News

Version: 11.4.0

S.T.A. DATA srl - C.so Raffaello, 12 - 10126 Torino - 011 6699345 - fax 011 6699375

1 3Muri - Commercial versions

3Muri has standard modules and add-on modules (protected by licence):

	Italian	Switzerland	Nederland	Greece
Code				
• Italian sismic calculation [Details]	✓	✗	✗	✗
• Eurocode [Details]	✗	✓	✗	✓
• Eurocode (NL) [NPR 9998/2015]	✗	✗	✓	✗
• SIA [Details]	✗	✓	✗	✗
Computing Modules				
• Local Mechanisms Analysis [Details]	✗	✗	✗	✗
• Static Analysis NTC08 [Details]	✓	✗	✗	✗
• Static Analysis EC6 [Details]	✗	✗	✗	✗
• Roof [Details]	✓	✗	✗	✓
• IFC	✗	✗	✗	✗
• Sensivity [Details]	✗	✗	✗	✗
• Multiprocessor [Details]	✗	✗	✗	✗
• SteelConnection [Details]	✗	✗	✗	✗
Language				
• Italian	✓	✗	✗	✗
• English	✗	✗	✓	✗
• German	✗	✓	✗	✗
• French	✗	✗	✗	✗
• Greek	✗	✗	✗	✓
• Albanian	✗	✗	✗	✗

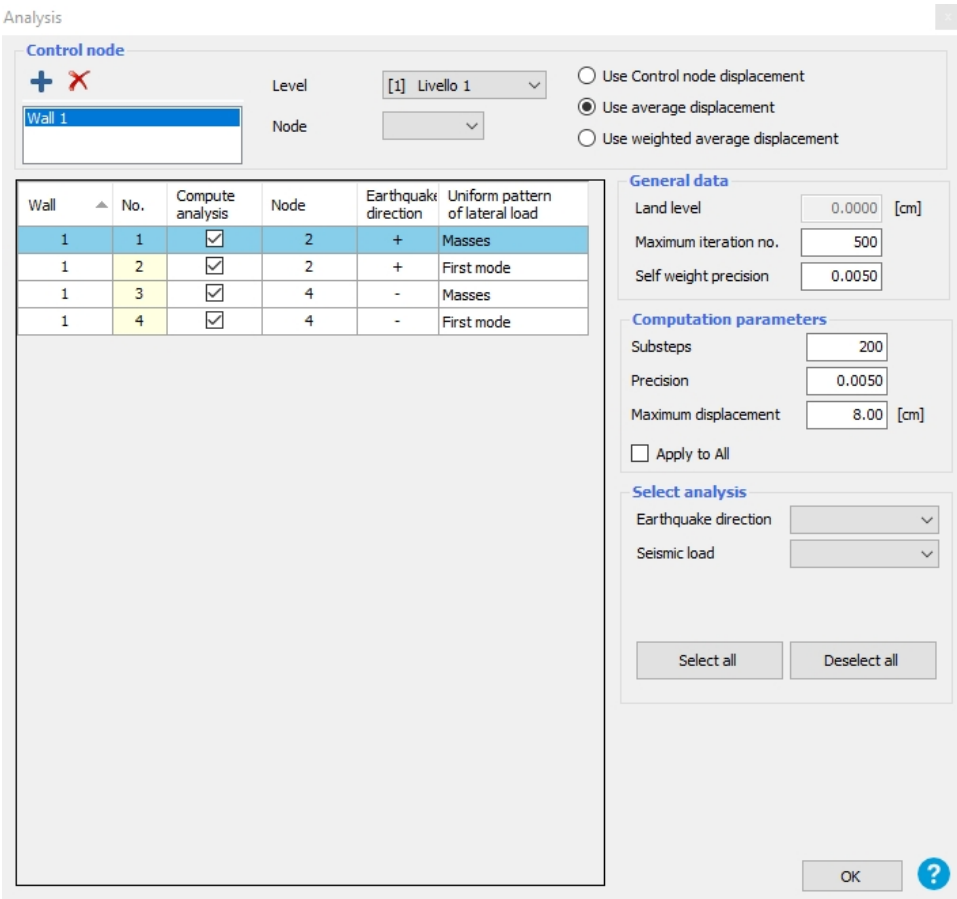
✔ : Standard module

✘ : Add-on module. Covered by license (contact distributors to get it)

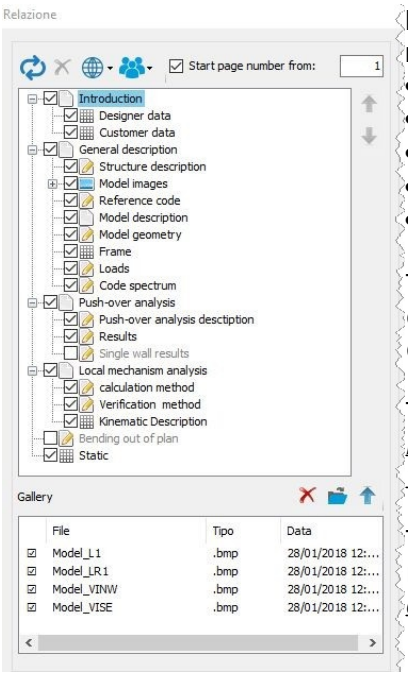
2 News

Pushover Analysis of single wall

Non-linear static analysis (pushover) is commonly known as a global analysis of the building.
All global analysis requires that the building possesses a good behavior like a box.
Sometimes, the design practice confronts us in cases where the limited stiffness of the slabs of the structure portions may affect in an important way the overall behavior of the building.
Walls connected to the structural context through slabs of limited stiffness may make it opportune of individual walls verifications.
A new feature allows you to graphically select more walls of which is desired to conduct the verification of the single wall.
For each wall, the program automatically generates 4 analysis to take into account the 2 directions for 2 different load distributions.



Report environment renovated



Previous versions of the 3Muri program generated distinct reports:

- Pushover analysis
- Local mechanisms analysis
- Static analysis
- Pushover analysis of single wall
- Bending out of plan

To facilitate the compilation of documents, they are now grouped together in a *single environment* in which the different types of analyzes constitute the main chapters.

The program creates *in automatic way updated model images both of structural plans and two of axonometric views*, and places them in a special chapter in the report.

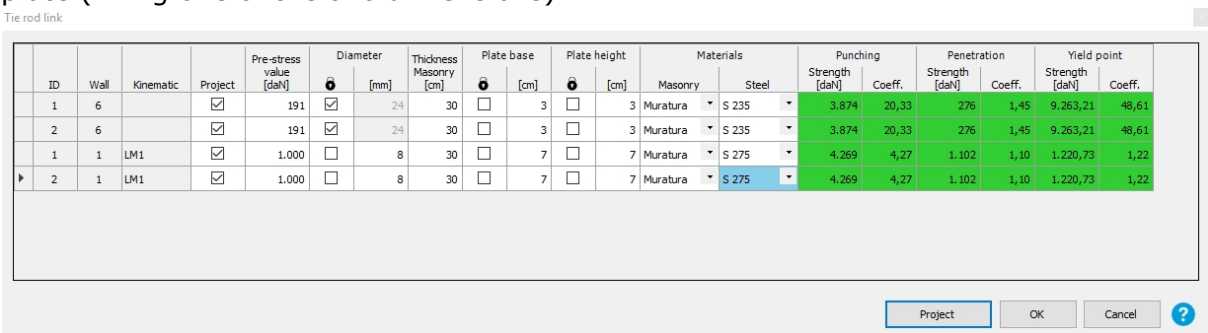
The new procedure with which the report was restructured allows to obtain a final *result ready for delivery*, considerably limiting any further reprocessing.

Tie rod link

The integrated Anchoring Calculation module allows the verification and the design of a tie rod-plate-masonry anchoring system.
The calculation includes the following verifications:

- punching verification of the masonry in the anchorage areas;
- penetration verification of the anchor (tension at the plate-masonry interface);
- yield verification of the tie rod.

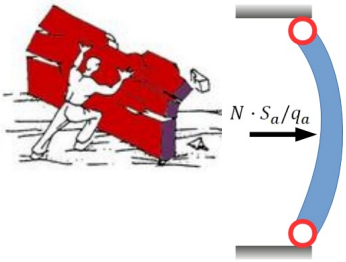
Given the characteristics of the masonry and the pre-stress value that the anchor must resist, it is possible to carry out the project with the shape of the rectangular square plate (fixing one of the two dimensions).



ID	Wall	Kinematic	Project	Pre-stress value [daN]	Diameter		Thickness Masonry [cm]	Plate base		Plate height		Materials		Punching		Penetration		Yield point	
						[mm]			[cm]		[cm]	Masonry	Steel	Strength [daN]	Coeff.	Strength [daN]	Coeff.	Strength [daN]	Coeff.
1	6		☑	191	☑	24	30	☐	3	☐	3	Muratura	S 235	3.874	20,33	276	1,45	9.263,21	48,61
2	6		☑	191	☑	24	30	☐	3	☐	3	Muratura	S 235	3.874	20,33	276	1,45	9.263,21	48,61
1	1	LM1	☑	1.000	☐	8	30	☐	7	☐	7	Muratura	S 275	4.269	4,27	1.102	1,10	1.220,73	1,22
2	1	LM1	☑	1.000	☐	8	30	☐	7	☐	7	Muratura	S 275	4.269	4,27	1.102	1,10	1.220,73	1,22

2.2 Version 11.3

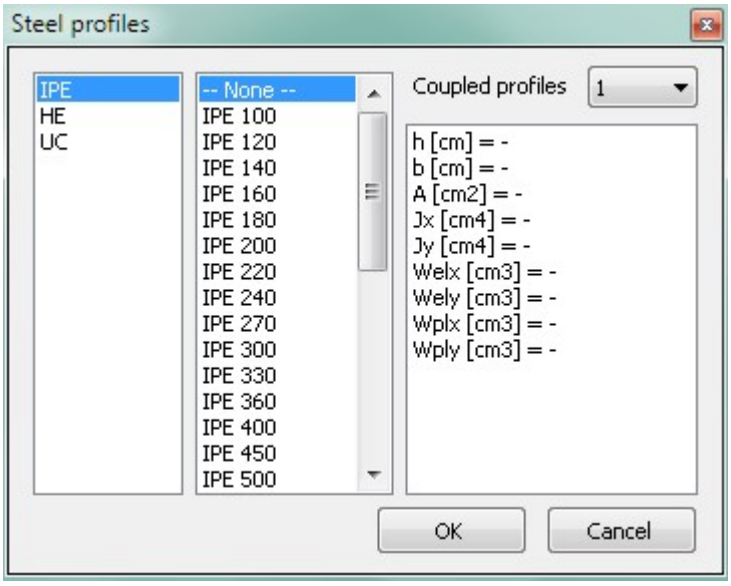
Verifications out of plan

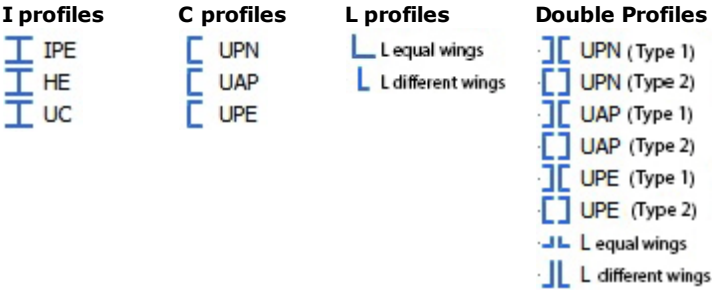


Bending type checks to take account the out plan behavior of the masonry.
These checks are of a local type, based on a control in terms of resistance by applying the resistance criteria specified in the current regulations.

N.	Ned [daN]	NRd [daN]	Sa [m/s2]	Med [daNcm]	MRd [daNcm]	PGAc [m/s2]	MRd/Med
19	2'691	17'189	0.23	15'340	34'050	2.84	2.22
20	12'641	53'553	0.23	72'047	144'856	2.57	2.01
21	12'284	51'032	0.23	70'015	139'909	2.55	2.00
22	3'284	19'710	0.23	18'719	41'056	2.80	2.19

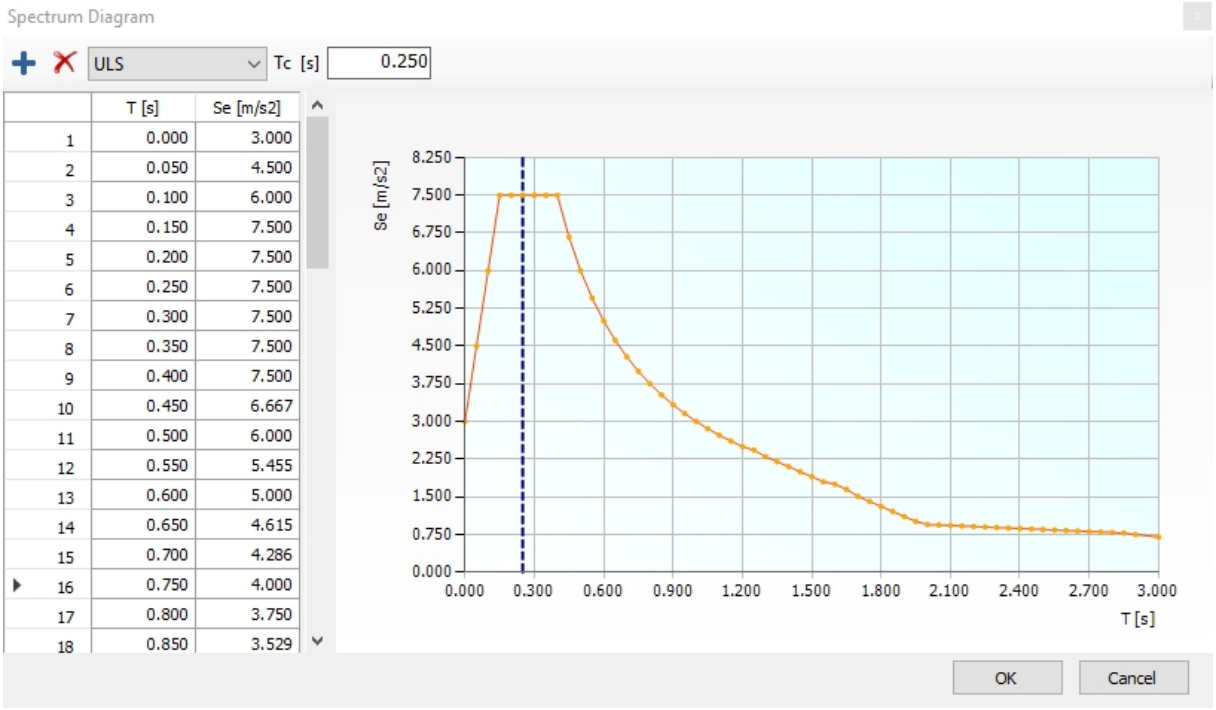
New library of metal profiles with new configurations





Definition of the spectrum by points

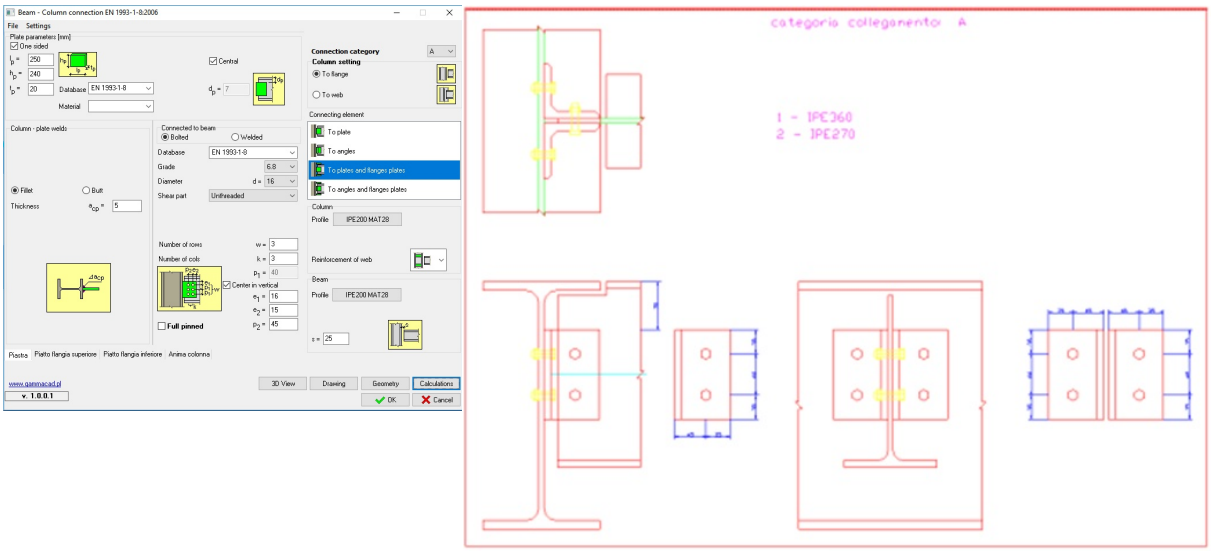
In addition to the classical process of defining the seismic spectrum based on the reference grid and to the definition of seismic parameters, it will be possible to define in a general way by points.



Link to SteelConnection

[OPTIONAL]

From the selection of the frame nodes it is possible to run the "Steel Connection" module for the verification and design of the connections.

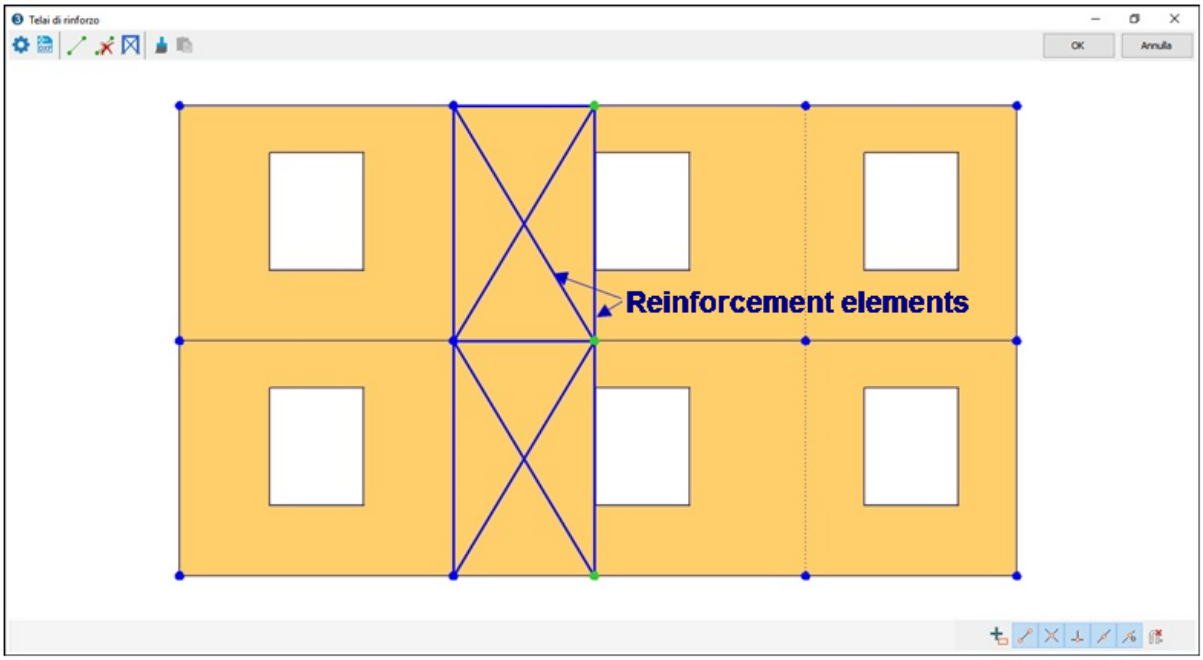


The "Steel Connection" module offers a wide range of types of steel connections, such as to cover all the needs of the designer. In addition to the calculation report, complete and exhaustive, they are also available designs of connections in DXF format.

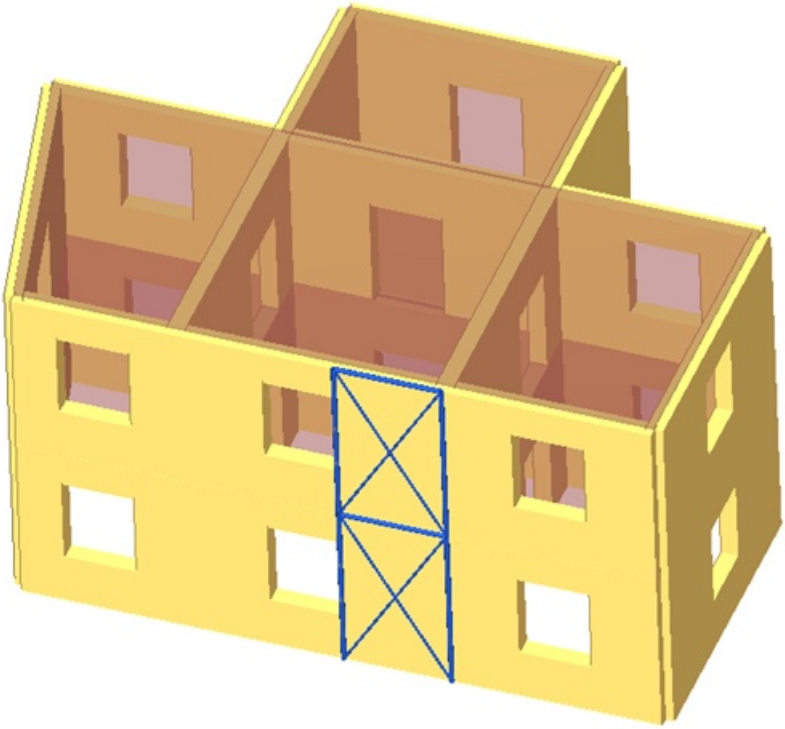
2.3 Version 11.0.0

Wall reinforcements through the frame

Selecting a wall and choosing the *"reinforcement frames"* item will open a dedicated CAD environment that shows the wall prospect.
Through the command bar it is possible to interact graphically defining the reinforcing elements.



Following insertion in perspective view, the reinforcing elements will also be shown in the axonometric view.

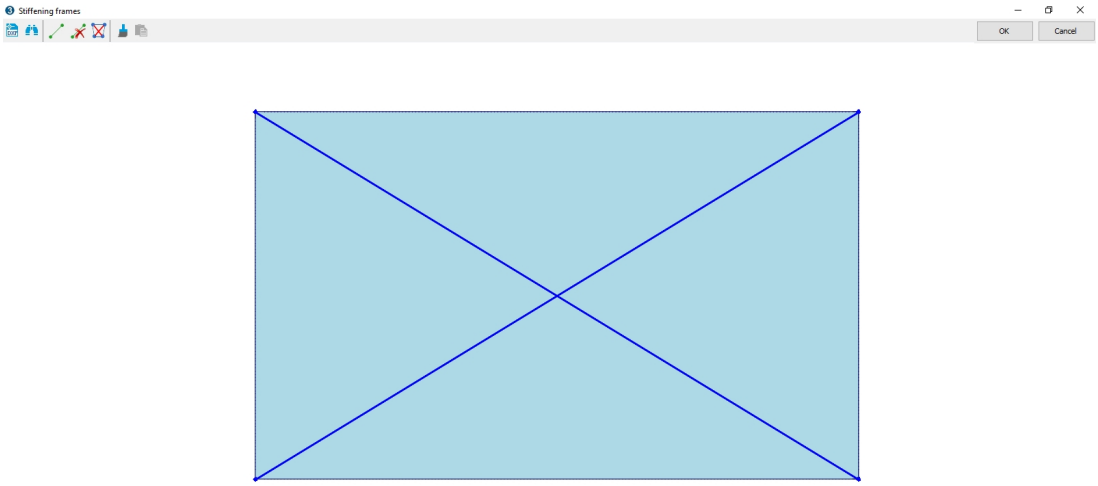


The structural elements that can be inserted are steel or wooden beams and tie rods (elements with only tensile strength).

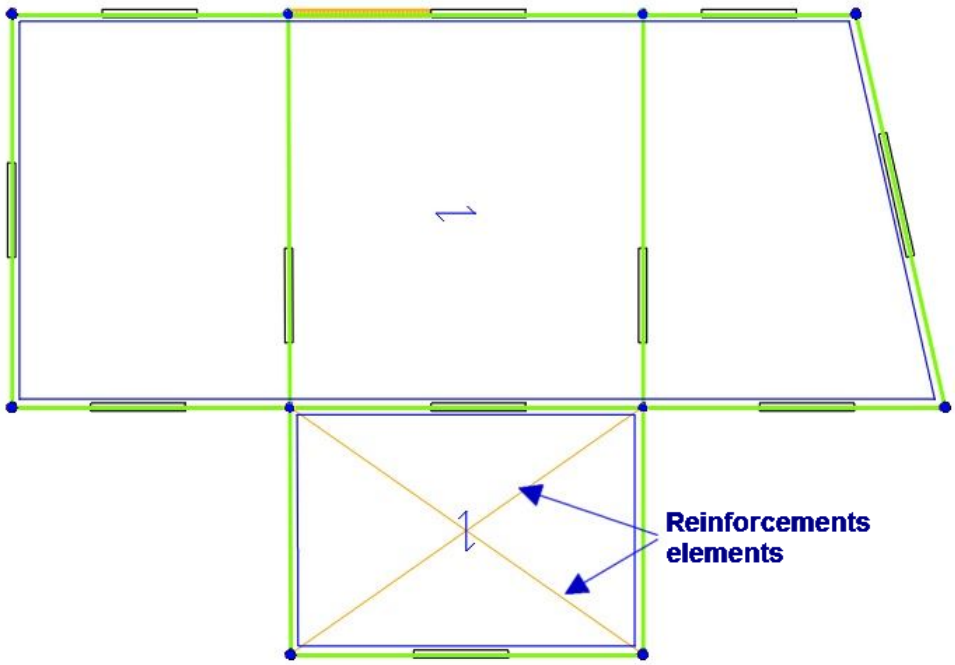
Slabs reinforcements through the frame

The slabs of existing buildings, very often made of wood, have extremely limited stiffness, allowing important deformations and distortions in plan.

It is not always possible to enter the completion slabs to limit distortions in the plan and therefore it is necessary inserting diagonal reinforcements on floors and roof slopes.
Selecting a slab and choosing the "stiffening frames" voice will open a dedicated CAD environment. The following figure shows a diagonal reinforcement applied on the plan of the slab.

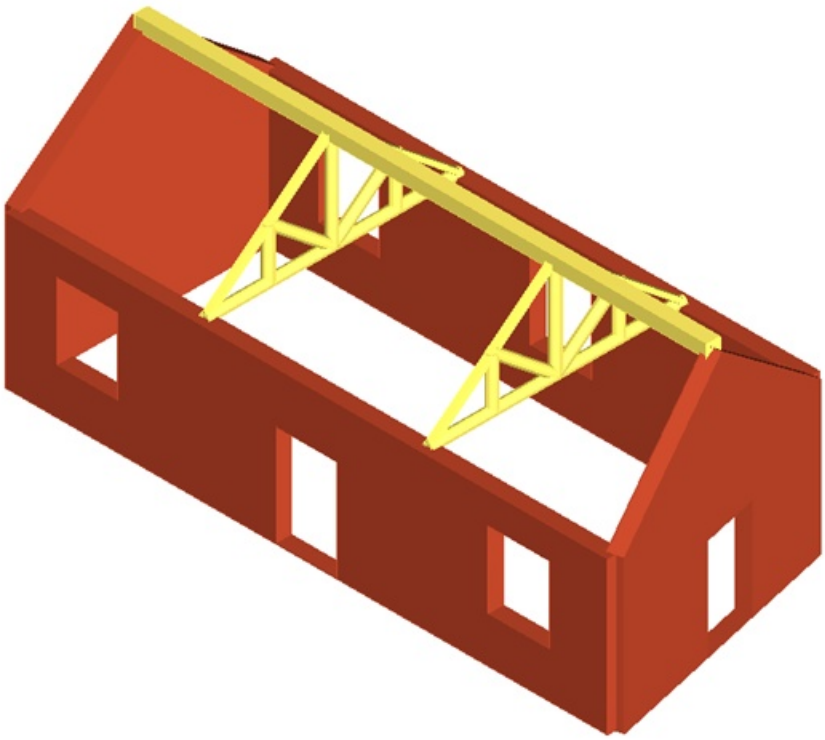


Once completed the input, the reinforcement is visible in the structure plan.



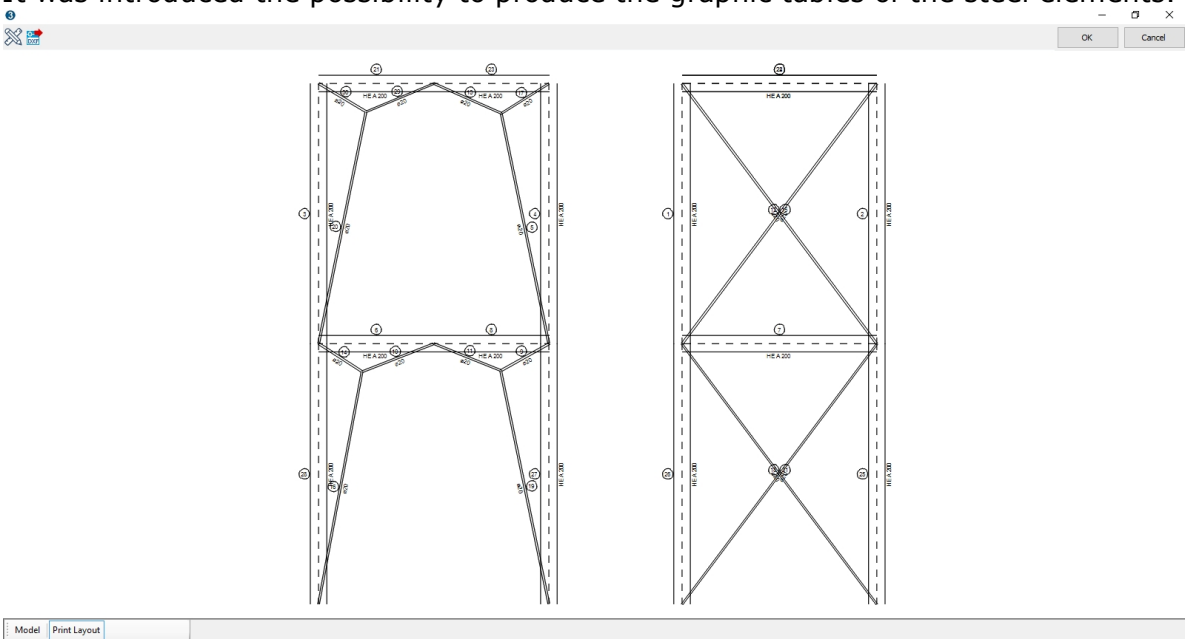
Reinforcement through truss beam

A further application of reinforcement systems is definitely focused to define truss beams in the roof elements. An example of application is shown in the figure below.



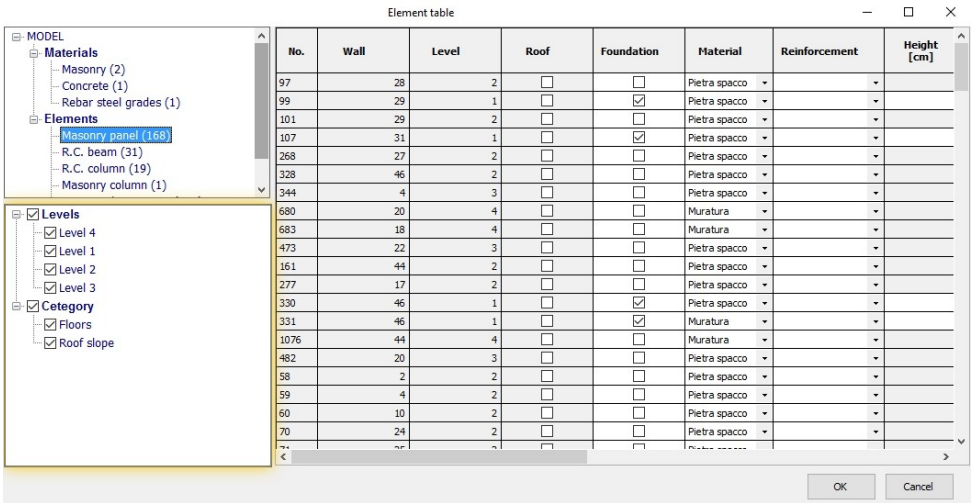
Graphic tables of steel elements

It was introduced the possibility to produce the graphic tables of the steel elements.



Update of the Elements Table

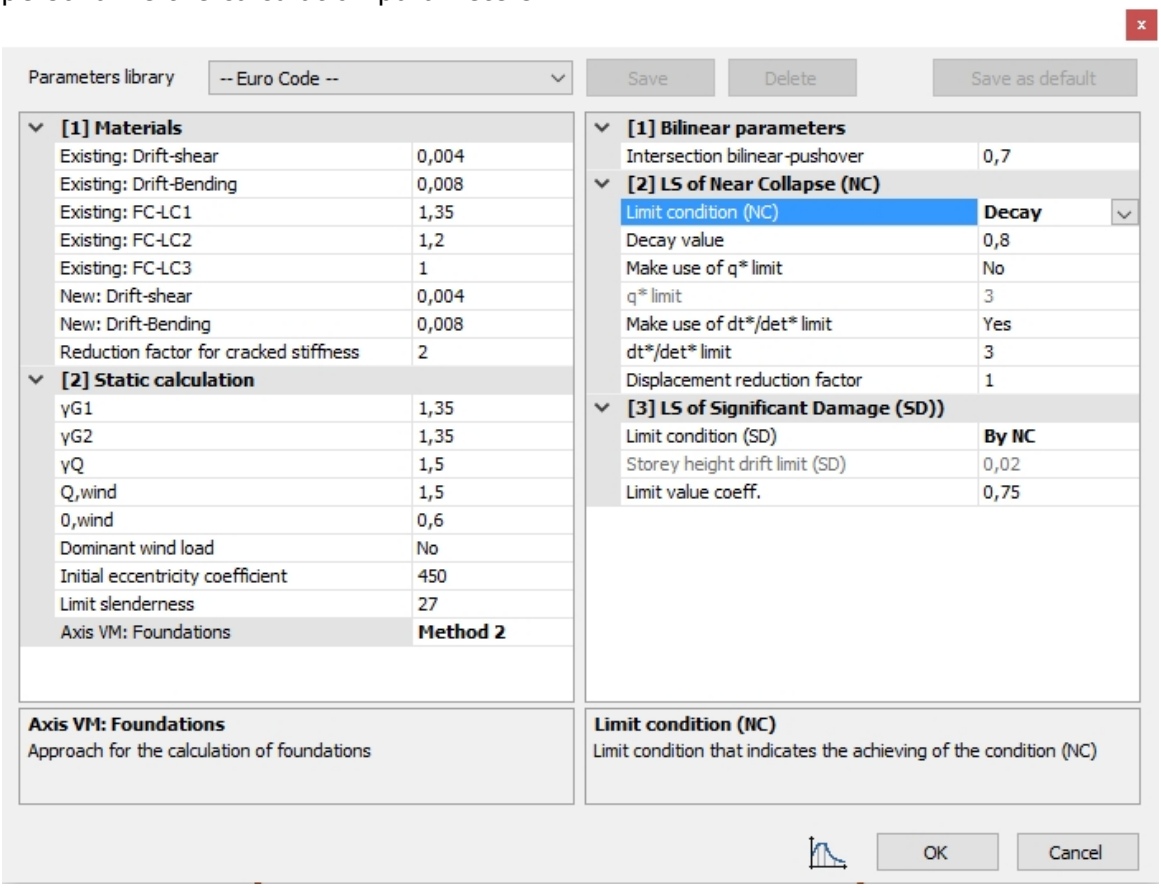
Displaying and editing of elements table have been optimized by inserting new and improved filters.



2.4 Version 10.9.1

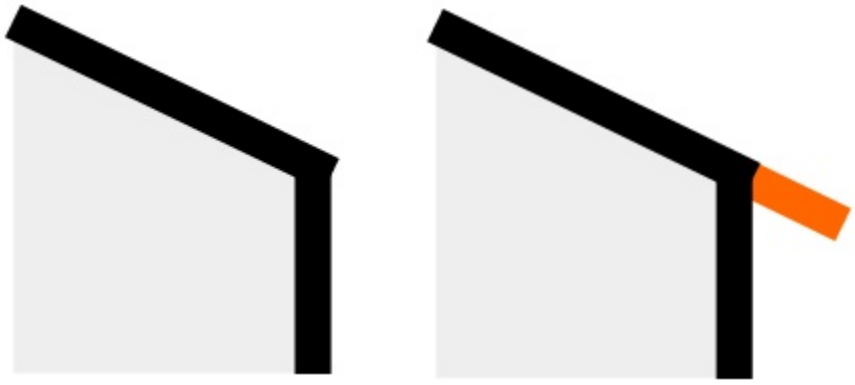
Personalized calculation parameters

For each standard with which you can perform the calculation, is now possible to personalize the calculation parameters.



Roof overhangs

Possibility of inserting the overhang of the roof slope.

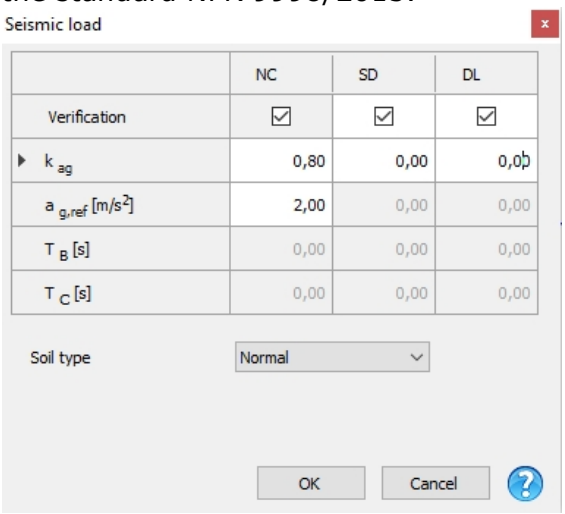


Without overhang **With overhang**

Standard NPR 9998/2015 [Eurocode (NL)]

[OPTIONAL]

Adaptation of the calculation method according to the Eurocode based on Dutch annexed.
This method provides the definition of seismic load according to the specifications of the standard NPR 9998/2015.



2.5 Version 10.9

Static analysis according to Eurocode 6

[OPTIONAL MODULE]

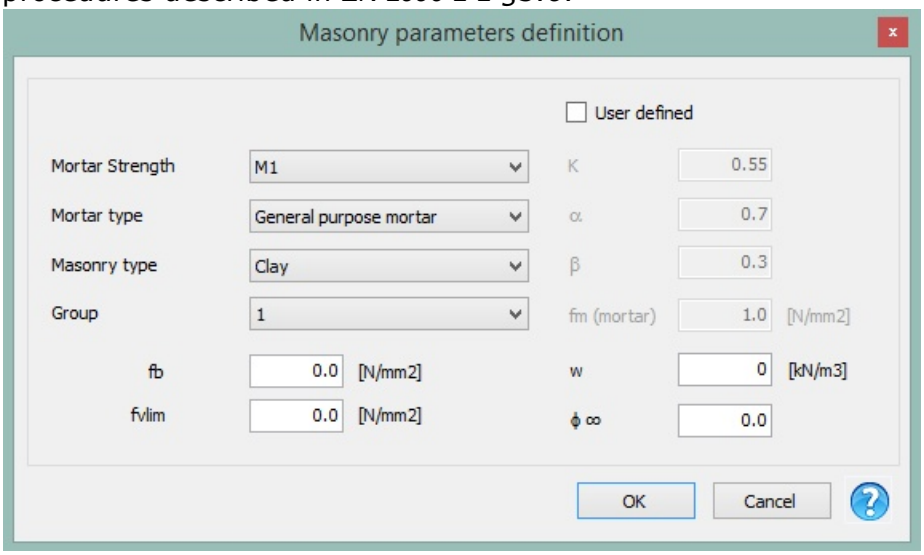
The Eurocode 6 is the European standard specifically dedicated to the verifications of masonry structures.
Each typology of building is interested by two main verification phases:

- Seismic
- Static

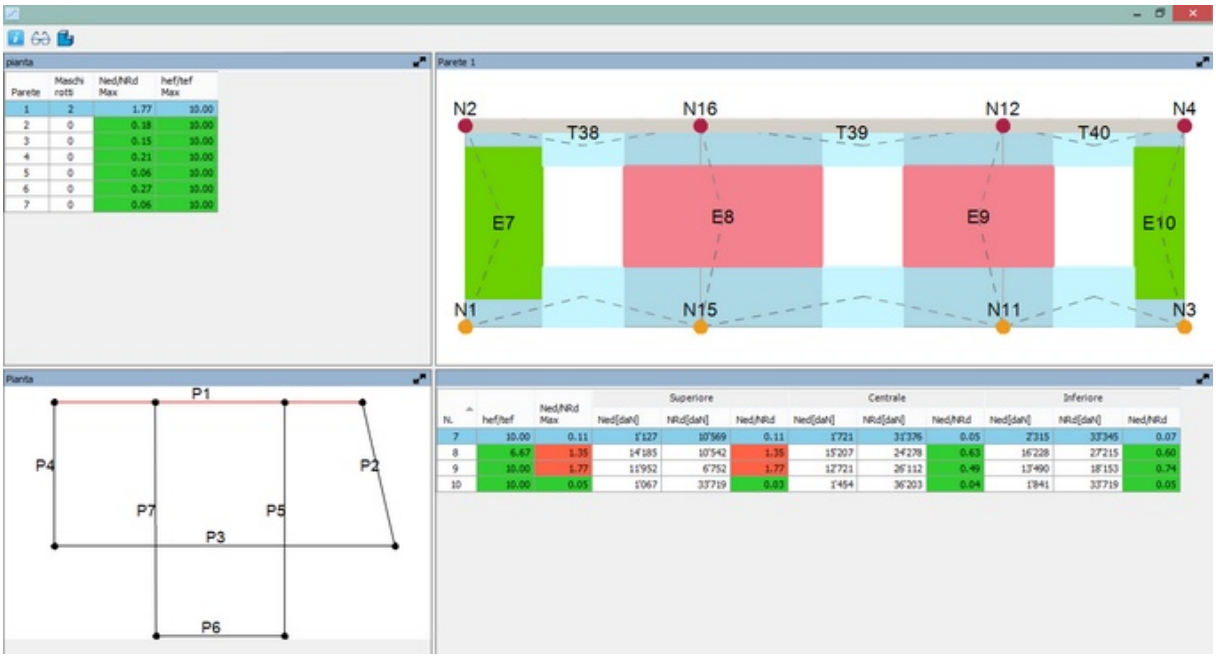
Eurocode 8 contains all the references about the verification procedures of "Seismic" type, there is no indication to the static verifications instead of them contained in the Eurocode 6.

It is now possible:

➔To impute the mechanical parameters of masonry materials according to the procedures described in *EN 1996-1-1 §3.6:*



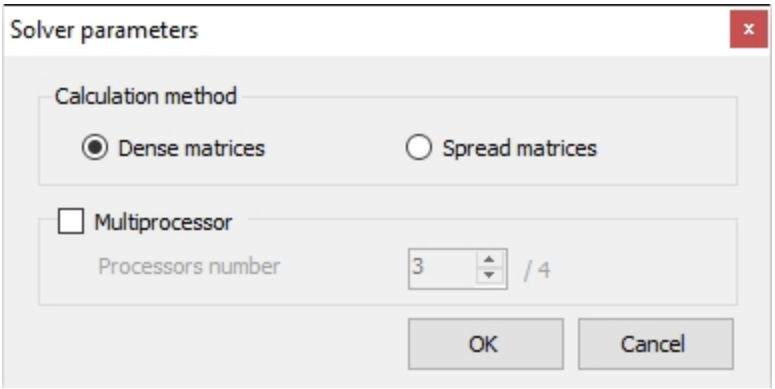
➔ Run the static verifications according to *EN 1996-1-1* § *6.1.2*



Solver parameters

[OPTIONAL MODULE]

There are now present two different calculation settings regarding the processor.

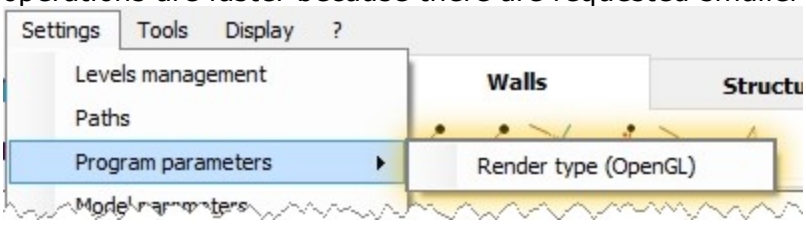


Calculation method:
- Dense matrix
- Sparse matrix

Multiprocessor:
By activating this option it is possible to conduct the calculation simultaneously on multiple processors.
In the case of multiple analysis (pushover 24) is possible to address an analysis on each processor available on the PC.The default proposes to use a number of processors equal to the maximum number minus one, the unused processor is left available to the system resources.
The saving in terms of time therefore depends on the number of processors available by the system.

Graphic optimization

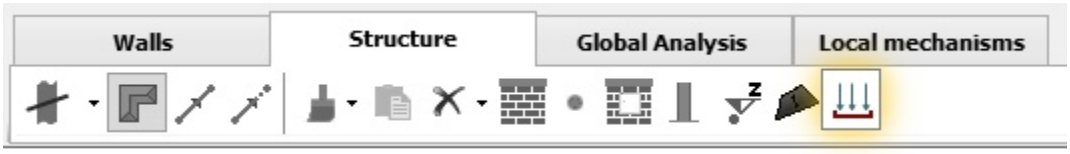
With the option "Render type *(OpenGL)*" **active**, the viewing in 3D is more accurate
With the *"Render type (OpenGL)"* option **disabled** the display 3d and the graphics operations are faster because there are requested smaller resources to the system.



Loads on roofs

The command "Loads", until now present in the structure environment, from now is also available in the roof environment in order to add concentrated or linear loads directly on the roof-slopes.

Some applications are:
- Protrusions of non-structural roof
- Dormers to which you do not want to give structural value but only as a load



2.6 Version 10.5

Sensitivity Analysis [OPTIONAL MODULE]

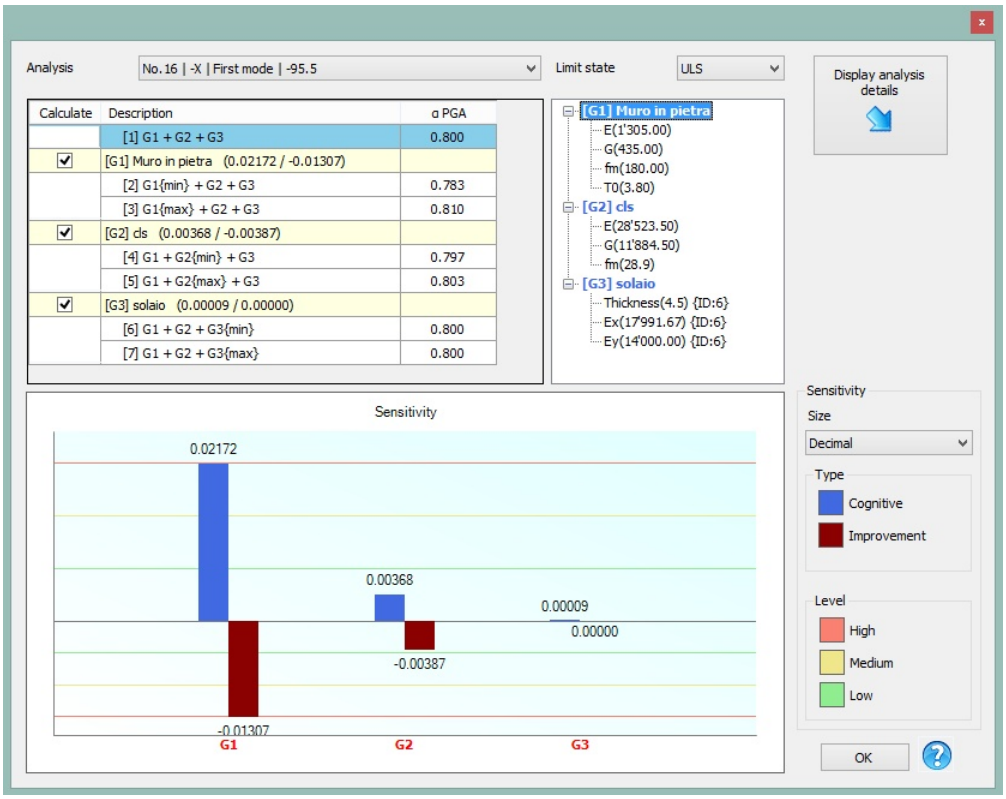
Sensitivity Analysis is a calculation method aimed to obtain better understanding of the structural functioning and accurate planning of the site investigation plan.
As known, doubts during modeling directly affect the evaluation of seismic safety. A specific example is materials mechanical properties, usually defined on the basis of reference values and for which, through investigation, it aims to limit the inescapable uncertainty.
Since the site tests have frequently high economic cost, the possibility to identify in advance (through the Sensitivity analysis) significant testing campaign points can limit investigation costs which result might not be of interest.

In the module are implemented two different types of "Sensitivity":

«Cognitive» Sensitivity *to answer the following questions:*
- Where do the surveys lead?
- Which "Model Assumptions" are more conservative?

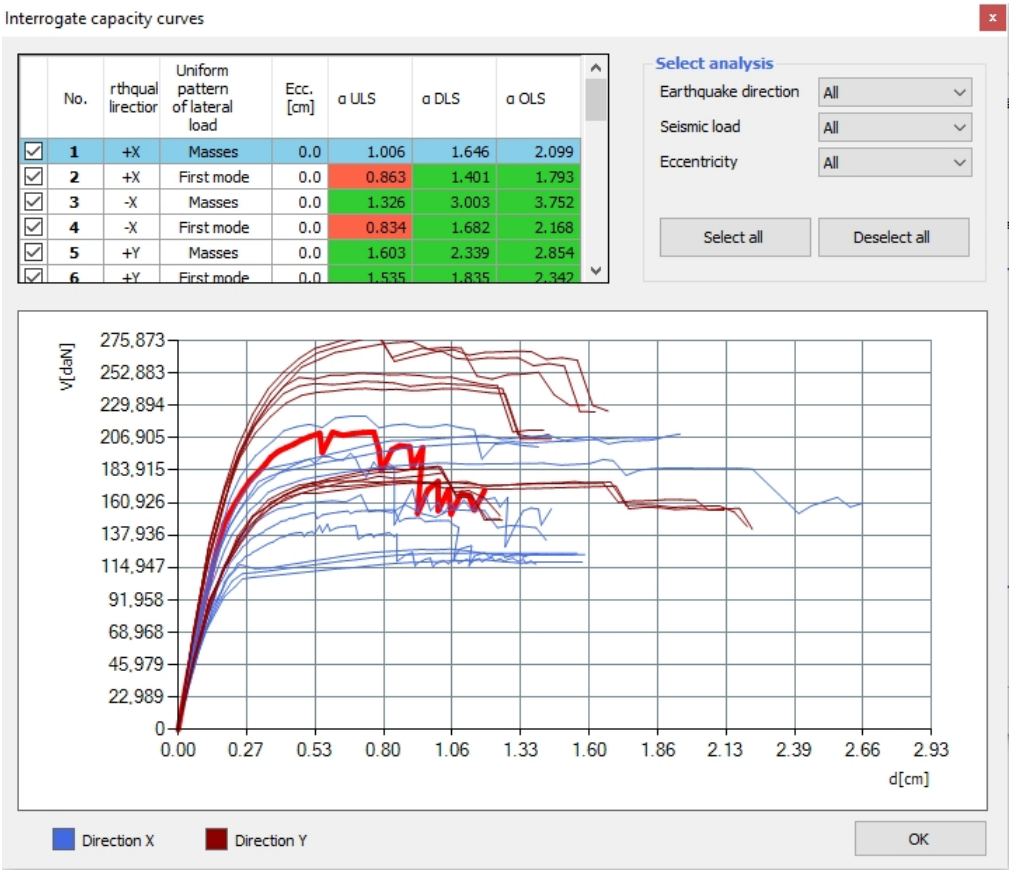
«Improving» Sensitivity *to answer the following questions:*
- Which type of improving intervention is more effective?

The result of this analysis is a "Sensitivity Index" as shown in a special diagram in the results window.



Overlapping capacity curves

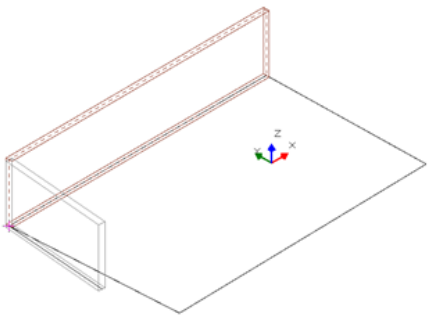
This feature allows the designer to simultaneously refer to the capacity curves of more analysis on a single diagram.



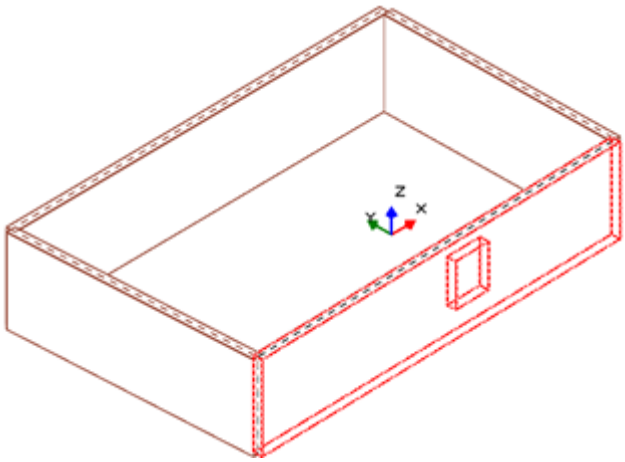
2.7 Version 10.1

3D modeling

To the current input in (2d) plan was joined a (3D) axonometric input mode, with the following characteristics.
You can insert the structural objects directly in an axonometric view.



Insertion of masonry walls in 3D



Insertion of openings on a masonry panel

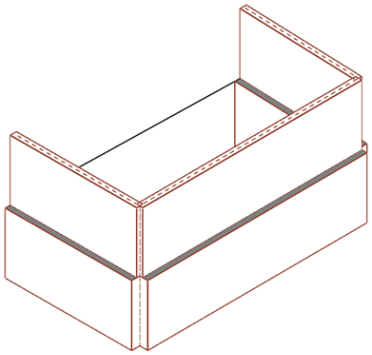
3D modeling is not developed only with the aim to provide an alternative to the already existing plan input but also provide a new mode of already allocated structures editing.
Currently the elements properties multiple editing is easy in plan but less performing in elevation. The editing in 3D will simultaneously change the characteristics of the elements at different levels.

Specific applications of 3D editing

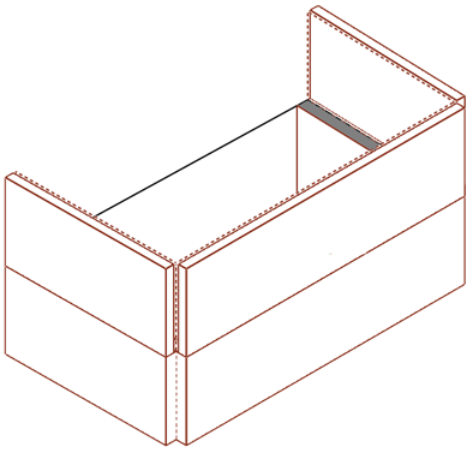
Following a multiple selection of structural objects (also on different levels), you can:

Align panels
It performs the concurrent editing of masonry walls with different thicknesses at different levels in order to align them with the same outer edge.
This operation, that so far was carried out by placing in the wall properties the eccentricity value to be manually calculated, is now fully automated.

[Before editing]
Walls aligned with the lower middle plan



[After editing]
Walls aligned with the outer edge



Assign wind exposed
The properties of a wind exposed panel can be assigned through a multiple selection.

Table editing
The editable table elements has always shown the characteristics of all the model's elements.
In this mode you can now show only the items affected by a selection.

3D Filters-The Parts

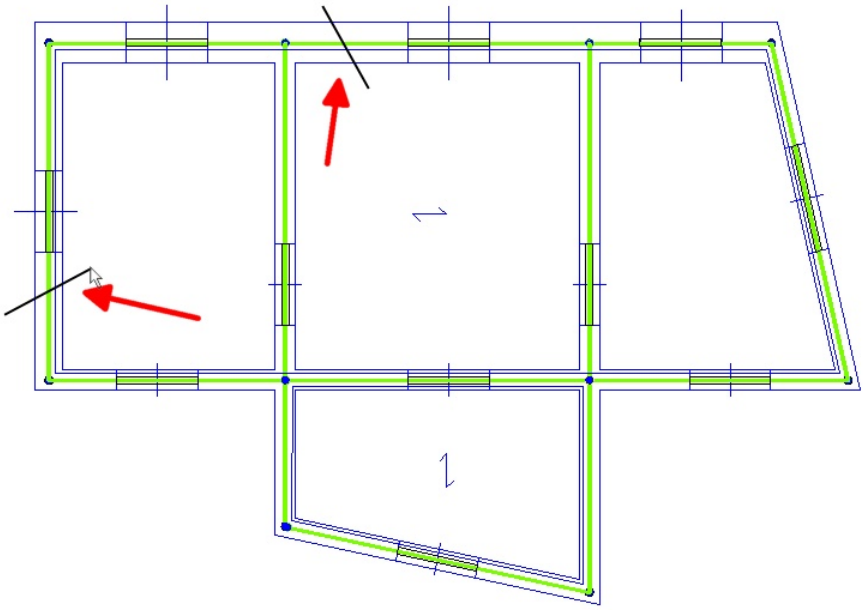
Allows to transform groups of structural elements in defined parts.
Working with parts makes easier the simultaneous editing phases of multiple elements.
You can display one or more parts, called active parts, at once.

There are two types of parts: user defined parts and logical parts.
The user defined parts are created by the user selecting the items that belong to the parties.
The logical parts are automatically created by the program, ordering the items in different criteria categories (material, elements).

Automatic detection of the thickness

During the input phase, you no longer have to worry about the various wall thickness in the different sections and perform the input with any thickness.

In a second step a dedicated command (by tracing a segment that intersects the wall) automatically calculates the thickness by extracting it from the DXF file and assigns it to the already defined section.



The section of the panel in order to automatically compute the thickness of the DXF file

---

Input by dimension lines

The input of the walls without the DXF background and without the need to use a coordinates box is possible through the cad dimensioning command with ability to directly enter the extension of the wall.
The direction of the walls can be easily identified through the use of guidelines.
The same dimensioning command is available for moving nodes, openings and concentrated loads.

IFC management – Output/Input

[OPTIONAL MODULE]

Export 3Muri to IFC

Once you have finished creating the model, the command "IFC export" in the File menu allows to create the IFC file.
This file includes the structural objects defined by the IFC standard in order to be visible with a player or BIM CAD.

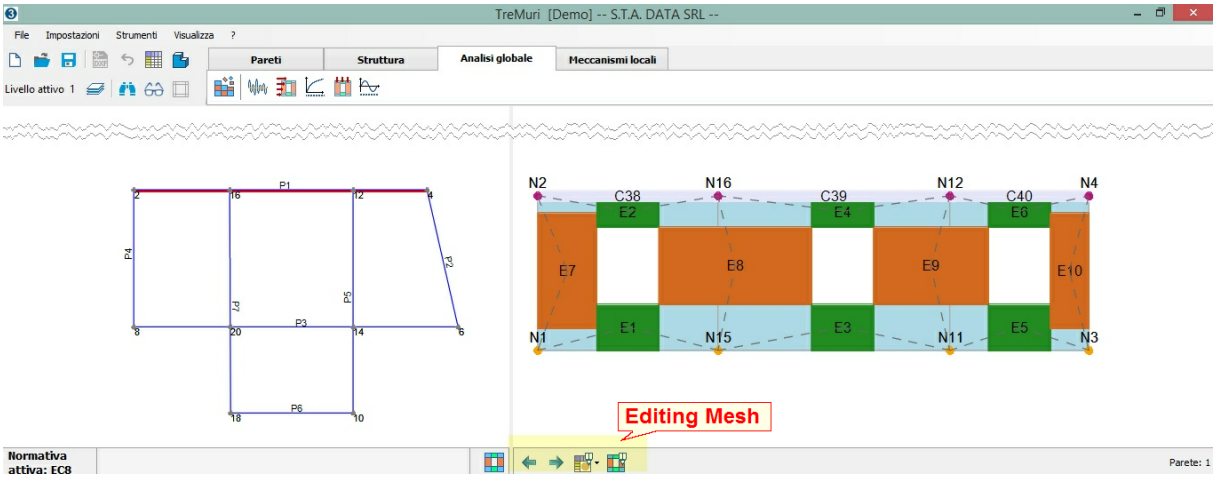
Advanced importation

Runs the OpenCAD application, a CAD system that makes interoperability its strong point.
With interoperability we mean the ability to cooperate and exchange information with the other products or services with resources's optimization.
The information exchange takes place through various graphic formats:
***IFC**, DGN (Bentley), DXF, SKP (File Sketchup), EMF, WMF, BMP, GIF , JPG/JPEG, TIF, DWG*

2.8 Version 10.0.2

New Editing Mesh

In mesh editing you can change the characteristics of the generated structure according to the design requirements.
The access in the editing mesh area is achieved through the appropriate button on the command bar.



By clicking on the appropriate button you can enter the editing area.

2.9 Version 10

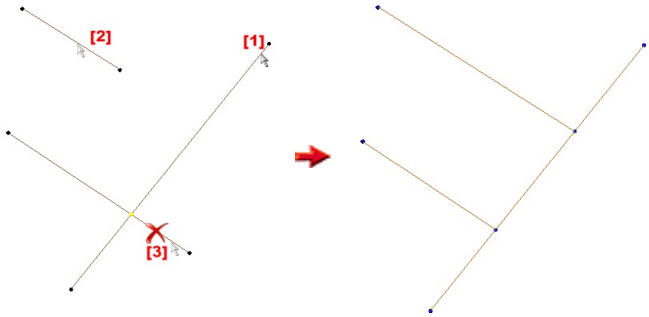
3Muri r. 10 is the new version of the software for the seismic calculation of the masonry structures.
The program has been completely rewritten in order to update it to the new operating systems, improve the present features and add new commands.

New graphics maintaining the ergonomics and ease of use to which all users are affectionate.
Some of the main functions are outlined below:

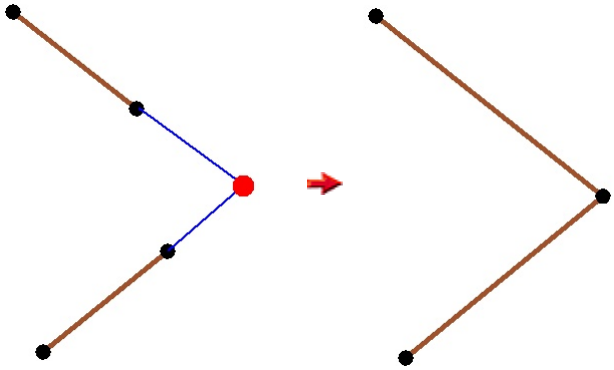
New walls' editing operations

Rectify walls: Allows to rectify the previously entered walls.

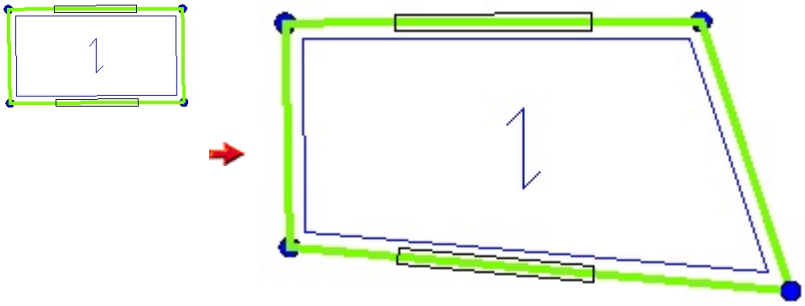
Extend/ Trim walls: allows to extend or shorten an already existing wall.



Fillet walls: Classic cad command for joining two walls that do not cross.

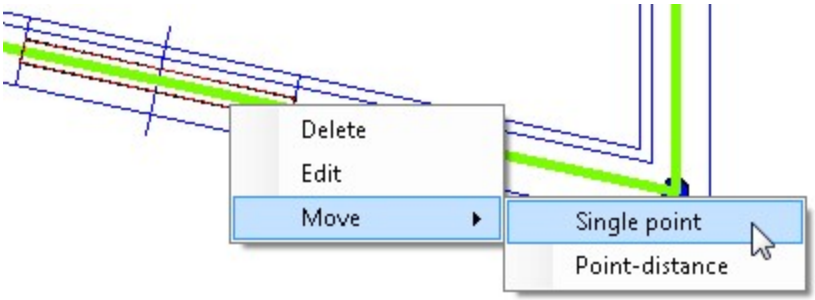


Stretch: This command allows to move an outer node of the wall.



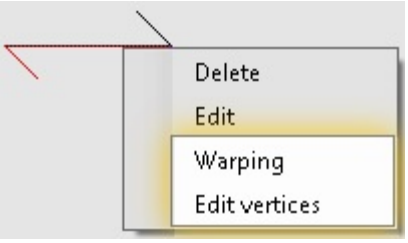
Movement of the openings.

Selecting with the right mouse button allows to move an already inserted opening.



It is necessary to define the displacement vector by clicking on two points, the start and end point.

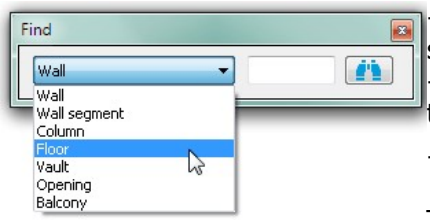
New editing modality of the slabs.



Slab warping direction:
Select a wall bounding the slab in order to redefine the warping direction of the slab.
Edit vertices:
Once inserted a slab you can edit the vertices hooking them to different nodes.

New search functions of the structural objects`

The "Find" command searches in the graphics for a wall, a wall segment, a slab, a pillar, a balcony if is known the identifier.



- Select from the menu the type of item you want to search.
- Enter the number of the element to be found in the text field
- Press to start the search.

The search result is shown by placing the searched element in the middle of the video, with the mouse pointer on it and a special marker that highlights it.

New DXF file importation functions

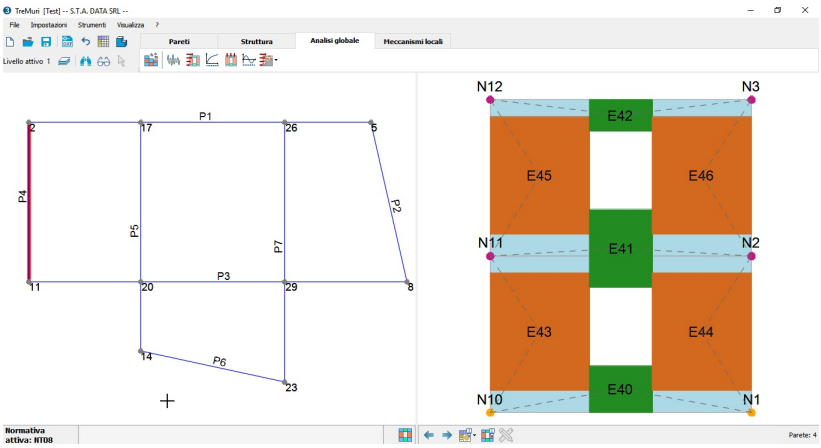
Cancel or easily move a DXF file translating it in the graphics area. These new functions are directly available by clicking on the right button of the mouse in the graphics area.

Editable elements table

The table appears by clicking on the proper button that shows the characteristics of all the information inserted by the user through the input windows while creating the model. The drop down menu bar on the left, facilitates the navigation in these tables.
The main characteristic of this table is the fact that it can be edited.
Every modification of the table brings the direct modification of the model`s characteristics.

New mesh display of the geometry

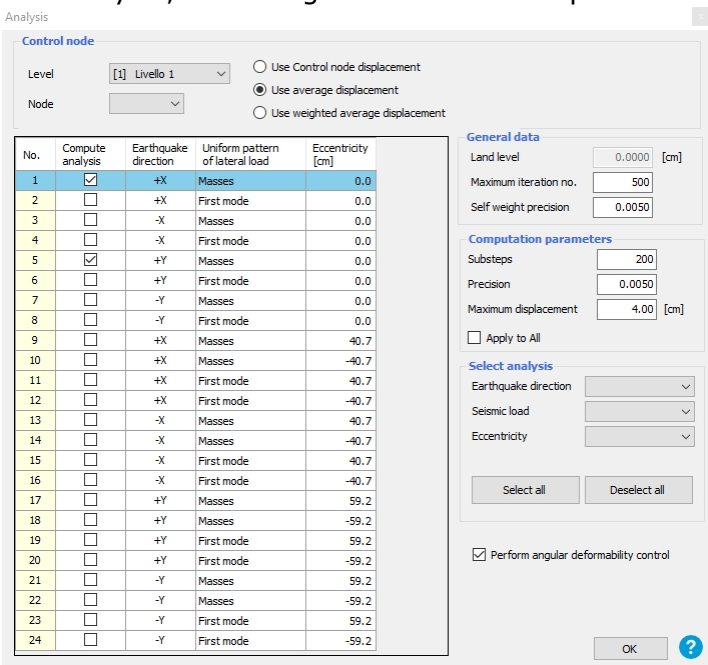
The graphics area is separated in two different areas, the plan on the left and on the right the view of the mesh of the selected wall.
The selection of a wall is possible through a simple click on the plan.



The cad commands (zoom, pan, etc.) are available in both areas plan (left) and mesh view (right)

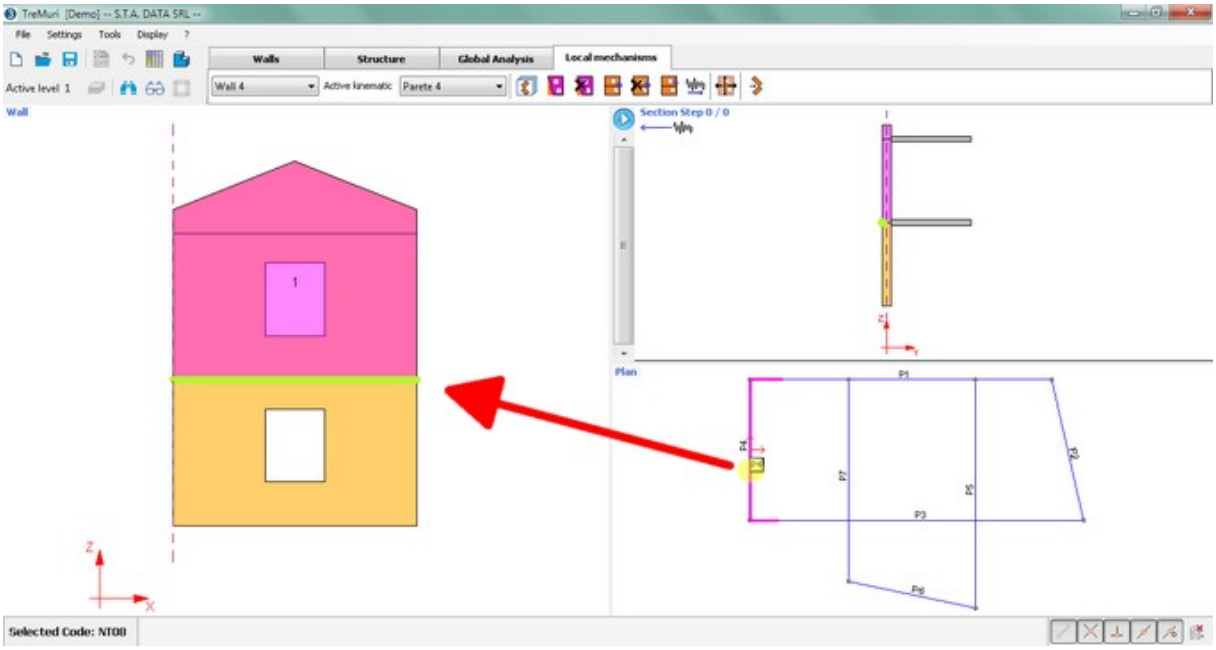
Improvements for the management of the analyses` parameters.

The calculation window shows on the table the "essential" data necessary to describe the analysis, on the right the calculation parameters.



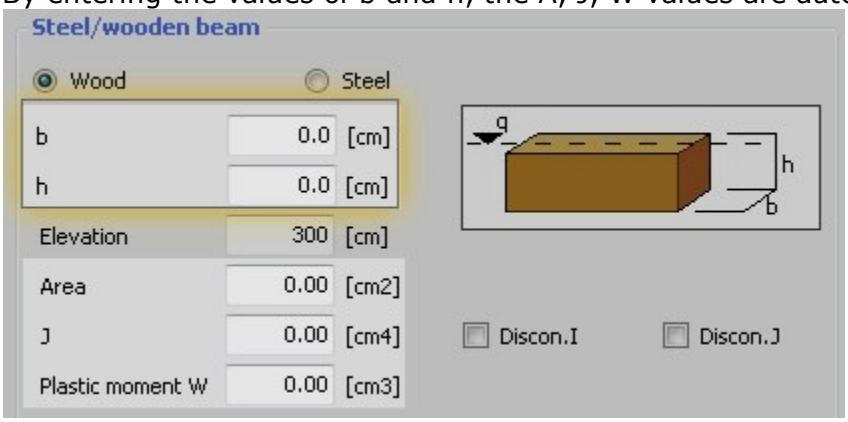
Better graphical interaction in the local mechanisms´ area

In order to load the view of the selected wall you can now click directly on the plan instead of using the drop down menu bar.
The cad area of the plant and view have been improved for a better graphical management of the input phase.



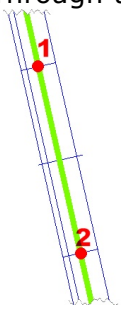
Filling of the properties of the wooden beam

By entering the values of b and h, the A, J, W values are automatically calculated.

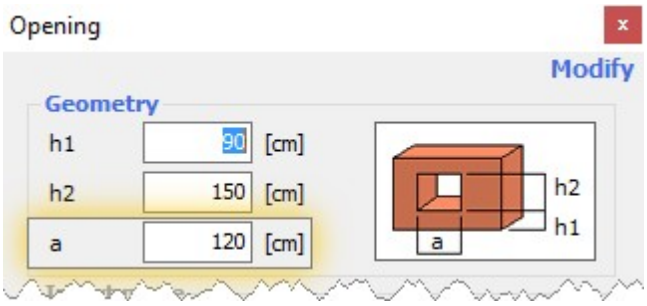


Insertion of an opening through two points.

Through this input mode it is not necessary to know the width of the opening.



There are necessary two clicks, one at the beginning [1] and one at the end of the opening [2]
The width is no required because it is calculated automatically from the distance between [1]-[2]



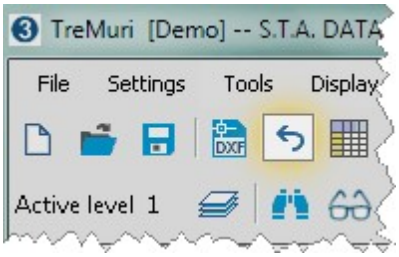
New snap utility

The snap are manageable in a total parametric way through the buttons bar down on the right.
They can be turned on or off according to the desired requirements.

New use of Pan and Zoom

The new zoom and pan commands are available directly on the rotation button of the mouse.

New “Undo” command

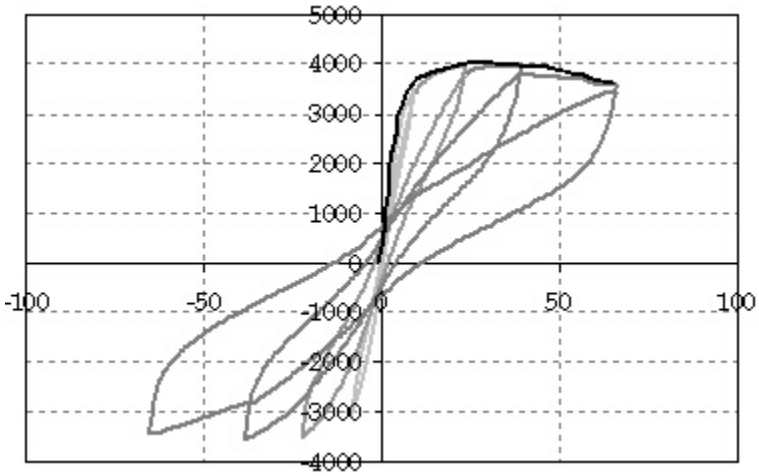


3 Structure Modelling

The code indications highlight the importance of carefully choosing the distribution of masses and rigidity (if necessary also considering the effect of non-structural elements) in order to obtain a structural model that is adequate for the global analysis. To that end, it is fundamental to do a preliminary knowledge phase, especially in the case of existing masonry structure, where the resistance structural system is not always immediately identifiable. This can be due to structural variations or different construction phases, change in the type of use for the building, and modifications to the original plans. The acquisition of this knowledge can make it clear what the resistant elements are (both for vertical actions as well as earthquake actions), as well as providing information about the characteristics of the materials.
A three-dimensional equivalent frame is the reference model, in which the walls are interconnected with horizontal partitions on the floors. In the specific case of a masonry structure, the wall can be schematized as a frame, in which the resistant elements (piers and spandrel beams) and the rigid nodes are assembled. The spandrel beams can be modelled only if they are adequately toothed by the walls, supported by structurally efficient architraves, and if possible a mechanism resistant to struts.
It is known that a less than perfect understanding of the positioning of the masses can lead to underestimation of the forces on the structures linked to the torsional effects. In fact, the increasing eccentricity in the center of the masses and the center of rigidity is that which exaggerates this aspect. Hence, code proposes consideration of accidental eccentricity to be applied to the center of the masses on every level of the structure. Accidental eccentricity is equal to ±5% of the maximum dimension of the level considered by the building in direction perpendicular to the seismic action.

3.1 Static Non-linear Analysis

Numerous computation and control measures, adopted in various countries with modern anti-seismic project legislation, propose a description of the structural response in terms of displacement, rather than forces, taking into account the greater sensitivity to damage based on imposed displacement. Italian code also provides a method that uses’non-linear static analysis.
In this context, non-linear static procedures play a central role, including the *Capacity Spectrum Method*, originally proposed by Freeman*et al.* 1975) and the’N2 Method (Fajifar 1999, 2000). These methodologies are simplified procedures in which the problem of evaluating the maximum expected response, consequent to’the occurrence of a determined seismic event, returns to the study of a non-linear system with a single grade of freedom equivalent to a model with n degrees of freedom, which represents the real structure ("Substitutive Structure Approach," Shibata and Sozen, 1976).
The characteristic that these procedures have in common is that of being based on the ’use of non-linear static analysis (*pushover)*to characterize the seismic-resistant system through *capacity curves:*: "static" analysis in that the external force is applied to the structure statically, and "non-linear" due to the behavioral model used for the structural resistance elements.
These curves are intended to represent the envelope of the hysteresis cycles produced during the seismic event and can be considered to be an indicator of the post-elastic behavior of the structure.

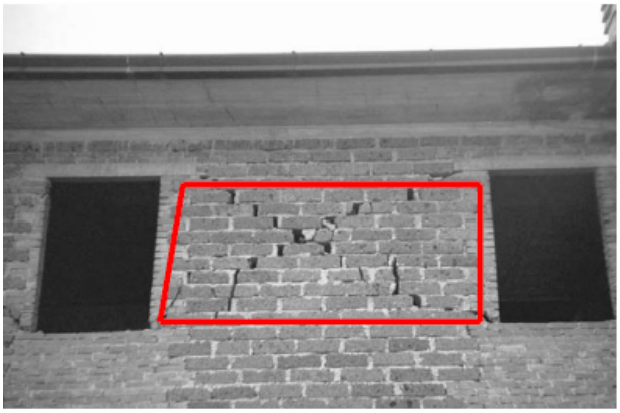


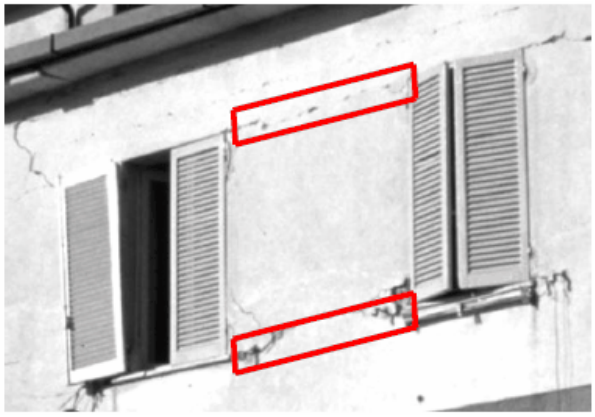
In this way, in the elastic analysis methods, the non-linear behavior is taken into account by introducing the structural factor, 'non-linear static analysis does not allow 'the structural response to evolve as each single element evolves in the non-linear field, providing information on the distribution of the anelasticity demand.
The curve obtained by the *pushover analysis* (which will then by transformed into a capacity curve, taking into account the system characteristics equivalent to grades of freedom) conventionally provides information on the'trend of the shear resulting at the base, with respect to the horizontal displacement of a control point on the structure. At each point on the curve, a specific damage state for the entire system can be linked,'and so it is possible to link determined displacement levels to the level of expected performance and the corresponding damage.
The curve is obtained by using *pushover analysis*, which predicts the 'assignment of a preset distribution of forces increasing in a static and monotonic manner. The distribution is kept unaltered even after the fail limit is reached. The analysis can also be conducted controlling for forces or for mixed force-displacement.
The load distribution applied is intended to represent the distribution of inertial forces induced by the 'seismic event. The profiles proposed are those in harmony with the first modal form, for masonry structures, more or less equivalent to those adopted for the 'linear static analysis, and that proportional to the mass. In particular, in the case of regular structures, the first distribution is chosen with the intention of better determining the structural response in the elastic field and secondly, in the non-linear field.
The "capacity" offered by the structure must then be determined, through the lens of a seismic check, with the "demand" requested by the external force, that is by a determined seismic event.
The energy dissipation effects, which offer an ulterior margin of resistance, which can not be explained using only linear elastic theory, are relevant in particular in the field of non-linear structural response: to take them into account the demand is reduced.
The expected response for the 'building, as a function of a determined action, is hence obtained through the identification of the *performance point* (whose coordinates in terms of spectrum displacement corresponds to d*max).
The maximum displacement value that can be offered by the building in a seismic event, is obtained in correspondence with the value of the shear that underwent a decline of 20% from the shear limit value. Based on the capacity curve of the real system defined in this way, it passes to the bilateral associated with the equivalent system; once found, the system period with one degree of freedom is identified, whose behavior permits the individuation of the seismic event's displacement demand.
From the observation of masonry buildings damaged by seismic events, two different damage mechanisms emerge:

Shear failure:

Compression-bending failure:

The practical observation of damages to existing structures, has led to the formulation of masonry micro-elements, elements which in their central part collect the shear behavior and in their peripheral parts collect the combined compressive and bending stress behavior.





From that observed above, the theoretical formulation of said macroelements emerges.

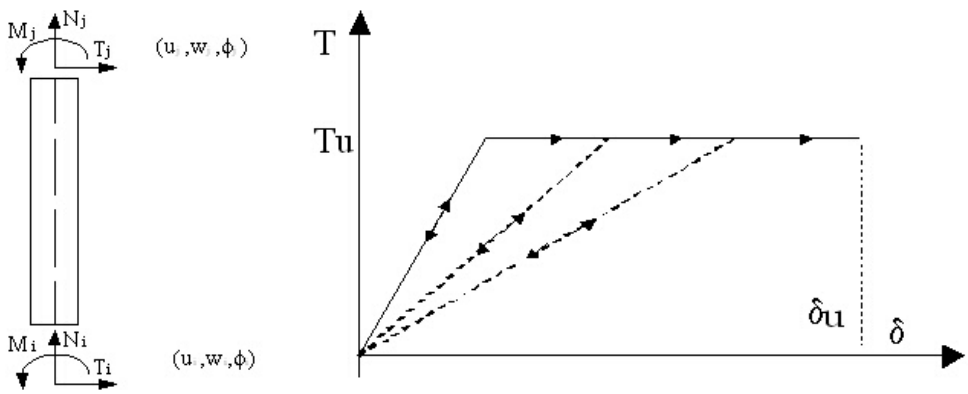
3.2 Masonry Macro-elements

A non-linear beam element model has been implemented in 3muri for modelling masonry piers and spandrels. Its main features are:

1) initial stiffness given by elastic (cracked) properties;
2) bilinear behaviour with maximum values of shear and bending moment as calculated in ultimate limit states;
3) redistribution of the internal forces according to the element equilibrium;
4) detection of damage limit states considering global and local damage parameters;
5) stiffness degradation in plastic range;
6) ductility control by definition of maximum drift (δu) based on the failure mechanism, according to the Italian seismic code and Eurocode 8:

$$\delta_m^{DL} = \frac{\Delta_m}{h_m} = \delta_u \begin{cases} 0.004 \text{ Shear} \\ 0.006 \text{ Compression-bending} \end{cases}$$

7) element expiration at ultimate drift without interruption of global analysis.



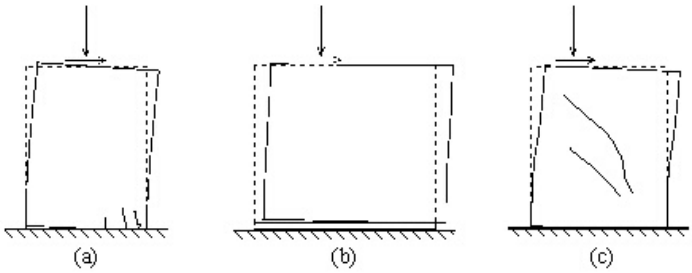
Non-linear beam degrading behavior

The elastic behaviour of this element is given by:

$$\begin{Bmatrix} T_i \\ N_i \\ M_i \\ T_j \\ N_j \\ M_j \end{Bmatrix} = \begin{bmatrix} \frac{12EJ}{h^3(1+\psi)} & 0 & -\frac{6EJ}{h^2(1+\psi)} & -\frac{12EJ}{h^3(1+\psi)} & 0 & -\frac{6EJ}{h^2(1+\psi)} \\ 0 & \frac{EA}{h} & 0 & 0 & -\frac{EA}{h} & 0 \\ -\frac{6EJ}{h^2(1+\psi)} & 0 & \frac{EJ(4+\psi)}{h(1+\psi)} & \frac{6EJ}{h^2(1+\psi)} & 0 & \frac{EJ(2-\psi)}{h(1+\psi)} \\ -\frac{12EJ}{h^3(1+\psi)} & 0 & \frac{6EJ}{h^2(1+\psi)} & \frac{12EJ}{h^3(1+\psi)} & 0 & \frac{6EJ}{h^2(1+\psi)} \\ 0 & -\frac{EA}{h} & 0 & 0 & \frac{EA}{h} & 0 \\ -\frac{6EJ}{h^2(1+\psi)} & 0 & \frac{EJ(2-\psi)}{h(1+\psi)} & \frac{6EJ}{h^2(1+\psi)} & 0 & \frac{EJ(4+\psi)}{h(1+\psi)} \end{bmatrix} \begin{Bmatrix} u_i \\ w_i \\ \phi_i \\ u_j \\ w_j \\ \phi_j \end{Bmatrix}$$

where $\psi = 24(1+\nu)\chi\left(\frac{r_i}{h}\right)^2 = 24(1+\frac{E-2G}{2G})1.2\frac{b^2}{12h^2} = 1.2\frac{E}{G}\frac{b^2}{h^2}$.

The non linear behavior is activated when one of the nodal generalized forces reaches its maximum value estimated according to minimum of the following strength criteria: flexural-rocking, shear-sliding or diagonal shear cracking.



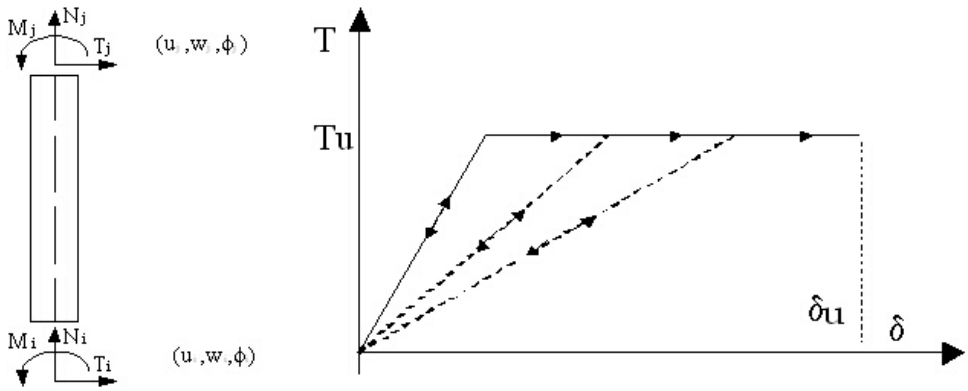
Masonry in-plane failure modes: flexural-rocking (a), shear-sliding (b) e diagonal-cracking shear (c) (Magenes et al., 2000)

----------OLD_TEXT----------
A non-linear beam element model has been implemented in 3muri for modelling masonry piers and spandrels. Its main features are:

1) initial stiffness given by elastic (cracked) properties;
2) bilinear behaviour with maximum values of shear and bending moment as calculated in ultimate limit states;
3) redistribution of the internal forces according to the element equilibrium;
4) detection of damage limit states considering global and local damage parameters;
5) stiffness degradation in plastic range;
6) ductility control by definition of maximum drift (δu) based on the failure mechanism, according to the Italian seismic code and Eurocode 8:

$$\delta_m^{DL} = \frac{\Delta_m}{h_m} = \delta_u \begin{cases} 0.004 \text{ Shear} \\ 0.006 \text{ Compression-bending} \end{cases}$$

7) element expiration at ultimate drift without interruption of global analysis.



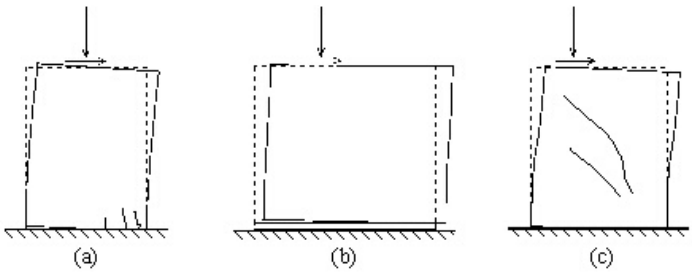
Non-linear beam degrading behavior

The elastic behaviour of this element is given by:

$$\begin{Bmatrix} T_i \\ N_i \\ M_i \\ T_j \\ N_j \\ M_j \end{Bmatrix} = \begin{bmatrix} \frac{12EJ}{h^3(1+\psi)} & 0 & -\frac{6EJ}{h^2(1+\psi)} & -\frac{12EJ}{h^3(1+\psi)} & 0 & -\frac{6EJ}{h^2(1+\psi)} \\ 0 & \frac{EA}{h} & 0 & 0 & -\frac{EA}{h} & 0 \\ -\frac{6EJ}{h^2(1+\psi)} & 0 & \frac{EJ(4+\psi)}{h(1+\psi)} & \frac{6EJ}{h^2(1+\psi)} & 0 & \frac{EJ(2-\psi)}{h(1+\psi)} \\ -\frac{12EJ}{h^3(1+\psi)} & 0 & \frac{6EJ}{h^2(1+\psi)} & \frac{12EJ}{h^3(1+\psi)} & 0 & \frac{6EJ}{h^2(1+\psi)} \\ 0 & -\frac{EA}{h} & 0 & 0 & \frac{EA}{h} & 0 \\ -\frac{6EJ}{h^2(1+\psi)} & 0 & \frac{EJ(2-\psi)}{h(1+\psi)} & \frac{6EJ}{h^2(1+\psi)} & 0 & \frac{EJ(4+\psi)}{h(1+\psi)} \end{bmatrix} \begin{Bmatrix} u_i \\ w_i \\ \phi_i \\ u_j \\ w_j \\ \phi_j \end{Bmatrix}$$

where $\psi = 24(1+\nu)\chi\left(\frac{r_i}{h}\right)^2 = 24(1+\frac{E-2G}{2G})1.2\frac{b^2}{12h^2} = 1.2\frac{E}{G}\frac{b^2}{h^2}$.

The non linear behavior is activated when one of the nodal generalized forces reaches its maximum value estimated according to minimum of the following strength criteria: flexural-rocking, shear-sliding or diagonal shear cracking.



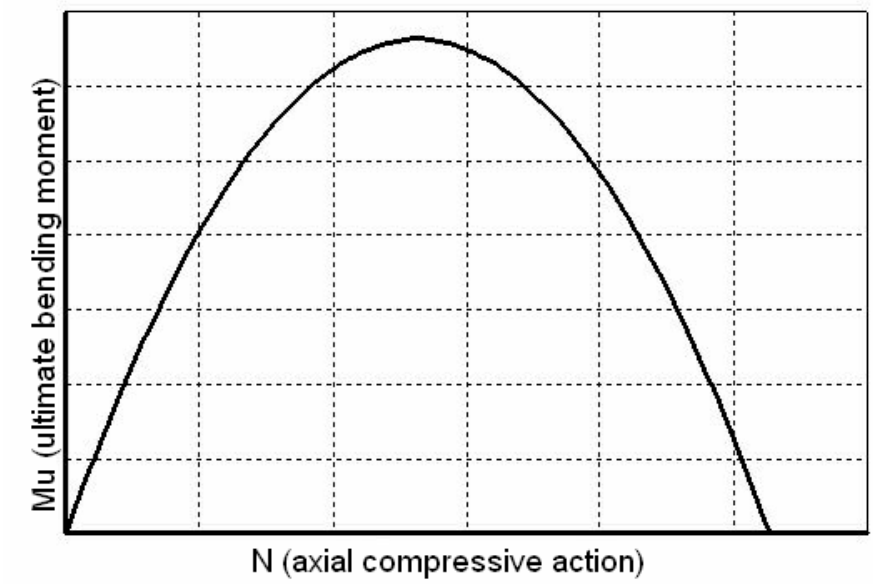
Masonry in-plane failure modes: flexural-rocking (a), shear-sliding (b) e diagonal-cracking shear (c) (Magenes et al., 2000)

3.2.1 bending: ROCKING BEHAVIOR

The ultimate bending moment is defined as

$$M_u = \frac{l^2 t \sigma_0}{2}\left(1 - \frac{\sigma_0}{0.85 f_m}\right) = \frac{Nl}{2}\left(1 - \frac{N}{N_u}\right).$$

Where l is the width of the panel, t is the thickness, N is the axial compressive action (assumed positive in compression), so is the normal compressive stress on the whole area (so=N/lt) and fm is the average resistance in compression of the masonry. This approach is based on a no-traction material where a non linear reallocation of the stress is performed (rectangular stress-block with factor =0.85)
In existing building the average resistance fm is to be divided by the "confidence factor" FC according to the structural knowledge level.



Strength criterion in bending-rocking

According to the element definition the global equilibrium must be satisfied: if the actual moment is reduced to ultimate bending moment value, the shear must be recalculated as

$$V_i = -V_j = \frac{M_i + M_j}{h}$$

3.2.2 Shear: Mohr-Coulomb criterion

The shear failure, according to Mohr-Coulomb criterion, defines an ultimate shear as

$$V_u = l't f_v = l't\left(f_{v0} + \mu\sigma_n\right) = l't f_{v0} + \mu N$$

Where l' is the length of the compressed section of the panel, t is the thickness, f_v is the shear resistance of the masonry, f_{v0} is the shear resistance of the masonry without compression, μ is the friction coefficient (usually 0.4) and σ_n is the normal average compressive stress, referred to the effective area.
In non linear static analysis according to the Italian code, the shear resistance f_v is to be divided by the "confidence factor" FC according to the structural knowledge level.
The use of the effective compressed length l' is due to the partialization of the section that occur when the eccentricity $e = |M|/N$ exceeds the limit value of l /6 in one of the ends (if e < l /6 all the points of the section are compressed).
In general the length l' can be expressed as

$$l' = 3\left(\frac{l}{2} - e\right) = 3\left(\frac{l}{2} - \frac{|M|}{N}\right)$$

If the current shear value V exceeds the ultimate value Vu it must be reduced but changing the shear value means to reduce the current bending moment values of Mi and Mj to grant the equilibrium according to the (2). A reduction of the moments causes a reduction of the eccentricity e and so a reduction of l': a limit value of l' has to be expressed to be consistent to ultimate shear and moment values.
According to the actual forces and the constrains the generic bending moment M can be expressed as αVh where α is a coefficient (α=0.5 for a double-bending constrain, α=1 for a cantilever) so:

$$l' = 3\left(\frac{l}{2} - \frac{\alpha V h}{N}\right)$$

Under the hypothesis that any possible reduction of the moments, caused by a shear reduction, doesn't change the static system, the ratio of the moments M_i and M_j must be unchanged: so α can be constant and expressed as

$$\alpha = \frac{M_{\max}}{M_{\max} + M_{\min}}$$

where M_{max} is the maximum absolute value between M_i and M_j ; note than α cannot be negative.

The shear resistance, according to Eurocodes and Italian codes, can be expressed as:

$$V_R = (f_{vo} + 0.4\sigma_o)l't = f_{vo}l't + 0.4N$$

Under the limit condition $V=V_R$

$$V_R = 3\left(\frac{l}{2} - \frac{\alpha V_R h}{N}\right) f_{vo}t + 0.4N = 1.5 f_{vo}lt + 0.4N - 3\alpha f_{vo}ht\frac{V_R}{N}$$

and then

$$V_R = \tfrac{1}{2}N\frac{3f_{vo}lt + 0.8N}{3\alpha f_{vo}ht + N}$$

l' can be expressed as:

$$l'_R = \frac{3}{2}\left(l - \frac{3\alpha f_{vo}lt + 0.8\alpha N}{3\alpha f_{vo}ht + N}h\right)$$

This is the value of the of the actual compressed section of the panel under the limit condition of shear

failure; furthermore must be $\frac{N}{0.85 f_m t} < l'_R \le l$; where the extremes of the interval are the conditions of the whole section compressed and the limit state for bending (the stress block is completed in the compressed section part).

If the previous inequality is not satisfied the value of *l'* is to be assumed as the correspondent extreme of the interval .

