

# New Features in Version ELCAD/AUCOPLAN 2021

October 2020

**AUCOTEC AG**

Hannoversche Str. 105  
D- 30916 Isernhagen

Phone: +49 (0)511 61 03-0  
Fax: +49 (0)511 61 40 74

[www.aucotec.com](http://www.aucotec.com)

**AUCOTEC, INC.**

2701 Troy Center Drive,  
Suite 440  
Troy, MI 48084

Phone: +1 630 485 5600  
Fax: +1 248 655 7800

**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.

**Trademarks:** AUCOPLAN® and ELCAD® are registered trademarks of AUCOTEC AG, Germany. Windows® is a registered trade mark of Microsoft Corporation, USA.

## Content

<b>1</b>	<b>New Features of ELCAD/AUCOPLAN Version 2021 .....</b>	<b>1</b>
<b>1.1</b>	<b>General Innovations .....</b>	<b>2</b>
1.1.1	Defining the maximum length of revision comments .....	2
1.1.2	Resetting all layers to the standard layer in the object tree .....	3
1.1.3	Optimized menu item Project Check .....	3
<b>1.2</b>	<b>New in Project Editing .....</b>	<b>5</b>
1.2.1	Archiving strip and plug master data of standard strips .....	5
1.2.2	Keeping the marking and revision information on copying drawings .....	6
<b>1.3</b>	<b>New in Drawing Editing .....</b>	<b>7</b>
1.3.1	Edit Form Immediately in the Change of Sheet dialog .....	7
1.3.2	New code numbers for the designations of the previous and the subsequent sheet .....	8
<b>1.4</b>	<b>New in Graphics Outputs .....</b>	<b>9</b>
1.4.1	Output of additional information to compact PDF files.....	9
<b>1.5</b>	<b>New in Item Editor .....</b>	<b>12</b>
1.5.1	Immediately editing strip data and defining standard strips.....	12
1.5.2	Assigning several cores at the same time .....	13
<b>1.6</b>	<b>New in List Editing .....</b>	<b>14</b>
1.6.1	New structure for the identification of standard strips.....	14
1.6.2	Checks Missing Symbols and Error Symbol Evaluation .....	15
1.6.3	Displaying the configuration of a list .....	18
1.6.4	Creating an overall parts list by using a function code .....	19
<b>1.7</b>	<b>New in Cable Planning .....</b>	<b>20</b>
1.7.1	Optimized control elements in the cable planning .....	20
1.7.2	Automatic update of cable designators .....	20
1.7.3	Status list Working Status: Display of duplicate terminal numbers .....	21
<b>1.8</b>	<b>New in Batch Processing.....</b>	<b>22</b>
1.8.1	Displaying plain texts instead of translate references in batches .....	22
1.8.2	Extended batch function "Create New Revision of Drawings" .....	22
1.8.3	Extended batch function "Equate Revisions of Drawings" .....	24

# **1 New Features of ELCAD/AUCOPLAN Version 2021**

With ELCAD/AUCOPLAN Version 2021, AUCOTEC continues the ongoing evolution of the past years.

ELCAD/AUCOPLAN has again significantly improved the way you can edit projects. For instance, there are new functions to check projects, advanced "Working Status" status lists and the advanced drawing management of previous and subsequent folders.

In ELCAD/AUCOPLAN 2021, we have incorporated a total of more than 90 enhancements that have been derived from your requirements and requests.

As in all our extensions, our focus has again been on the compatibility of your data and on further simplifications to the handling.

Isernhagen, October 2020

## 1.1 General Innovations

### 1.1.1 Defining the maximum length of revision comments

So far, revision comments could have a maximum length of 128 characters. The length of texts at symbols representing these comments was restricted to a maximum number of 58 characters.

As of this version of ELCAD, the length of the revision comments is restricted to 58 characters. Additionally, you can define the individual lengths of the mandatory comment and the additional comments under **Revisions/Designator Structure** in section **User/Project Options**. Values between 1 and 58 characters are permitted.

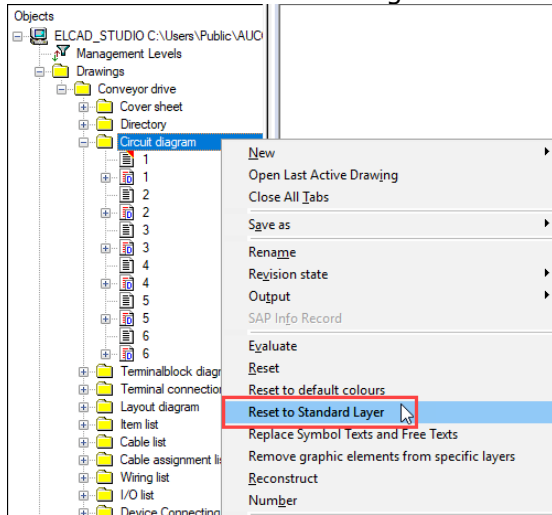
The screenshot shows the 'User/Project Options' dialog box with the 'Project' tab selected. The 'Designator Structure' section is active, and the 'Designation of the Additional Revision Comments' sub-section is highlighted with a red border. This sub-section contains a table with columns for the comment name and its maximum length. The 'Mandatory Comment' and six 'Additional Comment' rows are all set to a maximum length of 58 characters.

	Maximum Length
Mandatory Comment	58
Name of the Additional Comment 1	58
Name of the Additional Comment 2	58
Name of the Additional Comment 3	58
Name of the Additional Comment 4	58
Name of the Additional Comment 5	58
Name of the Additional Comment 6	58

### 1.1.2 Resetting all layers to the standard layer in the object tree

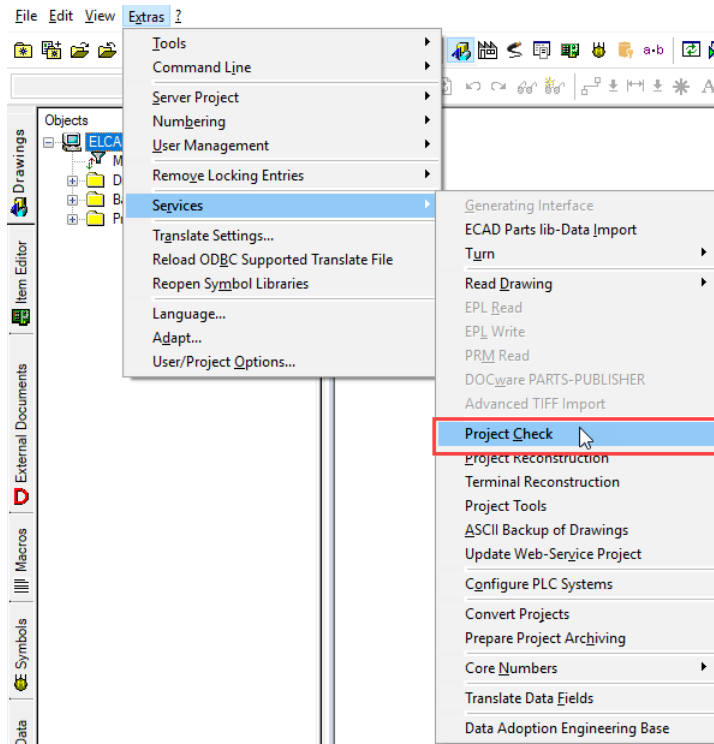
As of this version of ELCAD, you can reset the contents of all layers on the drawings to standard layer "0" via the shortcut menu of the object tree.

Click **Reset to Standard Layer** on the shortcut menu of the superordinate folder to reset all contents of all drawings to standard layer "0".

