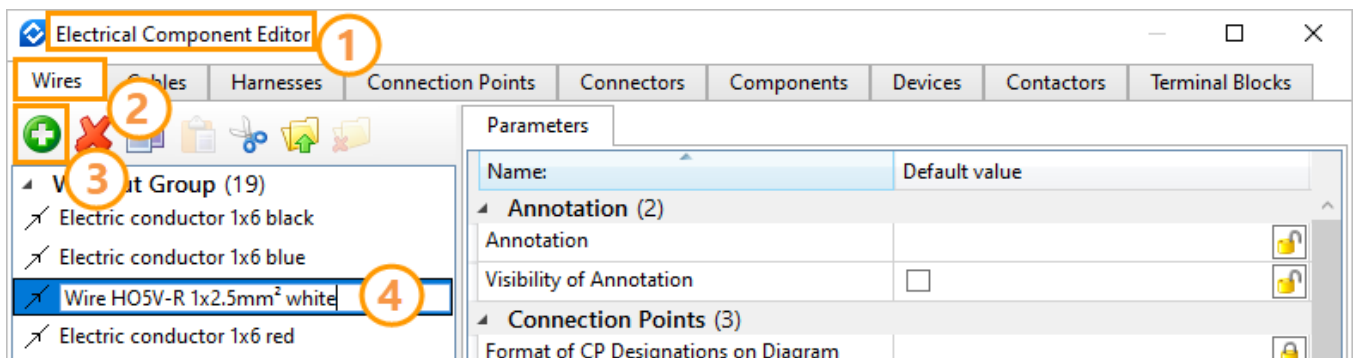
Continued: creating device.



Wire and harness prototypes

Let's create a new prototype of **Wire** type "Wire HO5V-R 1x2.5mm² white". The harness prototype is created in the same way.

The finished product can be found in **Component Editor** (cable products are stored as prototypes only).

Run  **Component Editor** command (1) and go to **Wires** tab (2). Click on  **New Element** button (3). A new prototype is added to the list of prototypes (4), you need to specify its name ("Wire HO5V-R 1x2.5mm² white").



By default, parameter fields are protected from being changed, as indicated by the icon . If you click on the icon, it will change to , and the fields will become editable.

For a new wire, be sure to fill in:

Description. The field participates in the compilation of **Report Name**, which contains the element name for list of elements and BOM. **Report Name** will be filled in automatically, since it uses a synonym for **Description** parameter.

Group Name. The field must be filled in so that the component is correctly grouped in the list of elements.

Include in Reports. The flag must be activated in order for a component with this prototype to be displayed in the list of elements, wires table, etc.

Purchased. The field must be filled in if the component is to appear in the list of purchased items. See here for more details.

Sheath Color. You need to select a color to set the appearance of the wire in 3D scene in **Shading** mode.

Sheath Material. You need to select a material to set the appearance of wire in 3D scene in **Render** mode. See here for more details.

Family Name. The important parameter that ensures the correct alphanumeric identification of the component on the diagram. Usually given by letter or letters.

Visibility of Tag Name. The checkbox must be activated in order for the component to be displayed in the diagram with its tag name.

Diameter. The field must be filled in to determine the diameter with which the wire will be drawn in 3D scene.

Length. The field must be filled in if it is necessary to display a certain wire length in the report. If you need to calculate the length systematically using a 3D model, the field can be left blank. See here for

more details.

Linear Density. The field must be filled in if it is necessary to display the weight of the wire in the report (BOM).

Minimal Bending Radius. The field must be filled in for the wire to be correctly drawn in 3D scene and, as a result, to calculate its length correctly.

For Reports (13)	
Color Name	
Data for PS from 3D	<input type="checkbox"/>
Description	Wire HO5V-R 1x2.5mm ² white
Group Name	Wires
Include in Reports	<input checked="" type="checkbox"/>
Manufacturer	
Part No.	
Purchased Item	<input type="checkbox"/>
Report Name	{Name} {Standard}
Shield	<input type="checkbox"/>
Short Name	Wire HO5V-R 1x2.5mm ² white
Short Report Name	{ShortName}
Standard	
Graphic Representation (5)	
2D Source	
3D Source	
Material	
Sheath Color	<input type="checkbox"/> (255,255,255)
Sheath Material	White Plastic
Parameters (1)	
Tag Name (6)	
Characteristics (6)	
Comment	
Cross Section	0
Diameter	1,8
Length	0
Linear Density	0
Minimal Bending Radius	18

These and other parameters are described in more detail here and here.

The wire prototype is ready, it remains to save the changes in **Component Editor** by clicking [OK].



The harness prototype is created in the same way. It is filled with wires and cables not in **Component Editor**, but is edited in-site (in a diagram or 3D model).

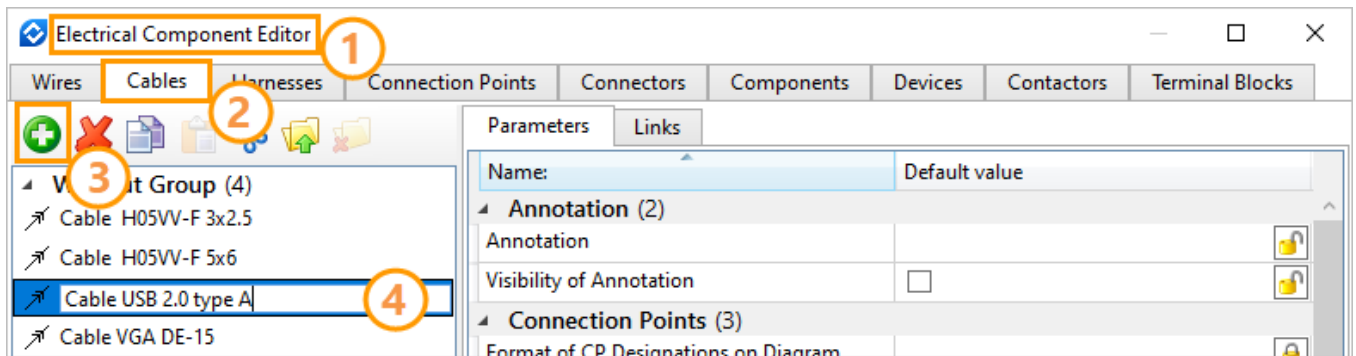
Continued: adding a wire to diagram.



Cable (cord) prototype

Let's create a new prototype of **Cable** type "Cable USB 2.0 Type A". A cord is a variety of cable. The difference lies in inclusion of connectors in the cord.

The finished product can be found in **Component Editor** (cable components are stored as prototypes only).

Run  **Component Editor** command (1) and go to **Cables** tab (2). Click on  **New Element** button (3). A new prototype is added to the list of prototypes (4), you need to specify its name ("Cable USB 2.0 Type A").



By default, parameter fields are protected from being changed, as indicated by the icon . If you click on the icon, it will change to , and the fields will become editable.

For a new wire, be sure to fill in:

Description. The field participates in the compilation of **Report Name**, which contains the element name for list of elements and BOM. **Report Name** will be filled in automatically, since it uses a synonym for **Description** parameter.

Group Name. The field must be filled in so that the component is correctly grouped in the list of elements.

Include in Reports. The flag must be activated in order for a component with this prototype to be displayed in the list of elements, wires table, etc.

Purchased. The field must be filled in if the component is to appear in the list of purchased items. See here for more details.

Sheath Color. You need to select a color to set the appearance of the wire in 3D scene in **Shading** mode.

Sheath Material. You need to select a material to set the appearance of wire in 3D scene in **Render** mode. See here for more details.

Family Name. The important parameter that ensures the correct alphanumeric identification of the component on the diagram. Usually given by letter or letters.


Visibility of Tag Name. The checkbox must be activated in order for the component to be displayed in the diagram with its tag name.

Diameter. The field must be filled in to determine the diameter with which the wire will be drawn in 3D scene.

Length. The field must be filled in if it is necessary to display a certain wire length in the report. If you need to calculate the length systematically using a 3D model, the field can be left blank. See here for more details.


Linear Density. The field must be filled in if it is necessary to display the weight of the wire in the report (BOM).

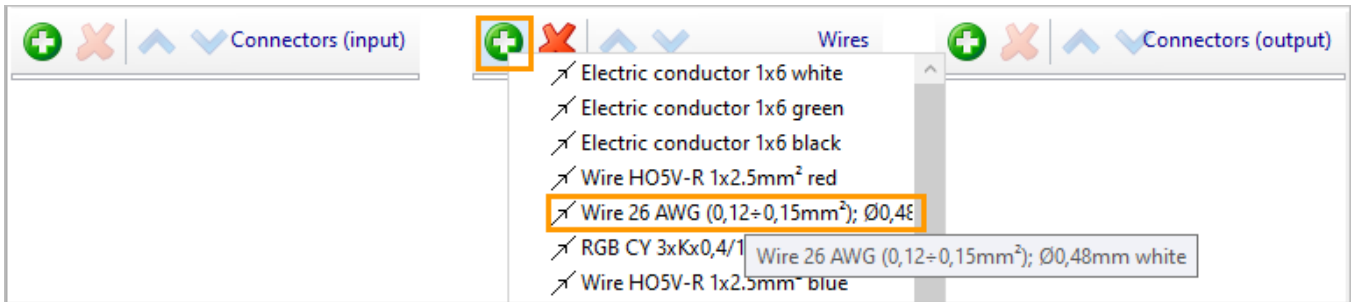
Minimal Bending Radius. The field must be filled in for the wire to be correctly drawn in 3D scene and, as a result, to calculate its length correctly.

For Reports (13)	
Color Name	<input type="text"/>
Data for PS from 3D	<input type="checkbox"/>
Description	Cable USB 2.0 type A
Group Name	Cables
Include in Reports	<input checked="" type="checkbox"/>
Manufacturer	
Part No.	
Purchased Item	<input checked="" type="checkbox"/>
Report Name	{Name} {Standard}
Shield	<input type="checkbox"/>
Short Name	
Short Report Name	{ShortName}
Standard	
Graphic Representation (5)	
2D Source	
3D Source	
Material	
Sheath Color	 (0,0,0)
Sheath Material	Black Plastic
Parameters (1)	
Tag Name (6)	
Characteristics (8)	
Comment	
Connector Soldering Pattern (
Connector Soldering Pattern (
Cross Section	0
Diameter	3,5
Length	1,6
Linear Density	0,0167
Minimal Bending Radius	7

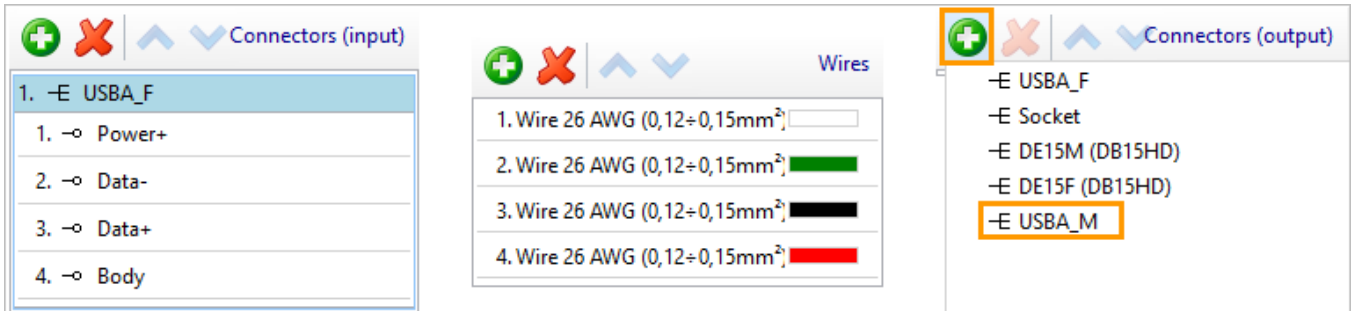
These and other parameters are described in more detail here and here.

It is necessary to specify **Cores** (conductors) on **Links** tab for cables and cords, since the set of wires in them is predefined. For cords, you also need to set **Connectors (Input)** and **Connectors (Output)** links. The cord can be "multi-headed", that is, it can contain more than two connectors. It is convenient to set cable and cord links in a special connection block located at the bottom of **Cables** tab of **Component Editor**.


Our cord should have four wires. We add the first wire using button  in **Conductors** section, then select the required type from the list and double-click on it. The rest of the wires will be added in the same way. The prototype of the cable will be ready at this stage.

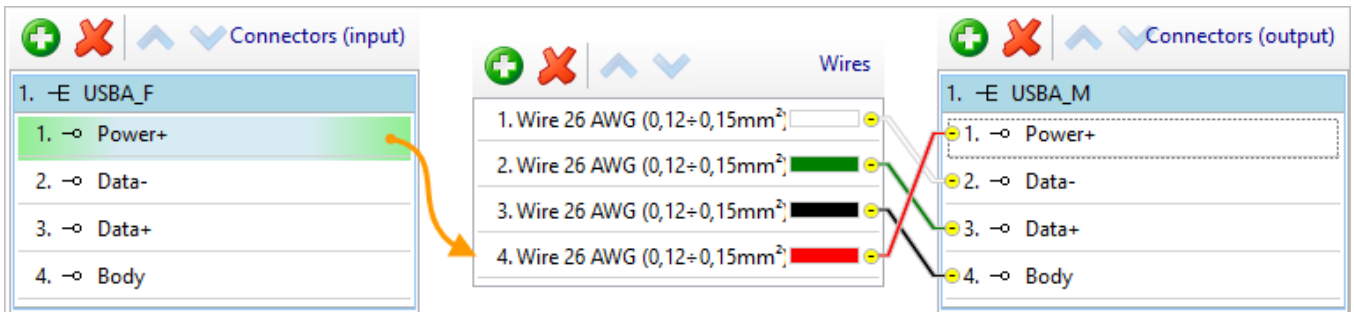


We add USBA_F and USBA_M connectors for the cord to the input and output, respectively.



Now we need to set the pinout in the cord. To do this, select the first contact of USBA_F connector (it will be highlighted in green) and, holding LMB, move the cursor to the red wire. The connection between wire and contact will be shown as a broken line in the same color as the wire.

We will also connect all other contacts with the required wires. To remove a link, just click on  at the end of the corresponding polyline.



The links we set will automatically fit into the cable parameters **Connector Soldering Pattern (Input)** and **Connector Soldering Pattern (Output)**. See here for more details.

If a connector or a wire of the required type is not available in the drop-down lists, create a new prototype on **Connectors** or **Wires** tabs in **Component Editor**. See here and here for more details.

The prototype of the cord is ready, it remains to save the changes in **Component Editor** by clicking [OK].

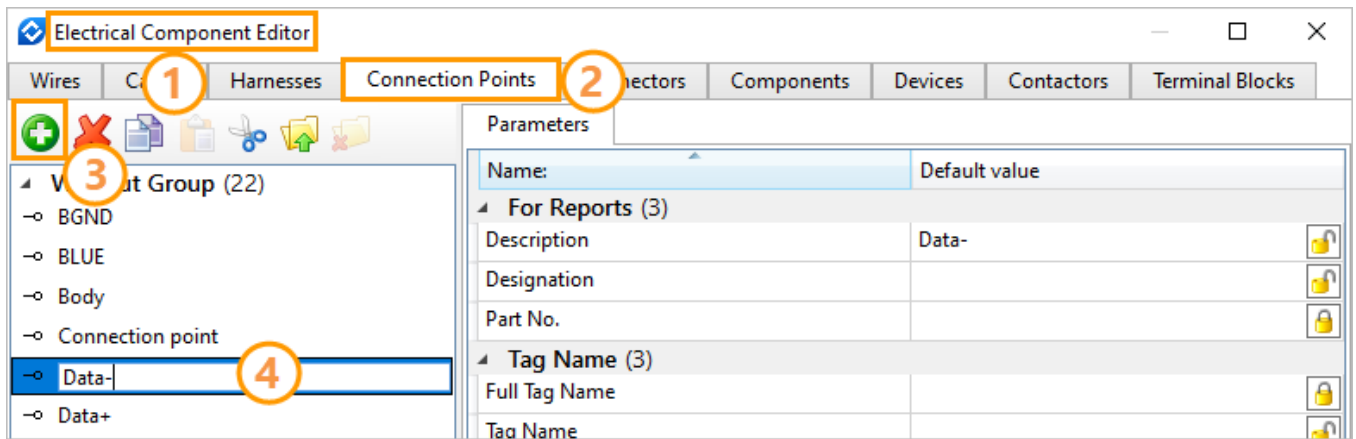
Continued: adding a cable to diagram.

Connection point prototype

Let's create a new prototype of **Connection Point** type "Data -".

The finished prototype can be found in **Component Editor**.

Call the command Product editor (1) and go to the Connection points tab (2). Click on the New item button (3). A new prototype (4) will be added to the list of prototypes, you need to specify its name ("Data -").



By default, parameter fields are protected from being changed, as indicated by the icon . If you click on the icon, it will change to , and the fields will become editable.

For a new connection point, you only need to fill in **Description**. Let's enter "Data -" in this field.

The rest of the parameters from **For Reports** and **Tag Name** groups do not need to be filled in for the connection point.

Additionally, you can specify:

Connection Type. You can select an option from the list if this CP will always has one type of connection, which must be displayed in wires table.

Terminal End Type. The terminal end can be selected from the list if this CP is always used with a terminal end of a certain type.

Additional parameters are described in more detail here.

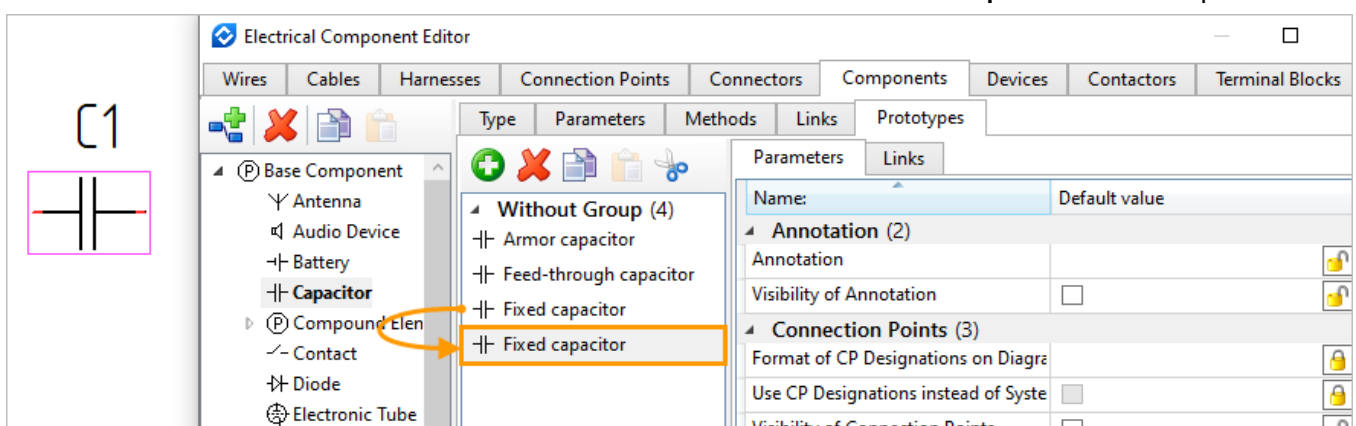
The connection point prototype is ready, it remains to save the changes in **Component Editor** by clicking [OK].

Save SE as Prototype command

Save SE as Prototype in Component Editor command allows you to add to **Component Editor** a prototype created on the basis of element available in the diagram.

This is relevant if the diagram contains an element whose parameters have been configured in the diagram, and you need to create other elements with the same parameters in the future. In this case, the original element can be either a library element or the one created in a context.

The command is launched from the context menu of an element, and **Component Editor** opens.




The tab of component of the corresponding type is active, and a new element is added to the prototype list, which should be renamed. By default, it is assigned the same name as the original

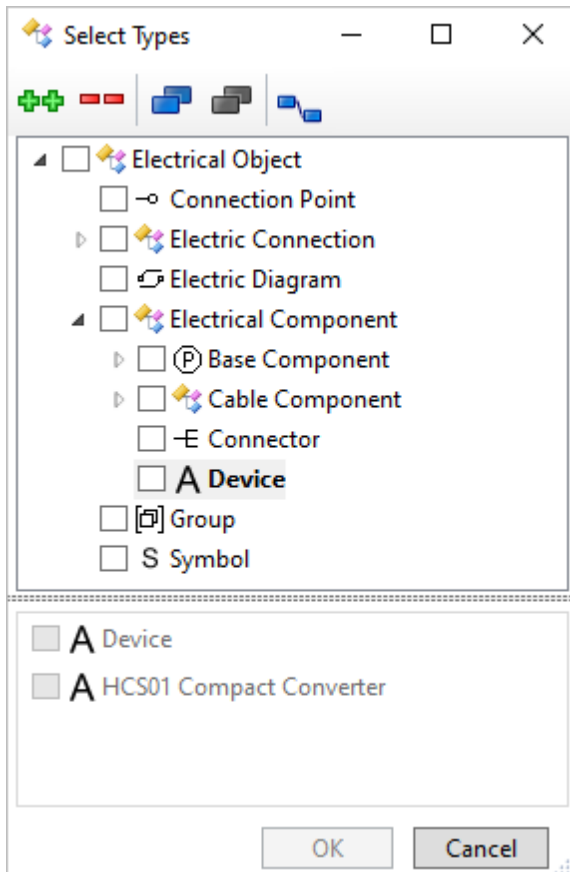
element. To rename, just double-click on the line with the name of prototype.

To save the prototype in **Editor**, click [OK] at the bottom of dialog.



Export and import of components



Users can share the content of **Component Editor**. To do this, use **Export Types** and **Import Types** commands, the icons of which are located in the lower part of **Component Editor** window.


Export Types  is intended for exporting the contents of **Component Editor** into a separate XML file. After launch, a window will open, at the top of which the type tree is displayed, and the corresponding prototypes - at the bottom.



Exported types and prototypes can be selected one at a time in the tree by checking the flags next to their names. The buttons above the tree are for group selection.

 button selects all types, and the prototypes related to them will not be selected. The button  cancels this action.

 button selects all prototypes, and the types to which they belong will not be selected. The button  cancels the action.

 button selects prototypes associated with the selected ones. For example, if a connector prototype is selected, the button will automatically select its associated connection points prototypes.

Thus, to select the entire contents of **Component Editor**, press  and .

After selecting the types, press [OK] and specify the name of XML file to which the components will be uploaded and select the folder to save it in Windows Explorer window.

Import Types  is intended for loading prototypes into **Component Editor** from an XML file.

After starting the command, a Windows Explorer window will open, in which you should select XML file from which the component data will be loaded and click [OK].

A dialog with a types tree similar to the one described above will open. The selection of prototypes for import is carried out according to the same principles using similar buttons.

When importing, you can configure how the system will process prototypes from an external source file whose names coincide with the names of prototypes in **Component Editor** (i.e. the current XML file, see details here). 4 processing options are available.

The settings described below affect only prototypes of the same name. Prototypes with names that differ from those already available in **Component Editor** will be simply added from an external XML file.

Skip Equal Objects. Prototypes from an external file with matching names will not be imported. It is advisable to choose this or the next option if the prototypes of the same name differ only in values of parameters, and the set of parameters in them is the same.

Overwrite Equal Objects. Prototypes from an external file will replace prototypes with matching names in **Component Editor**.


Combine Identical Objects (source → current). Parameters of prototypes of the same name from source file will be added to prototypes in **Component Editor**. Parameter values from an external file will replace the values of parameters of the same name in **Component Editor**. It is advisable to choose this or the next option if prototypes of the same name differ in a set of parameters.

Combine Identical Objects (current → source). Parameters of prototypes of the same name from source file will be added to prototypes in **Component Editor**. For the parameters of the same name, the values specified in **Component Editor** will be saved.

Update by Prototypes command

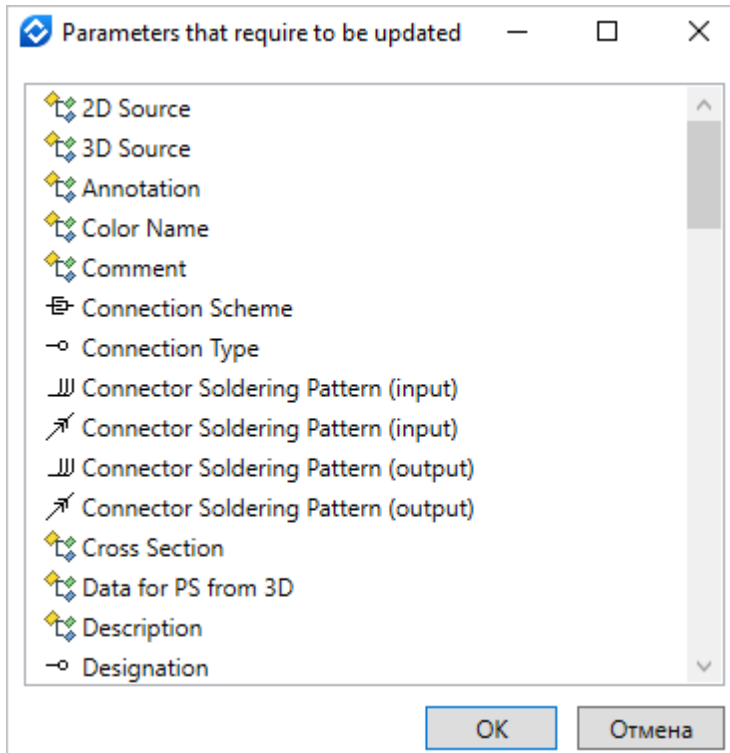
If prototypes have been changed in **Component Editor**, it may be necessary to transfer the changes to previously created grb files of components (for example, to library elements).

To do this, use the command:

Icon	Ribbon
	Electrical ? Prototypes ? Update by Prototypes
Keyboard	Textual Menu
	Electrical > Editors > Update by Prototypes

You must specify the folder with components files in Windows Explorer window in which the prototypes should be updated and click [Select Folder].


Then a window with a list of parameters will open. You can select the parameters that have been updated in it (some parameters may have been added, some may have changed their values). Selecting individual parameters in this window will speed up the update process. You can select all parameters using <Ctrl + A> key combination.



After clicking [OK], all files in the selected folder will be updated according to prototypes in **Component Editor**. If there are many files in the folder (as in the standard **Electrical Diagram Elements** library, for example), the update may take some time, since each file will be opened, recalculated and saved in the process.

Default Metadata command

To return the contents of **Component Editor** to the delivery state, use the command:

Icon	Ribbon
	Electrical ? Prototypes ? Default Metadata
Keyboard	Textual Menu
	Electrical > Editors > Default Metadata

After starting the command, the system will display a warning about the deletion of all user changes from **Component Editor**. To complete the command, click [Yes].

If the user has opened a diagram with elements whose prototypes are absent in **Component Editor**, the diagram will open correctly, the parameters of the elements will have the same values that they would have if all prototypes were present in **Component Editor**. However, you will not be able to create new elements with prototypes missing in the **Editor**. In this case, you need to use **Export and Import Types** commands.

Creating components of electrical diagrams

Base component

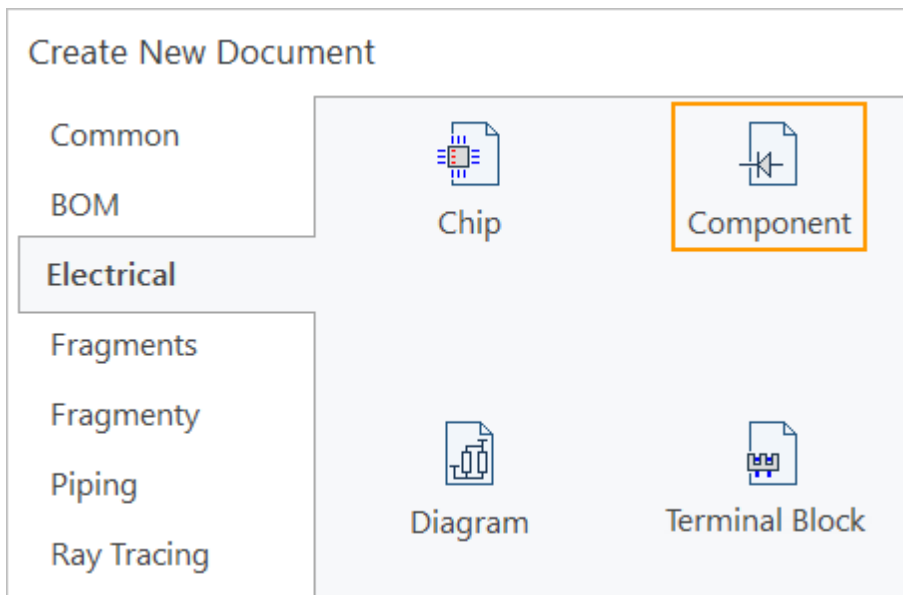
Graphic

The base component is the only type of component for which you need to create your own graphic symbol and "manually" create connection points in it. In this regard, its creation is somewhat different from the creation of other elements. The base component can only be created in the prototype, it cannot be created in the context of diagram.

A base component can be created to be self-inserted into a diagram (such as a resistor or light source). Also, its creation may be required for use as an element of a composite product (terminals of a terminal block, device contact), if the elements of the standard "Electrical diagram elements" library do not meet the requirements.

To create a base component, you need to create its graphic symbol, create connectors - future connection points, connect the graphic with a structural element in **Create Component** command.

