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**EasyDraw Software**
User manual



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**VIMAR**

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# MAIN MENU (FILE MANAGEMENT)

## CREATING A NEW DRAWING

To start a new drawing, run the Nuovo command, which allows to create a new drawing and set the page size of the drawing that appears in the background of the application. At the first save, you can assign the desired name to the drawing.

***Command      NUOVO***

### How to create a new drawing
Run the command.
If EasyDraw is already active and the current drawing has not been previously saved, a dialog box appears that allows saving.
Define the drawing page using the appropriate dialog box and confirm with **OK**.

## OPENING AN EXISTING DRAWING

By means of the APRI command, you can open existing drawings in the *.EDR format

***Command      APRI***

### How to open an existing drawing
Run the command.
The standard Windows dialog appears to open a file: select the desired file and confirm with **OK**.

## SAVING A DRAWING

You can save the current drawing in the EasyDraw native format (*.EDR) and set the default path for saving. *To save as *.DWG and *.DXF use the ESPORTA DWG-DXF command.*

***Command      SALVA***

### How to save a drawing:
If the drawing already has a name, EasyDraw only saves the changes applied since the previous version, while maintaining the same name. If, however, the drawing has no name, the command SALVACOME... is invoked. (see below).

***Command      SALVA CON NOME***

### How to "save as..." a drawing:
The drawing can be saved with a different name or in a different format.
Run the command.
When the standard Windows dialog appears, input the name of the file. Confirm the changes with **OK**.

### How to remember the directory path where to save the drawings
The program can remember the path to the directory for saving a drawing, allowing to set a custom location other than the default one.
Run the command SALVA or SALVA CON NOME.
When the standard Windows save dialog appears, select the Remember path option shown on bottom left of the window. When saving again, the Windows dialog opens in the set directory.

## EXCHANGING DRAWINGS IN AUTOCAD DWG/DXF FORMAT

DWG and DXF are the standard formats in the world of computer-aided drawing, and then you may frequently have to open a *.DWG or *.DXF drawing, and just as frequently have to save an EasyDraw drawing in one of these two formats.

When carrying out these operations, the user should consider that:

1. in the AutoCAD drawings, the **text** is drawn with a font that is used only by this program; when importing, EasyDraw automatically creates a style with the same name as the AutoCAD drawing, but associated with the system font on your computer, typically Arial. This is because the fonts used by Auto-CAD do not always find a correspondence in Windows TrueType fonts (depending on the version). It is advisable to start the STILETESTO command and match the exact font with each style that appears in the dialog box. In fulfilling this task, EasyDraw does the conversion minimizing the variations in the size of the text; however, the appearance of the text (the font), will change.

2. in the AutoCAD drawings, different types of line **may be defined**; EasyDraw carries out the conversion of these types of line into the most similar ones available. The unrecognized line types are turned into a continuous line.

3. after the conversion, the **quote** and the **hatches** in the drawings are transformed into blocks and, therefore, they are not recognized as separate editable entities.

The non-graphical data, except for **layers** and **text styles**, is entirely lost during the conversion. If the drawing has been created with an application that makes use of **extended data** (such as codes associated with entities), always create a backup copy in the native format *. EDR before saving as *.DWG or *.DXF, otherwise the drawing will be unusable by the application, though apparently intact.

## IMPORTING A DRAWING

The IMPORTA command allows you to open a drawing in a different format from *.EDR.

***Command***     ***IMPORTA DWG-DXF***

The formats recognized by EasyDraw are:

| file extension | description |
|---|---|
| DXF | the interchange format of AutoCAD®. |
| DWG | the native format of AutoCAD® (rel. 12-13-14 -2009). |

## EXPORTING A DRAWING

The ESPORTA command allows you to save a drawing in a different format from *.GWC.

***Command***     ***ESPORTA DWG-DXF***

The formats recognized by EasyDraw are:

| file extension | description |
|---|---|
| DXF | the interchange format of AutoCAD®. |
| DWG | the native format of AutoCAD® (rel. 12-13-14-2009). |

## EXPORTING - IMPORTING A BLOCK

***Command***     ***Esporta blocco.***

**How to save a block as a *.BLK file**

The SALVA UN BLOCCO SU FILE command allows you to save a block to a file, which can be invoked by other drawings.
Run the command.
• If the block has already been defined with the CREA BLOCCO command, type its name in the command line.
• When prompted, type the name of the file (with the *.BLK extension) to save the specified block at the previous request.
The generated file will be saved in the directory selected from the window

# PAGE OF THE DRAWING
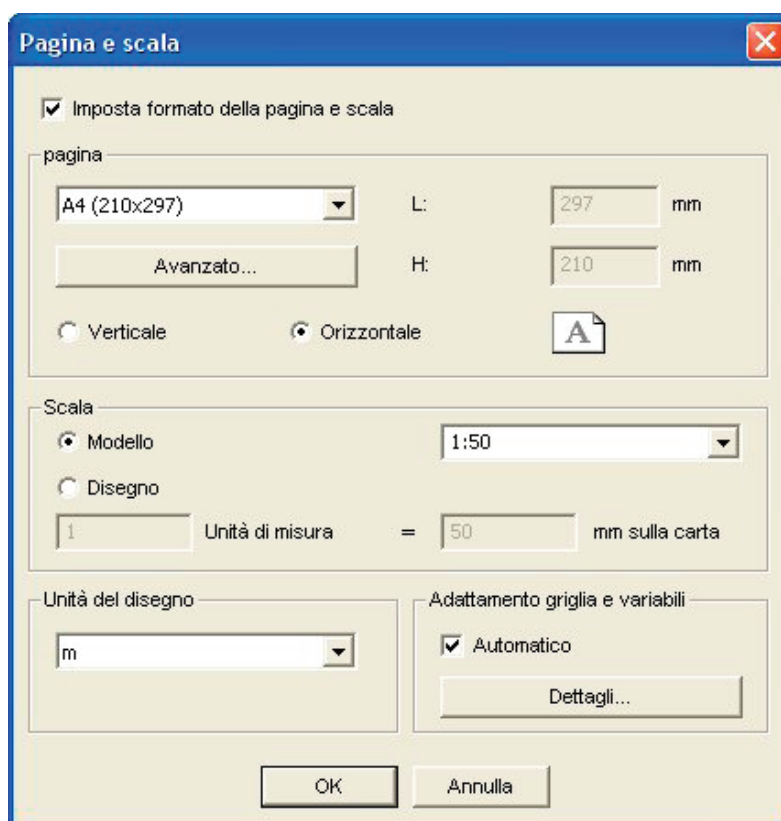
Every time you start a new drawing, it is necessary to set the size of the page where you are going to draw. The IMPOSTA PAGINA command allows to set the format, size, and scale of the page. This command automatically follows the starting of the program and the NUOVO command, but it can also be run independently in a project already started, to change the settings above.

After configuring the settings in the graphic area of EasyDraw, the dimensions of the set page are displayed: the dimensions are virtual and will not, therefore, be printed.

*Command*      *IMPOSTA PAGINA*

**How to set a drawing page**

Define the size of the drawing page choosing among the most commonly used (A0, A1, A2, etc..), or by setting the custom size after selecting *User Format*.



To change the source reference of the page, click *Advanced* (as shown in this manual, EasyDraw shows point 0.0 in the drawing area, otherwise known as origin: the *Advanced* button, allows to specify the point of origin depending on the set page). A second dialog opens:

# PRINTING

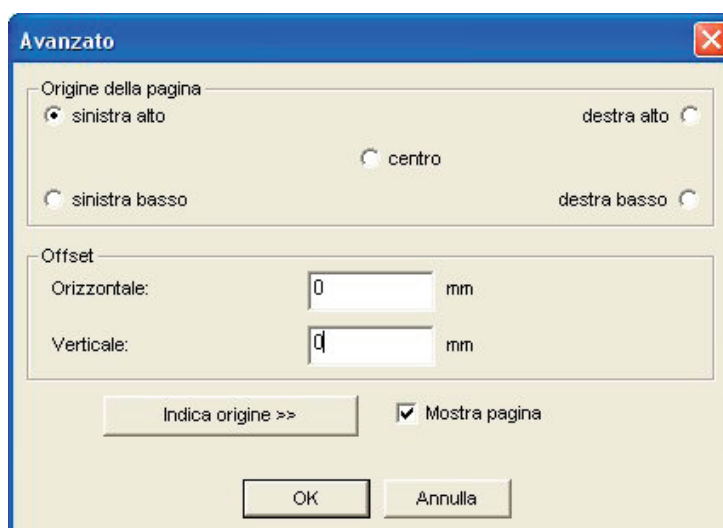EasyDraw allows you to print the drawing with any system printer or plotter installed on Windows. To this end, EasyDraw provides the following commands:
- IMPOSTA STAMPANTE, allowing you to open the Windows dialog box and to configure the print specifications,

- STAMPA, which allows to manage different parameters for printing on paper (reference to the drawing page, scale, sheet margins, color-depth conversion),

- STAMPA VELOCE, which allows to print the drawing directly with the configured device.


*Command*      *IMPOSTA STAMPANTE*

### How to set up the printer
This command opens the dialog box in Windows that allows the selection of the printer or plotter, the specification of the type of port used by the printer, paper size, orientation - portrait or landscape, printing resolution, and more advanced printing options for the printer in use.
The appearance of the dialog box and the related commands are structured according to the Windows default printer.
The margins of the printer can also be set from the STAMPA command.


*Command*      *STAMPA*

The STAMPA command opens a dialog box divided into three sections:
- Page          allows you to print a drawing made with a drawing page, retaining your settings and comparing them with those of the printer from a preview window.
- No page       allows you to print without considering the drawing page and to check the settings on the screen from a preview window.
- Settings      allows you to set the printer margins and the color-thickness conversion of the entities.

### Page section
The preview window displays the dimensions of the drawing page (in red) and of the sheet of paper (in white) including the margins (in grey).
At this point, the user can print the project:
• using the scale of the drawing page and moving the origin of the drawing from the edge of the paper sheet (horizontal and vertical *Offset*);
• adapting the dimensions of the drawing page and the sheet of paper (also considering of the printer margins - see Settings);
• dividing the result on multiple sheets. This option is useful when you want to print a large drawing and your printer supports a smaller format. In *Overlay* you will need to indicate the mm of sheet to be overlayed in order to merge the 4 sheets printed on paper.



### No Page Section
As already mentioned, this section is useful when the user wants to print the drawing without taking into account the dimensions of the sheet set by the IMPOSTA PAGINA command.
*Print All* can be selected when you want to print the drawing with the highest available margins in the printer.
Selecting *Print window* makes it possible to identify (*Indicate* button) a window on the drawing to be printed with the highest available margins of the sheet of paper.
With *Set a scale* you can indicate a ratio between the units of the drawing and the printed mm In the same section is also indicated the origin of the print.

**Set up section**
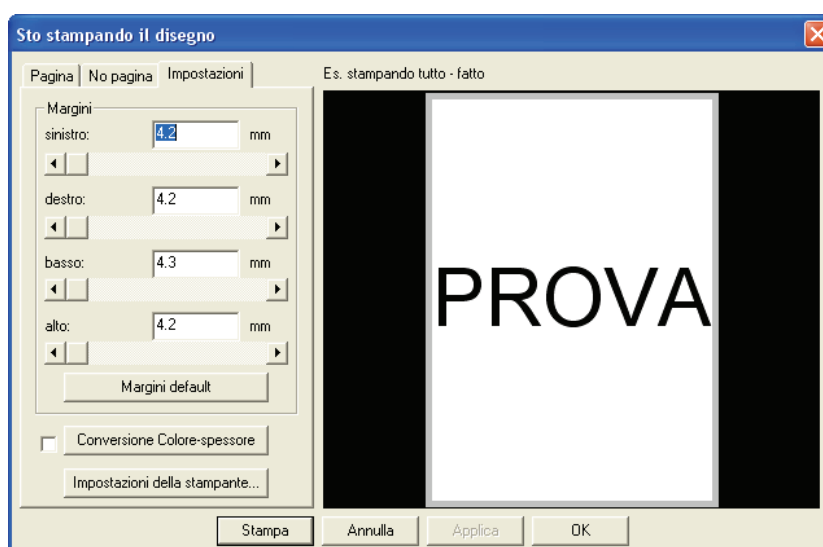


In this section are other essential printing parameters:

- **Margins of the default printer.**
  The user can, in fact, edit the individual print margins and see the results in the preview window, or automatically restore the standard margins of the device by clicking *Standard margins*.

- **Color-thickness conversion.**
  Every color representing each entity on the screen can be converted to a black thickness in the printing stage. The settings are kept indefinitely, unless a change is made.
  To add a color to the conversion list:
    • select the color to be converted in the list from the **Color** title.
    • select the thickness, in mm, to be assigned on paper to the entities of the color selected from the *Thickness mm* list; otherwise, type the value of the desired thickness.
    • press *Add*.
  To remove a color from the conversion list, select the combination and press *Remove*.
  To enable-disable the conversion, check or uncheck *Enable conversion*.

- **Set the printer**, which invokes the IMPOSTA PAGINA command (see above) to modify advanced parameters of the default device.

**How to print a drawing**
Run the STAMPA command.
Define in detail the type of printing you want.
Confirm the settings by clicking *Print*.

*MAINTENANCE OF THE DRAWING*

## COMPACTING THE DRAWING

The COMPATTA command allows you to remove all entities, blocks, layers, quote and text styles, defined in a drawing, but never actually used in it.

**Command     Compatta**

**How to delete unused entities**
Run the command.
Specify the type of entity to be deleted, choosing from the various options available:
B    to delete blocks
L    to delete layers
S    to remove text styles;
Q    to remove quote styles;
ST   to remove hatch styles;
A    to remove the canceled operations;
T    to remove all that is possible to eliminate.

The dialog box displays a brief summary of the deleted entities.

## CHECKING THE DRAWING

The VERIFICA command checks for any damaged entities, blocks, layers, quote, text and hatch styles in a drawing, to be recovered later using the RECU-PERA command.

**Command     Verifica**

**How to check the database:**
Run the command.
If the message  *No problematic objects in the drawing* appears, the objects in the drawing have no problem; conversely, the program lists the number and type of damaged items.

## RECOVERING THE DRAWING

The RECUPERA command checks for any damaged entities, blocks, layers, quote, text and hatch styles in a drawing, to be recovered later using the VERI-FICA command.

**Command     Recupera**

**How to recover a drawing:**
Run the command.
If the message  *No object needs to be recovered* appears, no objects have been recovered because there were no damaged objects; conversely, the pro-gram lists the number and type of recovered items.

## BACKGROUND COLOR

This command allows you to change the background color of the graphic area of EasyDraw.

*Command*  ***COLORE DELLO SFONDO***

**How to change the background color**
Run the command.
When the Windows color map appears, select the new color from the standard ones or create your own.
Confirm with **OK**.

## UNITS OF MEASUREMENT

With the UNITA' DI MISURA command, you can set the unit of measurement and the degree of precision to use it in the drawing.
When starting the command, the screen displays a dialog box where you can choose different types of conventional units of measurement and the precision of the measurements.

*Command*  ***UNITA' DI MISURA***

**How to set the unit of measurement**
Run the command.
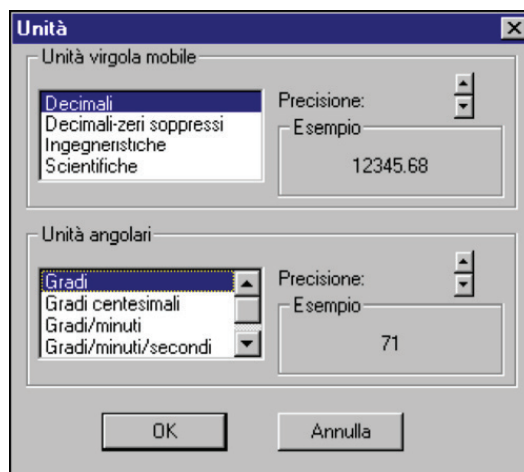


Specify the unit of measurement and its degree of accuracy.
Confirm with OK.

Particular attention deserves the unit of measurement *Decimals - suppressed zeros*. This type of representation is equal to the decimal place, except when the digits after the decimal point are all zeros: in this case, the zeros are not displayed.

## SYSTEM VARIABLES

This command allows you to set EasyDraw system variables, which store information on the current drawing and program configuration (the LTSCALE variable, for example, controls the scale of the dotted lines; the BKCOLOR variable controls EasyDraw's background color). You can change the setting of each variable taking into account its nature: the variable may be integer, point - which expresses coordinates x, y,- for example 19, 2), or string - text), and float (decimal value).

*Command        VARIABILI*



### How to edit a system variable

Run the command.

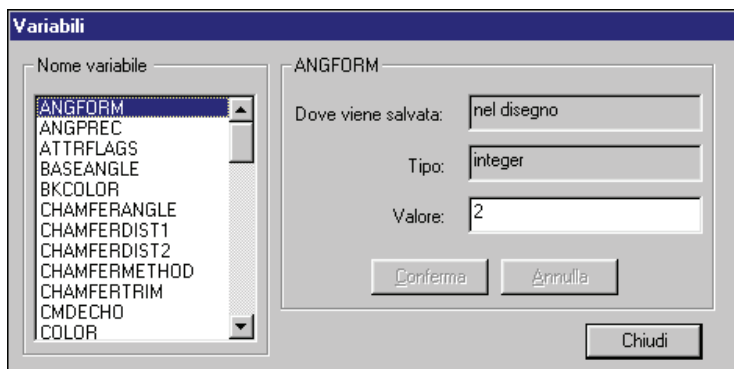When the dialog box appears, select the variable whose value you want to change and check the information that appears, related to the type (integer, point, string, float) and to the saving (in the configuration file, in the drawing with or without the ANNULLA command).

The Value field displays the current value and allows you to edit it, by setting the new value and confirming with *Confirm*. The *Cancel* button resets to the initial value.

The *Close* button is used to end the command.

## EXITING EasyDraw

Exiting EasyDraw means closing the current session of the program.

*Command        ESCI*

### How to exit EasyDraw

Run the command.

Confirm whether to save or discard the latest changes to the drawing before exiting or cancel the closing operation of EasyDraw.

# HOME MENU

## UNDO

EasyDraw allows you to work backwards with the operations performed when editing a drawing. Similarly, it is possible to restore what has been undone, without limitations. The undo command can even be persistent, that is, saved in the drawing file, available for later sessions.
To delete the information from the drawing that allow you to undo/redo the performed operations, use the ELIMINA command.

*Command*     *ANNULLA*

**How to cancel the previous command**
The ANNULLA command cancels the previous step.
Run the command.

## REDO

You can restore the last undone command or multiple commands that have been undone, until returning to the initial situation.

*Command*     *RIFAI*

**How to restore the previously undone command**
Run the command.

How to restore various undone commands
Run the command more than once.

## CUT, COPY, PASTE, also to other applications

You can copy entities to drawings during multiple sessions of EasyDraw, or to other applications. The selected entities are saved to the Windows clipboard. These commands are always available, also in the lower part of the application, located on the right.

**How to cut**

*Command*     *TAGLIA*
The TAGLIA command allows you to cut one or more entities so that you can paste them one or more times to another session of EasyDraw or to another application, deleting them permanently from the source.

**How to copy**

*Command*     *COPIA*
The COPIA command allows you to copy one or more entities so that you can paste them one or more times to another session of EasyDraw or to another application.

**How to paste**

*Command*     *INCOLLA*
The INCOLLA command allows you to paste the entities previously copied with the COPIA and TAGLIA commands one or more times to another session of EasyDraw or to another application.

## DELETE

Using this command you can delete the selected entities. This command is always available, also on bottom right of the application.

*Command*     *CANCELLA*

**How to delete an entity**
Run the command.
Select one or more entities to be deleted in the drawing.
Press <enter> to delete the selection.

*VIEW*

## ZOOM / IN / OUT

The ZOOM command allows to make the content that is displayed in the drawing area look bigger or smaller. EasyDraw allows you to set various types of zoom: by defining a window containing a detail of the drawing, by displaying the entire drawing, or by setting an enlargement/reduction scale.
For convenience, in the View panel there are a few shortcuts:
- Zoom+:                  enlarges by a factor of 2
- Zoom-:                  reduces by a factor of 2
- Previous Zoom:      restores the previous view
- Zoom Extension:    displays all objects in the drawing
- Zoom Page:            displays the entire page set for the drawing
- Redraw:                 regenerates the drawing, without changing the scale factor

In addition, the zoom controls can also be invoked directly from the toolbar at the bottom of the screen.

### *Command       ZOOM, Avvicina, Allontana*

**How to zoom the content in or out using a scale factor**
Run the command.
Specify the enlargement or reduction value typing the scale (value less than 1), or the enlargement factor (value greater than 1) in the command line. The negative scale value involves mirroring the content at line X=Y.

        For example, setting the value:

| | |
|---|---|
| 2 | the drawing is doubled; |
| 0.5 | the drawing is halved; |
| 1 | the view remains unchanged; |
| -1 | the view remains unchanged and the drawing is mirrored; |
| -2 | the drawing is mirrored and doubled; |
| -0.5 | the drawing is mirrored and halved; |

**How to zoom in or out a detail of the drawing through a window**
The content of a rectangular window is maximized to full screen.
Run the command.
Set the first angle of a rectangular cell where to locate the detail of the drawing to be scaled.
Set the second angle, opposite the first, of the rectangular cell.

**How to zoom in or out a detail of the drawing through a window**
The entire drawing can be zoomed in or out from a reference point by dragging the cursor.
Run the command.
Click in the vicinity of the reference point of the dynamic zoom.
Choose the option D or DINAMICO from the command line.
Drag the mouse: the detail gets closer or farther in a clearly visible manner.
Clicking in the vicinity of the point displays the desired detail.

**How to maximize the drawing to full screen**
Allows you to play the largest view of all the drawn entities.
Run the command.
Type E or ESTENSIONE in the command line.

**How to maximize the drawing to the drawing page**
Allows you to expand the view to the entire drawing page.
Run the command.
Type A or PAGINA in the command line.

**How to return to the previous view**
The user can view the previous view. It is understood that the view, and not the previous content of an edited drawing, is returned and that the previous view can be restored after the zoom control has been activated at least once.
Run the command.
Type P o PRECEDENTE in the command line.

## PAN

The PAN command allows to move the current view of the drawing in any direction of the screen, while maintaining its size unaltered.

***Command***     ***PANORAMICA***

### How to move the current view in a dynamic way
Run the command.
Specify the reference point for the movement.
Drag the mouse: the entities are "hooked" to the cursor.
Click in the vicinity of the point displaying the desired view.

## REDRAW

This command has the function to regenerate the drawing by clearing the screen, updating the entities, and showing the non-visible changes applied to certain entities.

***Command***     ***Ridisegna***

## *TOOLS*

## EDITING ENTITY PROPERTIES

EasyDraw allows you to edit the properties of selected entities, both individually and in groups. The modification of the properties of different entities, such as a line, a text and a quote, may also be limited to the common properties: in this case, only color and layer. If the system for modifying the common properties isn't adopted, but the "total" one is, the changes will affect only those entities that require it (if you change the text style when selecting a line and a text, the change will affect only the text).
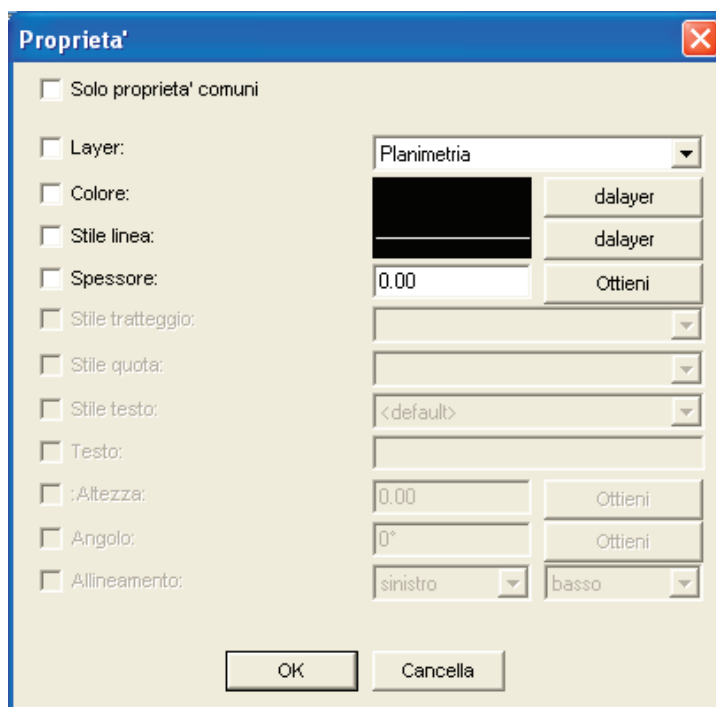
***Command***     ***PROPRIETA'***

### How to edit the properties of an entity
Run the command.
Select the entity or entities whose properties you want to change and confirm with <enter>.
Follow the instructions in the dialog box.

| | |
|---|---|
| **Only common properties** | by enabling this box, only the common properties to the selected entities can be accessed. Normally it is disabled. |
| **Color** | operate the button with the current color and edit it for the selected entities (the change does not affect the block definitions). |
| **Linetype** | operate the button with the current linetype and edit it for the selected entities (the change does not affect the entities such as text, quotes, attributes, etc.). |
| **Layer** | select the layer you want to change in the entity among those listed. |
| **Text style** | select the text style to be changed from those listed (only applies to text and attribute entities). |
| **Quote style** | select the quote style among those listed (only applies to quote entities). |
| **Hatch style** | select the quote style among those listed (only applies to hatch entities). |
| **Height** | Enter the new height for the entities or press the *Select* button to locate it with the filters on the current drawing (INT, MED, CEN, etc.) (only applies to text or attribute entities). |
| **Thickness** | Enter the new thickness for the entities or press the *Select* button to locate it with the filters on the current drawing (INT, MED, CEN, etc.) (only applies to purely graphic entities: lines, arches, circles, etc.). |
| **Angle** | Enter the new angle of inclination for the entities or press the *Select* button to locate it with the filters on the current drawing (INT, MED, CEN, etc.) (only applies to text or attribute entities). |
| **Align** | select the new justification for the selected text or attributes. |
| **Text** | type the new content for the selected text or attribute entities. |

## LAYER

A layer is the fundamental property of each drawing entity of EasyDraw. Layers are similar to overlays; think of a project whose different parts are madeof **transparent overlays**: at any time you can remove (or disable) one of these sheets, to make the drawing more understandable or simpler. When drawing with EasyDraw, the entities are always drawn on one layer, if no layer is defined, a "standard" one is created, called *<default>*. For each layer, the *color and linetype* properties are associated. When the color and *linetype* properties on a drawing entity have *dalayer* value, they automatically take on the values associated to the layer they belong to. As a consequence, properly distributing the entities along thematic layers allows you to draw with great efficiency: when drawing a planimetry, for example, by tracing all the lines representing the walls of a layer (e.g. walls), you can vary their color with a single operation, as well as make them invisible (frozen layer), or not editable (locked layer).
For each layer, you can assign a name and display it in various combinations with the others (this could also be used when printing, when you want, for example, remove the construction lines drawn on a specific layer).

## Control of the LAYERS

In the TOOL panel of the HOME menu, EasyDraw provides, a quick access dropdown list of the properties of the layers
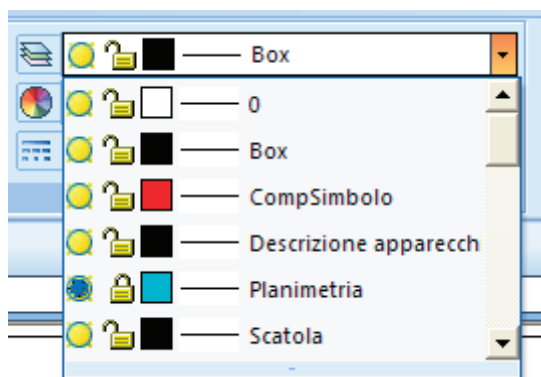
Through this list, you can:
- Make a layer current
  Select the description of the layer to make it current
- Display the color associated with the current layer (in the form of a square of the same color as the layer)
- View the linetype associated with the layer
- View and change the locked/unlocked status of a layer
  Click on the padlock icon to lock (padlock) or unlock (open padlock) a layer.
- *View and change the* frozen/unfrozen status of a layer

Click on the symbol of the sun to freeze (solar eclipse) or unfreeze (visible sun) a layer.

The LAYER command invokes the setting options respectively at the command line and in a user dialog
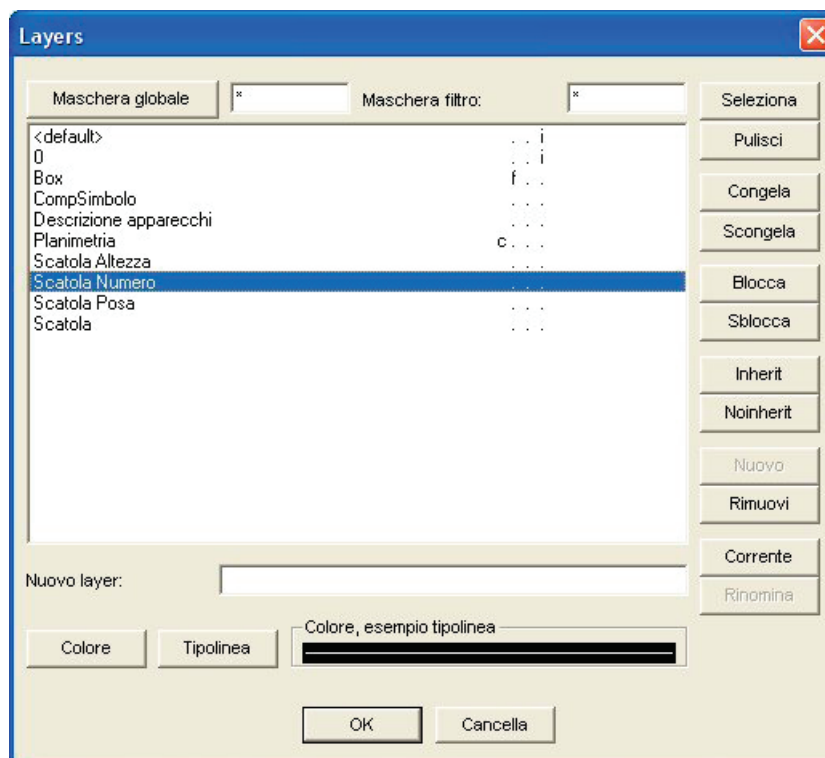


## Command    LAYER

### How to create a new layer

When organizing your own set of layers in a drawing, it is good to pay attention to the name to be assigned to each layer, as EasyDraw automatically sorts them into the window.
Initially, EasyDraw provides a default layer (called, precisely, default), white in color and with continuous linetype.
Run the command.
Type the name of the new layer to be created at the bottom of the dialog box under *New*.
The layer names can have a maximum length of 31 characters and contain special characters such as spaces, parentheses, hyphen (-), underscore (_), etc.
Press *New* to add the typed layer name to the list.

### How to rename a layer

At any time, you can change the name of a layer of a drawing without changing its characteristics.
Run the command.
Select the layer you want to rename by clicking its name.
Type the new name for the New entry.
Press *Rename* button to assign the name to the selected layer.

### How to make a layer current

EasyDraw is designed to always maintain a current layer, or a layer where to place the new drawing entities. If a different layer is made current, the new entity will be associated to the new layer.
It is important to point out that a frozen layer cannot be madecurrent (see next paragraph).
Run the command.
Select the layer you want to make current from the list in the dialog box.
Press *Current* to make the selected layer current.
or
Select the name of layer to be made current directly from the status bar (where the icon of the LAYER command is placed).

**How to freeze and unfreeze a layer**

As explained in the introduction, a layer can also be invisible while still being in the list of layers in a drawing. If you want less clutter on the page while drawing, you just need to freeze the layers that are not currently used. When a layer is frozen, entities are invisible.

Freezing the layers allows you to run the VIEW menu commands faster, facilitate the selection of the visible entities, and reduce regeneration time for complex drawings (the regeneration time is proportional to the size of the drawing).

Run the command.
Select the layer (or layers) to unfreeze from the list in the dialog box.
Press *Freeze and unfreeze* respectively to freeze and unfreeze the selected layers

**How to lock or unlock a layer**

Locking a layer means leaving a layer visible, but preventing it from being edited. The lock can be useful when you want to edit entities associated with particular layers, but also when you want to display entities on other layers. The entities on a locked layer cannot be edited, but they are still visible if the layer is unfrozen. A locked layer can be made current, in order to add new entities and use request commands on the existing ones.

The characteristics of a layer, such as color, line type, and status can be edited at any time.
Run the command.
Select the layer (or layers) to be locked from the list in the dialog box.
Press *Lock and Unlock* respectively to lock and unlock the selected layers

# COLOR
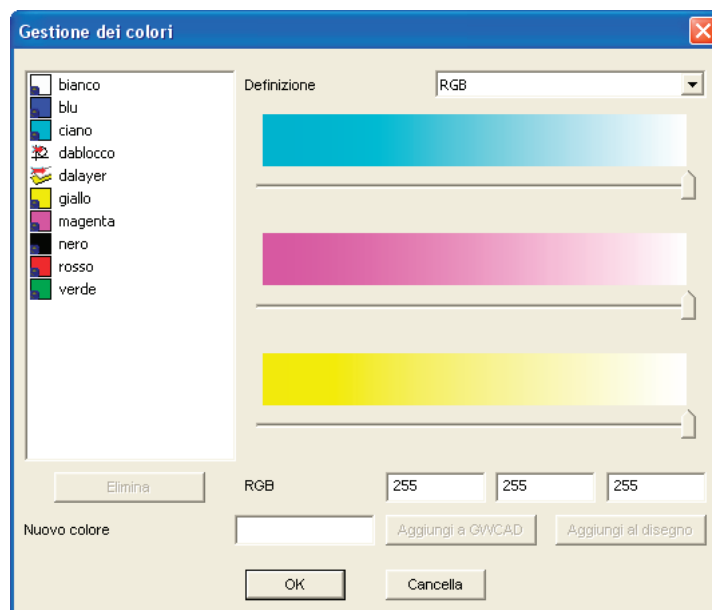
*Command*      *COLORE*

**How to set the working color**

This command allows you to set a color so that the newly drawn entities take that color.
Normally, the working color is DALAYER.

