



Getting started

Table of contents

	0
Part I Getting started	1
1 Table of contents	1
2 Guide A1 - Computer settings	2
3 Guide A2 - Create your first project	2
4 Guide A3 - Project settings	2
5 Guide B1 - Create a new circuit diagram	3
6 Guide B2 - Draw logic symbols	3
7 Guide B3 - Move, copy and paste	6
8 Guide B4 - Undo and redo	8
9 Guide B5 - Create cable definition	9
10 Guide B6 - Define wire name	11
11 Guide B7 - Define wire types	13
12 Guide B8 - Create and use macro	15
13 Guide B9 - Define components to logic symbols	17
14 Guide B10 - Define engraving plates to logic symbols	17
15 Guide B11 - Using coils and its contacts	18
16 Guide B12 - Use I/O symbols	20
17 Guide C1 - Create a layout page	23
18 Guide C2 - Draw layout symbol from circuit diagram	24
19 Guide C3 - Draw layout symbol from component	25
20 Guide C4 - Use manual layout symbols	25
21 Guide C5 - Use device mounting rails	26
22 Guide C6 - Use cable duct objects	26
23 Guide C7 - Use measurement objects	27
24 Guide D1 - Generate wire list	29
25 Guide D2 - Generate cable pull list and cable connection list	30
26 Guide D3 - Generate bill of material	32
27 Guide D4 - Generate order list	34
28 Guide D5 - Generate engraving plate list	36
29 Guide D6 - Generate PLC list	38
30 Guide D7 - Generate table of contents	39
31 Guide D8 - Re-generate existing list	40
32 Guide E1 - Open the database library	41
33 Guide E2 - Add component	42
34 Guide E3 - Create new layout symbol	43
35 Guide E4 - Link a layout symbol with component	45
36 Guide E5 - Create a parent symbol	46
37 Guide E6 - Create child symbol to a parent symbol	49

38	Guide E7 - Create a new page header	52
	Index	0

1 Getting started

1.1 Table of contents

This document will teach you Elwin step by step in some of the most common operations.

A. Basics

[Guide A1 - Computer settings](#)

[Guide A2 - Create your first project](#)

[Guide A3 - Project settings](#)

B. Edit circuit diagrams

[Guide B1 - Create a new circuit diagram](#)

[Guide B2 - Draw logic symbols](#)

[Guide B3 - Move, copy and paste](#)

[Guide B4 - Undo and redo](#)

[Guide B5 - Create cable definition](#)

[Guide B6 - Define wire name](#)

[Guide B6 - Define wire types](#)

[Guide B8 - Create and use macro](#)

[Guide B9 - Define components to logic symbols](#)

[Guide B10 - Define engraving plates to logic symbols](#)

[Guide B11 - Using coils and its contacts](#)

[Guide B12 - Use I/O symbols](#)

C. Edit layout pages

[Guide C1 - Create a layout page](#)

[Guide C2 - Draw layout symbol from circuit diagram](#)

[Guide C3 - Draw layout symbol from component](#)

[Guide C4 - Use manual layout symbols](#)

[Guide C5 - Use device mounting rails](#)

[Guide C6 - Use cable duct objects](#)

[Guide C7 - Use measurement objects](#)

D. Generate lists

[Guide D1 - Wire list](#)

[Guide D2 - Cable pull list and cable connection list](#)

[Guide D3 - Bill of material](#)

[Guide D4 - Order list](#)

[Guide D5 - Engraving plate list](#)

[Guide D6 - PLC list](#)

[Guide D7 - Table of contents](#)

[Guide D8 - Re-generate existing list](#)

E. Work with the database library

[Guide E1 - Open the database library](#)

[Guide E2 - Add component](#)

[Guide E3 - Create new layout symbol](#)

[Guide E4 - Link a layout symbol with component](#)

[Guide E5 - Create a parent symbol](#)

[Guide E6 - Create child symbol to a parent symbol](#)

[Guide E7 - Create a new page header](#)

1.2 Guide A1 - Computer settings

The computer settings are settings for you as an user of Elwin. The settings are not saved in a project or database. The settings will let you setup Elwin the way you like to have it. It will also define you as an user, like name, company and so on...

Step by step

1. Click **Project** in the menu bar, then click **Computer settings**.
2. The **Computer settings** dialog box opens.
3. Click the tab **User settings**. This tab is settings of you as an user. Enter as many fields as possible.
4. Click the tab **Edit options** to view edit option settings. Leave these settings unchanged until you are familiar with Elwin.

Click **OK** to save the settings.

1.3 Guide A2 - Create your first project

To be able to draw circuit diagrams, layout pages and generating lists, then you must create a project. Elwin will automatically create a folder with your new project. The folder will hold all the project files and sub-folders. Don't move or delete any of these files.

Step by step

1. Click **Project**, then click **New**, then click **Project**.
2. The **Create new project** dialog box opens. Select and open a folder where you like to create the new project.
3. Enter a project name. Use only letters and numeral characters.
4. Click **Create**.
5. The **Project settings** opens automatically when the project was successfully created.

[Continue with guide A3 - Project settings](#)

1.4 Guide A3 - Project settings

The project settings are individual settings of each project.

Step by step

1. Click **Project**, then click **Project settings**. The **Project settings** dialog box opens with several tabs. Let's start from the very left tab.
2. Click the tab **Project information**, which holds text field with information about the project data. Enter as much as you know about your project.
3. Next tab **Panel builder** is information about the panel builder of the project. These text fields may be used as delivery address of purchase order lists. Leave blank will not cause problems.
4. The tab **Databases** is very important. Click the *-button to select a database library for your project. In the standard Elwin folder "c:\program\elwin\Database\" you can find a database named **IEC_E.ewd**. Select this file and click **Open**. [Note!](#)
5. Select a font for your project from the standard Elwin folder "c:\program\elwin\font\". Select the font **Elwin.ewf**.
6. The tab **Databases** is settings for the page headers. Use the *-button to select **PH_CIRC** for circuit diagrams and **PH_LAYOUT** and for layout pages. Leave the other settings blank.

7. Select the tab **Logic symbols** which has settings for your most common used logic symbols. Enter as follows;
Wire name symbol: *WN*
Wire type symbol: *KS*
Cable lead symbol: *WCAB1*
Cable lead for the following lead: *WCAB2*
Terminal symbol: *X1*
Black box connection: *LX*
Wire cross reference symbol: *WREF*
Revision symbol: *REV_PAGE*
You write directly in the text box or use the database button to select the logic symbol.
8. Click **OK**.

NOTE!

To select correct database library from the very beginning is extremely important. Sure you can change the database at any time you like, but you will probably ruin the project's circuit diagrams and layout pages.

1.5 Guide B1 - Create a new circuit diagram

To be able to edit circuit diagrams, then you must first create each circuit diagram. The project tree holds all pages within the project.

Step by step

1. Right-click the project name in the project tree, in the popup menu click **Create new**, then click **Circuit diagram**.
or
Click **Page** in the menu bar, then click **Create new circuit diagram**.
2. The **New circuit diagram** dialog box opens.
3. Enter a page number of the new circuit diagram. It must be an integer value.
4. Enter optional description text of the page.
5. Leave the grid setting to 5mm.
6. Click **OK** to confirm the new circuit diagram.
7. The new circuit diagram was created to the project tree.

1.6 Guide B2 - Draw logic symbols

This guide will teach you how to draw logic symbols to the circuit diagram.

Step by step

1. Create a new circuit diagram ([Guide B1](#)) and open the circuit diagram by double-click the page number in the project tree.

Insert a black box which will represent the incoming power to the panel.

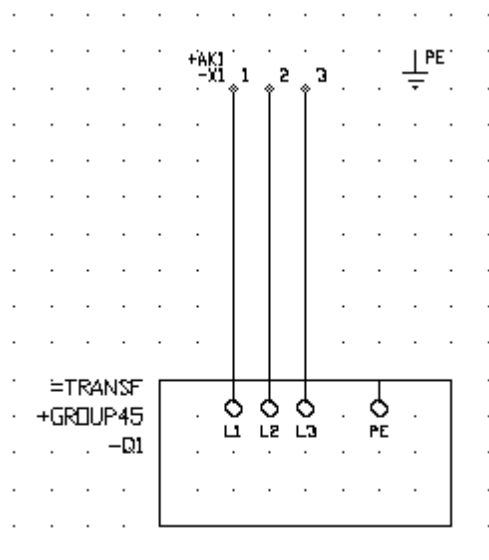
2. Click **Insert** in the menu bar, then click **Black box**.
3. Click for the upper left corner, then move the mouse-pointer to shape a rectangle object and click once again.
4. The **New black box** dialog box opens. Enter **TRANSF** in the installation, enter **GROUP45** in the location och **Q1** in the symbol designation. Click **OK**.

Now you will define the black box's connection points.

5. Click **Insert** in the menu bar, then click **Logic symbol**.
6. The **Database of Logic symbol** dialog box opens with all logic symbols. Filter the information by right-click the text **Type** in the upper part of the form, in the popup menu click **Black box connection**.
7. The database will now only show logic symbols of the type black box connections. Select the logic symbol **LX** and then click **OK**.
8. The logic symbol attaches to the mouse pointer. Move the symbol inside the black box and click.
9. The **Black box connection** dialog box opens. Enter **L1** in the connection point, select connection direction **Up** and let **LX** be un-changed. Click **OK**.
10. Repeat step 5-9 for the connection points **L2**, **L3** and **PE**.

OK, now is the incoming power symbol completed. Let's draw the terminals where the incoming cable is connected.

11. Click **Insert** in the menu bar, then click **Logic symbol**.
12. The **Database of Logic symbol** dialog box opens with all logic symbols. Filter the information by right-click the text **Type** in the upper part of the form, in the popup menu click **Terminal symbol**.
13. Select the logic symbol **X1** and then click **OK**.
14. The logic symbol attaches to the mouse pointer. Move the pointer above the L1 connection and click.
15. The **New logic symbol** dialog box opens. Enter **AK1** in the location and enter **X1** in the symbol designation.
16. Select the option **Show**, which means Elwin will show the symbol designation in the circuit diagram.
17. Select the options up and down for **Allow connections**.
18. Enter **1** in the terminal pin field..
19. Repeat step 11-18 with 2 and 3, but un-select the option **Show**.
20. Repeat step 11-18, but select the logic symbol **XPE1**, position the symbol right to the terminals and call the terminal pin **PE**.



Picture 1. Completion of step 1-20 should look something like this.

Let's insert the main breaker connected to -X1:1-3

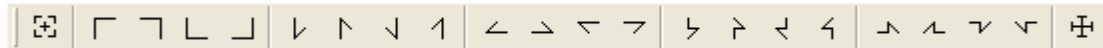
21. Click **Insert** in the menu bar, then click **Logic symbol**.
22. The **Database of Logic symbol** dialog box opens with all logic symbols. Filter the information by right-click the text **Type** in the upper part of the form, in the popup menu click **Parent symbol**.
23. Select the symbol **Q1** and then click **OK**.
24. The logic symbol attaches to the mouse pointer. Move the pointer above the X1:1 connection and

- click.
25. Enter **Q01** for symbol designation. And enter **AK1** for location.
 26. Click **OK**.

In the upper part of the circuit diagram we want power lines from left to right.

27. Click **Insert** in the menu bar, then click **Logic symbol**.
28. The **Database of Logic symbol** dialog box opens with all logic symbols. Filter the information by right-click the text **Type** in the upper part of the form, in the popup menu click **Wire cross reference symbol**.
29. Select the symbol **WREF** and then click **OK**.
30. Move the mouse pointer to the upper right corner of the circuit diagram and click.
31. The **New wire cross reference symbol** dialog box opens. Since this is an output from this circuit diagram, then select the option **Output - Right**.
32. Enter the cross reference designation to **L1**.
33. Click **OK**.
34. Repeat step 27-33, but enter designations as **L2** and **L3**.

Now it's time to get familiar with connection symbols.

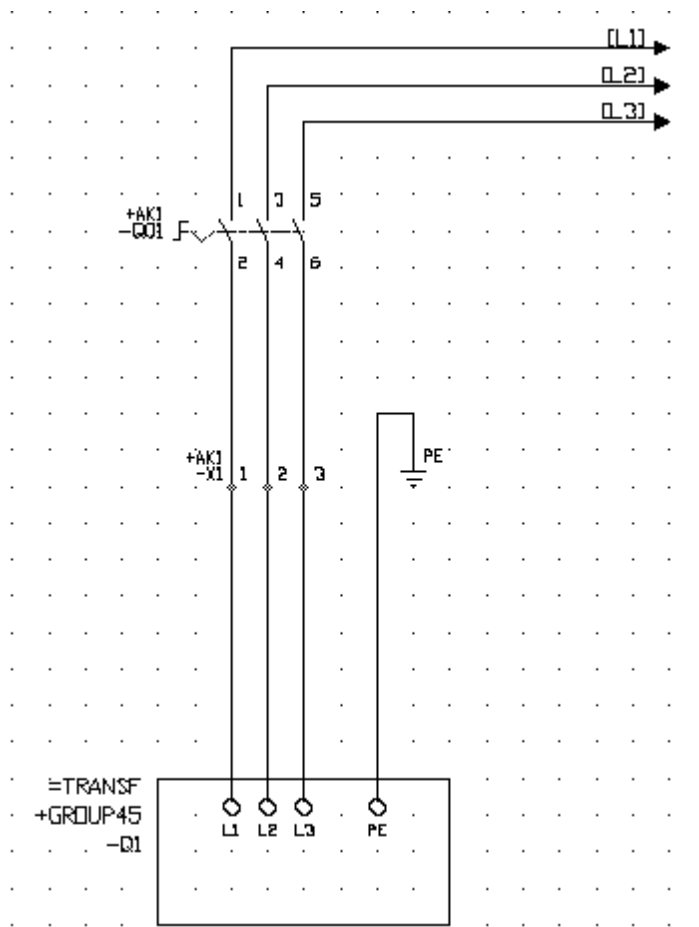


Since the ground symbol is searching for connection in the direction up, you need to manually insert connection symbols for the connection. The above picture shows all possible connection symbols.

35. Click the symbol "Down - Right", then the symbol attaches to the mouse pointer. Move the pointer directly over the transformer connection point PE and click.
36. Click the symbol "Left - Down", then move the mouse pointer just over the ground symbol and click.




Continue with the connection of the main breaker with the wire cross reference symbols.

37. Click the symbol "Down - Right", then move the mouse pointer over the main switch first phase and in the same level as the wire cross symbol **L1**.
38. Repeat step 37 for **L2** and **L3**.



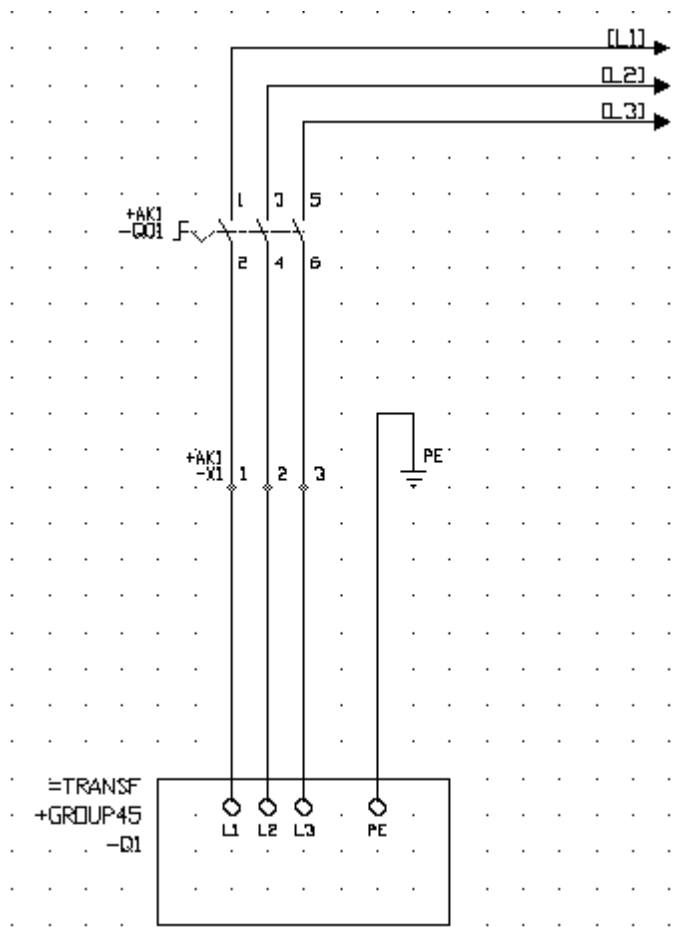
Picture 2. The circuit diagram should now look like above picture.

