3muri

User Manual

Release: 5.0.1

S.T.A. DATA

Indice

1 3Muri - Commercial versions

3Muri software is currently available in three different commercial versions:

Version	***Limits***
• Piccole Strutture	Max 2 levels and max 600 m^2 of floor surface
• Small Business	Max 100 nodes
• Professional	Max 13 levels

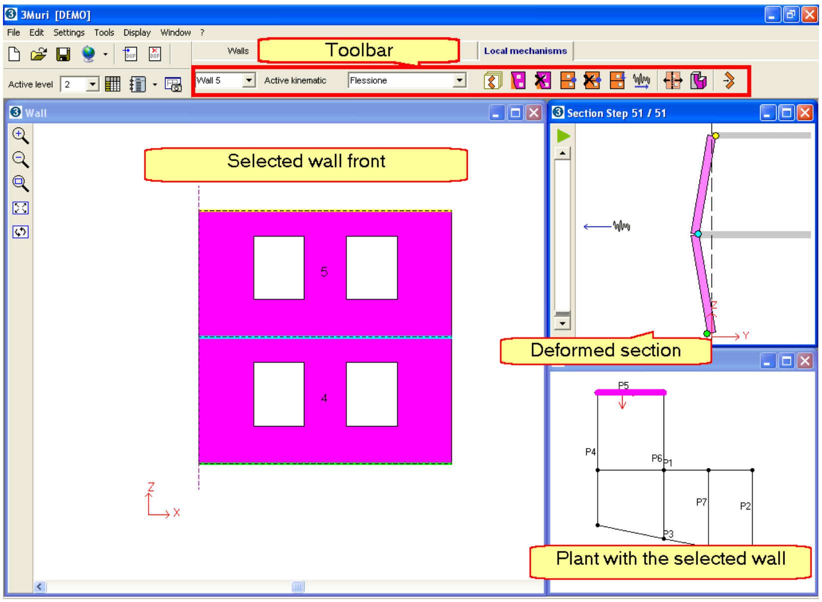
2 News

LM "Local Mechanisms" 3muri 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.



Materials Library

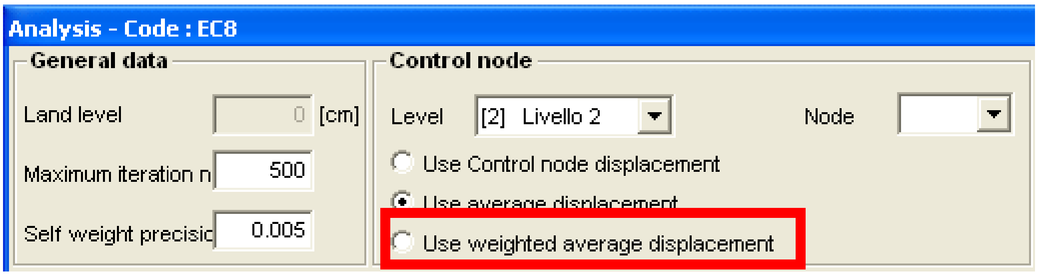
This function allows the designer to import on the project in exam the materials from different libraries (other Design Codes) or from the user library.
3Muri program has 3 main libraries types:

- 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.

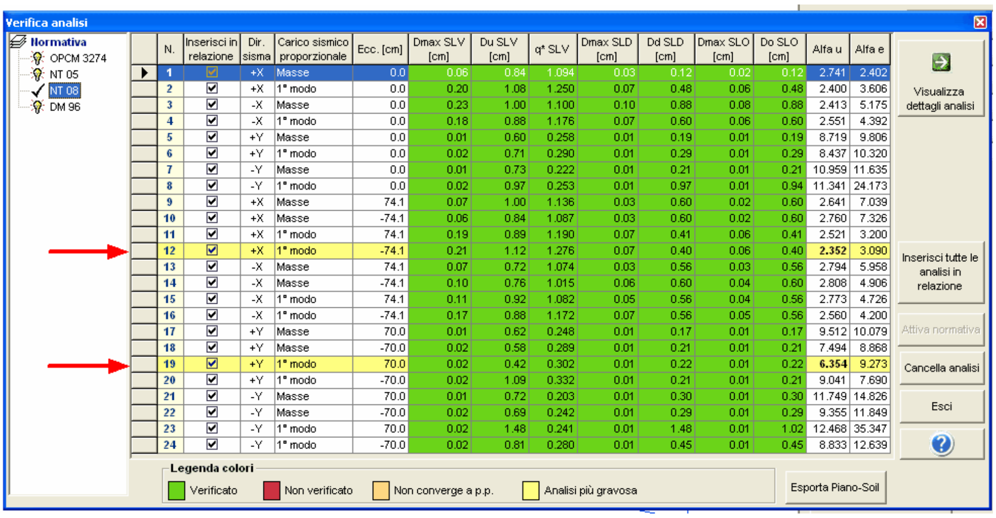
New parameters for the Pushover calculation

New output of the capacity curve for using weighted average displacement (mass weighted).

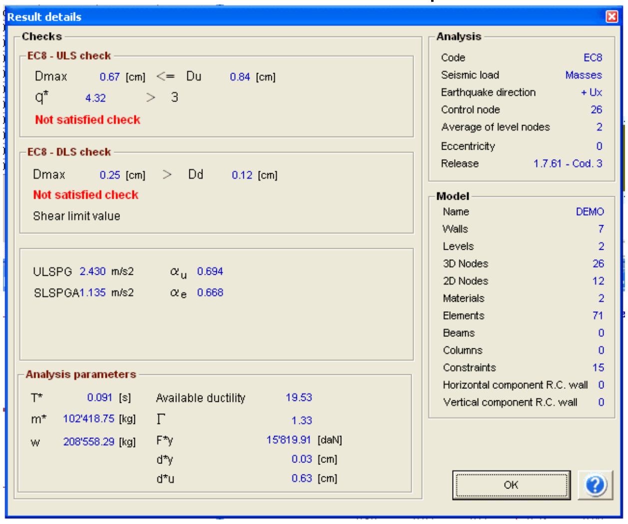


Improve results presentation

Underline of the most significative analysis.



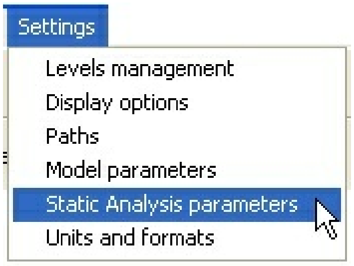
Presentation of new calculation parameters values.



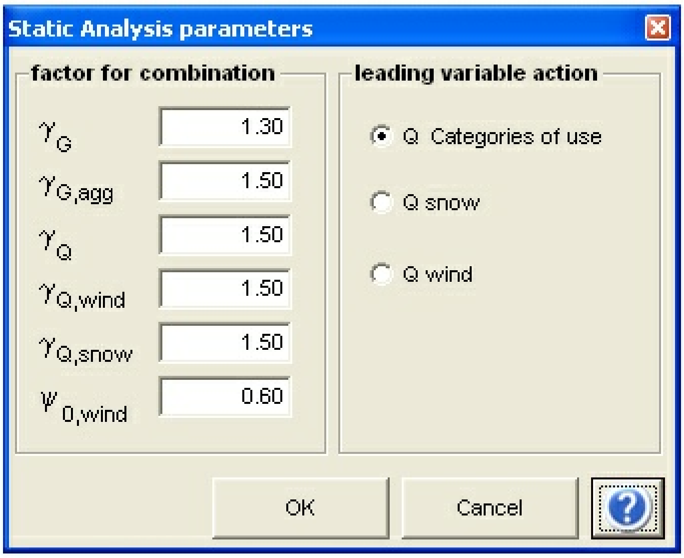
2.2 Version 4.0.311

Static 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.

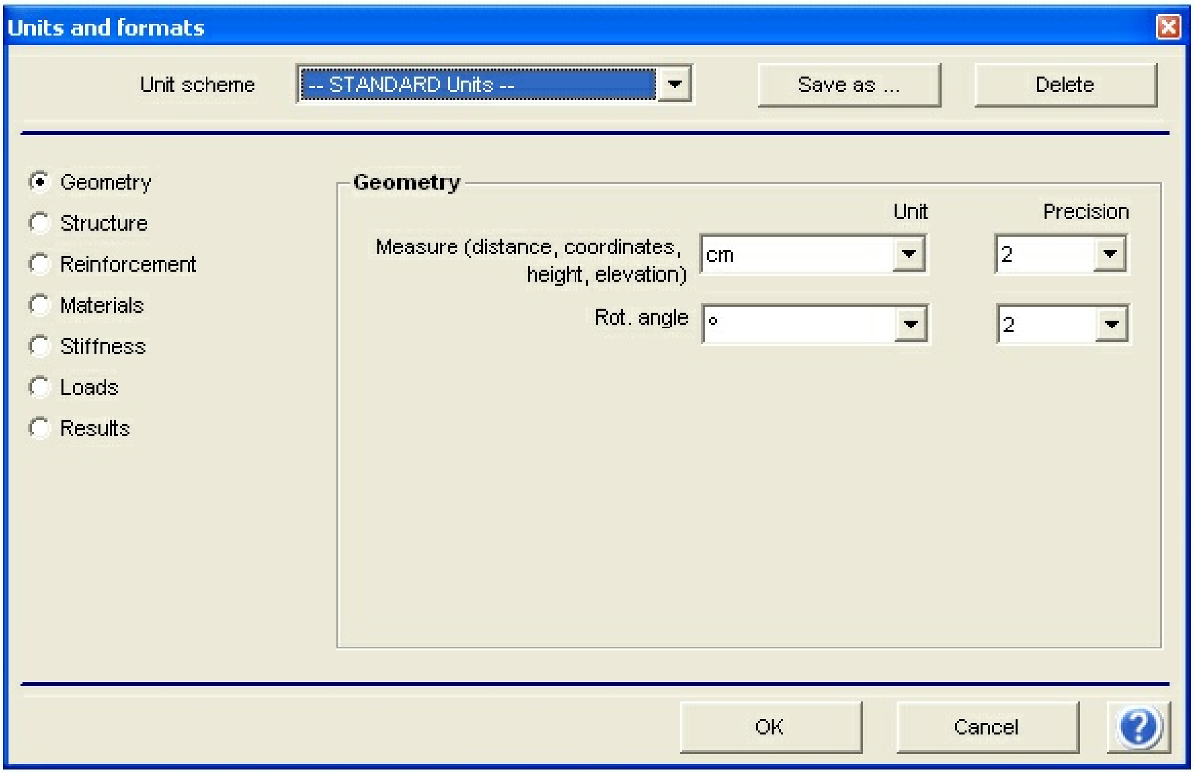


γ_G: factor for the permanent structural loads
$\gamma_{G,agg}$: factor for the permanent additional loads
γ_Q: factor for the live loads from the use destination of the building
$\gamma_{Q,vento}$:factor for the wind loads
$\gamma_{Q,neve}$: factor for the snow loads
$\Psi_{0,vento}$: factor for the wind loads

2.3 Version 4.0.3

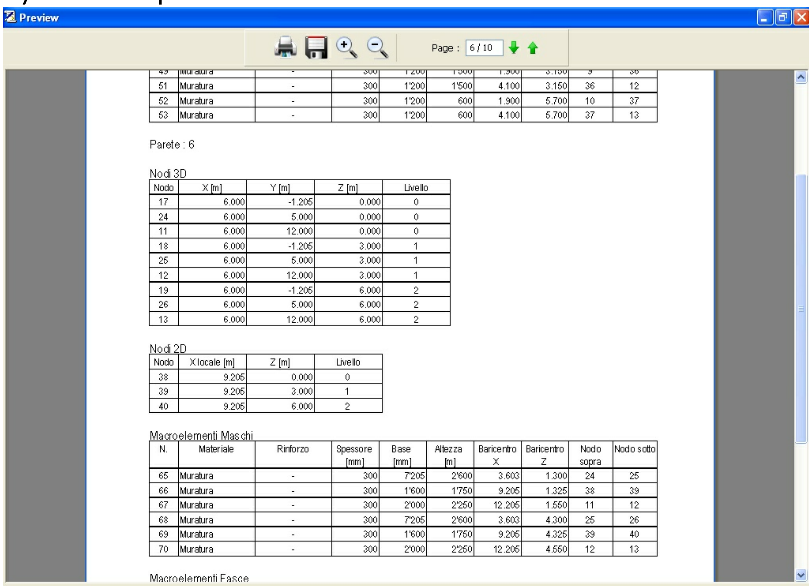
Units and formats

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.



Improved Export

An innovative component for reporting, allows you to export the report directly to RTF file, easily editable by any word processor (Microsoft Word, OpenOffice, etc...). With this new component the created tables are easily editable, and they appear as if they were created internally to word processor.



Preview

Page : 6 / 10

51	Muratura	-	300	1'200	1'500	4.100	3.150	36	12
52	Muratura	-	300	1'200	600	1.900	5.700	10	37
53	Muratura	-	300	1'200	600	4.100	5.700	37	13

Parete : 6

Nodi 3D

Nodo	X [m]	Y [m]	Z [m]	Livello
17	6.000	-1.205	0.000	0
24	6.000	5.000	0.000	0
11	6.000	12.000	0.000	0
18	6.000	-1.205	3.000	1
25	6.000	5.000	3.000	1
12	6.000	12.000	3.000	1
19	6.000	-1.205	6.000	2
26	6.000	5.000	6.000	2
13	6.000	12.000	6.000	2

Nodi 2D

Nodo	X locale [m]	Z [m]	Livello
38	9.205	0.000	0
39	9.205	3.000	1
40	9.205	6.000	2

Macroelementi Maschi

N.	Materiale	Rinforzo	Spessore [mm]	Base [mm]	Altezza [m]	Baricentro X	Baricentro Z	Nodo sopra	Nodo sotto
65	Muratura	-	300	7'205	2'600	3.603	1.300	24	25
66	Muratura	-	300	1'600	1'750	9.205	1.325	38	39
67	Muratura	-	300	2'000	2'250	12.205	1.550	11	12
68	Muratura	-	300	7'205	2'600	3.603	4.300	25	26
69	Muratura	-	300	1'600	1'750	9.205	4.325	39	40
70	Muratura	-	300	2'000	2'250	12.205	4.550	12	13

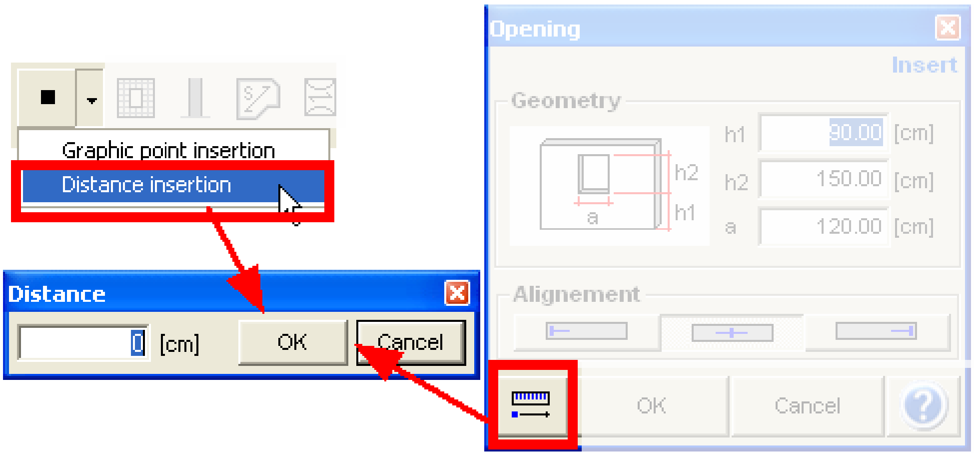
Macroelementi Fasce

Static Analysis

The calculations are now performed using the joints model as suggested in the existing code.

New Input on "distance" for nodes and openings

Insertion of a node/window can occur either through insertion of node/windows distances for the edge nodes.

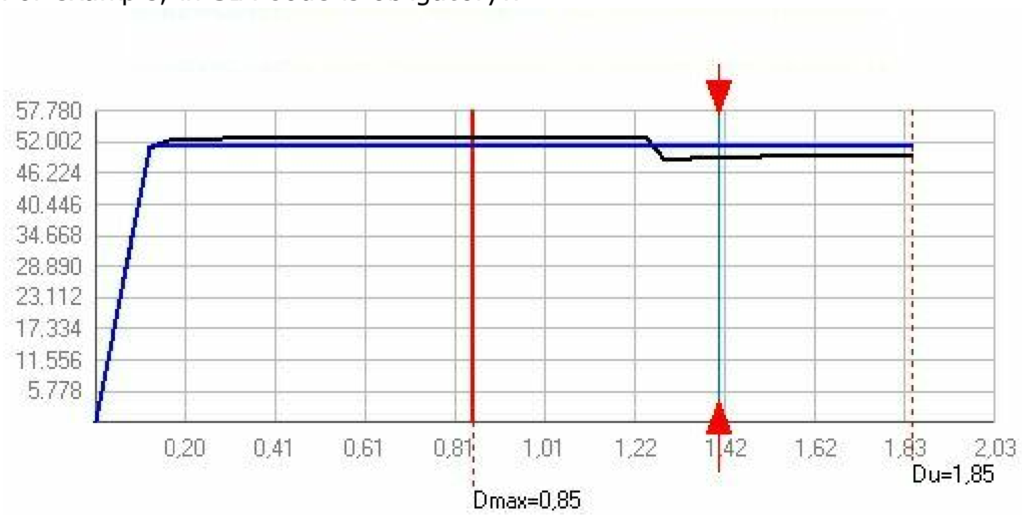


To insert "distance" segment nodes, it is necessary to position the mouse on a wall (highlighting it in red) and decide from which node to calculate the distance. The distance is then inserted, positioning the mouse closer to the node in question.

2.4 Version 4.0.2

New Last step

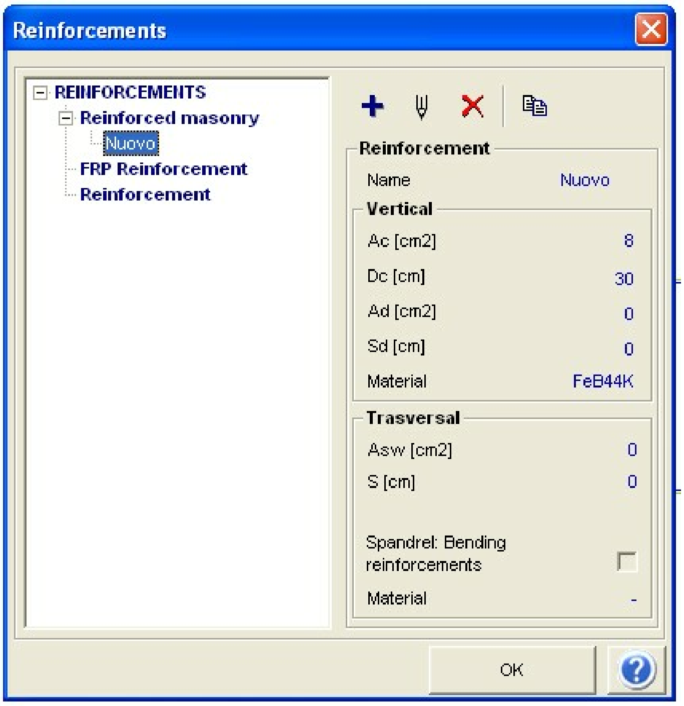
Some times, it could be necessary define a different value for ULS.
For example, in SIA code is obligatory!!



2.5 Version 4.0.1

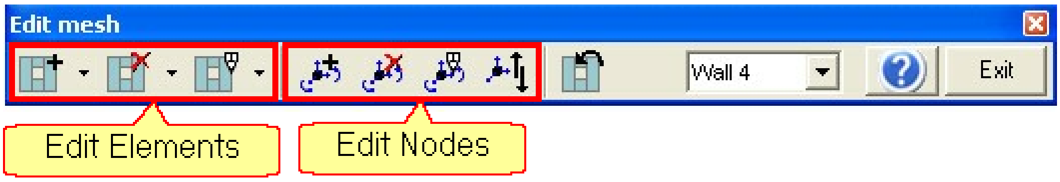
Reinforced masonry ed FRP

Now you can perform calculations of buildings with reinforced panels using FRP or reinforced masonry .



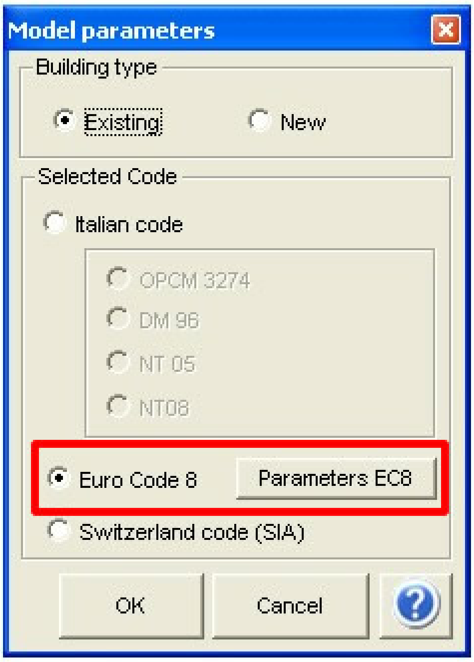
Powerfull Mesh Editing

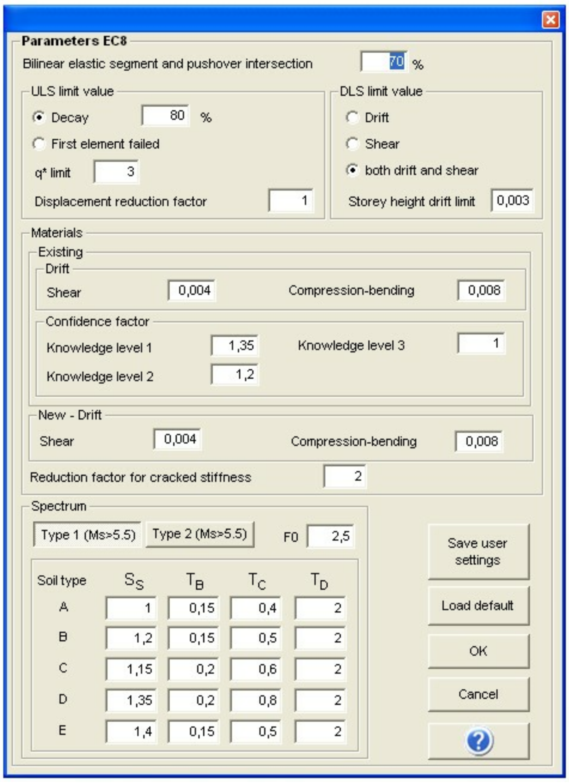
The procedures for automatic mesh generation are sufficiently advanced that it can capture a good 95% of cases that in practice a professional designer can meet. A new environment for editing, can enhance the existing environment in order to describe fully the building characteristics



Eurocode 8

Select Eurocode 8 among the codes listed in the window "Model parameters". It is possible to modify every settings selecting the button "EC8 parameters".





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.

English language

Now it is available 3Muri English version.

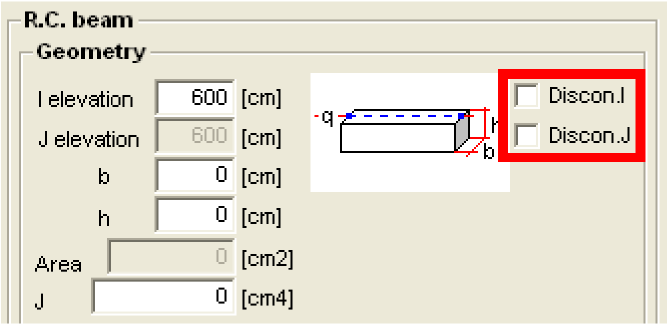
Info:

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.

2.6 Version 3.2.2

Internal Disjointures

R.C. beams as well as steel and wood beams can simply lean without being embedded. This new function allows the designer to define constraints for leaning, by inserting internal hinges, also in the non-linear field.

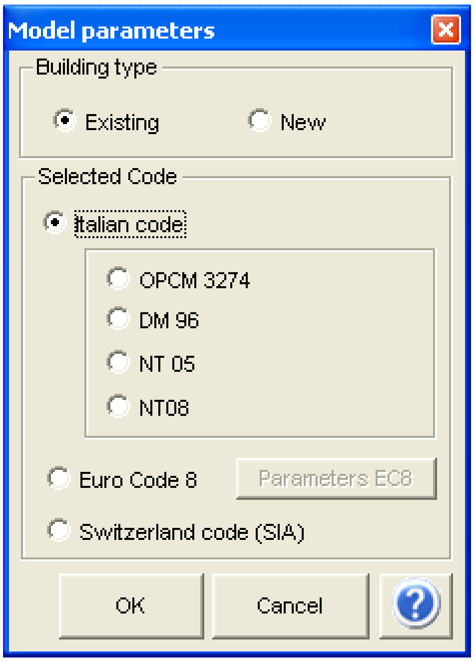


2.7 Version 3.2.0

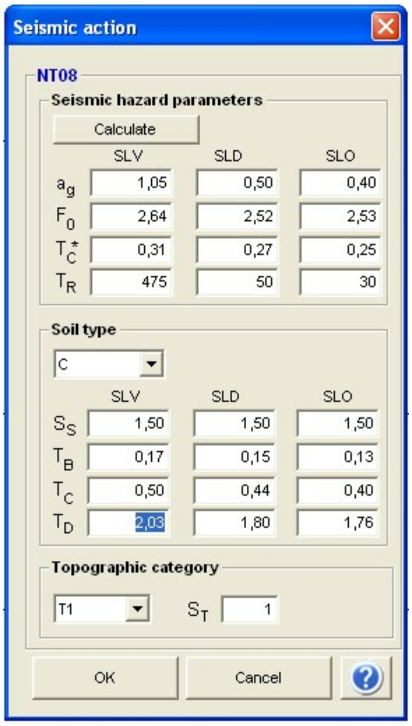
Here is a list of the main updates for this release of 3Muri.

Technical Norms for Construction - D.M. 14 January 2008

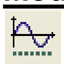
The template "Model Parameters" is shown when beginning a new project, allowing the designer to choose the code to be used. It can also be accessed through the Settings menu.



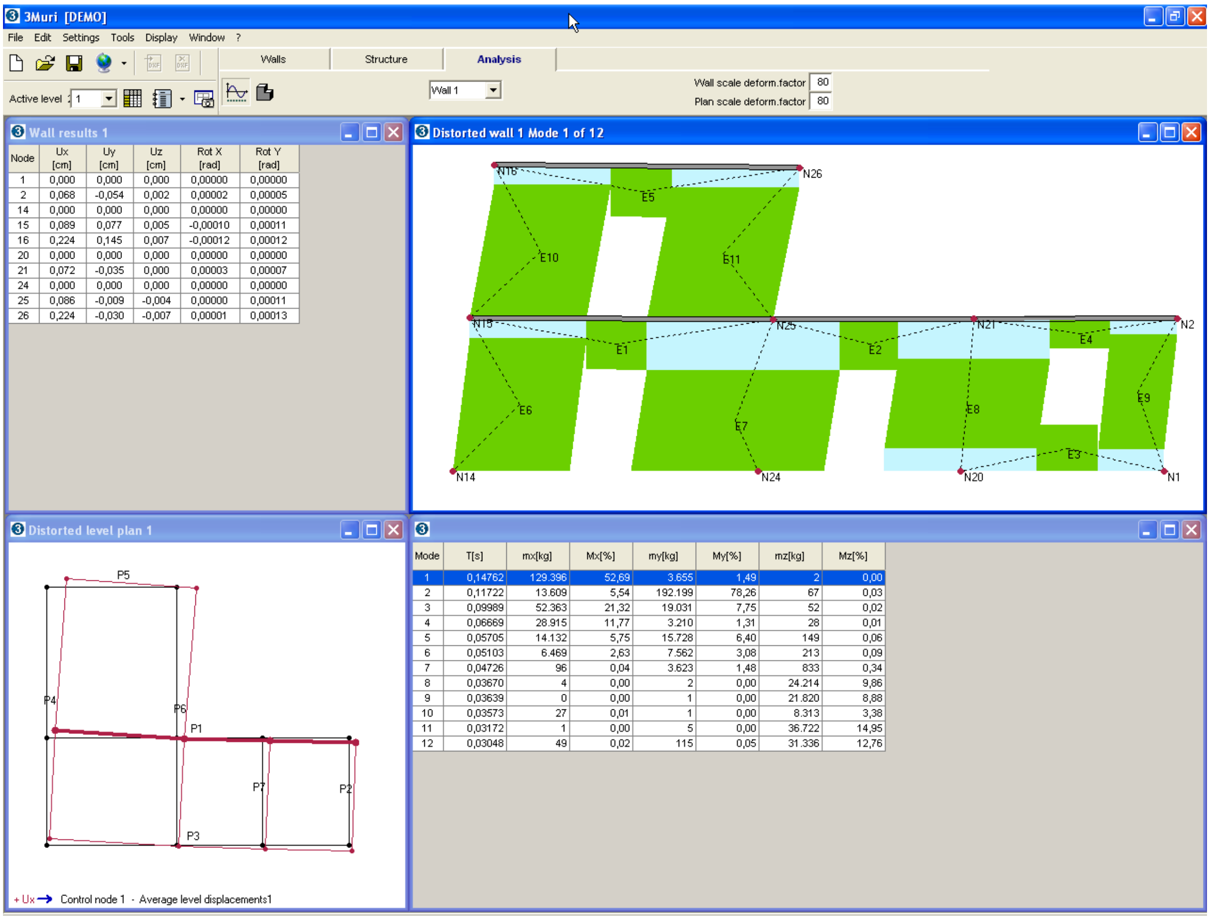
The new norms have created the necessity to identify the form of the spectrums through the reference lattice.



Modal Analysis



By selecting the number of modal forms, details are provided in relation to the participating masses and the modal deformation.


3Muri [DEMO]
File Edit Settings Tools Display Window ?
Walls
Structure
Analysis
Active level
Wall 1
Wall scale deform.factor 80
Plan scale deform.factor 80
Wall results 1
Distorted wall 1 Mode 1 of 12
Distorted level plan 1
Control node 1 - Average level displacements1

2.8 Version 3.1.0

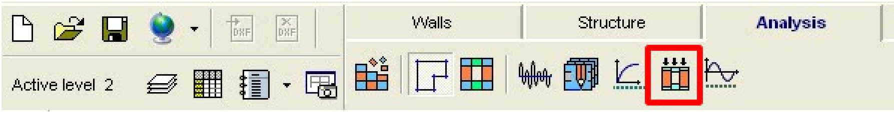
Here is a list of the main updates for this release of 3Muri.

Possibility to perform Static Checks

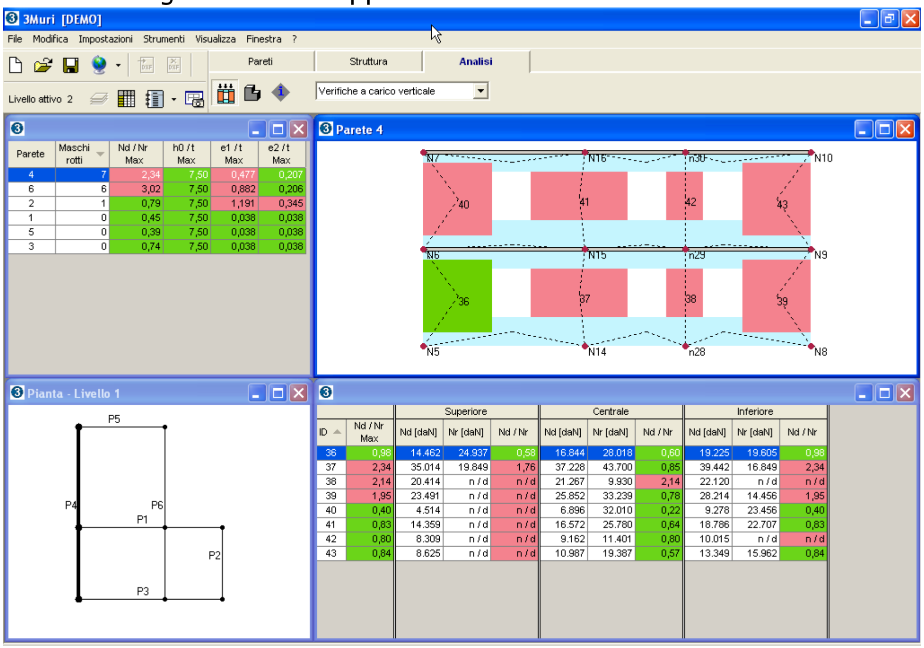
This is a module which performs static checks, according to the code in effect.
Here is a list of the checks:

Slenderness check:	$h_0/t \le 20$
Load eccentricity check:	$e_1/t \le 0.33$ $e_2/t \le 0.33$
Vertical loads check:	$N_d \le \Phi f_d A$

The static checks are performed in an area that is accessed using the associated button.

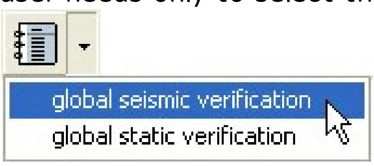


The following screen will appear:



Optimization of the area for Report creation

The view and pagination have been improved.
The program automatically creates both the seismic and the static checks reports. The user needs only to select the report to be created from the drop-down menu.



Multiple Language Management

It is possible to use the program and write a report in languages other than Italian.

The possibility to manage other languages is a separate module from the basic program. It is activated based on request.

Exporting in Piano soil

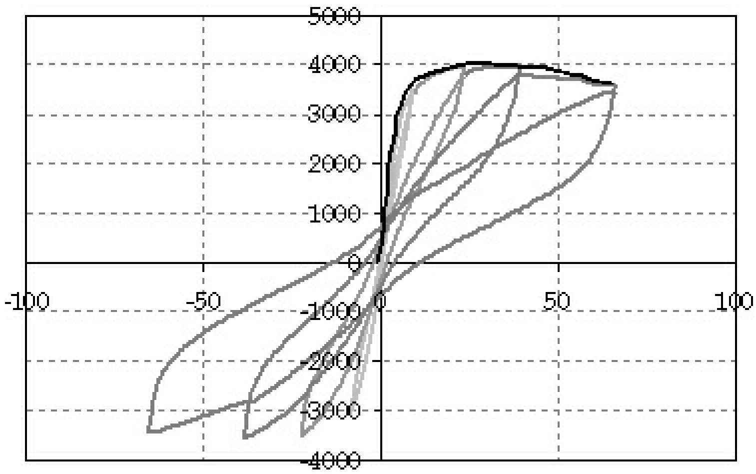
Procedure that exports a file containing foundation loads. This file is created so as to become input for the Piano soil program for computation of the foundation structure. [Piano soil is a product created and distributed by Aztec Informatica; www.aztec.it].

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.

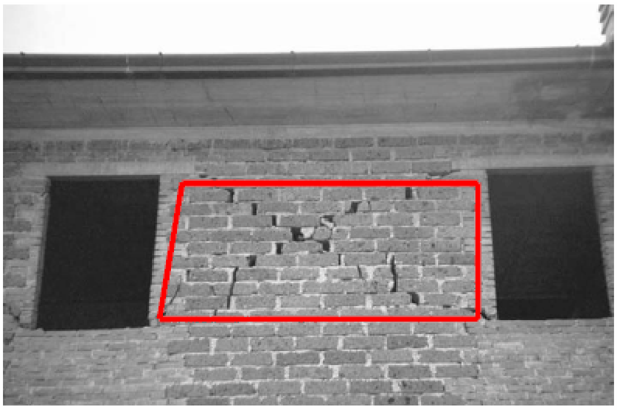
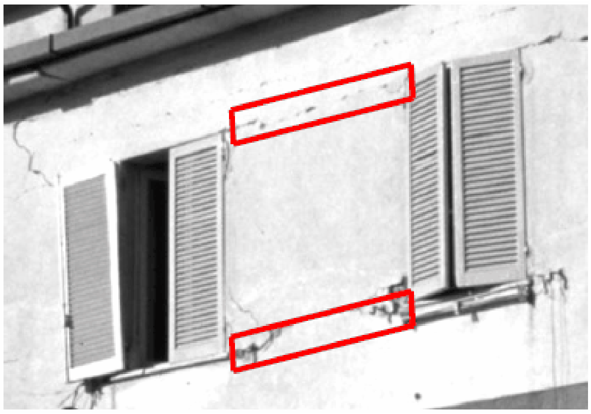


In this way, in the elastic analysis methods, the non-linear behavior is taken into account by introducing the structural factor, 'non-linear static analysis does not allow 'the structural response to evolve as each single element evolves in the non-linear field, providing information on the distribution 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.

