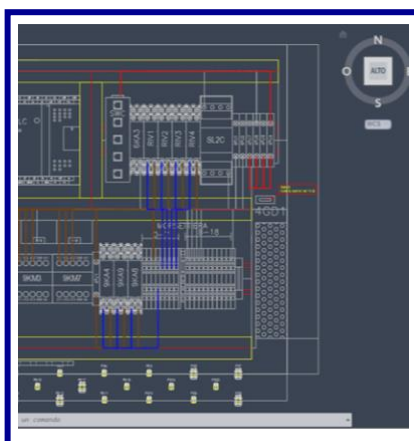
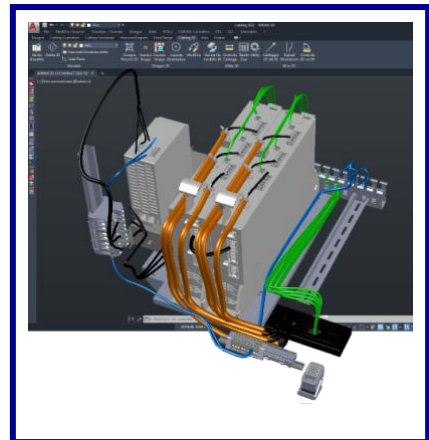


# NEWS RELEASE 2024

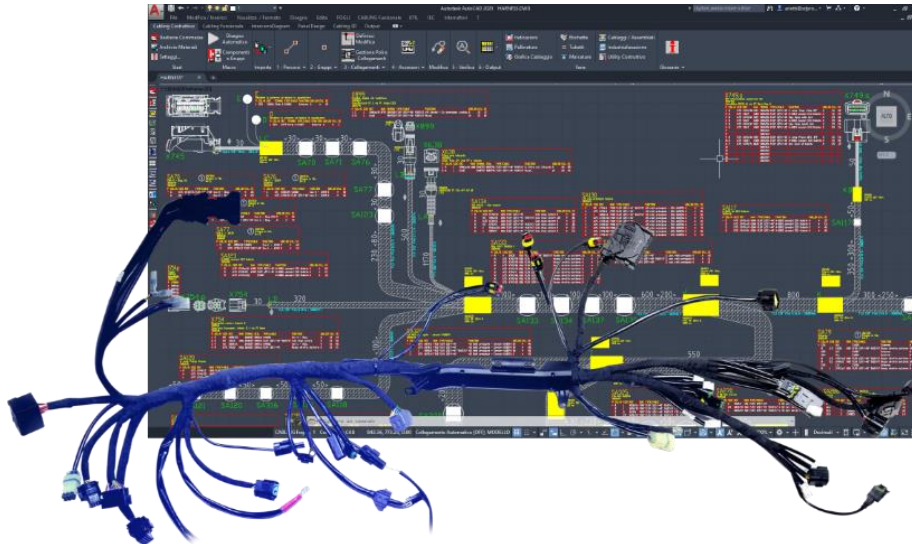


**CAD.Able s.r.l.**

C.so Giambone, 69  
10134 - Torino - Italy  
Tel. +39 011 614236  
[info@cadable.it](mailto:info@cadable.it)  
[www.cadable.it](http://www.cadable.it)

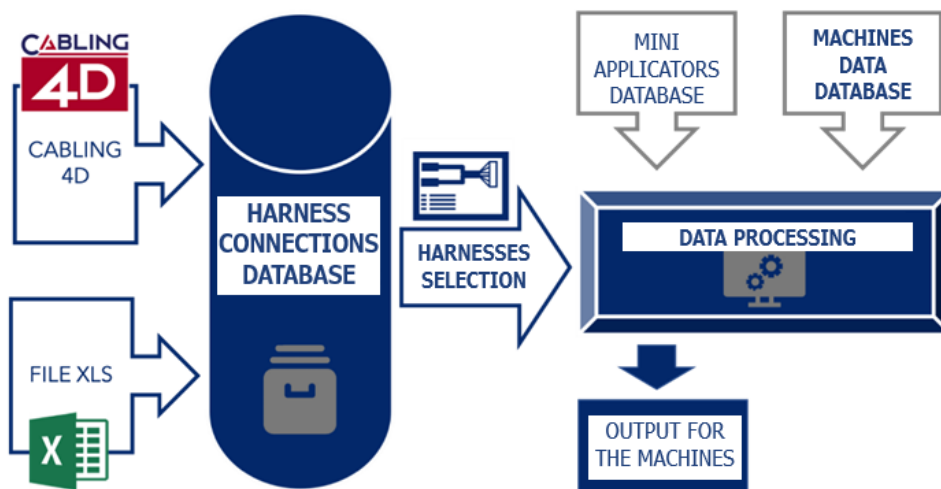
# THE NEW RELEASE 2024

**CABLING 4D** continue to consolidates its position in the market as a complete, powerful and reliable tool for **the design of cables and harnesses in the electrical sector.**



CABLING grows with a new product: **CABLING 4P or CABLING for PRODUCTION**

CABLING 4D is an independent software in a Windows environment that allows you to create/manage production job orders with different harnesses to be distributed on one or more automatic CUTTING/STRIPPING/CRIMPING machines



CABLING has in fact created a market standard used by both producers and designers of harnesses because it offers all the information necessary for both the production and the realization of the technical documentation. **Drawings can be shared between companies or between departments of the same company, creating synergy and reducing time and errors.**

In addition, other tools can be integrated into the design of the harnesses: this allows to increase the use of data containing in the harness projects of CABLING to give a feedback to other needs:

- Detailed wiring diagrams and electrical diagram that can be used as documentation and to facilitate technical assistance of you machinery: we are convinced that in sectors other than industrial automation, they must be more completed and readable in a simple and practical way.
- Modular interface with the cutting / strip / crimping machines and with the assembly departments
- Modular interface with harness testing systems
- Dynamic navigators on the project
- Time calculation for the production and assembly of harnesses.

Our mission today is undergoing a process of expansion, in correlation with the rapid changes in the market, in support of the electrical / electronic sector that today more than ever is the spearhead of industrial technology.

## **Importance makes a project necessary!**

CAD.Able, always in contact with the design departments, continues its development strategy of CABLING 4D by responding in an innovative way to current and future design needs, also in terms of reliability and time. This vision of ours, supported by the feedback from our customers, we trust is the right direction for the electrical sector.

The content of this document contains most of the new features introduced on the product.



# INSTALLABLE ON

---

## AutoCAD OEM version

Autodesk 2024 64 bit graphic engine



## APP to AutoCAD version

Installable on AutoCAD 2022, 2023 and 2024



# HARDWARE & SOFTWARE REQUIREMENT

---

*Minimum hardware & software configuration required:*

## HARDWARE

Personal Computer

- Processore Intel Core i5© or higher with SSE2 technology
- Processore AMD Ryzen 5™ or higher with SSE2 technology
- Mouse or other pointing device
- Possible plotter or printer
- 8 GB RAM memory

Video card

- 1 GB video card with 29 GB/s of bandwidth, compatible DirectX 11
- Screen DPI scale factor less than or equal to 125%
- Are compatible also the Ultra HD-4K monitors

USB Ports

- USB port for the hardware protection
- USB 2.0 port for installation

Free space on disk:

- CABLING 4D 2024                                 7.5 GB
- CABLING 4D CAD 2024                         9.5 GB (associated to SPAC Automation CAD 2024)

Internet access for the installation and on-line update

## SOFTWARE

O.S.

- Windows© 10 64 bit - version 1909 or successive
- Windows© 11 64 bit

App to AutoCAD

- Autodesk AutoCAD or Electrical rel. 2022/2023/2024 already installed and working.
- AutoCAD LT is not valid for the installation.

OEM version

- Autodesk AutoCAD© is not required with SPAC Automation OEM engine.

*Suggested hardware configuration:*

- 3GHz Intel® Core™ i7 or AMD Ryzen™ 7 Processor with SSE2 technology
- 4 GB video card with 106 GB/s of bandwidth, compatible DirectX 11
- USB 3.0 port for installation
- RAM memory: 16 GB
- Autodesk certified graphics hardware and drivers available at the following link  
<https://knowledge.autodesk.com/it/certified-graphics-hardware>

Attention

- System administrator permissions are required to install.



# INDEX

<b>HIGHLIGHTED</b> .....	<b>8</b>
<b>1 3D MODULE</b> .....	<b>9</b>
<b>2 MATERIALS DATABASE</b> .....	<b>12</b>
2.1 LIBRARIES MIGRATION .....	12
2.2 MATERIALS DATABASE .....	13
2.3 MATERIALS DATABASE - IMPORT .....	15
2.4 EXPORT/IMPORT DATA OF THE MATERIALS DATABASE .....	16
2.5 DEFINITION OF MOULDED CABLE CONDUCTORS .....	17
2.6 CABLES DEFINITION .....	18
2.7 EXTERNAL DIAMETER BASED ON THE WIRE SECTION .....	19
2.8 UPDATE HARNESS LAYOUT WITH DATA OF MATERIALS DATABASE .....	20
<b>3 HARNESS LAYOUT</b> .....	<b>21</b>
3.1 IMPORT GROUPS AND IMPORT CONNECTIONS .....	21
3.2 AUTOMATIC BALLOONS .....	23
3.3 EXCLUSIONS IN THE BILL OF MATERIALS .....	24
3.4 ERROR MANAGEMENT IN THE CONNECTION WINDOW .....	25
3.5 HARNESSES AND SUBSETS MANAGEMENT .....	26
3.6 UNPLUGGED GROUP PINS GRAPHICS .....	28
3.7 DRAW BUNDLES .....	30
3.8 IGNORE ACCESSORIES IN BILL OF MATERIALS .....	32
3.9 GROUPS INDICATIONS .....	34
3.10 ASSIGN FINISHING DEPARTMENT .....	35
3.11 DRAW BUNDLES .....	38
3.12 CHANGE OF A TERMINAL .....	40
3.13 CONNECTIONS LIST OUTPUT .....	41
3.14 GRAPHIC SYMBOL OF BRANCH .....	42
3.15 INSERIMENTO SALDATURE E COMPONENTI LUNGO I PERCORSI .....	43
3.16 BUNDLES TEXTS .....	44
3.17 GROUP ATTRIBUTES MANAGEMENT .....	45
<b>4 WIRING DIAGRAM</b> .....	<b>46</b>
4.1 SPLICES AND WIRE TERMINALS INSERTION .....	46
4.2 INSERTING COMPONENT SYMBOLS .....	49
4.3 MOTHER AND CHILDREN COMPONENTS MANAGEMENT .....	57
4.4 CROSS REFERENCE .....	59
4.5 DISPLAYING CONNECTIONS DRAWING ERRORS .....	60
4.6 VIEWING CONNECTION DATA .....	61
4.7 MODIFY UNPLUGGED GROUPS .....	63
4.8 DRAW CONNECTIONS .....	64
4.9 CONNECTION RETURNS .....	66
4.10 SHIELDS .....	68
4.11 RETURNS ON CONNECTIONS .....	69
<b>5 PANEL DESIGN</b> .....	<b>72</b>
5.1 CONNECTIONS DRAWING .....	74
5.2 MODIFY CONDUITS AND DIN BARS .....	78
<b>6 JOB-ORDERS MANAG. AND TITLE BLOCKS</b> .....	<b>79</b>
6.1 NEW JOB-ORDERS MANAGEMENT .....	79
6.2 MASTER TITLE MANAGEMENT .....	81
6.3 GENERATE MULTISHEET COMMAND .....	82
6.4 MASTER TITLE BLOCKS AND REVISIONS .....	83



**7 VARIOUS ..... 84**  
7.1 ADAPTATION TO WINDOWS TEXT SIZE .....84  
7.3 CHECK BLOCKS INSERTED IN THE DRAWING.....85  
7.3 IMPORT GROUPS AND CONNECTIONS .....87  
7.4 TABLE OUTPUTS.....88



## HIGHLIGHTED

# CABLING

## CAD FOR ELECTRICAL HARNESSSES

**CABLING 2024 changes the rules for the electrical diagram** because it makes it easier to insert components and takes advantage of the most reliable data source: the cables and harnesses designed with its layout environment.

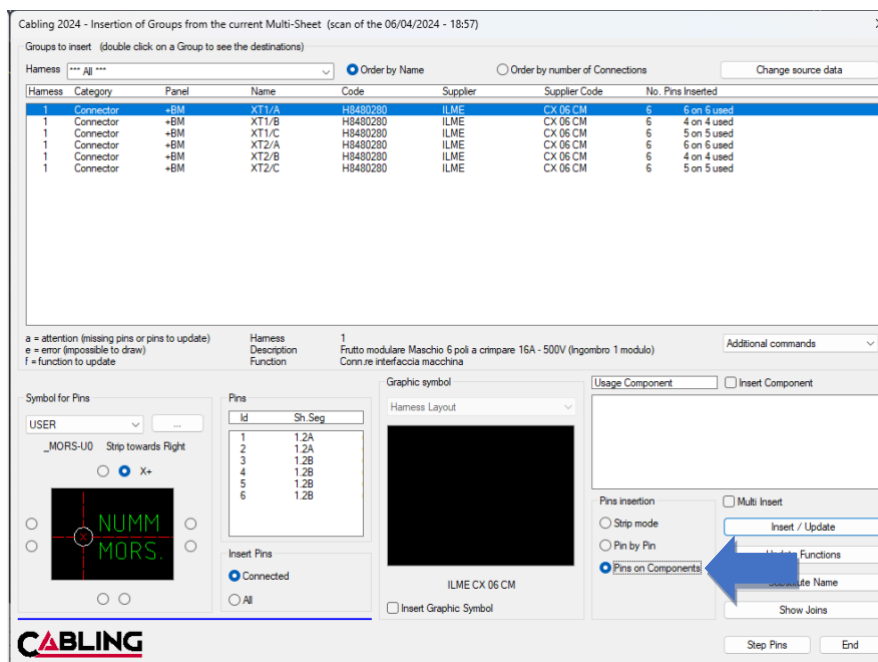
Now you can insert components in several ways:

- With the "Insert Components" command.
- By inserting the standard menu components
- By inserting the BLK user library Components
- Attaching a "Child" symbol to a "Mother" component.

You can easily generate output with the list of components inserted in the current Multisheet with their name, function and material code.

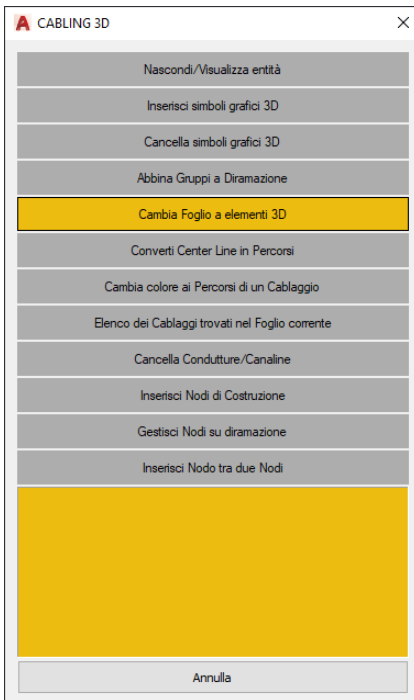
After inserting the Components, you can use the data of one or more harnesses and cables to insert the mating parts of the components in the desired position (groups such as connectors, terminals, etc.). We have added the new 'Insert Pins on Component' option which allows you to select a Component that becomes the 'Usage' of the Group to be inserted.

In the end you can easily import the connections between the inserted mating parts.





# 1 3D MODULE



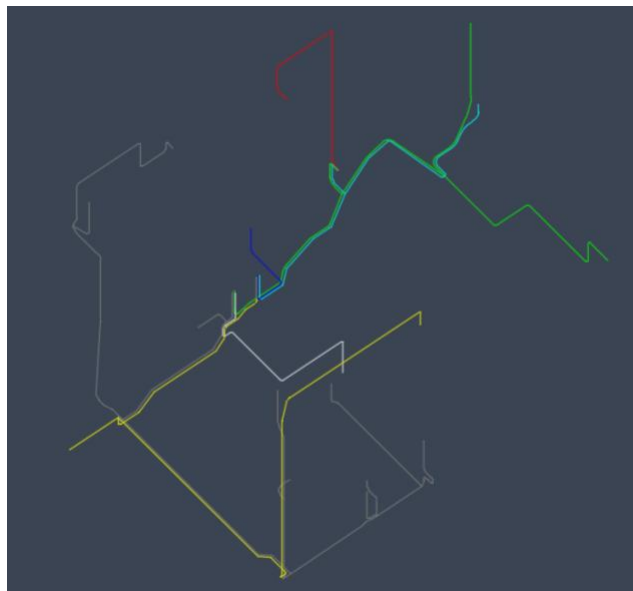
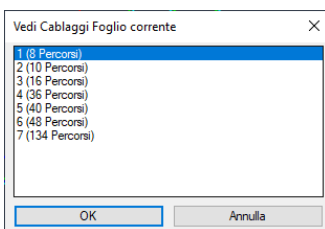
During the 2023 and 2024 releases, additional options were added to the utility command of the 3D module:

- Change Sheet to 3D elements
- Change color of the Bundles of an Harness
- List of Harnesses found in the current sheet.

The first option allows you to change the Sheet of 3D entities such as surfaces and solids by selecting them quickly and easily and indicating which layer/sheet to move the entities to. In the current sheet only the center-lines of the bundles remain, which can be used to determine the harnesses and the relative lengths.

The second option allows you to change the color of the bundles of a harness. A choose window asks which color to use (8 colors in total). An harness is recognized by the mutual continuity of the bundles that make it up. The change of colors helps not only to identify the harnesses, but also to see where they are interrupted due to small inaccuracies in the 3D model (if the harness arrives already designed by the mechanical department).

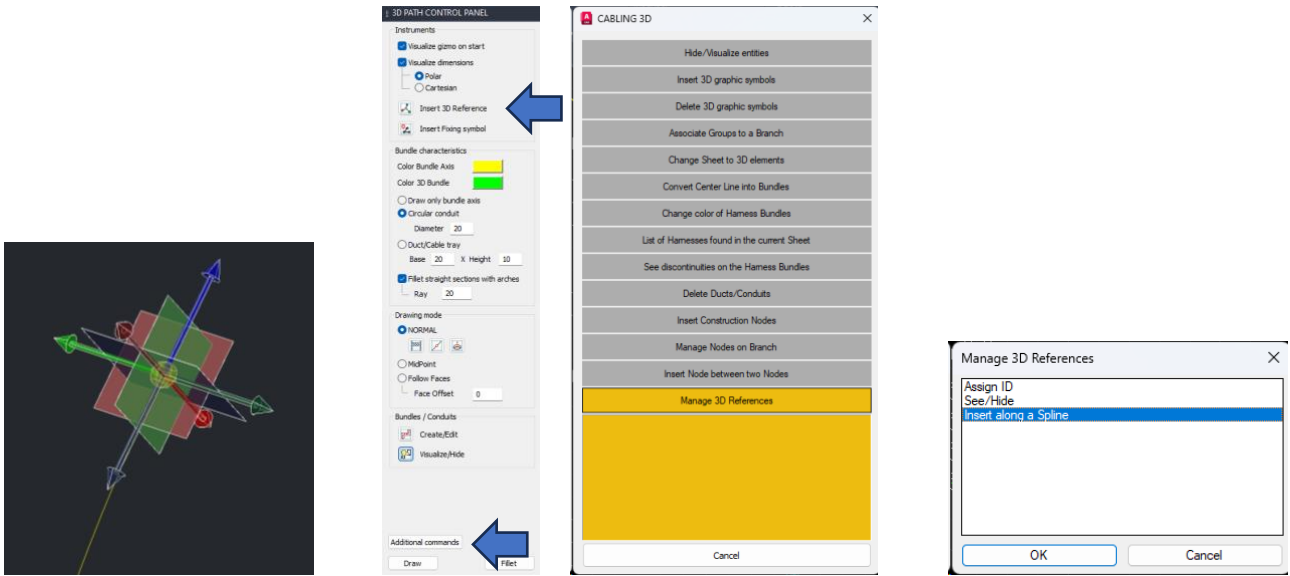
Similar to the color change, the 'List of Harnesses found in current sheet' option also helps to identify 3D Harnesses. A dialog opens with the list of found harnesses sorted by the number of bundles that make up them. With the OK button the window is closed and the selected harness is highlighted on the drawing.



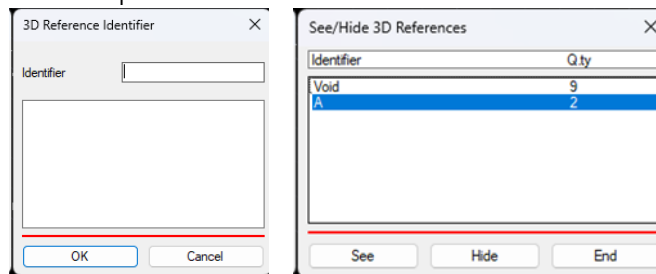
The '2D harness from 3D' command also has a novelty: if, after laying out a bundle in 2D to which other single 3D bundles are connected (without further branches), you are asked whether you want to draw them as multiple bundles (similarly to the command 2D bundles). You are not asked for the number of bundles and the lengths, which are acquired from the 3D. This speeds up the operations of laying out the 3D harness in 2D.



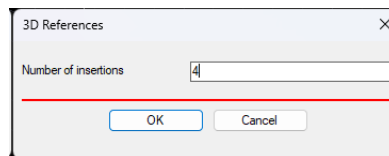
To allow you to work with Bundles made up of splines, that are free curvilinear bundles typical of the automotive field, we suggest to use the 3D References blocks, which can be inserted from the 'Draw 3D Bundles' command dialog box.



Since it is possible that numerous 3D references are inserted in the drawing, two new functions have been inserted in the 'Manage 3D References' command (with the menu that can be called up from the 'Additional Commands' button): 'Assign Identifier' and 'Show/Hide'. The first allows you to assign an identifier to the 3D References, the second allows you to display / hide the References with one or more identifiers. It is easy to name a set of 3D references that will make up a complex spline, isolate them by hiding unnecessary references, thus facilitating their selection. The 'Void' identifier represents unassociated references.

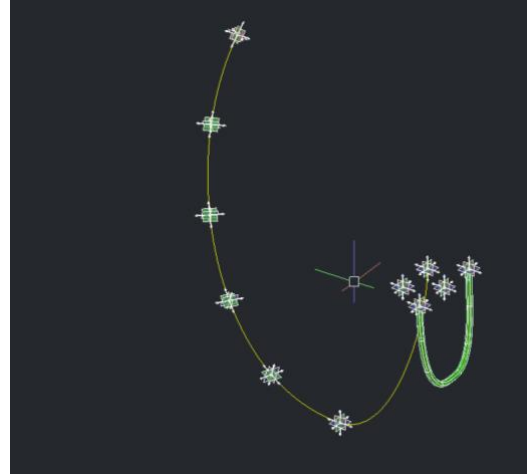
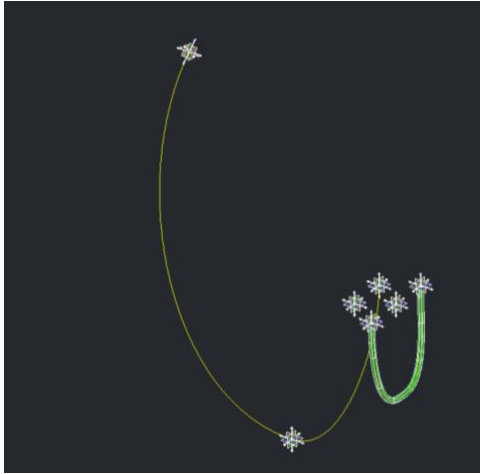


Finally, the 'Insert along a Spline' function is used to insert one or more equidistant 3D References along a previously drawn spline.



The references are aligned to the tangency of the curve at the point where they were inserted, they will break the original spline into several pieces and you can modify the position of the 3D references more precisely, which will automatically deform the curve to which they are connected.





These changes have made the 3D module even more powerful in its use, allowing you to work even with complex Bundles on 3D models imported from mechanical design.

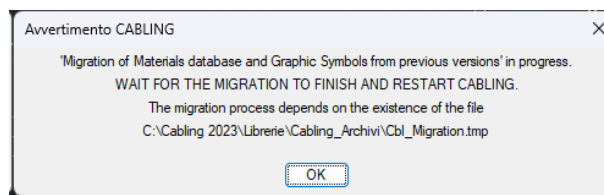


## 2 MATERIALS DATABASE

### 2.1 LIBRARIES MIGRATION

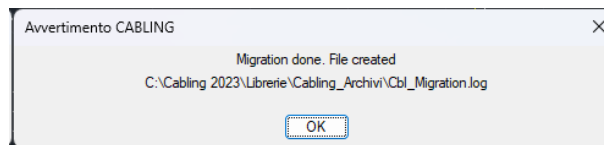
At the first launch of the new 2024 release, CABLING automates the 'Migration of Material database and Graphic Symbols from previous versions' to facilitate the user.

The previous versions of CABLING installed are searched and you are asked from which of them you intend to migrate the materials database, settings and graphic symbols. If you choose to migrate, the operation will start by creating a LOG file (..\Cabling\_Archivi\Cbl\_Migration.log) with a summary of the actions performed and any errors. Other users who may attempt to use CABLING 2024 will be notified of the ongoing migration process.

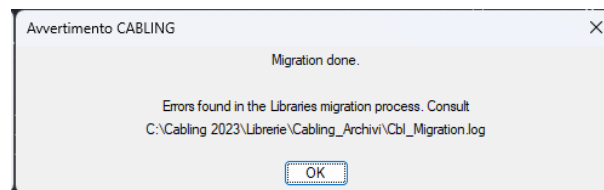


During the migration, the ..\Cabling\_Archivi\Cbl\_Migration.tmp file is created which indicates that the process is in progress. This file is reported in the message because, in the unlikely event of a migration problem, you can delete this file and the message will no longer appear when using CABLING.

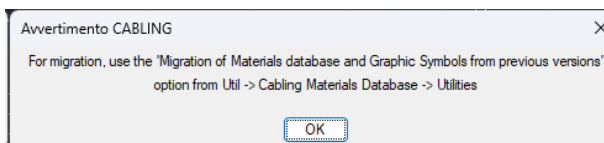
If the migration completes without problems, this message will be given:



If the migration ends with problems, this message will be given:



If you have chosen not to perform the migration, this message is given:



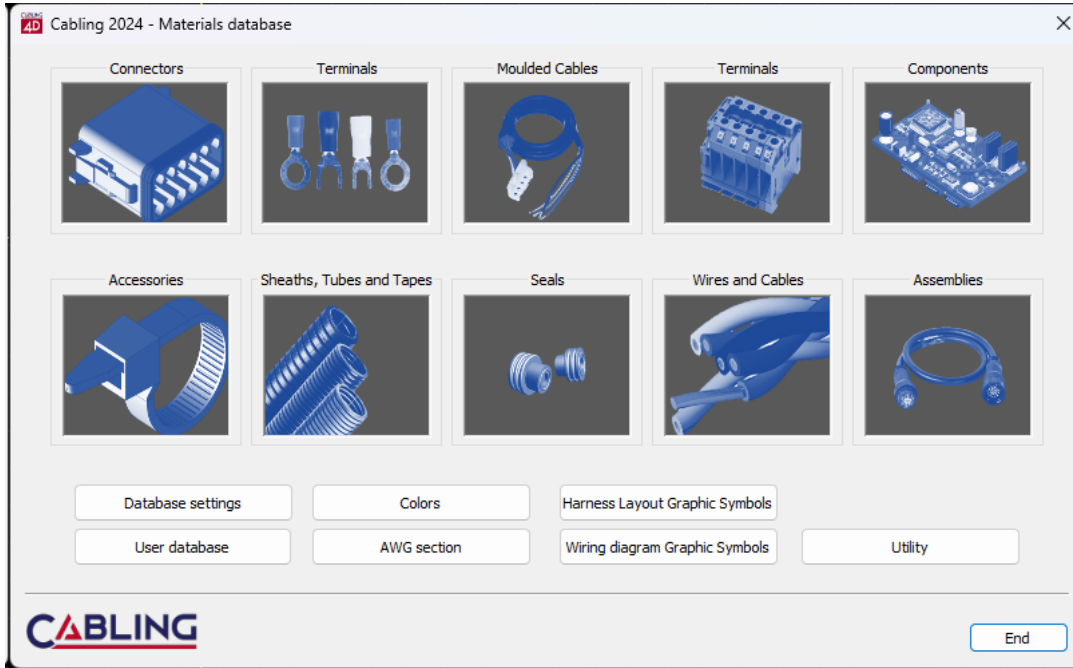
which indicates how to perform the migration later.

The migration creates "Installation" sub-folders with a backup copy of the installation files. If you perform subsequent migrations (only if necessary) the migration creates "Migration" sub-folders with an index "l..n" with the backup copy of the previous files.



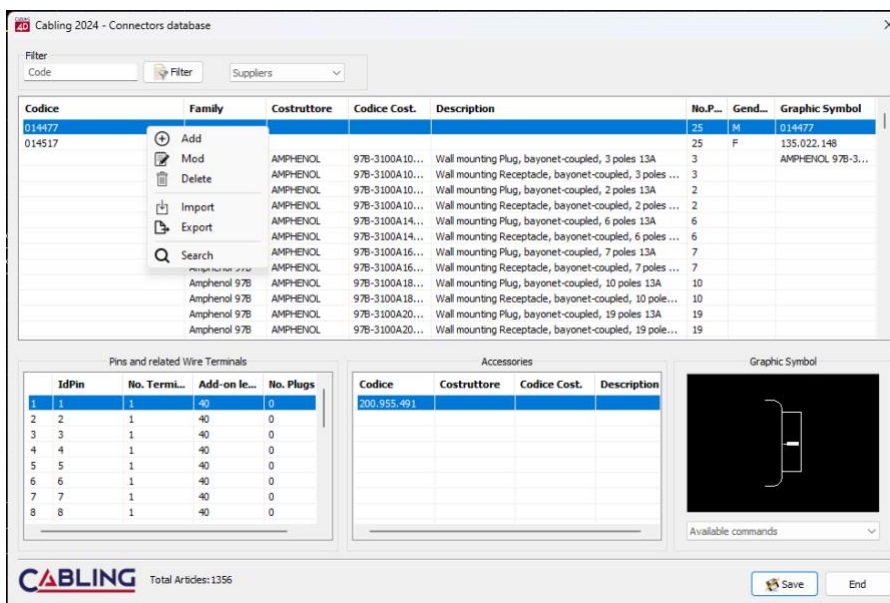
## 2.2 MATERIALS DATABASE

The 2024 version has changed the graphic interface of the materials database. When the 'Cabling Materials database' command is started, this new window is now opened where the buttons with the images of the categories of managed materials appear together with the additional buttons. 'Tool-tips' are also displayed with descriptions of the button actions.

