



## Graphical Connection Plans

In PCISCHEMATIC Automation, you can automatically create graphical connection plans for the active project.

This supplementary manual for the PCISCHEMATIC Automation manual describes how this is possible.

Additional information about Graphical Terminal Plans and Graphical Cable Plans can be found at the end of this manual.



# GRAPHICAL CONNECTION PLANS

In **Automation** it is possible to create graphical connection plans for the active project.

THIS CHAPTER DESCRIBES	
— ON GRAPHICAL CONNECTION PLANS .....	(P. 3)
— STARS IN CONNECTION PLANS .....	(P. 4)
— UNIT CONNECTION DIAGRAMS, ACCORDING TO IEC1083-3. ....	(P. 4)
— MORE THAN ONE CONNECTION PLAN IN SAME PROJECT .....	(P. 4)
— CREATING GRAPHICAL CONNECTION PLANS .....	(P. 5)
— SETTINGS IN BUILD GRAPHICAL CONNECTION PLAN .....	(P. 6)
— THE TAB GENERAL .....	(P. 6)
— THE TAB CRITERIA .....	(P. 8)
— THE TAB PAGES .....	(P. 11)
— THE TAB SYMBOLS .....	(P. 12)
— THE TAB WIRES .....	(P. 16)
— UPDATING EXISTING GRAPHICAL CONNECTION PLANS .....	(P. 17)

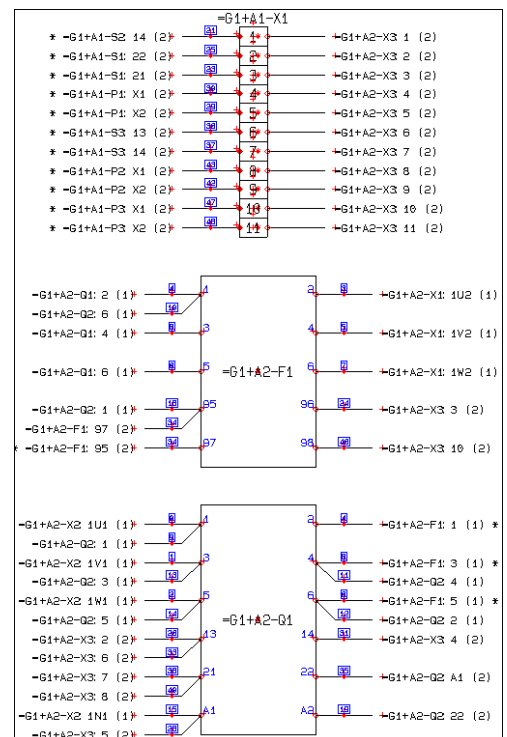
## ON GRAPHICAL CONNECTION PLANS

The graphical connection plan can either be created as an individual **Automation** project, or be inserted in the active project.

When the connection plan is created, you can edit the created connection plan by moving lines and symbols, just like on any other diagram page.

— If a connection plan does not turn out the way you want it to, you can change the settings, and create a new one.

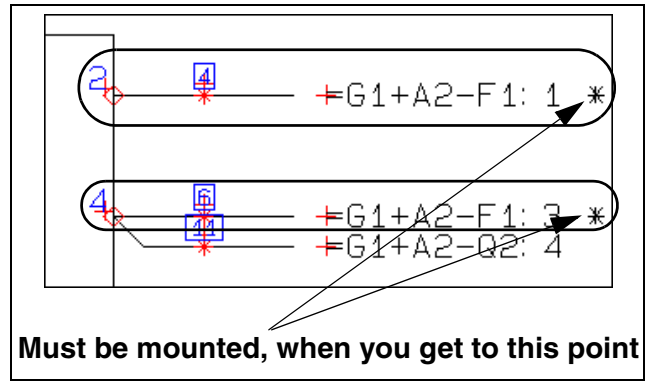
Connection plans are printed via the printing dialog box, as for ordinary pages in **Automation** projects.



**STARS IN CONNECTION PLANS**

When you mount based on the connection plan, and there is a star (\*) next to a wire, this means that this wire should already be mounted, when you get to this point.

This is only the case if you mount the components following the exact order, in which they appear in the connection plan — starting with the first page of the connection plan.



**UNIT CONNECTION DIAGRAMS, ACCORDING TO IEC1083-3**

The graphical connection plans which you can create here, are called *unit connection diagrams using interrupted lines* in the IEC1083-3 standard.

**MORE THAN ONE CONNECTION PLAN IN THE SAME PROJECT**

It is possible to create more than one graphical connection plans in the same project. When you wish to update a graphical connection plan, you must select one of the pages containing the connection plan in the project, and choose **Tools => Graphical connection plan**.