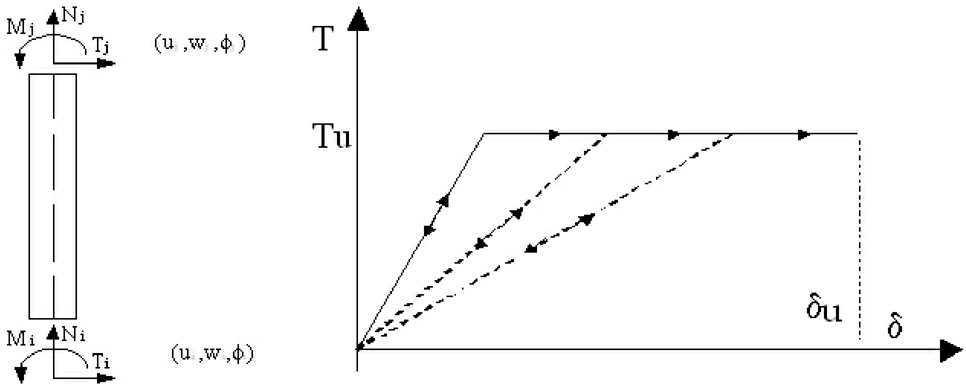
3.2 Masonry Macro-elements

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:

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

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



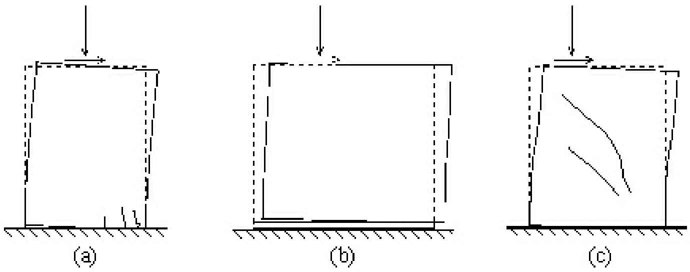
Non-linear beam degrading behavior

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)

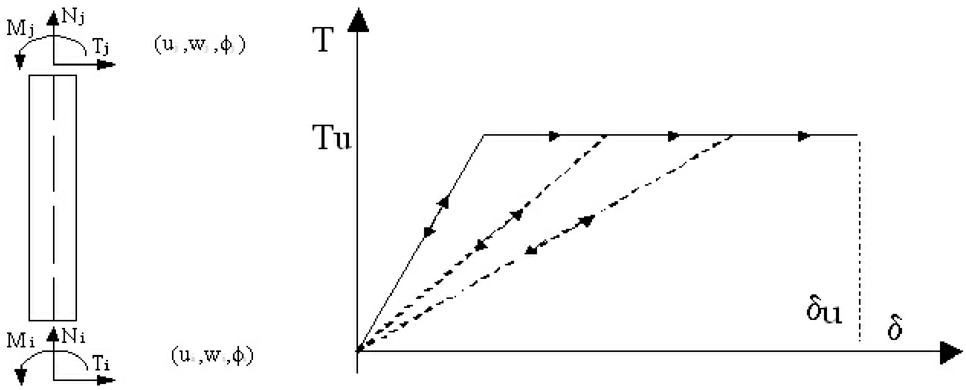
--------------------OLD_TEXT--------------------

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.



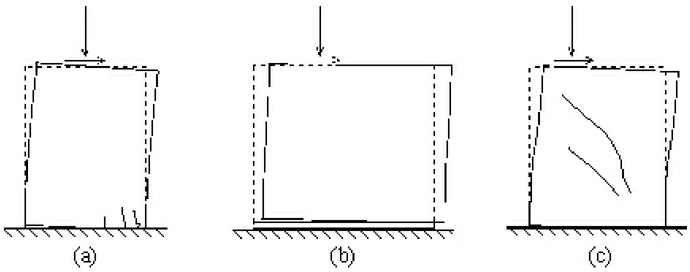
Non-linear beam degrading behavior

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.



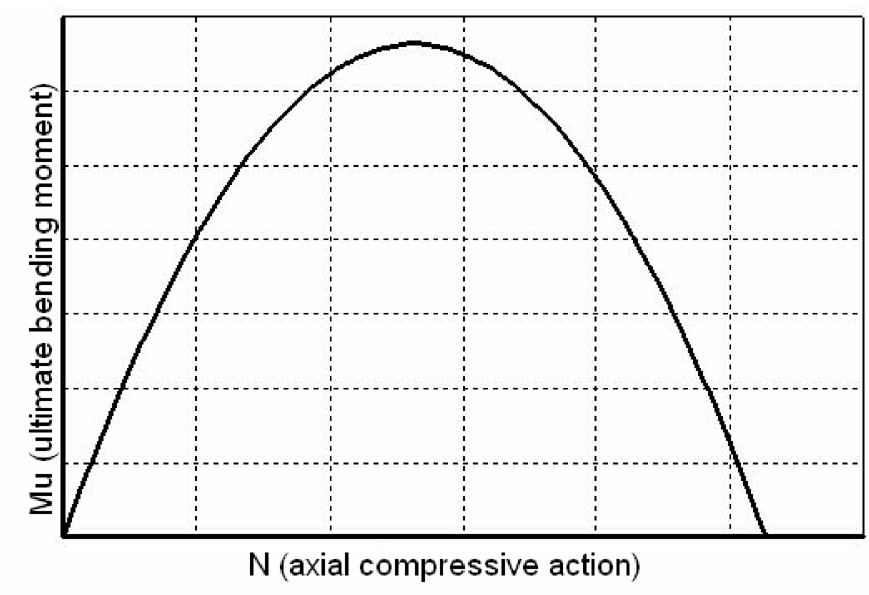
Masonry in-plane failure modes: flexural-rocking (a), shear-sliding (b) e diagonal-cracking shear (c) (Magenes et al., 2000)

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}

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

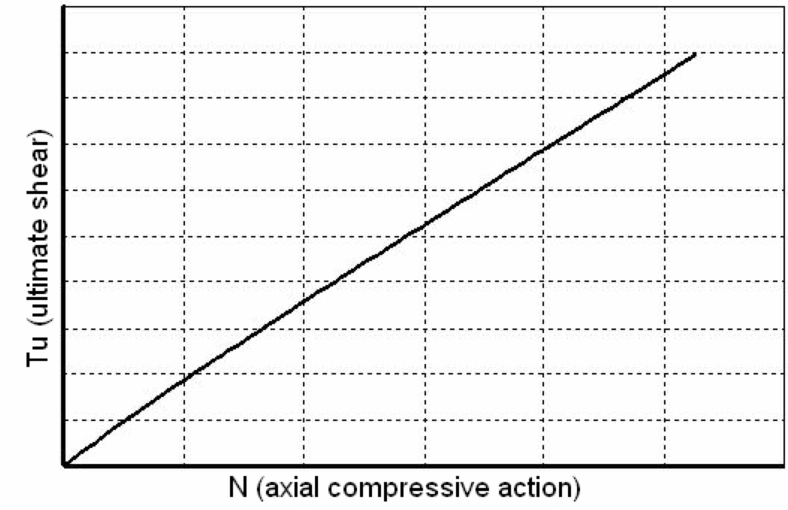
As for l'_R also l'_{lim} must be $\frac{N}{0.85 f_m t} < l'_{\lim} \le l$

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

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

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:

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



Mohr-Coulomb criterion for shear resistance

3.2.3 Shear: Turnšek and 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).

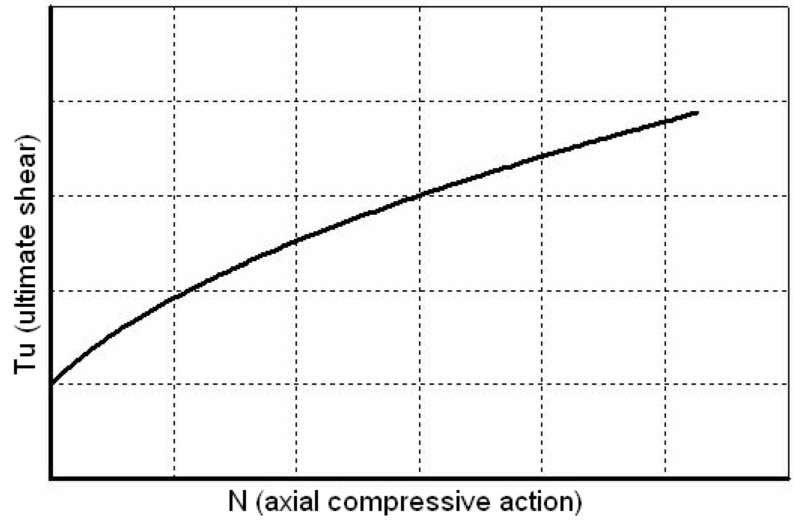


figure 5: Turnšek and Cacovic shear strength criterion

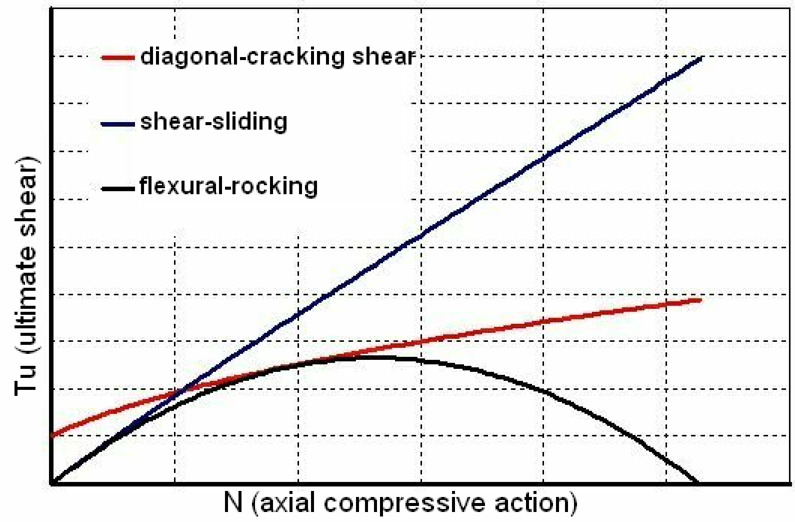


figure 6: Strength criteria comparison

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:

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

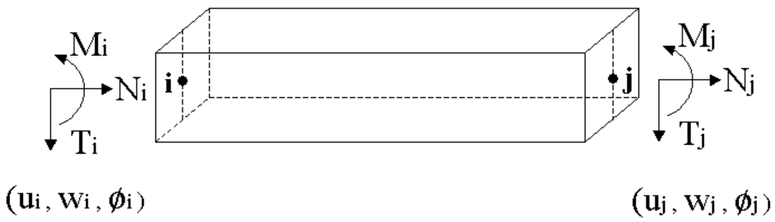
Where h is the height of the section of the panel, t is the thickness, f_{v0} is the shear resistance of the masonry without compression.
According to this the maximum bending moment is :

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

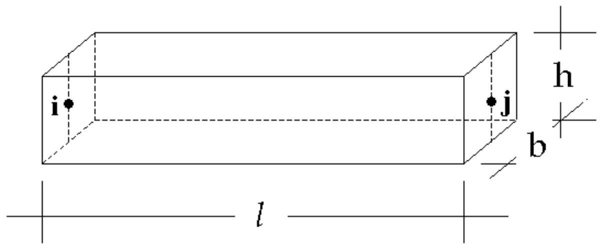
Where H_P is the minimum between the tension resistance of the stretched interposed element inside the lintel (for example a tie-road or tie-beam) and $0.4f_h ht$ where f_h the compression resistance of the masonry in the horizontal direction in the plane of the wall.

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...

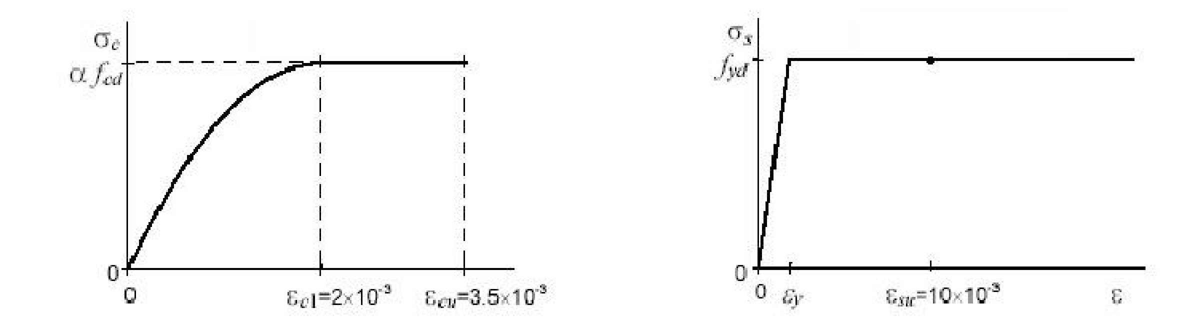
From "Point 8.5 Mixed structures with walls in ordinary or reinforced masonry - OPCM 3274":
In the area of masonry constructions, it is permitted to use structure with diverse technologies to support vertical loads, as long as the resistance to seismic action is entrusted entirely to elements of the 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 coefficientγc) 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

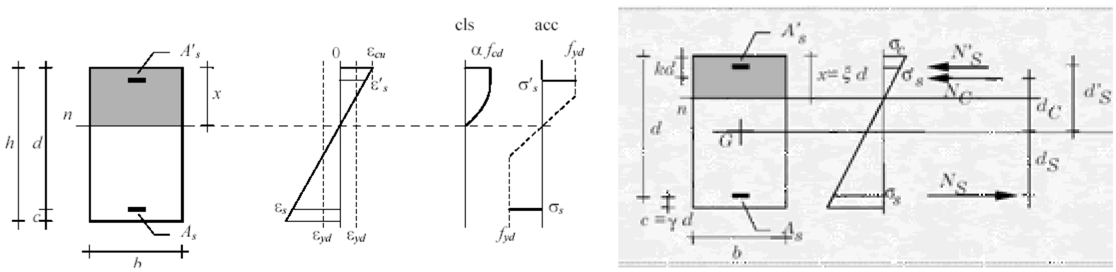
In accordance with point 5.4.1 and the relative specifications for existing buildings in chapter 11 of Ordinance 3274/03 and subsequent modifications and supplements, the check compares the values calculated for the moments with those calculated for resistance (limit values) on the basis of actually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

$N_{S'}$ = σs' As'

N_C = βξαfc 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 θu can be evaluated using direct experimentation, numeric modelling considering the contributions of concrete, steel and adherence, or using the following formulas:

$$\theta_u = \frac{1}{\gamma_{el}} 0{,}016 \cdot (0{,}3^{\nu}) \left[\frac{\max(0{,}01; \omega')}{\max(0{,}01; \omega)} f_c \right]^{0{,}225} \left(\frac{L_V}{h} \right)^{0{,}35} 25^{\left(\alpha \rho_{sx} \frac{f_{yw}}{f_c} \right)} (1{,}25^{100 \rho_d}) \quad (11.A.1)$$

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:

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

(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+(M0/Msdu)* 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 $\sum\varnothing$ 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 computation must be less than or equal to the sum of the resistance of the core reinforcement and the contribution of the other elements of the ideal lattice. In any case, the computed core reinforcement resistance must not be less than half of the shear computation.
The transversal reinforcement must be such that permits the following to be checked:
Vsdu ≤ Vcd + Vwd
in which:

$$V_{cd} == 0{,}60\ f_{ctd} \cdot b_w \cdot d \cdot \delta$$

$$V_{wd} = A_{sw} \cdot f_{ywd} \cdot \frac{0{,}90\ d}{s}\ (\text{sen}\ \alpha + \cos\alpha)$$

In these expressions a is the inclination of the transversal reinforcement with respect to the axis of the beam. Asw is the area of the transversal reinforcement found at the distance between centers s. d is a coefficient that takes into account the presence of normal strain and has the value:

δ $= 1$ if, in the presence of normal traction strain the neutral axis shears the section;

δ $= 0$ if, in the presence of normal traction strain the neutral axis results as external to the section;

d = $1+\dfrac{M_0}{M_{sdu}}$ in the presence of compression strain, *M0* and *Msdu* as defined above.
For raised shear resistant rebars it is recommended that the computation tension be limited to 0.8 *fywd*
Particular care should be taken with the dimensioning of the elements that undergo the straining actions for which it is possible that the computed shear resistance must be entirely entrusted to the core reinforcement.

To be precise, in compliance with that indicated in the code, in the case of existing buildings, the shear resistance is evaluated as for new constructions in non-seismic situations. In any case, the maximum contribution of the conglomerate considered should be equal elements without shear resistant transversal reinforcement.

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):

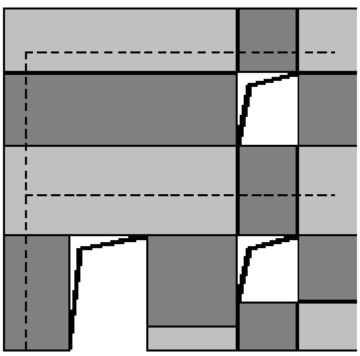
elastic phase permanence (E);
formation of a plastic hinge due to reaching the moment value limit (P);
collapse of the section after exceeding the maximum allowable rotation value (R).
Please note that the shear force characteristics are constant along theelement due to the concentrated actions in the 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

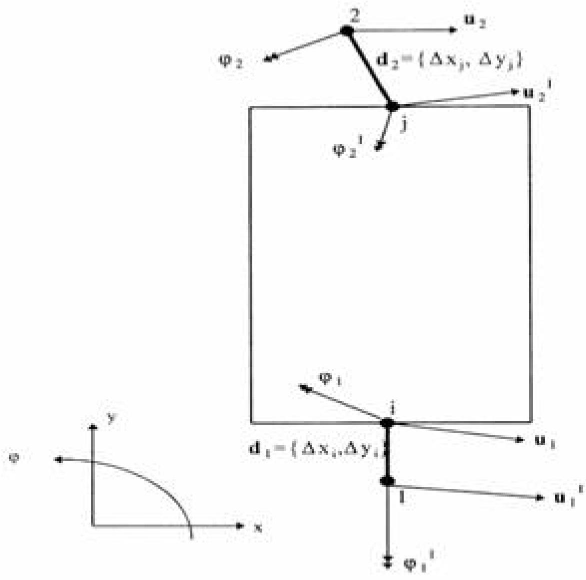
The three-dimensional modelling used is the direct result of observation of real building behavior and experimental tests. These allowed the introduction of some hypotheses about structural behavior of masonry constructions.
As mentioned above, damage mechanisms observed in buildings can be divided into 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.

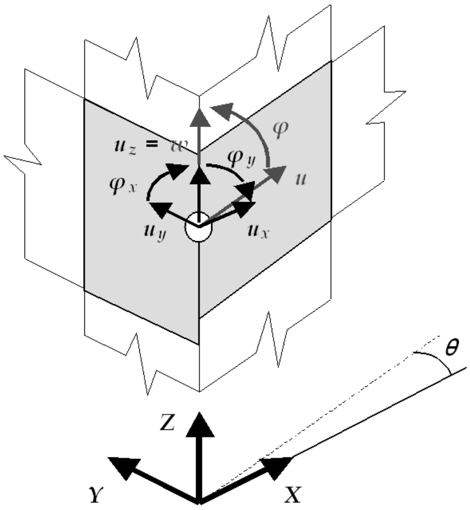
The rigidity matrix for elements without bending rigidity is easily found by eliminating all the limits that contain J from the element matrix. To manage the non-linearity, all of the elastic contributions due to the tie rods must be kept distinct. At each step, it must be verified if the tie rod that 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 φ *x* and φ *y*. 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:

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

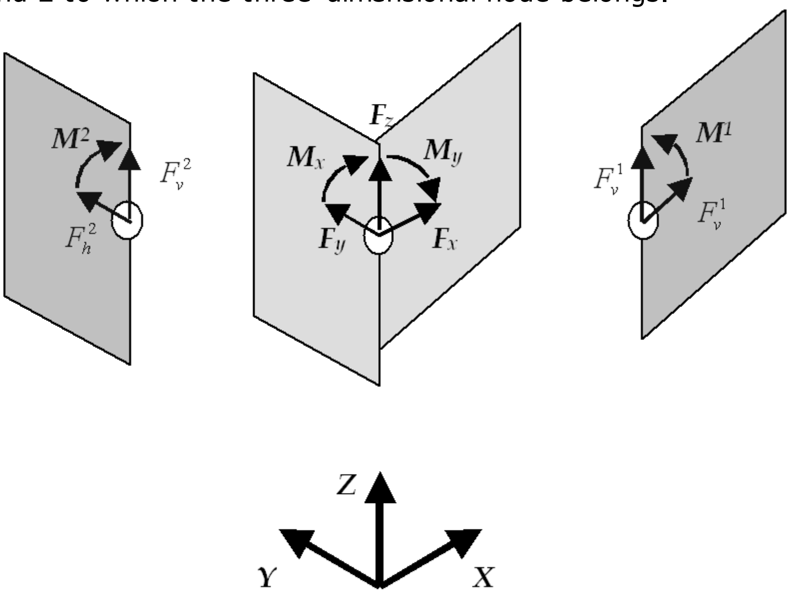
in which u, w, and φ indicate the three displacement components according to the degrees of liberty found in the fictitious node that belongs to the generic wall facing the plan according to angle φ. Similarly, the forces applied to the three-dimensional nodes are displaced according to the directions 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

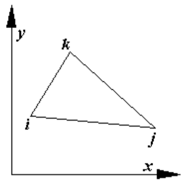
$$\begin{cases} F_x = F_h^1 \cos\theta_1 + F_h^2 \cos\theta_2 \\ F_y = F_h^1 \sin\theta_1 + F_h^2 \sin\theta_2 \\ F_z = F_v^1 + F_v^2 \\ M_x = M^1 \sin\theta_1 + M^2 \sin\theta_2 \\ M_y = -M^1 \cos\theta_1 - M^2 \cos\theta_2 \end{cases}$$

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

Differente languages and different codes are available in 3Muri.
3Muri has standard modules and add-on modules (protected by licence):

Code	**Italy**	**Switzerland**
• Italian sismic calculation [Details]	✓	✗
• Italian static calculation [Details]	✓	✗
• Eurocode 8 [Details]	✗	✓
• SIA [Details]	✗	✓
Language		
• Italian	✓	✗
• English	✗	✗
• German	✗	✓

✓ : Standard module

✗ : Add-on module. Covered by licence (contact distributors to get it)

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 - ***D.M. 14 settembre 2005***
➔Ordinanza 3274 come modificato dall' ***O.P.C.M. 3431 - 3 maggio 2005***
Le suddette normative prevedono al loro interno il calcolo dei parametri di vulnerabilità sismica secondo quanto prescritto dall' O.P.C.M. 3362 - 8 luglio 2004.
➔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. 14 settembre 2005

Le prescrizioni per questa normativa, sono le medesime riportate nella precedente tranne che per la computazione dei carichi.
Carico statico sui solai: Per questa normativa è necessario definire il solo fattore $\psi 2$
Carico sismico: Lo spettro di progetto per lo Stato limite di danno è differente da quello ultimo; è necessario definire le classi di importanza dell'edificio.

4.2.3 O.P.C.M. 3274 / 3431

Secondo quanto riportato nella normativa sismica OPCM-3274, si rendono necessarie due differenti verifiche; una per quanto riguarda gli stati limiti ultimi(SLU) e una per quanto concerne gli stati limite di danno(SLD).

4.2.3.1 Verifiche SLU

Gli elementi murari mobilitano la loro resistenza fino a quando raggiungono il valore massimo del drift per taglio o per presso flessione. Al raggiungimento di tale valore il contributo di resistenza apportato da quell'elemento viene meno. Il progressivo danneggiamento causa un decadimento dal suo valore di picco della curva push-over. Quando tale valore è arrivato all'80% di quello di picco si ricava il valore dello spostamento ultimo ("offerta" dell'edificio).
Dalla curva push-over dell'edificio si passa alla curva dell'oscillatore semplice associato, in modo da poter così calcolare il periodo del sistema equivalente che, attraverso lo spettro dettato dalla normativa permette di calcolare il valore massimo dello spostamento richiesto dal sisma ("domanda" del sisma).

I controlli : D_{max} *(del sisma)* < D_u *(dell'edificio)* ; $q^*<3$
indicano il corretto superamento della verifica.

q* indica il rapporto tra la forza di risposta elastica e la forza di snervamento del sistema equivalente.
Oltre ai parametri necessari a questa verifica, il programma calcola anche il valore dell'accelerazione limite a collasso che genera il valore dello spostamento richiesto dallo spettro pari a quello ultimo.

4.2.3.2 Verifiche SLD

Lo spostamento massimo a SLD (D_d) è il minor valore tra:
Spostamento corrispondente al massimo taglio alla base.
Spostamento che genera il drift ultimo di piano (valore dato dalla norma).
Lo spostamento massimo secondo lo spettro della normativa si ottiene riducendo l'accelerazione di un fattore pari a 2.5.

La verifica risulterà soddisfatta seguendo il seguente controllo:
D_{max} (del sisma a SLD) < D_d (dell'edificio)

4.2.4 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

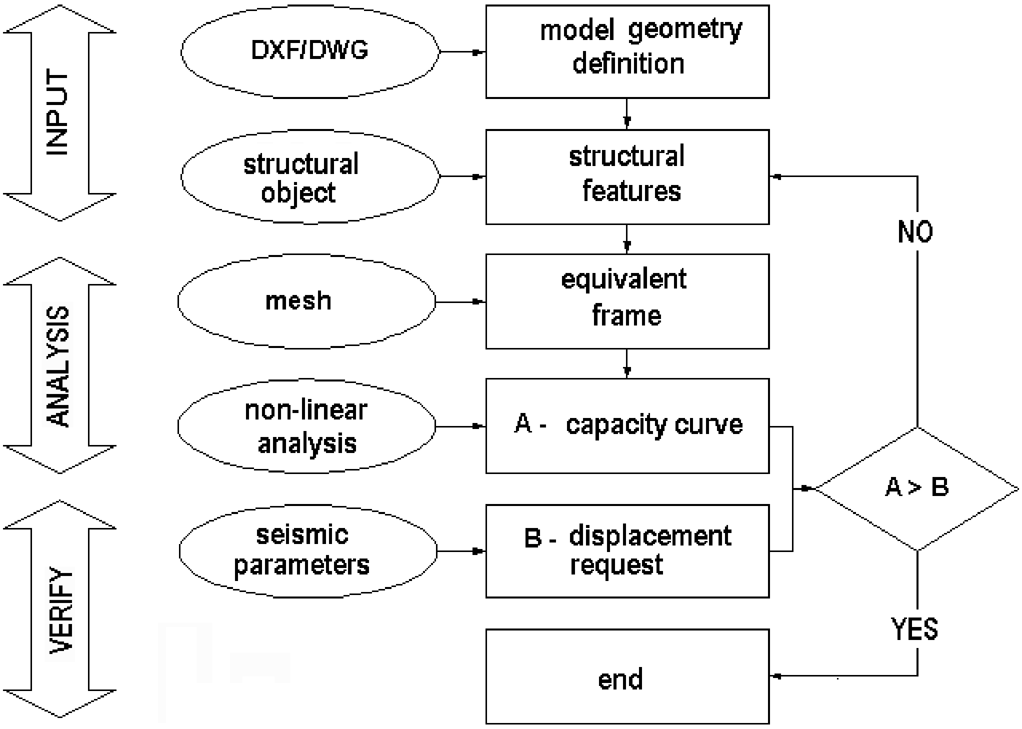
Reference code are:

- **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

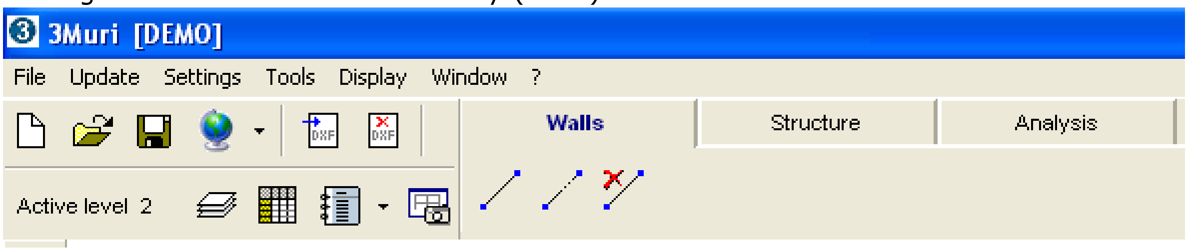
In this phase, the user inserts the data necessary for performing the analysis.

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.

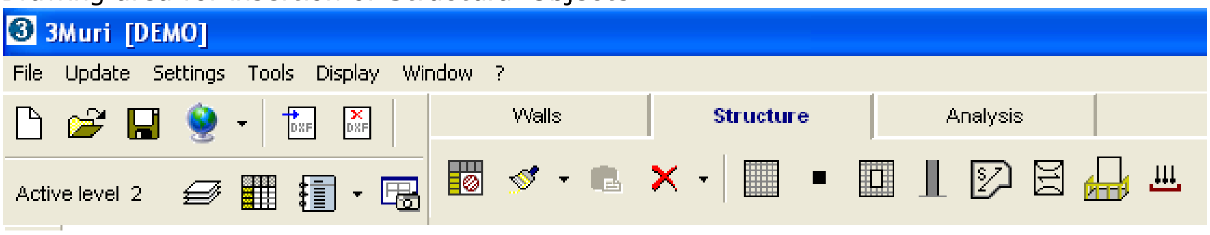
Practical rules for effective importation - Prepare the tables before importing:
- Position the origin of the reference system in one of the vertexes of the plan.
- Define the limits of the graphic area around the plan to be imported (CAD program limits command).
- Delete contiguous designs and images 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



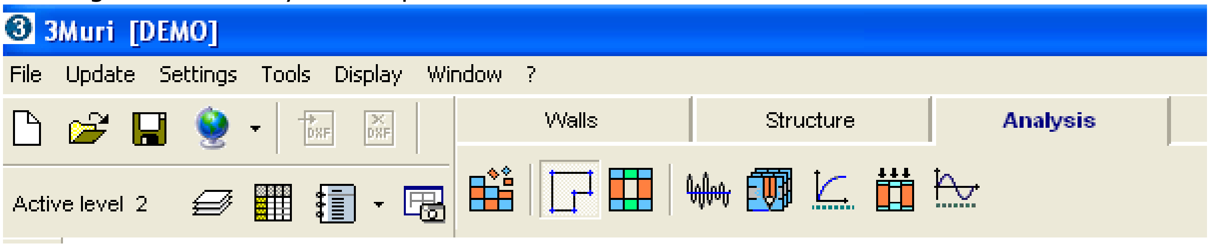
5.2 Analysis Phase

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.

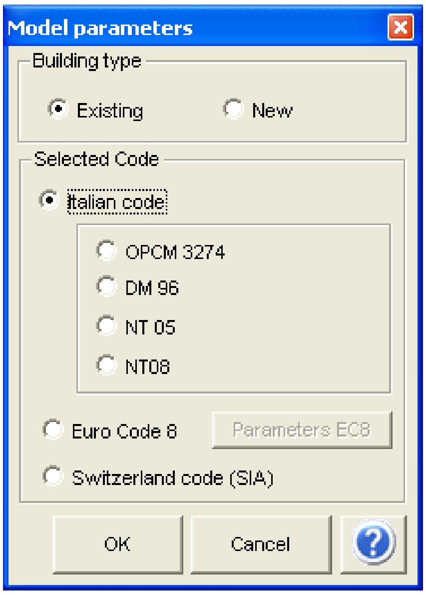
<u>Seismic parameters</u>
Definition of seismic parameters and evaluation of the parameters derived from the structure's capacity curve permits determination of the request in terms of displacement of the spectrum for the project at hand.
The check compares the two displacements (forces in the case of 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.

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

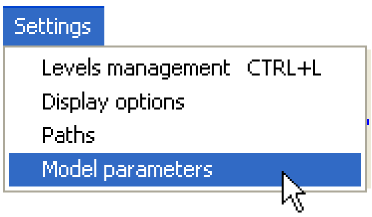
The window "Model parameters" is loaded creating a new project.



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.



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

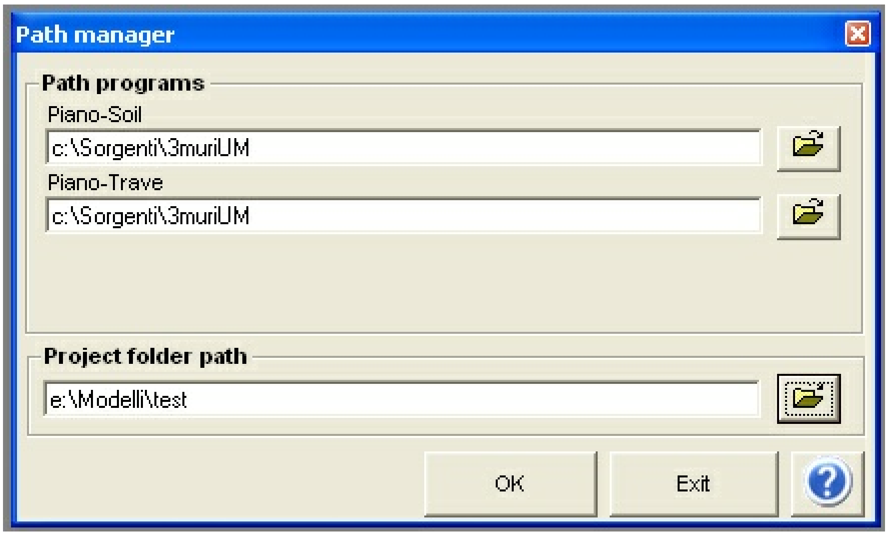
Model parameters

If selected code is **"Eurocode 8"**, it is possible to modify EC8 settings according to national annex.

Select "EC8 parameters" to manage Code parameters.

6.2 Path Selection

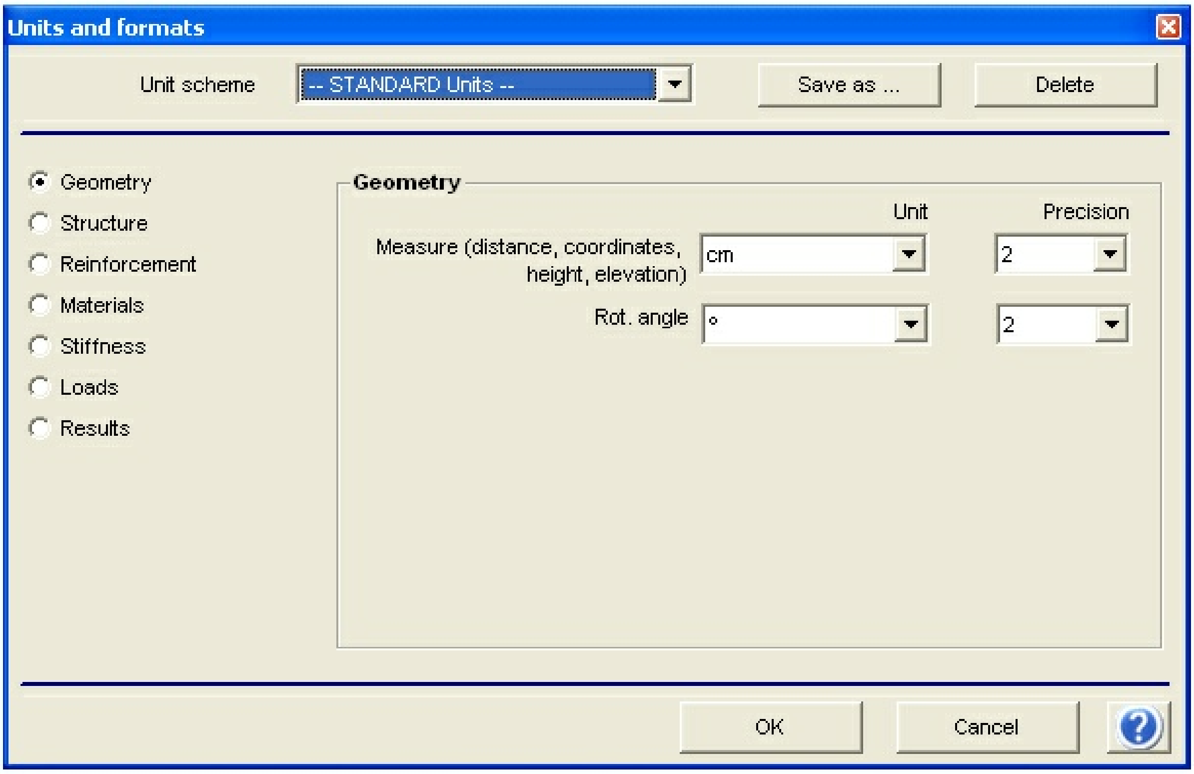
The project paths can be managed by 3Muri.



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

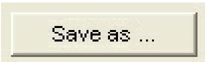
6.3 Units and formats

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".



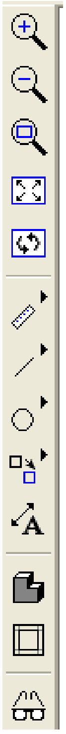
With the command "Save as..." we can save the parameters that we have modified.
Insert the name of the unit system defined by user.

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.

6.4 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.



Import DXF/DWG: imports a DXF or DWG file

Zoom: controls zoom on the project design. The zoom can also be controlled using the mouse wheel.

Redisplay: allow regeneration of the display in case of display errors,

Measurement tools: manages measurement of the design elements.

Insert line: insertion of generic lines that are vertical, horizontal, or perpendicular to other elements, that support insertion of the structure.

Insert circle: insertion of a three-point circle, or a circle given the center and the radius, to aid in insertion of the structure.

Edit drawing: permits editing of the support graphic entity for designing the structure

Copy: copies a graphic element.

Move: moves graphic entities.
The entity to be moved is selected, and then highlighted in red. After this, when the right mouse button is pressed, the program will apply the displacement vector.

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

Divide: divides an object into two parts.
Select the line with the left mouse button. Then, apply the division in the desire point.

Trim: truncates two intersecting lines.
Two lines are selected based on the direction in which the truncation should be applied.

Extend: Extend a line out to another object (line, circle, polyline).
First select a line to extend. Then, select the object to which it should be extended.

Delete: deletes graphic entities.
Objects are selected with the left mouse button, and then highlighted in red. Pressing the right mouse button will delete them.

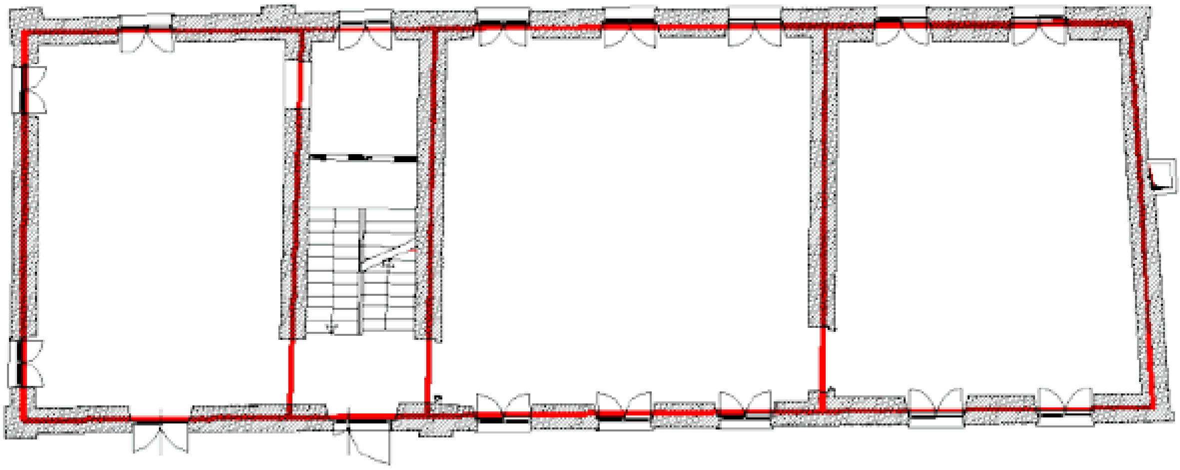
Undo delete: Undo the delete command.