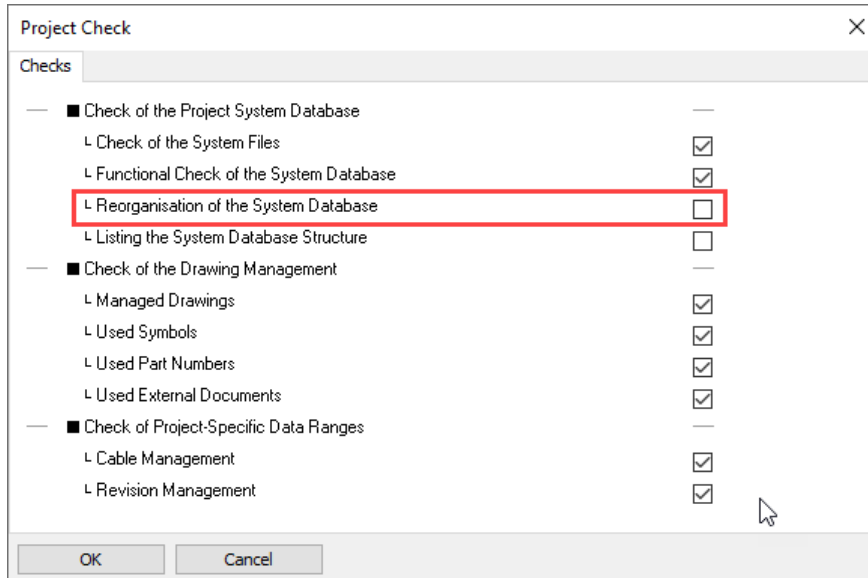


### 1.1.3 Optimized menu item Project Check

As of this version of ELCAD, menu item **Project Check** is available under **Extras/Services**. The two previous menu items **Project Check** and **Recover Database** have been combined in this new menu item.



1. Select the required options for the project check.



- The functionality of the previous **Recover Database** option is now available under-**Check of the Project System Database/Reorganisation of the System Database**.
  - As of this version of ELCAD, the additional option **Listing the System Database Structure** is available.
2. Click **OK** to start the project check.

#### To start the reorganization of the system database via the EDM interface

Additionally, a new switch is available which you can use to start the reorganization of the system database via the EDM interface. To do so, enter the following switch into instruction file **edm2elc.ini**:

```
[ProCheckOption]
CheckAll=1
Recover=1
```

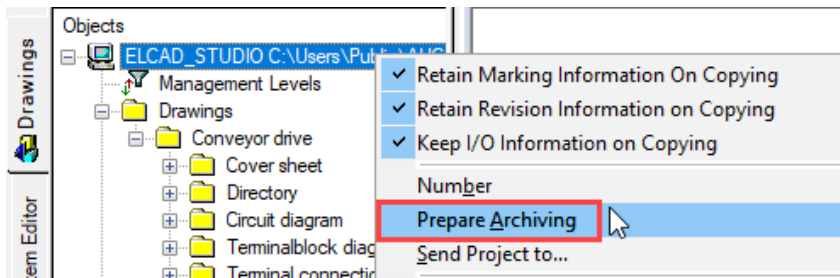
## 1.2 New in Project Editing

### 1.2.1 Archiving strip and plug master data of standard strips

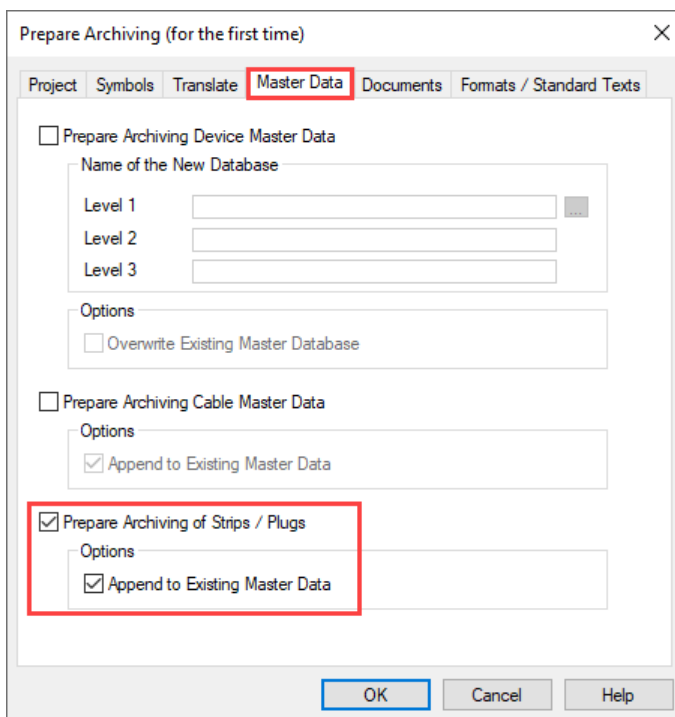
As of this version of ELCAD, a new option to archive standard strip data is available.

#### To prepare the archiving of strip and plug master data

1. Click **Prepare Archiving** in the object tree on the shortcut menu of the project.



2. Switch to the **Master Data** tab.



3. Activate option **Prepare Archiving of Strips / Plugs** if you want to include your configured strip / plug master database in the project archiving.
4. Additionally activate option **Append to Existing Master Data** if you want to append the terminal block data to already existing project master data during further archiving actions.

#### To switch off the archiving of strip / plug master data via the EDM interface

Additionally, a new switch is available which you can use to switch off the archiving of strip / plug master data via the EDM interface. To do so, enter the following switch into instruction file **edm2elc.ini**:

```
[Archiv]
NoMasterStrip=1
```



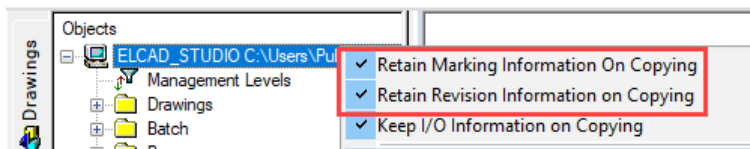
### 1.2.2 Keeping the marking and revision information on copying drawings

As of this version of ELCAD, you can define in the "Drawings" object tree whether ELCAD is to keep the marking and revision information on copying drawings. The new function is available on copying drawings within your project or from a template project.

By default, copying the information is activated.

#### To select the information to be copied

1. In the drawing editing, select the "Drawings" object tree.
2. On the shortcut menu of the object tree root, select option **Retain Marking Information On Copying** to keep the existing marking information on copying the drawings. Deactivate this option if you do not want to copy the marking information on copying the drawings.
3. On the shortcut menu of the object tree root, select option **Retain Revision Information On Copying** to keep the existing revision states on copying the drawings. Deactivate this option if you do not want to copy the revision information on copying the drawings.



#### Prerequisites for copying revision information

- Activate the Revision Management in the source project and in the target project.
- Your working project is the target project for your copied drawings.