In addition to the Mohr-Coulomb resistance, the value of the shear tension f_v must not exceed the limit value of $f_{v,lim}$:

$$f_v = \frac{T}{l't} \le f_{v,\lim}$$

If it exceeds the failure shear value can be fixed as

$$V_{\lim} = f_{v,\lim} l't$$

The effective compressed length l' has to be consistent with the value of V_{lim} and so may be different from l'_R: if the failure occurs for the an exceeding value of the limit shear tension, the element shear has to be reduced and this causes the reduction of the moments to grant the global equilibrium of the panel according to *α.*

The limit compressed length l'_{lim} , consistent with this failure mode, can be evaluated imposing $V= V_{lim}$.

$$V_{\lim} = \frac{3}{2}N\left(\frac{f_{v,\lim}lt}{3\alpha f_{v,\lim}ht + N}\right)$$

And so l'_{lim}

$$l'_{\lim} = \frac{3}{2}\left(l - \frac{3\alpha f_{v,\lim}lt}{3\alpha f_{v,\lim}ht + N}h\right)$$

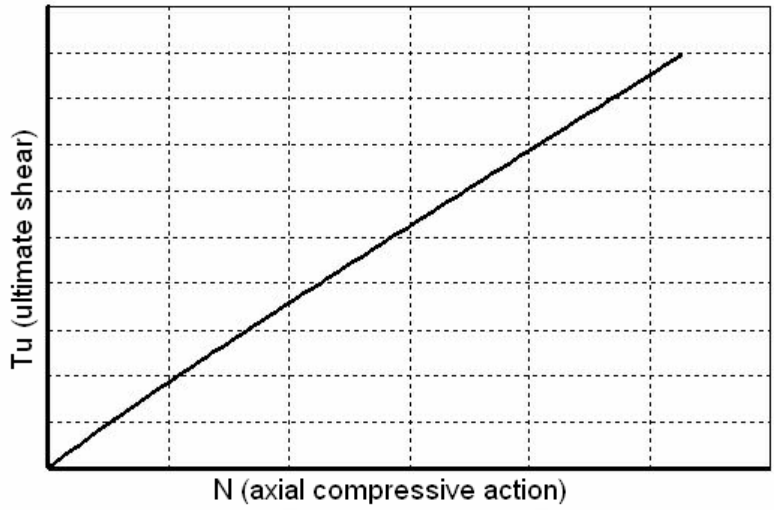
As for l'_R also l'_{lim} must be $\frac{N}{0.85 f_m t} < l'_{lim} \leq l$

Finally the limit shear V_u is the minimum between V_{lim} and V_R:

$$V \leq V_u = \min(V_R, V_{lim})$$

In case of the current shear overcomes the limit shear V_u, it is reduced to V_u and also the moments have to be reduced according to grant the same static scheme:

$$M_{max} = T_u \cdot \alpha \cdot h \qquad T \equiv T_u$$
$$M_{min} = T_u \cdot (1-\alpha) \cdot h$$



Mohr-Coulomb criterion for shear resistance

3.2.3 Shear: Turnšek Cacovic criterion

According to Italian code, only for existing building, the shear failure can be computed according to Turnšek and Cacovic criterion; the ultimate shear is defined as:

$$V_u = lt\frac{1.5\tau_o}{b}\sqrt{1+\frac{\sigma_o}{1.5\tau_o}} = lt\frac{f_t}{b}\sqrt{1+\frac{\sigma_o}{f_t}} = lt\frac{1.5\tau_o}{b}\sqrt{1+\frac{N}{1.5\tau_o lt}}$$

Where f_t and τ_0 are the design value of tension resistance in diagonal cracking of masonry and its shear value, b is a coefficient defined according to the ratio of height and length of the wall (b= h/l but 1≤ b ≤1.5).

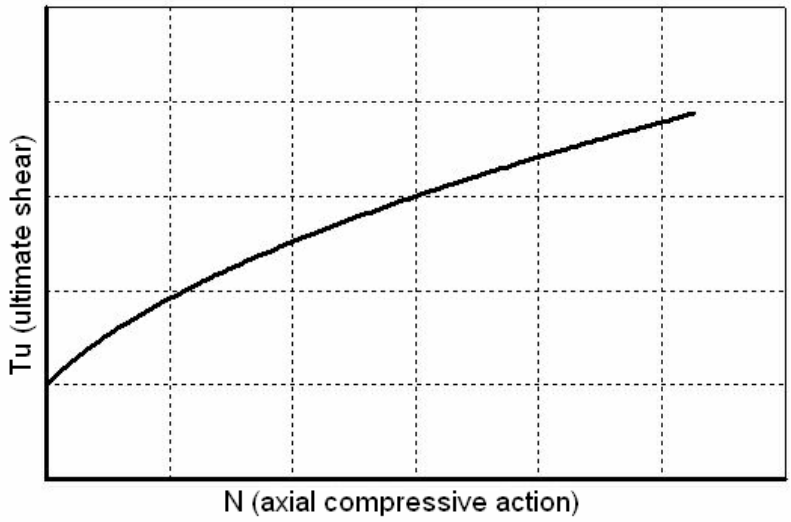


figure 5: Turnšek and Cacovic shear strength criterion

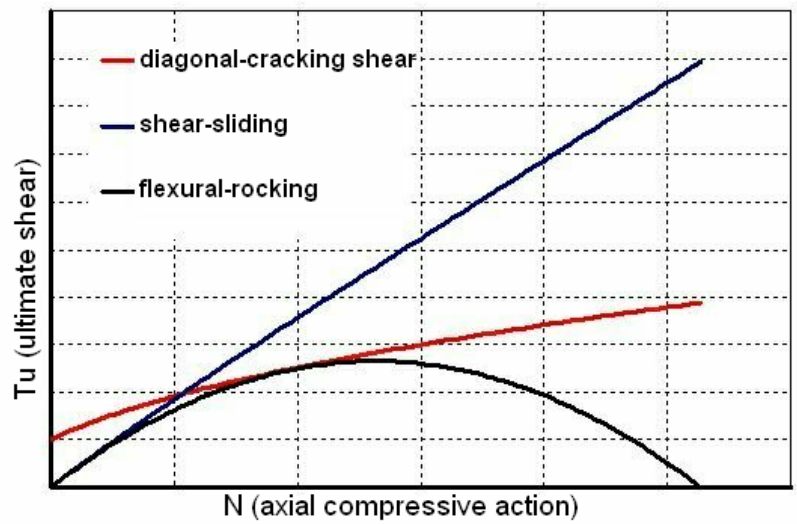


figure 6: Strength criteria comparison

3.2.4 masonry beams (lintels)

The previous strength criteria can be used only with effective axial compression, this is usually granted in piers but not for lintel where the shear resistance can be assumed as:

$$V_{u,\text{lintel}} = htf_{v0}$$

Where h is the height of the section of the panel, t is the thickness, f_{v0} is the shear resistance of the masonry without compression.
According to this the maximum bending moment is :

$$M_{u,\text{lintel}} = \frac{hH_p}{2}\left[1 - \frac{H_p}{0.85 f_h ht}\right]$$

Where H_P is the minimum between the tension resistance of the stretched interposed element inside the lintel (for example a tie-road or tie-beam) and $0.4 f_h ht$ where f_h the compression resistance of the masonry in the horizontal direction in the plane of the wall.
The balance will be provided similar to the one exhibited by the previous criteria.

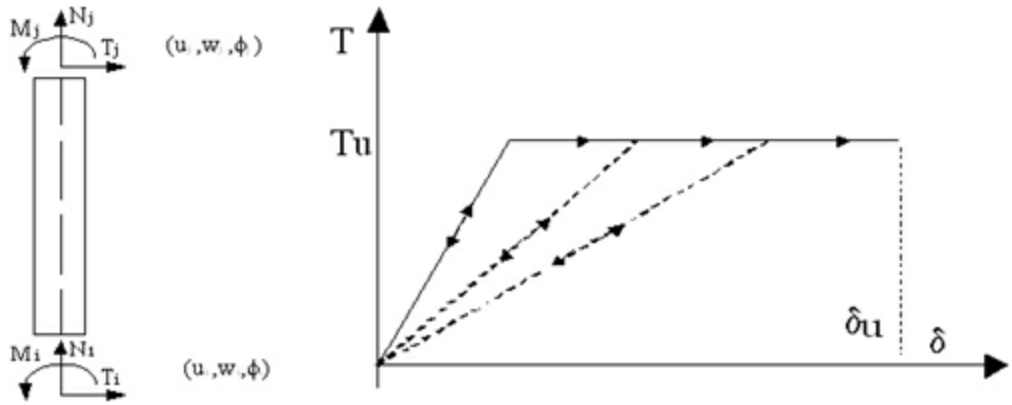
3.2.5 Reinforced masonry beams

The non-linear beam element aimed at modeling in reinforced masonry panels (or reinforced by elements made of FRP or other material) is based on a formulation similar to that used for ordinary masonry panels in which the adopted resistance criteria are appropriately modified in accordance with the recommendations proposed in the regulations, with regard to the reinforced masonry, and certain information contained in the document CNT- DT 200/2004 (Italian Regulations) (Instructions for the design, execution and monitoring in interventions to static reinforcements through the use of fiber-reinforced composites - Materials, reinforced concrete structures and prestressed concrete, masonry structures), regarding the case of using FRP reinforcement.

By analogy, therefore, to the nonlinear beam element formulated for ordinary masonry panels, the main particularities of this element are:

1) Initial stiffness according to the elastic characteristics (slotted) of the material: these contributions are calculated solely by reference to the contribution of the masonry, considering negligible - with regard to rigidity - that associated with the reinforcement;
2) Bi-linear behavior with maximum values of shear and time consistent with the values of the ultimate limit state;
3) Redistribution of internal stresses in the element such as to ensure balance;
4) Setting the state of damage according to the global and local parameters;
5) Degradation of stiffness in the plastic branch;
6) Control of ductility by defining maximum drift (du) differentiated according to the provisions of the current regulations depending on the damage mechanism acting on the panel. In particular, in accordance to the current standards it is taken on a limit value of 0.6% in the case of shear failure and at 1.2% in the case of breakage for buckling.
7) Elimination of the element, to achieve the s.l.u. analysis without interruption.

It is specified that it is neglected the contribution of resistance and rigidity out of the element plan.



Degrees of freedom (and the corresponding generalized stress characteristics) and non-linear behavior of the non-linear beam element

As specified in point 1), the stiffness matrix that governs the elastic behavior of that element has the same formulation as that of ordinary masonry panels.

The nonlinear behavior is activated when a nodal force reaches its maximum value defined as the minimum of the criterion adopted for the response to buckling and shear (accounted for using the shear-scrolling and shear-diagonal cracking as best explained later) appropriately modified to take account of the reinforcement presence. In addition there is a further associated control to breaking for pure compression or traction element.

In particular it is provided the insertion of two types of reinforcement: vertical (which may be concentrated and / or spread) and transverse. It must however be stated that the possibility to introduce the transverse reinforcement is subject to the presence of the vertical one: this because, as better specified in what follows, the criteria adopted for the evaluation of shear resistance are based on the assumption of truss behavior.

The variation in resistance criteria consequent the adoption of reinforced masonry is applied only to the default vertical resistant panels (piers).

It 'should be specified that in the case of the spandrel beams, since these elements are rotated by 90 ° compared to piers, by reference to the input data assigned to the reinforcement, the transverse reinforcement are those adopted to compute the response to the increased resistance to bending stress (the number of bars is counted starting from the spandrel beam width and spacing of the reinforcements, however, by providing a minimum of a bar at each end of the element when the size of the element do not result compatible with the inserted spacing).

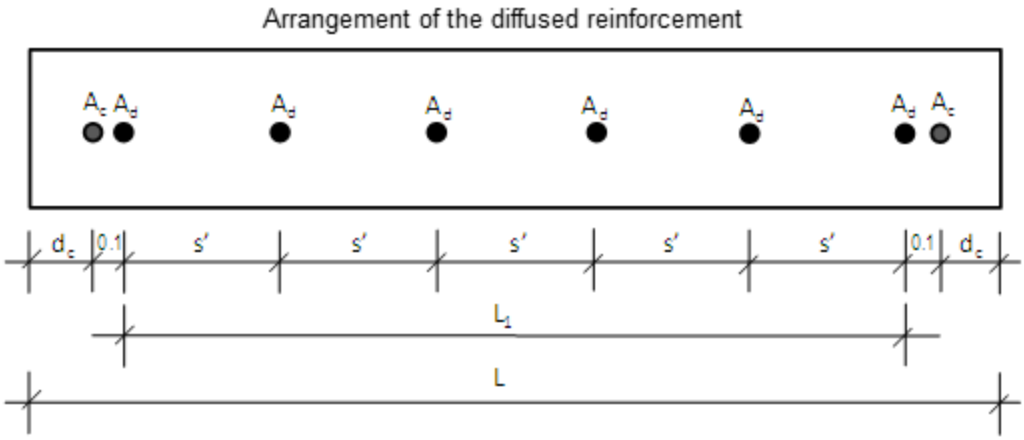
Clarifications on the treatment of the data provided as input in the case of reinforcements (in relation to the position taken for the reinforcements)
It recalls how in the vertical reinforcement mode, the input data results:

- Ac [m2] : total vertical reinforcement area concentrated to the extreme of the element; this reinforcement is assumed by default arranged symmetrically at both ends of the panel;
- dc [m] : distance from the centroid of the vertical reinforcement concentrated from the extreme of the element;
- Ad [m2]: widespread vertical reinforcement area; It means the total area of the individual reinforcement (including the one disposed to intrados and extrados of the panel) then placed at intervals sd;
- sd [m] : spacing of diffuse vertical reinforcement.

In the vertical reinforcement concentrated case the data provided as input are sufficient to univocally define the position of the reinforcements inside the panel.
In diffuse vertical reinforcement case, the position and number of the reinforcements inside the panel is calculated on the basis of the following criteria:

- is assumed a distance between the first bar (or group of bars) of the diffused reinforcement and the free edge of the panel or the concentrated reinforcement (where present) equal to 0.1 m;
- on the basis of the assumption of the previous and sd step provided as input to calculate the n' number of bars that actually can be arranged inside the panel [n'= 1 + L1/s con L1 =L-2*0.1-2*dc), assuming a symmetrical distribution with respect to the barycentric axis of the wall panel. Please note that the number of bars is approximated to the whole match (rounded up or down when the first decimal place is greater than or less than 5);
- based on the number of bars calculated according to the criterion shown in the previous step, it is updated where necessary the step of reinforcements (assuming s' equal to L1/(n'-1)) .

The following figure shows the arrangement of the reinforcements resulting on the basis of the above criteria.



Resistance limits for the pure compression/traction
The resistance to pure compression limit is calculated by adding the one offered of masonry (obtained by applying the coefficient equal to 0.85 to the resistance to compression of the masonry calculation) that associated with the present vertical reinforcements.
The pure tensile strength limit is calculated only by reference to the contribution offered by the vertical reinforcements.

The number and the position of the vertical reinforcements considered for the N-M interaction domain evaluation are computed according to the criteria illustrated above.

In particular, in the case of reinforced masonry, according to these directions, the N-M interaction domain is computed, assuming the conservation of planar sections, assuming for the masonry a rectangular compressions diagram, with depth 0.8 x, where x represents the neutral axis depth (calculated with respect to the compression edge), and solicitation of 0.85 fd (with fd the design compression strength of masonry). Furthermore, the maximum deformations are considered equal to emu = 0.0035 for the compressed masonry and esu = 0,01 for the tense steel.

In the case of using FRP reinforcement, the legislation proposes absolutely similar criteria with the following specifications:
- compared to the range proposed in the CNR-CT 200/2004 document for the depth to

be taken for the diagram of masonry compressive stresses (equal to 0.6 ÷ 0.8 x), it is taken as value equal to 0.8 similarly to the case of reinforced masonry;

- the value of the ultimate strain assumed for the reinforcement fiber-reinforced composite is assumed to be equal to the value supplied by the user input. This value can be calculated according to the criteria proposed in the document CNT- DT 200/2004 at § 5.3.2.

Resistance limits for shear

In the case of limit strength calculation associated with the shear response it is first of all appropriate to make the following clarification.
The choice of the shear strength criterion adopted, ie whether a policy to Mohr-Coulomb or a criterion to Turnšek and Cacovic (according to what is introduced in the chapter devoted to the illustration of the nonlinear beam element formulation for ordinary masonry panels), it is a direct consequence of the defined parameters and assigned for the type of characterizing masonry walls reinforced panel. Depending on the parameters assigned to the wall (if not - in the case of the criterion to Turnšek and Cacovic for existing masonry - fvm0 or - in the case of a criterion to the Mohr-Coulomb as proposed in the norm in the case of newly built walls), and then the shear strength criterion assumed, in what follows the criteria adopted are illustrated in the case of the presence of reinforcement.

- *Case I – Mohr- Coulomb criterion*

In this case, for the calculation of the reinforced masonry limit resistance is set reference to the criteria illustrated in current regulations.
In particular, the shear strength (Vt) is calculated as the sum of the contributions of the masonry (Vt, M) and reinforcement (V t, S), according to the following relationships:

$$V_t = V_{t,M} + V_{t,S}$$
$$V_{t,M} = d\, t\, f_{vd}$$

where:
d is the distance between the compression edge and the centroid tense reinforcement;
t is the wall thickness;
fvd It is assumed to fvk calculating the normal average tension (indicated by sn in the paragraph above) on the gross width section d (sn = P/dt);

$$V_{t,S} = (0{,}6\, d\, A_{sw}\, f_{yd}) / s$$

where:
d is the distance between the compression edge and the centroid tense reinforcement;
Asw is the shear reinforcement area disposed in a parallel direction to the shear force, with spacing s measured orthogonally to the direction of the shear force;
fyd is the calculation of the steel yield strength;
s is the distance between the reinforcing layers.

It must also be verified that the acting shear does not exceed the following value:

$$V_{t,c} = 0{,}3\, f_d\, t\, d$$

where:
t is the wall thickness
fd is the design masonry compressive strength.
As previously introduced, such resistance criterion requires a truss beam response of the panel whose operation can only be guaranteed by the presence of appropriate vertical reinforcing bars disposed in place.
In the case of using FRP reinforcement, according to what is proposed in the document

CNT- DT 200/2004, it has adopted a very similar approach to the one above, for reinforced masonry, taking care to replace the calculation of yield stress' steel (fyd) with the FRP (ffd) of the reinforcement project resistance, defined as the minimum between the rupture tension of the composite and the tension in the composite at which there is the debonding from the masonry.

- *Case II – Turnšek and Cacovic criterion*

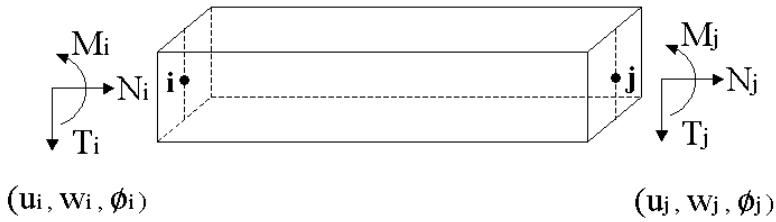
In the event that the values assigned as input for the masonry correspond to a criterion to Turnšek and Cacovic, Technical Rules of Construction (neither the CNT- DT 200/2004 document regarding reinforcement using FRP) does not propose any specific policy to keep account of the strength increase associated with the reinforcing.
Among the proposals in the literature, it was decided to adopt the approach proposed in Da Porto et al. 2009 (From Porto, F .; Modena, C .; Mosele, F. "cyclical behavior in the plane of a reinforced masonry system," Building in Brick, n. 130 July-August 2009, pp. 54-61.). These authors, on the basis of a calibration starting from the experimental test results of reinforced masonry panels, aiming to evaluate the shear strength increase provided by the presence of the reinforcement with a similar factor V t, S, introduced in the case of ' adoption of a policy to Mohr-Coulomb.
Thus ultimately the value of the shear strength is calculated according to the following expression:

$$V_t == lt\frac{1.5\tau_o}{b}\sqrt{1+\frac{N}{1.5\tau_o lt}}+0.6\frac{dA_{sw}f_{yd}}{s}$$

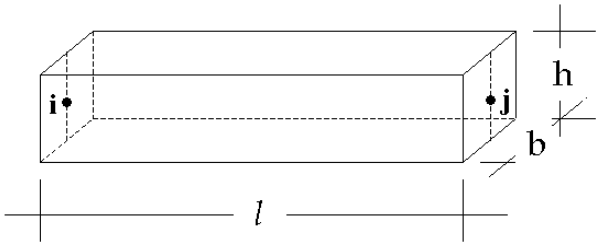
In which, for the meaning of symbols, reference is made to what previously introduced.

3.3 Non-linear R.C. element

A non-linear R.C. element is an element with six degrees of liberty, with limited resistance and elastic-perfectly plastic behavior.



Cinematic variables and forces characteristics for the R.C. beam element



Geometric measurements of the beam: Width (b) and height (h) of the section, and length (l) of the element

For each element, the linear elastic behavior is determined directly by the computation of the shear and bending rigidity contributions. These are computed based on the mechanical and geometric properties (Young elastic module E, shear module G, and the geometry of the beam): when computing these factors, reference is made only to the section in cement, ignoring the contribution of the reinforcement, while taking into account the reduction to the rigidity due to cracking, The various contributions are assembled in the elastic rigidity matrix for the individual element.

$$\begin{Bmatrix} T_i \\ N_i \\ M_i \\ T_j \\ N_j \\ M_j \end{Bmatrix} = \begin{bmatrix} \frac{12EJ}{l^3(1+\psi)} & 0 & \frac{6EJ}{l^2(1+\psi)} & -\frac{12EJ}{l^3(1+\psi)} & 0 & \frac{6EJ}{l^2(1+\psi)} \\ 0 & \frac{EA}{l} & 0 & 0 & -\frac{EA}{l} & 0 \\ \frac{6EJ}{l^2(1+\psi)} & 0 & \frac{EJ(4+\psi)}{l(1+\psi)} & -\frac{6EJ}{l^2(1+\psi)} & 0 & \frac{EJ(2-\psi)}{l(1+\psi)} \\ -\frac{12EJ}{l^3(1+\psi)} & 0 & -\frac{6EJ}{l^2(1+\psi)} & \frac{12EJ}{l^3(1+\psi)} & 0 & -\frac{6EJ}{l^2(1+\psi)} \\ 0 & -\frac{EA}{l} & 0 & 0 & \frac{EA}{l} & 0 \\ \frac{6EJ}{l^2(1+\psi)} & 0 & \frac{EJ(2-\psi)}{l(1+\psi)} & -\frac{6EJ}{l^2(1+\psi)} & 0 & \frac{EJ(4+\psi)}{l(1+\psi)} \end{bmatrix} \begin{Bmatrix} u_i \\ w_i \\ \phi_i \\ u_j \\ w_j \\ \phi_j \end{Bmatrix}$$

with

$$\psi = 24(1+\nu)\chi\left(\frac{r_i}{l}\right)^2 = 24(1+\frac{E-2G}{2G})1.2\frac{b^2}{12l^2} = 1.2\frac{E}{G}\frac{b^2}{l^2}$$

Elastic rigidity matrix of the R.C. beam element

The resistance limits, relative to the failure mechanisms in consideration coincide with the last value. This is because the elastic-perfectly plastic behavior hypothesis is in effect, without hardening.

Preliminary observations:

Two points from Ordinance 3274/03 and subsequent modifications and supplements are listed below. These are intended to clarify and assist with the choices made in the modelling area for these elements

From "Point 8.1.5.4 Non-linear static analysis - OPCM 3274":
...Masonry panels are characterized by bilinear elastic-perfectly plastic behavior, with resistance equivalent to the elastic limit and displacement to the elastic limit. The last is defined by the bending or shear response, in points 8.2.2 and 8.3.2. Linear R.C. elements (tie beams, coupling beams) are characterized by bilinear elastic-perfectly plastic behavior, with resistance equivalent to the elastic limit and displacement to the elastic limit. The last is defined by the bending or shear response...

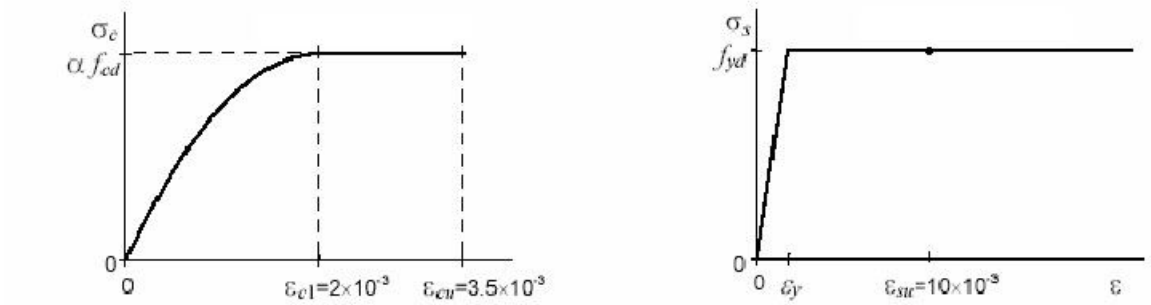
From "Point 8.5 Mixed structures with walls in ordinary or reinforced masonry - OPCM 3274":
In the area of masonry constructions, it is permitted to use structure with diverse technologies to support vertical loads, as long as the resistance to seismic action is entrusted entirely to elements of the same technology. In the case in which resistance is entrusted entirely to masonry walls, the requirements indicated above must be respected for the walls. In the case that the structural resistance is entrusted to other technologies (for example R.C. walls), the project design rules found in the associated chapters of the code must be followed. In the case that it is considered necessary to examine the combination of the masonry walls with the systems of different technology for resistance to seismic events, it must be verified using non-linear analysis methods (static or dynamic).

3.3.1 Resistance Criteria

Resistance mechanisms that are considered are: ductile bending (with or without normal forces) for each of the beam ends with the consequent formation of a plastic hinge and fragile to shears, in conformance with the criteria found in the code.

In addition, simple compression collapse limits are also taken into account (Checks on Safety Max Limits...the standard force must be less than that calculated for centered compression with an increase of 25% of the coefficient γc) and when thetraction limits for the reinforcement are exceeded.

Constituent link assumed for base materials steel and concrete.



Constituent link for base materials concrete and steel

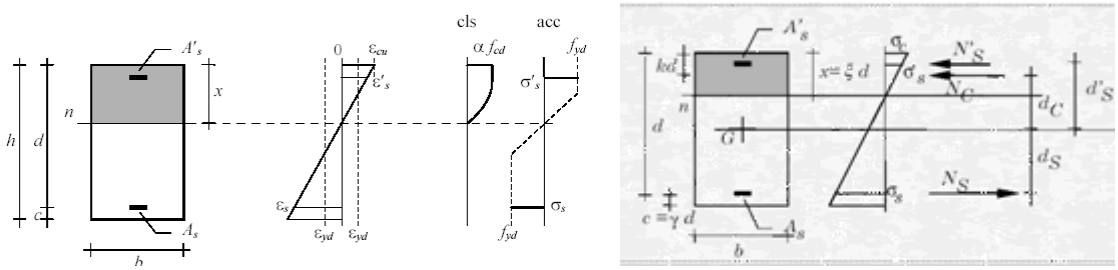
3.3.2 Bending Mechanism

In accordance with point 5.4.1 and the relative specifications for existing buildings in chapter 11 of Ordinance 3274/03 and subsequent modifications and supplements, the check compares the values calculated for the moments with those calculated for resistance (limit values) on the basis of actually existent bending reinforcement.

The M-N domain can be constructed by assigning a failure deformation and

determining the deformation diagram. Then, the tension diagram is determined using the constituent links. Finally, the results of compression and traction are calculated N_C , $N_{S'}$, N_S :

$N_S = \sigma s\ As$
$N_{S'} = \sigma s'\ As'$
$N_C = \beta\xi\alpha fc\ bd$



Deformation limit diagram and corresponding tension diagrams

These provide the equilibrium at transfer (a) and rotation (computed with respect to the geometric center of mass of the section):

$N = N_C + N_S + N_{S'}$ (a)
$M = N_C\ d_C + N_S\ d_S + N_{S'}\ d_{S'}$ (b)

Coordinates N and M correspond with a failure deformation and identify a point in the limit domain on the N-M plane.

Computation of section rotation and collapse

Calculation of section rotation with respect to the cord, to then be compared with collapse rotation, is done with reference to the definition found at point 11.3.2.1 of Ordinance 3274/03 and subsequent modifications and supplements:

"Deformative capacity is defined with reference to rotation ("rotation with respect to the cord") θ in the end section with respect to the conjunction line. This with the zero moment section at a distance equal to the span LV=M/V. This rotation is also equal to the relative displacement for the two sections divided by the span."

Calculation of the collapse rotation is done according to Annex 11.A (Ordinance 3274/03 and subsequent modifications and supplements).

"Rotation capacity with respect to the cord in collapse conditions θ_u can be evaluated using direct experimentation, numeric modelling considering the contributions of concrete, steel and adherence, or using the following formulas:

$$\theta_u = \frac{1}{\gamma_{el}} 0{,}016 \cdot (0{,}3^{\nu}) \left[\frac{\max(0{,}01;\omega')}{\max(0{,}01;\omega)} f_c \right]^{0.225} \left(\frac{L_V}{h} \right)^{0.35} 25^{\left(\omega_{\beta_{2x}} \frac{f_{yw}}{f_c} \right)} (1{,}25^{100\rho_d}) \qquad (11.A.1)$$

where γel=1.5 for primary elements and 1.0 for secondary elements (as defined in point 4.3.2 of the code), h is the height of the section, $\nu = N/(A_c f_c)$ is the normalized axial strain of the compression agent on the entirety of section Ac, $\omega = A_s f_y/(bhf_c)$ and mechanical percentages of longitudinal reinforcement in traction and compression (b,

h = base and height of the section), respectively. For the walls, all of the core longitudinal reinforcement should be included in the traction percentage. fc, fy, and fyw are the compression resistance of the concrete and the steel yield resistance, longitudinal and transversal. This is obtained as the average of the tests performed on site. If necessary, these can be corrected based on additional information, divided for confidence level in relation to the knowledge level attained, $\rho_{sx} = A_{sx} / b_w s_h$ the percentage of transversal reinforcement (sh=distance between centers of the stirrups in the critical zone), ρ_d the percentage of diagonal reinforcement in all directions, α is an efficiency factor given by:

$$\alpha = \left(1 - \frac{s_h}{2b_o}\right)\left(1 - \frac{s_h}{2h_o}\right)\left(1 - \frac{\sum b_i^2}{6h_o b_o}\right) \qquad (11.A.2)$$

(bo and ho) dimensions of the nucleus, bi distances of the longitudinal rebars held by tie-bars or stirrups found in the perimeter).

For the walls, or in the case of hardening steel the value given by the expression (11.A.1) must be divided by 1.6.
For elements that do not have adequate anti-seismic details, the value given by the expression (11.A.1) must be multiplied for 0.85.
In the presence of plain rebars and insufficient anchorage conditions, the value given by the expression (11.A.1) must be multiplied by 0.575."

Please note that calculation of the collapse rotation is done with exclusive reference to primary elements (as defined in 4.3.2 of Ordinance 3274/03 and subsequent modifications and supplements), as a precautionary measure. For this reason, coefficient γc is assumed to be equal to 1.5.

3.3.3 Shear mechanism

To check the last limit state for shearing forces, mono-dimensional elements with longitudinal reinforcement.

3.3.3.1 Elements without shear reinforcement

The use of elements without shear resistant transversal reinforcement is allowed for slabs, plates and other structures with analogous behavior, provided these elements have sufficient capacity to share the transversal loads.

3.3.3.1.1 Conglomerate check

The shear computation should not exceed the value that determines the formation of the oblique cracks, with reference to the computation traction resistance f_{ctd}. Also taking into account, in addition to the load effects, the coercive states that favor formation of cracks.

3.3.3.1.2 Longitudinal reinforcement check

The check transfers the diagram of the bending moment along the longitudinal axis, in the direction that creates an increase in the absolute value of the bending moment.
The checks can be performed respecting the condition:

$$V_{Sdu} \leq 0{,}25\, f_{ctd} \cdot r\,(1 + 50\,\rho_l) \cdot b_w \cdot d \cdot \delta$$

the symbols have the following meanings:

Vsdu = computation forcing sheer at the ultimate limit state;
fctd = computation traction resistance;
r = *(1.6-d)* where *d* is expressed in meters and in any case $d \leq 0{,}60$ m;
ρl =Asl/(bwd) and in any case, $\rho l \leq 0.02$
bw = width of the shear resistant frame;
d = effective section height;
δ = 1 in the absence of standard strain;
δ = 0 in the presence of appreciable normal traction strain;
δ = $1+(M_0/M_{sdu})$ in the presence of compression strain (or pre-compression). M0 is the decompression moment with reference to the fiber end of the section on which Msdu acts. Msdu is the computation maximum acting moment in the area where the shear check is performed. It should be assumed that this is at least equal to M0;
Asl = the area of the longitudinal traction reinforcement anchored beyond the intersection of the reinforcement axis with a possible 45° crack that is triggered in the section in question (see figure 3-I).

3.3.3.2 Elements with shear reinforcement

The level of resistance to shear forces by the cracked element is calculated by schematizing the beam as an ideal lattice. Ritter-Mörsch's represents a simplified model of this. The shear resistant lattice elements are the core transversal reinforcements, which function as wall sections, and the conglomerate of both the compressed flow and the core trusses.
The lattice is completed with longitudinal reinforcement.

3.3.3.2.1 Conglomerate check

The check compares the computed shear with a cautious expression for the compression resistance of the inclined trusses.
In the case in which the core contains pre-stretched rebars or injected cables with a diameter of Øbw/8, it is necessary to use the computation for the nominal width of the core:

$$b_{wn} = b_w - \frac{1}{2}\sum \varnothing$$

where ∑∅ is calculated for the most unfavorable level.
To verify conglomerate that is compressed obliquely, it is possible to use:

$$V_{Sdu} \leq 0{,}30\, f_{cd} \cdot b_w \cdot d$$

as *fcd* is the computed resistance when compressed.
The indicated shear resistance expression corresponds to cases where the transversal reinforcement consists of orthogonal stirrups at the central line (a=90°).
If the stirrups are inclined (45°≤ α <90°) the shear resistance expression should be taken to be equal to:

$$0{,}30\, f_{cd} \cdot b_w \cdot d\, (1+\cot\alpha)$$

with an upper limit of 0.45 *fcd* · *bw* · *d*.
In the case of raised rebars, most of that indicated above is not applicable.

3.3.3.2.2 Transversal core reinforcement check

The shear resistance VRd capacity of structural elements with specific shear reinforcement must be assessed on the basis of an appropriate Framed schematization. The resistant elements of the ideal truss are: the transverse

reinforcement, the longitudinal reinforcement, the compressed stream of concrete and struts of the angled core.

The shear resistance is defined:

VRd = min (VRsd, VRcd)

the design "tensile shear" resistance (VRsd) is calculated:

$$V_{Rsd} = 0,9 \cdot d \cdot \frac{A_{sw}}{s} \cdot f_{yd} \cdot (ctg\alpha + ctg\theta) \cdot \sin\alpha$$

With reference to the concrete core, the design "shear compression" resistance (VRcd) is calculated:

$$V_{Rcd} = 0,9 \cdot d \cdot b_w \cdot \alpha_c \cdot f'_{cd} \cdot (ctg\alpha + ctg\theta)/(1 + ctg^2\theta)$$

3.3.4 Non-linear behavior of reinforced cement elements

The beam elements in reinforced cement are based on a non-linear type correction. This starts from the elastic prediction, which compares the calculated forces with the resistance limits which follow from the above-mentioned criteria.
Relative to the bending resistance mechanism, plastic hinges are formed when the resistance moment is reached. This limits the capacity to transmit bending forces when the ultimate rotation is reached.
The beam remains in the elastic field until either one of the two ends reaches the limit moment. This check is performed for both sections.
If, for example, at theend*i*of theelement the moment limit value is exceeded, the plastic hinge is created. The moment is maintained at a constant equal to the limit value. The total relation, which was before entirely elastic, becomes partly elastic and partly plastic, localized at theend. The moment at the limit, *j* while still in the elastic field must be balanced with the current displacement condition of theelement*i*in which the plastic hinge section is found. So, it is no longer that which was provided by the initial elastic prediction basedon the hypothesis that the rotations developed at the end are of an exclusively elastic nature. Instead, it is balanced with the displacement state, which at the limit takes into account atend*i*only the elastic part and in*j*the rotation which is still entirely elastic.
The assessment of the balanced moment with that displacement state occurs immediately when thelinear elastic equation is used, in which the appropriate surrounding conditions are applied. For example, in the case above in which the plastic hinge is created in i, imposing the known values at theend*i*, equal to the limit moment, and that of*j*- entirely elastic rotation. In this way, the program can compute the elastic and plastic parts of the rotation*i*and the balance moment*j*balanced with the current displacement state at the end, considering only the elastic part of the rotation at theend where the plastic hinge is formed.
Depending on the various possible situations, the surrounding conditions selected whenusing theelastic line equation are as follows:

Case**plasticized end i (Pi)–end j in elastic phase (Ej)**: the surrounding conditions selected are Mi = MLimit and φj (*known from the initial elastic prediction.*) from which the number for the elastic rotation at the end is found end*i*φi, eland consequently, also the plastic φi,P; known φj andφi, el it is possible to calculate the Mj moment balanced with that displacement state.
Case**end i in elastic phase (Ei)–end j plasticized (Pj)**: the surrounding conditions selected are Mj = MLimit and φi(*known from the initial elastic prediction.*) from which the number for the elastic rotation at the end is found *j* φj, el and consequently, also the plastic φj,P; known φi and φj,el it is possible to calculate the Mi moment balanced with that displacement state.
Case**both ends i and j plasticized (Pi - Pj)**:the surrounding conditions selected are

Mi = Mj = MLimit from which the figures for the elastic rotation at the two ends are found φi,el and φj,el from which it is possible to calculate the plastic figures φi,P and φj,P.

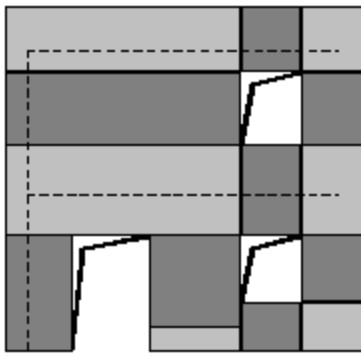
At this point, when the bending moment at the ends of the element have been correctly computed the next step is the rotation check. This is calculated with respect to the cord identified in the section at the zero moment, with respect to the ultimate rotation calculated according to that indicated in the code.
In the case in which the limit value is exceeded, the moment is over and the rotation imparted becomes entirely plastic. At this point the force characteristics (shear and moment) found in theother end are calculated in accordance with the new static schema for the beam. This means for theend in which the bending collapse which became a plastic hinge occurred.
To sum up, the conditions which can occur in each end sectionare a result relative to the bending mechanism (with or without normal force):
elastic phase permanence (E);
formation of a plastic hinge due to reaching the moment value limit (P);
collapse of the section after exceeding the maximum allowable rotation value (R).
Please note that the shear force characteristics are constant along theelement due to the concentrated actions in the nodes. These are calculated so as to guarantee the'equilibrium with the moments developed at the ends.
With regards to the shear resistance check, this is performed by comparing the calculated shear value, which is compatible with the equilibrium of the element on the basis of the moments developed at the ends, with that limit. If this check is not satisfied, and the shear resistance is less than that calculated then theelement will be evaluated as collapsed, and hence no longer able to support forces, due to the fragile breakage mechanism hypotheses.
Please note the dependence of the maximum resistance limits (for bending and shear) on the normal compression strain. It follows that these comparison values are not a constant property of theelement. They can vary during the analysis, following redistribution of the actions towards the elements which contribute together tothe total equilibrium of the structural system.

3.4 Three-dimensional Modelling

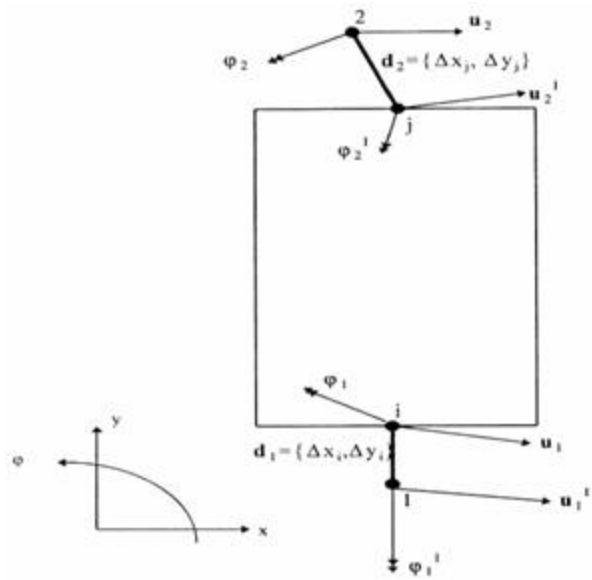
The three-dimensional modelling used is the direct result of observation of real building behavior and experimental tests. These allowed the introduction of some hypotheses about structural behavior of masonry constructions.
As mentioned above, damage mechanisms observed in buildings can be divided into two categories. These depend on the type of wall response and their mutual degree of connection: so-called first mode mechanisms, in which walls or portions of walls receive orthogonal forces on their floor; and second mode mechanisms in which the wall responds to the seismic action on its floor.
It is necessary to understand and identify the structure resistant to vertical and horizontal loads internal to the masonry construction to obtain a reliable simulation. usually, these elements are walls and horizontal structures.
Walls are assigned the role of resistant element, both with regards to horizontal and vertical loads. The horizontal structures have the role of distributing the vertical load resting on them to the walls and then dividing, as part of the floors' stiffening elements, the horizontal actions on the impacted walls.
With regards to the horizontal actions, the chosen model neglects the resistance contribution of the walls in orthogonal direction to their floor, given their notable flexibility.
Hence, the collapse mechanisms outside the floor are not modelled. However, this is not a limitation as these are phenomena connected to the local response of the individual walls. The onset of these can be decidedly limited by appropriate preventative actions.
Similarly, the flexional response of the floors is not simulated. This is significant in checking their resistance, but can be ignored in terms of the global response. Loads on the floor are divided by the walls in function of the area of influence and warping direction. The floor contributes as a slab with suitable level resistance.

3.4.1 Wall modelling

Dividing the wall into vertical areas which correspond to the various levels, and noting the location of the openings, the portions of masonry, masonry piers, and spandrel beams, where deformability and damage are concentrated, can be determined. This can be verified by observing the damage caused be real earthquakes, and with experimental and numerical simulations. These areas are modeled with finite two-dimensional macro-elements, which represent masonry walls, with two nodes and three degrees of liberty per node *(ux, uz, roty)* and two additional internal degrees of liberty.
The resistant portions of the wall are considered as rigid two-dimensional nodes with finite dimensions, to which the macro-elements are connected. The macro-elements transfer the actions along the level's three degrees of liberty, at each incident node. In the description of each single wall, the nodes are identified by a pair of coordinates (*x, z*) in the level of the wall. The height, z, corresponds to that of the horizontal structures. The degrees of liberty are solely *ux, uz,* and *roty* (for two-dimensional nodes).
Thanks to the division of elements into nodes, the wall model becomes completely comparable to that of a frame plan.



During assembly of the wall, the possible eccentricities between the model nodes and the ends of the macro-elements are considered. Given the axes that are the center of mass for the elements, these cannot coincide with the node. Hence in the rigid blocks, it is possible that eccentricity may be found between the model node and that of the flexible element.



This operation is performed by applying a rigidity limit matrix to the same element's rigidity matrix.

Structural modelling also requires the possibility of inserting beams, (elastic prisms with constant sections), identified in the level by the position of the two edge nodes. Once the length (prevalent dimension), the area, the inertial moment, and the elastic module are known, it is possible to reconstruct the rigidity matrix, applying elastic joint rules, and assuming that they remain indefinitely in the elastic field, the normal formulation of elastic joints are applied (Petrini, et al. , 2004; Corradi dell'Acqua, 1992).

In addition to the presence of actual beams (architraves or r.c. tie beams), the model assumes the presence of tie rod structures. These metallic structures completely lack bending rigidity and lose all effectiveness if they are compressed. This detail adds an additional non-linear element to the model. The total rigidity of the system must decrease if a stretched tie rod is compressed, and it must increase in the opposite case.

Another characteristic of these elements is the possibility to assign an initial deformation $\varepsilon 0$, which determines a force $Fc = EA\,\varepsilon_0$. From a static point of view, once the overall vector of the precompression forces ***fc*** is determined, it is enough to apply it to the structure as if it were an external load.

The rigidity matrix for elements without bending rigidity is easily found by eliminating all the limits that contain J from the element matrix. To manage the non-linearity, all

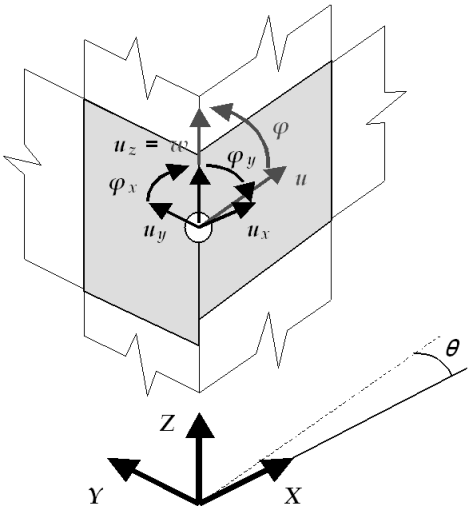
of the elastic contributions due to the tie rods must be kept distinct. At each step, it must be verified if the tie rod that previously was stretched is now compressed or vice versa. If the situation changes, the total rigidity matrix for the model must be corrected.

3.4.2 Spatial Modelling

In spatial modelling, the walls are resistant elements, with regards to vertical and horizontal loads. On the other hand, the horizontal structures (floors, vaults, ceilings) transfer their vertical loads to the walls and divide the horizontal actions onto the incident walls. In this way, the structure is modelled by assembly of the level structures: the walls and the horizontal structures, both lacking bending rigidity outside of the level.
The procedure for modelling macro-elements for masonry walls which receive forces from their own level was illustrated above. This instrument constitutes an important starting point for modelling of the overall behavior, based on the behavior of the walls on their level. In any case, extension of the procedure to three-dimensional modelling is not simple. The correct strategy is that of conserving the modelling of the walls on their level and assembling them with the horizontal structures, including those for which the membrane behavior is modelled.
In this way, the model of the structure takes on mass and rigidity on all of the three dimensional degrees of liberty. At the same time, it locally takes into account the individual degrees of liberty of the levels (two-dimensional nodes).
In this way, an essential structural model is created, without adding the complication of computation of the response outside of the local level. This can of course be verified later.
Once a single overall reference is established for the structural model, the local references are introduced for each wall. It is assumed that the walls rest on the vertical plane and they are found in the plan of the generic wall i through the coordinates of a point, the origin of the local reference *Oi (xi, yi, zi)*, with respect to an overall Cartesian reference system *(X,Y,Z)*. The angle i is computed with respect to axis *X*.
In this way, the local reference system for the wall is unambiguously defined and the macro-element modelling can take place with the same modality used for the levels. Macro-elements, such as beams and tie rods, maintain the behavior of the level and do not require reformulation.
Connection nodes, belonging to a single wall, maintain their degrees of liberty at the local reference level. Nodes that belong to more than one wall (localized in the incidences of the walls) must have degrees of liberty in the overall reference (three-dimensional nodes). These nodes, due to the hypothesis that ignores the bending rigidity of the walls, do not need a rotational degree of liberty around the Z axis, as they are not connected to any element able to provide local rotational rigidity limits.
Three-dimensional rigid nodes, representing angle iron or hammer situations, are obtained as an assemblage of virtual two-dimensional rigid nodes identified in each of the incident walls. These have displacement components generalized using five degrees of liberty: three displacement *ux*, *uy* and *uz*. Two rotational φx and φy. The relationships between the five displacement and rotation components of the three-dimensional node and the three for the fictitious two-dimensional node, belonging to the single wall are given by:

$$\begin{cases} u = u_x \cos\theta + u_y \sin\theta \\ w = u_z \\ \varphi = \varphi_x \sin\theta - \varphi_y \cos\theta \end{cases}$$

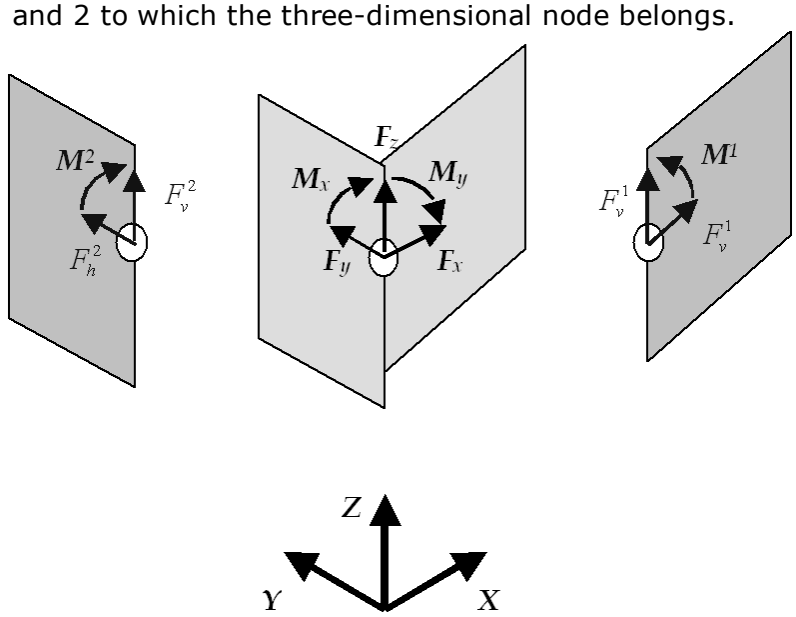
in which u, w, and φ indicate the three displacement components according to the degrees of liberty found in the fictitious node that belongs to the generic wall facing the plan according to angle φ. Similarly, the forces applied to the three-dimensional nodes are displaced according to the directions identified by the middle level of the walls and then applied to the macro-elements in their level of resistance.



The reactive forces transmitted by the macro-elements that belong to the individual walls to the fictitious two-dimensional nodes are carried over to the overall reference based on

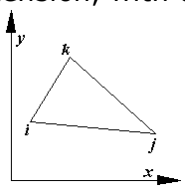
$$\begin{cases} F_x = F_h^1 \cos\theta_1 + F_h^2 \cos\theta_2 \\ F_y = F_h^1 \sin\theta_1 + F_h^2 \sin\theta_2 \\ F_z = F_v^1 + F_v^2 \\ M_x = M^1 \sin\theta_1 + M^2 \sin\theta_2 \\ M_y = -M^1 \cos\theta_1 - M^2 \cos\theta_2 \end{cases}$$

in which, as seen in the figure, the boundaries with apex 1 and 2 respectively make reference to the force limits corresponding with the virtual nodes identified in the walls 1 and 2 to which the three-dimensional node belongs.



In this way, modelling of the wall can take place on the level, recovering that described in the preceding chapter. The nodes that only belong to a single wall remain

two-dimensional. They maintain only three degrees of liberty, rather than five.
The floors, modelled as finished orthotropic membrane three-node elements, with two degrees of liberty per node (displacements ux and uy), are identified with a warping direction, with respect to that characterized by an elastic module E1. E2 is an elastic model with a direction perpendicular to the warping, while n is the Poisson coefficient and G2,1 is the elasticity tangential model. E1 and E2 represent the degree of connection that the floor, thanks to the effects of the tie beams and tie rods, exercises on the element nodes on the level of the wall. G2,1 represent the shear rigidity of the floor on its level and the division of the actions among the walls depends on this.
It is possible to position a floor element connecting it to the three-dimensional nodes. This is because the floor element functions principally to divide the horizontal actions between the various walls in proportion to their rigidity and its own. In this way it makes the model three-dimensional in a way that brings it close to the true structural performance.
The finished reference element to be considered is the level element, in a level state of tension, with three nodes.



The rigidity matrix involves the individual three-dimensional incidental nodes on the floor. The contribution of the vertical loads, self or borne, is attributed in terms of nodal mass added to all the nodes, including those with three degrees of liberty, that belong to the incident walls at the height of the level of the floor. This added mass is calculated based on the area of influence of each node, taking into account the warping direction of the floor.

4 Reference code

4.1 Europe

Reference code is **Eurocode 8**.

4.2 Italy

➔Norme Tecniche per le Costruzioni - ***D.M. 14 gennaio 2008***

➔Norme Tecniche per le Costruzioni in zona Sismica - ***D.M. 16 gennaio 1996***

4.2.1 N.T. - D.M. 14 gennaio 2008

Le prescrizioni per questa normativa mostrano le seguenti peculiarità:

Carico sismico: La definizione degli spettri mediante il carico sismico, non è più legata alla zonizzazione ma alle coordinate geografiche (latitudine, longitudine), secondo quanto prescritto dal *"reticolo di riferimento"* in base alle indicazioni riportate nell' *Allegato A delle Norme Tecniche*.

Carico statico sui solai: Per questa normativa è necessario definire il solo fattore $\psi 2$

Stati Limite: Gli stati limite da prendere in esame sono i seguenti (paragrafo 3.2.1 delle Norme Tecniche):
- Stato Limite di Salvaguardia della Vita (SLV)
- Stato Limite di Danno (SLD)
- Stato Limite di Operatività (SLO)

4.2.2 N.T. - D.M. 16 gennaio 1996

Secondo quanto riportato nella normativa, si rende necessaria la verifica di resistenza strutturale che equivale a controllare che la struttura sia in grado di sopportare le azioni sismiche previste dalla normativa.
Il programma calcola il valore del carico sismico per l'edificio modellato e lo confronta con il massimo carico sopportabile dall'edificio corrispondente al valore di picco della curva di capacità.

La verifica risulterà soddisfatta seguendo il seguente controllo:
F_h *(carico sismico richiesto dalla norma)* < F_u *(carico ultimo dell'edificio)*

4.3 Switzerland

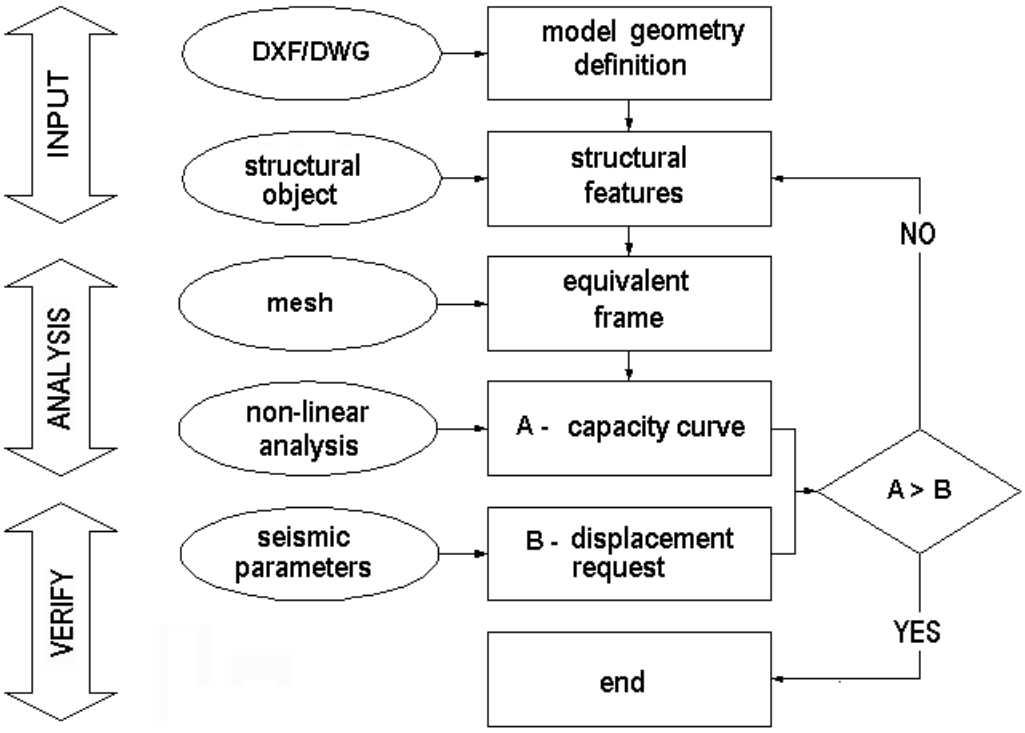
Reference code are:

➔ **SIA 2018**
➔ **SIA 266**
➔ **SIA 261**
➔ **SIA 260**

5 General schema of the program

3Muri executes Non-linear static analysis on masonry buildings .

The process to follow in the verification of the structure to examine consists of the following phases:



5.1 Input phase

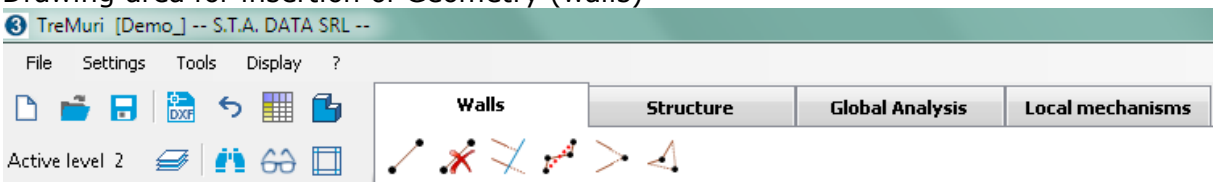
In this phase, the user inserts the data necessary for performing the analysis.

Define geometry
The geometric characteristics of the structure, that is the placement of the walls in the plan and the height of the floors, constitute the foundation for insertion of the "structural objects" found in the next phase.
The geometric data, mainly segments, are inserted directly in drawing mode, or by tracing a DXF or DWG file.

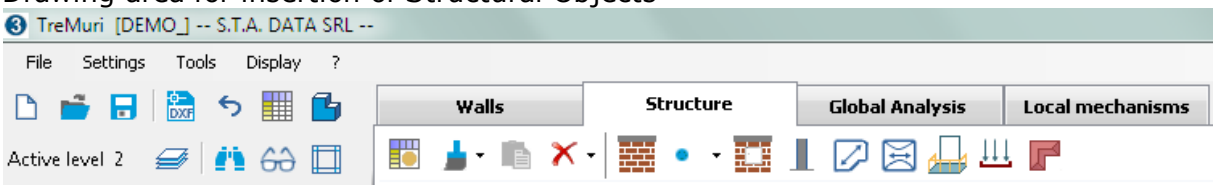
Practical rules for effective importation - Prepare the tables before importing:
- Position the origin of the reference system in one of the vertexes of the plan.
- Define the limits of the graphic area around the plan to be imported (CAD program limits command).
- Delete contiguous designs and images around the plan, maintaining only items that are truly useful. Delete any screens that may be present.
- Check the unit of measurement selected. 3muri uses unit that you can see in Units and formats geomety setting(default:"cm"). In this way, it is possibly to correctly scale the design before importation, and to define the scaling factor to be used.
- Select the plan and blow up everything. (There should not be any blocks.)
- Save the design in dxf/dwg format, version "2000."

Structural characteristics
The structure is composed of "structural objects" which constitute the resistant elements.
The objects are mainly vertical masonry walls with possible reinforcements (tie rods, tie beams, columns), floors for the distribution of horizontal actions, and linear elements (beams, columns) made from various material types (R.C., steel, wood).
Every object is characterized by its material and additional geometric parameters (thickness, inertial characteristics, resistance properties).
Reinforcement parameters are requested for R.C. structures as non-linear analysis is performed for these elements.

Drawing area for insertion of Geometry (walls)



Drawing area for insertion of Structural Objects



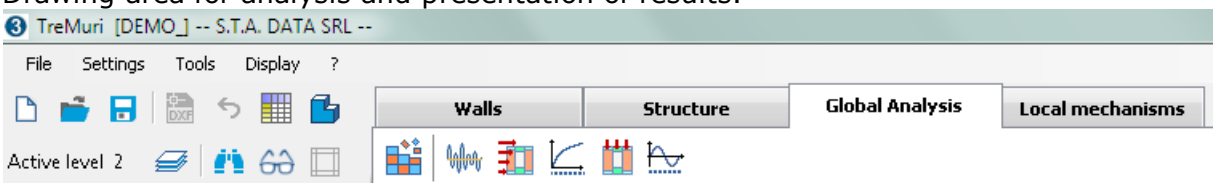
5.2 Analysis Phase

Structural analysis is divided in two phases: in the first an equivalent frame model is automatically created. After this, non-linear static analysis (push-over) follows, from which the structural capacity curve is derived (strain curve - displacement of the control point).

Define equivalent frame
Using the 3Muri model, the data for the equivalent frame are derived, starting from the geometry and the inserted structural objects.
After the analysis a mesh is created, which schematizes piers, spandrel beams, beams, tie-beams, and columns. These elements can also be manually modified if the situation requires.

Non-linear analysis
The analysis is conducted increasing the loads in monotonic mode, and then deriving the horizontal displacement of the structure.
Once the conventional displacement is exceeded, which is calculated automatically, the structure is considered to have collapsed. The horizontal force-horizontal displacement curve can be constructed, which represents the capacity curve, or the behavior of the structure with changes to the horizontal loads.
Note that this curve is independent of earthquakes, as it is a characteristic intrinsic to the structure, a function of the geometry and resistance characteristics of the materials.

Drawing area for analysis and presentation of results.



5.3 Check

The check compares the displacement offered by the structure and that required by code.

<u>Seismic parameters</u>
Definition of seismic parameters and evaluation of the parameters derived from the structure's capacity curve permits determination of the request in terms of displacement of the spectrum for the project at hand.
The check compares the two displacements (forces in the case of D.M. 1996), that offered by the structure and that required by code.
If the first is greater than the second then the structure satisfies the check. If not, the structure must be modified, changing the necessary parameters.

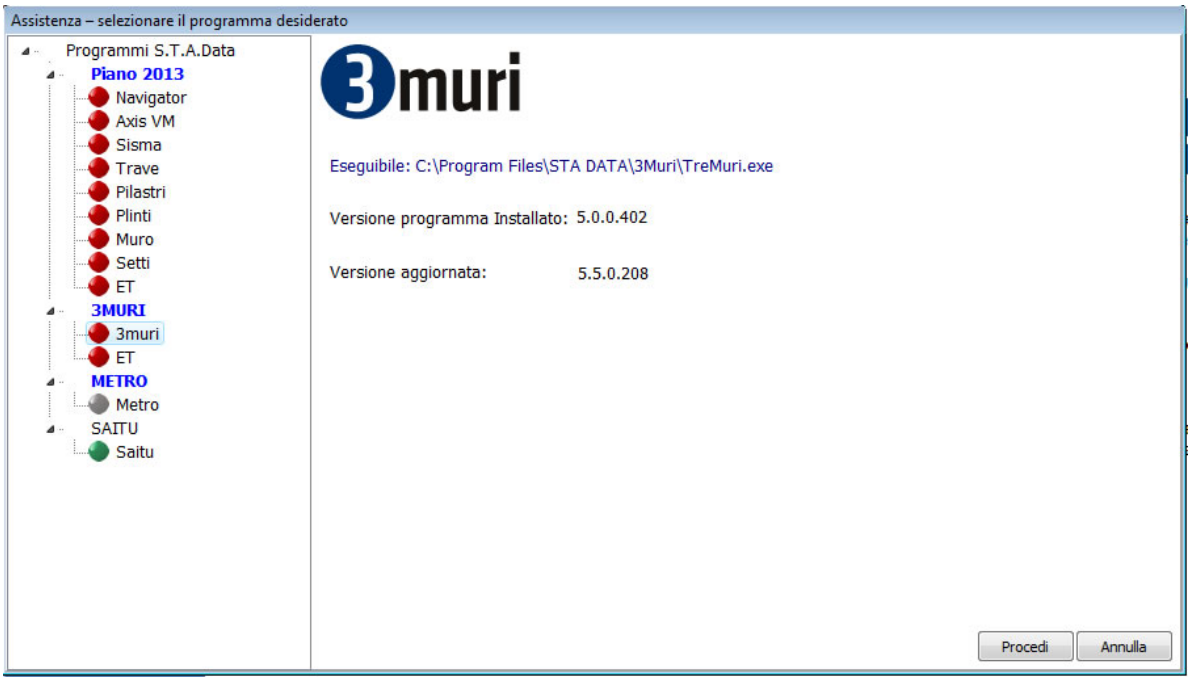
5.4 Edil Cloud

Edil Cloud è lo strumento creato da S.T.A. Data per il salvataggio e la condivisione online di progetti e documenti, le richieste di assistenza e il controllo dello stato di aggiornamento dei programmi installati sul pc.

Direttamente dalla barra dei menù è possibile accedere ai servizi di Edil Cloud. Un menù a tendina permette di selezionare il servizio desiderato.

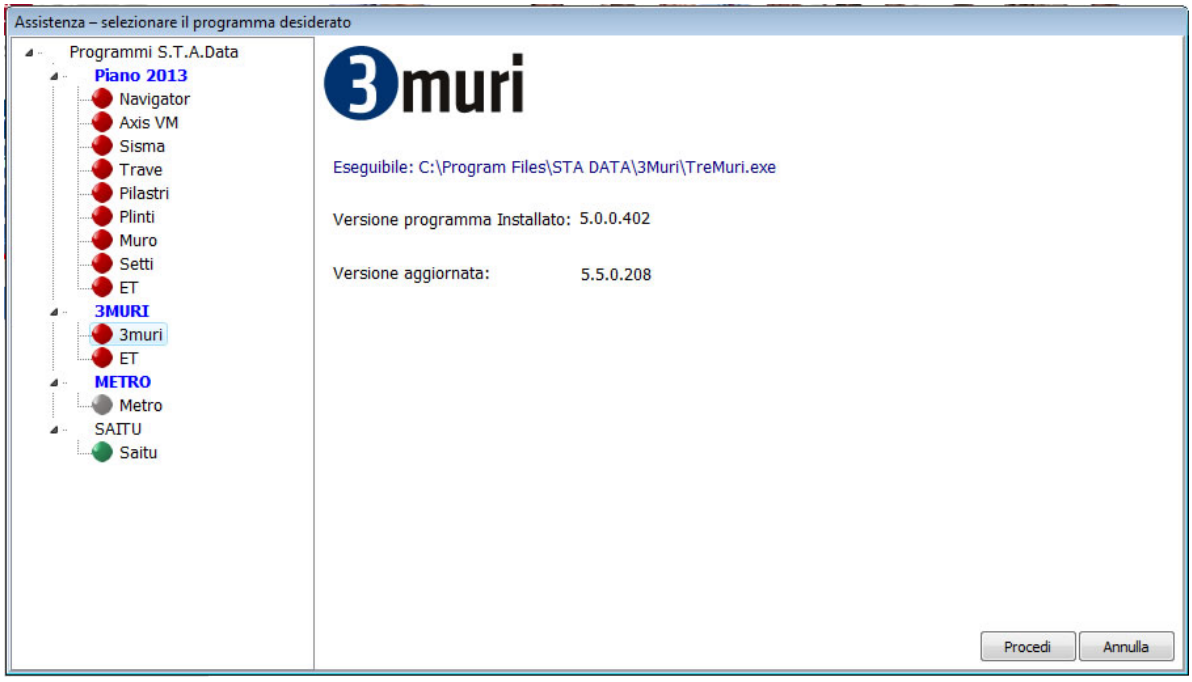
Al primo avvio del programma è richiesto l'inserimento delle proprie credenziali di accesso all'**Area Clienti** del sito www.stadata.com.
Le credenziali possono essere salvate spuntando l'opzione "Memorizza le credenziali di accesso". In questo modo, ai successivi utilizzi del programma, l'accesso al proprio account sarà automatico.
Una volta effettuato il login, il programma si apre nell' ambiente del servizio prescelto**.**

Nell'**Area Aggiornamenti** è possibile verificare lo stato di aggiornamento dei programmi STA Data installati sul computer rispetto alle ultime versioni disponibili.



Cliccando su **Procedi** il browser integrato è apre l'area di download del sito web www.stadata.com da cui si potranno scaricare le versioni più aggiornate disponibili.

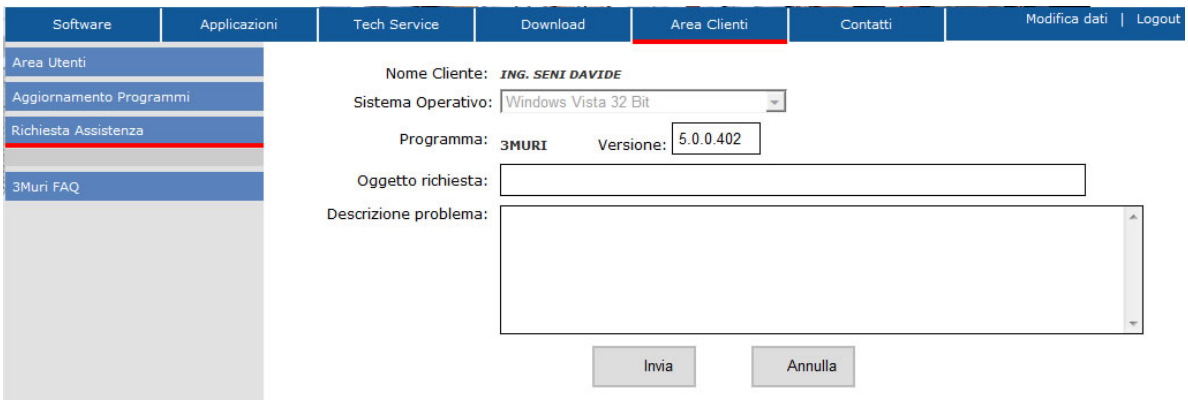
Nell'**Area Gestione Assistenze** è possibile richiedere o verificare lo stato di un'assistenza.



Cliccando su **Procedi i**l browser apre la pagina personale del sito www.stadata.com per la gestione delle richieste di assistenza.

Il form di richiesta assistenza è già precompilato con i propri dati. Per inviare una richeista è sufficiente compilare i campi **Oggetto richiesta** e **Descrizione problema**. Se la richiesta di assistenza è associata ad un particolare lavoro o documento, è possibile salvare online i files necessari tramite le funzioni di archiviazione ed indicarne il percorso nel testo della richiesta.

Per visualizzare l'elenco delle assistenze richieste, senza effettuare una nuova richiesta, è sufficiente fare click su **Annulla**.

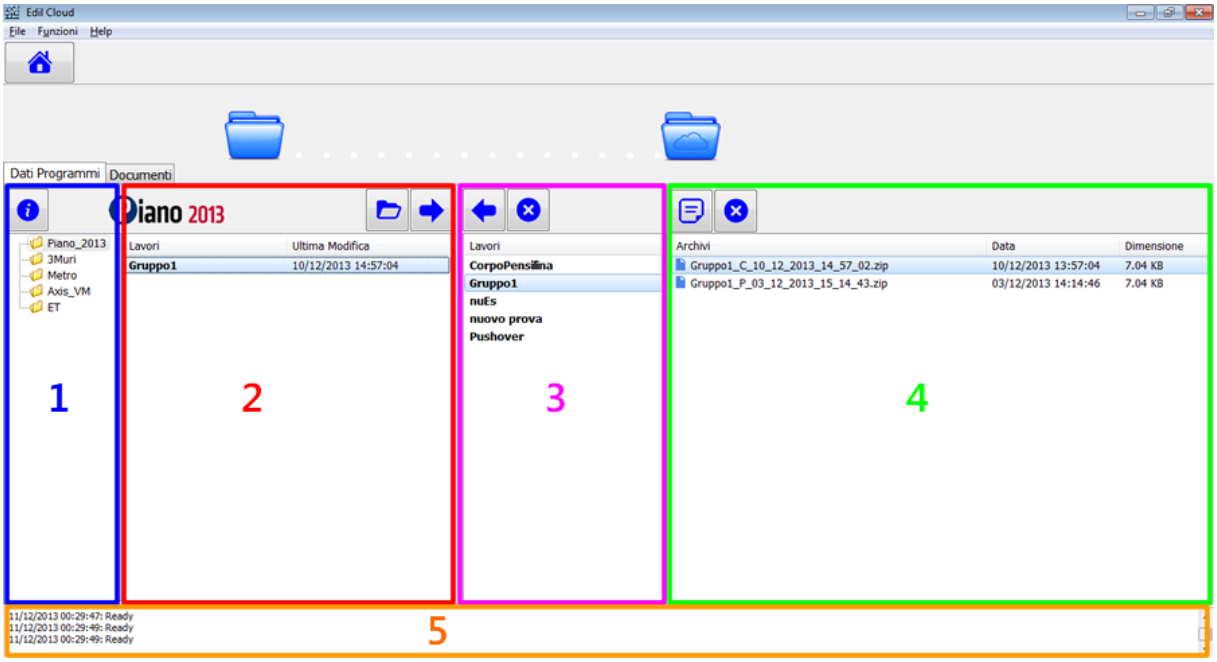


L'**Area di Archiviazione** permette la gestione dei lavori e dei documenti archiviati online.

L'**area di gestione dei lavori** contiene una serie di funzionalità pensate specificatamente per l'organizzazione dei lavori realizzati con i software STA Data.

La finestra può essere divisa in 5 sezioni:

1. Elenco dei programmi;
2. Lavori presenti sul pc in uso;
3. Lavori presenti su cloud;
4. Versioni dello stesso lavoro salvate su cloud;
5. Barra di stato;



Utilizzando i comandi presenti in ogni finestra si può facilmente gestire l'archivio dei propri lavori.

L'**area di gestione documenti** è pensata per il salvataggio su cloud di file e cartelle generici.

La finestra può essere divisa in 5 sezioni:

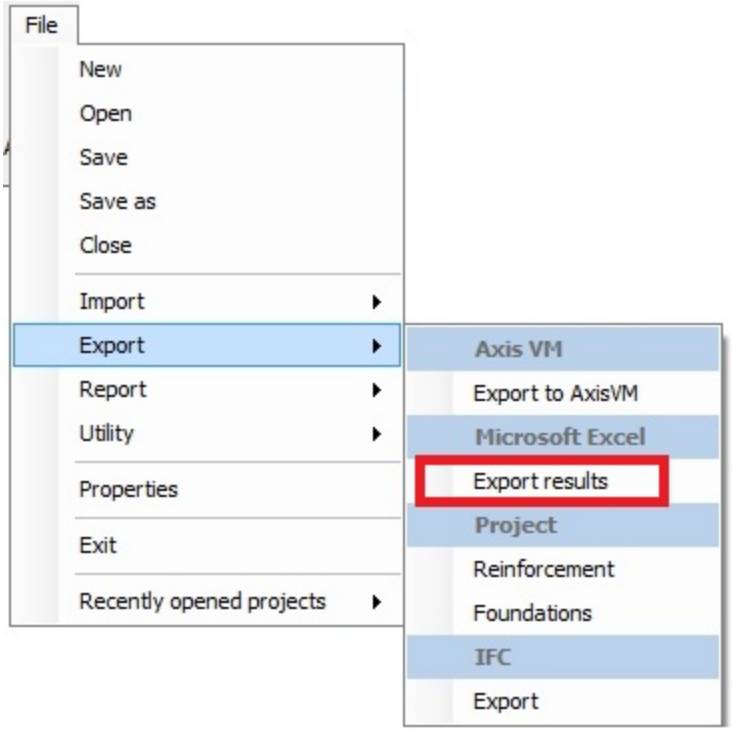
1. Area locale: selezione e visualizzazione dispositivo di archiviazione e cartella.
2. Area locale: selezione e visualizzazione file e cartelle.
3. Area Cloud: selezione e visualizzazione cartelle. In questa finestra è possibile copiare sia file che cartelle.
4. Area Cloud: selezione e visualizzazione file. In questa finestra è posibile copiare solo file.
5. Barra di stato.

Utilizzando il semplice comando di trascinamento con il mouse si possono trasferire i file e le cartella dal proprio pc all'alrchivio remoto e viceversa.

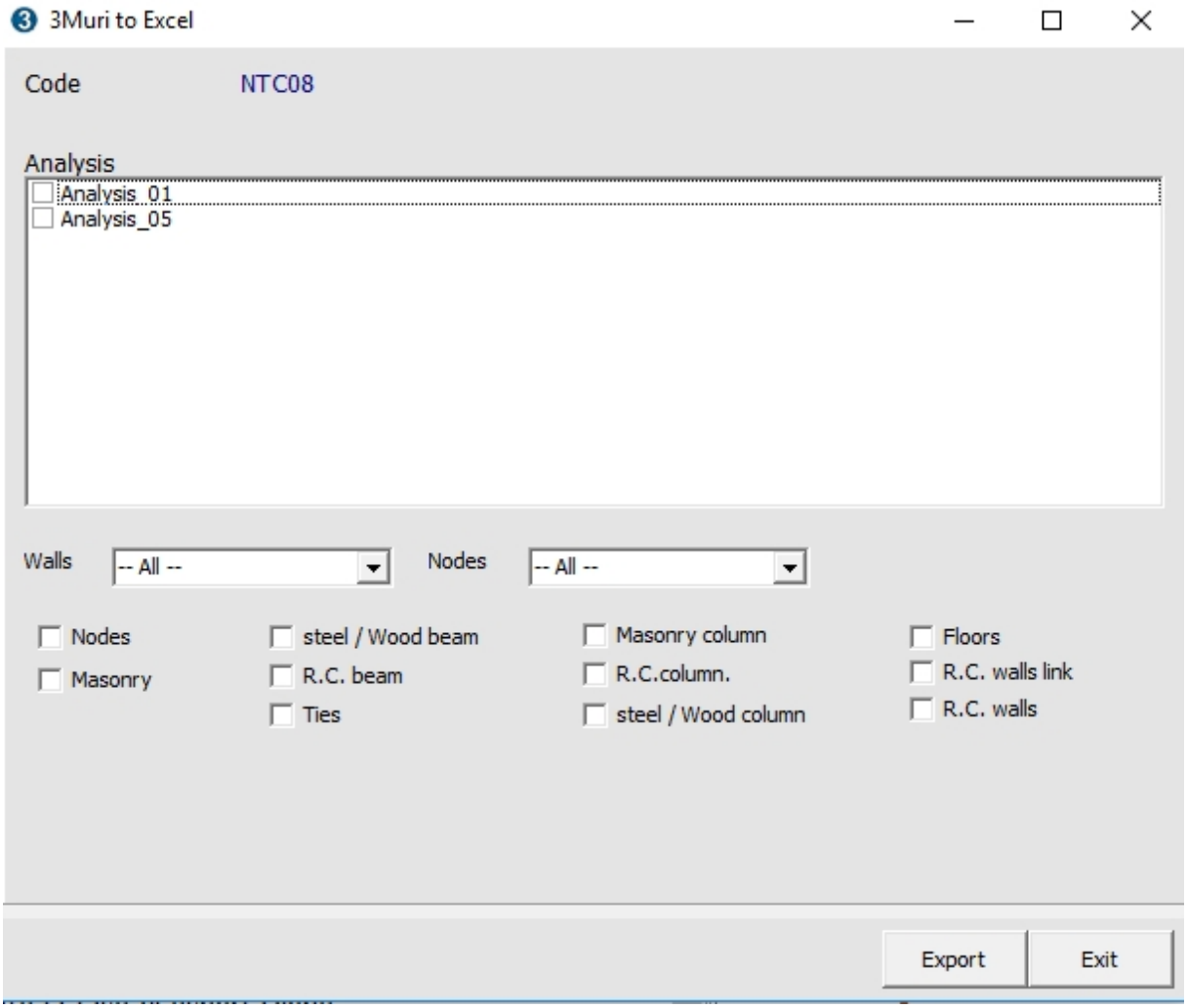
5.5 Export to Excel files

This feature allows to create an Excel file containing the calculation results of the performed analysis.

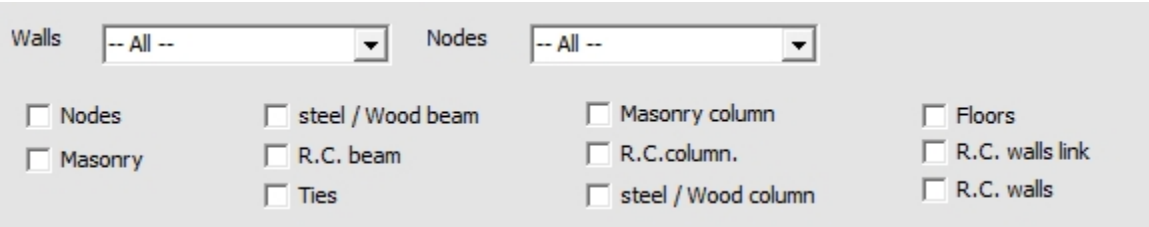
To access this feature you must select the corresponding command in the File menu.



It is presented the following window.



This feature allows you to decide which analysis results you want to export, of which elements and in which walls.



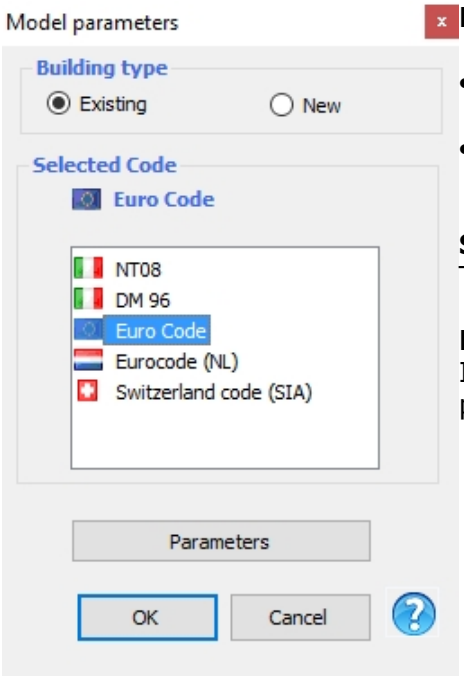
An output Excel file, contains all the required data through the filters during the export phase.

6 Basic concepts for using the program

To correctly use the program, it is important to understand its fundamental rules.
In the drawing work area, lines and points ditinguished in *support graphic entities, walls, and structure elements.*

6.1 Model parameters

The window "Model parameters" is loaded creating a new project.

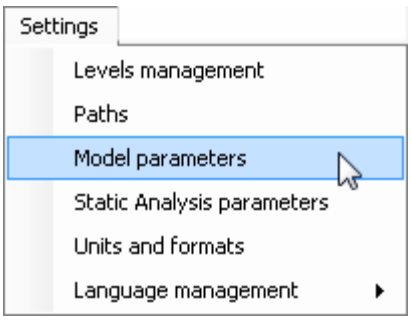


Building type:

- *Existing building*: the user can insert new or exiting materials as well.
- *New building*: the user can insert only new materials.

Selected code:
The user can choose reference code.

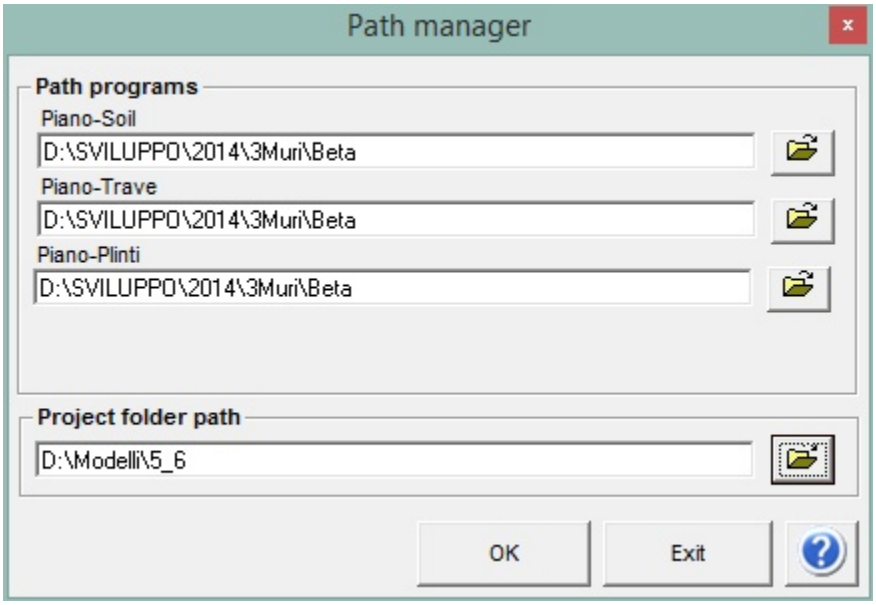
Parameters:
It is possible to modify the calculation settings parameters so in that way you can personalize them.



You can modify model parameters using "Settings" menu every time you need.

6.2 Path Selection

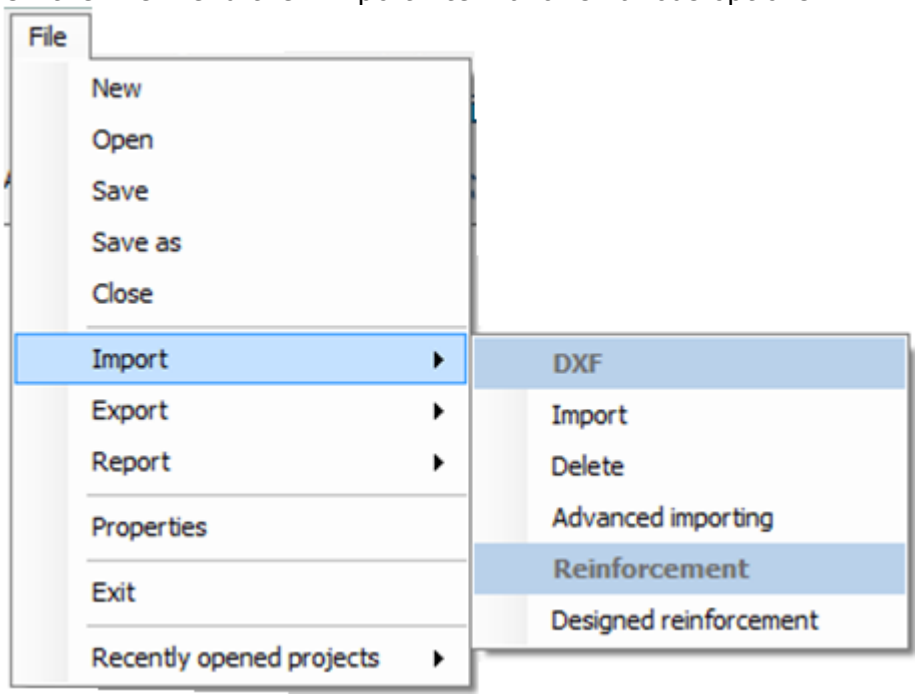
The project paths can be managed by 3Muri.



"Project folder path" indicates the path where projects created by the user are saved.
"Path programs" indicates where the module is installed.Piano soil .

6.3 Import

On the File menu the "Import" item allows various options.



DXF>Import
Allows the import of A DXF file

DXF>Delete
Allows deleting of previously imported DXF files

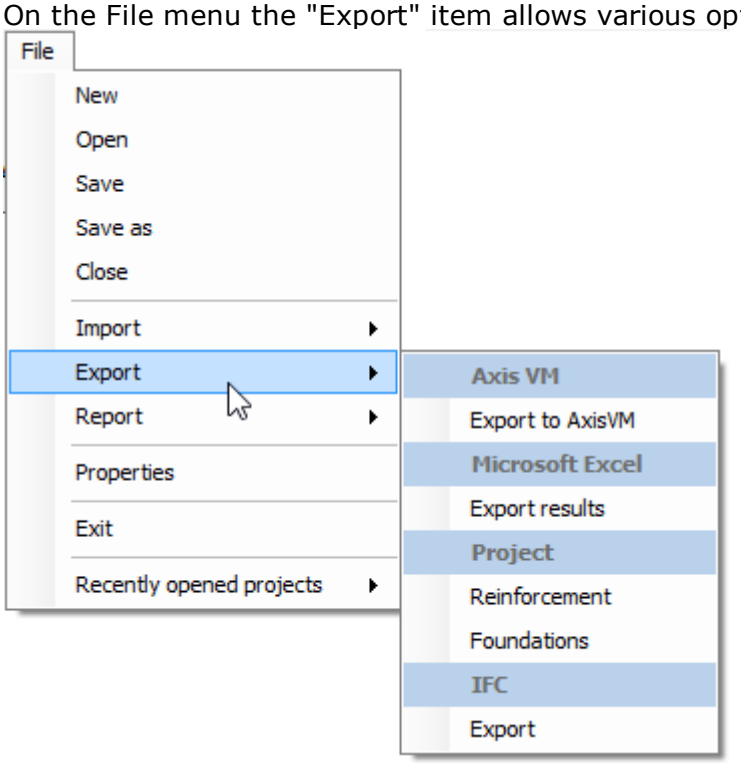
DXF>Advanced importation
Runs the OpenCAD application, a CAD system that makes interoperability its strong point.
With interoperability we mean the ability to cooperate and exchange information with the other products or services with resources's optimization.

The information exchange takes place through various graphic formats:
IFC, DGN (Bentley), DXF, SKP (File Sketchup), EMF, WMF, BMP, GIF , JPG/JPEG, TIF, DWG

Reinforcement>Designed reinforcement
Imports the reinforcement after the preliminary design process.

6.4 Export

On the File menu the "Export" item allows various options.



Axis VM>Export model
Allows the export of a model created with the 3Muri program to AxisVM (FEM solver).

Microsoft Excel> Export results
Allows the exportation of the computation results to Microsoft Excel.

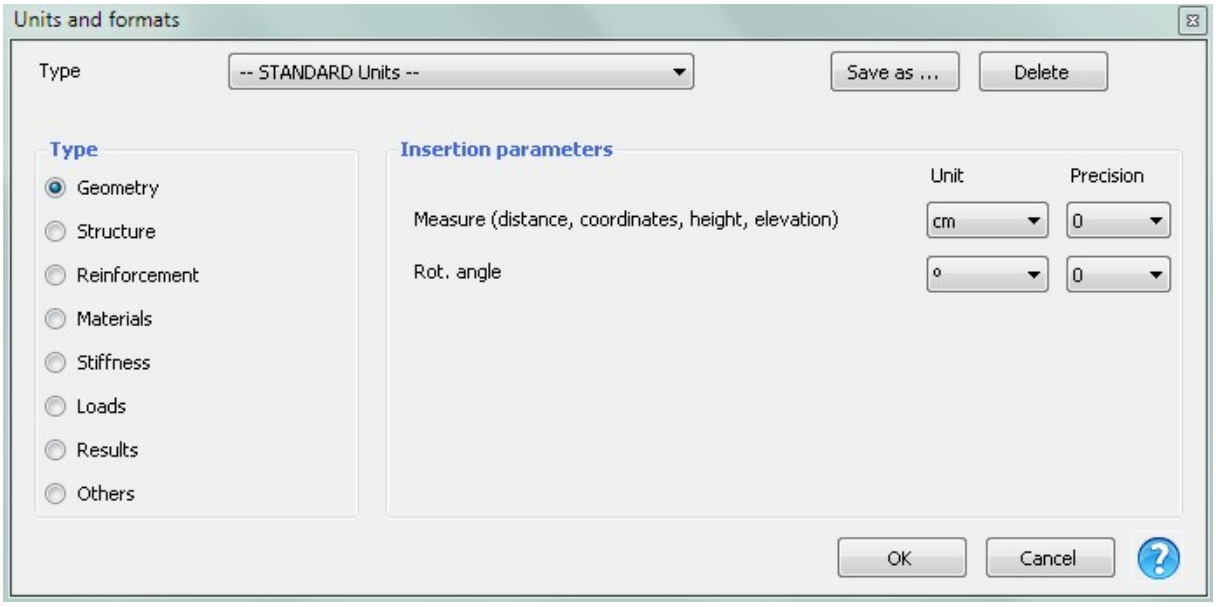
Design>Reinforcement
Provides the calculation of the beams' reinforcement subjected to preliminary design process.

Design>Foundation
Allows the calculation of the foundation structures starting from the reaction forces of the 3Muri model.

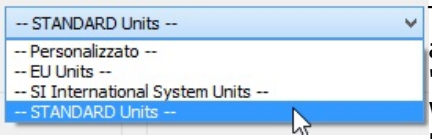
IFC>Export
A model created in 3Muri can be exported in IFC format.

6.5 Units and formats

Settings > Units and Formats

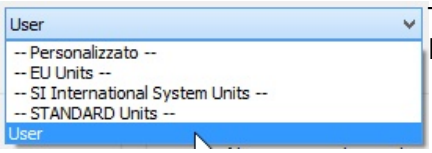


It allows to configure the units (SI and/ or English system) and formats of the variables used on the program (number of decimal used for the visualization or exponential format).
It's possible to use default settings, or create and save the personalized settings.



The drop-down menu contains the list of units systems available.
"STANDARD Units" is the default schema unit.
When we modify the properties of the default style, it becomes
automatically "Personalized".

With the command "Save as..." we can save the parameters that we have modified.
Insert the name of the unit system defined by user.



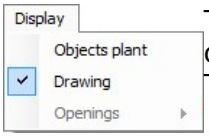
The name of the new "Unit schema" appears in the bottom of defaults schema.

The created units systems remain available inside the program, not only for the model test but even for every successive work.

6.6 Support graphic entities

Support graphic entities are obtained through a combination of commands found in the following image. It allows insertion of support graphic entities that can be used as guidelines for the creation of a model. No structural objects can be associated with support graphic entities. Its use allows the designer to have guidelines available which can be used to proceed in the creation of the model. An imported design in DXF or

DWG format is considered to be a support graphic entity.



The environment dedicated to graphical support is activated through the display menu.
The command bar is displayed at the bottom of the screen.

Insert line: These icons allow to insert generic lines, vertical, horizontal or perpendicular to other elements, that helps inserting the structure.

Insert circle: allows the entry of a circle by three points or a circle given its center and radius, that helps inserting the structure.

Offset: copies a line at a certain distance.
After having selected the line, insert the distance and direction in which the copying should occur.

Copy: copies a graphic element.

Move: moves graphic entities.

Fillet: allows to fillet to lines.

Extend: Extend a line out another.

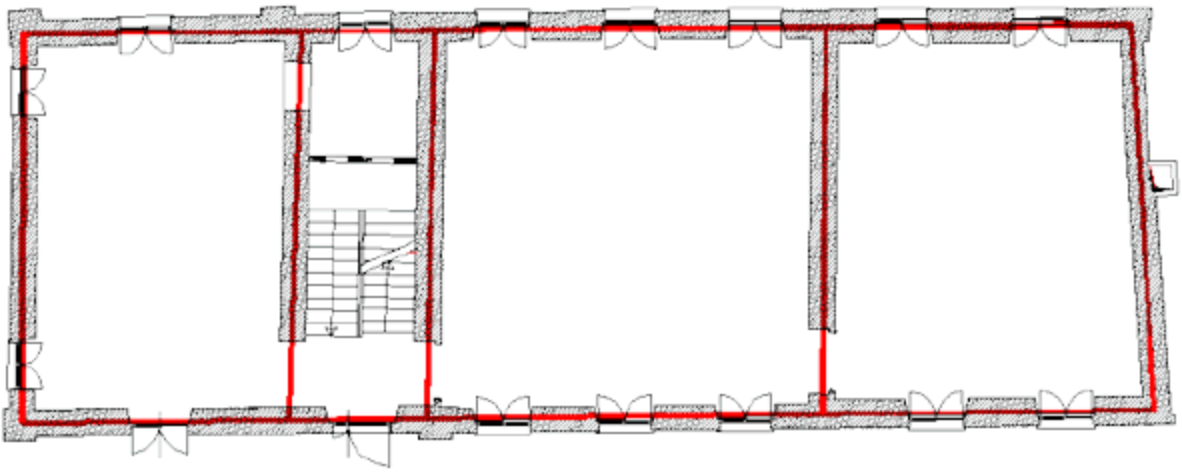
Trim: truncates two intersecting lines

Delete: deletes graphic entities

PLEASE NOTE: DO NOT USE THESE COMMANDS FOR STRUCTURE MANAGEMENT. THEY ARE SIMPLY AN AID FOR INSERTION OF THE STRUCTURE.

6.7 Wall

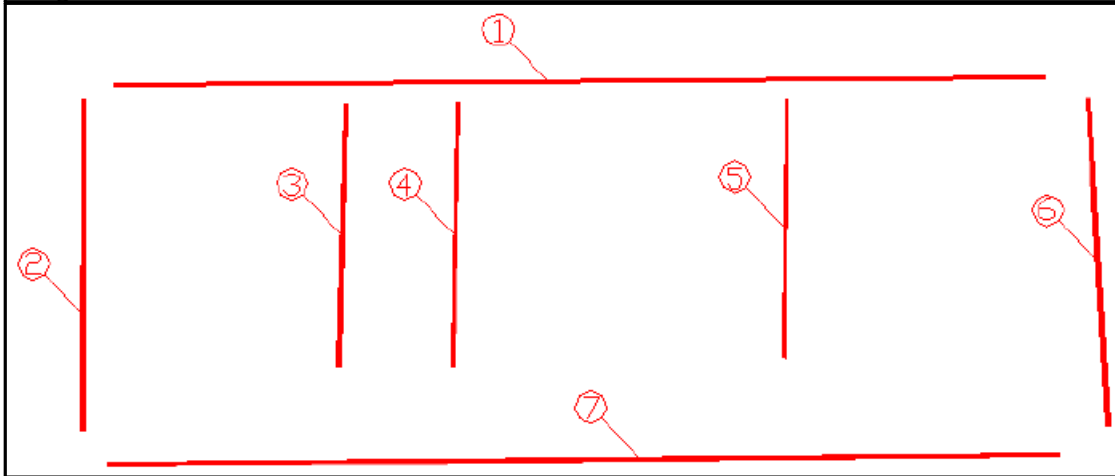
The lines that represent the walls are the basis for the definition of: masonry panels, beams, tie rods, and columns.
The wall represents the synthesis, taken from the architectural design, of the structure to be modelled, both on the horizontal as well as the vertical plane.
Synthesis because it is necessary to include all the principal resistance aspects of the structure, simplifying, if necessary, the scheme that is graphically inserted.
In the following images, you can see how the walls synthesize a combination of masonry walls, representing them with their axes (the red lines represent the walls).



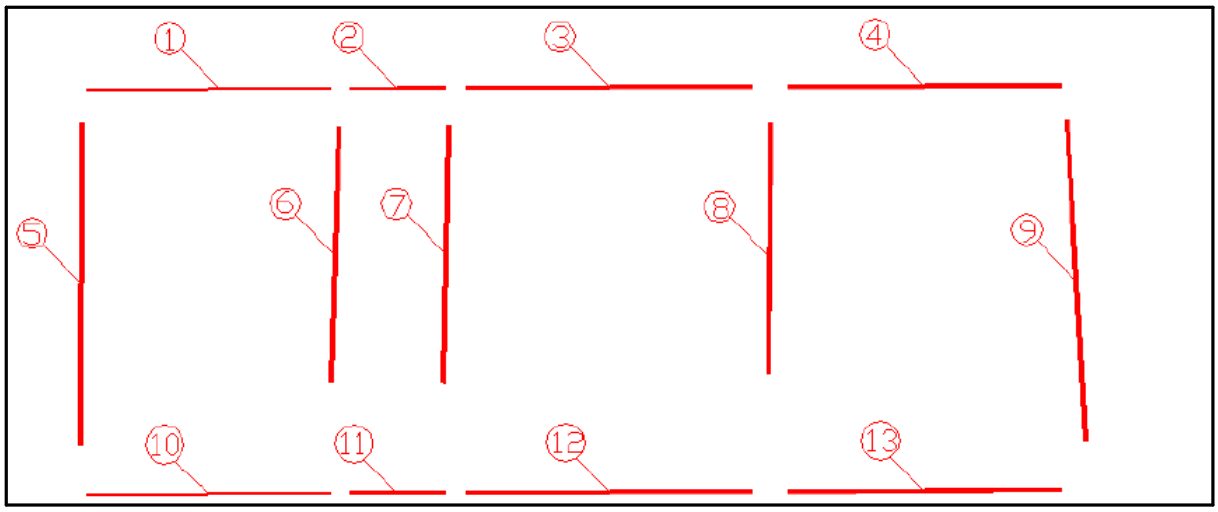
Exploding the wall system it becomes clear why various contiguous segments with structural environment definitions, belonging to the same tangent, must be modelled using a single wall. If wall segments do not have definition in the structural environment on any level, then in place of a single wall, multiple walls are inserted on the same tangent. Here, though, they are NOT contiguous.