Text: Allows insertion of text boxes in the drawing and the dimesioning relative to the structure.

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

6.5 Wall

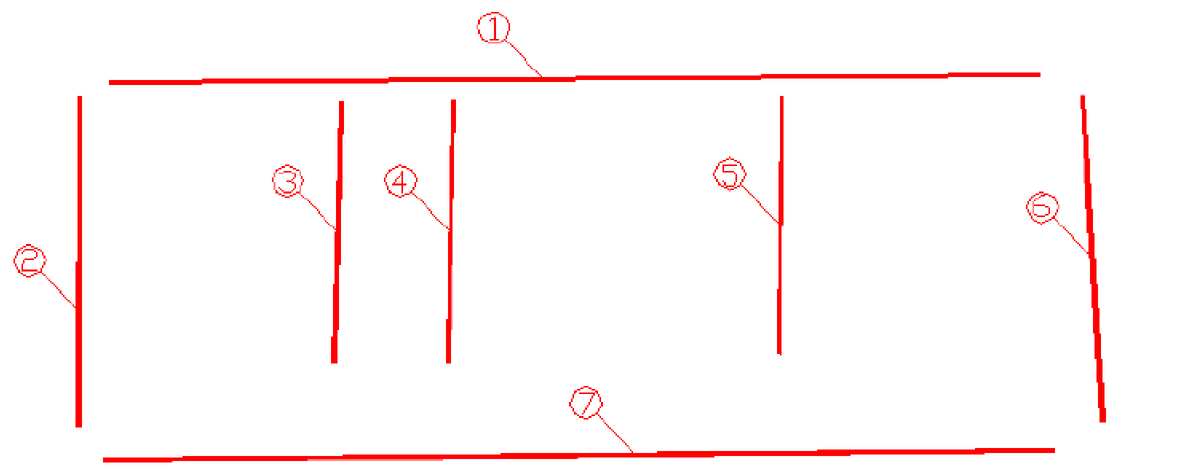
The lines that represent the walls are the basis for the definition of: masonry panels, beams, tie rods, and columns.
The wall represents the synthesis, taken from the architectural design, of the structure to be modelled, both on the horizontal as well as the vertical plane.
Synthesis because it is 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).



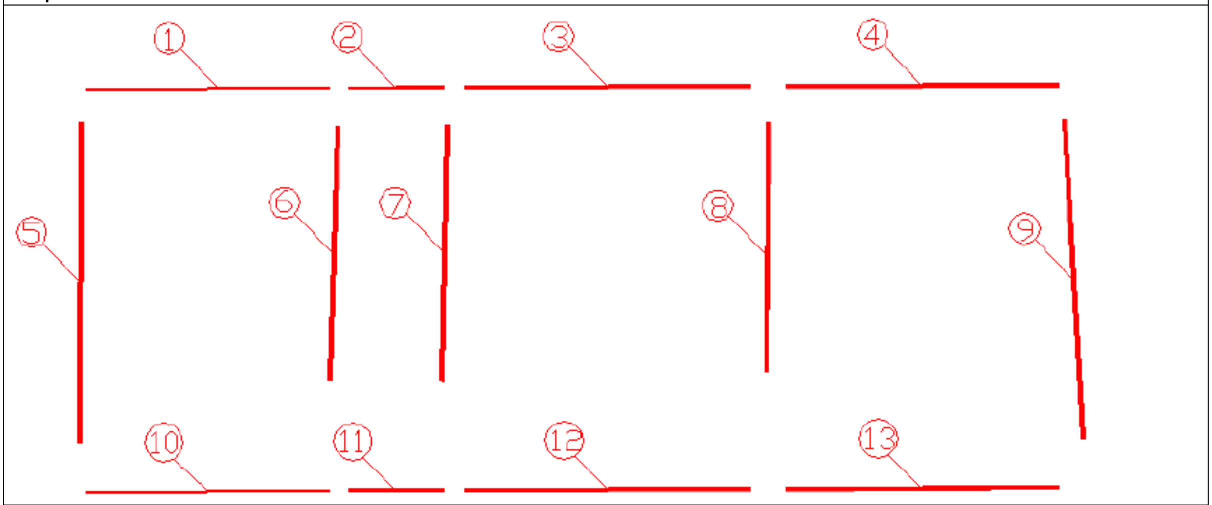
Exploding the wall system it becomes clear why various contiguous segments with structural environment definitions, belonging to the same tangent, must be modelled using a single wall. If wall segments do not have definition in the structural environment on any level, then in place of a single wall,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

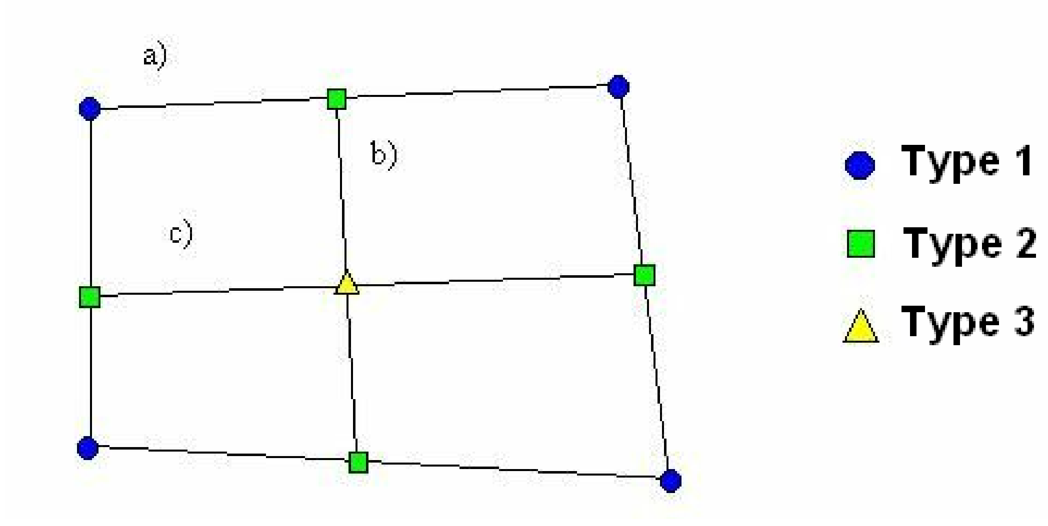


The walls can be managed on all levels, and can be deleted, added to, or modified in all design phases.

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).

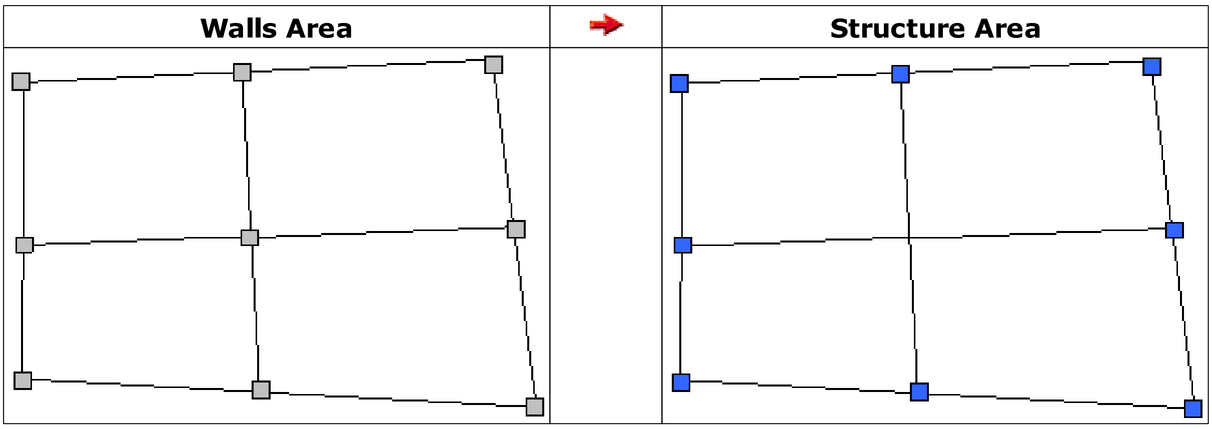
During the insertion phase, a third type of node can be created. This is automatically derived from the computation of the intersection between walls. For example, between the intersection of walls b) and c).
These TYPE 3 nodes (which are indicated with a yellow triangle) are found in an intermediate 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.

6.6 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.

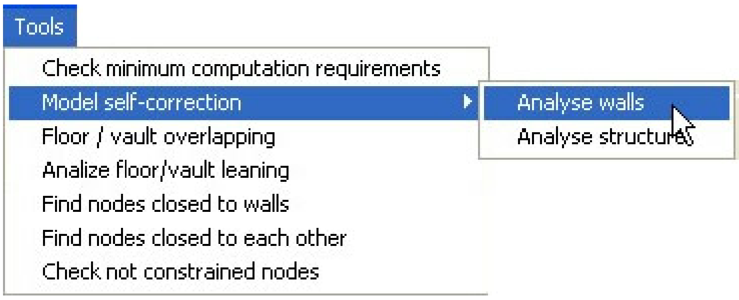


A column can be inserted only in correspondence with a wall endpoint or segment point.

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.

6.7 Checking Models

During the model creation phase, disorganized support graphic entities or simple human error can lead a designer to make involuntary mistakes. To assist with this problem, the program includes an automatic procedure which checks that all of the basic rules for project creation have been met.
This correction procedure can be accessed from the Tools menu.



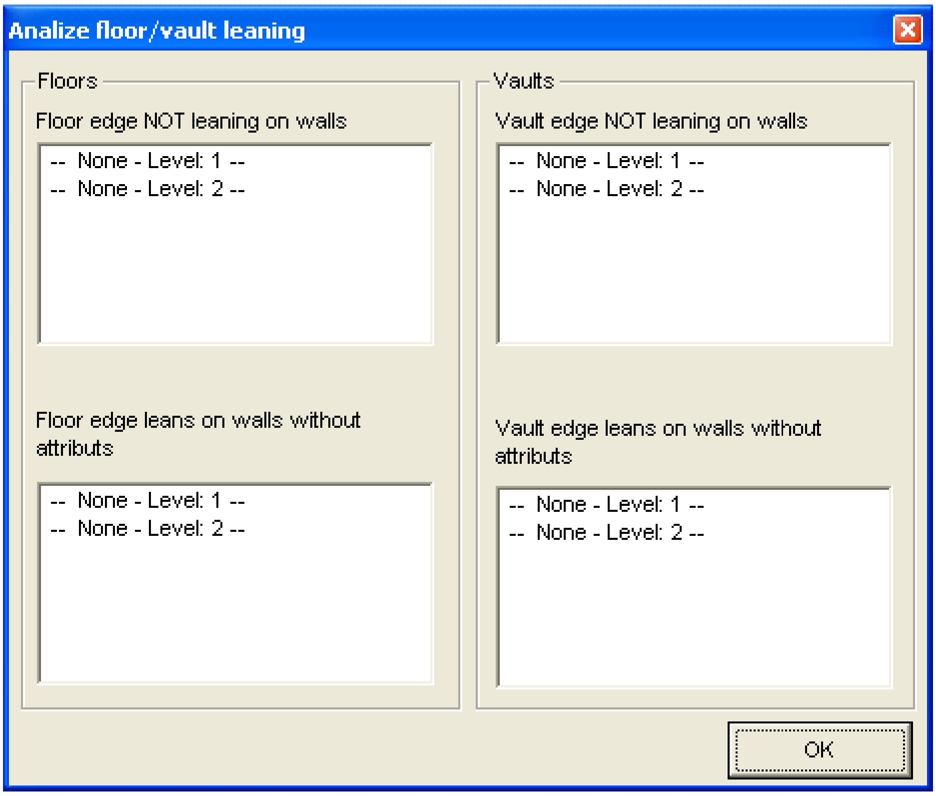
Check minimum computation requirements:
Checks the "box" behavior of the building, checking that there are no nodes that belong to asingle wall. If this check comes back negative, the user is told at which point the problem was found.

Model self correction > Analyse walls:
This performs an element nodes check in the wall environment, correcting any errors that may be found. It checks that nodes of the three typologies illustrated above have been correctly inserted.

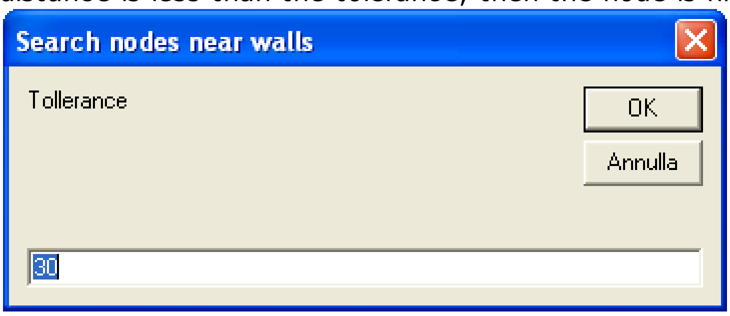
Model self correction > Analyse Structure:
This checks for the relations between the segment points and the wall endpoints. This is useful to keep proliferation of segment points under control for successive insertions in the same positions.

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

Analyze floor/vault leaning:
This 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.

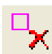


Find nodes near the walls:
Allows the identification of problems linked to the graphic insertion of a wall that has an end at an intermediate point of another wall. If the node is not found on the wall and the distance is less than the tolerance, then the node is highlighted.



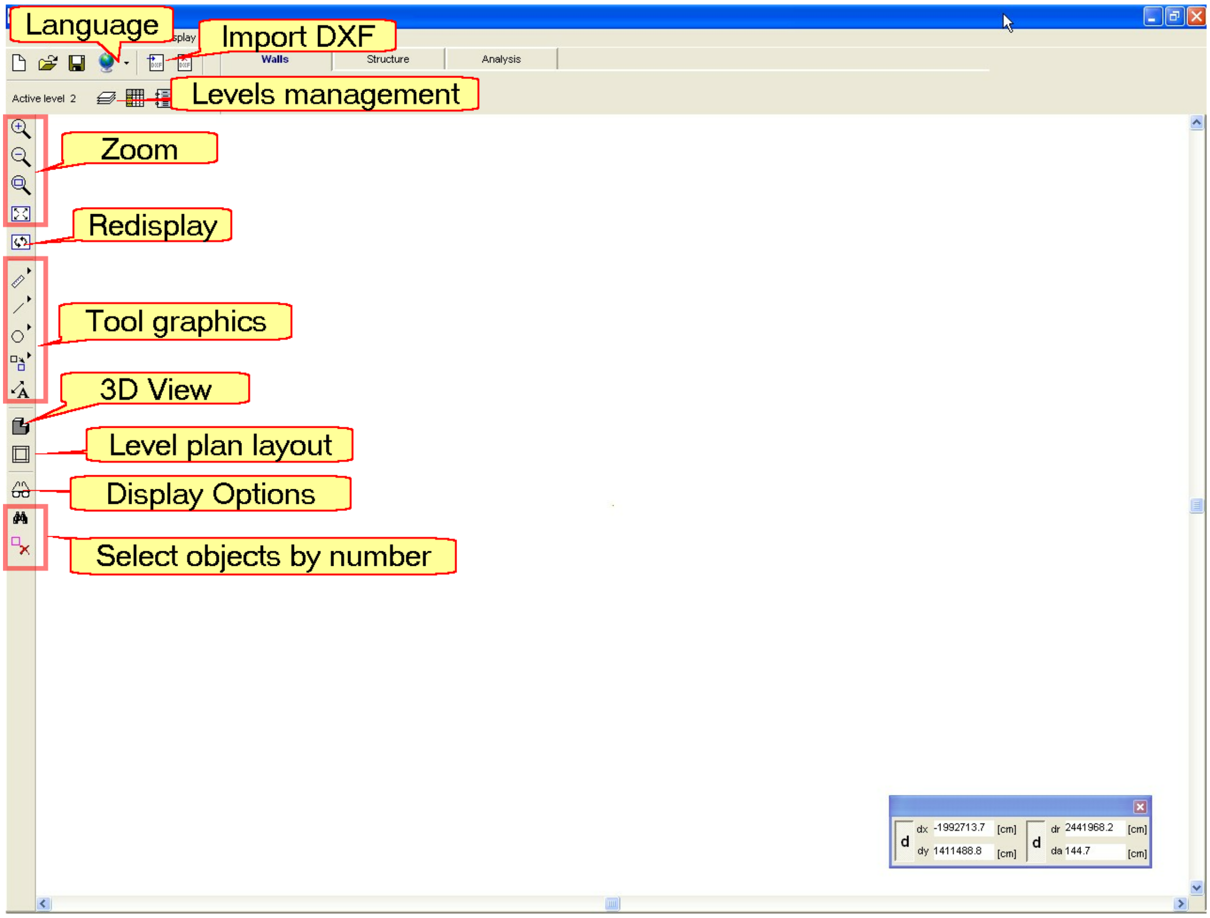
Find nodes that are near to each other:
Allows the identification of problems linked to the graphic insertion of walls that must have a shared end. If the nodes do not coincide and the distance between the two is less than the tolerance, then the node is highlighted.

Constrained nodes check:
Checks that there are no foundation nodes without their respective constraints. This check is important when the foundation has various elevations.

The button that allows deletion of the graphic symbols for selection of elements can be found in the lower part of the left vertical toolbar.

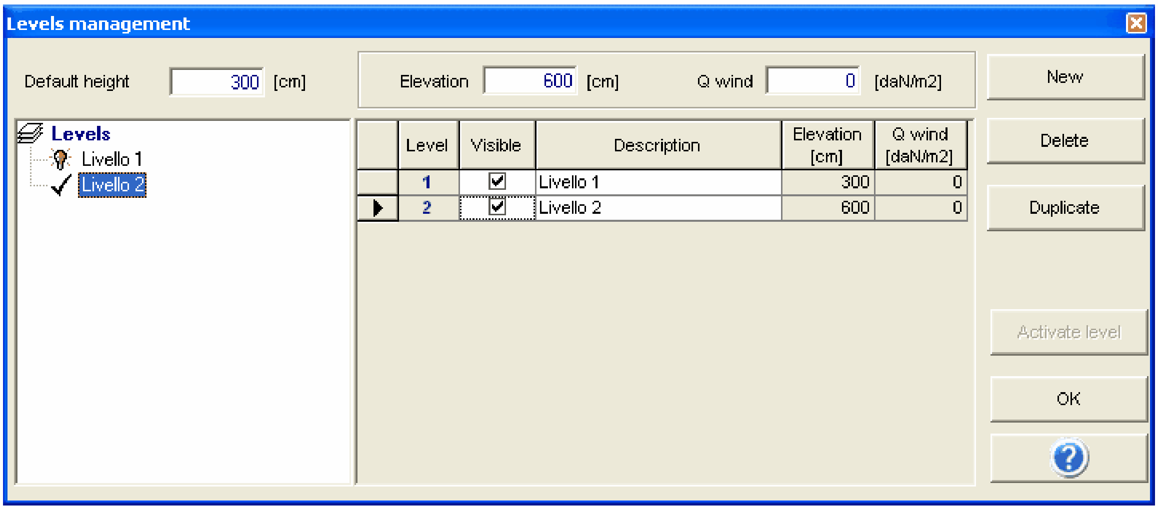
It is recommended that all users take advantage of these verification procedures, both when creating the model and when finishing before proceeding to computation.

7 Main commands



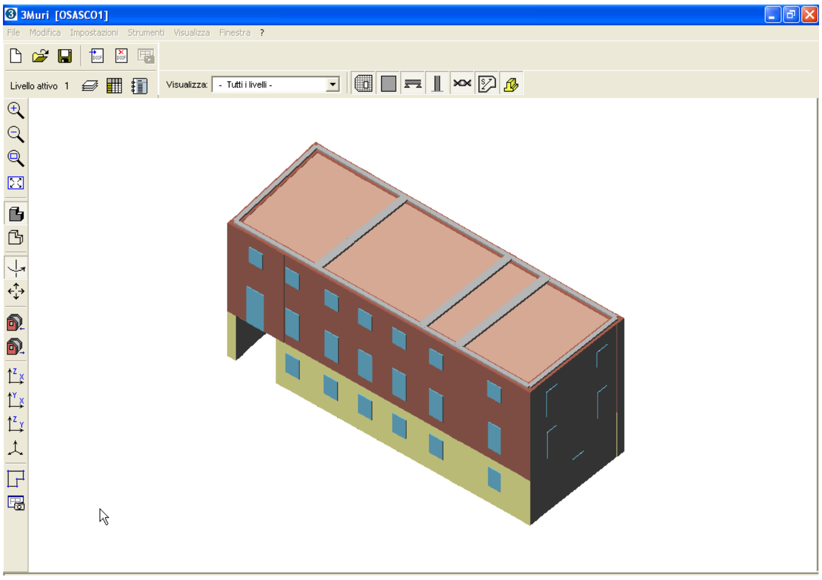
7.1 Levels management

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.

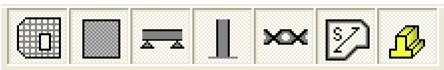


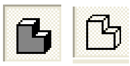
7.2 3D View

Accesses a window that shows the structure in 3D.

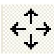


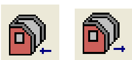
Commands for the "3D View" window:

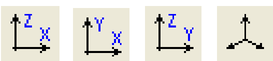
Elements Display: activates/deactivates display of walls, beams, columns, tie rods, floors, and foundations.

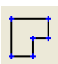
 3D View: allows the display to be shown in rendering or in wire-frame.

 Rotate: allows the model to be rotated, using the right mouse button.

Move: allows the model to be moved, using the right mouse button.

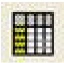
 Move the point of view: the two commands allow the point of view for the structure to be moved closer or farther away by pressing the left mouse button until the model is dissected.

 Perspective Views: allows the display to be shown according to the main floors of the reference system.

 Back to design: Exits from the "3D View" window and returns to the design area.

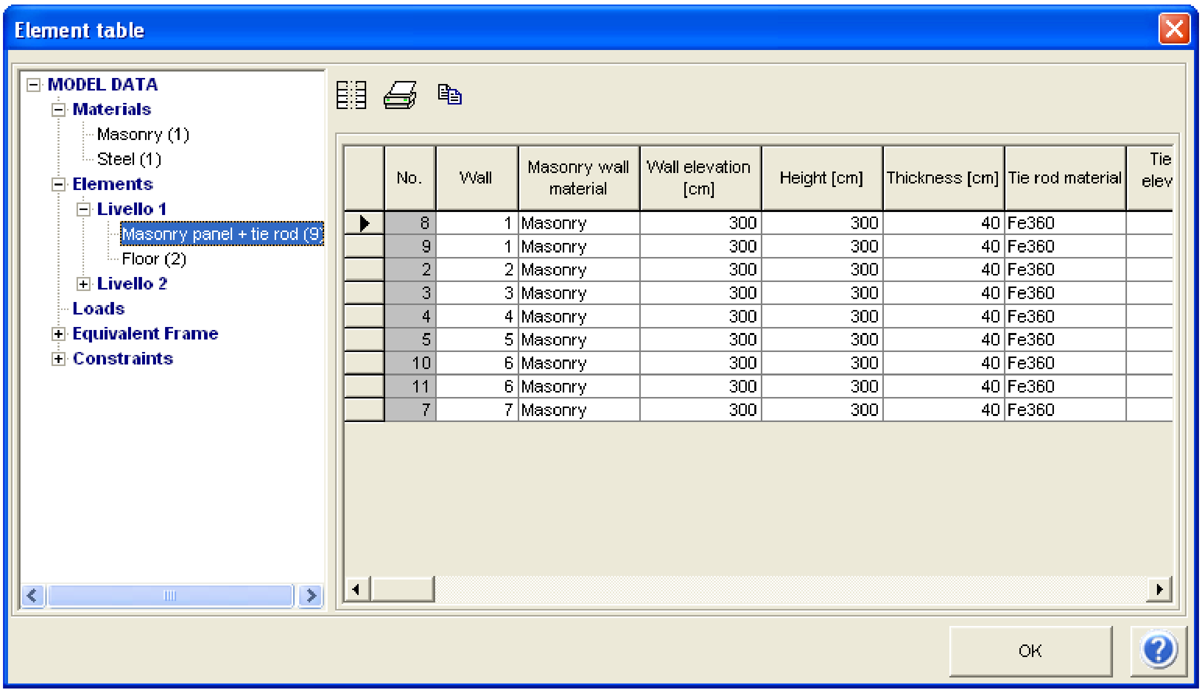
Save image: Saves an image from the "3D View" window.

7.3 Table



When the table is opened, using the button, a window will open that shows the characteristics of everything the user has inserted through the interface during the model creation phase. The tree structure on the left makes navigation through the tables easier. The tree is organized into five main branches:

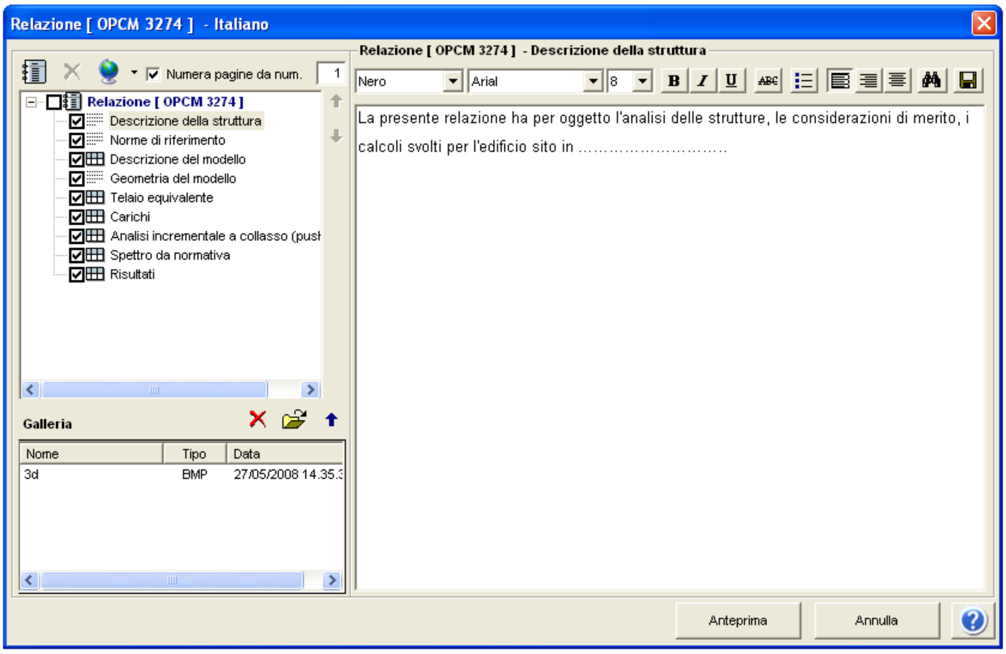
Materials: This contains the material typologies used in the project, with their mechanical characteristics.
Elements: This contains the elements used, divided by typology (according to that indicated in the characteristics definition window, described below), grouped by level.
Equivalent frame: This contains everything that has to do with mesh definition, from individuation of the geometric position of the noes, to definition of the characteristics of the elements, grouped by wall.
Constraints: This contains the identification number of the constraints, with the rigidity relative to the degrees of liberty for the node in question. In the table, the letter "V" indicates a perfect constraint with infinite rigidity.



No.	Wall	Masonry wall material	Wall elevation [cm]	Height [cm]	Thickness [cm]	Tie rod material	Tie elev
8	1	Masonry	300	300	40	Fe360	
9	1	Masonry	300	300	40	Fe360	
2	2	Masonry	300	300	40	Fe360	
3	3	Masonry	300	300	40	Fe360	
4	4	Masonry	300	300	40	Fe360	
5	5	Masonry	300	300	40	Fe360	
10	6	Masonry	300	300	40	Fe360	
11	6	Masonry	300	300	40	Fe360	
7	7	Masonry	300	300	40	Fe360	

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 seismic check report or the static check report.
When the button is activated on the tool bar, the report creation window opens.

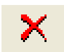
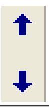


On the left the subjects for designing masonry buildings are found in order. The user can decide what to include in the report by selecting the box on the left of the descriptions.

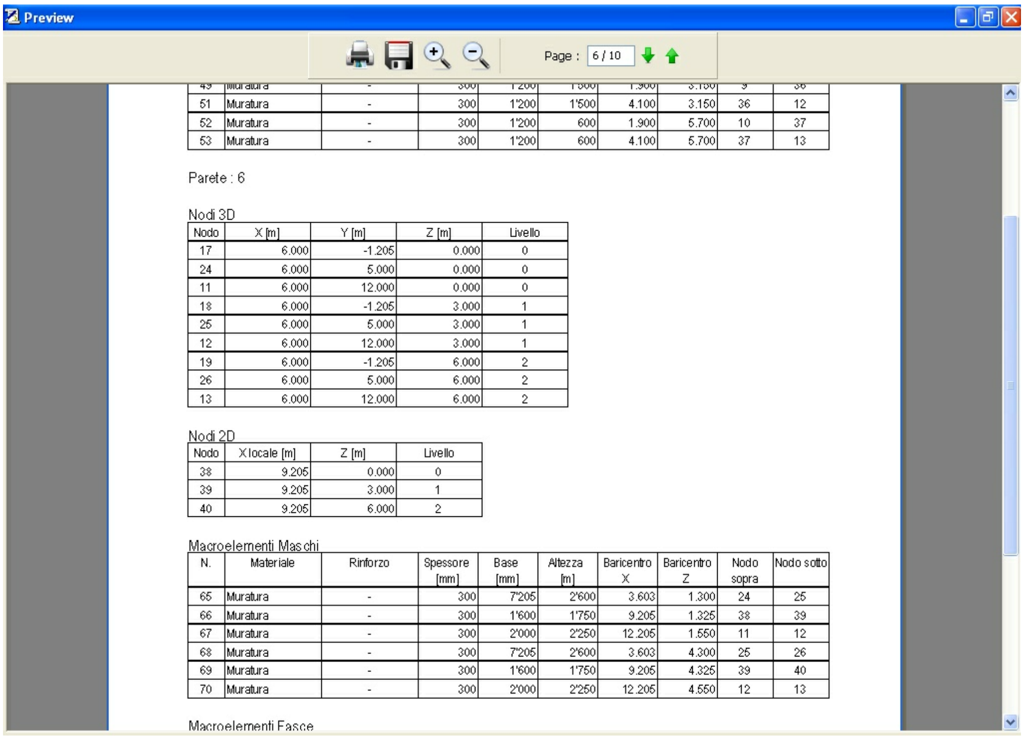
 With this button, the user can select the language for the report.

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 saves the image seen on the screen at that moment.

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.



51	Muratura	-	300	1'200	1'500	4.100	3.150	36	12
52	Muratura	-	300	1'200	600	1.900	5.700	10	37
53	Muratura	-	300	1'200	600	4.100	5.700	37	13

Parete : 6

Nodi 3D

Nodo	X [m]	Y [m]	Z [m]	Livello
17	6.000	-1.205	0.000	0
24	6.000	5.000	0.000	0
11	6.000	12.000	0.000	0
18	6.000	-1.205	3.000	1
25	6.000	5.000	3.000	1
12	6.000	12.000	3.000	1
19	6.000	-1.205	6.000	2
26	6.000	5.000	6.000	2
13	6.000	12.000	6.000	2

Nodi 2D

Nodo	X locale [m]	Z [m]	Livello
38	9.205	0.000	0
39	9.205	3.000	1
40	9.205	6.000	2

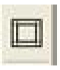
Macroelementi Maschi

N.	Materiale	Rinforzo	Spessore [mm]	Base [mm]	Altezza [m]	Baricentro X	Baricentro Z	Nodo sopra	Nodo sotto
65	Muratura	-	300	7'205	2'600	3.603	1.300	24	25
66	Muratura	-	300	1'600	1'750	9.205	1.325	38	39
67	Muratura	-	300	2'000	2'250	12.205	1.550	11	12
68	Muratura	-	300	7'205	2'600	3.603	4.300	25	26
69	Muratura	-	300	1'600	1'750	9.205	4.325	39	40
70	Muratura	-	300	2'000	2'250	12.205	4.550	12	13

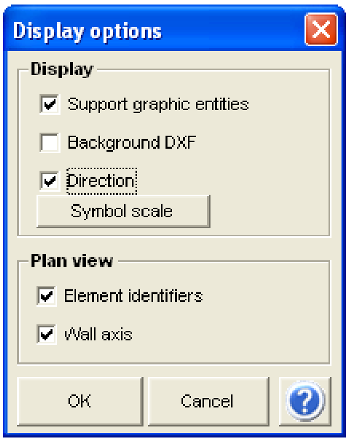
Macroelementi Fasce

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

7.5 Display Parameters

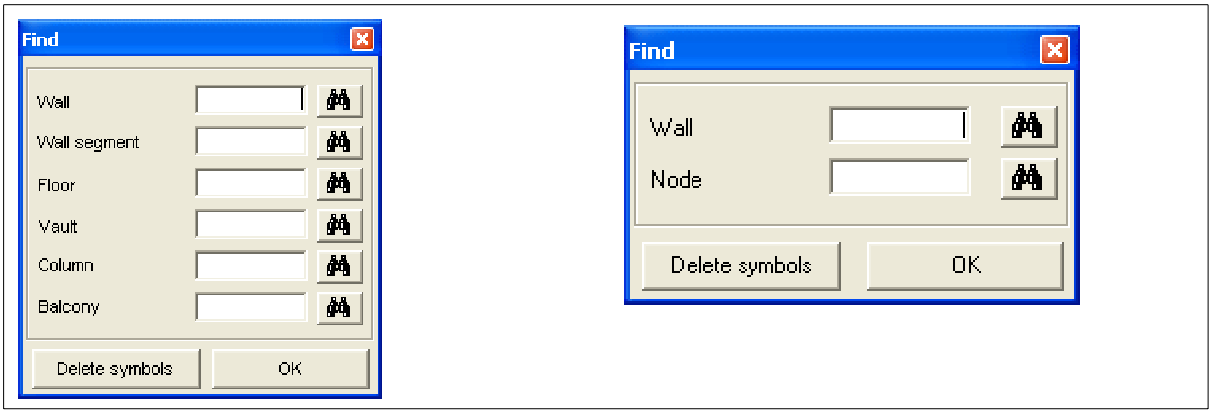
 Level plan layout: Shows the level plan in schematic mode. The various typologies of structural elements defined in the structural environment are shown.

 Display Options: This instrument allows the user to decide what to show in the video, choosing between the various choices seen in the window.



"Direction" shows the local reference system for the walls so that the designer can understand the eccentricities sign of the masonry panels with respect to the wall.

 Select objects by number: The command "Find" allows a wall, wall segment, floor, column, or a balcony to be found in the drawing area, if its identifier is known.



Remember that wall segments are segments of walls that are assigned definitions (e.g. M33, T122, C54). In the space, the identification number must be inserted -- not the letter which precedes it, which only indicates the type of element.

 Create image from the screen: This produces an image file (*.bmp) from the screen being used in the report area.

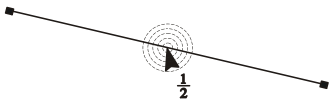
7.6 Snap

The program includes an automatic recognition system for the important points of the support graphic entities, on the typical .dxf files imported from a generic CAD system or on the walls.

The snaps described above are available during the wall insertion phase.
These snaps do not need to be activated by the user. They automatically become available based on the position taken by the mouse arrow on the visualized entity.

	End
	Middle
	Intersection
	Perpendicular
	to the Wall
	Selection

When the cursor is positioned on the drawing area, the program will find the model entity that is closest to the cursor among the entities that are found or that intersect the identification area.
The shape of the cursor indicates the type of entity that has been identified.

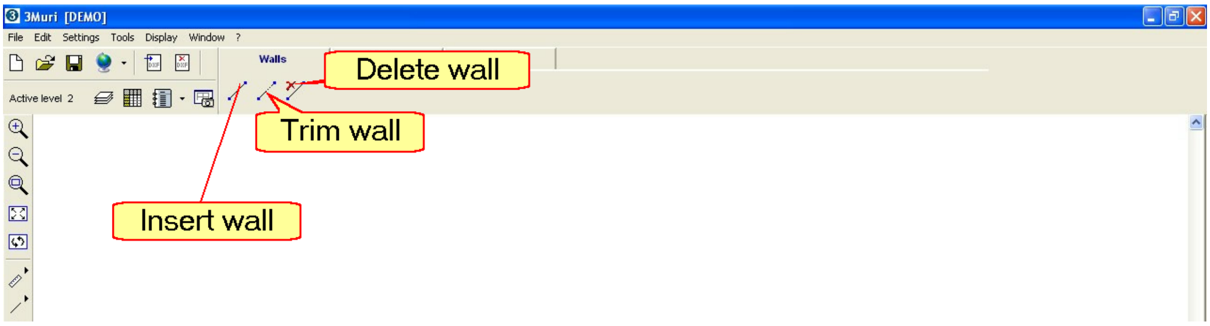


The cursor will become red and take the from indicated above to identify the selection. When the cursor is working in this mode, the entity selection mode is also active.

7.7 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.

