

# **ELCAD/AUCOPLAN 7**

## **News In 7.4.0**

Issue: November 12, 2007

**AUCOTEC AG**

Oldenburger Allee 24, D-30659 Hannover  
Phone: +49 (0)511 61 03-0, Fax: +49 (0)511 61 40 74

**Copyright:** All rights, especially the right of reproduction and distribution as well as translation, are reserved. No part of this book may be reproduced, stored in retrieval system, or transmitted in any form or by any means, electronic, mechanical, photocopying, microfilming, recording, or otherwise, without prior permission from **AUCOTEC AG**.

**Exclusion of liability:** Texts and software have been prepared with the greatest of care. The publishers as well as the authors cannot assume any legal or other liability of any nature for potential faulty statements and their consequences, which shall apply also for the software potentially included.

**Trademark:** AUCOPLAN<sup>®</sup> and ELCAD<sup>®</sup> are registered trade marks of the AUCOTEC AG, Germany. MS-DOS<sup>®</sup> and Windows<sup>®</sup> are registered trade marks of the Microsoft Corporation, USA.

# Contents

<b>1</b>	<b>New Features of ELCAD/AUCOPLAN 7.4.0 .....</b>	<b>1</b>
1.1	<b>General Innovations .....</b>	<b>2</b>
1.1.1	Release for Windows Vista .....	2
1.1.2	AUCOPLAN Studio .....	2
1.1.3	Windows-Compliant Dialogs .....	2
1.1.4	International Version of ELCADFluid .....	3
1.1.5	Optimization of the Print Functionality .....	3
1.2	<b>New Features in Drawing Editing .....</b>	<b>3</b>
1.2.1	Editing of Several Drawings via Tabs .....	3
1.2.2	Simple Cut & Paste of Drawings .....	3
1.2.3	Placing Symbols and Connections in Drawings .....	4
1.2.4	Checking Symbols During Drag & Drop .....	4
1.2.5	Colour PDF and Printer Output .....	4
1.2.6	Multiple Representation of Cores and Wires.....	5
1.2.7	Working Status and Connector Status Always Visible.....	5
1.2.8	Creating a Part While Editing a Symbol.....	5
1.2.9	Part Assignment via Drag & Drop with More than One Part Number .....	5
1.3	<b>New Features in Project Editing.....</b>	<b>6</b>
1.3.1	Strip-Related Terminal Diagrams Without Batch Processing .....	6
1.3.2	Exempting Code Numbers from the Revision .....	6
1.3.3	Search for the Appropriate Master Data Record.....	6
1.3.4	Free Code Number for Creating Cross-References .....	6
1.3.5	Expanded Translate Text Selection .....	7
1.3.6	Extension of the Replace Function for Texts.....	7
1.3.7	Specifying Data for Sub-Devices in a Symbol .....	7
1.3.8	Deletion of Existing Graphic Lists before Renewed Creation.....	8
1.3.9	Archiving with Several Databases.....	8
1.3.10	Expanded Functions for Form Editing.....	8

<b>1.4</b>	<b>New Features in Terminal Editing</b> .....	<b>9</b>
1.4.1	Terminal Editing from the Terminal Diagram.....	9
1.4.2	Freezing Terminals in the Item Editor .....	9
1.4.3	Extended Standard Strip Editing Options .....	9
1.4.4	Different Terminal Types in a Terminal Diagram.....	10
1.4.5	Cross-References between Terminals Displayed Separately.....	10
<b>1.5</b>	<b>New Features in Cable Editing</b> .....	<b>11</b>
1.5.1	Freezing Cores .....	11
1.5.2	Cross-Section Doubling by Means of Parallel Cores .....	11
1.5.3	Navigation from Cable Planning .....	11
1.5.4	Inserting Cables from the Database .....	11
<b>1.6</b>	<b>New Features in List Editing</b> .....	<b>12</b>
1.6.1	Device-Related List of Connectors.....	12
1.6.2	Keeping Manual Entries When Regenerating Contents Tables..	12
<b>1.7</b>	<b>New Features in ELCADmaintenance</b> .....	<b>12</b>
1.7.1	Navigation to Graphic Representations Extended .....	12
<b>1.8</b>	<b>New Interface Features</b> .....	<b>13</b>
1.8.1	EDM Operation for ELCADmaintenance and ELCADview Extended .....	13
1.8.2	Extension of the DXF / DWG Output .....	13
1.8.3	DWG Output Extended to AutoCAD 2005, 2006 and 2007 .....	13
1.8.4	Support for Weidmüller M-Print Pro.....	13
1.8.5	Exporting Several Strip Layouts At Once .....	14
1.8.6	Individual Specification of a Compact PDF File.....	14