To create a base component, use **Component** prototype on **Start** page (Electrical group).



You can also use **Create Component** command, which automatically creates a new document based on **Base Component** prototype.

Icon	Ribbon
	Electrical ? 2D Elements ? Create Component
Keyboard	Textual Menu
	Electrical > Elements > Create Component


It is important to create electrical components in the corresponding prototypes of documents. Special macros have been added to prototypes, and a special library is connected to them, which ensures the correct operation of elements in the diagram.

Graphic symbol of element is a parametric drawing based on construction lines. Lines must be linked to variables that have been created in advance in prototypes of electrical diagram elements:





Angle controls the native rotation angle of the element (in the diagram, the value of variable is changed using button in the automenu).


AngleAssembly takes the value of the rotation angle from the "top level" (it is important for the correct rotation of terminals in terminal block, contacts in the connector, etc.).

A ensures the correct rotation of the component by taking into account its own rotation angle and the rotation angle of the top-level element.

Symmetry controls the symmetric display of the element (the value of variable in the diagram is changed using  button in the automenu)

Step ensures that the connection points coincide with the grid points (if they do not match, it will be impossible to connect to them).

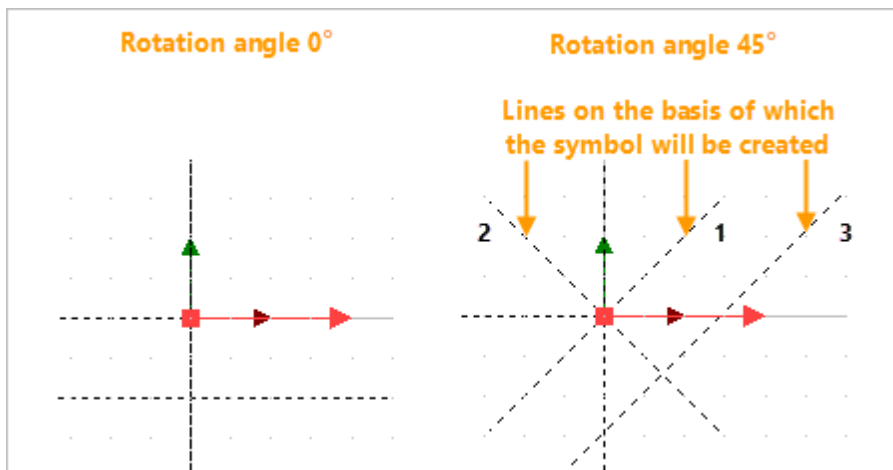
Variables				
Name	Expression	Value	Comm...	
Group:				
	Angle	0	0	
	Symmetry	0	0	
	Step	2.5	2.5	
	AngleAssembly	0	0	
	A	Angle+AngleAssembly	0	

All variables, except for **A**, are external ones (marked with  symbol), since changing the angle, symmetrically reflecting and changing the grid step is required from an external document (diagram).

Let's consider the creation of a graphic symbol of transistor n-p-n.

The finished component can be found in the library "Electrical diagram elements / Active / Transistors".

Five construction lines are pre-created in **Component** prototype. Let's set **Angle** variable to 45 degrees to see them. For convenience, we will designate the rotated construction lines as the first, second and third ones.



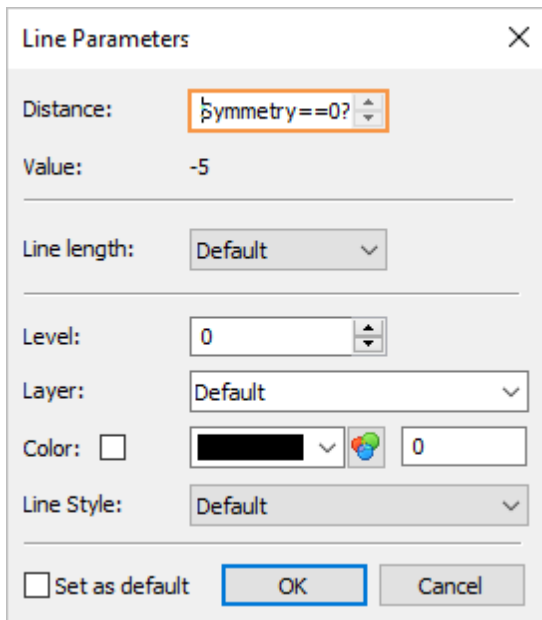
The two lines remaining in their places are the vertical and horizontal lines passing through the origin of the drawing coordinate system.

1 - the line directly controlled by **Angle** variable. The horizontal axis of the future graphic symbol.

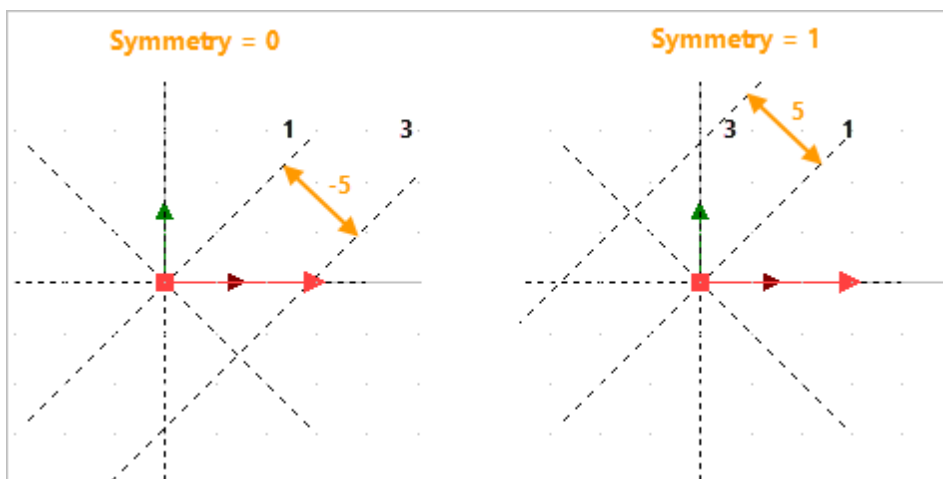
2 - a line drawn to the first at an angle of 90 degrees. Vertical axis of the future symbol.

3 - line directly controlled by **Symmetry** variable. Will be used for symmetric reflection of symbol, also linked to the Step variable.

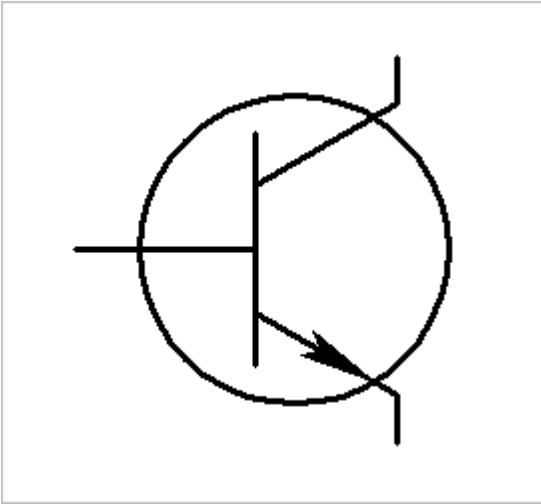
To correctly build the drawing, it is necessary to create it with reference to the three lines described above. With a default rotation of 0 degrees, two of them coincide with the vertical and horizontal lines. To eliminate incorrect snapping and make it more visual, we rotated the drawing by 45 degrees. Let's take a closer look at line 3 by calling the dialog of its parameters from the context menu. The expression is specified for it In Distance window: $\text{Symmetry}==0?-2*\text{Step}:2*\text{Step}$. The expression specifies the distance at which the third construction line is postponed from the first one.






The expression means the following: if the value of **Symmetry** variable is 0, the distance between the third construction line and the first one will be $-2 \times$ the value of **Step** variable. Otherwise, the construction line will be located at a distance $2 \times$ the value of **Step** variable. The default value for **Step** is 2.5.



In the first case, construction line 3 is located at a distance of -5 , in the second - at a distance of 5 . Let's start creating a drawing. We will need to get the result like in the image below.



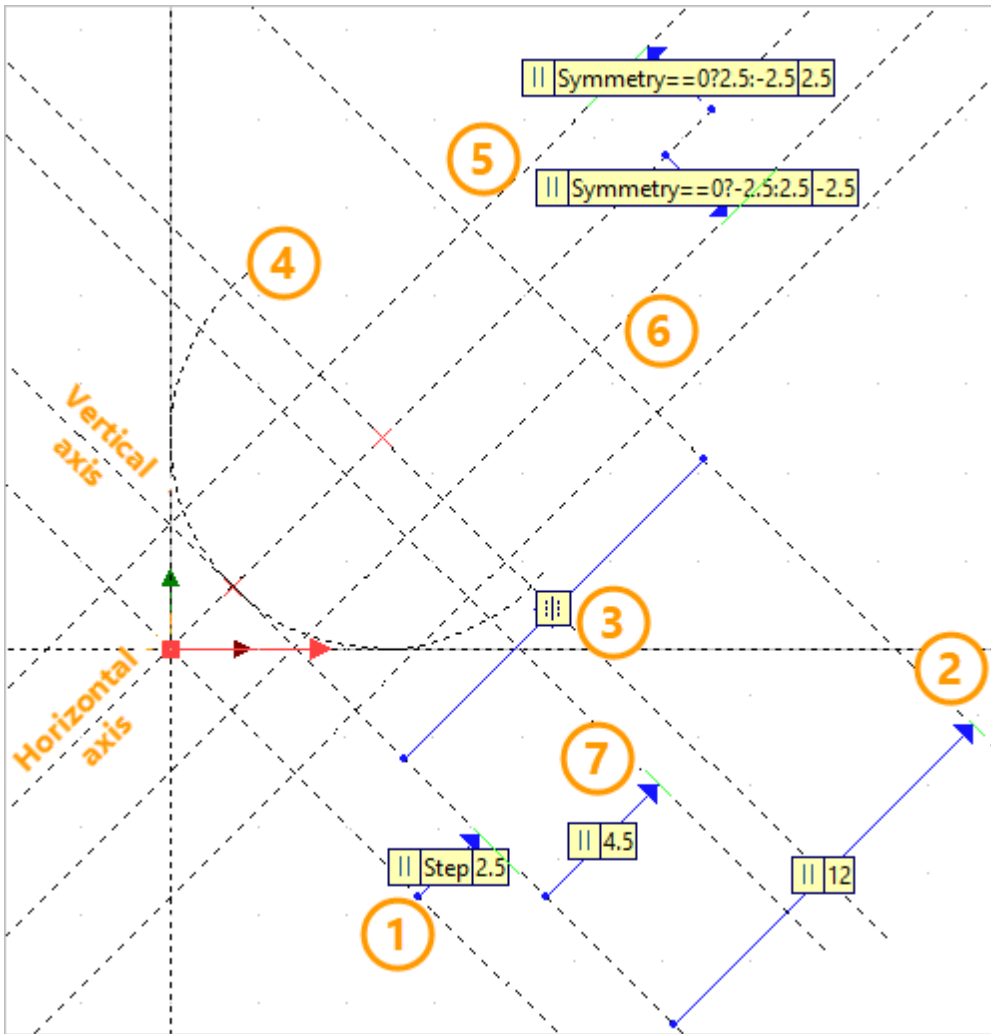
To create construction lines use the command  **L: Line**, graphic lines - command  **G: Graphic Line**, nodes - command  **N: Node**.

Let's draw a line (1) at a distance **Step** from the vertical axis (line 2 in the picture above).

Line (2) will be drawn at a fixed distance of 12 mm: it is needed to draw a circle, which itself is not connected to the connection points. Line (3) will be constructed as an axis of symmetry, it will allow us to find the center of the future circle. Draw a circle (4) from the point of intersection of line (3) and the horizontal axis (line 1 in the picture above).

Lines (5) and (6) are needed to create the emitter and collector lines, and it should be possible to swap these lines using **Symmetry** variable. In this regard, write the expressions in **Distance** field discussed above. In the expressions, opposite signs are used, since the distances from the horizontal axis to lines (5) and (6) must be equal, but created in opposite directions.

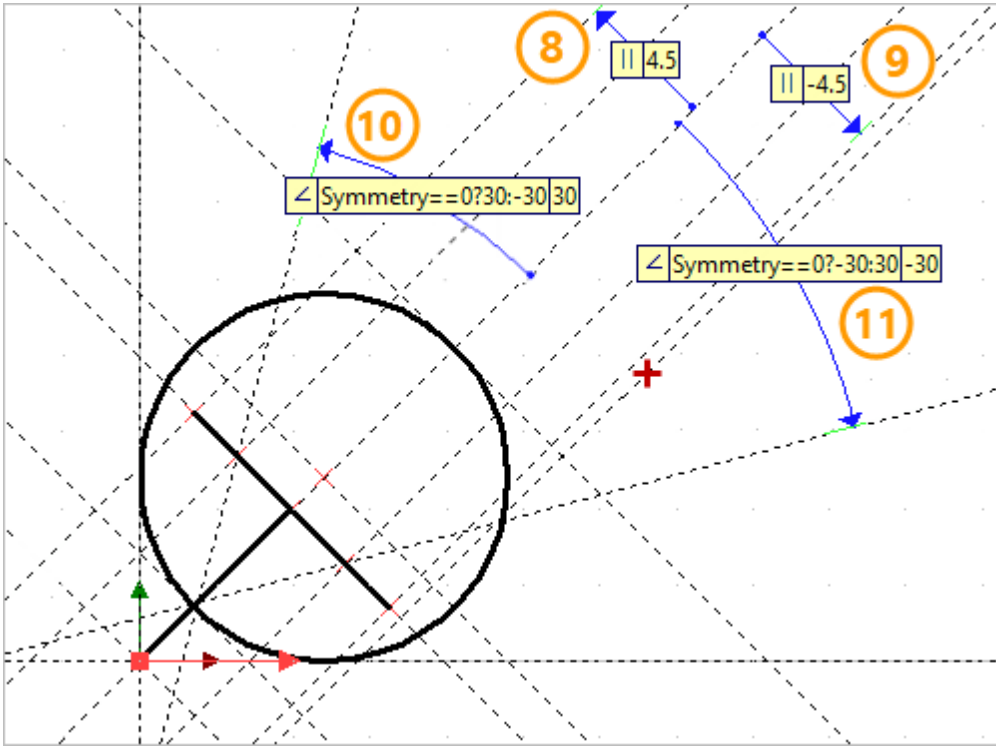
Line (7) is drawn at a fixed distance of 4.5 mm from the vertical axis.



Lines (8) and (9) will be drawn at a distance of 4.5 and -4.5 from the horizontal axis of the symbol.

Lines (10) and (11) will be drawn at an angle of 30 and -30 degrees to the horizontal axis, linking them to **Symmetry** variable.

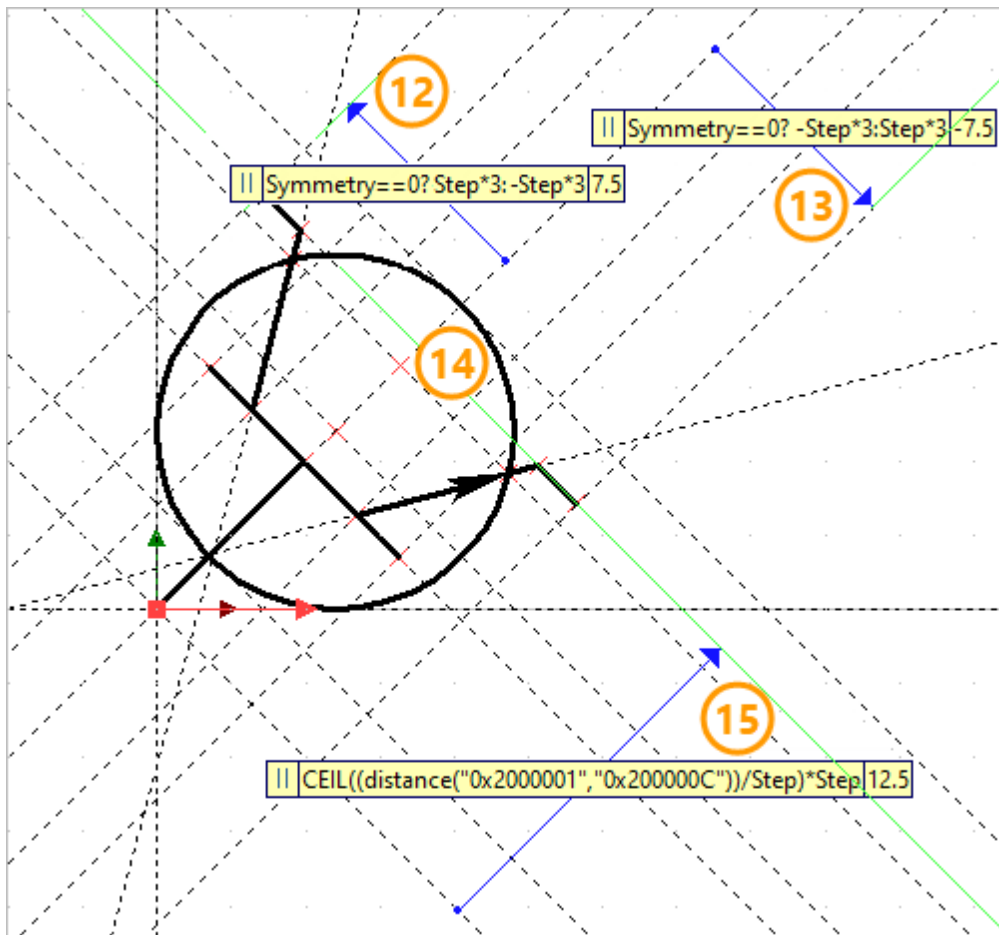
The line originally created in the prototype to demonstrate how **Symmetry** variable works will be removed as unnecessary (marked with a red X in the figure).



Lines (12) and (13) define the position of the connection points on the emitter and collector, so their distances must be linked to **Step** variable, and they must also be swapped depending on the value of **Symmetry** variable.

Line (14) is drawn through 2 nodes created at the intersection of lines (10) and (11) with a circle (4). This line will allow you to correctly draw the arrow of the emitter. At the point of intersection of line (14) with the horizontal axis, create an additional node.

Line (15) defines the position of the connection points on the emitter and collector, so its distance must be related to **Step** variable. There should be a small distance between lines (14) and (15), and (15) should intersect the grid line. To calculate the position of line (15), measure the distance from the node at the coordinate system origin to the node on line (14) using the distance function (alphanumeric codes in brackets are system IDs of nodes, they can be found in **Information** dialog). Divide this distance by **Step** and round it up using **CEIL** function. We get an integer, which we multiply by **Step**. Thus, line (15) will be positioned slightly further than line (14), and coincide with the grid line.



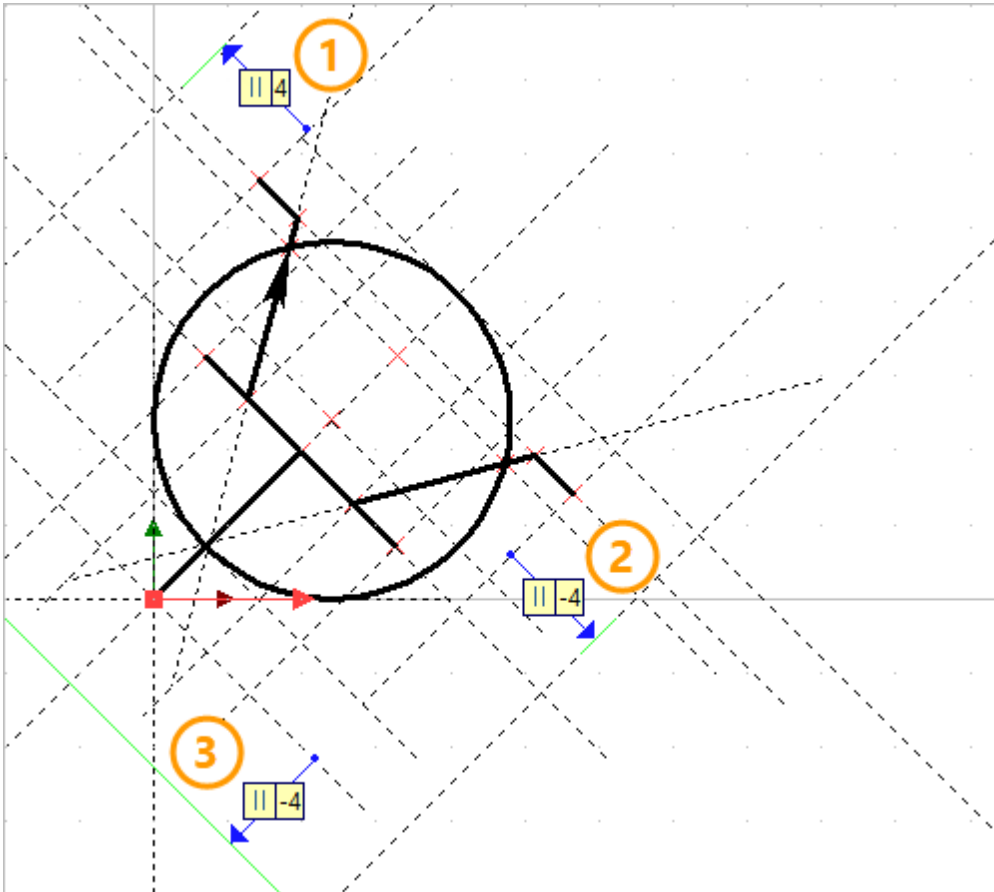
We outline the drawing with graphic lines. Draw the emitter line with two segments, placing the arrow at the end of one of them.



The correctness of the created drawing must be checked by changing the values of variables.

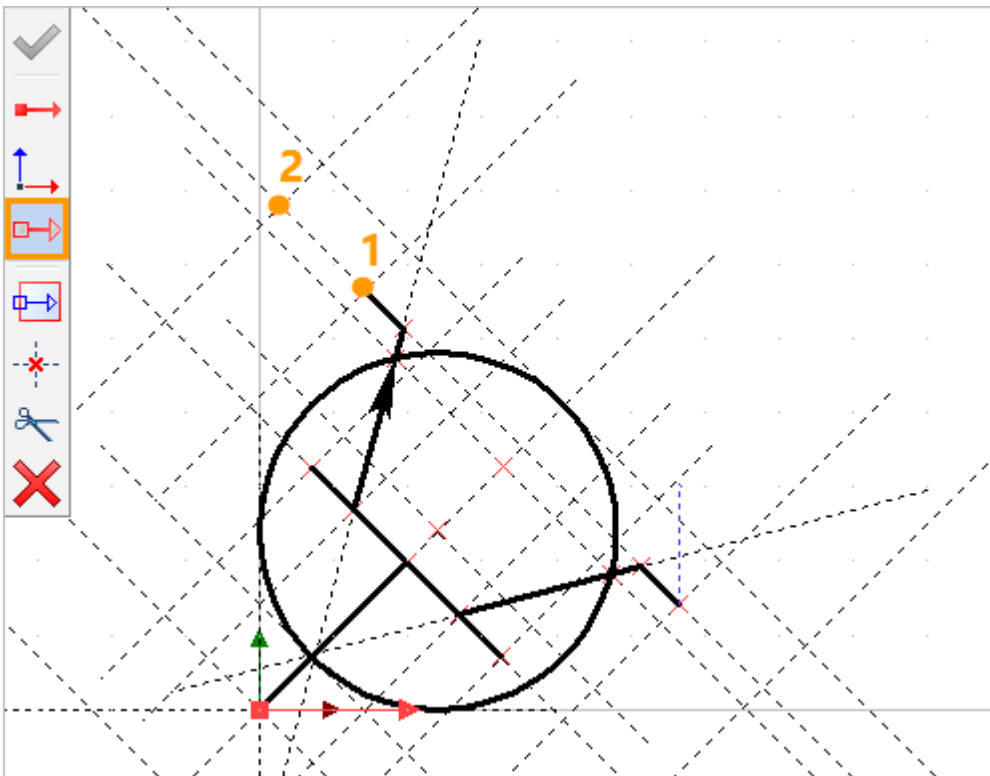
Creating pins

Let's add 2D connection points to the drawing. It is recommended to rotate the drawing by 45 degrees (see the reason here).

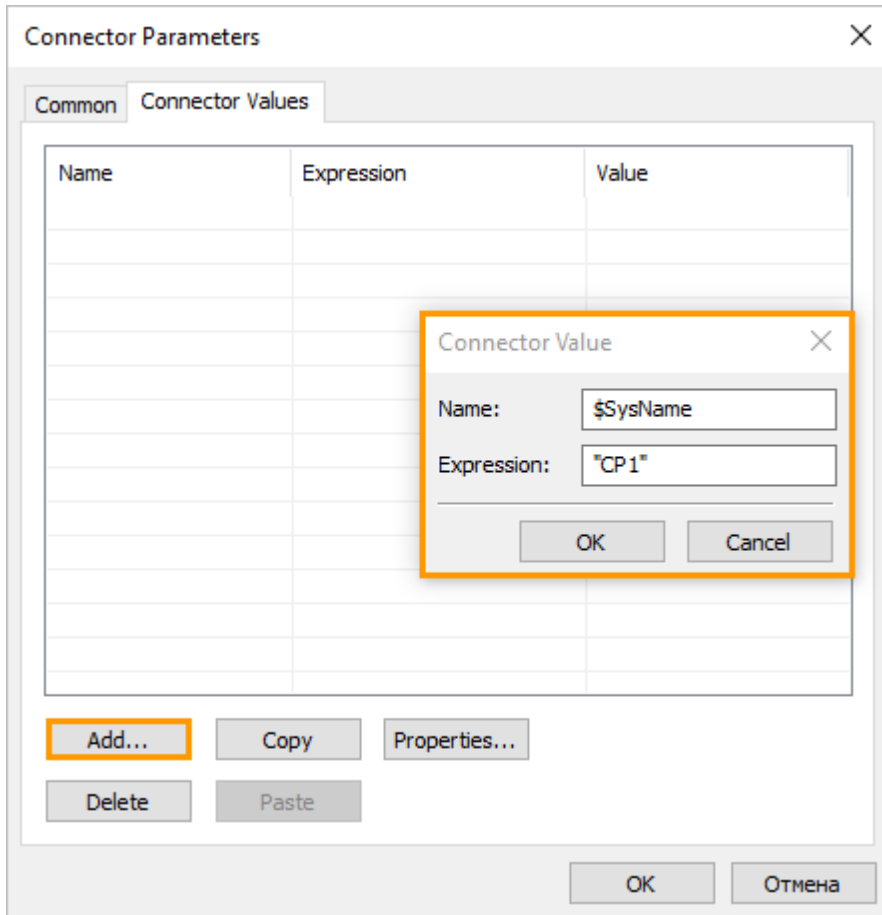
The pins in base component are 2D connectors by their nature. 2D connector is a vector and requires two points to create. Let's set aside lines from the end points of the base, emitter and collector lines at an arbitrary fixed distance.



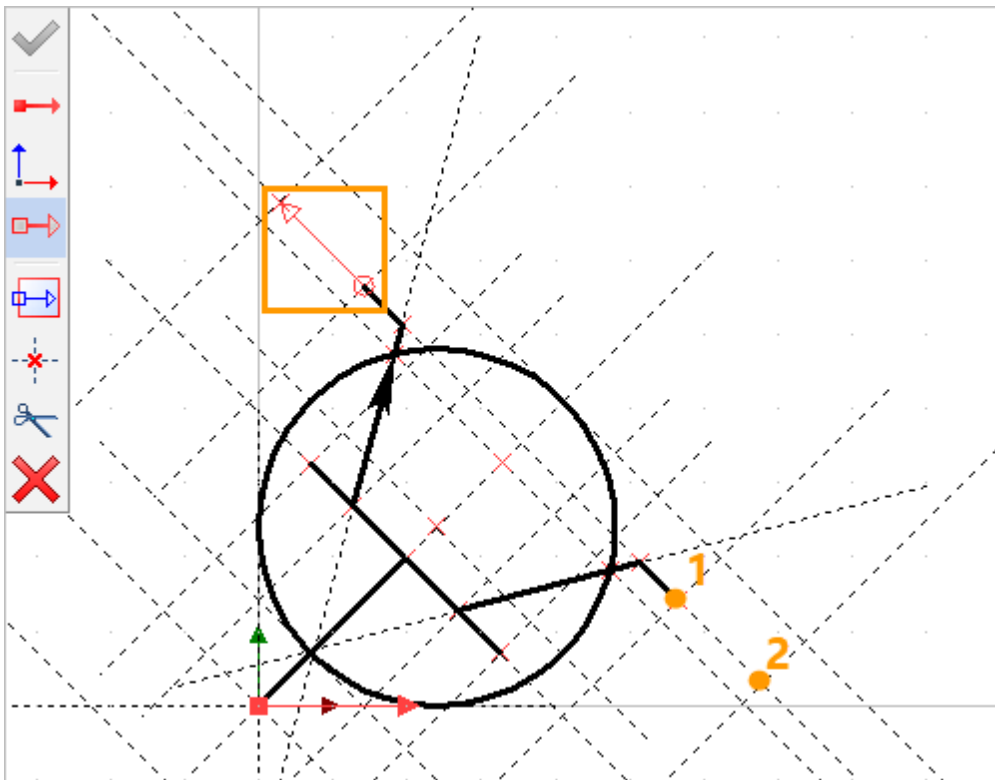
Let's call the command  **FV: Fixing Vector** and select the automenu option  **Create connectors**. Select 2 points of the future connector (first you need to select a point near the image line), after which a dialog of its values will open.