8 Geometric Definitions



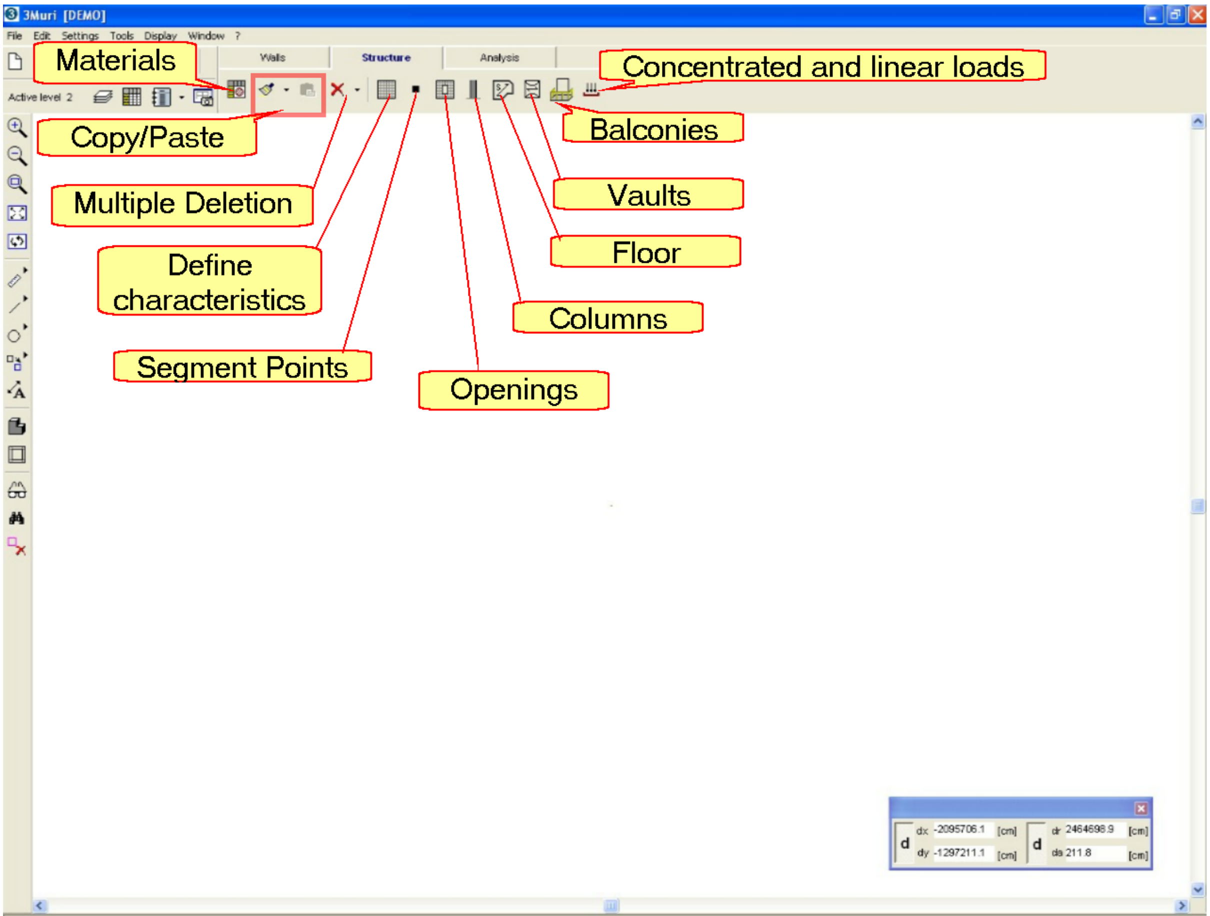
Insert wall: inserts a wall.
Remember that, as stated above, "wall" means a continuous length of masonry, R.C. walls, beams, or tie rods -- it can also be multiple segments all resting on the same tangent.
After selecting the icon, use the left mouse button to insert successive nodes that identify one or more walls. To exit from the command, use the right mouse button.
Insertion of elements can occur in assisted mode using snaps (on graphic entities or on walls). Alternatively, it is possible to insert the information for the effective length of the walls by entering the coordinates (cartesian or polar).

The buttons with the letter "d" are active when the relative coordinates appear in the corresponding text boxes. If the button is deactivated, the absolute coordinates are shown in the corresponding text boxes.
During the model generation phase it is possible to directly access the desired coordinate by entering the corresponding letter. For example, enter "x" to insert coordinate x.

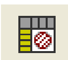
Trim wall: lengthens or shortens an existing wall.
First the wall to be lengthened or shortened is selected, and then the reference wall.

Delete wall: Deletes a wall.

9 Characteristics of the Structure



9.1 Materials

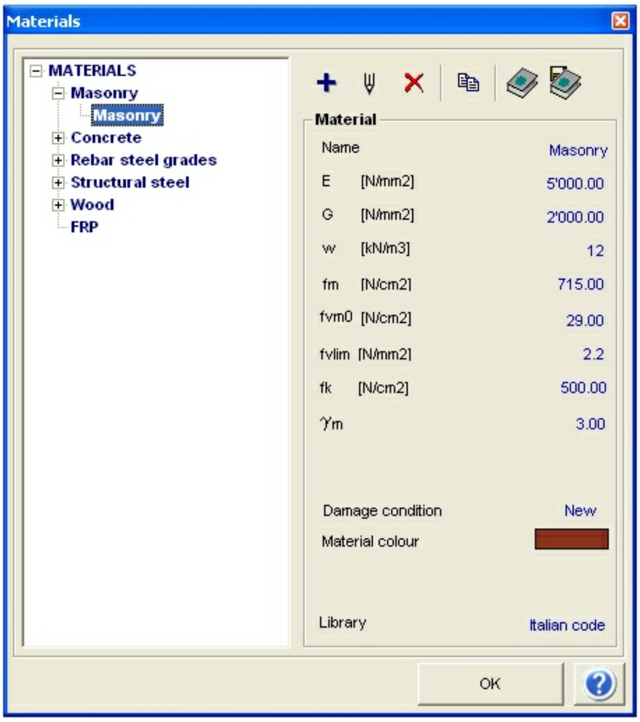


Select the icon, and window will open. In the window are the characteristics of masonry materials (concrete, steel, and wood) generally used in structural objects (masonry panel, tie rods, beams, columns, and floors). It is possible to modify or create new mechanical characteristics for the materials. 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

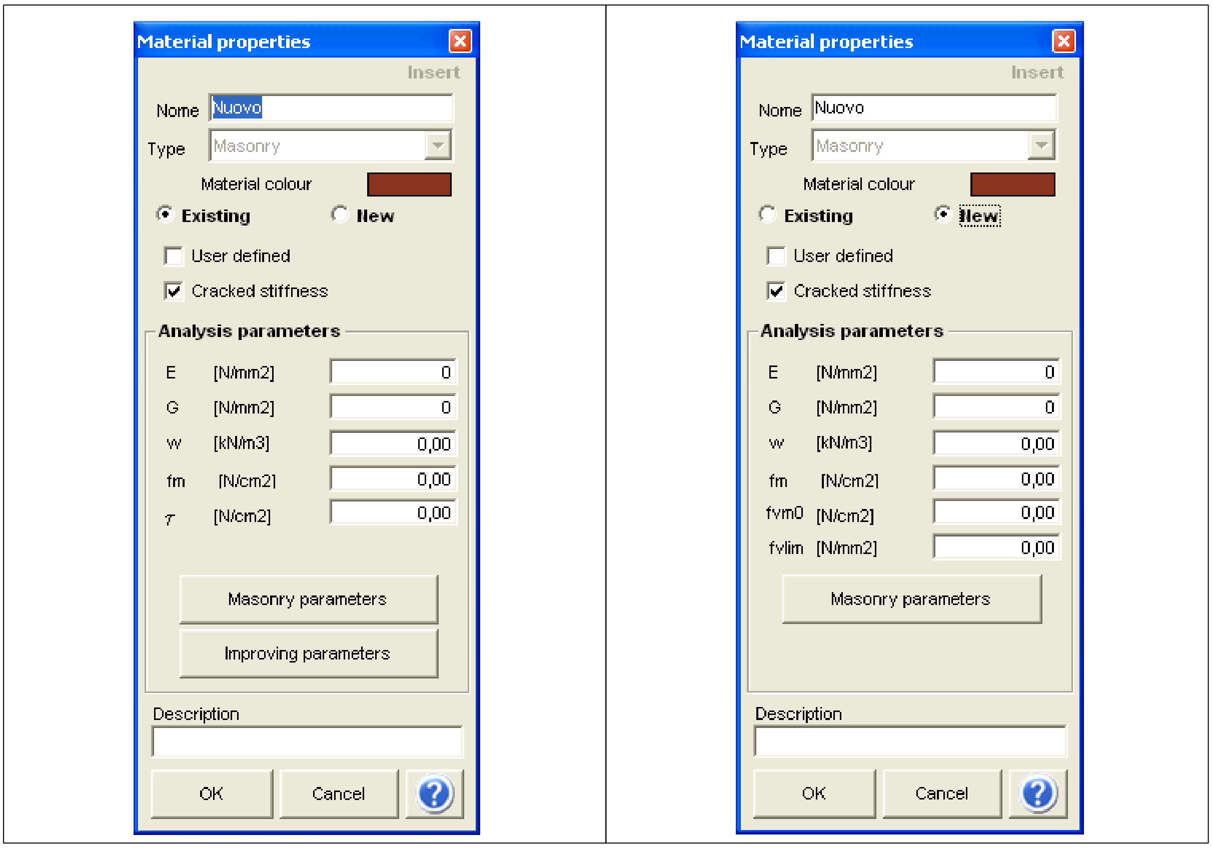
	Copy a material
	Library explorer
	Save User library

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



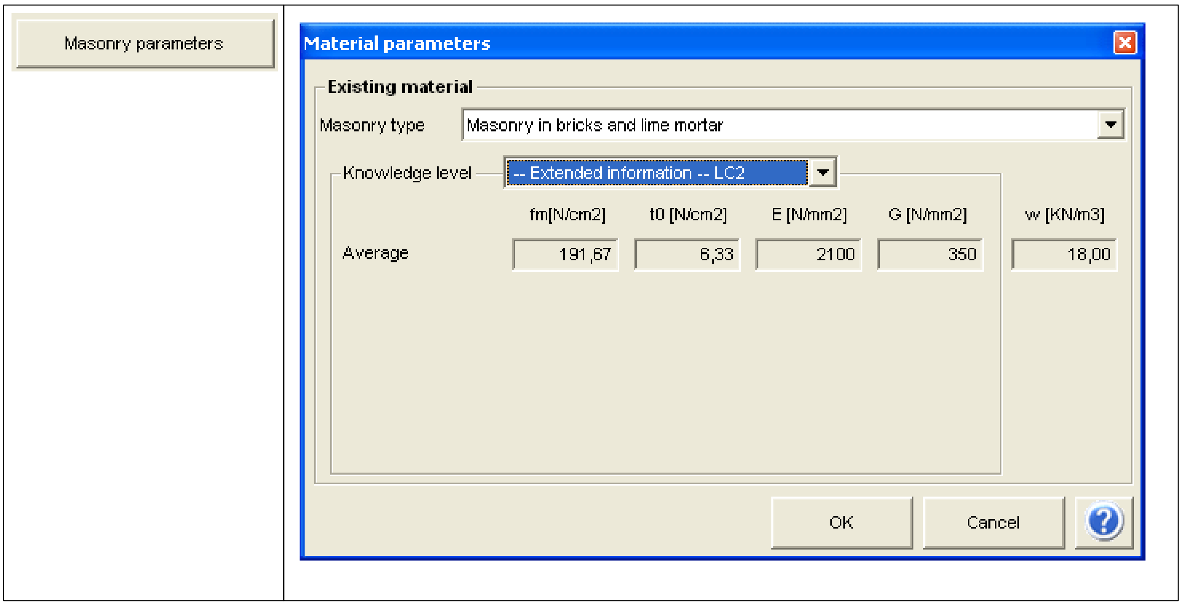
When a new typology of masonry material is inserted, there are two options:

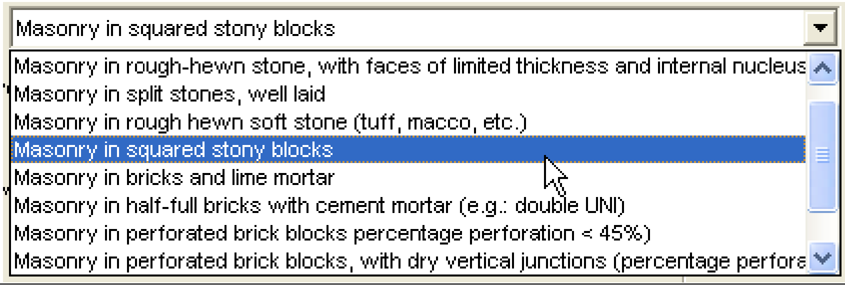
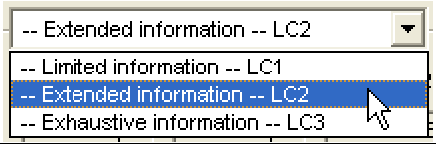
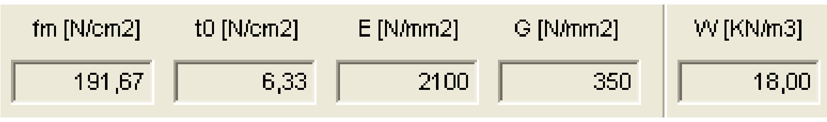
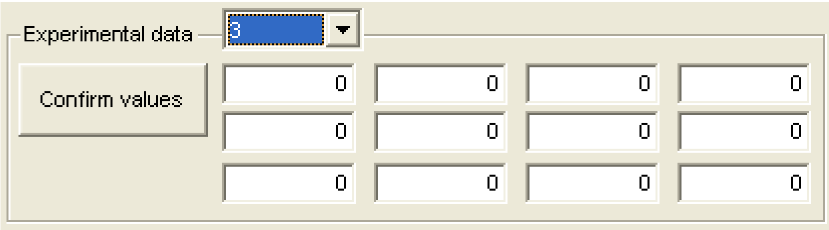
Existing Material	New Material



In the window for insertion of masonry material characteristics, there are buttons that help the user to identify these parameters.
Alternatively, the user can decide to directly insert the values.

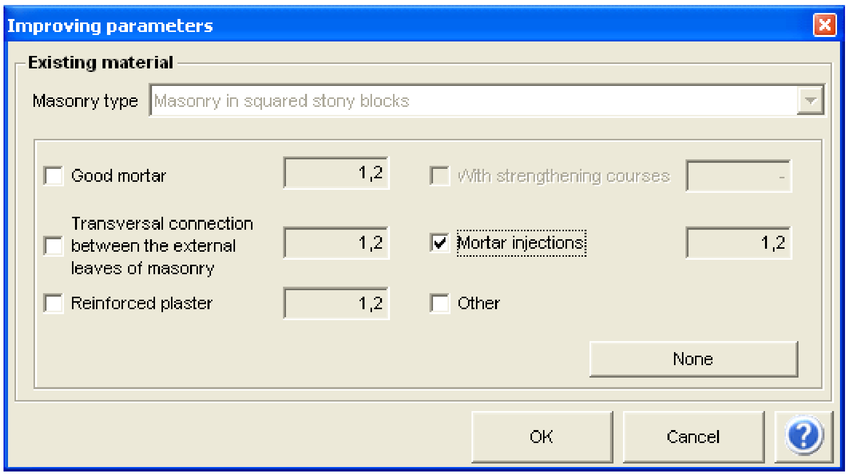
9.1.1 Existing Material



<table>
<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.


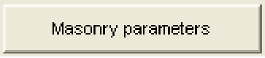
Clicking on "Confirm values" defines the computation values.</td></tr>
</table>

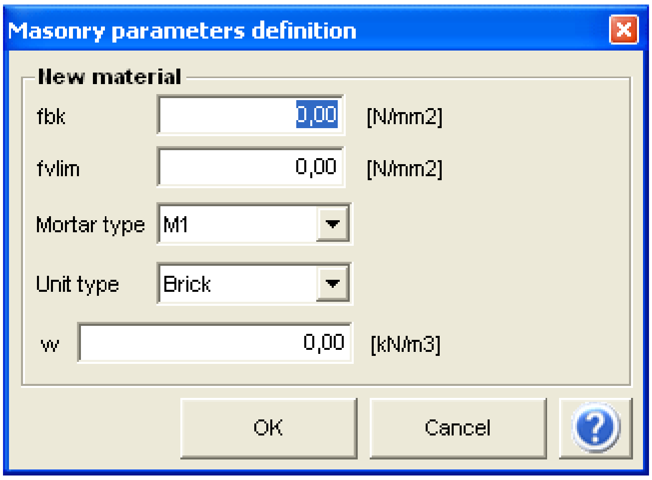
After having defined the material characteristics, it is possible to define improvement parameters, according to that indicated in the code.

<table>
<tr><td>
</td><td>
</td></tr>
</table>

In the update window for masonry materials properties, there is a link to the indications found in the code with regards to masonry.

9.1.2 New Material





fbk: characteristic compression resistance
fv,lim: Limit shear resistance
Tipo malta: mortar classification

9.1.3 Materials Library

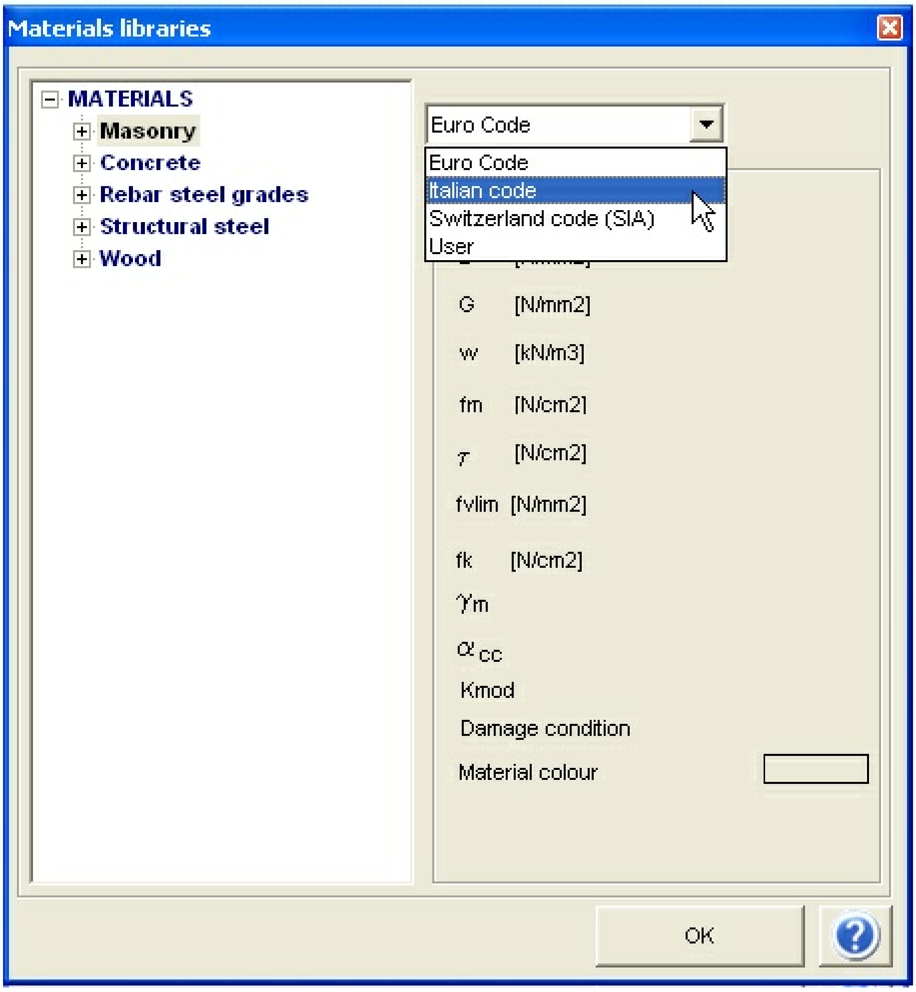
This function allows the designer to import on the project in exam the materials from different libraries (other Design Codes) or from the user library.
3Muri program has 3 main libraries types:

- 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.

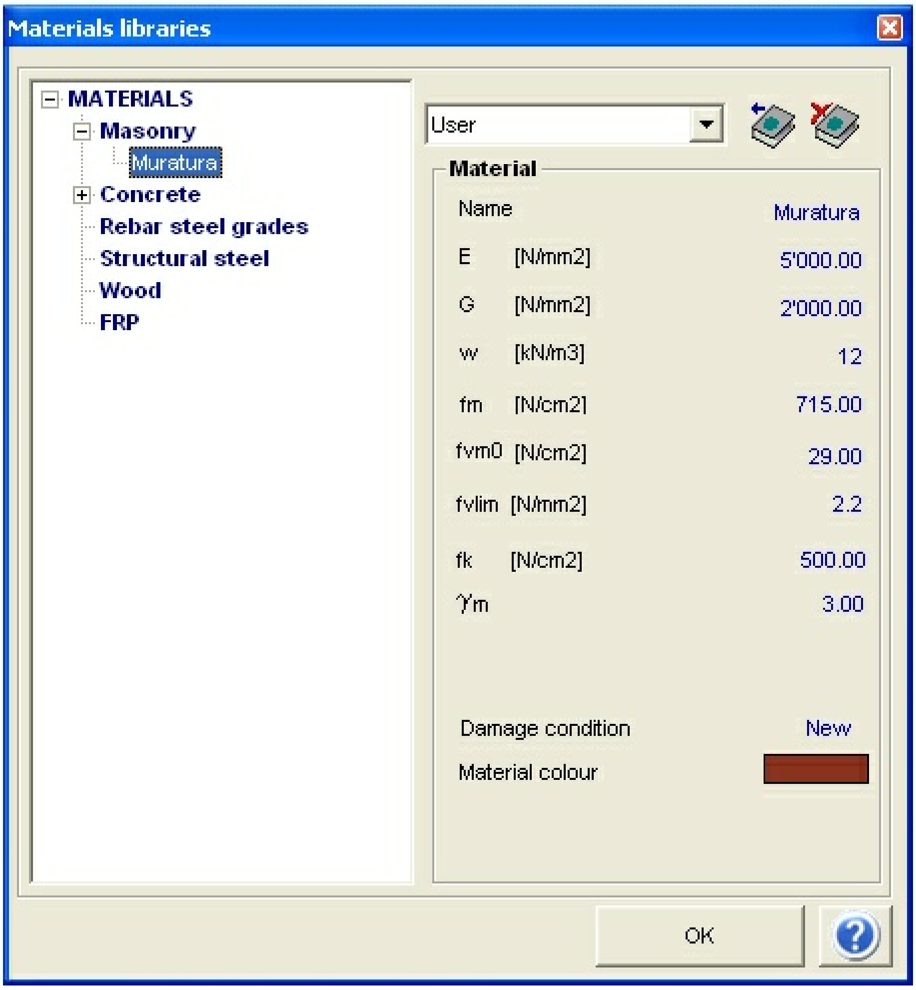
User Library
After defining a new or existing material, will be shown in the tree to the left of the window material.
The defined material is now available within the project, if this material is usually re-used for other projects different from the project on which you are working, you can save it on 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.

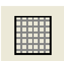


 **Copy in the project**
Allows to copy the selected material in the project library, making it available for the active project.

 **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)

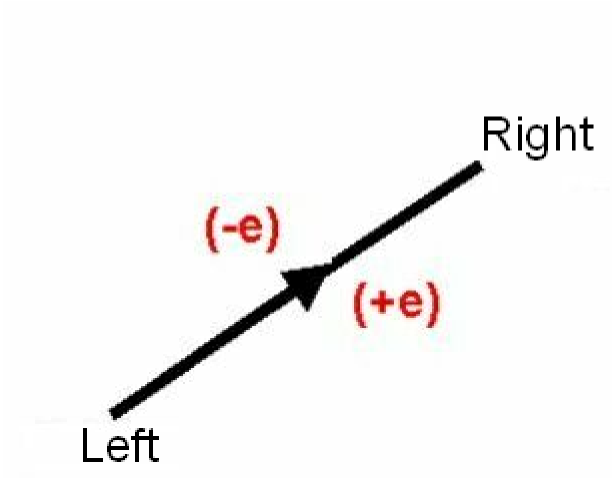
 Define characteristics: once the button is activated, the cursor changes shape and allows selection of one or more objects, whose structural characteristics can then be defined. Clicking the right mouse button, a window opens. In this the structural objects to be assigned to the selected walls can be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

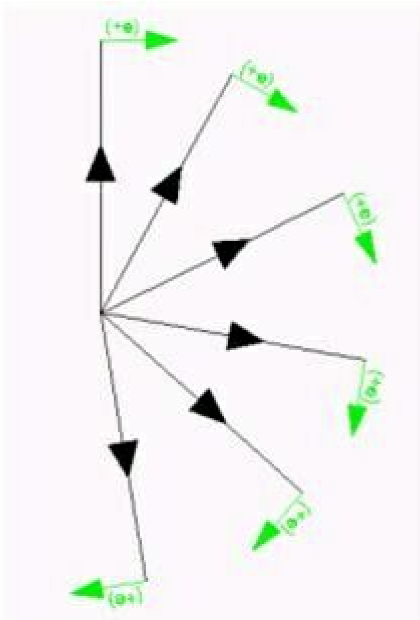
For all the elements that can be inserted there are two areas: one for insertion of geometry, and the other for insertion of material.
Insertion of material provides the possibility to choose the materials that will enter into play in the definition of the structural element. For example for an R.C. 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

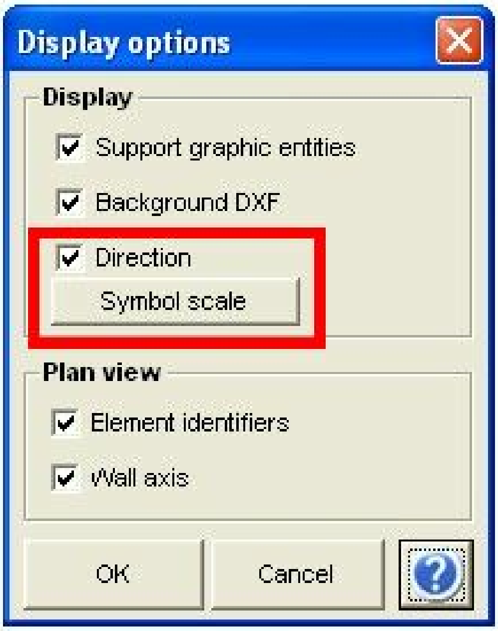
Elevation: The maximum elevation of the panel
Height: Height of the panel. Calculated from the point of maximum elevation to the ground.
Thickness: Thickness of the masonry.
Static checks: the corresponding box includes the figures for eccentricity and the wind exposure conditions. Eccentricity indicates the shift of the masonry panel with respect to the wall. (This is inserted in the walls area)

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.

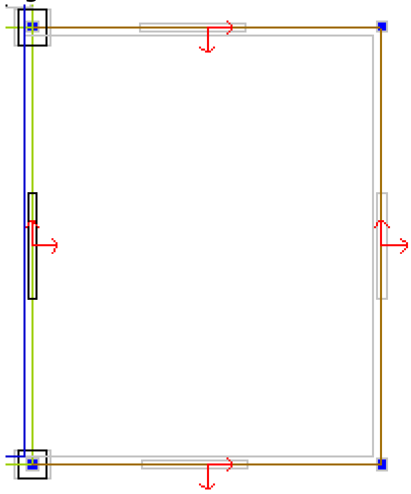




 With the display options button you can choose to show the local reference system directly on the model plan.



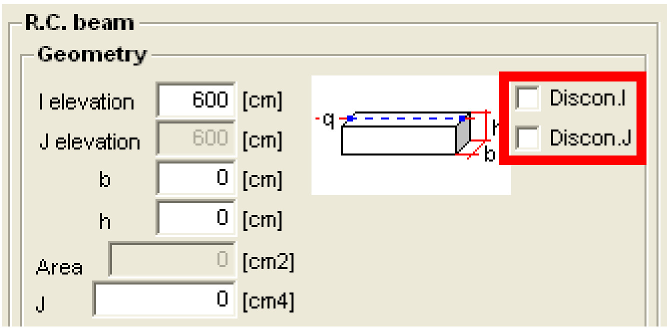
Using the options command, the local reference system is shown on each wall, allowing individuation of the eccentricity sign.



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.



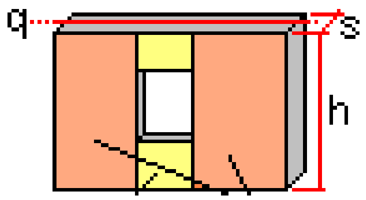
9.2.1.3 Steel/wooden beam

-I Elevation / J Elevation: Identifies the elevation of the two ends of the beam, to allow 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.

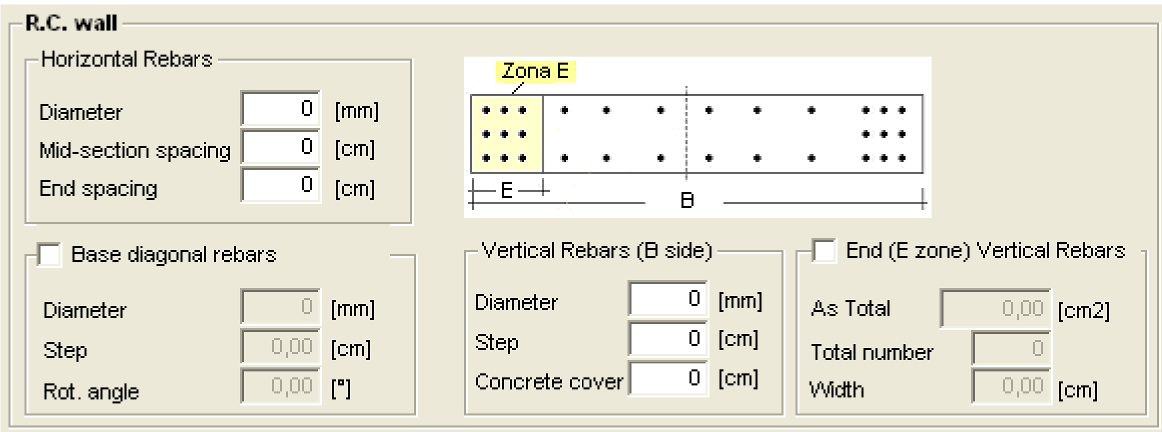
9.2.1.4 R.C. wall

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.
General considerations with regards to the reinforcement that help to identify if the toothing state is satisfactory or not. Checks with regards to the code requirements for anti-seismic details and on the typologies of rebars (plain, deformed).

R.C. walls are inserted using two different typologies of elements:

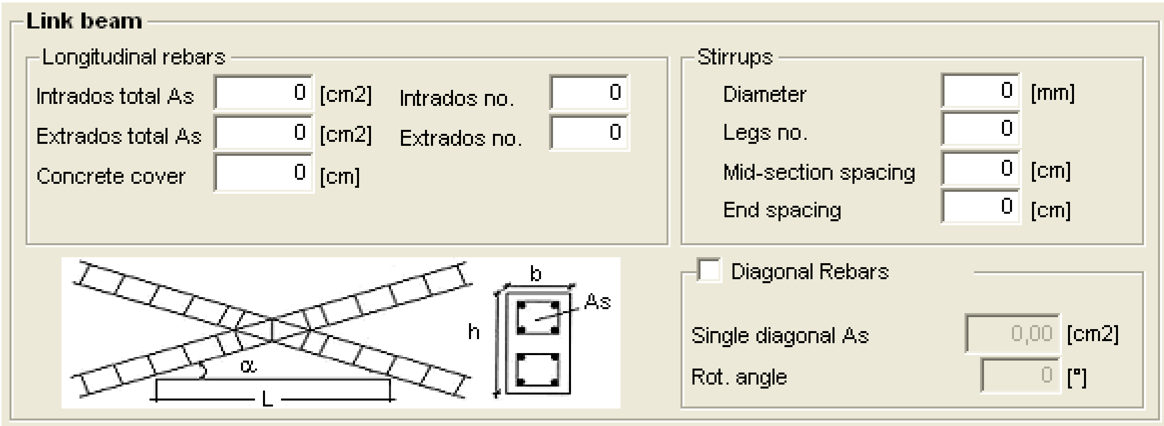


Wall:
-Diameters, steps from the rebars and concrete cover for the vertical and horizontal rebars.
-Possibility to define different vertical reinforcements in the end areas (zone E)
-Diameter and steps from the base diagonal rebars



Link beam:
-Diameters, steps from the rebars and concrete cover for the vertical and horizontal

rebars.
-Possibility to define different vertical reinforcements in the end areas (zone E)
-Diameter and steps from the base diagonal rebars
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. It is also possible to use diagonal rebars.



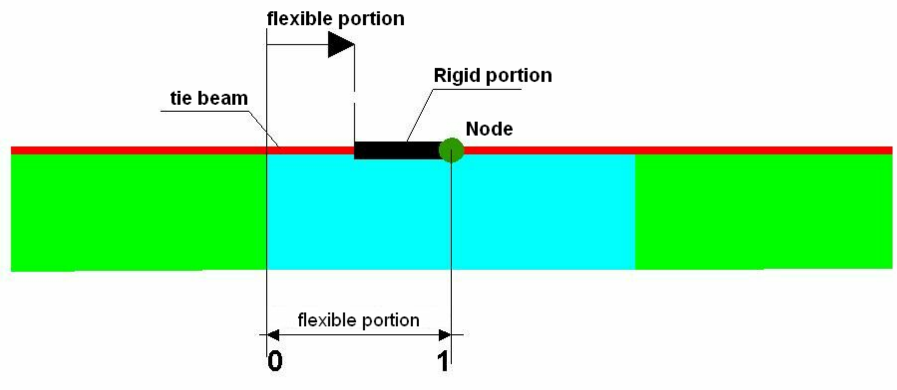
<u>Tie rod:</u>
Insertion of a tie rod not linked to masonry walls is functional only if inserted on part of a single wall (divided by insertion of segment points). In this way, other structural elements on the same alignment exist which can absorb the actions created by the tie rod.
Elevation: elevation in which the tie rod is placed
Diameter: diameter of the iron which constitutes the tie rod
Tension: stretching of the tie rod

9.2.1.5 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 Complex Elements

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.



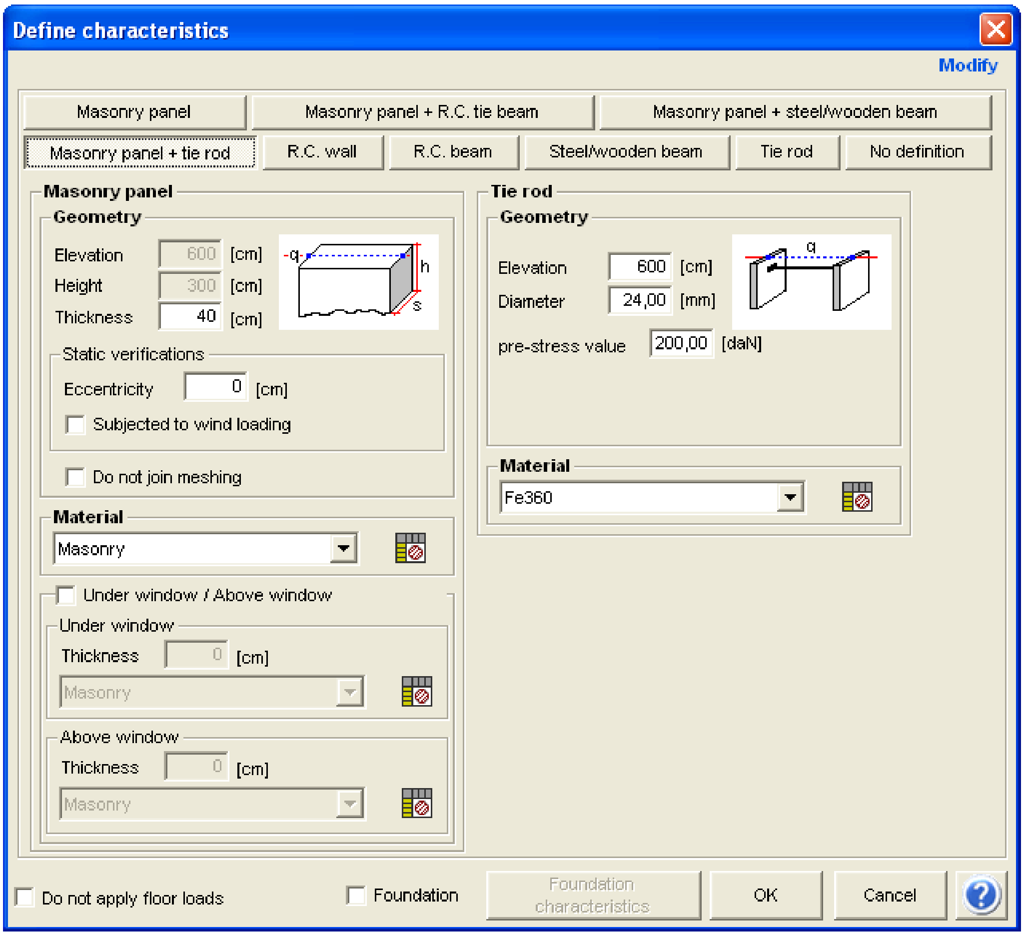
9.2.2.2 Masonry Panel + Beam

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.

