User's Manual
1.BASIC

ACE HELLAS
INTEGRATED SOLUTIONS

CONTENTS

I. GENERAL DESCRIPTION OF THE NEW INTERFACE
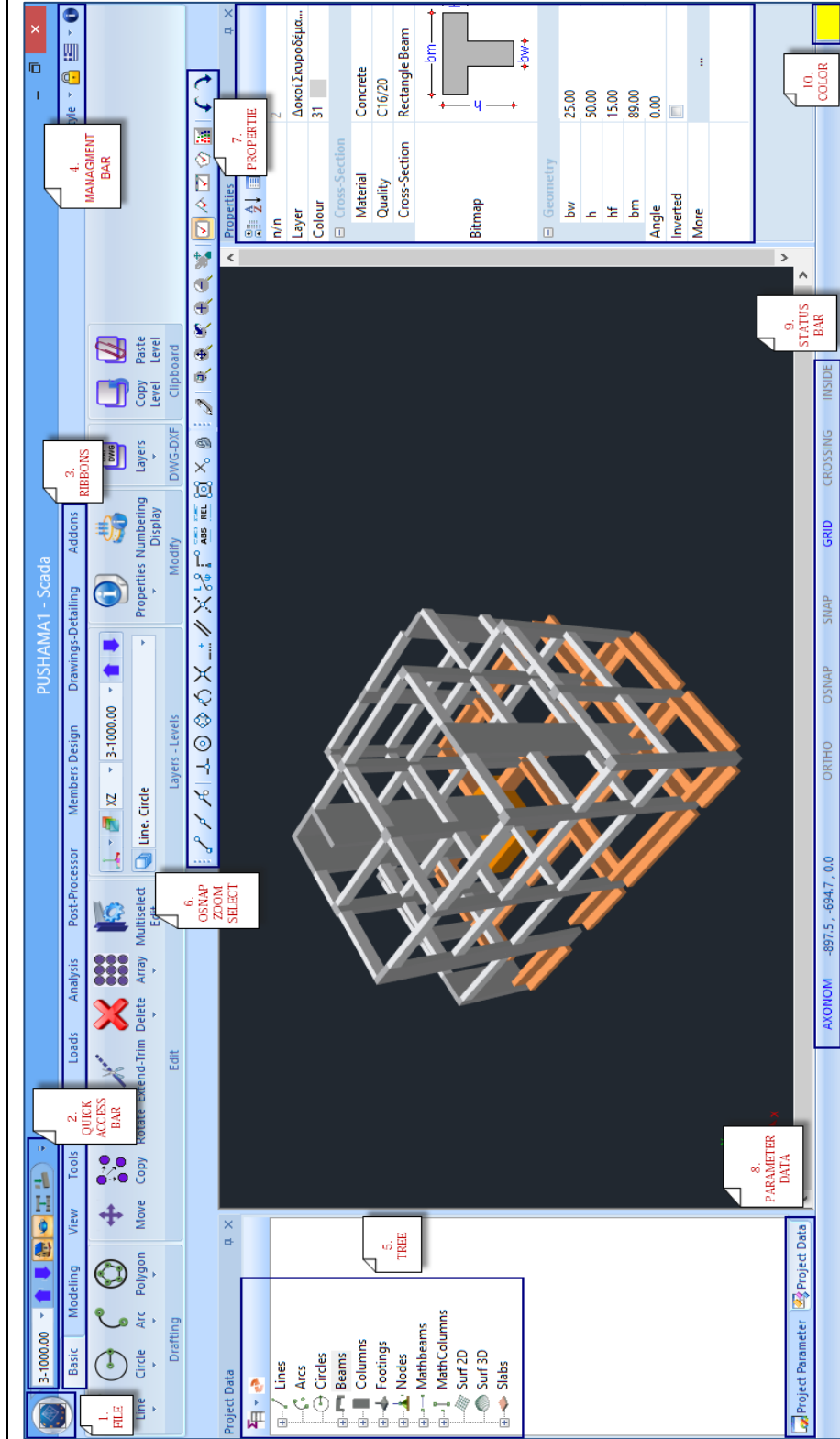
II.   DETAILED DESCRIPTION OF THE NEW INTERFACE

In the new upgraded SCADA Pro, all program commands are grouped in 12 units.

| Basic | Modeling | View | Tools | Slabs | Loads | Analysis | Post-Processor | Members Design | Drawings-Detailing | Addons | Optimization |
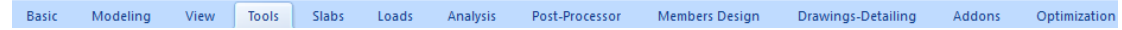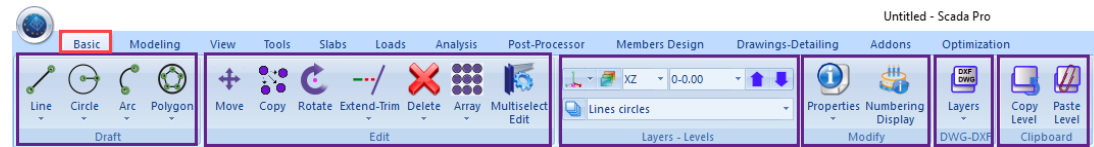
BASIC

The "Basic" UNIT includes the six following Tabs:

- **Draft**
- **Edit**
- **Layers-Levels**
- **Modify**
- **DWG-DXF**
- **Clipboard**

1.   Draft
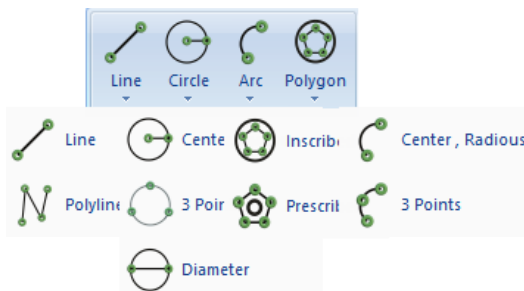
"Draft" command group contains drawing commands:
- **Line**
- **Circle**
- **Arc**
- **Polygon**

Each command includes a subgroup of commands that define the geometry and the drawing procedure.

1.1 Line

For drawing **lines**, use this command:
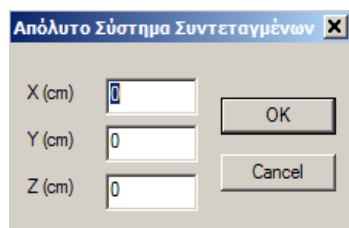
Line :
Select the command and define the start point and the end point,
. **Graphical definition**: Left click to define the first and then the second point, or
. **Definition using the osnap commands or the absolute/relative coordinates**:

Define the start point and then select the command . In the dialog box, type the absolute coordinates.

Otherwise, select command **REL** and type the relative coordinates.

Activate the checkbox ☑ Relative to point , type the relative coordinates and drag the cursor close to a given point. A little square will show the point at which you defined the relative coordinates, relative to the given point.

N Polyline: Use this command for drawing **polylines**.
Select the command and follow the previous procedure (See "Line" command.").

1.2 Circle

Use this command for drawing **circles**.

Center, Radius:

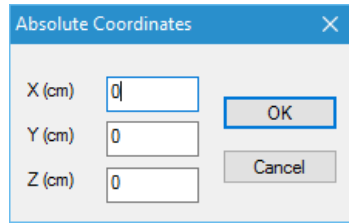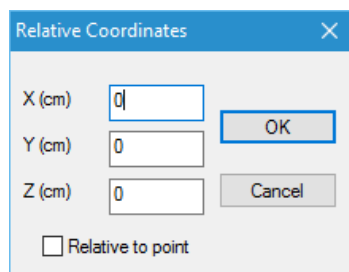- **Graphical definition**: Define the center of the circle and then define the radius of the circle.

Definition using the osnap commands or the absolute or relative coordinates: Define the center of the circle or the end point of the circle radius by hovering the cursor over a point and then selecting **ABS** or/.

Three Points:

- **Graphical definition**: Define three consecutive points on the perimeter of the circle.

Definition using the osnap commands or the absolute or relative coordinates: Define the first point of the perimeter of the circle and then select **ABS** or **REL** to define the other two points.

Diameter:

- **Graphical definition**: Define two points referring to the two end points of the circle diameter.

- **Definition using the absolute or relative coordinates**: Define the first point and then select [ABS] or [REL] to define the end point of the circle diameter.

1.3 Arc

Use this command for drawing **arcs**.
The definition of the arcs can be done through the *Center, Radius,* and the *Three Points* option, similarly to the definition of the circles.

1.4 Polygon

Use this command for drawing **Inscribed or prescribed polygons**.

Inscribed polygon:

Type in the dialog box the number of sides of the inscribed polygon. In the following, define graphically or numerically the radius of the prescribed circle.
ATTENTION!! The number of the sides must be at least 3
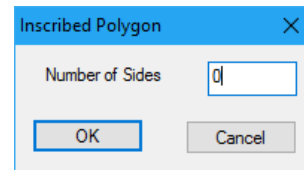
Prescribed polygon:
Similar to the inscribed polygon procedure.

2. Edit

"Edit" command group contains editing commands:

- **Move**
- **Copy**
- **Rotate**
- **Extend-Trim**
- **Delete**
- **Array/Offset/Clone/Creation/Move with Attachments/Match Properties**
- **Multi Select Edit**

2.1 Move

Move This command is used for moving one or more objects from their present location.

Firstly, select the command and then the objects to be transferred. The selection can be done individually, by a rectangular window etc. To end the selection process, press the right mouse button and then pick the reference point (line edge, column vertices etc).

2.2 Copy

Copy This command is used for making multiple copies of the selected physical or drawing objects.

ATTENTION:
Use Copy command to copy physical objects without a mathematical model. To copy objects with mathematical model use Clone Creation command.
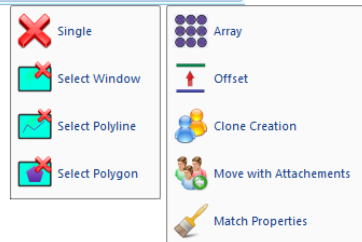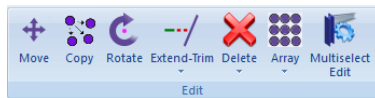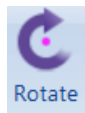
First, select the command and then the objects you want to copy. The selection can be done individually, by a rectangular window etc. To end the selection process, press the right mouse button and then pick the reference point (line edge, column vertices etc.). Finally, select the point where you will copy the objects.

2.3 Rotate

Rotate This command is used for rotating an object on the drawing.