 * Run the command.
 * Type option **D** to open the dialog box for selecting the color, or type the name of the color, or its RGB values (e.g. 130,40,200).



 * Select how to define the color and use the cursors, or act on the RGB valuesand confirm with **OK**.
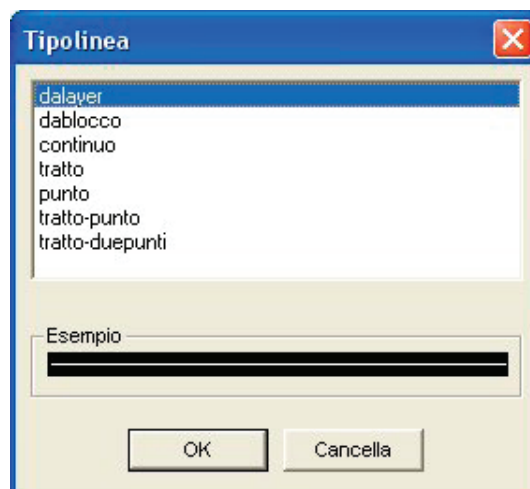 * By assigning a name to the defined color, you can save it to the current drawing (Add to drawing) or permanently to the program (Add to EASY-DRAW).

## LINE TYPE

***Command    Tipo di linea***

**How to set the working line type**
  * Run the command.
  * Type option **D** to open the dialog box for selecting the line type, or type the name of the line type to be set directly.



  * Select the line type to be set among those proposed (normally DALAYER)..

## GRID

The GRIGLIA command allows you to set a grid of points that are shown in the graphics area. The position of the first point (origin) and the cell size (rectangle described by four points) are the parameters that determine the size of the grid.

The cursor is automatically hooked to the grid points when the *Magnet* option is active. The grid is virtual and it does not represent an entity of the drawing, so it cannot be selected, deleted, or printed. The grid is represented when the *Show Grid* option is active, by means of a different color from the one of the set background of EasyDraw.

***Command    Griglia***

**How to set the grid cells**
Run the command.



Insert the cell size along the X and the Y axis in the *Cell* fields (as in the example: 20.00, 20.00). To edit the cell, you must enter two other values or use the *Indicate cell* >> button, which allows you to drag the cursor to define the two dimensions simultaneously.
Manually enter the coordinates of the point of origin of the cell in the *Origin* fields (for example: 0.00, 0.00). To change the point of origin, type another coordinate or use the *Indicate origin* >> button, which allows you to specify a point in the graphic area.
Enable or disable the *Show grid* option if you want to view it.
Enable or disable the *Magnet option on the grid* if you want to hook the points between the grid and the cursor. It is important to remember that the magnet is active when the points are being placed (for example, while drawing the entities).

*Command*     *Griglia on/off*

### How to display/hide the grid

The GRIGLIA ON/OFF command allows to show or hide the configured grid using the IMPOSTA GRIGLIA command This command acts as an on/off switch. The same function can be achieved by starting the IMPOSTA GRIGLIA command and by enabling or disabling the *Show Grid* option.

*Command*     *Calamita su griglia on/off*

### How to enable/disable the limited movement of the cursor

It is possible to set the movement of the cursor in increments according to the set distances between the grid points. The cursor is positioned at the grid point that is closest to its location on the screen. This command acts as an on/off switch. The same function can be achieved by starting the IMPOSTA GRIGLIA command and by enabling or disabling the *Show Grid* option.

The grid magnet, like the filters, is active only during the request for placing the points (for example, while drawing entities).

## ORTHOGONAL

The ORTO command limits the movements of the cursor on the vertical or horizontal axis. This command facilitates the drawing of parallel or perpendicular entities to each other. It is not considered when tracing lines expressly indicating the coordinates in the command line and it acts only on the second point of an entity that the user is drawing.

*Command*     *Ortogonale*

### How to enable the orthogonal mode

Run the command.

### How to disable the orthogonal mode

This command can only be disabled if previously enabled. The cursor movement resumes without constraints.
Run the command.

## DISTANCE

By specifying two points, the command provides the distance between them and the difference between the values of the respective abscissa and ordinate.

*Command*     *Distanza*

### How to measure the distance between two points

Run the command.
Specify the reference point.
Specify the second point. The three measures appear at the command line.

## LIST OF PROPERTIES

This command lists the set of information relating to one or more selected entities.

| type of entity | listed characteristics |
| --- | --- |
| Line | entity type, layer of origin, coordinates of the start point and end point, thickness of the line. |
| Circle | entity type, layer of origin, center point coordinates and radius length, thickness of the line. |
| Arch | type of entity, layer of belonging, center coordinates, radius length, rotation angle of the initial and the final point, thickness of the line section. |
| Solid | type of entity, layer of belonging, coordinates of the four points of the polygon. |
| Text or Attribute | type of entity, layer of belonging, text style, coordinates of the reference point, text height, angle of rotation, alignment with respect to the reference point. |
| Block | type of entity, layer of belonging, color and line type, block name, coordinates of the reference point, scale, rotation, mirrored yes/no. |
| Quote | type of entity, layer of belonging, quote style, coordinates of the quoted points, quote dimensions. |
| Hatch | type of entity, layer of belonging, hatch style, number of sides that make up the hatched polygon, coordinates of the points that define the polygon. |
| Group | type of entity, layer of belonging, color and line type, coordinates of the reference point, scale, rotation, mirrored yes/no. |

**Imagine** type of entity, layer of belonging, name defined in the raster image manager, reference parameters (center, angle of rotation, scale, mirrored yes/no), parameters of the polygon (coordinates of the points that define the image).

*Command* *Lista Delle Proprieta'*

**How to display the list of information of an entity**
Run the command.
Select the entity for which you want to get the list of information, by clicking near it.
Confirm with <enter> to get the list of features to the command window.

## COMMAND HISTORY

When running commands that provide for the display of text strings, such as LIST, it is necessary to temporarily expand the text area, to be able to read the entire content. The STORIA command displays, in a separate window, the entire sequence displayed in the command area at EasyDraw startup, with a limit of 5000 characters.

*Command* *STORIA DEI COMANDI*

*Command* *AIUTO*

This command opens this explanatory manual.

## *INFO*

## MANUALE DI ISTRUZIONE ALL'USO DI EASYDRAW

This command opens this instructions manual in PDF format.

## VERSIONE

This command opens a box that summarizes the version references and software copyright.
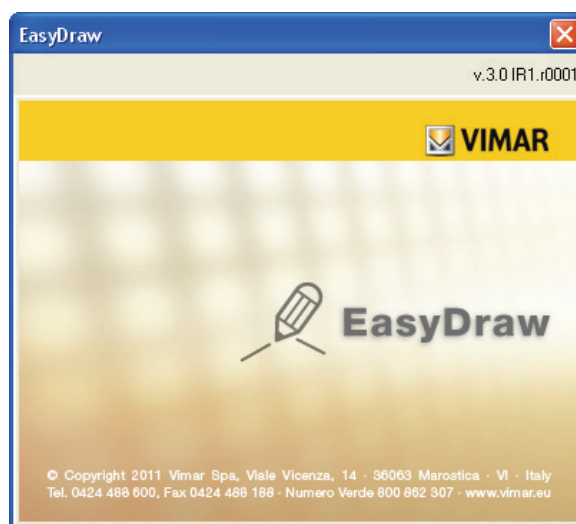
# EDIT MENU

## DUPLICATE

The DUPLICA command allows you to copy one or more entities, one or more times, within the current drawing.

*Command        DUPLICA*

### How to duplicate an entity once (single copy)
Run the command.
Select one or more entities that need to be duplicated within the drawing and click <enter> to abort the selection and continue with the command.
Determine the reference point of the entity to be duplicated, specifying a point on it.
Specify where to place the new duplicated entity.

### How to duplicate an entity multiple times (Multiple Copy)
Run the command.
Select one or more entities that need to be duplicated more than once within the drawing and <enter> to abort the selection and continue with the command.
Type M or MULTIPLO in the command line.
Determine the reference point of the entity to be duplicated, specifying a point on it.
Specify the points in succession where to place the new duplicated entity.

## SERIES

This command is used to draw a series of entities or selection groups according to a rectangular or polar route.

*Command        SERIE*

### How to draw a rectangular series
This is the default method used by EasyDraw for the series arrangement of an entity or a selection group, prompting the user for the number of rows and columns for placing the series. It can be sized by defining the dimensions of the entire series or of the single cell.
Run the command.
Specify the entity to be placed in series
Type R or RETTANGOLARE.
Specify the number of rows.
Specify the number of columns.
Type option A or AVANZATO to define the method of representation of the series. It is possible to:
refer to the single cell (option C or DEFINISCI CELLA);
refer to the entire series (option S or DEFINISCI CELLA);
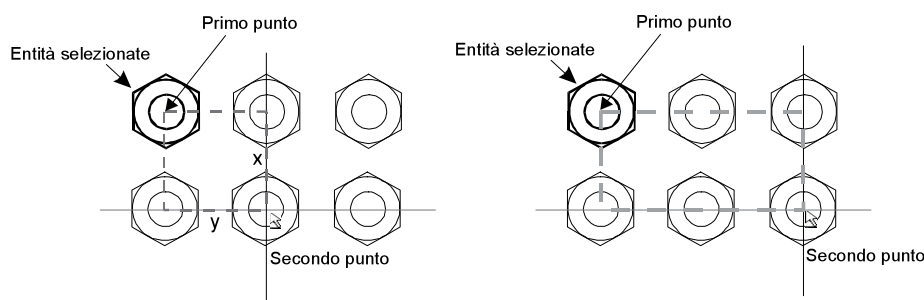


Option B or ANGOLO BASE in the ADVANCED submenu is used to set the value of the angle of rotation of the entire series, indicating a point in the graphical area with a click, or by typing a value between 0 and 360° in the command line.
The R or PUNTO DI RIF. option in the ADVANCED submenu is used to change the point of reference of the entities for drawing them in series, indicating a point in the graphical area with a click, or by typing the coordinates in the command line.
After defining the method of representation, the base angle, or point of reference of the entity, set the size of the cell or the entire series by typing the coordinates in the command line or by dragging the mouse cursor.

### How to draw a polar series
This is the default method used by EasyDraw for the polar arrangement of an entity or a selection group, prompting the user for the number of copies, the angle of rotation, and whether the copies should be rotated.
Run the command.

Specify the entity to be placed in series
Typr P or POLAR.
Enter the number of angular steps, or the number of copies to be generated, including the original. This value divides into segments the angle that will be set later.
Specify the number of radial steps. If the value is <1> a single concentric series is obtained with respect to the central point of the series.
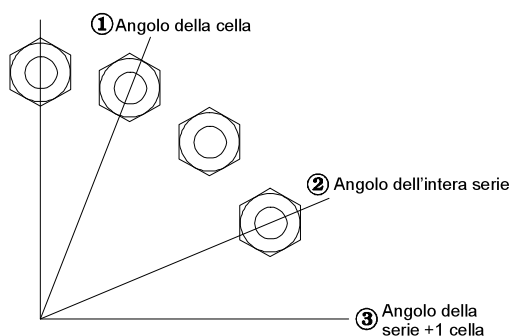Define the center point of the series.
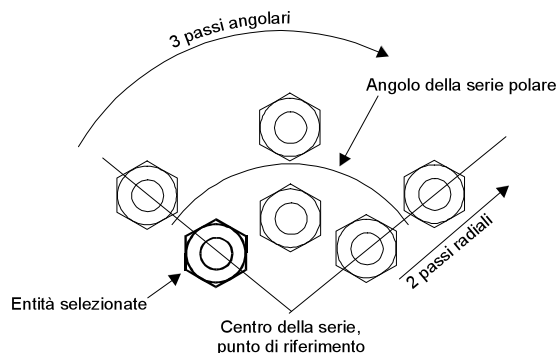


Type option A or AVANZATO to define the method of representation of the series. It is possible to:
Define the angle of the single cell (option C or DEFINISCI CELLA);
Define the angle of the entire series (option S or DEFINISCI SERIE);
Define the angle of the entire series + 1 extra cell (option U or ANGOLO PIU' UNO).



Option P or ANGOLO PIENO is used to position the entity in series dividing a round angle by the number of angular steps previously set, so as to automatically represent the equidistant entities.
Option B or ANGOLO BASE in the ADVANCED submenu is used to set the value of the angle of rotation of the entire series, indicating a point in the graphic area with a click or by typing a value between 0 and 360° in the command line.
The R or PUNTO DI RIF. option in the ADVANCED submenu is used to change the point of reference of the entities for drawing them in series, indicating a point in the graphical area with a click, or by typing the coordinates in the command line.
After defining the method and modes of representation, set the size of the cell or the entire series by typing the coordinates in the command line or by dragging the mouse cursor.
Confirm whether or not to rotate the entities around the central point.

## MOVE
This command lets you move an entity from one position to another

*Command        SPOSTA*
**How to move an entity**
Run the command.
Select one or more entities that need to be moved within the drawing and <enter> to abort the selection and continue with the command.
Specify the reference point for the entire selection.
Specify the new position.

# ROTATE

This command allows to rotate an entity or a selection group according to an angle set by the user or by referring to the angle of rotation of an existing entity.

***Command***        ***RUOTA***

**How to rotate an entity**
This is the default method to rotate an entity.
Run the command.
Select the entity to rotate.
Specify the reference point for the rotation.
Set the value of the angle in degrees in the command line or specify the angle width by dragging the cursor.
Choose whether to delete the original entity. The default is the last confirmed.

**How to rotate an entity according to a reference**
You can also rotate an entity with reference to the rotation angle of an existing entity.
Run the command.
Select the entity to rotate.
Specify the reference point for the rotation.
Type R in the command line and press <enter>.
Specify the reference point for the entity of reference.
Specify the angle of rotation for the entity of reference.
Choose whether to delete the original entity. The default is the last confirmed one.

# MIRROR

This command allows you to mirror one or more selected entities along an axis defined by two points. Please note that the text type entity is not mirrored but it is kept readable from left to right. For mirror copies with axes of 0°, 90°, 180° and 270°, we recommend the orthogonal mode (see command ORTO).

***Command***        ***SPECCHIA***

**How to mirror an entity**
Run the command.
Select the entity to mirror.
Specify the first point of the mirror axis with respect to which mirror the entity.
Specify the second point of the mirror axis.
Determine whether to delete the original entity. The default is the last confirmed one.

# SCALE

You can reduce or enlarge an entity or a selection group by setting a scale factor or a reference.

***Command***        ***SCALA***

**How to scale an entity**
This is the default method to scale an entity. Setting a scale factor greater than 1, the entity is enlarged, while it decreases if the scale factor is less than 1. In any case, it involves the change of the original quotes.
Run the command.
Select the entity to scale.
Specify the reference point.
Set the scale factor in the command line or drag the cursor.
Choose whether to delete the original entity. The default is the last confirmed one.

How to scale an entity according to a reference
You can also scale an entity with reference to the size of an existing entity.
Run the command.
Select the entity to scale.
Specify the reference point.
Type R in the command line and press <enter>.
Specify the reference point for the entity of reference.
Choose the size of the entity of reference.
Choose whether to delete the original entity. The default is the last confirmed one.

## STRETCH

Using this command you can stretch the selected entities.

*Command        STIRA*

**How to stretch an entity**
Run the command.
Select one or more entities that need to be stretched from a selection window.
Specify the reference point for the movement to be made.
Specify the second point of the movement.

## BREAK

The SPEZZA command is used to delete a part of the entity by indicating the cutting points.

*Command        SPEZZA*

**How to break an entity**
Run the command.
Select the entity to break.
EasyDraw interprets the point indicated for the selection as the first cutting point.
Specify the second cutting point.

## CROP

This command performs the cutting of one or more entities at the points indicated by the user (cutting edges).

*Command        RITAGLIA*

**How to crop an entity**
Run the command.
Select the entities to be used as the cutting edges of the entity to crop.
Select the entities to be cropped one by one.

## EXTEND

The ESTENDI command lengthens the selected entities until reaching certain limits defined by other entities.

*Command        ESTENDE*

**How to extend an entity**
Run the command.
Identify the entities to be reached, i.e. the entities that will limit the extension.
Select the entity to extend. The extension takes effect immediately.
Confirm with <enter> to stop the action of the command.

## CONNECT

The RACCORDA command allows you to connect two lines, circles, arches, and solids by means of an arch with a specified radius. In addition, this command fits the lengths of the original entities so that they terminate exactly on the connection arch.

*Command        RACCORDA*

**How to connect two line segments with a cut**
EasyDraw makes the cut by default.
Run the command.
Specify the connecting radius by typing the value in the command line or by confirming the default value.
Select the first line to be connected.
Select the second line to be connected. To abort the command, press <enter>.

**How to connect two line segments without a cut**
The N or NONTAGLIA option allows to connect two entities without cutting.
Run the command.

Type N or NONTAGLIA in the command line.
Specify the connecting radius by typing the value in the command line or by confirming the default value.
Select the first line to be connected.
Select the second line to be connected. To abort the command, press <enter>.
The NONTAGLIA option remains in effect until the option T or TAGLIA is specified.

**How to connect two circles or two arches**
EasyDraw connects the circles and arches in the same manner as it links two lines. Depending on the selected points and on the radius, several connections are possible between entities.

## TRIM

The CIMA command connects two non-parallel lines between them, extending or trimming them to intersect or merge them to a trimmed line. You can keep the lines intact or cut them and extend them to the trim line.
Trim distances indicate the distance between the point of intersection of the two lines and the point where the trim line must be intersected.
If both trim distances are zero, the two lines are extended to the point of intersection and trimming is not performed.

*Command*          *CIMA*

**How to trim two lines with a cut**
EasyDraw makes the cut by default.
Run the command.
Specify the first trim distance by typing the value in the command line or by confirming the default value.
Specify the second trim distance by typing the value in the command line or by confirming the default value.
Select the first line to be trimmed.
Select the second line to be trimmed. To abort the command, press <enter>.

**How to trim two lines without a cut**
The N or NONTAGLIA option allows to connect two entities without cutting.
Run the command.
Type N or NONTAGLIA in the command line.
Specify the first trim distance by typing the value in the command line or by confirming the default value.
Specify the second trim distance by typing the value in the command line or by confirming the default value.
Select the first line to be trimmed.
Select the second line to be trimmed. To abort the command, press <enter>.
The NONTAGLIA option remains in effect until the option T or TAGLIA is specified.

**How to trim specifying length and trim angle**
You can trim the two lines specifying at what point of the first line the trim line starts and the angle they form together.
Run the command.
Specify the first trim distance by typing the value in the command line or by confirming the default value.
Type A or ANGLE in the command line.
Specify the trim angle that is to be formed between the first line to be trimmed and the trim line.
Select the first line to be trimmed.
Select the second line to be trimmed. To abort the command, press <enter>.

## OFFSET

The OFFSET command automatically draws an entity similar to another selected, at a user-defined distance; in other words, we can say that the command performs the offset of an entity.
You can offset lines, circles, arches. The offset of the circles creates larger or smaller concentric circles, depending on the side that is confirmed: in fact, if a point inside the perimeter is indicated, the circle will be smaller; on the contrary, if a point external to the circumference is indicated, the circle will be larger.

*Command*          *OFFSET*

**How to offset an entity specifying the distance**
Run the command
Select the entity to offset.
Specify the location where you want to place the offset entity, by clicking with the mouse pointer, or by typing the value of the offset distance in the command line.
Indicate the side where to place the offset of the entity compared to the source entity, or vary the distance.
Select another entity to offset or type <enter> to end the command.

## EXPAND BLOCK

*Command*      ***ESPLODI BLOCCO***

**How to expand the block**
Using the ESPLODE command you can break down a block reference into the entities that compose it.
By expanding a block reference, entities can be edited or added to those that define it.
* Run the ESPLODI BLOCCO command.
* Select the blocks to be expanded on the current drawing.

If attributes are also defined in the expanded block, their values will be lost. If a block is expanded by mistake, you can retrieve all information with the AN-NULLA command.

## BLOCK ATTRIBUTES

*Command*      ***MODIFICA STRUTTURA***

**How to change the already defined attributes**
When an attribute is not yet associated with a block, you can modify its characteristics.
Run the MODIFICA STRUTTURA command.
Type the capital letter corresponding to the characteristic to be edited.
If necessary, type the changes.

*Command*      ***MODIFICA CONTENUTO***

**How to change the already defined attributes**
If attributes were defined in a block, even if they already have a value (such as, for example, the one by default) it can be changed.
Run the MODIFICA CONTENUTO command.
Select the block to modify the attributes.
Select the attribute to be modified in the dialog box and type in the new value.

*Command*      ***MODIFICA VALORE***

**How to edit the attribute valuesin a block**
If attributes were defined in a block, even if they already have a value (such as, for example, the one by default) it can be changed with the EDITA ATTRIBUTI NEI BLOCCHI command.
Run the command EDITA ATTRIBUTI NEI BLOCCHI.
Select the block to modify the attributes.
Select the attribute to be modified in the dialog box and type in the new value.

*Command*      ***SPOSTA ATTRIBUTO***

**How to move an attribute within a block without expanding it**
Run the SPOSTA ATTRIBUTO command.
Select the attribute you want to move and place it freely.

*Command*      ***SCALA ATTRIBUTO***

**How to edit the size of an attribute within a block without expanding it**
Run the SCALA ATTRIBUTO command.
Select the attribute you want to scale and indicate its new size.

*Command*      ***ATTRIBUTO ON/OFF***

**How to edit the visibility of an attribute within a block without expanding it**
Run the ATTRIBUTO ON/OFF command
Select the block: the list of attributes to hide/show will be presented.

## DRAWING AIDS: FILTERS

Filters are tools that operate on point insertion; the points indicated by the user undergo the transformations to meet the demands of the user. For example, if you want to start a line from the center of a circle, when prompted to insert a starting point, simply select the filter *center of*, and then touch the circle
Filters allow you to easily find the reference points of the entities shown in the drawing, and more.

The filters can be invoked as independent controls, since they remain active until another selection is made, or they can be used in a transparent manner within another command, when a point is requested, and be automatically disabled when done with the insertion.

Filters can be invoked by typing in the command line, by selecting the corresponding icon on the "filters" bar, or from the corresponding menu. The Easy-Draw filters can also be operated combined with each other: for example, if you have to locate a point with the same X coordinate of the END of a line and the same Y coordinate of the CENTER of a circle.

EasyDraw can operate in STICKY mode, automatically displaying the reference points of the entity where the cursor is located. Near the cursor itself appears a message informing about the type of point identified; to confirm the entered reference point, press the left mouse button.

To limit the action of the STICKY only to the desired entities, only the last 15 entities "hovered" by the cursor are considered. To start a line from the center of a circle when prompted to enter the starting point, it will be sufficient "hover" the cursor on the circumference of the circle (thus including it in the group of entities on which STICKY acts), then approach the center of the circle; when the word *center* appears near the cursor, press the left mouse button.

## AUTOMATIC FILTERS (STICKY)

The STICKY command allows the automatic recognition of the reference points of the entity, as soon as the cursor approaches them. This command enables all filters available in EasyDraw except ALLINEA (note the icon menu of the filters); to exclude unwanted filters, click the icons that represent them. To disable the automatic recognition, run the command from the icon again.
The STICKY mode also activates the CONTINUATION (CONTIN) filter which allows to place a point on the continuation of an arch or a line section.

*Command*          ***FILTRI AUTOMATICI***

**How to enable the automatic recognition of geometric points**
Run the command from the icon.
Exclude any unwanted filters.

How to enable the automatic recognition of geometric points
Run the command from the icon.

## END POINT

A filter that allows to locate the closest end to the selected point on an entity.

*Command*          ***FINE***

## MEDIUM POINT

Filter which allows to hook the midpoint of a line or arch.

*Command*          ***MEDIO***

## CENTER

Filter which allows to detect the center of a selected circle or arch.

*Command*          ***CENTRO***

## INSERTION POINT or BASE POINT

A filter that allows to locate the insertion point of a block (originally indicated when defining the block), a text (depending on the type of justification/style), or an attribute. If you select an internal attribute of a block, EasyDraw identifies the insertion point of the attribute and not the block.

*Command*          ***Inserimento***

## INTERSECTION POINT

A filter that allows to find the point of intersection between two entities such as lines, circles, and arches. If the entities do not intersect visibly, EasyDraw equally locates the point of intersection, imagining to extend the entities until their hypothetical intersection.

*Command*          ***Intersezione***

## NEAREST POINT

This filter allows to locate the nearest point to the selection and belonging to a line, arch, circle, or solid.

**Command**          **Vicino**

## PERPENDICULAR POINT

A filter that allows to find the point that, along with the previously selected one, detects the normal line to the selected entity.

**Command**          **Perpendicolare**

## TANGENT POINT

A filter that allows to find the point that, along with the previous one selected, detects the tangent line to the selected entity.

**Command**          **Tangente**

## ABSCISSA

Filter which returns the coordinates of a selected point on the X axis and limits the next entry to the single Y coordinate.

**Command**          **ASCISSA**

## ORDINATE

Filter which returns the coordinates of a selected point on the Y axis and limits the next entry to the single X coordinate.

**Command**          **ORDINATA**

## PARALLEL POINT

A filter that allows you to locate a point that, virtually joined to another point identified above, defines a parallel to the reference entity.

**Command**          **Parallelo**

## VIRTUAL INTERSECTION

A filter that allows you to locate the virtual point of intersection, which is the point that would be obtained if two or more entities were extended until they meet.

**Command**          **Intersezione virtuale**

## ALIGNED

A filter that limits the movement of the cursor to a line or an arch when drawing an entity. It only works as a transparent command and it can be invoked multiple times.

**Command**          **ALLINEATO**

Enabling this filter once, the user has the possibility of aligning:
- to a line, indicating the extremes in succession.
- to an entity hatch, by selecting the entity itself or by the E or ENTITA' option;
- to an arch, indicating the points of an arc with the A or ARCH option;
- to a circle by specifying the center and the radius by choosing C or CENTRO.
Activating the filter a second time, the cursor is limited to the intersection of the two selections.
The R or RIMUOVI option allows to remove a previous selection.

# DRAWING MENU

## LINE

A line is the simplest entity in EasyDraw and it is characterized by a start point, an end point, and thickness, as well as by layer, color, and line type. The LINEA command allows to draw more lines in sequence, matching the end point of a line with the start point of the next. Each segment of the broken line, however, is an autonomous entity.

*Command*          *LINEA*

**How to draw a line segment**
Run the command.
Specify the start point.
Specify the end point.
Press <enter> to end the command.

**How to draw consecutive line segments**
Run the command.
Specify the start point.
Specify the end point.
Specify the end point of each successive segment
**or**
<enter> to end the command and interrupt the sequence of the line segments
**or**
type C or CHIUDI in the command line for ending a line with precision: in fact, the command ensures the precise connection between the first point and the last vertex of the sequence of segments.

**How to change the line thickness**
During the execution of the command, and always before a new segment, you can vary the thickness of the line. EasyDraw lets you get, then, consecutive segments of different thickness, but not segments with different thickness between the start point and end point.
Run the command.
Type S or SPESSORE in the command line.
Enter the new thickness value for the next segment.
Press <enter> to end (the new thickness setting is saved), or continue drawing one line segment or more consecutive segments.

## CIRCLE

EasyDraw allows to draw a circle by defining three points on the circumference, or by determining its center and its radius. As for the LINEA command, also the circumference of the circle may have a thickness.

*Command*          *CERCHIO*

**How to draw a circle by center and radius**
This is the default method that EasyDraw uses for drawing a circle. It consists of defining first the center of the circle, and then its radius.
Run the command.
Specify the center of the circle.
Specify the radius by entering its value or by dragging the cursor (on screen you will clearly see the size of the circle vary).

**How to draw a circle passing through three points**
This method consists in drawing a circle passing through three required points in succession.
Run the command.
Type 3 or 3PUNTI in the command line.
Specify the first point.
Specify the second point.
Specify the third point.

**How to change the circumference thickness**
As for the line, also the thickness of the circumference can be changed.
Run the command.
Type S or SPESSORE in the command line and press <enter>.
Enter the new thickness value for the circumference to be drawn.
Press <enter> to end (the new thickness setting is saved), or draw the circle using one of two methods described above.

**How to draw a circle using a reference point of another entity**
Thanks to the filters, it is possible to draw a circle having a common point to a geometric point of another entity or tangent to one or more entities (three at the most, using the method of the three points).

## ARCH

EasyDraw allows you to draw an arch passing through three points or determining the center and two points. As for the LINEA and CIRCLE commands, also the arch may have a thickness.

*Command        ARCO*

### How to draw an arch with a center and two points

This is the default method that EasyDraw uses for drawing an arch. It consists of initially defining the circle center from which to extract the arch, and then indicate the start and end point of the arch itself.
Run the command.
Specify the center of the arch.
Specify the start point, or drag the mouse to your liking (you will see that the size of the hypothetical circle from which the arch is "extracted" varies).
Specify the end point, or drag the mouse to your liking (you will see the arch developing clockwise or counter-clockwise, depending on the position of the cursor).

*Warning: end point means to locate the intersection of the arch extension with a line passing through the center and through the desired end point. This request does not detect the end of the arch but its angle relative to the center: in fact, in most cases, the end point does not lie exactly on the arch, if not by coincidence. You may notice that the cursor arrow (the one that shows the exact point on the screen) is separated from the crosshair.*

### How to draw an arch belonging to three points

This method consists in drawing an arch passing through three required points in succession.
Run the command.
Type 3 or 3PUNTI in the command line.
Specify the first point.
Specify the second point.
Specify the third point.

### How to change the circumference thickness

As for the line and the circle, also the thickness of the arch can be varied.
Run the command.
Type S or SPESSORE in the command line and press <enter>.
Enter the new thickness value for the arch to be drawn.
Press <enter> to end (the new thickness setting is saved), or draw the arch using one of two methods described above.

## SOLID

EasyDraw provides the SOLIDO command for drawing triangular or quadrangular areas, single or consecutive, requiring four points in succession. The fill color of the solid is the current one

*Command        SOLIDO*

### How to draw a single triangular solid entity

Run the command.
Specify the first point of the solid area.
Specify the second point, determining the first angle of the solid area.
Specify the third point, diagonally opposite to the first.
Press <enter> when prompted for the fourth point to specify the previous point, i.e. the third). On the screen appears the triangular area filled with the current color.
Press <enter> to end the command.

### How to draw a single quadrangular solid entity

Run the command.
Specify the first point of the solid area.
Specify the second point, determining the first angle of the solid area.
Specify the third point, diagonally opposite to the first.
Specify the fourth point. On the screen appears the quadrangular area filled with the current color.
Press <enter> to end the command.

### How to draw more triangular or quadrangular consecutive solid entities

Run the command.
Specify the first point of the solid area.
Specify the second point, determining the first angle of the solid area.
Specify the third point, diagonally opposite to the first.
Press <enter> when prompted for the fourth point to obtain a triangular area, or specify it to get a quadrangular shape
This command continues to ask for the third and fourth point of the next area, considering the previous two valid as first and second for the current solid area
Press <enter> to end.

## TEXT

EasyDraw uses the Windows system fonts for the text. The TESTO command allows the user to create a complete sentence on several lines with a single command. The text can be stored (in the MTEXT.MDB archive located in the VIMAR\easydraw_easycap\DB) for reuse.
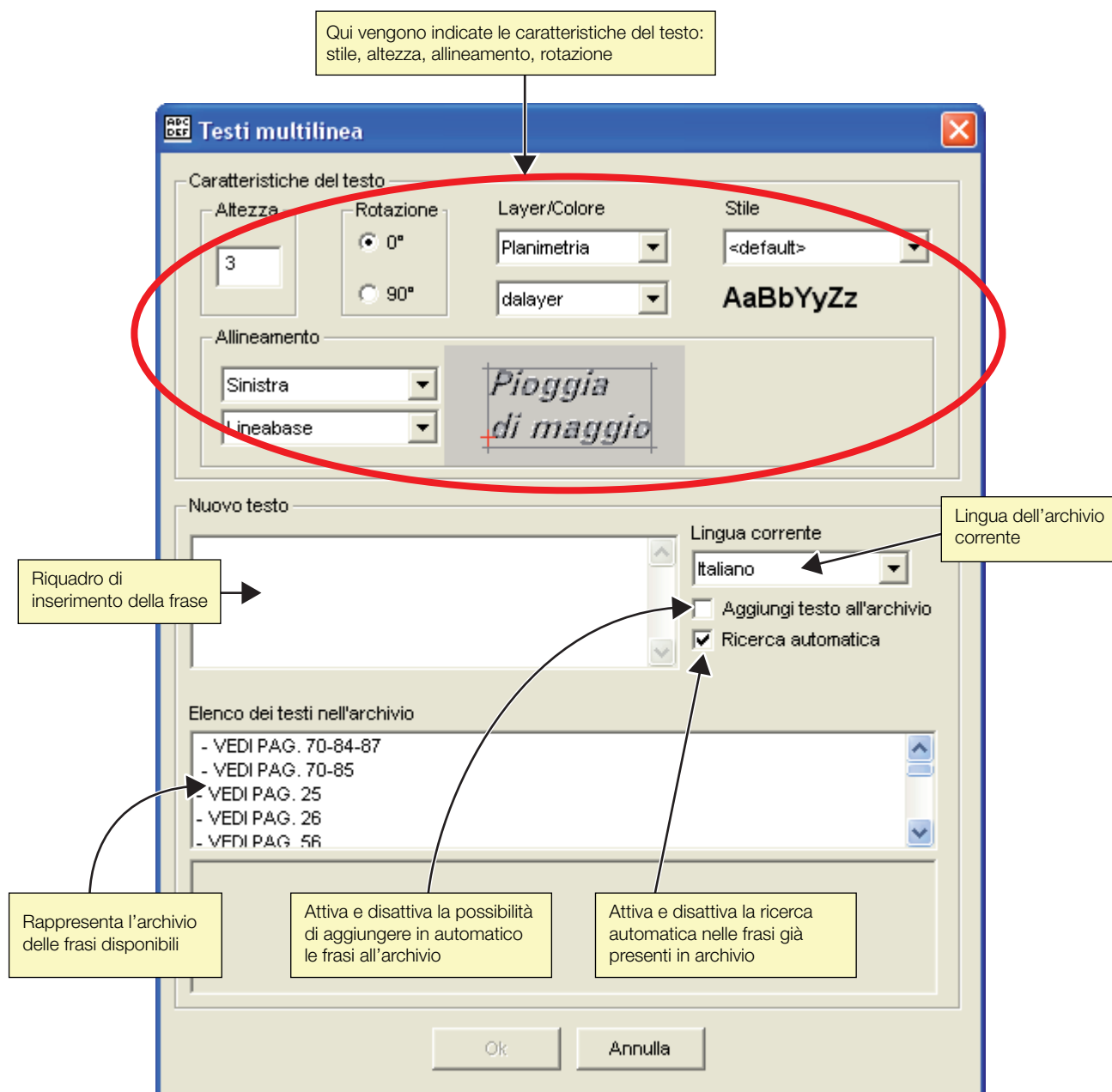


In the dialog box you can specify the language you are using.
The phrase archive is divided into 4 languages.
By activating the *Add text to archive* option, you can add phrases to the archive.
Activating the **Auto Search** you can search for a phrase in the archive by typing the first few letters. The phrase can be selected with a double click.
The *Stile* combo allows to invoke previously created text styles with the STILETESTO command.