## 1.3 New in Drawing Editing

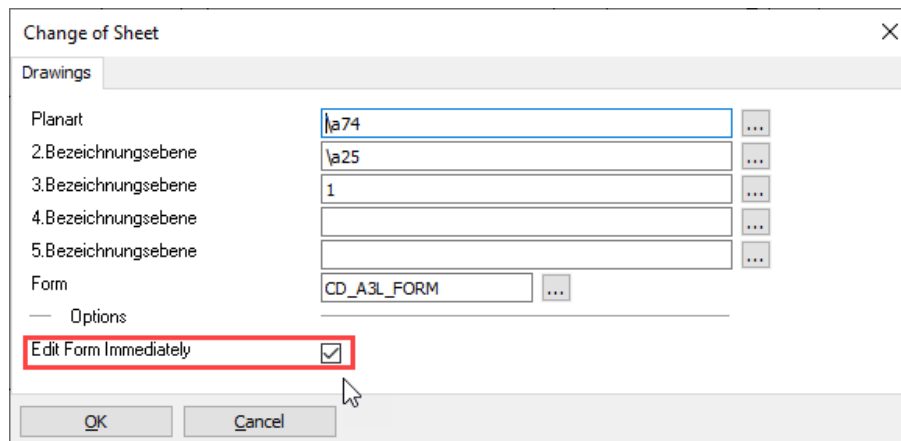
### 1.3.1 Edit Form Immediately in the Change of Sheet dialog

As of this version of ELCAD, a new option **Edit Form Immediately** is available in the **Change of Sheet** dialog.

- Using this option, you can edit the form entries of the sheet in an easy and comfortable way before you change to a new sheet.
- ELCAD then opens the new sheet directly with the changed data.
- That way, you do no longer have to call the editing dialog of the form separately.

#### To immediately edit the form before a change of sheets

1. Activate option **Edit Form Immediately** in the **Change of Sheet** dialog.



2. Confirm the dialog by clicking **OK**.

### 1.3.2 New code numbers for the designations of the previous and the subsequent sheet

As of this version of ELCAD, the following new code numbers are available for the use in forms:

- Code number **87** for the designation of the previous sheet
- Code number **88** for the designation of the subsequent sheet.

ELCAD composes the contents of the new code numbers of the superordinate folder designation in the "Drawings" object tree and the sheet designation (sheet number).

If required, add these codes to your form and select the position on the form where you want to display the information on the drawing.

FORM :CD_A3L_FORM	
Sheet Naming Graphics Symbol Info	
Release	2021 alpha
Scaling: Scale	1:1
Unit	mm
Scaling	1
Ext.Macro	EAS:
Page Format	A3L
Comm. Page Format	297 x 420mm
Macro Worksheet	
Revision Designator	0
Date of Revision	21.10.2020
Internal Filename	000235
Release Symbol	7.3.1
Version Symbol	1.0
Level forwards	\\a22/2
Level backwards	\\a25/2

Example: Form with code numbers 87 and 88 included

### To skip the strip designators at terminal block diagrams

In terminal block diagrams, the folder above the sheet often contains the designation of the terminal block. If you want to display the superordinate folder instead of the strip designator, e.g. **Terminal Block Diagram**, enter the following switch into file **project.ini**:

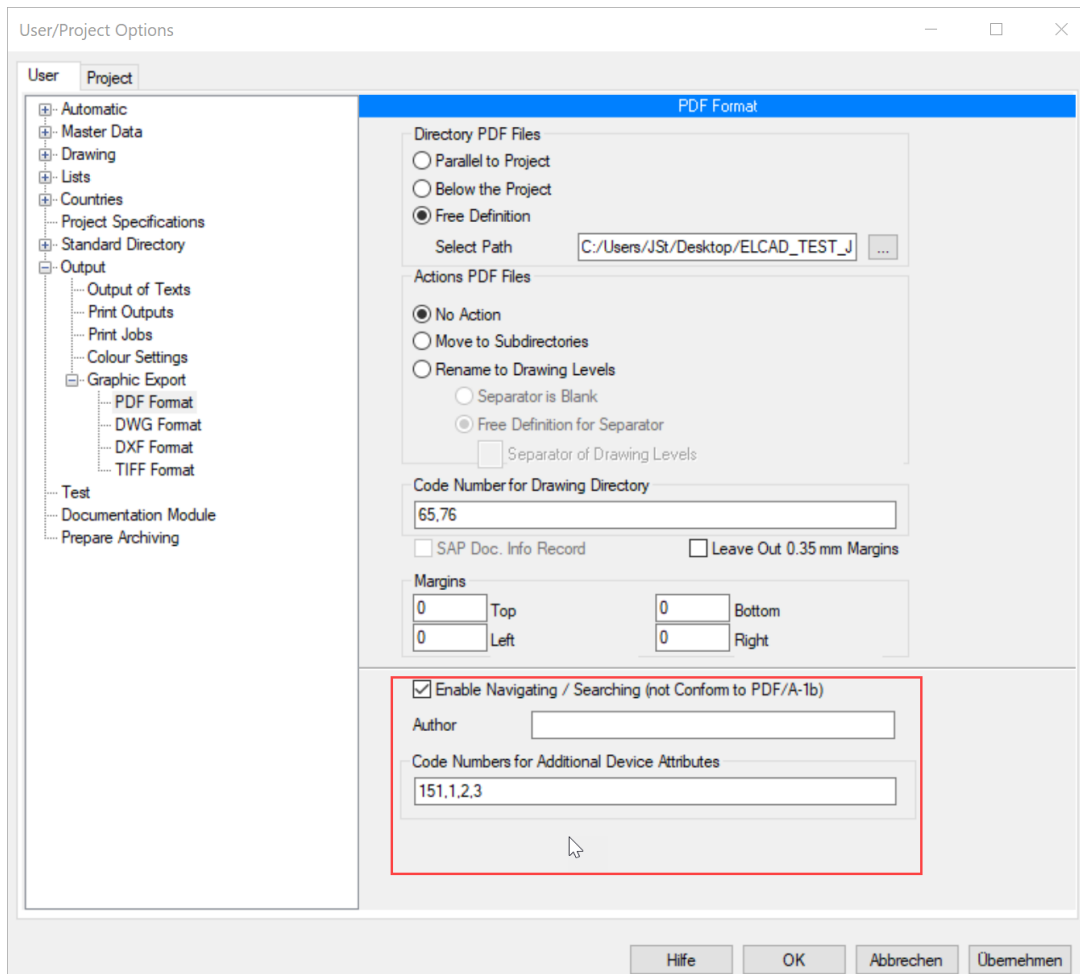
```
[Automatic]
NoStripNameToLastLevelDesignation=1
```

## 1.4 New in Graphics Outputs

### 1.4.1 Output of additional information to compact PDF files

As of this version of ELCAD, you can output additional item information saved by ELCAD to a compact PDF file, if required. Material information, locations or spare part numbers are examples of this additional information.

#### Output/PDF Format/Code Numbers



If you enable option **Enable Navigation and Searching** in this dialog, the additional information is available every time you output compact PDF files by a batch action.

Enter up to ten **Code Numbers for Additional Device Attributes** that you want to display as additional information in the PDF. In this example, the code numbers for Manufacturer (151), Designation Blocks Plant (1), Function (2) and Location (3) have been inserted.

A prerequisite for the output of the information is that the symbols are evaluated. Click **Evaluate** on the shortcut menu of the required drawing folder in the navigation pane.