The two figures shown below clarify the correct way to create the model.
Wall 1 must remain a single piece and not be divided in four walls.

Single wall: CORRECT MODEL



Separate Walls: INCORRECT MODEL

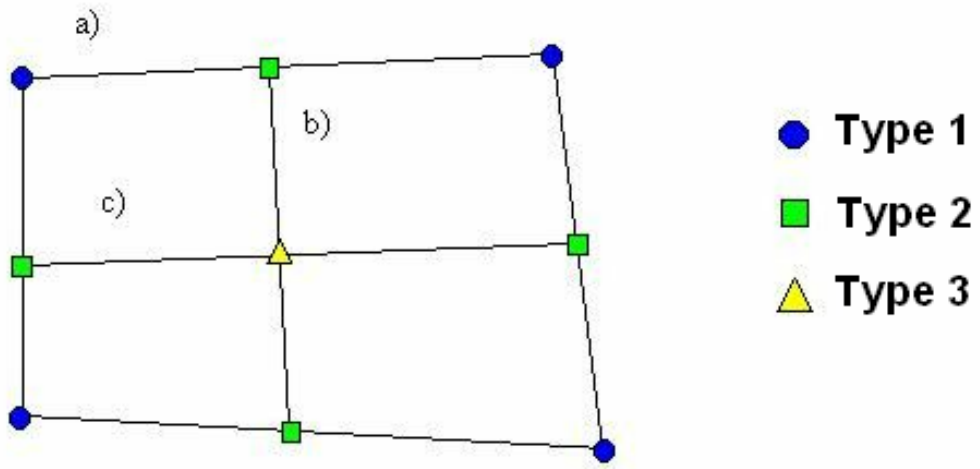


The walls can be managed on all levels, and can be deleted, added to, or modified in all design phases.

When a wall is inserted, the SNAP to the existing nodes or the development of another already inserted wall is automatically activated.
The walls are segments that go from node to node (TYPE 1 wall endpoints are indicated with a small blue ball -- it is a vertical wall endpoint)

Walls whose initial point is found inside of another wall generate a node that does NOT graphically divide the contact wall. TYPE 2 wall endpoints are indicated with a green square. In the figure below, the wall endpoint is for wall b) and is a contact node for a).

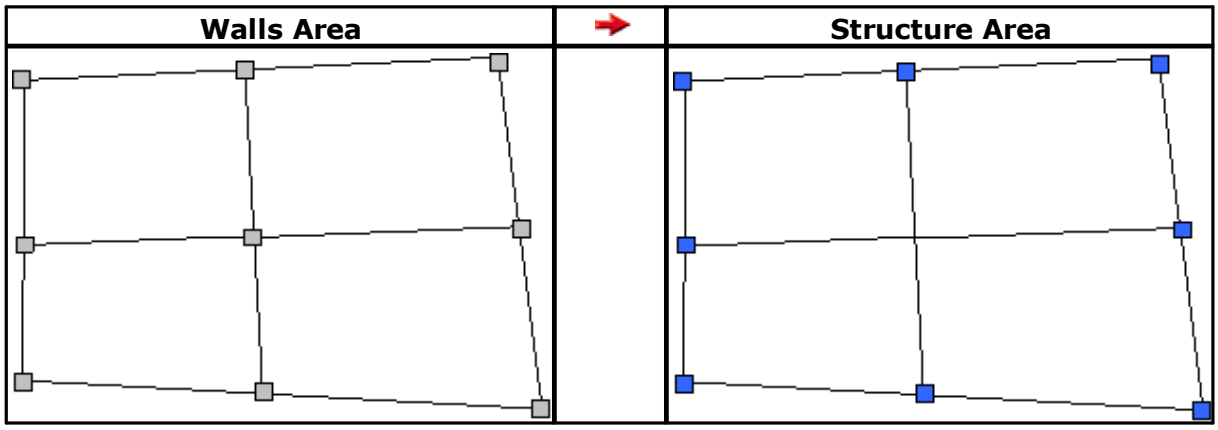
During the insertion phase, a third type of node can be created. This is automatically derived from the computation of the intersection between walls. For example, between the intersection of walls b) and c).
These TYPE 3 nodes (which are indicated with a yellow triangle) are found in an intermediate position at the intersection of the walls. They are represented visually because they can be useful for insertion of structural objects such as panels, beams, and tie rods.



The wall is a graphic entity that can only be inserted using the wall command (found in the Walls area). It represents a sort of "stand in" that the designer will have to complete in the Structure area using the Structural Objects.

6.8 Structure

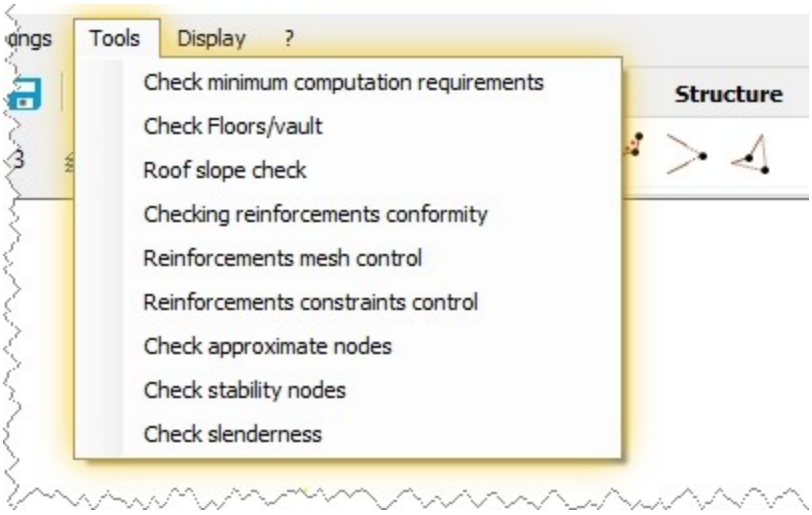
In the Structure area, the walls can be "dressed" with structural objects such as masonry, columns, beams, tie rods, and R.C. walls.
When the Structure area is activated, all the walls are transformed into segments which become objects that can be "dressed." Each wall can be divided into segments by inserting "segment points”.
Segment points are a point of structural discontinuity (e.g. masonry walls with differing thicknesses). They can be inserted along a wall segment or above an existing wall segment.
(e.g. at the intersection of two walls).
Note that the ends of all the walls (nodes on type 1 and type 2 wallsare automatically transformed into segment points for the Structure area. This does not occur for type 3 wall endpoints, where segment points can be inserted only if necessary.



A column can be inserted only in correspondence with a wall endpoint or segment point. In the case being considered, to insert a column in correspondence with the intersection of the two internal walls of the structure, it is necessary to insert a segment point.

6.9 Checking Models

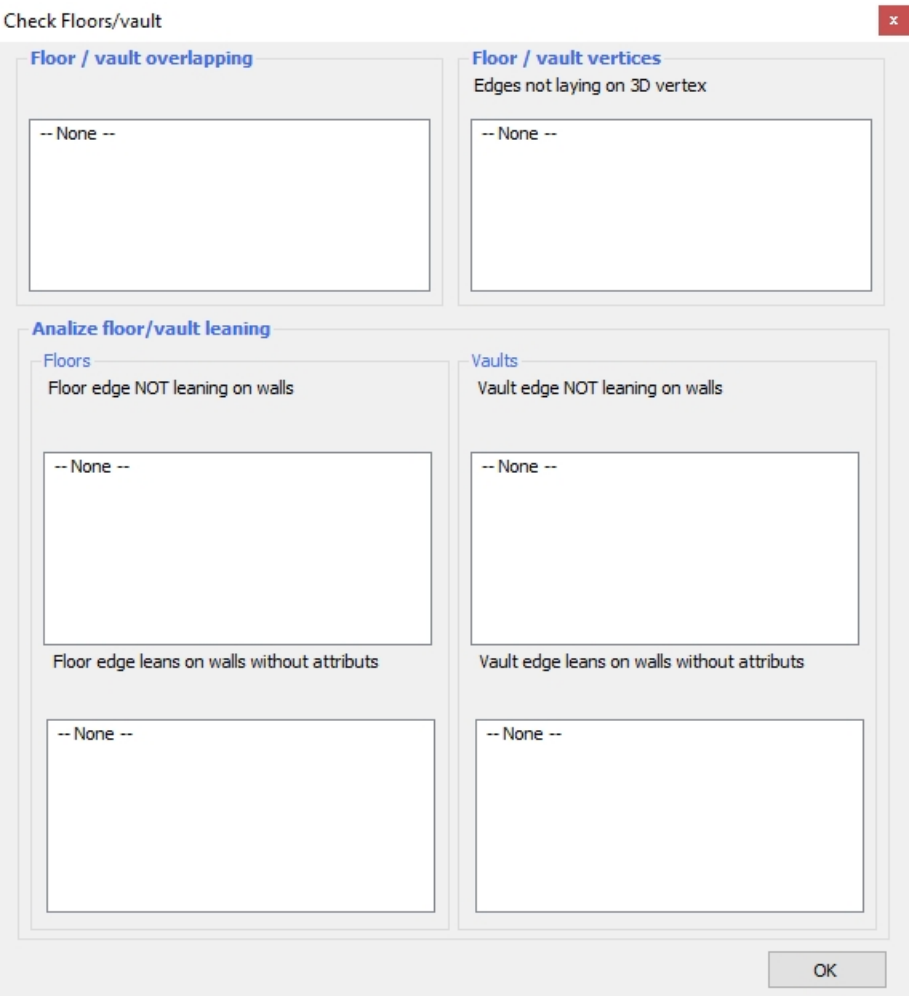
During the model creation phase, disorganized support graphics or a simple human error can lead the designer to make involuntary mistakes. To overcome these shortcomings the program comes with an automatic procedure that controls the fulfillment of the basic rules of model creation.
This correction process is directly available from the Tools menu.



Check minimum computation requirements:
Checks the "box" behavior of the building, checking that there are no nodes that belong to a single wall. If this check does not give a positive outcome than is reported to the user the point where the problem occurred.

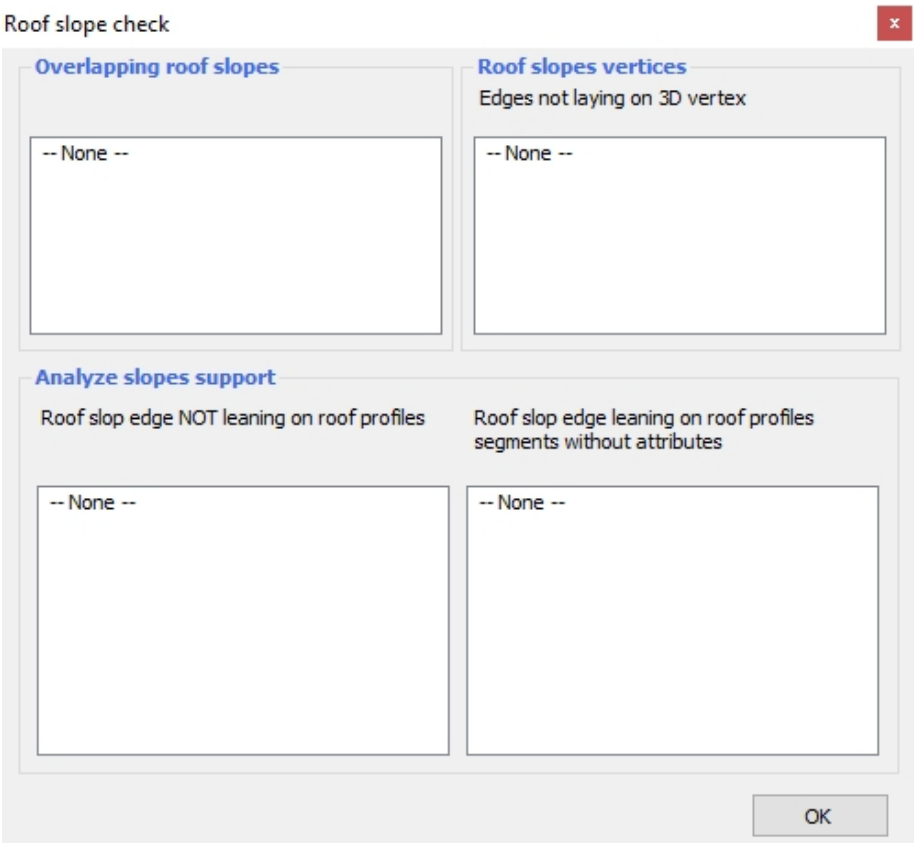
Check floors/vault overlapping:
This command checks for the presence of overlapping floors to avoid the insertion of more than one floor on the same plan.

This command checks for structural elements able to support the floor plan along its entire perimeter.
When the check is finished, critical errors that are found are displayed in the following window.



<u>Check overlapping pitched:</u>
Checks for overlapping pitched to prevent the user to enter more pitched on the same plant.

This command checks for structural elements able to support the pitched along its entire perimeter.
When the check is finished, critical errors that are found are displayed in the following window.

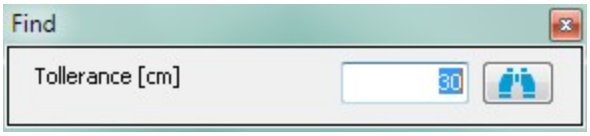


Reinforcements compliance check:
The structural reinforcements inserted with the special command are realized through framed elements.
Each bar of the frame, looking to be "compliant" shall be completely contained within the encumbrance of the wall prospect.

Reinforcements mesh check:
Check the suitability of the mesh reinforcements to the calculation.
A mesh of reinforcement is defined acceptable if the frame is correctly paired and connected to the structure.
The defendant frame must form a scheme not labile and the connections to the wall structure can take place exclusively to portions of levels and foundation.

Reinforcements bonds check:
Any additional nodes created with the definition of the reinforcing frames, may require external constraint that defines the connection with the foundation.
This check validates the suitability of the reinforcements foundation constraints.

Check approximated nodes:
Allows the identification of problems linked to the graphic insertion of a wall that has an end at an intermediate point of another wall. If the node is not found on the wall and the distance is less than the tolerance, then the node is highlighted.
Allows the identification of problems linked to the graphic insertion of walls that must have a shared end. If the nodes do not coincide and the distance between the two is less than the tolerance, then the node is highlighted.
Another referred to case is the one represented by the insertion of element nodes on different levels that should belong to the same vertical but in fact appear staggered because of a graphical input incorrect.

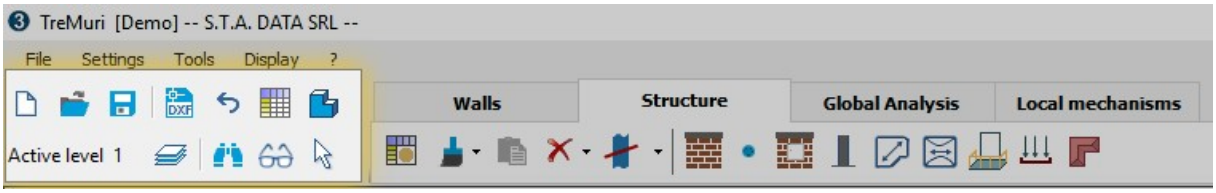


Check Stability nodes:
Checks that there is no nodal lability or absence of constraints. This control is important when foundations are located at different heights or when you examine cases in which are inserted wall panels that arise in false on the beams.

Check slenderness:
After the mesh generation the wall structure is divided into pier and spandrel beam. During the modeling phase, in an attempt to faithfully follow the drawing, it can happen that are generated some piers elements with limited dimensions that on many occasions aren't able to support the weight of the structure that rests above creating instability problems.
This check, highlights the piers with limited dimensions so that the designer can intervene by eliminating those elements.

The use of these verification procedures is recommended to all users, both during the realization phase of the model and at the end before proceeding to calculation.

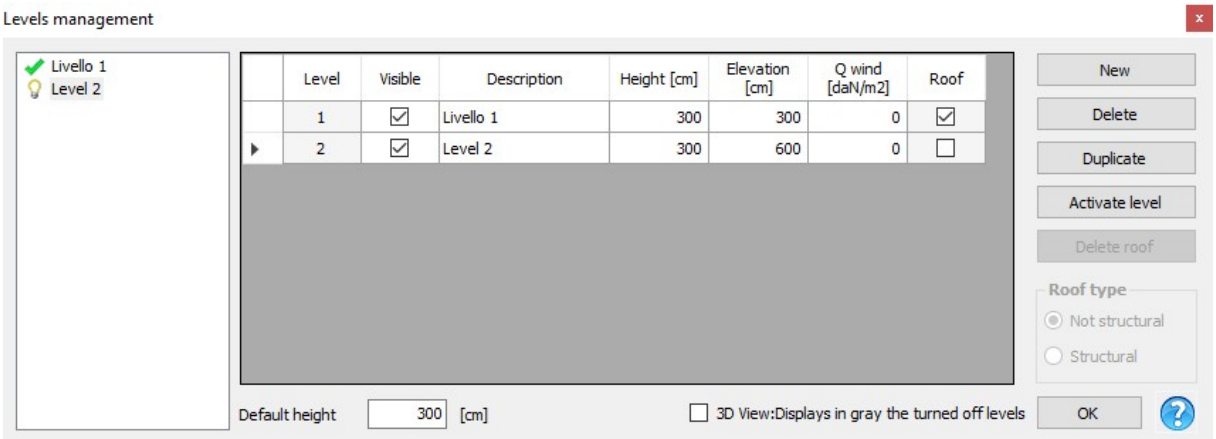
7 Main commands



	New model
	Open
	Save
	Import Dxf
	Undo
	Elements Table
	3D View
	Levels Properties
	Find
	Display options
	Allows multiple selection (only available in 3D mode input)

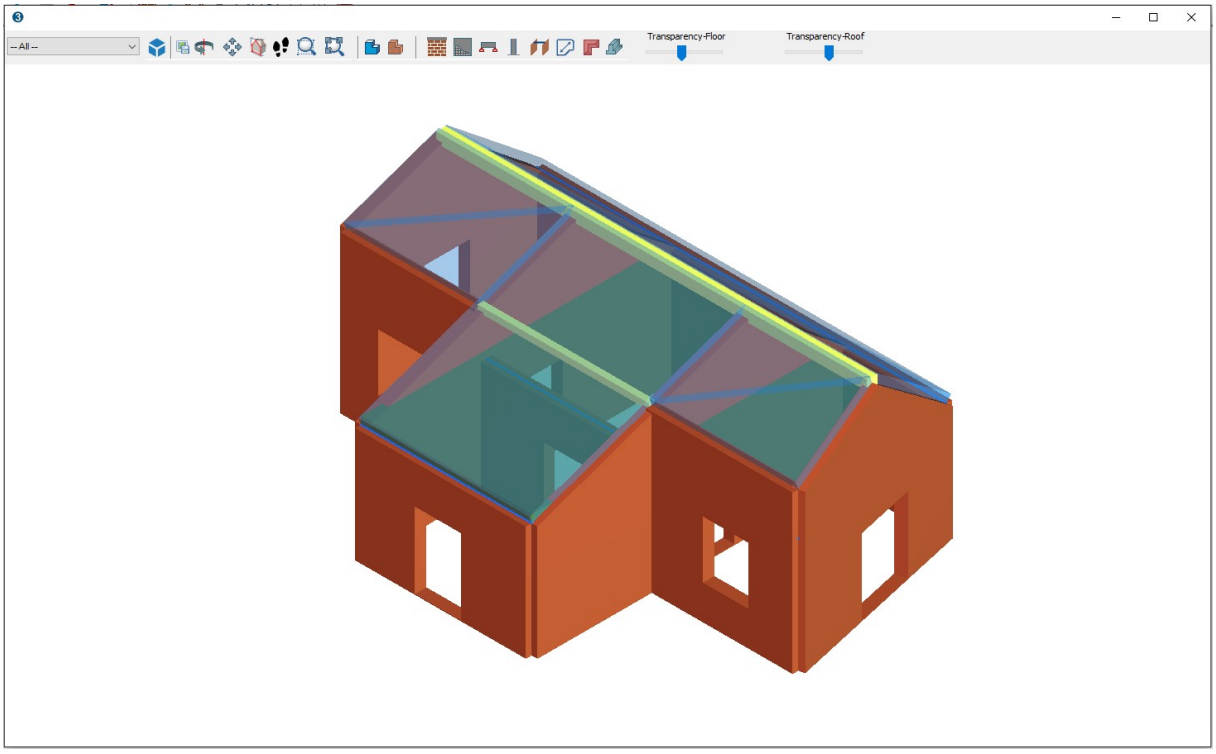
7.1 Levels management

This command permits management of the levels of the structure.
In the associated window, levels can be inserted, using the "New" button. It is also possible to copy a selected level, using the "Copy" button. In addition, the elevation of levels can be modified. To modify a level, it is necessary to render it active using the "Activate Level" button. It is also possible to deactivate the display of a level.
In the last column on the right, it is possible to insert wind loads (average per level). This value is necessary only if static checks are to be performed.

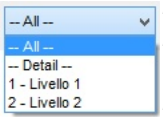


7.2 3D View

Enables the access to a window that shows the structure in 3D view.

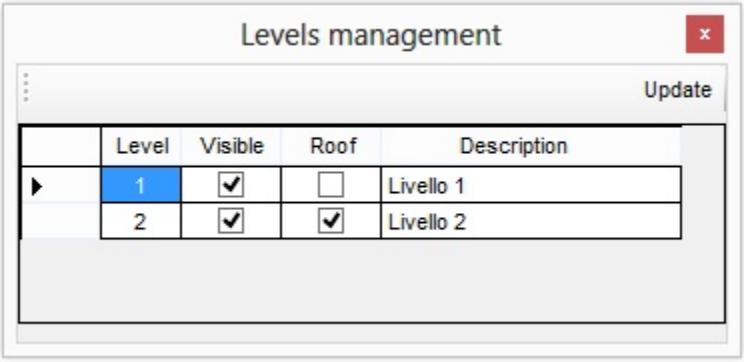


The stories' view filters can be managed by means of a drop down menu:



"--All--" : shows the entire structure
"n-n Storey" : shows the n-th storey (eg, Storey 1, Storey 2, etc ...)

"--Detail--" : appears an appropriate dialog window with more accurate filter options



Instead of seeing all stories or just one at a time you can see a specific combination of stories.
This feature is very useful in cases of particularly complex buildings, whose overall vision may not let us have a good understanding of the position of some fake elements compared to the elements already inserted at the lower storeys.

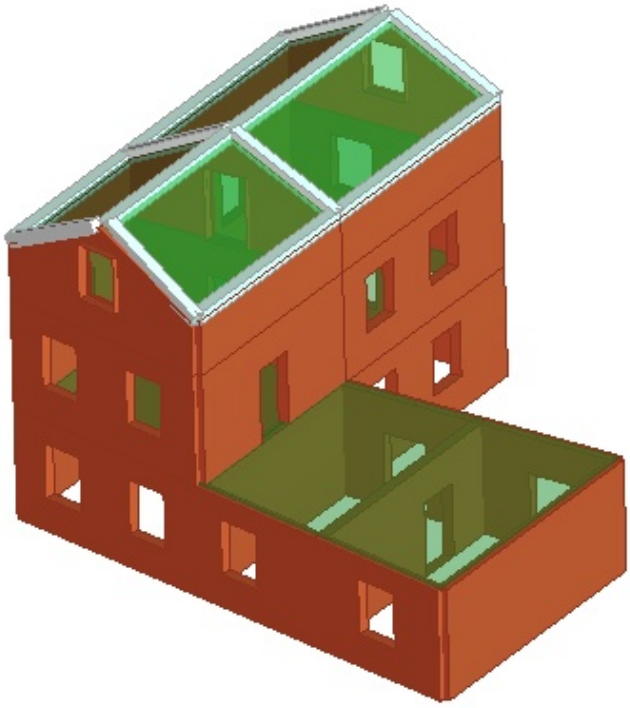
The possibility to hide the roof elements, will be particularly useful in those cases where it is decided not to consider the structural roof and have a clear vision of what will be omitted from the calculation in these cases.

Switch to 3D Input area: This 3D view is only for display purposes, if you want to edit the structure in 3D view it is necessary to switch to the specific working area.

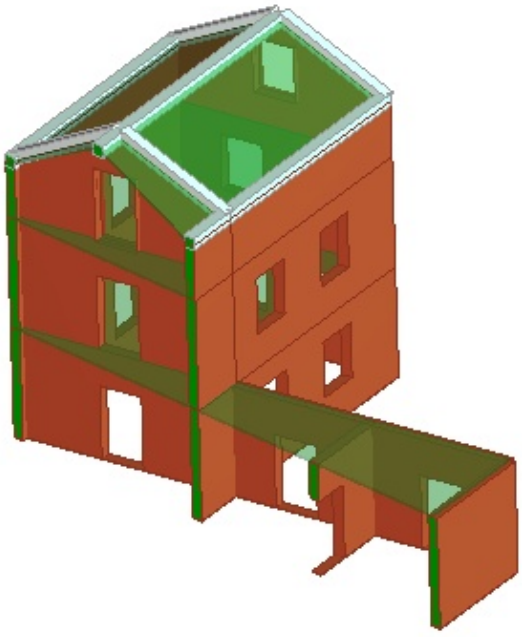
Rotate: A CAD command of the "Orbit" type allows the model to be rotated using the left mouse button. Press [Esc] to exit the command.

Move: A CAD command of the "Pan" type allows the model to be moved using the left mouse button. Press [Esc] to exit the command.

Section: Allows to have a cross-section axonometric view as shown in the following picture



3D without section

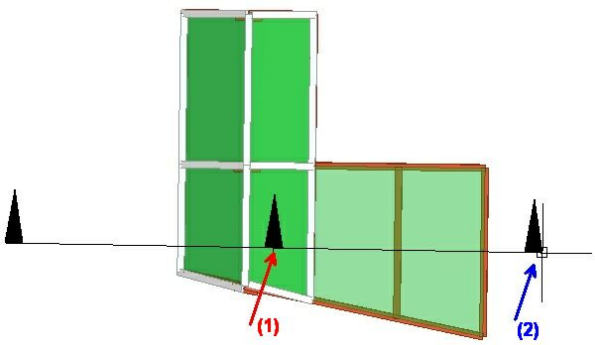


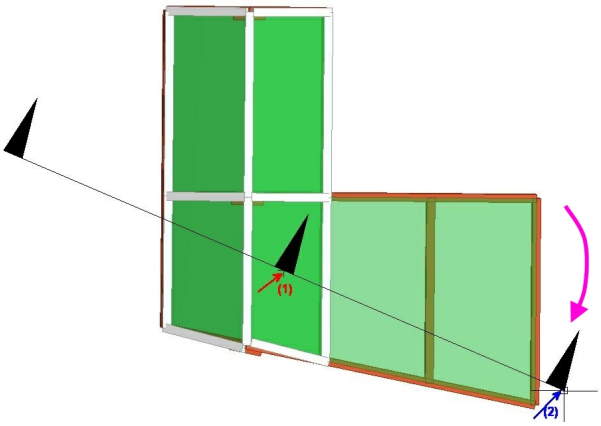
3D with Section

How to insert a section:
By pressing the button associated to this command, you need to identify two points that identify the section plane.

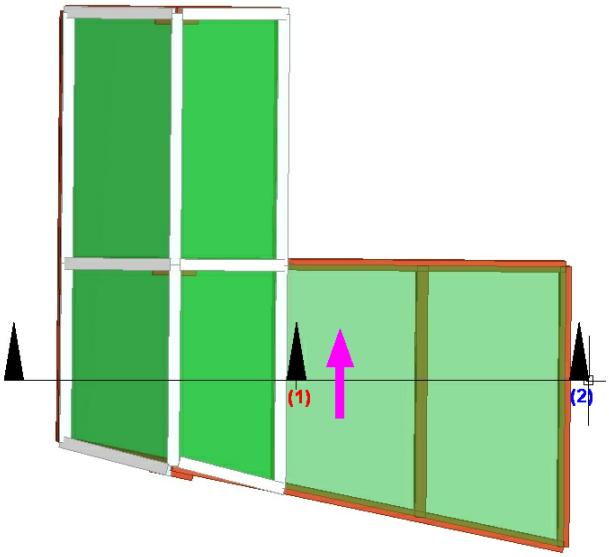
The section planes are always to be considered as vertical, starting from this hypothesis, the identification of the section plane is done by clicking on two points that identify the straight line projection of the plan.

The first **(1)** entered point identifies a first passage point of the section plane.
By moving the mouse the section plane revolves around **(1)**.
Placing **(2)** will uniquely identify both the location of the plane and the direction of the view of the section.

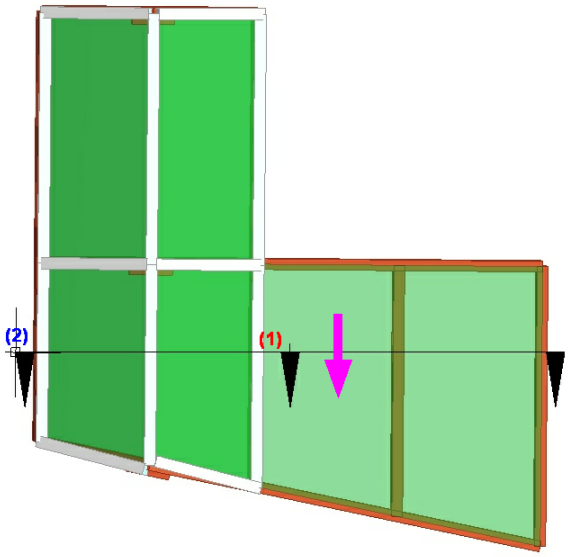




Here is an example with two points of view:



View from the bottom to the top (2) on the right of (1)



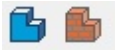
View from the top to the bottom (2) to the left of (1)

To return to the "Standard" view (with no sections) you need to press the button by which you activated the command 

 Walk Through: An interactive system of navigation "through" / "around" the model. To use this function, perform the following steps:

Press the appropriate button to enter the Walk through mode
Use the keyboard to navigate in the model:
[W]: move forward
[S]: move backwards
[A]: shift left
[D]: shift right
[Page ↑]: shift upward
[Page ↓]: shift downward
Move the mouse for positioning the camera
[Esc]: Stops the navigation at the last highlighted perspective view.
 The corresponding button looks pressed, press it again to cancel the command and return to the classic view.

 Zoom: Classical zoom functions: window size, zoom-in, zoom-out

 3D Views: Displays the view in rendering or in wire-frame.

 View Items:
enables / disables the display of elements: panels, RC walls, beams, columns, tie-rods, floors, roofs, foundations.

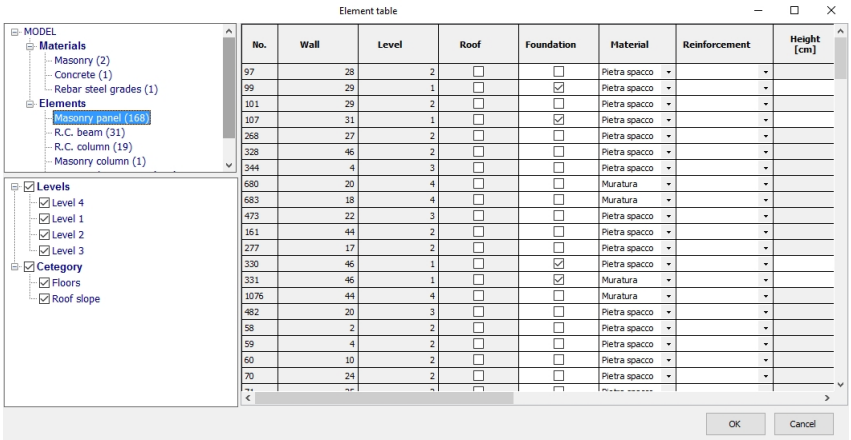
 Save image: allows you store an image of the "3D view" window.

 **WARNING!!!**
In order to ensure a constant update of 3D, you need to close the window of the axonometric view before proceeding with any updates / changes to the model.

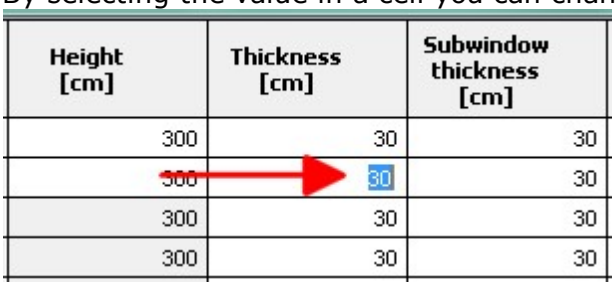
7.3 Table


While opening the table through the proper button, a window that shows the user's inserted data through the graphics will appear. The drop down menu on the left makes the navigation inside the table easier.
The drop down menu is organized in four main branches:

Materials: This contains the material typologies used in the project, with their mechanical characteristics.
Reinforcements: Contains the features of the masonry reinforcement type.
Elements: This contains the elements used, divided by typology (according to that indicated in the characteristics definition window, described below), grouped by level.
Loads: Contains the characteristics of the concentrated/linear loads applied to the structure.

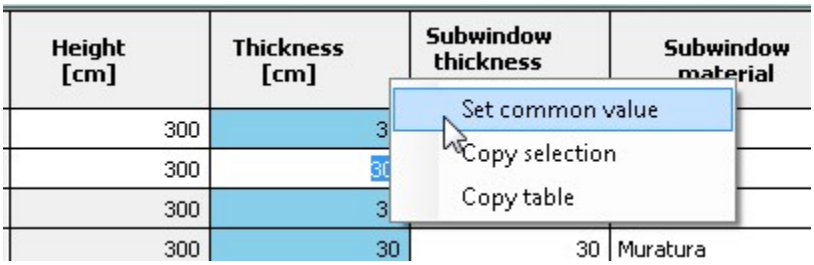


The present data can be edited and stored with the new value.
Every modification in the table implies changes of the template data.
Only the fields with a white background are entitled to modification, the gray background indicates a not editable fixed field.
There are two types of modifications in the table:
- Single edit:
By selecting the value in a cell you can change it directly.

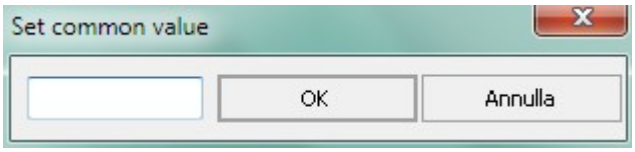


Height [cm]	Thickness [cm]	Subwindow thickness [cm]
300	30	30
300	30	30
300	30	30
300	30	30

- Multiple edit:
After you select a column, you can pop up a context menu by placing the mouse on the column header and pressing the right button.



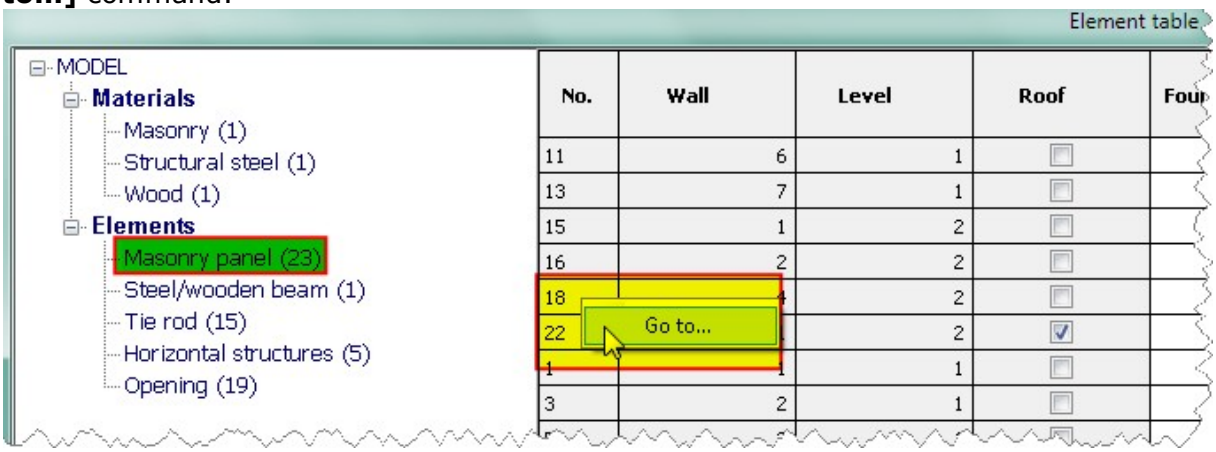
Select "Set common value" shows a small window to define a single value common to all elements.



This option is useful when you decide to apply some modification to a group of elements.

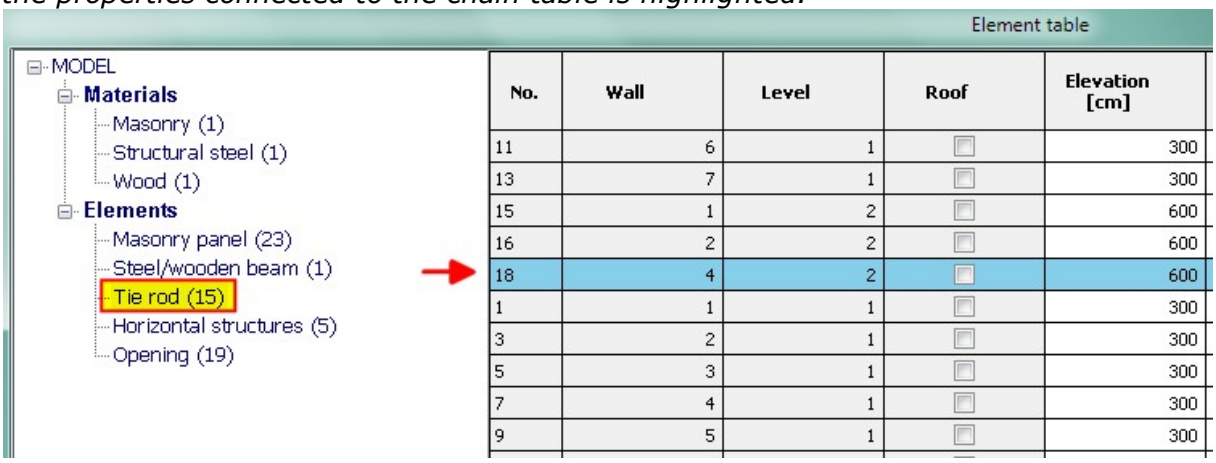
All the "***composed***" structural elements (panel+ridge; panel+chain; panel+steel/ wood beam) are divided into their constituent elements.
This breakdown allows for example to change the characteristics of all the walls regardless if they are dominated by chains, curbs or by any other evidence.

Through the access to the table elements, is possible to distinguish a "composed" element from a "simple" one by the symbol that indicates the connection with another element.
Selecting the symbol with a click on the mouse's right button displays the **[Go to...]** command.

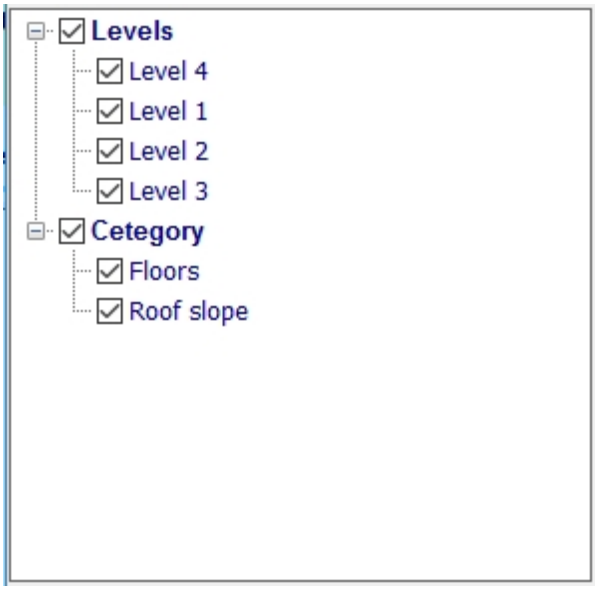


By selecting this command, you get redirected to another table where the row corresponding to the element coupled with the first element is highlighted.
<u>Ie:</u>
All the elements that make up a composed entity have the same identification.

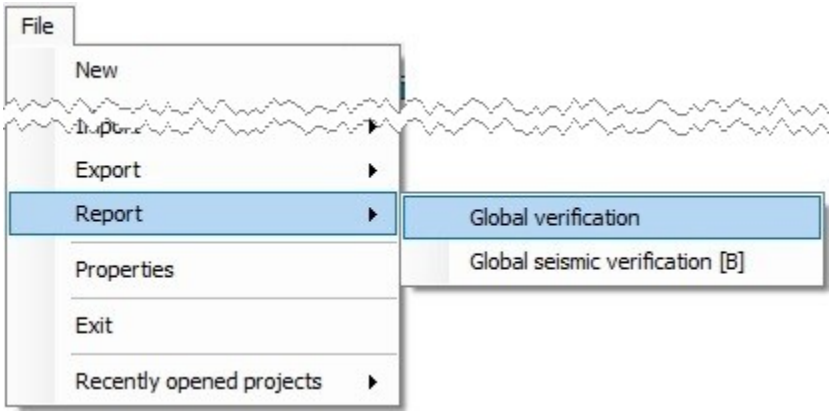
For example, if from the "wall panel" table, is selected the [Go to ...] corresponding to the panel 18 you get redirected to the "chain" table where the same element 18 with the properties connected to the chain table is highlighted.



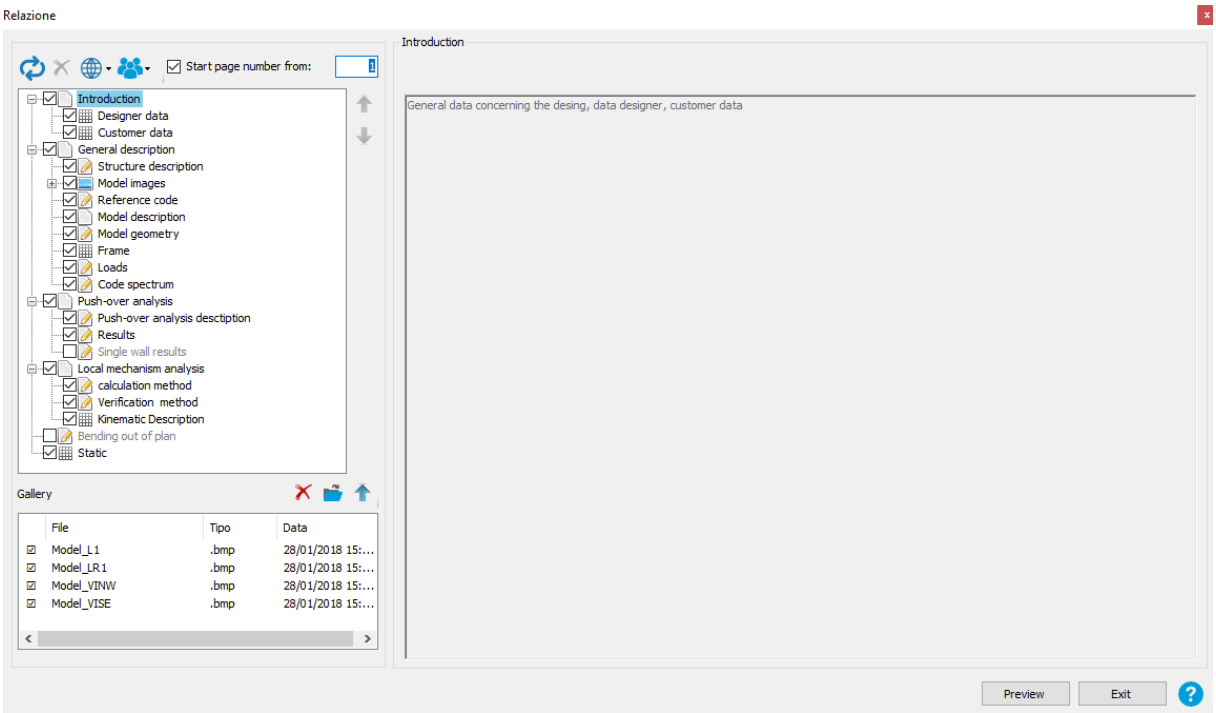
In the lower part of the table is shown a filter function tree that allows filtering the contents of the table based on levels and categories of interest.



7.4 Report



The report tool allows the user to create project reports automatically.
Using the arrow on the right of the report button, the user can prepare either the complete seismic verification report or the synthetic one.
When the button is activated on the tool bar, the report creation window opens.



On the left the subjects for designing masonry buildings are found in order. The user can decide what to include in the report by selecting the box on the left of the descriptions.

This command regenerates the original report by canceling all changes made by the user in this environment.

It allows to delete an image from the report

This button allows you to select the language for the report.

Designer-costumer data:

- A special mask requires the data of the designer (memorized for all future projects)
- A special mask requires the customer's data

In the lower part of the screen there is a gallery of images that the user has saved during the design phase, using the save image command.

This command (present in the 3D view) saves the image seen on the screen at that moment.

In all other graphic environments, the access to the image capture command is made using the context menu that is obtained by pressing the right mouse button in the graphics area.

When entering in the report environment, a gallery of the main images (plans, levels, 3D views) is automatically created and automatically arranged in the "General Description" chapter.

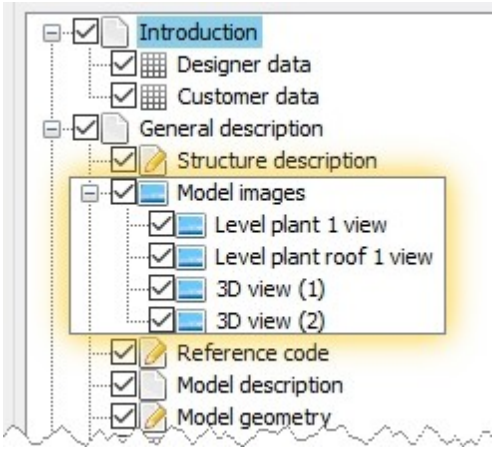
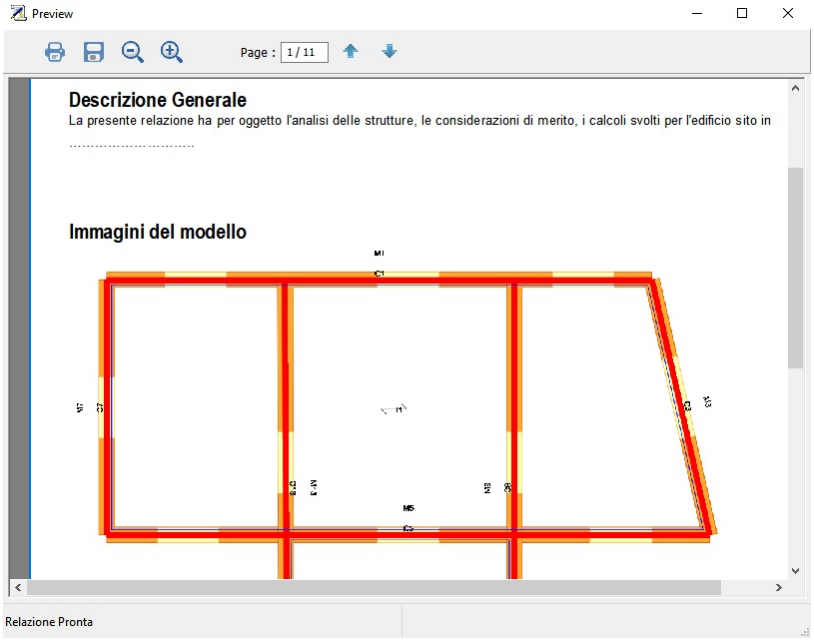


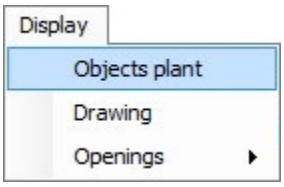
Image Gallery Commands	
	Deletes an image from the Image Gallery
	Imports an external image to the Image Gallery
	Inserts an image in the report scheme
	Moves an image inserted in the report, allowing the user to decide where to place it

When the "Preview" button is activated, a print preview is shown, allowing the document to be seen.



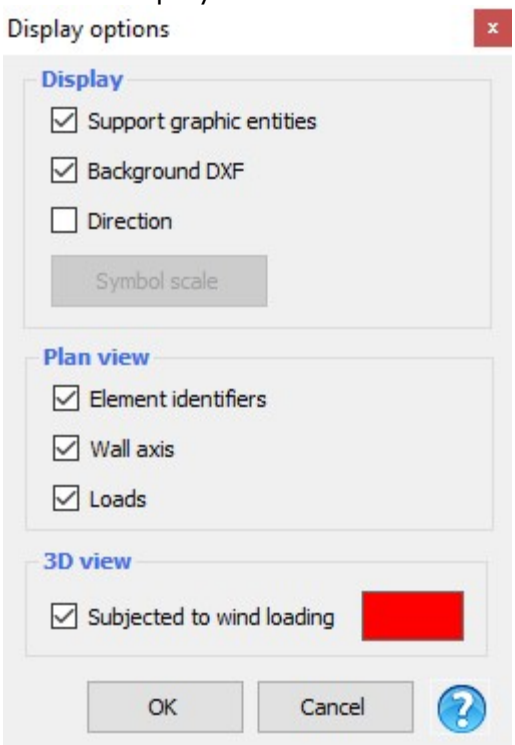
With "save command" you can export a "RTF" file.

7.5 Display Parameters



Plan view of the level: This command schematically displays the plan of the active layer with the representation of the various types of structural elements defined in the structure.

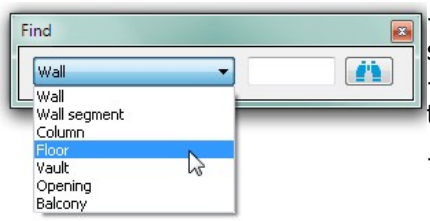
Display Options: This instrument provides to the user the opportunity to decide what to display.



"Direction" shows the walls' local reference system determining the wall panels' eccentricities sign referring to the wall.

"*Loads*" allows to show/hide loads concentrated/linear showed in the plan.
"*Exposed to wind*" in the 3D input/editing mode you can choose to directly see the walls affected by the wind load in a different color.
The color used to indicate the wind exposed panels can be changed by selecting it in this window.

Finds objects by their number: The "Find" command searches in the graphics for a wall, a wall segment, a slab, a column, a balcony if is known the identifier.



- Select from the menu the type of item you want to search.
- Enter the number of the element to be found in the text field
- Press to start the search.

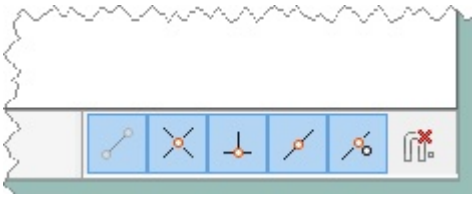
The search result is shown by placing the searched element in the middle of the video, with the mouse pointer on it and a special marker that highlights it.

Please note that the wall sections are parts of wall (eg M33, T122, C54). In the proper area must be entered the identifying number and not the precedent letter in order to indicate the item type.

This command (present in the 3D view) allows saving the image displayed at the time on the screen.
In all the other graphical environments, the access to the image capture command is done through the context menu that is obtained by pressing the right mouse button in the graphics.

7.6 Snap

The program is equipped with an automatic recognition system of the remarkable points on the support graphics, on a typical imported dxf generic cad system or walls.
The same just described snap are available during the walls insertion.
On the bottom right, in the graphics area is available a snap command bar.

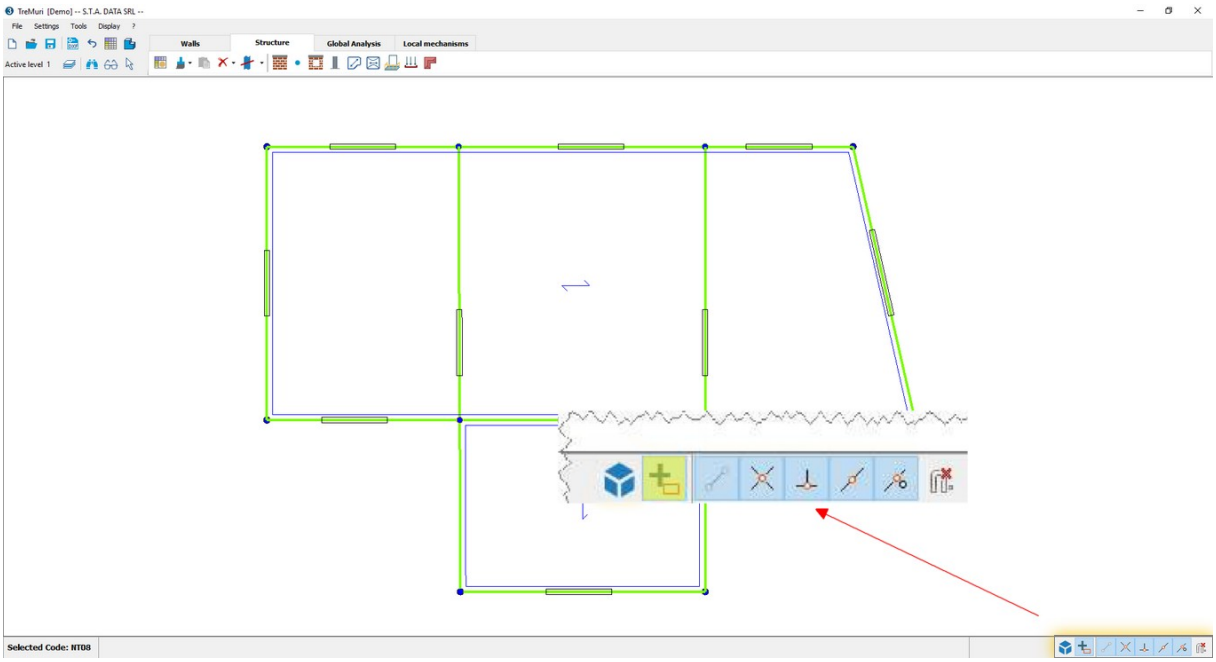


Below the list of available Snap:

	Extreme
	Intersection
	Perpendicular
	Middle
	Line
	Uncheck all Snap

7.7 Dynamic Input

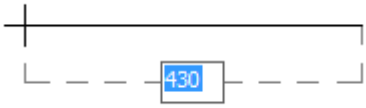
The command ***Dynamic Input*** is accessible from the command bar placed at the bottom right of the screen input.
The corresponding button appears pressed with the activated "dynamic input" mode, to disable this mode it is necessary to press again the same button



When this work mode is enabled, the input/modification commands are supported by a line of dynamic elevation that shows the dimension and location of the various objects. Let's see some application examples of this command.

Elements Length

This input mode is used by the graphical support of walls and lines.



The entry of this type of element is carried out by two points:

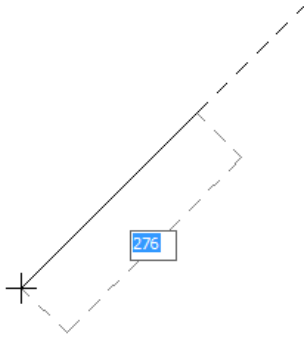
1. The first point can be defined by the graphics snaps or through the insertion of the absolute coordinate.

You can switch from the [X] coordinate to the [Y] coordinate by pressing the [TAB] Keyboard.

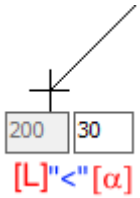
2. The second point can be defined in coordinates.

Polar [Distance=*{numerical}* ; Angel=*{mouse}*]

Shows the distance from the first point, at intervals of 45° is enabled the display of a guideline to which you can attach the wall/line tracking.

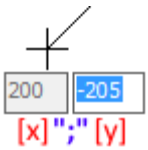


Polar [Distance=*{numerical}* ; Angel=*{numerical}*]



After entering the length of the elevation by pressing the symbol **"<"** on the keyboard you switch to a second text field that allows defining the angle that forms with the positive semi-axis of X.

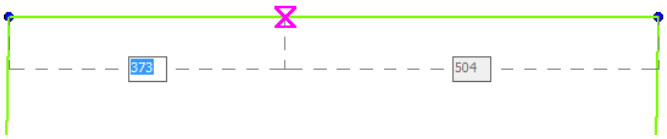
Relative [X=*{numerical}* ; Y=*{numerical}*]



Relative coordinates with the first point line origin.
Enter the [x] value in the text field set up for the length; pressing the button under the **";"** symbol activates a second text field set up for the [y] value.

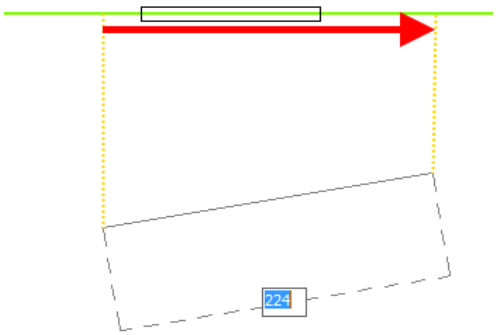
Placing objects on walls

The main application of this command is focused to insert nodes and openings at a specified distance from the extremes.



Moving Objects on walls

Movement of an opening or node of the element according to a vector projected on the wall of interest.

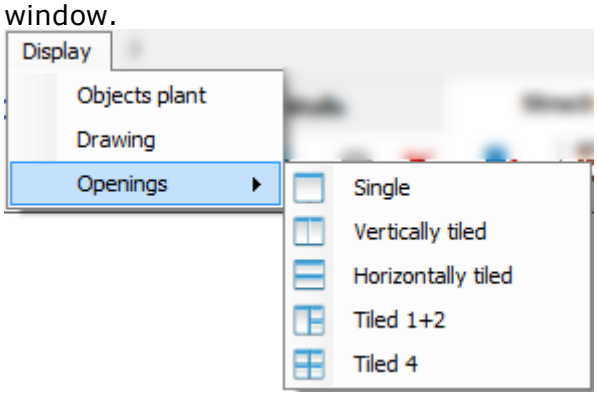


7.8 Selection Mode

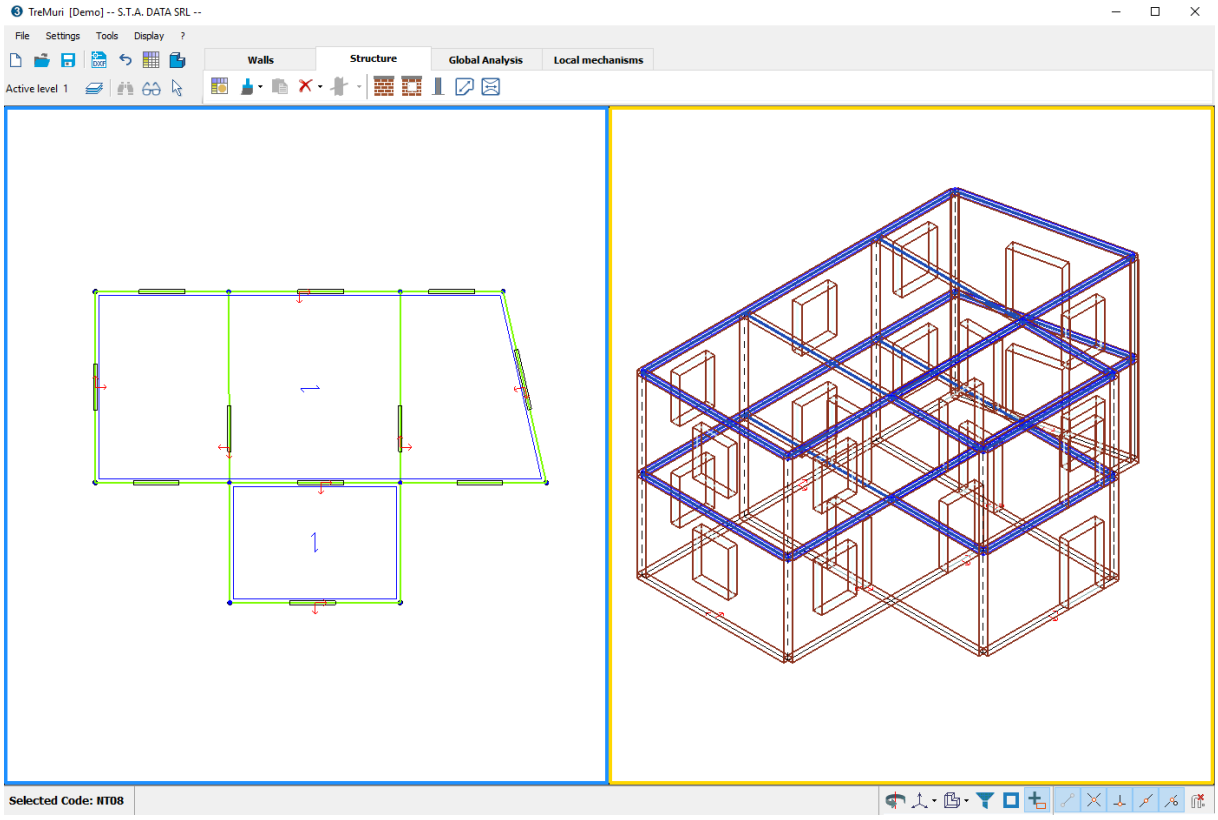
The program offers various selection modes.
Single selection: Each element is selected by clicking on the entity
Multiple selection: This selection mode has three sub-modes.
Various elements can be chosen in sequence, or through window mode selection. All elements that are contained within the window and intersect its limits are selected.

7.9 Multiple view

From the Display menu is possible to select the display mode of the preferred multiple window.



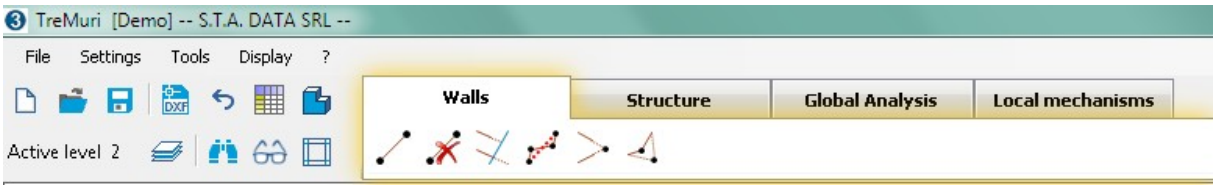
On each graphical area is possible to define a different display mode.



The active view is surrounded by an orange border.

NB: The multiple view mode is available only in the structure environment.

8 Geometric Definitions



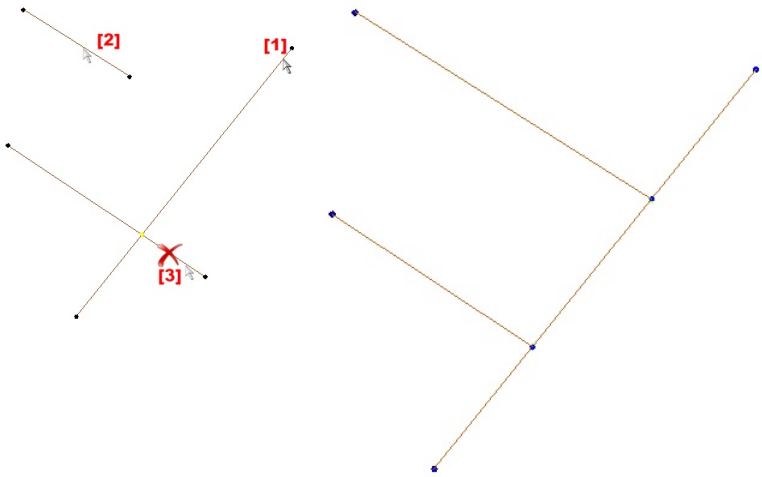
Insert wall: allows to insert a wall.
For wall is meant a continuous length of masonry, R.C. walls, beams, tie rods (it can also be multiple segments all resting on the same tangent).
Insert a new wall: by clicking on the mouse's left button can be defined the subsequent nodes that identify one or more walls;
Terminate inserting a wall: right click on the mouse to exit the command.
The insertion of the elements can be easily performed through the snap (to the graphics or on the walls).

Remove wall: allows to delete a previously inserted wall

Extend/Trim Wall: allows to lengthen or shorten an existing wall.
1. select the wall **on which to extend the other walls**
2. select one or more walls **to extend**

Example:
[1]: select the wall on which to extend / cut
[2]: select the wall to be extended
[3]: Select the wall to be cut from the side you want to delete

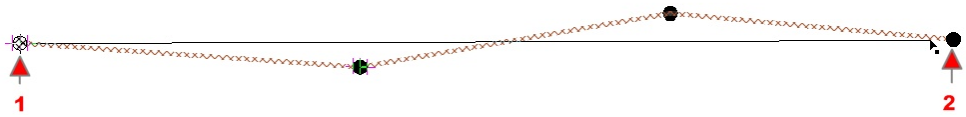
Before **After**



Rectify walls: Allows to rectify the previously entered walls.
The figure below shows three different walls with a slope close to each other. This may result from a not very accurate background DXF that led to repolish the plant through segments not well aligned.

When the change of slope is so contained, the designer may decide to replace the sequence of three walls with a single wall (shown in red in the figure above) in order to simplify the calculation model without losing precision.
The rectification command replaces a sequence of adjoining walls with a single one.

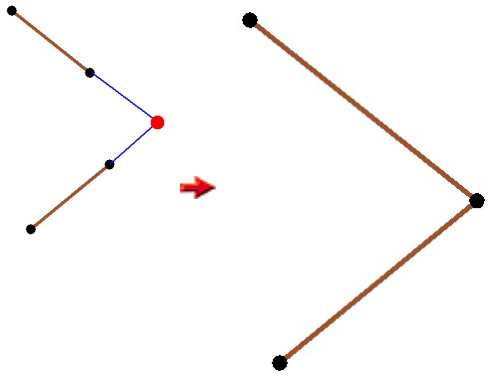
Following the steps of how to use this command:
- sequentially select the contiguous walls with the left mouse button;
- confirm the selection by pressing the right mouse button;
- identify the rectification directrix indicating the two extreme points (the points should be identified between the extremes of the selected walls).



The result of the rectification operation will be the following in the [**Walls**] area.

The result of the rectification operation will be the following in the [**Structure**] area.
In the structure area, the wall now defined as one, will be separated into multiple segments (through the element nodes) at the extremes of the wall, removed after the operation rectification.

Fillet walls: allows to join two walls that do not cross.
Select two walls to be connected with the left mouse button

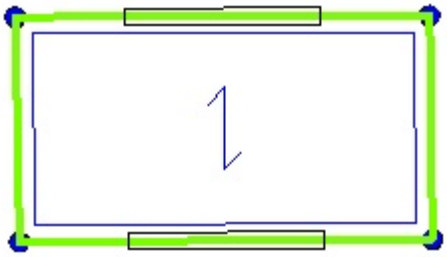
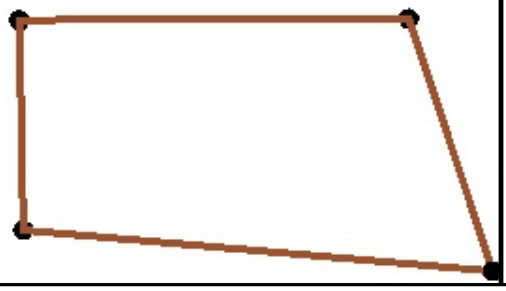
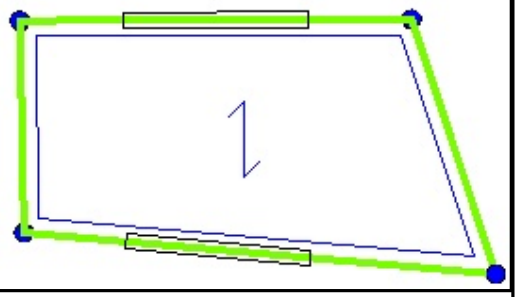


Stretch: This command allows to move an outer node of the wall.

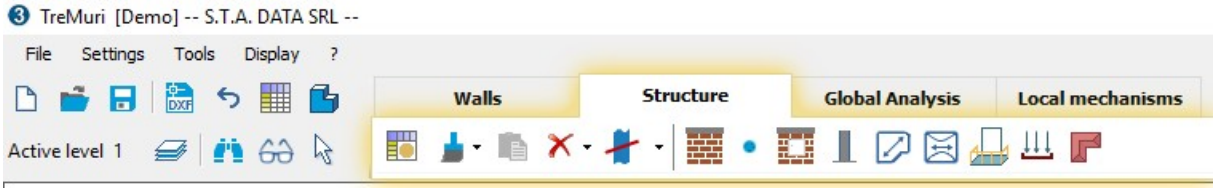
After using the command:
- Select the node to be moved with the left mouse button
- Click at the point where you want to place the node

The editing operation involves the automatic adjustment of all the structural objects directly attached to that node.

	Walls Area	Structure Area

	Before		
	After		

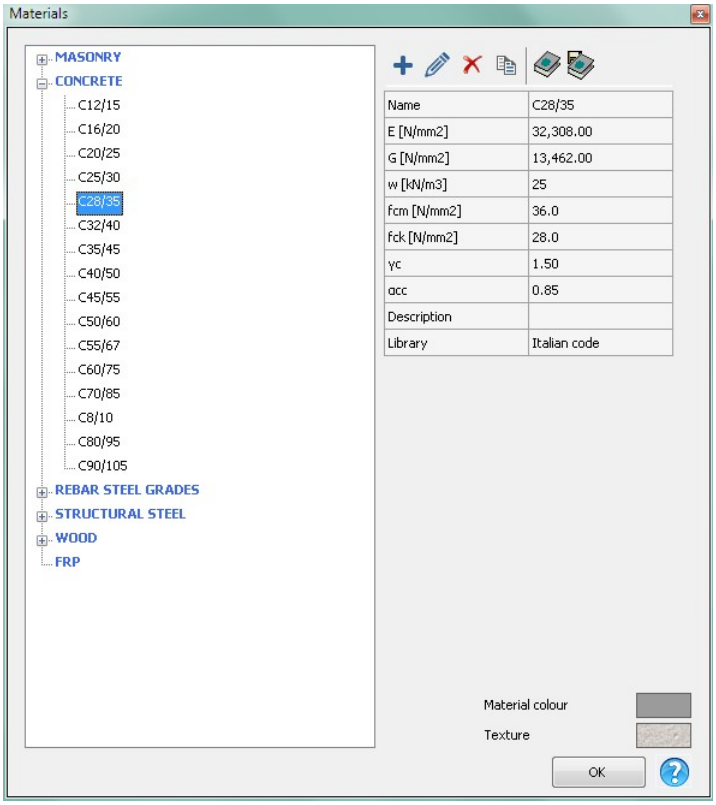
9 Characteristics of the Structure



	Materials
	Copy Propriety
	Paste Propriety
	Delete the selected structural elements
	Supported thickness (read the panels thickness from DXF file)
	Assign attributes of the wall segments
	Graphic point insertion/Distance insertion
	Opening
	Column
	Slab
	Vault
	Balcony
	Load
	Roof

9.1 Materials

Select the icon, and window will open. In the window are the characteristics of masonry materials (concrete, steel, and wood) generally used in structural objects (masonry panel, tie rods, beams, columns, and floors). It is possible to modify or create new mechanical characteristics for the materials. Use the right mouse button and select "Modify" or "New". It is also possible to use the appropriate buttons.
The defined mechanical characteristic values both for the predefined materials as well as for those that must be defined refer to average values.
The concept of knowledge level is present only for the definition of existing material typologies and serves to define the confidence factor that the program will apply to the average resistance.



	Create a new material of the selected typology
	Modify an already defined material
	Delete a material
	Copy a material
	Library explorer
	Save User library

Each material is associated with a color chosen by the user. It is then used in the 3D display window.

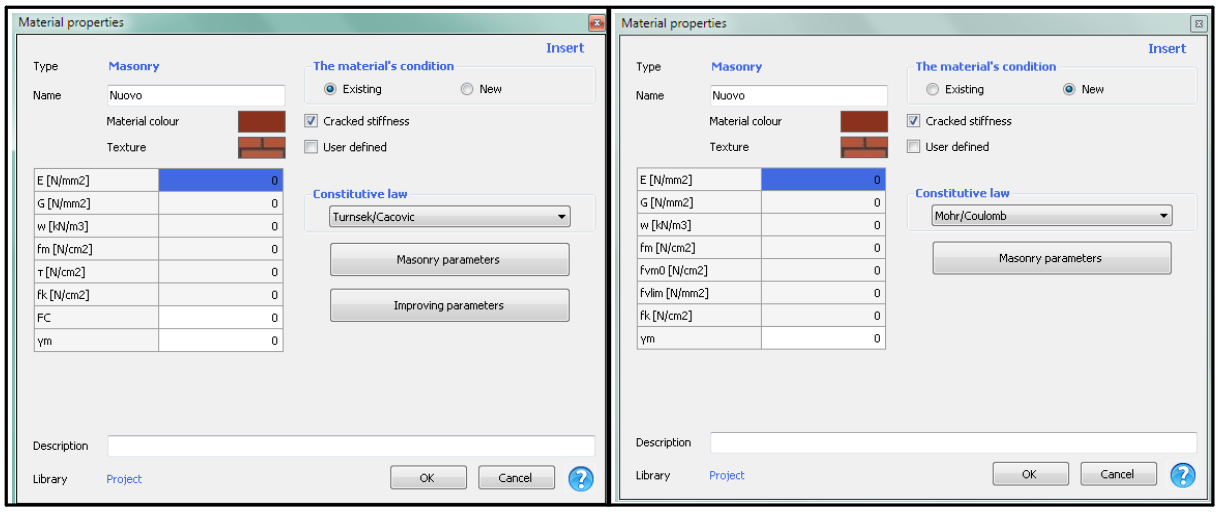
9.1.1 Muratura

Enter topic text here.

9.1.1.1 NT08/Eurocodice/SIA

When a new typology of masonry material is inserted, there are two options:

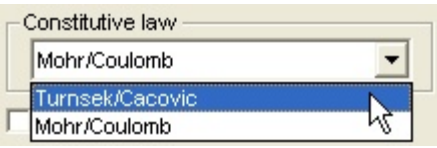
Existing Material	**New Material**



E: longitudinal elasticity module
G: Shear elasticity module
w: specific weight
fm: Average compressive strength
fvm0:*(Mohr-Coulomb)* The average shear strength without axial action
fvlim: *(Mohr-Coulomb)* The shear strength limit (suggested value 2.2 N/mm2 *§7.8.2.2.2 - D.M.14-01-2008*)
τ:*(Turnšek Cacovic)* Shear Strength
fk: Characteristic compressive strength
γm: Material security factor
FC: Confidence factor
fm, fvm0, fvlim, τ:The values listed on the form are to be considered NOT reduced for the confidence factor (CF), the reduction will be applied directly in the calculation phase.

In the window that allows you to enter the characteristics of masonry material, there are displayed buttons that support the user in identifying these parameters.
Alternatively to the use of such windows, the user may decide to directly enter the values of the characteristics.

For both existing and new material you can decide the type of shear bond to be used:
- Turnšek Cacovic criteria
- Mohr-Coulomb criteria



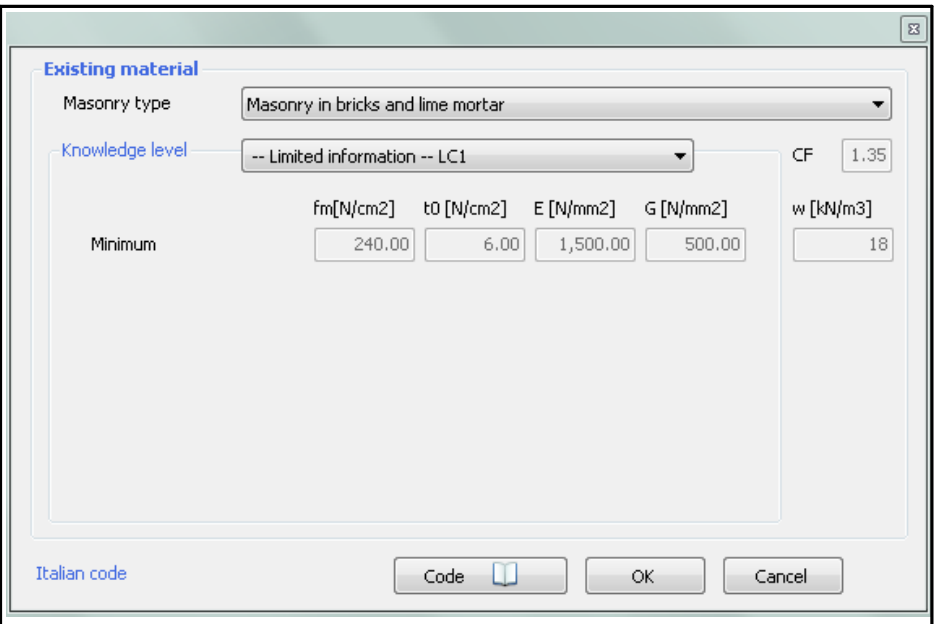
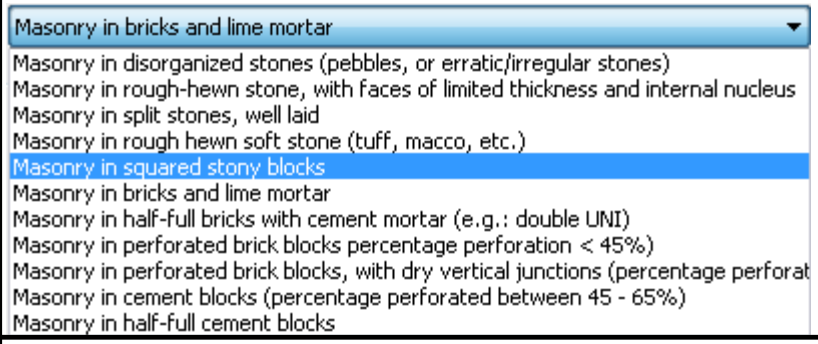
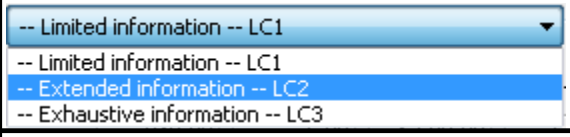
The **Turnšek Cacovic** criteria represents a type of diagonal shear failure and it is recommended to use especially for *existing walls.*
The **Mohr-Coulomb** criteria represents a type of sliding shear failure and it is recommended to use especially for the new masonry.

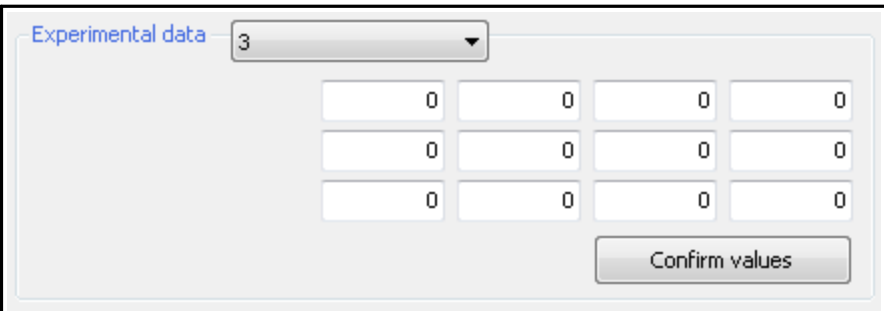
In the case of existing masonry made of clay, it makes sense deciding to adopt a Mohr-Coulomb failure criteria in order to examine a failure criteria more appropriate for the type of examined walls.

In the lower part of the screen displays a button **[Masonry parameters]** that invokes a help to fill the mechanical properties.

	Turnšek Cacovic	Mohr-Coulomb
Existing material	Filling Parameters 1	Filling Parameters 2
New material	*No help to fill*	Filling Parameters 2

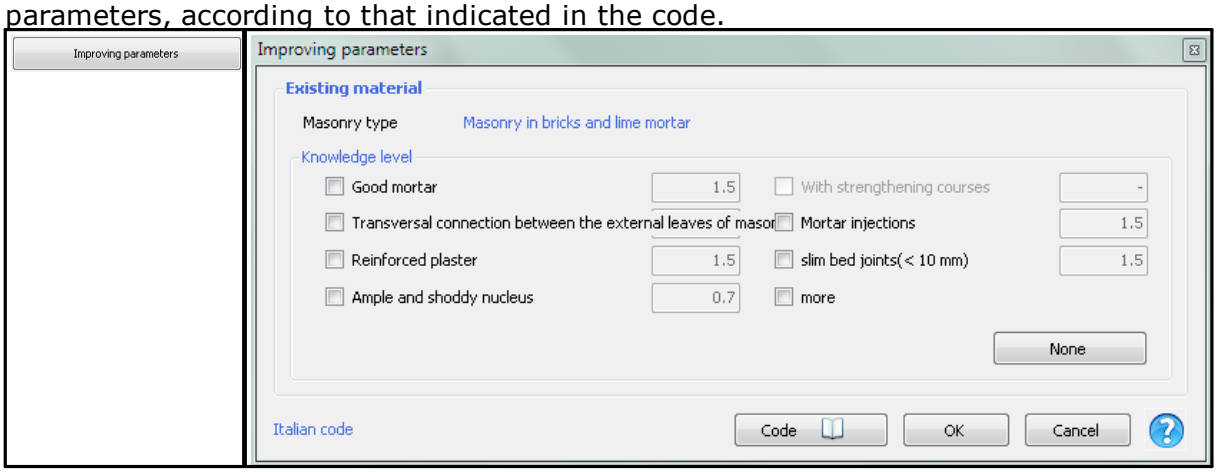
9.1.1.1.1 Filling Parameters 1

<table>
<tr><td> 
</td><td> 
</td></tr>
<tr><td>Type of masonry</td><td> 
</td></tr>
<tr><td>Knowledge Level</td><td> 
</td></tr>
<tr><td>The values for the characteristics are automatically provided.</td><td> 
</td></tr>
<tr><td>If working with knowledge level 3</td><td>The experimental values derived from the tests are requested.</td></tr>
</table>

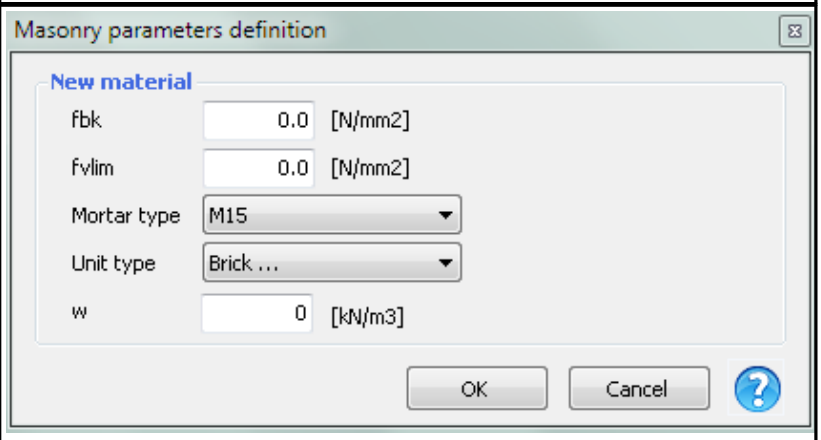


Clicking on "Confirm values" defines the computation values.

After having defined the material characteristics, it is possible to define improvement parameters, according to that indicated in the code.

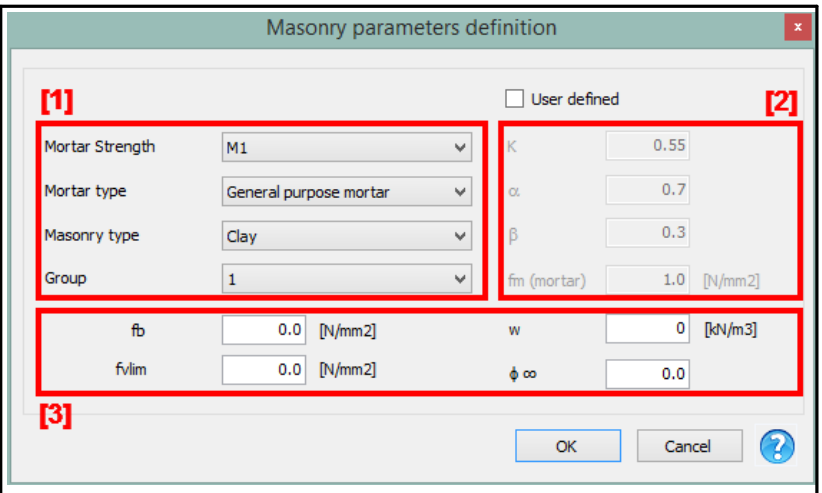


9.1.1.1.2 Filling Parameters 2

Technical Standard of Constructions	**§11.10.3**
Masonry parameters	 **fbk**: characteristic compression strength **fvlim**: shear strength limit **Mortar type:** classification of mortars

Eurocode	**EN 1996 §3.6**

Masonry parameters



If **"user defined"** <u>is not selected</u> it is possible to use the parameters in [1] in order to define the coefficients in [2]:

- §3.6-Tab 3.3 ➔ K
- §3.6.1.2(2) ➔ α, β

If **"user defined"** <u>is selected</u> it is possible to insert directly the coefficients in the space provided [2]:

- Input value for K, α, β, fm(mortar)

In [3] the parameters that must be entered manually

- fb : characteristic resistance of the block
- §3.6.2(3)NOTE➔ fvlim (0,065 fb or fvlt)
- ϕ∞
- w

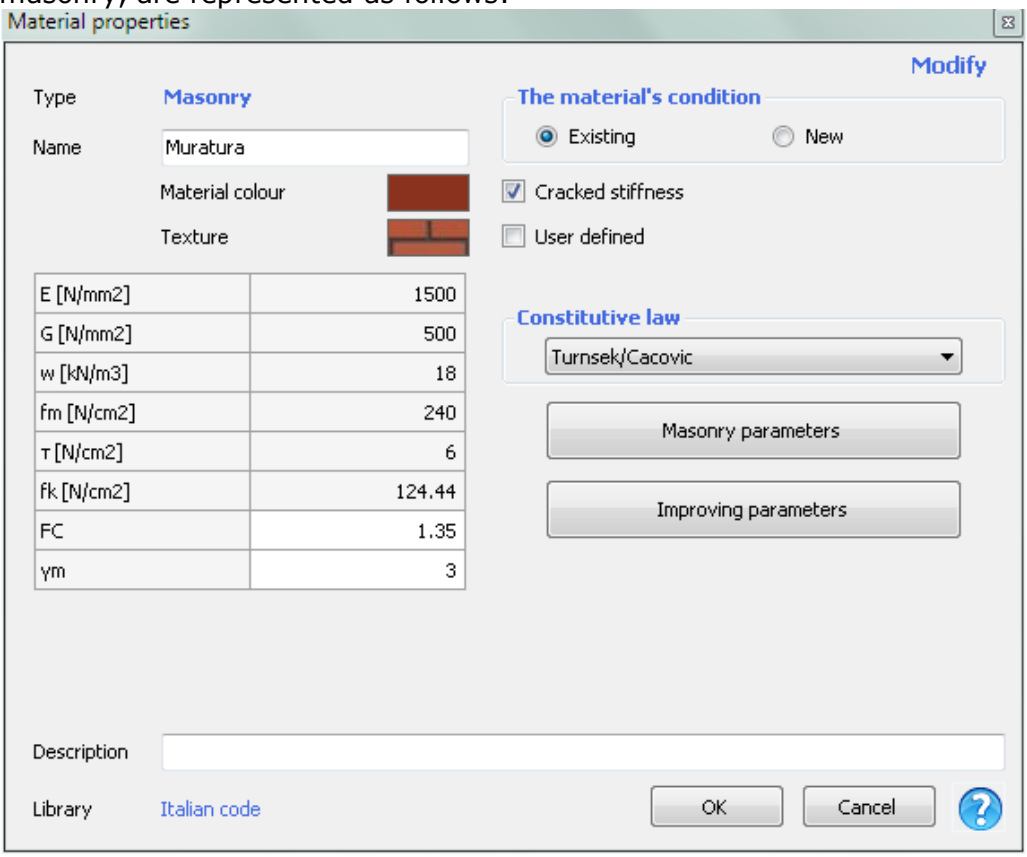
When you press [OK] are performed the following operations:

- §3.6.1.2(1) ➔ $f_k = k \cdot f_b^{\alpha} \cdot f_m^{\beta}$ ➔ $f_m = f_k/0.7$
- §3.6.2 -Table 3.4 ➔ fvko ➔ fvmo =fvko/0.7 (avarage value)
- §3.7.2(2) ➔ E (elasticity modulus)

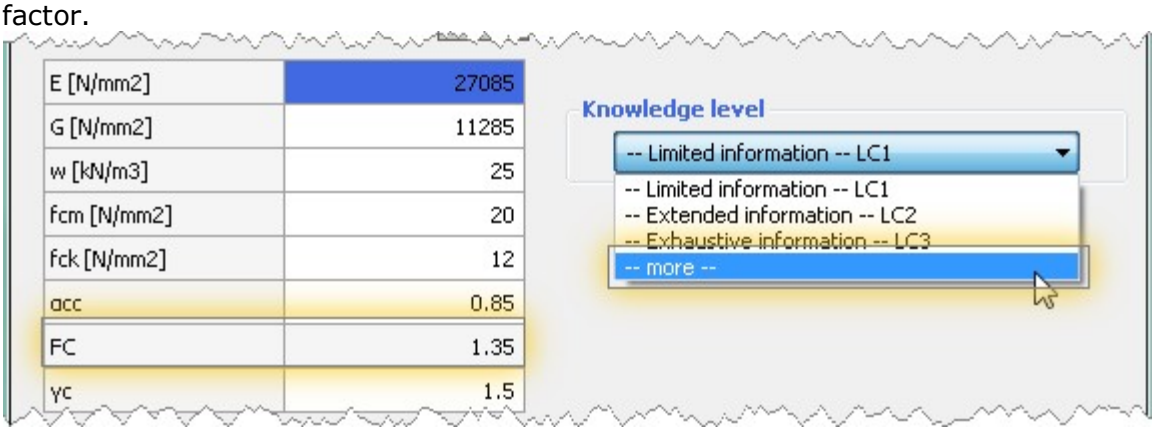
§3.7.3(1) ➔ G (shear modulus)

9.1.2 Others materials

The windows for the definition of the mechanical properties of different materials than masonry, are represented as follows:



By selecting "**More**" in the drop down menu, you can directly enter the confidence factor.



9.1.3 Materials Library

This function allows the designer to import on the project in exam the materials from different libraries (other Design Codes) or from the user library.
3Muri program has 3 main libraries types:

- Library Project: Materials collection contained in this project, shown in the material

dialog window (these materials are only available for the active project).

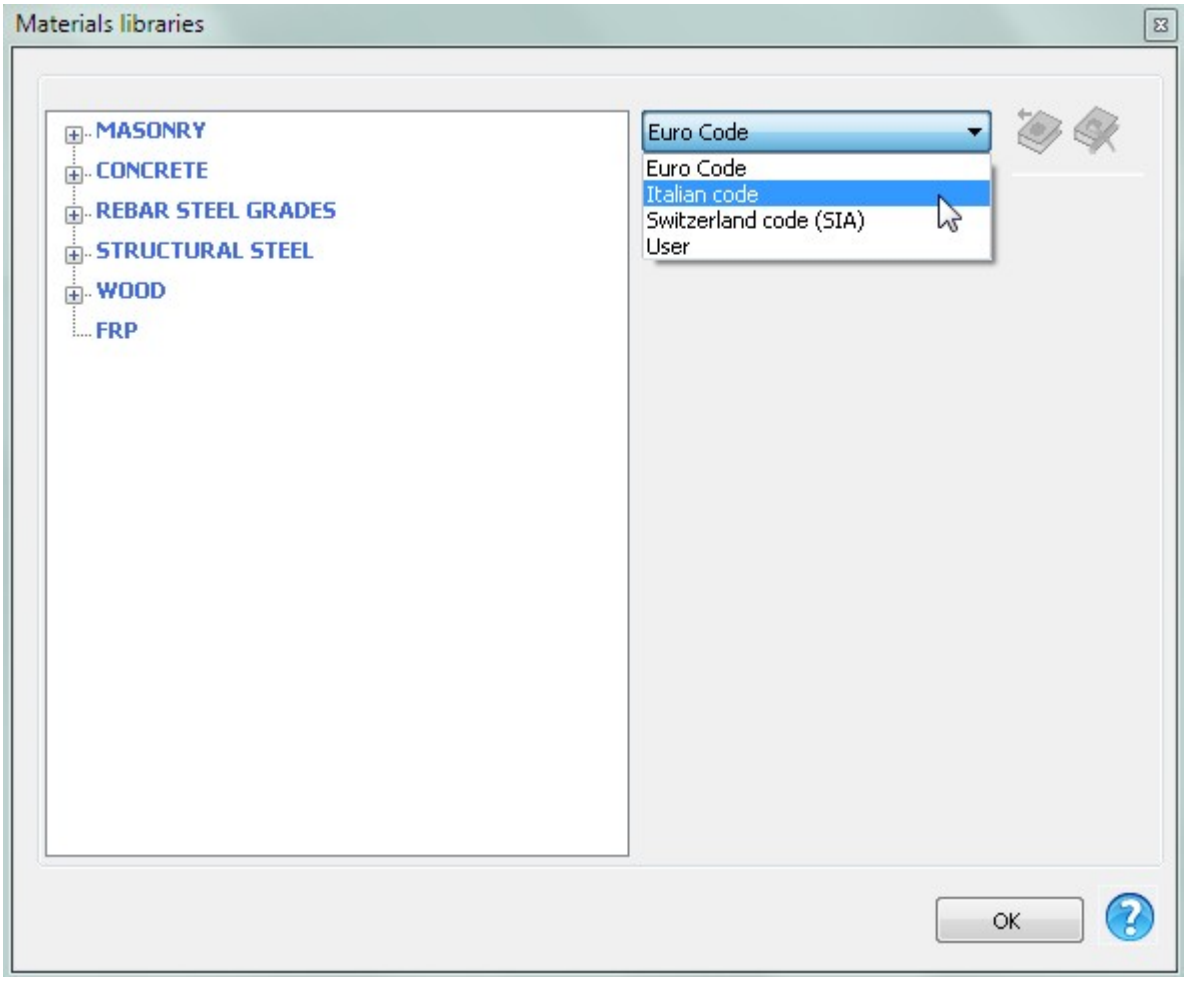
- Design Code Library: The material properties are defined as indicated by the various Design Codes. There is a library for any Design Code. At the moment you open a new work is uploaded to the library project the contents of the selected corresponding Design Code.
- Library User: It is empty by default and is filled by the user according to his needs.

If you use very often the same types of masonry materials it can be stored in the user library to use it in future projects.

User Library

After defining a new or existing material, will be shown in the tree to the left of the window material.
The defined material is now available within the project, if this material is usually re-used for other projects different from the project on which you are working, you can save it on the user library to be able to retrieve and use later in different models.
To use the material created in the current model or in a different model, after you create it you must select the name and press "save in the library"

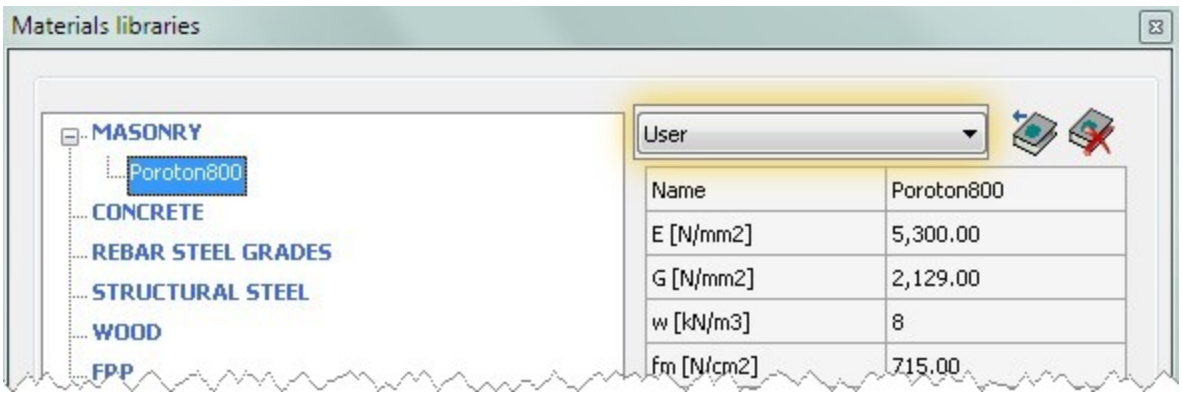
When you open a different model and you want to import a material into the design library from the library user, proceed as follows:
Library: Open the Material Library



The materials presented in the tree on the left are those in the selected library in the drop down menu.

The availability of design code libraries is affected by having the Design Code in its license contract.

Selecting "User" shows the user library.
Entering in the tree you can select the material that you want to import into the project.



Copy in the project
Allows to copy the selected material in the project library, making it available for the active project.

Delete from library:
Allows to delete the selected material from the user library.

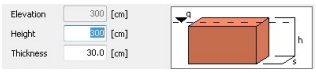
9.2 Definition of Structural Objects

To refine the computation procedure, the program examines the non-linear behavior of the elements. (see the theoretical information found in the introduction) Given the definite non-linear behavior of the macro-elements, it is necessary to perform an mixed structural analysis that is sufficiently accurate. This must examine the non-linear behavior of the other elements that work together with the masonry as beams and columns. (many of the parameters required in the element input phase are necessary for correct computation of the non-linear analysis)

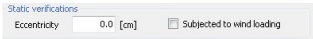
Define characteristics: once the button is activated, the cursor changes shape and allows selection of one or more objects, whose structural characteristics can then be defined. Clicking the right mouse button, a window opens. In this the structural objects to be assigned to the selected walls can be chosen.

For all the elements that can be inserted there are two areas: one for insertion of geometry, and the other for insertion of material.
Insertion of material provides the possibility to choose the materials that will enter into play in the definition of the structural element. For example for an R.C. beam it is necessary to insert the characteristics of the concrete and the steel.
The geometry area changes depending on the element and it is described in detail below.

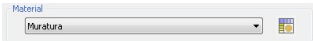
9.2.1 Simple Elements



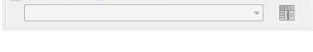
-*Elevation*: The maximum elevation of the panel
-*Height*: Height of the masonry panel calculated from the maximum elevation point with the downward direction.
-*Thickness*: Defines the wall thickness.