TIPS

- Instead of step 5-7, just click the icon  in the **Logic symbol toolbox** (project setting must be according to step 7 in [Guide A3 - Project settings](#)).
- Instead of step 11-13, just click the icon  in the **Logic symbol toolbox** (project setting must be according to step 7 in [Guide A3 - Project settings](#)).
- Instead of step 27-29, just click the icon  in the **Logic symbol toolbox** (project setting must be according to step 7 in [Guide A3 - Project settings](#)).

1.7 Guide B3 - Move, copy and paste

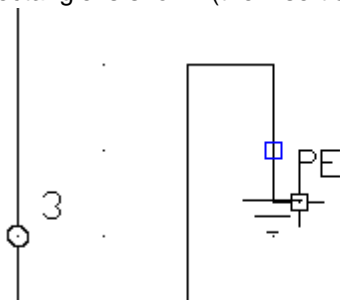
To move, copy or paste objects in Elwin functions similar to traditional Windows software. Please note you can't move the wire connection, but you move the symbols and the wire connection will move indirectly.



Picture 1. We start from the [Guide B2](#) circuit diagram.

Move single objects

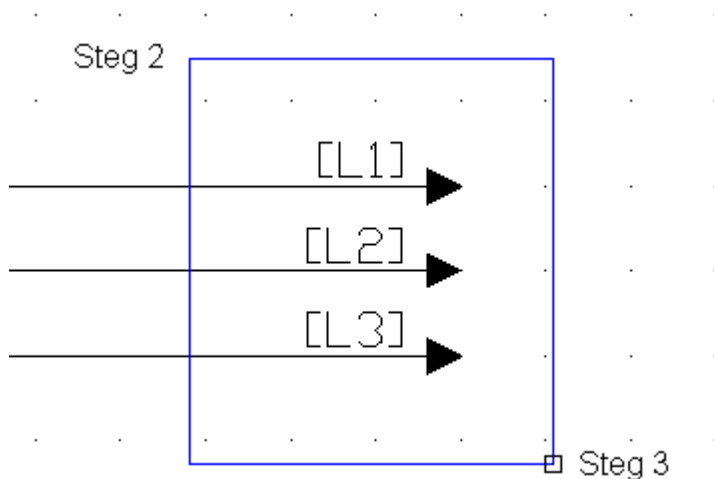
1. The ground symbol is to be moved down.
2. Click the ground symbol graphics object or text, then the objects gets selected and a blue rectangle is shown (the insertion point).



3. Click the blue rectangle (insertion point) and the symbol attaches the mouse pointer.
4. Move the mouse pointer (and symbol) to a new position and click.

Move single or several objects

1. The mission is to move all three wire cross symbols further to the right.
2. Click an empty spot just over the objects to the left.



3. Move the mouse pointer over the objects to form a rectangle and click.
4. The three objects get selected (blue rectangle indicates selected object(s)).
5. Click **Edit** in the menu bar, then click **Move object(s)**.
6. The selected objects attach to the mouse pointer, then move the pointer to a new position and click.

Copy single object

1. Right-click the objects graphics or text.
2. In the popup menu, click **Copy object**.
3. Elwin copies the object to an internal memory and Elwin is ready to paste the object.

Copy several objects

1. Select the objects you want to copy. A blue rectangle indicates selected object.
2. Click **Edit** in the menu bar, then click **Copy object(s)**.
3. Elwin copies the objects to an internal memory and Elwin is ready to paste the objects.

Paste copied object (single or several objects is the same procedure)

1. Click **Edit** in the menu bar, then click **Paste**.
2. All objects in the internal Elwin memory attaches to the mouse pointer.
3. Move the pointer in position and click. Some objects may need to be configured, but Elwin will automatically open the properties dialog box when necessary.

NOTE!

- The function **Cut** is similar to **Copy**, but with the difference **Cut** will delete the object(s) after the copy.

TIPS!

- Instead of using the menu bar, you can use the keyboard to quickly activate **Move**, **Copy**, **Cut** and **Paste**. See the menu bar for keyboard strokes.

1.8 Guide B4 - Undo and redo

The functions Undo and Redo works in similar with other Windows software. If you edit something, then you have the possibility to undo your edit. And if you undo by mistake then you can redo your edit.

Undo

1. Open a circuit diagram, for example [Guide B2](#) circuit diagram.
2. Double-click the main switch (for example the text Q01). This will open the properties of the logic symbol.
3. Change the symbol designation to Q02.
4. Click **OK**.
5. Elwin will now change symbol designation of the circuit diagram object. See text Q02.
6. To undo your edit in step 2-4, then click **Edit** in the menu bar, then click **Undo**.
7. Elwin will now change back the symbol designation to Q01.

Redo

8. To redo the edit in step 2-4, then click **Edit** in the menu bar, then click **Redo**.
9. Elwin will now change back the symbol designation to Q02.

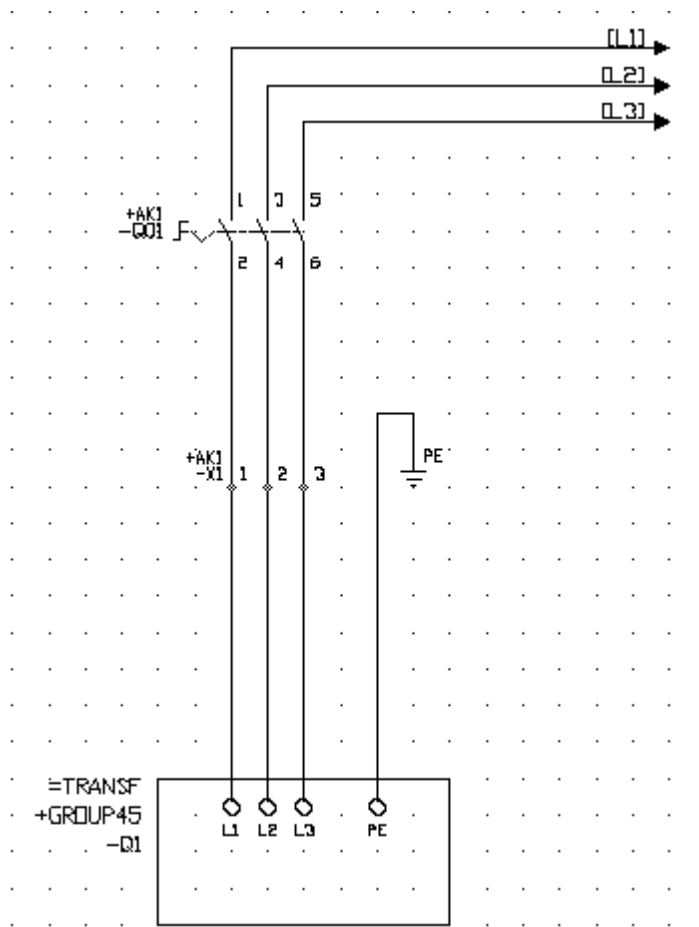
You can redo/undo all edit functions of the pages and there is no limit of steps.

NOTE!

As standard setting of a new project, **Undo** and **Redo** will reset upon page change. It means you can't undo something if you change to a new page. In the project setting you have the option to change this behaviour of Elwin.

1.9 Guide B5 - Create cable definition

This guide will teach how to define a cable in a circuit diagram.



Picture 1. We start from the [Guide B2](#) circuit diagram.

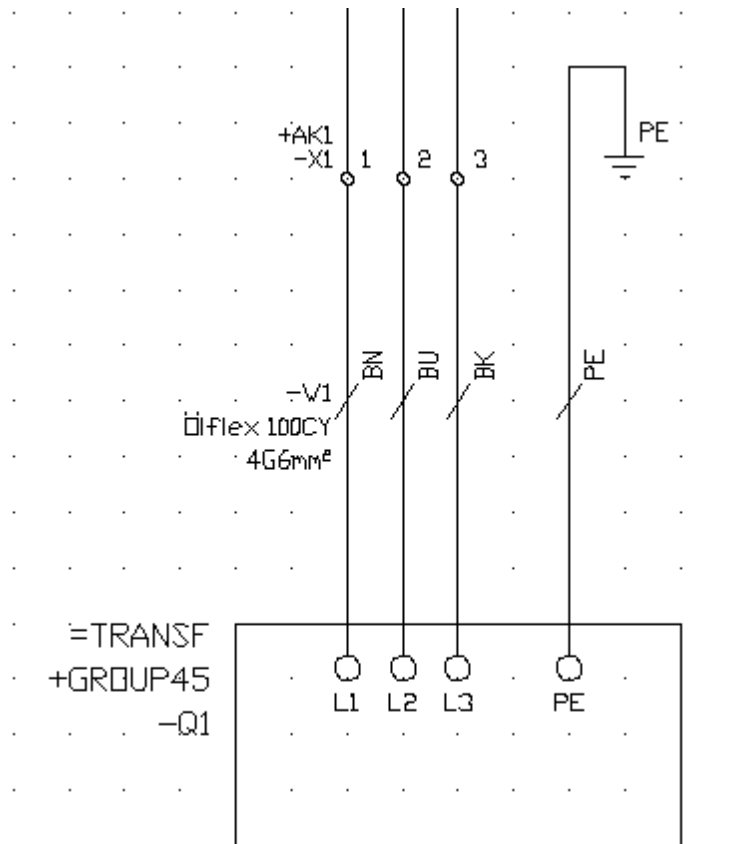
The cable definition is for the connection from the transformer station to "our" terminals.

Step by step

1. Open the circuit diagram by double-click the page number in the project tree.
2. Click **Insert** in the menu bar, then click **Logic symbol**.
3. The **Database of Logic symbol** dialog box opens with all logic symbols. Filter the information by right-click the text **Type** in the upper part of the form, in the popup menu click **Cable lead symbol**.
4. Select the logic symbol **WCAB1** and then click **OK**.
5. The logic symbol attaches to the mouse pointer. Move the pointer between **X1:1** and **L1**, then click.
6. The **New Cable lead symbol** dialog box opens. Enter **W1** in the cable designation field.
7. Click the database button (*) next to the **Cable name and type**.
8. The **Database of Cable** dialog box opens with all cable definitions in the database library.
9. Since we want to insert a 6mm² cable, then enter **6** in the top row of column **Size**. This will filter out all other cable sizes but 6.
10. Select **Öflex 100CY 4G6mm²** and then click **OK**.
11. The definition is completed, so click **OK** once again.


The first lead is completed now let's do the second, third and ground.

12. Elwin will automatically define the next cable lead and attach the logic symbol to the mouse pointer.
13. Move the mouse pointer to the next cable lead and click.
14. Repeat step 12-13 for the third and ground parts.
15. Press ESC-button to abort automatic insertion of the screen part (SH)..



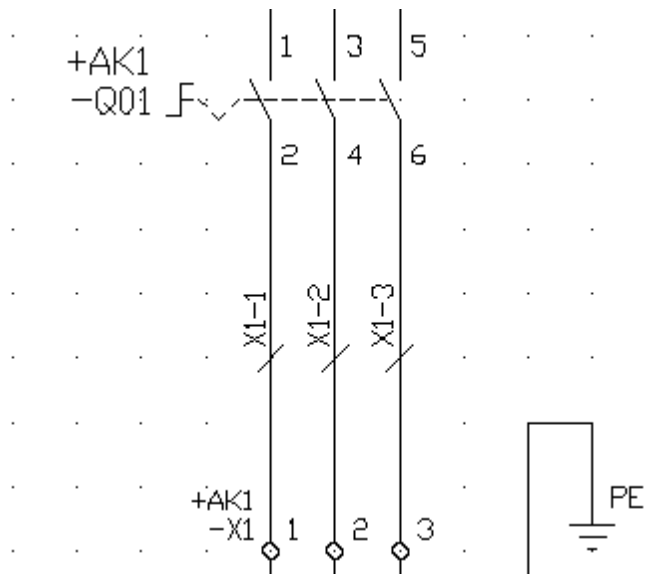
Picture 2. Result upon completion of the last step.

TIPS

- Instead of step 2-4, click the icon  in the **Logic symbol toolbox** (project setting must be according to step 7 in [Guide A3 - Project settings](#)).


1.10 Guide B6 - Define wire name

This guide will teach how to define a wire name of a wire connection in a circuit diagram.



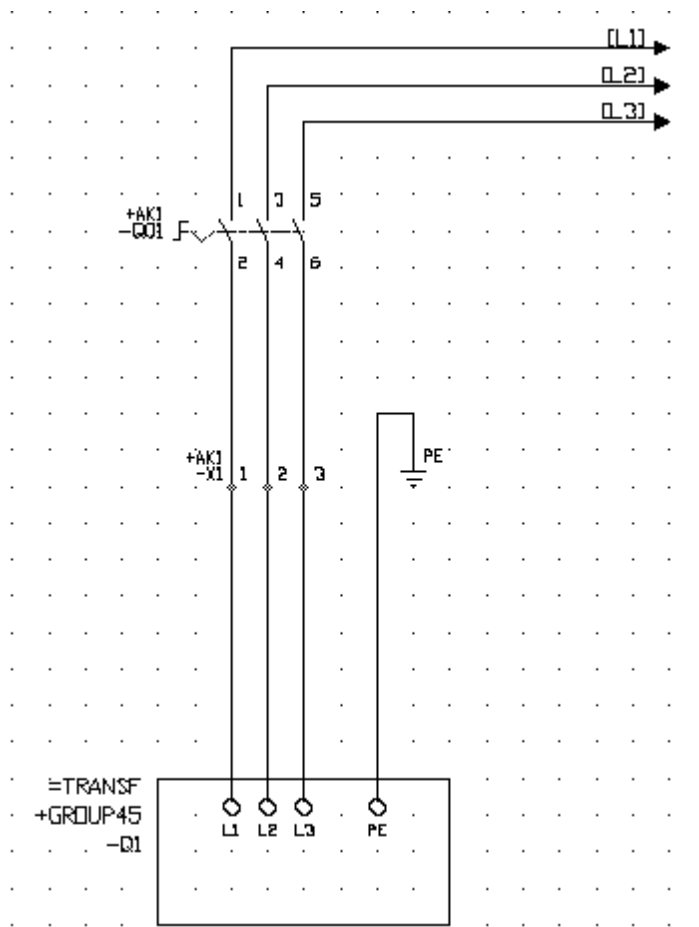
Picture 2. Result upon completion of the last step.

TIPS

- Instead of step 2-4, click the icon  in the **Logic symbol toolbox** (project setting must be according to step 7 in [Guide A3 - Project settings](#)).

1.11 Guide B7 - Define wire types

This guide will teach how to define a wire type of a wire connection in a circuit diagram.