## CREATING GRAPHICAL CONNECTION PLANS

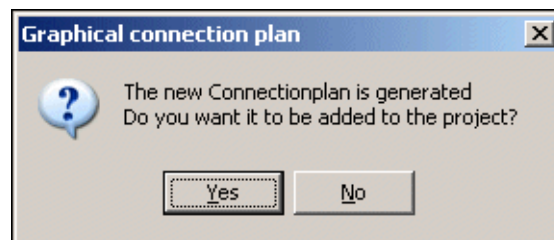
To create a graphical connection plan, choose **Tools => Graphical connection plan** in **Automation**. You then enter the dialog box **Create new graphical connection plan**, which is described in the following.

### Updating a graphical connection plan

If you have selected a page in a project, which contains an existing graphical connection plan, you enter the dialog box **Update graphical connection plan**, which is described in “Rebuilding existing connection plans” on page 17.

### Inserting the connection plan in the project

If you have *not* chosen a connection plan page in the project, a new graphical connection plan is created. When the connection plan is created, you are asked whether to insert it in the project:



*Answer Yes - to insert the connection plan in the project*

If you answer **Yes** to insert the connection plan in the project, together with a tab page named **Connectionplan**.

*Answer No - to insert the connection plan in the project*

If you answer **No** in the dialog box, the connection plan is created as an independent **Automation** project with the name **Connectionplan**.

### *Page type for connection plans*

The pages in the connection plan are assigned the page type **CNP**. In **Automation** they therefore have no influence on the electrical and mechanical lists in the project.



#### **NOT UPDATED AUTOMATICALLY**

IF YOU MAKE CHANGES IN THE DIAGRAM PAGES IN THE ORIGINAL **Automation** PROJECT, THE CONNECTION PLAN IS *NOT* UPDATED AUTOMATICALLY. TO UPDATE THE CONNECTION PLAN, YOU MUST THEREFORE CHOOSE **TOOLS => GRAPHICAL CONNECTION PLAN** AGAIN — AS DESCRIBED IN “REBUILDING EXISTING CONNECTION PLANS” ON PAGE 17.

### SETTINGS IN BUILD GRAPHICAL CONNECTION PLAN

The dialog box **Build graphical connection plan** contains the following tabs:

TAB	FUNCTION	PAGE
<b>GENERAL</b>	HERE YOU SPECIFY THE ORIENTATION OF THE GRAPHICAL CONNECTION PLAN.	(P. 6)
<b>CRITERIA</b>	HERE YOU SPECIFY CRITERIA FOR WHICH COMPONENTS YOU WISH TO INCLUDE IN THE CONNECTION PLAN.	(P. 8)
<b>PAGES</b>	HERE YOU CHOOSE A DRAWING HEADER FOR THE CONNECTION PLAN.	(P. 11)
<b>SYMBOLS</b>	HERE YOU CHOOSE WHICH TERMINAL SYMBOL TO APPLY, AND WHETHER TO USE BLACKBOX SYMBOLS.	(P. 12)
<b>WIRES</b>	HERE YOU CHOOSE WHICH TEXTS TO PLACE NEXT TO THE WIRES IN THE DOCUMENTATION.	(P. 16)

The functions on the tabs are described in the following.

#### Create the graphical connection plan

When you have made the desired choices on the tabs, click on **OK** in the right-hand side of the dialog box. Then the graphical connection plan for the chosen components is created/updated.

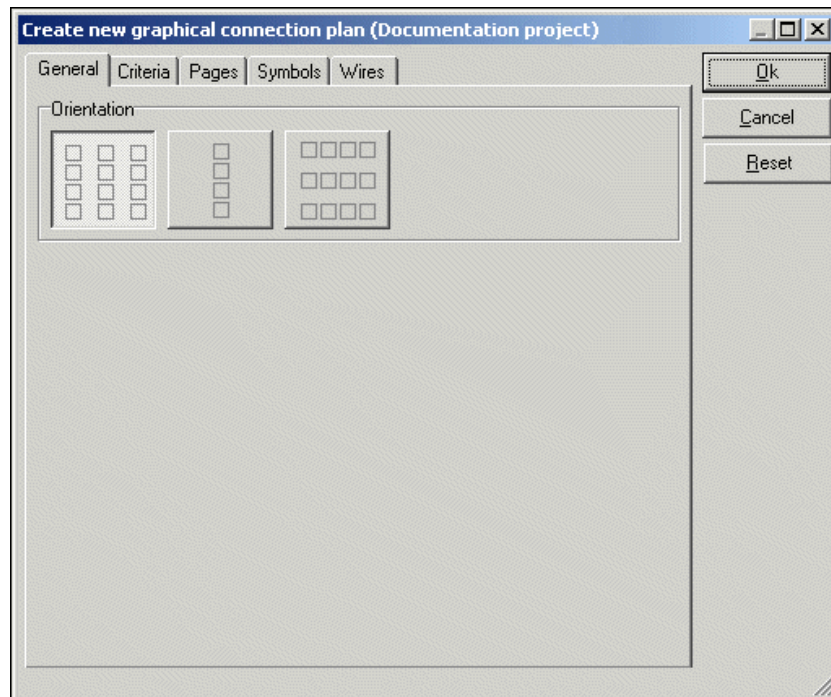
If you click **Cancel**, you exit the dialog box, and no connection plan is created.