# **1 New Features of ELCAD/AUCOPLAN 7.4.0**

With ELCAD / AUCOPLAN 7.4.0 you get a version that contains a large number of improvements and innovations. For this purpose AUCOTEC has analyzed a large number of customer requests and put together the most important points in version 7.4.0. The most prominent extensions are the compatibility of your data and easier handling.

In this way we have in many places achieved an even easier handling of your operating steps and a further simplification of the operation of the functions. Moreover you will find a detailed documentation of the errors eliminated in version 7.4.0 in the readme files of the installation CD.

## 1.1 General Innovations

### 1.1.1 Release for Windows Vista

With ELCAD / AUCOPLAN 7.4.0, the Windows Vista ® operating system is now officially supported and enabled. Please refer to the product CD for information concerning peculiarities of the installation procedure and the operation under Windows Vista ®.

In return the Windows NT 4.0 operating system is no longer supported by version 7.4.0, several years after Microsoft stopped supporting Windows NT 4.

### 1.1.2 AUCOPLAN Studio

With version 7.4.0, AUCOPLAN Studio has been implemented as integration solution for ELCAD Studio and AUCOPLAN Basic. It offers solutions for alphanumeric planning of the detail engineering. AUCOPLAN Studio is a separate module available at extra expense.

With AUCOPLAN Studio, you can alphanumerically define and centrally change devices independent of graphics and circuit diagram. Object and frame representations are automatically updated, as is the rewiring information as soon as rewiring is activated. Alphanumeric and graphic editing can be arbitrarily mixed. Changes in the objects are carried out online for all representations.

AUCOPLAN Studio integrates external engineering data from the most varied sources of your company into your projects. You can import arbitrary drive lists, load and field device definitions into your projects. With AUCOPLAN Studio, imported data can be further processed in tabular form, specified and linked to the graphic. You can adopt the identification alternatively from the drawings or the imported device definitions.

Devices can be ordered already without functional assignment in the circuit diagram, and cabinet layout can be commissioned. The circuit diagram can be created later on.

Do you start with the cabinet layout?

With AUCOPLAN Studio you can copy complete cabinets from other projects. AUCOPLAN Studio fills the necessary device list on the basis of the device data copied simultaneously and automatically generates the parts list required for ordering.

### 1.1.3 Windows-Compliant Dialogs

In ELCAD / AUCOPLAN 7.4.0, a large number of dialogs of project and drawing editing have been revised according to the current Windows standards. They are thus easier to handle and better to operate. Wherever meaningful, functions and online help entries have been added.

### **1.1.4 International Version of ELCADFluid**

With version 7.4.0, an international, English-language version of ELCAD-Fluid with symbols and master data is available.

### **1.1.5 Optimization of the Print Functionality**

Version 7.4.0 optimally supports even complex printer configurations for printer output. Driver settings such as "Print on both sides", "Paper tray selection" or "Paper size" are taken into account and maintained.

## **1.2 New Features in Drawing Editing**

### **1.2.1 Editing of Several Drawings via Tabs**

With ELCAD / AUCOPLAN 7.4, you can place up to 30 drawings in tabs and thus quickly access them. Among other things this permits you to rapidly switch between the individual drawings or to efficiently copy drawing elements from one drawing to another one.