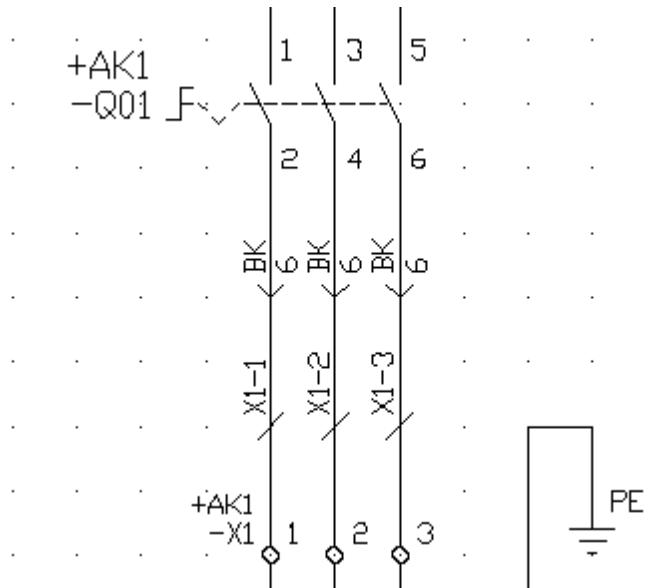


Picture 1. We start from the [Guide B2](#) circuit diagram.

The internal wires from X1 to the main switch is to be defined as 6mm² black wire.

Step by step

1. Open the circuit diagram by double-click the page number in the project tree.
2. Click **Insert** in the menu bar, then click **Logic symbol**.
3. The **Database of Logic symbol** dialog box opens with all logic symbols. Filter the information by right-click the text **Type** in the upper part of the form, in the popup menu click **Wire type symbol**.
4. Select the logic symbol **KS** and then click **OK**.
5. The logic symbol attaches to the mouse pointer. Move the pointer between **X1:1** and **Q01**, then click.
6. The **New Wire** dialog box opens. On the tab **Wire type specification** click the database button (*) next to the text field **Wire type**.
7. The **Database of Wire** dialog box opens with all wire definitions in the database library.
8. Since we want to insert a 6mm² wire, then enter **6** in the top row of column **Size**. This will filter out all other wire sizes but 6.
9. Select **H07V-K 6mm² BK** and then click **OK**.
10. The definition is completed, so click **OK** once again.
11. Repeat step 2-10 for the wires from X1:2 and X1:3.



Picture 2. Result upon completion of the last step.

NOTE!

When you print or export the circuit diagram, you may want to hide the wire type symbol. In the **Preferences** tab in **Project settings** dialog box you set

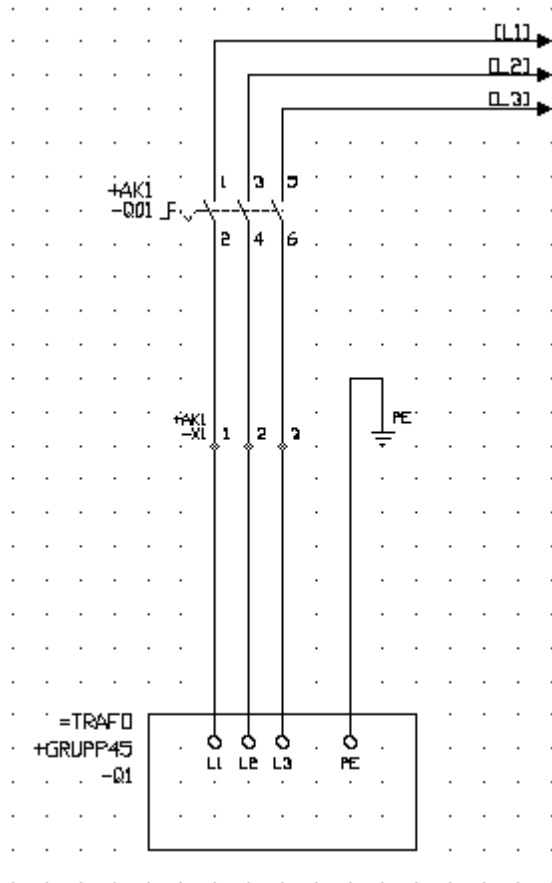
TIPS

- Instead of step 2-4, click the icon  in the **Logic symbol toolbox** (project setting must be according to step 7 in [Guide A3 - Project settings](#)).

1.12 Guide B8 - Create and use macro

A very useful function is macros. This guide will teach you to create and use macro.

A macro is objects copied to a file. You can insert the macro file to circuit diagrams and it will save you design time. You can create standard solution for circuits you repeatedly use.



Picture 1. We start from the [Guide B2](#) circuit diagram.

Step by step (Create macro)

1. Open the circuit diagram by double-click the page number in the project tree.
2. Select the objects you like to include in the macro.
3. Click **Edit** in the menu bar, then click **Create macro**.
4. The standard windows **Save as** dialog box opens. Enter a file name and select a folder where you like to save the macro.
5. Click **Save**.
6. The **Macro description** dialog box opens. Enter an optional description of the macro.
7. Click **OK**.
8. The macro is saved and is waiting to be used.

Step by step (Use the macro)

1. Open a circuit diagram (or create a new).
2. Click **Insert** in the menu bar, then click **Macro**.
3. The **Select macro** dialog box opens. Select the folder with your macro.
4. The list in the right side of the dialog box shows all available macros in the selected folder.
5. Select the macro you like to insert.
6. The macro's objects attach to the mouse pointer. Move the pointer and objects in position, then click.
7. Elwin saves the objects to the page. Some objects may need to be edited, and Elwin will automatically open the edit dialog boxes.

1.13 Guide B9 - Define components to logic symbols

You can define components to the logic symbols. When you are finished with your project, then you will be able to create a bill of material with the components from the logic symbols. You can also create order lists, sorted per supplier.

You can at any time add, change or remove components from a logic symbol.

Step by step

1. Open the circuit diagram from [Guide B2](#) by double-click the page number in the project tree.
2. Double-click the main switch object (Q01) to open its properties.
3. Click the tab **Components**. This tab shows all defined components to the logic symbol.
4. Click the button **Add component**.
5. The **Choose component** dialog box opens, which shows all components from the database library.
6. Search the component P1-32/V/SVB-SW, by enter **P1*** in the top row of column order number.
7. Select the component and click **OK**.
8. The component is now in the logic symbol dialog box. Enter how many units of the component you like for the logic symbol. The standard is 1.
9. Click **OK** to save the changes to the logic symbol.

See also

[Guide D3 - Generate bill of material](#)

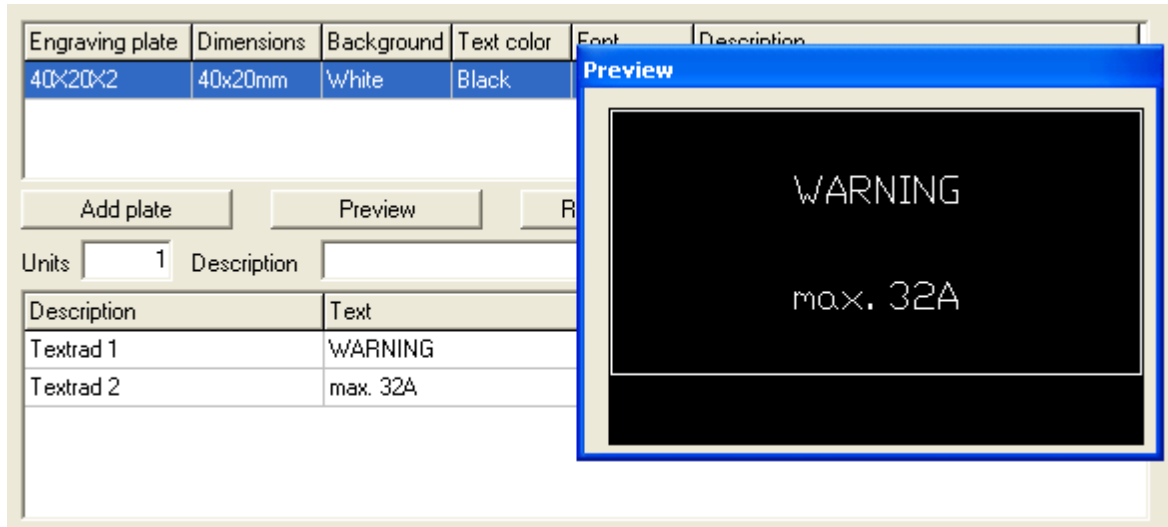
[Guide D4 - Generate order list](#)

1.14 Guide B10 - Define engraving plates to logic symbols

You can define engraving plates to logic symbols. Each logic symbol may have up to four different plates. When you have edited the project, then you simply generate an engraving plate layout list with all engraving plates from the project.

Step by step

1. Open the circuit diagram from [Guide B2](#) by double-click the page number in the project tree.
2. Double-click the main switch object (Q01) to open its properties.
3. Click the tab **Engraving plates**. This tab shows all defined engraving plates to the logic symbol.
4. Click the button **Add plate**.
5. The **Database of engraving plates** dialog box opens, which shows all plates from the database library.
6. Select the engraving plate **40X20X2**, then click **OK**.
7. Enter the text for the engraving plate in the column **Text**.
8. Select number of identical engraving plates and an optional description of the plate.
9. Click **OK** to save the changes to the logic symbol.



Picture 1. Result upon completion of the last step.

TIPS

- Use the button **Preview** to see how the plate will look like.
- You can assign up to four different engraving plates to each logic symbol, but by using auxiliary symbols you increase the number.

1.15 Guide B11 - Using coils and its contacts

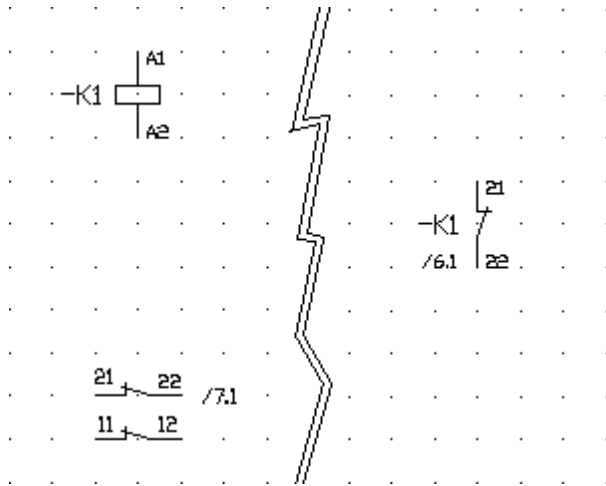
To use coils and contacts in the design of circuit diagrams are very common. In Elwin you do in two ways. This guide will teach you how to use both ways.

This procedure can not only be used for coils and relays, but also other logic symbols with cross-references contacts, like circuit breakers, main switches, safety switches, pushbuttons and so on.

Step by step (procedure 1)

1. Open the circuit diagram from [Guide B2](#) by double-click the page number in the project tree.
2. Click **Insert** in the menu bar, then click **Logic symbol**.
3. The **Database of Logic symbol** dialog box opens. Search and select the parent symbol **K1** and click **OK**.
4. The logic symbol attaches to the mouse pointer. Move the symbol in position and click.
5. The **New logic symbol** dialog box opens. Enter **K1** in the symbol designation.
6. Click **OK**.
7. Now we will define the contacts of the relay. We only like to show a cross-reference of the contacts, because the real connections will be on another circuit diagram.
8. Click **Insert** in the menu bar, then click **Logic symbol**.
9. The **Database of Logic symbol** dialog box opens. Search and select the auxiliary symbol **KX_002** and click **OK**.
10. The logic symbol attaches to the mouse pointer. Move the symbol in any position and click.
11. The **New logic symbol** dialog box opens. Enter **K1** in the symbol designation.
12. Un-select the option **Show**, which will hide the symbol designation in the circuit diagram.
13. Click **OK**.
14. Right click the auxiliary symbol, in the popup menu click **Rotate**, then click **+270° rotation**.
15. Create a new circuit diagram ([Guide B1](#)) and open the new circuit diagram.
16. Click **Insert** in the menu bar, then click **Logic symbol**.

17. The **Database of Logic symbol** dialog box opens. Search and select the child symbol **KS_001** and click **OK**.
18. The logic symbol attaches to the mouse pointer. Move the symbol in position and click.
19. The **New logic symbol** dialog box opens. Enter **K1** in the symbol designation.
20. Click **OK**.



Picture 1. Result upon completion of step.

Step by step (procedure 1)

1. Open the circuit diagram from [Guide B2](#) by double-click the page number in the project tree.
2. Click **Insert** in the menu bar, then click **Logic symbol**.
3. The **Database of Logic symbol** dialog box opens. Search and select the parent symbol **K1_002** and click **OK**.
4. The logic symbol attaches to the mouse pointer. Move the symbol in position and click.
5. The **New logic symbol** dialog box opens. Enter **K2** in the symbol designation.
6. Click **OK**.
7. Create a new circuit diagram ([Guide B1](#)) and open the new circuit diagram.
8. Click **Insert** in the menu bar, then click **Logic symbol**.
9. The **Database of Logic symbol** dialog box opens. Search and select the child symbol **KS_001** and click **OK**.
10. The logic symbol attaches to the mouse pointer. Move the symbol in position and click.
11. The **New logic symbol** dialog box opens. Enter **K1** in the symbol designation.
12. Click **OK**.

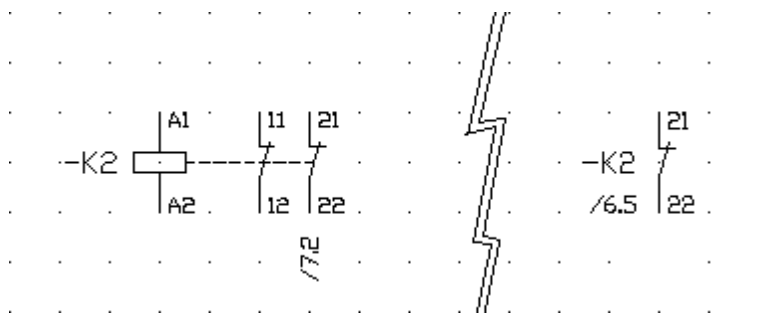


Bild 2. Resultat efter Sätt 2 att rita kontaktspegel.

COMMENT

Procedure 1 is much more flexible than procedure 2, but procedure 2 are faster to design. Note the child symbol is designed in the same way in both procedures.

1.16 Guide B12 - Use I/O symbols

Elwin supports PLC cross-references in a simple way, but advanced enough to fit all ways of designing your I/O logic symbols.