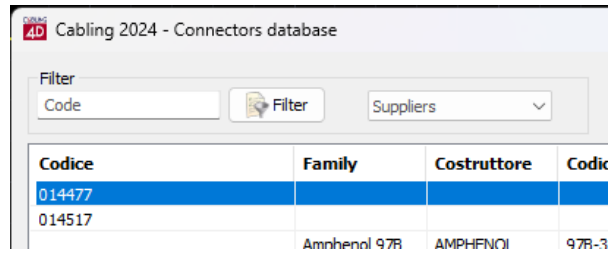


The windows of the various material categories have also been completely revised with the following new features:

- List of materials on grid with resizable columns
- Contextual menus on the various window grids
- Search on main window (the old search mode has also been retained)
- Tooltips on fields that are not fully viewable



014517	Amphenol 97B	AMPHENOL	97B-3100A10...	Wall mounting Plug, bayonet-coupled, 3 poles 13A	25	F	135.022.148
	Amphenol 97B	AMPHENOL	97B-3100A10...	Wall mounting Receptacle, bayonet-coupled, 3 poles 13A	3		AMPHENOL 97B-3...
	Amphenol 97B	AMPHENOL	97B-3100A10...	Wall mounting Plug, bayonet-coupled, 2 poles 13A	2		
	Amphenol 97B	AMPHENOL	97B-3100A10...	Wall mounting Receptacle, bayonet-coupled, 2 poles ...	2		

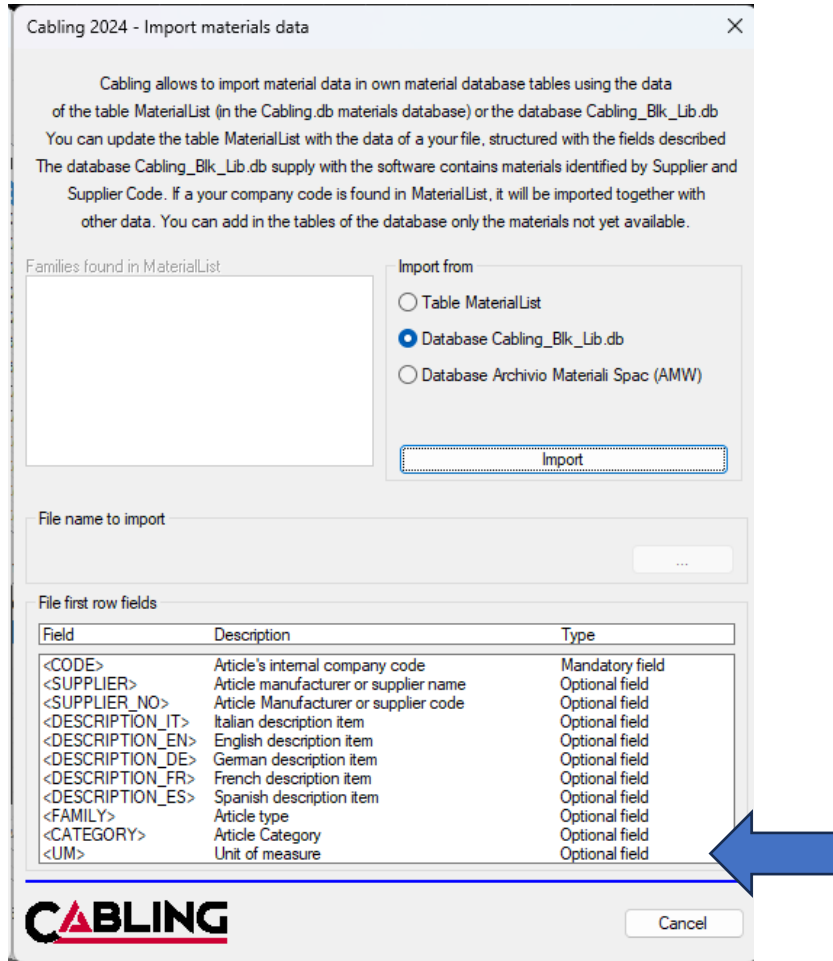


The new windows simplify the user interface and make them more modern and usable.

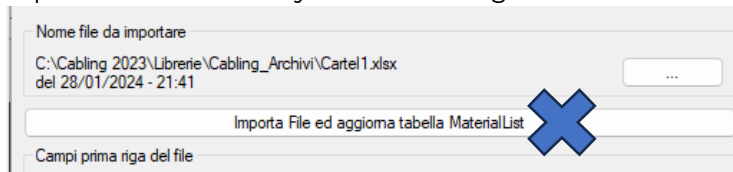


## 2.3 MATERIALS DATABASE - IMPORT

The 2024 version allows you to import materials into the MaterialList table, also including the unit of measurement, which however can only be used when importing accessories.



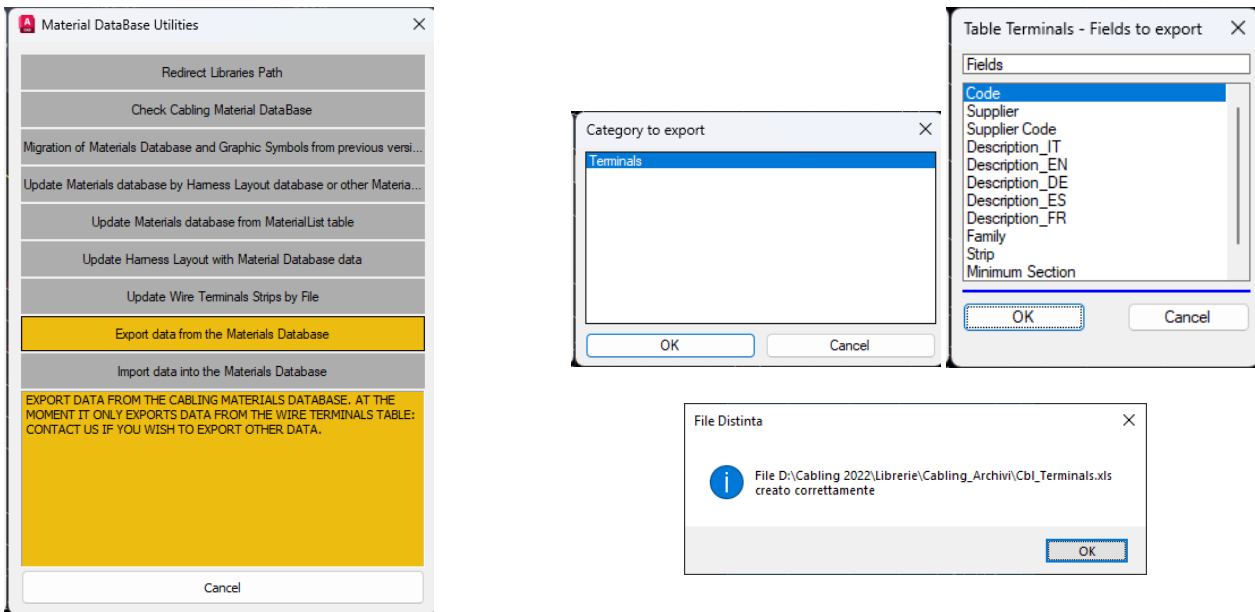
Furthermore, the 'Import File and update MaterialList table' button has been eliminated: instead, confirmation of the import is asked directly after choosing the file with the button with the dots.



## 2.4 EXPORT/IMPORT DATA OF THE MATERIALS DATABASE

Two new options have been added to the materials database Utilities which currently only concern one type of material from the Cabling material archive, but which can be completed by us based on your requests: 'Export materials database data' and 'Import data into the materials database'.

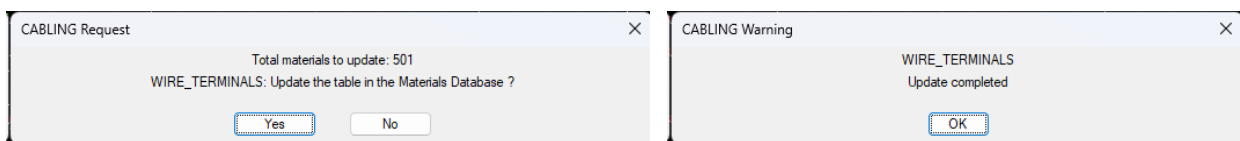
The basic concept is the need to export data from the Cabling archive to an Excel file, be able to modify it and be able to re-import it into the Cabling archive always referring to the material code as a unique field. In this way the 'maintenance' of the archive will be easier than the changes that can be made individually with the functions of the material database.



Today the only category that can be exported is the Wire Terminals category which creates the Cbl\_Terminals.xls file in the Cabling\_Archivi folder of the Library. After modifying the file without changing the first column of the file, it is possible to re-import it with 'Import data into Material Database'.

The screenshot shows an Excel spreadsheet with the following data:

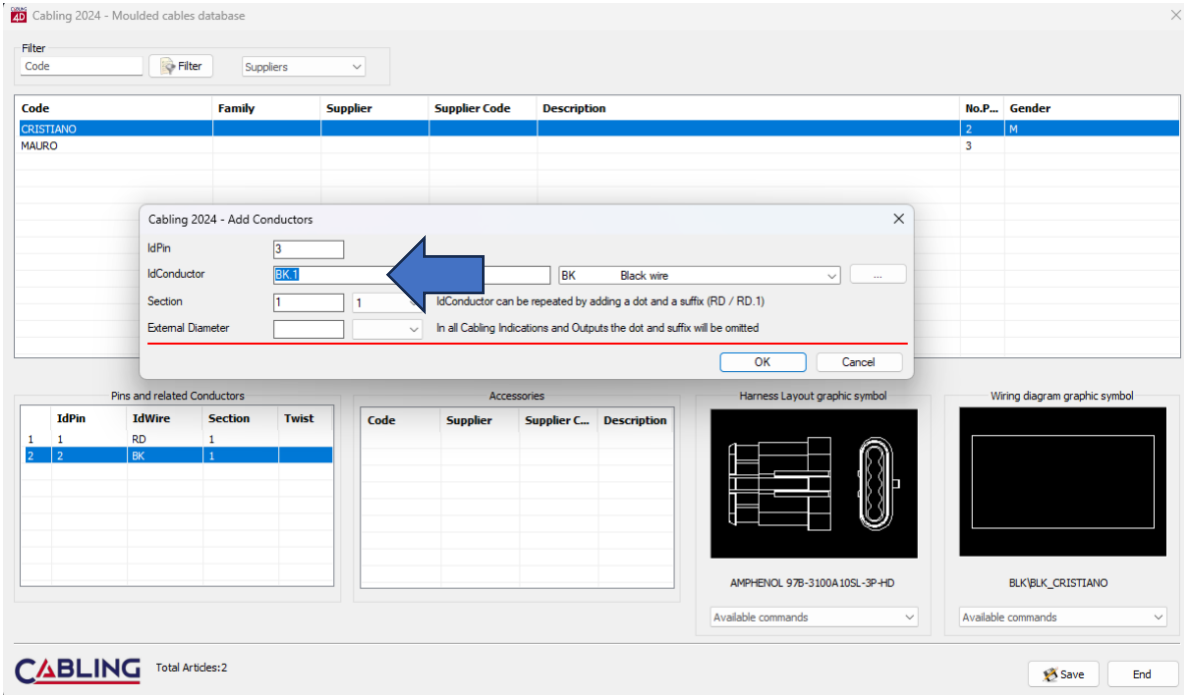
Wire_Terminals	Code	Supplier	SupplierCode	StripLength	SectionMin	SectionMax
2	9001	RAYCHEM	CWT-9001	5		
3	9002	RAYCHEM	CWT-9002	5		
4	9003	RAYCHEM	CWT-9003	5		
5	9004	RAYCHEM	CWT-9004	5		
6	9005	RAYCHEM	CWT-9005	5		
7	XS000001	RAYCHEM	SGRS-1	5		
8	XS000002	RAYCHEM	SGRS-2	5		
9	XS000003	RAYCHEM	SGRS-3	5		
10	XS000004	COVI	SA01	13	0.75	8
11	XS000005	COVI	SA02	13	4	12
12	XS000010	CEMBRE	L10-M	20		





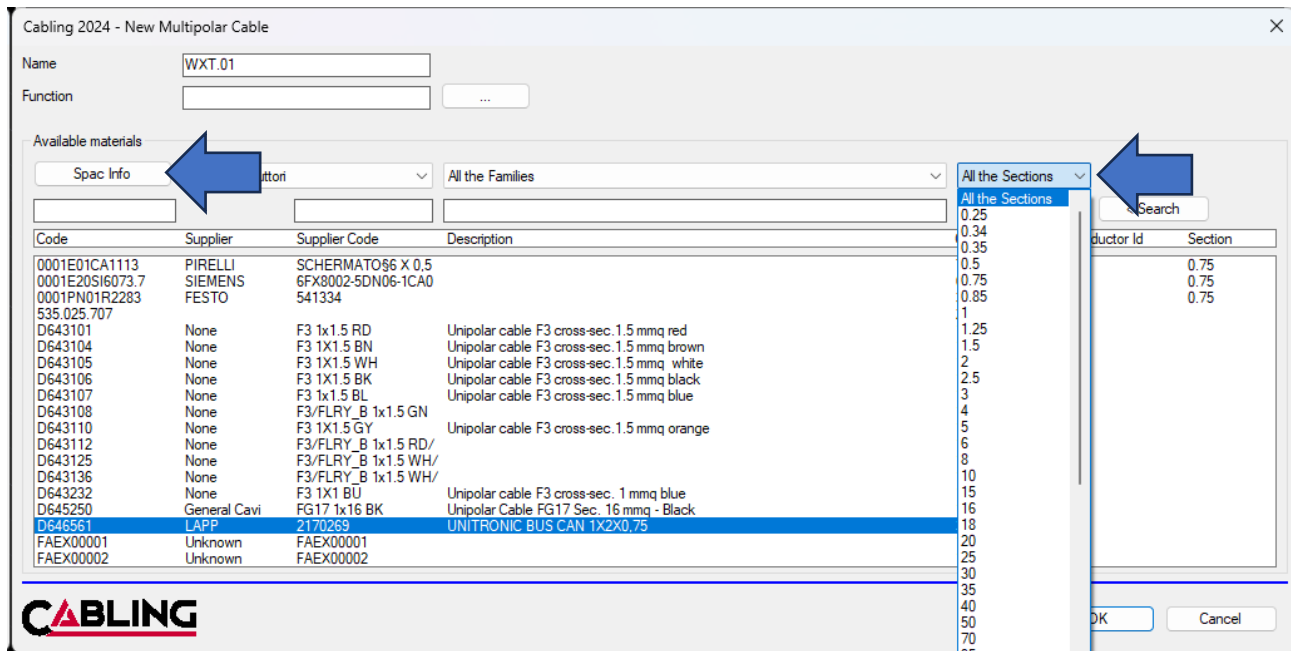
## 2.5 DEFINITION OF MOULDED CABLE CONDUCTORS

In the definition of a moulded cable it is now possible to add a repeated Conductor Id (color), which was not possible until now. Simply add a point and a suffix to the ConductorId which are displayed in the Cabling windows but hidden in the CABLING Indications and Outputs.



## 2.6 CABLES DEFINITION

When defining a new multi-core cable it is also possible to choose the material based on one of its sections.



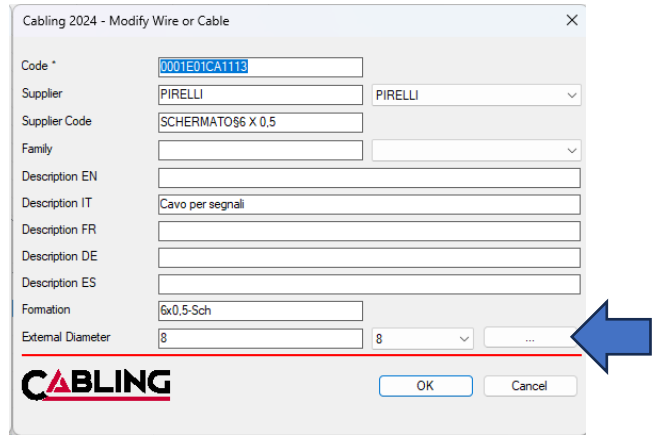
Furthermore, the new Spac Info button has been added which, if CABLING is combined with SPAC Automazione, allows you to query the SPAC cable materials archive (DBCABLES.DB) to view further information not available in the CABLING materials archive, with equal of material code. In particular:

- Category
- External Color
- Curvature radius [mm]
- Conductive material
- Insulation material
- Screen material
- Sheath material
- Operating temperature
- Voltage [V]
- Dielectric strength [kV/mm]
- Electrical Resistance [ohm/km]
- Weight [kg/km]
- Price [euro/m]
- Note
- Technical notes



**2.7 EXTERNAL DIAMETER BASED ON THE WIRE SECTION**

In the materials database, adding and modifying single-core wires in the Wires and Cables archive provides further help in defining the external diameter.



The new button has been added [...] indicated by the blue arrow which opens a new window where you will have the list of the main sections in mm<sup>2</sup> with the corresponding external diameter. These are theoretical data typical of the automotive sector and must be verified with the actual type of wire which may have diameters depending on the type of insulation.

Section	Diameter
0.35	1.3
0.50	1.6
0.75	1.9
1.00	2.1
1.50	2.4
2.00	2.6
2.50	3
4.00	3.7
6.00	4.3
8.00	5.5
10.00	6
16.00	7.9
25.00	9.4
30.00	11
35.00	11.6
50.00	15.5

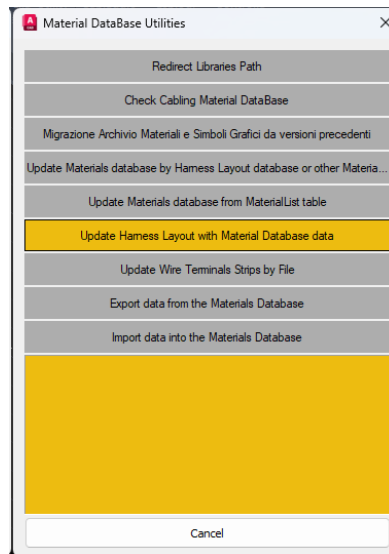
We remind you that the external diameter favors two aspects of design with 4D CABLING:

1. Check that the diameter is compatible with the minimum and maximum diameter attributed to the individual rubbers, typical of the automotive sector
2. Check that the maximum diameter of the wire bundles calculated with the CABLING functions is adequate with the diameter of the tube or protection sheath.



## 2.8 UPDATE HARNESS LAYOUT WITH DATA OF MATERIALS DATABASE

With the 'Update Construction with Materials Archive data' option called up with the 'Utility' button in the materials archive window, it is now possible to also update the material codes.



For example, if you have defined a connector with only the Supplier and Supplier code, the code that is stored on the drawing is the composition of the two values with the separation character '\$'.

TYCO Code\$ 284844-4 → Supplier TYCO → Supplier Code 284844-4

If the code is subsequently redefined, the correspondence between the drawing and the materials archive is lost. With this procedure, if the Supplier and Supplier code are found in the archive, the code is acquired and replaced in the drawing in place of TYCO\$ 284844-4.

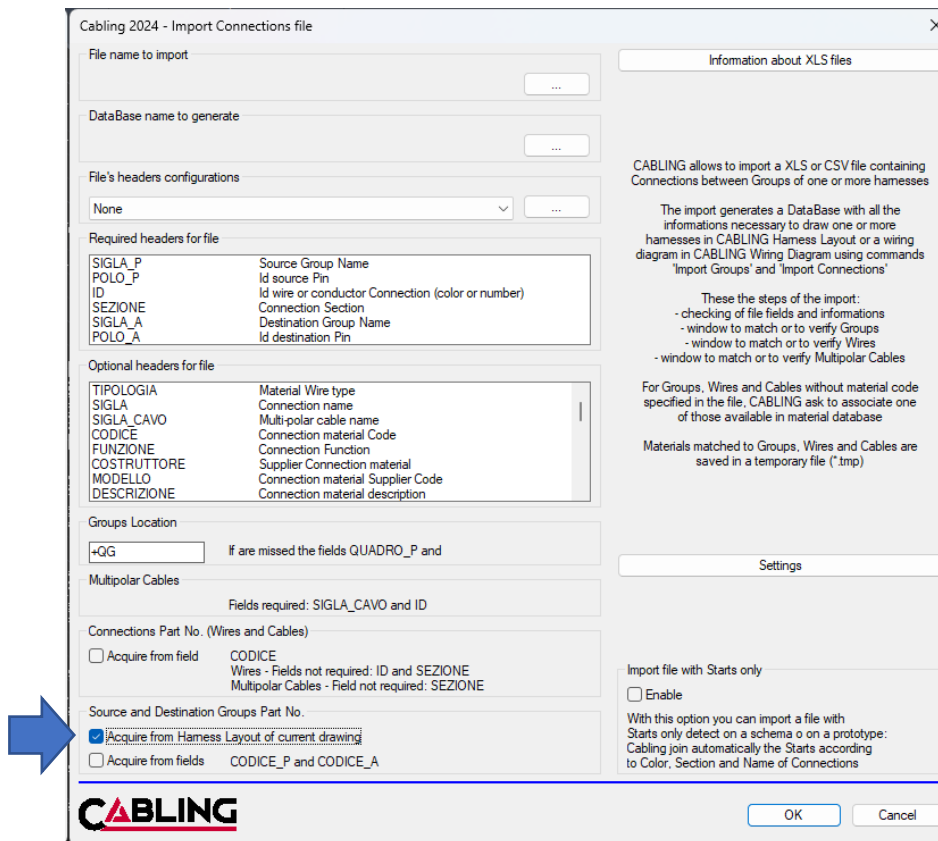
This procedure allows a high degree of freedom in the management of materials.



## 3 HARNESS LAYOUT

### 3.1 IMPORT GROUPS AND IMPORT CONNECTIONS

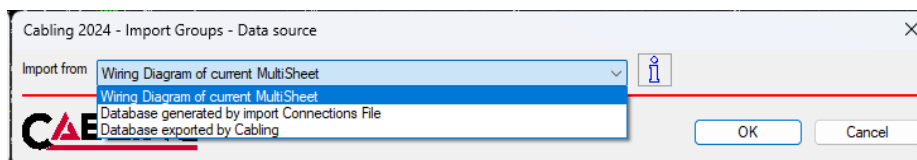
Importing the Connections file in the 'Import Groups' and 'Import Connections' command has a new checkbox that allows you to acquire the material codes of the groups from the harness layout of the current drawing.



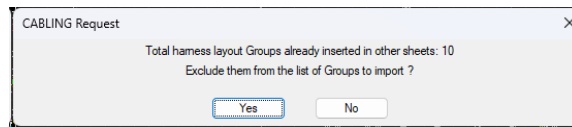
Enabling this mode acquires the material code of the Groups from the current drawing based on the Initials and the Panel. For this reason it no longer becomes necessary:

- Match the materials of the Groups in the appropriate window (which opens in any case)
- Enter the material codes in the fields of the XLS or CSV file that is imported.

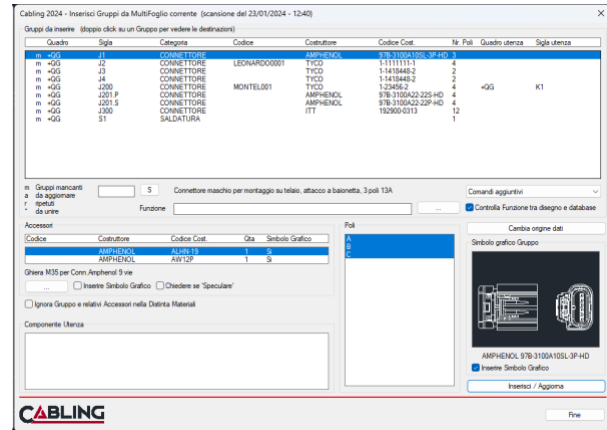
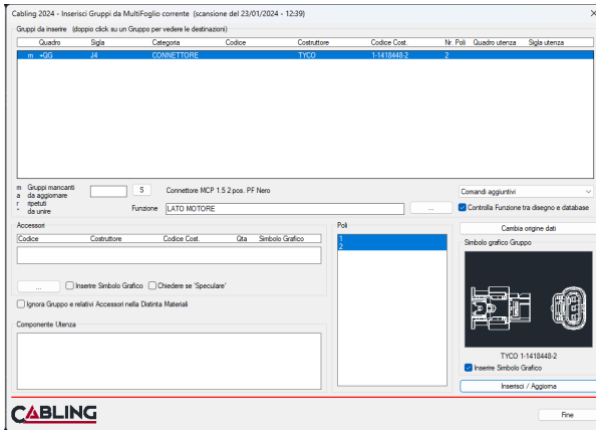
The import of Groups has been enriched with a new functionality to help designers in creating multiple cables/harnesses.



After pressing OK in the previous window CABLING checks whether Groups have been inserted in other Sheets other than the current one and if it finds any, asks the following question:



By pressing Yes (first image) the list of groups is reduced by excluding those already inserted in the other sheets, otherwise they are all displayed.

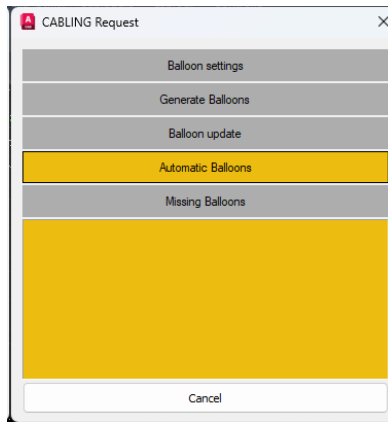


This allows you to progressively reduce the number of groups to be inserted in multiple sheets of the same Multisheet and speed up the design of the cablea/harnesses.

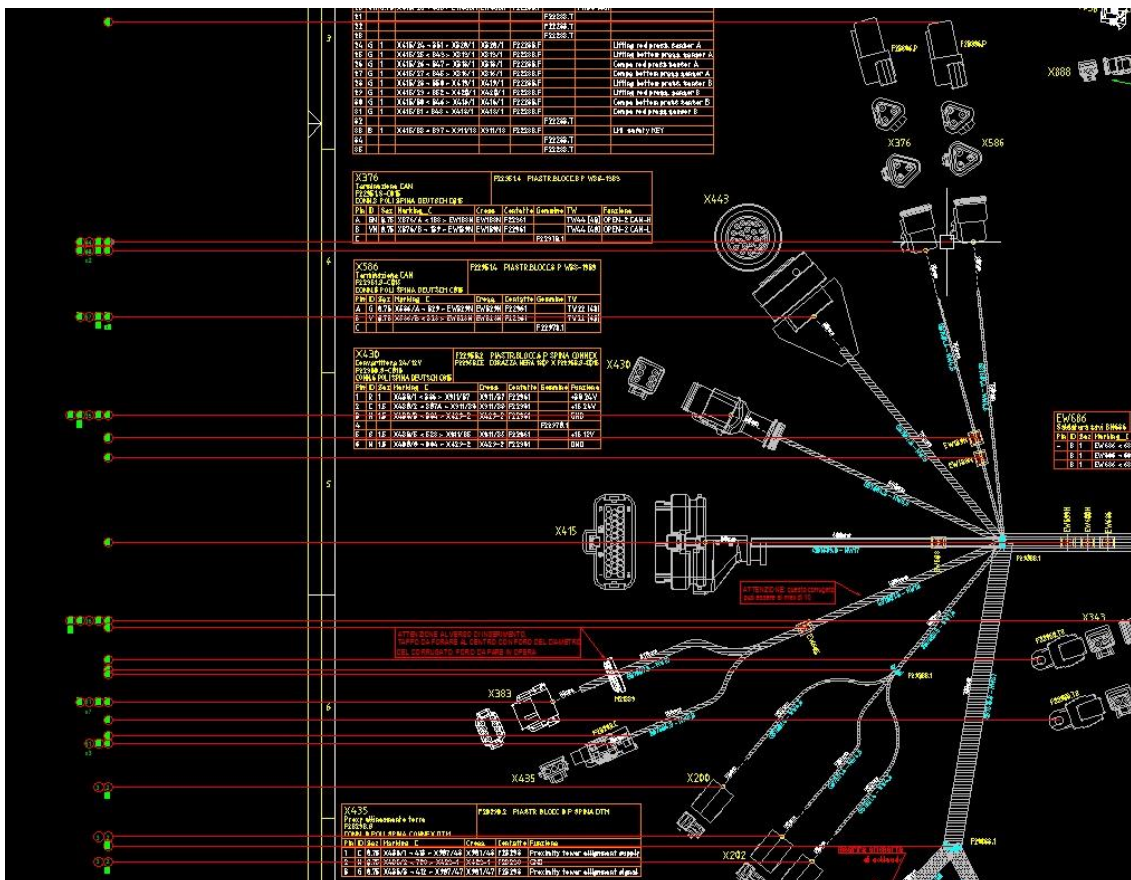


### 3.2 AUTOMATIC BALLOONS

The Balloons of the elements of the Harness Layout was completed with an Automatic mode.

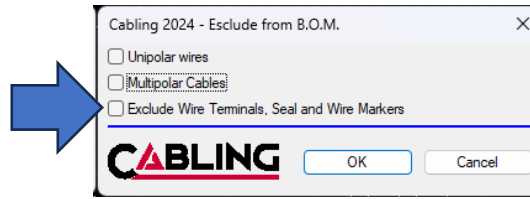


Pressing the right mouse button opens the contextual menu in which you will find the new item 'Automatic Balloons'. It will ask you to select individually or with boxes the entities you wish to balloon and, at the end of the selection, it will ask you to define one or more line segments from which the Balloons will be started with the lines reaching the selected elements. If you have selected elements that have already been marked, a message will be given and confirmation will be asked whether to continue, re-marking all the selected elements. Another step forward to speed up the design and also create a ordered drawing.

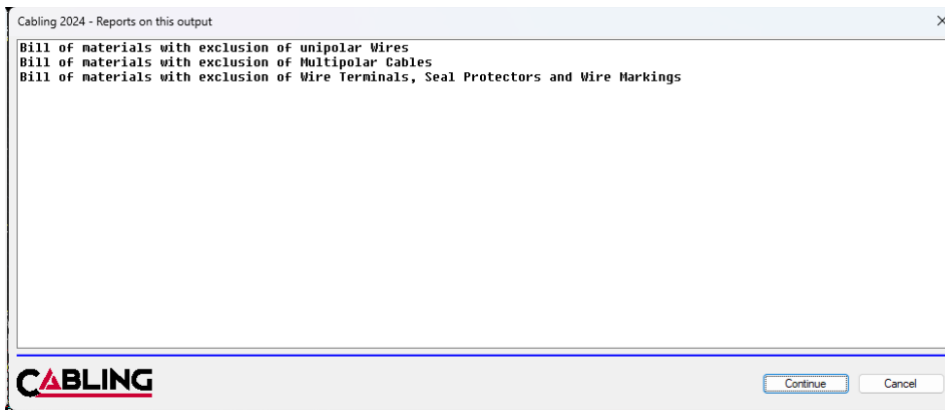


### 3.3 EXCLUSIONS IN THE BILL OF MATERIALS

In the bill of materials it is now possible to also exclude wire terminals, seal protectors and wire markers, materials that go together with multi-core wires and cables and with their exclusion.



The warning message is always given to remind you of the exclusions made.