In the window that appears select one of the options from the drop-down list:
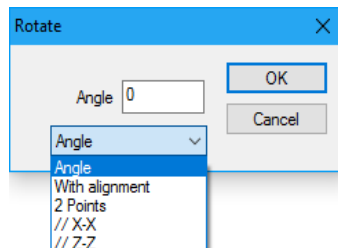
- Angle: Type the value of the rotation angle in degrees (counterclockwise is positive) and then select the objects that are going to be rotated. Finally, select the characteristic point that will be the center of the rotation.
- With alignment: Select the objects that are going to be rotated, then select the rotation point and finally the line for the alignment.
- 2 Points: Select the objects that are going to be rotated then select the rotation point and finally the two points. The rotation is performed similarly to the alignment option.
- //X-X: With this option, the objects are rotated around the X-X axis. Type the value of the rotation angle in degrees and then select the objects that are going to be rotated. Finally, select the characteristic point that will be the center of the rotation.
- //Z-Z: With this option, the objects are rotated around the Z-Z axis. Type the value of the rotation angle in degrees and then select that are going to be rotated. Finally, select the characteristic point that will be the center of the rotation.

2.4 Extend-Trim

The Extend-Trim command is used as follows:
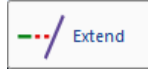
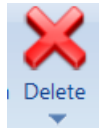First, select the Extend command and then the line acting as boundary up to which the extention will be performed. Finally, select the object (beam, line etc.) to be extended. Continue with the next object or exit the command with the right mouse button.

Similar to the Extend procedure.

2.5 Delete

This command is used for erasing physical and/or mathematical entities from the model. First, select the command and then the objects to be erased. The erase can be performed individually, by a rectangular window, by a polygon or by a polyline
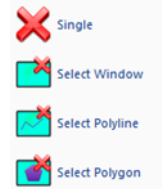
⚠ Note that the mathematical model and the entities of a specific layer can be erased from the Edit Layer window.

⚠ It is also possible to erase an object if its ID number is known: Firstly, select the command and then use the "Select Group-Number" command to select the desirable object.

2.6 Array

This command is used for creating multiple copies of selected objects in a rectangular, circular or linear pattern.

In the window that appears, first select the type of the array (rectangular, circular or linear). Then, based on the type of the array, define the number of the objects in each direction, the distance and the angle. Finally, select the element to be copied.
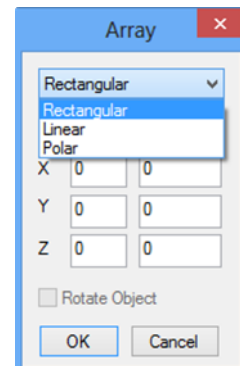
💡 **EXAMPLE 1**

Rectangular Array

Select the "Rectangular Array" command and type the total number of objects on X and Z direction (including the original one) and the two corresponding distances. Select the object to be copied and exit the command with the right mouse button.

The objects are placed in the grid as shown in the following picture.
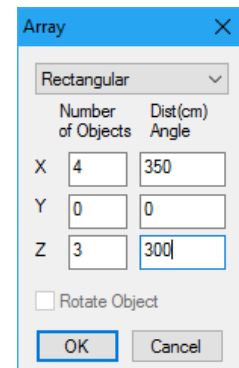
☞ To place objects along the negative X or Z axis type negative values of distance.
☞ The Linear array is a subcase of the rectangular array that makes copies only in the X direction

💡 **EXAMPLE 2**

Polar Array

Select the "Polar Array" command and type the total number of objects (including the original one) and the angle of the arc of the polar array. Activate the "Rotate Object" check box to rotate the objects as they are copied.
Select the object to be copied and two characteristic points that define the radius and the direction in which the center point of the array is created.
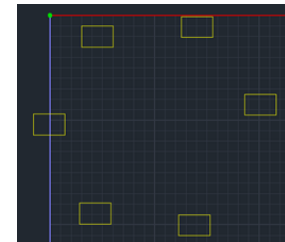
⚠ **NOTE:** The Array command also works with mathematical members.

2.7 Offset

This command is used for creating a line parallel to another line at a specified distance.

Select the command and in the dialog box type a value of the offset distance from the original object in cm. Select the object to offset and click again to indicate the offset direction. Repeat the command or right click to exit.

2.8 Clone creation

This command is used for creating one or more **exact copies** of physical and mathematical entities.

First, select the command and then the objects to be cloned. The selection can be done individually, by a rectangular window, by a polygon etc. To end the s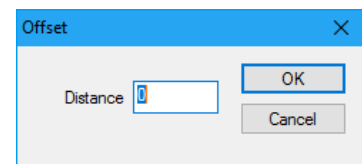election process, press the right mouse button and then pick the reference point (line edge, column vertices etc.). Finally, select the point where the objects will be cloned.

⚠ *This command differs from the "Copy" command as it can be used for mathematical elements. It creates "clones", which are elements with the identical geometrical and the inertial characteristics.*

2.9 Move with Attachments

This command is used for **moving in the parallel direction** one or more objects, simultaneously. First, select the command and then the objects to be moved by using the "window or polygon" option. The objects inside the window are moved, while the objects that are intersected by the window are stretched. During moving and stretching, the mathematical model of the selected elements is also included.

Select the command and use relative REL or absolute ABS coordinates to define the transfer.

💡 **EXAMPLE**

Select the command and use the window option to select the nodes of the 1st column.
Note that the window contains the nodes that are going to be moved and intersects the elements that will be stretched, i.e., only one of the two end nodes will be moved.

Select the relative coordinates and define the relative distance, on X direction and you will receive the configuration on the right.
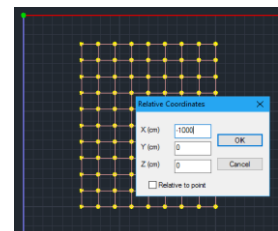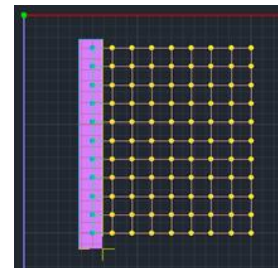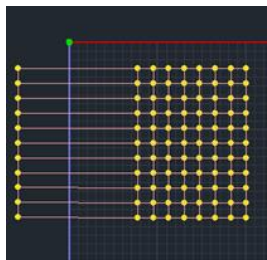
Notice the new position of the nodes and the elongation of the elements that haven't lose their connection with the nodes.

2.10 Match Properties

This command is used for assigning the properties of the selected object to other similar objects.

Select the command and then select the object the properties of which you would like to assign to others. In the dialog box that appears, select the properties you want to assign and finally, select (using any selection tools) similar objects to which you are going to assign the selected properties.
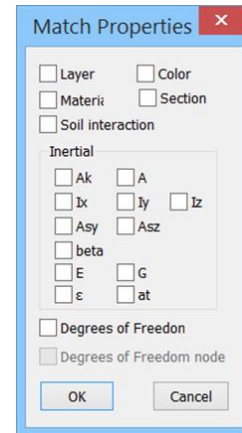
2.11 Multiselect Edit

Multiselect Edit This command is used to **manage and modify the properties** of an object or a group of objects.

Select the command and then select the objects. By pressing the right mouse button the following dialog box is displayed:

The dialog box contains ten different Tabs, each one including the properties of the selected object or objects. The parameters can be modified by typing a new value in the corresponding field.
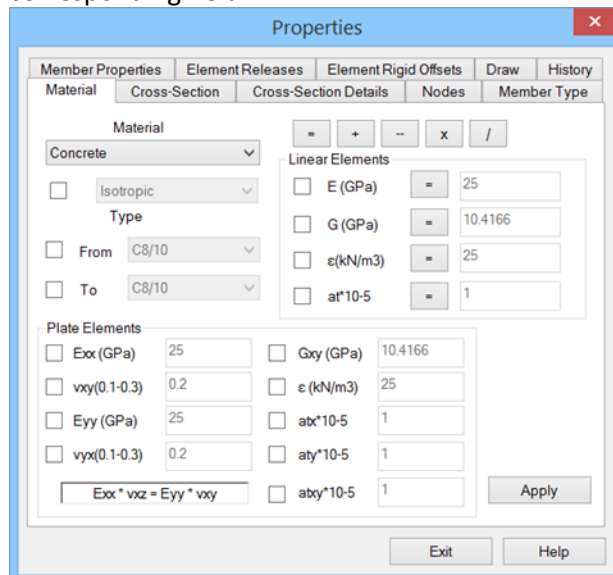
2.11.1 Material Tab

In this Tab you can modify the following parameters:

- The type of material, selecting from the list of materials.

- The "orthotropic or isotropic" property, (it is worth mentioning that an "orthotropic" material has different properties in different directions. By choosing orthotropic the parameters must be defined according to the relation : $E_{xx} * v_{yx} = E_{yy} * v_{xy}$).

- The material grade. You can select the new material grade with or without prior knowledge of the original grade
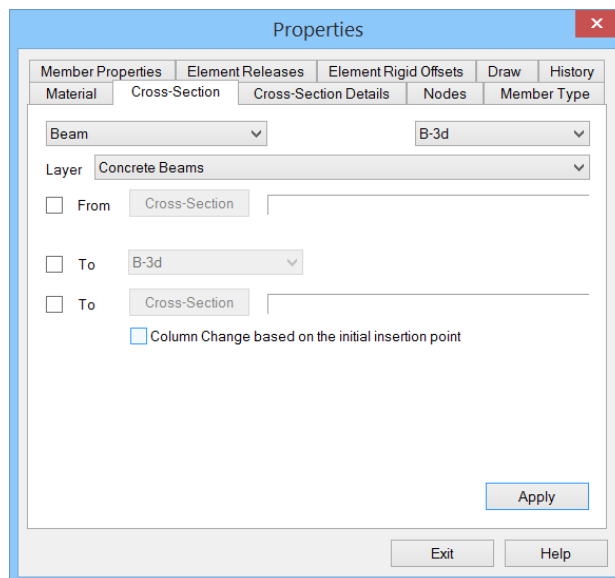
- The physical properties of the linear (beam/truss) or surface elements. To modify a value, activate the relative property, select an operation and type the new value.

For example, if you want to multiply the elastic module E by 2, select [x] and type 2. Select [Apply] to save the changes.

2.11.2 Cross-Section Tab

In this Tab you can modify the following parameters:

- The structural element category `Beam ∨` .

- The linear element type `B-3d ∨` .

- The layer of the element   Layer `Concrete Columns ∨`

  ☑ From `Cross-Section` [ ]

  ☐ To   `B-3d ∨`

- The Cross-Section ☑ To `Cross-Section` [ ]

  .

💡 **EXAMPLE**

Change the cross-section of rectangular beams from 25/60 to 35/70 in a certain level.

The presence of the mathematical model is not a prerequisite for applying this change.

Select the "Multi Select Edit" command and by using the "window selection" option, select all the elements of the level. The dialog box is displayed by pressing the right mouse button. Since beams of different cross-section, as well as columns, walls etc., may have been selected, a filter is required.