You need to click [Add], and enter **Name** - \$ SysName, **Expression** - "CP1" (in quotes) in the window that opens, and then click [OK] twice. This recording format allows the system to recognize that this connector is the connection point for an electrical component.



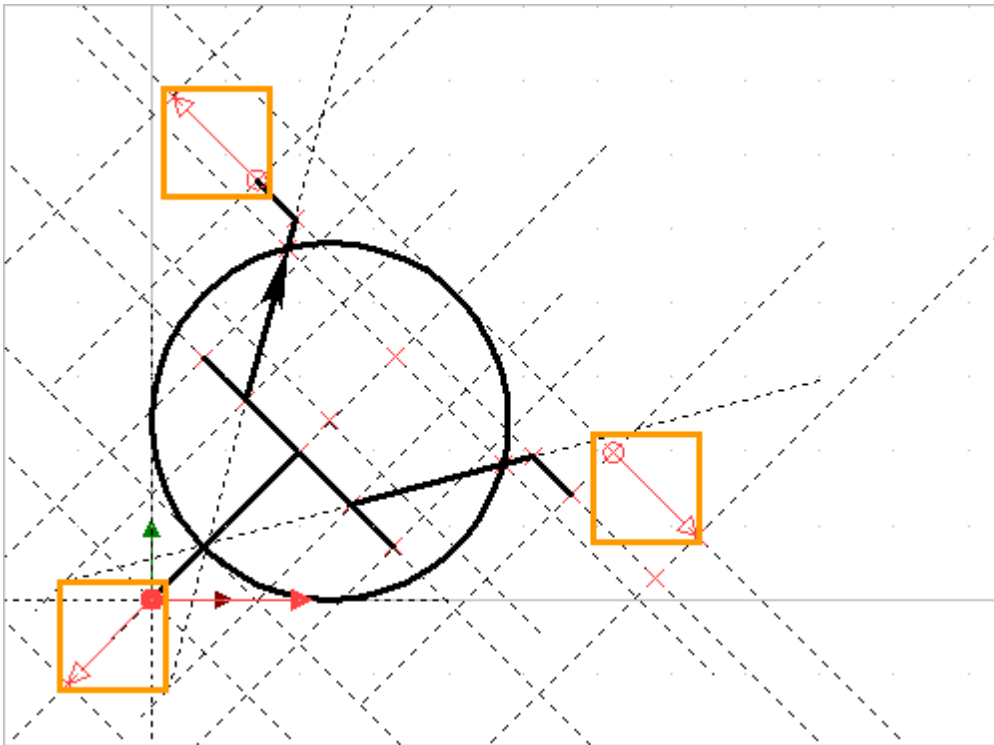
The created connector appears in the drawing. Without leaving the command, select the next pair of connection points.



The second connector will have the same **Name** and **Expression** "CP2" since the points are numbered in order from 1.

After creating the third connector (with the value "CP3") and exiting the command, we get the

following result.



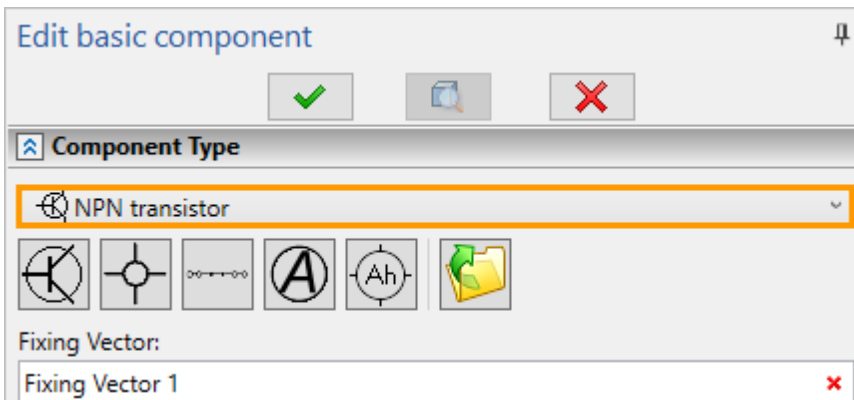
Creating component

After creating the graphic symbol and connectors, run **Edit Component** command to create a link between the graphics of component and its prototype.

Icon	Ribbon
	Electrical ? 2D Elements ? Edit Component
Keyboard	Textual Menu
	Electrical > Elements > Edit Component

Component Type section is used to select a prototype of component from **Component Editor**.

You can select it from the drop-down list or from the complete list of prototypes called by **Select from dialog** button. The last selected prototypes are shown on the icons below the drop-down list. The creation of n-p-n transistor prototype is described here.

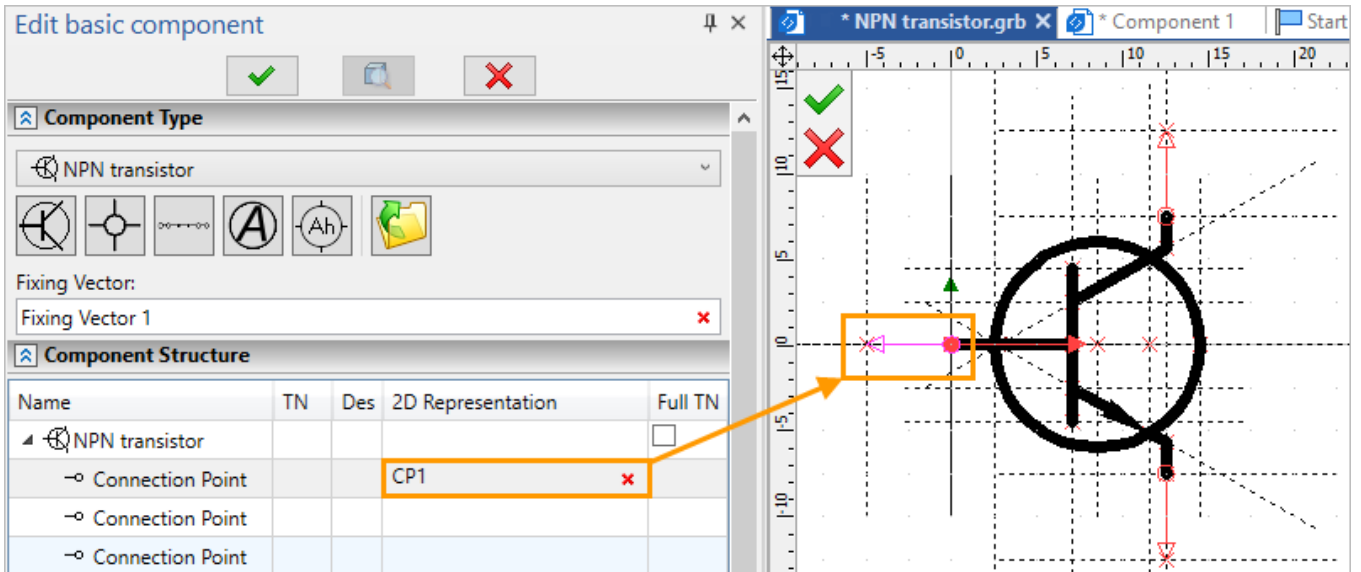


Component Structure section displays the contacts of the selected prototype.

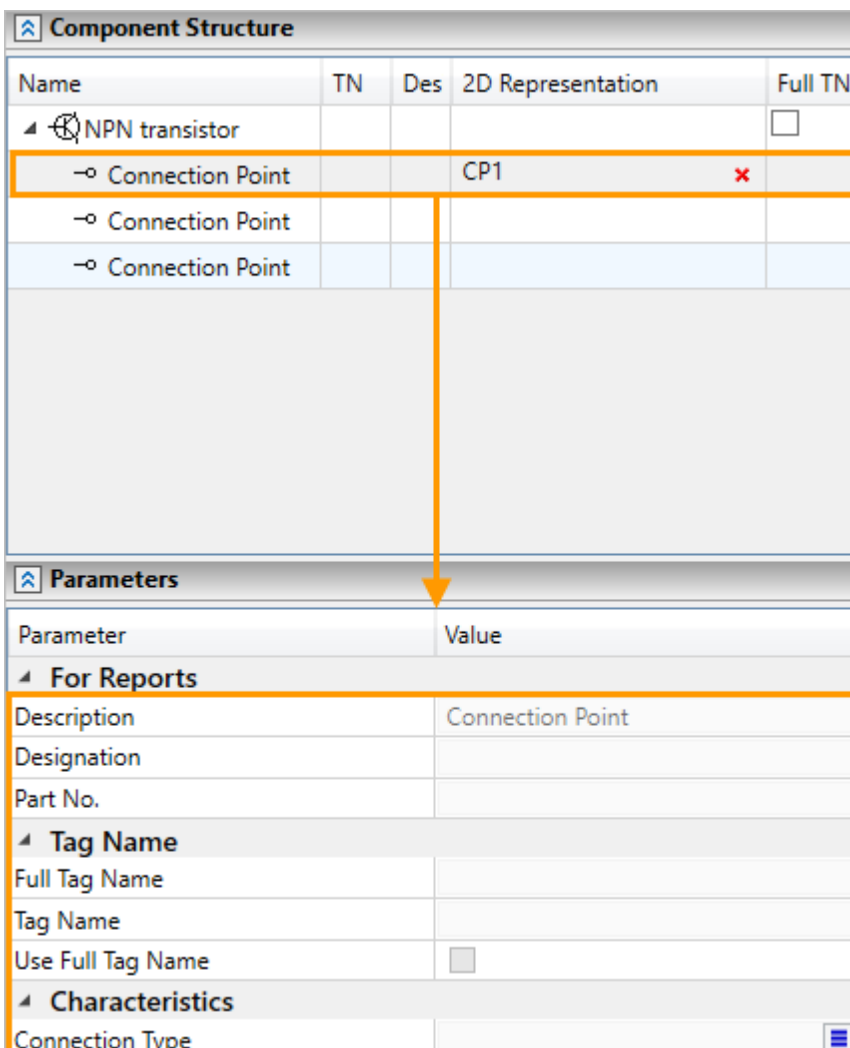
You need to match the connection points in the list with the connectors that were created in the previous step. To do this, place the cursor in **2D Representation** cell and select the connector on the drawing corresponding to the first connection point. Similarly, set the correspondence to the rest of

component pins.


To deselect a connector, use **x** icon.



Parameters section displays the parameters of element that is selected in **Component Structure** section. This means that the parameters of transistor or connection point will be displayed, depending on which line of **Component Structure** section the cursor is positioned in.



The parameters of base component are described in more detail here and here.

When the correspondence between connection points and the connectors is set, click  in the automenu or at the top of the dialog.

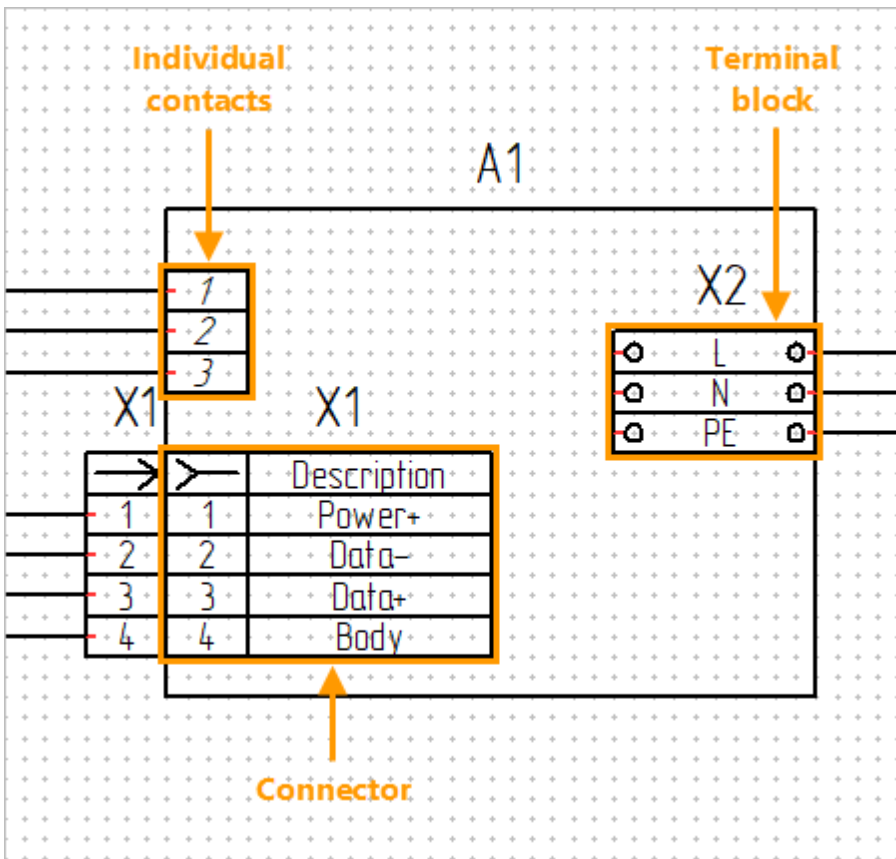
After saving the document, the component can be used as a diagram element.

Device

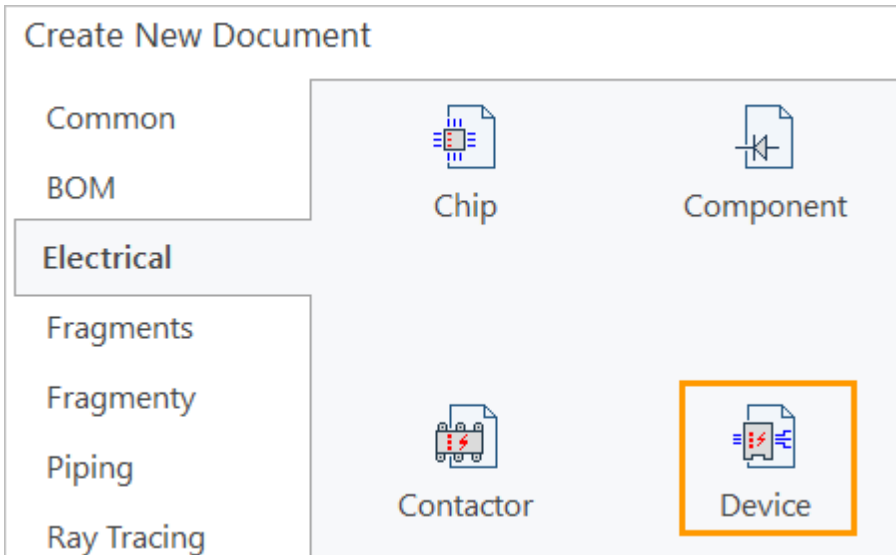
Graphic symbol

A device in T-FLEX Electrical terminology is a composite device, which can include individual contacts and components of different types. The device can be created in a prototype and in context of diagram.


The image of the device on diagram consists of a rectangular frame and graphic symbols of elements that are part of it. The figure below shows an example of a device that includes individual contacts, a terminal block and a connector.



To create the device, the prototype of the same name on **Start** page is used (Electrical group).



You can also use **Create Device** command, which automatically creates a new document based on **Device** prototype and launches the mode of structure edit.

Icon	Ribbon
	Electrical ? 2D Elements ? Create Device
Keyboard	Textual Menu
	Electrical > Elements > Create Device

It is important to create electrical components in the corresponding prototypes of documents. Special macros have been added to prototypes, and a special library is connected to them, which ensures the correct operation of elements in the diagram.

Graphic symbol of device itself is a parametric drawing (rectangular frame) based on construction lines. Variables are used to adjust the size of the frame:

Contact_hor controls the width of the border by specifying the number of red vertical lines within it.

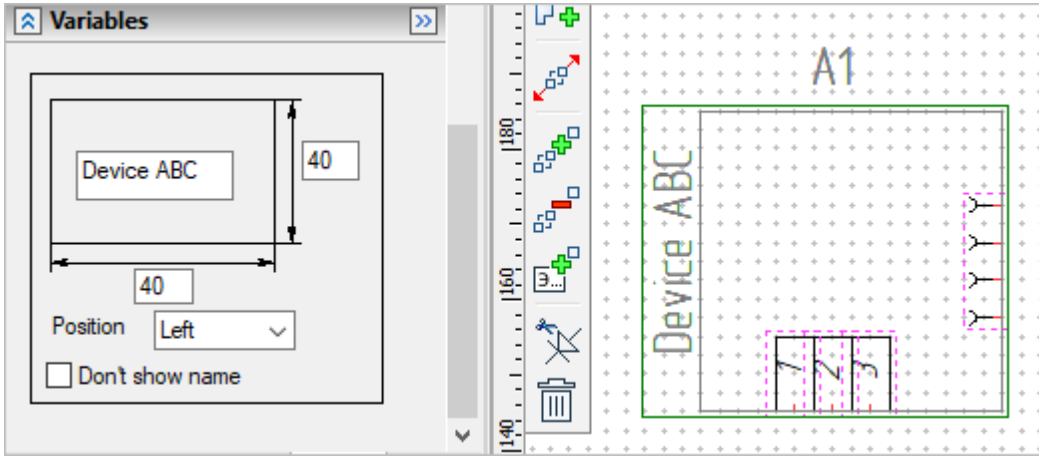
Contact_vert controls the height of the device by setting the number of red horizontal lines inside it.

Variables				
Name	Expression	Value	Comment	
Group:				
A	Angle+AngleAssembly	0		
Angle	0	0		
AngleAssembly	0	0		
Contact_hor	7	7		
Contact_vert	9	9		
Step	2.5	2.5		
TextLength	0	0		
hor	$2 * \text{Step} * (\text{Contact_hor} + 2 + (\text{C}...$	75		
n1_hor	$\text{ceil}(\text{Contact_hor} / 2)$	4		
n1_vert	$\text{ceil}(\text{Contact_vert} / 2)$	5		
n_hor	$\text{floor}(\text{Contact_hor} / 2)$	3		
n_vert	$\text{floor}(\text{Contact_vert} / 2)$	4		
vert	$\text{Step} * (\text{Contact_vert} + 2 + (\text{Con}...$	47.5		

Variables **A**, **Angle**, **AngleAssembly**, **Step** are described here.

The rest of variables in the document are auxiliary and are used to draw the frame.

When the device is inserted into the diagram and selected, the frame settings are displayed in **Variables** section.



The input field in the center of the frame is intended for specifying the device's signature on the diagram.


Position parameter determines on which side of the frame the caption will be placed.

Don't show name checkbox allows you to hide the device signature on the diagram.

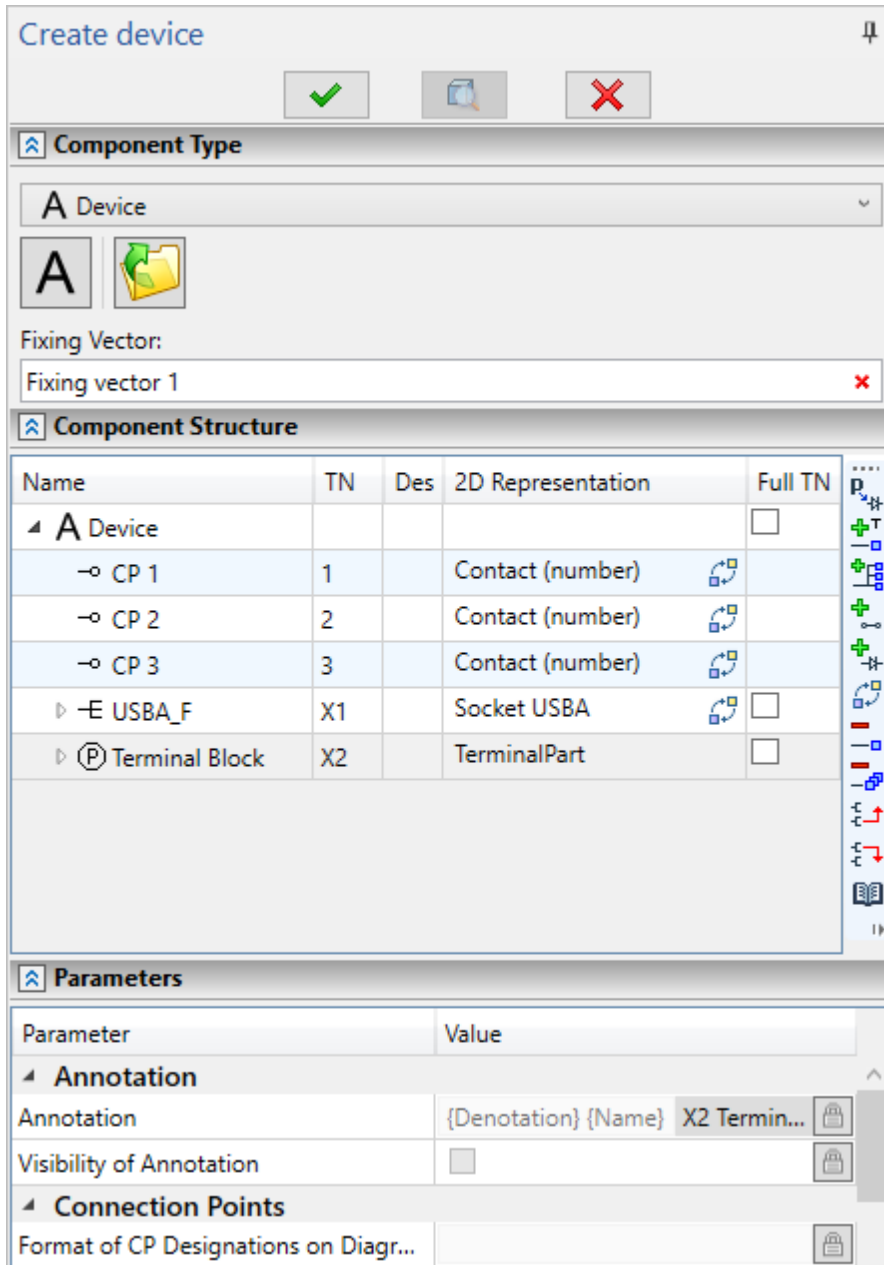
The input boxes next to the dimension arrows control the size of frame in the diagram. Dimensions are set in millimeters (the value in the input field is equal to **Step** grid multiplied by the number of vertical or horizontal steps).

Create device

After setting the frame size, you need to run **Edit Component** command.

Icon	Ribbon
	Electrical ? 2D Elements ? Edit Component
Keyboard	Textual Menu
	Electrical > Elements > Edit Component

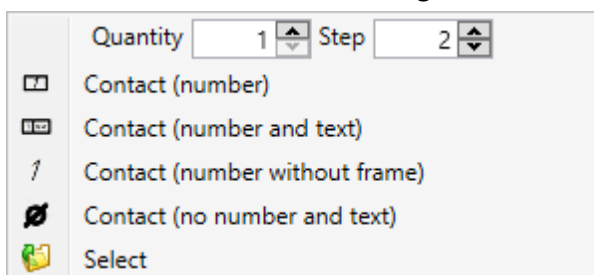
The type of component in the document is determined automatically, and the system opens the dialog for editing the component structure.



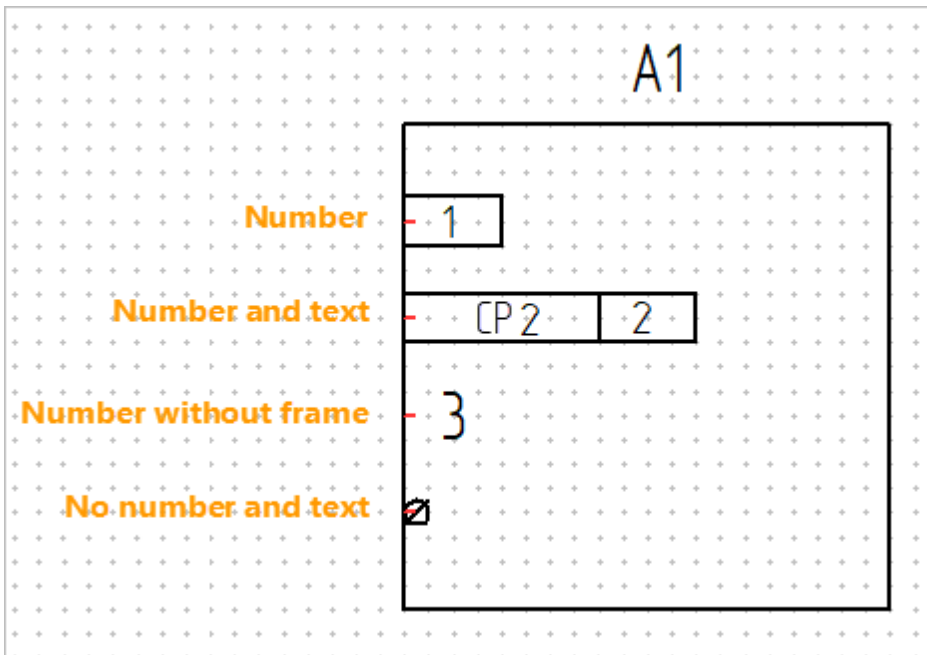
In **Component Type** block, select the prototype of the device from **Component Editor**. By default, the basic prototype of device is selected. button opens a list of all device prototypes created in Editor.



In **Component Structure** block, the elements that make up the device are displayed. It is empty by default. To add elements to it, use the mini-menu buttons:


Add connection point allows to add single pins. After clicking, a drop-down menu will open, in which you can set **Quantity** of contacts and **Step** (it means the number of grid steps between adjacent contacts, it matters when adding several CPs at once).





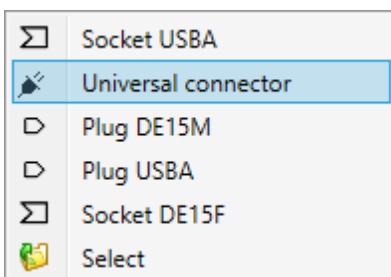
Then, in the menu, you need to select a fragment of contact in the device. The contact will follow the cursor and must be placed on the page.



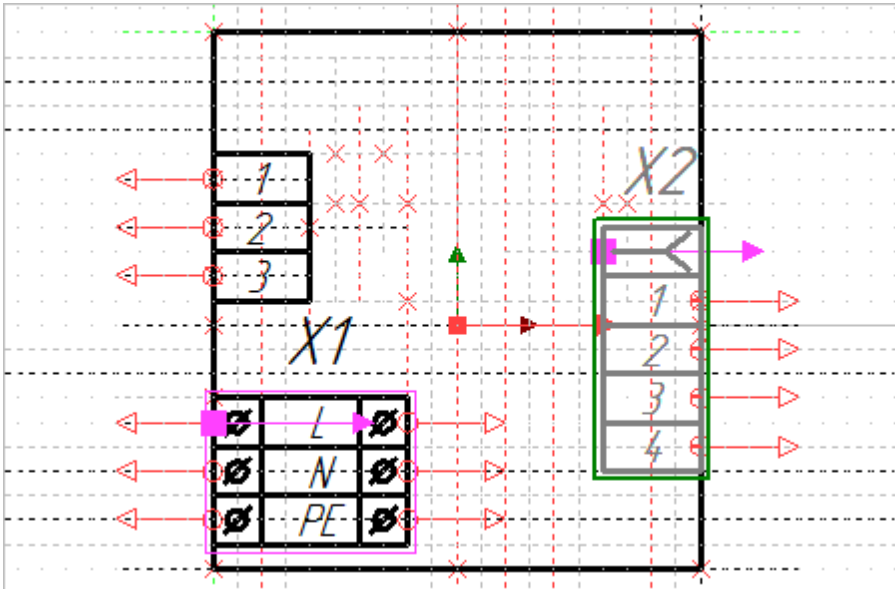
The connection points are marked with red markers, and the contact must be oriented so that the connection line can be connected directly to the marker without crossing the device's symbol. To ensure the correct orientation of the contact, you can use the buttons  **Mirror** and  **Orientation** in the automenu. They are available when a contact has already been selected, but not yet placed, as well as after placing a contact, if it is selected in **Component Structure** block.

 **Set 2D representation** allows you to replace one type of contact with another. For example, if **Contact (number)** is added to the device, this option can be used to replace it with **Contact (number and text)**. Group replacement is also possible (if several contacts are selected in **Component Structure** block using <Shift> or <Ctrl>).

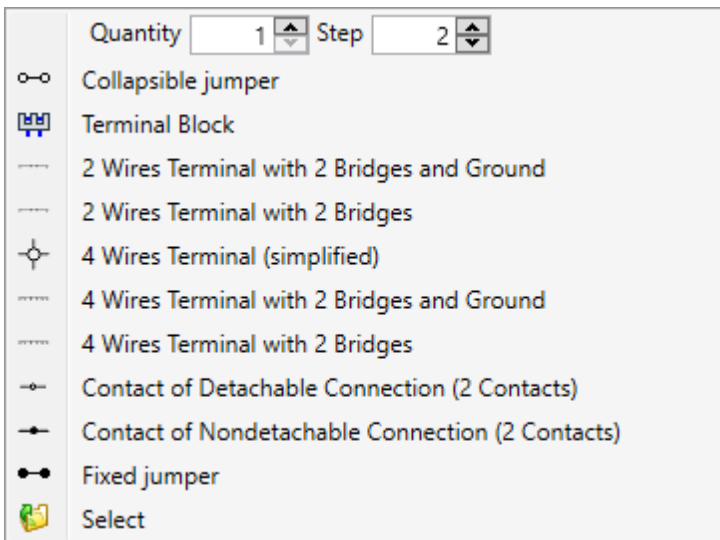
 **Add connector** is used to add connectors to the diagram. After clicking, a drop-down menu will open in which you need to select one of the connectors added to the library. To open a complete list of connectors, click on  **Select** button.



