User Manual

Release: 13.5.

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

Indice

0

1 3Muri - Commercial versions

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

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

✓ : Standard module

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

2 Latest updates

A renovated solver

Each calculation engine constitutes a "container of knowledge" transformed into mathematics and is made up of two main components:

- Technical scientific knowledge
- Regulatory knowledge

Normative skills alone are not sufficient to perform a calculation since normative texts contain information too limited to be considered completely thorough.

The information contained in the texts of current regulations therefore constitute the guide to good planning and, at the same time, the minimum requirements that must be met. To follow these directives, the scientific instruments that the world of research with its countless publications is able to provide, cannot be ignored.

The world of university research is in constant movement, in the constant search for procedures that can better represent reality through numerical modeling.

Software companies that work closely with a University have the potential to be able to collect a large number of case studies that arise from the direct experiences of us designers who use these technological tools to study real cases.

Tackling the practical problems that professionals have to overcome on a daily basis helps us to go further in scientific research, finding different methods to solve problems that the current Regulations cannot solve independently or look for solutions that show results that are more consistent with what was found on the site.

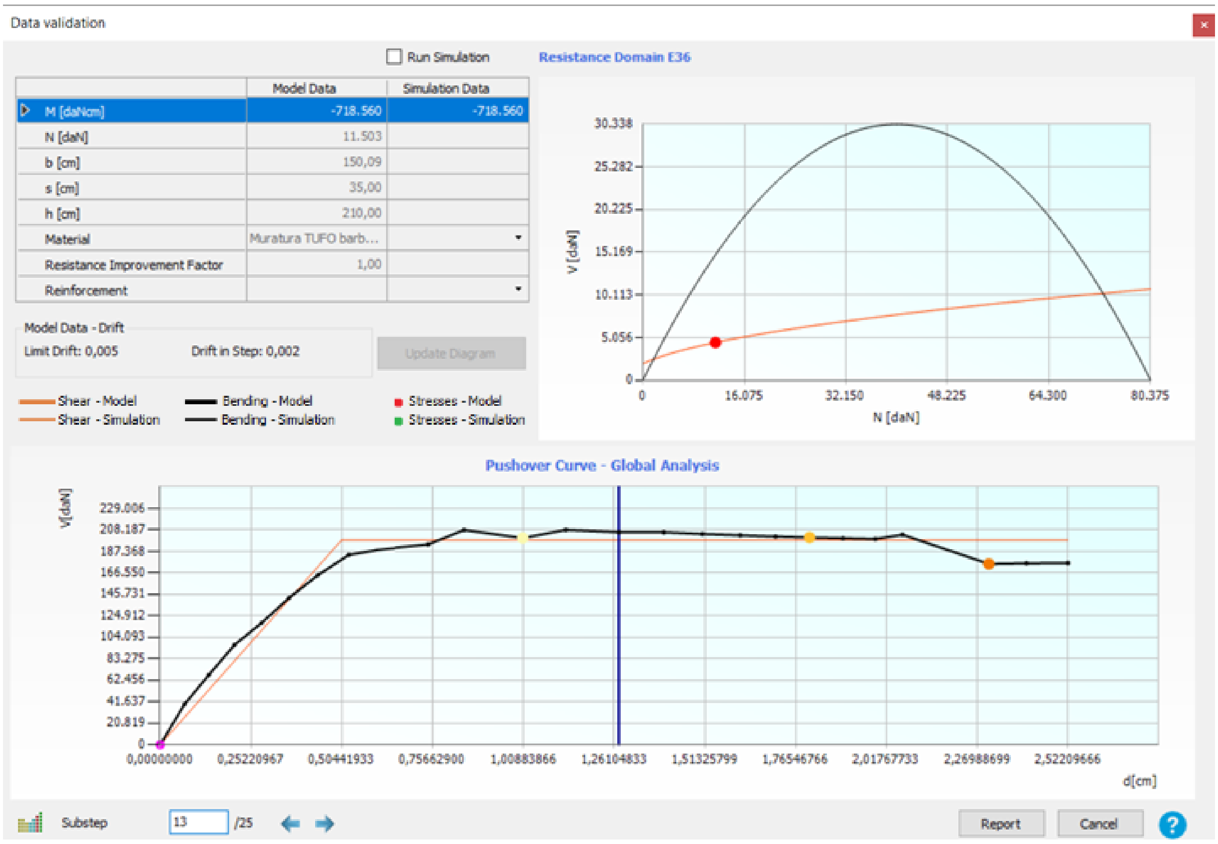
The masonry can be schematized by different constitutive bonds to be adopted based on the characteristics of the masonry texture. Regarding this, the legislation is clear: the choice of reinforcing with FRP, FRCM or reinforced masonry is not made explicit for all the bonds. For this reason, well-founded and sufficiently tested theoretical assumptions are necessary to be considered valid for the greatest number of practical cases possible. This is just one of the countless examples in which scientific assumptions allow us to perform calculations that are aligned to the reality being examined, without which this would not be possible.

Fortunately, scientific research is constantly evolving, and with this also the software on which it depends. We should not therefore be surprised if a calculation with a new engine provides slightly different results from the previous one, but be aware that this calculation is different because as research advances, a model can represent reality more accurately.

Validation Tools

As specified by the current legislation, the designer is required to carry out manual operations to validate the calculation carried out using the calculation software and the results provided by it. This operation, usually carried out with the aid of simple spreadsheets created by the user, can often be onerous, cumbersome and demanding, because of the degree of complexity reached nowadays by structural calculation software.

This new function is intended to provide the user with a powerful validation tool dedicated to the drafting of a report containing the formulations, graphs and results illustrated step-by-step, helping the professional to overcome this "annoying", but essential burden of validation of the calculation.



The validation window provides the geometric and mechanical data of the wall segment in question, the stresses to which it is subjected, the limit drift of the element and the drift in steps.
There is also a graphic area in which the resistance domains of the element are traced and, by means of two circular indicators, the stress state of the element.
It is also possible to start a simulation by varying the parameters of the element (thickness, material, insertion of a reinforcement etc ...) thus evaluating how the behavior of the element varies according to the changes made.
Finally, it is possible to create a report to be included in relation with formulations, graphs and results illustrated step-by-step.

Kinematic analysis: maintenance of mechanisms with mesh regeneration

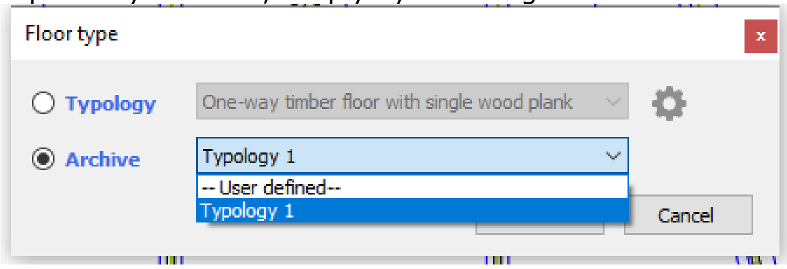
In previous versions, the remaking of the model mesh forced the elimination of any local mechanisms previously defined.

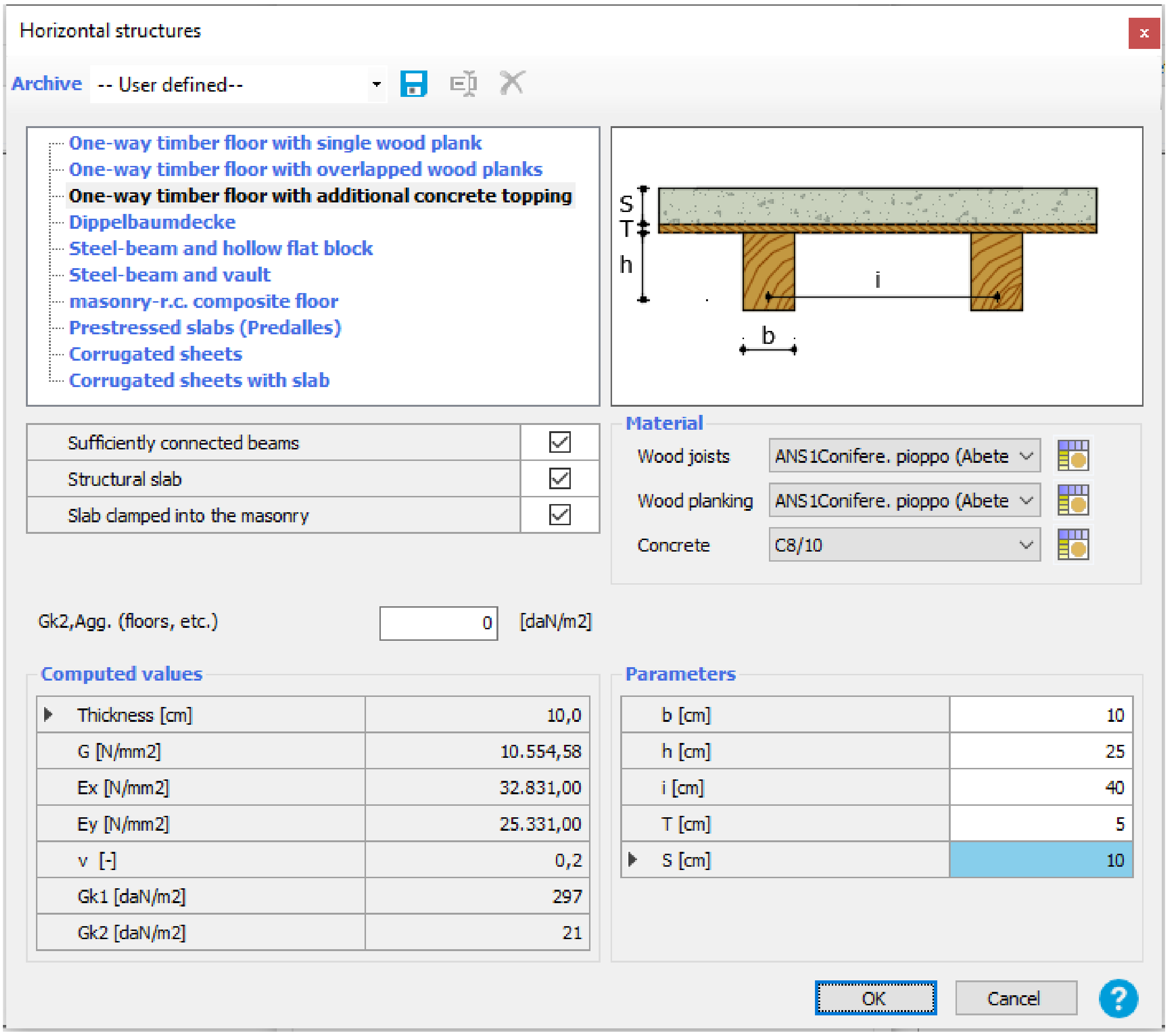
With the addition of this novelty, it will be possible to keep, even after the model mesh has been remade, all the mechanisms whose definition is aligned with the data defined in the structure.

For this reason, a series of automatic checks have been included with the aim of helping the user in assessing the compliance of the mechanism with the model and therefore in the choice of keeping the defined mechanisms or not.

Upgrade floors

Very often in the verification of existing structures, the floors have variated geometries based on the types present. The input of some types requires a large amount of data. Usually, in a structure a certain type is almost never present a single time, therefore this new feature allows you to manage "Archives", consisting of families of floors that can be repeatedly inserted, simply by selecting the name from the appropriate drop-down list.





Furthermore, the calculation report was integrated with a descriptive part of the horizontals based on the data entered, in order to make the reporting complete and thorough, not only from the point of view of mechanical performance, but also of geometries and materials.

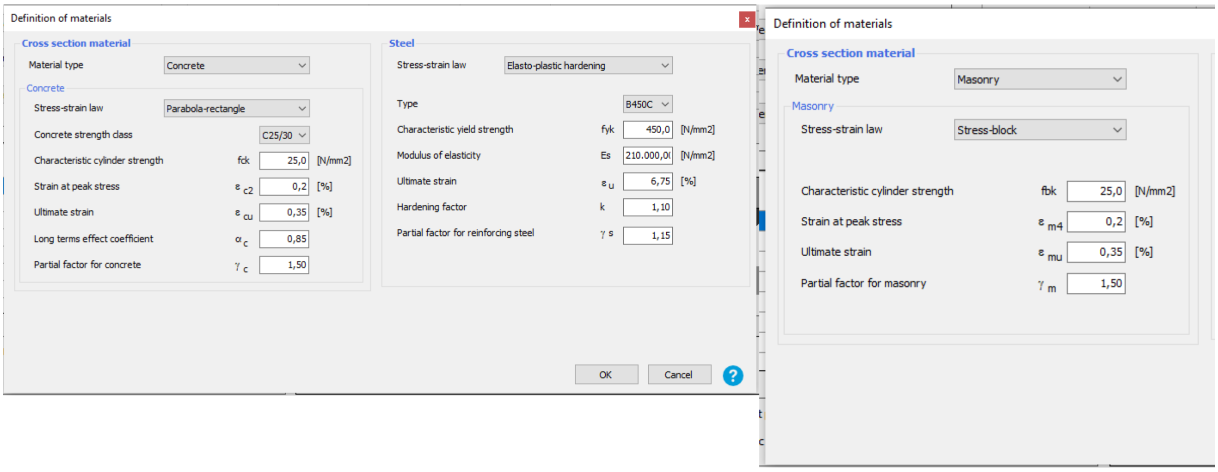
Name	Materials	Description
U1	Wood joists: ANS1Conifere. pioppo (Abete Nord 1)	One-way timber floor with single wood plank b [cm] = 10,0; h [cm] = 25,0; i [cm] = 40,0; T [cm] = 5,0
U2	Concrete: C8/10	masonry-r.c. composite floor b [cm] = 10,0; i [cm] = 50,0; h floor [cm] = 25; S [cm] = 5,0

Floor

No.	Archive	Elevation [cm]	Thickness [cm]	G [N/mm2]	Ex [N/mm2]	Ey [N/mm2]	Mass loading	Type
1	U1	300	4,0	10,00	18.750,00	0,00	Unidirectional	One-way timber floor with single wood plank
2	U2	300	5,0	10.554,58	45.595,80	25.331,00	Unidirectional	masonry-r.c. composite floor

Upgrade Flexional/Bending

The module allows the verification of bending, straight bending and deviated bending of sections in reinforced concrete and masonry.

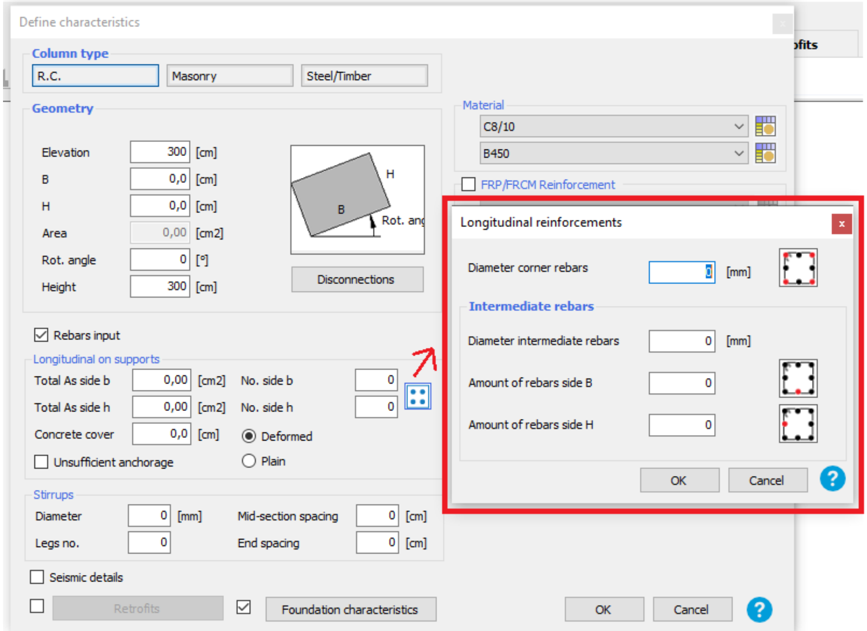


The following changes have been introduced within the module:

- The introduction of the masonry material.
- Update of the calculation algorithm that allows to take into account different constitutive laws for materials:
  - Reinforced concrete: Parabola-rectangle, Bi-linear, stress-block.
  - Masonry: Bi-linear, stress-block.
  - Steel: Elastic-perfectly plastic, elasto-plastic hardening and elasto-brittle behavior.
- Ability to copy and paste the action table from Excel.

Optimization of longitudinal reinforcement input

In order to facilitate the interaction between the different environments, the data input of the reinforcement inside the column has been improved, allowing a more efficient flow of data from the global model to the calculation sheets relating to local verifications (and vice versa).

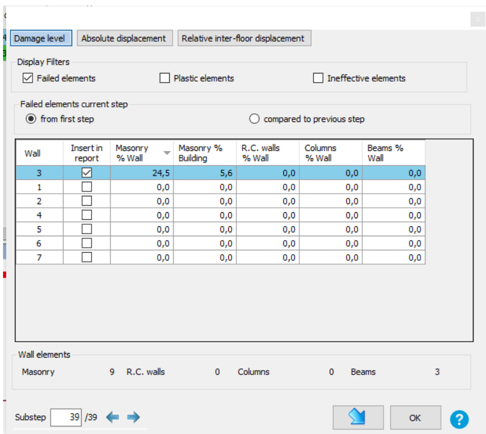


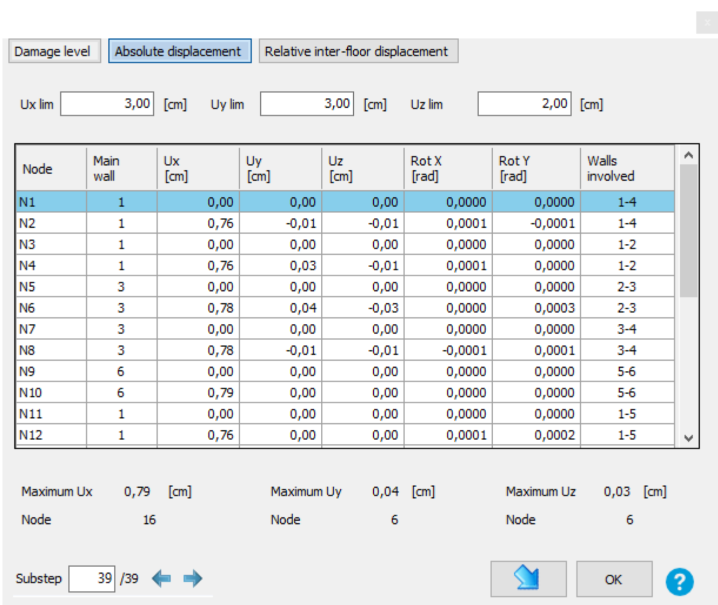
Upgrade model controls

In order to simplify all the modeling and interpretation operations of the results, a new window has been inserted within the results environment, allowing the elements subject to crises (damage and breakage) to be checked with precision and speed and check the levels of deformation of each part of the structure. This tool has a dual utility: checking the correctness of the model and understanding in which points of the structure it makes sense to act to plan an improvement or adaptation.

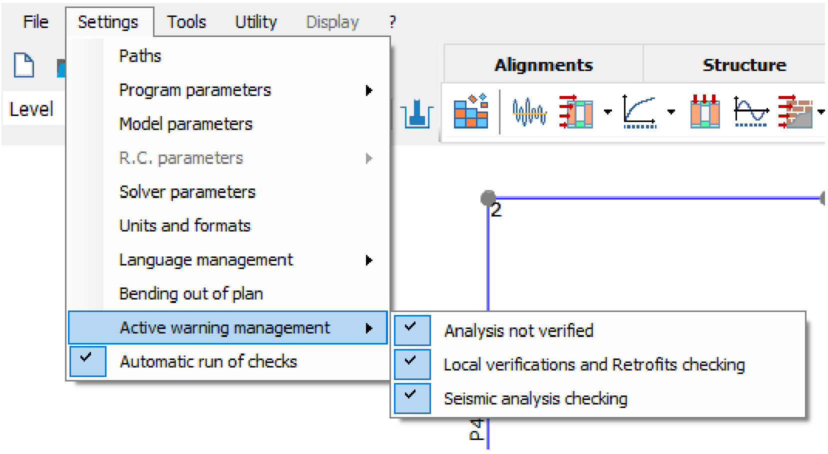
Damage level control

Displacement control (absolute and relative inter-floor)

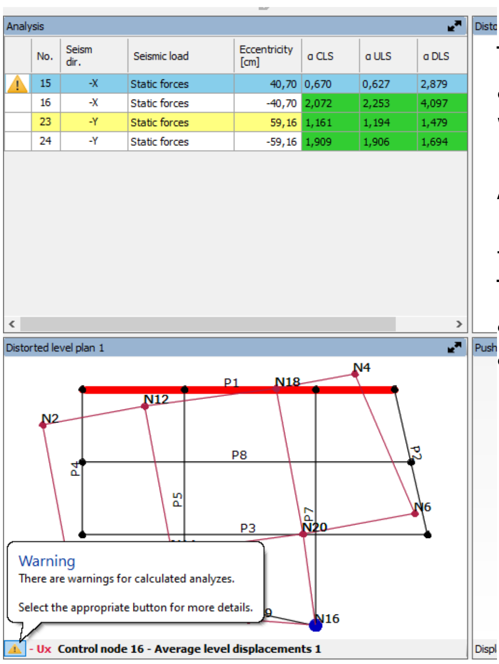




The addition of new controls is accompanied by a new management of warnings relating to the model in question.



The new warnings contain information regarding calculation criticalities (eg convergence problems) with suggestions regarding possible causes, "precision" and accuracy checks of the result, aimed at making appropriate corrections to the calculation, in order to obtain more reliable results, some indicators can provide "suggestions" aimed at improving the model to obtain a more reliable result.



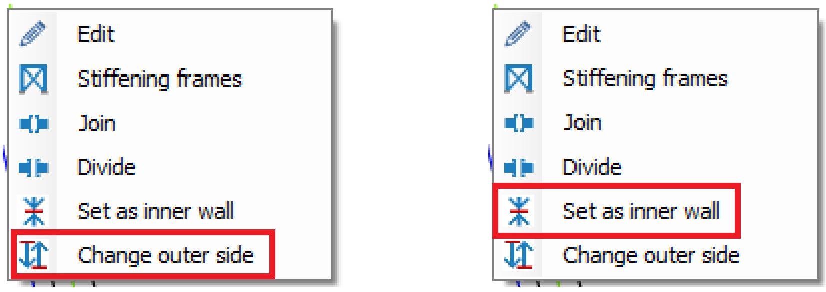
The appearance of the symbol ⚠ in the list of analysis carried out indicates the presence of warnings relating to the analysis.

At the same time, at the bottom left, a pop-up message will appear notifying the presence of alerts for which you can view the details.
This mode allows you to have an overview of all the analysis and understand any critical issues that act across the board and not just on the single analysis.

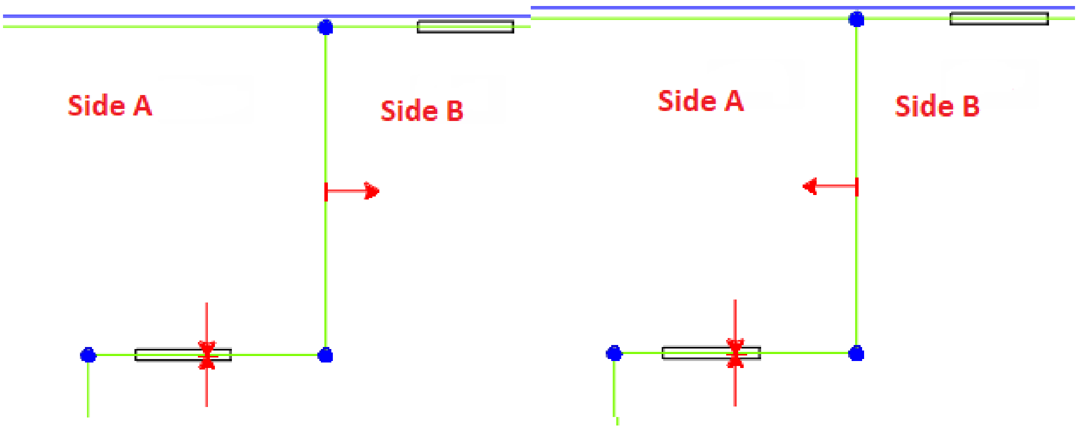
Automatic recognition of outer walls

In previous versions, the distinction between external and internal walls, for the application of a FRP / FRCM composite material reinforcement or the insertion of the wind load, had to be input manually on each element.
Thanks to the novelty introduced, the recognition of outer walls will be totally automatic,

leaving the user maximum freedom to change the property of the element manually in order to automatically update all the attributes dependent on it.
In order to change the exposure of the element manually, it will be necessary to activate the panel exposure filter (Display Options> Symbols> Panel Exposure) and then select the options on the context menu.



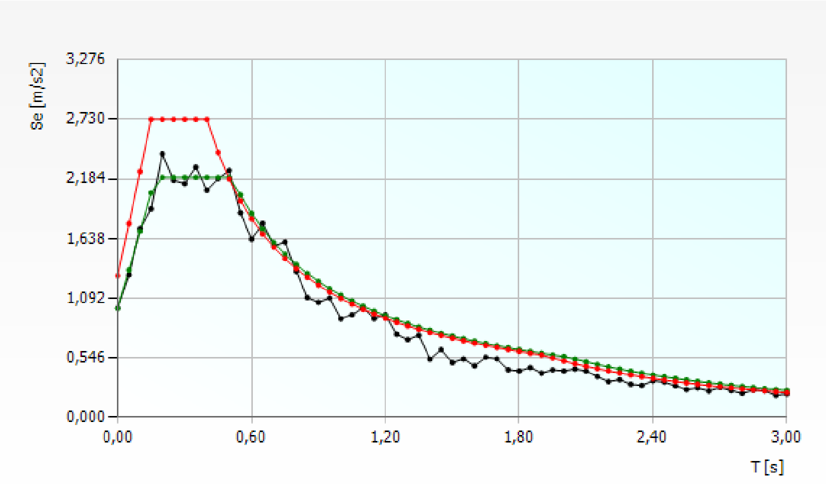
This new functionality is supported by the introduction of a new symbology in the structure environment, in order to quickly identify the exposure of each element.



Local seismic response

Based on what is reported in Appendix 1- Annex 1 of Ordinance no. 55 of 24 April 2018, a procedure for regularizing a response spectrum produced by a numerical simulation is indicated, transforming it into a standard spectrum.

Thanks to the new module "Local Seismic Response" it will therefore be possible to transform the response spectrum, resulting from the numerical simulations in the MS3 studies, into a spectrum with a standard form, consisting of a segment with linear increasing acceleration, a segment with constant acceleration, a segment in which the acceleration decreases with 1 / T and, therefore, at a constant speed.



At the end of the procedure, all the parameters for entering the elastic spectrum (ag, TB, TC, TD, F0, S) will also be available:

Spectrum Parameters Local Seismic Response

Spectrum	ag [m/s²]	F0 [-]	S [-]	TB [s]	TC [s]	TD [s]
from calculation LSR	1,00	2,20	1,00	0,17	0,51	2,01
from reticulum NTC						

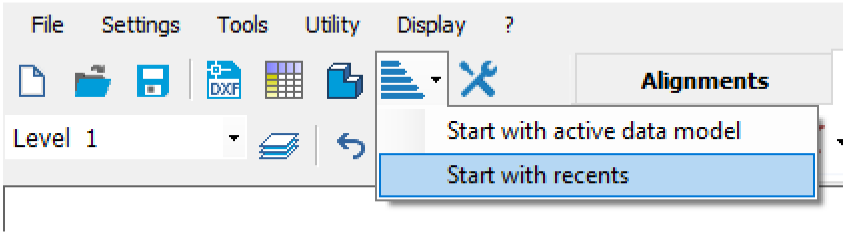
This will allow to obtain a correct identification of the vulnerability parameters, both in terms of acceleration and return period, otherwise impossible through a single input point by point.

Add or delete level (bottom)

Working in historical contexts of sets of buildings, immediately understanding the overall conformation of the structural unit is not trivial and, perhaps, it is necessary to intervene later by acting on the lower levels.
Therefore, to make it easier to create the model within the modeling environment, the ability to modify the levels of the structure has been added by adding, deleting and / or duplicating en the lower levels.

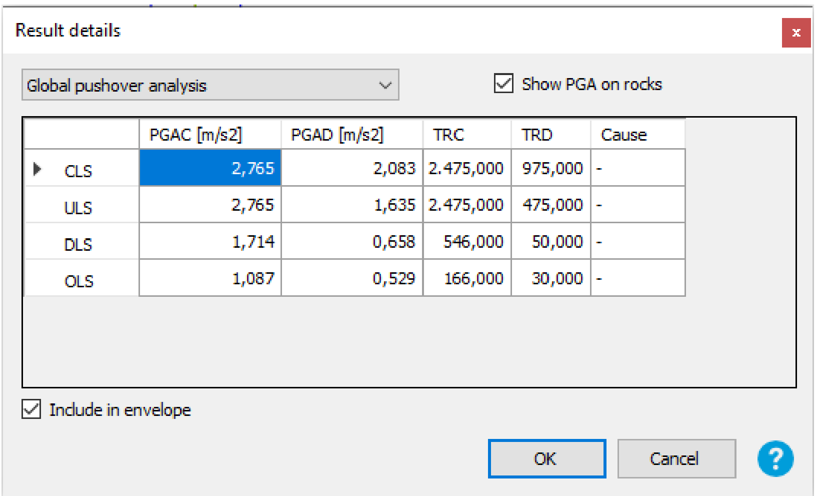
SismoTest - Export of analysis parameters

SismoTest is the 3Muri module dedicated to the seismic classification of buildings, according to the D.M. n. 58 of 28/2/2017 and subsequent additions. This module puts into practice what is required by the Guidelines, making them easily applicable, providing suggestions in the evaluation phases, producing the required documents.



Given the importance of interventions aimed at reducing the possibility of triggering local mechanisms, such as, for example, the insertion of chains and tie rods against the overturning of the walls in masonry buildings, in order to facilitate the designer in producing the necessary documentation , the possibility of automatically importing the results of the kinematics analysis into the Sismotest module will be added to the current functions.

It will also be possible to automatically import the results of any single wall analysis and / or out-of-plane bending analysis.



2.1 Version 13.5.

A renovated solver

Each calculation engine constitutes a "container of knowledge" transformed into mathematics and is made up of two main components:

- Technical scientific knowledge
- Regulatory knowledge

Normative skills alone are not sufficient to perform a calculation since normative texts contain information too limited to be considered completely thorough.

The information contained in the texts of current regulations therefore constitute the guide to good planning and, at the same time, the minimum requirements that must be met. To follow these directives, the scientific instruments that the world of research with its countless publications is able to provide, cannot be ignored.

The world of university research is in constant movement, in the constant search for procedures that can better represent reality through numerical modeling.

Software companies that work closely with a University have the potential to be able to collect a large number of case studies that arise from the direct experiences of us designers who use these technological tools to study real cases.

Tackling the practical problems that professionals have to overcome on a daily basis helps us to go further in scientific research, finding different methods to solve problems that the current Regulations cannot solve independently or look for solutions that show results that are more consistent with what was found on the site.

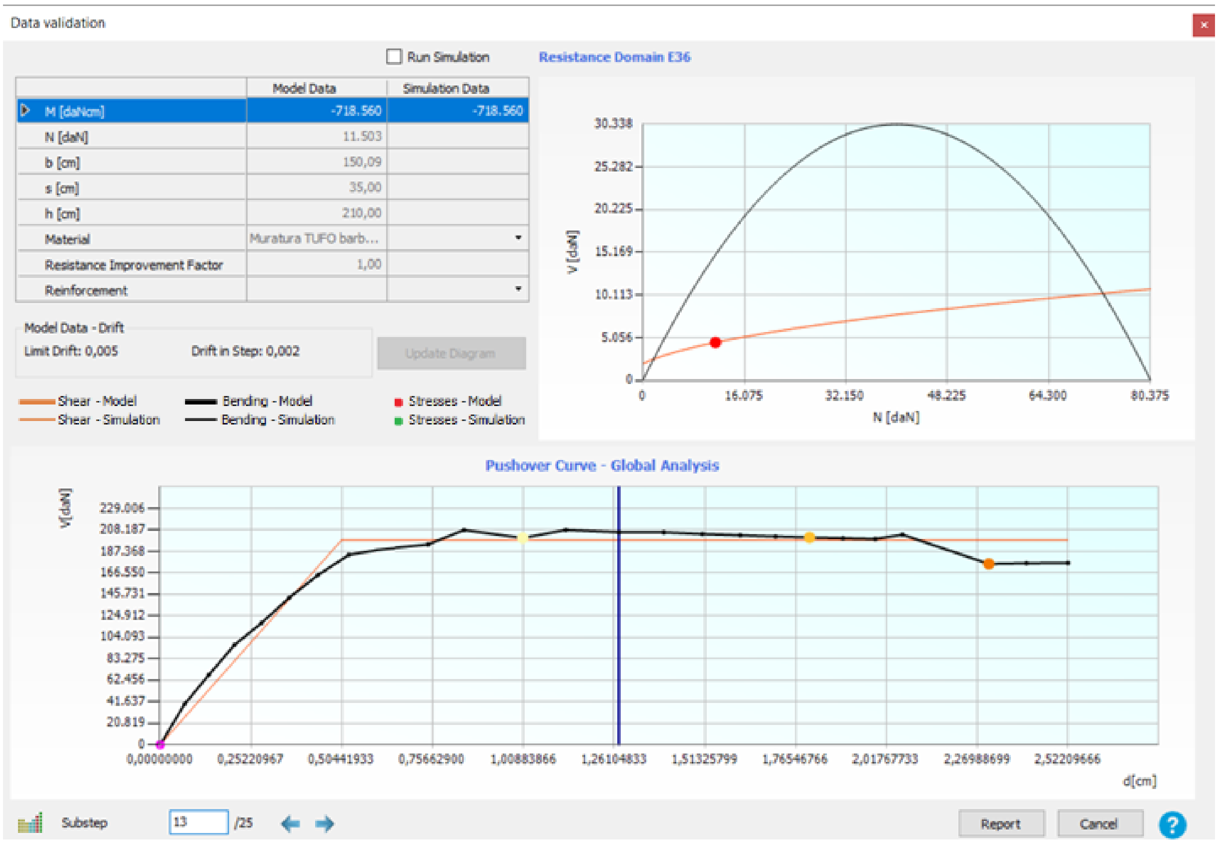
The masonry can be schematized by different constitutive bonds to be adopted based on the characteristics of the masonry texture. Regarding this, the legislation is clear: the choice of reinforcing with FRP, FRCM or reinforced masonry is not made explicit for all the bonds. For this reason, well-founded and sufficiently tested theoretical assumptions are necessary to be considered valid for the greatest number of practical cases possible. This is just one of the countless examples in which scientific assumptions allow us to perform calculations that are aligned to the reality being examined, without which this would not be possible.

Fortunately, scientific research is constantly evolving, and with this also the software on which it depends. We should not therefore be surprised if a calculation with a new engine provides slightly different results from the previous one, but be aware that this calculation is different because as research advances, a model can represent reality more accurately.

Validation Tools

As specified by the current legislation, the designer is required to carry out manual operations to validate the calculation carried out using the calculation software and the results provided by it. This operation, usually carried out with the aid of simple spreadsheets created by the user, can often be onerous, cumbersome and demanding, because of the degree of complexity reached nowadays by structural calculation software.

This new function is intended to provide the user with a powerful validation tool dedicated to the drafting of a report containing the formulations, graphs and results illustrated step-by-step, helping the professional to overcome this "annoying", but essential burden of validation of the calculation.



The validation window provides the geometric and mechanical data of the wall segment in question, the stresses to which it is subjected, the limit drift of the element and the drift in steps.
There is also a graphic area in which the resistance domains of the element are traced and, by means of two circular indicators, the stress state of the element.
It is also possible to start a simulation by varying the parameters of the element (thickness, material, insertion of a reinforcement etc ...) thus evaluating how the behavior of the element varies according to the changes made.
Finally, it is possible to create a report to be included in relation with formulations, graphs and results illustrated step-by-step.

Kinematic analysis: maintenance of mechanisms with mesh regeneration

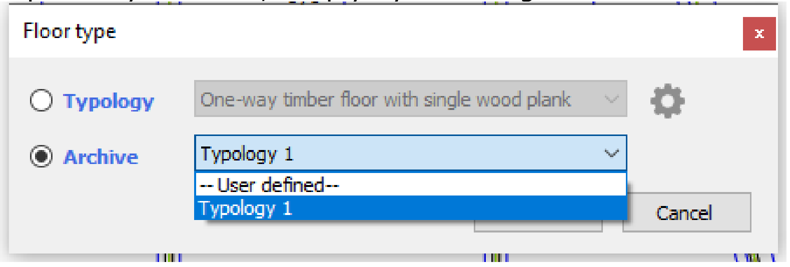
In previous versions, the remaking of the model mesh forced the elimination of any local mechanisms previously defined.

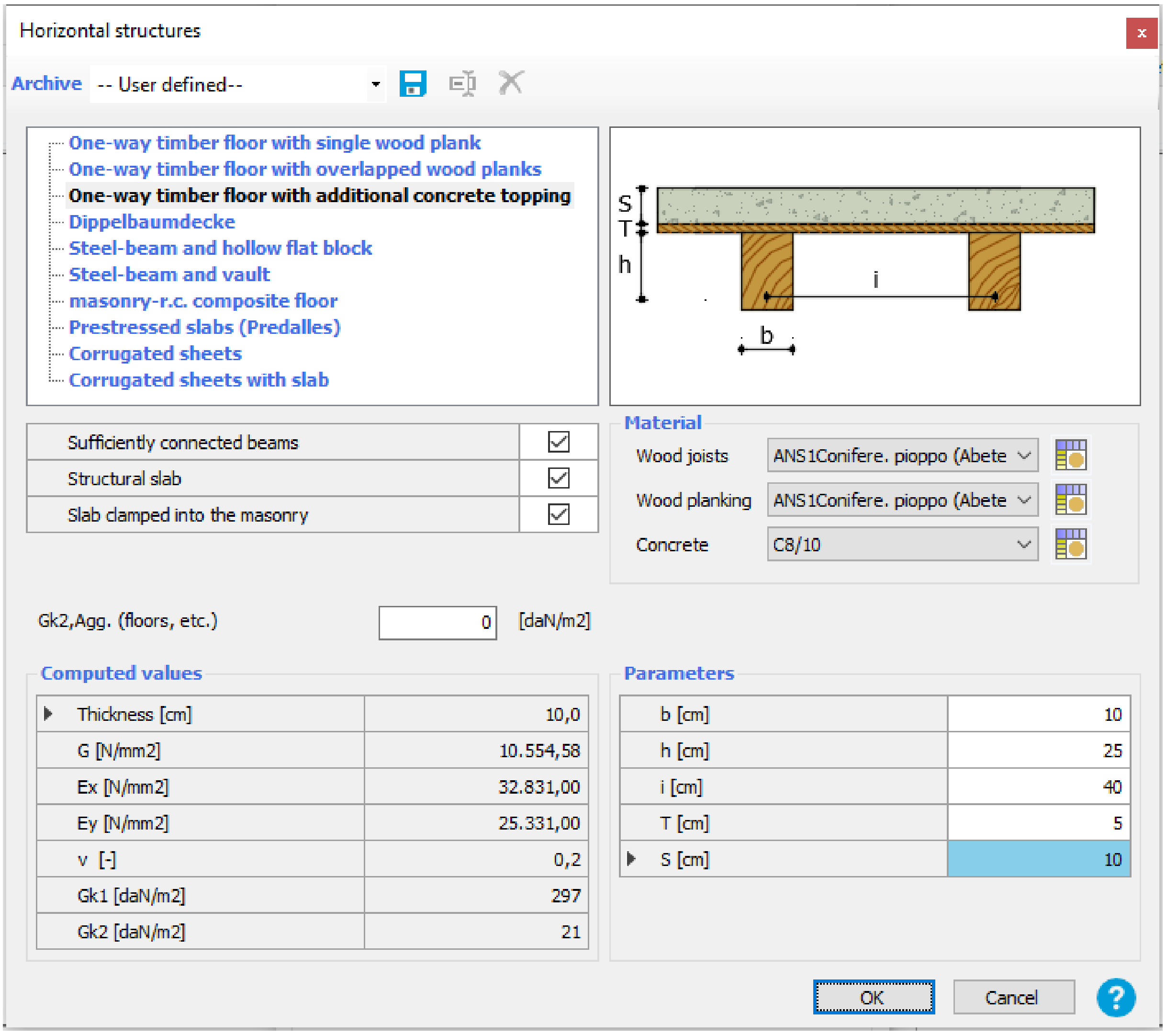
With the addition of this novelty, it will be possible to keep, even after the model mesh has been remade, all the mechanisms whose definition is aligned with the data defined in the structure.

For this reason, a series of automatic checks have been included with the aim of helping the user in assessing the compliance of the mechanism with the model and therefore in the choice of keeping the defined mechanisms or not.

Upgrade floors

Very often in the verification of existing structures, the floors have viariated geometries based on the types present. The input of some types requires a large amount of data. Usually, in a structure a certain type is almost never present a single time, therefore this new feature allows you to manage "Archives", consisting of families of floors that can be repeatedly inserted, simply by selecting the name from the appropriate drop-down list.





Furthermore, the calculation report was integrated with a descriptive part of the horizontals based on the data entered, in order to make the reporting complete and thorough, not only from the point of view of mechanical performance, but also of geometries and materials.

Name	Materials	Description
U1	Wood joists: ANS1Conifere. pioppo (Abete Nord 1)	One-way timber floor with single wood plank b [cm] = 10,0; h [cm] = 25,0; i [cm] = 40,0; T [cm] = 5,0
U2	Concrete: C8/10	masonry-r.c. composite floor b [cm] = 10,0; i [cm] = 50,0; h floor [cm] = 25; S [cm] = 5,0

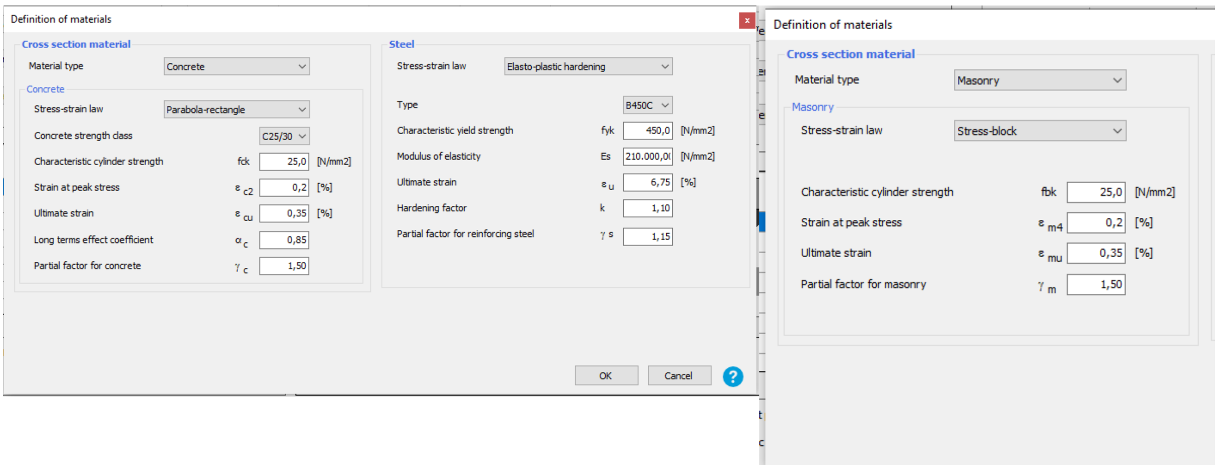
Floor

No.	Archive	Elevation [cm]	Thickness [cm]	G [N/mm2]	Ex [N/mm2]	Ey [N/mm2]	Mass loading	Type
1	U1	300	4,0	10,00	18.750,00	0,00	Unidirectional	One-way timber floor with single wood plank
2	U2	300	5,0	10.554,58	45.595,80	25.331,00	Unidirectional	masonry-r.c. composite floor

---

Upgrade Flexional/Bending

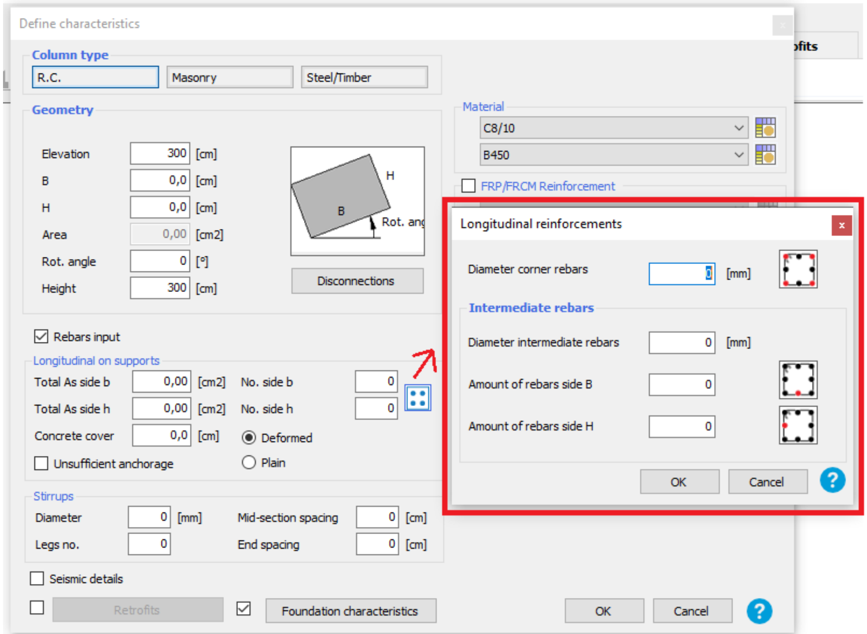
The module allows the verification of bending, straight bending and deviated bending of sections in reinforced concrete and masonry.



The following changes have been introduced within the module:
- The introduction of the masonry material.
- Update of the calculation algorithm that allows to take into account different constitutive laws for materials:
  - Reinforced concrete: Parabola-rectangle, Bi-linear, stress-block.
  - Masonry: Bi-linear, stress-block.
  - Steel: Elastic-perfectly plastic, elasto-plastic hardening and elasto-brittle behavior.
- Ability to copy and paste the action table from Excel.

Optimization of longitudinal reinforcement input

In order to facilitate the interaction between the different environments, the data input of the reinforcement inside the column has been improved, allowing a more efficient flow of data from the global model to the calculation sheets relating to local verifications (and vice versa).

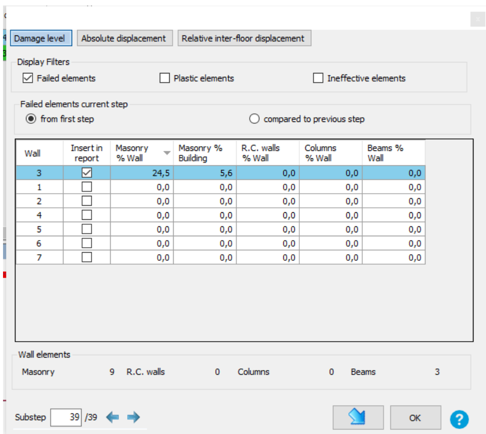


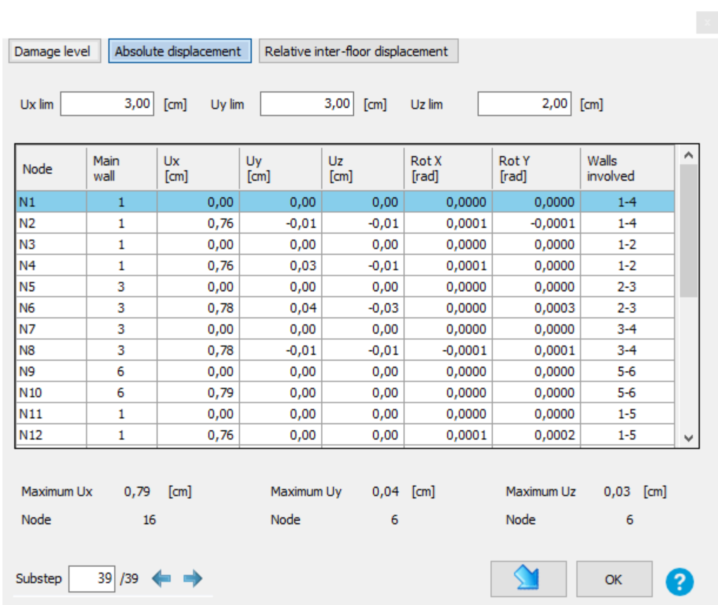
Upgrade model controls

In order to simplify all the modeling and interpretation operations of the results, a new window has been inserted within the results environment, allowing the elements subject to crises (damage and breakage) to be checked with precision and speed and check the levels of deformation of each part of the structure. This tool has a dual utility: checking the correctness of the model and understanding in which points of the structure it makes sense to act to plan an improvement or adaptation.

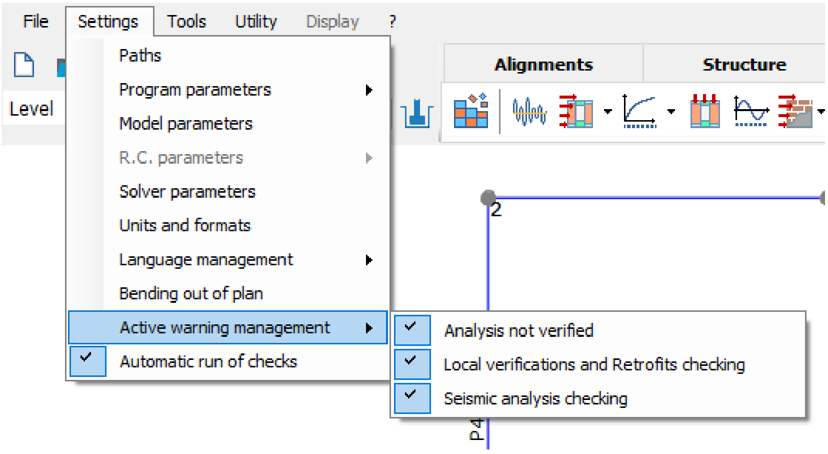
Damage level control

Displacement control (absolute and relative inter-floor)

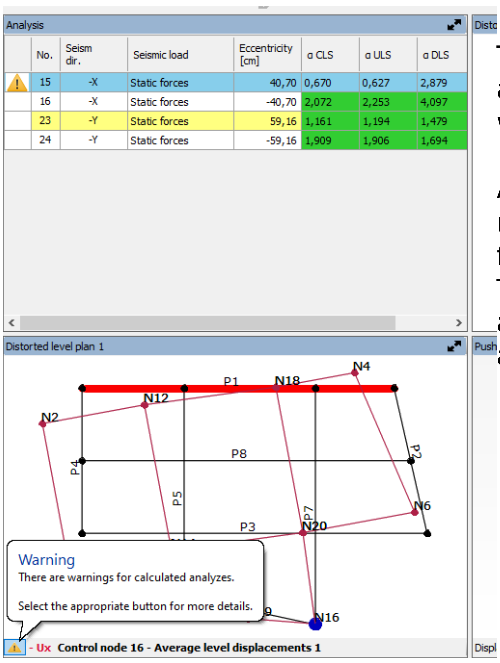




The addition of new controls is accompanied by a new management of warnings relating to the model in question.



The new warnings contain information regarding calculation criticalities (eg convergence problems) with suggestions regarding possible causes, "precision" and accuracy checks of the result, aimed at making appropriate corrections to the calculation, in order to obtain more reliable results, some indicators can provide "suggestions" aimed at improving the model to obtain a more reliable result.



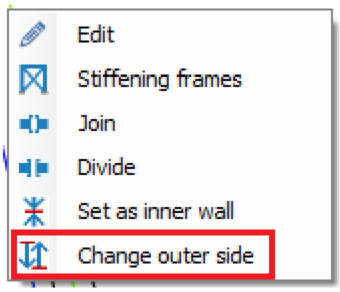
The appearance of the symbol  in the list of analysis carried out indicates the presence of warnings relating to the analysis.

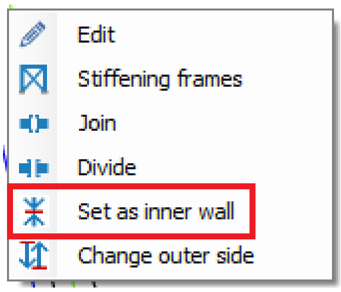
At the same time, at the bottom left, a pop-up message will appear notifying the presence of alerts for which you can view the details.
This mode allows you to have an overview of all the analysis and understand any critical issues that act across the board and not just on the single analysis.

Automatic recognition of outer walls

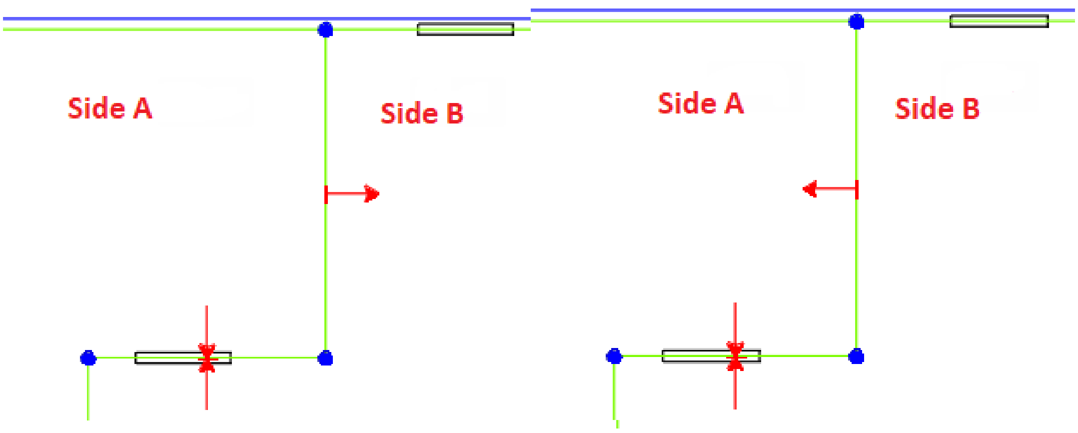
In previous versions, the distinction between external and internal walls, for the application of a FRP / FRCM composite material reinforcement or the insertion of the wind load, had to be input manually on each element.

Thanks to the novelty introduced, the recognition of outer walls will be totally automatic, leaving the user maximum freedom to change the property of the element manually in order to automatically update all the attributes dependent on it.
In order to change the exposure of the element manually, it will be necessary to activate the panel exposure filter (Display Options> Symbols> Panel Exposure) and then select the options on the context menu.





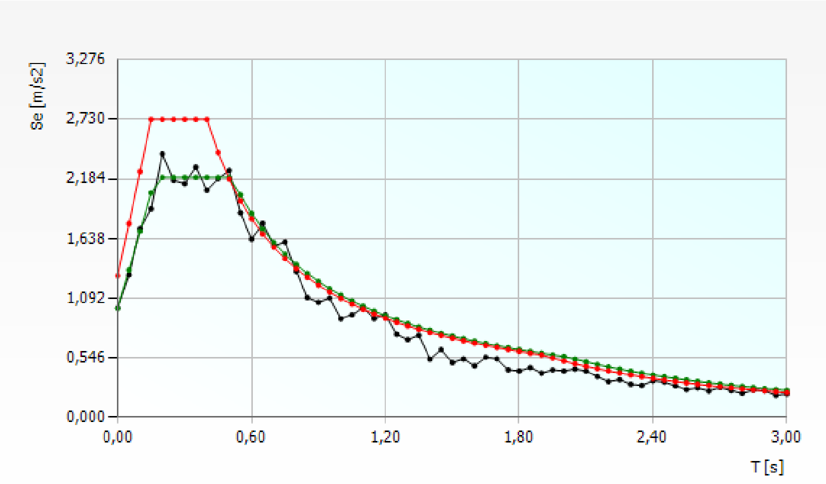
This new functionality is supported by the introduction of a new symbology in the structure environment, in order to quickly identify the exposure of each element.



Local seismic response

Based on what is reported in Appendix 1- Annex 1 of Ordinance no. 55 of 24 April 2018, a procedure for regularizing a response spectrum produced by a numerical simulation is indicated, transforming it into a standard spectrum.

Thanks to the new module "Local Seismic Response" it will therefore be possible to transform the response spectrum, resulting from the numerical simulations in the MS3 studies, into a spectrum with a standard form, consisting of a segment with linear increasing acceleration, a segment with constant acceleration, a segment in which the acceleration decreases with 1 / T and, therefore, at a constant speed.



At the end of the procedure, all the parameters for entering the elastic spectrum (ag, TB, TC, TD, F0, S) will also be available:

Spectrum Parameters Local Seismic Response

Spectrum	ag [m/s²]	F0 [-]	S [-]	TB [s]	TC [s]	TD [s]
from calculation LSR	1,00	2,20	1,00	0,17	0,51	2,01
from reticulum NTC						

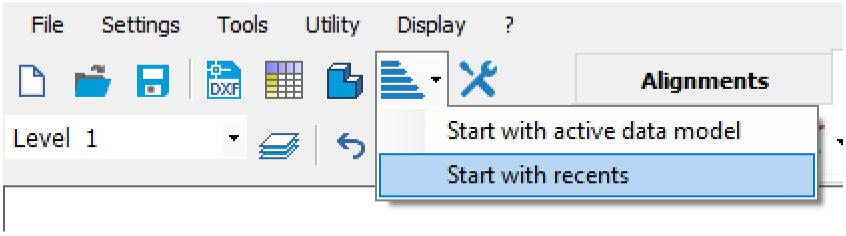
This will allow to obtain a correct identification of the vulnerability parameters, both in terms of acceleration and return period, otherwise impossible through a single input point by point.

Add or delete level (bottom)

Working in historical contexts of sets of buildings, immediately understanding the overall conformation of the structural unit is not trivial and, perhaps, it is necessary to intervene later by acting on the lower levels.
Therefore, to make it easier to create the model within the modeling environment, the ability to modify the levels of the structure has been added by adding, deleting and / or duplicating en the lower levels.

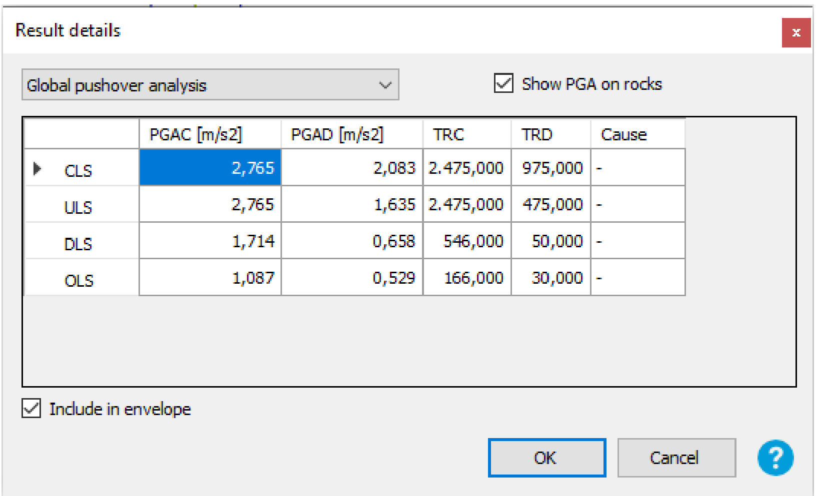
SismoTest - Export of analysis parameters

SismoTest is the 3Muri module dedicated to the seismic classification of buildings, according to the D.M. n. 58 of 28/2/2017 and subsequent additions. This module puts into practice what is required by the Guidelines, making them easily applicable, providing suggestions in the evaluation phases, producing the required documents.



Given the importance of interventions aimed at reducing the possibility of triggering local mechanisms, such as, for example, the insertion of chains and tie rods against the overturning of the walls in masonry buildings, in order to facilitate the designer in producing the necessary documentation , the possibility of automatically importing the results of the kinematics analysis into the Sismotest module will be added to the current functions.

It will also be possible to automatically import the results of any single wall analysis and / or out-of-plane bending analysis.



2.2 Version 13.2.0.10

Multilayer panels

A wall panel made by the presence of 2 panels and possibly a cavity, occasionally filled with a material of minor characteristics, cannot be identified from a regulatory point of view. This "problem" compels the designer to resort to scientific procedures of "proven validity".
The latter have been implemented within the program in order to facilitate the input of these elements, avoiding having to conduct homogenization calculations separately.

FRCM Discontinuous

In the CNR DT215 / 2018, in its Appendix 1, a simplified procedure is indicated to calculate a FRCM type reinforcement with discontinuous layout.
This procedure is now implemented within the program.

Vulnerability of TR < 30 years

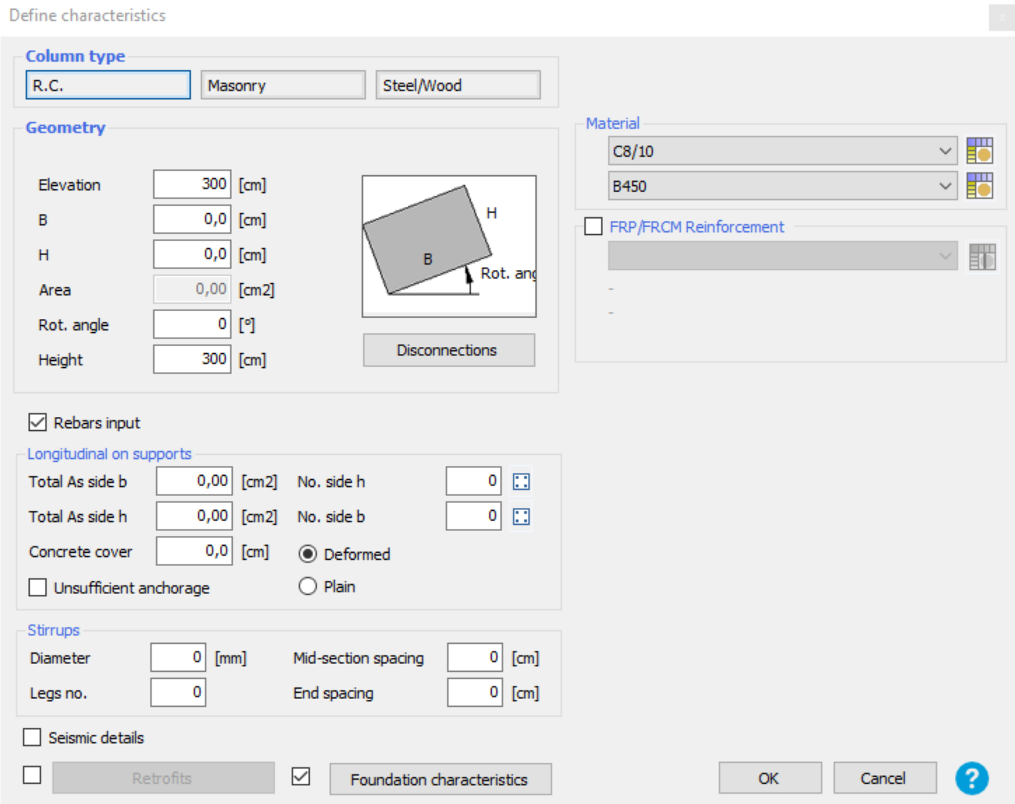
In professional practice, for buildings subjected to a major seismic action, it could occur that for the minimum value of the return period provided by the legislation (30 years) the return period of TRc capacity has not yet been found.
For this reason, in Annex A of Ministerial Decree 65 of 07-03-2017 "Guidelines for the classification of the seismic risk of buildings" a simplified procedure is indicated that allows the identification of the return period of less than 30 years.
This procedure is now available within the program for calculating the vulnerability on return periods.

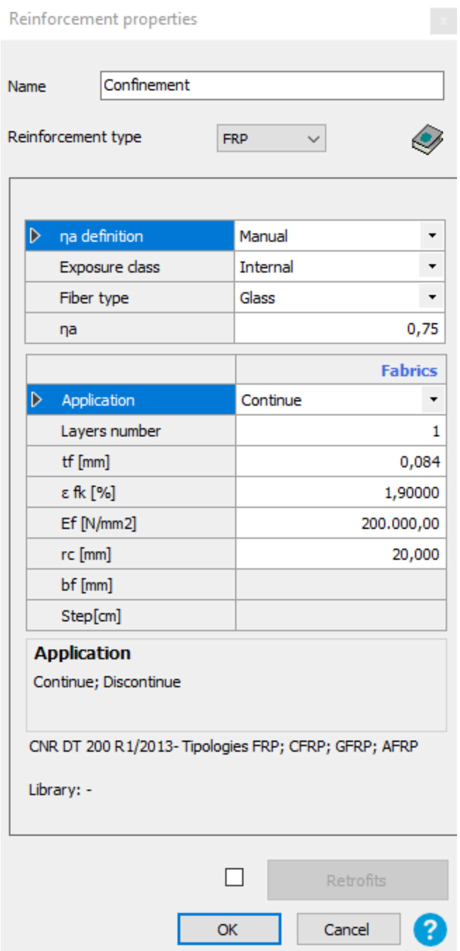
2.3 Version 13.2.0.8

FRP / FRCM - Reinforced concrete columns

It often happens in professional practice that columns are "weak links" of the structural complex, since they can be too slender and excessively loaded with respect to their size and mechanical performance.
In the text of the CNR-DT 200 R1 / 2013, applications of reinforcement using FRP are suggested as reinforcement interventions through the confinement of the columns.
The confinement can be made with fabrics and foils. The fabrics are applied on the contour as a continuous external wrapping (covering) or discontinuous (circling).
The wrapping of elements allows to increase the bearing capacity and the ductility of the column.

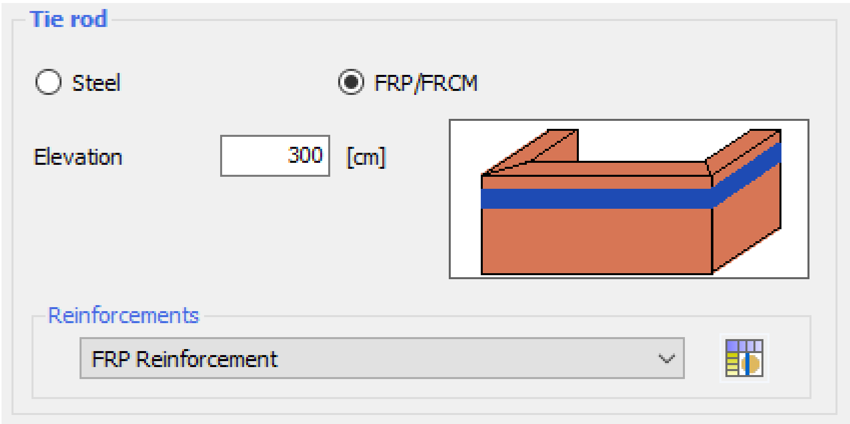
With this new functionality, it will be possible to define all the types of intervention mentioned. The definition of these reinforcements, as in the case of the other elements, can be made either by the user or by selection from manufacturers' libraries, with the advantage of being able to directly use the geometric and mechanical parameters provided by the manufacturers themselves.


Define characteristics
Column type
R.C.
Masonry
Steel/Wood
Geometry
Elevation 300 [cm]
B 0,0 [cm]
H 0,0 [cm]
Area 0,00 [cm2]
Rot. angle 0 [°]
Height 300 [cm]
H
B
Disconnections
Material
C8/10
B450
FRP/FRCM Reinforcement
Rebars input
Longitudinal on supports
Total As side b 0,00 [cm2]
Total As side h 0,00 [cm2]
Concrete cover 0,0 [cm]
No. side h 0
No. side b 0
Deformed
Plain
Unsufficient anchorage
Stirrups
Diameter 0 [mm]
Legs no. 0
Mid-section spacing 0 [cm]
End spacing 0 [cm]
Seismic details
Retrofits
Foundation characteristics
OK
Cancel


Reinforcement properties
Name Confinement
Reinforcement type FRP
ηa definition Manual
Exposure class Internal
Fiber type Glass
ηa 0,75
Fabrics
Application Continue
Layers number 1
tf [mm] 0,084
ε fk [%] 1,90000
Ef [N/mm2] 200.000,00
rc [mm] 20,000
bf [mm]
Step[cm]
Application
Continue; Discontinue
CNR DT 200 R1/2013- Tipologies FRP; CFRP; GFRP; AFRP
Library: -
Retrofits
OK
Cancel

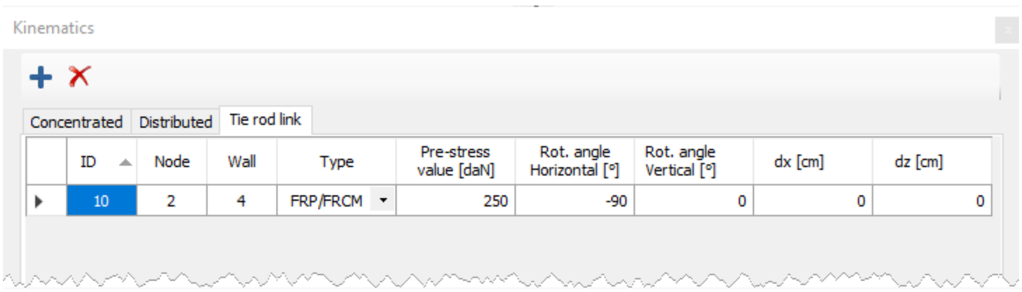
FRP / FRCM tie rods

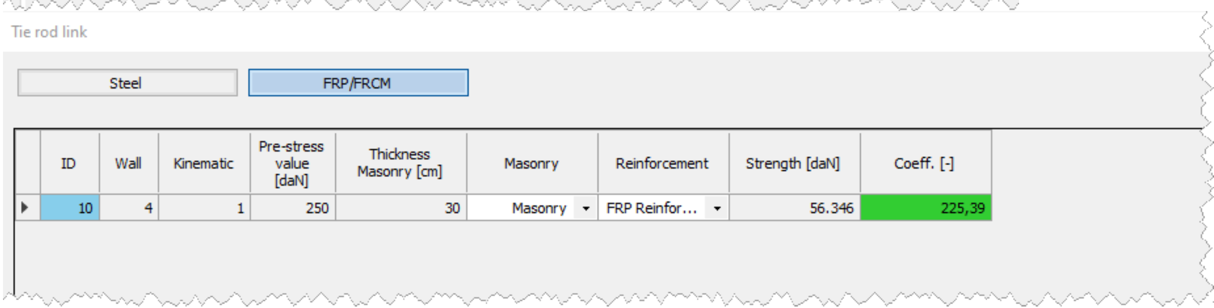
A big problem in the landscape of the existing design is very often related to the limited "box-like behavior" of the structures as a result of deformable floors weakly attached to the walls and the absence of curb elements.
A widely used intervention technique in these cases is the "floor hooping" which consists in the insertion of tie rods along the structural perimeter, which link the building and create a better overall functioning.
Until now, these tie rods have always been defined by steel bars, from now on it will be possible to decide to also use tie rods made of composite materials (FRP, FRCM) by inserting strips of fiber-reinforced fabric.



If a local analysis leads to a failure on the verification, the design practice very often leads us to provide the insertion of tie rods with the aim of preventing the activation of the mechanism.

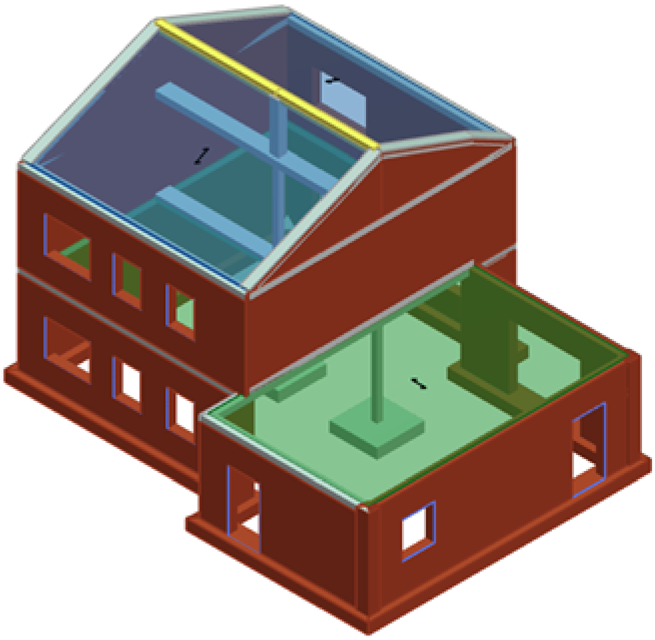
Thanks to this new feature, it will be possible to insert a "FRP / FRCM" type tie rod link within the kinematics in analysis and also check the inserted tie rod element.





3D Model Update

With the aim of improving the three-dimensional representation of the model, some graphic features have been included such as: connection of the edges of the elements and direction of warping of the floors.



A further increase in the functionality of the three-dimensional drawing of objects is represented by the possibility of being able to view the steel profiles with their sections and orientations within the model.
This functionality is available for both beams and pillars but also for reinforcing elements such as braces and for the encirclements of openings.
The dimensions of the sections shown will be obtained directly from the profile database within the program.

Simple sections	I ; [; L
Double sections	II ;][; JL

2.4 Version 13.2.0.3

3D Steel Profiles

The feature of three-dimensional drawing of objects has been improved, in particular the possibility of being able to view the steel profiles with their sections and orientations within the model.

This feature is available both for beams and columns but also for reinforcing elements such as bracing and hoops of openings.
The dimensions of the sections shown will be obtained directly from the profile database of the program.

Simple sections	I; [; L
Double sections	II ; [[; L L

Masonry column verification

How many times, during the design of a masonry building, we focus our attention on the verification of masonry panels, floors and vaults, neglecting the importance of masonry columns?
A wall column has good performance characteristics if its size is sufficient, while too small sections can create problems of slenderness and consequent instability, (since the masonry is not effective at traction) and excessive eccentricity can significantly penalize the outcome of any static verification.
With this module it is possible to carry out static checks of masonry columns, which are the main problem of buildings with extensive frames.

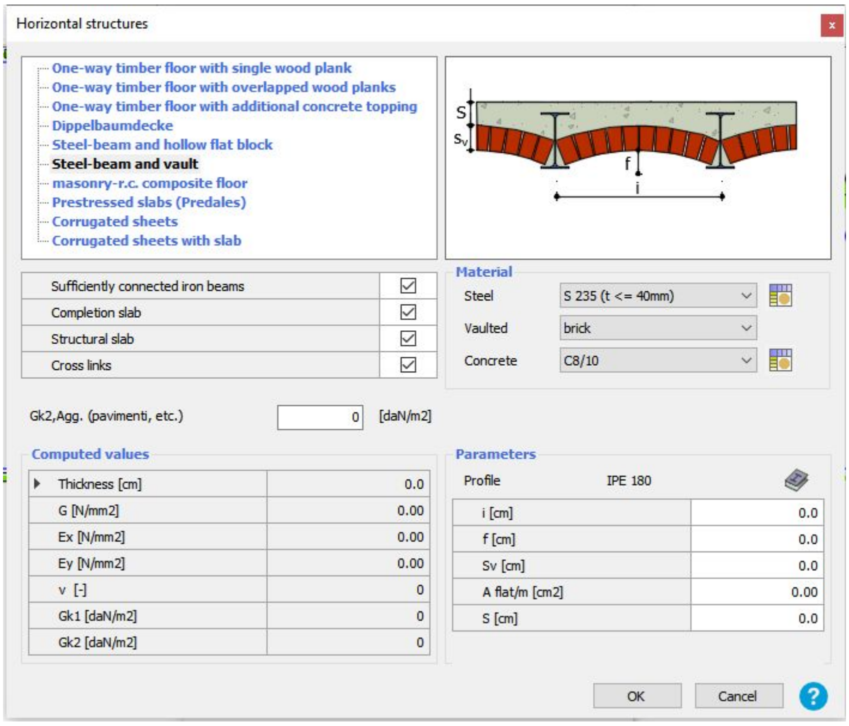
2.5 Version 13.2.0.0

New input slabs

The new input of the floors provides for both a renewal of the imputed environment and the inclusion of new types of horizontals.

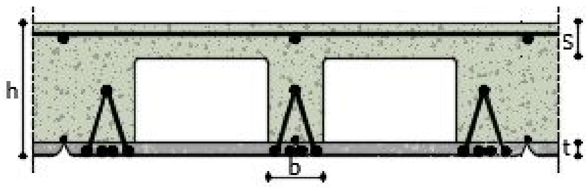
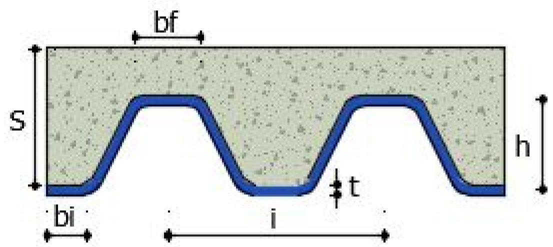
Renewal of the input

A renewed input environment of the slabs allows an easier definition of the horizontals.
Based on the selected type, the insertion of the mechanical characteristics of the materials that come into play is guided by direct references to the material libraries that allow to obtain both mechanical characteristics and weights.
For floors made with steel beams, a direct reference to an integrated profile facilitates insertion.
This new input method allows you to request less data from the user directly and at the same time to find more information.
In previous versions, the input of the floors calculated only the stiffnesses and not their weight, now the load is also calculated automatically including any carried weights such as blocks and floors / foundations.



New types

To the types already present, some new types are added.

Floor with Predalles slabs	
Floors in corrugated sheet metal ***(with and without completion insole)***	

Insertion of the SIA 269/8 regulation

The new Swiss standard SIA 269/8 is now available.

2.6 Version 12.6.1

Optimization of the alignment input commands

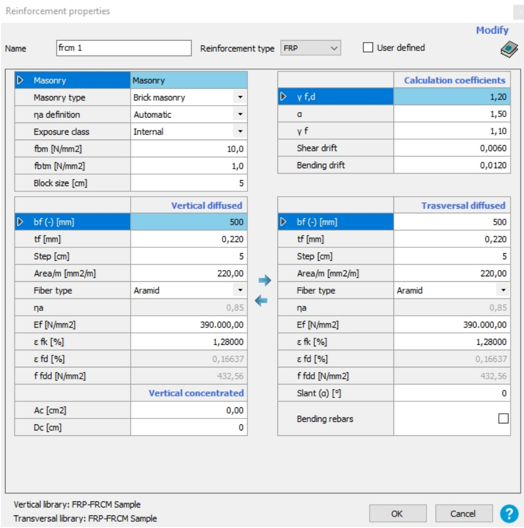
The alignment editing commands, such as extend/trim and move node, have evolved remarkably in this version.
Have been added automatic checks of the model consistency; graphic entities used to define three-dimensional objects where at each graphic modification must present a new configuration of the coherent model for the objects that weigh down to the individual alignments.
Next to the standard "extend/trim" command, an advanced one has been introduced respect to the previous one, which allows more detailed editing operations.

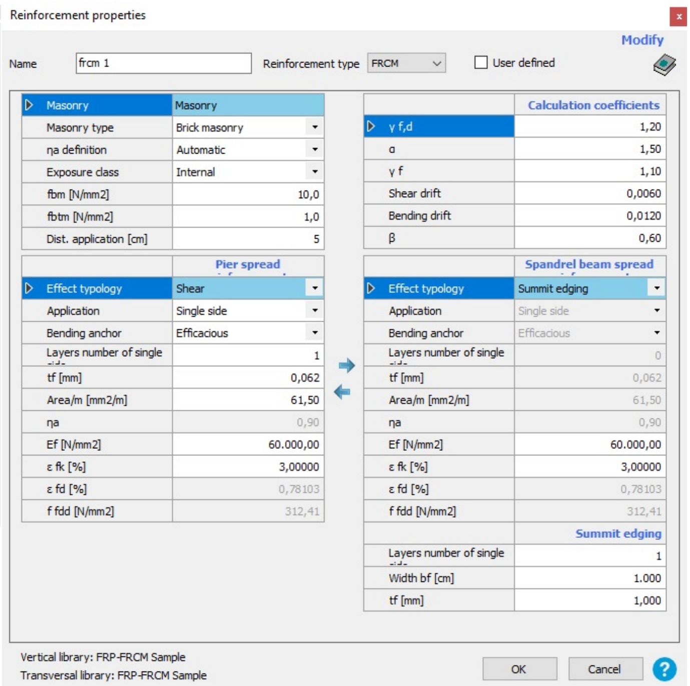
2.7 Version 12.6

FRCM for masonry walls

As part of the completion of the structural reinforcements applicable to the model, the module relating to masonry panels is extended, including FRP, reinforcements and reinforced masonry, introducing the possibility of applying FRCM reinforcements.
The treatment now implemented for FRP reinforcements follows the indications of CNR-DT 200 R1/2013.
FRCM reinforcements are accurately illustrated in CNR-DT 215/2018 - *"Instructions for the Design, Execution and Control of Static Consolidation Interventions through the use of Fiber-Reinforced Composites in Inorganic Matrix "*.
To the now classic fiber-reinforced composites FRP (Fiber Reinforced Polymer), made with long glass fibers, carbon or immersed aramid in polymeric matrices (such as epoxy resins), in the structural rehabilitation interventions, the FRCM fiber-reinforced composites flank more and more frequently nowdays.
The inorganic matrix has numerous advantages over the organic one of the FRP, especially for applications to masonry structures, thanks to its greater affinity with this type of supports.
Given the increasing diffusion of these reinforcements, it becomes necessary having greater accuracy in the calculation through the indications given in the text of CNR-DT 215/2018.

FRP (CNR-DT 200 R1/2013) **FRCM (CNR-DT 215/2018)**





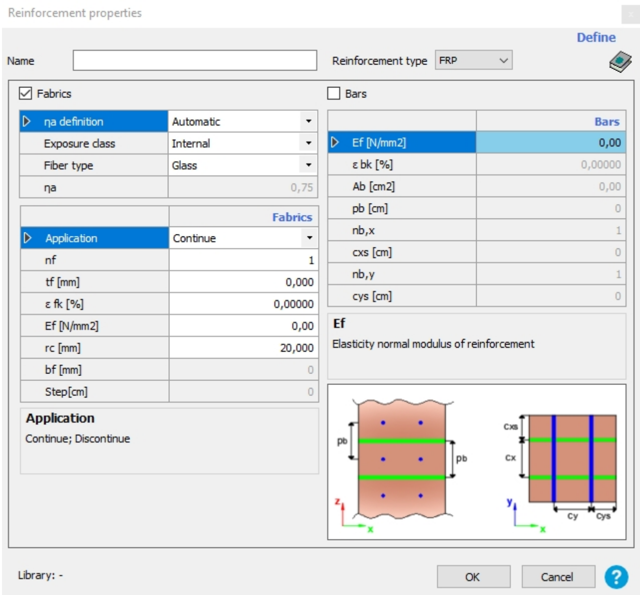
FRP and FRCM for masonry columns

In the professional practice it often happens that masonry colonnades are "weak links" of the structural complex, since they can be too slender and excessively loaded respect to their size and the mechanical performance of the masonry.

In the text of CNR-DT 200 R1/2013 and CNR-DT 215/2018, applications of reinforcements by FRP/FRCM are suggested as reinforcement interventions by confining the masonry columns.

The confinement can be accomplished with fabrics, lamina and bars. The fabrics are applied on the contour as a continuous (covering) or discontinuous (encirclements) external bandage; the bars are arranged inside the column to create opportunely spread reinforced seams. The bandage of elements allows to increase their bearing capacity.

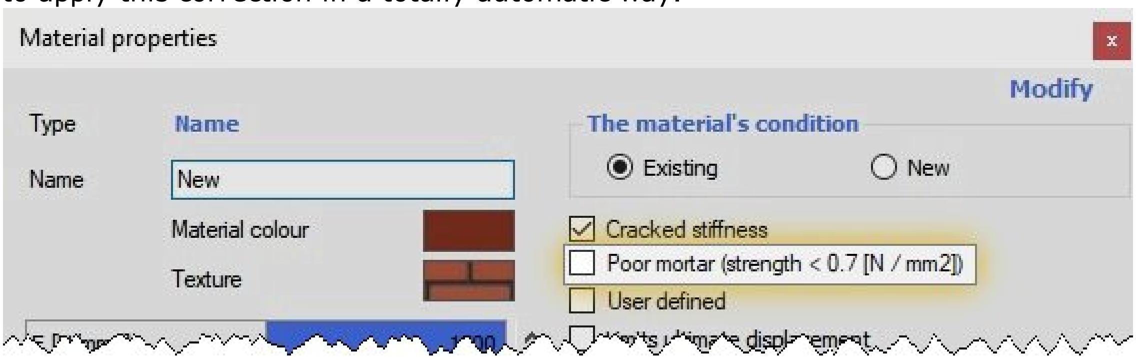
The input of these reinforcements is facilitated by a dedicated mask.



Masonry material - automation of reduced parameters

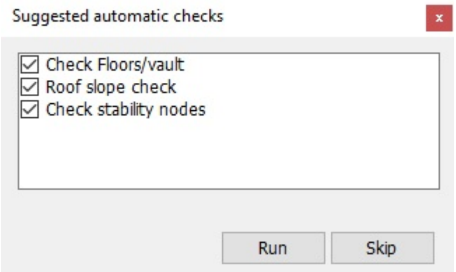
Further improvements have been implemented in the mechanical properties identification of the materials according to what is indicated in the *Application Circular No. 7 of 21 January 2019*. In this circular, regarding the behavior of the most recurrent wall typologies, a table shows indications on the possible values of the mechanical parameters, provided that precise conditions are respected (lime mortar of modest characteristics, absence of appeals etc ...).

For the sole purpose of seismic verification, in the case that the mortar has particularly poor characteristics (compressive average strength fm estimated to be less than 0.7 N/mm2), it is suggested to apply to the table values a reduction coefficient of 0.7 for the resistances and 0.8 for the elastic modules. At the moment, if the designer has to work with extremely poor mortars (<0.7N / mm2), he must proceed to manually reduce the performance of the material according to the two coefficients indicated above.
The checkbox "*extremely poor mortar*" in the mask of the material definition, will allow to apply this correction in a totally automatic way.



Update and integration of model checks

The currently present checks have been optimized and filtered according to the structure typology that is being modeled.
The automated procedures remind us, during the use, the most appropriate moment to conduct these checks without the designer having to remember to perform them in advance.

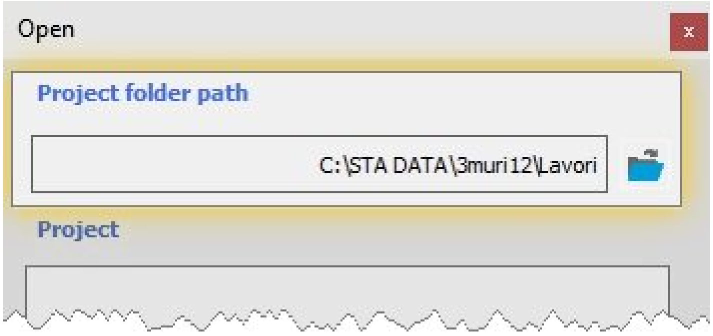


Rerun the generation of the mesh when modifying a model

The mesh represents the "mathematical" scheme of the objects introduced into the structure environment.
Whenever the user makes changes to the structure it is essential to regenerate the mesh, otherwise the calculations would be performed with the previous scheme to our modifications and therefore would not be updated.
From this version, the program performs some checks before proceeding with the calculation, aimed at verifying the mesh-structure coherence and if there is no correspondence the user would be notified.

Modify the path of the projects folder

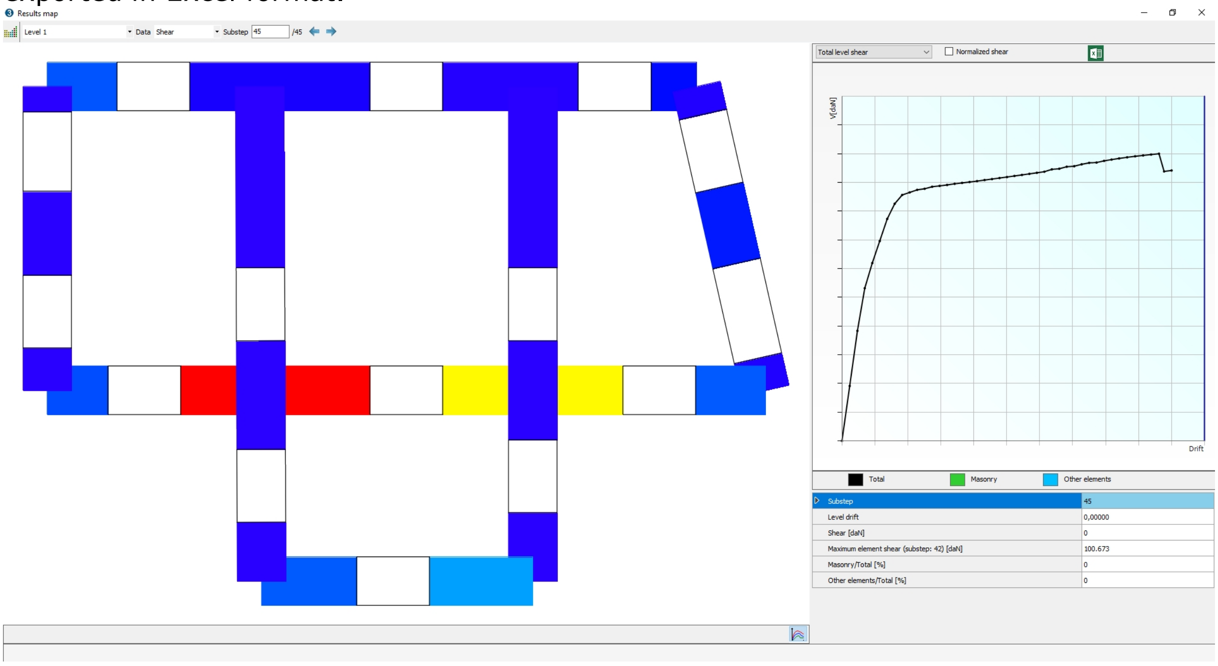
From this version it is possible to modify the path of the project folder directly within the "Open" window.



2.8 Version 12.5

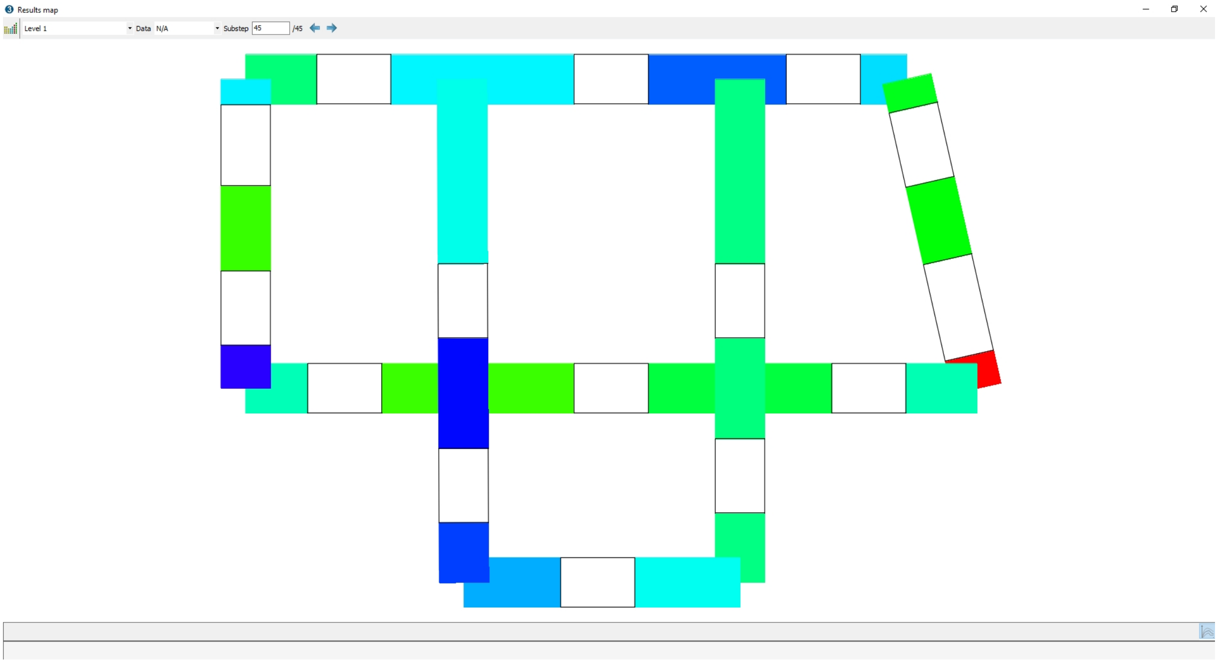
Colors map - Plan shear and trend diagram of the shear plan

For each level of the model a color map is shown, similar to the one used for the foundation module, in which the plan shear is represented. The different colors allow the user to view the values for the different pushover analyzes. On the right is the diagram with the trend of the shear plan for each step of the pushover analysis. The data can be exported in Excel format.



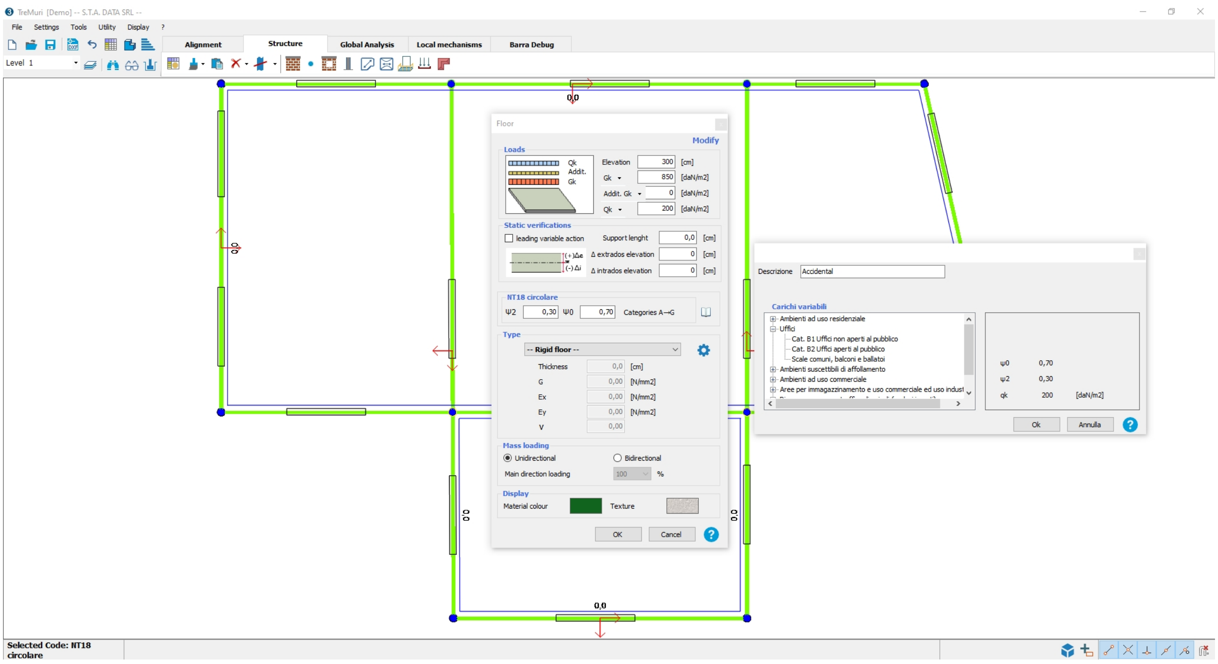
Colors map - Plan tensions

For each level of the model, a colors map is shown, similar to the one used for the foundation module, in which the plan tensions are represented. The different colors allow the user to view the stresses for the different pushover analysis.



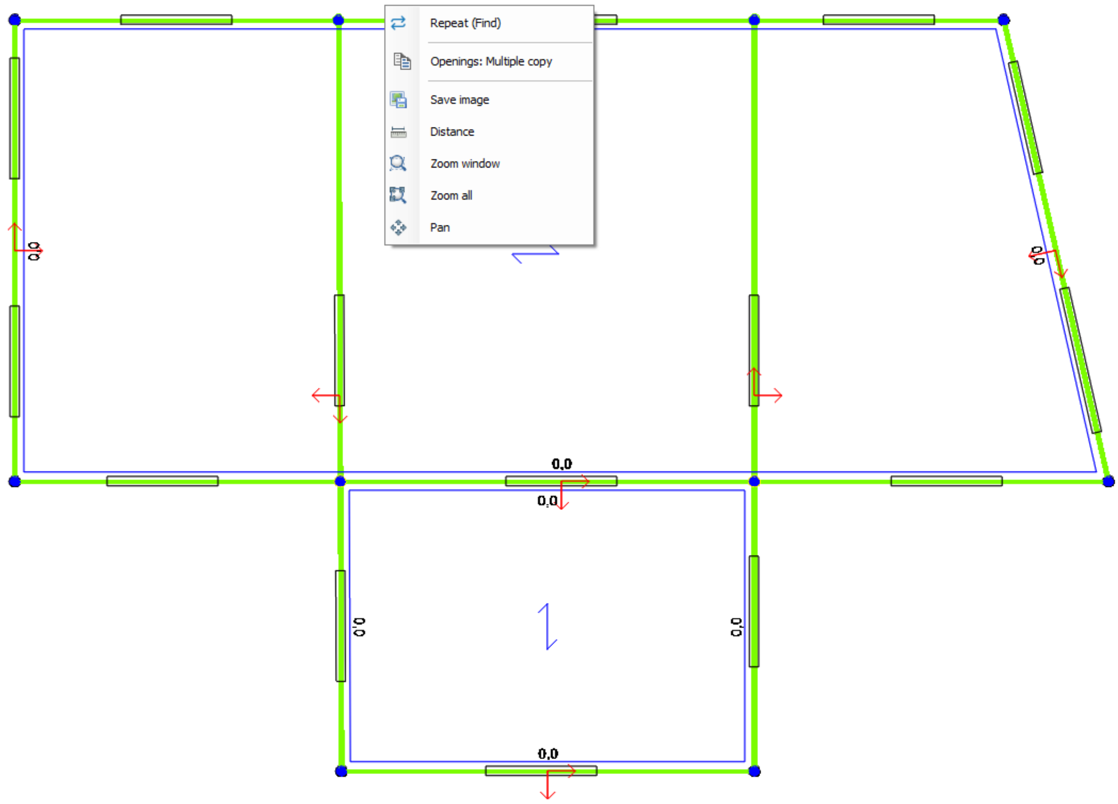
Calculation of the coefficient ζv

With the introduction of the most recent reference code review, some safety evaluations have been spelled out, among which the calculation of the coefficient ζe, for seismic analysis, and ζv for the static analysis. Regarding the coefficient ζv, the software allows the selection of the accidental loads that interest the calculation and in the results of the static analysis it give back the value of the calculated coefficient ζv.



Repeat last command

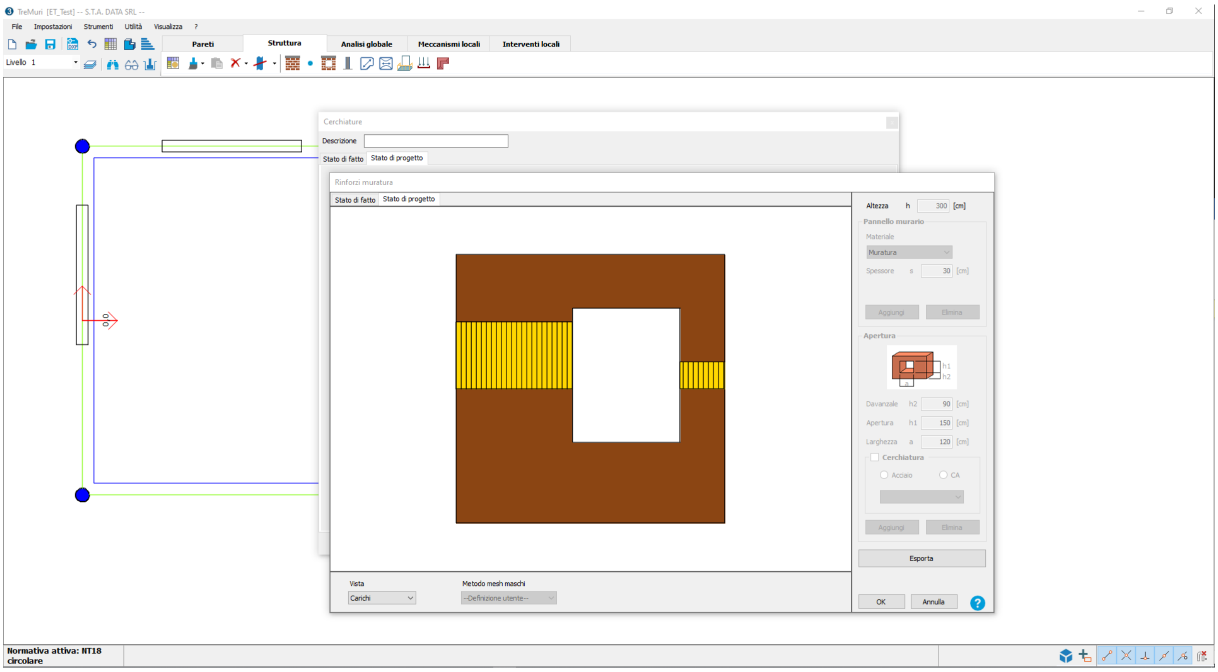
The possibility to recall the last command executed is added by clicking on the right mouse button to the usual graphic selection menu.



OPTIONAL MODULES

Encirclements

It allows the user to verify the encirclements of an opening in terms of local intervention. If the user want to previously dimensioned an encirclement, in order to consider it in the global model, the module communicates directly with the model from which it obtains the data already entered: the geometric configuration, the loads and the materials.



Steel lintels

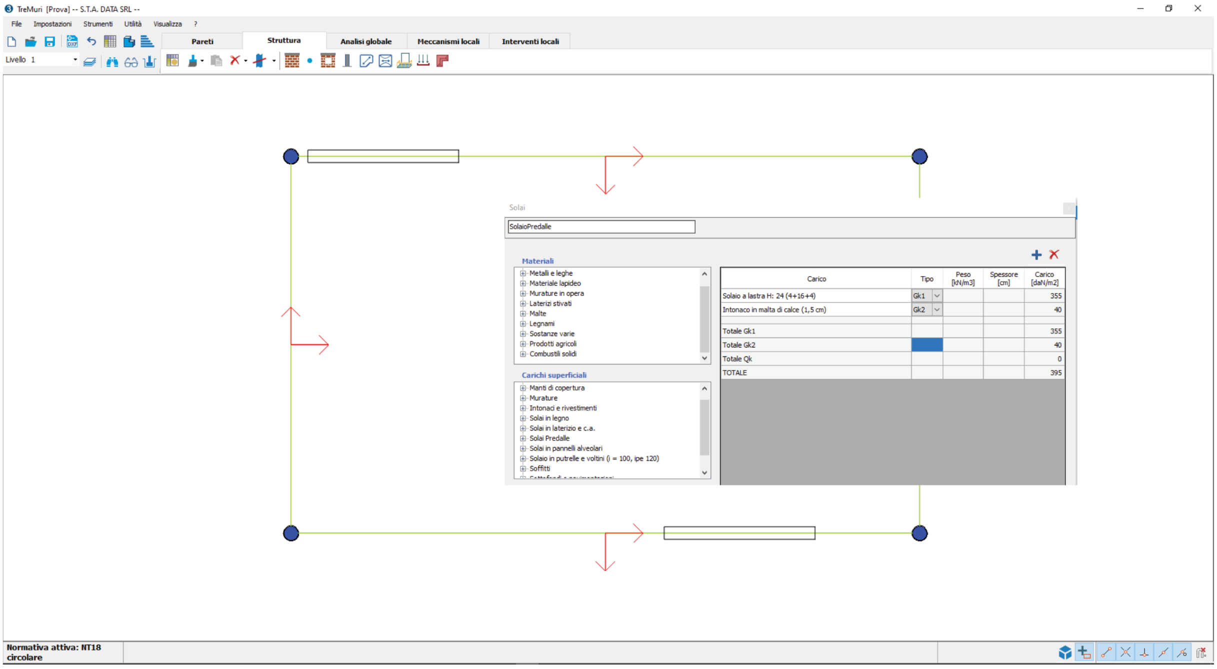
It allows the verification of a bending and deforming lintel against the stresses induced by the weight of the masonry and by an eventual of overlying slabs. The profile that is being verified can be selected from a library of the most common profiles in use or, alternatively, it is possible to insert manually its geometric and mechanical characteristics. The masonry and the eventual portion of the slab included in an imaginary equilateral triangle, built above the lintel, is considered as being charged on the lintel itself. The consequent moment stress and deformation is calculated considering the static scheme of simply supported beam.

Masonry Lintel

It allows the verification of a masonry lintel, structure frequently realized on the walls partitions of masonry buildings and with an arched behavior. The masonry and the eventual portion of the slab included in an imaginary equilateral triangle, built above the lintel, is considered as being charged on the lintel itself. The consequent moment stress is calculated considering the static scheme of simply supported beam.

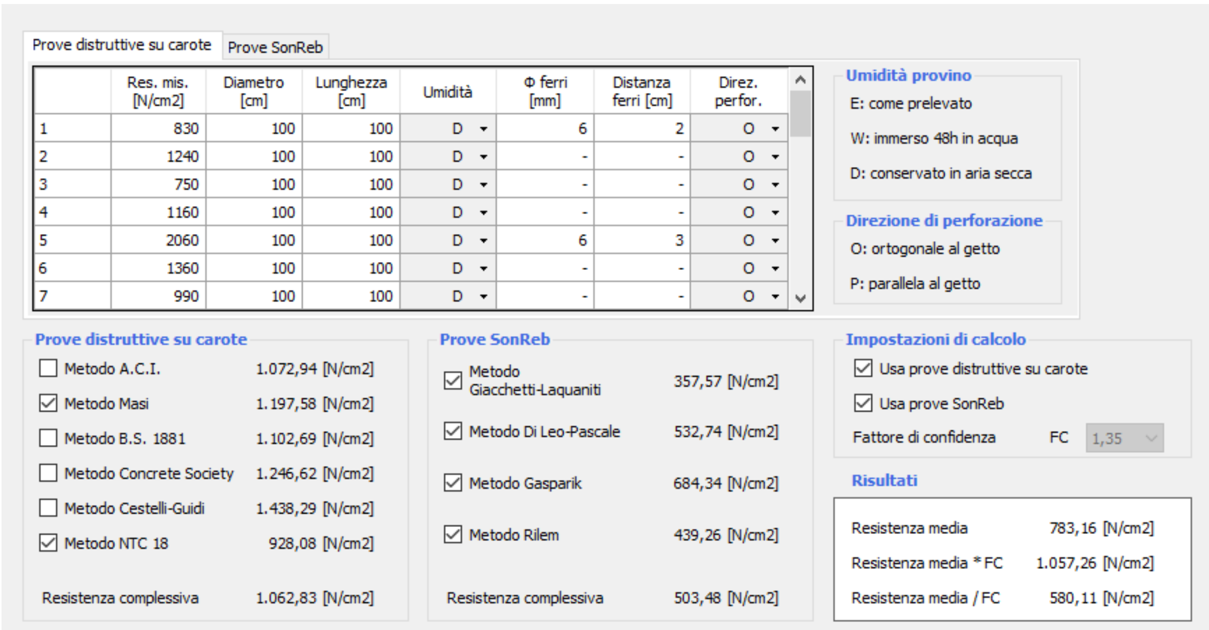
Load analysis

It allows to perform the calculation of the slabs loads, by going to set: materials, geometries and variable loads, each of them has its own library.



Concrete in situ

The results of the tests carried out in situ to assay the concrete strength require succeeding re-elaborations to obtain the strength values to be used in the calculations. This module allows the user to apply the different theories regarding the corrections of the rupture values obtained from tests on extracted cores and the elaboration of the SonReb test results.

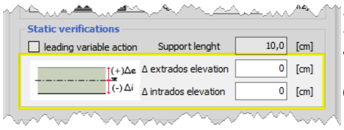


2.9 Version 12.2.1.9

Net Height

The net height that is attributed to each level is given by the difference in the elevations relative to the average plans of the slabs, height which, from the point of view of

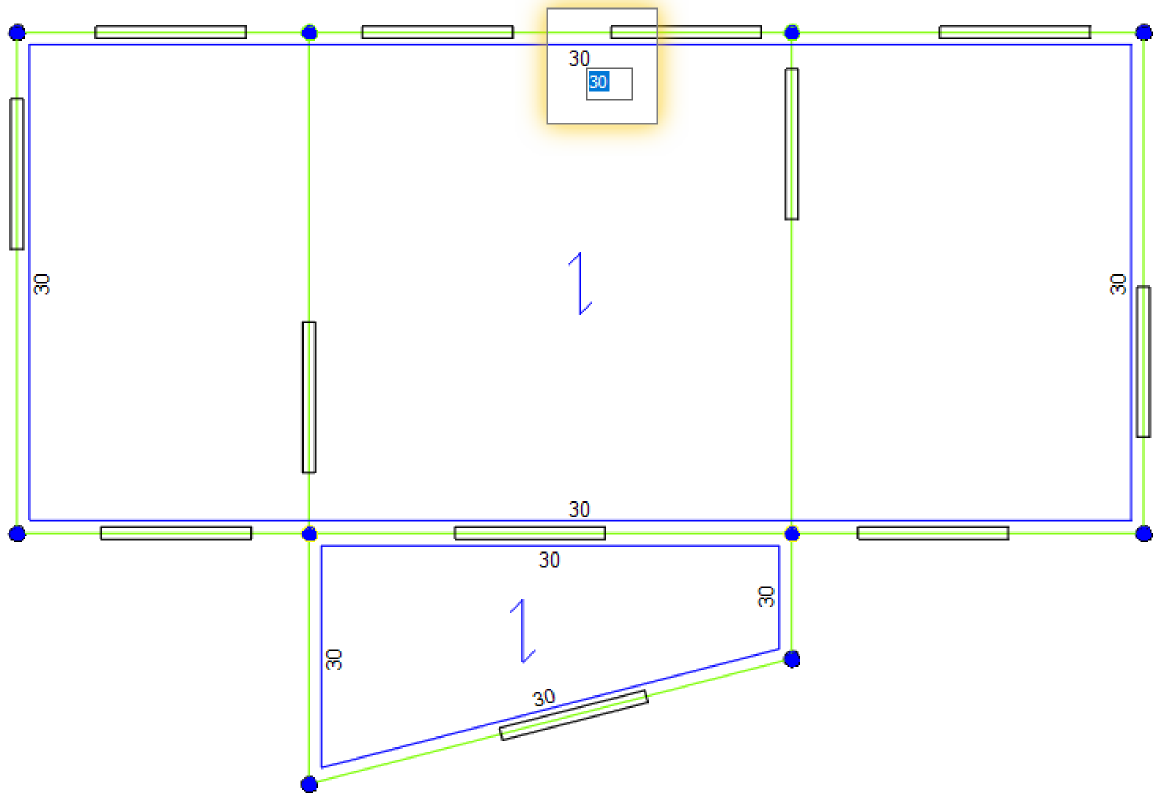
calculation, appears to be precautionary.
In some cases, such as in the case of static analysis, it may be necessary to refine the calculation by considering a reduced height (net height or effective height) which takes into account the contribution given by the thickness of the slabs.



In order to consider it is sufficient to assign,
within the mask, a value to the parameters Δ extrados/intrados elevation.
By inserting a new slab inside the model, the fields Δ extrados/ intrados elevation are
automatically filled with suggested values based on the thickness attributed to the slab.

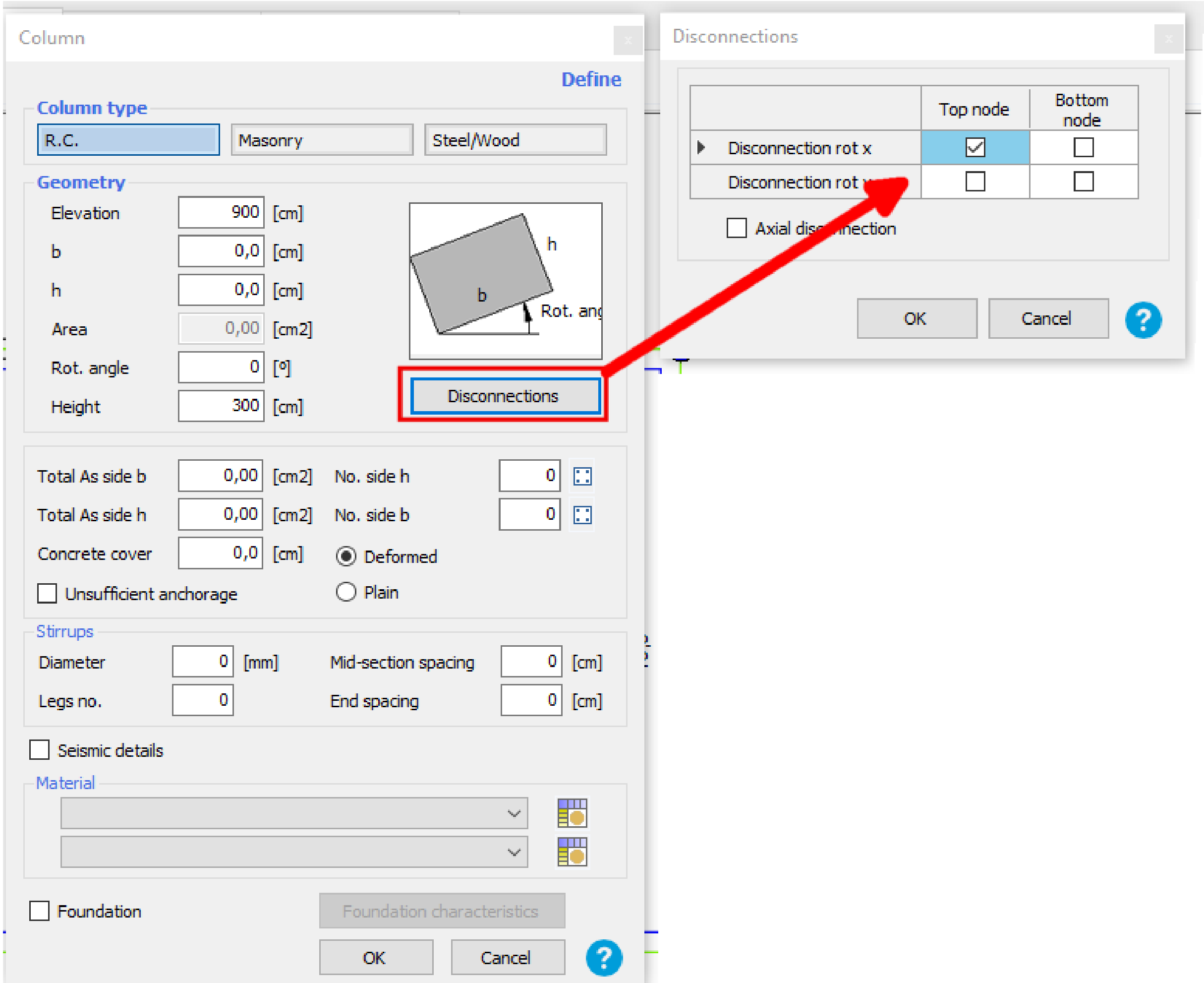
Support Length

In order to calculate the eccentricity required for the static verification of the building, it is important to define the values of the support lengths of the slabs.
If it is necessary to assign different values to each side of the slab, it is possible to edit the values corresponding to the support lengths directly from the structure environment.



Columns disconnections

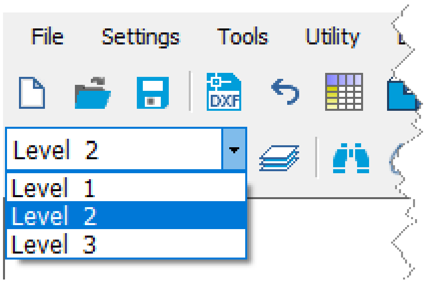
Through the disconnection button it is possible to modify the constraint degree of the extreme nodes of the columns.



ù

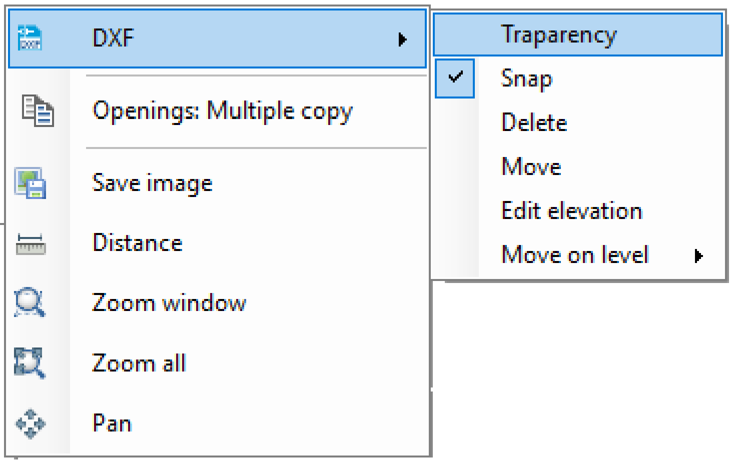
Levels management

From the command bar it is now possible to move directly from one level to another without accessing to the level management



Dxf transparency

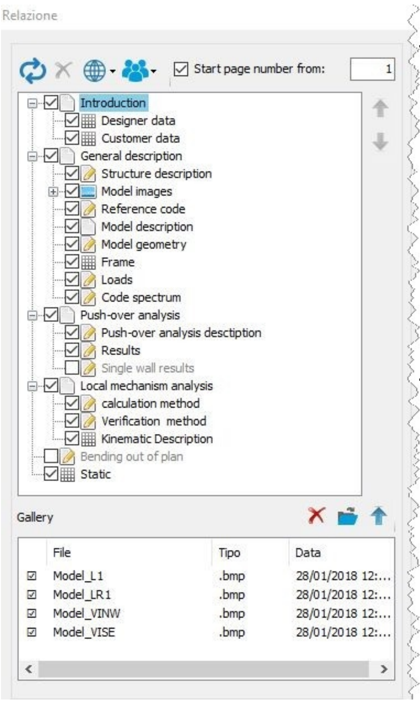
In some cases, the basic dxf graphic risks being excessively invasive, making it difficult to work on the elements already inserted.
In such cases it may be useful to adjust the transparency of the dxf in order to make the operations on the graphics easier.



2.10 Version 12.1.0

Renewed calculation report

The automatic generation of the report often produces a "semi-finished" document or a draft that requires of course significant intervention from the user before arriving at a definitive and deliverable version. This phase is almost always repetitive and somehow boring, but automating it saves time.
The different reports are now integrated into a single environment that allows the user to create a complete document of all the different types of calculation and verification.



The new calculation report allows the user to insert inside the final report, in both automatic and manual mode, some images of both plan and axonometric views.

Inside the report, the walls that suffered the highest failure percentage are automatically reported for the most severe analyzes.

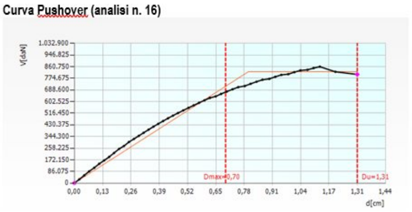
The contents have also been expanded, both from the theoretical point of view (constitutive bonds) and from the presentation of the results (capacity curves and damage maps).

An introductory chapter illustrates the non-linear constitutive bonds of the masonry and a table, created automatically, contains the textual descriptions of the types of masonry inserted.

Nome	Tipo	Colore	Descrizione
Muratura	Muratura		
C32/40	Calcestruzzo		
B450	Acciaio armatura		NTC08
S 235 (t <= 40mm)	Acciaio strutturale		UNI EN10025-2 (laminati)
M1-pietrame	Muratura		Pietrame eterogeneo
M2-blocchi	Muratura		Blocchi in pietra squadrati
M3-mattoni	Muratura		Mattoni pieni
GL24h	Legno		Lamellare
POROTON_30	Muratura		
M1-pietrame_Migliorato	Muratura		Pietrame eterogeneo
M3-mattoni_Migliorato	Muratura		Mattoni pieni

A color legend allows us to visually associate the materials in the table with those shown in 3 view.

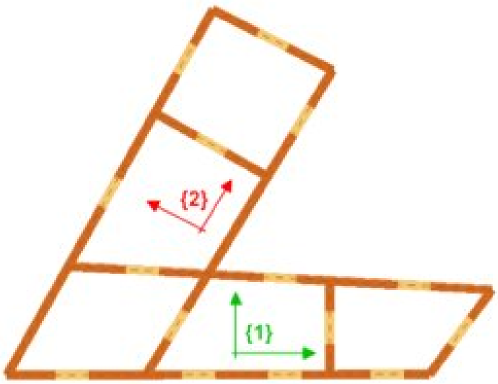
Curva Pushover (analisi n. 16)



The results chapter contains images of the capacity curve and accurate tables of vulnerability indices calculated on both accelerations and return periods for the most severe analyzes in the two main directions.

Pushover in generic directions

Normally the pushover is performed according to the two main X-Y directions (system {1} in the figure).
Some planimetric configurations could however give rise to the doubt that these directions are not actually the most significant.



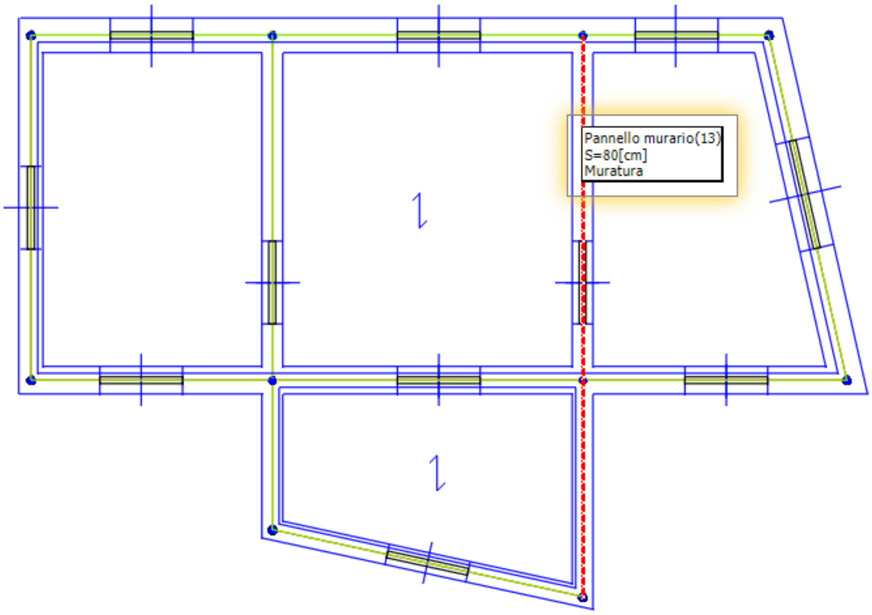
A planimetric layout as shown in the figure to the left side is formed by two main bodies:

- **Body-1**: represented quite well by the system {1} because it consists of walls that are approximately parallel to the system itself.
- **Body-2**: Generally inclined respect to Body-1 it is probably best represented by an inclined system {2}.

A new calculation mode will allow the user to perform pushovers in different directions simultaneously, simply by defining the angle in input.

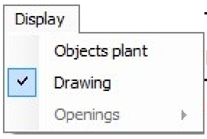
Tooltip of the elements

Within the structure environment it is possible to display the dedicated tooltip by moving the cursor over the required element.
The latter contains inside information concerning the element under examination such as height, width, thickness, thus resulting in a valid alternative to quickly find all the necessary information.

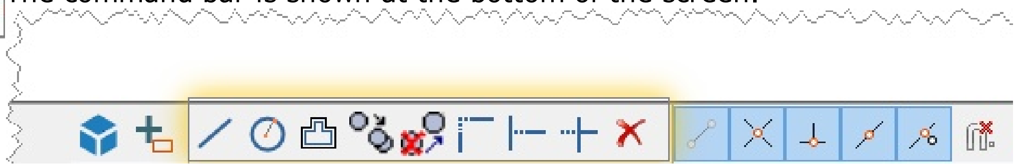


2.11 Version 11.5.0

The support graphic is obtained through a set of commands presented in the following figure, which allow to insert graphic entities that can be used as guidelines for the creation of the model. Structural objects are not associable to the supporting graphic entities. Its use allows the user to have available guidelines with which to proceed with the creation of the model. An imported DXF or DWG drawing is considered as supporting graphic.



The environment dedicated to graphics support is activated via the display menu.
The command bar is shown at the bottom of the screen.



 Line insertion: these icons allow to insert generic, vertical, horizontal or perpendicular lines to other elements, to support the insertion of the structure.

 Circle insertion: allows the user to insert a circle for three points or a circle given the center and the radius, to support the insertion of the structure.

 Offset: allows to duplicate a line at a certain distance.
After selecting the line it is required the distance and the direction in which the duplication must be performed.

 Duplicate: allows the user to copy a graphic element.

 Move: allows the user to move a graphic element.

 Trim to intersection: allows to trim to intersection two lines

 Extend: Allows the user to extend one line over another

 Cut: Allows the user to cut one line over another

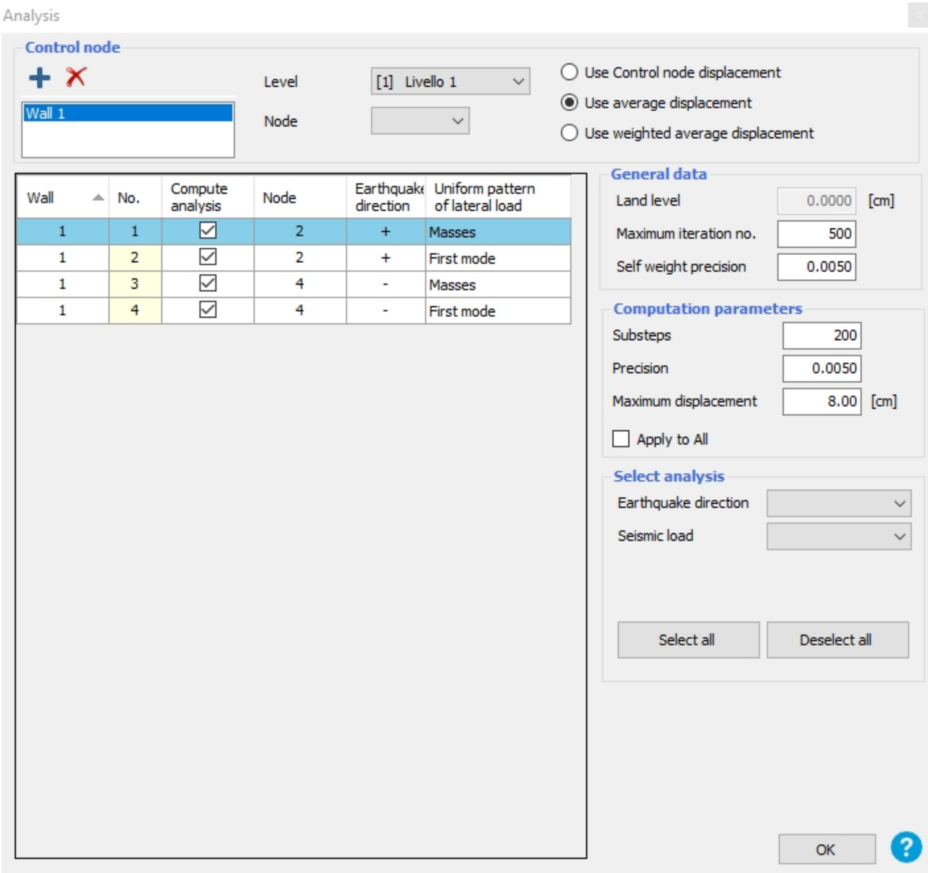
 Delete: delete graphic entities

WARNING: DO NOT USE THESE COMMANDS FOR THE STRUCTURE MANAGEMENT, BUT ONLY AS A HELP FOR ITS INSERTION.

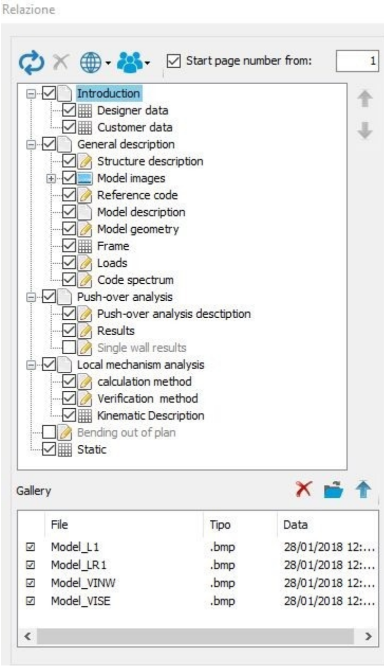
2.12 Version 11.4

Pushover Analysis of single wall

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


Analysis
Control node
Wall 1
Level
[1] Livello 1
Node
Use Control node displacement
Use average displacement
Use weighted average displacement
Wall
No.
Compute analysis
Node
Earthquake direction
Uniform pattern of lateral load
1 1 2 + Masses
1 2 2 + First mode
1 3 4 - Masses
1 4 4 - First mode
General data
Land level 0.0000 [cm]
Maximum iteration no. 500
Self weight precision 0.0050
Computation parameters
Substeps 200
Precision 0.0050
Maximum displacement 8.00 [cm]
Apply to All
Select analysis
Earthquake direction
Seismic load
Select all
Deselect all
OK

Report environment renovated



Previous versions of the 3Muri program generated distinct reports:

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

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

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

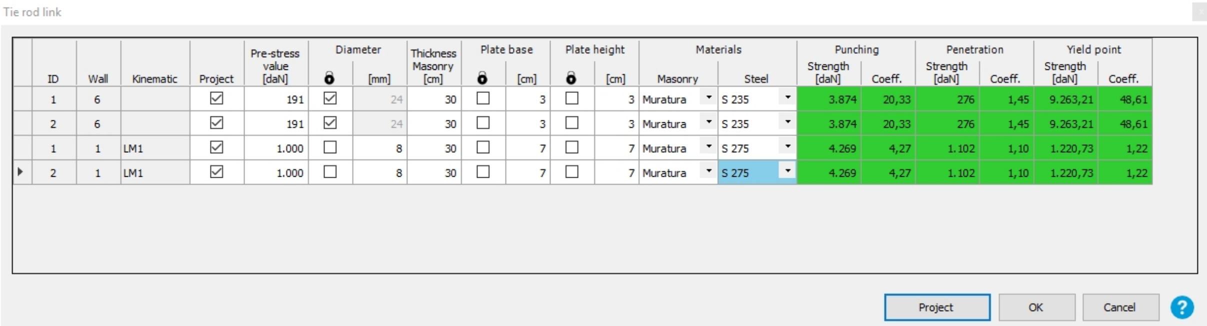
The new procedure with which the report was restructured allows to obtain a final *result ready for delivery*, considerably limiting any further reprocessing.

Tie rod link

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

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

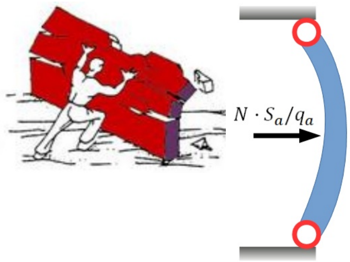
Given the characteristics of the masonry and the pre-stress value that the anchor must resist, it is possible to carry out the project with the shape of the rectangular square plate (fixing one of the two dimensions).



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

2.13 Version 11.3

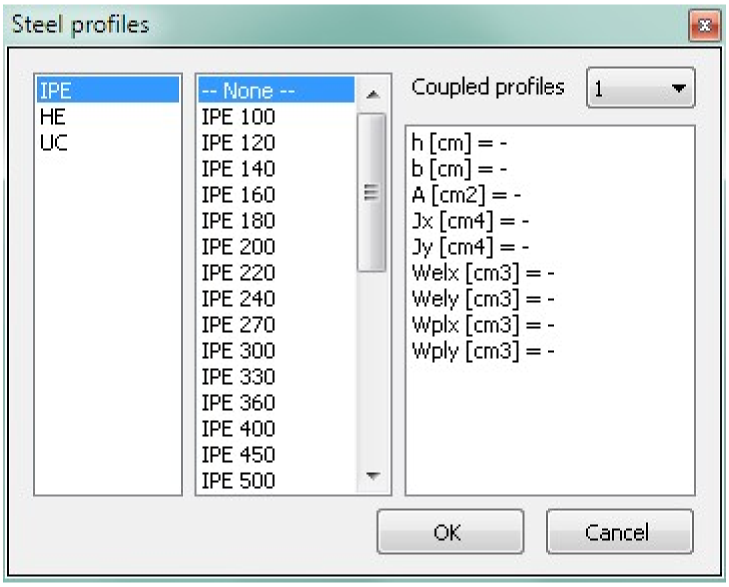
Verifications out of plan



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

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

New library of metal profiles with new configurations



I profiles
- IPE
- HE
- UC

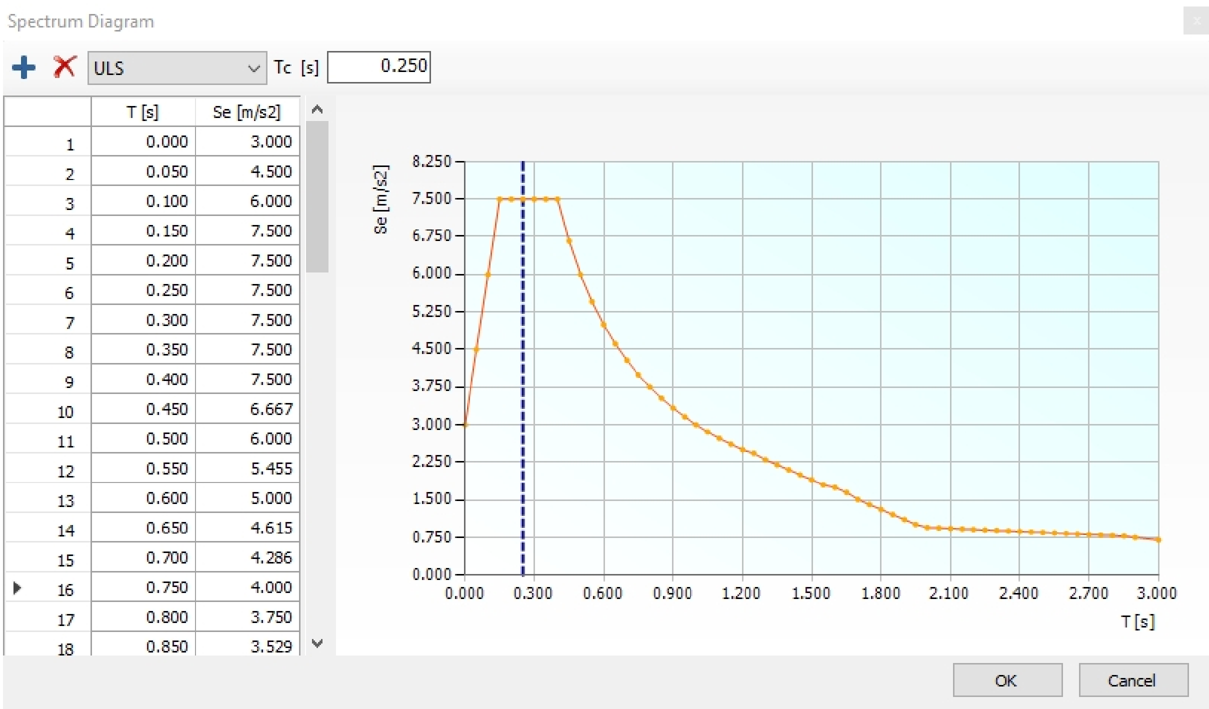
C profiles
- UPN
- UAP
- UPE

L profiles
- L equal wings
- L different wings

Double Profiles
- UPN (Type 1)
- UPN (Type 2)
- UAP (Type 1)
- UAP (Type 2)
- UPE (Type 1)
- UPE (Type 2)
- L equal wings
- L different wings

Definition of the spectrum by points

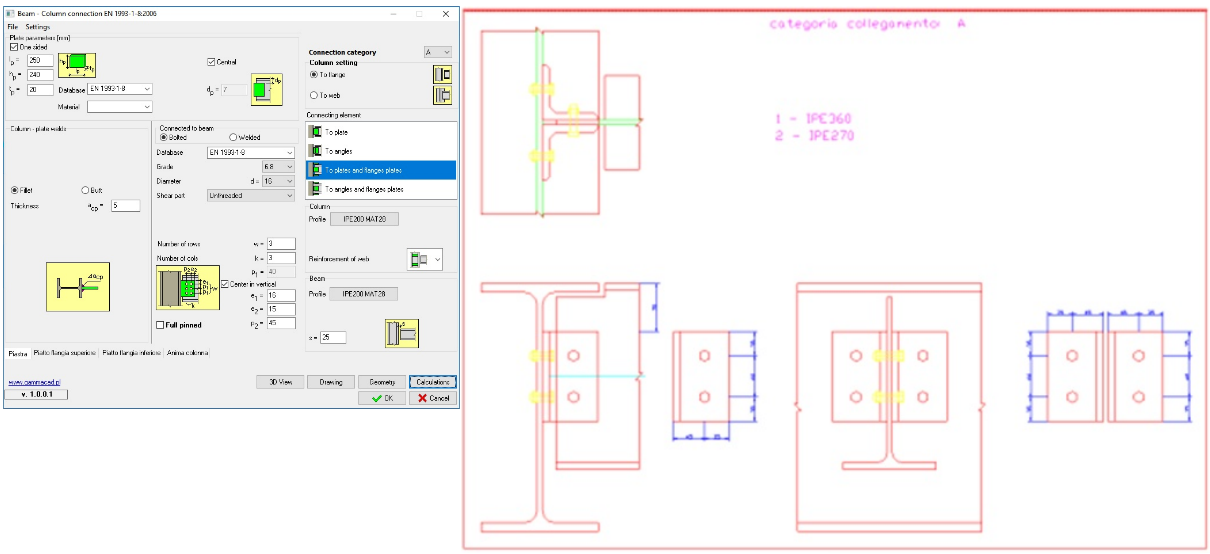
In addition to the classical process of defining the seismic spectrum based on the reference grid and to the definition of seismic parameters, it will be possible to define in a general way by points.