Select "Beam", "B-3d", (if you have not already generated the mathematical model, the selection of the type of the linear elements is not necessary) and "Concrete Beams" layer.

Activate the checkbox "From" and press the "Cross Section" button. In the beam cross-section dialog box, type the original dimensions (25/60) and then press "OK".

Activate the checkbox "To", press the "Cross Section" button and type the new dimensions (35/70).
Press the "Apply" button and "Exit" button, to apply the modification.
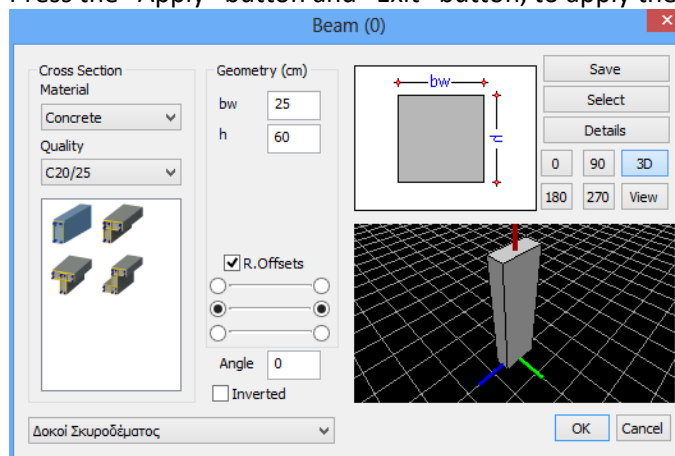
In case you want to change the dimensions of all the beams of the level, you can follow the same procedure but without activating the checkbox "From".
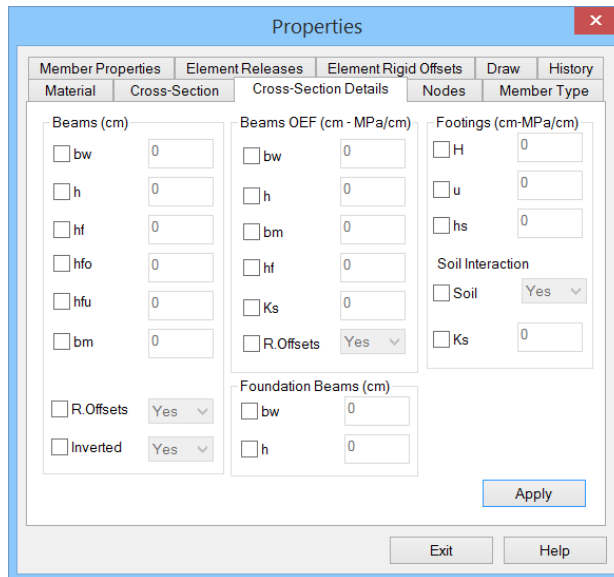
You can also change only the type of the linear member (when you have the mathematical model). Activate the checkbox ☑ To [B-3d ▾] and select the new type from the list.

2.11.3 Cross-Section Tab

In this Tab, you can modify the geometric characteristics of:
- Selected beams.
- Selected beams OEF.
- Selected footings.

Selected foundation beams.

💡 **EXAMPLE**

Select the "Multi Select Edit" command and select all the desired beams. In the displaying dialog box, to change the width b_w of all the selected beams, activate the field b_w and type the new value of 30 cm in the following field ☑ bw [30].

In case the mathematical model exists, you can also activate or deactivate the rigid offsets.

2.11.4 Nodes Tab

In this Tab, you can modify the degrees-of-freedom of the selected nodes:
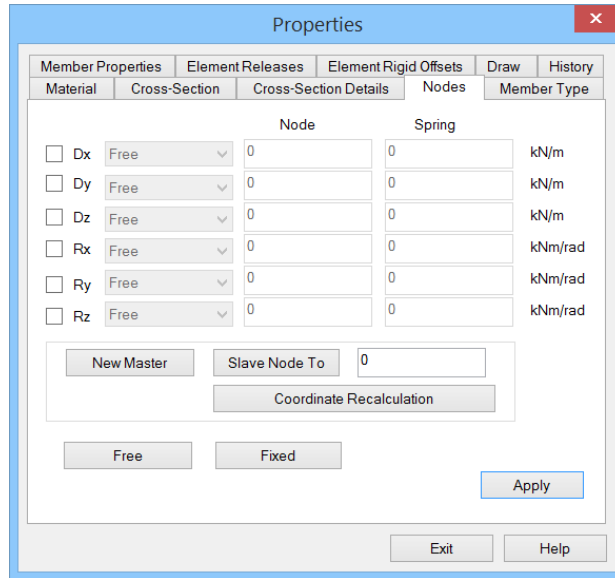
Activate the degrees-of-freedom and select the restraint/constraint status:

- <u>Free</u>: Translation/rotation is allowed

- <u>Fixed</u>: Translation/rotation is not allowed

- <u>Slave</u>: The translation/rotation of the node depends on the translation/rotation of another node (Master Node). The Master node ID must be filled in the respective field and the constraint is automatically activated. It must be noted that different degrees-of-freedom may be constrained in different nodes.

- <u>Spring</u>: The translational/rotational stiffness is assigned to the node.

 To fully constrain one node to another one. The ID of the second one must be typed in the respective field.

 To constrain the selected nodes at a new Master Node which is created at the geometrical center of the selected nodes.

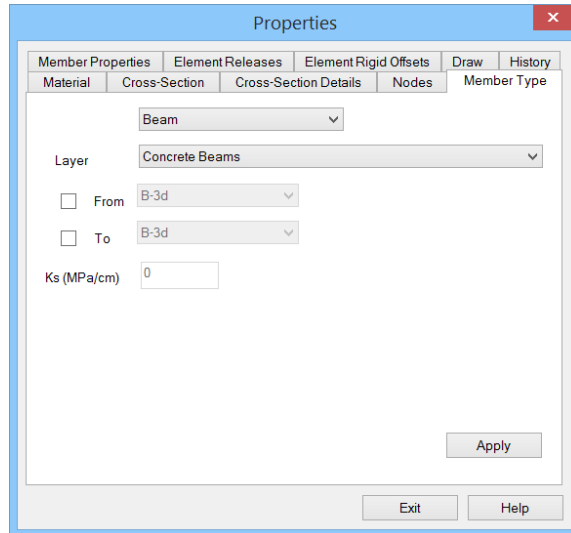 To calculate automatically the new coordinates of the Master Node.

 To automatically set free or set fixed the degrees-of-freedom of the selected nodes.

Click on the "Apply" button to save the changes and then press "Exit".

2.11.5 Member Type Tab

In this Tab, you can modify the type of element.

Select the type of the element and its layer from the drop-down list. Activate the checkbox "From" to filter the selected elements, or activate directly the checkbox "To" to change the type of all the elements of the selected layer.

For the foundation, elements selected from the drop-down list the label "on Elastic Foundation (oef)" and type a value (MPa/cm) to activate the soil spring constant Ks.

2.11.6 Member Properties Tab

In this Tab, you can modify the physical properties of the selected members of the same type or the members with a specific cross-section by activating the respective filter.

Select a new cross-section activating the checkbox ☑ To [Cross-Section] and the program automatically fills the properties. You can modify or set the values manually, by activating the corresponding checkbox, selecting an operation (=) and typing the new value.

For example, if you want to multiply the area by a factor of 2 select [x] and type 2.
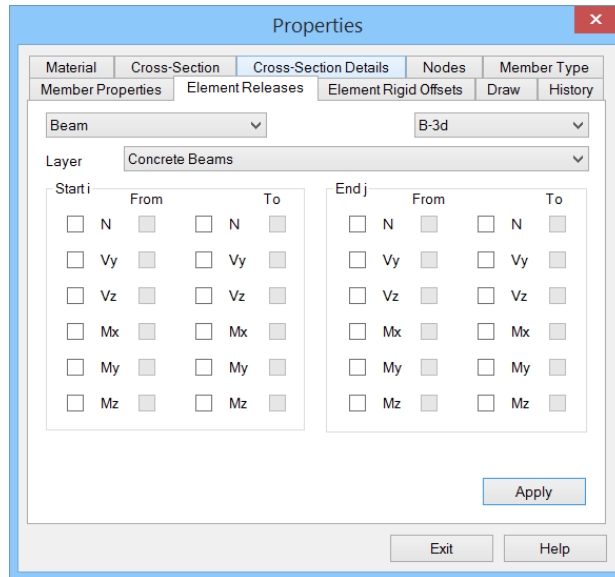
Click the "Apply" button to save the changes.
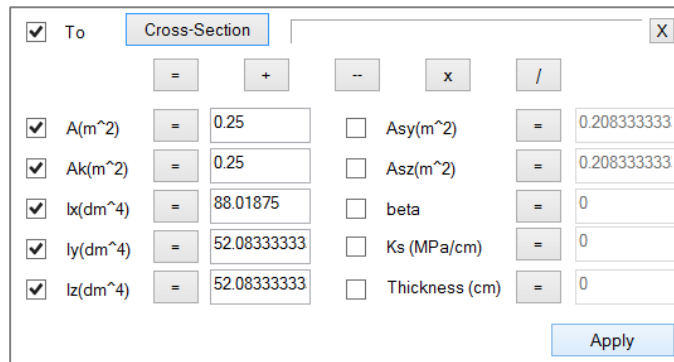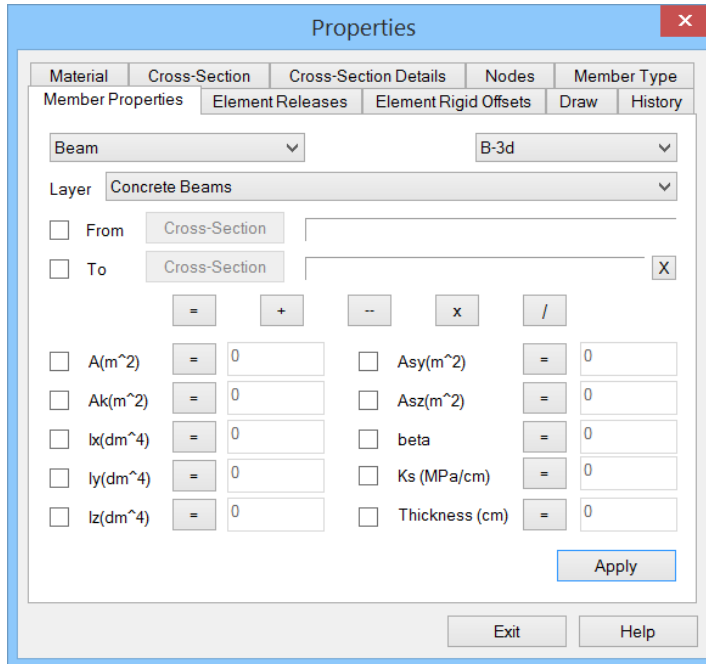
2.11.7 Element Releases Tab

In this Tab you can modify can the start and end releases of all selected elements with the same type.