For this purpose double-click on a drawing in the Drawing tree while pressing and holding the Shift key, or open a drawing in the Drawing tree with the right mouse button using the function OPEN IN NEW TAB.

The status of the tabs is stored for the last 10 projects and restored with the next project selection.

### **1.2.2 Simple Cut & Paste of Drawings**

With ELCAD / AUCOPLAN 7.4.0, you can easily start the graphic copying function by pressing the left mouse button and drawing a window while keeping the mouse button pressed.

Upon releasing the mouse button, a pop-up window appears with the menu items COPY GRAPHIC SECTION and CUT GRAPHIC SECTION. Subsequently the reference point for the graphic section is queried, and the data is copied into the clipboard.

If you have opened several drawings in different tabs, you can now copy the graphic section to other drawings by simply changing the tab.

A click on right mouse button in the free area of a drawing creates a pop-up window with the function PASTE GRAPHIC SECTION. The insertion point is queried, and the data from the clipboard is inserted.

### 1.2.3 Placing Symbols and Connections in Drawings

In version 7.4.0, graphics editing has been considerably simplified with respect to the placement of symbols and connections.

#### Placing connections

When drawing connections, possible connector points for symbols are now marked, and the prioritized connector point is specially highlighted. You do not have to directly "hit" the connector points to establish the connection. Moreover connections are now automatically bent if one moves the cross-hair pointer more than three grid steps (default value) away from the coordinate of the origin.

#### Connection preview when placing symbols

When placing symbols while the symbol is attached to the cursor, the program calculates the possible connections for the surrounding drawing online and displays them graphically. You can arbitrarily change the desired connection direction before the placement and then place the symbol together with the connections in one step.

### 1.2.4 Checking Symbols During Drag & Drop

When you drag a part from the component tree onto a symbol in a drawing, the system checks whether the part has been assigned the name of the symbol to which it is to be assigned. This check is carried out only for the assignment of the first part number.

If more than one matching symbol is specified, a list of the possible symbols is offered. A graphic preview is also possible

If the check shows that the name has not been assigned to the part, a corresponding note is displayed; the symbols are not automatically exchanged, however. Nonetheless you can assign the desired part to the symbol.

### 1.2.5 Colour PDF and Printer Output

With ELCAD / AUCOPLAN 7.4.0, you can now output coloured drawings to Windows printers (GDI) and as PDF files.

You find the required settings in the menu `OPTIONS / USER / OUTPUT / COLOUR SETTINGS`

The former possibility to colour print in special cases by means of a manually adjusted printer 1000 is no longer available due to this solution.

## 1.2.6 Multiple Representation of Cores and Wires

If you use cores and wires in connections spanning several sheets, you can from version 7.4.0 onward display the data (e.g. core designation or colour) on both sheets.

For this purpose the connections spanning several sheets must be constructed by means of connector symbols (symbol type 3) or potentials (symbol type 14).

## 1.2.7 Working Status and Connector Status Always Visible

To recognize a connector type (e.g. internal or external) and the working status of symbols and connections, it used to be necessary to call an appropriate test function. With version 7.4.0 it is now possible to permanently visualize this information in the drawing.

In the drawing management, you find in the menu EXTRAS / TEST DRAWING.../ CONFIGURATION... a setting for the permanent display of markings. The settings you specify here are stored in the aucotec.ini. Modifications of the connection status as well as the working status of symbols or connections can thus immediately be visualized in the drawing.

## 1.2.8 Creating a Part While Editing a Symbol

Once a symbol has been edited, there is a check whether all part numbers specified in the symbol editing mask are already present in the predefined master database.

If this is not the case, there is a query whether the new part number is to be entered in the device master database

This function observes the write rights for master data and can be deactivated.