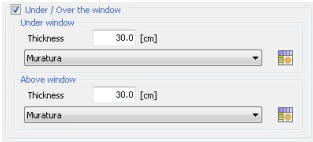
-*Static verifications*: the corresponding frame contains the eccentricity data and the wind exposure condition. The eccentricity indicates the offset of the masonry panel to the wall (inserted in the walls environment).



The mechanical properties of the masonry material.

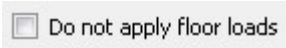


The mechanical properties of the reinforcement in reinforced masonry/FRP reinforcements.



Various thickness and material characteristics of the sub and above window.

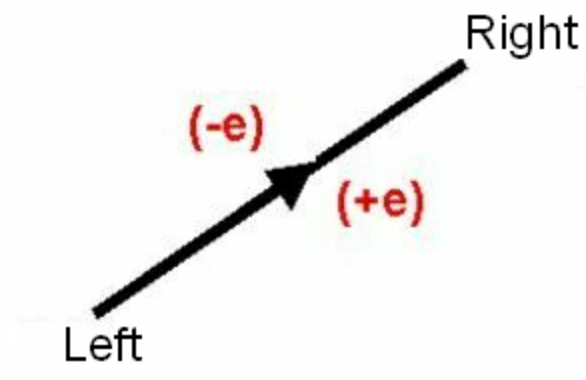
Two adjacent wall panels separated by a node element with the same geometric/mechanical characteristics are normally joined during the mesh generation.
With this option during the mesh generation the two masonry walls will remain separated.

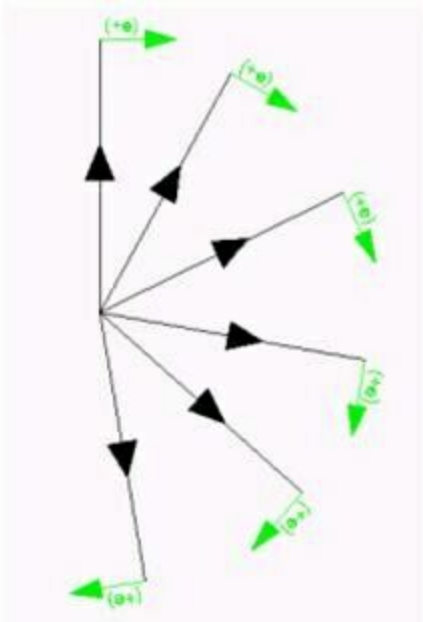


If you want a wall to have only seismic functions but not floor masses transfer you can activate this box.

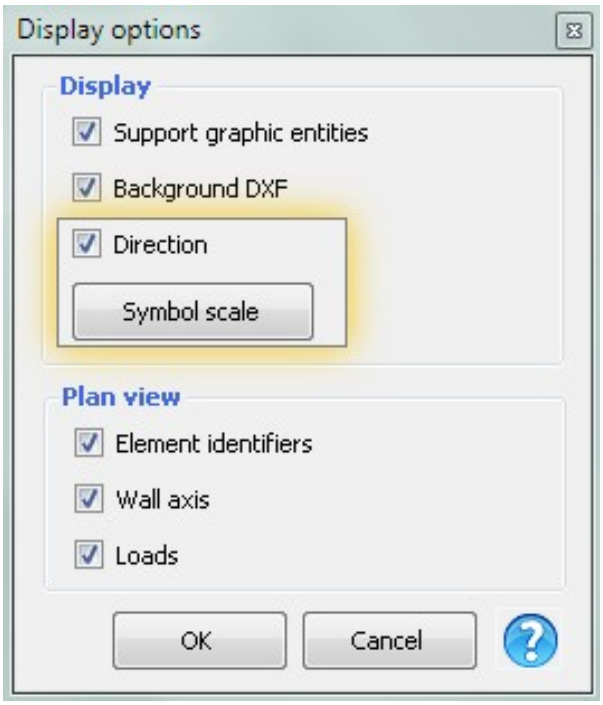
Defines the opportunity to decide that the node at the base of the panel is part of the foundation and the ability to insert the foundation's characteristics.

The eccentricity of structural objects must be inserted with the sign in the following way: Following the wall from the left-most vertical wall endpoint, going towards the right, the positive eccentricity is on the right of the wall. (see figure below)
If you do not intend to use the static checks module (chapter 18 of this manual), these parameters are not necessary.

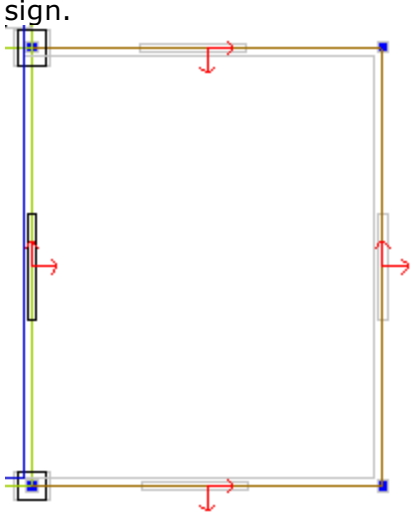




With the display options button you can choose to show the local reference system directly on the model plan.



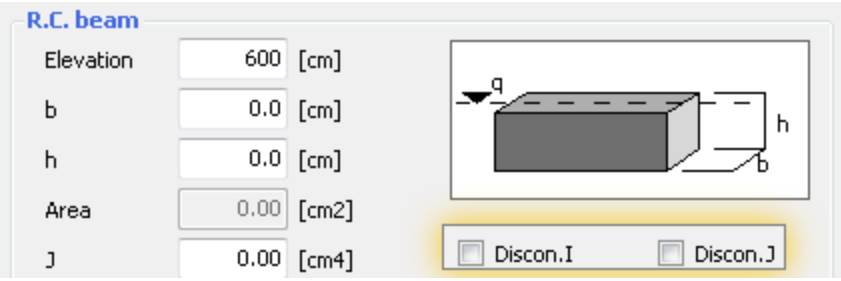
Using the options command, the local reference system is shown on each wall, allowing individuation of the eccentricity sign.



9.2.1.2 R.C. beam

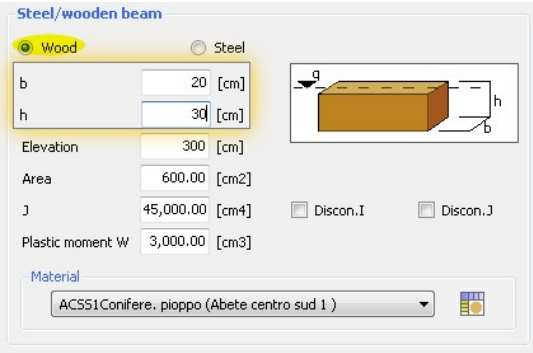
-*I Elevation / J Elevation*: Individuates the elevation of the two beam ends. This allows insertion of inclined beams. (insertion of two identical elevations creates a horizontal beam). In this version of the program, only horizontal beams can be inserted (I Elevation = J Elevation).
-*Geometric characteristics of the section*: base, height, area, inertia.
-*Reinforcement*: Area of the longitudinal reinforcements and number of rebars, distinguished based on their position (higher or lower in the section), as well as steps from the stirrup spacing, area and concrete cover. The reinforced areas to be inserted are the totals, and not individual rebars.
-*Seismic details*: Identifies the use of construction techniques that guarantee good performance of structural elements in terms of seismic events (e.g.: the choice of good distribution of longitudinal rebars and stirrups).
-*Discon. I, J*: This allows disconnections (internal hinges) to be inserted at the ends of the beam.

This function allows the designer to define constraints for leaning, by inserting internal hinges, also in the non-linear field.
Insertion of disconnections is managed using the associated tick boxes.
I and J indicate, respectively, the first and second wall segment ends, with respect to the sign convention dictated by the local reference.
The end where the disconnection will be inserted is decided by ticking the appropriate box.

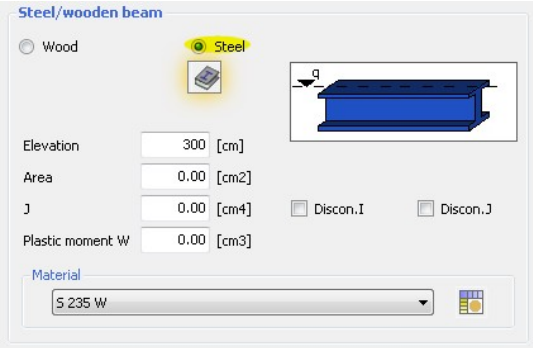


9.2.1.3 Steel/wooden beam

-*Elevation*: Identifies the elevation of the two ends of the beam, in order to allow the insertion of the inclined beams. In this version of the program, only horizontal beams can be inserted (I Elevation = J Elevation)
-*Geometric characteristics of the section*: area, inertia, and plastic resistance module.
-Discon*. I, J*: This allows disconnections (internal hinges) to be inserted at the ends of the beam.



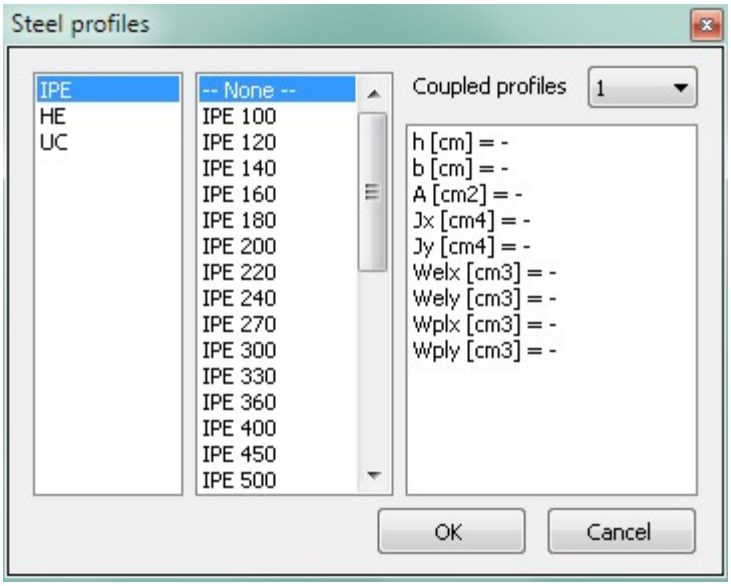
By selecting [**Wood**] is immediately available the space to insert the base and height of the section, the calculation of A, J, and W is automatic.
For non-rectangular sections you can directly enter the values of A, J and W.



By selecting [**Steel**] is immediately available the button that brings up the library profiles.

With this button you can invoke the library of the metal profiles from which you want to retrieve the characteristics.
By selecting the profile's family and size are presented the mechanical characteristics that will be used in the calculation.

 With this button you can display the steel profiles library from where you can get the features.



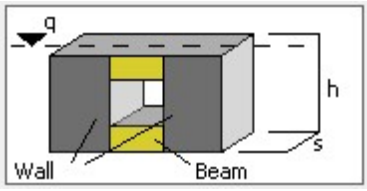
It can also generate a composed section placing up to 4 profiles side by side.
The "Coupled profiles" option is not available for the columns.
It is also possible to generate a section composed of up to 4 profiles disposed side by side.
The option "profiles disposed side by side" is not available for the pillars.

9.2.1.4 R.C. wall

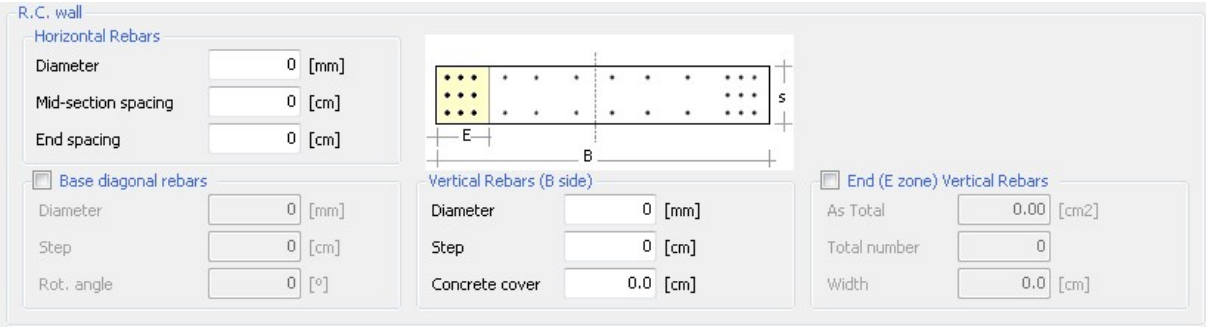
The first step for R.C. walls insertion is definition of the general data:
- Elevation: the maximum elevation of the R.C. wall
- Height: height of the R.C. wall, calculated from the point of maximum elevation to the ground.
- Thickness: thickness of the R.C. wall.

There are options:
1. to define whether the anchorage state of the reinforcement is satisfactory or not,
2. if there are anti-seismic details
3. for the types of bars (smooth/improved adherence).

R.C. walls are inserted defining the wall and the connection beam (see figure below):

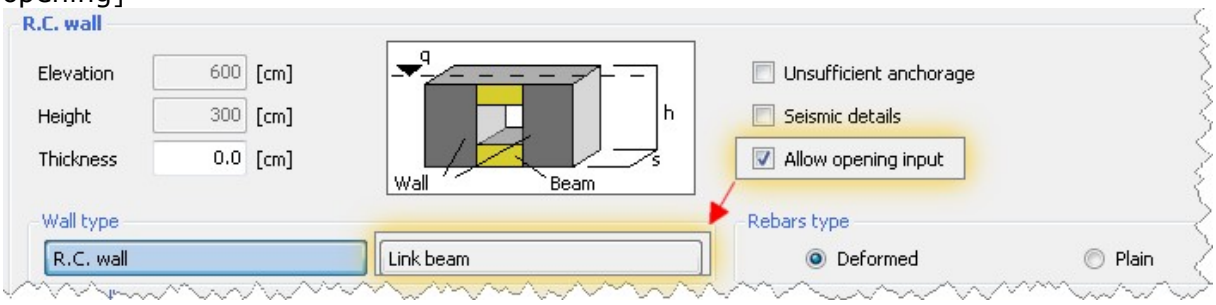


Wall:
-Diameters, steps and concrete cover for the vertical and horizontal rebars.
-Possibility to define different vertical reinforcements in the extreme areas (E zone)
-Diameter and steps of the diagonal based rebars

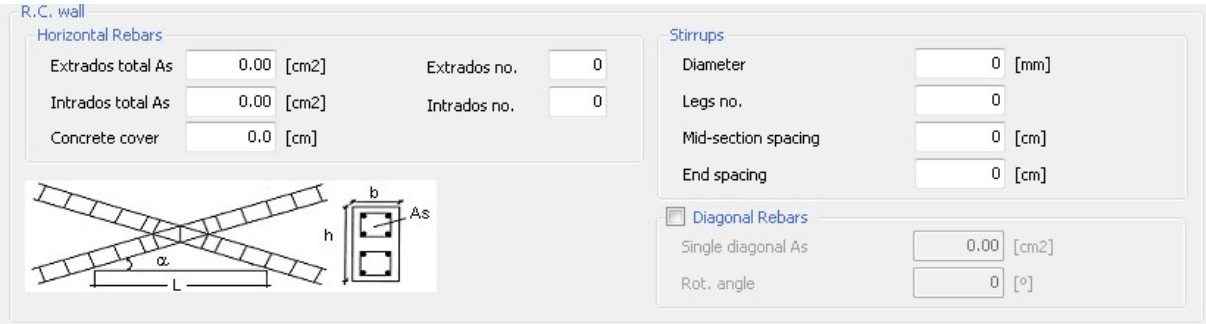


Connection beam:

The insertion of the connection beams is allowed only if you select [Allow insertion opening]



-Diameters, steps and concrete cover for the vertical and horizontal rebars.
-Possibility to define different vertical reinforcements in the extreme areas (E zone)
-Diameter and steps of the diagonal based rebars
-Reinforcement: Area of the longitudinal reinforcements and number of rebars, distinguished after their position (top or bottom of the section), as well as the stirrup spacing steps, area and concrete cover. *The reinforcement areas to be inserted are those of the total and not individual rebars. It is also possible to use diagonal rebars*.

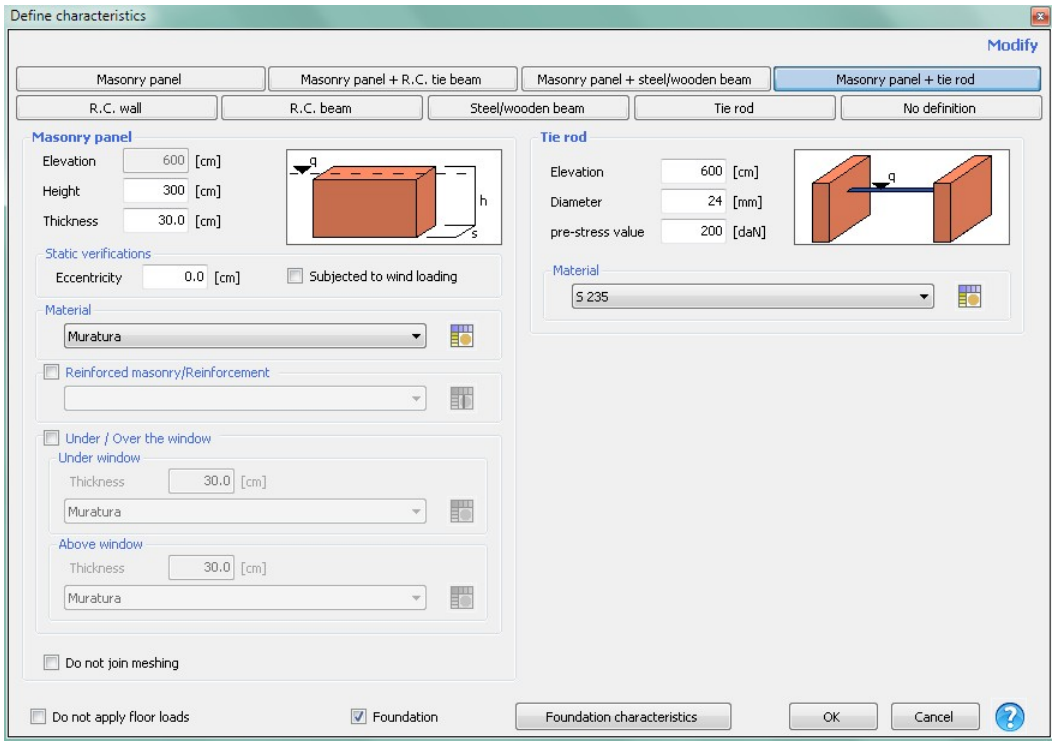


9.2.1.5 Tie rod

The insertion of a tie rod not connected with a masonry wall, is functional only if inserted on a part of a single wall (divided through the insertion of segment points) able to include other structural elements of the same alignment which are able to absorb the actions provided by the tie rod.
-*Elevation*: elevation in which the tie rod is placed
-*Diameter*: diameter of the iron which constitutes the tie rod
-*Tension*: stretching of the tie rod

9.2.2 Composite Elements

If the structural element is composed and not simple, the definition window is divided into two parts to allow the insertion of the mechanical characteristics of both structural objects.

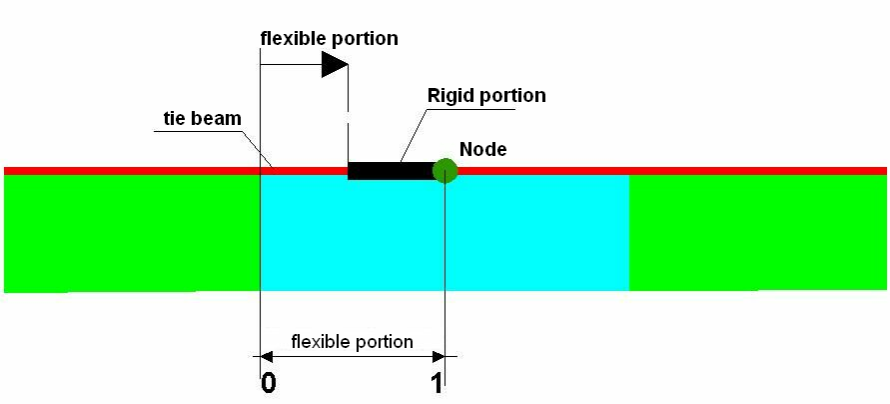


Once chosen the structure type you can edit the geometric characteristics of the element and access the materials library.

In the bottom left of the dialog, there is the possibility to decide not to load the desired item with the slab above (eg. The floor is not resting directly on the element).

9.2.2.1 Masonry panel + Tie beam

Pairing of a masonry panel with an R.C. beam linked to the same wall (the panel and the beam are part of the same vertical plane. The definition of the panel and the beam is the same used for the elements taken individually.
The flexible portion of the tie beam is inserted as a number between 0 and 1. This multiplies the distance between the node in question and the edge of the continuous spandrel beam and represents the length of the flexible part of the tie beam. This extends to the inside of the rigid node, starting from the edge of the spandrel beam.



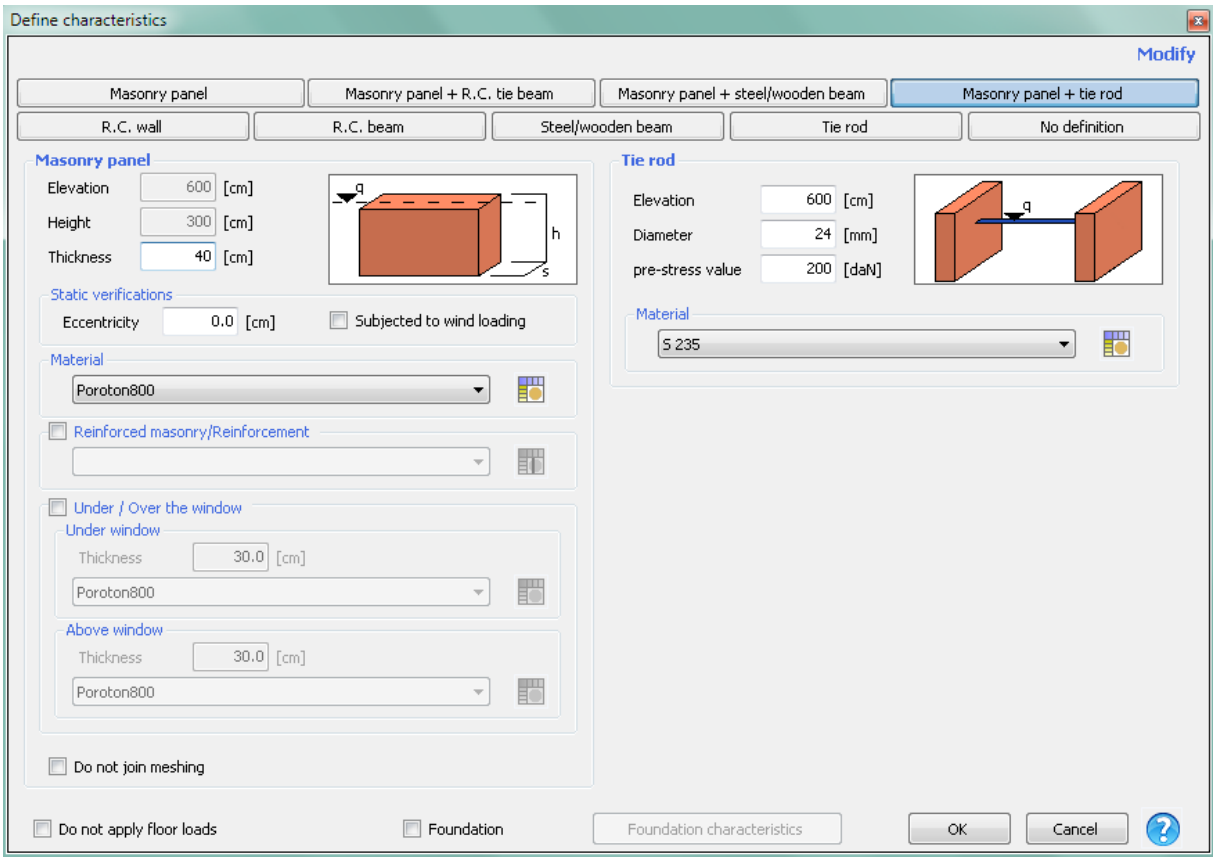
9.2.2.2 Masonry Panel + Beam

This is a masonry panel paired with a steel or wood beam. The parameters that must be inserted are the same as those for the elements taken individually.

9.2.2.3 Masonry Panel + Tie Rod

This is a masonry panel paired with a tie rod. The parameters that must be inserted are the same as those for the elements taken individually.
The combined elements are very useful for strengthening masonry panels with elements such as tie beams, steel or wood beams, or tie rods.

Whenever the use of a combined structural element is required, the definition window is divided into two parts. In this way, the mechanical characteristics of both structural objects can be inserted.

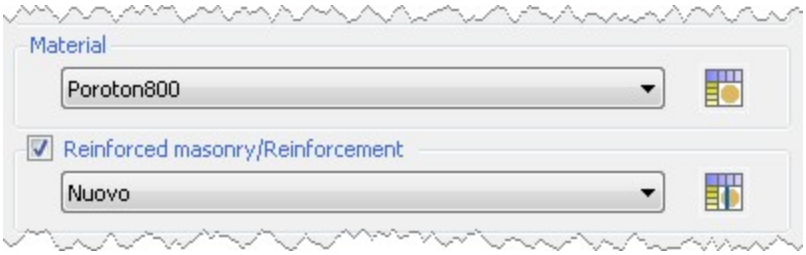


Once the structural typology is chosen, the geometric characteristics of the elements can be edited and the materials catalog can be accessed.

In the lower left corner of the window, it is possible to choose if the element will receive the load of the floor above it. (e.g. the floor does not rest directly upon the element.)

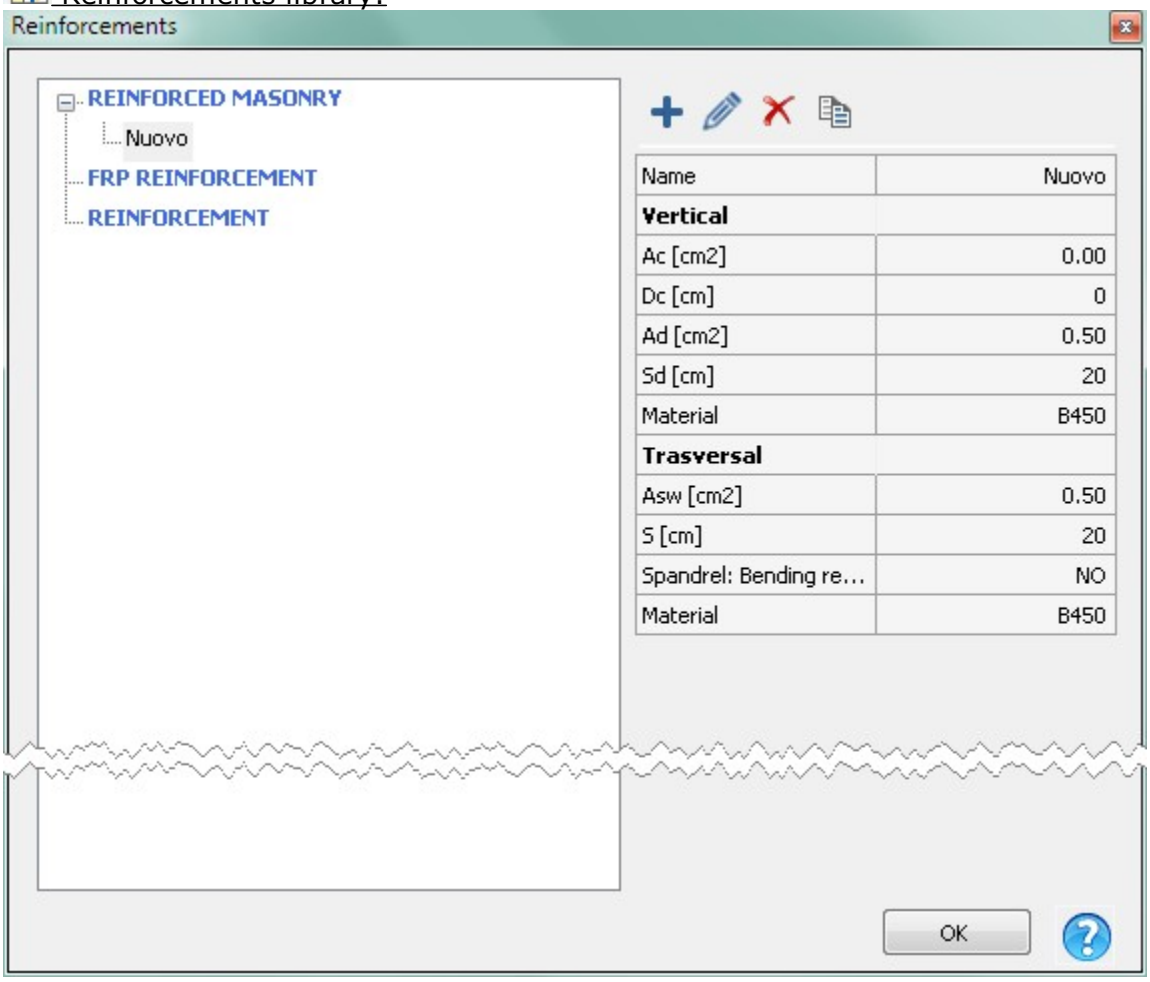
9.2.3 Reinforcements

All structural items, such as masonry panel and with it compounds, contain in their definition a reinforcement.



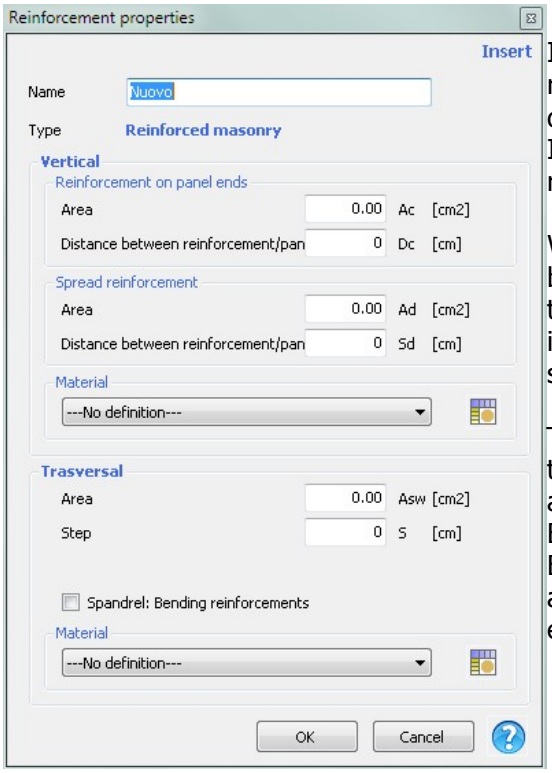
Checking the properly box, the reinforcements library form is ready to use.

Reinforcements library:



Main reinforcements types:
- Reinforced masonry
- FRP fabrics
- Reinforcement

New reinforcement:
Allow to define properties of new reinforcement type.



In defining the characteristics of the reinforcement you can decide the its distribution by area and spacing.
It is also possible to define a concentred reinforcement.

When you assign a reinforcement of a particular building panel, the requirements defined in the type of reinforcement will be allocated to individual masonry macroelements (pier and spandrel)

The vertical reinforcement will be assigned only to pier; trasversal reinforcementwill be assigned to spandrel too if the box "Spandrel: Bending reinforcements" will be checked.
Every concentred reinforcement is automatically definied as simmetrical as to element ends.

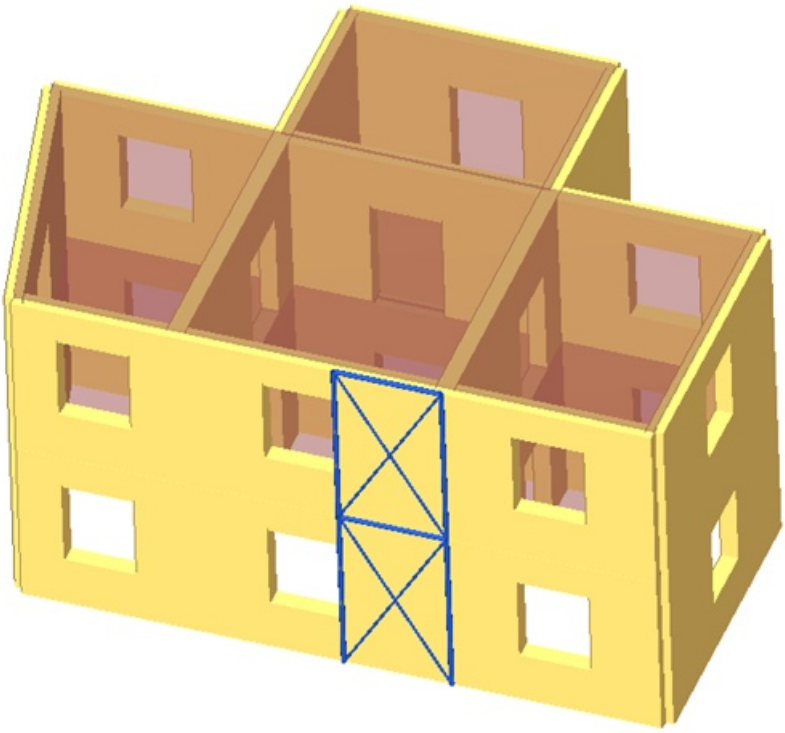
Other functions:

	Edit reinforcement
	Delete reinforcement
	Duplicate reinforcement

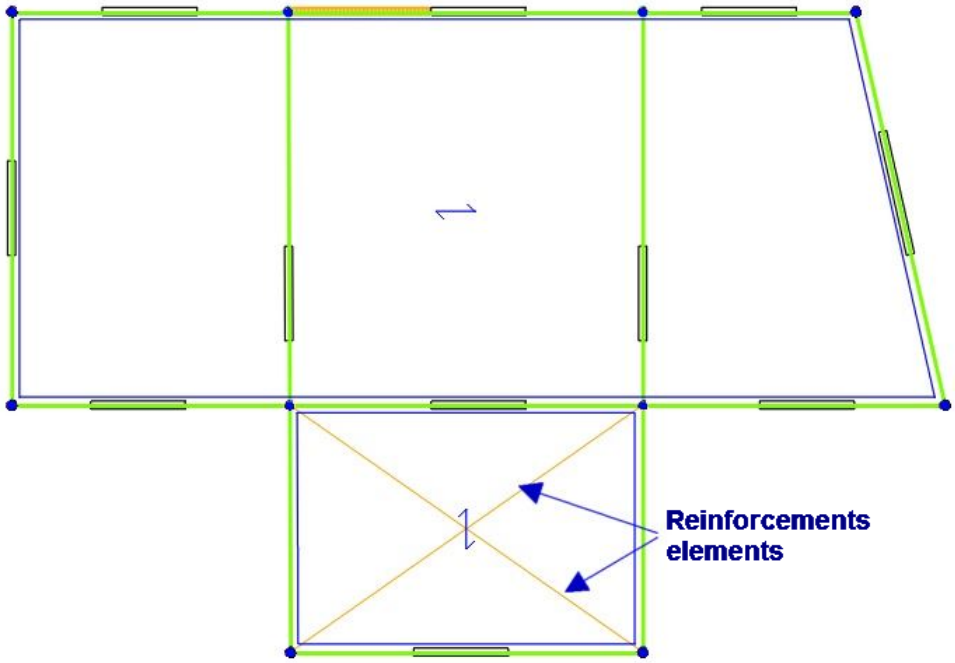
ATTENTION!!!
It is NOT possible to insert a reinforcement only on the spandrels.

9.2.4 Rinforzi con telai

You can reinforce a structure with framed elements by placing them in the walls and in the slabs:



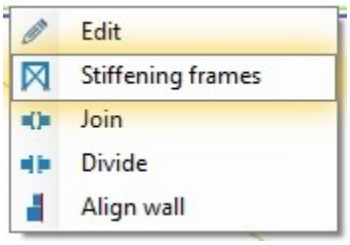
Frame reinforcements in the walls



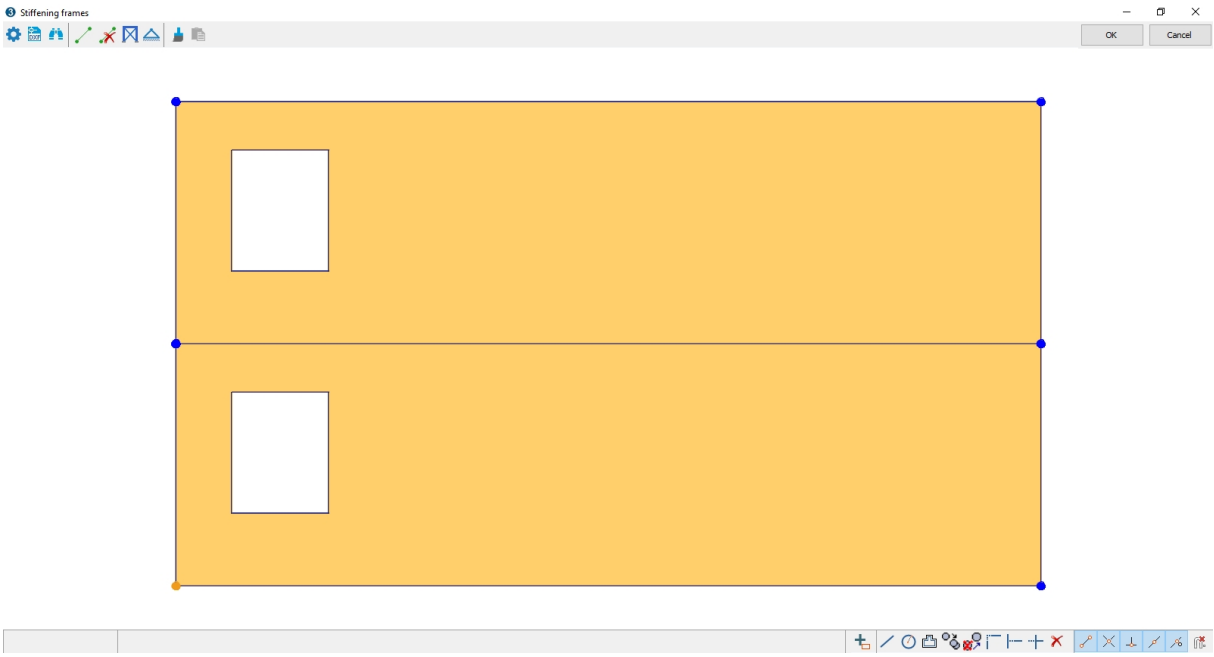
Frame reinforcements in slabs.

9.2.4.1 Frame reinforcements in walls

Select a wall with the right mouse button in the structure environment than the context menu highlights the *"Reinforcement Frames".*
Selecting this item leads to the reinforcements insertion environment for the examined wall.

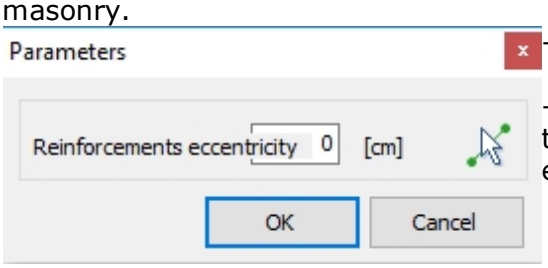


The environment shows that the wall in question opens with the command bar dedicated to the reinforcements.



At the bottom right, the bar dedicated to the graphic input already used in other areas of the program, for the detail of the individual commands, see the general session (Support graphic; Snap) .

At the top left, the bar dedicated to the insertion of the reinforcements.

	3D visualization properties that allows to set the offset of strengthening systems to make them visible in 3D. For the calculation, the reinforcements are positioned in the mid-plane of the wall, to reporting purposes in the 3D view will therefore be hidden the encumbrance of the masonry. An exclusively graphic offset will make visible such items outside of the masonry.  -Enter the display eccentricity -Press the Select button to identify the elements to which to apply this eccentricity.
	DXF Import Support
	Find reinforcing element based on its number
	Insert reinforcement Enter value
	Delete reinforcement
	Reinforcement characteristics definition Enter value
	Insert additional constraints Enter value
	Copy properties
	Paste properties

9.2.4.1.1 Insert/Delete reinforcement

Are two commands dedicated to defining routes that identify the location of the reinforcements.
The insert command, allows the placement of the reinforcement by graphical input to indicate the start and end point of the reinforcement.
The delete command, requires the selection of the reinforcing segment to be deleted.

The use of these commands is made easier by the snap to graphics and from DXF input.
The well defined elements are free of any geometric / mechanical characteristic to be defined by the appropriate command described in the following.

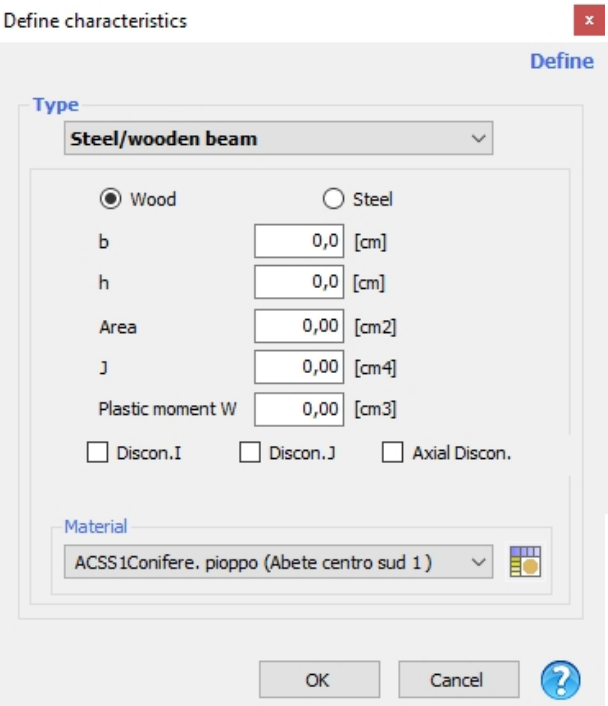
9.2.4.1.2 Reinforcement characteristics definition

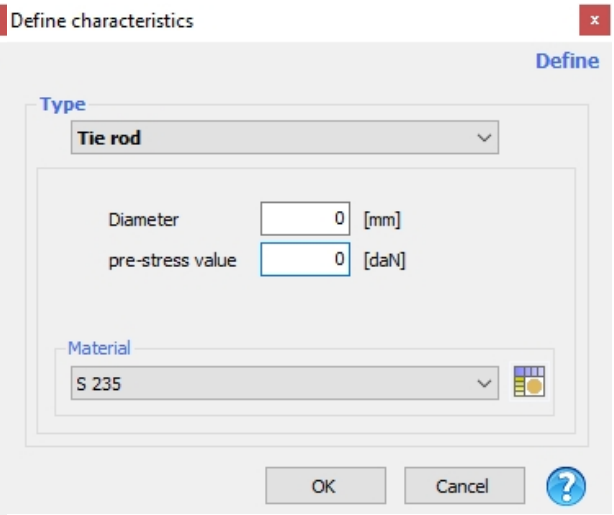
Allows to define the geometric / mechanical properties of the reinforcing elements drawn with the insert command  .

By calling this command, you need to perform a selection of reinforcements to which you want to assign the properties; once you completed the selection will appear the characteristics definition window.

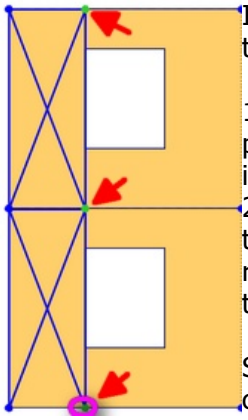
By selecting **"Steel / Wood Beam "** from the drop down menu is possible to define the characteristics of a beam in wood or steel.

By selecting **"Tie rod"** from the dropdown menu is possible to define the characteristics of an element only resistant to traction.





9.2.4.1.3 Additional constraints



In the reinforcements prospect view we can recognize two different types of nodes:

1. **Blue** : structural nodes that exist independently from the presence of the reinforcements (for example produced by the intersection with the plug walls).
2. **Green** : product nodes by the inclusion of reinforcements, are then created with the sole purpose of connecting the elements of the reinforcing frame and attach them to the structure (the figure shows the nodes created for reinforcements).

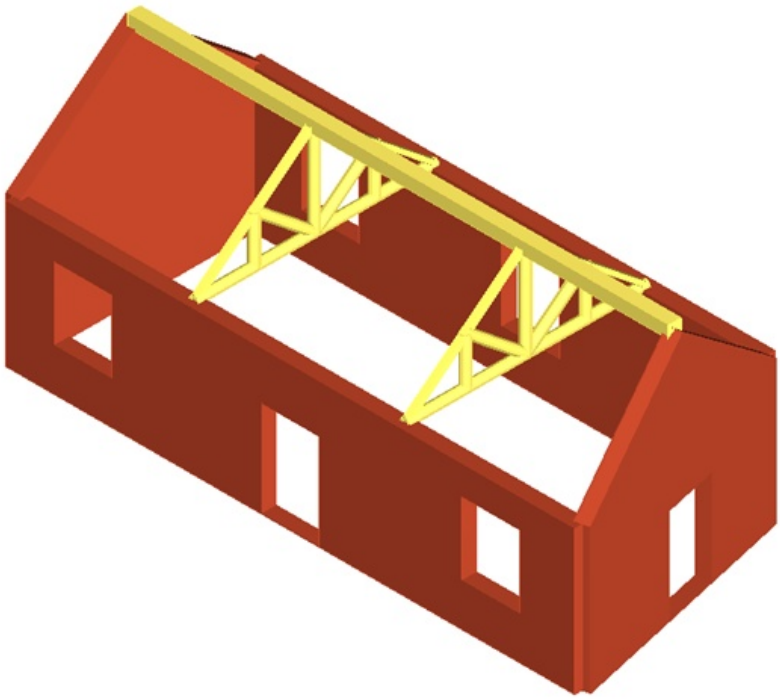
Some of these nodes may be positioned in correspondence of a constraint of the foundation.

This constraint can not be automatically defined, but must be manually entered by the user using the appropriate command. A blocked node will appear in **Orange**.

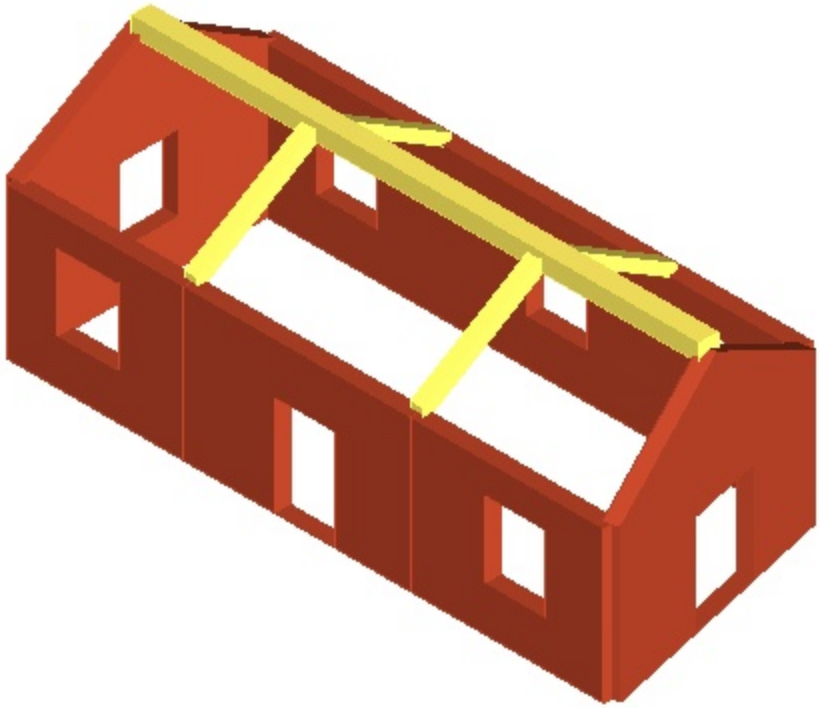
Deleting a constraint is done by clicking on the node with the right mouse button and selecting "*Delete*"

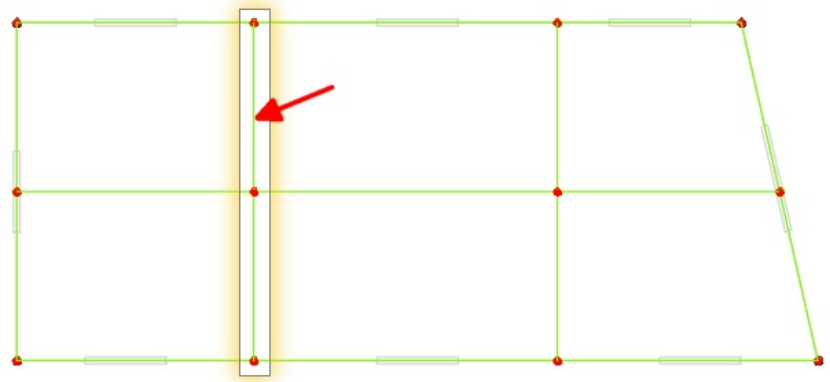
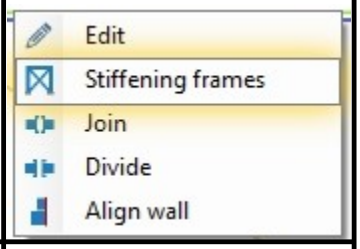
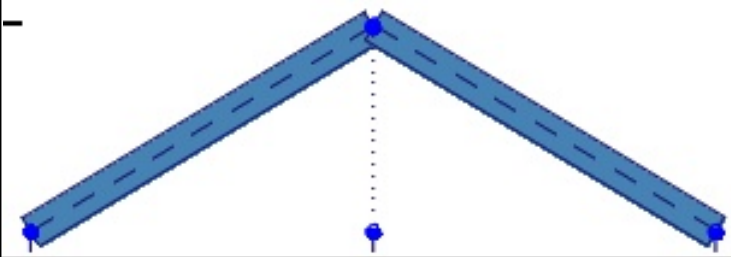
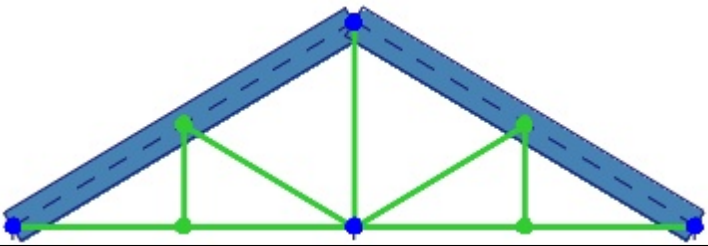
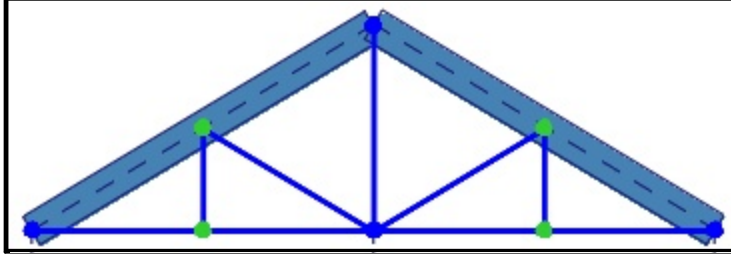
9.2.4.1.4 Reinforcement through truss beam

Below, an example of an application that allows to use the reinforcements insertion to define a truss beam as shown in the following figure.



Here below the steps leading to the creation of the truss beam:

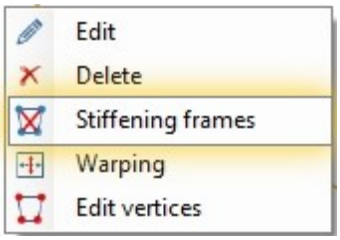
	In the roof environment previously define the ridge beams and the elements that constitute the upper stringers of the truss beam (by tracing of the roof profiles). **CAUTION!!** : The tracking of these elements must occur in advance with respect to the definition of the reinforcement.

	Select the roof profile corresponding to the current top of the grid and enter the definition area of the reinforcements. 
	There are displayed in the appropriate environment, the upper current of the truss.
	 Use the appropriate command to draw the additional elements that define additional members of the reticular.
	 Assign the geometrical/ mechanical characteristics to the recently drawn elements.

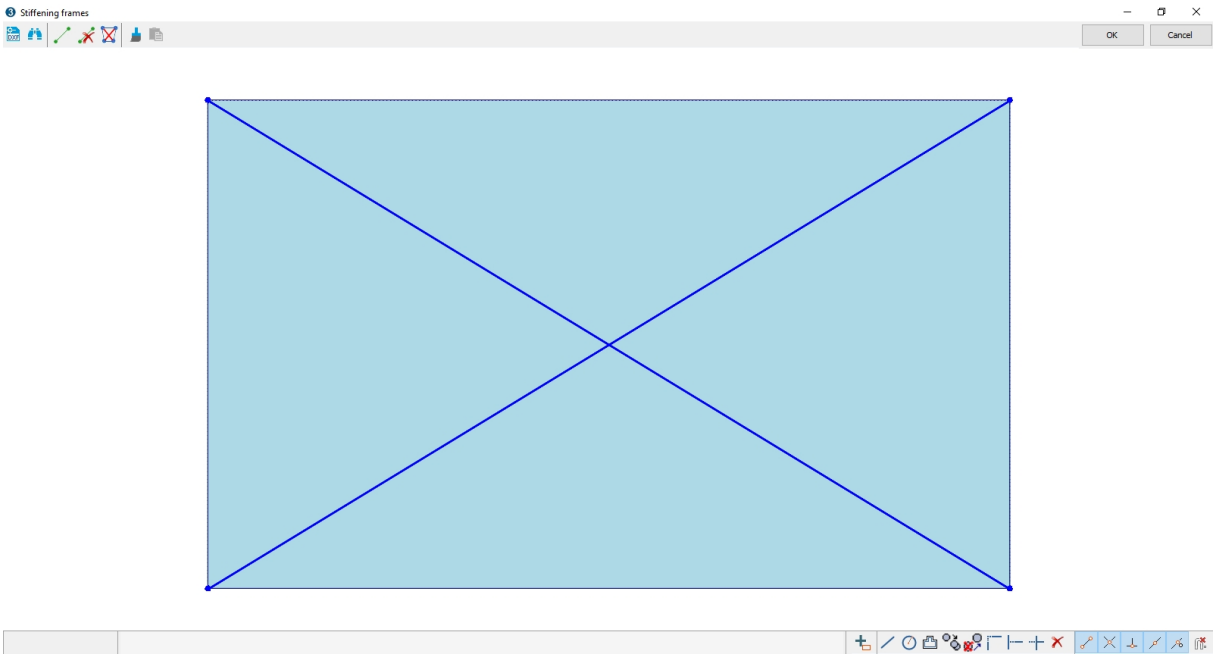
Coming out of this environment and recalling the 3D view you see the desired result shown in the first image.

9.2.4.2 Reinforcements with frames of horizontal elements

Into the Structure environment, by selecting with the right button of the mouse a slab, the context menu shows the item "Reinforcement Frames".
By selecting this item you access to the insertion environment of reinforcements for the slab under consideration.



The environment that opens up displays the slab in question with the command bar dedicated to the reinforcement.



At the bottom right, the toolbar dedicated to the input graph already used in other environments of the program, for the detail of each commands please refer to the general session (Supporting graphics; Snap).

At the top left, the dedicated bar for the reinforcement insertion.

	Import support DXF
	Find reinforcing element based on its number
	Reinforcement insertion
	Delete reinforcement
	Reinforcement characteristics definition
	Copy properties
	Paste Properties

9.2.4.2.1 Inserimento/Elimina rinforzo

Are two commands dedicated to defining routes that identify the location of the reinforcements.
The insert command, allows the placement of the reinforcement by graphical input to indicate the start and end point of the reinforcement.
The delete command, requires the selection of the reinforcing segment to be deleted.

The use of these commands is made easier by the snap to graphics and from DXF input.
The well defined elements are free of any geometric / mechanical characteristic to be defined by the appropriate command described in the following.

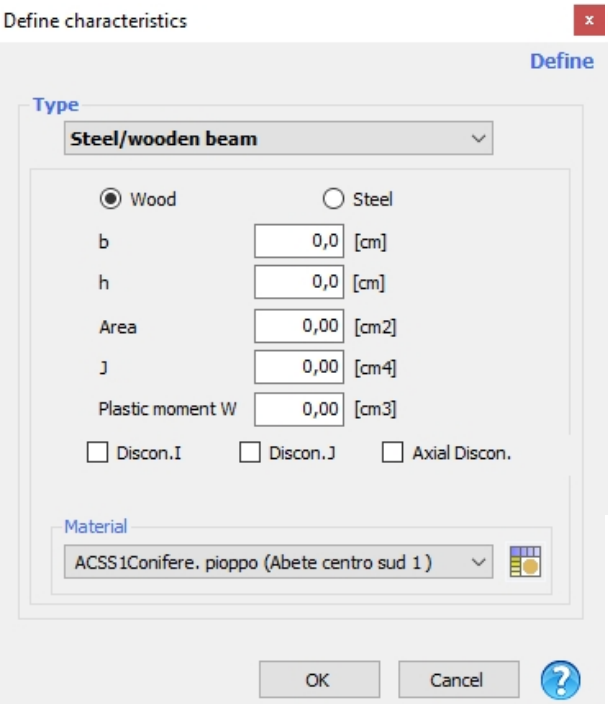
9.2.4.2.2 Definizione caratteristiche rinforzo

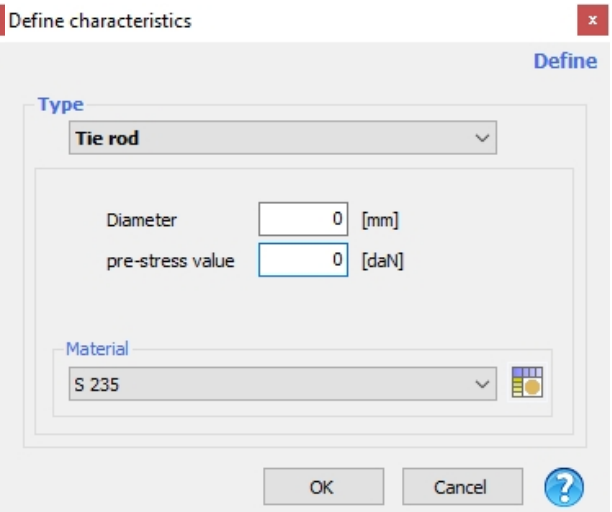
Allows to define the geometric / mechanical properties of the reinforcing elements drawn with the insert command  .

By calling this command, you need to perform a selection of reinforcements to which you want to assign the properties; once you completed the selection will appear the characteristics definition window.

By selecting **"Steel / Wood Beam "** from the drop down menu is possible to define the characteristics of a beam in wood or steel.

By selecting **"Tie rod"** from the dropdown menu is possible to define the characteristics of an element only resistant to traction.





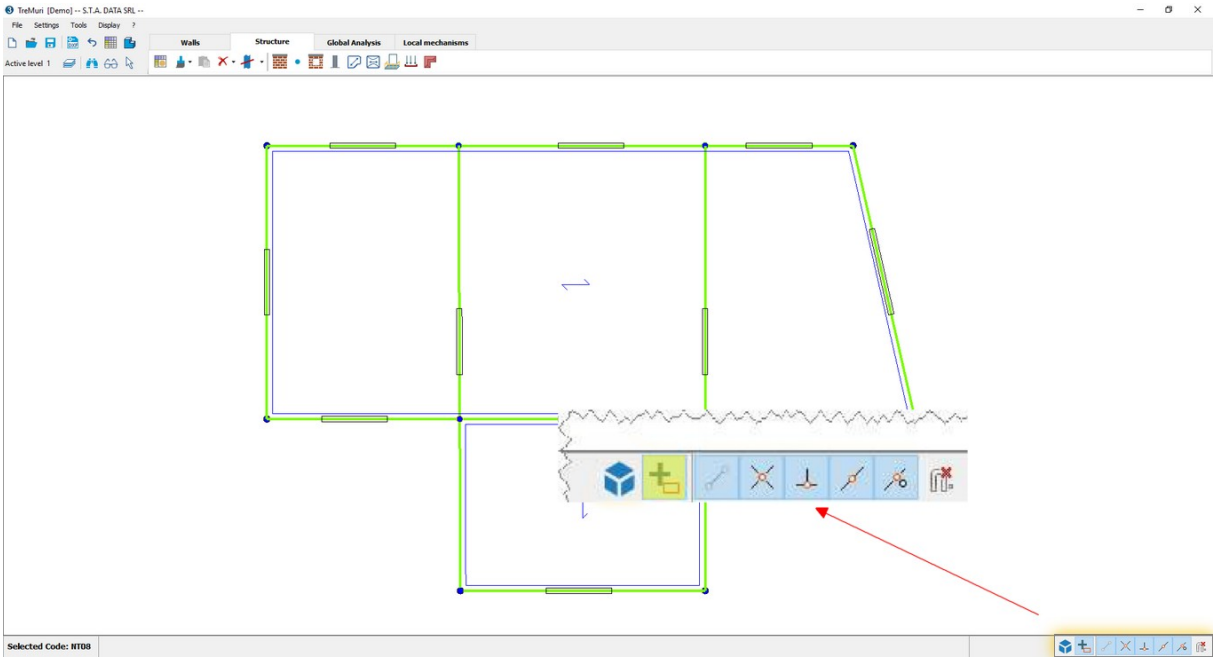
9.2.5 Segment Points

- Segment points can be inserted using the left mouse button.

This function can be used to assign various materials to a single wall, or to insert a

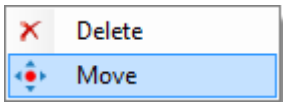
segment point at the intersection of more than one wall. For example, if one wants to define a single wall with different masonry typologies, or with masonry of different thicknesses, it is necessary to define the segment points in the points in which the thickness or the material changes. All Type 2 nodes are segment nodes. Hence they can always be used as wall endpoints to define a floor. Type 3 nodes are not segment nodes. They cannot be used to insert a floor unless a segment node is inserted using the command. (for more information on nodes in the walls/structure environment, see the areas description)

The entry in a precise distance during the input phase can be performed using the command "Dynamic Input" available through the specific button on the command bar.



Through the specific command is activated the input mode with elevation; the Dynamic input button remains pressed and during the object input a double elevation indicates the distance at which it will be placed.
The dimension values are contained within a textfield, by entering a numerical value you can impose a predetermined distance, by pressing the [TAB] key you can move to the next textfield.

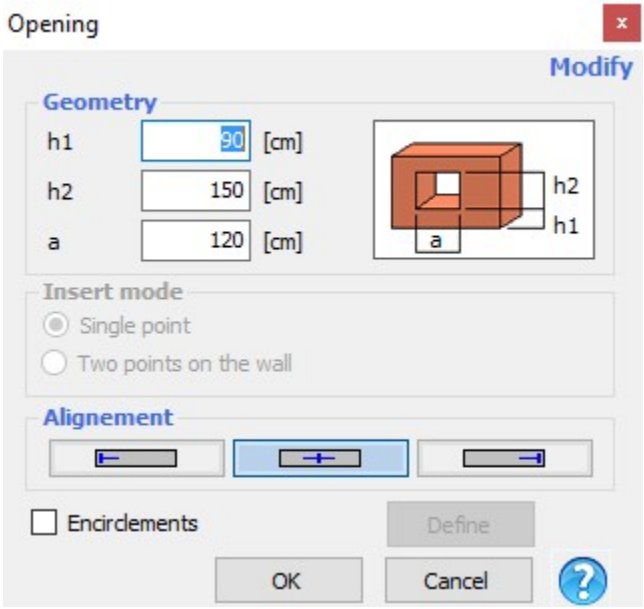
By selecting an already inserted entity with the right mouse button (opening or node) you can move it.



It is necessary to define the displacement vector by clicking in two points, starting point and end point.

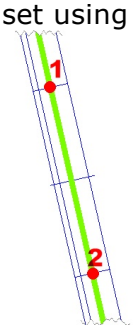
9.2.6 Openings

Allows the insertion of an opening in a wall.
A window through which is possible to modify the geometric characteristics of the opening will appear, once given the OK, it can be proceeded with the insertion of the openings in the desired positions. Exit by clicking the right mouse button. It is possible to select the alignment for the insertion of the opening.
During the opening insertion phase, the window will remain active, allowing the dimensions of the openings to be changed without having to close and restart the insertion command.

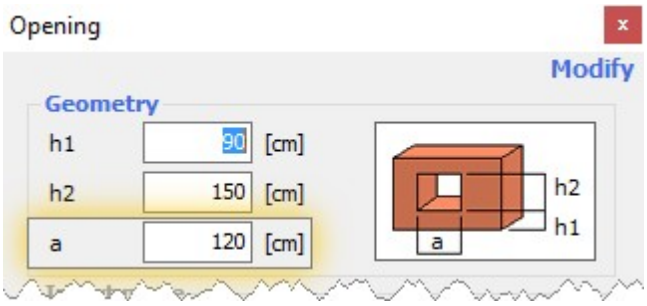


The insertion of an opening can be performed in two different ways:

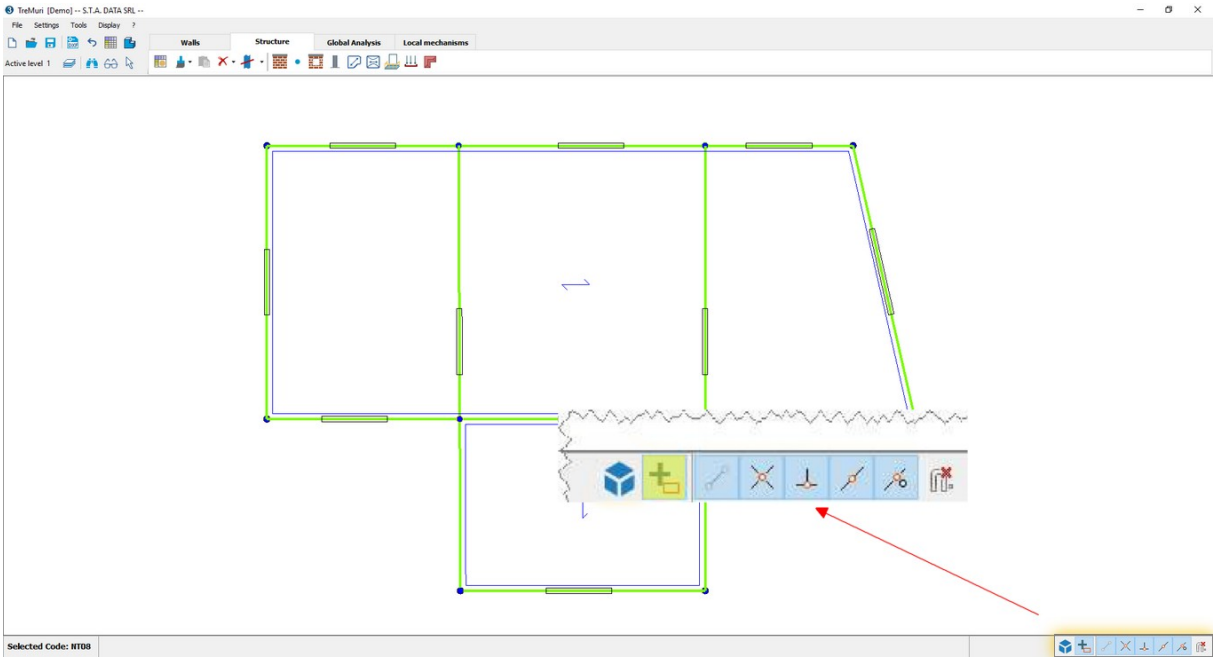
Single point: it is necessary to define the size of the opening and click at the "thread" chosen for inclusion.
Two points on the wall: heights (h1, h2) are necessary but not the width (a) which is set using two clicks in the graphics area (at the beginning and end of the opening).



There are necessary two clicks, one at the beginning [1] and one at the end of the opening [2].
The width is no required because it is calculated automatically from the distance between [1]-[2].

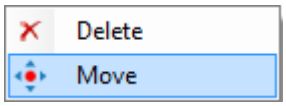


The entry in a precise distance during the input phase can be performed using the command "Dynamic Input" available through the specific button on the command bar.



Through the specific command is activated the input mode with elevation; the Dynamic input button remains pressed and during the object input a double elevation indicates the distance at which it will be placed.
The dimension values are contained within a textfield, by entering a numerical value you can impose a predetermined distance, by pressing the [TAB] key you can move to the next textfield.

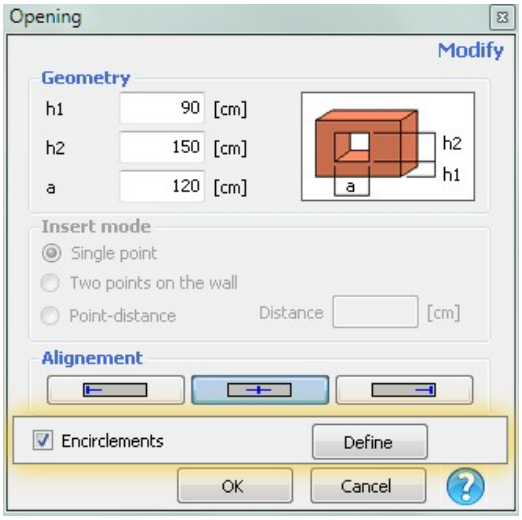
By selecting an already inserted entity with the right mouse button (opening or node) you can move it.



It is necessary to define the displacement vector by clicking in two points, starting point and end point.

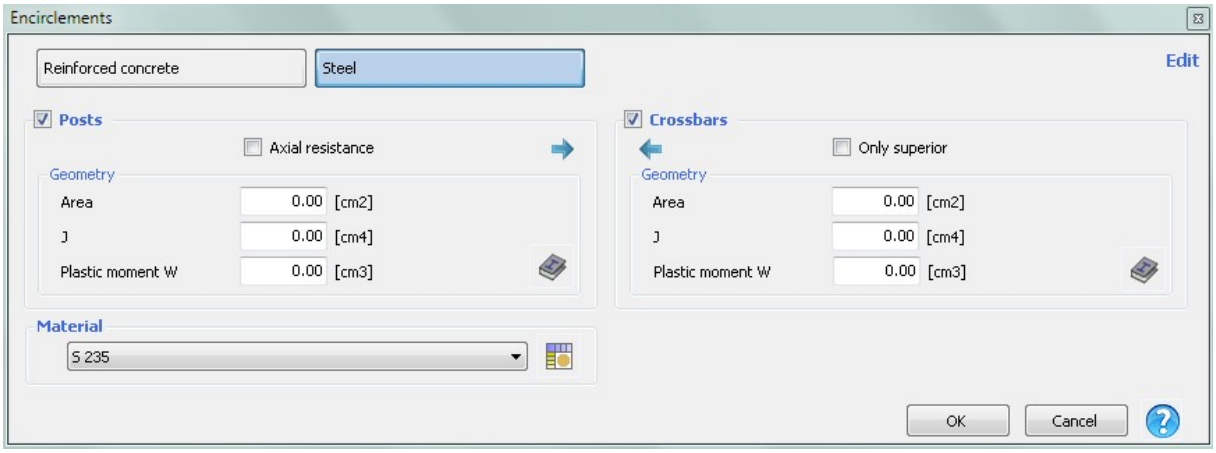
9.2.7 Encirclements

The encirclements are a kind of reinforcement that can be applied directly to the openings of the structure.
To insert an encirclement you need to check the box "encirclements", right from the "Opening" command, to activate the setting button that allows you to insert the features.

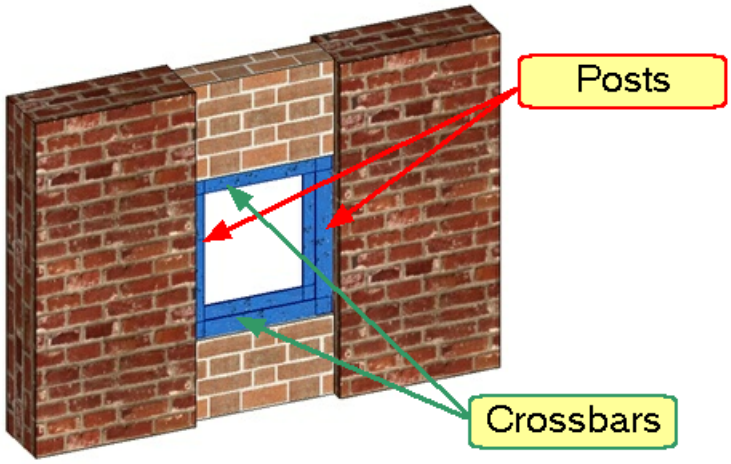


By pressing the definition button, it appears the input window of the geometrical and mechanical characteristics.
From this window you can choose from the two main types of encirclements:

- concrete encirclements
- steel encirclements



For each of the two types, you can set through a checkbox, if you want to insert only the posts, only crossbars, or both.



The "Posts" section, allows you to choose whether to consider the contribution of

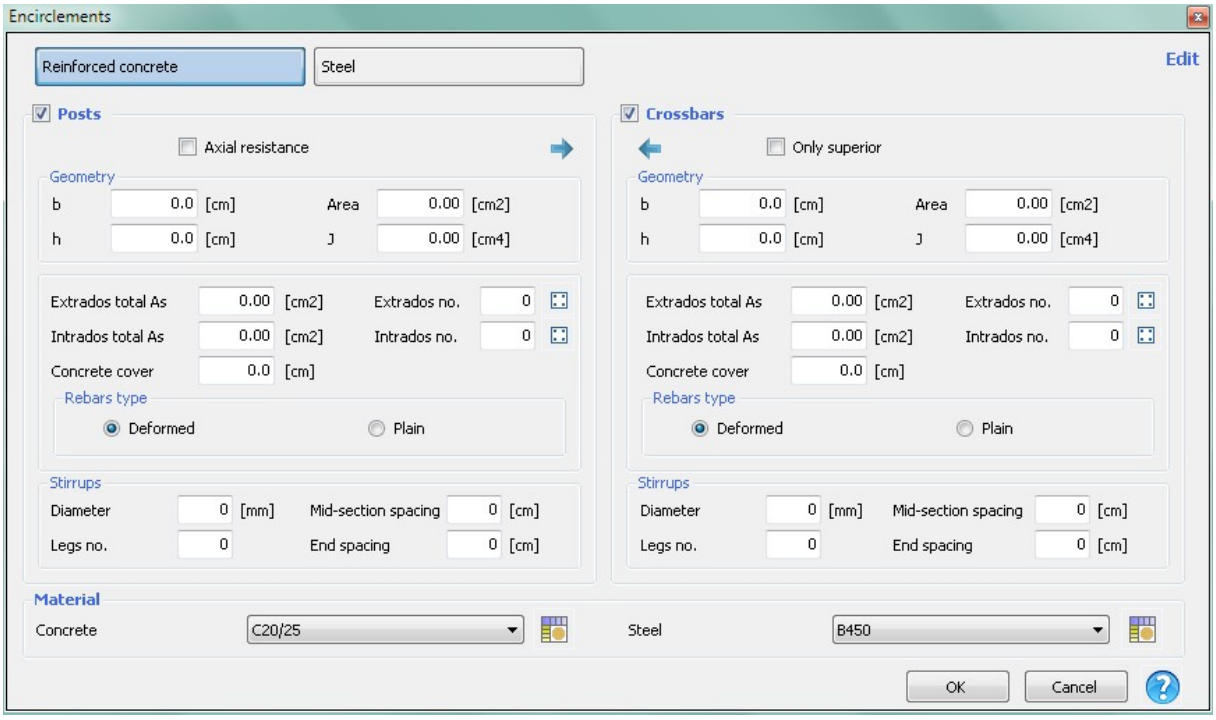
the axial strength of the post or not.
By choosing to account the axial strength of the post, you allow it to absorb a part of the loads from the upper floor along with the masonry wall to which is attached.
It's widespread practice, not to consider this contribution for safety reasons, this why the corresponding checkbox is disabled by default.

The "Crossbars" section, allows the designer to decide whether to consider both or only the upper cross member with the function of lintel.

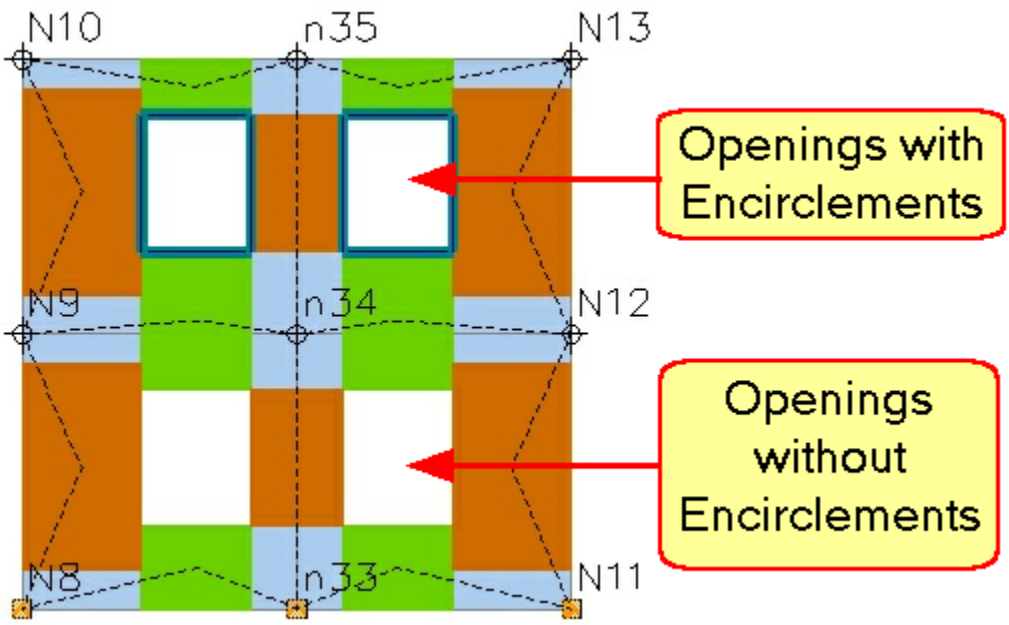
These buttons allow you to copy the features entered for posts to the crossbars and vice versa.
As for the posts and crossbars using the same section, this command helps the input.

All the mechanical properties such as J, W, the iron's surface, are to be considered related to the flexural strength in the plane of the wall where you enter the opening.

The input of encirclements made of reinforced concrete, is much larger because all the parameters of the reinforcements are needed, so that we can consider their behavior in the non-linear field.

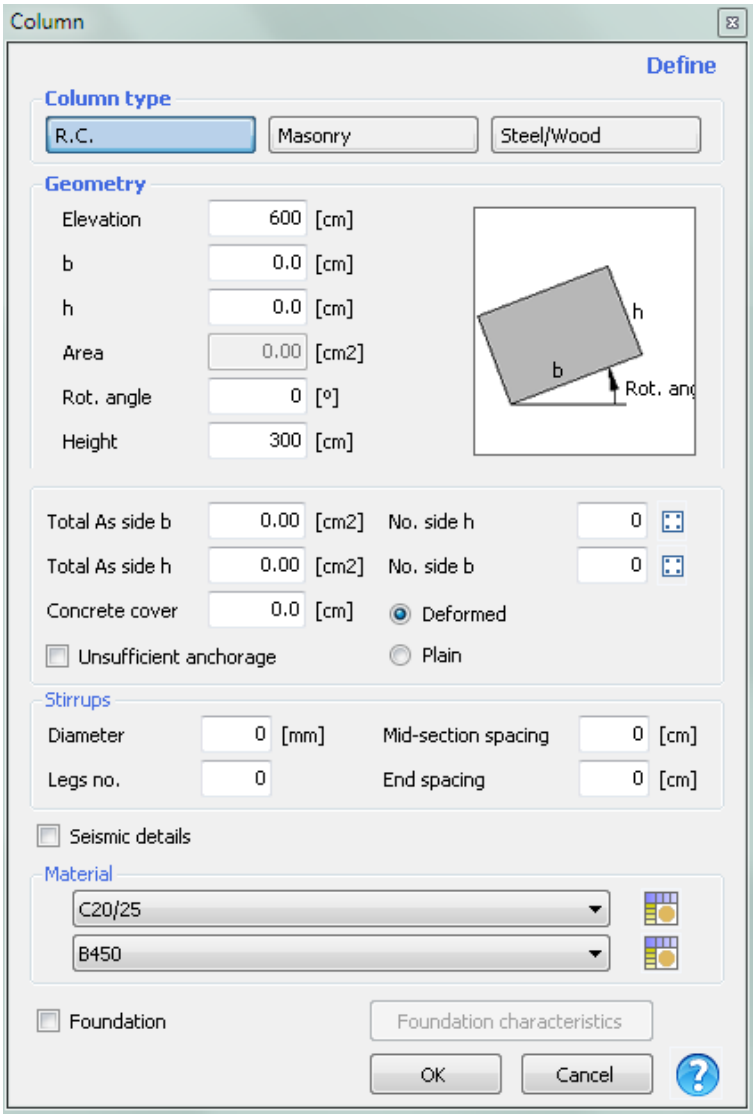


Following the introduction, will be visually presented directly in the statement of the mesh wall.



9.2.8 Columns

Insert a column in correspondence with one or more nodes.
First the node or nodes where the columns will be placed are selected. Then, using the right mouse button, access the window in which the geometric characteristics and the materials of the element are defined.
There are three different types of columns that can be inserted: R.C., masonry or steel/wood. Based on the column typology chosen, the mechanical characteristics necessary to perform the non-linear computation will be requested. For R.C. columns, the reinforced areas that must be inserted are the totality along the side and not those of the individual irons.



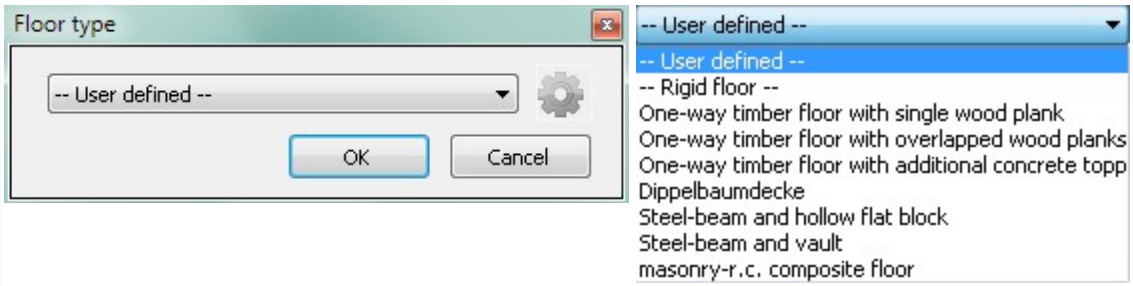
At the lower left, it is possible to activate a box that imposes foundation constraints at the base of the column.

9.2.9 Floor

allows to insert a slab.

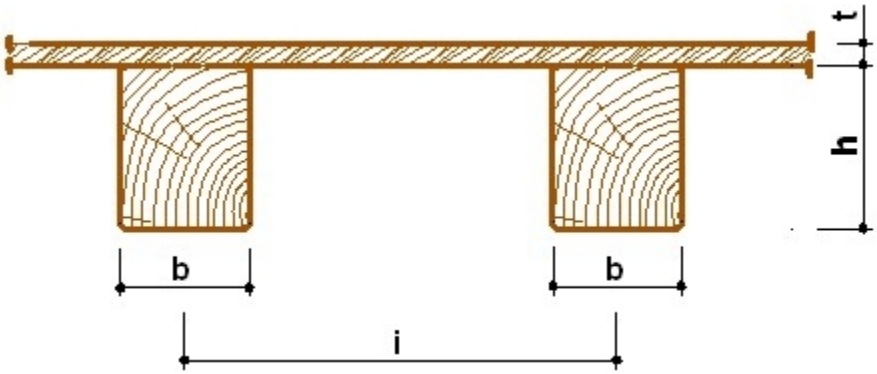
Professional version, Small Business, Small structures

A dialog in which the user can select the type of slab to insert is displayed.

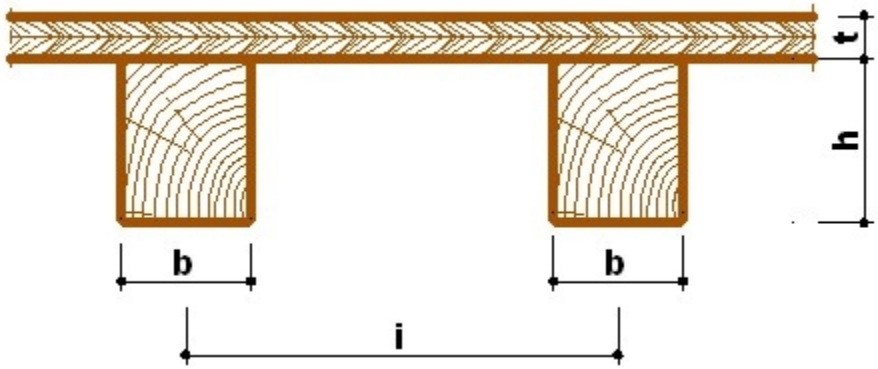


The horizontal elements window allows to set the mechanical characteristics of various types of slabs among the most common; the program examines the following:

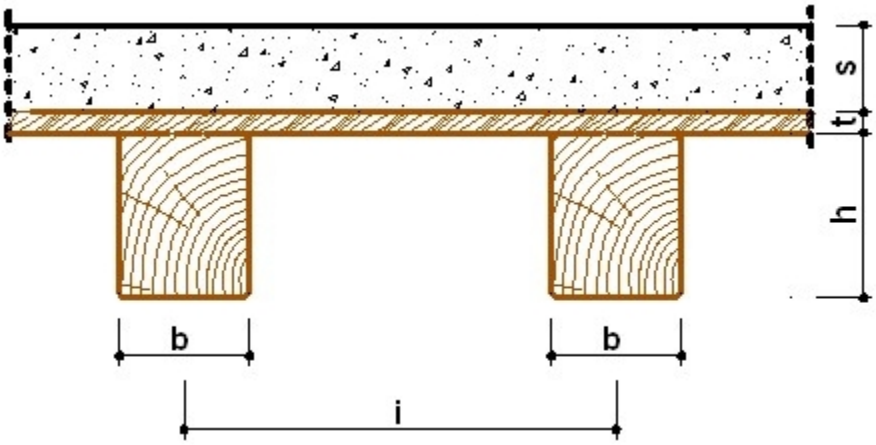
One-way timber floor with single wood plank



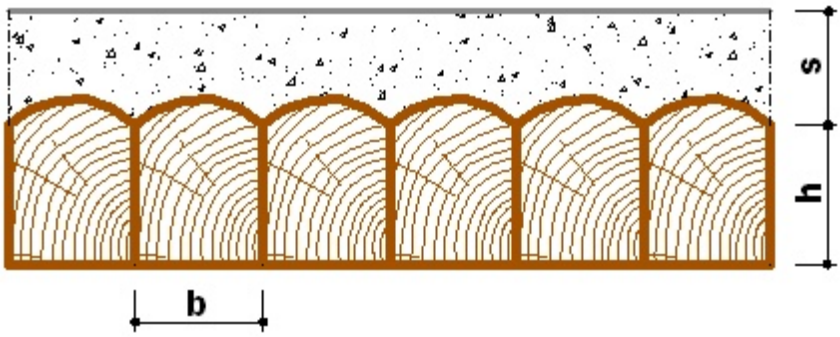
One-way timber floor with overlapped wood planks



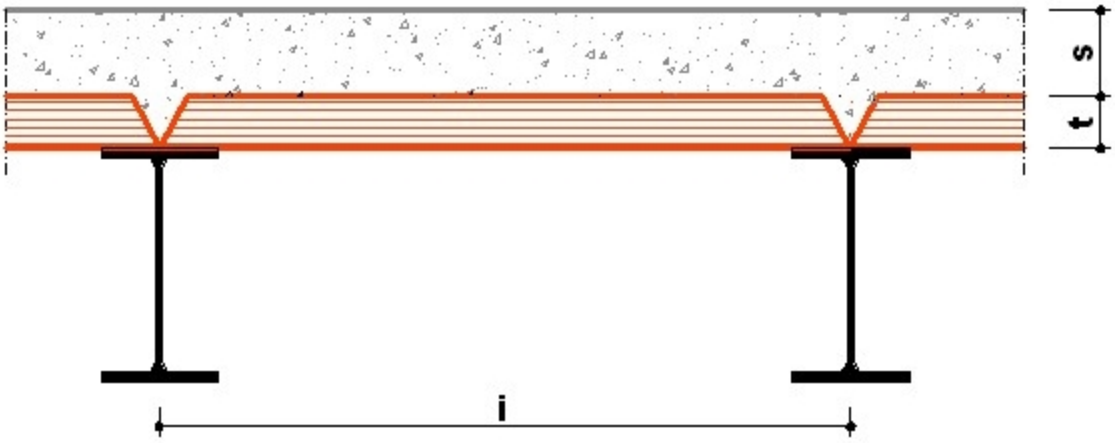
One-way timber floor with additional concrete topping



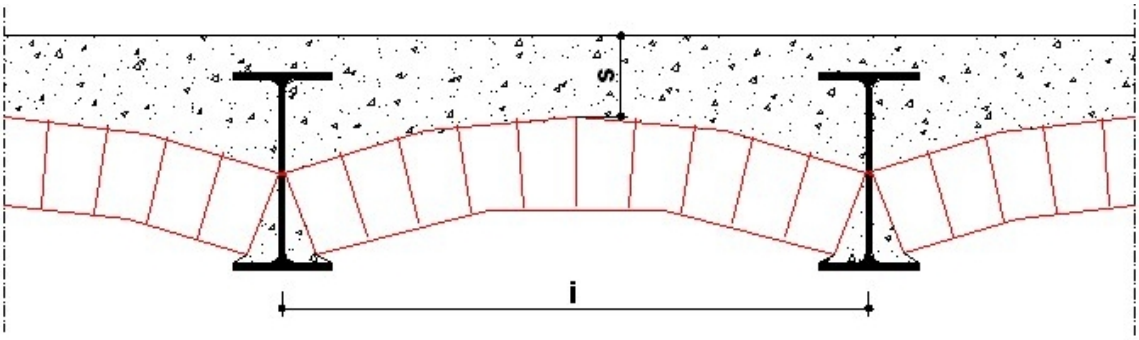
Dippelbaumdecke
Legno con soletta



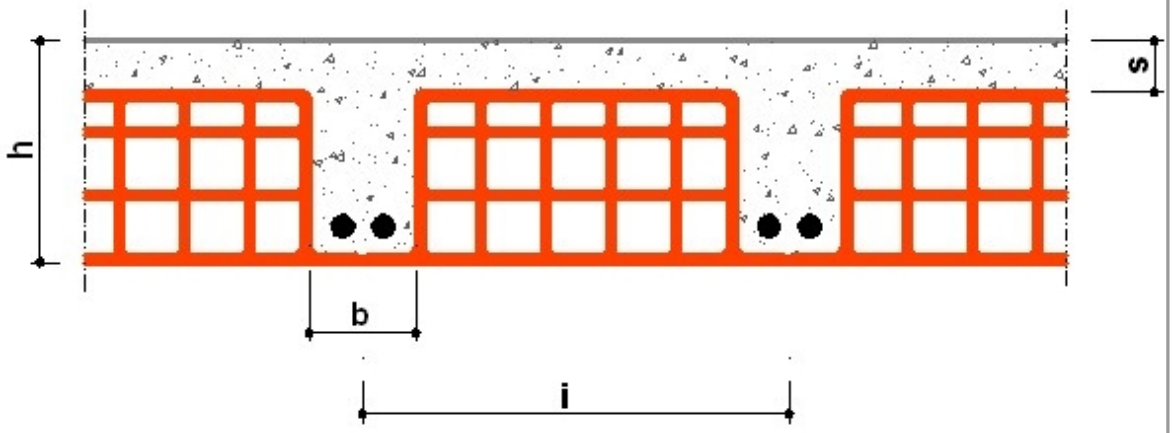
Steel-beam and hollow flat block



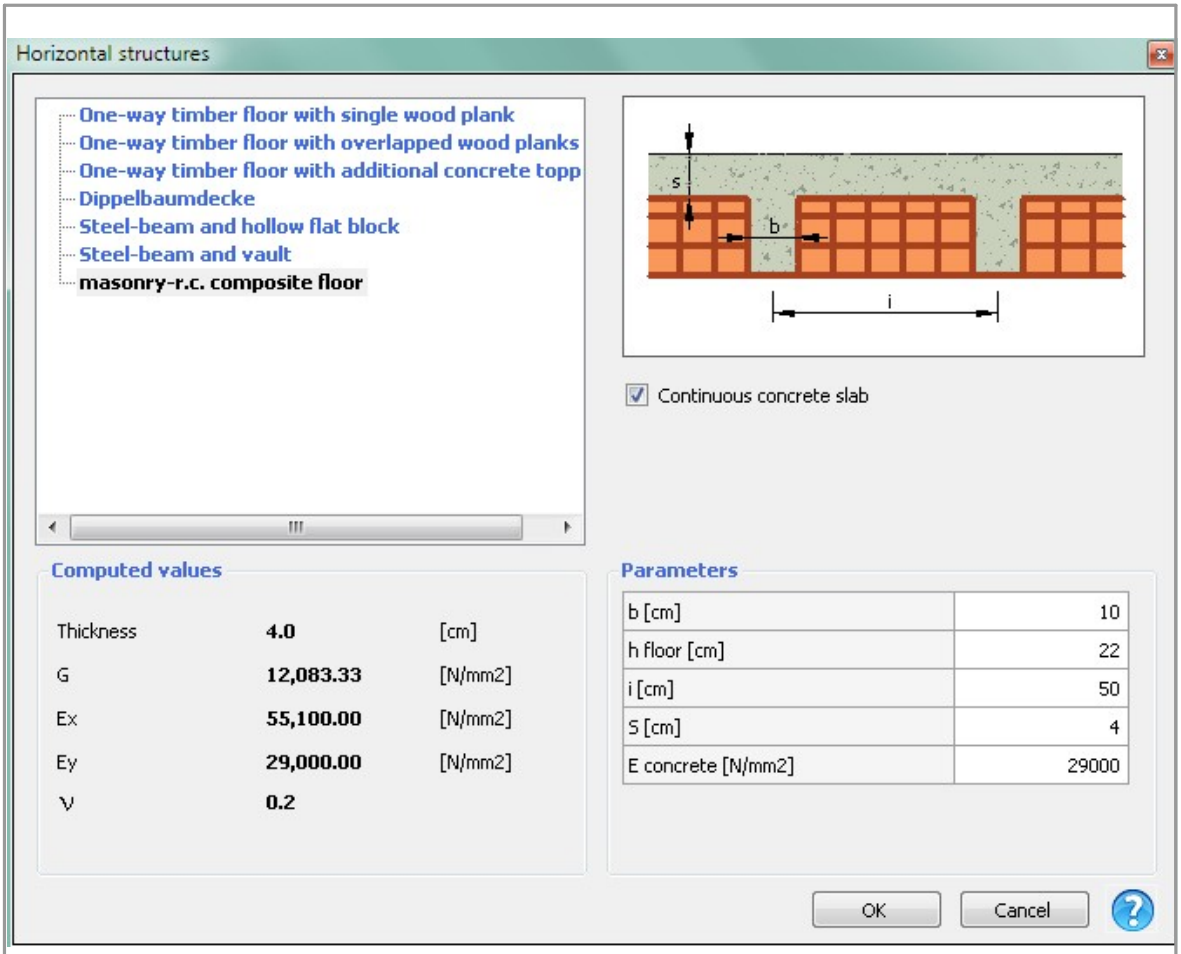
Steel-beam and vault



masonry-r.c. composite floor

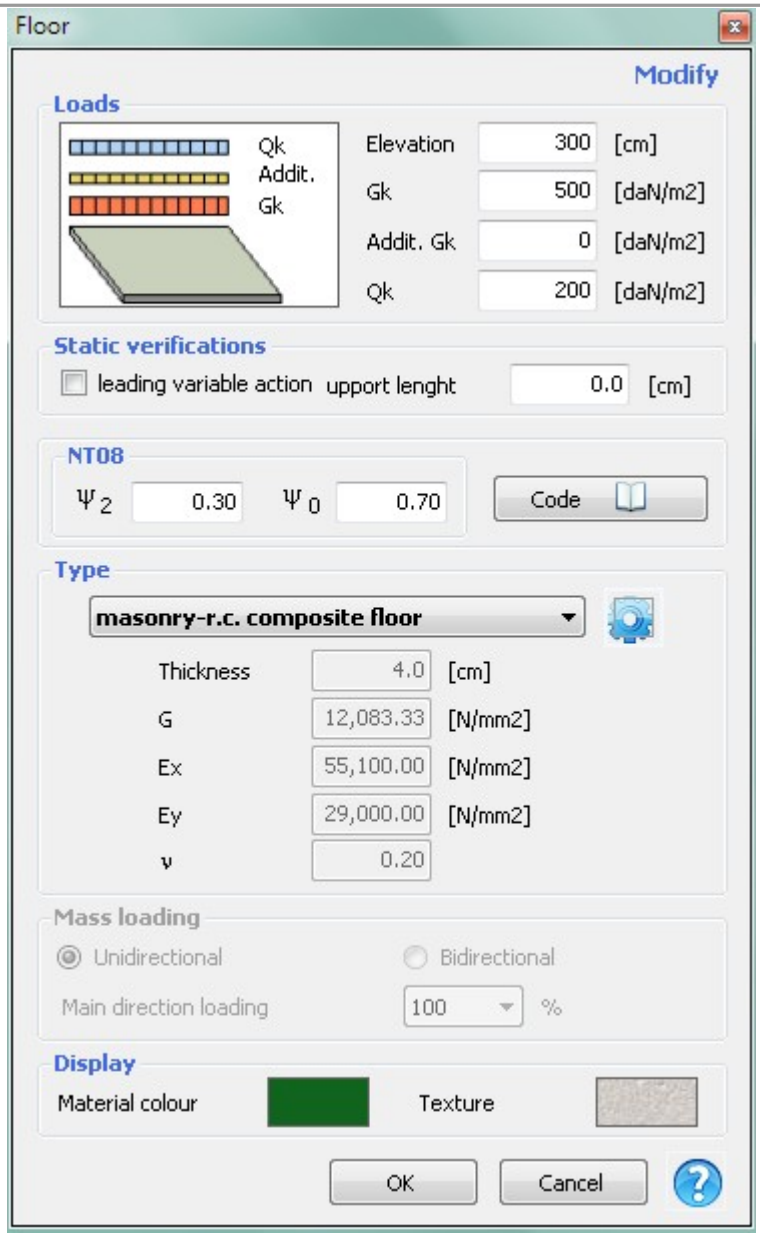


For each of the above slabs typology, the user can decide which of the structural components are well connected to the masonry. (i.e. guaranteeing the connection is equivalent to guaranteeing an increased contribution to the resistance for the global system).