To release an internal force at the start or end point, activate the check box ( ) and then activate the "To" field. Please note that the symbols are:

 = Fixed

 = Free

☞ This command is useful for making changes in a group of elements and not individually. For making changes in an element individually, it is preferable to select the element and edit it directly into the "Properties" Tab

The user may also use the "From" field as a filter but in that case, the original values of dx /dy /dz of the rigid offset must be typed manually.

☐ dx

☐ dy

To define a new value, activate the corresponding checkbox ☐ dz , select an operation

| = | + | -- | x | / |

and then type the value.

💡 EXAMPLE

Set the "Start" rigid offsets of all the beams on X direction (dx) to zero in a specific level.

Select the "Multi Select Edit" command and select all the elements of the specific level through the "window select" option. The dialog box is displayed by pressing the right mouse button.

From the drop-down list select Beam as the element type

For the Start, point activates the "To" and "dx" fields, select the operation "=" and type 0.
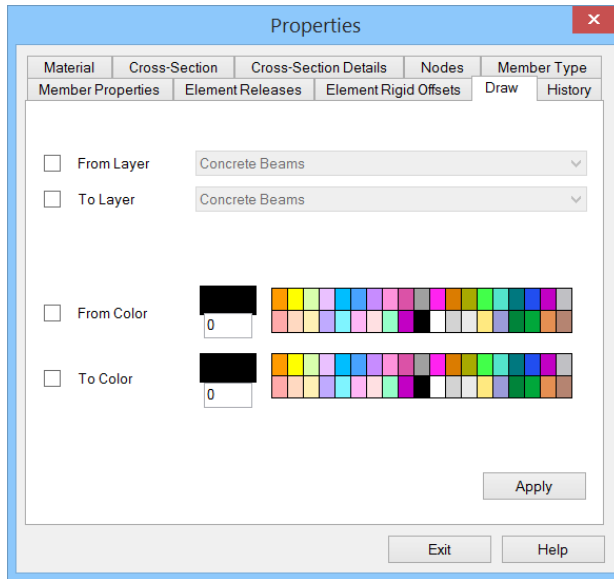
To

☑ dx [=] [0]

Press "Apply" and "Exit" to finish the command.

2.11.8 Draw Tab

In this Tab, you can modify the layer and the color of the selected elements.

Activating "From Layer" works as a filter. It is optional and requires to select the layer that belongs to the elements that will change the layer

Activate the "To Layer" to change the layer of the selected or "filtered" elements.
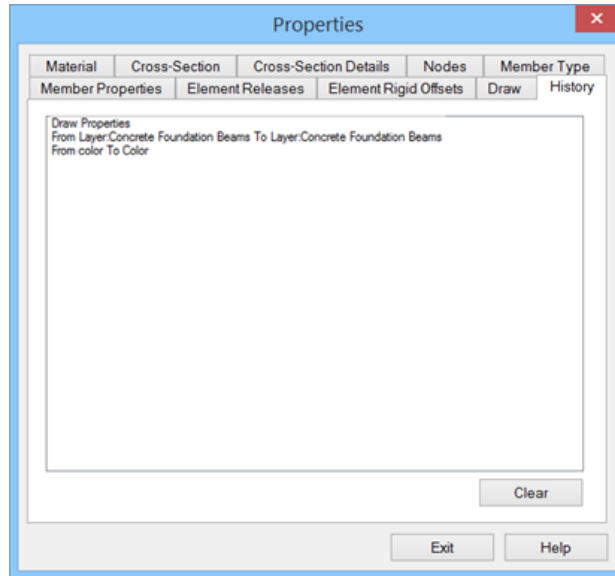
☑ To Layer   Concrete Beams ▼

Use the same procedure to change the color.

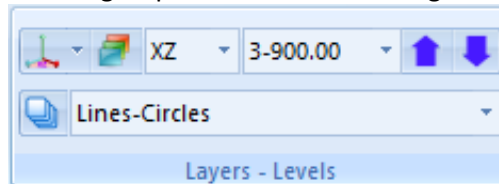☑ To Color   [■] [color palette]
                 [0]

2.11.9 History Tab

In this Tab all, the changes on the selected elements are displayed as a log. Select "Clear" to clear the "History".
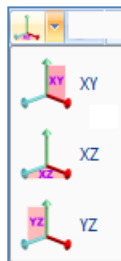
3. Layers - Levels

The "Layers - Levels " command group contains the following commands:

- **Create layers and levels**
- **Edit layers and levels**
- **Move layers and levels**
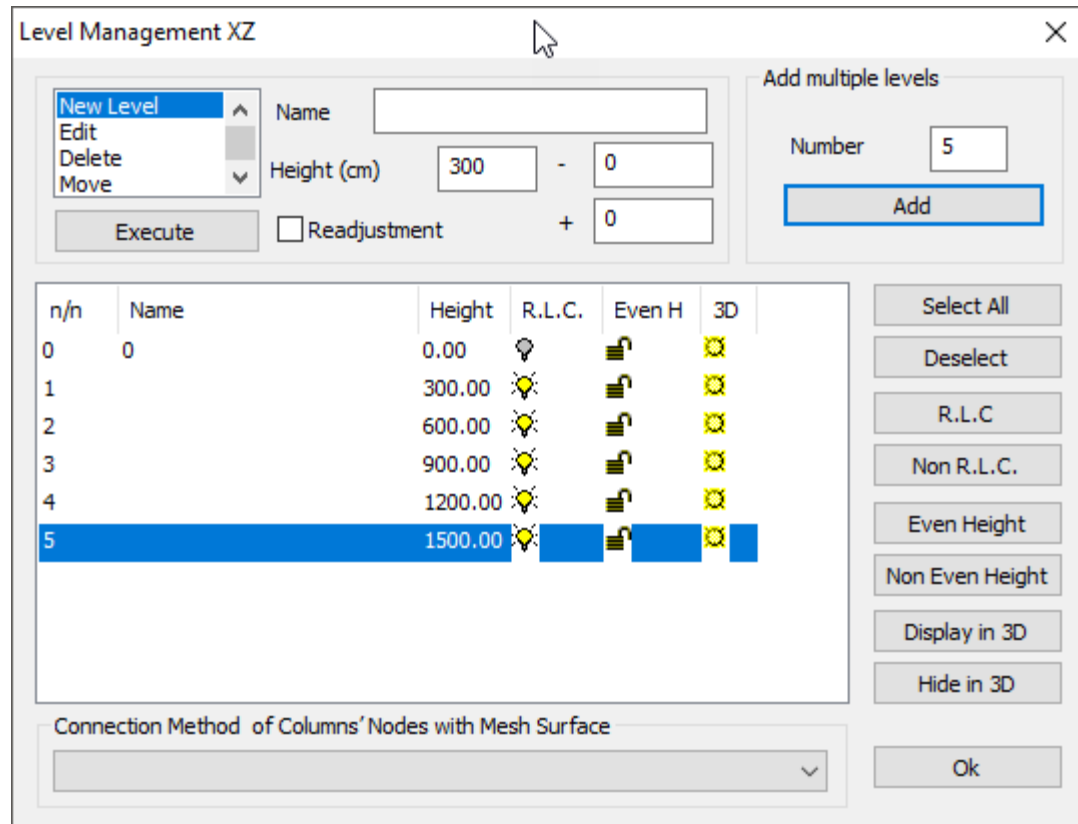- **Transition between levels**

⚠ By hovering the cursor over a command, a tooltip appears that shows the name of the command and a short description.
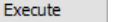

This command is used for **creating** and **editing** the levels of the structure.
By default, the XZ levels edit is selected.
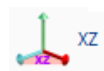
In the dialog box the edit of XZ levels (plan views) appears:
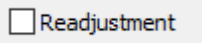
- To **modify an existing level**, select "Edit" and then select the level from the list (the name of the layer becomes blue). Type a new name, or set a different height and click Execute.

- To **create a new level** select "New Level" and then type a name and the height. Fill in the "−" and "+" range in case of minor height variation, the existence of sloped elements or vertical surface elements, to include them (and their masses) in the specific level and also display them.

💡 **EXAMPLE:**

The 2nd level is located at 700 cm height but there are elements at 600 cm height.
During the creation/edit of the 2nd level, the user types 150 (cm) in the "−" field. In this way, when you activate the 2nd level, all entities between 550cm and 700cm will be displayed.

Activating the ☐ Readjustment check box, allows the altitude of existing levels to be readjusted to a possible altitude change of an intermediate level.

💡 **EXAMPLE:**

Let's say three levels are created at altitudes of 0,300,600,900 and you want to change the height of the 1st level without changing the individual heights of the floors. You select the level, the Edit command, and the Readjustment command. Change the Altitude value and select Update. The result that arose is the one below:
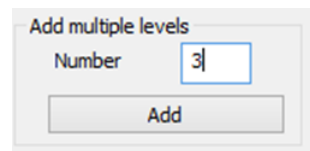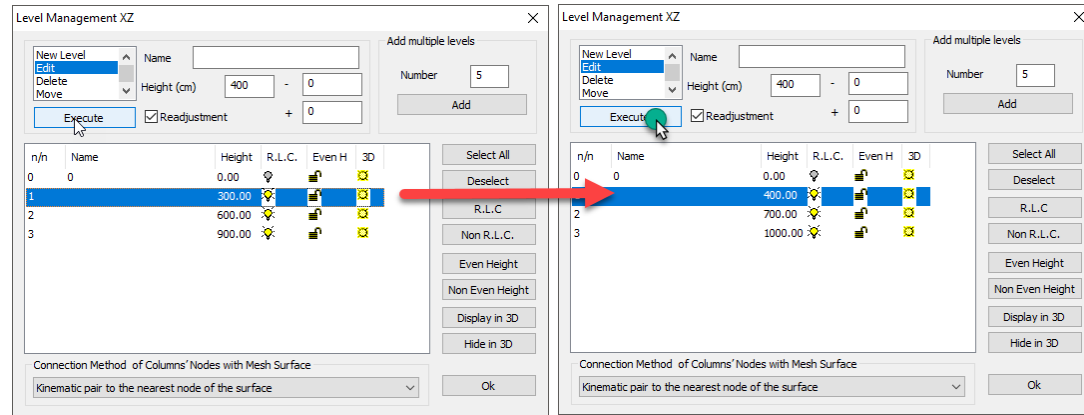
To **create simultaneously multiple levels** type the height of the typical floor and in the field "Add multiple levels" type the number of the levels. By pressing the [Add] button, all the levels of the structure are created automatically.