## 1.2.9 Part Assignment via Drag & Drop with More than One Part Number

If you wish to assign parts to symbols via drag & drop yet the symbols contain more than one part number, then you can obtain the assignment for this function with the CTRL key pressed. You can then select to which one of the part numbers in the symbol the part is to be assigned.

## 1.3 New Features in Project Editing

### 1.3.1 Strip-Related Terminal Diagrams Without Batch Processing

With ELCAD / AUCOPLAN 7.4.0, you can now, in addition to the former terminal diagram creation by batch operation, also create online terminal diagrams in relation to a project, a folder, a location or a strip. You can start this terminal diagram creation method in the item editor on terminal strips or higher-level folders.

You can regenerate these online terminal diagrams without carrying out a complete wiring run. Moreover you can regenerate the terminal diagrams automatically before opening them. You can use the Sort out function to delete terminal diagrams of no longer existing strips.

The creation and storage of the terminal diagrams is configured in the options. Essential for the configuration is the unambiguous assignment of the sheet designation to the strip designation because otherwise the program can no longer find the terminal diagrams belonging to a strip.

### 1.3.2 Exempting Code Numbers from the Revision

Starting with version 7.4.0, the code numbers to be exempted from the revision can be stored in the revision settings. You can now enter up to 100 code numbers whose change does from now on not trigger a revision.

### 1.3.3 Search for the Appropriate Master Data Record

When editing symbols of the types 1, 5 or 6, you can now search for parts intended for the symbol (for the first part number) **also in the master data**. This function is available only if you work with a master database.

A second selection button [...] then appears after the first part number enabling a search for appropriate parts. For this selection, only parts are offered that fit the symbol, i.e. parts that contain this symbol name in the master data.

### 1.3.4 Free Code Number for Creating Cross-References

With ELCAD / AUCOPLAN it is now possible to use two arbitrary code numbers in the range 1 ... 32767 for configuring the cross-reference setup.

When using an arbitrary code number in the cross-reference, you should by all means specify a separator in the settings. When you change the cross-reference setup, the respective project is reconstructed and evaluated anew.

### 1.3.5 Expanded Translate Text Selection

When entering a text in a mask or an input field, a question mark can be used to start the selection of a translate text. Up to now this was restricted to the selection of a text from a translate file per input field.

In version 7.4.0, the selection options for translate texts have been expanded. By putting together translate references, separated by blanks, it is now possible to select several translate texts from several language files.

### 1.3.6 Extension of the Replace Function for Texts

The Replace function for texts has been expanded. You can now also

- use it to exchange free texts
- empty any existing texts
- specify the symbol type for the exchange

After the exchange the new texts become manual entries. The entries in the dialog are stored and offered as default values at the next activation.

### 1.3.7 Specifying Data for Sub-Devices in a Symbol

With ELCAD / AUCOPLAN 7.4.0 you can now use a symbol to specify the technical / commercial attributes of several sub-devices or accessories and transmit them to the parts lists. Up to now this was restricted per part number to the code numbers 149 (quantity) and 3050 .. 3059 (supplementary information per part).

You can now use additional code numbers for each part number (code 80), whose contents if occurring repeatedly are adopted for the respective part number in the parts list. For code numbers 1 .. 8 and several other system code numbers this function is not available.

The number of part numbers (code 80) and the attributes used several times in the symbol must be identical.

#### **Example:**

The first nominal current (code 31) for the first part number is transmitted from the symbol to the parts list.

The second nominal current (code 31) for the second part number is transmitted from the symbol to the parts list.

### 1.3.8 Deletion of Existing Graphic Lists before Renewed Creation

Before exporting lists to drawings, you can now delete all existing drawings in the drawing folder. When you create a smaller number of documents than before, you do not have to manually remove the old lists.

To activate this function, set the following switch in the aucotec.ini:

```
[ExportSheets]
```

```
DeleteOldSheets=
```

If 0 is entered here, the drawings are not automatically deleted.

If 1 is entered here, the old drawings are automatically deleted.