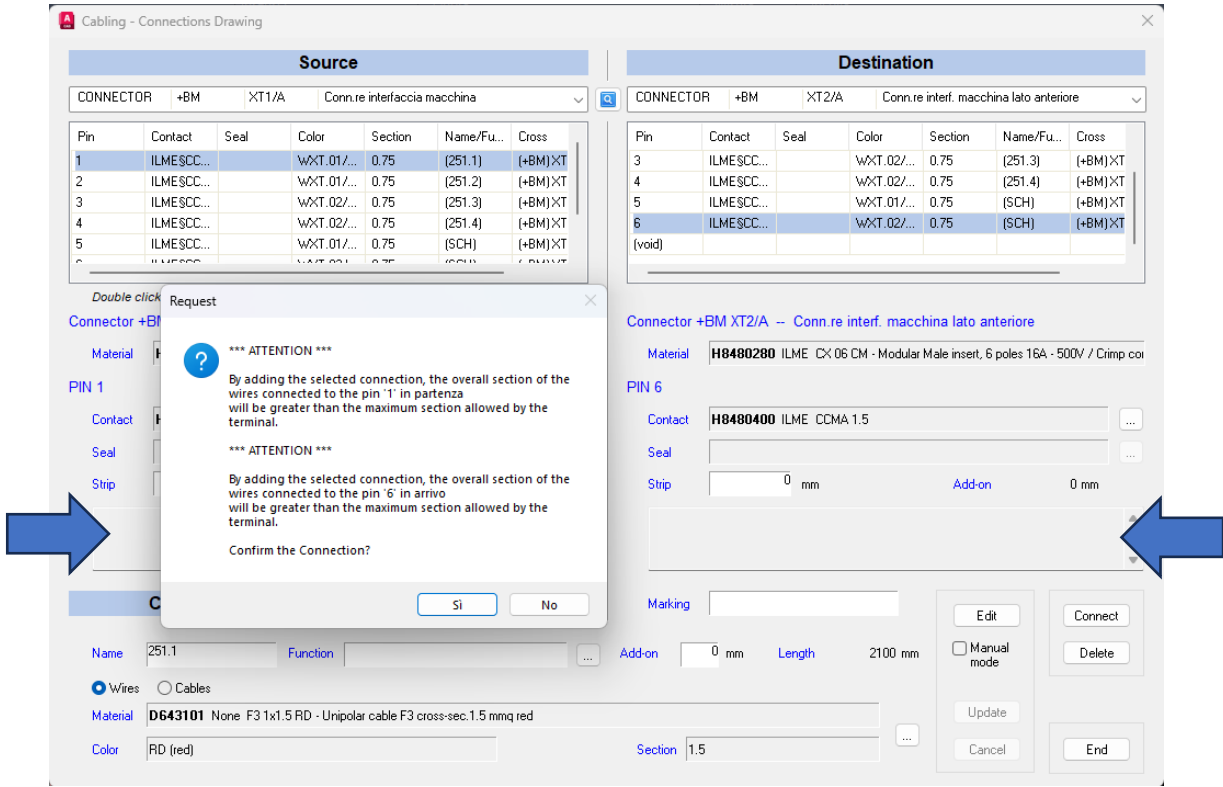
However, the harness labels are always counted.



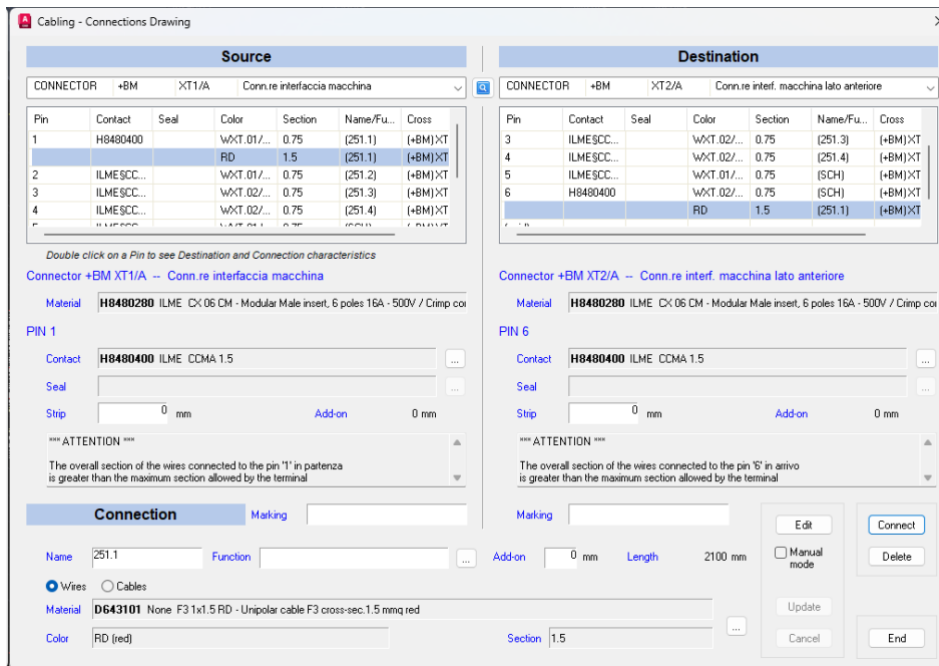


### 3.4 ERROR MANAGEMENT IN THE CONNECTION WINDOW

The connection dialog box remains unchanged but any error messages are displayed in a further window when the 'Connect' or 'Update' button is pressed.

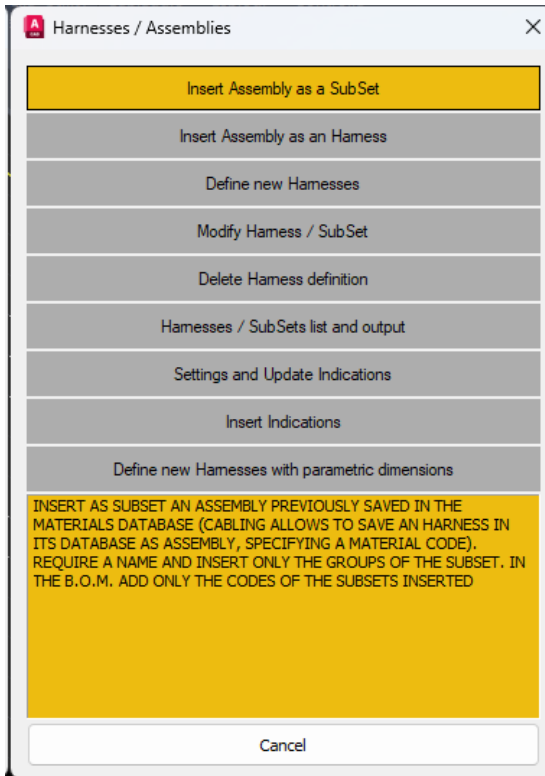


In this way there are no misunderstandings regarding the messages that appeared previously based on the choices on the main window (areas indicated by the arrows). Messages on these areas are now displayed on already existing connections, if there are errors (see image below).



### 3.5 HARNESSES AND SUBSETS MANAGEMENT

Some changes have been made in the Harnesses / Assemblies command to improve its functioning. Let's start by saying that the command is used to create/insert harnesses and insert SubSets. The first are layouts to which a name and other characteristics can be applied, the latter are synthetic layouts which are considered as a single object for the bill of materials.



Harnesses and SubSets have their own Harness Indication which you can insert into the drawing to identify them with the data you have configured with 'Settings and Updating Indications'.

Harnesses can be inserted or defined: in the first case, insert an assembly previously saved in the materials database, in the second, define the harness by assigning the data to a layout present in the drawing.

SubSets can only be inserted by choosing an assembly previously saved in the materials database.

You can modify the data of the Harnesses and SubSets, but in any case the associated Panel and Name that identifies them must remain unique.

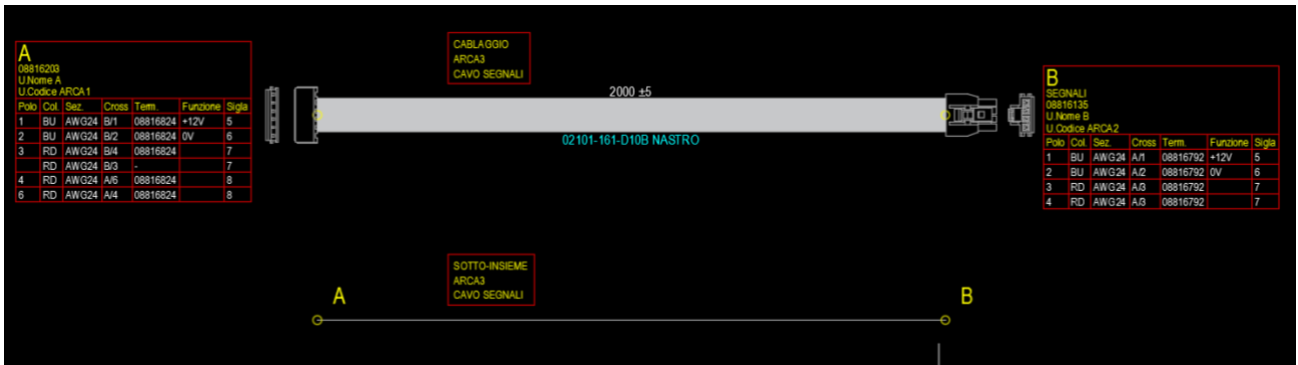
You can Delete the definition of a Harness, which returns to being a standard layout while if you select a SubSet you are asked if you want to delete it from the drawing.

In summary, this command allows:

- To insert an assembly as a harness with the related data
- To assign the data of a new harness to a drawing layout
- To insert a SubSet which is considered as a single object for the bill of materials
- To insert multiple SubSets to create a topographical view of the system, possibly also inserting the components to which they are connected
- To have a list of Harnesses / SubSets inserted in a window and to create output in a Table or in a XLS or CSV File formats
- The bill of materials can be divided by individual Harness.

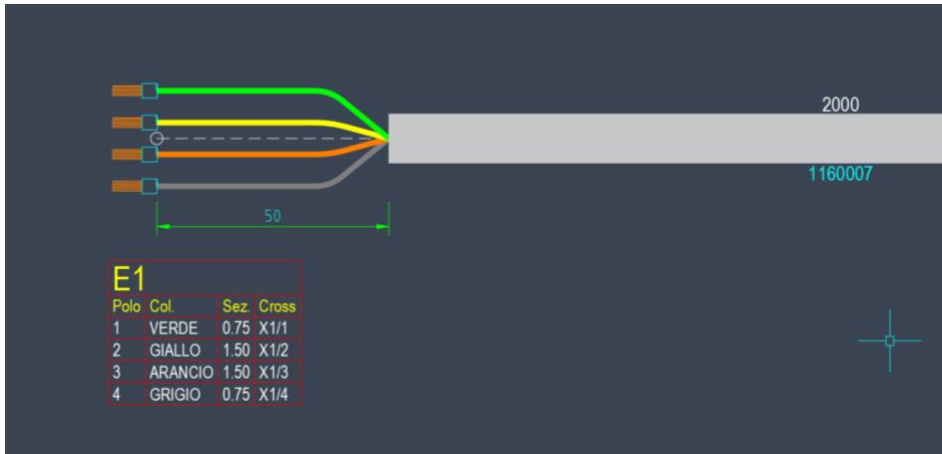
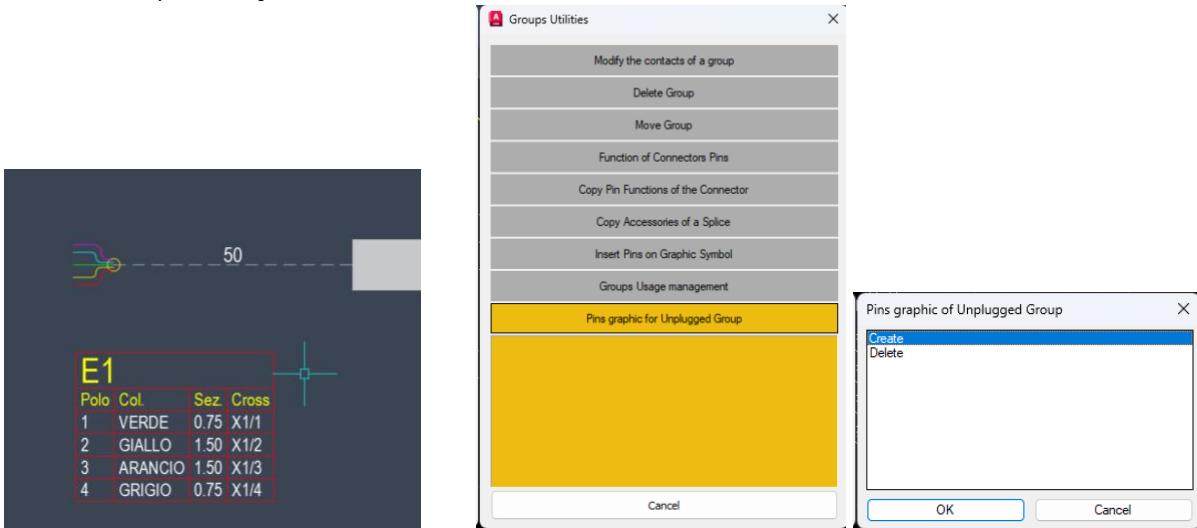
The main improvement was that the SubSets are seen together with the Harnesses, so you will see them in the Harness / SubSets List window and you can insert the associated indications. In the image below you will find the same assembly inserted as harness (in the upper part) and inserted as a SubSet (in the lower part), both with the associated indication.





### 3.6 UNPLUGGED GROUP PINS GRAPHICS

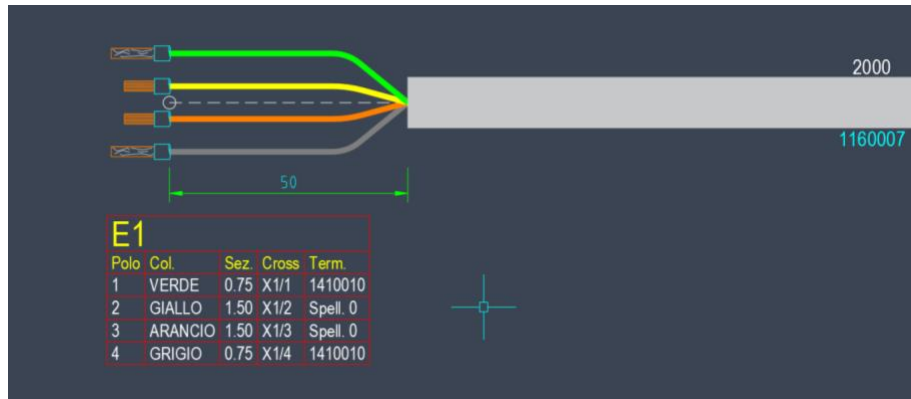
As an alternative to the usual symbol of the Unplugged Group, in CABLING 2024 it is possible to create new, more attractive and clear graphics with the menu command: CABLING Harness Layout → Groups Utility.



It is necessary that the graphic symbol is not isolated and connected to a straight Bundle without material, as in the previous images. For Creation you are asked to select the unplugged group, while deletion erases all previously created graphics.

In the presence of terminals, the graphics of the symbols placed at the ends of the colored lines are changed, as shown in the image below.





The old graphic symbol is deleted and the node and the adjacent Bundle, although visible in the drawing, are forced to a color that is not displayed during printing, thus avoiding erroneous interpretations during consultation.

Finally the dimension with the length is added and the connections, if the color has been defined as RGB in the CABLING material archive, are colored.

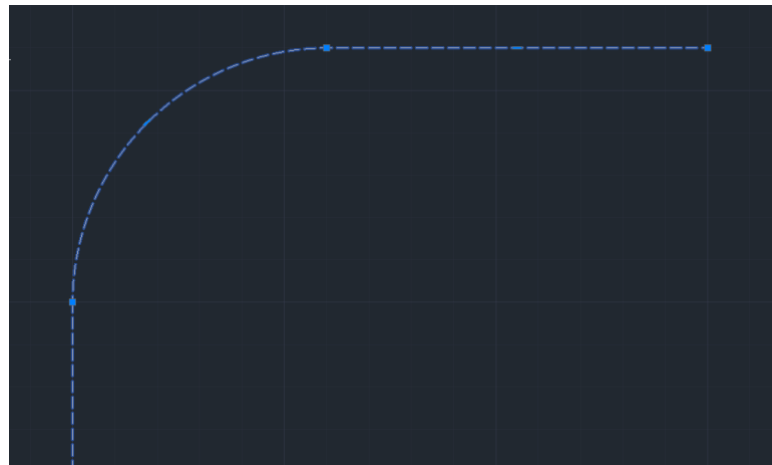
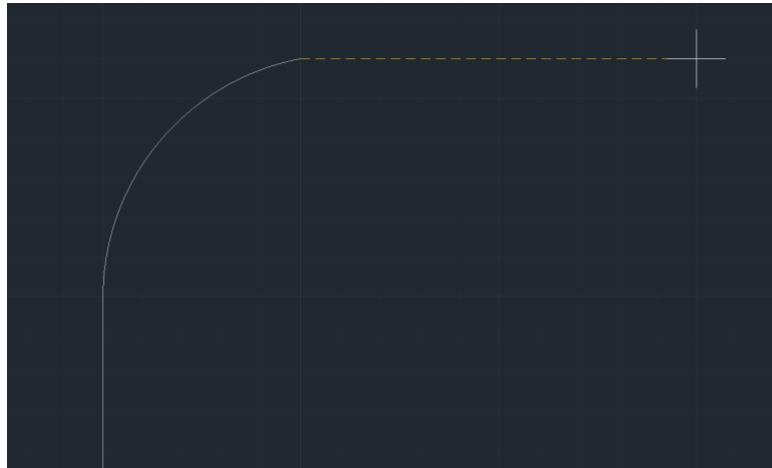
Connection diameters are not managed and are represented with a default diameter.



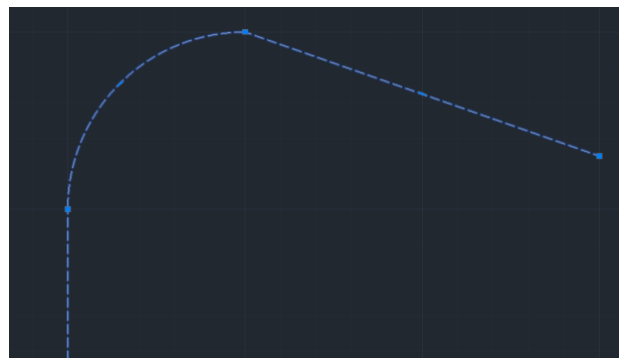
### 3.7 DRAW BUNDLES

**Very important innovation** in the drawing of the Bundles of the harness layout: after drawing an arc, the following happens:

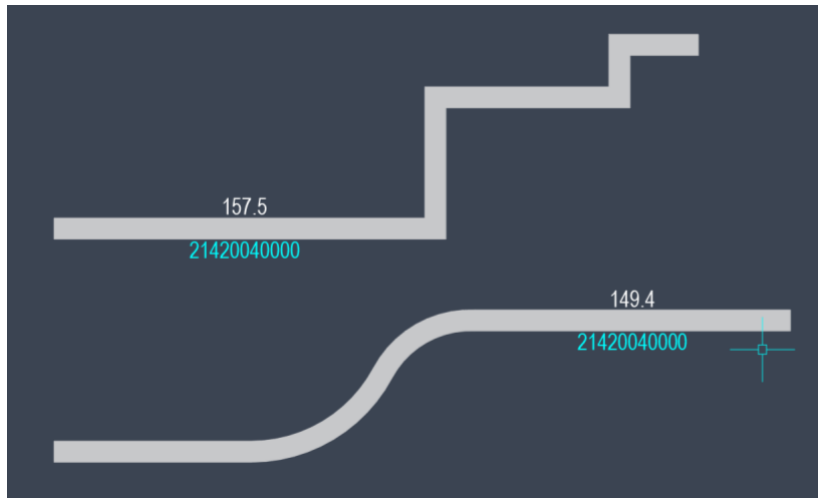
1. Cabling automatically returns to 'Line' mode (thus avoiding having to go back to the menu activated by the right mouse button)
2. If you draw a further linear segment, Cabling automatically modifies the arc to make it tangent to the last line (between the first and second images, notice that the point of the end of the arc and the start of the line has been moved a little to the right)



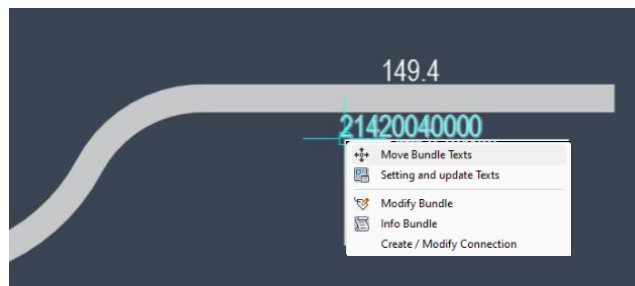
In this way we have **speeded up the design** and made the design of the Bundles that make up the harnesses more **correct and aesthetically valid**. The automatism does not occur if the angle of the line with respect to the end of the arc exceeds 15 degrees.



Finally, a new update for the design of the Bundles: the insertion point of the Bundles texts, which until now were inserted at the midpoint, is calculated in a new way.

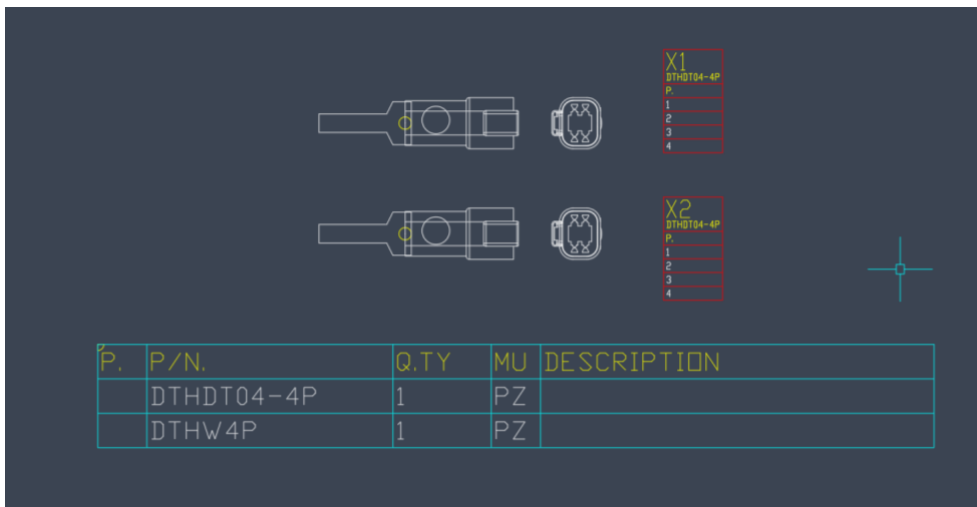
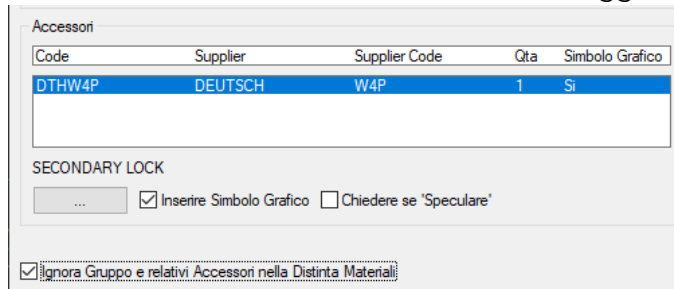


In a Bundle with only linear sections, the texts are inserted at the midpoint of the longest linear section, in a Bundle with linear and curved sections, the texts are inserted at the midpoint of the longest linear section. In all other cases the texts are inserted at the midpoint. The possibility of moving the texts is always available with the contextual menu option, which is recommended to be recalled with the mouse cursor on the start/insertion point of the texts (see image below).

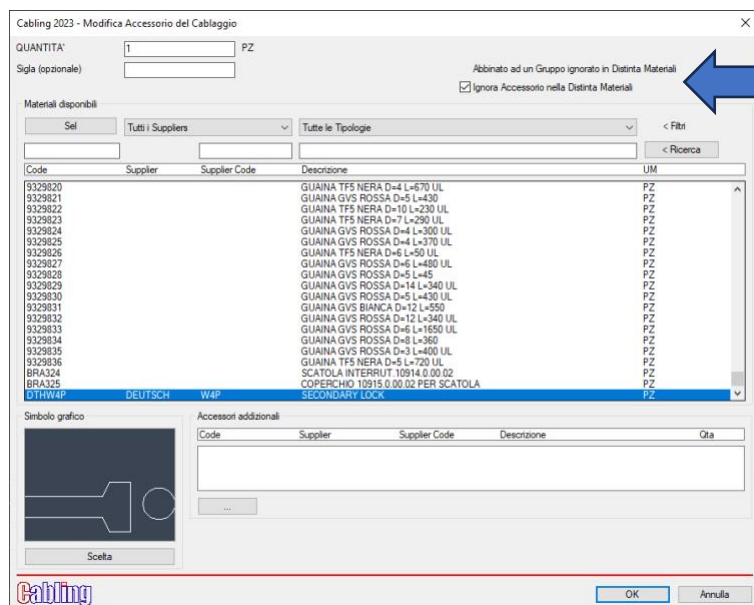


### 3.8 IGNORE ACCESSORIES IN BILL OF MATERIALS

The two connectors X1 and X2 in the example are the same and have a visible accessory which is counted in the bill of materials for X1 and not for X2: in fact, during the insertion of X2 we enabled the 'Ignore Group and related Accessories in the Bill of Materials' toggle.



To allow you to still have the X2 accessory in the bill of materials you can now edit the accessory and this window will appear where 2 new information has been added: that the accessory is associated to an ignored group in the bill of materials and the checkbox 'Ignore Accessory...' is enabled.

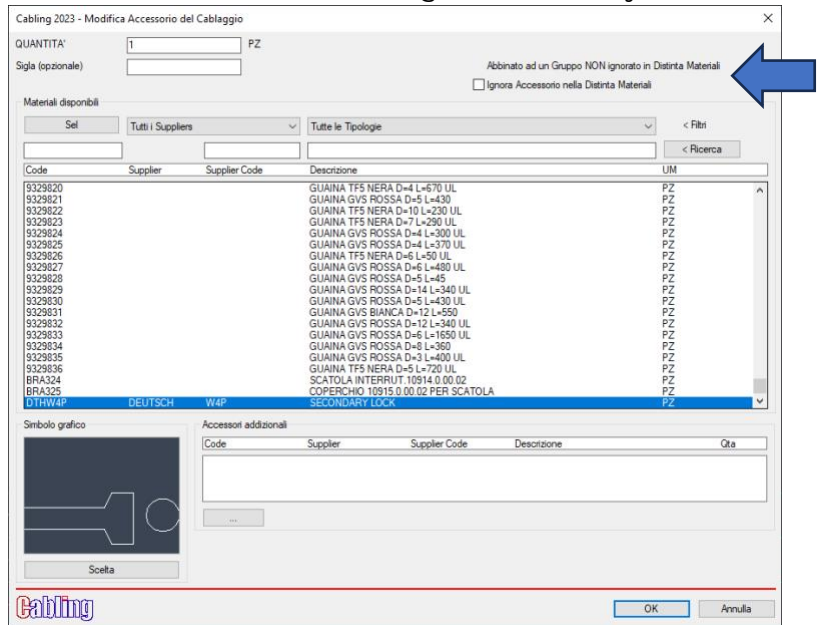




If you disable the 'Ignore Accessory...' toggle it is included in the BOM, and the connector to which it is associated continues to be ignored:

P.	P./N.	Q.TY	MU	DESCRIPTION
	DTHDT04-4P	1	PZ	
	DTHW4P	2	PZ	

Conversely, for the X1 connector that is not ignored, if you modify the accessory, this window will appear where 2 new information has been added: that the accessory is associated to a group NOT ignored in the bill of materials and the checkbox 'Ignore Accessory...' is disabled.



If you enable the 'Ignore Accessory...' toggle it is excluded in the BOM, and the connector to which it is associated continues to be displayed:

P.	P./N.	Q.TY	MU	DESCRIPTION
	DTHDT04-4P	1	PZ	



**3.9 GROUPS INDICATIONS**

Four new items have been added to the Group Indications:

- Strip
- Total Section
- Total Connections
- Pos Wiring Diagram
- Department

The 'Strip' appears alongside the 'Unplugged Group Strip' field which has been left for compatibility with previous versions. Inserts the stripping of any connection on any type of Group.

The 'Total Section' represents the sum of the sections of the connections on the individual pins, whether expressed in square mm or AWG. It is entered for each Pin only once on the first Connection.

<b>SU-7</b>								
Totale Collegamenti: 7								
Polo	Col.	Sez.	Cross	Funzione	Sigla	Spell.	S.Tot.	
-	G	0.5	C200/7	+15	79	12.5	3.5	
	G	0.5	C231/7	+15	89	12.5		
	G	0.5	E.500LA/3	TABLET +15		12.5		
	G	0.5	C300/7		98	12.5		
	G	0.5	C231/12	+15	95	12.5		
	G	0.5	F001A/19	+15 PROT_TORCIA POSTERIORE	143	12.5		
	G	0.5	C080/2	TORCIA CASSETTO	56	12.5		

The 'Total Connections' is inserted at the top of the table with the information of how many Connections the Group has, whatever type it is.

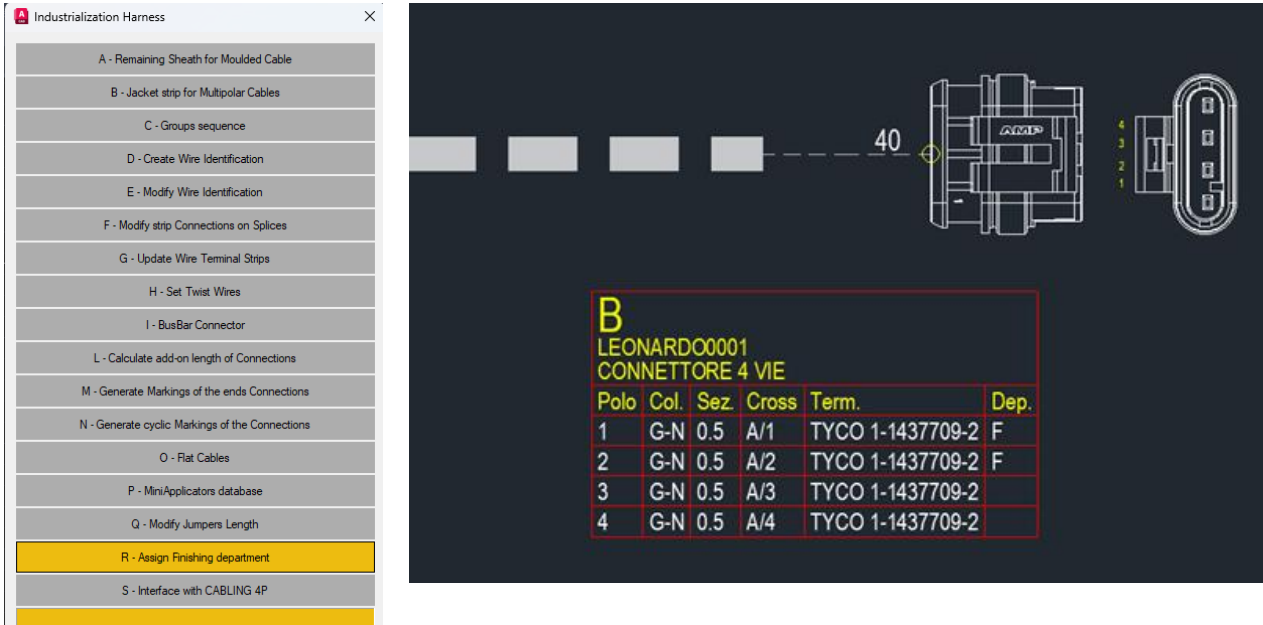
The 'Pos Wiring Diagram' defines the Sheet.Segment position of the individual Pins of the Group on an wiring diagram created with CABLING 4D. It represents the possibility of consulting the wiring diagram more quickly starting from the layout drawing of an harness.