Step by step

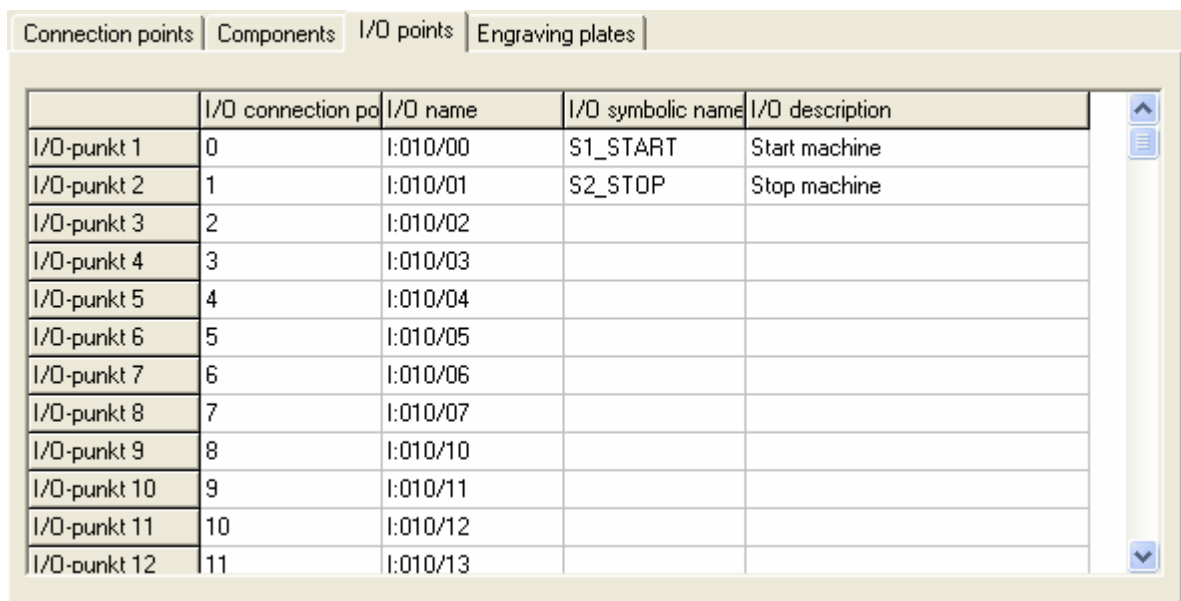
[Draw PLC parent symbol](#)

[Draw PLC child symbol \(type 1\)](#)

[Draw PLC child symbol \(type 2\)](#)

Step by step (Draw PLC parent symbol)

1. Create a new circuit diagram ([Guide B1](#)) and open the new circuit diagram.
2. Click **Insert** in the menu bar, then click **Logic symbol**.
3. The **Database of Logic symbol** dialog box opens. Search and select the PLC parent symbol **IO16** and click **OK**.
4. The logic symbol attaches to the mouse pointer. Move the symbol in position and click.
5. The **New logic symbol** dialog box opens. Enter **A1** in the symbol designation.
6. Click the tab **I/O points**.
7. Enter information in each row and column.
8. Click **OK** to save the logic symbol.



	I/O connection point	I/O name	I/O symbolic name	I/O description
I/O-punkt 1	0	I:010/00	S1_START	Start machine
I/O-punkt 2	1	I:010/01	S2_STOP	Stop machine
I/O-punkt 3	2	I:010/02		
I/O-punkt 4	3	I:010/03		
I/O-punkt 5	4	I:010/04		
I/O-punkt 6	5	I:010/05		
I/O-punkt 7	6	I:010/06		
I/O-punkt 8	7	I:010/07		
I/O-punkt 9	8	I:010/10		
I/O-punkt 10	9	I:010/11		
I/O-punkt 11	10	I:010/12		
I/O-punkt 12	11	I:010/13		

Picture 1. Result of editing the I/O points.

Step by step (Draw PLC child symbol type 1)

1. Create a new circuit diagram ([Guide B1](#)) and open the new circuit diagram.
2. Click **Insert** in the menu bar, then click **Logic symbol**.
3. The **Database of Logic symbol** dialog box opens. Search and select the PLC child symbol **8DI-DO** and click **OK**.
4. The logic symbol attaches to the mouse pointer. Move the symbol to the very top left position on the circuit diagram and click.

5. The **New logic symbol** dialog box opens. Enter **A1** in the symbol designation.
6. Double-click the first row in the column **I/O Connection point**.
7. The **Select I/O point** dialog box opens with the projects all I/O's. Select I/O Connection point **0** of symbol designation **A1** and click **OK**.
8. Repeat step 6-7 for all I/O points.
9. Click **OK**.

Symbol texts		I/O points		
	I/O connection po	I/O name	I/O symbolic name	I/O description
I/O-punkt 1	0	I:010/00	S1_START	Start machine
I/O-punkt 2	1	I:010/01	S2_STOP	Stop machine
I/O-punkt 3	2	I:010/02		
I/O-punkt 4	3	I:010/03		
I/O-punkt 5	4	I:010/04		
I/O-punkt 6	5	I:010/05		
I/O-punkt 7	6	I:010/06		
I/O-punkt 8	7	I:010/07		

Picture 2. Result of editing the I/O child points.

Step by step (Draw PLC child symbol type 2)

1. Create a new circuit diagram ([Guide B1](#)) and open the new circuit diagram.
2. Click **Insert** in the menu bar, then click **Logic symbol**.
3. The **Database of Logic symbol** dialog box opens. Search and select the PLC child symbol **1DI-DO** and click **OK**.
4. The logic symbol attaches to the mouse pointer. Move the symbol to the very top left position on the circuit diagram and click.
5. The **New logic symbol** dialog box opens. Enter **A1** in the symbol designation.
6. Double-click the first row in the column **I/O Connection point**.
7. The **Select I/O point** dialog box opens with the projects all I/O's. Select I/O Connection point **8** of symbol designation **A1** and click **OK**.
8. Click **OK**.

	I/O connection po	I/O name	I/O symbolic name	I/O description
I/O-punkt 1	8	I:010/10	K1_OK	Safety relay activated

Picture 3. Result of editing an I/O child point.

RESULT

Picture 6. Logic symbol of PLC child symbol.

NOTE!

So it's possible to mix I/O child parent symbols with different number of I/O points. You can also define a PLC parent symbol and directly make the connections (without using PLC child symbols).

1.17 Guide C1 - Create a layout page

A layout page offers new type of symbols for your editing. To a layout page you can insert layout symbols, which are pre-defined graphics object from the database library. It will save you time, when you don't have to draw everything from scratch.

You can also insert objects called cable duct and device mounting rails. By selecting an object from the database and only enter the length, you will quick and easy design a layout.

All editing of a layout page is in millimeters and for each page you will set the layout size needed for the objects.

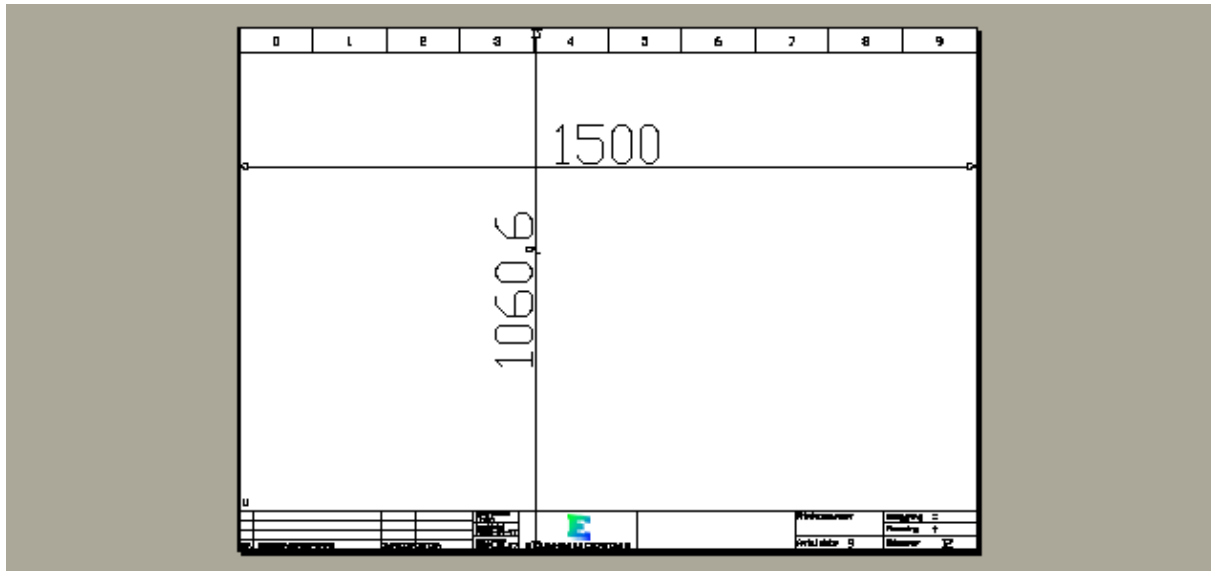
The measurement function will help you to insert measurements to the layout.

Step by step

1. Right-click the project name in the project tree, in the popup menu click **Create new**, then click **Layout page**.
or
Click **Page** in the menu bar, then click **Create new layout page**.
2. The **New layout page** dialog box opens.
3. Enter a page number of the new layout page. It must be an integer value.
4. Enter optional description texts of the page.
5. Leave the grid setting to 5mm.
6. The fields **Layout page width** and **Layout page height** sets the full size of the page. Enter 1500 and 1000 (See below picture).
7. Click **OK** to confirm the new page.
8. The new page was created to the project tree.

NOTE!

If 1500 and 1000mm was entered in the fields **Layout page width** and **Layout page height** then the result will be as the below picture. As indicated by the measurement objects Elwin automatically extended the height to keep the aspect ratio of the page. You can change the width and height of the layout page any time you like, but keep in mind some objects may end up outside the page if you decrease the page size. When you change the page size Elwin will automatically scale all objects.

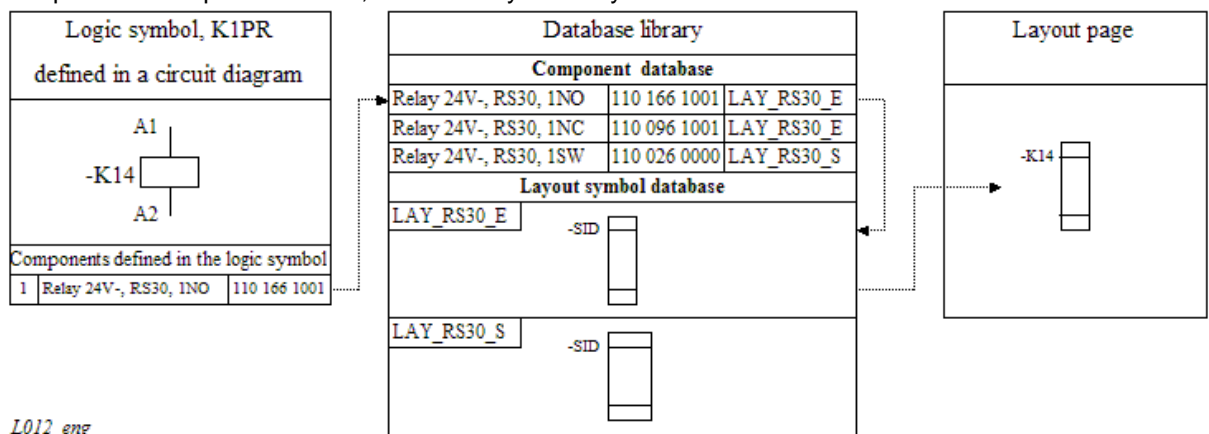


Picture 1. An example of a layout page.

1.18 Guide C2 - Draw layout symbol from circuit diagram

A layout symbol is a graphical representation of a component. All layout symbols are from the database library.

In the component database you can assign layout symbols to each component. When you assign components to a logic symbol in the circuit diagrams, then there will be a link to a layout symbol. So by only knowing the logic symbol you draw a layout symbol without knowing anything about the component. Complicated? Yes, but in reality it's easy.



L012_eng

Picture 1. Relationship between logic symbol and layout symbol.

Before you start this guide make sure you have completed [Guide B9](#).

Step by step

1. Open the layout page from [Guide C1](#) by double-click the page number in the project tree.
2. Click **Insert** in the menu bar, then click **Layout symbol**, then click **From circuit diagram**.
3. The **Select layout symbol from circuit diagram** dialog box opens.
4. Select the option **Show only logic symbols with component**.
5. The upper list shows all logic symbols with defined components. Select the logic symbol **Q01**.
6. The lower list shows components defined in the selected logic symbol. The column **Used** shows

how many times you have designed the component in a layout page, but since we have not designed it yet it shows **No**.

7. Select the row with the main switch P1-32/V...
8. Click **OK**.

The **Layout symbol properties** dialog box opens. The upper part of the dialog shows information about the link to the logic symbol. This information is basically the definitions made in step 2-6. The list **Select layout version to use** shows the available drawings in the layout symbol L_P1-32.

9. Select the layout version **1. Rotary handle**.
10. Click **OK**.
11. The layout symbol attaches to the mouse pointer. Move the object in position and click.

1.19 Guide C3 - Draw layout symbol from component

The function draws a component from the database library. By selecting a component you will be able to draw a layout. If a layout symbol is specified for the component then Elwin will use it, but if no layout symbol is specified then Elwin will use width and height information and just form a rectangle.

You may use this function for components which are not in the circuit diagram, like the control panel, mounting devices and so on. By selecting the option **Component to lists** the component will be included in the bill of material and order lists.

This guide will show how to draw a control panel from the database library.

Step by step

1. Open the layout page from [Guide C1](#) by double-click the page number in the project tree.
2. Click **Insert** in the menu bar, then click **Layout symbol**, then click **From component database**.
3. Search and select the component **1050.500**.
4. Click **OK**.

The **Layout symbol properties** dialog box opens. The upper part of the dialog shows information about the link to the component and layout symbol. This information is basically the definitions made in step 2-4. The list **Select layout version to use** shows the available drawings in the layout symbol L_AE1050.500.

5. Select the layout version **1. Front view+Mounting plate**.
6. Select the option **Component to lists**.
7. Click **OK**.
8. The layout symbol attaches to the mouse pointer. Move the object in position and click.

NOTE

- If a component is missing a layout symbol, then Elwin will use the width and height information.

1.20 Guide C4 - Use manual layout symbols

When you need to make a quick and simple layout symbol without using the other more advanced functions, then you can use manual layout symbol. The manual symbol has no links to the database library or logic symbols in the project.

A manual layout symbol is more or less only a rectangle with 9 text objects attached to the rectangle. When you define the manual layout you enter the actual size for height and width. By using this method

instead of single objects you will keep layout symbol objects grouped as one object. You will also have the possibility to let the layout symbol break device mounting rails.

This guide will make a manual layout symbol.

Step by step

1. Open the layout page from [Guide C1](#) by double-click the page number in the project tree.
2. Click **Insert** in the menu bar, then click **Layout symbol**, then click **Manual layout symbol**.
3. Enter **45mm** in the field manual width.
4. Enter **70mm** in the field manual height.
5. Select the option **Break mounting rails** which will make the layout symbol break a device mounting rail object.
6. In the first row of column **Text**, enter **K1**.
7. Double-click the first row of column **Show**, so the text indicates **Yes**. This will make this text row visible in the layout.
8. Enter the text height value to **2** in the column **Height**.
9. Enter the value **10** in the columns **X-pos** and **Y-pos**.
10. Click **OK**.
11. The layout symbol attaches to the mouse pointer. Move the object in position and click.

1.21 Guide C5 - Use device mounting rails

This guide will guide you how to use device mounting rails.

Step by step

1. Open the layout page from [Guide C1](#) by double-click the page number in the project tree.
2. Click **Insert** in the menu bar, then click **Device mounting rail**.
3. Dialogrutan **New device mounting rail** dialog box opens. Click the database button (*) in the upper part of the dialog.
4. The **Database of Device mounting rail** dialog box opens, which shows all rails in the database library.
5. Search and select **TS35x7.5** and click **OK**.
6. Enter the length of the mounting rail by enter 200 in the **Length** field.
7. Select the option **No rotation**.
8. Click **OK**.
9. The mounting rail attaches to the mouse pointer. Move the object in position and click.

NOTE!

- The length of the rail may be edited by selecting the object and click in the selection points in either ends.
- Try and see what happens if you place a layout symbol on the rail. ([Use guide C4](#)).

1.22 Guide C6 - Use cable duct objects

This guide will guide you how to use cable duct objects.