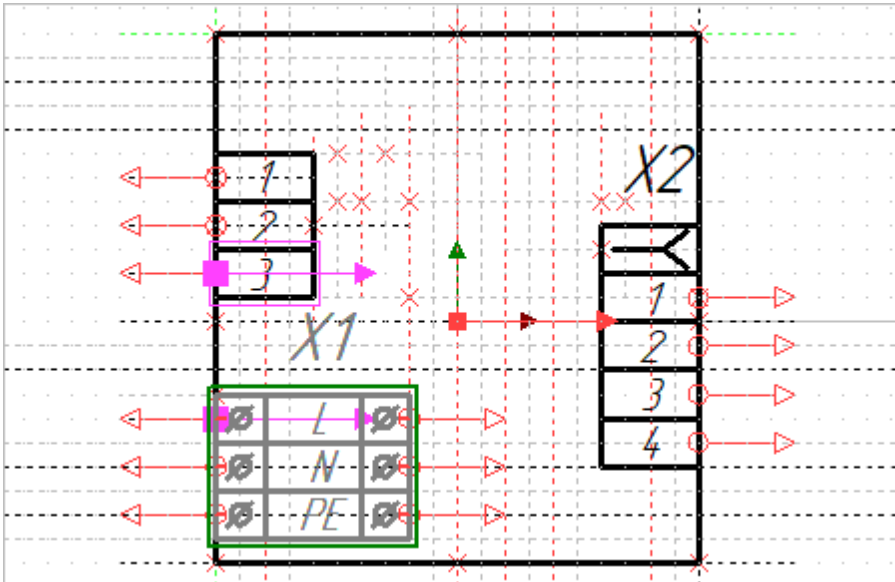
The connector will follow the cursor and must be placed on the page. The principle and possibilities of orientation of connector in the device are the same as for the individual contacts (see above). It should be borne in mind that for connectors from the library, contacts are placed on the column with contact numbers.



Add terminal block is used to add terminals and terminal blocks to the device. After clicking, a drop-down menu will open, in which you can set **Quantity** of terminals and **Step** (it means the number of grid steps between adjacent terminals, it matters when adding several terminals at once).

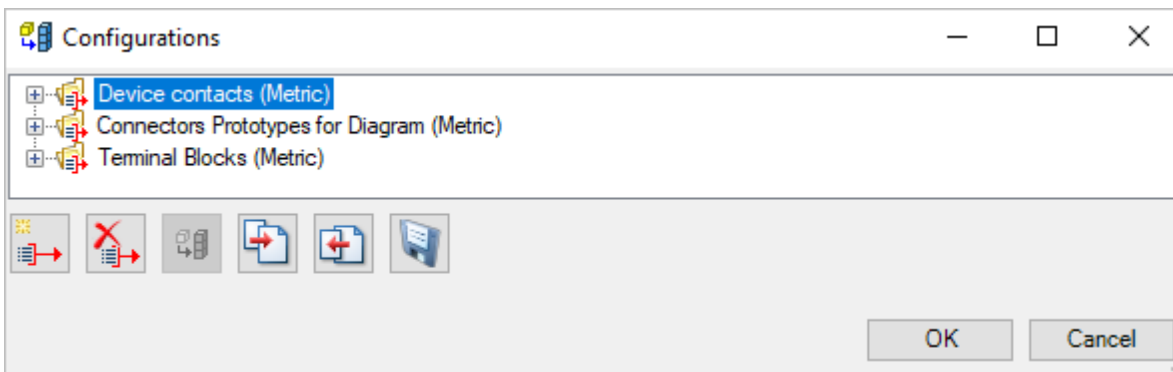


Then you need to select one of the components added to the library. The element will follow the cursor and must be placed on the page. The principle and possibilities of orientation of terminals and terminal blocks in the device are the same as for the individual contacts (see above).



+ Add component is used to add basic components to the device. In the drop-down menu, you can also set **Quantity** of elements and **Step** between them.

The folders from which the system borrows terminals, connectors, pins and contacts are configured in **Library Configuration** command. This command is run in transparent mode in abbreviated form by **Library Configuration** button in the mini-menu. In the window that appears, you can see from which folders certain elements that may be part of the device are taken. For more information on setting up configurations, see here.



- Delete is used to delete selected elements in **Component Structure** block. You can select one element or several using <Shift> or <Ctrl>.

- Delete all is used to delete all elements in **Component Structure** block.


The information in **Parameters** block is context sensitive, that is, it changes depending on which item is selected in **Component Structure** block. The parameters of the device are described in more detail here.

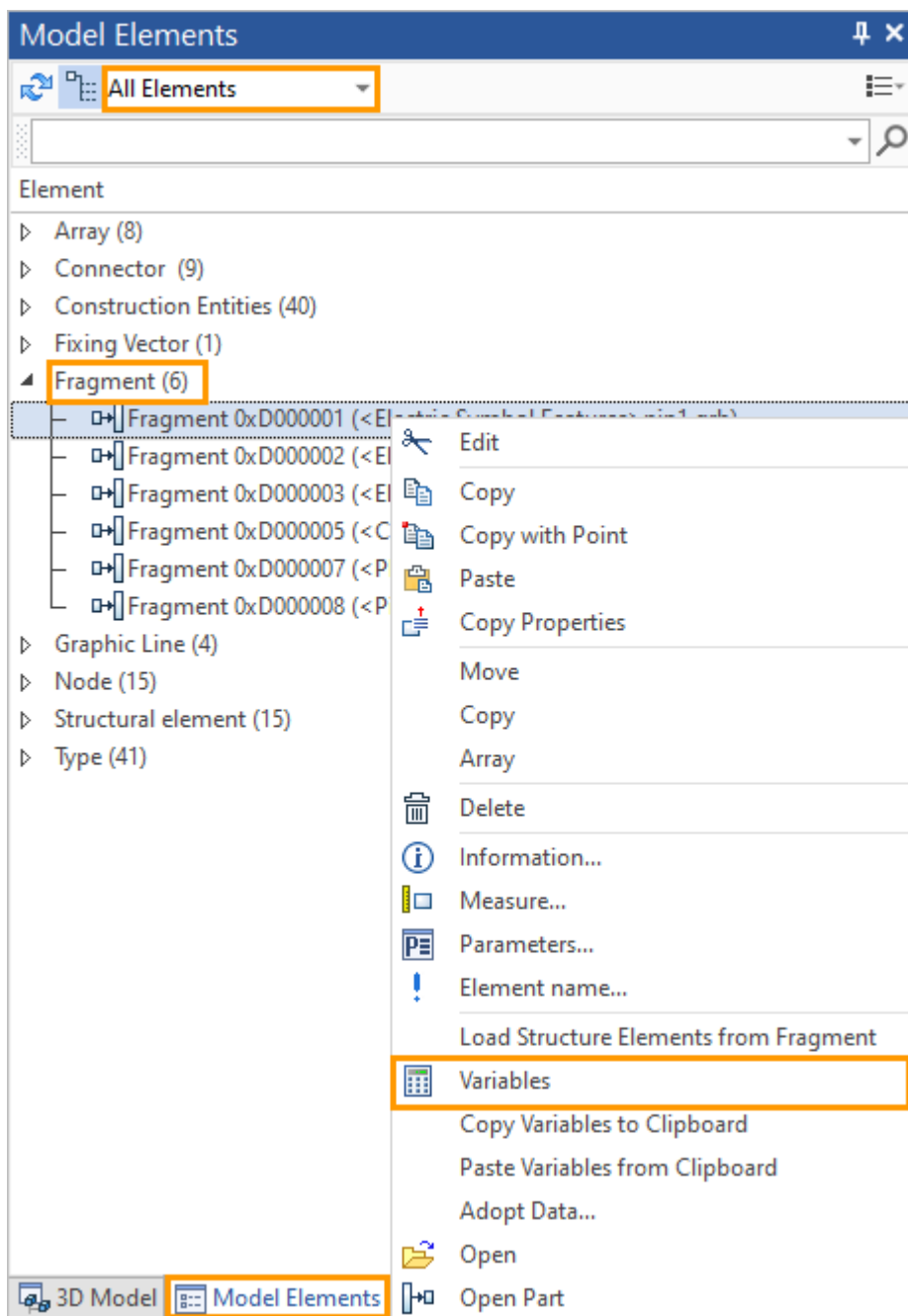
When the device is full of components, press **✓** in the automenu or at the top of the dialog.

After saving the document, the device can be used as a diagram element.

Elements added to the device can be additionally configured using **Variables** dialog. To call this dialog for connectors and terminal strips, open the context menu and select **Variables**. For connectors, you can configure the visibility of columns, for terminal strips - the column width (variable **TextLength**), visibility of terminal number (variable **Pos_vis**) and visibility of terminal names (variable **Head_vis**).

To call a dialogue with individual contacts, open **Model Elements** window, select **All elements** option,

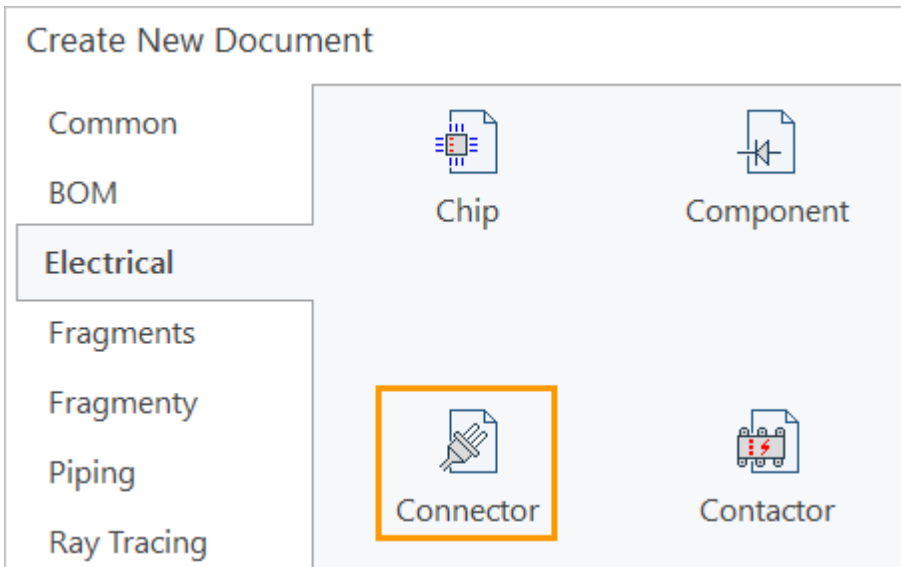
find the required element in **Fragment** folder and call  **Variables** command from the context menu. In the dialog that opens, you can correct the contact number (variable **\$Position**), visibility of the contact name (variable **Text_vis**) and contact number (variable **Pos_vis**).



Connector


Create connector

To create a connector, use the prototype of the same name on **Start** page (**Electrical** group).

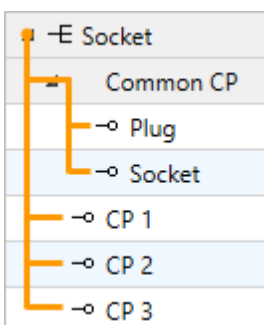



After opening the document, click  **Edit Component** in the Ribbon.

You can also use **Create Connector** command, which automatically creates a new document based on the **Connector** prototype and runs the structure editing command.



Icon	Ribbon
	Electrical ? 2D Elements ? Create Connector
Keyboard	Textual Menu
	Electrical > Elements > Create Connector

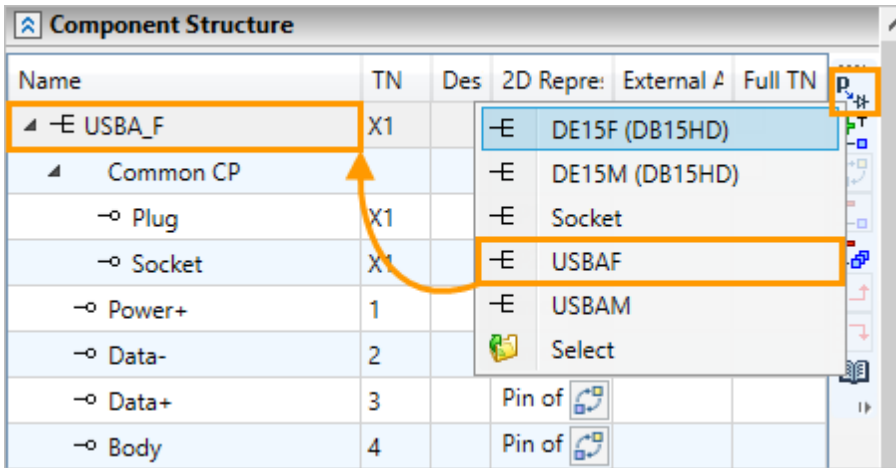
The structure of connector is displayed in **Component Structure** block and includes:



- **Socket** (head element) at the top level. By default, the "neutral" prototype is selected from  **Component Editor**.
- **Common CP** (common connection points) in a separate subgroup. Read more about them here.
- **Connection points** of connector on the lower level, which are not present by default - they need to be added.

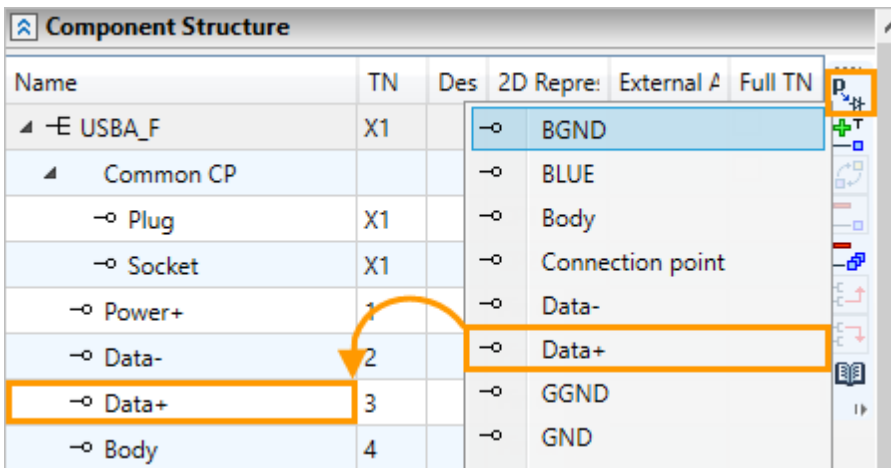
The following commands are available in the mini automenu.

 **Set Prototype** allows you to select a prototype from  **Component Editor** or replace a previously selected prototype.



If the cursor is positioned in the line with the connector name in **Component Structure** block, when replacing the prototype, the question will be asked "Load connector structure from prototype? Current structure will be lost." Answering [Yes] assumes that the connector prototype will be replaced and the connection points will be replaced with it.

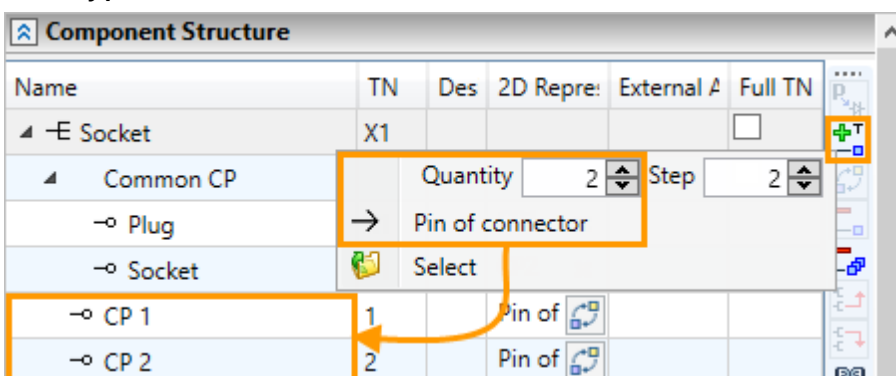
If the cursor is positioned on the line of connection point, the CP prototypes for replacement will become available in the drop-down list.



Add Connection Point adds connection points to the connector.

In the drop-down list, set **Quantity** of contacts and their type. The default **Step** is 2, but it can also be set manually. The step value must be even.


Contacts are added to the diagram as 2D fragments, and by default there is one **Pin of Connector** option that takes into account all graphic options. The connection point added in this way is assigned a "neutral" prototype from **Component Editor**. It can be replaced with the command **Set Prototype**.





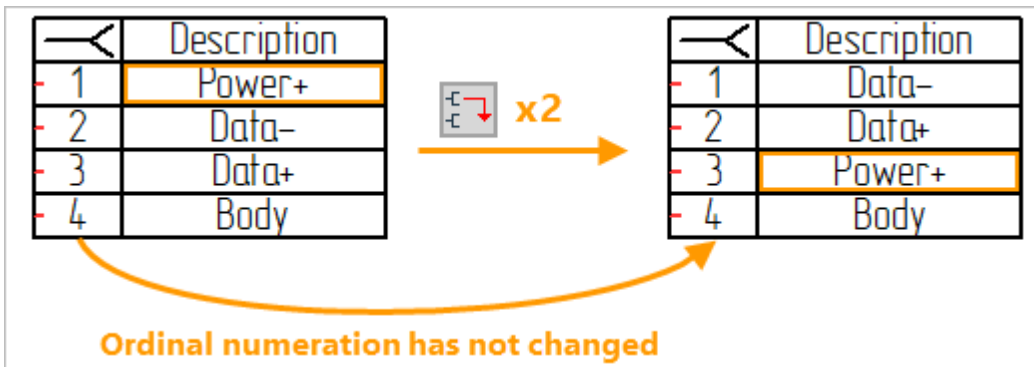
Set 2D Representation allows you to replace one fragment of a connector pin with another. By



default, only one fragment **Pin of Connector** is available, so the command does not play a special role. However, if the user needs to create his own fragment with a connector contact, he will be able to use the function of this icon.

 **Delete** is used to delete selected elements in **Component Structure** block. You can select one element or several using <Shift> or <Ctrl>.

 **Delete All** is used to delete all elements in **Component Structure** block.

 **Up** and  **Down** are used to change the order of connection points in the connector. In this case, only the names of CPs will be reversed, and the ordinal numbering will be saved.

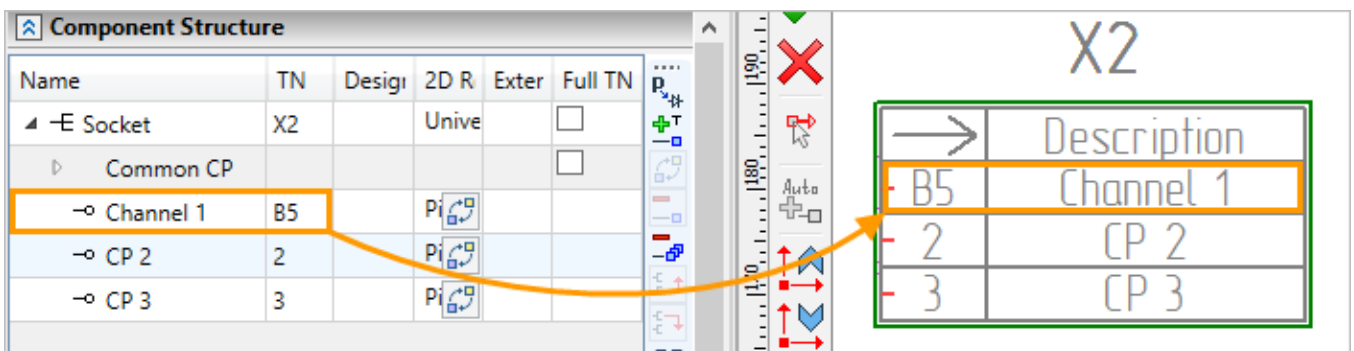


The folders from which the system borrows connector pins are configured in  **Library Configuration** command. This command is run in transparent mode in abbreviated form by  button in the mini automenu. Since the default connector pin allows you to create all the variations of its graphic representation, this command is unlikely to be required. For more information on setting up configurations, see here.


The following columns are displayed in **Component Structure** block.

Name displays the value of Description parameter of the selected element. It is convenient to rename the connector or the selected connection point in the column.

Tag Name displays the value of Tag Name parameter for the selected element. It is convenient to assign arbitrary designations to the connector contacts in the column.



Designation displays the value of Designation parameter of the connection point. It does not play an important role for the connector.


2D Representation view displays the fragment used to graphically display the connection point in the graphic symbol.  icon allows you to replace a fragment with another one, see details here.


External address is used to manually fill in **Address external** column in the table connector. See here for more details.


Full TN activates **Use full tag name** option for the selected element. It does not play an important role for the connector.

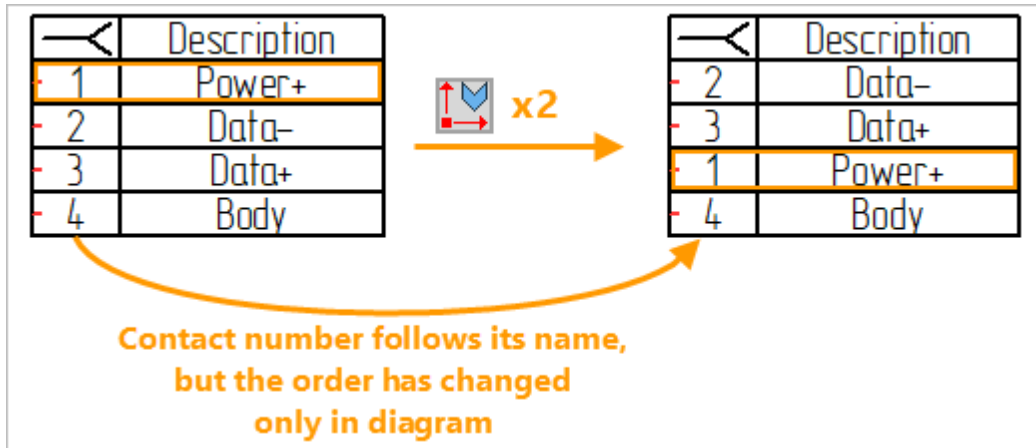
The values entered in the cells of **Component Structure** block are transferred to the corresponding parameters of the elements.


The following commands are available in the automenu:


 **Repeat element insertion (place automatically)** adds a connection point with a "base" prototype to the connector.

 **Move up** moves the selected contact up the list. The serial numbers and names of contacts will move together on the diagram, but there will be no changes in **Component Structure** block.

 **Move down** moves the selected contact down the list with similar conditions.




 **Orientation** rotates the connector. The function is available only for the "head" element and is not available for the connector pins.

 **Mirror** moves all pins of the connector to the other side. The function is available only for the "head" element and is not available for the connector pins.



 **Separated-pins view** toggles connector graphics to separated-pins view. See here for more details.

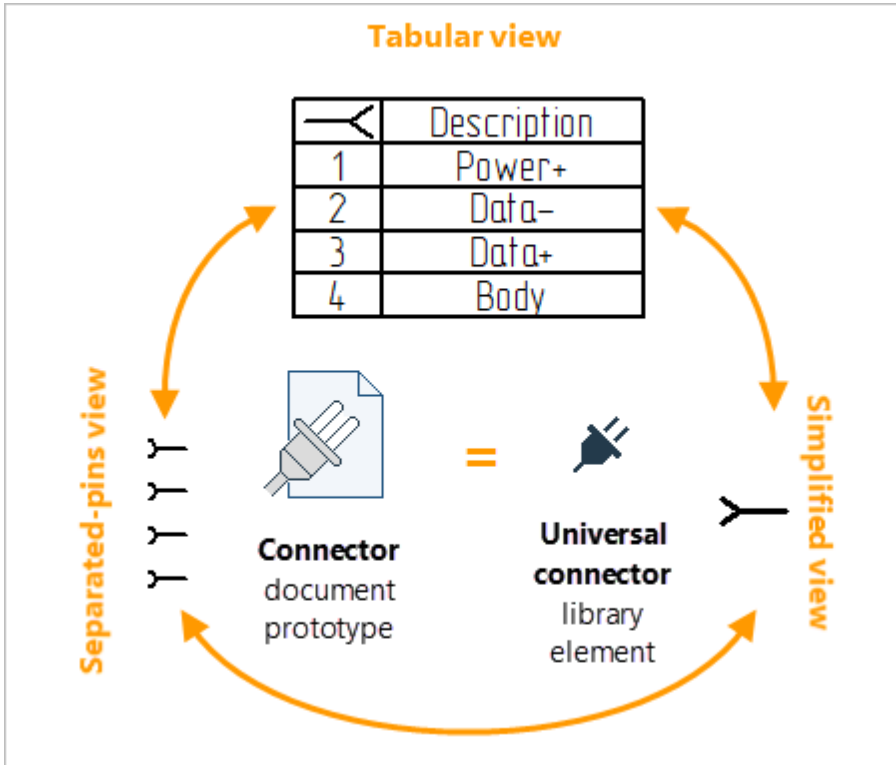
The information in **Parameters** block is context sensitive, that is, it changes depending on which element is selected in **Component Structure** block. The connector parameters are described in more detail here and here.

When the composition and parameters of connector are configured, click  in the automenu or at the top of the dialog.

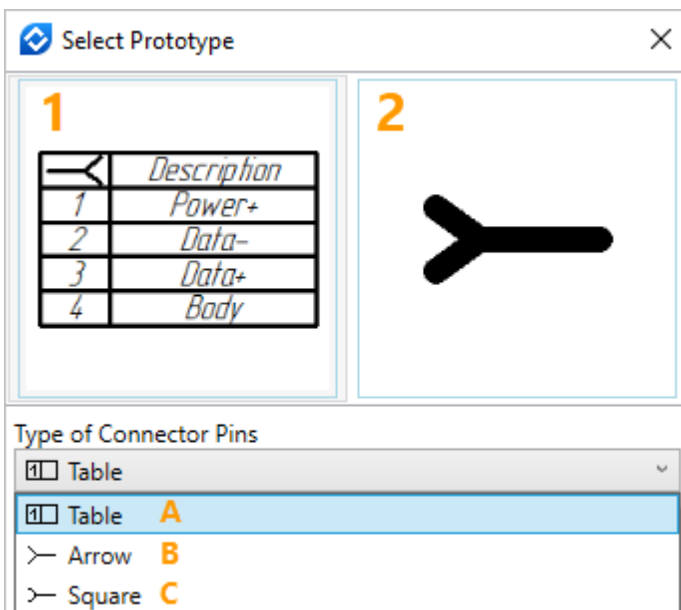
The graphic representation of the connector (tabular, simplified, separated pins) will be selected when inserting the connector into the diagram, for more details see here.

Graphic symbol

 **Connector** document prototype stores graphics of three views at once: tabular, simplified, and separated pins. When creating a connector, it is only necessary to fill in its structure, and the graphical representation can be set when inserting an element into the diagram or after that. Also, all graphical representations are available in  **Universal connector** in the standard library, which is used to create a connector in context of diagram.



When you insert an earlier created connector, a window will appear in which you need to select a graphical representation - tabular, simplified or separated-pins.

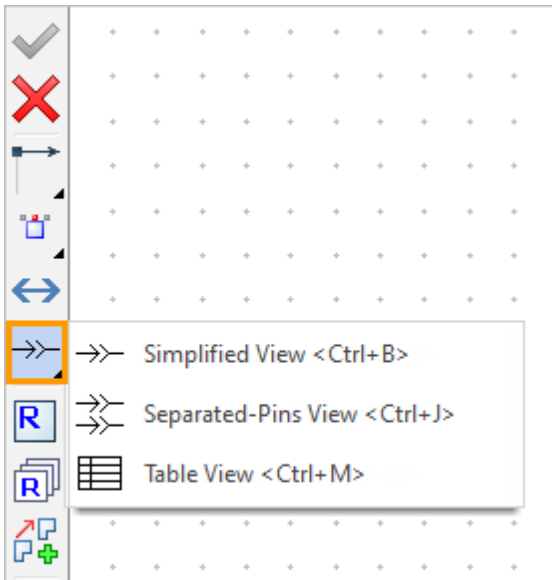


1A is a tabular connector.

2B/C - simplified connector.

1B/C - separated-pins connector.

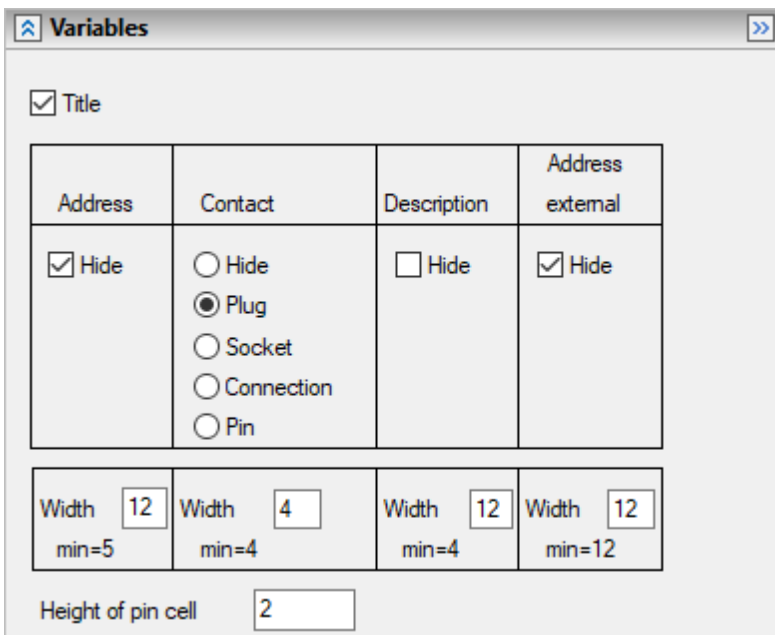
You can also switch the view of connector after inserting it into the diagram. To do this, select a connector, open the drop-down list in the automenu and select the required view. This method is also relevant for a connector created in the context of diagram.