Finally, the 'Department' displays the assignment of one or more pins (excluding splices and moulded cables) to the Finishing department. This information is especially useful for harness manufacturers as it indicates that the terminal associated to the pin will have to be crimped onto the wire in the finishing department with manual machinery and not with automatic cutting/crimping machines.

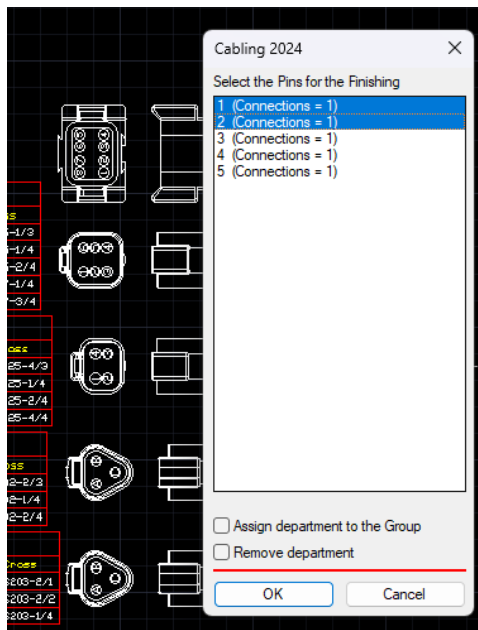


### 3.10 ASSIGN FINISHING DEPARTMENT






The Department is especially useful for wiring manufacturers as it indicates that the terminal tied to the single pole will have to be crimped onto the wire in the Finishing department with manual machinery and not with automatic cutting machines.



The department that can be associated is the Finishing (with the letter F for indications and outputs). With the 'R - Assign Finishing Department' command you are asked to cyclically select a group (with the exception of splices and moulded cables) and a window with the list of pins is opened. You can select no pin if you want to delete the department, all the connector pins or the desired ones. By pressing 'All Pins' the entire Connector goes to the Finishing department. The assigned department is saved and the information can appear in the Group indications, if the 'Pin Department' (Dept) field is present.



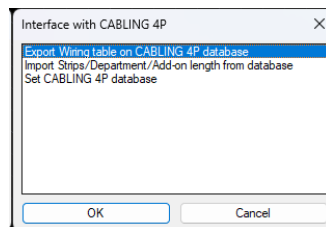
The department information is also managed in numerous other outputs, including the wiring table both in XLS and CSV files and in tabular and graphic form (see example).

TABELLA CONFORMAZIONE CAVI / CONFORMATI				
PARTENZA/SOURCE		FUNCTION	CABLE NAME	LENGHT (mm)
	A/1		G-N Cavo AEX G-N 0.5f MMQ 120° PE	1130
			G-N Cavo AEX G-N 0.5f MMQ 120° PE	205
	A/2		G-N Cavo AEX G-N 0.5f MMQ 120° PE	1130
	A/3		G-N Cavo AEX G-N 0.5f MMQ 120° PE	1130
	A/4		G-N Cavo AEX G-N 0.5f MMQ 120° PE	1210

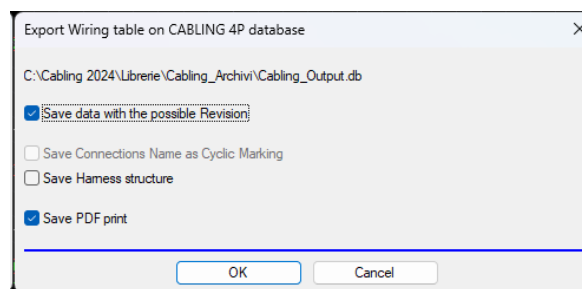
The Department information is also exported for CABLING 4P (for Production), the new program for the intelligent management of cutting of wires using automatic cutting/stripping/crimping machines.

The program, independent of the AutoCAD platform, allows you to manage the Finishing Department by excluding automatic crimping on the machines of the pins to which it has been assigned, a process that can be carried out subsequently in the finishing department with manual machines (bench presses, pliers...).

The Department can also be modified in CABLING 4P and can be sent back to CABLING 4D via the new command 'S - Interface with CABLING 4P' which contains three functions:



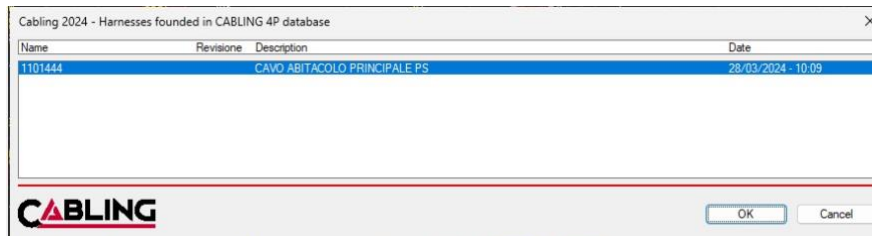
- **Export wiring table for Cabling 4P:** allows you to export the wiring table from a CABLING 4D harness layout, including any assignments to the Finishing department.



The dialog box allows you to choose what and how to export the harness data designed with CABLING 4D. It is also possible to export The harness structure to allow changes to the structure even in 4P CABLING 4P (Production). This topic will be explained in detail in separate documentation.



- **Importa Spellature/Reparto/Ricchezza da Database:** permette di importare queste tre informazioni nel layout costruttivo corrente. Il layout verrà aggiornato in modo tale che una eventuale modifica consenta una esportazione della tabella di cablaggio aggiornata. La seguente finestra permette di scegliere il cablaggio di CABLING 4P da importare.
- **Import Strips/Department/Add-on Length from Database:** allows you to import these three information into the current harness layout. The layout will be updated so that any modification allows an export of the updated wiring table. The following window allows you to choose the CABLING 4P wiring to import.



- **Configure export for Cabling 4P:** allows you to define the name and position of the database managed by CABLING 4P. Normally it coincides with the Cabling\_Output.db database that you find in the Cabling\_Archivi folder of your Path Libraries.



### 3.11 DRAW BUNDLES

In some situations it is possible that the Graphics Bundles command creates errors as in the image below.

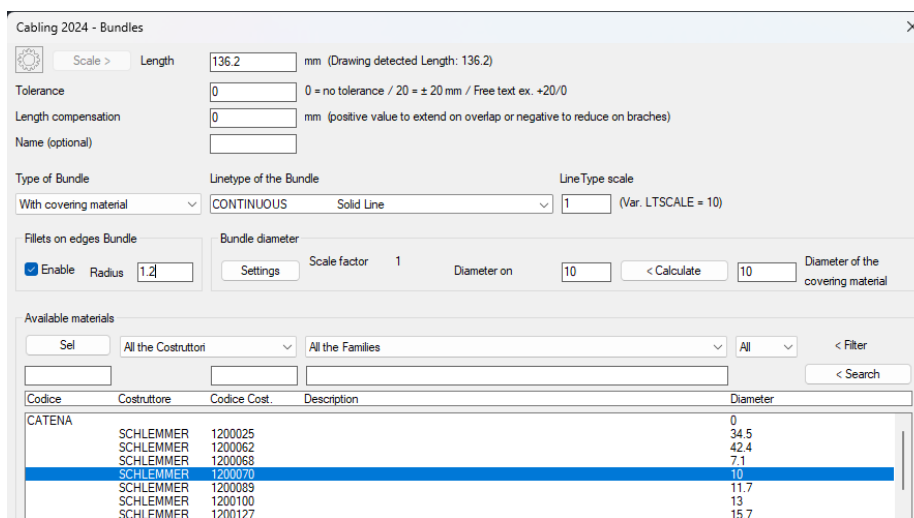


This is due to the fact that on the vertices of the bundle with radius situations similar to the zoomed image below can be created.

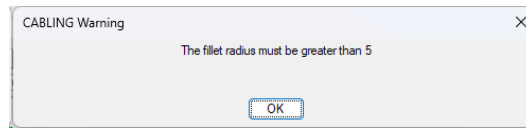


There are two solutions in the dialog:

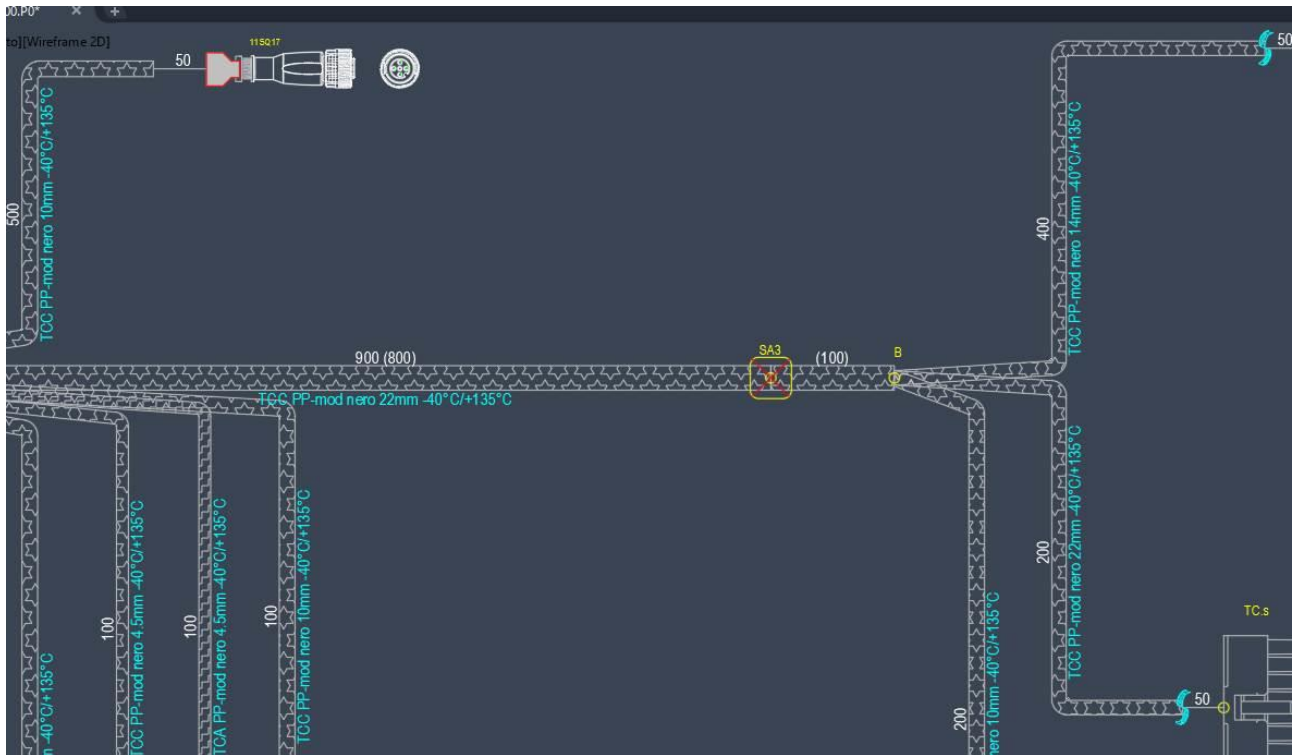
- Increase the fillet radius
- Decrease the Bundle diameter.



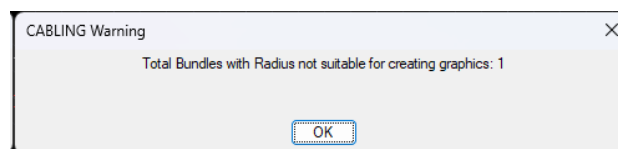
Now by pressing OK in the dialog box, if the diameter is not sufficient, this alert is given:



The underlying vertex is connected correctly so the bundle graphics will be correct.

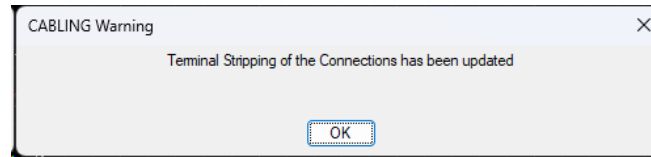


During the creation of the Graphis of Bundles with materials, in order not to block the command and create errors, bundles that have unsuitable curvature radius are automatically discarded. In this case the following error message is given:



### 3.12 CHANGE OF A TERMINAL

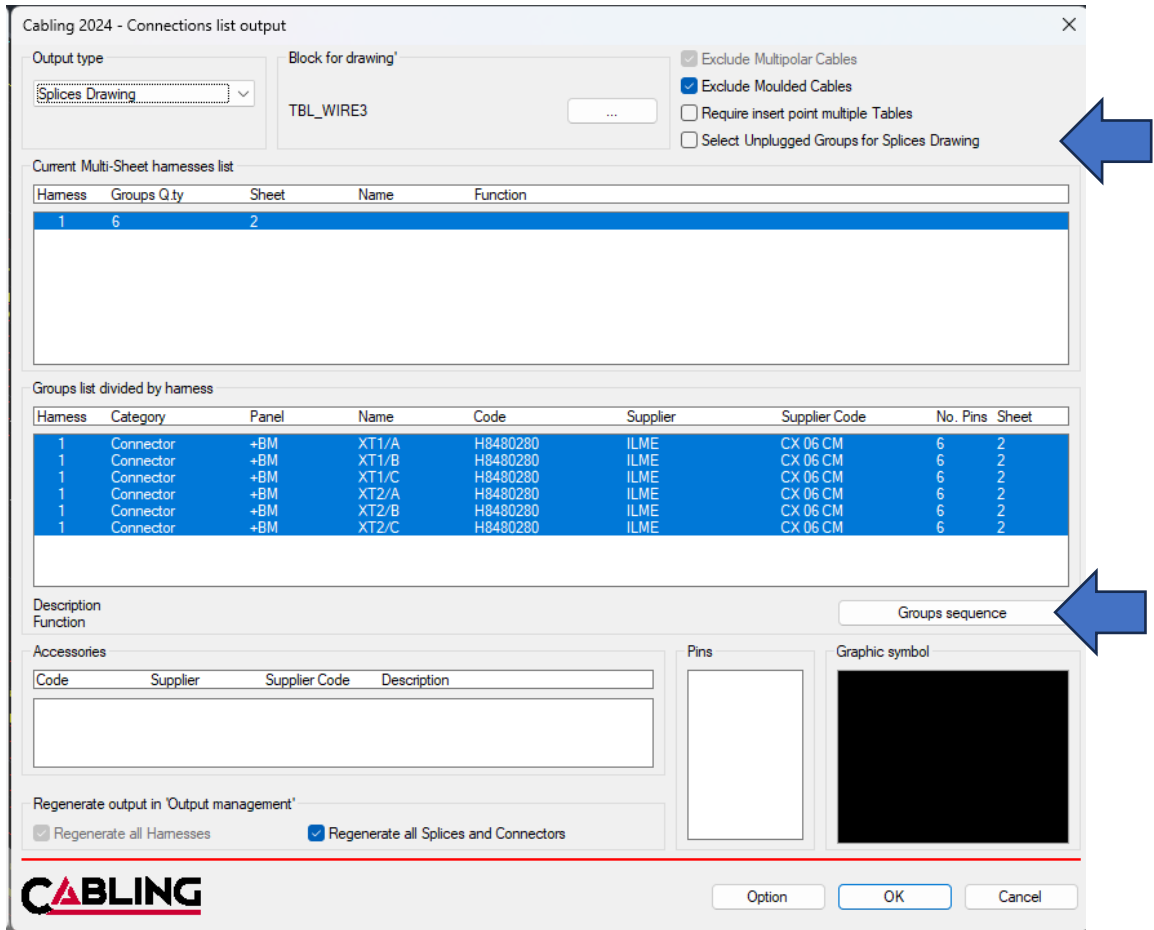
If you edit a terminal and replace it with another that has different strip, it is updated on all connections joined to the terminal. At the end the following message is given.



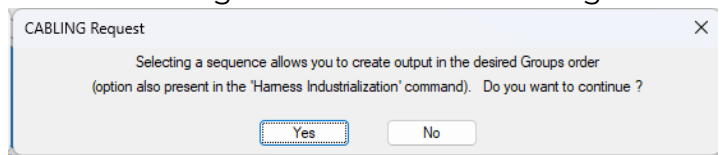


**3.13 CONNECTIONS LIST OUTPUT**

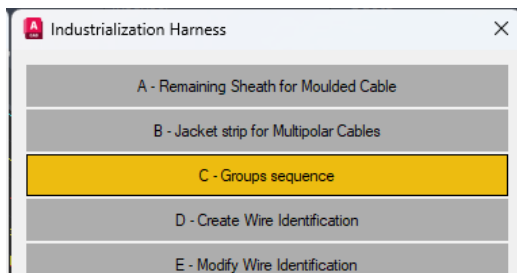
The 'Select Unplugged Groups for Splice Drawing' checkbox has been added to the main window, thus avoiding the next question after closing the window. The 'Group Sequence' button has been added which allows you to select the group sequence with which to perform the output, valid for the 'Groups' and 'Groups and Junctions' sorting.



You have to confirm before closing the window and starting the selection of Groups.



The command is identical to the one found in 'Industrialization Harness'.

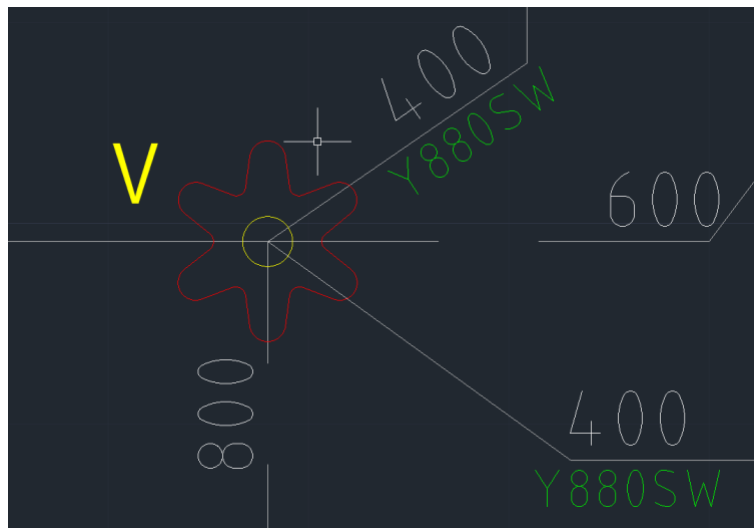


### 3.14 GRAPHIC SYMBOL OF BRANCH

If you wish to insert a graphic symbol representing a branch, you can now do so. After inserting the branches you can use the 'Harness Layout graphics' command with the 'Update groups graphic symbol' option.

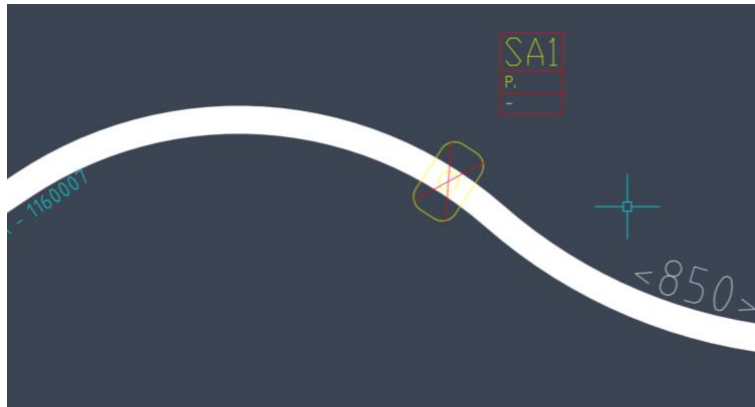
Selecting the node of a branch will insert the BL-BRANCH.DWG block which must be present in the Libreria ..\Librerie\Cabling\_Blz folder. If the block is not found a message is given on the AutoCAD command line.

As with other graphic symbols, it is deleted if you delete the associated branch node and you can use the associated context menu to change the abbreviation and function of the branch.

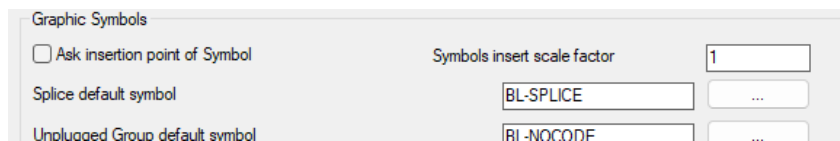


### 3.15 INSERIMENTO SALDATURE E COMPONENTI LUNGO I PERCORSI

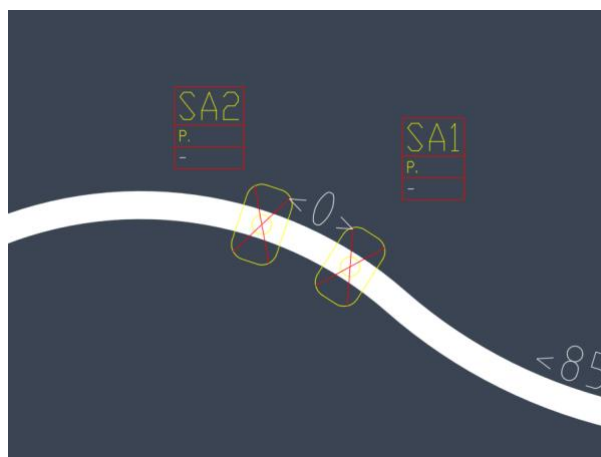
The insertion of a splice, which now has a new, narrower graphic shape to allow more than one to be placed close together, is done by aligning with the Bundle at the insertion point. The same thing goes for the components that must be placed along a sheath (diodes, resistors...).



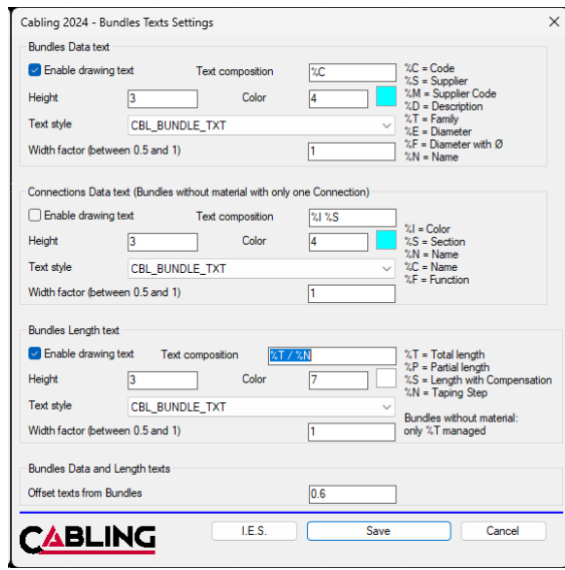
To change the default splice graphic symbol you can use the 'General Settings' button in the Harness layout settings and use the new BL-SPLICE symbol.



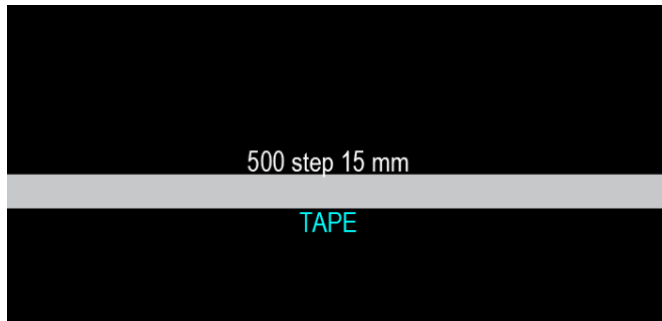
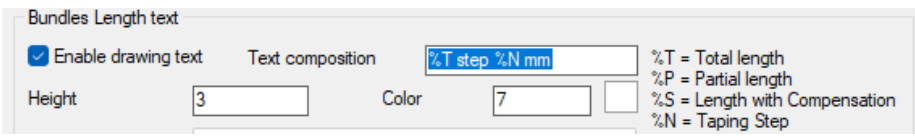
Furthermore, it is now possible to force the length of Bundles with materials to zero (previously this was not possible). Obviously the zero length Bundles are not counted in the bills of materials and in the sheaths cutting list but they can be useful for the 'Splices Table' output.



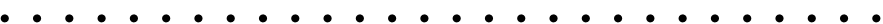
**3.16 BUNDLES TEXTS**



In the Bundle Texts settings it is now possible to add the %N taping step parameter which appears in the length texts if the bundle is of the taped type and in which the step value is important to display.



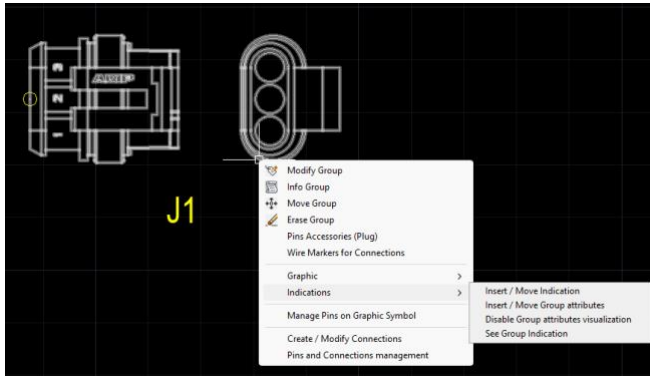
In the absence of taping, the 'step' and 'mm' texts are automatically removed from the indication, which will only show the length (500)



## 3.17 GROUP ATTRIBUTES MANAGEMENT

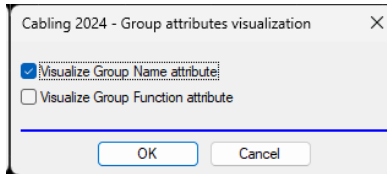
The management of Groups attributes has been changed to make operation easier.

- **Contextual menu**

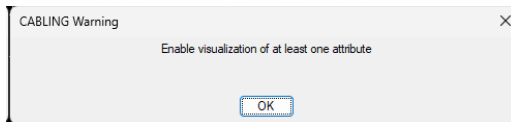


There are now two options:

- **Insert / Move Group Attributes**  
If they are missing, it allows you to insert the Group Name and Group Function attributes with a new window, regardless of the settings:



Instead, the saved settings for the textual characteristics of the two attributes are used. In previous versions the following message was given if no attributes were enabled.

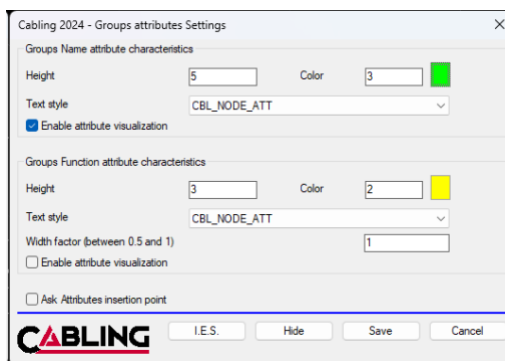


- **Disable Group Attributes display**  
Allows you to disable the display of all Group attributes.

- **Editing a Group**

Modifying a group does not involve the automatic insertion of the enabled attributes as happened until now.

- **Group Attributes setting windows**



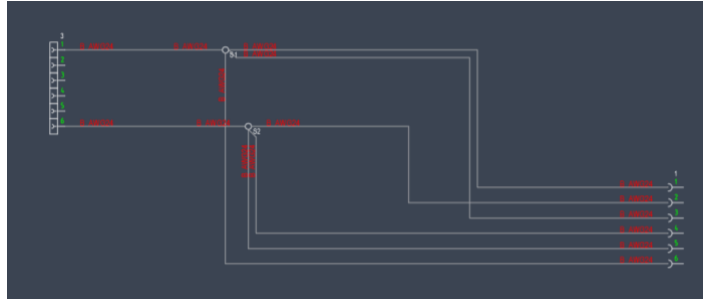
The window has been simplified by removing two checkboxes. Now when you are asked to display the attributes of the Groups even on just one node they are activated on all but the display and characteristics of the text (height, color...) become independent for each node.



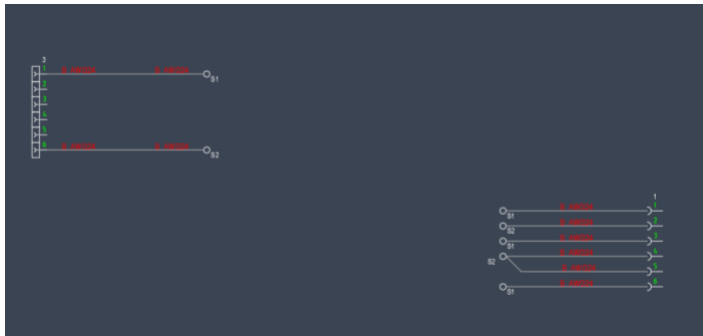
# 4 WIRING DIAGRAM

## 4.1 SPLICES AND WIRE TERMINALS INSERTION

The example of an electrical diagram below represents the standard (still available) with which it is possible to insert solders and wire terminals and connect them to the destinations.



The example of an electrical diagram below is the same as the previous one but with a new method that allows the same splices and wire terminals to be inserted several times aligned with the destinations. The new mode makes it easier to draw and understand connections. In the outputs and in the harness layout, all repeated soldering and terminals will be considered as a single node, making the two methods identical as electrical schema.