Step by step

1. Open the layout page from [Guide C1](#) by double-click the page number in the project tree.
2. Click **Insert** in the menu bar, then click **Cable duct**.
3. Dialogrutan **New cable duct** dialog box opens. Click the database button (*) in the upper part of the dialog.
4. The **Database of Cable duct** dialog box opens, which shows all ducts in the database library.

5. Search and select **BxH-25x60** and click **OK**.
6. Select the option **No rotation**.
7. Enter the length of the cable duct by enter 300 in the **Length** field.
8. Select **5. Sloping lines** in the list box **Cable duct layout**.
9. Select **3. BxHxL --x--x--** in the list box **Text layout**.
10. Click **OK**.
11. The cable duct attaches to the mouse pointer. Move the object in position and click.

NOTE!

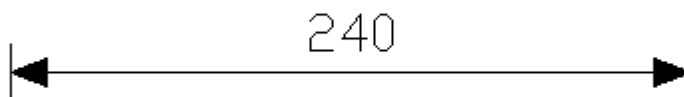
- The length of the cable duct may be edited by selecting the object and click in the selection points in either ends.
- If you select the **Cable duct layout** and **Text layout** to **1. Project**, then the cable duct will get the setting from the project setting. Open the project settings by click **Project** in the menu bar, then click **Project settings**, then select the tab **Layout page**.

1.23 Guide C7 - Use measurement objects

For layout pages you may insert measurement objects. There are several settings for the measurement objects so this guide will just give you a good start in using measurements. When this guide is completed it's recommended to do some own practise.

Step by step

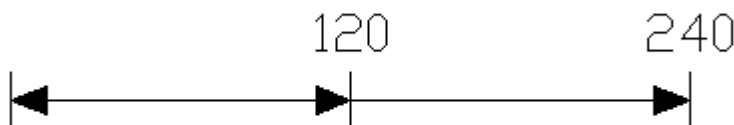
1. Open the layout page from [Guide C1](#) by double-click the page number in the project tree.
2. Click **Draw** in the menu bar, then click **Measurement object**.
3. Move the mouse pointer to the start position and click.
4. The measurement object will now be shaped from the start position to the mouse pointer position. Move to the end position and click.



Picture 1. Completion of step 4.

Add an extra measurement point

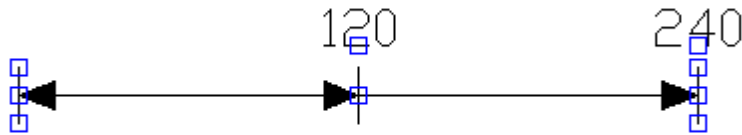
5. Right-click the measurement object.
6. In the popup menu click **Add measurement point**.



Picture 2. Completion of step 6.

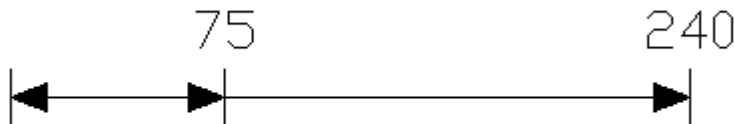
Move position of start, end or points.

- Click any of the graphics to select the measurement object.



Picture 3. Completion of step 7.

- Click the selection point on the horizontal line under 120.
- Move the point to a new position and click.



Picture 4. Completion of step 9.

Change the measurement object to measure segments

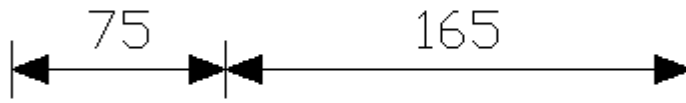
- Right-click the measurement object.
- In the popup menu click **Properties**.
- The **Measurement object properties** dialog box opens.
- Select the option $\leftarrow\rightarrow\leftarrow\rightarrow\leftarrow\rightarrow$ in the group box **Arrow appearance**.
- Click the button **Add** to add a segment.
- Click **OK**.



Picture 5. Completion of step 15.

Change the segment size

- Click any of the graphics to select the measurement object.
- As in step 8, click any of the three size selection points on the horizontal line.
- Move the mouse pointer to the new position and click.

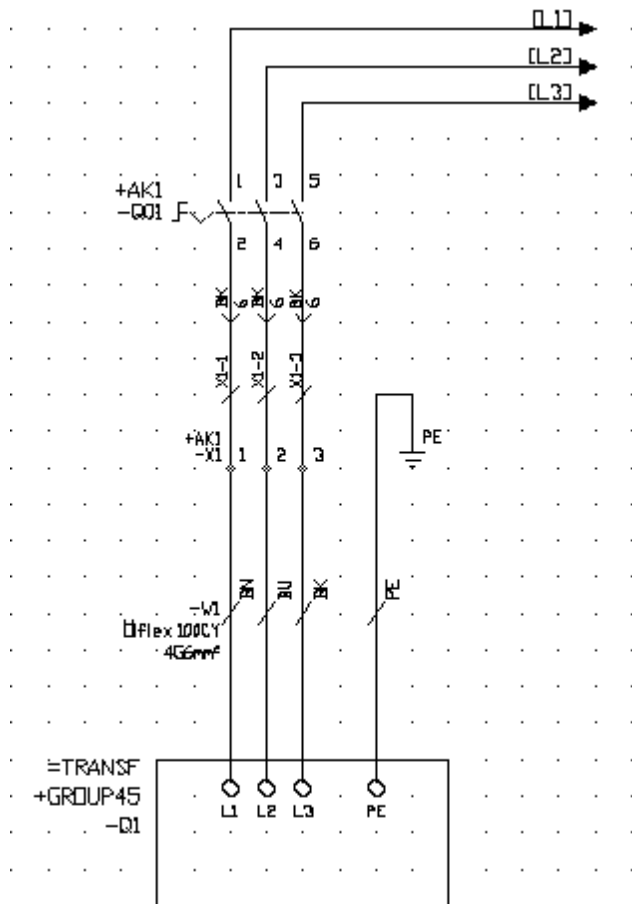


Picture 6. Completion of step 18.

1.24 Guide D1 - Generate wire list

A wire list is generating all wire connections in the selected location. The list is analyzing the projects logic symbol connection, searching for wire names and wire types, then finally outputs a sorted wire list.

As base for this guide, we will use guide B2, B6 and B7.



Picture 1. The circuit diagram created in guide B2, B6 and B7.

Step by step

1. Select a page header of **Wire list** (if it's already selected then jump to step 8).
2. Click **Project** in the menu bar, then click **Project settings**.
3. Click the tab **Page header**.

4. Click the database button (*) next to the **Wire list** text field.
5. The **Database of Page header** dialog box opens. Search and select **PH_WIRE**.
6. Click **OK**.
7. Click **OK** of the dialog box **Project settings**.
8. Right-click the project name in the project tree, in the popup menu click **Create new**, then click **Wire list**.
9. A wire list was created and is located in the project tree. But the list properties needs to be adjusted.
10. Right-click the wire list (___XX. Wire list) in the project tree, in the popup menu click **Properties**.
11. Enter **AK1** in the **Location**, which will change data input of the list. By selecting AK1 we will only get wires inside AK1 control panel.
12. Click **OK**.
13. The list will automatically be re-generated.
14. Open the list by double-click the list page number in the project tree.

0	1	2	3	4	5	6	7	8	9	
Wire list										
Wire type	Size	Color	Length	Source	Terminal	Cross-reference	Wire name	Destination	Terminal	Cross-reference
H07V-K	6mm ²	BLACK		-C01	2	/L1	X1-1	-X1	1	/L1
H07V-K	6mm ²	BLACK		-C01	4	/L1	X1-2	-X1	2	/L1
H07V-K	6mm ²	BLACK		-C01	6	/L1	X1-3	-X1	3	/L1

Picture 2. Result of step 14.

NOTE!

- Double-click the **cross-references** to navigate to the page.

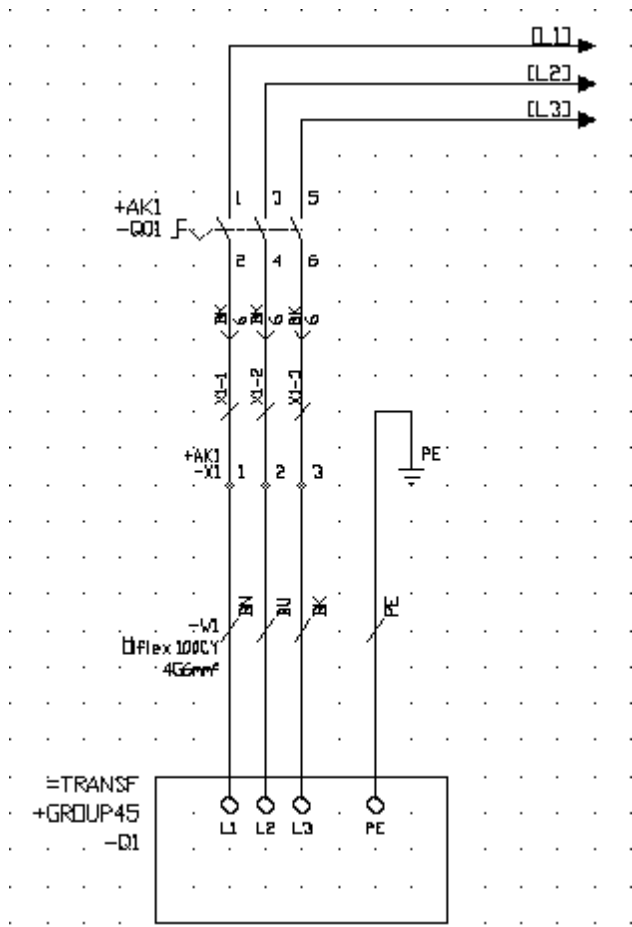
1.25 Guide D2 - Generate cable pull list and cable connection list

The lists **Cable pull list** and **Cable connection list** is generating all cables defined in the selected location. The lists are analyzing the projects cable logic symbols, its lead's connections source and destination, then finally outputs a sorted list.

The intention of the cable pull list is to get all the projects cables listed. You can not get the details of each cable lead connection from a cable pull list. For details about the connections, then use the cable connection list.

In difference with cable pull list, the cable connection list is presenting the cable lead's connection information. The cable connection list generator is creating one page per cable.

As base for this guide, we will use guide B2 and B5.



Picture 1. The circuit diagram created in guide B2 and B5.

Step by step

1. Select a page header of **Cable pull list** and **Cable connection list** (if it's already selected then jump to step 11).
2. Click **Project** in the menu bar, then click **Project settings**.
3. Click the tab **Page header**.
4. Click the database button (*) next to the **Cable pull list** text field.
5. The **Database of Page header** dialog box opens. Search and select **PH_CABLE**.
6. Click **OK**.
7. Click the database button (*) next to the **Cable connection list** text field.
8. The **Database of Page header** dialog box opens. Search and select **PH_CCONN**.
9. Click **OK**.
10. Click **OK** of the dialog box **Project settings**.
11. Right-click the project name in the project tree, in the popup menu click **Create new**, then click **Cable pull list**.
12. A cable list was created and is located in the project tree.
13. Open the list by double-click the list page number in the project tree.

0	1	2	3	4	5	6	7	8	9
Cable pull list									
Cable designation	Source	Destination	Cable name and type		Comment	Cross-reference cable connection			
-W1	+AK1-X1	=TRANSF+GROUP45-Q1	Flex 100CY 4G6mm²			/12-1			

Picture 2. Result of the cable pull list generation.

14. Right-click the project name in the project tree, in the popup menu click **Create new**, then click **Cable pull list**.
15. A cable list was created and is located in the project tree.
16. Open the list by double-click the list page number in the project tree.

0	1	2	3	4	5	6	7	8	9
---	---	---	---	---	---	---	---	---	---

Cable connection list

Cable designation	Cable type	Supplier	Size	Ground/Shield	Comment
-W1	Hilflex 100CY 4G6mm ²	Lappkabel	6mm ² Leads 3	X/X Length	

Lead	Cross-reference	Source	Terminal	Cross-reference	Wire name	Destination	Terminal	Cross-reference
BN	/1.1	+AK1-X1	1	/1.1		=TRANSF+GROUP45-G1	L1	/1.1
BU	/1.1	+AK1-X1	2	/1.1		=TRANSF+GROUP45-G1	L2	/1.1
BK	/1.1	+AK1-X1	3	/1.1		=TRANSF+GROUP45-G1	L3	/1.1
PE	/1.1	=TRANSF+GROUP45-M1	PE	/1.1		-X1	PE	/1.2
SH								

Picture 3. Result of the cable connection list generation.

NOTE!

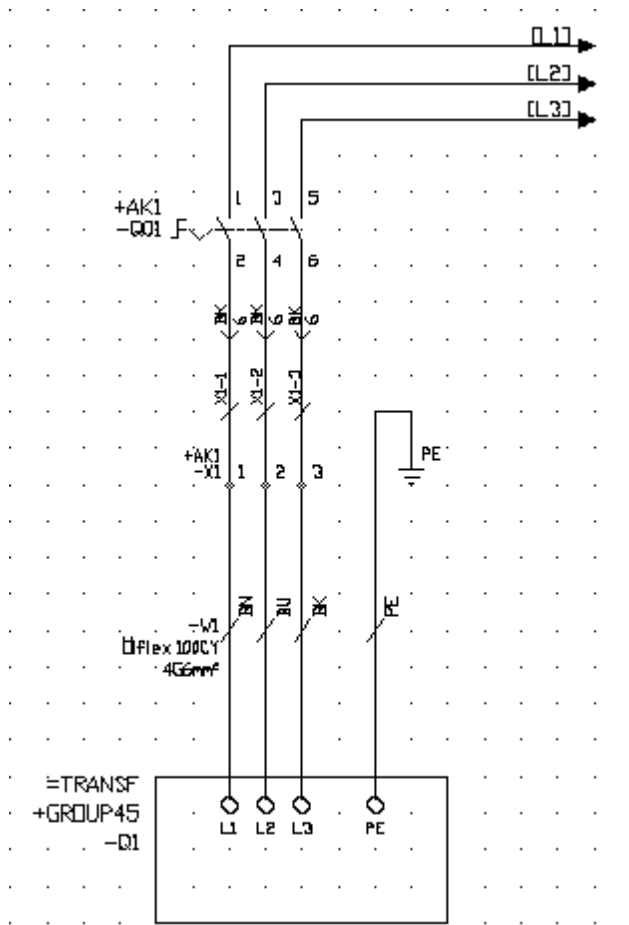
- Double-click the **cross-references** to navigate to the page.
- The column **Cross-reference cable connection** is reference to a possible cable connection list. When you update the cable pull and cable connection lists, then always update first all cable connection lists in the project.

1.26 Guide D3 - Generate bill of material

A bill of material list is searching and generating all components from the logic symbols defined in the circuit diagrams. The list is gathering the information from the logic symbol and combines it with data from the database library, then finally outputs a bill of material list.

The bill of material list will also collect information from the layout pages. If you insert a component to a layout page, then you can select the component to be included to bill of material lists.

As base for this guide, we will use guide B2, B9 and C3, but the terminal symbols X1:1, X1:2 and X1:3 also have components defined.



Picture 1. The circuit diagram created in guide B2 and B9.

Step by step

1. Select a page header of **Bill of material** (if it's already selected then jump to step 8).
2. Click **Project** in the menu bar, then click **Project settings**.
3. Click the tab **Page header**.
4. Click the database button (*) next to the **Bill of material** text field.
5. The **Database of Page header** dialog box opens. Search and select **PH_BOM**.
6. Click **OK**.
7. Click **OK** of the dialog box **Project settings**.
8. Right-click the project name in the project tree, in the popup menu click **Create new**, then click **Bill of material**.
9. A bill of material list was created and is located in the project tree.
10. Open the list by double-click the list page number in the project tree.