### 1.3.9 Archiving with Several Databases

For archiving projects, the switchover between different databases by means of daba\_cfg is now taken into account.

The parts are now initially searched for in the predefined database and then in the databases specified by means of the daba\_cfg settings in order to adjust the technical data.

### 1.3.10 Expanded Functions for Form Editing

In the Drawing tree, starting with version 7.4.0 new functions are available with a right mouse click on drawing objects.

- EDIT FORM
- EXCHANGE FORM
- EDIT SYMBOL (only with open drawing)

The graphics editing menu for forms has been extended by the menu item EDIT / FORM / EDIT SYMBOL.

## 1.4 New Features in Terminal Editing

### 1.4.1 Terminal Editing from the Terminal Diagram

With version 7.4.0 you can now use the graphic editing mode of a terminal diagram as the starting point to edit a terminal or strip symbol by a right mouse click on the respective symbol, to start the terminal editor and to move to the selected terminal in the item editor.

If the strip or terminal is no longer evaluated, i.e. the terminal diagram would not be up to date, an error message is issued.

### 1.4.2 Freezing Terminals in the Item Editor

In the item editor you can now freeze all terminals of a project, of a location or of an individual strip in one step. The frozen terminals are shown in blue in the item editor. Freezing prevents retrospective changes by automatic number assignment.

After the action you receive a message

- whether and if so how many strips and terminals have been frozen
- that strips were already frozen
- that strips are currently being edited by another user (e.g. in the terminal editor)

### 1.4.3 Extended Standard Strip Editing Options

Whether or not a terminal strip is a standard strip can from version 7.4.0 on be set or cancelled in the terminal editor. Depending on this some functions of the terminal editor are either blocked or enabled.

In the item editor, the terminal strips and all higher-order folders contain the menu item DELETE STANDARD STRIP (called with the right mouse button). Only unused standard strips can be deleted.

After the deletion you receive a message

- whether and if so how many strips have been deleted
- that strips could not be deleted
- that strips are currently being edited by another user

Moreover you can use the menu item SAVE AS in the item editor to copy standard strips.

### **1.4.4 Different Terminal Types in a Terminal Diagram**

Up to now only two different terminal symbols could be used in a terminal diagram. However, this could not be used to display terminals belonging to the same block.

In version 7.4.0 a third terminal symbol is now used, one that at the end of a terminal signals that the next terminal belongs to the same block.

To achieve correct sorting of the terminals for representation in the terminal diagram according to block and level, the terminal sorting options were extended.

### **1.4.5 Cross-References between Terminals Displayed Separately**

The unity of terminals which are displayed separately and wired can be traced in a printed documentation only if there are cross-references between the individual partial representations. This is supported from version 7.4.0 on.

For this purpose add a dialog for the cross-reference (code 10) of the connector to each terminal connector (code 45). The cross-references are created even if the symbol has not yet been evaluated.

## 1.5 New Features in Cable Editing

### 1.5.1 Freezing Cores

By analogy to the freezing of terminals, starting with version 7.4.0 cores can also be frozen if the core information was assigned automatically.

Thus the core information is not assigned anew by the program following a possible reset.

For this purpose you find the menu items FREEZE CORES and UNFREEZE CORES in the item editor under Cables. On the Cable folder you can edit all cores of all cables simultaneously. Cores can also be frozen and unfrozen via the menu items EXTRAS / AUTOMATIC in the working area Cable Management.

After freezing, the cores have the status "assigned manually" and are shown in blue in the item editor.

### 1.5.2 Cross-Section Doubling by Means of Parallel Cores

With the cable management being switched on, you can now increase the cross section by using two parallel cores between two connector points.

### 1.5.3 Navigation from Cable Planning

With version 7.4.0, you can switch over from the core in cable planning directly to the drawing with the core representation. At the same time the associated cable is opened in the item editor.

For this purpose use a click with the right mouse button on a core in the core table to call the menu item PROPERTIES. By means of this new menu item the drawing where the core is located is opened for editing, and the core is marked.