If you click **Reset**, the settings in the dialog box are reset to the default settings for the program.



#### GENERAL

On the tab **General** you specify the orientation of the graphical connection plan:



You have the following three options:

**Columns vertically**

Click here to create a connection plan, where the components are placed in as many vertical columns as possible on each page.

**One column vertically**

Click here to create a connection plan with one vertical column of components on each page.

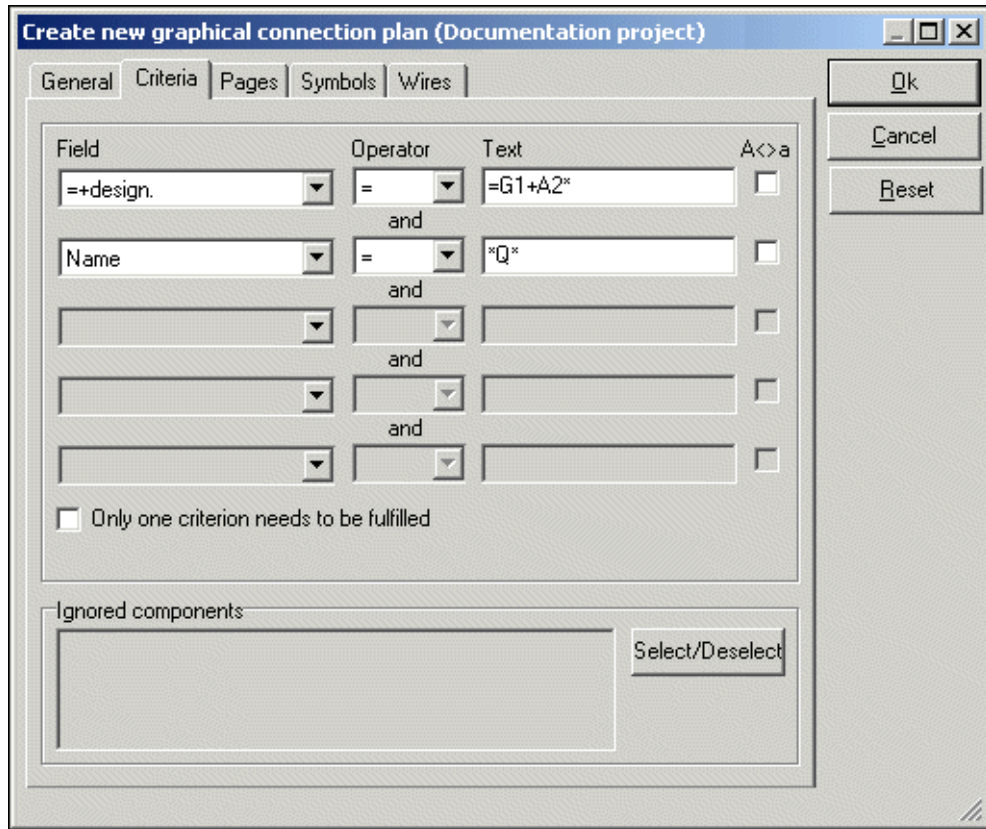
**Rows horizontally**

Click here to create a connection plan, where the components are placed in as many horizontal rows as possible on each page.



**CRITERIA**

On the tab **Criteria** you set up the criteria for which components to include in the connection plan:

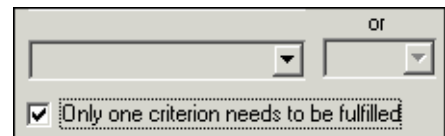


You can specify up to five criteria.

**Only one criterion needs to be fulfilled**

When you choose **Only one criterion needs to be fulfilled** (logical OR), the text *and* changes to *or* in the dialog box.

In the dialog box above this would implicate that you would get all components with reference designations starting with **=G1+A2**, and all components with a **Q** in the symbol name.



If **Only one criterion needs to be fulfilled** is *not* selected, the text *and* is displayed between the criteria lines. This specifying that *all* of the criteria must be fulfilled (logical AND). In the dialog box above, this implicates that you will get all components having a reference designation beginning with **=G1+A2** and a **Q** in the symbol name.



### Specifying a criterion

When clicking under **Field**, you specify which datafield the criterion is set for. This being one of the symbol texts **Name**, **Type**, **Article No.** or **Function**, or a reference designation (**=+design.**).

If you set **Operator** to =, you can type in which text, that the contents of the datafield must be identical to, under **Text**.

If the contents of the datafield must be different from what you type in under **Text**, set the **Operator** to <>.