0	1	2	3	4	5	6	7	8	9
---	---	---	---	---	---	---	---	---	---

Bill of material

Symbol designation	Units	Description	Article number Order number	Manufacturer Supplier	Cross-reference
+AK1-X1	3	Feed-through terminal, 0,5-10/0,5-6mmf	ZDU 6 160 862 0000	Wekhüller Wekhüller	/11
+AK1-X1	1	End plate, beige, for ZDU6	ZAP/TW5 160 883 0000	Wekhüller Wekhüller	/11
+AK1-001	1	Main switch, 3-pole, 32A, rear mounting	035484 P1-32A/V/SVB-SV	Moeller Electric Moeller Electric	/11
-LAYOUT	1	Compact enclosure AE, 500x500x210	1050500 1050500	Rittal Rittal	/10

Picture 3. Result of the bill of material generation.

NOTE!

- Double-click the **cross-references** to navigate to the page.

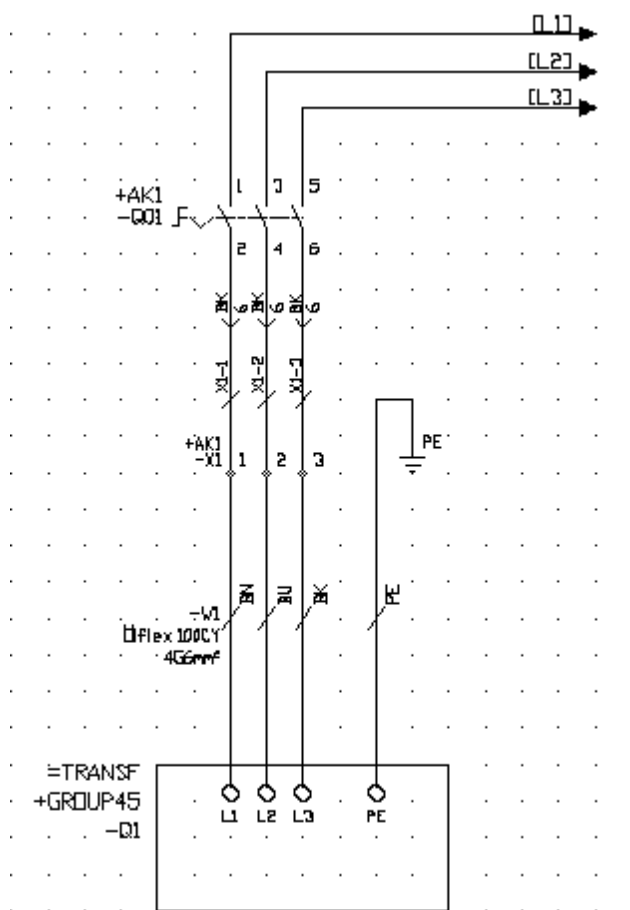
1.27 Guide D4 - Generate order list

An order list is searching and generating all components from the logic symbols defined in the circuit diagrams. The list is gathering the information from the logic symbol and combines it with data from the database library, then finally outputs a sorted order list.

The order list will also collect information from the layout pages. If you insert a component to a layout page, then you can select the component to be included to component lists.

In difference with the bill of material list, an order list will not save the link from one specific component to one specific logic symbol. Each component will only be presented once and if you have selected the same component for several logic symbols, then Elwin will add the units to the component. You will also get the order list as one list of each supplier.

As base for this guide, we will use guide B2, B9 and C3, but the terminal symbols X1:1, X1:2 and X1:3 also have components defined.



Picture 1. The circuit diagram created in guide B2 and B9.

Step by step

1. Select a page header of **Order list** (if it's already selected then jump to step 8).

2. Click **Project** in the menu bar, then click **Project settings**.
3. Click the tab **Page header**.
4. Click the database button (*) next to the **Purchase order list** text field.
5. The **Database of Page header** dialog box opens. Search and select **PH_CO_ORDER**.
6. Click **OK**.
7. Click **OK** of the dialog box **Project settings**.
8. Right-click the project name in the project tree, in the popup menu click **Create new**, then click **Order list**.
9. An order list was created and is located in the project tree. Please note there will be one list per supplier.
10. Open the list by double-click the list page number in the project tree.

Row	Units	Description	Order number	Article number	Manufacturer
1	1	Main switch, 3-pole, 32% main mounting	PL-05/W/3V8-0U	003484	Moeller Electric

Supplier: Moeller Electric
 Telephone:
 Fax:
 Contact person:
 Our order number:
 Shipment number:
 Delivery time:
 Number articles: 1

1 of 1

Picture 2. Supplier 1 of the order list generation.

Row	Units	Description	Order number	Article number	Manufacturer
1	1	Device enclosure (E 300/300/20)	300300	100300	Rittal

Supplier: Rittal
 Telephone:
 Fax:
 Contact person:
 Our order number:
 Shipment number:
 Delivery time:
 Number articles: 1

1 of 1

Picture 3. Supplier 2 of the order list generation.

Row	Units	Description	Order number	Article number	Manufacturer
1	1	Feed-through terminal, 0.5-1L/703-Smf	361 662 000	Z01 6	Verstücker
2	1	End plate, beige, Per Z016	361 662 000	Z01 TV3	Verstücker

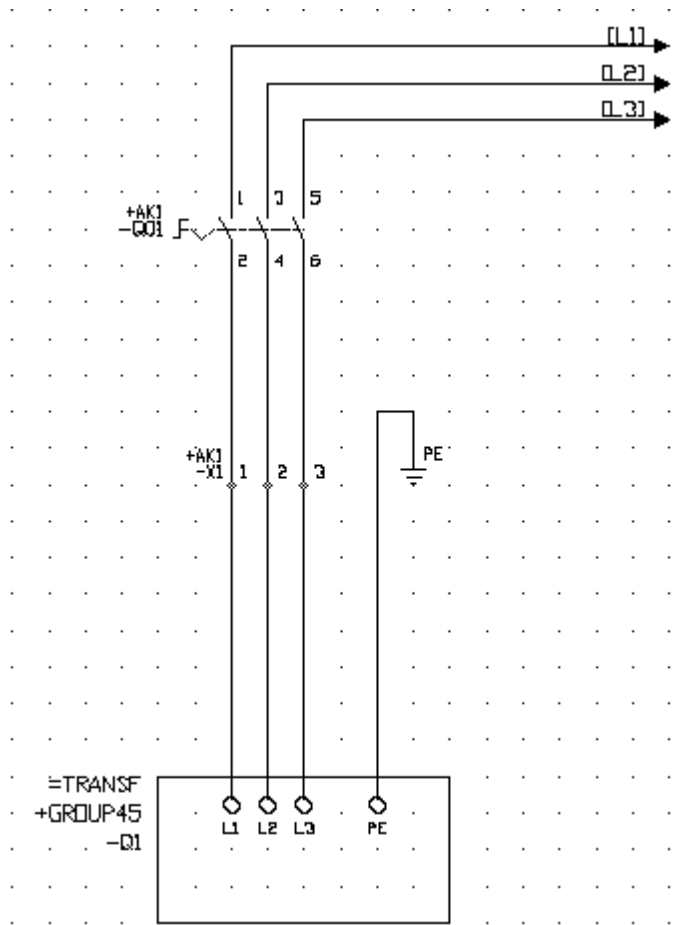
Picture 4. Supplier 3 of the order list generation.

1.28 Guide D5 - Generate engraving plate list

An engraving plate list is searching and generating all engraving plates from the logic symbols defined in the circuit diagrams. The list is gathering the information from the logic symbol and combines it with data from the database library, then finally outputs sorted engraving plate layouts.

The engraving plate list generator will also include the engraving plates defined in the layout pages.

As base for this guide, we will use guide B2 and B10.



Picture 1. The circuit diagram created in guide B2 and B9.

Step by step

1. Select a page header of **Engraving plate list** (if it's already selected then jump to step 8).
2. Click **Project** in the menu bar, then click **Project settings**.
3. Click the tab **Page header**.
4. Click the database button (*) next to the **Engraving plate list** text field.
5. The **Database of Page header** dialog box opens. Search and select **PH_EPL**.
6. Click **OK**.
7. Click **OK** of the dialog box **Project settings**.
8. Right-click the project name in the project tree, in the popup menu click **Create new**, then click **Engraving plate list**.
9. An engraving plate list was created and is located in the project tree.
10. Open the list by double-click the list page number in the project tree.

PLC list									
Symbol	Symbol	PLC name	PLC name	Description	Created	Modified			
-M	1	M1.0	START	Start machine	.1	.1			
-M	2	M1.1	STOP	Stop machine	.1	.1			
-M	3	M1.2			.1	.1			
-M	4	M1.3			.1	.1			
-M	5	M1.4			.1	.1			
-M	6	M1.5			.1	.1			
-M	7	M1.6			.1	.1			
-M	8	M1.7	STOP	Stop machine	.1	.1			
-M	9	M1.8			.1	.1			
-M	10	M1.9			.1	.1			
-M	11	M1.10			.1	.1			
-M	12	M1.11			.1	.1			
-M	13	M1.12			.1	.1			
-M	14	M1.13			.1	.1			
-M	15	M1.14			.1	.1			
-M	16	M1.15			.1	.1			
-M	17	M1.16			.1	.1			
-M	18	M1.17			.1	.1			

Picture 1. Result of the list generation.

TIPS!

Double-click the **cross-references** to navigate to the page.

1.30 Guide D7 - Generate table of contents

The list table of contents creates a list with the pages within the project location. You can have several table of contents within the project, as long as the lists are not in the same location.

As an option you can design the page header, so Elwin creates miniature snap-shots of each page.

When you generate the table of contents list, then make sure you update all the other lists before generating the table of contents.

As base for this guide, we will use guide A, B and C.

Step by step

1. Select a page header of **Table of contents** (if it's already selected then jump to step 8).
2. Click **Project** in the menu bar, then click **Project settings**.
3. Click the tab **Page header**.
4. Click the database button (*) next to the **Table of contents** text field.
5. The **Database of Page header** dialog box opens. Search and select **PH_TOC2**.
6. Click **OK**.
7. Click **OK** of the dialog box **Project settings**.
8. Right-click the project name in the project tree, in the popup menu click **Create new**, then click **Table of contents**.
9. A table of contents list was created and is located in the project tree.
10. Open the list by double-click the list page number in the project tree.

D	L	E	S	4	5	6	7	8	9								
Table of contents																	
	Page number: 1	Description: Main circuiting	Created: 1/5/2006	Modified by: XM		Page number: 13-1	Description: PLC child symbol	Created: 2/5/2006	Modified by: XM								
	Page number: 2	Description: Child symbol	Created: 1/3/2006	Modified by: XM		Page number: 13-2	Description: PLC child symbol	Created: 2/3/2006	Modified by: XM								
	Page number: 8	Description: Use of PLC child symbol	Created: 1/3/2006	Modified by: XM		Page number: 13-3	Description: PLC child symbol	Created: 2/3/2006	Modified by: XM								
	Page number: 9	Description: Use of PLC child symbol	Created: 1/5/2006	Modified by: XM		Page number: 14-1-1	Description: PLC child symbol	Created: 2/5/2006	Modified by: XM								
	Page number: 10	Description: Layout	Created: 1/3/2006	Modified by: XM		Page number: 14-2-1	Description: PLC child symbol	Created: 2/3/2006	Modified by: XM								
<div style="display: flex; justify-content: space-between;"> 13-4 14-8 </div> <table border="1" style="width: 100%; border-collapse: collapse;"> <tr> <td style="width: 50%;"></td> <td style="width: 50%;"></td> </tr> <tr> <td style="width: 50%;"></td> <td style="width: 50%;"></td> </tr> <tr> <td style="width: 50%;"></td> <td style="width: 50%;"></td> </tr> <tr> <td style="width: 50%;"></td> <td style="width: 50%;"></td> </tr> </table>																	

Picture 1. Result page 1 of the list generation.

D	L	E	S	4	5	6	7	8	9								
Table of contents																	
	Page number: 14-3-1	Description: PLC child symbol	Created: 2/5/2006	Modified by: XM		Page number: 14-3-1	Description: PLC child symbol	Created: 1/5/2006	Modified by: XM								
	Page number: 15-1	Description: PLC child symbol	Created: 1/3/2006	Modified by: XM													
	Page number: 16-1	Description: PLC child symbol	Created: 1/3/2006	Modified by: XM													
	Page number: 17-1	Description: PLC child symbol	Created: 1/5/2006	Modified by: XM													
	Page number: 17-2	Description: PLC child symbol	Created: 1/3/2006	Modified by: XM													
<div style="display: flex; justify-content: space-between;"> 13-4 14-8 </div> <table border="1" style="width: 100%; border-collapse: collapse;"> <tr> <td style="width: 50%;"></td> <td style="width: 50%;"></td> </tr> <tr> <td style="width: 50%;"></td> <td style="width: 50%;"></td> </tr> <tr> <td style="width: 50%;"></td> <td style="width: 50%;"></td> </tr> <tr> <td style="width: 50%;"></td> <td style="width: 50%;"></td> </tr> </table>																	

Picture 2. Result page 2 of the list generation.

TIPS!

Double-click the **page number** to navigate to the page.

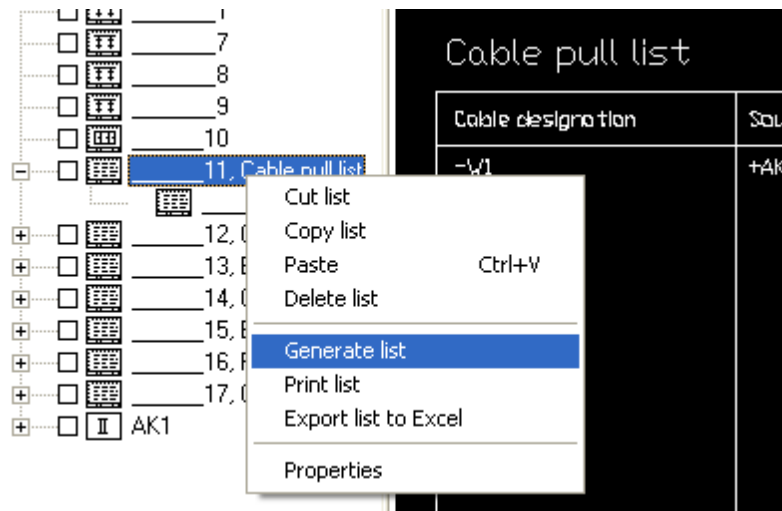
1.31 Guide D8 - Re-generate existing list

When you have edited a circuit diagram or layout page, then the generated lists may be out of date and you need to re-generate the information. Elwin will only re-generate a list if you edit its properties.

There are several ways to re-generate.

Step by step (one single list)

1. Right-click the list page in the project tree.
2. In the popup menu, click **Generate list**.



3. The list gets re-generated.

Step by step (all list in the project)

1. Click **Page** in the menu bar, then click **Generate all lists**.
2. All lists in the project get re-generated.

NOTE!

When you re-generate all lists in the project, Elwin will automatically generate in the correct order. First possible cable connection lists, then "not mentioned lists" and finally the table of contents.

1.32 Guide E1 - Open the database library

What is a database library? Well, basically it includes all information which you will repeatedly use and don't want to design several times. It includes the page header, the logic symbols, layout symbols, component definitions, cable and wire definitions, cable ducts, device mounting rails and so on... Simply, everything you will need to design a control panel project.

The database library comes in two files with the same name, but with different file extensions (*.ewd and *.ewd00).

When you change anything in the database library and save the library, then the change will immediately take effect in all projects which are using the database. This is very powerful, but may cause serious problems if not handled correctly.

We start easy in this guide and just open the database for edit.