## Shortcut menu in the objects window/Save as/Compact PDF file

Dateiname:

Dateityp:

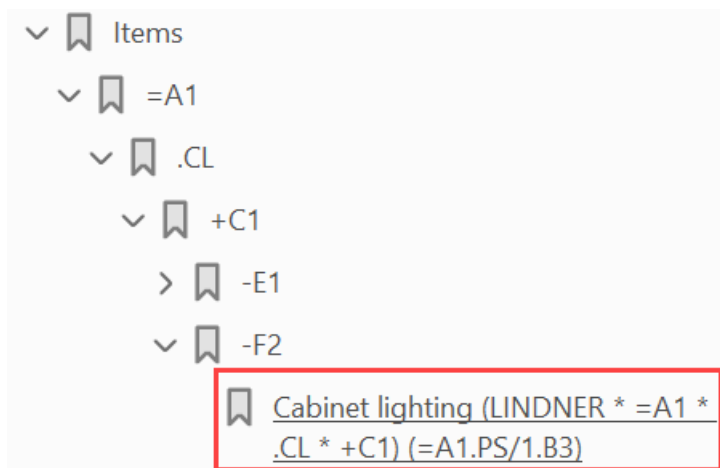
☒ Open PDF File Immediately

☒ Enable Navigation and Searching

☐ Set Water Mark

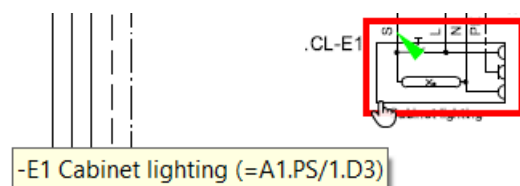
Activate option **Enable Navigation and Searching** to output the additional information in the compact PDF file. This option enables you to navigate from the items in the PDF to the respective representations, for instance to the cabinet layout or to the devices on the connector diagram.

## Information on an item in the PDF tree structure



The references to the drawings on which the item can be found are visible below the item.

## Information on an item via tooltip on the drawing



The additional item information is included on the drawing as tooltips. The following abbreviations represent the references to the diagram types:

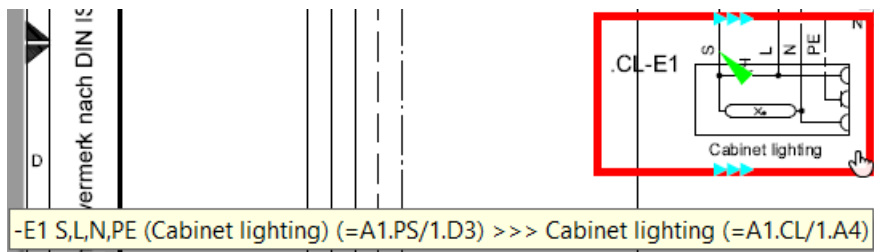
## Diagram types

- • CD >>> Circuit diagram, here: Cabinet lamp
- • TD >>> Terminal block diagram
- • TC >>> Connector diagram
- • LA >>> Layout plan

## Long texts for references to diagram types

Set the following switch in file **aucotec.ini** to use the long text of the current user interface language for the references instead of the abbreviations:

**UseLongPlanTypeName=1**



## Using your own designations

If you want to use your own designations for the diagram types instead of the abbreviations, you can define them manually in file **aucotec.ini**.

You can specify the designations as translate references to use them for multilingual exports. To do so, enter the translate references into the following section of file

**aucotec.ini**:

```
[ViewerTreeInfo]
SetExtSheetInfos=0
CD=\a25
TD=\a26
TC=\a27
LA=\a28
UseLongPlanTypeName=1
```

Alternatively, you can assign fixed character strings:

```
[ViewerTreeInfo]
SetExtSheetInfos=0
CD=Circuit plan
LA=Mounting plan I
```

## Not showing the additional information in compact PDF files

If you want to use the output of compact PDF files as before, i.e. without the additional information, enter the following switch into file **aucotec.ini**:

```
[Pdf]
OldIntelligentMode=1
```

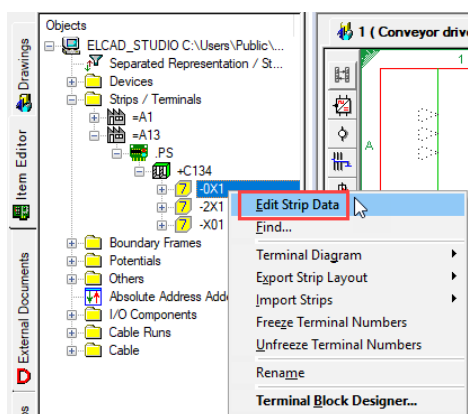
## 1.5 New in Item Editor

### 1.5.1 Immediately editing strip data and defining standard strips

As of this version of ELCAD, you can edit the strip data of a terminal block directly in the Item Editor. Moreover, option **Standard Strip** is now available in the **Input of strip data** dialog.

#### To edit the strip data directly in the Item Editor

1. In the Item Editor, select **Edit Strip Data** on the shortcut menu of a standard strip.



2. Make the required settings in the **Input of strip data** dialog. Check the **Standard Strip** checkbox if you want to define the terminal block to be a standard strip.

 The screenshot shows the 'Input of strip data' dialog box. It has a 'Properties' tab and contains two sections: 'Strip Data' and 'Terminal\_Diagram'. In the 'Strip Data' section, there are fields for 'Part number 1' (ABB\_07Di92), 'Part number 2' (WAG\_750-550), 'Comment', and 'Rail Length' (2). The 'Standard Strip' checkbox is checked and highlighted with a red rectangle. In the 'Terminal\_Diagram' section, there is a field for 'Terminal diag. form' (CD\_A3L\_FORM), a 'do not create' checkbox, and a 'Max. jumper length' field (0). At the bottom are 'OK' and 'Cancel' buttons.

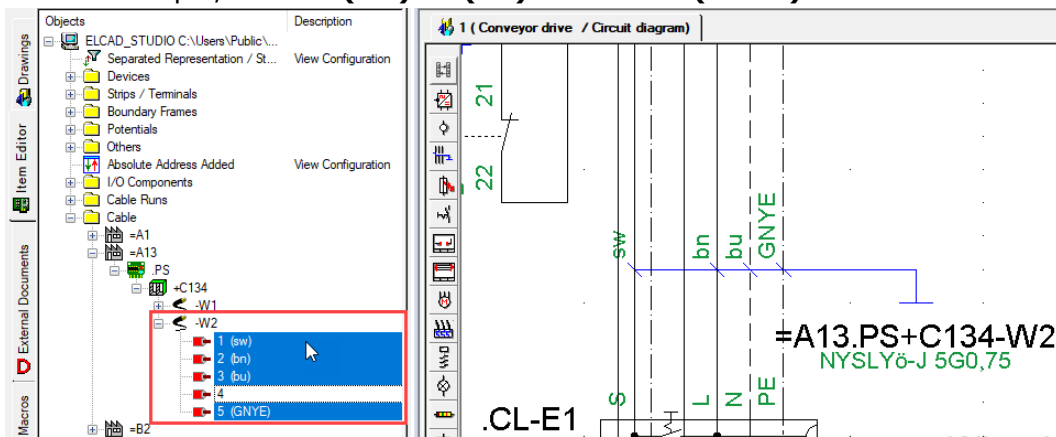
### 1.5.2 Assigning several cores at the same time

As of this version of ELCAD, you can select several cores of a cable at the same time in the Item Editor to assign them to the symbols on the drawing one after the other. You can place cores onto a core symbol or onto a connection on the drawing.

- A precondition for you to be able to select several cores at the same time is that you have not assigned these cores yet.

#### To assign several cores at the same time on a drawing

1. In the Item Editor, select the unassigned cores individually or by a multiple selection using the left mouse key.
- To select several cores at the same time, click one core after the other while keeping the Shift or Ctrl key pressed.
- In this example, cores **1 (sw)**...**3 (bu)** and core **4 (GNYE)** are selected.



2. Use drag-and-drop, i.e. drag the selected cores from the Item Editor onto the required symbol on the drawing while keeping the left mouse key pressed.

In the status line, you can see which target symbols are suitable for the placement of the cores:

**Core Found**

ELCAD highlights the target symbol in purple.