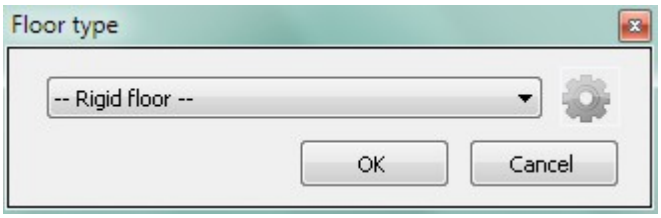
After entering the geometric mechanical parameters, by turning the OK button the user is prompted to neatly select the nodes on which the slab will rest thus a structural reference to define the warping direction of the slab (parallel, perpendicular or by choice).

At ended selection is displayed the following dialog.



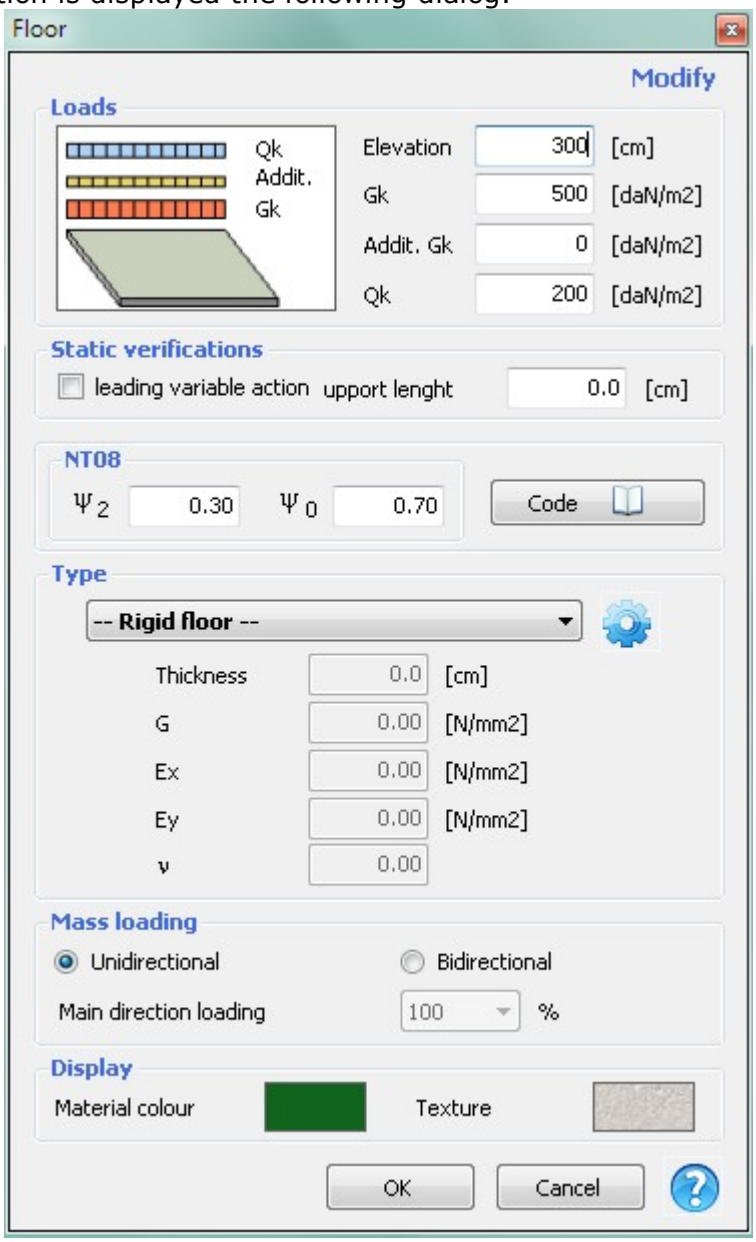
Smart Version

The displayed dialog shows to the user that through this feature will proceed the insertion of the rigid deck.



By clicking the OK button the user is prompted to neatly select the nodes on which the slab will rest thus a structural reference to define the warping direction of the slab (parallel, perpendicular or by choice).

At ended selection is displayed the following dialog.



In the upper part, can be inserted the load actions on the slab as either permanent (Gk) or variable (Qk), that can be combined according to the coefficients indicated in the code. If the user desires, it is possible to use the "Code" button to get additional information about the combination coefficients choice.

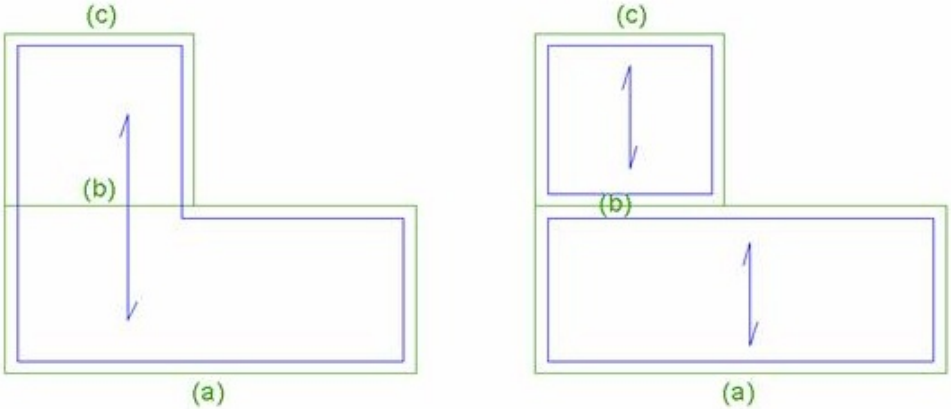
The permanent loads (G_k) are defined as permanent structural loads (G_1)
The permanent loads ($G_{k,agg}$) are defined as the weight of all non-structural elements (G_2).

"Static checks" contains the necessary parameters to perform the static checks. It is necessary to check that the slab being examined is covered and indicate the support length of the floor on the wall. If the user does not intend to perform static checks, but merely seismic checks, it is not necessary to insert these parameters.

In addition, it is possible to decide whether the floor divides its mass in a single direction or along the two directions of the level. If the user decides to divide the masses bi-directionally, it is necessary to indicate the vertical load percentage for the principal direction. (calculating the mass that bears on the secondary direction) If the user decides to use a predefined floor type from the horizontal structures window, the discharge typology is automatically defined by the structural typology. Hence, it is not possible to change it in the floor insertion window.

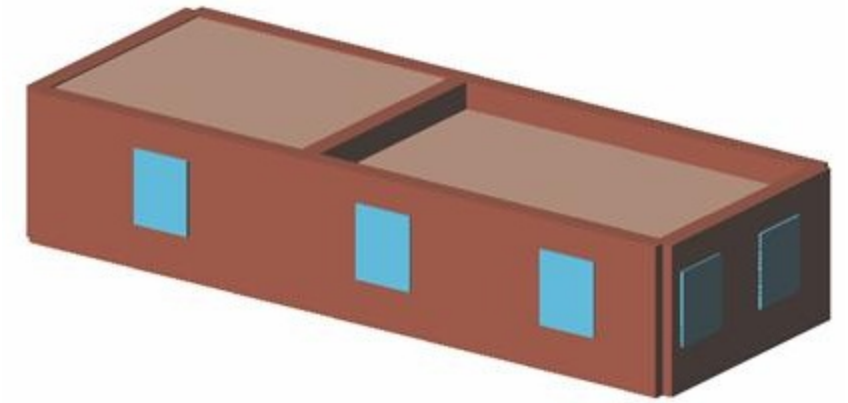
Bending modules Ex and Ey refer to the local axes system (x, y) in which "x" is identified based on the warping direction and "y" is perpendicular to the warping direction.

When inserting the floor, it is sufficient to highlight the external perimeter of the building. The program automatically recognizes the bearing structural elements on which to discharge the mass, without having to separate the floor into additional sub-areas.

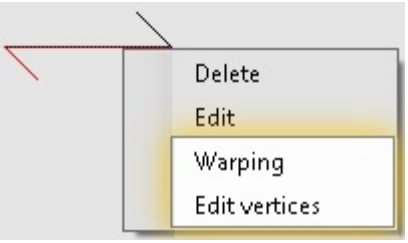


[Wall (b) is borne by the floor independently from the chosen insertion mode]

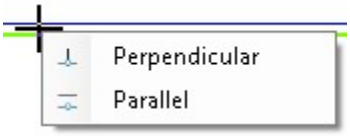
If there are different elevations of the floor on the same level, it is possible to define these by inserting the effective floor elevation in the respective insertion window.



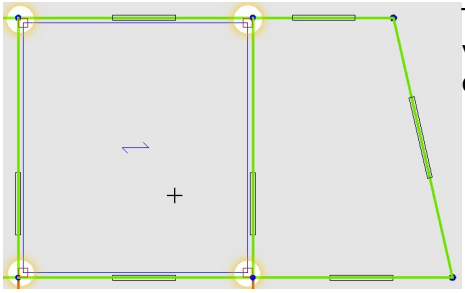
The program does not create additional computation nodes in correspondence with the position of the slabs. It continues to use those already defined, taking into account the contribution due to the transfer of the floor with respect to these limit nodes between one level and another.
It is not possible to insert floor with an elevation superior to the current level, unless there is already a defined level above it.
In order to create reliable models, it is important to construct the model so that the level elevation is the average value for all the elevations of the various floors defined on that level.



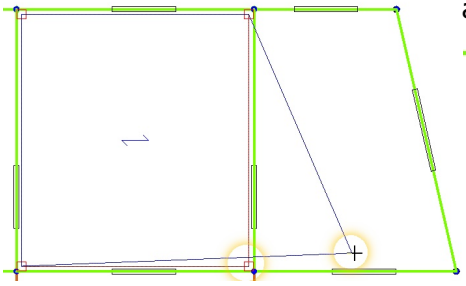
Slab warping direction:
Select a wall bounding the slab in order to redefine the warping direction of the slab.



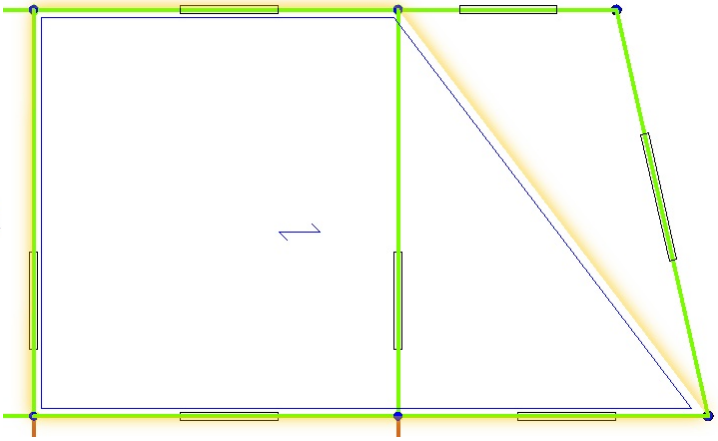
Edit vertices:
Once inserted a slab you can edit the vertices hooking them to different nodes.

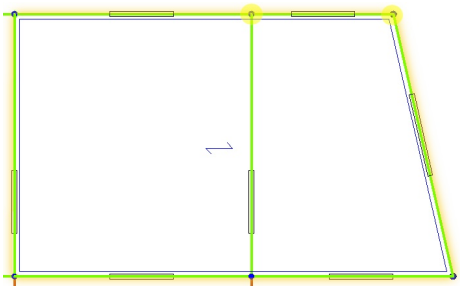


The "handles" are shown in correspondence to the vertices, if dragged they allow to define the vertices of the polygon in correspondence with other nodes.



By dragging a single vertex it is repositioned to another node.



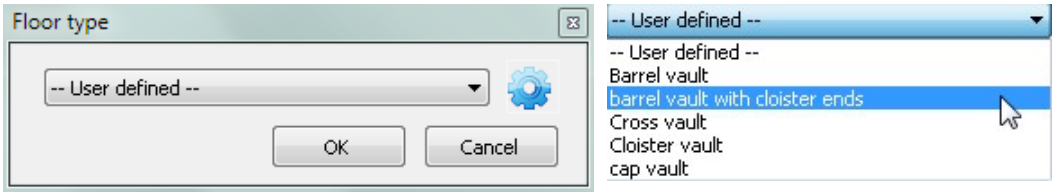


By invoking the command you can repeat the operation for other vertices

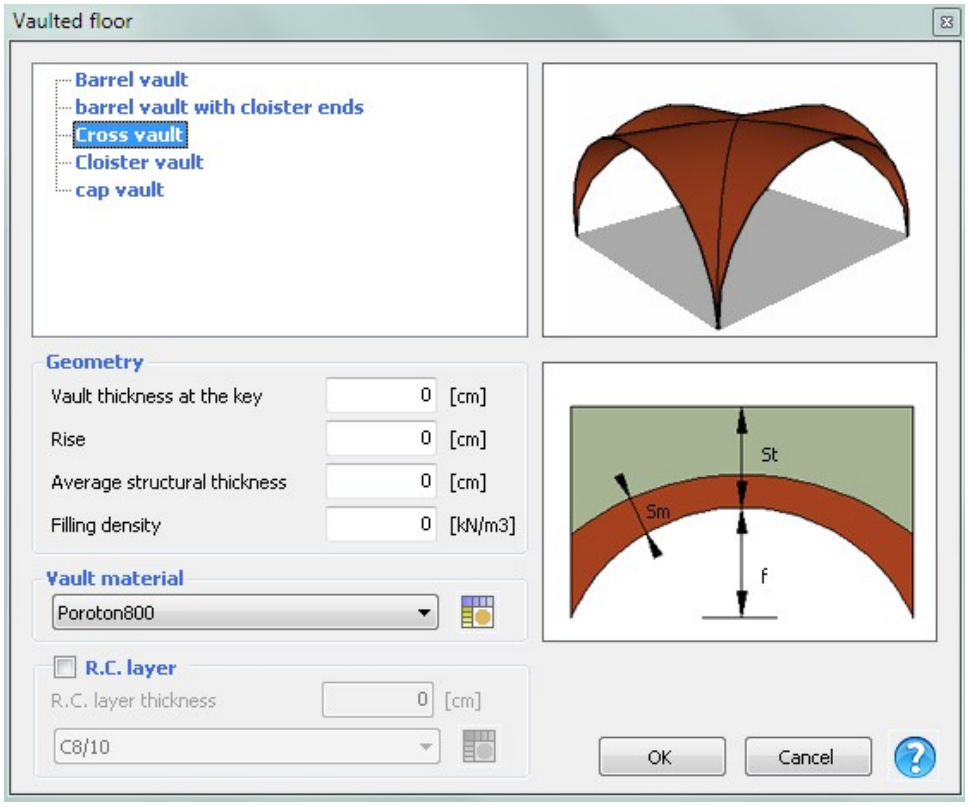
9.2.10 Vaults

allows insertion of vaults.

A window opens in which the user can select the desired vault type.

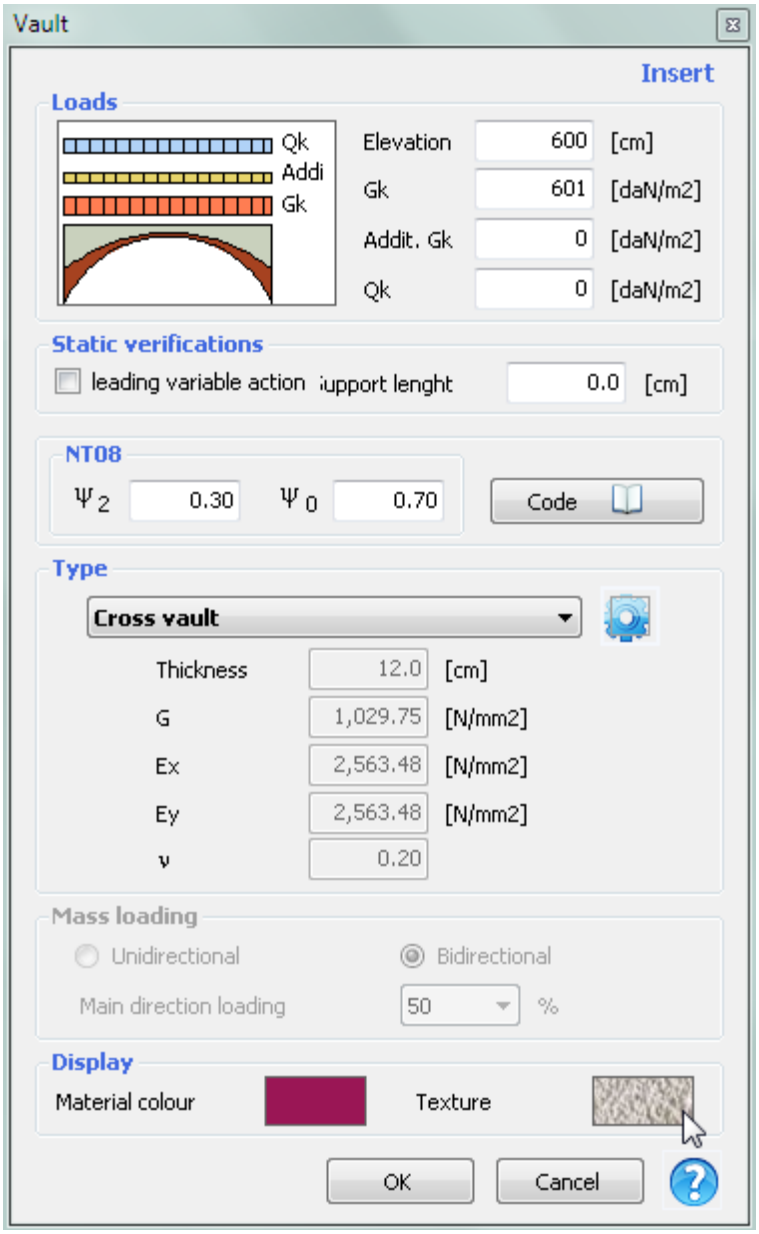


For each vault typology (listed above), the user must define the main parameters.



After having inserted the geometric mechanical parameters, click the OK button. Then carefully select the nodes on which the vault will rest. After, select a reference structural element to define the direction for the vault's discharge (parallel, perpendicular, or user defined).

When selection is finished, the following window will appear.

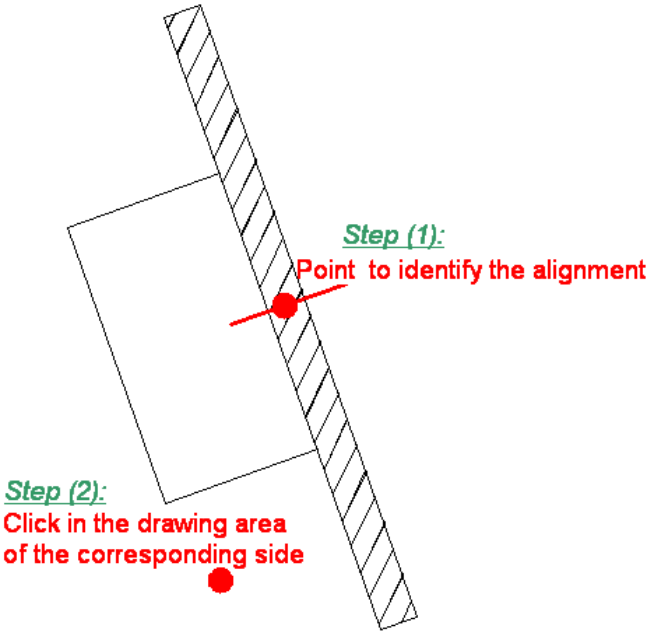


This window is very similar to the floor window.
In this case, the vault's permanent structural load (Gk) is automatically calculated.
The user must insert the additional permanent Gk (e.g.: weight of the trimming work) and the accidental loads (Qk).

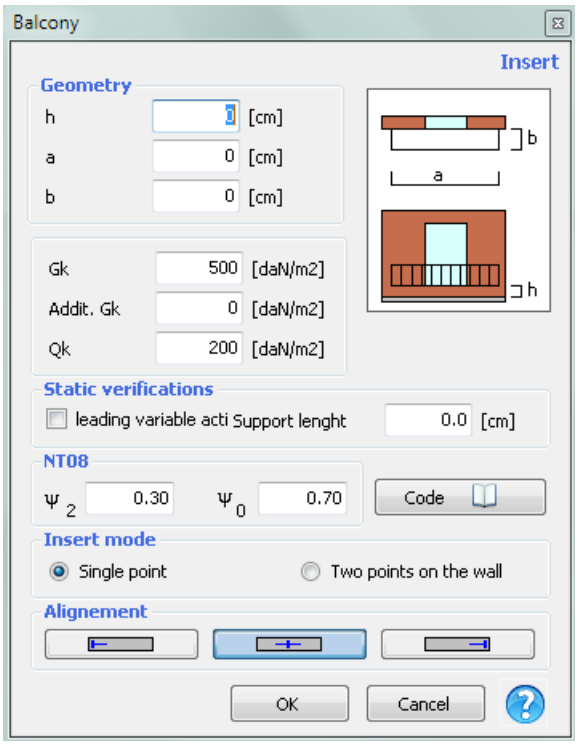
9.2.11 Balconies

Allows the insertion of balconies.
The insertion occurs through the insertion of the following parameters:
-Geometry: the geometry of the floor plan (axb); h indicates the difference between the elevation of the balcony and that of the lower level.
- Gk and Qk indicate the permanent and accidental loads.
-Multiplier coefficient as defined by code.

Method for insertion:
Single point: A point on the wall is selected to identify the fixed alignment; the side of the wall on which the overhang will be created is identified by clicking in the drawing area of the corresponding side.



Two points on the wall: The length of the balcony is inserted graphically through the insertion of the starting and ending points without the use of fixed alignments for the insertion.



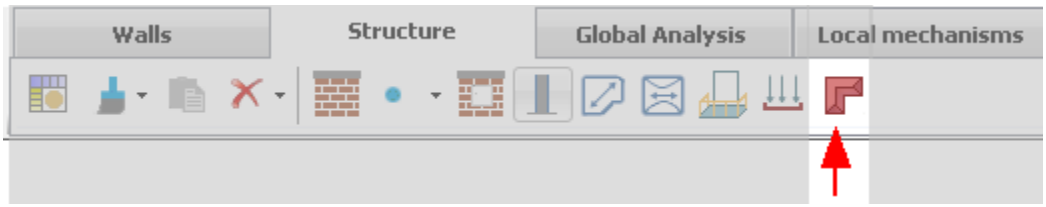
9.2.12 Coperture

[OPTIONAL MODULE]

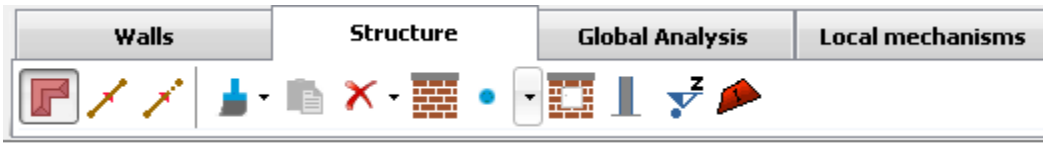
3Muri allows to model roofs on different levels, therefore for entering or editing a roof

the first thing to do is to select the level you want to work on.

The insertion of the roof elements is done by using the relevant button of the environment structure.



By pressing this button the environment "roofs" is activated along with the typical environment toolbar.



Allows to exit from the roof environment and return to the structure environment

Profile

Allows to insert a "roof profile"

Extend Profile

Extends an existing profile

Copy attributes

Copies the definitions of the structural elements characteristics.

Paste attributes

Paste the properties of the selected element using the copy command.

Multiple Deletion

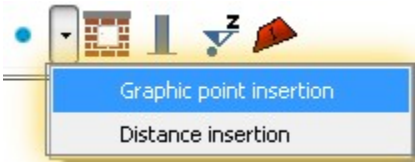
Delete various already inserted elements.

Assign attributes

Assigns the attributes of the structural objects

Graphic point insertion/ Distance insertion

Allows the insertion of a node "element node" (with the help of the graphic support) or within a predetermined distance from an edge node.



Opening

Allows to insert an opening in a panel.

Column

Allows to insert a column matching one or more nodes.

Elevation

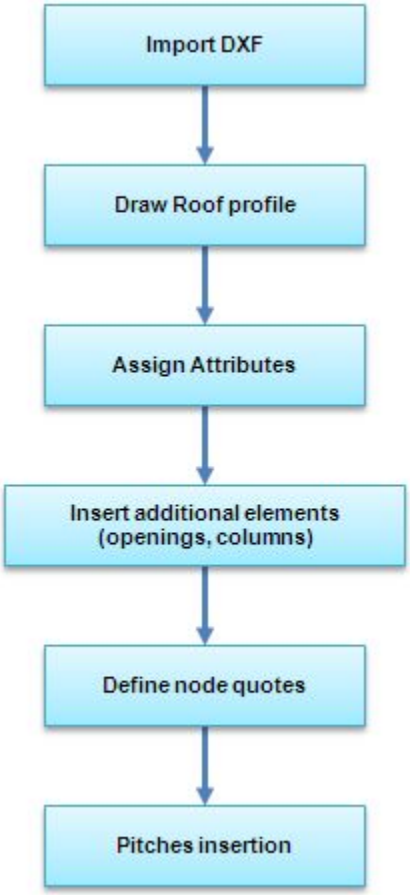
Assigns the height of nodes

Pitch

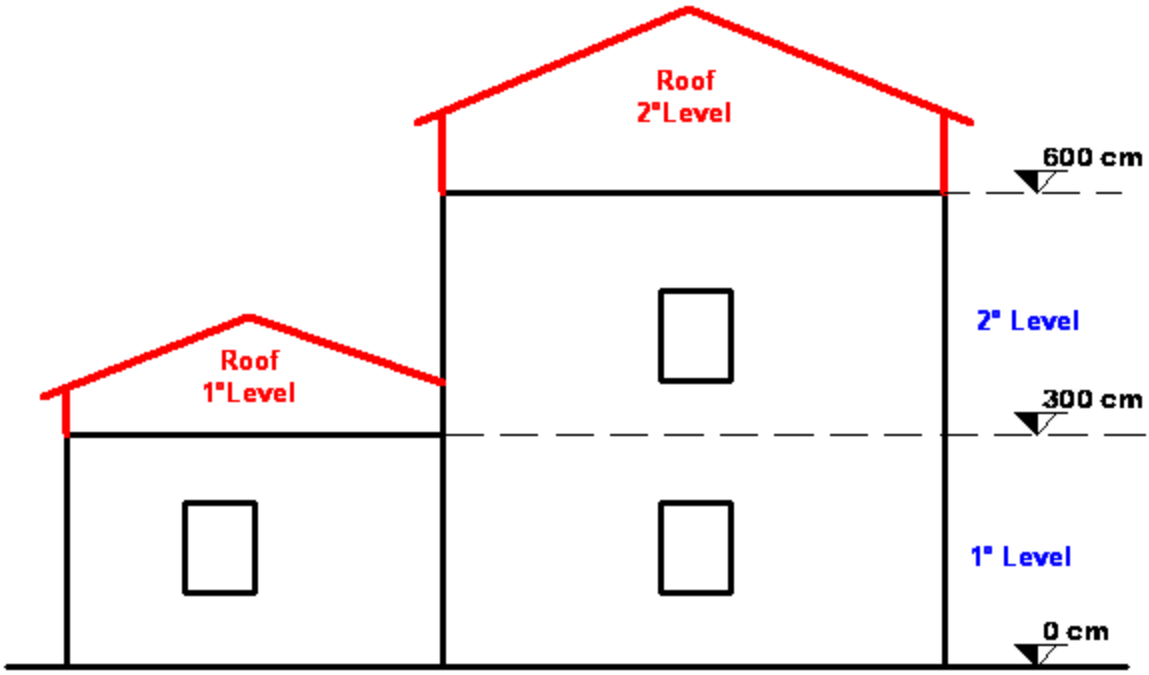
Allows to insert the pitches

Info:
This module is available with the acquisition of the appropriate licence, the "Standard" version of the product contains no such form. For more information contact your distributor.

9.2.12.1 Modeling Mode



The roofs are made from a set of structural elements that are part of the active level, so it is possible to define a roof for each level.
This input method allows to define a system of roofs of differentiated heights.
A roof is considered to be part of the level to which it corresponds to its lower elevation (see figure below).



The roof can be modeled as "non-structural" or "structural", with the following difference:

NON-structural:

This is the typical case of a timber roof in an existing structure. In this case the designer may not want to run the risk to assign the seismic lift and the ability to transfer the forces to a low stiffness system (such as timber) because of the limited information on the good anchoring with the masonry.

In such cases it's better to ignore the strength and stiffness of these elements. This way they won't come into play at the meshing moment and get transformed in loads applied to the underlying structure.

The same portions in masonry (e.g. tympani), in the pitches without good stiffness, might lead to mechanisms out of plane; in this case it would be appropriate to exclude the stiffness of such masonry elements.
In this case, the mesh of the building would be the same with the case where the mesh had been performed before inserting the roof.

Structural:

In cases where the pitch owns a significant stiffness, you can use it in order to have a distribution of forces more consistent with the reality.
In this case it is necessary that all the structural elements of the roof are involved in the mesh of the structure.
The pitches are made up of surfaces (NOT always flat) that are discredited using triangular mesh with membrane type elements (the same element used for floors).
Given the irregularity of the existing structures, it often happens that, in order to follow accurately the wall profile, not coplanar beams of a single pitch are laid, this is why you can also enter non-flat pitches.

The masonry walls are modified in height and shape to properly follow the perimeter of the pitch.

Example of mesh shown in the analysis environment

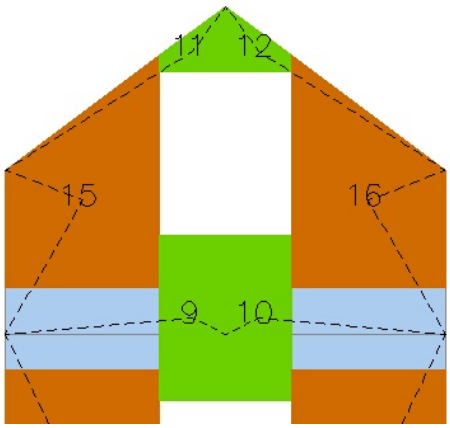
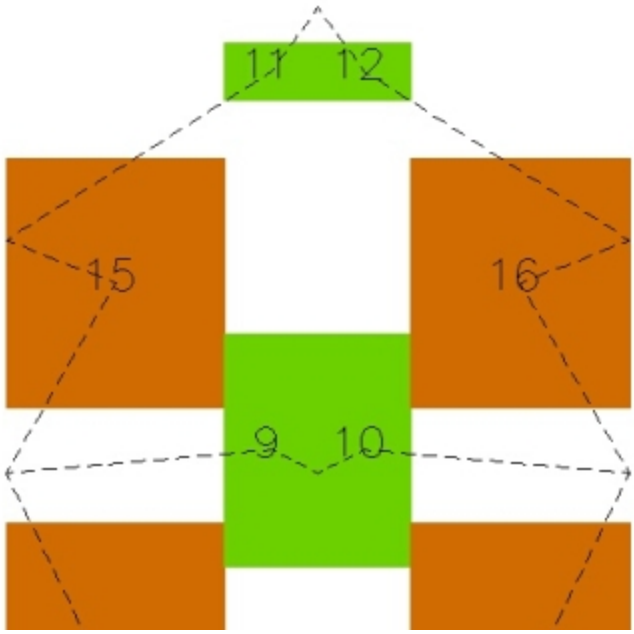
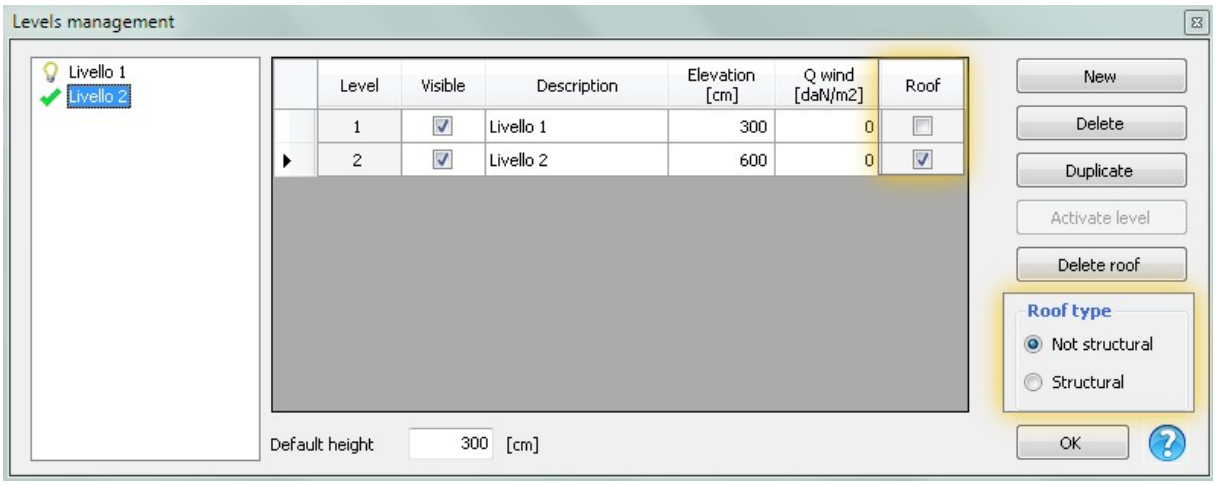


Diagram of the pier/spandrel elements with the actual heights used for the calculation



The typology of the "non-structural" or "structural" modeling is conducted by the appropriate function in the window "Levels management".
This option is a property of the individual level, allowing to decide whether a roof is both structural or less depending on the belonging plan.



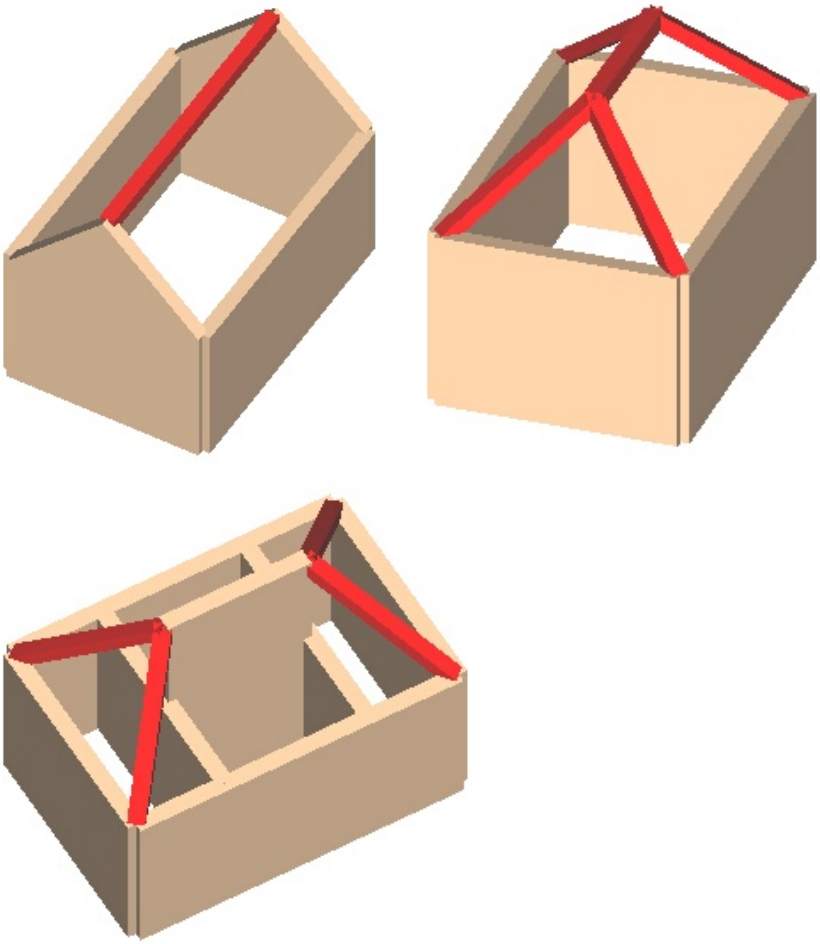
When you enter a roof for a level the check mark appears in the box of the column **"Roof"**.

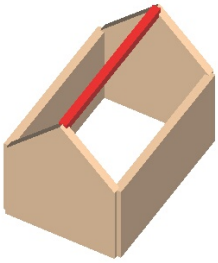
You may decide to remove a roof from a level by simply clicking on the ***"Delete roof".***

9.2.12.2 Roof Elements

The insertion of roof elements is done in the proper environment which can be accessed by clicking 

The roof profiles (in red in the figure below) are useful only for modeling the roof and can be created and edited ONLY in the roof environment. So they are not accessible in the walls environment.



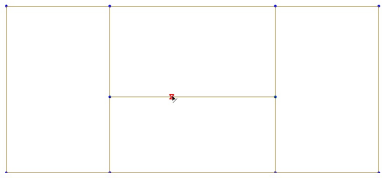


Once you enter in the "roof" environment you can see only the elements already present in the "walls" environment.

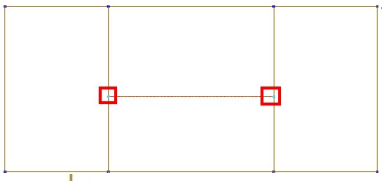
You need to add a single roof "*profile*" with the aim to describe the ridge

Extend Profile:

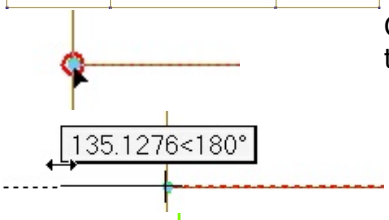
It allows to extend an already set profile, let's look at the utilization phases:



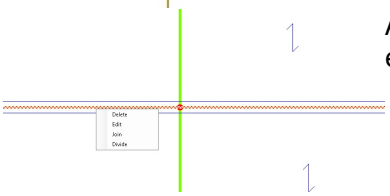
Select the profile or wall that you want to extend



The endpoint joints of the selected segment are highlighted in blue.

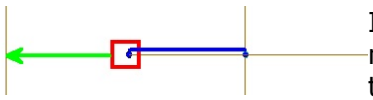


Click on an endpoint node (selected) and drag to extend the segment.



An "edit" or "delete" command takes effect only on the extended portion of the object.

A segment produced by an elongation CANNOT be affected by an elongation, it is therefore not possible to stretch a segment which is himself an elongation.
If there is this requirement, you should delete the elongation of interest and then reinsert it.

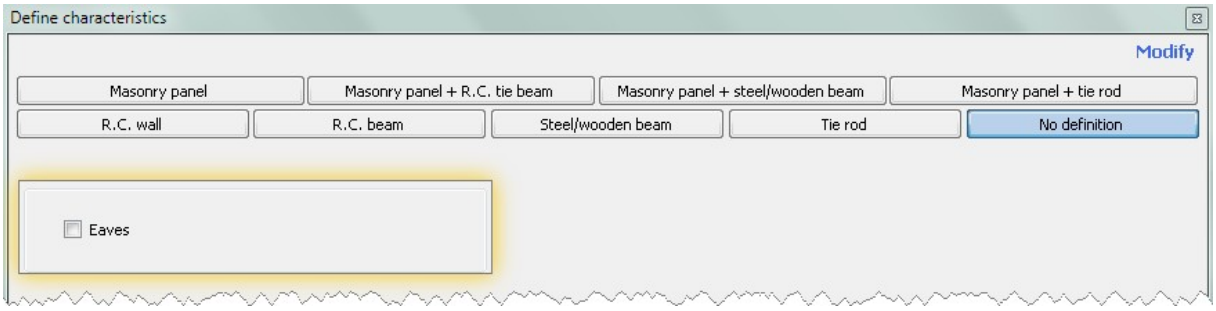


If you wish to extend the highlighted node on the wall is not possible because the highlighted line in blue is already the product of an elongation.
1. delete the inserted part: right click on it and choose "Delete"
2. redo an extension for the entire stretch.

The "extend" command cannot be used to trim a wall.

Assign attributes:

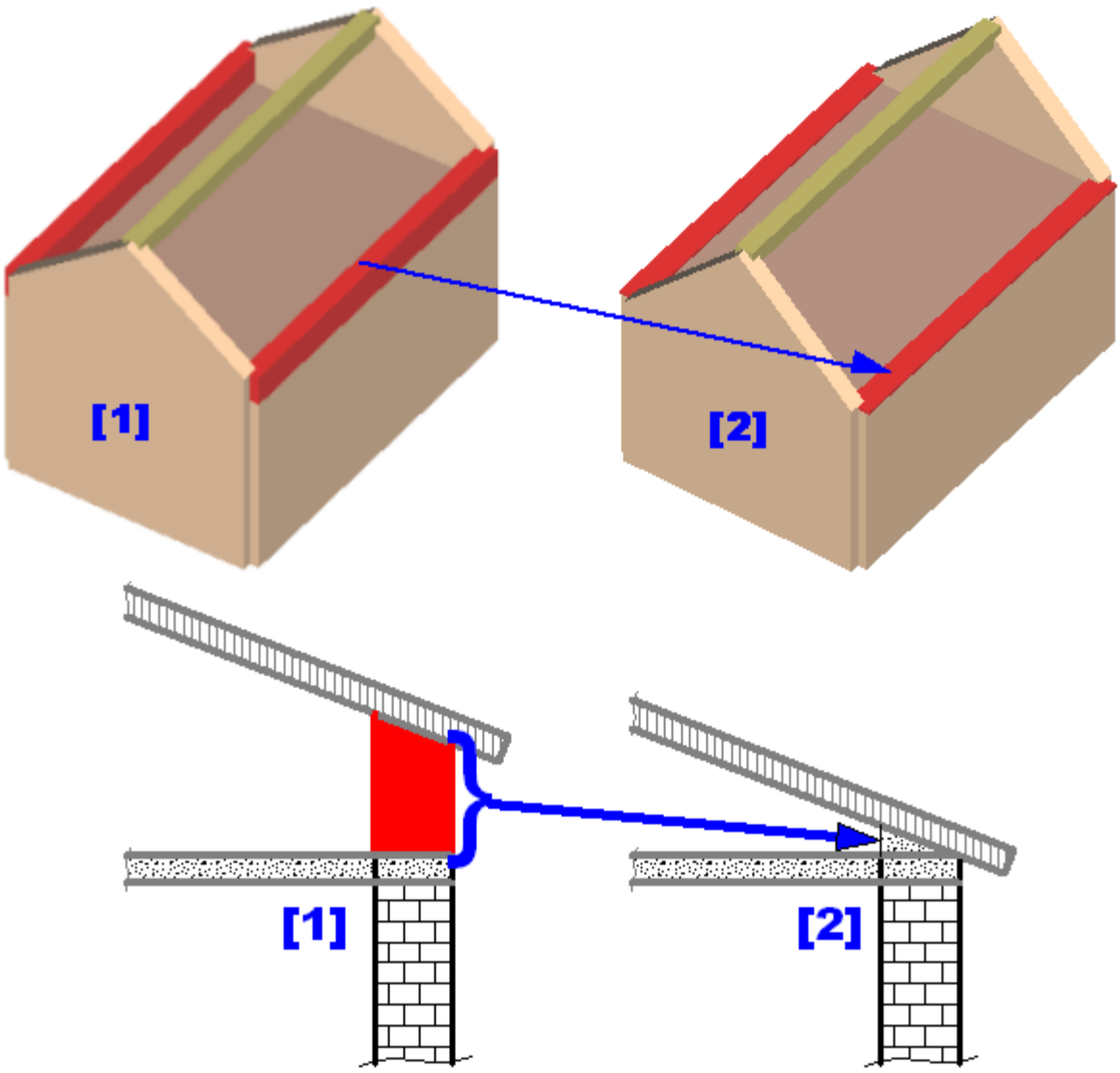
This command is very similar to the "Definition of structural objects" (Simple elements, Composite elements) already known to the structure environment.
In addition to the classic features, there is also the possibility to define a **"Pitch support".**



When is it possible to use the definition ***"Pitch support"?***
All the times when a pitch is supported by a structural element already defined in the structure, let's now see some examples:

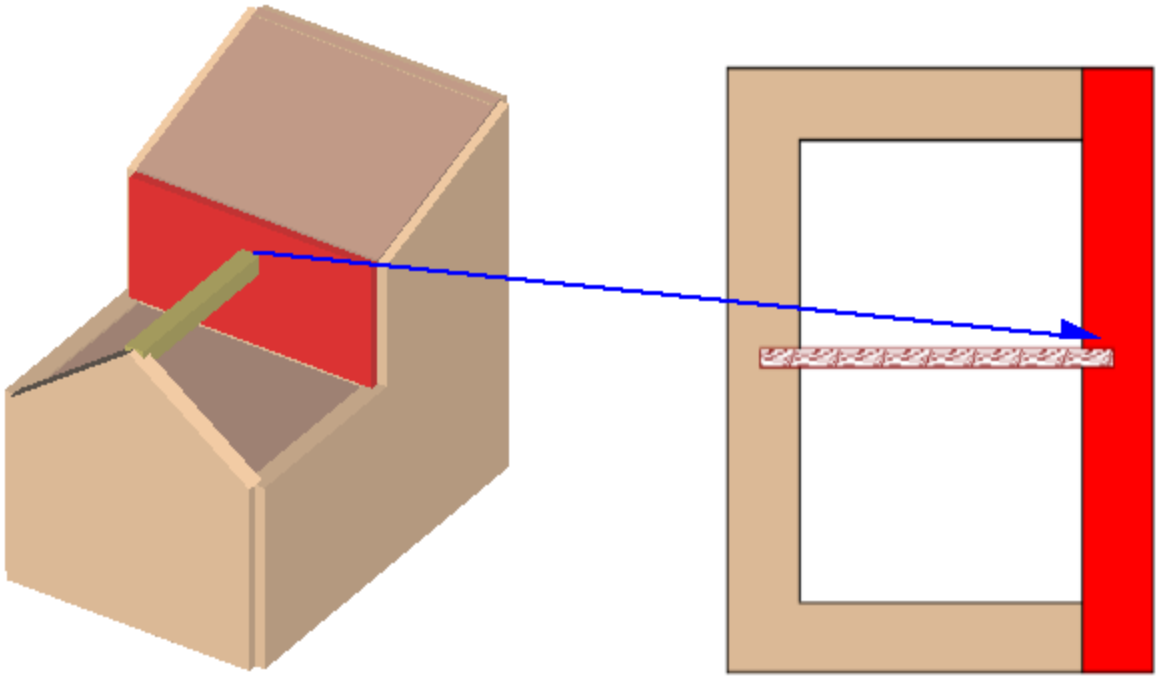
1. Heigt of the eaves coincides with the height of the level:

In Figure [1] the pitch is supported in correspondence of the eave by a masonry wall defined in the roof environment➔ it is **NOT** *"pitch support*" but "*wall panel*".
In Figure [2]] the pitch is supported in correspondence of the eave by a masonry wall defined in the structural area that doesn't exist in the roof area ➔it is *"pitch support*".



2. Roof elements that are supported by structural elements defined for the next levels

In the case of buildings formed by structural bodies defined at different heights, each one with its specific roof, you can create cases similar to that described by the following figure.
The wall highlighted in red is roof supporting masonry wall as well as wall of the second structural level, in this case you shouldn't define the masonry wall in the roof environment as it is already defined in structure environment but it is necessary to define this segment as **"pitch support".**



Opening:

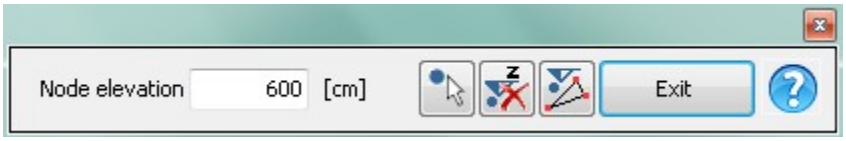
In the walls of the roof environment can be inserted openings exactly like in the structural area.
More details in the corresponding chapter "Structure Features>Definition of Structural Objects>Openings".

Column:

As well as in the structural environment can be inserted columns in reinforced concrete, masonry, steel and wood structure.
More details in the corresponding chapter "Structure Features>Definition of Structural Objects>Columns"

9.2.12.3 Define the heights

The allocation of the structural elements' heights that make up the roof is defined by assigning the heights to the nodes.
Pressing the appropriate button a dialog box appears for the definition of the height.



The "*Quote node*" field presents by default the maximum height of the active layer that corresponds to the minimum height of the begging of the roof.
Phases of the dimensioning of the nodes:
1. Enter the height value of the nodes
2. Press , the pointer changes shape and the definition window of the height disappears.
3. Select the nodes affected by that height:

Selection mode	Description
Single click on the node	
Multiple selection with window mode	

4. Press the right mouse button to confirm the selection; the definition window of the height reappears in the foreground.
A ***dimensioned node***, is shown by magenta.
By placing the pointer on the node, the value of the allocated height is showed in square brackets

A ***NON dimensioned node***, is shown by blue.
By placing the pointer on the node the sign "[-]" appears to indicate that the height is not allocated.

5. Repeat the steps described so far until you have entered all the necessary heights.
6. After entering all the heights press **"Exit"** to exit.

Cancel the input of a height

Allows to remove the definition of the node height.

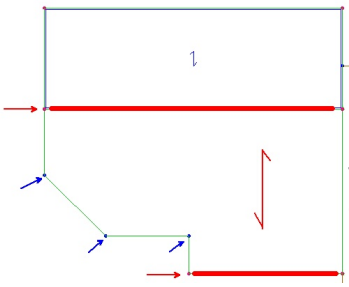
Why is necessary cancelling the insertion of a height?
The not defined (canceled) heights, if affected by a pitch profile, can be calculated automatically by interpolation between the outer heights.

Once defined the units of **A** and **B**, those of **C** and **D** are obtained automatically by interpolating tract A-B.

Calculate height

There are many cases in which the ground survey is not accurate enough and doesn't

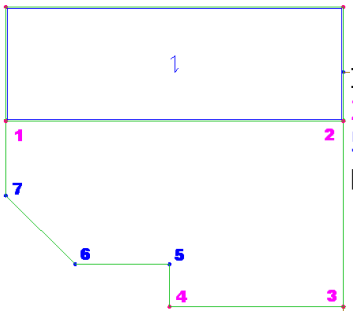
allow to know precisely the proportion of all the nodes.



For example, in this case we know the height only of the points that define the ridge and the eaves (highlighted in red) of the pitch that we want to design but we don't know the heights of some intermediate points (marked with a blue arrow).

The "*calculate height*" command allows to automatically calculate the heights that aren't defined.
The calculation is done by interpolation from the elevation of 3 dimensioned nodes.

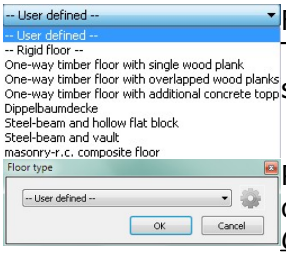
Select in sequence first the 3 nodes (dimensioned) that define the plane and then those for which you want to calculate the height.



In the case of the example described above you can select 1-2-4-5-6-7 so that the program calculates the dimensions of 5-6-7 so that they are coplanar with 1-2-4 to create a flat pitch.

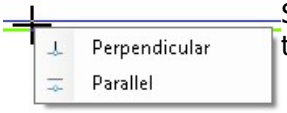
9.2.12.4 Pitches

The insertion of the pitches is done by using the relevant button.

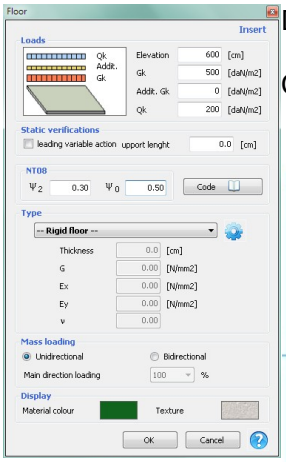


First, you must choose the type of structure of the pitch.
The types presented are the same as described above for the slabs.

Pressing "**OK**", you must proceed by clicking on the nodes that define the perimeter of the pitch.
Only the dimensioned nodes (color magenta) can be selected in the definition of the pitch profile, so it is essential to dimension the nodes before defining the pitch.
Click with the right mouse button to close the polygon that defines the pitch.

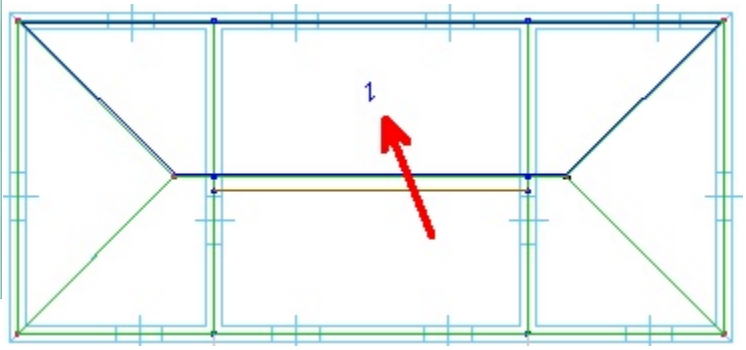


Select with the left mouse button a profile / wall roof to bring up the context menu of the direction of framework.



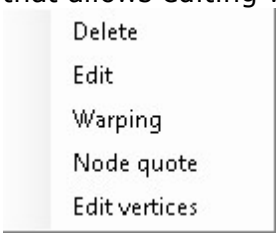
Define loads and characteristics of the pitch.

Confirm with "**OK**" to see the inserted pitch.



Edit pitch

Clicking with the right mouse button on one side of the pitch, the contextual menu that allows editing will appear.

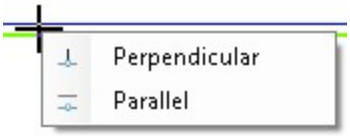


Cancel: Cancel pitch

Modify: Appears the input window used to define the loads and allows editing.

Pitch warping:

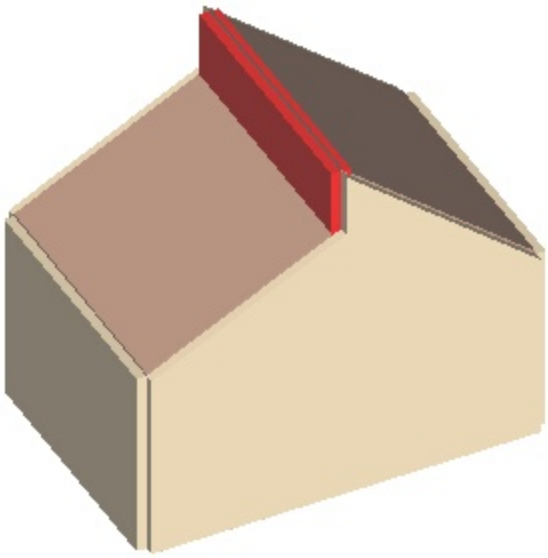
If the pitch's warping previously inserted, it is not correct and you want to change it, by selecting this command is again required the selection of the structural reference for warping; by clicking on a stand is resubmitted the menu for the selection of the warping.



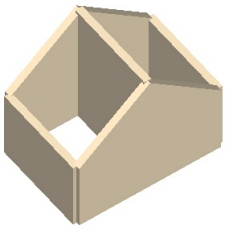
Pitch nodes heights:

Allows to act on the height of the nodes of the pitch without changing the height of the node itself.

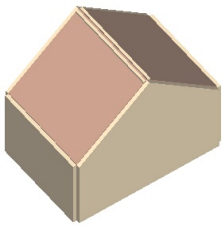
The most common application of this command is to create a model shown in the following figure (in red):



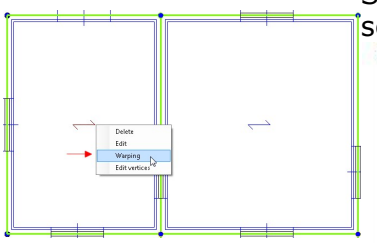
Let's see the steps of creation of this model:



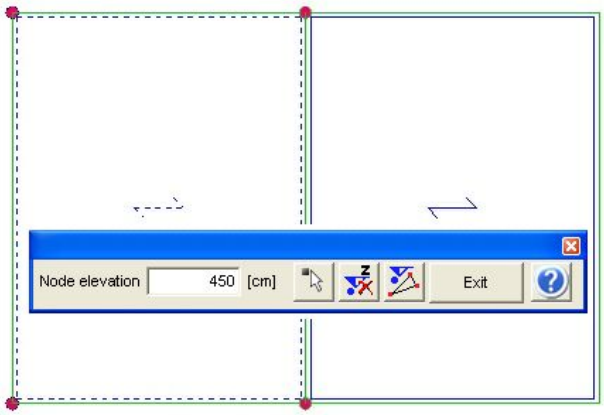
Create the model with gable and walls that extend up to the maximum height of the ridge.



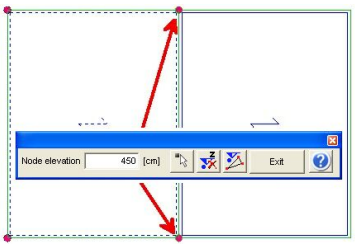
Insert the pitches with the above procedure



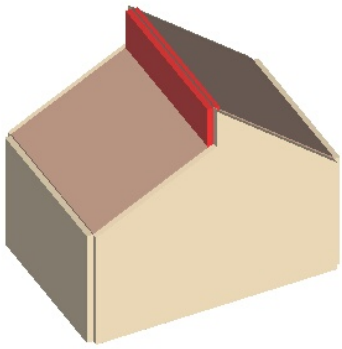
Select the pitch of the left with the right mouse button and select *"Height pitch nodes ".*



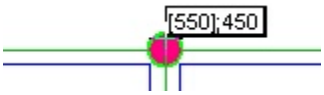
The defining heights nodes window appears with highlighted dimensioned nodes that affect only the selected pitch.



In the field "*Node height* " insert the top dimension that you want to define on the ridge in order to create the folder and assign it only to the two nodes of interest.



WARNING !!: It is NEVER possible to assign a height to the pitch nodes LOWER then the height assigned to the node.



When a node has been defined with the double height, it appears a label with the list of height values associated to the node.
The values between "[]" are the heights associated directly to the node.
Values without "[]" are the heights associated to the node as a typical height of an end pitch (through pitch editing).

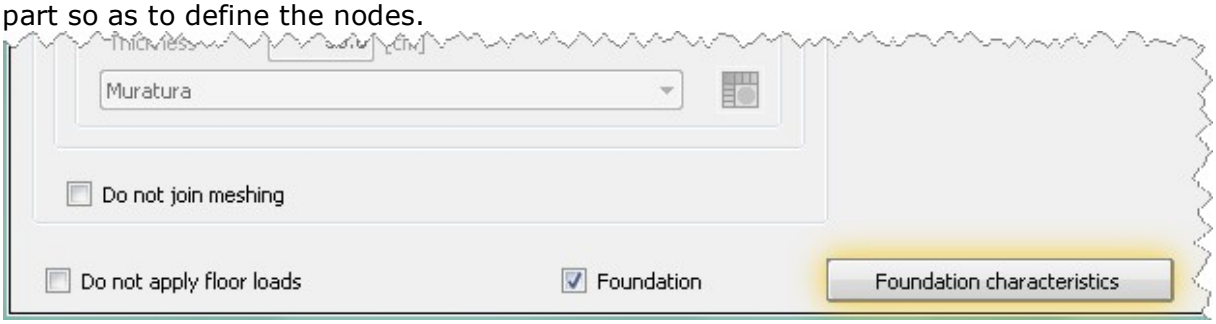
Edit vertices:
It activates the handles corresponding to the vertices of the slab polygon.
By dragging the individual vertices it is possible to change the incidences.

Add overhang:
After calling the command, it is necessary to:
- select the side on which to place the overhang
- enter the distance/extension of the overhang
- click in the blank area by indicating on which side to insert the overhang

9.2.13 Fondazione

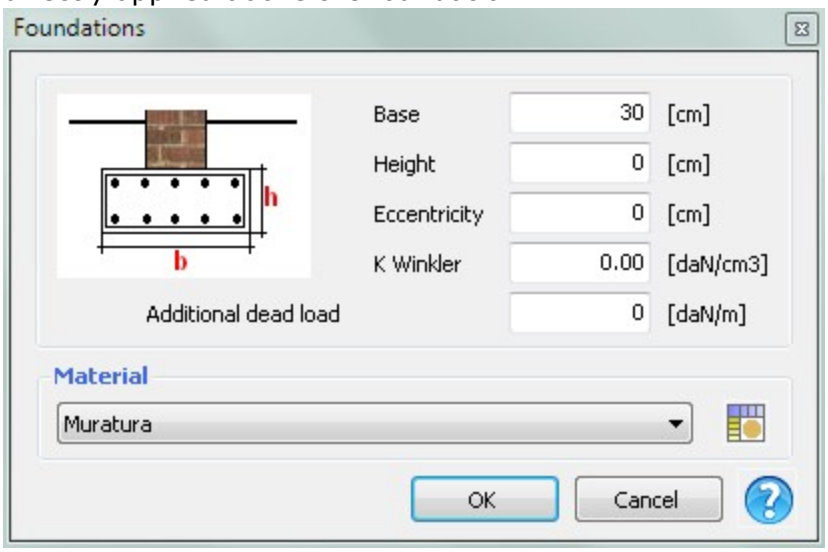
9.2.13.1 Masonry Panels Foundation

There is the possibility to activate a box named "Foundation" so that the user during the insertion phase can decide if each panel goes directly to the foundation in its low part so as to define the nodes.

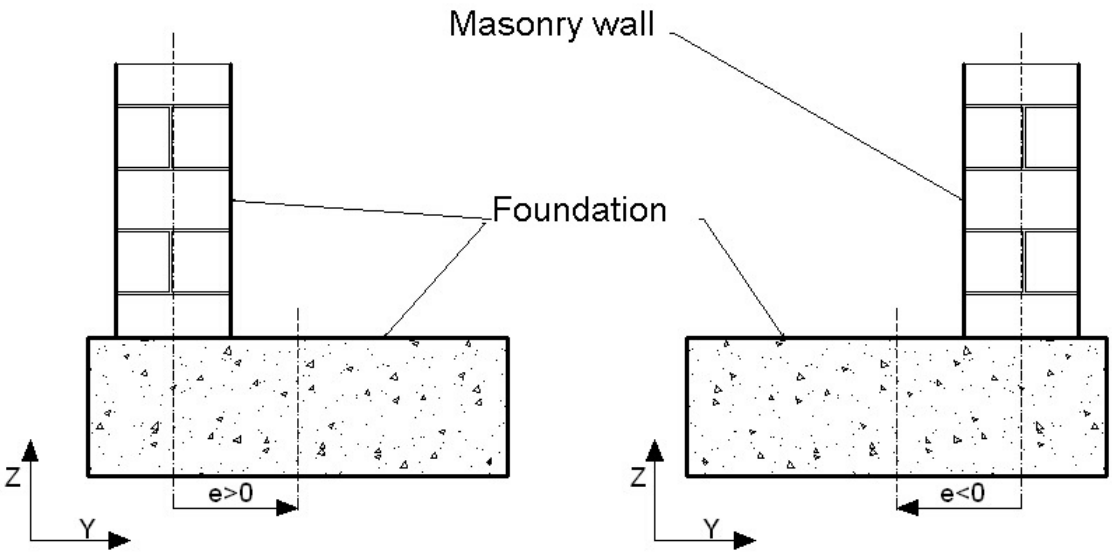


This item appears active and not editable when you insert the first level because the nodes at the base are certainly part of the foundation; it appears disabled but editable in the next levels. The choice of a panel that goes directly to the foundation means to constrain all the freedom degrees of the nodes to the base (both translation and

rotation). Different boundary conditions can be inserted only during the mesh editing in the analysis while viewing the prospectus of a mesh wall.
After deciding that a structural element taken in consideration is part of the foundation, the "Foundation Characteristics" button is activated in order to define the foundation system's characteristics necessary to calculate the tensions in contact with the ground.
The displayed dialog defines:
- the foundation beam's dimensions, the material
- The Kwinkler
- The eccentricity of the masonry panel compared to foundation beam
- a dead load directly applied above the foundation

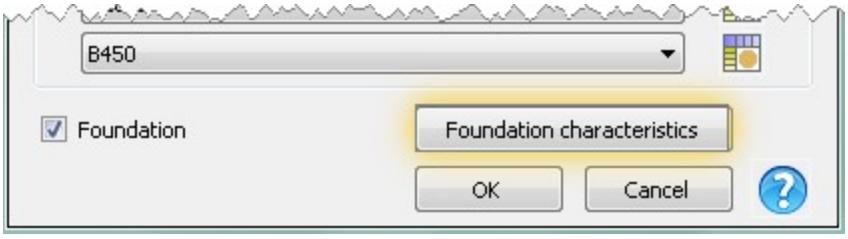


The foundation's eccentricity is calculated with the convention sign shown in the following figure.



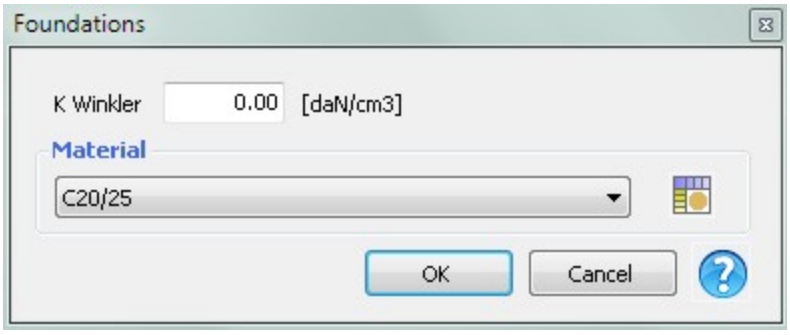
9.2.13.2 Columns Foundation

There is the possibility to activate a box named "Foundation" so that the user during the insertion phase can decide if each column goes directly to the foundation in its low part so as to define the nodes.



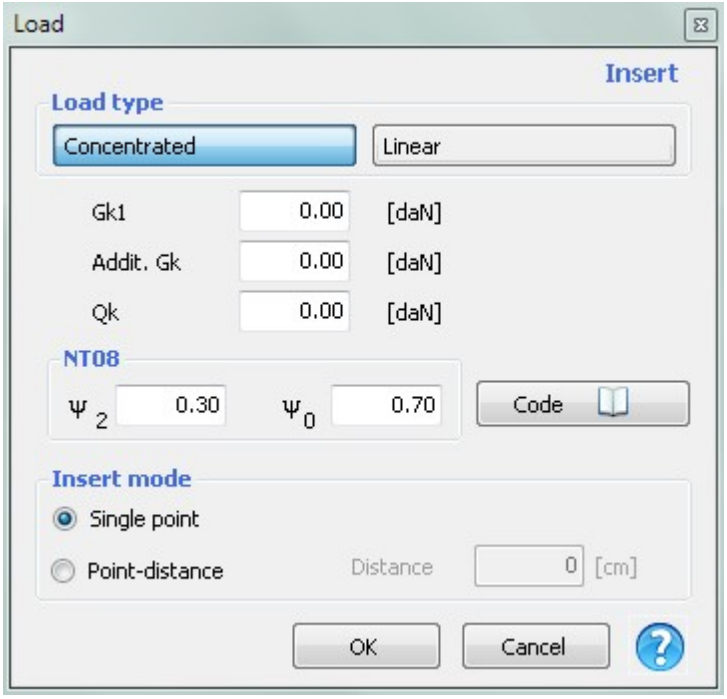
This item appears active and not editable when you insert the first level because the nodes at the base are certainly part of the foundation; it appears disabled but editable in the next levels. The choice of a column that goes directly to the foundation means to constrain all the freedom degrees of the nodes to the base (both translation and rotation). Different boundary conditions can be inserted only during the mesh editing in the analysis while viewing the prospectus of a mesh wall.
After deciding that a structural element taken in consideration is part of the foundation, the "Foundation Characteristics" button is activated in order to define the characteristics of the ground.
The displayed dialog defines:

- The Kwinkler
- the foundation's material



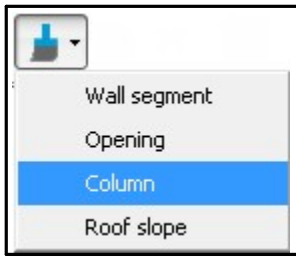
9.2.14 Concentrated and linear loads

Allows the insertion of concentrated or linear loads, both at permanent part as well as accidental.
The window shows the multiplier coefficient for the actions according to code requirements.



9.2.15 Structure Editing

Copy attributes

	Copies the definitions of the structural elements characteristics. Using the drop-down menu, choose the typology of structural element whose properties will be copied. Select the reference structural element to be copied.

Paste attributes

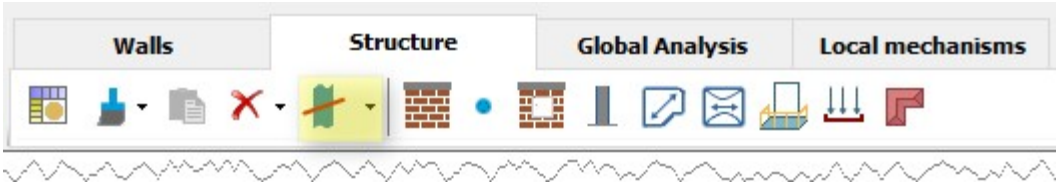
Paste the properties of the selected element using the copy command.
Select, in order, the structural objects that will have the copied properties assigned to them.
End selection of multiple items by pressing the right mouse button.
A video with the characteristics of the structural elements to be assigned to the selected objects will show. Click "OK" to confirm the definition of the characteristics.

Multiple Deletion:

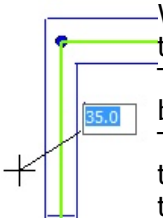
	Delete various already inserted elements. Using the dropdown menu, decide the type of element to be deleted. Select, in sequence, the elements to be deleted. The right mouse button can be used to stop multiple selection and proceed to deletion.

9.2.16 Supported thickness

During the input phase it is possible to stop worrying about the wall's thickness on the various segments and perform the input with any thickness; afterwords you can directly find the thickness from the DXF background.
The command is available directly in the structure area.

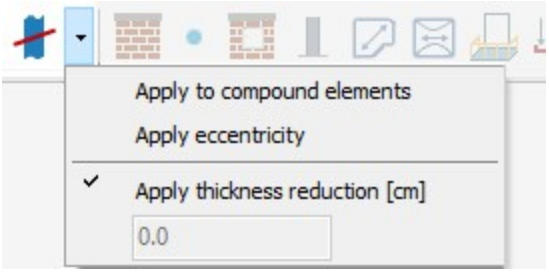


The command runs immediately by pressing the apposite button:



With a double click you can define a segment section that intersects only the DXF that represents the wall's size.
Through this section is read out the wall's thickness directly from the background design and it appears a text box that displays the read value.
This numeric value can be edited directly in the text field or by proceeding to the alignment of a new section segment thus automatically confirming the previously read thickness.
The alignment of a new section line is repeated automatically until you press the right mouse button, that stops this command

The available various options are shown by pressing on the arrow on the right of the command button:



Apply to the compound elements:
The "Supported thickness" command automatically detects the thickness of the masonry walls; often an element can be "compound", for example the combination of a panel + R.C. tie beam. With this option checked [✓] the base of the R.C. tie beam will also be updated.

Apply eccentricity:

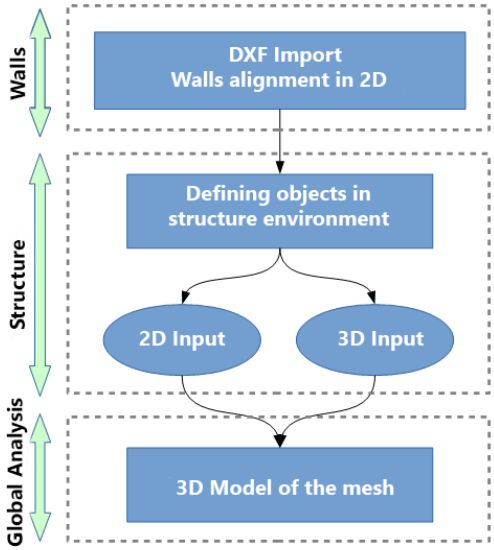
With this option checked [✓], based on the wall's position in the DXF file compared to the axis of the drawn wall, it calculates the eccentricity of the masonry wall.

Apply thickness reduction:

Very often the drawings represent the walls with the "architectural final design" thickness, therefore it is important to be able to reduce the thickness of the wall, for example, the coat's thickness (Ex: for 1cm of coat on each face, I will insert 2cm of thickness reduction in order to automatically calculate the structural thickness).
With this option activated, the text field displays the reduced thickness value.

9.3 Input 3D

As an alternative to the input/edit in 2D mode you can also use the one 3D mode.
The entry of structural objects can occur directly in an axonometric view.

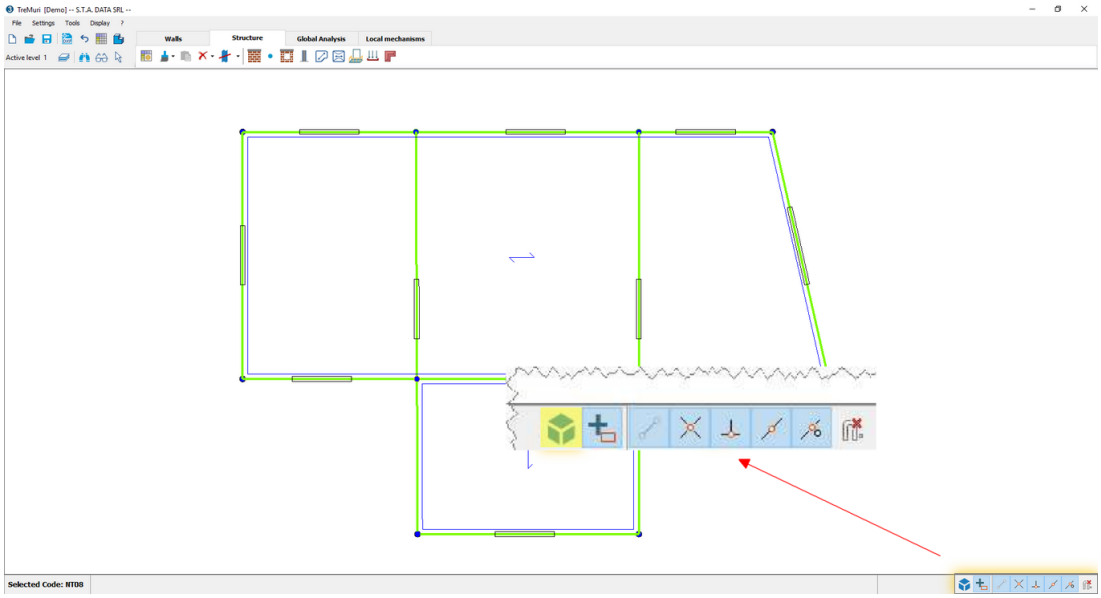


The process goes through the tracking of the walls in plan (the same as in the 2D mode) to define the main alignments in structure.

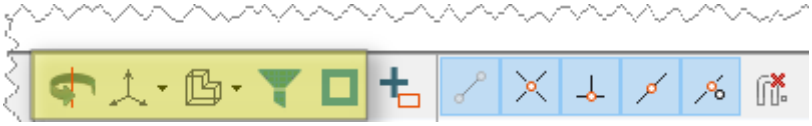
In the structural area is possible to decide whether to continue with the input in plan or to switch to 3D mode.

Whatever the input methods are, in the analysis area will equally be generated a 3D calculation model.

The access to the 3D area is performed by using the specific button on the toolbar at the bottom right of the structure area.

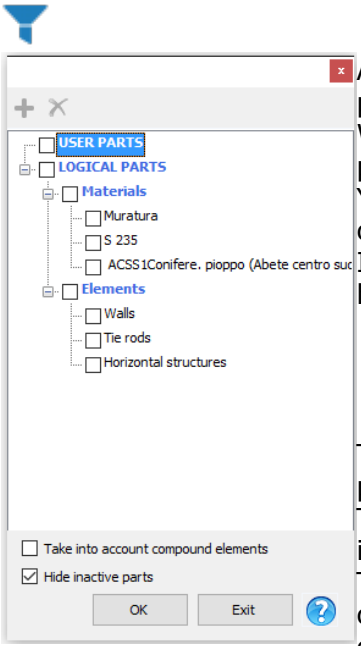


Once entered in the 3D mode the building is shown in an axonometric view and the toolbar will change.



	Rotation in the axonometric view
	Different axonometric views
	Display Modes (solid, Hidden, Wireframe)
	3D Filters
	Back to the 2D input mode

9.3.1 3D Filters-the parts



Allows to transform groups of structural elements in defined parts.
Working with parts makes easier the simultaneous editing phases of multiple elements.
You can display one or more parts, called active parts, at once.
In addition, if the "check" cell is disabled, the commands will be applied or refer only to the activated parts entities.

There are two types of parts: user defined parts and logical parts.
The user defined parts are created by the user selecting the items that belong to the parties.
The logical parts are automatically created by the program, ordering the items in different criteria categories (material, elements).
You can activate an existing part by clicking it on the list.

Create an user defined **new part** (a group of model's entities).
To each new part you must assign a name. Then you define the new part by selecting entities in the active window.

Allows to **delete** user defined **parts** selected from the list.
This command doesn't delete the model's entity.

9.3.2 Objects' input

The commands of the bar structure present in the 3D input mode are also available in 2D mode.

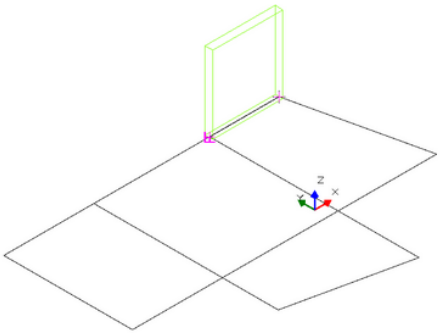
Some commands developed in 2D mode however, are not currently present in the 3D mode; the following table clearly shows the comparison.

	2D	3D	Description
	✓	✓	Materials
	✓	✓	Copy Properties
	✓	✓	Paste Properties
	✓	✓	Delete selected structural elements
	✓	✗	Perceived thickness (read walls' thickness from dxf file)

	✓	✓	Insert wall segments attributes
	✓	✗	Insert node ("divide" operation available in the element properties)
	✓	✓	Opening
	✓	✓	Column
	✓	✓	Slab
	✓	✓	Vault
	✓	✗	Balcony
	✓	✗	Load
	✓	✗	Roof

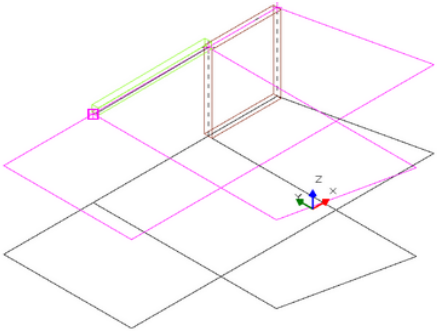
All the structured objects inserted in 2D mode require the position where to be located and then the mechanical characteristics, the 3D mode instead requires to define the geometry first and then the location.

Masonry wall, Panel+R.C. tie beam, Masonry Wall+steel/wooden beam, Masonry Wall+tie rod, R.C. wall, Column



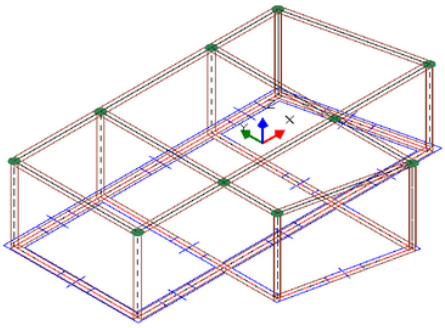
For this group of elements that have an extension in elevation along the level's height, during the insertion phase is possible to perform the input directly in 3D mode using the snaps to the walls tracking reported to the lower elevation of the level.

R.C. Beam, Steel/wooden beam, Tie rod



For this group of elements, the extension of which is essentially limited to the summit elevation of the levels, there are guidelines in magenta that represents the tracking of the walls plant to this elevation in order to facilitate the snap.

Floor, Vault



In case of floors and vaults input, the nodes affected by snap operations through the polygon vertices are shown in green.

All the information regarding the definition of the single elements parameters are shown in a dedicated section of this manual: Definition of Structural Objects

9.3.3 Edit objects

The multiple editing of the elements properties is easy in plan but less performing in elevation. The editing in 3D will simultaneously change the characteristics of the elements at different levels.
We can therefore conclude that the elements editing can be divided into *Basic Editing* and *Advanced Editing.*

9.3.3.1 Basic editing

Select a single element with the right mouse button to show the contextual menu that recalls the editing operations that interest the selected element.

Masonry wall, Masonry wall+R.C. tie beam, Masonry wall+steel/wooden beam, Masonry wall +tie rod, R.C. wall

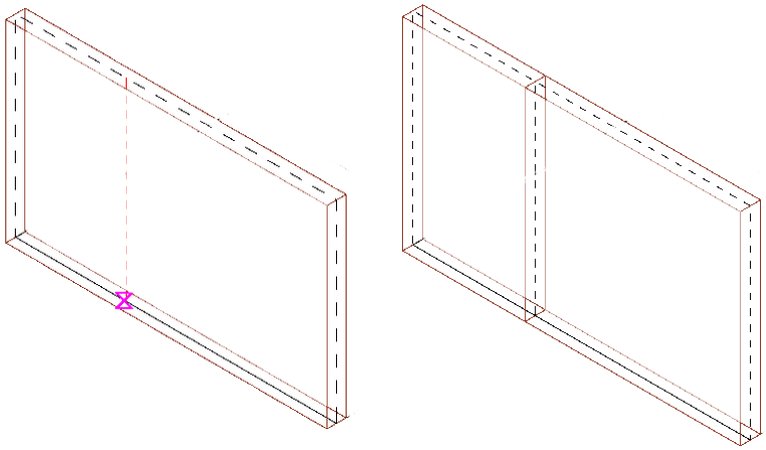
Delete: Delete element

Edit: Shows the element properties window

Join: two contiguous wall segments part of the same wall, separated only by an element node.
It is required the selection of a second element to join the one from which has been executed the command.

Divide: An element is divided into an identified point by selection in graphics, a vertical dashed line shows the point of interruption.

Guidelines insertion *Divided element*



Select wall:
This operation activates the simultaneous selection of all the elements of a single wall.

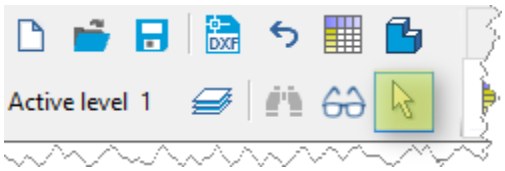
Floor, Vault, Column

Delete: Delete element

Edit: Shows the element properties window

9.3.3.2 Advanced editing

These operations are available only after a multiple selection that can be performed by using the specific button on the command bar.
This command is only available on Structure in 3D mode.



This command is interrupted by pressing the right mouse button.
The context menu of Advanced editing appears immediately after confirming the selection:

Align walls

Assign exposed to wind

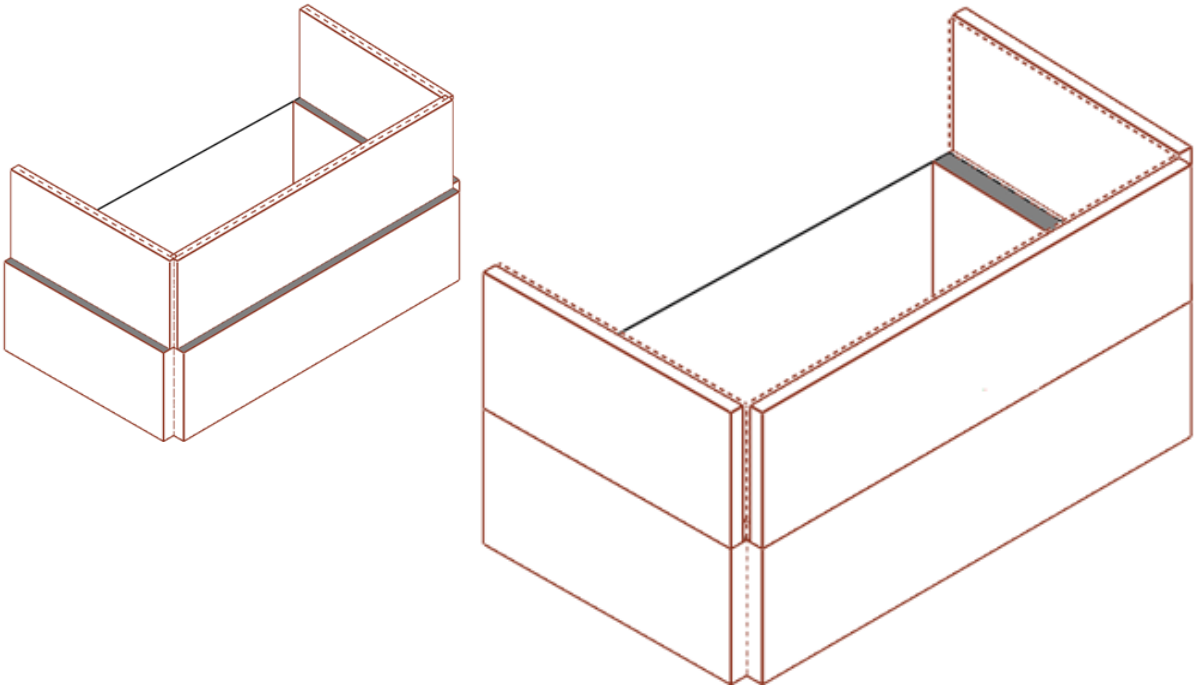
Edit table

Define user's part

Align walls:
It performs the concurrent editing of masonry walls with different thicknesses at different levels in order to align them with the same outer edge.

[Before editing]
Walls aligned with the lower middle plan

[After editing]
Walls aligned with the outer edge

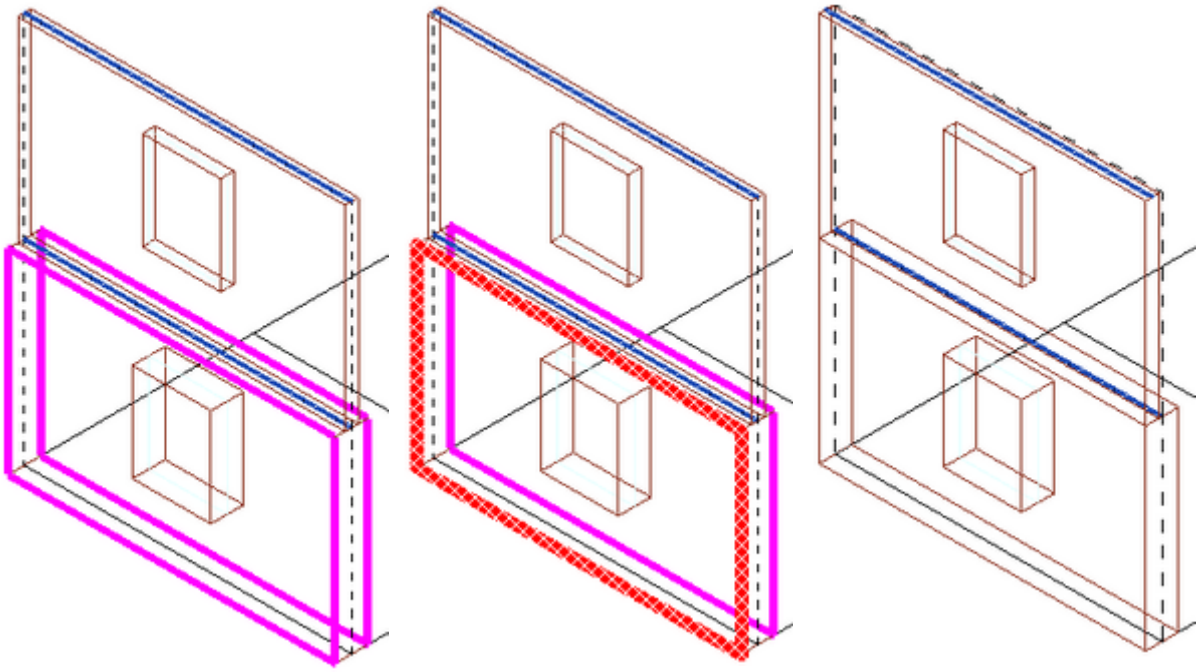


As soon as you call back the command the following operations must be performed in this order:

(1)-Show the reference element to perform the alignment

(2)-Show the alignment face

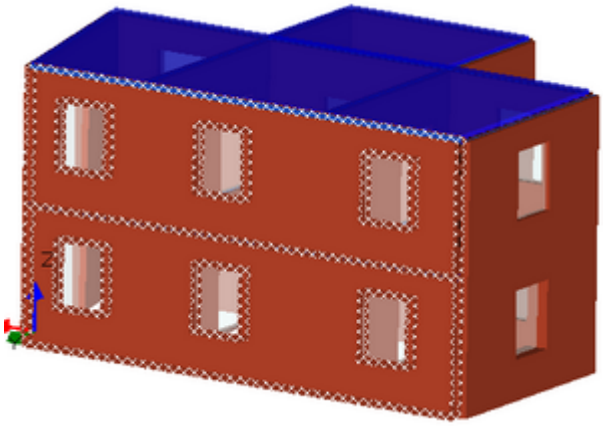
(3)-Final result



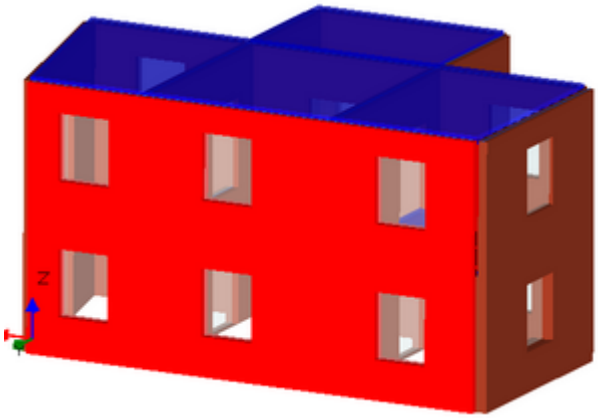
Assign wind exposed

The selected panels are automatically modified to assign the "exposed to the wind" property used in static verifications.

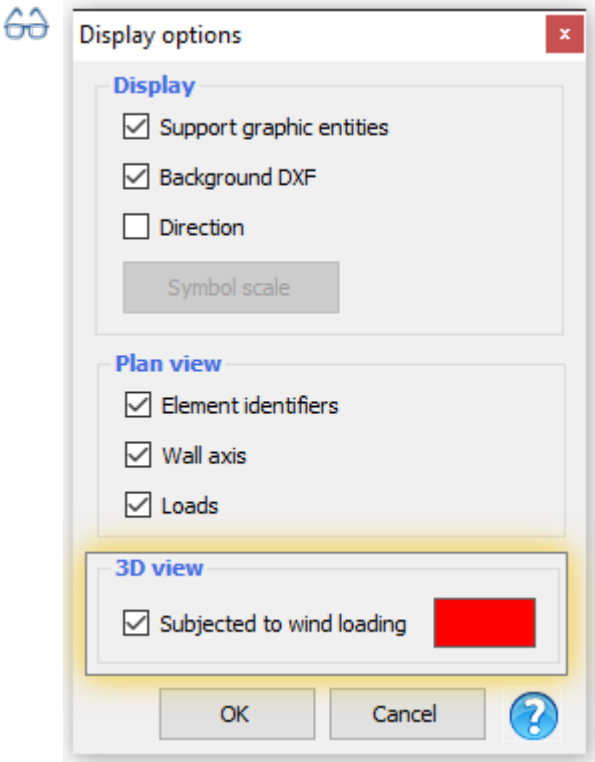
Selected walls



The color is updated

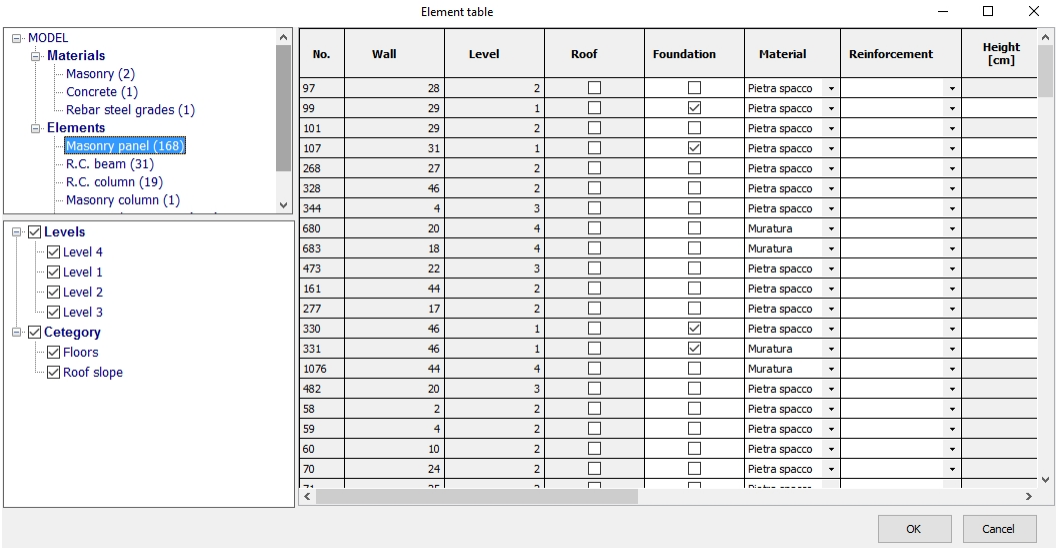


The color of the elements exposed to wind is defined from the mask of the display filters.



Edit table

After a partial selection of the structure you can access the element table that displays only the selected elements.
Possible changes in the table will therefore affect only the selected elements.

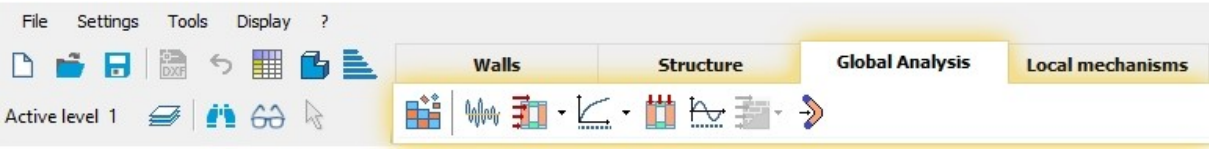


Define user's part

After a partial selection of the structure "part" entities containing the selected elements is automatically created.

In order to create a name it is necessary to define the name that will represent the part.

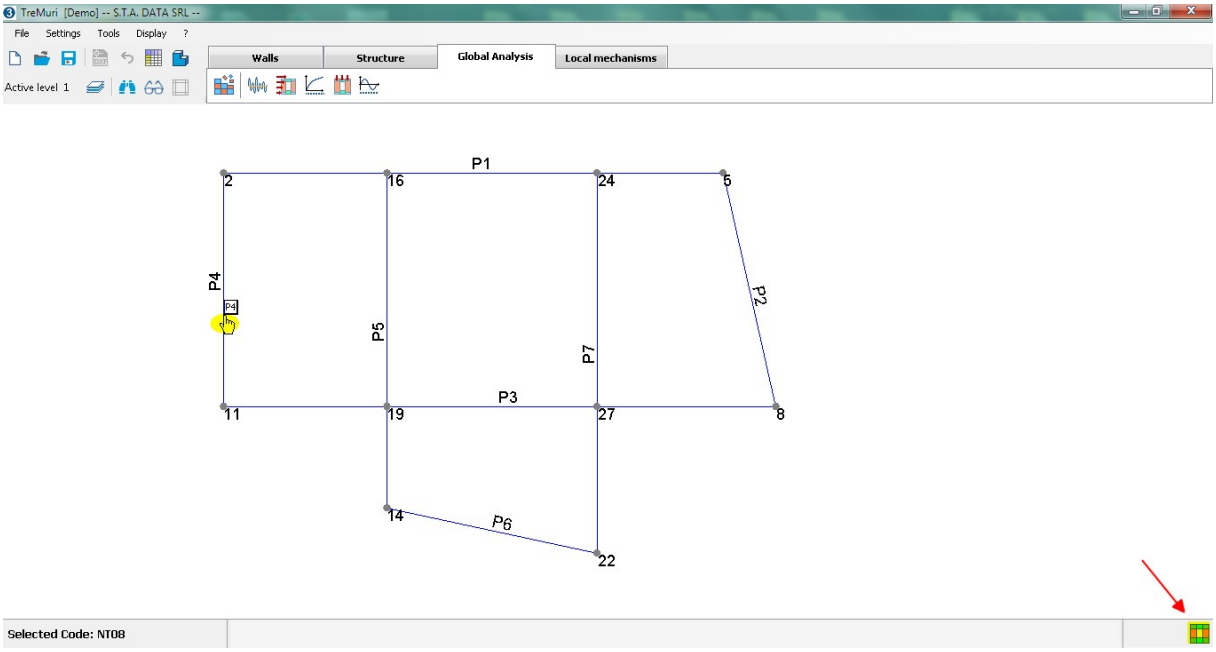
10 Analysis



	Compute model mesh Enter value
	Seismic Action Enter value
	Pushover Calculation Enter value
	Display pushover results Enter value
	Global static verification Enter value
	Modal analysis Enter value
	Sensitivity analysis Enter value
	Bending out of plan

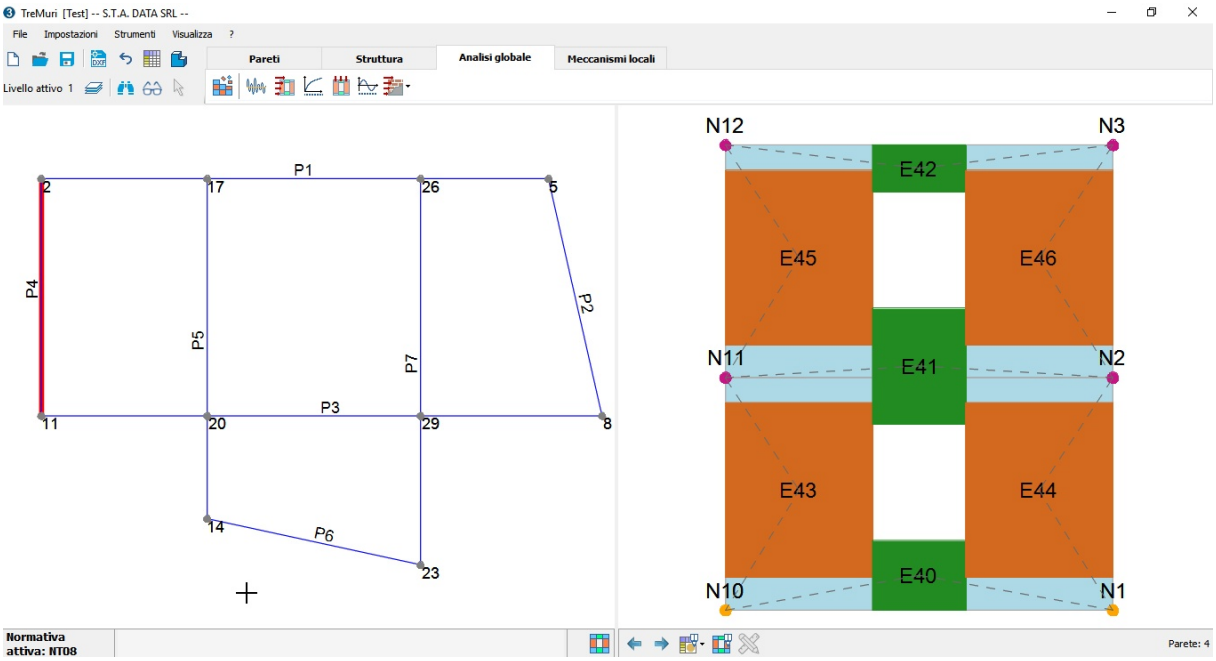
10.1 Mesh definition

From the graphics of the analysis area it's possible to click on a wall in order to display it in the mesh front.