To change the characteristics and content of a multiline text already entered in the scheme run the command MODIFICA PROPRIETA' in the ribbon menu *Tools*.

Procedure: after selecting the text to be edited, a dialog box appears, similar to the one used for the creation of the text itself; the only differences concern the inability to vary the angle of rotation of the text.

# TEXT STYLE

With EasyDraw IT is possible to define custom text styles, identified by a name. In the text style appear text characteristics such as font (linked to the Windows fonts, it can also be True Type), height (fixed or stated from time to time), appearance (bold, italic, strikethrough, underline). STILETESTO opens the dialog box for defining or changing text styles.
The *default* style is created by EasyDraw when you write a text without defining any style. Please note that when you change the characteristics of a style, all texts written in that style will take on the new changes.

***Command          Stile di testo***
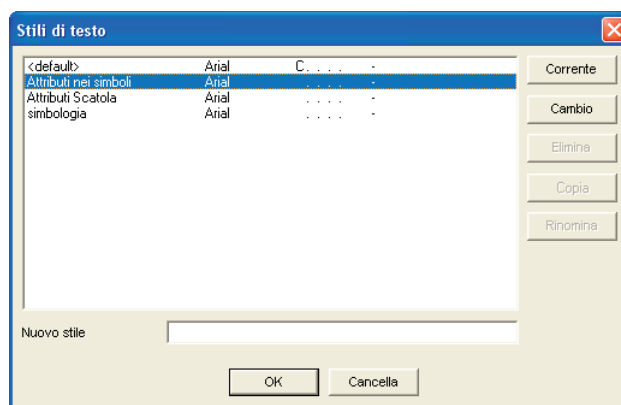


**How to create a text style**
Run the command.
When the dialog box appears, type the name of the style to be created in the New style field.
Confirm the *New* button.
Specify the type of font and style, set the fixed height, if desired, and note the settings configured in the sample box.
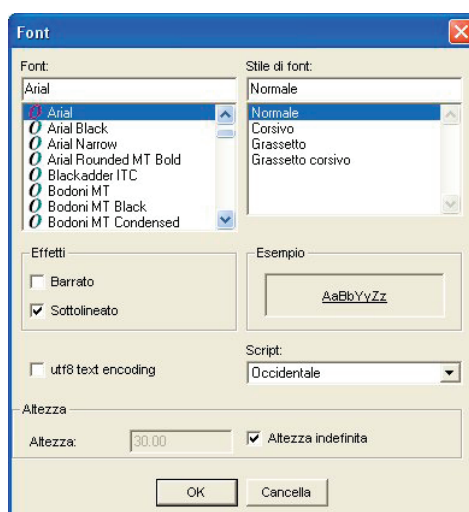


Confirm with **OK**.

**How to edit an existing text style**
Run the command.
When the dialog box appears, click the text style to be edited and confirm with *Edit*.
Changing the style settings.
Confirm with **OK**.

**How to edit a current text style**
Run the command.
When the dialog box appears, click the text style to be edited and confirm with *Edit*.
Changing the style settings.
Confirm with **OK**.
All text written in this current style take on the new settings.

**How to delete a text style**

Run the command.
When dialog box appears, click the text style to be deleted and confirm with *Delete*.
Confirm with **OK**.

**How to make another text style current**

Run the command.
When the dialog box appears, click the style name to be made current and confirm with the *Current* button.
Confirm with **OK**.
All text that will be inserted will adopt the text style just made current.

**How to rename a text style**

You can change the name of a text style without changing its characteristics.
Run the command.
When the dialog box appears, click the style name to be renamed.
In the *New* field enter a name for the selected style and confirm with *Rename*.
Confirm with **OK**.

*Command*        *TESTO*

**How to create a text**

By means of the TESTO command you can add text to the current drawing.
  • Run the command.
  • Type the text in the command line.
  • Specify the style of the text, pressing <enter> to use the default style or by typing the name of a custom style, created by means of the STILETESTO command. The ? is used to display the list of available text styles.
  • Set the height of the text, confirming the default value with <enter> or typing a new value to the command line.
  • Set the reference point of the text by clicking to confirm the default point of reference, which does not appear in the options and is visible on the screen, or can be invoked by typing **P** or **POSIZIONE** in the command line.
or
1. changing settings by typing **A** or **ALTO**, **B** or **BASSO**, **D** or **DESTRO**, **C** or **CENTRALE**, **S** or **SINISTRO**, **M** or **MEZZO** (midpoint in height), **BA** or **LINEA DI BASE** (line of support of the text) in the command line, followed by <enter>.
2. click to confirm the new reference point.
   * Set the rotation angle by typing the measurement of the angle, or by specifying the reference point and the direction of angulation.

# HATCH

The TRATTEGGIO command allows to fill an enclosed area with a predefined hatch type (style) using the STILE DI TRATTEGGIO command.
Hatches are useful for highlighting specific characteristics of some areas and are used to represent the different materials that make up a drawn entity.
The TRATTEGGIO command can fill an existing enclosed area, or you can draw a hatched area after indicating the points that define it. EasyDraw handles the obtained hatch like an entity, allowing editing (recognition of reference points, deletion, modification of the properties,...). To select a hatch entity, you need to select its outline; the lines forming the pattern are not, in fact, selectable entities.

# HATCH STYLES

With EasyDraw it is possible to define custom hatch styles, each identified by a name. The hatch style contains the characteristics of a single hatch unit, which, depending on the values it takes, allows to obtain different fill patterns. STILE DI TRATTEGGIO opens the dialog box for defining or changing hatch styles.
The *default* style is created by EasyDraw when you draw a hatch without defining any style. Please note that when you change the characteristics of a style, all hatches drawn in that style will take on the new changes.

**General concepts**

A hatch style is created by making reference to a single *hatch unit (cell)*, which is then repeated to fill the hatch entities. The elementary entity used for drawing the pattern is the family of lines. A family of lines is a set of parallel lines, identified by a point of origin, a direction, and a step. Each cell can contain an arbitrary number of families of lines.

The properties of a family of lines are:
**Direction**  characterizes the angle of the lines,
**Origin**     is the source of the first line,
**Step**       indicates the gap between the parallel lines

These properties are defined by means of the dialog box that appears when clicking the EDIT button of the STILE DI TRATTEGGIO command.

***Command        Stile di tratteggio***
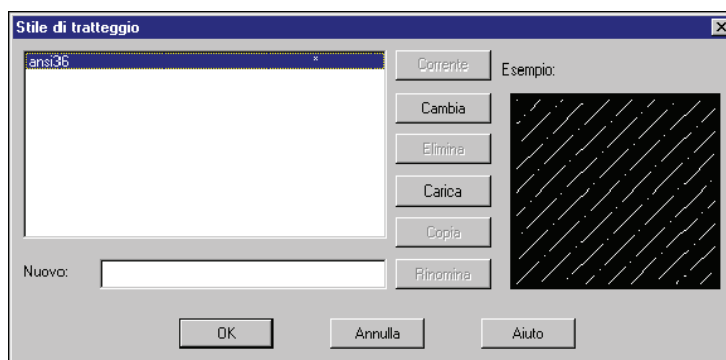


The *Edit* button in the STILE DI TRATTEGGIO dialog has the dual function to modify an existing hatch, or to create one from scratch. From this assumption, it is possible to guess the dual functions of creation/editing of some of the dialog boxes (Move/Origin, Angle/Direction, Scale/Distance).

### How to create a new hatch style

As already mentioned above, a hatch style is defined by reference to families of lines; it is necessary, therefore, to define the "generating" lines of these families. For each line must be indicated the origin, the inclination angle, and the distance (in drawing units) that separates a line from its parallel. After defining the line, press the *New hatch* button (you will see the example appear in the left area of the dialog box: if the zoom is not particularly suitable for the display, you will need to enter a different zoom ratio in the *Zoom example* and press the *Apply* button).

With the Mirror button, you can mirror the lines drawn along the axis of the sample window.

With the *Remove last segment* button you can delete the last created line. The *Discard pattern* button permanently deletes the just created hatch. *Restore* is used to restore the condition of the hatch before the editing.

When a hatch is too thick, EasyDraw does not normally display it: by enabling *Fill if too thick* it is possible to represent the dotted line with a solid fill.

Run the command.

When the dialog box appears, type the name of the style to be created in the *New* field.

Confirm with the *New* button.

Customize the display parameters of the new style.

Confirm the two OK in succession.

### How to edit an existing hatch style

To change the morphological characteristics of the hatch, it is necessary to indicate the changing values in the Move/Origin, Angle/Direction, Scale/Distance boxes, and then act on the related modifier keys. You can also use the *Mirror* button to make mirror copies of the hatch style; *Remove last hatch* to remove the last inserted line in the Hatch style; *Discard pattern*  to delete the madeor modified drawing (*Reset*  to return to the initial conditions).

Run the command.

When dialog box appears, click the hatch style to be edited and confirm with *Edit*.

Changing the style settings.

Confirm the two OK in succession.

### How to edit a current hatch style

Run the command.

When dialog box appears, click the hatch style to be edited and confirm with *Edit*. The current hatch style is marked by an asterisk (*).

Changing the style settings.

Confirm the two OK in succession.

All the hatches drawn with this current style take on the new settings.

### How to retrieve a file containing already saved patterns

Run the command.

When the dialog box appears, confirm with the *Load* button.

Select the file with *.PTT extension and identify one or more styles among the ones shown, ensuring to "mark" the styles you want to load.
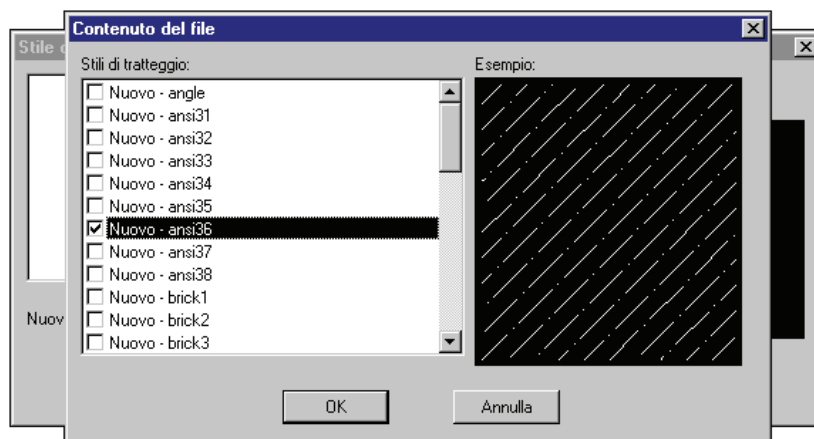
Confirm the two OK in succession.
The just invoked hatch style will be added to the list of hatch styles.

### How to delete a hatch style

Run the command.
When dialog box appears, click the hatch style to be deleted and confirm with *Delete*.
Confirm with OK.
You cannot delete the current hatch style.

### How to make another hatch style current

Run the command.
When the dialog box appears, click the style name to be made current and confirm with the *Current* button.
Confirm with OK.
The new hatches will be drawn with the new current hatch style.

### How to rename a hatch style

You can change the name of a hatch style without changing its characteristics.
Run the command.
When the dialog box appears, type the name to be assigned to the selected style in the *New* field.
Click the style name you want to rename and press the *Rename* button.
Confirm with OK.

### How to duplicate a hatch style

You can duplicate a hatch style.
Run the command.
When the dialog box appears, type the name of the duplicated style in the *New* field.
Click the style name you want to duplicate and press the *Rename* button.
Confirm with OK.

### *Command*          *TRATTEGGIO*

### How to hatch an enclosed area

Run the command.
• Specify the hatch style by pressing <enter> to use the default style, or by typing the name of a custom style, created by means of the STILETRATTEGGIO command. Type ? to get the list of hatch styles defined in the current drawing.
Type S or SELEZIONA to fill an existing entity already closed.
Choose one of the advanced options I or INTERNO, O or ESTERNO, G or IGNORA, V or VISIBILE, P or PIENO available, and then click in the vicinity of a point within the entity to be hatched.

### How to create a hatched area

Run the command.
• Specify the hatch style by pressing <enter> to use the default style, or by typing the name of a custom style, created by means of the STILETRATTEGGIO command
The ? is used to display the list of the already defined styles.
Specify the first point from which to draw the hatched area.
In succession, determine the various sections of the hatched area, making use of options A or ARCO, C or CENTRO, D or DIREZIONE, 2P or 2PUNTI to outline an arch-shaped segment, L or LINEA to outline a linear hatch, CH or CHIUDI to automatically close the area (i.e. join the last point with the first one).

## QUOTES

**Quote line**
It is a line that indicates the direction and extent of a quote.

**Quote lines**
These lines extend perpendicularly from the entity to be quoted, to the quote line.

**Arrow ends**
These are symbols added to the ends of the quote line.

**Quote text**
It is a string that indicates the extent of the quoted entity but EasyDraw allows to modify it by inserting a prefix or a suffix to it, or by changing its style and size.

**Associative quote**
All EasyDraw quotes are *associative*. An associative quote is a quote where extension lines, quote line, arrows, and quote text are a single entity and they are not interpreted by the program as a single entity. In other words, thanks to this feature, the user can change the quoted entity without bothering to edit the quote, because it is automatically updated in real time. **An expanded quote (ESPLODE command) cannot be reconverted into an associative quote.**

## QUOTE STYLE

A *quote style* is a group of quote settings, marked by a name, which determines the appearance of a quote. These settings can be changed and saved to new quote styles.

***Command        Stile di quota***
By means of the STILE DI QUOTA command, EasyDraw allows the creation and manipulation of new styles, or the modification of the default style (standard). When running the command, in fact, the window below appears, which contains commands to create a new style (*New* button), to change an existing style (*Change* button), to rename it (*Rename* button), to delete it (*Delete*) button), or to make it current with respect to the default one (*Current* button).

The following describes the parameters of representation provided by EasyDraw.

**Properties section**
Allows to configure the quote text.
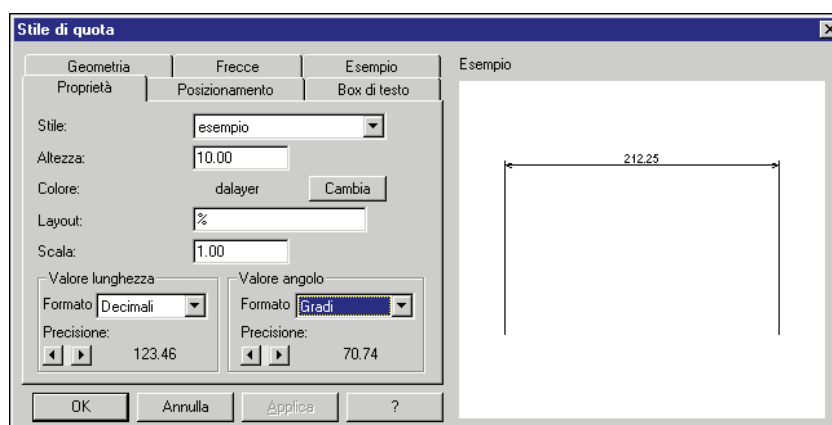


| Style | style of the quote text. |
|---|---|
| **Height** | height of the quote text. |
| **Color - Edit button** | color of the quote text. |
| **Layou**t | the % character will be automatically replaced by the quote value. % can be preceded or followed by prefixes or suffixes, such as "cm%". |
| **Scale** | scale of the quote text. |
| **Length value** | units of measurement and accuracy of numerical linear quote text. |
| **Angle value** | units of measurement and accuracy of numerical angular quote text. |

## Position Section

Defines the position of the quote text with respect to the extension lines and quote line.
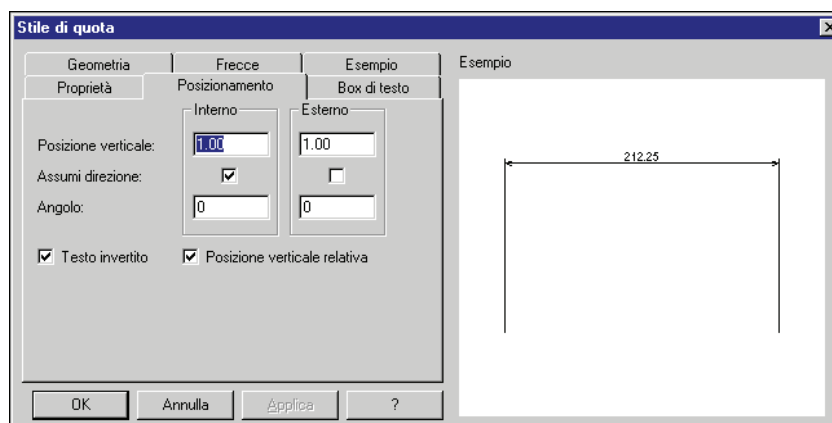


| | |
|---|---|
| **Vertical position** | absolute or relative position of the quote text with respect to the quote line. |
| **Take direction** | enables and disables the rotation of the quote text. |
| **Angle** | rotation of the quote text. If the value is other than 0, the text is placed inside the quote. |
| **Inverted text** | possibility to invert the quote text. |

## Text Box section

Allows you to place the quote text in a box.
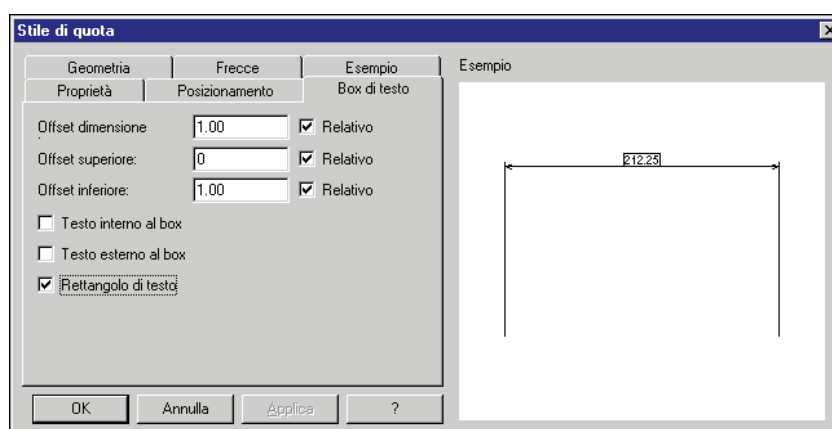


| | |
|---|---|
| **Dimension offset** | defines the absolute or relative size of the box horizontally. |
| **Upper offset** | defines the absolute or relative distance between the upper side of the box and the quote text. |
| **Lower offset** | defines the absolute or relative distance between the lower side of the box and the quote text. |
| **Text inside the box** | the quote line breaks at the edges of the box and the text remains inside the box. |
| **Text rectangle** | if enabled, a box is represented. |

## Geometry Section

Allows you to customize the appearance of the extension lines and quote line.
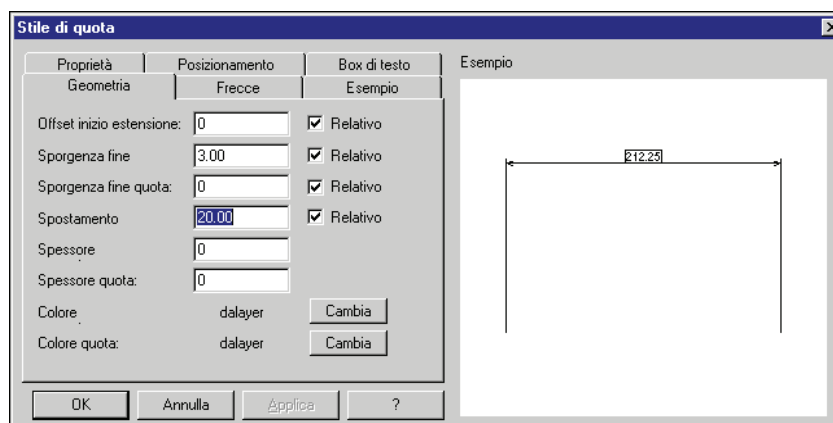


**Extension start offset**        indicates the absolute or relative distance between the beginning of the extension lines and the extent.

**Overhang at the end of extension**        indicates the absolute or relative extension of the extension lines with respect to the quote line.

**Overhang at the end of quote**     indicates the absolute or relative extension of the quote line with respect to the extension lines.

**Quote movement**        indicates the absolute or relative movement of the dimension line.

**Extension thickness**        thickness of the extension lines.

**Quote thickness**        thickness of the quote line.

**Extension color**        color of the extension lines.

**Quote color**        color of the quote line.

## Arrow Section

Allows to customize the appearance of the arrows.
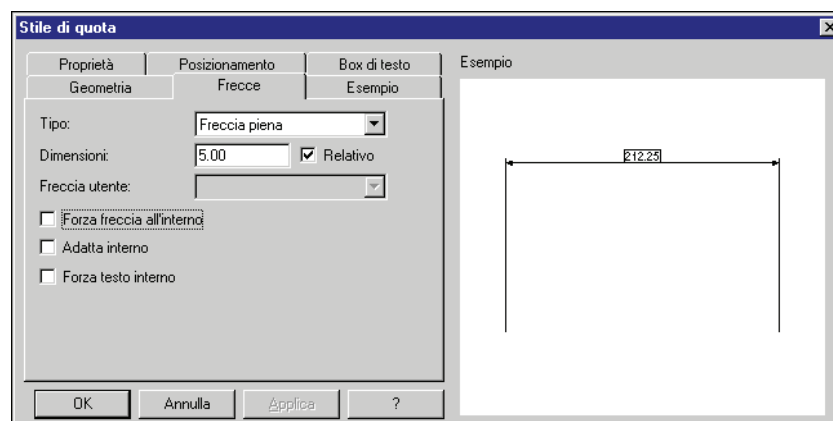


**Type**        various arrows available.

**Size**        absolute or relative size of the arrow symbol.

**User arrow**        customization of the arrows.

**Force inside**        When enabled, the arrows are always internal.

**Adapt inside**        if enabled and if the text with the arrows is longer than the quote line, the text will be external to the quote line but the arrows will remain internal.

**Force text inside**        When enabled, the text will always be internal and cannot be moved.

## Example Section

Allows you to set the representation of the example shown in the box.
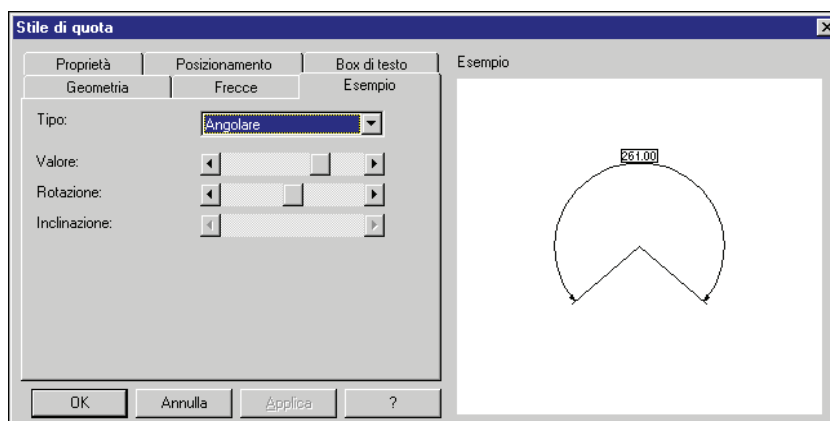


**Type**          you can see an example of a desired quote type.

**Value**         value of the example quote.

**Rotation**      angle of rotation of the example.

**Inclination**   inclination of the quote (available only for the aligned quotes).


**OK Button**
Used to confirm the customizations made to the quote style.

**Undo Button**
Used to undo the customization made.

**Apply Button**
Used to view the configurations madein the example shown.

**Help Button**
Used to invoke EasyDraw online help.


### How to create a new quote style
Run the command.
When the dialog box appears, type the name of the style to be created in the Name field.
Confirm with the *New* button.
Customize the display parameters of the new style.
Confirm with **OK**.

### How to edit an existing quote style
Run the command.
When the dialog box appears, click the quote style to be edited and confirm with the *Change* button.
Changing the style settings.
Confirm with **OK**.

### How to edit a current quote style
Run the command.
When the dialog box appears, click the text style to be edited and confirm with *Edit*.
Changing the style settings.
Confirm with **OK**.
All quotes designed with this current style take on the new changed settings.

### How to delete a quote style
Run the command.
When the dialog box appears, click the quote style to delete and confirm with the *Delete* button.
Confirm with **OK**.

**How to make another quote style current**
Run the command.
When the dialog box appears, click the style name to be made current and confirm with the *Current* button.
Confirm with **OK**.
The new quotes will be drawn with the new current quote style.

**How to rename a quote style**
You can change the name of a quote style without changing its characteristics.
Run the command.
When the dialog box appears, click the style name to be renamed.
In the field *Name* enter a name for the selected style and confirm with *Rename*.
Confirm with **OK**.

## CREATION OF LINEAR QUOTES

This command allows you to list an entity in a linear fashion. Each quote is drawn on the current layer with the current text style and quote style (see QUOTESTYLE and TEXTSTYLE).
You can create quotes by selecting an entity or by indicating the position of the extension lines.

*Command*        *LINEAR QUOTE*

**How to create a linear quote indicating the points**
Run the command.
Enter the name of an existing quote style or confirm the default quote style. Type ? to get the list of quote styles already defined.
Specify the entity to be quoted by typing E or ENTITIES to select an entity, or by specifying the start point, and then the end point of a quote.
Set the characteristics of the representation of the quote:
respond to the command line:
V or VERTICAL to obtain a vertical quote.
O or HORIZONTAL to obtain a horizontal quote.
A or ALIGNED to obtain an aligned quote.
T or TEXT to display any text instead of the numerical value.
R or ROTATED to tilt the quote around the two specified points at will.
The changes are immediately shown on the screen.
drag the cursor to the desired location and press <enter>.
or
specify the position where to place the quote by entering the coordinates in the command line, or by dragging the cursor.

## CREATION OF CONTINUOUS LINEAR QUOTES

The continuous linear dimensions managed by EasyDraw are *base lines or continuous lines* and they are a series of related quotes, which, respectively, allow to obtain more quotes of the same base line, or a long quote divided into shorter segments. To create one of these types, a linear quote must already exist for the entity that you want to quote.

*Command*        *CONTINUOUS*

**How to create base line linear quotes**
The base line quote has the extension line coincident to the extension line of the original quote and adopts the same quote style.
Select the existing quote. The extension line used as a reference will be the one nearest to the point indicated in the selection of the quote.
Type T or TEXT in the command line to set the text to replace the numerical value and press <enter>.
Specify the position of the second extension line. This command is aborted by pressing <enter>.

**How to create continuous line linear quotes**
The continuous line quote has the first extension line coincident to the extension line of the original quote and adopts the same quote style.
Select the existing quote. The extension line used as a reference will be the one nearest to the point indicated in the selection of the quote.
Type T or TEXT in the command line to set the text to replace the numerical value and press <enter>.
Specify the position of the second extension line. This command is aborted by pressing <enter>.

## CREATION OF RADIAL QUOTES

The DIAMETER, ARCH, and RADIUS commands allow to create quotes that are based, respectively, on a diameter, an arch, and a chord.

***Command        Diameter quote***

**How to create a quote of a diameter**
Run the command.
Enter the name of an existing quote style or confirm the default quote style. Type ? to get the list of quote styles already defined.
Find the entity to be quoted typing E or ENTITY
**or**
type C or CENTER to confirm the center and radius in sequence
**or**
confirm the initial point, and then the end point.
Type L or LAYOUT to edit the quote text or confirm the position of the quote.

***Command        Arch quote***

**How to create a quote of an arch**
Run the command.
Enter the name of an existing quote style or confirm the default quote style. Type ? to get the list of quote styles already defined.
Find the entity to be quoted typing E or ENTITY
**or**
type 3P or 3POINTS to confirm the three points of the quote in sequence
**or**
confirm the center and, subsequently, the starting and end point of the quote.
Type L or LAYOUT to edit the quote text or confirm the position of the quote.

***Command        Radius quote***

**How to create a quote of a radius**
Run the command.
Enter the name of an existing quote style or confirm the default quote style. Type ? to get the list of quote styles already defined.
Find the entity to be quoted typing E or ENTITY
**or**
confirm the center and, subsequently, the opening radius of the quote.
Type L or LAYOUT to edit the quote text or confirm the position of the quote.

## CREATION OF ANGULAR QUOTES

The ANGULAR QUOTES command allows to dimension the angle of a sector or the angle between two different entities. The quote line is arch-shaped.

***Command        Angle quote***

**How to create a quote of an angle**
Run the command.
Enter the name of an existing quote style or confirm the default quote style. Type ? to get the list of quote styles already defined.
Find the entity to be quoted typing E or ENTITY
**or**
confirm, in succession, the starting point and its direction, the second point and its direction.
Type L or LAYOUT to edit the quote text or confirm the position of the quote.

***Command        Reset quote***

**How to reset the position of the quote text**
Run the command.
Select the quote you want to reset and press <enter>.

## BLOCKS

A *block* is a set of entities grouped in a single one. As soon as they are grouped, the entities are given a name, which is used to enter the group within the drawing whenever the user wishes.
The introduction of blocks within a drawing significantly speeds up the editing because a block can be entered multiple times within the same drawing, avoiding to redraw the entities that compose it every time, and also because you can quickly correct some parts of the scheme defined as blocks.

In a block it is possible to store all information relating to layer, color, and line type, ensuring that the entities are drawn with their original characteristics.

**Important**
When drawn on the *<default>* or *0* layer, with DALAYER color and linetype, the entities of a block take on the properties of the layer on which the block is inserted.
However, when the entities have a block color and linetype DABLOCCO, they take on the properties of the block reference entity.

A block can be composed of other blocks without physical limitations: it is, however, impossible to create a block that contains another one with the same name.

*Command*          *DEFINES*

**How to define a block in the current drawing**
Start the BLOCK command.
Select the entities to be associated with the block.
Specify the point that will be the base point or insertion point of the new block.
Type the name to be assigned to the block in the current drawing.
The selected entities disappear; they can be inserted into the drawing again with the INSER command.

*Command*          *Insert block*

**How to insert a block in the current drawing**
To insert a block previously saved in the current drawing, in addition to the name, you need to specify the insertion point, scale, and rotation angle.
Start the INSER (or DINSER) command. This command creates the entity *Block reference*.
At the prompt, type the name of the block previously saved with the BLOCK command (typing ? you can see, in the text box, the list of defined blocks).
Specify, the insertion point of the block using coordinates, or by dragging it.
At the prompt, type the scale or indicate the reference length for the insertion of the block.
Type the angle of rotation or indicate the one of reference in the drawing.

## MANAGING IMAGES

With EasyDraw it is possible to carry out projects using drawing tools or scanned planimetry images (raster image). The use of scanned planimetry images avoids the drawing phase (if the image quality is good), or, at least speeds it up by being able to retrace the raster image with the same drawing tools.

*Command*          *MANAGES*

**How to insert an image**
Run the command.
Select the desired raster file (*.BMP or *.JPG)
Specify the inserting point.
Drag the mouse to zoom the image to the desired size.
Click to confirm the inserted image.

## CALIBRATING IMAGES

You can scale the entire raster image indicating the value of the distance between two known points. After selecting an image, it is necessary to define the known distance between two points, indicating in succession the first point, the second point, and the value of the distance between the two points in the drawing unit used.

*Command*          *CALIBRATE*

**How to calibrate an image**

You want to import a planimetry image acquired by means of a scanner, and then add the electrical system to it. It is necessary that the planimetry and the electrical symbols have the same scale. Proceed as follows:
Identify two points on the planimetry which have between them a known distance, for example the width of a door.
Run the command and select the image from the scanned planimetry.
Enlarge the door with the ZOOM command.
Select the two ends of the door in succession, and then type the value in the quote, such as 100 (centimeters, of course).

The raster file will be scaled up to bringing the two points at 100 drawing units. Proceed with the drawing, assuming that each drawing unit corresponds to a centimeter on the planimetry image
.