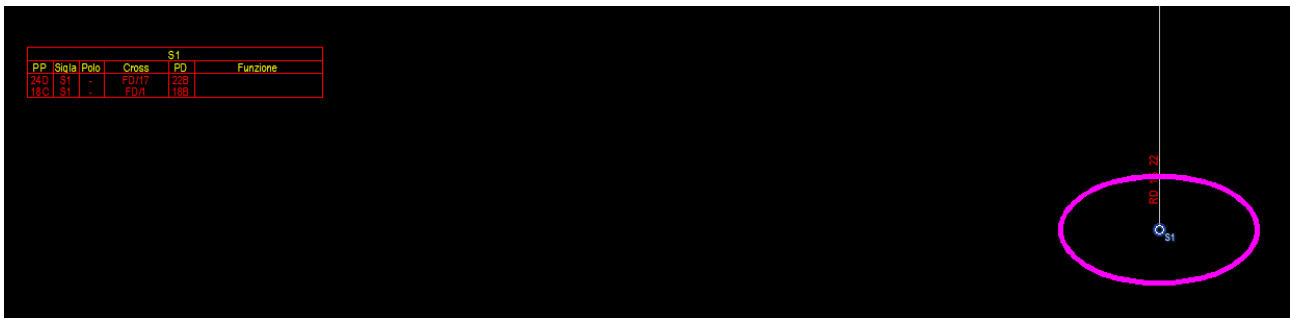
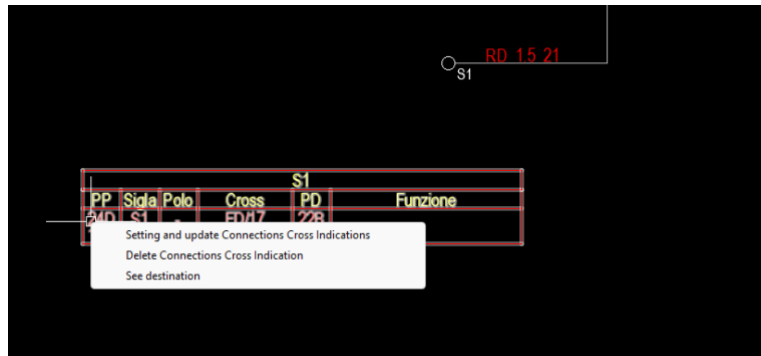
To simplify consultation of the wiring diagram you can create tables for each individual splice/wire terminal with a summary of the `_MORS*` blocks inserted and the relative Sheet.Segment position. The Sheet.Segment give the position of splices/wire terminals in the schema using the master title block divisions.

PP	Sigla	Polo	Cross
24D	S1	-	FD/17
18C	S1	-	FD/1

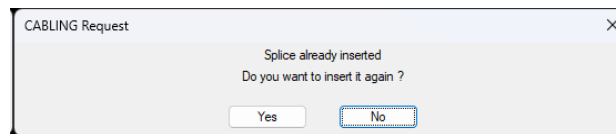
Just use the 'Cross Connection Indications' option which behaves differently on Splices and Terminals compared to other types of Groups.

After inserting the tables on the drawing they can be moved, deleted, regenerated and if you select one of the positions you can view the corresponding `_MORS*` block (See destination), facilitating consultation of the diagram.

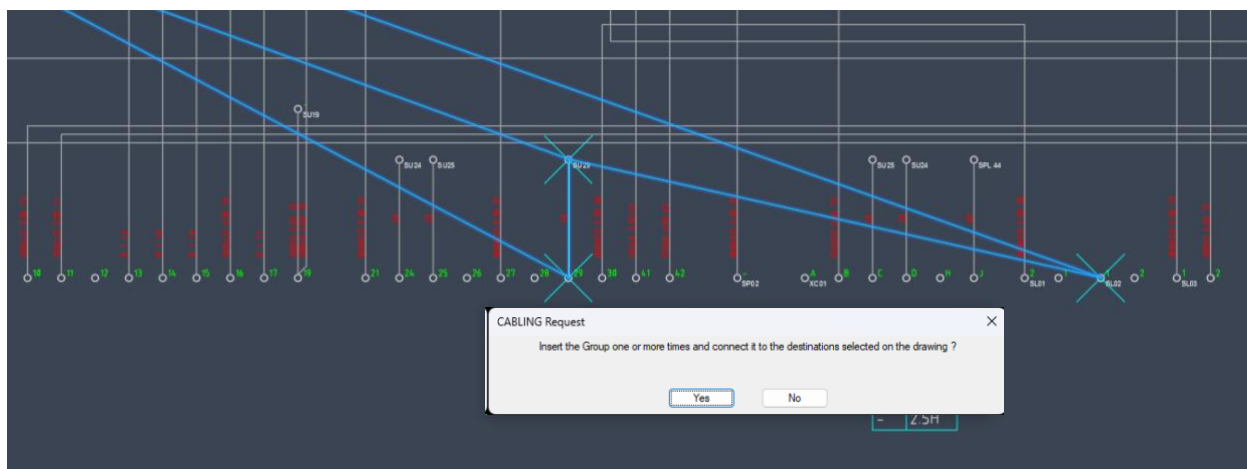




Also if you **import groups from an harness diagram or from a file**, the insertion of a wire terminal or splice group can be carried out several times in the electrical diagram to simplify the connections, even subject to a confirmation request.

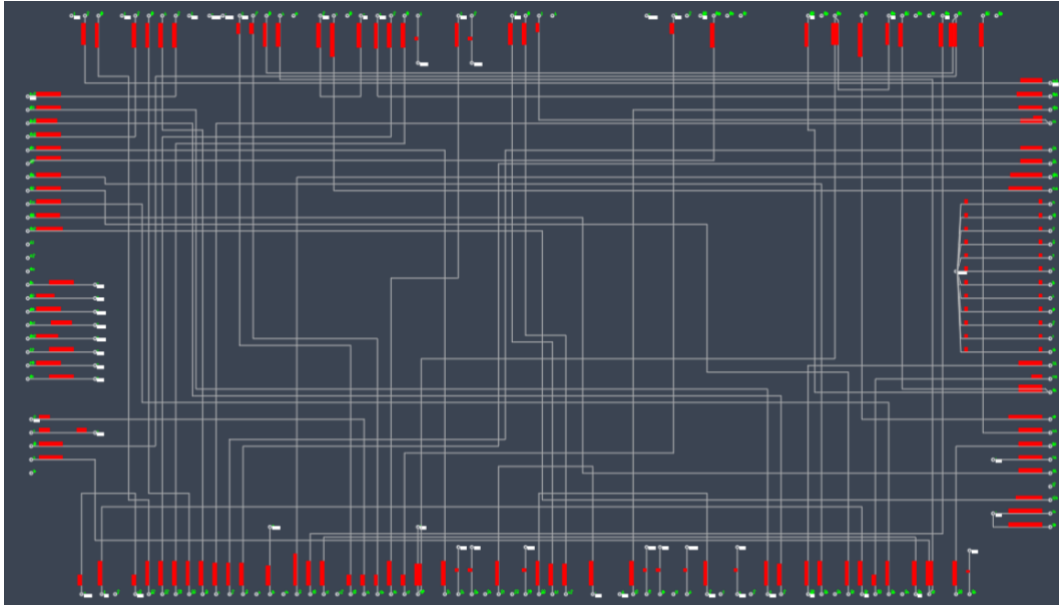


To facilitate the positioning of wire terminals and splices in the correct position in order to be connected to the destinations, they are highlighted together with the theoretical connections (in blue) and confirmation is asked to insert and connect them to the destinations selected on the drawing.

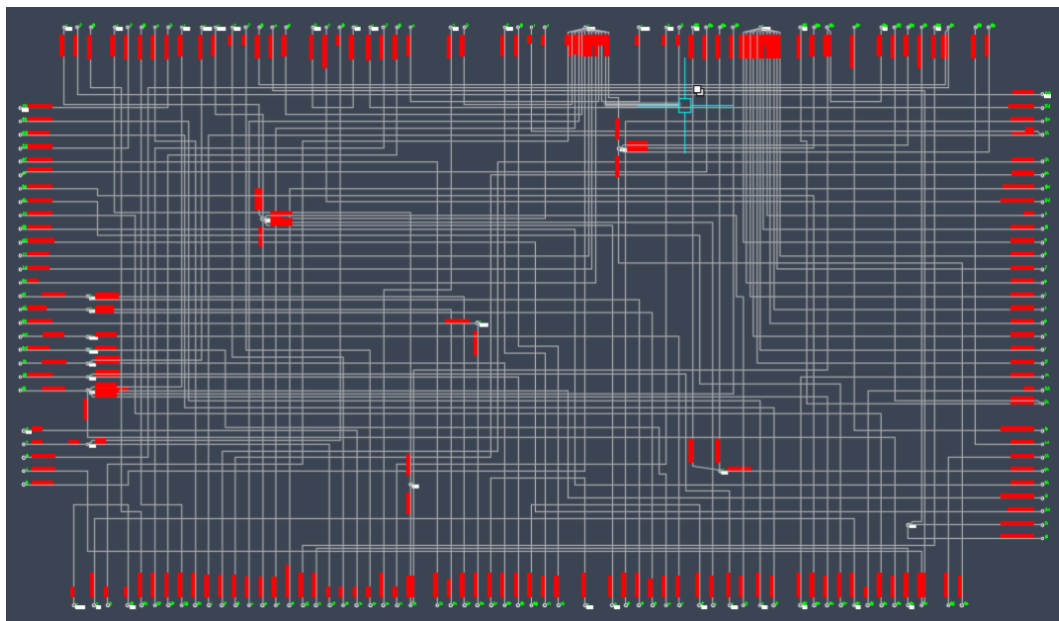


You are asked to "Select one or more of the destinations highlighted on the drawing" and after inserting the wire terminal or splice `_MORS*` block it is possible to connect it to the selected destination(s). The request is repeated unless all targets are connected. This automates the process and speeds up design.

This is a drawing with the new solution:



The diagram below is identical but with a single insertion of Wire Terminals and Splices which are connected to the respective destinations.





**4.2 INSERTING COMPONENT SYMBOLS**

In CABLING 4D 2024 we have given specific answers to each type of electrical diagram you wish to create.

Until now:

- Possibility of inserting symbols associated with their definition with a code in the materials database.

The main new features of the 2024 version:

- Possibility of associating more than one symbol with a material code
- Insert graphic symbols without them being combined with a material code
- In the case of multiple graphic symbols associated with the same acronym, management of the Mother component with one or more Children
- Cross Reference indications to be inserted next to the Mother to list the children included in the scheme
- Navigation between children and mother and between cross indication and children
- It is no longer necessary to insert the free groups that represented the component inputs.

All this makes CABLING the most flexible system for designing an electrical diagram based on your technical needs.

Graphic symbols are divided into two main categories:

IEC, JIC, VDE or Electronic Standard graphic symbols	USER BLK graphic symbols
They are graphic symbols already present in the CABLING 4D installation and are in the respective IEC, JIC, VDE or Electronics folders in the Libraries directory	They are symbols that the user can create and are saved in the BLK folder (or in one of its subfolders) in the Libraries directory

The graphic symbols of both categories can be associated with a code in the definition of the components in the material database and subsequently inserted into the drawing or inserted freely onto the drawing without being combined with a material. It is possible to create diagrams with mixed graphic symbols, some associated with a material code and some not.

By inserting only the graphic symbols without materials you will obtain a basic electrical diagram, with the graphic symbols combined with the materials a complete electrical diagram.

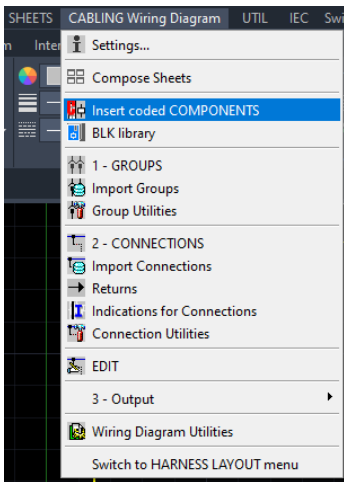
All the graphic symbols inserted are identified in the diagram by two information:

- **Name**
- **Panel**

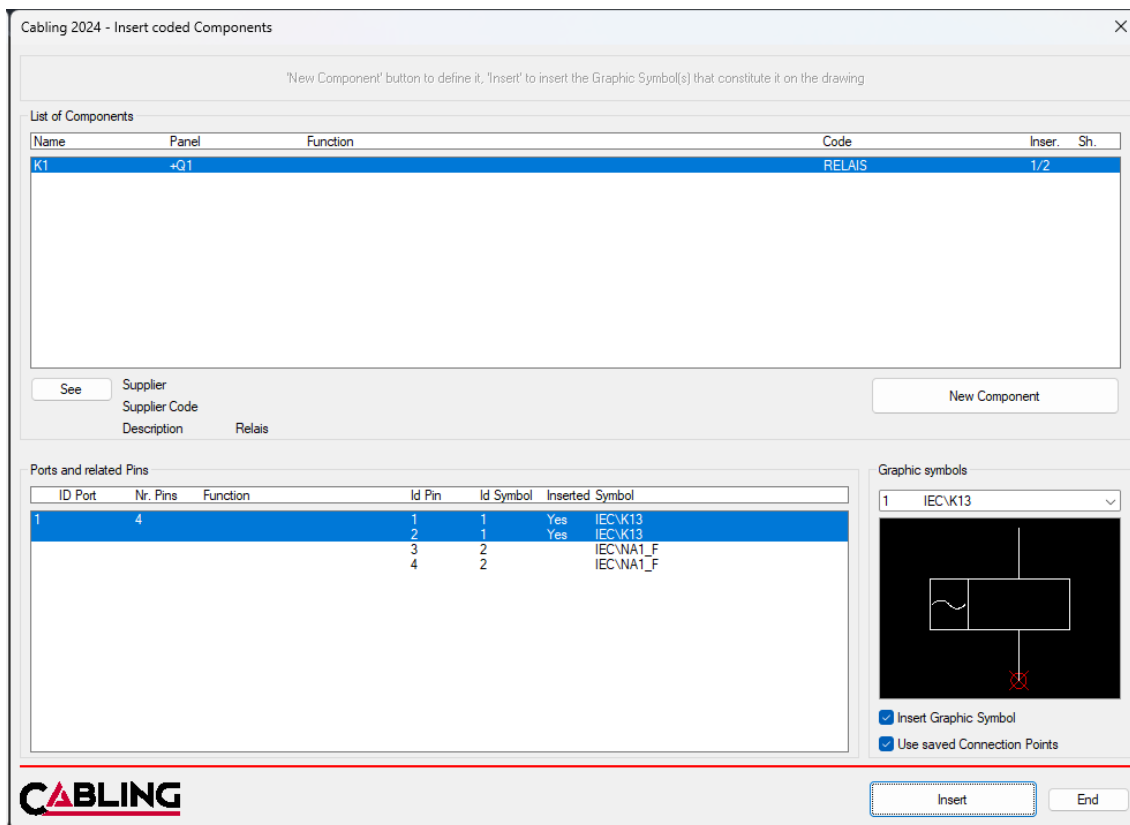
Inserimento Componenti codificati	Inserimento simboli grafici senza materiale
Inserimento di componenti generici con un unico simbolo grafico associato oppure di componenti con più simboli grafici di cui il primo viene considerato 'Madre' e i successivi 'Figli'	Inserimento di simboli grafici senza materiale associato. Potete inserirne uno solo come componente generico oppure più di uno di cui il primo viene considerato 'Madre' e i successivi 'Figli'
L'inserimento parte dal comando 'Inserisci COMPONENTI codificati' del menu	L'inserimento parte dalla scelta del simbolo grafico nei menu IEC, JIC, VDE o Elettronica oppure dal comando Libreria BLK per i simboli Utente



## Insertion of components with Code



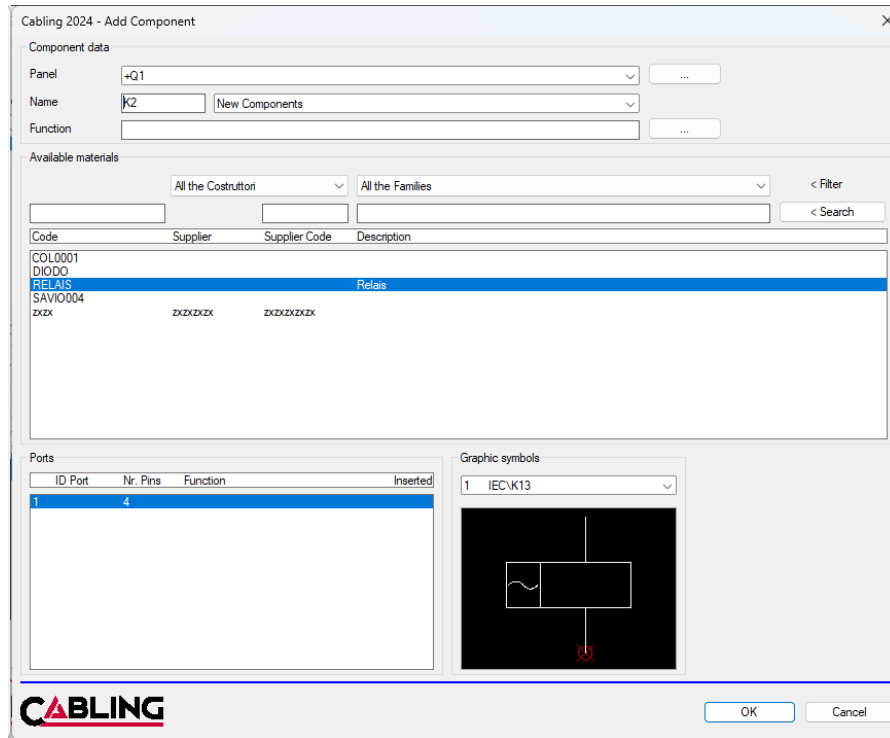
The main window that is opened cyclically for the insertion of components, whether generic or Mothers with Children, is the following:



It lists all the components inserted in the current Multisheet with the name, the panel, the function, the material code (if present) and the sheet in which they were inserted.

With the 'View' button you can see the selected component in the list, with 'Cross Indication' you can insert or move the Cross indication of the selected component (if it is a 'Mother'), with 'New Component' you open the window below where you can choose a material from those defined in the database and indicate the component name and panel.



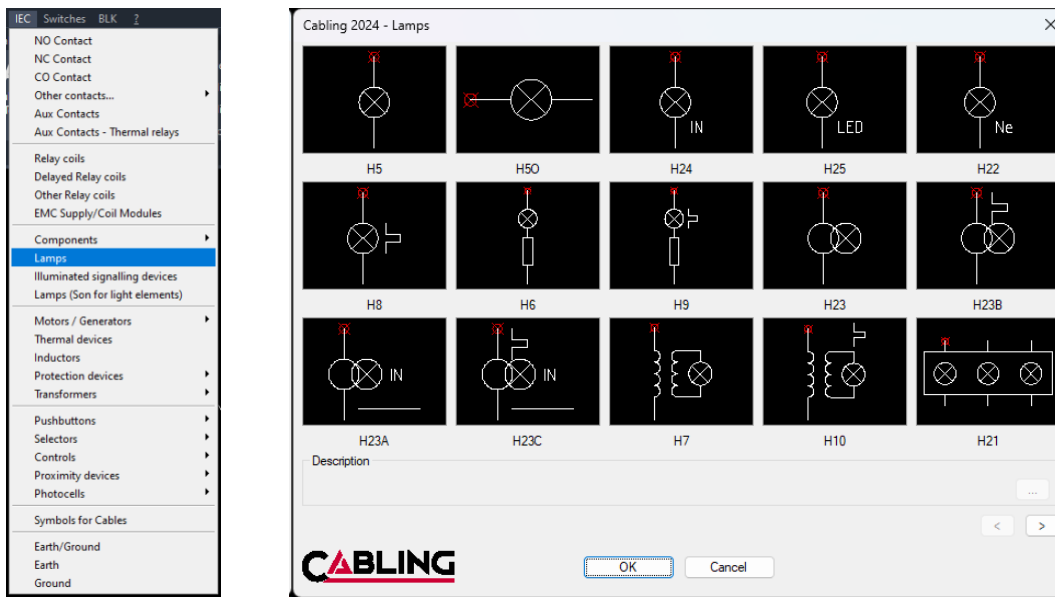


The window is similar to the one used so far in previous versions of Cabling but by pressing OK the component is not inserted but added to the list of the main window.

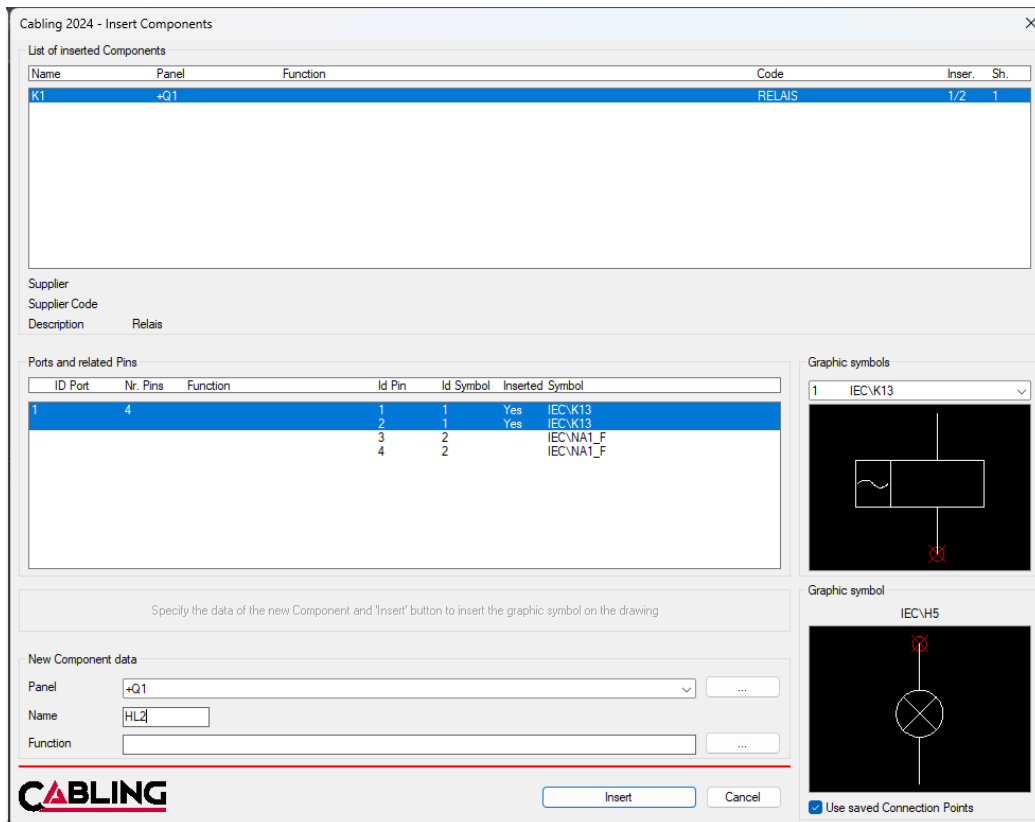
It is then possible to insert the symbol or symbols that you have associated with the component code with the 'Insert' button.



### Insertion of graphic symbols without material

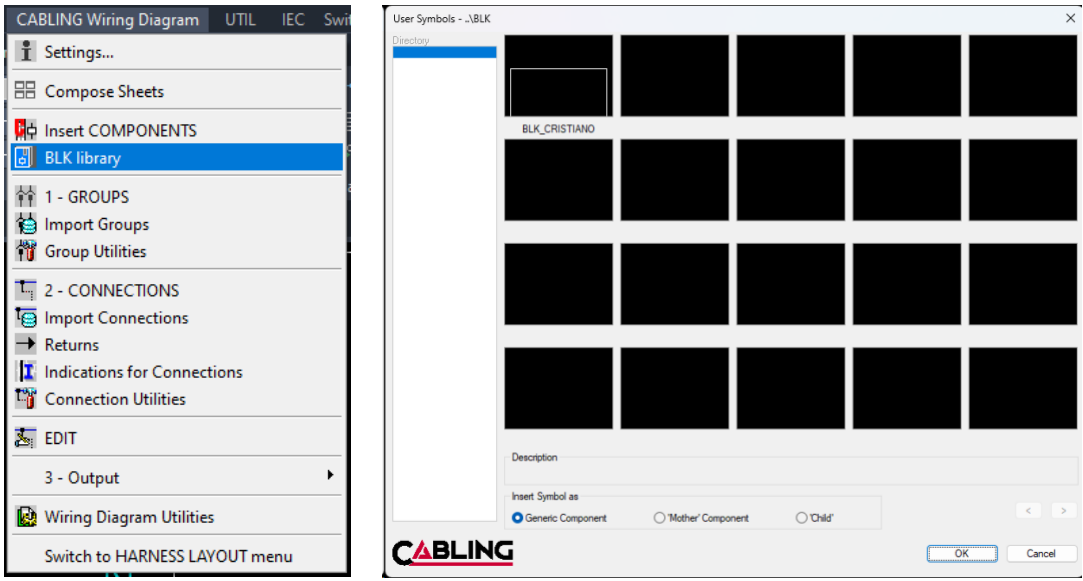


After choosing one of the graphic symbols in the window, the following dialog box opens where in the upper part the components already inserted are visible and in the lower part the request for the data of the new component which, by pressing 'Insert', is inserted into the drawing.



The Library components are already divided into generic, mother and child: for example a relais with one or more open, closed or changeover contacts. Different story for the User Components of the 'BLK Library', on which you have to decide what type they are.

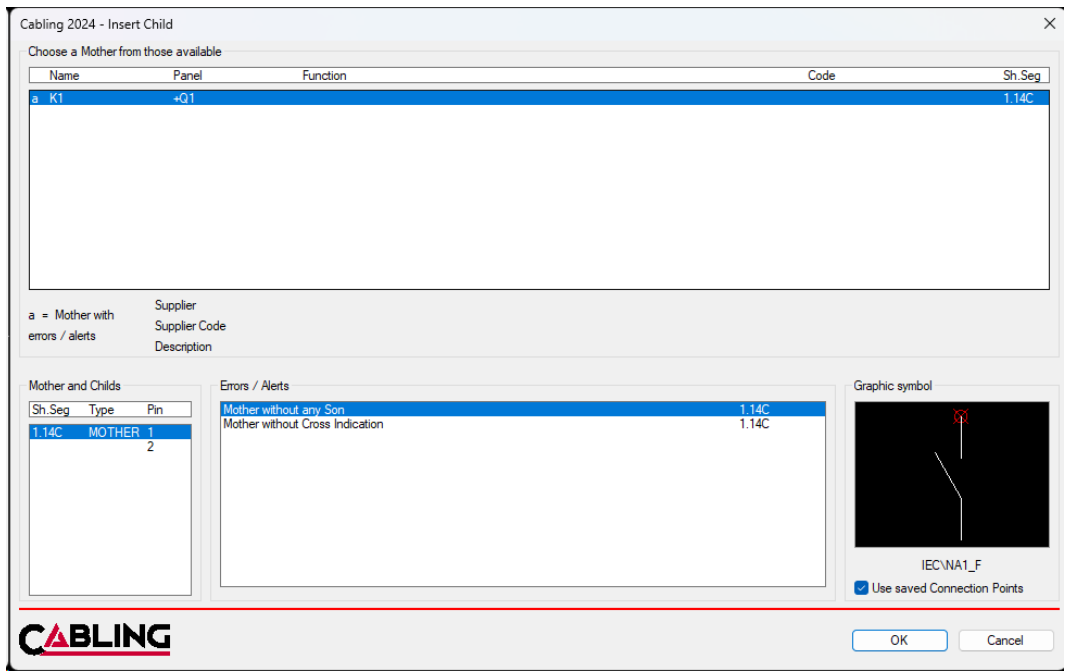




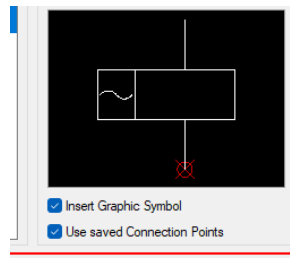
The window that opens after choosing the graphic symbol is the same as in the previous case, except that you have not chosen to insert the symbol as a 'Child'.

**Insertion of children graphic symbols**

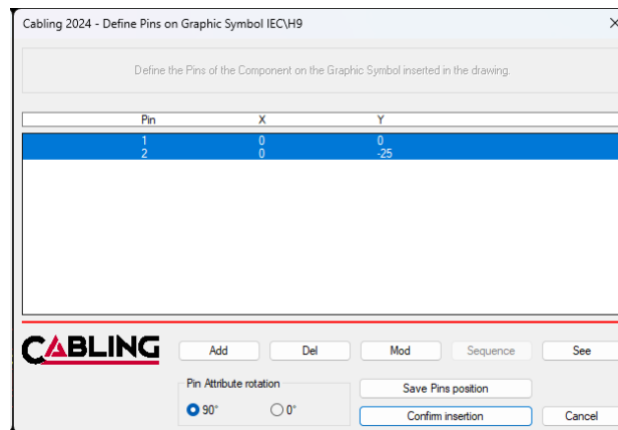
In cases where you insert a standard graphic symbol from the IEC, JIC, VDE or Electronic libraries that is recognized as a child (contacts of various kinds) or if you choose to insert a user graphic symbol from the BLK library as 'Child', another dialog box are opened that allows you to choose the 'mother' to which to attach the 'Child'.



**'Insert Graphic Symbol' and 'Use Saved Connection Points' checkboxes**

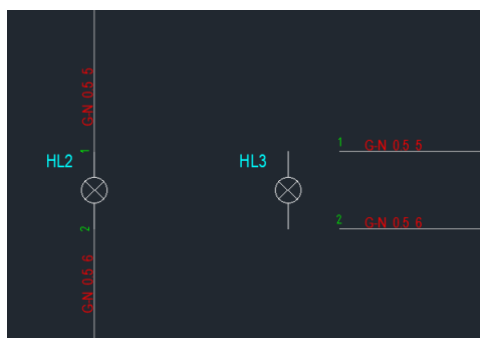


Every time you insert a previously unused graphic symbol, CABLING requests the position of the pins where the connections start/arrive. For this reason, a specific window is available in which to enter the positions, which can be saved associated to the name of the block.

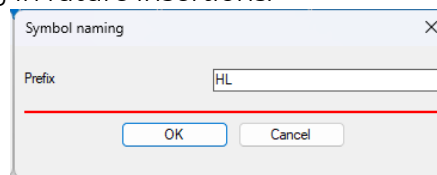


This window opens even if the 'Use Saved Connection Points' checkbox is disabled. Why was this possibility left? Because the insertion of the pins can be modified 'on the fly' for specific needs.

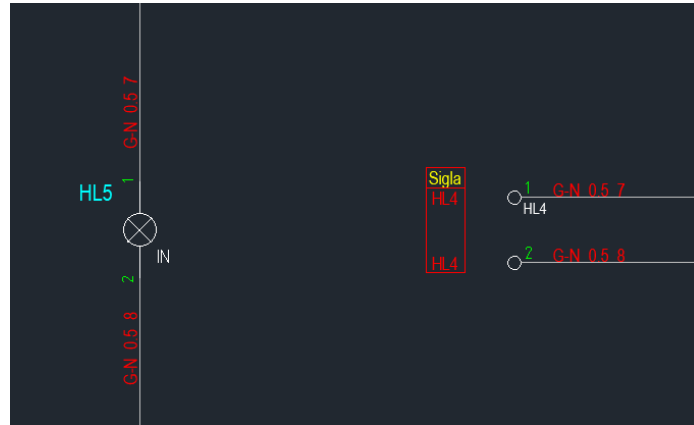
Because the 'virtual' attachment points (or pins) can be moved to the most appropriate point (even without being saved) based on the type of design you want to create. On the left in the image the typical standard insertion, on the right an alternative insertion (widely used in wired electrical interconnection diagrams in the automotive sector). From the connection point of view, nothing changes, but this possibility is very important.



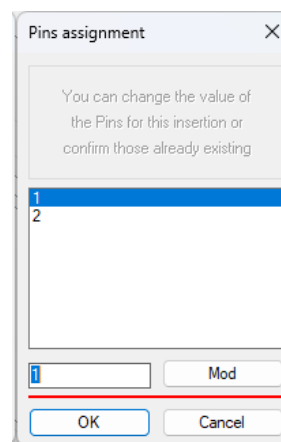
In the window, by saving the positions, it is also possible to save the prefix of the symbol for automatic progressive marking in future insertions.