To **delete a level and its elements** select "Delete" option from the drop-down list, activate the ☐ Readjustment check box, click on the level and then press the "Execute" button.

To **move a level in the next position/row** select "**Move**" in the drop-down list, click on the level and then press the "Execute" button. The level is moved one position below. This command is useful when you want to interpose a new level between two existing levels.

To **delete a level but not its elements** select the last "**Delete**" option from the drop-down list, click on the level and then press the "Execute" button.

After the creation, the levels are displayed in a list containing the ID number, the name, the height, the Rigid Link Constrain ("R.L.C.") option, the height variation ("Even Height") and the 3D visibility.
The following symbols show whether the parameters are active or inactive:

💡 🔓 ❄ Active

💡 🔒 ❄ Inactive

To modify the last three parameters, you can use the commands on the right.

First, select the level or the levels (ctrl and left click or "Select All") and the following command:

- **"R.L.C."** or **"Non R.L.C."**

- **"Even Height"** to lock the height 🔒 . That means that all new elements will be created in this height regardless of the actual height.

- **"Display in 3D"** to display the levels on the 3D view.

- **"Hide in 3D"** to hide the level on the 3D view.

NOTE
⚠ It is not possible to enter levels with the same altitude

**Connection Method of Column's Nodes with Mesh Surface**

SCADA Pro allows the combination of linear and surface elements in the same project.

On the bottom part of the window, the user may select the connection method between the nodes of the columns and the surface mesh. Four methods are available using kinematic constraints or connecting elements.
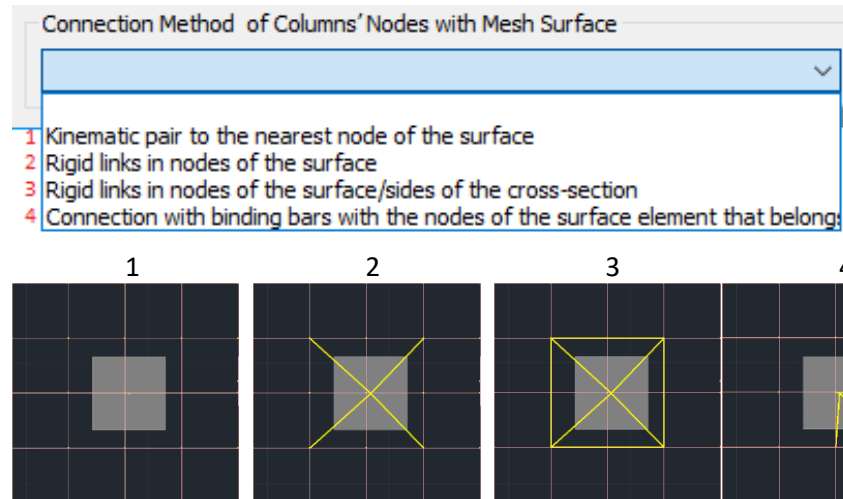

Select the level, then the type of connection and press Execute.

In the dialog box the edit of the XY levels appears:
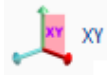
- In the field **"From"** type the Z coordinate of the first level.

- In the field **"To"** type the Z coordinate of the last level.

- In the field **"Step"** type the relative distance between the levels along the Z axis.

- To display the elements located within a range from the level's position fill the **"-"** and **"+"** fields.
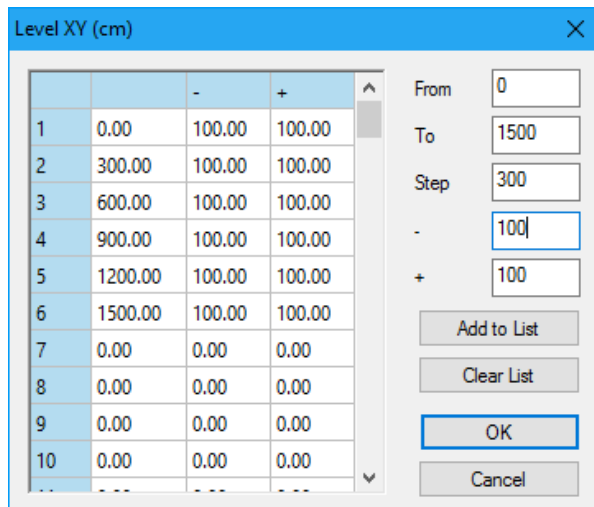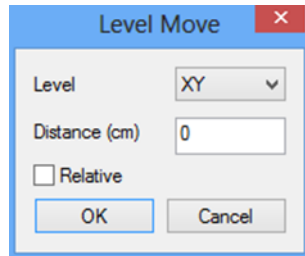
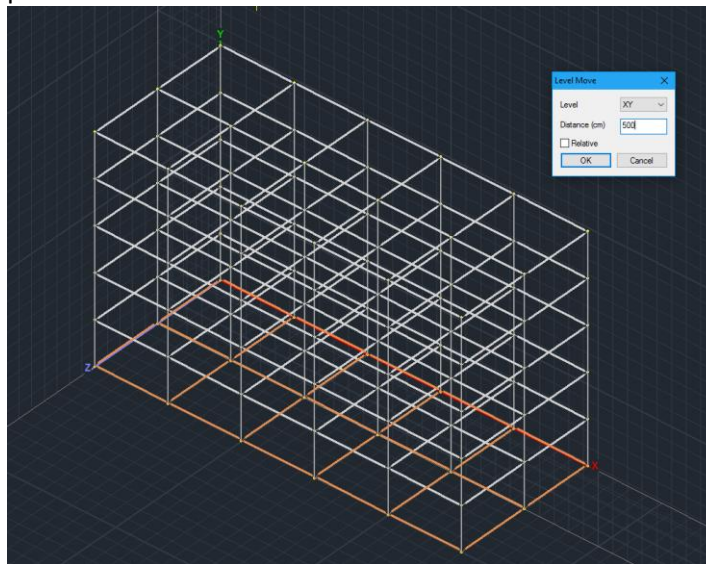Edit the YZ levels similarly to the XY levels.

 This command is used for **moving the planes** XY, XZ, YZ.

Select the command and the following dialog box is displayed:

For the level under consideration, activate the checkbox "Relative" and type the distance that will be moved.

In the example of the image below, the XY level is transferred 500 cm away from its original position.
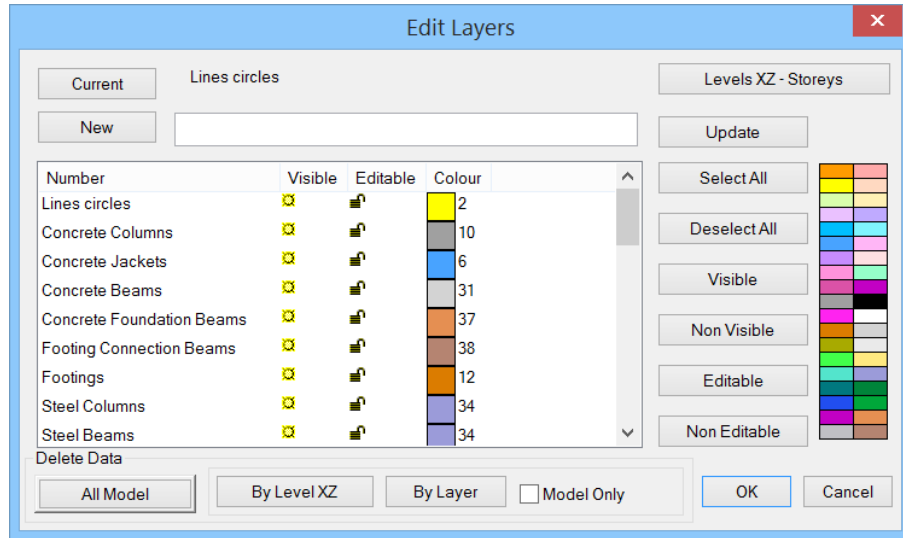

⚠ Changing the view from 3D to 2D and back restores the levels to their original position.

 This command is used for **editing** the existing layers and for **creating** new ones.
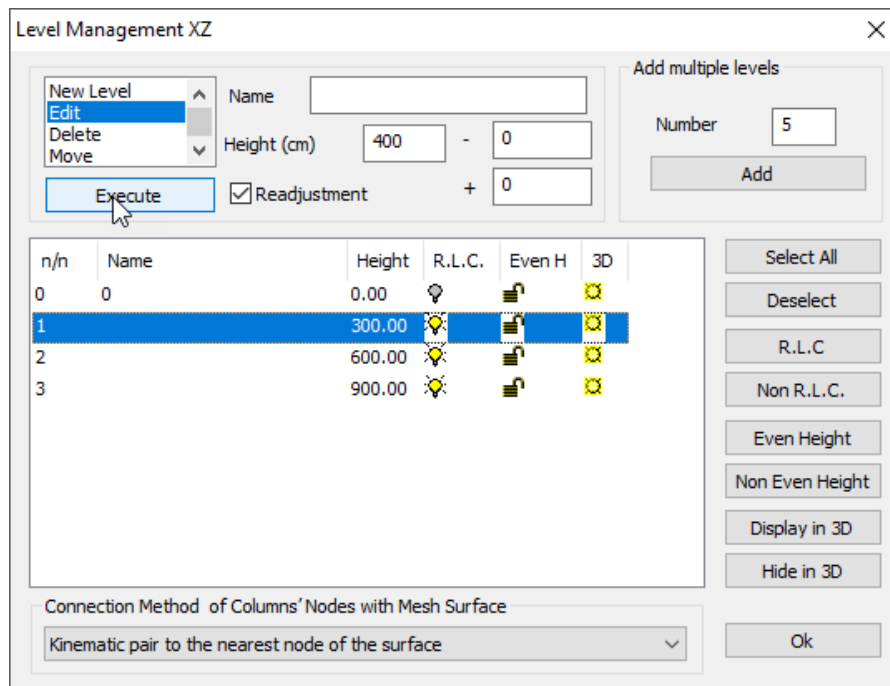
By clicking on the icon the following dialog box appears:

- <u>Current</u>: This command is used for setting the current layer. Select a Layer from the list and click "Current".
- <u>New</u>: This command is used for creating a new layer. Type a description and click "New".
- <u>Levels XZ - Storeys</u>: This command is used for opening the XZ Level Management dialog box. (see command)

Level Management XZ

- <u>Update</u>: This command is used for saving an existing layer under a different name. Select a layer from the list, then type the new name and click "Update".