Link to SteelConnection

[OPTIONAL]

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

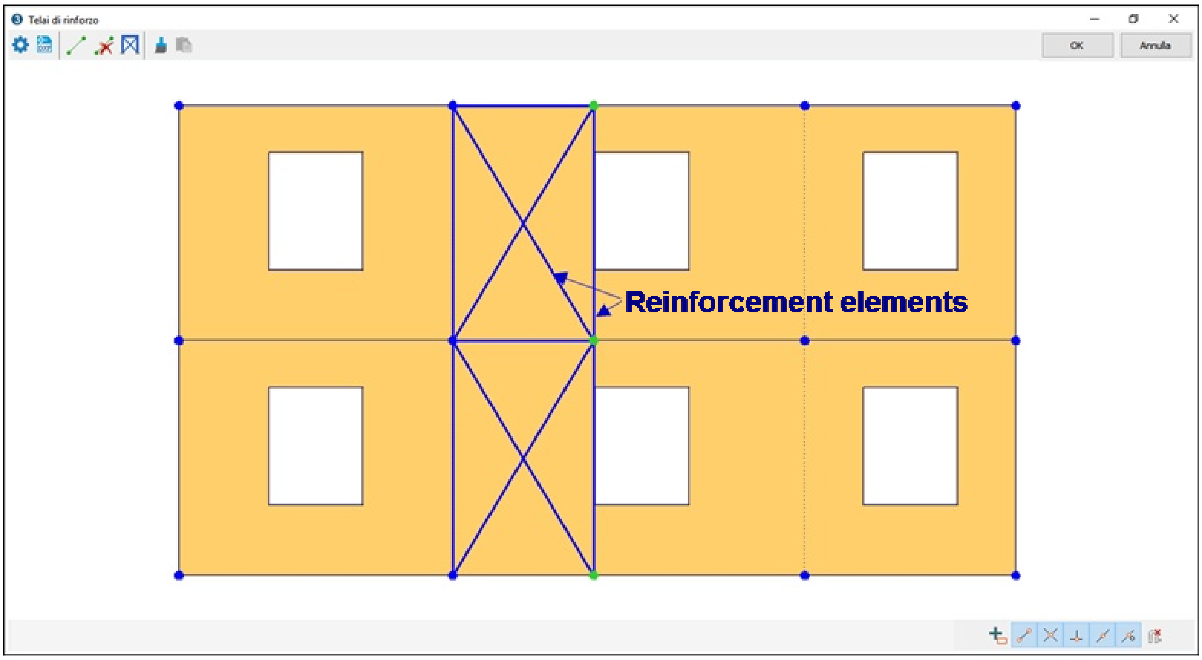


The "Steel Connection" module offers a wide range of types of steel connections, such as to cover all the needs of the designer. In addition to the calculation report, complete and exhaustive, they are also available designs of connections in DXF format.

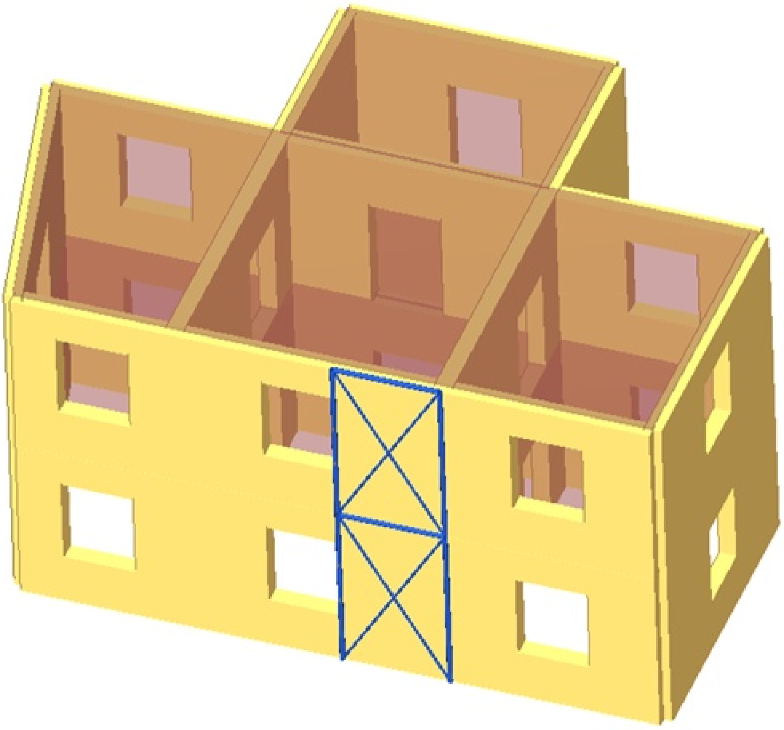
2.14 Version 11.0.0

Wall reinforcements through the frame

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



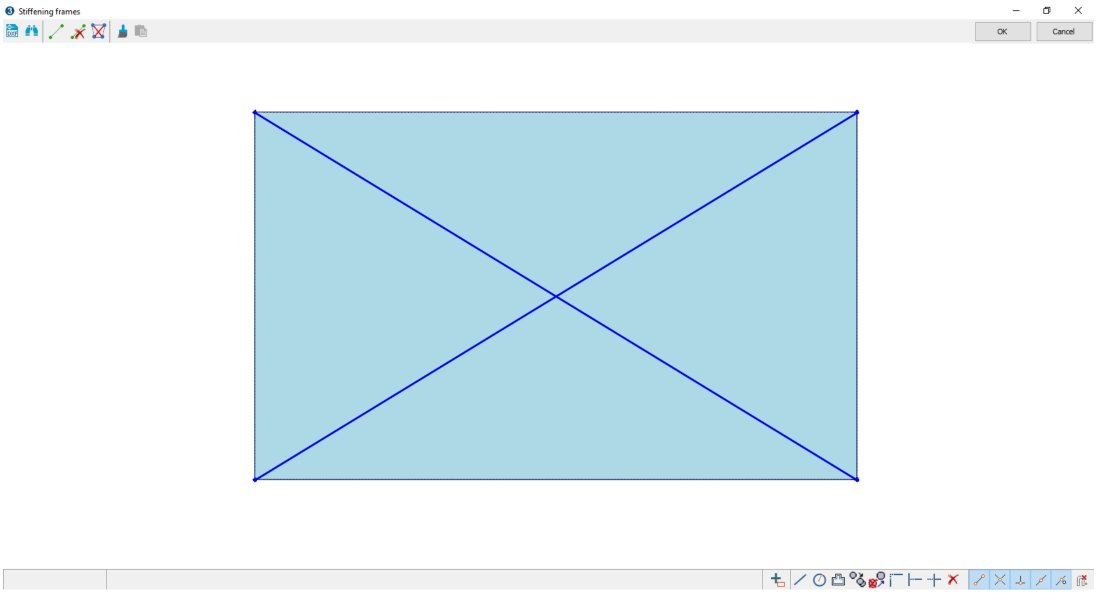
Following insertion in perspective view, the reinforcing elements will also be shown in the axonometric view.



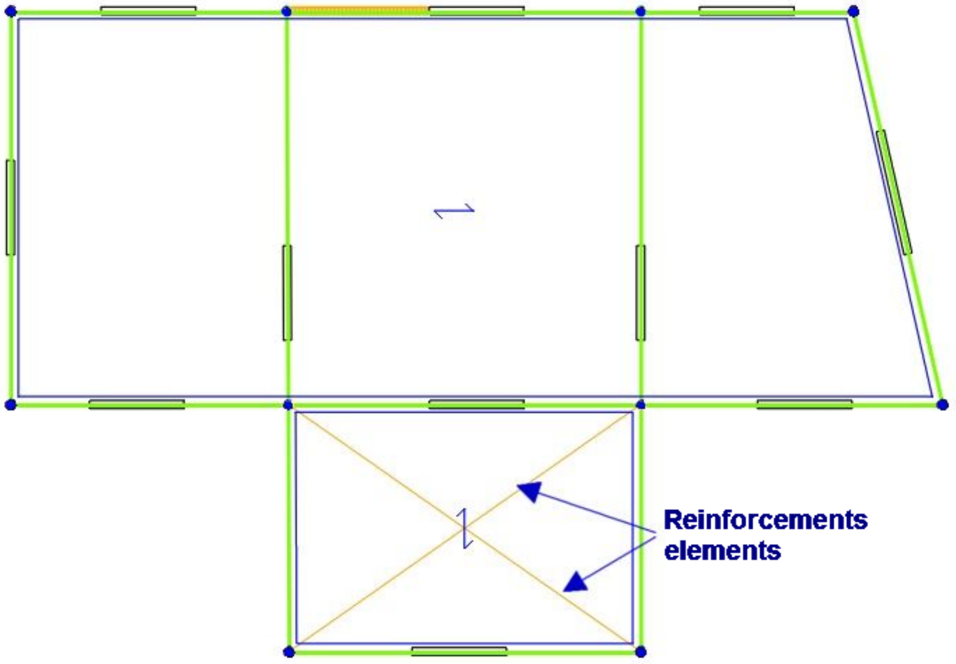
The structural elements that can be inserted are steel or wooden beams and tie rods (elements with only tensile strength).

Slabs reinforcements through the frame

The slabs of existing buildings, very often made of wood, have extremely limited stiffness, allowing important deformations and distortions in plan.
It is not always possible to enter the completion slabs to limit distortions in the plan and therefore it is necessary inserting diagonal reinforcements on floors and roof slopes.
Selecting a slab and choosing the "stiffening frames" voice will open a dedicated CAD environment. The following figure shows a diagonal reinforcement applied on the plan of the slab.

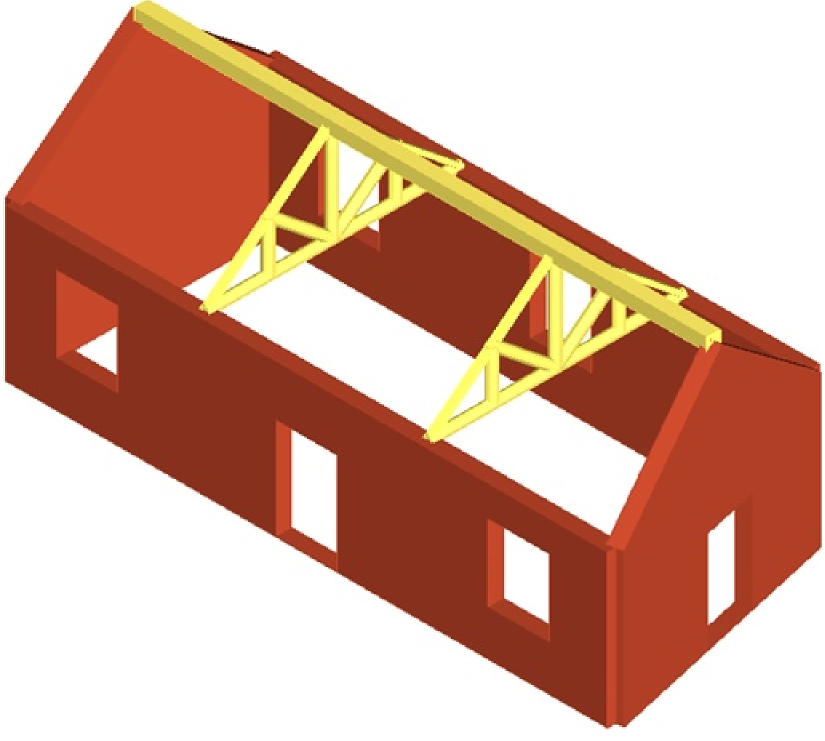


Once completed the input, the reinforcement is visible in the structure plan.



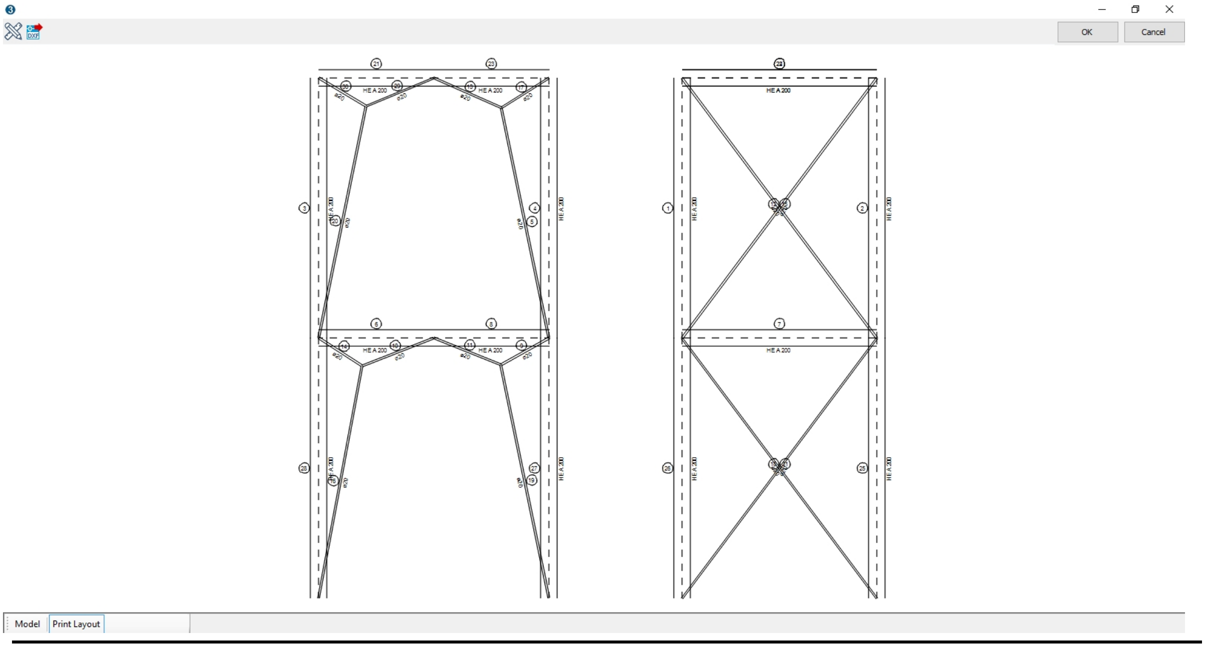
Reinforcement through truss beam

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



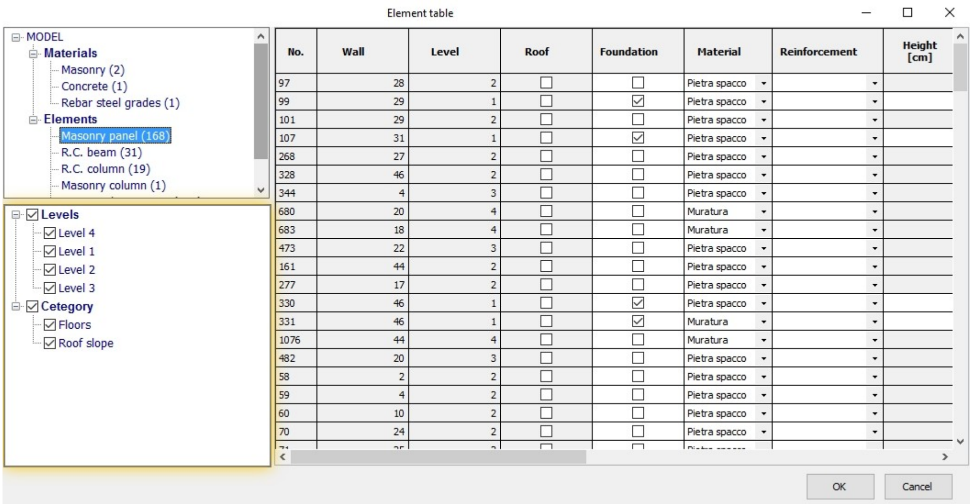
Graphic tables of steel elements

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



Update of the Elements Table

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

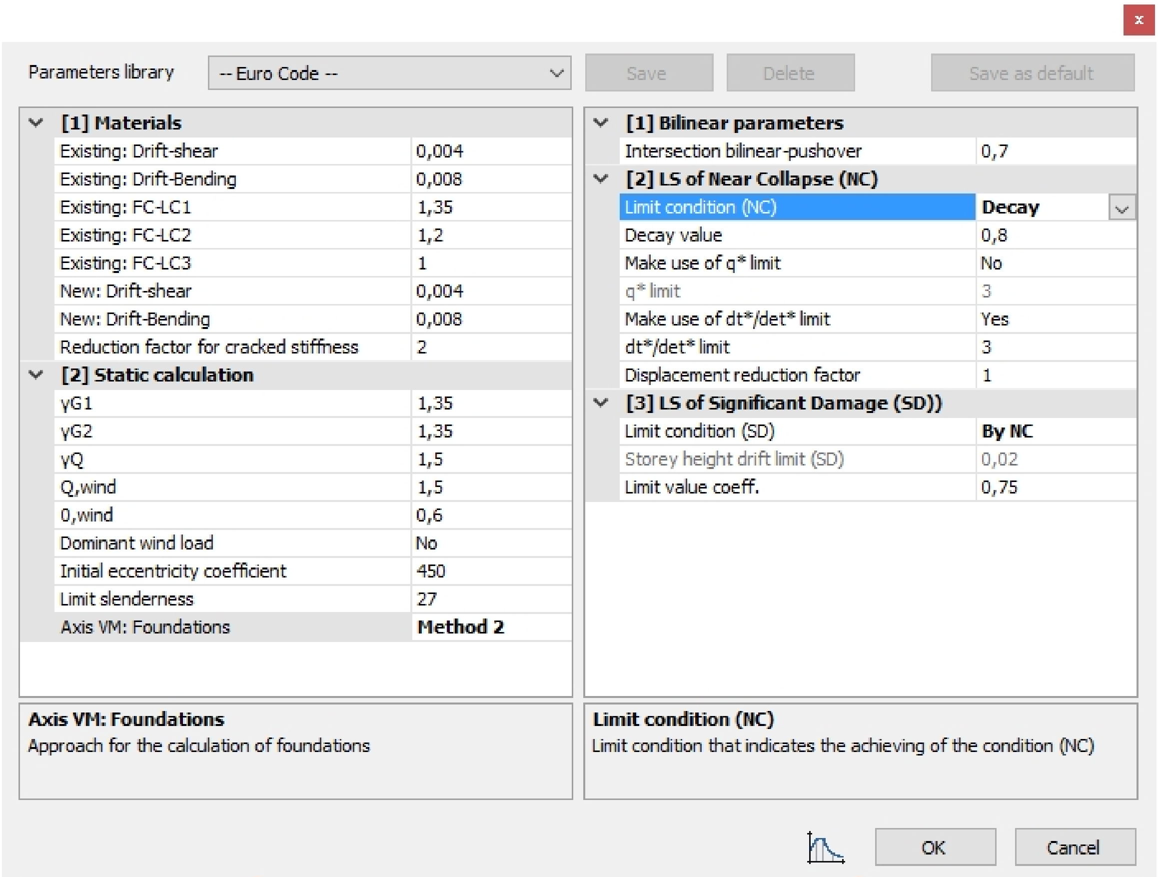


No.	Wall	Level	Roof	Foundation	Material	Reinforcement	Height [cm]
97	28	2	☐	☐	Pietra spacco		
99	29	1	☐	☑	Pietra spacco		
101	29	2	☐	☐	Pietra spacco		
107	31	1	☐	☑	Pietra spacco		
268	27	2	☐	☐	Pietra spacco		
328	46	2	☐	☐	Pietra spacco		
344	4	3	☐	☐	Pietra spacco		
680	20	4	☐	☐	Muratura		
683	18	4	☐	☐	Muratura		
473	22	3	☐	☐	Pietra spacco		
161	44	2	☐	☐	Pietra spacco		
277	17	2	☐	☐	Pietra spacco		
330	46	1	☐	☑	Pietra spacco		
331	46	1	☐	☑	Muratura		
1076	44	4	☐	☐	Muratura		
482	20	3	☐	☐	Pietra spacco		
58	2	2	☐	☐	Pietra spacco		
59	4	2	☐	☐	Pietra spacco		
60	10	2	☐	☐	Pietra spacco		
70	24	2	☐	☐	Pietra spacco		

2.15 Version 10.9.1

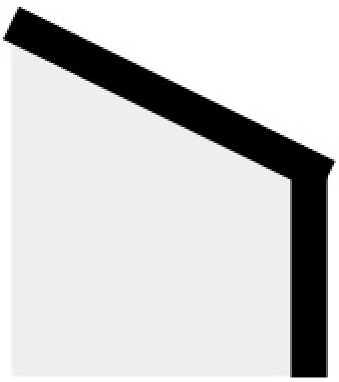
Personalized calculation parameters

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



Roof overhangs

Possibility of inserting the overhang of the roof slope.



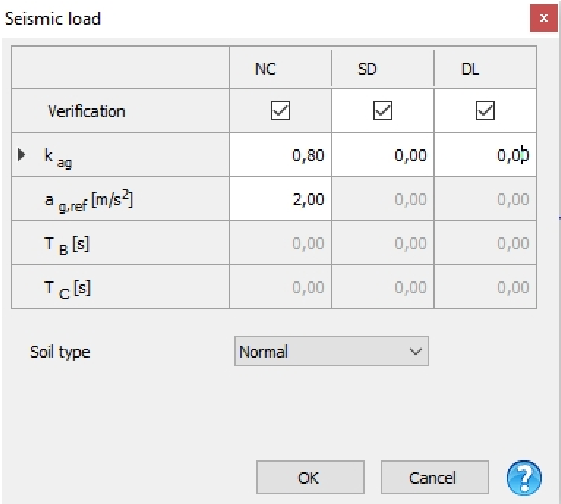
Without overhang

With overhang

Standard NPR 9998/2015 [Eurocode (NL)]

[OPTIONAL]

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



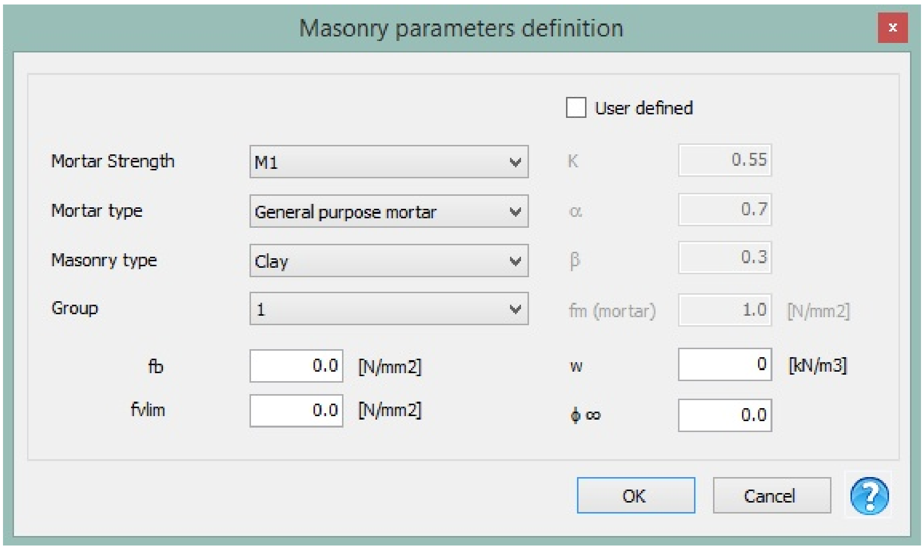
2.16 Version 10.9

Static analysis according to Eurocode 6

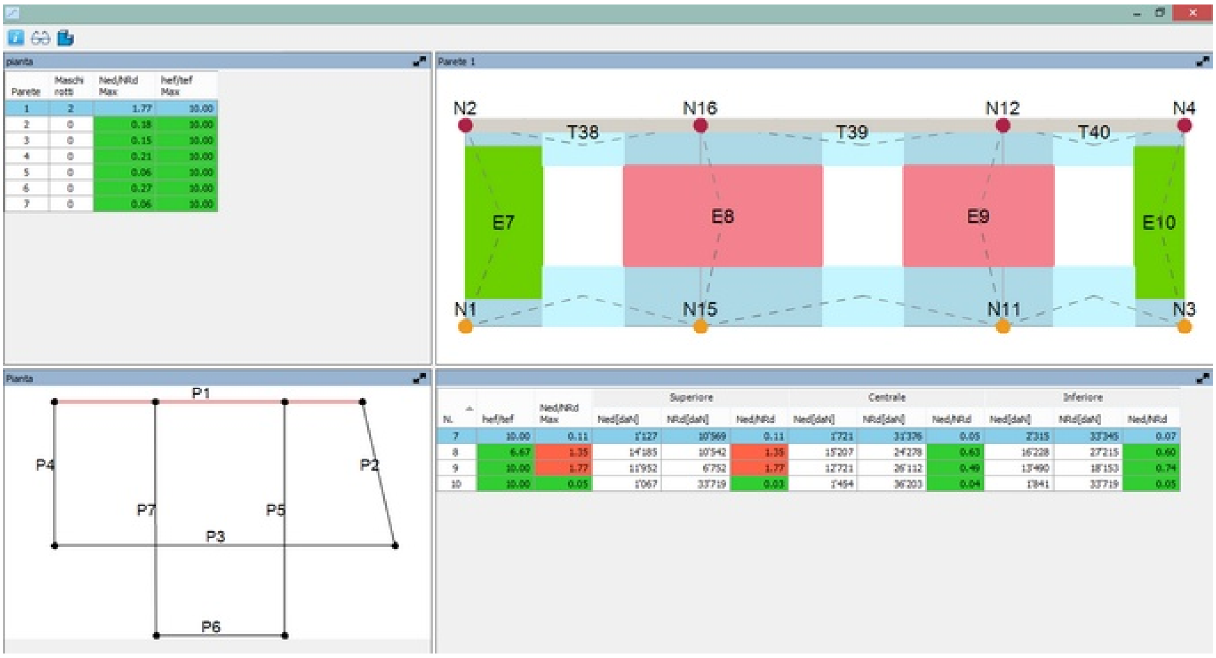
[OPTIONAL MODULE]

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

Eurocode 8 contains all the references about the verification procedures of "Seismic" type, there is no indication to the static verifications instead of them contained in the Eurocode 6.
It is now possible:
➔To impute the mechanical parameters of masonry materials according to the procedures described in *EN 1996-1-1 §3.6:*



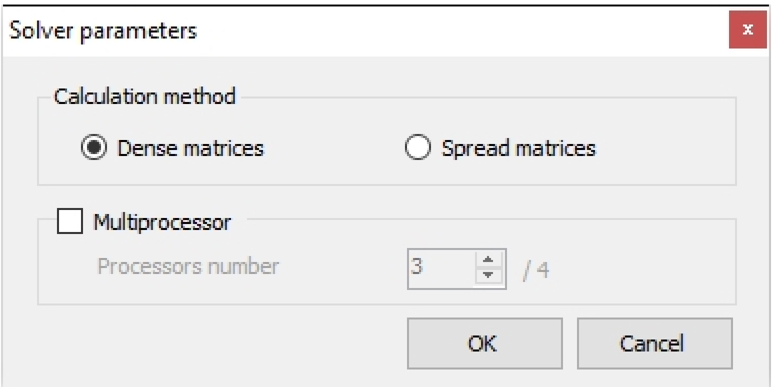
➔ Run the static verifications according to *EN 1996-1-1* § *6.1.2*



Solver parameters

[OPTIONAL MODULE]

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

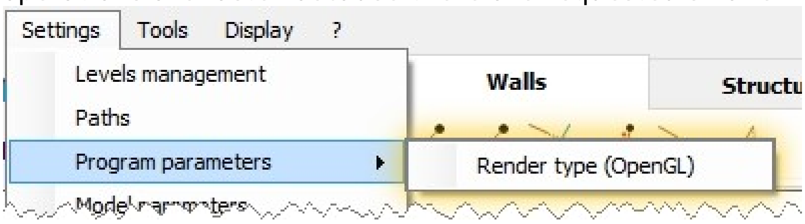


Calculation method:
- Dense matrix
- Sparse matrix

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

Graphic optimization

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



Loads on roofs

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

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

2.17 Version 10.5

Sensitivity Analysis [OPTIONAL MODULE]

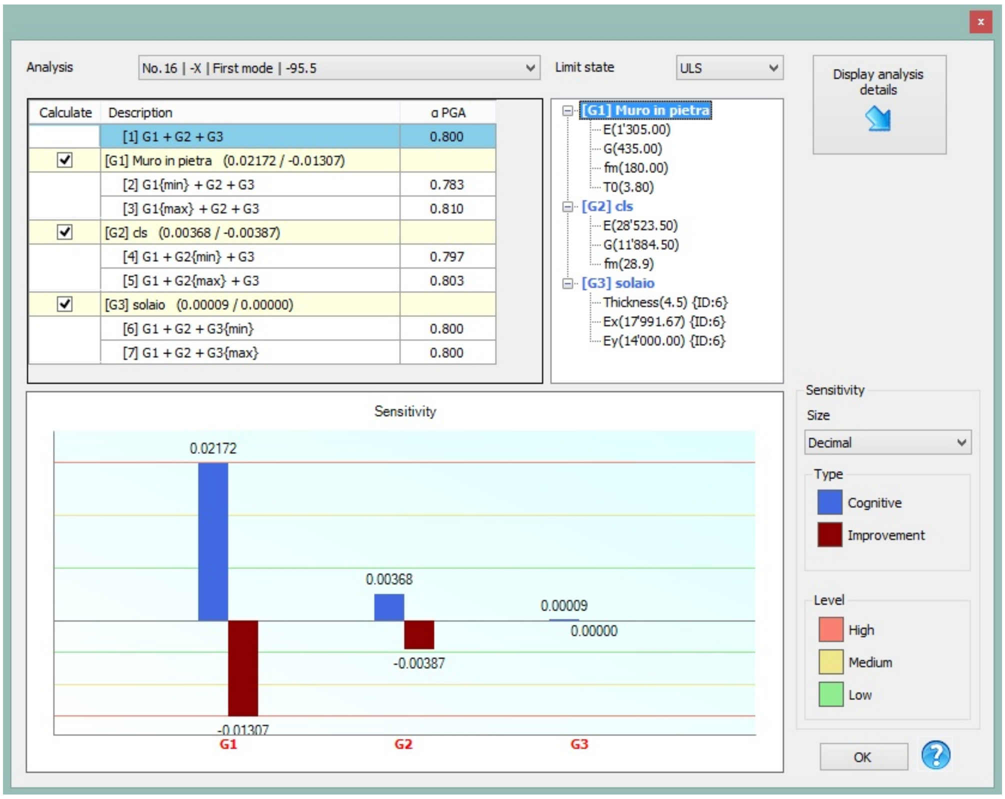
Sensitivity Analysis is a calculation method aimed to obtain better understanding of the structural functioning and accurate planning of the site investigation plan.
As known, doubts during modeling directly affect the evaluation of seismic safety. A specific example is materials mechanical properties,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

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

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

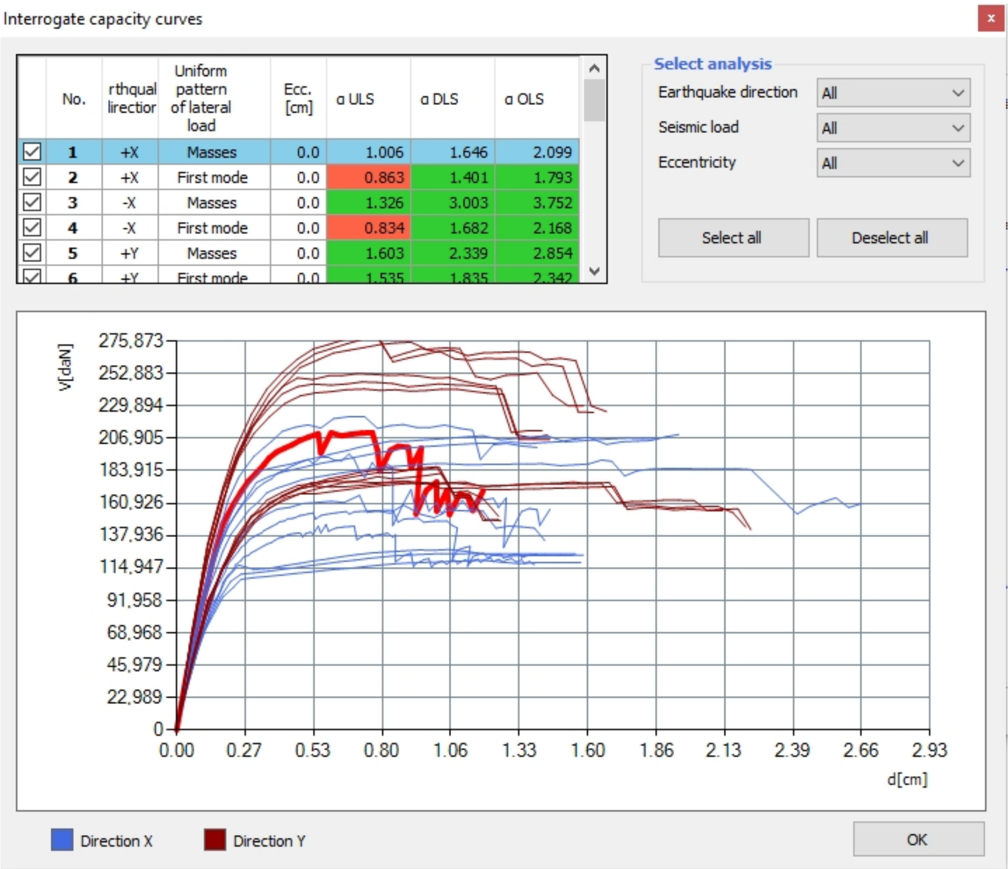
«Improving» Sensitivity *to answer the following questions:*
- Which type of improving intervention is more effective?

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



Overlapping capacity curves

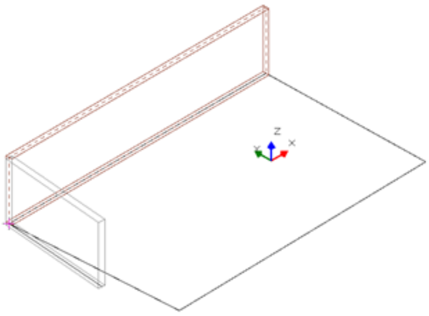
This feature allows the designer to simultaneously refer to the capacity curves of more analysis on a single diagram.



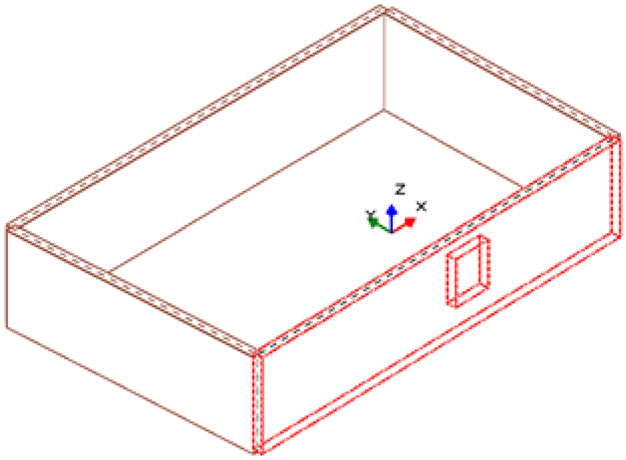
2.18 Version 10.1

3D modeling

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



Insertion of masonry walls in 3D



Insertion of openings on a masonry panel

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

Specific applications of 3D editing

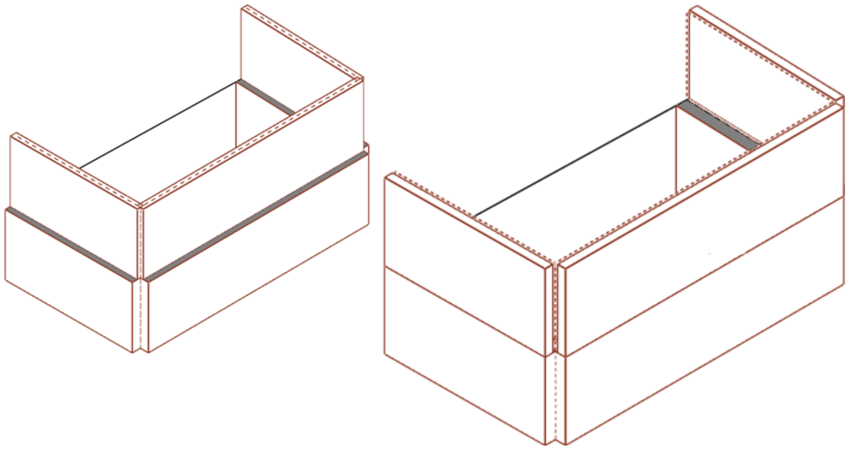
Following a multiple selection of structural objects (also on different levels), you can:

Align panels

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

[Before editing]
Walls aligned with the lower middle plan

[After editing]
Walls aligned with the outer edge



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

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

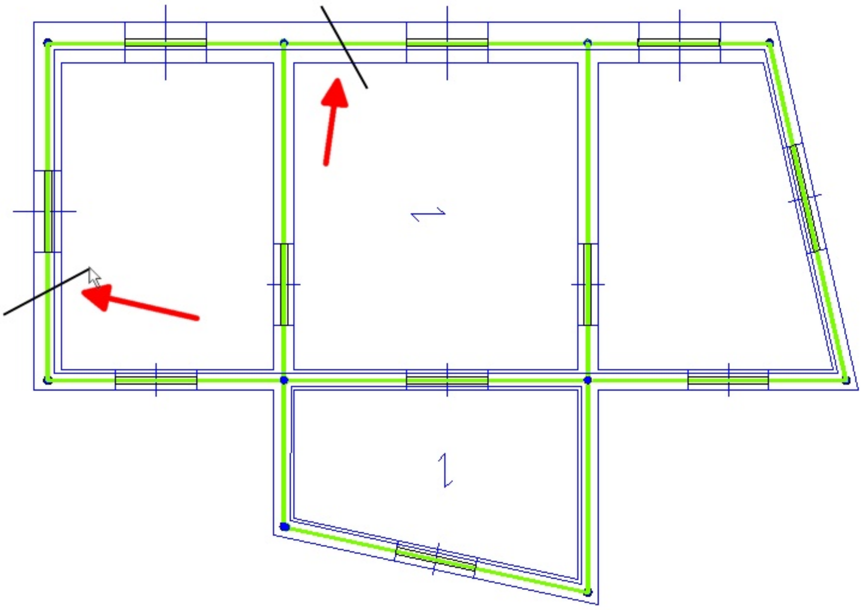
3D Filters-The Parts

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

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

Automatic detection of the thickness

During the input phase, you no longer have to worry about the various wall thickness in the different sections and perform the input with any thickness.
In a second step a dedicated command (by tracing a segment that intersects the wall) automatically calculates the thickness by extracting it from the DXF file and assigns it to the already defined section.



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

---

Input by dimension lines

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

IFC management – Output/Input

[OPTIONAL MODULE]

Export 3Muri to IFC

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

Advanced importation

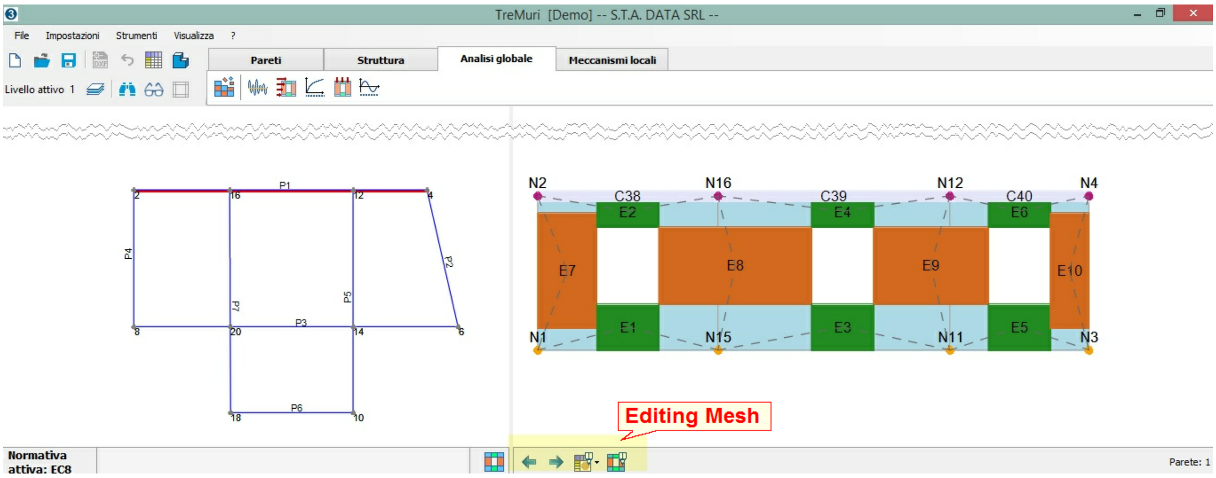
Runs the OpenCAD application, a CAD system that makes interoperability its strong point.
With interoperability we mean the ability to cooperate and exchange information with the other products or services with resources's optimization.
The information exchange takes place through various graphic formats: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

2.19 Version 10.0.2

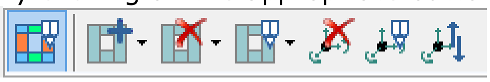
New Editing Mesh

In mesh editing you can change the characteristics of the generated structure according

to the design requirements.
The access in the editing mesh area is achieved through the appropriate button on the command bar.



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



2.20 Version 10

3muri 10 years

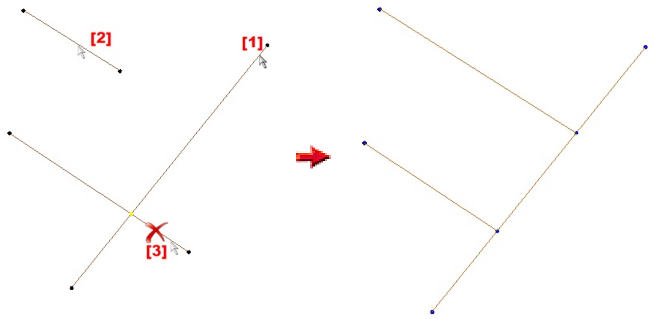
3Muri r. 10 is the new version of the software for the seismic calculation of the masonry structures.
The program has been completely rewritten in order to update it to the new operating systems, improve the present features and add new commands.

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

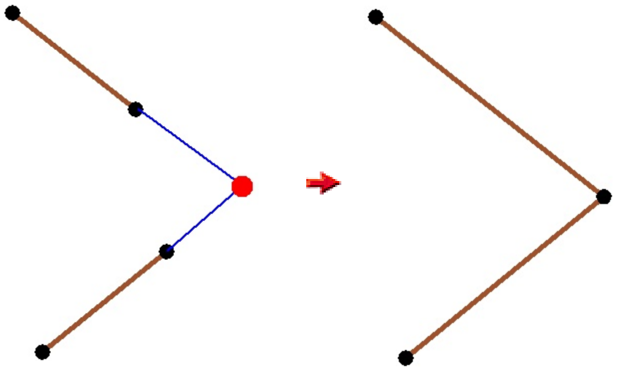
New walls' editing operations

Rectify walls: Allows to rectify the previously entered walls.

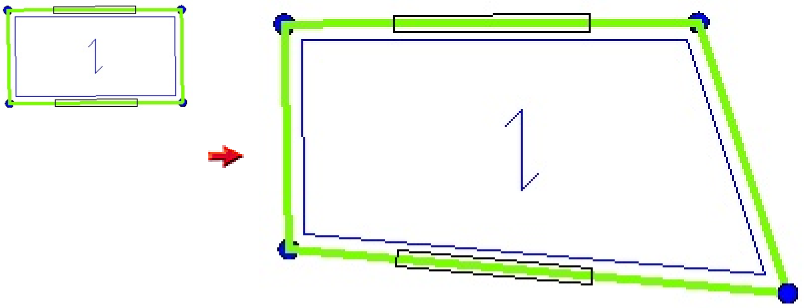
Extend/ Trim walls: allows to extend or shorten an already existing wall.



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

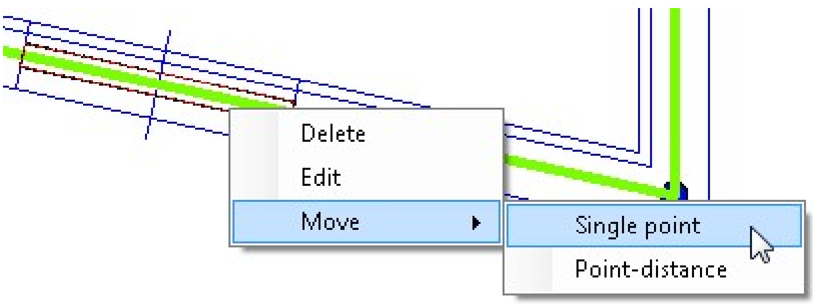


Stretch: This command allows to move an outer node of the wall.



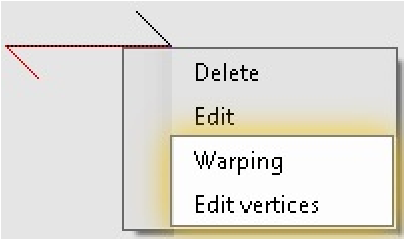
Movement of the openings.

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



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

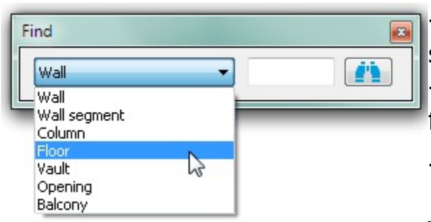
New editing modality of the slabs.



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

New search functions of the structural objects`

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



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

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

New DXF file importation functions

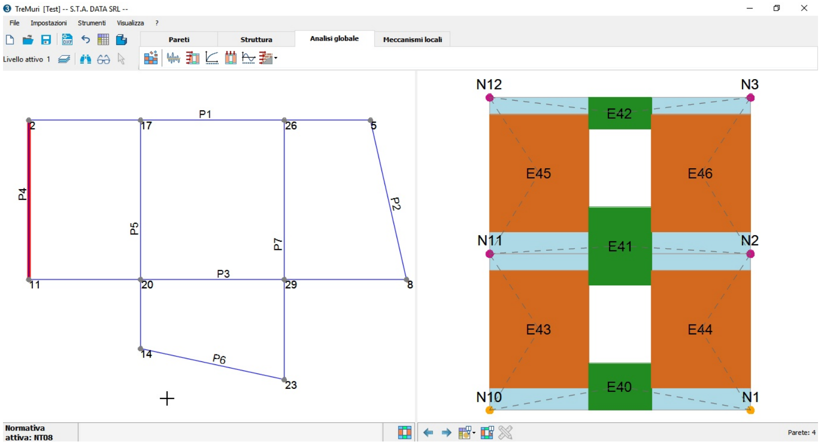
Cancel or easily move a DXF file translating it in the graphics area. These new functions are directly available by clicking on the right button of the mouse in the graphics area.

Editable elements table

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

New mesh display of the geometry

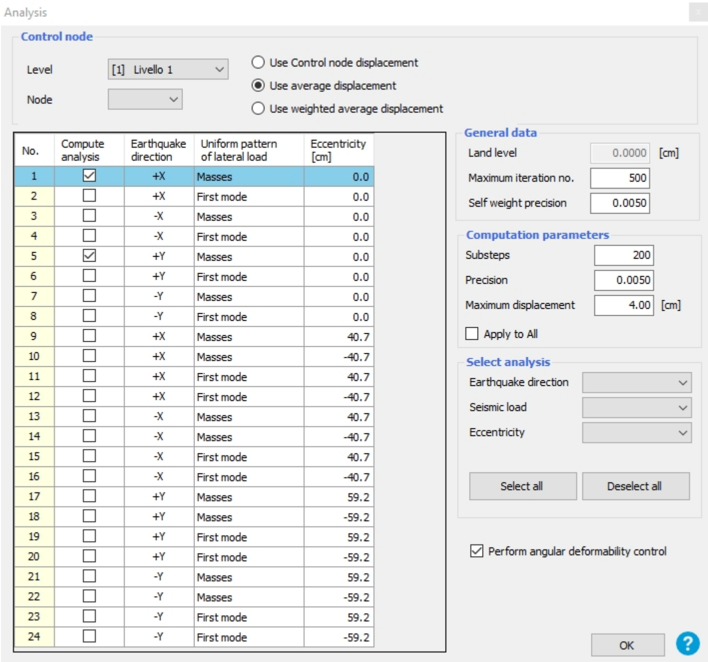
The graphics area is separated in two different areas, the plan on the left and on the right the view of the mesh of the selected wall.
The selection of a wall is possible through a simple click on the plan.



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

Improvements for the management of the analyses` parameters.

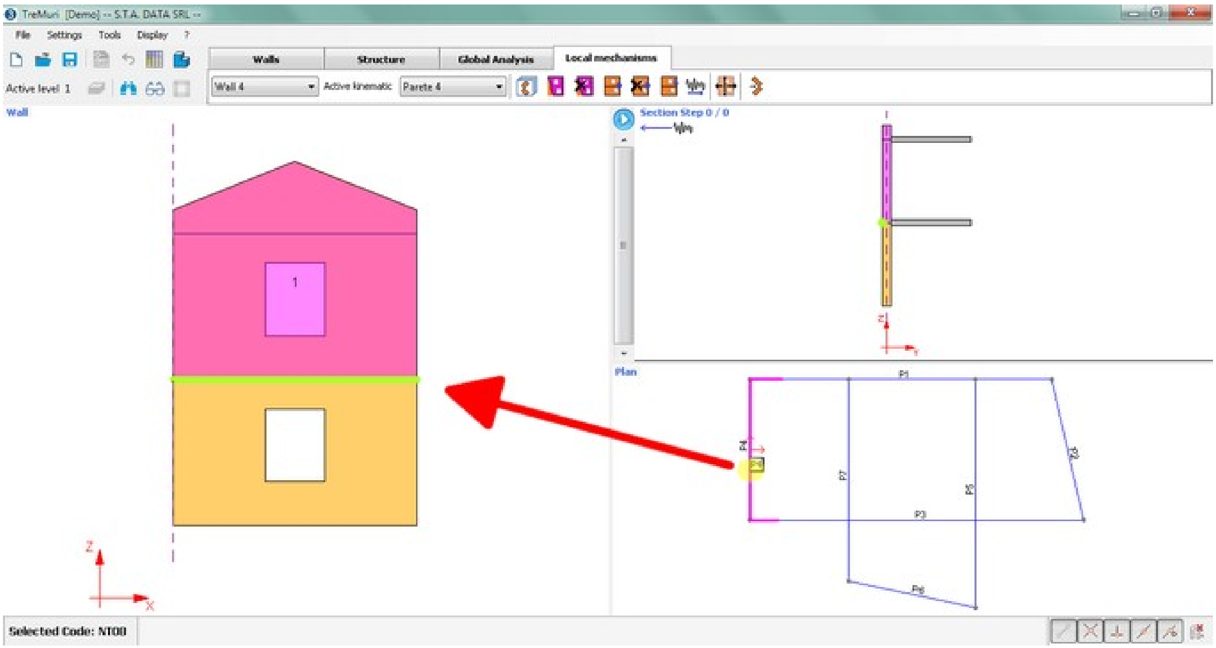
The calculation window shows on the table the "essential" data necessary to describe the analysis, on the right the calculation parameters.



Better graphical interaction in the local mechanisms´ area

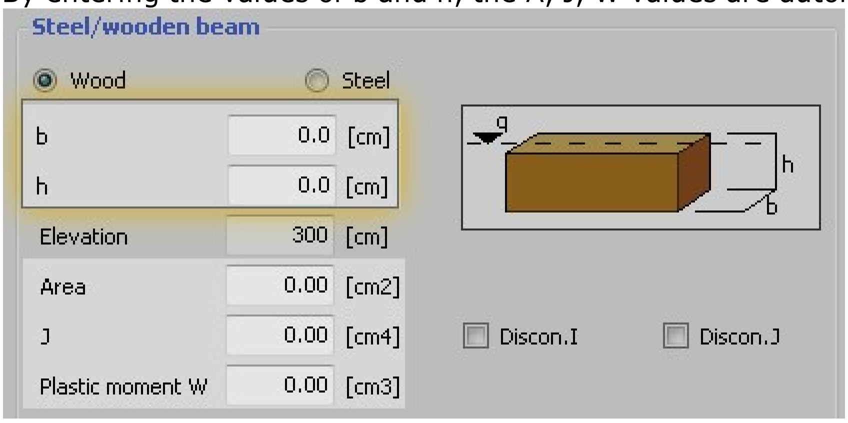
In order to load the view of the selected wall you can now click directly on the plan instead of using the drop down menu bar.

The cad area of the plant and view have been improved for a better graphical management of the input phase.



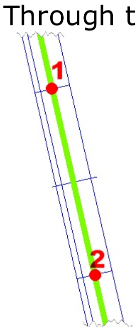
Filling of the properties of the wooden beam

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

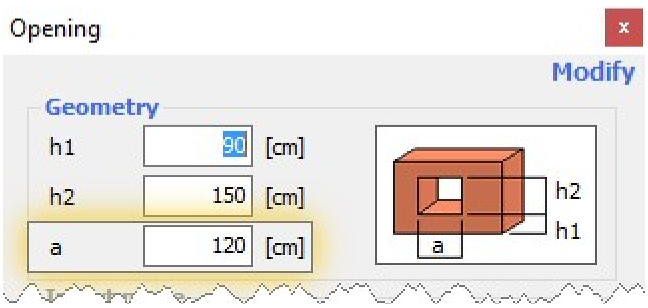


Insertion of an opening through two points.

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

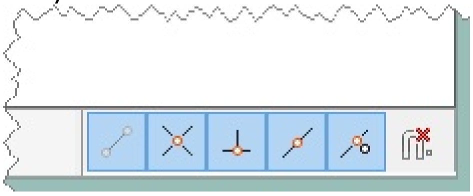


There are necessary two clicks, one at the beginning [1] and one at the end of the opening [2]
The width is no required because it is calculated automatically from the distance between [1]-[2]



New snap utility

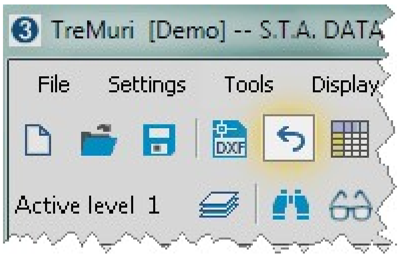
The snap are manageable in a total parametric way through the buttons bar down on the right.
They can be turned on or off according to the desired requirements.



New use of Pan and Zoom

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

New “Undo” command

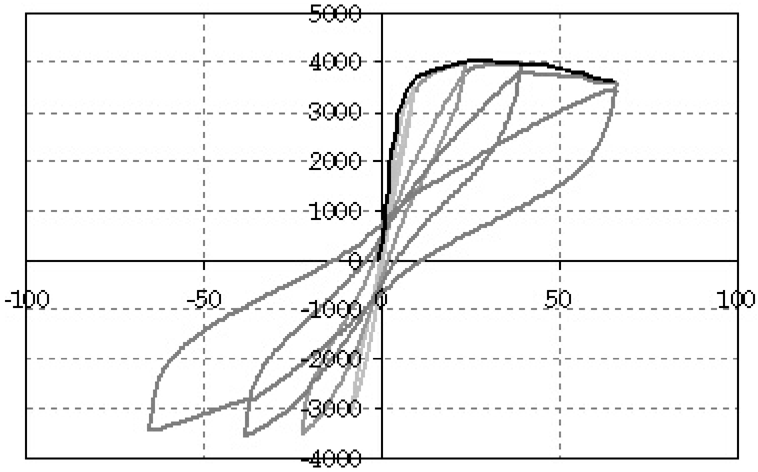


3 Structure Modelling

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

3.1 Static Non-linear Analysis

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



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

The curve obtained by the *pushover analysis* (which will then by transformed into a capacity curve, taking into account the system characteristics equivalent to grades of freedom) conventionally provides information on the'trend of the shear resulting at the base, with respect to the horizontal displacement of a control point on the structure. At each point on the curve, a specific damage state for the entire system can be linked,'and so it is possible to link determined displacement levels to the level of expected performance and the corresponding damage.

The curve is obtained by using *pushover analysis*, which predicts the 'assignment of a preset distribution of forces increasing in a static and monotonic manner. The distribution is kept unaltered even after the fail limit is reached. The analysis can also be conducted controlling for forces or for mixed force-displacement.

The load distribution applied is intended to represent the distribution of inertial forces induced by the 'seismic event. The profiles proposed are those in harmony with the first modal form, for masonry structures, more or less equivalent to those adopted for the 'linear static analysis, and that proportional to the mass. In particular, in the case of regular structures, the first distribution is chosen with the intention of better determining the structural response in the elastic field and secondly, in the non-linear field.

The "capacity" offered by the structure must then be determined, through the lens of a seismic check, with the "demand" requested by the external force, that is by a determined seismic event.

The energy dissipation effects, which offer an ulterior margin of resistance, which can not be explained using only linear elastic theory, are relevant in particular in the field of non-linear structural response: to take them into account the demand is reduced.

The expected response for the 'building, as a function of a determined action, is hence obtained through the identification of the *performance point* (whose coordinates in terms of spectrum displacement corresponds to d*max).

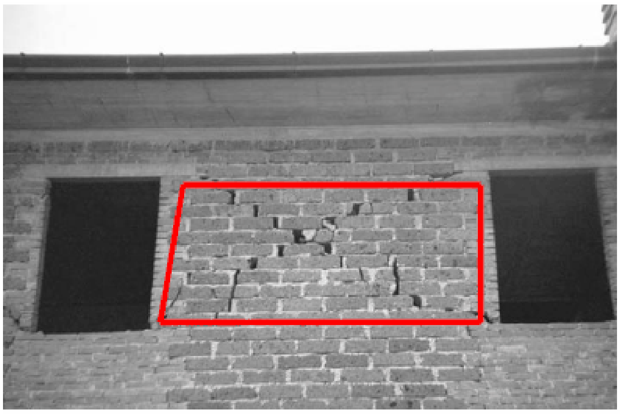
The maximum displacement value that can be offered by the building in a seismic event, is obtained in correspondence with the value of the shear that underwent a decline of 20% from the shear limit value. Based on the capacity curve of the real system defined in this way, it passes to the bilateral associated with the equivalent system; once found, the system period with one degree of freedom is identified, whose behavior permits the individuation of the seismic event's displacement demand.

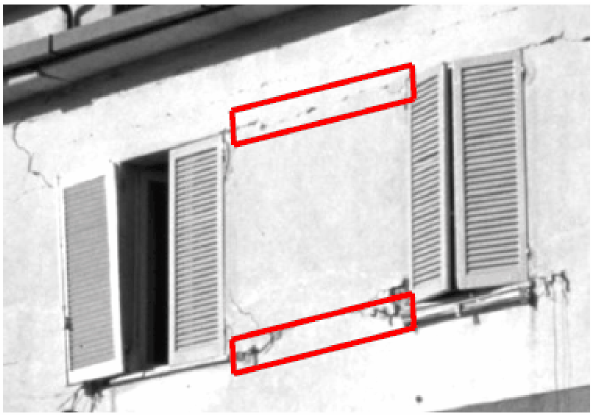
From the observation of masonry buildings damaged by seismic events, two different damage mechanisms emerge:

Shear failure:

Compression-bending failure:

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





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

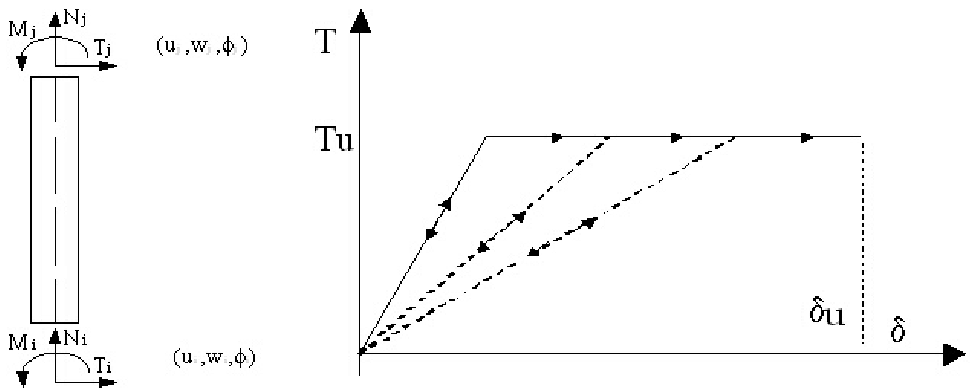
3.2 Masonry Macro-elements

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

1) initial stiffness given by elastic (cracked) properties;
2) bilinear behavior 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 defining maximum drift (δu). The maximum drift can be different for shear or axial bending. The regulations provide for different limit values depending on the failure type.

$$\delta_m^{DL} = \frac{\Delta_m}{h_m} = \delta_u$$

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



Non-linear beam degrading behavior

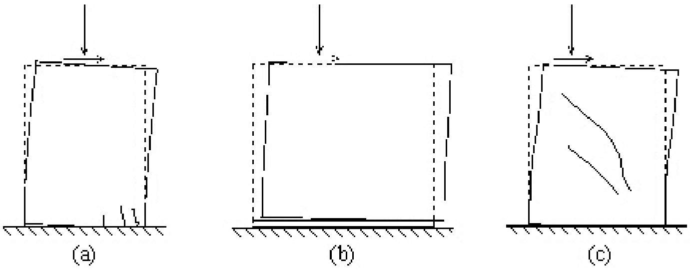
The elastic behaviour of this element is given by:

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

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)

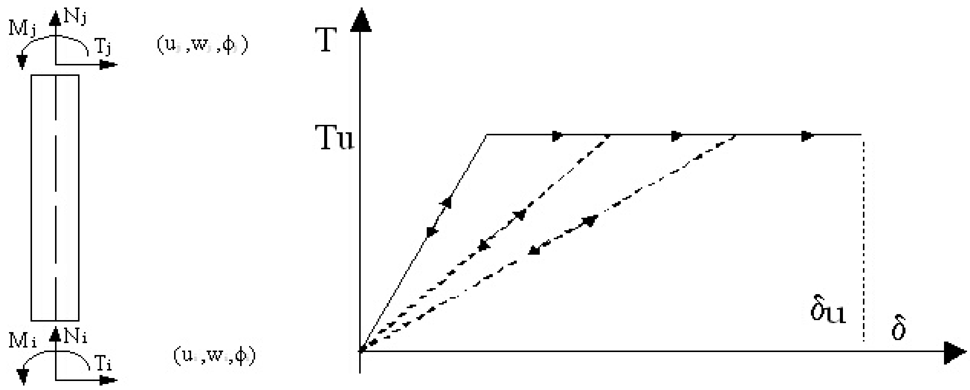
--------------------OLD_TEXT--------------------

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:

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

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



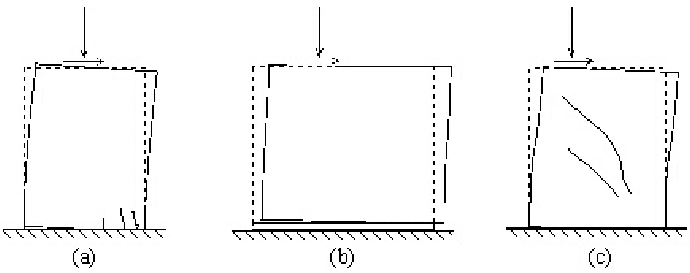
Non-linear beam degrading behavior

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)

3.2.1 Bending: Ultimate moment

The resistance to compression bending can be evaluated by a parabolic curve which relates the normal stress and the ultimate moment, according to the hypothesis of material with no tensile strengh. The ultimate bending moment is defined as:

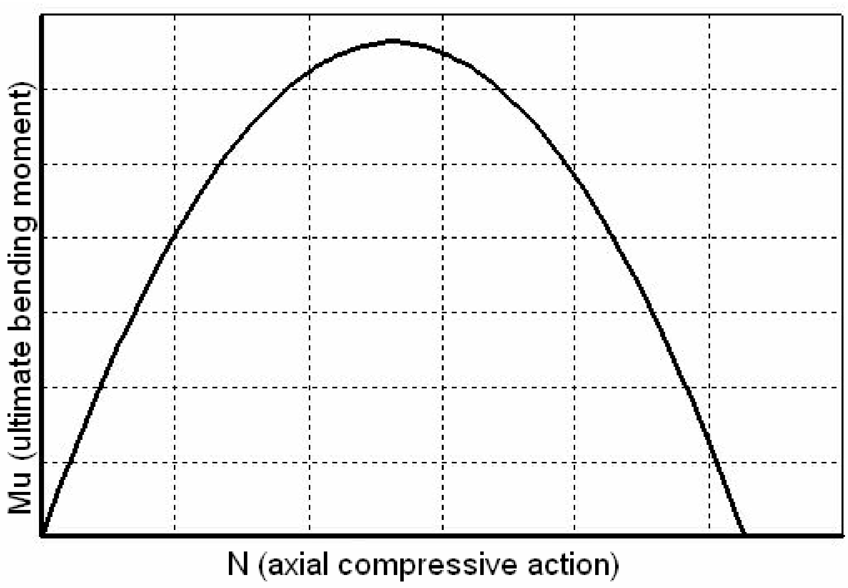
$$M_u = \frac{l^2 t\sigma_0}{2}\left(1 - \frac{\sigma_0}{0.85 f_m}\right) = \frac{Nl}{2}\left(1 - \frac{N}{N_u}\right).$$

Where:
-l is the length of the panel,
-t is the thickness,
-σ_0 is the average compression tension
-N is the axial compressive action (assumed positive in compression)

-N_u is the maximum axial compressive action of the panel an it is equal to 0.85 f_m l t
-fm is the average resistance in compression of the masonry.

This approach is based on a no-tension 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

3.2.2 Shear: Mohr-Coulomb criterion

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

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

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 $V=V_R$

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

and then

$$V_R = \tfrac{1}{2} N \frac{3 f_{vo} l t + 0.8N}{3\alpha f_{vo} h t + N}$$

l' can be expressed as:

$$l'_R = \frac{3}{2}\left(l - \frac{3\alpha f_{vo} l t + 0.8\alpha N}{3\alpha f_{vo} h t + N} h\right)$$

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 $V = V_{lim}$.

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

And so l'_{lim}

$$l'_{lim} = \frac{3}{2}\left(l - \frac{3\alpha f_{v,lim} l t}{3\alpha f_{v,lim} h t + N} h\right)$$

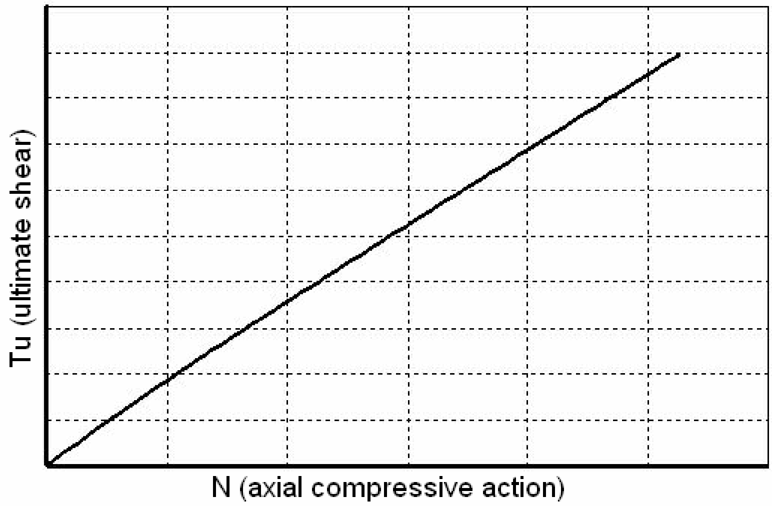
As for l'_R also l'_{lim} must be $\frac{N}{0.85 f_m t} < l'_{lim} \le l$

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:

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



Mohr-Coulomb criterion for shear resistance

3.2.3 Shear: Turnšek 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:

$$b = \begin{cases} 1,5 & h/l > 1,5 \\ h/l & 1 \le h/l \le 1,5 \\ 1 & h/l < 1 \end{cases}$$

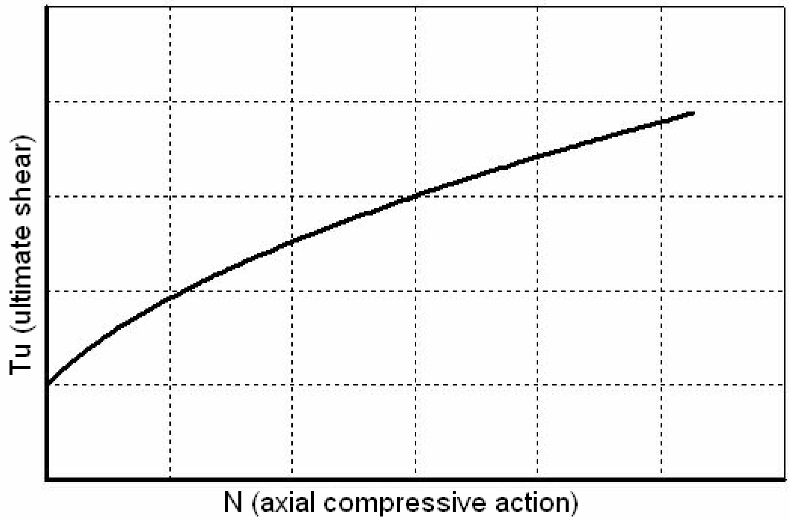


figure 5: Turnšek and Cacovic shear strength criterion

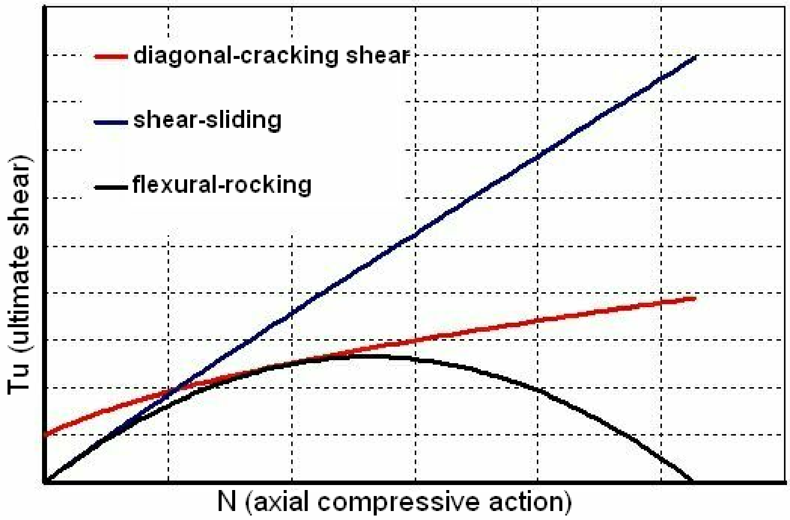


figure 6: Strength criteria comparison

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. In the case that the effective compression of lintels cannot be evaluated, 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.
The maximum resistant moment, associated with to the compression-bending mechanism, 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.
The equilibrium will be provided similar to the one exhibited by the previous criteria.

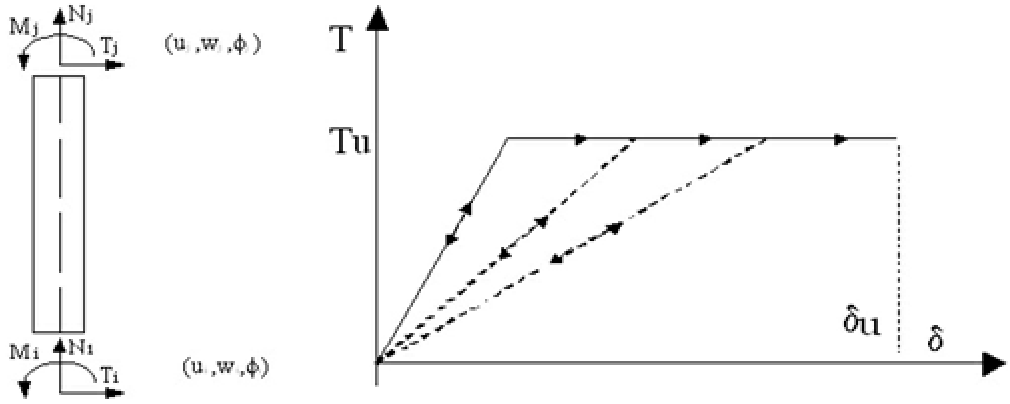
3.2.5 Reinforced masonry beams

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

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

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

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



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

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

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

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

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

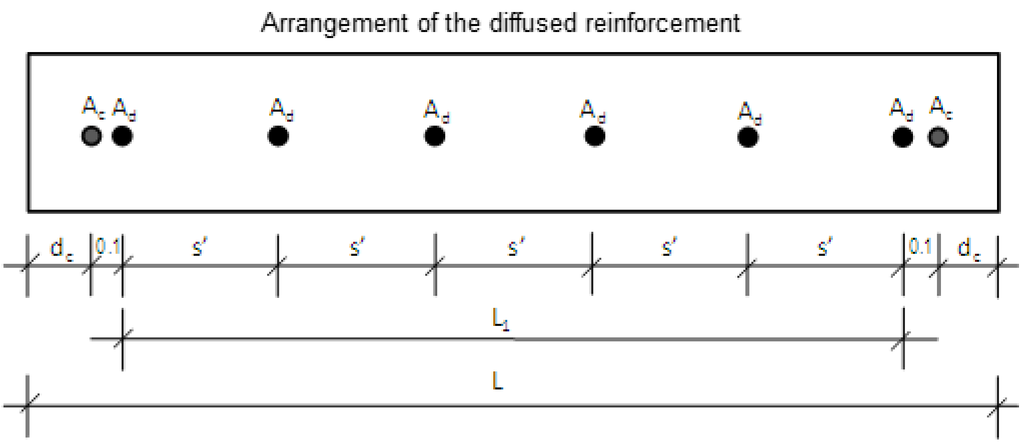
It 'should be specified that in the case of the spandrel beams, since these elements are rotated by 90 ° compared to piers, by reference to the input data assigned to the reinforcement, the transverse reinforcement are those adopted to compute the response to the increased resistance to bending stress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

Clarifications on the treatment of the data provided as input in the case of reinforcements (in relation to the position taken for the reinforcements)
It recalls how in the vertical reinforcement mode, the input data results:
o Ac [m2] : total vertical reinforcement area concentrated to the extreme of the element; this reinforcement is assumed by default arranged symmetrically at both ends of the panel;
o dc [m] : distance from the centroid of the vertical reinforcement concentrated from the extreme of the element;
o Ad [m2]: widespread vertical reinforcement area; It means the total area of the individual reinforcement (including the one disposed to intrados and extrados of the

panel) then placed at intervals sd;
o sd [m] : spacing of diffuse vertical reinforcement.

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

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



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

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

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

compressed masonry and esu = 0,01 for the tense steel.

In the case of using FRP reinforcement, the legislation proposes absolutely similar criteria with the following specifications:

- compared to the range proposed in the CNR-CT 200/2004 document for the depth to be taken for the diagram of masonry compressive stresses (equal to 0.6 ÷ 0.8 x), it is taken as value equal to 0.8 similarly to the case of reinforced masonry;
- the value of the ultimate strain assumed for the reinforcement fiber-reinforced composite is assumed to be equal to the value supplied by the user input. This value can be calculated according to the criteria proposed in the document CNT- DT 200/2004 at § 5.3.2.

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

- *Case I – Mohr- Coulomb criterion*

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

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

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

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

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

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

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

where:
t is the wall thickness

fd is the design masonry compressive strength.
As previously introduced, such resistance criterion requires a truss beam response of the panel whose operation can only be guaranteed by the presence of appropriate vertical reinforcing bars disposed in place.
In the case of using FRP reinforcement, according to what is proposed in the document CNT- DT 200/2004, it has adopted a very similar approach to the one above, for reinforced masonry, taking care to replace the calculation of yield stress' steel (fyd) with the FRP (ffd) of the reinforcement project resistance, defined as the minimum between the rupture tension of the composite and the tension in the composite at which there is the debonding from the masonry.

o *Case II – Turnšek and Cacovic criterion*

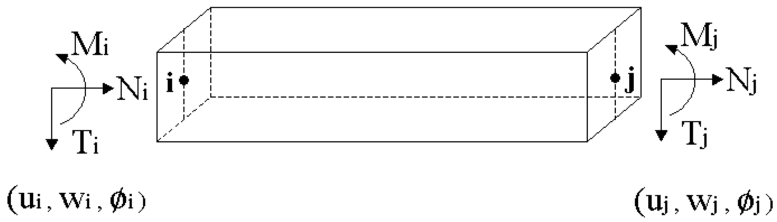
In the event that the values assigned as input for the masonry correspond to a criterion to Turnšek and Cacovic, Technical Rules of Construction (neither the CNT- DT 200/2004 document regarding reinforcement using FRP) does not propose any specific policy to keep account of the strength increase associated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

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

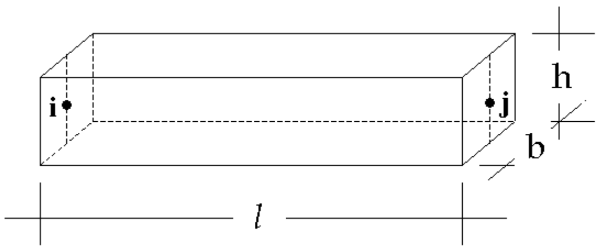
In which, for the meaning of symbols, reference is made to what previously introduced.

3.3 Non-linear R.C. element

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



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



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

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

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

with

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

Elastic rigidity matrix of the R.C. beam element

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

Preliminary observations:

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

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

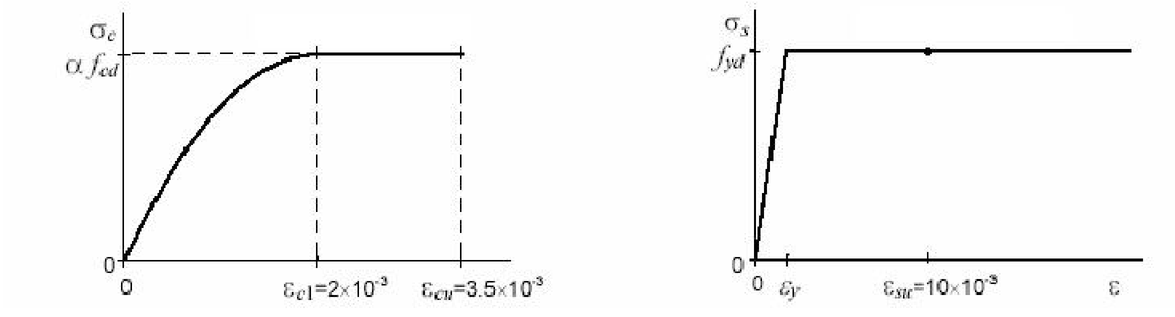
From "Point 8.5 Mixed structures with walls in ordinary or reinforced masonry - OPCM 3274":
In the area of masonry constructions, it is permitted to use structure with diverse technologies to support vertical loads, as long as the resistance to seismic action is entrusted entirely to elements of the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

3.3.1 Resistance Criteria

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

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

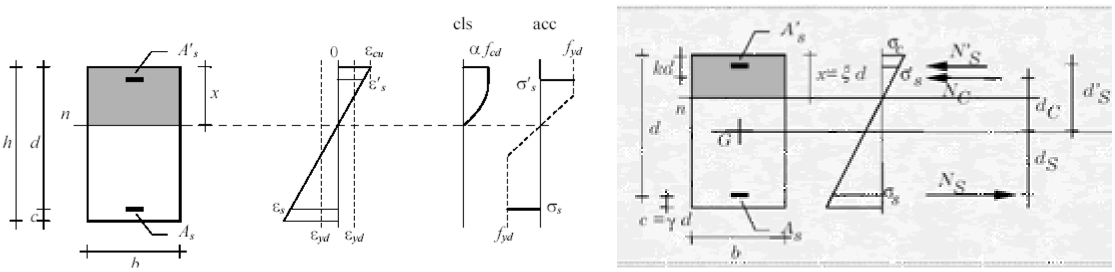
In accordance with what is specified on current legislation, the verification compares the values calculated for the moments with those calculated for resistance (limit values) on the basis of actually existent bending reinforcement.

The M-N domain can be constructed by assigning a failure deformation and determining the deformation diagram. Then, the tension diagram is determined using the constituent links. Finally, the results of compression and traction are calculated N_C , $N_{S'}$, N_S :

$N_S = \sigma s\ As$

$N_{S'} = \sigma s'\ As'$

$N_C = \beta\xi\alpha fc\ bd$



Deformation limit diagram and corresponding tension diagrams

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

$N = N_C + N_S + N_{S'}$ (a)

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

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

Computation of section rotation and collapse

Calculation of section rotation with respect to the chord, to then be compared with collapse rotation, is done with reference to the current legislation:

"Deformative capacity is defined with reference to rotation ("rotation with respect to the chord") $\boldsymbol{\theta}$ 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."

"Rotation capacity with respect to the chord in collapse conditions $\boldsymbol{\theta}_u$ can be evaluated using direct experimentation, numeric modeling considering the contributions of concrete, steel and adherence, or using the following formulas:

$$\theta_u = \frac{1}{\gamma_{el}} 0{,}016 \cdot (0{,}3^{\nu}) \left[\frac{\max(0{,}01; \omega')}{\max(0{,}01; \omega)} f_c \right]^{0.225} \left(\frac{L_V}{h} \right)^{0.35} 25^{\left(\alpha \rho_{sx} \frac{f_{yw}}{f_c} \right)} (1{,}25^{100 \rho_d})$$

where γel=1.5 for primary elements and 1.0 for secondary elements, 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:

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

(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 of the rotation at the chord must be divided by 1.6.
For elements that do not have adequate anti-seismic details, the value of the rotation at the chord must be multiplied for 0.85.
In the presence of plain rebars and insufficient anchorage conditions, the rotation at the chord must be multiplied by 0.575."

Please note that calculation of the collapse rotation is done with exclusive reference to primary elements , 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:

$$V_{Rd} \geq V_{Ed}$$

The shear resistance is evaluated with:

$$V_{Rd} = \max\left\{\left[0{,}18 \cdot k \cdot (100 \cdot \rho_1 \cdot f_{ck})^{1/3} / \gamma_c + 0{,}15 \cdot \sigma_{cp}\right] b_w \cdot d;\ (v_{\min} + 0{,}15 \cdot \sigma_{cp}) \cdot b_w d\right\}$$

With
f_{ck} expressed in MPa
$k = 1 + (200/d)^{1/2} \leq 2$
$v_{min} = 0{,}035 k^{3/2} f_{ck}^{1/2}$
and where
d is the effective section height (in mm);
$\rho l = A_{sl} / (b_w \cdot d)$ is the geometric relation of the tensile reinforcement (≤ 0,02) that has a length at least greater than
(l_{bd} + d) beyond the section considered, where l bd is the anchorage length ;
$\sigma cp = N_{Ed} / A_c$ [MPa] is the mean compression on the section (≤ 0,2 f_{cd});
b_w is the minimum width of the section (in mm).

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:

$$b_{wn} = b_w - \frac{1}{2}\sum Ø$$

where $\sum Ø$ is calculated for the most unfavorable level.
To verify conglomerate that is compressed obliquely, it is possible to use:

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

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

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

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

3.3.3.2.2 Transversal core reinforcement check

The shear resistance VRd capacity of structural elements with specific shear reinforcement must be assessed on the basis of an appropriate Framed schematization. The resistant elements of the ideal truss are: the transverse reinforcement, the longitudinal reinforcement, the compressed stream of concrete and struts of the angled core.

The shear resistance is defined:

VRd = min (VRsd, VRcd)

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

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

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

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

3.3.4 Non-linear behavior of reinforced cement elements

The beam elements in reinforced cement are based on a non-linear type correction. This starts from the elastic prediction, which compares the calculated forces with the resistance limits which follow from the above-mentioned criteria.
Relative to the bending resistance mechanism, plastic hinges are formed when the resistance moment is reached. This limits the capacity to transmit bending forces when the ultimate rotation is reached.
The beam remains in the elastic field until either one of the two ends reaches the limit moment. This check is performed for both sections.
If, for example, at the end *i* of the element 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 the end. The moment at the limit, *j* while still in the elastic field must be balanced with the current displacement condition of the element *i* in which the plastic hinge section is found. So, it is no longer that which was provided by the initial elastic prediction based on 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 at end *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 the linear 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 the end *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 the end where the plastic hinge is formed.
Depending on the various possible situations, the surrounding conditions selected when using the elastic 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 and φi, 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.

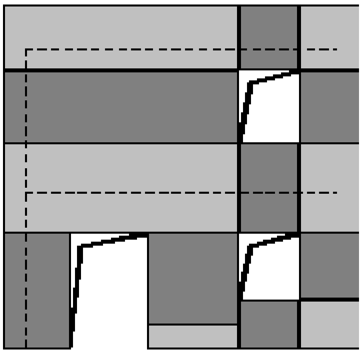
At this point, when the bending moment at the ends of the element have been correctly computed the next step is the rotation check. This is calculated with respect to the cord identified in the section at the zero moment, with respect to the ultimate rotation calculated according to that indicated in 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 the other end are calculated in accordance with the new static schema for the beam. This means for the end in which the bending collapse which became a plastic hinge occurred.
To sum up, the conditions which can occur in each end section are 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 the element 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 the element 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 the element. They can vary during the analysis, following redistribution of the actions towards the elements which contribute together to the total equilibrium of the structural system.

3.4 Three-dimensional Modelling

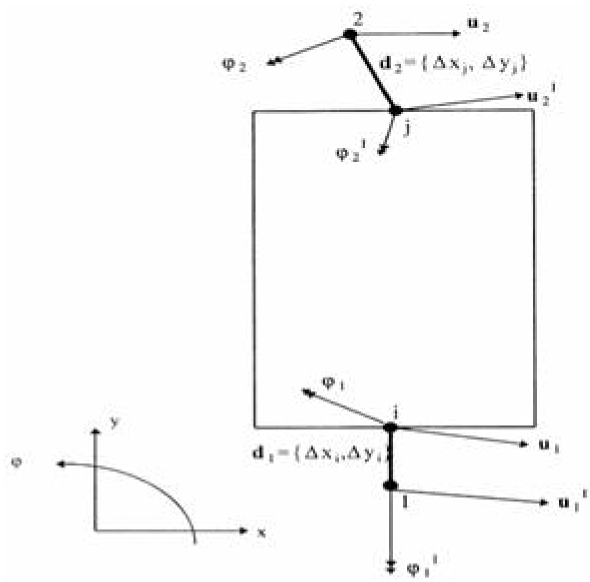
The three-dimensional modelling used is the direct result of observation of real building behavior and experimental tests. These allowed the introduction of some hypotheses about structural behavior of masonry constructions.
As mentioned above, damage mechanisms observed in buildings can be divided into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

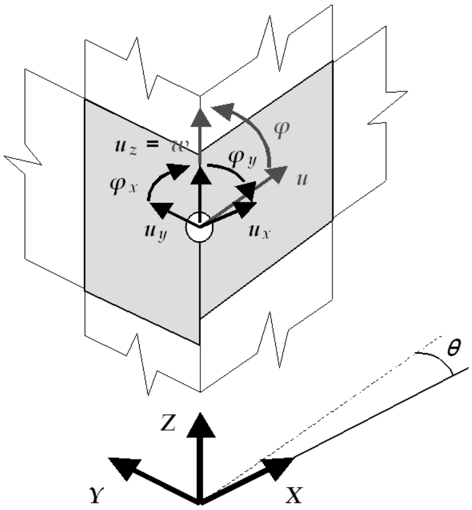
The rigidity matrix for elements without bending rigidity is easily found by eliminating all the limits that contain J from the element matrix. To manage the non-linearity, all of the elastic contributions due to the tie rods must be kept distinct. At each step, it must be verified if the tie rod that 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 modeling, 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 modeled by assembly of the level structures: the walls and the horizontal structures, both lacking bending rigidity outside of the level. The procedure for modeling macro-elements for masonry walls which receive forces from their own level was illustrated above. This instrument constitutes an important starting point for modeling of the overall behavior, based on the behavior of the walls on their level. In any case, extension of the procedure to three-dimensional modeling is not simple. The correct strategy is that of conserving the modeling of the walls on their level and assembling them with the horizontal structures, including those for which the membrane behavior is modeled.
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 modeling 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:

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

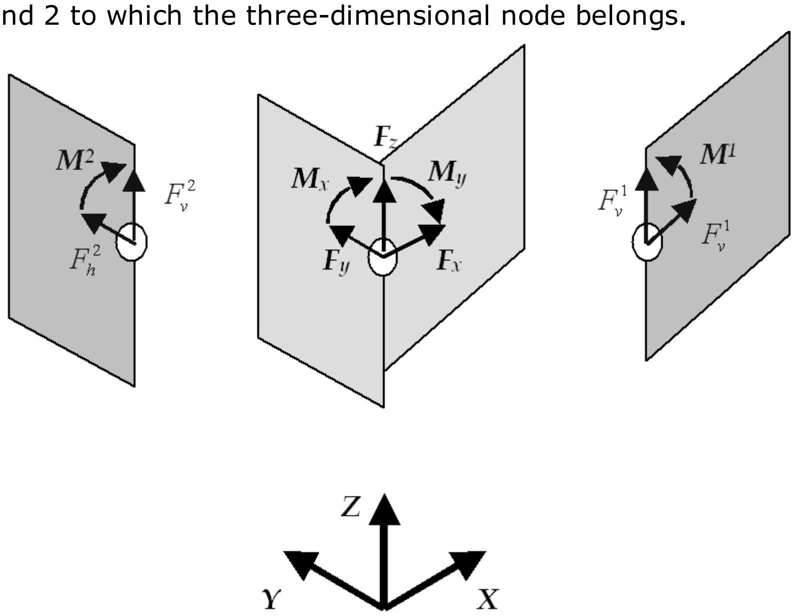
in which u, w, and φ indicate the three displacement components according to the degrees of liberty found in the fictitious node that belongs to the generic wall facing the plan according to angle φ. Similarly, the forces applied to the three-dimensional nodes are displaced according to the directions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

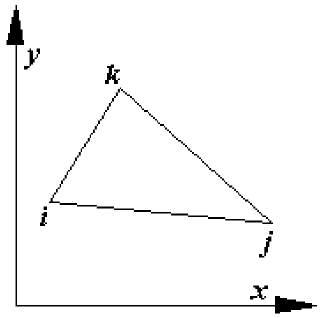
$$\begin{cases} F_x = F_h^1 \cos\theta_1 + F_h^2 \cos\theta_2 \\ F_y = F_h^1 \sin\theta_1 + F_h^2 \sin\theta_2 \\ F_z = F_v^1 + F_v^2 \\ M_x = M^1 \sin\theta_1 + M^2 \sin\theta_2 \\ M_y = -M^1 \cos\theta_1 - M^2 \cos\theta_2 \end{cases}$$

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, modeling 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, modeled 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 ν 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 freedom, that belong to the incident walls at the height of the level of the floor. This added mass is calculated based on the area of influence of each node, taking into account the warping direction of the floor.

4 Reference code

4.1 Europe

The reference seismic code is:

Seismic: **Eurocode 8**

Static: **Eurocode 6**

4.2 Italy

Technical standards for construction- ***D.M. 17 gennaio 2018***

Technical standards for construction - ***D.M. 14 gennaio 2008***

Technical standards for construction on Seismic areas - ***D.M. 16 gennaio 1996***

4.2.1 N.T. - D.M. 17 January 2018

Limit States: The limit states to be considered are the following:
- Limit State of Collapse (LMC)
- Limit State of Safeguarding life (LSL)
- Limit State of Damage (LSD)
- Limit State of Operationality (LSO)

Compared to the previous Technical Standars for Construction of 2008 the main news are:
- Introduction of the Limit State Collapse (LMC): before this Technical Standards, this limit state was not required for masonry buildings.
- Adaptation interventions (NTC2018-§8.4.3.): Some cases are presented in which the verification can be considered overcome even with vulnerability indexes <1.
- Constitutive bonds of the masonry: The limit values of the element drift are updated according to the LMC
- Load distributions (NTC2018-§7.3.4.2.): Introduction of a distribution proportional to the form of several significant modes.

On the S.T.A. DATA YouTube channel there are videos/courses regarding this topic. By subscribing to the channel it is possible to always stay up to date.

Circulaire # 7 January 21, 2019

It is possible to select the calculation method explained on the Circulaire # 7 January 21, 2019, which plays a fundamental role for masonry structures.

Given the innovations introduced by the new circulaire, it is expected to obtain significantly different results.

4.2.2 N.T. - D.M. 14 January 2008

The requirements for this standards show the following peculiarities:

Seismic load: The definition of the spectra by the seismic load, is no longer linked to the zoning but to the geographical coordinates (latitude, longitude), according to what is prescribed by the *"reference grid"* based on the indications given in Annex A of the *Technical Standards.*

Static load on the slabs: For this standard it is necessary to define only the factor ψ2

Limit States: The limit states to be considered are the following (paragraph 3.2.1 of the Technical Regulations):
- Limit State of Safeguarding life (LSL)
- Limit State of Damage (LSD)
- Limit State of Operationality (LSO)

4.2.3 N.T. - D.M. 16 January 1996

According to what is reported in the standard, it is necessary the verification of structural resistance that is equivalent in order to verify that the structure is able to endure the seismic actions foreseen by the regulations.
The program calculates the value of the seismic load for the modeled building and compares it with the maximum load that can be endure by the building corresponding to the peak value of the capacity curve.

The verification will be satisfied according to the following control:
F_h (seismic load required by the standard) < F_u (ultimate load of the building)

4.3 Switzerland

The reference codes are:

➔ **SIA 269/8**
➔ **SIA 2018**
➔ **SIA 266**
➔ **SIA 261**
➔ **SIA 260**
➔ **Eurocode 6:** Static verification

4.4 Netherland

The reference codes are:

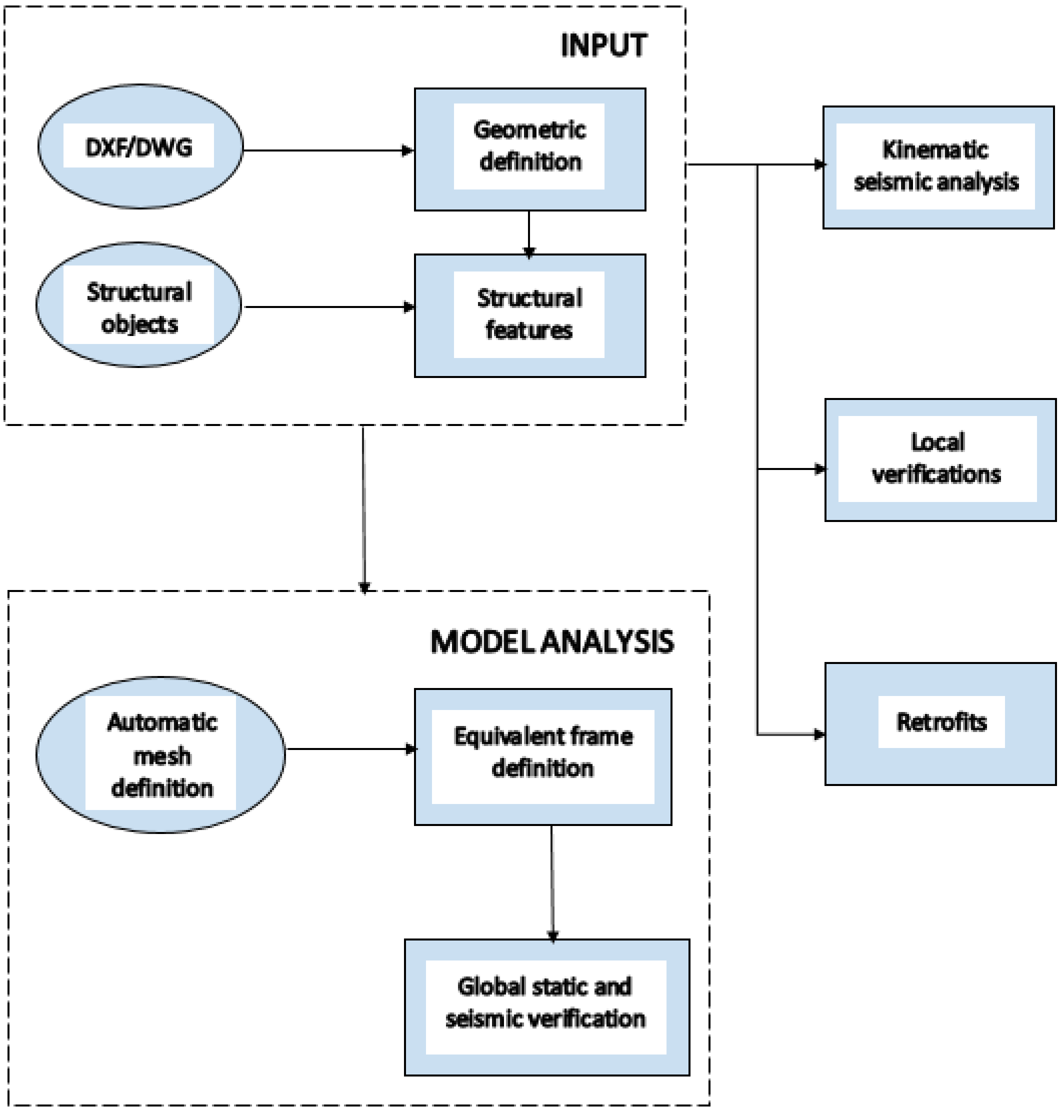
➔ **NPR 9998-2018**

5 General schema of the program

The program 3Muri is a software created to execute the following analysis on masonry buildings:

- Non-linear static seismic analysis
- Analysis of vertical loads
- Kinematic analysis of masonry elements
- Local verification of structural and non-structural elements
- Management tool for retrofits on the structure

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



5.1 Input phase

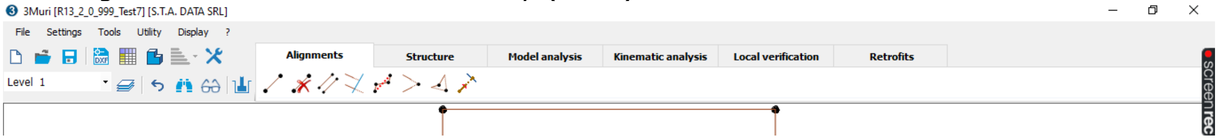
In this phase, the user inserts the data necessary for performing the analysis.

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.

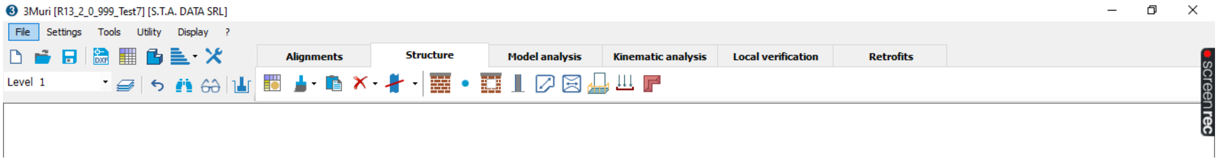
Practical rules for effective importation - Prepare the tables before importing:
- Position the origin of the reference system in one of the vertexes of the plan.
- Define the limits of the graphic area around the plan to be imported (CAD program limits command).
- Delete contiguous designs and images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

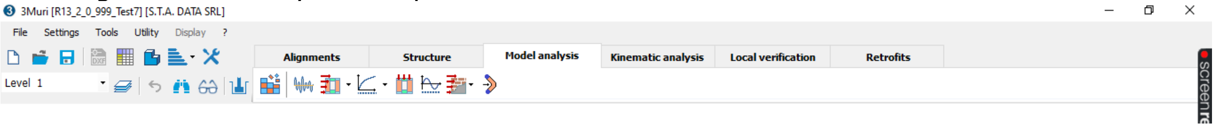
Structural analysis is divided in two phases:
Mesh generation: equivalent frame model is automatically created.

Calculation: The calculation phase is divided is divided into the phases mentioned bellow, depending on the type of calculation that is made.

-Non-linear static analysis (push-over) from which the capacity curve of the structure is obtained (stress-displacement curve of the control point).
-Global static verification
-Bending out of plan
-Sensitivity analysis

Equivalent frame definition
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.

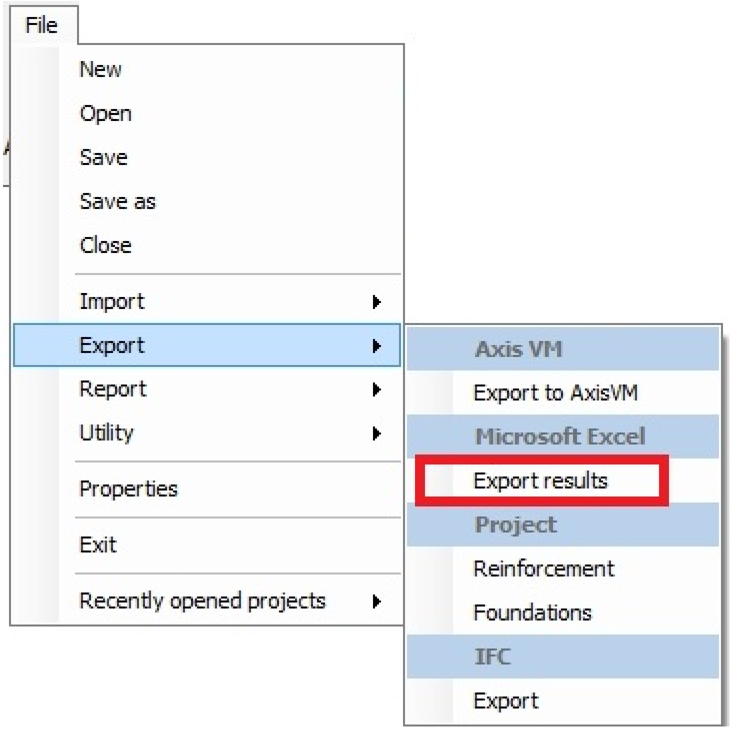
Drawing area for analysis and presentation of results.



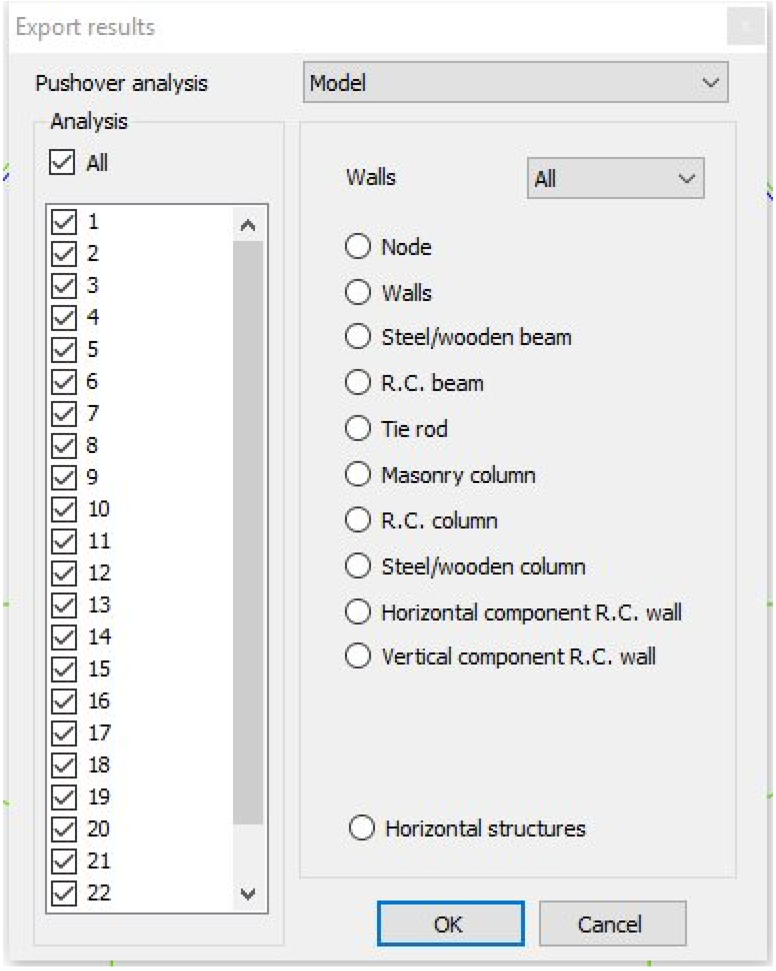
5.3 Export to Excel files

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

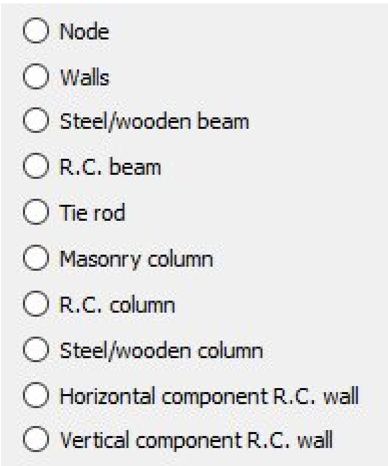
To access this feature you must select the corresponding command in the File menu.



It is presented the following window.



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



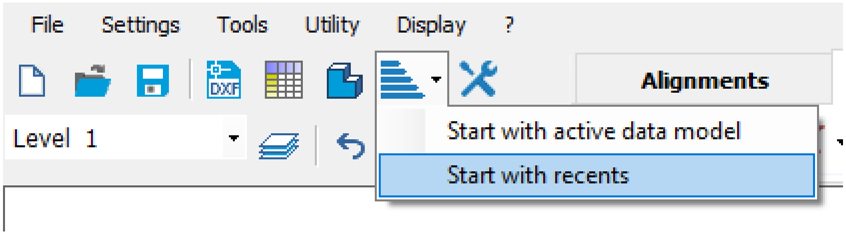
An output Excel file, contains all the required data through the filters during the export phase.

Wall	Step	ID Element	Analisi_1	T1 [N]	N1 [N]	M1 [Nm]	T2 [N]	N2 [N]	M2 [Nm]	Sx1 [m]	Sz1 [m]	R1 [rad]	Sx2 [m]	Sz2 [m]	R2 [rad]	Drift	Analisi_5	T1 [N]
1	0	1		835,15	170185,75	-2164,34	-835,15	-170185,75	-132,32	0	0	0	0	-0,0001	0	0		835
1	0	2		1457,28	171289,81	-197,92	-1457,28	-171289,81	-3809,59	0	0	0	0	-0,0001	0	0		1457
1	0	3		365,07	-17231,55	240,3	-365,07	17231,55	-1317,27	0	-0,0001	0	0	-0,0001	0	0		365
1	0	4		-894,98	104810,26	-2277,19	894,98	-104810,26	4917,39	0	-0,0001	0	0	-0,0001	0	0		-894
1	0	5		2371,03	107153,98	2203,91	-2371,03	-107153,98	-9198,44	0	-0,0001	0	0	-0,0001	0	0		2371
1	0	6		469,54	-16888,43	-362,2	-469,54	16888,43	-1022,95	0	-0,0001	0	0	-0,0001	0	0		469
2	0	7		492,16	84956,22	-429,38	-492,16	-84956,22	-924,07	0	0	0	0	-0,0001	0	0		492
2	0	8		2242,06	185086,94	6146,58	-2242,06	-185086,94	-12312,24	0	0	0	0	-0,0001	0	0		2242
2	0	9		-4790,36	99096,4	8458,25	4790,36	-99096,4	5673,32	0	-0,0001	0	0	-0,0001	0	0		-4790
2	0	10		-574,3	51437,38	1863,86	574,3	-51437,38	-169,66	0	-0,0001	0	0	-0,0001	0	0		-57
2	0	11		2098,23	125285,96	16334,47	-2098,23	-125285,96	-22524,27	0	-0,0001	0	0	-0,0001	0	0		2098
2	0	12		176,46	42585,66	-187,76	-176,46	-42585,66	-276,51	0	0	0	0	-0,0001	0	0		176
2	0	13		99,22	102913	-1039,61	-99,22	-102913	778,57	0	0	0	0	-0,0001	0	0		99
3	0	14		-2825,5	183200,36	-19751,99	2825,5	-183200,36	27522,12	0	0	0	0	-0,0001	0	0		-282
3	0	15		-1031,03	71859,89	-1085,1	1031,03	-71859,89	3920,43	0	0	0	0	0	0	0		-1031
3	0	16		-2278,94	119807,24	-17700,26	2278,94	-119807,24	23967,35	0	0	0	0	0	0	0		-2278
3	0	17		-3630,67	122897,49	-23358,6	3630,67	-122897,49	34069,09	0	-0,0001	0	0	-0,0001	0	0		-3630
3	0	18		-889,23	42048,42	-1253,23	889,23	-42048,42	3876,46	0	0	0	0	-0,0001	0	0		-889
3	0	19		-688,12	57170,11	-16387,58	688,12	-57170,11	18417,53	0	0	0	0	-0,0001	0	0		-688
5	0	20		-5583,83	208369,49	9168,09	5583,83	-208369,49	7304,2	0	-0,0001	0	0	-0,0002	0	0		-5583
5	0	21		647	198567,07	-11875,47	-647	-198567,07	9966,81	0	-0,0001	0	0	-0,0002	0	0		6
5	0	22		13,21	55726,14	-22,39	-13,21	-55726,14	-12,98	0	0	0	0	-0,0001	0	0		13
5	0	23		-1104,34	197307,98	-1482,39	1104,34	-197307,98	4438,7	0	0	0	0	-0,0001	0	0		-1104
5	0	24		730,05	193491,78	82,19	-730,05	-193491,78	-2036,53	0	0	0	0	-0,0001	0	0		730
5	0	25		-35,73	52175,67	17,82	35,73	-52175,67	77,83	0	0	0	0	-0,0001	0	0		-35
1	1	1		-164503,96	98723,23	290124,29	164503,96	-98723,23	162261,59	0	0	0	0,0002	0	0	0,0001		5588

Walls

5.4 Seismic classification

From the main menu of the program it is possible to start SismoTest, the module dedicated to the seismic classification of buildings, according to the D.M. n. 58 of 28/2/2017 and subsequent additions.

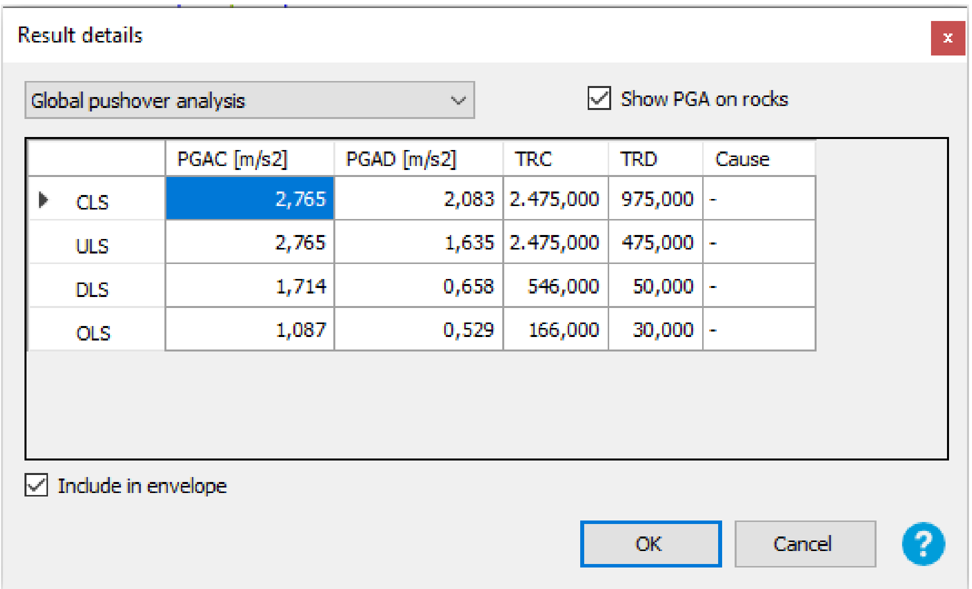


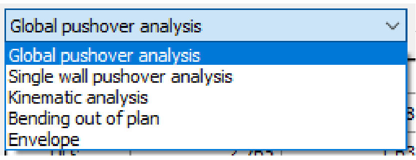
The module can be started in two different ways:

Start with active data model It allows to start the SismoTest module using the results of the analysis carried out on the current model as input.

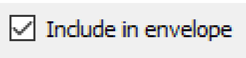
Start with recents Allows you to start the SismoTest module by manually entering the input data (not considering the results of the analysis carried out on the model).

In the case of "Start with active data model" it is necessary to choose which analysis to consider for the seismic classification.
Using the following window it is possible to view, for each analysis carried out, the results obtained:



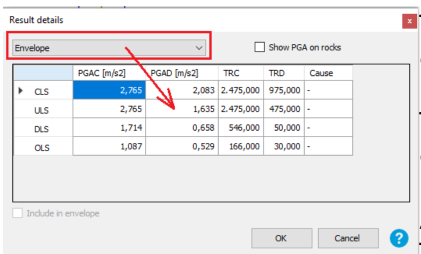


Allows you to select the type of analysis for which you want to view the results obtained.



If the "Include in envelope" box is checked (for the considered analysis) the relative results will be included in the choice of data shown in the "Envelope" configuration.

E.g.:
I check the "Include in envelope" option for the "Global Pushover" and "Kinematic Analysis" analyzes.
By selecting the "Envelope" option, I will find in the table the data deriving from the envelope of the results of the global and kinematic pushover analyzes.



The "Envelope" configuration allows you to view which data will be used as input for the seismic classification.

This configuration is obtained by enveloping the results obtained from the analyzes carried out on the model.

ATTENTION:
To include the results of an analysis within the envelope it is necessary to check the "Include in envelope" option present for each type of analysis.

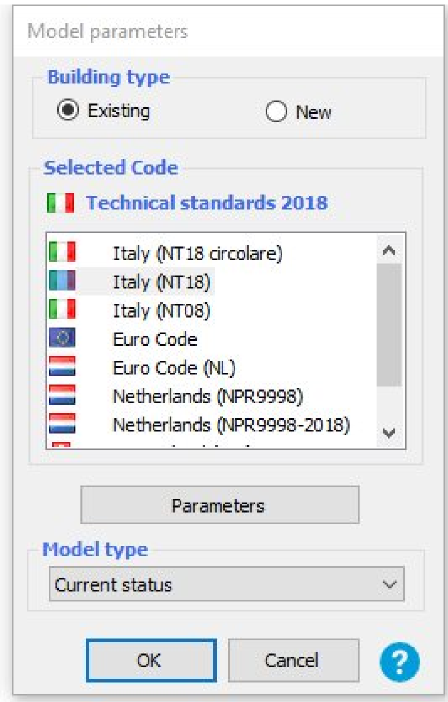
Once the input data has been established, after clicking on "OK", the SismoTest module is started.

6 Basic concepts for using the program

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

6.1 Model parameters

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

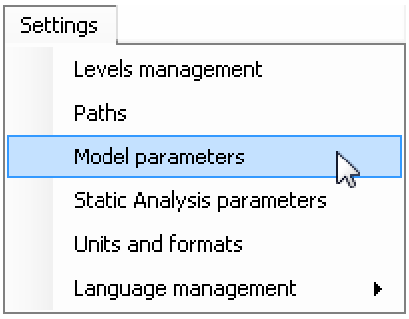


Building type:

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

Selected code:
The user can choose reference code.

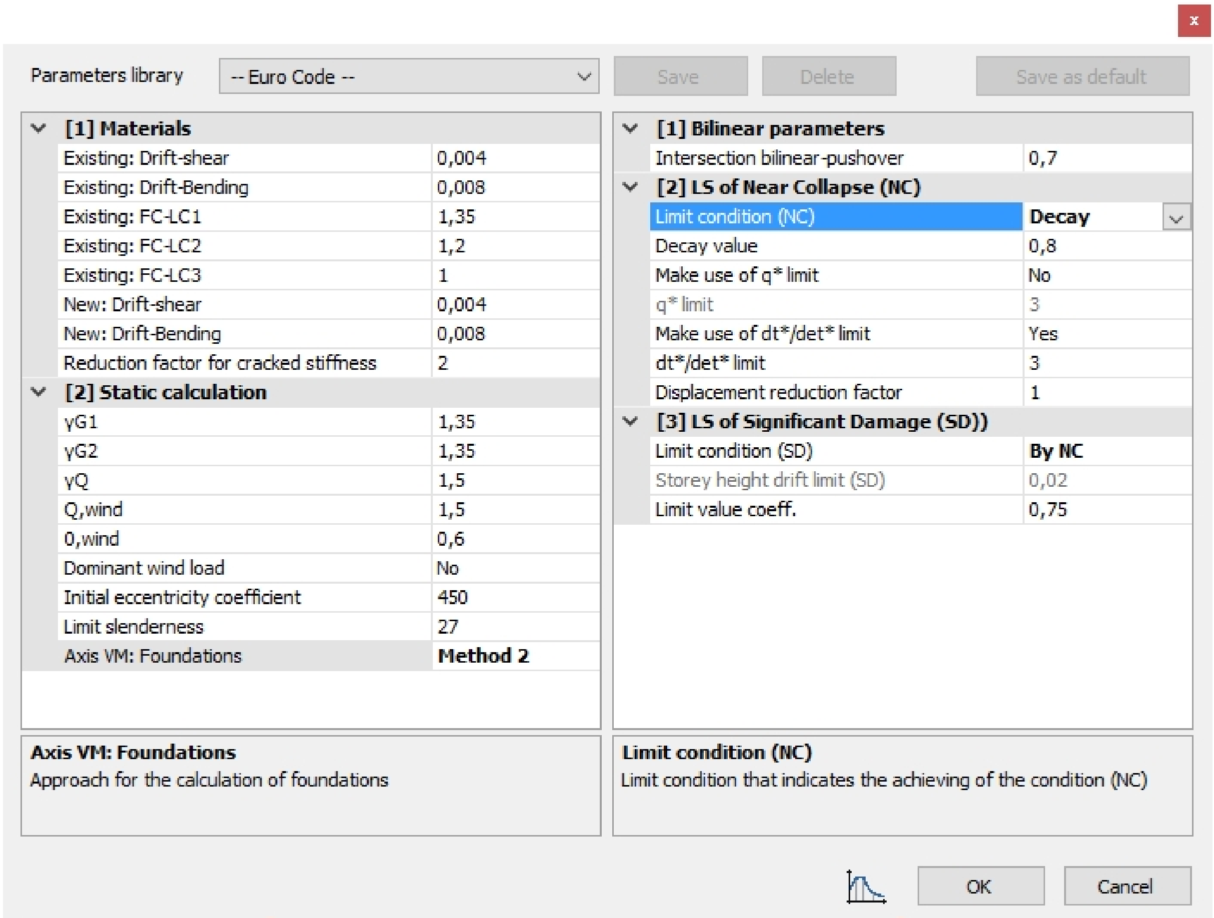
Parameters:
It is possible to modify the calculation settings parameters so in that way you can personalize them.



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

Calculation parameters

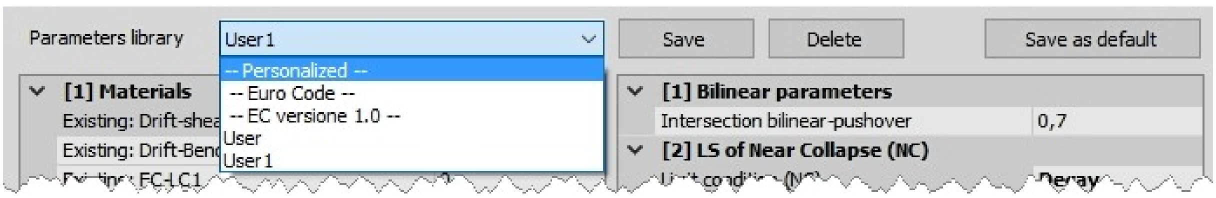
The window that allows the configuration of calculation data is as follows:



The parameters provided in this screen depend on the active standard.
For each standard they are available in the drop down menu several libraries of parameters defined by a <Name>:

1. if that name is NOT enclosed between "-" (<Name>) it means that it is a *project library* and will only be available in the project in which it was created independently of the PC on which the project is open.
2. if that name is enclosed between "--" (--<Name>--) it means that it is a *program library* that will be available for each project calculated with the same rule.

The *project library (1)* is used when you make a change and should only be used for this project.
The *program library (2)* is usually used when we need to change the parameters and use them for all future projects (eg, the national annexes of the Eurocodes, once the parameters are adjusted to those of the state in which they work, will be used for all projects in the same state).

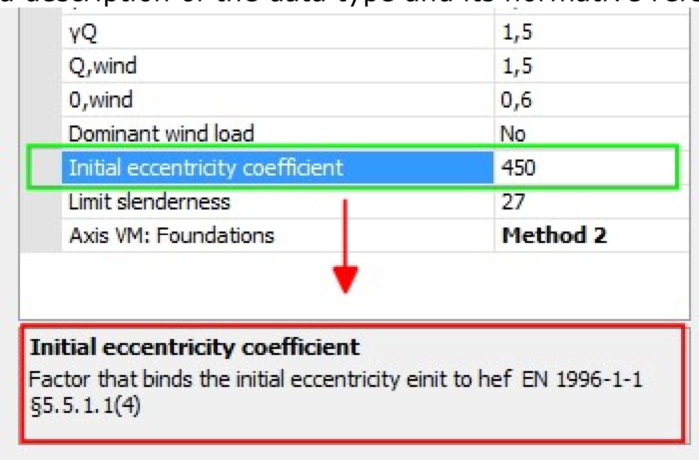


Creating libraries:
When modifying an input parameter, the drop-down moves immediately to --*Personalized*-- to indicate that the parameters are being edited.
To confirm and make the changes active, press the **[Save]** button and enter the name to

assign to the new library.
Confirming the insertion of the library is immediately selected as current.
The newly created library is *"project" type (1)*.
Pressing **[Save As Default]** saves a copy of the library as "Program Library" (2) (the name will be enclosed in "-").
The program libraries present are usually different configurations of the parameters with the same rule, only one of them can be generated by using the **[Save As Default]** command.
Rerun **[Save as default]** on a different *project library (1)* lead to overwrite the *program library (2)* previously created.
Ex. As in the previous figure, select "User02", press **[Save as default]** leads to overwrite "--User01--" which is the only *"program library" (2)* available to the user.

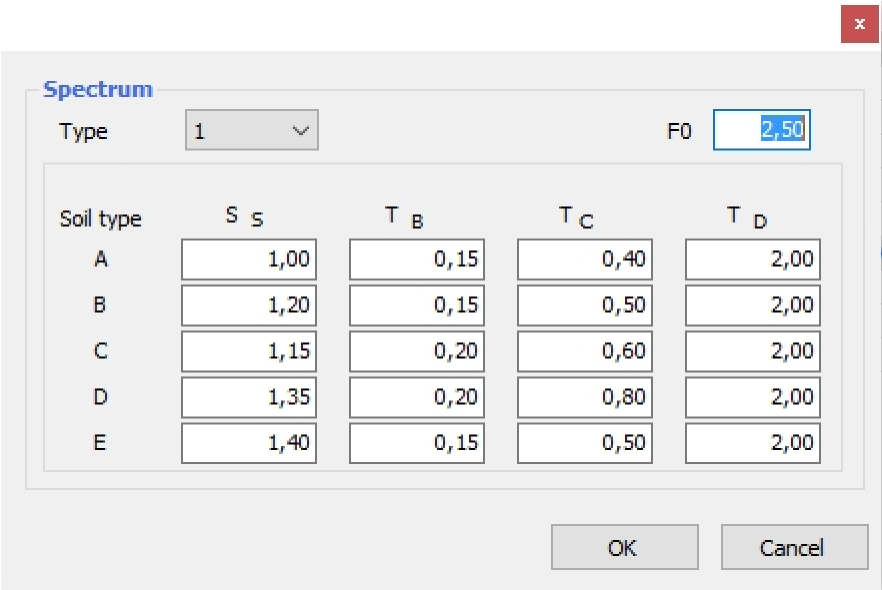
The [Delete] command allows you to delete a *project library (1)* created, *the program library (2)* are not erasable.

Contextual description of the parameters subject to change:
By selecting the field corresponding to a specific value, the bottom part of the grid shows a description of the data type and its normative reference.



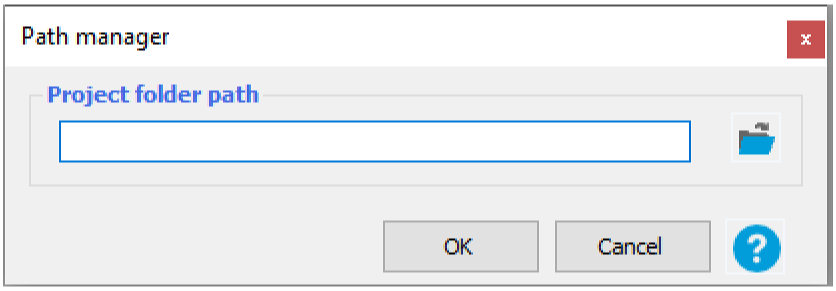
Spectrum (available only with Eurocode standard)

With this button you can access the table containing the parameters for the definition of the seismic spectrum depending on the soil classes.
Ms: magnitude value.
F0: maximum value of the amplification factor of the accelerated spectrum (usually assumed to be 2.5).



6.2 Path Selection

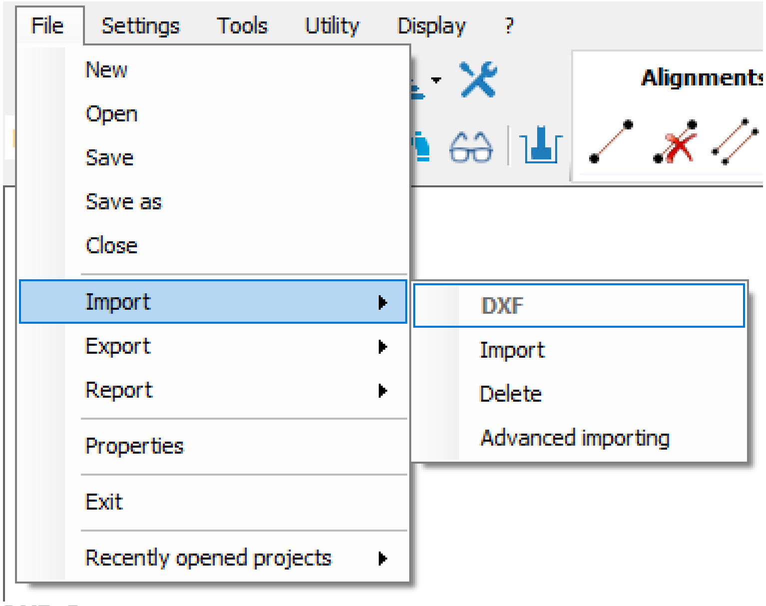
The project paths can be managed by 3Muri.



"Project folder path" indicates the path where projects created by the user are saved.

6.3 Import

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



DXF>Import
Allows the import of A DXF file

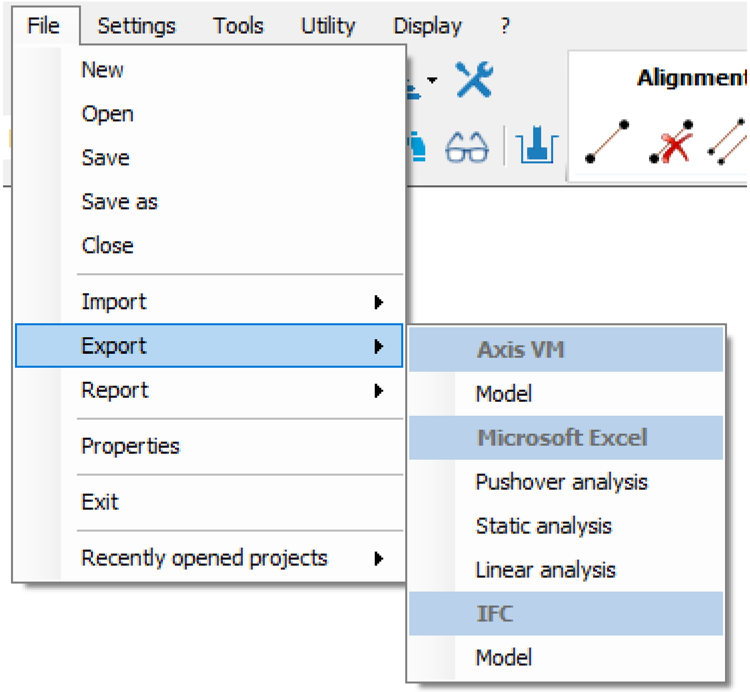
DXF>Delete
Allows deleting of previously imported DXF files

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

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

6.4 Export

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



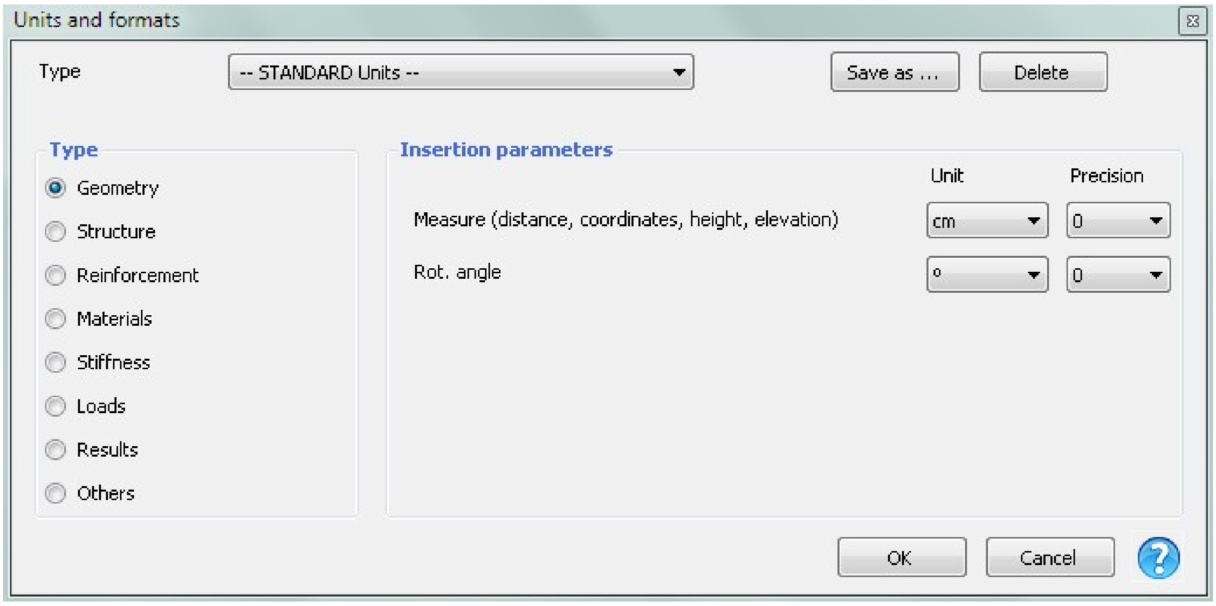
Axis VM>Export model
Allows the export of a model created with the 3Muri program to AxisVM (FEM solver).

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

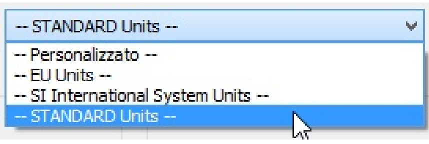
IFC>Export
A model created in 3Muri can be exported in IFC format.

6.5 Units and formats

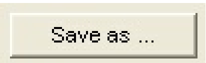
Settings > Units and Formats



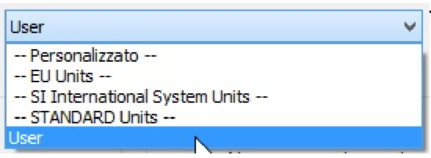
It allows to configure the units (SI and/ or English system) and formats of the variables used on the program (number of decimal used for the visualization or exponential format).
It's possible to use default settings, or create and save the personalized settings.



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



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

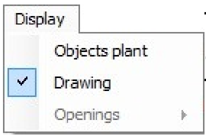


The name of the new "Unit schema" appears in the bottom of defaults schema.

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

6.6 Support graphic entities

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



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

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

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

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

 Copy: copies a graphic element.

 Move: moves graphic entities.

 Fillet: allows to fillet to lines.

 Extend: Extend a line out another.

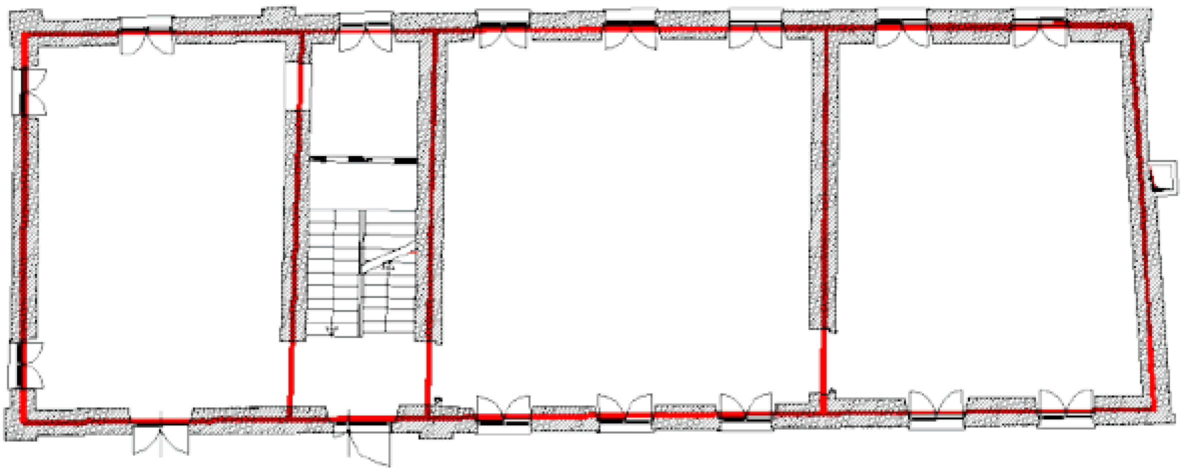
 Trim: truncates two intersecting lines

 Delete: deletes graphic entities

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

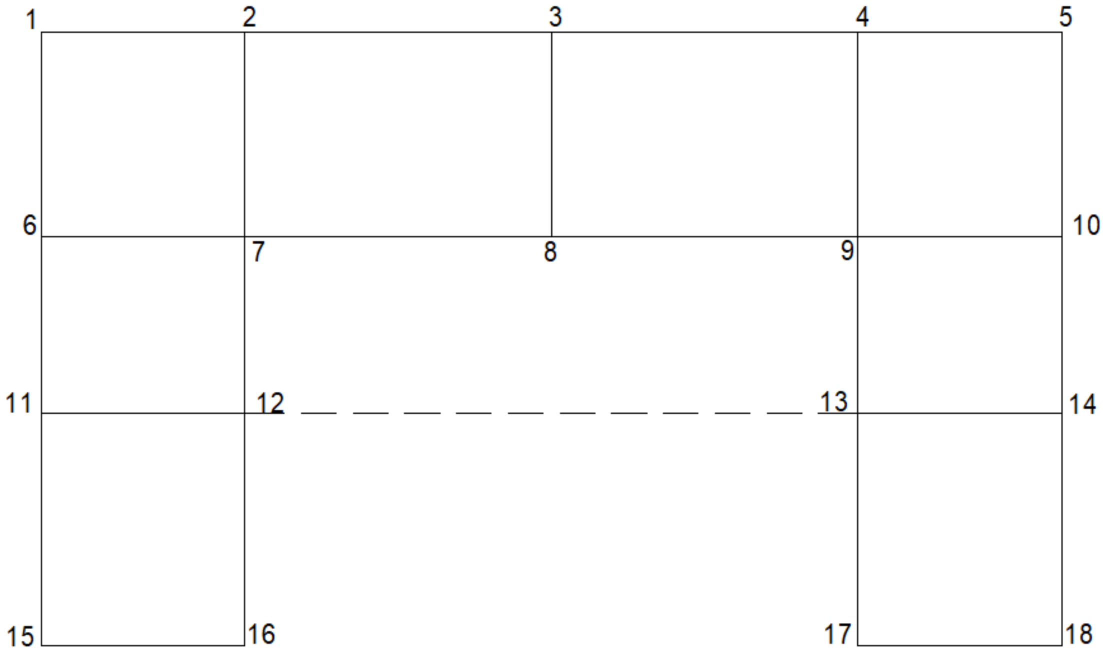
6.7 Walls

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 modeled, 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).



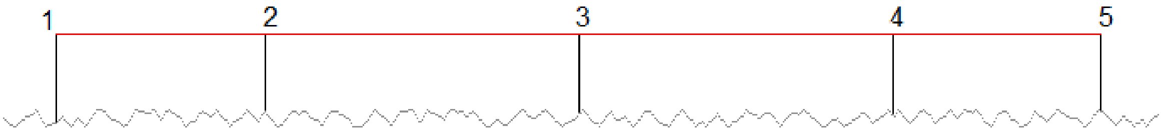
Exploding the wall system it becomes clear why various contiguous segments with structural environment definitions, belonging to the same tangent, must be modeled using a single wall. If wall segments do not have definition in the structural environment on any level, then in place of a single wall, multiple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

Following are some examples that clarify what has been said. Taking as a reference the underlying scheme, suppose that our building is composed of two levels.



- Level 1: the wall panel starts from node 1 and ends at node 5 (without interruption)
- Level 2: the wall panel starts from node 1 and stops at node 3 and then starts from node 4 up to node 5.