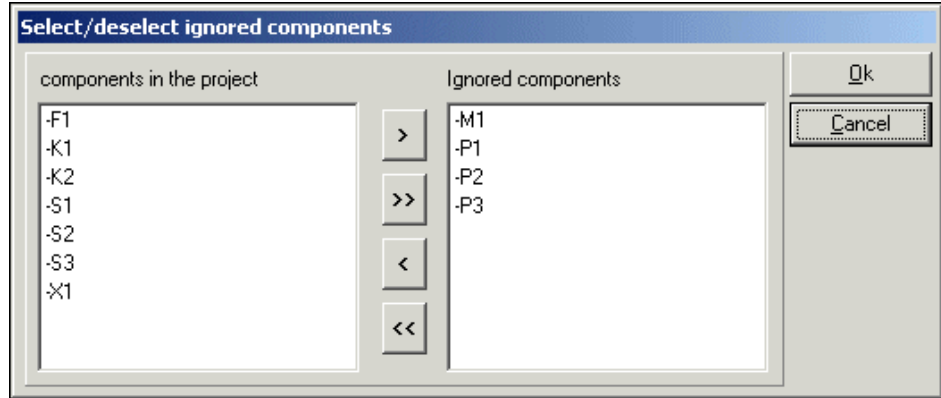
To specify whether the contents of the datafield must be larger than or smaller than the specified text, set the operator to > or < respectively. Please note that you hereby specify whether the text comes before or after the criterion alphabetically.

When you select **A<>a**, you specify that there shall be made a distinction between upper case and lower case letters.

WILDCARDS
<p>When for instance you wish to document all components having the letter <b>Q</b> in the name, you need a way of specifying this to the program.</p> <p>For this purpose, you can use the so-called wildcards, which are also used in Windows. These characters are <b>?</b> and <b>*</b>.</p>
<p>When you place a <b>?</b> somewhere in the text, this means that there <i>must</i> be one (and only one) character on this position.</p> <p>When you place a <b>*</b>, this indicates that there can be an arbitrary number of characters on this position.</p>
<p>If for instance you wish to extract all components with exactly three letters, with the letter <b>Q</b> as the second character, you write <b>?Q?</b>. This specifies that there shall be one — and only one — character before <b>Q</b>, and one — and only one — character after.</p> <p>If you wish to extract all components with an <b>A</b> anywhere in the text, write <b>*A*</b>.</p>
<p>— Please note that when you write <b>?</b> there <i>must</i> be a character on this position, but when you write <b>*</b> this also covers the case where there are no characters.</p>

### Ignored components

Unless specified otherwise, the program documentates all components in the project. If this is not what you wish — and you have not excluded one or more components via the specified criteria —, just click on **Select/Deselect** in the lower-right corner of the tab. You then enter the following menu:



When you wish to ignore a component, click on the component in the left-hand side, and then click on the right-arrow in the dialog box.



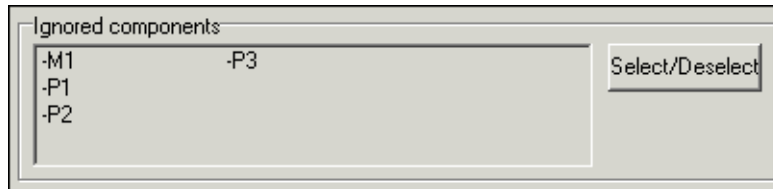
When you wish to include an ignored component, click on the component in the right-hand side, and then click on the left-arrow.



By clicking on the double left-arrow and the double right-arrow respectively, you transfer all components from the one column to the other.

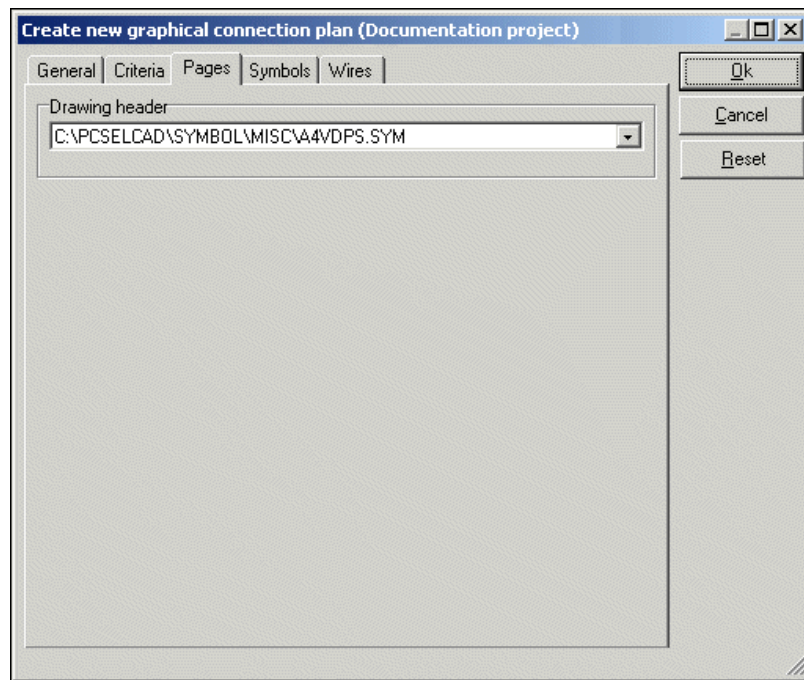
Click **OK** when you have selected/deselected the desired components.