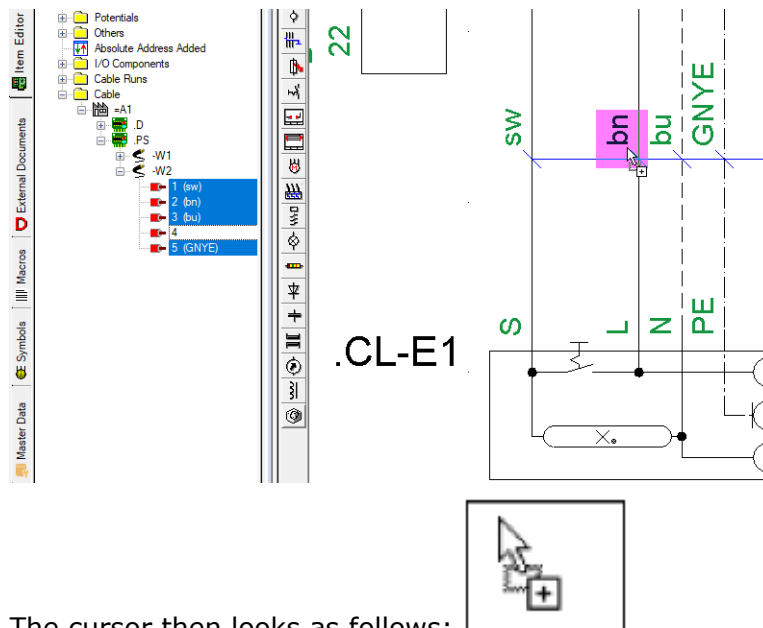
The cores are assigned in the sequence as they are represented in the Item Editor. In this example, the first core is **1 (sw)**.

As of the second core, the status line shows which core you place next:

Placement of the Next Marked Core >> =A13.PS+C134-W2 : 2



- Place the cross-hairs onto the required symbol.



The cursor then looks as follows:

- Release the left mouse key on the required symbol to place the core onto the target symbol. The core is then associated with the core symbol.
- Repeat this process to place all further cores selected in the Item Editor onto the drawing.

If you wish to abort the process, press the ESC key.

## 1.6 New in List Editing

### 1.6.1 New structure for the identification of standard strips

As of this version of ELCAD, code number **8100** is available. It indicates whether a specific terminal block is a standard strip. If the terminal block is a standard strip, ELCAD sets a "1" in code **8100**. The precondition to define a standard strip is that you activate checkbox **Standard Strip** in the Terminal Block Designer.

#### View/List Editing

Strip_Designator	Comment	Start_Terminal	End_Terminal	Number_PE	Number_N	Total	Terminal_Diagram	Standard_Strip
=A1.D+C1-1X1		1	4	0	0	4	Bandantrieb  Klemmenplan  =	
=A1.D+C1-3X1		2	9	0	0	9	Bandantrieb  Klemmenplan  =	
=A1.D+C1-3X2		1	L-	0	0	12	Bandantrieb  Klemmenplan  =	
=A1.ES+C1-3X2		1	8	0	0	8	Bandantrieb  Klemmenplan  =	
=A13.PS+C134-0X1		1	5	0	0	5		1
=A13.PS+C134-2X1		1	6	0	0	6		

If the respective terminal block is not a standard strip, field **Standard Strip** remains empty on generating the list. You can display this information both for new terminal lists and for already existing terminal lists.

#### Extras/Structure/Extend

Insert code number **8100** into the structure of the terminal list in case you require that information.

Edit structure

<div> <div>Enter</div> <div>Change</div> <div>Delete</div> <div>Up</div> <div>Down</div> <div>S-key</div> <div>Print</div> <div>Accept</div> <div>Close</div> </div>							
	No.	Designation	Code	Lgth.	Typ	Dec.places	Upc
1	1	Standard_strip	8100	6	0	0	0
2	2	Part_of_object_list	8006	6	0	0	0
3	3	Plant	1	16	0	0	0

- Together with the already existing code **8006** (terminal is part of a parts list) you can, for instance, display the number of terminals to be ordered.
- If you have added codes **8006** and **8100** to the structure of an overall parts list, you can determine the actual number of terminals to be ordered by using filters. To do so, use the following filter string:

Terminal List [KLEMMENLISTE]

Options

List comment: Terminal list

Convert reference texts to plain texts: ☐

Compare with Database: ☒

Filter String: `Part_of_object_list = "1" and (Standard_strip = "0"`

Name: Only show terminals to be ordered

OK Cancel

## 1.6.2 Checks Missing Symbols and Error Symbol Evaluation

As of this version of ELCAD, the checks **Symbols/Missing Symbols** and **Evaluation/Error Symbol Evaluation** are available as options in status list Working Status.

A precondition for option **Error Symbol Evaluation** is that you activate option **Automatic/Evaluate/Error List**.

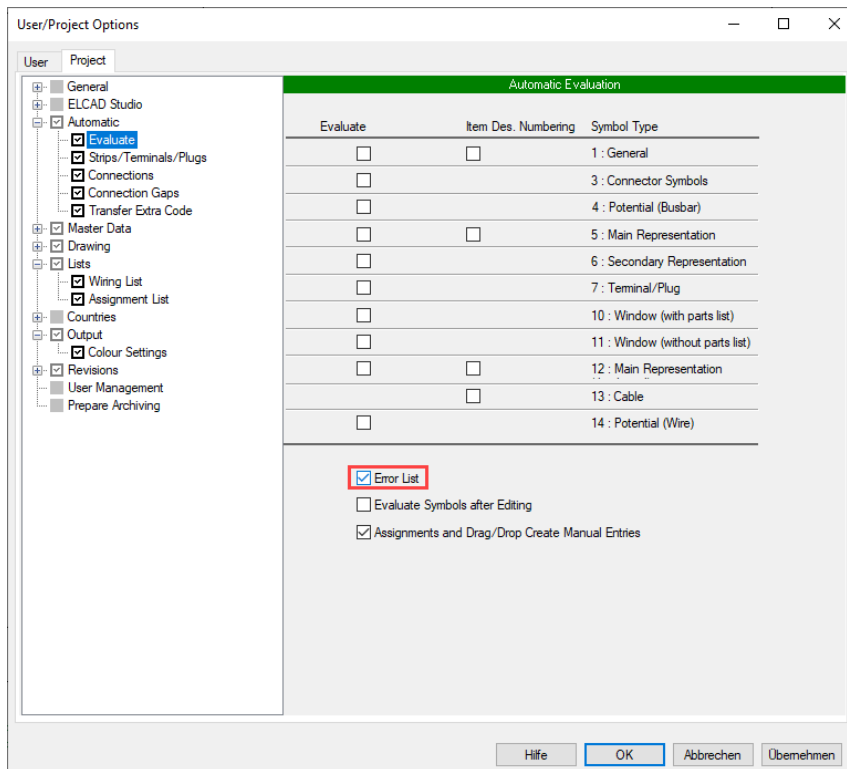
A precondition for this evaluation is that you have re-evaluated the drawings beforehand.

- If you activate option **Missing Symbols**, you can identify missing symbols in your project and list symbols that should be there, for instance via calling a macro.
- If you activate option **Error Symbol Evaluation**, you can display all registered evaluation errors.