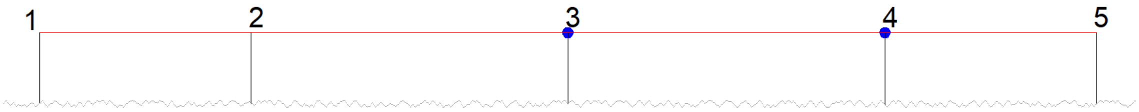
A correct modeling provides the design of a single wall (as shown in the image below) that will be



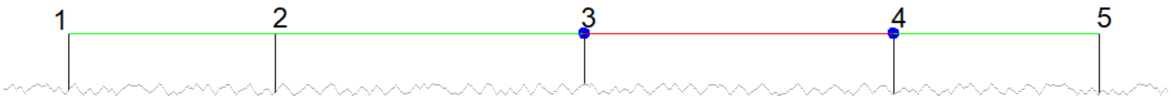
After drawing the wall, you can assign its properties to it.
In the case of level 1 it is simple because the wall panel is continuous.
In the case of level 2, an additional step is required because the wall panel is not continuous but c

In this case it is necessary to:

1) Within the structure environment, insert points 3 and 4 of the additional "element nodes" in such a way as to be able to define different characteristics to the various sections

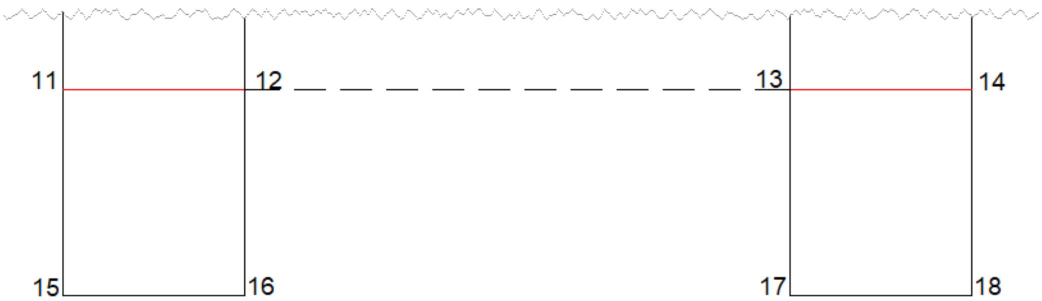


2) Assign the characteristics to the walls 1-3, 4-5

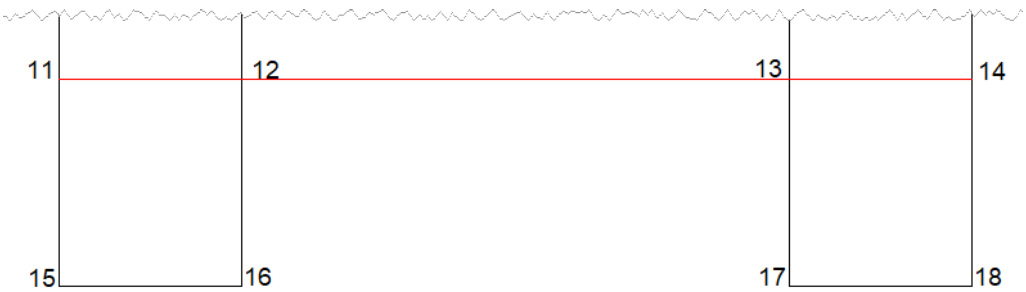


If we want to report another example, we take as a reference the alignment from node 11 to node

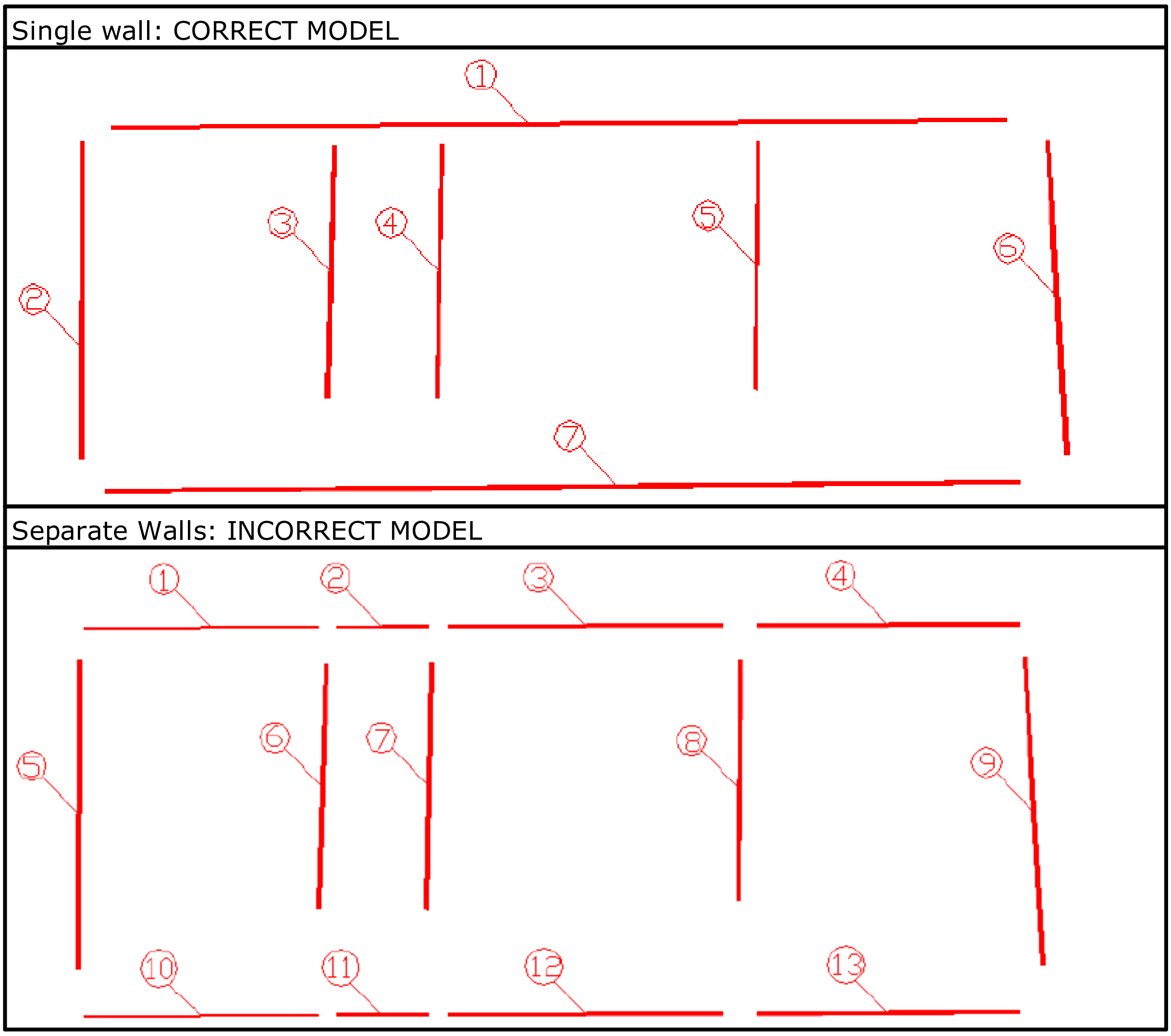
RECOMMENDED MODELING



NOT RECOMMENDED MODELING



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.

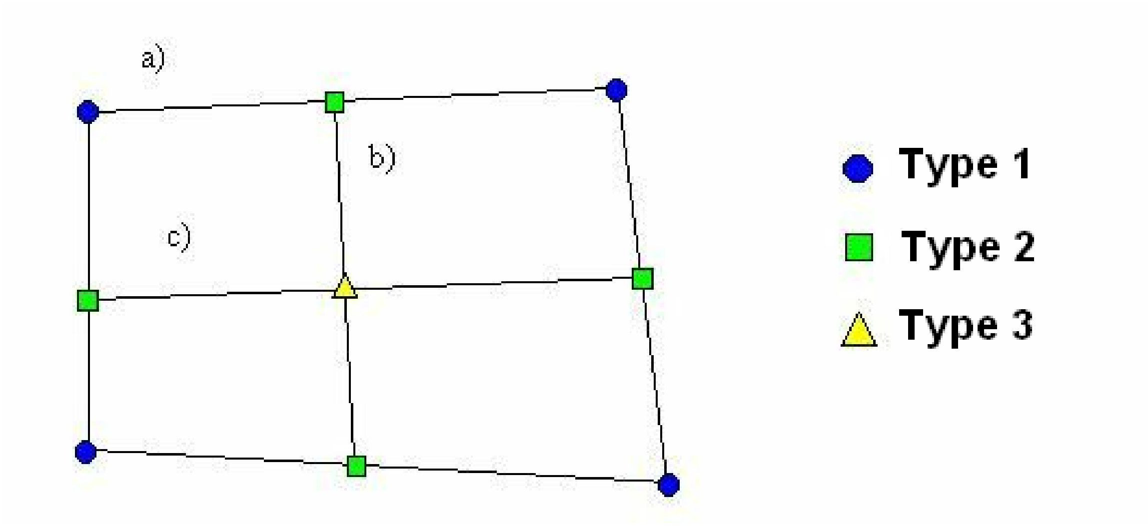


The walls can be managed on all levels, and can be deleted, added to, or modified in all design phases.

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

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

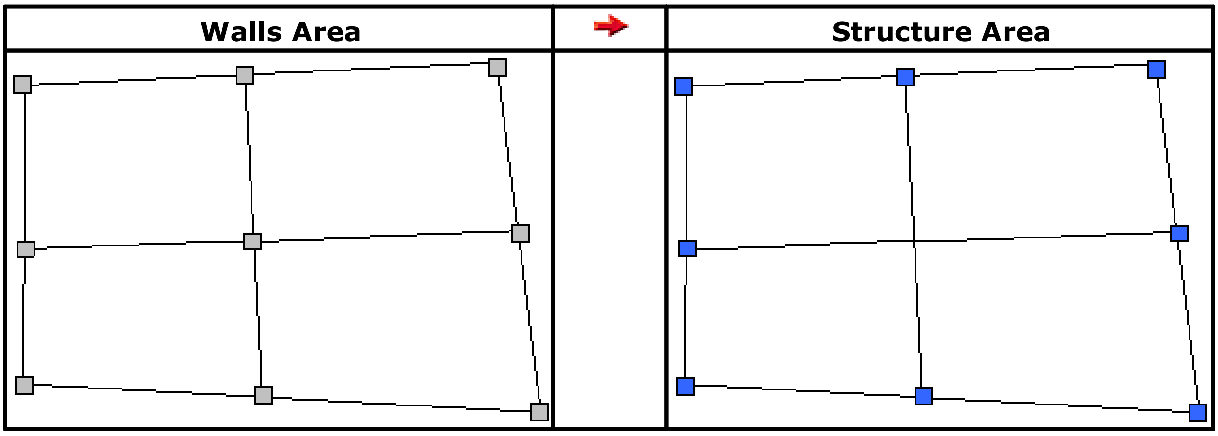
During the insertion phase, a third type of node can be created. This is automatically derived from the computation of the intersection between walls. For example, between the intersection of walls b) and c).
These TYPE 3 nodes (which are indicated with a yellow triangle) are found in an intermediate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



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

6.8 Structure

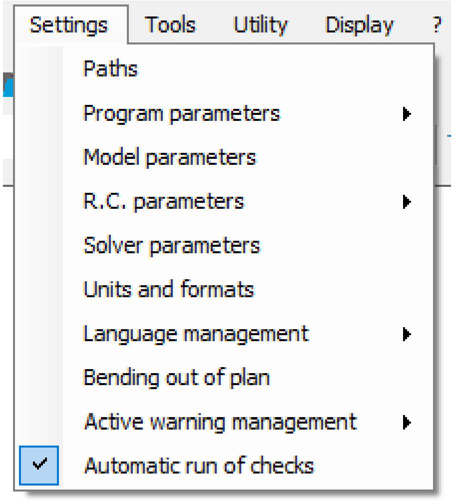
In the Structure area, the walls can be "dressed" with structural objects such as masonry, columns, beams, tie rods, and R.C. walls.
When the Structure area is activated, all the walls are transformed into segments which become objects that can be "dressed." Each wall can be divided into segments by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



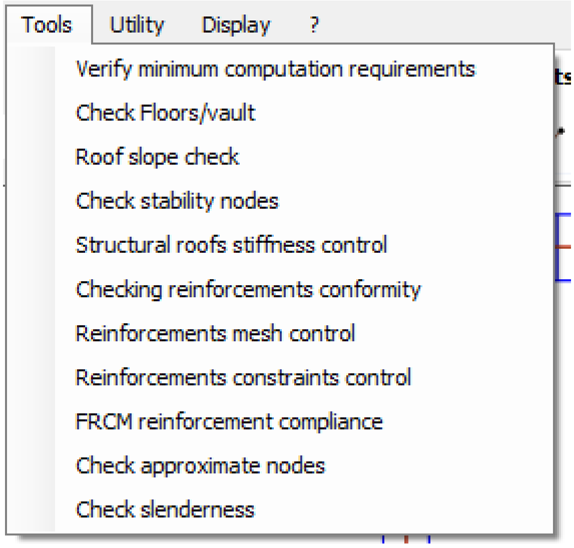
A column can be inserted only in correspondence with a wall endpoint or segment point. In the case being considered, to insert a column in correspondence with the intersection of the two internal walls of the structure, it is necessary to insert a segment point.

6.9 Model Checking

During the model creation phase, disorganized support graphics or a simple human error can lead the designer to make involuntary mistakes. To overcome these shortcomings the program comes with an automatic procedure that controls the fulfillment of the basic rules of model creation.
This correction procedure can be made automatic from the "settings" menu by selecting the "automatic run of checks" option.



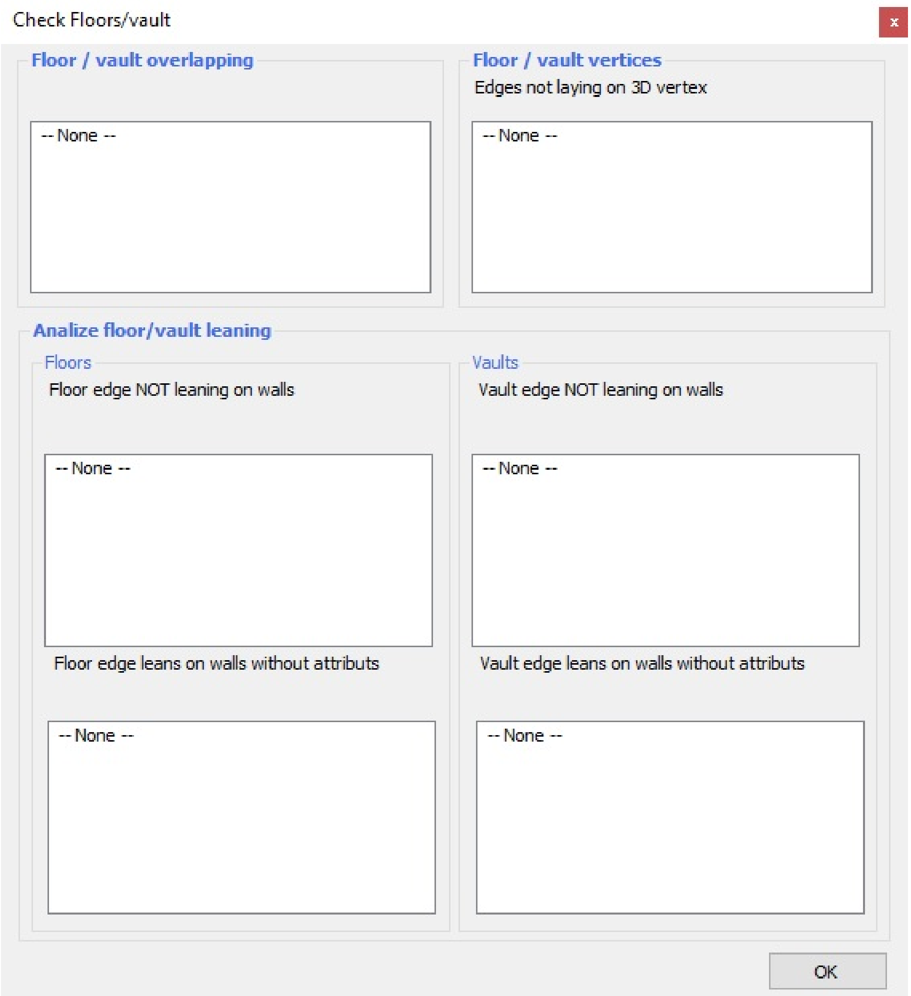
From the "tools" menu it is possible to perform these checks manually:



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

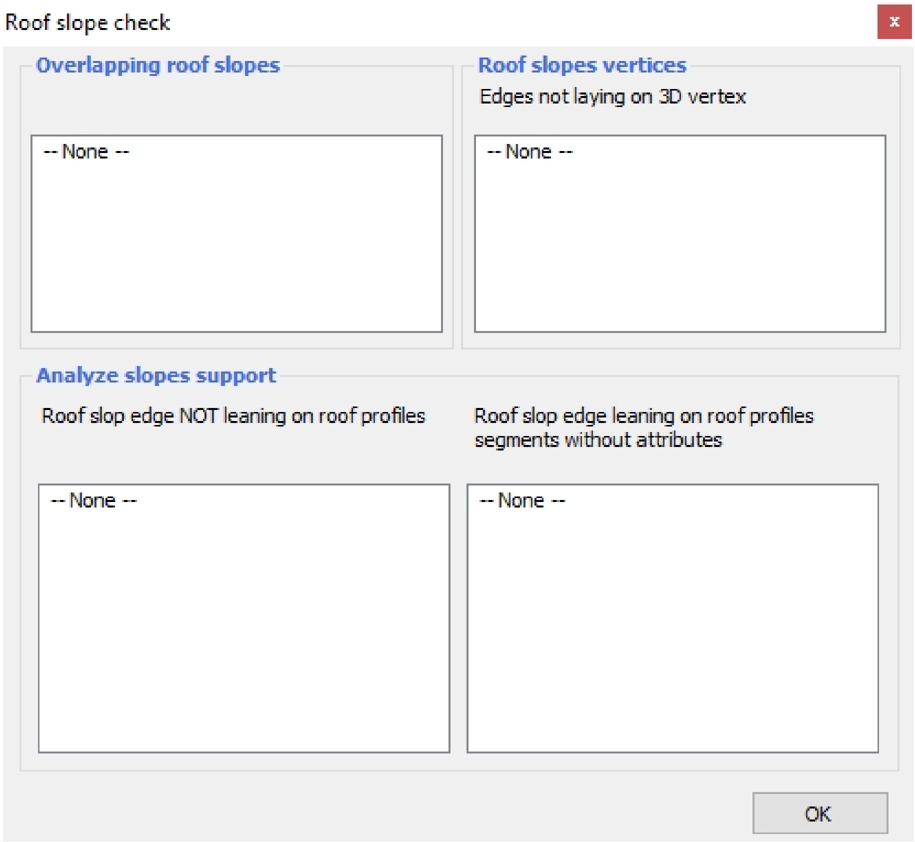
Check floors/vault overlapping:
This command checks for the presence of overlapping floors to avoid the insertion of more than one floor on the same plan.

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



Check overlapping roof slope:
Checks for overlapping roof slopes to prevent the user to enter more roof slopes on the same level.

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



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

Structural roofs stiffness control:
This check is carried out on pitches belonging to a structural roof and is passed when the stiffness of the pitches is different than zero ($Ey \neq 0$). This control does not include pitches with bracing elements.

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

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

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

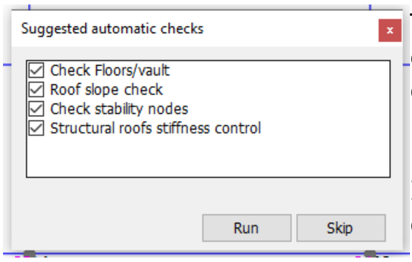
FRCM reinforcement compliance:

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

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

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

The "Automatic run of checks" option predisposes that in the significant points (eg the generation of the mesh), the program remembers which and how many checks it is good to conduct.
This operation is performed with an opportune mask that appears on the screen.



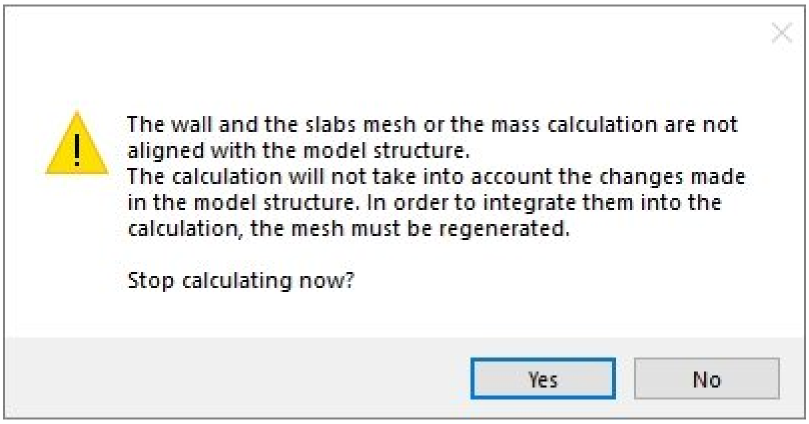
The list of proposed checks is prepared according to the characteristics of the model, in order to show only the checks of interest.

[Run]: Run all the checks selected with the checkbox.
If some checks are not of interest, they can be excluded directly by unchecking the corresponding box.

[Skip]: No checks are performed and it goes directly to the next step.

Mesh update

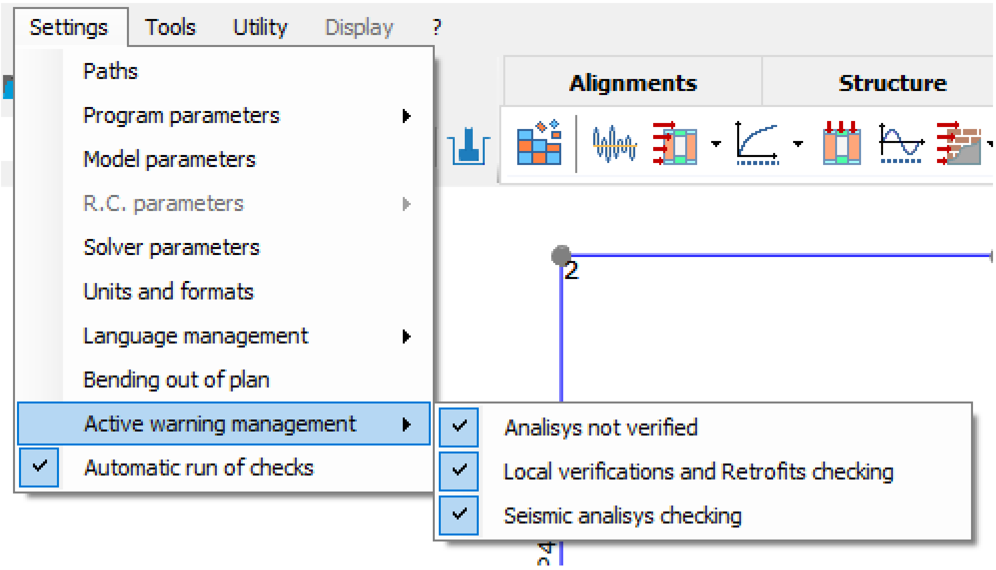
The mesh used in the calculation phase is generated starting from what is modeled in the structure environment, so each time changes are made in the structure it is therefore necessary to regenerate the mesh.
If the user passes to the calculation after making changes without having regenerated the mesh, the following information message is shown.



By answering [Yes] it is possible to perform the mesh with the appropriate command, or return to the model modification phase if the request to perform the calculation was made by mistake.
Answering [No] it proceed with the calculation with a NOT aligned mesh to what is imputed in the structure.
Clearly this is not a recommendable choice, however there are some limited cases in which it can be useful (e.g. rerun the calculation with a different control node, temporarily neglecting the changes made in the structure).

6.10 Active warning management

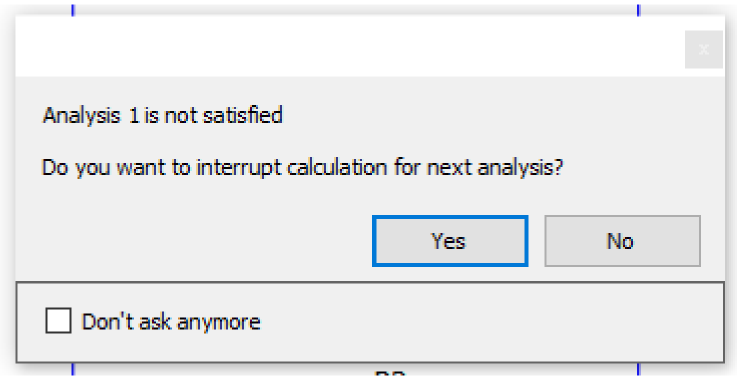
From the "settings" menu, selecting the "Active warning management" option, it is possible to choose which alerts to activate.

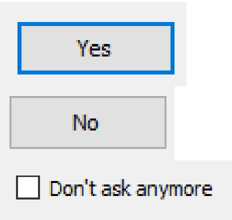


Analysis not verified

Activate the warning about any unverified analysis.

By activating the warning, in the case of unverified analysis, the program presents the following window that notifies the analysis failure.



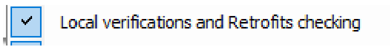


Selecting this option will not proceed with the calculation of the remaining analysis.

Selecting this option will proceed with the calculation of the remaining analysis.

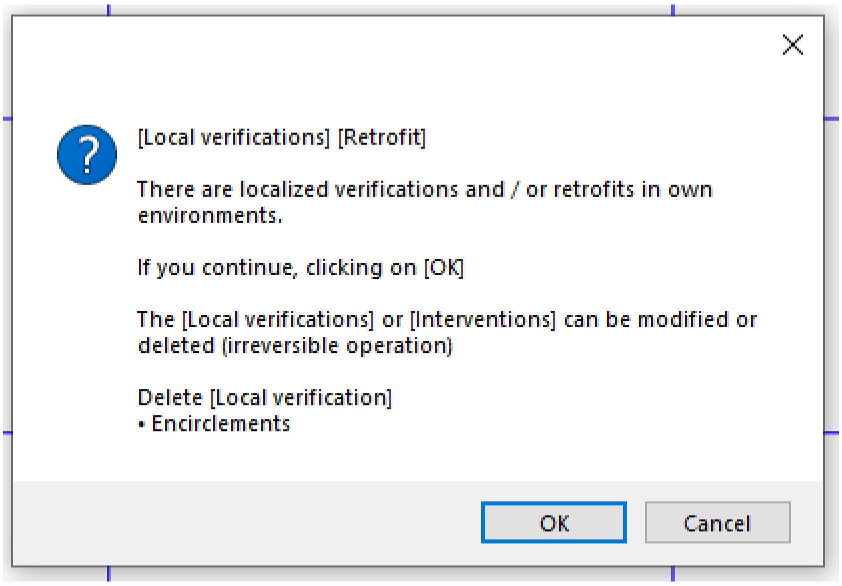
By selecting this option, the program will no longer notify any warnings relating to the failure of the analysis.

Local verifications and retrofits



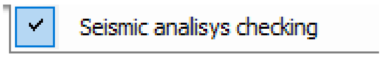
Activates the warnings relating to the calculation cards created within the "Local verification" environment and to any "Retrofits" applied to the elements of the model.

The alerts are structured as shown in the following image:



Depending on the calculation sheet created, following changes made on the structural objects, the program warns that the calculation sheet will be modified / deleted.

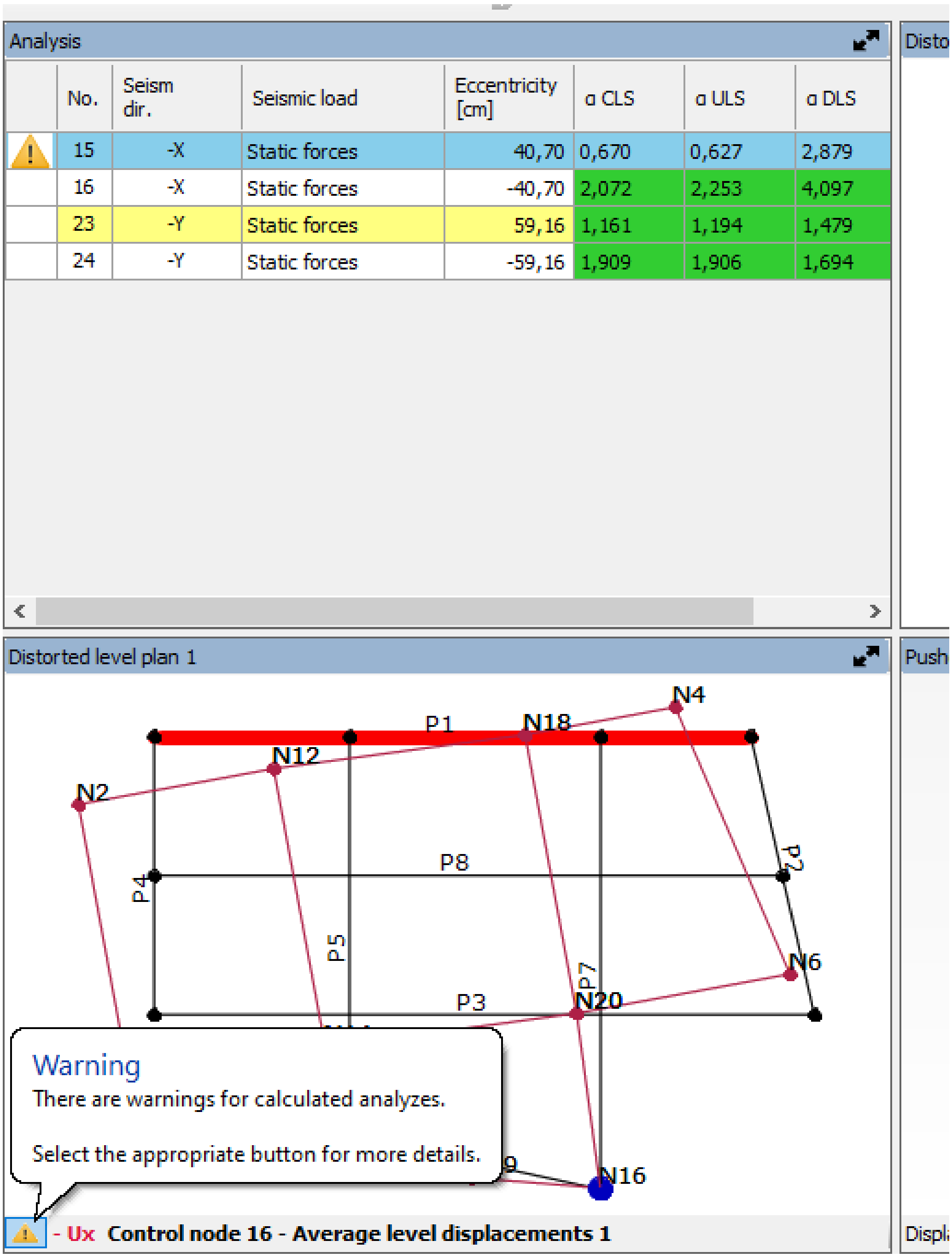
Seismic analysis



Activates the warnings related to the checks carried out during the calculation of a pushover analysis (no convergence, participant mass check, number of curve steps etc ...).

In the list of analyzes carried out, the symbol  will appear indicating the presence of warnings relating to the analysis.

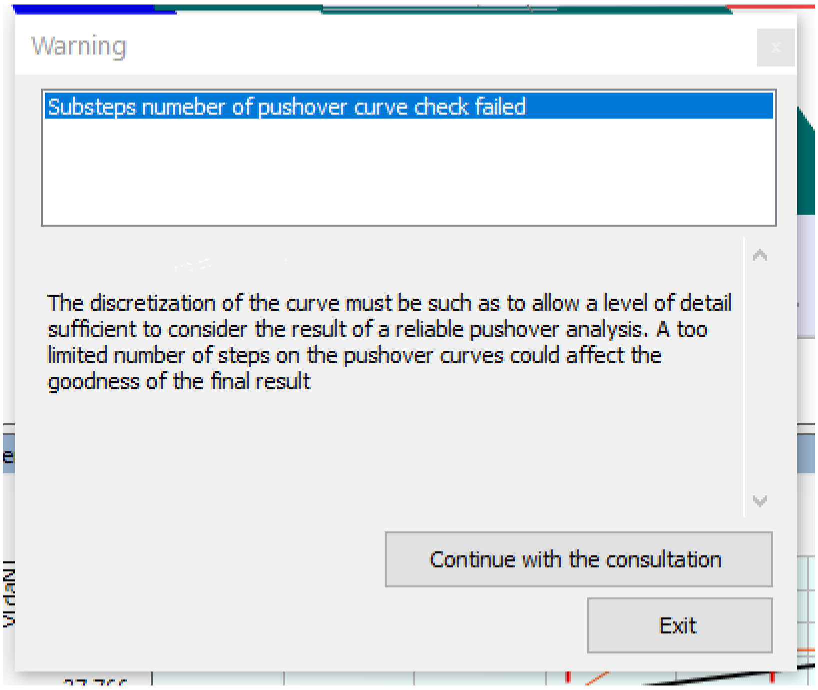
Also, at the bottom left, a pop-up message will appear notifying the presence of alerts for which the details can be visualized.

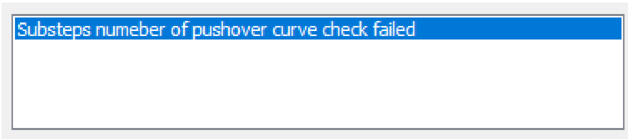


By clicking on the button  you can access the details window.

6.10.1 Warning detail

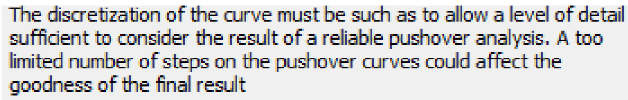
The window relating to the details of the checks carried out is as follows:



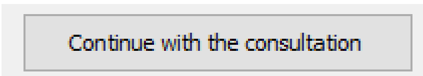


In the upper part of the window is presented the list of checks that the selected analysis did not pass.

By clicking on each row it is possible to view the details relating to each individual control.

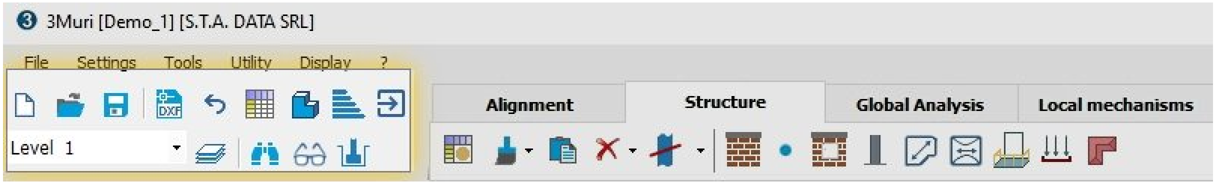


In the central part, after selecting a specific check, a brief explanatory message will appear specifying what the control consists of and, in some cases, what could be the reasons that led to its failure.



Using this button it is possible, depending on the control, to access descriptive parts rather than further detail windows useful for understanding and analyzing the results obtained.

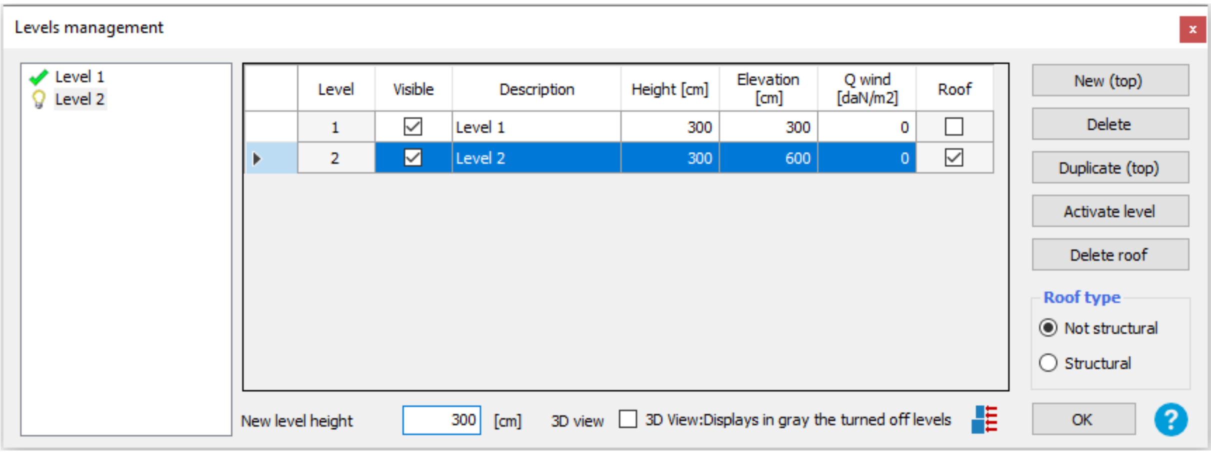
7 Main commands



	New model
	Open
	Save
	Import Dxf
	Undo
	Elements Table
	3D View
	Seismic classification
	Maintenance Plan
	Switch to local interventions
	Levels Properties
	Find
	Display options
	Foundations analysis

7.1 Levels management

This command permits management of the levels of the structure.

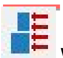


New (top)	Insert new empty level. The level will be inserted above the selected layer. ***[Default height]*** *allows to define the height of the new levels created.*
New (bottom)	Insert new empty level. The level will be inserted bellow the selected layer. ***[Default height]*** *allows to define the height of the new levels created.*
Delete	Delete level. You can delete both the top and bottom levels.
Duplicate	Duplicate previous level.
Activate level	To modify a level, it can be activated with the "Activate level" button.
Delete roof	Delete a roof

[Roof type] Allows to decide whether the roof on the level highlighted in the table is to be considered as "not structural" or "structural".

[ **/**  **]** The visualization of a single level can be deactivated (Hide level) by clinking on the light bulb icon.
[3D view: displays in gray the turned off levels] *allows to show the "hidden" levels in gray on the 3D view even if the are deactivated.*

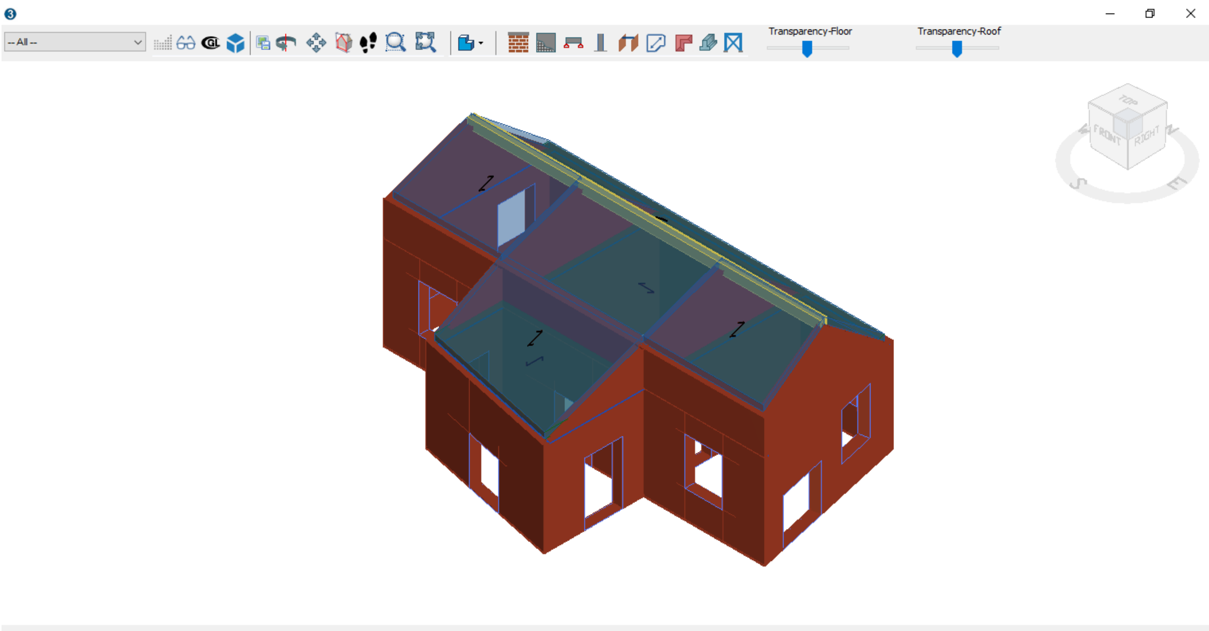
On the table, other parameters can be found:
- Wind loads (average per level). This value is necessary only if static checks are to be performed.
- A check box that informs whether or not the roof is present for a given level.

 **Wind load calculation**

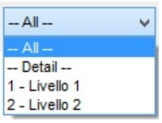
It allows the wind load analysis according to Italian standards. (Only available for Italy)

7.2 3D View

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

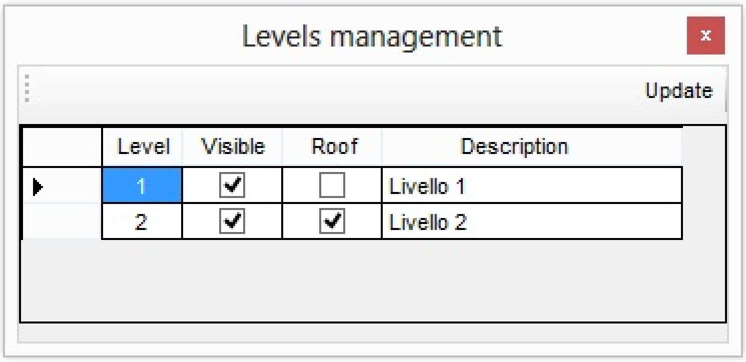


The stories' view filters can be managed by means of a drop down menu:



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

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

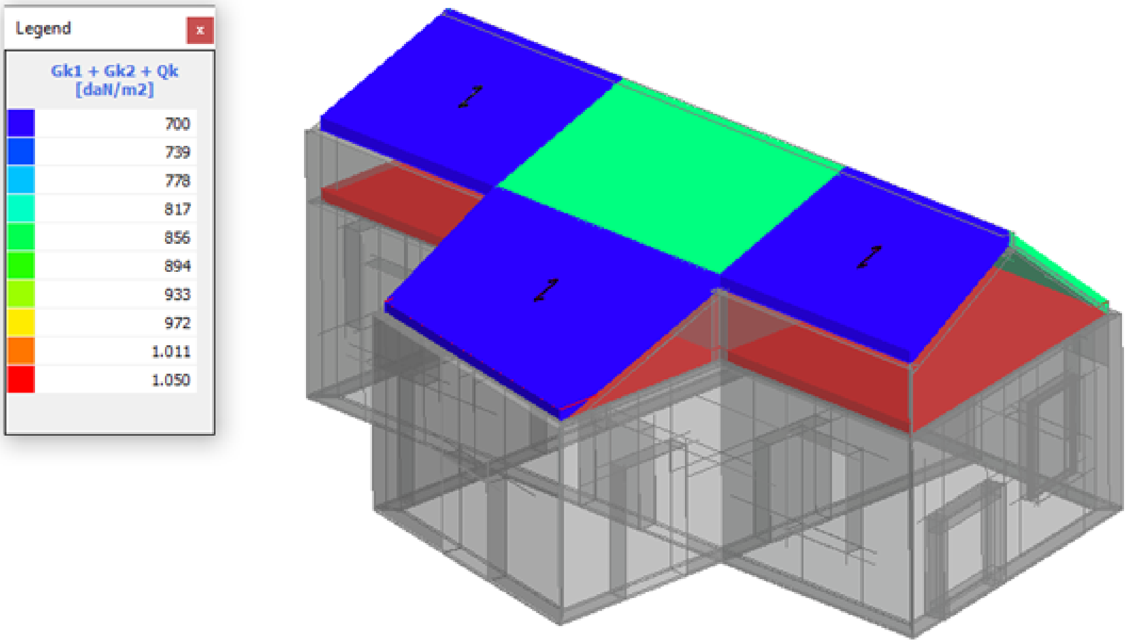


	Level	Visible	Roof	Description
▶	1	☑	☐	Livello 1
	2	☑	☑	Livello 2

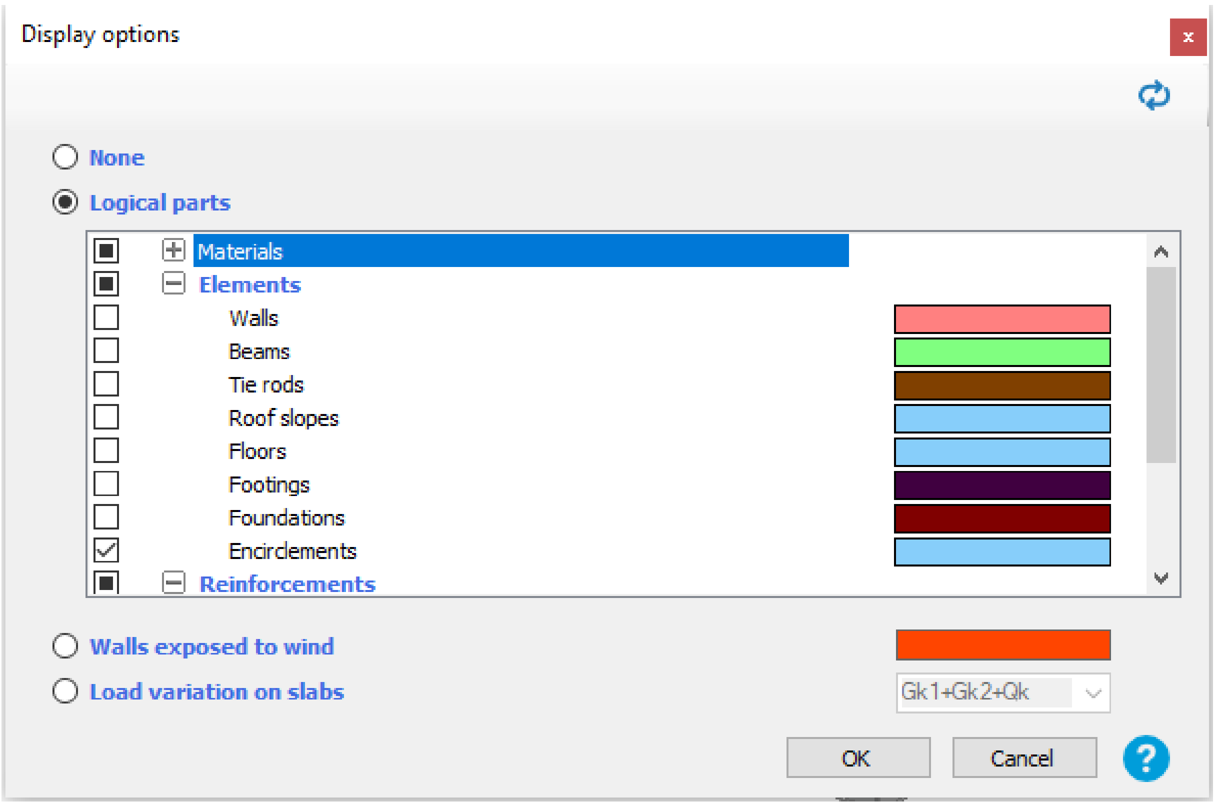
Instead of seeing all stories or just one at a time you can see a specific combination of stories.
This feature is very useful in cases of particularly complex buildings, whose overall vision may not let us have a good understanding of the position of some fake elements compared to the elements already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

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

 Legend color slabs: If the display option *Load variation on slabs* is enabled, it allows the visualization of the color legend.



 Display options: Opens the window for setting the display options



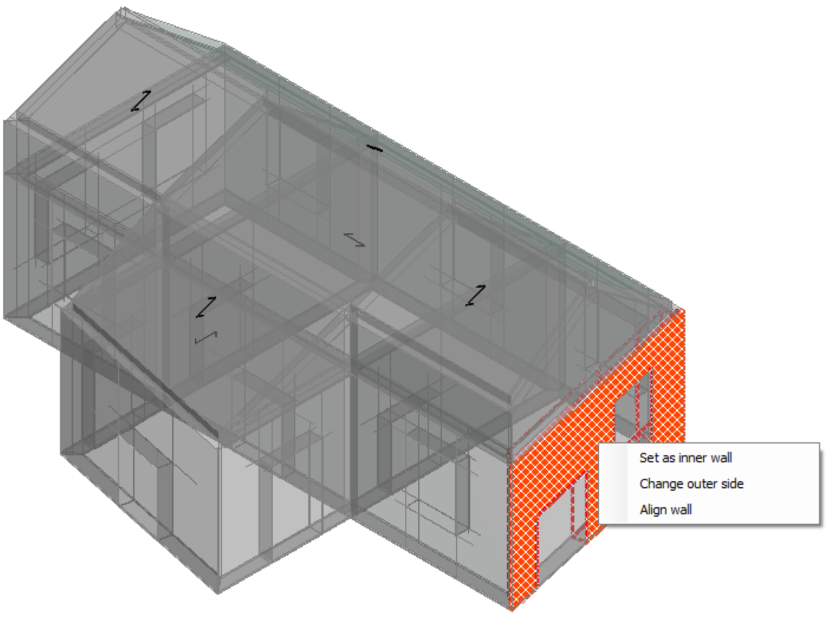
There are three different types of display options:

Logical parts It allows to draw the 3D elements with customized colors according to the material they are made of, according to their type (walls,

beams, columns, etc.) or according to the reinforcements or retrofits applied to them.
To include an item in the display options, simply check the box on the left.
The color can be customized by double clicking on the colored rectangle; default colors can be restored by clicking on the icon .
If an object satisfies several filters at the same time (for example for its type and for its material), it will be colored according to the last valid option.

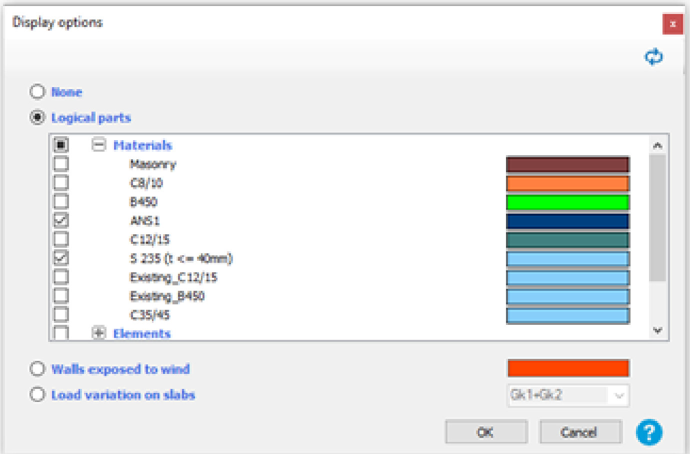
Walls exposed to wind It allows to color the walls exposed to the wind.
The color can be customized by double clicking on the colored rectangle; the default color can be restored by clicking on the icon .

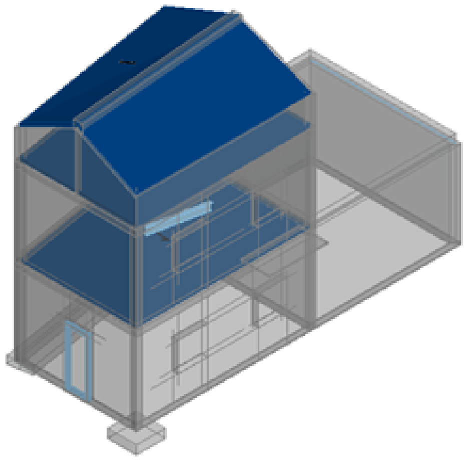
This view allows, in addition to seeing at a glance all the elements "exposed to wind", to assign the attribute of "external / internal panel" to the panels, assigning the element exposure accordingly.



Load variation on slabs It allows to color the horizontal elements (floors, roof slopes, vaults) with a color scale according to the load acting upon them, according to the combination chosen in the drop-down menu on the right.
The user can then consult the legend by clicking on the icon .

All objects that do not satisfy the active display options are colored in gray and made partially transparent.



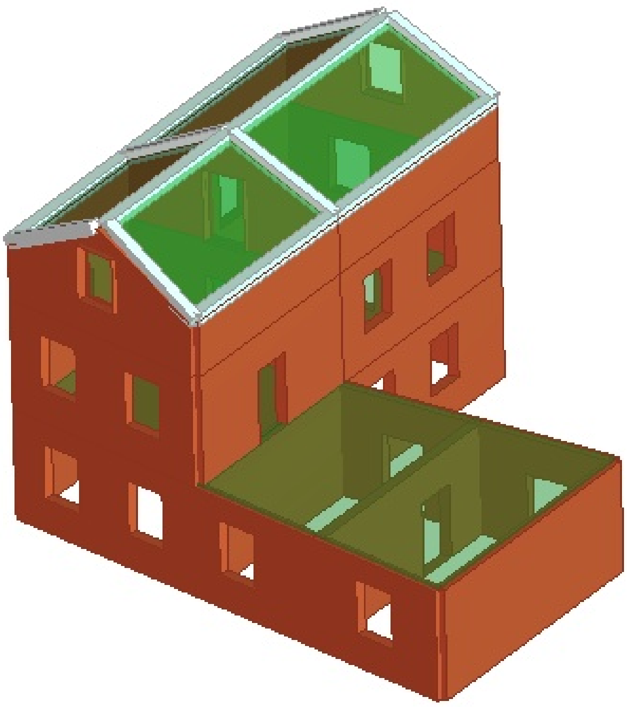


Switch to 3D Input area: This 3D view is only for display purposes, if you want to edit the structure in 3D view it is necessary to switch to the specific working area.

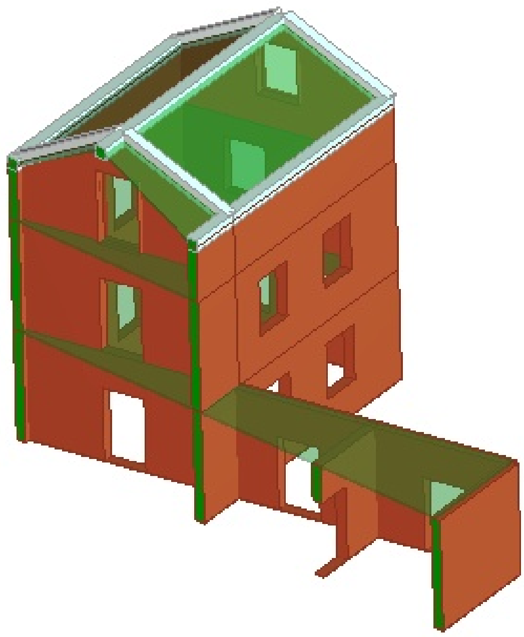
Rotate: A CAD command of the "Orbit" type allows the model to be rotated using the left mouse button. Press [Esc] to exit the command.

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

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



3D without section



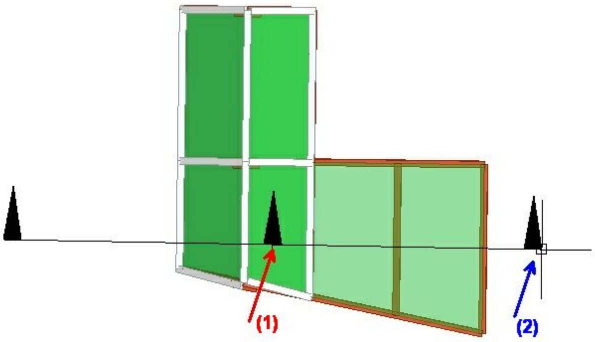
3D with Section

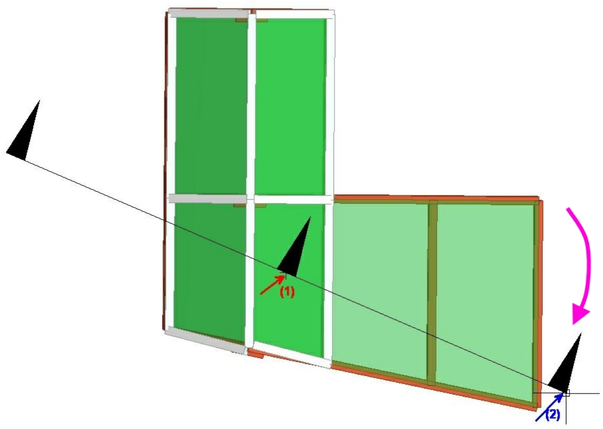
How to insert a section:

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

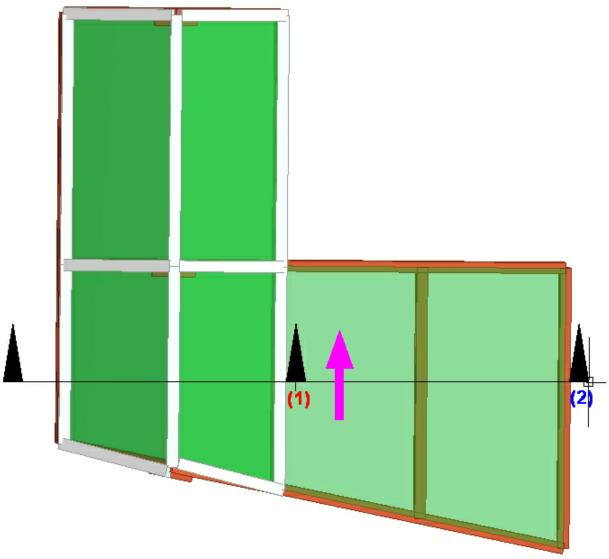
The section planes are always to be considered as vertical, starting from this hypothesis, the identification of the section plane is done by clicking on two points that identify the straight line projection of the plan.

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

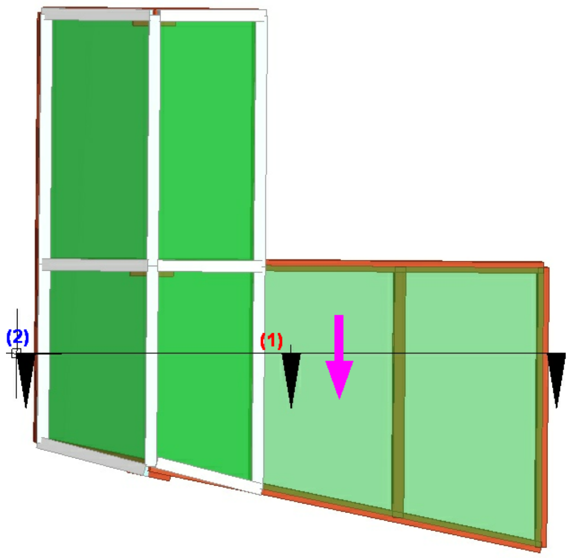




Here is an example with two points of view:



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



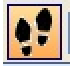
View from the top to the bottom (2) to the left of (1)

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

 Walk Through: An interactive system of navigation "through" / "around" the model.

To use this function, perform the following steps:

Press the appropriate button to enter the Walk through mode
Use the keyboard to navigate in the model:
[W]: move forward
[S]: move backwards
[A]: shift left
[D]: shift right
[Page ↑]: shift upward
[Page ↓]: shift downward
Move the mouse for positioning the camera
[Esc]: Stops the navigation at the last highlighted perspective view.

 The corresponding button looks pressed, press it again to cancel the command and return to the classic view.

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

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

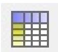
 View Items:
enables / disables the display of elements: panels, RC walls, beams, columns, tie-rods, floors, roofs, foundations.

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

OPENGL View: Button that activates/deactivates the OPENGL view within the three-dimensional view of the model.

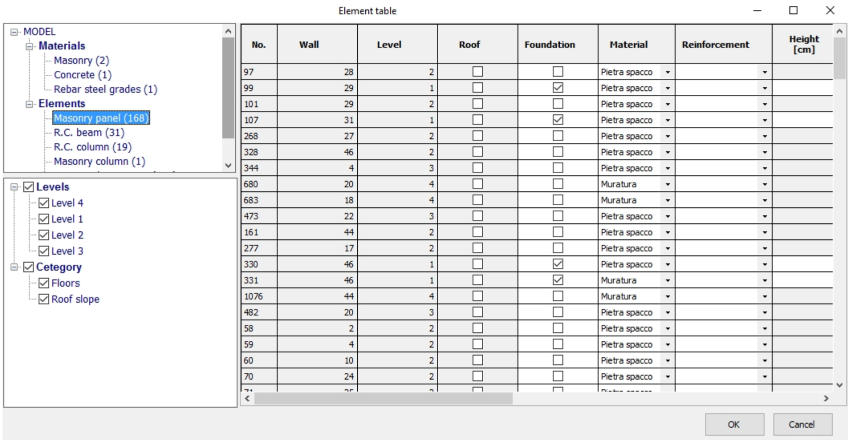
WARNING!!!
In order to ensure a constant update of 3D, you need to close the window of the axonometric view before proceeding with any updates / changes to the model.

7.3 Table

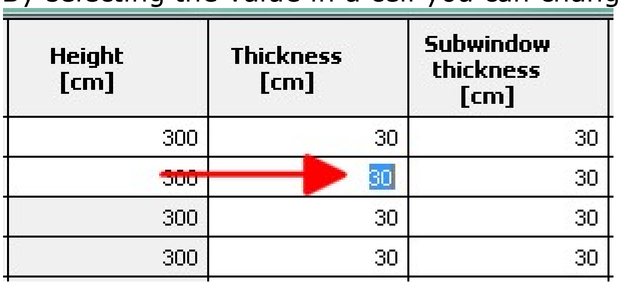


While opening the table through the proper button, a window that shows the user's inserted data through the graphics will appear. The drop down menu on the left makes the navigation inside the table easier.
The drop down menu is organized in four main branches:

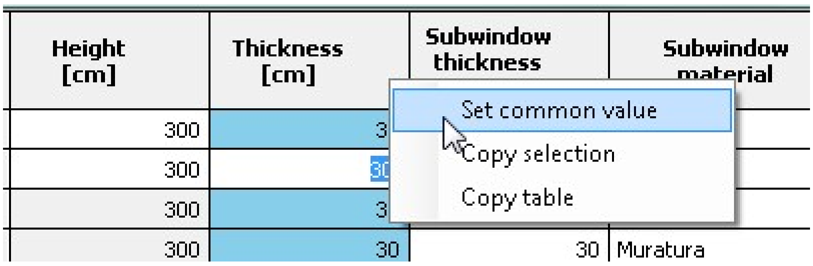
Materials: This contains the material typologies used in the project, with their mechanical characteristics.
Reinforcements: Contains the features of the masonry reinforcement type.
Elements: This contains the elements used, divided by typology (according to that indicated in the characteristics definition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



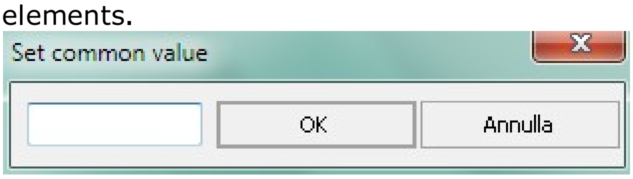
The present data can be edited and stored with the new value.
Every modification in the table implies changes of the template data.
Only the fields with a white background are entitled to modification, the gray background indicates a not editable fixed field.
There are two types of modifications in the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



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



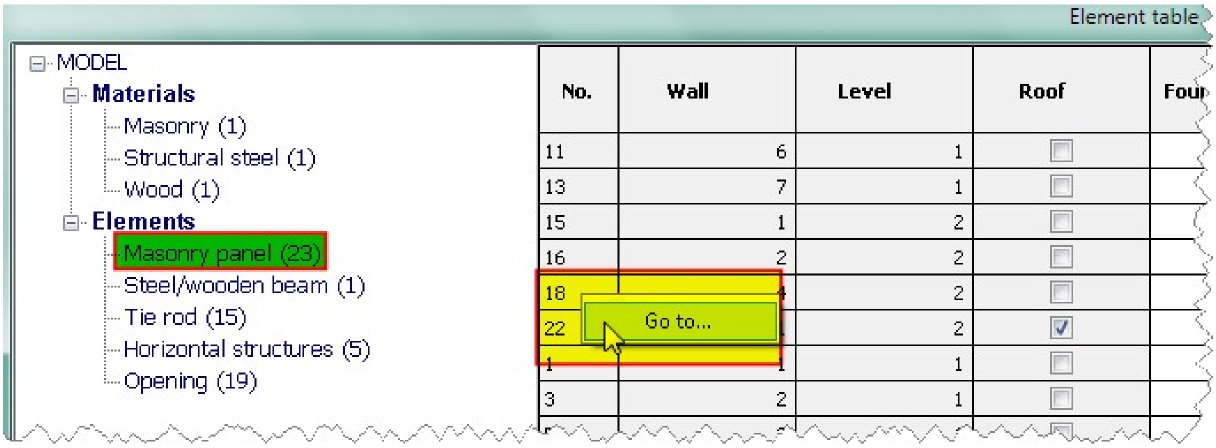
Select "Set common value" shows a small window to define a single value common to all elements.



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

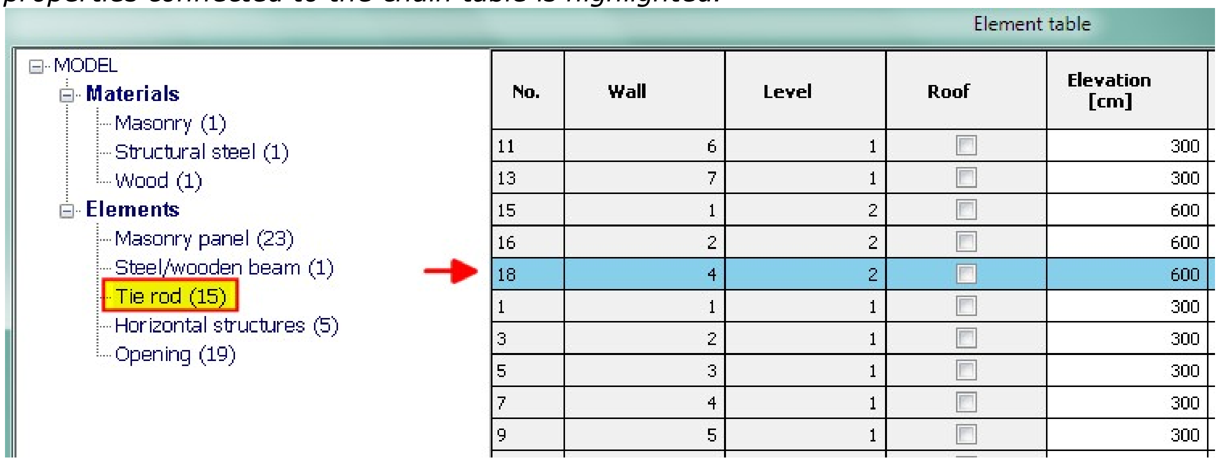
All the "***composed***" structural elements (panel+ridge; panel+chain; panel+steel/wood beam) are divided into their constituent elements.
This breakdown allows for example to change the characteristics of all the walls regardless if they are dominated by chains, curbs or by any other evidence.

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

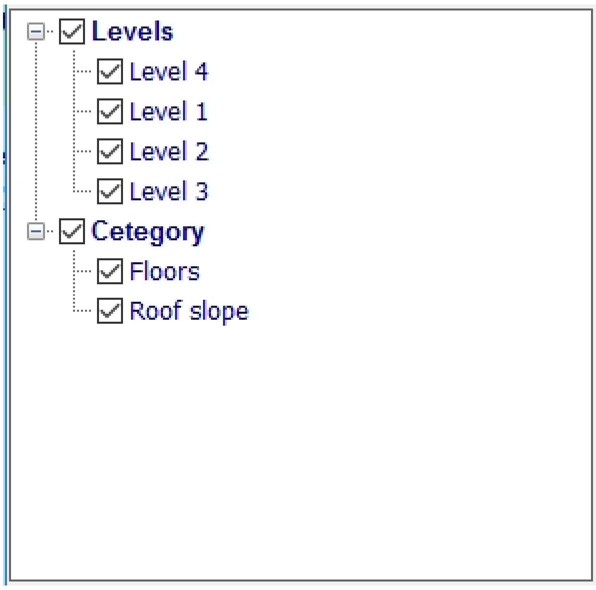


By selecting this command, you get redirected to another table where the row corresponding to the element coupled with the first element is highlighted.
Ie:
All the elements that make up a composed entity have the same identification.

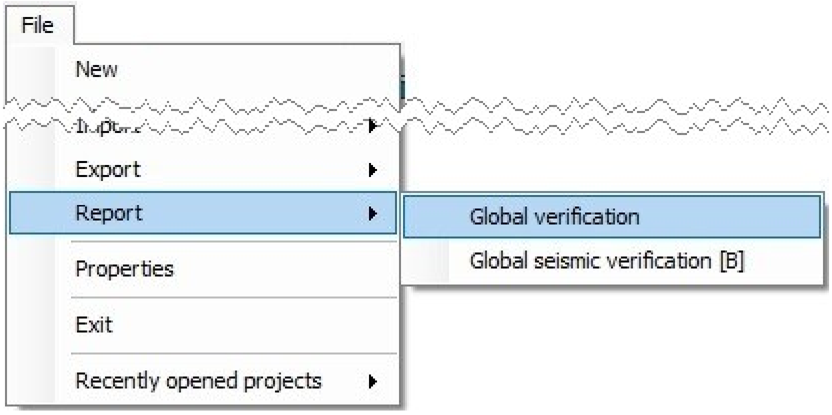
For example, if from the "wall panel" table, is selected the [Go to ...] corresponding to the panel 18 you get redirected to the "chain" table where the same element 18 with the properties connected to the chain table is highlighted.



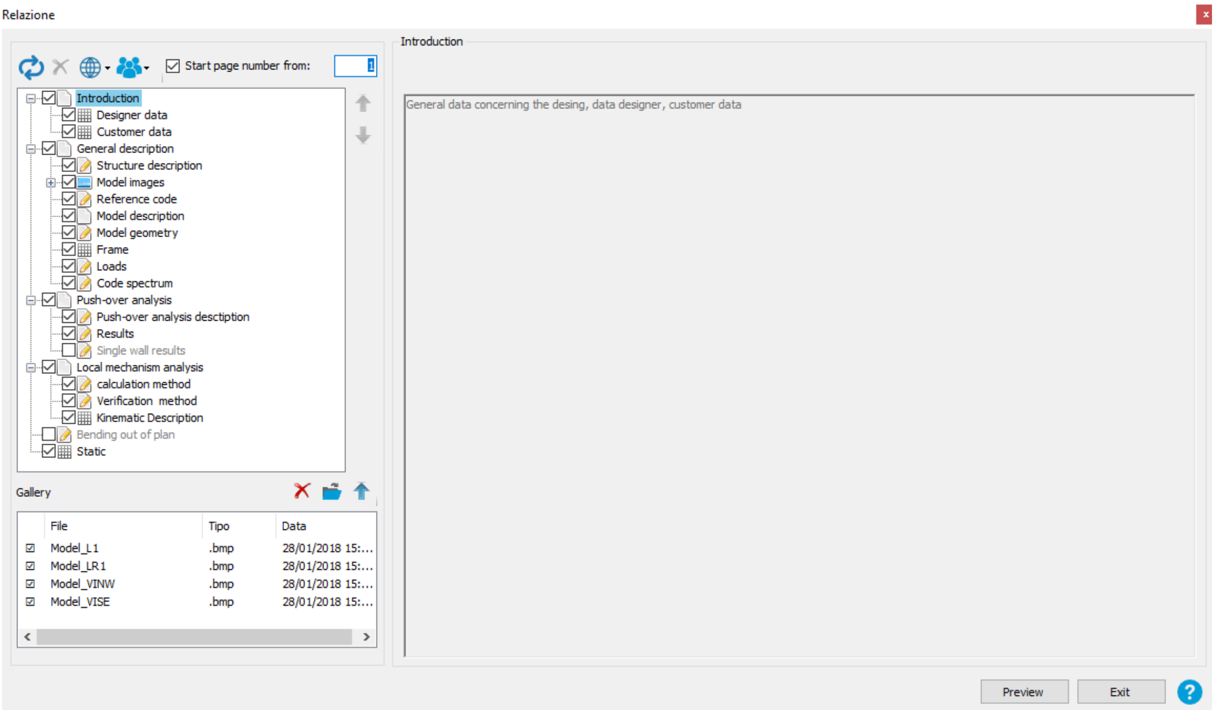
In the lower part of the table is shown a filter function tree that allows filtering the contents of the table based on levels and categories of interest.



7.4 Report



The **Report** item in the File Menu produces the project report.



On the left side the chapters of the Report are listed divided by calculation types. Details are available within each chapter.

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

 It allows to delete an image from the report

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

 Designer-costumer data:

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

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

 This command (present in the 3D view) allows saving the image currently displayed on the screen.

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

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

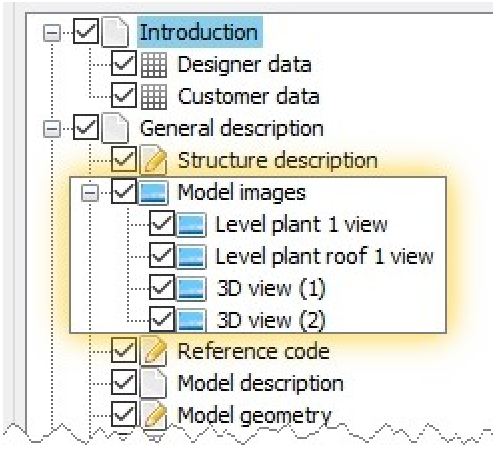
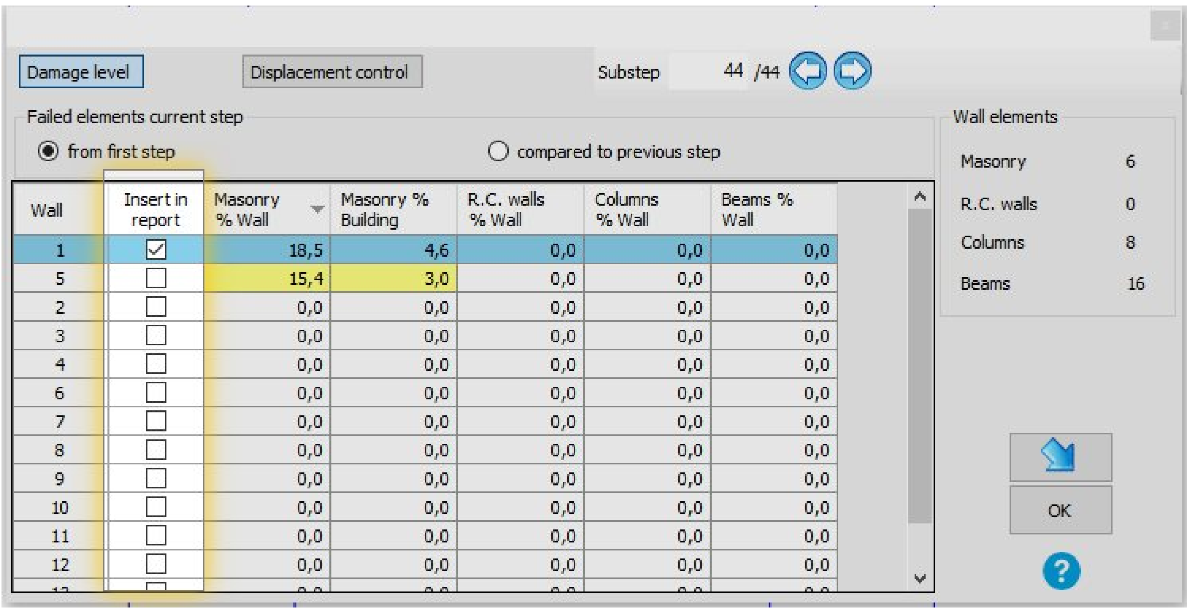


Image Gallery Commands	
	Deletes an image from the Image Gallery
	Imports an external image to the Image Gallery
	Inserts an image in the report scheme
	Moves an image inserted in the report, allowing the user to decide where to place it

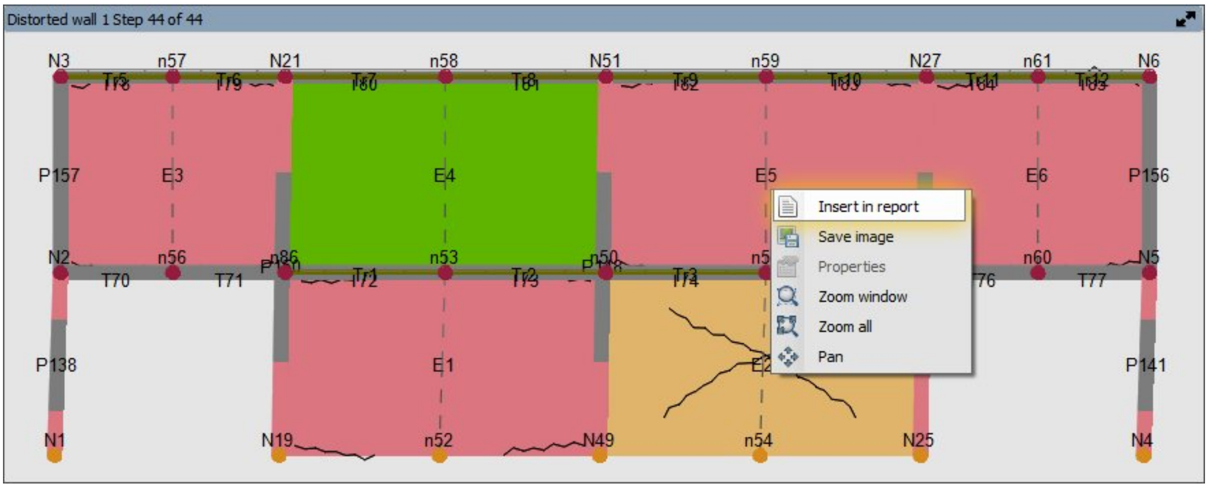
Inside the report are described the most severe analysis with their relative walls.
This is done automatically by the program which identifies the most severe analysis and, from these analysis, goes to evaluate which are the walls that have obtained the bigger failure.

Through the command "display results", in the column "insert in report", the walls that will be inserted automatically in the report are highlighted.

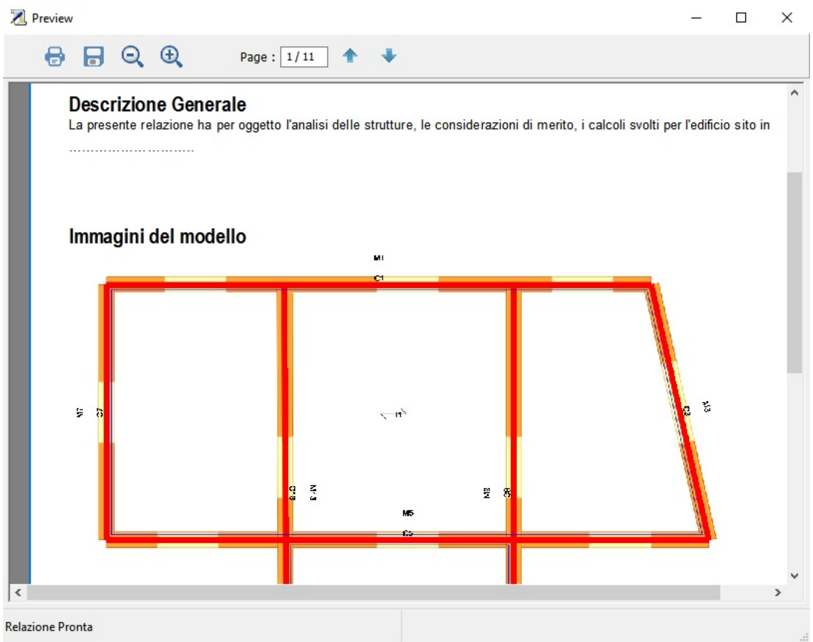


It is possible to customize the insertion by modifying/adding the choices made automatically by the program.

Furthermore, the user, within the results environment, can establish at any time which walls to a
By clicking on the right mouse button, in fact, appears the window (visible in the image below) fro



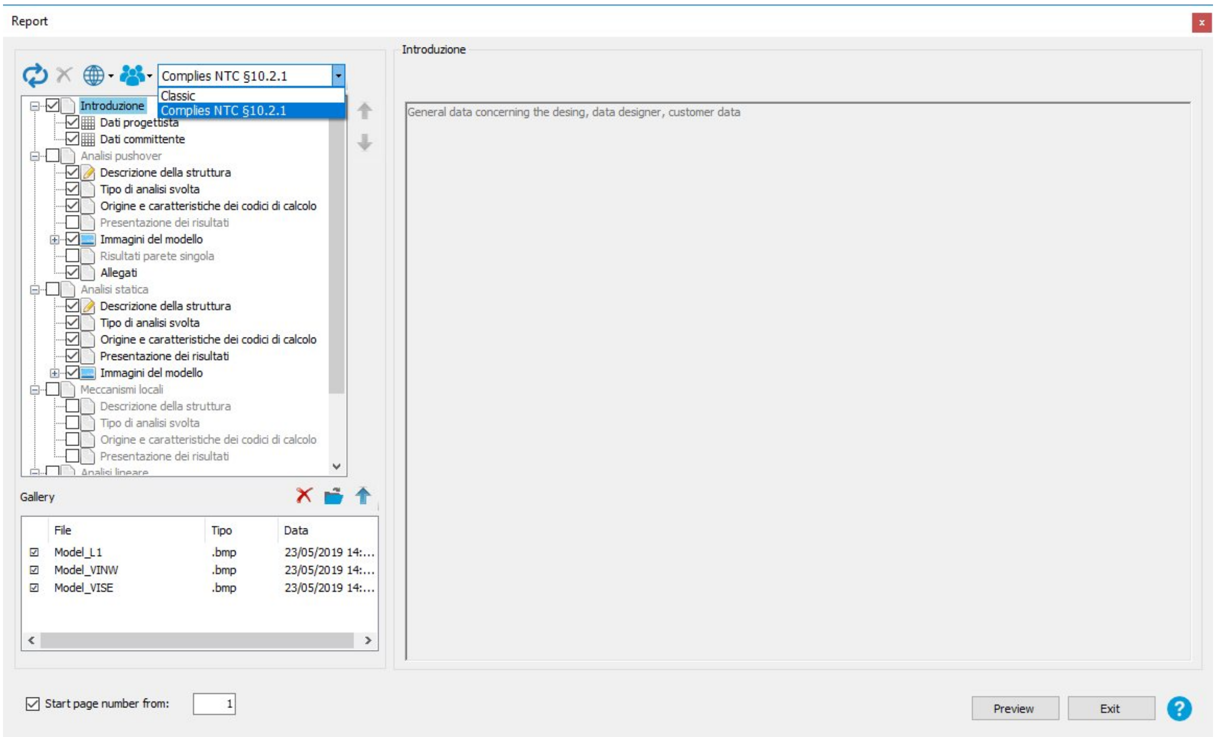
Activating the "Preview" button displays the print preview which allows a first view of the document.



The Save button allows the user to export the report in "RTF" format, readable and editable by any word processor.

NTC18

It is also possible to export a report that complies with the requirements in the § 10.2.1 of the 2018 Italian Technical Standards.



The report thus exported will be structured as follows:

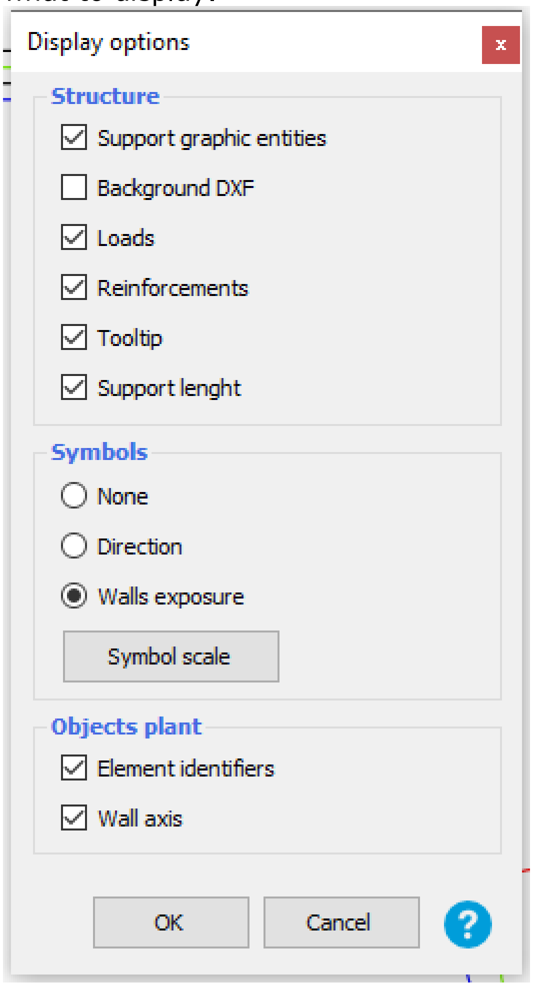
1) Description of the structure	Description of the structure under exam
2) Type of analysis performed	Description of the structural analysis typology carried out, by the method adopted for solving the structural problem, the load combinations adopted and, in the case of non-linear calculations, the load paths followed
3) Origin and characteristics of calculation codes	Title, author, producer, version, license of the calculation code used
4) Results Presentation	Normative framework of the intervention, project parameters, description of the materials adopted, design and modeling criteria, summary of the results with drawings and graphic schemes
5) Attachments	Collection of Spreadsheets provided automatically by the program

It is possible, for each type of analysis, to remove the part that the user do not want to insert in the report by removing the tick from the corresponding item.

7.5 Display Parameters

Plan view of the level: This command (accessible from the Display menu) schematically displays the plan of the active layer with the representation of the various types of structural elements defined in the structure environment.


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



Structure
"*Background Dxf*" activates the display of any imported .dxf files within the model space

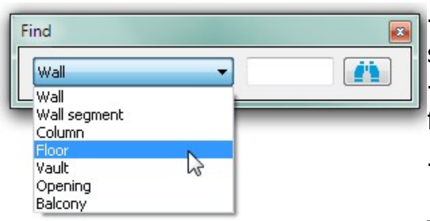
"*Loads*" allows to show/hide loads concentrated/linear showed in the plan.
"Tooltip" allows to show / hide, within the structure environment, the tooltips that are shown in the plan for each element
"*Support length* "activates the display of the values assigned to the "support length" parameter of the floors

Symbols
"Direction" shows the walls' local reference system determining the wall panels'

eccentricities sign referring to the wall.
"*Display panels*" activates the symbology relating to the display of each panel. In fact, it is possible to indicate whether a wall is internal / external. This feature allows you to see the walls directly affected by the wind load, assigning their exposure to wind.

Objects plant
"Element indentifiers" activates the display, in the object plan view, of the identifiers of the elements
"Wall axis" activates the display, in the object plan view, of the axes of the walls

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



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

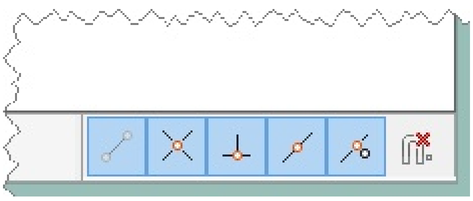
The search result is shown by placing the searched element in the middle of the screen, with the mouse pointer on it and a special marker that highlights it.

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


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

7.6 Snap

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



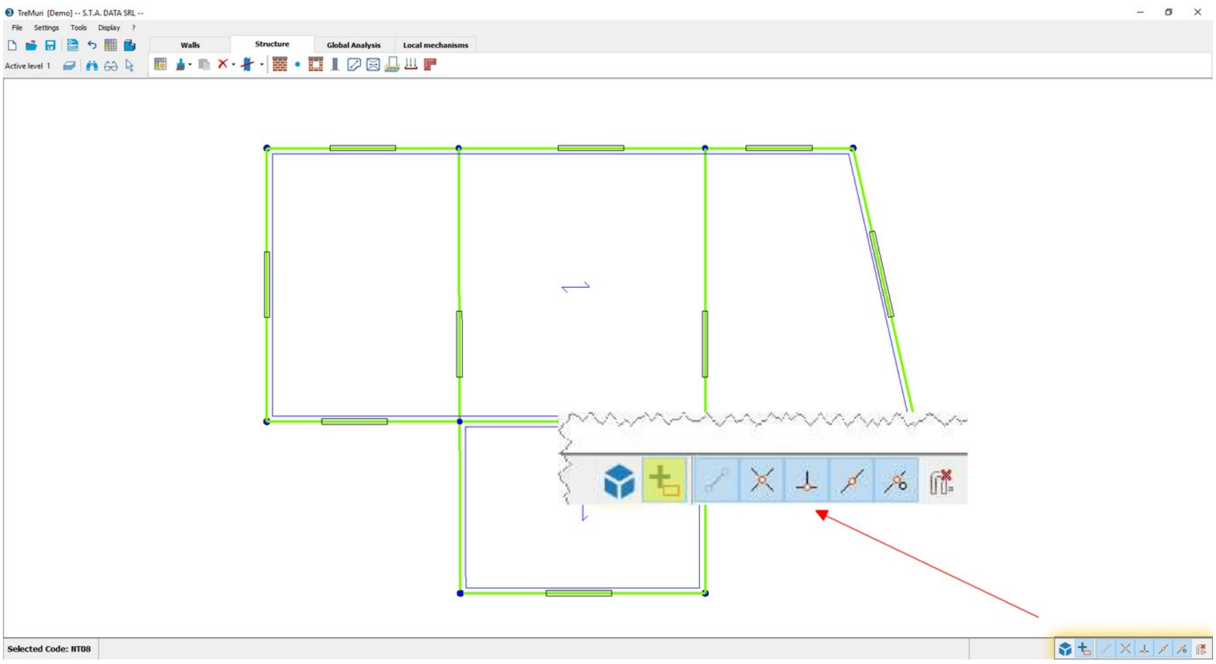
Below the list of available Snap:

	Extreme

	Intersection
	Perpendicular
	Middle
	Line
	Uncheck all Snap

7.7 Dynamic Input

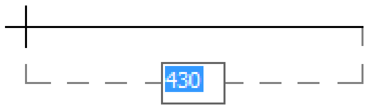
The command ***Dynamic Input*** is accessible from the command bar placed at the bottom right of the screen input.
The corresponding button appears pressed with the activated "dynamic input" mode, to disable this mode it is necessary to press again the same button.



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

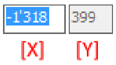
Elements Length

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



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

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

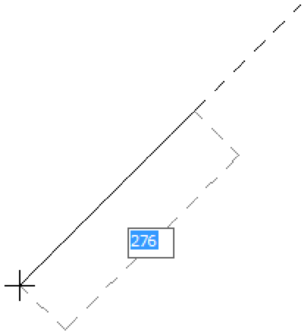


You can switch from the [X] coordinate to the [Y] coordinate by pressing the [TAB] Keyboard.

2. The second point can be defined in coordinates.

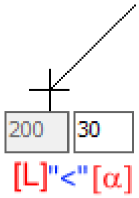
Polar [Distance=*{numerical}* ; Angle=*{mouse}*]

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

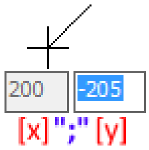


Polar [Distance=*{numerical}* ; Angle=*{numerical}*]

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



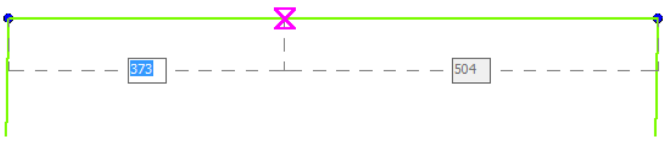
Relative [X=*{numerical}* ; Y=*{numerical}*]



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

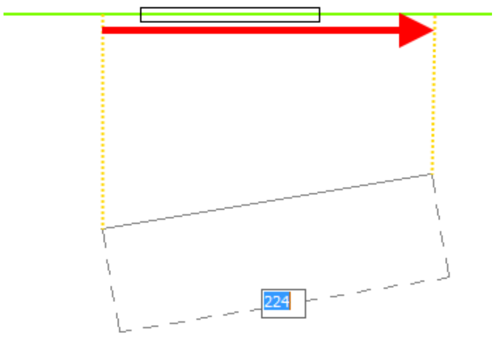
Placing objects on walls

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



Moving Objects on walls

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

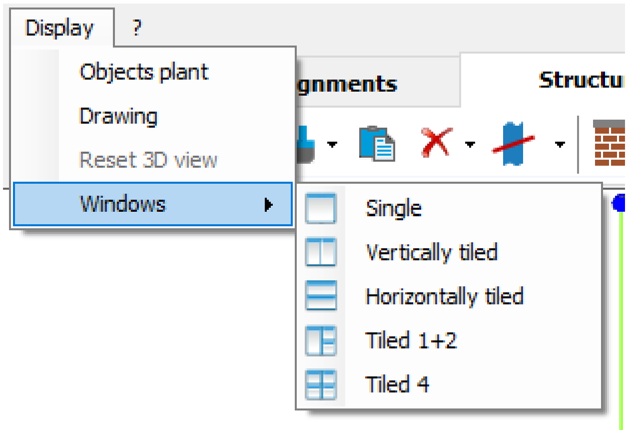


7.8 Selection Mode

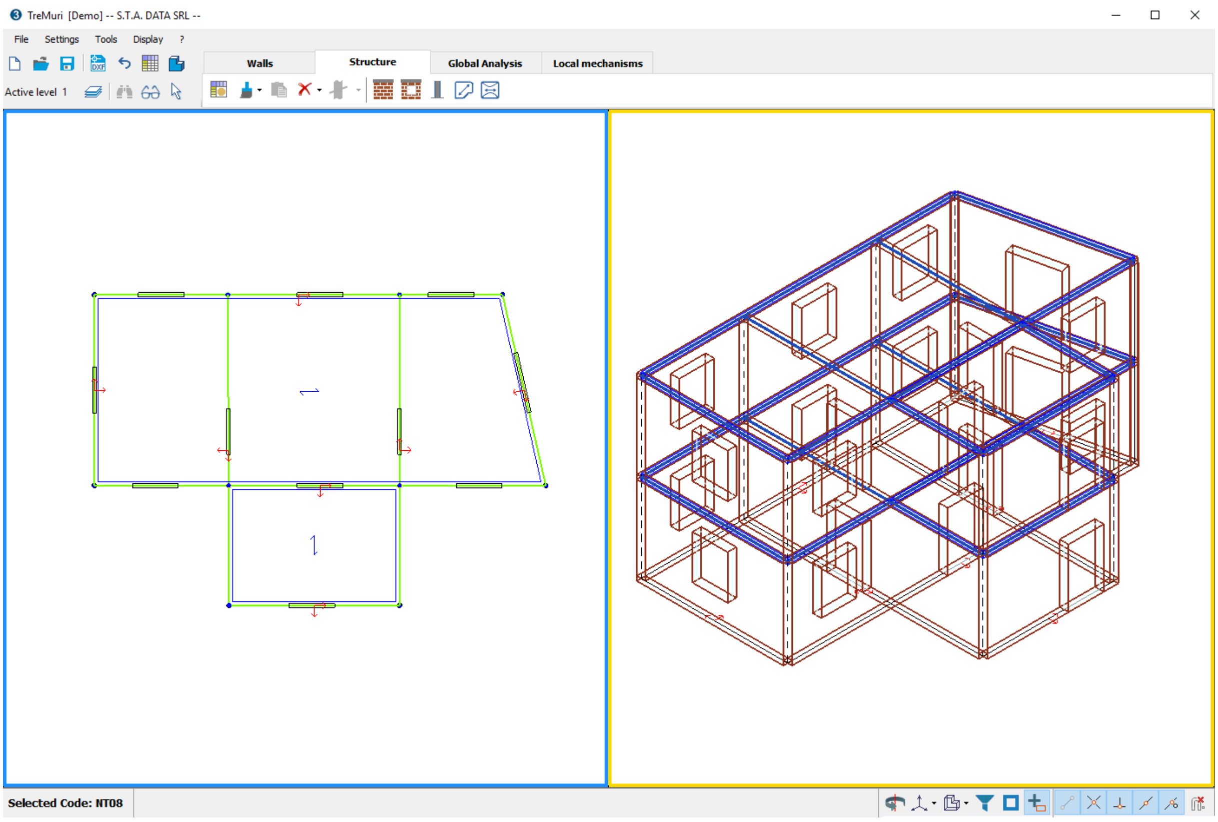
The program offers various selection modes.
Single selection: Each element is selected by clicking on the entity
Multiple selection: This selection mode has two sub-modes.
1. Selection in sequence of more elements
2. Selection in window mode (all elements that are intersected by the window frame and those contained within are part of the selection).

7.9 Multiple view

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



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



The active view is surrounded by an orange border.

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

7.10 Switch to 3Muri IL

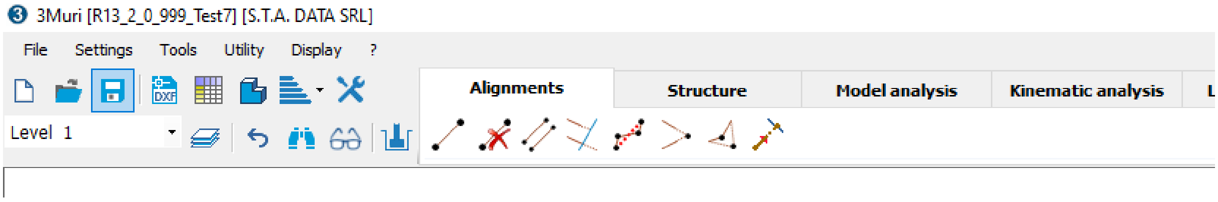
3Muri IL is an application for the verification of local interventions created and distributed by S.T.A.DATA srl

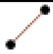
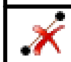
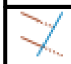
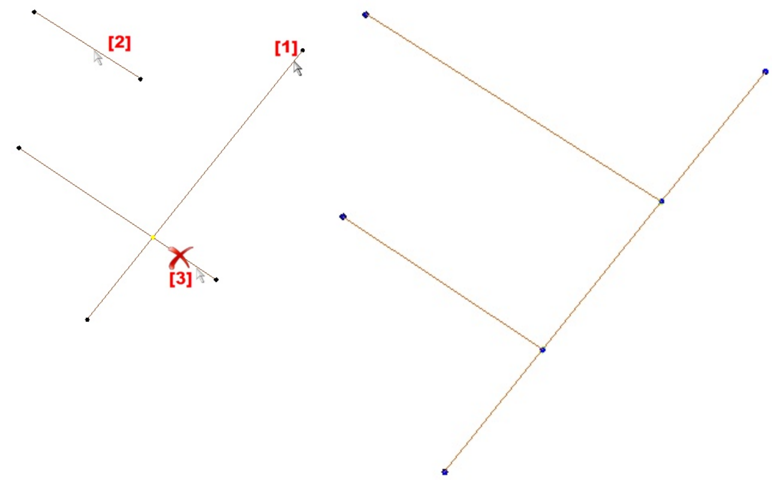
By clicking the appropriate button inside the main command bar, the user can export the model within 3Muri IL (Local Interventions).
Before proceeding with the export, the program warns the user that to open the model within the 3Muri IL software, it is necessary to close the current application.

After confirming, the model is automatically opened in the 3Muri IL software, inside which it is possible to see the exported model.

The interoperability between the two softwares is bidirectional. In fact, it is possible at any time to switch from one software to another without the need to remodel the elements of the structure.

8 Geometric Definitions

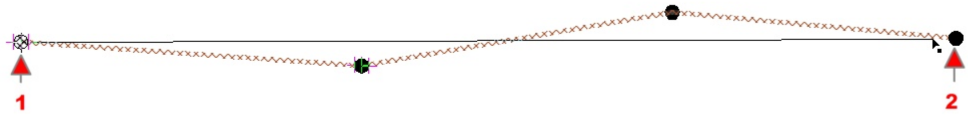


	**Insert walls**: allows to insert an alignment. For alignment is meant a continuous length of masonry, R.C. walls, beams, tie rods (it can also be multiple segments all resting on the same tangent). **Insert a new alignment**: by clicking on the mouse's left button can be defined the subsequent nodes that identify one or more alignment; **Terminate inserting a alignment:** right click on the mouse to exit the command. The insertion of the elements can be easily performed through the snap (to the graphics or on the alignments).
	**Remove walls:** allows to delete a previously inserted alignment
	**Extend/Trim walls**: allows to lengthen or shorten an existing alignment. 1. select the alignment **on which to extend the other alignment** 2. select one or more alignments **to extend** **Example:** [1]: select the alignment on which to extend / cut [2]: select the alignment to be extended [3]: Select the alignment to be cut from the side you want to delete **Before** **After** 
	**Rectify wall:** Allows to rectify the previously entered alignments. The figure below shows three different alignments with a slope close to each other. This may result from a not very accurate background DXF that led to repolish the plant through segments not well aligned.

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

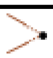

When the change of slope is so contained, the designer may decide to replace the sequence of three alignments with a single alignment (shown in red in the figure above) in order to simplify the calculation model without losing precision.
The rectification command replaces a sequence of adjoining alignments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

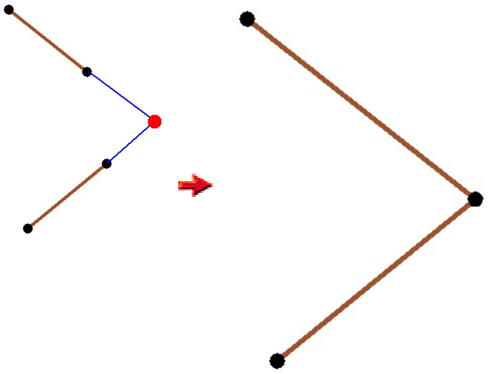
Following the steps of how to use this command:
- sequentially select the contiguous alignments with the left mouse button;
- confirm the selection by pressing the right mouse button;
- identify the rectification directrix indicating the two extreme points (the points should be identified between the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



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

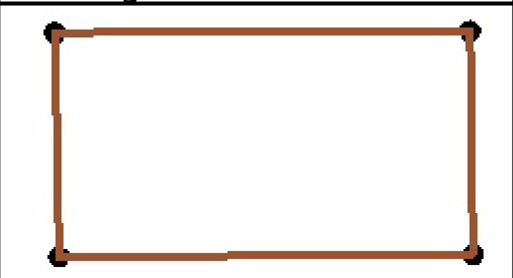
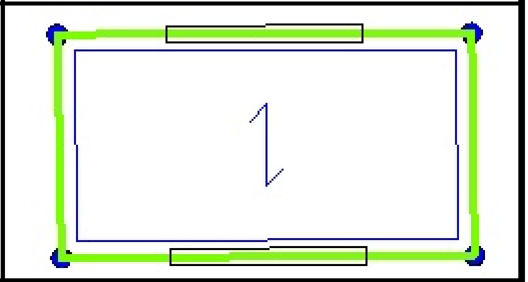
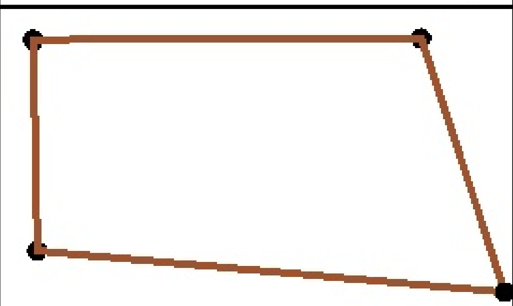
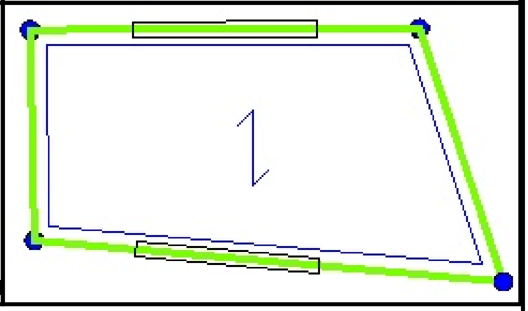
The result of the rectification operation will be the following in the [Structure] area.
In the structure area, the alignments now defined as one, will be separated into multiple segments (through the element nodes) at the extremes of the alignment, removed after the operation rectification.

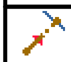

</td>
</tr>
<tr>
<td></td>
<td>
<u>Fillet walls</u>: allows to join two alignments that do not cross.
Select two alignment to be connected with the left mouse button


</td>
</tr>
<tr>
<td></td>
<td>
<u>Stretch</u>: This command allows to move an outer node of the alignment.

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

The editing operation involves the automatic adjustment of all the structural objects directly attached to that node.
</td>
</tr>
</table>

	Alignment Environment	Structure Environment
Before		
After		

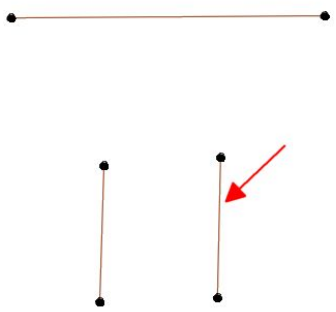
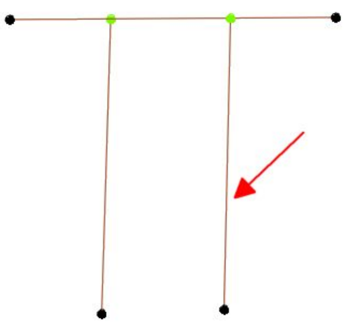
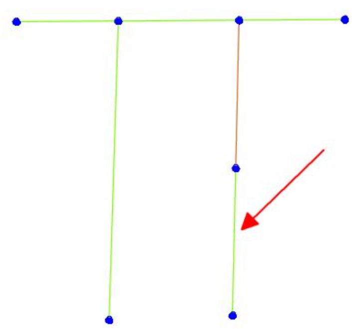


Advanced Extend/Trim

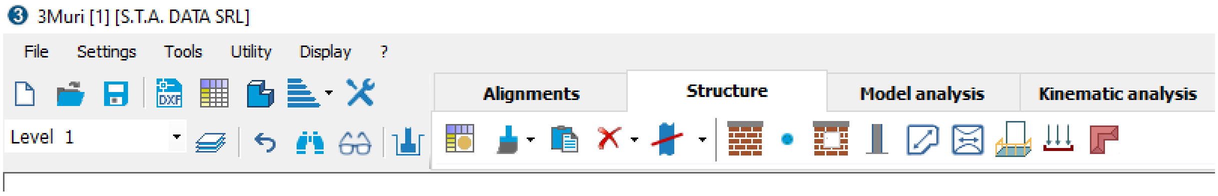
This command represents an evolution of the classic "extend/trim" command of any CAD system aimed at the classic imputation of wall structures.
In classic mode, an extend applied to an alignment moves one of the two extremes up to an intersecting element, also involving the properties of the member itself.
In advanced mode, the same option does not imply a displacement of the extreme but an automatic creation of a new portion without structural properties. This functionality does not alter the conformation of the structural objects that insist on an extreme because the point is maintained with the "element node" properties.

Example:

The right alignment (indicated by the arrow) represents the wall affected by the advanced extend function, while the left alignment is affected by the classic extend.

before of extend	**after extend (alignments)**	**after extend (structure)**
		

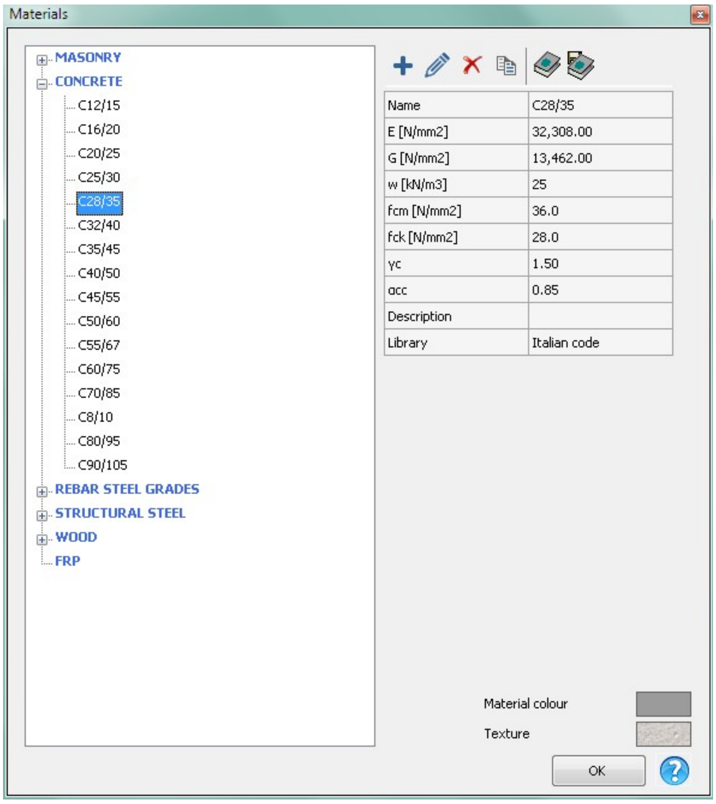
9 Characteristics of the Structure

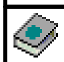


	Materials
	Copy Property
	Paste Property
	Delete the selected structural elements
	Supported thickness (read the panels thickness from DXF file)
	Assign attributes of the wall segments
	Graphic point insertion/Distance insertion
	Opening
	Column
	Slab
	Vault
	Balcony
	Load
	Roof

9.1 Materials

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



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

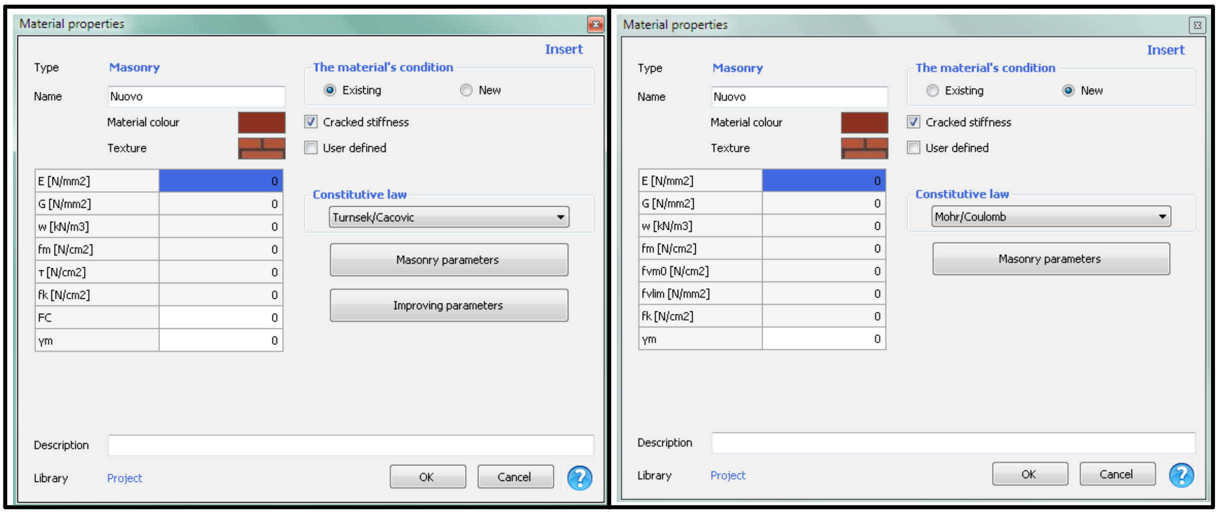
Each material is associated with a color chosen by the user. It is then used in the 3D display window.

9.1.1 Muratura

9.1.1.1 NT08/Eurocodice/SIA

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

Existing Material	New Material

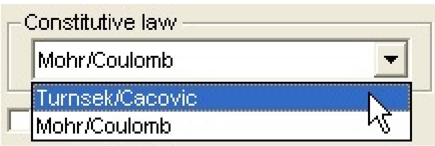


E: longitudinal elasticity module
G: Shear elasticity module
w: specific weight
fm: Average compressive strength
fvm0:*(Mohr-Coulomb)* The average shear strength without axial action
fvlim: *(Mohr-Coulomb)* The shear strength limit (suggested value 2.2 N/mm2 *§7.8.2.2.2 - D.M.14-01-2008*)
τ:*(Turnšek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

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

For both existing and new material you can decide the type of shear bond to be used:

- Turnšek Cacovic criteria
- Mohr-Coulomb criteria



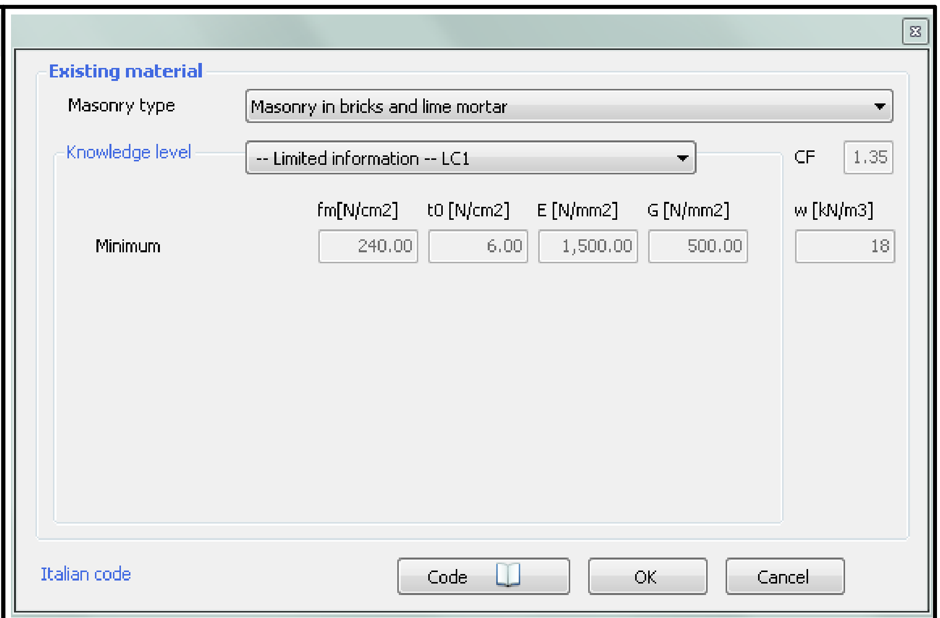
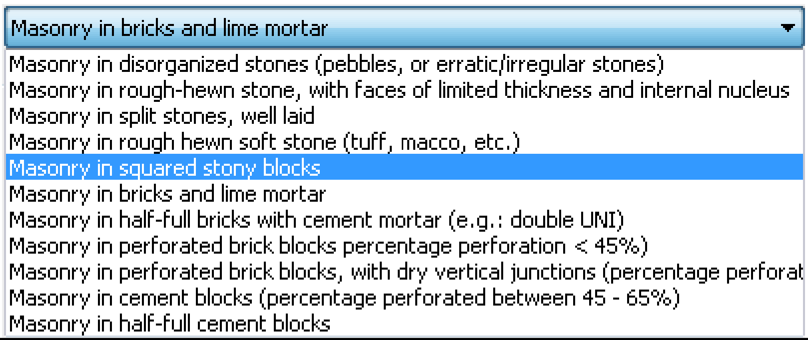
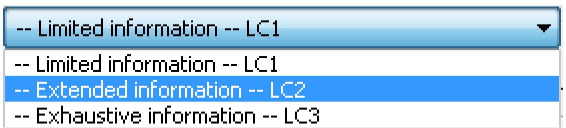
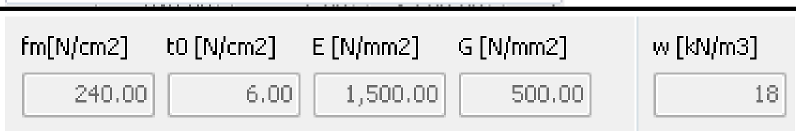
The **Turnšek Cacovic** criteria represents a type of diagonal shear failure and it is recommended to use especially for *existing walls.*
The **Mohr-Coulomb** criteria represents a type of sliding shear failure and it is recommended to use especially for the *new masonry*.

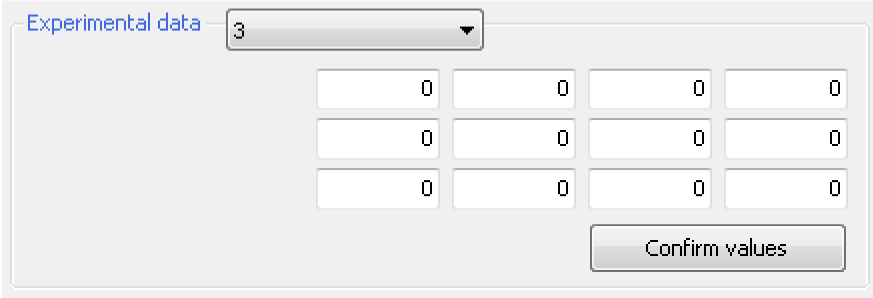
In the case of existing masonry made of clay, it makes sense deciding to adopt a Mohr-Coulomb failure criteria in order to examine a failure criteria more appropriate for the type of examined walls.

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

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

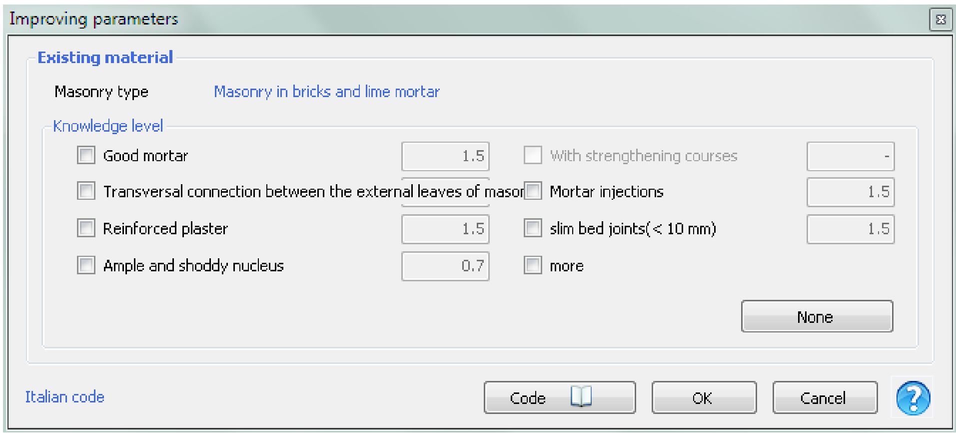
9.1.1.1.1 Filling Parameters 1

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

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

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>

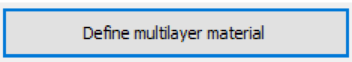
Definition of multilayer

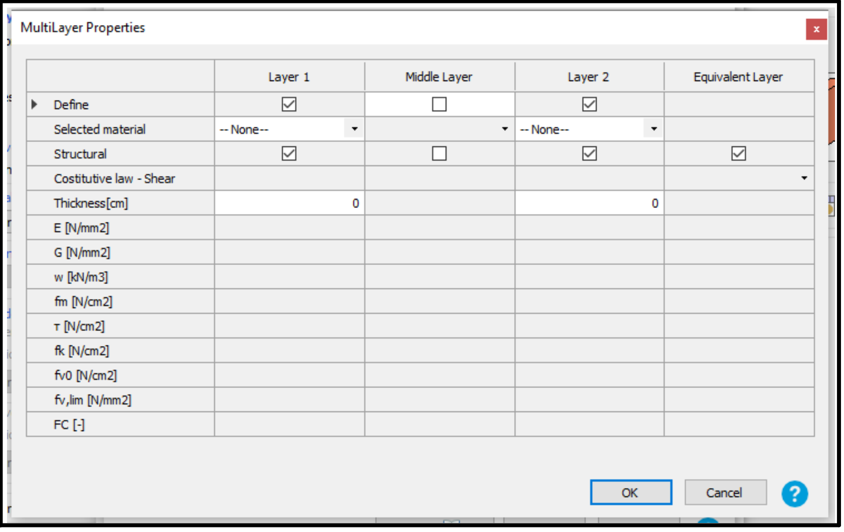
A wall panel made by the presence of 2 panels and possibly a cavity, occasionally filled with a material of minor characteristics, cannot be identified from a regulatory point of view. This "problem" compels the designer to resort to scientific procedures of "proven validity".

From a first analysis, a multilayer wall can be in 2 different conditions:

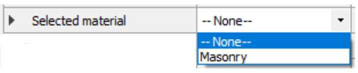
1. Panels integral with each other: In this case it is assumed that the two panels are simultaneously effective at the seismic bearing capacity. It is therefore possible to create an equivalent masonry material that arises from the parallel combination of the 2 layers. This procedure is easily achievable through a process of homogenization of the mechanical characteristics.
2. Panels NOT in solidarity with each other: In this case it would be good to consider problems of local instability of a single panel, excluding, if appropriate, the contribution of one of the two panels. This solution is to be considered suitable only in the modeling of current status since, in a project status it would be advisable to think of joining interventions of the layers, thus referring back to what is reported in point (1).

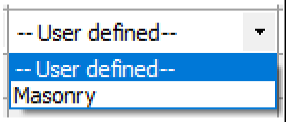
What is described in point (1) is what contemplates the functionality of the program described here; if the building in question has wall panels described in case (2), they must be examined with particular attention, evaluating the approximations that are deemed most suitable for the case analyzed.

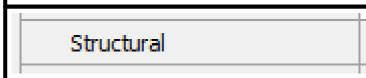
<table>
<tr><td>
</td><td>In this window it is possible to define the characteristics of each of the layers that make up the equivalent layer.</td></tr>
</table>

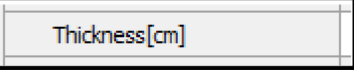
<table>
<tr><td></td><td>
</td></tr>
</table>

General prameters

<table>
<tr><td>
</td><td>The field indicates how many and which layers will be defined.
Insert the check in correspondence of the layers that you want to insert.</td></tr>
<tr><td>


</td><td>From the "selected material" tab it is possible to load the characteristics of one of the materials present, for each layer.

In the case of the intermediate layer it is possible to make a "- user defined -". This option is useful in the case in which (eg. brick masonry) the mechanical characteristics of the material of which the intermediate layer is made are not known. In this case it will be necessary to define only the "thickness" and "w" parameters.</td></tr>
<tr><td>
</td><td>Determines whether the inserted layer is structural or not.

Attention:
At least 2 of the 3 layers must have the "structural" attribute.</td></tr>
<tr><td>
</td><td>Thickness of the inserted layer.</td></tr>
</table>

Costitutive law - Shear Thickness[cm] E [N/mm2] G [N/mm2] w [kN/m3] fm [N/cm2] τ [N/cm2] fk [N/cm2] fv0 [N/cm2] fv,lim [N/mm2] FC [-]	Descriptive parameters associated with the selected material.

Definition of the parameters of the equivalent layer

$$E_{eq} = \frac{\sum_{i=1}^{3} s_i E_i}{s_{eq}}$$

$$G_{eq} = \frac{\sum_{i=1}^{3} s_i G_i}{s_{eq}}$$

$$w_{eq} = \frac{\sum_{i=1}^{3} s_i w_i}{s_{eq}}$$

$$f_{m,eq} = E_{eq} * MIN(\frac{f_{m1}}{E_1}; \frac{f_{m2}}{E_2}; \frac{f_{m3}}{E_3})$$

$$\tau_{eq} = G_{eq} * MIN(\frac{\tau_1}{G_1}; \frac{\tau_2}{G_2}; \frac{\tau_3}{G_3})$$

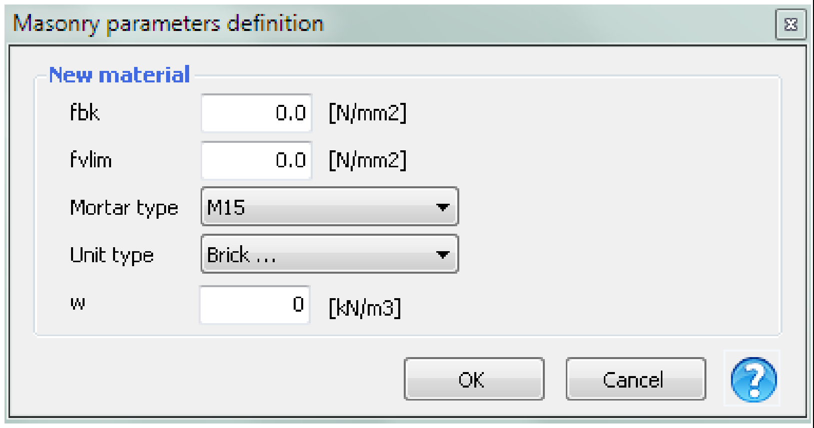
$$f_{k,eq} = E_{eq} * MIN(\frac{f_{k1}}{E_1}; \frac{f_{k2}}{E_2}; \frac{f_{k3}}{E_3})$$

$$f_{v0,eq} = G_{eq} * MIN(\frac{f_{v0,1}}{G_1}; \frac{f_{v0,2}}{G_2}; \frac{f_{v0,2}}{G_3})$$

$$f_{v,lim} = 0.065 * f_{m,eq}$$

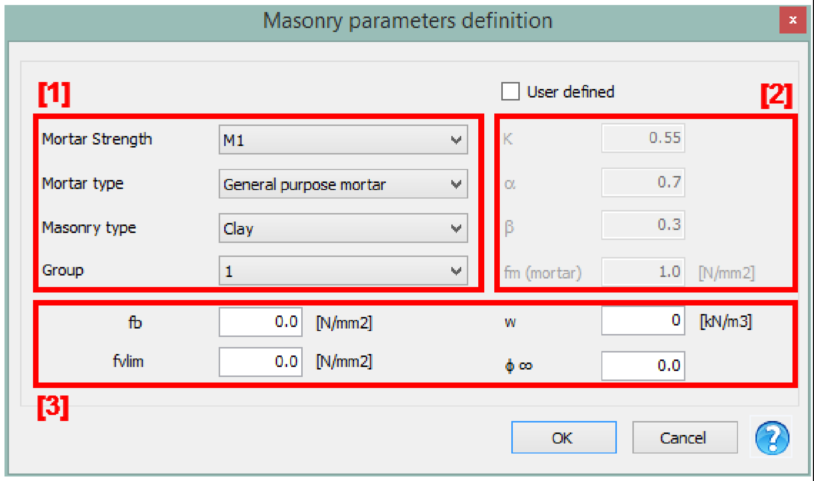
9.1.1.1.2 Filling Parameters 2

Technical Standard of Constructions | **§11.10.3**



fbk: characteristic compression strength
fvlim: shear strength limit
Mortar type: classification of mortars

Eurocode | **EN 1996 §3.6**



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

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

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

	• Input value for K, α, β, fm(mortar) In [3] the parameters that must be entered manually • fb : characteristic resistance of the block • §3.6.2(3)NOTE→ fvlim (0,065 fb or fvlt) • $\phi\infty$ • w When you press [OK] are performed the following operations: • §3.6.1.2(1) → $f_k = k \cdot f_b^{\alpha} \cdot f_m^{\beta}$ → $f_m = f_k/0.7$ • §3.6.2 -Table 3.4 → fvko → fvmo =fvko/0.7 (avarage value) • §3.7.2(2) → E (elasticity modulus) §3.7.3(1) → G (shear modulus)

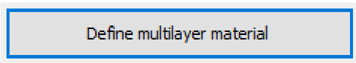
Definition of multilayer

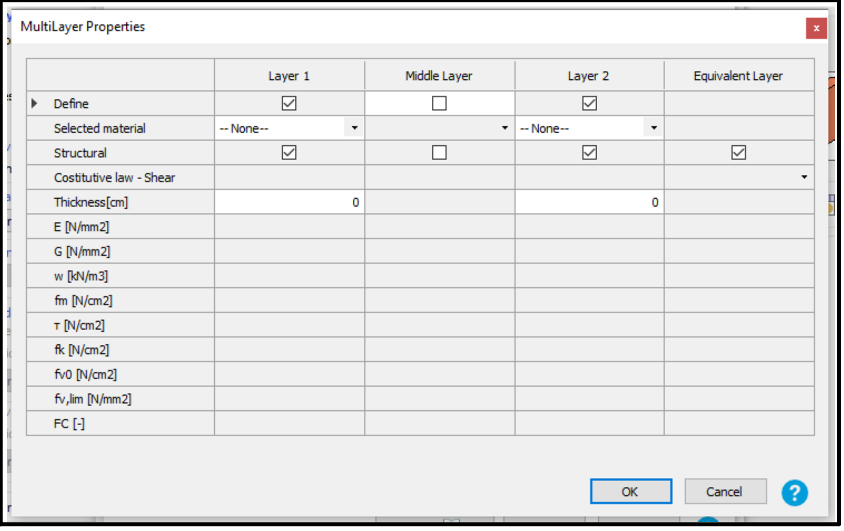
A wall panel made by the presence of 2 panels and possibly a cavity, occasionally filled with a material of minor characteristics, cannot be identified from a regulatory point of view. This "problem" compels the designer to resort to scientific procedures of "proven validity".

From a first analysis, a multilayer wall can be in 2 different conditions:

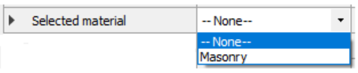
1. Panels integral with each other: In this case it is assumed that the two panels are simultaneously effective at the seismic bearing capacity. It is therefore possible to create an equivalent masonry material that arises from the parallel combination of the 2 layers. This procedure is easily achievable through a process of homogenization of the mechanical characteristics.
2. Panels NOT in solidarity with each other: In this case it would be good to consider problems of local instability of a single panel, excluding, if appropriate, the contribution of one of the two panels. This solution is to be considered suitable only in the modeling of current status since, in a project status it would be advisable to think of joining interventions of the layers, thus referring back to what is reported in point (1).

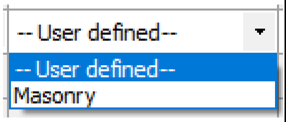
What is described in point (1) is what contemplates the functionality of the program described here; if the building in question has wall panels described in case (2), they must be examined with particular attention, evaluating the approximations that are deemed most suitable for the case analyzed.

	In this window it is possible to define the characteristics of each of the layers that make up the equivalent layer.



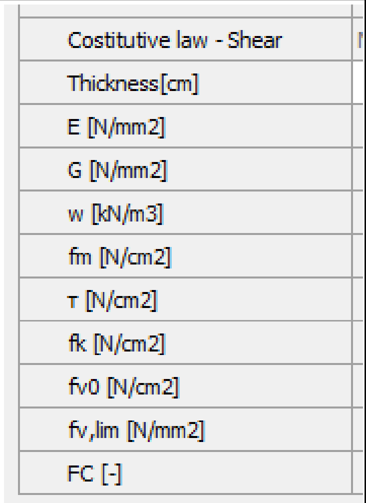
General prameters

<table>
<tr><td>
</td><td>The field indicates how many and which layers will be defined.
Insert the check in correspondence of the layers that you want to insert.</td></tr>
<tr><td>


</td><td>From the "selected material" tab it is possible to load the characteristics of one of the materials present, for each layer.

In the case of the intermediate layer it is possible to make a "- user defined -". This option is useful in the case in which (eg. brick masonry) the mechanical characteristics of the material of which the intermediate layer is made are not known. In this case it will be necessary to define only the "thickness" and "w" parameters.</td></tr>
<tr><td>
</td><td>Determines whether the inserted layer is structural or not.

Attention:
At least 2 of the 3 layers must have the "structural" attribute.</td></tr>
<tr><td>
</td><td>Thickness of the inserted layer.</td></tr>
</table>



Descriptive parameters associated with the selected material.

Definition of the parameters of the equivalent layer

$$E_{eq} = \frac{\sum_{i=1}^{3} s_i E_i}{s_{eq}}$$

$$G_{eq} = \frac{\sum_{i=1}^{3} s_i G_i}{s_{eq}}$$

$$w_{eq} = \frac{\sum_{i=1}^{3} s_i w_i}{s_{eq}}$$

$$f_{m,eq} = E_{eq} * MIN(\frac{f_{m1}}{E_1}; \frac{f_{m2}}{E_2}; \frac{f_{m3}}{E_3})$$

$$\tau_{eq} = G_{eq} * MIN(\frac{\tau_1}{G_1}; \frac{\tau_2}{G_2}; \frac{\tau_3}{G_3})$$

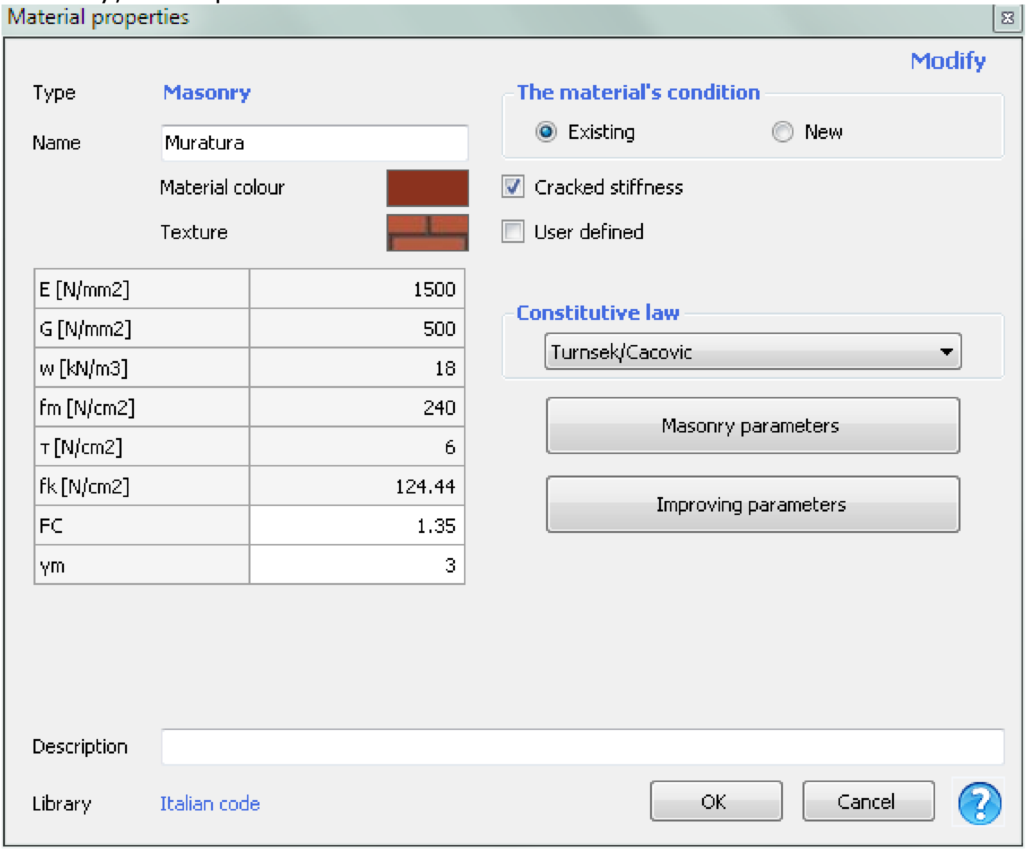
$$f_{k,eq} = E_{eq} * MIN(\frac{f_{k1}}{E_1}; \frac{f_{k2}}{E_2}; \frac{f_{k3}}{E_3})$$

$$f_{v0,eq} = G_{eq} * MIN(\frac{f_{v0,1}}{G_1}; \frac{f_{v0,2}}{G_2}; \frac{f_{v0,2}}{G_3})$$

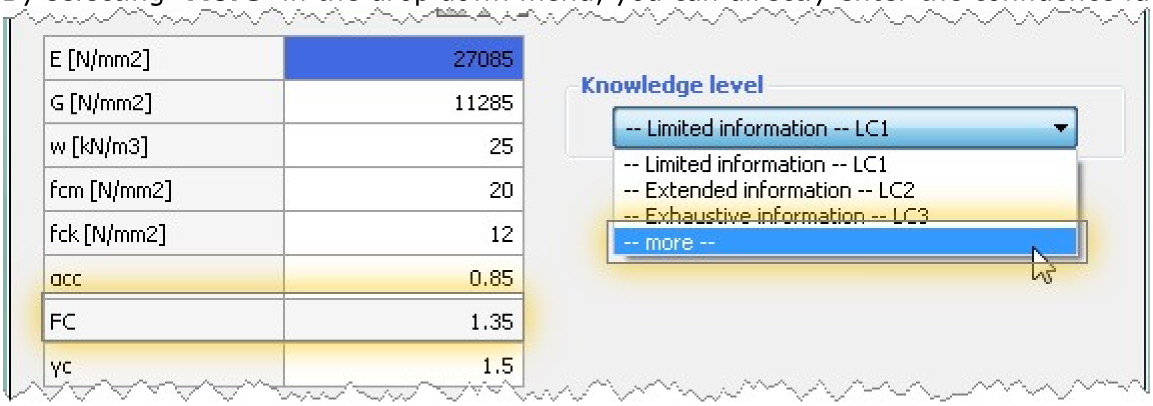
$$f_{v,lim} = 0.065 * f_{m,eq}$$

9.1.2 Other materials

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



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



9.1.3 Materials Library

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

- Library Project: Materials collection contained in this project, shown in the material dialog window (these materials are only available for the active project).

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

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

User Library

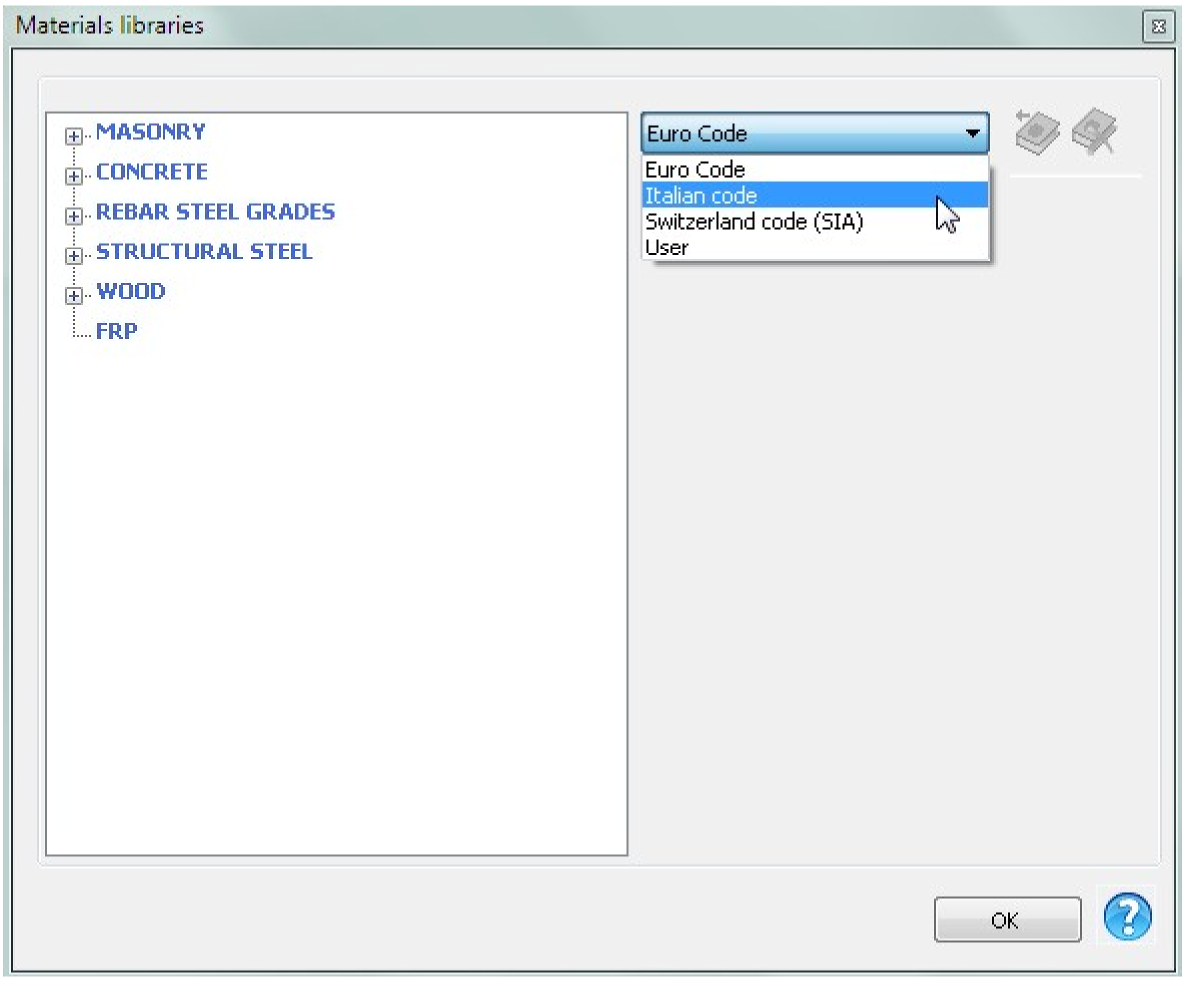
After defining a new or existing material, will be shown in the tree to the left of the window material.