The ignored components are now displayed at the bottom of the tab **Criteria**:



## PAGES

On the tab **Pages** you select which drawing header to use in the graphical connection plan.



### Drawing header

In the field **Drawing header** you specify which drawing header to use in the created connection plan.

When you click the down-arrow in the **Drawing header** field, you get the list of drawing headers in **Automation**, and can choose which drawing header to use. You can only use drawing headers, which are in this list.

#### *User-defined drawing headers*

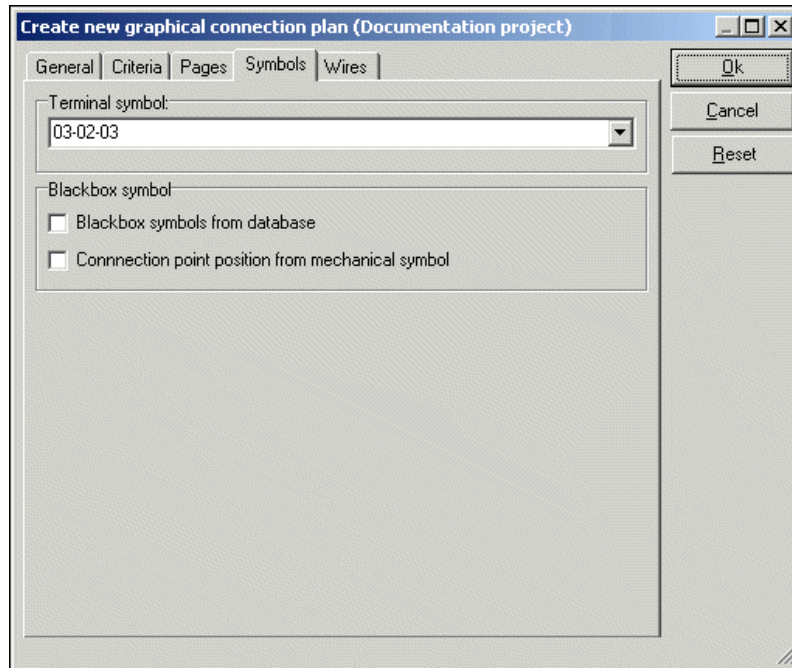
If you have designed your own drawing header for connection plans, you must first add it to the list of drawing headers in **Automation**.

When you have done this, you will be able to choose the drawing header on the tab **Pages**.

— Read more about this, and about designing drawing headers, in the **Automation** manual.

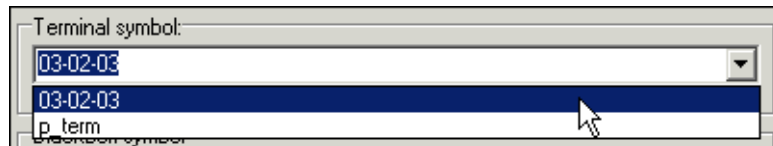
## SYMBOLS

Here you specify which terminal symbol to use in the graphical connection plan, and how to represent the individual components:

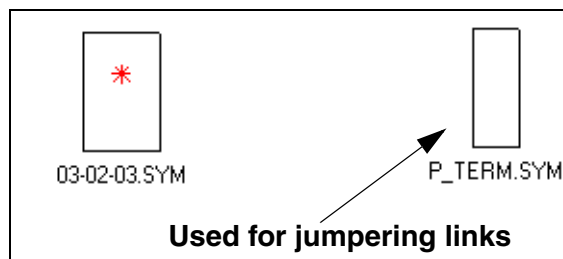


### Terminal symbol

In the upper part of the dialog box you choose which terminal symbol to use on the connection plan. When you click the down-arrow, you can choose between the previously used symbols.



The terminal symbols, which you can choose from by default, looks like this:



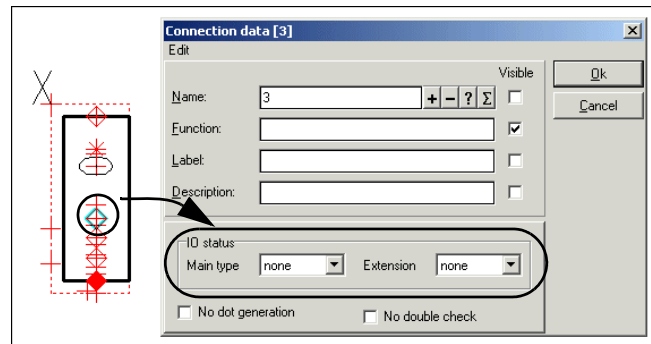
These symbols are only examples on what you can use for representing terminals in the connection plan.