### 1.5.4 Inserting Cables from the Database

With ELCAD / AUCOPLAN 7.4.0 you can now use the menu items INSERT / PART in the graphics editing mode to also insert cables from the master data.

You can specify the core symbols to be used for the cables in EXTRAS OPTIONS / DRAWING / STANDARD SYMBOLS.

Upon actuating the menu item, the cable is selected from the predefined cable database. Once you have selected a cable, the preset core symbols are offered for placement. Using the mouse wheel while simultaneously pressing and holding the Ctrl and Alt keys, you can switch between the specified core symbols. The core symbol gets the default values from the cable master data when placed.

## 1.6 New Features in List Editing

### 1.6.1 Device-Related List of Connectors

With ELCAD / AUCOPLAN 7.4.0, you can specify during the generation of a wiring list whether you want to use it later on for a device-related list of connectors.

During the preparation of a device-related list of connectors, the program processes the structure of the wiring list with swapped connector destination information in duplicate. Thus it is ensured that all connectors of a device are available on the right as well as on the left side of the connection.

With an appropriate output format it is possible to output a device-related list of connectors from these wiring lists as a list or a drawing.

### 1.6.2 Keeping Manual Entries When Regenerating Contents Tables

In contents tables, manual entries can be kept with version 7.4.0. For this purpose the structure of the contents table is automatically extended by the code number 5000 and filled. Data records not containing contents in code number 5000 are maintained during regeneration.

For contents tables not yet containing the code number 5000 you get a program message during regeneration telling you that additional entries may be lost.

To ensure that code number 5000 is empty following manual list editing it is explicitly emptied.

## 1.7 New Features in ELCAD*maintenance*

### 1.7.1 Navigation to Graphic Representations Extended

When you open AUCOPLAN projects with ELCAD*maintenance*, you will from version 7.4.0 on find extended menus for the following PM&CT objects when actuating the right mouse button.

- Measurement tags
- Instrumentation
- Signals
- Function groups

Thereby you can navigate to the representations of the objects.

## **1.8 New Interface Features**

### **1.8.1 EDM Operation for ELCADmaintenance and ELCADview Extended**

With version 7.4.0, the following EDM actions are supported when operating simultaneously with an ELCADmaintenance license.

- GUEST
- WORK
- VIEW
- SHEET
- SHEETLEVEL

For all other actions, the EDM functionality is immediately terminated and an ActionError is recorded in the log file. The called action is entered with a note.

### **1.8.2 Extension of the DXF / DWG Output**

When outputting ELCAD / AUCOPLAN drawings in the DXF or DWG format, now the images (logos etc.) inserted into the drawings and symbols are also output.

To restrict the file size, a maximum size is specified for the images to be output. The size can be adjusted via an ini switch, however.

### **1.8.3 DWG Output Extended to AutoCAD 2005, 2006 and 2007**

With version 7.4.0, the graphic output in the DWG format has been extended by the versions AutoCAD 2005, 2006 and 2007.

### **1.8.4 Support for Weidmüller M-Print Pro**

For users who print their terminal data with the software package M-Print Pro from Weidmüller, ELCAD / AUCOPLAN 7.4.0 not only offers the interface but can moreover start the M-Print Pro program, select the label material in the program and directly start the printer output.

### **1.8.5 Exporting Several Strip Layouts At Once**

With version 7.4.0 it is now possible to simultaneously export the layout of several terminal strips to the strip layout systems from Phoenix, Wago and Weidmüller.

To do this, mark several terminal strips in the item editor, then select EXPORT STRIP LAYOUT in the context menu of the right mouse button.

In order to support several strips with the Weidmüller RailDesigner, this output has been converted to the current XML format.

### **1.8.6 Individual Specification of a Compact PDF File**

When creating a compact PDF file from the Drawing tree, you can from version 7.4.0 on individually specify the folder and the file designation without being bound to the settings for the PDF output.