The defined material is now available within the project, if this material is usually re-used for other projects different from the project on which you are working, you can save it on the user library to be able to retrieve and use later in different models.

To use the material created in the current model or in a different model, after you create it you must select the name and press "save in the library" 

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

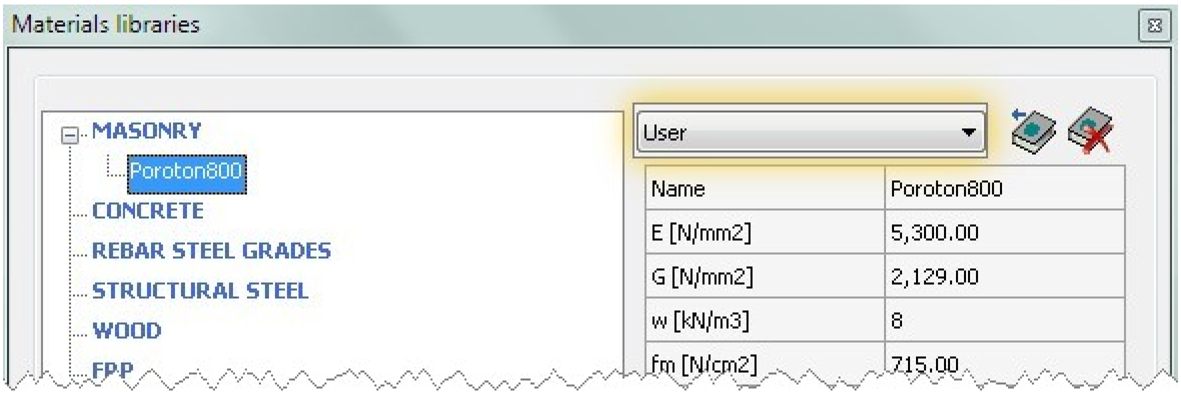
Library: Open the Material Library

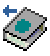


The materials presented in the tree on the left are those in the selected library in the

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

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



 **Copy in the project**
Allows to copy the selected material in the project library, making it available for the active project.
**Delete from library:**
Allows to delete the selected material from the user library.

9.2 Definition of Structural Objects

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

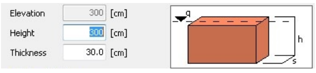
*To refine the computation procedure, the program examines the **non-linear behavior** of the elements. (see the theoretical information found in the introduction) Given the definite non-linear behavior of the macro-elements, it is necessary to perform an mixed structural analysis that is sufficiently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

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

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

necessary to insert the characteristics of the concrete and the steel.
The geometry area changes depending on the element and it is described in detail below.

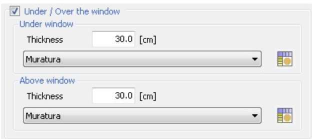
9.2.1 Simple Elements



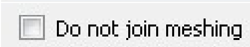
-*Elevation*: The maximum elevation of the panel
-*Height*: Height of the masonry panel calculated from the maximum elevation point with the downward direction.
-*Thickness*: Defines the wall thickness.

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

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



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

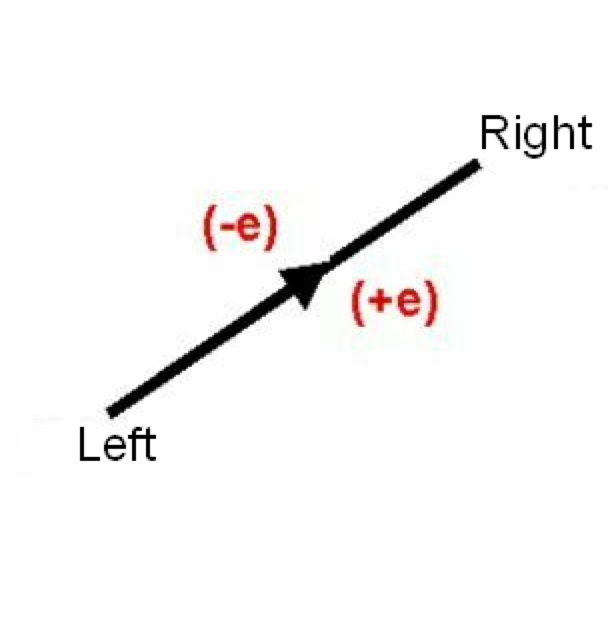


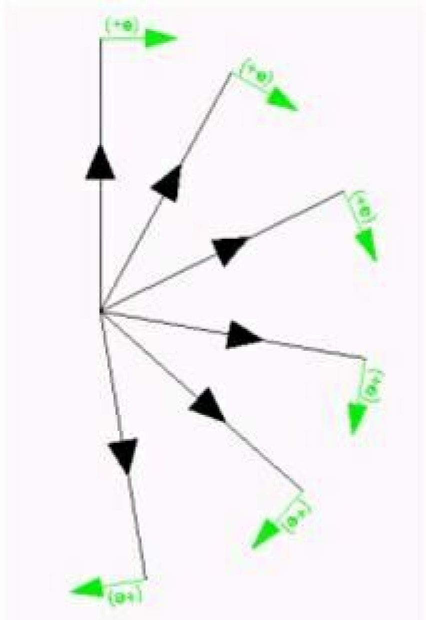
Two adjacent wall panels separated by a node element with the same geometric/mechanical characteristics are normally joined during the mesh generation.
With this option during the mesh generation the two masonry walls will remain separated.

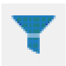
If you want a wall to have only seismic functions but not floor masses transfer you can activate this box.

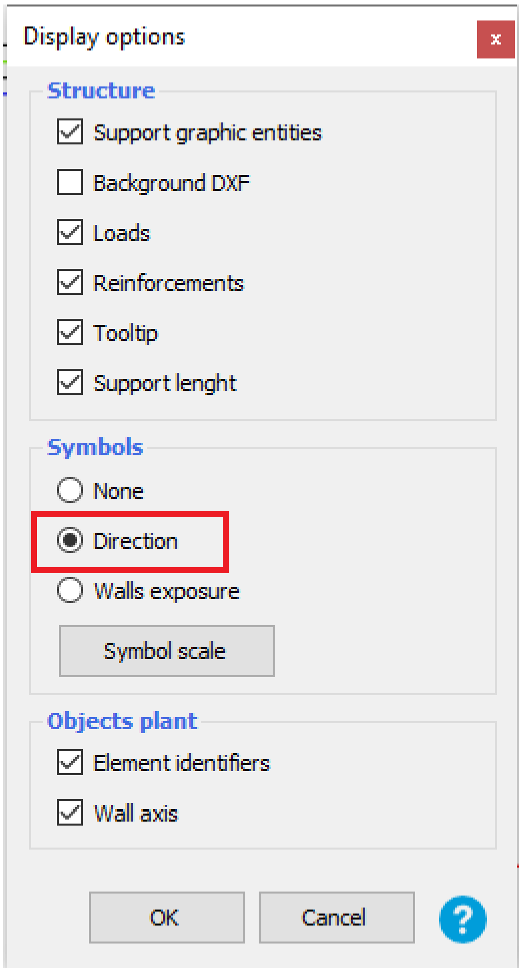
Defines the opportunity to decide that the node at the base of the panel is part of the foundation and the ability to insert the foundation's characteristics.

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

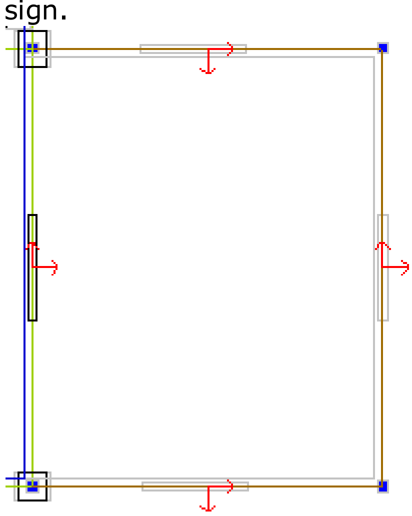




With the display options button you can choose to show the local reference system directly on the model plan.

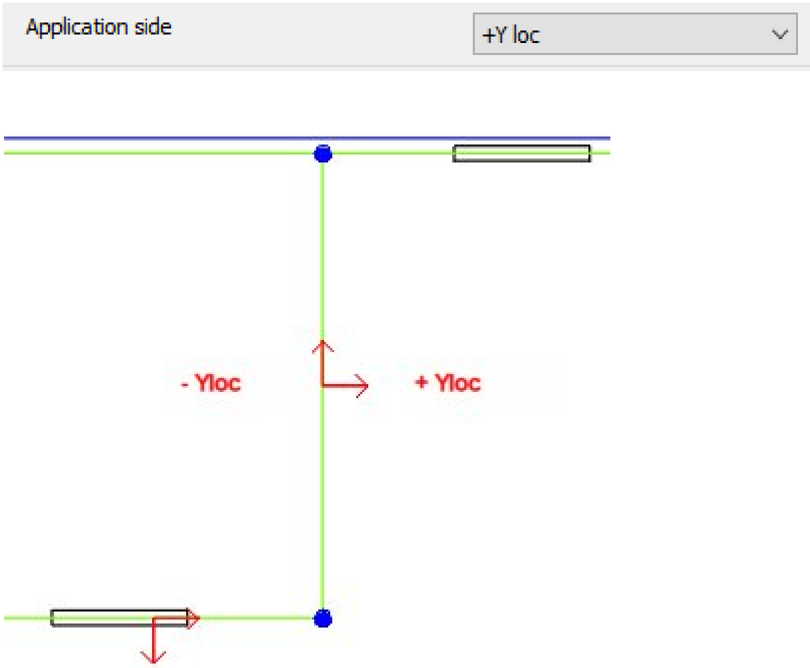


Using the option "Direction", the local reference system is shown on each wall, allowing individuation of the eccentricity sign.



The reference system also makes it possible to understand the application side of the elements. The identification of the application side depends on the orientation of the Y axis of the reference system.

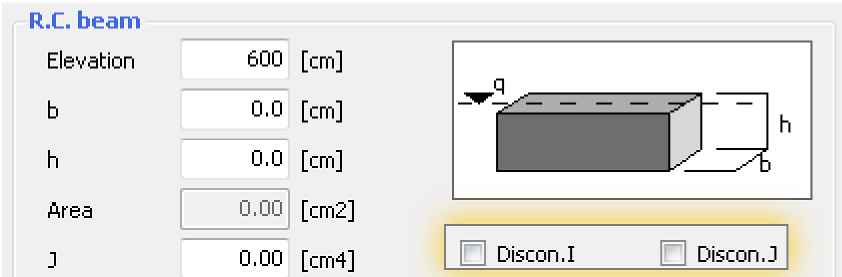
It is possible to choose the application side using the "application side" option present in the element characteristics definition windows:



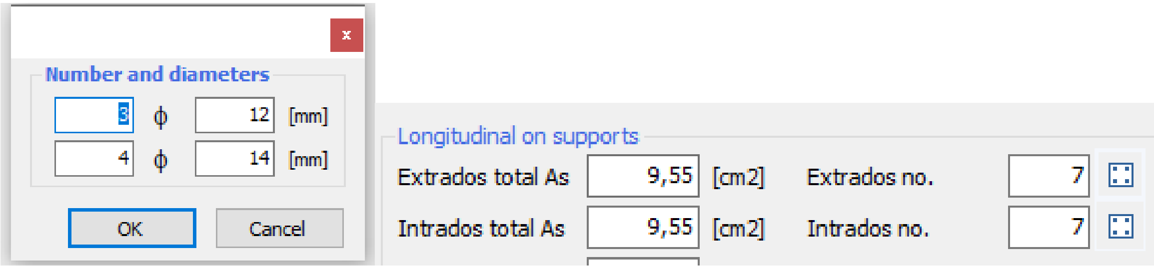
9.2.1.2 R.C. beam

-*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.

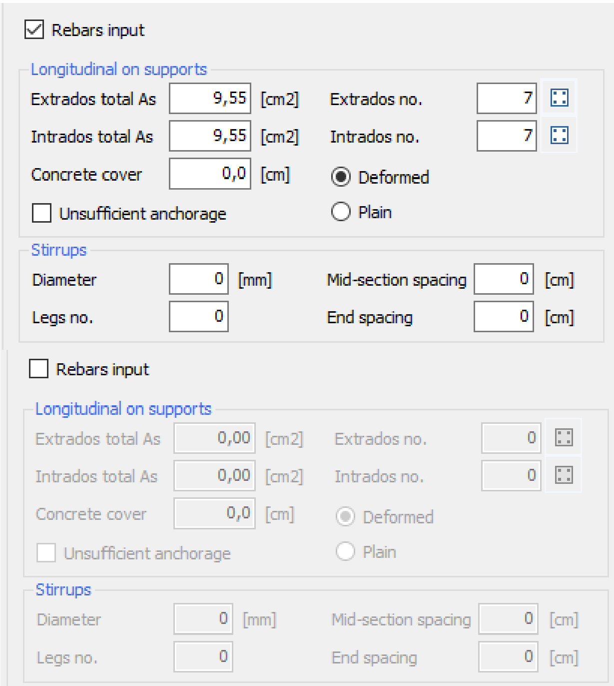
This function allows the designer to define constraints for leaning, by inserting internal hinges, also in the non-linear field.
Insertion of disconnections is managed using the associated tick boxes.
I and J indicate, respectively, the first and second wall segment ends, with respect to the sign convention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



This button helps the user to insert the amount of reinforcement (quantity and diameter).When choosing the number and diameters of the reinforcement (3?12+4?14 on the example below) the software automatically calculates the total area and number of rebars.



- Reinforced concrete beams insertion



By selecting the "rebars input" option, the quantity of longitudinal and transverse reinforcement must be manually defined by the user

When unselecting the "rebars input" option, it is not necessary to insert the reinforcement parameters.

Reinforcement is mandatory for the pushover calculation and can be defined in two ways:
-Manually, by activating this option.
-Using the environment [Local verification] (the element will be pre-dimensioned).

At the end of the pre-dimensioning, the reinforcement quantities are automatically reported in the appropriate spaces.

NB: The pushover is a non-linear calculation, for this reason the presence of

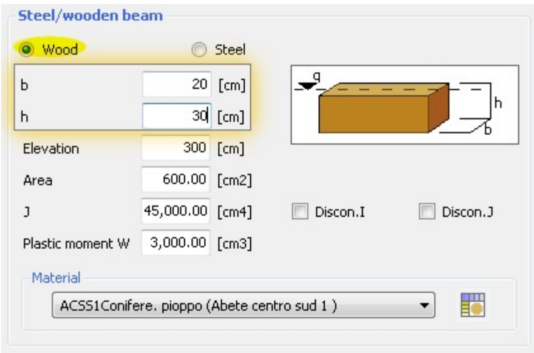
the reinforcement is fundamental.
Other types of calculation present in the program are linear (eg. analysis for vertical loads only), therefore reinforcement is not necessary.

9.2.1.3 Steel/wooden beam

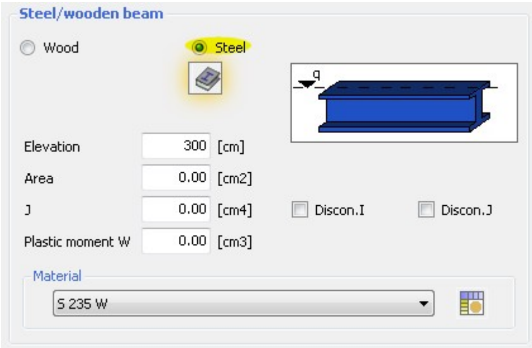
-*Elevation*: Identifies the elevation of the two ends of the beam, in order to allow the insertion of the inclined beams. In this version of the program, only horizontal beams can be inserted (I Elevation = J Elevation)
-*Geometric characteristics of the section*: area, inertia, and plastic resistance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

Why is the Axial Discon. option only available in the roof environment?

This option is present only in the roof environment and not in the structure environment because it is designed for a "rafter" type beam (typically made of wood). Normally, in wooden roofs the ridge beam is defined, and the rafter elements rest orthogonally on top of that ridge beam. If the axial stiffness of these elements is considered, it would happen that the rafter element would "help" the ridge beam to support the loads (behaving like a truss). In most cases, the ridge beam should carry all the loads from the roof (the rafters rest on top of the ridge beam). Neglecting the axial stiffness of the rafters allows the vertical component of the load on the ridge beam not to be transmitted to the rafters, and consequently allows the ridge beam to support all the rafters.



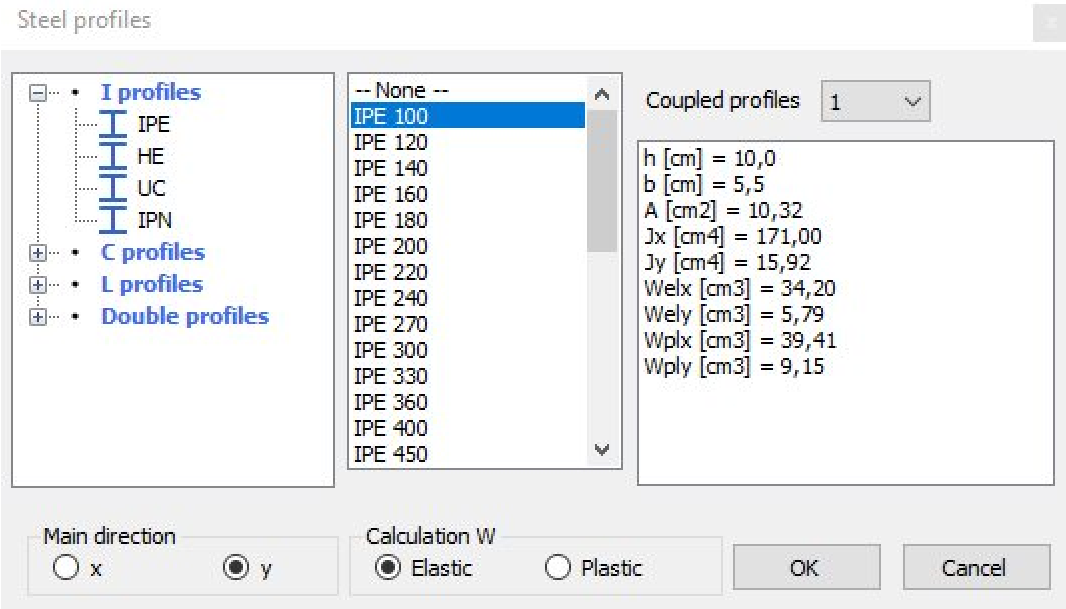
By selecting [**Wood**] is immediately available the space to insert the base and height of the section, the calculation of A, J, and W is automatic.
For non-rectangular sections you can directly enter the values of A, J and W.



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

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

With this button it is possible to recall the steel profiles library from where you can get the features.
Selecting the family and profile size,the mechanical characteristics that will be used in the calculation are presented.



I profiles
IPE
HE
UC

It is also possible to generate a section composed of several profiles by disposing of up to 4 side by side.

C profiles
UPN
UAP
UPE

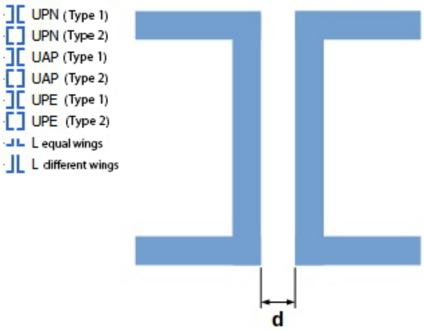
Given the absence of double symmetry in this family of profiles, the side-by-side the disposition will not be possible, but it will be possible to arrange them as "double" according to the configurations shown below.

L profiles
L equal wings
L different wings

Given the absence of double symmetry in this family of profiles, the side-by-side the disposition will not be possible, but it will be possible to put them as "double" according to the configurations shown below.

Double profiles

The previous families are disposed in the various "double" configurations available, indicated by the respective icon.
For each configuration it is also possible to define the distance (d) between the two profiles.



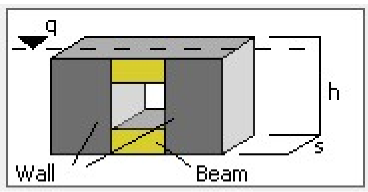
It is also possible to generate a section composed of up to 4 profiles disposed side by side.
The option "profiles disposed side by side" is not available for the pillars.

9.2.1.4 R.C. wall

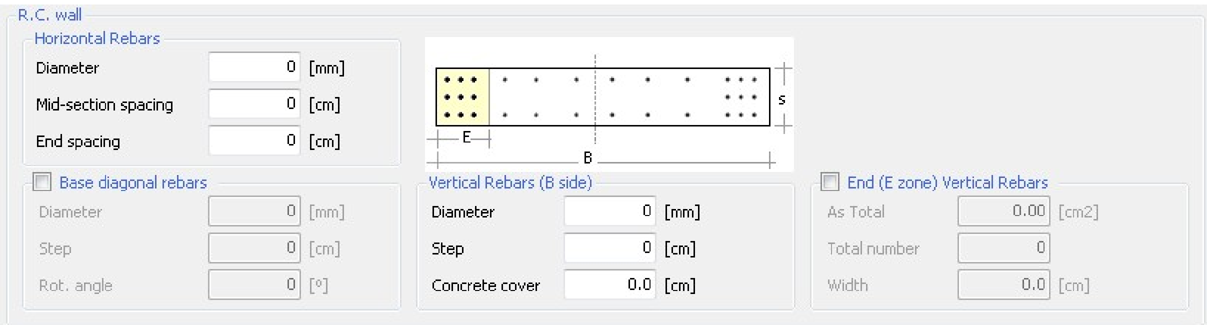
The first step for R.C. walls insertion is definition of the general data:
- Elevation: the maximum elevation of the R.C. wall
- Height: height of the R.C. wall, calculated from the point of maximum elevation to the ground.
- Thickness: thickness of the R.C. wall.

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

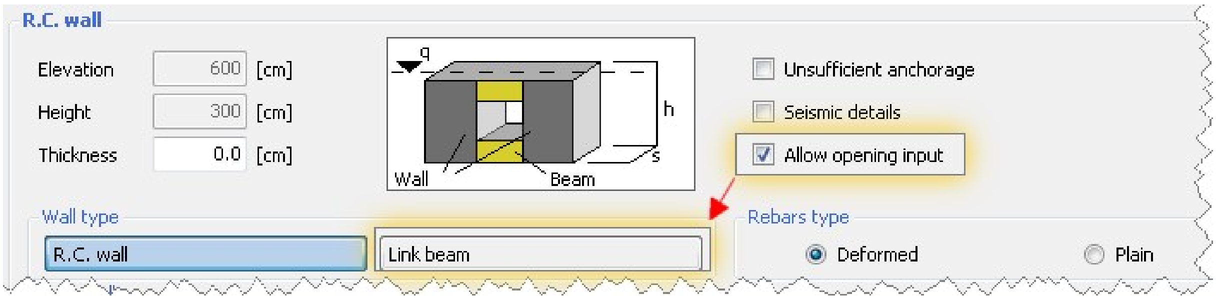
R.C. walls are inserted defining the wall and the connection beam (see figure below):



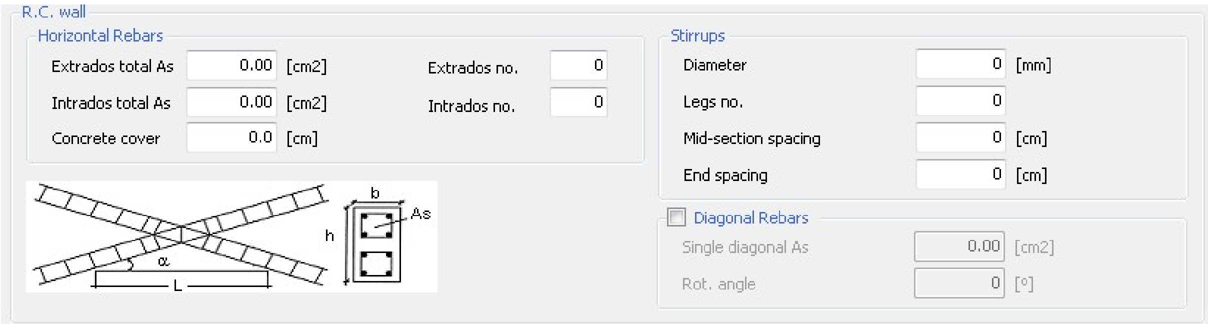
<u>Wall</u>:
-Diameters, steps and concrete cover for the vertical and horizontal rebars.
-Possibility to define different vertical reinforcements in the extreme areas (E zone)
-Diameter and steps of the diagonal based rebars



<u>Connection beam:</u>
The insertion of the connection beams is allowed only if you select [Allow insertion opening]



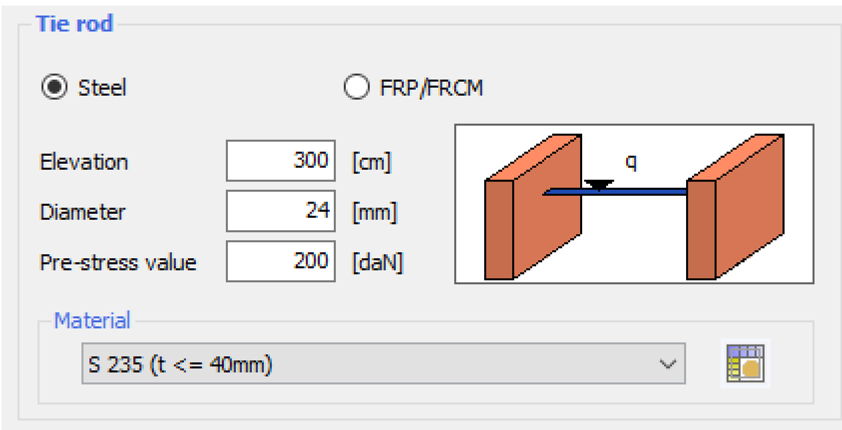
-Diameters, steps and concrete cover for the vertical and horizontal rebars.
-Possibility to define different vertical reinforcements in the extreme areas (E zone)
-Diameter and steps of the diagonal based rebars
-Reinforcement: Area of the longitudinal reinforcements and number of rebars, distinguished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



9.2.1.5 Tie rod

Steel tie rod

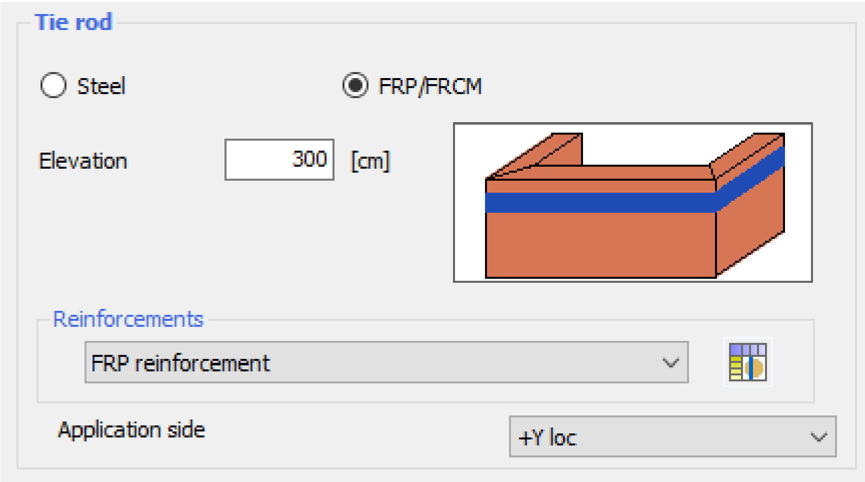
The tie rod is a tension-only element that can be defined using the following parameters:
- Elevation: Elevation in which the tie rod is placed.
- Diameter: Diameter of the steel bar that constitutes the tie rod
- Pre-stress value: Tension of the tie rod
- Material: Material of which the tie rod is made



FRP/FRCM tie rod

The tie rod is made up of a composite material and can be defined using the following parameters:

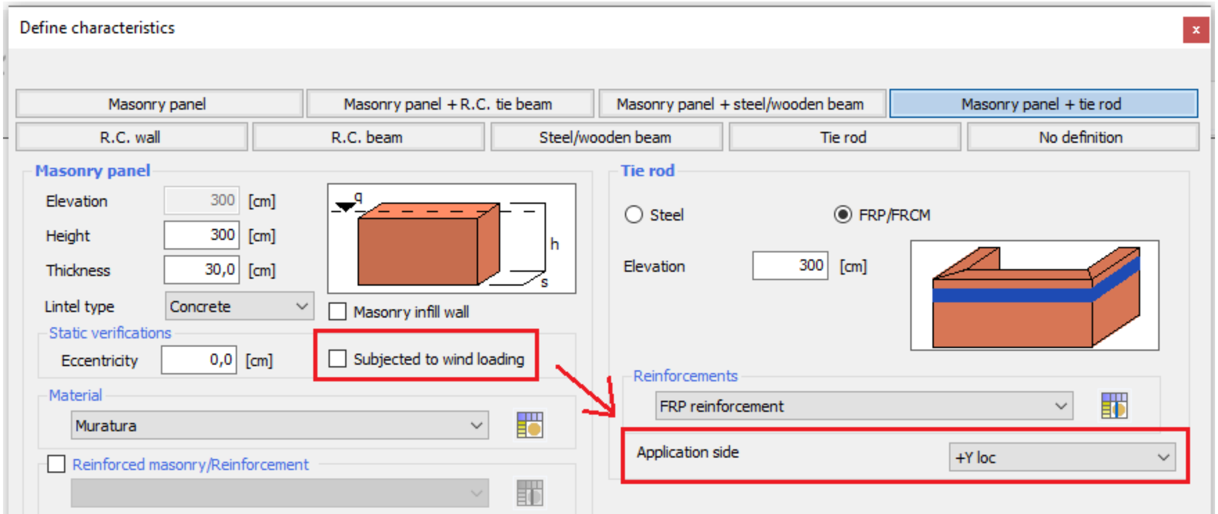
- Elevation: Elevation in which the tie rod is placed.
- Reinforcements: Type of FRP/FRCM material
- Application side: allows you to choose the side (internal / external) on which to apply the tie rod.



The choice of the application side differs depending on whether the panel in question is exposed / not exposed to wind.

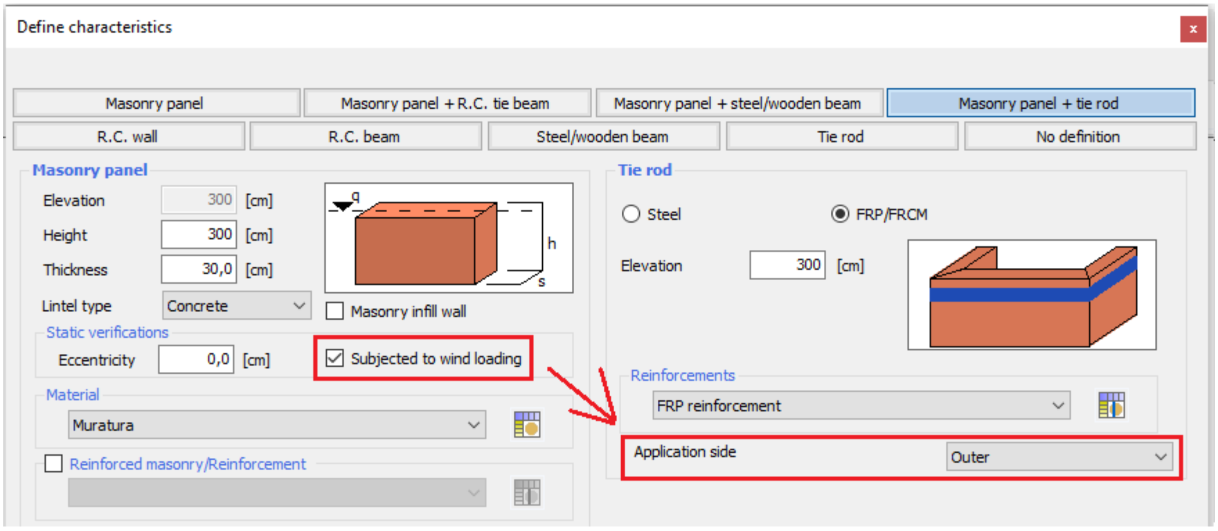
Wall not exposed to wind

The application side is based on the orientation of the reference system associated with the element.



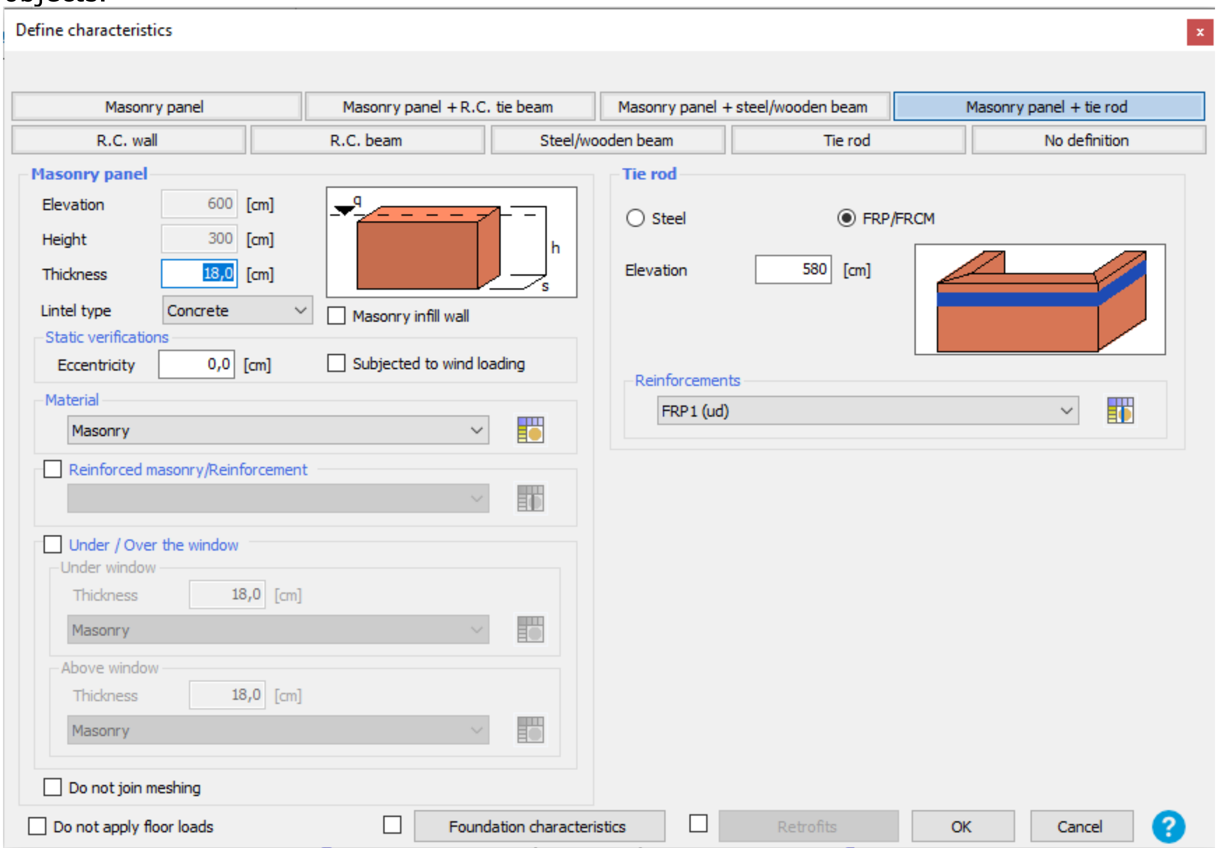
Wall exposed to wind

The application side is based on the exposure of the element sides.



9.2.2 Composite Elements

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

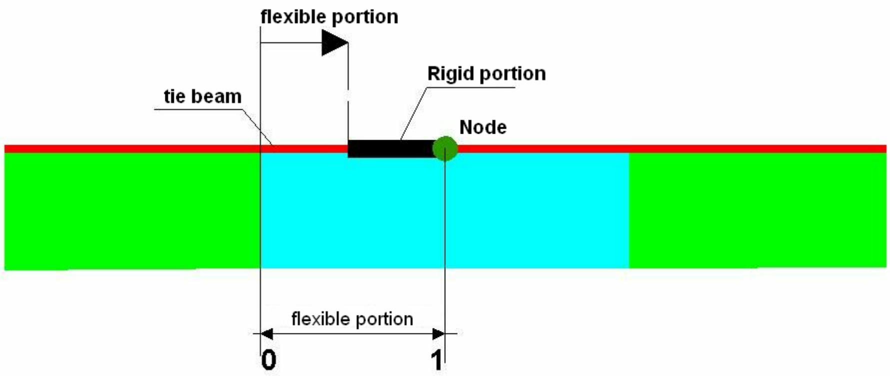


Once chosen the structure type you can edit the geometric characteristics of the element and access the materials library.

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

9.2.2.1 Masonry panel + Tie beam

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



9.2.2.2 Masonry Panel + Beam

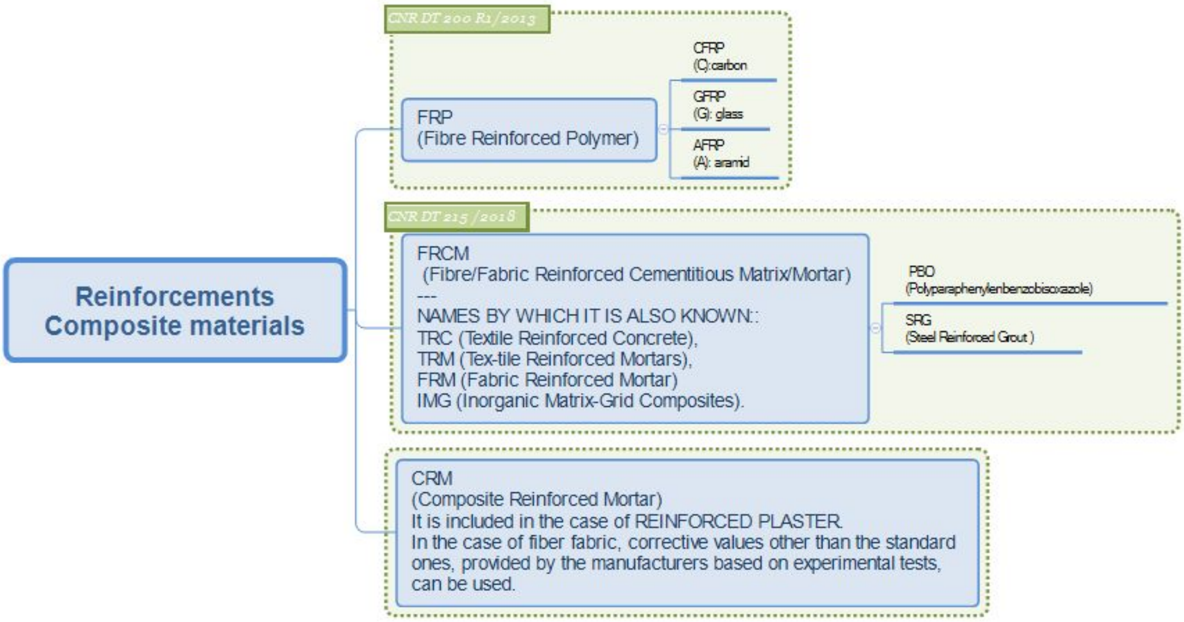
This is a masonry panel paired with a steel or wood beam. The parameters that must be inserted are the same as those for the elements taken individually.

9.2.2.3 Masonry Panel + Tie Rod

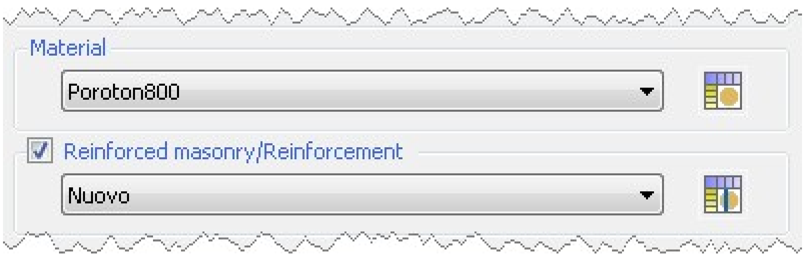
This is a masonry panel paired with a tie rod. The parameters that must be inserted are the same as those for the elements taken individually.
The combined elements are very useful for strengthening masonry panels with elements such as tie beams, steel or wood beams, or tie rods.

9.2.3 Reinforced masonry reinforcements/FRP/FRCM_2

Before going into the calculation procedures of the elements reinforced with FRP or FRCM, it is good to clarify the various types of existing "reinforcements with composite materials" with a diagram.

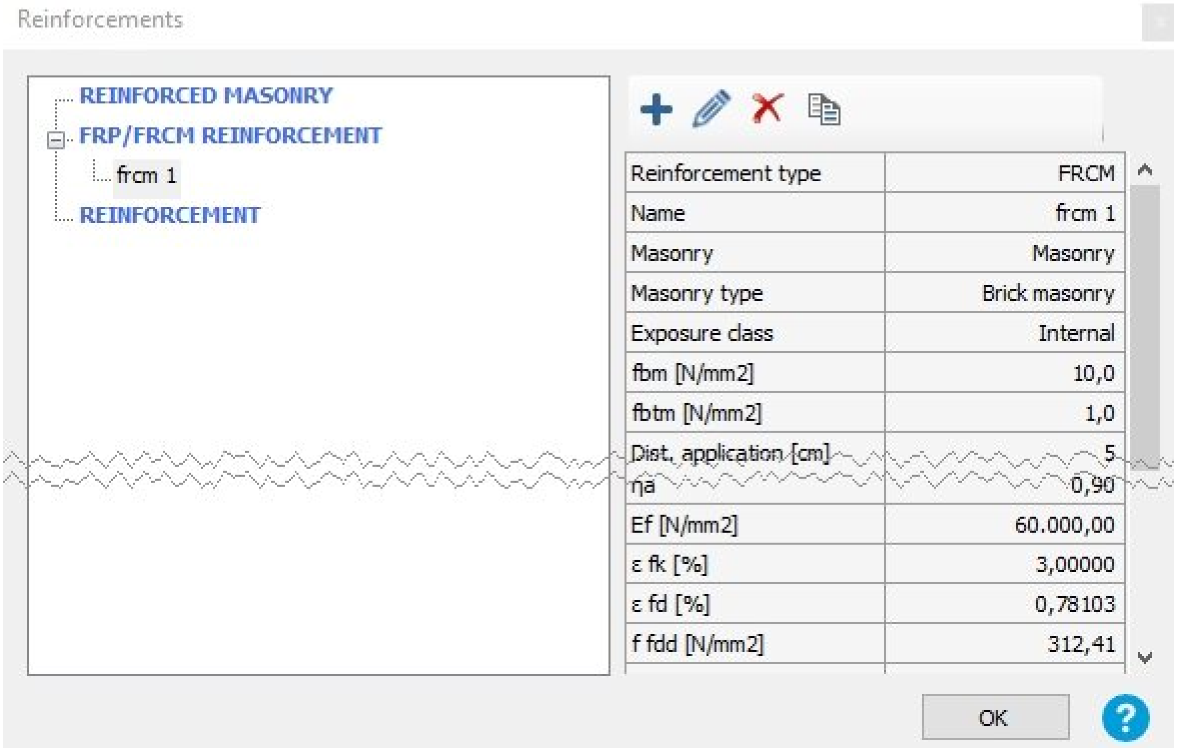


All structural items, such as "masonry panel" and those with it compounds, contain in their definition the possibility of defining reinforcement systems.



Activating the checkbox enables the possibility to define the typology of reinforcement.

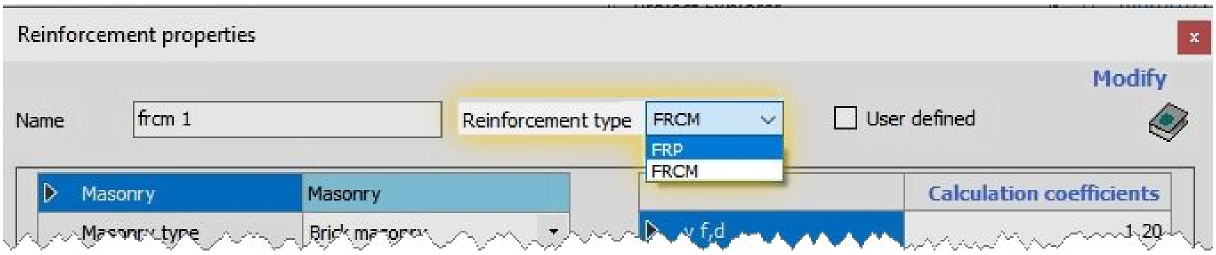
Reinforcements library:



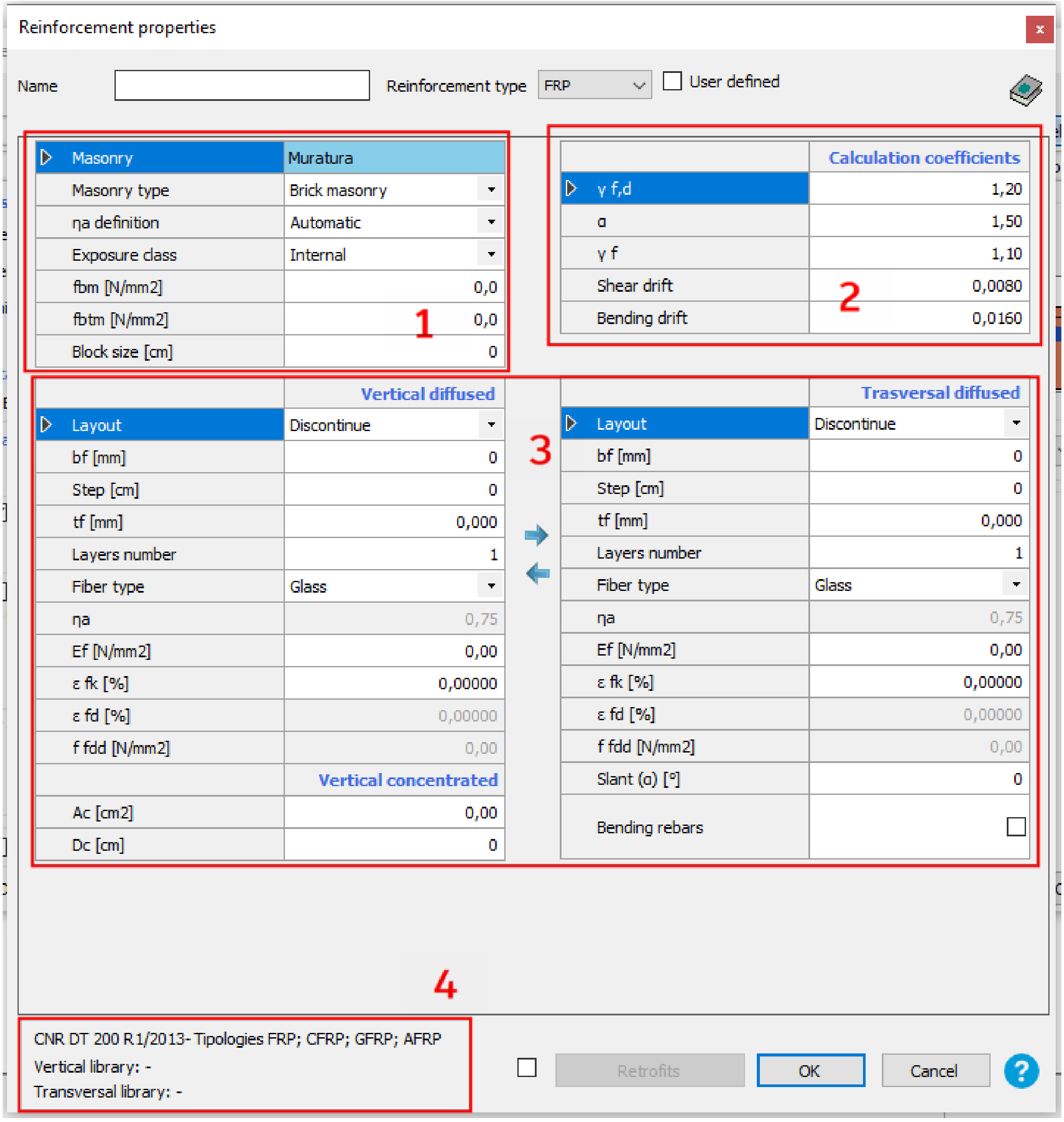
The three main typologies of reinforcement are presented here:
- Reinforced masonry
- FRP/FRCM reinforcement
- Reinforcement

9.2.3.1 FRP reinforcement

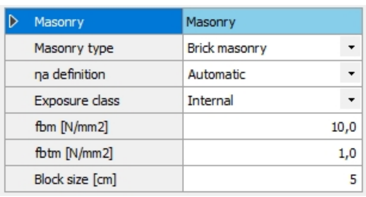
+ FRP/FRCM reinforcement:
By choosing the "insert" option the following mask appears within which we could see all the necessary input data collected.
In the upper part, a list with the FRP/FRCM items shows the typology of reinforcement selected.



Specifically, by leaving the FRP session active, all the input required below will comply with the current FRP legislation which is the CNR-DT 200 R1/2013.



1



It shows the name of the material from which it was created
It depends on the type of masonry on which it is applied
Modality for entering the environmental conversion factor, if automatic, regulatory procedures are used, if manual, the value is entered directly manually.
Required parameter to define the environmental conversion factor (CNR DT200 R1 / 2013, Table 3.2)
Average compressive strength
Tensile strength of the block
Width of the reinforced element

2

	Calculation coefficients
γ f,d	1,20
α	1,50
γ f	1,10
Shear drift	0,0080
Bending drift	0,0160

CNR DT200 R1/2013, § 3.4.1
Intermediate separation reduction factor
CNR DT200 R1/2013, § 3.4.1

3

Layout	Discontinue (Discontinue / Continue)

Type of fiber application.
The reinforcement can be applied "continuous" or "discontinuous".
Depending on the selected option, the input parameters will be different

bf [mm]	0
Step [cm]	0
tf [mm]	0,000
Layers number	1
Fiber type	Glass
ηa	0,75
Ef [N/mm2]	0,00
ε fk [%]	0,00000
ε fd [%]	0,00000
f fdd [N/mm2]	0,00
	Vertical concentrated
Ac [cm2]	0,00
Dc [cm]	0

Width of the reinforcement
Strips step, measured orthogonally to the direction of the fibers
Thickness of the reinforcement
Number of layers of the reinforcement
Fiber material (glass, aramid, carbon, steel)
Automatically calculated value if "automatic mode" is requested in the previous session
Normal elasticity modulus in the direction of the force
Characteristic strain
Maximum strain attributable to the reinforcement
Tension design value of the reinforcement separation

Area of the concentrated reinforcement
Reinforcement/panel edge distance

The transversal reinforcement has the same information as the vertical one, with the addition of the following two fields.

Slant (α) [°]	0
Bending rebars	☐

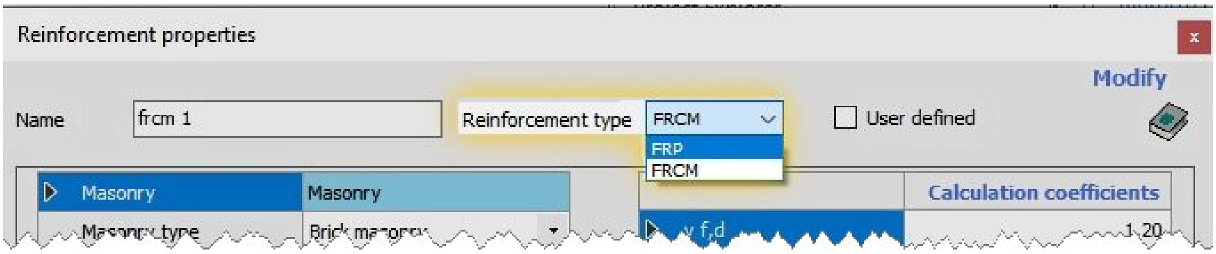
Inclination angle
Parameter that indicates the presence or absence of the bending reinforcement

4

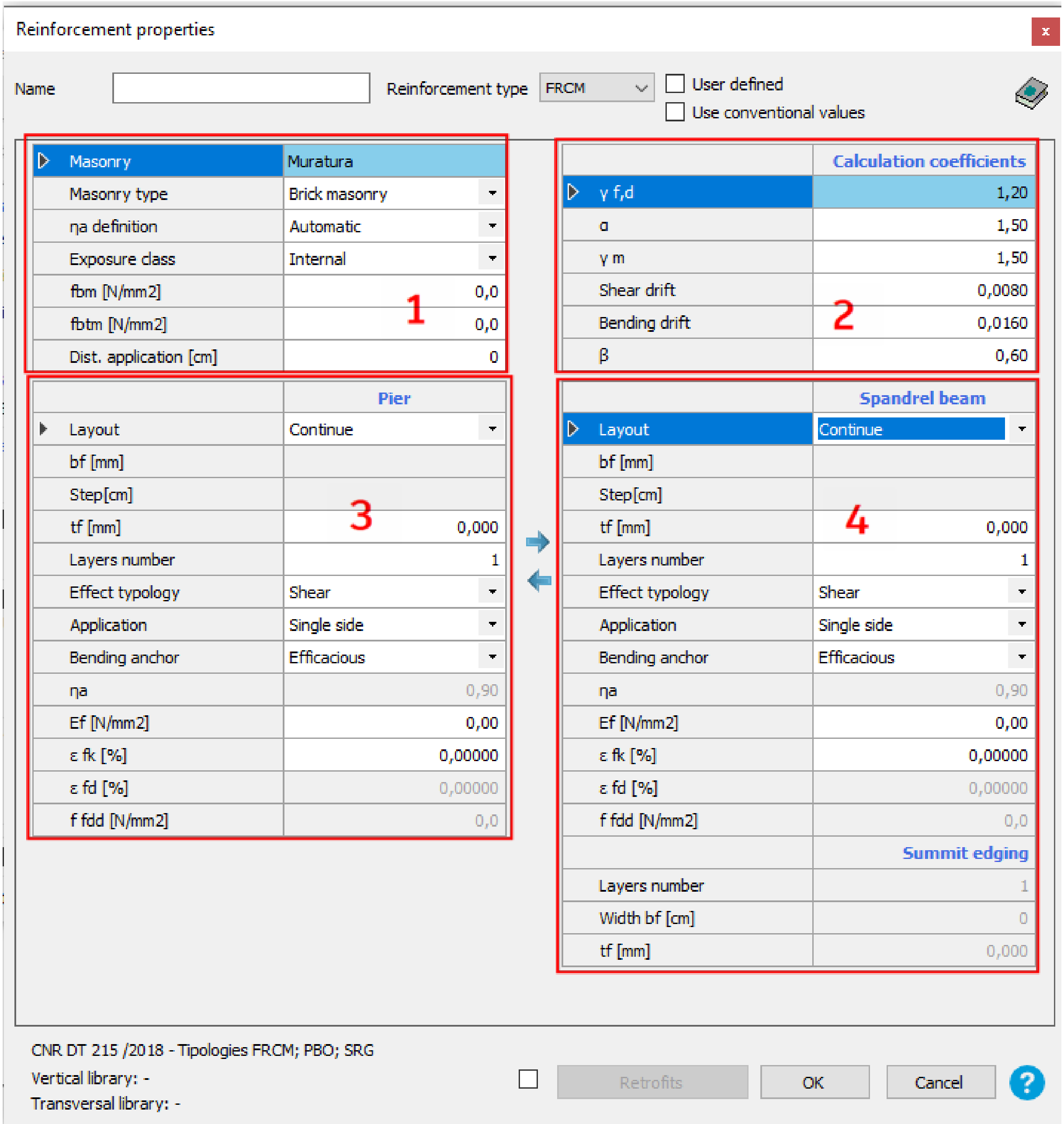
Informative section

9.2.3.2 FRCM reinforcement

FRP/FRCM reinforcement:
By choosing the "insert" option the following mask appears within which we could see all the necessary input data collected.
In the upper part, a list with the FRP/FRCM items shows the typology of reinforcement selected.



Specifically, by leaving the FRCM session active, all the input required below will comply with the current FRCM legislation which is the CNR-DT 215/2018.



The information in the CNR DT215/2018 are not always thorough for the calculation, for this reason some references refer to the CNR-DT 200 R1/2013 for completeness.

1

Masonry	Masonry
Masonry type	Brick masonry
ηa definition	Automatic
Exposure class	Internal
fbm [N/mm2]	10,0
fbtm [N/mm2]	1,0
Dist. application [cm]	5

It shows the name of the material from which it was created
It depends on the type of masonry on which it is applied
Modality for entering the environmental conversion factor, if automatic, regulatory procedures are used, if manual, the value is entered directly manually.
Required parameter to define the environmental conversion factor (CNR DT215/2018, Table 3.1)
Average compressive strength
Tensile strength of the block
Distance from the edge of the reinforcement application

2

	Calculation coefficients
γ f,d	1,20
α	1,50
γ f	1,10
Shear drift	0,0060
Bending drift	0,0120
β	0,60

CNR DT200 R1/2013, § 3.4.1
Intermediate separation reduction factor
CNR DT200 R1/2013, § 3.4.1

CNR-DT 215/2018, §12 Appendix 1

3-4 Pier and Spandrel beam

Layout	Discontinue (Discontinue / Continue)

Type of fiber application.
The reinforcement can be applied "continuous" or "discontinuous".
Depending on the selected option, the input parameters will be different.

bf [mm]	
Step[cm]	
tf [mm]	0,000
Layers number	1
Effect typology	Shear
Application	Single side
Bending anchor	Efficacious
ηa	0,90
Ef [N/mm2]	0,00
ε fk [%]	0,00000
ε fd [%]	0,00000
f fdd [N/mm2]	0,0

Reinforcement width (only if discontinuous application)
Step of the strips, measured orthogonally to the direction of the fibers (only for discontinuous application)
Equivalent thickness of the reinforcement element
Total number of layers
Application on a single side or on both sides for the shear
CNR-DT 215/2018, §4.1.1
Equivalent thickness of reinforcement element
Quality of the bending anchor
Automatically calculated value if "automatic mode" is requested in the previous session
Normal elasticity modulus in the direction of the force
Characteristic strain
Maximum strain attributable to the reinforcement
Tension design value of the reinforcement separation

Active session if Effect type=edging
Number of horizontal layers on which the reinforcement is placed
Reinforcement width which usually coincides with the width of the edging

Equivalent thickness of the reinforcement

The calculation of the summit edging follows what is indicated in CNR-DT 215/2018, §4.3.

Use conventional values

In the upper part of the window, the option "use conventional values" activates the possibility to enter directly conventional values.

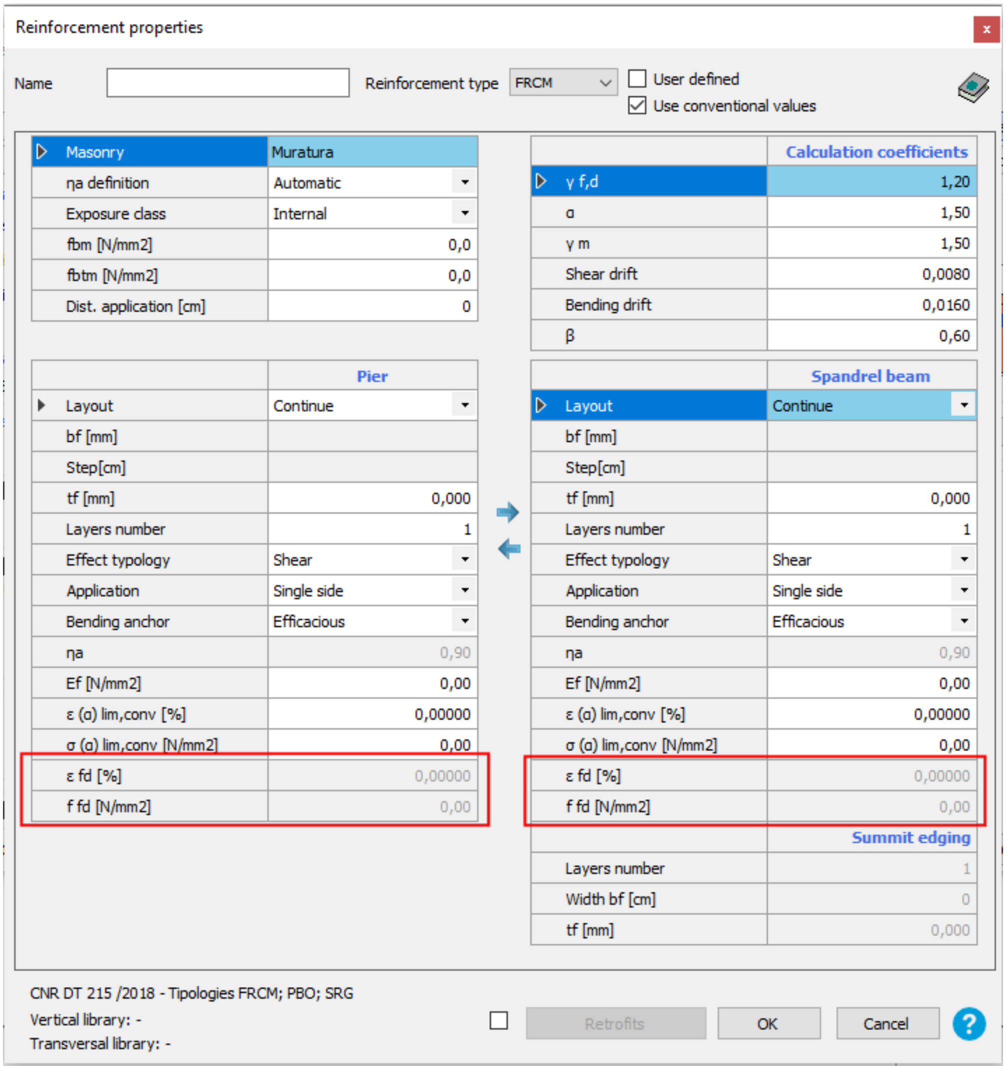
	Pier
Effect typology	Shear
Application	Single side
Bending anchor	Efficacious
Layers number	1
tf [mm]	0.000
Area/m [mm2/m]	0.00
ηa	0.90
Ef [N/mm2]	0.00
ε (α) lim,conv [%]	0.00000
σ (α) lim,conv [N/mm2]	0.00
ε fd [%]	0.00000
f fd [N/mm2]	0.00

For more information on conventional values, refer to the chapter of the mechanical parameters of composites.

9.2.3.3 Mechanical parameters of composites

The mechanical parameters in terms of strain and limit strengths, to be used for the calculation of a reinforced wall panel, are not those of the dry fabric but depend on innumerable factors.

The program has an internal procedure which, given the product parameters and some complementary information, calculates the necessary parameters for the verification using the indications of the CNR DT200 R1/2013.
The user can take advantage of this procedure, or select the "user definition" item and manually enter the parameters for the calculation.
In the case of manual insertion, all the "complementary" parameters will not be required, but it will be essential to define the strains and strengths.



If the user leave the program to calculate strengths and strains, the calculation procedure is indicated below with the corresponding regulatory references relating to *checks on detachment from the support*.

CALCULATION of K_b - CNR DT200 R1/2013, § 4.1.2

b_f

b

b_f/b

$$k_b = \sqrt{\frac{3 - \frac{b_f}{b}}{1 + \frac{b_f}{b}}}$$

CALCULATION of Γ_{fd} - CNR DT200 R1/2013, § 4.1.2

k_b, k_G, FC, f_{bm}, f_{btm}

$$\Gamma_{Fd} = \frac{k_b \cdot k_G}{FC} \cdot \sqrt{f_{bm} \cdot f_{btm}}$$

K_B is a corrective coefficient of geometric type;

K_G is an additional correction coefficient calibrated on the basis of experimental results.

It is assumed to be equal to:

- 0.031 mm, in the case of brick masonry
- 0.048 mm, in the case of tuff masonry
- 0.012 mm, in the case of limestone masonry or Lecce stone

In the absence of experimental data f_{btm} can be assumed equal to 0.10 f_{bm}

CALCULATION of f_{fdd} - CNR DT200 R1/2013, § 4.1.3

$\gamma_{f,d}$, t_f, E_f, Γ_{fd}

$$f_{fdd} = \frac{1}{\gamma_{f,d}} \cdot \sqrt{\frac{2 \cdot E_f \cdot \Gamma_{Fd}}{t_f}}$$

Design tension of the reinforcement concerning to the separation of the extremity

CALCULATION of $f_{fdd,2}$ - CNR DT200 R1/2013, § 4.1.4

$$f_{fdd,2} = \alpha \cdot f_{fdd}$$

CALCULATION of ε_{fdd} - CNR DT200 R1/2013, § 4.1.4

$f_{fdd,2}$, E_f

$$\varepsilon_{fdd} = \frac{f_{fdd,2}}{E_f}$$

CALCULATION of ε_{fd} - CNR DT200 R1/2013, § 4.2.2.1

η_a

ε_{fk}

γ_f

ε_{fdd}

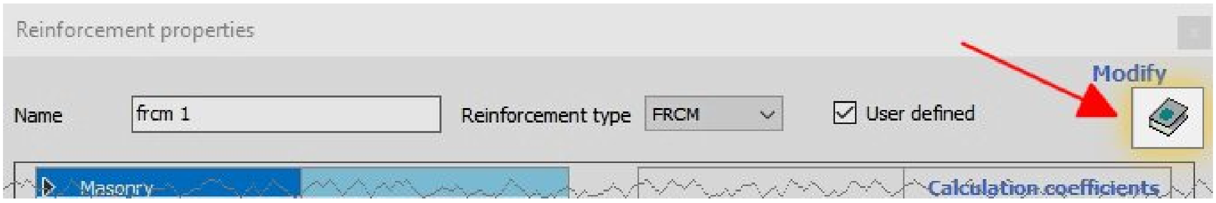
$$\varepsilon_{fd} = \mathrm{MIN}\left\{\eta_a {}^* \frac{\varepsilon_{fk}}{\gamma_f}; \varepsilon_{fdd}\right\}$$

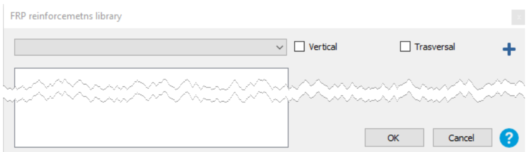
FRCM reinforcements:

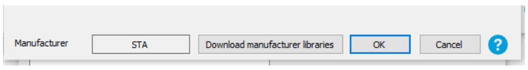
In the particular case of this typology of reinforcement, the reference standard is CNR-DT 215/2018.
In the chapter concerning the calculation of the mechanical characteristics (§3.1) of the aforementioned CNR, reference is made to the concept of "conventional" stresses and strains which represents the strength of the reinforcement system obtained by detachment tests from conventional supports and as such is dependent on the type of support.
The use of the conventional limit strain and the conventional limit tension competent, allows to design FRCM reinforcement interventions avoiding *the explicit verification in comparison of the detachment phenomenon from the support* or sliding of the fibers in the matrix in correspondence with the end of the reinforcement, otherwise necessary in cases where such crisis mode is possible.

9.2.3.4 Reinforcement library

In the upper right part, a button allows to access to the library of possible reinforcements.





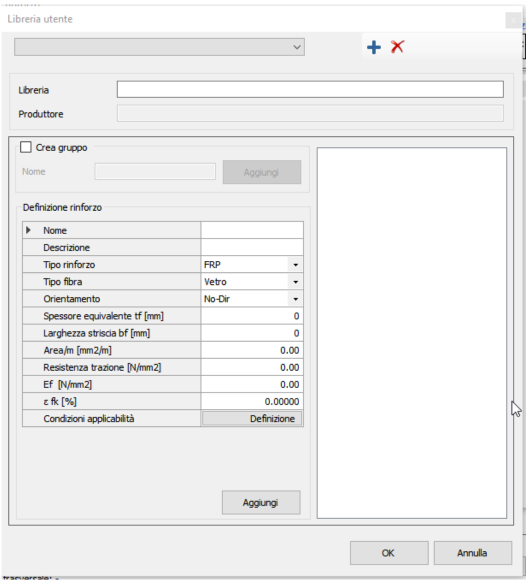


From the drop down window it is possible to view the present libraries and select the desired one.
By inserting the tick in the "vertical" and/or "transversal" options, it is possible to choose the arrangement of the reinforcement.
Using the "+" symbol it is possible to create a new library to be added to the list of those already present.
By clicking on the "Download manufacturer libraries" button it is possible to consult and download the manufacturer libraries.

Once downloaded, the new library will be visible in the dropdown shown above.

If, in the design practice, the designer always refer to a specific manufacturer, is possible to generate an own library which will be available for all future projects.

To create a new user library it is necessary to define some parameters:

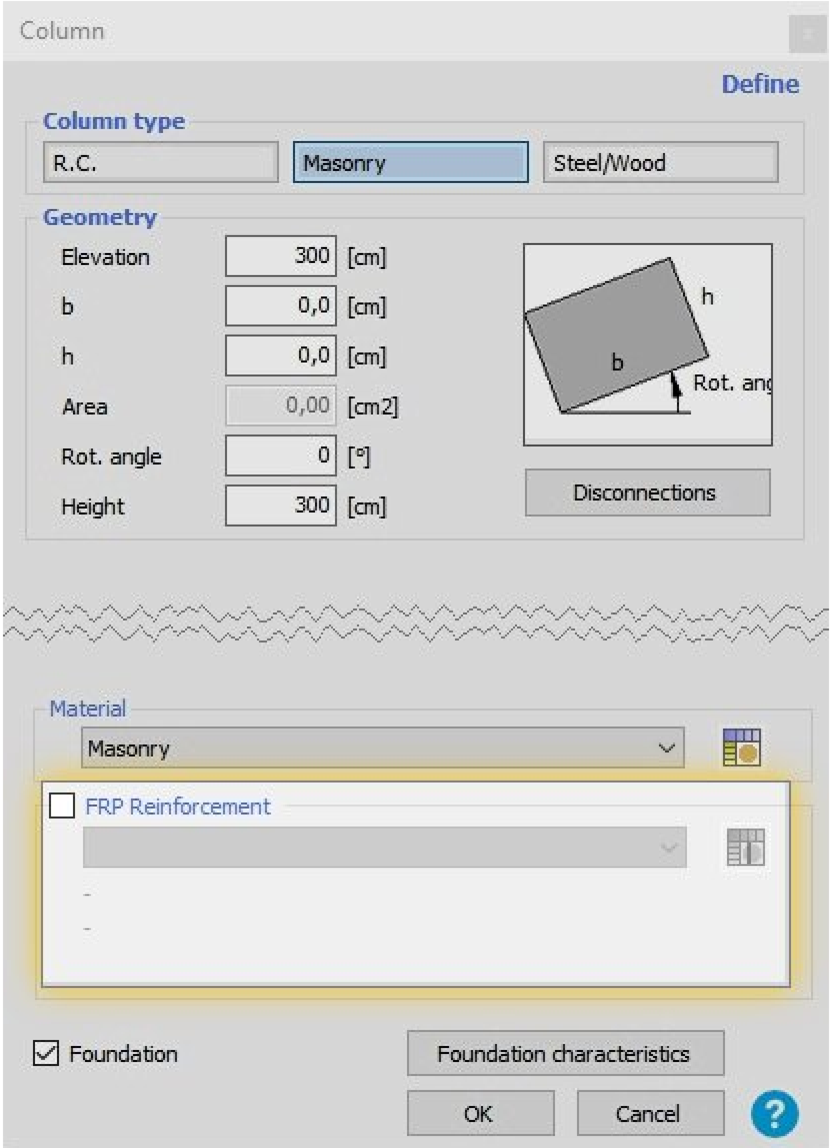


Some of these parameters are descriptive parameters (name, description), others are necessary data for the definition of the reinforcement type, the fiber type, the orientation (uni/bi-directional).
The following data (equivalent thickness, strip width, area, tensile strength, E_f and ε_{fk}) are all parameters available from the manufacturers' technical data sheets.

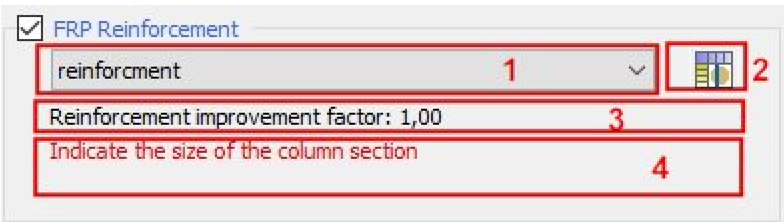
9.2.3.5 Reinforcement of masonry columns for confinement

It is possible to define reinforcements for confinement on **masonry** columns with FRP or FRCM composite materials.
Theoretical references:
FRP: CNR-DT 200 R1/2013 - §5.6.1
FRCM: CNR-DT 215/2018 - §4.4

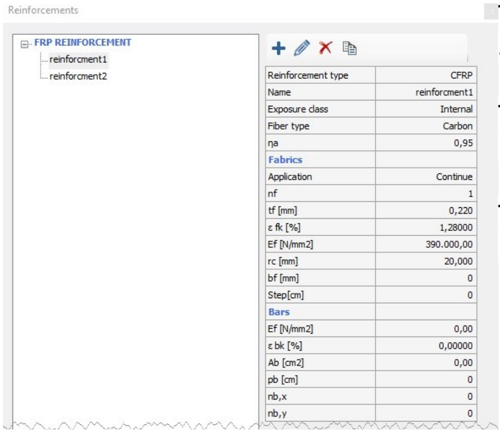
Reinforcement interventions for confinement act on mechanical performance, improving the compressive strength of the column.
The program calculates a "reinforcement improvement factor", consisting of a number greater than 1 which, multiplied by the compressive strength of the masonry, generates the improvement effect.
A session dedicated to FRP/FRCM reinforcement appears in the lower part of the window dedicated for inserting the masonry columns.



The reinforcement session is divided into 4 parts.
1. Reinforcement selection area among those present in the list, to be available in the list it must be defined in the definition environment.
2. Reinforcement definition environment
3. Reinforcement improvement factor
4. Information messages area, to warn in case of non-compliance with some indications contained in the normative text



Reinforcement definition environment



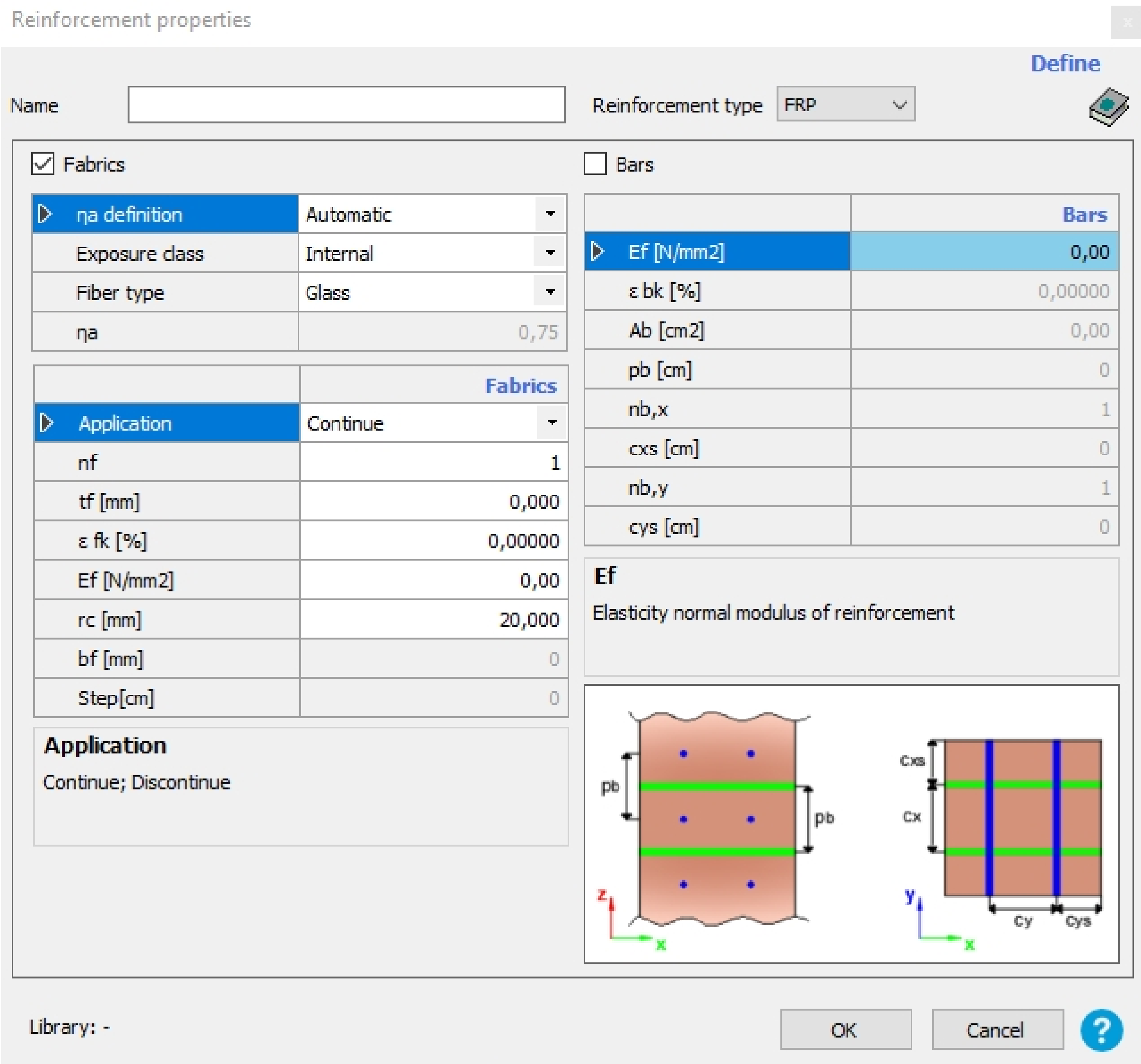
The left side indicates the list of reinforcements already defined.

The right part summarizes the characteristics of the reinforcement selected in the left list.

The command bar at the top allows to Insert, Modify, Delete or Copy a reinforcement.

Insert / Modify reinforcement element

By selecting the insert/modify command, the following mask is shown

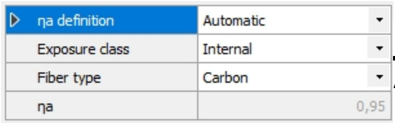


In the upper part, in addition to the field where is possible to define the name of the reinforcement, a drop-down window allows the choice of the FRP/FRCM typology.

The Reinforcement library button allows the access to the preloaded reinforcements.

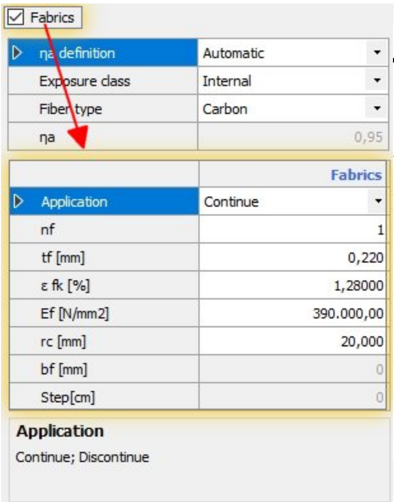
Based on what has been chosen in this drop-down window, the tables at the bottom show the parameters of the corresponding reinforcement.

With reference to the CNRs listed above, for the purposes of confinement, FRPs can be introduced both as fabrics and as bars; meanwhile the FRCM are provided only by fabrics.



Environmental conversion factor:

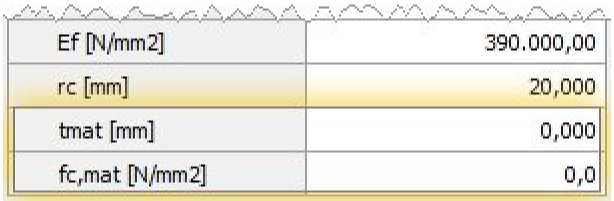
At the top left, a table allows the definition of the "environmental conversion factor".
In "Automatic" mode, it uses the CNR indications to calculate its value directly.
In "Manual" mode, the factor can be entered manually by the user.



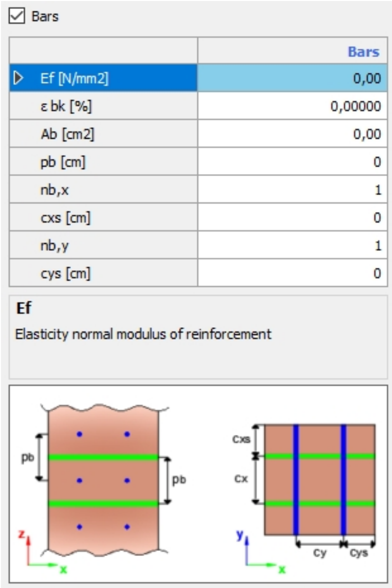
Fabrics:

By selecting the "Fabrics" item, the area relating to the input of these elements is activated at the bottom.
Application: Continue/Discontinue
nf: number of layers
tf: equivalent thickness of the single fabric
ε fk: ultimate strain
Ef: normal elasticity modulus of the reinforcement
rc: corner rounding radius
bf: [available only if the application mode is "Discontinue"]: Width of the reinforcement strip
step: [available only if the application mode is "Discontinue"]: Step of the reinforcement strip

In the case of FRCM reinforcement, the "Discontinuous" application mode is not contemplated and the last two lines of the table are replaced with two others typical of the FRCM.



tmat: total thickness of the reinforcement
fc,mat: compressive strength of the matrix



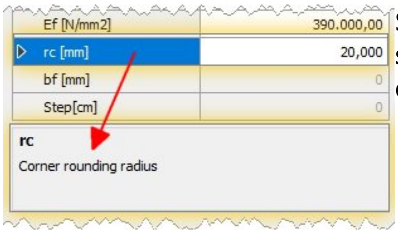
Bars:

[Available only with FRP]
The "Bars" insertion session is activated by selecting the appropriate box at the top.

Ef: normal elasticity modulus of the reinforcement
ε fk: ultimate strain
Ab: area of the single bar
pb: rows step of bars measured along the z axis of the column
nb,x: number of bars oriented according to the x direction of the column section
cxs: distance of the first bar in the x direction from the edge of the column
nb,y: number of bars oriented according to the y direction of the column section
cys: distance of the first bar in the y direction from the edge of the column

The steps between the cx and cy bars are not required as input because they are calculated automatically from the dimensions of the column.

The reference systems and distances are clarified by the illustrative image placed immediately below the insertion table.



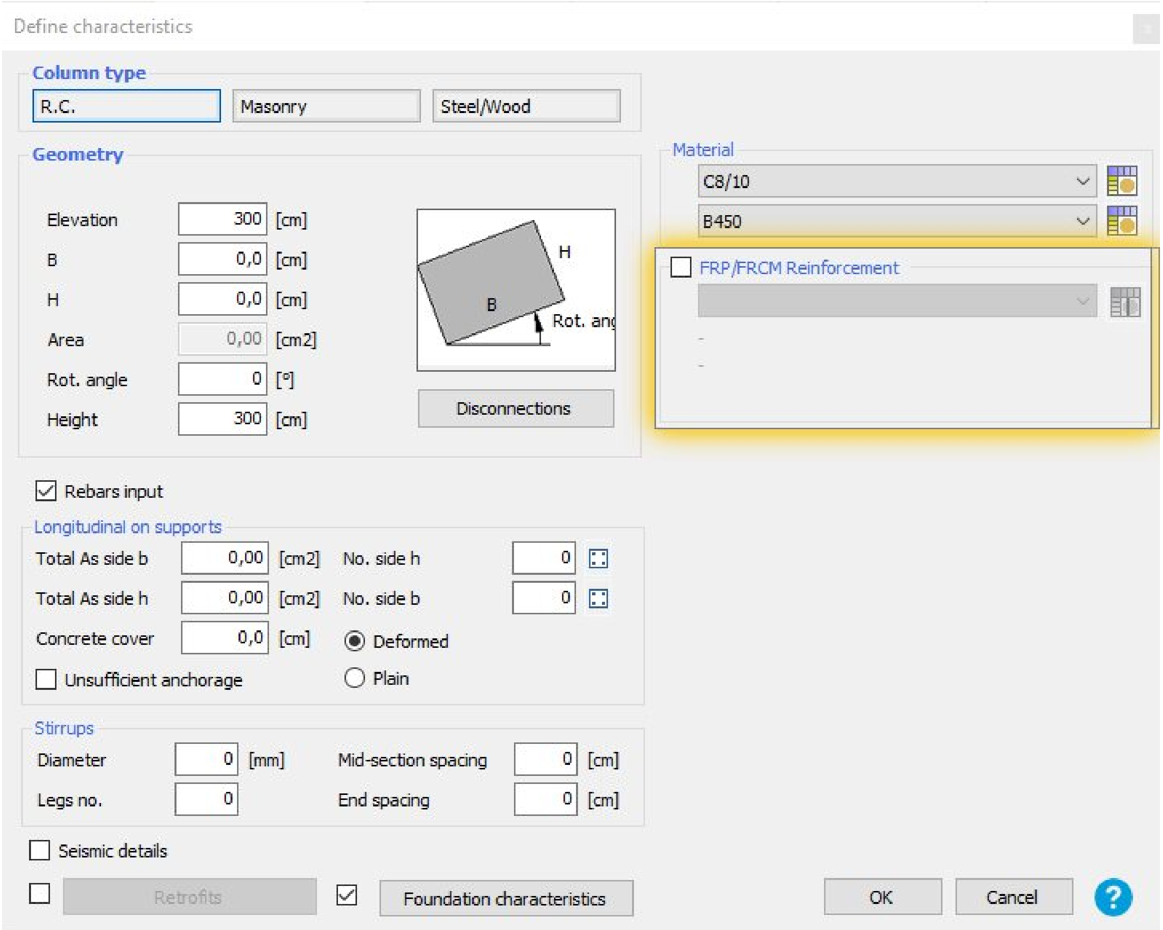
Special rectangular boxes show the description of the field selected in the table to always have on hand the meaning of each single field.

9.2.3.6 Reinforcement of concrete colums for confinement

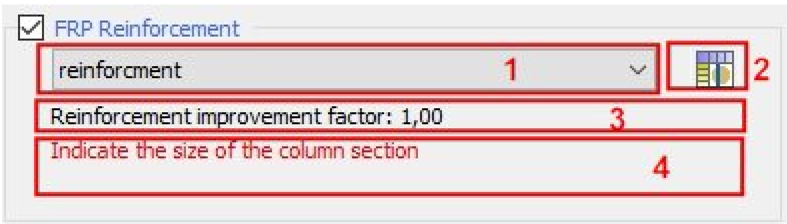
It is possible to define reinforcements for confinement on **concrete** columns with FRP or FRCM composite materials.
Theoretical references:
FRP: CNR-DT 200 R1/2013 - §4.5
FRCM: CNR-DT 215/2018 - §5.3

Reinforcement interventions for confinement act on mechanical performance, improving the compressive strength of the column.
The program calculates a "reinforcement improvement factor", consisting of a number greater than 1 which, multiplied by the compressive strength of the concrete, generates the improvement effect.

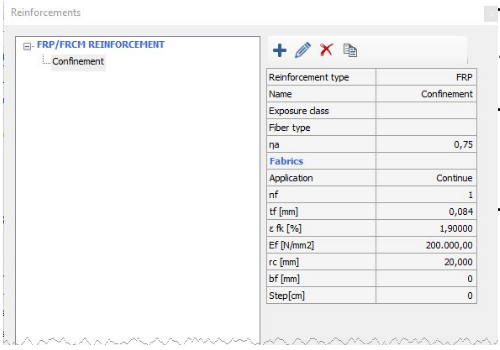
A session dedicated to FRP/FRCM reinforcement appears in the lower part of the window dedicated for inserting the concrete columns.



The reinforcement session is divided into 4 parts.
1. Reinforcement selection area among those present in the list, to be available in the list it must be defined in the definition environment.
2. Reinforcement definition environment
3. Reinforcement improvement factor
4. Information messages area, to warn in case of non-compliance with some indications contained in the normative text



Reinforcement definition environment



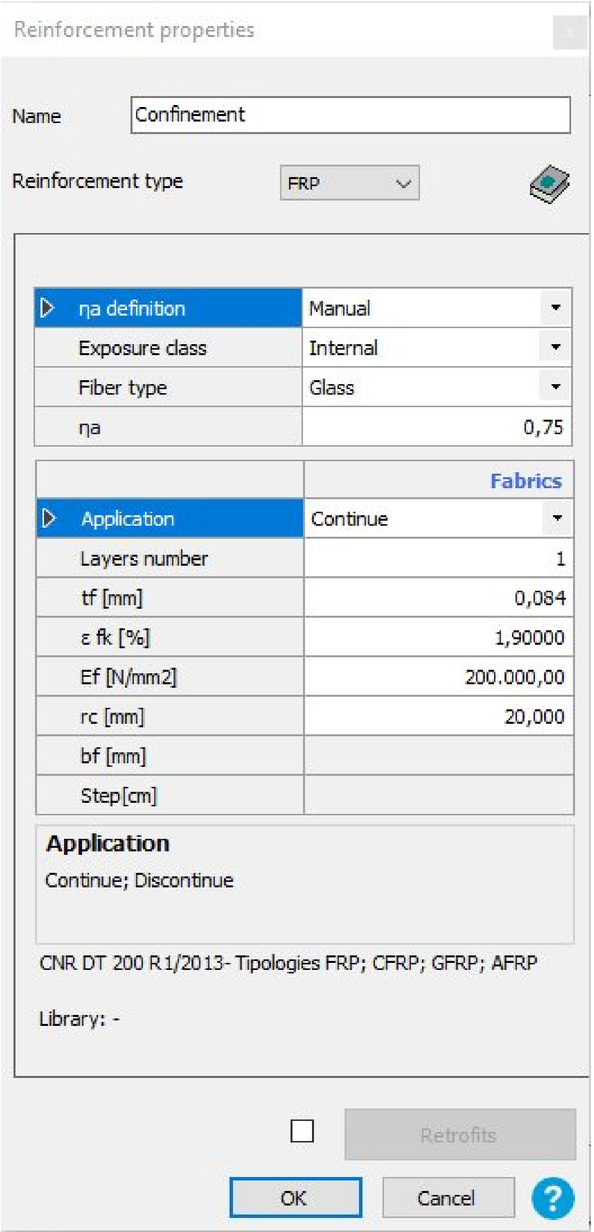
The left side indicates the list of reinforcements already defined.

The right part summarizes the characteristics of the reinforcement selected in the left list.

The command bar at the top allows to Insert, Modify, Delete or Copy a reinforcement.

Insert / Modify reinforcement element

By selecting the insert/modify command, the following mask is shown

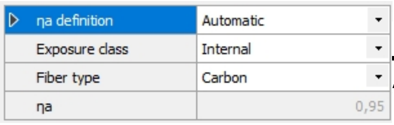


In the upper part, in addition to the field where is possible to define the name of the reinforcement, a drop-down window allows the choice of the FRP/FRCM typology.

The Reinforcement library button allows the access to the preloaded reinforcements.

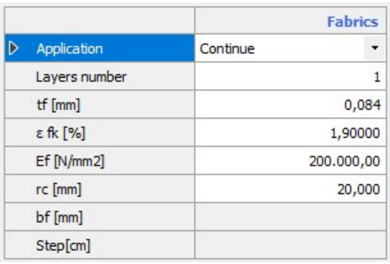
Based on what has been chosen in this drop-down window, the tables at the bottom show the parameters of the corresponding reinforcement.

With reference to the CNRs listed above, for the purposes of confinement, FRPs can be introduced both as fabrics and as bars; meanwhile the FRCM are provided only by fabrics.



Environmental conversion factor:

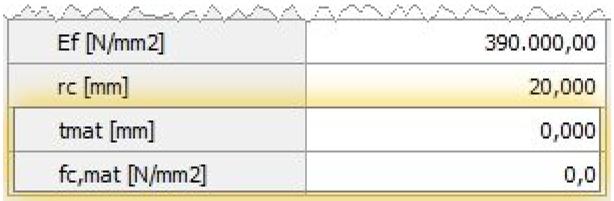
At the top left, a table allows the definition of the "environmental conversion factor".
In "Automatic" mode, it uses the CNR indications to calculate its value directly.
In "Manual" mode, the factor can be entered manually by the user.



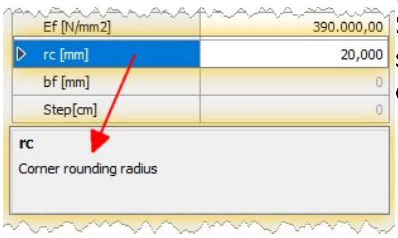
Fabrics:

Application: Continue/Discontinue
nf: number of layers
tf: equivalent thickness of the single fabric
ε fk: ultimate strain
Ef: normal elasticity modulus of the reinforcement
rc: corner rounding radius
bf: [available only if the application mode is "Discontinue"]: Width of the reinforcement strip
step: [available only if the application mode is "Discontinue"]: Step of the reinforcement strip

In the case of FRCM reinforcement, the "Discontinuous" application mode is not contemplated and the last two lines of the table are replaced with two others typical of the FRCM.



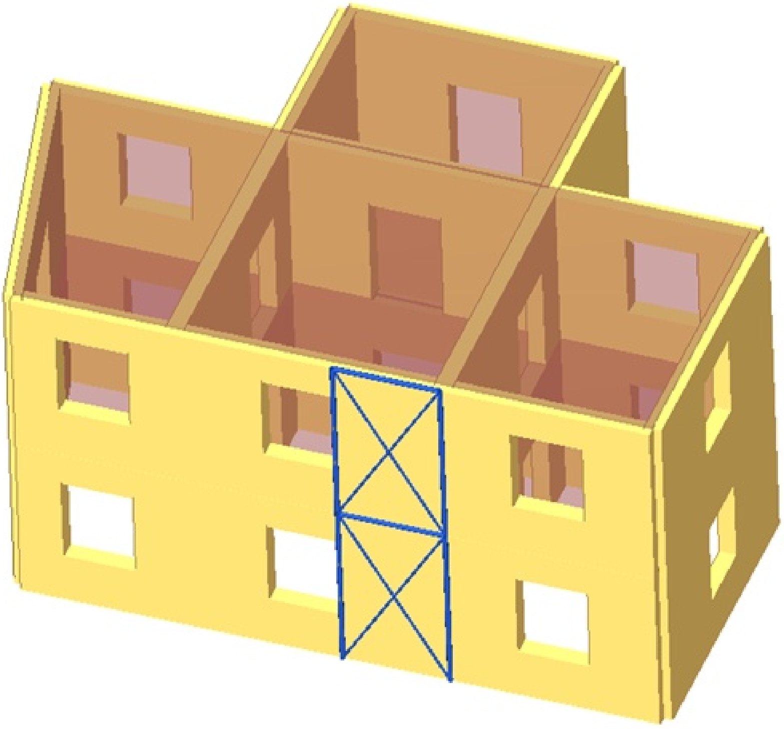
tmat: total thickness of the reinforcement
fc,mat: compressive strength of the matrix



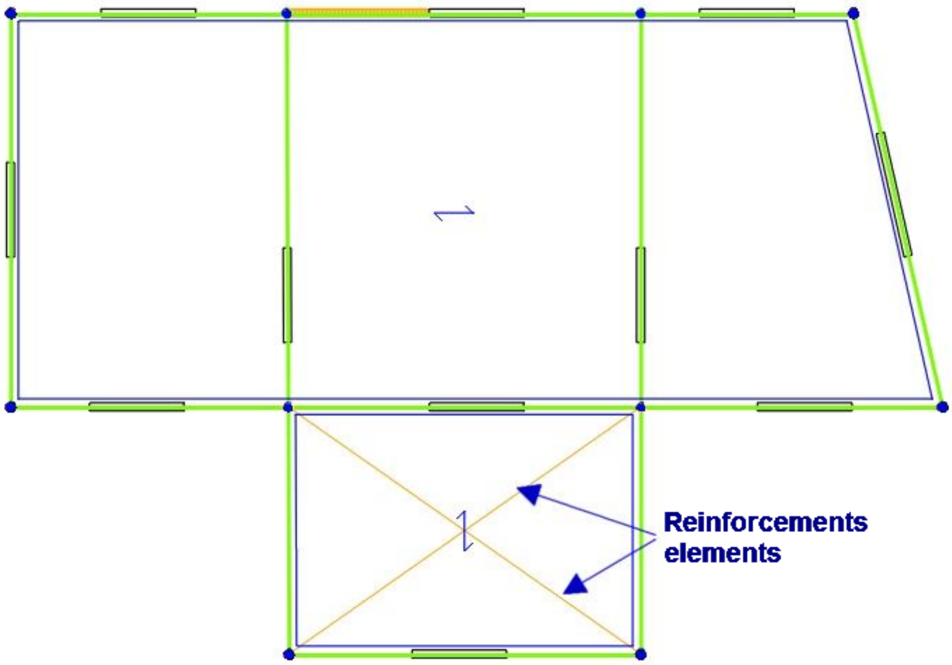
Special rectangular boxes show the description of the field selected in the table to always have on hand the meaning of each single field.

9.2.4 Frame reinforcement

It is possible to reinforce a structure with framed elements by placing them in the walls and in the slabs:



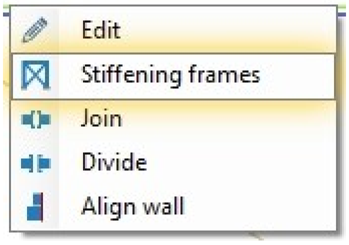
Frame reinforcements in the walls



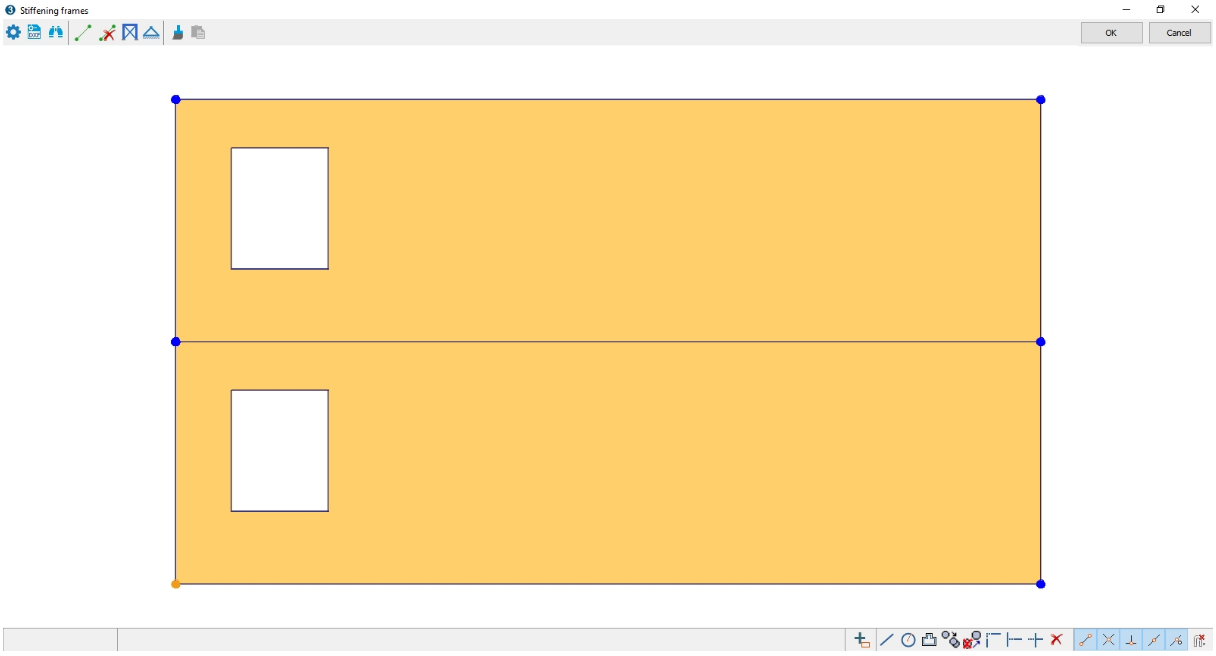
Frame reinforcements in slabs.

9.2.4.1 Frame reinforcements in walls

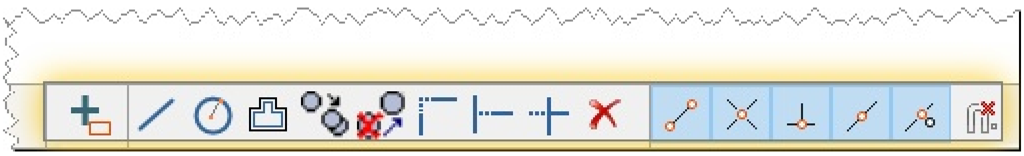
Select a wall with the right mouse button in the structure environment than the context menu highlights the *"Reinforcement Frames".*
Selecting this item leads to the reinforcements insertion environment for the examined wall.



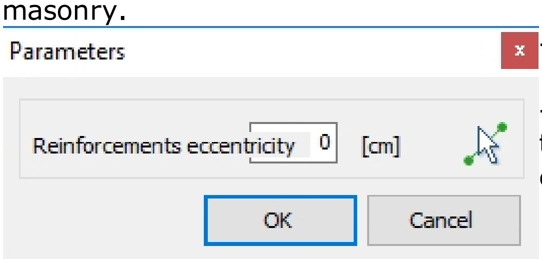
The environment shows that the wall in question opens with the command bar dedicated to the reinforcements.



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



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

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

9.2.4.1.1 Insert/Delete reinforcement

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

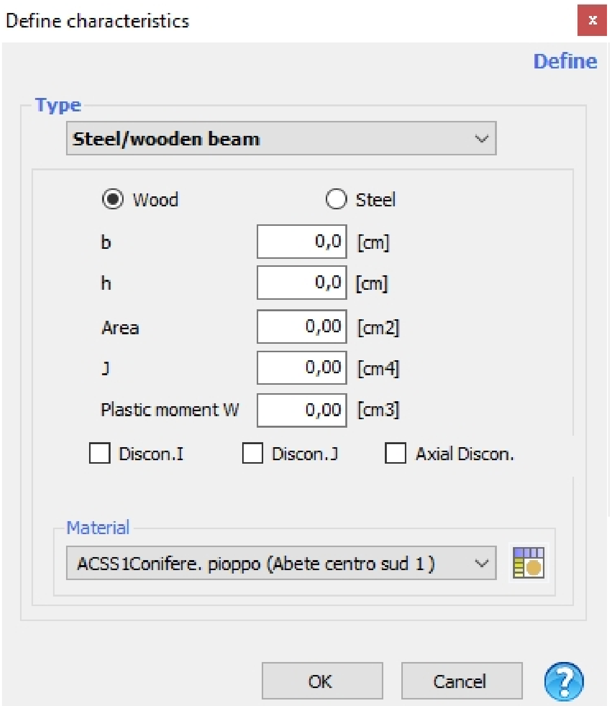
The use of these commands is made easier by the snap to graphics and from DXF input.
The well defined elements are free of any geometric / mechanical characteristic to be defined by the appropriate command described in the following.

9.2.4.1.2 Reinforcement characteristics definition

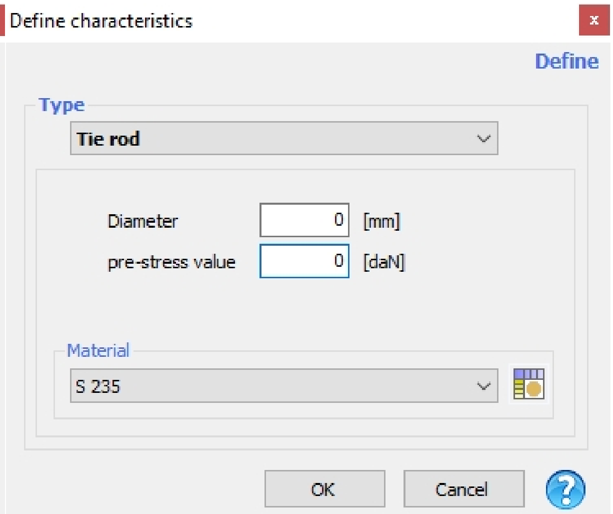
Allows to define the geometric / mechanical properties of the reinforcing elements drawn with the insert command .

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

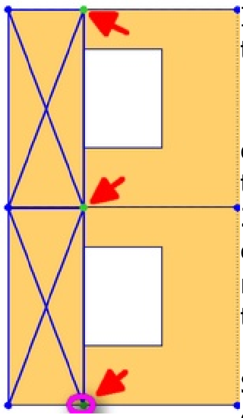
By selecting **"Steel / Wood Beam "** from the drop down menu is possible to define the characteristics of a beam in wood or steel.



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



9.2.4.1.3 Additional constraints



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

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

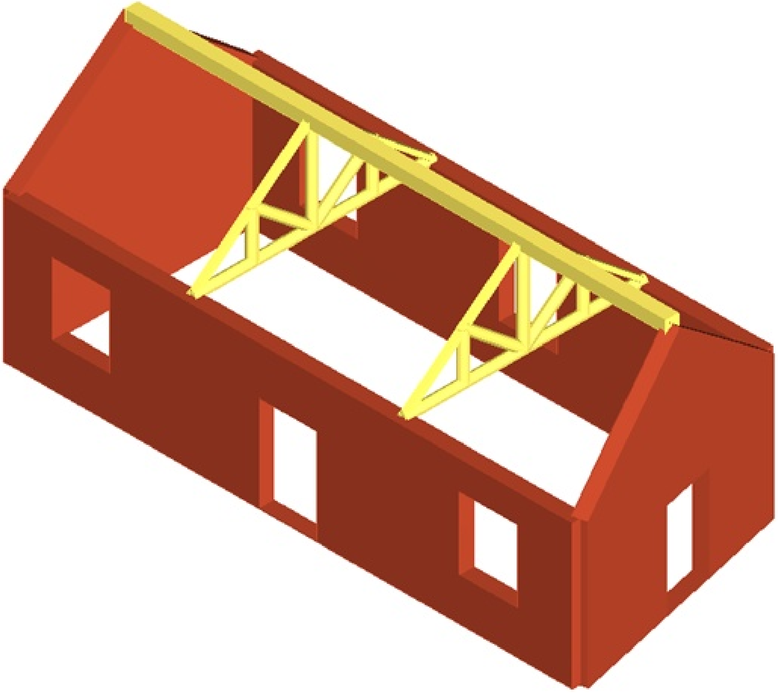
Some of these nodes may be positioned in correspondence of a constraint of the foundation.

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

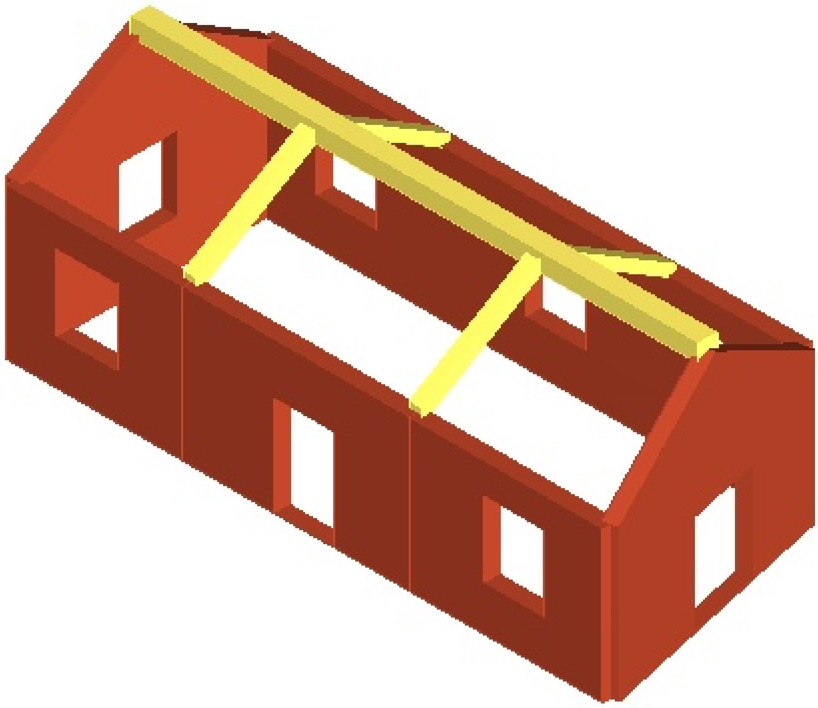
Deleting a constraint is done by clicking on the node with the right mouse button and selecting "*Delete*"

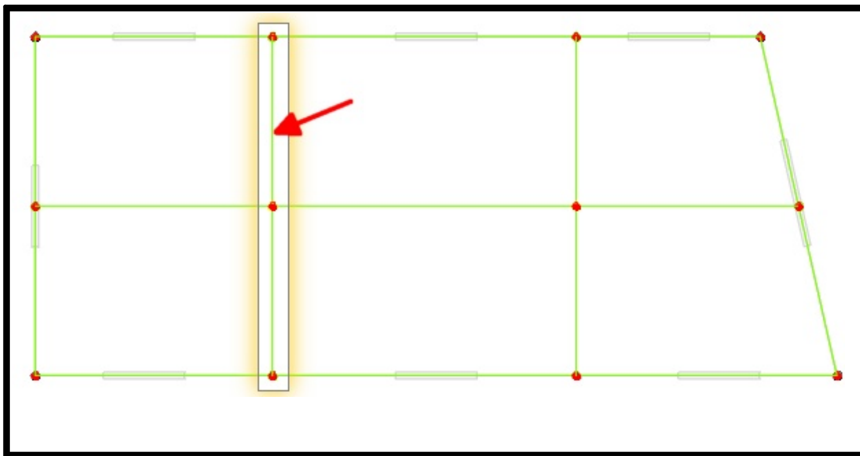
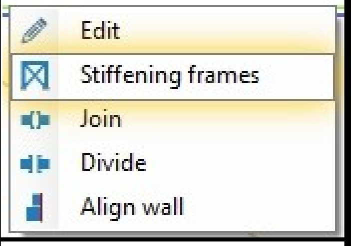
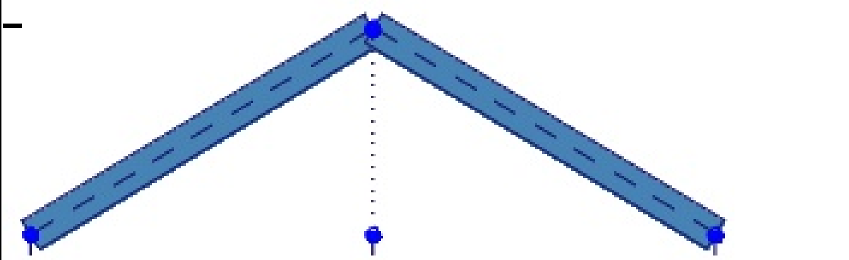
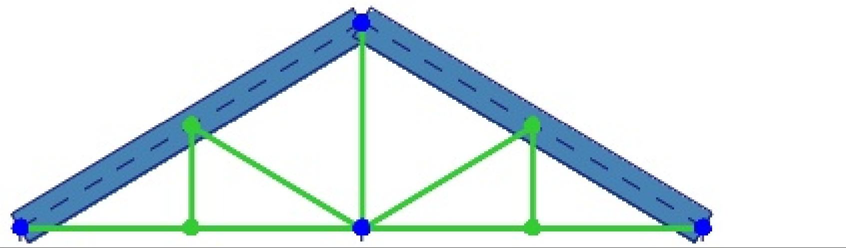
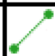
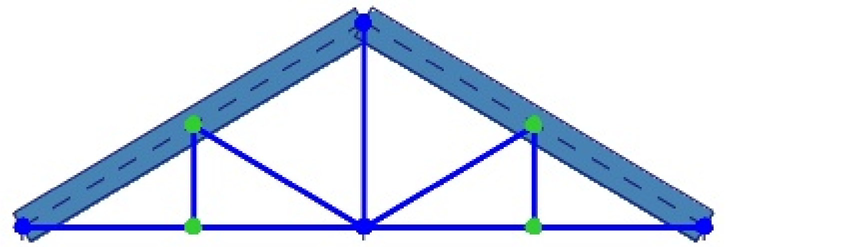
9.2.4.1.4 Reinforcement through truss beam

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



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

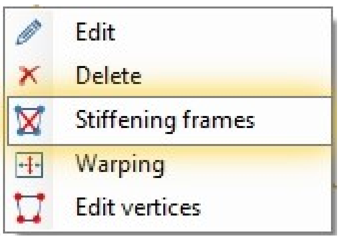
	In the roof environment previously define the ridge beams and the elements that constitute the upper stringers of the truss beam (by tracing of the roof profiles). **CAUTION!!** : The tracking of these elements must occur in advance with respect to the definition of the reinforcement.

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

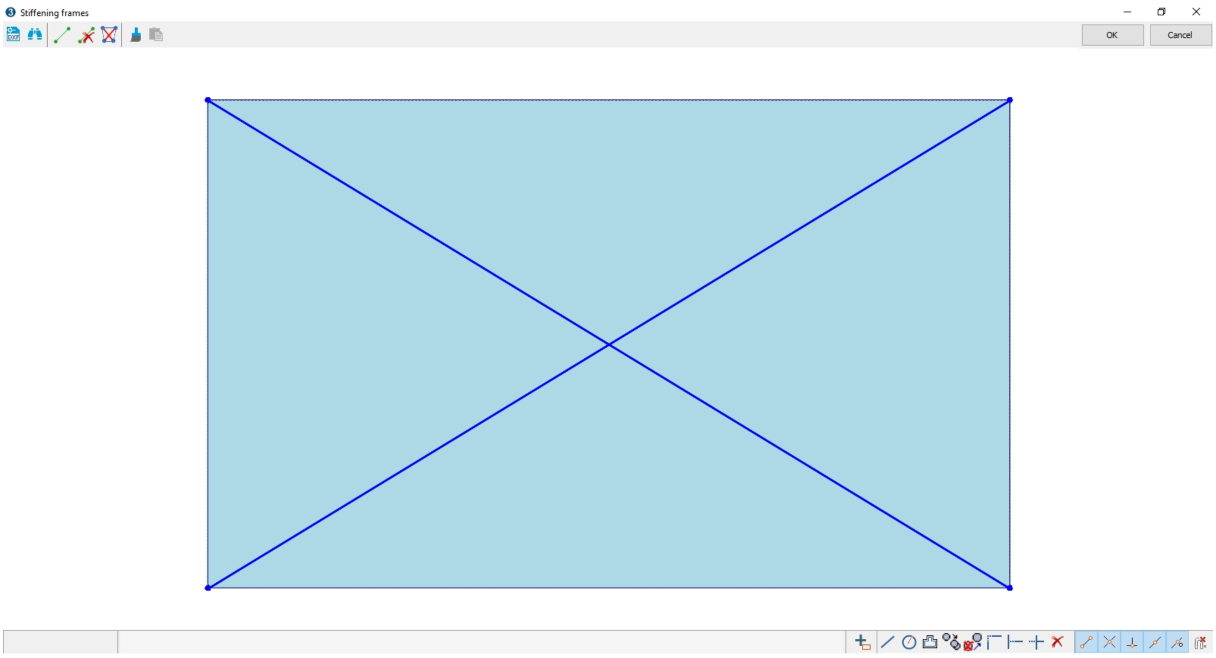
Coming out of this environment and recalling the 3D view you see the desired result shown in the first image.

9.2.4.2 Reinforcements with frames of horizontal elements

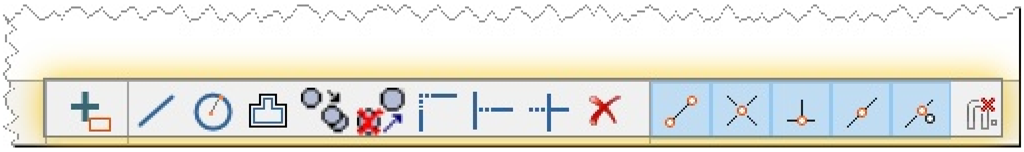
Into the Structure environment, by selecting with the right button of the mouse a slab, the context menu shows the item "Reinforcement Frames".
By selecting this item you access to the insertion environment of reinforcements for the slab under consideration.



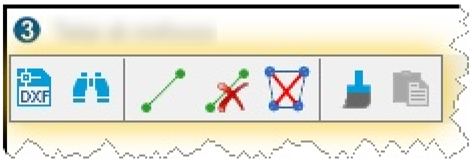
The environment that opens up displays the slab in question with the command bar dedicated to the reinforcement.

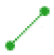
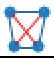


At the bottom right, the toolbar dedicated to the input graph already used in other environments of the program, for the detail of each commands please refer to the general session (Supporting graphics; Snap).

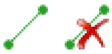


At the top left, the dedicated bar for the reinforcement insertion.



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

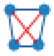
9.2.4.2.1 Insertion/elimination of reinforcement

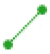


Are two commands dedicated to defining routes that identify the location of the reinforcements.
The insert command, allows the placement of the reinforcement by graphical input to indicate the start and end point of the reinforcement.
The delete command, requires the selection of the reinforcing segment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

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

9.2.4.2.2 Definition of reinforcement characteristics

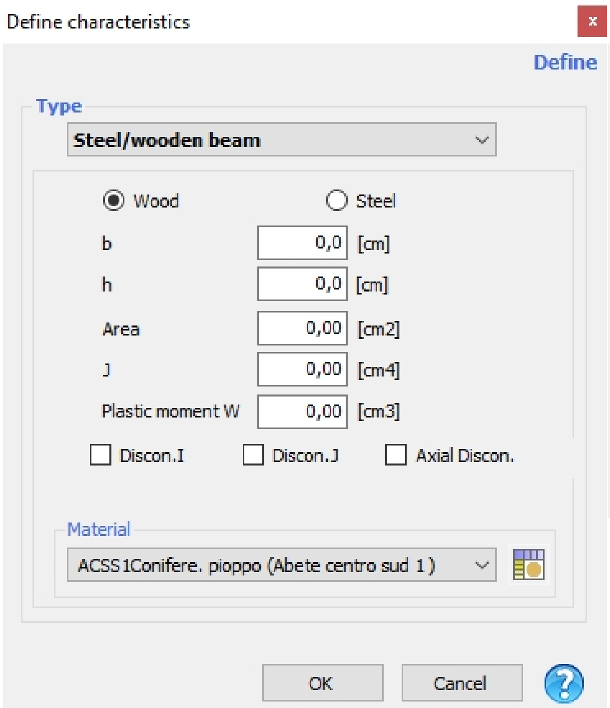


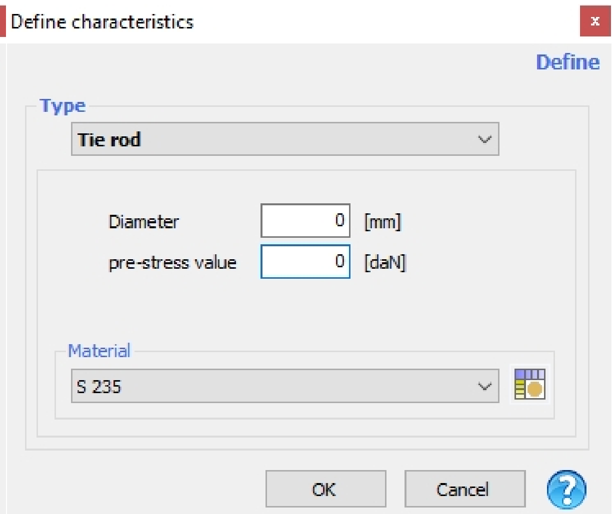
Allows to define the geometric / mechanical properties of the reinforcing elements drawn with the insert command  .

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

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

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



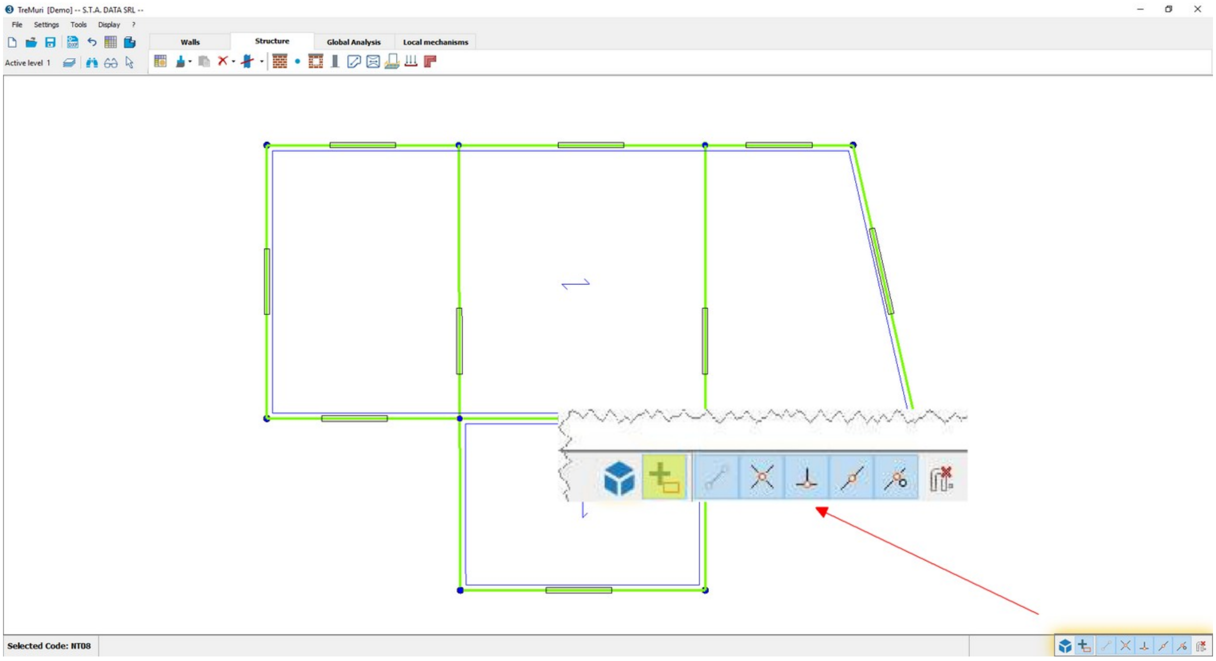


9.2.5 Segment Points

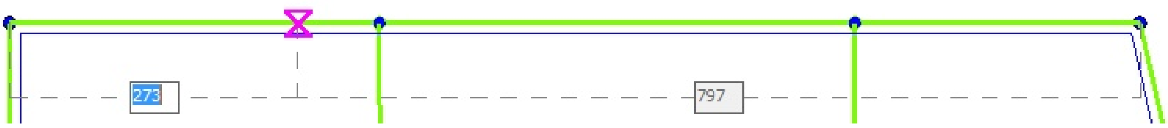
- Segment points can be inserted using the left mouse button.

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

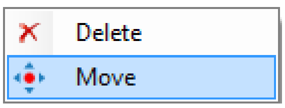
The entry in a precise distance during the input phase can be performed using the command "Dynamic Input" available through the specific button on the command bar.



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



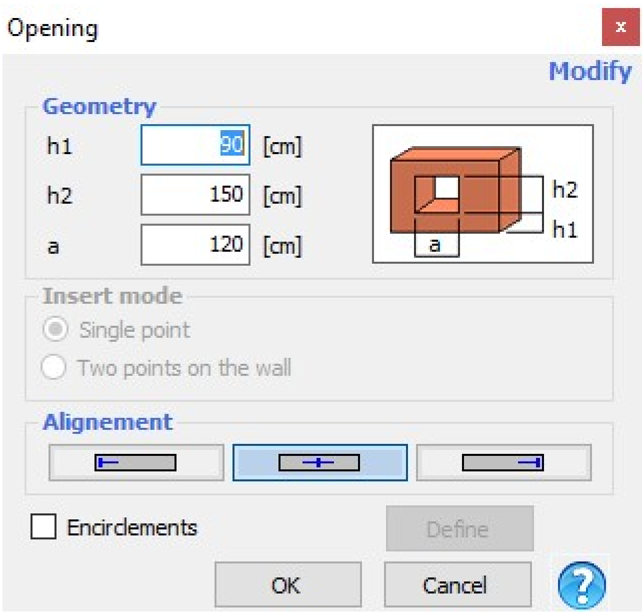
By selecting an already inserted entity with the right mouse button (opening or node) you can move it.



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

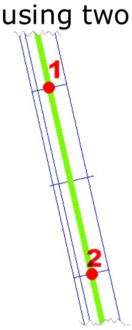
9.2.6 Openings

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

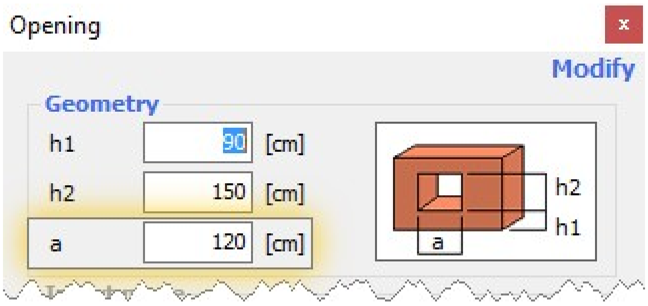


The insertion of an opening can be performed in two different ways:

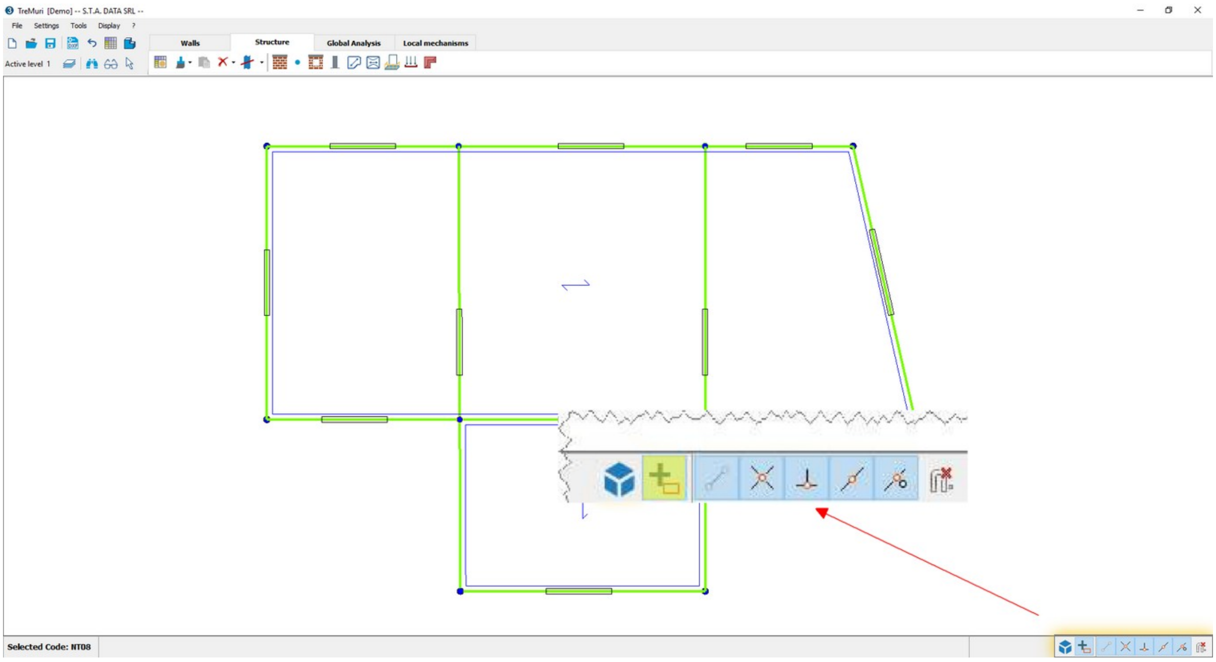
Single point: it is necessary to define the size of the opening and click at the "thread" chosen for inclusion.
Two points on the wall: heights (h1, h2) are necessary but not the width (a) which is set using two clicks in the graphics area (at the beginning and end of the opening).



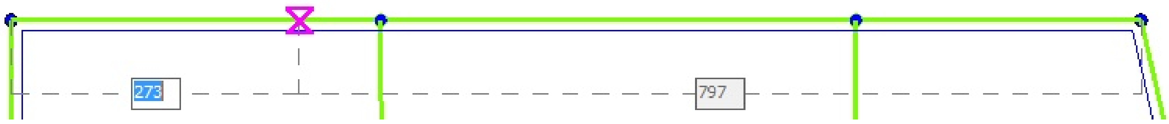
There are necessary two clicks, one at the beginning [1] and one at the end of the opening [2].
The width is no required because it is calculated automatically from the distance between [1]-[2].



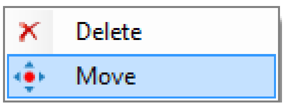
The entry in a precise distance during the input phase can be performed using the command "Dynamic Input" available through the specific button on the command bar.



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



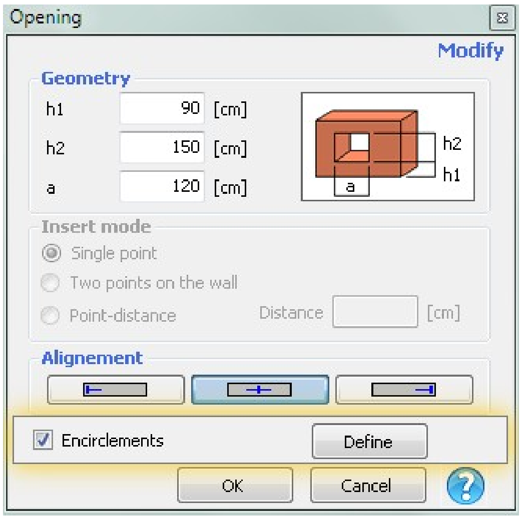
By selecting an already inserted entity with the right mouse button (opening or node) you can move it.



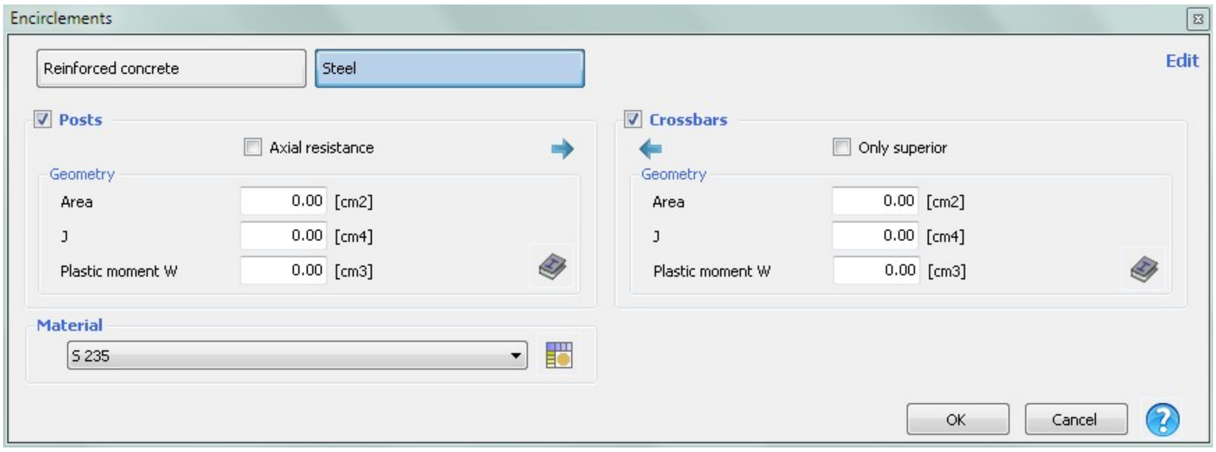
It is necessary to define the displacement vector by clicking in two points, starting point and end point.

9.2.7 Encirclements

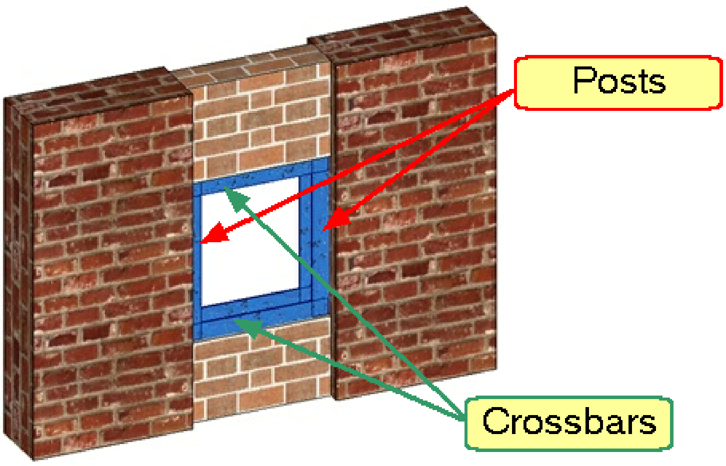
The encirclements are a kind of reinforcement that can be applied directly to the openings of the structure.
To insert an encirclement you need to check the box "encirclements", right from the "Opening" command, to activate the setting button that allows you to insert the features.



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

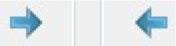


For each of the two types, you can set through a checkbox, if you want to insert only the posts, only crossbars, or both.



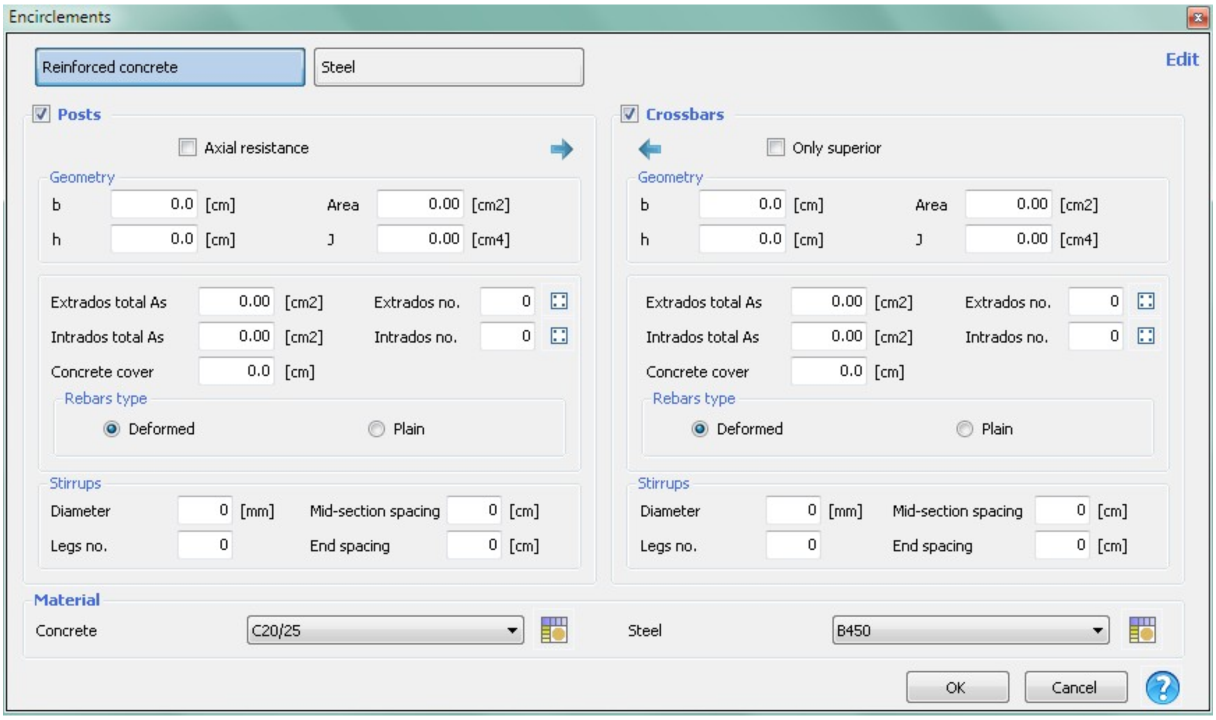
The "Posts" section, allows you to choose whether to consider the contribution of the axial strength of the post or not.
By choosing to account the axial strength of the post, you allow it to absorb a part of the loads from the upper floor along with the masonry wall to which is attached.
It’s widespread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

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

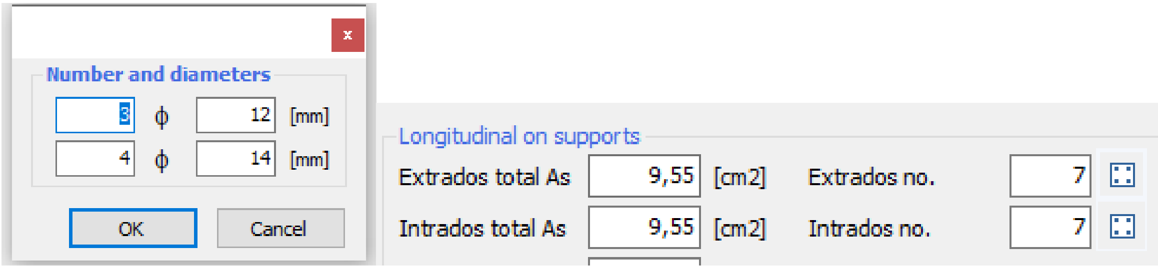
 These buttons allow you to copy the features entered for posts to the crossbars and vice versa.
As for the posts and crossbars using the same section, this command helps the input.

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

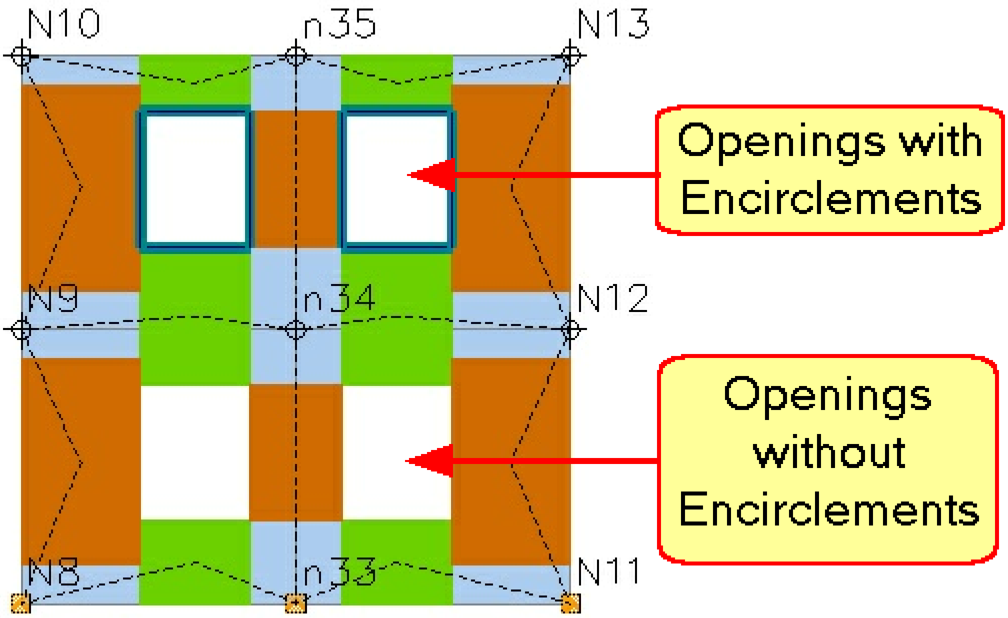
The input of encirclements made of reinforced concrete, is much larger because all the parameters of the reinforcements are needed, so that we can consider their behavior in the non-linear field.