If you wish to use another terminal symbol, you can just type in the name of the symbol in the **Terminal symbol** text field.

### Terminal symbol for displaying Jumpering links in the connection plan

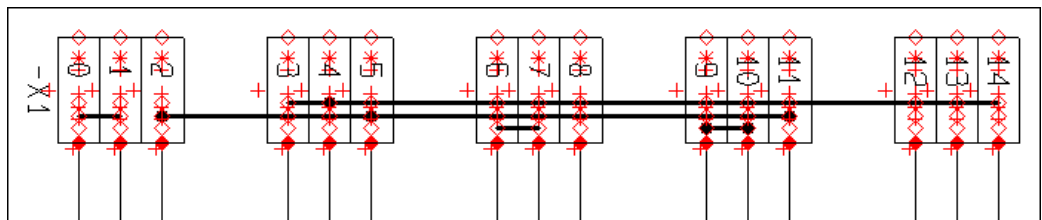
To display *jumpering links* in the connection plan, the applied symbol must have extra connection points. These must have **IO status Main type** and **Extension** set to **none** — as displayed in the figure to the right.

Read more about designing symbols in the **Automation** manual.



The symbol **P\_TERM.SYM** can be applied for displaying jumpering links in the connection plan.

Jumpering links are displayed in connection plans as straight lines between the connection points in the terminal symbols.



### Path to the terminal symbol

The file name for the terminal symbol can be typed in with or without a *path* specifying which folder the symbol is placed in.

When you type in the full path along with the symbol name, the program will always be able to locate the symbol.

As you can see above, only the symbol name is written in the **Terminal symbol** field. This is because the symbols are placed in a folder, which has an *alias* in the **Alias for libraries** dialog box.

You can enter this dialog box by choosing **Settings => Directories**, and then click on **Show directory alias**.

— Read more about alias in the **Automation** manual.

**Blackbox symbols**

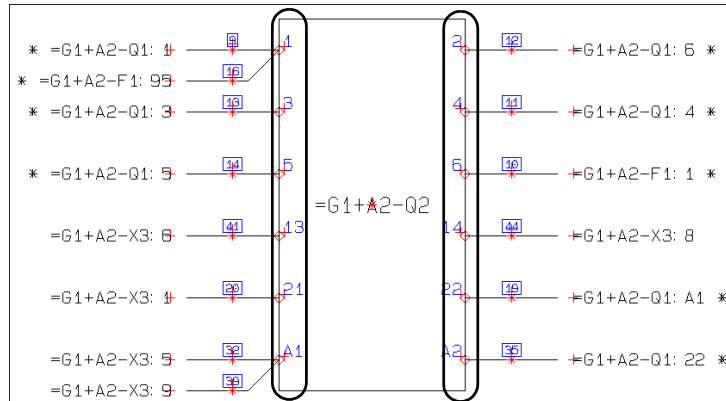
When you create a graphical connection plan, the program automatically generates blackbox symbols, which represents the components in the connection plans.

These symbols are boxes with connection points on the sides.

Which connection points, that are placed *where* on the symbols, can be determined in two ways:

*Let the program place the connection points*

If you do *not* select **Blackbox symbols from database**, the connection points on the symbols in the connection plan, are placed the following way:



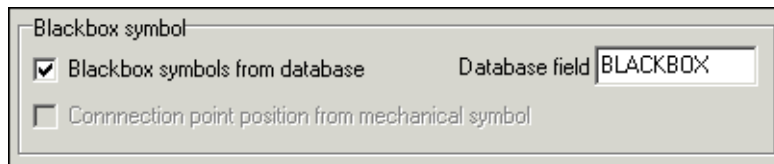
The principle is that connection points with *odd numbered* connection names are placed on the *left-hand side* of the blackbox symbol, and connection points with *even numbered* connection names on the *right-hand side* of the blackbox symbol.

*Fetch connection point positions from mechanical symbols*

If you choose to use the blackbox symbols, which the program creates (this meaning you have *not* selected **Blackbox symbols from database**), and select **Connection point position from mechanical symbol**, the program fetches the positions of the connection points from the mechanical symbols from the components database. Then the connection points are placed on the blackbox symbols, as they are placed on the mechanical symbols.

*Blackbox symbols from database*

If you select **Blackbox symbol from database**, you can specify from which field in the database, that the program shall fetch the symbols for the connection plan.



In the field **Database field** you specify which field in the database that contains the file names for these symbols.