### To include the checks in status list Working Status

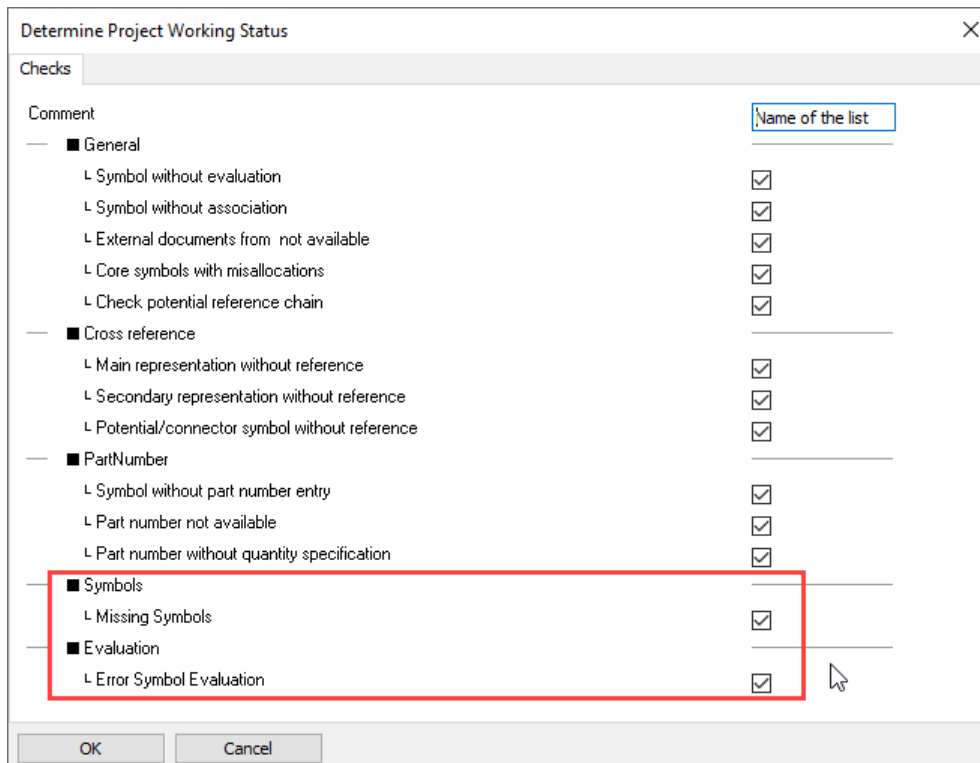
1. Activate option **Error List** in the user or project options under **Automatic/Evaluate**.
2. Activate **Missing Symbols** in the status list.
3. Activate **Error Symbol Evaluation** in the status list.

Regenerate **the existing lists**.



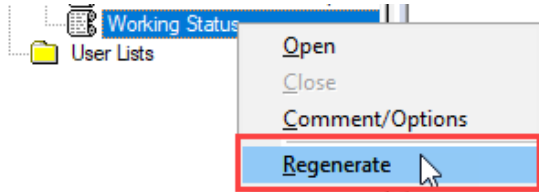
### To activate the checks

1. Click **Regenerate** on the shortcut menu of the status list.
2. Activate the required checks **Missing Symbols** and **Error Symbol Evaluation** in the **Determine Project Working Status** dialog.



### To regenerate an existing status list Working Status

1. **Regenerate** status list Working Status via the respective command on the shortcut menu.



### Integrating the checks into existing processes

If you are using processes in your project which already contain process action **Regeneration of Lists**, you can add the checks **Missing Symbols** and **Error Symbol Evaluation** to the process workflow by clicking **Change**.

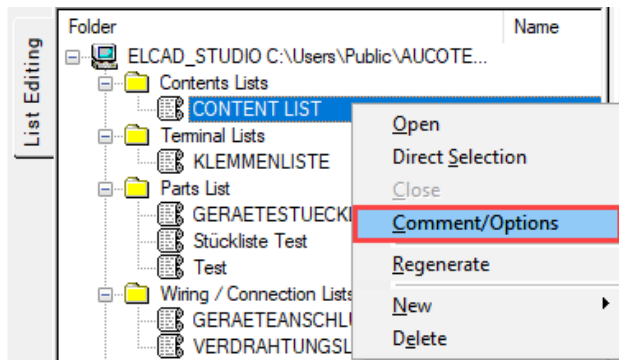
After the regeneration, you will find the identified errors in status list Working Status.

### 1.6.3 Displaying the configuration of a list

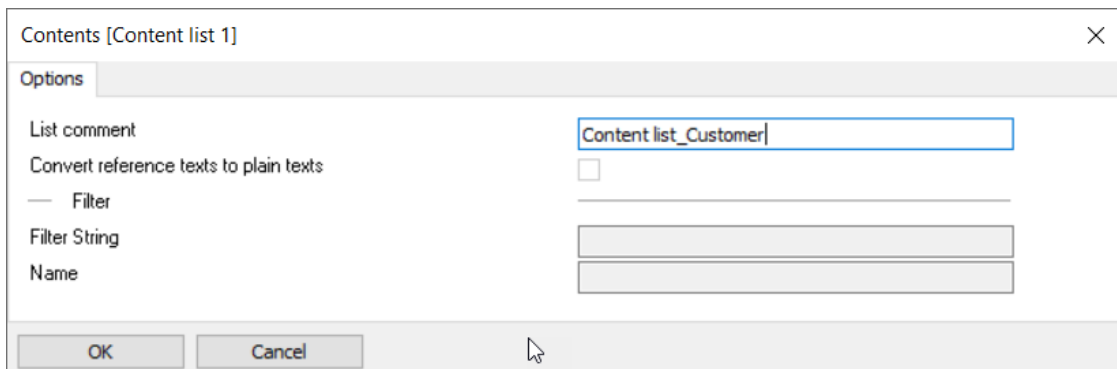
As of this version of ELCAD, you can display the configuration of a created or already regenerated list using **Comment/Options**. Depending on the list type, ELCAD offers different dialog items. The respective dialog is different for a directory, a parts list, a terminal list and a wiring list.

#### To display Comment/Options of a list

1. In the "List Editing" object tree on the shortcut menu of a list, e.g. a directory, click **Comment/Options**.



2. In this dialog, you can enter a list comment and display the configuration of the list.



3. Click **OK** to close the dialog.

If you want to edit the configuration of a list, use the **Regenerate** menu item on the shortcut menu of the list.

#### To deactivate the options dialog before the regeneration

Additionally, a new switch is available which you can use to deactivate the options dialog which is by default displayed before a regeneration. If you want to deactivate the options dialog, enter the following switch in file **aucotec.ini**:

```
[System]
RegenerationWithNoOptionMask=1
```

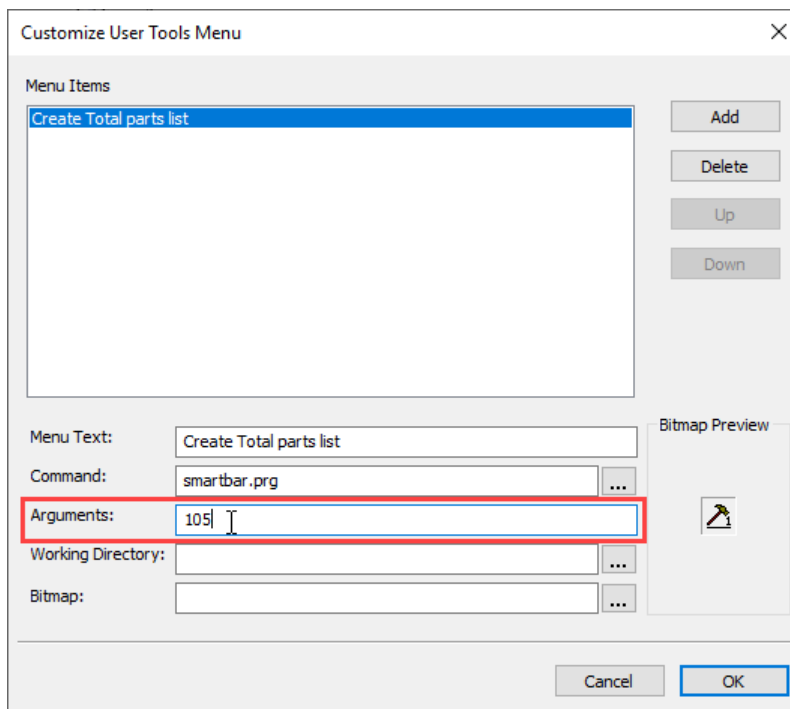
### 1.6.4 Creating an overall parts list by using a function code

As of this version of ELCAD, there is a new function code which you can use to integrate an overall parts list into the **Customize** menu. Use function code **105** to create an overall parts list.