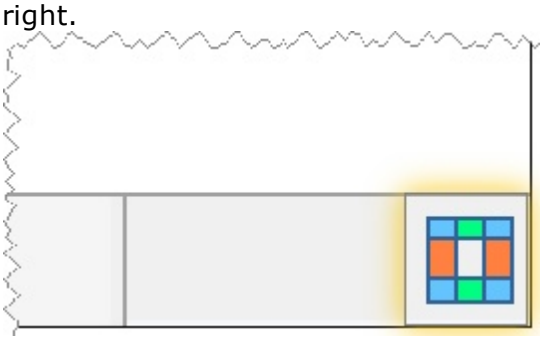


The graphics area is separated in two different areas, the plan on the left and on the

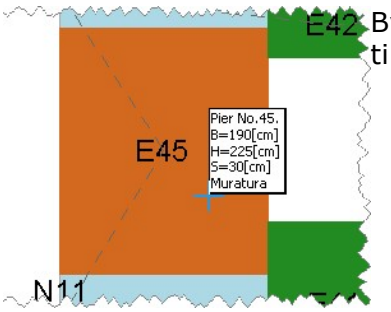
right the front of the mesh of the selected wall.



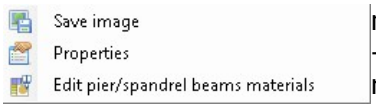
The access to such setting is also possible with the appropriate button at the bottom right.



The cad commands (zoom, pan, etc.) are available in both areas plan (left) and mesh view (right)



By placing the mouse pointer on the mesh elements a tool tip shows the details of the item.



Pressing the right button on an element activates a context menu with the following commands:
- Save image: save the image of the active view in the report's gallery
- Properties: allows to change the properties shown in the tool tip
- Edit materials peer/spandrels beam: Allows direct editing of the element's material to perform an improvement step.

At the bottom left in the front mesh appears a command bar.

	Previous wall
	Next wall
	Edit Material: If the designer has to provide some adaptation on the structure he/she may act through localized interventions improving the mechanical properties of the individual wall elements. (Details...)
	Edit Mesh: The procedure for automatic mesh generation (calculates mesh) is able to capture almost all of the case studies in the more usual design practice. For the limited cases where this is not possible, the user can enter in the appropriate setting. Currently the command is not available.

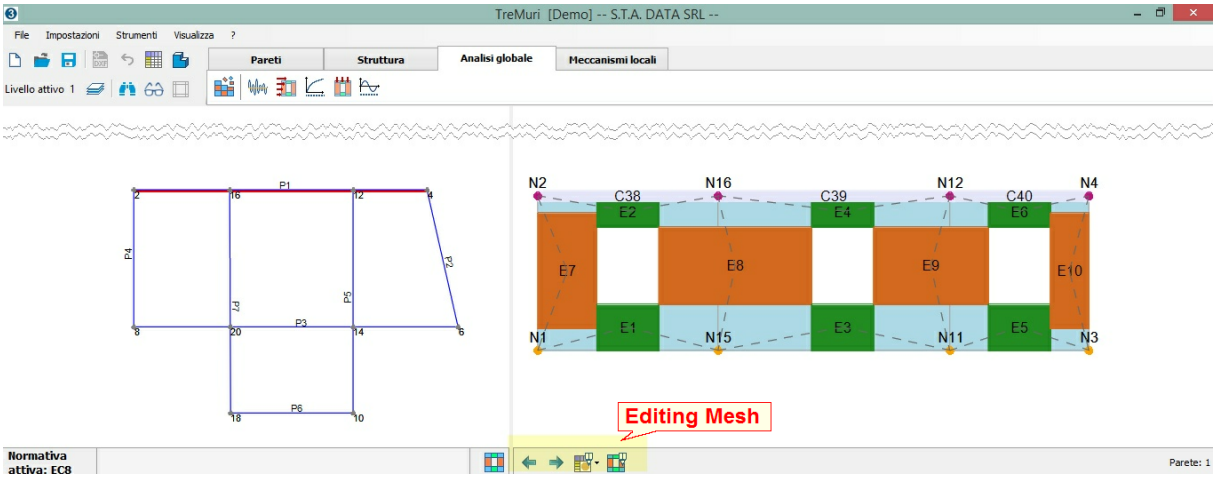
Compute model mesh: calculates the mesh model (if not already performed at 'beginning of the analysis procedure) or to recalculate in case of modifications.

10.2 Mesh editing

Professional, Small Business, Small structures versions

The environment mesh editing allows to modify the characteristics of the mesh generated automatically by a special procedure

In mesh editing you can change the characteristics of the generated structure according to the design requirements.
The access in the editing mesh area is achieved through the appropriate button on the command bar.



By clicking on the appropriate button you can enter the editing area.

Editing functions are divided in two groups: Editing Elements, Editing Nodes.

All edit functions, except "Change node type", have "Cancel" command.
While changing wall or closing the mesh editing environment, you will be asked to confirm the changes. If the answer is affirmative, the mesh wall is changed according to the instructions with an automatic recalculation of the rigid nodes. The negative answer cancels the current changes.

The button used to access the mesh editing area appears pressed during editing operations, to stop editing and return to the traditional working area you need to click on the same button.

Please note that the 3Muri program operates in a temporary session, so the changes will not be finalized until you save the template. Saving takes place by the user's request or automatically before a calculation.

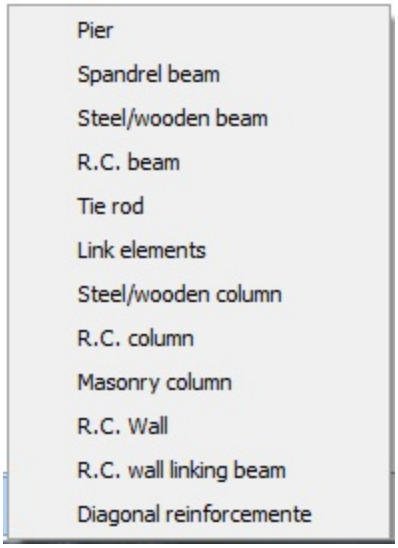
Smart Version

This *Smart* function is not present in the version.

10.2.1 Editing Elements

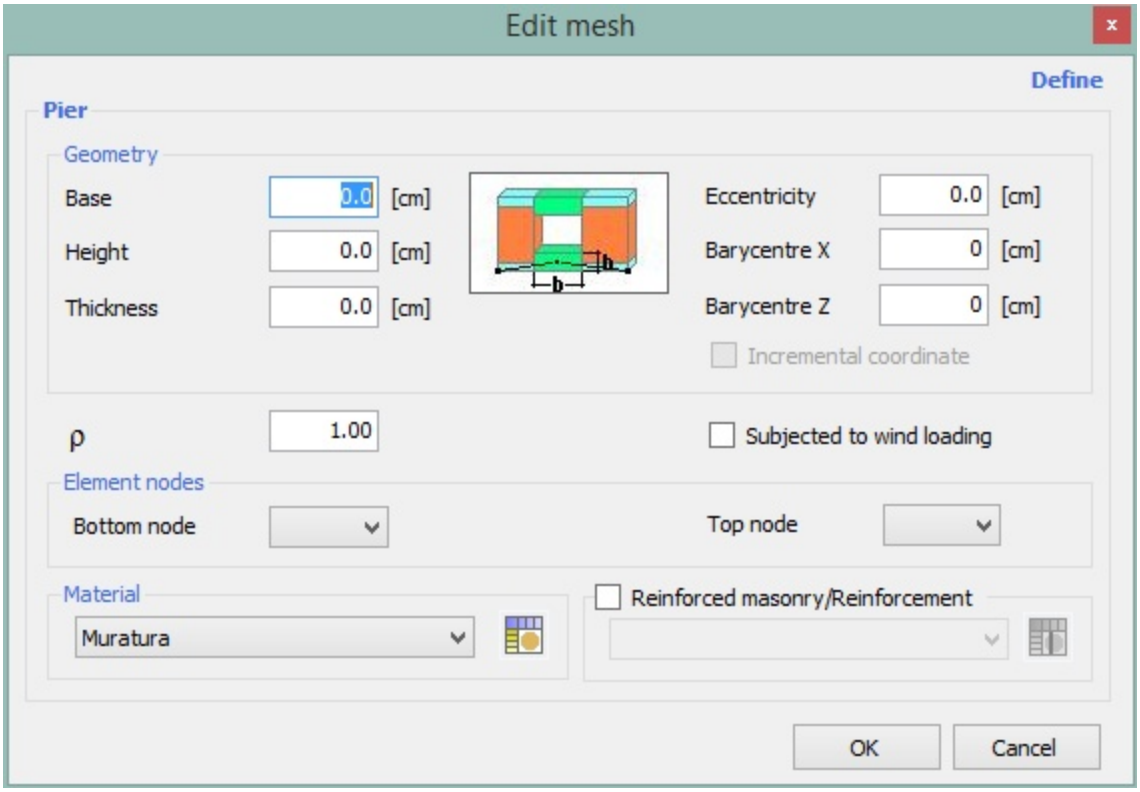
Professional, Small Business, Small Structures versions

The editing functions act on the entities in the associated menu on each button of the command bar.



Link elements: Rigid link or truss. These items are included in automatic mesh algorithm to create a equivalent frame according to solver's rules.
Rigid beam: Are included in "blind" pier definition (a "blind" pier is due to a panel without opening)
Truss: Are included in all those situations where it was generated a spandrel but, for example due to openings at full height, it is not possible, its generation; so the connection of nodes to the equivalent frame is by a truss.

 Add element:
After choosing the item to be add, a window with all item data is loaded.
Suppose for example the case of a pier:



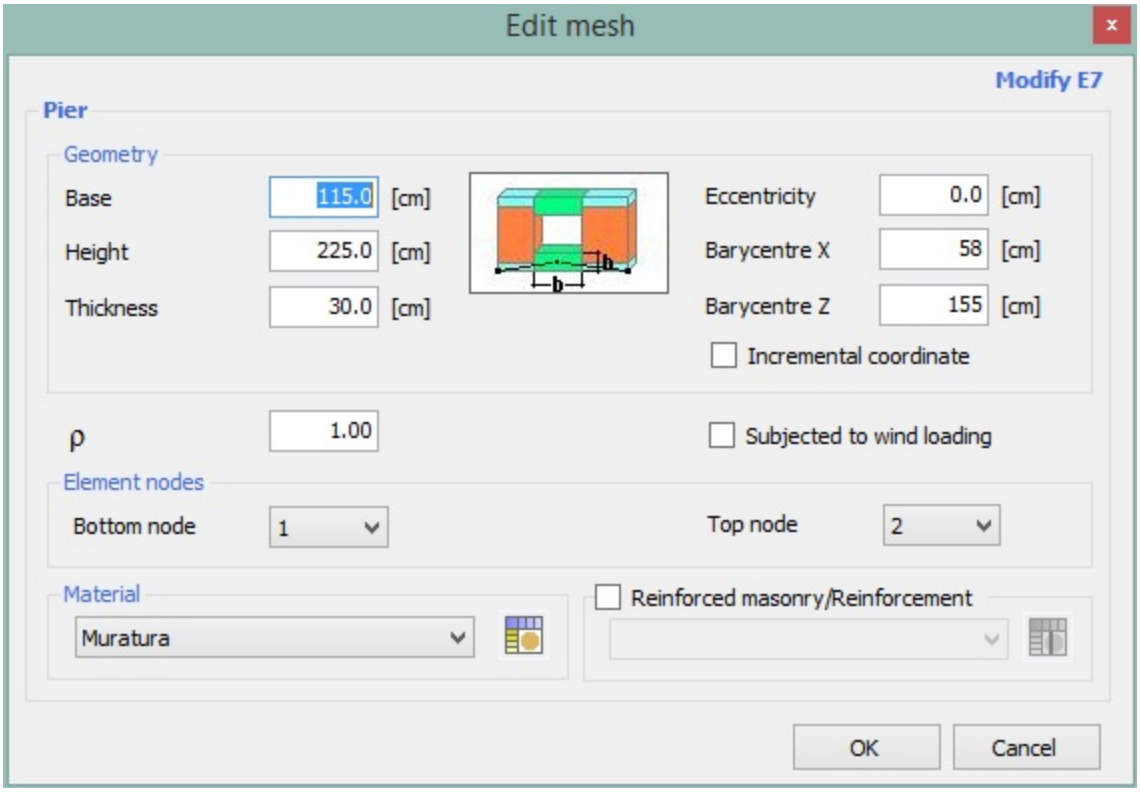
Element nodes: Allow user to modify element nodes of selected item; in this way the user can modify equivalent frame geometry.
Move barycenter: allow the use to move selected macroelement by insert component for translate vector (in local wall system: x, z).
Reduction factor for slenderness(ρ):factor used for static verifications in accordance with the code. If you do not perform these checks this field is not significant

Input boxes of the form must be completed in all parts.

Delete item: Select one or more items to delete.

Modify item:
It is possible to select one or more items to modify. A form with all modifiable item data is loaded. In case of single item selection, in the form are available all item data. In case of multiple items selection, in the form are available some item data.

Suppose for example the case of a pier:



The case of a multiple selection:

✓ Only new data inserted in input boxes are modified. To keep the original data of selected items, leave the input boxes blank. For example, to move up 10 cm all selected piers, it is necessary:

- checking the box "Incremental coordinate"
- insert in "Delta Z" input box the value 10
- click on "ok" button.

All the characteristics of the selected piers remain unchanged, except barycenter coordinated and therefore piers are shifted by 10 cm upwards

✓ It is impossible to modify elements nodes

Smart Version

This function is non available for the *Smart* version

10.2.2 Editing Nodes

Professional, Small Business, Small structures versions

Editing nodes functions are available for 2D node as well as 3D.

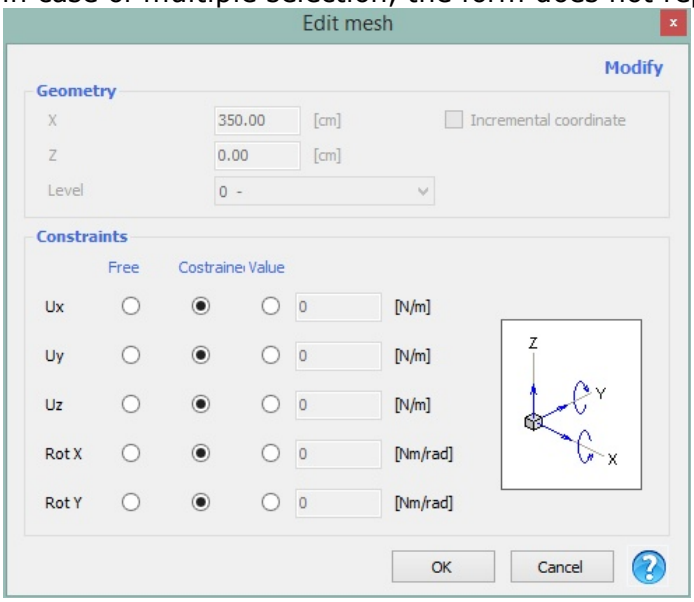
 Delete nodes:

Delete all selected node if they are not related to items on the current wall or on other walls in case of 3D nodes.

 Modify node:

It is possible to select one or more nodes to be modify. A form that includes all the data editable is loaded. In case of individual selection, the form contains all data node;

in case of multiple selection, the form does not report any data of selected nodes.



From this window you can define the boundary conditions.

Modify node type (3D->2D)(2D->3D):

Allows modifying node type. A 2D node can be modified as 3D node and vice versus. This function is available only for one node at a time.

Passage 3D -> 2D

The node that is a 3D type node before the modification is transformed into a 2D node (Nodal derating).

It is checked if the selected node belongs to more than one wall.

- ✓ If the node belongs to a *single wall* the type that it gets derated (this functions is prevented only if the node is connected to a column or a R.C. wall)
- ✓ If it belongs to *two walls*, including the current one, the node type is derated on both walls, this is useful when you want to delete a 3D node on a wall as it no longer makes sense to exist as a result of the changes made by the user. This operation means that the 3D node becomes a double 2D node (one node for each walls) with the same spatial coordinates; this allows the user to decide what to do of each node, move it or delete it. Remember that the program solver can not accept that two 2D nodes coexist with the same coordinates (coinciding nodes).
- ✓ If it belongs to *more than two walls*, including current one, the operation is prevented.

Passage 2D -> 3D

The node that is a 2D type node before the modification is transformed into a 3D node (Nodal reclassifications).

In this phase the program prompts the user if it exists any incidental wall. The program checks if the input wall intersect the current wall and if that happens, it will search for the incidental wall a node that geometrically coincides with the node that has to be reclassificated; even this node will be transformed in 3D node type.

N.B.: *It is impossible to apply undo on these functions because the UNDO is valid only for current wall. These functions involved more than current wall so they are not canceled in "edit mesh".*

Smart Version

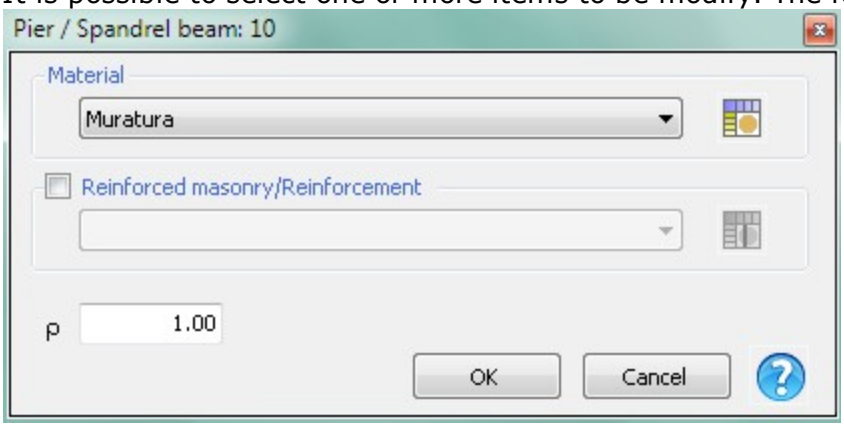
This function is non available for the *Smart* version

10.3 Editing Materials

This function allows you to edit only the materials related to pier and spandrel without intervening on the geometry of the mesh and then on the characteristics of the equivalent frame.

Select the item to modify.

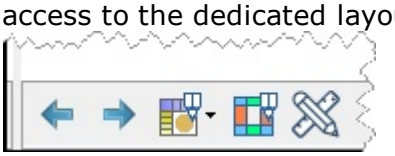
It is possible to select one or more items to be modify. The following form is loaded:



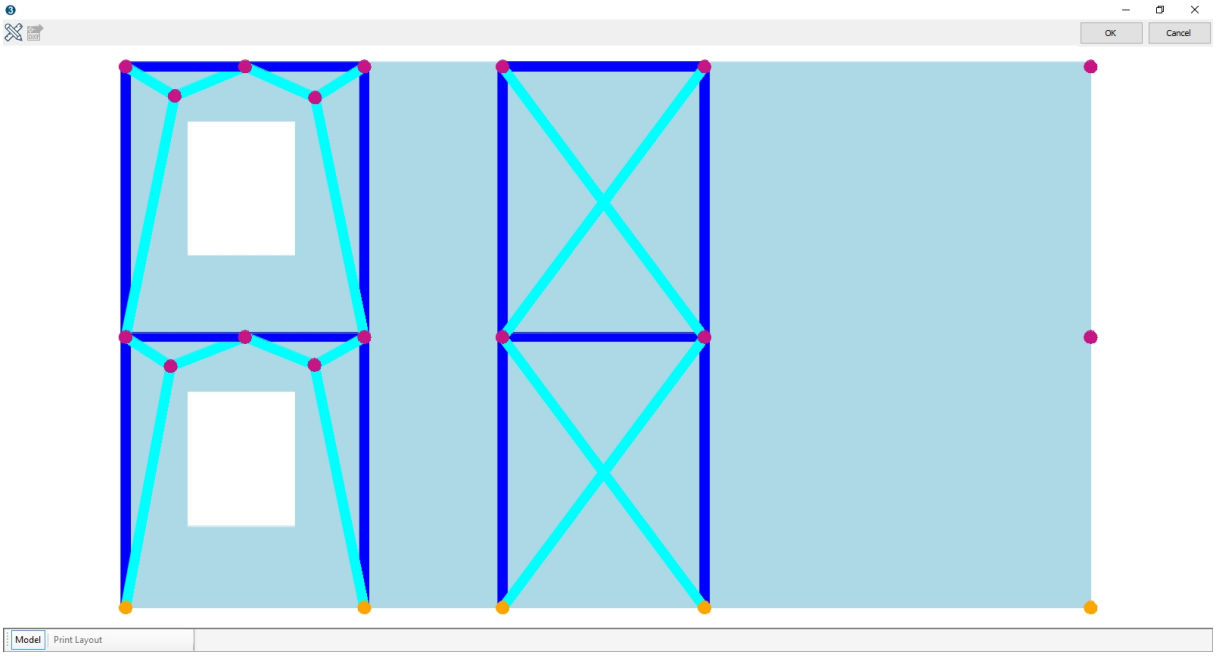
In case of single selection, the form contains all data item; in case of multiple selection, the form does not report any data of selected items. If you want to modify only one of two items of the form, simply leave unchanged the other one.

10.4 Steel elements graphic layout

Pressing the appropriate command in the bottom right of the mesh command bar, you access to the dedicated layout environment.

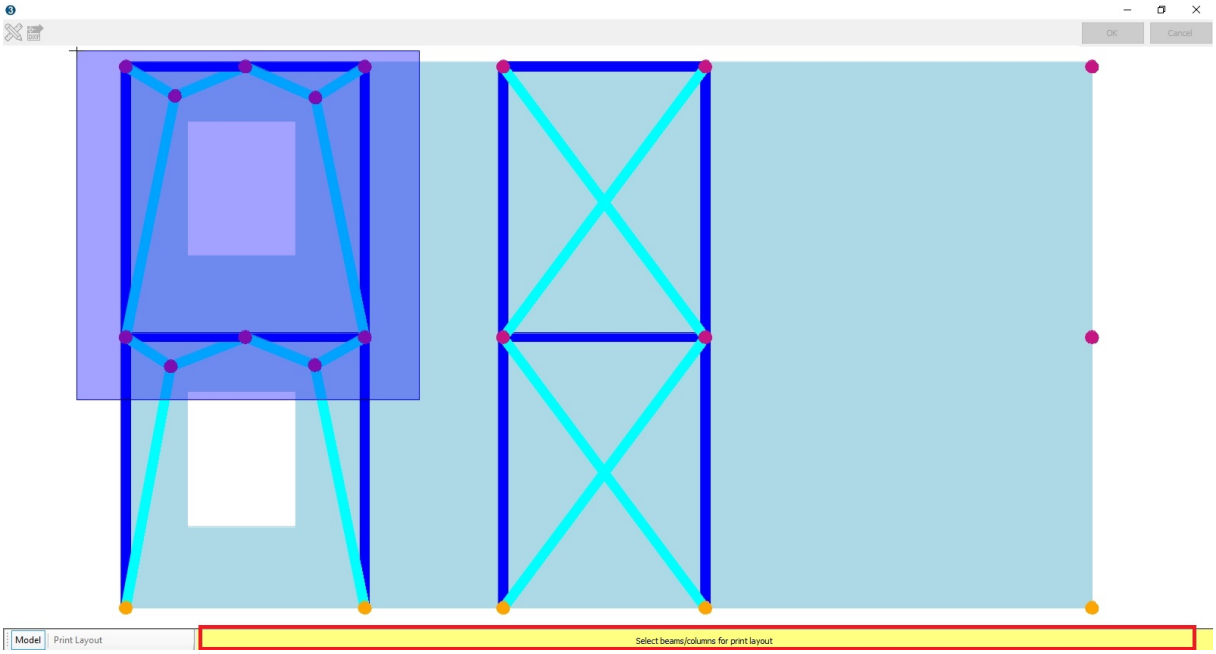


This environment shows the reinforcing elements on the façade of the wall.

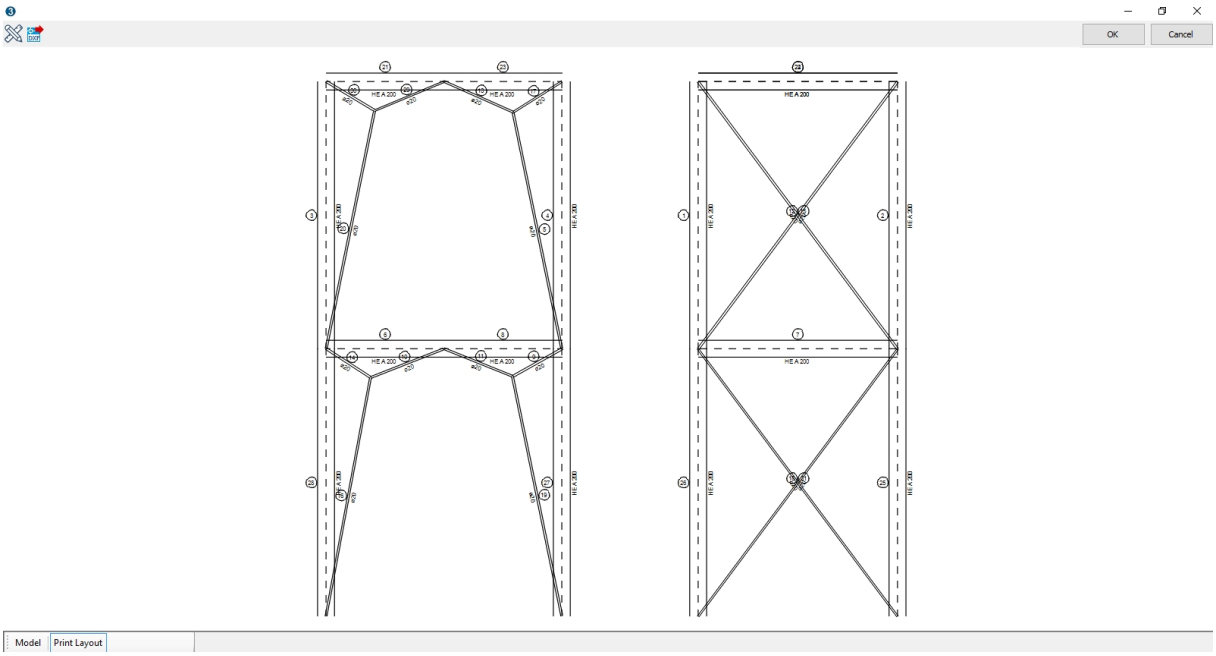



Create graphics table:

Activating the command prompts you to select the steel elements of which you want to produce the graphic tables.



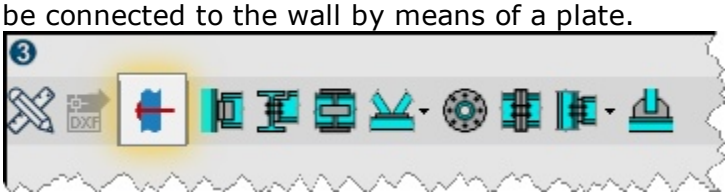
After the selection the viewing immediately goes into *Print Layout* mode of the framed structures.



The export button allows you to produce the tables in DXF format.

Calculation of the tie rod-plate link

This environment is used to define the "extreme tie rod" nodes which are required to be connected to the wall by means of a plate.



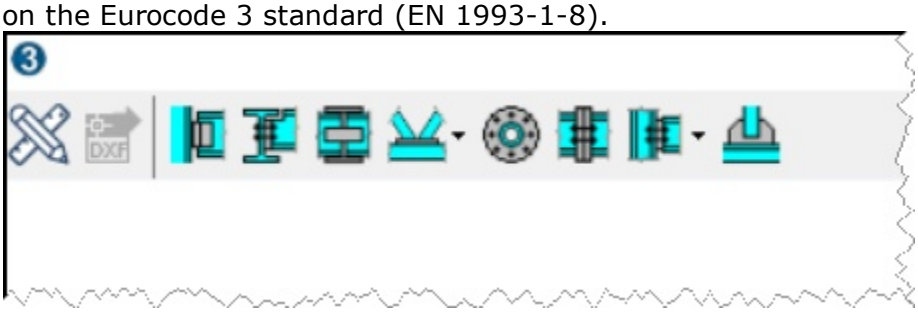
Selecting this command is required to indicate the nodes to be calculated.
The design/verification of the tie rod-plate connections are carried out through the appropriate command in the Utility menu that combines the calculation of all the tie rods, both those from a global model and from local mechanisms.

Calculation of Steel Elements [Steel Connection Module]

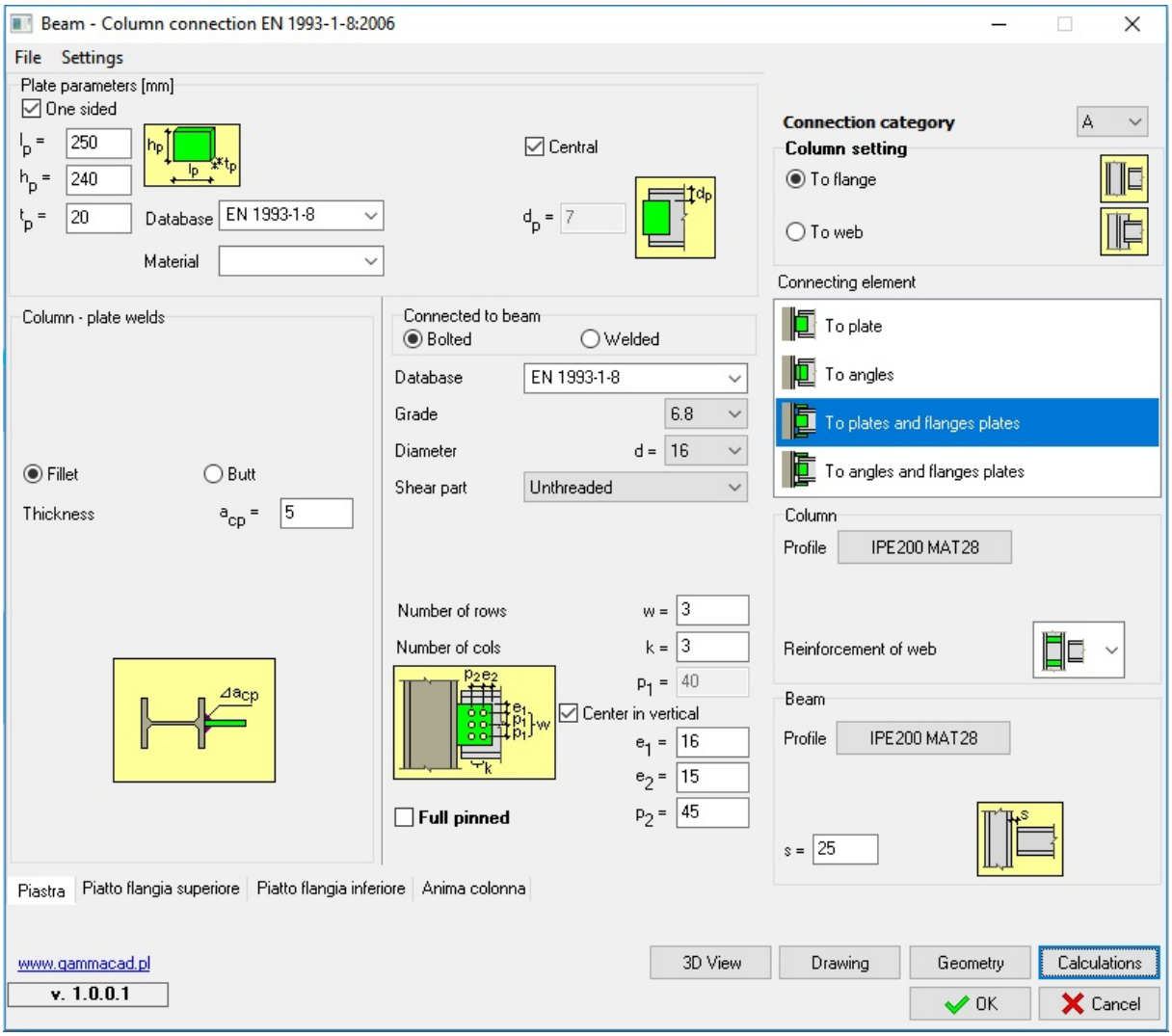
[OPTIONAL]

Allows you to select rods that converge to a node of the frame structure and automatically run the SteelConnection module.

This module is dedicated to *dimensioning and verification of steel connections* based on the Eurocode 3 standard (EN 1993-1-8).



The following image shows one of the SteelConnection calculation modules

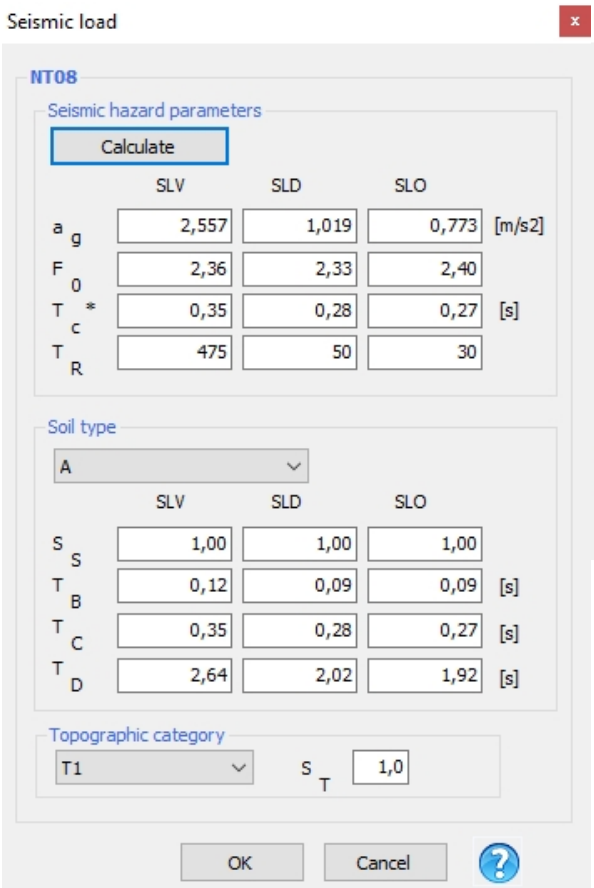


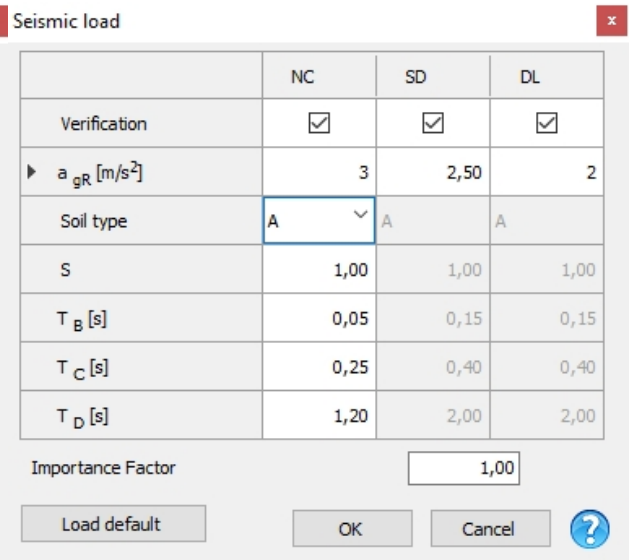
10.5 Pushover Seismic Analysis

10.5.1 Selection of the seismic conditions

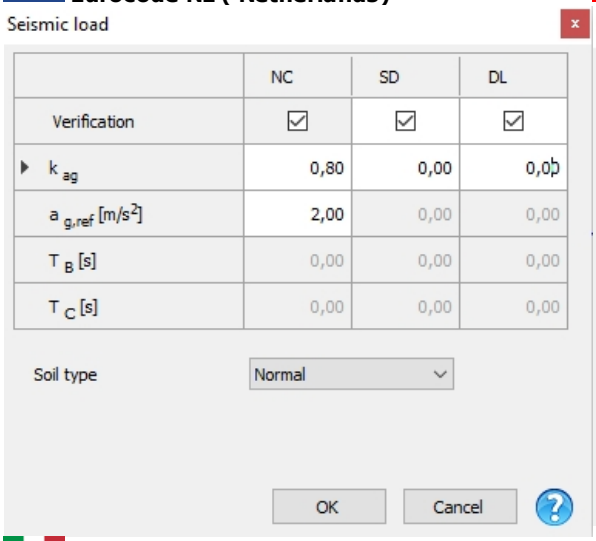
Seismic load: allows to set the earthquake zone and the class of the soil according to the indications of the code. For more details that indicated in the following windows, it refers to as described in the corresponding code.

NTC08 **Eurocode**

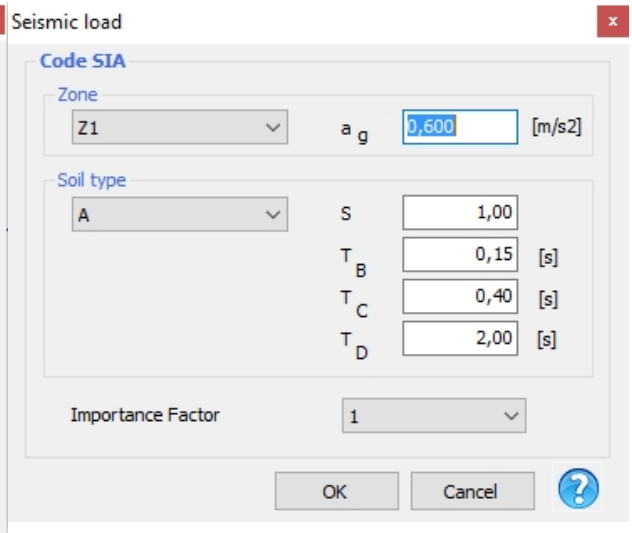




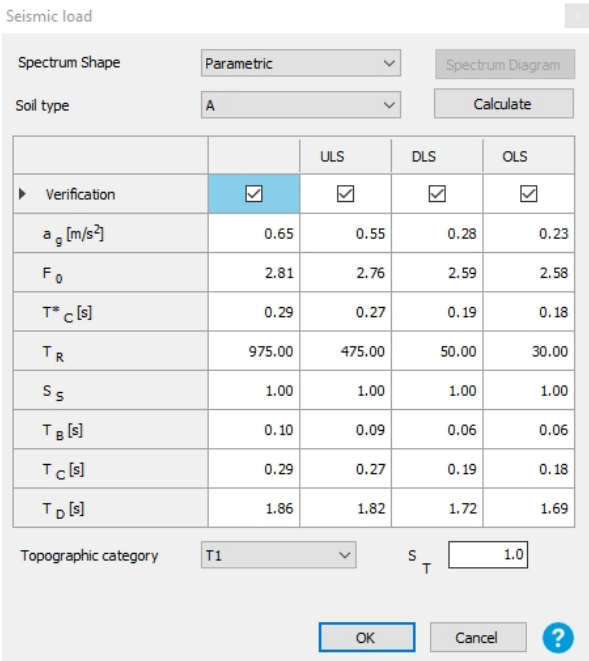
Eurocode NL (Netherlands)



SIA



NTC17



NTC17

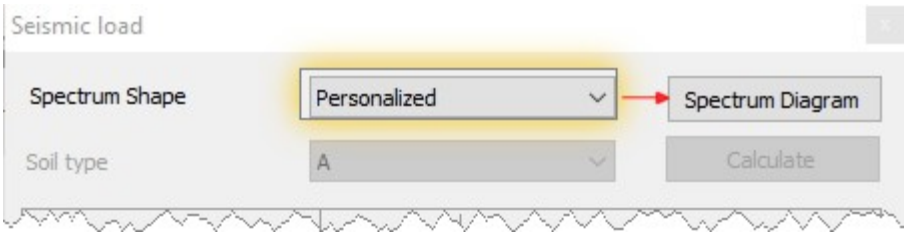
It is important to emphasize that, unlike previous standard, there is a possibility to select the limit states on which to conduct the verification by using the check box.

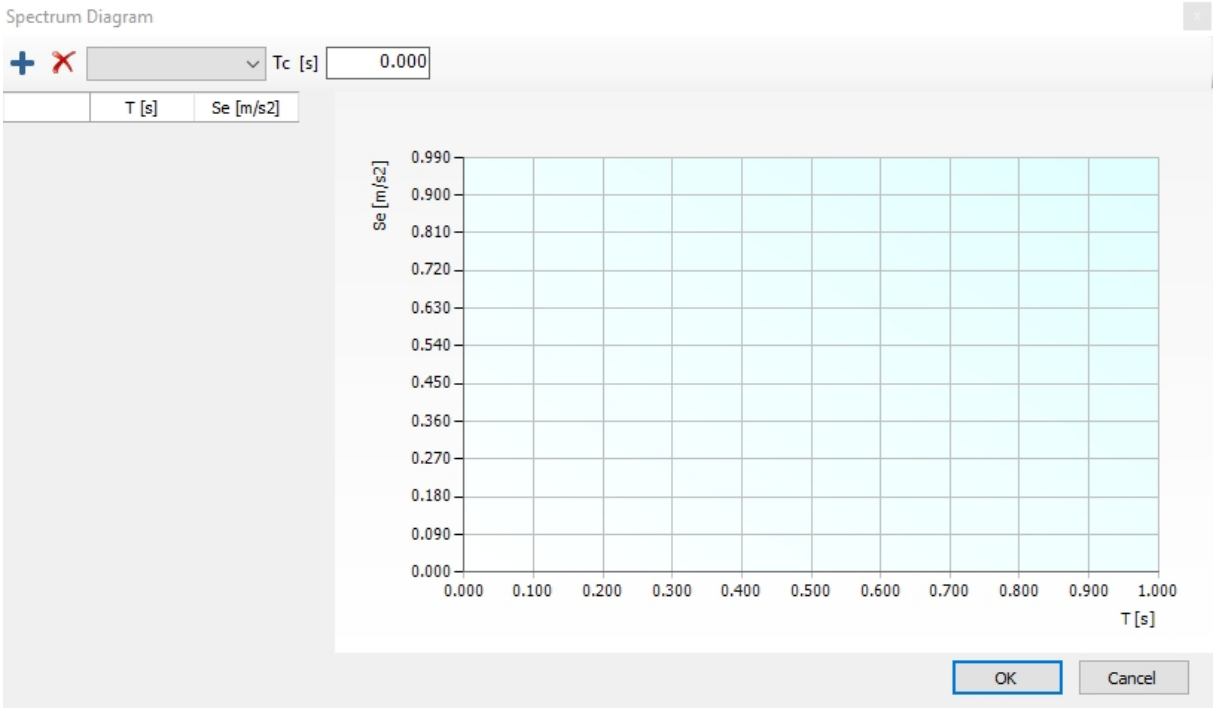
Eurocode

By using the check box is possible to select of which limit state will conduct the verification.

Definition of the spectrum by points

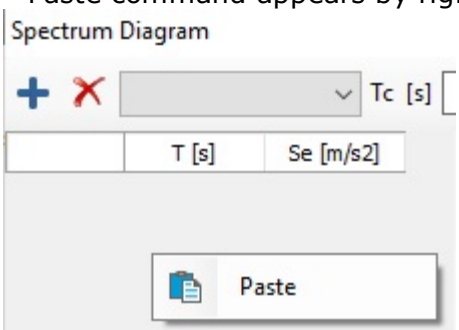
Selecting "Personalized" Spectrum Shape it activates the [Spectrum Diagram] command

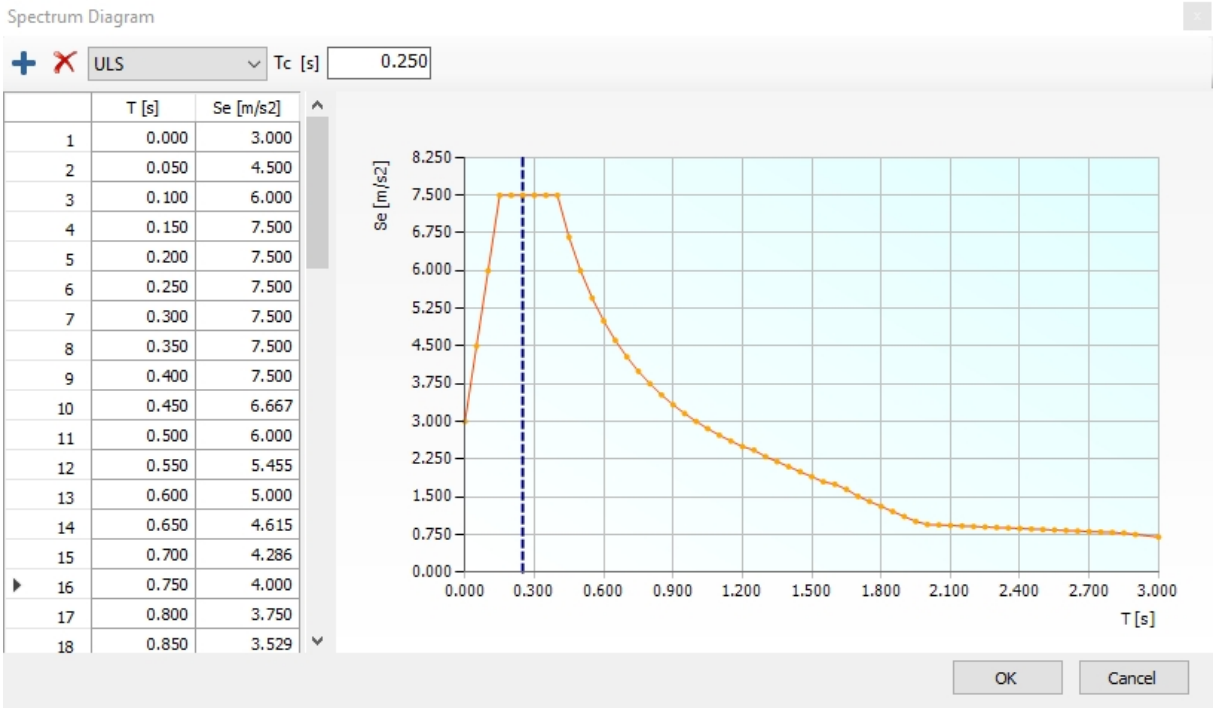




There are two different ways to impute the shape of the curve:

- Using the [+] button to create the table one row at a time
- Through the paste command for data coming from external sources (eg Excel). The Paste command appears by right-clicking in the table area.





The value of Tc (corresponding period at the beginning of the constant speed section) must be manually defined by the user using the text field above or by clicking directly on a point in the diagram.
The drop-down window in the bar contains the list of limit states, select each limit state to define individually the spectrum points.

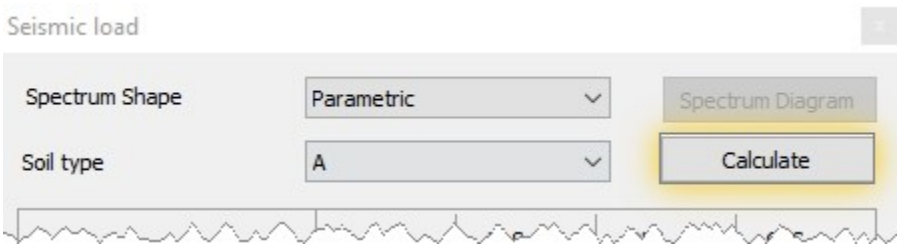
If the point definition is used, the results window will display the vulnerability index as the multiplying factor of the spectral form defined herein, such as to identify the ultimate condition of the required limit state.

NTC08

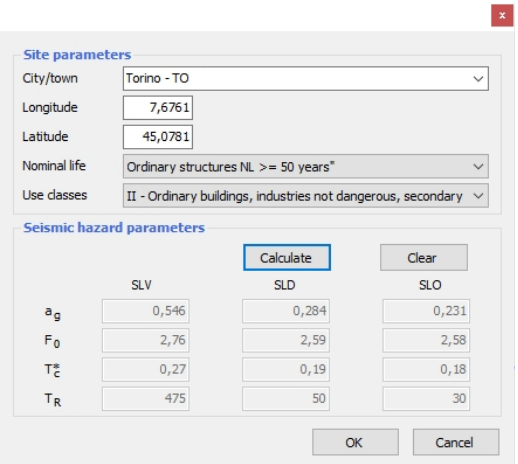
In "Norme Tecniche del Gennaio 2008", the seismic spectrums depend on the geographical coordinates of the site, instead of the earthquake zone (as in previous rules). In the window "seismic action" the "parameters of seismic hazard" are defined by the button "Calculate".

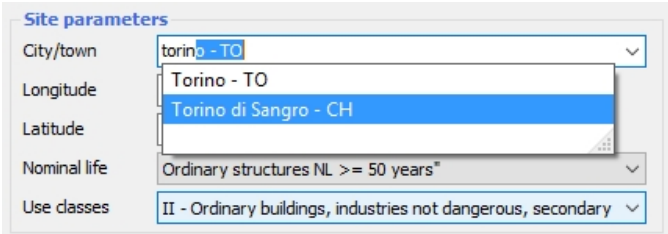
NTC08

In the Technical Standards of January 2008, seismic spectra no longer depend on the seismic zone (as in the previous norms) but from the geographic coordinates of the site.
In the window "*Seismic action*" the "*parameters of seismic hazard*" are defined by the button "Calculate".



Choosing this button, the following window is shown:

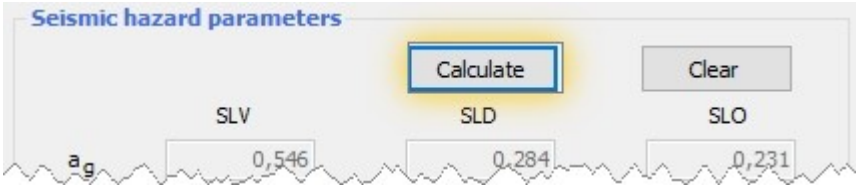




You can select directly the municipality using the internal database or insert the latitude and longitude of the site.

World Geodetic System 84 (WGS 84)

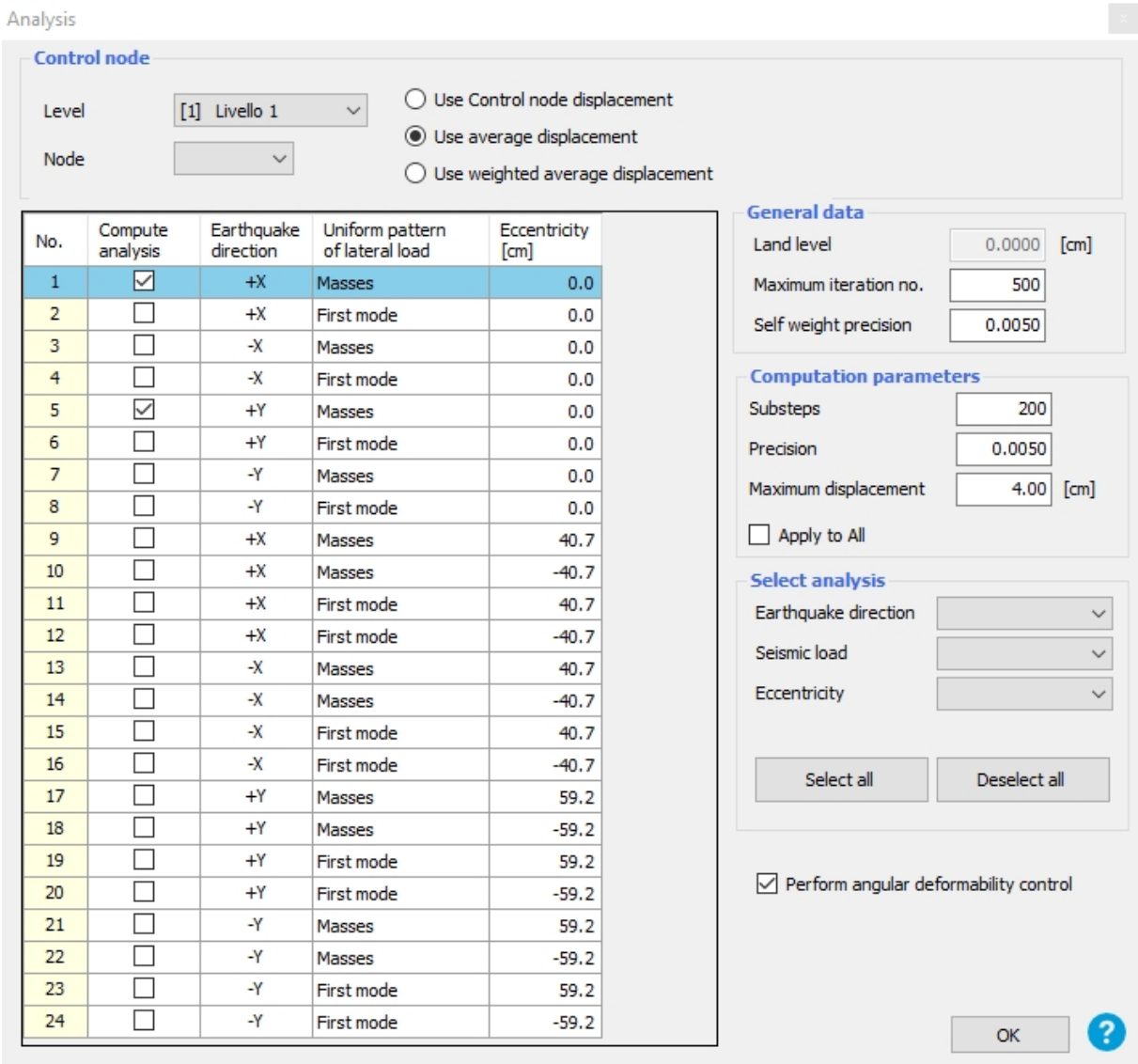
Calculate the necessary values to define the shape of the spectrum for each limit state resuming them in the lower part of the window.



10.5.2 Computation Settings

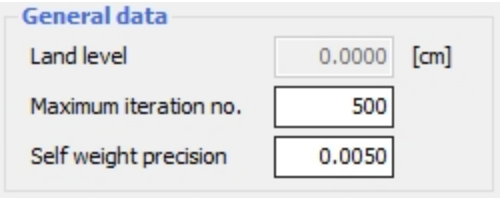
To access to the global analysis of the structure, select "Global Calculation" from the drop-down list.
In this phase, the computation is performed using the selected code.

Many of the computation parameters defined in the "Settings" window are already set so as to work with most examinable structures. Others are automatically computed by the program based on the geometry of the model. The earthquake direction to be considered and the choice of the control node are chosen by the designer based on the indications found in the code.
The choice of the distribution of seismic forces (proportional to the masses or the first mode of vibration) is up to the designer.
The bearing capacity curve can be drawn monitoring displacement, in place of the control node of the average of the project, by selecting the appropriate text box.

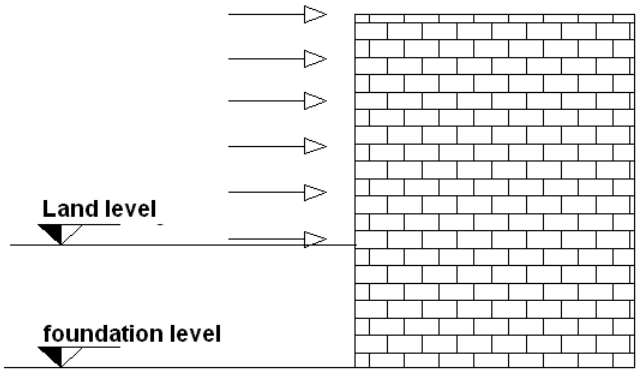


No.	Compute analysis	Earthquake direction	Uniform pattern of lateral load	Eccentricity [cm]
1	☑	+X	Masses	0.0
2	☐	+X	First mode	0.0
3	☐	-X	Masses	0.0
4	☐	-X	First mode	0.0
5	☑	+Y	Masses	0.0
6	☐	+Y	First mode	0.0
7	☐	-Y	Masses	0.0
8	☐	-Y	First mode	0.0
9	☐	+X	Masses	40.7
10	☐	+X	Masses	-40.7
11	☐	+X	First mode	40.7
12	☐	+X	First mode	-40.7
13	☐	-X	Masses	40.7
14	☐	-X	Masses	-40.7
15	☐	-X	First mode	40.7
16	☐	-X	First mode	-40.7
17	☐	+Y	Masses	59.2
18	☐	+Y	Masses	-59.2
19	☐	+Y	First mode	59.2
20	☐	+Y	First mode	-59.2
21	☐	-Y	Masses	59.2
22	☐	-Y	Masses	-59.2
23	☐	-Y	First mode	59.2
24	☐	-Y	First mode	-59.2

General data:



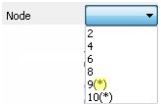
Land level: represents the elevation of the land level. The program assigns the lowest point of the structure elevation 0. The possibility of inserting this elevation allows the user to define the point where the seismic load initiates. The value of this elevation must be between the foundation elevation (generally zero) and the maximum elevation of all the constrained nodes.



Maximum iteration no.: represents the maximum number of analysis steps that the solver must perform before stopping the computation if no convergences are found.
Accuracy pbw: represents the degree of accuracy attained by the calculation of the first step of the calculation (where it is present only pbw)

Control node options: definition of a control node is obligatory for computation. It is recommended that the node is chosen in correspondence with the highest level of the structure.

- *Control node displacement*: the capacity curve is drawn only with the control node displacement.
- *Average displacement*: the capacity curve is drawn based on the average displacement of all nodes of the level at which the control node belongs
- *Weighted average displacement*: the capacity curve is drawn based on the average weighted displacement (weighed on the masses) of all nodes of the level at which the control node belongs. If the floor was infinitely rigid, this displacement would equate to the displacement of the center of gravity.



The check node can be chosen from the menu.
The **(*)** symbol indicates a top node of the roof.
Very often, it is preferred not to perform the calculation with a roof top type node because the deformation behavior at a ridge may not be representative of the actual behavior of the structure deformation.

This window performs multiple analysis in distinct cascades, for direction, orientation, type of seismic load, and eccentricity.

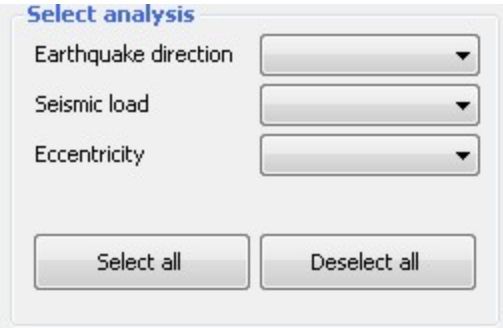
Direction: indicates the earthquake direction.

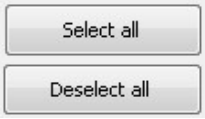
Orientation: positive if in concordance with the positive direction of the axis examined.

Seismic load: Proportional to the mass or the first node to vibrate.

Eccentricity: Accidental eccentricity of the center of mass with respect to the rigidity center computed automatically according to the code.

Using the associated space, multiple analysis can be performing by activating the selection filters.

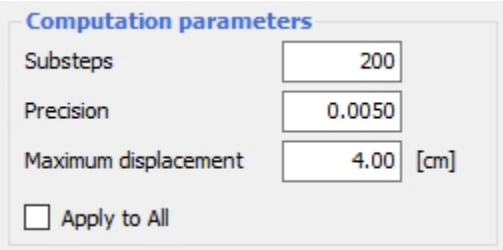




Enables the calculation of all the currently disabled analyzes

Disables the calculation of all the currently active analyzes

The parameters of each analysis can be set through the appropriate area.



Substeps: represents the number of displacement steps computed by the solver for the seismic load pattern.

Tolerance: represents the degree of tolerance reached by the non-linear computation.

Maximum displacement: represents the maximum displacement that the structure's control node can withstand.

Apply to All: With the active tick the values are applied to all analyzes; if the check is disabled they are only applied to the selected analysis.

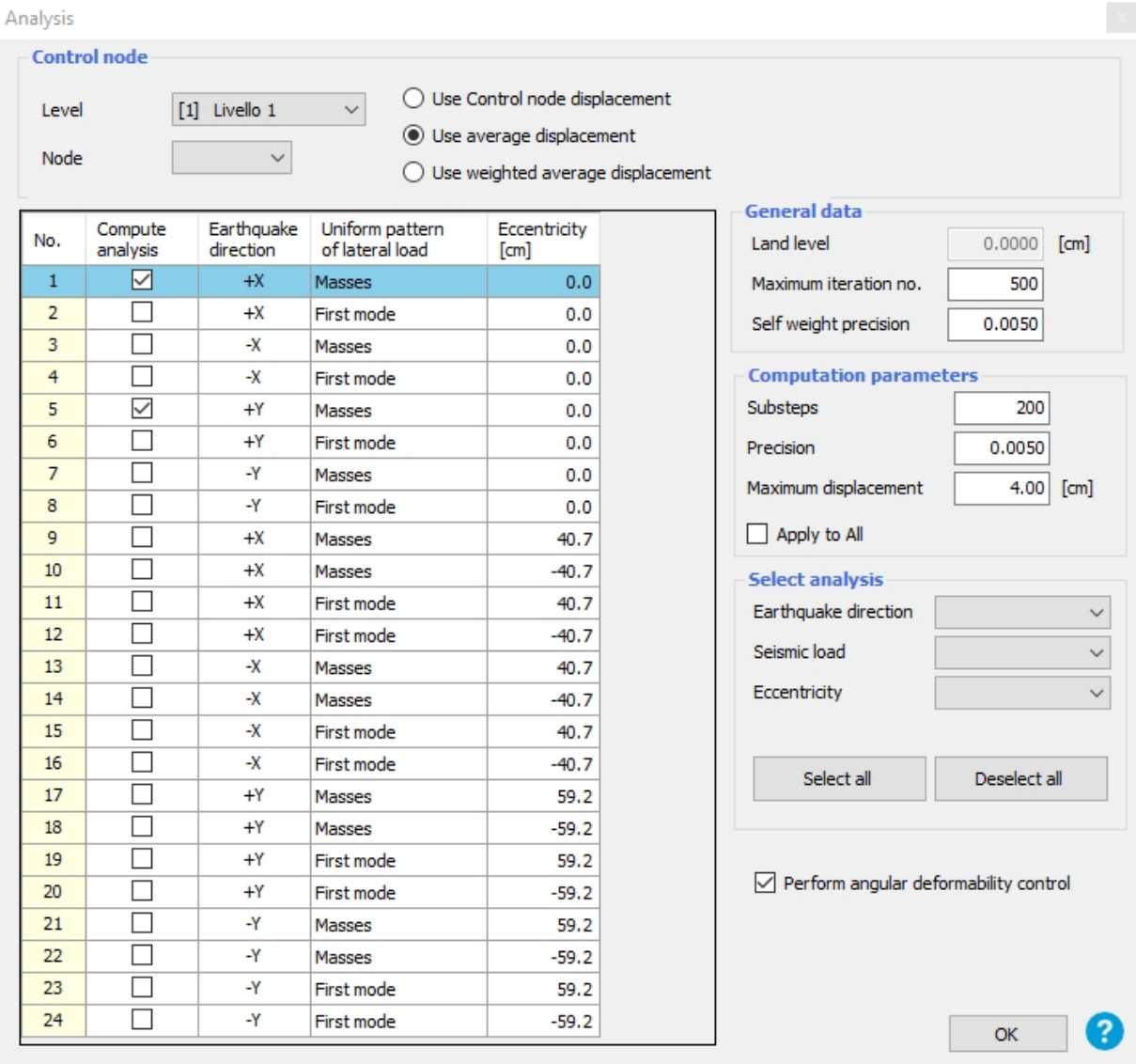
10.5.2.1 Global analysis


To access to the global analysis of the structure, select "Global Calculation" from the drop-down list.
In this phase, the computation is performed using the selected code.

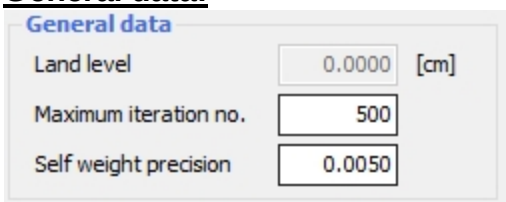
Many of the computation parameters defined in the "Settings" window are already set so as to work with most examinable structures. Others are automatically computed by the program based on the geometry of the model. The earthquake direction to be considered and the choice of the control node are chosen by the designer based on the indications found in the code.

The choice of the distribution of seismic forces (proportional to the masses or the first mode of vibration) is up to the designer.

The bearing capacity curve can be drawn monitoring displacement, in place of the control node of the average of the project, by selecting the appropriate text box.

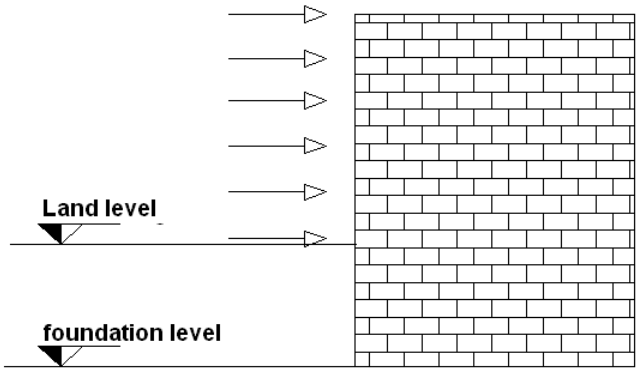


General data:



Land level: represents the elevation of the land level. The program assigns the lowest point of the structure elevation 0. The possibility of inserting this elevation allows the user to define the point where the seismic load initiates. The value of this elevation

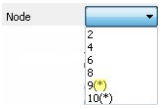
must be between the foundation elevation (generally zero) and the maximum elevation of all the constrained nodes.



Maximum iteration no.: represents the maximum number of analysis steps that the solver must perform before stopping the computation if no convergences are found.
Accuracy pbw: represents the degree of accuracy attained by the calculation of the first step of the calculation (where it is present only pbw)

Control node options: definition of a control node is obligatory for computation. It is recommended that the node is chosen in correspondence with the highest level of the structure.

- *Control node displacement*: the capacity curve is drawn only with the control node displacement.
- *Average displacement*: the capacity curve is drawn based on the average displacement of all nodes of the level at which the control node belongs
- *Weighted average displacement*: the capacity curve is drawn based on the average weighted displacement (weighed on the masses) of all nodes of the level at which the control node belongs. If the floor was infinitely rigid, this displacement would equate to the displacement of the center of gravity.



The check node can be chosen from the menu.
The **(*)** symbol indicates a top node of the roof.
Very often, it is preferred not to perform the calculation with a roof top type node because the deformation behavior at a ridge may not be representative of the actual behavior of the structure deformation.

This window performs multiple analysis in distinct cascades, for direction, orientation, type of seismic load, and eccentricity.

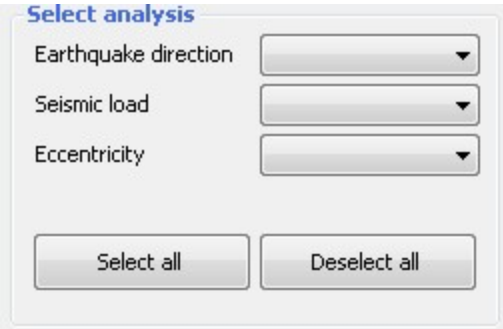
Direction: indicates the earthquake direction.

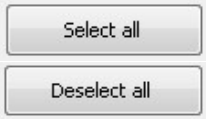
Orientation: positive if in concordance with the positive direction of the axis examined.

Seismic load: Proportional to the mass or the first node to vibrate.

Eccentricity: Accidental eccentricity of the center of mass with respect to the rigidity center computed automatically according to the code.

Using the associated space, multiple analysis can be performing by activating the selection filters.

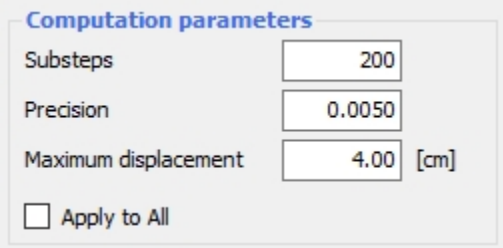




Enables the calculation of all the currently disabled analyzes

Disables the calculation of all the currently active analyzes

The parameters of each analysis can be set through the appropriate area.



Substeps: represents the number of displacement steps computed by the solver for the seismic load pattern.

Tolerance: represents the degree of tolerance reached by the non-linear computation.

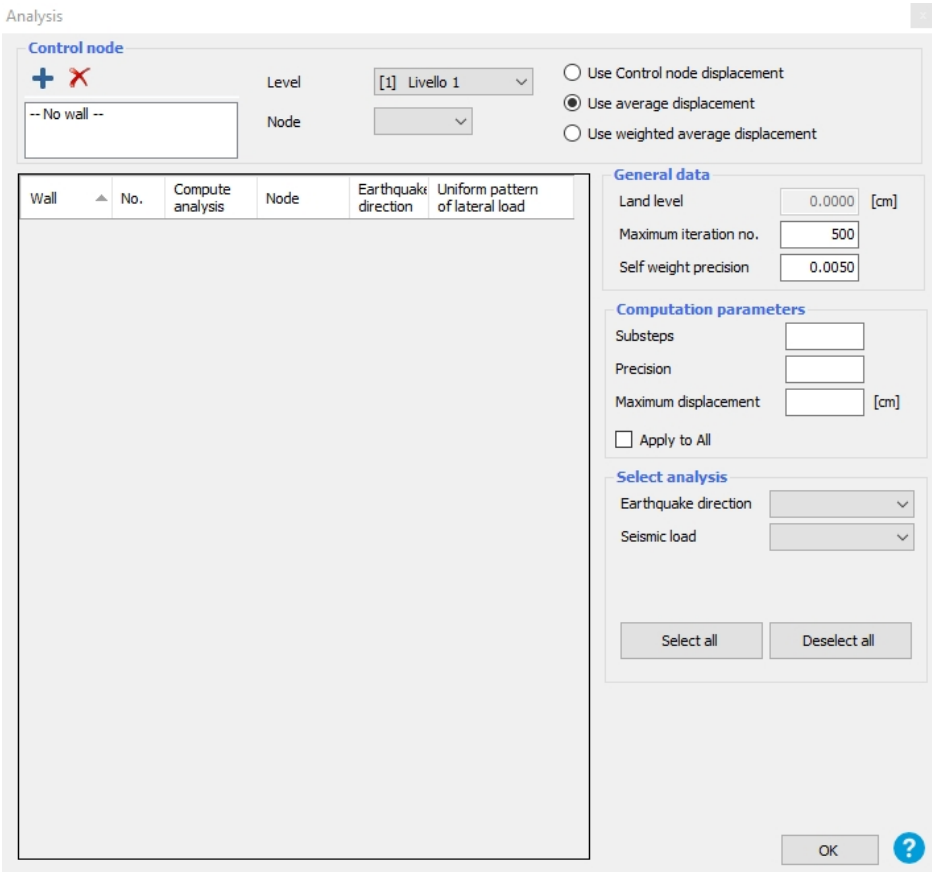
Maximum displacement: represents the maximum displacement that the structure's control node can withstand.

Apply to All: With the active tick the values are applied to all analyzes; if the check is disabled they are only applied to the selected analysis.

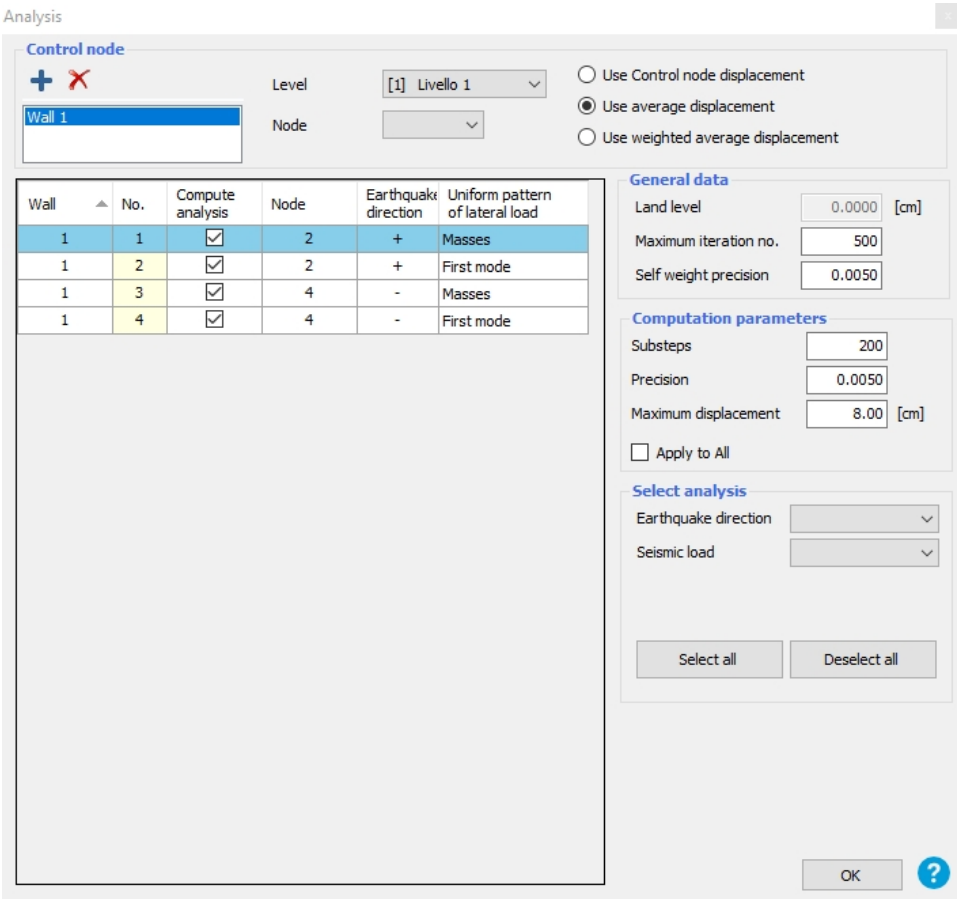
10.5.2.2 Single wall analysis


To access to the pushover analysis of the single wall, select "Single Wall Calculation" from the drop-down list.
In this phase, the computation is performed using the selected code.

The main screen that appears is as follows

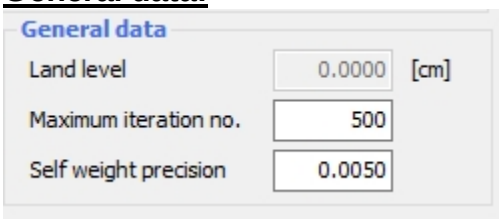


The aim of this analysis is to allow a pushover analysis considering the structure, not in its entirety, but in its individual walls, with the possibility of analyzing one or more walls together.
To start the analysis, as a first step yopu have to select the interested walls, by adding them from the appropriate command on the main screen (+).



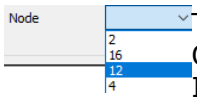
For each selected wall the software provides four differentiated analyzes for the direction of the earthquake and the distribution of the earthquake forces.

General data:



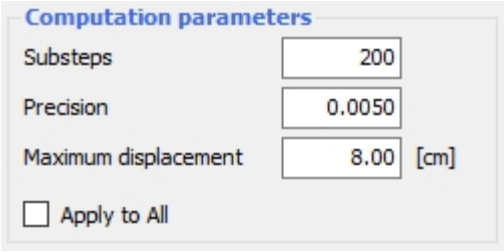
The General Data section is the same as in the global analysis dialog box described above.

Control node options: definition of a control node is obligatory for computation.



The check node can be chosen from the drop-down menu.
Only the nodes on the selected wall for the analysis are present.
In the calculation of single walls, the choice of control node has less importance than global verification since it is not affected by the deformation in the plan;for this reason, for single wall verification, the program "suggests" a node in order to facilitate the insertion of the calculation parameters.
There is no assurance about the efficiency of the proposed node, it is advisable to check the results at the end of the calculation.

The parameters of each analysis can be set through the appropriate area "Calculation Parameters".

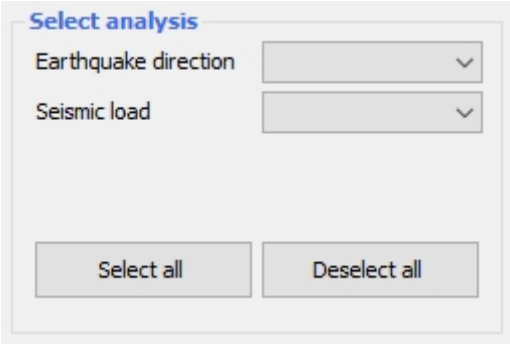


Substeps: represents the number of displacement steps computed by the solver for the seismic load pattern.

Tolerance: represents the degree of tolerance reached by the non-linear computation.

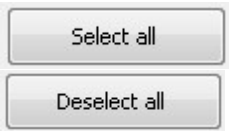
Maximum displacement: represents the maximum displacement that the structure's control node can withstand.

Apply to All: With the active tick the values are applied to all analyzes; if the check is disabled they are only applied to the selected analysis.



Earthquake Direction: indicates the earthquake direction.

Seismic load: Proporzionale alle masse o al primo modo di vibrare.

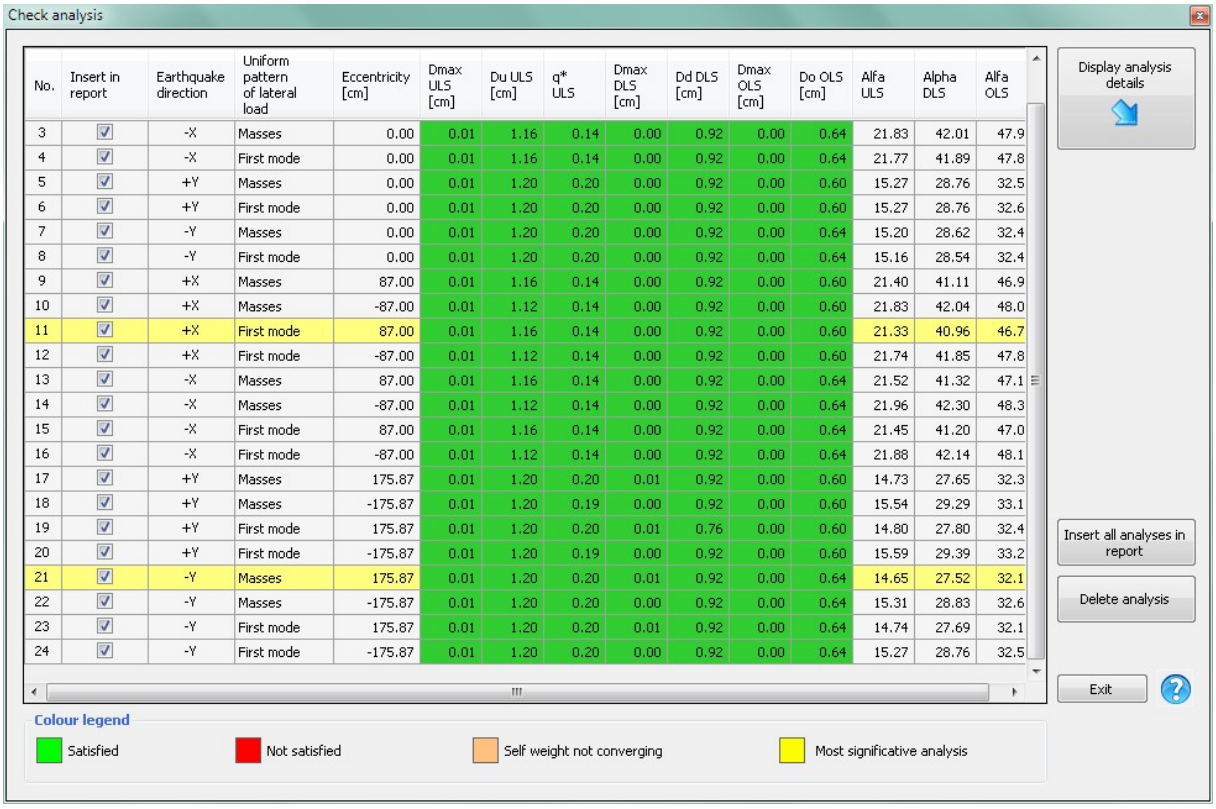


Enables the calculation of all the currently disabled analyzes

Disables the calculation of all the currently active analyzes

10.5.3 Display results

This window shows the results of the seismic computations performed on the model, based on that indicated in the code.

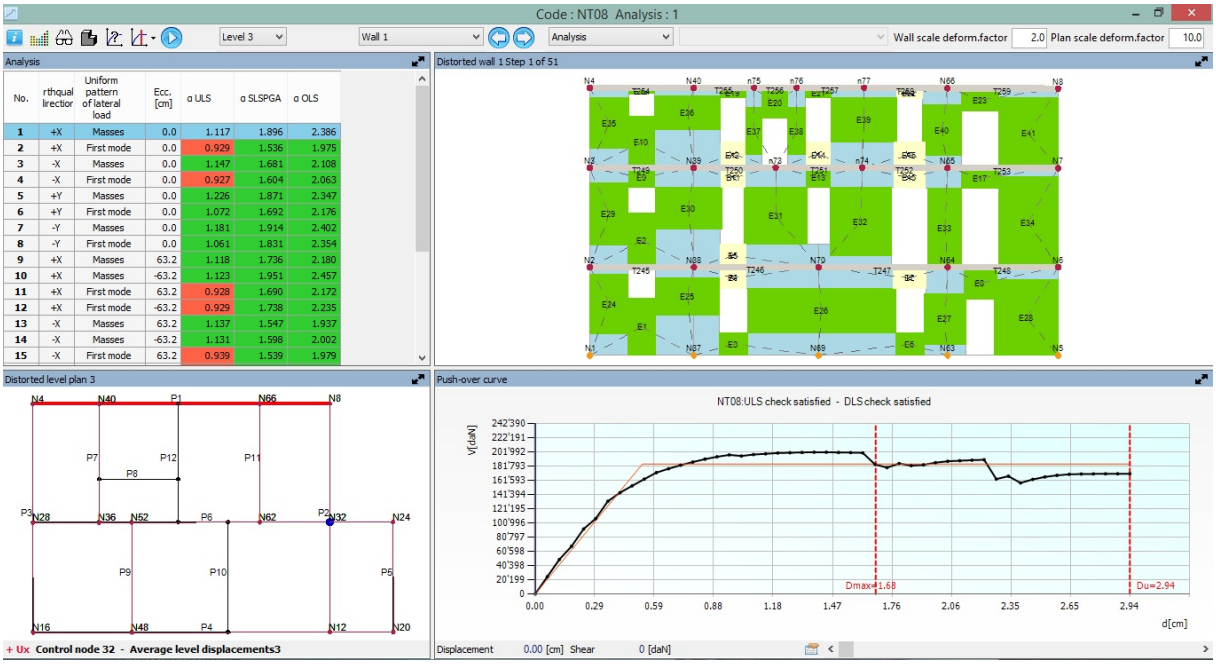


No.	Insert in report	Earthquake direction	Uniform pattern of lateral load	Eccentricity [cm]	Dmax ULS [cm]	Du ULS [cm]	q* ULS	Dmax DLS [cm]	Dd DLS [cm]	Dmax OLS [cm]	Do OLS [cm]	Alfa ULS	Alpha DLS	Alfa OLS
3	☑	-X	Masses	0.00	0.01	1.16	0.14	0.00	0.92	0.00	0.64	21.83	42.01	47.9
4	☑	-X	First mode	0.00	0.01	1.16	0.14	0.00	0.92	0.00	0.64	21.77	41.89	47.8
5	☑	+Y	Masses	0.00	0.01	1.20	0.20	0.00	0.92	0.00	0.60	15.27	28.76	32.5
6	☑	+Y	First mode	0.00	0.01	1.20	0.20	0.00	0.92	0.00	0.60	15.27	28.76	32.6
7	☑	-Y	Masses	0.00	0.01	1.20	0.20	0.00	0.92	0.00	0.64	15.20	28.62	32.4
8	☑	-Y	First mode	0.00	0.01	1.20	0.20	0.00	0.92	0.00	0.64	15.16	28.54	32.4
9	☑	+X	Masses	87.00	0.01	1.16	0.14	0.00	0.92	0.00	0.60	21.40	41.11	46.9
10	☑	+X	Masses	-87.00	0.01	1.12	0.14	0.00	0.92	0.00	0.60	21.83	42.04	48.0
11	☑	+X	First mode	87.00	0.01	1.16	0.14	0.00	0.92	0.00	0.60	21.33	40.96	46.7
12	☑	+X	First mode	-87.00	0.01	1.12	0.14	0.00	0.92	0.00	0.60	21.74	41.85	47.8
13	☑	-X	Masses	87.00	0.01	1.16	0.14	0.00	0.92	0.00	0.64	21.52	41.32	47.1
14	☑	-X	Masses	-87.00	0.01	1.12	0.14	0.00	0.92	0.00	0.64	21.96	42.30	48.3
15	☑	-X	First mode	87.00	0.01	1.16	0.14	0.00	0.92	0.00	0.64	21.45	41.20	47.0
16	☑	-X	First mode	-87.00	0.01	1.12	0.14	0.00	0.92	0.00	0.64	21.88	42.14	48.1
17	☑	+Y	Masses	175.87	0.01	1.20	0.20	0.01	0.92	0.00	0.60	14.73	27.65	32.3
18	☑	+Y	Masses	-175.87	0.01	1.20	0.19	0.00	0.92	0.00	0.60	15.54	29.29	33.1
19	☑	+Y	First mode	175.87	0.01	1.20	0.20	0.01	0.76	0.00	0.60	14.80	27.80	32.4
20	☑	+Y	First mode	-175.87	0.01	1.20	0.19	0.00	0.92	0.00	0.60	15.59	29.39	33.2
21	☑	-Y	Masses	175.87	0.01	1.20	0.20	0.01	0.92	0.00	0.64	14.65	27.52	32.1
22	☑	-Y	Masses	-175.87	0.01	1.20	0.20	0.00	0.92	0.00	0.64	15.31	28.83	32.6
23	☑	-Y	First mode	175.87	0.01	1.20	0.20	0.01	0.92	0.00	0.64	14.74	27.69	32.1
24	☑	-Y	First mode	-175.87	0.01	1.20	0.20	0.00	0.92	0.00	0.64	15.27	28.76	32.5

This window summarizes the check parameters according to each norm, indicating whether the results were satisfactory or not.
The analyzes that have minimum "Alfa" values are more restrictive, so the results window shows the two analyzes with the minimum "Alfa SLV" (one for the X direction and one for the Y direction).
On the right of the window there are commands with the following functions:

Display analysis details	Enter in the window that allows you to show the verification details.
Insert all analyses in report	Allows to print the parameters of all the analyzes included in the report
Delete analysis	Clear the results of a performed analysis

10.5.3.1 Display analysis details



The results' environment is divided in 4 main areas:

Analysis table

Contains the performed analyzes summary for the model taken in consideration.
The first columns describe the type of analysis, the last show the vulnerability indexes for each of the three limit states.
The background color, green or red, distinguishes between the exceeded analysis by those that aren't.
The yellow color shows the two analyzes that have the lowest vulnerability indexes (more significant for calculation's purposes).

Analysis

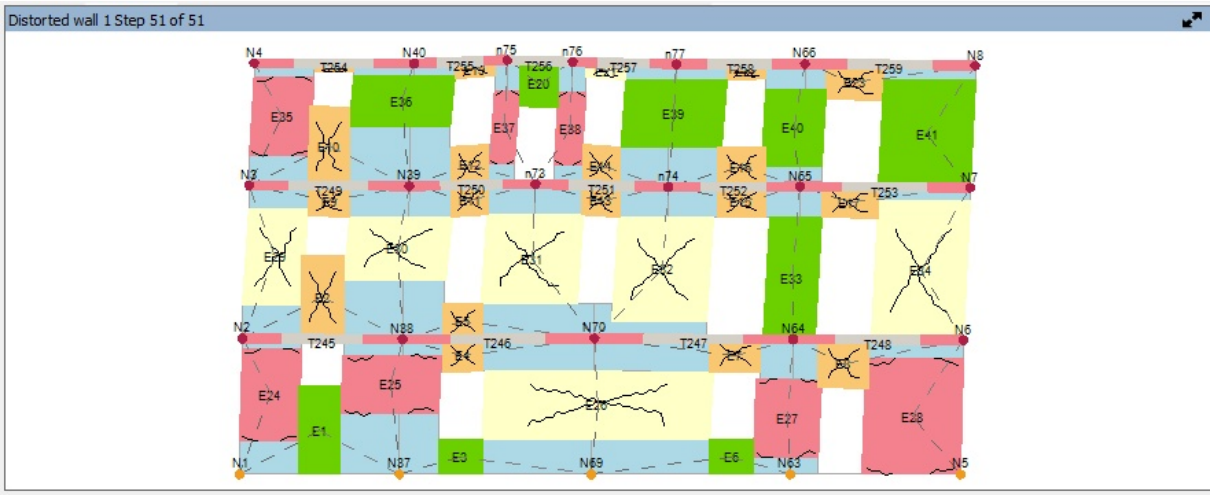
No.	rthqual lirectior	Uniform pattern of lateral load	Ecc. [cm]	dm/dt	α ULS	α DLS
1	+X	Masses	0.0	0.91	0.555	0.816
2	+X	First mode	0.0	0.71	0.461	0.717
3	-X	Masses	0.0	0.70	0.569	0.785
4	-X	First mode	0.0	0.73	0.466	0.731
5	+Y	Masses	0.0	0.64	0.639	0.866
6	+Y	First mode	0.0	0.71	0.529	0.800
7	-Y	Masses	0.0	0.72	0.623	0.862
8	-Y	First mode	0.0	0.76	0.551	0.819

The active analysis is highlighted in blue, it's results are detailed in the other 3 areas.

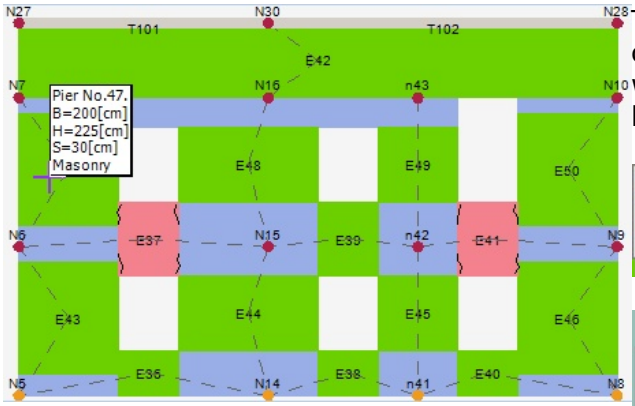
Clicking on a row in the table will automatically switch to the details of another analysis.

Distorted Wall

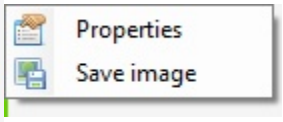
Shows the wall's deformation, the elements color indicates the type of the identified damage immediately through the color legend.
The deformation is shown in correspondence of each single step.
This tool shows a great potential for the management of any adjustment interventions on the existing structure, as it proves to be very effective for the identification of the intervention areas.

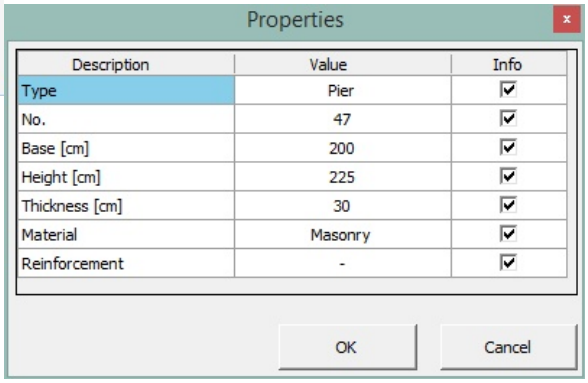


Positioning the pointer over an item displays a tool tip with the masonry's characteristics.



The content of the tool tip can be customized through a special window which is accessed by pressing the right button.



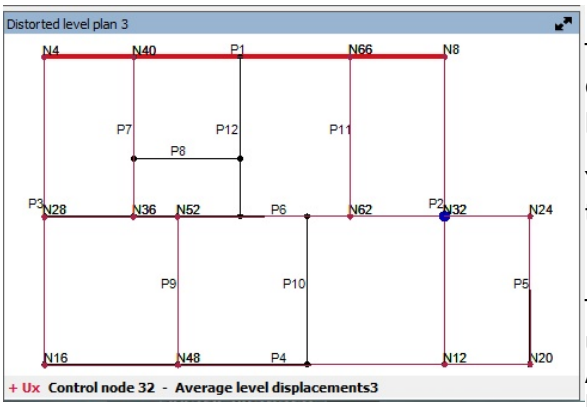


The window of the mesh prospectus is a real CAD, the functions zoom and pan are available by using the mouse.

A double-click on the wheel zooms extension bringing it back to the maximized window.
Pressing the right mouse button the option "Save Image" which allows you to save the image to be used in the report is available.

Distorted level Plant

An overlapped view of the deformed plant to the not deformed one helps to identify any torsional effects.



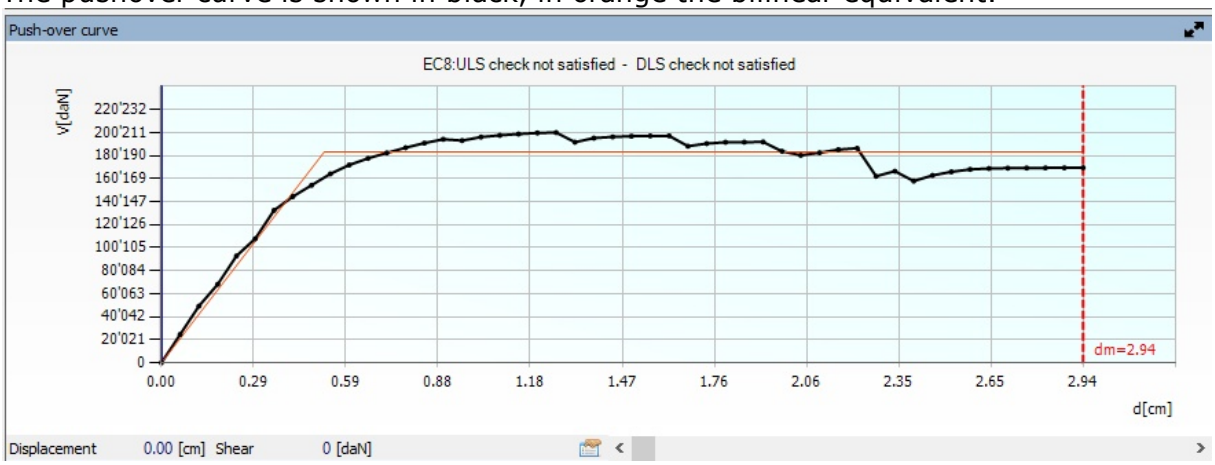
The control node is shown in the lower part of the window and highlighted in blue in the plant.

You can select a wall in the plant to see it in the statement of deformation.

The zoom and pan functions are available by using the mouse.
A double-click on the wheel zooms extension bringing it back to the maximized window.
Pressing the right mouse button the option "Save Image" which allows you to save the image to be used in the report is available.

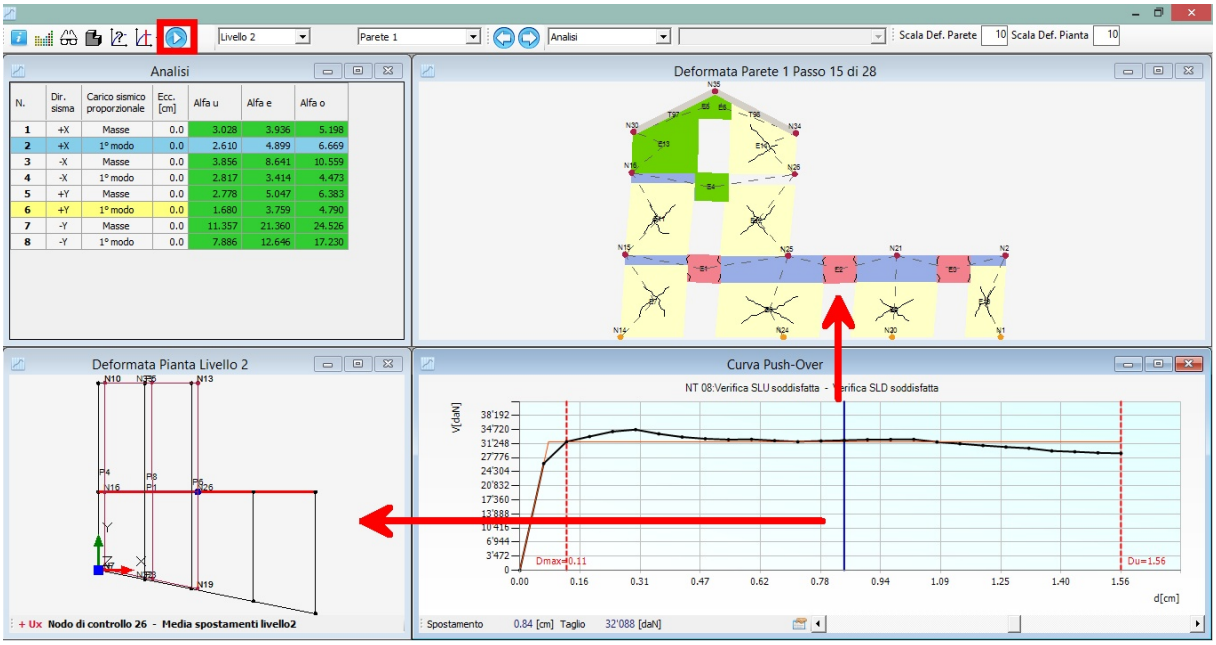
Push-over Curve

Shows the capacity curve.
The pushover curve is shown in black, in orange the bilinear equivalent.



Du: Offered displacement of the structure
Dmax: displacement demand of the seisma

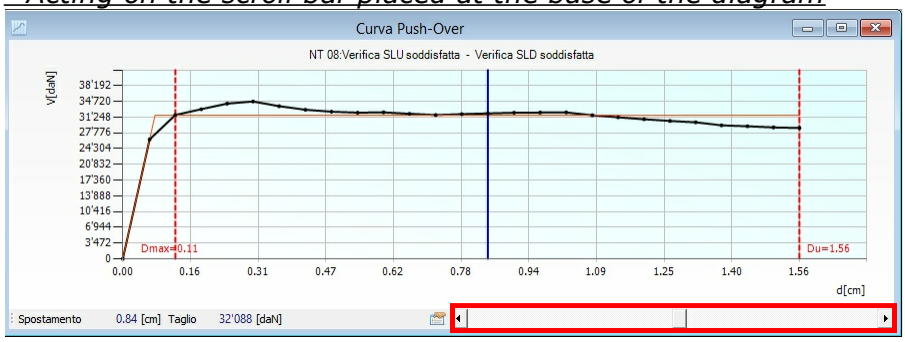

Auto run: By using this command, you can start a video showing the progressive deformation of the structure.
Is displayed a capacity curve indicator that allows us to understand at what point in the evolution of load are we; The prospectus and the plant show the deformed state.