Tabular view

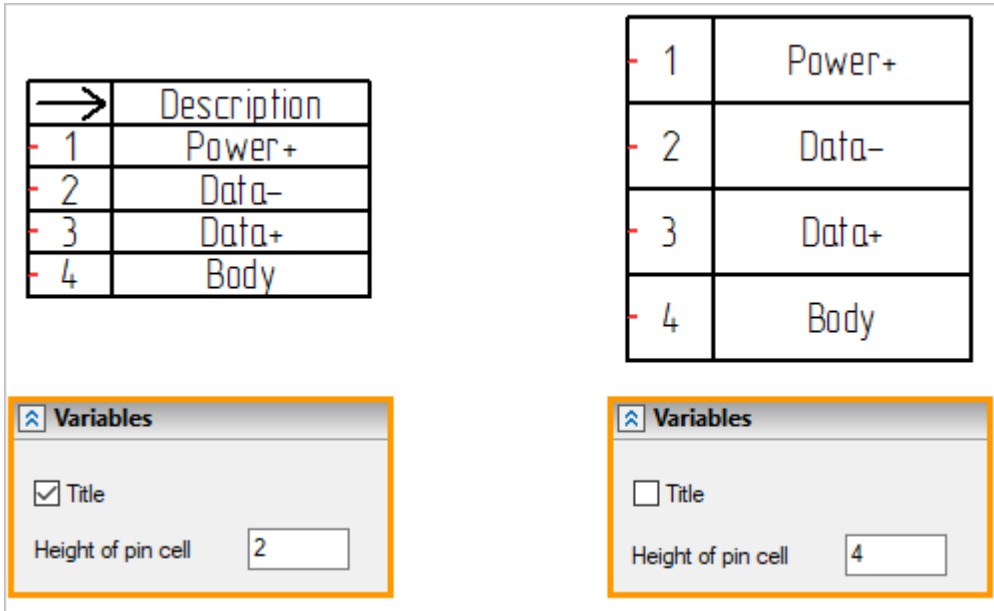
A tabular connector is stored on **Page 1** in **Connector** prototype document.

The following settings are available in **Variables** block:



Title controls the visibility of the table header.

Height of pin cell is set by the parameter of the same name. The height can be set only as an even number (to ensure the correct construction of connector graphic symbol).

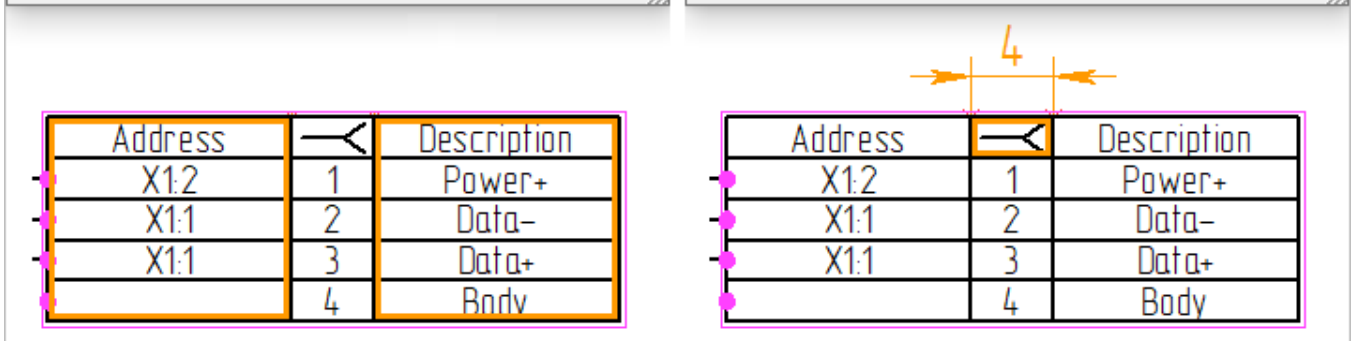
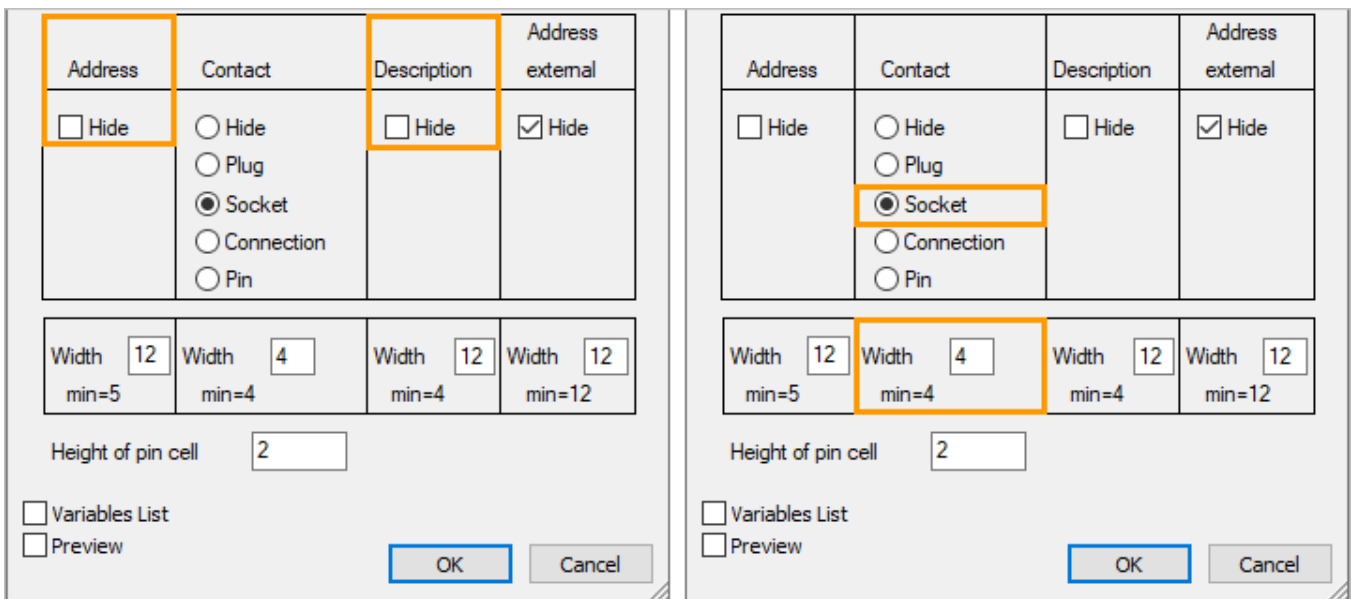



The rest of parameters in the dialog are divided into columns, which they control on the diagram.

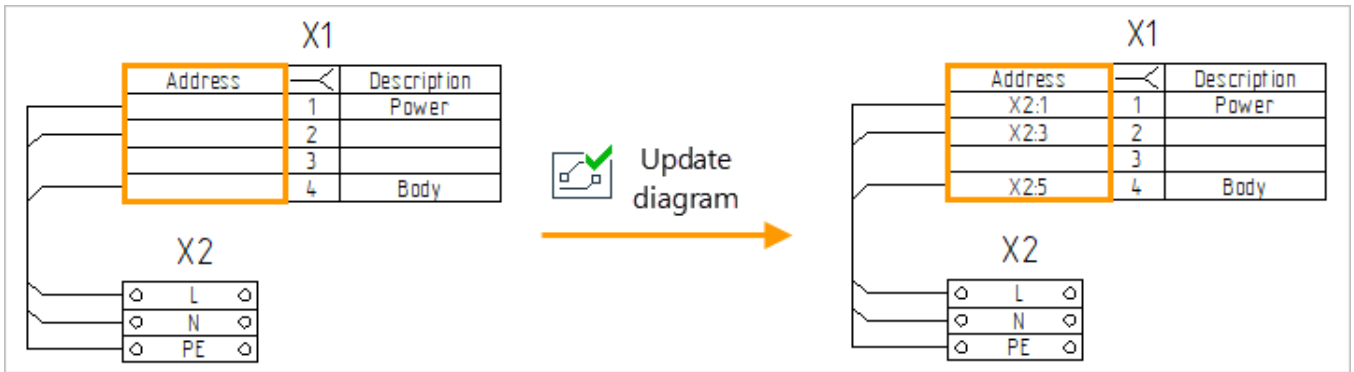
Hide controls the visibility of the corresponding column in graphic symbol.

In **Contact** column, **Plug**, **Socket**, **Connection** and **Pin** options are also available to control the symbol in the title string.

Width of each column is controlled by parameter of the same name. The minimum column width is indicated next to each input field, depending on the length of text in the title.



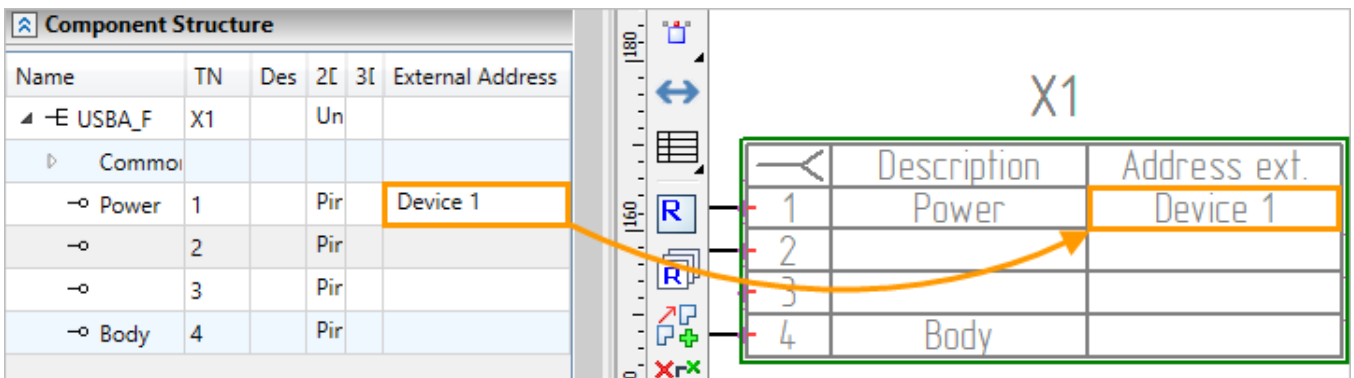
Address column is empty by default. If the connector is connected to another element, run  **Update Diagram** command and the **Address** column will be filled.



Contact column is filled with tag names of connector pins.

Description column is filled with the names of connector pins.

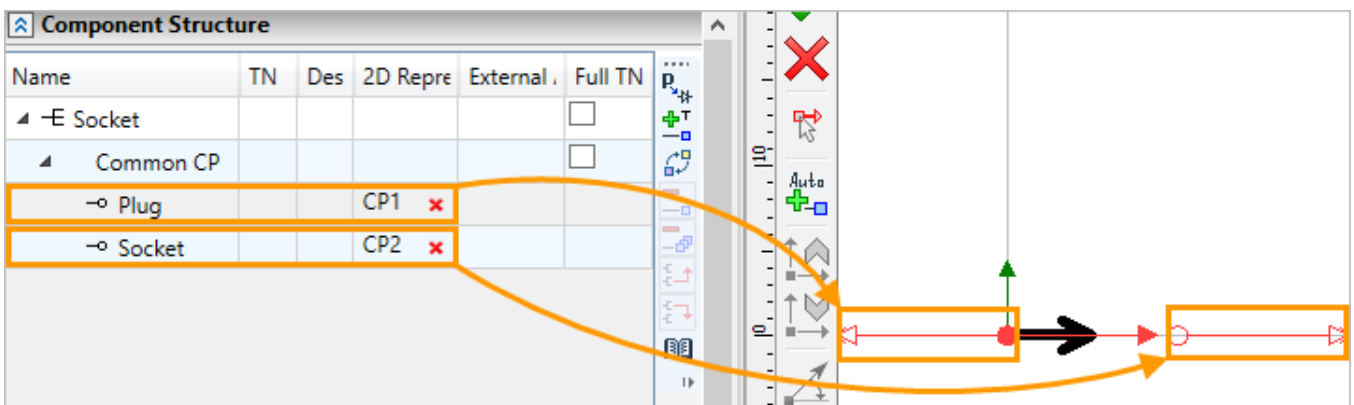
External address column is empty by default. It can only be filled manually in **Component Structure** block when the connector is selected in the diagram.



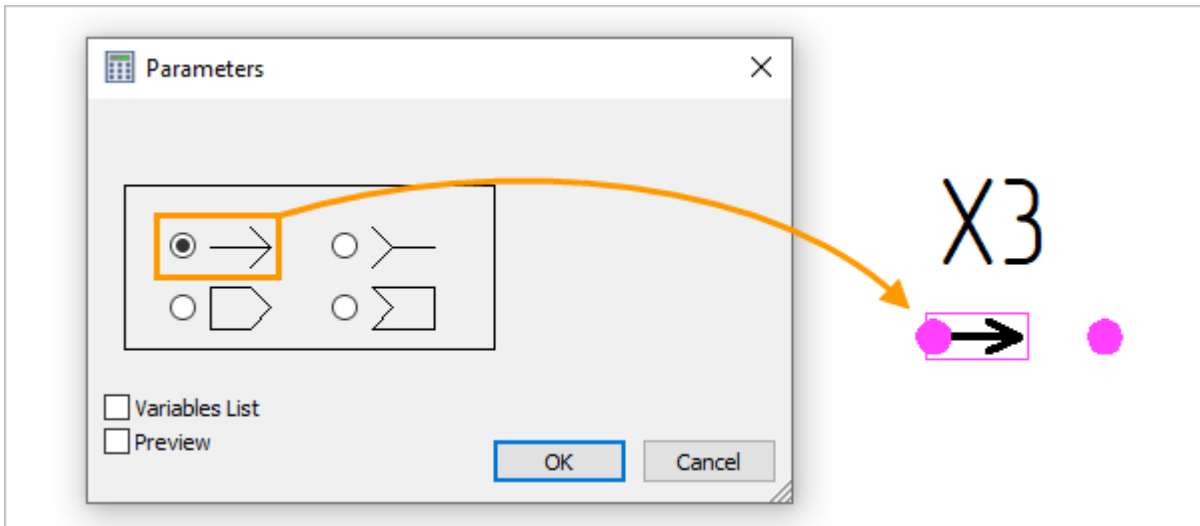
Simplified view

A simplified connector is stored on **Page 2** in **Connector** prototype document.

The simplified connector has common connection points on the cable and mounting side. The common CP assumes that several connection points in **Component Structure** are represented by one element (connector) on the diagram. The specified types (plug and socket) in this case have no physical meaning, the difference between types is necessary at the program level.



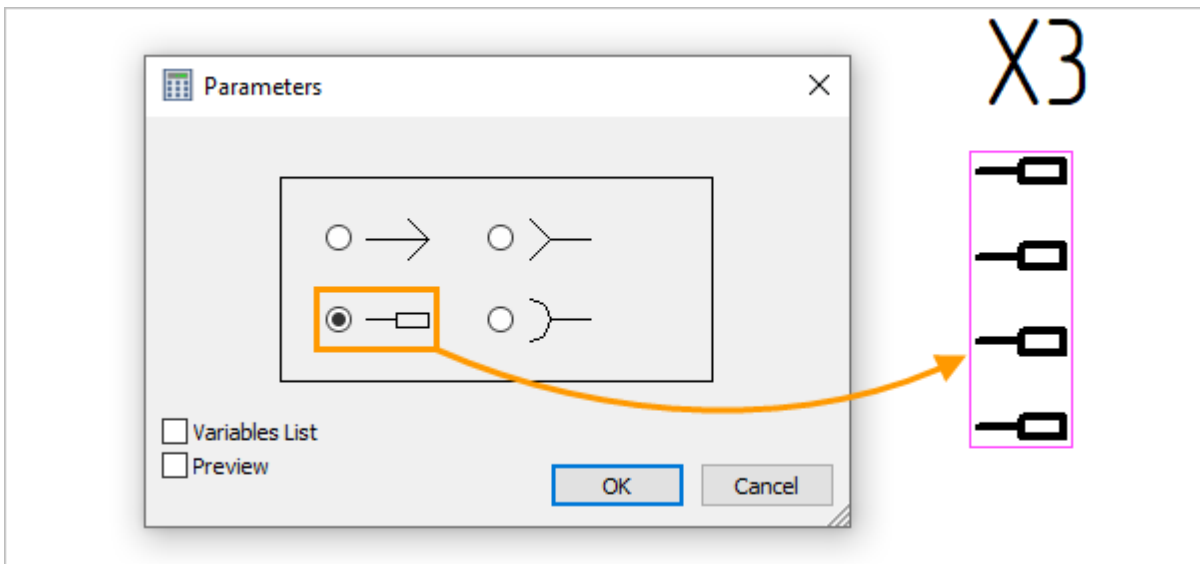
You can select a symbol of simplified connector in **Variables** dialog box and in the command of the same name:



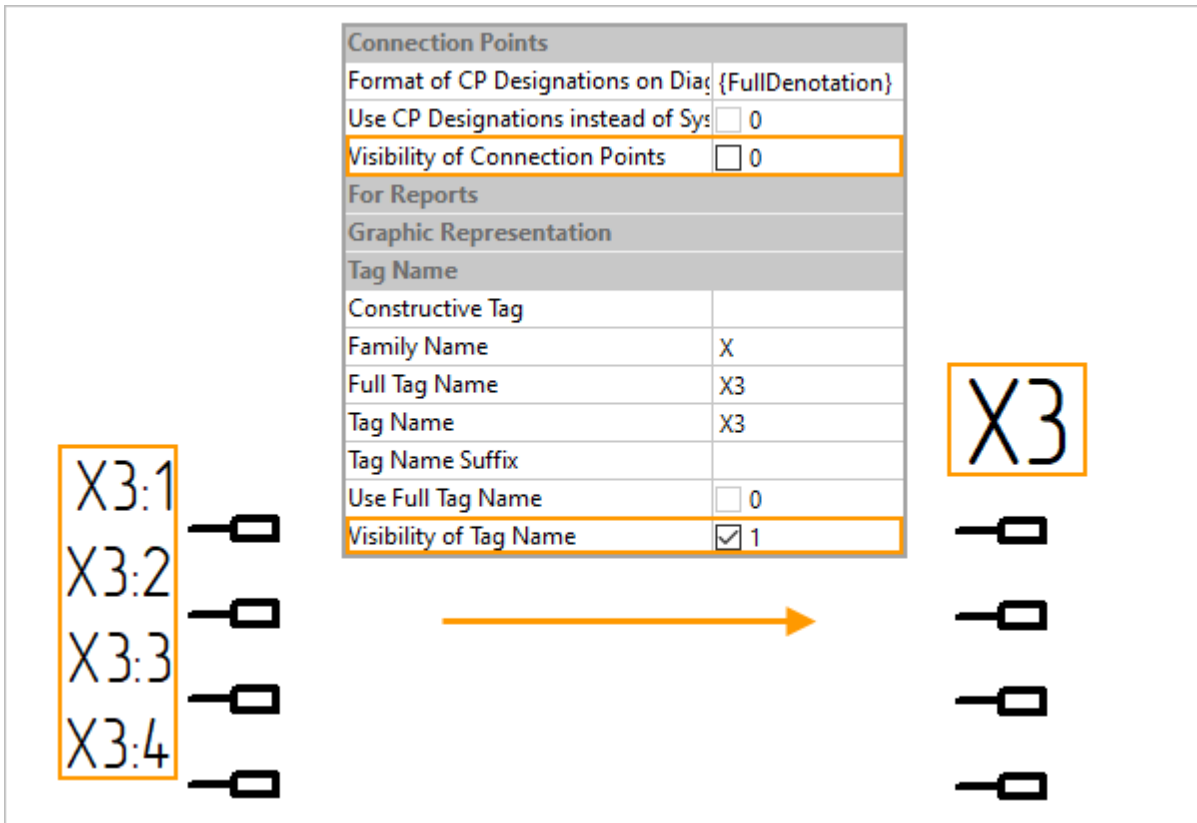
Separated-pins view

The separated-pins view does not have its own page in **Connector** document prototype because it is derived from a tabular view.

The selection of symbol of connector pin is available in **Variables** dialog box and in the command of the same name:

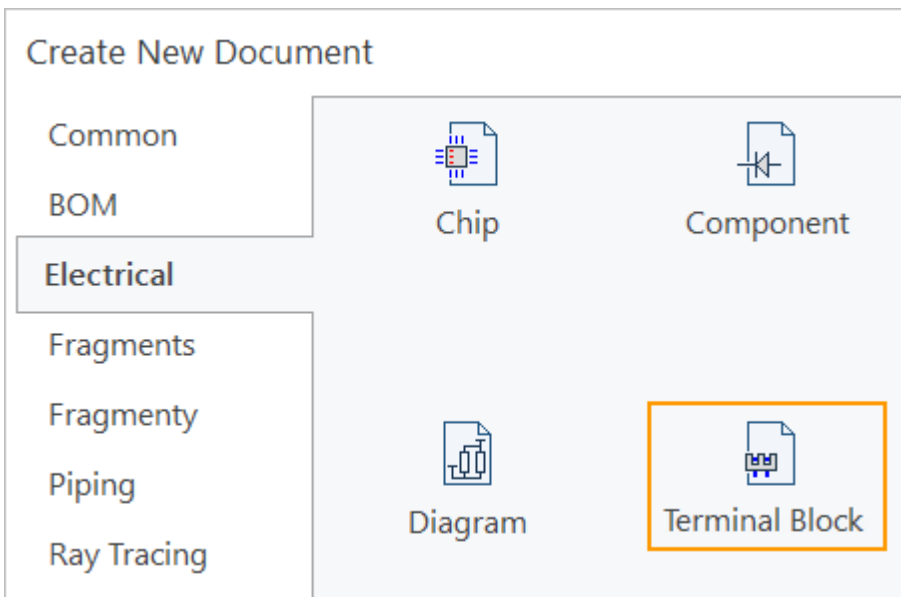


By default, the diagram displays the full tag names of connector pins in separated-pins view. To switch to displaying the connector's own tag name, enable **Visibility of tag name** and deactivate **Visibility of Connection Points** in connector parameters:




Terminal block

The prototype **Terminal Block** is used on **Start** page to create a terminal block (Electrical group).





After opening the document, click  **Edit Component** in the Ribbon.

You can also use **Create Terminal Block** command, which automatically creates a new document based on the **Terminal Block** prototype and starts the structure editing command.


Icon	Ribbon
	Electrical ? 2D Elements ? Create Terminal Block
Keyboard	Textual Menu
	Electrical > Elements > Terminal Block

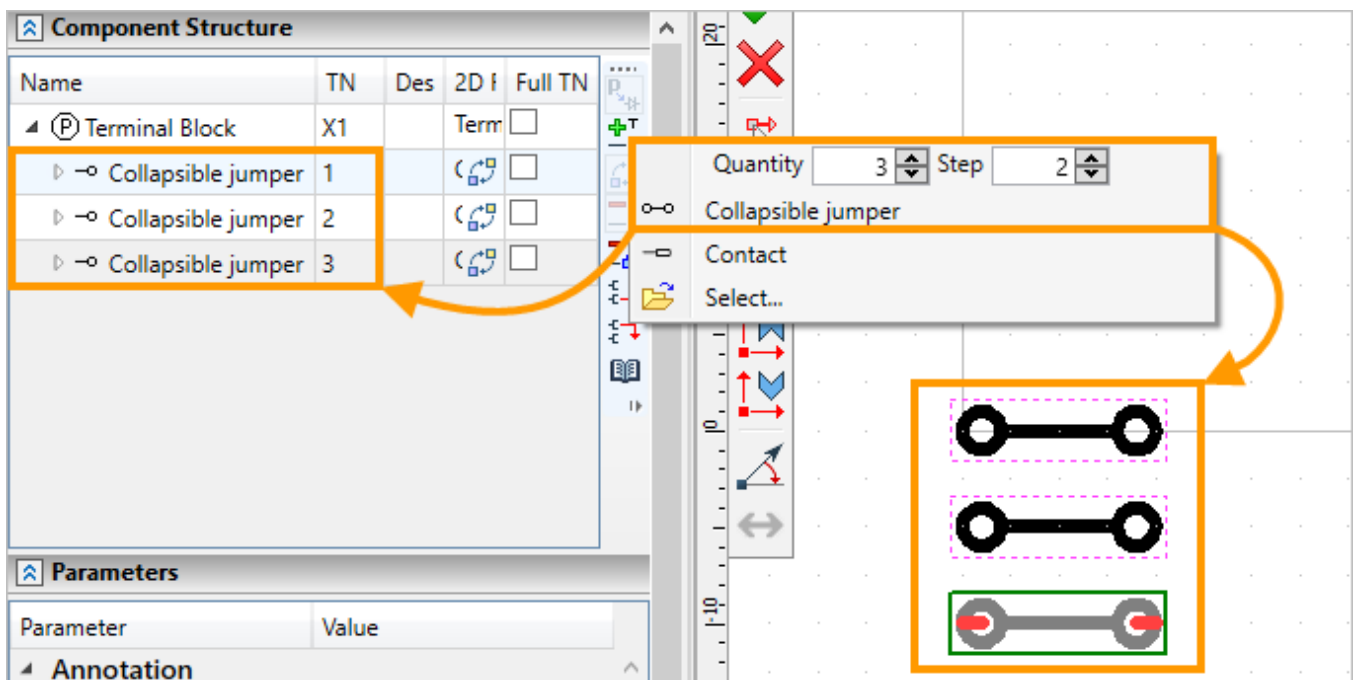
Component Structure block displays the elements that are part of the terminal block. It is empty by default.


To add elements to it, use the mini-automenu buttons:

 **Set prototype** allows you to select a prototype from  **Component Editor** or replace a previously selected prototype.

When replacing, depending on which line the cursor is positioned on (in the line of **Terminal Block** or one of the terminals), prototypes of terminal blocks or terminals will be available in the drop-down list. Replacing the prototype terminal block will not affect the terminals included in it.



 **Add terminal** is used to add terminals to the terminal block. Set the **Number** of terminals and their type in the drop-down list. The default **Step** is 2, but it can also be set manually. The step value must be even.

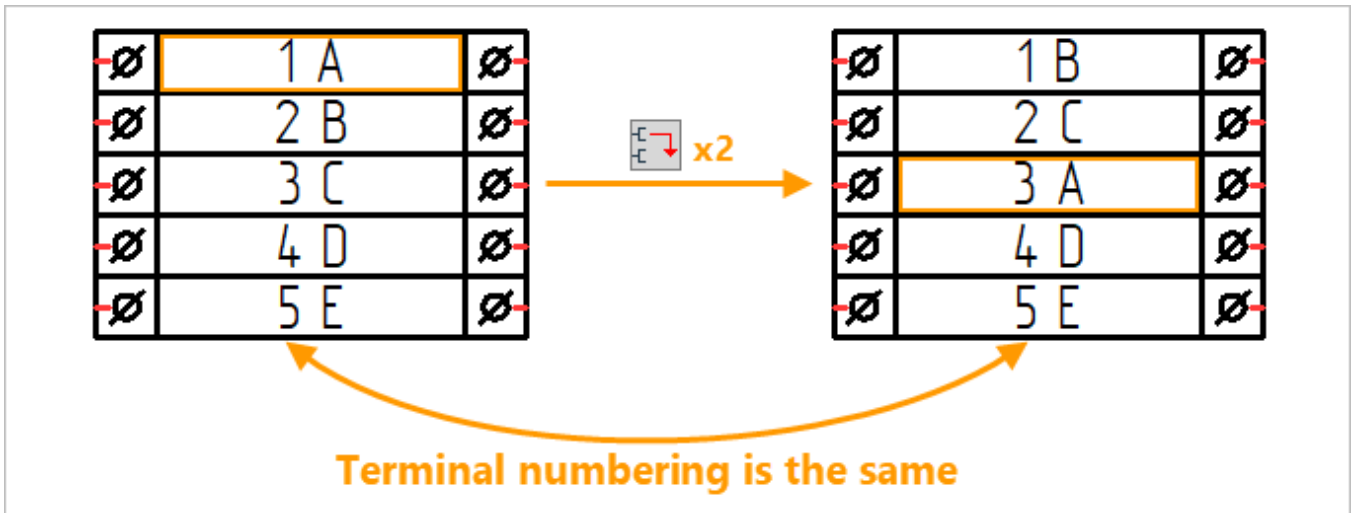


 **Set 2D representation** allows you to replace the symbol of terminals with another. In this case, the terminal parameters will remain the same.

 **Delete** serves to delete the selected elements in **Component Structure** block. You can select one element or several using <Shift> or <Ctrl>.