#### To integrate an overall parts list into the Customize menu

1. Select menu item **Extras/Tools/Customize**.
2. Click the **Add** button.
3. In line **Menu Text**, enter the designation of the new menu item to be created.
4. In line Command, enter **smartbar.prg**.
5. In line Arguments, enter function code **105**.
6. Optionally, enter a **Working Directory** and a **Bitmap**.
7. Click **OK** to close the dialog.

After that, you can integrate the new function into the toolbar, so that you can start the new menu item via a button on the toolbar.

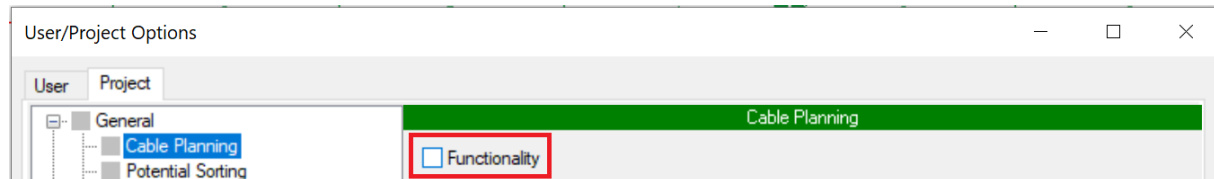


## 1.7 New in Cable Planning

### 1.7.1 Optimized control elements in the cable planning

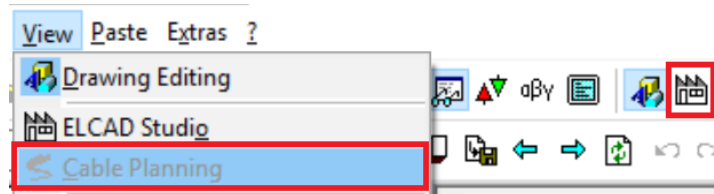
As of this version of ELCAD, the control elements to call the cable planning are optimized. If the cable planning is not activated under **General/Cable Planning** in your project, the respective control elements are inactive.

#### Extras/User/Project Options



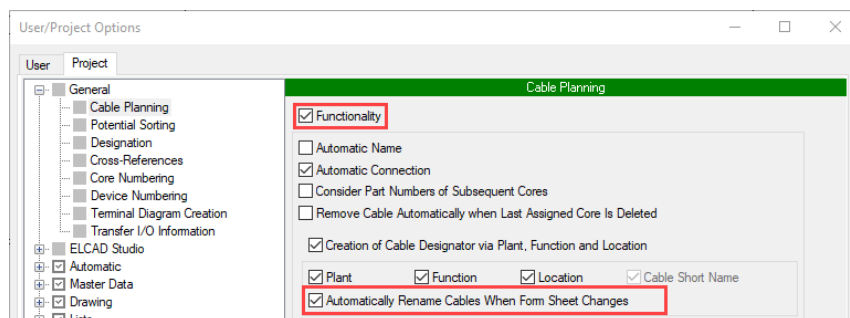
This improvement affects the menu entry and the respective icon on the toolbar to call the cable planning.

#### View/Cable Planning



### 1.7.2 Automatic update of cable designators

As of this version of ELCAD, you can define that cables represented on drawings are automatically renamed when the plant, function and location of the form sheet is renamed. The following preconditions are required:



#### To automatically update the cable designators

1. Open the project options under **General/Cable Planning**.
2. Activate the cable management by checking the **Functionality** checkbox.
3. Check checkbox **Automatically Rename Cables When Form Sheet Changes**.



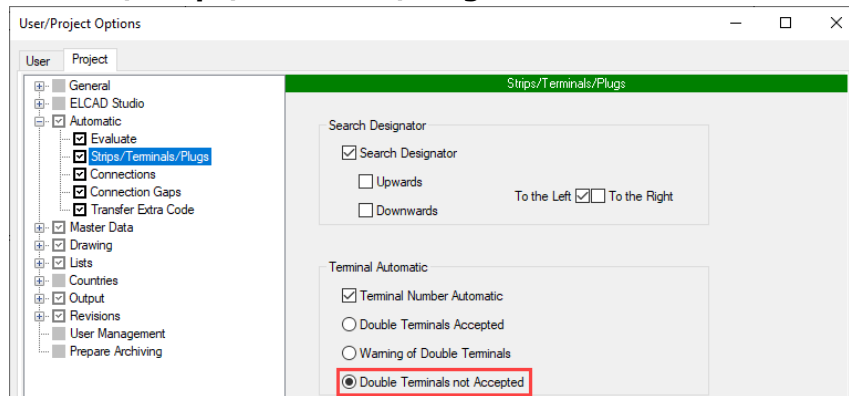
More than 50 percent of the cores of a cable must be represented on the drawing.

### 1.7.3 Status list Working Status: Display of duplicate terminal numbers

As of this version of ELCAD, you can display duplicate terminal numbers as errors in the status list Working Status once you have evaluated the symbols.

#### To activate information on duplicate terminal numbers

- Activate option **Double Terminals not Accepted** in the project options under **Automatic/Strips/Terminals/Plugs**.



From now on, any terminal numbers that you create twice in ELCAD are no longer accepted by ELCAD.

A precondition for ELCAD to display duplicate terminal numbers in status list Working Status is that you have re-evaluated the drawings in advance.

	Status	Description	Item	Comment
1	Error	Duplicate Terminal Number	=A1.PS+C1-0X1:2	
2	Error	The core is in a wiring node without connectors.	=A1.D+C1-W3:3	



If you regenerate status list Working Status, it contains error messages regarding those terminal numbers that exist twice.



## 1.8 New in Batch Processing

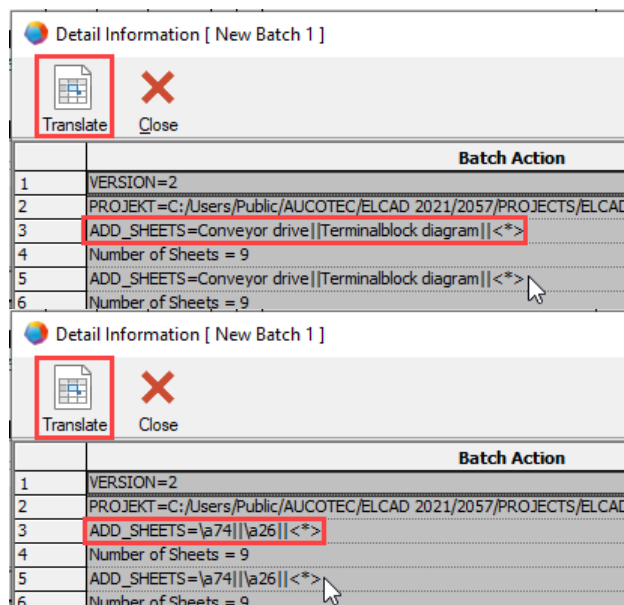
### 1.8.1 Displaying plain texts instead of translate references in batches

As of this version of ELCAD, the **Detail Information [New Batch]** dialog contains a **Translate** button which you can use to switch between the plain texts of the batch actions and their translate references. As of this version of ELCAD, the plain texts are visible by default.