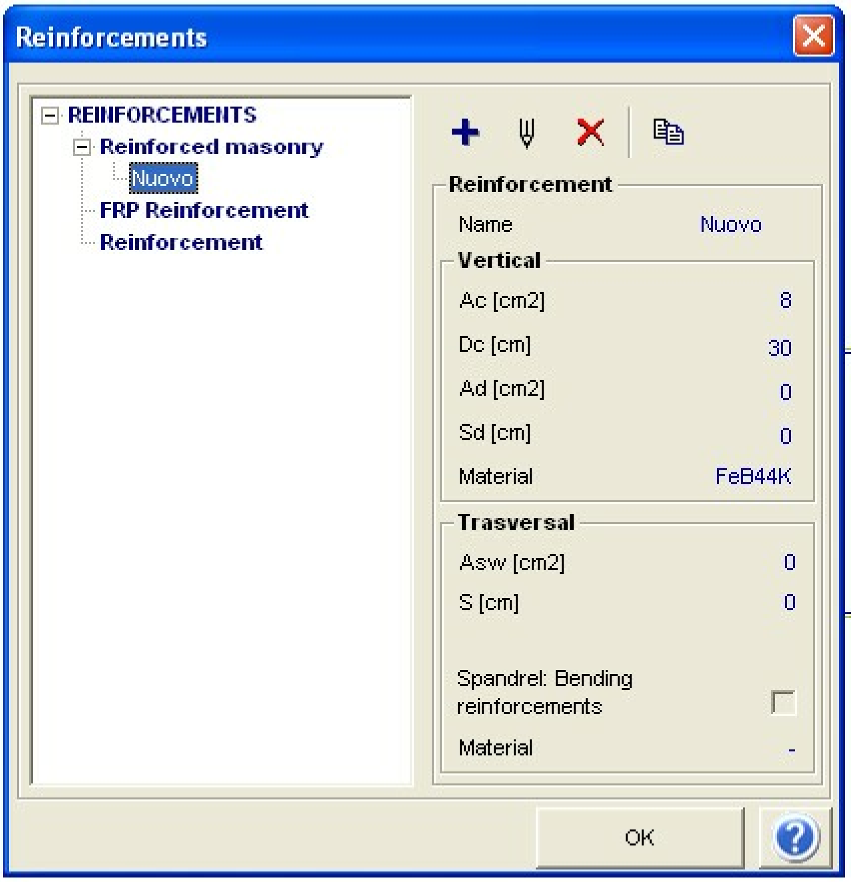
In the lower left corner of the window, it is possible to choose if the element will receive the load of the floor above it. (e.g. the floor does not rest directly upon the element.)

9.2.3 Reinforcements

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

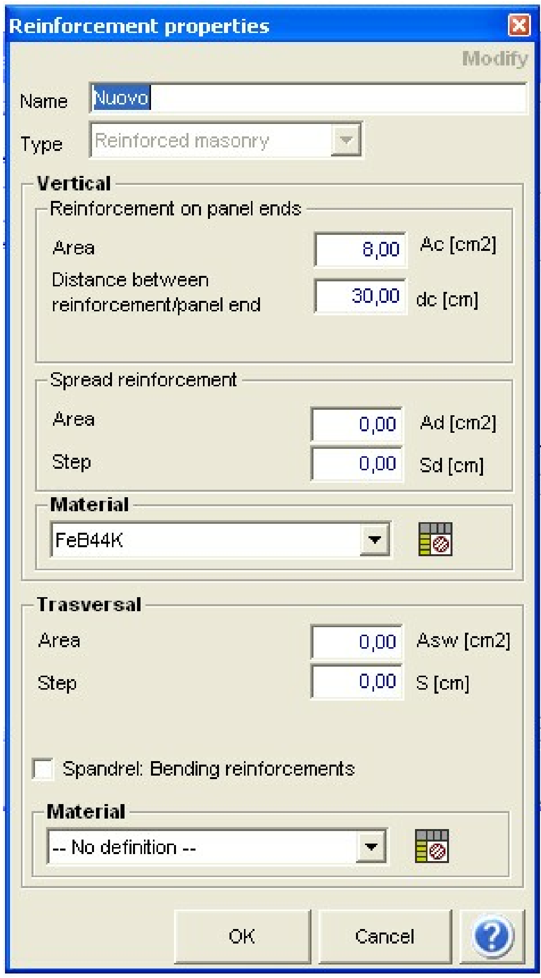
VChecking the properly box, the reinforcemtns library form is ready to use.

 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.

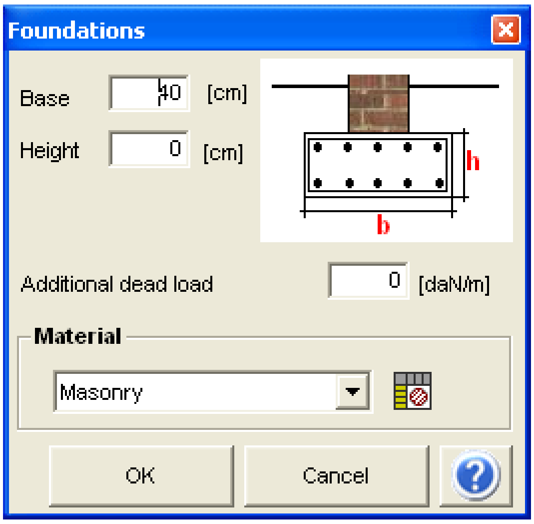
Other functions:

	Edit reinforcement
	Delete reinforcement
	Duplicate reinforcement

9.2.4 Foundation

It is possible to activate a text box called "Foundation". In this way, during the insertion phase, the user can decide if each wall goes directly into the foundation, so as to define the constraints. This option appears as active and non-editable when the first level is inserted. Here the base nodes are definitely in the foundation. At the higher levels, the option appears as nonactive, but editable. Selecting that a panel goes directly in the foundation means constraining all the degrees of liberty for the base nodes, both at transfer and at rotation. Different constraint conditions can be inserted only during the mesh editing phase during environment analysis while displaying the front view of a wall with the mesh.
After having decided that a given structural element is in the foundation, the button "Foundation characteristics" is activated. In this way, the characteristics of the foundation necessary for calculating the tension in contact with the ground can be defined.
The window that appears allow definition of the dimensions for the foundation beam, the

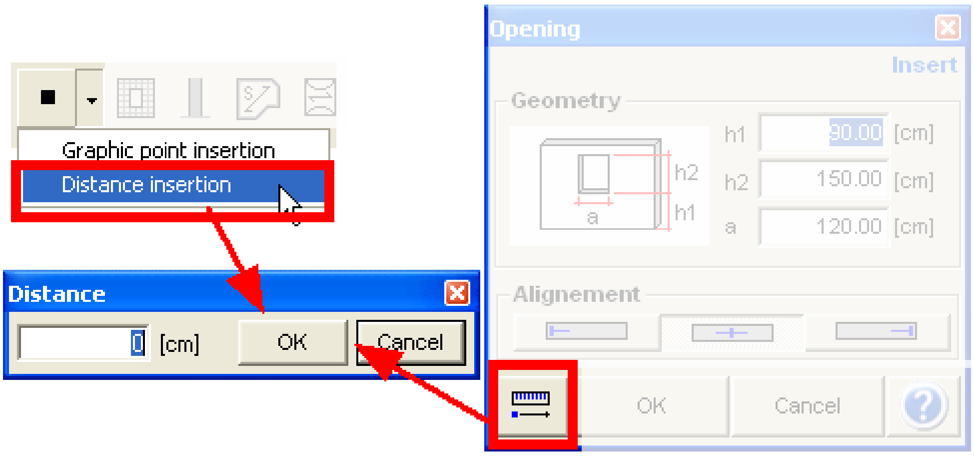
material, and, if necessary, a permanent load applied directly above the foundation.



9.2.5 Segment Points

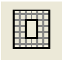
 Segment points can be inserted using the left mouse button.
This function can be used to assign various materials to a single wall, or to insert a segment point at the intersection of more than one wall. For example, if one wants to define a single wall with different masonry typologies, or with masonry 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 3nodes 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 description of environments)

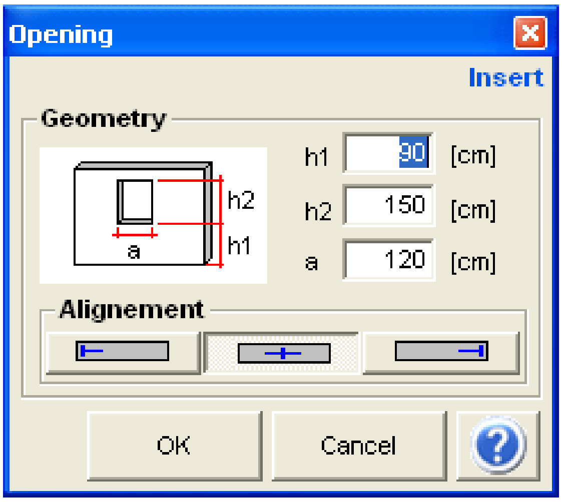
Insertion of a node/window can occur either through insertion of node/windows distances for the edge nodes.



To insert "distance" segment nodes, it is necessary to position the mouse on a wall (highlighting it in red) and decide from which node to calculate the distance. The distance is then inserted, positioning the mouse closer to the node in question.

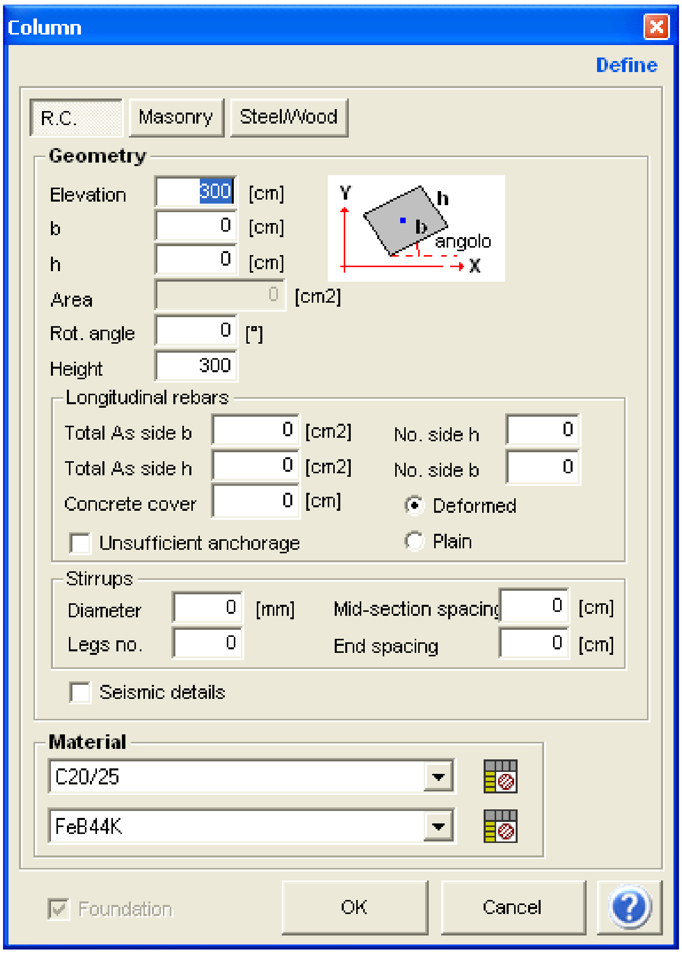
9.2.6 Openings


Allows the insertion of an opening in a wall.
A window will appear in which it is possible to modify the geometric characteristics of the opening, once given the OK, it proceeds to the insertion of the openings in the desired positions. To exit, click 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 through the use of snaps or with the insertion of the distance using the same method used for segment points.



9.2.7 Columns

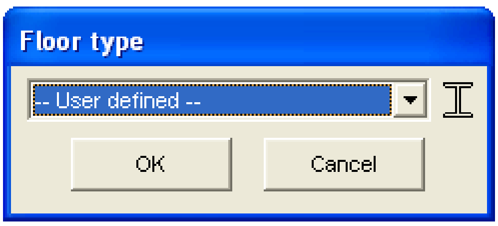

Insert a column in correspondence with one or more nodes.
First the node or nodes where the columns will be placed are selected. Then, using the right mouse button, access the window in which the geometric characteristics and the materials of the element are defined.
There are three different types of 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.

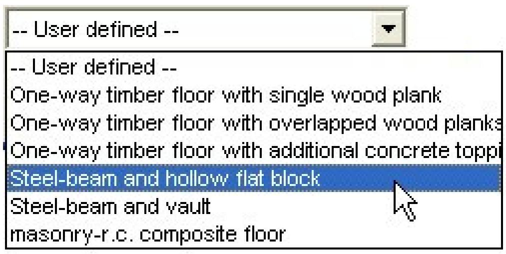


At the lower left, it is possible to activate a box that imposes foundation constraints at the base of the column.

9.2.8 Floor

 Inserts a floor.
A window opens in which the user can select the desired floor type.



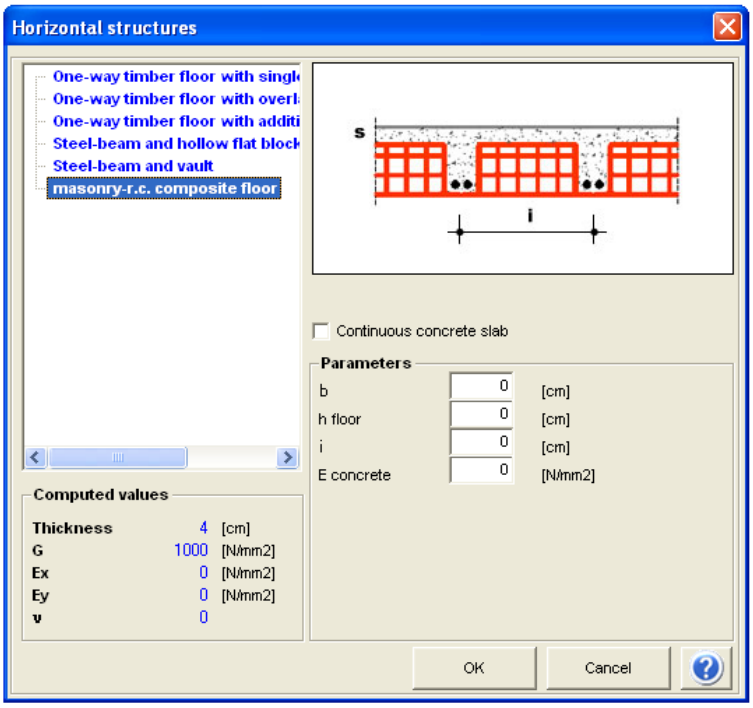


The horizontal structures window allows definition of the mechanical characteristics for the most common floor typologies. The program examines the following:

- One way timber floor, with single wood plank
- One way timber floor, with overlapped wood planks
- One way timber floor with additional concrete topping

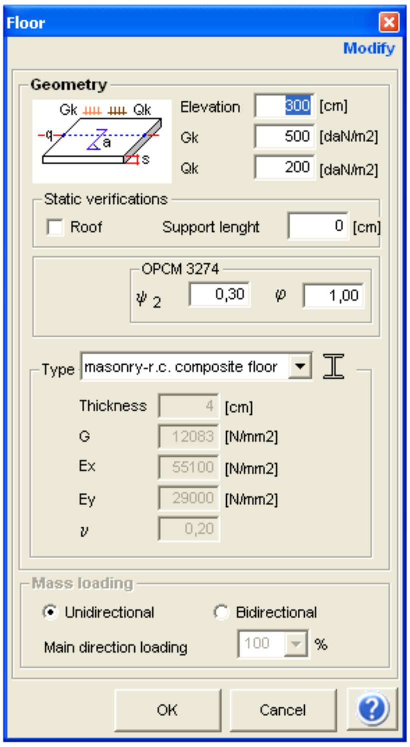
- Steel beam and hollow flat block
- Steel beam and vault
- Masonry R.C. composite floor

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

When selection is finished, the following window will appear.



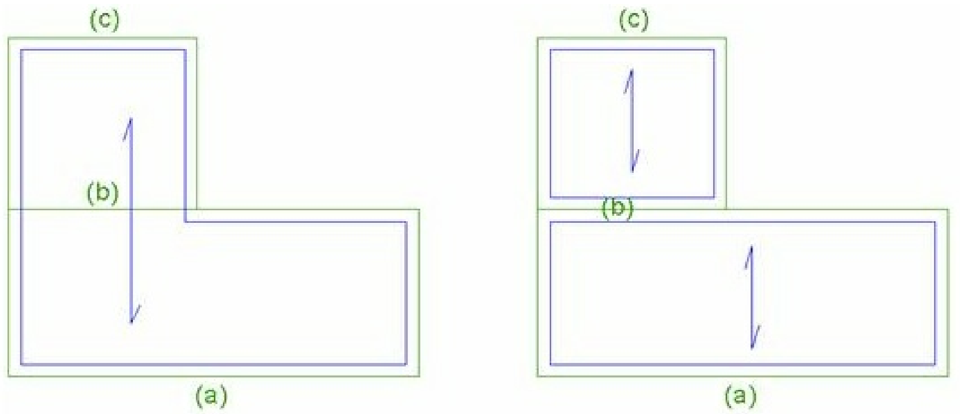
In the upper part, insert the load actions on the floor as either permanent (Gk) or variable (Qk). These can be combined according to the coefficients indicated in the code. If the user desires, it is possible to use the "Code" button to get additional information about choosing the combination coefficients.

“Static checks contains the parameters necessary to perform the static checks. It is necessary to check that the floor being examined is covered and indicate the support lengthof the floor on the masonry. 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.

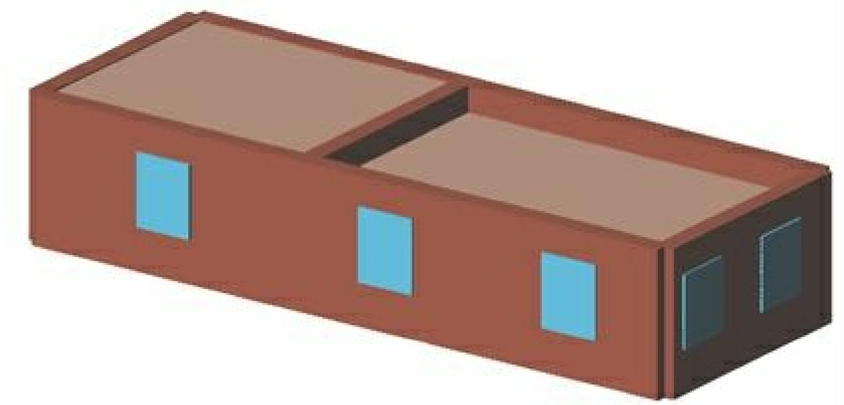
Bending modules Ex and Ey refer to the local axes system (x, y) in which "x" is identified based on the warping direction and "y" is perpendicular to the warping direction.

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 of the insertion mode chosen]

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.

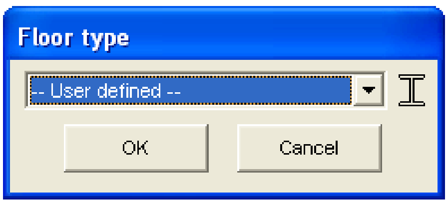


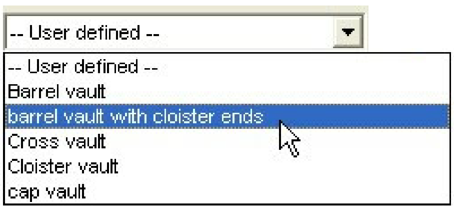
The program does not create additional computation nodes in correspondence with the position of the floors. It continues to use those already defined, taking into account the contribution due to the transfer of the floor with respect to these limit nodes between one level and another.
It is not possible 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.

9.2.9 Vaults

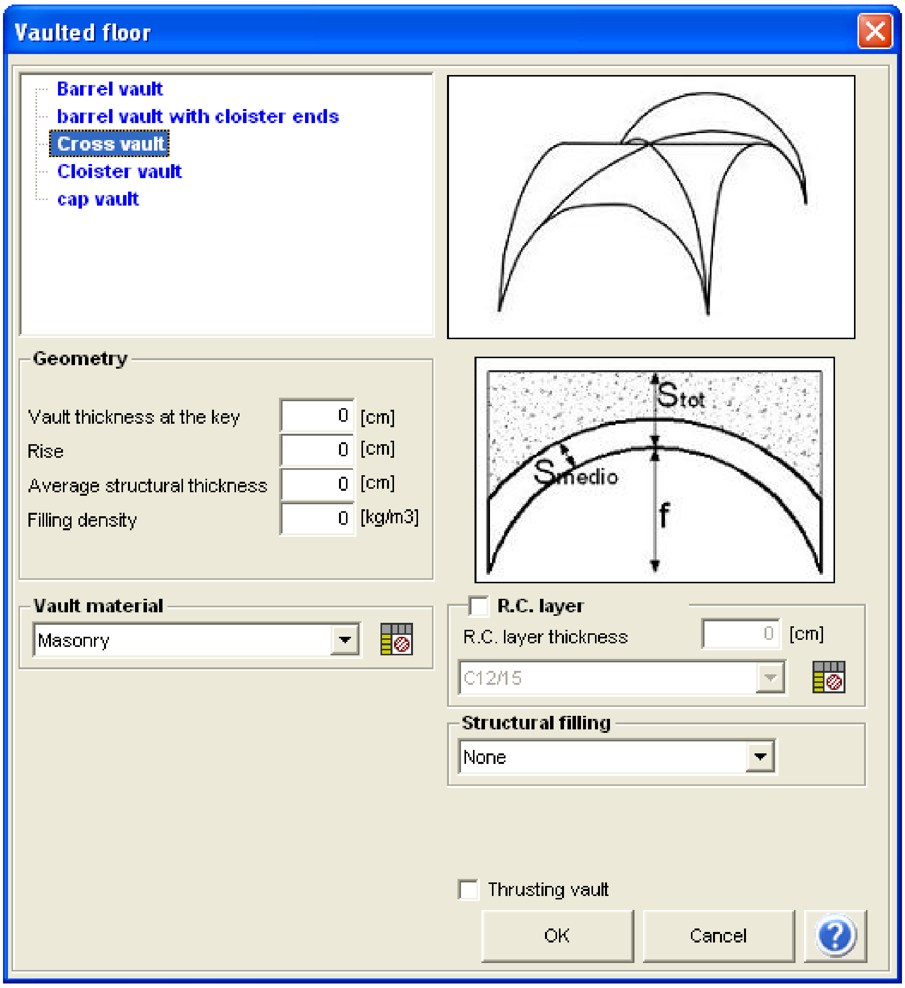
 allows insertion of vaults.

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



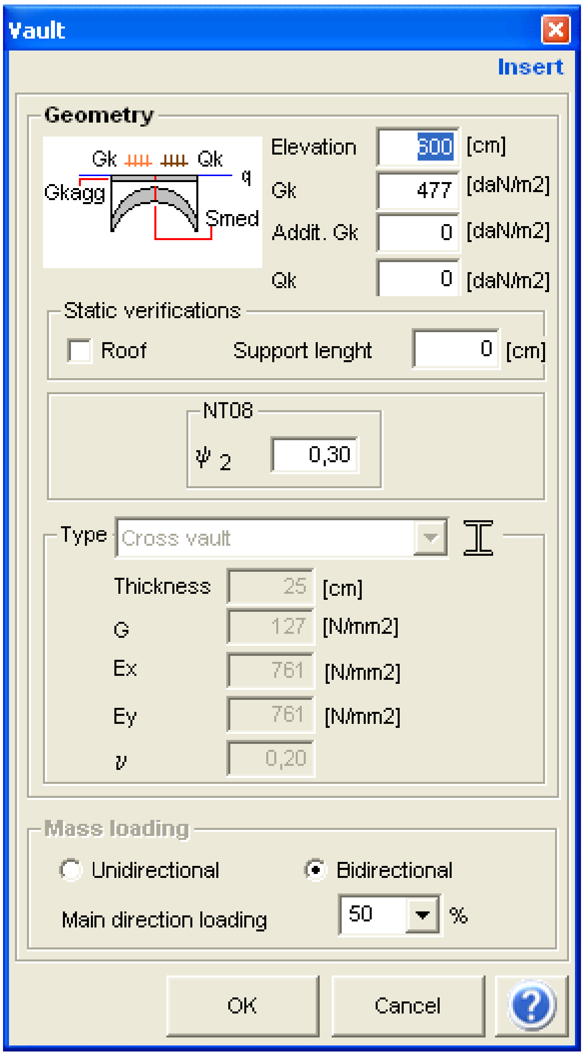


For each vault typology (listed above), the user must define the main parameters.



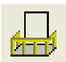
After having inserted the geometric mechanical parameters, click the OK button. Then carefully select the nodes on which the vault will rest. After, select a reference structural element to define the direction for the vault's discharge (parallel, perpendicular, or user defined).

When selection is finished, the following window will appear.

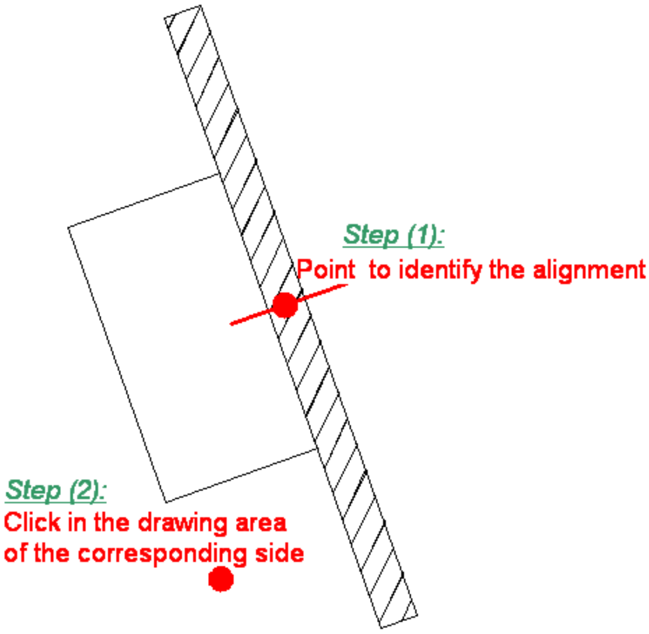


This window is very similar to the floor window.
In this case, the vault's permanent structural load (Gk) is automatically calculated. The user must insert the additional permanent Gk (e.g.: weight of the trimming work) and the accidental loads (Qk).

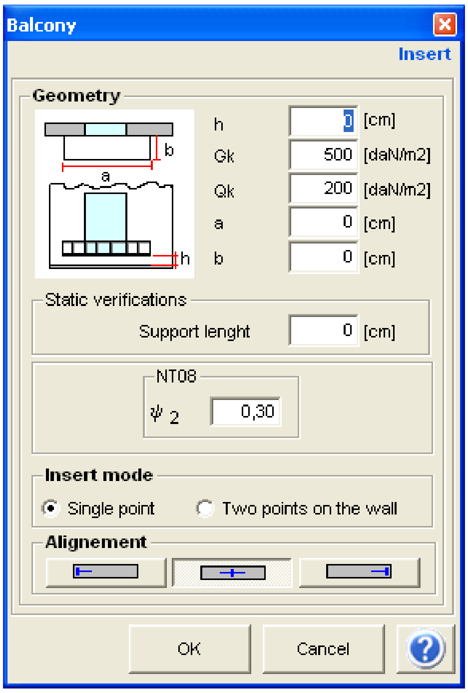
9.2.10 Balconies

 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.

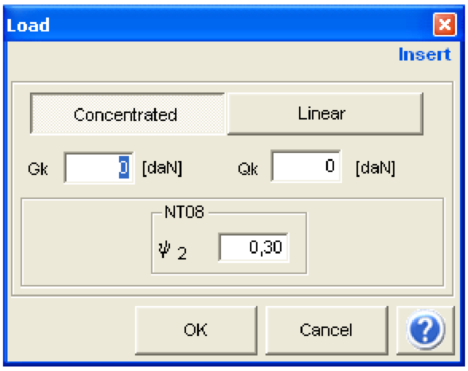


Two points on the wall: The length of the balcony is inserted graphically through the insertion of the starting and ending points without the use of fixed alignments for the insertion.



9.2.11 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.



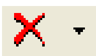
9.2.12 Structure Editing

 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.

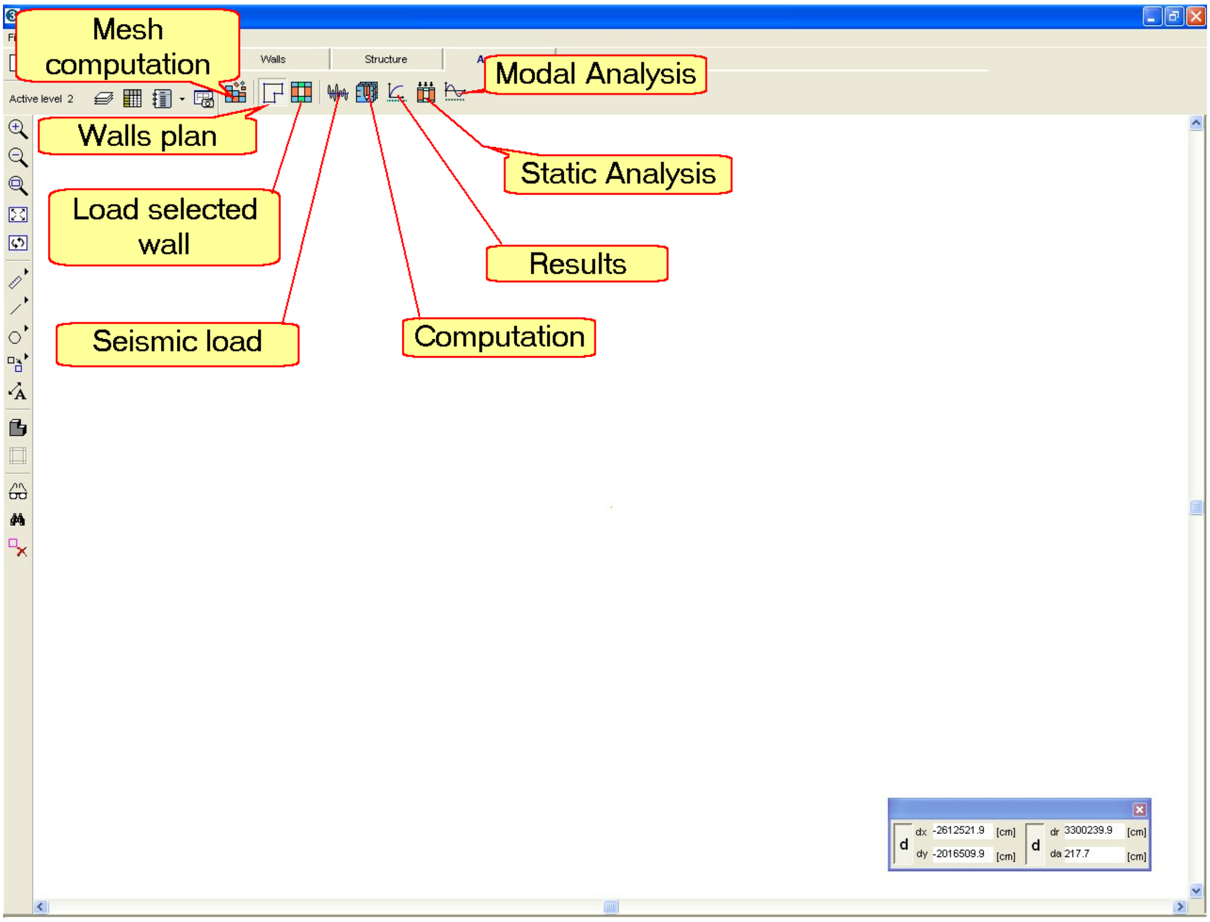
 Paste attributes

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.

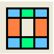
 Multiple Deletion:

	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.

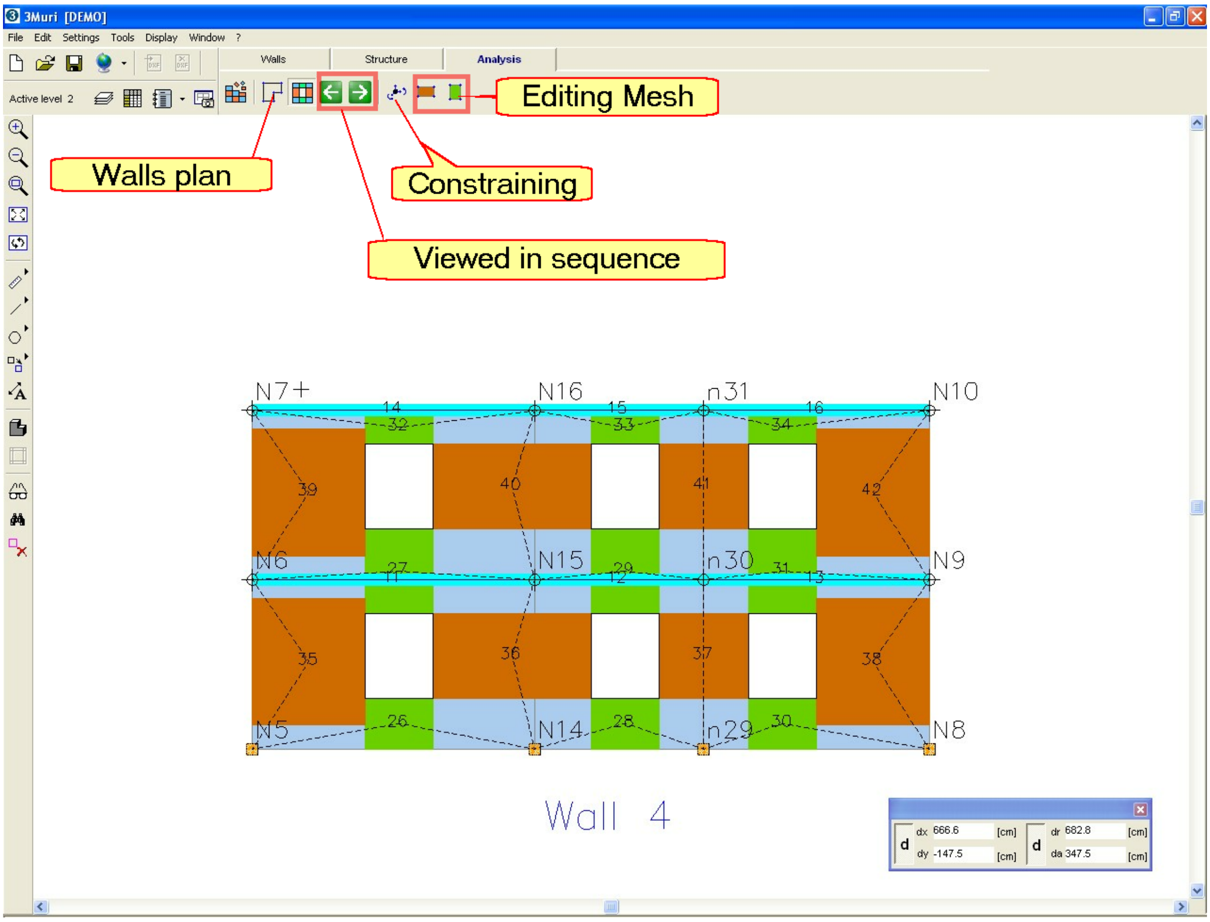
10 Analysis



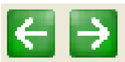
10.1 Mesh definition

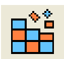
Load selected wall: This command display a mesh front view of a selected wall.

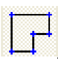
After proceeding to loading of the wall mesh in the drawing area, the selected wall mesh is shown. The toolbar changes as seen in the figure below:

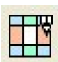


The display shows the nodes in 2D (the letter "n" followed by the node's identifying number) and 3D (the letter "N" followed by the node's identifying number) and the macro-elements delimited by the openings.

Two buttons will appear on the toolbar. Using these, the walls can be viewed in sequence.

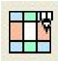
 Mesh computation: computes the model mesh. It necessary to recalculate it, if modifications are made to the structural model.

Walls plan: shows the disposition of the walls in the plan, with numbers identifying the walls and nodes. This is used to return to plan view after having seen the wall mesh (Load selected wall).

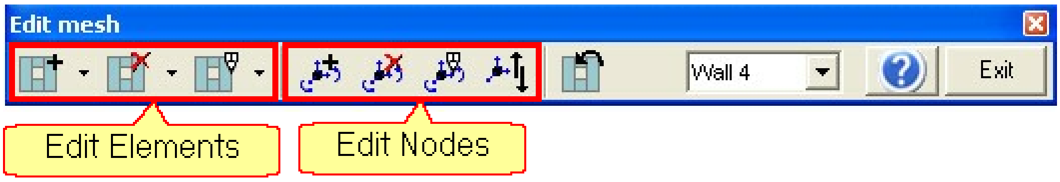
 Mesh editing: The procedure of automatic mesh generation (calculate mesh) can seize almost all of the more usual case in practice project.For the limited cases where this is not possible,the user can enter into Edit mesh environment.

 Modifica materiale: Qualora in seguito ad un calcolo, il progettista si trovi a dover prevedere degli interventi di adeguamento sulla struttura, può agire mediante interventi localizzati migliorando le proprietà meccaniche dei singoli elementi murari. (Details...)

10.2 Mesh editing

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

Edit funcions are divided in two groups: Edit Elements, Edit Nodes.

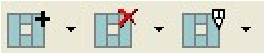


All edit functions, except "Change node type", have "Cancel" command.
When the user change working wall or close edit mesh form, the program asks if he confirms changes made. Answering yes, wall edit mesh save in temporary working session and automatic calculation of rigid nodes is applied. In case of negative answer, all editing actions are cancelled.

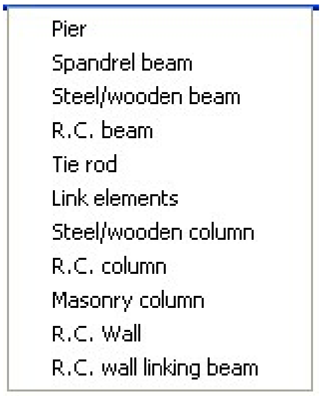
Please note that the program 3Muri operates in a temporarily working session , so all the changes are not finalized until you save the template. Salvaging takes place or by a user request or automatically before a calculation.

 Undo
Allow to cancel tha last editing operation. If the same editing operation. If the same change has been applied to multiple items or multiple nodes simultaneously, the undo is applied to an item or node at a time.

10.2.1 Editing Elements

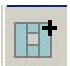


The editing functions acting on the entities in the menu associated with each button on the command bar.