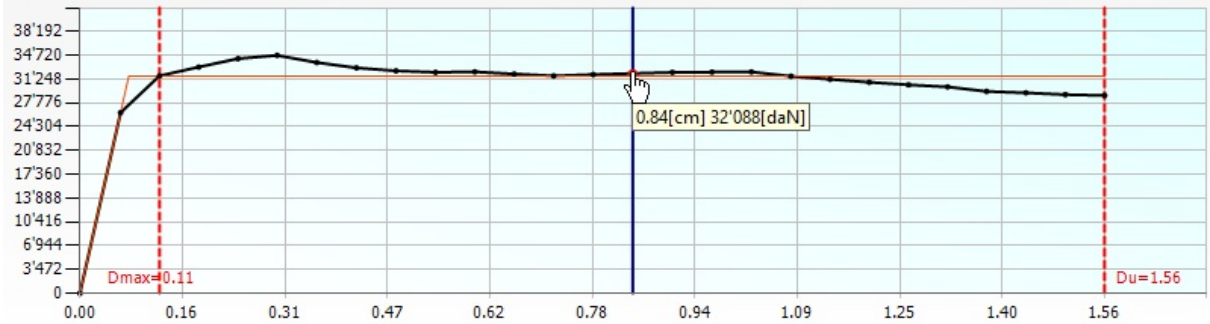
N.	Dir. sisma	Carico sismico proporzionale	Ecc. [cm]	Alfa u	Alfa e	Alfa o
1	+X	Masse	0.0	3.028	3.936	5.198
2	+X	1° modo	0.0	2.610	4.899	6.669
3	-X	Masse	0.0	3.856	8.641	10.559
4	-X	1° modo	0.0	2.817	3.414	4.473
5	+Y	Masse	0.0	2.778	5.047	6.383
6	+Y	1° modo	0.0	1.680	3.759	4.790
7	-Y	Masse	0.0	11.357	21.360	24.526
8	-Y	1° modo	0.0	7.886	12.646	17.230

You can place the marker at a specific point in two different ways:

- Acting on the scroll bar placed at the base of the diagram



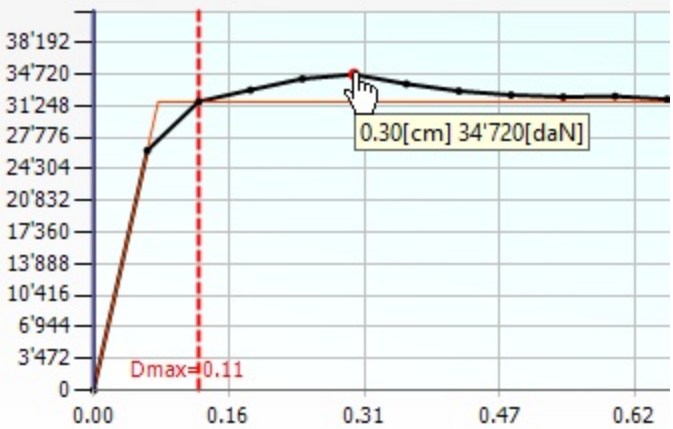
- By clicking on the diagram at the step where you want to place the indicator



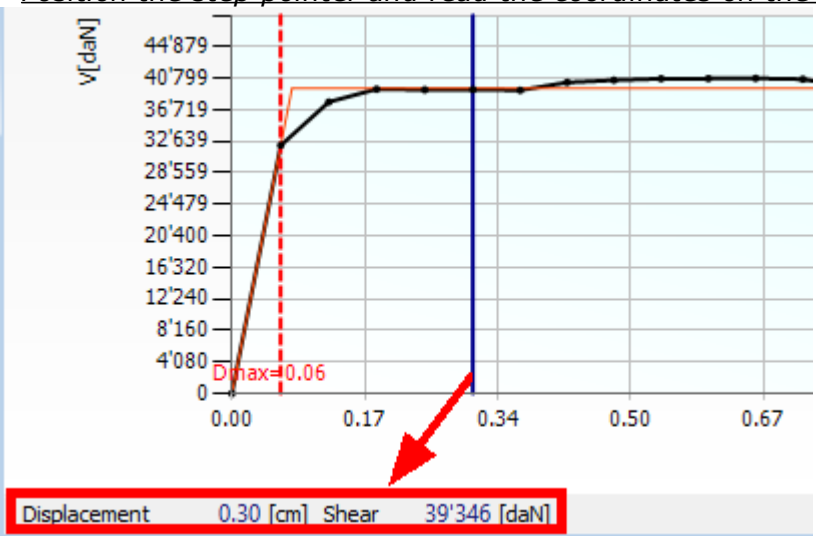
Interrogate the curve's coordinates

You can interrogate the values of the curve in three different ways:

- *Position the pointer on the curve at a step and read the value that appear*

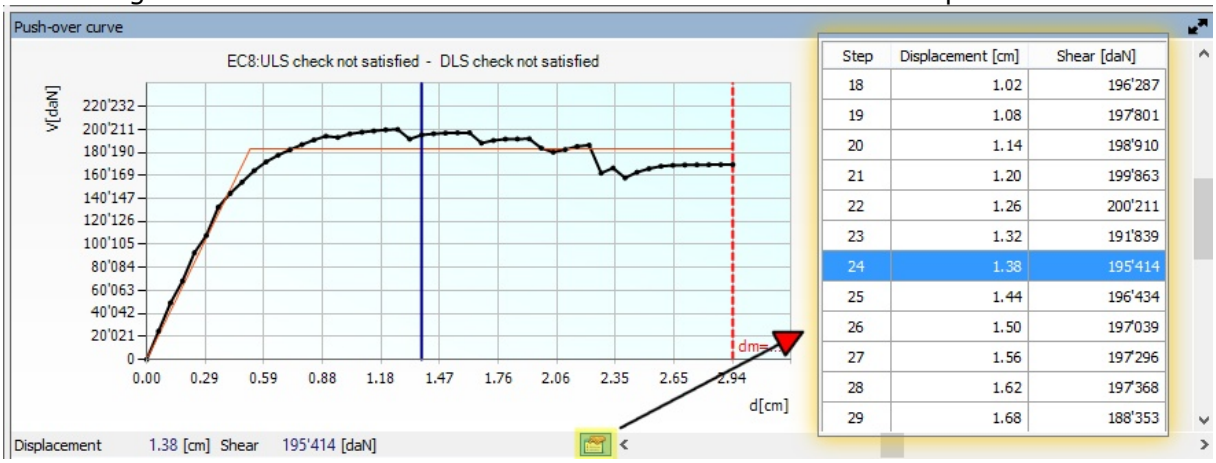


- *Position the step pointer and read the coordinates on the bottom bar*



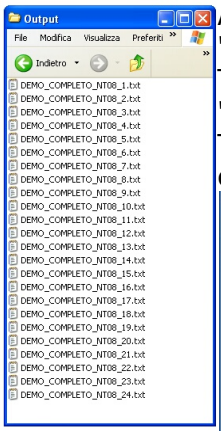
- *Refer to the table with coordinates*


Using this button shows the table with the coordinates of each point on the curve.

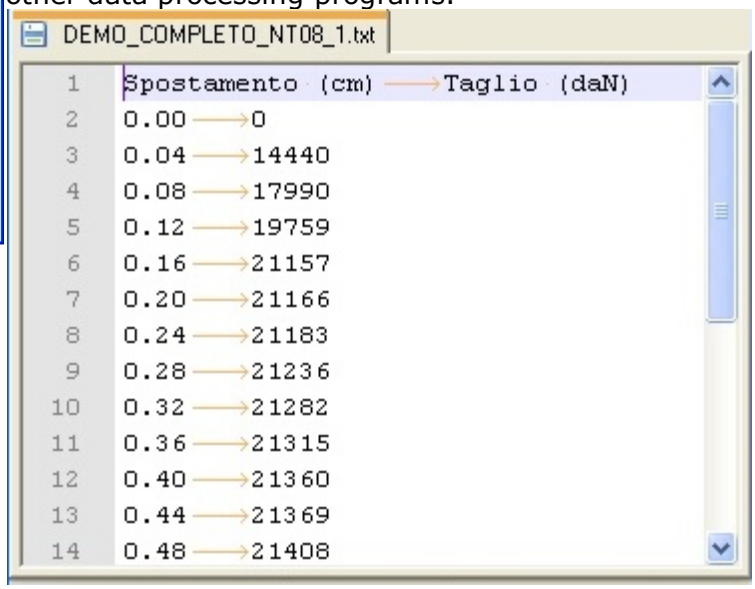


The row corresponding to the active step appears highlighted.
Clicking the right mouse button on the table allows you to copy the contents of the table.

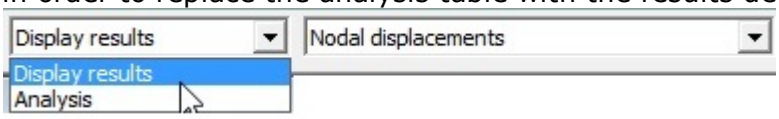
Exporting the pushover's diagram points

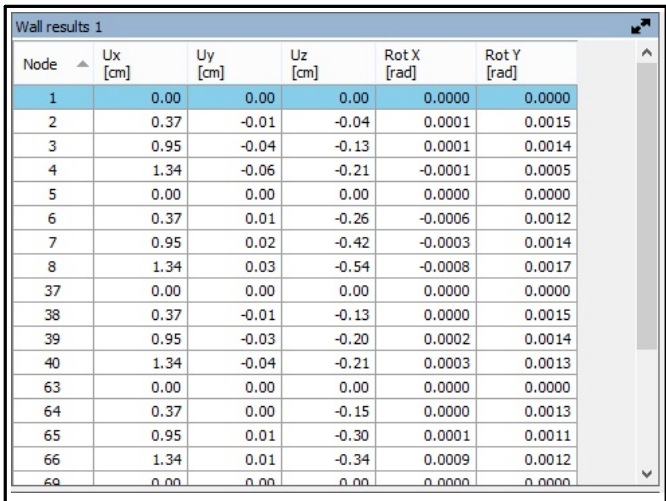


At finished calculation, within the project folder in the directory "output", is saved a file for each of the performed analyzes.
The file will have the name "ModelName"_"Regulations"_"N.ANALYSIS".TXT.
The points of the curve are separated by tabs for easier processing by other data processing programs.

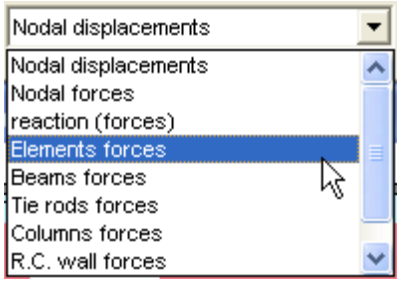


On the command bar, you can change the option from "Analysis" to "Display results" in order to replace the analysis table with the results details table.





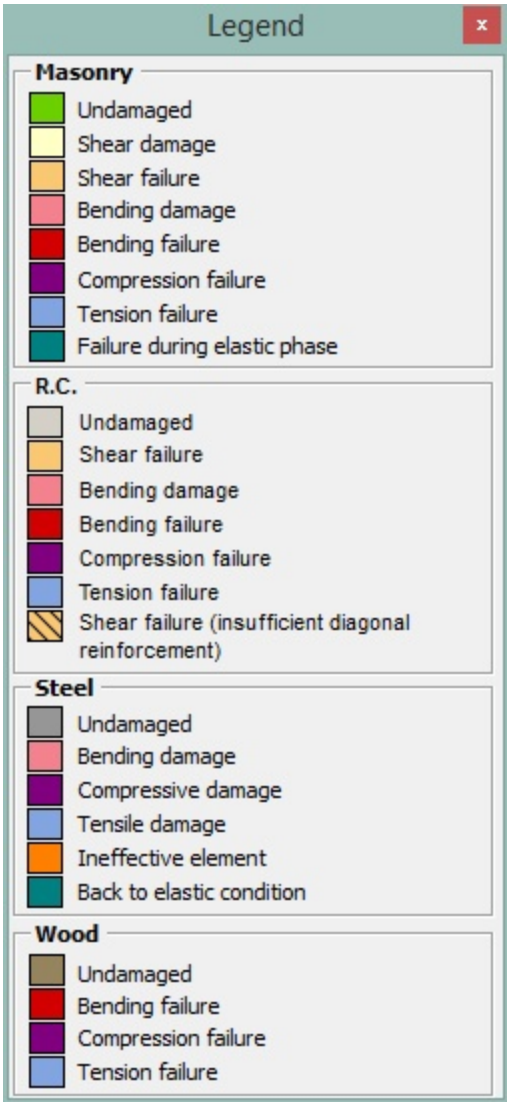
Node	Ux [cm]	Uy [cm]	Uz [cm]	Rot X [rad]	Rot Y [rad]
1	0.00	0.00	0.00	0.0000	0.0000
2	0.37	-0.01	-0.04	0.0001	0.0015
3	0.95	-0.04	-0.13	0.0001	0.0014
4	1.34	-0.06	-0.21	-0.0001	0.0005
5	0.00	0.00	0.00	0.0000	0.0000
6	0.37	0.01	-0.26	-0.0006	0.0012
7	0.95	0.02	-0.42	-0.0003	0.0014
8	1.34	0.03	-0.54	-0.0008	0.0017
37	0.00	0.00	0.00	0.0000	0.0000
38	0.37	-0.01	-0.13	0.0000	0.0015
39	0.95	-0.03	-0.20	0.0002	0.0014
40	1.34	-0.04	-0.21	0.0003	0.0013
63	0.00	0.00	0.00	0.0000	0.0000
64	0.37	0.00	-0.15	0.0000	0.0013
65	0.95	0.01	-0.30	0.0001	0.0011
66	1.34	0.01	-0.34	0.0009	0.0012
69	0.00	0.00	0.00	0.0000	0.0000



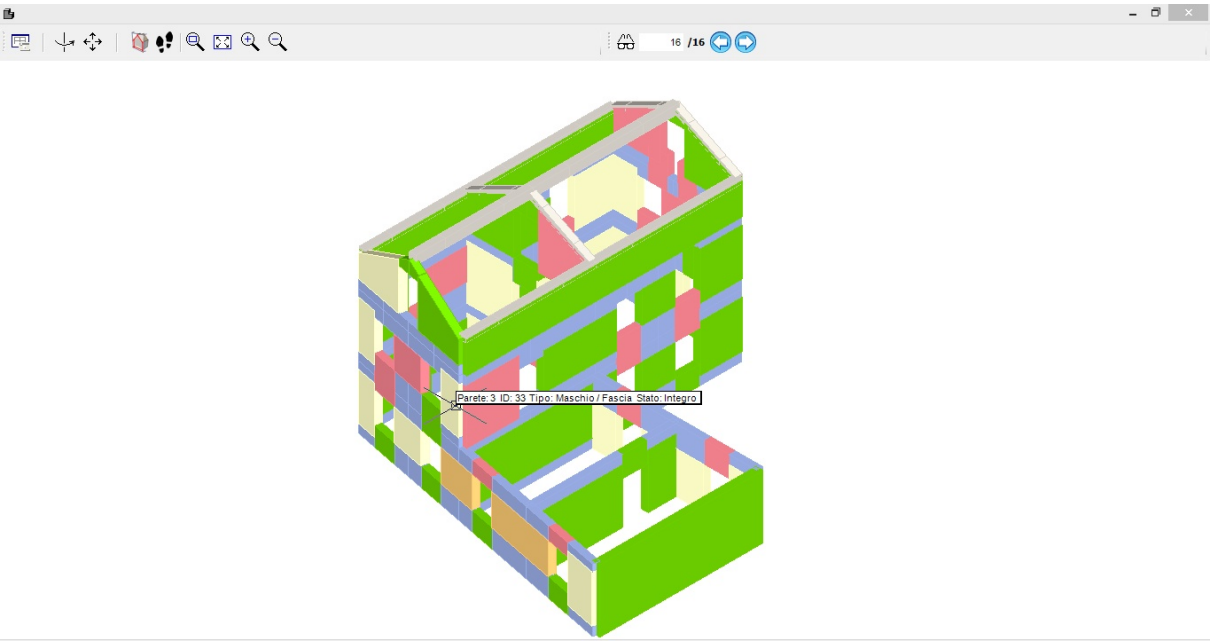
“ In this window, every value can be selected. With a right click on the button of the mouse in the selected area "Copy data" appears; by selecting it can be loaded on the clipboard in order to be pasted in the application that we consider most suitable. (word, excel, ecc..). ”

Check Details:
Represents a summary window that displays the details of the required analyzes and checks.

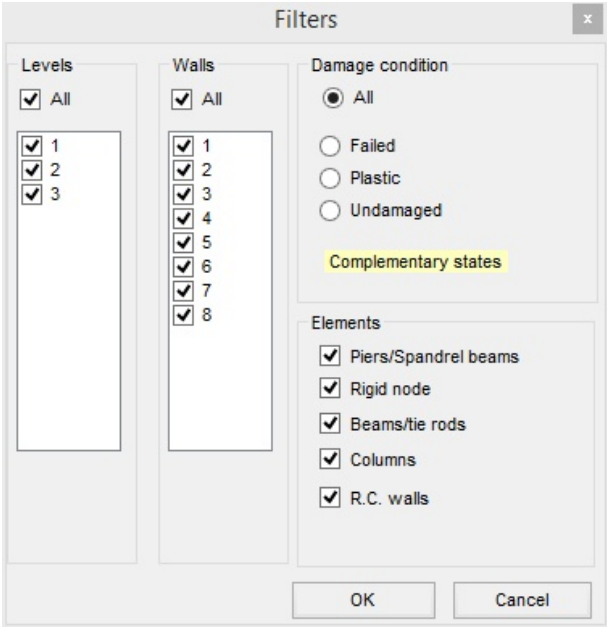
Colors legend: Shows a colors map in order to identify the type of damage to the structure (the map shows the damage of the masonry elements, of the R.C., steel and wood).



View 3D of the mesh: Allows you to see the 3D of the mesh distinguished according to the step of damage and with the color map just described.



Comands in 3D view:



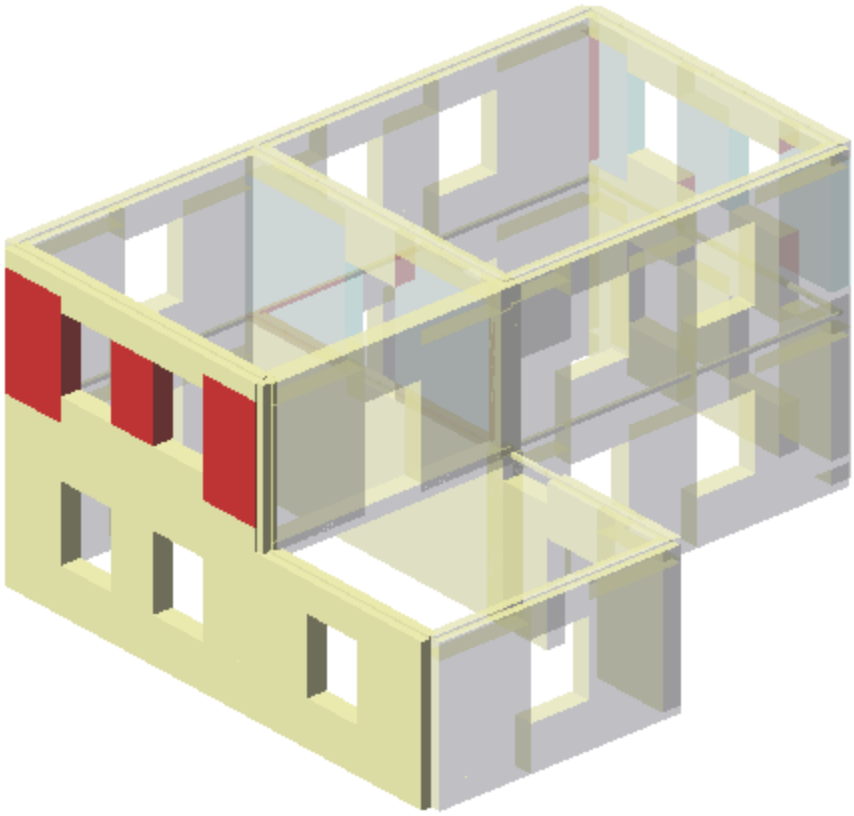
Display filters:

- *Levels*: Are shown only the selected levels
- *Walls*: Are shown only the selected walls.
- *Elements*: Are shown only the selected elements.

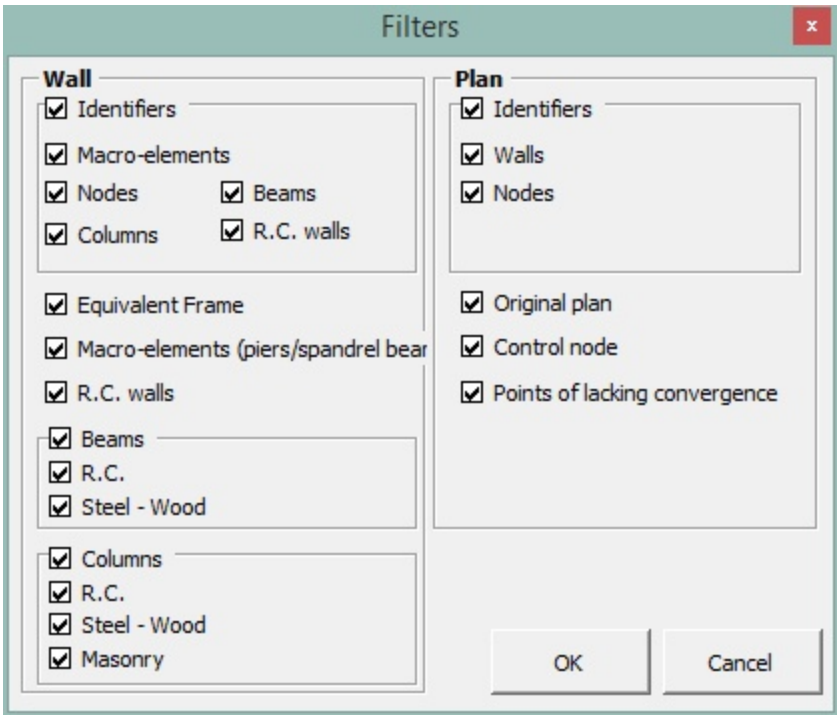
Levels / Walls not selected may be shown as transparency.

You can decide to show the items with the corresponding color of the legend only if they have a well-defined state of damage.

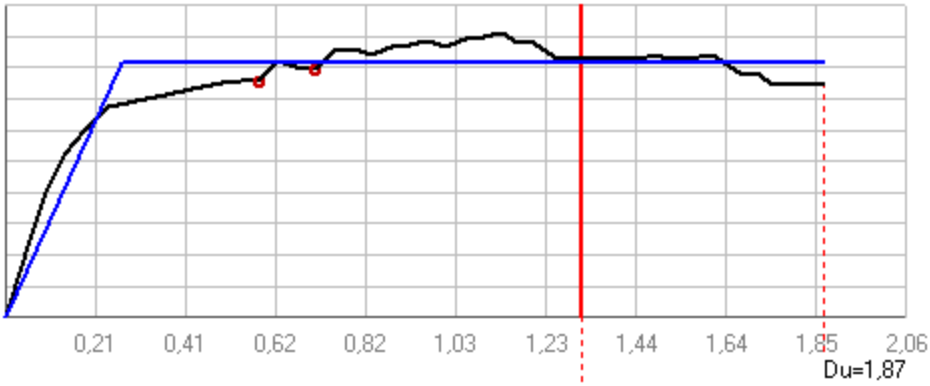
Viewing the property of transparency and showing only the broken elements is easier to locate the only points of structural weakness.



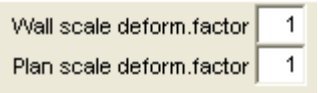
Display filters: Allow the user to decide what to display in two views (level plan, wall) of the deformation.



This window allows you to decide whether to display the points of possible failure convergence in the non-linear pushover which are shown on the chart with small red circles.



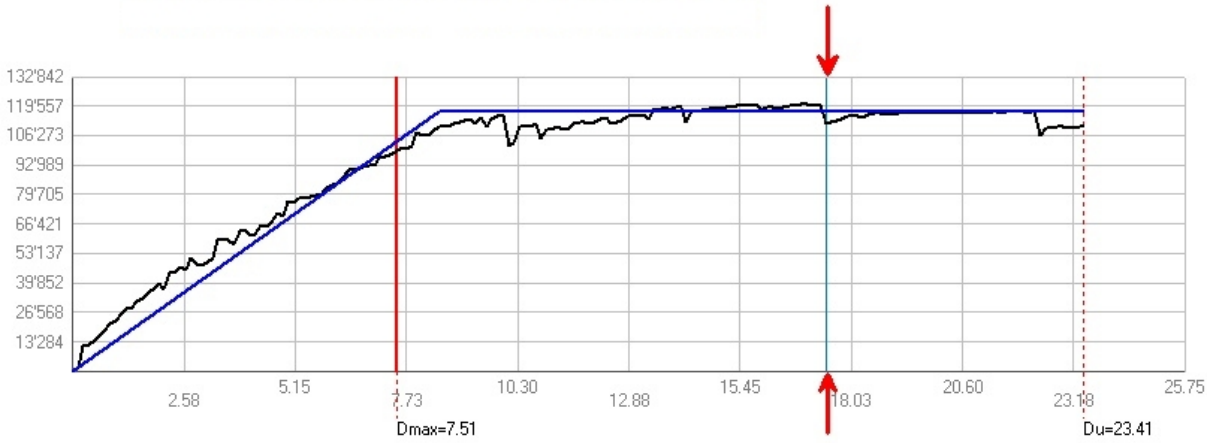
The presence of some non-convergence points should not worry but the presence of a high number of outside convergence steps must be index of a modeling that can be improved.



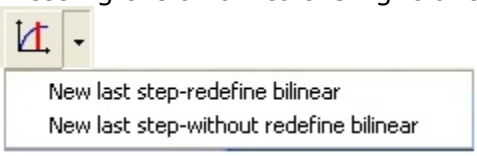
These commands present in the analysis bar, allow to decide the scale of display of the deformations both on the prospect of the wall and on the level plant.

New Last step

If the user considers it appropriate, may decide to find a different displacement value for the ultimate limit state, as long as lower than the value associated with the decay of resistance limit.
Use the scroll bar to go through the underpasses of the analysis and to identify by using the appropriate blue marker a new "last step".

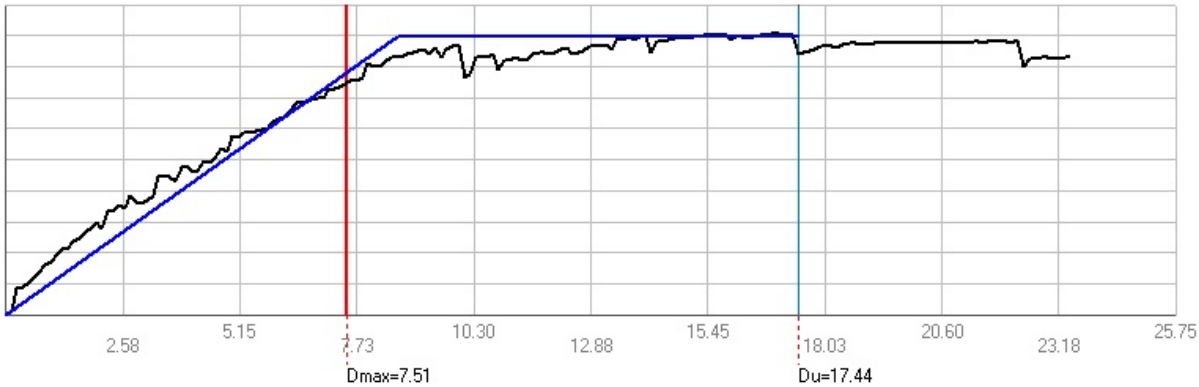


Pressing the arrow to the right of the "New last step", there are two options.



New step - bilinear recalculation:

The bilinear is redefined taking into account all the points of the curve until the new ultimate displacement.



When to you use this command?
The simultaneous rupture of many elements that could affect the overall stability. An example would be the failure of all pillars of a porch which is located at the main entrance of a building. This break could be considered by the designer to continue limiting the analysis.

New step-without bilinear recalculation:
In this case the displacement is redefined at the new last step but it is not redefined the bilinear by maintaining the original one.

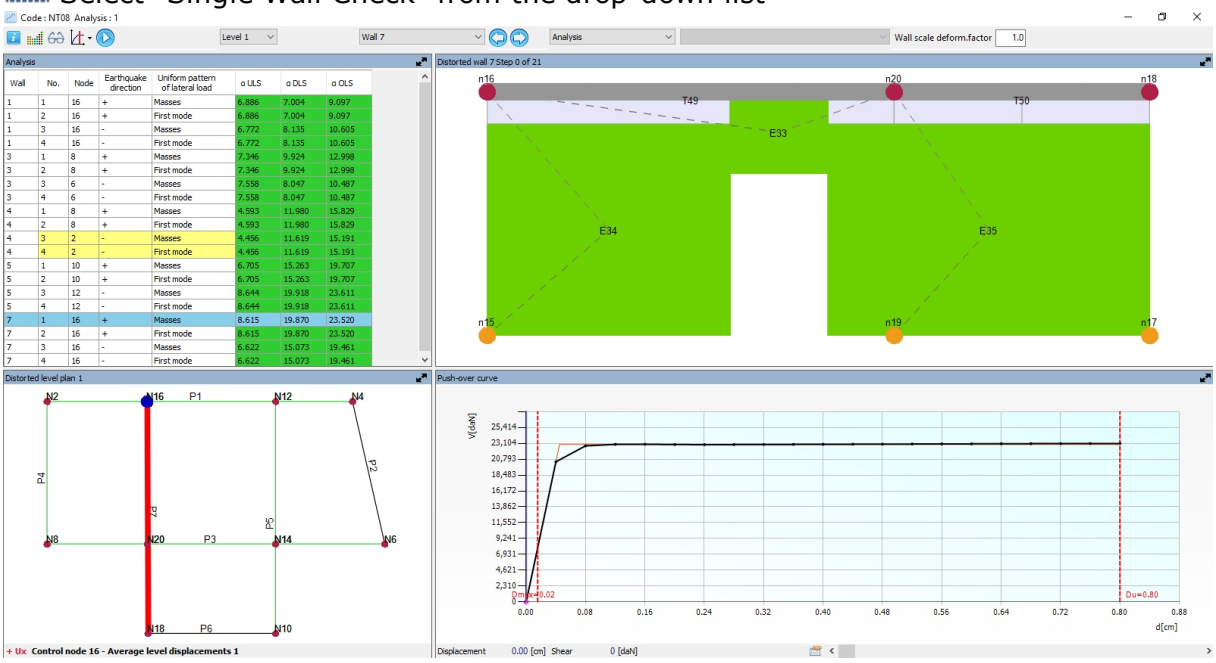
When to you use this command?
This command is used when the designer considers useful interrupt the analysis in a previous step due to an element considered of significant importance, the stability of which probably can not give rise to failures that would alter the continuation of the deformation of the structural complex.

CAUTION!!! For very low values of displacement is not possible to define new values of ultimate displacement; such low values may not allow the regeneration of the bilinear equivalent because you can not abide by the requirements of tracking (eg, the intersection of the bilinear pushover and conservation of energy dissipation).

In this case, the program informs that there is no bilinear that satisfies these requirements.
Confirming this window, the program defines a new step by replacing the last bilinear with a linear stretch that dissipates the same energy of the pushover (in this case can not therefore be imposed as a point of passage for the bilinear).

10.5.3.2 Show details single wall analysis

Select "Single Wall Check" from the drop-down list



The result environment is divided into 4 main areas:

Analysis Table

Contains the performed analyzes summary for the model taken in consideration.
The first columns describe the type of analysis, the last show the vulnerability indexes for each of the limit states.
The background color, green or red, distinguishes between the exceeded analysis by those that aren't.
The yellow color shows the two analyzes that have the lowest vulnerability indexes (more significant for calculation's purposes).

Analysis

Wall	No.	Node	Earthquake direction	Uniform pattern of lateral load	α ULS	α DLS	α OLS
1	1	16	+	Masses	6.886	7.004	9.097
1	2	16	+	First mode	6.886	7.004	9.097
1	3	16	-	Masses	6.772	8.135	10.605
1	4	16	-	First mode	6.772	8.135	10.605
3	1	8	+	Masses	7.346	9.924	12.998
3	2	8	+	First mode	7.346	9.924	12.998
3	3	6	-	Masses	7.558	8.047	10.487
3	4	6	-	First mode	7.558	8.047	10.487
4	1	8	+	Masses	4.593	11.980	15.829
4	2	8	+	First mode	4.593	11.980	15.829
4	3	2	-	Masses	4.456	11.619	15.191
4	4	2	-	First mode	4.456	11.619	15.191
5	1	10	+	Masses	6.705	15.263	19.707
5	2	10	+	First mode	6.705	15.263	19.707
5	3	12	-	Masses	8.644	19.918	23.611
5	4	12	-	First mode	8.644	19.918	23.611
7	1	16	+	Masses	8.615	19.870	23.520
7	2	16	+	First mode	8.615	19.870	23.520
7	3	16	-	Masses	6.622	15.073	19.461
7	4	16	-	First mode	6.622	15.073	19.461

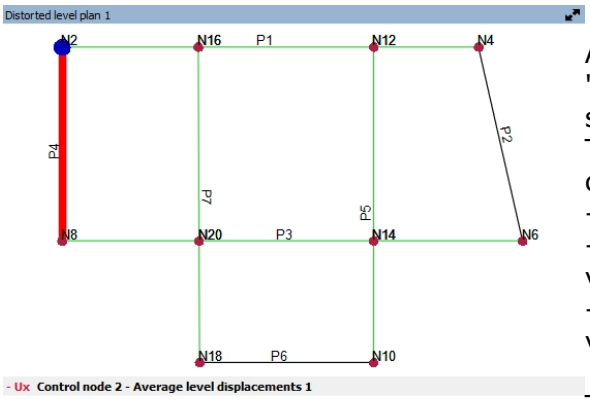
The active analysis is highlighted in blue, it's results are detailed in the other 3 areas.

By clicking on a row in the table will automatically switch to the details of another analysis.

Distorted Wall

The deformed wall of the prospectus follows the same rules described for the global analysis.

Plan



A thick red-colored stretch highlights the "active" wall (the one whose results are shown).
The walls can be drawn with a thin stretch of color:
- Black: Wall not calculated
- Green: Calculated with exceeded verification
- Red: calculated with NOT exceeded verification

The control node is indicated at the bottom of the window and is highlighted in blue in the plant based on the currently calculated wall.

It is possible to select a wall in the plant to see it in the prospect of the deformation.

Zoom and pan functions are available with the mouse.
A double click on the wheel zooms extension

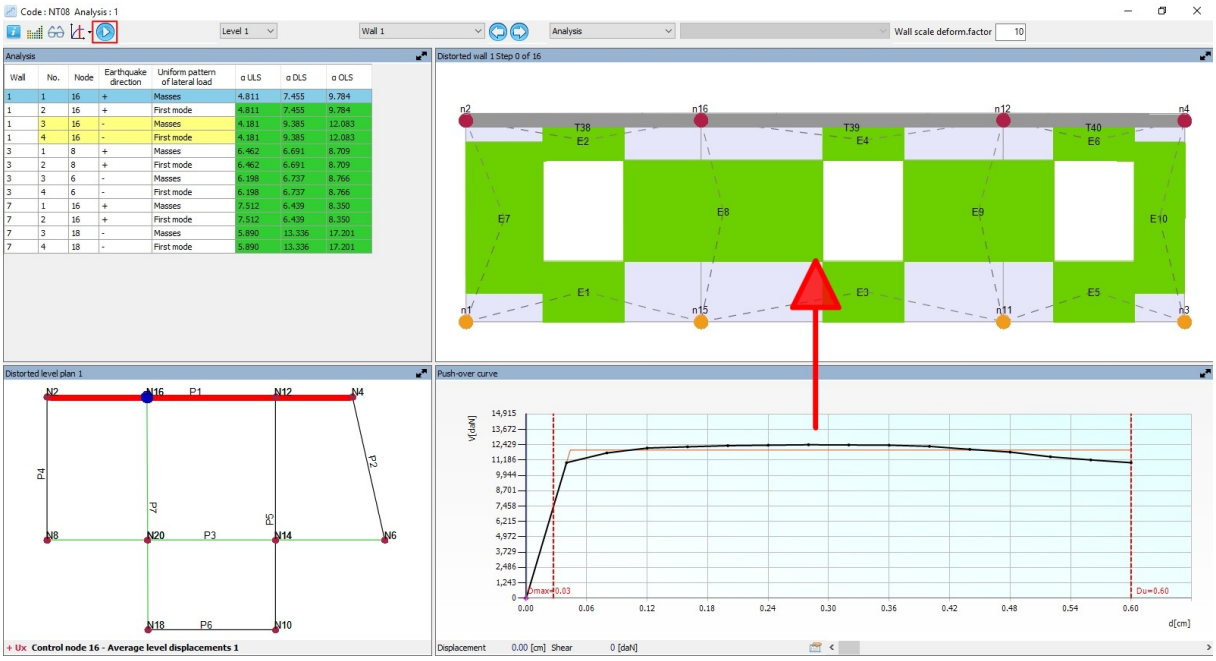
maximized it back to the window.
By pressing the right mouse button, the "Save Image" option is available to save the image to be used in the calculation report.

In the case of the calculation of the individual walls, the plant shown is undeformed.

Push-over curve

The capacity curve diagram follows the same rules.

Auto run: By using this command, you can start a video showing the progressive deformation of the structure.
Is displayed a capacity curve indicator that allows us to understand at what point in the evolution of load are we; The prospectus and the plant show the deformed state.

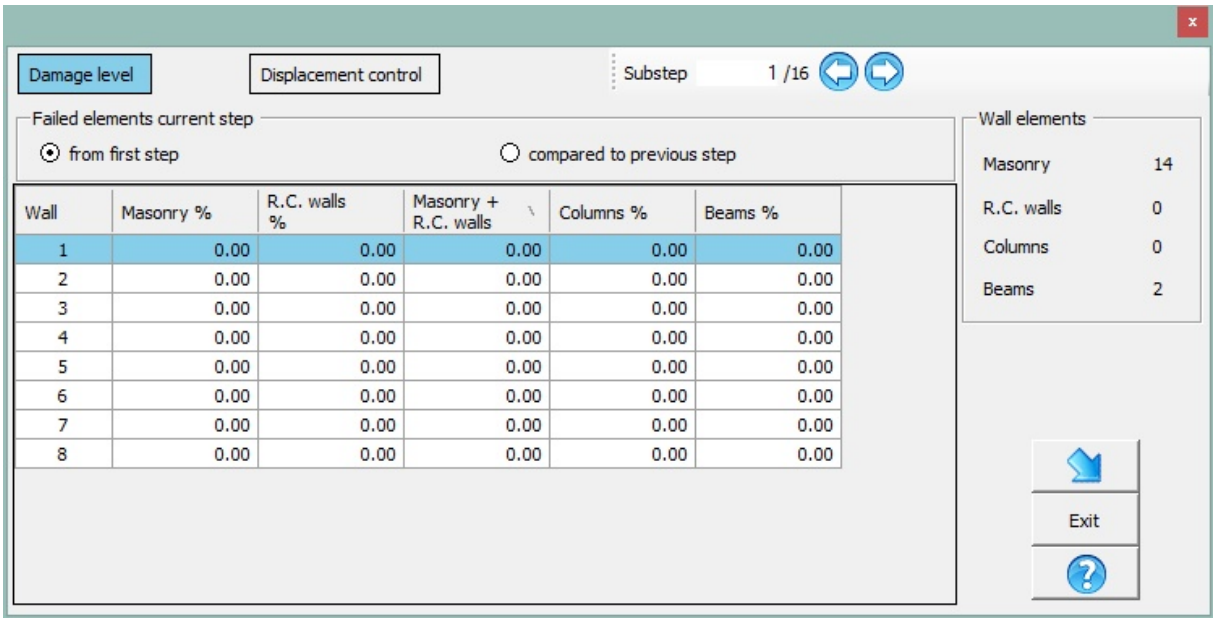


10.5.3.3 Display Results

This environment displays a table showing on rows the percentage of the damaged elements for each wall. The rows and therefore the walls, are sorted according to the percentage of damaged elements.
Under this system you can immediately identify the most damaged wall (the first on the list).

Allows to immediately load the view of the selected wall (the corresponding row) in order to examine it to provide interventions.

The percentage of broken elements presented in the table can be defined from the beginning of the loading history or in the current step of the analysis.



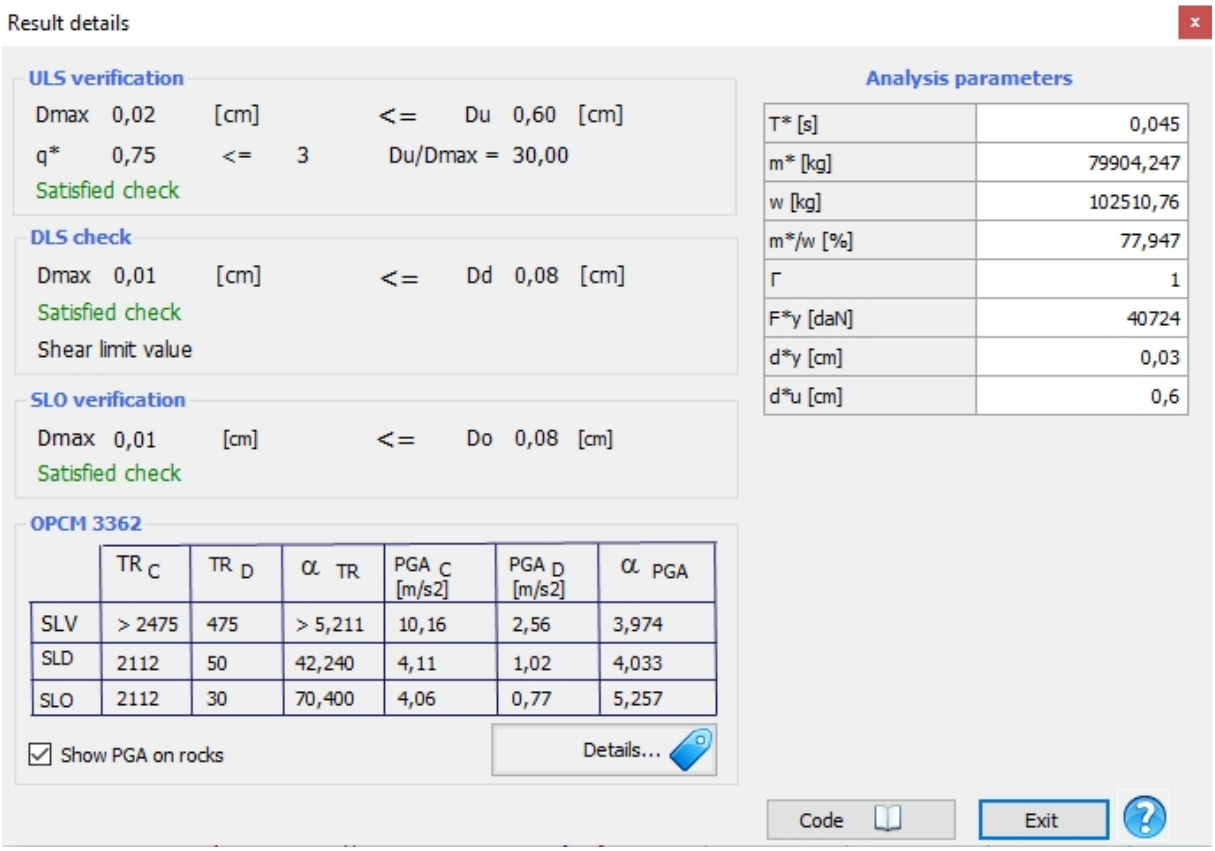
Wall	Masonry %	R.C. walls %	Masonry + R.C. walls	Columns %	Beams %
1	0.00	0.00	0.00	0.00	0.00
2	0.00	0.00	0.00	0.00	0.00
3	0.00	0.00	0.00	0.00	0.00
4	0.00	0.00	0.00	0.00	0.00
5	0.00	0.00	0.00	0.00	0.00
6	0.00	0.00	0.00	0.00	0.00
7	0.00	0.00	0.00	0.00	0.00
8	0.00	0.00	0.00	0.00	0.00

A second environment of "displacement control" orders the walls according to the relative interstorey displacement in order to identify where the greater displacement occurs.

Main wall	Bottom node	Top node	Relative displacement [cm]	Level	Walls involved
1	16	30	0.00	Roof	1-4
1	26	34	0.00	Roof	1-6
4	7	27	0.00	Roof	3-4
5	10	28	0.00	Roof	4-5
5	9	10	0.00	2	4-5
5	13	29	0.00	Roof	5-6
1	15	16	0.00	2	1-4
1	1	2	0.00	1	1-2
3	3	4	0.00	1	2-3
7	22	23	0.00	1	3-7
6	19	31	0.00	Roof	3-6
5	12	13	0.00	2	5-6

10.5.3.4 Results Details

Represents a summary window that displays the details of the analyzes and required checks.



This window displays a summary of the test parameters required by each regulation.

NTC08

The table for the "Seismic Vulnerability" evaluation shows the α parameters derived from the homonyms reports for each of the limit states:

$\alpha_{PGA}=PGA_C/PGA_D$; $\alpha_{TR}=TR_C/TR_D$

PGA_C: Limit capacity acceleration for each limit state (independent from the seismic spectrum).
PGA_D: Spectral acceleration for each of the limit states (depends on the seismic spectrum).
TR_C: Return period of the limit capacity seismic action for each of the limit states.
TR_D: Spectral return period for each of the limit states.

If the box **"Show PGA on rock"** is selected, all the (PGA) accelerations are measured,

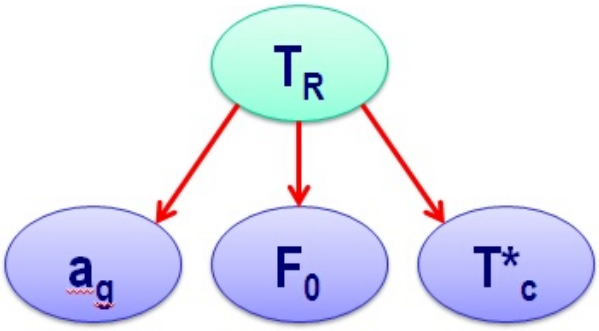
by convention, on *rigid ground (A)*.
Some data sheets require the calculation of these accelerations on the ground of reference, in this case you will need to multiply the calculated value by the program by the "S" ($S=S_S*S_T$) factor, defined in the spectrum parameters.
The product between acceleration and "S" factor is obtained by removing the check from "Show PGA on rock" button.

The return periods are those presented *in "Annex B"* of the *"Technical Standards"* (the definition of the reference grid). The parameters contained in the tables that define the reference grid can not be extrapolated, if the TRC values are outside the table the **">"** o **"<"** symbols are shown to indicate the exceeded of its upper or lower limit.

By selecting ***"Details ..."*** the following table is shown:

Vulnerabilità Sismica

					TR_C				TR=cost	
	TR_C	TR_D	α_{TR}	PGA_D [m/s2]	PGA_C (TR) [m/s2]	F_0 (TR)	T_C^*(TR)	α_{PGA}(TR)	PGA_C [m/s2]	α_{PGA}
SLV	189	475	0.398	2.56	1.82	2.31	0.32	0.712	1.86	0.728
SLD	71	50	1.420	1.02	1.19	2.32	0.29	1.172	1.19	1.172
SLO	71	30	2.367	0.77	1.19	2.32	0.29	1.544	1.17	1.515



$$\alpha_{PGA} = \frac{PGA_C(T_R)}{PGA_D}$$

$$\alpha_{PGA} = \frac{PGA_C}{PGA_D}$$

This table highlights two different procedures of α_{PGA} calculation which are based on two different theoretical starting assumptions, the choice is up to the designer according to which he considers most appropriate for the specific case. Both theories can be considered valid and reliable.
Remind that the three values for the definition of the (a_g, F_0, T^*_c) spectrum are calculated from the reference grid starting from a given value of the return period T_R.
The value of PGA_C on rigid ground corresponds to the value of a_g.
Method1 (TR=cost): Calculate PGA_C by varying the value of a_g until reaching the condition of the corresponding limit state, maintaining F_0 and T^*_c constants defined on the basis of the T_R value defined by the the seismic spectrum.
Method2 (TR_C): Calculate PGA_C by varying the value of T_R until reaching a tern of (a_g, F_0, T^*_c) values corresponding to the condition of the considered limit state.
The value of a_g thus calculated corresponds to required PGA_C.

Eurocode

Limit state	PGA [m/s2]	α
NC	14,976	7,488
SD	11,232	3,744
DL	3,199	1,600

The table for the evaluation of the *"Seismic Vulnerability"* reports the α parameters derived from homonyms reports for each of the limit states:

$\alpha_{PGA}=PGA/a_{gR}$

PGA: Limit capacity acceleration for each of the limit states (independent of the seismic spectrum).

Eurocode (NL)

NC

dt 0,02 [cm] <= dm 0,60 [cm]

qu = 0,43

Satisfied check

SD

dt 0,02 [cm] <= dm 0,45 [cm]

Satisfied check

DL

Sd 0,02 [cm] <= d*y 0,04 [cm]

Satisfied check

Limit state	α	
NC	30,000	} dm/dt
SD	22,500	
DL	2,314	d*y/Sd

The table for the evaluation of the *"Seismic Vulnerability"* reports the α parameters derived from homonyms ratios for each of the limit states derived by the ratio between the displacements.
Given the formulation of the spectrum defined in the NPR 9998/2015, it is clear that it is not possible to identify a vulnerability index on the basis of spectral accelerations will therefore be referred to vulnerabilities calculated as the ratio between the displacements.

In the area named "Analysis Parameters" appear the following factors:
T*: Period of the equivalent system
m*: mass of the equivalent system
W: total mass until the achievement of a tern of values

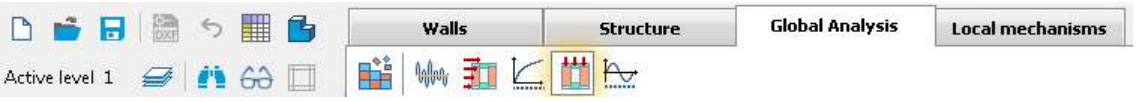
Available ductility: the ratio between the ultimate displacement and the elastic limit displacement
Γ: modal participation factor
F^*_y: plasticization strength of the equivalent system
d^*_y: plasticization displacement of the equivalent system
d^*_u: ultimate displacement of the equivalent system

10.6 Static Analysis

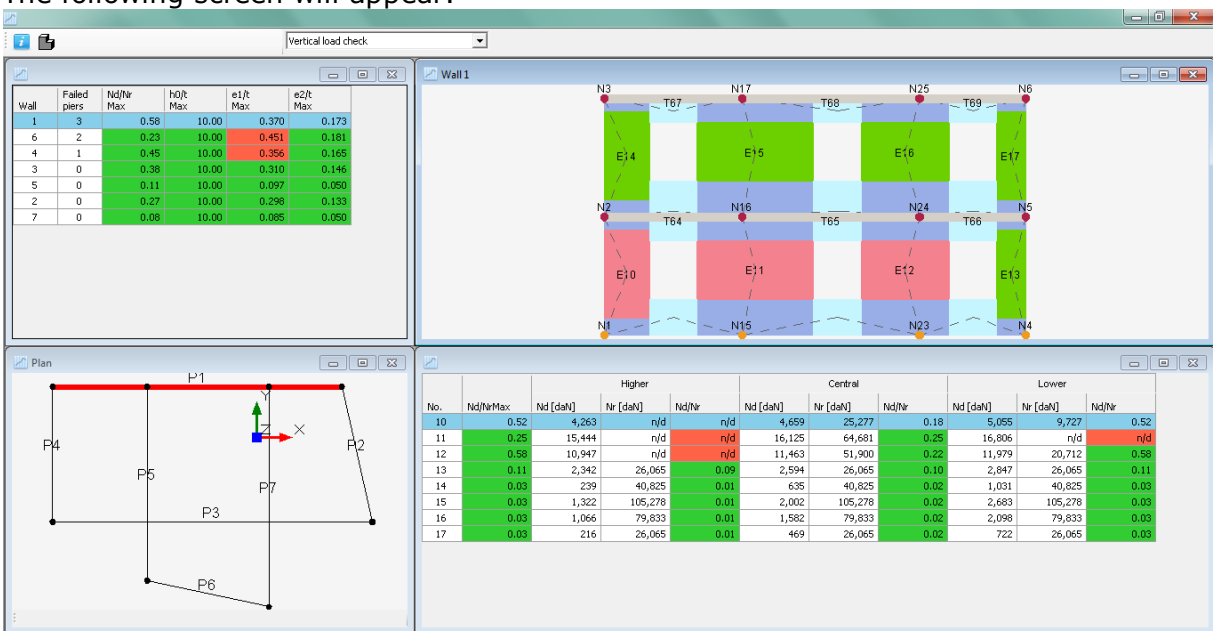
[OPTIONAL MODULE]

This is a module which performs static checks on the structure, according to the code in effect.
The program uses the meshes already created to perform the non-linear analysis, adapting the equivalent frame theory to perform the static checks in the linear field.

The static checks are performed in an area that is accessed using the associated button.



The following screen will appear:

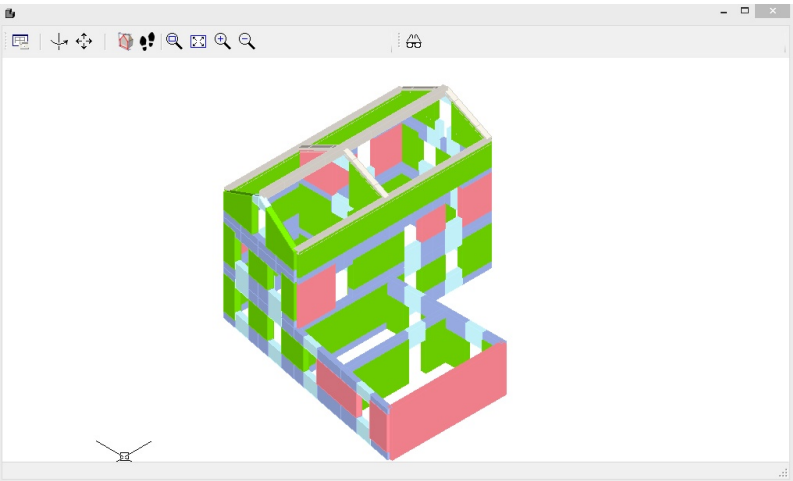


Wall	Failed piers	Nd/Nr Max	h0/t Max	e1/t Max	e2/t Max
1	3	0.58	10.00	0.370	0.173
6	2	0.23	10.00	0.451	0.181
4	1	0.45	10.00	0.356	0.165
3	0	0.38	10.00	0.310	0.146
5	0	0.11	10.00	0.097	0.050
2	0	0.27	10.00	0.298	0.133
7	0	0.08	10.00	0.085	0.050

		Higher			Central			Lower		
No.	Nd/NrMax	Nd [daN]	Nr [daN]	Nd/Nr	Nd [daN]	Nr [daN]	Nd/Nr	Nd [daN]	Nr [daN]	Nd/Nr
10	0.52	4,263	n/d	n/d	4,659	25,277	0.18	5,055	9,727	0.52
11	0.25	15,444	n/d	n/d	16,125	64,681	0.25	16,806	n/d	n/d
12	0.58	10,947	n/d	n/d	11,463	51,900	0.22	11,979	20,712	0.58
13	0.11	2,342	26,065	0.09	2,594	26,065	0.10	2,847	26,065	0.11
14	0.03	239	40,825	0.01	635	40,825	0.02	1,031	40,825	0.03
15	0.03	1,322	105,278	0.01	2,002	105,278	0.02	2,683	105,278	0.03
16	0.03	1,066	79,833	0.01	1,582	79,833	0.02	2,098	79,833	0.03
17	0.03	216	26,065	0.01	469	26,065	0.02	722	26,065	0.03

This video is very similar to that which presents the results of non-linear analysis. Here we describe it in detail.

If the user wishes, axonometric visualization can be used to find the elements that did not pass the check.



In order to help the user interpret the results, some of the tables offer the possibility of reordering the rows according to the column characteristics.

Info:
This module is available with the acquisition of the appropriate licence, the "Standard" version of the product contains no such form. For more information contact your distributor.

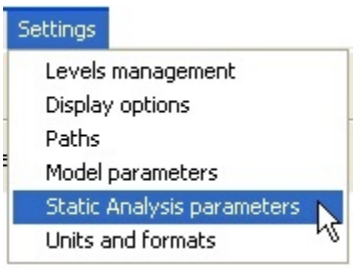
10.6.1 Italian Standart NTC08

Let's present below the checks that are carried out:

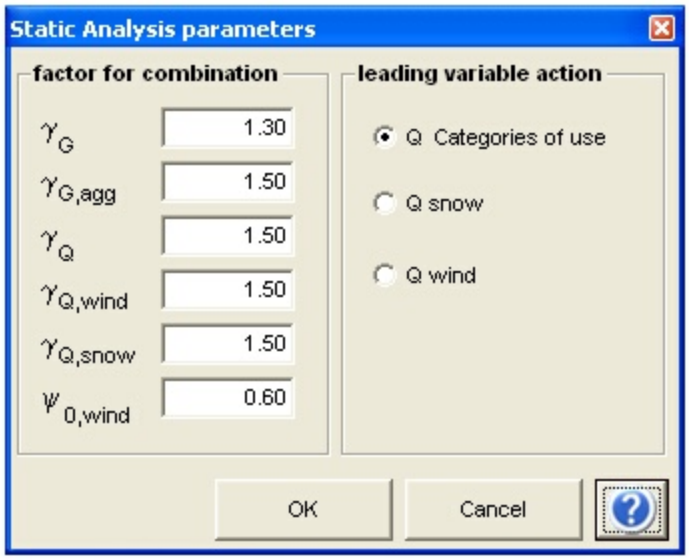
Slenderness check:	**h0/t≤20**
	h0: effective length of the wall equal to ρ•h *t: thickness of the wall*
Load eccentricity check:	$e_1/t \leq 0.33$ $e_2/t \leq 0.33$
	$e_1 = \lvert e_s \rvert + \lvert e_a \rvert$; $e_2 = \frac{e_1}{2} + \lvert e_v \rvert$ *es : total eccentricity of the vertical loads;* *ea : eccentricity due to execution tolerance;* *ev : eccentricity due to wind;*
Vertical loads check:	$N_d \leq \Phi\ f_d\ A$
	***Nd**: vertical load at the base of the wall;* ***A**: area of the horizontal section of the wall, after subtracting the openings;* ***fd**: computation resistance of the masonry;* ***Φ**: coefficient for wall resistance reduction*

If the active standards are the "Technical Standards for Construction 2008", the multiplication coefficients of the loads are editable using the command "Static calculation parameters "

Choosing from the Settings menu the item "Static analysis parameters", is possible to set static loads combinations.

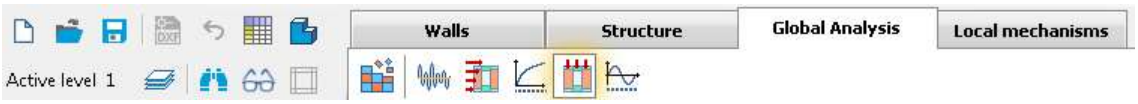


All combinations factors are impost in a parametrically and directly mode in the correspondent window.

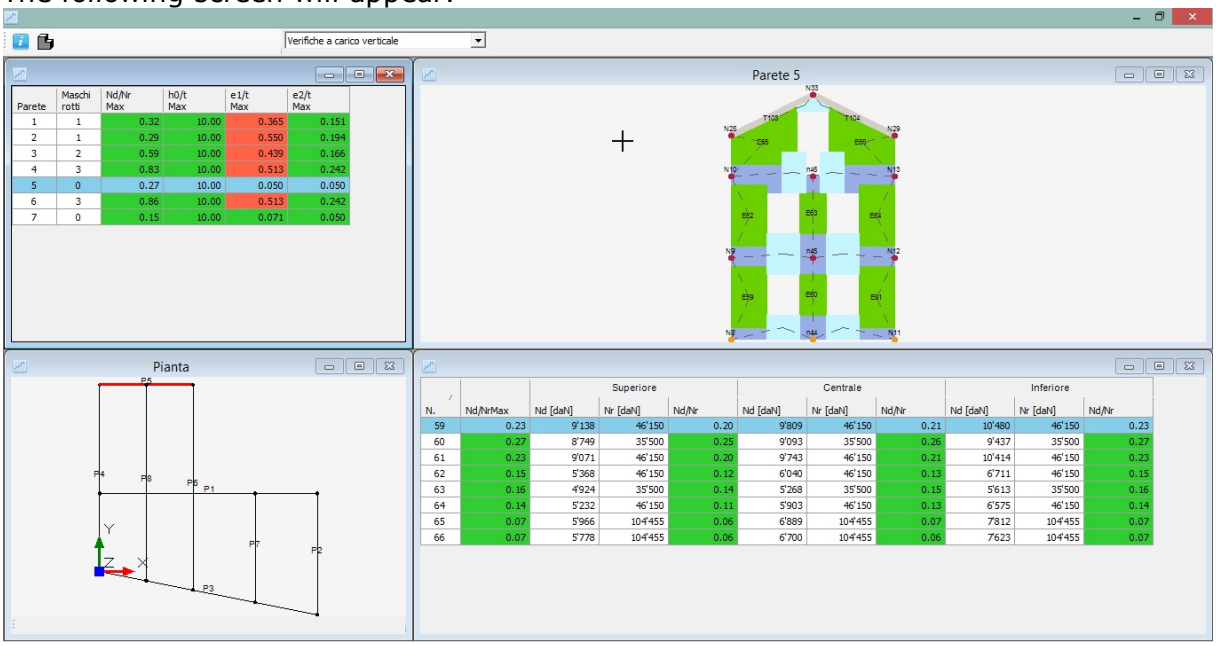


γ_G: factor for the permanent structural loads
$\gamma_{G,agg}$: factor for the permanent additional loads
γ_Q: factor for the live loads from the use destination of the building
$\gamma_{Q,vento}$:factor for the wind loads
$\gamma_{Q,neve}$: factor for the snow loads
$\Psi_{0,vento}$: factor for the wind loads

The static checks are performed in an area that is accessed using the associated button.



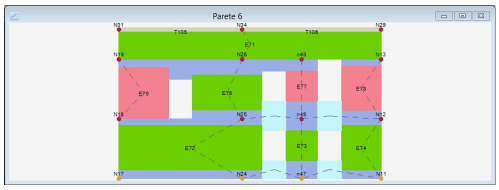
The following screen will appear:



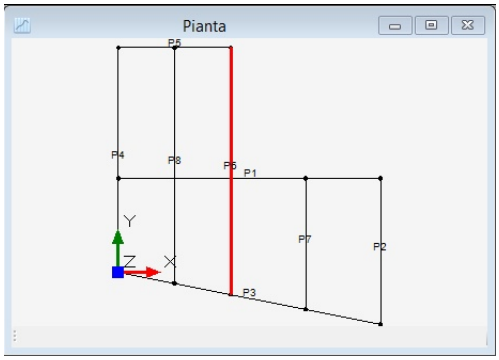
Parete	Maschi rotti	Nd/Nr Max	h0/t Max	e1/t Max	e2/t Max
1	1	0.32	10.00	0.365	0.151
2	1	0.29	10.00	0.550	0.194
3	2	0.59	10.00	0.439	0.166
4	3	0.83	10.00	0.513	0.242
5	0	0.27	10.00	0.050	0.050
6	3	0.86	10.00	0.513	0.242
7	0	0.15	10.00	0.071	0.050

		Superiore			Centrale			Inferiore		
N.	Nd/NrMax	Nd [daN]	Nr [daN]	Nd/Nr	Nd [daN]	Nr [daN]	Nd/Nr	Nd [daN]	Nr [daN]	Nd/Nr
59	0.23	9'138	46'150	0.20	9'809	46'150	0.21	10'480	46'150	0.23
60	0.27	8'749	35'500	0.25	9'093	35'500	0.26	9'437	35'500	0.27
61	0.23	9'071	46'150	0.20	9'743	46'150	0.21	10'414	46'150	0.23
62	0.15	5'368	46'150	0.12	6'040	46'150	0.13	6'711	46'150	0.15
63	0.16	4'924	35'500	0.14	5'268	35'500	0.15	5'613	35'500	0.16
64	0.14	5'232	46'150	0.11	5'903	46'150	0.13	6'575	46'150	0.14
65	0.07	5'966	104'455	0.06	6'889	104'455	0.07	7'812	104'455	0.07
66	0.07	5'778	104'455	0.06	6'700	104'455	0.06	7'623	104'455	0.07

Let's describe it in detail.



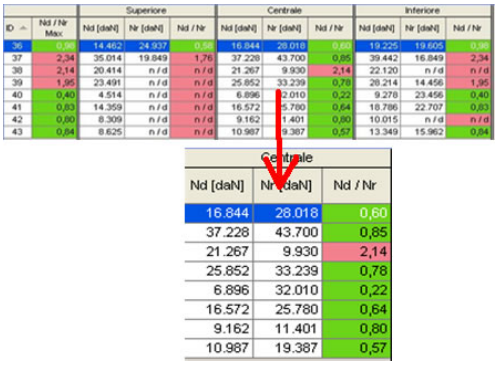
In the upper right side appears the wall mesh. In this case, does not exist the legend with colors indicating different phases of damage. Elements that passed the check appear in green and in different color those that do not exceed the check.



At the lower left side, is shown the plan view. The wall shown in the precedent view is highlighted with a thick line.

Parete	Maschi rotti	Nd / Nr Max	h0 / t Max	e1 / t Max	e2 / t Max
4	7	2,34	7,50	0,477	0,207
6	6	3,02	7,50	0,882	0,206
2	1	0,79	7,50	1,191	0,345
1	0	0,45	7,50	0,038	0,038
5	0	0,39	7,50	0,038	0,038
3	0	0,74	7,50	0,038	0,038

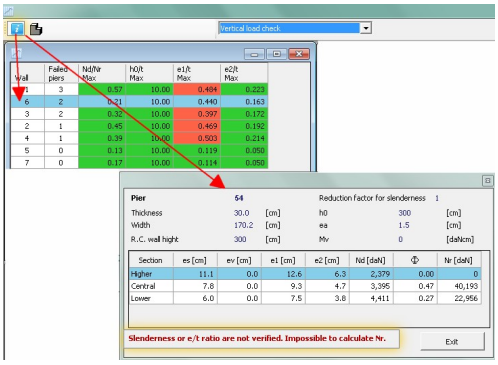
On the upper left side there is a list of the walls in the model, with the number of elements that did not pass the check and the values associated with the individual checks. The values found in the table are for the wall elements examined in which the limit values are the most restrictive of all the piers.
Clicking on the line of a wall (highlighting it in blue) brings that wall to the view on the right side.



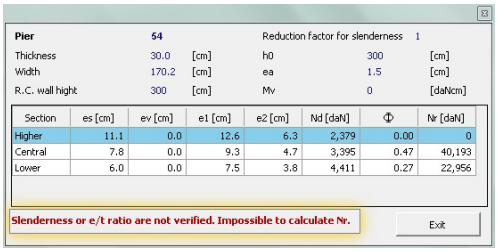
Nd [daN]	Nr [daN]	Nd / Nr
16.844	28.018	0,60
37.228	43.700	0,85
21.267	9.930	2,14
25.852	33.239	0,78
6.896	32.010	0,22
16.572	25.780	0,64
9.162	11.401	0,80
10.987	19.387	0,57

At the lower right side, the elements detail window is shown for the selected wall.
For each masonry element, the checks are performed for three different sections (higher, central, lower).
For each section the value for normal forces strain is shown (Nd: computed based on the masses and the combinations of the loads) and the normal resistant strain (Nr $=\Phi\ f_d\ A$).

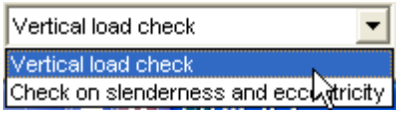
The check is satisfied if the ratio Nd/Nr≤ 1. In this case, the corresponding cell appears in green.
In some cases, as shown in the example, Nr cannot be calculated (n/d: not defined). This happens when the slenderness or eccentricity checks are not satisfactory.



When a masonry pier is chosen from the list and the information button is pressed, a window will appear which contains the calculation details.



The window shows all the details of the parameters used in the computation of the various check coefficients.
The text in red near the bottom gives relative informations to conditions where the check was not satisfied.
This window can remain open and be moved to any point of the drawing area while working (floating window). This gives to the user the possibility to select various elements in different wall and still have the details for each individual check visible.

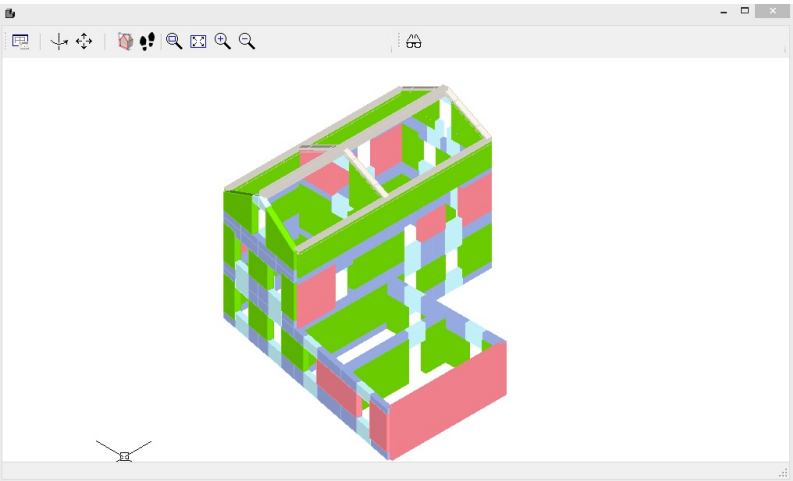


Through the associated menu on the results bar, it is possible to switch to visualization of the compression results from the slenderness and eccentricity results

		Higher	Central	Lower
ID	h0 /t	e1 /t	e2 /t	e1 /t
35	7,50	0,320	0,144	0,263
36	7,50	0,306	0,145	0,276
37	7,50	0,360	0,172	0,331
38	7,50	0,332	0,150	0,274
39	7,50	0,451	0,167	0,268
40	7,50	0,401	0,175	0,311
41	7,50	0,448	0,202	0,369
42	7,50	0,470	0,186	0,309

Here we see the check details for slenderness and eccentricity. The green values indicate that the check was passed.


If the user wishes, axonometric visualization can be used to find the elements that did not pass the check.



In order to help the user in the results interpretations, some of the tables offer the possibility of reordering the rows according to the column characteristics.

Wall check summary table

Parete	Maschi rotti	Nd / Nr Max	h0 / t Max	e1 / t Max	e2 / t Max
1	0	0,45	7,50	0,038	0,038
2	1	0,79	7,50	1,191	0,345
3	0	0,74	7,50	0,038	0,038
4	7	2,34	7,50	0,477	0,207
5	0	0,39	7,50	0,038	0,038
6	6	3,02	7,50	0,882	0,206

This table is ordered based on the wall identifiers. The type of orientation is clarified by the arrow found at the top of the column.

Parete	Maschi rotti	Nd / Nr Max	h0 / t Max	e1 / t Max	e2 / t Max
4	7	2,34	7,50	0,477	0,207
6	6	3,02	7,50	0,882	0,206
2	1	0,79	7,50	1,191	0,345
1	0	0,45	7,50	0,038	0,038
5	0	0,39	7,50	0,038	0,038
3	0	0,74	7,50	0,038	0,038

Clicking on the appropriate column will reorder the values according to the characteristics chosen. In the figure at the side, the table is ordered based on the number of failed elements.

Parete	Maschi rotti	Nd / Nr Max	h0 / t Max	e1 / t Max	e2 / t Max
6	6	3,02	7,50	0,882	0,206
4	7	2,34	7,50	0,477	0,207
2	1	0,79	7,50	1,191	0,345
3	0	0,74	7,50	0,038	0,038
1	0	0,45	7,50	0,038	0,038
5	0	0,39	7,50	0,038	0,038

...or it can be ordered based on what the check penalizes most.

Compression check details table

ID	Nd / Nr Max	Nd [daN]	Nr [daN]	Nd / Nr	Nd [daN]	Nr [daN]	Nd / Nr	Nd [daN]	Nr [daN]	Nd / Nr
1	0,38	24.834	111.564	0,22	33.733	111.564	0,30	42.633	111.564	0,38
4	0,45	7.160	22.917	0,31	8.743	22.917	0,38	10.327	22.917	0,45
5	0,05	-2.663	29.147	-0,09	-647	29.147	-0,02	1.369	29.147	0,05
6	0,18	1.985	44.626	0,04	5.071	44.626	0,11	8.156	44.626	0,18
7	0,21	2.998	44.626	0,07	6.083	44.626	0,14	9.168	44.626	0,21

This table can also be reordered.
It is ordered based on the wall identifiers and the efficiency of the compression load (only the global one for the element, and not for individual sections).

The current formulation that leads to the calculation of the eccentricity es2 considers in the calculation of the N2 (load coming from the overlying floors) in addition to its contribution also the respective d2 eccentricity value for all of the levels higher than that under consideration.

Info:

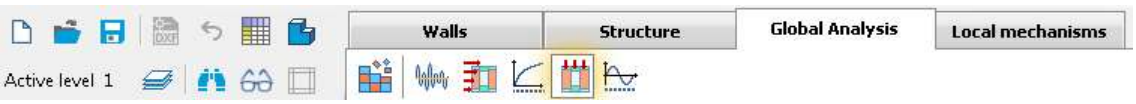
This module is available with the acquisition of the appropriate licence, the "Standard" version of the product contains no such form. For more information contact your distributor.

10.6.2 Eurocodice 6

Let's present below the checks that are carried out:

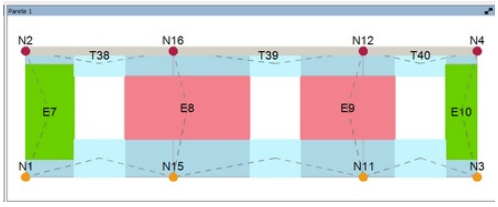
Slenderness check: *EN 1996-1-1 § 5.5.1.4*	$\mathbf{h_{ef}/t_{ef} \leq \lambda_{lim}}$: *di default=27*
	h_{ef}: *effective height of the wall equal to* $\rho \bullet h$ t_{ef}: *effective thickness of the wall equal to* $\rho_t \bullet h$
Verification subjected to vertical loading *EN 1996-1-1 § 6.1.2*	$\mathbf{N_{Ed} \leq N_{Rd} = \Phi\ f_d\ A}$
	N_{Ed}: *design value of the vertical load applied to a masonry wall;* ***A****: is the loaded horizontal gross cross-sectional area of the wall;* ***fd****: design compressive strength of the masonry;* ***Φ****: is the capacity reduction factor*

The static checks are performed in an area that is accessed using the associated button.

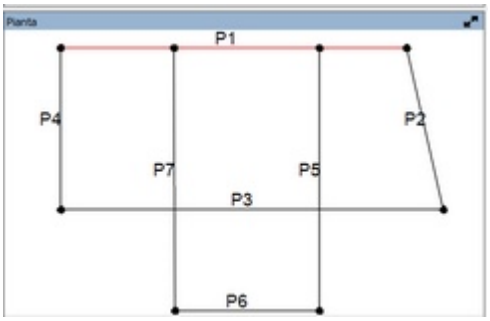


The following screen will appear:

This video is very similar to that which presents the results of non-linear analysis. Let's describe it in detail.



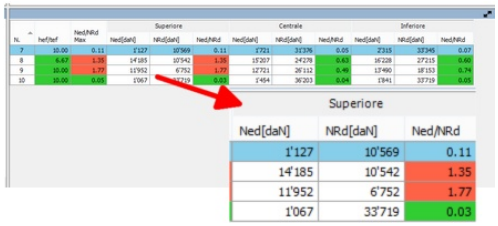
In the upper right side appears the wall mesh. In this case, does not exist the legend with colors indicating different phases of damage.
Elements that passed the check appear in green and in different color those that do not exceed the check.



At the lower left side, is shown the plan view. The wall shown in the precedent view is highlighted with a thick line.

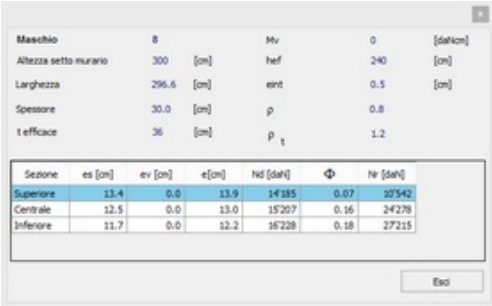
Parete	Maschi rotti	Ned/NRd Max	hef/tef Max
1	2	1.77	10.00
2	0	0.18	10.00
3	0	0.15	10.00
4	0	0.21	10.00
5	0	0.06	10.00
6	0	0.27	10.00
7	0	0.06	10.00

On the upper left side there is a list of the walls in the model, with the number of elements that did not pass the check and the values associated with the individual checks. The values found in the table are for the wall elements examined in which the limit values are the most restrictive of all the piers.
Clicking on the line of a wall (highlighting it in blue) brings that wall to the view on the right side.



At the lower right side, the elements detail window is shown for the selected wall.
For each masonry element, the checks are performed for three different sections (higher, central, lower).
For each section the value for normal forces strain is shown (N_{Rd}: computed based on the masses and the combinations of the loads) and the normal resistant strain (N_{Rd} **=Φ f_d A)**.
The check is satisfied if the ratio $N_{Ed}/N_{Rd} \leq 1$. In this case, the corresponding cell appears in green, otherwise in red color.
In some cases, as shown in the example, N_{Rd} cannot be calculated (n/d: not defined). This happens when the slenderness or eccentricity checks are not satisfactory.

When a masonry pier is chosen from the list and the information button is pressed, a window will appear which contains the calculation details.



The window shows all the details of the parameters used in the computation of the various check coefficients.
The text in red near the bottom gives relative informations to conditions where the check was not satisfied.
This window can remain open and be moved to any point of the drawing area while working (floating window). This gives to the user the possibility to select various elements in different wall and still have the details for each individual check visible.

		Higher	Central	Lower
ID	h0 /t	e1 /t	e2 /t	e1 /t
35	7,50	0,320	0,144	0,263
36	7,50	0,306	0,145	0,276
37	7,50	0,360	0,172	0,331
38	7,50	0,332	0,150	0,274
39	7,50	0,451	0,167	0,268
40	7,50	0,401	0,175	0,311
41	7,50	0,448	0,202	0,369
42	7,50	0,470	0,186	0,309

Here we see the check details for slenderness and eccentricity. The green values indicate that the check was passed.

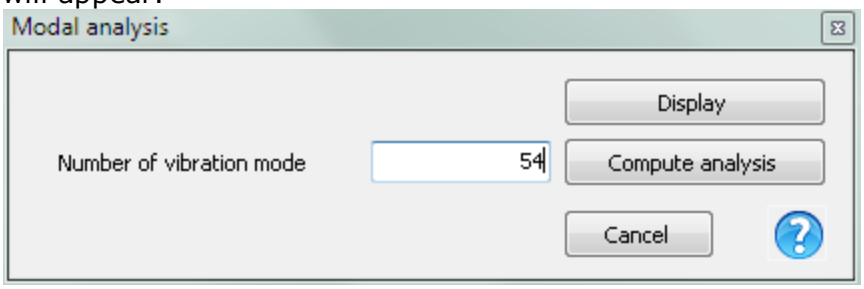
In order to help the user in the results interpretations, some of the tables offer the possibility of reordering the rows according to a characteristic shown in a column by simply clicking on the title of the column.

Info:
This module is available with the acquisition of the appropriate licence, the "Standard" version of the product contains no such form. For more information contact your distributor.

10.7 Modal Analysis

This is an area dedicated to computation of modal forms and the parameters associated with them.

When the appropriate button found in the analysis bar is pushed, the following window will appear:

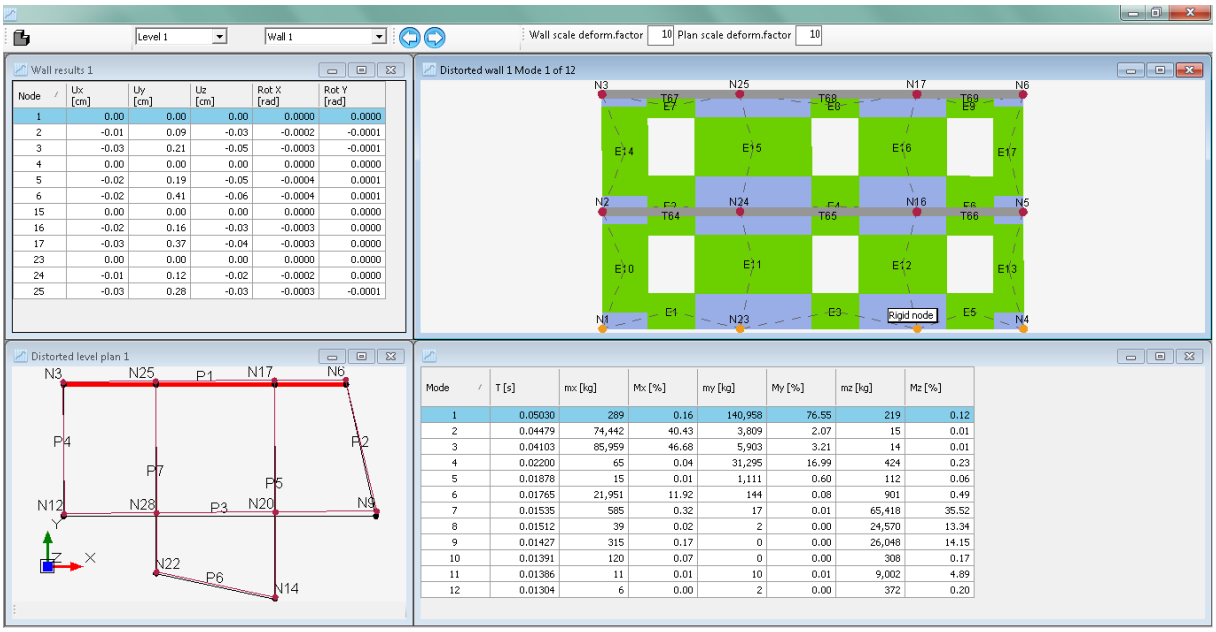


A number of predefined modal forms are offered.

If the computation has already been performed, the results are shown.

Compute analysis The computation is performed, and the results are shown.

When the calculation is finished, the presentation of the results is automatically shown.



Node	Ux [cm]	Uy [cm]	Uz [cm]	Rot X [rad]	Rot Y [rad]
1	0.00	0.00	0.00	0.0000	0.0000
2	-0.01	0.09	-0.03	-0.0002	-0.0001
3	-0.03	0.21	-0.05	-0.0003	-0.0001
4	0.00	0.00	0.00	0.0000	0.0000
5	-0.02	0.19	-0.05	-0.0004	0.0001
6	-0.02	0.41	-0.06	-0.0004	0.0001
15	0.00	0.00	0.00	0.0000	0.0000
16	-0.02	0.16	-0.03	-0.0003	0.0000
17	-0.03	0.37	-0.04	-0.0003	0.0000
23	0.00	0.00	0.00	0.0000	0.0000
24	-0.01	0.12	-0.02	-0.0002	0.0000
25	-0.03	0.28	-0.03	-0.0003	-0.0001

Mode	T [s]	mx [kg]	Mx [%]	my [kg]	My [%]	mz [kg]	Mz [%]
1	0.05030	289	0.16	140,958	76.55	219	0.12
2	0.04479	74,442	40.43	3,809	2.07	15	0.01
3	0.04103	85,959	46.68	5,903	3.21	14	0.01
4	0.02200	65	0.04	31,295	16.99	424	0.23
5	0.01878	15	0.01	1,111	0.60	112	0.06
6	0.01765	21,951	11.92	144	0.08	901	0.49
7	0.01535	585	0.32	17	0.01	65,418	35.52
8	0.01512	39	0.02	2	0.00	24,570	13.34
9	0.01427	315	0.17	0	0.00	26,048	14.15
10	0.01391	120	0.07	0	0.00	308	0.17
11	0.01386	11	0.01	10	0.01	9,002	4.89
12	0.01304	6	0.00	2	0.00	372	0.20

In the table at the lower right, a list of modal forms is shown.

The table appears in this way:
Mode: Numeric identifier for the modal form
T[s]: Fundamental period
mx[kg]: Participating mass direction X
Mx[%]: Percentage of participating mass direction X
my[kg]: Participating mass direction Y
My[%]: Percentage of participating mass direction Y
mz[kg]: Participating mass direction Z
Mz[%]: Percentage of participating mass direction Z

If a single line from the table is selected, deformation of the wall and the plan is shown for the corresponding mode.

10.8 Local Mechanisms Analysis

[OPTIONAL MODULE]

In the existing masonry buildings are often missing systematic linking elements between walls, at the level of the floors,which means a possible vulnerability towards of local mechanisms, that can affect not only the collapse out of the plane of individual wall panels, but more extensive portions of the building.

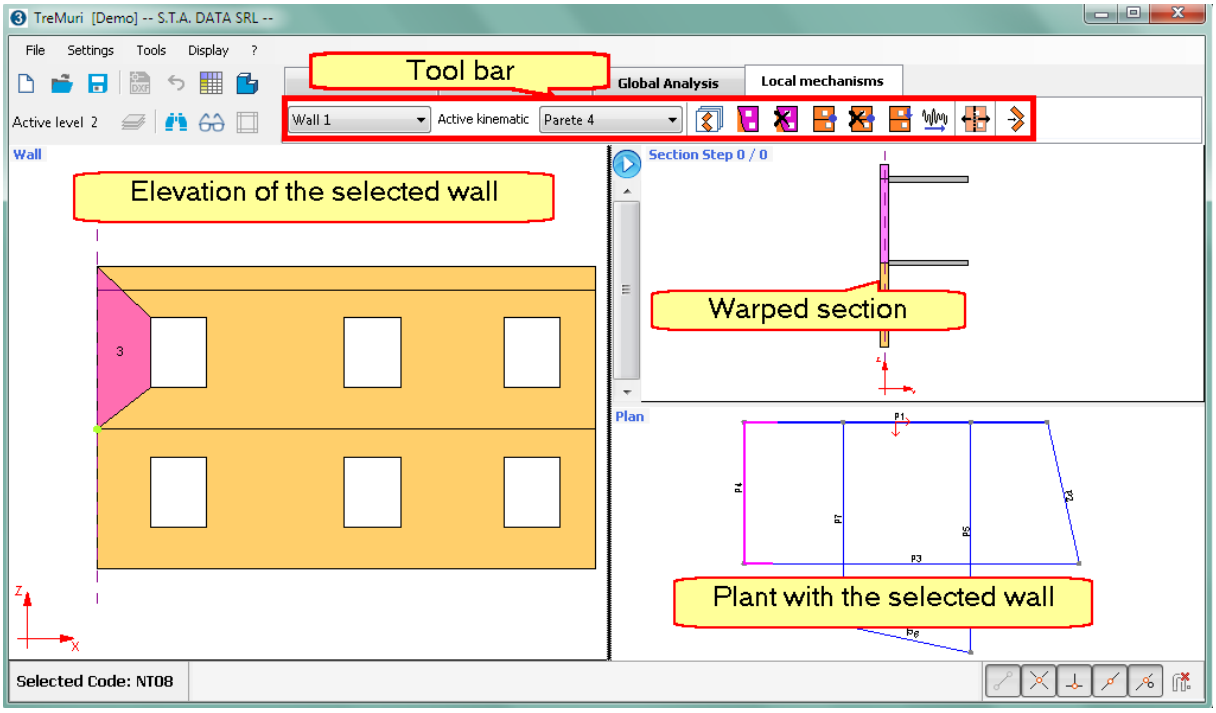
"Tremuri LM" is a calculation module inside the Tremuri program, which is dedicated to the evaluation of the building safety against such mechanisms.
The module **"Tremuri LM"** exploits the versatility and the input ergonomics of the program TreMuri to finalize a spatial model on which the user can investigate the possible mechanisms.

Before proceeding with the local mechanisms verification through **"Tremuri LM"** it is necessary:

➔To create the spatial model of the structure, the same that is used to perform the global and statics verifications through the "*Walls*" and "*Structure*" setting.

➔ Compute model Mesh through the "*Analysis*" setting
➔ Insert the parameters of seismic spectrum through the "*Analysis*" setting

The image below shows the contents of the toolbar of local mechanisms.



ATTENTION!!!!

All the data input generated on the Local Mechanisms setting will be erase automatically with the regeneration of the Mesh!!!
To conserve the local mechanisms already defined, save a copy of the model before proceeding with the generation of the mesh.

Info:
This module is available with the acquisition of the appropriate licence, the "Standard" version of the product contains no such form. For more information contact your distributor.

*** Bibliography:**

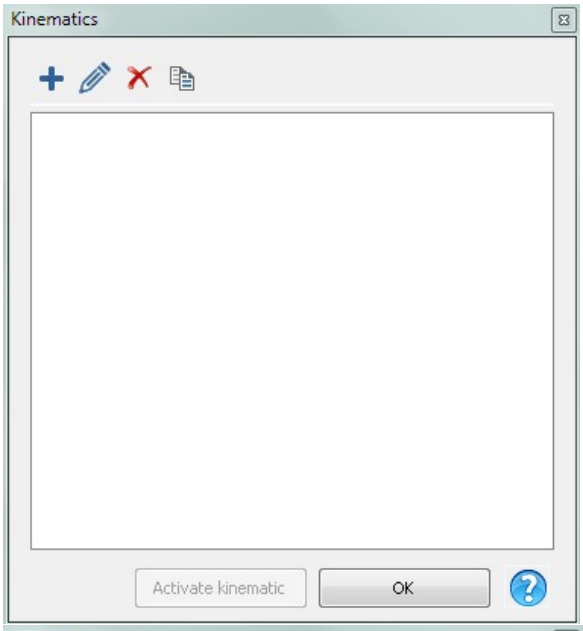
Beolchini G. C., Milano L., Antonacci E. (A cura di). *Repertorio dei meccanismi di danno, delle tecniche di intervento e dei relativi costi negli edifici in muratura – Definizione di modelli per l'analisi strutturale degli edifici in muratura*, Volume II – Parte 1a. Convenzione di

Ricerca con la Regione
Marche; Consiglio Nazionale delle Ricerche – Istituto per la Tecnologia delle Costruzioni – Sede di
L'Aquila; Dipartimento di Ingegneria delle Strutture, delle Acque e del Terreno (DISAT) – Università
degli Studi di L'Aquila. L'Aquila, 2005.

10.8.1 Mechanisms input

After generating the mesh and inserting the seismic load it is possible to introduce the mechanisms that want to examine.

Kinematics:
Pressing this button shows the window that allows to select the mechanisms containing in the "archive".

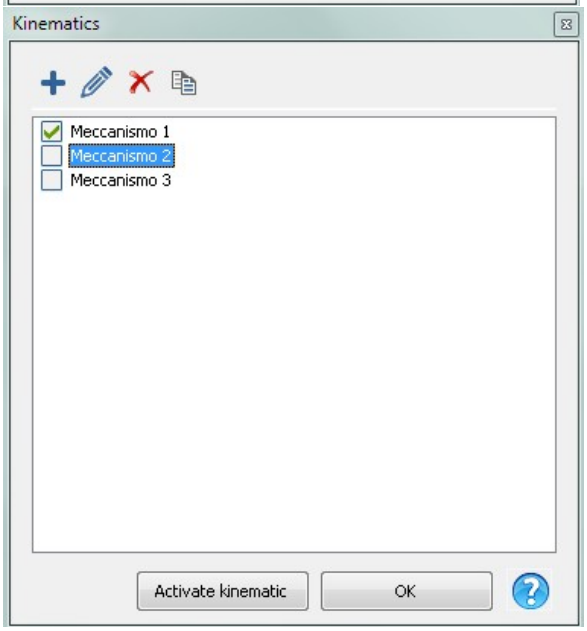


New Mechanisms (enter the name of the mechanism)

Modify mechanisms'name

Delete Mechanism

Duplicate Mechanism



After the introduction of new mechanisms to be examined, they are appended to the list below with the name that you want to insert.

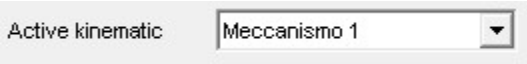
The "Kinematics" presented are like "containers" that can hold in their internal any kind of mechanism (tilting, bending, etc. ..). The examined type of mechanism will be generated based on input made during the creation phase of the kinematic, for example based on the type of constraints that want insert.

Activate kinematic

Used to activate one of the "kinematics containers", indicating on which kinematic decide to work.

The active mechanism is represented by checking the box (☑) to the left of the name.

Confirming with OK the window closes and displays the name of the active kinematic combo box "active kinematics" shown in the toolbar.

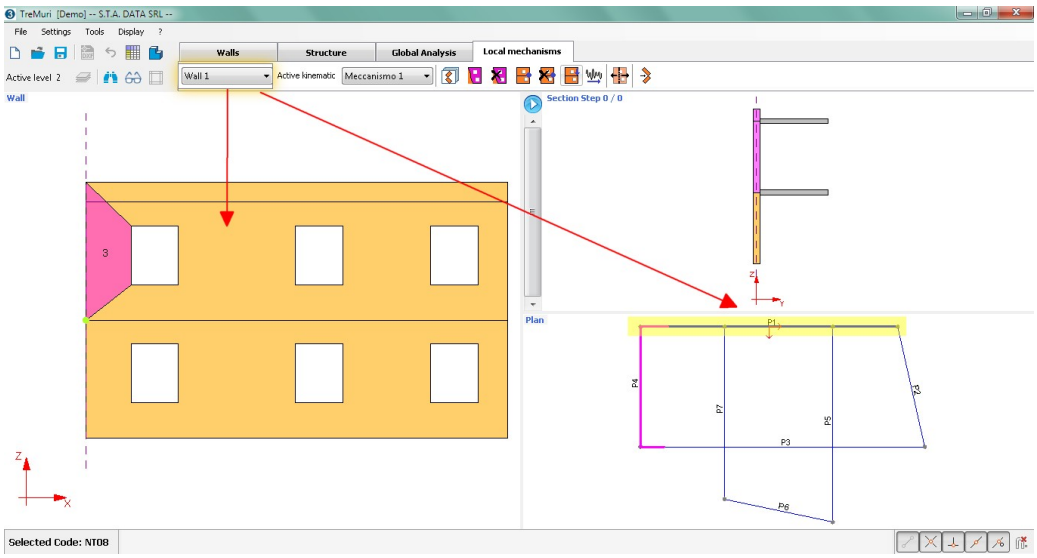


It is possible to use this combo box to change the active kinematic.

10.8.2 Mechanisms definition

The toolbar of the **"Local Mechanisms"** setting, allows the definition of a single mechanism.

By selecting a curtain wall from the combo box, is shown the front of the selected wall and on plan highlighted in bold.



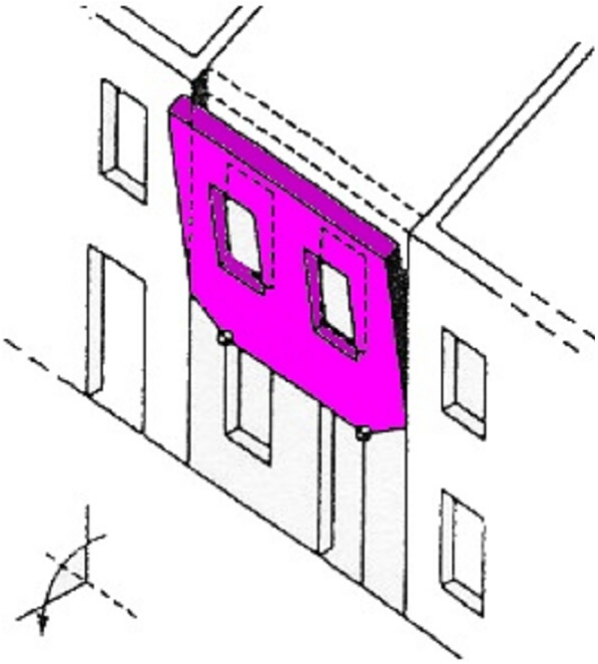
The mechanism input consist in three steps:

➔ Inserting **Kinematics Blocks**

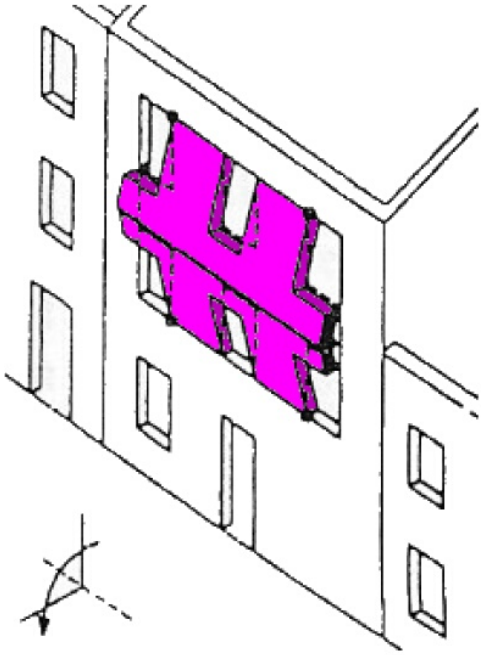
➔ Inserting **Vincoli Constraints**
➔ Inserting **Loads**

10.8.2.1 Kinematics blocks

Kinematic Block means a part of masonry considered "infinitely rigid" on kinematic terms, subject to a movement of tilting respect another block or to the rest of the wall. The image below (*) shows two examples of kinematic blocks.



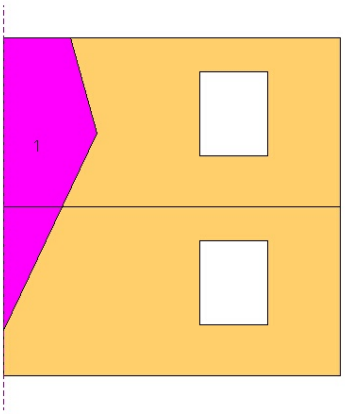
Example of a mechanism consisting of a single block



Example of a mechanism consisting of two blocks

Insert block:
It allows to enter the surface of the block by defining a closed polygon.
Pressing the button, the mouse pointer becomes sensitive to the graphics of the selected wall front activating the snap at the present nodes and lines.
To close the polygon on the first apex, press the right mouse button.