 **Delete All** is used to delete all elements in **Component Structure** block.

 **Move up** and  **Move down** are used to change the order of terminals in terminal block. In this case, only the names of terminals will be reversed, and the sequential numbering will be retained.

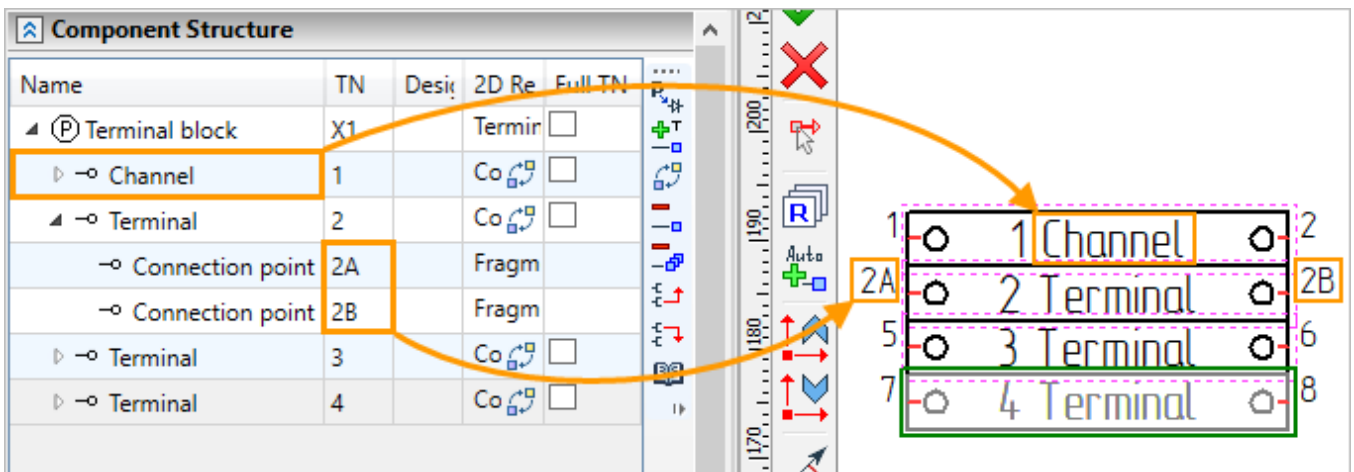


The folders in which the system "searches" for terminals are configured in **Library Configuration** command. This command is run in transparent mode in an abbreviated form by a button in the mini-menu. See here for more details on setting up configurations.

The following columns are displayed in **Component Structure** block.

Name displays the value of parameter of the same name of selected element. It is convenient to rename the terminal block or the selected terminal in the column.

TN displays the value of parameter of the same name for the selected element (i.e. its tag name). It is convenient to assign arbitrary tag names of terminal contacts in the column.



Designation displays the value of the connection point parameter of the same name. It does not play an important role for the terminal block.

2D Representation displays the fragment used to graphically display the terminal. The icon allows you to replace a fragment with another one, see details here.

Full TN activates **Use full tag name** checkbox for the selected element. It does not play an important role for the terminal block.

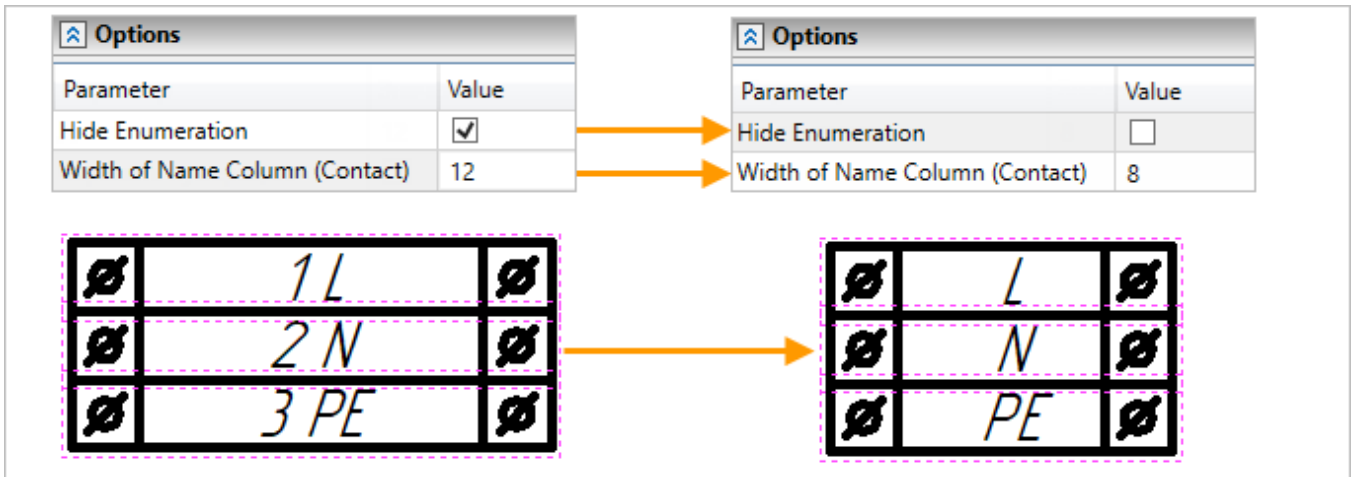
The values entered in the fields of **Component Structure** block are transferred to the corresponding parameters of the elements.

The following parameters are available in **Options** block.


Width of Name column (contact) controls the number of characters in terminal symbols, changing the width of terminal block. Available only when focusing on the head item in **Component Structure**


block.

Hide Enumeration controls the visibility of terminal tag names in the symbol of terminal block.

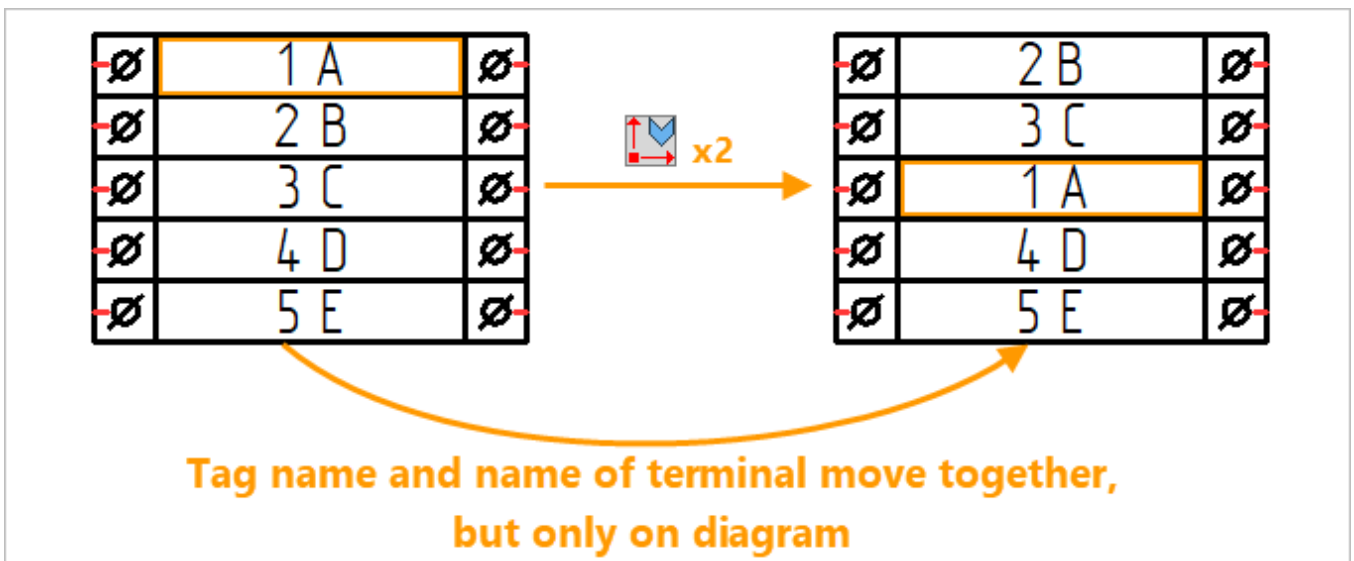


The following commands are available in the automenu:

 **Repeat element** adds a terminal with a "basic" prototype to the terminal block.

 **Move up** moves the selected terminal up the list. The serial numbers and names of terminals will move together in the diagram, but there will be no changes in **Component Structure** block.


 **Move down** moves the selected terminal down in the list with similar conditions.



 **Orientation** rotates the terminal block.

 **Mirror** transfers the terminals of terminal block to the other side.

The information in **Parameters** block is context sensitive, that is, it changes depending on which item is selected in **Component Structure** block. The parameters of terminal block are described in more detail here and here.

When the composition and parameters of the terminal block are configured, click  in the automenu or at the top of the dialog.

Contactor

Relay is an electronic component with possibility of being placed on the electrical diagram in its entity or in a spaced form. That's why the relay is created different from the electronic component and there is a special prototype for relay.

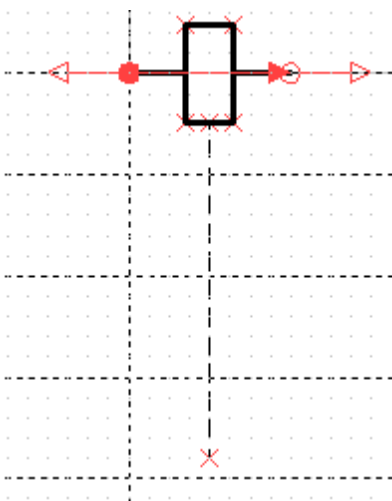
The structural element "Relay" is located in the components editor on the **Electronic components** tab and its prototypes can be created by the [same rules](#) as electronic components.

There are already two connection points by default in the prototype so there is no need to create them. User should only add several contact pairs from the library configuration.

Let us create a new document on the basis of the "Relay" document on the "Electrical" tab.

It is available to create "Relay" component only on the basis of the "Relay" prototype.


The prototype already contains base drawing for the relay creation. There is a fixing vector for insertion into a diagram, two connection points and schematic symbol of the main part of the relay.



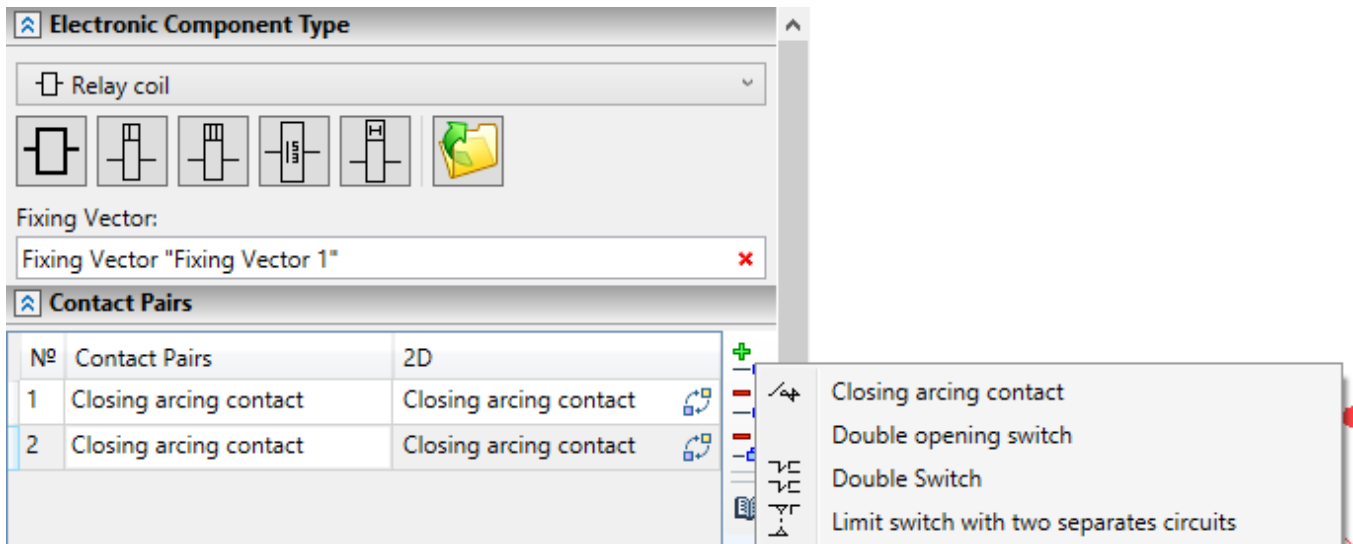
Except the [standard variables](#) there is a variable "Contact_vert" in the prototype. It is used to add additional slots for contact pairs.

The contact pairs should be placed at the intersection of the construction lines in the prototype.

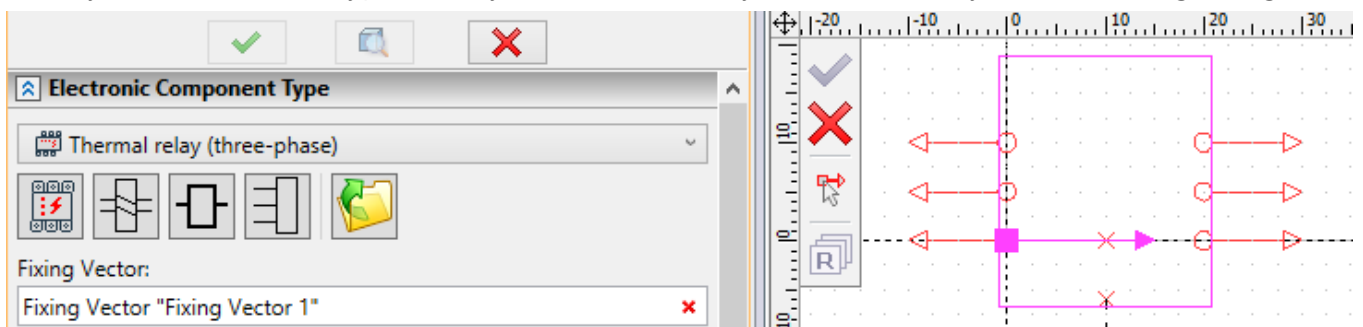
Use the following command to create a relay:


Icon	Ribbon
	Electrical ? Elements ? Create Relay
Keyboard	Textual menu
	Electrical > Elements > Create Relay

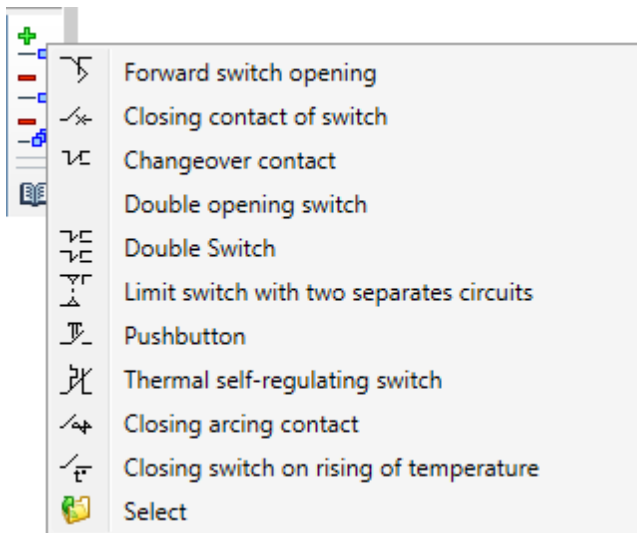
You can select a relay in the drop-down list. You can find the prototype in [Components editor](#) on Electronic components tab.



When you select another type of relay from the list the symbol of the relay on the drawing changes.




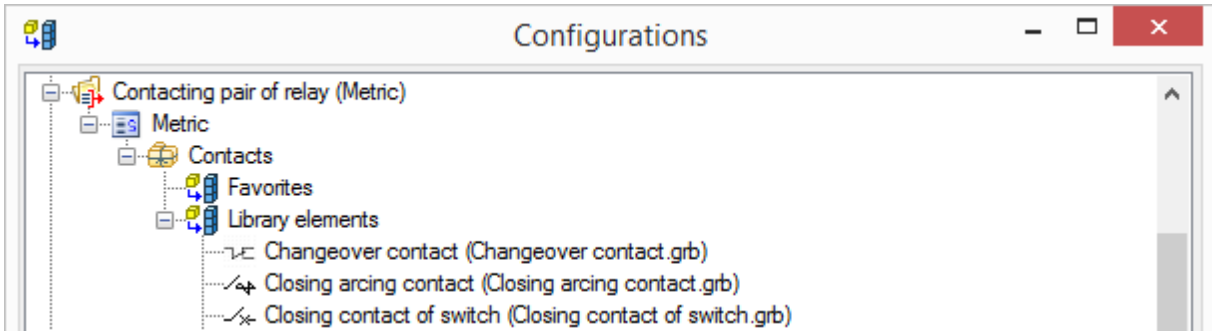
The option  is used to add contact pairs of the relay. Standard contacts are displayed in the list. If custom contacts added to the [library configuration](#) will be also displayed in the list.



The option **Delete Contact Pair**  deletes the selected in the list contact pair.

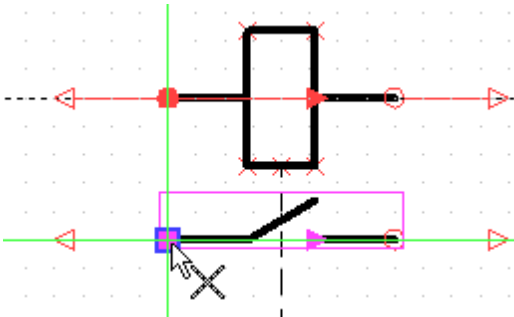
The option **Delete All Contact Pairs**  deletes in the list.

The **Library Configurations**  opens the **Configurations** window in which the connected library with contacts is displayed.

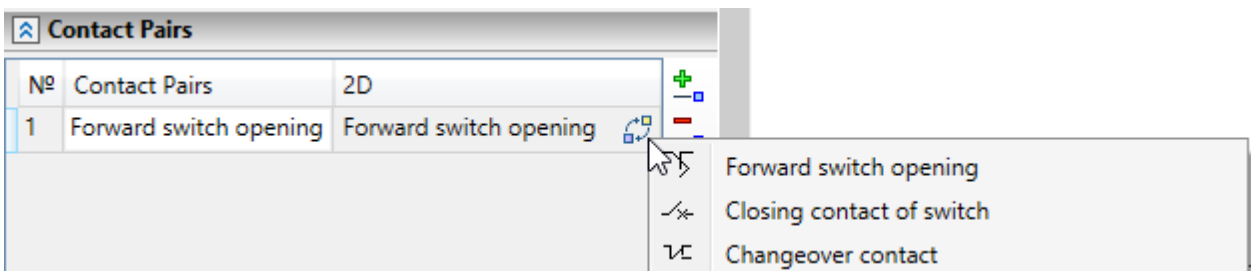


User can [add his own components](#) to the configuration.

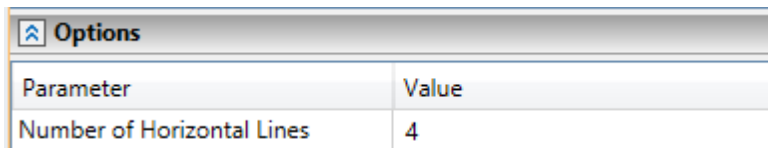
You can create contact pairs at the intersection of the existing in the prototype vertical lines and single horizontal line - slots.



You can change contact pair if it is necessary using the special option.



Use the special option in the command parameters window to add slots for contact pairs. The value is associated with the "Contact_vert" variable.




After adding all the contact pairs press and save the relay into the library.

Chip

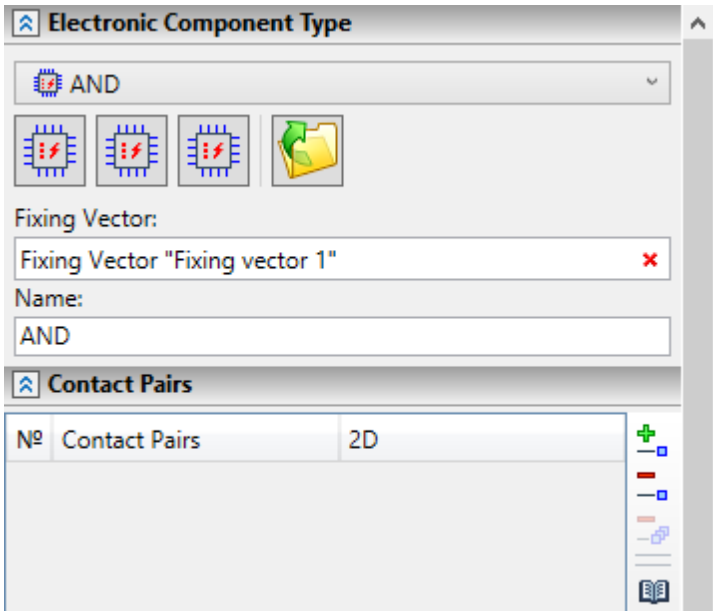
The "Chip" prototype can be found on the **Electronic components** tab in the **Components editor** dialog.

A new element on the basis of "Chip" type is created based on the "Device" prototype document on the "Electrical" tab.

Use the following command to create a chip:

Icon	Ribbon
	Electrical ? Elements ? Create Chip
Keyboard	Textual menu
	Electrical > Elements > Create Chip

After calling the command a parameters window appears. In the window you should select a chip type and add any number of logic gates.




Symbol

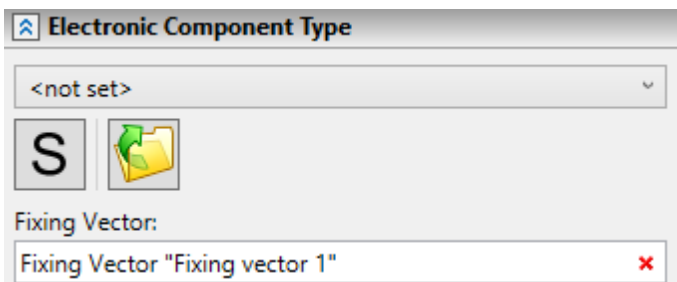
A symbol is a graphic image that can be placed on the diagram.

You should create only schematic symbol in the document, according to the rules described in the Base component - Graphic section.

To create a symbol use the command:

Icon	Ribbon
	Electrical ? Elements ? Create Symbol
Keyboard	Textual menu
	Electrical > Elements > Create Symbol

You should select the prototype for the symbol in the parameters window. There is a single prototype that is used for creation of all symbols.




Add components to configuration


The library configuration stores data about components and allows to insert components into diagrams using [Insert component](#) command. Also it allows to add connection points, contacts and connectors to another components using **Create...** commands .

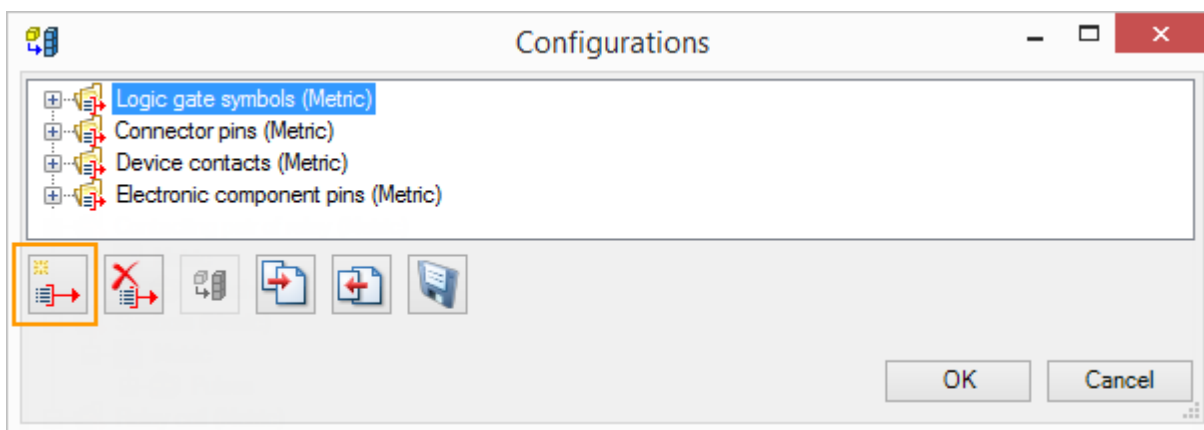
If the user component is not added to the library configuration it does not displayed in system menus.

Let us add connector library configuration to be able add it to devices.

To add components to the library configuration use the command:

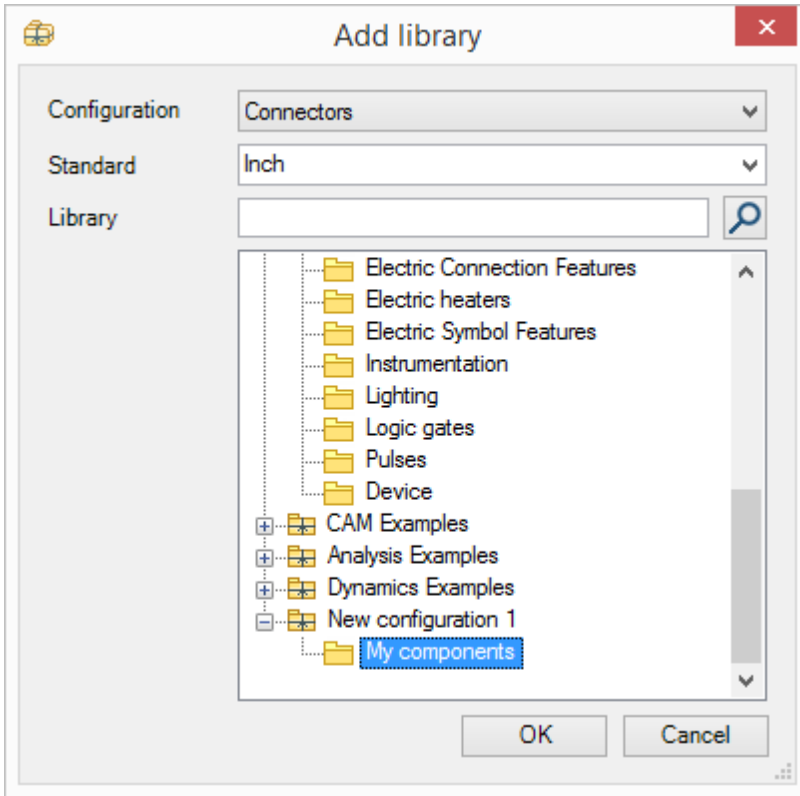
Icon	Ribbon
	Electrical -> Editors -> Library Configurations
Keyboard	Textual menu

Select option  in the appeared window.

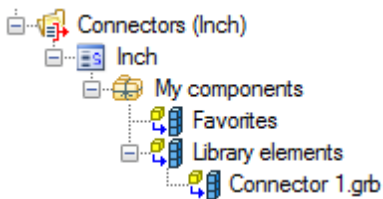


In the appeared window you should select configuration "Connectors", standard "Inch" and library with the connector. Press . More information about the exiting library configurations [can be found in the following chapter](#).

Library configurations for each type of components are available in the list.




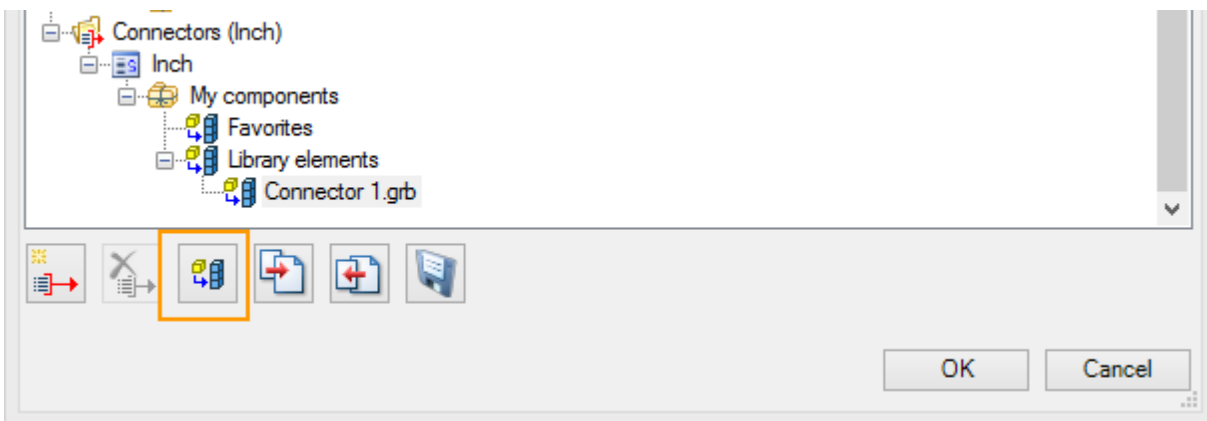
A new library configuration will be displayed in the window.



The component will be now displayed in the lists of the listed above commands.

If you want the connector to be displayed as favorite, you should move it to the corresponding folder.