- Select All / Deselect All: These commands are used for making group changes in layers. Additional layers can be selected by using left click and "Ctrl".
- Visible / Non Visible : These commands are used for displaying or hiding the elements of the corresponding layer.
- Editable / Non Editable : These commands are used for allowing or not the modification of the elements of the corresponding layer.
- Delete Data: in this Tab, the user may erase the mathematical model of the project or only a portion of it.
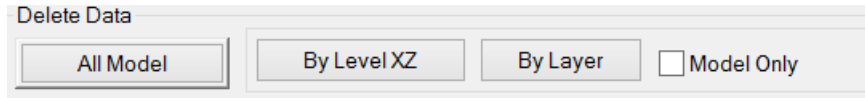
 : This command is used for erasing the mathematical model of the project.

 + : Erase the elements belonging to the selected layers from the current XZ level.
Firstly, select one or more layers, then deactivate "Model Only" checkbox and finally click "By Level XZ".

 + Erase the mathematical model of the elements which belong to the selected layers from the current XZ level.
Firstly, select one or more layers, then activate "Model Only" checkbox and finally click "By Level XZ".

 + Erase all the elements, which belong to the selected layers.
Firstly, select one or more layers, then deactivate "Model Only" checkbox and finally click "By Level XZ".

 + : Erase the mathematical model of all the elements which belong to the selected layers.
Firstly, select one or more layers, then activate "Model Only" checkbox and finally click "By Level XZ".
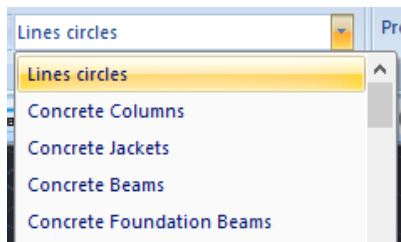
To open the layers' drop-down list, press the left mouse button. Note that the active layer is always highlighted.

4. Modify

" Modify " command group contains commands for displaying and modifying the properties of the physical or mathematical model:
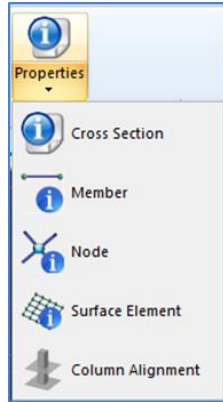
- **Cross section**
- **Member**
- **Node**
- **Surface element**
- **Column Alignment**

4.1 Properties

From the drop-down list select the command (Cross Section, Member, Node or Surface Element) and then click on the element. On the right of the screen, the "Properties" window will appear with the properties of the selected element.

⚠ Alternatively, you can display the properties by selecting the element pressing the left mouse button. Remember to exit any previous command by pressing the right mouse button.

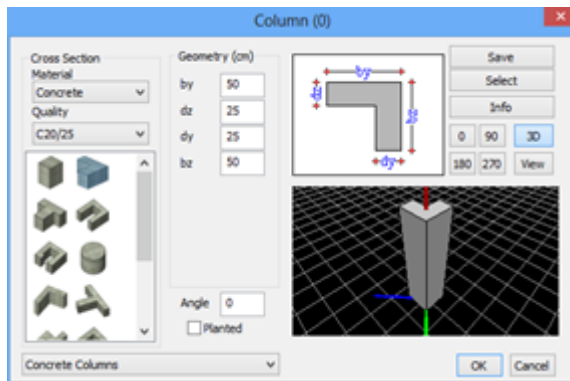| Properties		Properties		Properties		Properties	
n/n	4	n/n	5	n/n	100	n/n	45
Layer	Cmcrete Beam	Layer	Cmcrete Colur	Layer	Mathbeams	Layer	MathColumns
Colour	15	Colour	10	Colour	15	Colour	25
⊟ Cross-Section		⊟ Cross-Section		Type	B-3d	⊟ Coordinates	2509.81, 800.00,...
Material	Concrete	Material	Concrete	⊞ Cross-Section		X	2509.81
Quality	C20/25	Quality	C20/25			Y	800.00
Cross-Section	Rectangle Beam	Cross-Section	Rectangle secti...			Z	735.39
				Bitmap		⊟ Degrees of Freedom	
Bitmap		Bitmap				Dx	Free
						Dy	Free
				⊟ Nodes		Dz	Free
				First Node	44	Rx	Free
				Final Node	45	Ry	Free
⊟ Geometry		⊟ Geometry		⊟ "Cross Section"		Rz	Free
bw	30.00	by	200.00	Material	Concrete	Master Node	...
h	60.00	bz	30.00	Quality	C20/25	Free Node	...
Angle	0.00	Angle	0.00	⊟ Rigid Offsets (cm)		Fixed Node	...
Inverted	☐	Planted	☐	dx (Start)	-25.00	Slave Node	...
More	...	More	...	dx (End)	100.00	More	...
				dy (Start)	0.00		
				dy (End)	0.00		
				dz (Start)	10.00		
Layer		**Layer**		**Layer**		**Layer**	
Layer Definition		Layer Definition		Layer Definition		Layer Definition	

4.1.1 Cross Section

Select the command and then select the element graphically in 2D or 3D display or from the "Tree" list. The selected element is colored red to be identified easily.
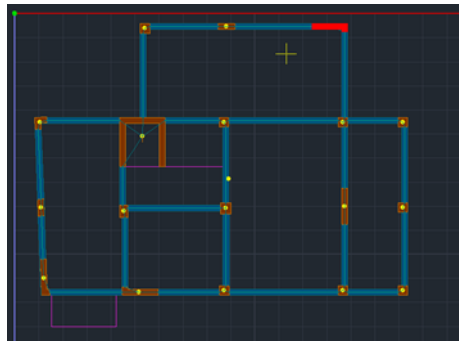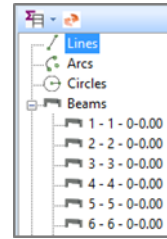
Depending on the type of the selected element, the "Properties" list displays the physical and the geometric characteristics of the section.

The changes may be applied directly into the "vertical bar" or into the "More" option that opens the dialog box with the characteristics of the section.
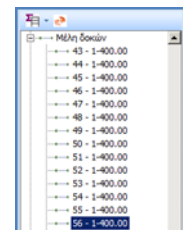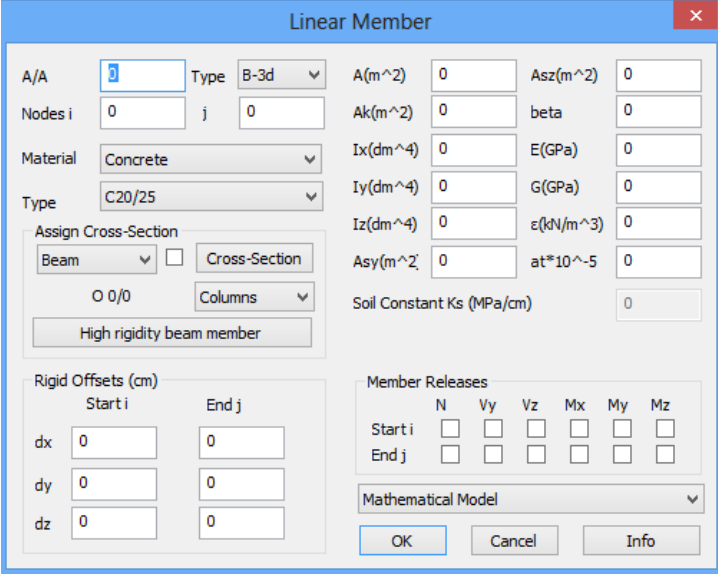
For example, picking a column and clicking "More", the column's properties dialog box appears:

4.1.2 Member

Select the command and then select the member graphically in 2D or 3D display or from the "Tree" list. The selected member is colored red to be identified easily.

Depending on the type of the selected element, the "Properties" list displays the physical and the geometric characteristics of the linear member.

The changes may be applied directly into the "vertical bar" or into the "More" option that opens the dialog box with the characteristics of the linear member.
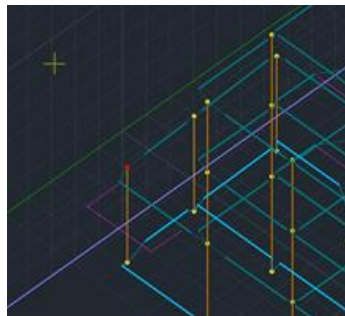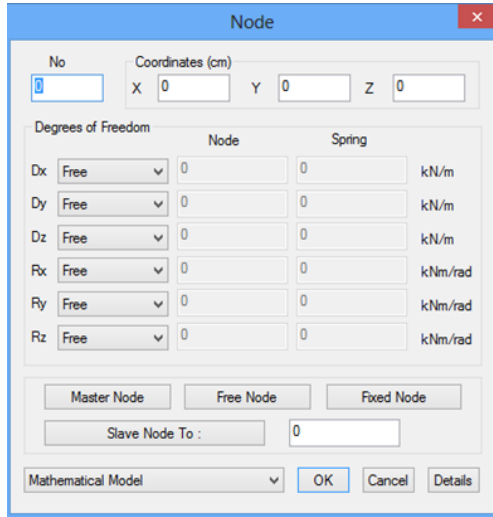

4.1.3 Node

Select the command and then select the node graphically in 2D or 3D display or from the "Tree" list. The selected node is colored red to be identified easily.
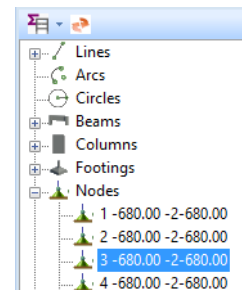
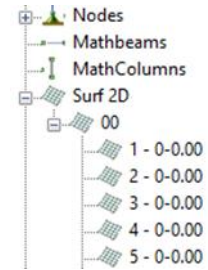In the "Properties" list, the physical and the geometric characteristics of the node are displayed.

The changes may be applied directly into the "vertical bar" or into the "More" option that opens the dialog box with the characteristics of the node.
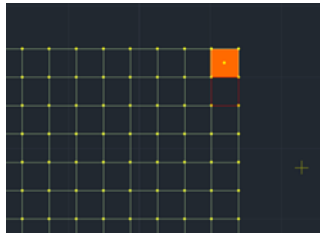
4.1.4 Surface Element

Select the command and then select a surface element graphically in 2D or 3D display or from the "Tree" list. The user may also select a single surface finite element.
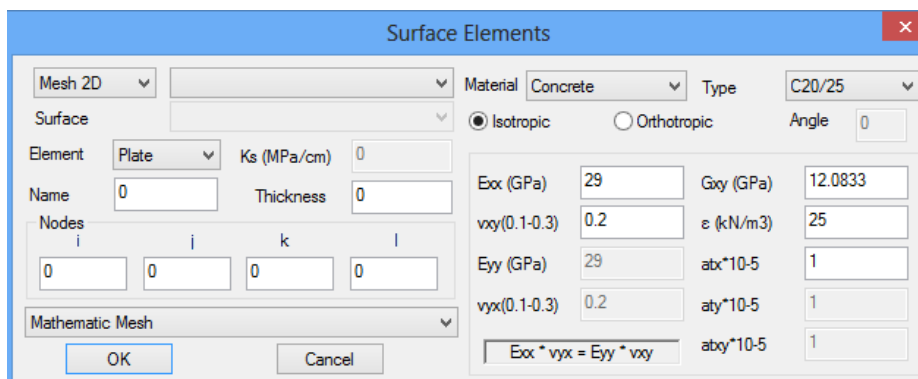
The selected element is colored red to be identified easily.