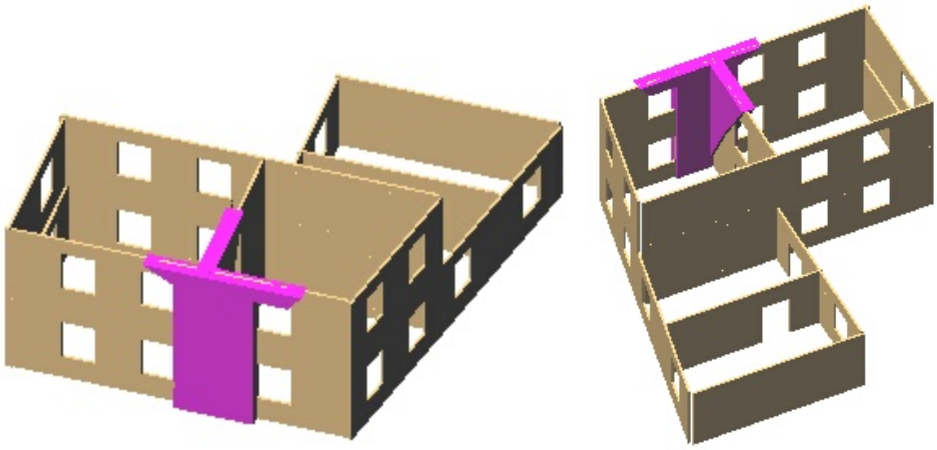
Here is an example of cinematic block defined on 4 apex.



The possibility to draw a closed perimeter allows the user to trace the edges in correspondence of the panel crack found in site.

(*)

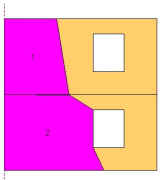
Each single kinematic can contain any number of kinematics blocks in the same and different walls.
The image below shows a drawn system block based on the visible cracks of the structure.
A portion of the masonry of the wall plug (wedge) participate in the tilting of the perimeter wall.



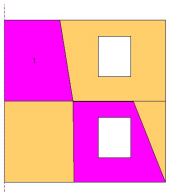
Axonometric views such as those described above are visible by pressing the "3D View" button .

Different blocks in the same kinematic must be connected together through the constraints.
The absence of constraints implies that two blocks are linked together in a rigid mode. To ensure that this is true, it is fundamental that the delimitated areas by the two blocks have at least two points in common.
For example, the case of the image above shows two blocks from two different walls, where is given the absence of constraints along the intersection of the blocks, it generates an overall behavior like two blocks formed one unique body.

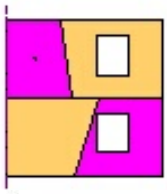
Therefore apply the following construction rules:



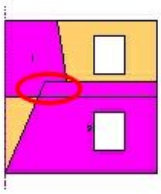
CORRECT
Blocks 1 and 2 have a common side



NOT CORRECT!!!!
The blocks 1 and 2 have only one common side.



NOT CORRECT!!!!
The blocks 1 and 2 have no common point

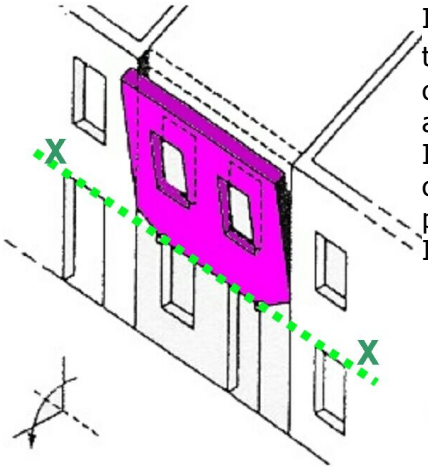


NOT CORRECT!!!!
The blocks 1 and 2 are overlapped

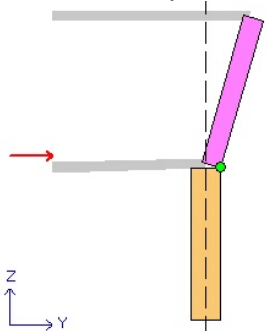
Delete blocks:
Selecting one or more blocks in sequence, confirming by pressing the right mouse the selected blocks are deleted.

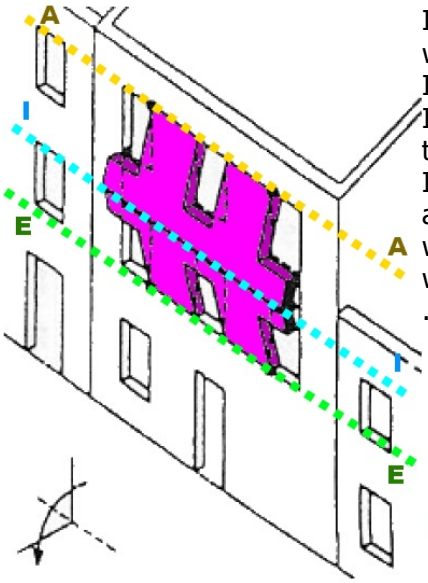
10.8.2.2 Constraints

The kinematics blocks do not have any default constraint .
The constraint conditions must be specified in an appropriate mode depending on the mechanism type that would like to examine.

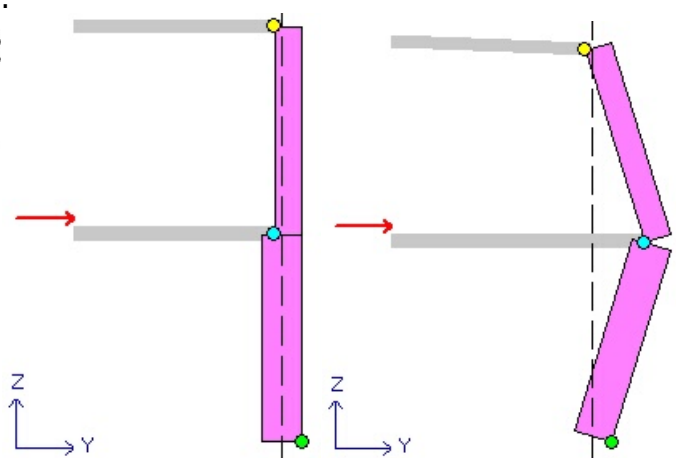


If we want to consider a tilting case of a wall portion like the one represented in the image (*), the considered case is the case of a block that rotates around the X-X axis .
In correspondence to this point you must enter a constraint between a kinematic block and a fixed wall portion.
In this case you must insert the **"External Hinge"**

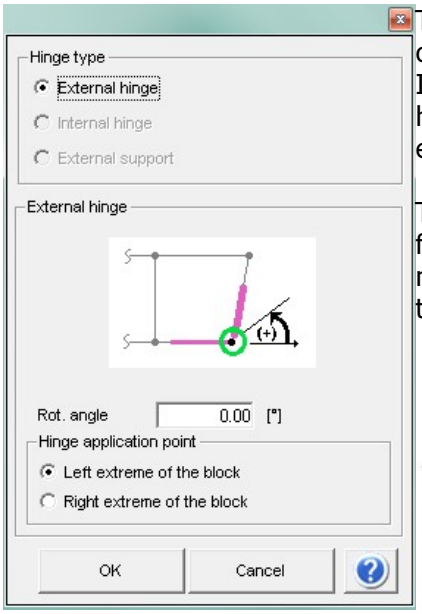




In this case (*) the bottom block is placed directly on a wall portion that is not deformed .
In the **E-E** position will be put the **"External Hinge"**
In the **I-I** position confines two blocks, so will be put the **"Internal Hinge".**
In the **A-A** position the deformation mechanism will not allow any movement out of the plan. The points of this wall can only move vertically in the plane of the wall, so will be put a **"Support"**.
.

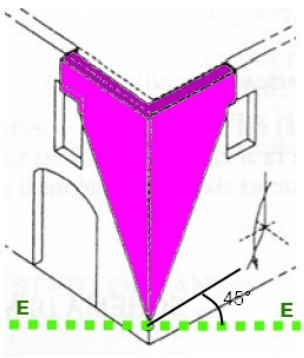


Insert constraint:
Pressing this button will display the input window of constraints.



The various constraint types can be inserted only in the order they are presented.
If you want to insert an internal hinge or a support you have to insert already an external hinge (there is no equilibrium static scheme if there is no external hinge).

The "Angle" box means the angle that the external hinge form with the active wall. When the angle is zero, this means that the rotation axis of the constraint is parallel to the wall.



This function can be useful to examine tilting cases of the cantonal. (*)

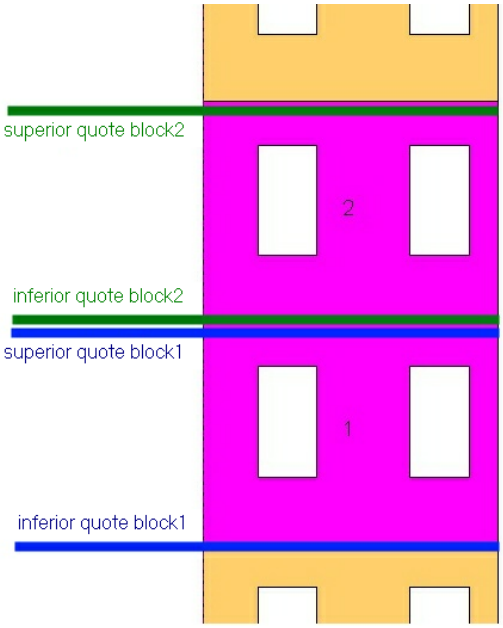
The insertion of each type of constraint occurs after the definition of the blocks. The

constraints are always positioned at the points of maximum and minimum elevation of the block.
So we can deduce the necessity to insert the constraints relatively of each block.

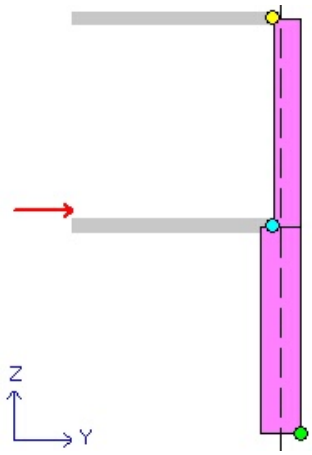
After selecting the constraint type and pressing the OK button it is necessary to select the reference kinematic block.

Clicking on the block, the constraint is inserted to the:
- **Bottom** quote of the block if **External Hinge**
- **Top** quote of the block if **Internal Hinge**
- **Top** quote of the block **if Support**

The quote concept "Top" or "Bottom" is relative on the block and is explained in the image on the right.



If the angle of the constraint is zero, its axis is contained in the plane of the selected block wall. The axes of the constraints are also the axes around which rotate the blocks, this means that in this case it is assumed that the **earthquake direction** is perpendicular to the shown wall.
The **sisma verse** is indicated by an red arrow in the section shown on the right of the screen.



If you want to change the sisma verse you can use the appropriate button shown to the left of the section.

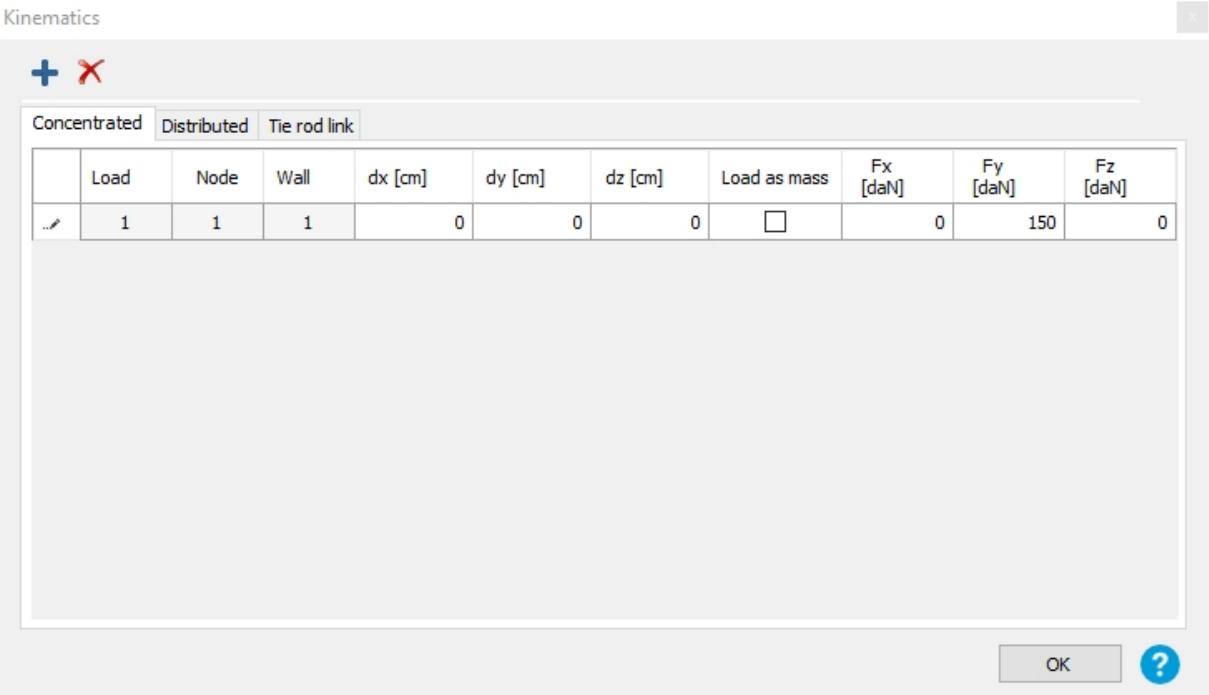
In the section view, the constraints are represented with colored circles with the corresponding colors to each type of constraint at the wire fixed inside or outside depending on where you generate in physical mode the rotation point for the defined mechanism and for the assigned sisma verse.

Delete constraint:
Allows to remove a constraint.

10.8.2.3 Loads

It is possible to insert additional loads on Kinematic blocks caused by:
Pre-stress value of the tie rod, Vault Push, loads from the structural elements that impact directly on the Kinematic block, etc. ..

Pressing the "Loads" button will appear the dialog window like the image below:

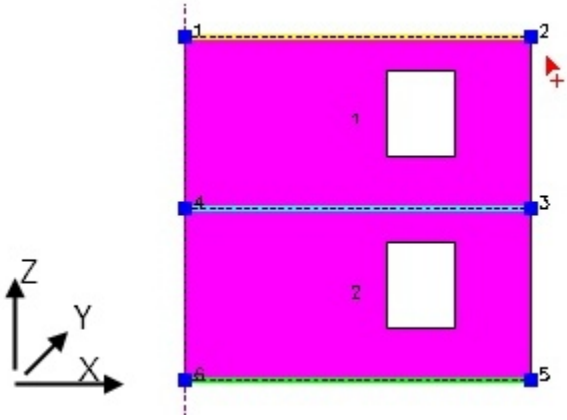


The buttons named **"Concentrated"** and **"Linear"** allow to put a concentrated load or linearly distributed; depending on the enabled button is shown the table with the list of loads already placed on the considered kinematics. The first time the table is clearly empty.

The item "**Tie rod link**" allows to define a load coming from an inserted tie rod in order to prevent the activation of the mechanism. Entering the tie rod load in this table allows you to pass the pre-stress value to the Tie Rod link verification module.

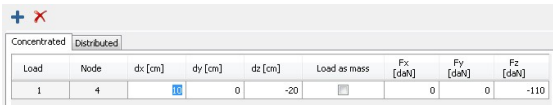
Insert new load:

Pressing the button, the snap becomes "selection snap" ( ) and on the wall front are shown selected nodes.



Select the node nearest to the point where you want insert the load.
After the selection, will appear the dialog window like the image below.

Concentrated load

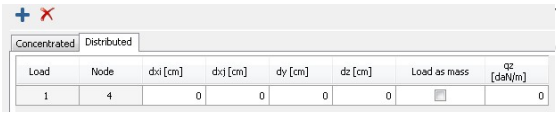


Concentrated | Distributed

Load	Node	dx [cm]	dy [cm]	dz [cm]	Load as mass	Fx [daN]	Fy [daN]	Fz [daN]
1	4	10	0	-20		0	0	-110

Node: Indicate the number of the selected node in graphical modality.
Force: If you insert a vertical load (Fz) different from zero, this load does not generate mass, without creating any horizontal component of seismic type (α*Fz; α: factor of vertical load).
Mass: If you insert a vertical load (Fz) different from zero, this load generates mass, creating a horizontal component of seismic type.
The loads applied to the mechanism in "indirect" mode, for example when the loads came from the superior wall are usually considered forces but not masses.
dx/dz: are the relative coordinates of the load application point, compared to a node centered system.
Fx / Fy / Fz: are components of the force in the system wall.

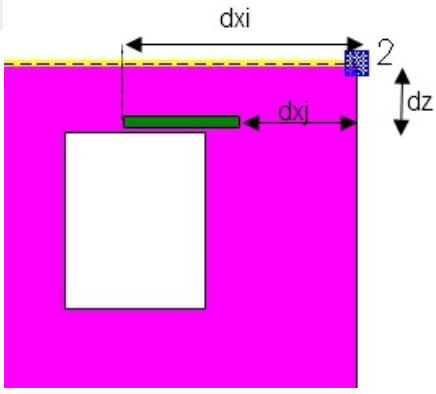
Distributed load



Concentrated | Distributed

Load	Node	dxi [cm]	dxj [cm]	dy [cm]	dz [cm]	Load as mass	qz [daN/m]
1	4	0	0	0	0		0

qz: distributed vertical load .
dxi / DXJ / dz: are the relative coordinates of the load application points compared to a node centered system.



Confirming the inserting of the load, the table is updated with as many rows as there are loads included.

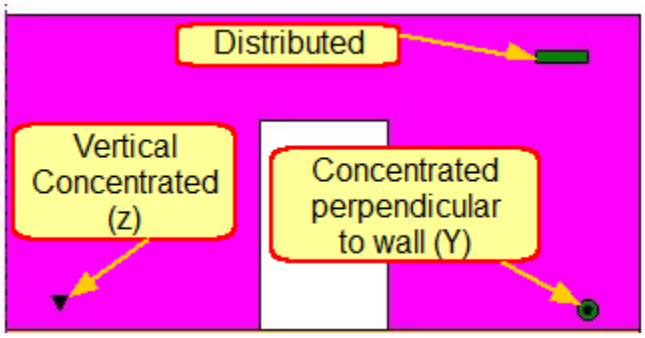
Concentrated | Distributed

Load	Node	dxi [cm]	dxj [cm]	dy [cm]	dz [cm]	Load as mass	qz [daN/m]
3	4	0	0	0	0	☐	0

It is not allowed to edit directly the numbers in this table, to edit these values you must select the row and press the "**Edit**" button .

Delete: allow to remove the load corresponding to the selected line.

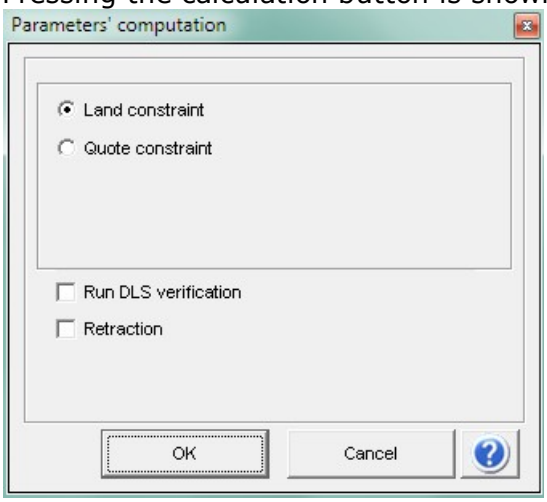
The loads are shown in the below graphic with the following agreement.



10.8.2.4 Calculation

When the input is complete, you can proceed with the calculation.
With the module **"3Muri LM"** is possible to run the Verification of the Linear Kinematic Analysis.

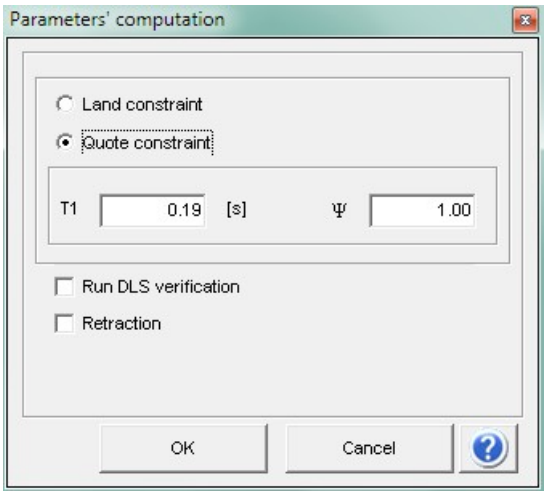
Pressing the calculation button is shown the following dialog window:



Select a **Land constraint** where the verification is for a single element or a portion of the building that still rests on the ground.

Select a **Quote constraint** where the local mechanism interest a portion of the building at a certain quote.

In this case, the calculation window will show some additional calculation parameters.



T1 is the first period of vibration of the whole structure in the considered direction. The default value is calculated using the simplified formula according to the Design Code.

$T_1 = C_1 \cdot H^{3/4}$ assuming C1 = 0050 and H: height of the building

A more accurate calculation can be derived from the modal analysis of the structure.

ψ is the first vibration mode in the considered direction, standardized at a summit of the building, in the absence of more accurate valuation is assumed ψ = Z / H, where H is the height of the structure regard to the foundation..

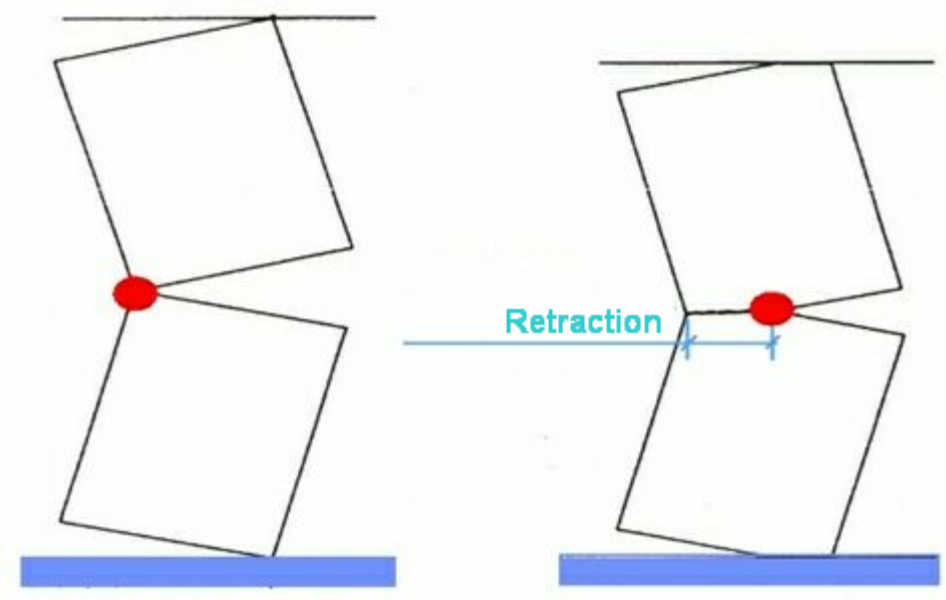
Z is the height, compared to the foundation of the building, the center of gravity of the constraint lines between the blocks interested by the mechanism.

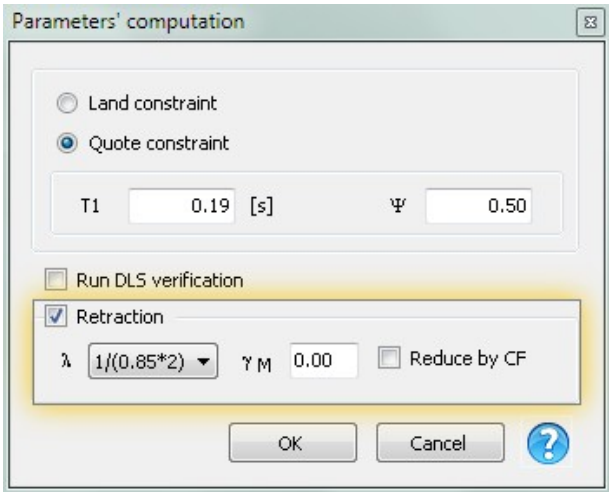
The box **"Run verification DLS"** allows you to verify the Damage Limit State. Normally this box is not selected because this verification is not required.

The box **"Retraction"** manages the retraction of the hinge.

The hinges of the mechanism constitute the points about which the various blocks rotate relative to one another.

The point of rotation depends on the compressive strength of the masonry. In the case in which the resistance is infinite, the center of rotation coincides with the edge at the base. In the case of limited compressive strength, the rotation center of the kinematic will be positioned within the thickness of the wall.





λ: Constant of calculation of the retraction function (recommended = 1/(0.85*2))
γ_M: Security factor of the resistance of the material (recommended = 1)
Reduce by CF: application of the reduction to take into account from the Confidence Factor (recommended = NOT selected)

The retraction is identified by the relation

$$a=\lambda\frac{P}{fd\,L}$$

a: retraction
P: total vertical load acting on the hinge
L: length of the hinge
fd: design strength

The design strength fd is detected by the appropriate formula:

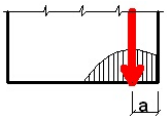
$$f_d=\frac{f_m}{F_C\cdot\gamma_M}$$

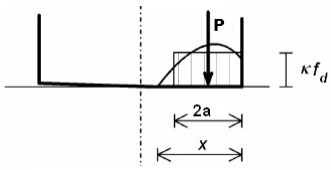
fm: average compressive strength

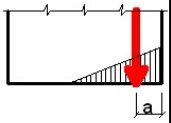
γ_M: Security factor of the strength of the material (suggested value = 1)

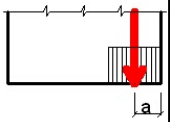
in case of linear analysis such safety factor must be set equal to 2.
Actually the NTC do not comment the value that has to be attributed to this parameter for the case of retraction calculation of the hinge in a linear kinematic analysis. The retreat of the hinge is different from the analysis (called "linear kinematics").
The retraction of the rotation hinge corresponds to the physical phenomenon of edge breakage of the wall and therefore it is a "non-linear" phenomenon and the safety coefficient of the materials can reasonably be set equal to 1.

CF: Confidence Factor of the material
The NTC do not comment on the need to consider the CF in the calculation of design resistance, some techniques bibliographies consider it in the calculation.
It is important to remember that the confidence factor is already introduced in the formula that permits the acceleration calculation of the activation mechanism.
Consider it also in the calculation of the design resistance is equivalent to consider it twice. It is therefore recommended that you do NOT reduce by CF.

<table>
<tr>
<td> 

λ=1/(0.85*2)
RECOMMENDED!</td>
<td>This value of the calculation constant has its origins in the theory of vertical bending which is based on the stress-block method on which it bases the origins of the constitutive connection for bending (NTC § 7.8.2.2.1 regulations).


In this case, from equilibrium at vertical-traverse it is possible to write:
$$P = k \cdot f_d \cdot L \cdot 2a$$
The length of retraction is then half the length of the stress-block equivalent, thus appears to be equal to:
$$a = \frac{P}{2 \cdot k \cdot f_d \cdot L} = \frac{1}{2 \cdot k} \cdot \frac{P}{f_d \cdot L} = \lambda \cdot \frac{P}{f_d \cdot L}; \quad \lambda = \frac{1}{2 \cdot k}$$
K: Used in the bending formula of NTC § 7.8.2.2.1 regulations assumed is equal to 0.85.
<u>Reference:</u>
Galasco e Frumento, ANALISI SISMICA DELLE STRUTTURE MURARIE,
E127 - Progetto Costruzione Qualità, Gruppo Editoriale SIMONE</td>
</tr>
<tr>
<td> 

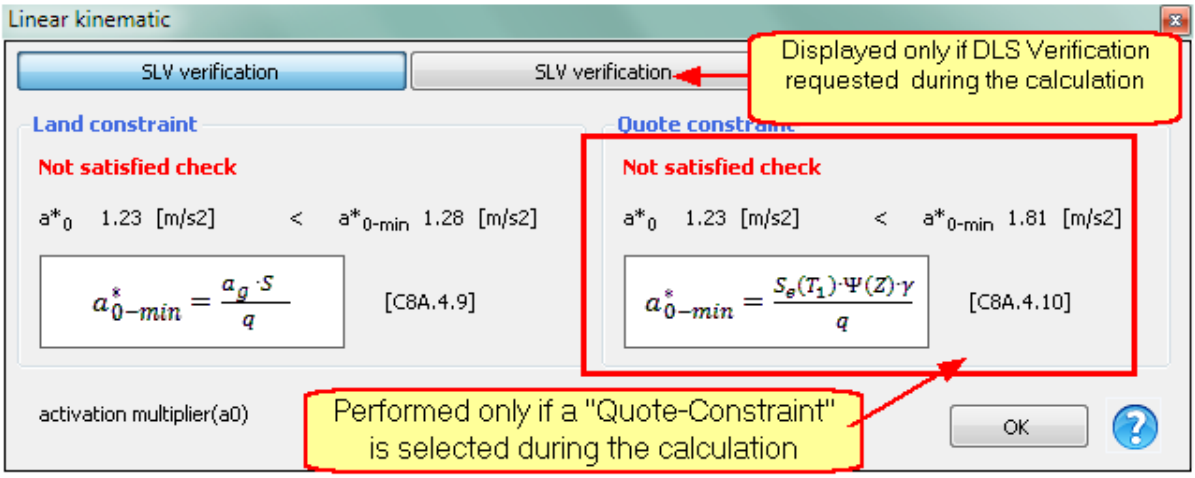
λ=2/3</td>
<td>This value has its origins from a hypothesis of triangular distribution of the tension.</td>
</tr>
<tr>
<td> 

λ=1/2</td>
<td>This value has its origins from a hypothesis of rectangular (constant) distribution of the tension.</td>
</tr>
<tr>
<td>More...</td>
<td>In this case, it's possible to enter any multiplier that the user think is appropriate</td>
</tr>
</table>

Pressing the OK button the calculation is performed.

10.8.2.5 Results

The results dialog window appears when the calculation is complete.
Pressing "SLV Verification" or "SLD Verification" shows the corresponding results.



a*0: The spectral seismic acceleration of the activation of the mechanism
ag: function of the probability of exceeding the selected Limit State and the reference life
S: is the coefficient that takes into account the soil type and the topographical conditions
q: structure factor

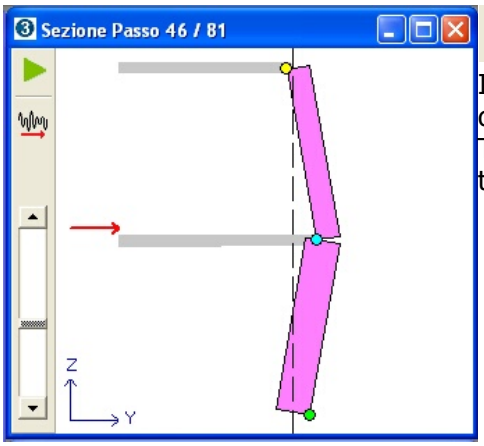
Se(T1): elastic spectrum, function of the probability of exceeding the selected Limit State (in this case 63%) and the reference period as VR, calculated for the period T1;
ψ (Z): is the first vibration mode in the considered direction, standardized at a summit of the building, in the absence of more accurate valuation is assumed ψ = Z / H, where H is the height of the structure regard to the foundation.
γ : modal coefficient participation (in the absence of more accurate valuation can be taken γ = 3N / (2N +1) with N number of floors of the building).

In the case of:

- **Land constraint** should only be conducted the verification with simplified structure factor q (linear kinematic analysis)
- **Quote Constraint** should be conducted both calculations (the verification with simplified structure factor q and the verification taking into account that the spectrum of response is related to the probability of exceeding of 10% over the

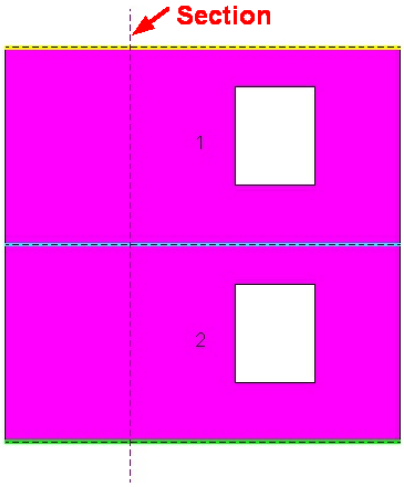
reference period VR.

When the calculation is done, appear the window that shows the section, and allows to see a motion movie with the deformity progressive of the section.



"Play" button:
It allows you to start the motion movie showing the deformation evolution of the structure.
The vertical scroll bar allows you to place in any of the intermediate steps of the movie.

The deformed section is drawn in a precise point on the wall front.



The section is represented in the wall front by a vertical underscore line.


The button "move section line" allows you to replace the section line by clicking a point in the graphics area.

10.9 Sensitivity Analysis

[OPTIONAL MODULE]

The Sensitivity analysis is a calculation method aimed to obtain better understanding of the structural functioning and accurate planning of the site investigation plan.
As known, doubts during modeling directly affect the evaluation of seismic safety. A specific example is materials mechanical properties, usually defined on the basis of reference values and for which, through investigation, it aims to limit the inescapable uncertainty.

Since the site tests have frequently high economic cost, the possibility to identify in advance (through the Sensitivity analysis) significant testing campaign points can limit investigation costs which result might not be of interest.
This kind of analysis can reduce the current uncertainty level on the structures analysis through the evaluation of the importance level of every parameter for which you want to investigate the significance.
A sensitivity index allows focusing the investigation only where it is necessary.
The methodology includes the identification of parameters groups that express the uncertainty degree, through the execution of multiple different non-linear analysis it identifies a level of sensitivity for each parameter in order to furnish a weight in terms of importance.

The main aim of the sensitivity analysis, to be executed through a sequence of nonlinear static analysis (pushover), is to identify the aleatory parameters that most affect the seismic capacity of the building.
There must be especially performed 2N + 1 analysis:

- the first by taking as a reference for the aleatory parameters the reasonable mean value
- in the other 2N analyzes all these parameters are maintained at their central value (average) of the range, with the exception of a parameter (or group of parameters) for which is taken the lower or upper limit of the above range.

The execution of pushover analysis implies the choice of different combinations and load conditions, in relation to:
1) forces distribution (for example proportional to the masses, derived from a triangular deformed shape or proportional to the first modal shape);
2) seismic action direction (X or Y);
3) the direction of this action (positive or negative);
4) the accidental eccentricity in each direction (usually defined in the regulations as 5% of the maximum size of the building in the orthogonal direction to that of the analysis).
Although in the final safety check all the different options should be considered (as explicitly required by the regulations), in order to limit the computational effort of the sensitivity analysis it can be selected the worst condition (corresponding to the minimum vulnerability index) by comparing the resulting capacity assuming plausible averages.

Bibliographic references:

CNR-DT212 / 2013: Istruzioni per la valutazione affidabilistica delle sicurezza sismica costruzioni esistenti, Consiglio Nazionale delle Ricerche, Roma.

ANIDIS / 2015: L'incompleta conoscenza nella valutazione sismica di edifici esistenti:
definizione del fattore di confidenza attraverso analisi di sensibilità .
Serena Cattari, Jamil Haddad, Sergio Lagomarsino

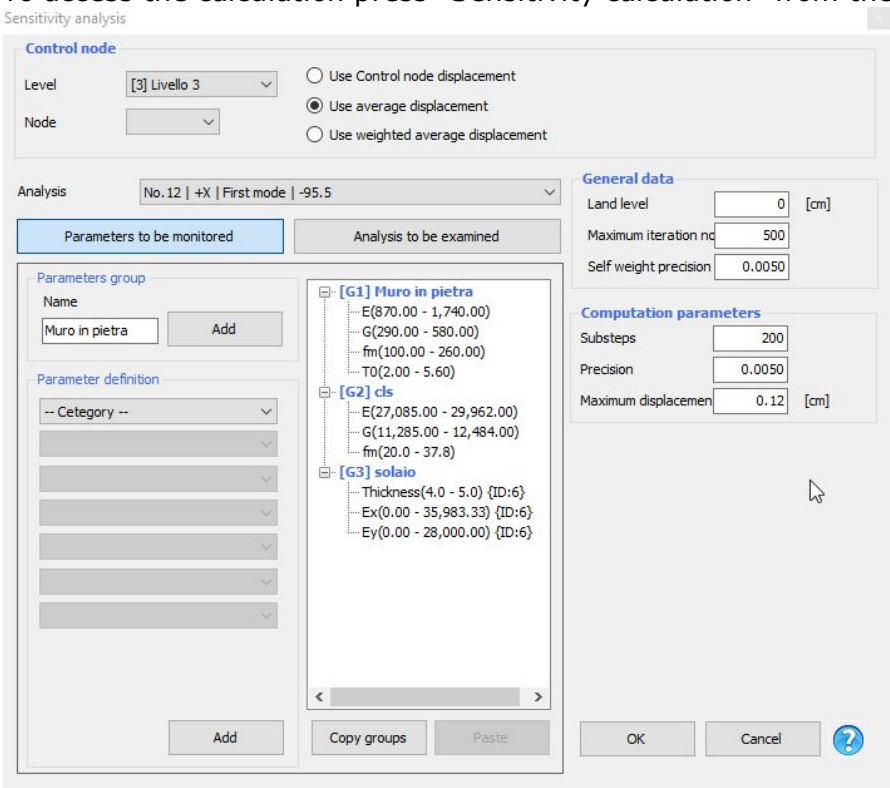
Bull Earthquake Eng: Sensitivity analysis for setting up the investigation protocol and de?ning proper con?dence factors for masonry buildings.
S. Cattari · S. Lagomarsino · V. Bosiljkov · D. D'Ayala

Info:
This module is available with the acquisition of the appropriate licence, the "Standard" version of the product contains no such form. For more information contact your

distributor.

10.9.1 Calculation

To access the calculation press "Sensitivity calculation" from the drop down menu.

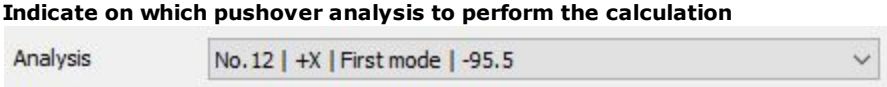


Input phases of the parameters to be monitored:

- Specify the pushover analysis in which to perform the calculation
- Insert groups [G1], [G2], [G3]..etc... according to which characteristics are to be examined
- Select the name of the group on the menu on the right in order to indicate the group in which to add a parameter
- Use the session "Define parameter" in order to define the parameter in the selected group

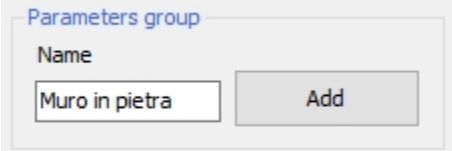
Following details of the various stages

Indicate on which pushover analysis to perform the calculation



Create a new group:

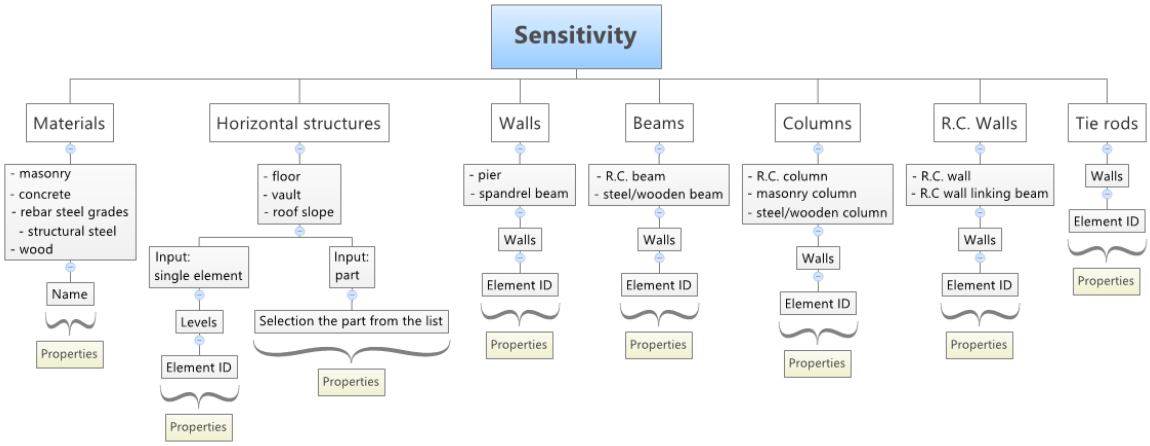
Enter the name in the provided space and press [Add]

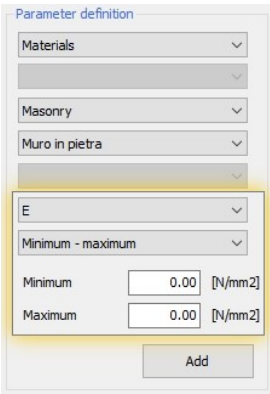


On the right is displayed on the blue menu the inscription **[G<*n*>] <*GroupName*>**
<*n*>: Identification number of the group created automatically by the program
<*GroupName*> : The name of the group inserted during the input phase

Parameter definition

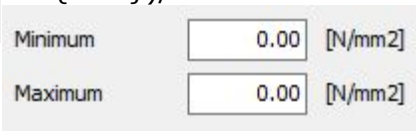
To define a parameter you must first select a previously created group in order to define the belonging group.
A series of lists that must be filled in order from top to bottom to indicate the parameter type.
Choosing an option in the first list activates a second list and so on for subsequent lists.
The diagram below summarizes all possible options that can be created by progressively loading the necessary parameters.



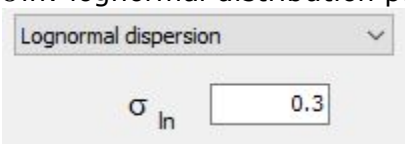


Following the first lists that are used to locate the item on which to intervene, a list shows the parameters subject to sensitivity (in the example of the figure on the left it has been decided to take action on the E parameter).

The last list allows to define two different ways through which can be specified the variation range of that parameter:
Minimum-maximum: The variation range is defined by entering the minimum and maximum value.
If X is a generic parameter, it is assumed X{average}= (X{min} + X{max})/2



Log-normal dispersion: X{average} corresponds to the characteristic entered in the model.
X{min}=X{average}/ $e^{\sigma ln}$
X{max}=X{average} $e^{\sigma ln}$
σln: lognormal distribution parameter not symmetric



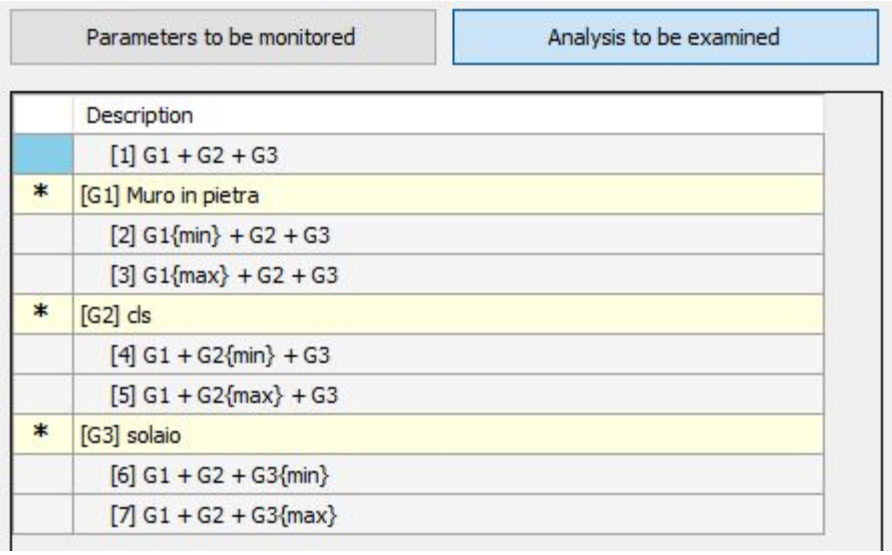
[Add] button:
Once selected the edited parameter and defined the variation range, by pressing the appropriate button will be added the parameter to the selected group.

P.S.:

Immediately after pressing [Add], the lists that have led to the selection of the newly defined parameter remain active, allowing you to call up a different parameter from the last list without having to redefine all of the above and be able to immediately insert a new parameter to the group.
This feature proves to be extremely useful to define different parameters in the same group linked by common characteristics.
Example: Insert in sequence various mechanical properties of the same material

Analysis to be examined

After defining groups and parameters you can select "Analysis to be examined" to switch to the list of analyzes to be carried out created according to the defined groups.

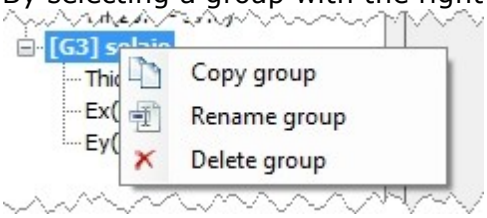


The analysis is identified by a numeric identifier shown in square brackets [] .
The analysis [1] is created with all the parameters set to their average value in the defined range.
Subsequent analyzes are grouped in pairs in the parameters group therefore the parameters are defined in the min or max range.

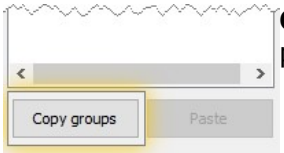
Edit parameters

Edit groups

By selecting a group with the right mouse button it appears the edit commands menu.



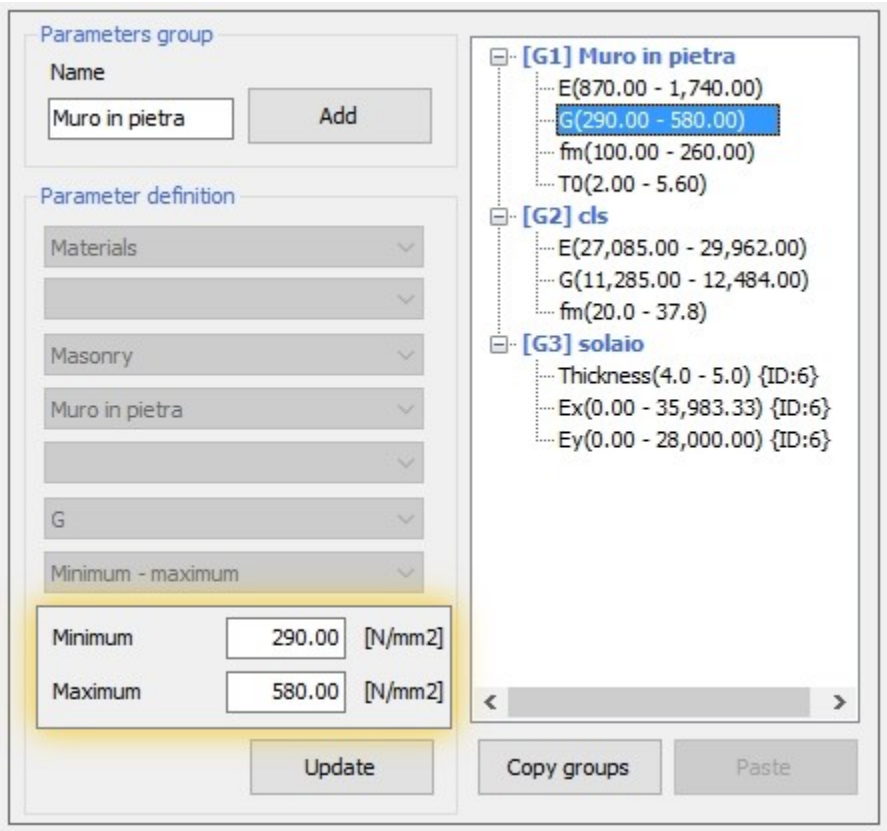
Copy group: Allows copying a group that can be pasted within another analysis. If you decide to monitor different analysis with the same group of parameters, following the "copy" command it can be pasted without manually re-entering the parameters.
Rename group: Allows changing the group name
Delete group: Deletes a group with all the parameters contained in it.

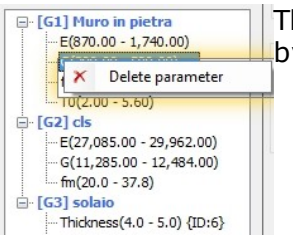


Copy groups: Allows copying all the groups of a predetermined pushover analysis for pasting them into another.

Edit parameters

Selecting a parameter from the menu, the "parameter definition" session shows the sequence that led to defining the characteristics of the selected parameter, allowing to figure out at which wall, floor or element it refers.
It is clearly not possible to edit the process that has led to define this parameter after the insertion but it is allowed to edit ranges where the parameter in question may vary.





The command through which can be canceled a parameter appears by selecting it with the right mouse button.

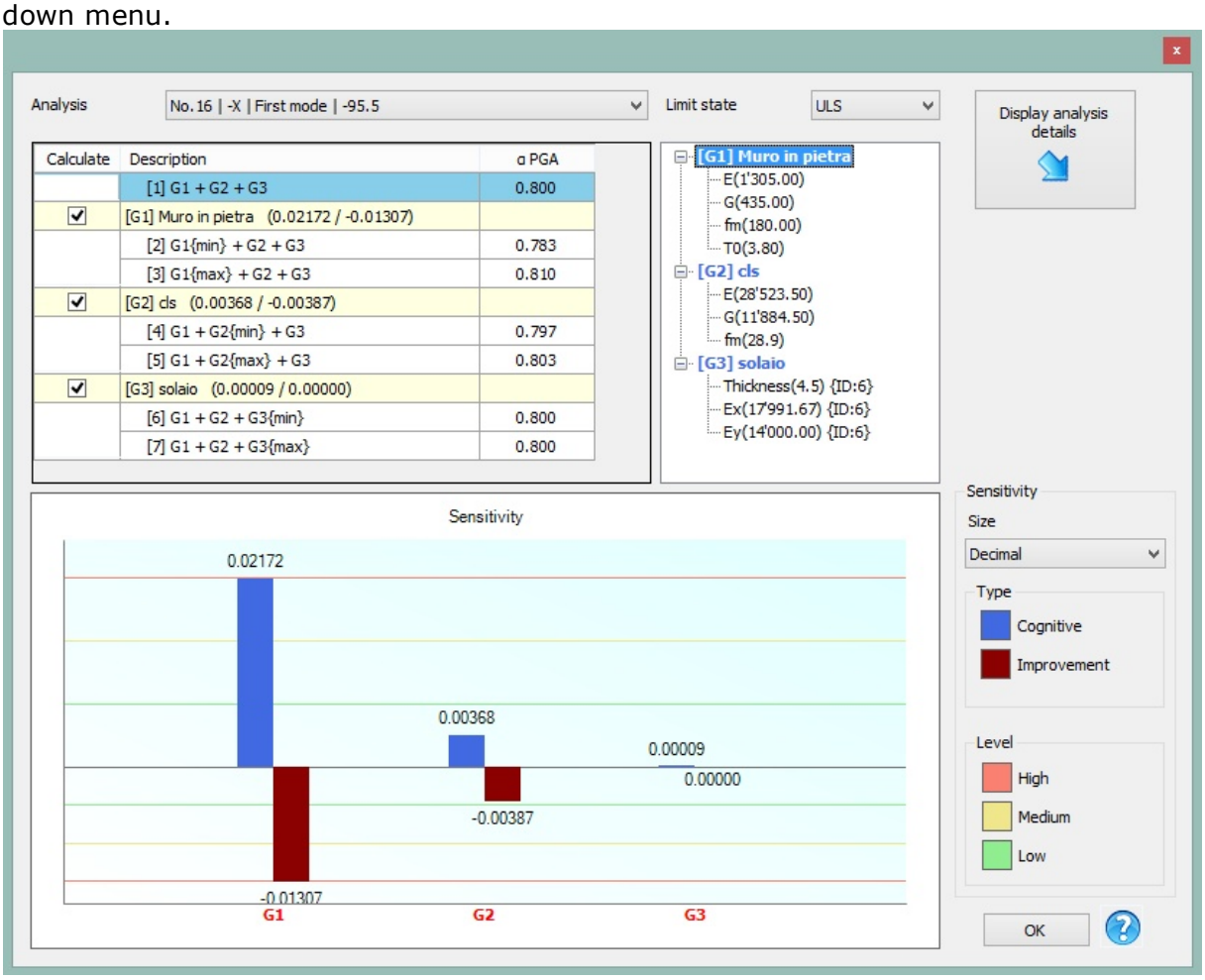
Press [OK] to proceed with the sensitivity calculation (perform in cascade the analysis of the list).
A dedicated window poses the question "Do you want to run the selected analysis?"
[Yes]: The calculation begins

[No]: Exits without performing any calculations and the parameters defined in the groups are saved

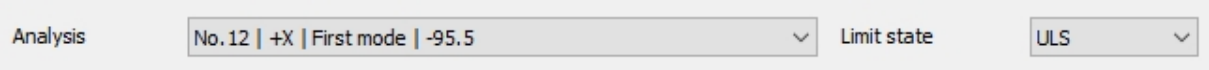
Press[Cancel] to exit the calculation window abandoning any change.

10.9.2 Results

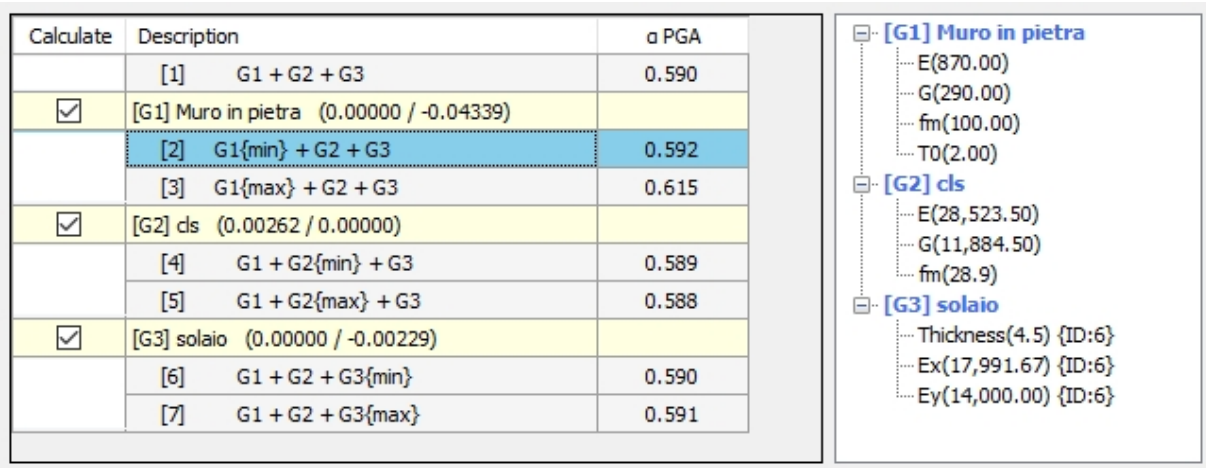
To access the sensitivity analysis results, press "Sensitivity results" from the drop down menu.



In the upper part of the screen is necessary to select the analysis and the limit state of the required results.



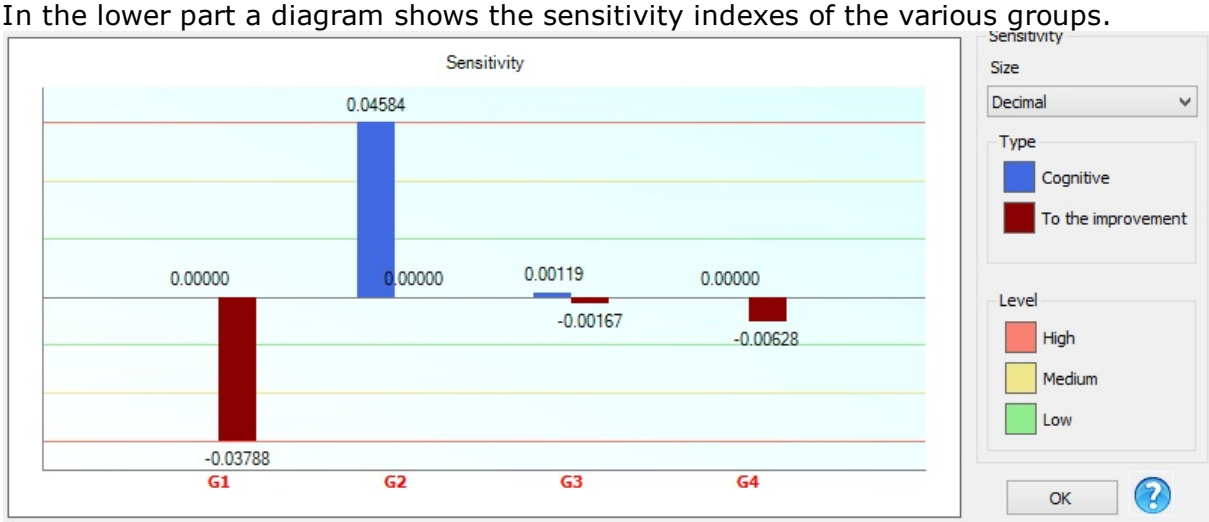
The table shows the list of the performed analysis.
Except for the analysis [1] that is in correspondence of all the groups average values, the following are grouped according to the group that is set to min or max values.
For each analysis is displayed the vulnerability index on the accelerations and for each group the matching sensitivity index.



Calculate	Description	α PGA
	[1] G1 + G2 + G3	0.590
☑	[G1] Muro in pietra (0.00000 / -0.04339)	
	[2] G1{min} + G2 + G3	0.592
	[3] G1{max} + G2 + G3	0.615
☑	[G2] cls (0.00262 / 0.00000)	
	[4] G1 + G2{min} + G3	0.589
	[5] G1 + G2{max} + G3	0.588
☑	[G3] solaio (0.00000 / -0.00229)	
	[6] G1 + G2 + G3{min}	0.590
	[7] G1 + G2 + G3{max}	0.591

By selecting a single analysis with the mouse, it is possible to see on the right menu precisely the values used in the calculation of the corresponding analysis.

In the lower part a diagram shows the sensitivity indexes of the various groups.



Two different types of sensitivity are shown:

«Cognitive» Sensitivity:

The existence of a parameters combination that may provide worse conditions than the average is searched from the examination of all the carried out analysis combinations.
Following an example of cognitive sensitivity index calculation for the [G1] group.

$$\alpha_{mean} = \alpha_{[1]} \;\; ; \;\; \alpha_{min} = min(\alpha_{mean}; \overbrace{\alpha_{[2]}; \alpha_{[3]}}^{[G1]}) \;\; ; \;\; I_s = \frac{\alpha_{mean} - \alpha_{min}}{\alpha_{mean}}$$

Given the mathematical formulation, this parameter will always be **>0** or eventually **=0** if the reference parameter group is not significant for the result or if the average value corresponds to the worst condition.

The use of this parameter is a prerequisite for the choice of the points of the structure in which to conduct the tests/surveys.

«Improving» Sensitivity:

The existence of a parameters combination that may provide better conditions than the average is searched from the examination of all the carried out analysis combinations.
Following an example of improving sensitivity index calculation for the [G1] group.

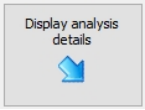
$$\alpha_{mean} = \alpha_{[1]} \;\; ; \;\; \alpha_{max} = max(\alpha_{mean}; \overbrace{\alpha_{[2]}; \alpha_{[3]}}^{[G1]}) \;\; ; \;\; I_s = \frac{\alpha_{mean} - \alpha_{max}}{\alpha_{mean}}$$

Given the mathematical formulation, this parameter will always **<0** or eventually **=0** if the reference parameter group is <u>not significant</u> for the result or if the average value corresponds to the <u>best condition</u>.

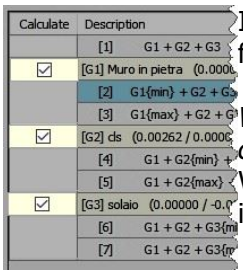
The use of this parameter is a prerequisite for the choice of intervention that is more efficient to improving.

The height of "*colored columns*" in the diagram, represents a "weight" in terms of importance of the individual group compared to the others.
High columns, correspond to "sensitive" parameters for the result that is worth investigating whether it is cognitive sensitivity or elements that is useful to reinforce in case of improving sensitivity.

In order to improve the reading of the sensitivity indexes it is possible to show the numeric value in *Decimal, Scientific* or *Percent* format.



By selecting one of the analyzes from the list and by pressing this button you can enter the analysis details.



In the performed analysis table, a check mark may exclude a group from the sensitivity index count.

When does it makes sense to exclude a group from the sensitivity calculation?
When from details of the analysis calculation comes out a result that is considered to be insignificant or even distort for sensitivity.

10.10 Bending analysis out of plan

The out of plan verifications can be performed separately, and the equivalent forces can be taken to the non-structural elements, assuming qa = 3.
More precisely, the seismic action perpendicular to the wall can be represented by a distributed horizontal force, equal to Sa / qa times the weight of the wall.
For the earthquake-resistant walls, it can be assumed for Sa the following expression:

$$S_a = \alpha \cdot S \cdot \left[1.5 \cdot (1 + Z/H) - 0.5\right] \geq \alpha \cdot S$$

where:
α: the ratio of the maximum acceleration of soil ag to subsoil type A for the limit state under examination and gravity acceleration g;
S: coefficient that takes account of the subsoil category and the topographical conditions;
Z:quote of barycenter of the non-structural element measured from the foundation plan;
H: height of the building measured from the foundation plan;

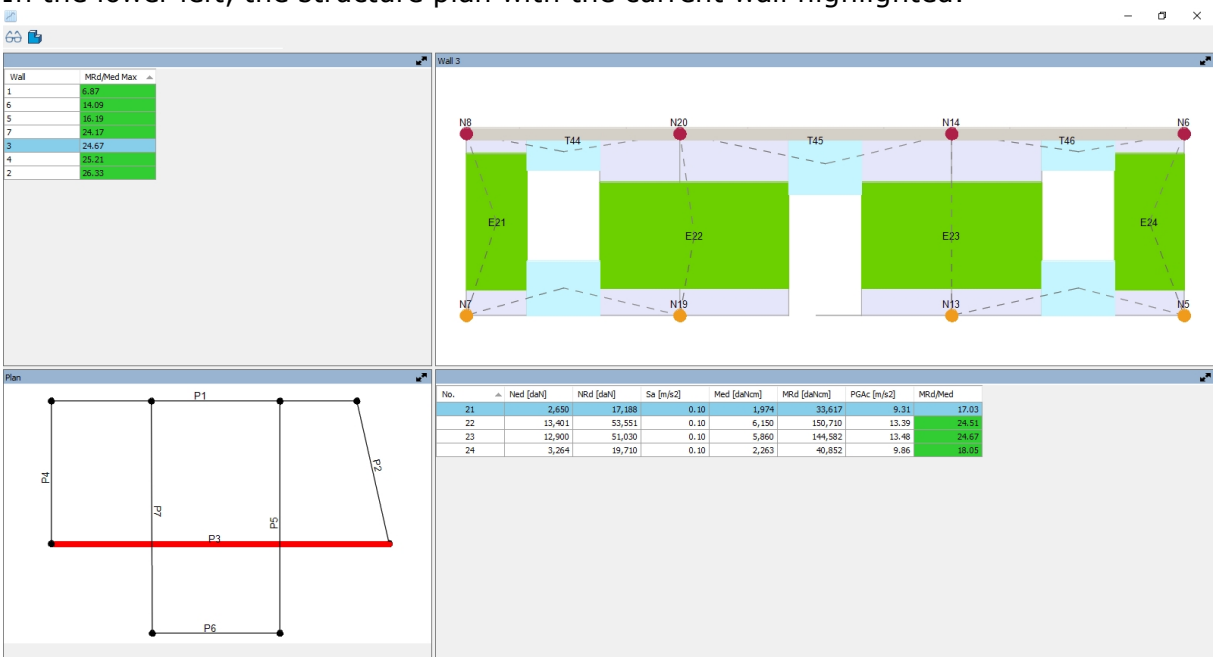
Known the Sa/qa ratio you can compute the force Fh=N*Sa/qa representing the horizontal force at the center of gravity of the pier.
We have the maximum moment at mid-height of the pier assuming the configuration of pier hinged in correspondence of the slabs that produces a moment produced by the transformation of the load Fh in distributed load q=fh/he equal to Med=q $he^2/8$.

The resistant ultimate moment is provided by the following formulation:

$$M_{Rd} = \left(l \cdot t^2 \cdot \frac{\sigma_0}{2}\right)\left(1 - \frac{\sigma_0}{0.85\, f_d}\right)$$

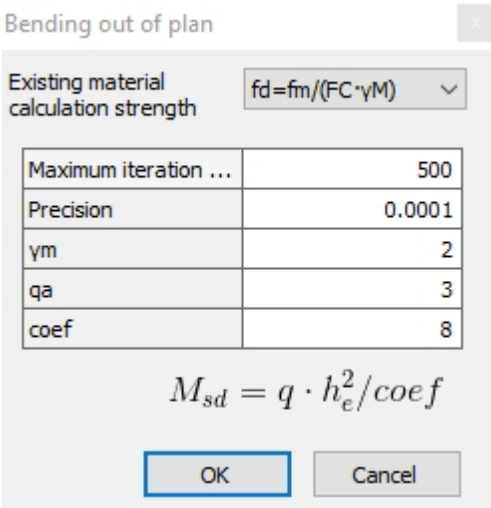
The verification will be exceeded if the MRd/Med ratio is greater than one.

The result environment is divided into 4 main areas.
In the top left, the list of walls with the corresponding value of the lower Mrd/Med ratio between all the wall piers.
In the upper right, the wall panel with the piers that did not pass the checkout highlighted in red.
In the bottom right, check details for each single pier.
In the lower left, the structure plan with the current wall highlighted.



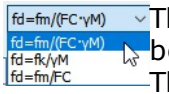
Moving from one wall to another can be simply by clicking on the plan in the point of interest or by using the list of walls by clicking on the corresponding row.

From the "*Settings> Bending out of plan*" menu, you can set the main parameters of the calculation:



The resistance calculation of the existing material is not clearly explained how it should be calculated within the regulations for this type of verification; for this reason, we

propose different alternative formulas, suggesting as the default what we feel most appropriate for this verification.

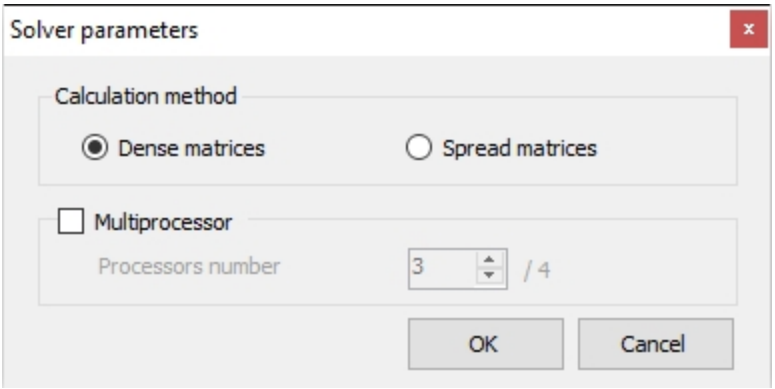


The first formulation of the list is proposed by us as a default because we believe it is scientifically more reasonable.
The second formulation, apparently consistent with the legislation, it is actually affected by the problem that the calculation of fk is approximated if obtained from fm and is likely to be extremely punitive.

10.11 Solver parameters

[OPTIONAL]

It is possible to access in that environment directly from the menu "settings."
Here are present two different calculation settings regarding the processor.



Calculation method:

- Dense matrices
- Spread matrices

The 3Muri program performs the calculation through the FEM method (finite elements), in the base of which exists the knowledge of the stiffness matrix K.
The classical engineering applications give rise to stiffness matrices with a high number of null values inside them.
In numerical analysis, a spread matrix is a matrix whose values are almost all equal to zero.

When the sparse matrices are stored and managed on a computer, it is useful and often also a necessity, to use specialized algorithms and data structures that take into account the spread nature of matrix. Performing the operations by using the structures and the usual matrix algorithms (dense method) results a very slow operation, and also leads to great waste of memory, if the matrix to be managed have big dimensions. The spread data are, by their nature, easily compressible, and their compression almost always involves a significantly lower usage of memory. It is also true, however, that some very large spread matrices are impossible to manage with the standard algorithms.

The stiffness matrices of masonry structures calculated with the equivalent frame method, averagely are of small dimensions. In such cases, a calculation to spread matrices does not show many advantages. Paradoxically, in the buildings with contained size the "conversion" time from dense to spread could lead to more overall calculation times; The approach to the spread matrices is therefore not always convenient in terms of time saving. We therefore feel to advise to all our customers to use this method of calculation in cases where the performance of the PC in use are not

sufficient to produce a result, or in cases of buildings with important dimensions.

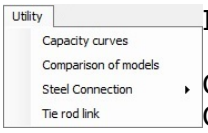
Multiprocessor:

By activating this option it is possible to conduct the calculation simultaneously on multiple processors.
In the case of multiple analysis (pushover 24) is possible to address an analysis on each processor available on the PC.
The default proposes to use a number of processors equal to the maximum number minus one, the unused processor is left available to the system resources.
The saving in terms of time therefore depends on the number of processors available by the system.
Example:
In the case of PC with 4 processors dedicated to the calculation, the total time taken on the 24 analysis will be about 1/4 of the time spent with a single processor!

Info:
This module is available with the acquisition of the appropriate licence, the "Standard" version of the product contains no such form. For more information contact your distributor.

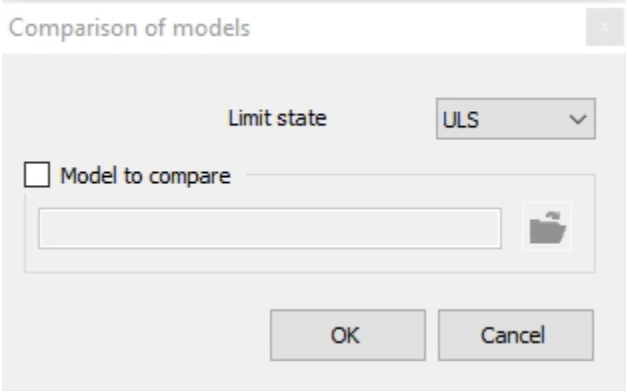
10.12 Utility

From the File menu, you can access the **"Utility"** category.



In this category you can find the following items:

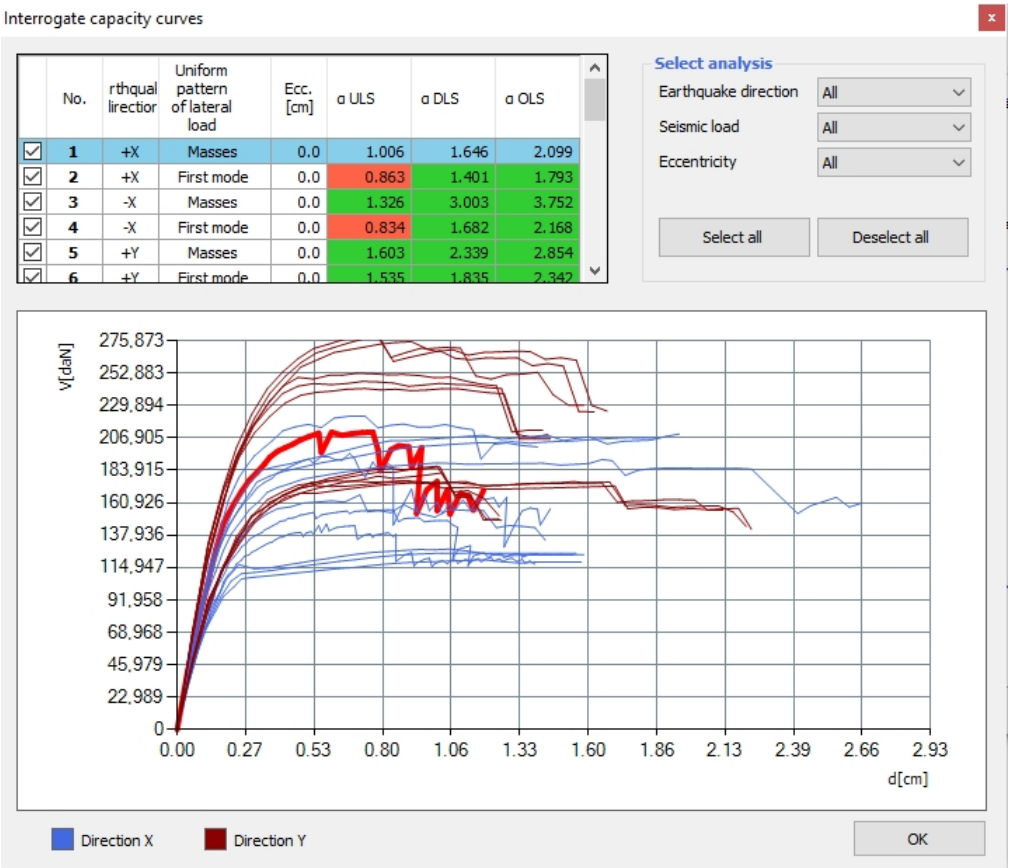
Capacity curves
Comparision of models
It compares different models for a given limit state.



Steel Connection
Tie rod link

10.12.1 Overlapping capacity curves

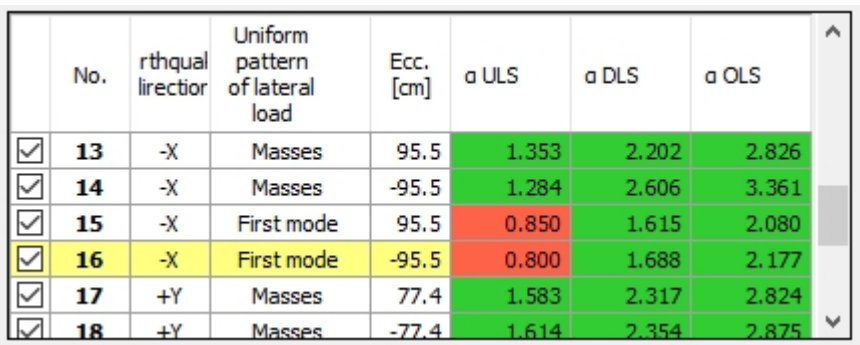
This feature allows the designer to simultaneously refer to the capacity curves of more analysis on a single diagram.



The capacity curves area is composed of three principal elements:
- Analysis table
- Overlapped pushover curves
- Selection filters

Analysis table

Contains a summary of the analyzes for the current model.
The first columns describes the analysis type, the last show the vulnerability indexes for each of the three limit states.
The background color, green or red, distinguishes the positive analyzes from the negative ones.
The yellow color shows the two analyzes that have the lowest vulnerability indexes (more significant for calculation purposes).

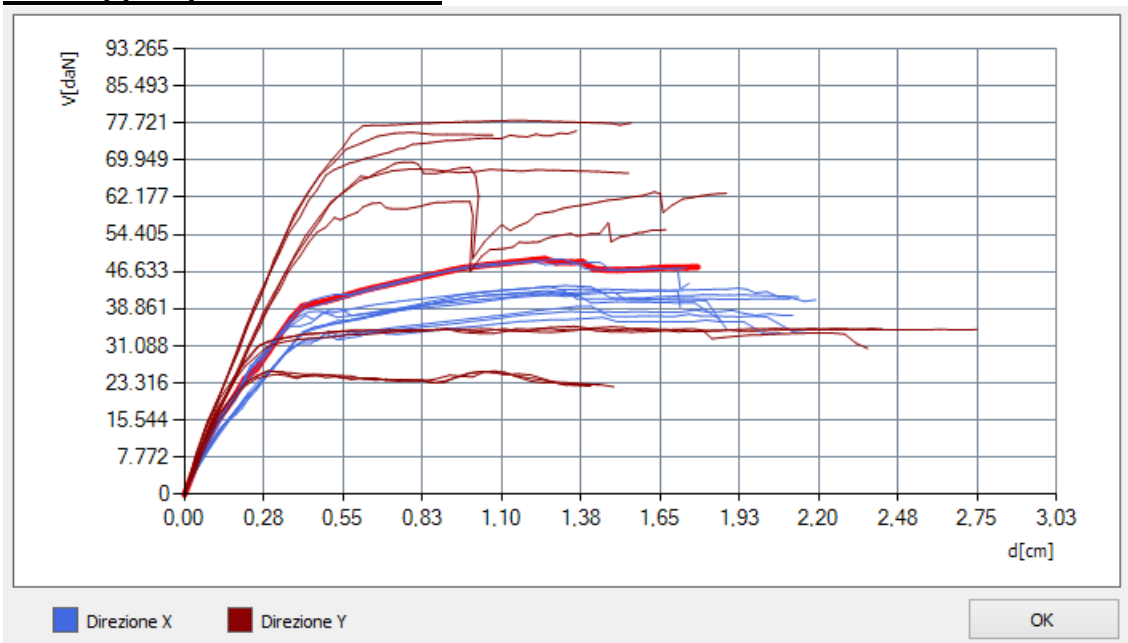


No.	rthqual lirectior	Uniform pattern of lateral load	Ecc. [cm]	α ULS	α DLS	α OLS
13	-X	Masses	95.5	1.353	2.202	2.826
14	-X	Masses	-95.5	1.284	2.606	3.361
15	-X	First mode	95.5	0.850	1.615	2.080
16	-X	First mode	-95.5	0.800	1.688	2.177
17	+Y	Masses	77.4	1.583	2.317	2.824
18	+Y	Masses	-77.4	1.614	2.354	2.875

The selected analysis is highlighted in blue, that shows the relative push-over curve, highlighted by a red thick line, in the "Overlapped push-over curves" graph.
Another analysis is highlighted in blue by clicking on a row in the table and the corresponding curve is selected on the graph.

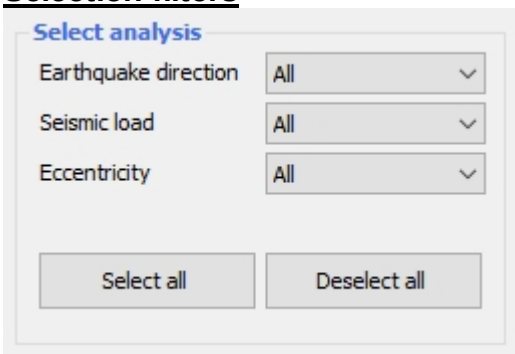
It is also possible to disable some analysis by manually unchecking the first column, the corresponding push-over curves will be automatically excluded from the graph.

Overlapped pushover curves

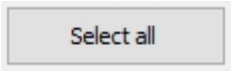


Shows the capacity curves of all the calculated and active analysis.
The curves shown in blue are relative to the X direction while those represented in brown are relative to the Y direction.
The selected curve is highlighted by a red thick line.
It is also possible to change the selection by clicking on another curve, the relative analysis in the "Analysis table" will be automatically highlighted in blue.

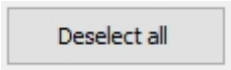
Selection filters



By using the proper space, there can be selected multiple analysis by activating the selection filters.

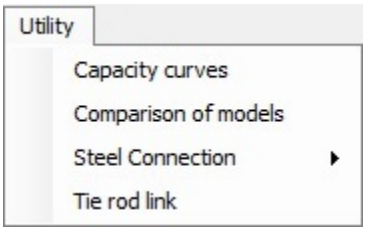


Activate all the analysis and relative curves currently disabled.



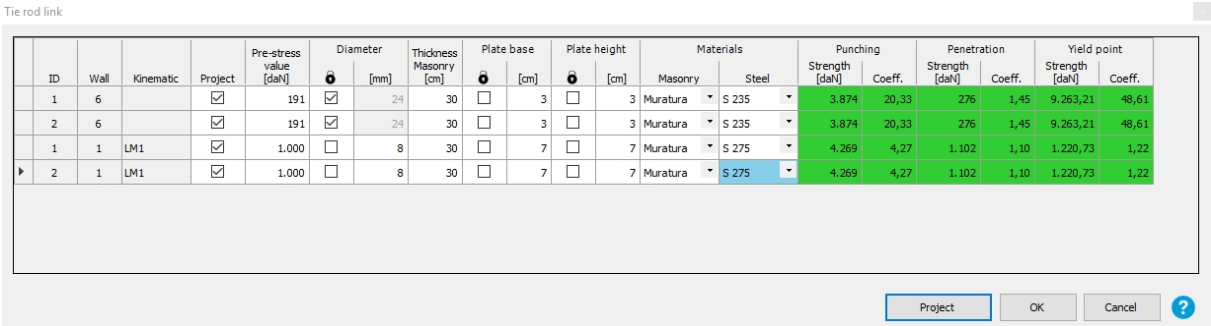
Disables all the analysis and relative curves currently enabled.

10.12.2 Tie rod link



[Utility > Tie rod link]

Calling this command shows the window that checks the inserted tie rods.



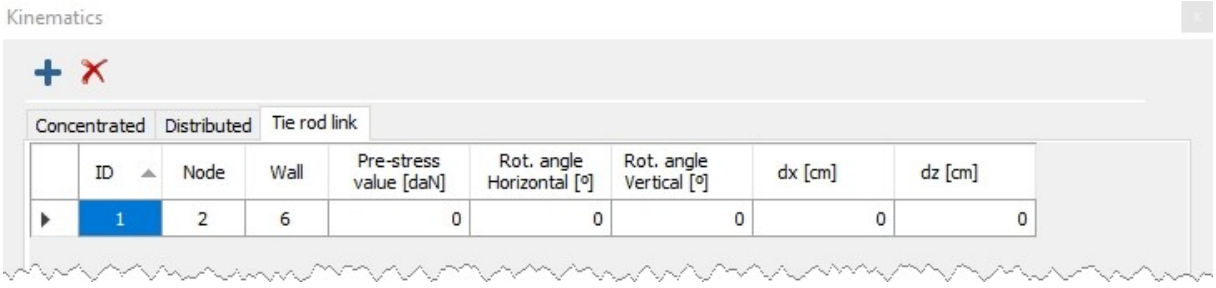
ID	Wall	Kinematic	Project	Pre-stress value [daN]	Diameter	[mm]	Thickness Masonry [cm]	Plate base	[cm]	Plate height	[cm]	Materials Masonry	Materials Steel	Punching Strength [daN]	Punching Coeff.	Penetration Strength [daN]	Penetration Coeff.	Yield point Strength [daN]	Yield point Coeff.
1	6		☑	191	☑	24	30	☐	3	☐	3	Muratura	S 235	3.874	20,33	276	1,45	9.263,21	48,61
2	6		☑	191	☑	24	30	☐	3	☐	3	Muratura	S 235	3.874	20,33	276	1,45	9.263,21	48,61
1	1	LM1	☑	1.000	☐	8	30	☐	7	☐	7	Muratura	S 275	4.269	4,27	1.102	1,10	1.220,73	1,22
2	1	LM1	☑	1.000	☐	8	30	☐	7	☐	7	Muratura	S 275	4.269	4,27	1.102	1,10	1.220,73	1,22

: This column allows to block, in the design phase, the corresponding dimension (eg in the case of the plate the user can block the base dimension and only increase the height).

The tie rods considered belong to two different types:

Tie rods as reinforcement intervention of local mechanism

They are the tie rods inserted only for the purpose of preventing the activation of the mechanism and are not present in the global model.
Column [Kinematisms]: the name of the mechanism is shown
Column [Pre-stress value]: is the computed pre-stress value in the mechanism load definition window in the tie rod session



ID	Node	Wall	Pre-stress value [daN]	Rot. angle Horizontal [°]	Rot. angle Vertical [°]	dx [cm]	dz [cm]
1	2	6	0	0	0	0	0

Columns [Tie rod diameter] / [Plate base] / [Plate height]: these are the columns that are designed/checked
Columns [Punching] / [Penetration] / [Yield]: these are the checks carried out with the related safety factor

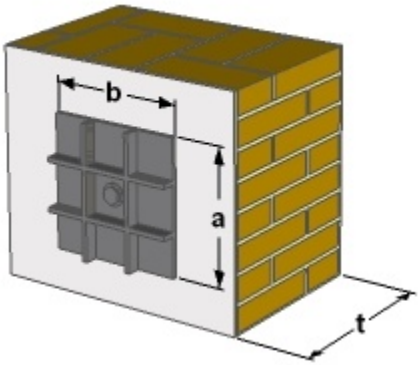
Tie rod from a global model

The calculation is carried out for the tie rod ends defined with the appropriate function in Graphical layout of steel elements.
This environment is used to define the "extreme tie rod" nodes which are required to be connected to the wall by means of a plate.

Selecting this command is required to indicate the nodes of which you want to proceed with the calculation.

10.12.2.1 Tie rod link - Verifications

The verification of the tie rod-plate-masonry system provides the control of three rupture mechanisms.



1 - Punching verification of the masonry in anchorage areas

The puncture resistance is given by:

$$T_{pun} = f_v \cdot [2 \cdot (b + t) + 2 \cdot (a + t)] \cdot t$$

where :
- t is the thickness of the masonry
- b is the width of the plate
- a is the height of the plate
- fv is the calculating shear strength of the masonry

2 - Penetration verification of the anchor

The penetration of the anchoring in the masonry is given by the overcoming of the masonry compressive strength of the plate contact pressure. The penetration resistance of the anchor is given by:

$$T_{pen} = f_d \cdot a \cdot b$$

where :
- fd is the calculating compressive strength of the masonry

3 - Yielding verification of the tie rod

The yield strength of the tie rod is given by:

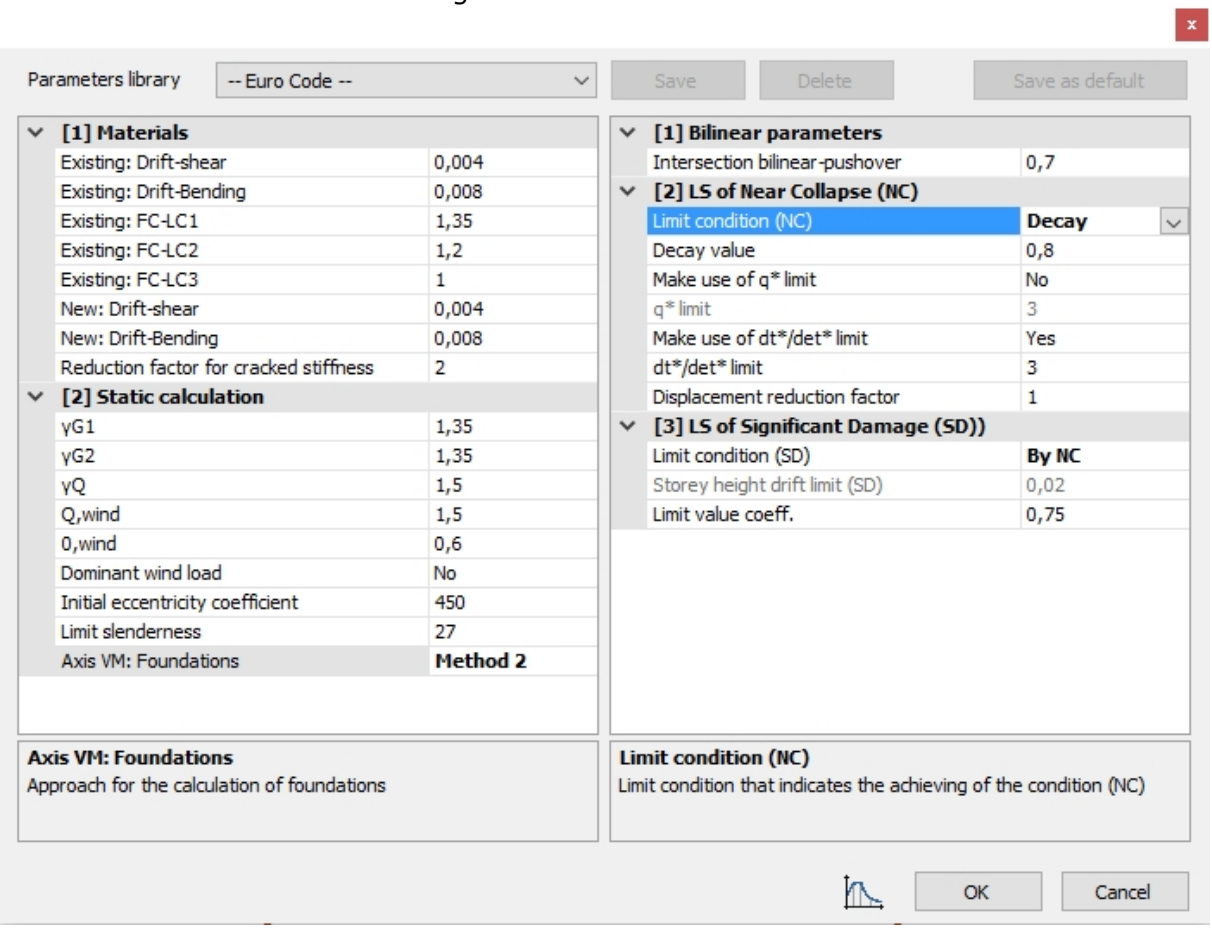
$$T_{tir} = f_y \cdot \frac{\phi^2 \cdot \pi}{4}$$

where :
- Φ is the diameter of the tie rod

- fy is the calculating strength of the tie rod

11 Setting code parameters

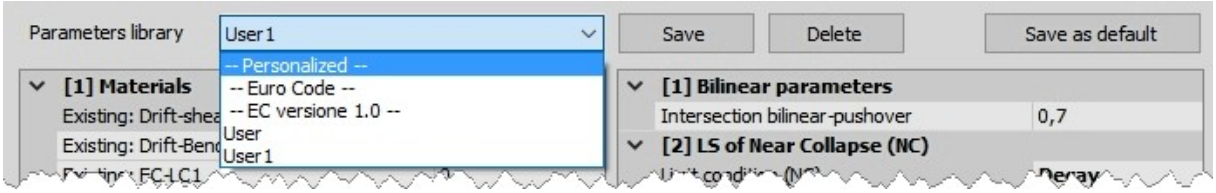
The window that allows the configuration of calculation data is as follows:



The parameters provided in this screen depend on the active standard.
For each standard they are available in the drop down menu several libraries of parameters defined by a <Name>:

1. if that name is NOT enclosed between "-" (<Name>) it means that it is a *project library* and will only be available in the project in which it was created independently of the PC on which the project is open.
2. if that name is enclosed between "--" (--<Name>--) it means that it is a *program library* that will be available for each project calculated with the same rule.

The *project library (1)* is used when you make a change and should only be used for this project.
The *program library (2)* is usually used when we need to change the parameters and use them for all future projects (eg, the national annexes of the Eurocodes, once the parameters are adjusted to those of the state in which they work, will be used for all projects in the same state).

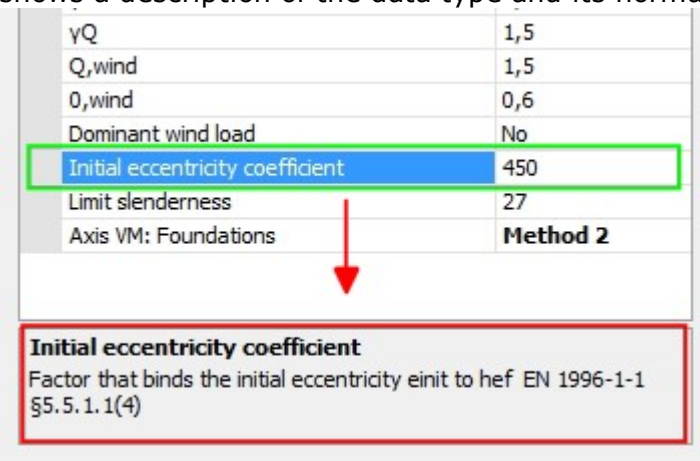


Creating libraries:

When modifying an input parameter, the drop-down moves immediately to -- *Personalized*-- to indicate that the parameters are being edited.
To confirm and make the changes active, press the **[Save]** button and enter the name to assign to the new library.
Confirming the insertion of the library is immediately selected as current.
The newly created library is *"project" type (1*).
Pressing **[Save As Default]** saves a copy of the library as "Program Library" (2) (the name will be enclosed in "-").
The program libraries present are usually different configurations of the parameters with the same rule, only one of them can be generated by using the **[Save As Default]** command.
Rerun **[Save as default]** on a different *project library (1)* lead to overwrite the *program library (2)* previously created.
Ex. As in the previous figure, select "User02", press **[Save as default]** leads to overwrite "--User01--" which is the only *"program library" (2*) available to the user.

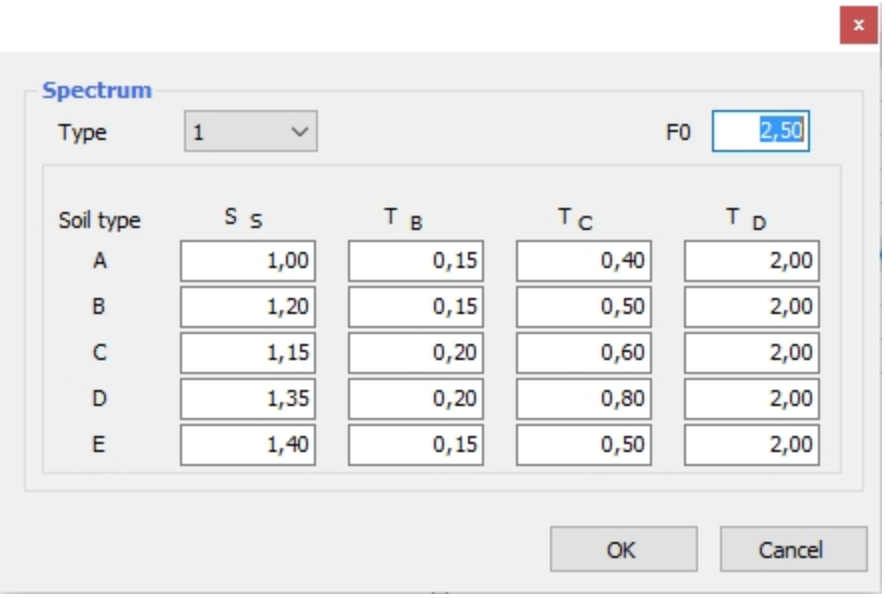
The [Delete] command allows you to delete a *project library (1)* created, *the program library (2*) are not erasable.

Contextual description of the parameters subject to change:
By selecting the field corresponding to a specific value, the bottom part of the grid shows a description of the data type and its normative reference.



Spectrum (available only with Eurocode standard)

With this button you can access the table containing the parameters for the definition of the seismic spectrum depending on the soil classes.
Ms: magnitude value.
F0: maximum value of the amplification factor of the accelerated spectrum (usually assumed to be 2.5).

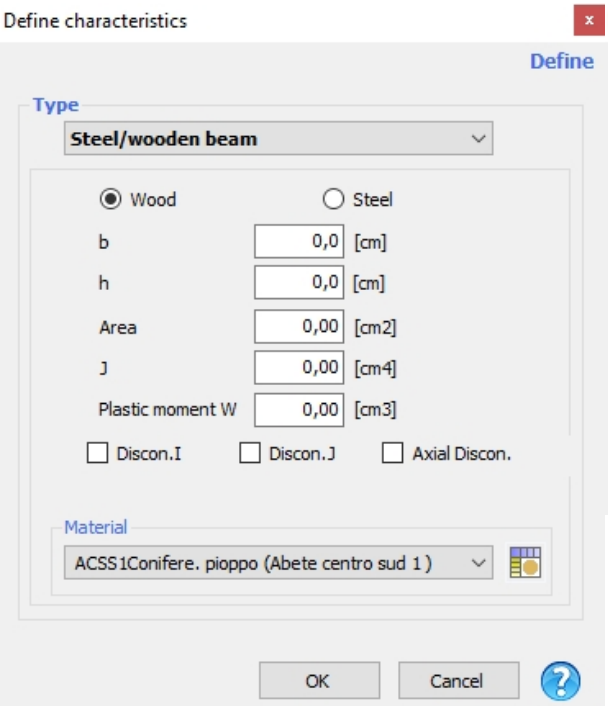

Spectrum
Type
1
F0
2,50
Soil type
S S
T B
T C
T D
A 1,00 0,15 0,40 2,00
B 1,20 0,15 0,50 2,00
C 1,15 0,20 0,60 2,00
D 1,35 0,20 0,80 2,00
E 1,40 0,15 0,50 2,00
OK
Cancel

12 Reinforcement frames

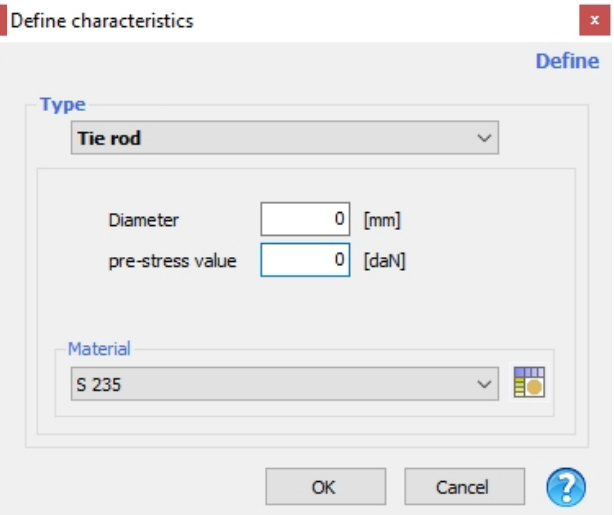
Allows to define the geometric / mechanical properties of the reinforcing elements drawn with the insert command .

By calling this command, you need to perform a selection of reinforcements to which you want to assign the properties; once you completed the selection will appear the characteristics definition window.

By selecting **"Steel / Wood Beam "** from the drop down menu is possible to define the characteristics of a beam in wood or steel.



By selecting **"Tie rod"** from the dropdown menu is possible to define the characteristics of an element only resistant to traction.



13 SismoTest

SismoTest is the 3Muri module dedicated to the Seismic Classification of Buildings, according to D.M. n. 65 of 7/3/2017.
With the 2017 Budget Law, approved December 21, 2016, a campaign to seismic improvement of existing facilities it has begun.
This is the so-called "Sismabonus", an opportunity to stimulate a volunteer plan for assessing and preventing the seismic risk of buildings. The novelties of the Sismabonus decree are many and aim to promote seismic prevention through numerous facilitations, including:
- the extension to seismic zones 1, 2 and 3 and therefore a widening of the areas concerned, which in the previous legislation were limited to zones 1 and 2;
- deductions for five years and then a return certainly more affordable than the 10 years provided for types of non-seismic intervention;
- the maximum amount is equal to € 96,000 with the possibility of the assignment of credit that allows you to take advantage of funding even if you do not have fiscal capacity.
To access the Fiscal Benefit, the property owner must appoint a professional to evaluate the risk class and the preparation of the intervention project.

The SismoTest module of 3Muri puts into practice the requirements of the Guidelines, making them easy to apply, providing suggestions to the assessment, producing the required documents.