# ARCHITECTURAL MENU AND PLANIMETRY

This menu contains all the commands necessary for the management of planimetry images on the same drawing where you are going to add an electric system

## SQUARING

With this command, you can invoke a squaring to insert on the current page.

The squaring is actually a block that is inserted and scaled according to the initial settings of the drawing (scale and unit of measurement). EasyDraw provides the user with a folder ("squaring") with 10 squaring blocks already saved in the most common formats (A4, A3, A2, etc...).

In every provided squaring there is a cartouche (table to be filled) located at the bottom right: once inserted, you can edit its references using the commands operating on the blocks.

When the command is run, a dialog box appears:



La casella *Posizione fissa* abilitata consente di inserire la squadratura centrata rispetto all'origine del foglio
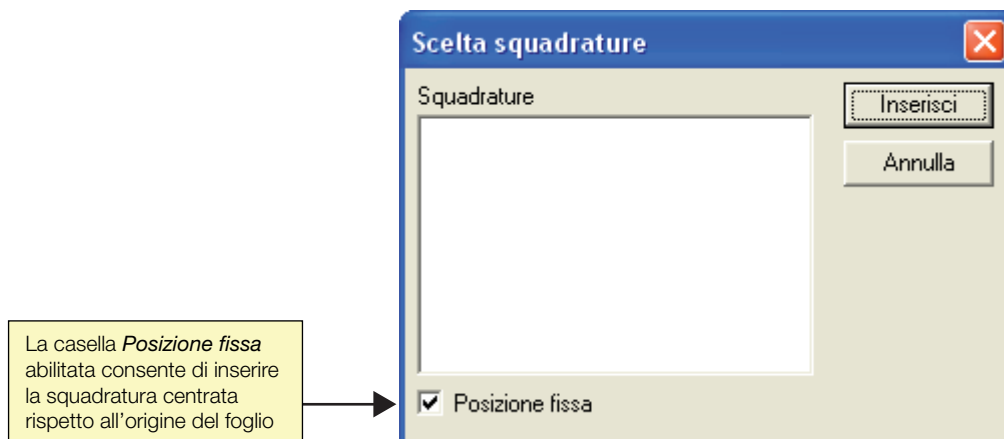
Select a squaring and click *Insert* (or double-click the squaring itself).

## WALL

A command that allows you to draw walls, both internal and external, or a series of consecutive walls. When drawing the walls you can invoke the PROPERTIES command without ending the WALL command (see below), in order to accelerate the user operation of editing the default values.

**Drawing the walls**

Start the WALL command. In the command bar is displayed: *Starting point / Properties*.

Select the point where the stretch of wall begins (click with the left mouse button).

Drag the mouse until you reach the length and direction of the wall you want.

Select the point where the stretch of wall ends (click with the left mouse button).

You can also enter the length of the wall using the keyboard. See example below.



Punto di partenza del muro

Quota relativa alla lunghezza corrente del muro

Posizione attuale del mouse
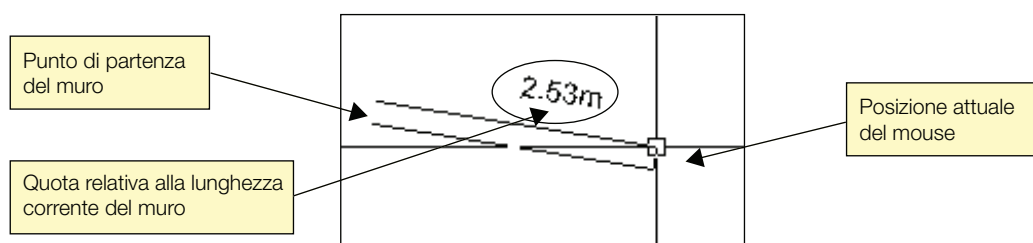
While drawing a wall, a dynamic quote appears that indicates the current length of the wall. To change the inserted length, type a new value in the current unit of measurement and press *Enter* again. Then select the direction of the created wall with the mouse and click to confirm.

You cannot change the length of a wall with this procedure after you have confirmed the direction of the wall itself.

**Drawing the partition walls**

Start the WALL command. In the command bar is displayed: *Starting point / Properties*.

Place the mouse on the wall from which you want remove the partition. A connection preview and a quote between the partition and a corner of the main wall are displayed (to change the side of reference of the quote, bring the mouse at the other end of the main wall).

Select the point where to hook the partition wall (click with the left mouse button).

Drag the mouse until you reach the length and direction of the wall you want.

Select the point where the stretch of wall ends (click with the left mouse button).

The created partition can now be managed as a normal wall.

**Connect a partition to another wall or partition**

Start the WALL command. In the command bar is displayed: *Starting point / Properties*.

Move closer to the wall to which you want to connect the partition. A connection preview and a quote related to the distance between the partition are displayed (to change the side of reference of the quote, bring the mouse at the other end of the main wall).

Drag the mouse until you reach the hooking position you want.

Confirm the location (click with the left mouse button).

**Drawing arch wall sections**

To draw sections of an arch wall run the command WALL and select the Arch wall option on the control bar.

The program asks you to specify the details of the wall segment (via mouse click), and then its curvature (see example below).
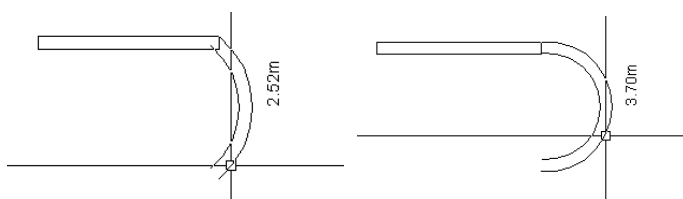


Selezionato il punto di arrivo, spostare il cursore del mouse per raggiungere il grado di curvatura desiderato. Confermare la scelta con un click del tasto sinistro del mouse.

## WALL PROPERTIES

By running the WALL command and selecting *Properties* on the command bar, the following dialog appears:



Indica lo spessore che dovranno avere i muri disegnati da questo momento in poi. Il campo offre alcuni valori di spessore tra i più utilizzati. L'unità di misura è quella corrente.

Impostare il riferimento del muro. Rappresenta il riferimento con cui verranno espresse le misure durante il disegno dei muri sulla planimetria.

## DOOR

A command that allows to insert a door in a wall section. When inserting, you can invoke the PROPERTIES command without ending the DOOR command (see below), in order to accelerate the user operation of editing the default values. The displayed requests are the following:

*- Select wall / Properties:*

drag the cursor from side to side of the wall to define the position of the frame; also drag the cursor to move the door at the point where you want to insert it and confirm with a click. To end confirm with *Enter*. By clicking near the *Properties* option, in the command line, the command PROPERTIES is run, which allows to change the door parameters still to be entered.

## DOOR PROPERTIES

By running the DOOR command and selecting *Properties* on the command bar, the following dialog appears:

Inserire la larghezza della porta.

E' possibile impostare la porta con anta, senza anta e a doppia anta per mezzo del menù a tendina *Tipo*.
Nel caso venga scelta la porta con anta, il campo *Anta* consente di definire la posizione dei cardini (destra o sinistra).

**Proprietà della porta**

Dimensioni
Altezza: 2 m
Larghezza: 1 m

Aspetto
Tipo: standard
Anta: destra

OK   Annulla

## WINDOW

A command that allows you to insert a window. When inserting, you can invoke the PROPERTIES command without ending the WINDOW command (see below), in order to accelerate the user operation of editing the default values.
The displayed requests are the following:

- *Select wall / Properties:*
  drag the cursor from side to side of the wall to define the position of the frame; also drag the cursor to move the window at the point where you want to insert it and confirm with a click. To end confirm with *Enter*. By clicking near the *Properties* option, in the command line, the command PROPERTIES is run, which allows to change the window parameters still to be entered.

## WINDOW PROPERTIES

By running the WINDOW command and selecting *Properties* on the command bar, enter the width of the window to insert.

## PILLAR

A command that allows you to insert a pillar. When inserting, you can invoke the PROPERTIES command without ending the PILLAR command, in order to accelerate the user operation of editing the default values.
The displayed requests are the following:

- *Select one point / Properties:*
  A preview of the pillar is connected to the mouse. Select a point in the project where you want the pillar. By clicking near the *Properties* option, in the command line, the command PROPERTIES is run, which allows to change the pillar parameters still to be entered.

- *Select the angle:*
  After hooking the pillar to the drawing you must select the angle of rotation.

## PILLAR PROPERTIES

By running the PILLAR command and selecting *Properties* on the command bar, the following dialog appears:

L'utente deve inserire i parametri dimensionali del pilastro.
Le misure vengono espresse nell'unità di misura corrente.

selezionare quale punto della colonna si vuole agganciare al mouse durante l'inserimento.

## RECESS / PILASTER

A command that allows you to create a recess or a pilaster on a wall already drawn.

The required operations are the following:
1. zoom in a part of the wall where you want to create a recess / pilaster for better drawing accuracy;
2. run the command RECESS / PILASTER;
3. select the side of the wall to create a recess / pilaster;
4. select a point on the wall, on the selected side, where to start drawing the recess / pilaster;
5. trace all the segments required to draw the recess / pilaster;
6. click the point where the wall the drawing of the recess / pilaster is meant to finish;
7. press <enter> (or right mouse button) to finish the drawing

Nicchia          Lesena
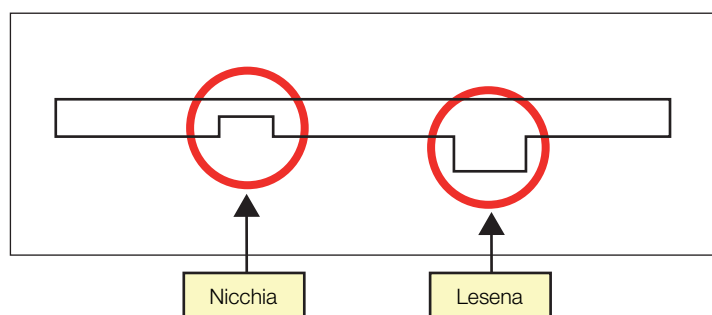
## MOVE / EDIT OBJECT

This command can be applied to only one object at a time and it acts differently depending on the type of selected entity.
Function *"Stretch wall"*: selecting the end of a wall, you can lengthen or shorten the wall itself. After running the MOVE/MODIFY OBJECT command, select one end of a wall, set the new length with the mouse and confirm with a click.
Function *"Translate wall"*: this function allows you to translate the position of a wall relative to another to which it is attached, provided that the two walls are perpendicular to each other. After running the MOVE/MODIFY OBJECT command, select a distant wall from its end, set the new position with the mouse and confirm with a click.
Function *"Move windows or doors"*: selecting a door or a window, you can move the architectural entity as if inserting it again; this allows you to select another wall where you want the door or window and the position on that wall. After running the MOVE/MODIFY OBJECT command, select a door or window, choose the wall where you want to place them, and confirm by clicking the new position.

## PROPRIETIES

Command that invokes the dialogs for changing the parameters of the architectural entities. You can access these dialogs in two ways:
by running the insert commands (wall, door, window, pillar) and selecting PROPERTIES on the command bar
by running the PROPERTIES command from the icon or drop-down menu and selecting the desired architectural entity
If the PROPERTIES command is applied to walls and pillars already included in the project, you cannot make any changes.

## CONNECT WALL

This command prompts the user to select with a click the ends (short side on top) of the walls to connect. The selected walls will be connected at the point of intersection of their extensions.

## DECOR LIBRARIES

EasyDraw contains a library of scale symbols with various pieces of furniture ranging from bathroom fixtures, to furniture for the home and office.
To insert a piece of furniture you need to select the image of the symbol in the ribbon.

The prompt for its inclusion in the drawing appears on the screen:
- *Insertion Point*:
Click on the picture at the point where you want to insert the symbol.
- *Angle/Ref angle/<>*:
Requires to indicate the angle of rotation of the symbol or the reference angle.

## RECOMMENDED EQUIPMENT

To obtain standard equipment according to Vimar, depending on the type of installation you want to accomplish, you can use the installation Wizard.
Through quick selection by the user, the Wizard provides, in fact, a set of objects which can then be modified to adapt to the needs of the real case.

Page 1:
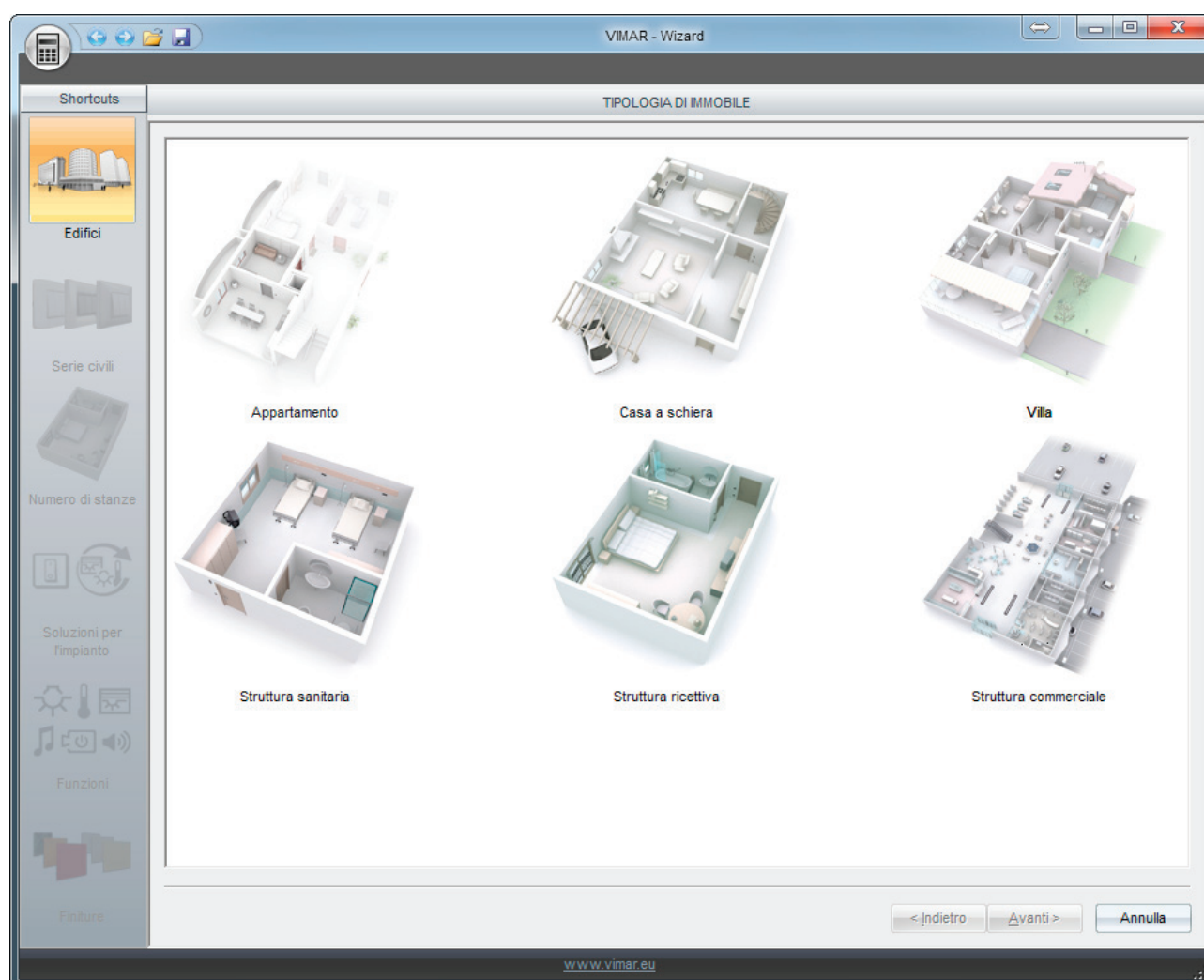


The first page lists the different types of property, to choose one just click on the corresponding picture. The chosen building will be highlighted and you can then move to the next page with the *Next* button.

Page 2:
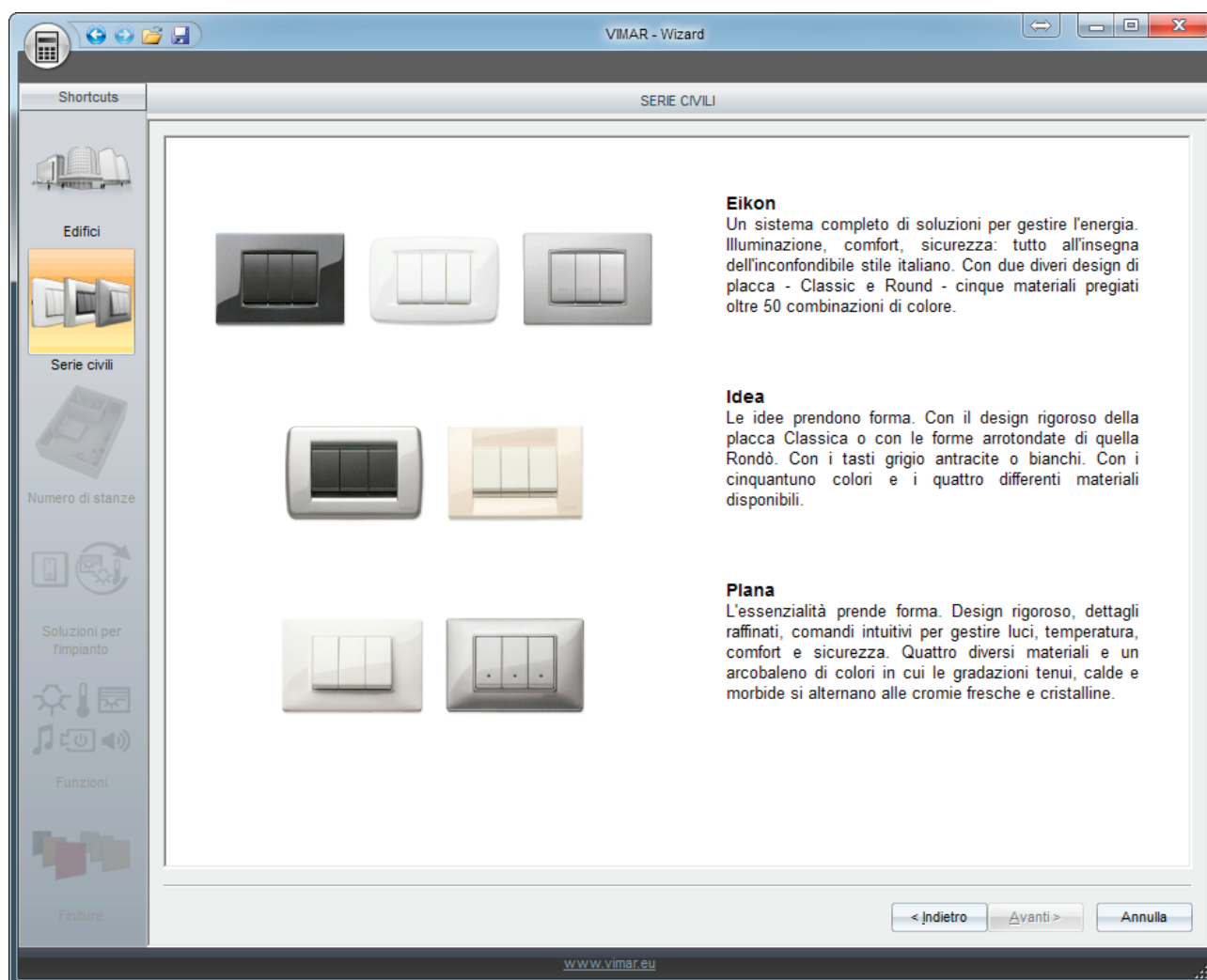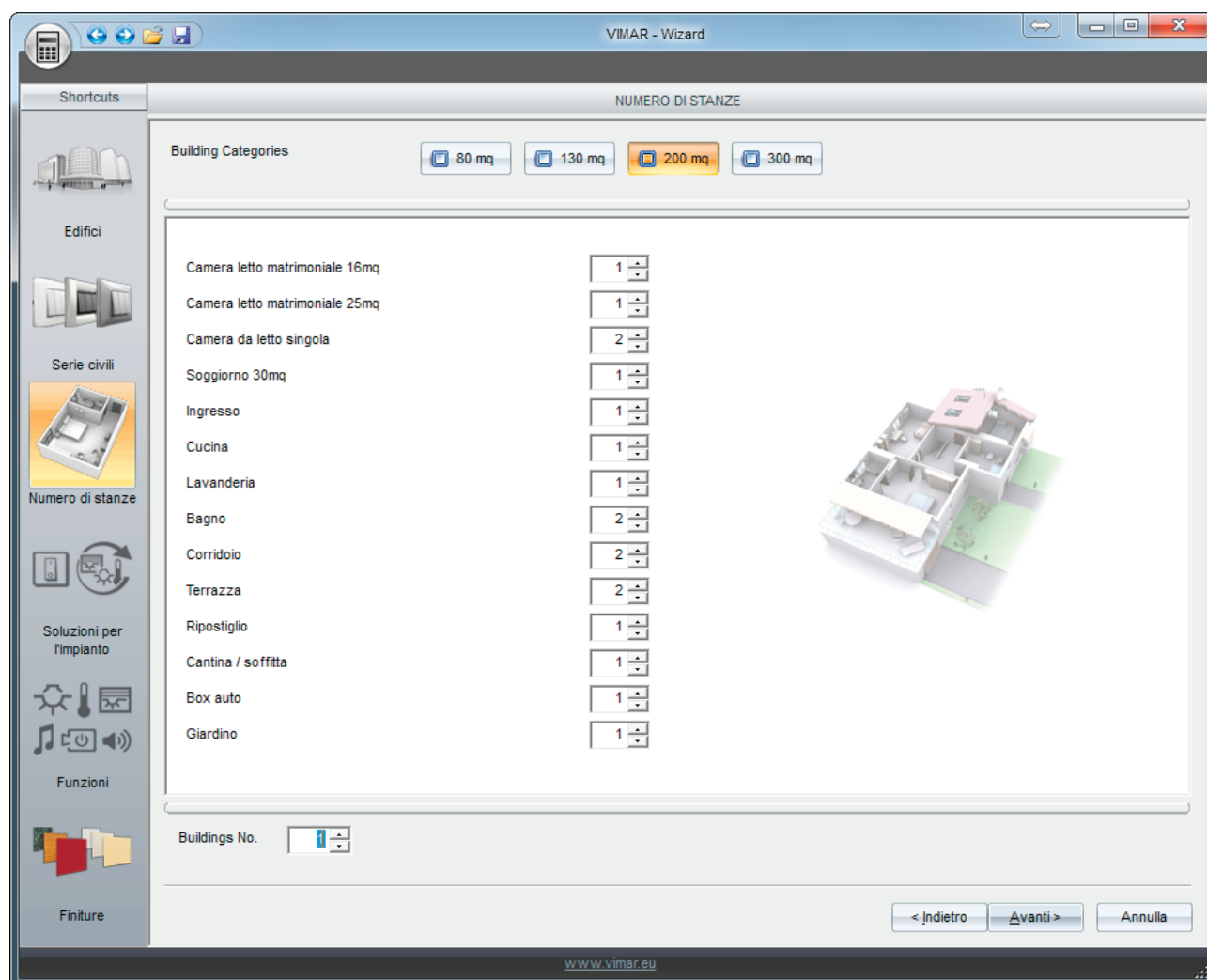


The second step is related to the choice of the civil series to use.

Keep in mind that some home automation functions may not be available for one or more series.

In any case, the choice made can be changed later returning to this page via the *Back* button, or via the button representing the page on the left side.

Page 3:



On this page you can refine your choice on the building.

There can be multiple categories (top) that refer to the size of the building itself. Each category is associated with a certain number of rooms and for each of them an amount is suggested. We must therefore choose the category that is closest to the real case and then change the number of the rooms. To exclude a room you can set the appropriate quantity to zero.

At the bottom is a multiplier of buildings (usually set to 1) which serves to multiply the equipment in the case of multiple identical buildings.

Page 4:



This the page is reserved for different types of available installations, depending on the selected type of building.

Based on the previously chosen building, you can choose among several levels from the top. The levels affect the variety of the equipment generated by the Wizard.

When you can choose among multiple alternative installations, as in the case in the figure, the choice of a level suggests the installation necessary for its achievement. It is however possible to change the type of installation, obtaining alternative equipment to the one actually required for the achievement of the selected level.

Page 5:
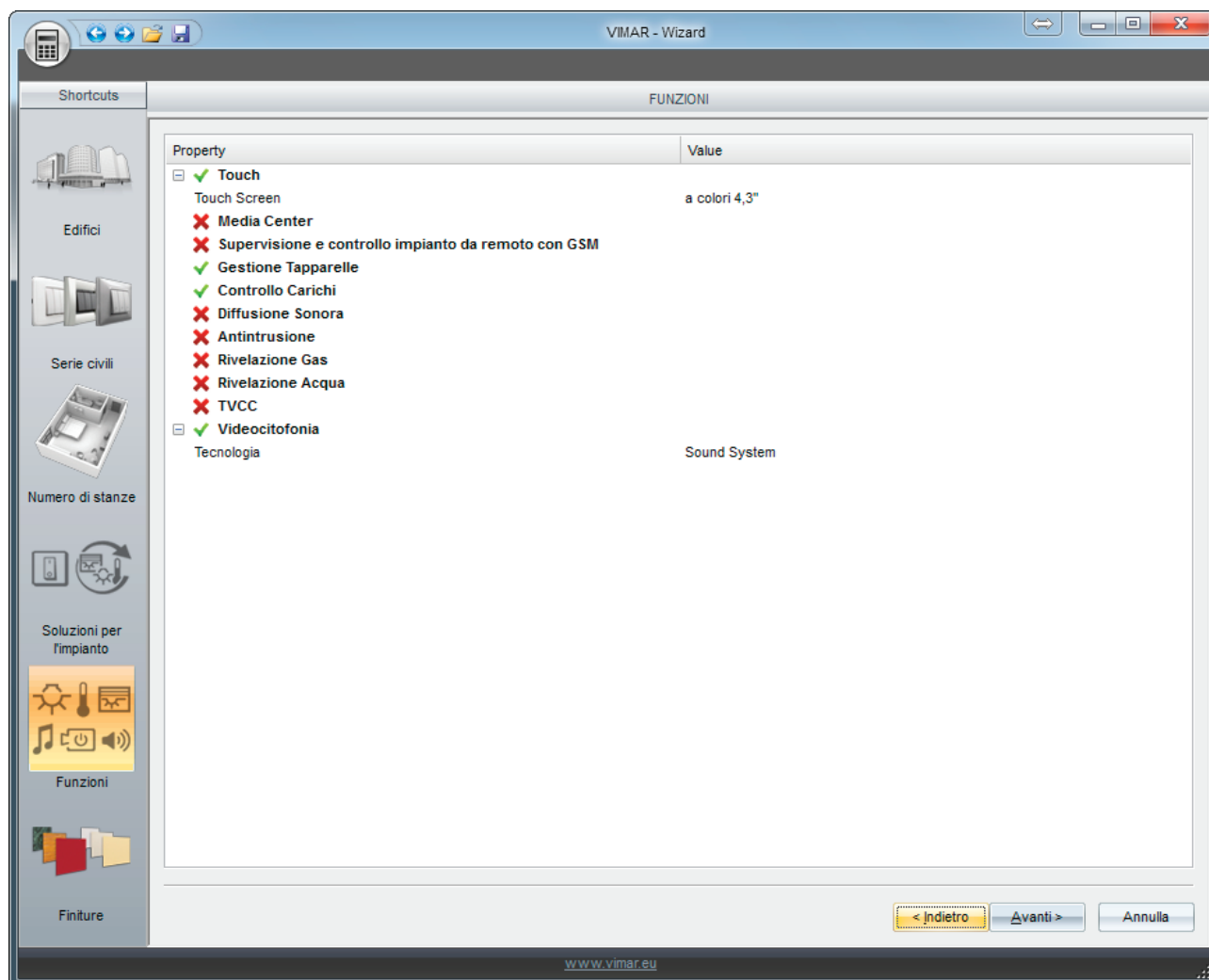


Depending on the type of installation you choose, you may be able to choose between several accessory features. Some of these features may already be pre-selected.
In some cases, the selection of a feature also provides for the choice of some options that determine the insertion of certain items instead of others.
This screen is not invoked in some cases (for example, in the case of tertiary).

Page 6:
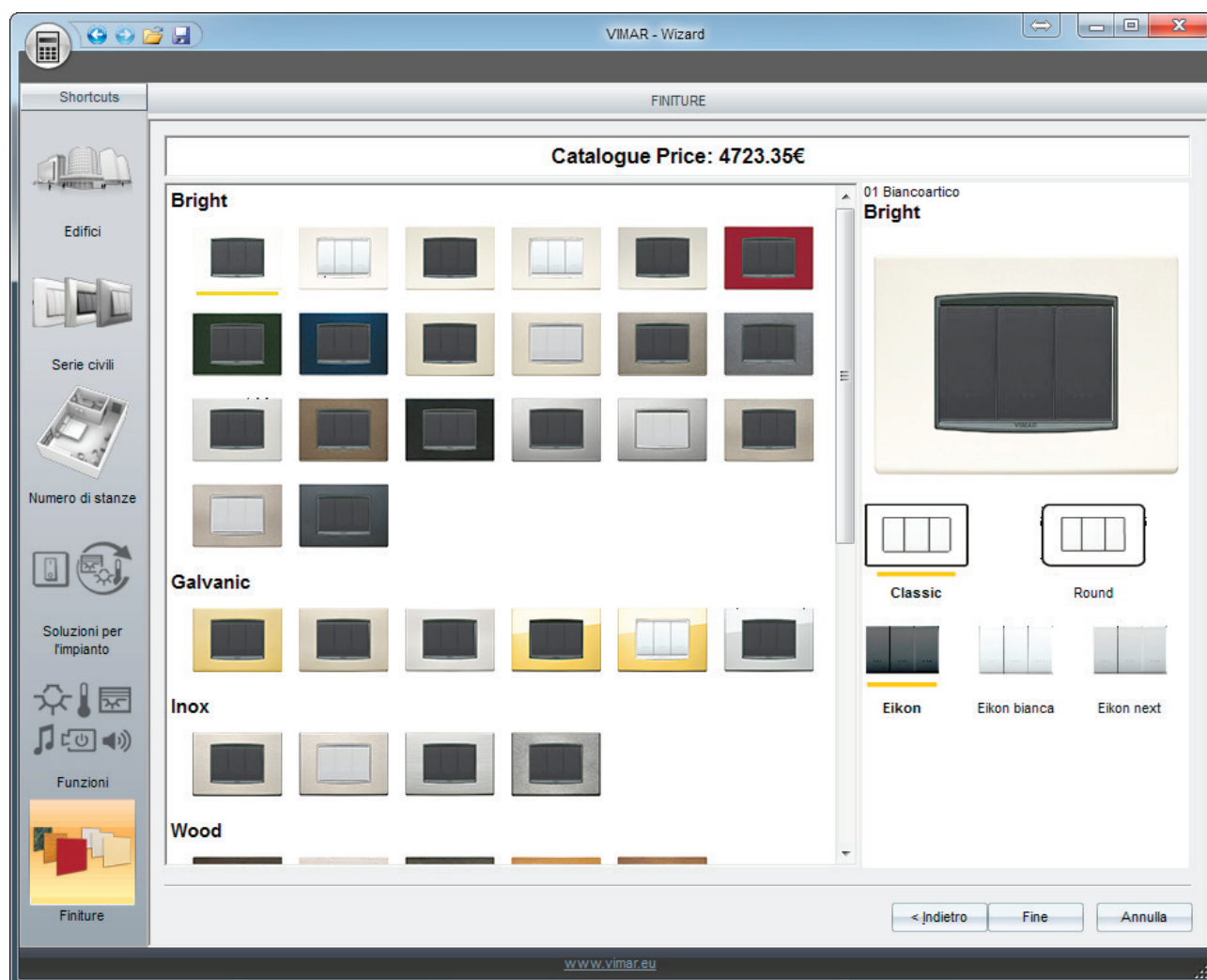


After defining all that you want for your installation, it is time to move to the choice of finishes to be used. The last page of the wizard displays a list of plates by material, color, and shape among those available for the series chosen on page 2. You can also choose the color of the fruit.
The right dialog displays a preview on the selections, while at the top is shown in the price list for the installation and the chosen features; you can see how the price varies depending on the choice of the finishes.

All the choices made until the last page can be changed going back to the previous steps, the Wizard keeps track of all the selections and adapts to the changes.
By choosing the "Finish" button on the last page, the Wizard closes and transfers on the drawing the equipment grouped by room.

Bagno 1

Bagno 2

Dispositivi di sistema

Terrazza 1

Terrazza 2

Lavanderia

Corridoio 1

Corridoio 2

Cucina

Camera letto matrimoniale 16mq

Ingresso

Riposiglio

Cantina / soffitta

Camera letto matrimoniale 25mq

Soggiorno 30mq

Box auto

Giardino

Camera da letto singola 1

Camera da letto singola 2

## SYMBOL LIBRARIES

Except for purely "graphic" ones, symbols are associated with entries for metric calculation in EasyCap.