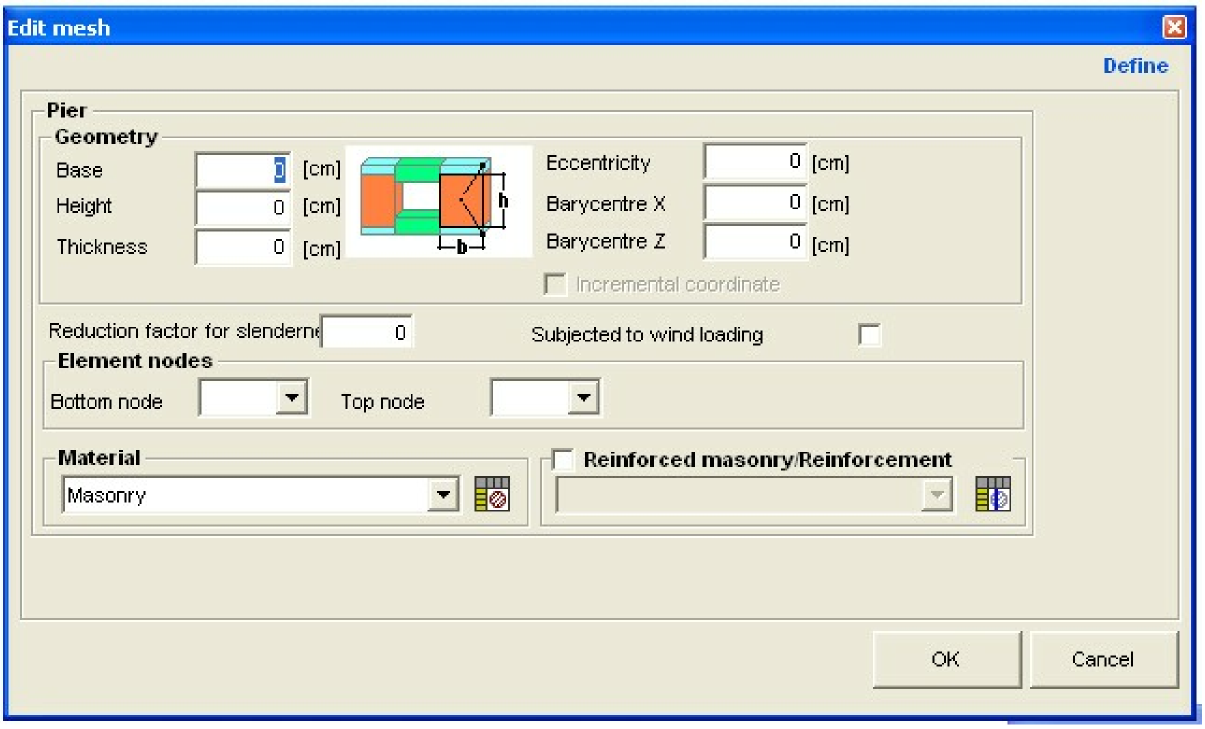


Link elements: Rigid link or truss. These items are included in automatic mesh algorithm to create a equivalent frame according to solver's rules.

Rigid link: Are included in "blind" pier definition (a "blind" pier is due to a pannel 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.

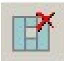


Add item:
After choosing the item to be add, a form with all item data is loaded.
Suppose for example the case of pier:



Element nodes: Allow user to modify element nodes of selected item; in this way the user can modify equivalent frame geometry.
Move barycentre: allow the use to move selected macroelement by insert component fo 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

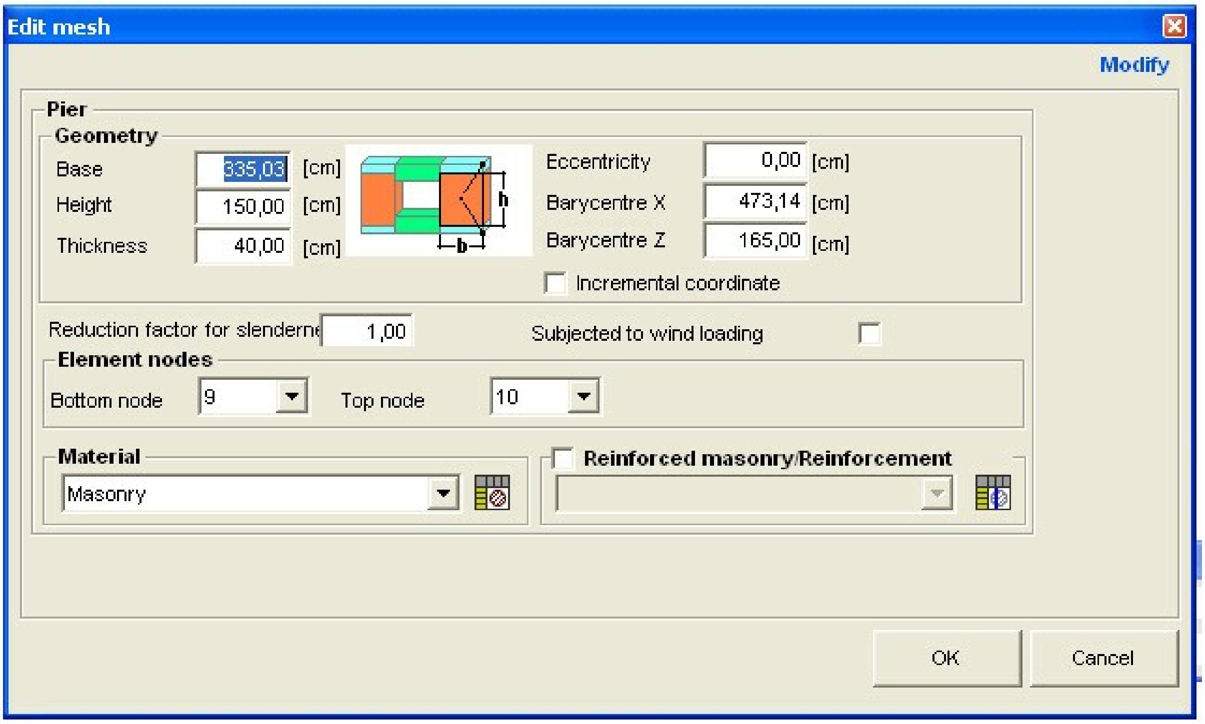
Input boxes of the form must be completed in all parts.



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 pier:



Multiple selection:
✓ Only new data inserted in input boxes are modified. To keep the original data of selected items, leave the input boxes blank. Fro instance, to move up of 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.

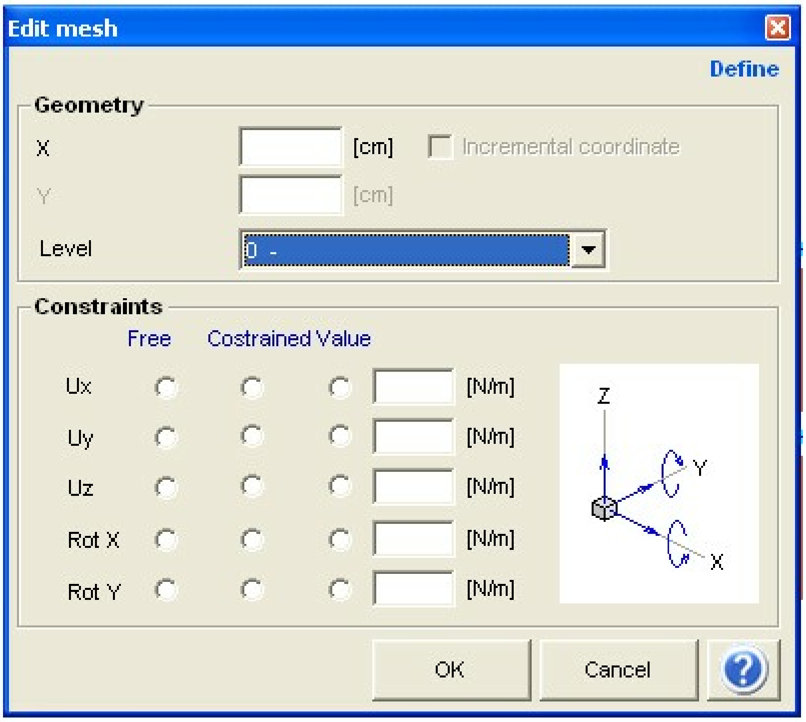
All the characteristics of the selected piers remain unchanged, except barycentre coordinated and therefore piers are shifted by 10 cm upwards.

✓ It is impossible to modify elements nodes.

10.2.2 Editing Nodes

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

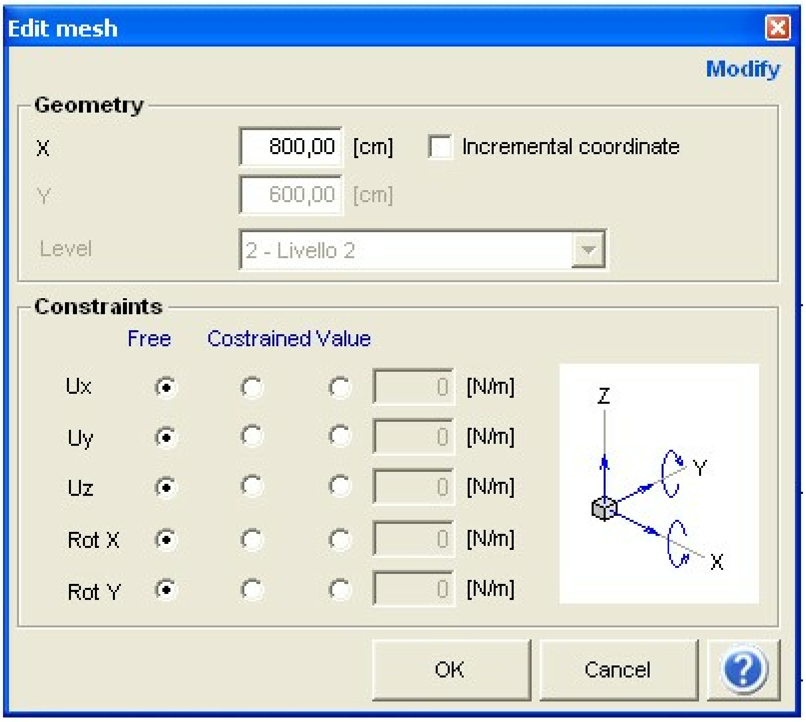
Add node: a form with all node data is loaded.



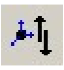
Input boxes of the form must be completed in all parts.

Delete node:
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.



I this form is possible to define Constraining conditions.

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

Allow to modify node type. This function is available one node a time.

3D -> 2D

To apply this function it is necessary to verify if the selected node belongs to several walls.

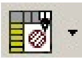
- ✓ If the node belong to a *single wall* the type is changed (only if the node is connected to a column or a R.C. wall this functions is prevented).
- ✓ If it belongs to *two walls*, including current one, the node type is changed both, this is useful when you want to delete a 3D node on a wall as it no longer makes sense to exist as a result of 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 and that allows the user to decide what to do of each node, move 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.

2D -> 3D

In this phase the program prompt user if exist any incidental wall. The program checks if the input wall intersect current wall and it looking for in the incidental wall a node that geometrically coincides with the input node; the node in curente wall and the node in incidental wall area trasformed in 3D node typ.

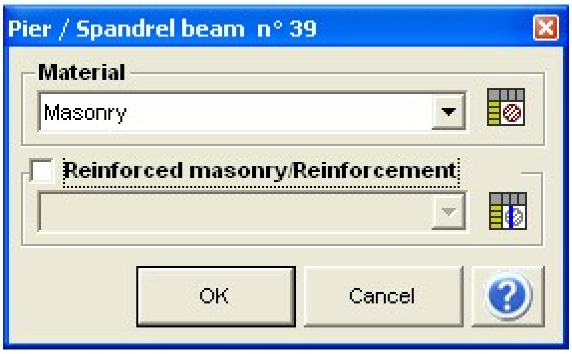
***N.B.:** It is impossible to apply undo on these functions because of UNDO is valid onlyfor current wall. These functions involved mome tahn current wall son they are not* *cancelled in "edit mesh".*

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:

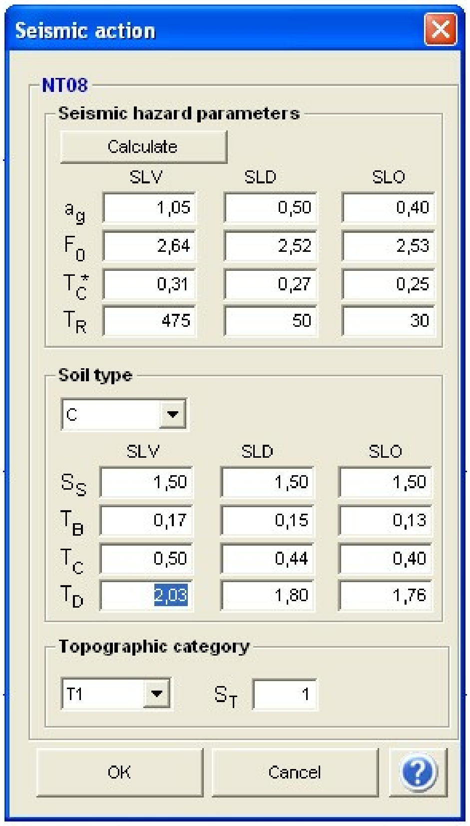


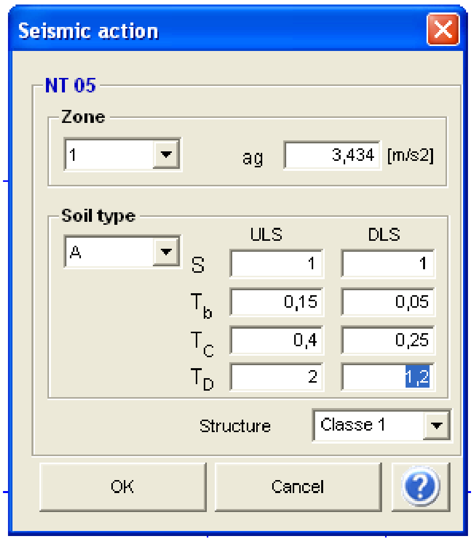
In case of single selection, the form contains all data item; in case of multiple selection, the form does not report any data of selected items. If you want to modify only one of two items of the form, simply leave unchanged the other one.

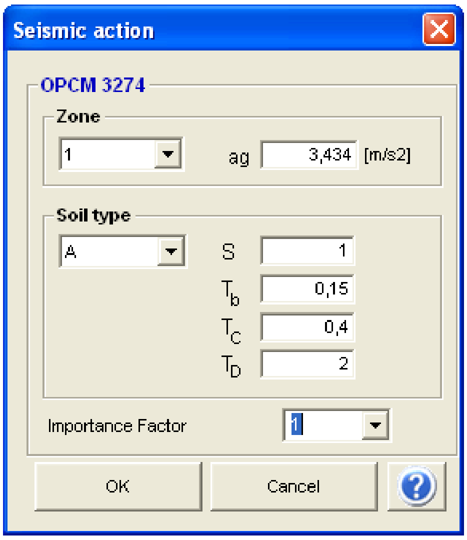
10.4 Pushover Seismic Analysis

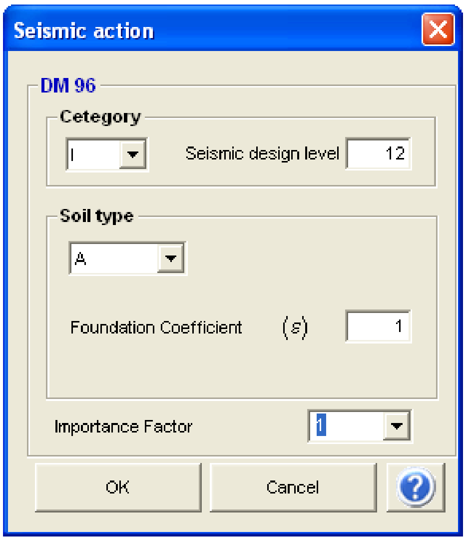
10.4.1 Selection of the seismic conditions

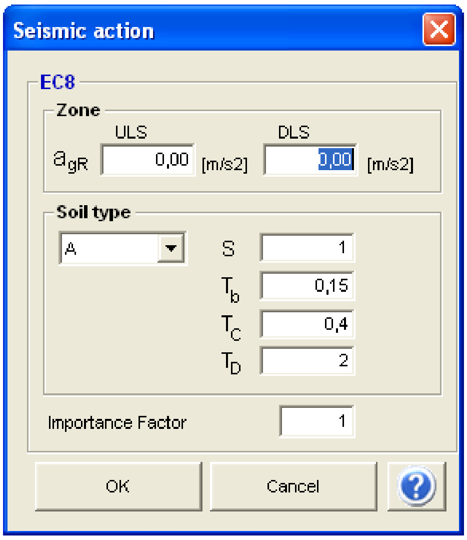
Seismic load: allows to set the earthquake zone and the class of the soil according to the indications of the code. For more details that indicated in the following windows, it refers to as described in the corresponding code.

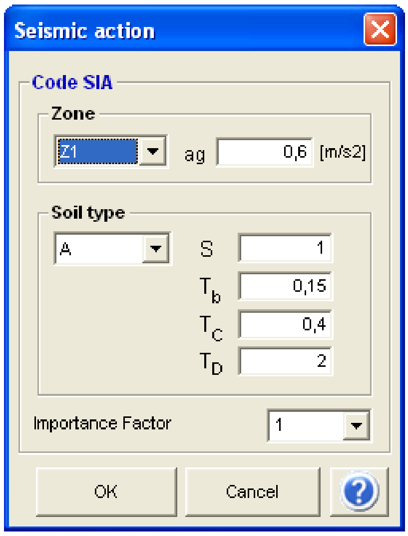

Seismic action
NT08
Seismic hazard parameters
Calculate
SLV SLD SLO
ag 1,05 0,50 0,40
F0 2,64 2,52 2,53
TC* 0,31 0,27 0,25
TR 475 50 30
Soil type
C
SLV SLD SLO
SS 1,50 1,50 1,50
TB 0,17 0,15 0,13
TC 0,50 0,44 0,40
TD 2,03 1,80 1,76
Topographic category
T1
ST 1
OK
Cancel


Seismic action
NT 05
Zone
1
ag 3,434 [m/s2]
Soil type
A
ULS DLS
S 1 1
Tb 0,15 0,05
TC 0,4 0,25
TD 2 1,2
Structure Classe 1
OK
Cancel

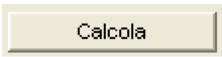

Seismic action
OPCM 3274
Zone
1
ag 3,434 [m/s2]
Soil type
A
S 1
Tb 0,15
TC 0,4
TD 2
Importance Factor 1
OK
Cancel


Seismic action
DM 96
Cetegory
I
Seismic design level 12
Soil type
A
Foundation Coefficient (ε) 1
Importance Factor 1
OK
Cancel

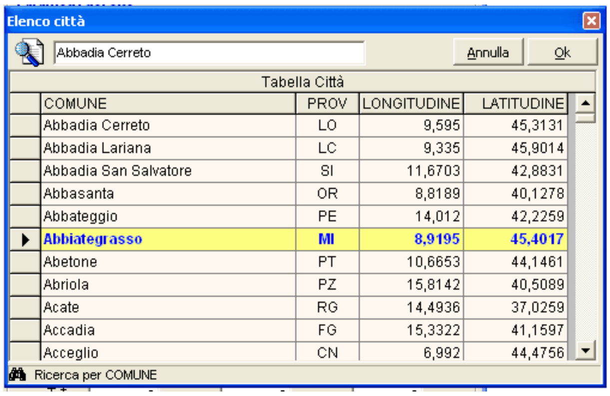




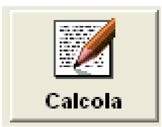
In "Norme Tecniche del Gennaio 2008", the seismic spectrums depend on the geographical coordinates of the site, instead of the earthquake zone (as in previous rules). In the window "seismic action" the "parameters of seismic hazard" are defined by the button "Calculate".



Choosing this button, the following window is shown:



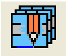
You can select the municipality using the internal database or insert the latitude and longitude of the site.

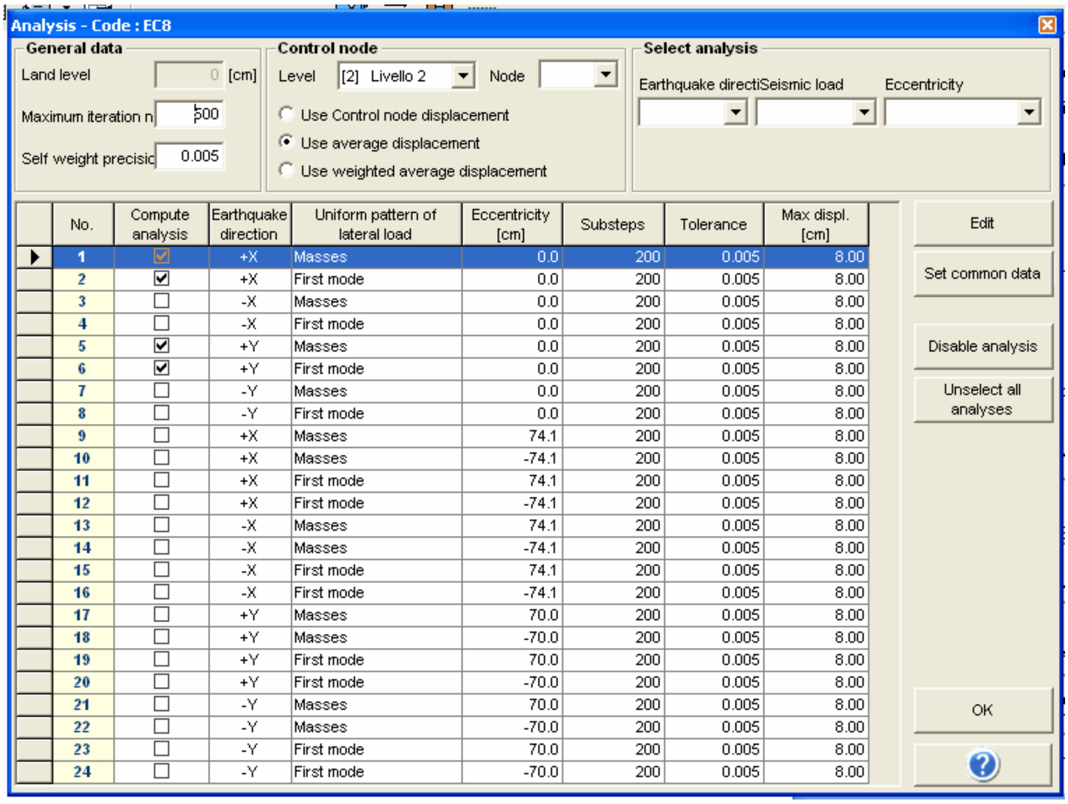


Calculate the necessary values to define the shape of the spectrum for each limit state resuming them in the lower part of the window.

Export results to Seismic action window.

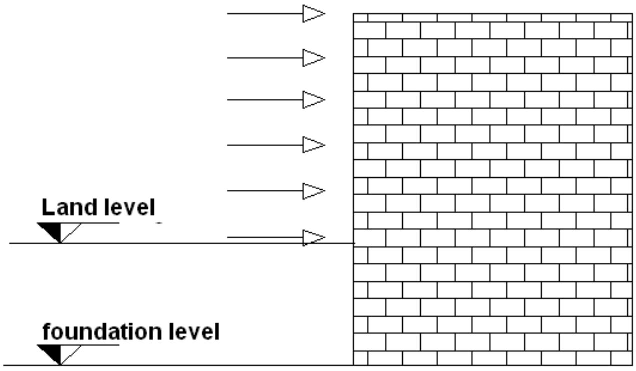
10.4.2 Computation Settings

Performs computation of the structure.
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 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]	Substeps	Tolerance	Max displ. [cm]
1	☑	+X	Masses	0.0	200	0.005	8.00
2	☑	+X	First mode	0.0	200	0.005	8.00
3	☐	-X	Masses	0.0	200	0.005	8.00
4	☐	-X	First mode	0.0	200	0.005	8.00
5	☑	+Y	Masses	0.0	200	0.005	8.00
6	☑	+Y	First mode	0.0	200	0.005	8.00
7	☐	-Y	Masses	0.0	200	0.005	8.00
8	☐	-Y	First mode	0.0	200	0.005	8.00
9	☐	+X	Masses	74.1	200	0.005	8.00
10	☐	+X	Masses	-74.1	200	0.005	8.00
11	☐	+X	First mode	74.1	200	0.005	8.00
12	☐	+X	First mode	-74.1	200	0.005	8.00
13	☐	-X	Masses	74.1	200	0.005	8.00
14	☐	-X	Masses	-74.1	200	0.005	8.00
15	☐	-X	First mode	74.1	200	0.005	8.00
16	☐	-X	First mode	-74.1	200	0.005	8.00
17	☐	+Y	Masses	70.0	200	0.005	8.00
18	☐	+Y	Masses	-70.0	200	0.005	8.00
19	☐	+Y	First mode	70.0	200	0.005	8.00
20	☐	+Y	First mode	-70.0	200	0.005	8.00
21	☐	-Y	Masses	70.0	200	0.005	8.00
22	☐	-Y	Masses	-70.0	200	0.005	8.00
23	☐	-Y	First mode	70.0	200	0.005	8.00
24	☐	-Y	First mode	-70.0	200	0.005	8.00

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.

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 are drawn only with the control node displacement.
- *Average displacement*: the capacity curve are drawn with the average displacement of all level's nodes.
- *Weighted average displacement*: the capacity curve are drawn with the weighted average displacement (mass weighted).

If the floors are considerate rigid, this value is the same with the barycentre displacement.

Level average reference node: Identifies the possibility to perform predictive calculations to draw the bearing capacity diagram with reference to the value of the average displacement of the level.

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

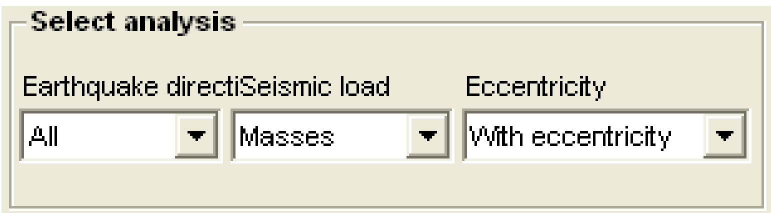
Direction: indicates the earthquake direction.

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 analyses can be performing by activating the selection filters.

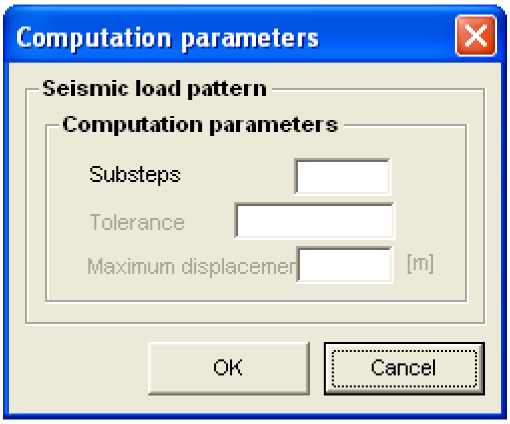


In the left part of the window, the code to be used can be selected.

On the right hand side of the same window there are buttons that can be used to regulate the parameters for the analyses.

Edit	Select the computation parameters for every individual analysis
Set common data	Select the computation parameters used for all analyses
Select analysis	Enables computation of a type of analysis currently deactivated
Unselect all analyses	Disables computation of all analyses

The parameters for each analysis can be selected in the following window.

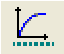


Substeps: represents the number of displacement steps computed by the solver for the seismic load pattern.

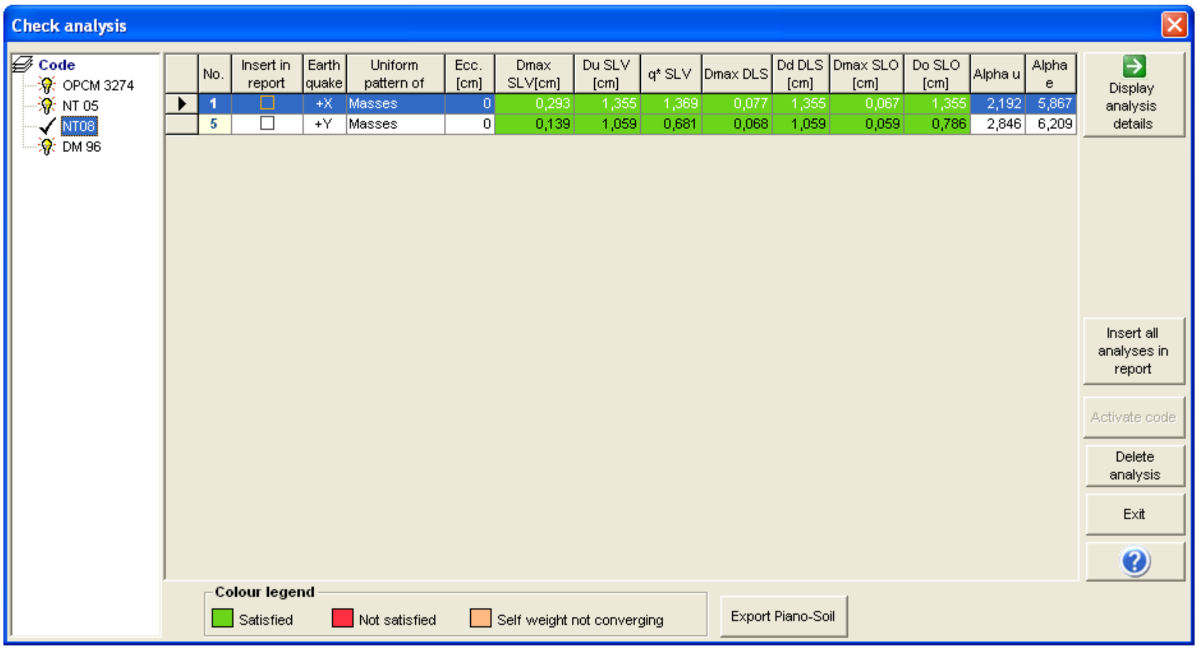
Tolerance: represents the degree of tolerance reached by the non-linear computation.

Maximum displacement: represents the maximum displacement that the structure's control node can withstand.

10.4.3 Display results



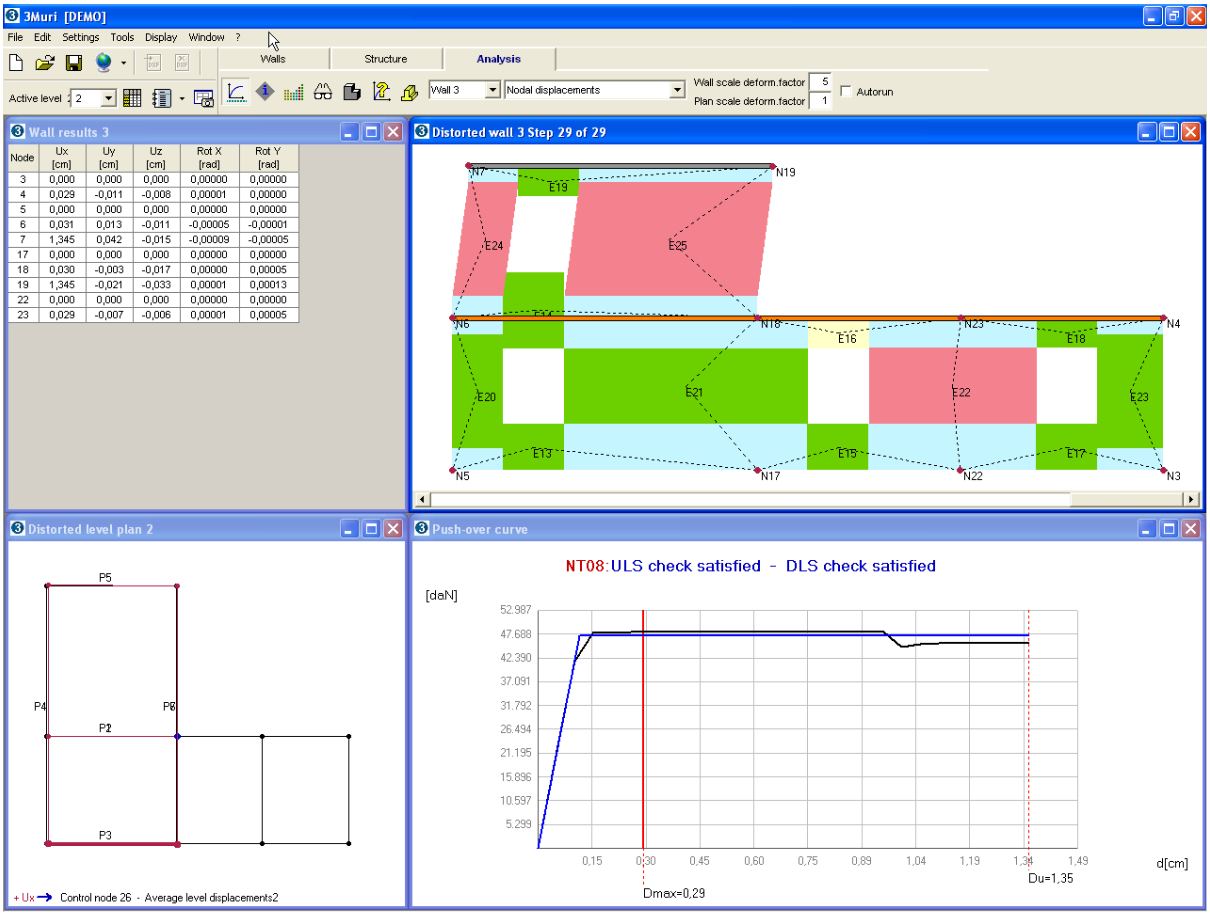
This window shows the results of the seismic computations performed on the model, based on that indicated in the code.



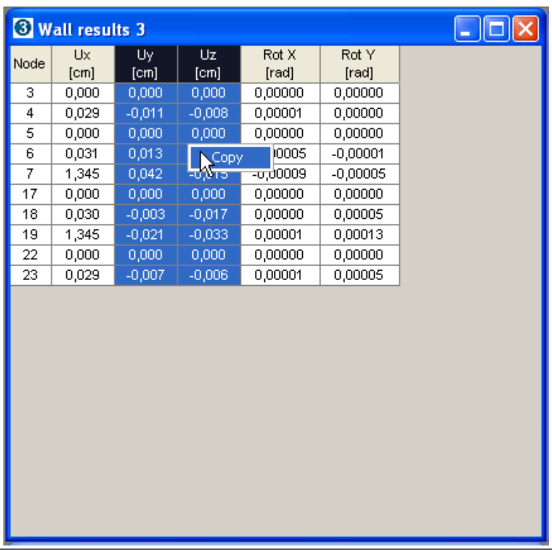
This window summarizes the check parameters according to each norm, indicating whether the results were satisfactory or not.
On the right of the window there are commands with the following functions:

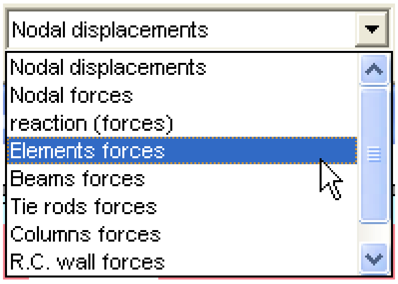
Display analysis details	Display analysis details
Insert all analyses in report	Print the parameters of all the analyses in the report
Activate code	Activates visualization of the results according to the chosen code
Delete analysis	Deletes the results of the analyses performed
Export Piano-Soil	With this button, a file containing the foundation loads is exported This file is created so as to become input for the Piano soil program for computation of the foundation structure. (Piano soil is a product created and distributed by Aztec Informatica; www.aztec.it)

10.4.3.1 Display analysis details



The push-over curve and the outline of the equivalent system bilateral are shown in the window. Based on the user defined code, the corresponding conditions to be satisfied are shown.
Displacements, force and node deformation characteristics for the various walls that make up the structure are shown in another window. This is shown for each analysis substep.





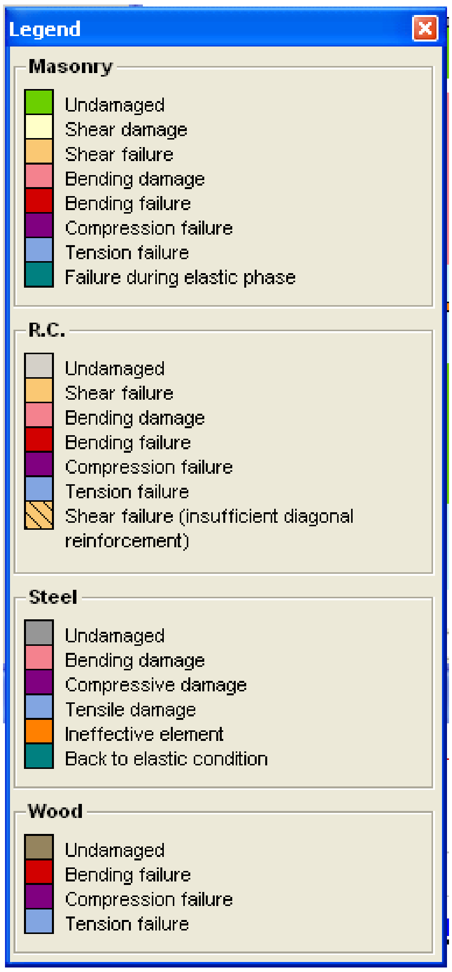
"In this window each value can be selected. Click the right mouse button and "Copy data" will appear in the selected area. If it is selected, the information will be saved to the clipboard and it could be paste to the most useful application (Word, Excel, etc.)."