Step by step

1. Click **Project** in the menu bar, then click **Open**.
2. The **Open Elwin file** dialog box opens.
3. Change the file format to **Elwin database (*.ewd)**.
4. Search in traditional Windows procedure to the folder with the database library "c:\program\elwin\database library\". This folder contains a database library with the name **IEC_E**.
5. Click **Open**.
6. The database opens and the objects are grouped in the project tree.

NOTE!

- A database library may be opened even though it may currently be used by another user.

1.33 Guide E2 - Add component

Each component you want to include in a project's circuit diagram and layout page must be defined in the database library. This guide will instruct the procedure of adding a component to the database.

Step by step

1. Open the database library (see [Guide E1](#)).
2. Right-click **Component** in the project tree.
3. Click **Add component** in the popup menu.
4. Click the tab **Properties**.
5. Enter a description of the component in the text field **Description**.
6. Enter the order number of the component in the text field **Order number**.
7. Enter the article number of the component in the text field **Article number**.
8. Select a manufacturer by clicking the database button (*) right to the text field **Manufacturer**.
9. Select a supplier by clicking the database button (*) right to the text field **Supplier**.

The screenshot shows a 'Properties' dialog box with the following fields and values:

Field	Value
Description	Main switch, 3-pole, 32A, rear mounting
Order number	P1-32/W/SVB-SW
Article number	055484
Manufacturer	Moeller Electric
Supplier	Moeller Electric
Component group	
Technical description	Without e-stop function, rear mounting, front IP65
	Black rotary handle&locking collar, lockable 0-pos

Picture 1. Result of step 1-9.

10. Click the tab **Layout appearance**.
11. Enter the width of the component in the field **Width**.
12. Enter the height of the component in the field **Height**.

Properties | Layout appearance | Price | Status | Information

Size width: mm

Size height: mm

Weight: kg

Recommended logic symbol: * X

Recommended layout symbol: * X

Picture 2. Result of step 10-12.

13. Click the tab **Layout appearance**.

14. Enter the number of units in one pack (=if you order this article number how many units will you get).

Properties | Layout appearance | Price | Status | Information

Units in pack:

Price/Pack: Discount: % Currency: *

Price/unit:

Picture 3. Result of step 13-14.

15. Click **OK** to add the component to the database library.

NOTE

- There are several more settings for a component, but this guide will only instruct the minimum settings.

1.34 Guide E3 - Create new layout symbol

A layout symbol is simply a graphical object with lines, rectangles, circles and text objects. The graphics intention is to form a true view or layout of a component. When the layout symbol is finished, then you can link the layout symbol with one or several components. This link will enable the user of the component to very easy find and use a layout symbol.

One layout symbol can hold up to 8 versions (or sub layout symbols). The versions intention is to enable you to define the same layout in different angles, different detail level, and so on... It's up to you as a designer of the layout symbol to use the versions. One version is minimum.

This guide will show how to define and draw a layout symbol. When this guide is finished then continue with [Guide E4](#) to link our new layout symbol with a component.


Step by step

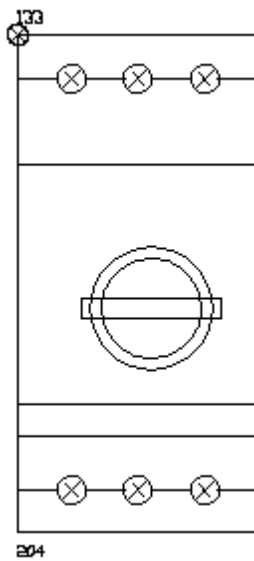
1. Open the database library (see [Guide E1](#)).
2. Right-click **Layout symbol** in the project tree.
3. Click **Add layout symbol** in the popup menu.
4. The **New layout symbol** dialog box opens. Click the tab **Properties**.
5. Enter a database name of the layout symbol in the text field **Name**. Please note the name must be unique in the database's layout symbols.
6. Enter an optional description text of the layout symbol in the text field **Description**.
7. Click the tab **Editing**.
8. This tab holds information about the editing area you need for the design of the layout. If the component you will draw is 100mm height and 200mm width, then set the area about 50% more. The layout symbol we will draw in this guide is 45mm width, so enter 80mm in the **Width of edit area**. Our component height are 93mm so enter 140mm in the **Height of edit area**.
9. Select the option **Grid** and then enter the grid value to 1mm.
10. Click **OK**.

Elwin created the layout symbol and has put it in the project tree in group **Layout symbol**. One layout version was automatically created.

11. Open the group **Layout symbol** in the project tree by double-click the text **Layout symbol**.
12. Open the **Layout symbol name** you just entered in step 5 by double-click the name.
13. Elwin created a layout version called **No name**. Change this name by right-click **No name**.
14. In the popup menu click **Properties**.
15. The **Layout version properties** dialog box opens. Enter a new name of the version, like **Front view**.
16. Click **OK**.

Now it's time to open the layout version and draw the graphics..

17. Double-click the layout version, i.e. **Front view**.
18. As you see Elwin has automatically created a rectangle and a text object (133).
19. Change the size of the rectangle to 45mm width and 93mm height, by click in the lower right corner and form a new size. While you are forming the new size, Elwin is presenting the size of the rectangle in the lower part of the screen. You can also use the arrow keys on the keyboard. When correct size is set then click (or press Enter key).
20. The insertion point is defined by the symbol . Move the symbol in appropriate position. Normally a good position is the upper left corner, but you are free to set it anywhere. The insertion point will not be visible when the user use the layout symbol.
21. Draw a picture according to the picture 1. The most important are text object 133 (installation, location and symbol designation) and text object 204 (cross-reference). The outer rectangle is also important so the space required are clearly shown for the user of the layout symbol. The objects inside are nice to have, but really not necessary.



Picture 1. Result of step 21.

22. This is a component which you will mount on a device mounting rail. So you need to define the outer rectangle as "break device mounting rail".
23. Right-click the outer rectangle object.
24. In the popup menu select **Breaks device mounting rail** (if the option is already selected then ignore and press ESCape key).
25. Now it's time to save the database and really activate all changes. Click **Project** in the menu bar, then click **Save**.

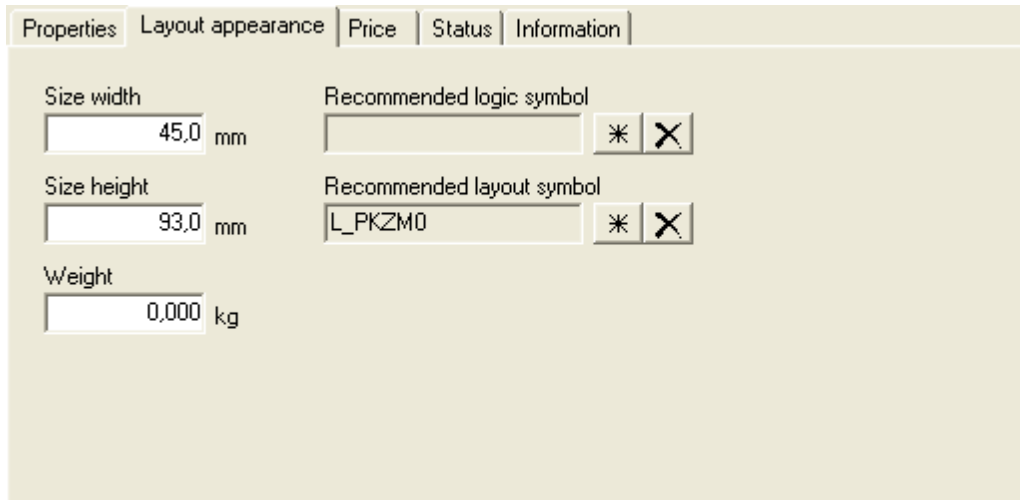
1.35 Guide E4 - Link a layout symbol with component

If you create and edit a layout symbol, then you really should link the symbol to a component. The component will then use the layout symbol on the layout pages.

You can at any time change component's linked layout symbol, but be aware Elwin will need to change all existing layout pages.

Step by step

1. Open the database library (see [Guide E1](#)).
2. Create a component as in [Guide E2](#).
3. Create a layout symbol as in [Guide E3](#).
4. Right-click the component in the project tree.
5. In the popup menu click **Properties**.
6. Click the tab **Layout appearance**.
7. Click the database button (*) to the right of **Recommended layout symbol**.
8. The **Database of layout symbols** dialog box opens. Select a layout symbol in the list, which you like to link to the component.
9. Click **OK**.
10. Finally, click **OK** to save the changes to the component.



Picture 1. Result of step 9.

1.36 Guide E5 - Create a parent symbol

This guide will show how to create a parent logic symbol. The symbol will have two connection points and a variable text. In the end of guide we will use the symbol in a circuit diagram.

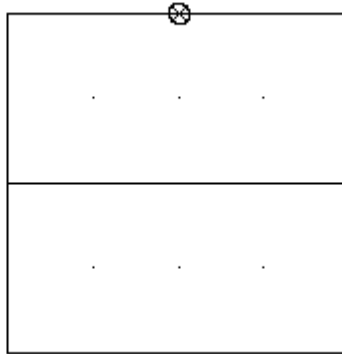
Step by step

1. Open the database library (see [Guide E1](#)).
2. Double-click the group **Logic symbol** in the project tree.
3. Right-click **Parent symbol**, in the popup menu click **Add parent symbol**.
4. Click the tab **Properties**.
5. Enter a database name of the logic symbol in the text field **Name**. Please note the name must be unique in the database's logic symbols. Enter **TEST_H**.
6. Enter an optional description text of the logic symbol in the text field **Description**. Enter **Test symbol of guide E5**.
7. In the text field **Symbol designation** enter a character which will be the pre-defined when the user will use the symbol. Enter **SP**.
8. Click the tab **Editing**.
9. This tab sets the size of editing area needed for the design of the logic symbol. Enter **50mm** and **50mm** in the fields **Width** and **Height**.
10. Select the option **Grid** and enter the grid value to **2.5mm**.
11. Click **OK** to create the logic symbol.


Elwin created the logic symbol and has put it in the project tree in group **Logic symbol** och **Parent symbol**. Now it's time to edit the logic symbol's graphics and logic information. Each logic symbol must be defined in four different versions.

12. Open the logic symbol by double-click the version **Main version**.
13. As standard Elwin has created the text object 133, which is the "Installation+location+symbol designation" text type. We leave this object for now.
14. Draw a rectangle and a line to form a symbol like below. Use **Draw** in the menu bar to select the tools.

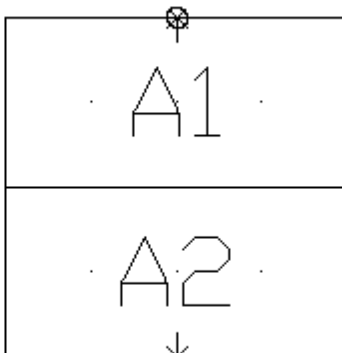
133



Picture 1. Result of step 14.

15. Now we will add connection points. Click **Insert** in the menu bar, then click **Connection point**.
16. The connection point object attaches to the mouse pointer. Move the pointer to the insertion point  and click.
17. The **Insert new connection point dialog** box opens. Enter a connection point name which will be the pre-defined when an user uses the symbol. Enter **A1**.
18. The text field **Description** is description of the connection point. This text will be visible for the user. Enter **Supply+**.
19. The list box **Type** defines in which direction this connection point will search for another connection point. Select **120. Connection direction up**.
20. **Text height**, **Adjust** and **Rotation** are properties of the connection point name. Enter 2mm, center and 0 rotation.
21. Click **OK**.
22. Click the text (A1) to select the object. Click the insertion point of the text object (A1) to move the text in center position. When in position then click once again.
23. One connection point is now defined. Repeat ste 15-22 for the second connection point. This point will have pre-defined connection point name **A2**, description text **Common**, type **122. Connection direction down**.

133

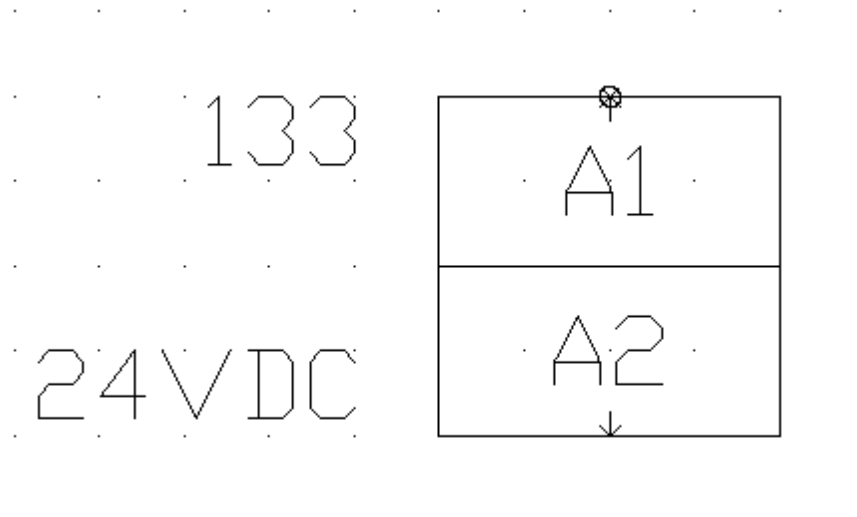


Picture 2. Result of step 23 (please note the two arrows are just indicating a connection point, they will not show in the circuit diagram)

24. Move the text object 133 in level with the upper part of the rectangle.
25. Change the insertion point of the text object 133, by double-click and change the properties **Text adjust** to right-top.

Now we will define a variable text. A variable text is text for the user of the symbol to specify.

26. Click **Draw** in the menu bar, then click **Text**.
27. Select **Type** to **146. Variable text**.
28. In the text field **Description**, enter a description of the variable text. This text will be visible for the user of the symbol. Enter **Voltage**.
29. The text field **Text** will be the pre-defined text for the user. The field may be left blank, but we enter **24VDC**.
30. **Text height**, **Text adjust** and **Rotation** are properties of the variable text. Enter 2mm, right-top and 0 rotation.
31. Click **OK**.
32. The text object attaches the mouse pointer. Move the pointer in position and click.



Picture 3. Result of step 32.

The logic symbol is now finished for one version (main version). Now you have to define three more (+90, +180 and +270), but Elwin has a function to auto create the rotate versions. The auto rotation won't create perfect versions, but it will give you a good start of edit.

33. Right-click the **Main version** of our symbol **TEST_H**, in the popup menu click **Create rotated versions**.
34. A question dialog box opens, click **Yes** to verify the auto creation.
35. Save the database library by clicking **Project** in the menu bar, then click **Save**.

Now we will open a project and use our logic symbol.

Step by step

36. Open the project created in [Guide A2](#).
37. Create a new circuit diagram, by follow [Guide B1](#).
38. Open the new circuit diagram by double-click the page number in the project tree.
39. Click **Insert** in the menu bar, then click **Logic symbol**.
40. The **Database of logic symbol** dialog box opens. Search and select our logic symbol **TEST_H**.
41. Click **OK**.
42. The logic symbol attaches to the mouse pointer. Move the symbol in position and click.
43. The **New logic symbol** dialog box opens. Note the symbol designation is pre-defined as entered in

- step 7. Enter the symbol designation to **SP1**.
44. Click the tab **Connection points**.
 45. Note there are two connection points as defined in 15-23. You have the possibility to change A1 and A2.
 46. Click the tab **Symbol texts**.
 47. Note there is one symbol text(variable text) as defined in step 26-32.
 48. Click **OK**.