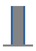
This button helps the user to insert the amount of reinforcement (quantity and diameter).When choosing the number and diameters of the reinforcement (3?12+4?14 on the example below) the software automatically calculates the total area and number of rebars.



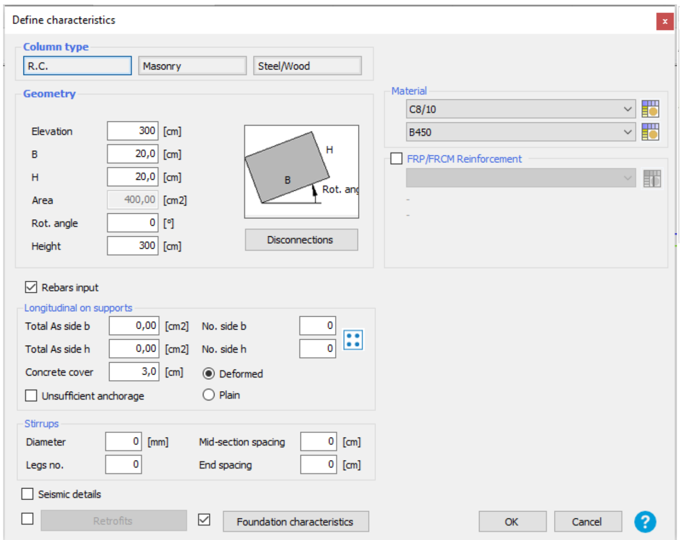
Following the introduction, will be visually presented directly in the statement of the mesh wall.

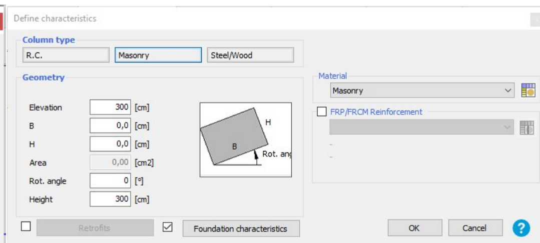


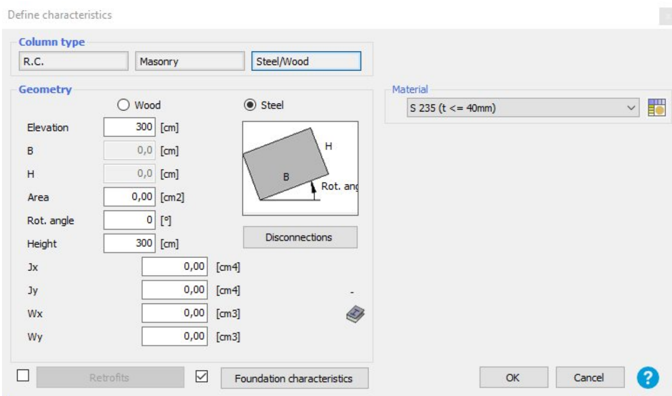
9.2.8 Columns

 It allows you to insert a column in correspondence with one or more nodes.
You are first asked to select the node or nodes in which you want to place the columns, then with the right mouse button you access a window in which you can define the geometrical and material characteristics of the element.

The insertable columns can be of three different types: R.C., masonry or steel/wood; depending on the type of column chosen, the inclusion of the mechanical characteristics necessary to perform the non-linear calculation is required. *In the R.C. columns, the areas of reinforcement to be inserted in the appropriate space are the total areas along the side and not those of the single rebar.*



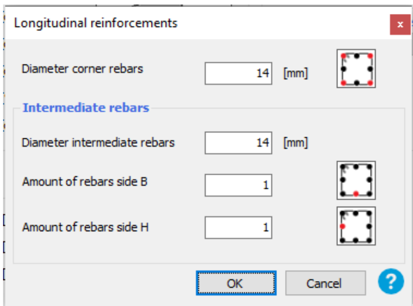




At the lower left, it is possible to activate a box that imposes foundation constraints at the base of the column.

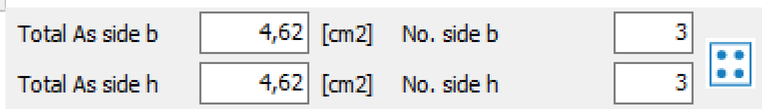
R.C. Columns:
Pilastri C.A. :

Questo pulsante aiuta il progettista ad inserire i quantitativi di armatura (aree e numero ferri).



Dal conteggio del numero di barre per il lato B e il lato H vanno esclusi i ferri d'angolo.
Ad esempio, in un presenza di un pilastro la cui armatura consista di soli 4 ferri d'angolo e nient'altro, il numero di barre per il lato B e per il lato H sarà pari a 0.

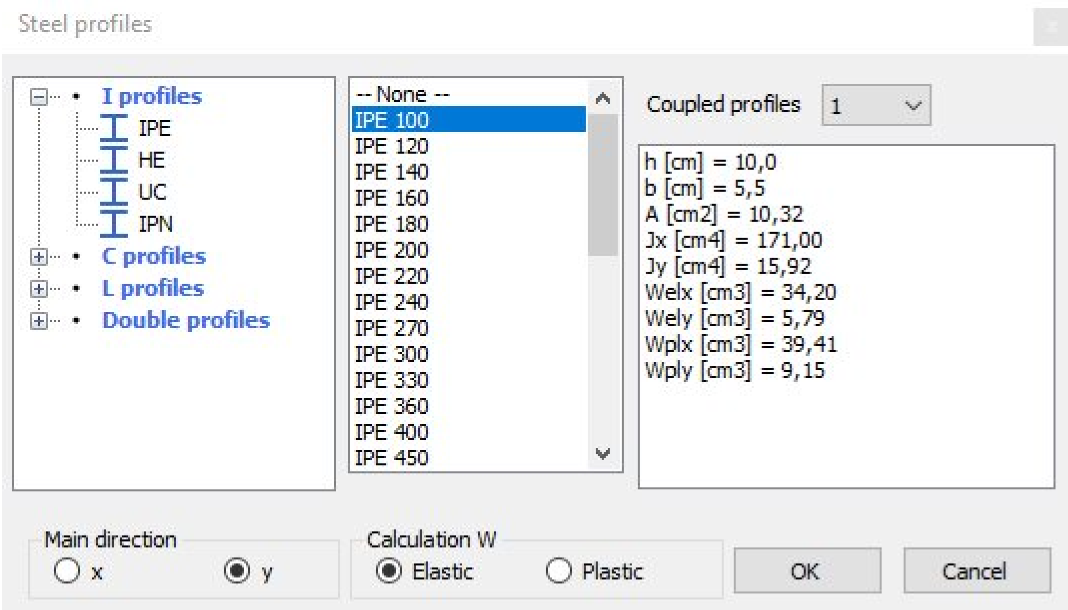
Dopo aver confermato, il programma calcolerà in automatico l'area di armatura risultante:



Through the appropriate section (shown on the side) it is possible to insert FRP/FRCM reinforcement for confinement.

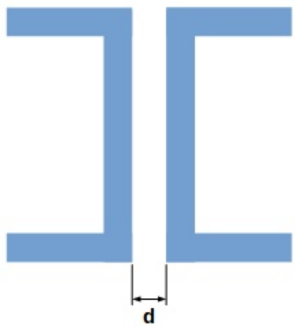
Steel Columns :

With this button it is possible to recall the steel profiles library from where you can get the features.
Selecting the family and profile size,the mechanical characteristics that will be used in the calculation are presented.



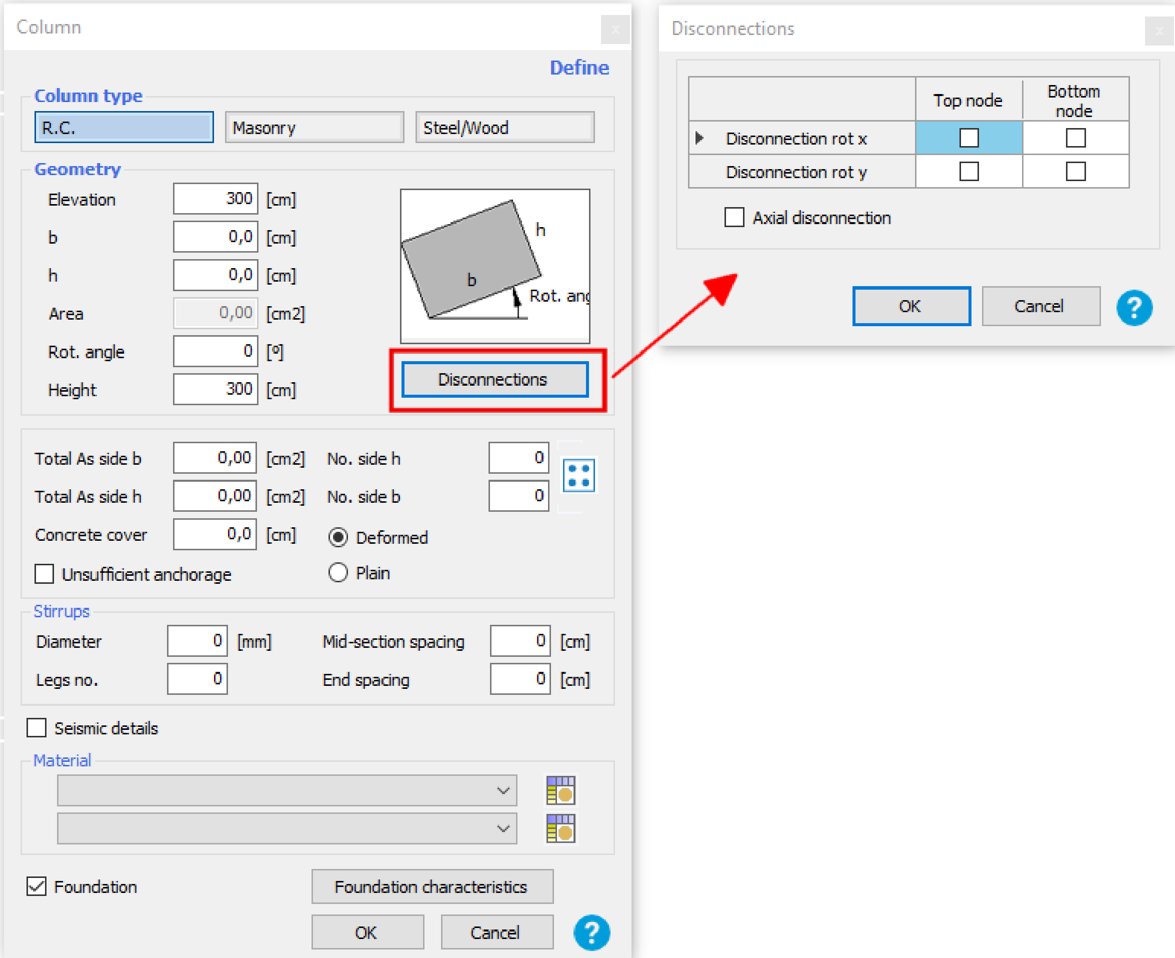
I profiles It is also possible to generate a section composed of several profiles by disposing of up to 4 side by side.

IPE HE UC	
C profiles UPN UAP UPE	Given the absence of double symmetry in this family of profiles, the side-by-side the disposition will not be possible, but it will be possible to arrange them as "double" according to the configurations shown below.
L profiles L equal wings L different wings	Given the absence of double symmetry in this family of profiles, the side-by-side the disposition will not be possible, but it will be possible to put them as "double" according to the configurations shown below.
Double profiles UPN (Type 1) UPN (Type 2) UAP (Type 1) UAP (Type 2) UPE (Type 1) UPE (Type 2) L equal wings L different wings	The previous families are disposed in the various "double" configurations available, indicated by the respective icon. For each configuration it is also possible to define the distance (d) between the two profiles.

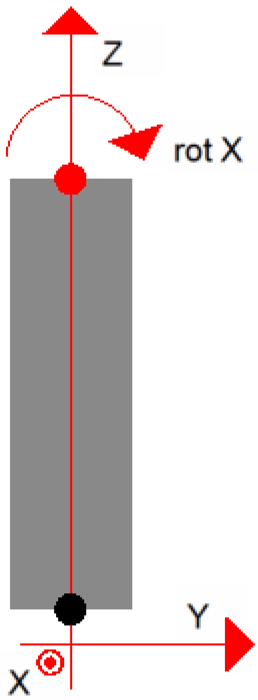


At the bottom left, there is the possibility of activating the box that imposes the foundation constraint at the base of the column.

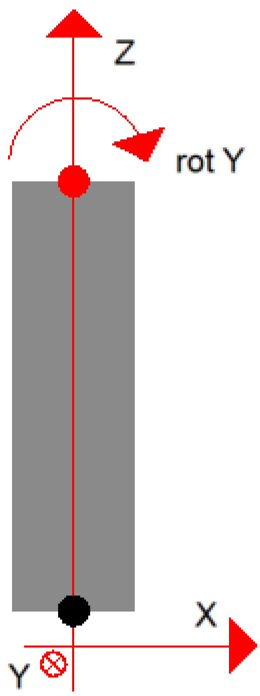
Through the disconnection button it is possible to modify the constraint degree of the columns extreme nodes.



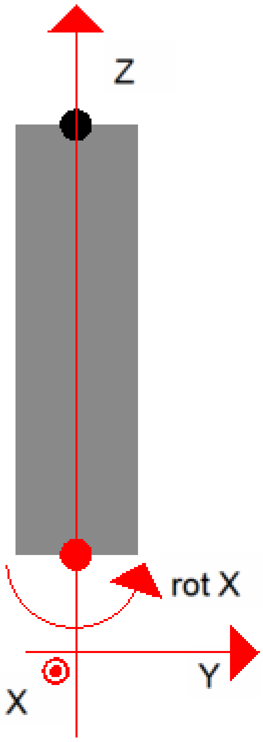
Rot x disconnection - UPPER NODE



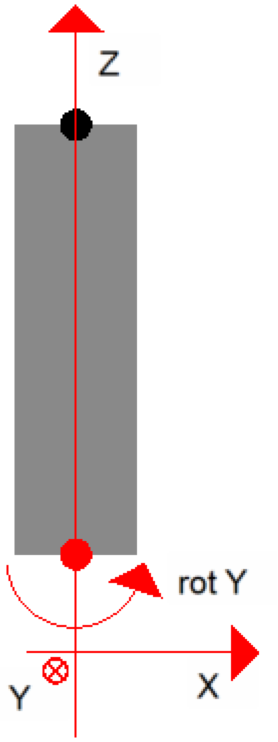
Rot y disconnection - UPPER NODE



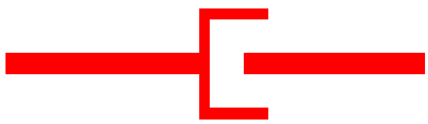
Rot x disconnection - LOWER NODE



Rot y disconnection - LOWER NODE



AXIAL DISCONNECTION

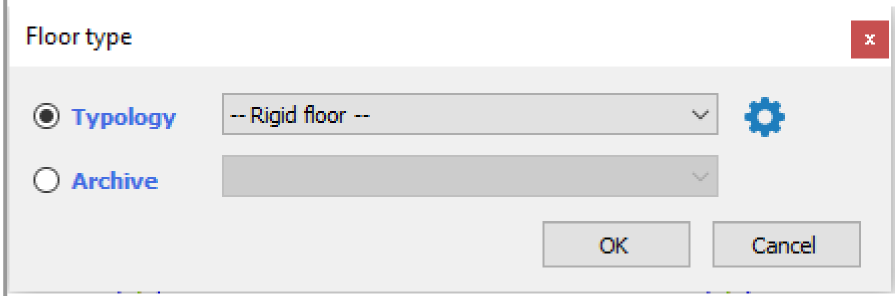


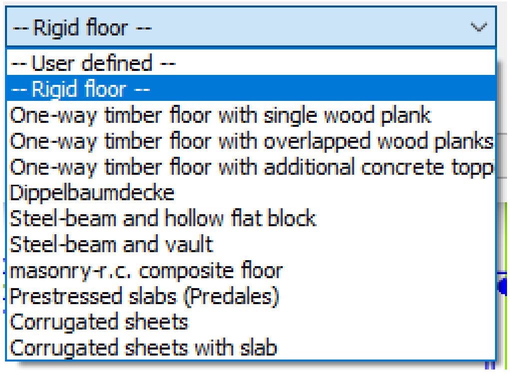
9.2.9 Floor

 allows to insert a slab.

Effective deformability floor

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





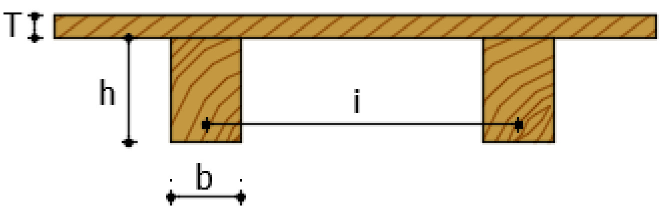
- The "**Typology**" field allows to select one of the possible types of floors.

- The "**Archive**" field allows to assign to the floor one of the types present in the archive.

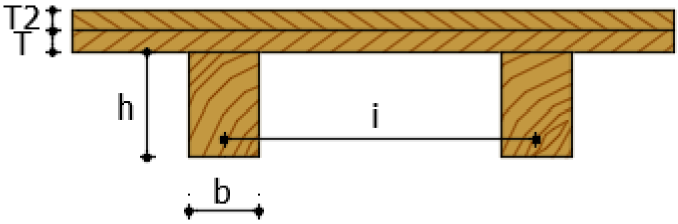
To create the floor archive, consult the dedicated section.

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

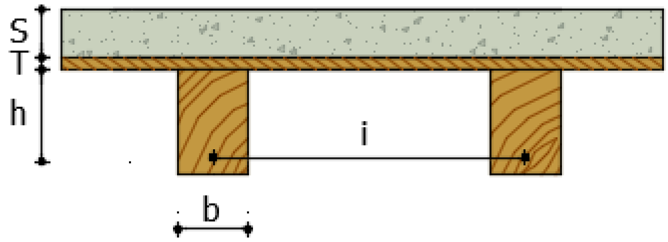
One-way timber floor with single wood plank



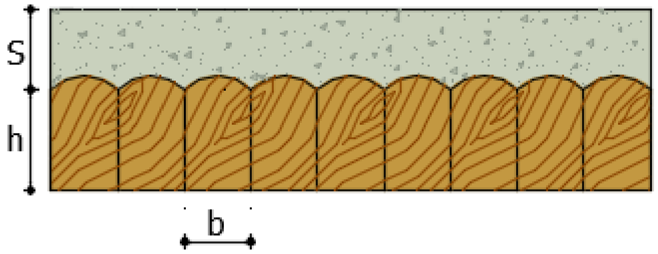
One-way timber floor with overlapped wood planks



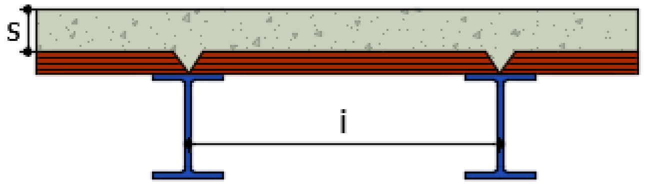
One-way timber floor with additional concrete topping



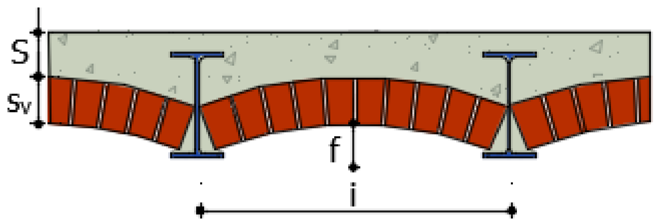
Dippelbaumdecke



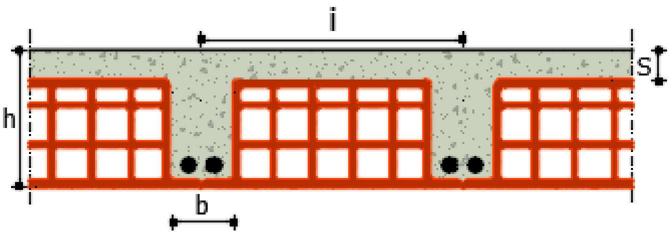
Steel-beam and hollow flat block



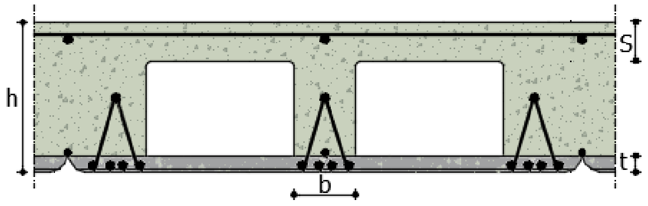
Steel-beam and vault



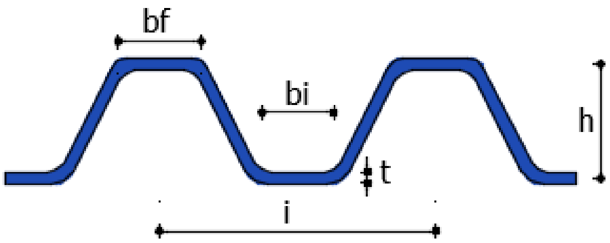
masonry-r.c. composite floor



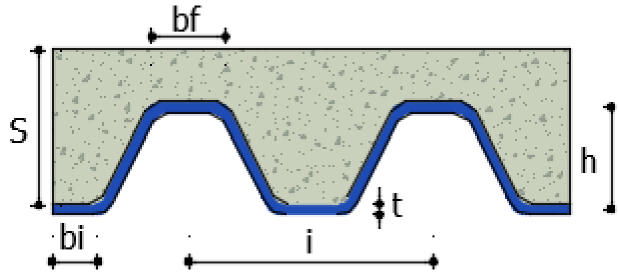
predalles



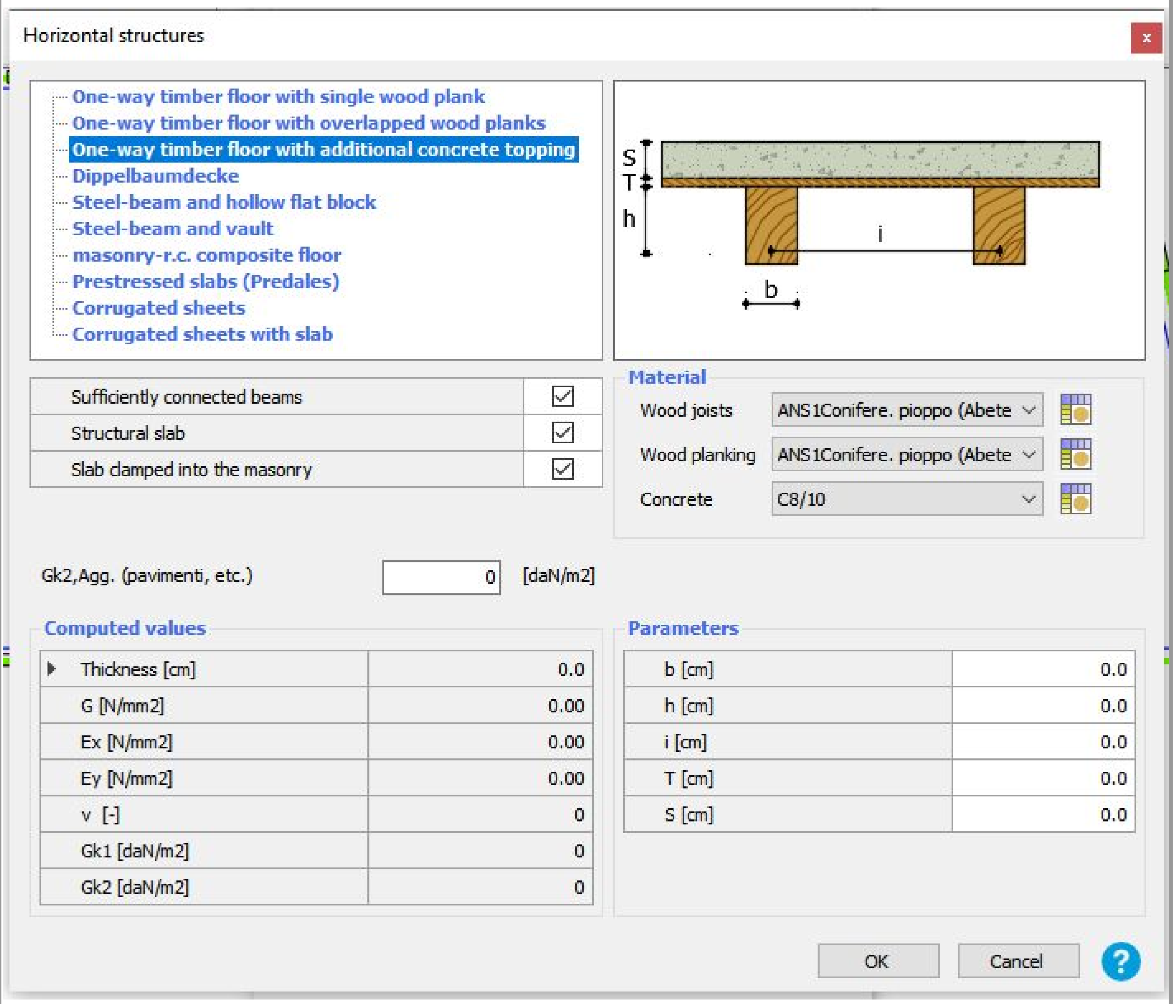
corrugated sheet floor



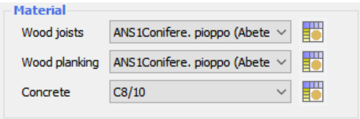
corrugated sheet floor with additional concrete topping



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

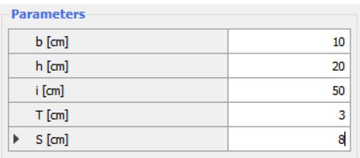


Material libraries



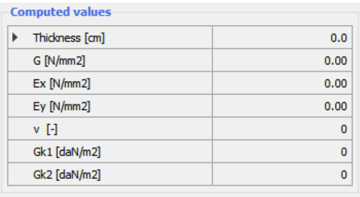
Material libraries which can be accessed to attribute their characteristics.

Geometric parameters

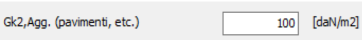


Geometric parameters relating to the elements that make up the floor under examination.
The number and type of parameters depend on the type of floor.

Calculated values



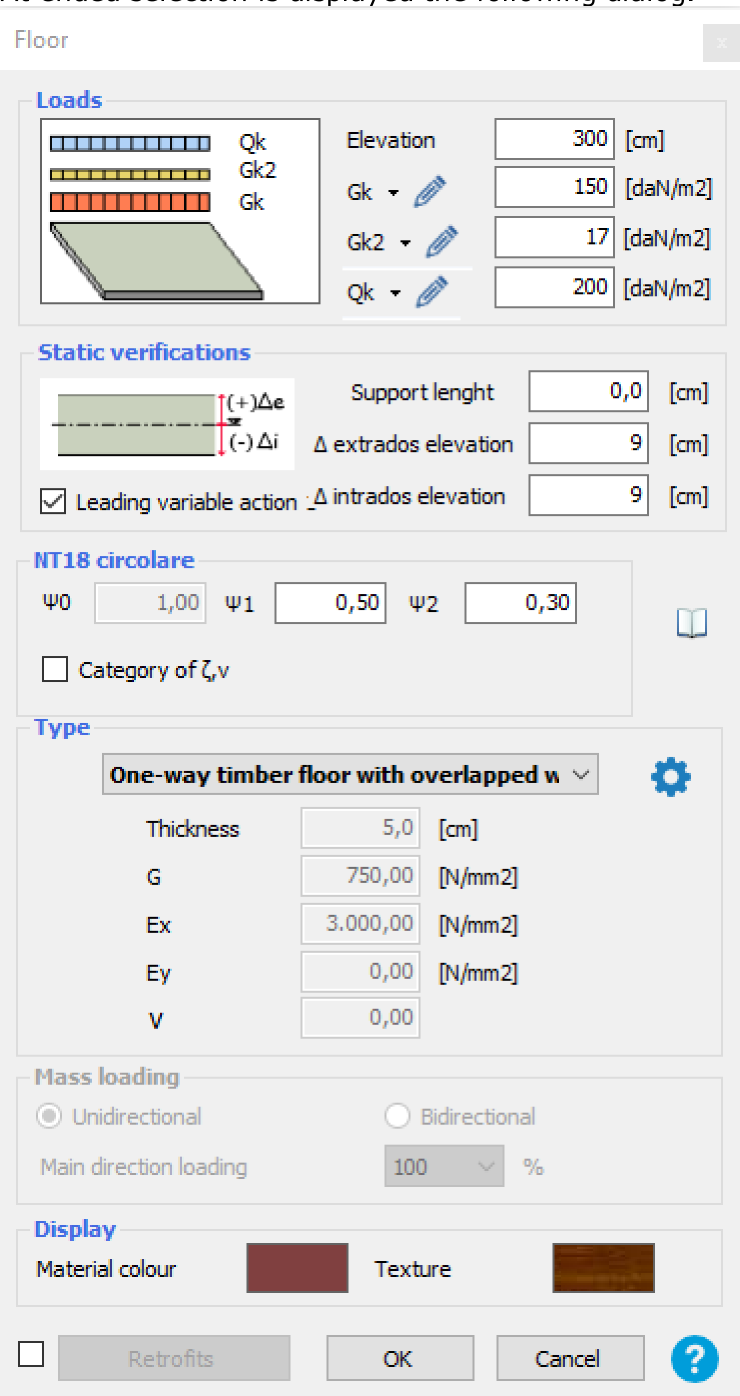
The stiffness (Ex,Ey) of the floor and the relative loads (structural and otherwise) are automatically calculated through the defined geometric values and the relative characteristics of the materials.
The extent of these quantities will be a function, in addition to materials and geometries, of the choice to consider or not a structural element rather than well connected or clamped to the masonry.



Through the item Gk2,Agg it is also possible to define an additional not structural load that takes into account additional elements not defined through the geometric parameters (flooring etc..).

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

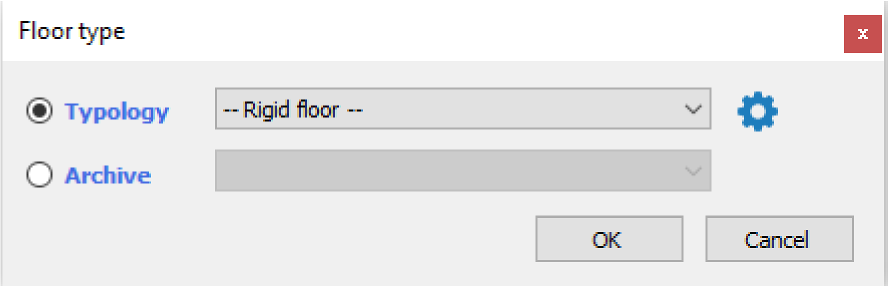
At ended selection is displayed the following dialog.



Loads shown in the mask are automatically calculated with informations in the material library.

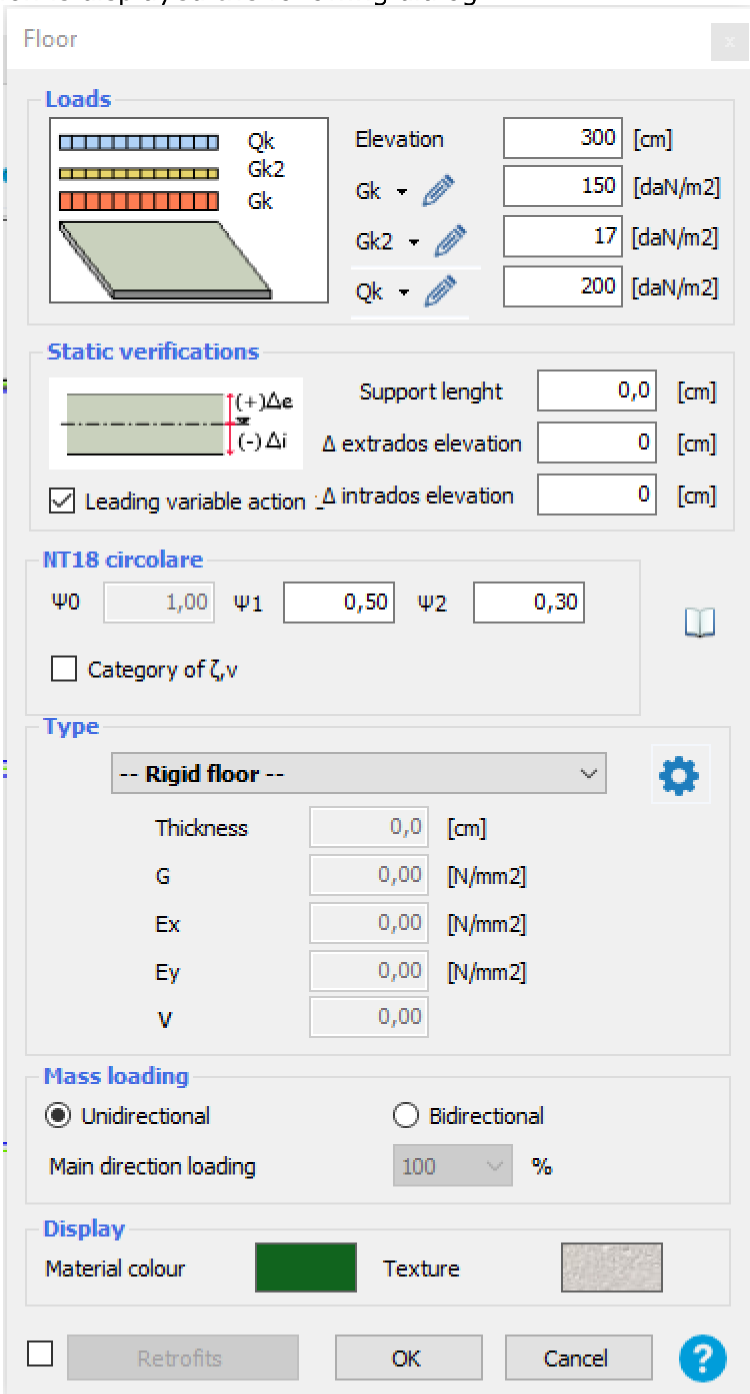
Rigid floor

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



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

At ended selection is displayed the following dialog.



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

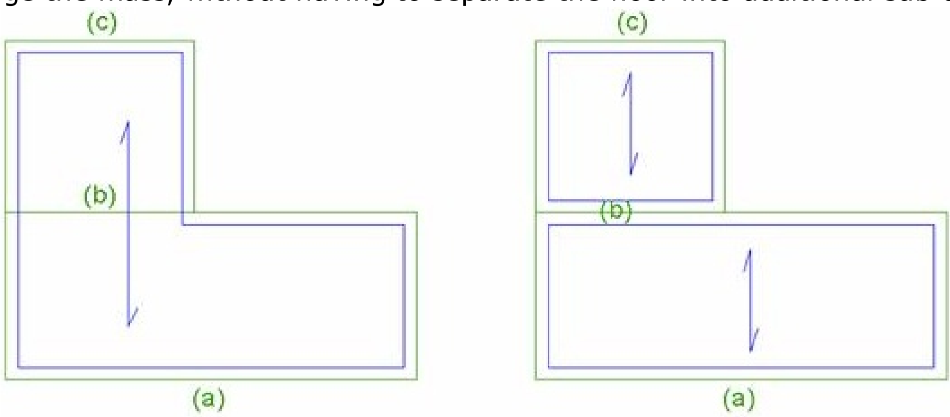
The permanent loads (G_k) are defined as permanent structural loads (G_1)
The permanent loads ($G_{k,agg}$) are defined as the weight of all non-structural elements (G_2).

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

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

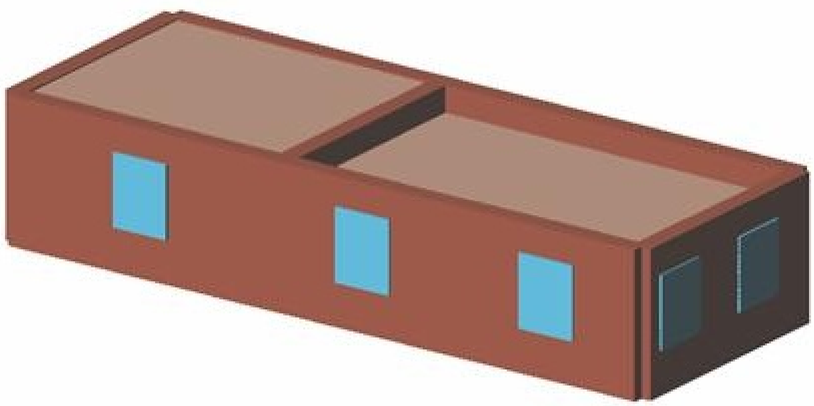
Bending modules Ex and Ey refer to the local axes system (x, y) in which "x" is identified based on the warping direction and "y" is perpendicular to the warping direction.

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



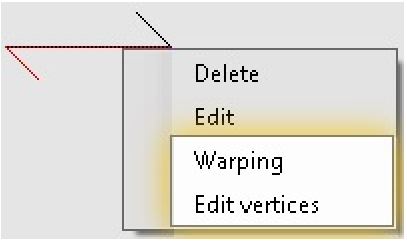
[Wall (b) is borne by the floor independently from the chosen insertion mode]

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

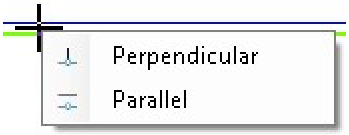


The program does not create additional computation nodes in correspondence with the position of the slabs. It continues to use those already defined, taking into account the contribution due to the transfer of the floor with respect to these limit nodes between one level and another.
It is not possible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

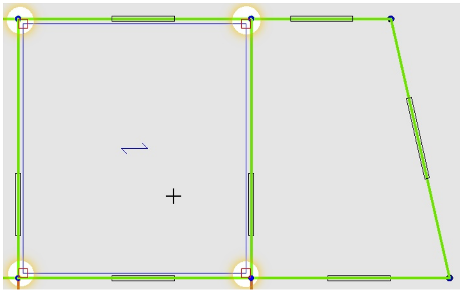
level.



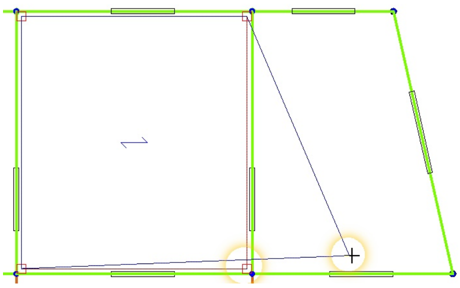
Slab warping direction:
Select a wall bounding the slab in order to redefine the warping direction of the slab.



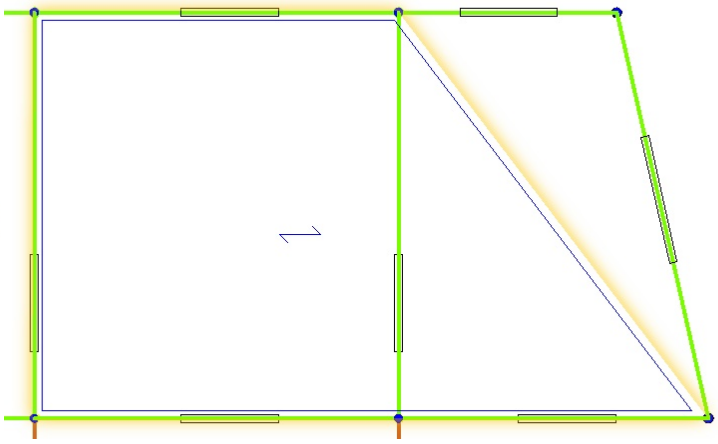
Edit vertices:
Once inserted a slab you can edit the vertices hooking them to different nodes.

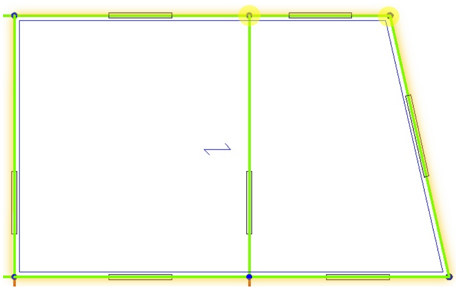


The "handles" are shown in correspondence to the vertices, if dragged they allow to define the vertices of the polygon in correspondence with other nodes.



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

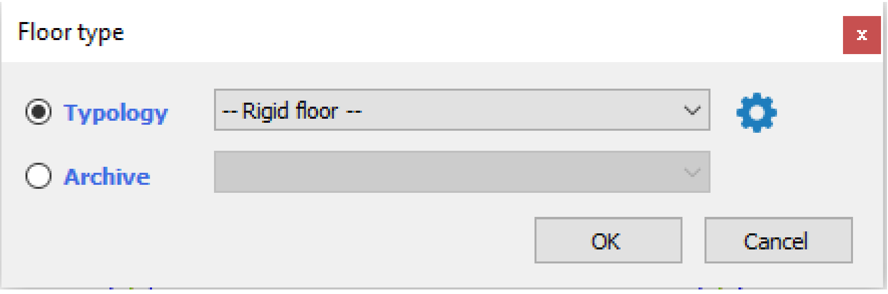




By invoking the command you can repeat the operation for other vertices

9.2.9.1 Floor archive

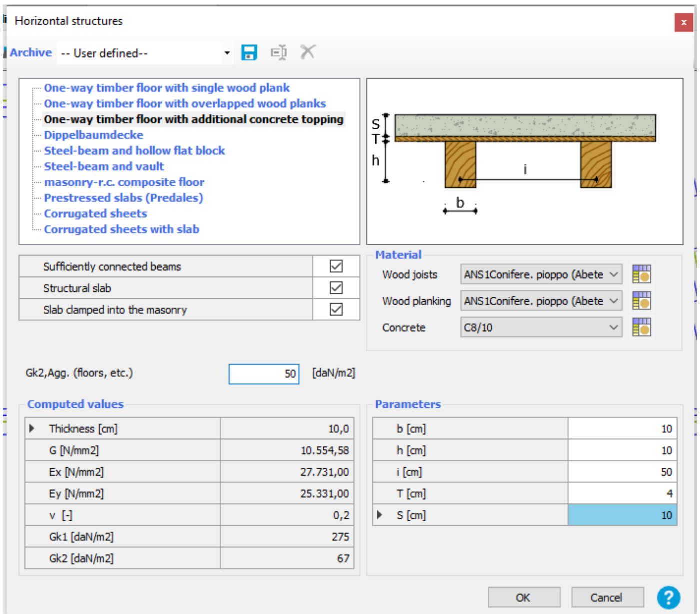
Once the slab insertion command has been selected , the following window opens:



By clicking the "archive" option, it is possible to assign one of the previously created types to the floor.

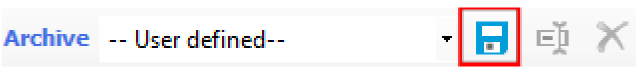
It is possible to add a new typology to the archive using the following steps:

1. Definition of the characteristics of the floor



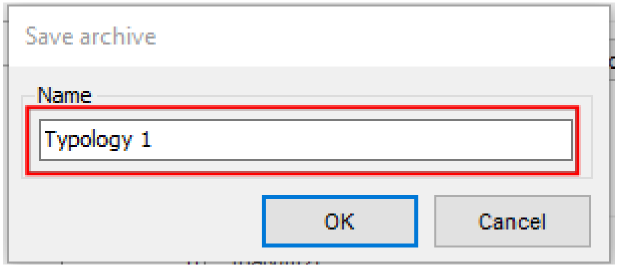
Enter all the parameters that define the floor (materials, characteristics etc ...).

2. Save the new type in the archive

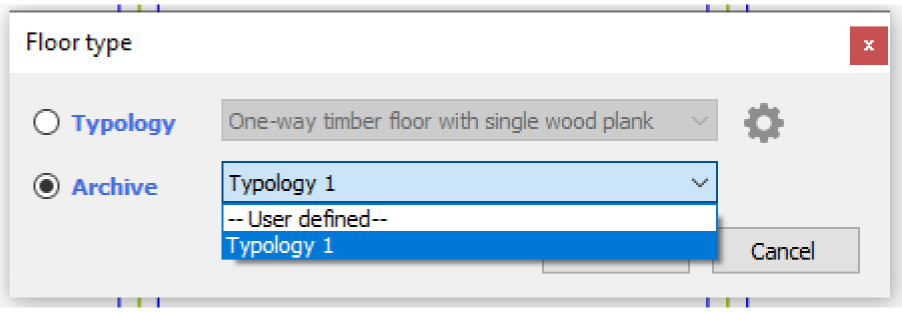


Click on the "save" button

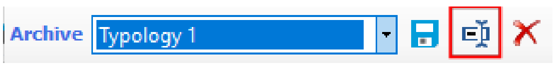
Assign a "Name" to the new typology to be inserted



Once these operations have been completed, the new type of floor will be added to the archive and it will be possible to assign it to any other floor added later.



It is possible to perform editing operations on the types of slabs created:



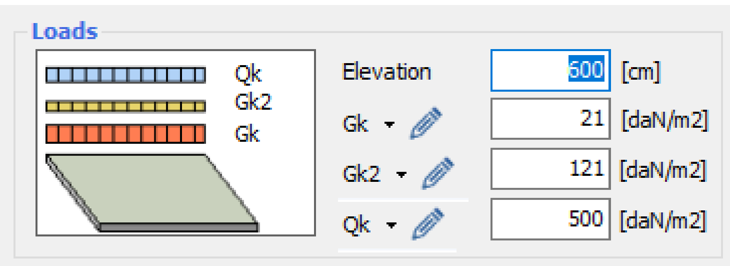
Command "**rename**": allows you to change the name assigned to the typology inserted in the archive.

Command "**delete**": allows you to delete the type of floor inserted from the slabs archive.

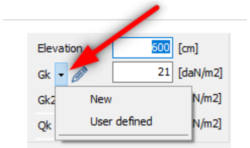
9.2.9.2 Floor load definition

There are three types of load that can be introduced:

- Structural permanent load: G_{k1}
- Non-structural permanent load: G_{k2}
- Variable: Q_k

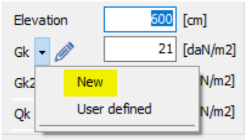


Define G_{k1} and G_{k2}



[New]:
Start an assisted procedure, aimed at identifying the intensity of the permanent load.

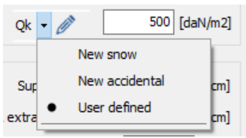
[User Defined]:
It is possible to manually enter the load value and the respective coefficients.



[New]:
By selecting the"new"option it is possible to select each material or element present in the material and surface loads lists and add it to the current load analysis by dragging or double clicking.
The total, given by the sum of the surface loads entered, is updated as the elements of the stratigraphy are added.
It is also possible to add customized materials whose characteristics are known, by entering specific weight and thickness, or, directly, the load per surface unit.

N.B.: In the case of modeling "effective deformability" floors, for which the program automatically calculates the stiffnesses of the floor by inserting the materials and geometries, the use of this function will not be necessary as the program conducts a pre-dimensioning of the loads based on the data entered.

Define Q_k



[New]:
- Select the "new snow" option which allows to calculate the actions due to snow, as illustrated by the DM of 17 January 2018 (NTC) and by the explanatory Circular no. 7 21st January 2019
- Select the "new accidental" option that allows you to insert a new accidental load as indicated in Table 3.I.II (NTC)

Following the "assisted" procedure on new or accidental snow:
- the text field containing the intensity of the load and the multiplying factors Ψ0, Ψ1, Ψ2; they are automatically filled in and made non-editable as they are linked to the choice of using an "assisted" input. To be able to edit the value it is necessary to use the "User definition" mode.

- In the drop-down menu, an entry is added with the name that the user has decided to assign to the procedure for calculating the load intensity during assisted input.

- The category identified based on the definition of the intended use is automatically defined.

The same load can be assigned to several floors simply by selecting it from the list.

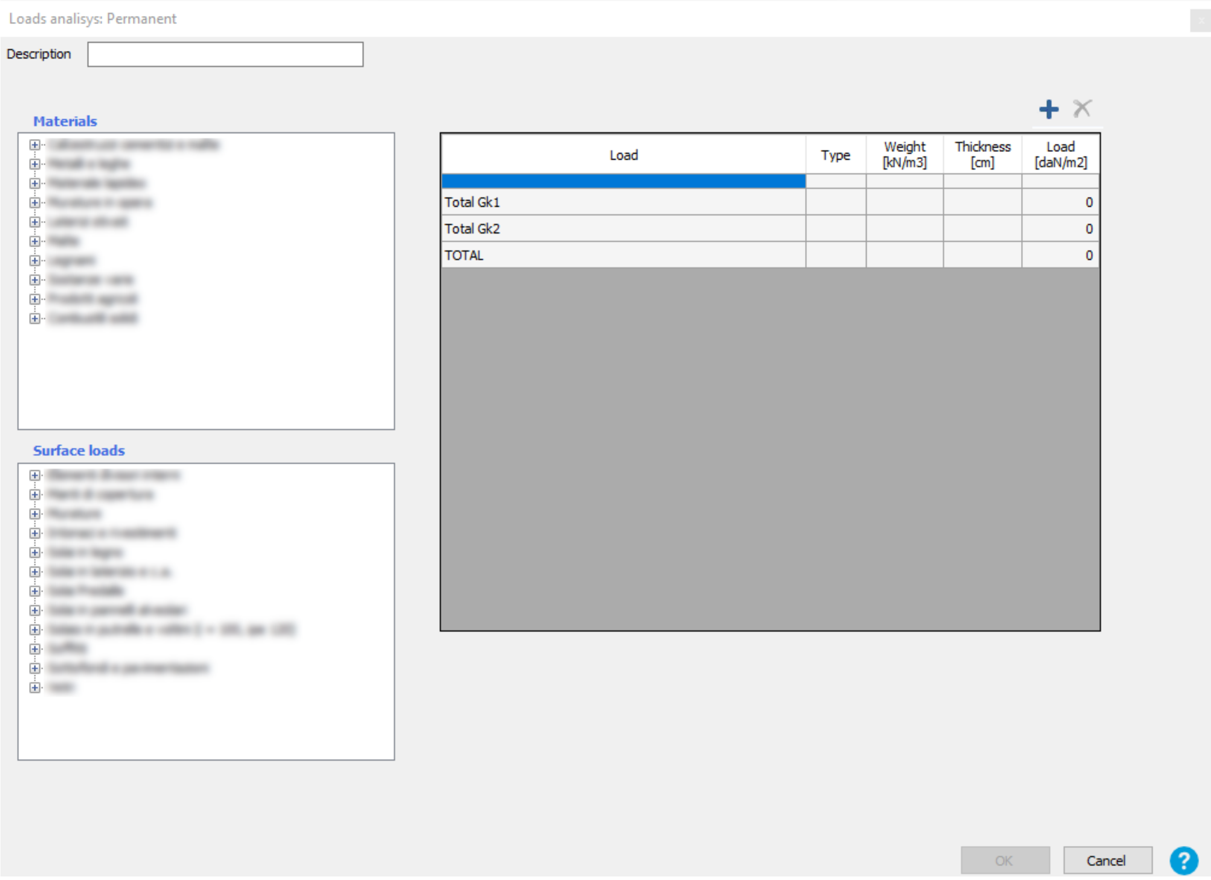
The assigned load will show the symbol "√" alongside

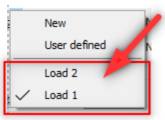
[User Definition]:
- Select the "User defined" option, which allows you to manually

enter the amount of the load

9.2.9.2.1 Permanent loads

Start an assisted procedure, aimed at identifying the intensity of the permanent load managed by the following window.





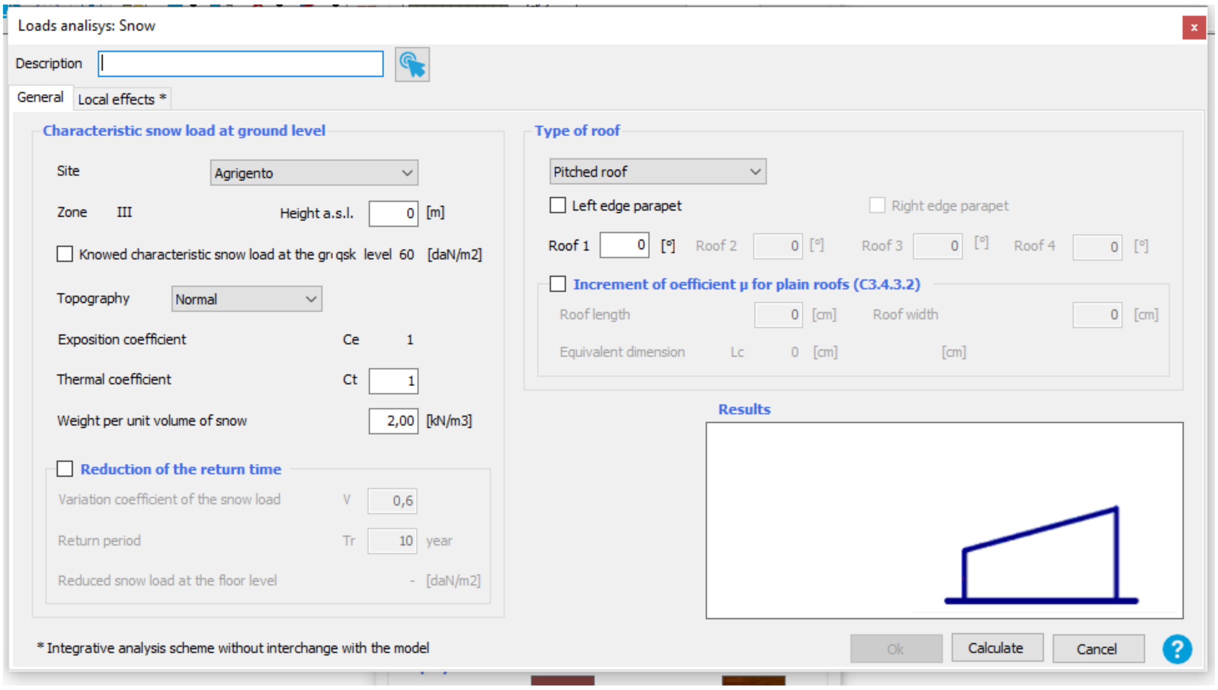
Following the "assisted" procedure:
- the text field containing the intensity of the load is automatically filled in and made non-editable as it is linked to the choice of using an "assisted" input.
To be able to edit the value it is necessary to use the "User definition" mode.
- In the drop-down menu, an entry is added with the name that the user has decided to assign to the procedure for calculating the load intensity during assisted input.

The same load can be assigned to several floors simply by selecting it from the list.

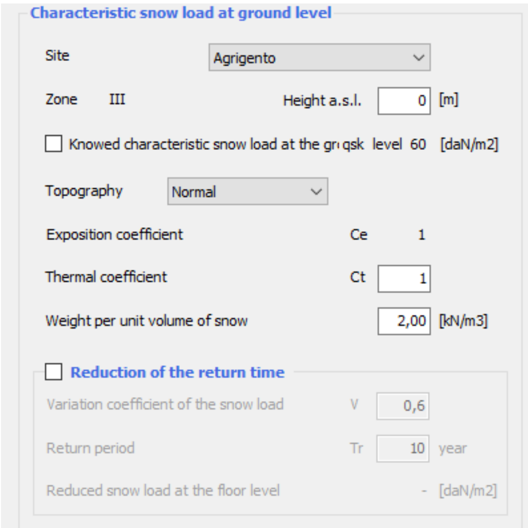
The assigned load will show the symbol "✓" alongside

9.2.9.2.2 Snow loads

The image below shows the work environment for snow loads:

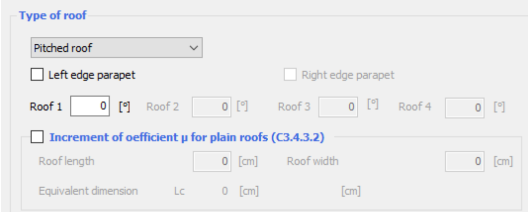


The window is divided into three sections, related to the characteristic snow load value at ground level, the type of roof and the results.



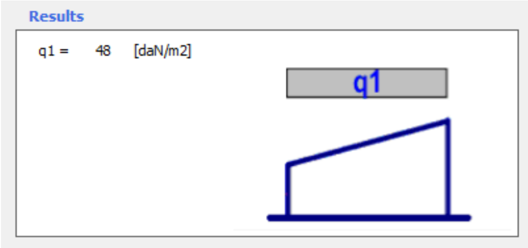
1. Characteristic snow load at ground level

The section provides the insertion of the geographical and topographical data of the project site, with the possibility of reducing the return period of the action if more accurate research is available.



2. Type of roof

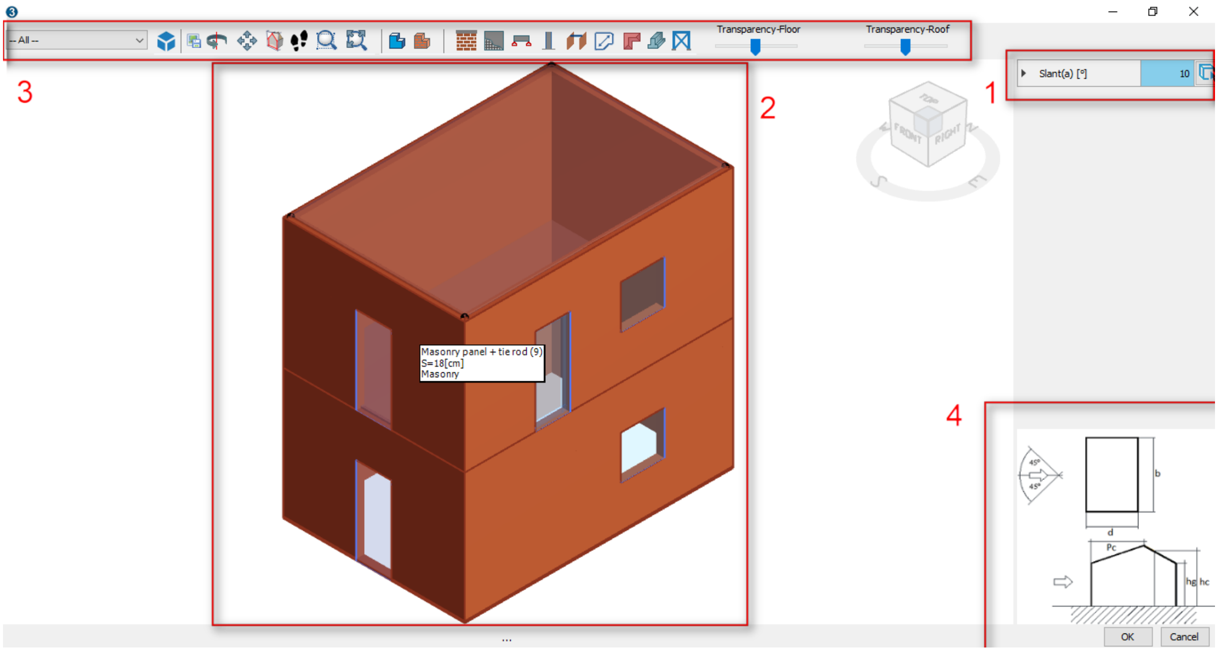
It is possible to choose between the different roof configurations listed in the NTC 2018 (Fig. 3.4.2, Fig. 3.4.3, Fig. C3.4.3, Fig. C3.4.4, Fig. C3.4.5); for each of these, the program will request the input data necessary to calculate the snow load.



3. Results

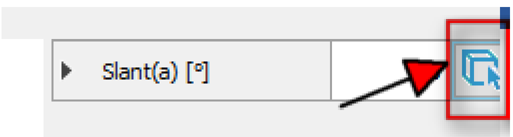
Depending on the roof configuration under consideration, the program will return the results of the global actions due to snow.

Button that allows interaction with the model, making it easy to enter data



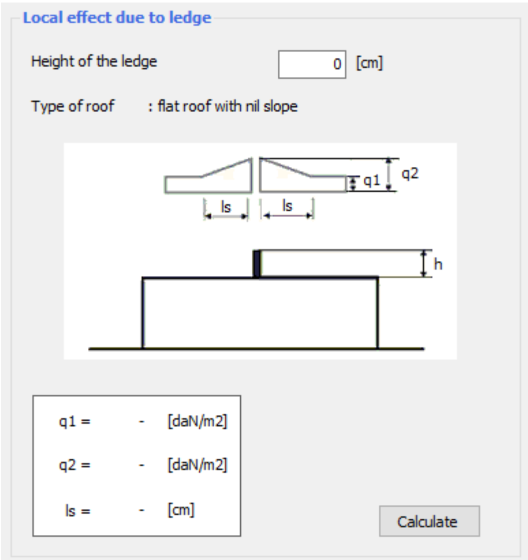
1 Data to be found for calculation purposes
2 Three-dimensional model from which it is possible to obtain the data necessary for the calculation
3 Display filters
4 Illustrative image

By clicking on the button indicated in the following image, it is possible to find the data useful for the calculation.



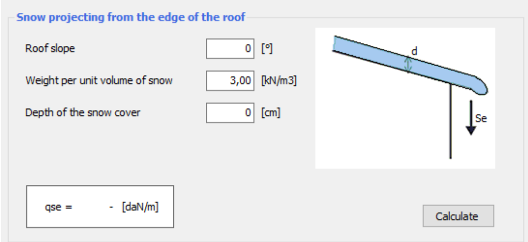
9.2.9.2.2.1 Local effects

The Local Effects window is divided into three sections, respectively relating to local effects due to ledge, snow projecting from the edge of the roof and snow load on covers and other obstacles.



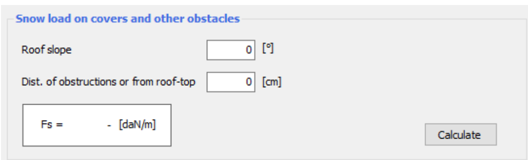
1. Local effect due to ledge

For the characteristic value of snow on ground level, the section calculates the local effects due to the snow load in the presence of a ledge of known height.



2. Snow projecting from the edge of the roof

For the characteristic value of snow on ground level, the section calculates the local effects due to the snow load projecting from the edge of a roof with a known slope.

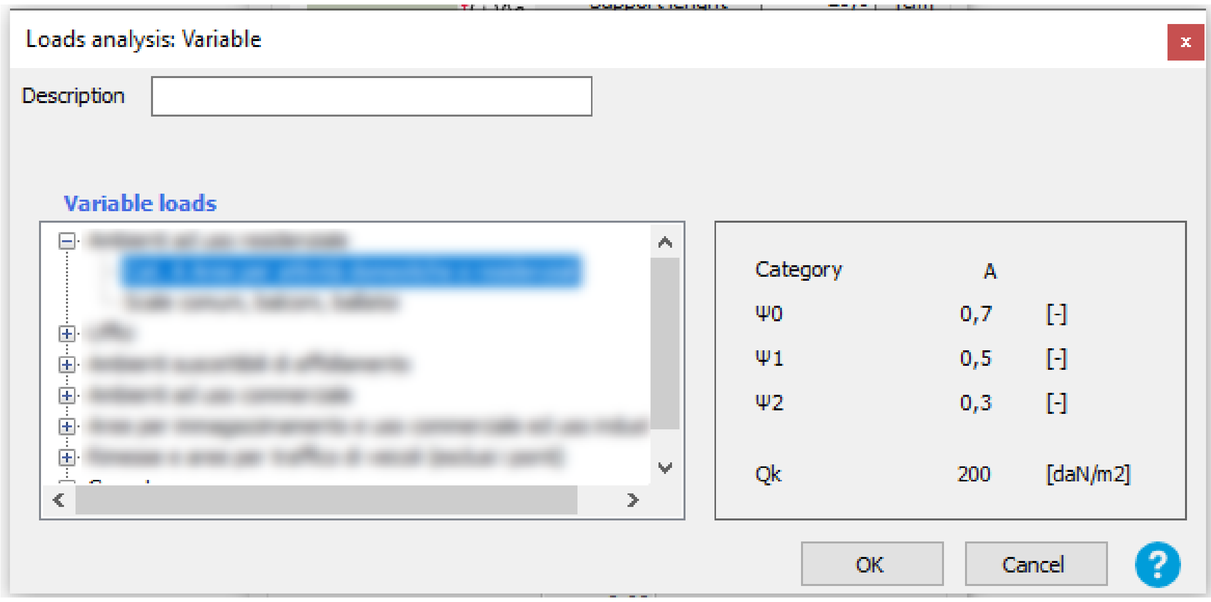


3. Snow load on covers and other obstacles

For the characteristic value of snow on ground level, the section calculates the local effects due to the snow load in the presence of covers and / or obstacles on the roof slope.

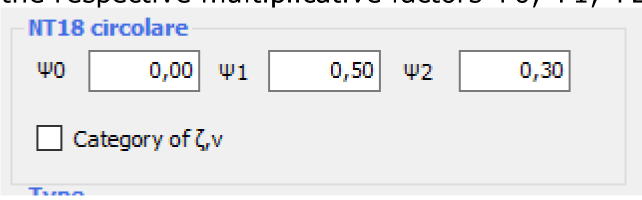
9.2.9.2.3 Accidental loads

Allows to select a new accidental load as indicated in Table 3.I.II (NTC)



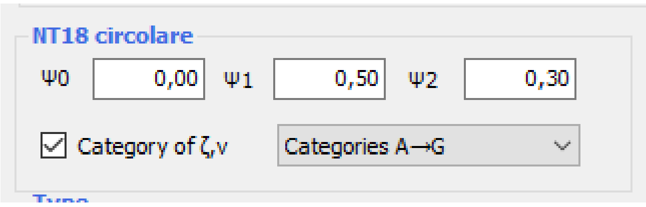
9.2.9.2.4 User defined

By choosing the "user defined" option, the load intensity field can be edited along with the respective multiplicative factors Ψ0, Ψ1, Ψ2



In this case, the "Category" associated with the intended use is not known because it has not been defined.
However, the category is useful for calculation purposes only if the calculation of the "ζv" factor is required.

By activating the box "calculation ζv" a drop-down allows us to define the category if necessary.

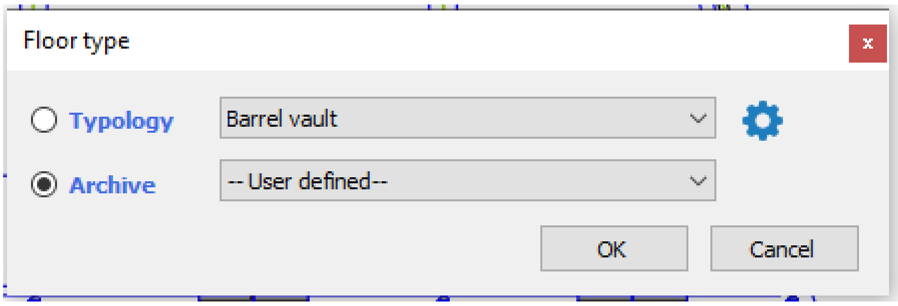


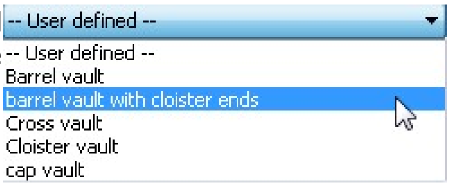
9.2.10 Vaults

Allows insertion of vaults.

Effective deformability floor

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

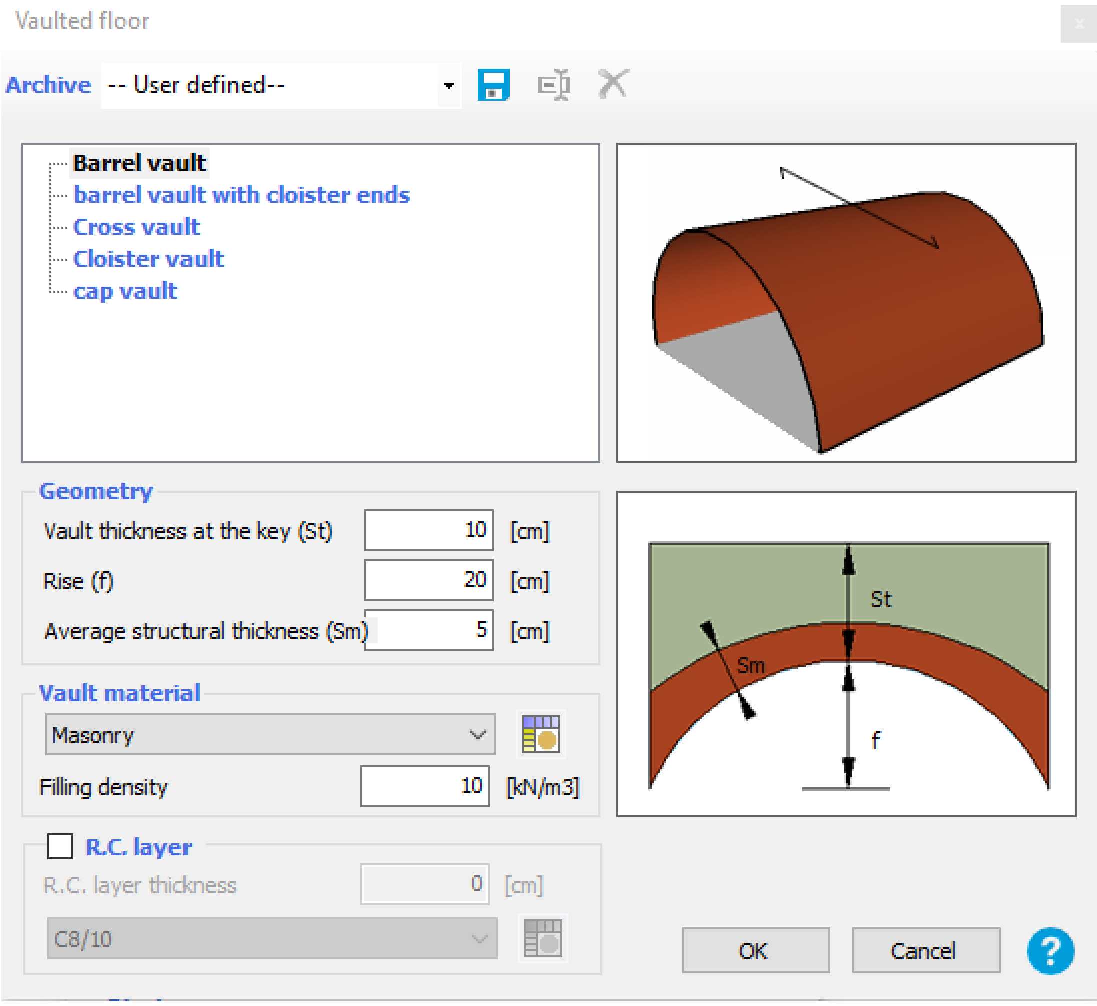




- The "**Typology**" field allows to select one of the possible types of floors.
- The "**Archive**" field allows to assign to the floor one of the types present in the archive.

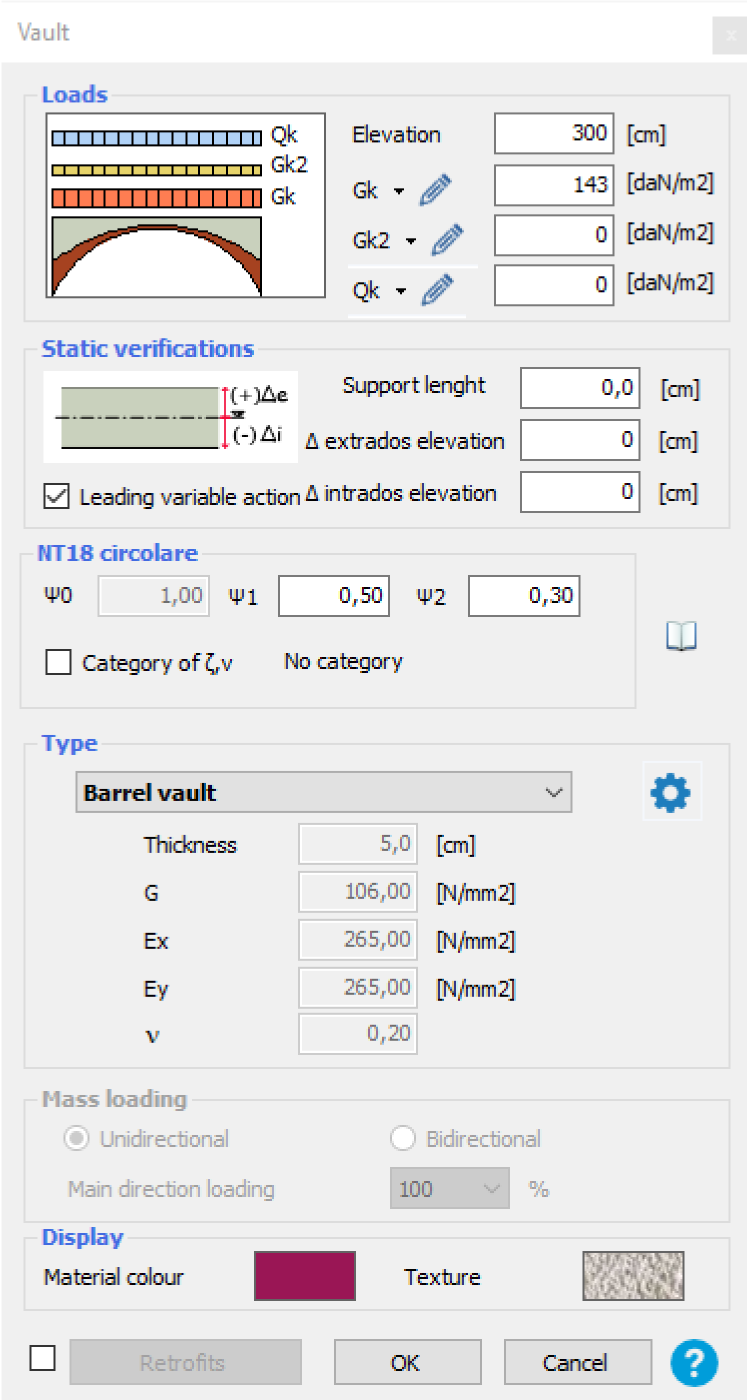
To create the floor archive, consult the dedicated section.

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).

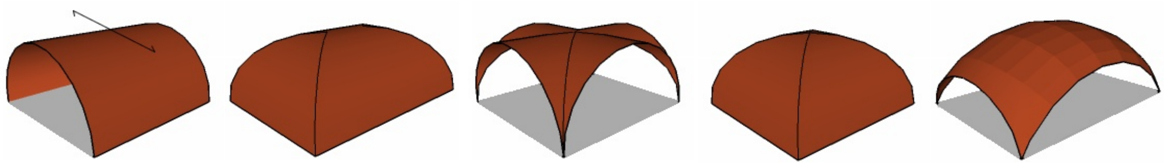
When selection is finished, the following window will appear.



This window is very similar to the floor window.
The permanent loads (G_k) must be interpreted as permanent loads (G_1), for the vaults it is calculated automatically based on the density of the filling material (this contribution does not include the weight of the "material of the vault" that is of interest the "average structural thickness").
To facilitate the input, the user is not required to enter the curvature of the vault by points, therefore the exact curvature is not known to the program; for this reason the weight of the vault is to be considered only indicative, the user is therefore free to modify this value at will with one calculated in a more rigorous way based on the characteristics of the specific vault.

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

Barrel vault	**Barrel vault with cloister ends**	**Cross Vault**	**Cloister Vault**	**Cap Vault**



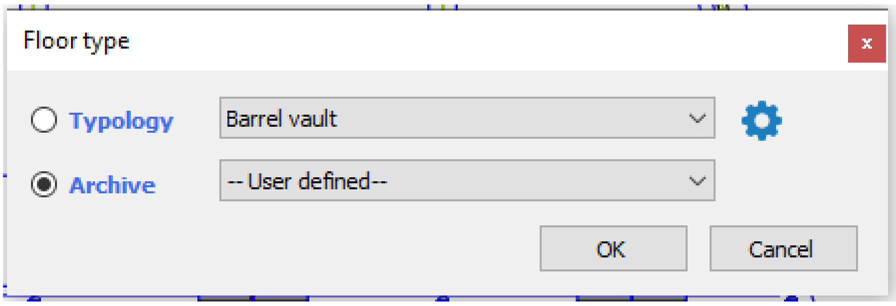
Rigid floor

This feature is not available in this version as the rigid deck theory is implemented.
This window is very similar to the floor window.
The permanent loads (G_k) must be interpreted as permanent loads (G_1), for the vaults it is calculated automatically based on the density of the filling material (this contribution does not include the weight of the "material of the vault" that is of interest the "average structural thickness").

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

9.2.10.1 Vault archive

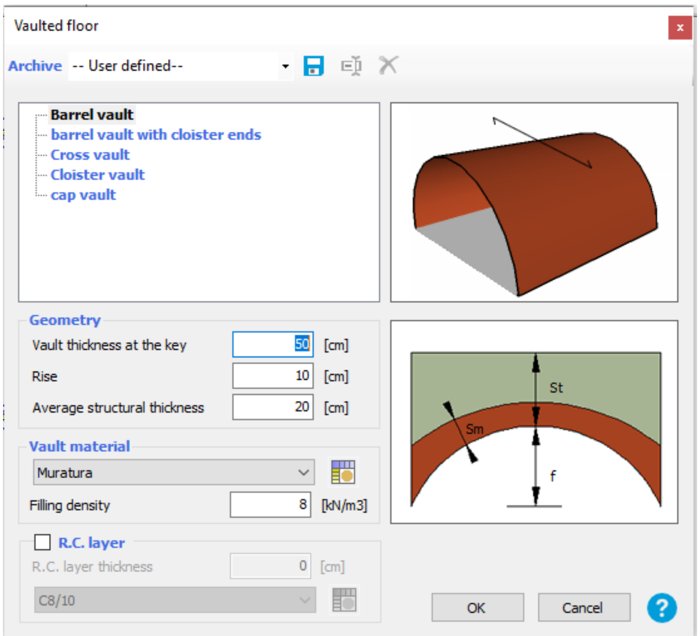
Once the vault insertion command has been selected, the following window opens:



By clicking the "archive" option, it is possible to assign one of the previously created types to the vault.

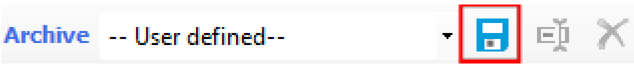
It is possible to add a new typology to the archive using the following steps:

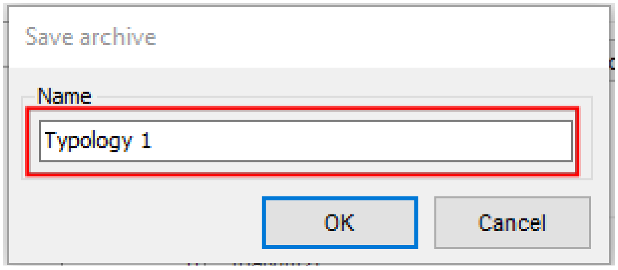
1. Definition of the characteristics of the floor



Enter all the parameters that define the vault (materials, characteristics etc ...).

2. Save the new type in the archive

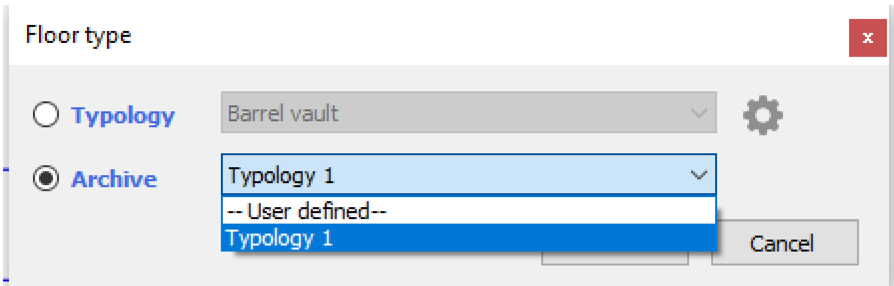




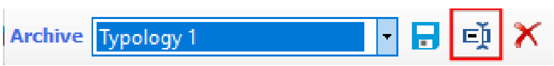
Click on the "save" button

Assign a "Name" to the new typology to be inserted

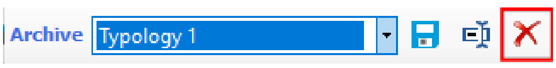
Once these operations have been completed, the new type of floor will be added to the archive and it will be possible to assign it to any other vault added later.



It is possible to perform editing operations on the types of slabs created:



Command "**rename**": allows you to change the name assigned to the typology inserted in the archive.

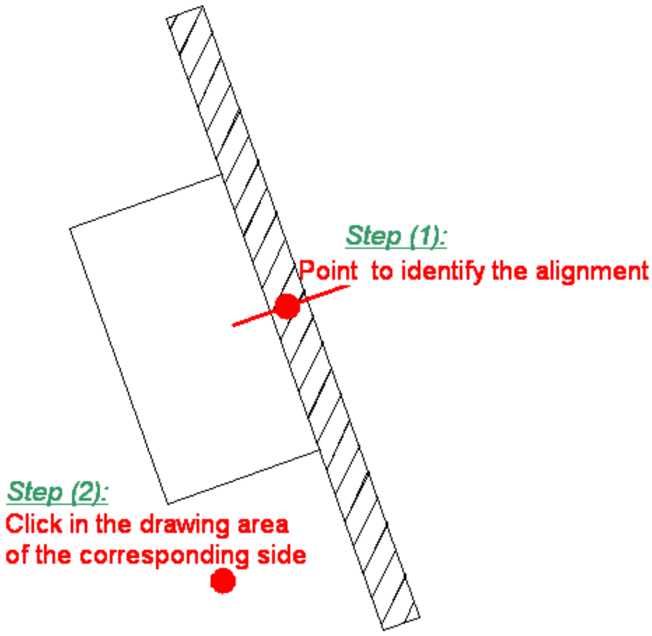


Command "**delete**": allows you to delete the type of floor inserted from the vault archive.

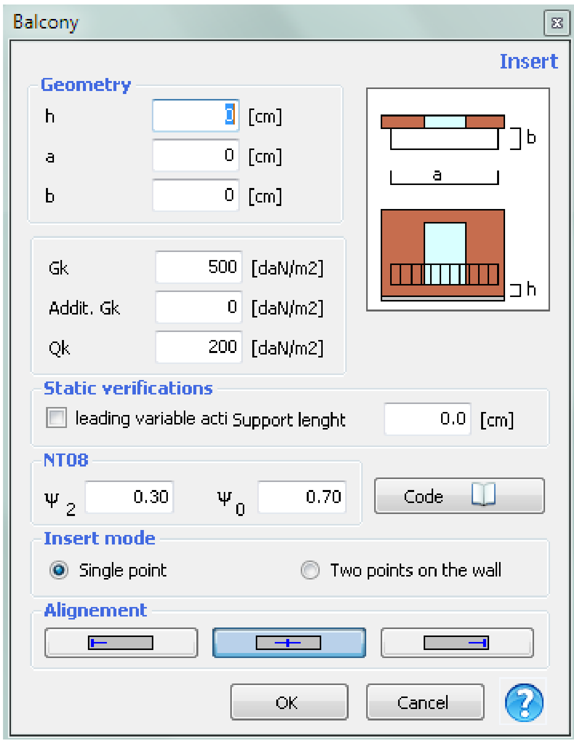
9.2.11 Balconies

Allows the insertion of balconies.
The insertion occurs through the insertion of the following parameters:
-Geometry: the geometry of the floor plan (axb); h indicates the difference between the elevation of the balcony and that of the lower level.
- G_k: structural permanent loads (G_1)
- $G_{k,agg}$: self weight of all non structural elements (G_2).
- Q_k: accidental loads
-Multiplier coefficient as defined by code.

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



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

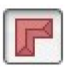


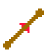
9.2.12 Roofs

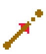
3Muri allows to model roofs on different levels, therefore for entering or editing a roof the first thing to do is to select the level you want to work on.

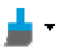
The insertion of the roof elements is done by using the relevant button of the environment structure.

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

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

 **Profile** Allows to insert a "roof profile"

 **Extend Profile** Extends an existing profile

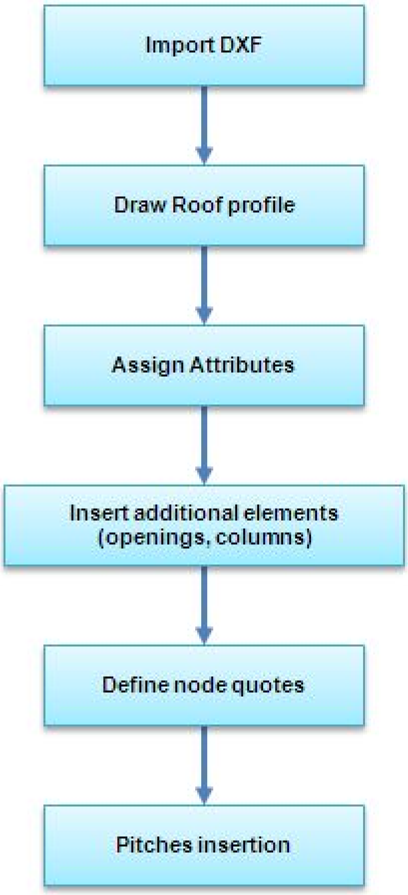
 **Copy** Copies the definitions of the structural elements characteristics.

structural elements properties	
Paste properties	Paste the properties of the selected element using the copy command.
Delete selected structural elements	Delete various already inserted elements.
Insert wall segment attributes	Assigns the attributes of the structural objects
Insert node	Allows the insertion of a node "element node" (with the help of the graphic support) or within a predetermined distance from an edge node.
Opening	Allows to insert an opening in a panel.
Column	Allows to insert a column matching one or more nodes.
Node elevation	Assigns the height of nodes
Roof slope	Allows to insert the pitches
Load	Allows to insert concentrated/linear loads

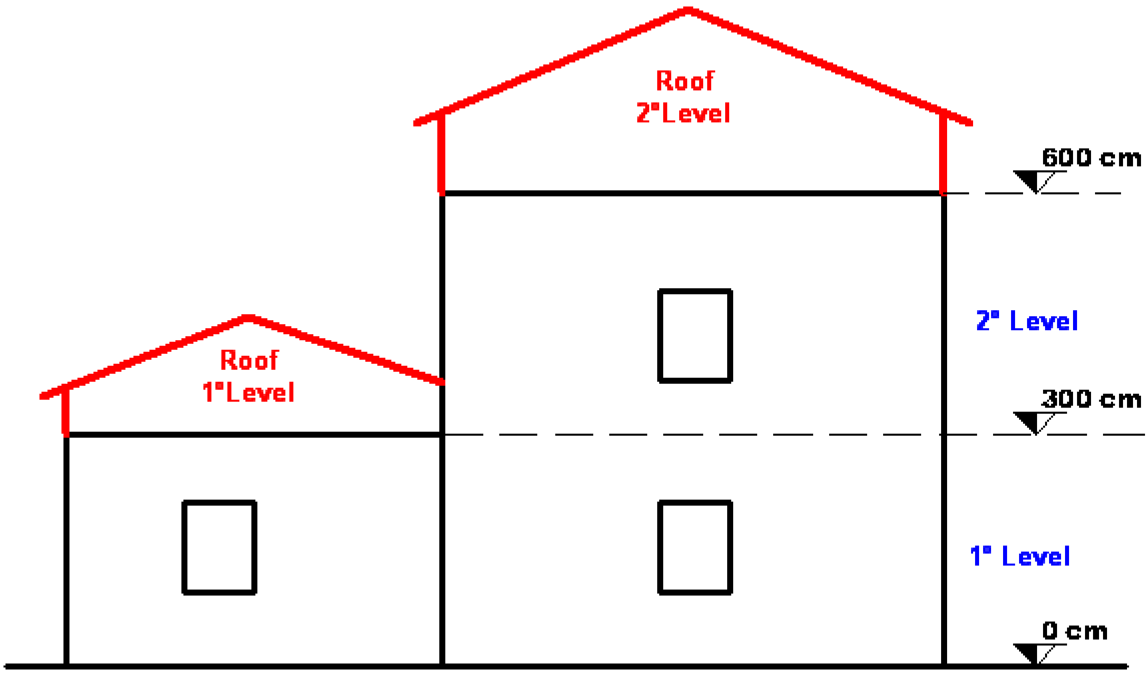
Info:

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

9.2.12.1 Modeling Mode



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



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

NON-structural:

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

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

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

Structural:

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

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

Example of mesh shown in the analysis environment

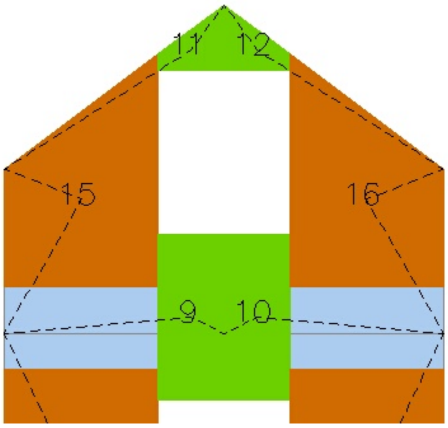
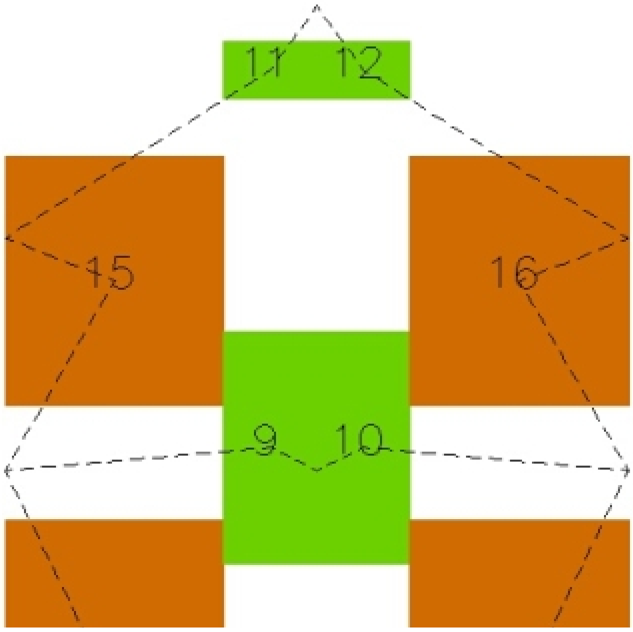
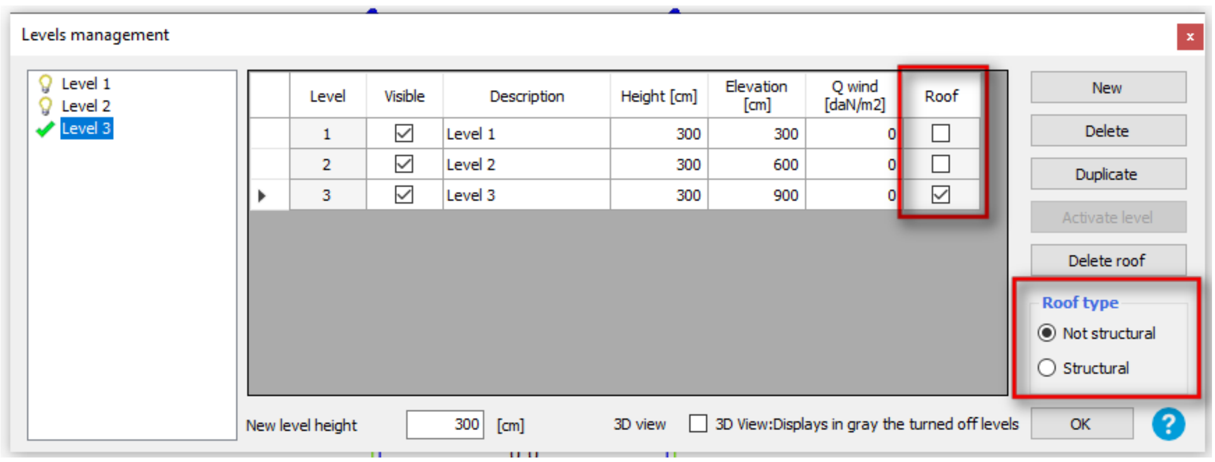


Diagram of the pier/spandrel elements with the actual heights used for the calculation



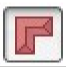
The typology of the "non-structural" or "structural" modeling is conducted by the appropriate function in the window "Levels management".
This option is a property of the individual level, allowing to decide whether a roof is both structural or less depending on the belonging plan.



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

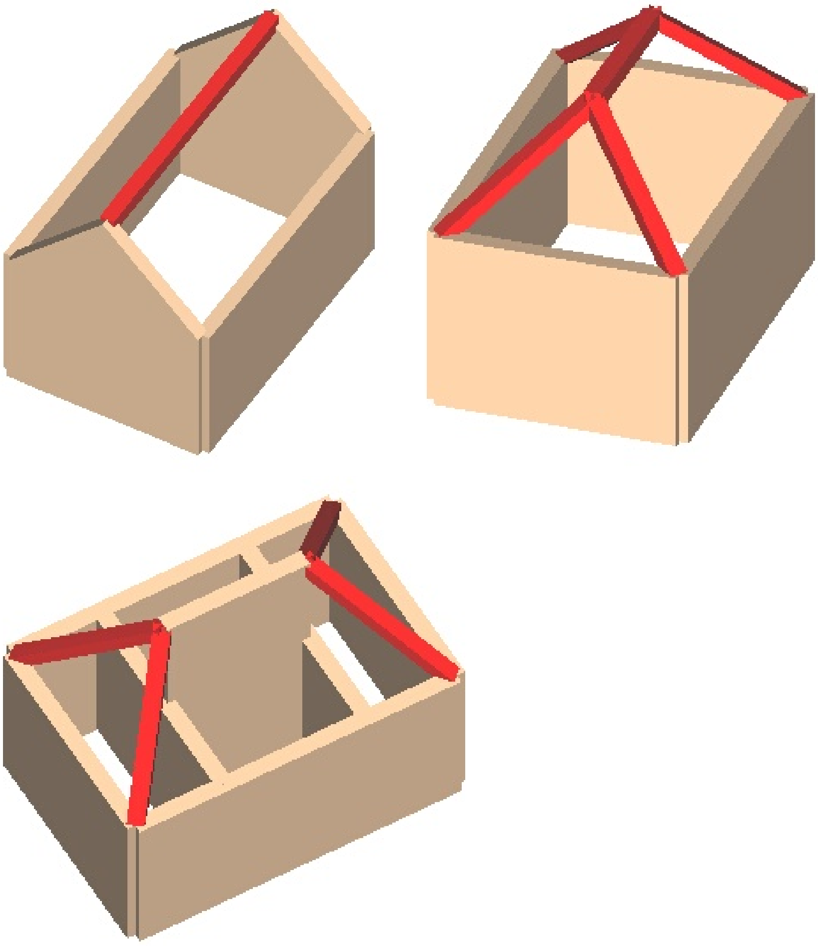
You may decide to remove a roof from a level by simply clicking on the ***"Delete roof".***

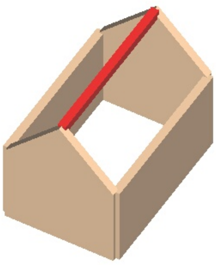
9.2.12.2 Roof Elements

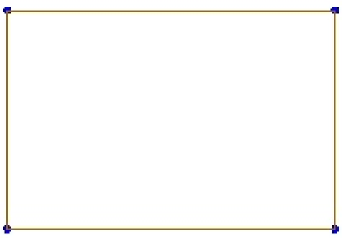
The insertion of roof elements is done in the proper environment which can be accessed by clicking 

Profile:

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





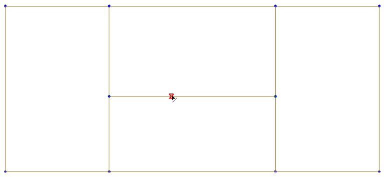


Once you enter in the "roof" environment you can see only the elements already present in the "walls" environment.

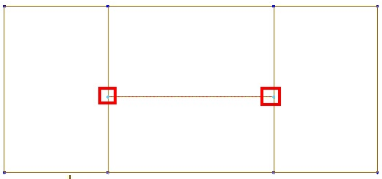
You need to add a single roof "*profile*" with the aim to describe the ridge

Extend Profile:

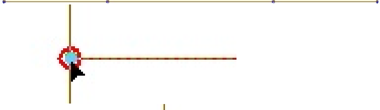
It allows to extend an already set profile, let's look at the utilization phases:



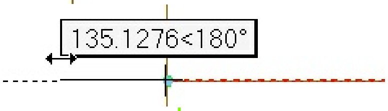
Select the profile or wall that you want to extend

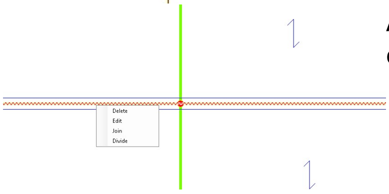


The endpoint joints of the selected segment are highlighted in blue.



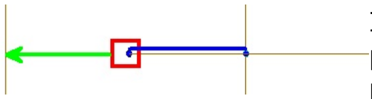
Click on an endpoint node (selected) and drag to extend the segment.



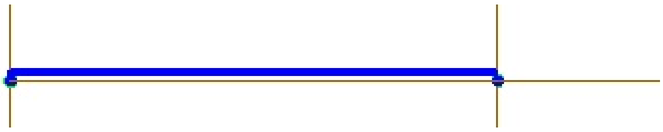


An "edit" or "delete" command takes effect only on the extended portion of the object.

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



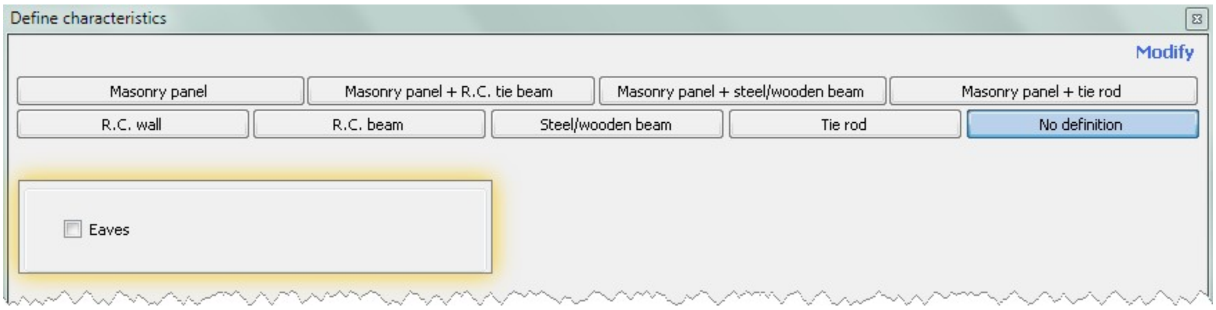
If you wish to extend the highlighted node on the wall is not possible because the highlighted line in blue is already the product of an elongation.
1. delete the inserted part: right click on it and choose "Delete"
2. redo an extension for the entire stretch.



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

Assign attributes:

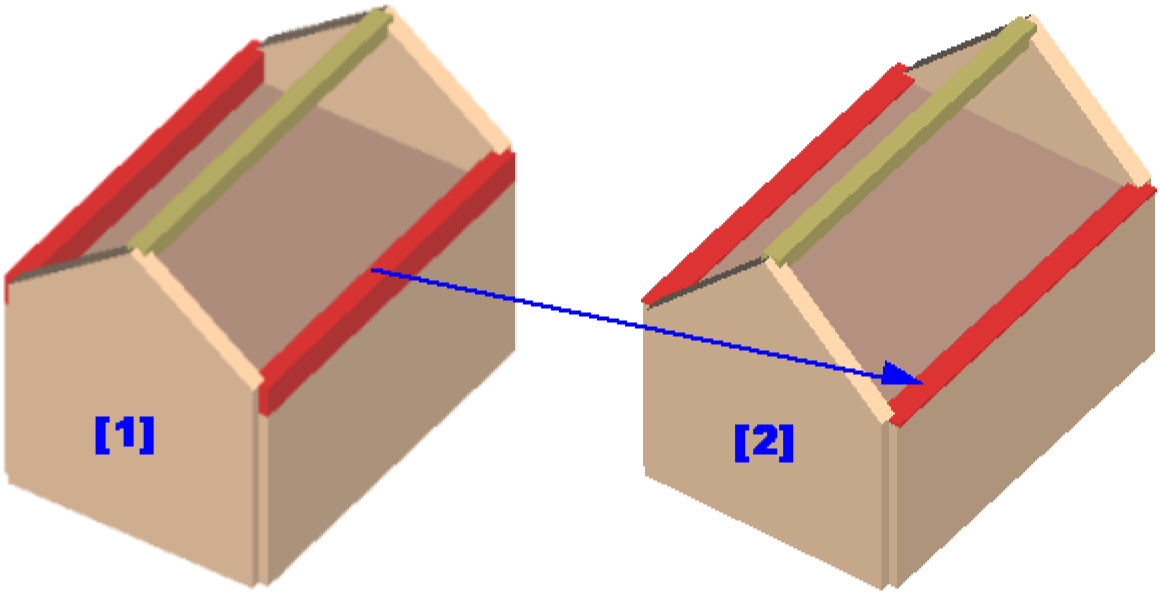
This command is very similar to the "Definition of structural objects" (Simple elements, Composite elements) already known to the structure environment.
In addition to the classic features, there is also the possibility to define a **"Pitch support".**

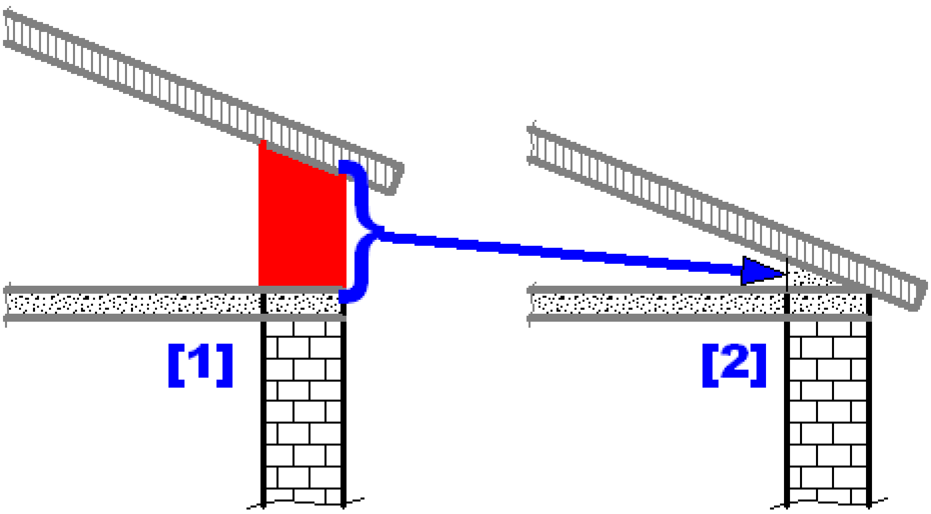


When is it possible to use the definition ***"Pitch support"?***
All the times when a pitch is supported by a structural element already defined in the structure, let's now see some examples:

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

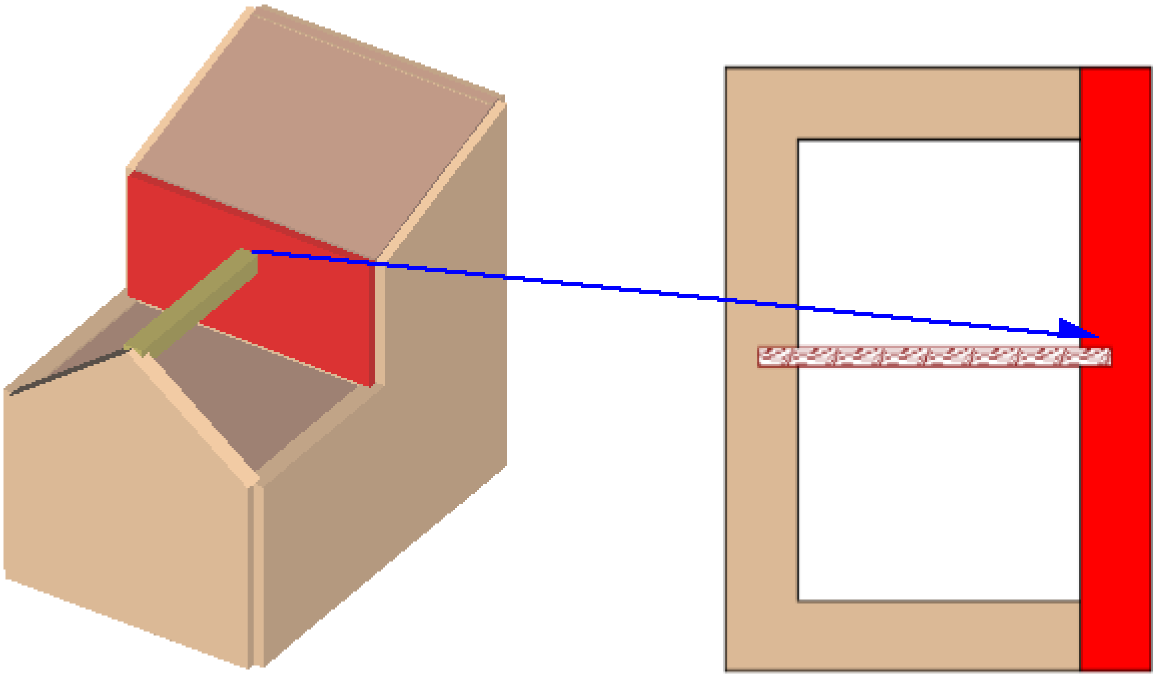
In Figure [1] the pitch is supported in correspondence of the eave by a masonry wall defined in the roof environment→ it is **NOT** *"pitch support"* but "*wall panel*".
In Figure [2]] the pitch is supported in correspondence of the eave by a masonry wall defined in the structural area that doesn't exist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





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

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



Opening:

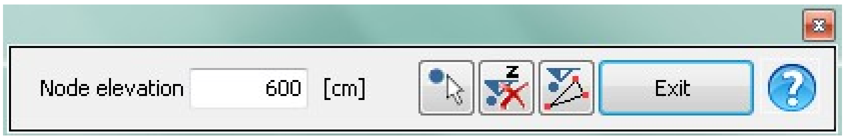
In the walls of the roof environment can be inserted openings exactly like in the structural area.
More details in the corresponding chapter "Structure Features>Definition of Structural Objects>Openings".

Column:

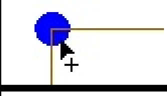
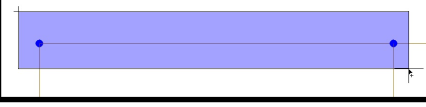
As well as in the structural environment can be inserted columns in reinforced concrete, masonry, steel and wood structure.
More details in the corresponding chapter "Structure Features>Definition of Structural Objects>Columns"

9.2.12.3 Define the heights

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

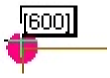


The "*Quote node*" field presents by default the maximum height of the active layer that corresponds to the minimum height of the begging of the roof.
Phases of the dimensioning of the nodes:
1. Enter the height value of the nodes
2. Press , the pointer changes shape  and the definition window of the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

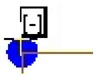
Selection mode	Description
Single click on the node	
Multiple selection with window mode	

4. Press the right mouse button to confirm the selection; the definition window of the height reappears in the foreground.

A ***dimensioned node***, is shown by magenta.
By placing the pointer on the node, the value of the allocated height is showed in square brackets



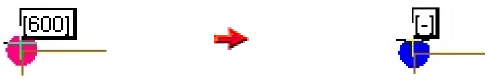
A ***NON dimensioned node***, is shown by blue.
By placing the pointer on the node the sign "[-]" appears to indicate that the height is not allocated.



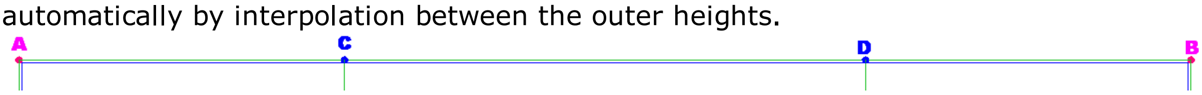
5. Repeat the steps described so far until you have entered all the necessary heights.
6. After entering all the heights press **"Exit"** to exit.

 Cancel the input of a height

Allows to remove the definition of the node height.



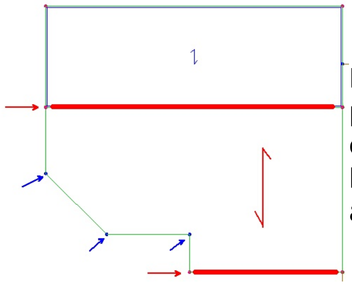
Why is necessary cancelling the insertion of a height?
The not defined (canceled) heights, if affected by a pitch profile, can be calculated automatically by interpolation between the outer heights.



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

 Calculate height

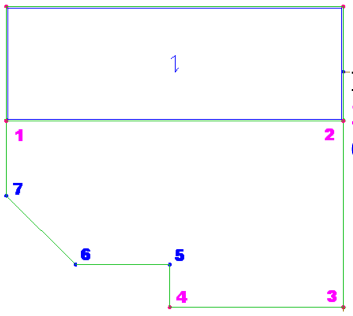
There are many cases in which the ground survey is not accurate enough and doesn't allow to know precisely the proportion of all the nodes.



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

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

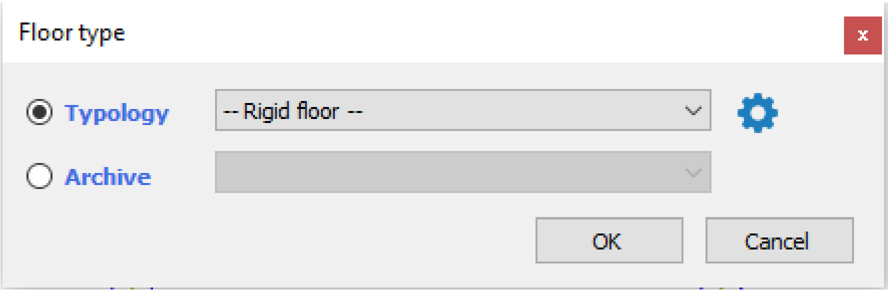
Select in sequence first the 3 nodes (dimensioned) that define the plane and then those for which you want to calculate the height.



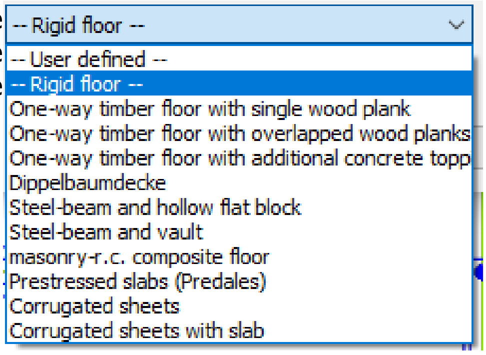
In the case of the example described above you can select 1-2-4-5-6-7 so that the program calculates the dimensions of 5-6-7 so that they are coplanar with 1-2-4 to create a flat pitch.

9.2.12.4 Pitches

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

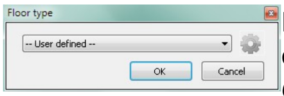


- The "**Typology**" field allows to select one of the possible types of pitches. The types presented are the same as those already described for the floors.

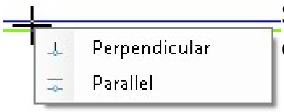


- The "**Archive**" field allows to assign to the pitch one of the types present in the archive.

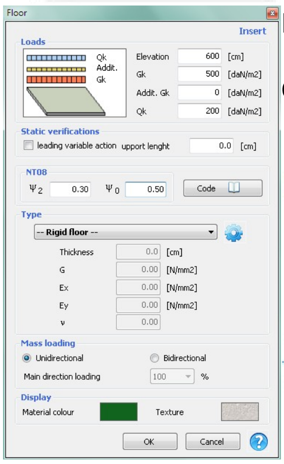
To create the pitch archive, consult the dedicated section.



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

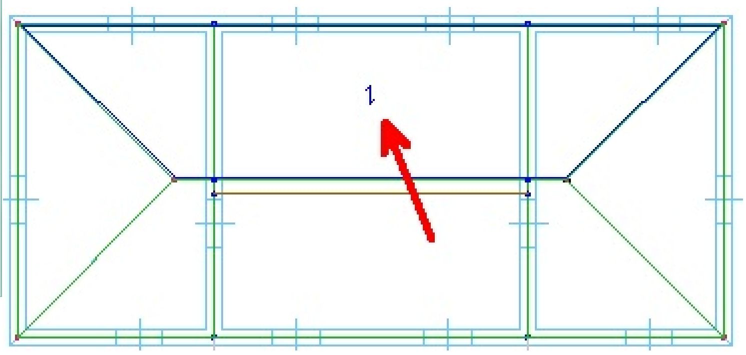


Select with the left mouse button a profile / wall roof to bring up the context menu of the direction of framework.



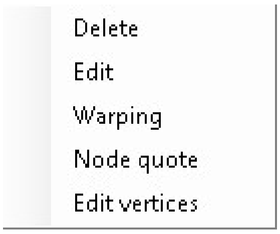
Define loads and characteristics of the pitch.

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

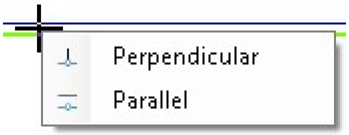


Edit pitch

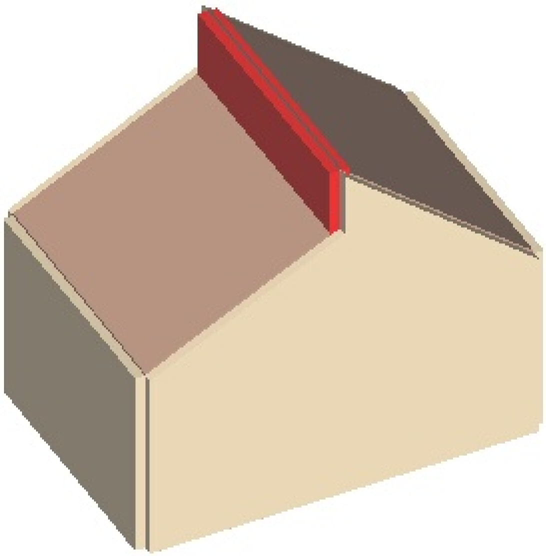
Clicking with the right mouse button on one side of the pitch, the contextual menu that allows editing will appear.



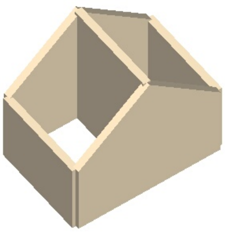
Cancel: Cancel pitch
Modify: Appears the input window used to define the loads and allows editing.
Pitch warping:
If the pitch's warping previously inserted, it is not correct and you want to change it, by selecting this command is again required the selection of the structural reference for warping; by clicking on a stand is resubmitted the menu for the selection of the warping.



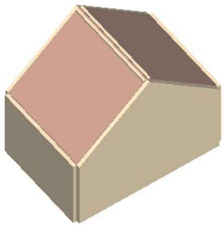
Pitch nodes heights:
Allows to act on the height of the nodes of the pitch without changing the height of the node itself.
The most common application of this command is to create a model shown in the following figure (in red):



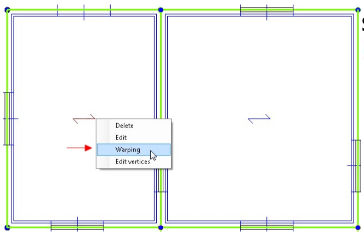
Let's see the steps of creation of this model:



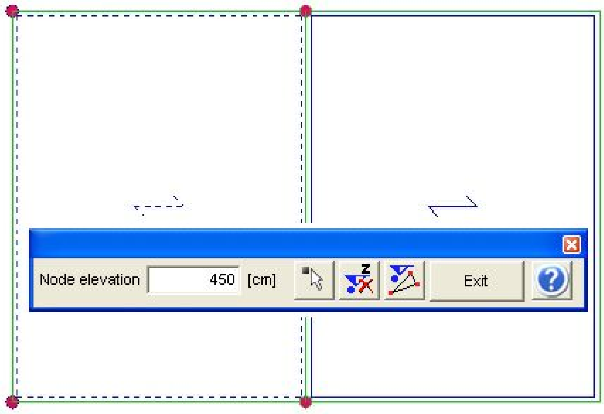
Create the model with gable and walls that extend up to the maximum height of the ridge.



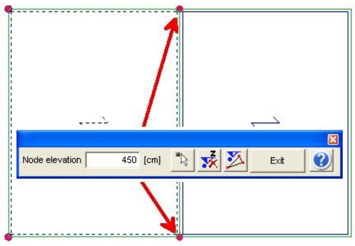
Insert the pitches with the above procedure



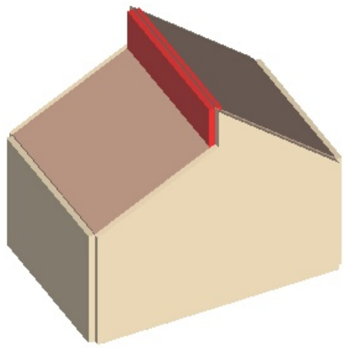
Select the pitch of the left with the right mouse button and select *"Height pitch nodes ".*



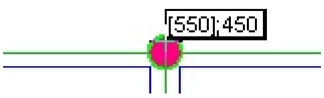
The defining heights nodes window appears with highlighted dimensioned nodes that affect only the selected pitch.



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



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



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

Edit vertices:

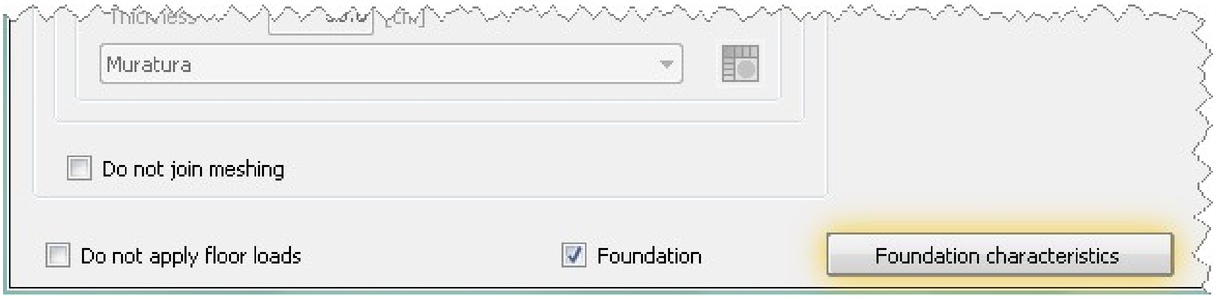
It activates the handles corresponding to the vertices of the slab polygon.
By dragging the individual vertices it is possible to change the incidences.

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

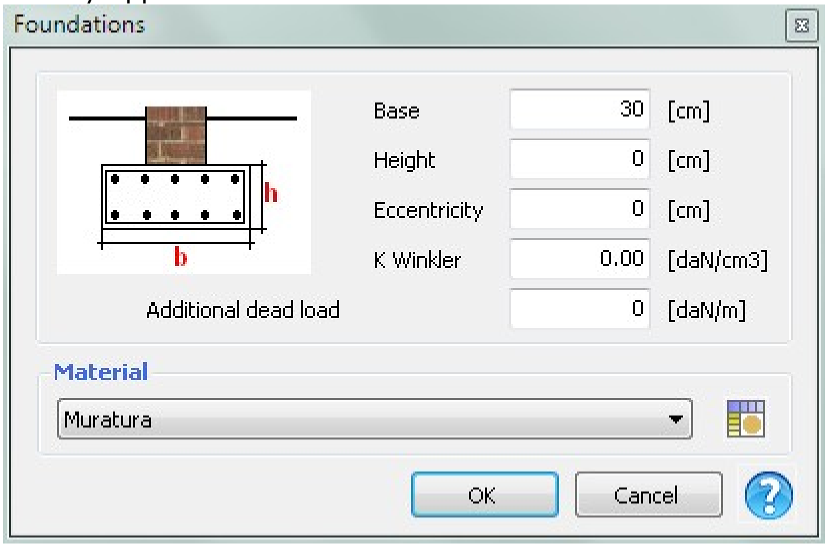
9.2.13 Foundation

9.2.13.1 Masonry Panels Foundation

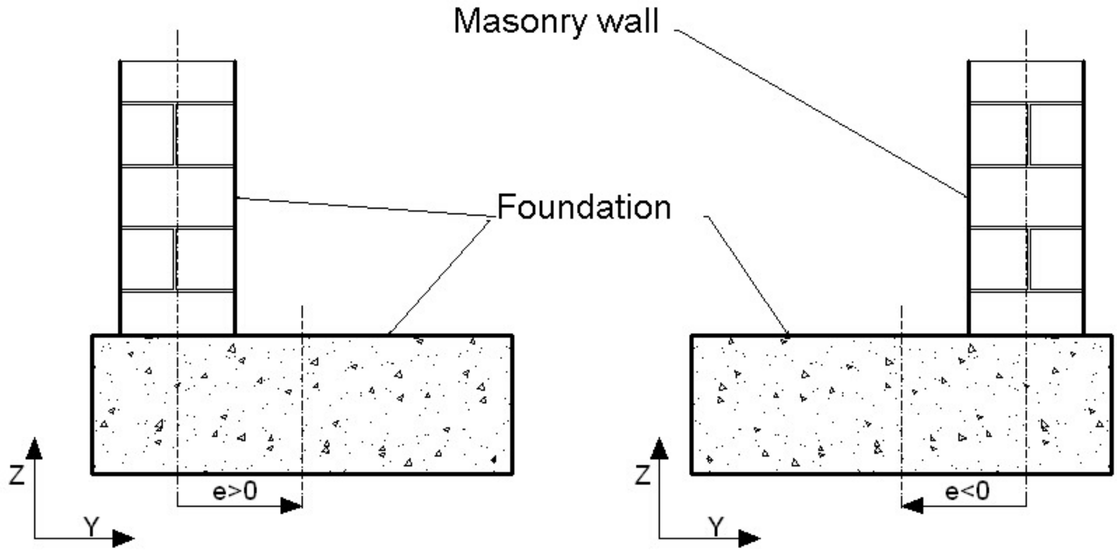
There is the possibility to activate a box named "Foundation" so that the user during the insertion phase can decide if each panel goes directly to the foundation in its low part so as to define the nodes.



This item appears active and not editable when you insert the first level because the nodes at the base are certainly part of the foundation; it appears disabled but editable in the next levels. The choice of a panel that goes directly to the foundation means to constrain all the freedom degrees of the nodes to the base (both translation and rotation). Different boundary conditions can be inserted only during the mesh editing in the analysis while viewing the prospectus of a mesh wall.
After deciding that a structural element taken in consideration is part of the foundation, the "Foundation Characteristics" button is activated in order to define the foundation system's characteristics necessary to calculate the tensions in contact with the ground.
The displayed dialog defines:
- the foundation beam's dimensions, the material
- The Kwinkler
- The eccentricity of the masonry panel compared to foundation beam
- a dead load directly applied above the foundation

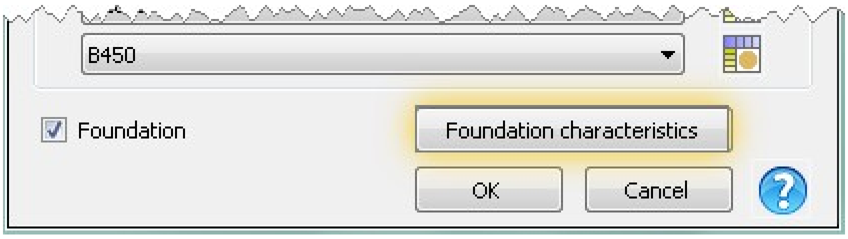


The foundation's eccentricity is calculated with the convention sign shown in the following figure.



9.2.13.2 Columns Foundation

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

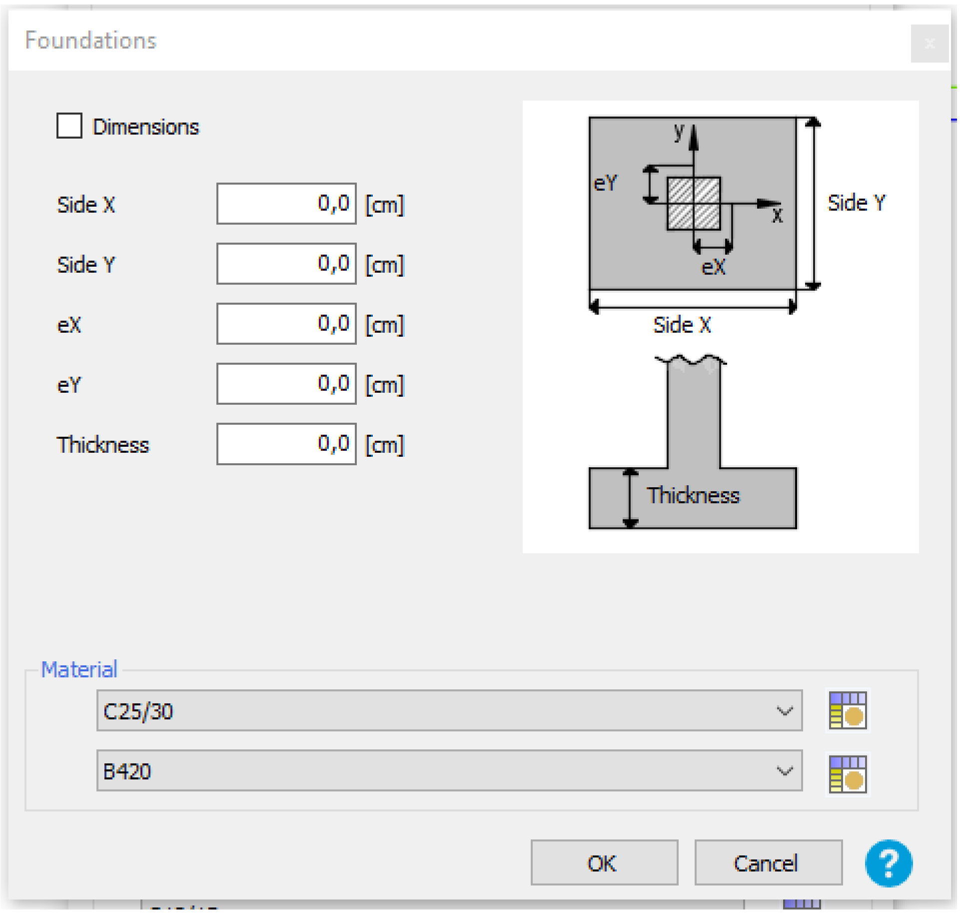


This item appears active and not editable when you insert the first level because the nodes at the base are certainly part of the foundation; it appears disabled but editable in the next levels. The choice of a column that goes directly to the foundation means to constrain all the freedom degrees of the nodes to the base (both translation and rotation). Different boundary conditions can be inserted only during the mesh editing in the analysis while viewing the prospectus of a mesh wall.

After deciding that a structural element taken in consideration is part of the foundation, the "Foundation Characteristics" button is activated in order to define the characteristics of the ground.

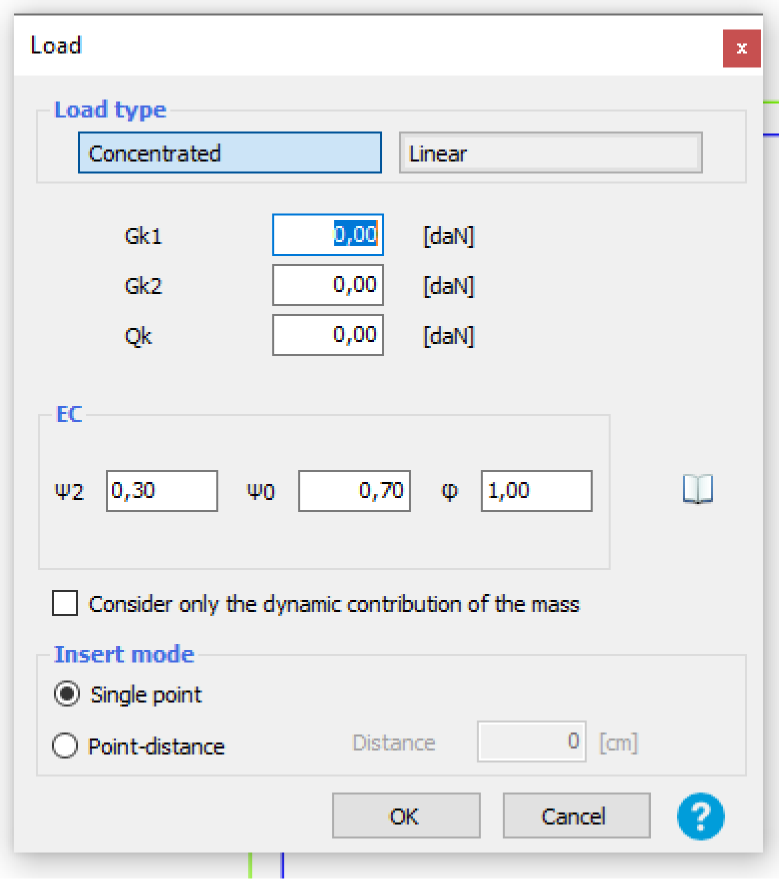
The displayed dialog defines:

- The Kwinkler
- The foundation's material
- Geometry and position respect to the column



9.2.14 Concentrated and linear loads

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



Consider only the dinamic contribution of the mass

This option can be used when it is needed to consider only the dynamic contribution of a mass and not that of the vertical loads.

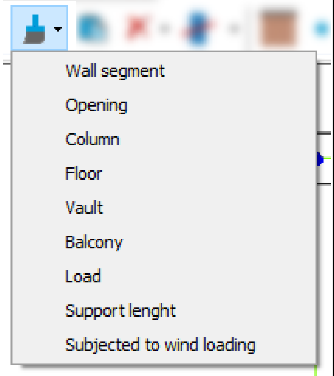
Practical example:

Think of a masonry built in contiguity with another wall facing, there are connections between the two facing but capable of transmitting only the horizontal forces, letting the mass of the masonry discharge directly onto its foundation without weighing on the contiguous facing. In this case the weight of the masonry must not weigh on the structure and also the contribution of resistance is not desired to come into play, but the dynamic effect of the masonry on the adhering structure remains.

The plan view of the loads can be shown / hidden using the filters

9.2.15 Structure Editing

Copy attributes

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

 Paste attributes

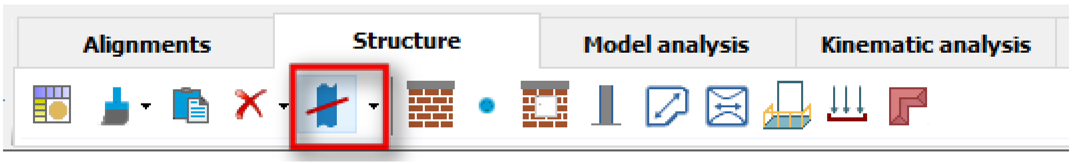
Paste the properties of the selected element using the copy command.
Select, in order, the structural objects that will have the copied properties assigned to them.
End selection of multiple items by pressing the right mouse button.
A video with the characteristics of the structural elements to be assigned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

 Multiple Deletion:

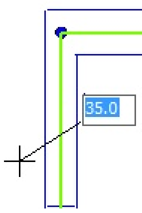
	Delete various already inserted elements. Using the dropdown menu, decide the type of element to be deleted. Select, in sequence, the elements to be deleted. The right mouse button can be used to stop multiple selection and proceed to deletion.

9.2.16 Supported thickness

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

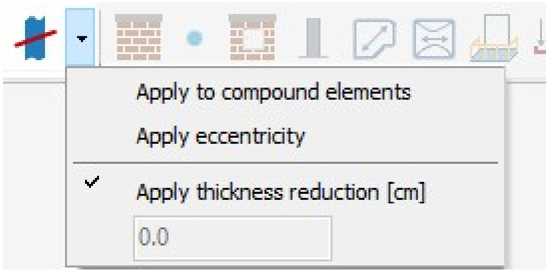


 **The command runs immediately by pressing the apposite button:**



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

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



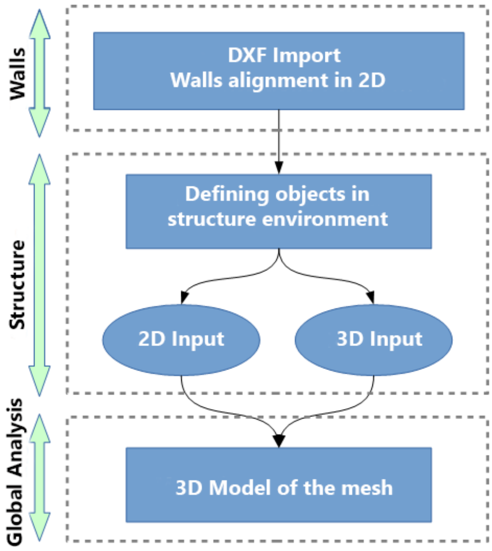
Apply to the compound elements:
The "Supported thickness" command automatically detects the thickness of the masonry walls; often an element can be "compound", for example the combination of a panel + R.C. tie beam. With this option checked [✓] the base of the R.C. tie beam will also be updated.

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

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

9.3 Input 3D

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

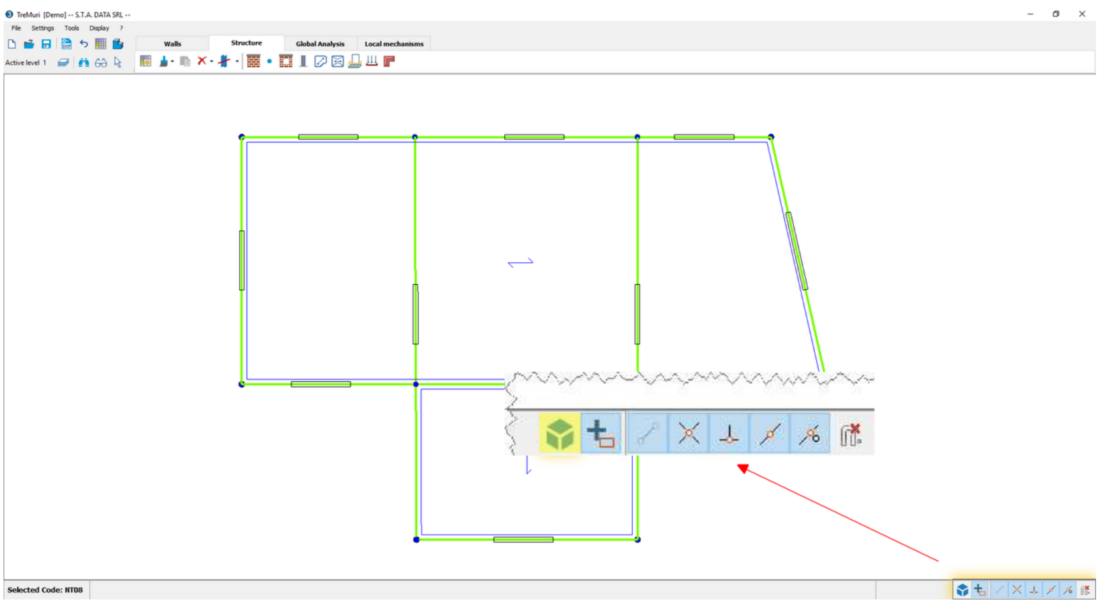


The process goes through the tracking of the walls in plan (the same as in the 2D mode) to define the main alignments in structure.

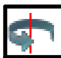
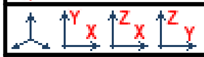
In the structural area is possible to decide whether to continue with the input in plan or to switch to 3D mode.

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

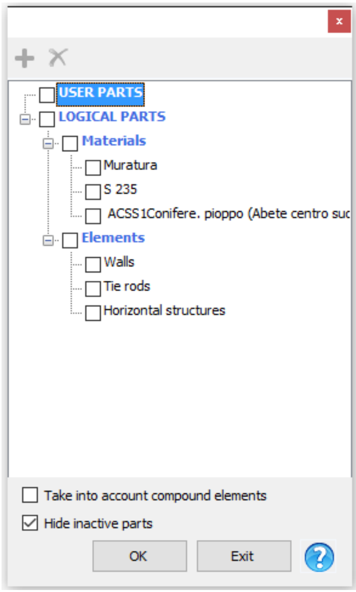
 The access to the 3D area is performed by using the specific button on the toolbar at the bottom right of the structure area.



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

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

9.3.1 3D Filters-the parts



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

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

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

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

9.3.2 Objects' input

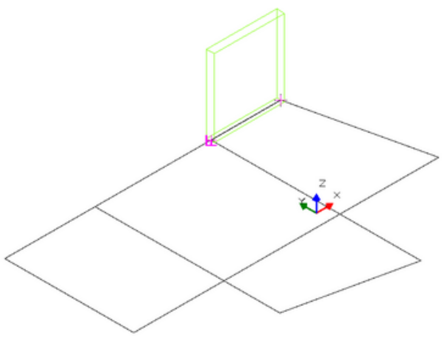
The commands of the bar structure present in the 3D input mode are also available in 2D mode.

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

	2D	3D	Description
	✓	✓	Materials
	✓	✓	Copy Properties
	✓	✓	Paste Properties
	✓	✓	Delete selected structural elements
	✓	✗	Perceived thickness (read walls' thickness from dxf file)
	✓	✓	Insert wall segments attributes
	✓	✗	Insert node ("divide" operation available in the element properties)
	✓	✓	Opening
	✓	✓	Column
	✓	✓	Slab
	✓	✓	Vault
	✓	✗	Balcony
	✓	✗	Load
	✓	✗	Roof

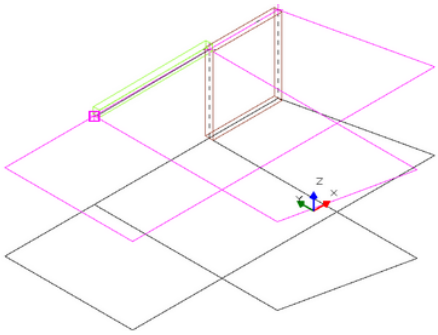
All the structured objects inserted in 2D mode require the position where to be located and then the mechanical characteristics, the 3D mode instead requires to define the geometry first and then the location.

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



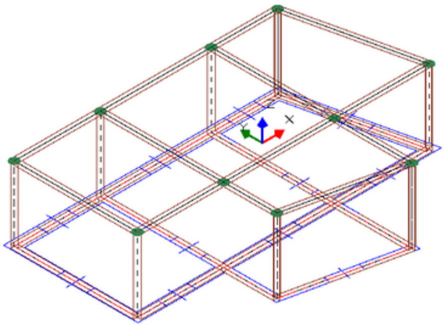
For this group of elements that have an extension in elevation along the level's height, during the insertion phase is possible to perform the input directly in 3D mode using the snaps to the walls tracking reported to the lower elevation of the level.