An entry for metric calculation is a complex code comprising:
• **code of articles**, taken from the lists of manufacturers, with the relative prices;
• **time needed for the installation** and **qualification of the installer** with cost;
• **general expenditures**, expressed in % of the cost of material and installation;
• **profit margin**, expressed in % of the cost of material and installation;

The SIMBOLI command invokes the dialog box containing the electrical blocks and graphical symbols in general. The symbols are organized by a tree divided into folders by topic.
To the right is the image of the block with its description.
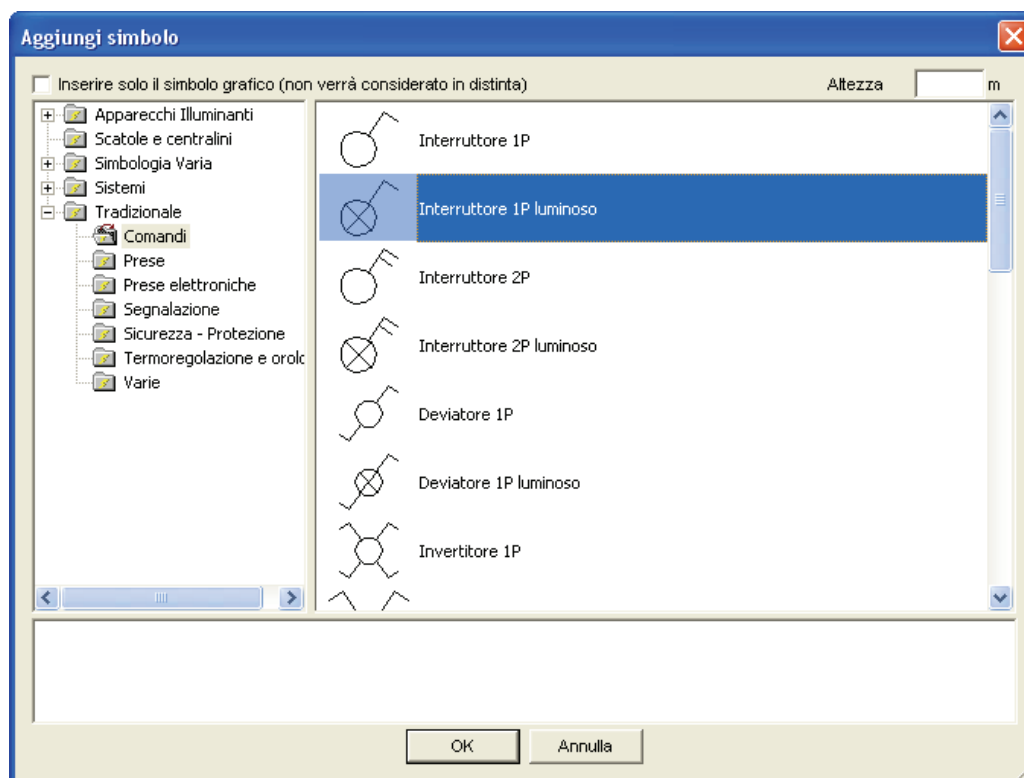
After identifying the symbol to be placed in the drawing, double-click its name, or click it once, and then confirm with *OK* in order to insert it. The program then starts to search the archives.

The electrical symbols can also be invoked by clicking the corresponding image in the ribbon of the program.

If the symbol is associated with an electrical device, you will see a box (below) that allows you to select the family and type of device you want.
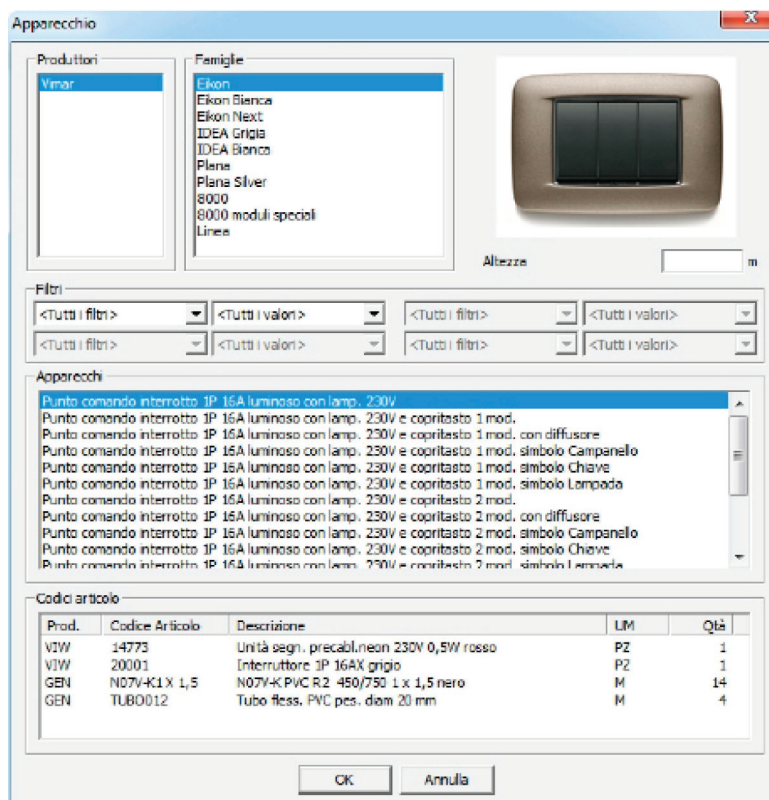


Click *OK* to confirm the selection.

A prompt for inserting in the drawing appears on the screen:

### *Insertion Point:*
Click on the picture at the point where you want to insert the symbol.

### *Angle/Ref angle/<>:*
Requires to indicate the angle of rotation of the symbol or the reference angle. The insertion angle is very simple if the *Ortogonale* function is enabled.

After inserting a symbol in the project, the application makes it available again; this allows to speed up the insertion of more symbols referring to the same item. Right-click, or press *Enter* to finish.

## NEW SYMBOL

This command allows you to automatically create the block in *.BLK format, and then its image in *.BMP format.
To create a new symbol, simply draw the entities it will consist of (lines, circles, text, etc...).
After finishing the drawing, run the NUOVO SIMBOLO command and respond to the requests:

### *Select entities:*
Specify the entities that make up the symbol to save.

### *Insertion Point:*
Specify the point that will become the base point or reference point of the symbol.

At this point, the screen shows the dialog box that prompts the user to specify the box where to save the symbol.
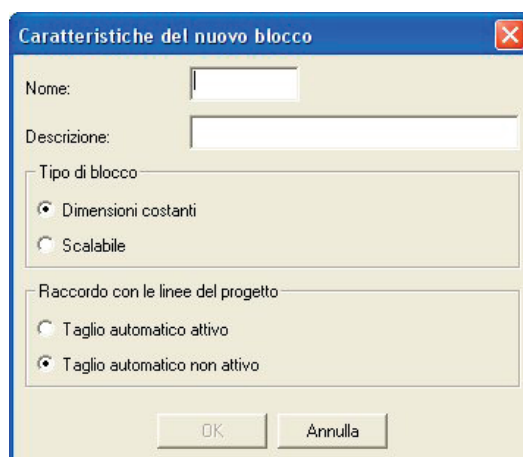


**Name (file) and Description**: allow the definition of the name (maximum 8 characters) to be given to the *.BLK and *.BMP files, as well as the inclusion of a description (maximum 100 characters), the same one that appears in the selection dialog of the symbols next to the image.

**Block type**: allows you to define whether the new block shall have a fixed size (fixed blocks), or proportionate dimensions to the planimetry (scalable block). Click the item in question.

**Connection with the lines in project**: allows you to define whether the new block shall break the lines where it is going to be superimposed or not.

By clicking**OK** the block and the bitmap image will be created. If a block defined by that name already exists, EasyDraw asks for confirmation to overwrite.

## INTERACTIVE GRAPHICAL COMPOSITION

EasyDraw allows you to define the standard civil equipment also 'by box', with a graphic and interactive approach.
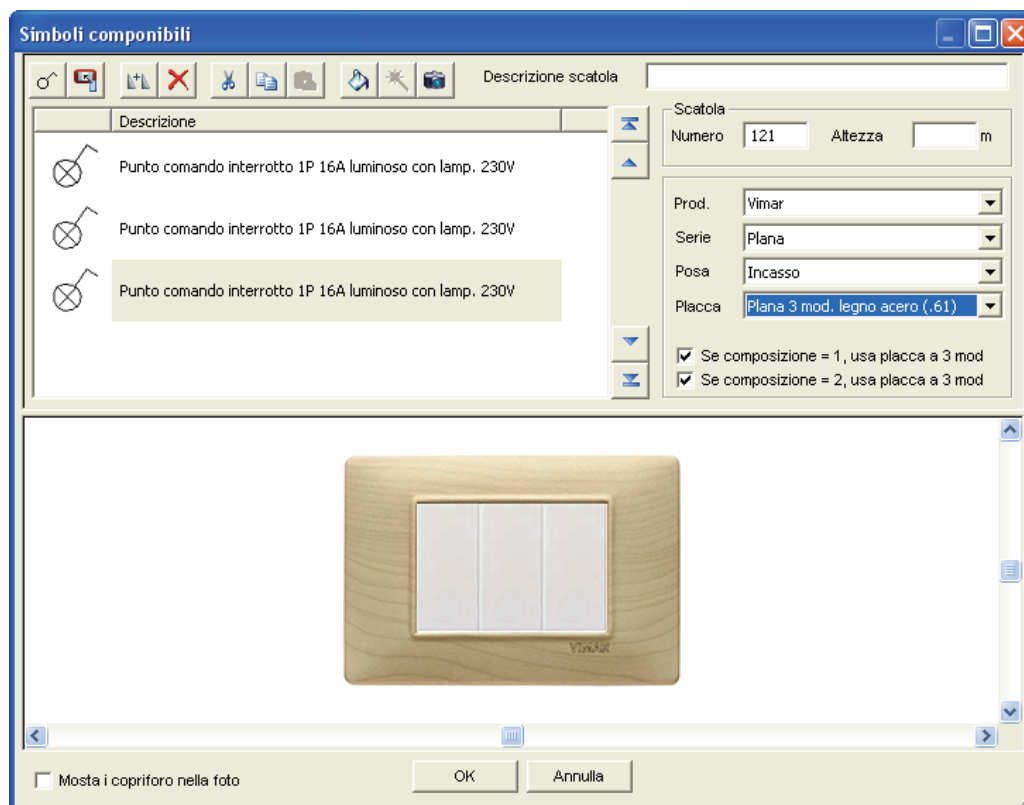


It is possible to:
- Select series, laying, and finishing plate, watching in real time the changes in appearance of the composition of the equipment
- Assign a description, a number, and the height from the ground of the composition (the height is given for information purposes and used for the realization of axonometric projections).
- Choose the appliances that will constitute the composition, possibly changing their order.
- Automatically complete the composition with the necessary hole plugs (which may or may not be represented in the planimetry by the relative symbols).
- Force the minimum size of the box holding the equipment to 3 modules with equipment consisting of one or two modules, for better uniformity of the installation.
- Export a 'photo' of the front view of the composition, as shown in the interface, for use in the planimetry, or in third-party applications.

## EDIT COMPOSITION

This command allows you to change the series and type of equipment associated with a symbol already in the drawing.
The change takes place indicating the symbol and, through the dialog box for selecting the manufacturer and device previously shown by the SIMBOLI command, the user is allowed to access to the list of devices again. The new data on the type of equipment is confirmed with *OK*.

## EXPAND COMPOSITIONS

This command allows you to expand a composition, making individual devices that compose it editable. The accessories for the installation added to the compositions are automatically deleted.

## NUMBER COMPOSITIONS
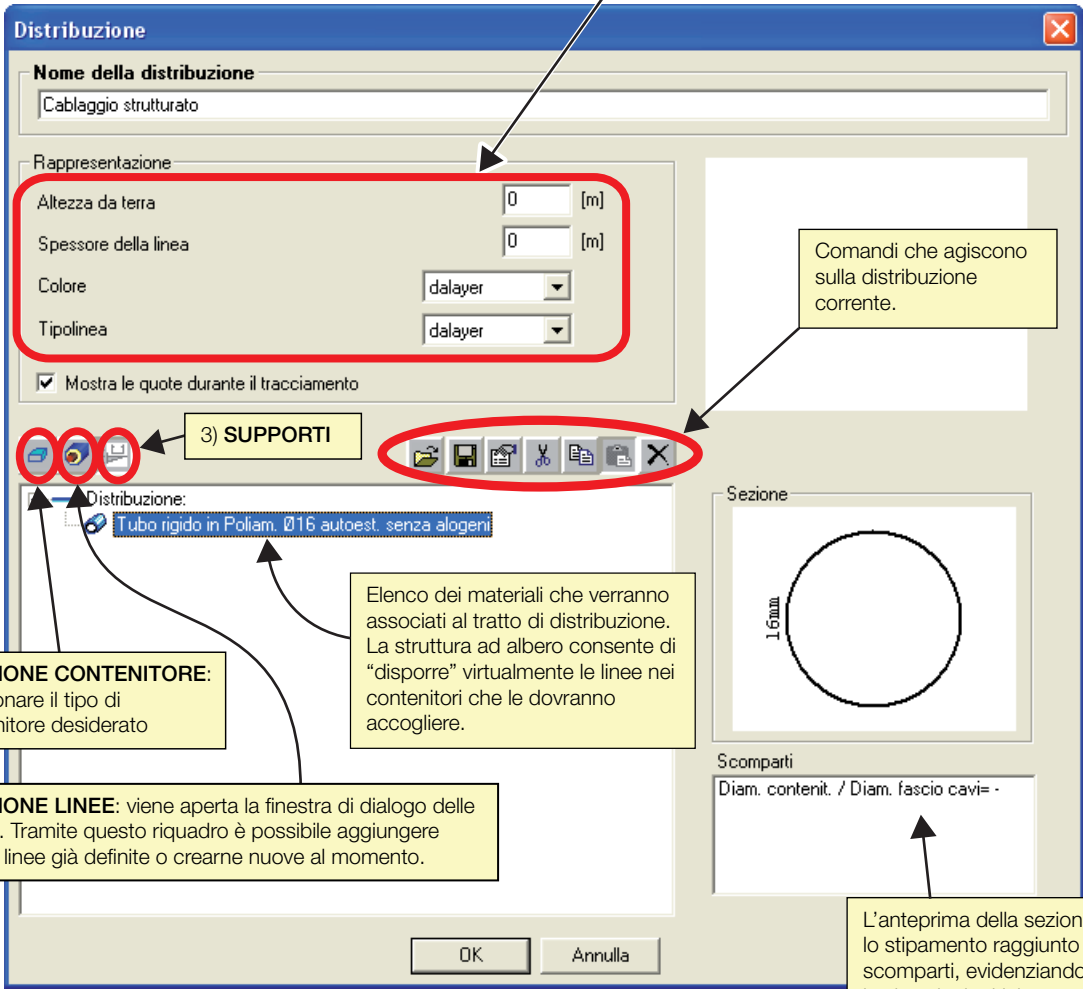
This command allows you to assign an automatic sequential numbering to the boxes included in the project. If you wish to manually change this number, you can use the command EDITA ATTRIBUTI NEI BLOCCHI.

## DISTRIBUZIONE

This command allows you to draw a distribution path with a representation of single-cable type. Each time you start the command, after selecting the starting point of the distribution, a dialog box appears for setting the parameters of the representation.

Permette di inserire una descrizione complessiva della composizione creata. Tale descrizione viene utilizzata come "nome della distribuzione" quando viene salvata nell'archivio utente.

Impostare l'*Altezza* da terra della tratta di distribuzione, lo *Spessore* (di rappresentazione), il *Colore* e il *Tipolinea* da utilizzare per disegnare il tratto di distribuzione.

Comandi che agiscono sulla distribuzione corrente.

3) **SUPPORTI**

Elenco dei materiali che verranno associati al tratto di distribuzione. La struttura ad albero consente di "disporre" virtualmente le linee nei contenitori che le dovranno accogliere.

1) **GESTIONE CONTENITORE**: selezionare il tipo di contenitore desiderato

2) **GESTIONE LINEE**: viene aperta la finestra di dialogo delle LINEE. Tramite questo riquadro è possibile aggiungere nuove linee già definite o crearne nuove al momento.

L'anteprima della sezione mostra lo stipamento raggiunto negli scomparti, evidenziando in rosso le situazioni critiche.

If the indicated point coincides with the section of a distribution already outlined, the dialog will inherit its content, and it will be proposed automatically pre-filled with the same data structure (containers and lines).

You can add an unlimited number of containers, pipes, and lines within the same section, you can also insert a container inside another container (e.g. a pipe inside a channel); this behavior is sometimes essential to ensure segregation of the circuits.

After selecting the pipes, channels, and lines, you can begin to track the distribution by pressing OK.

Here are the required drawing options:

*Starting point:*
indicate the starting point of the distribution section, identifying a point in the graphic area or by entering the coordinates in the command line. After the detection of the first point, the request changes to allow the making of the uprights:

*Upright/next point:*
next point allows you to continue tracking the distribution by delineating new sections; each section is a summary of the current valuesof the elevation, codes, and associated notes.
The Upright option allows you to create the uprights prompting the user to enter the new elevation value.
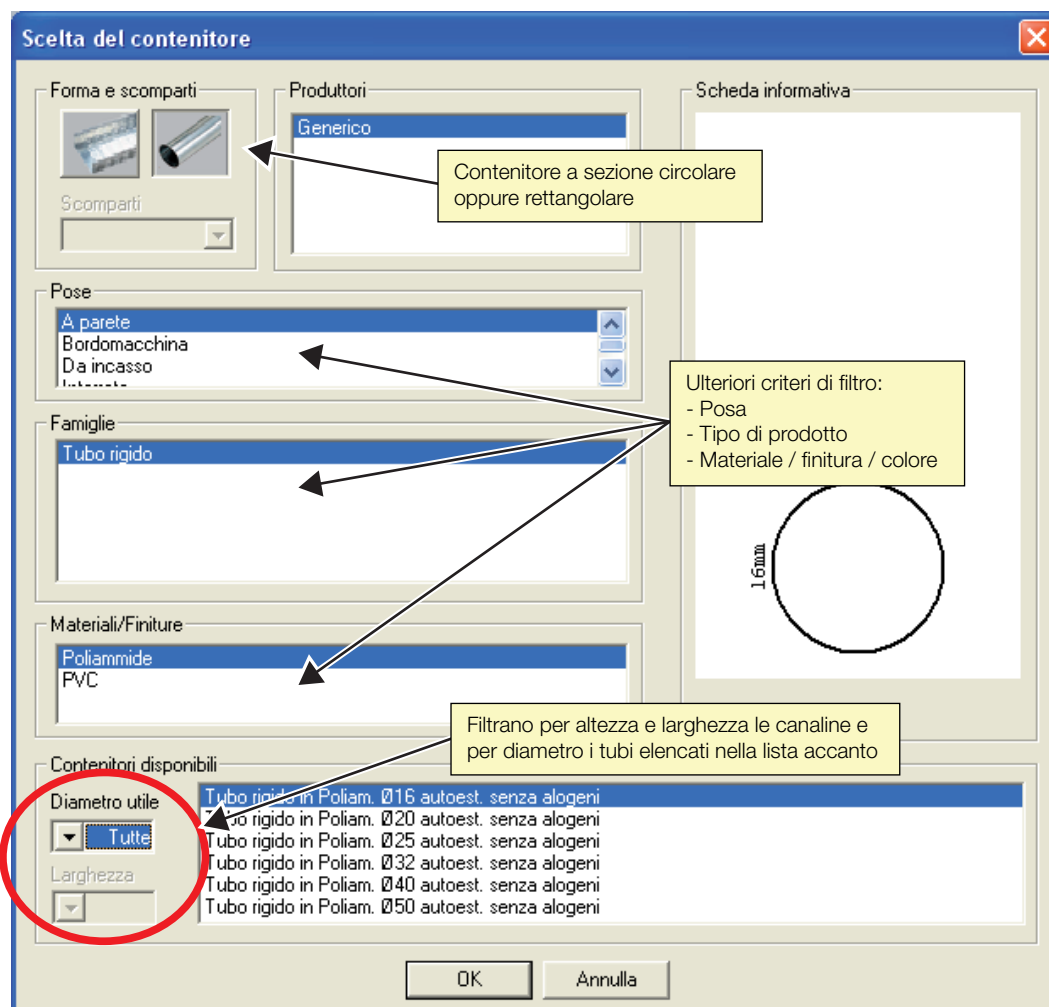
*New elevation:*
enter a value greater than the current to create the ascending upright or a smaller value to create the descending upright. The point where the distribution varies in elevation is represented by a symbol.

*Upright/next point:*
press Enter or right click to finish the tracking

## 1) Container management

Allows you to add a container to the section of distribution, choosing a channel or a pipe from the archive. Selecting an icon for the container will open the dialog box shown below:



For each section of the distribution, you can add an unlimited number of pipes, but only one container/channel.

## 2) Line management

EasyDraw allows you to define and store any number of distribution lines in each project. The lines can also be exported to external files to be imported into any other projects.
The GESTIONE LINEE command includes all the necessary functions.
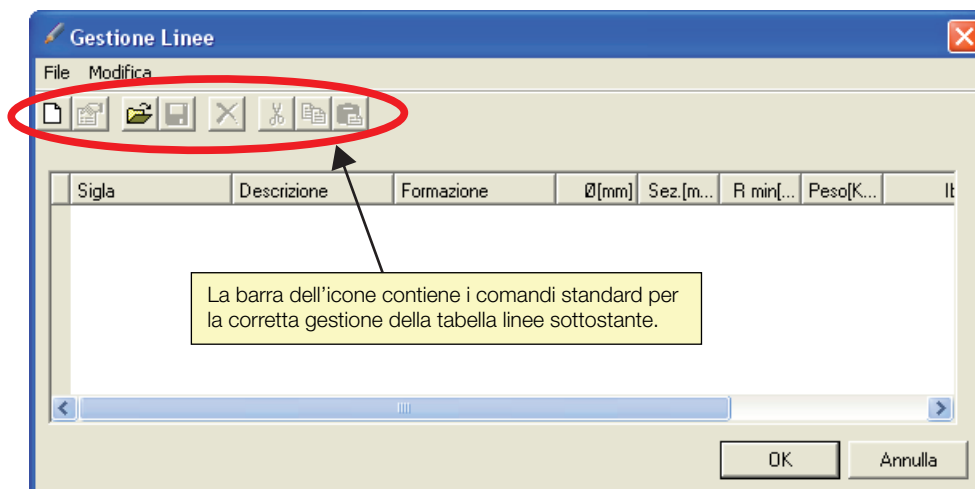


*New line:*
Press this icon to define a new line in the project. The new line will be defined and will immediately be modifiable through the Proprietà command of the line, invoked automatically.

*Import-Export lines*
The lines defined in an EasyDraw project can be exported to external files to be imported in later projects, or other applications. These files have *.LDF extension and free name and location. It is thus possible to define the lines of frequent use only once, and then import them in a single operation in projects where they are needed.
To export:	select the lines you want to export, and then press the Export icon; type the name of the file when prompted by the program.
To import:	press the Import icon, then select the file with *.LDF extension you want to import.

*Delete line:*
Select the lines you want to delete and press the Delete line icon.

*Line properties*
This command allows you to edit an existing line. This icon is enabled when a single line is selected.
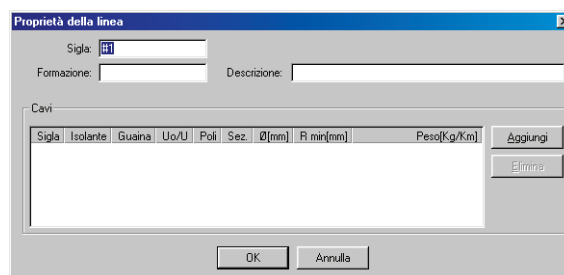


A line is identified by a code in a project, initially assigned automatically ("#" followed by a sequential number).

The other properties of the line are:

**DESCRIPTION**
To be used to describe the characteristics of the line. It will show in the legend of the lines.

**FORMATION**
Describes the cables that make up the line. It is automatically populated when you define the line, but it remains editable by the user.

**DEFINITION OF A LINE**
To manually define a line, you simply need to add the desired cables to the list. Press the Add button. The panel shown below appears.
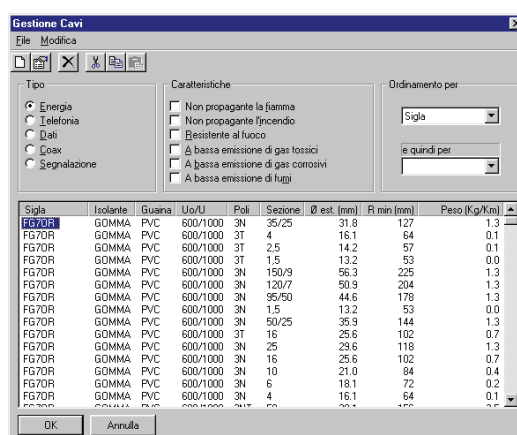


**SELECTION OF A CABLE**
EasyDraw has a complete archive of cables, which you can access through the CABLE MANAGEMENT panel.

The archive is organized by type.
To sort the list of cables, select the first criteria in the Sort dropdown and the second in the and then for dropdown.

After identifying the desired cable, to add it to the line while being defined, select it and press OK, or double-click it.
Press Cancel to exit without selecting any cables.

**ADD A CABLE**
Press the New icon, and then define the new cable through the command PROPRIETA' DEL CAVO, which is launched automatically.

**DELETE and CUT / COPY / PASTE**
Select the cable to act on, then select the corresponding icon.

**PROPRIETA' DEL CAVO**
This command allows you to edit the selected cable.
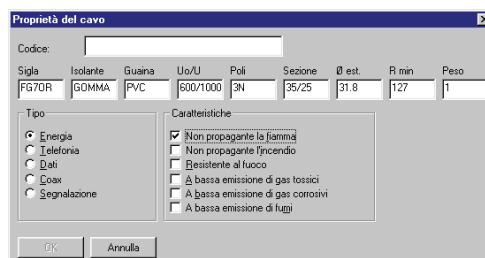


This command allows you to edit the selected cable.

## 3) Supports

Allows you to select the supports for the channels.
*This icon is only enabled by choosing containers where the necessary data has been entered.*

## MODIFICA DISTRIBUZIONE

With this command it is possible to operate on the sections already drawn to modify their contents (containers with related accessories and lines), or graphical features (height from the ground, width, color, linetype).
After running the command, select the sections to be edited, according to the new settings.
Please note that you cannot change the height of sections containing uprights.

## ACCESSORI DISTRIBUZIONE

This command allows you to select and add distribution accessories.
The program shows only the compatible accessories with the type of channel/pipe selected.
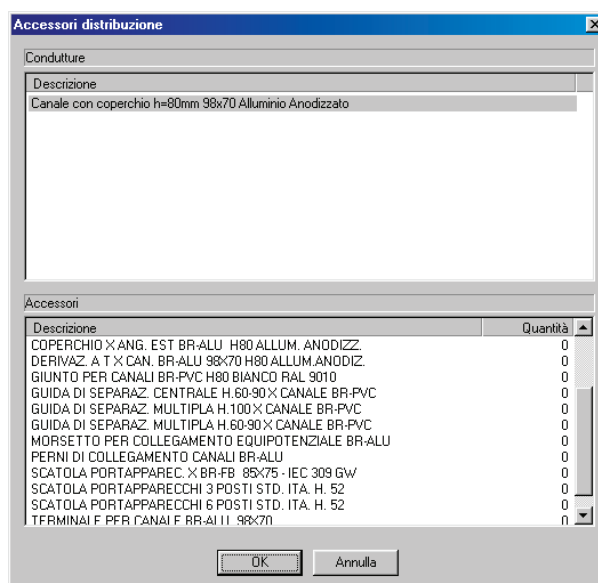


To add distribution accessories, select one or more distribution sections.
When opening the dialog box, select a code for the accessories and indicate the desired quantity. Repeat the operation with the other codes.
Press OK to finish.

## DISTRIBUTION IDENTIFIER

With this function you can add an identifier to a planimetry that allows to illustrate, in a simple and detailed way, the characteristics of the selected distribution section. The identifier can be composed freely, by selecting the options below.
Available options:
- A section view, complete with visual indication of the reached level of cramming, possibly divided by compartment;
- The symbol of the container (the code under which the product is identified in the bill of quantities);
- The description of the container;
- The list of the lines contained in each compartment, divided by code, description, and formation.

It is important to note how in the dialog it is possible to set the size of the identifier; the control takes place by means of the height of the text, to which also are connected the dimensions of the sectional view.
The inserted height is the one that will be taken by the texts on paper, after printing the drawing respecting the current scale.
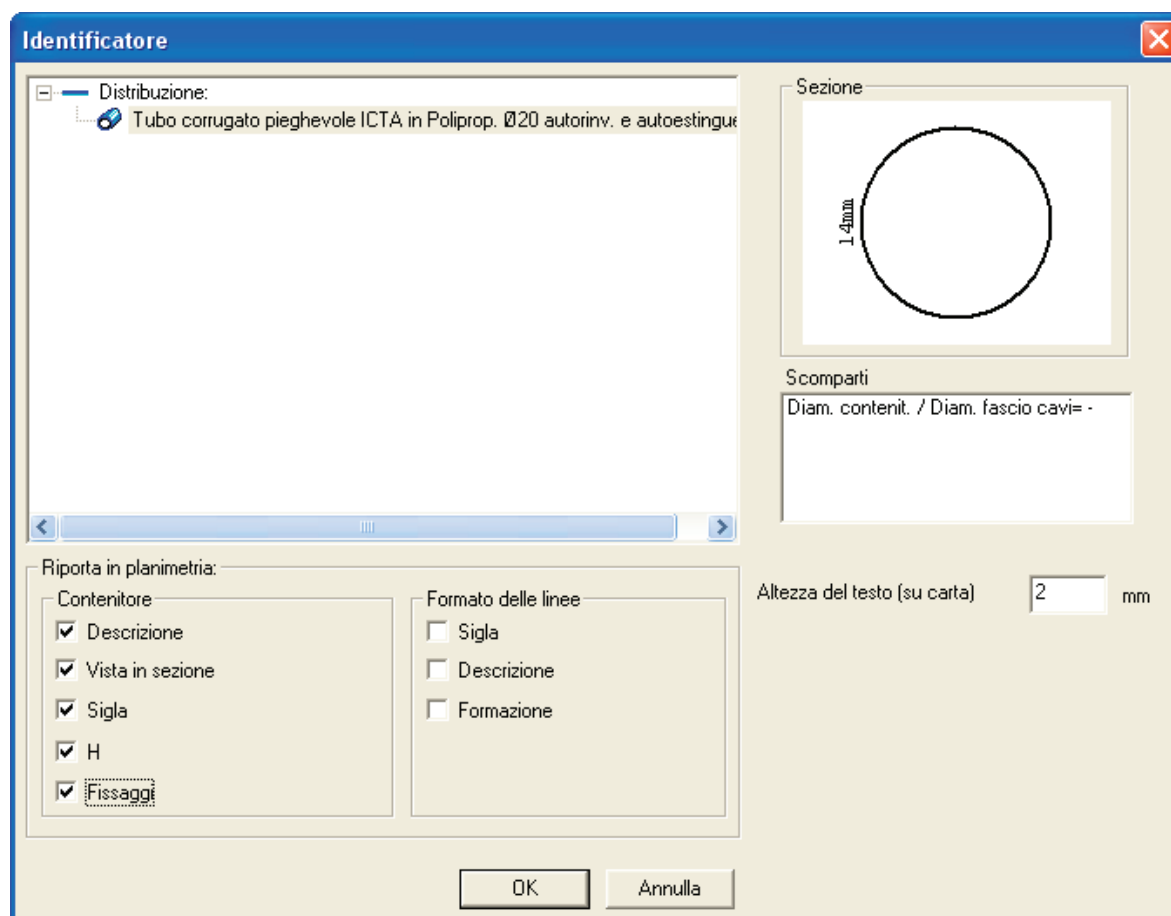


When the command is invoked, you are prompted to select a distribution section; when the dialog shown above appears, select the container to be identified, enabling the desired options.

After confirming with the OK button, you can enter the vertices of a broken line, which will connect the section with the identifier; press <enter> to end the broken line and insert the symbol.

Note : An identifier is inserted into the drawing as a block, and then you need to expand it in order to change it (with the ESPLODE command).
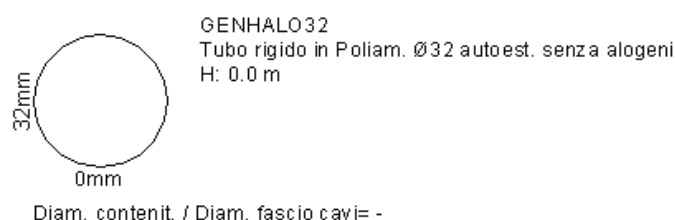
## LEGEND OF THE SYMBOLS

This command allows you to edit an existing line. This icon is enabled when a single line is selected.

Selezionare quale parte del progetto prendere in considerazione



Indicare l'altezza del testo espressa in millimetri
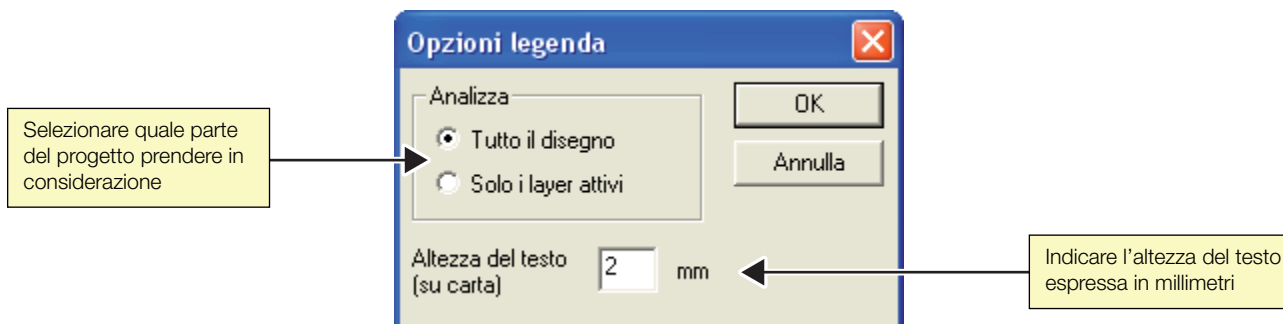
## LEGEND OF THE LINES

A legend of the lines included in the project is generated.
By running the command, the same dialog box as the LEGEND OF THE SYMBOLS appears.
Confirm your selections by clicking OK and define the insertion point of the legend in the project with a click of the mouse.
This command is run from the command line entering name LEGLINEE.