Here the deformation alignment, both for the plan and the wall is shown. The change in color highlights the state of advancement of damage in the various macro-elements.
By observing the colors found in the damaged wall map, it is easy to understand which

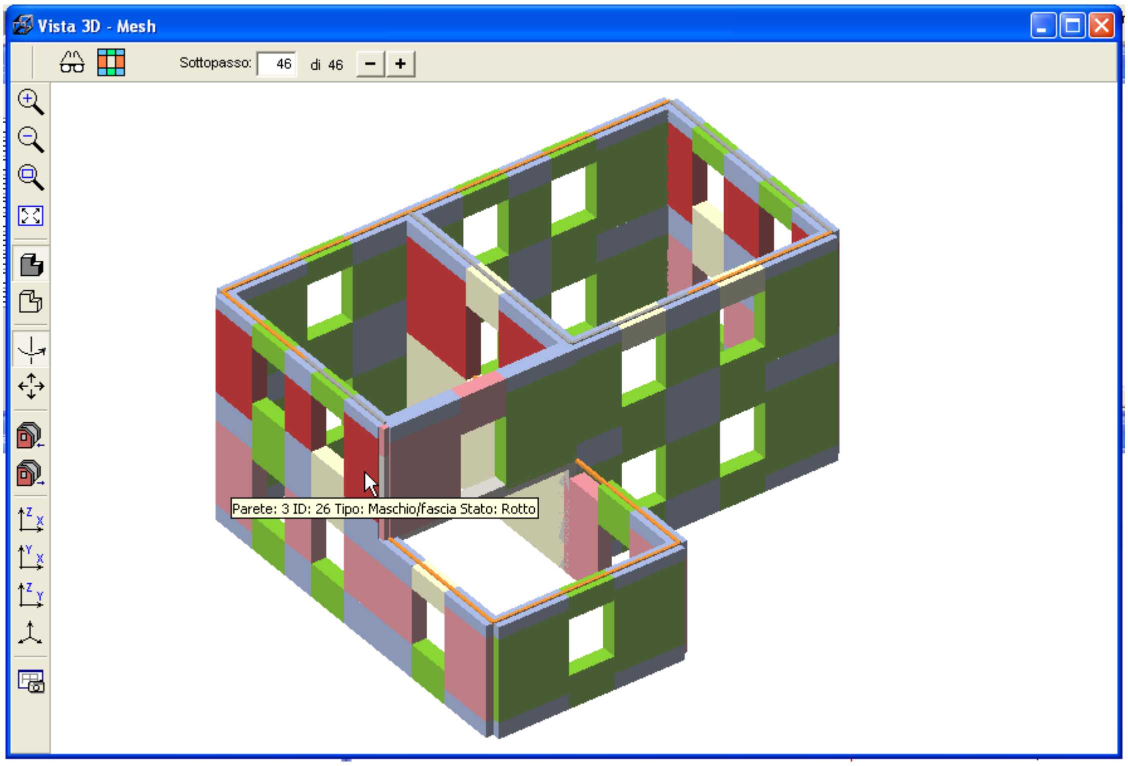
macro-elements are damaged, and the cause of the damage (shear, compression bending). It is also possible to examine the tendency towards damage for all non-damaged elements, as well as determining whether they become plastic due to shearing or compression bending.
This type of visualization is not only for masonry elements, but also for reinforced concrete, steel, or wood.
This instrument is also extremely useful for the management of necessary corrective changes toexisting buildings. It makes it very easy to identify the zones in which to make changes.

Selecting the command "Autorun" an animation of deformation can be seen, which shows the various phases of damage advancement.


Color legend: Shows a color map which identifies the various types of structural damage. The map shows the damage for masonry elements, as well as those in reinforced concrete, steel, or wood.

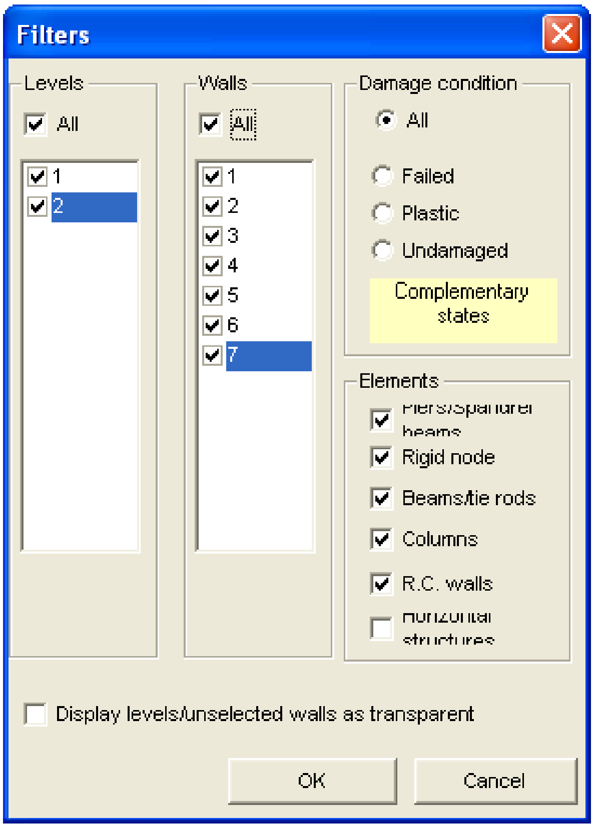



Display3D mesh: Shows the 3D mesh, showing it in function of the state of damage using the color map described above.



3D view commands:

 Display filters:

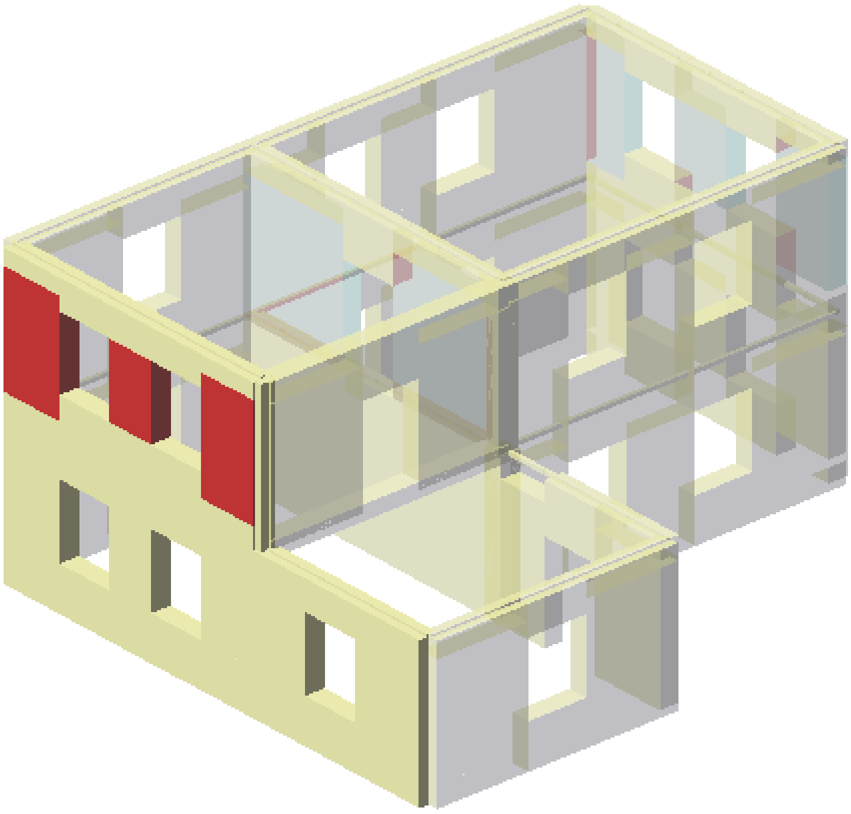


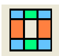
-*Levels*: Only the selected levels are displayed
-*Walls*: Only the selected walls are displayed
-*Elements*: Only the selected elements are displayed.

The levels/walls that are not selected can be displayed as transparent.

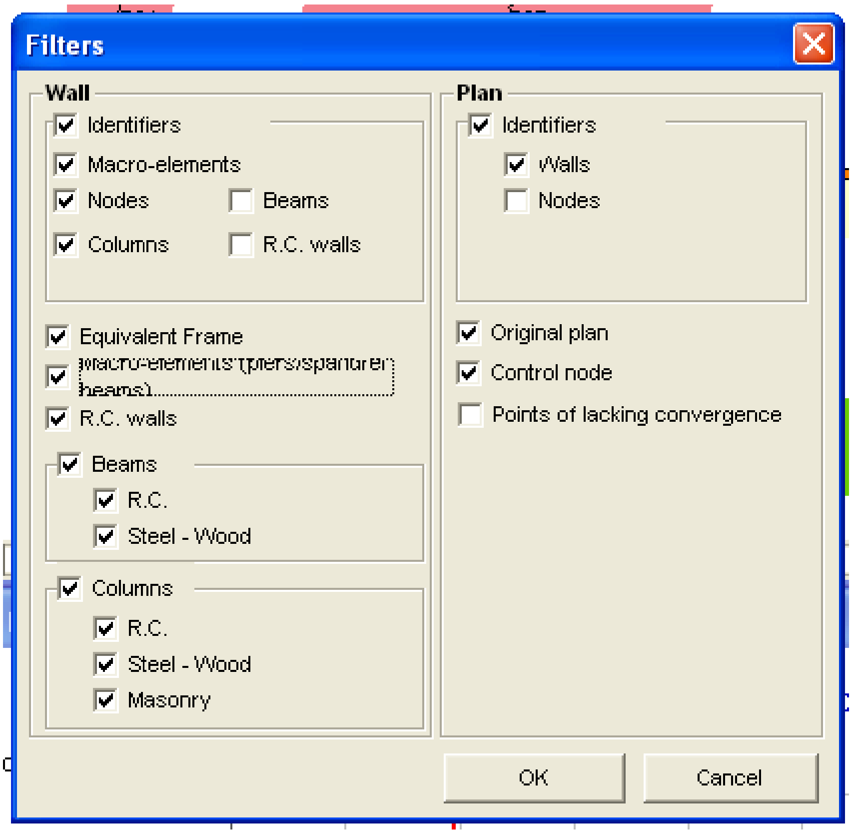
It is also possible to display only one type of elements demage state. Complementary demage state type are colored in light yellow (as shown in dialogue box).

Displaying the structure as transparent and showing only the failed elements makes it easier to identify where the structure is weak.

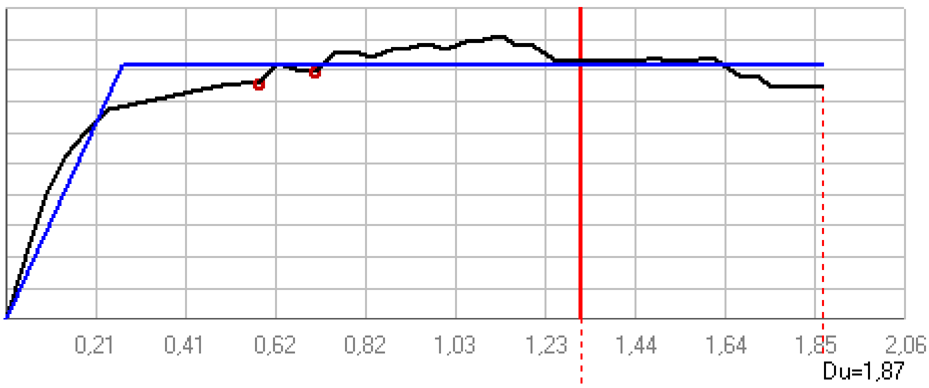


Load wall: When a wall is selected in 3D view, a window to verify details for the wall in question will appear.

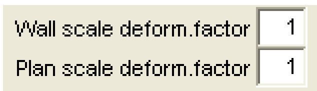
Display filters:The user can decide what to display in the two deformation windows (plan, wall).



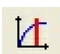
With this window it is possible to decide whether or not to show the points lacking convergence in the non-linear analysis. These are shown in the pushover diagram with small red circles.



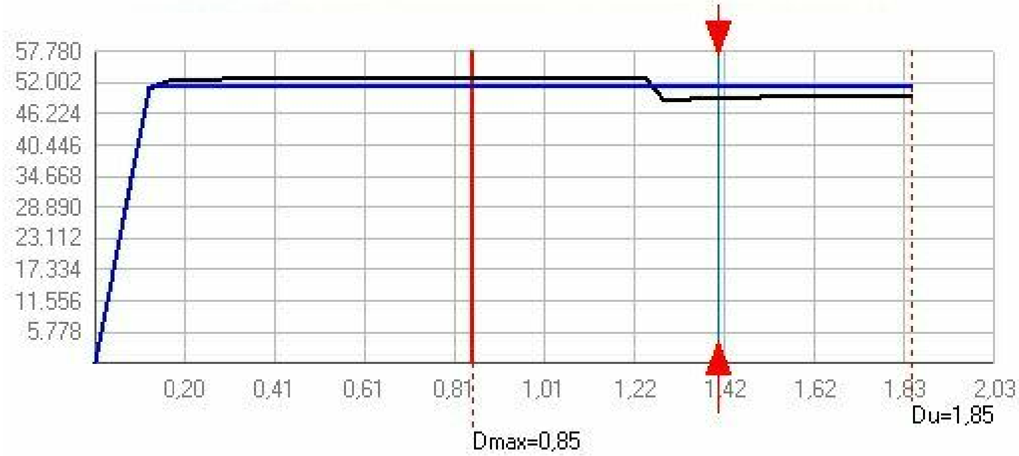
The presence of some points lacking convergence should not cause worry. However, the presence of a high number of non-converging steps can indicate a model that needs improvement.



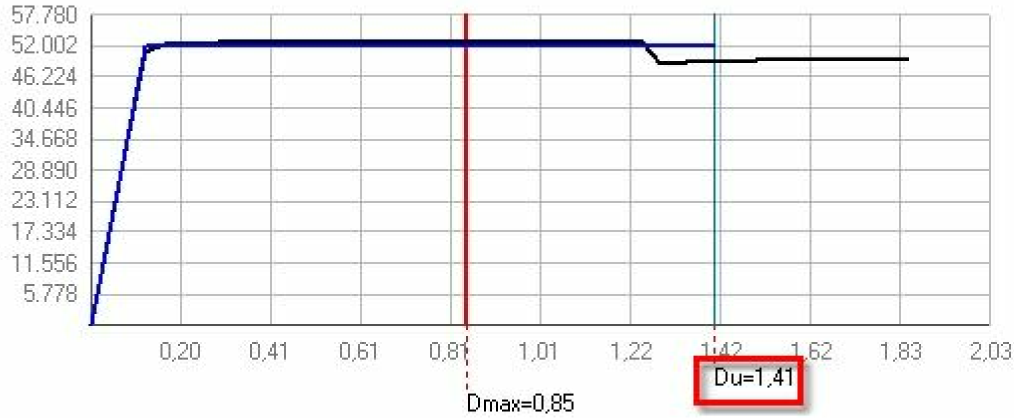
These commands, which can be found in the analysis toolbar, allow selection of the display scale to be used to visualize the deformations, both in the wall and the plan view.

 New Last step: If the customer considers it opportune, he can decide a different value for the ULS. The new value provided must be inferior to the resistance decay limit. Use the sliding bar for moving step-by-step the blu line cursor . In this way it is possible

to select the appropriate position for the new “last step”.



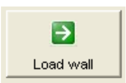
Pressing the push-button “New last step”,the program defines a new ultimate displacement value and it redefines the bilinear equivalent.



ATTENTION! For very small displacement values it is not possible to define new ultimate step; small values could not able regeneration of the bilinear equivalent because intersection of the bilinear with pushover can't respect the constructive prescription (i. e.: the dissipated energy conservation is not possible).

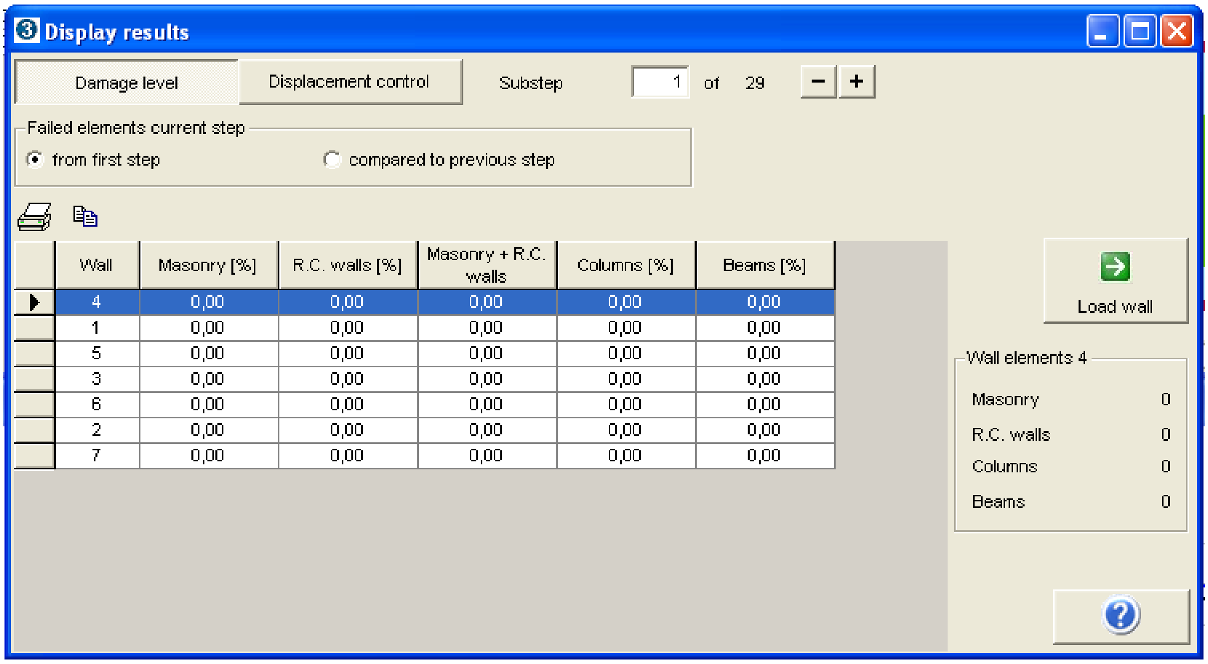
10.4.3.2 Display Results

This window shows a table with information about the percentage of damaged elements in each wall. The walls are shown in order, based on the percentage of damaged elements. With this system, the most damaged wall is easily identified, as it is always first on the list.

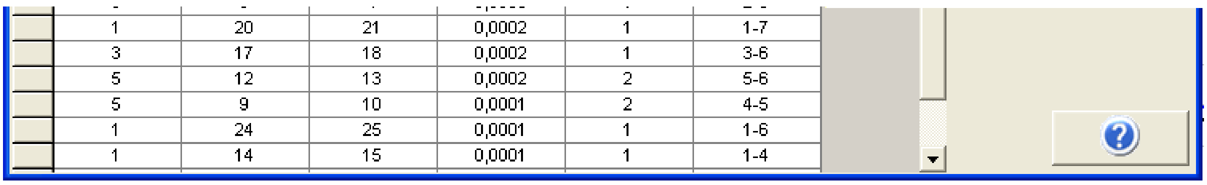


It is possible to immediately load the image of the selected wall (the corresponding line). In this way, it can be viewed to take action.

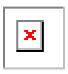
The percentage of failed elements presented in the table can be defined by the beginning of the load history, or by the current analysis step.



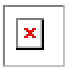
A second area for "displacement control" places the walls in order, based on the relative displacement of the floor. In this way, the zone with the greatest displacement can be identified.



10.4.3.3 Foundations Analysis

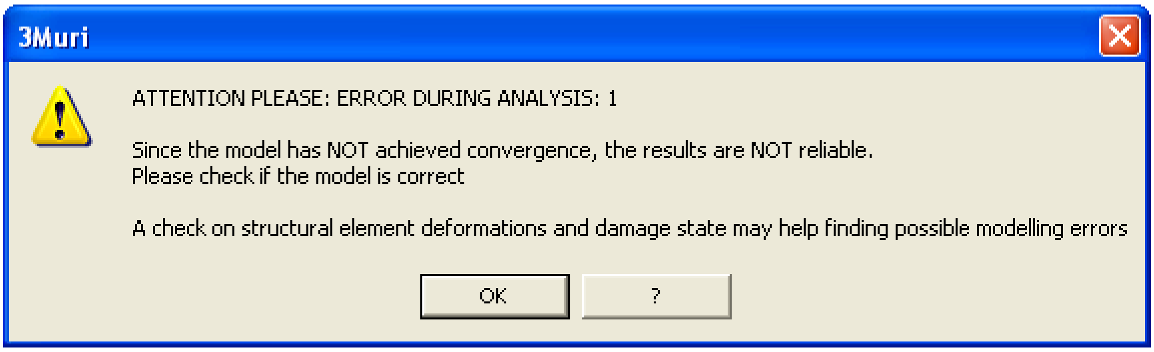


Pushing this button will display a table that shows a list of wall segments.
For each of these, the tension in contact with the soil (foundation-ground) of the current step and the maximum value between the first step and that corresponding to the displacement value equal to Dmax is shown.



10.4.3.4 Convergence Problems

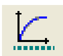
In some cases the program may indicate a lack of convergence for the Self weight.

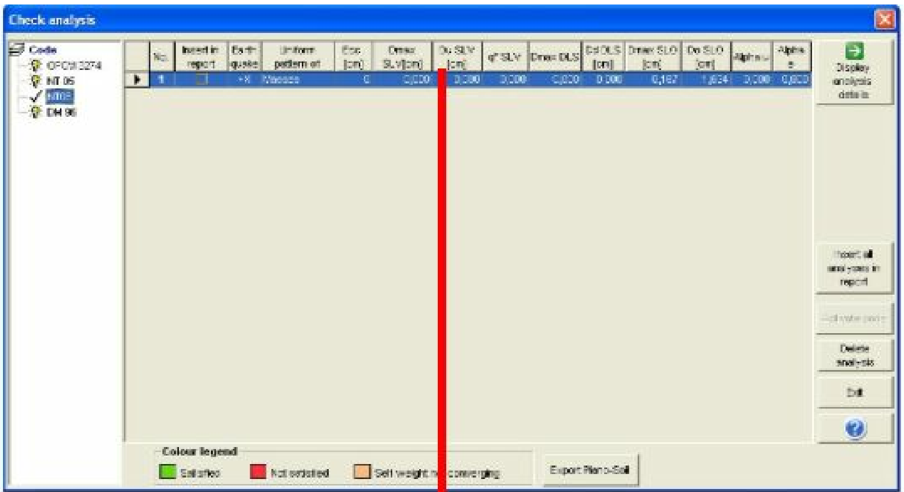


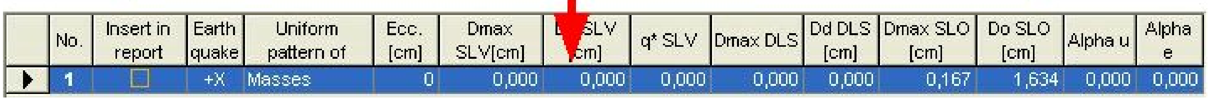
Often, this notice is shown when there is an error of modeling or under-sizing of some structural elements. For example, not enough beam reinforcement.
Under-sizing don't allows the element in question to resist in the static field. In this case a lack of convergence is created in the non-linear analysis.

The best way to identify the points where structural weakness led to the problem, is to view the wall deformation with its respective level of degradation.
When a lack of convergence is found, the user is still able to see the results, showing only the deformed area (check environment).

What is the problem?

1. Push the button to see the results .
2. A window will be displayed that shows the analysis that does not converge with all of the null results.





	No.	Insert in report	Earth quake	Uniform pattern of	Ecc. [cm]	Dmax SLV[cm]	Du SLV [cm]	q* SLV	Dmax DLS	Dd DLS [cm]	Dmax SLO [cm]	Do SLO [cm]	Alpha u	Alpha e
▶	1	☐	+X	Masses	0	0,000	0,000	0,000	0,000	0,000	0,167	1,634	0,000	0,000

3. Pressing the button Show details opens a window that shows deformation of individual walls.

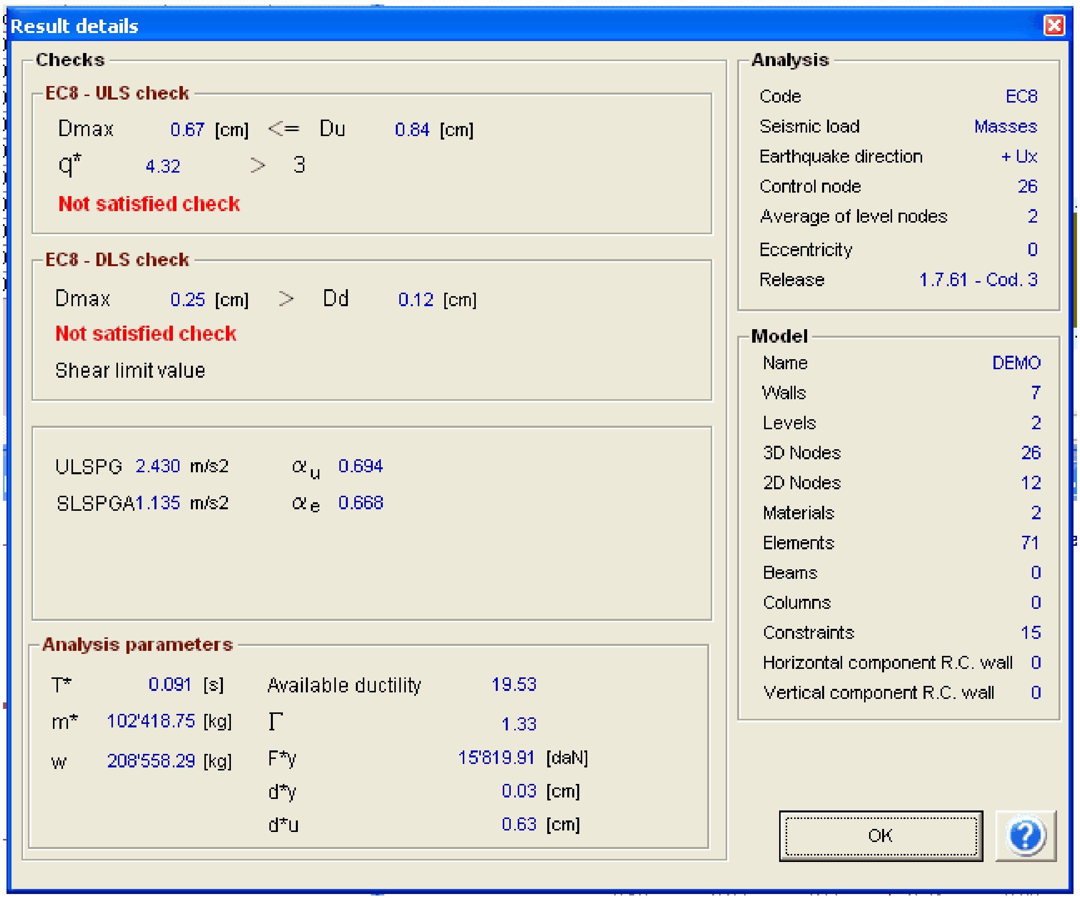
4. By observing each individual wall it is possible to search for elements that were severely deformed or failed under the vertical load.

What are the possible causes?

1. Loads on the floors that are too high (load values are estimated incorrectly).
2. The mechanical parameters of the masonry are not appropriate.
3. The reinforcement of the reinforced concrete elements is underestimated for a self weight analysis.

10.4.3.5 Results

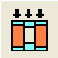
This is a window that offers a summary of the results from the requested analysis and checks.



This window shows the check parameters required by each code.

In the area "Analysis Parameters" the following factors are shown:
T*: Equivalent system period
m*: Equivalent system mass
W: total mass
Available ductility: ratio between the ultimate displacement and the elastic displacement limit
Γ: modal partecipate factor
F^*_y: plasticization force of the eqivalent system
d^*_y: plasticization displacement of the eqivalent system
d^*_u: ultimate displacement of the eqivalent system

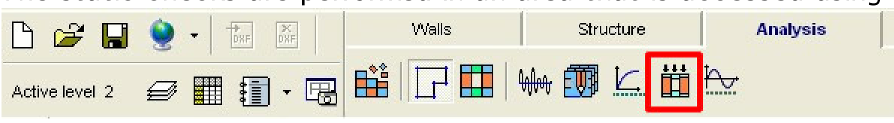
10.5 Static Analysis



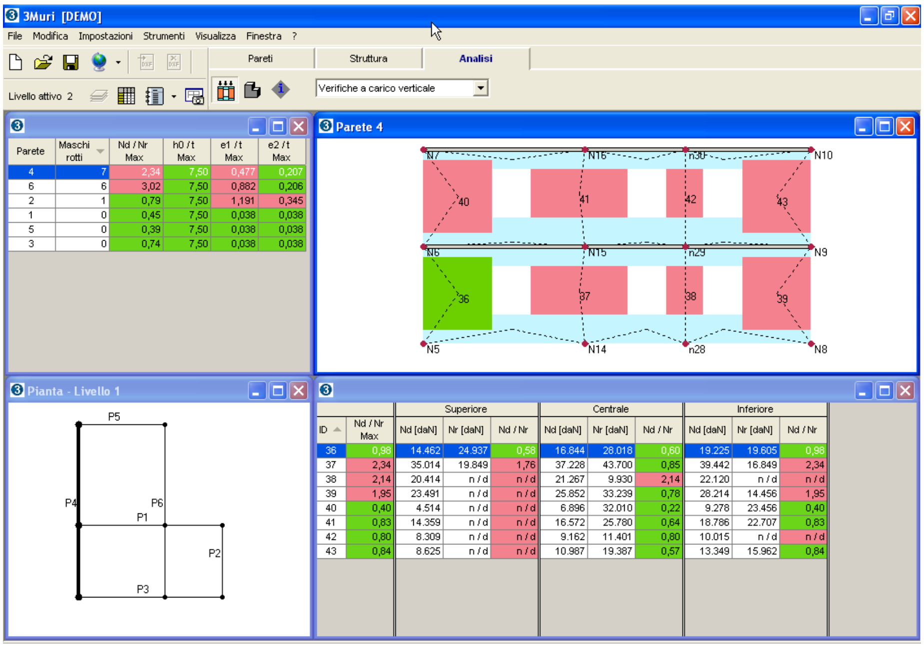
This is a module which performs static checks on the structure, according to the code in effect.
The program uses the meshes already created to perform the non-linear analysis, adapting the equivalent frame theory to perform the static checks in the linear field.
Below are the checks that are performed:

Slenderness check:	**$h0/t \le 20$**
	h0: effective length of the wall equal to r·h *t: thickness of the wall*
Load eccentricity check:	**$e_1/t \le 0.33$** **$e_2/t \le 0.33$**
	$e_1 = \| e_s \| + \| e_a \|$; $e_2 = \frac{e_1}{2} + \| e_v \|$ *es : total eccentricity of the vertical loads;* *ea : eccentricity due to execution tolerance;* *ev : eccentricity due to wind;*
Vertical loads check:	**$N_d \le \Phi\ f_d\ A$**
	***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.*

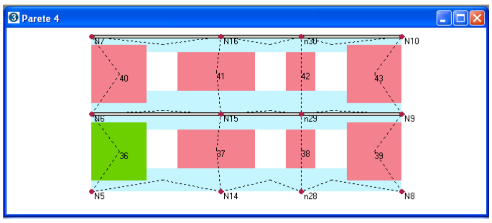
The static checks are performed in an area that is accessed using the associated button.



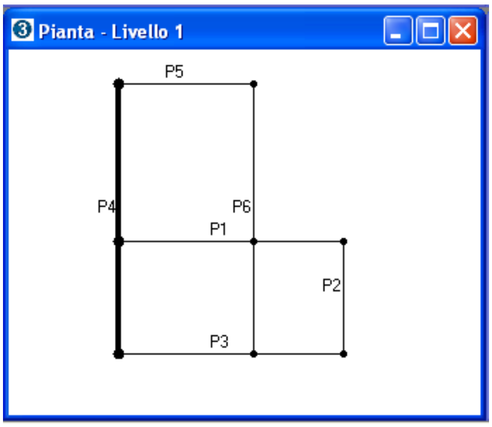
The following screen will appear:



This video is very similar to that which presents the results of non-linear analysis. Here we describe it in detail.



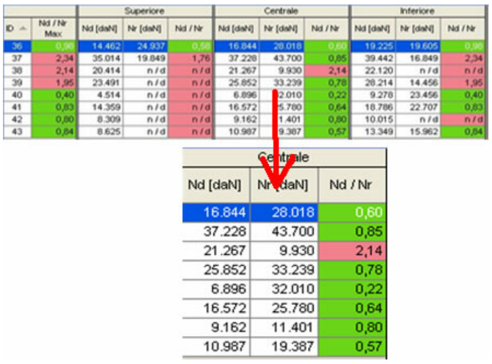
In the upper right the wall mesh appears.
In this case, the legend with colors indicating different phases of damage does not appear.
Elements that passed the check appear in green. Those that did not appear in other colors.



At the lower left, the plan view is shown. The wall shown in the precedent view is highlighted with a thick line.

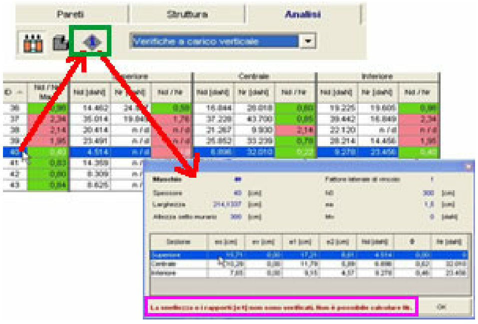
Parete	Maschi rotti	Nd / Nr Max	h0 / t Max	e1 / t Max	e2 / t Max
4	7	2,34	7,50	0,477	0,207
6	6	3,02	7,50	0,882	0,206
2	1	0,79	7,50	1,191	0,345
1	0	0,45	7,50	0,038	0,038
5	0	0,39	7,50	0,038	0,038
3	0	0,74	7,50	0,038	0,038

On the upper left there is a list of 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.

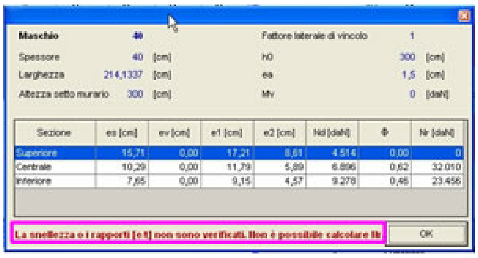


Nd [daN]	Nr [daN]	Nd / Nr
16.844	28.018	0,60
37.228	43.700	0,85
21.267	9.930	2,14
25.852	33.239	0,78
6.896	32.010	0,22
16.572	25.780	0,64
9.162	11.401	0,80
10.987	19.387	0,57

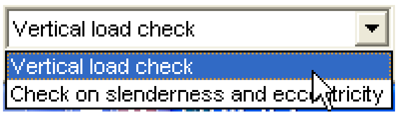
At the lower right, the elements detail window is shown for the selected wall.
For each masonry element, the checks are performed for three different sections (higher, central, lower).
For each section the value for normal forces strain is shown (Nd: computed based on the masses and the combinations of the loads) and the normal resistant strain (Nr =Ffd A).
The check is satisfied if the ratio Nd/Nr £ 1. In this case, the corresponding cell appears in green.
In some cases, as in the example here, Nr cannot be calculated (n/d: not defined). This happens when the slenderness or eccentricity checks are not satisfactory.



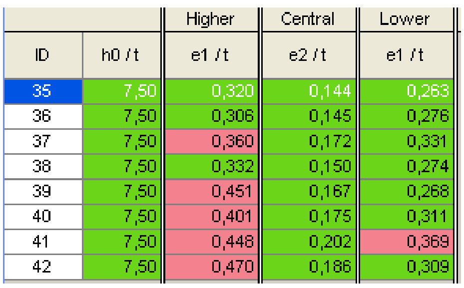
When a masonry pier is chosen from the list and the information button is pressed, a window will appear which contains the computation details.



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 information relative 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 the user the possibility to select various elements in different wall and still have the details for each individual check visible.



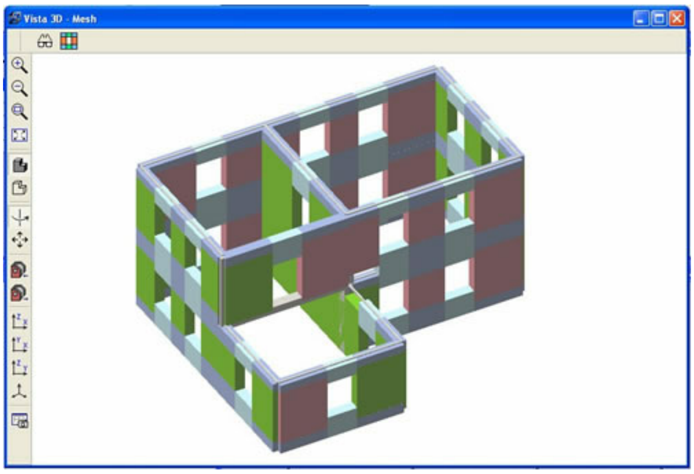
Through the associated menu on the results bar, it is possible to switch to visualization of the compression results from the slenderness and eccentricity results.



		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 interpret the results, 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 order 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

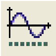
...or it can be ordered based on what the check penalizes the most.

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

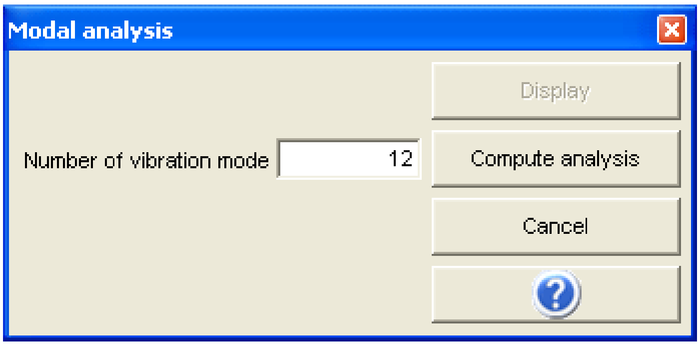
This table can also be reordered.
It is ordered based on the wall identifiers and the efficiency of the compression curve (only the total for the element, and not for individual sections).