In the "Properties" list, the physical and the geometric characteristics of the element are displayed.

The changes may be applied directly into the "vertical bar" or into the "More" option that opens the dialog box with the characteristics of the surface element.
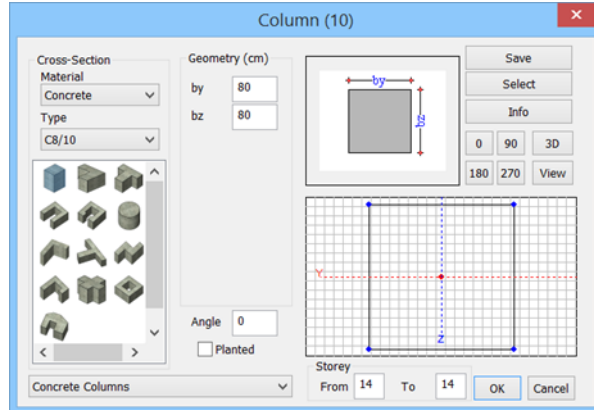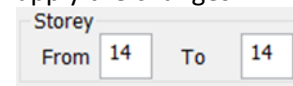
4.1.5 Column Alignment

This command is used for changing the parameters of a column section in one or more levels simultaneously.

Select the command and click on the corner or the side of the column that will be used as a reference. The following dialog box appears:

Apply the changes and define the first and the last level in which you want to apply the changes.

⚠ The same command can also be found in the command list that opens by approaching a column and by pressing the right mouse button (see Chapter 2 Modeling –Columns)
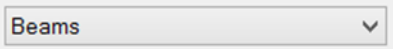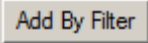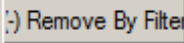
4.2 Numbering Display

This command is used for displaying on the physical or mathematical model information like numbering, degrees of freedom, a moment of inertia etc.

By selecting the command the following dialog box appears:
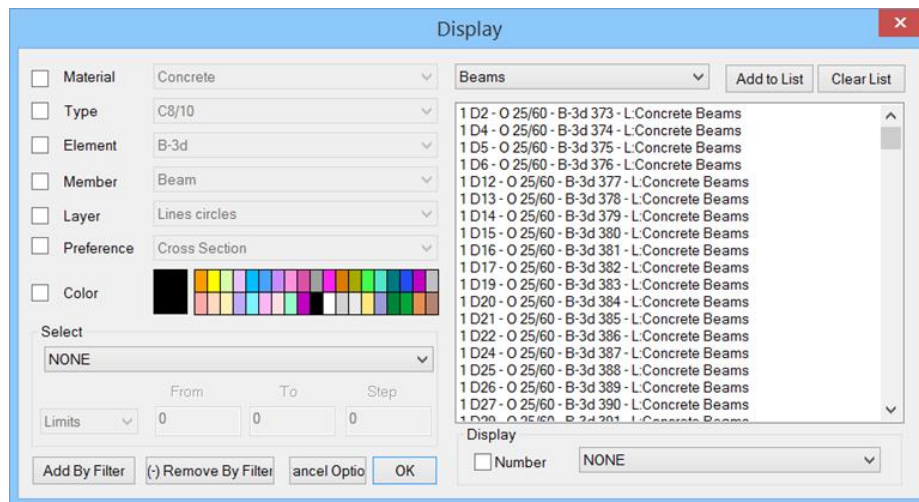
Select the elements by using filters such as "Material", "Type", "Element Type" etc., or one of the groups of the drop-down list [Beams ▼] and then click the "Add to List" button.

Use the commands [Add By Filter] and [Remove By Filter] to add or remove the elements respectively by using a filter.

Select from the list the elements under consideration (left click and ctrl to be highlighted). In the "Display" field, activate the checkbox "Number" to display the numbering of the selected elements.
From the list, select the type of information you want to be displayed.
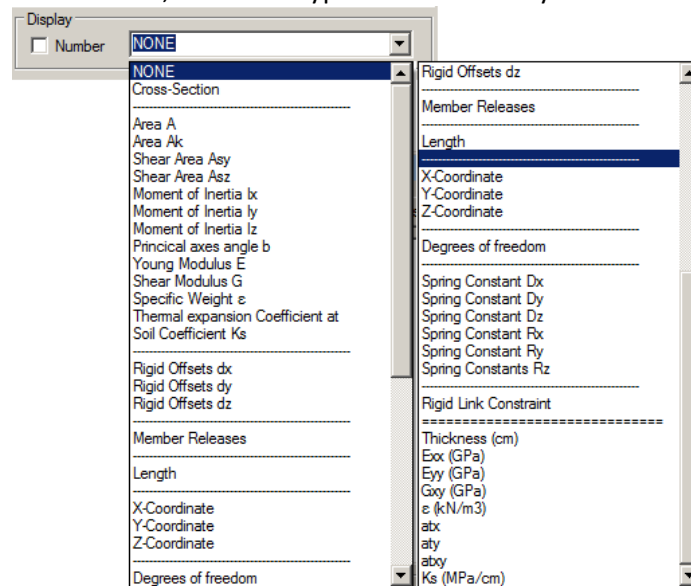
In the "Select" field, you can set additional filters concerning the maximum and minimum values or lower and upper bounds. For example, use this command to display the maximum and minimum values of Cross Section area of beams or display those between the two bounds ("From" and "To") with a specific step etc.
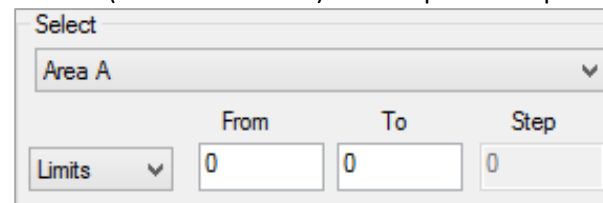
To remove the displayed information, select "NONE" in the drop-down list, then deactivate the checkbox "Number" and press the "Remove By Filter" button.
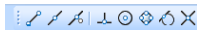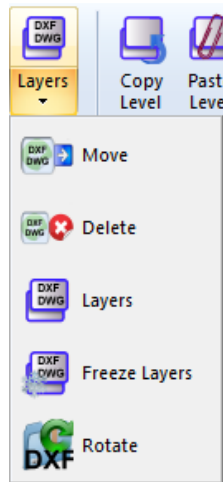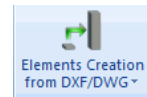
5. DWG-DXF
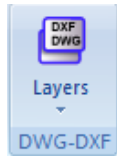
DWG-DXF command group regards auxiliary import files. The user can import DWG/DXF files in SCADA Pro and use them as a guide for the elements' import. At each level, you can also import the corresponding plan view.

The import of an auxiliary file has a double significance:
1. The characteristic points of the included drawing entities can act as snap points.

2. The closed pattern lines, which represent the columns and the beams, are converted to physical sections by using the corresponding tool, automatically. (see Unit "Modeling").

"DWG-DXF" command group contains editing tools about the auxiliary import of files:

- **Move**
- **Delete**
- **Layers**
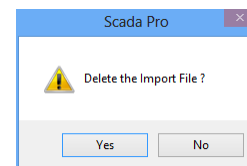- **Freeze Layers**
- **Rotate**

5.1 Move

Is a command to move a DWG or DXF file already imported in SCADA Pro. Select the command and left click to define "From" and "To" points.

5.2 Delete

This command is used to delete a DWG/DXF file, already imported in SCADA Pro. Select the command "Delete" and then press "Yes" in the displayed dialog box.
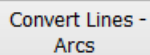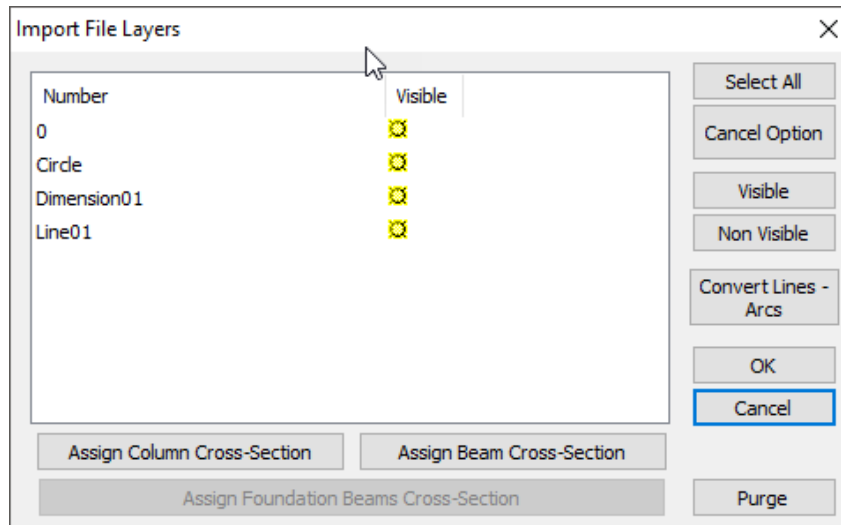
5.3 Layers

This command is used to define the layers of the imported file that will be visible or not on the screen.

Select one or more layers, click on "Visible" or "Non Visible" and "OK".

Convert Lines - Arcs Automatic conversion of lines, arcs, and circles in the corresponding design objects of SCADA Pro.

⚠ When you call an auxiliary file in the environment of SCADA Pro, objects that are imported are not SCADA Pro entities, but only auxiliary lines offering "snapping" at various points.