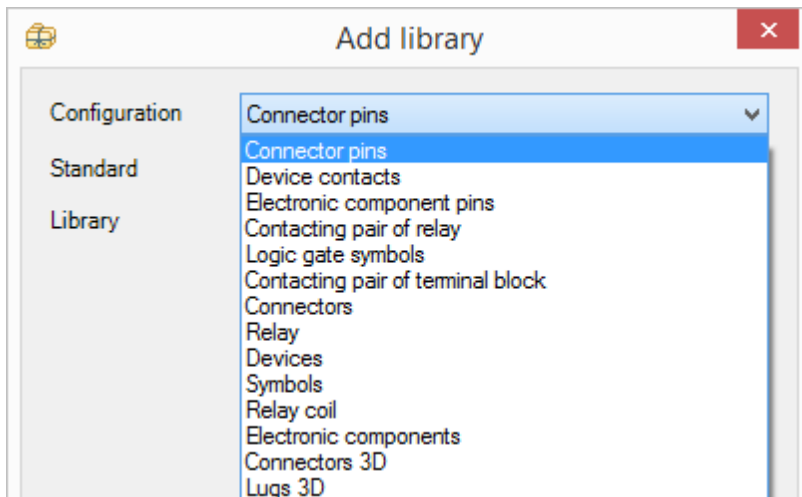
Use the **Add/Delete element as element by default**  option.



Press [OK] to finish.

Libraries of electrical diagram elements

There are several library configurations in configurations window, in which you can add components.



Name of library configuration

Comment

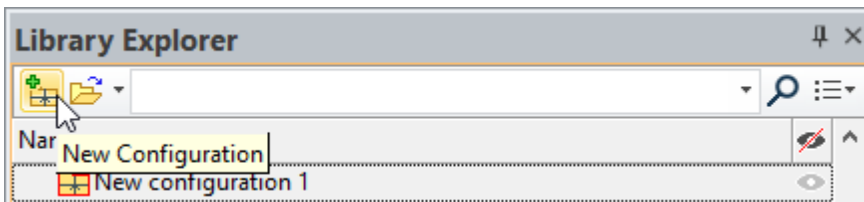
Connectors pins	used to store connectors that can be added to the devices
Devices contacts	used to store connection points that can be added to the device
Electronic components pins	used to store connection points that can be added to the electronic components
Contact pair of relay	used to store contact pairs that can be used in the relay
Logic gate symbols	used to store logic gate symbols that can be added to the chip
Contact pair of terminal block	used to store contact pairs that can be added to the terminal block
Connectors	used to store user elements of the "Connector" type so they can be displayed in the Insert component command and upon adding to the device and cable.
Relay	used to store user components of the "Relay" type so they can be displayed in the Insert component command.
Devices	used to store user components of the "Device" type so they can be displayed in the Insert component command.
Symbols	used to store user components of the "Symbol" type so they can be displayed in the Insert component command.
Relay coil	used for coils that can be added to the relay
Electronic components	used to store user elements of the "Electronic component" type so they can be displayed in the Insert component command and upon adding to the device and cable.
Connectors 3D	used to store 3D connectors
Cable terminals 3D	used to store 3D cable terminals

- Cable labels 3D used to store 3D cable labels
- Cable-end splittings 3D used to store 3D cable-end splittings
- Cable adapters 3D used to store 3D cable adapters

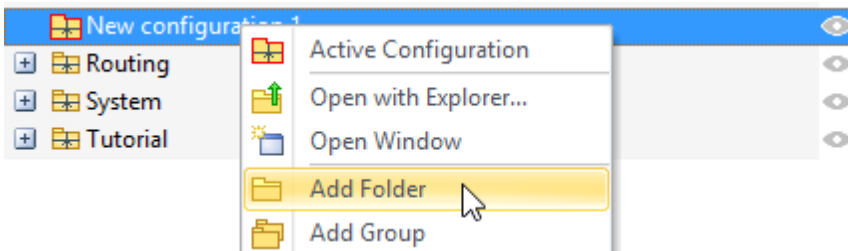
Add components to library

All the created components should be saved to the library. If the documents with components were not added to the library, they should be stored with the diagram document. The files from the library can be added to the library configuration.

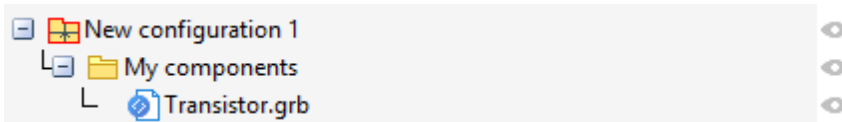
Let us create "My components" folder. The created file of the component is saved in the folder. After that we create a new library configuration in the **Library explorer** <Alt + 1>.



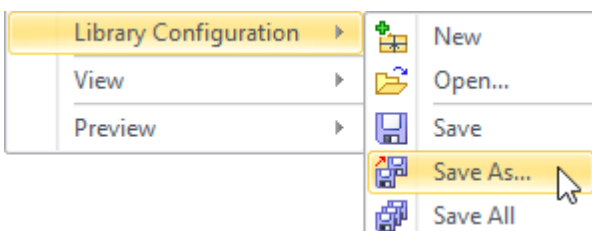
We add the created folder "My components" into the library explorer.



The folder and the file will be displayed in the library explorer.



The library should be saved using the corresponding item in its context menu.



Designing of electrical components in 3D

Creation of associations between a diagram and a 3D model of the component, laying of cable components and creation of reports for the finished project are created here.

General approach for creation of 3D model by diagram

When the diagram is ready you can create a 3D assembly. At first, the diagram should be inserted into the 3D assembly document using the [Insert Diagram](#) command.

After that you should prepare 3D fragments by [creating special 3D connectors](#) according to certain rules. The connectors will be associated with the 2D-connectors of the components on the diagram.

Next we should associate components on the diagram and in the assembly. Use [Component in 3D](#) command to do this. You should select the component on the diagram and component in the assembly to associate them using the command. If association of the connection points was not created automatically you can select a connection point on the diagram and the corresponding 3D connector in the assembly manually.

After all the associations are set you should [lay cable components](#). You can lay all wires at once, for the components between which associations were created, or lay each of the cable components by selecting them on the diagram manually. In the first case the [Cable components by diagram](#) command is used. Corresponding commands are used for laying of cable components: [Wire](#), [Cable](#). The cable components can be added to a harness using the [Harness](#) command.

After that you should specify position of cable components in the assembly. This can be done using the manipulators or by specifying a predefined path.

You can create layout of the harness on a separate page using the [Harness layout](#) command. The harness can be unloaded to separate assembly using a special command.

When the assembly is ready you can create [reports](#).

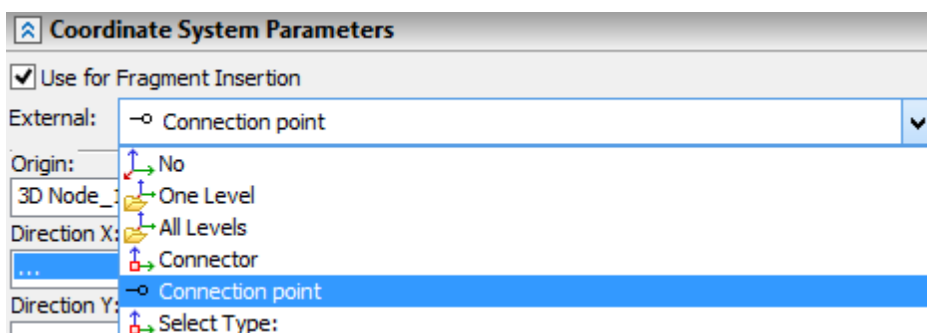
Prepare component 3D model

It is necessary to create special connectors for all component models that are included into the assembly. Connectors allows to associate diagram elements and 3D objects in the assembly. On the basis of this association all cable components will be laid in the assembly.

If it is necessary to add several similar components to the assembly, it is easier to create connectors in the component model document. Otherwise, connectors can be created in the in the assembly.

To add a connector, use the command **3D: Create Local Coordinate System**.

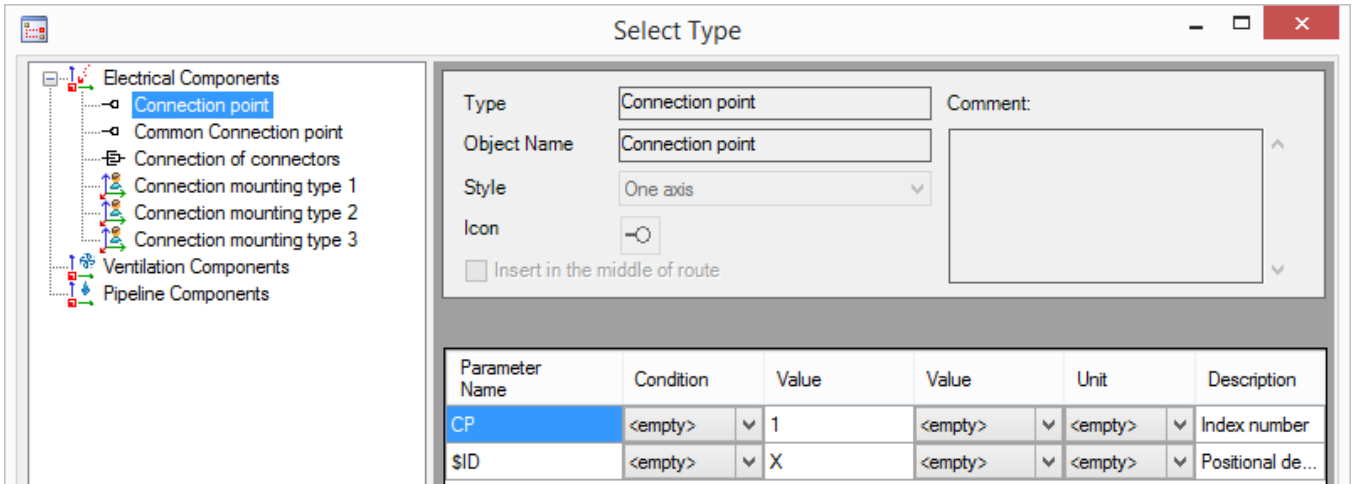
When you create LCS, it is necessary to set that the LCS is a connection point.



Special types for the connection points are already created in the system. Their list is displayed after

you select the **Select type** item from the drop-down list.

All existing connection points types are listed in the window.

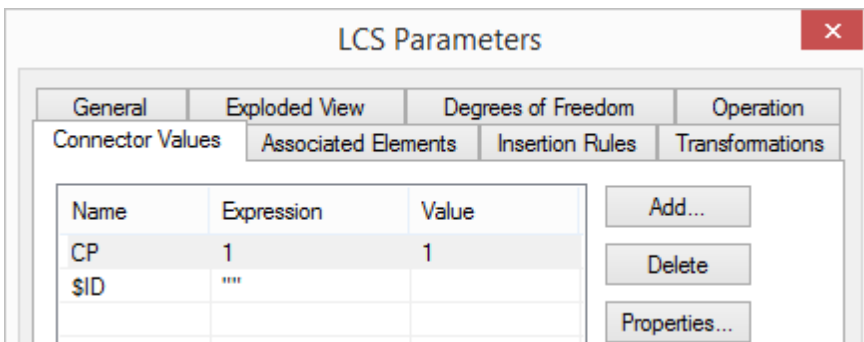


Connection point - normal connection point.

Common connection point is used only for connectors. It has a tag name for connector, so it can quickly connect to entire connector.

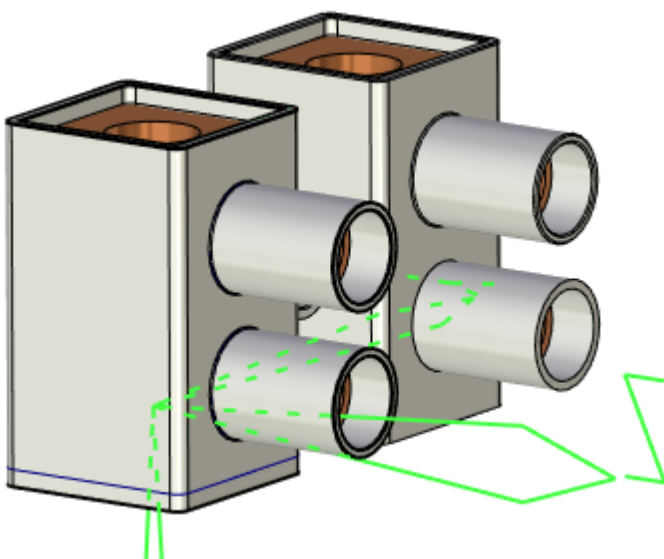
Connector joint is used where plug and joint are connected.

After the connection point is selected you should set connector parameter. By default, the name "CP" is already set, because it is assigned after selecting the LCS type. You should specify only designation.



The designation should correspond the designation of connection point on the diagram.

After all the parameters are set, you should specify position of LCS on the 3D model.



You should repeat this actions for all 3D fragments of the assembly.


Component in 3D

After all 3D models of components are added to the assembly, and 3D connectors are created, you should specify association between the 3D models and the components on the diagram.

You should set associations for automatic cable components creation.

After you select a component on the diagram the model associated with the component will be highlighted.

Use the following command to specify the associations:

Icon	Ribbon
	Electrical ? Cable Components ? Component in 3D
Keyboard	Textual menu
<>	Electrical > Cable Components > Component in 3D


After calling the command you should select **2D Representation** in the parameters window.

The 2D representation is selected on the diagram.

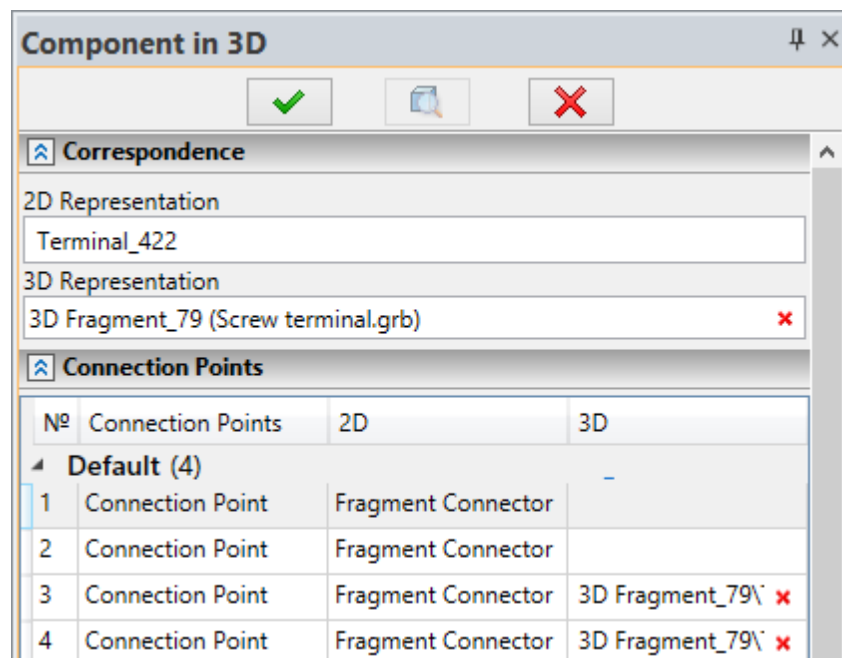
Use the following option for selection:

	<P>	2D Representation
---	-----	-------------------

Next you should select a 3D Representation in assembly using the option:

	<L>	3D Representation
---	-----	-------------------

After you select 3D representation on the diagram, association of all its connection points with connection points of the component on the diagram is specified.



In case when the system can't specify association of connection points on 2D and 3D automatically, you should specify it manually.


Use the option:

	<C>	Connection Point 3D
---	-----	---------------------

You should select an item in the **Connection Points** list and select an associated 3D connector in the assembly.

Connection Points			
Nº	Connection Points	2D	3D
Default (8)			
1	L1	Fragment Connector "CP1"	3D Fragment_62\1 (Connection point) ✘
2	L2	Fragment Connector "CP1"	3D Fragment_62\2 (Connection point) ✘
3	L3	Fragment Connector "CP1"	
4	PE	Fragment Connector "CP1"	
5	U	Fragment Connector "CP1"	
6	V	Fragment Connector "CP1"	
7	W	Fragment Connector "CP1"	
8	PE	Fragment Connector "CP1"	

To clear all 3D representations use the option:

	<K>	Break Links with 3D Representations
---	-----	-------------------------------------

Cable components

There are several ways to lay cable components:

- You can create all cable components on the basis of the diagram using the [Cable Components by Diagram](#) command. You select a diagram and all components are associated according to the diagram. The way is convenient for a small assemblies and for assemblies where harnesses should be created.
- Create each of the cable components type separately using special commands. This way is convenient for large assemblies as it allows to avoid mistakes.

To create a wire it is necessary to select a connection line on the diagram or connection point in the assembly. When you select a connection point on the diagram, the wire is created automatically. Two connection points are connected according to the diagram.

You can select connection lines on the diagram or select already created wires in 3D assembly to create a cable.


You can select connection lines on the diagram or select already created wires and cables in 3D assembly to create a harness.

When you click on the existing wire a manipulator appears. The manipulator allows to specify position of the cable component.

Cable components by diagram

After specifying of association between all 2D and 3D representations we can create cable components.

Use the following command to create all wires:

Icon	Ribbon
	Electrical ? Cable components ? Cable Components by Diagram
Keyboard	Textual menu
	Electrical > Cable components > Cable Components by Diagram

Use the following option to select single connection line on the diagram:

	<L>	Select Connection Line on the Diagram
---	-----	---------------------------------------

You can select normal or group connection line on the diagram.




Use the following option to select all connection lines on the diagram:

	<A>	Select All Connection Lines on the Diagram
---	-----	--

To delete all connection lines from the list use the option:

	<D>	Clear
---	-----	-------

A list of the selected electrical connections is displayed in the bottom part of the window. Here are displayed connection points and wires. You can select wires from the list manually.

<input checked="" type="checkbox"/>	Conductor	From	To
▲	Connection_522 (10)		
<input checked="" type="checkbox"/>	 Universal grey wire	KM2:4:9	X2:11:23
<input checked="" type="checkbox"/>	 Universal grey wire	KM2:4:9	QF9:2
<input checked="" type="checkbox"/>	 Universal grey wire	KM2:4:9	X2:4:9

In the "From" and "To" columns you can specify connection points that will be connected using wire. It allows to specify the best way of components connection.

You should activate flag in the left column to create a wire in the scene.

Parameters

In the **Diagram** field you should select a diagram on the basis of which all cable components will be created.

Diagram	Electric Diagram_33
Geometry Type	Spline
Representation in Model	Wire
Tangency on Start:	80
Tangency on End	80

Geometry type. A geometry that will be used for the cable components upon creation should be selected here: **Spline** or **Polyline**.

Representation in model. You should select one of the three representation for the cable products in the scene: **Wire**, **Faceted**, **Solid**.

Wire representation creates a path between connection points. It is the least resource-intensive way of displaying cable products in the scene, but the least demonstrative and effective.

Faceted representation creates faceted representation for cable components. The representation is used by default.

Solid representation created solid bodies for cable components. Can is required when projecting assembly to receive vector projection. It is the most resource-intensive way of creating cable components.


Tangency on start and **Tangency on end** allows to specify minimal length along the tangent at the start and in the end.

Cable components creation

Creation of different type of cable components is described here.

Wire

Use the following command to create a wire:

Icon	Ribbon
	Electrical ? Cable Components ? Wire
Keyboard	Textual menu
	Electrical > Cable Components > Wire

Use the following command to select a connection line:

	<Y>	Select Connection Line on the Diagram
---	-----	---------------------------------------

You can select a normal connection line or a group connection line of the diagram.

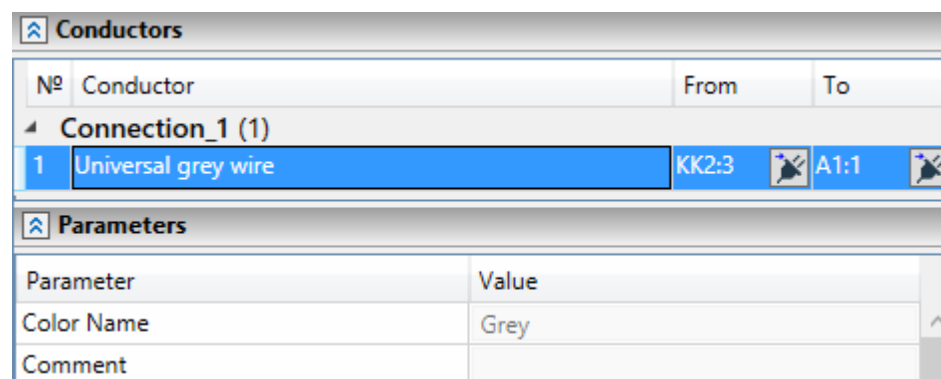
Use the following option to cancel connection line selection:

	<D>	Clear
---	-----	-------

[Automenu options common for all commands](#)

You can select a wire that is used for the connection in the parameters window. Parameters are displayed for the wire. You can also change connection points for the wire using items **From** and **To**.


You can select endings for the wire. Press on the  icon and select ending from the drop-down list.



The wire parameters are displayed the **Parameters** section. They may be changed, if necessary.

Cable

Use the following command for a cable creation:

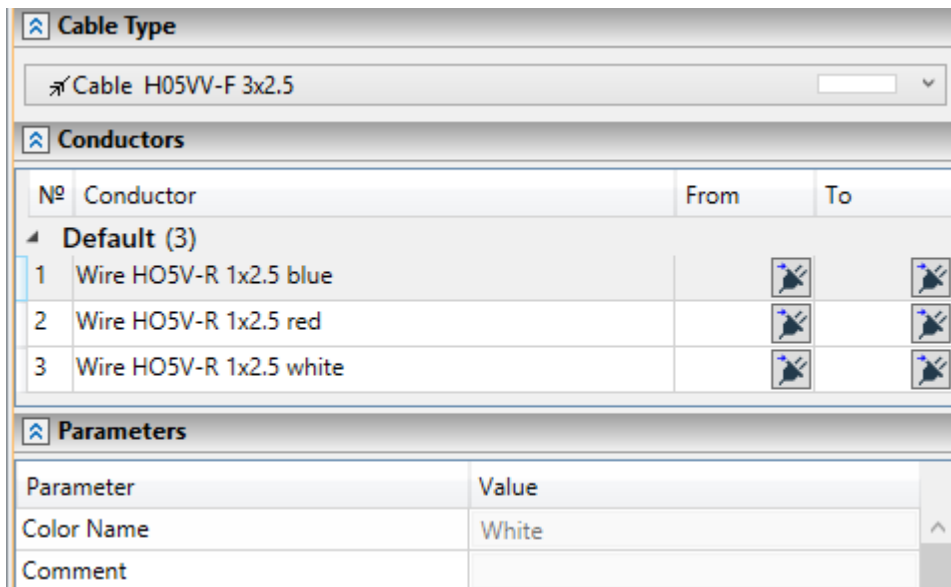
Icon	Ribbon
	Electrical ? Cable Components ? Cable
Keyboard	Textual menu
	Electrical > Cable Components > Cable

After the command is called the parameters window appears.

You should select type of the cable there.

A list of conductors and parameters will be displayed for the type. You can't change type of the conductor for the cable.

You can select endings for the cable. Press on the  icon and select ending from the drop-down list.



The cable parameters specified in the components editor are displayed in the **Parameters** section. They may be changed, if necessary.

You can select wires that will be included in the cable using one of the following ways:

1. Select connection line on the diagram. A cable will be created for the connection line in 3D.


	<Y>	Select Connection Line on the Diagram
---	-----	---------------------------------------

You can select normal or group connection line of the diagram.

Use the following option to cancel connection line selection:


	<D>	Clear
---	-----	-------

2. You can select already existing in the 3D scene wires. Use the following option:


	<U>	Add Conductor to Cable Component
---	-----	----------------------------------

After activation you should select wires that will be added to the cable.



You can separate wires from the cable using the option:

		Conductor as Separate Component
---	--	---------------------------------

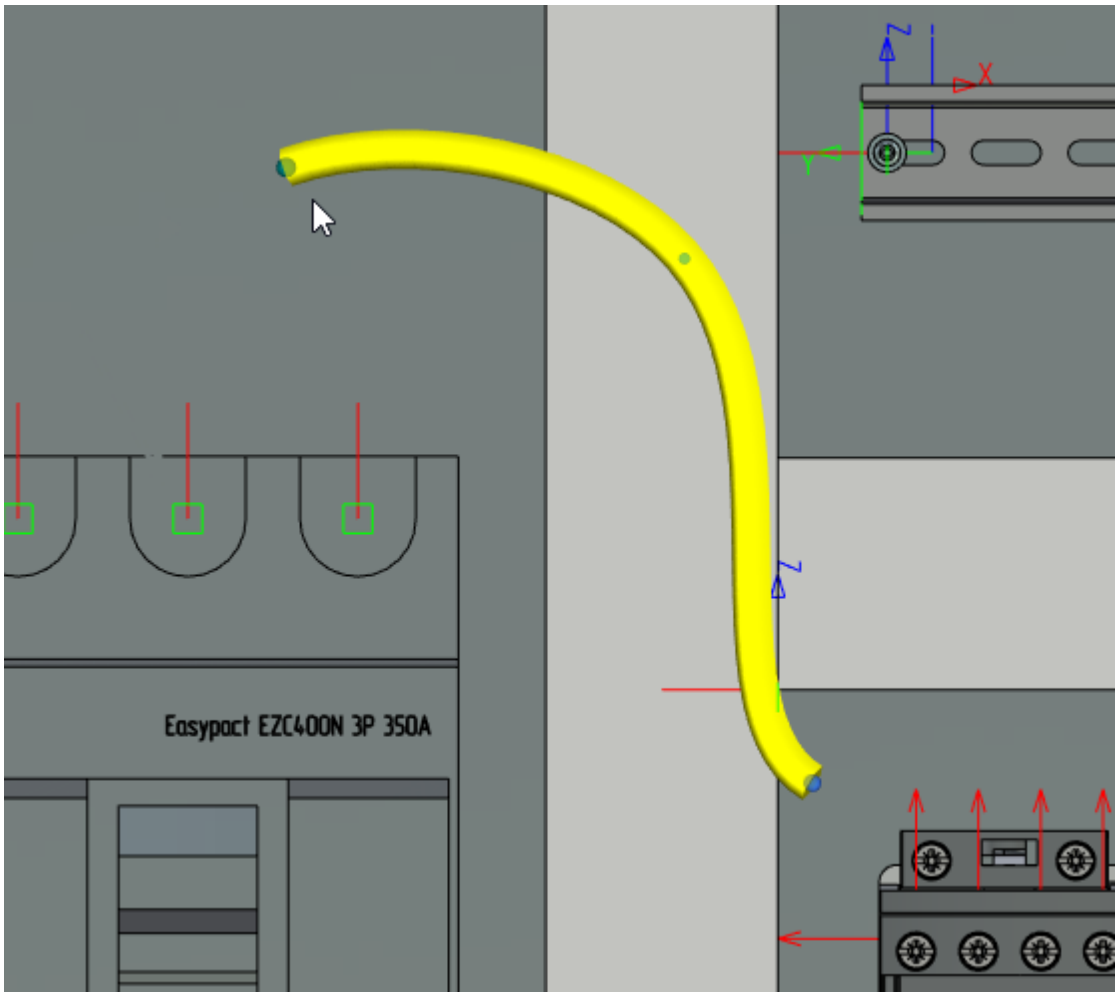
To separate all wires from the cable use the option:

	All Conductors as Separate Components
---	---------------------------------------

You can create a sheath of the cable. You should specify two points in the 3D scene. For this use the following options:

	<F>	Start Point of Sheath
	<L>	End Point of Sheath


You should select two connection points in the 3D scene to create a sheath. A sheath body will appear after that. There are manipulators on the ends of the sheath body. You can change position of start and end of the cable sheath using the manipulators. When you click on the sheath, standard manipulators for cable components appear. They allow to set the required position of the cable.



[Automenu options common for all commands](#)

Harness

Use the following command for a harness creation:

Icon	Ribbon
	Electrical ? Cable Components ? Harness
Keyboard	Textual menu
	Electrical > Cable Components > Harness

To create a harness follow the steps:

1. Select harness type in the parameters window;
2. Select 3D path or route for the harness;
3. Add conductors to the harness.

You can select conductors that will be included in the harness using one of the following ways:

You can select a connection line on the diagram. A harness for the connection line will be created.


	<Y>	Select Connection Line on the Diagram
---	-----	---------------------------------------

You can select normal or group connection line of the diagram.

Use the following option to cancel a connection line selection:


	<D>	Clear
---	-----	-------

Use the following option to add wires and cables to the harness:


	<U>	Add Conductor to Cable Component
---	-----	----------------------------------

After activation you should select wires and cables that will be added to the harness in the 3D scene.