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



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

Floor, Vault



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

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

9.3.3 Edit objects

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

9.3.3.1 Basic editing

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

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

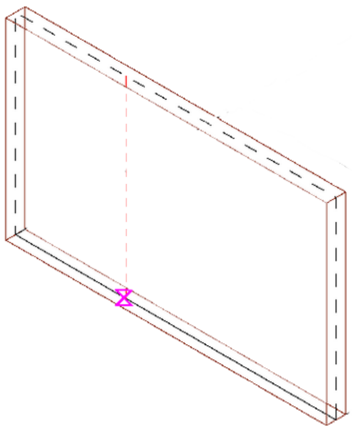
Delete: Delete element

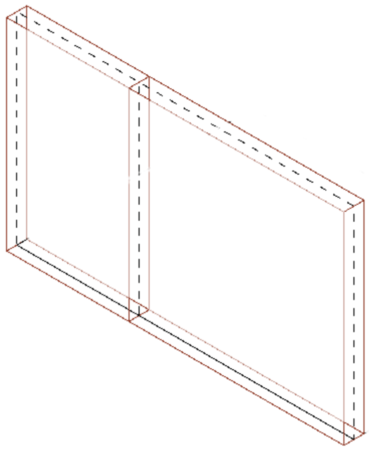
Edit: Shows the element properties window

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

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

Guidelines insertion *Divided element*





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

Floor, Vault, Column

Delete: Delete element

Edit: Shows the element properties window

9.3.3.2 Advanced editing

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

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

Align walls

Assign exposed to wind

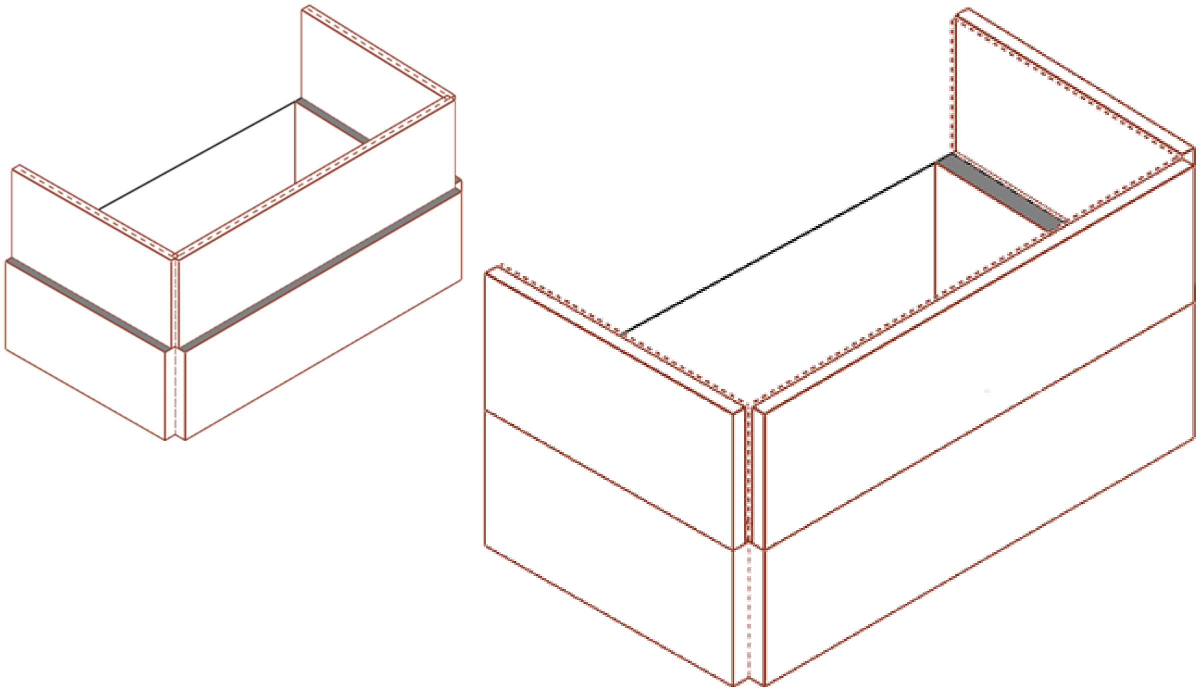
Edit table

Define user's part

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

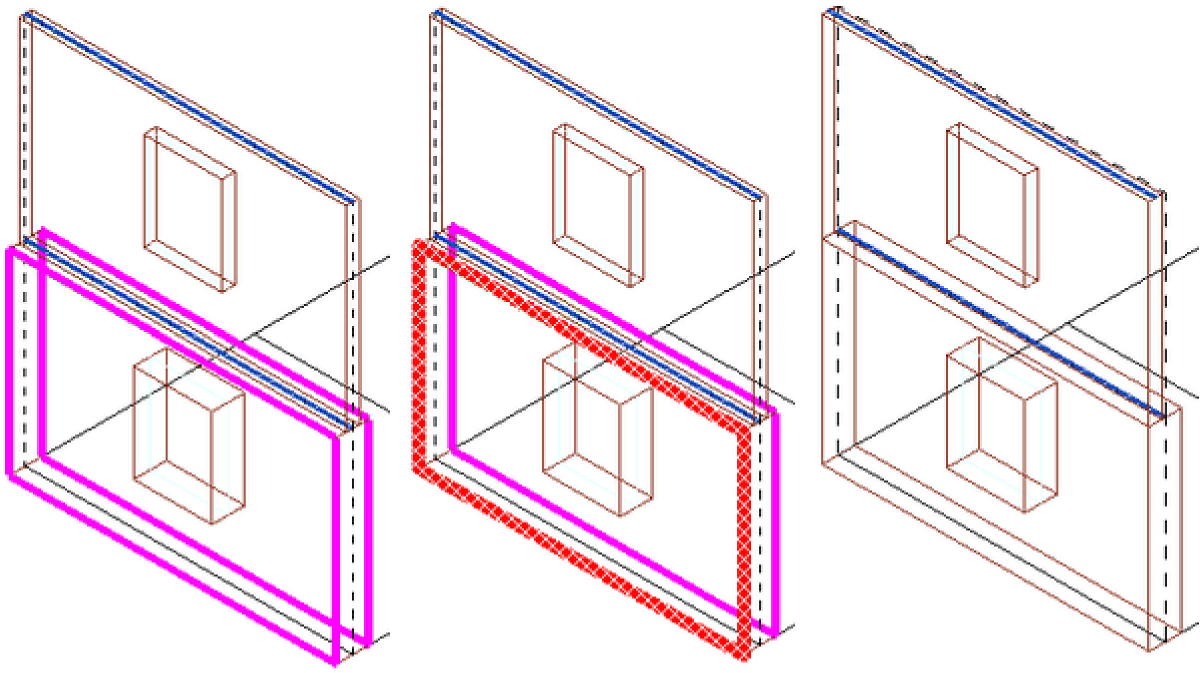
[Before editing]
Walls aligned with the lower middle plan

[After editing]
Walls aligned with the outer edge



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

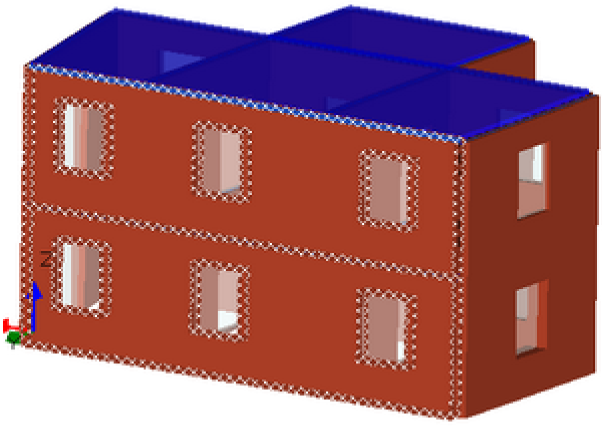
(1)-Show the reference element to perform the alignment (2)-Show the alignment face (3)-Final result



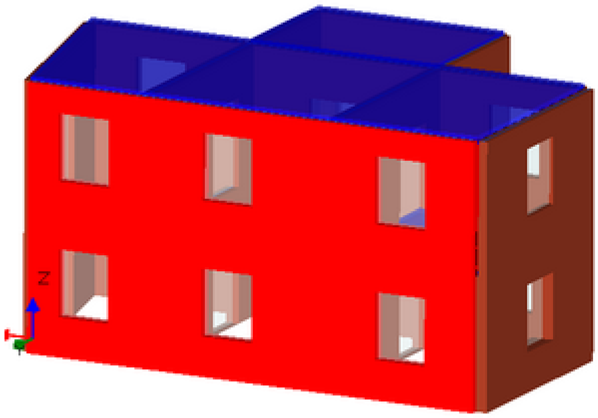
Assign wind exposed

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

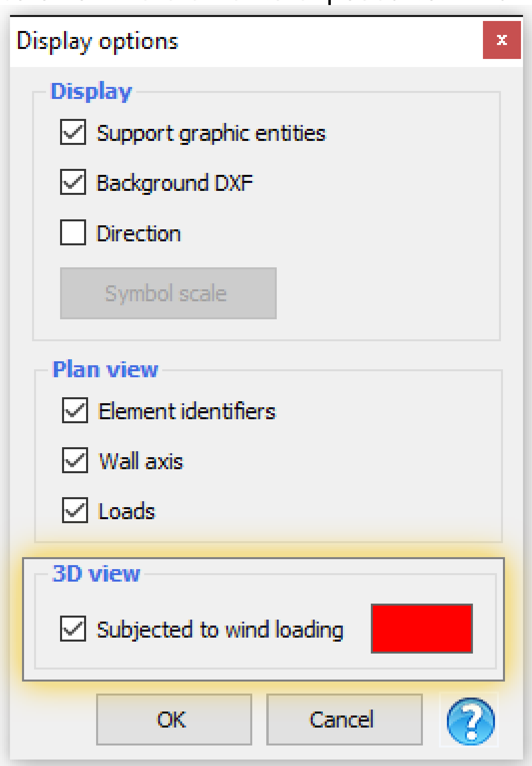
Selected walls



The color is updated



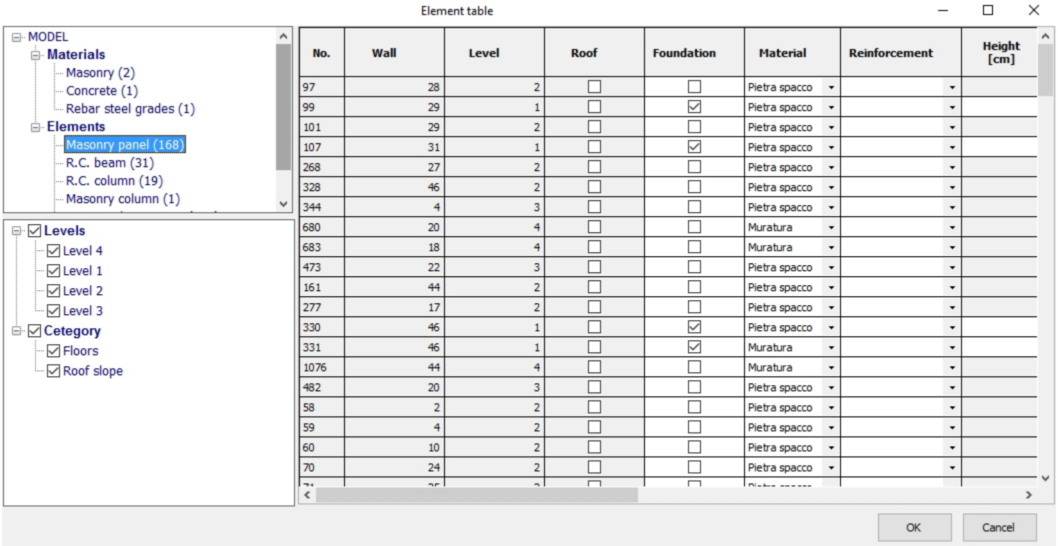
The color of the elements exposed to wind is defined from the mask of the display filters.



Edit table

After a partial selection of the structure you can access the element table that displays only the selected elements.

Possible changes in the table will therefore affect only the selected elements.

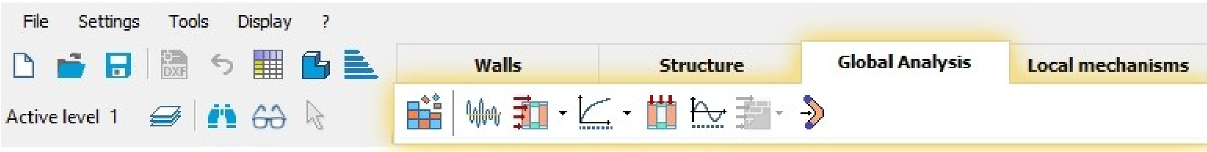


Define user's part

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

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

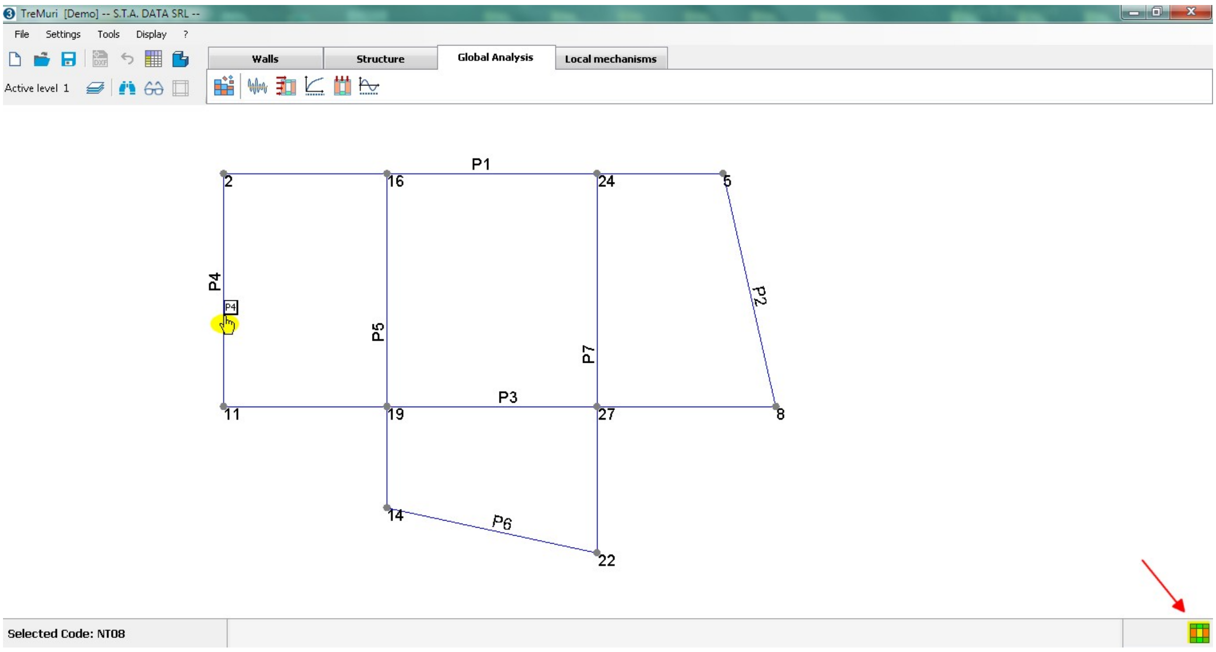
10 Analysis



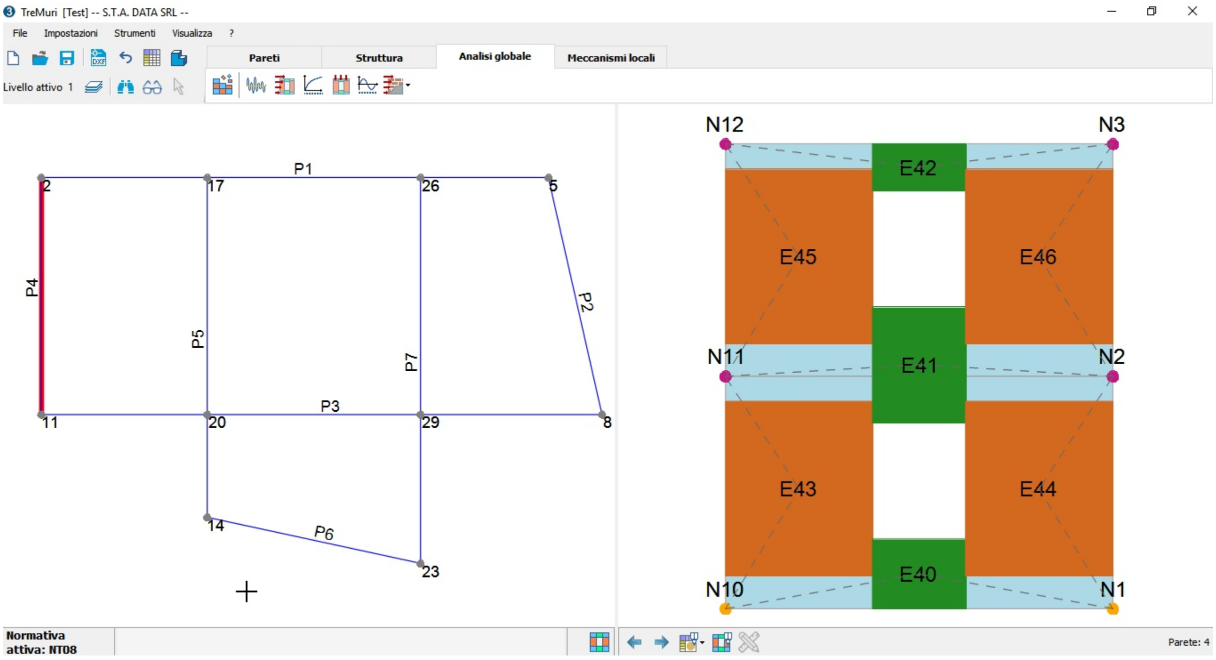
	Compute model mesh Enter value
	Seismic Action Enter value
	Pushover Calculation Enter value
	Display pushover results Enter value
	Global static verification Enter value
	Modal analysis Enter value
	Sensitivity analysis Enter value
	Bending out of plan

10.1 Mesh definition

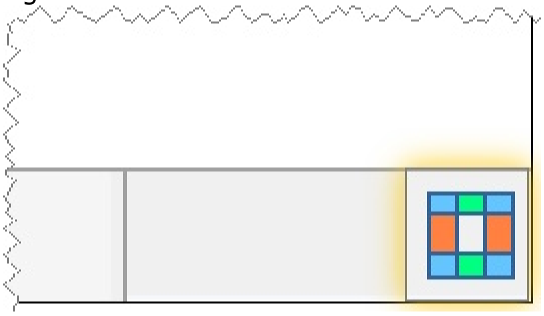
From the graphics of the analysis area it's possible to click on a wall in order to display it in the mesh front.



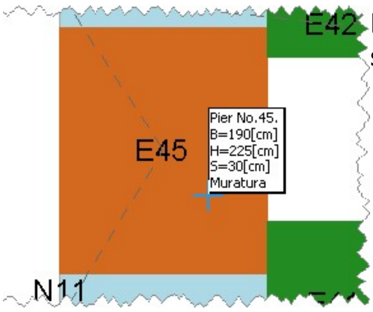
The graphics area is separated in two different areas, the plan on the left and on the right the front of the mesh of the selected wall.



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



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



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

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

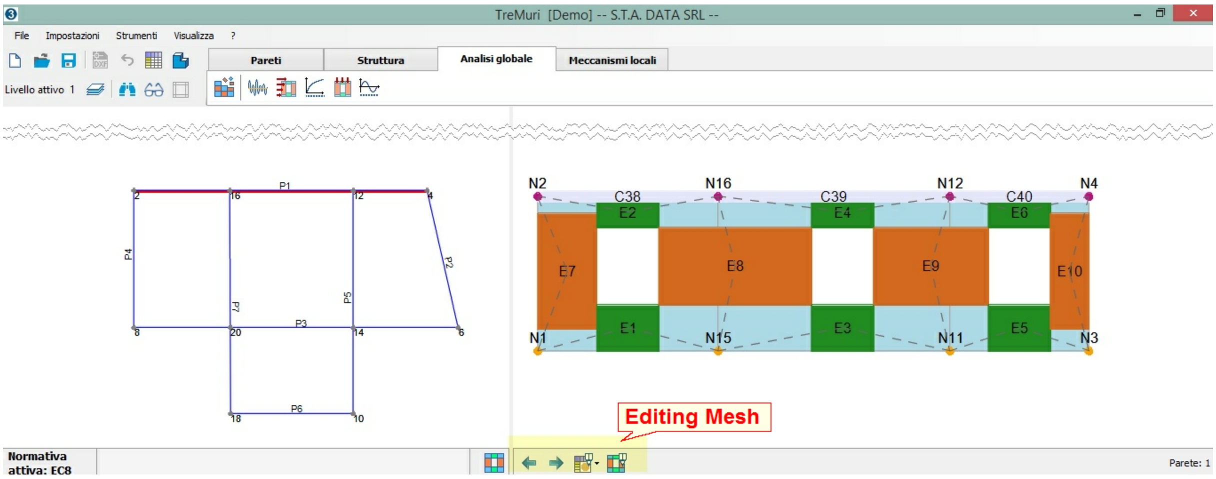
	Previous wall
	Next wall
	Edit Material: If the designer has to provide some adaptation on the structure he/she may act through localized interventions improving the mechanical properties of the individual wall elements. (Details...)
	Edit Mesh: The procedure for automatic mesh generation (calculates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

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

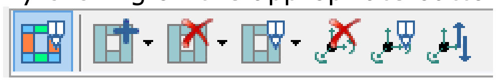
10.2 Mesh editing

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

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



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



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

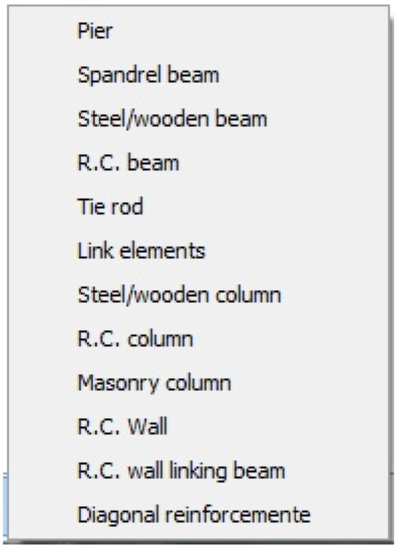
All edit functions, except "Change node type", have "Cancel" command.
While changing wall or closing the mesh editing environment, you will be asked to confirm the changes. If the answer is affirmative, the mesh wall is changed according to the instructions with an automatic recalculation of the rigid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

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

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

10.2.1 Editing Elements

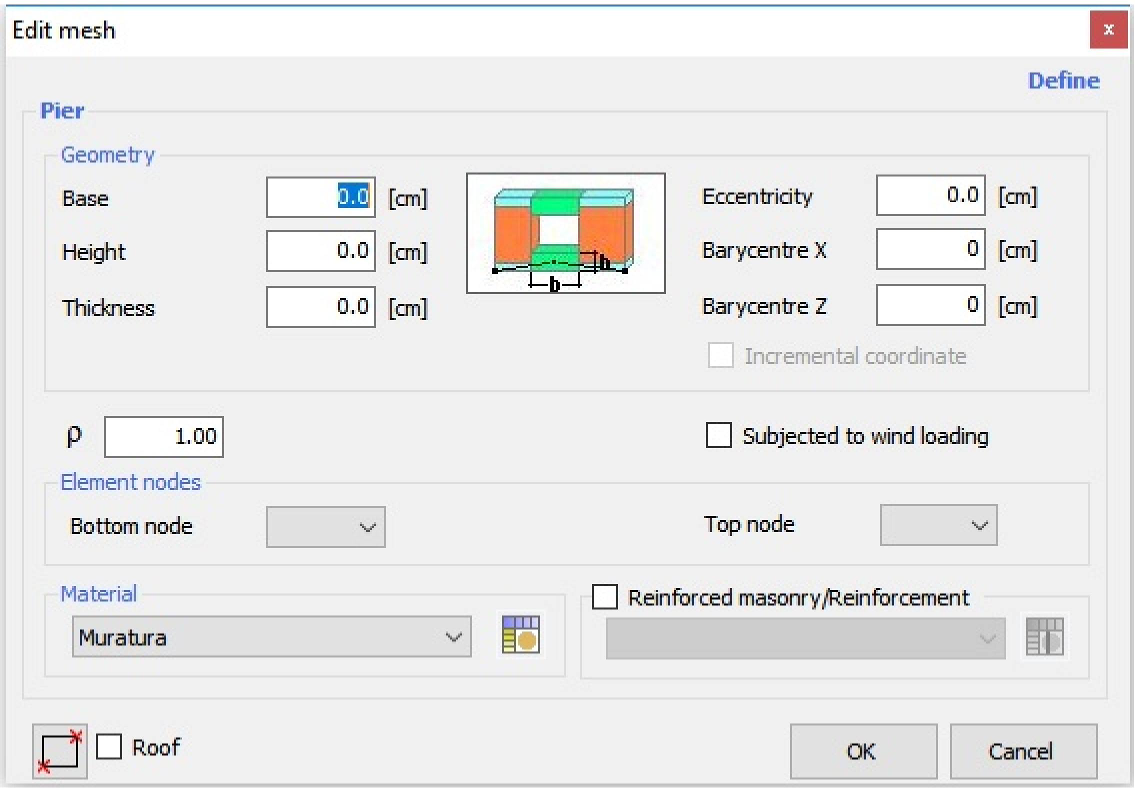
The editing functions act on the entities in the associated menu on each button of the command bar.



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

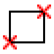
 Add element:

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

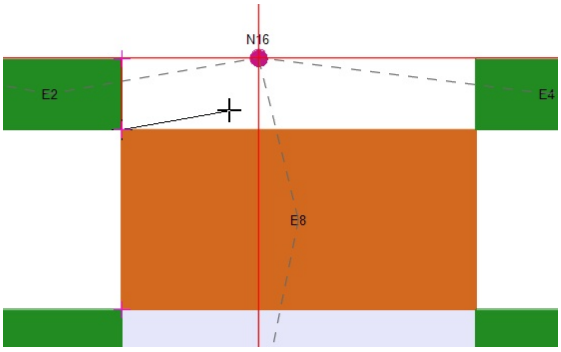


Element nodes: Allow user to modify element nodes of selected item; in this way the user can modify equivalent frame geometry.
Move barycenter: allow the use to move selected macroelement by insert component for translate vector (in local wall system: x, z).
Reduction factor for slenderness(ρ):factor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

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

: This command allows to insert a pier/spandrel beam through the graphic input.
- Insert the encumbrance rectangle through the two opposite vertices of the rectangle
- click on the first element incidence node
- click on the second node of incidence of the element
As soon as the encumbrance of the pier and its incidences has been defined, the input mask with the preloaded data is presented again, simply just define the thickness and the wall material before confirming the insertion.

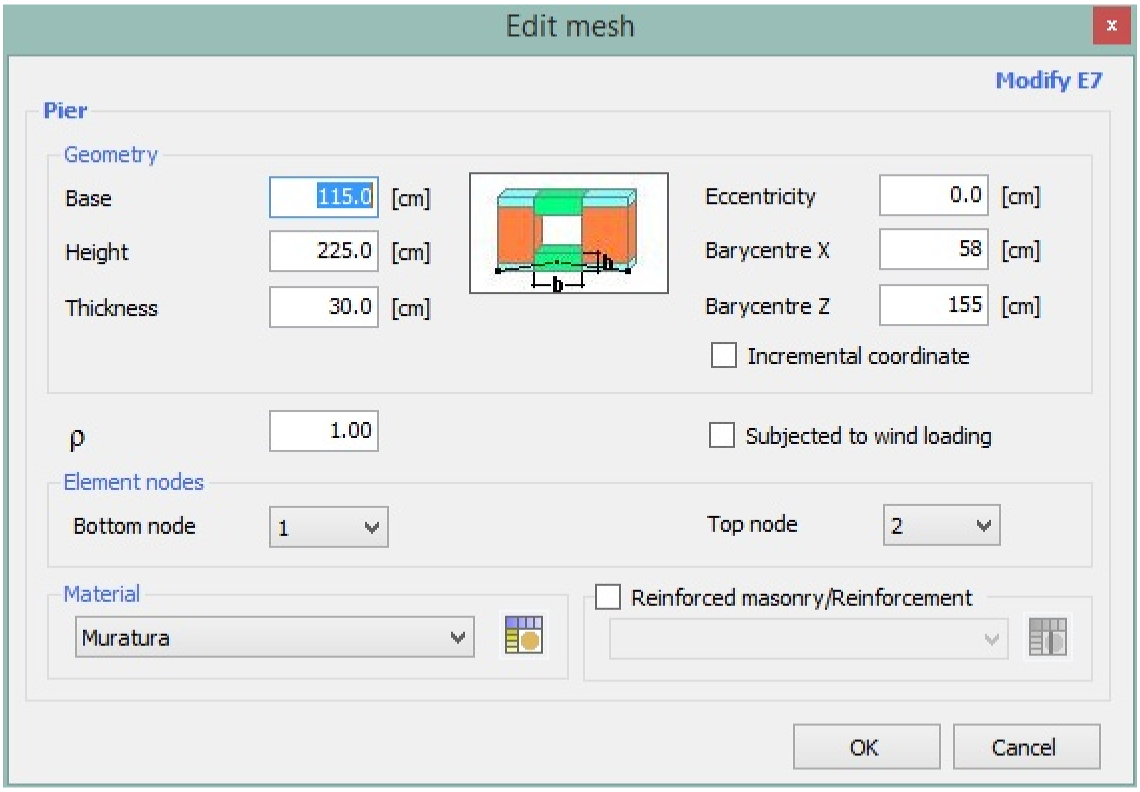
Addition of the Rigid Node:
A first click allows the user to select the node to which the rigid polygon is linked, immediately the pair of axes that divides the node's around into 4 quadrants is defined.
Subsequent clicks are used to define the closed polygon that delimits a quadrant of the rigid node.
The polygons of the node must be inserted for quadrants because each quadrant could have different thickness and material.



 Delete item: Select one or more items to delete.

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

Suppose for example the case of a pier:



The case of a multiple selection:

✓ Only new data inserted in input boxes are modified. To keep the original data of selected items, leave the input boxes blank. For example, to move up 10 cm all selected piers, it is necessary:
  - checking the box "Incremental coordinate"
  - insert in "Delta Z" input box the value 10
  - click on "ok" button.

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

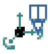
✓It is impossible to modify elements nodes

10.2.2 Editing Nodes

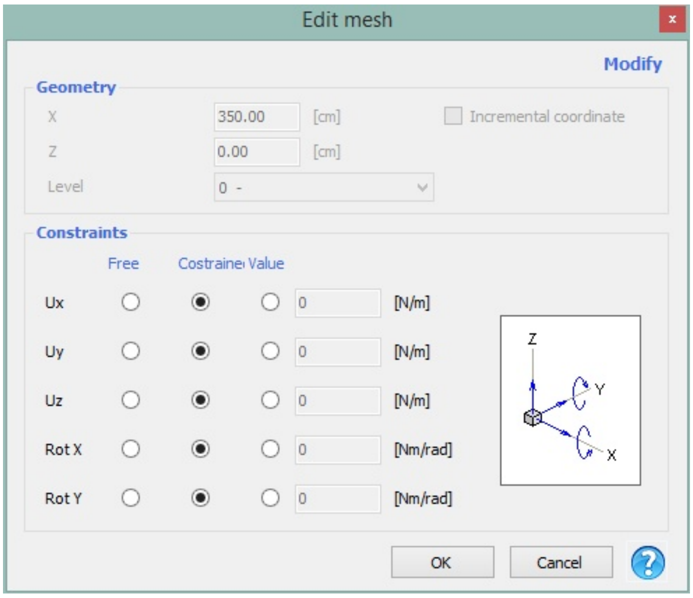
Editing nodes functions are available for 2D node as well as 3D.

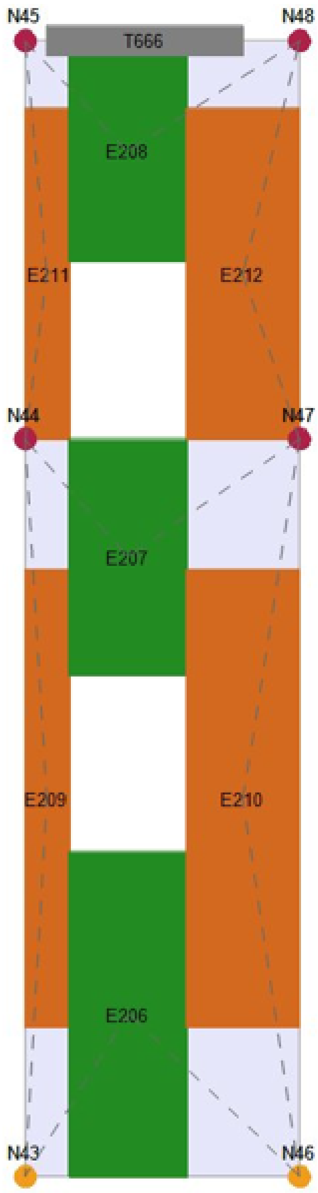
 Delete nodes:

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

 Modify node:

It is possible to select one or more nodes to be modify. A form that includes all the data editable is loaded. In case of individual selection, the form contains all data node; in case of multiple selection, the form does not report any data of selected nodes.


Edit mesh
Modify
Geometry
X
350.00
[cm]
Incremental coordinate
Z
0.00
[cm]
Level
0 -
Constraints
Free
Costraine Value
Ux
[N/m]
Uy
[N/m]
Uz
[N/m]
Rot X
[Nm/rad]
Rot Y
[Nm/rad]
Z
Y
X
OK
Cancel



Inside the mesh elevation, the nodes take on a different color depending on the degree of constraints they possess:

Yellow: node with restricted displacements and rotations
Red: node with free movements and rotations

From this window you can define the boundary conditions.

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

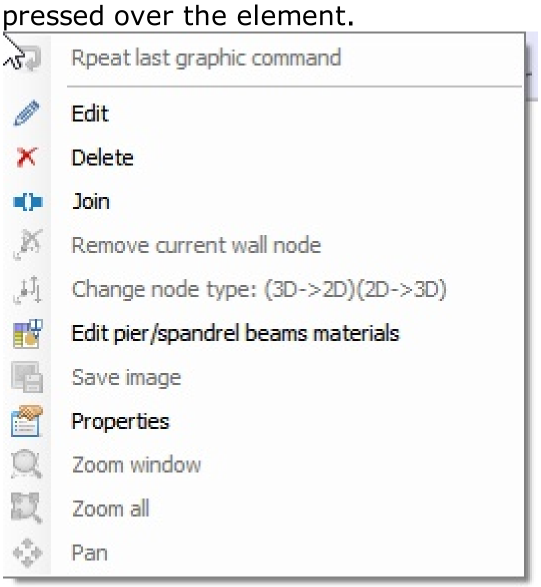
Passage 3D -> 2D
The node that is a 3D type node before the modification is transformed into a 2D node (Nodal derating).
It is checked if the selected node belongs to more than one wall.
✓ If the node belongs to a *single wall* the type that it gets derated (this functions is prevented only if the node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

Passage 2D -> 3D
The node that is a 2D type node before the modification is transformed into a 3D node (Nodal reclassifications).
In this phase the program prompts the user if it exists any incidental wall. The program checks if the input wall intersect the current wall and if that happens, it will search for the incidental wall a node that geometrically coincides with the node that has to be reclassificated; even this node will be transformed in 3D node type.

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

10.2.3 Element editing

These commands are available in the context menu that appears when the right button is pressed over the element.



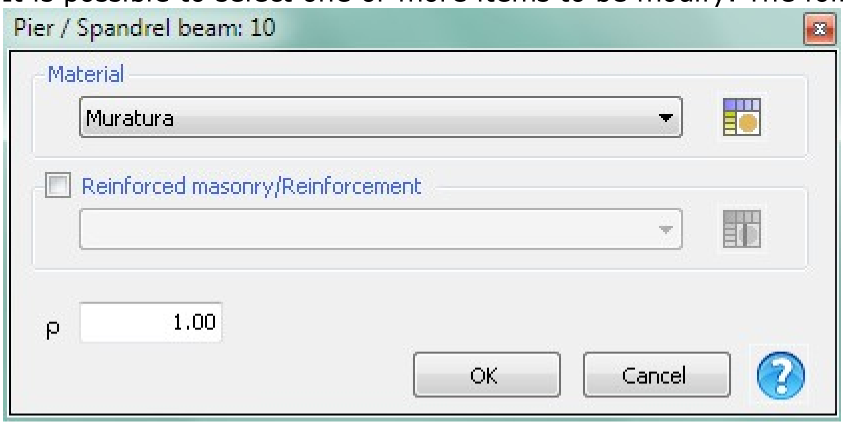
Repeat last graphic insertion	Repeat the last insertion command executed
Edit	Edit the properties of an element
Delete	Delete an element
Join	It allows you to merge the selected pier with one on a higher level creating a unique one that involves two levels. This command makes the procedure for creating double volumes much easier.
Delete node for current wall	In the case of a 3D node, it provides the cancellation from the node and the creation of another 2d node for the central wall in order to close the frame.
Change node type (3D->2D)(2D->3D)	Change the node category from 3D to 2D and vice versa

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:

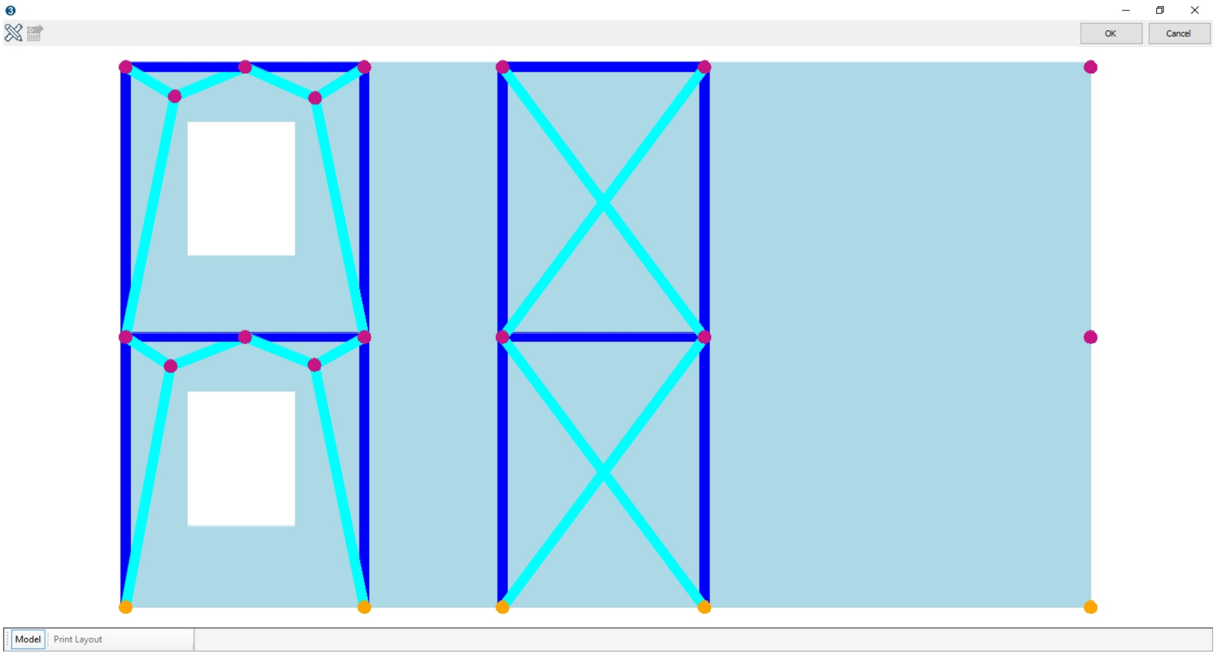


In case of single selection, the form contains all data item; in case of multiple selection, the form does not report any data of selected items. If you want to modify only one of two items of the form, simply leave unchanged the other one.

10.3.1 Steel elements graphic layout

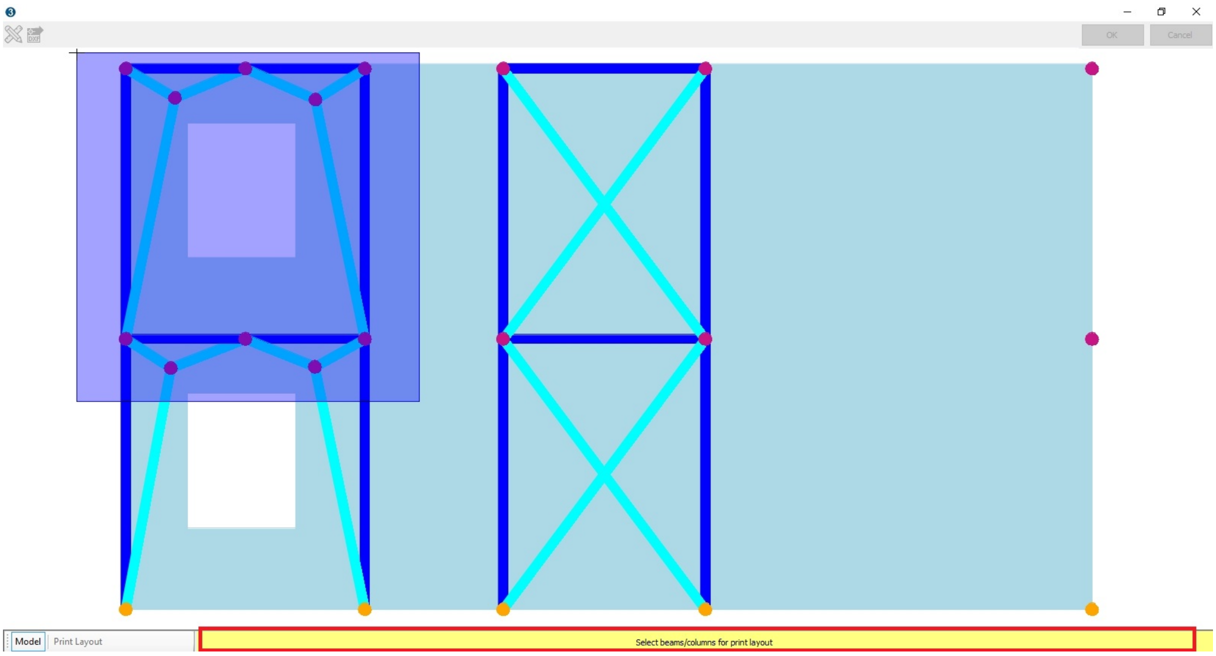
Pressing the appropriate command in the bottom right of the mesh command bar, you access to the dedicated layout environment.

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

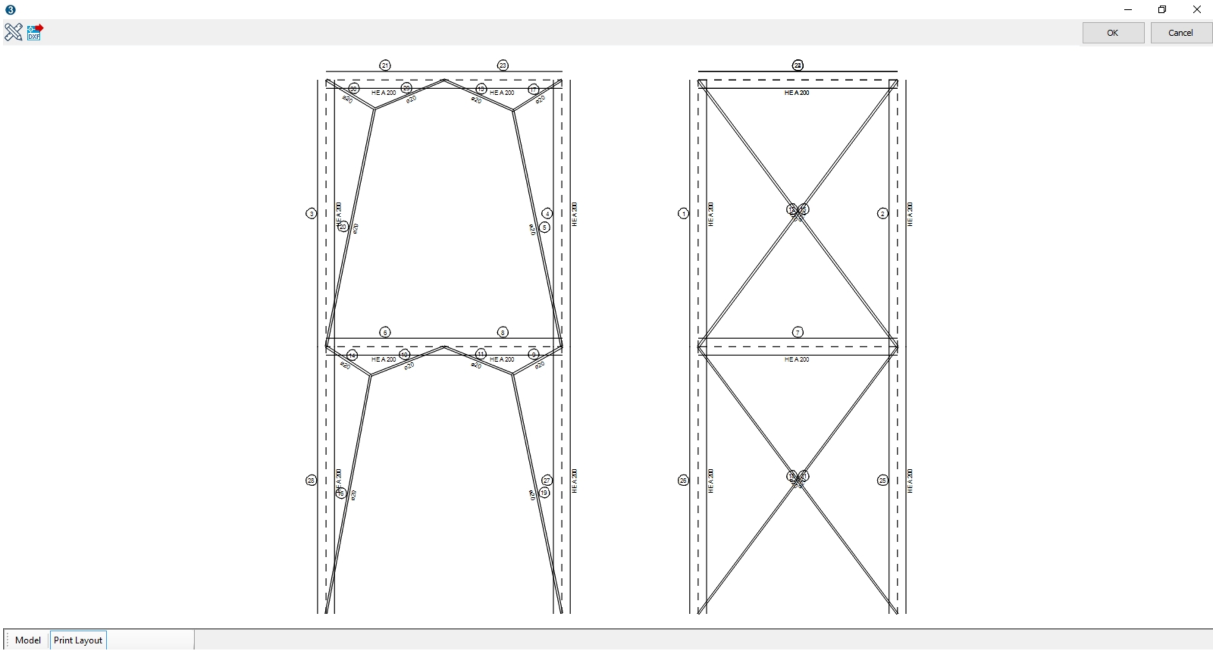


 Create graphics table:

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



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



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

Calculation of the tie rod-plate link

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

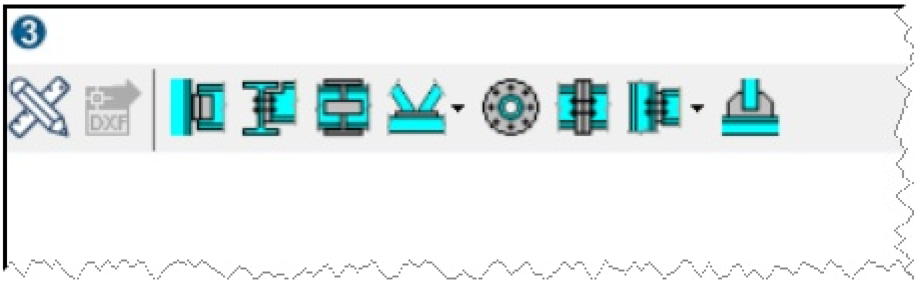
Selecting this command is required to indicate the nodes to be calculated.
The design/verification of the tie rod-plate connections are carried out through the appropriate command in the Utility menu that combines the calculation of all the tie rods, both those from a global model and from local mechanisms.

Calculation of Steel Elements [Steel Connection Module]

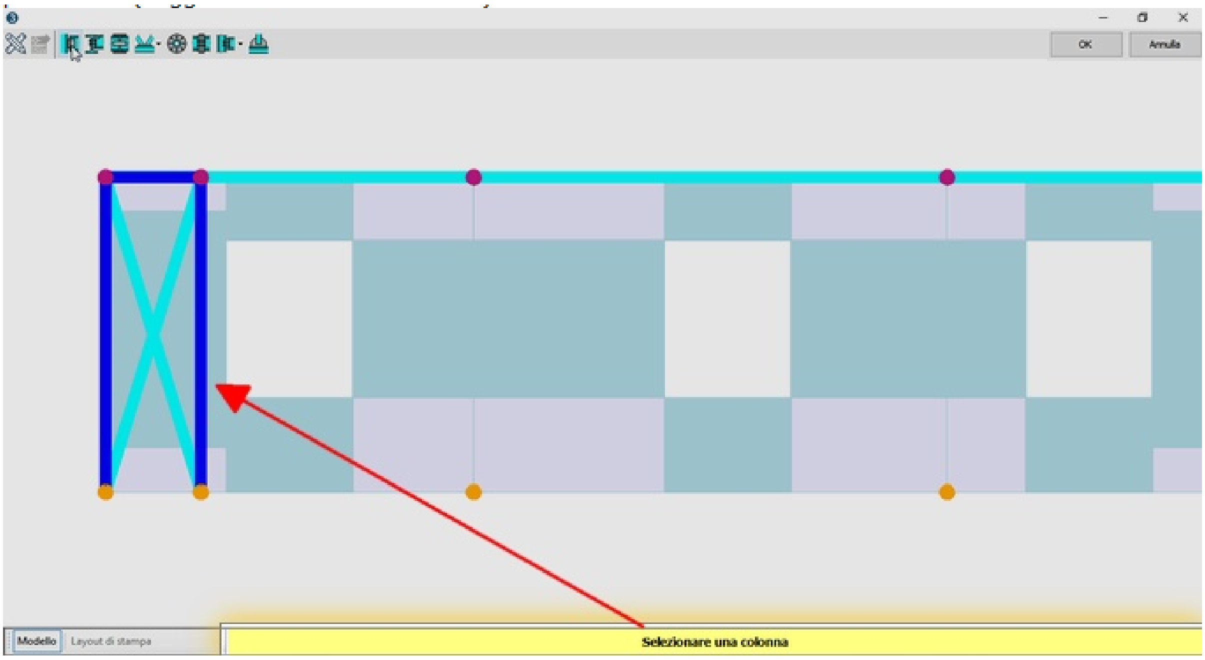
[OPTIONAL]

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

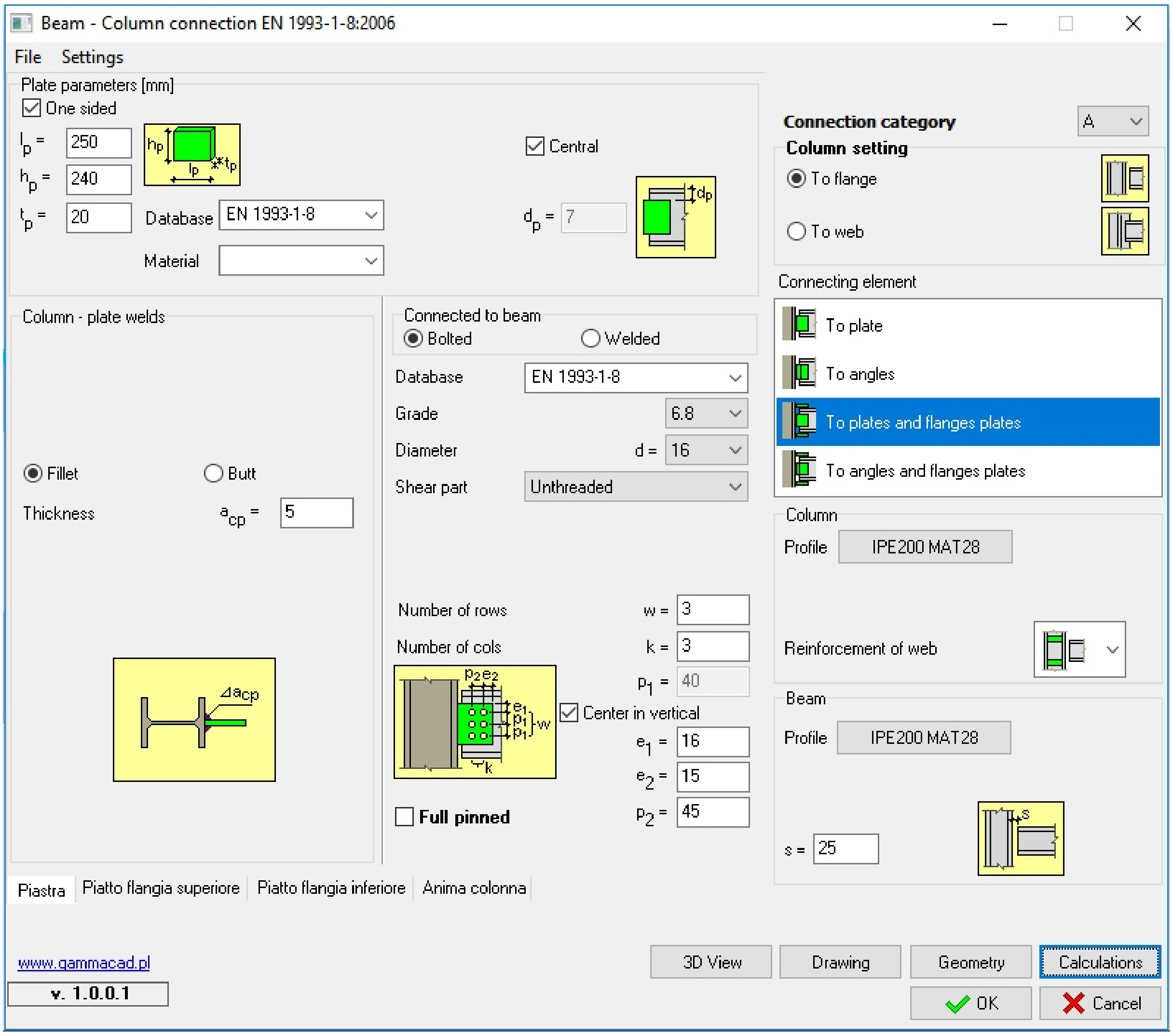
This module is dedicated to *dimensioning and verification of steel connections* based on the Eurocode 3 standard (EN 1993-1-8).



Use the command bar to select the type of joint you want to check.
After selection, it is necessary to individually identify the members that converge on the node of interest in the pre-established order (suggested in the lower bar).



The following image shows one of the SteelConnection calculation modules

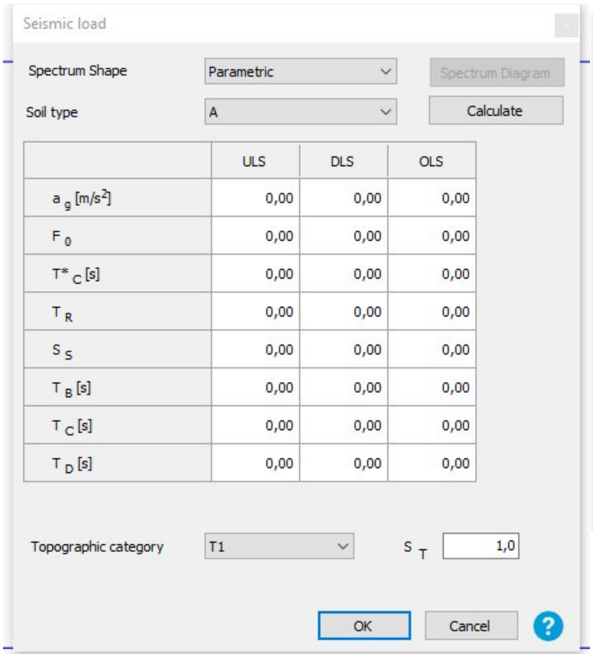


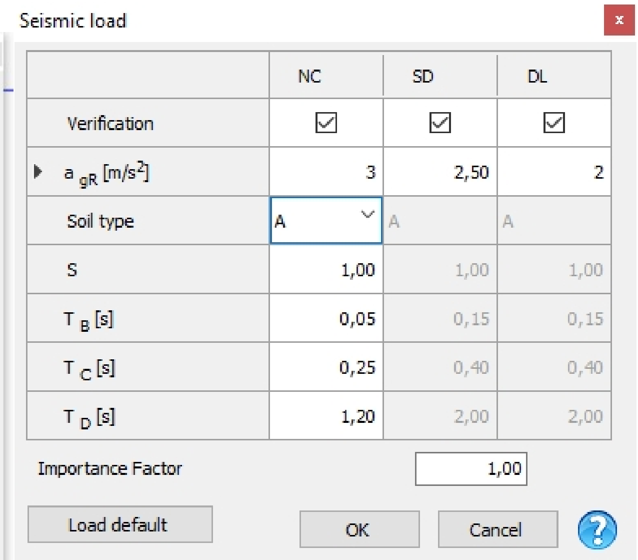
10.4 Seismic Analysis

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.

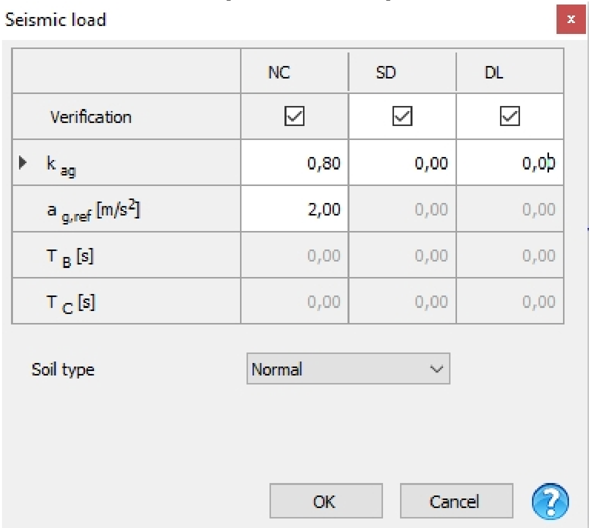
NTC08 **Eurocode**

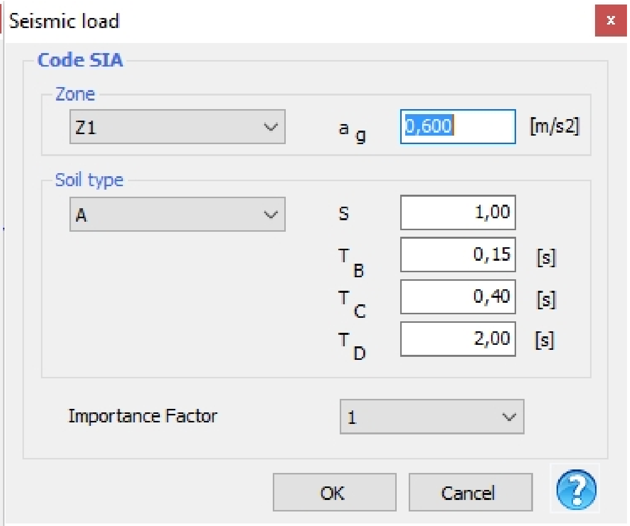




Eurocode NL (Netherlands)

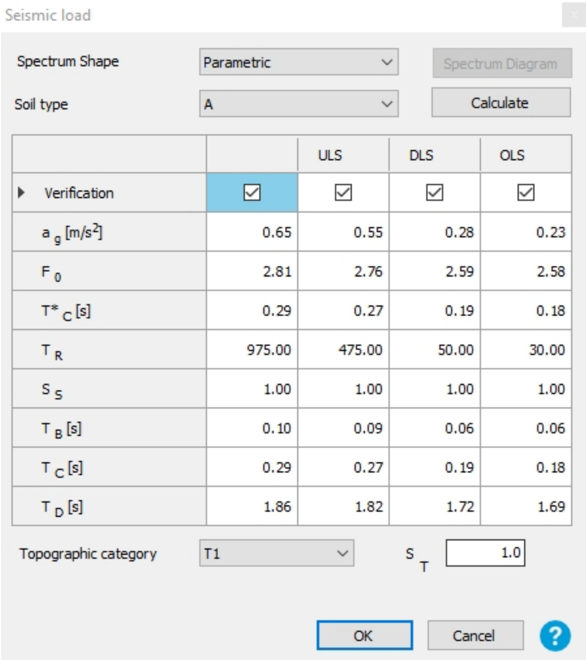
SIA 269/8

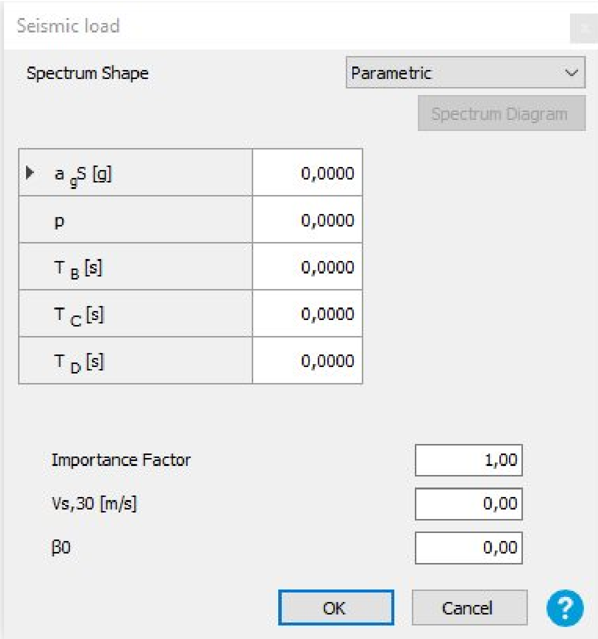




NTC18

NPR 9998-2018

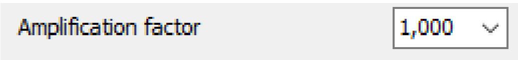




NTC18

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

The amplification factor allows to increase the seismic spectrum by the indicated factor.



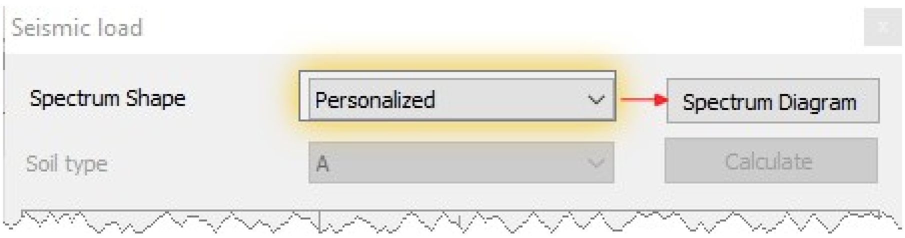
This function can be used to examine the bidirectional behavior of the structure.

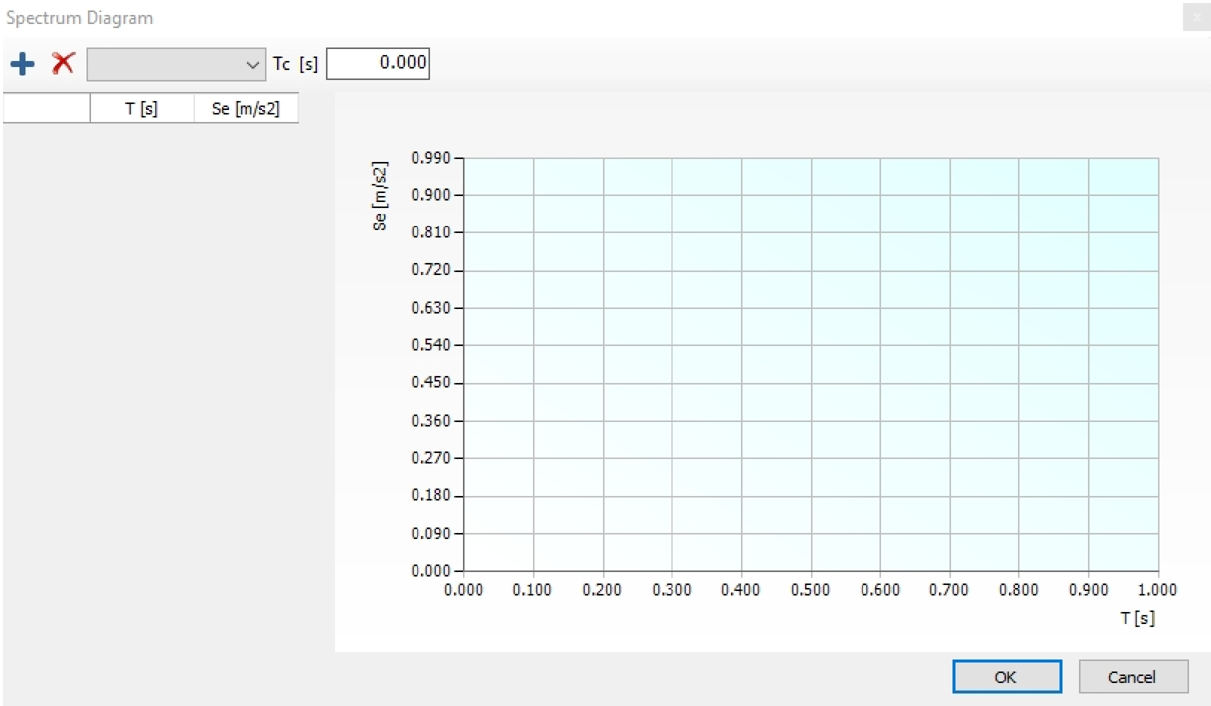
Eurocode

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

Definition of the spectrum by points

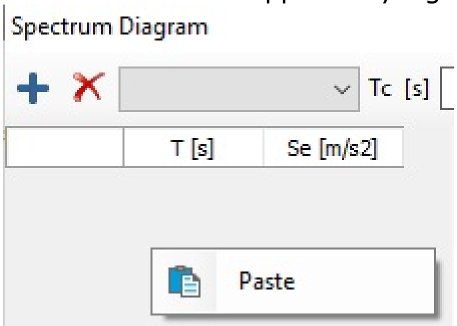
Selecting "Personalized" Spectrum Shape it activates the [Spectrum Diagram] command

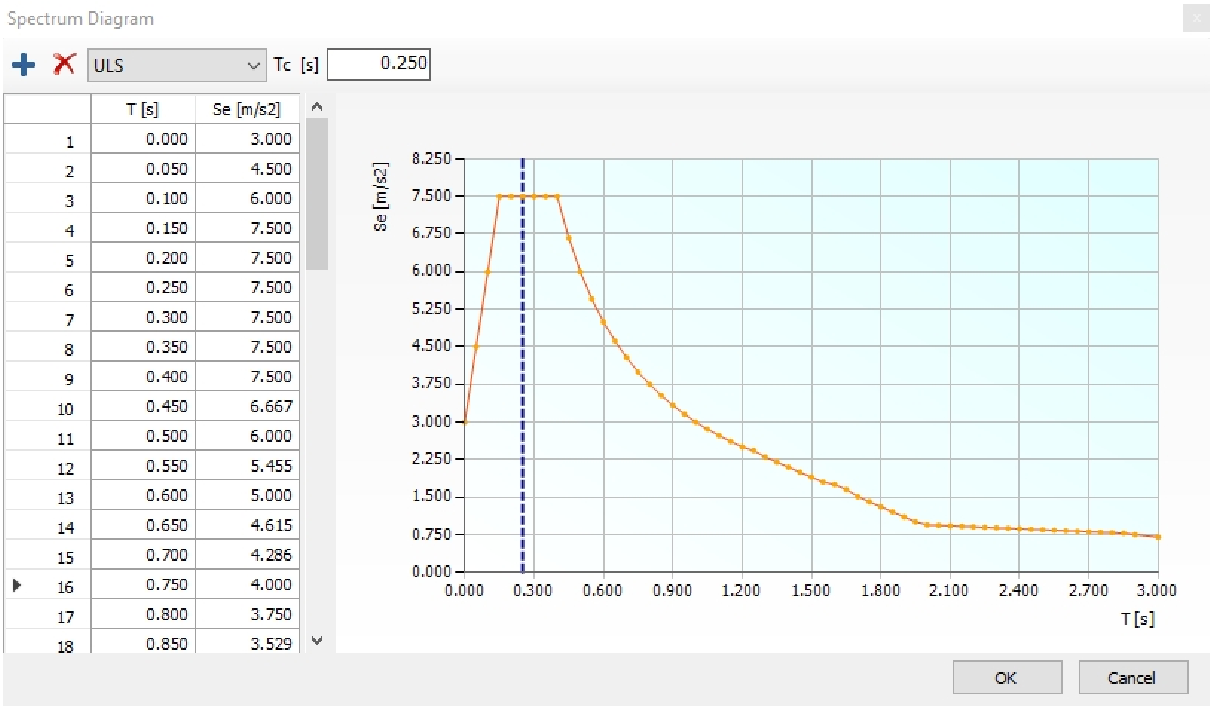




There are two different ways to impute the shape of the curve:

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



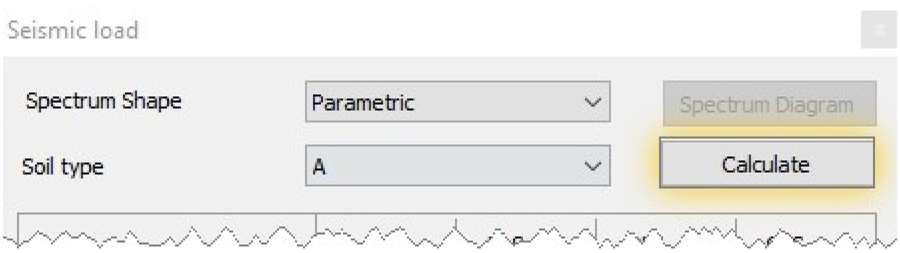


The value of Tc (corresponding period at the beginning of the constant speed section) must be manually defined by the user using the text field above or by clicking directly on a point in the diagram.
The drop-down window in the bar contains the list of limit states, select each limit state to define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

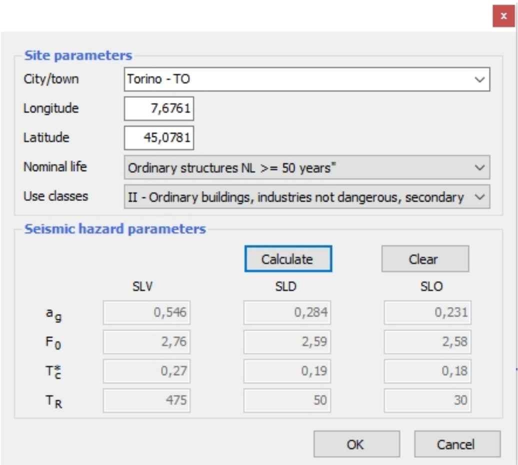
If the point definition is used, the results window will display the vulnerability index as the multiplying factor of the spectral form defined herein, such as to identify the ultimate condition of the required limit state.

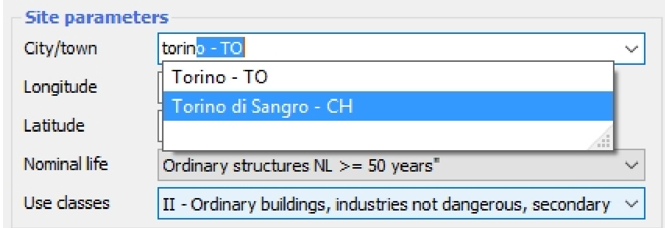
NTC08; NTC18

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



Choosing this button, the following window is shown:

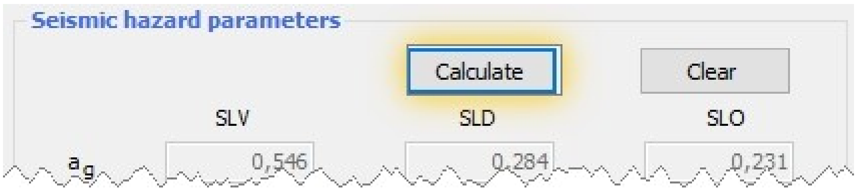




You can select directly the municipality using the internal database or insert the latitude and longitude of the site.

World Geodetic System 84 (WGS 84)

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



10.4.2 Not Linear Analysis - Pushover

10.4.2.1 Computation Settings

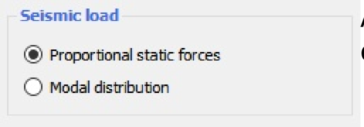
To access to the global analysis of the structure, select "Global Calculation" from the drop-down list.
In this phase, the computation is performed using the selected code.

Many of the computation parameters defined in the "Settings" window are already set so as to work with most examinable structures. Others are automatically computed by the program based on the geometry of the model. The earthquake direction to be considered and the choice of the control node are chosen by the designer based on the indications found in the code.

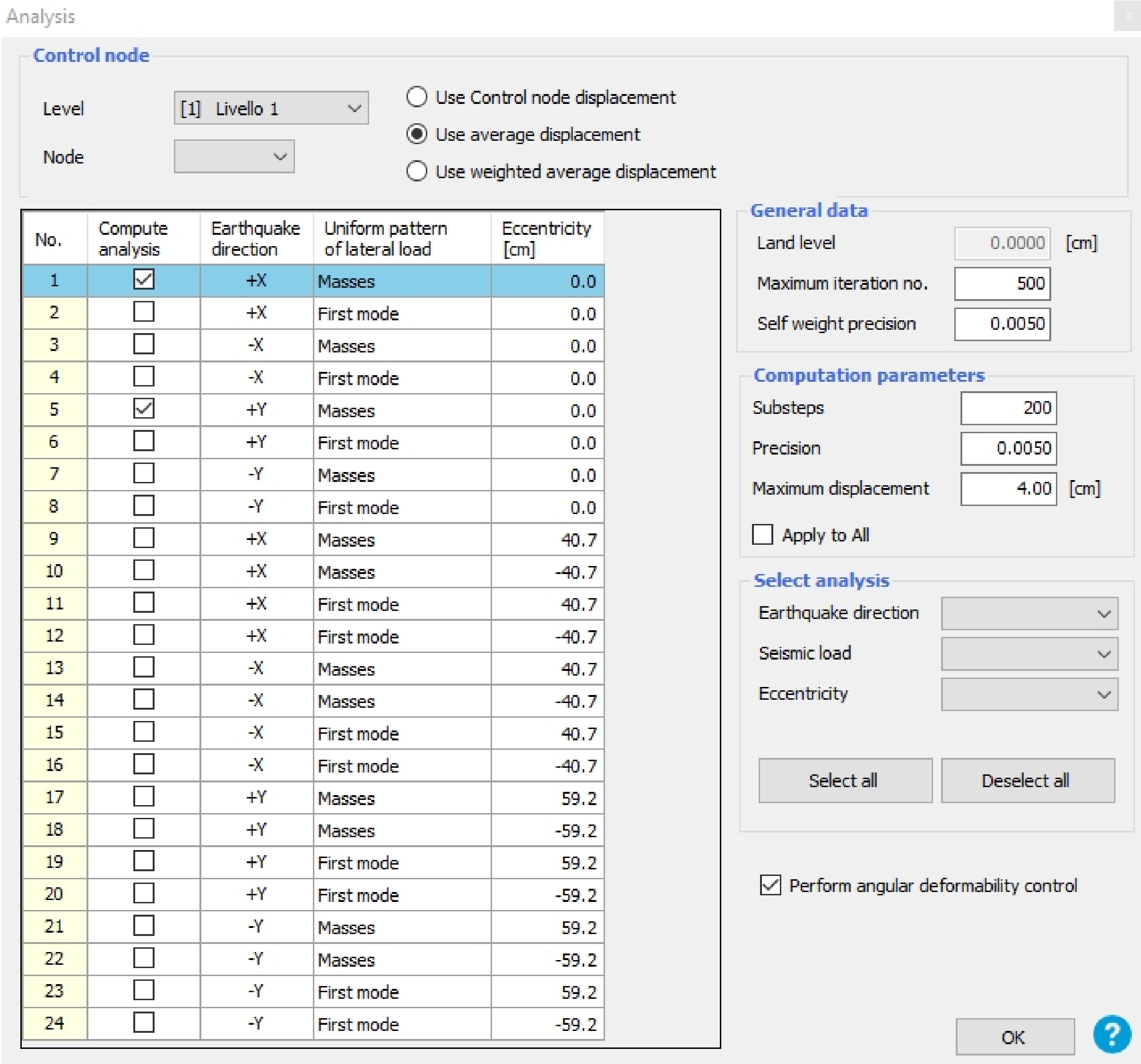
The bearing capacity curve can be drawn monitoring displacement, in place of the control node of the average of the project, by selecting the appropriate text box.

The choice of the seismic force distributions is up to the designer, the available options are:

- Uniform: distribution of forces, deduced from a uniform trend of accelerations along the height of the construction;
- Static forces: proportional distribution to static forces $F_i = F_h \cdot z_i \cdot W_i / \sum z_j \cdot W_j$
- Modal distribution: this distribution is an alternative to "Static forces" and is calculated on the basis of the identified significant modes following the calculation of modal forms.

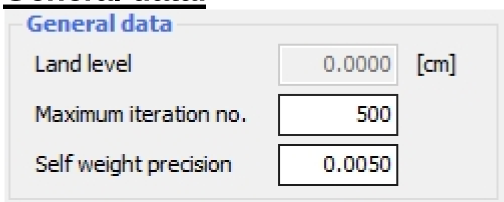


At the bottom right side,a panel allows to select which distribution to use between static and modal forces.

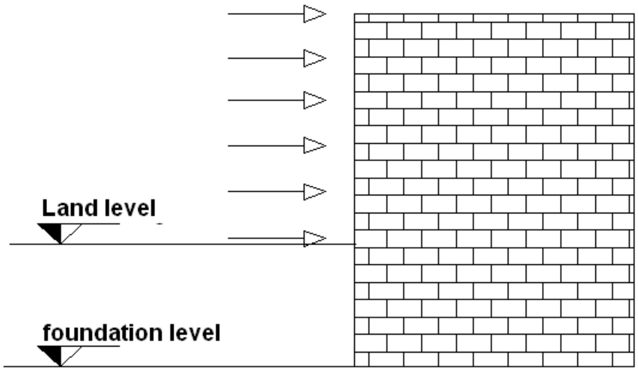


No.	Compute analysis	Earthquake direction	Uniform pattern of lateral load	Eccentricity [cm]
1	☑	+X	Masses	0.0
2	☐	+X	First mode	0.0
3	☐	-X	Masses	0.0
4	☐	-X	First mode	0.0
5	☑	+Y	Masses	0.0
6	☐	+Y	First mode	0.0
7	☐	-Y	Masses	0.0
8	☐	-Y	First mode	0.0
9	☐	+X	Masses	40.7
10	☐	+X	Masses	-40.7
11	☐	+X	First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

General data:



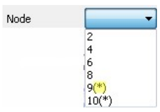
Land level: represents the elevation of the land level. The program assigns the lowest point of the structure elevation 0. The possibility of inserting this elevation allows the user to define the point where the seismic load initiates. The value of this elevation must be between the foundation elevation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



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

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

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



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

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

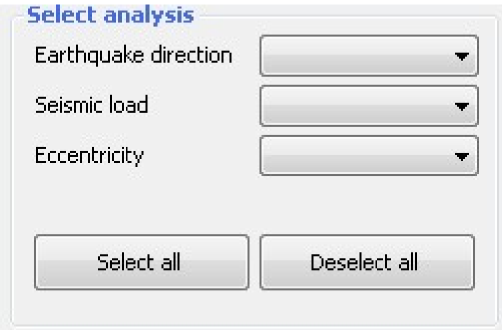
Direction: indicates the earthquake direction.

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

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

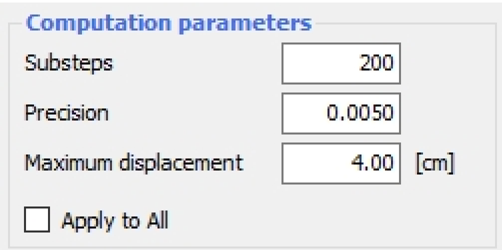
Eccentricity: Accidental eccentricity of the center of mass with respect to the rigidity center computed automatically according to the code.

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



Enables the calculation of all the currently disabled analyzes
Disables the calculation of all the currently active analyzes

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



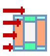
Substeps: represents the number of displacement steps computed by the solver for the seismic load pattern.

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

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

Apply to All: With the active tick the values are applied to all analyzes; if the check is disabled they are only applied to the selected analysis.

10.4.2.1.1 Pushover analysis

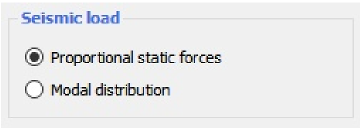
 To access to the global analysis of the structure, select "Global Calculation" from the drop-down list.
In this phase, the computation is performed using the selected code.

Many of the computation parameters defined in the "Settings" window are already set so as to work with most examinable structures. Others are automatically computed by the program based on the geometry of the model. The earthquake direction to be considered and the choice of the control node are chosen by the designer based on the indications found in the code.

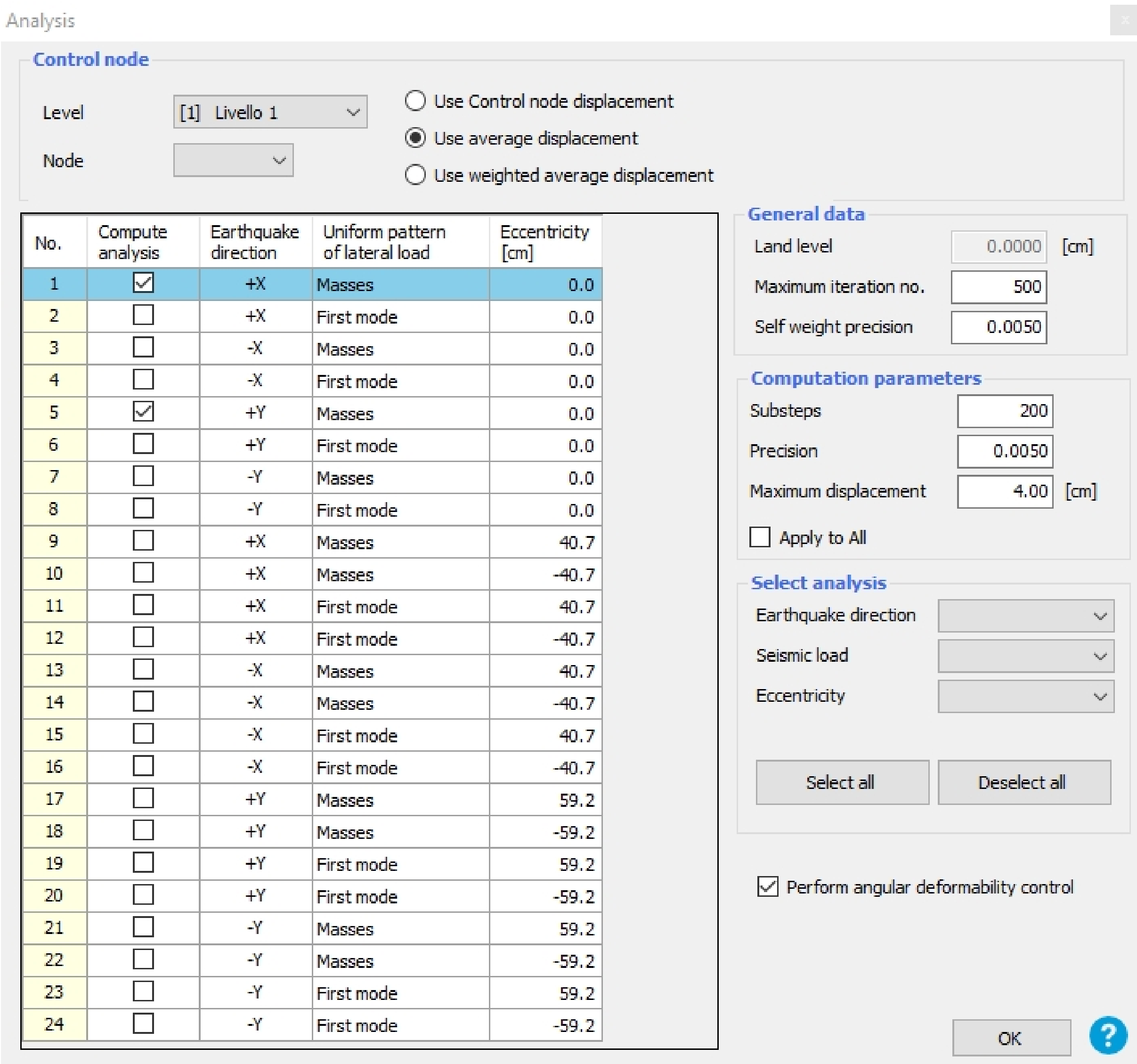
The bearing capacity curve can be drawn monitoring displacement, in place of the control node of the average of the project, by selecting the appropriate text box.

The choice of the seismic force distributions is up to the designer, the available options are:

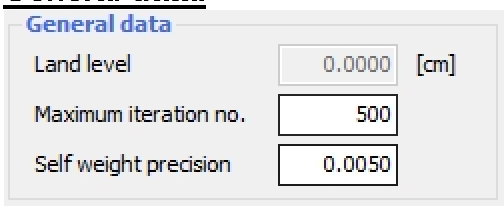
- Uniform: distribution of forces, deduced from a uniform trend of accelerations along the height of the construction;
- Static forces: proportional distribution to static forces $F_i = F_h \cdot z_i \cdot W_i / \sum z_j \cdot W_j$
- Modal distribution: this distribution is an alternative to "Static forces" and is calculated on the basis of the identified significant modes following the calculation of modal forms.



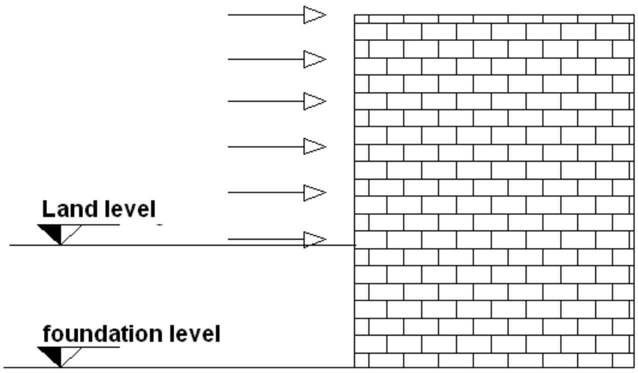
At the bottom right side,a panel allows to select which distribution to use between static and modal forces.



General data:



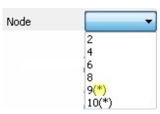
Land level: represents the elevation of the land level. The program assigns the lowest point of the structure elevation 0. The possibility of inserting this elevation allows the user to define the point where the seismic load initiates. The value of this elevation must be between the foundation elevation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



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

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

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



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

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

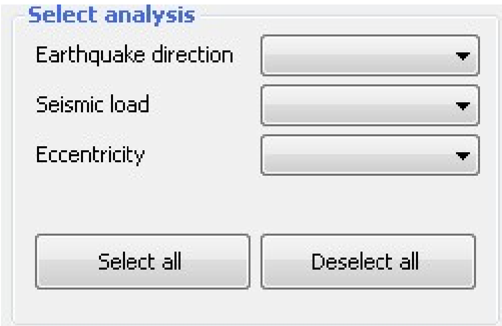
Direction: indicates the earthquake direction.

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

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

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

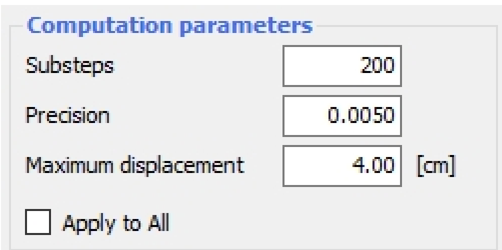
Using the associated space, multiple analysis can be performing by activating the selection filters.



Enables the calculation of all the currently disabled analyzes

Disables the calculation of all the currently active analyzes

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



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

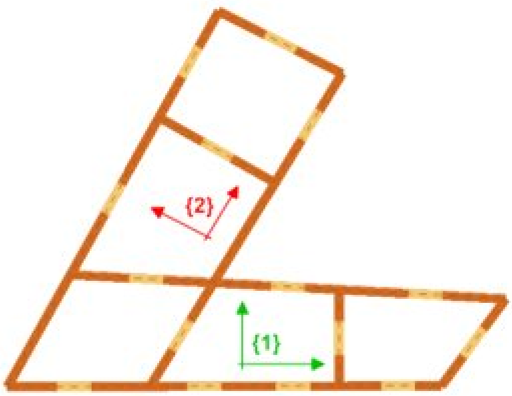
Tolerance: represents the degree of tolerance reached by the non-linear computation.

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

Apply to All: With the active tick the values are applied to all analyzes; if the check is disabled they are only applied to the selected analysis.

10.4.2.1.1.1 Analysis with seismic angle

Generally the pushover analysis is performed according to the two main directions X-Y Some planimetric configurations could however give rise to the doubt that these directions are not actually the most significant.



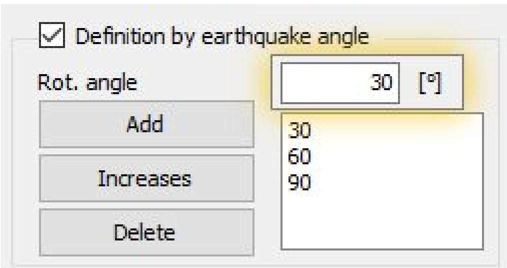
We can take for example the planimetric layout represented by the image on the left.
It is represented by two main bodies:

- Body-1: represented quite well by the system {1} as formed by walls that are approximately parallel to the system itself.

- Body-2: being inclined compared to the Body-1, it is better represented by an inclined system {2}

Performing a pushover analysis relative to the main axis direction, we obtain representative results of the real behavior of the building part that is represented by the reference system {1} but not as representative of the building part that is represented by the reference system {2}.
This last part could in fact be better represented by the results of a pushover analysis performed according to the two directions indicated by the reference system {2}.

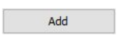
This is possible by using the calculation mode that allows to perform pushovers in different directions simultaneously, simply by defining the angle in input.



An angle analysis can degenerate into a classical analysis by imposing specific angle values (as shown in the following table).

Pushover analysis	Pushover analysis with angle
+ X	0°
- X	180°
+ Y	90°
- Y	270°

After defining the angle, the user can use the "add", "increase" and "delete" buttons.

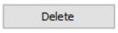


By clicking the add button, for the corresponding value of the chosen angle, it is possible to insert in the list the angle whose analysis is to be conducted.
At each angle there are 6 analyzes:

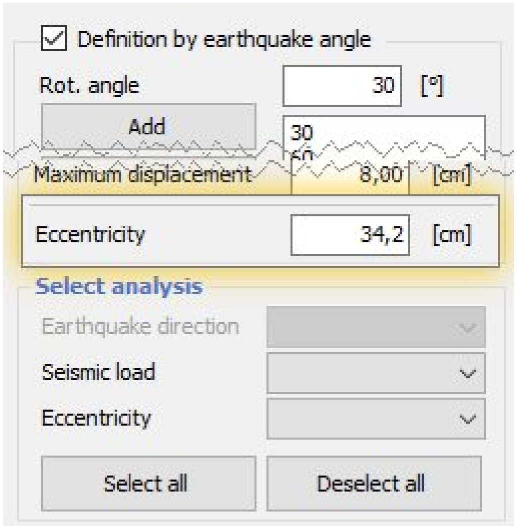
- 3 analyzes corresponding to a distribution of forces taken from a uniform trend of accelerations along the height of the construction;
- 3 analyzes corresponding to a distribution proportional to the static forces;

Among these the analyzes are distinguished in turn depending on the value that assume the eccentricity: nil, positive or negative.

It allows to define the value of the seism angle incrementally, based on the value of the previous angle.
For example, by entering the value 30 ° inside the specific cell and clicking the "increase" button, we add six analyzes to the list (with a seism angle of 30 °).
By leaving the value of the angle unchanged, selecting again "increase", we add to the list a further six analyzes with a seism angle of 60 °, or the value of the seism angle of the previous analyzes increased by 30 °.

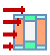


By selecting one of the previously defined angles and clicking the "delete" button, all six corresponding analyzes are deleted from the list.
It is possible to exclude one or more analyzes from the calculation by removing the check for the analysis in question in the "calculate analysis" column.

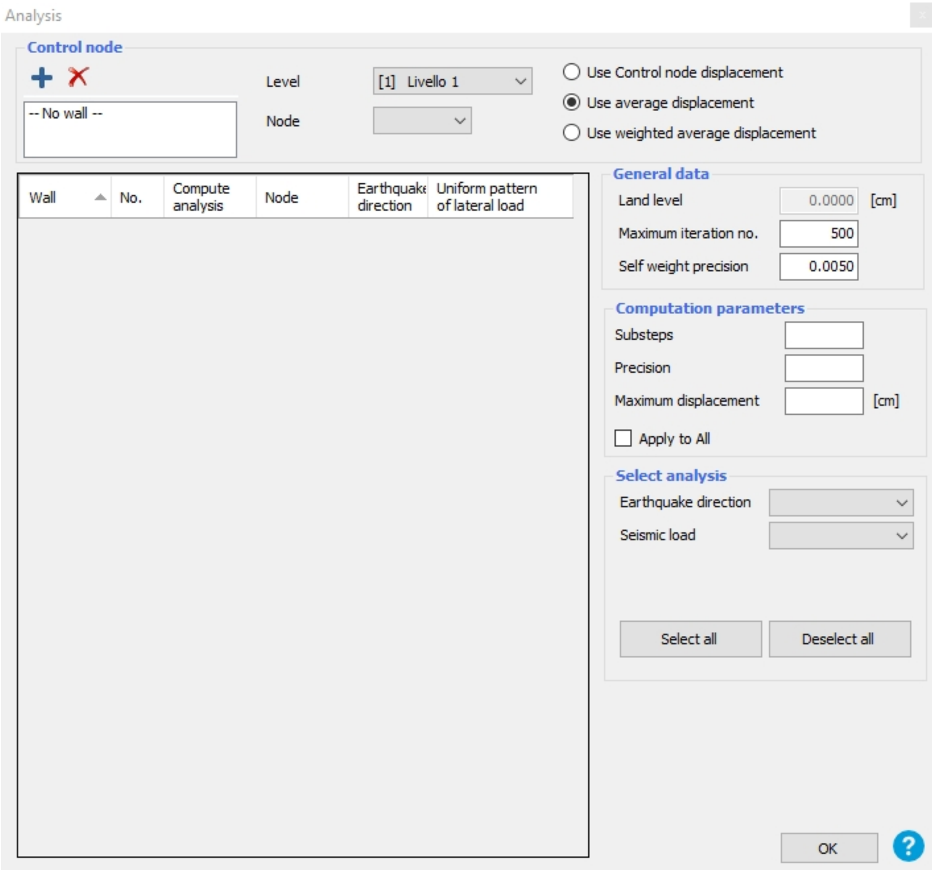


At the insertion time it is suggested an eccentricity value modifiable at any time.

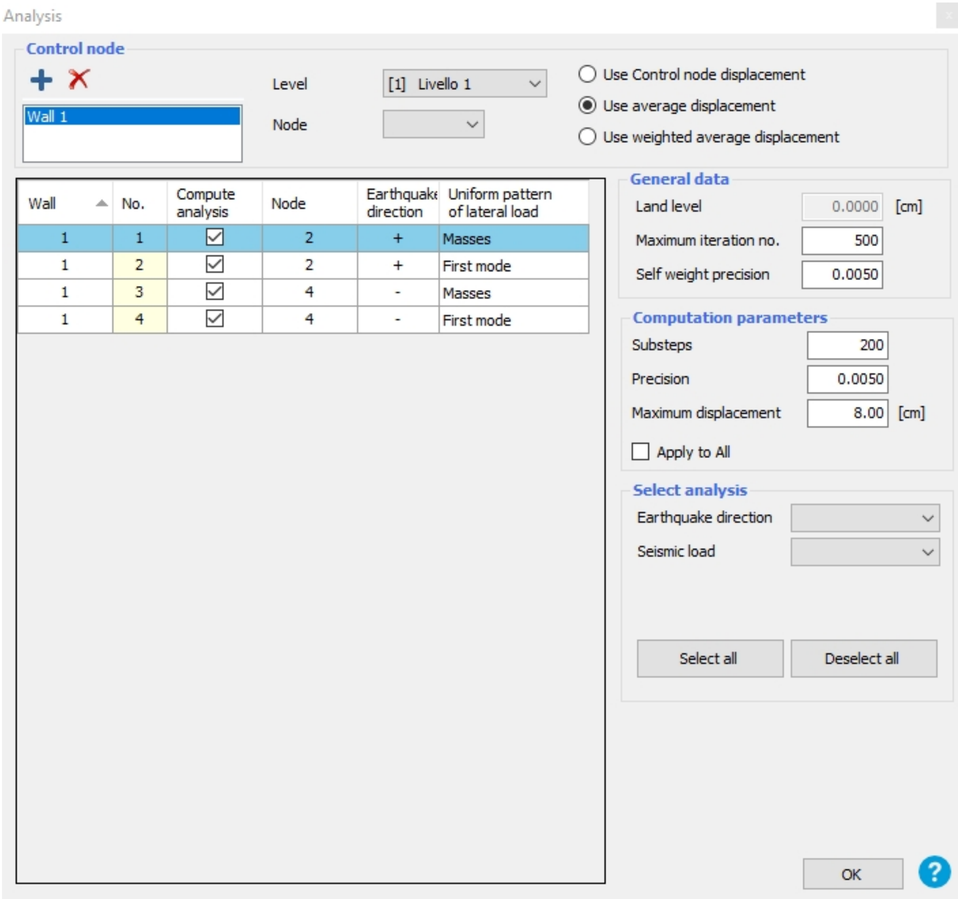
10.4.2.1.2 Single wall analysis

 To access to the pushover analysis of the single wall, select "Single Wall Calculation" from the drop-down list.
In this phase, the computation is performed using the selected code.

The main screen that appears is as follows

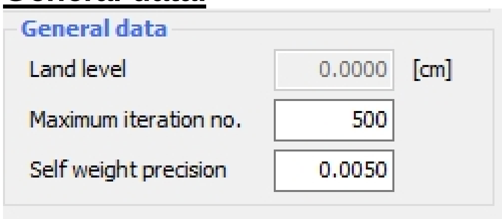


The aim of this analysis is to allow a pushover analysis considering the structure, not in its entirety, but in its individual walls, with the possibility of analyzing one or more walls together.
To start the analysis, as a first step yopu have to select the interested walls, by adding them from the appropriate command on the main screen ().



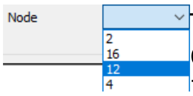
For each selected wall the software provides four differentiated analyzes for the direction of the earthquake and the distribution of the earthquake forces.

General data:



The General Data section is the same as in the global analysis dialog box described above.

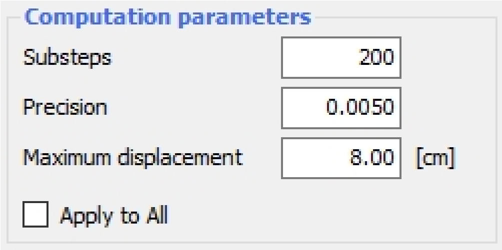
Control node options: definition of a control node is obligatory for computation.



The check node can be chosen from the drop-down menu.
Only the nodes on the selected wall for the analysis are present.
In the calculation of single walls, the choice of control node has less importance than global verification since it is not affected by the deformation in the plan;for this reason, for single wall verification, the program "suggests" a node in order to facilitate the insertion of the calculation parameters.
There is no assurance about the efficiency of the proposed node, it is advisable to check the results at the end of the calculation.

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

Parameters".

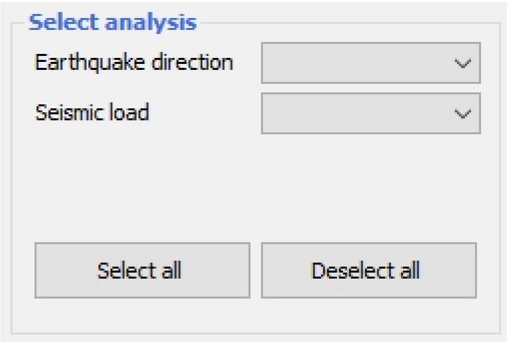


Substeps: represents the number of displacement steps computed by the solver for the seismic load pattern.

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

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

Apply to All: With the active tick the values are applied to all analyzes; if the check is disabled they are only applied to the selected analysis.



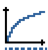
Earthquake Direction: indicates the earthquake direction.

Seismic load: Proporzionale alle masse o al primo modo di vibrare.

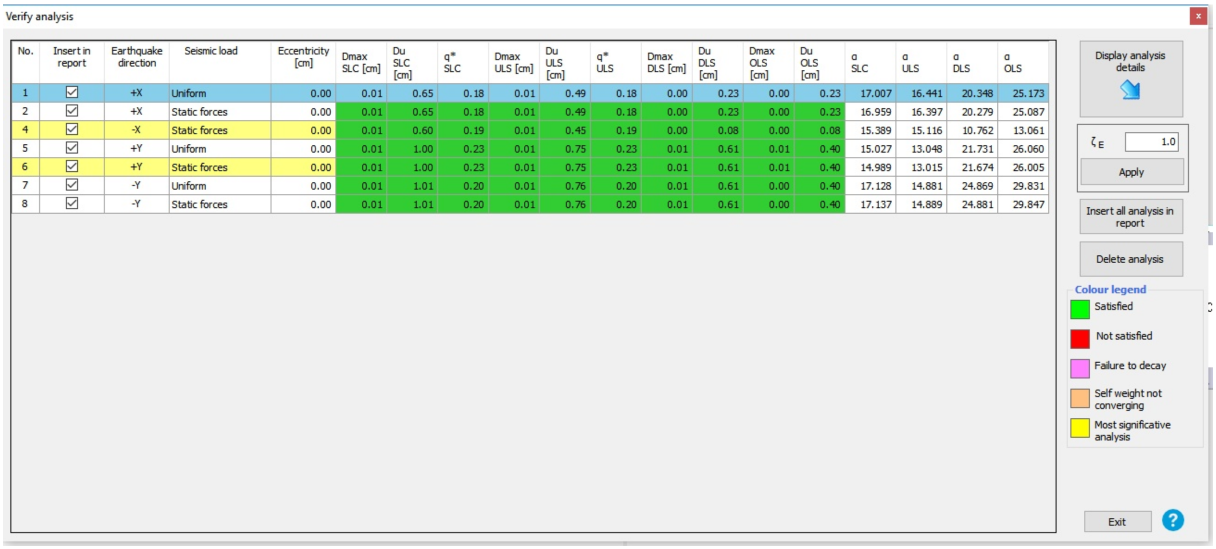
Enables the calculation of all the currently disabled analyzes

Disables the calculation of all the currently active analyzes

10.4.2.2 Display results

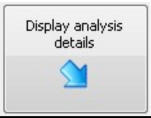
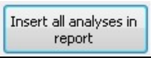
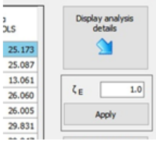


This window shows the results of the seismic computations performed on the model, based on that indicated in the code.



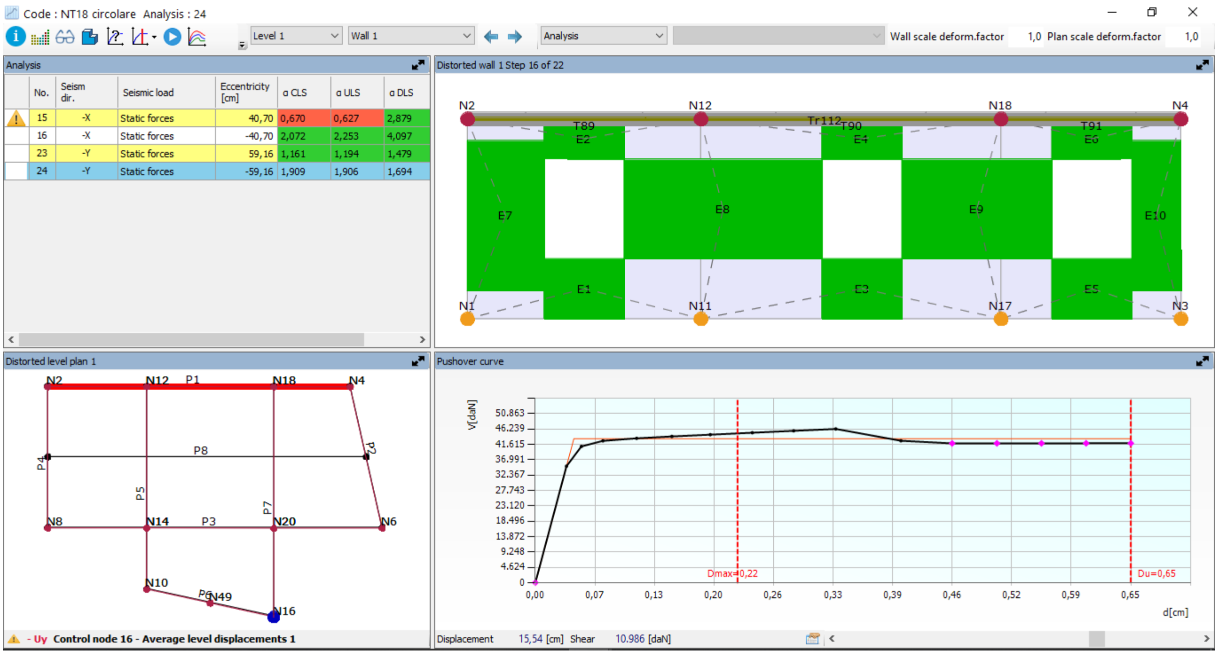
No.	Insert in report	Earthquake direction	Seismic load	Eccentricity [cm]	Dmax SLC [cm]	Du SLC [cm]	q* SLC	Dmax ULS [cm]	Du ULS [cm]	q* ULS	Dmax DLS [cm]	Du DLS [cm]	Dmax OLS [cm]	Du OLS [cm]	α SLC	α ULS	α DLS	α OLS
1	☑	+X	Uniform	0.00	0.01	0.65	0.18	0.01	0.49	0.18	0.00	0.23	0.00	0.23	17.007	16.441	20.348	25.173
2	☑	+X	Static forces	0.00	0.01	0.65	0.18	0.01	0.49	0.18	0.00	0.23	0.00	0.23	16.959	16.397	20.279	25.087
4	☑	-X	Static forces	0.00	0.01	0.60	0.19	0.01	0.45	0.19	0.00	0.08	0.00	0.08	15.389	15.116	10.762	13.061
5	☑	+Y	Uniform	0.00	0.01	1.00	0.23	0.01	0.75	0.23	0.01	0.61	0.01	0.40	15.027	13.048	21.731	26.060
6	☑	+Y	Static forces	0.00	0.01	1.00	0.23	0.01	0.75	0.23	0.01	0.61	0.01	0.40	14.989	13.015	21.674	26.005
7	☑	-Y	Uniform	0.00	0.01	1.01	0.20	0.01	0.76	0.20	0.01	0.61	0.00	0.40	17.128	14.881	24.869	29.831
8	☑	-Y	Static forces	0.00	0.01	1.01	0.20	0.01	0.76	0.20	0.01	0.61	0.00	0.40	17.137	14.889	24.881	29.847

This window summarizes the check parameters according to each norm, indicating whether the results were satisfactory or not.
The analyzes that have minimum "Alfa" values are more restrictive, so the results window shows the two analyzes with the minimum "Alfa SLV" (one for the X direction and one for the Y direction).
On the right of the window there are commands with the following functions:

<table>
<tr><td></td><td>Enter in the window that allows you to show the verification details.</td></tr>
<tr><td></td><td>Allows to print the parameters of all the analyzes included in the report</td></tr>
<tr><td></td><td>Clear the results of a performed analysis</td></tr>
<tr><td></td><td><u>NTC18</u>
The outcome of the verification is defined as passed when the vulnerability index is greater than 1.
On the basis of NT18, this lower limit can be reduced according to the type of intervention envisaged.
By entering the value and pressing [Apply] it is possible to define this limit for the project in question.</td></tr>
<tr><td></td><td><u>Satisfied</u>: The verification has been satisfied
<u>Non satisfied</u>: The verification is not been satisfied
<u>Failure to decay:</u> By the design code the pushover analysis continues until the imposed decay occurs.
In some cases the limit displacement could be reached before the decay occurs, and in these cases the analysis is marked, the verifications will not be carried out but the results of deformation and damage will be visible together with the pushover curve in order to allow the investigation of the causes that led to failure achievement of the resistance loss.
<u>Non-convergence at self weight</u>: When convergence on self weight is not achieved. Very often it occurs due to the inability of the structure to bear inserted elements.
<u>More significant analysis</u>: highlights the two most significant analyzes for the two main directions.</td></tr>
</table>

10.4.2.2.1 Display analysis details

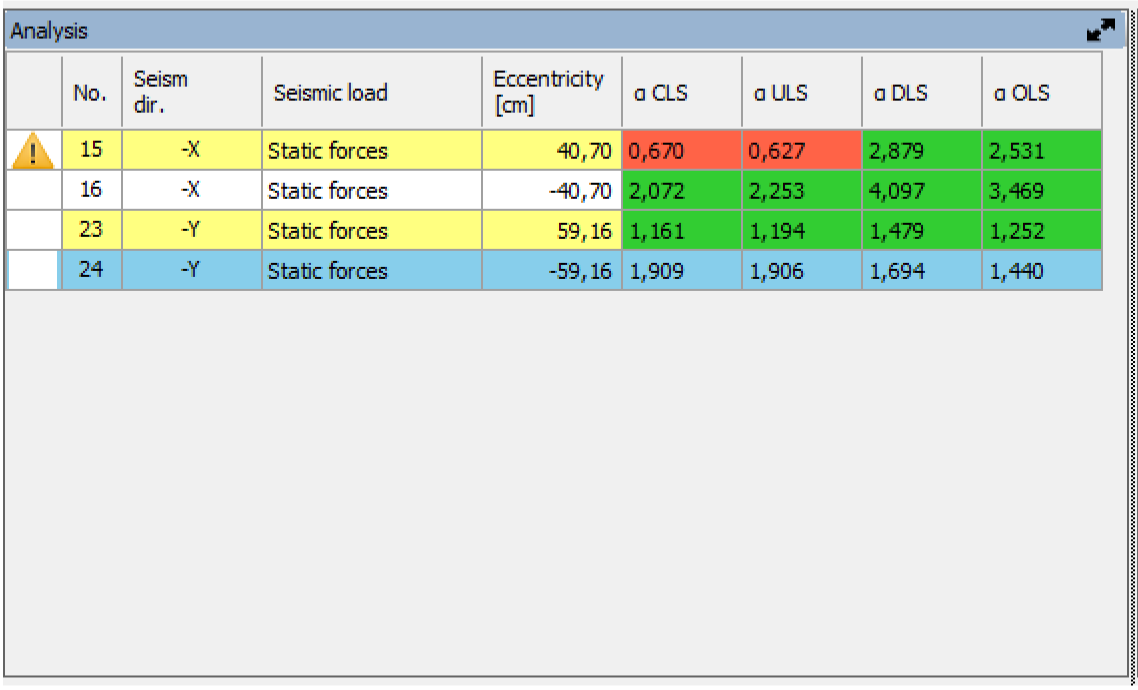
Select "Pushover analysis verification" from the the drop-down list



The results' environment is divided in 4 main areas:

Analysis table

Contains the performed analyzes summary for the model taken in consideration.
The first columns describe the type of analysis, the last show the vulnerability indexes for each of the three limit states.
The background color, green or red, distinguishes between the exceeded analysis by those that aren't.
The yellow color shows the two analyzes that have the lowest vulnerability indexes (more significant for calculation purposes).



Analysis

	No.	Seism dir.	Seismic load	Eccentricity [cm]	α CLS	α ULS	α DLS	α OLS
!	15	-X	Static forces	40,70	0,670	0,627	2,879	2,531
	16	-X	Static forces	-40,70	2,072	2,253	4,097	3,469
	23	-Y	Static forces	59,16	1,161	1,194	1,479	1,252
	24	-Y	Static forces	-59,16	1,909	1,906	1,694	1,440

The active analysis is highlighted in blue, it's results are detailed in the other 3 areas.

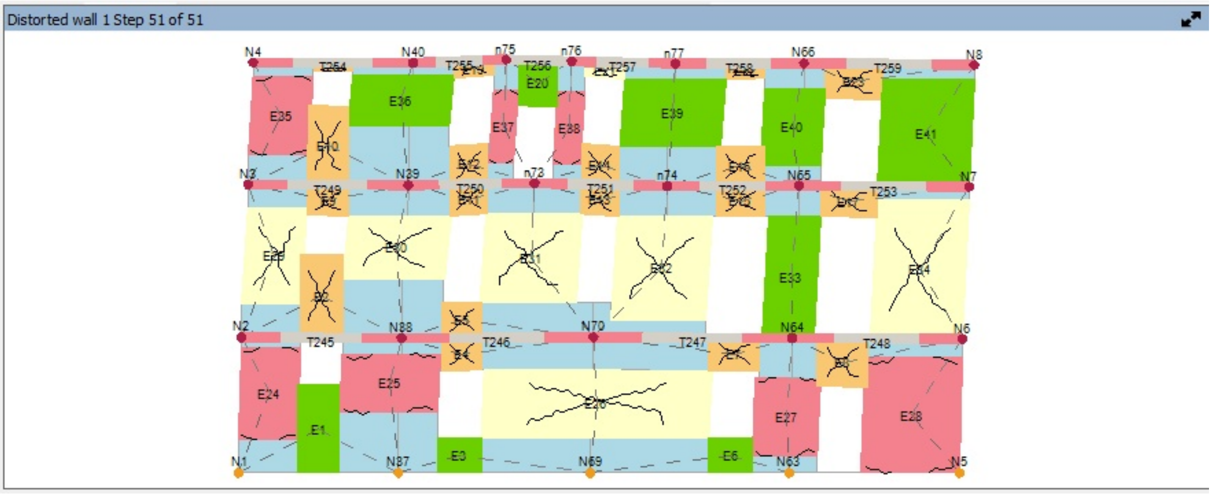
Clicking on a row in the table will automatically switch to the details of another analysis.

The results obtained are subjected to some checks aimed at guaranteeing the goodness of the results and to help the designer in the critical analysis of them.

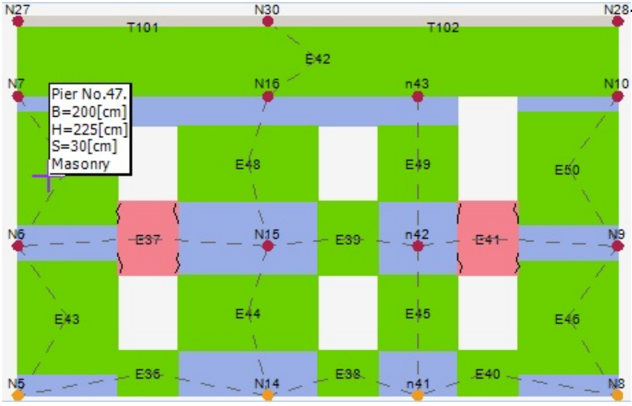
In the case of unsuccessful checks, the symbol ⚠ is shown in the first column.
For more information on the checks carried out click HERE.

Distorted Wall

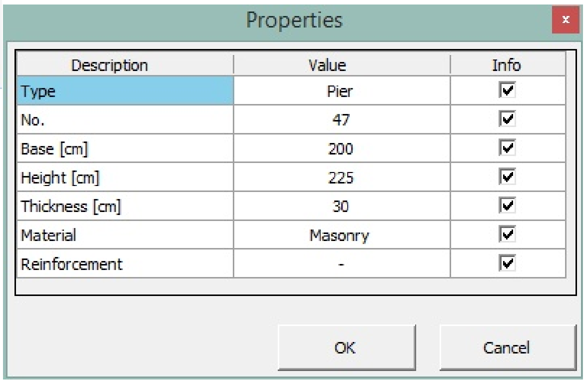
Shows the wall's deformation, the elements color indicates the type of the identified damage immediately through the color legend.
The deformation is shown in correspondence of each single step.
This tool shows a great potential for the management of any adjustment interventions on the existing structure, as it proves to be very effective for the identification of the intervention areas.



Positioning the pointer over an item displays a tool tip with the masonry's characteristics.



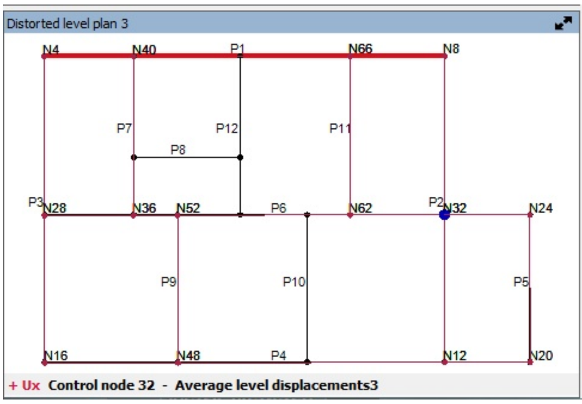
The content of the tool tip can be customized through a special window which is accessed by pressing the right button.



The window of the mesh prospectus is a real CAD, the functions zoom and pan are available by using the mouse.
A double-click on the wheel zooms extension bringing it back to the maximized window.
Pressing the right mouse button the option "Save Image" which allows you to save the image to be used in the report is available.

Distorted level Plan

An overlapped view of the deformed plant to the not deformed one helps to identify any torsional effects.



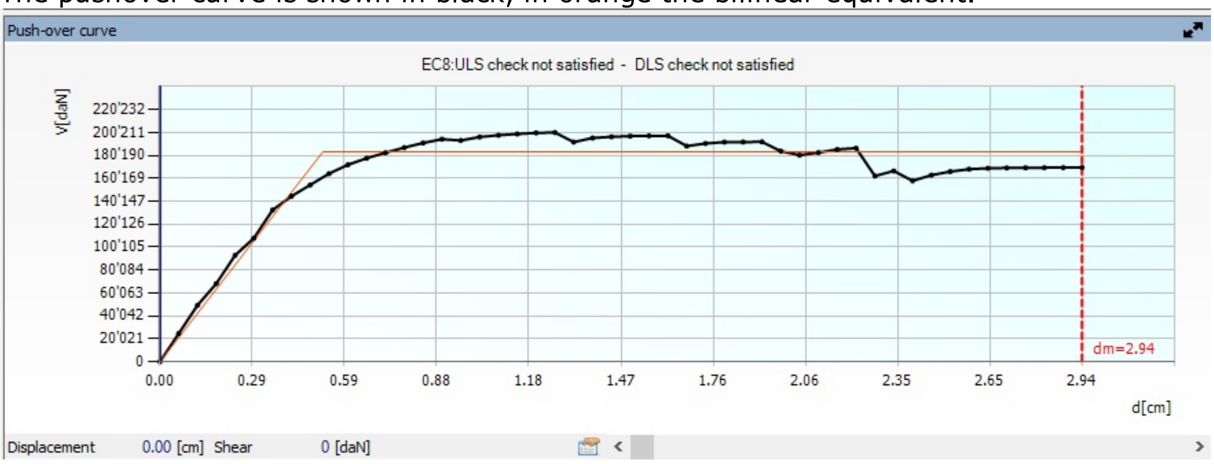
The control node is shown in the lower part of the window and highlighted in blue in the plant.

You can select a wall in the plant to see it in the statement of deformation.

The zoom and pan functions are available by using the mouse.
A double-click on the wheel zooms extension bringing it back to the maximized window.
Pressing the right mouse button the option "Save Image" which allows you to save the image to be used in the report is available.

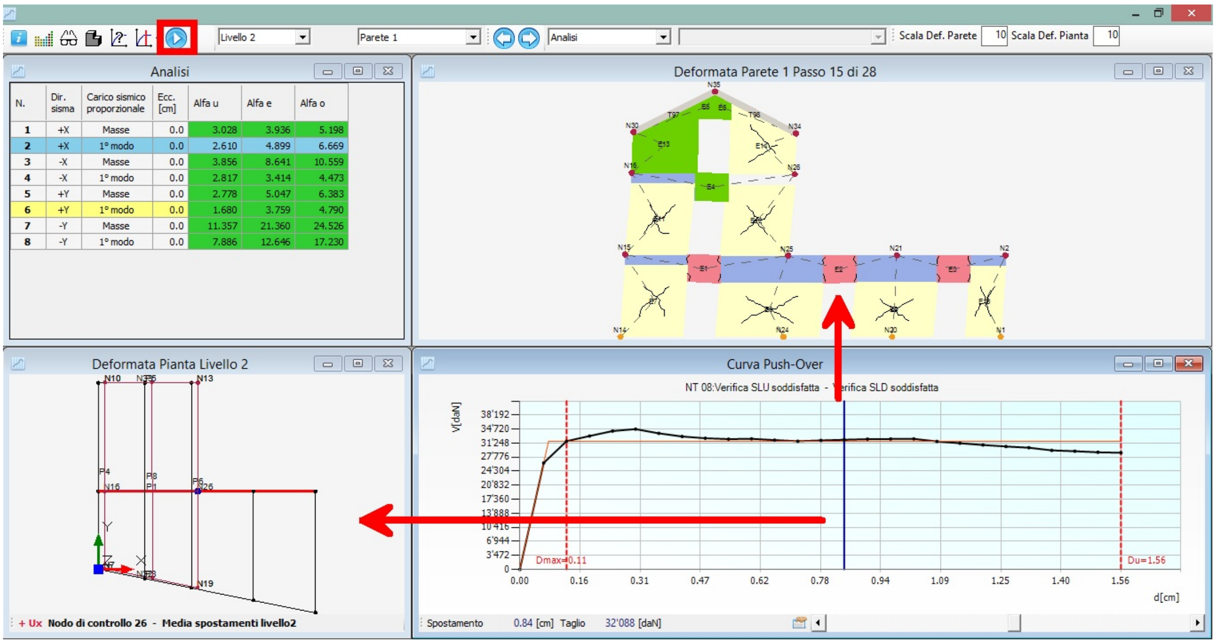
Push-over Curve

Shows the capacity curve.
The pushover curve is shown in black, in orange the bilinear equivalent.



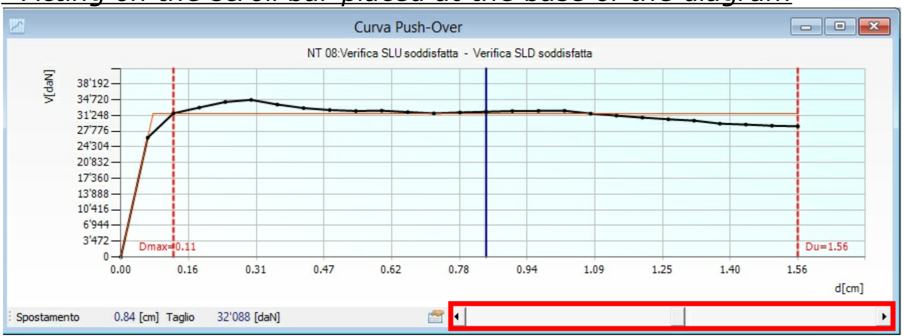
Du: Offered displacement of the structure
Dmax: displacement demand of the seismic force

Auto run: By using this command, you can start a video showing the progressive deformation of the structure.
Is displayed a capacity curve indicator that allows us to understand at what point in the evolution of load we are; the elevation and the plan show the deformed state.



You can place the marker at a specific point in two different ways:

- Acting on the scroll bar placed at the base of the diagram

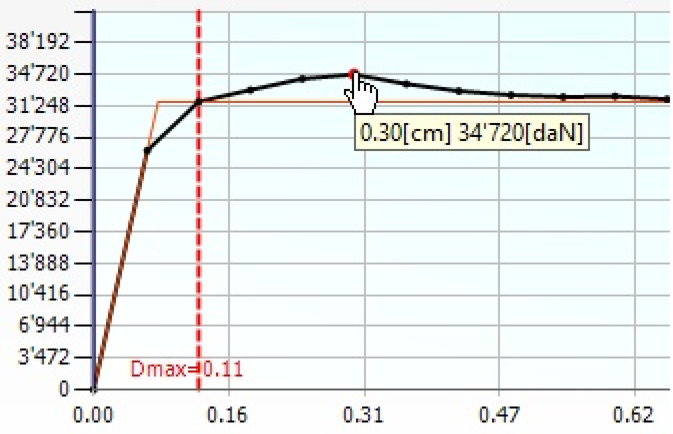


- By clicking on the diagram at the step where you want to place the indicator

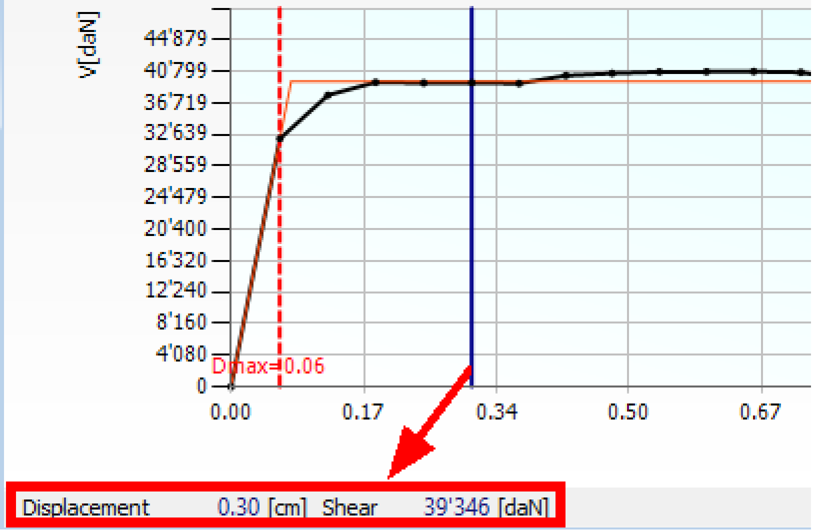
Interrogate the curve's coordinates

You can interrogate the values of the curve in three different ways:

- *Position the pointer on the curve at a step and read the value that appear*

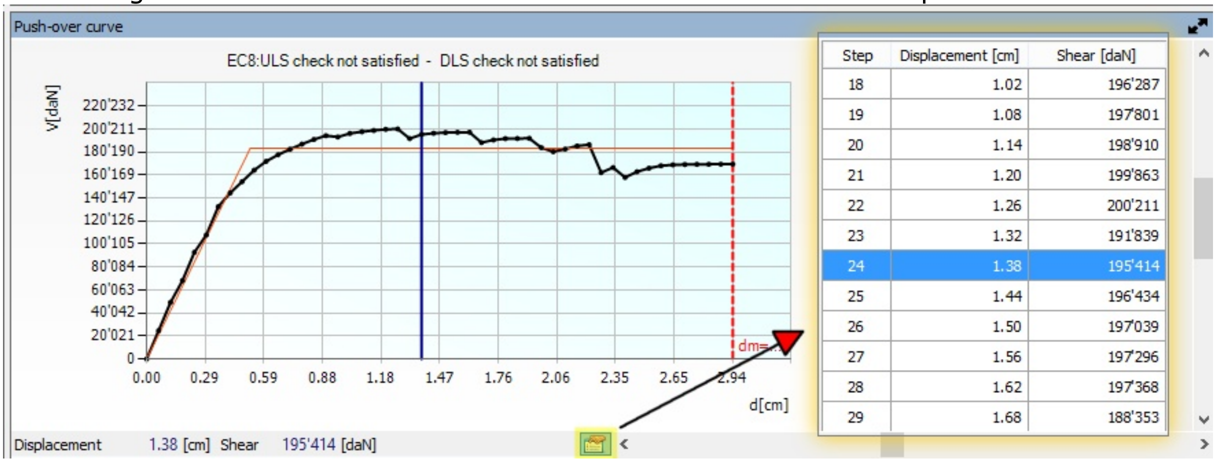


- *Position the step pointer and read the coordinates on the bottom bar*



- *Refer to the table with coordinates*

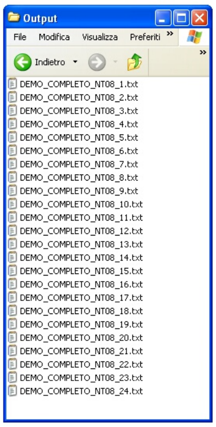
 Using this button shows the table with the coordinates of each point on the curve.



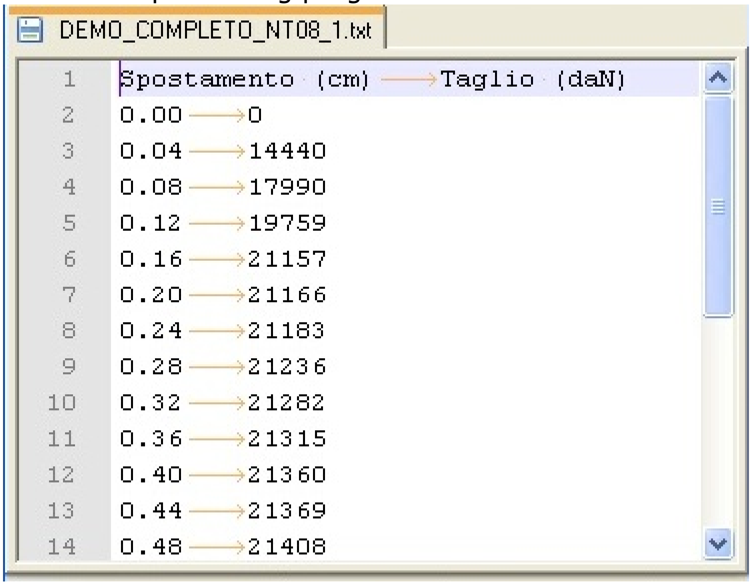
Step	Displacement [cm]	Shear [daN]
18	1.02	196'287
19	1.08	197'801
20	1.14	198'910
21	1.20	199'863
22	1.26	200'211
23	1.32	191'839
24	1.38	195'414
25	1.44	196'434
26	1.50	197'039
27	1.56	197'296
28	1.62	197'368
29	1.68	188'353

The row corresponding to the active step appears highlighted.
Clicking the right mouse button on the table allows you to copy the contents of the table.

Exporting the pushover's diagram points



At finished calculation, within the project folder in the directory "output", is saved a file for each of the performed analyzes.
The file will have the name "ModelName"_"Regulations"_"N.ANALYSIS".TXT.
The points of the curve are separated by tabs for easier processing by other data processing programs.

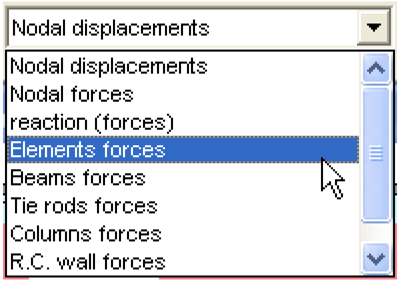

DEMO_COMPLETO_NT08_1.txt

```
1  Spostamento (cm)	Taglio (daN)
2  0.00	0
3  0.04	14440
4  0.08	17990
5  0.12	19759
6  0.16	21157
7  0.20	21166
8  0.24	21183
9  0.28	21236
10 0.32	21282
11 0.36	21315
12 0.40	21360
13 0.44	21369
14 0.48	21408
```

On the command bar, you can change the option from "Analysis" to "Display results" in order to replace the analysis table with the results details table.


Wall results 1

Node	Ux [cm]	Uy [cm]	Uz [cm]	Rot X [rad]	Rot Y [rad]
1	0.00	0.00	0.00	0.0000	0.0000
2	0.37	-0.01	-0.04	0.0001	0.0015
3	0.95	-0.04	-0.13	0.0001	0.0014
4	1.34	-0.06	-0.21	-0.0001	0.0005
5	0.00	0.00	0.00	0.0000	0.0000
6	0.37	0.01	-0.26	-0.0006	0.0012
7	0.95	0.02	-0.42	-0.0003	0.0014
8	1.34	0.03	-0.54	-0.0008	0.0017
37	0.00	0.00	0.00	0.0000	0.0000
38	0.37	-0.01	-0.13	0.0000	0.0015
39	0.95	-0.03	-0.20	0.0002	0.0014
40	1.34	-0.04	-0.21	0.0003	0.0013
63	0.00	0.00	0.00	0.0000	0.0000
64	0.37	0.00	-0.15	0.0000	0.0013
65	0.95	0.01	-0.30	0.0001	0.0011
66	1.34	0.01	-0.34	0.0009	0.0012



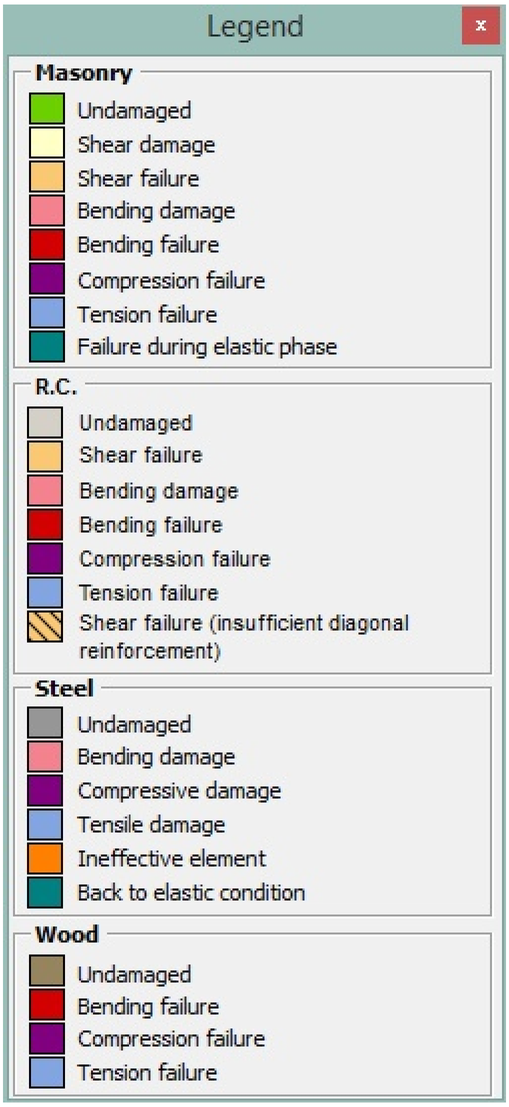
"In this window, every value can be selected. With a right click on the button of the mouse in the selected area "Copy data" appears; by selecting it can be loaded on the clipboard in order to be pasted in the application that we consider most suitable. (word, excel, etc..)."

Check Details:

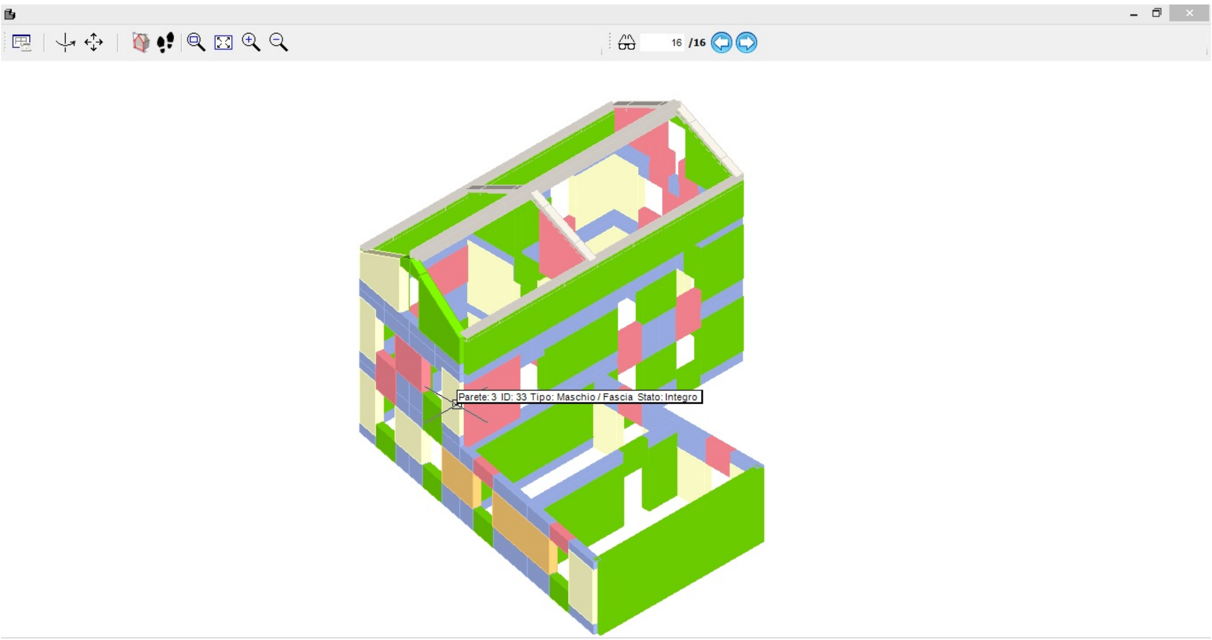
Represents a summary window that displays the details of the required analyzes and checks.

Find:
Identify objects by their number. For more details click here.

Colors legend: Shows a colors map in order to identify the type of damage to the structure (the map shows the damage of the masonry elements, of the R.C., steel and wood).



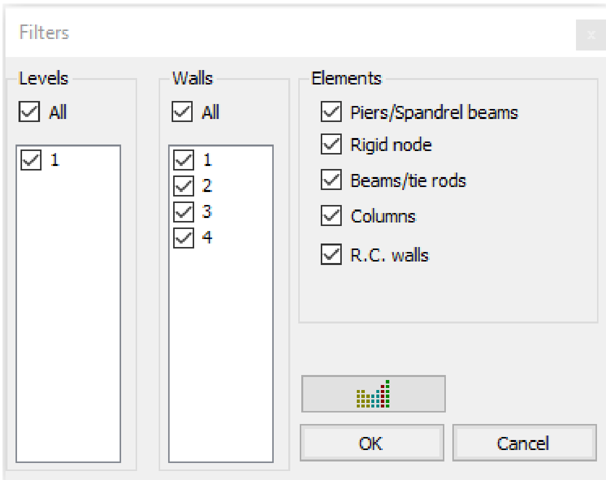
View 3D of the mesh: Allows you to see the 3D of the mesh distinguished according to the step of damage and with the color map just described.



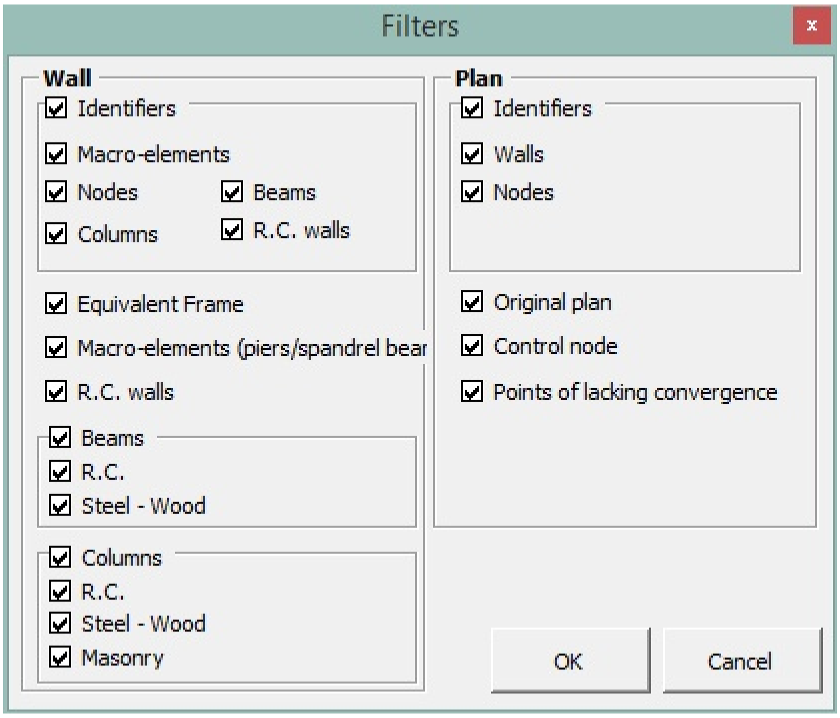
Comands in 3D view:

Display filters:

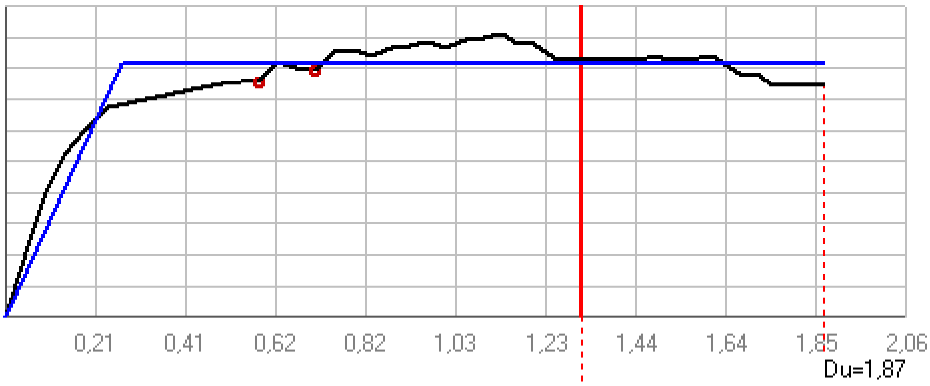
- *Levels*: Are shown only the selected levels
- *Walls*: Are shown only the selected walls.
- *Elements*: Are shown only the selected elements.