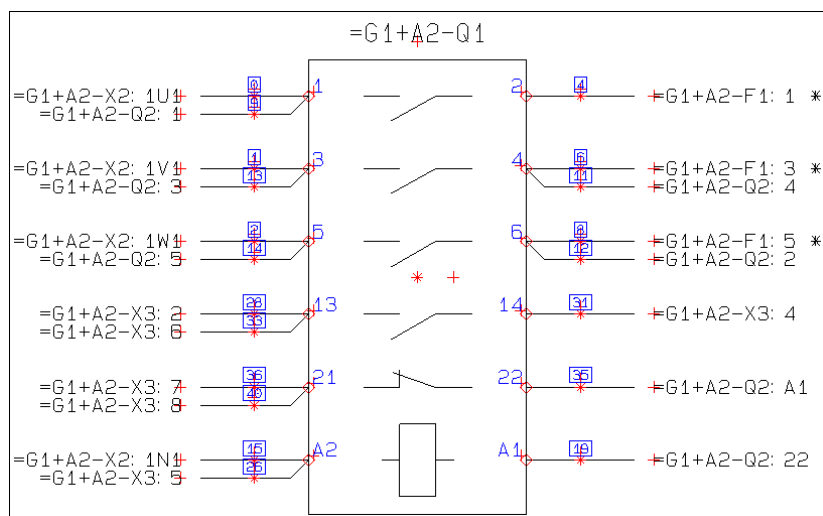


**USER-DEFINED SYMBOLS FOR CONNECTION PLANS**

IT IS *NOT* INTENDED THAT YOU SHALL APPLY THE PRE-DEFINED MECHANICAL SYMBOLS, THAT ARE ATTACHED TO THE SYMBOLS.

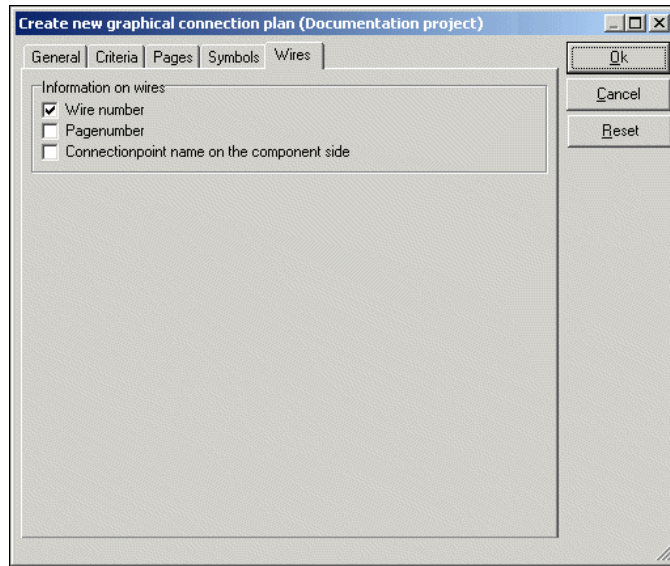
ON THE CONTRARY, THIS GIVES YOU THE OPTION OF DESIGNING YOUR OWN SYMBOLS FOR THE CONNECTION PLANS.

On the connection plan symbols, which you create yourselves, you can therefore graphically display which connection points on the component, that has to do with make contacts, break contacts etc., as displayed in the figure below:



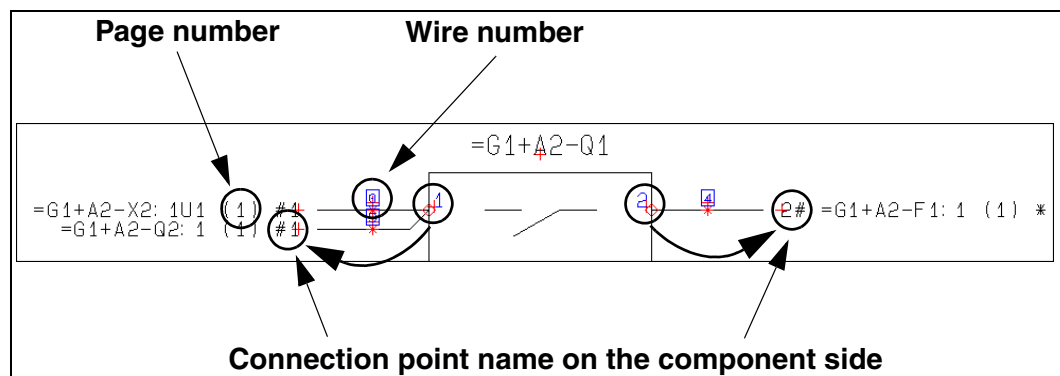
## WIRES

On the tab **Wires** you specify which information you wish to display on the connections.