If the insertion is in **'Insert graphic symbols without material'** mode there is no 'Insert Graphic Symbol' checkbox, which instead appears in the 'Insert components with code' mode. Disabling this last option (even if it is rarely used) allows you not to insert the graphic symbol but to store the component code information together with the code, panel and function in the pins that are alternatively inserted. The insertion of the graphic symbol is avoided and space is saved in the drawing. With or without insertion of the graphic symbol, the component information becomes the usage, information which for example can be used in the harness layout obtained from a wiring diagram.



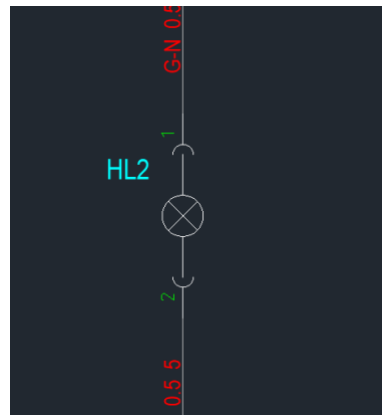
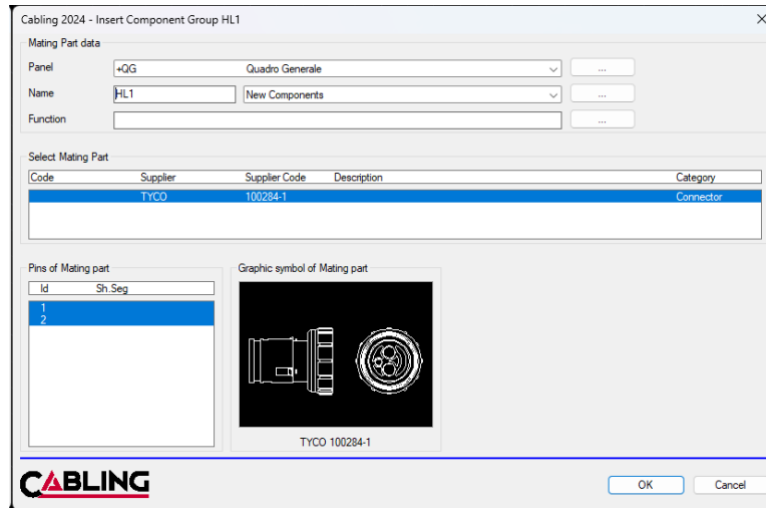
In the case of **'Insertion of graphic symbols without material'**, before inserting the symbols you are asked to change the values of the Pins or to confirm those already present.



However, this window is not opened in case of 'Insertion of components with code' because the matching of the pins values is already done in the definition of the component in the materials database during the matching with the functional graphic symbols.

Finally, in the case of **'Insertion of coded components'**, if a **mating part** (for example a connector) has been provided in the materials database, a further window is opened. In this window you are asked for the Name of the mating part which may be different from that of the user component.





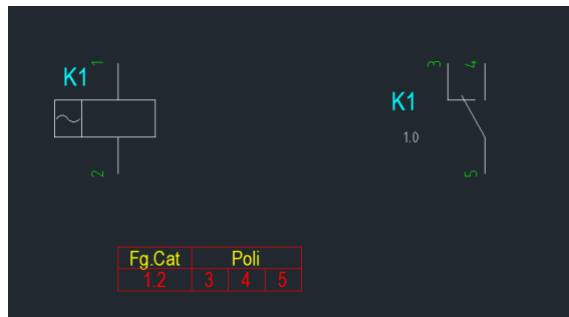


### 4.3 MOTHER AND CHILDREN COMPONENTS MANAGEMENT

More types of components are foreseen in CABLING 2024:

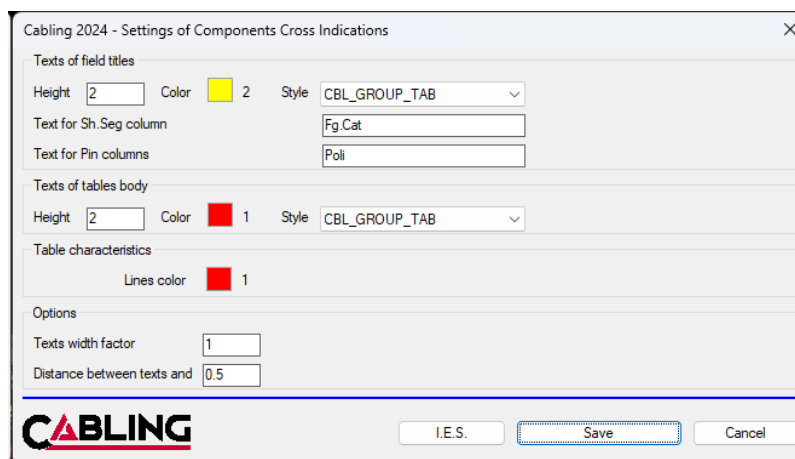
- Terminals/Terminal blocks
- Connectors
- Terminals and Splices
- Unplugged Groups
- Generic components
- Mother components and Child components

Mother and Child Components have been designed to allow you to divide a component into multiple graphic symbols. For example, a coil (mother) with one or more contacts (children) or an electronic board divided into several parts. Mother and children are considered as a single component and are identified with the same name and membership panel. The mother has a PRES attribute with M, the children have an attribute starting with F.



To allow us to understand where the mother and children are in the scheme, two methods have been provided, one linked to the mother and the other to the children. The mother may have associated a Cross Indication which reports the Sheet.Segment position of the children and their pinout, i.e. the value of the Pins. For the children, the REG attribute is filled in which indicates the Sheet.Segment position of the mother.

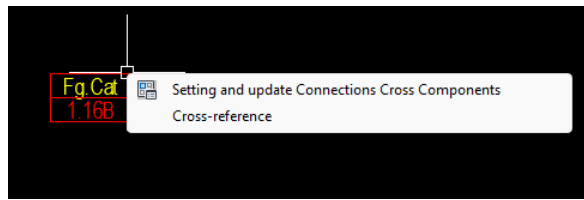
The Cross Indications include a window with parameter settings:



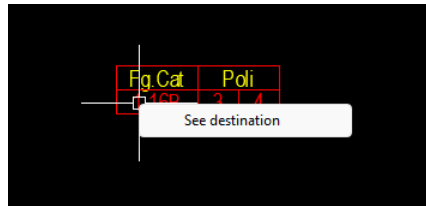
Only two columns are provided, one for the Sheet.Segment of the children and the other for the values of the Pins. The Pins column is further subdivided by individual values up to a maximum of 8. If a child has more than 8 pins, the Pins column is omitted.



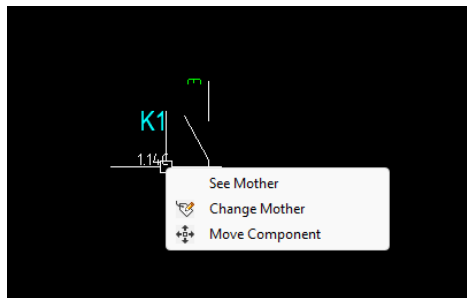
To view the window you need to press the right button on a previously inserted Cross Indication.



If you press the right button above one of the texts, a contextual menu will be presented to see the destination, i.e. the child to which the selected text belongs.



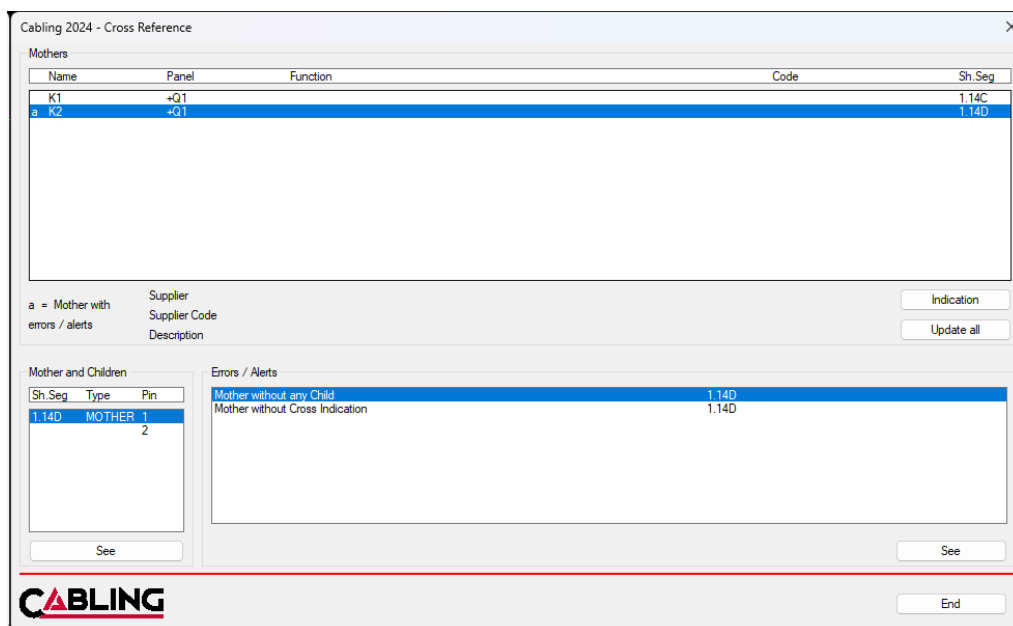
If you press the right button above one of the child, a contextual menu will be presented to see the Mother or change her.



## 4.4 CROSS REFERENCE

The Cross-Reference command directly opens the window below in which the mothers found in the current multisheet are listed. The list shows the Name, the Panel, the function, any material code and the Sheet.Segment position of the individual mothers. If a mother has errors or warnings, the letter 'a' is displayed on the corresponding line and in the 'Errors / Reports' section they are listed. It is possible to see the symbols that have a problem with the 'See' button.

With the 'Indications' button you can insert the indication of the selected mother or move it if it is already present. With the 'Update all' button all the mothers and children are updated, i.e. the present indications are updated and the Sheet.Segment of the corresponding mother is inserted in the children.



By selecting the individual mothers, the Sheet.Segment information, the type (mother or child) and the pins used are displayed in the 'Mother and Children' section. The error/warning messages are:

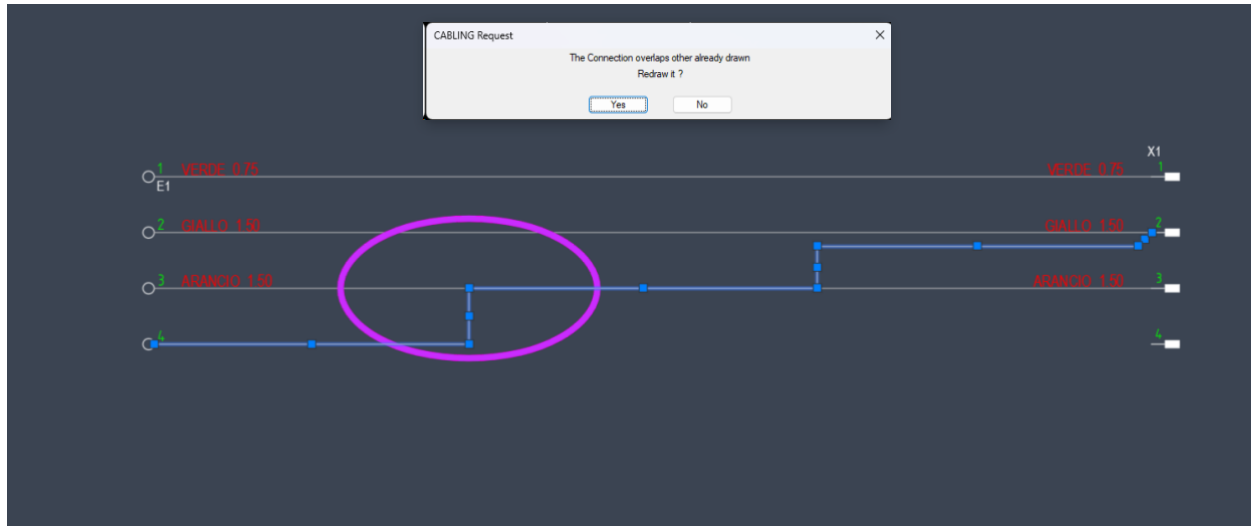
- More than one Mother with the same Name found
- Mother without any Child
- Mother with the same Name as a generic component
- Attribute not valued
- Repeated Pin
- Mother without Cross Indication
- Child(ren) without Mother
- Child with the same Name of a generic Element



## 4.5 DISPLAYING CONNECTIONS DRAWING ERRORS

During the drawing of the electrical and wiring diagrams there was already a function that checked that there were no overlapping wires.

With the 2024 version of CABLING, in addition to the message, cyan colored signs are inserted in the drawing indicating the position where the overlap occurred.



In this way it is easier to redesign the connection, especially on diagrams with numerous connections.

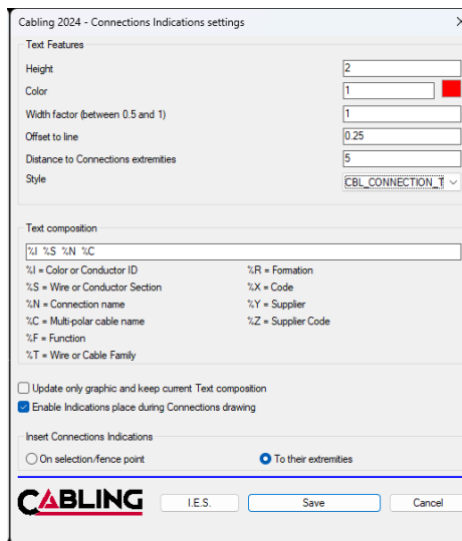


## 4.6 VIEWING CONNECTION DATA

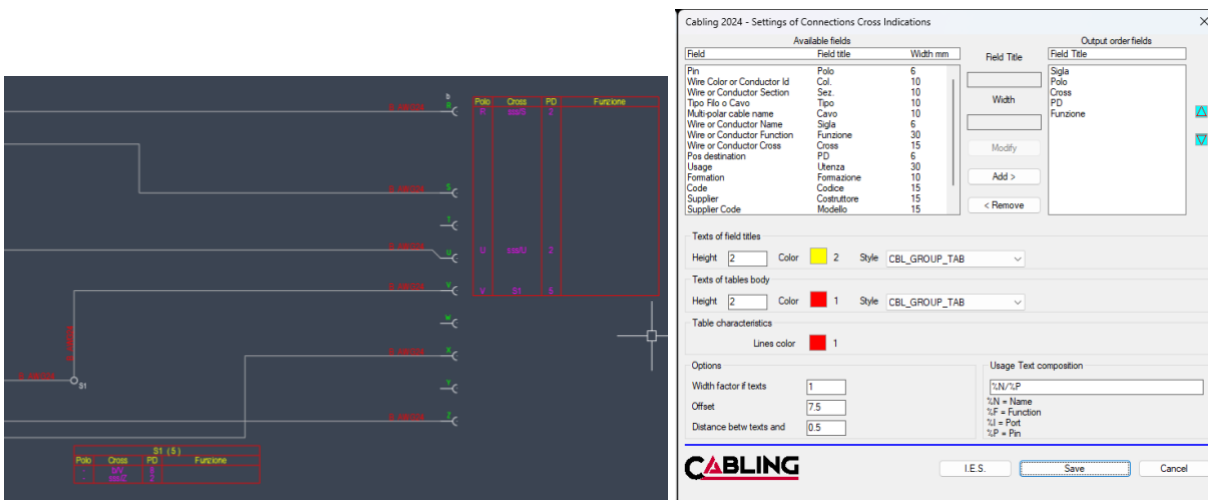
Wiring diagram connection data can be displayed in many ways:

- Via the connection indications
- Via cross indications

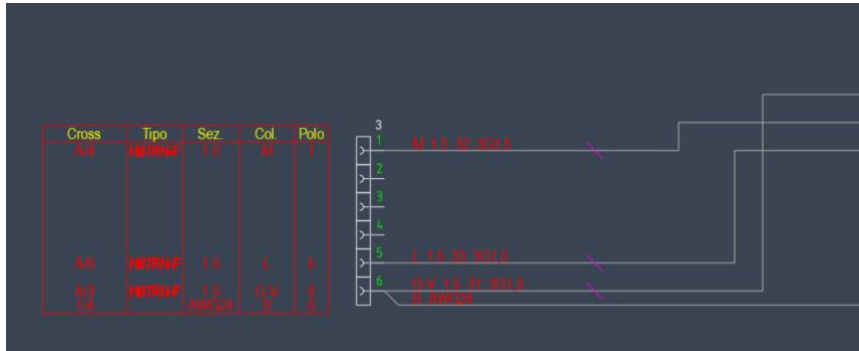
The Connection Indications are by default inserted at the source and destination of the connections, unless the connection consists of a single line of reduced length, in which case only one indication is inserted. The indications can be deleted without loss of data and added at other points of the connection. The content of the indications can be set with the settings in the window below, where appropriate %\* keys can be interleaved with spaces or other characters. The characteristics of the text representing the indication can be set: style, height, color and other parameters. The default parameters proposed were studied together with the pins step and the visibility in the printing. To open the settings window, press the right mouse button after starting the 'Connection Indications' command.



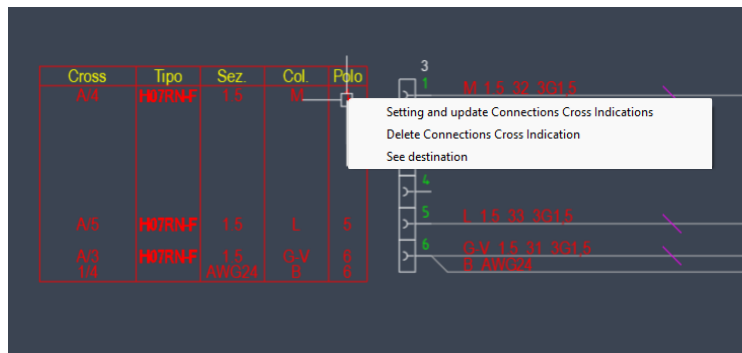
Cross Indications are created next to the Pins of the wiring diagram to indicate the connection data and the destination pin. Indications can be erased without loss of data. The content and characteristics of the indications can be set with the settings in the window below. To open the settings window, press the right mouse button after starting the 'Cross Indications' command.



The 2024 version of the command creates the Cross Connections Indications based on the selected pins, which can also belong to different Groups. For this reason, the Group Name field has been added. Previously the indications concerned only the pins of a single group, with the possibility of cross tables that interfered with each other.

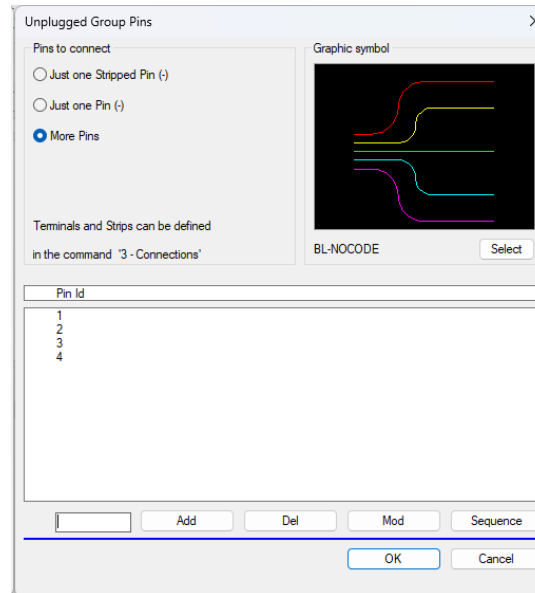


Another important new feature is the possibility of having the 'See destination' option on the contextual menu (right mouse button clicked on the departure or arrival texts) which focuses on the destination/arrival point.

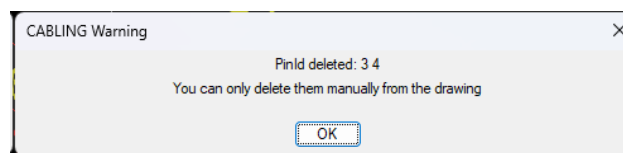


## 4.7 MODIFY UNPLUGGED GROUPS

In the wiring diagram it is possible to modify an Unplugged Group also by inserting additional pins. Starting from the usual window where it is possible to change the name and the function, after pressing the 'Edit Group' button a new window opens where it is possible to add new Pins.



If Pins already inserted are deleted from window, an error message is given:

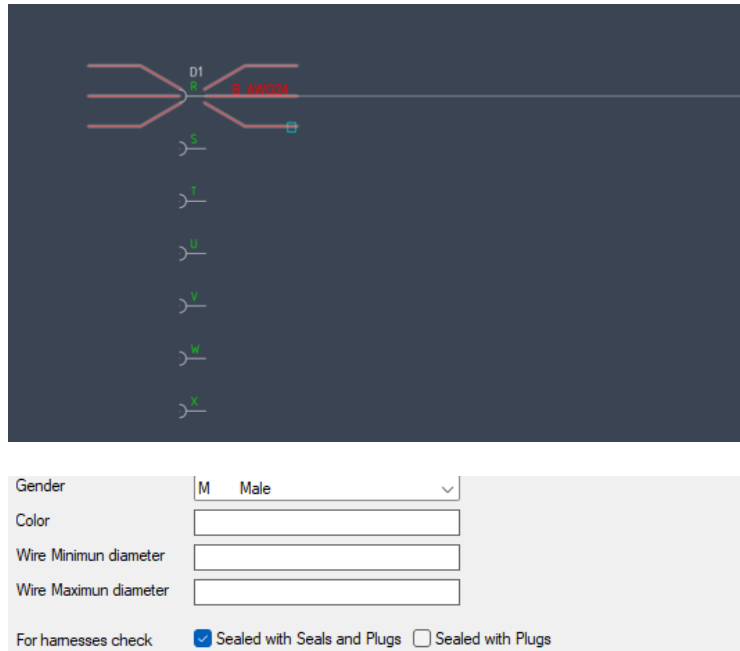


After having possibly modified the name and the function, if new pins have been added you are asked to insert them on the drawing one at a time.



## 4.8 DRAW CONNECTIONS

If when drawing the connections in the wiring diagram you select a pin of a connector that already has a connection and if the connector has been defined as 'Sealed' in the Cabling materials archive, then an alert message is given indicating the inconsistency to connect two or more wires to one pin of a waterproof connector.

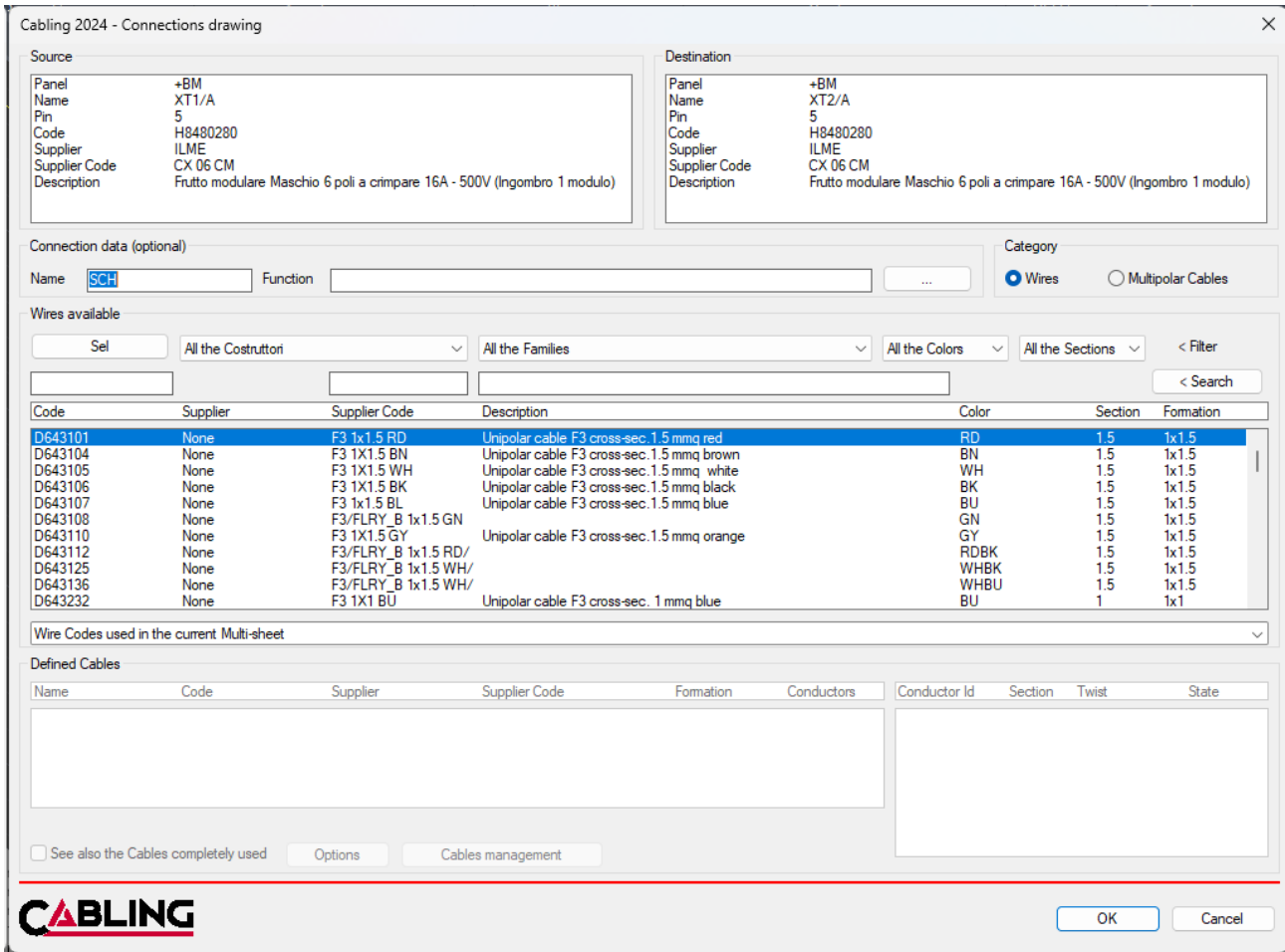


The connections drawing window has been revised with the following new features:

- It has been simplified
- A list of 'Wire Codes used in the current Multisheet' has been added: if you select one of the codes it becomes the current code in the upper list, unless the set filters do not allow it. In the latter case, an alert message is given.
- For Multicore Cables, 3 superfluous buttons have been removed and the list of cables shows by default those that still have usable conductors. By pressing the 'Also see the Cables completed used' checkbox all used cables in the current multisheet are displayed in the top list.







The improvement of this window allows greater ease of use of the program and a reduction in search times for wire codes already used in the Multisheet.

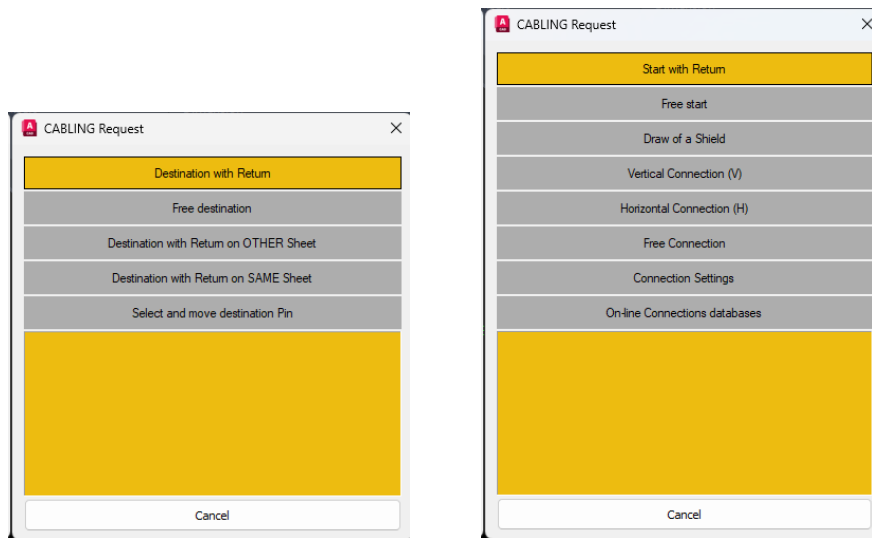


## 4.9 CONNECTION RETURNS

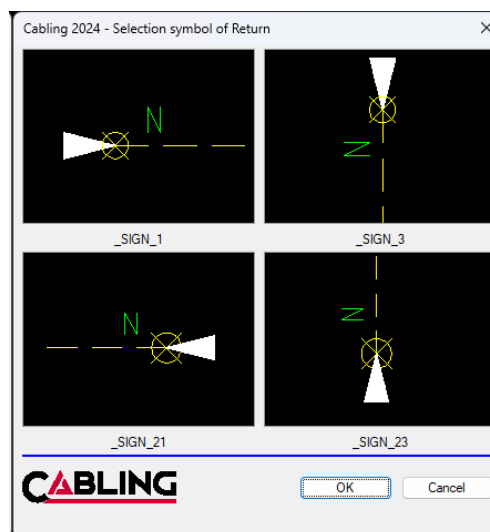
Drawing Connections that start/end in different sheets can be done in several ways:

- Create a sheet composition, connect the departure and arrival with a connection and finally break the connection with 2 Returns, one for the start and one for the destination. The splitting of sheets, at the end, will put everything back in place, with a Return on one sheet and the other on another.
- Use the 'Destination with Return to another sheet' option which allows you to insert a Start on the first part of the connection, change sheets and insert the Destination Return together with the second part of the connection. This method assumes that you know exactly the target sheet and that you have already drawn the target group.

In the 2024 version, a new mode has been added: Free Destination with Return, which is coupled with Free Start with Return.

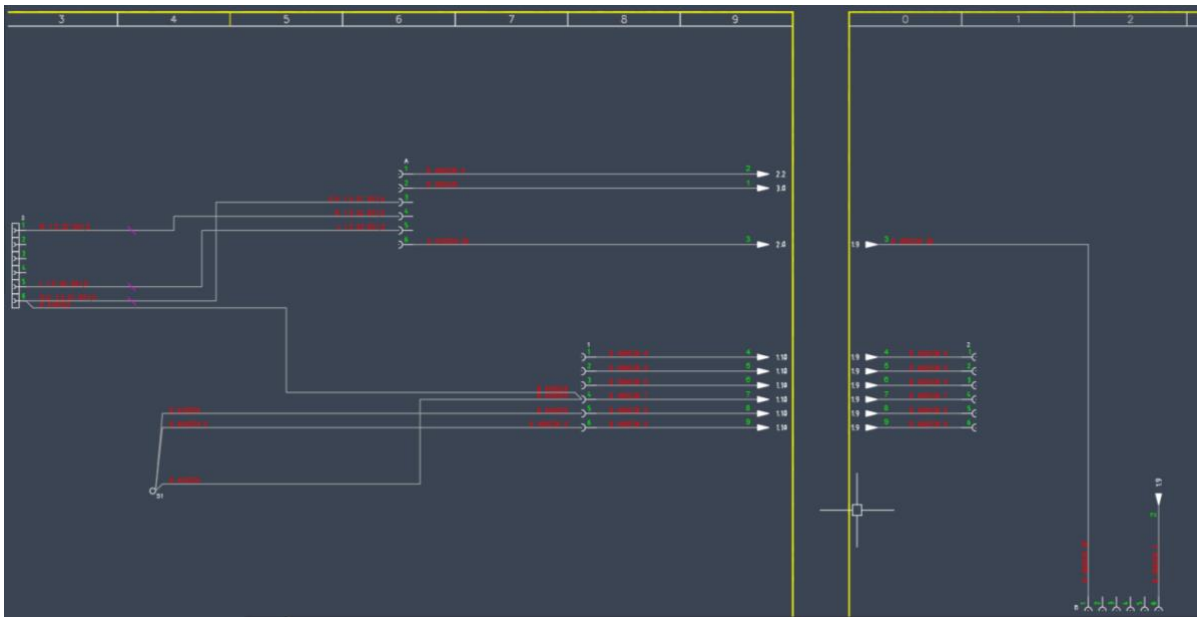


In other words, you can manage starts and destinations separately on the individual sheets, because CABLING will offer you the list of references already inserted allowing you to choose one, to check if a connection already has start and destination and therefore cannot be used and, finally, to acquire automatically the characteristics of the start or destination already entered.