By displaying the plain texts, you can identify at first sight which batch action affects which object on the navigation pane. Moreover, you can directly see to which drawing sheets the respective action is applied. Sheets or folders containing sheets can, for instance, be defined objects for a batch action. By repeatedly clicking the **Translate** button, you can switch between the representation of the translate reference and the representation of the plain text.

#### To switch the plain texts of a batch

1. Double-click the required batch.
2. Click **Detail Info** to open the **Detail Information** dialog.
3. Click **Translate** to switch the plain texts of the batch actions over to translate references.



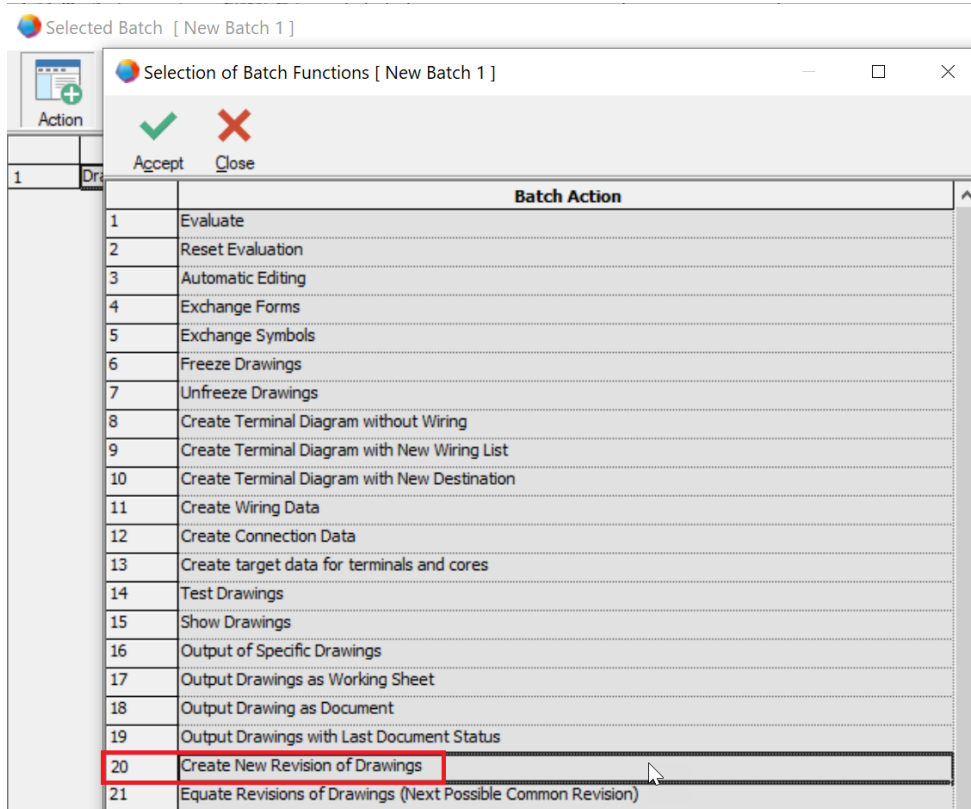
### 1.8.2 Extended batch function "Create New Revision of Drawings"

So far, you can use batch action **Create New Revision of Drawings** to create revisions of all drawings of a batch. As of this version of ELCAD, the batch function has an additional option. By checking the respective checkbox, you can execute the revision for changed documents only.

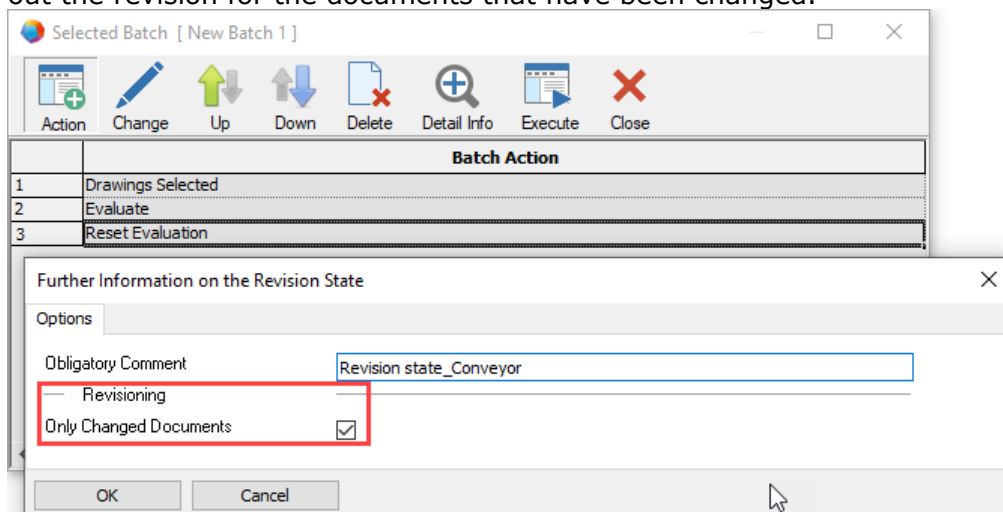
In addition to that, you can set the drawing revisions to a common revision state. ELCAD uses the next possible revision name that is permitted for all drawings as the common revision.

**To carry out a revision for changed documents only**

1. Double-click the required batch in the object window.
2. Select the row in the batch where you want to insert the batch function.
3. Click **Actions**.
4. Select batch action **Create New Revision of Drawings**.
5. Click **Apply**.



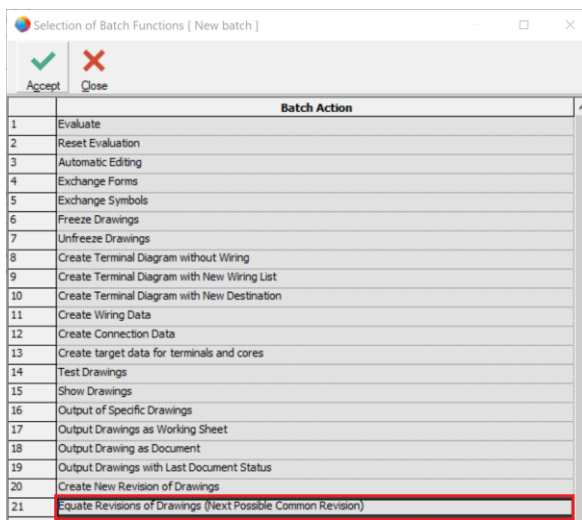
In the following dialog, check checkbox **Only Changed Documents** to only carry out the revision for the documents that have been changed.



### 1.8.3 Extended batch function "Equate Revisions of Drawings"

#### To equate the revision of the drawings

1. Double-click the required batch in the object window.
2. Select the row in the batch where you want to insert the batch function.
3. Click **Action**.
4. Select batch action **Equate Revisions of Drawings (Next Possible Common Revision)**.
5. Click **Accept**.



6. Then, set the required options that you want to use on setting the revisions.