You can exclude wires and cables from the harness using the option:

		Conductor as Separate Component
---	--	---------------------------------

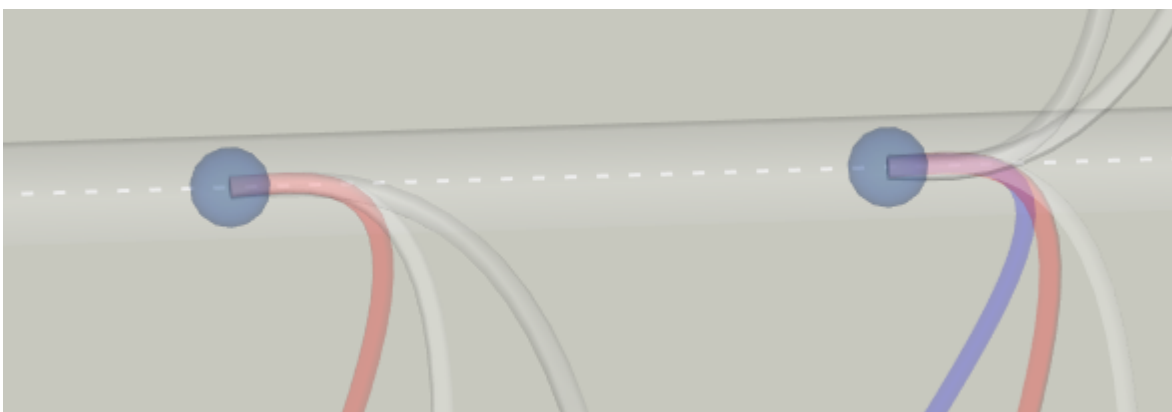
To exclude all wires and cables from the harness use the option:

		All Conductors as Separate Components
---	--	---------------------------------------



To optimize the position of entry points for wires into the harness, the following option is used:

		Optimize Conductors
---	--	---------------------

When harness is created, user can manually set entry point for wires and cables. The option allows to optimize wires so they enter the nearest point and have minimal length. To create an entry point you should click in the harness with pressed <Ctrl>.

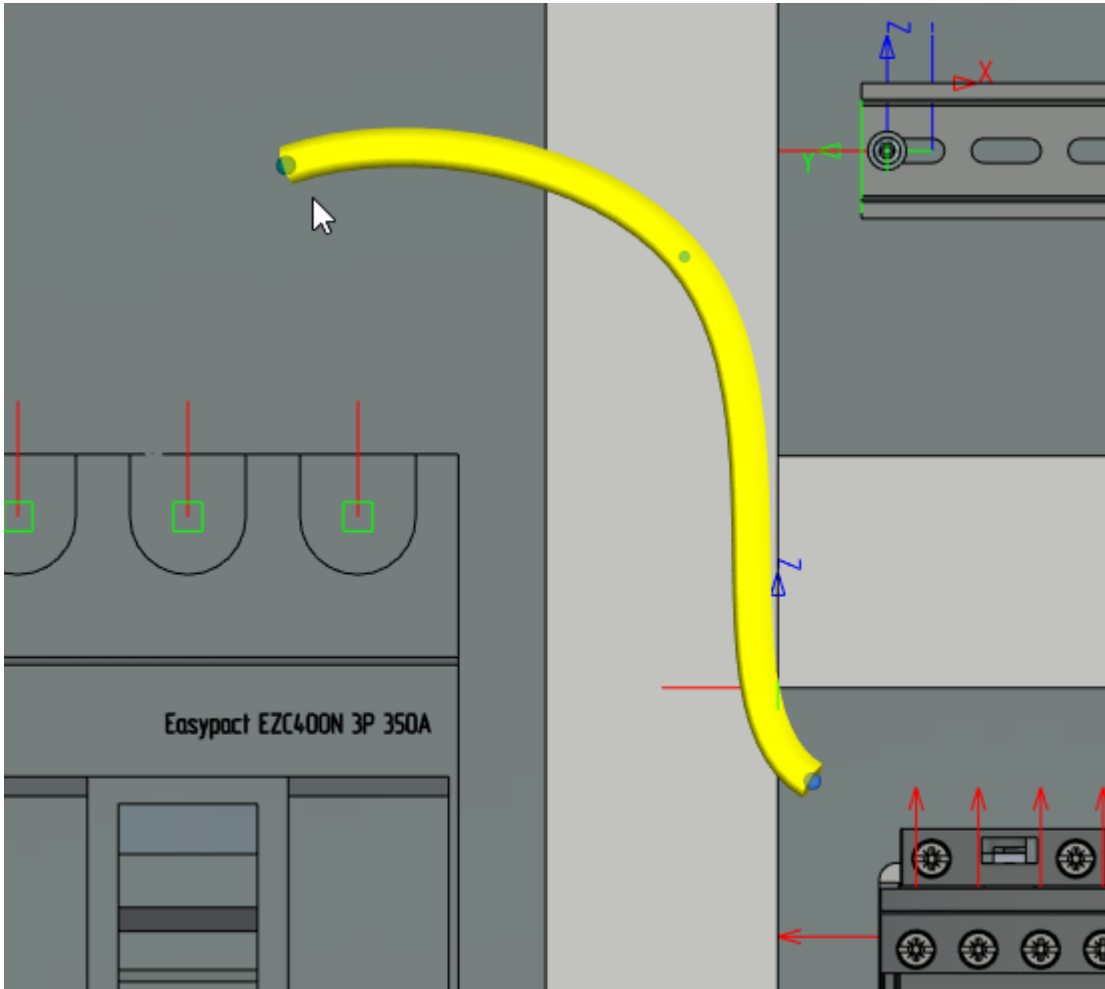


You can create a harness sheath by specifying two connection points in the 3D scene:

	<F>	Start Point of Sheath
	<L>	End Point of Sheath

You should select two connection points in the 3D scene to create a sheath. A sheath body will appear after that. There are manipulators on the ends of the sheath body. You can change position of start

and end of the cable sheath using the manipulators. When you click on the sheath standard manipulators for cable components appear. They allow to set the required position of the harness.



[Automenu options common for all commands](#)

Common automenu options

The following options are available in the automenu of each of the commands **Wire**, **Cable**, **Harness**:

The following option allows to select route along which the conductor or segment will pass:

	<P>	Select Route of Conductor/Sheath Segment
--	-----	--

Use the following option to cancel a route selection:

	<C>	Cancel Route of Conductor/Sheath Segment
--	-----	--


The following options allow to add reference points:

	<H>	Add Reference Point to the End of Segment
	<J>	Add Reference Point to the Beginning of Segment

The **Geometry type** option allows to specify type of the geometry for the conductor: spline or polyline.

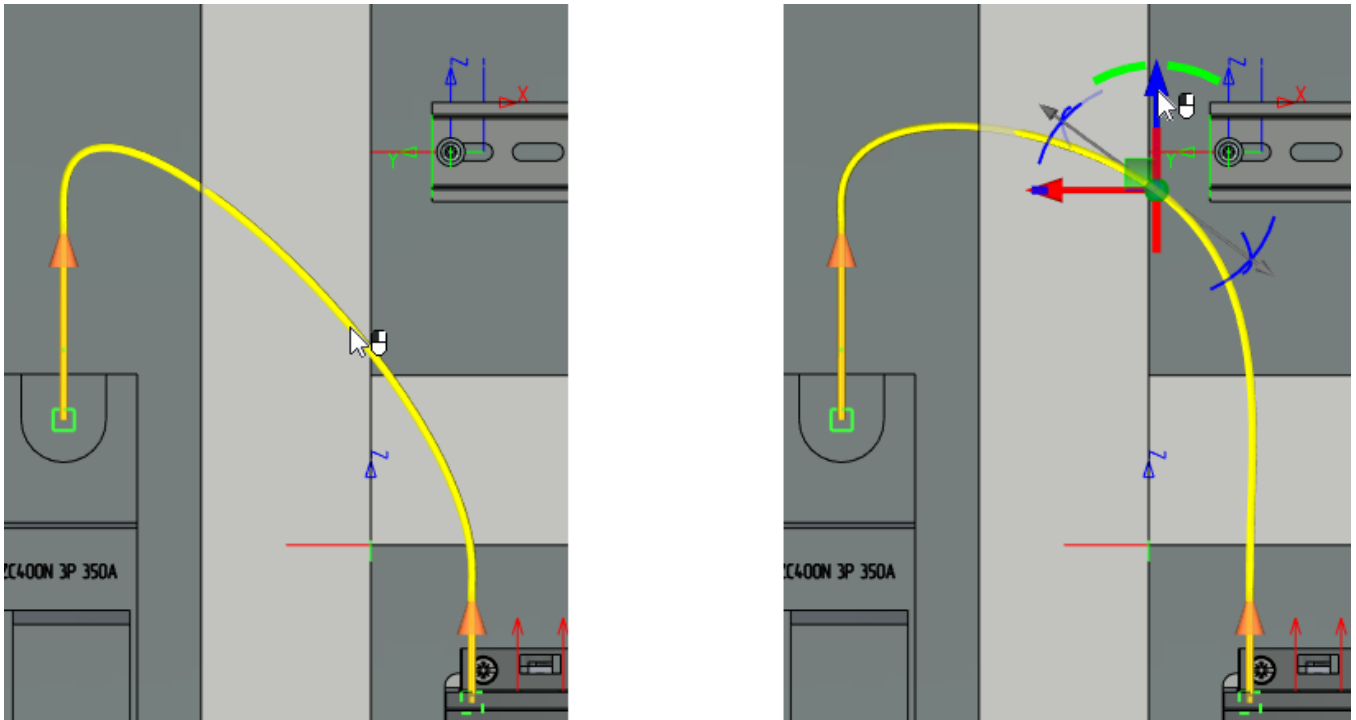
	<0>, <1>	Geometry type
--	----------	---------------

The **Representation** option allows to select a representation of the conductor in the 3D scene: **Wire**, **Faceted**, **Solid**.

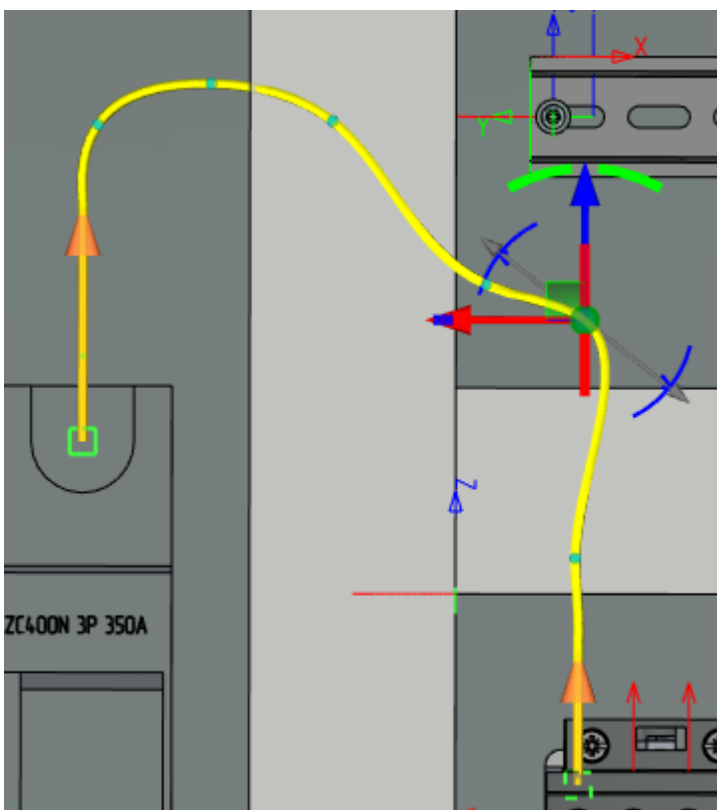
	<6>, <7>, <8>	Representation
---	---------------	----------------

Cable components manipulators

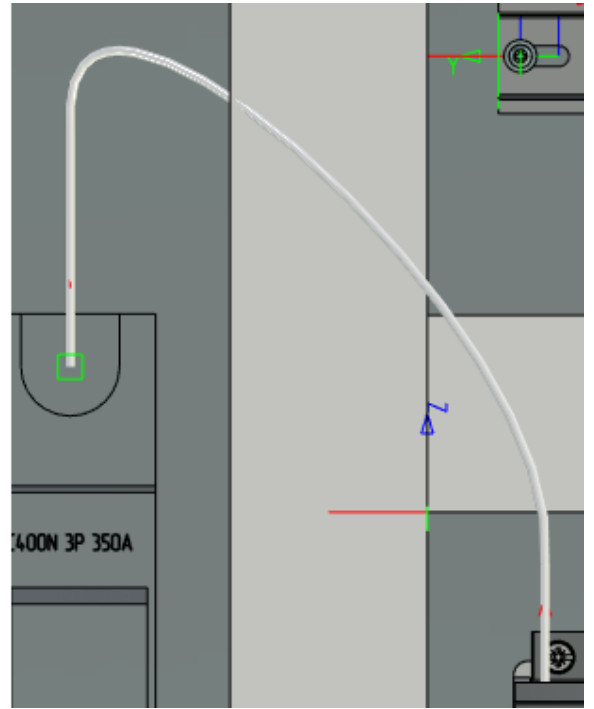
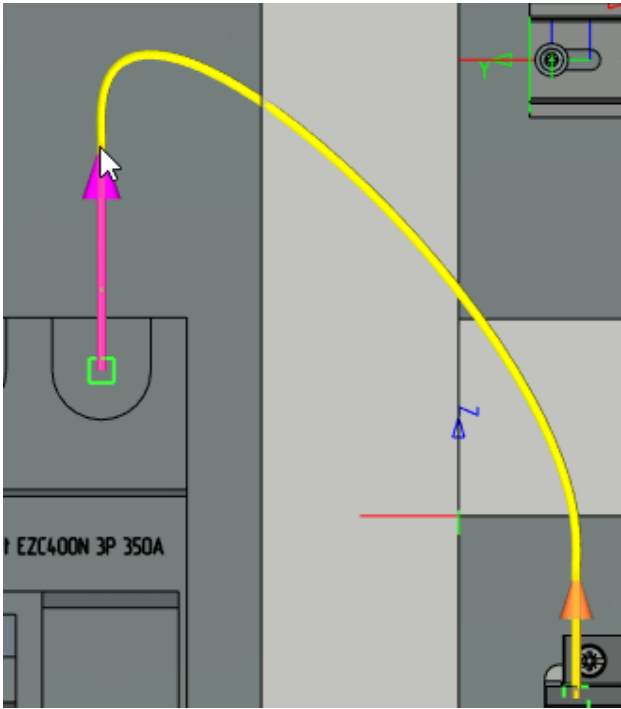
You can change cable component position using manipulator. You need to select cable component, call the **Edit** command and click on the cable component. The manipulator will appear.



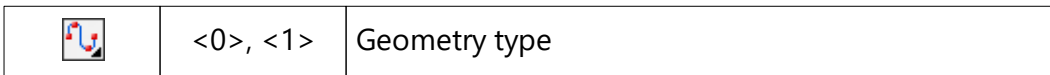
You can create any number of points and specify position of each of them using the manipulator.



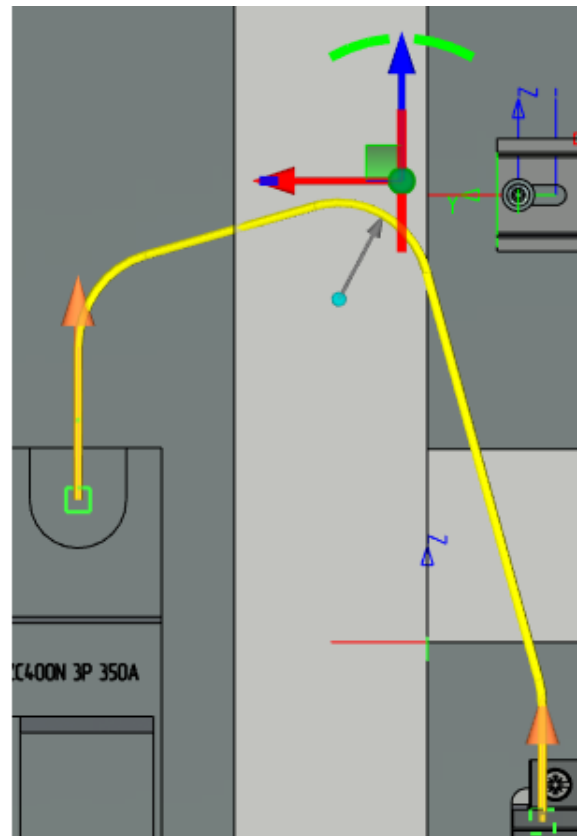
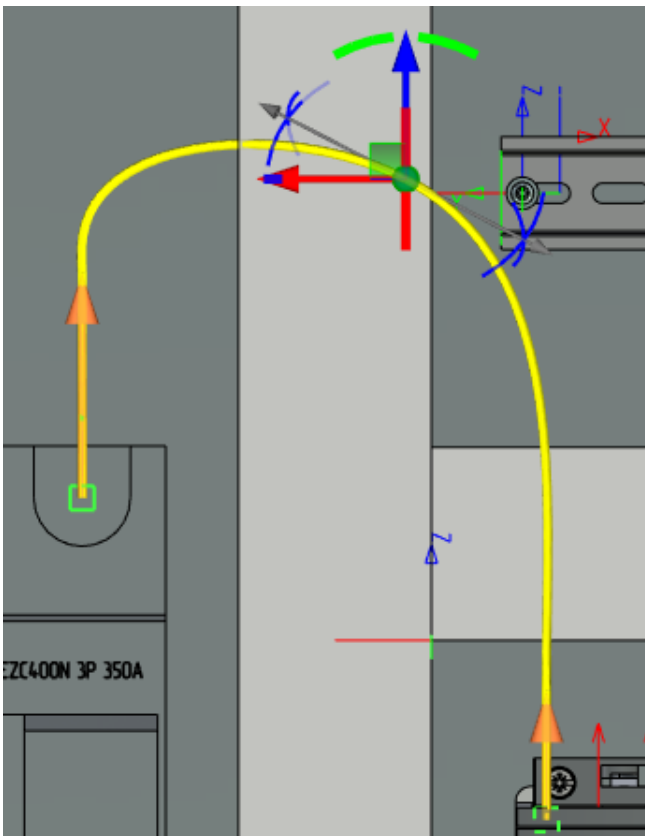
You can specify a straight segment for cable component using the special manipulator.



You can specify the geometry type using the option:




You can specify type of geometry: spline or polyline.



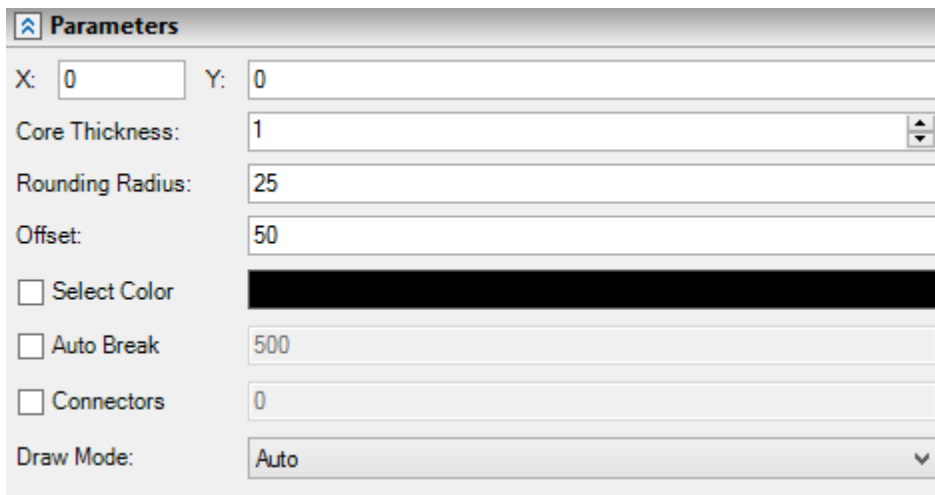
Harness layout

Use the following command to create a harness layout:

Icon	Ribbon
	Electrical ? Cable Components ? Harness Layout
Keyboard	Textual menu
	Electrical > Cable Components > Harness Layout

After the command is called, the parameters window appears.

Parameters group



X, Y specify point for the harness layout positioning on the drawing.

Core Thickness specifies thickness of the core on the layout.

Rounding Radius specifies visual radius of the rounding on the layout.

Offset specifies offset of layout lines according to each other.

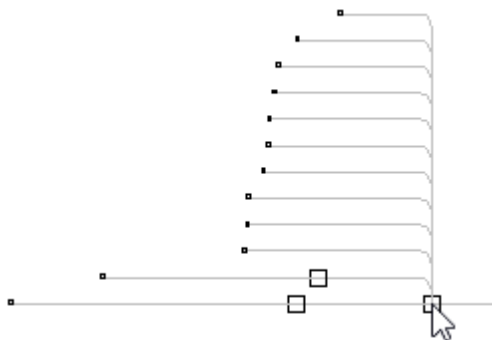
Select color - specifies color of the created lines.

Auto break specifies value of lines auto break.

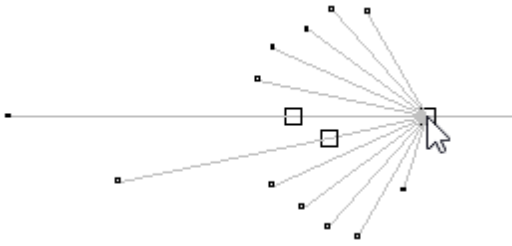
Connectors allows to show connectors on the harness layout and to specify their size.

Draw Mode allows to select drawing type: manual or automatic.

When you select automatic type, zero length segments are created to minimize intersections of harness segments. This is done to simplify visual perception.



When you select manual type the segments are not created. User can manually rotate segments of the harness.



The following option is available in the automenu:

	<D>	Create Dimensions Automatically
--	-----	---------------------------------

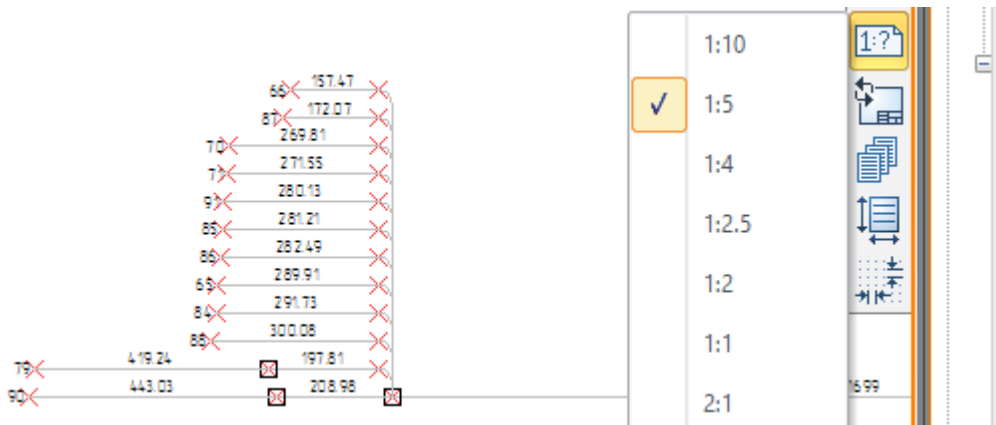
If the option is active, all dimensions will be automatically created on the layout.

A **Create dimensions with arrows** option allows to create dimensions with arrows on the harness layout. If the option is disabled the dimension will be displayed above the core.

	<F>	Create Dimensions with Arrows
--	-----	-------------------------------

It is recommended to create a new page for harness layout using the command **PG: Pages**.

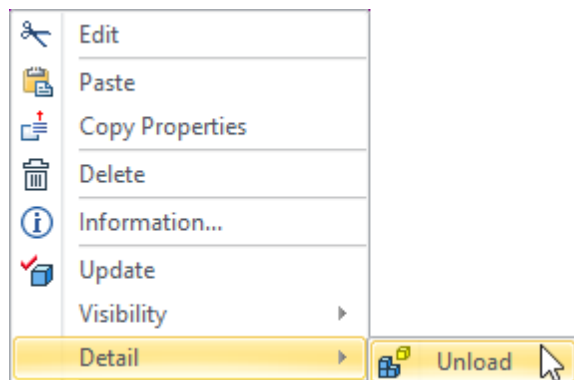
After the layout is created you can manage scale, so the layout will fit the page.



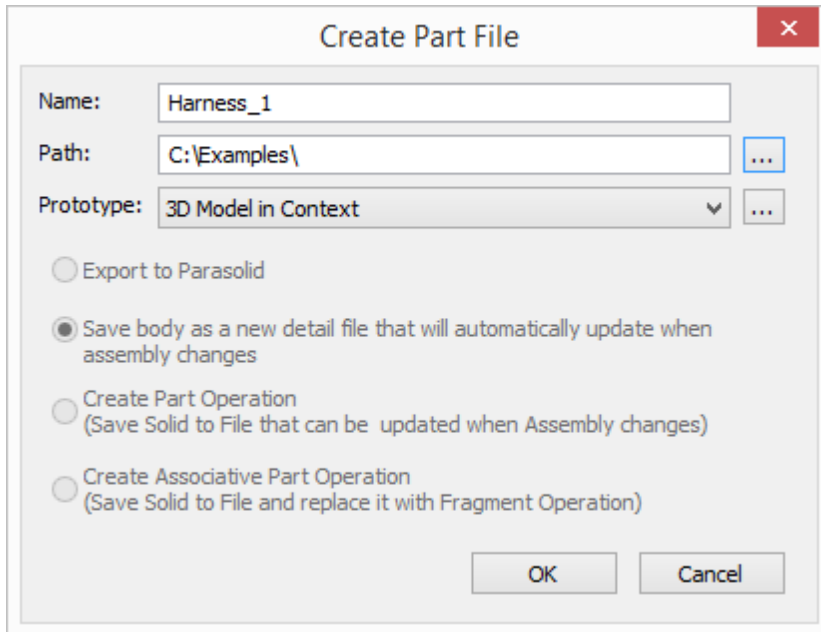
Unload harness to detail

You can unload a harness from the assembly in a separate detail to create a separate report for it.

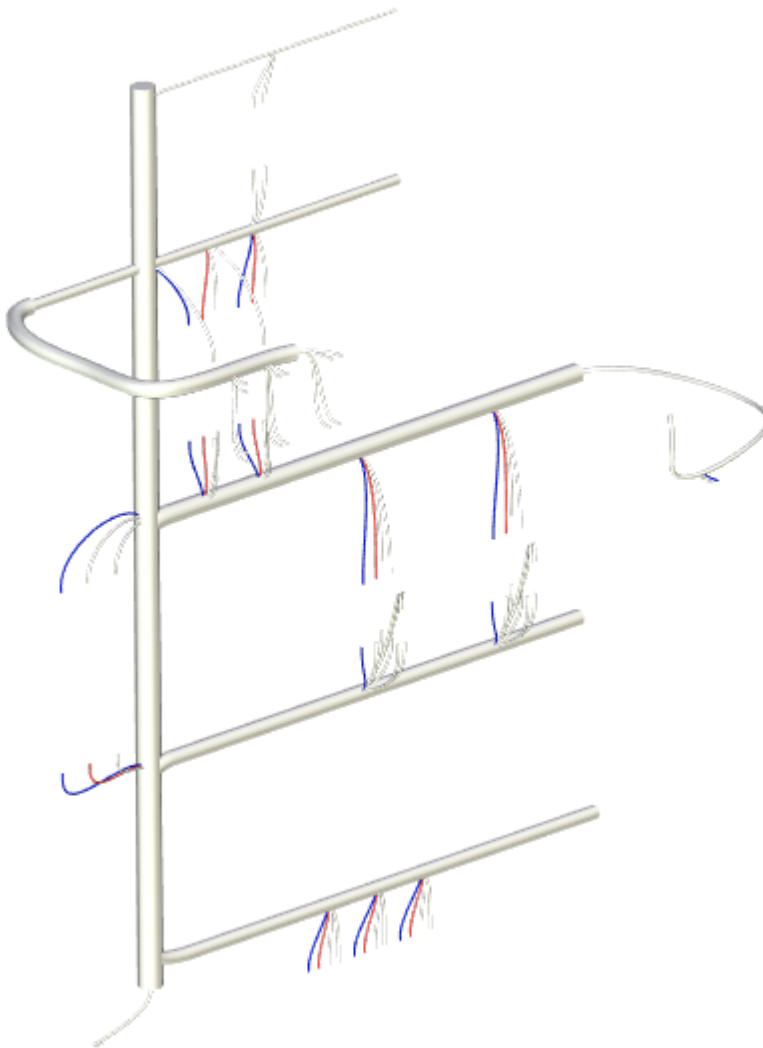
You should select the harness in the model tree and select **Detail > Unload** in the context menu.



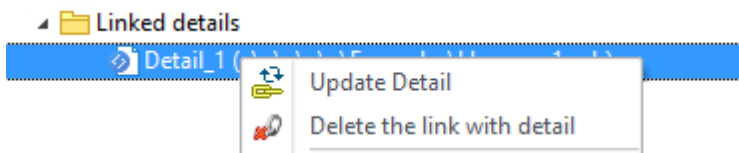
You should specify path for saving the file in the appeared dialog.



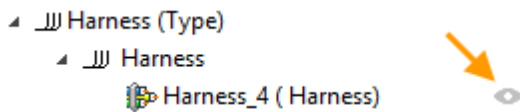
The harness will be unloaded in a separate file. You can create a report for the harness.



The data in the unloaded file can be updated according to the assembly changes. You can't transfer changes in the harness from the unloaded file back to the assembly.



You can hide harness in the assembly when it is inserted in another assembly.



Reports in 3D assembly

When all cable components are created user can create reports.

You can create a ["From/to" table](#) and [Connection table](#) that will be supplemented with information of the cable components.

You can also create additional reports.

Use projection to create a components layout diagram. Call the command **3J: Create 2D projection**.

When you create projection for cable components, the solid representation is activated automatically. It allows to display cable components on the projection.

To create a bill of materials you should open the product structure window using <alt+9> and select "Electrical. BOM" type.

Contact us to request information about T-FLEX software, our
Academic Program, or if you have ideas on cooperating
with Top Systems

www.tflex.com/mail

Contact Us



www.tflex.com

(7-499) 973-20-34
(7-499) 973-20-35