## LEGEND OF THE CONTAINERS

A legend of the containers included in the project is generated.
Running the command the same dialog box as the LEGEND OF THE SYMBOLS or LEGEND OF THE LINES appears.
Confirm your selections by clicking OK and define the insertion point of the legend in the project with a click of the mouse.
This command is run from the command line entering name LEGLINEE.

# LIST OF MATERIALS
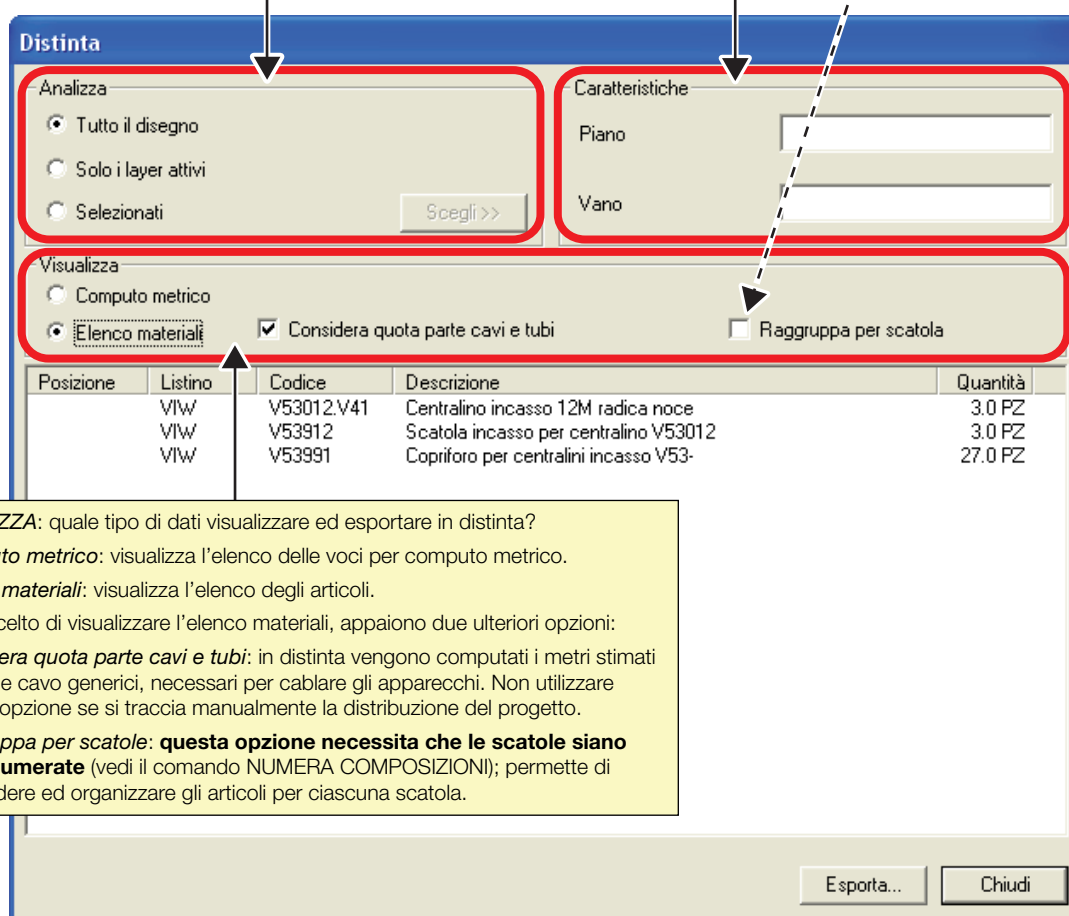
This command allows you to extract a list of materials from the drawing and/or a metric calculation of the project (or part thereof). By running the command, the following dialog box appears that allows to work on the type of bill to create, or on what to include in the bill.

*ANALIZZA*: cosa computare nella distinta?
- *Tutto il disegno*: ogni apparecchio inserito nel progetto senza distinzione.
- *Solo i layer attivi*: tutti gli apparecchi non nascosti.
- *Selezionati*: solo gli apparecchi selezionati; fare un click su *Scegli*>> e selezionare una o più scatole e/o apparecchi presenti nel progetto. Terminata la selezione degli apparecchi premere *Invio*.

*CARATTERISTICHE*: gestisce la suddivisione degli elementi analizzati.
- *Piano*: testo opzionale per identificare un gruppo.
- *Vano*: testo opzionale per identificare un sottogruppo.
- *Posizione*: associa a tutti gli articoli della distinta una posizione fissa. **Questa opzione è in alternativa all'opzione *Raggruppa per scatola*.**

**Distinta**

**Analizza**
- ⦿ Tutto il disegno
- ○ Solo i layer attivi
- ○ Selezionati          *Scegli* >>

**Caratteristiche**

Piano [                    ]

Vano  [                    ]

**Visualizza**
- ○ Computo metrico
- ⦿ Elenco materiali      ☑ Considera quota parte cavi e tubi       ☐ Raggruppa per scatola

| Posizione | Listino | Codice | Descrizione | Quantità |
|---|---|---|---|---|
|  | VIW | V53012.V41 | Centralino incasso 12M radica noce | 3.0 PZ |
|  | VIW | V53912 | Scatola incasso per centralino V53012 | 3.0 PZ |
|  | VIW | V53991 | Copriforo per centralini incasso V53- | 27.0 PZ |

*VISUALIZZA*: quale tipo di dati visualizzare ed esportare in distinta?
- *Computo metrico*: visualizza l'elenco delle voci per computo metrico.
- *Elenco materiali*: visualizza l'elenco degli articoli.

Se si è scelto di visualizzare l'elenco materiali, appaiono due ulteriori opzioni:
- *Considera quota parte cavi e tubi*: in distinta vengono computati i metri stimati di tubo e cavo generici, necessari per cablare gli apparecchi. Non utilizzare questa opzione se si traccia manualmente la distribuzione del progetto.
- *Raggruppa per scatole*: **questa opzione necessita che le scatole siano state numerate** (vedi il comando NUMERA COMPOSIZIONI); permette di suddividere ed organizzare gli articoli per ciascuna scatola.

[ *Esporta...* ] [ Chiudi ]

By pressing *Export…* a window opens that allows you to print or save the bill to file.

[ Esporta... ] [ Chiudi ]
    File in formato testo...
    File per Easycap...
    Stampa...

- *File in text format*: file *.TXT, which can be easily imported into other programs.
- *Interchange File*: specific file for EasyCap. The file extension varies depending on the selected type of bill: *.IEM for lists material, *.IMC for metric calculations.
- *Print*: prints the bill on paper.

# AXONOMETRY

Using this command you can get an axonometric view of the entire project or part thereof, according to a user-customizable point of view, obtained with the direct setting of the Azimuth and Zenith angles

This command syntax is as follows:

- *Selezionare oggetti:*
  select the entities you want to represent in three-dimensional view.

Muovendo le frecce si modifica la prospettiva della proiezione

È possibile impostare la proiezione con l'inserimento diretto dei valori Azimuth e Zenith

**Proiezione assonometrica**

Imposta angoli di Azimuth e Zenith (scrivi i valori o trascina le frecce)

90°

180° 0°

270°

90°

0°

90°

Da: Asse X 270.00    Piano XY 90.00

Anteprima

L'anteprima mostra in tempo reale come si presenta la proiezione in base ai valori di Azimuth e Zenith impostati

OK    Annulla

- *Posiziona l'assonometria:*
  position the axonometric view anywhere on the drawing, making use of the zoom and spostamento commands.

For greater clarity…
**Azimuth** is the angle that the observer-subject connecting line forms with the axis X.
**Zenith** is the angle that the observer-subject connecting line forms with the XY plane.

0°

0°
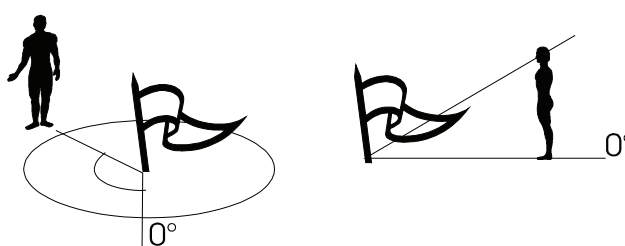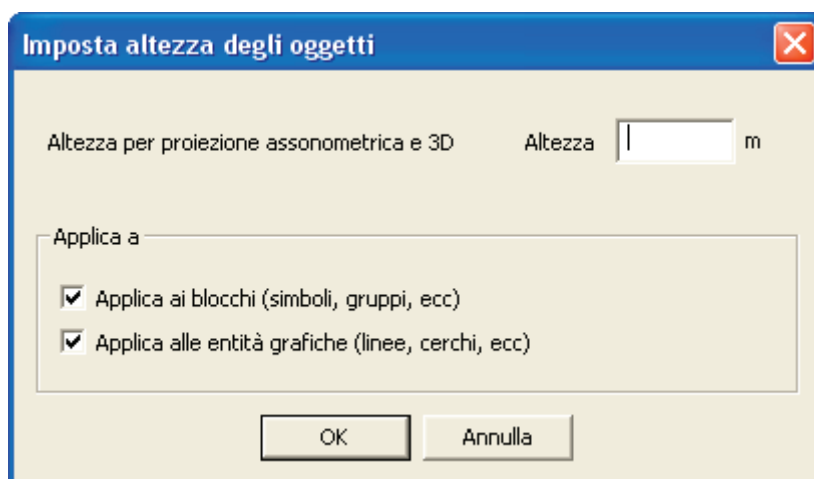
Azimuth and Zenith

## SET h

In the projection, the objects (symbols, groups, etc.) and the graphical entities (circles, lines, etc.) in the drawing are automatically placed at the assigned height.
For all symbols, the distribution lines etc., the height can be set directly when inserting the object.

The inclusion of graphical entities does not provide this option, and then you must tell the program what height the object inserted into the projection should be placed.

# EasyDraw & BY-ME

## NEW BY-ME PROJECT

**Main Menu -> NEW -> Nuovo Progetto BY-ME**
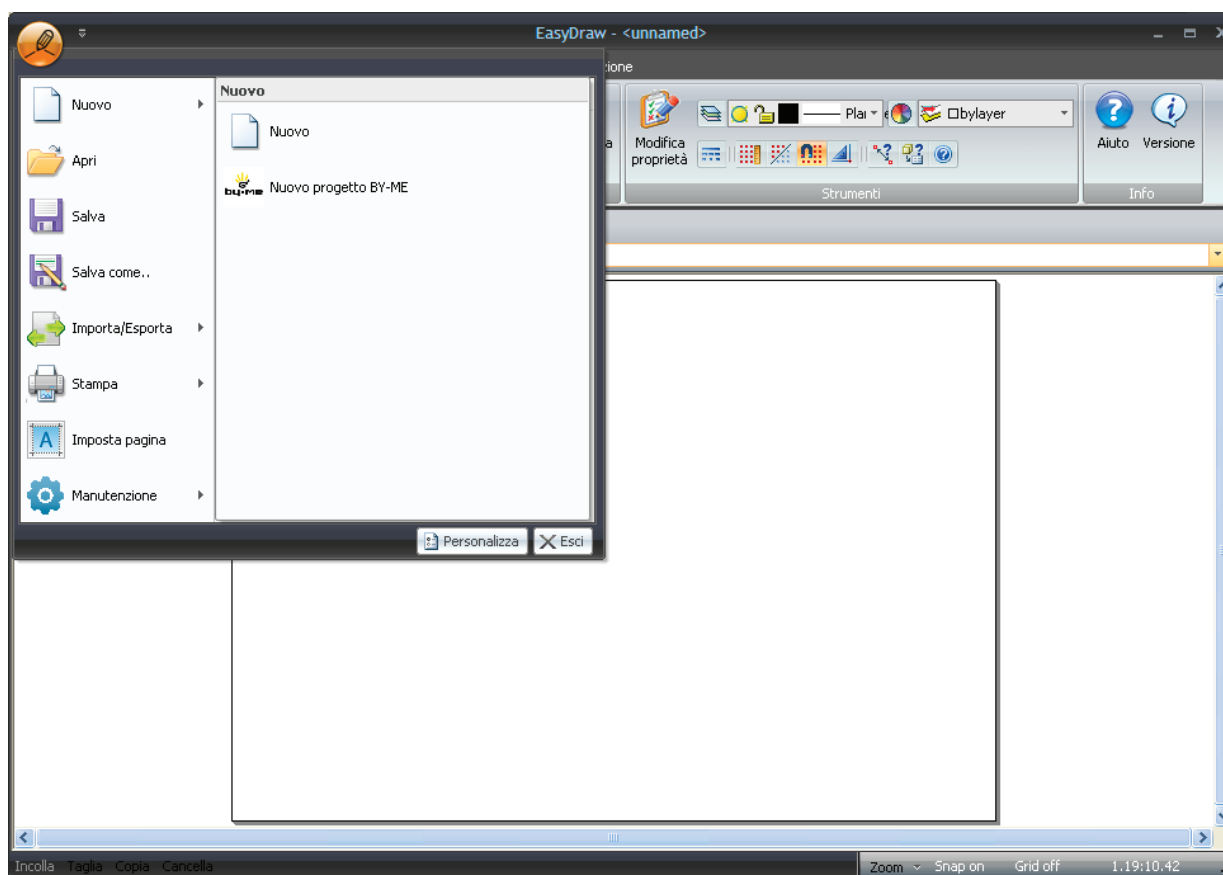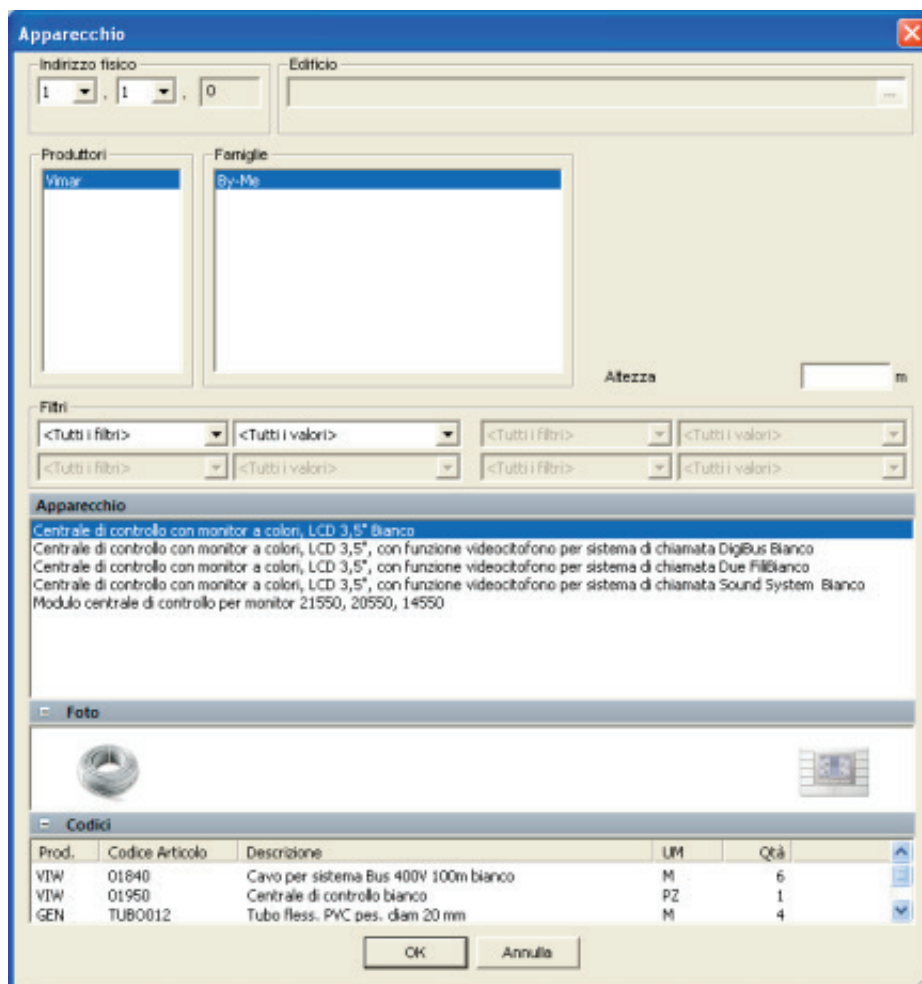


The "Nuovo Progetto BY-ME" command prompts you to choose what type of installation to create:
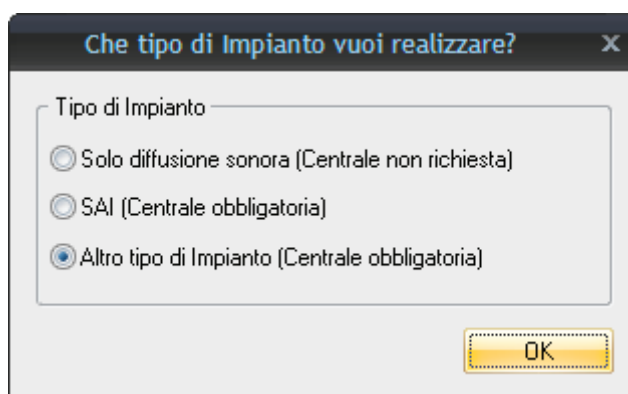
If you intend to manage a "sound diffusion" system, a control unit is not required, and then you can choose the first option: "Only sound diffusion"
If the system has to handle various BY-ME applications (SAI*, automation, climate, load, sound, scenery, etc.) you will need to choose the "Other type of installation" option and you will be prompted to enter a control unit.
By default, the proposed "Control unit with color monitor, LCD 3.5" White" "(code 01950)". You can select a control unit to your liking among those available in the "Device" list



The control unit is placed on the Area/Line 1.1.



**\* If in the project you intend to manage the configuration of SAI (anti-intrusion) devices you need to move the control unit to the Area/Line 0.0.**

## PLANIMETRY MENU
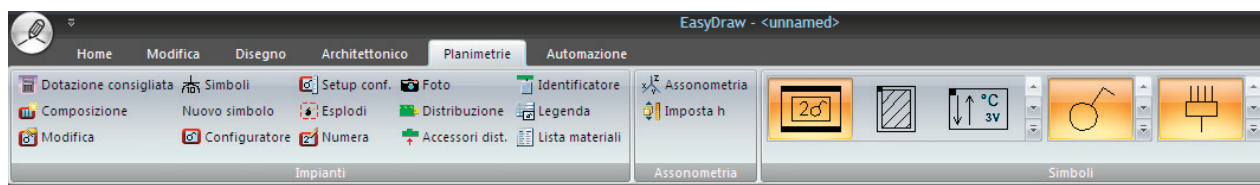


The Planimetry menu contains a series of commands dedicated to the drawing of the installation: commands for inserting electrical symbols, draw BUS lines, create compositions, etc. Exceptions are the commands to draw the planimetry which are within the Architectural menu.

These commands have already been described in the user manual accompanying the Software. In this quick guide we will examine only those involved in the new home automation functions.

**Symbol Command**
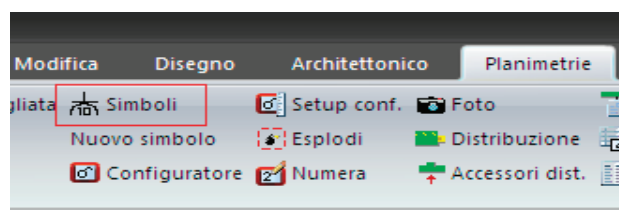


The "Simboli" command shows the EasyDraw symbols organized into thematic folders.  The BY-ME devices are contained in the folder
**Sistemi->Automazione BY-ME**
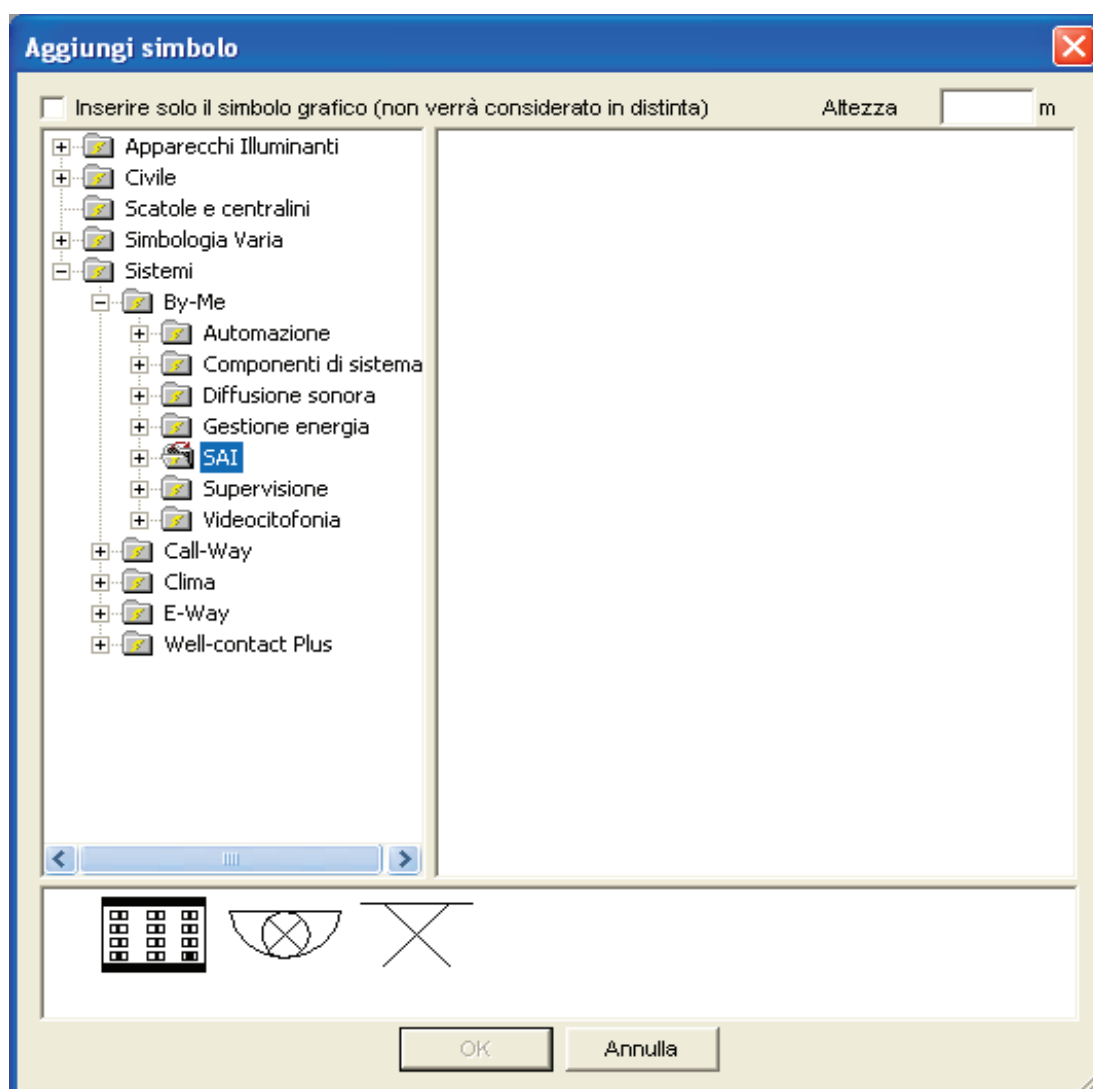
**Inserting a symbol**

The selected symbol can belong to one of the following categories:
    a) BY-ME device
    b) utility
    c) power supply for BUS BY-ME systems
    d) traditional device that cannot be configured

    Depending on the circumstances, the "Insert Device" interface will require various information.


**a) Inserting a BY-ME device**

    If the device is BY-ME, it will prompt for the Area and Line to link to it

    Example:
    Entering "tre pulsanti basculanti" command
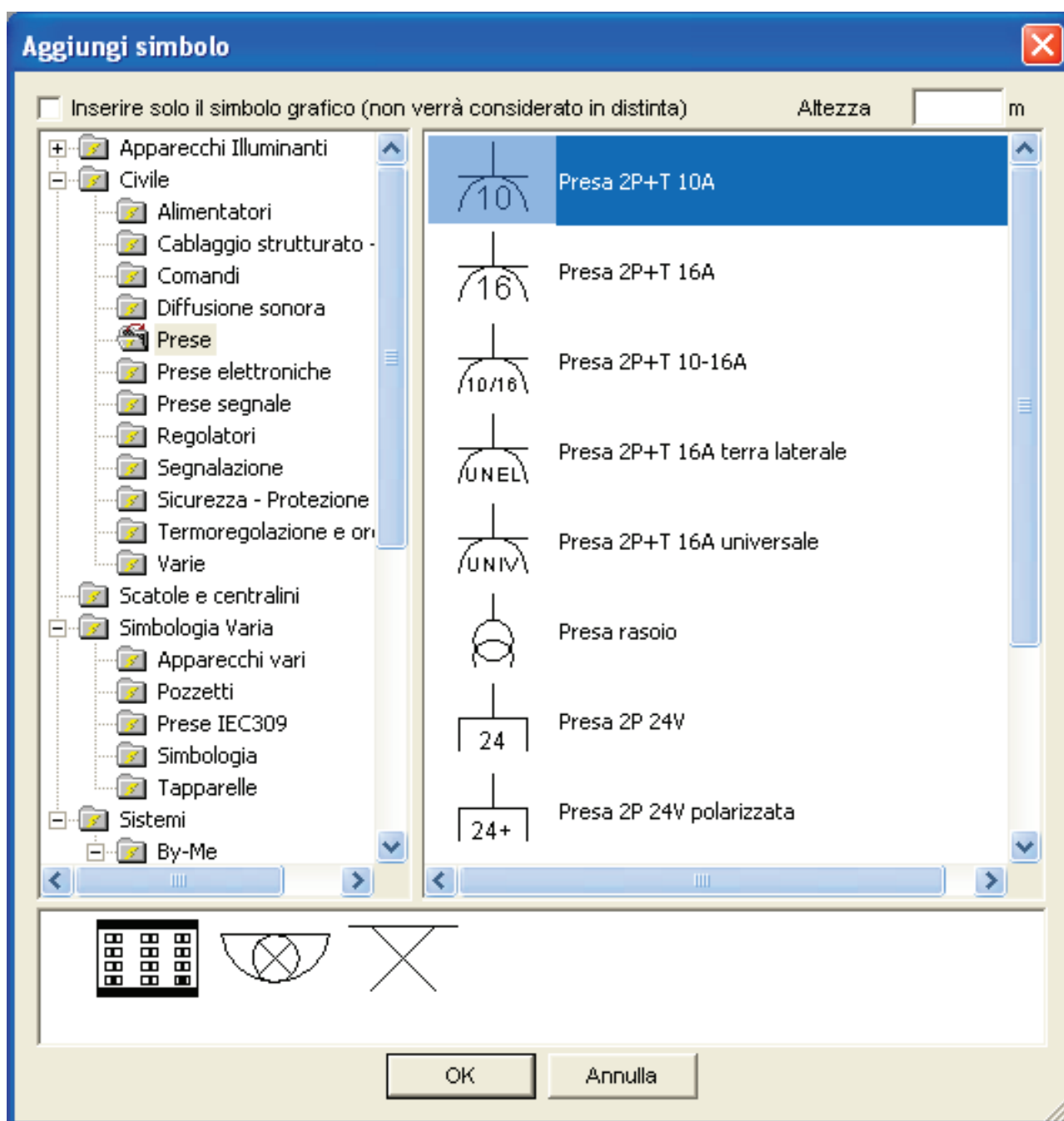
Area and Line can be changed at the discretion of the user. The combos show the areas and lines defined in the project. To create new ones you must access the "Areas and Lines" in the "System Topology".

**The user can only choose Area and Line, but cannot assign a physical address to the device.**

The physical address is calculated automatically by the software when inserting the device into the drawing. In fact, you will see that in the "Physical Address" field, the address of the device is empty.
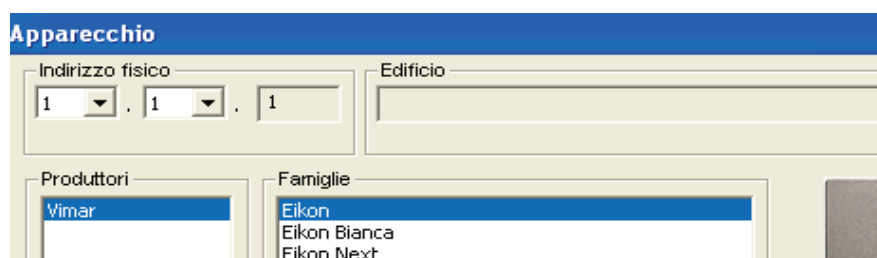
By pressing "OK", the device will be inserted in the drawing:





By "double-clicking" the device, you can return to the previous dialog. You will notice now that the Physical Address has been calculated.

**Each BY-ME device inserted in the project has a Physical Address.**

## a) Inserting a utility

The utility (lamp, shutter, socket, etc.) is a device that does not have a physical address because it is managed by an actuator. The utility, however, has a description that is used to identify it in the project.

Select the composition ➡ select the desired device ➡ select the "By-me" drop-down menu ➡ click "group configuration" ➡ select the relay functional block ➡ click "select utility".