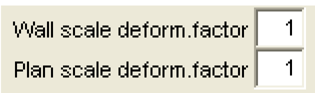
Display filters: Allow the user to decide what to display in two views (level plan, wall) of the deformation.



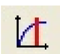
This window allows you to decide whether to display the points of possible failure convergence in the non-linear pushover which are shown on the chart with small red circles.



The presence of some non-convergence points should not worry but the presence of a high number of outside convergence steps must be index of a modeling that can be improved.



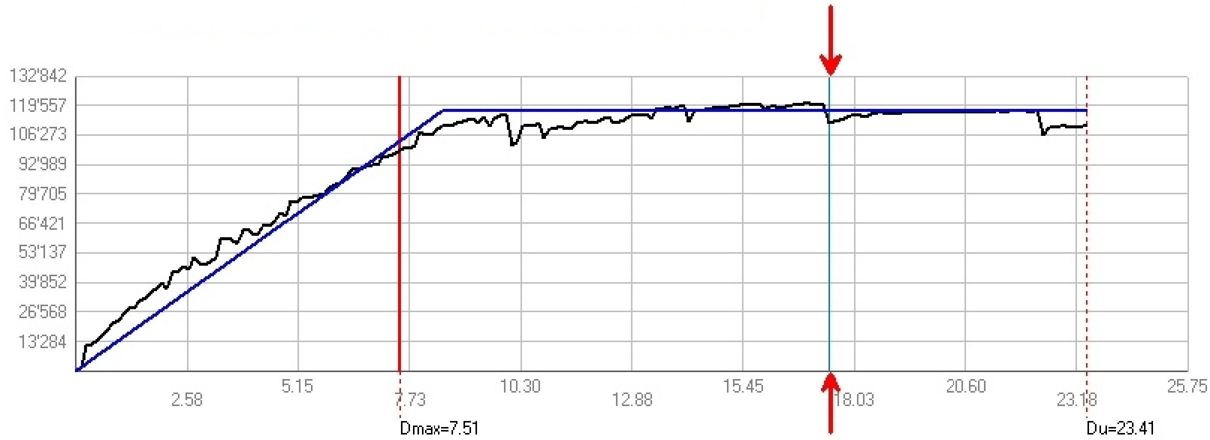
These commands present in the analysis bar, allow to decide the scale of display of the deformations both on the prospect of the wall and on the level plant.



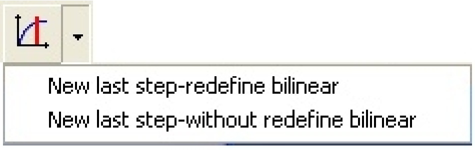
New Last step

If the user considers it appropriate, may decide to find a different displacement value for the ultimate limit state, as long as lower than the value associated with the decay of resistance limit.

Use the scroll bar to go through the underpasses of the analysis and to identify by using the appropriate blue marker a new "last step".

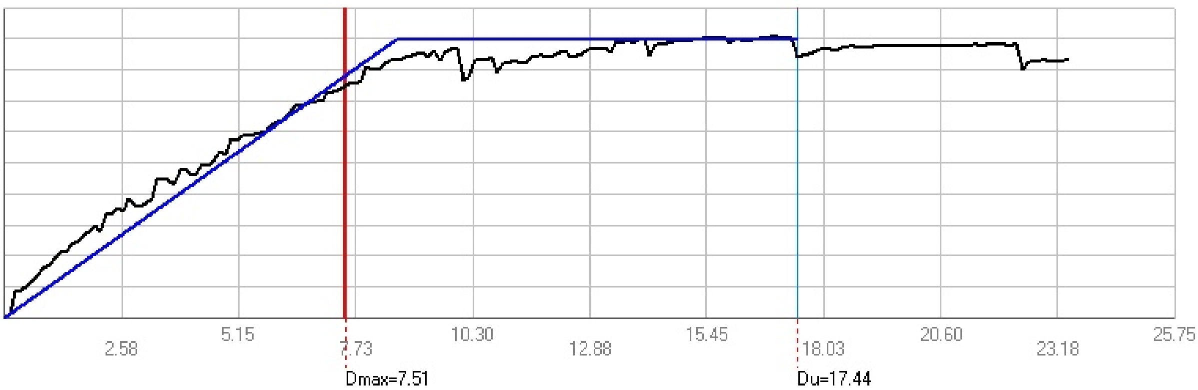


Pressing the arrow to the right of the "New last step", there are two options.



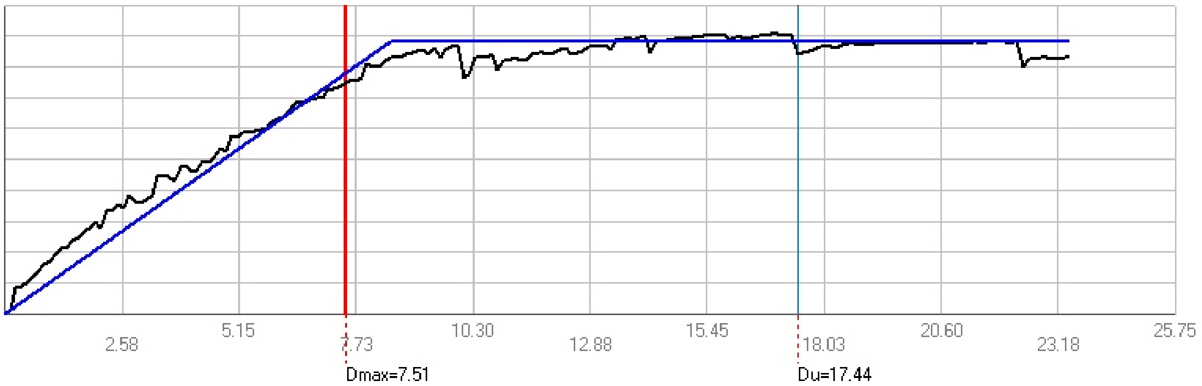
New step - bilinear recalculation:

The bilinear is redefined taking into account all the points of the curve until the new ultimate displacement.



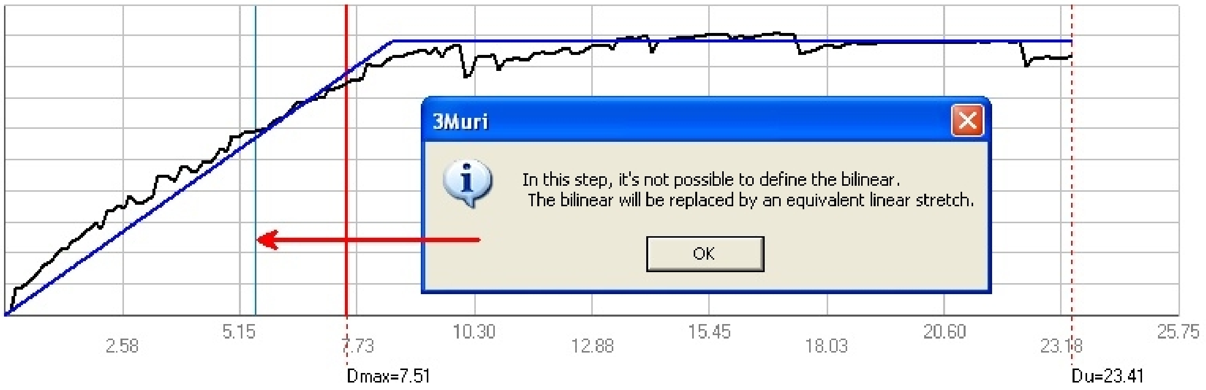
When to you use this command?
The simultaneous rupture of many elements that could affect the overall stability. An example would be the failure of all pillars of a porch which is located at the main entrance of a building. This break could be considered by the designer to continue limiting the analysis.

New step-without bilinear recalculation:
In this case the displacement is redefined at the new last step but it is not redefined the bilinear by maintaining the original one.



When to you use this command?
This command is used when the designer considers useful interrupt the analysis in a previous step due to an element considered of significant importance, the stability of which probably can not give rise to failures that would alter the continuation of the deformation of the structural complex.

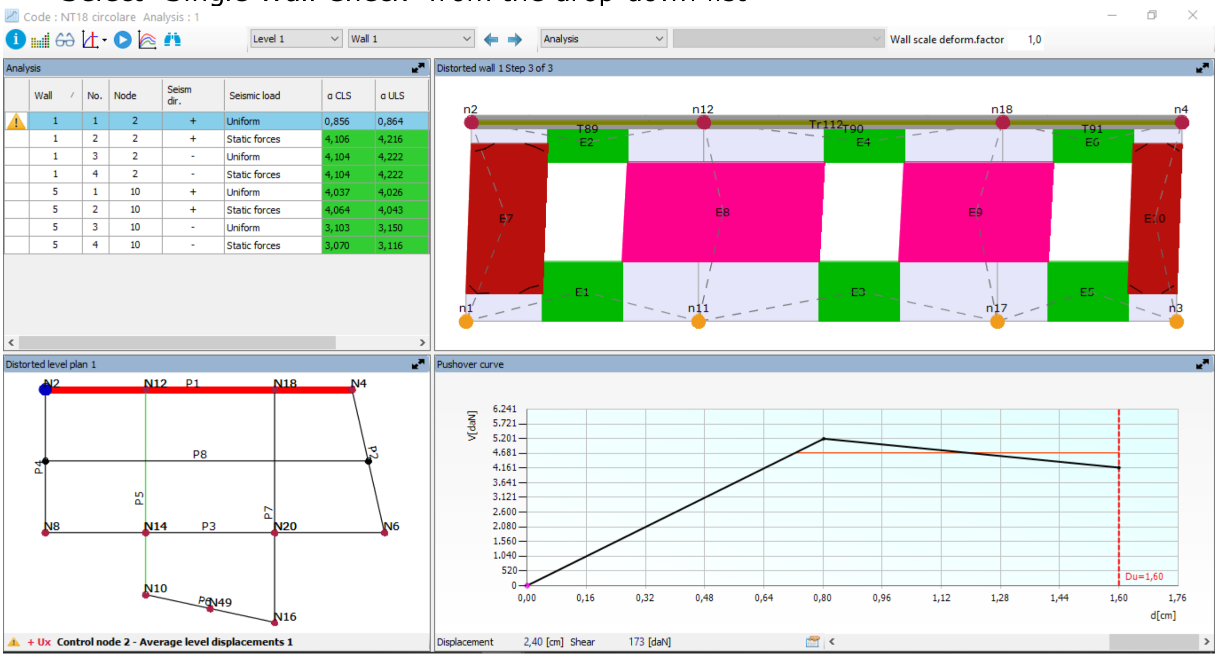
CAUTION!!! For very low values of displacement is not possible to define new values of ultimate displacement; such low values may not allow the regeneration of the bilinear equivalent because you can not abide by the requirements of tracking (eg, the intersection of the bilinear pushover and conservation of energy dissipation).



In this case, the program informs that there is no bilinear that satisfies these requirements.
Confirming this window, the program defines a new step by replacing the last bilinear with a linear stretch that dissipates the same energy of the pushover (in this case can not therefore be imposed as a point of passage for the bilinear).

10.4.2.2.2 Show details single wall analysis

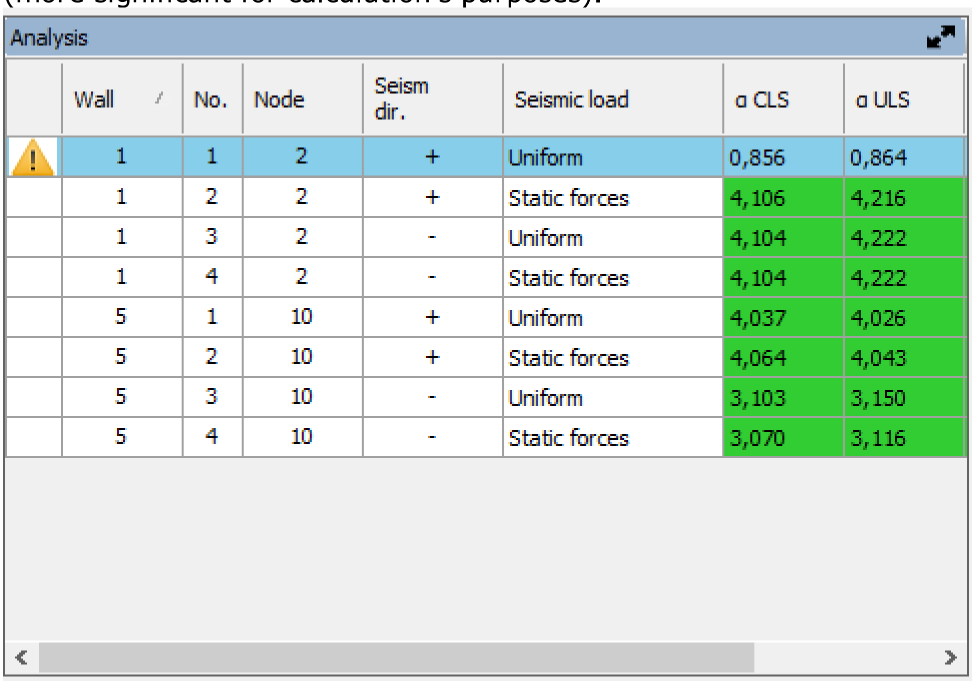
Select "Single Wall Check" from the drop-down list



The result environment is divided into 4 main areas:

Analysis Table

Contains the performed analyzes summary for the model taken in consideration.
The first columns describe the type of analysis, the last show the vulnerability indexes for each of the limit states.
The background color, green or red, distinguishes between the exceeded analysis by those that aren't.
The yellow color shows the two analyzes that have the lowest vulnerability indexes (more significant for calculation's purposes).



Analysis

	Wall	No.	Node	Seism dir.	Seismic load	α CLS	α ULS
⚠	1	1	2	+	Uniform	0,856	0,864
	1	2	2	+	Static forces	4,106	4,216
	1	3	2	-	Uniform	4,104	4,222
	1	4	2	-	Static forces	4,104	4,222
	5	1	10	+	Uniform	4,037	4,026
	5	2	10	+	Static forces	4,064	4,043
	5	3	10	-	Uniform	3,103	3,150
	5	4	10	-	Static forces	3,070	3,116

The active analysis is highlighted in blue, it's results are detailed in the other 3 areas.

By clicking on a row in the table will automatically switch to the details of another analysis.

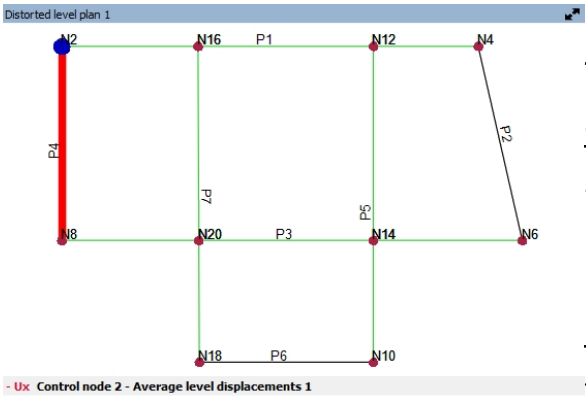
The results obtained are subjected to some checks aimed at guaranteeing the goodness of the results and to help the designer in the critical analysis of them.

In the case of unsuccessful checks, the symbol is shown in the first column.
For more information on the checks carried out click HERE.

Distorted Wall

The deformed wall of the elevation follows the same rules described for the global analysis.

Plan



A thick red-colored stretch highlights the "active" wall (the one whose results are shown).
The walls can be drawn with a thin stretch of color:
- Black: Wall not calculated
- Green: Calculated with verified analysis
- Red: calculated with analysis NOT verified

The control node is indicated at the bottom of the window and is highlighted in blue in the plant based on the currently calculated wall.

It is possible to select a wall in the plant to see it in the eleveation deformation.

Zoom and pan functions are available with the mouse.
A double click on the wheel zooms extension maximized it back to the window.
By pressing the right mouse button, the "Save Image" option is available to save the image to be used in the calculation report.

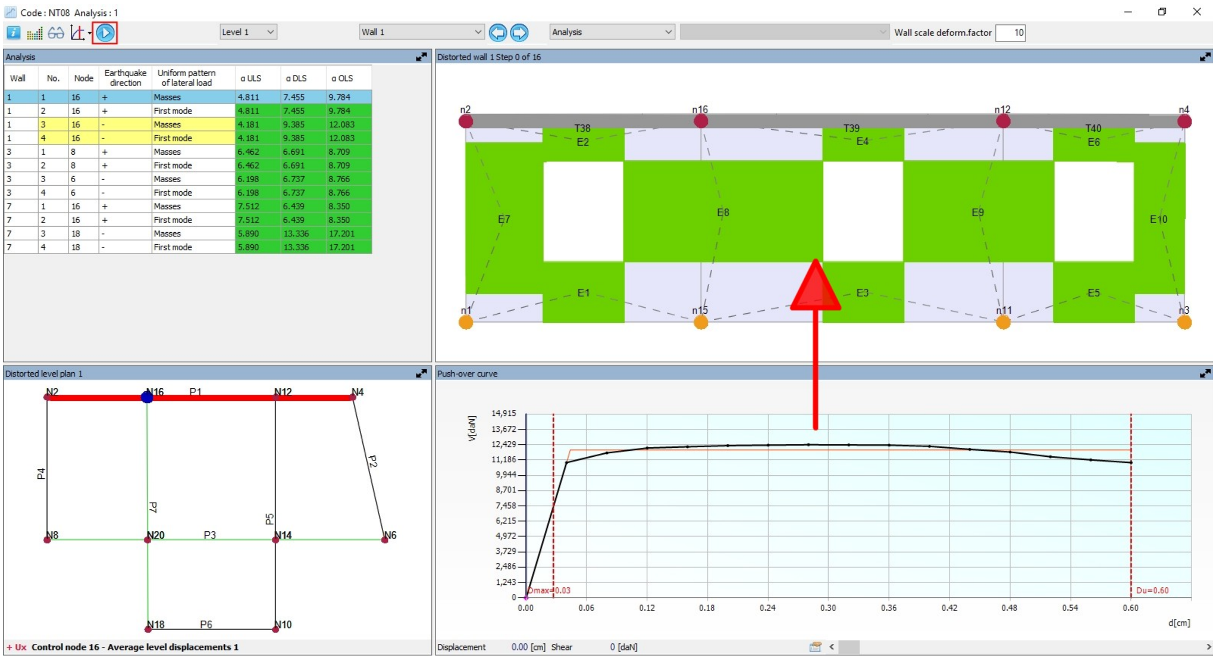
In the case of the calculation of the individual walls, the plant shown is undeformed.

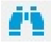
Push-over curve

The capacity curve diagram follows the same rules.

Auto run: By using this command, you can start a video showing the progressive deformation of the structure.
Is displayed a capacity curve indicator that allows us to understand at what point in the

evolution of load are we; The elevation and the plan show the deformed state.



 Find:

Identify objects by their number. For more details click here.

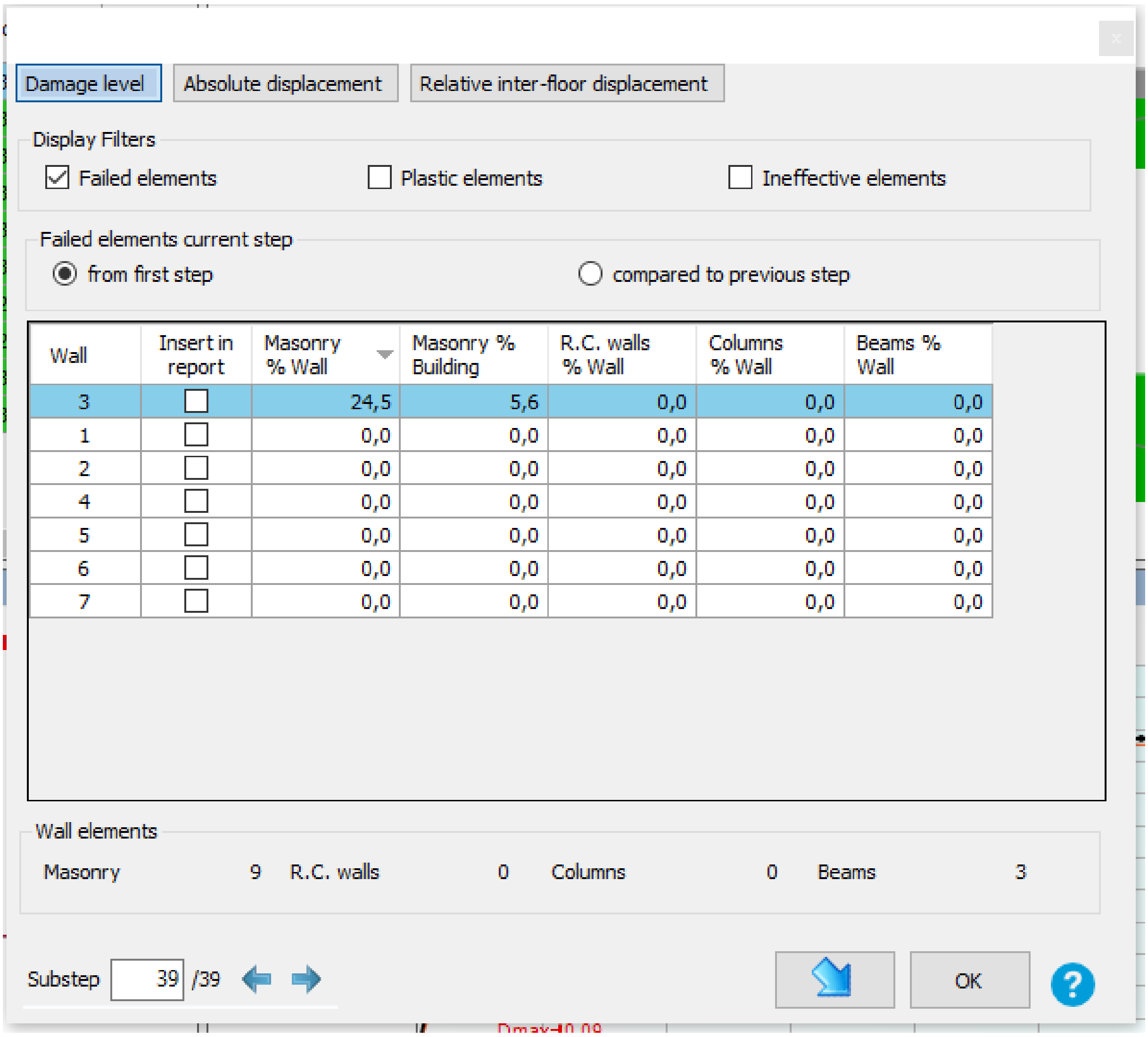
10.4.2.2.3 Display Results

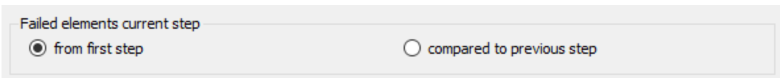
This environment displays a table showing on rows the percentage of the damaged elements for each wall. The rows and therefore the walls, are sorted according to the percentage of damaged elements.

Under this system you can immediately identify the most damaged wall (the first on the list).

Allows to immediately load the view of the selected wall (the corresponding row) in order to examine it to provide interventions.

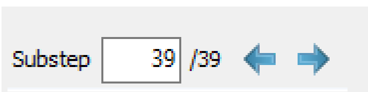
The percentage of broken elements presented in the table can be defined from the beginning of the loading history or in the current step of the analysis.





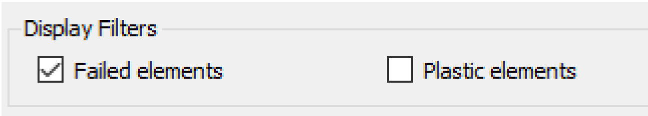
Allows to choose which substep of the curve to display the percentage of damaged elements.

It allows to view the number of elements, divided by material, which have suffered the damage indicated in the appropriate section.



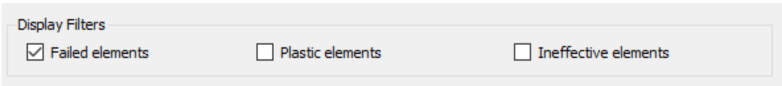
Allows to select the substep in which you want to see the data of displacements and damage.

NTC08 Eurocode NL (Netherlands) NPR 9998-2018 SIA 2018



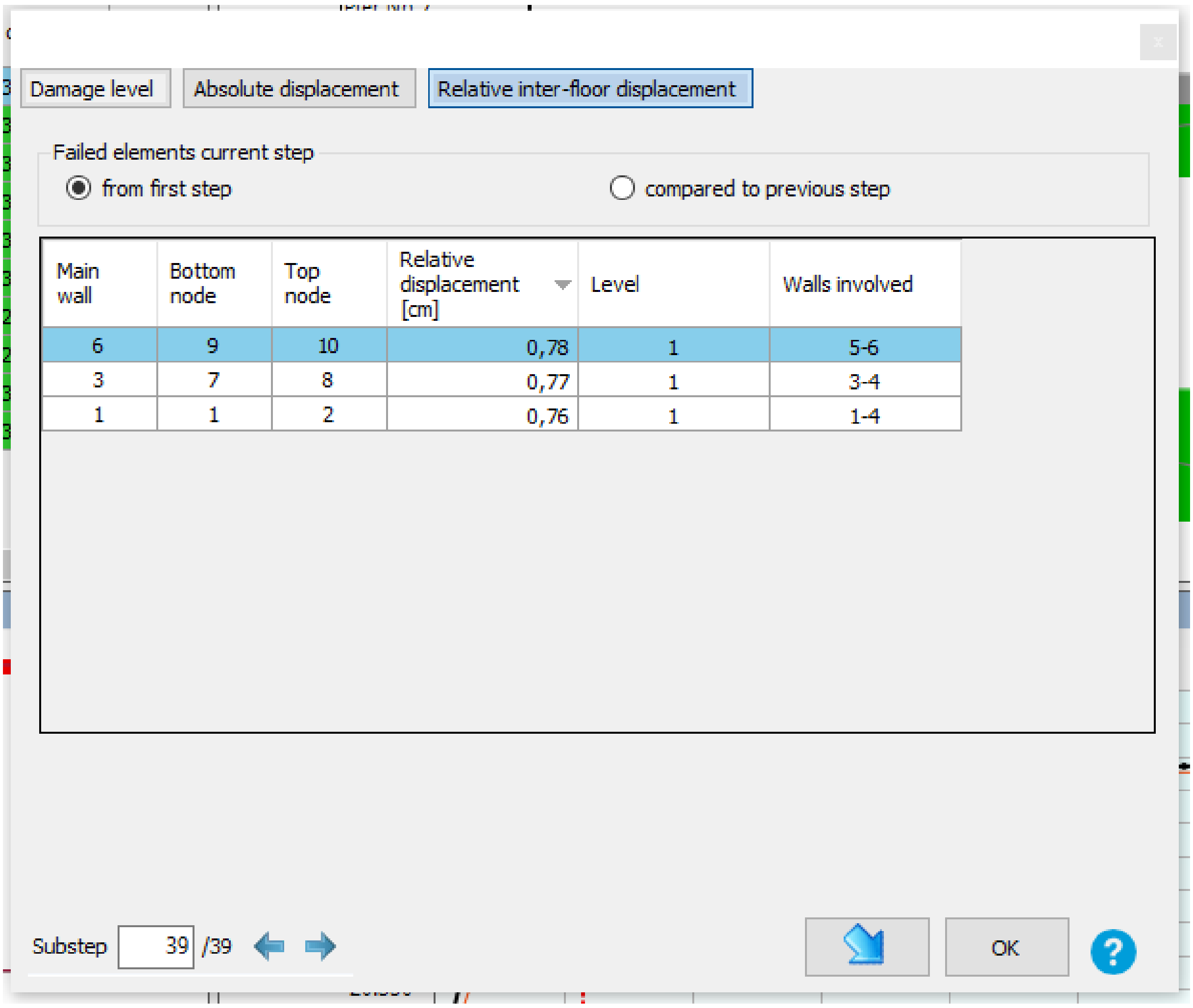
The percentage of damaged elements is displayed including all broken elements present within the model.

NTC18 **Eurocode**

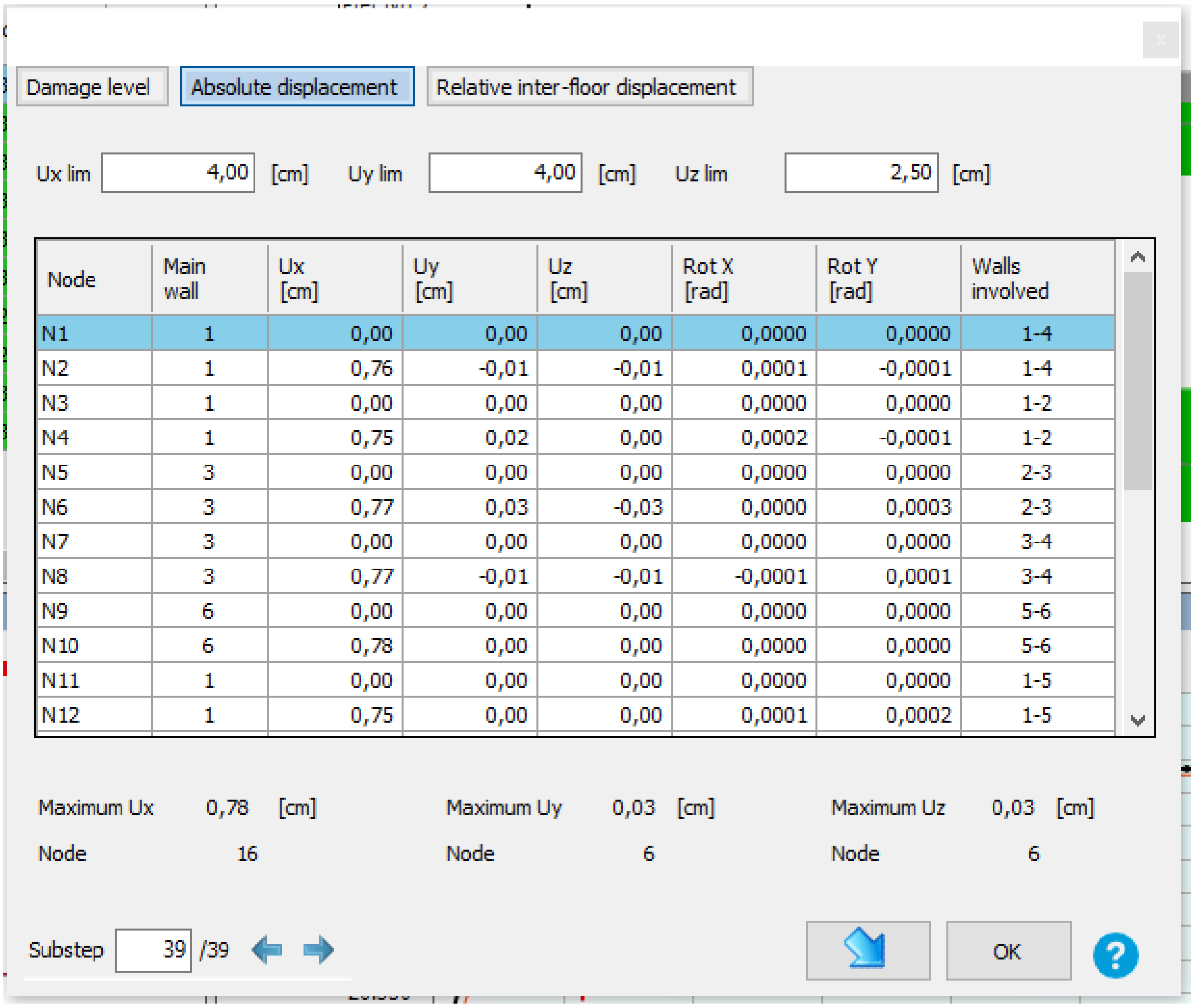


It is possible to filter the display of broken elements on the basis of the damage status assumed by the elements.

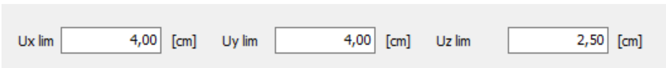
A second environment of "displacement control" orders the walls according to the relative inter-floor displacement in order to identify where the greater displacement occurs.



A third "displacement control" environment orders the associated nodes and walls according to the chosen limit displacement.



Node	Main wall	Ux [cm]	Uy [cm]	Uz [cm]	Rot X [rad]	Rot Y [rad]	Walls involved
N1	1	0,00	0,00	0,00	0,0000	0,0000	1-4
N2	1	0,76	-0,01	-0,01	0,0001	-0,0001	1-4
N3	1	0,00	0,00	0,00	0,0000	0,0000	1-2
N4	1	0,75	0,02	0,00	0,0002	-0,0001	1-2
N5	3	0,00	0,00	0,00	0,0000	0,0000	2-3
N6	3	0,77	0,03	-0,03	0,0000	0,0003	2-3
N7	3	0,00	0,00	0,00	0,0000	0,0000	3-4
N8	3	0,77	-0,01	-0,01	-0,0001	0,0001	3-4
N9	6	0,00	0,00	0,00	0,0000	0,0000	5-6
N10	6	0,78	0,00	0,00	0,0000	0,0000	5-6
N11	1	0,00	0,00	0,00	0,0000	0,0000	1-5
N12	1	0,75	0,00	0,00	0,0001	0,0002	1-5

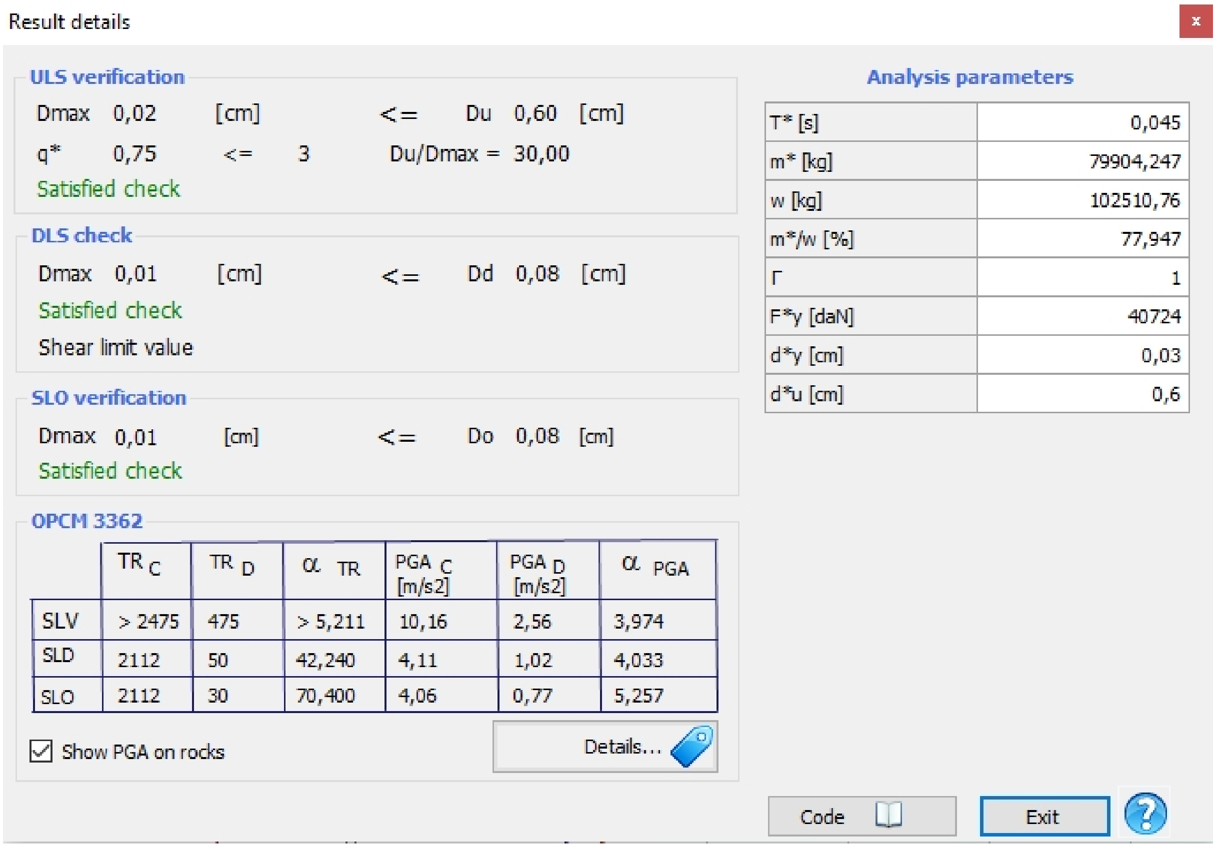


They allow to set a value of the limit displacement in the three directions "x", "y" and "z".
All values greater than or equal to the limit displacements indicated will be highlighted in the relative columns.

They indicate which of all the nodes have the greatest displacement value along the three directions "x", "y" and "z".

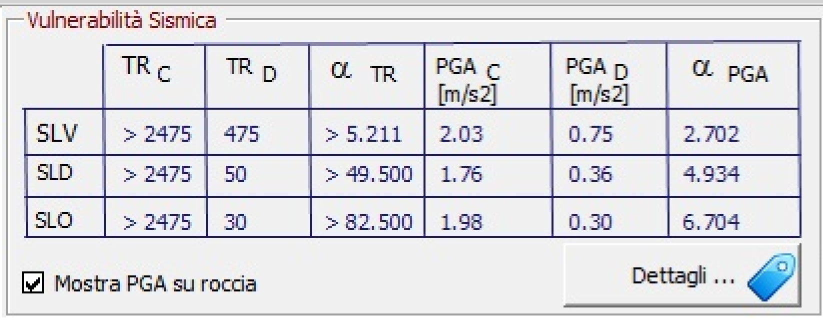
10.4.2.2.4 Results Details

Represents a summary window that displays the details of the analyzes and required checks.



This window displays a summary of the test parameters required by each regulation.

NTC08



The table for the "Seismic Vulnerability" evaluation shows the α parameters derived from the homonyms reports for each of the limit states:

α_{PGA}=PGA_C/PGA_D ; α_{TR}=TR_C/TR_D

PGA_C: Limit capacity acceleration for each limit state (independent from the seismic spectrum).
PGA_D: Spectral acceleration for each of the limit states (depends on the seismic spectrum).
TR_C: Return period of the limit capacity seismic action for each of the limit states.
TR_D: Spectral return period for each of the limit states.

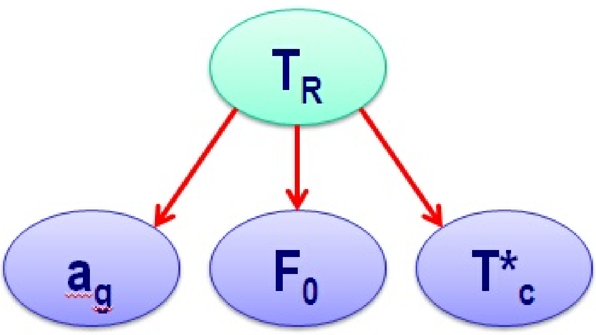
If the box **"Show PGA on rock"** is selected, all the (PGA) accelerations are measured, by convention, on *rigid ground (A)*.
Some data sheets require the calculation of these accelerations on the ground of reference, in this case you will need to multiply the calculated value by the program by the "S" ($S=S_S*S_T$) factor, defined in the spectrum parameters.
The product between acceleration and "S" factor is obtained by removing the check from "Show PGA on rock" button.

The return periods are those presented *in "Annex B"* of the *"Technical Standards"* (the definition of the reference grid). The parameters contained in the tables that define the reference grid can not be extrapolated, if the TRC values are outside the table the **">"** o **"<"** symbols are shown to indicate the exceeded of its upper or lower limit.

By selecting ***"Details ..."*** the following table is shown:

Vulnerabilità Sismica

					TR_C				TR=cost	
	TR_C	TR_D	α_{TR}	PGA_D [m/s2]	PGA_C (TR) [m/s2]	F_0 (TR)	T_C^*(TR)	α_{PGA}(TR)	PGA_C [m/s2]	α_{PGA}
SLV	189	475	0.398	2.56	1.82	2.31	0.32	0.712	1.86	0.728
SLD	71	50	1.420	1.02	1.19	2.32	0.29	1.172	1.19	1.172
SLO	71	30	2.367	0.77	1.19	2.32	0.29	1.544	1.17	1.515



$$\alpha_{PGA} = \frac{PGA_C(T_R)}{PGA_D}$$

$$\alpha_{PGA} = \frac{PGA_C}{PGA_D}$$

This table highlights two different procedures of α_{PGA} calculation which are based on two different theoretical starting assumptions, the choice is up to the designer according to which he considers most appropriate for the specific case. Both theories can be considered valid and reliable.
Remind that the three values for the definition of the (a_g, F_0, T^*_c) spectrum are calculated from the reference grid starting from a given value of the return period T_R.
The value of PGA_C on rigid ground corresponds to the value of a_g.
Method1 (TR=cost): Calculate PGA_C by varying the value of a_g until reaching the condition of the corresponding limit state, maintaining F_0 and T^*_c constants defined on the basis of the T_R value defined by the the seismic spectrum.
Method2 (TR_C): Calculate PGA_C by varying the value of T_R until reaching a tern of (a_g, F_0, T^*_c) values corresponding to the condition of the considered limit state.
The value of a_g thus calculated corresponds to required PGA_C.

Eurocode

Limit state	PGA [m/s2]	α
NC	14,976	7,488
SD	11,232	3,744
DL	3,199	1,600

The table for the evaluation of the *"Seismic Vulnerability"* reports the α parameters derived from homonyms reports for each of the limit states:

α_{PGA}=PGA/a_{gR}

PGA: Limit capacity acceleration for each of the limit states (independent of the seismic spectrum).

Eurocode (NL)

NC

dt 0,02 [cm] <= dm 0,60 [cm]

qu = 0,43

Satisfied check

SD

dt 0,02 [cm] <= dm 0,45 [cm]

Satisfied check

DL

Sd 0,02 [cm] <= d*y 0,04 [cm]

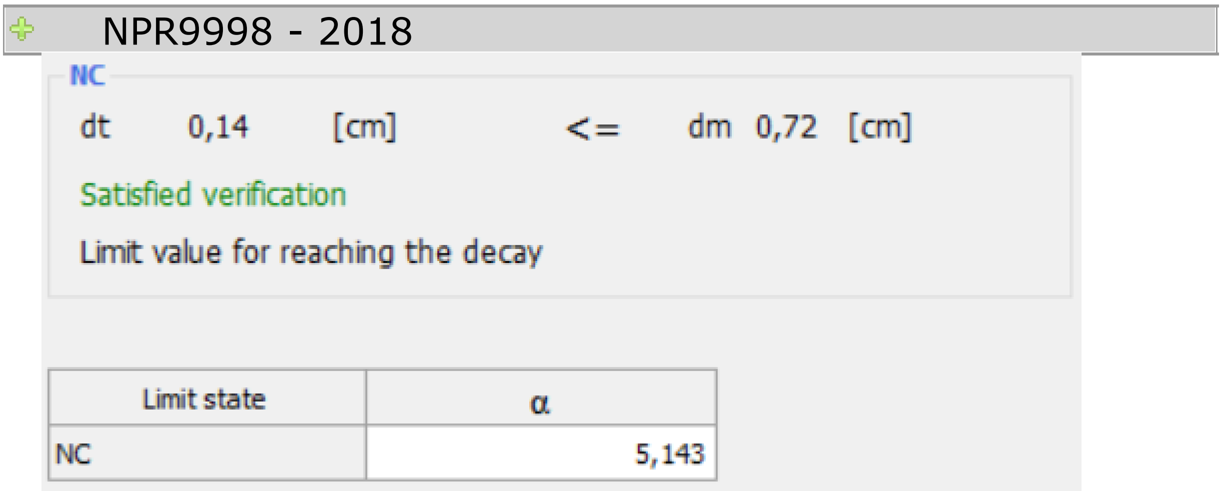
Satisfied check

Limit state	α	
NC	30,000	} dm/dt
SD	22,500	
DL	2,314	d*y/Sd

The table for the evaluation of the *"Seismic Vulnerability"* reports the α parameters derived from homonyms ratios for each of the limit states derived by the ratio between the displacements.
Given the formulation of the spectrum defined in the EUROCODE (NL), it is clear that it is not possible to identify a vulnerability index on the basis of spectral accelerations will therefore be referred to vulnerabilities calculated as the ratio between the displacements.

In the area named "Analysis Parameters" appear the following factors:
T*: Period of the equivalent system
m*: mass of the equivalent system

W: total mass until the achievement of a tern of values
Available ductility: the ratio between the ultimate displacement and the elastic limit displacement
Γ: modal participation factor
F^*_y: plasticization strength of the equivalent system
d^*_y: plasticization displacement of the equivalent system
d^*_u: ultimate displacement of the equivalent system



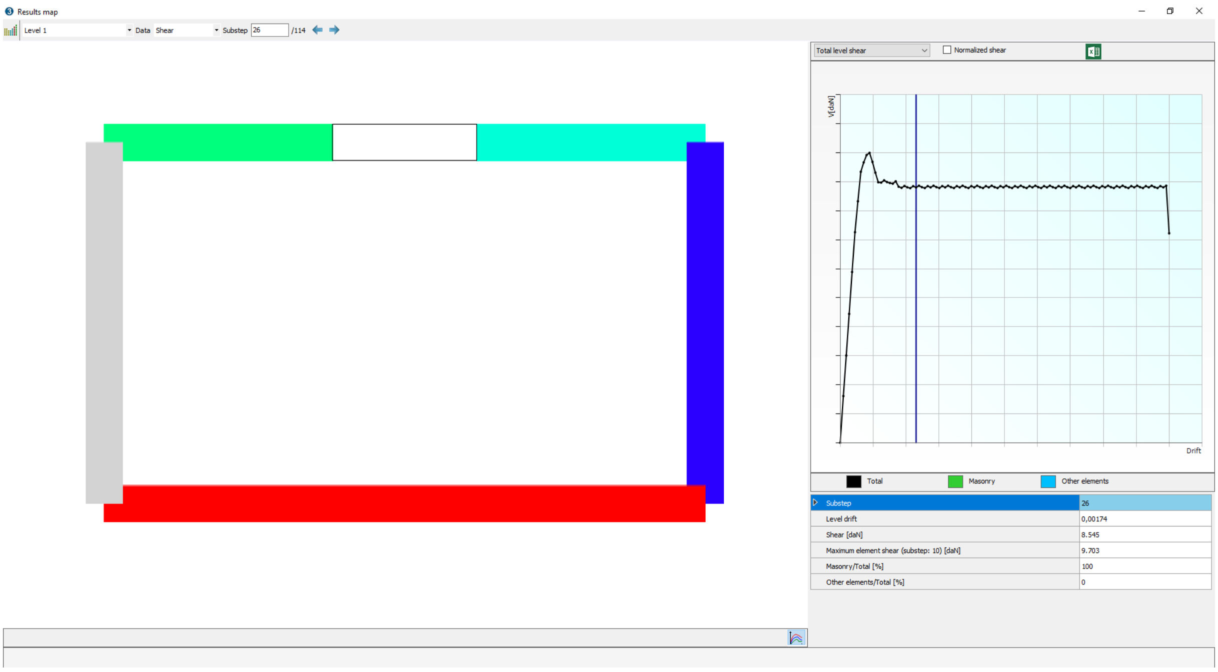
Limit state	α
NC	5,143

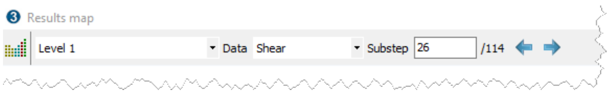
The table for the evaluation of the *"Seismic Vulnerability"* reports the α parameter derived from the homonyms ratio of the limit states derived by the ratio between the displacements.

In the area named "Analysis Parameters" appear the following factors:
T*: period of the idealized equivalent SDOF system
m*: mass of the equivalent system
W: total weight
M: total mass
m*/M: ratio between the mass of the equivalent system and the total mass
Γ: transformation factor
F^*_y: ultimate strength of the idealized system (SDOF)
d^*_y: yield displacement of the idealized SDOF
d^*_m: ultimate displacement of the idealized system (SDOF)

10.4.2.2.5 Colors map

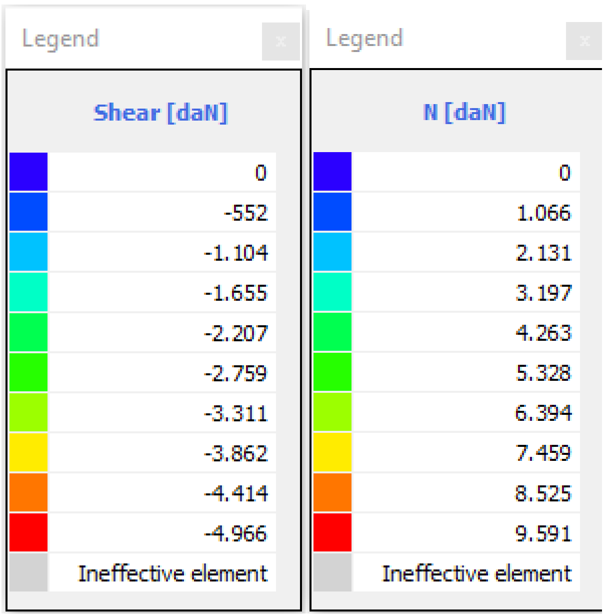
 Within the results environment of the pushover analysis, by clicking on this icon, it is possible to view the building's colors map relating to the results of the carried out seismic analysis.



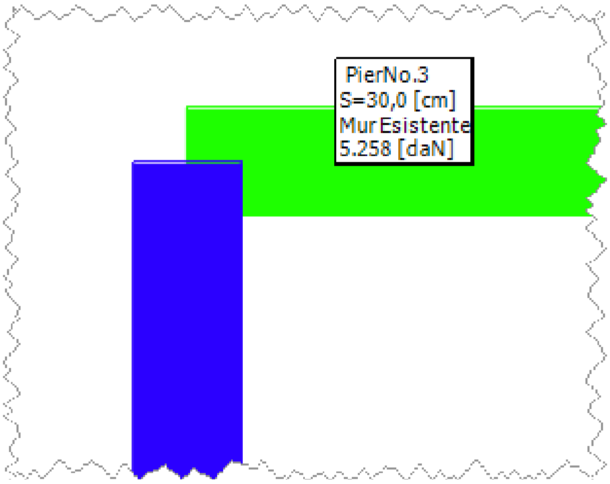


Using the "data" drop-down menu in the upper left corner it is possible to view the map relating to normal stress, tensions or plan shear.
It is possible also to filter the maps by level.

In the case of the plan shear, it is also possible to manually edit the underpass that the user want to analyze and move backwards and forwards through the arrows.

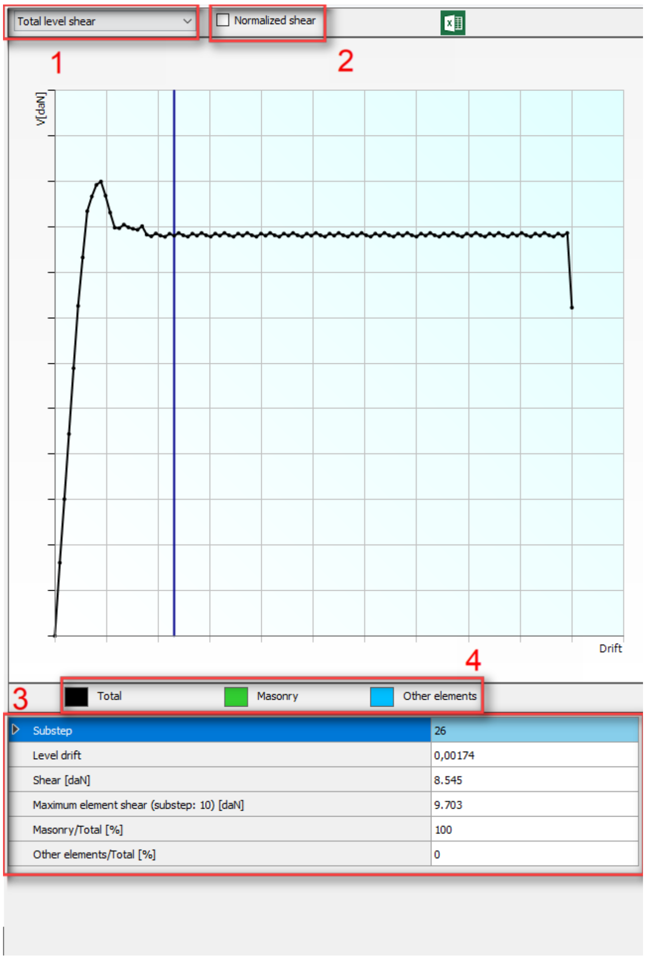


Through the "colors legend" command the user can see the legend associated with the colors map.

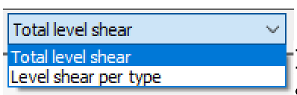


Through the tooltips associated with each element, it is possible to analyze the entity of the stresses acting.

In the right part of the window there are the diagrams related to the plan shear:



1

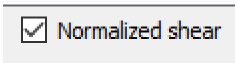


It is possible to choose between the "total plan shear" or "plan shear by typology" option

a) total plan shear: allows to view the total shear diagram of all the elements present in a given plan

b) plan shear by typology: allows the user to view, on two distinct curves, the plan shear relating to the masonry elements and that relating to the non-wall elements.

2



Option that allows, for each underpass, to view the value of the plan shear compared to the overall total shear

3

Substep	26
Level drift	0,00174
Shear [daN]	8.545
Maximum element shear (substep: 10) [daN]	9.703
Masonry/Total [%]	100
Other elements/Total [%]	0

Underpass: indicates the number of the analyzed underpass
Plan drift: indicates the value of the plan drift associated with the underpass taken into consideration (corresponds with the abscissa value of the diagram)
Shear: Indicates the ratio between the plan shear value and the overall shear value with the same underpass
Maximum shear of the element : maximum shear value (and related underpass)
Masonry/Total [%]: Percentage of shear attributable to masonry elements
Other elements/Total [%]: Shear percentage attributable to non-masonry elements

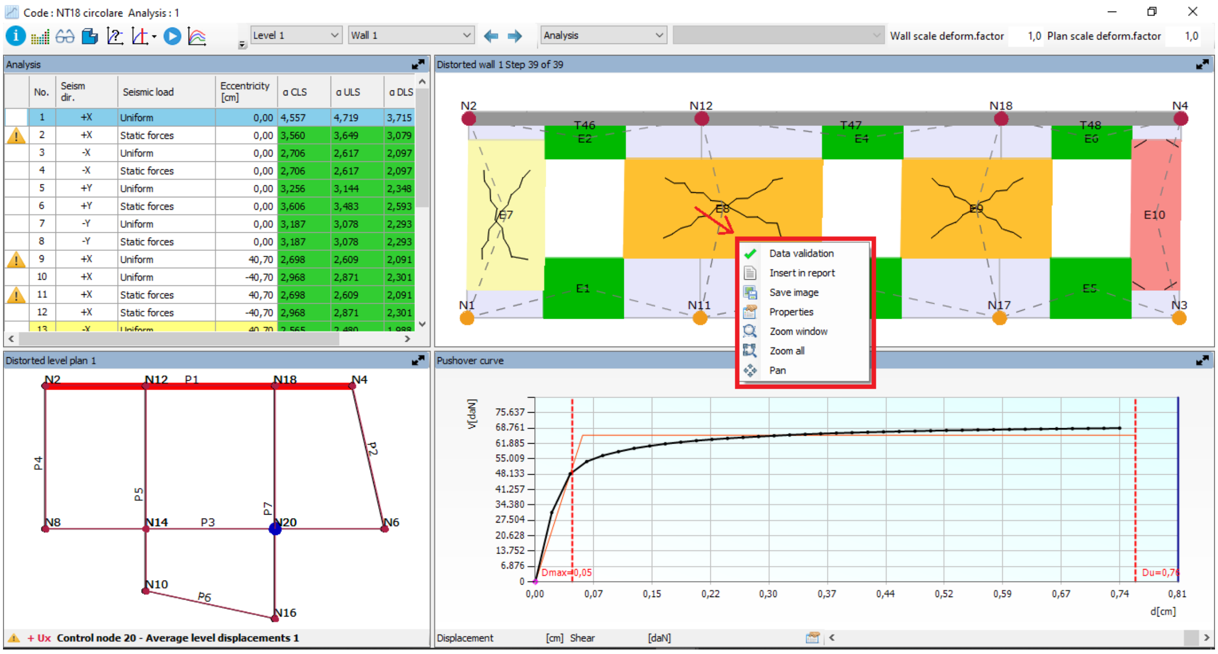
4

Colors legend associated with the diagram of the plan shear

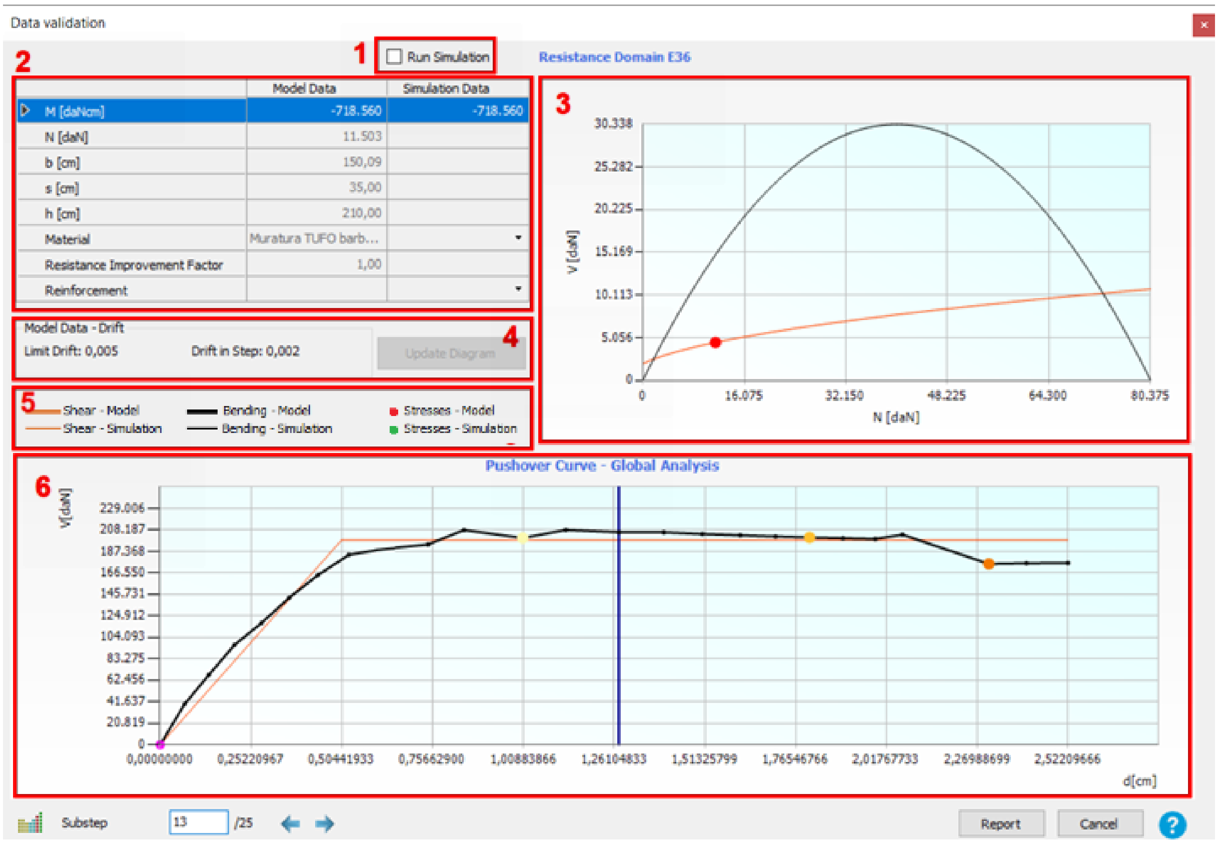
10.4.2.2.6 Data validation

This validation tool allows the user to evaluate the stress state of the masonry walls in a quick, simple and intuitive way, graphically displaying, for each step of the non-linear seismic analysis, the drift value, the resistance domain and the mechanism of collapse.
This function is designed both to help the user understand the behavior of the structure, but above all to help and guide him in the phase of validating the results provided by the software.
For this purpose, within the results window, at each point of the global pushover curve, for each masonry wall it is possible to view the state (first yielding, incipient faliure, breaking) in which the element is located. This allows the designer to have a transversal view of the behavior, and therefore of the influence, of the individual wall panels within the structure.
In the event that improvements need to be made to an existing building (reinforcements with FRP and FRCM), using this validation tool, the designer can easily understand on which wall panels it is appropriate to intervene, or if the interventions are effective or if they can be optimized.

From the window containing the results of the pushover analysis, by right clicking on the wall of interest, the following drop-down menu opens:



Once you have clicked on the "data validation" option, the following window appears:



1 It allows to start a simulation by entering the simulation data in the third column of the table at the top left, and click on “Update diagram”.

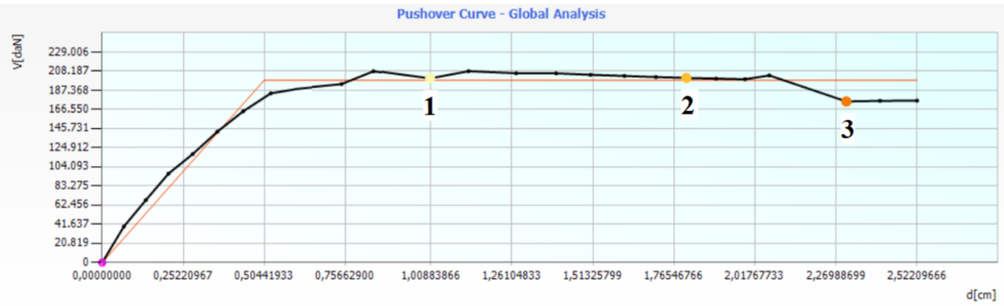
2 The table contains in the second column the geometric and mechanical data of the masonry wall in question, while in the third column, which can be activated by checking the “Run simulation” box, the respective parameters used to determine the simulated resistance domains can be entered.

3 Resistance domains diagram.

4 Values of the element's limit drift (depends on the breaking mechanism) and the drift to the pace. As is known, the element reaches the break when the drift at the step reaches the drift limit.

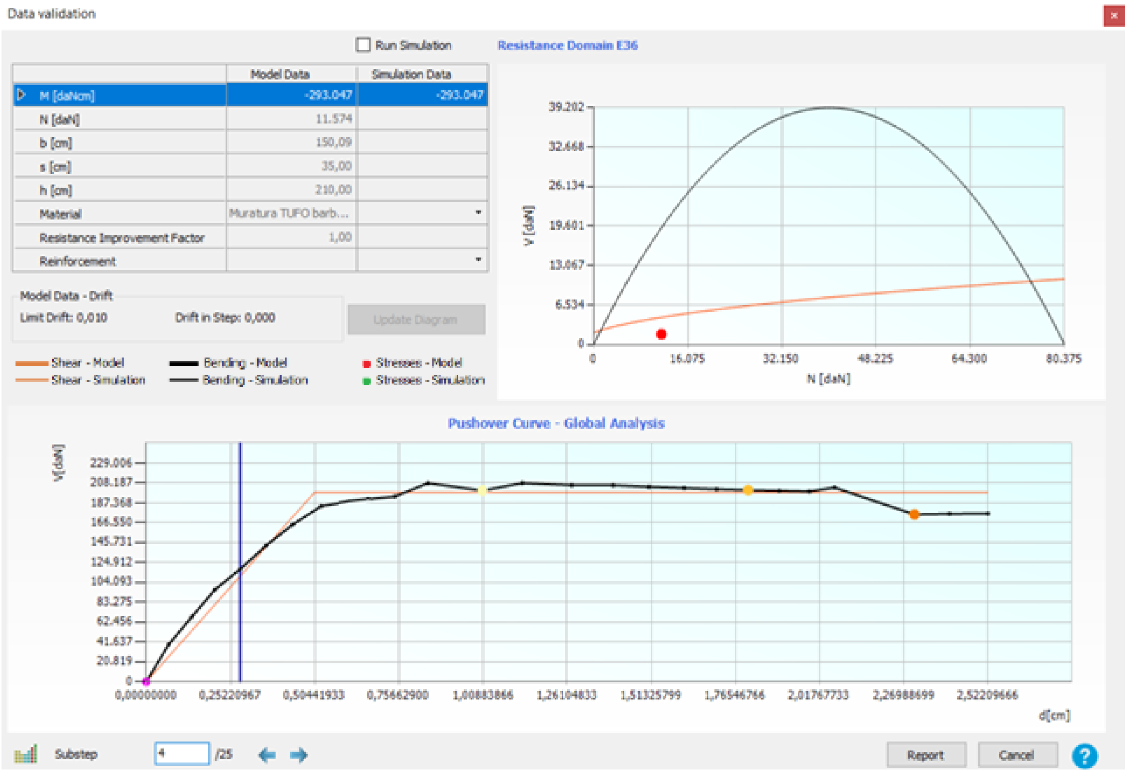
5 Legend related to the diagrams of the resistance domains.
The orange line indicates the shear resistance domain, while the black line indicates the bending resistance domain. The solid lines represent the resistance domain evaluated for the geometric and mechanical characteristics of the modeled masonry wall, while the dashed lines indicate the simulated resistance domains. The two circular indicators, red and green, respectively indicate the stress state of the element resulting from the pushover analysis and the simulated stress state.

6 Global pushover curve on which there are colored indicators that indicate the calculation step in which the state change of the wall occurs (beginning of plastification, incipient breaking, breaking).

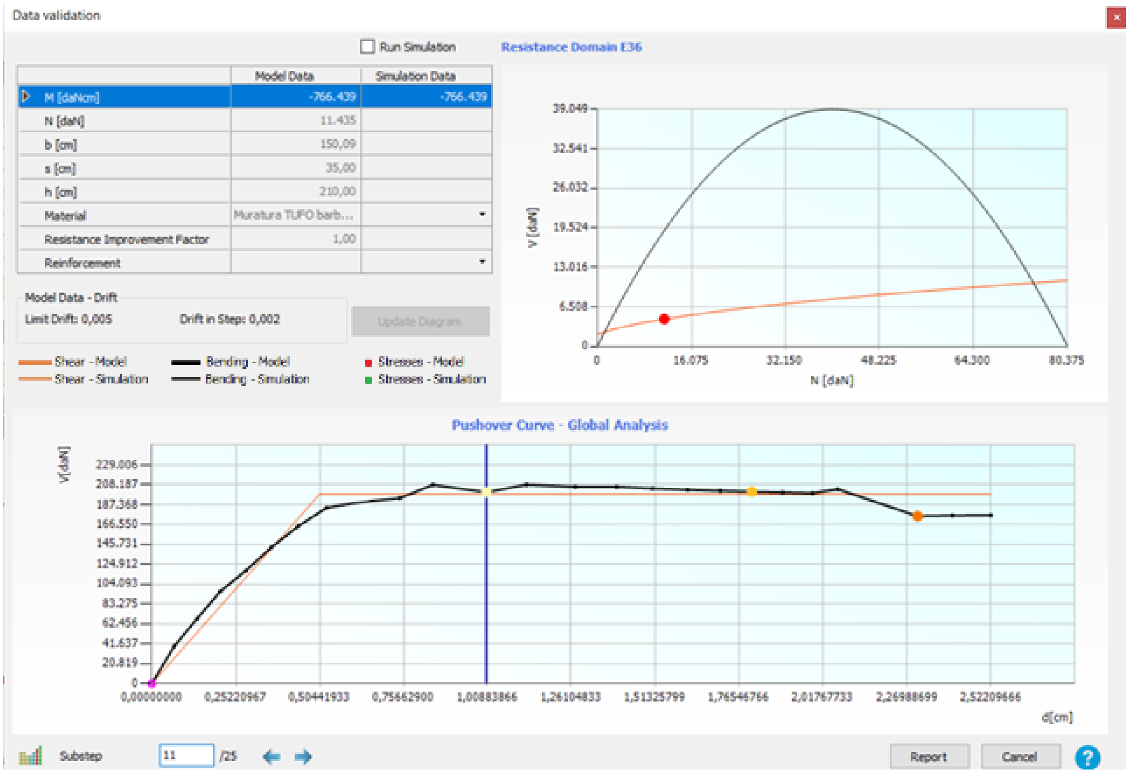


The colored indicators, which designate the state in which the wall under examination is located, allow us to quickly understand that the wall under examination exhibits an elastic behavior until point 1 is reached, step in which the first plastification by shear takes place. Moving further along the curve, the wall changes state in correspondence with the step identified by point 2, that is when he reaches 75% of the drift limit. In this step, in fact, the color legend indicates that the element is in a state of incipient breakage due to shear, which occurs at the step identified in point 3, when the drift limit is reached. As can be easily understood from this result, it can be deduced that the wall in question is well sized and effectively contributes to the strength of the building as its breakage is close to the collapse of the entire building, coinciding with the last point of the curve.

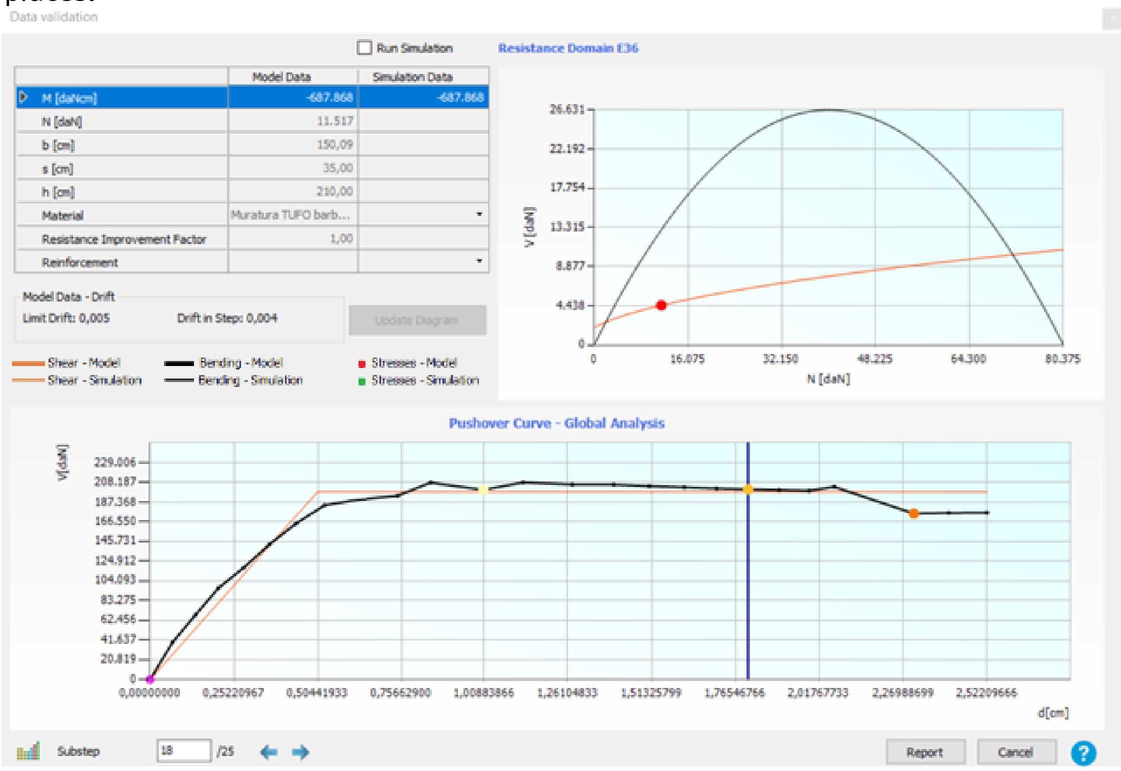
For each point of the pushover curve, it is also possible to visualize the resistance domains of the masonry wall. As already said, in the stretch of curve between step 0 and point 1, the masonry wall is in the elastic phase, in fact, from the resistance domain we can observe that the point indicating the state of stress at the step (red indicator), is placed within the domain.



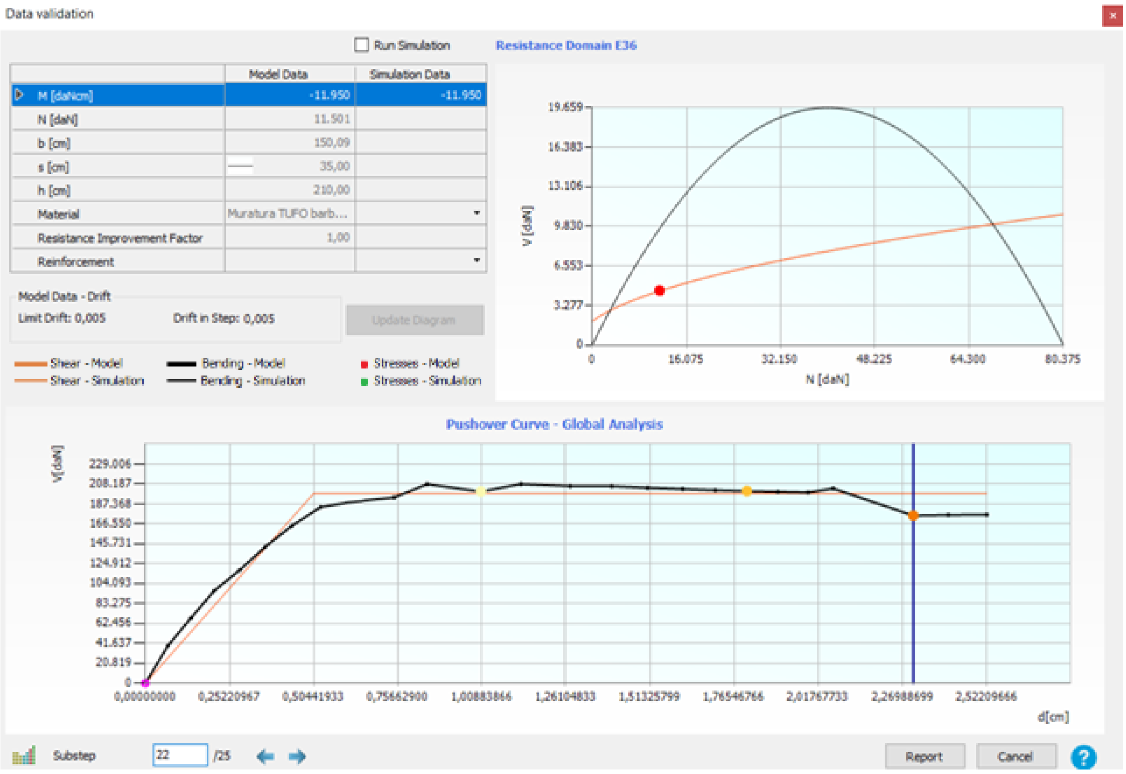
Positioning instead in correspondence with the step of the pushover curve where the first yielding occurs (point 1), from the graph it can be observed that the stressing point (red indicator) is exactly on the curve corresponding to the shear resistance criteria of the masonry wall (curve orange), but if we observe the drift value, it is lower than the breaking value (drift in step = 0.001; drift limit = 0.005).



By creating the diagrams of the resistance domains for the step of the pushover curve corresponding to point 2 (incipient collapse due to shear), it is observed that the red indicator always coincides with the orange curve, while the drift at the step is equal to 0.004 or coincides with 75% of the drift limit rounded to three decimal places.

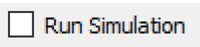


Finally, positioning in correspondence with the step of the curve in which the breaking of the wall occurs, we observe that the red indicator always coincides with the orange curve, while the drift at the step is equal to 0.005 or coincides with the drift limit.



At the bottom left, using the following buttons, you can access the damage status legend and move between the various points of the pushover curve.

Run simulation



This function offers the possibility of simulating the resistance domain of a masonry wall in real time, simply by varying a few parameters that define its geometric and mechanical characteristics, or those of any reinforcement applied, without having to modify anything in the global model and without having to re-calculate.

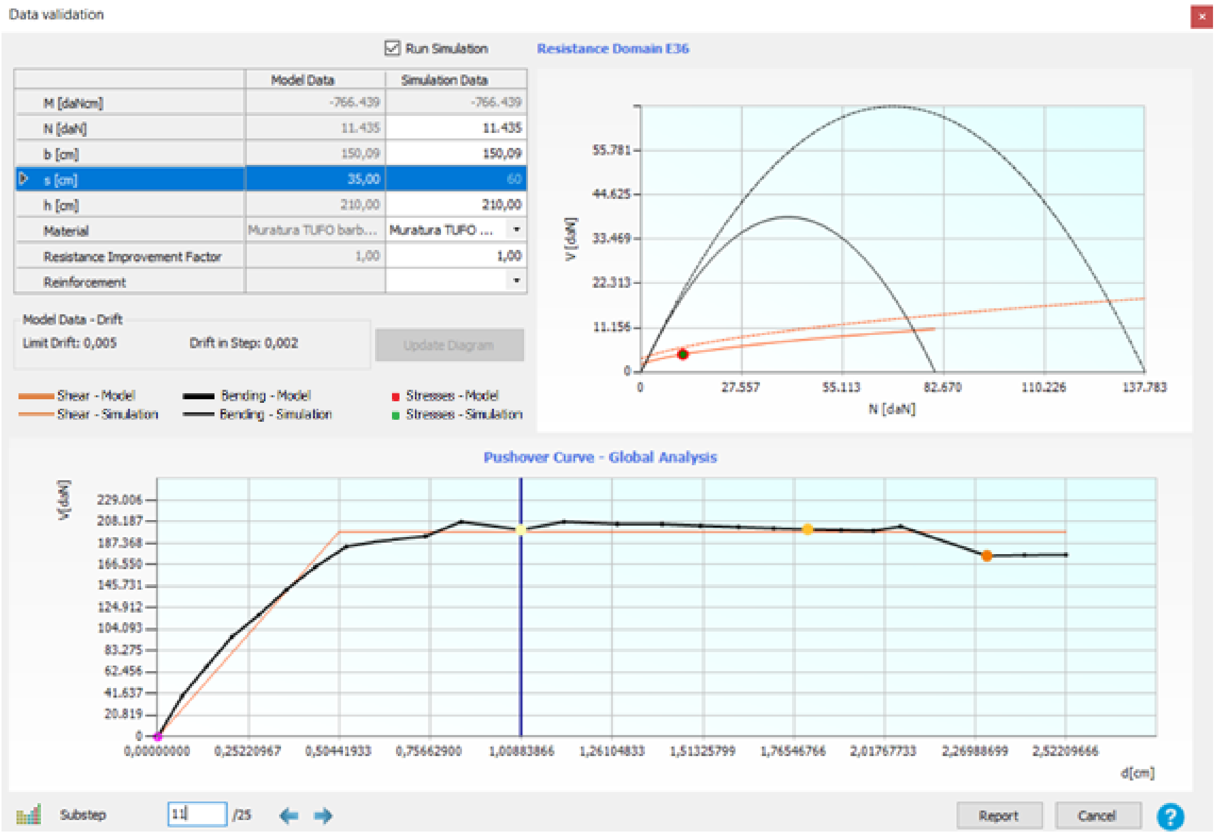
If you want to start a simulation, you need to select the step of interest of the pushover curve, check the "Run simulation" box, enter the simulation data in the third column of the table at the top left, and click on "Calculate domain" .

Example

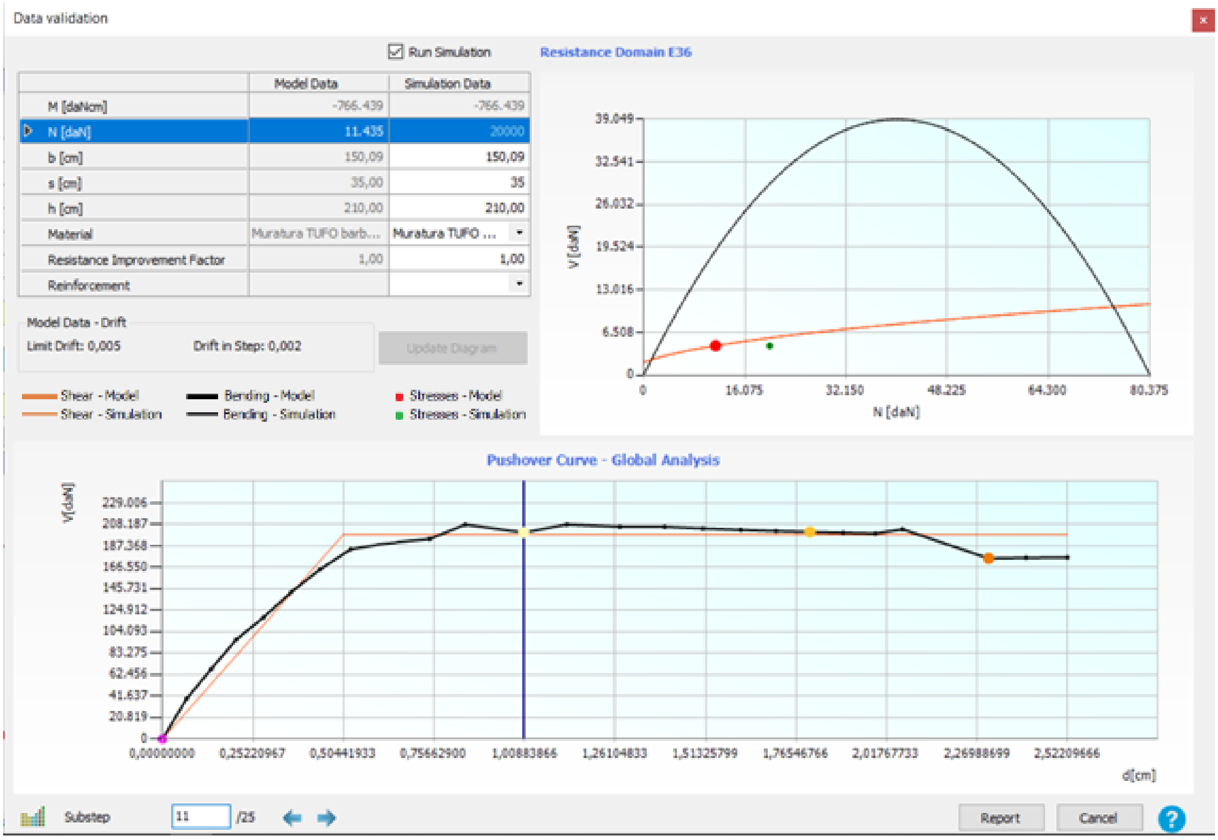
In the following example, the simulation is started by positioning at point 1 of the pushover curve and increasing the thickness of the masonry.

As it can be seen from the following figure, the increase in thickness of the masonry wall has the effect of increasing its resistance, and therefore, under the same stress (the red indicator coincides with the green one due to the fact that in the simulation the normal stress has not been changed), in the simulation the element is in the elastic phase, while from the data entered in the model it would be in the plastic phase.

Ultimately, if the masonry wall had the geometric characteristics introduced with the simulation, it would be able to withstand a greater shear force.



A similar result could be obtained by simulating the case in which the normal stress is greater, for example by increasing the weight of the floor that rests on this wall as a consequence of a reinforcement intervention on a floor which on the one hand improves the performance of the slab itself, and on the other hand it increases the load on the walls. In this case the curves of the domains remain unchanged as they are a function only of the geometric and mechanical parameters of the masonry and of the possible presence of the reinforcement, while the green indicator, which indicates the state of simulated stress, has moved to the right, re-entering the domain.



What is described above should make us reflect on all those cases in which we try to make an improvement on the structure by lightening the floors.
This could bring the solicitor closer to the origin of the domain and consequently expose them outside the domain itself, creating a deterioration rather than the desired improvement.
These observations make us understand that the most efficient improvement choices should be made after having examined the resistance domain to clarify on which area it is good to act.

Quantifying the extent of improvement of a reinforcement

The application of a reinforcement (reinforced masonry, FRP or FRCM) acts on two different ways:

- Resistance
- Ductility (displacements)

The concept of "improvement" is therefore produced by the union of the two characteristics listed above, which can occur simultaneously or one of the two can have a greater weight than the other.
Strength affects plasticity, ductility affects rupture; ductility itself can be indirectly influenced by resistance by postponing the plasticity of the element due to greater resistance.
Modifying resistance and / or ductility alters the stiffness of the building as a whole and consequently the load that can potentially act on the element in question.
According to the theory of resistance domains, the performance of the element depends on the vertical load acting on it. It therefore follows a significant alteration of the shape of the resistance domains and of the element's ultimate ductility.
It is therefore impossible to numerically quantify the extent of the improvement

resulting from the intervention as load, strength and ductility are linked to each other and cannot be schematized in a simplified way through a simple improvement factor of the compressive strength.
To define the benefits produced by an intervention of this nature, it is therefore necessary to simultaneously examine both the changes in the resistance domain (and not just the compressive capacity) and the increase in ductility provided by the reinforcement itself.

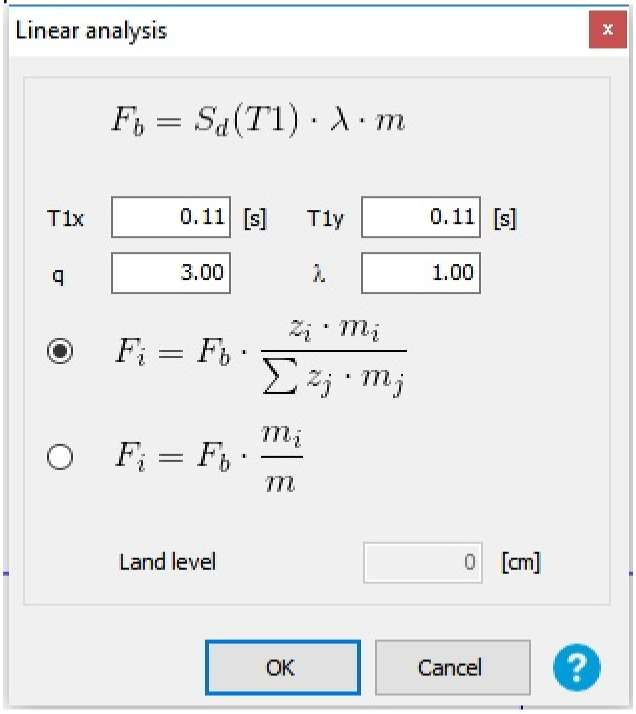
In some cases it may happen that the result obtained by recalculating the global analysis does not reflect what was previously simulated through the change of some characteristics.
In fact, when the simulation is carried out on an element, it is considered as an isolated element exempt from the interactions with the rest of the structure that will come into play only following the global calculation.
The modification of geometric or mechanical characteristics or the addition of any reinforcements have an influence not only on the element itself but on the behavior of the entire structure, modifying its configuration in terms of ductility, stiffness and load.
It is therefore important to consider the simulation result as approximate (compared to what would be obtained by redoing the global analysis of the structure) and to use the tool as an aid to the designer who otherwise, faced with an element that is not verified, would have to make changes to the structure based on more or less vague criteria, sometimes modifying the structure in an ineffective way, and spending time redoing the mesh each time and recomputing the seismic analysis.

10.4.3 Linear analysis

10.4.3.1 Calculation

> *Linear analysis Calculation* (select the item from the drop-down)

The calculation window is shown immediately with the request of the necessary parameters.



The seismic base shear force Fb, for each horizontal direction in which the building is analysed, shall be determined using the following expression:

$$F_b = S_d(T1) \cdot \lambda \cdot m$$ **EN 1998-1:2005 §4.3.3.2.2(4.5)**

where:

Sd(T1) : is the ordinate of the design spectrum at period T1;

T1 : is the fundamental period of vibration of the building for lateral motion in the direction considered, this value can be obtained using the simplified formula [**EN 1998-1:2005 §4.3.3.2.2(4.5)**] or through a modal analysis ;

m: is the total mass of the building, above the foundation or above the top of a rigid basement;

λ: is the correction factor, the value of which is equal to: λ=0.85 if T1≤2Tc and the building has more than two storeys, or λ=1

The seismic action effects shall be determined by applying, to the two planar models, horizontal forces Fi to all storeys.

$$F_i = F_b \cdot \frac{z_i \cdot m_i}{\sum z_j \cdot m_j}$$ **EN 1998-1:2005 §4.3.3.2.3(3)**

$$F_i = F_b \cdot \frac{m_i}{m}$$ **Alternative formulation normatively not contemplated**

Dove:

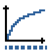
Fi: is the horizontal force acting on storey i;

Fb: is the seismic base shear due to the seismic action

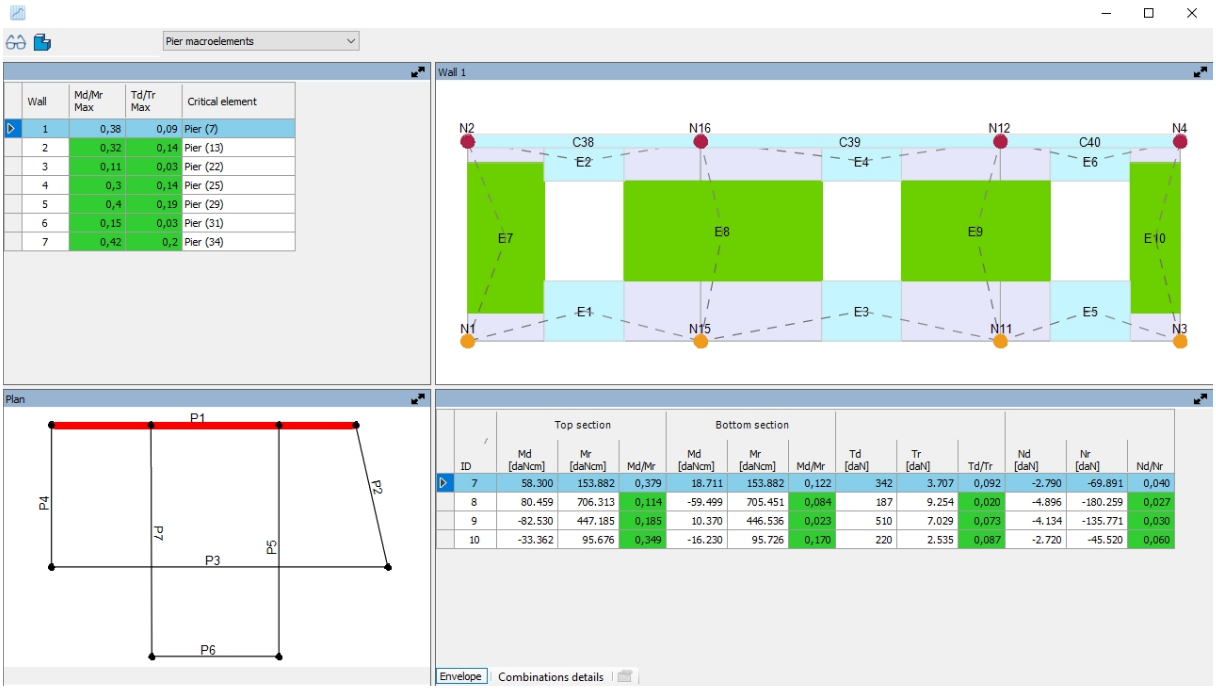
zi, zj: are the heights of the masses mi mj

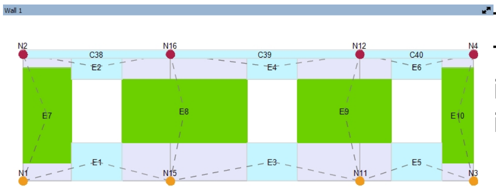
Pressing [OK] starts the calculation procedure.

10.4.3.2 Results

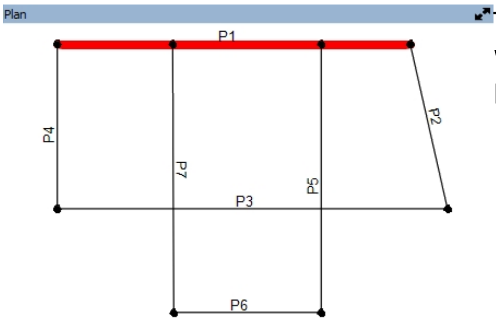
 ***>Verification of linear analysis*** (select the item from the drop-down)

The following screen appears on the screen:





The wall mesh appears at the top right.
The elements that go beyond the verification appear in green color and those that do not exceed it appear in different color.



The plan view appears at the bottom left, highlighted with a stretch we often see the wall shown in the previous elevation.

	Wall	Md/Mr Max	Td/Tr Max	Critical element
▷	1	0,38	0,09	Pier (7)
	2	0,32	0,14	Pier (13)
	3	0,11	0,03	Pier (22)
	4	0,3	0,14	Pier (25)
	5	0,4	0,19	Pier (29)
	6	0,15	0,03	Pier (31)
	7	0,42	0,2	Pier (34)

A table shows the list of walls at the top left, for each one the most burdensome safety factor among all the elements present in the wall is shown.

In the lower right corner a table shows the verification detail for each element of the wall

ID	Top section Md [daNcm]	Mr [daNcm]	Md/Mr	Bottom section Md [daNcm]	Mr [daNcm]	Md/Mr	Td [daN]	Tr [daN]	Td/Tr	Nd [daN]	Nr [daN]	Nd/Nr
7	58.300	153.882	0,379	18.711	153.882	0,122	342	3.707	0,092	-2.790	-69.891	0,040
8	80.459	706.313	0,114	-59.499	705.451	0,084	187	9.254	0,020	-4.896	-180.259	0,027
9	-82.530	447.185	0,185	10.370	446.536	0,023	510	7.029	0,073	-4.134	-135.771	0,030
10	-33.362	95.676	0,349	-16.230	95.726	0,170	220	2.535	0,087	-2.720	-45.520	0,060

The values shown in the table for each element constitute the most difficult conditions produced by the envelope of all calculation combinations. The envelope configuration is shown when the corresponding label is selected at the bottom of the screen.

Envelope | Combinations details

Selecting "Combination details" shows the combination table for the element selected in the "Envelope" table.

Pier: 7

Comb.	Desc.	Top section Md [daNcm]	Mr [daNcm]	Md/Mr	Bottom section Md [daNcm]	Mr [daNcm]	Md/Mr	Td [daN]	Tr [daN]	Td/Tr	Nd [daN]	Nr [daN]	Nd/Nr
1	+Ex+0.3Ey	58.300	153.882	0,379	18.711	153.882	0,122	342	3.707	0,092	-2.787	-69.891	0,040
2	+Ex-0.3Ey	58.239	153.761	0,379	18.691	153.761	0,122	342	3.706	0,092	-2.785	-69.891	0,040
3	-Ex+0.3Ey	57.677	153.888	0,375	17.871	153.888	0,116	336	3.707	0,091	-2.787	-69.891	0,040
4	-Ex-0.3Ey	57.617	153.767	0,375	17.851	153.767	0,116	335	3.706	0,090	-2.785	-69.891	0,040
5	+0.3Ex+Ey	58.153	154.025	0,378	18.440	154.025	0,120	340	3.707	0,092	-2.790	-69.891	0,040
6	+0.3Ex-Ey	57.951	153.622	0,377	18.374	153.622	0,120	339	3.706	0,092	-2.782	-69.891	0,040
7	-0.3Ex+Ey	57.966	154.026	0,376	18.188	154.026	0,118	338	3.707	0,091	-2.790	-69.891	0,040
8	-0.3Ex-Ey	57.764	153.624	0,376	18.122	153.624	0,118	337	3.706	0,091	-2.783	-69.891	0,040

Envelope | Combinations details

For elements such as columns and R.C. walls in which there is a bidirectional behavior, the "detail" button calls a window with the stresses decomposed for the two directions.

Envelope | Combinations details

Pilastro C.A.

ID	Sezione superiore Mdx [daNcm]	Mdy [daNcm]	Mrx [daNcm]	Mry [daNcm]	Md/Mr	Sezione inferiore Mdx [daNcm]	Mdy [daNcm]	Mrx [daNcm]	Mry [daNcm]	Md/Mr	Tdx [daN]	Tdy [daN]	Trx [daN]	Try [daN]	Td/Tr	Nd [daN]	Ndr [daN]	Nd/Nr
55	0	-12'403	693'030	693'030	0.02	3'864	-13'250	693'030	693'030	0.02	855	129	355'658	355'658	0.00	-835	-214'232	0.00

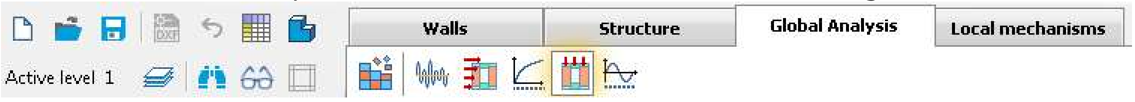
Please note : Given the formulation of the constitutive bonds of the masonry material, if a masonry element possesses Nd> 0 (traction) or Nd / Nr> 1 (not exceeded verification with simple compression) it will not be possible to calculate the resistant moment Mr which will then be equal to zero and the ratio Md/Mr = u/d (undefined).

10.5 Static Analysis

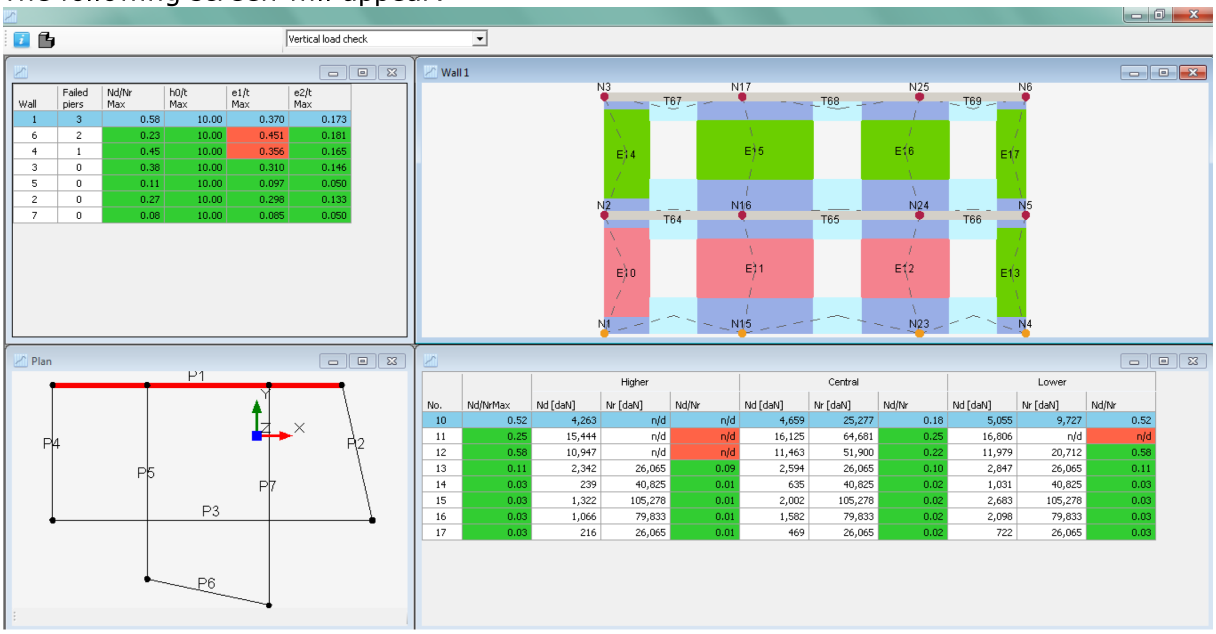
This is a module which performs static checks on the structure, according to the code in effect.
The program uses the meshes already created to perform the non-linear analysis, adapting the equivalent frame theory to perform the static checks in the linear field.

All checks are carried out by means of combinations of static loads, based on the provisions of the current legislation in force at the time of the calculation.

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

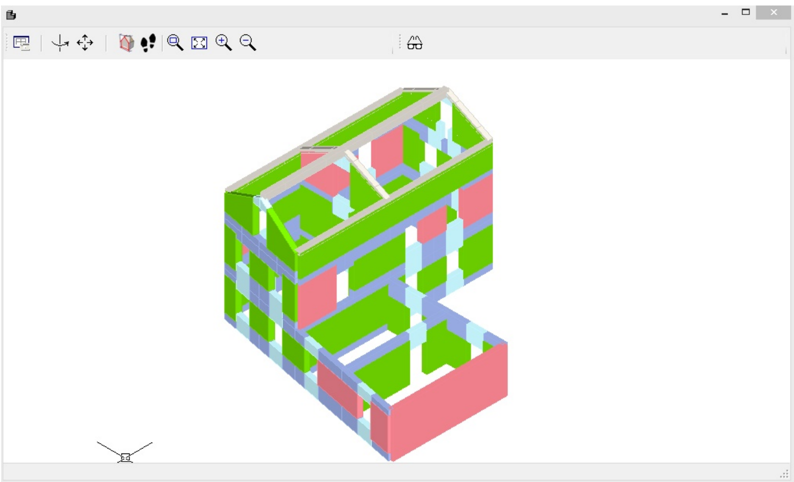


The following screen will appear:



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

If the user wishes, axonometric visualization can be used to find the elements that did not pass the check.



In order to help the user interpret the results, some of the tables offer the possibility of reordering the rows according to the column characteristics.

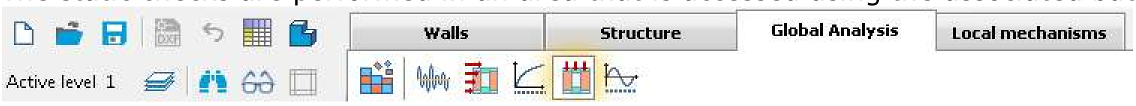
10.5.1 Italian Standart NTC08

Let's present below the checks that are carried out:

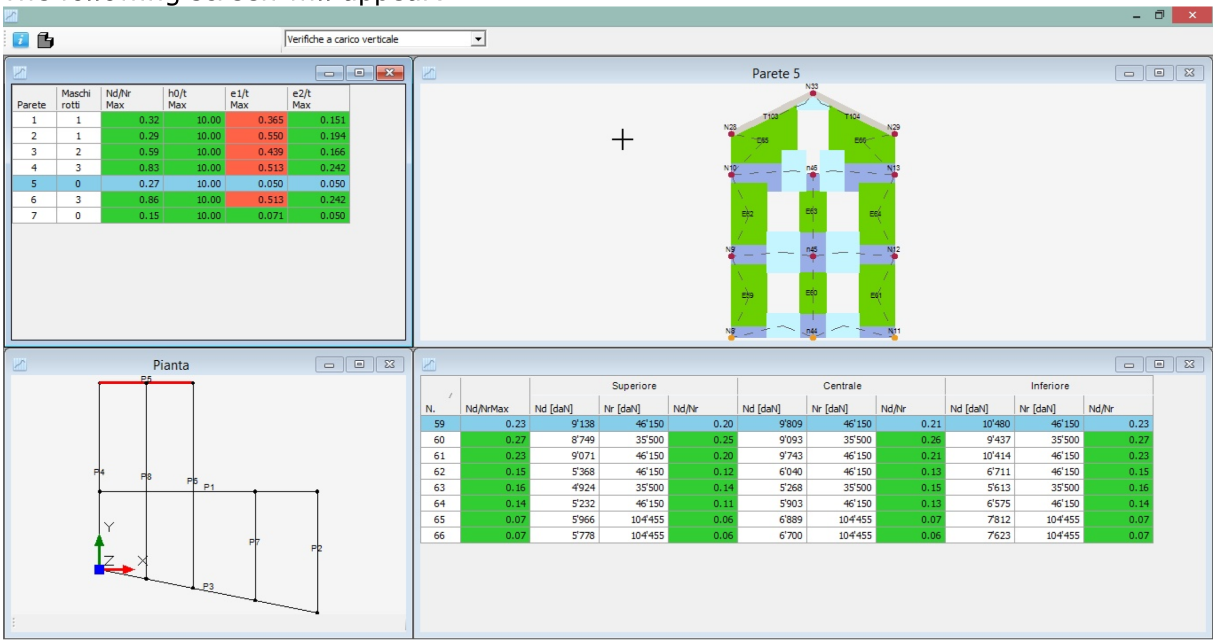
Slenderness check:	**$h0/t \leq 20$**
	h0: effective length of the wall equal to ρ•h *t: thickness of the wall*
Load eccentricity check:	**$e_1/t \leq 0.33$** **$e_2/t \leq 0.33$**
	$e_1 = \| e_s \| + \| e_a \|$; $e_2 = \frac{e_1}{2} + \| e_v \|$ *es : total eccentricity of the vertical loads;* *ea : eccentricity due to execution tolerance;* *ev : eccentricity due to wind;*
Vertical loads check:	**$N_d \leq \Phi\ f_d\ A$**
	***Nd**: vertical load at the base of the wall;* ***A**: area of the horizontal section of the wall, after subtracting the openings;* ***fd**: computation resistance of the masonry;* ***Φ**: coefficient for wall resistance reduction*

If the active standards are the "Technical Standards for Construction 2008", the multiplication coefficients of the loads are editable using the command "Static calculation parameters"

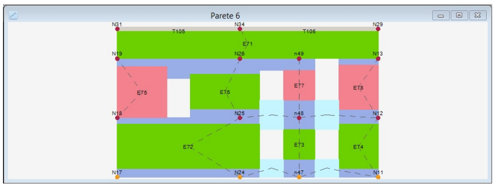
The static checks are performed in an area that is accessed using the associated button.



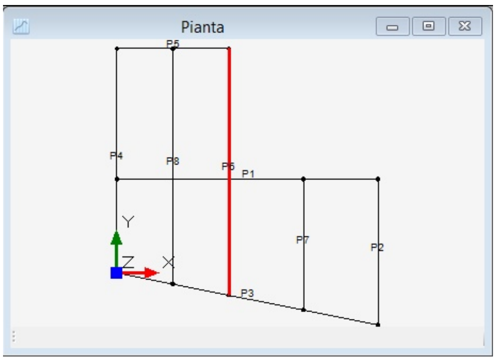
The following screen will appear:



Let's describe it in detail.



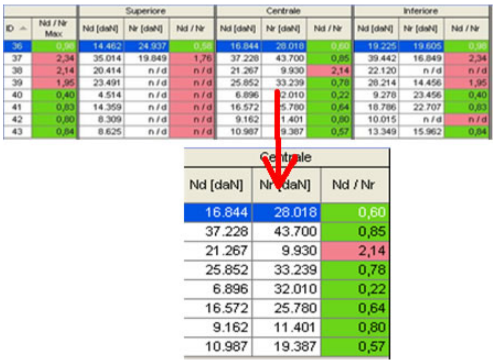
In the upper right side appears the wall mesh.
In this case, does not exist the legend with colors indicating different phases of damage.
Elements that passed the check appear in green and in different color those that do not exceed the check.



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

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

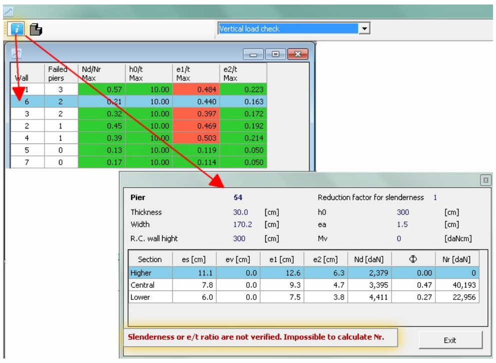
On the upper left side there is a list of the walls in the model, with the number of elements that did not pass the check and the values associated with the individual checks. The values found in the table are for the wall elements examined in which the limit values are the most restrictive of all the piers. Clicking on the line of a wall (highlighting it in blue) brings that wall to the view on the right side.



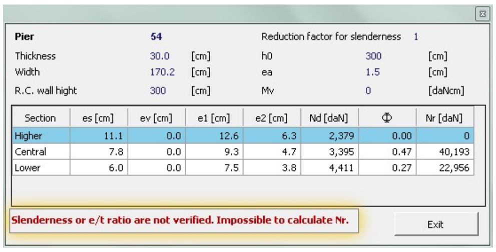
Nd [daN]	Nr [daN]	Nd / Nr
16.844	28.018	0,60
37.228	43.700	0,85
21.267	9.930	2,14
25.852	33.239	0,78
6.896	32.010	0,22
16.572	25.780	0,64
9.162	11.401	0,80
10.987	19.387	0,57

At the lower right side, the elements detail window is shown for the selected wall.
For each masonry element, the checks are performed for three different sections (higher, central, lower).
For each section the value for normal forces strain is shown (Nd: computed based on the masses and the combinations of the loads) and the normal resistant strain (Nr =**Φ** f_d **A**).

The check is satisfied if the ratio Nd/Nr≤ 1. In this case, the corresponding cell appears in green.
In some cases, as shown in the example, Nr cannot be calculated (n/d: not defined). This happens when the slenderness or eccentricity checks are not satisfactory.



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



Pier 54 — Reduction factor for slenderness 1

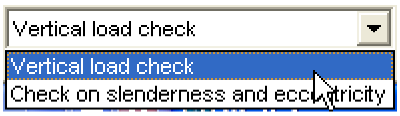
Thickness	30.0	[cm]	h0	300	[cm]
Width	170.2	[cm]	ea	1.5	[cm]
R.C. wall hight	300	[cm]	Mv	0	[daNcm]

Section	es [cm]	ev [cm]	e1 [cm]	e2 [cm]	Nd [daN]	Φ	Nr [daN]
Higher	11.1	0.0	12.6	6.3	2,379	0.00	0
Central	7.8	0.0	9.3	4.7	3,395	0.47	40,193
Lower	6.0	0.0	7.5	3.8	4,411	0.27	22,956

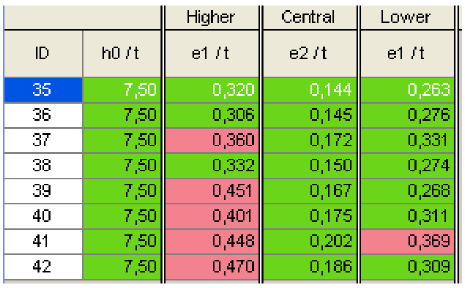
Slenderness or e/t ratio are not verified. Impossible to calculate Nr.

Exit

The window shows all the details of the parameters used in the computation of the various check coefficients.
The text in red near the bottom gives relative informations to conditions where the check was not satisfied.
This window can remain open and be moved to any point of the drawing area while working (floating window). This gives to the user the possibility to select various elements in different wall and still have the details for each individual check visible.



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



		Higher	Central	Lower
ID	h0 /t	e1 /t	e2 /t	e1 /t
35	7,50	0,320	0,144	0,263
36	7,50	0,306	0,145	0,276
37	7,50	0,360	0,172	0,331
38	7,50	0,332	0,150	0,274
39	7,50	0,451	0,167	0,268
40	7,50	0,401	0,175	0,311
41	7,50	0,448	0,202	0,369
42	7,50	0,470	0,186	0,309

Here we see the check details for slenderness and eccentricity. The green values indicate that the check was passed.

In order to help the user in the results interpretations, some of the tables offer the possibility of reordering the rows according to the column characteristics.

Wall check summary table

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

This table is ordered based on the wall identifiers. The type of orientation is clarified by the arrow found at the top of the column.

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

Clicking on the appropriate column will reorder the values according to the characteristics chosen. In the figure at the side, the table is ordered based on the number of failed elements.

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

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

Compression check details table

ID ▲	Nd / Nr Max	Nd [daN]	Nr [daN]	Nd / Nr	Nd [daN]	Nr [daN]	Nd / Nr	Nd [daN]	Nr [daN]	Nd / Nr
1	0,38	24.834	111.564	0,22	33.733	111.564	0,30	42.633	111.564	0,38
4	0,45	7.160	22.917	0,31	8.743	22.917	0,38	10.327	22.917	0,45
5	0,05	-2.663	29.147	-0,09	-647	29.147	-0,02	1.369	29.147	0,05
6	0,18	1.985	44.626	0,04	5.071	44.626	0,11	8.156	44.626	0,18
7	0,21	2.998	44.626	0,07	6.083	44.626	0,14	9.168	44.626	0,21

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

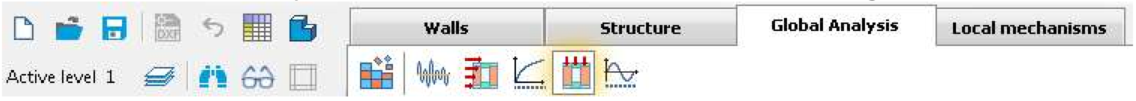
The current formulation that leads to the calculation of the eccentricity es2 considers in the calculation of the N2 (load coming from the overlying floors) in addition to its contribution also the respective d2 eccentricity value for all of the levels higher than that under consideration.

10.5.2 Italian Standard NTC2018

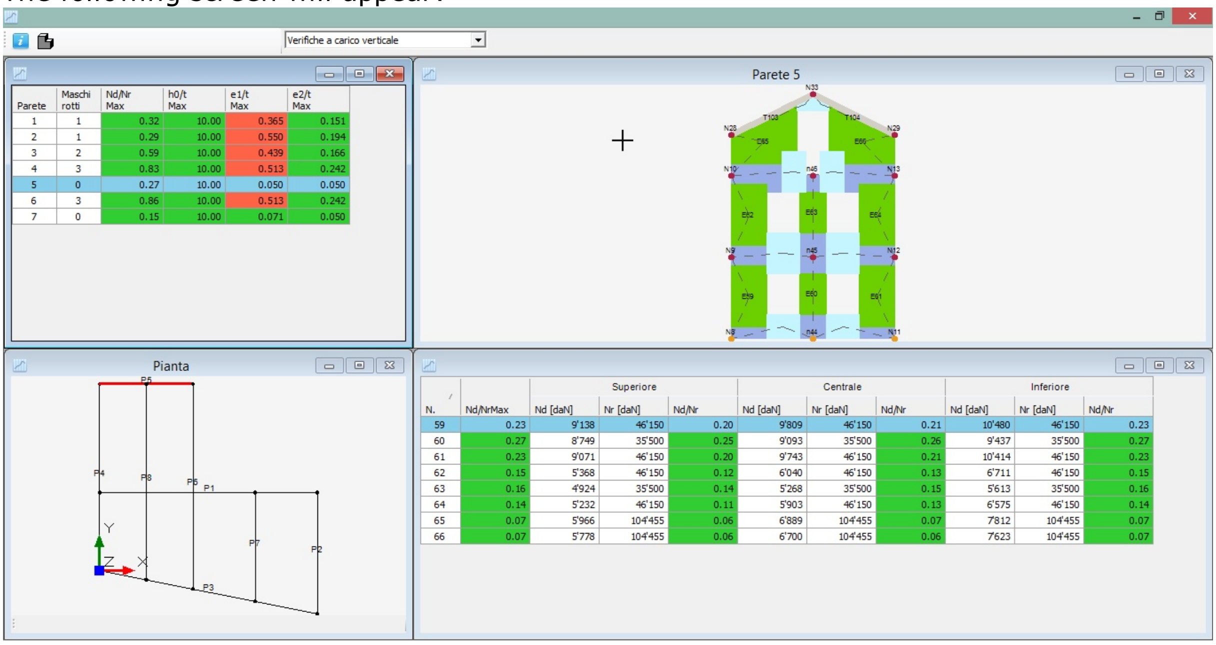
Slenderness Control: § [4.5.1]	**h0/t≤20**
	h0: free deflection length of the wall equal to ρ•h *t: wall thickness* *ρ: factor that takes into account the effectiveness of the constraint provided by orthogonal walls* *h: internal height of the level*
Eccentricity control of loads: § [4.5.11]	$e_1/t \leq 0.33$ $e_2/t \leq 0.33$
	$e_1 = \mid e_s \mid + \mid e_a \mid$; $e_2 = \quad + \mid e_v \mid$ *es : total eccentricity of vertical loads;* *ea: eccentricity due to execution tolerances;* *ev: eccentricity due to the wind;*

Verification of vertical loads: § [4.5.6.2]	$N_d \le \Phi\ f_d\ A$
	Nd: *vertical load calculation agent at the base of the wall;* ***A***: *area of the horizontal section of the wall net of openings;* ***fd***: *masonry calculation resistance;* *Φ: reduction coefficient of the wall resistance*

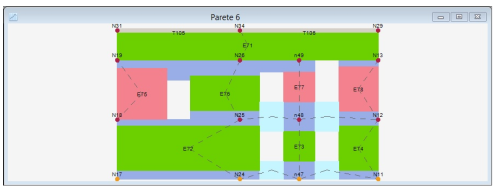
The static checks are performed in an area that is accessed using the associated button.



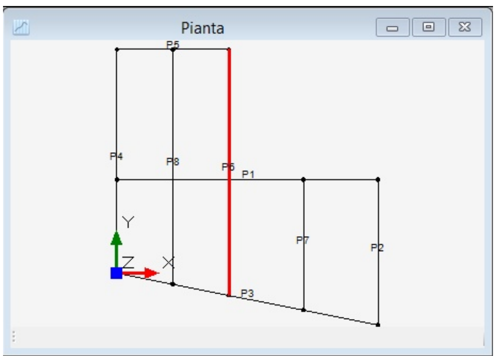
The following screen will appear:



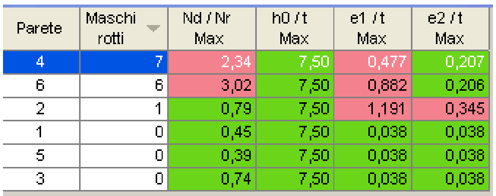
Let's describe it in detail.



In the upper right side appears the wall mesh.
In this case, does not exist the legend with colors indicating different phases of damage.
Elements that passed the check appear in green and in different color those that do not exceed the check.

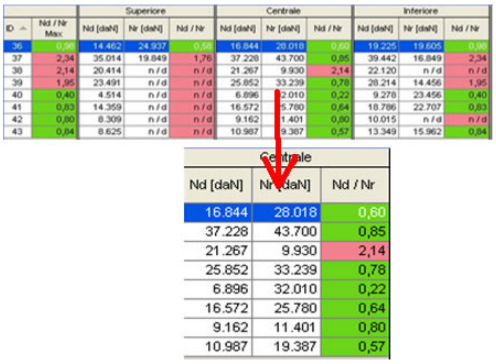


At the lower left side, is shown the plan view. The wall shown in the precedent view is highlighted with a thick line.



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

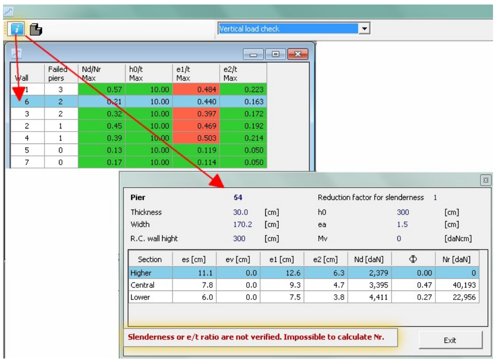
On the upper left side there is a list of the walls in the model, with the number of elements that did not pass the check and the values associated with the individual checks. The values found in the table are for the wall elements examined in which the limit values are the most restrictive of all the piers. Clicking on the line of a wall (highlighting it in blue) brings that wall to the view on the right side.



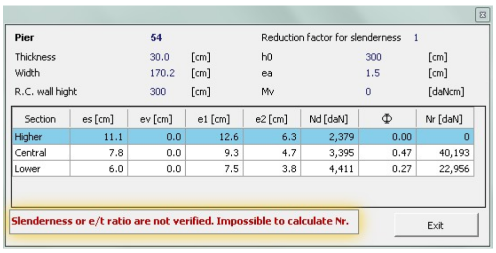
Nd [daN]	Nr [daN]	Nd / Nr
16.844	28.018	0,60
37.228	43.700	0,85
21.267	9.930	2,14
25.852	33.239	0,78
6.896	32.010	0,22
16.572	25.780	0,64
9.162	11.401	0,80
10.987	19.387	0,57

At the lower right side, the elements detail window is shown for the selected wall.
For each masonry element, the checks are performed for three different sections (higher, central, lower).
For each section the value for normal forces strain is shown (Nd: computed based on the masses and the combinations of the loads) and the normal resistant strain (Nr $= \Phi f_d A$).

The check is satisfied if the ratio Nd/Nr≤ 1. In this case, the corresponding cell appears in green.
In some cases, as shown in the example, Nr cannot be calculated (n/d: not defined). This happens when the slenderness or eccentricity checks are not satisfactory.



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



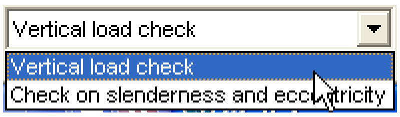
Pier	54		Reduction factor for slenderness	1	
Thickness	30.0	[cm]	h0	300	[cm]
Width	170.2	[cm]	ea	1.5	[cm]
R.C. wall hight	300	[cm]	Mv	0	[daNcm]

Section	es [cm]	ev [cm]	e1 [cm]	e2 [cm]	Nd [daN]	Φ	Nr [daN]
Higher	11.1	0.0	12.6	6.3	2,379	0.00	0
Central	7.8	0.0	9.3	4.7	3,395	0.47	40,193
Lower	6.0	0.0	7.5	3.8	4,411	0.27	22,956

Slenderness or e/t ratio are not verified. Impossible to calculate Nr.

Exit

The window shows all the details of the parameters used in the computation of the various check coefficients.
The text in red near the bottom gives relative informations to conditions where the check was not satisfied.
This window can remain open and be moved to any point of the drawing area while working (floating window). This gives to the user the possibility to select various elements in different wall and still have the details for each individual check visible.



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

		Higher	Central	Lower
ID	h0 /t	e1 /t	e2 /t	e1 /t
35	7,50	0,320	0,144	0,263
36	7,50	0,306	0,145	0,276
37	7,50	0,360	0,172	0,331
38	7,50	0,332	0,150	0,274
39	7,50	0,451	0,167	0,268
40	7,50	0,401	0,175	0,311
41	7,50	0,448	0,202	0,369
42	7,50	0,470	0,186	0,309

Here we see the check details for slenderness and eccentricity. The green values indicate that the check was passed.

In order to help the user in the results interpretations, some of the tables offer the possibility of reordering the rows according to the column characteristics.

Wall check summary table

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

This table is ordered based on the wall identifiers. The type of orientation is clarified by the arrow found at the top of the column.

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

Clicking on the appropriate column will reorder the values according to the characteristics chosen.
In the figure at the side, the table is ordered based on the number of failed elements.

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

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

Compression check details table

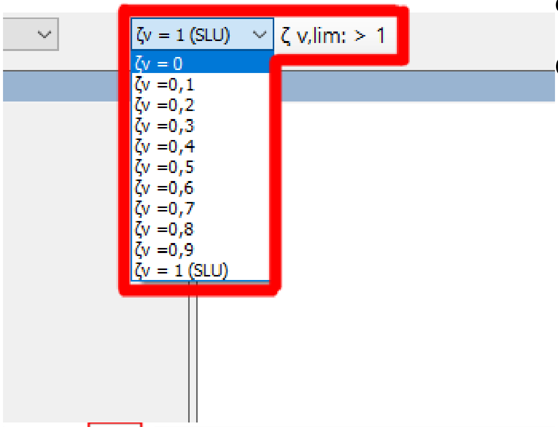
ID	Nd / Nr Max	Nd [daN]	Nr [daN]	Nd / Nr	Nd [daN]	Nr [daN]	Nd / Nr	Nd [daN]	Nr [daN]	Nd / Nr
1	0,38	24.834	111.564	0,22	33.733	111.564	0,30	42.633	111.564	0,38
4	0,45	7.160	22.917	0,31	8.743	22.917	0,38	10.327	22.917	0,45
5	0,05	-2.663	29.147	-0,09	-647	29.147	-0,02	1.369	29.147	0,05
6	0,18	1.985	44.626	0,04	5.071	44.626	0,11	8.156	44.626	0,18
7	0,21	2.998	44.626	0,07	6.083	44.626	0,14	9.168	44.626	0,21

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

The current formulation that leads to the calculation of the eccentricity es2 considers in the calculation of the N2 (load coming from the overlying floors) in addition to its contribution also the respective d2 eccentricity value for all of the levels higher than that under consideration.

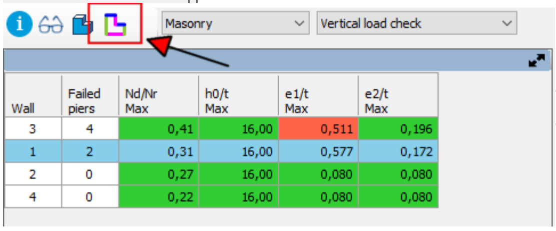
It is also possible:

Calculate the factor ζv, i defined as:
the ratio between the maximum value of the variable vertical overload that can be tolerated by the i-th part of the construction and the value of the variable vertical overload that would be used in the design of a new



construction.

Circular n ° 7 of 21 January 2019, § [C8.3]



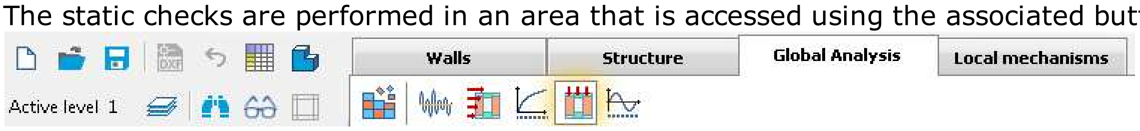
Display a color map which shows, for each element, the normal stress and stress values for each element.

10.5.3 Eurocode 6

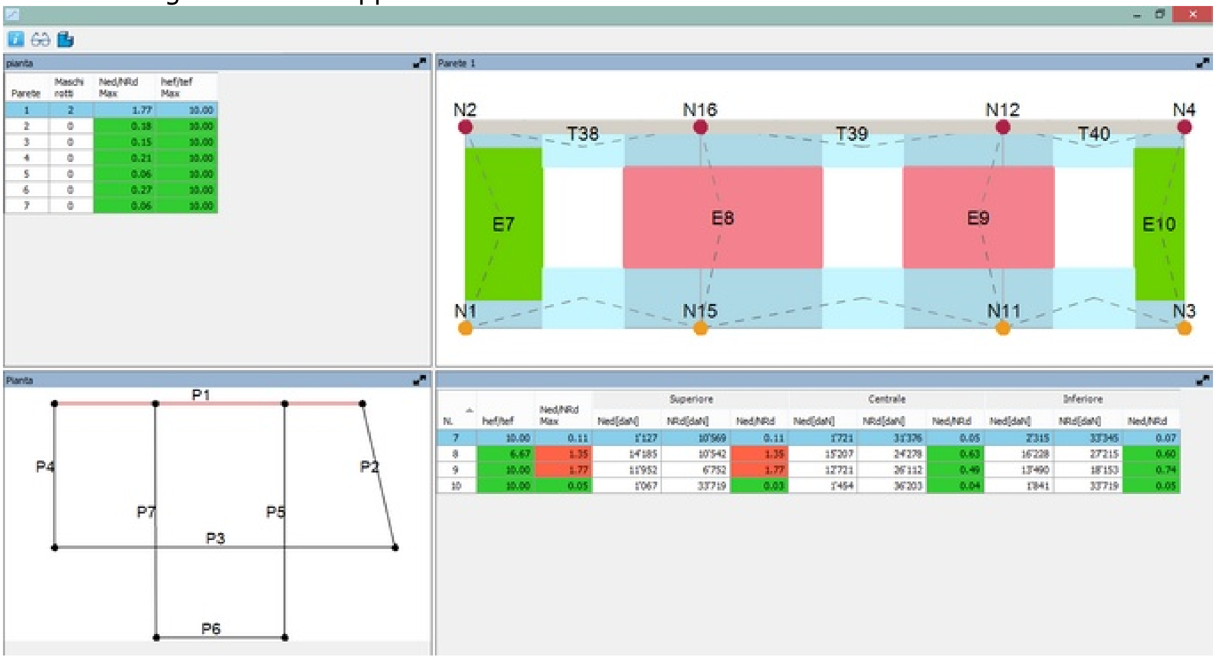
Let's present below the checks that are carried out:

Slenderness check: *EN 1996-1-1 § 5.5.1.4*	$h_{ef}/t_{ef} \leq \lambda_{lim}$: *di default=27*
	h_{ef}: *effective height of the wall equal to* $\rho \cdot h$ t_{ef}: *effective thickness of the wall equal to* $\rho_t \cdot h$
Verification subjected to vertical loading *EN 1996-1-1 § 6.1.2*	$N_{Ed} \leq N_{Rd} = \Phi\ f_d\ A$
	N_{Ed}: *design value of the vertical load applied to a masonry wall;* **A**: *is the loaded horizontal gross cross-sectional area of the wall;* **fd**: *design compressive strength of the masonry;* Φ: *is the capacity reduction factor*

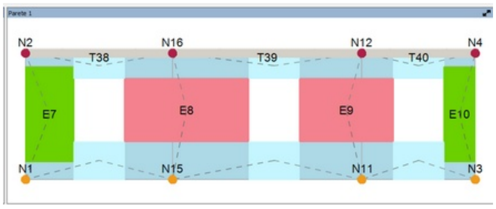
The static checks are performed in an area that is accessed using the associated button.



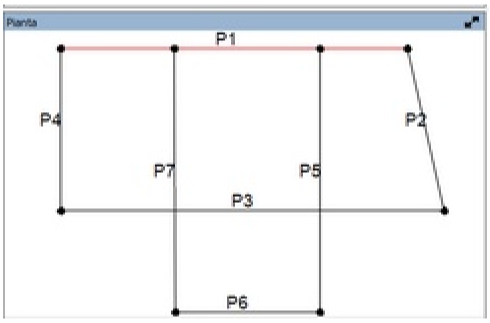
The following screen will appear:



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



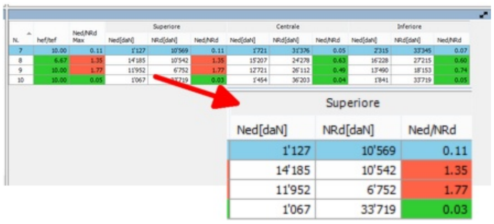
In the upper right side appears the wall mesh.
In this case, does not exist the legend with colors indicating different phases of damage.
Elements that passed the check appear in green and in different color those that do not exceed the check.



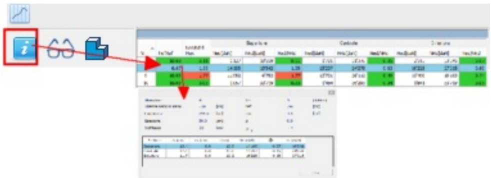
At the lower left side, is shown the plan view. The wall shown in the precedent view is highlighted with a thick line.

Parete	Maschi rotti	Ned/NRd Max	hef/tef Max
1	2	1.77	10.00
2	0	0.18	10.00
3	0	0.15	10.00
4	0	0.21	10.00
5	0	0.06	10.00
6	0	0.27	10.00
7	0	0.06	10.00

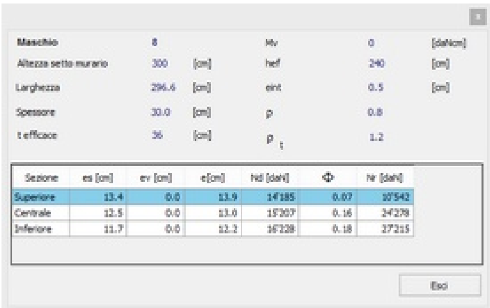
On the upper left side there is a list of the walls in the model, with the number of elements that did not pass the check and the values associated with the individual checks. The values found in the table are for the wall elements examined in which the limit values are the most restrictive of all the piers. Clicking on the line of a wall (highlighting it in blue) brings that wall to the view on the right side.



At the lower right side, the elements detail window is shown for the selected wall.
For each masonry element, the checks are performed for three different sections (higher, central, lower).
For each section the value for normal forces strain is shown (N_{Rd}: computed based on the masses and the combinations of the loads) and the normal resistant strain (**$N_{Rd} = \Phi\ f_d\ A$).**
The check is satisfied if the ratio $N_{Ed}/N_{Rd} \leq 1$. In this case, the corresponding cell appears in green, otherwise in red color.
In some cases, as shown in the example, N_{Rd} cannot be calculated (n/d: not defined). This happens when the slenderness or eccentricity checks are not satisfactory.



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



The window shows all the details of the parameters used in the computation of the various check coefficients.
The text in red near the bottom gives relative informations to conditions where the check was not satisfied.
This window can remain open and be moved to any point of the drawing area while working (floating window). This gives to the user the possibility to select various elements in different wall and still have the details for each individual check visible.

		Higher	Central	Lower
ID	h0 /t	e1 /t	e2 /t	e1 /t
35	7,50	0,320	0,144	0,263
36	7,50	0,306	0,145	0,276
37	7,50	0,360	0,172	0,331
38	7,50	0,332	0,150	0,274
39	7,50	0,451	0,167	0,268
40	7,50	0,401	0,175	0,311
41	7,50	0,448	0,202	0,369
42	7,50	0,470	0,186	0,309

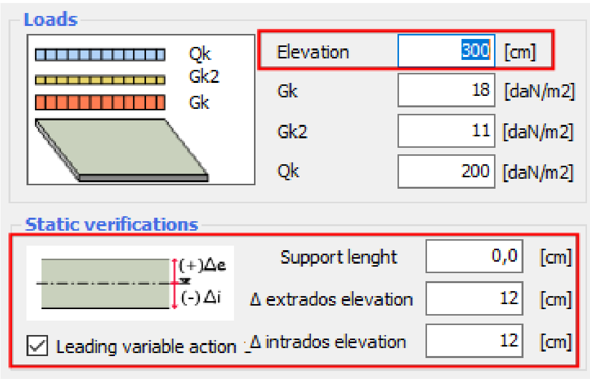
Here we see the check details for slenderness and eccentricity. The green values indicate that the check was passed.

In order to help the user in the results interpretations, some of the tables offer the possibility of reordering the rows according to a characteristic shown in a column by simply clicking on the title of the column.

10.5.4 Net height

The height that is attributed to each level is given by the difference in the relative quotas to the average levels of the slabs, a height which, from the point of view of the calculation, appears to be cautionary.
In some cases, such as in the case of static analysis, it may be necessary to refine the

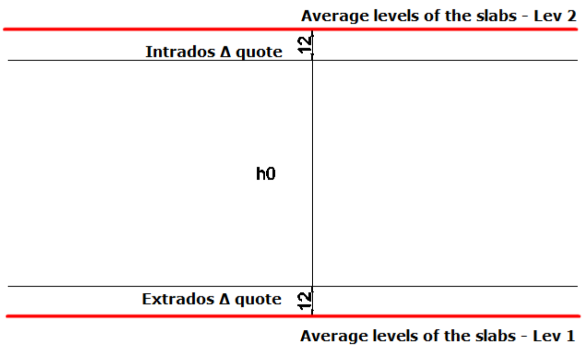
calculation by considering a reduced height (net height or effective height) that takes into account the contribution given by the thickness of the slabs.



In order to consider it, is sufficient to assign, within the mask, a value to the parameters extrados/intrados Δ quote.

By inserting a new slab inside the model, the fields extrados/intrados Δ quota are automatically filled in with suggested values based on the thickness attributed to the slab.

The field can be re-edited at any time by entering values that are considered appropriate.

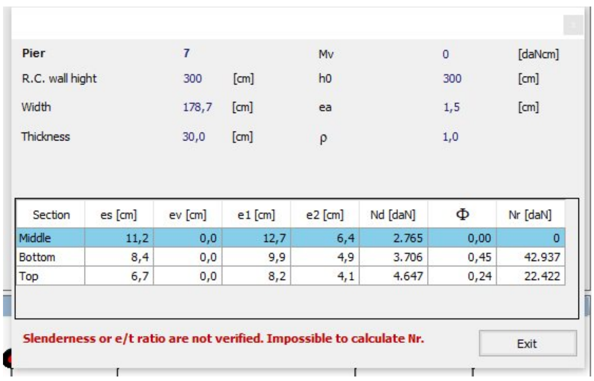


The value of h_{net} will be determined as the difference between the height of the level and the sum given by extrados Δ quota and the intrados Δ quota.

Here is an example that refers to the image on the left side.

Level height = 300 cm
Height masonry r.c. composite slab for level 2 = 24 cm
extrados Δ quota = 12 cm
Height masonry r.c. composite slab for 1 = 24 cm
intrados Δ quota = 12 cm

h_{net} = Level height - (extrados Δ quota + intrados Δ quota)
= 300 - 24 = 288 cm

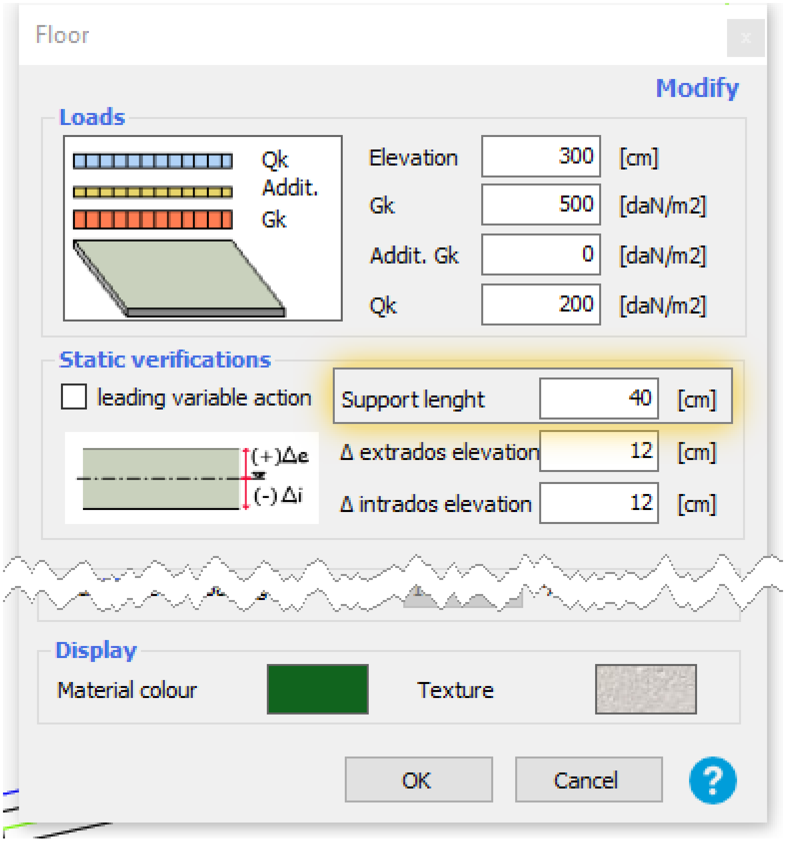


Section	es [cm]	ev [cm]	e1 [cm]	e2 [cm]	Nd [daN]	Φ	Nr [daN]
Middle	11,2	0,0	12,7	6,4	2.765	0,00	0
Bottom	8,4	0,0	9,9	4,9	3.706	0,45	42.937
Top	6,7	0,0	8,2	4,1	4.647	0,24	22.422

It is possible to display the value of the net height corresponding to each pier from the details window, available within the results environment related to the static analysis.