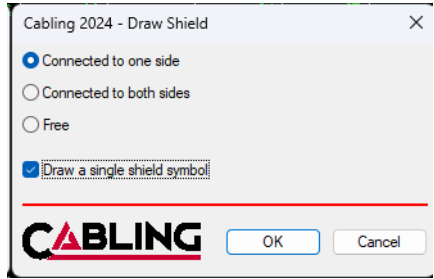
Furthermore, CABLING 2024 proposes the reference number/name equal to the connection Name, widely used in some design sectors, or a progressive number if the connection Name is not used.

However, there are cases in which the number/name must be used several times in the electrical diagram. Since CABLING does not allow multiple departures/arrivals with the same number/name, the concept of Branching was introduced: with the same acronym/name you can enter a value in the Branching. Therefore, from the 2024 version CABLING always manages two returns, one starting and one arriving, and identifies them not only by the number/name but also by the possible value of the Branching.

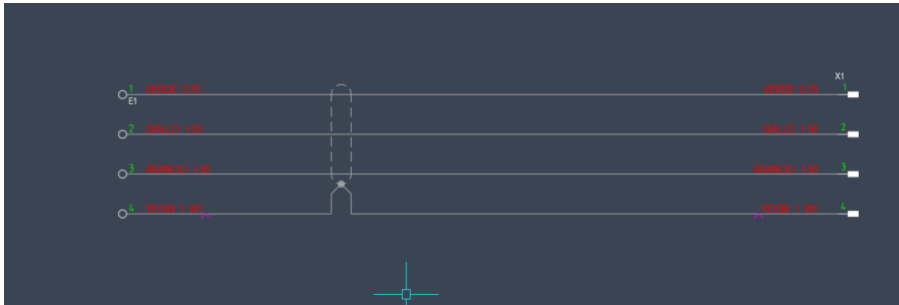


## 4.10 SHIELDS

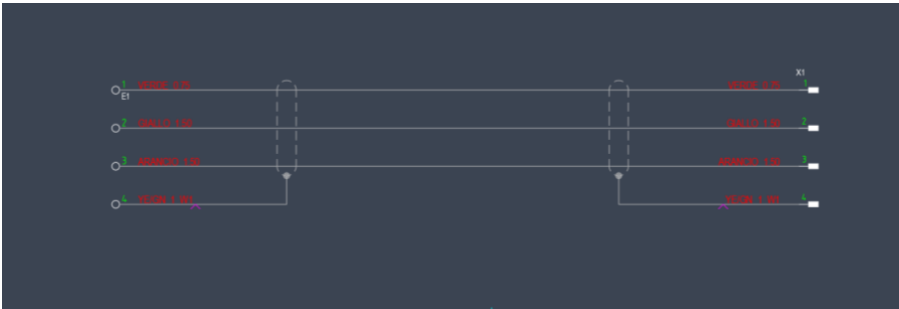
The design of the shields has been completed with a new mode: the possibility of inserting a single shield symbol. The window below opens which has the new 'Draw a single shield symbol' checkbox.



In the case of single shield symbol:



In the case of double shield symbol:

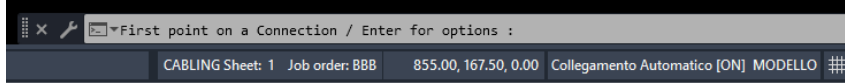


The new mode simplifies the wiring diagram while maintaining the characteristics of the connections on the shields unchanged.

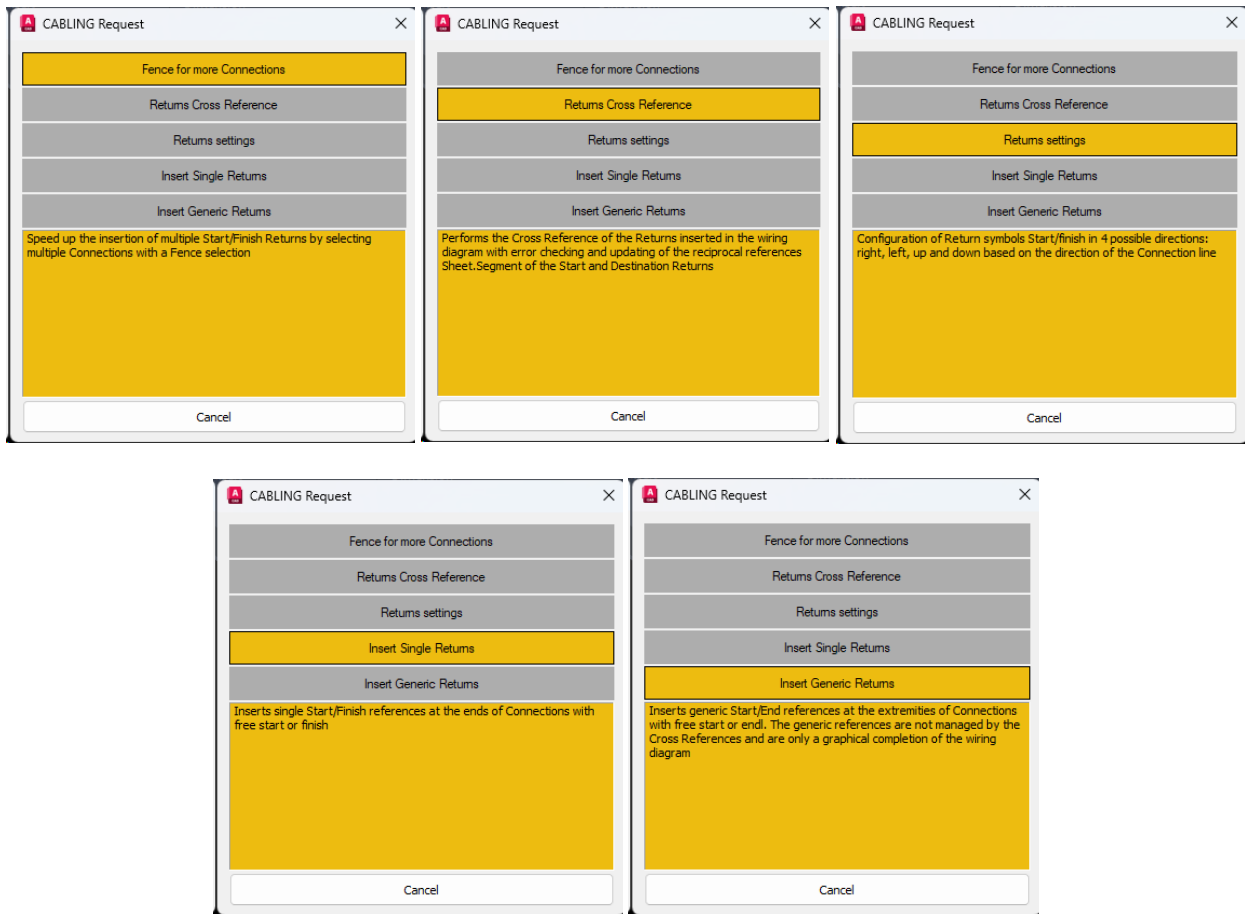


## 4.11 RETURNS ON CONNECTIONS

The command has now become unique (both from the menus and from the right click on a Connection) with the request on the command line for the insertion point of the Start and Destination Returns.



If you press Enter to view the options, these additional functions appear (each displayed with its explanation):



The old Returns command, present on the CABLING menus, is the one that you can now recall with the 'Insert Single Returns' option.

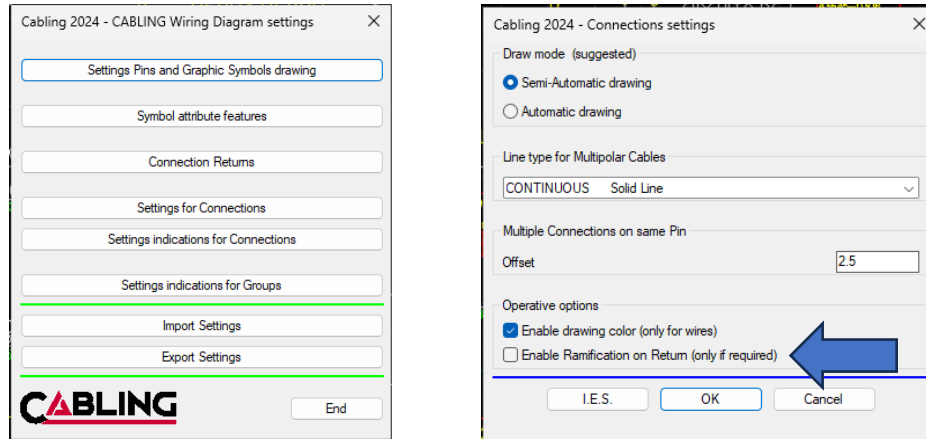
The returns break the connection with a Start and Destination Return and are identified by a Name (or signal). It is not permitted to have more than two references with the same Name.

The Name of the Returns is independent of the Name of the connection (or wire number as in SPAC) because in CABLING the Name of the connection is not mandatory. If you use the connection Name, it is good practice to match the connection Name with that of the Returns.

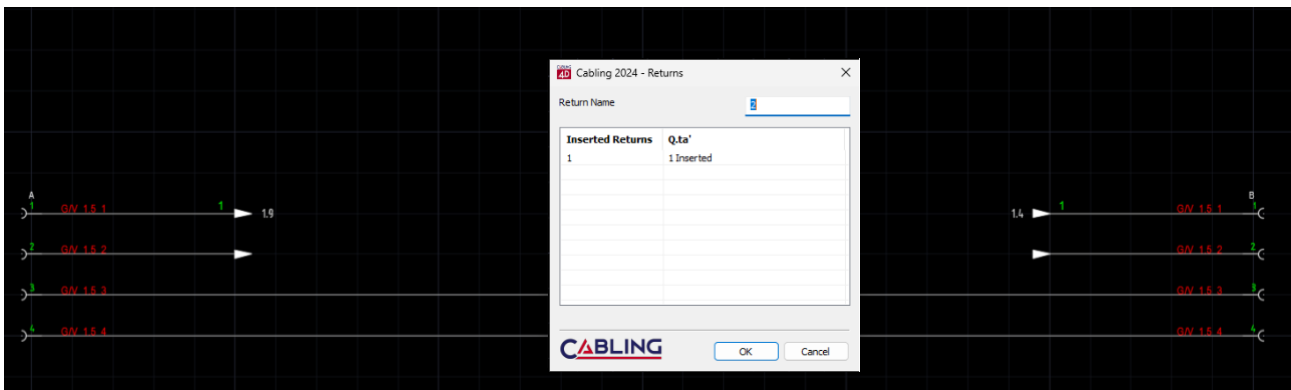


In CABLING 2024 the possibility of working with Ramification was also introduced, i.e. the possibility of having the same return name but with the addition of a value (even null) for the Ramification.

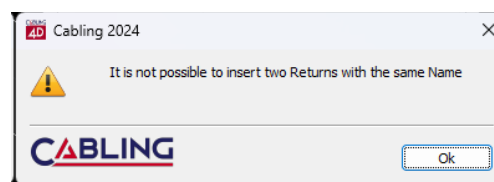
To activate this mode it is necessary to go to the wiring diagram settings and use the 'Connection Settings' button to activate the new 'Enable Ramification on References' option.



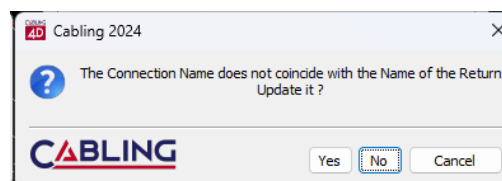
When you insert a Return, a new window opens showing the list of Returns inserted and a box where you can enter the Return name (which can be a number or a generic string).



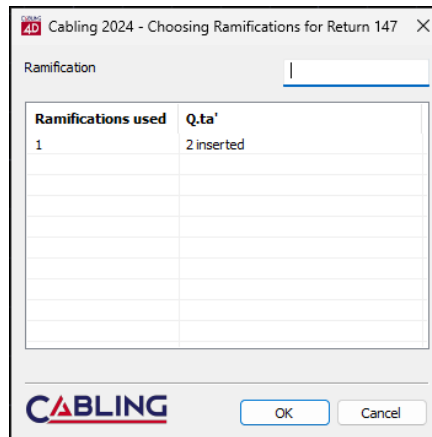
If the name has already been used, an error message is given:



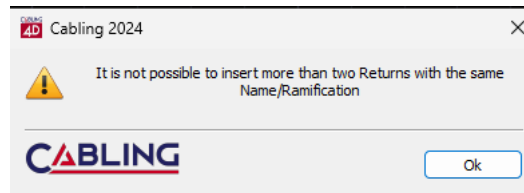
If the return name is different from the Connection name, confirmation is asked:



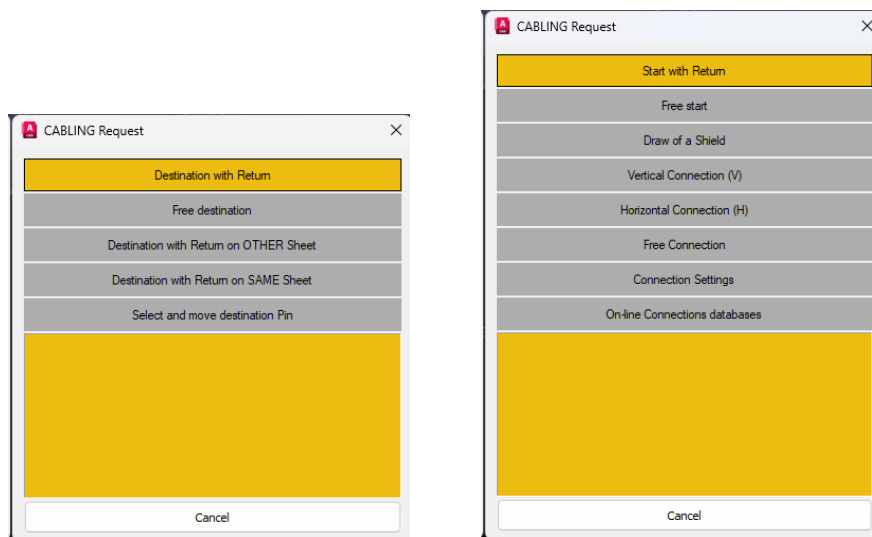
If 'Enable Ramification on Returns' is active, a new window opens which displays the Ramifications already used for the chosen Return name and a text box where you can enter the text (even null) of the Ramification.



If you have already previously inserted a Return + Ramification name in the drawing, the following message will be given:



Finally, we would like to remind you that the following drawing possibilities with Returns were included when drawing the connections:



Destination with Return, Start with Return, Destination with Returns on ANOTHER Sheet and Start with Returns on SAME Sheet are the methods for drawing connections with Returns (some already existing, others new) which use the same windows to define the Name of the return and possibly of the Ramification.



## 5 PANEL DESIGN

The Panel Design module has been profoundly revised in its operation, while maintaining the basic logic.

It was designed to design two-dimensional electrical panels/panels.

A brief summary of the entities that are managed:

- Passage Conduits
- DIN bars
- Panel accessories
- Bundles with covering material, Bundles without covering material, Bundles belonging to multi-core cables, Bundles belonging to moulded cables.
- Wire bunches
- Real connections
- Virtual connections
- Components
- Terminals Blocks
- Various group (connectors, wire terminals, etc.)

Normally, in the design of an electrical panel, real connections are used that start from one point and arrive at another. The points are the pins or pins of components, terminals, connectors or single terminals or splices.

Cabling, however, allows you to draw some parts of the panel with the harness mode, i.e. instead of drawing real connections it defines virtual connections.

The connectors can be managed with the associated node or with the individual pins inserted on the graphic symbol of the connector.

The terminals block are managed with two virtual pins: A and B, which generally correspond to High and Low.

The components are managed with the pins that distinguish them, which can be grouped into one or more ports. The components also include any mating part connectors to be connected to the other element of the panel.

A drawing of a panel can be made to scale to be inserted into the desired title block. The scale is stored so that subsequent bills of materials and cutting list are correct in length.

A real connection is a single entity that connects two points of the panel and which can pass one or more conduits, in wire bunches, in Bundles with or without covering material or free passage in the panel. Each connection can be modeled as desired and the free passage points fitting with the set radius.





The ends of the connections arriving on components, connectors and terminals are managed based on the information present in the Cabling materials database:

- If stripping is foreseen, it is applied
- If terminals are provided, they are also applied based on the section of the connection (or connections)
- In the absence of a terminal, it is still checked that the connection section is suitable with the minimum and maximum section possibly set for the individual pins of the component.

For multicore and moulded cables, the lengths of the unstripped common Bundle and of the ends are managed, allowing you to create outputs with the lengths of the jacket strip and of the individual conductors.

Each connection, in addition to the length of the real or virtual connection calculated, can be integrated by the following riches:

- Add-on Length of the single connection
- Departure/arrival add-on on the departure and arrival components.

Each connection can have a name (or wire number), a function and extremities or cyclic markings can be created along the wire.

Thanks to the add-on lengths, the starting and finishing terminals or stripping and the markings, the connections can be exported with all the information necessary for cutting even with automatic cutting/stripping/crimping machines.

An important Panel Design tool is the possibility of importing Excel files with the connections belonging to the electrical panel: these files can be generated by other systems or compiled by the user to speed up the definition of the connections.

It is also possible to import data from an wiring diagram created in the Cabling 4D functional environment.

In any case, Panel Design is one of the few systems that allows you to directly design a panel without the need for data from an electrical diagram.

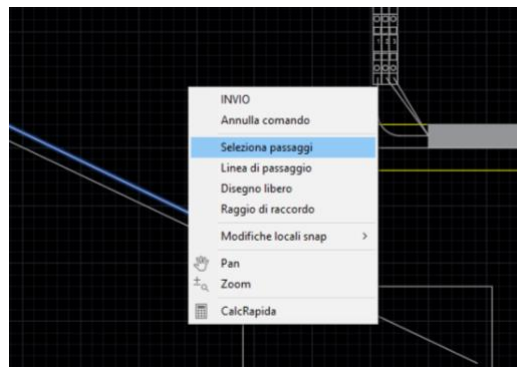
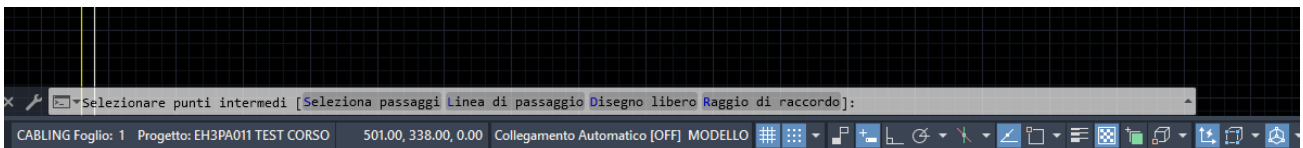


## 5.1 CONNECTIONS DRAWING

The connection design has been completely revised. We always start by selecting the start and end and then modeling the connection, unless the start and end are in a traditional harness which is made up only of Bundles joined to the group nodes.

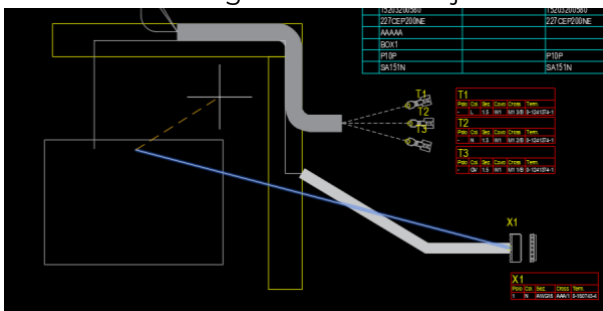
Options are also proposed that you can select in different ways:

- Selecting the option with the mouse on the AutoCAD command line
- By entering the letter corresponding to the option (in blue) and pressing Enter
- Using the right mouse button and selecting the option from the context menu.



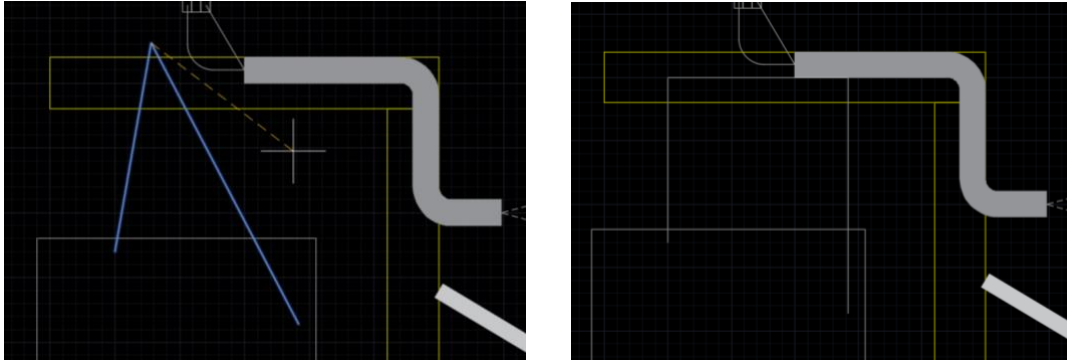
After selecting start and end:

Phase 1 – drawing of the line that joins start and end with the possibility of intermediate points



Phase 2 – There are these cases:

- if you select intermediate points that create a connection that intersects conduits or wire bunches, a connection is automatically created within them.

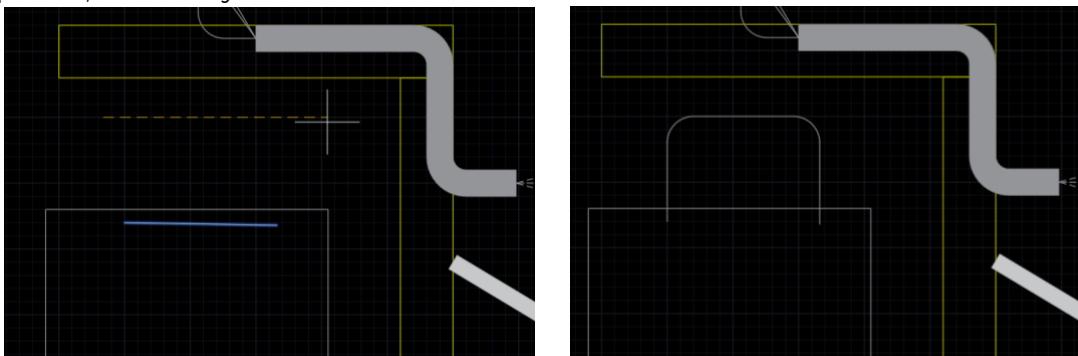


- if the connection created with the intermediate points does not intersect any conduit/wire bunch, the connection remains unchanged, eventually with a fillet in the corner of the connection. If the first and last segment of the connection are not horizontal or vertical, within a certain range they are automatically made horizontal or vertical.

In the two previous cases or if no intermediate points are selected, Panel Design opens the connection dialog box.

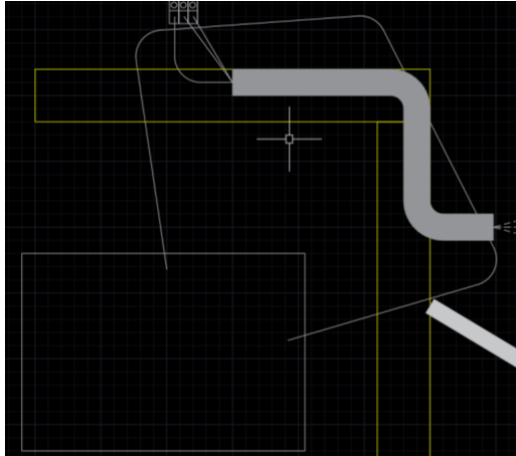
If you select one of the options instead, here's what happens:

- **Select crossing**  
Panel Design asks you to select in sequence the elements over which to pass the connection. If in the selection no elements that are contiguous in any way are chosen, a message is given. At the end, the connection is created which, in the initial and final part, can have further segments to join the starting pin to the first selected object and to join the arrival pin to the last selected object.
- **Passing line**  
Panel Design asks you to select two points in the drawing and creates a connection as if the two selected points constituted a virtual conduit through which the connection passes, eventually with a fillet in the corner of the connection.



- Free design

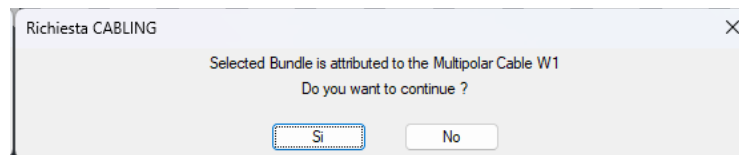
Panel Design asks you to select intermediate points without any intersections with conduits and wire bunches being considered. At the end the connection is created and eventually with a fillet in the corner of the connection.



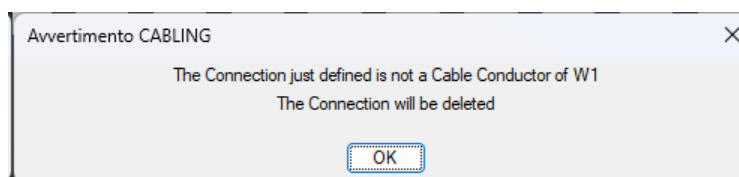
- Connection radius

Panel Design asks on the AutoCAD command line for the fillet radius for connection. If you enter the value zero no fillet will be created.

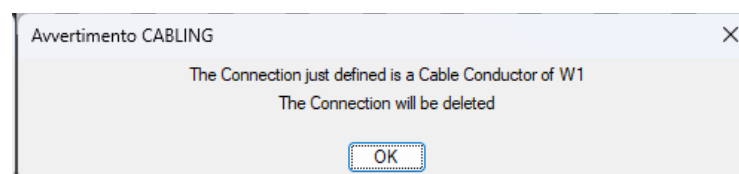
With the 'Select routing' option, if you select as the first element a Bundle to which a Multipolar or Moulded cable has been assigned, the following request will be made:



If you press yes, the selection process stops and the connection dialog box opens, where it is mandatory to choose a multi-core cable with the same name. If you choose a single core wire, an error message is given and the connection is deleted.

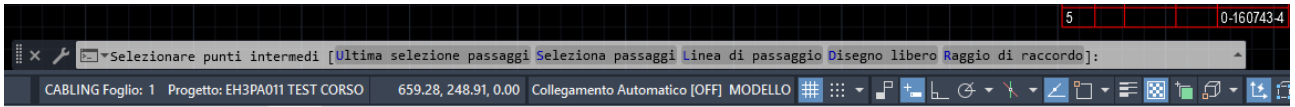


Similarly, if you draw a conductor of a multi-core cable without having selected a Bundle attributed to a multi-core cable, the following message is given:



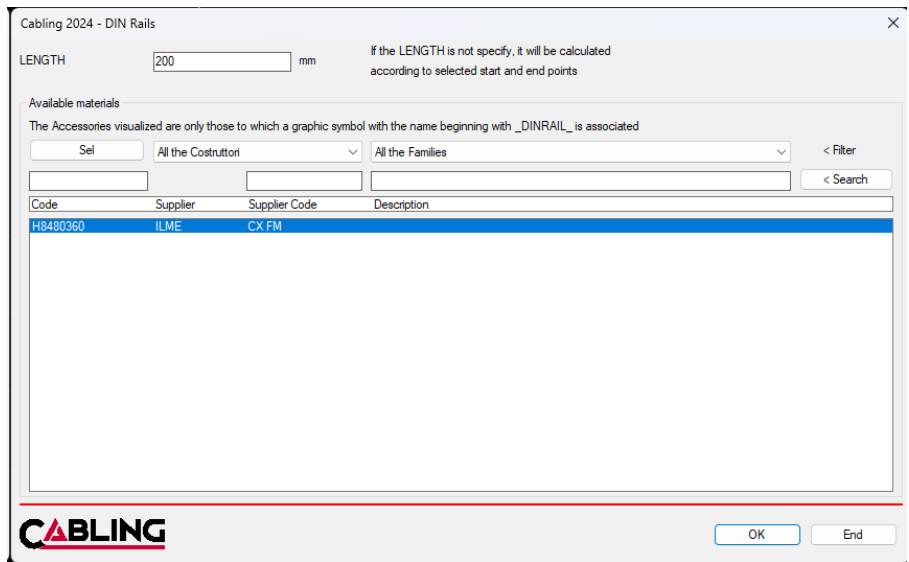
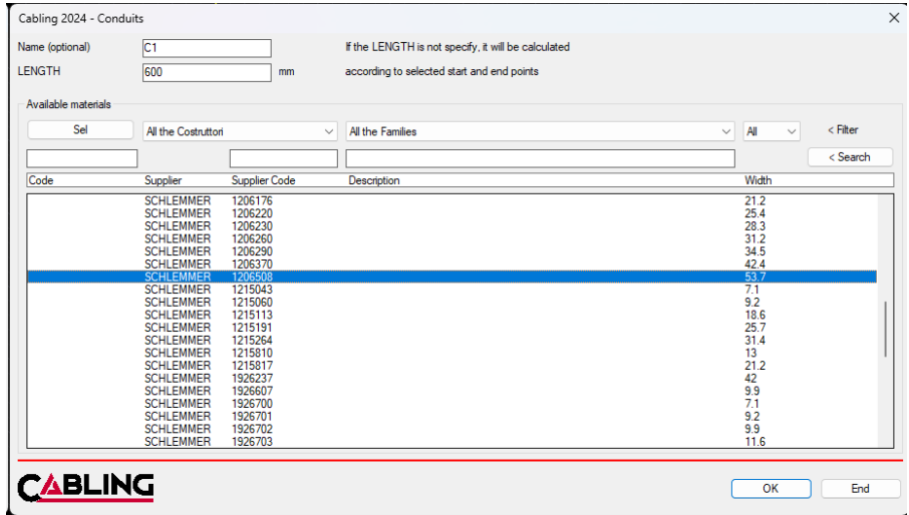
In the 'Select routing' option, if you choose a sequence of conduits, wire bunches and Bundles, it is stored.