 Example:  Inserting a socket
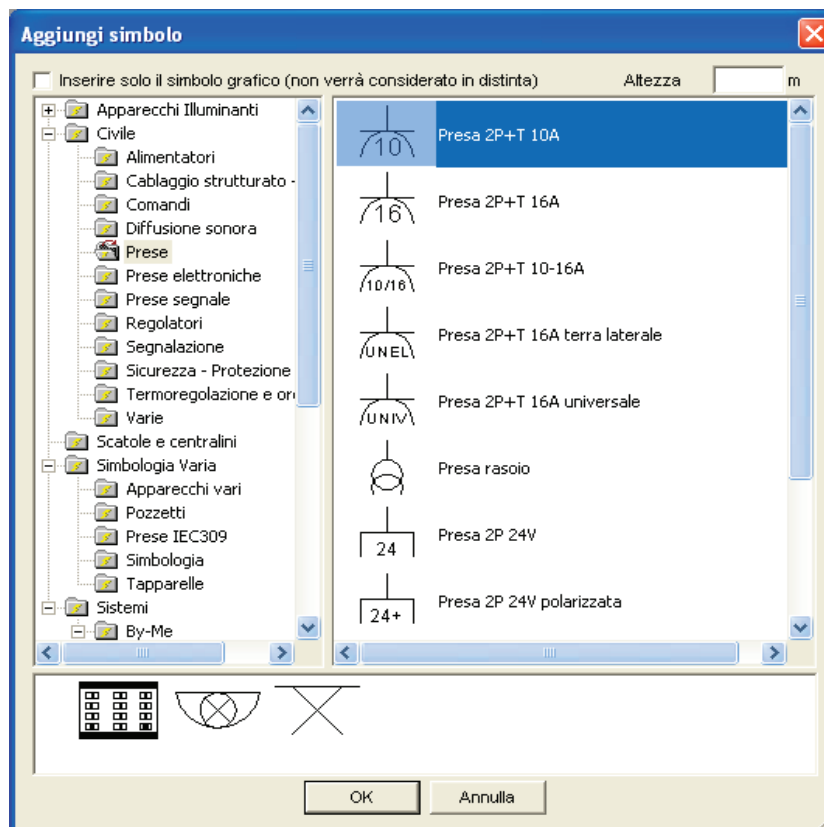


By pressing the "..." button, you can create a new description or choose one of the existing ones.
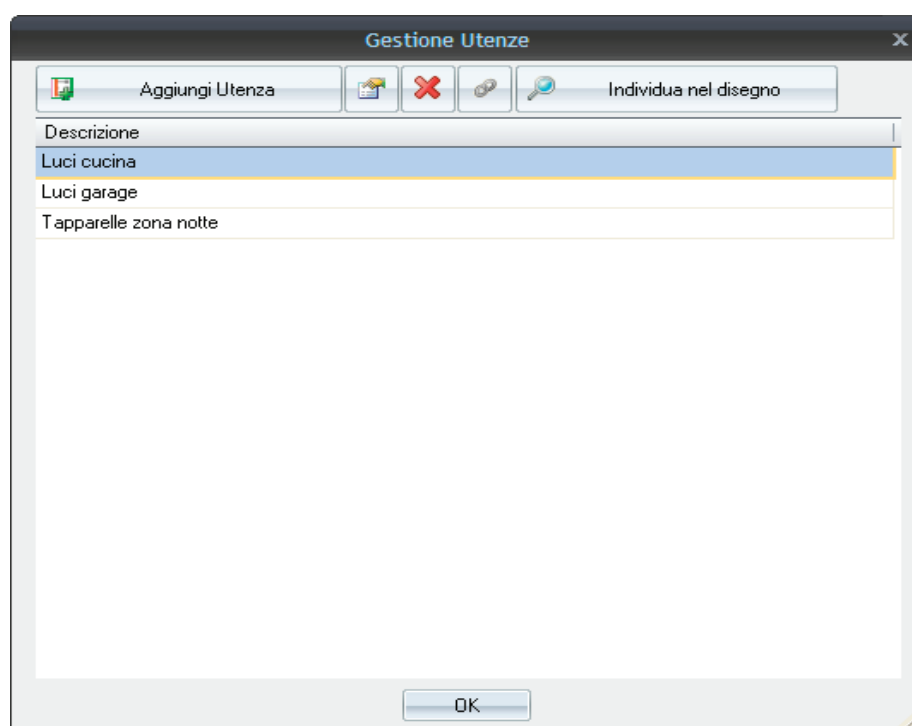
The following example shows how to create a new description by clicking "Add utility".
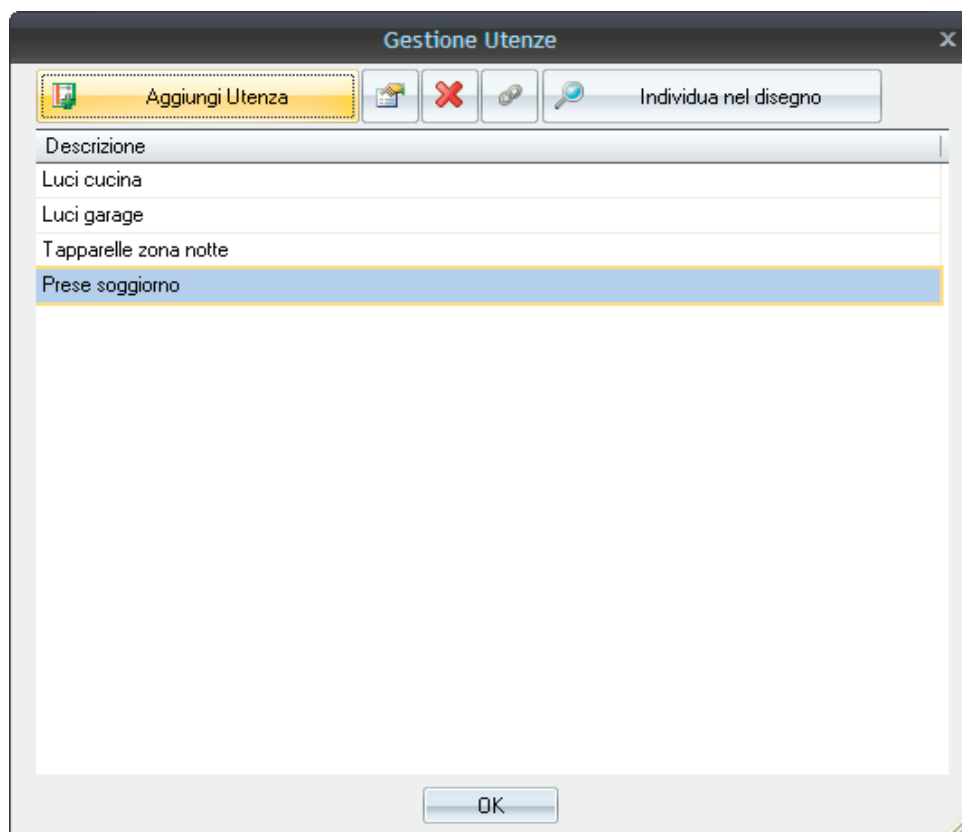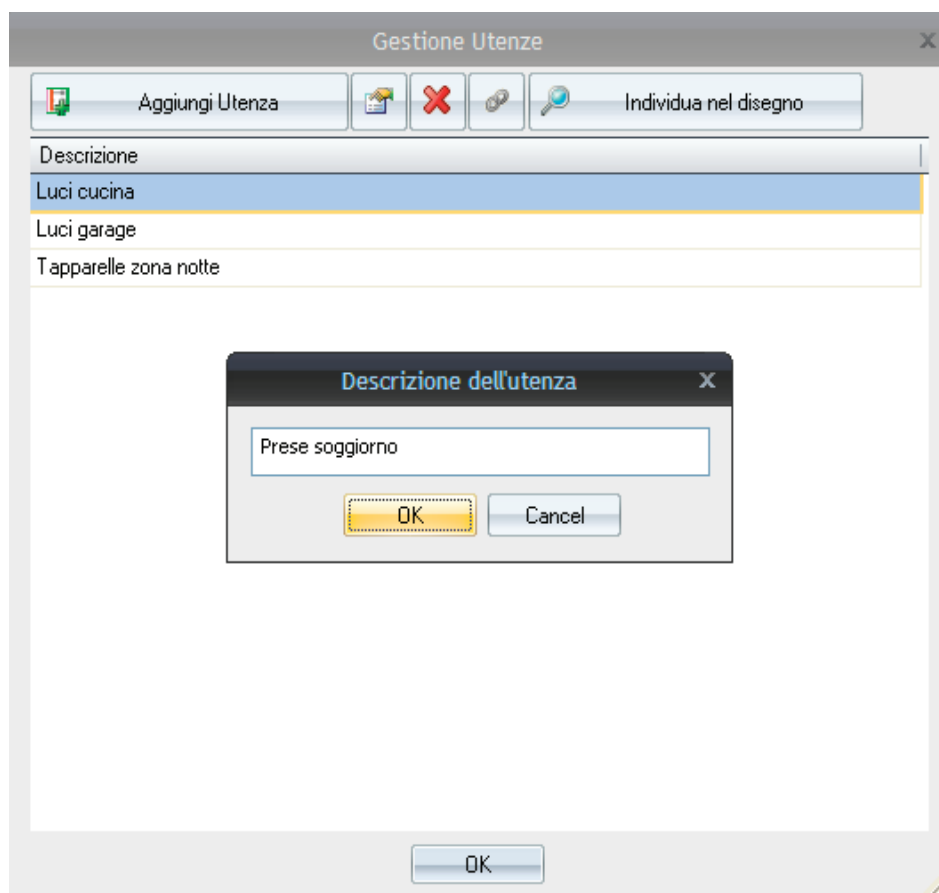
By pressing "OK" the description "Living-room sockets" will be associated with the utility
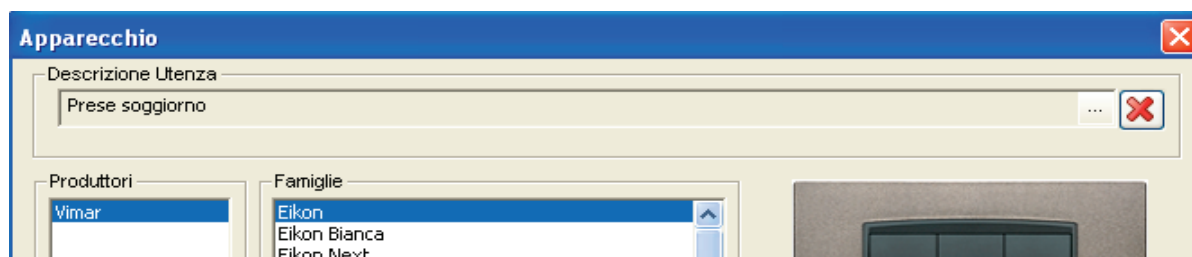


### c) Inserting a power supply for BUS BY-ME systems

A power supply for BUS BY-ME systems does not have a "Physical Address"; however, you will be prompted to set Area and Line. In this case, Area and Line are only part of the topological information allowing the software to know where the power supply is connected. This is to make certain consistency checks of the circuit:  check if the number of power supplies per line or to the distances between power supplies and devices are respected.

Example Inserting a power supply
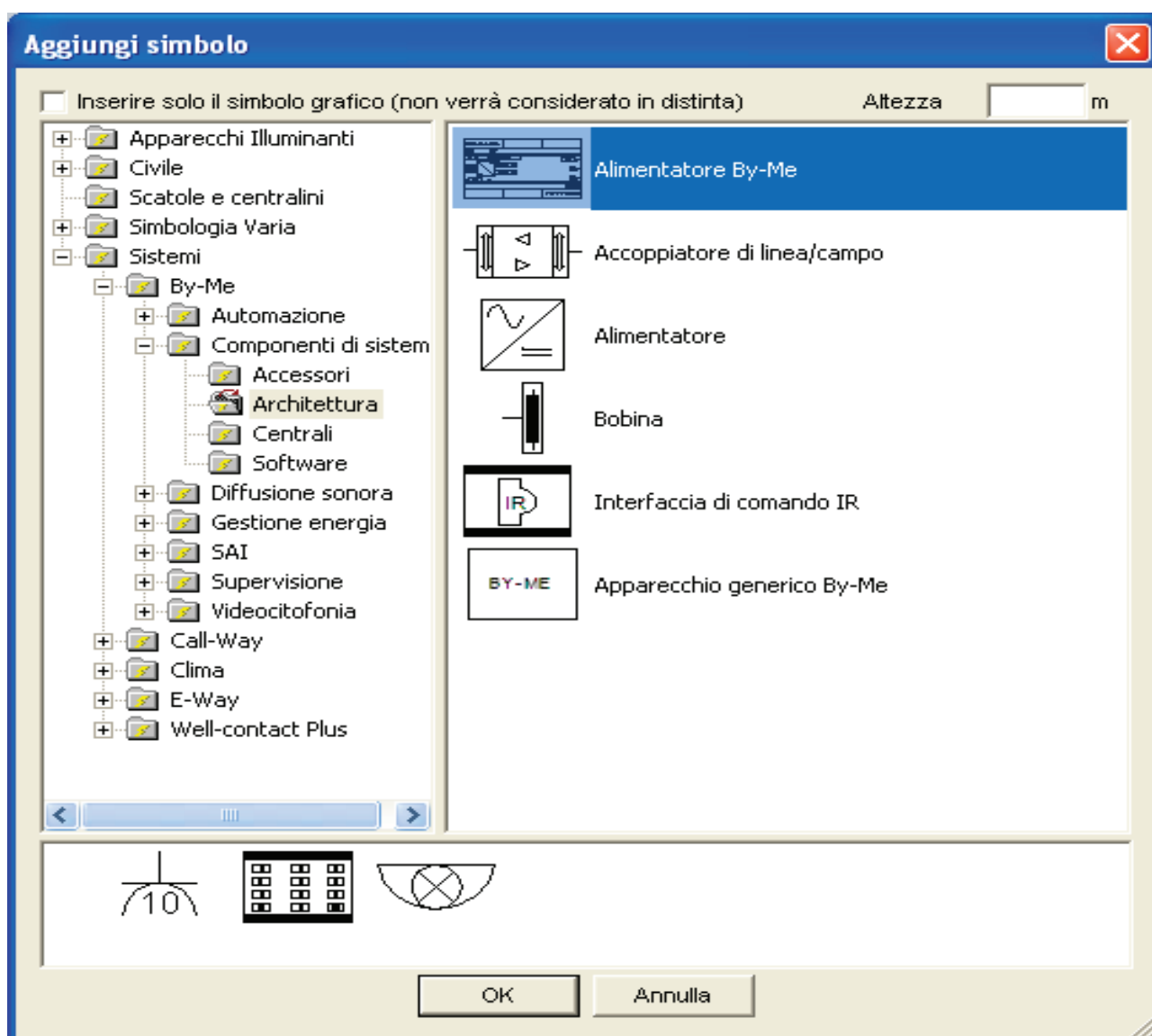
The "Insert Device" interface will require Area and Line only for "information" purposes to carry out precise consistency checks of the circuit.



**a) Inserting a traditional device**

If the chosen device does not belong to any of the categories mentioned above, it is considered generic and shows the Standard Device Input interface.

# AUTOMATION MENU



## SYSTEM TOPOLOGY PANEL
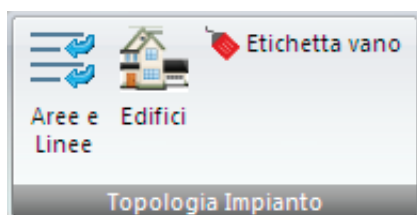


In "System Topology", the commands allow you to add/edit areas, lines, and buildings.

## BY-ME PANEL Groups and Scenarios
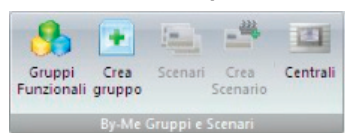


In the "BY-ME Groups and Scenarios" panel, the commands allow you to Add/Change Functional groups and Scenarios and manage the Control units of the system, up to a maximum of 8.

### Command Crea Gruppo



The Crea Gruppo Funzionale command allows you to configure a group of BY-ME devices in the project.

The Functional Group may include:

      a) Only BY-ME devices

      b) BY-ME devices + utilities

**a) Create Group Command:  only BY-ME devices**

To create a functional group it is not mandatory to connect the utilities. The association to the utilities can be performed at any time, even after the creation of the group.

Example:
In the drawing have been inserted two BY-ME devices: a "three tilting keys" and a "relay actuator with 4 outputs"
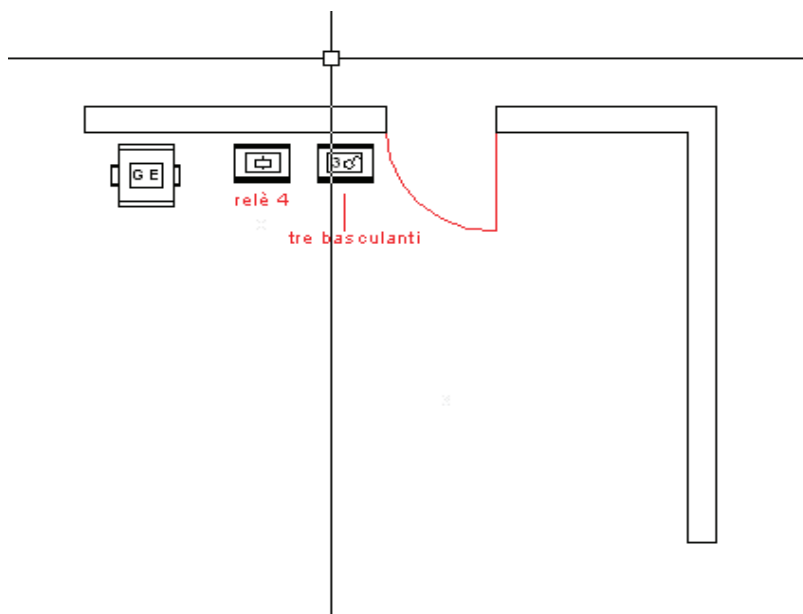


By clicking on "Create Group" you are prompted to sequentially select the devices that will be part of the Group.  The order of selection does not matter. Select, for example, the "three tilting keys"
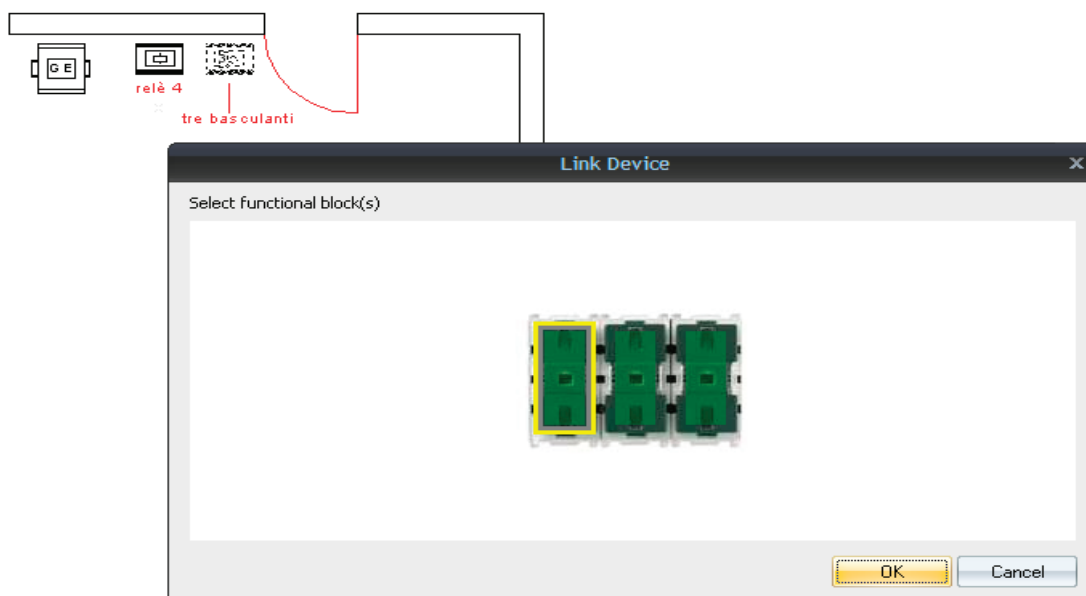


The selected device can have multiple functional blocks. The functional blocks compatible with the devices selected in sequence will appear in green. The selection is multiple, you can choose more than one functional block. The selected blocks will appear in "red".
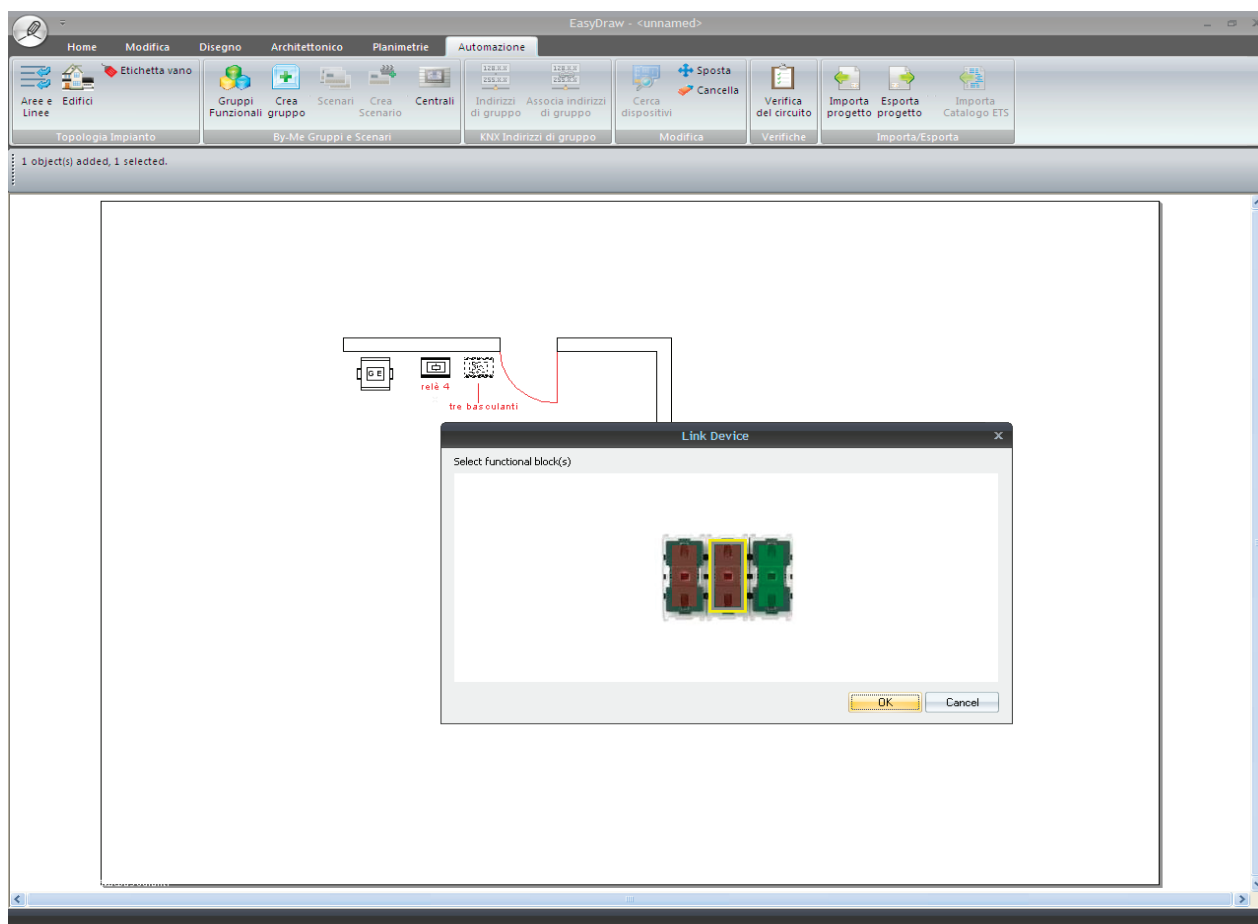
By pressing "OK", the control returns to the drawing and you can continue to select other devices. For example the actuator.
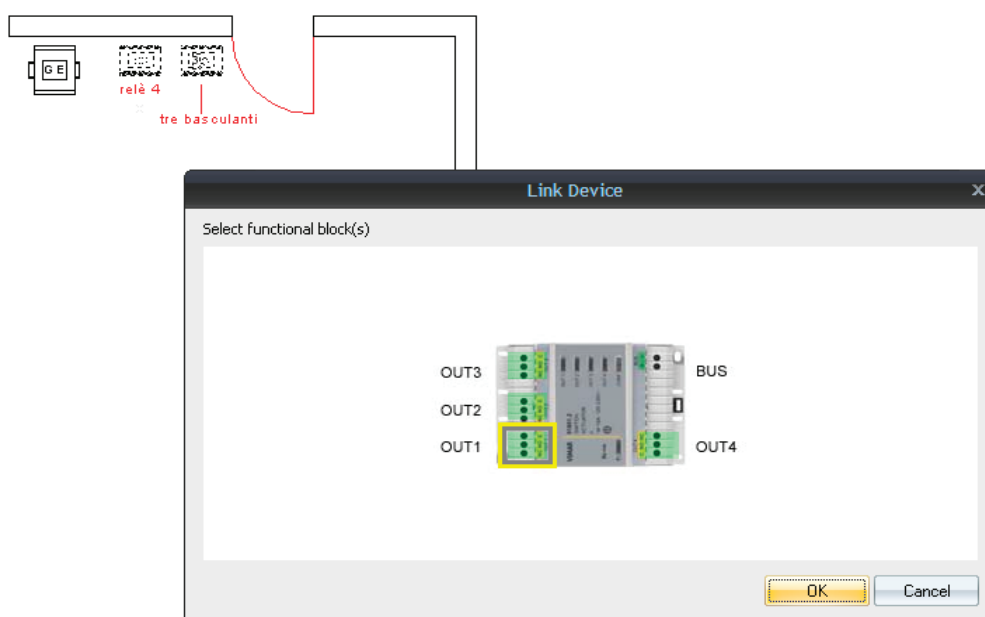


Also in this case the selected device has more functional blocks. The blocks compatible with the buttons on the "three tilting keys" selected before will be shown in "green". Choose, for example, OUT1.
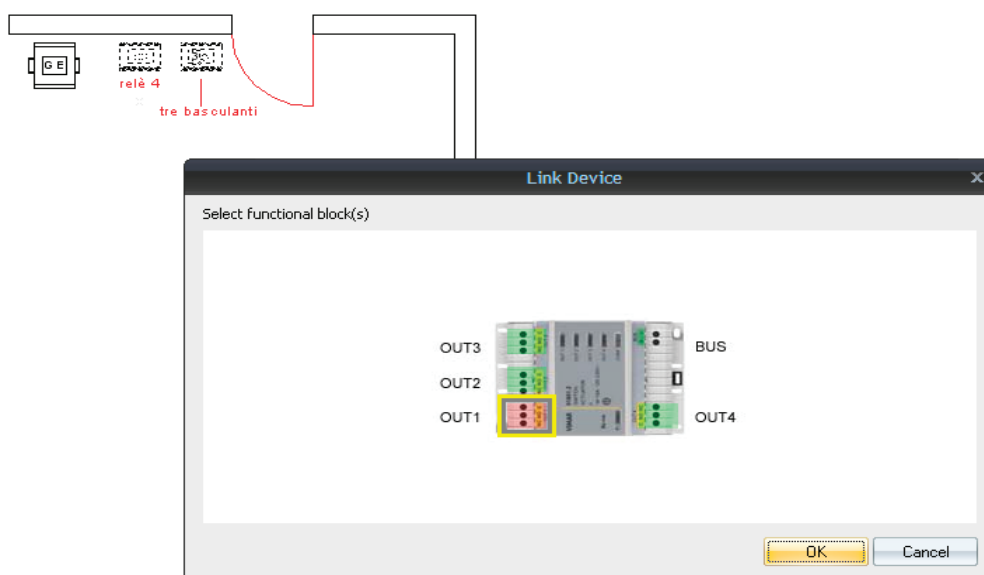
By pressing "OK" the control returns to the drawing and you can continue or "create the group" by pressing "enter" or "right-clicking".
As a Functional Group represents the function (or Application) the devices that compose it must perform, if the possible functions are n, the software prompts the user to choose one.
In the example, the software determines that the group you are creating can perform two different functions (or applications): automation or loads.
The user must choose the desired application.
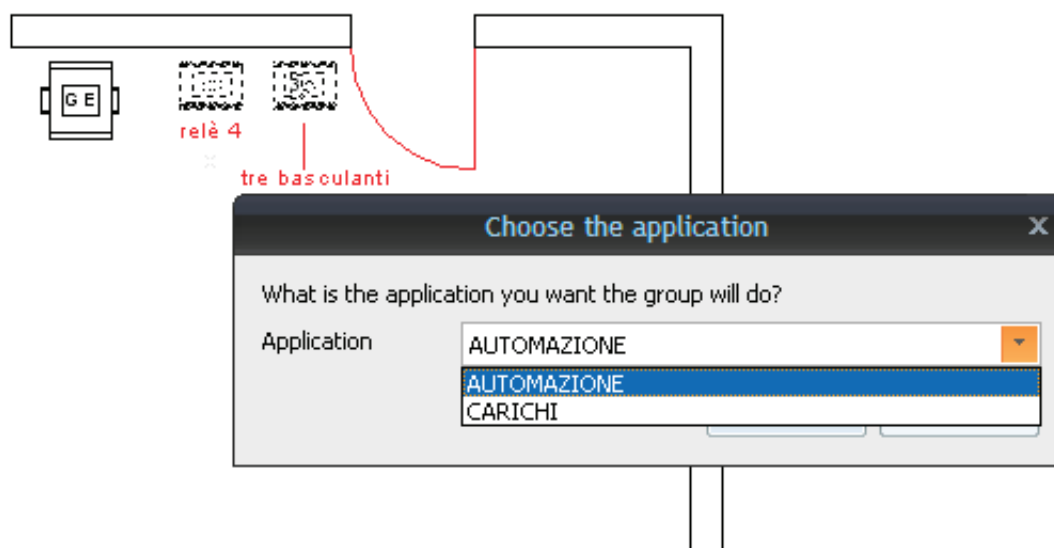
After choosing the Application, the group is created and shown in the "Functional groups" interface. Through this interface, you can edit the properties of the various devices, but also modify the newly created Group: add new devices, delete existing ones, edit the description of the Group.

**IMPORTANT: For the line coupler, the communicator, and the Web Server, you must create a dedicated group.**

**b) Crea Gruppo dispositivi BY-ME configurabili + utenze command**

As well as creating a Group, you can choose the utilities to be connected.

For example, you create a group formed by the "three tilting keys", "relay" and a "socket".
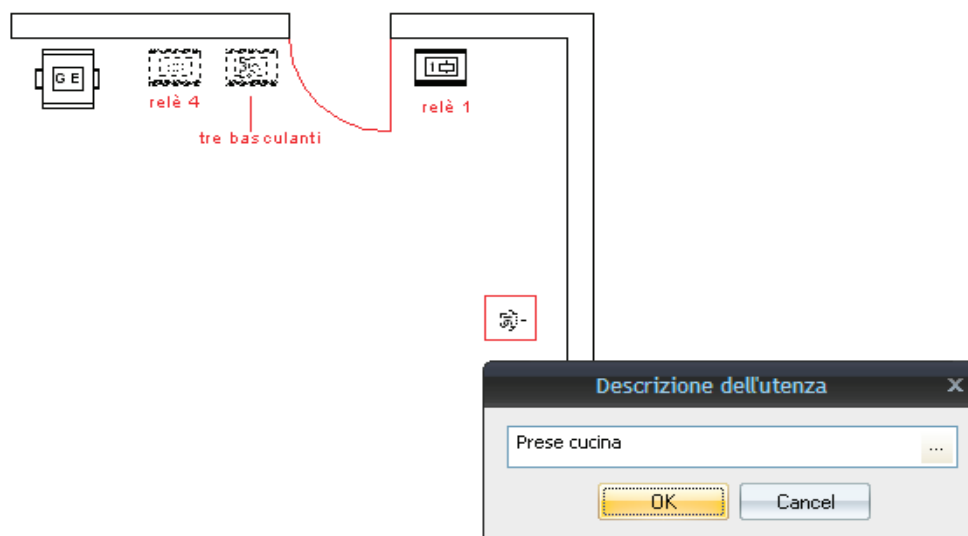
If you select a utility (e.g. socket) without description, the software will prompt you to enter it.  In the example the description "Kitchen Socket" is added. Closing the Group with "enter" or by "right-clicking", the terminal of "relay 4" OUT1 will be connected to the utility "Kitchen Socket".
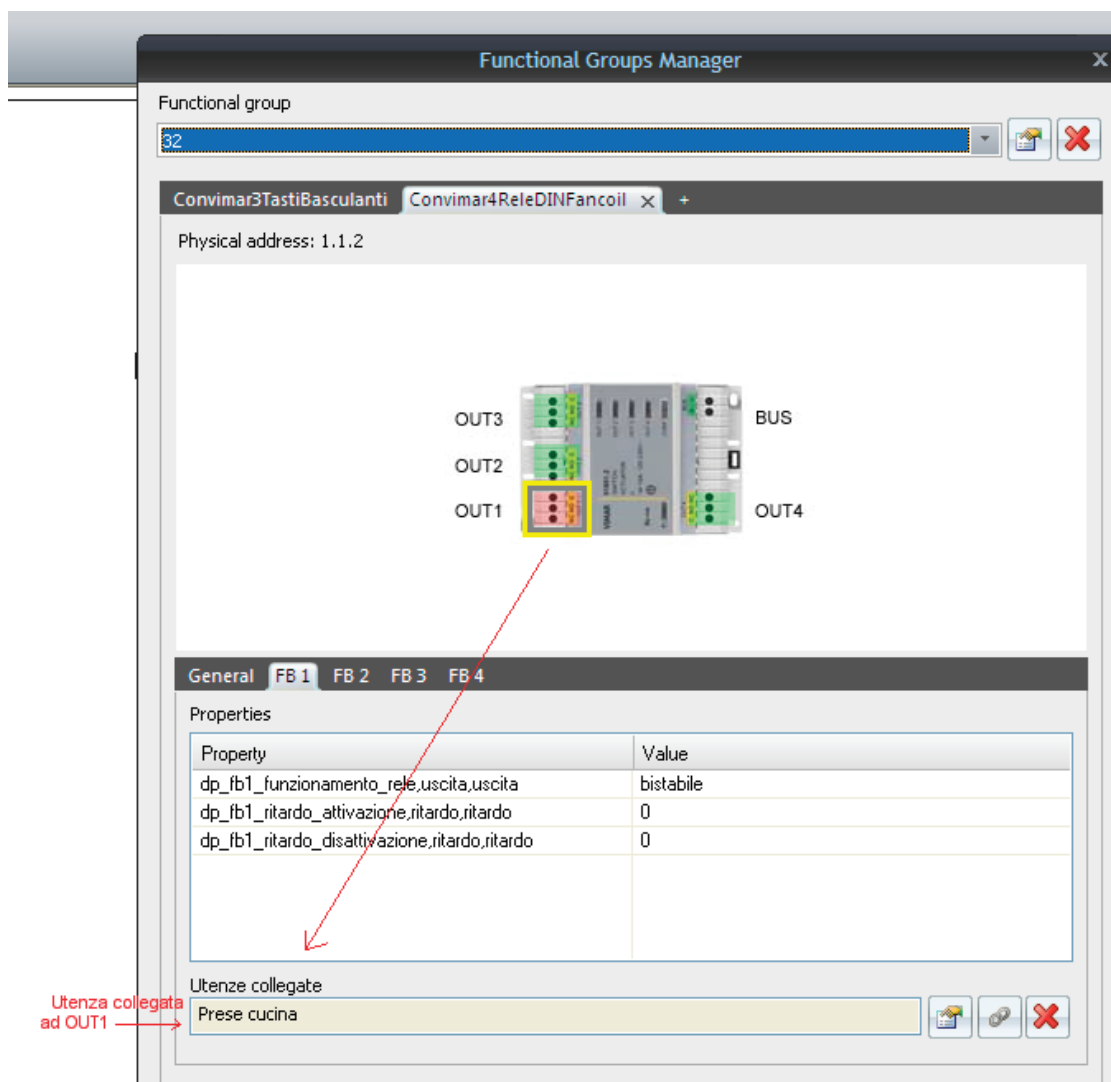


**Compared to the previous case (a), the software did not request the application from the user.**

This is because some utilities indirectly suggest the type of function the Group will perform. For example, in the database of the EasyDraw, the "sockets" are associated with the application "loads". Therefore, the software automatically selects the Application "loads" to create the group.

**NOTE: in a group cannot be connected utilities that belong to different applications.**

Creating a group by selecting BY-ME devices and utilities has a double advantage:

      1 ) The utility is directly connected to the actuators of the group

      2 ) The utility suggests to the software which application to use, without prompting the user.