To convert them to SCADA Pro design objects (lines, arcs, circles), choose from the list the layer or layers to which they belong and then the command "Convert Lines, Arches".
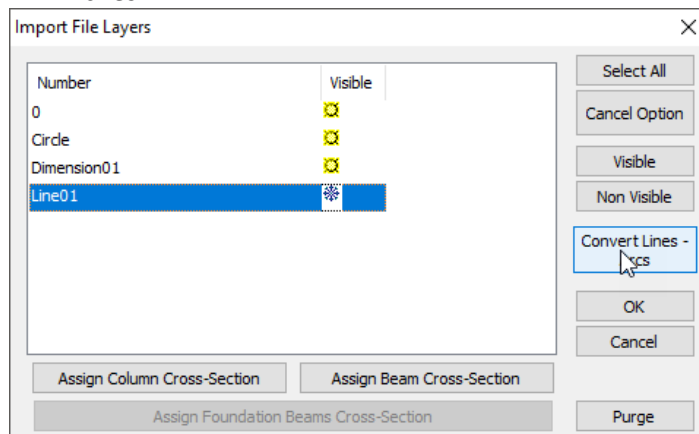
Automatically all the design objects of the selected layer are converted to SCADA Pro design objects with their respective properties.
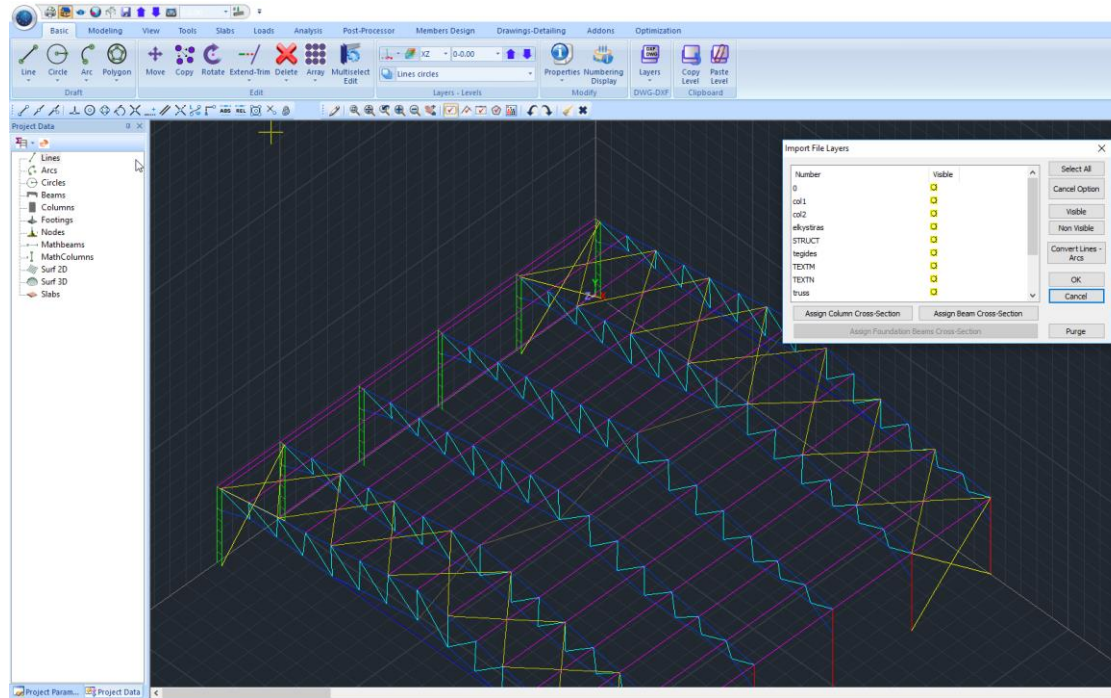
3D dwg-dxf

🔸 SCADA Pro gives you the opportunity, not only to import a 3D dwg-dxf file but also to assign automatically the respective sections to the drawing lines.

First, select the 3D display of SCADA Pro and through the command Insert, insert the 3D design.

Selecting   opens the list with all project's layers.

For the automatic attribution of the cross sections on the drawing lines to be achieved, it is necessary that the lines are matched correctly to the respective layers.

Choose a layer from the list and depending on whether it includes columns or beams, select the corresponding command at the bottom of the window

Automatically the cross sections window opens to select the one that will be assigned on all the lines of the selected layer. Also, choose the angle, the insertion point and the SCADA Pro's layer in which they will be inserted.
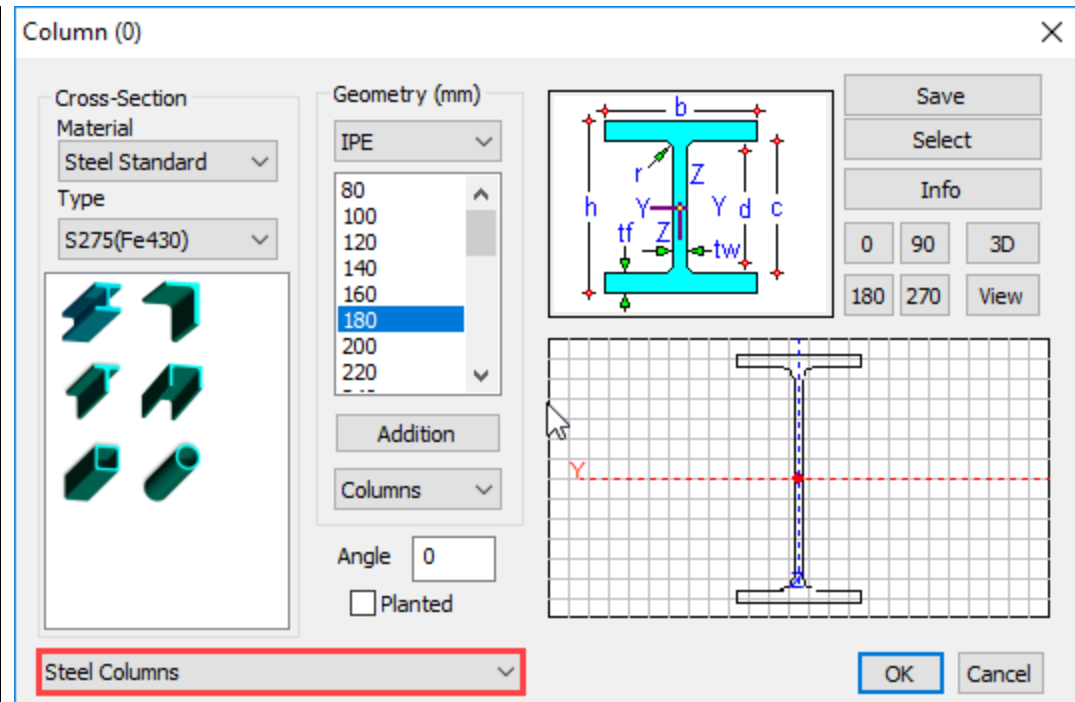
The Members are created automatically containing all the physical and mathematical properties.
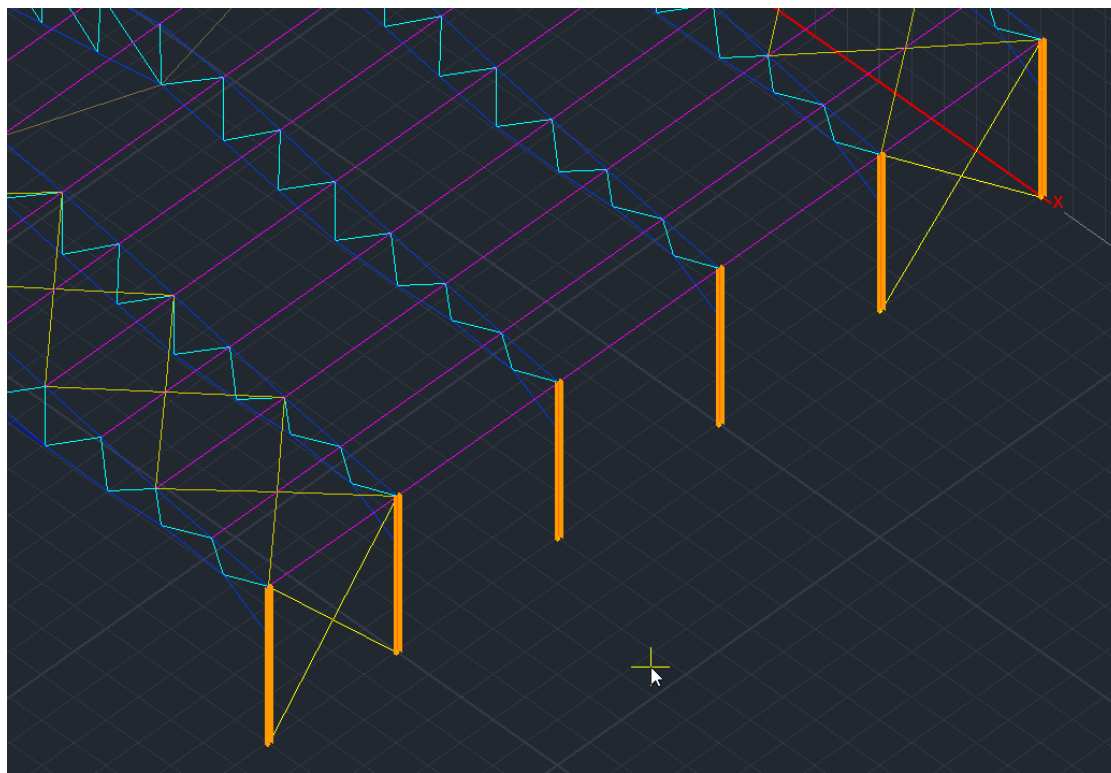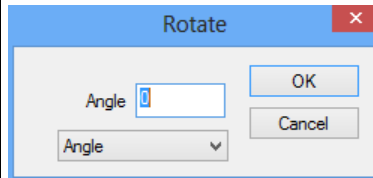
5.4 Freeze Layers

This command is used to freeze the layers of the DWG/DXF file, which has been already imported in SCADA.
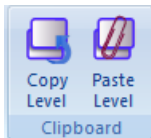
5.5 Rotate

This command is used to rotate the entire DWG/DXF file along with the principal axis X-X or Z-Z. This command is especially useful, in case you want to work in XY or XZ level. Select the command and the following dialog box appears:

Type the rotation angle and select the rotation axis. Then, click to define the center point of the rotation.

6. Clipboard

The "Clipboard" group command contains the commands "Copy Level" and "Paste Level".

The command "Copy Level", selects all **physical elements** contained on the current level, and the command "Paste level" pastes the copied elements to another level.

A level that contains **physical elements** is displayed on the screen. Select the command "Copy Level" and use the arrows to change level. Then select the command "Paste level" to copy here all the physical elements of the previous level. Repeat the procedure to copy on a different level.

ATTENTION:
⚠ Do not use the commands "Copy - Paste level" after the creation of the mathematical model. Remember that the "copy" refers only to physical and design entities, and not mathematical.

⚠ The commands "Copy - Paste level" copies all the physical elements of the level without erasing or replacing them. Special attention is needed in order not to create duplicated copies.

⚠ The Copy - Paste Level command copies all the physical elements of the level to the paste level. A warning message makes sure you do not duplicate one another.
Select either to delete all existing elements at the selection level by replacing them with those of the selected one, or keep them in mind, however, taking into account that if the

items are already in the same position, they will not be overwritten but overlapped by causing a model error!