10.6 Modal Analysis



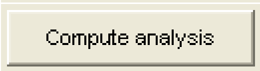
This is an area dedicated to computation of modal forms and the parameters associated with them.

When the appropriate button found in the analysis bar is pushed, the following window will appear:



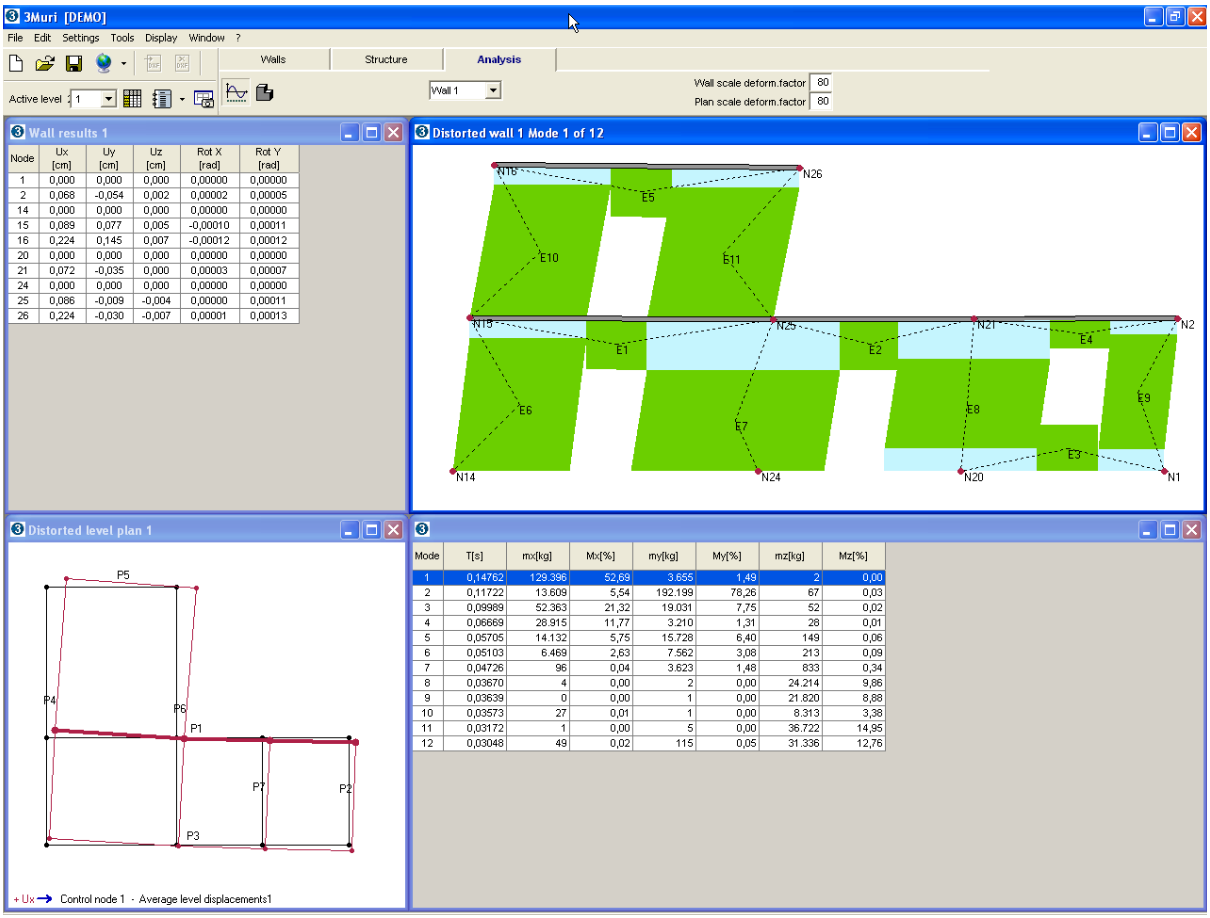
A number of predefined modal forms are offered.

If the computation has already been performed, the results are shown.



The computation is performed, and the results are shown.

When the calculation is finished, the presentation of the results is automatically shown.



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.

10.7 Local Mechanisms Analysis

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.

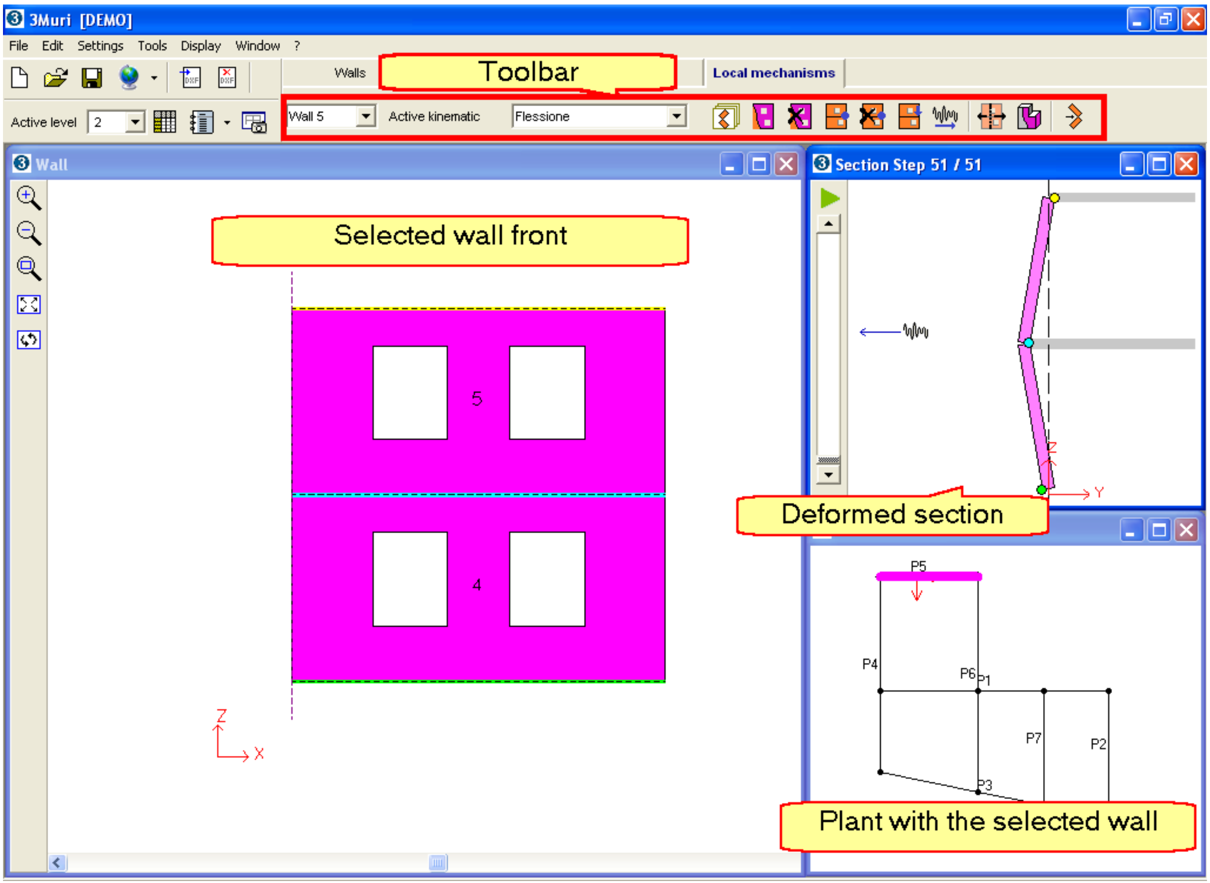
"Tremuri LM" is a calculation module inside the Tremuri program, which is dedicated to the evaluation of the building safety against such mechanisms.
The module **"Tremuri LM"** exploits the versatility and the input ergonomics of the program TreMuri to finalize a spatial model on which the user can 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.

*** 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

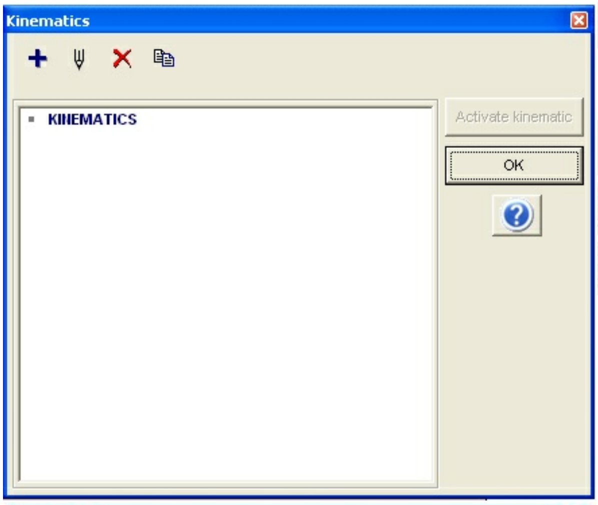
per l'analisi strutturale degli edifici in muratura, Volume II – Parte 1a. Convenzione di Ricerca con la Regione Marche; Consiglio Nazionale delle Ricerche – Istituto per la Tecnologia delle Costruzioni – Sede di L'Aquila; Dipartimento di Ingegneria delle Strutture, delle Acque e del Terreno (DISAT) – Università degli Studi di L'Aquila. L'Aquila, 2005.

10.7.1 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".

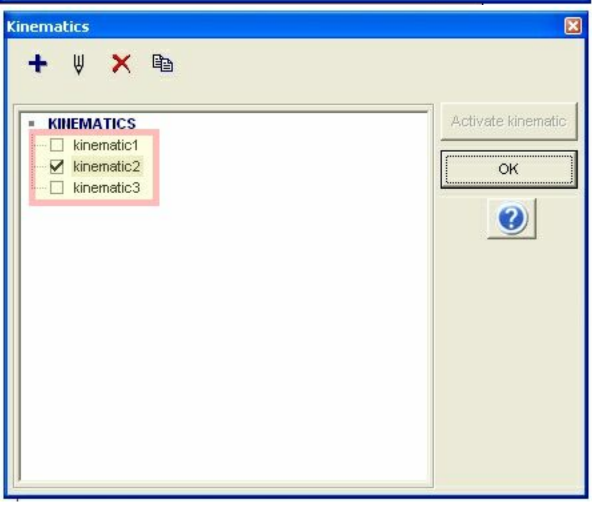



New Mechanisms (enter the name of the mechanism)

 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.

Used to activate one of the "kinematics containers", indicating on which kinematic decide to work.

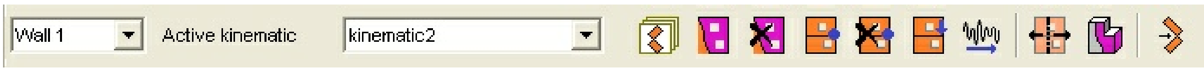
The active mechanism is represented by checking the box (☑) to the left of the name.

Confirming with OK the window closes and displays the name of the active kinematic combo box "active kinematics" shown in the toolbar.

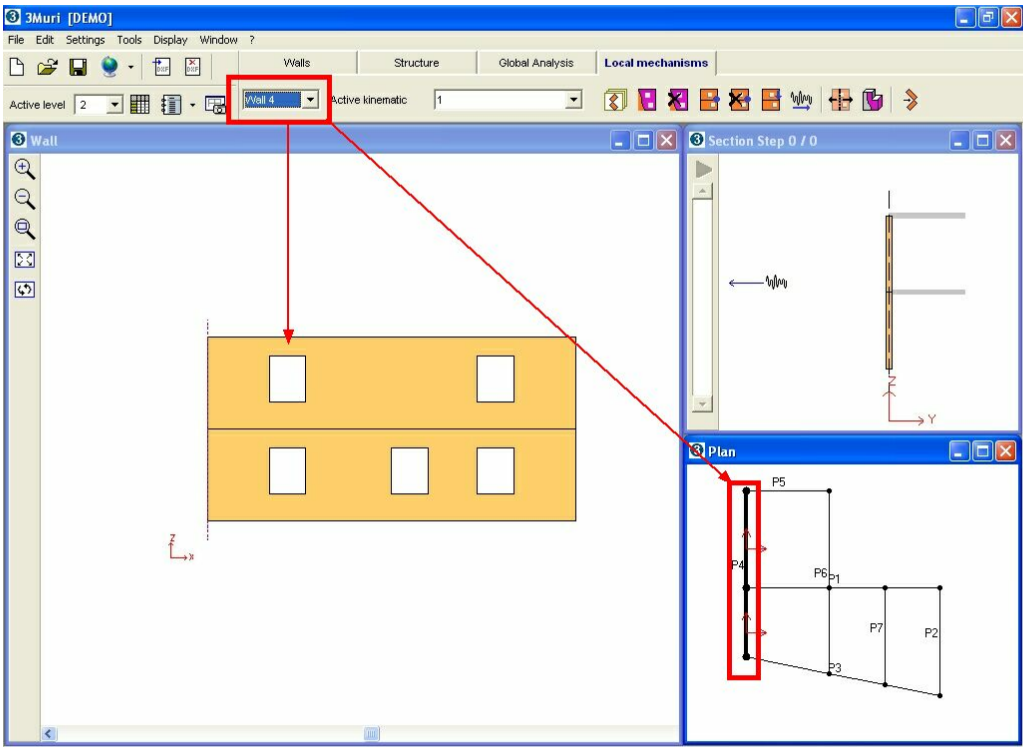
It is possible to use this combo box to change the active kinematic.

10.7.2 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.

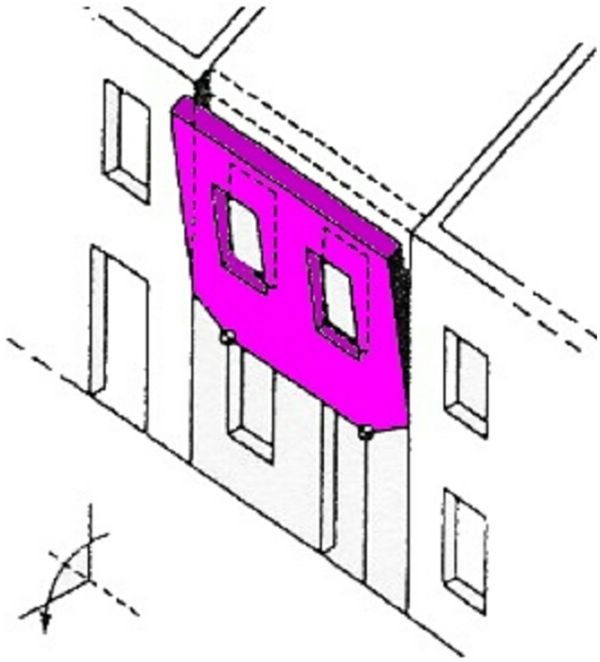


The mechanism input consist in three steps:

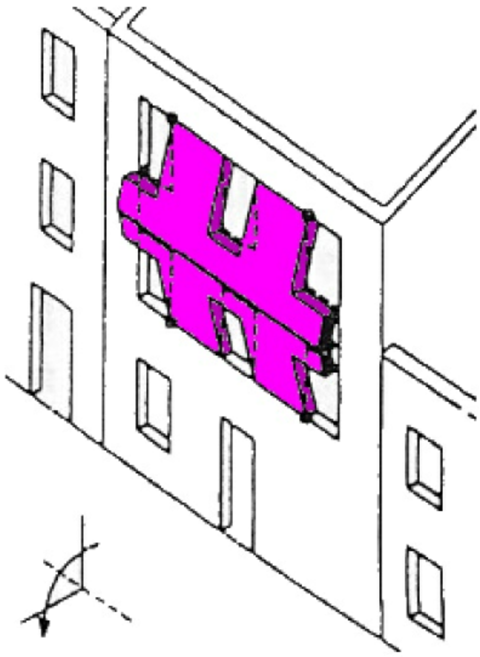
- Inserting **Kinematics Blocks**
- Inserting **Vincoli Constraints**
- Inserting **Loads**

10.7.2.1 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.



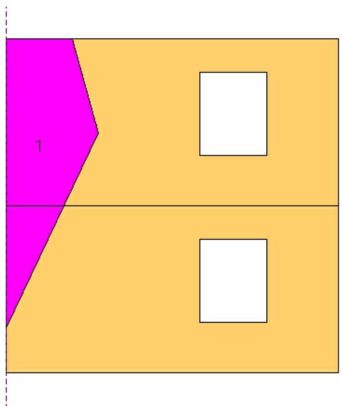
Example of a mechanism consisting of a single block



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.

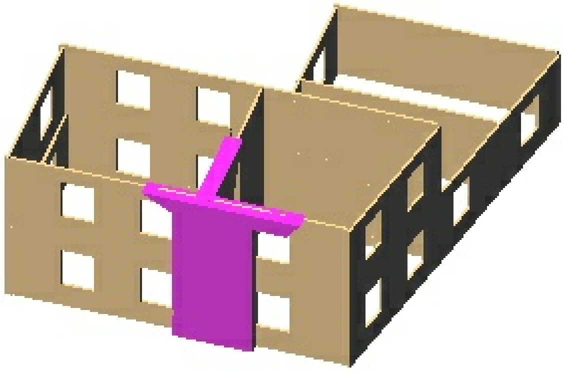


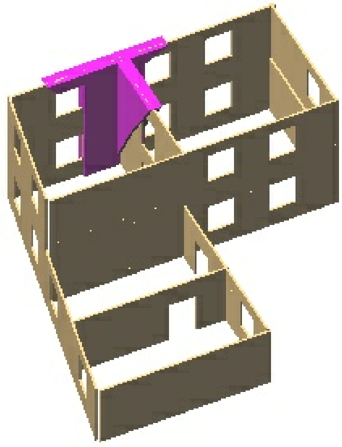
The possibility to draw a closed perimeter allows the user to trace the edges in correspondence of the panel crack found in site.

(*)

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.

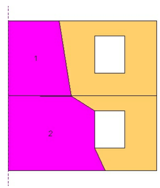




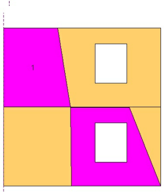
Axonometric views such as those described above are visible by pressing the "3D View" button .

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.

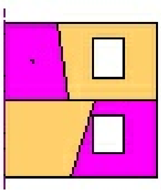
Therefore apply the following construction rules:



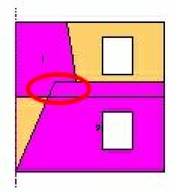
CORRECT
Blocks 1 and 2 have a common side



NOT CORRECT!!!!
The blocks 1 and 2 have only one common side.



NOT CORRECT!!!!
The blocks 1 and 2 have no common point

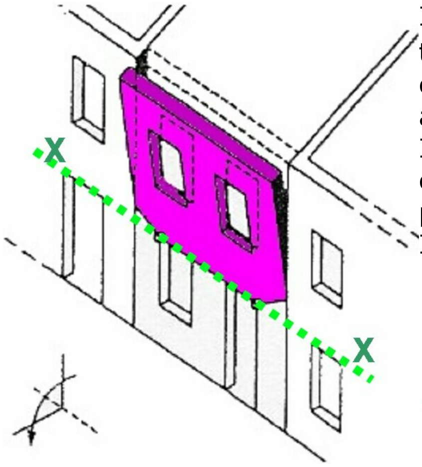


NOT CORRECT!!!!
The blocks 1 and 2 are overlapped

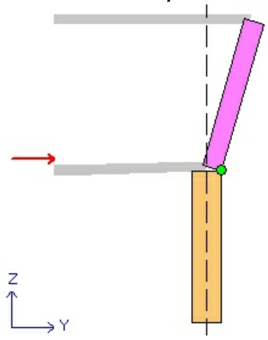
**Delete blocks:**
Selecting one or more blocks in sequence, confirming by pressing the right mouse the selected blocks are deleted.

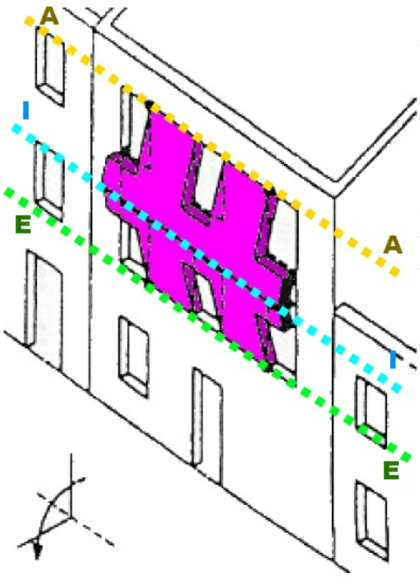
10.7.2.2 Constraints

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.

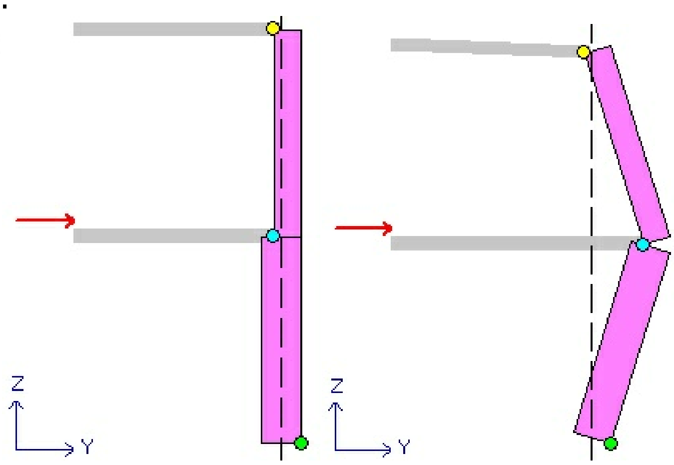


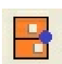
If we want to consider a tilting case of a wall portion like the one represented in the image (*), the considered case is the case of a block that rotates around the X-X axis .
In correspondence to this point you must enter a constraint between a kinematic block and a fixed wall portion.
In this case you must insert the **"External Hinge"**

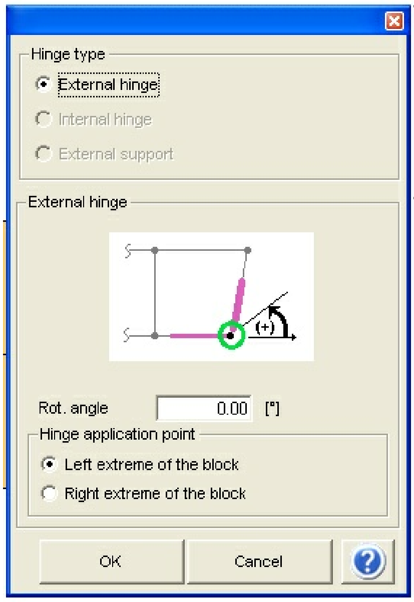




In this case (*) the bottom block is placed directly on a wall portion that is not deformed .
In the **E-E** position will be put the **"External Hinge"**
In the **I-I** position confines two blocks, so will be put the **"Internal Hinge".**
In the **A-A** position the deformation mechanism will not allow 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".**
.

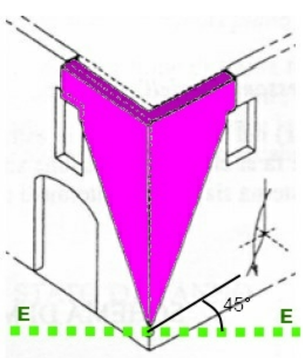


**Insert constraint:**
Pressing this button will display the input window of constraints.



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.



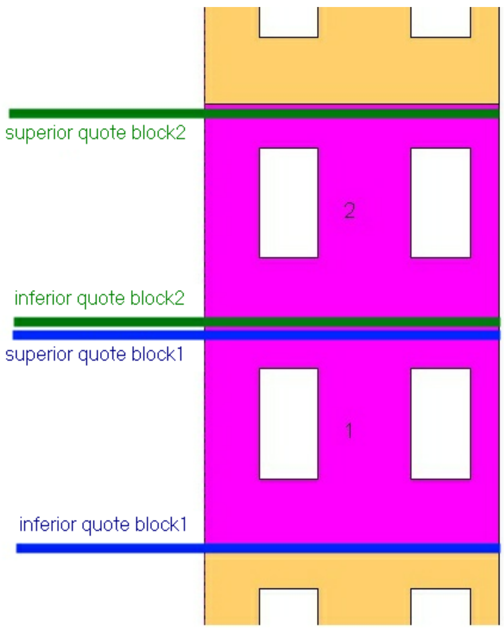
This function can be useful to examine tilting cases of the cantonal. (*)

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.

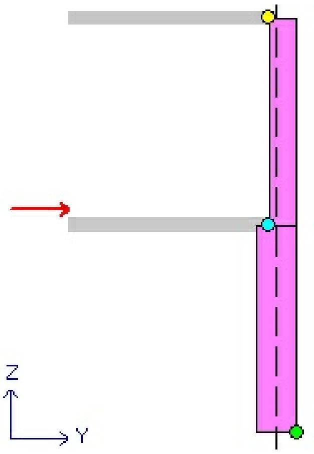
Clicking on the block, the constraint is inserted to the:
- **Bottom** quote of the block if **External Hinge**
- **Top** quote of the block if **Internal Hinge**
- **Top** quote of the block **if Support**

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.

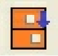


 If you want to change the sisma verse you can use the appropriate button shown to the left of the section.

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.

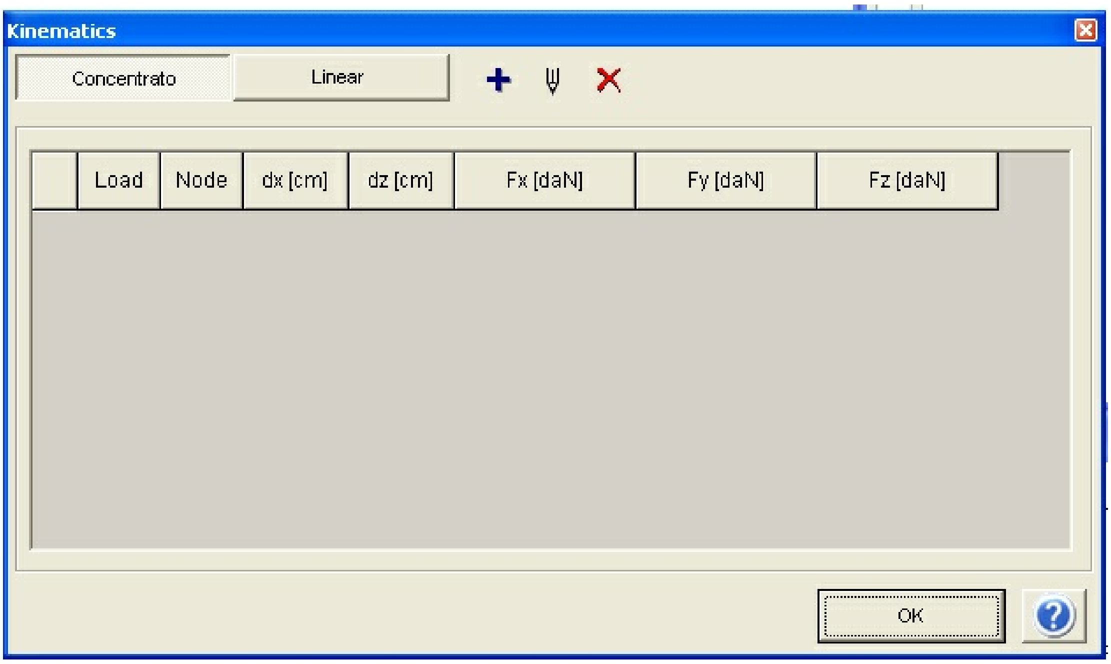
 **Delete constraint:**
Allows to remove a constraint.

10.7.2.3 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. ..

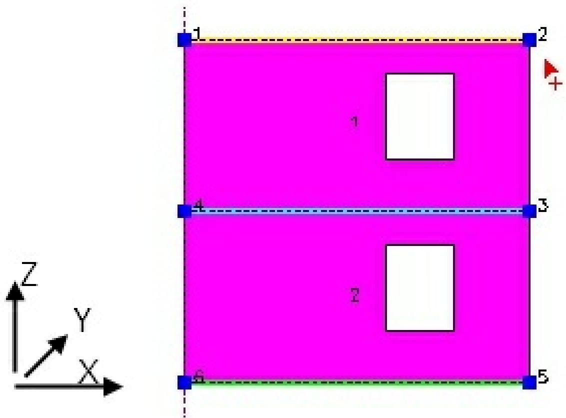
Pressing the "Loads" button will appear the dialog window like the image below:



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.

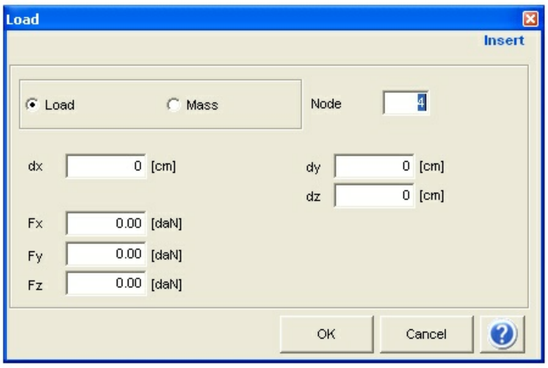
**Insert new load:**

Pressing the button, the snap becomes "selection snap" () and on the wall front are shown selected nodes.



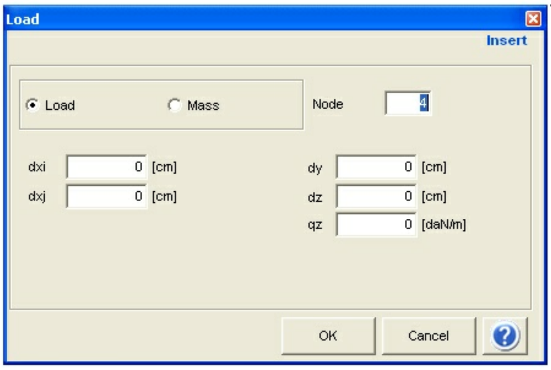
Select the node nearest to the point where you want insert the load.
After the selection, will appear the dialog window like the image below.

Concentrated load

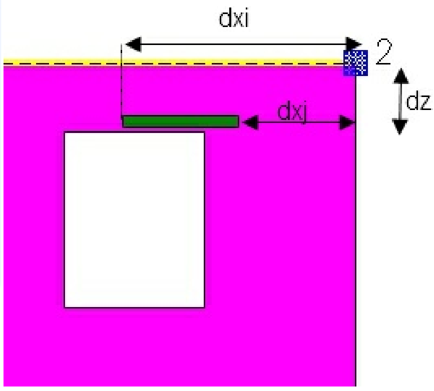


Node: Indicate the number of the selected node in graphical modality.
Force: If you insert a vertical load (Fz) different from zero, this load does not generate mass, without creating any horizontal component of seismic type (α*Fz; α: factor of vertical load).
Mass: If you insert a vertical load (Fz) 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



qz: distributed vertical load .
dxi / DXJ / dz: are the relative coordinates of the load application points compared to a node centered system.



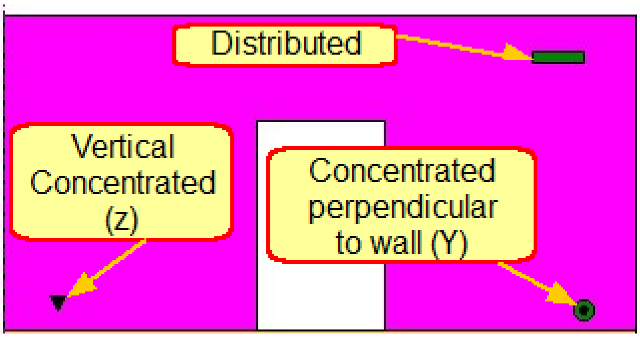
Confirming the inserting of the load, the table is updated with as many rows as there are loads included.

	Load	Node	dxi [cm]	dxj [cm]	dz [cm]	qz [daN]
▶	1	4	-200	-100	0	-100

 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.

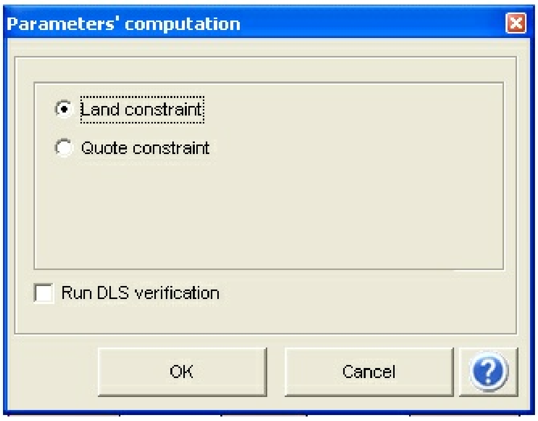
The loads are shown in the below graphic with the following agreement.


Distributed
Vertical Concentrated (z)
Concentrated perpendicular to wall (Y)

10.7.2.4 Calculation

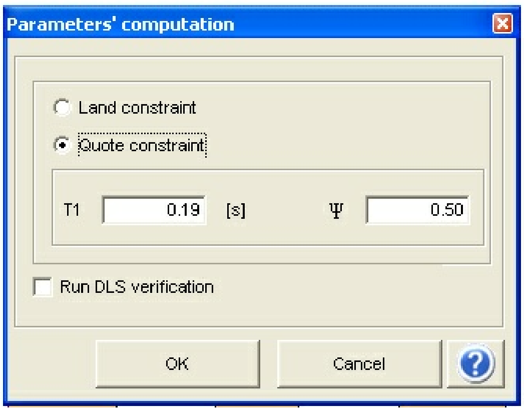
When the input is complete, you can proceed with the calculation.
With the module **"3Muri LM"** is possible to run the Verification of the Linear Kinematic Analysis.

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

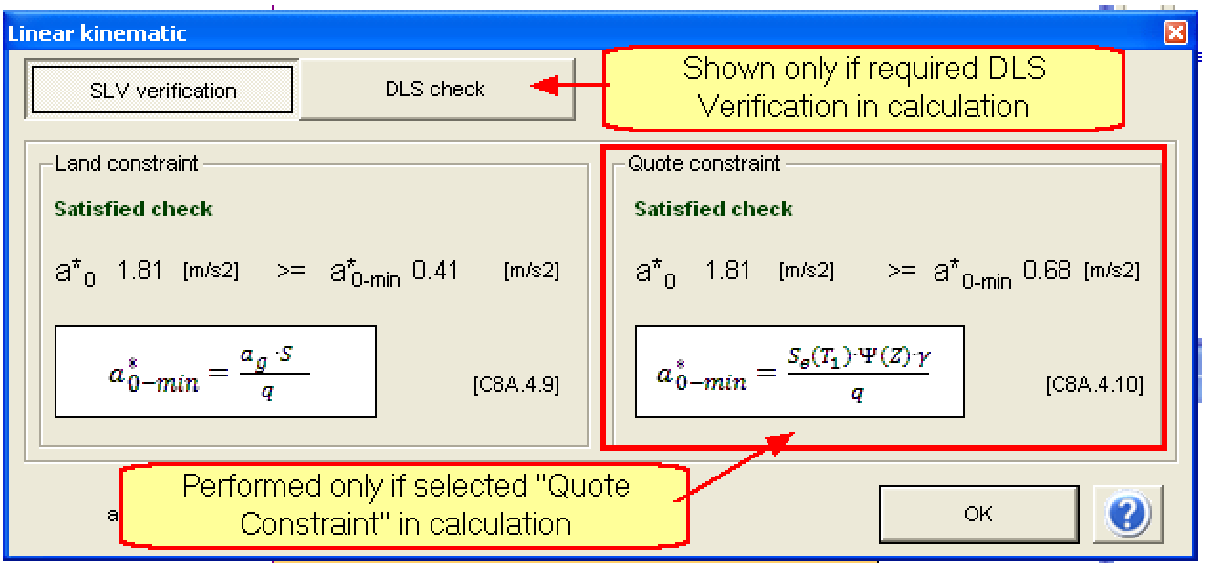
constraint lines between the blocks interested by the mechanism.

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.

Pressing the OK button the calculation is performed.

10.7.2.5 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

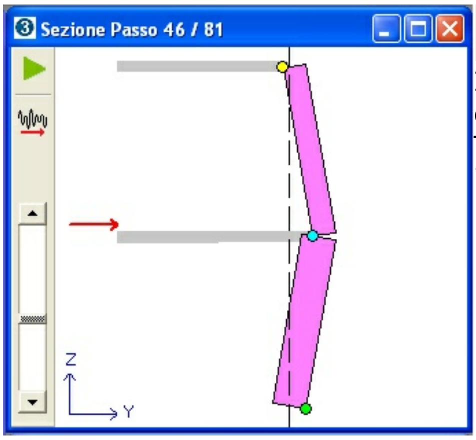
Se(T1): elastic spectrum, function of the probability of exceeding the selected Limit State (in this case 63%) and the reference period as VR, calculated for the period T1;
ψ (Z): is the first vibration mode in the considered direction, standardized at a summit of the building, in the absence of more 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

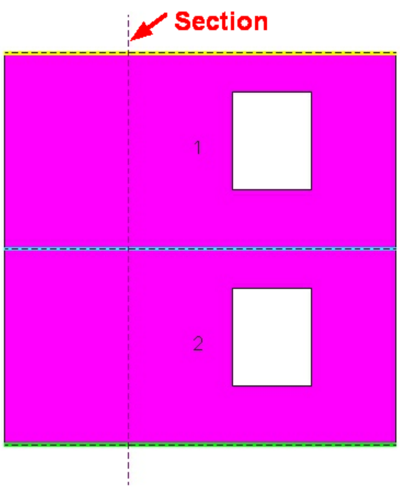
factor q (linear kinematic analysis)
- **Quote Constraint** should be conducted both calculations (the verification with simplified structure factor q and the verification taking into account that the spectrum of response is related to the probability of exceeding of 10% over the reference period VR.

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.