**NOTE: in reality, the utility is physically connected to an actuator. If the group does not have at least one actuator, the utility will not be connected.**

**"Gruppi Funzionali" Command**
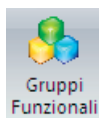


The interface of the functional groups is the "control panel" of all the functional groups in the project and defined in the **Current control unit.**
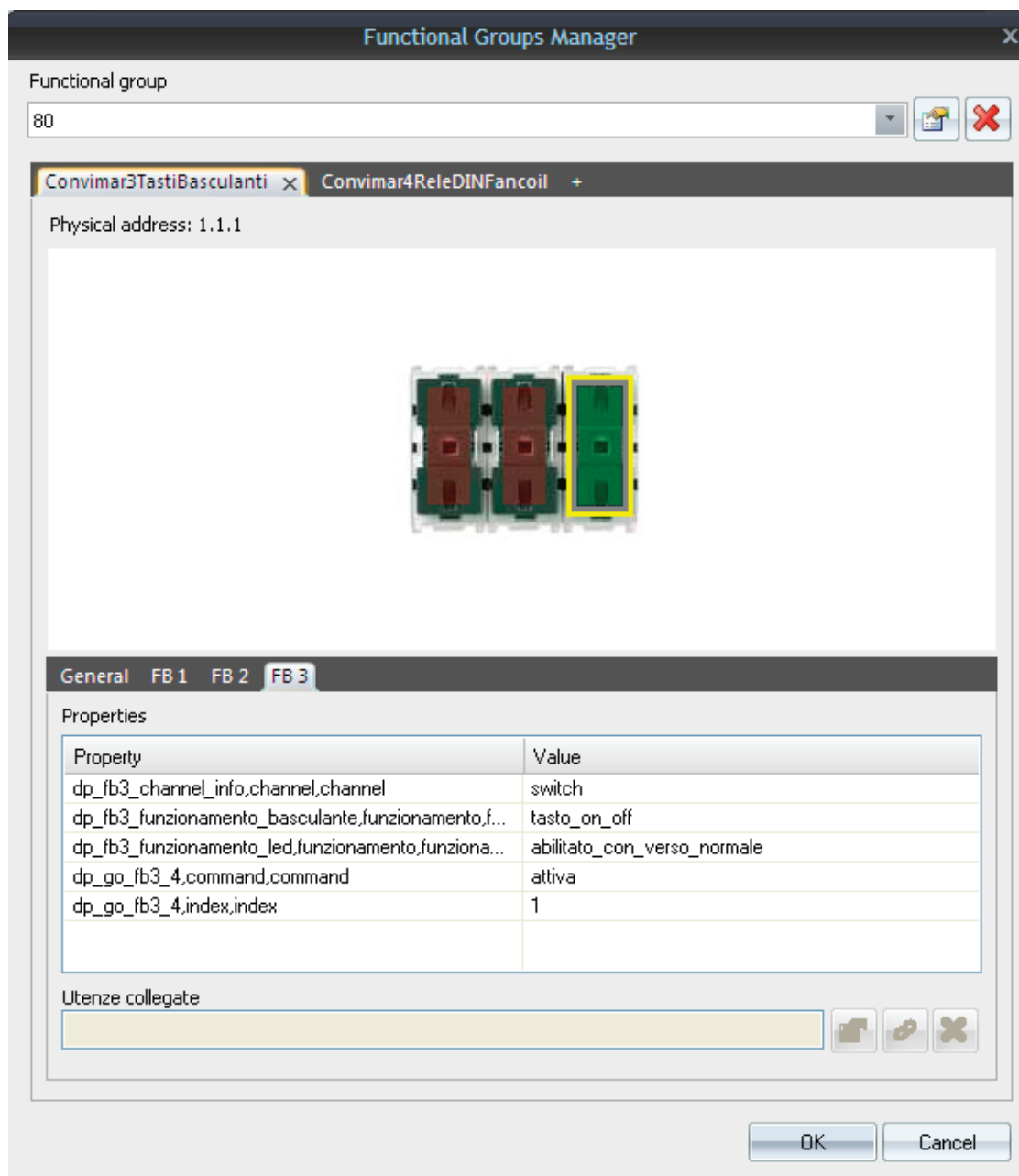
**Group Description**



The last created Group is highlighted on top of the combo (e.g. 80). "80" is the unique index that is assigned automatically by the software to a Group.  This description is not an end in itself, but it is the same one that appears on the display of the Physical Control Unit when the project is online. The expert installer may modify this description using the "properties" on the right.

**Group Devices**

The devices that make up the Group are organized in a property sheet in the center of the interface where each page is a device.
 For each device (in this example the "three tilting keys" and the "relay 4") the following is shown:
The list of function blocks connected to the group (in red)
The list of properties of each functional group (drop-down menu FB1, FB2, etc. positioned in the lower part of the screen).
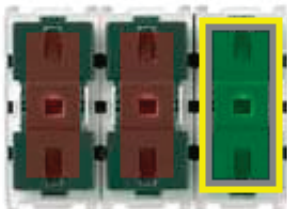


**Utilities connected to the Group**

The lowest part of the interface, shows the associations between the Group and the utilities. Only if the Functional Block is an actuator, can you edit these associations (removing them, adding them, etc.).

## EDITING A GROUP

You can edit the Group by adding new functional blocks or deleting existing ones

### a) Add/Remove a Functional Block to a Group

To add a Functional Block just click on the image corresponding to the Block to be added/removed. For example, to add the third button on the right, click on the green button. The software will prompt for confirmation. By pressing "Yes", the new block will be added to the group. To delete a Block proceed in the same way by clicking on the Block to be removed.
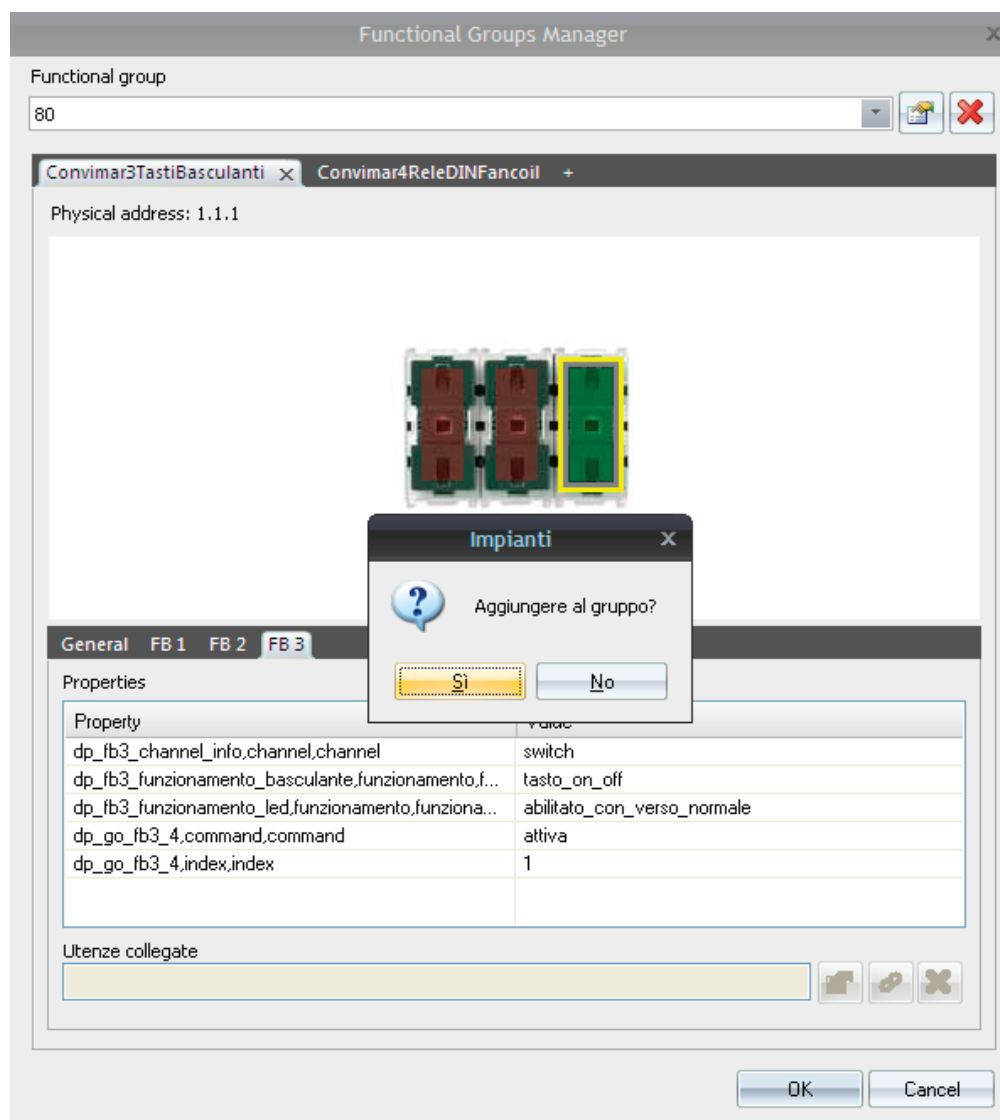
**b) Add a new device to the drawing**

It is possible to add a new device to the Group in the drawing.
For example, in the drawing has been added another 1 output relay (relay 1)
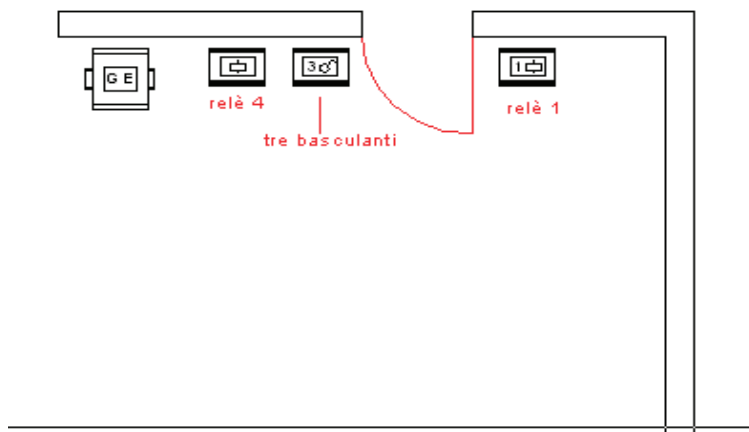


To add the relay to Group 80, simply click on the tab "+" (highlighted in the red rectangle)
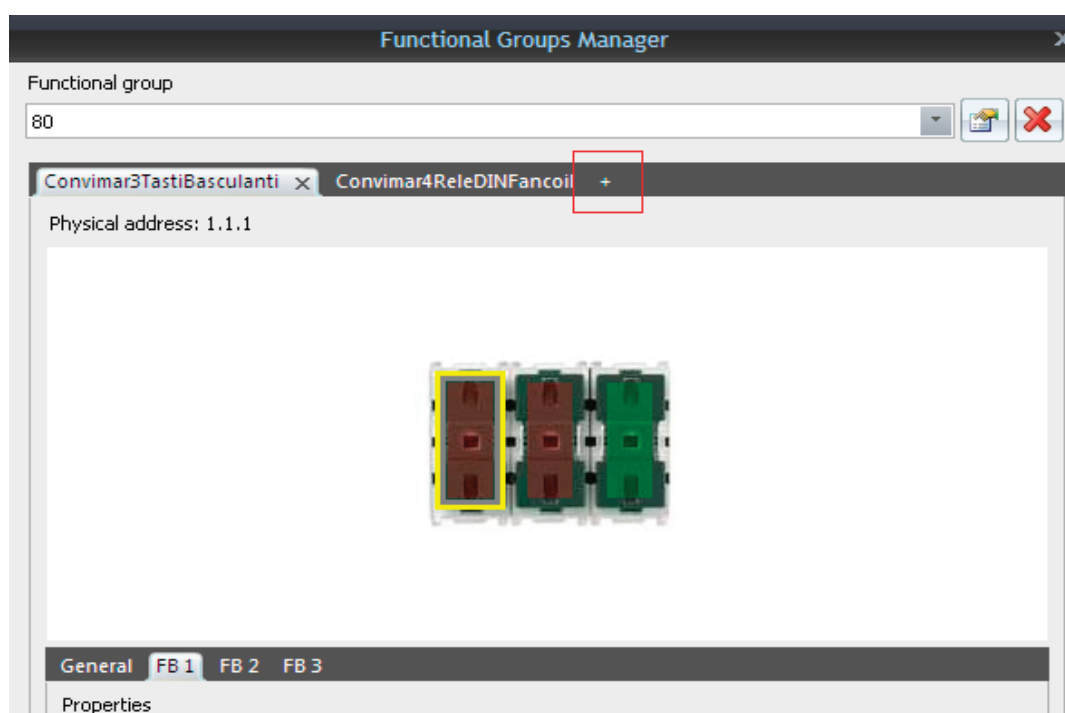


By clicking "+", the control passes to the drawing and you can select the new device
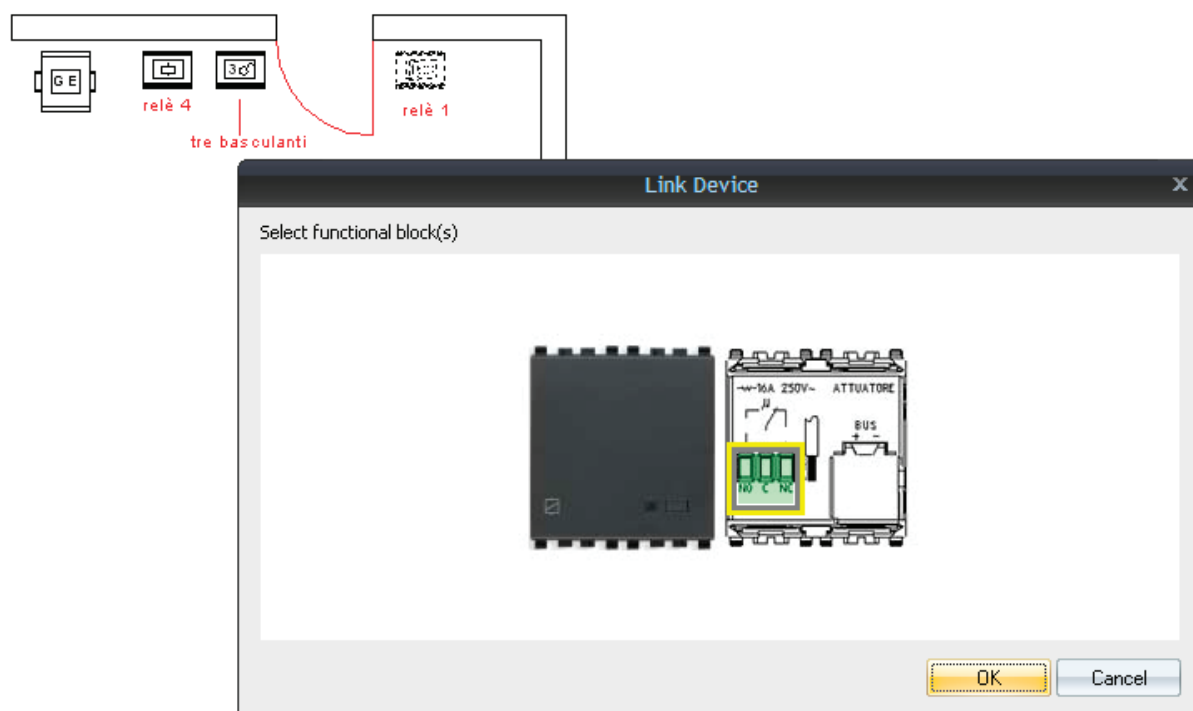
In "green" is highlighted the relay output. By clicking it and pressing "OK", the new relay will be added to the Group.
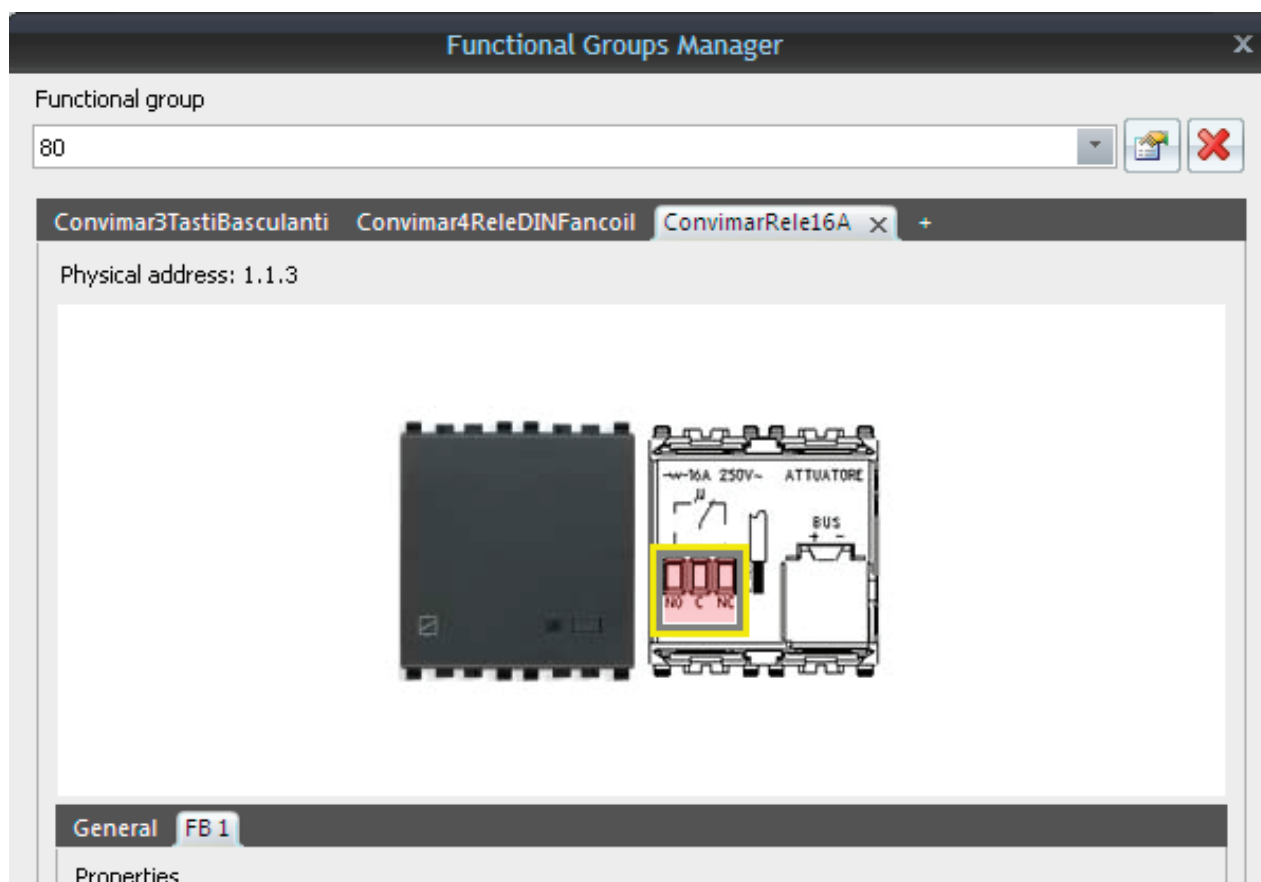
**Functional groups pre-filtered by Device**

As mentioned above, the "Functional Groups" interface shows *all* Functional Groups of the project related to the current Control unit. You can only access the Groups for a specific device by double-clicking them and clicking "Functional groups of the device" that appears on top right. This opens the usual Functional groups interface, but this time the combo at the top will only show up the Groups related to the selected device.
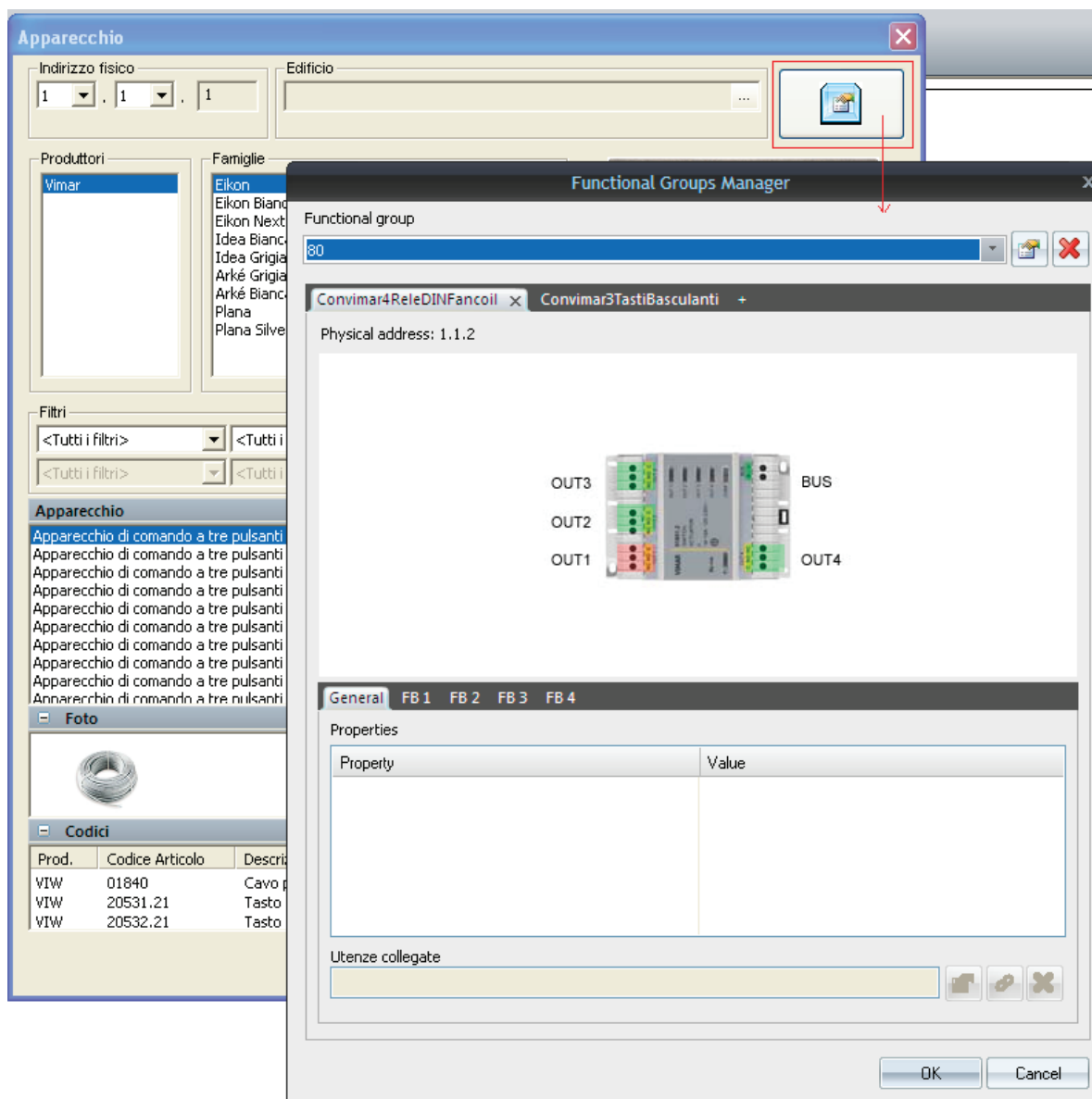


**Functional groups of the Control Unit**

By "double-clicking" a Control unit, the "Functional groups of the device" button displays the groups stored in the Control unit..

**VERIFICATION PANEL**

**Verifica del Circuito Command**
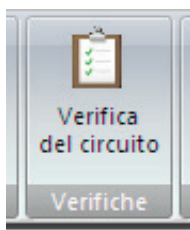


The "Verifica del circuito" command, checks if the installation complies with the BY-ME installation drawing rules.
Some of these checks take into account the distances between the devices. The distance checks are performed only if the BUS that connects the devices is being traced (see DISTRIBUZIONE command).
Clicking the command opens the following interface.
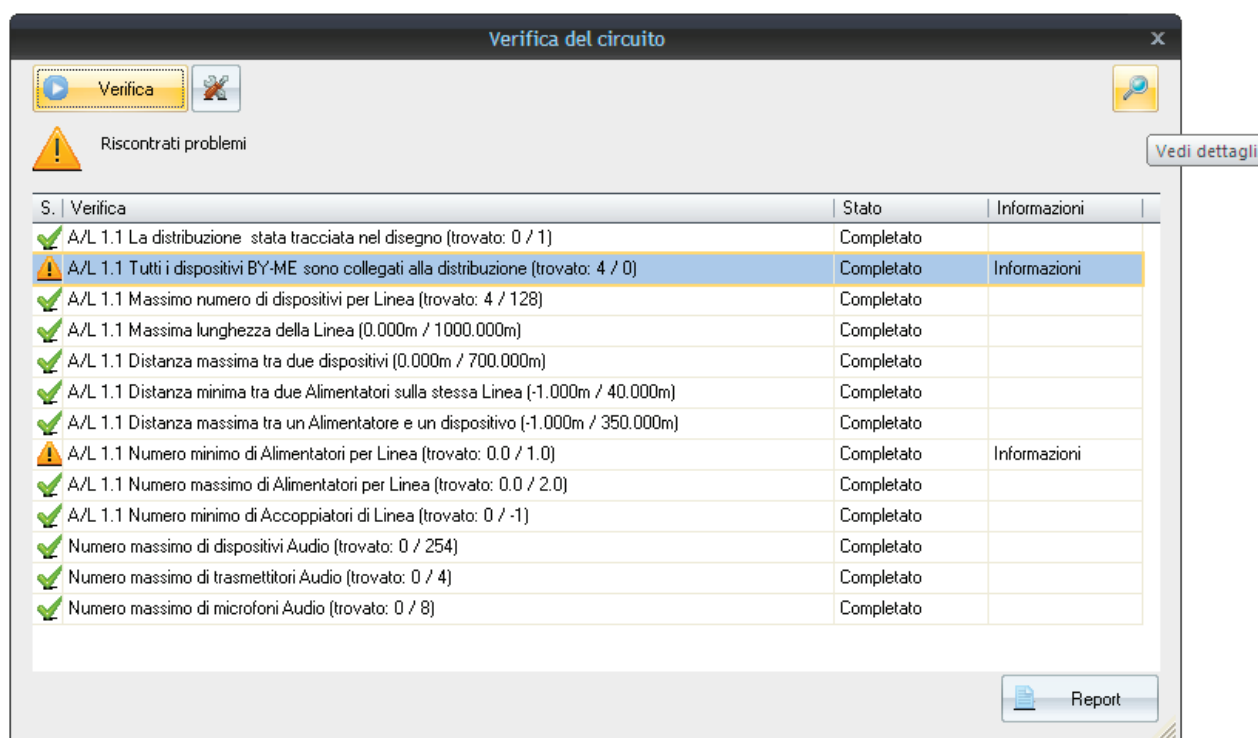
Press "Check" to start the verification of the circuit. The user can decide which checks to perform through the "Settings" interface ( ). By default, all checks are selected.
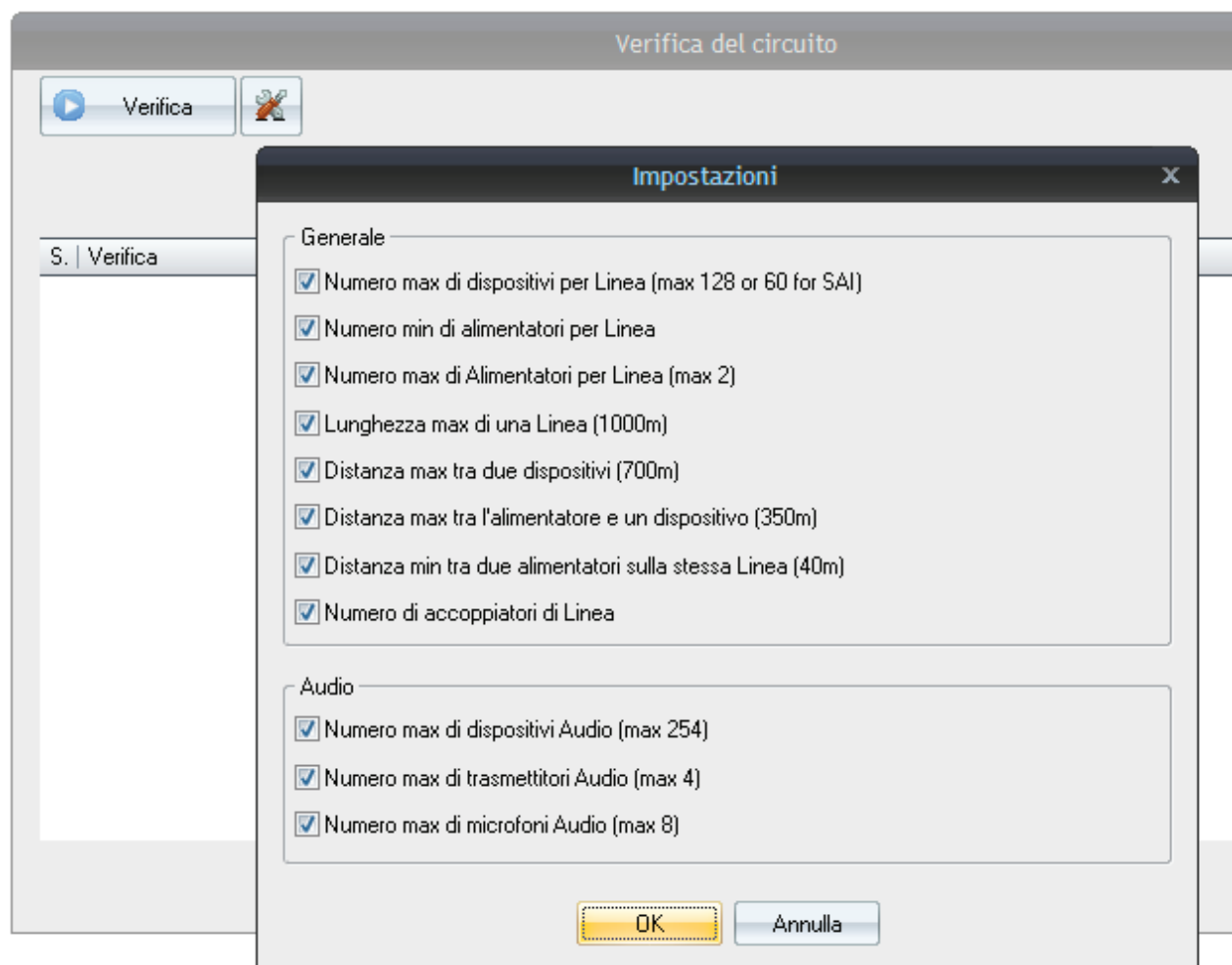
Each check can have positive or negative outcome. In case of failure, the "attention" symbol appears and the "information" string will show the cause of the error.

By clicking on the magnifying glass in the upper right corner you can see the cause of the failure.



**Connecting the devices to the Distribution**

As mentioned above, consistency checks on the distances between devices are not performed if the BUS that connects them has not been represented graphically.
A device is connected to the BUS if it is **crossed by it** or if it **ends in its vicinity**. Examples:
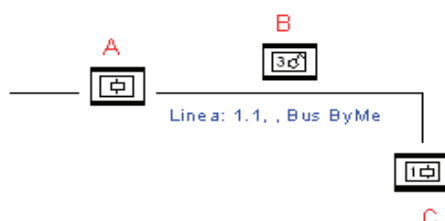


A: It is connected to the BUS because the distribution crosses it
B: It is not connected to the BUS because the distribution does not end on the device
C: It is connected to the BUS because the distribution ends on the device.

**3D Connecting the devices to the Distribution**

The interface for inserting a device allows to enter the installation height of the device. Same thing goes for the BUS line: you can specify how high to trace the BUS.
The procedure that checks the links between devices also considers whether these heights are specified.

## IMPORT/EXPORT PANEL
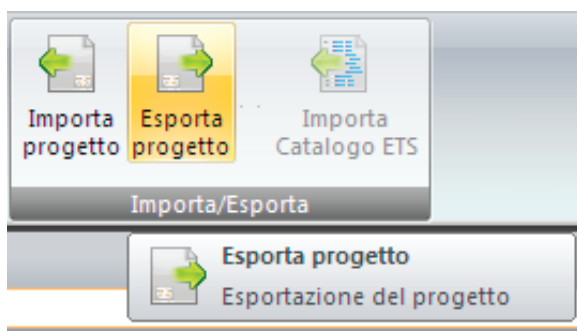


**SPOSTA and CANCELLA commands**

A KNX or BY-ME device has certain properties that can be inserted in a descriptive manner or deducted from the drawing:

   - Compartment: it can be assigned through the appropriate interface or inferred from the position of the device in a planimetry.

   - Line: it can be assigned through the appropriate interface or inferred from the position of the device near a BUS line.

The "SPOSTA" command allows, then, to move one or more devices and to update their "compartment" and "line" properties automatically without the need to set them up manually from the relevant dialog boxes. The drawing always wins: that is, if in the dialog box is a device is associated with a compartment "x" and it is then moved/added to the planimetry in a compartment "y", it will inherit the name of the compartment "y". Same thing goes for the line:  if in the dialog box a device is associated with the line "x", and then it is moved/inserted in the vicinity of a distribution section that contains the line "y", it will inherit the line "y"
The "CANCELLA" command deletes the symbol from the project.
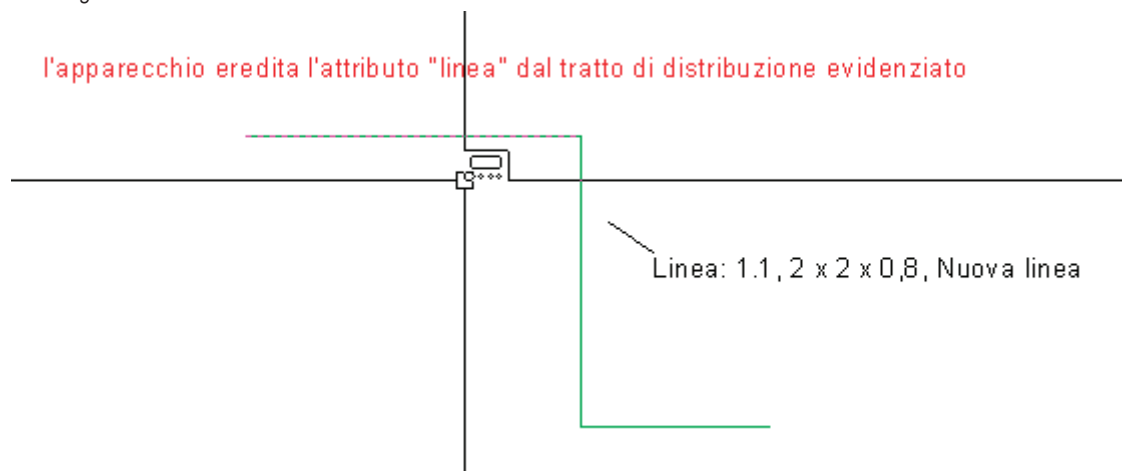
*Figure 1: Moving a device near a Line*

*Figure 2: Moving a device to an "identified" compartment*