This allows you to have a further option: 'Last selection steps'. It allows you to use the last previously used sequence, speeding up the drawing of connections.



## 5.2 MODIFY CONDUITS AND DIN BARS

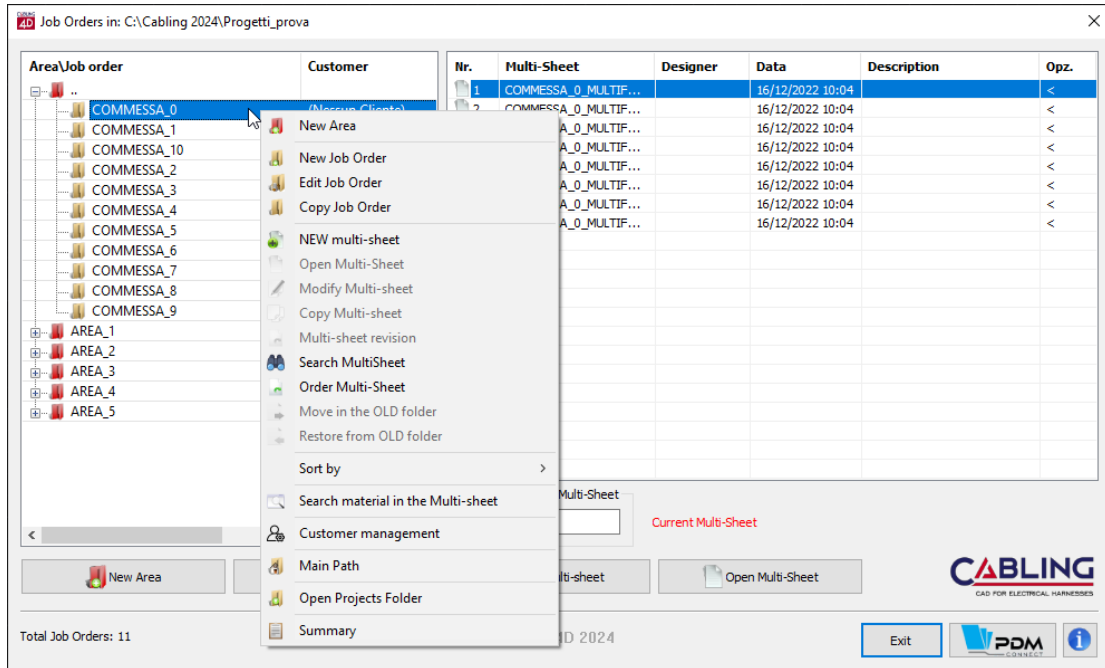
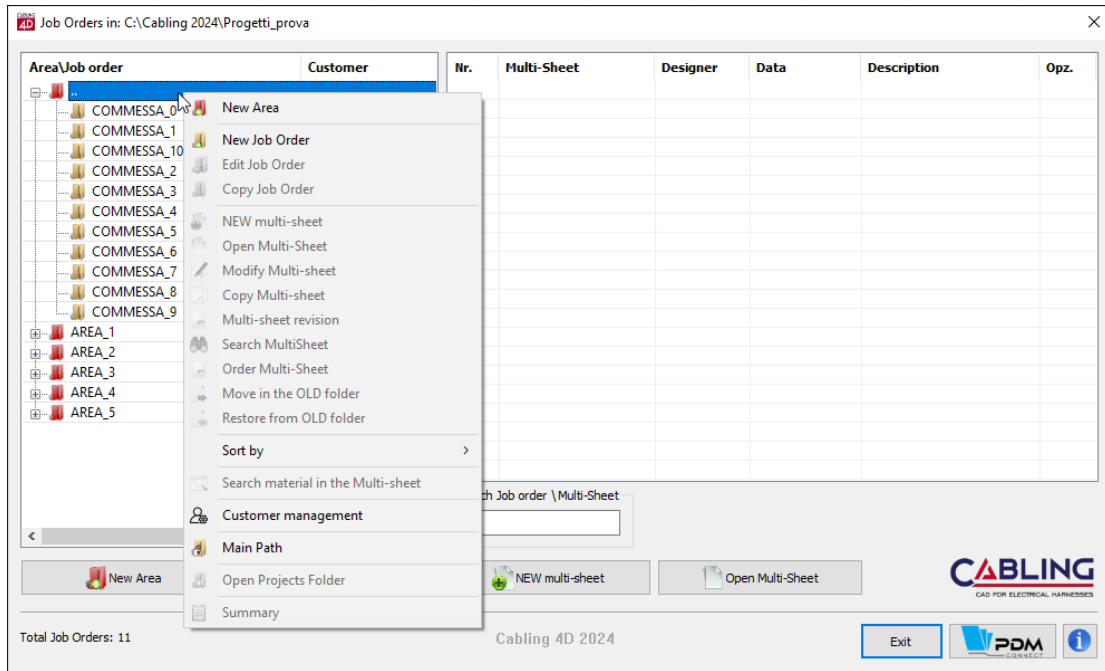
Now it is possible to modify the conduits and DIN bars/rails inserted in the drawing. It is not permitted to change the material but the length and name (only conduits).



# 6 JOB-ORDERS MANAG. AND TITLE BLOCKS

## 6.1 NEW JOB-ORDERS MANAGEMENT

The new interface of the Job-Orders Management command has been completely revised while maintaining the same potential as the previous version, but simplifying use and making it much more user friendly.



Job Orders in: C:\Cabling 2024\Progetti\_prova

Area/Job order	Customer	Nr.	Multi-Sheet	Designer	Data	Description	Opz.
COMMESSA_0	(Nessun Cliente)	1	COMMESSA_0_MULTIF...		16/12/2022 10:04		<
COMMESSA_1	(Nessun Cliente)	2	COMMESSA_0_MULTIF...				<
COMMESSA_10	(Nessun Cliente)	3	COMMESSA_0_MULTIF...				<
COMMESSA_2	(Nessun Cliente)	4	COMMESSA_0_MULTIF...				<
COMMESSA_3	(Nessun Cliente)	5	COMMESSA_0_MULTIF...				<
COMMESSA_4	(Nessun Cliente)	6	COMMESSA_0_MULTIF...				<
COMMESSA_5	(Nessun Cliente)	7	COMMESSA_0_MULTIF...				<
COMMESSA_6	(Nessun Cliente)	8	COMMESSA_0_MULTIF...				<
COMMESSA_7	(Nessun Cliente)						<
COMMESSA_8	(Nessun Cliente)						<
COMMESSA_9	(Nessun Cliente)						<
AREA_1							
AREA_2							
AREA_3							
AREA_4							
AREA_5							

Search Job order \ Multi-Sheet

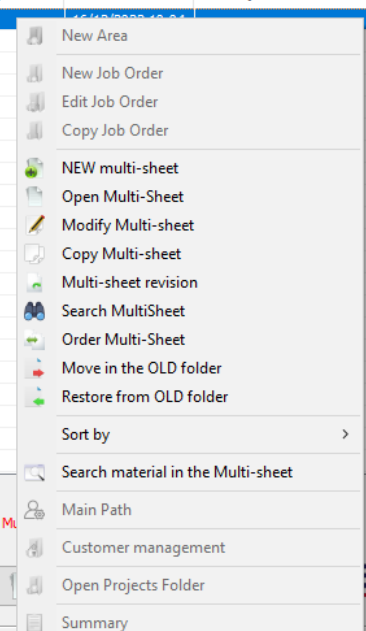
Buttons: New Area, New Job Order, NEW multi-sheet, Open Multi-Sheet

Total Job Orders: 11

Cabling 4D 2024

Exit

PDM



- New Area
- New Job Order
- Edit Job Order
- Copy Job Order
- NEW multi-sheet
- Open Multi-Sheet
- Modify Multi-sheet
- Copy Multi-sheet
- Multi-sheet revision
- Search MultiSheet
- Order Multi-Sheet
- Move in the OLD folder
- Restore from OLD folder
- Sort by
- Search material in the Multi-sheet
- Main Path
- Customer management
- Open Projects Folder
- Summary

Job Orders in: C:\Cabling 2024\Progetti\_prova

Area/Job order	Customer	Nr.	Multi-Sheet	Designer	Data	Description	Opz.
COMMESSA_0	(Nessun Cliente)	1	COMMESSA_0_MULTIF...		16/12/2022 10:04		<
COMMESSA_1	(Nessun Cliente)	2	COMMESSA_0_MULTIF...		16/12/2022 10:04		<
COMMESSA_10	(Nessun Cliente)	3	COMMESSA_0_MULTIF...		16/12/2022 10:04		<
COMMESSA_2	(Nessun Cliente)	4	COMMESSA_0_MULTIF...		16/12/2022 10:04		<
COMMESSA_3	(Nessun Cliente)	5	COMMESSA_0_MULTIF...		16/12/2022 10:04		<
COMMESSA_4	(Nessun Cliente)	6	COMMESSA_0_MULTIF...		16/12/2022 10:04		<
COMMESSA_5	(Nessun Cliente)	7	COMMESSA_0_MULTIF...		16/12/2022 10:04		<
COMMESSA_6	(Nessun Cliente)	8	COMMESSA_0_MULTIF...		16/12/2022 10:04		<
COMMESSA_7	(Nessun Cliente)						<
COMMESSA_8	(Nessun Cliente)						<
COMMESSA_9	(Nessun Cliente)						<
AREA_1							
AREA_2							
AREA_3							
AREA_4							
AREA_5							

Search Job order \ Multi-Sheet

Buttons: New Area, New Job Order, NEW multi-sheet, Open Multi-Sheet

Total Job Orders: 11

Cabling 4D 2024

Exit

PDM





## 6.2 MASTER TITLE MANAGEMENT

The management of MASTER title blocks has been improved and completed. The rule always remains the same:

1. Insertion of a single title block on sheet 0
2. Insertion of a title block for each sheet excluded on sheet 0

In the case of a title block for each sheet the dimensions can be different, and this is the reason why mode 2 is used.

With the 'Number Sheets' command (see image below) or 'Title Management' you can set the positions of the sheet numbers for the title block on sheet 0 (mode 1) or for that of the current sheet (mode 2). For each title block it is possible to set which data (numbers) to display and in what position, while the height of the characters is common to all master title blocks.

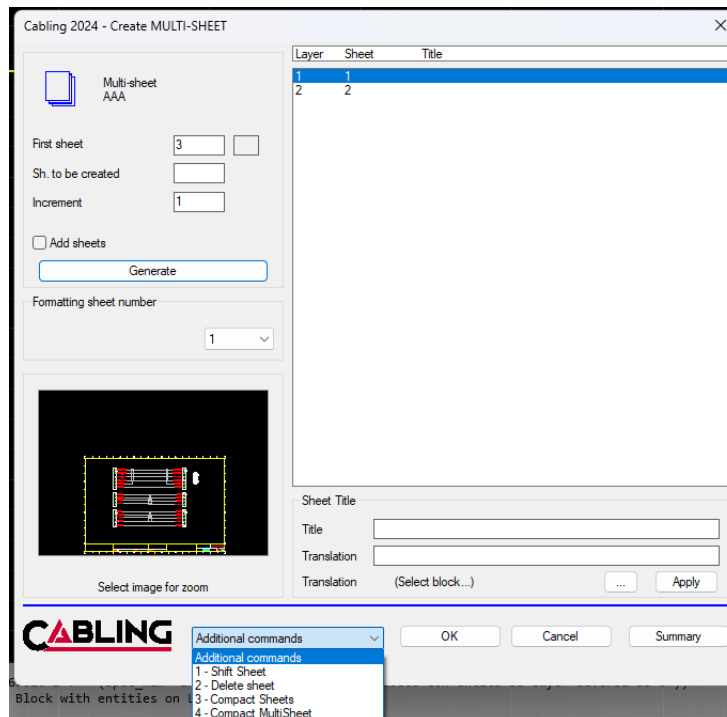
Data	X	Y	Select <	Height
Sheet NUMBER	408.5	22.5	Select <	3
PREVIOUS Sheet	382.5	30	Select <	3
FOLLOWING Sheet	408.5	17.5	Select <	3
TOTAL Sheets	382.5	15	Select <	3

In the numbering, the sheet numbers will be inserted according to the positions of the title block on sheet 0 (mode 1) or on the positions of the title blocks (even different from each other) inserted on sheets other than 0 (mode 2). In mode 2, numbering is not performed on sheets where there is no title block.



## 6.3 GENERATE MULTISHEET COMMAND

CABLING 2024 has also revised the Generate MultiSheet command: 3 additional commands have been provided that can be called up in the popup list at the bottom of the window.



- 1 Shift Sheet  
It is the similar command that you find in the menu and has been inserted in this window to simplify the use of Cabling and also because the menu command will be eliminated in future versions.
- 2 Delete Sheet  
It is the similar command that you find in the menu and has been inserted in this window to simplify the use of Cabling and also because the menu command will be eliminated in future versions.
- 3 Compact Sheets  
It is a new command that allows you to compact the sheets starting from the chosen sheet by eliminating the missing sheet numbers: if you have sheets 2 3 4 6 8 and compact from number 4 you will get 2 3 4 5 6. If no compaction is possible it is given a message.
- 4 Compact MultiSheet  
It is a new command that allows you to compact the whole Multisheet



## 6.4 MASTER TITLE BLOCKS AND REVISIONS

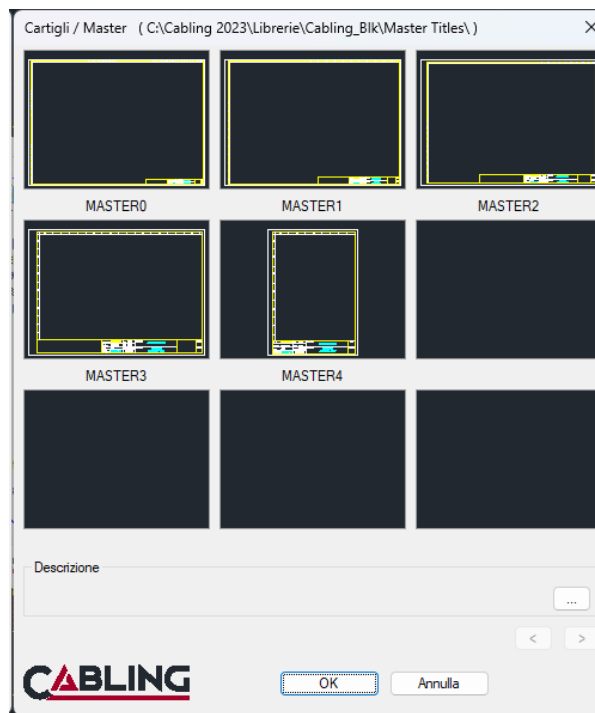
CABLING 2024 has a new folder location containing title blocks and revisions:

<path librerie>\\Cabling\_Blk\\Master Titles

If you are new user (and therefore have not imported libraries from previous versions) you will find 5 MASTER title blocks in this folder. Title blocks are AutoCAD blocks containing attributes that have a name starting with "Master".

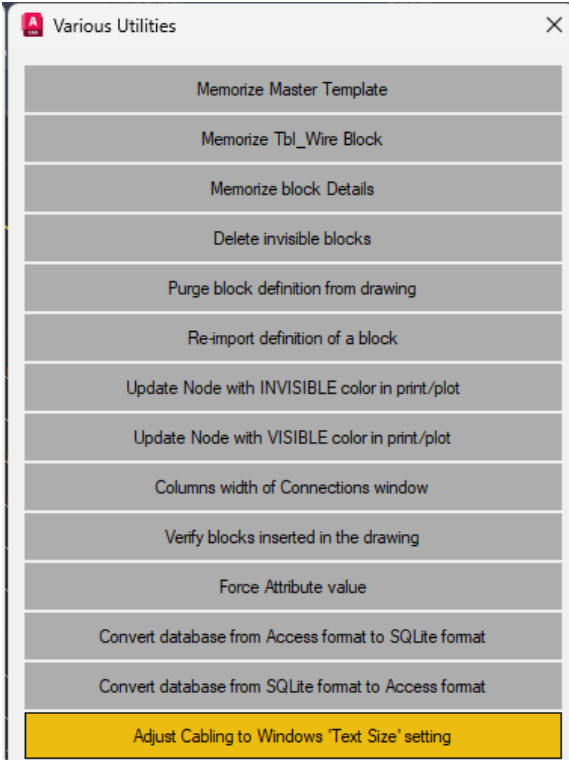
- Master0 → Title block in A0 format (1189 x 841 mm)
- Master1 → Title block in A1 format (841 x 594 mm)
- Master2 → Title block in A2 format (594 x 420 mm)
- Master3 → Title block in A3 format (420 x 297 mm)
- Master4 → Title block in A4 format (210 x 297 mm)

The blocks have been defined with details in Italian and English, depending on the installation language.



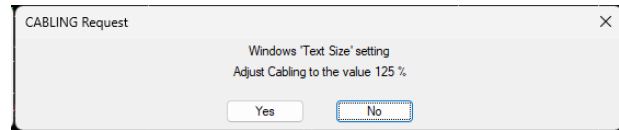
# 7 VARIOUS

## 7.1 ADAPTATION TO WINDOWS TEXT SIZE



To adapt the CABLING dialog boxes if the text size has been changed in Windows, we have made a new utility available in the command UTIL → Utility Cabling → Adapt Cabling to the Windows 'Text Size' setting.

Launching this utility prompts:

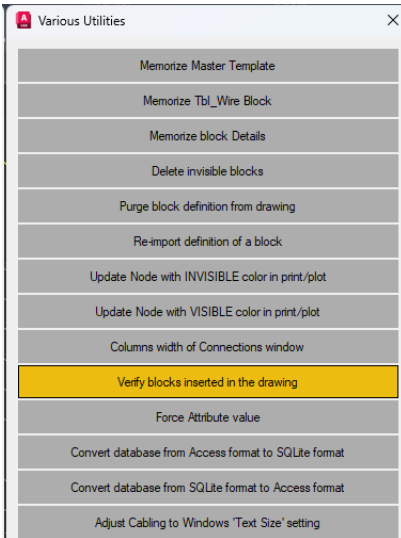


which modifies the windows to match the size (typically 25% larger) of the Windows text size.



## 7.3 CHECK BLOCKS INSERTED IN THE DRAWING

The function, contained in the Util → Utility Cabling command, has now been enhanced to make it easier for users to identify any problems in the current drawing.



Please remember that when you insert a new Block into the current drawing AutoCAD carries out two phases:

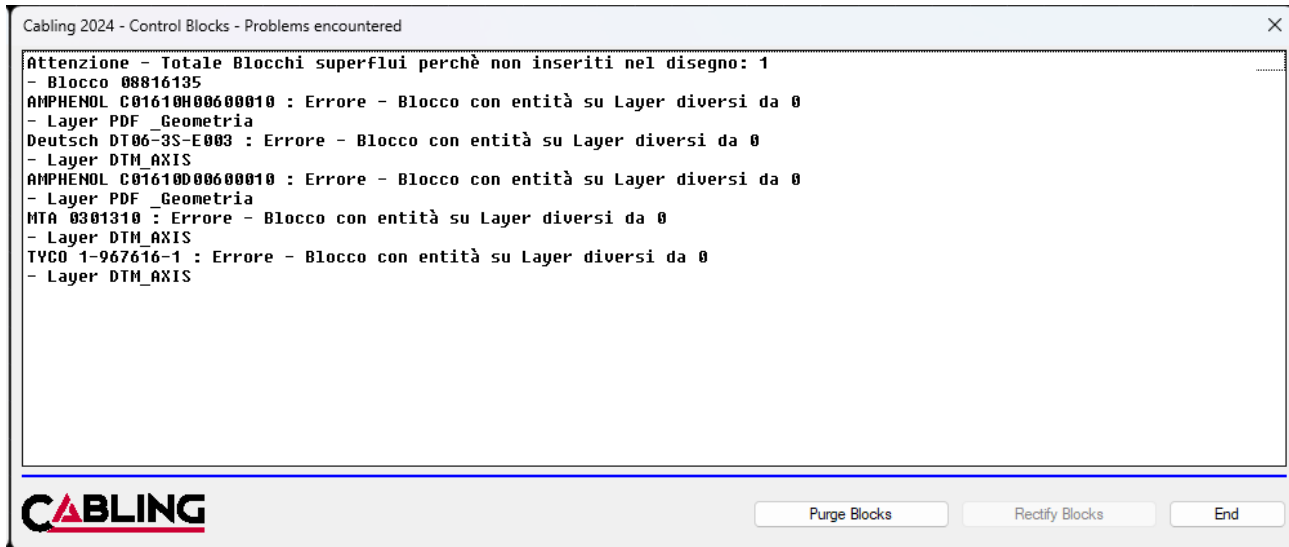
- Inserts the block definition
- Inserts the block into the drawing, possibly scaled and rotated.

If all blocks with a certain name are deleted, AutoCAD still retains its definition in the drawing. It is possible to delete the block definition with the AutoCAD 'Purge...' command found in File → Drawing Utilities.

For the blocks to be used with CABLING it is recommended that all the entities (line, circles, attributes...) that constitute it are on layer 0 (zero), always present in the drawing.

If a block contains entities in a sheet that is not zero (for example on the "DTM\_AXIS" layer) the layer cannot be deleted from the drawing until all blocks with that name are deleted and the block definition is deleted.

The 'Check Blocks inserted in the drawing' function now allows you to report superfluous blocks (i.e. those that are no longer inserted in the drawing) and which can therefore be eliminated and those that contain entities on the layer other than zero.



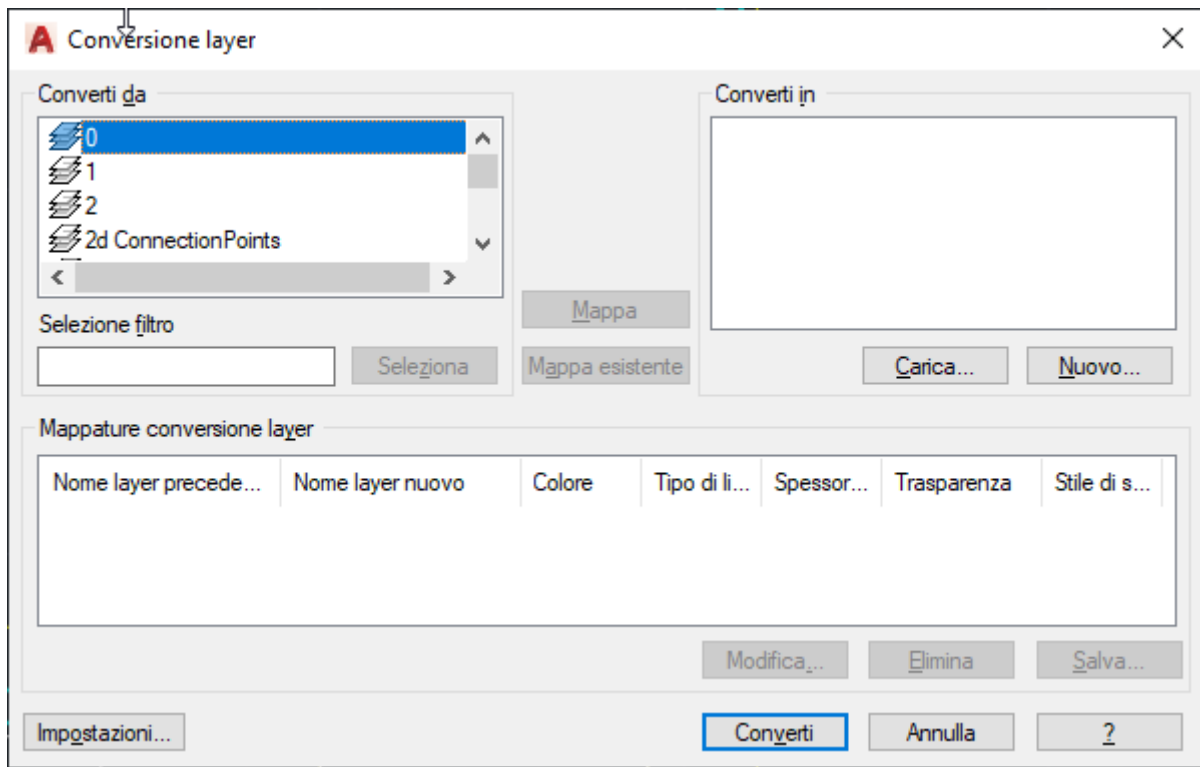
Translation of the content of the dialog:  
 Warning - Total Blocks superfluous because not included in the drawing: 1  
 TYCO 1-967616-1 : Error - Block with entities on Layers other than 0

The 2 new buttons allow you to Purge superfluous blocks and Rectify blocks made up of entities not on the '0' sheet. In the example, after correcting the blocks it was possible (with the AutoCAD Purge command) to delete the DTM\_AXIS and PDF\_GEOMETRY layers. The result is not guaranteed if the block contains other blocks within it, blocks that we strongly advise against using.

If you are unable to resolve drawing problems with this new tool, we recommend using the AutoCAD LAYTRANS command (type on the command line), which converts the layers of the current drawing into the specified standard layers.



In the Layer Conversion dialog box, specify the layers in the current drawing you want to convert and the layers you want to convert them to. Conversion maps layers in the current drawing to different layer names and properties in the specified drawing or standards file, and then converts them using those mappings.



(sorry, italian version of the dialog)

**Convert from:** Contains the list of layers present on the drawing

**Convert To:** Lists all layers that the current drawing's layers can be converted to.

**New:** Defines a new layer to show in the Convert to list for conversion. If you select a layer in the Convert To list before choosing New, the properties of the selected layer are used as default values for the new layer. You cannot create a new layer that has the same name as an existing layer.

Selecting the layers from 'Convert from' and a layer from 'Convert to' and clicking on 'Map' performs the mapping of the selected layers.

Subsequently, clicking on 'Convert' starts the conversion of the mapped layers. If the current layer conversion mappings have not been saved, you are prompted to save them before starting the conversion.

We have included the explanation of this command to allow users to be able to change the state of the layers if the drawing comes from an external source. In many cases there are numerous layers (often belonging to block entities) that are not in line with the CABLING logic, which only provides numerical layers.



### 7.3 IMPORT GROUPS AND CONNECTIONS

If you wish to import a connections file previously exported from CABLING you may find some specific characters in the file, such as (1) or (2) which indicates the entry side of the wires into a splice and the character > which indicates that the terminal field (CONTATTO\_COD\_P or CONTATTO\_COD\_A) is subject to double crimping.

Although these characters are configurable in CABLING and you can decide not to insert any of them in the output, we have still provided the possibility of importing a file that contains them. The following characters are now excluded:

- >
- - >
- (1)
- (2)

This allows you to import the file without being forced to modify it manually. Below is an example of a file with the characters described above.

SIGLA_P	CODICE_P	POLO_P	CONTATTO_COD_P	SIGLA_CAVO	CODICE	ID	SEZIONE	LUNGHEZZA	SIGLA_A	CODICE_A	POLO_A	CONTATTO_COD_A
A	AMPHENOL\$97B-3100A10SL-3P-HD	A	ITT\$121668-0126 >		FAEX00001	G-N	0,5	1000	B	AMPHENOL\$97B-3100A10SL-3P-HD	B	
A	AMPHENOL\$97B-3100A10SL-3P-HD	A	- >		FAEX00001	G-N	0,5	1000	B	AMPHENOL\$97B-3100A10SL-3P-HD	A	
A	AMPHENOL\$97B-3100A10SL-3P-HD	C			FAEX00001	G-N	0,5	400	S1			-(1) >
B	AMPHENOL\$97B-3100A10SL-3P-HD	C			FAEX00001	G-N	0,5	600	S1			-(2) >

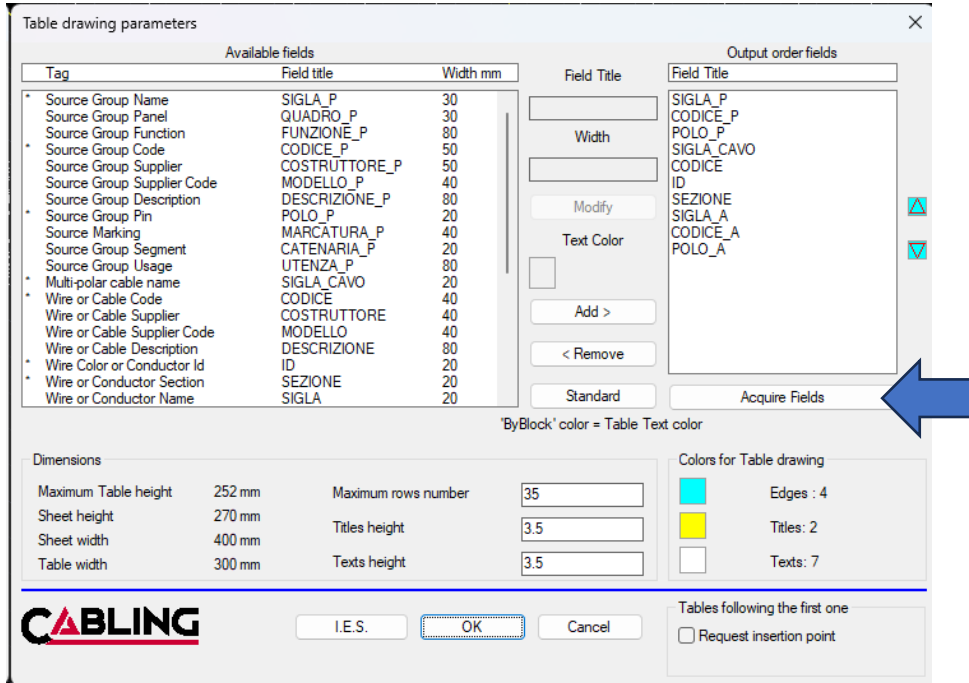
The file so far had to be modified like the example below.

SIGLA_P	CODICE_P	POLO_P	CONTATTO_COD_P	SIGLA_CAVO	CODICE	ID	SEZIONE	LUNGHEZZA	SIGLA_A	CODICE_A	POLO_A	CONTATTO_COD_A
A	AMPHENOL\$97B-3100A10SL-3P-HD	A	ITT\$121668-0126		FAEX00001	G-N	0,5	1000	B	AMPHENOL\$97B-3100A10SL-3P-HD	B	
A	AMPHENOL\$97B-3100A10SL-3P-HD	A			FAEX00001	G-N	0,5	1000	B	AMPHENOL\$97B-3100A10SL-3P-HD	A	
A	AMPHENOL\$97B-3100A10SL-3P-HD	C			FAEX00001	G-N	0,5	400	S1			-
B	AMPHENOL\$97B-3100A10SL-3P-HD	C			FAEX00001	G-N	0,5	600	S1			-



**7.4 TABLE OUTPUTS**

The new 'Acquire Fields' button has been added to the table output settings which allows you to select a previously generated output from the drawing to acquire its characteristics, in particular the order of the fields and their title.



After the selection, the window is reopened with the new parameters acquired.

*The specifications of this document are to be considered confidential. CAD.Able may make changes to the features described in this document at any time for technical or commercial reasons. The examples in this document are only examples.*