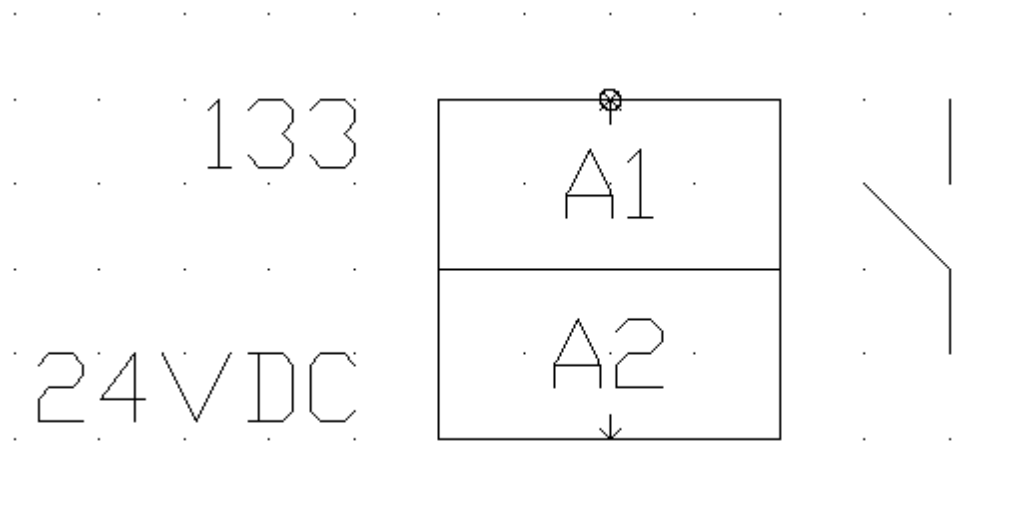
The logic symbol are now inserted to a circuit diagram. Open the database again and change some graphics (add a line or re-size the rectangle), then open the circuit diagram again and note the logic symbol has changed.

1.37 Guide E6 - Create child symbol to a parent symbol

In guide E5 we created a parent symbol with two connection points. In this guide we will complement the parent symbol with a cross reference symbol with two connections, i.e. an auxiliary contact will be added.

Step by step

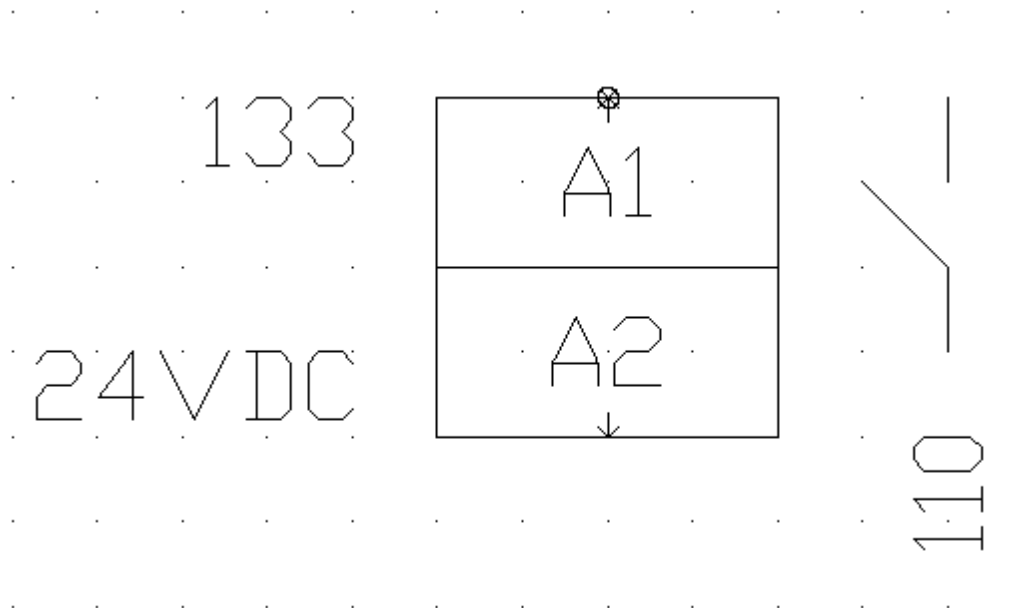
1. Open the database library (see [Guide E1](#)).
2. Open the logic symbol by double-click the version **Main version** of logic symbol **TEST_H** in group the **Logic symbol/Parent symbol**.
3. Draw a contact symbol as in below picture 1. Use the tool **Draw** in the menu bar, then **Line**.



Picture 1. Result of step 3.

4. Click **Insert** in the menu bar, then click **Child cross reference point**.
5. The **Insert new child cross reference point** dialog box opens. Select **Type** to **110. Child cross reference point**.
6. In the description text field enter a text to explain the function of the contact. This text will be visible for the user of the logic symbol. Enter **Fault contact**.
7. The text field **Text** has no real function for a child cross reference point so better leave this field blank.
8. **Text height**, **Text adjust** and **Rotation** are properties of the future cross reference text in the circuit diagram. Enter 2mm, Right-Center and 90 rotation.
9. Click **OK**.
10. The text object attaches the mouse pointer. Move the pointer in position just below the contact and

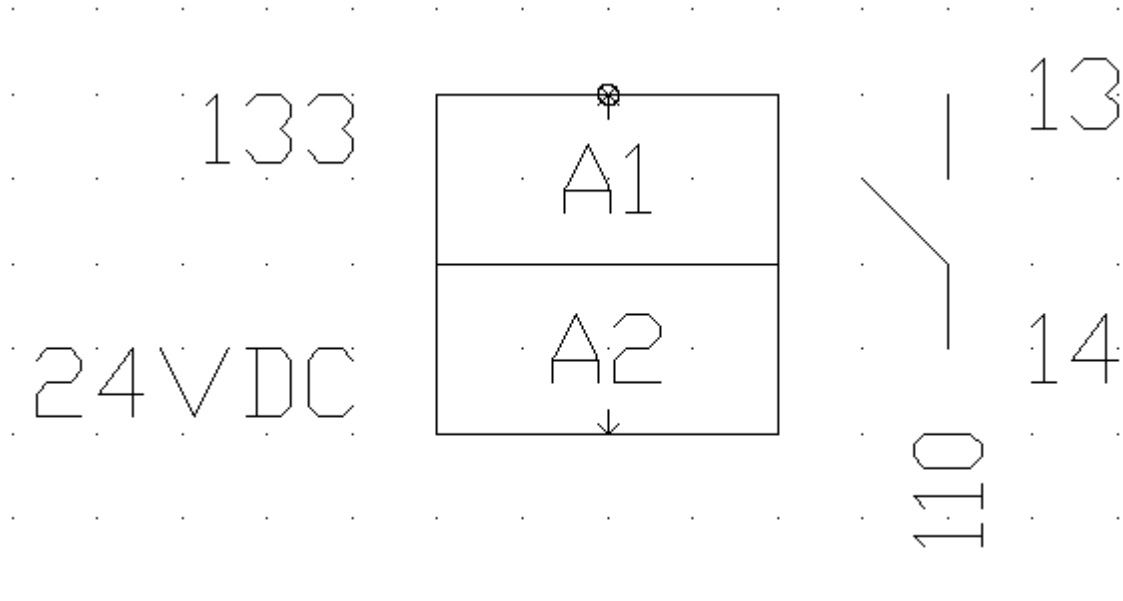
click.



Picture 2. Result of step 10.

Nu ska spegelpunktens anslutningstexter definieras.

11. Click **Insert** in the menu bar, then click **Child connection point**.
12. The **Insert new child connection point** dialog box opens. Select **Type** to **112. Child connection point**.
13. In the list box **Member of cross reference point**, select the **Fault contact** we just created in step 4-10. Since we only created one child cross reference point in this logic symbol, the list box has only one item.
14. In the text field **Text** enter a pre-defined connection point name for the connection. Enter **13**.
15. **Text height**, **Text adjust** and **Rotation** are properties of the connection text. Enter 2mm, Left-Center and 0 rotation.
16. Click **OK**.
17. The text object attaches the mouse pointer. Move the pointer in position just to the right of the contact and click.
18. Repeat step 11-17, but enter **Text** to **14**.



Picture 3. Result of step 18.

The logic symbol is now ready with the new cross reference connection point. Since all four version of the logic symbol must be identical in function we have to repeat step 1-18 for the rest of the versions. Or use step 19-20 to create automatic versions.

19. Right-click the **Main version** of our symbol **TEST_H**, in the popup menu click **Create rotated versions**.
20. A question dialog box opens, click **Yes** to verify the auto creation.
21. Save the database library by clicking **Project** in the menu bar, then click **Save**.

Now let's use the logic symbol in a circuit diagram.

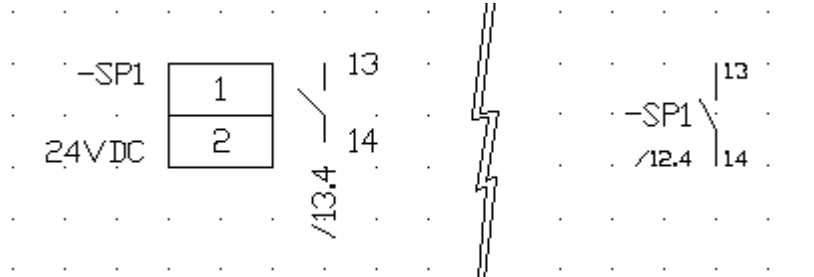
22. Open the same project you used in [Guide E5](#).
23. Double-click the circuit diagram you created and edited in [Guide E5](#).
24. Since we have edited the logic symbol used in the circuit diagram, an error message (605) will popup with the message we must edit our logic symbol. Click **OK**.
25. The **Logic symbol properties** dialog box opens automatically. Click the tab **Child symbols**.
26. The list box will show all child symbols in the logic symbol. You can change the cross points (13 and 14) if you like.
27. Click **OK**.

Now let's define a child logic symbol to cross-reference our logic symbol (SP1).

28. Create a new circuit diagram where to use the contact ([Guide B1](#)).
29. Open the new circuit diagram by double-click the page number in the project tree.
30. Insert a child logic symbol to cross-reference our logic symbol, by clicking **Insert** in the menu bar, then click **Logic symbol**.
31. The **Database of logic symbol** dialog box opens.
32. Filter the number of logic symbols by right-click the first row in the column **Type**.
33. In the popup menu click **Child symbol**.
34. Search and select **KS_010**.
35. Click **OK**.
36. The logic symbol attached the mouse pointer. Move the pointer in position and click.
37. The **New logic symbol** dialog box opens. Enter symbol designation to **SP1**, since we like cross-

reference our logic symbol.

38. Click the tab **Connection points**.
39. Make sure the connection point names are the same as in our parent symbol (13 and 14).
40. Click **OK**.



Picture 4. As you can see a cross-reference text was automatically created between the two logic symbols (/13.4 and /12.4). Double-click the cross-reference text navigate between the circuit diagrams.

NOTE

- Note you need to have the same symbol designation for a cross-reference text to be created. Also, the same number of cross connections and names. Try and change the connection points and see what happens.

1.38 Guide E7 - Create a new page header

A page header defines the size, area and objects of a blank circuit diagram, a layout page or list. A page header of the type list you will also define which text fields to show.

In this guide we will create a page header of **Wire list**.

Steg för steg

1. Open the database library (see [Guide E1](#)).
2. Right-click **Page header** in the menu bar, then click **Add page header**.
3. The **New page header** dialog box opens.
4. Click the tab **Standard**.
5. Enter a name of the new page header. Please note the name must be unique in the database's page headers. Enter the name to **PH_TEST**.
6. Enter an optional description of the page header in the text field **Description**.
7. In the list box **Type** select **10. Wire list**.
8. Click **OK**.
9. Double-click the new page header in the group **Page header** and **Wire list** in the project tree.
10. Draw a horizontal line in the bottom of the page header (see picture 1) and two vertical lines to divide into sections. Use **Draw** in the menu bar, then click **Line**.
11. Create three texts as in picture 1. Use **Draw** in the menu bar, then click **Text**. In the dialog box select **Type** to **6. Static text**.
Create the texts:
Page number
Previous page number
Next page number

There are several text fields which are good idea to show in a page header. Add text type 907, 908 and 930 in the middle section. Add 960, 970 and 915 in the left section.

The text types in the series 900-999 are insertable to all types of page headers. For each type list page header there a one series of text types added. These series are all unique to each page header list. The text type series for a **Wire list** is series 1000-1099.

This page header will show Wires in the project, so for this purpose we need to define some text fields.

19. Click **Draw** in the menu bar, then click **Text**.
20. Select the type to **1016.(source)Inst+Loc+Symbol+Conn.point(compress)**.
21. Text height 2,0mm
22. Text adjust to Left and middle.
23. Rotation 0.
24. Click **OK**.
25. Position the text object in upper part of the page header.
26. Since this is a text type of the list, Elwin will simulate how the list will look like.

	0	1	2	3	4	5	6	7	8	9
0										
1	1016									
2	1016									
3	1016									
4	1016									
5	1016									
6	1016									
7	1016									
8	1016									
9	960 970 Print date 915	907 908 909					Page number 903 Previous page number 922 Next page number 923			

Picture 3. Result of step 26.

The number of list rows is set by the **Page header properties**. The properties will also set the gap between the rows, list in groups and so on.

27. We like to increase the number of list rows. Right-click the page header (PH_TEST) in the project tree, in the popup menu click **Properties**.
28. Click the tab **Lists**.
29. Enter the list rows per page to **30**.
30. We don't like to show the lists in groups, so enter **0** in the field **List rows in group**.
31. The gap between list rows are **5mm** which is OK, so do not change anything.
32. The field **Gap between groups** are not applicable since we entered 0 in the field **List rows in**

group.

33. Click **OK** and note how the list changed.
34. Add the text types 1026, 1030 and 1031.

	0	1	2	3	4	5	6	7	8	9
0										
	1016		1026		1030		1031			
1	1016		1026		1030		1031			
	1016		1026		1030		1031			
	1016		1026		1030		1031			
2	1016		1026		1030		1031			
	1016		1026		1030		1031			
	1016		1026		1030		1031			
3	1016		1026		1030		1031			
	1016		1026		1030		1031			
	1016		1026		1030		1031			
4	1016		1026		1030		1031			
	1016		1026		1030		1031			
	1016		1026		1030		1031			
5	1016		1026		1030		1031			
	1016		1026		1030		1031			
	1016		1026		1030		1031			
6	1016		1026		1030		1031			
	1016		1026		1030		1031			
	1016		1026		1030		1031			
7	1016		1026		1030		1031			
	1016		1026		1030		1031			
	1016		1026		1030		1031			
8	1016		1026		1030		1031			
9	960 970 Print date 915		907 908 909					Page number 903 Previous page number 922 Next page number 923		

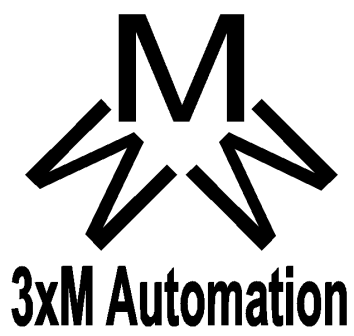
Picture 4. Result of step 34.

35. Save the database library by clicking **Project** in the menu bar, then click **Save**.

Test our new page header by doing [Guide D1](#) but with our page header PH_TEST.

	0	1	2	3	4	5	6	7	8	9
0										
1		-001e -0014 -0016		-X1a -X12 -X13		X1-1 X1-2 X1-3		H07V-K H07V-K H07V-K		
2										
3										
4										
5										
6										
7										
8										
9		<> <your company> Print date 6/5/2016							Page number 5-1 Previous page number 4 Next page number 6	

Picture 5. Result of [Guide D1](#) with our page header PH_TEST.



Copyright © 2006-2017 3xM Automation