You here have the following options to choose from:

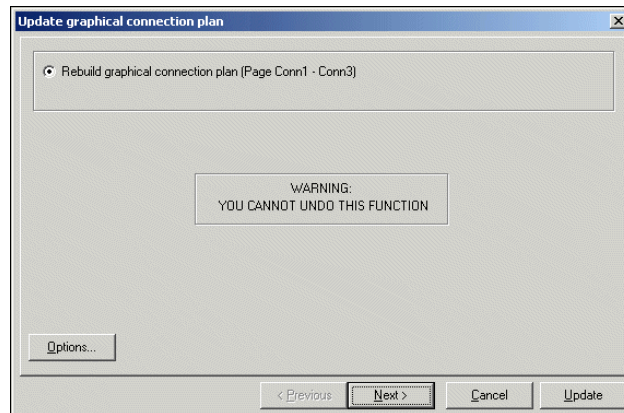
OPTION	FUNCTION
<b>WIRE NUMBER</b>	PLACES A WIRE NUMBER SYMBOL ON THE CONNECTION.
<b>PAGE NUMBER</b>	WRITES THE NUMBER OF THE PAGE, ON WHICH THE COMPONENT IS PLACED IN THE PROJECT, IN BRACKETS AFTER THE NAME ON THE CONNECTION POINT IN THE CONNECTION PLAN.
<b>CONNECTIONPOINT NAME ON THE COMPONENT SIDE</b>	WRITES THE NAME OF THE COMPONENTS CONNECTION POINT, IN THE TEXT, WHICH SPECIFIES WHICH COMPONENT, THAT THE DISPLAYED WIRES IN THE CONNECTION PLAN COMES FROM. THEREFORE, IF YOU REMOVE A COMPONENT FROM THE CONNECTION PLAN, YOU WILL STILL BE ABLE TO SEE WHICH CONNECTION POINT ON THE COMPONENT, THAT THE INDIVIDUAL WIRES SHALL BE CONNECTED TO.





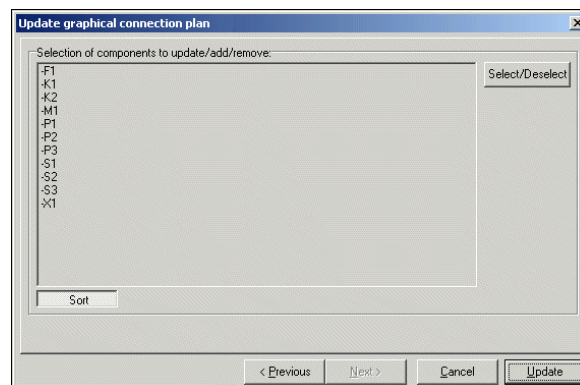
## REBUILDING EXISTING CONNECTION PLANS

If you have selected one of the pages for a created connection plan in the active **Automation** project, and choose **Tools => Graphical connection plan**, you enter the dialog box **Update graphical connection plan**:



If you click on **Options**, you enter the dialog box **Settings for connection plan**, which contains the same tabs and options, as described above.

When you then click on **Next**, you enter the following dialog box:



### *Sorting the components*

By default, the components are sorted according to the normal guidelines in **Automation** — see the **Automation** manual.

If you want the components to appear in another order, you can click on the components, and *drag* them to the desired position.

If you want the components sorted in the created connection plan again, click on the **Sort** button in the lower-left corner of the screen. When the components are sorted, the button will be activated.

In the generated connection plan, the components in the first column are documented first — starting with the uppermost component. Then the second column is documented, etc.

### *Click on Update*

Then click on **Update**, and the graphical connection plan is rebuild/updated.

# GRAPHICAL TERMINAL- AND CABLE PLANS

This chapter describes changes in the graphical terminal plans and cable plans in **Automation**.

THIS CHAPTER DESCRIBES	
— MORE THAN ONE GRAPHICAL PLAN IN THE SAME PROJECT.....	(P. 18)
— SETTINGS FOR CRITERIA FOR ALL TYPES OF GRAPHICAL PLANS.....	(P. 18)
— SCREEN ON CABLES DISPLAYED .....	(P. 18)

## MORE THAN ONE GRAPHICAL PLAN IN THE SAME PROJECT

It is now both possible to have more than one graphical terminal plan, and more than one graphical cable plan in the same project.

To update one of these cable-/terminal plans, you must select one of the pages in the relevant cable-/terminal plan, when you choose **Tools => Graphical cable-/terminal plan**.

By setting up criteria — see “Settings for Criteria for all types of graphical plans” on page 18 — you can hereby create graphical cable-/terminal plans, which represents different parts of the project.

### *Creating a new graphical cable-/terminal plan*

When you wish to create a *new* graphical cable-/terminal plan, you must first choose a page in the active project, which is *not* a graphical cable-/terminal plan, when you choose **Tools => Graphical cable-/terminal plan**.

## SETTINGS FOR CRITERIA FOR ALL TYPES OF GRAPHICAL PLANS

You can also set up *criteria* for which terminal rows to include in graphical terminal plans, and to set up *criteria* for which cables to include in graphical cable plans.

This is done via the tab **Criteria**, as described in “Criteria” on page 8.

Furthermore, **Select/Deselect** is moved to the tab **Criteria**.

## SCREENS ON CABLES DISPLAYED

Screens on cables are displayed in graphical terminal-/cable plans.