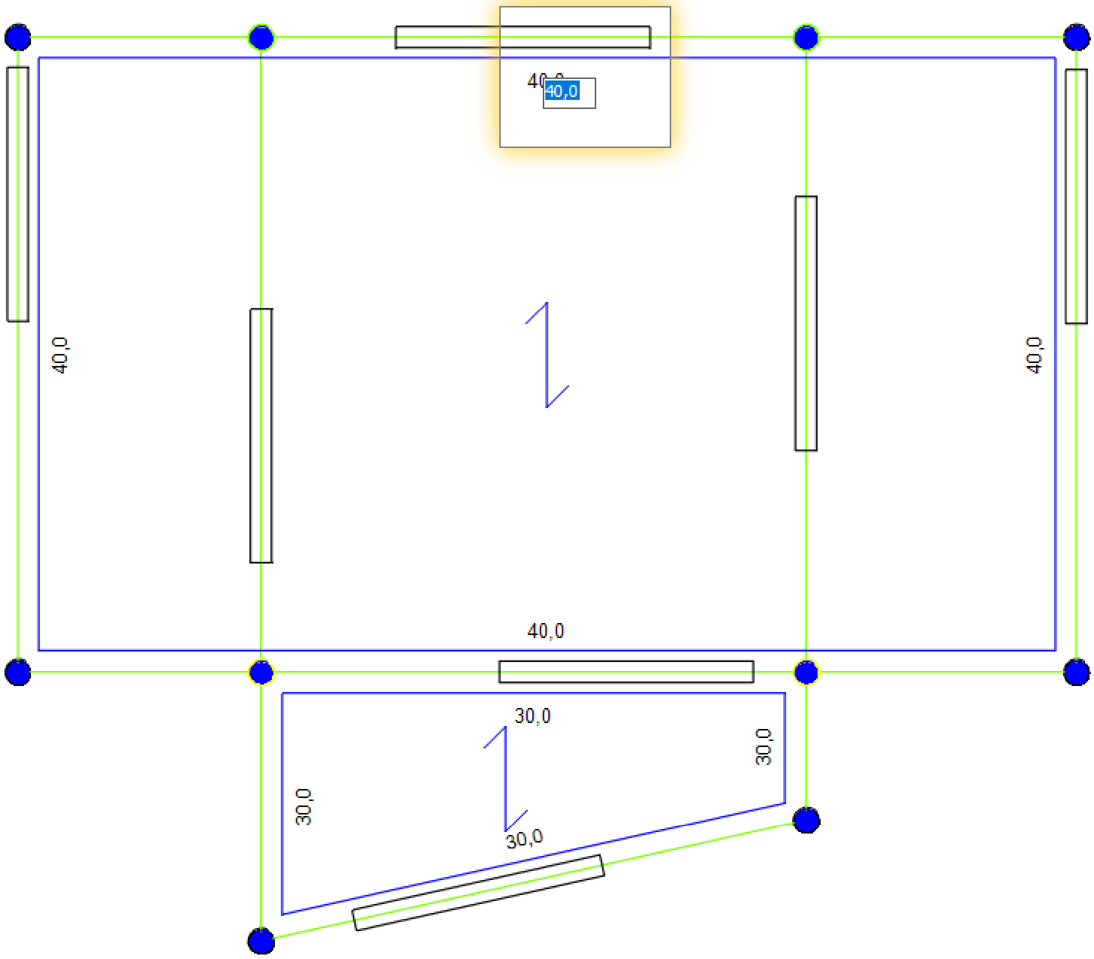
10.5.5 Support length

In order to calculate the eccentricity required for the static verification of the building, it is important to define the values of the support lengths of the slabs.
It is possible to assign a value to the support length inside the mask of the slabs.



The value entered in this way will be attributed, equally, to all sides of the slab.

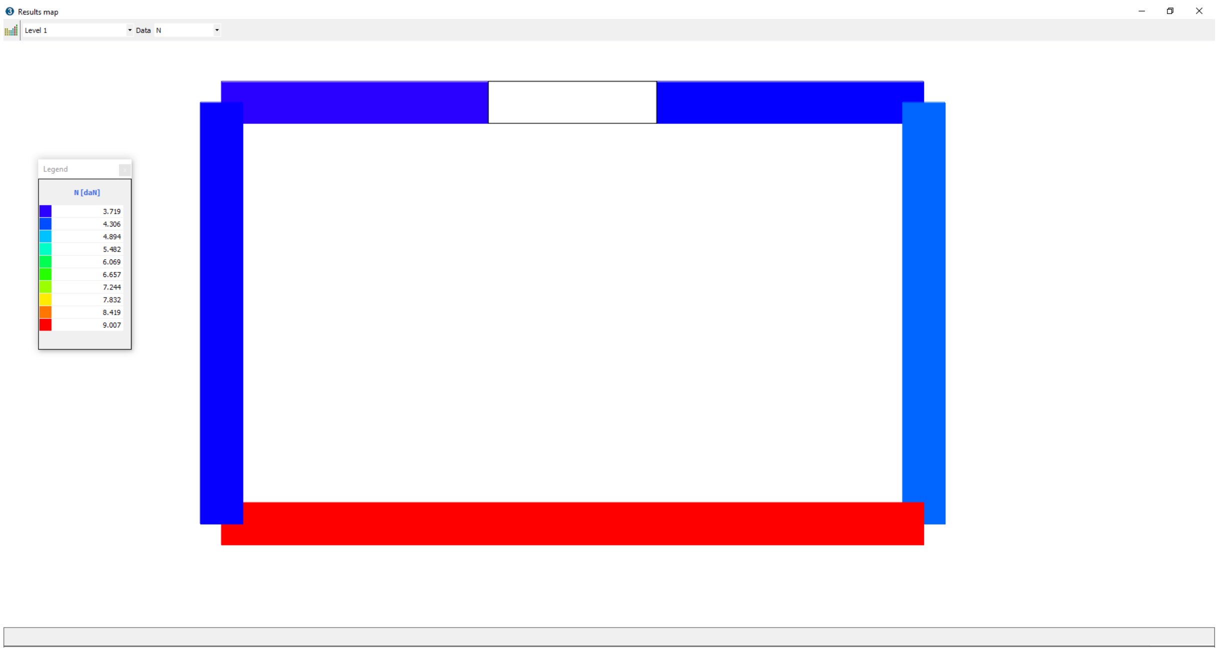
If it is necessary to assign different values to each side of the slab, it is possible to edit the values corresponding to the support lengths directly from the structure environment.

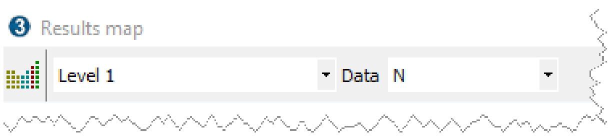


To modify the single value referred to the side, simply click on the dimension with the right button of the mouse and enter the new value.

10.5.6 Colors map

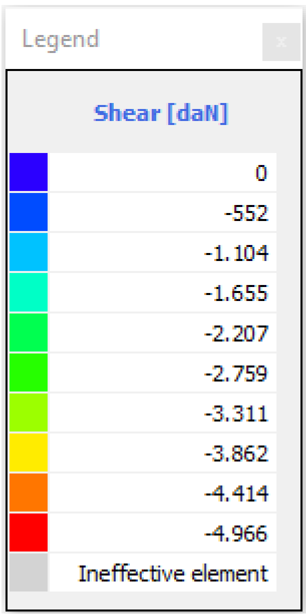
 Within the static analysis results environment, by clicking on this icon, it is possible to view the building's colors map relating to the results of the carried out static analysis.

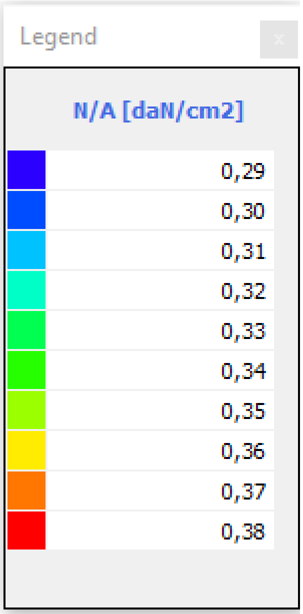


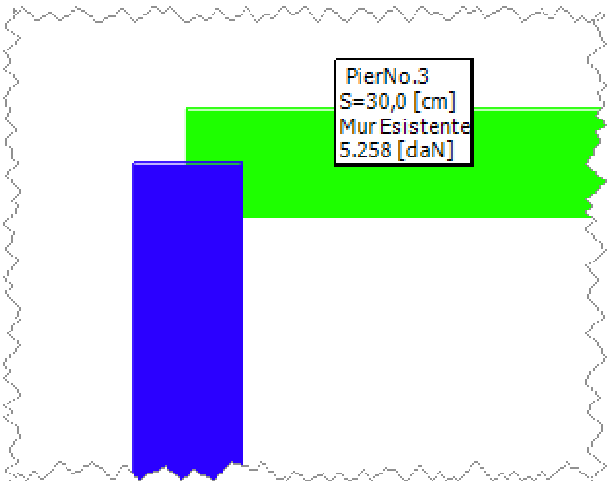


Through the "data" drop-down menu in the upper left corner it is possible to view the map relating to normal stress or that relating to tensions.
The user can also filter the maps by level.

Using the "colors legend" command it is possible to view the legend associated with the colors map.

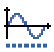






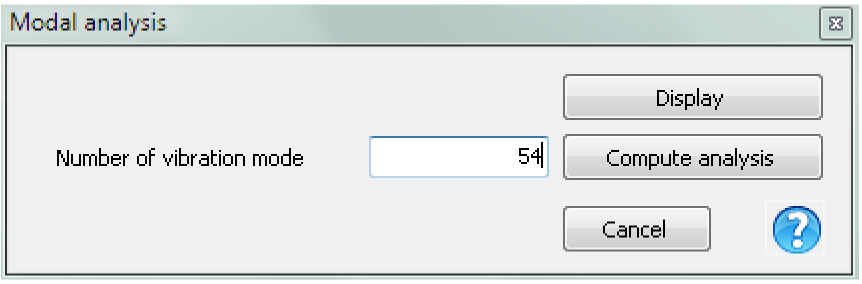
Using the tooltips associated with each element, it is possible to analyze the entity of the stresses acting

10.6 Modal Analysis

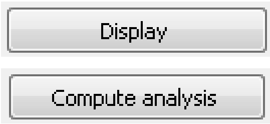


This is an area dedicated to computation of modal forms and the parameters associated with them.

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



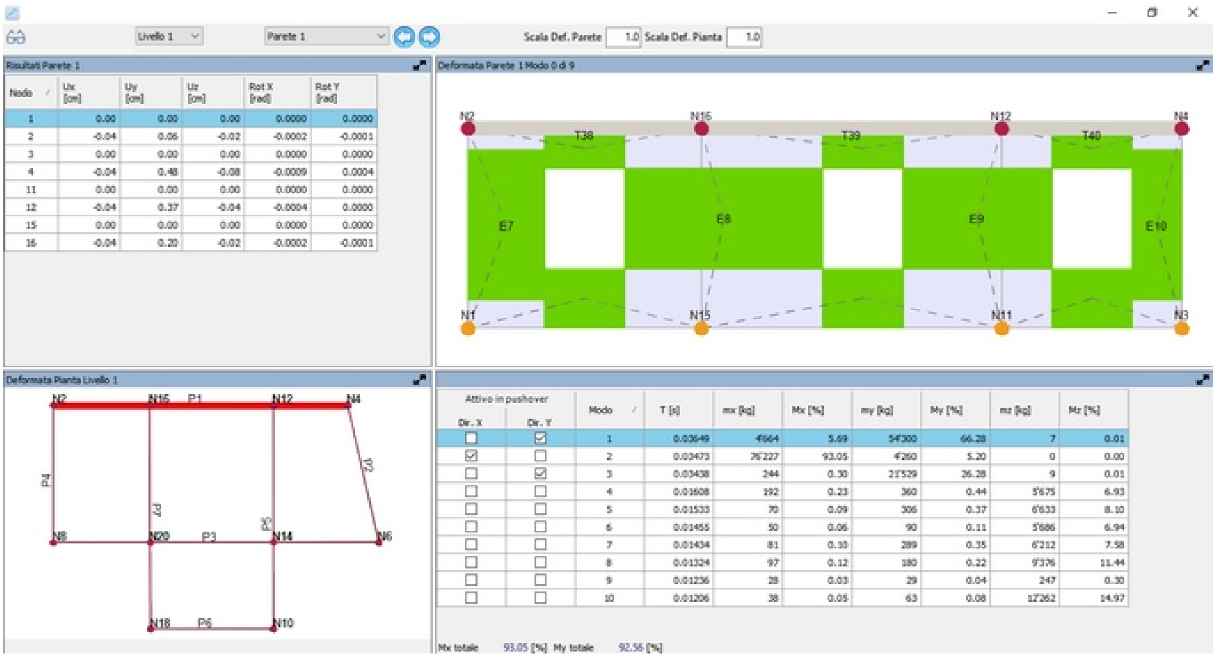
A number of predefined modal forms are offered.



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

The computation is performed, and the results are shown.

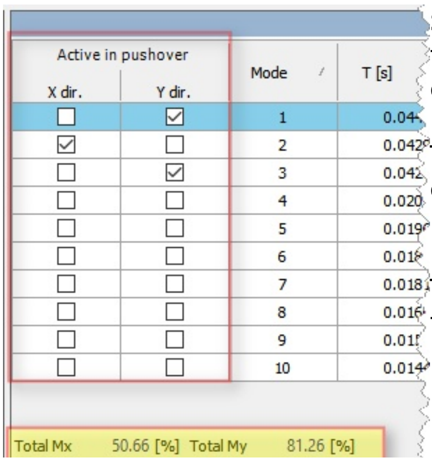
When the calculation is finished, the presentation of the results is automatically shown.



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.



Selecting the box to the left of the Mode, is possible to find the contribution of that modal form in the definition of the seismic load distribution in pushover.

The distribution of the seismic load can be defined as a combination of several modes until the desired total participant mass is reached.

The total mass, expressed as a percentage, is shown in the lower part of the table for each of the directions.

10.7 Kinematic 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.

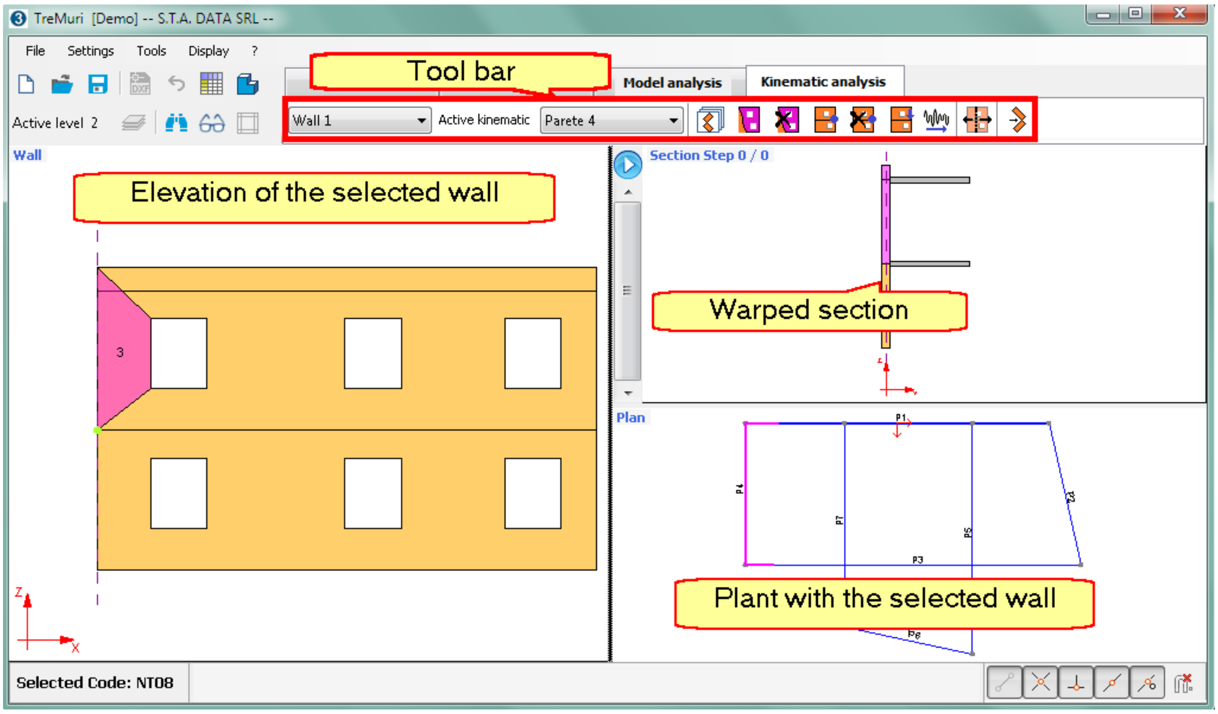
"Tremuri LM" is a calculation module inside the Tremuri program, which is dedicated to the evaluation of the building safety against such mechanisms.
The module **"Tremuri LM"** exploits the versatility and the input ergonomics of the program TreMuri to finalize a spatial model on which the user can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

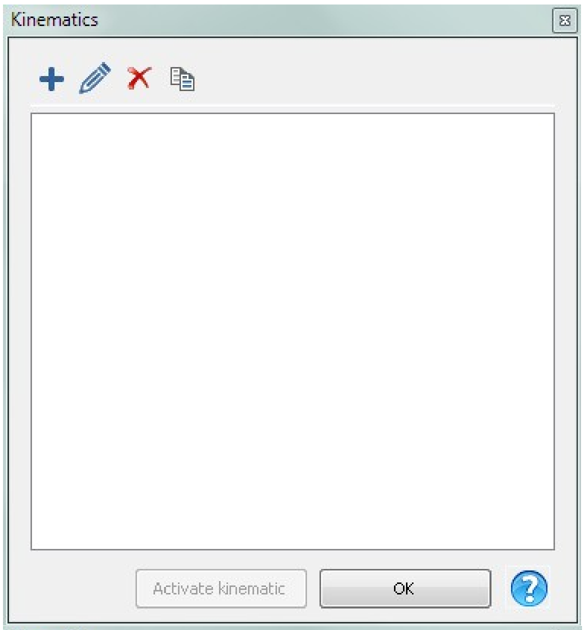
Beolchini G. C., Milano L., Antonacci E. (A cura di). *Repertorio dei meccanismi di danno, delle tecniche di intervento e dei relativi costi negli edifici in muratura – Definizione di modelli per l'analisi strutturale degli edifici in muratura*, Volume II – Parte 1a. Convenzione di Ricerca con la Regione 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".

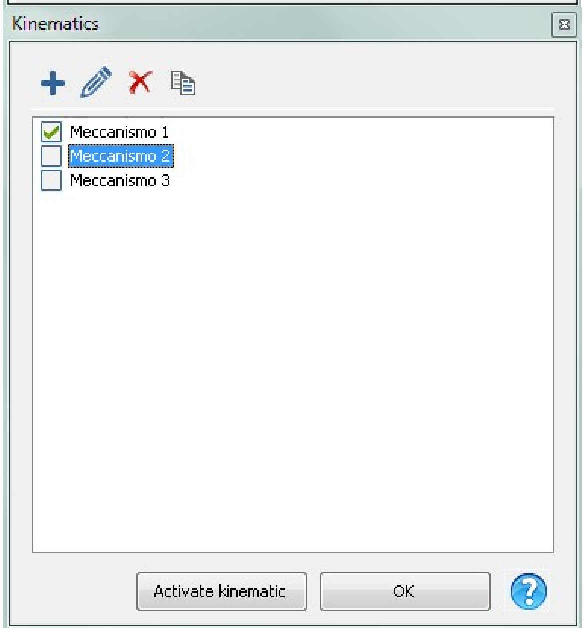


New Mechanisms (enter the name of the mechanism)

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.

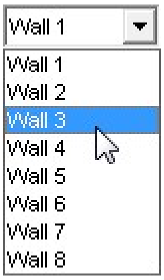
The active mechanism is represented by checking the box (☑) to the left of the name.

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

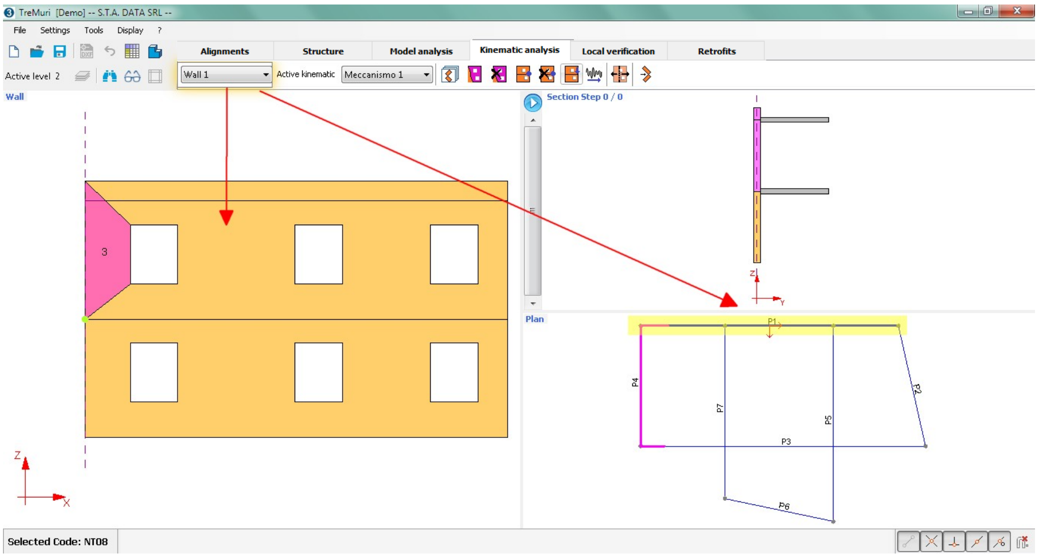
It is possible to use this box to change the active kinematic.

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.

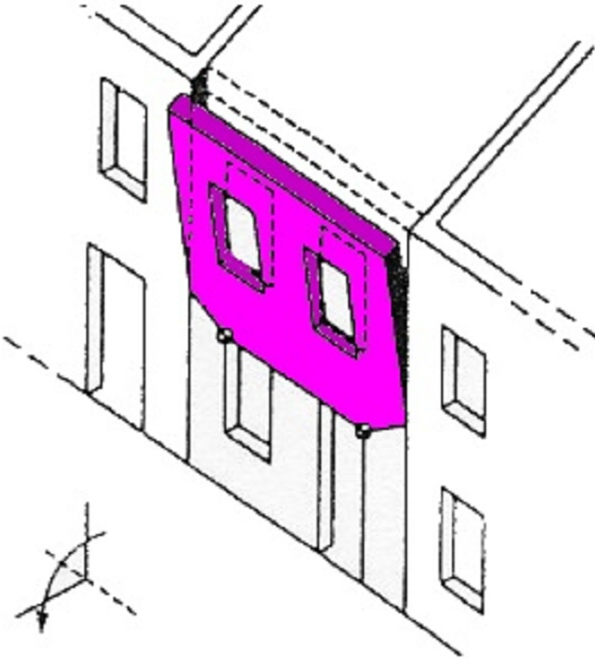


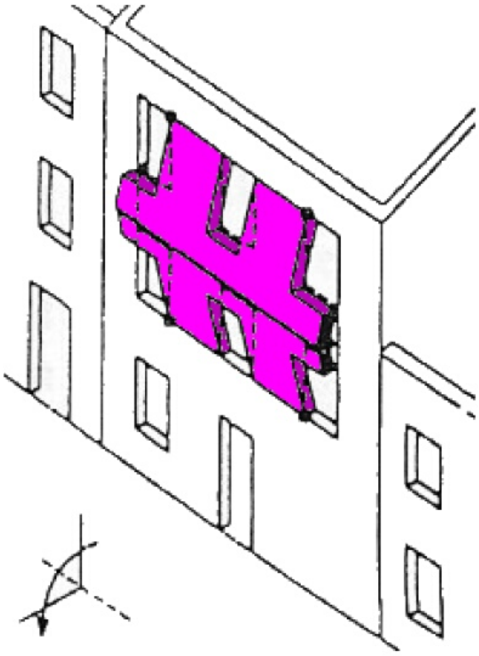
The mechanism input consist in three steps:

- 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.



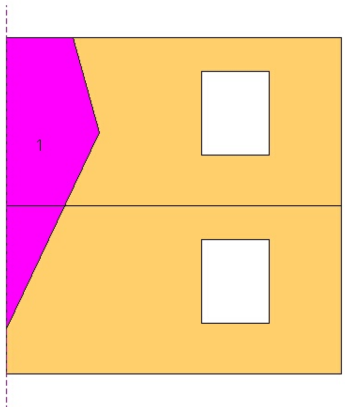


Example of a mechanism consisting of a single block

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.

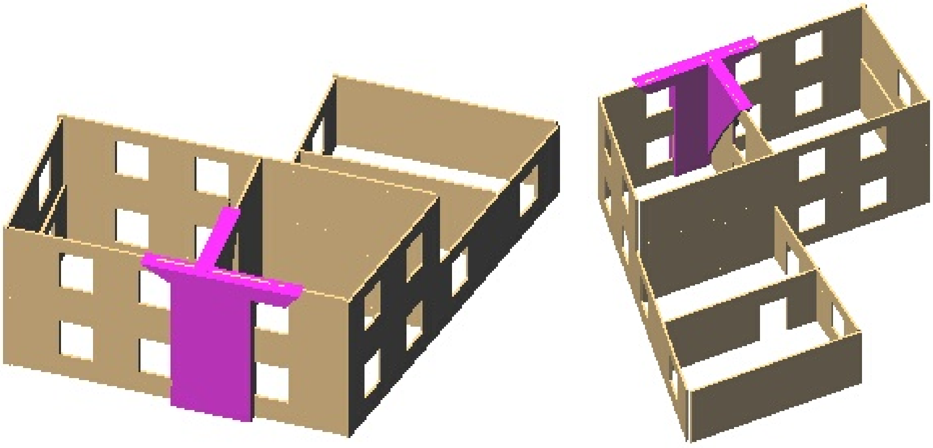


The possibility to draw a closed perimeter allows the user to trace the edges in correspondence of the panel crack found in site.

 (*)

Each single kinematic can contain any number of kinematics blocks in the same and different walls.

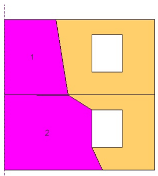
The image below shows a drawn system block based on the visible cracks of the structure.
A portion of the masonry of the wall plug (wedge) participate in the tilting of the perimeter wall.



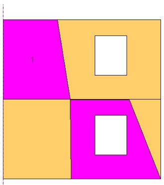
Axonometric views such as those described above are visible by pressing the "3D View" button .

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.

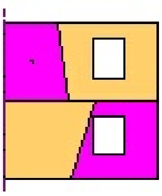
Therefore apply the following construction rules:



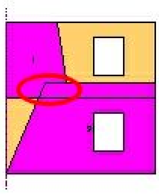
CORRECT
Blocks 1 and 2 have a common side



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



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

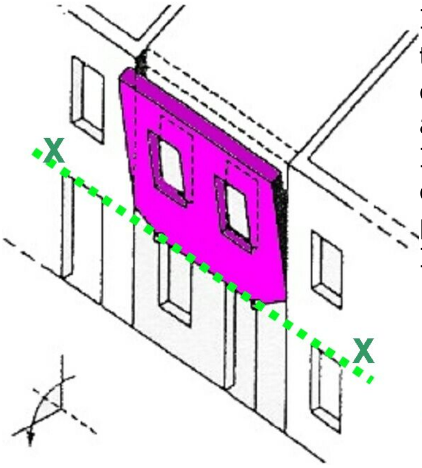


NOT CORRECT!!!!
The blocks 1 and 2 are overlapped

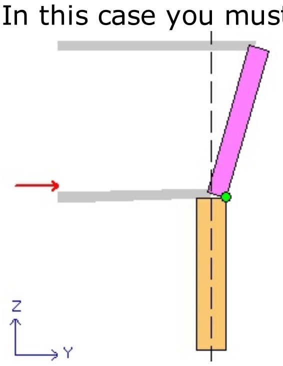

Delete blocks:
Selecting one or more blocks in sequence, confirming by pressing the right mouse the selected blocks are deleted.

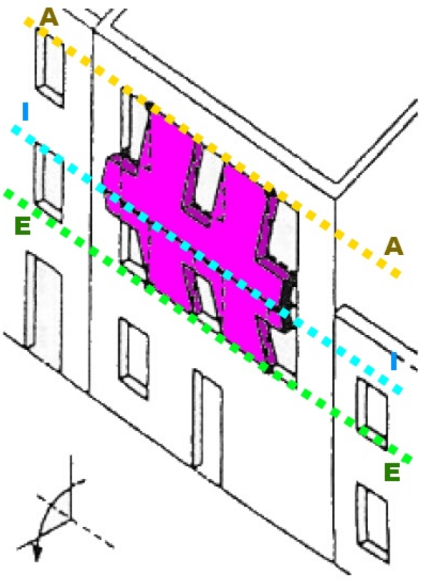
10.7.2.2 Constraints

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.

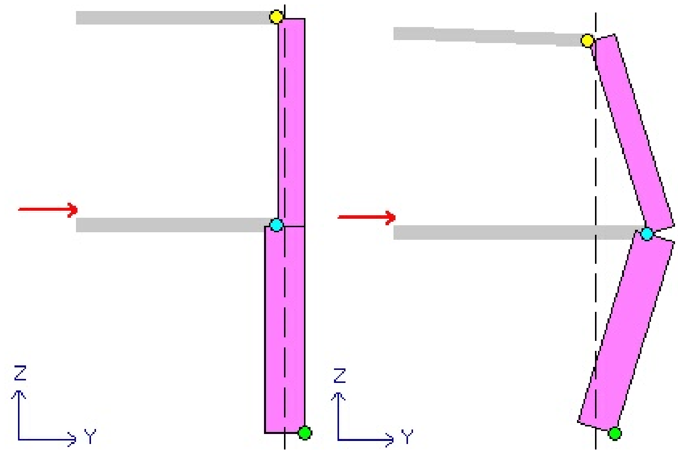


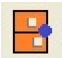
If we want to consider a tilting case of a wall portion like the one represented in the image (*), the considered case is the case of a block that rotates around the X-X axis .
In correspondence to this point you must enter a constraint between a kinematic block and a fixed wall portion.
In this case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



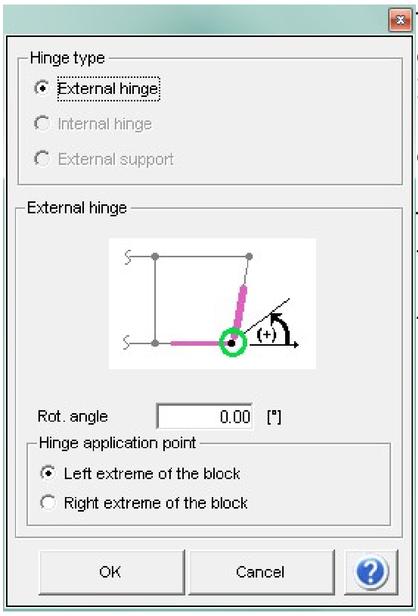


In this case (*) the bottom block is placed directly on a wall portion that is not deformed .
In the **E-E** position will be put the **"External Hinge"**
In the **I-I** position confines two blocks, so will be put the **"Internal Hinge".**
In the **A-A** position the deformation mechanism will not allow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



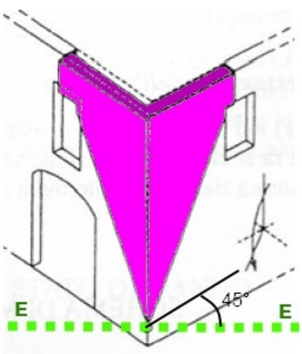

Insert constraint:

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).

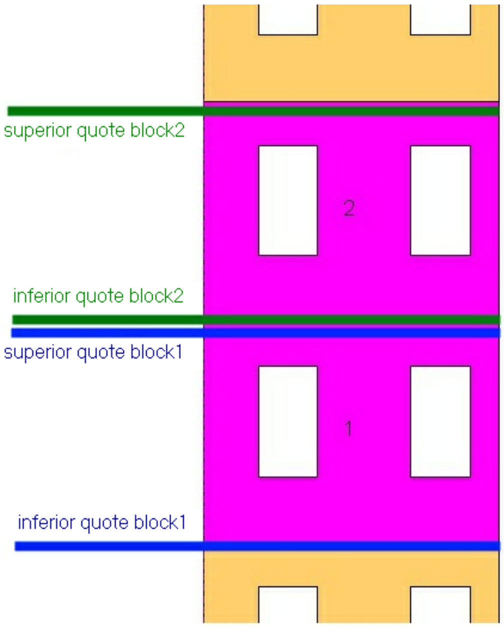
The "Angle" box means the angle that the external hinge form with the active wall. When the angle is zero, this means that the rotation axis of the constraint is parallel to the wall.



This function can be useful to examine tilting cases of the building's corner. (*)

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.

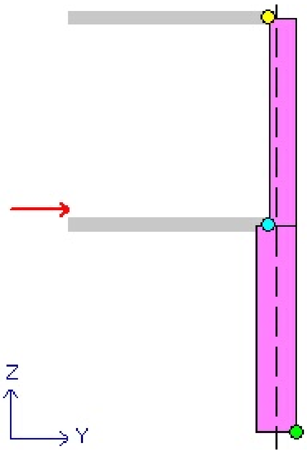


<u>Clicking on the block, the constraint is inserted to the:</u>

- **Bottom** quote (height) of the block if **External Hinge**
- **Top** quote (height) of the block if **Internal Hinge**
- **Top** quote (height) 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 **earthquake direction** is indicated by an red arrow in the section shown on the right of the screen.

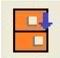


 If you want to change the earthquake direction you can use the appropriate button shown to the left of the section.

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 earthquake direction.

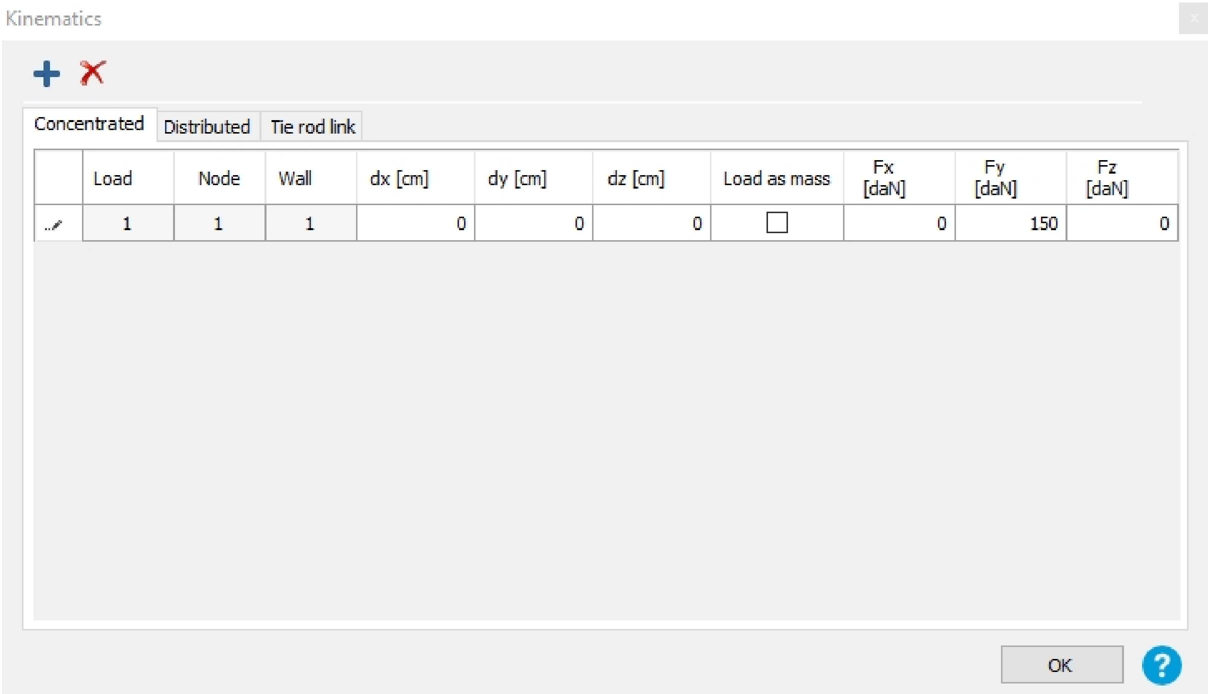
 **Delete constraint:**
Allows to remove a constraint.

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:

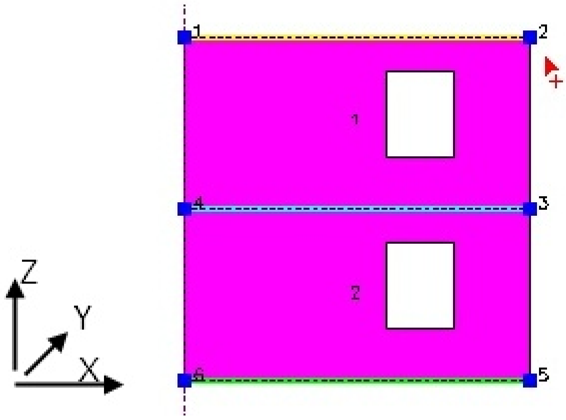


The buttons named **"Concentrated"** and **"Linear"** allow to put a concentrated load or linearly distributed; depending on the enabled button is shown the table with the list of loads already placed on the considered kinematics. The first time the table is clearly empty.

The item "**Tie rod link**" allows to define a load coming from an inserted tie rod in order to prevent the activation of the mechanism. Entering the tie rod load in this table allows you to pass the pre-stress value to the Tie Rod link verification module.

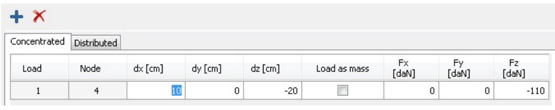
+ Insert new load:

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



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

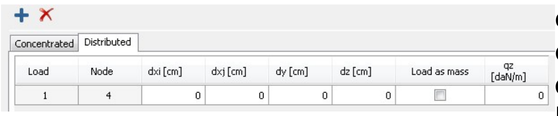
Concentrated load


Concentrated | Distributed

Load	Node	dx [cm]	dy [cm]	dz [cm]	Load as mass	Fx [daN]	Fy [daN]	Fz [daN]
1	4	10	0	-20	☐	0	0	-110

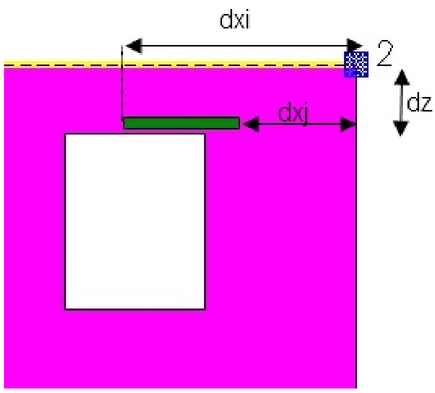
Node: Indicate the number of the selected node in graphical modality.
Force: If you insert a vertical load (Fz) different from zero, this load does not generate mass, without creating any horizontal component of seismic type (α*Fz; α: factor of vertical load).
Mass: If you insert a vertical load (Fz) different from zero, this load generates mass, creating a horizontal component of seismic type.
The loads applied to the mechanism in "indirect" mode, for example when the loads came from the superior wall are usually considered forces but not masses.
dx/dz: are the relative coordinates of the load application point, compared to a node centered system.
Fx / Fy / Fz: are components of the force in the system wall.

Distributed load


Concentrated | Distributed

Load	Node	dxi [cm]	dxj [cm]	dy [cm]	dz [cm]	Load as mass	qz [daN/m]
1	4	0	0	0	0	☐	0

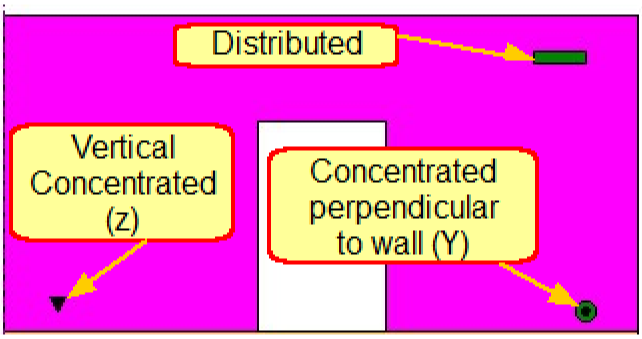
qz: distributed vertical load .
dxi / dxj / dz: these are the relative coordinates of the load application points compared to a node centered system.
dy: coordinate of the point of application of the load, with respect to the axis orthogonal to the plane of the wall.



It is not allowed to edit directly the numbers in this table, to edit these values you must select the row and press the "**Edit**" button .

Delete: allow to remove the load corresponding to the selected line.

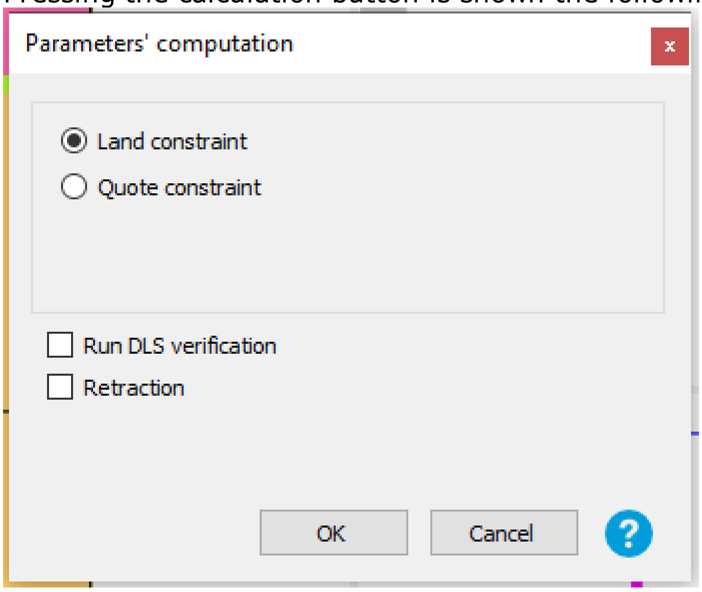
The loads are shown in the below graphic with the following agreement.



10.7.2.4 Calculation

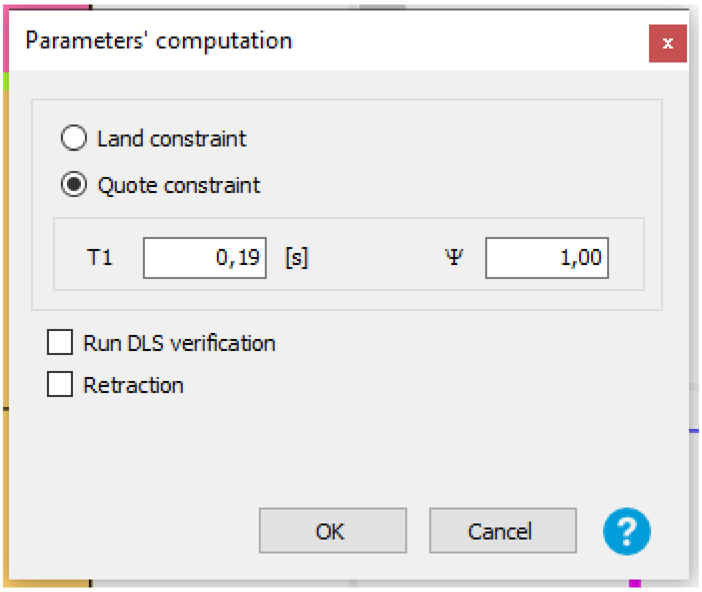
When the input is complete, you can proceed with the calculation.
With the module **"3Muri LM"** is possible to run the Verification of the Linear Kinematic Analysis.

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 (height).
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.

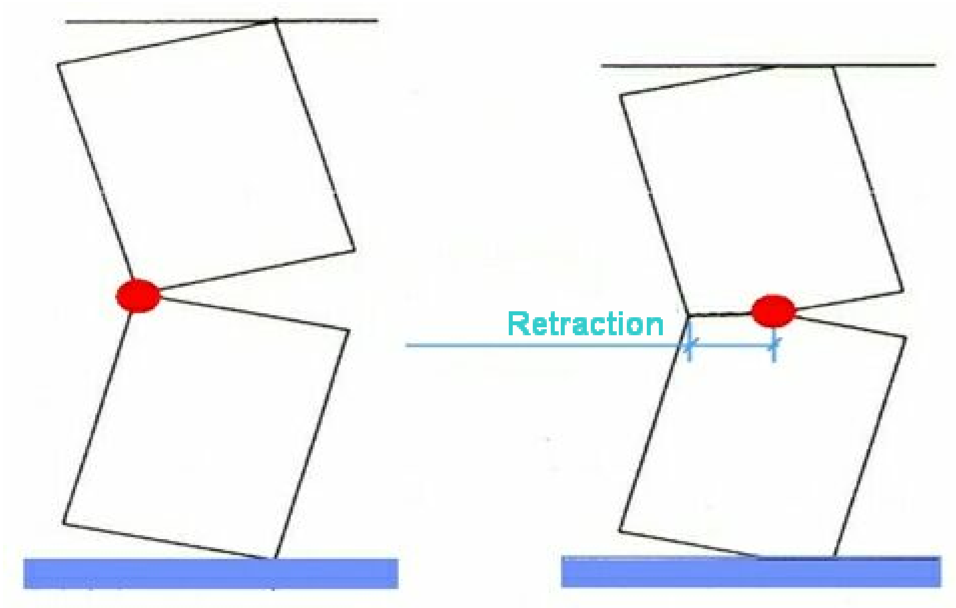
ψ is the first vibration mode in the considered direction, standardized at a summit of the building, in the absence of more accurate valuation is assumed ψ = Z / H, where H is the height of the structure regard to the foundation.
Z is the height, compared to the foundation of the building, the center of gravity of the constraint lines between the blocks interested by the mechanism.

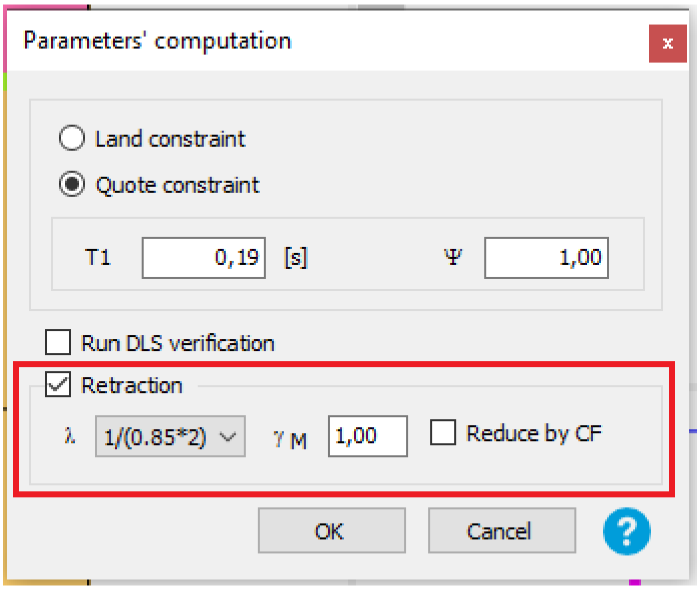
The box **"Run verification DLS"** allows you to verify the Damage Limit State.
Normally this box is not selected because this verification is not required.

Legal extract: C8A.4.2.3 Safety checks - Damage limit state

```
.....
In the case of local mechanisms, the limit state limit state of damage corresponds
to the onset of cracks that do not affect the entire structure but only a part of
it; therefore in the case of existing masonry buildings, also in consideration of
the justified conservation needs, although as it is desirable to satisfy this
limit state, its verification is not required.
....
```

The box **"Retraction"** manages the retraction of the hinge.
The hinges of the mechanism constitute the points about which the various blocks rotate relative to one another.
The point of rotation depends on the compressive strength of the masonry. In the case in which the resistance is infinite, the center of rotation coincides with the edge at the base. In the case of limited compressive strength, the rotation center of the kinematic will be positioned within the thickness of the wall.





λ: Constant of calculation of the retraction function (recommended = 1/(0.85*2))
γ_M: Security factor of the resistance of the material (recommended = 1)
Reduce by CF: application of the reduction to take into account from the Confidence Factor (recommended = NOT selected)

The retraction is identified by the relation

$$a=\lambda\frac{P}{fd\ L}$$

a: retraction
P: total vertical load acting on the hinge
L: length of the hinge
fd: design strength

The design strength fd is detected by the appropriate formula:

$$f_d = \frac{f_m}{F_C \cdot \gamma_M}$$

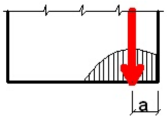
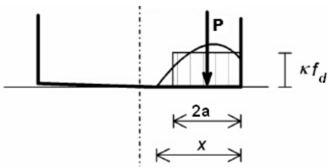
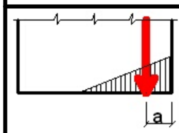
fm: average compressive strength

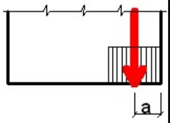
γ_M: Security factor of the strength of the material (suggested value = 1)

in case of linear analysis such safety factor must be set equal to 2.
Actually the NTC do not comment the value that has to be attributed to this parameter for the case of retraction calculation of the hinge in a linear kinematic analysis. The retreat of the hinge is different from the analysis (called "linear kinematics").
The retraction of the rotation hinge corresponds to the physical phenomenon of edge breakage of the wall and therefore it is a "non-linear" phenomenon and the safety coefficient of the materials can reasonably be set equal to 1.

CF: Confidence Factor of the material

The NTC do not comment on the need to consider the CF in the calculation of design resistance, some techniques bibliographies consider it in the calculation.
It is important to remember that the confidence factor is already introduced in the formula that permits the acceleration calculation of the activation mechanism.
Consider it also in the calculation of the design resistance is equivalent to consider it twice. It is therefore recommended that you do NOT reduce by CF.

 λ=1/(0.85*2) ***RECOMMENDED!***	This value of the calculation constant has its origins in the theory of vertical bending which is based on the stress-block method on which it bases the origins of the constitutive connection for bending (NTC § 7.8.2.2.1 regulations).  In this case, from equilibrium at vertical-traverse it is possible to write: $P = k \cdot f_d \cdot L \cdot 2a$ The length of retraction is then half the length of the stress-block equivalent, thus appears to be equal to: $a = \frac{P}{2 \cdot k \cdot f_d \cdot L} = \frac{1}{2 \cdot k} \cdot \frac{P}{f_d \cdot L} = \lambda \cdot \frac{P}{f_d \cdot L}; \quad \lambda = \frac{1}{2 \cdot k}$ **K:** Used in the bending formula of NTC § 7.8.2.2.1 regulations assumed is equal to 0.85. ***<u>Reference:</u>*** *Galasco e Frumento, ANALISI SISMICA DELLE STRUTTURE MURARIE,* *E127 - Progetto Costruzione Qualità, Gruppo Editoriale SIMONE*
 λ=2/3	This value has its origins from a hypothesis of triangular distribution of the tension.

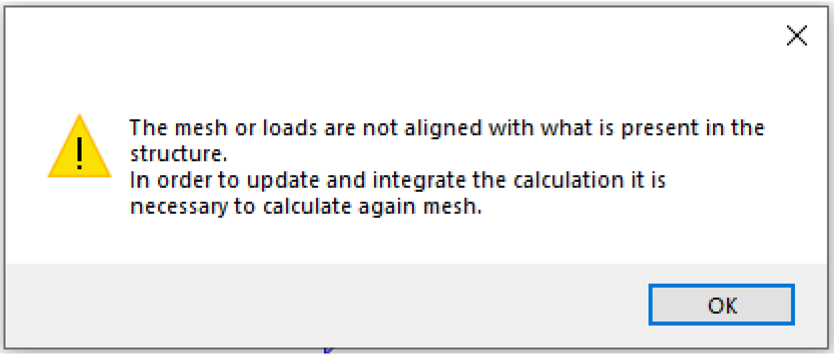
 λ=1/2	This value has its origins from a hypothesis of rectangular (constant) distribution of the tension.
More...	In this case, it’s possible to enter any multiplier that the user thinks is appropriate

Pressing the **OK** button the calculation is performed.

Checks

Mesh / Loads not aligned

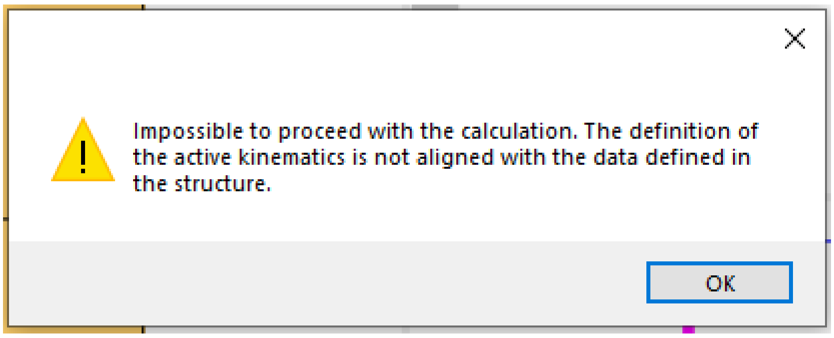
In the event that the mesh or the loads are not aligned with what is present in the structure, when the "Kinematic Analysis" environment is accessed, the following warning appears:



The warning suggests us to re-mesh the model, to be sure of making an updated calculation.

Calculation - Invalid Mechanism

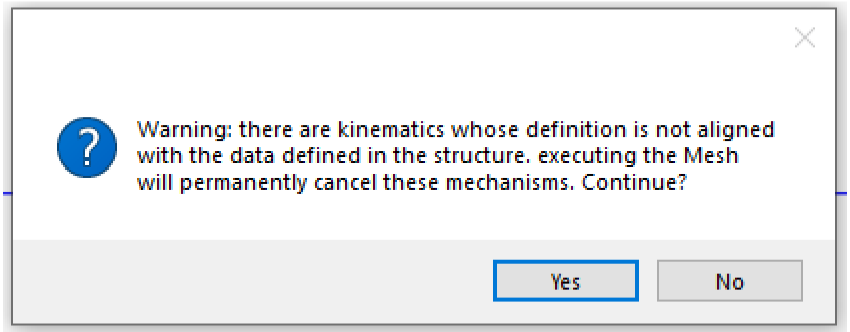
If you decide not to re-mesh the model (following changes made within the structure environment) it may happen that a previously defined mechanism is no longer valid. In this regard, the program warns us with the following message:

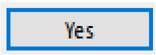


In this case it will be necessary to redefine the kinematics to proceed with the calculation.

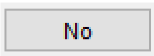
Mesh - Invalid Mechanisms

If you proceed with the mesh of a model in which invalid mechanisms have been defined, the program warns us with the following message:





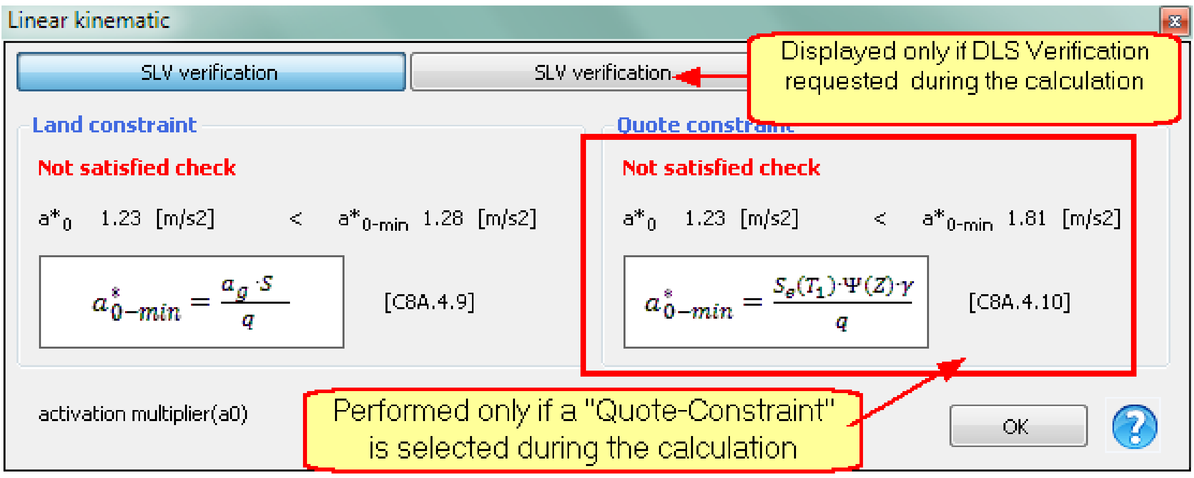
It proceeds by meshing the model, automatically eliminating all the mechanisms whose definition is not aligned with the data defined in the structure.



Stops, without meshing the model.

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

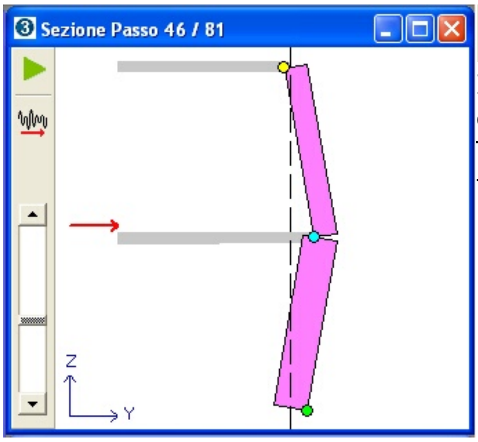
q: structure factor

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:

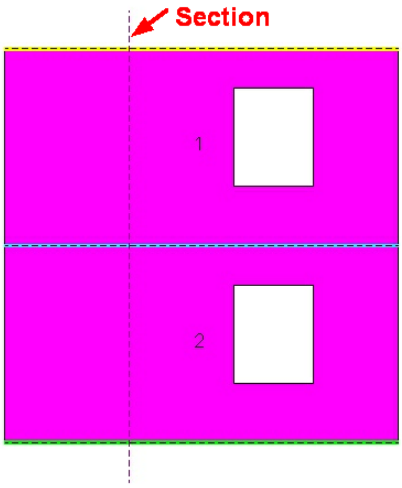
- **Land constraint** should only be conducted the verification with simplified structure factor q (linear kinematic analysis)
- **Quote Constraint** should be conducted both calculations (the verification with simplified structure factor q and the verification taking into account that the spectrum of 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.



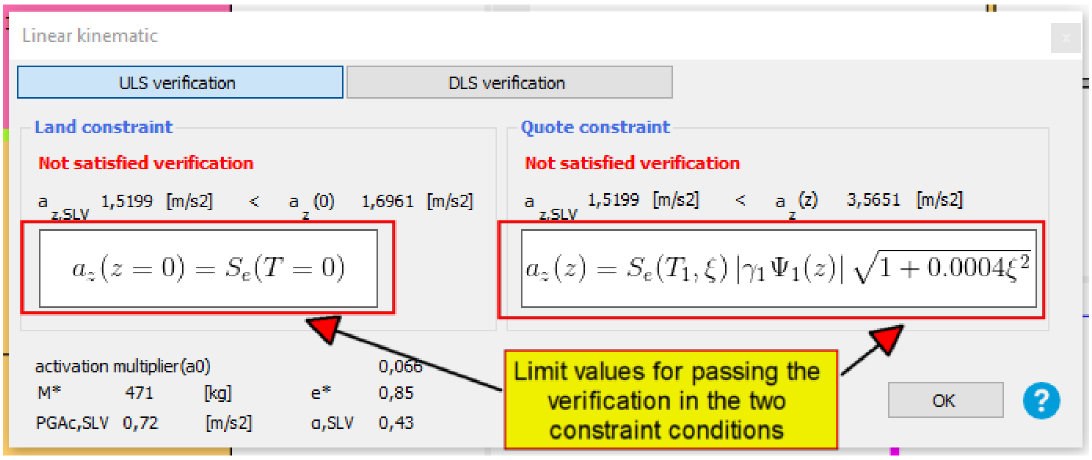
The section is represented in the wall front by a vertical underscore line.

 The button "move section line" allows you to replace the section line by clicking a point in the graphics area.

10.7.2.5.1 Specifications for NT 2018

These specifications become necessary and supplementary if the current legislation in Italy, NT 2018 with the corresponding reference circular, is chosen in the model parameters as active legislation.

At the end of the calculation, the results window is shown.
Pressing **"Check SLV"** or **"Check SLD"** shows the corresponding results.



Life limit state (SLV)
In the event that the verification concerns an isolated element or a portion of the building, however substantially resting on the ground, the condition for passing the verification is provided by the following inequality:

$$a_{z,SLV} \geq a_z(z=0) = S'_{e,SLV}(T=0) = a_{g,SLV} \cdot S$$

$$a_{z,SLV} = \frac{\alpha_0 \cdot g}{e^* \cdot FC} \cdot q$$

In the event that the local mechanism affects a portion of the building located at a certain altitude, it must be taken into account that the absolute acceleration at the altitude of the portion of the building affected by the kinematics is generally amplified compared to that on the ground; for this reason, passing the test is provided by the following inequality:

$$a_{z,SLV} \geq \max(a_z(z=0);\ a_z(z))$$

$$a_z(z) = \sqrt{\sum a_{z,k}^2(z)}$$

$$a_{z,k}(z) = S_{e,SLV}(T_k,\xi_k) \cdot |\gamma_k \cdot \Psi_k(z)| \cdot \sqrt{1 + 0.0004 \cdot \xi_k^2}$$

where:

- **Se (Tk, ξk)**: elastic response spectrum to the ground, evaluated for the equivalent period T and the equivalent viscous damping ξ (5%) of the non-structural element, the plant or the local mechanism considered;
- **γk:** k-th modal participation coefficient of the construction;
- **ψk (z):** value of the k-th modal form at height z in the position on the plan where the local mechanism to be verified is located;
- **aZ, k**: contribution of the k-th mode to the maximum plane acceleration.

On the basis of chapter §C7.2.3 it is allowed to refer only to the first mode of vibration that is significant for the significant direction, therefore k = 1 is assumed.

$$a_z(z) = S_{e,SLV}(T_1,\xi_1) \cdot |\gamma_1 \cdot \Psi_1(z)| \cdot \sqrt{1 + 0.0004 \cdot \xi_1^2}$$

Limit state of damage (SLD)

In the case of existing masonry buildings, also in consideration of the justified conservation needs, even though it is desirable to satisfy this limit state, its verification is not mandatory.

The same considerations, expressed for the Life Limit State (SLV), can be considered valid for the Damage Limit State (SLD), both in the case in which the verification concerns an isolated element or a portion of the building in any case substantially resting on the ground:

$$a_{z,SLD} \geq a_z(z=0) = S_{e,SLD}(T=0) = a_{g,SLD} \cdot S$$

$$a_{z,SLD} = \frac{\alpha_0 \cdot g}{e^* \cdot FC}$$

both in the case in which the local mechanism affects a portion of the building located at a certain height:

$$a_{z,SLD} \geq \max(a_z(z=0);\ a_z(z))$$

$$a_z(z) = S_{e,SLD}(T_1, \xi_1) \cdot |\gamma_1 \cdot \Psi_1(z)| \cdot \sqrt{1 + 0.0004 \cdot \xi_1^2}$$

10.8 Sensitivity Analysis

The Sensitivity analysis is a calculation method aimed to obtain better understanding of the structural functioning and accurate planning of the site investigation plan.
As known, doubts during modeling directly affect the evaluation of seismic safety. A specific example is materials mechanical properties, usually defined on the basis of reference values and for which, through investigation, it aims to limit the inescapable uncertainty.
Since the site tests have frequently high economic cost, the possibility to identify in advance (through the Sensitivity analysis) significant testing campaign points can limit investigation costs which result might not be of interest.
This kind of analysis can reduce the current uncertainty level on the structures analysis through the evaluation of the importance level of every parameter for which you want to investigate the significance.
A sensitivity index allows focusing the investigation only where it is necessary.
The methodology includes the identification of parameters groups that express the uncertainty degree, through the execution of multiple different non-linear analysis it identifies a level of sensitivity for each parameter in order to furnish a weight in terms of importance.

The main aim of the sensitivity analysis, to be executed through a sequence of nonlinear static analysis (pushover), is to identify the aleatory parameters that most affect the seismic capacity of the building.
There must be especially performed 2N + 1 analysis:
- the first by taking as a reference for the aleatory parameters the reasonable mean value
- in the other 2N analyzes all these parameters are maintained at their central value (average) of the range, with the exception of a parameter (or group of parameters) for which is taken the lower or upper limit of the above range.

The execution of pushover analysis implies the choice of different combinations and load conditions, in relation to:
1) forces distribution (for example proportional to the masses, derived from a triangular deformed shape or proportional to the first modal shape);
2) seismic action direction (X or Y);
3) the direction of this action (positive or negative);
4) the accidental eccentricity in each direction (usually defined in the regulations as 5% of the maximum size of the building in the orthogonal direction to that of the analysis).
Although in the final safety check all the different options should be considered (as explicitly required by the regulations), in order to limit the computational effort of the sensitivity analysis it can be selected the worst condition (corresponding to the minimum vulnerability index) by comparing the resulting capacity assuming plausible averages.

Bibliographic references:

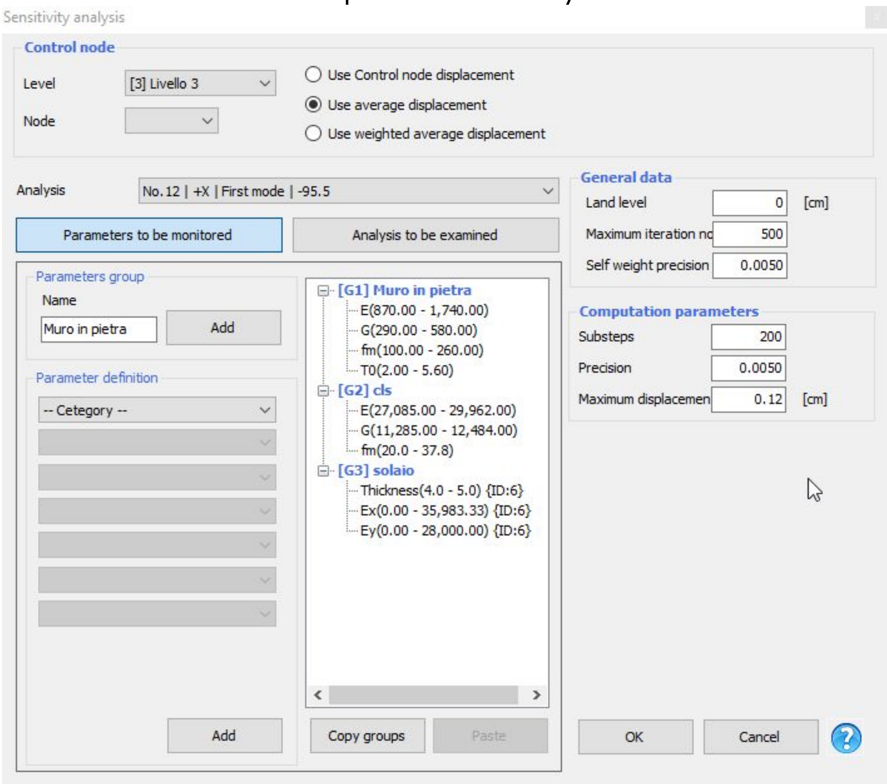
CNR-DT212 / 2013: Istruzioni per la valutazione affidabilistica delle sicurezza sismica costruzioni esistenti, Consiglio Nazionale delle Ricerche, Roma.

ANIDIS / 2015: L'incompleta conoscenza nella valutazione sismica di edifici esistenti:
definizione del fattore di confidenza attraverso analisi di sensibilità .
Serena Cattari, Jamil Haddad, Sergio Lagomarsino

Bull Earthquake Eng: Sensitivity analysis for setting up the investigation protocol and de?ning proper con?dence factors for masonry buildings.
S. Cattari · S. Lagomarsino · V. Bosiljkov · D. D'Ayala

10.8.1 Calculation

To access the calculation press "Sensitivity calculation" from the drop down menu.

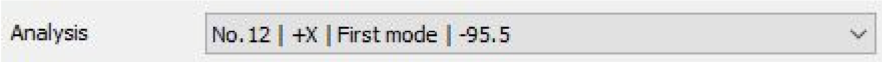


Input phases of the parameters to be monitored:

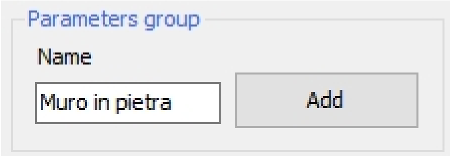
- Specify the pushover analysis in which to perform the calculation
- Insert groups [G1], [G2], [G3]..etc... according to which characteristics are to be examined
- Select the name of the group on the menu on the right in order to indicate the group in which to add a parameter
- Use the session "Define parameter" in order to define the parameter in the selected group

Following details of the various stages

Indicate on which pushover analysis to perform the calculation

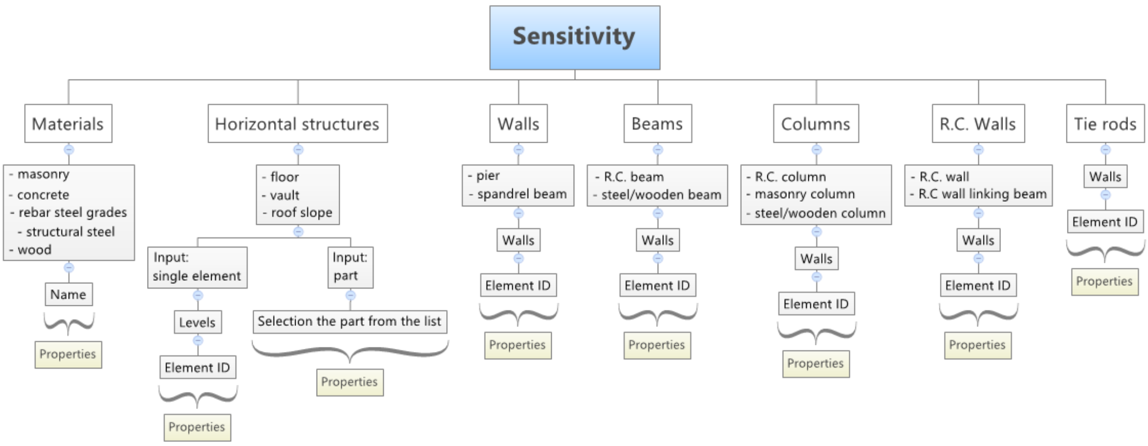


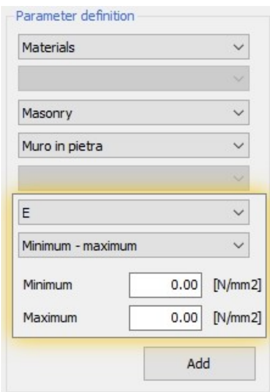
Create a new group:
Enter the name in the provided space and press [Add]



On the right is displayed on the blue menu the inscription **[G<*n*>] <*GroupName*>**
<*n*>: Identification number of the group created automatically by the program
<*GroupName*> : The name of the group inserted during the input phase

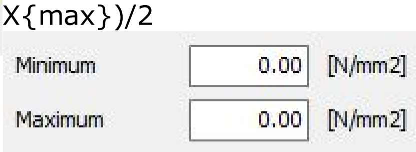
Parameter definition
To define a parameter you must first select a previously created group in order to define the belonging group.
A series of lists that must be filled in order from top to bottom to indicate the parameter type.
Choosing an option in the first list activates a second list and so on for subsequent lists.
The diagram below summarizes all possible options that can be created by progressively loading the necessary parameters.





Following the first lists that are used to locate the item on which to intervene, a list shows the parameters subject to sensitivity (in the example of the figure on the left it has been decided to take action on the E parameter).

The last list allows to define two different ways through which can be specified the variation range of that parameter:
Minimum-maximum: The variation range is defined by entering the minimum and maximum value.
If X is a generic parameter, it is assumed X{average}= (X{min}+ X{max})/2



Log-normal dispersion: X{average} corresponds to the characteristic entered in the model.

$X\{min\}=X\{average\}/\ e^{\sigma ln}$

$X\{max\}=X\{average\}\ e^{\sigma ln}$

σln: lognormal distribution parameter not symmetric

[Add] button:

Once selected the edited parameter and defined the variation range, by pressing the appropriate button will be added the parameter to the selected group.

P.S.:

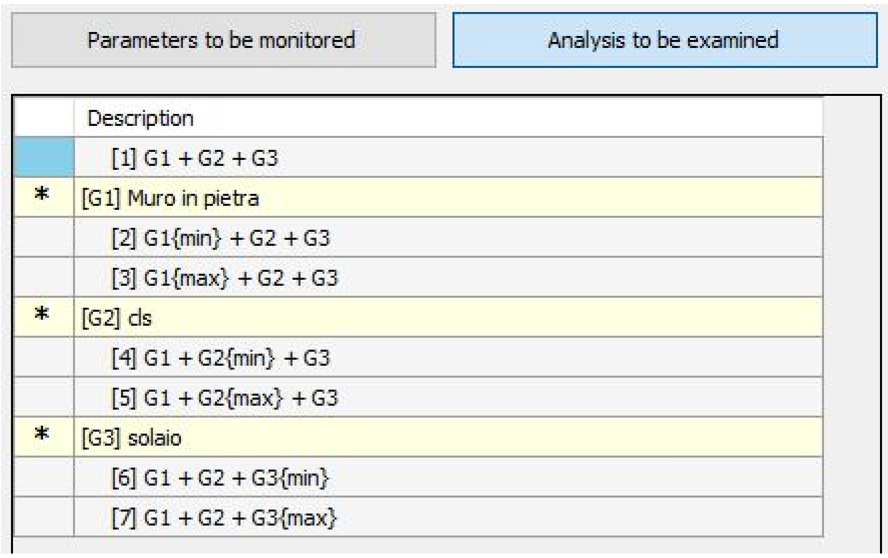
Immediately after pressing [Add], the lists that have led to the selection of the newly defined parameter remain active, allowing you to call up a different parameter from the last list without having to redefine all of the above and be able to immediately insert a new parameter to the group.

This feature proves to be extremely useful to define different parameters in the same group linked by common characteristics.

Example: Insert in sequence various mechanical properties of the same material

Analysis to be examined

After defining groups and parameters you can select "Analysis to be examined" to switch to the list of analyzes to be carried out created according to the defined groups.



The analysis is identified by a numeric identifier shown in square brackets [] .

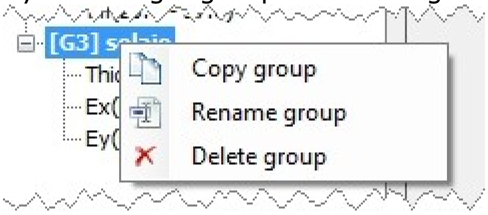
The analysis [1] is created with all the parameters set to their average value in the defined range.

Subsequent analyzes are grouped in pairs in the parameters group therefore the parameters are defined in the min or max range.

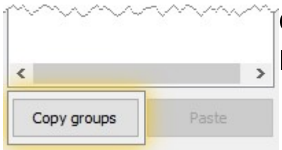
Edit parameters

Edit groups

By selecting a group with the right mouse button it appears the edit commands menu.



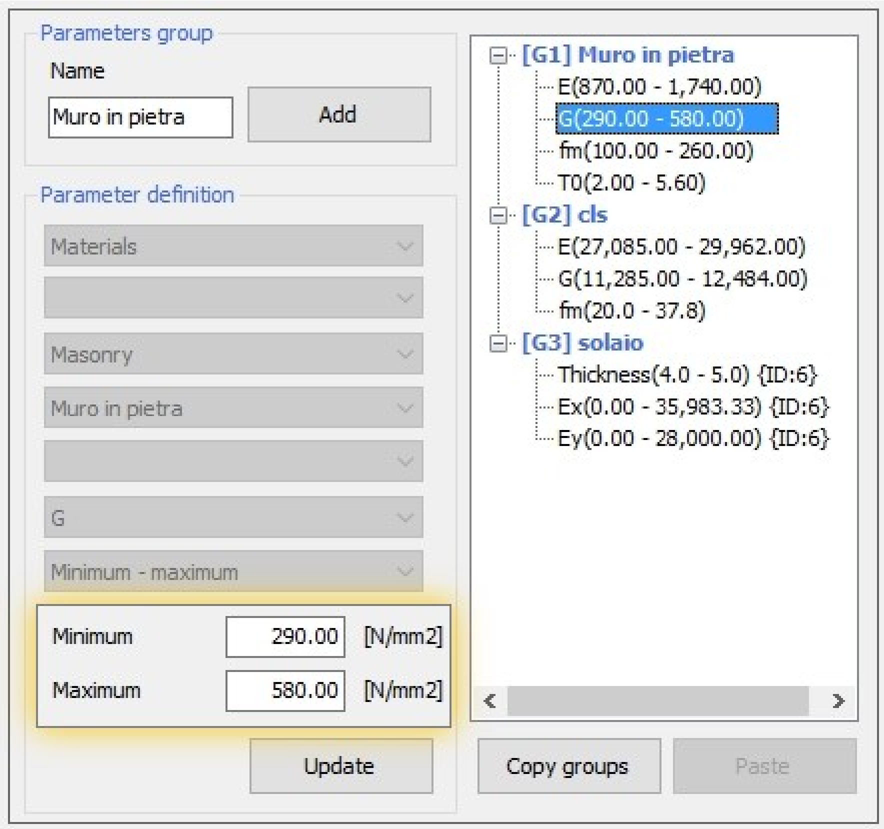
Copy group: Allows copying a group that can be pasted within another analysis. If you decide to monitor different analysis with the same group of parameters, following the "copy" command it can be pasted without manually re-entering the parameters.
Rename group: Allows changing the group name
Delete group: Deletes a group with all the parameters contained in it.

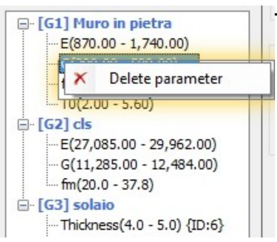


Copy groups: Allows copying all the groups of a predetermined pushover analysis for pasting them into another.

Edit parameters

Selecting a parameter from the menu, the "parameter definition" session shows the sequence that led to defining the characteristics of the selected parameter, allowing to figure out at which wall, floor or element it refers.
It is clearly not possible to edit the process that has led to define this parameter after the insertion but it is allowed to edit ranges where the parameter in question may vary.





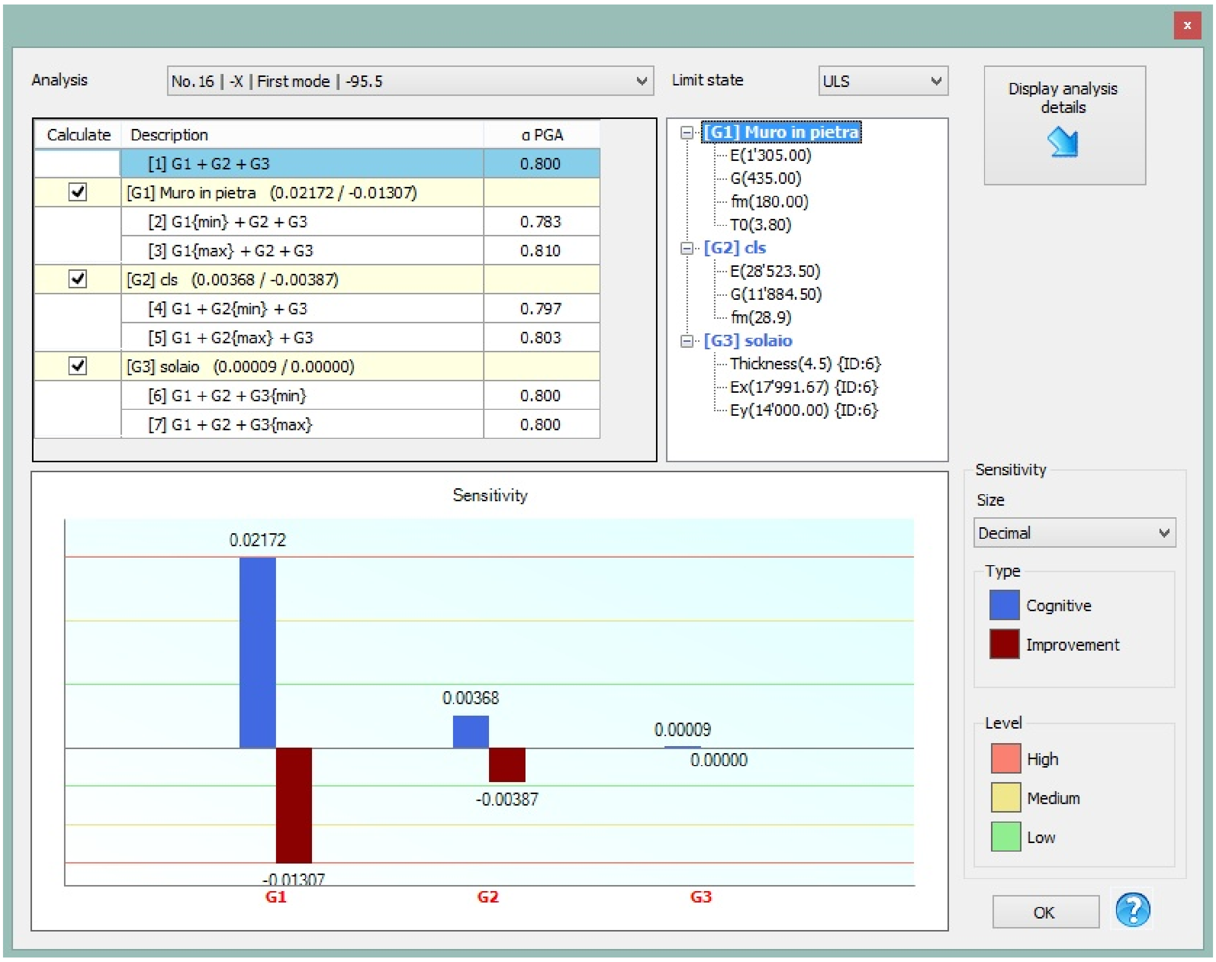
The command through which can be canceled a parameter appears by selecting it with the right mouse button.

Press [OK] to proceed with the sensitivity calculation (perform in cascade the analysis of the list).
A dedicated window poses the question "Do you want to run the selected analysis?"
[Yes]: The calculation begins
[No]: Exits without performing any calculations and the parameters defined in the groups are saved

Press[Cancel] to exit the calculation window abandoning any change.

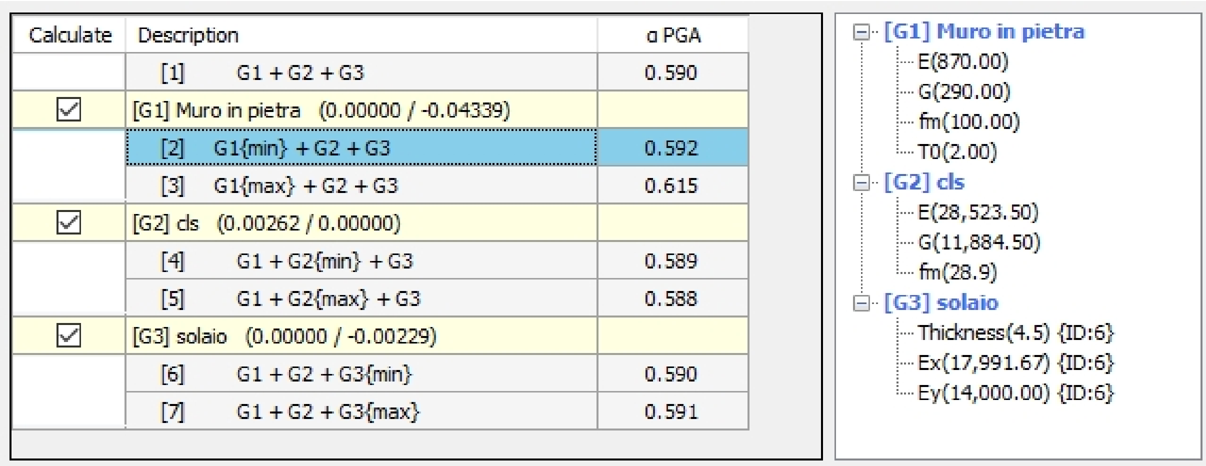
10.8.2 Results

To access the sensitivity analysis results, press "Sensitivity results" from the drop down menu.



In the upper part of the screen is necessary to select the analysis and the limit state of the required results.

The table shows the list of the performed analysis.
Except for the analysis [1] that is in correspondence of all the groups average values, the following are grouped according to the group that is set to min or max values.
For each analysis is displayed the vulnerability index on the accelerations and for each group the matching sensitivity index.



Calculate	Description	α PGA
	[1] G1 + G2 + G3	0.590
☑	[G1] Muro in pietra (0.00000 / -0.04339)	
	[2] G1{min} + G2 + G3	0.592
	[3] G1{max} + G2 + G3	0.615
☑	[G2] cls (0.00262 / 0.00000)	
	[4] G1 + G2{min} + G3	0.589
	[5] G1 + G2{max} + G3	0.588
☑	[G3] solaio (0.00000 / -0.00229)	
	[6] G1 + G2 + G3{min}	0.590
	[7] G1 + G2 + G3{max}	0.591

By selecting a single analysis with the mouse, it is possible to see on the right menu precisely the values used in the calculation of the corresponding analysis.

In the lower part a diagram shows the sensitivity indexes of the various groups.

Two different types of sensitivity are shown:

«Cognitive» Sensitivity:

The existence of a parameters combination that may provide worse conditions than the average is searched from the examination of all the carried out analysis combinations. Following an example of cognitive sensitivity index calculation for the [G1] group.

$$\alpha_{mean} = \alpha_{[1]} \quad ; \quad \alpha_{min} = min(\alpha_{mean}; \overbrace{\alpha_{[2]}; \alpha_{[3]}}^{[G1]}) \quad ; \quad I_s = \frac{\alpha_{mean} - \alpha_{min}}{\alpha_{mean}}$$

Given the mathematical formulation, this parameter will always be **>0** or eventually **=0** if the reference parameter group is not significant for the result or if the average value corresponds to the worst condition.

The use of this parameter is a prerequisite for the choice of the points of the structure in which to conduct the tests/surveys.

«Improving» Sensitivity:

The existence of a parameters combination that may provide better conditions than the average is searched from the examination of all the carried out analysis combinations. Following an example of improving sensitivity index calculation for the [G1] group.

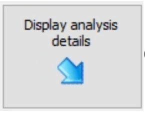
$$\alpha_{mean} = \alpha_{[1]} \;;\; \alpha_{max} = max(\alpha_{mean}; \overbrace{\alpha_{[2]};\alpha_{[3]}}^{[G1]}) \;;\; I_s = \frac{\alpha_{mean} - \alpha_{max}}{\alpha_{mean}}$$

Given the mathematical formulation, this parameter will always **<0** or eventually **=0** if the reference parameter group is not significant for the result or if the average value corresponds to the best condition.

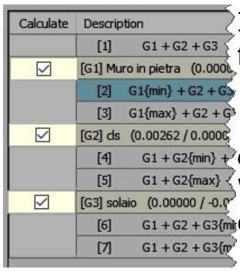
The use of this parameter is a prerequisite for the choice of intervention that is more efficient to improving.

The height of "*colored columns*" in the diagram, represents a "weight" in terms of importance of the individual group compared to the others.
High columns, correspond to "sensitive" parameters for the result that is worth investigating whether it is cognitive sensitivity or elements that is useful to reinforce in case of improving sensitivity.

In order to improve the reading of the sensitivity indexes it is possible to show the numeric value in *Decimal, Scientific* or *Percent* format.



By selecting one of the analyzes from the list and by pressing this button you can enter the analysis details.



In the performed analysis table, a check mark may exclude a group from the sensitivity index count.

When does it makes sense to exclude a group from the sensitivity calculation?
When from details of the analysis calculation comes out a result that is considered to be insignificant or even distort for sensitivity.

10.9 Bending analysis out of plan

The out of plan verifications can be performed separately, and the equivalent forces can be taken to the non-structural elements, assuming qa = 3.
More precisely, the seismic action perpendicular to the wall can be represented by a distributed horizontal force, equal to Sa / qa times the weight of the wall.
For the earthquake-resistant walls, it can be assumed for Sa the following expression:

$$S_a = \alpha \cdot S \cdot [1.5 \cdot (1 + Z/H) - 0.5] \geq \alpha \cdot S$$

where:
α: the ratio of the maximum acceleration of soil ag to subsoil type A for the limit state under examination and gravity acceleration g;
S: coefficient that takes account of the subsoil category and the topographical conditions;
Z:quote of barycenter of the non-structural element measured from the foundation plan;
H: height of the building measured from the foundation plan;

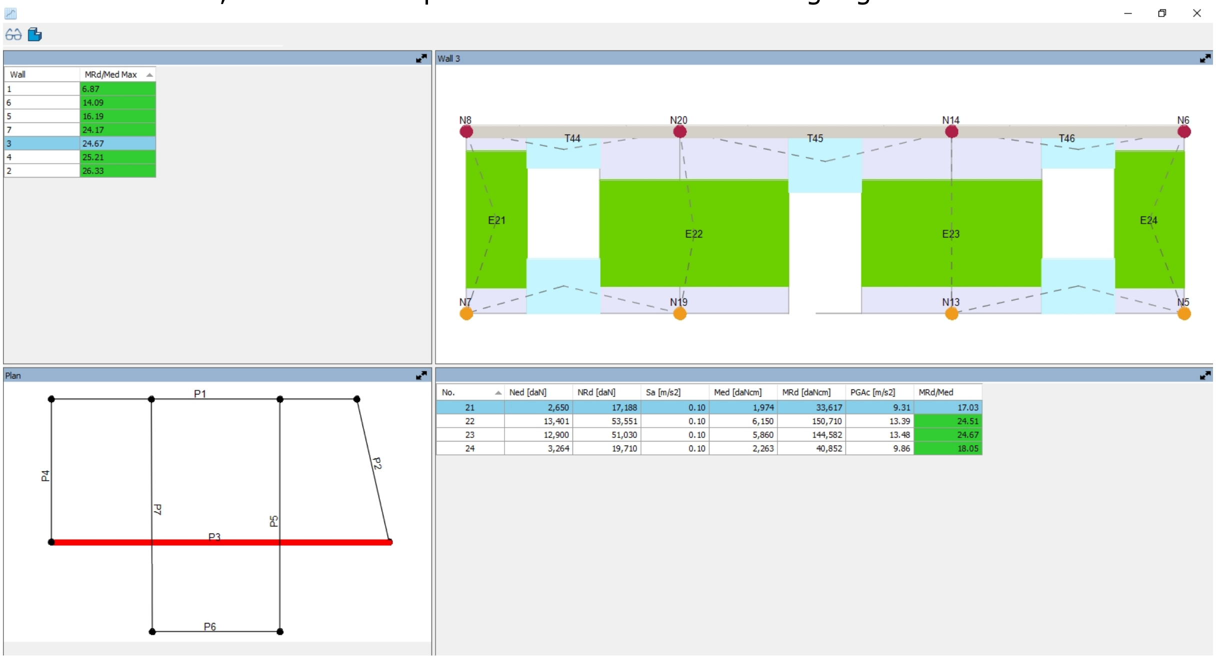
Known the Sa/qa ratio you can compute the force Fh=N*Sa/qa representing the horizontal force at the center of gravity of the pier.
We have the maximum moment at mid-height of the pier assuming the configuration of pier hinged in correspondence of the slabs that produces a moment produced by the

transformation of the load Fh in distributed load q=fh/he equal to Med=q he²/8.
The resistant ultimate moment is provided by the following formulation:

$$M_{Rd} = \left(l \cdot t^2 \cdot \frac{\sigma_0}{2}\right)\left(1 - \frac{\sigma_0}{0.85\, f_d}\right)$$

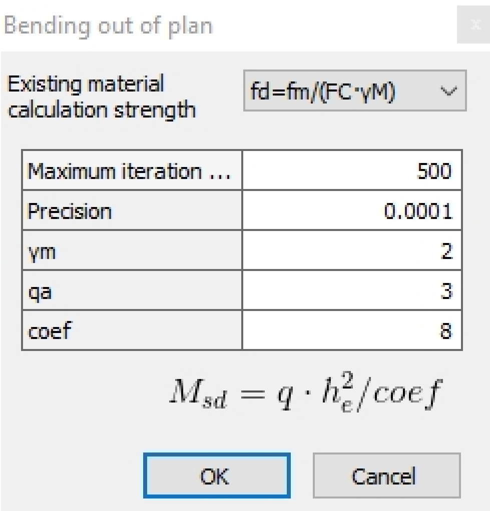
The verification will be exceeded if the MRd/Med ratio is greater than one.

The result environment is divided into 4 main areas.
In the top left, the list of walls with the corresponding value of the lower Mrd/Med ratio between all the wall piers.
In the upper right, the wall panel with the piers that did not pass the checkout highlighted in red.
In the bottom right, check details for each single pier.
In the lower left, the structure plan with the current wall highlighted.

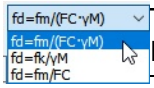


Moving from one wall to another can be simply by clicking on the plan in the point of interest or by using the list of walls by clicking on the corresponding row.

From the "*Settings> Bending out of plan*" menu, you can set the main parameters of the calculation:



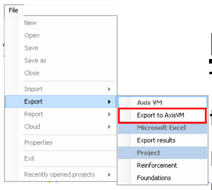
The resistance calculation of the existing material is not clearly explained how it should be calculated within the regulations for this type of verification; for this reason, we propose different alternative formulas, suggesting as the default what we feel most appropriate for this verification.



The first formulation of the list is proposed by us as a default because we believe it is scientifically more reasonable.
The second formulation, apparently consistent with the legislation, it is actually affected by the problem that the calculation of fk is approximated if obtained from fm and is likely to be extremely punitive.

10.10 FoundationFEM

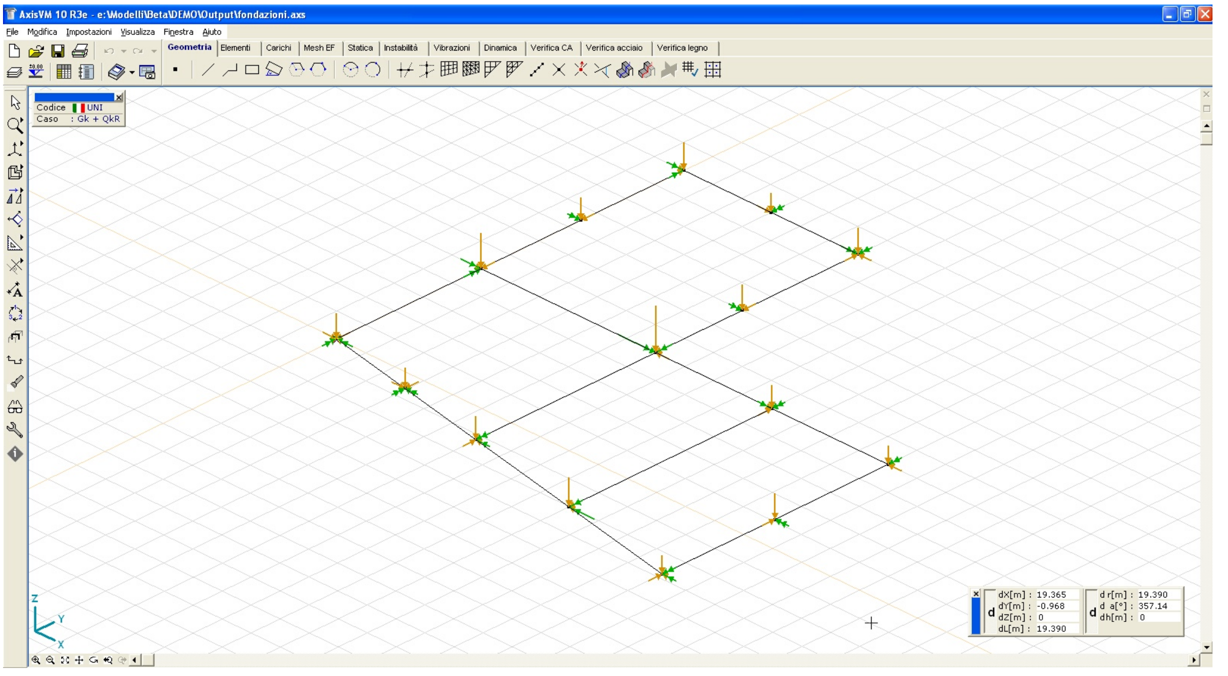
The program 3Muri has several modes of export for the calculation of the foundation structures.
The exportation takes place to the software Axis VM (finite element program distributed by S.T.A. DATA srl.).



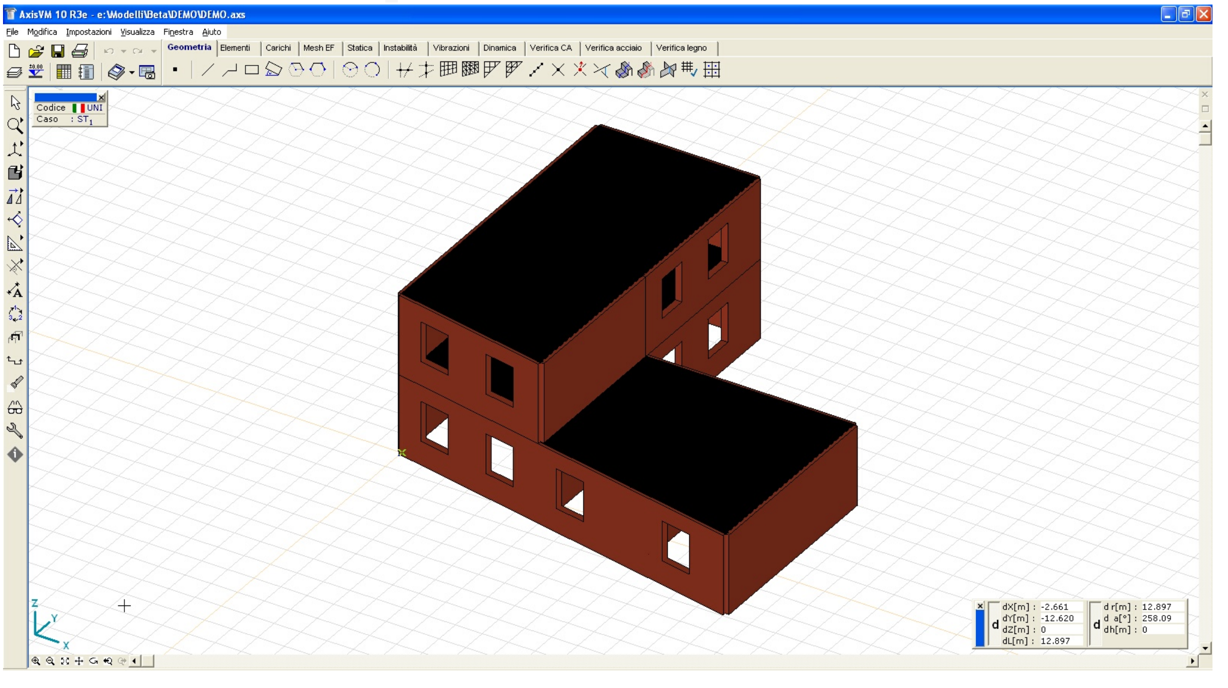
Export foundations:
Through this form you can transfer the results of the loads of the foundation from the program 3Muri to Axis VM. The module Loads foundation creates a new model in which the lines corresponding to the plant of the walls are drawn automatically to allow the tracking the plant of the foundations directly from the interface of the program Axis VM.

Export Model:
Through this form you can transfer the entire model by the program 3Muri to the program AxisVM.

Example export foundations



Example export the entire model

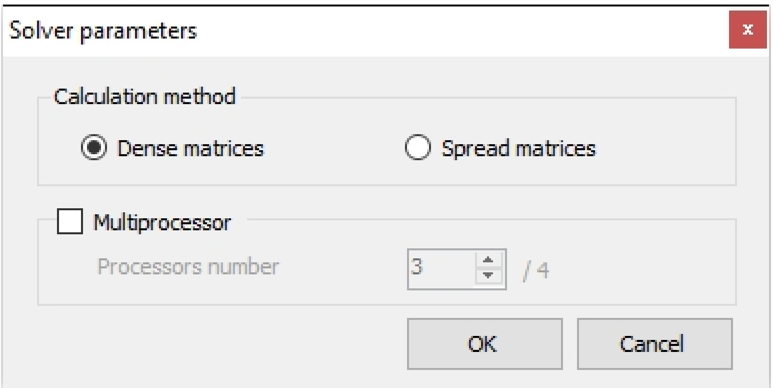


Exporting the entire model is a good way to take masonry stiffness into account when calculating foundations.

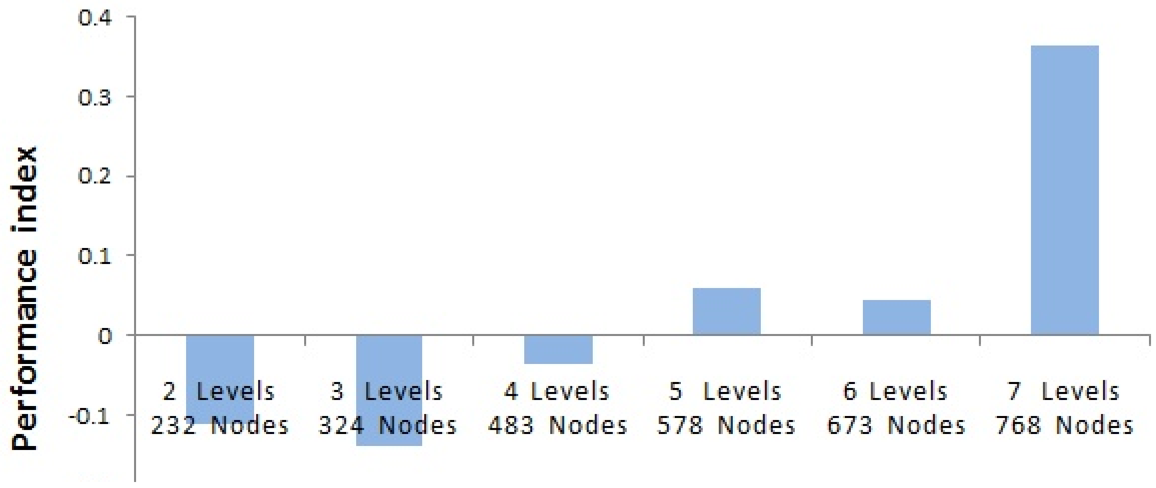
10.11 Solver parameters

It is possible to access to this environment directly from the menu "settings."

Here are present two different calculation settings regarding the processor.



As an example, below is a diagram produced by a test on a 2-level building.
An index has been obtained from the calculation times *(Sparse Calculation Time) - (Dense Calculation Time) / (Dense Calculation Time).*
When this index is negative it means that the computation time with the sparse matrix procedure is slower.
A positive index represents the increased speed of the sparse method.
From the starting model of two levels, others were produced by the duplication of the last level in order to progressively increase the size of the model and to study the yield of the sparse matrix approach.



The example highlights how there is a greater efficiency of the method with sparse matrices starting from the fifth level.
Starting from the seventh level, the time gain is significantly greater due to the achievement of maximum memory utilization with the dense matrix method.

Calculation method:

- Dense matrices
- Spread matrices

The 3Muri program performs the calculation through the FEM method (finite elements), in the base of which exists the knowledge of the stiffness matrix K.
The classical engineering applications give rise to stiffness matrices with a high number of null values inside them.
In numerical analysis, a spread matrix is a matrix whose values are almost all equal to

zero.

When the sparse matrices are stored and managed on a computer, it is useful and often also a necessity, to use specialized algorithms and data structures that take into account the spread nature of matrix. Performing the operations by using the structures and the usual matrix algorithms (dense method) results a very slow operation, and also leads to great waste of memory, if the matrix to be managed have big dimensions. The spread data are, by their nature, easily compressible, and their compression almost always involves a significantly lower usage of memory. It is also true, however, that some very large spread matrices are impossible to manage with the standard algorithms.

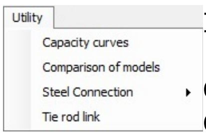
The stiffness matrices of masonry structures calculated with the equivalent frame method, averagely are of small dimensions. In such cases, a calculation to spread matrices does not show many advantages. Paradoxically, in the buildings with contained size the "conversion" time from dense to spread could lead to more overall calculation times; The approach to the spread matrices is therefore not always convenient in terms of time saving. We therefore feel to advise to all our customers to use this method of calculation in cases where the performance of the PC in use are not sufficient to produce a result, or in cases of buildings with important dimensions.

Multiprocessor:

By activating this option it is possible to conduct the calculation simultaneously on multiple processors.
In the case of multiple analysis (pushover 24) is possible to address an analysis on each processor available on the PC.
The default proposes to use a number of processors equal to the maximum number minus one, the unused processor is left available to the system resources.
The saving in terms of time therefore depends on the number of processors available by the system.
Example:
In the case of PC with 4 processors dedicated to the calculation, the total time taken on the 24 analysis will be about 1/4 of the time spent with a single processor!

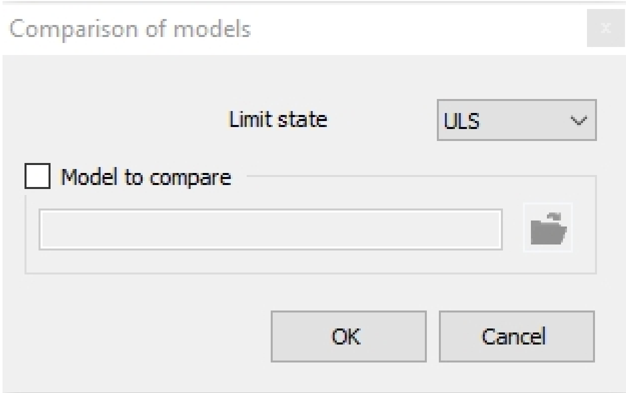
10.12 Utility

From the File menu, you can access the **"Utility"** category.



In this category you can find the following items:

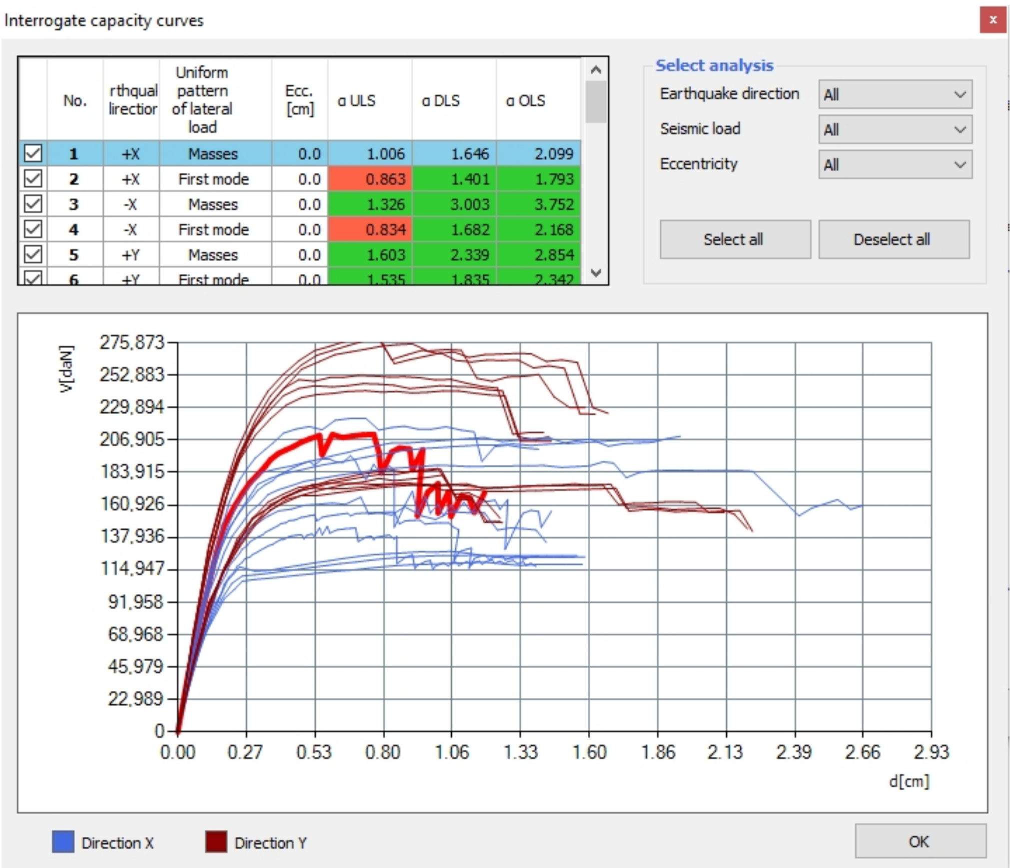
Capacity curves
Comparision of models
It compares different models for a given limit state.



Steel Connection
Tie rod link

10.12.1 Overlapping capacity curves


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



The capacity curves area is composed of three principal elements:
- Analysis table
- Overlapped pushover curves
- Selection filters

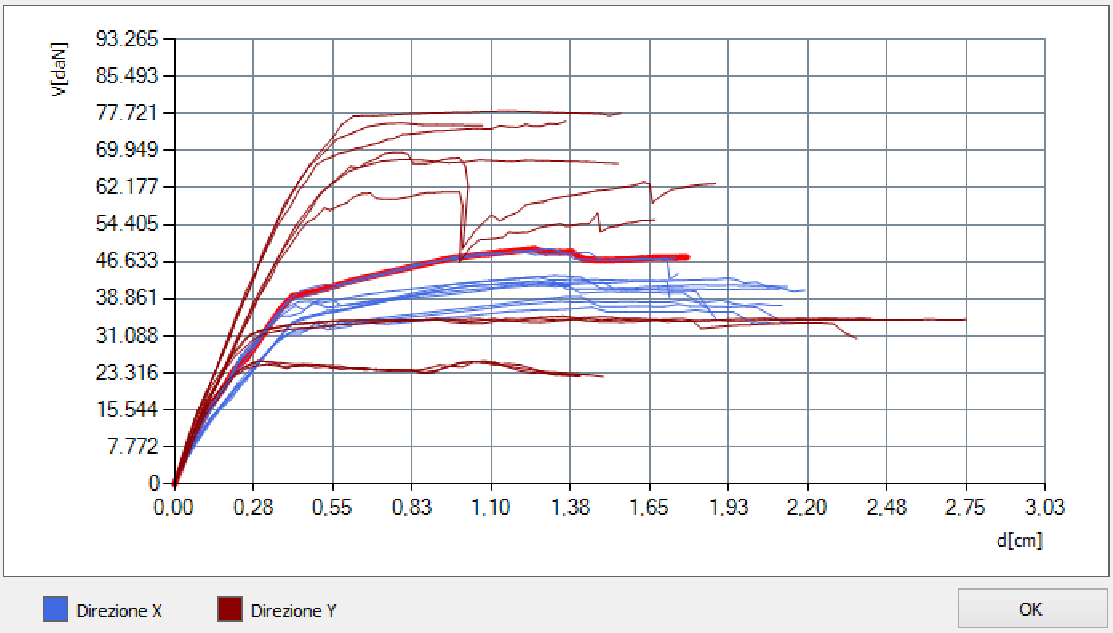
Analysis table

Contains a summary of the analyzes for the current model.
The first columns describes the analysis type, the last show the vulnerability indexes for each of the three limit states.
The background color, green or red, distinguishes the positive analyzes from the negative ones.
The yellow color shows the two analyzes that have the lowest vulnerability indexes (more significant for calculation purposes).

	No.	rthqual lirectior	Uniform pattern of lateral load	Ecc. [cm]	α ULS	α DLS	α OLS
☑	**13**	-X	Masses	95.5	1.353	2.202	2.826
☑	**14**	-X	Masses	-95.5	1.284	2.606	3.361
☑	**15**	-X	First mode	95.5	0.850	1.615	2.080
☑	**16**	-X	First mode	-95.5	0.800	1.688	2.177
☑	**17**	+Y	Masses	77.4	1.583	2.317	2.824
☑	**18**	+Y	Masses	-77.4	1.614	2.354	2.875

The selected analysis is highlighted in blue, that shows the relative push-over curve, highlighted by a red thick line, in the "Overlapped push-over curves" graph.
Another analysis is highlighted in blue by clicking on a row in the table and the corresponding curve is selected on the graph.
It is also possible to disable some analysis by manually unchecking the first column, the corresponding push-over curves will be automatically excluded from the graph.

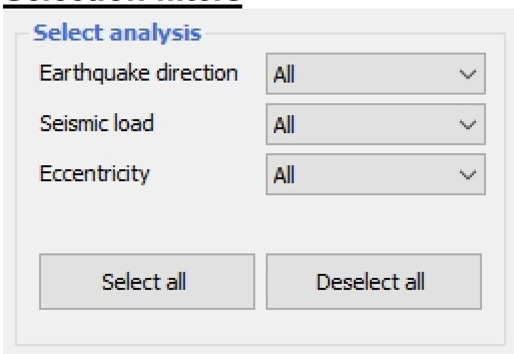
Overlapped pushover curves



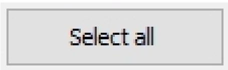
Shows the capacity curves of all the calculated and active analysis.
The curves shown in blue are relative to the X direction while those represented in brown are relative to the Y direction.
The selected curve is highlighted by a red thick line.
It is also possible to change the selection by clicking on another curve, the relative

analysis in the "Analysis table" will be automatically highlighted in blue.

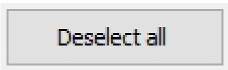
Selection filters



By using the proper space, there can be selected multiple analysis by activating the selection filters.

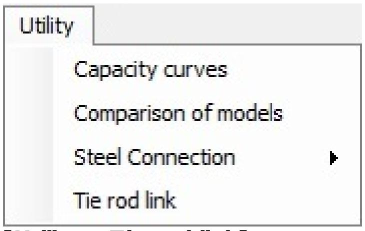


Activate all the analysis and relative curves currently disabled.



Disables all the analysis and relative curves currently enabled.

10.12.2 Tie rod link

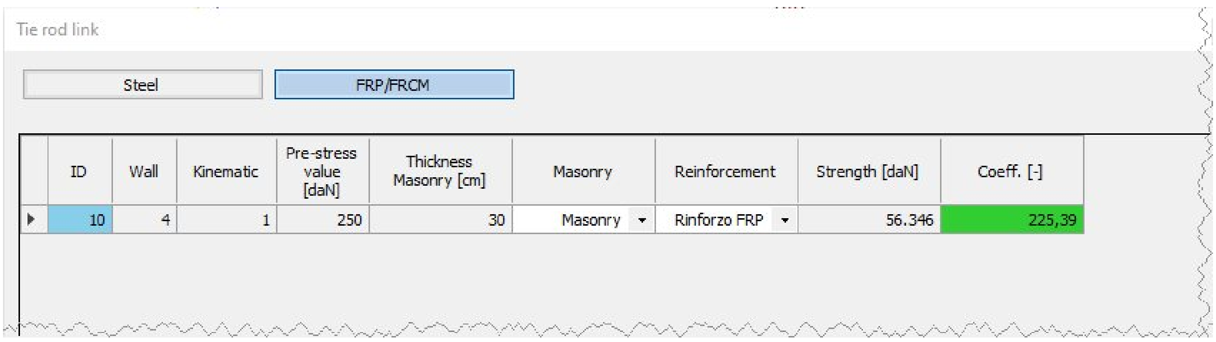


[Utility > Tie rod link]

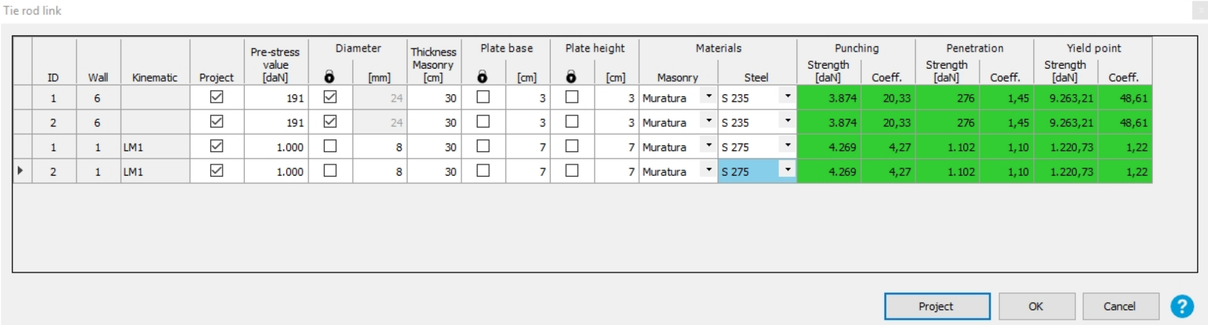
Calling this command shows the window that checks the inserted tie rods.

The tie rods can be of two types:

FRP/FRCM tie rods



Steel tie rods



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

: This column allows to block, in the design phase, the corresponding dimension (eg in the case of the plate the user can block the base dimension and only increase the height).

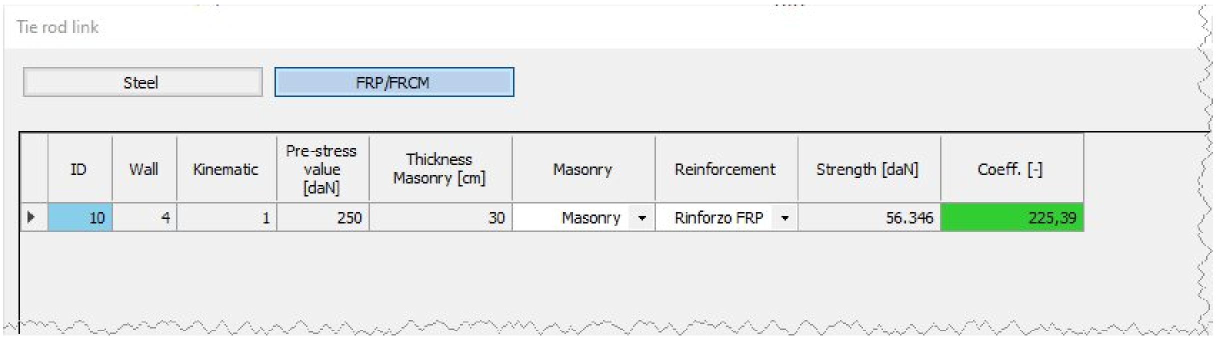
The tie rods considered belong to two different types:

Tie rods as reinforcement intervention of local mechanism

They are the tie rods inserted only for the purpose of preventing the activation of the mechanism and are not present in the global model.

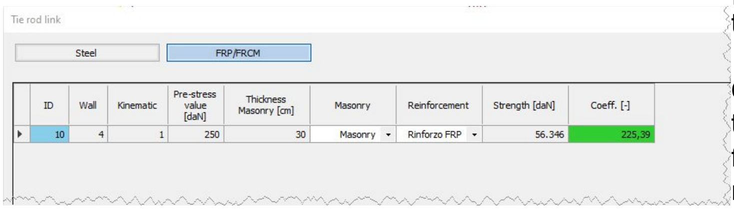
They can be inserted both in steel and in FRP / FRCM.

First of all it is necessary to define, within the environment dedicated to kinematics, the position, the type and the value of the tension of the tie rod to be inserted.



ID	Wall	Kinematic	Pre-stress value [daN]	Thickness Masonry [cm]	Masonry	Reinforcement	Strength [daN]	Coeff. [-]
10	4	1	250	30	Masonry	Rinforzo FRP	56.346	225,39

After defining this data, you need to reselect the "Tie rod link" option to view the calculation data for the tie rod.

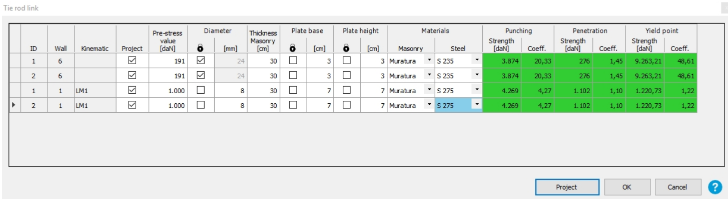


ID	Wall	Kinematic	Pre-stress value [daN]	Thickness Masonry [cm]	Masonry	Reinforcement	Strength [daN]	Coeff. [-]
10	4	1	250	30	Masonry	Rinforzo FRP	56.346	225,39

[Kinematics] column: the name of the mechanism is shown

Column [Pre-stress value]: it is the tension computed in the window for defining the load of the mechanism in the tie rod session

[Coeff.] Column: ratio between [Resistance] / [Pre-stress value]
[Kinematics] column: the name of the mechanism is shown


Tie rod link

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

Project OK Cancel

Column [Pre-stress value]: it is the tension computed in the window for defining the load of the mechanism in the chains session

Columns [Tie rod diameter] / [Plate base] / [Plate height]: these are the columns that are designed / tested

[Punching] / [Penetration] / [Yielding] columns: these are the checks carried out with the relative safety factor.

Tie rod from a global model

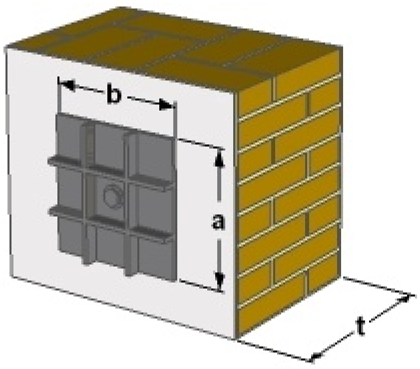
The calculation is carried out for the tie rod ends defined with the appropriate function in Graphical layout of steel elements.

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

Selecting this command is required to indicate the nodes of which you want to proceed with the calculation.

10.12.2.1 Tie rod link - Verifications

The verification of the tie rod-plate-masonry system provides the control of three rupture mechanisms.



1 - Punching verification of the masonry in anchorage areas

The puncture resistance is given by:

$$T_{pun} = f_v \cdot [2 \cdot (b + t) + 2 \cdot (a + t)] \cdot t$$

where :
- t is the thickness of the masonry
- b is the width of the plate
- a is the height of the plate
- fv is the calculating shear strength of the masonry

2 - Penetration verification of the anchor

The penetration of the anchoring in the masonry is given by the overcoming of the masonry compressive strength of the plate contact pressure. The penetration resistance of the anchor is given by:

$$T_{pen} = f_d \cdot a \cdot b$$

where :
- fd is the calculating compressive strength of the masonry

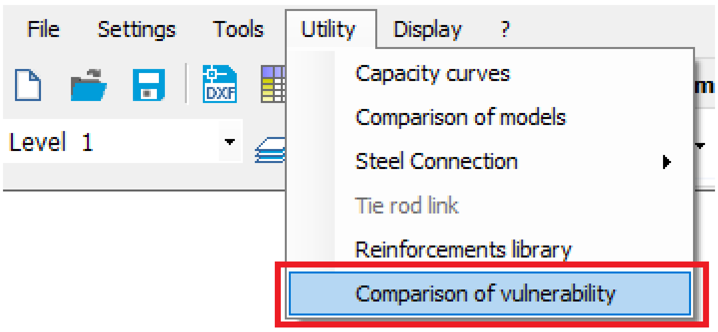
3 - Yielding verification of the tie rod

The yield strength of the tie rod is given by:

$$T_{tir} = f_y \cdot \frac{\phi^2 \cdot \pi}{4}$$

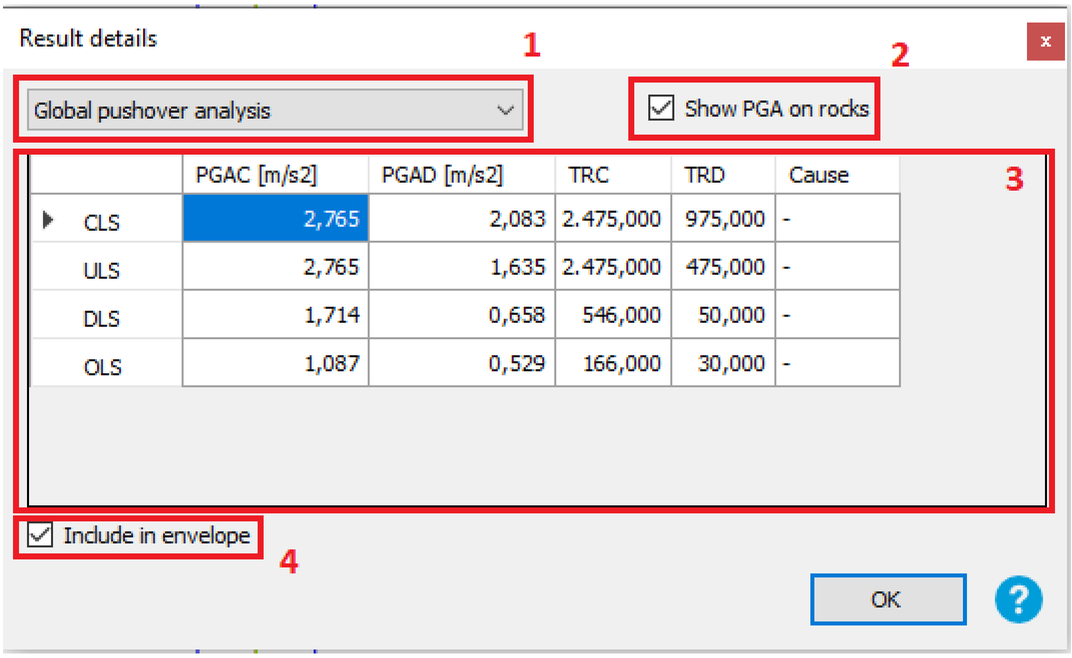
where :
- Φ is the diameter of the tie rod
- fy is the calculating strength of the tie rod

10.12.3 Comparison of vulnerability

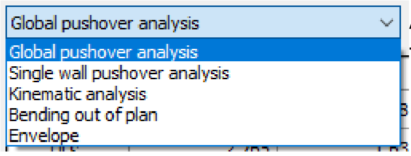


In the menu **[Utility> Vulnerability comparison]** it is possible to view a summary of the values of the vulnerabilities obtained for all the analyzes carried out.

The window that allows you to consult the values is the following:



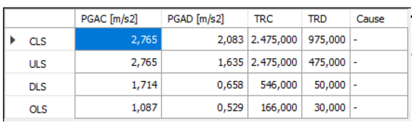
	PGAC [m/s2]	PGAD [m/s2]	TRC	TRD	Cause
CLS	2,765	2,083	2.475,000	975,000	-
ULS	2,765	1,635	2.475,000	475,000	-
DLS	1,714	0,658	546,000	50,000	-
OLS	1,087	0,529	166,000	30,000	-

1 

Allows to select the type of analysis for which you want to view the results obtained.

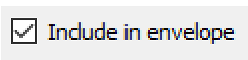
2 

If the box "Show PGA on rocks" is checked, all accelerations (PGA) are evaluated, by convention, on rigid ground (A).
Some technical data sheets require the calculation of the aforementioned accelerations on the reference soil, in this case it will be necessary to multiply the value calculated by the program by the factor "S" (S = SS * ST), defined in the parameters of the spectrum.
The product between acceleration and "S" factor is obtained by removing the selection from the "Show PGA on rocks" box.

3 

	PGAC [m/s2]	PGAD [m/s2]	TRC	TRD	Cause
CLS	2,765	2,083	2.475,000	975,000	-
ULS	2,765	1,635	2.475,000	475,000	-
DLS	1,714	0,658	546,000	50,000	-
OLS	1,087	0,529	166,000	30,000	-

Table containing the summary, for each limit state, of the results obtained from the selected analysis.

4 

If the "Include in envelope" box is checked (for the considered analysis) the relative results will be included in the choice of data shown in the "Envelope" configuration.

ES:
I check the "Include in envelope" option for the "Global Pushover" and "Kinematic Analysis" analyzes.
Selecting the "Envelope" option (1) I will find in table (3) the data deriving from the envelope of the results of the

global and kinematic pushvover analyzes.

PGAc not defined

In cases of spectrum defined by user input or custom spectrum it is not possible to determine the data relating to the capacity PGA.
The PGAc shown in this window is directly linked to the capacity return period of the structure. In the pushover calculation is identified by means of a rigorous numerical procedure, that requires the presence of the "reference grid" which allows a direct association between the periods of return and acceleration dependent on it.

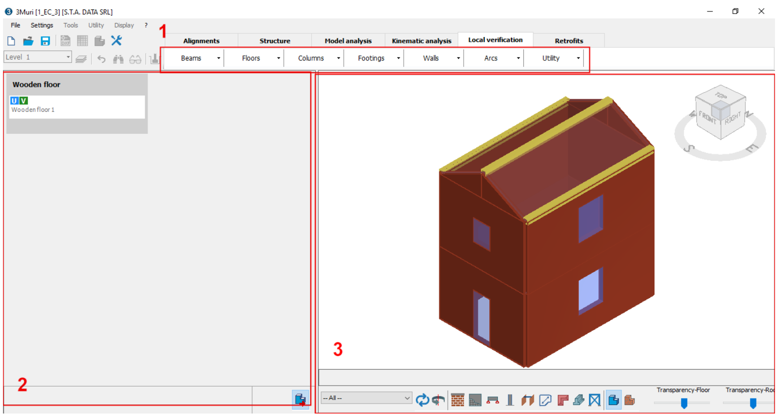
11 Local verification

The "Local verification" environment is dedicated to the local verification of masonry, reinforced concrete, steel and wood elements.

11.1 Environment interface

All the modules and functions are contained within the "Local verification" TAB.

The environment is structured as follows:



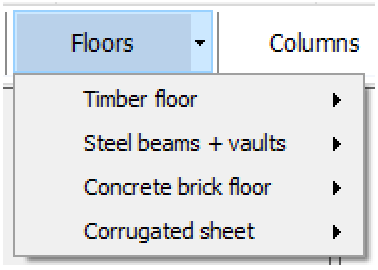
1.

This interactive menu presents an item for each type of element.
By clicking on the small arrow on the side, you can access the list of available calculation modules.

2.

Within this side area the calculation cards created are displayed and managed. The content of this area depends on the type of elements selected in the main menu [1].

Example:
I select the item "Floors" and I am offered all the types of floors of which it is possible to make a calculation.



3.

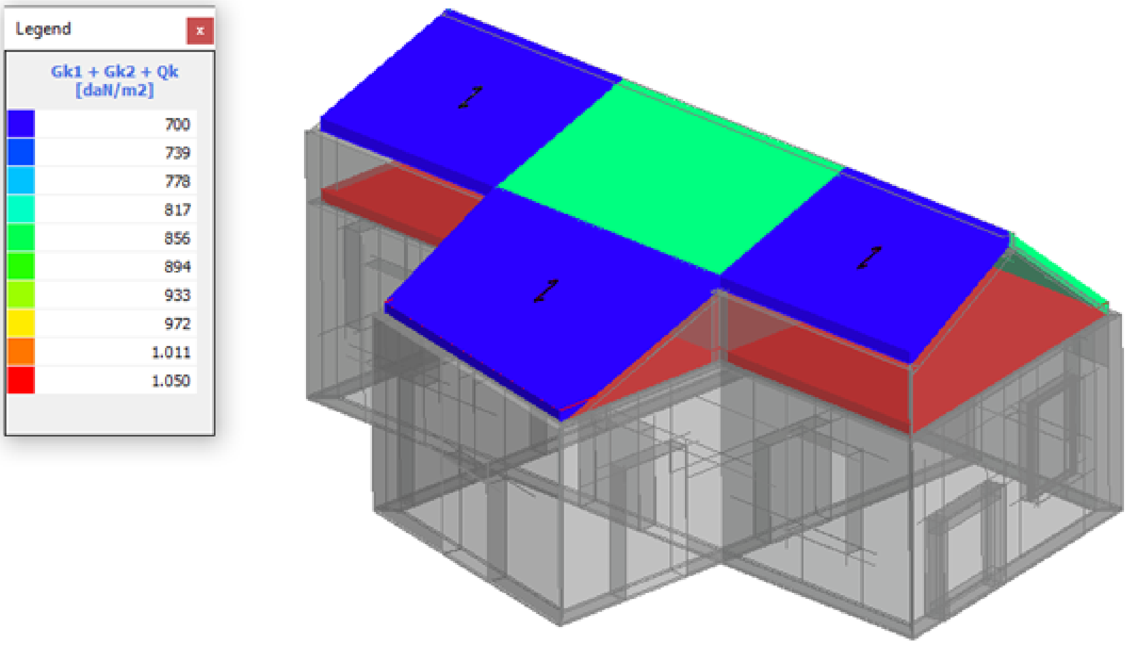
Model space in which it is possible to view the three-dimensional model of the structure and, interacting with it, create the tabs of the elements to be calculated.

It is possible to perform local verifications in two ways:

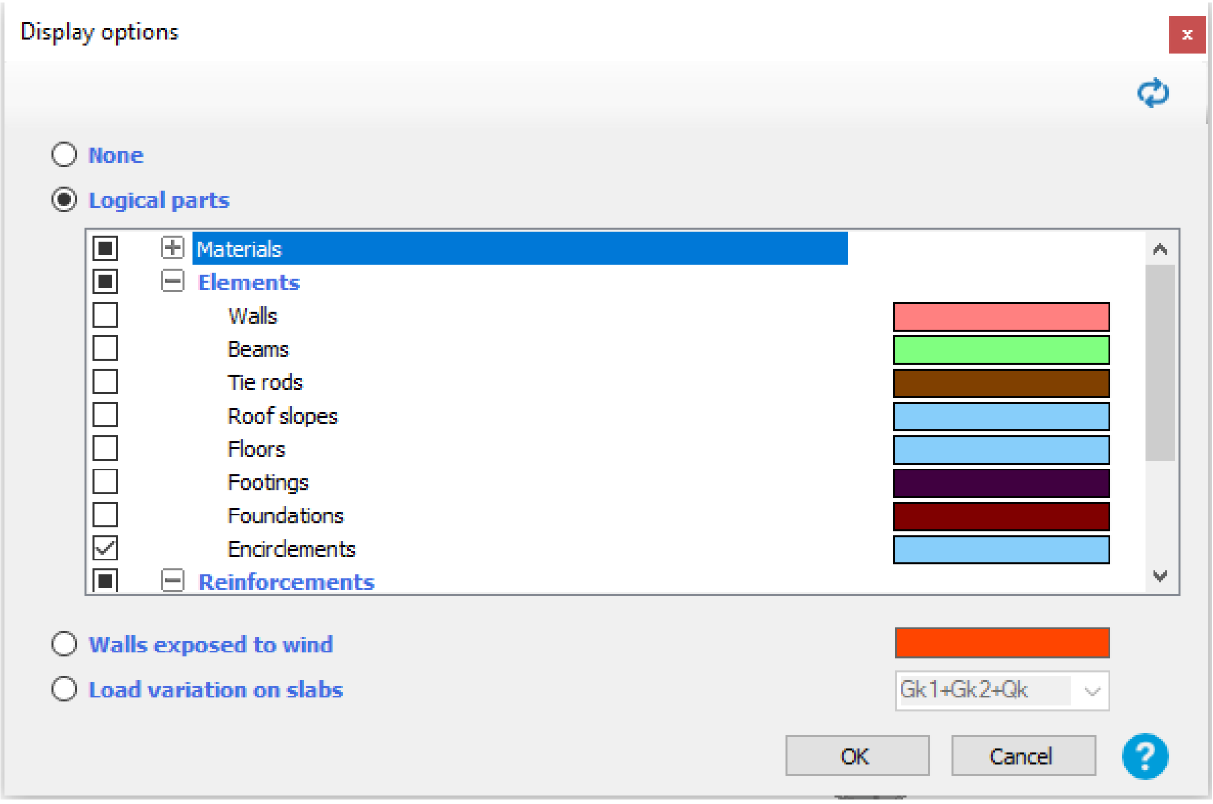
- Using the data from the model
- Without using the model data, entering it manually.

11.1.1 3D

Legend color slabs: If the display option Load variation on slabs is enabled, it allows the visualization of the color legend.



Display options: Opens the window for setting the display options



There are three different types of display options:

Logical parts

It allows to draw the 3D elements with customized colors according to the material they are made of, according to their type (walls, beams, columns, etc.) or according to the reinforcements or retrofits applied to them.
To include an item in the display options, simply check the box on the left.
The color can be customized by double clicking on the colored rectangle; default colors can be restored by clicking on the icon .
If an object satisfies several filters at the same time (for example for its type and for its material), it will be colored according to the last valid option.

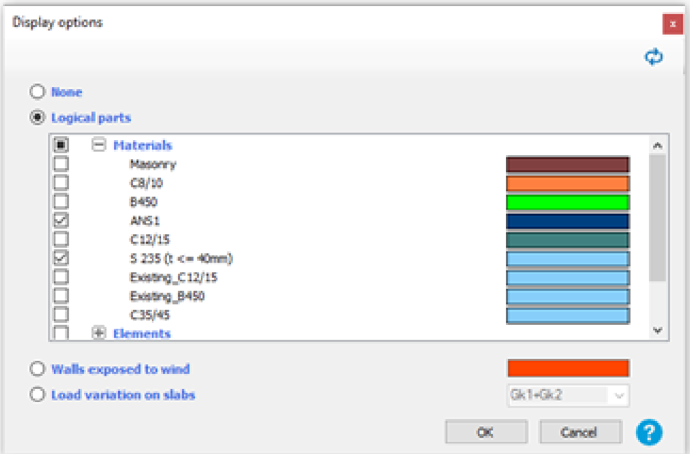
Walls exposed to wind

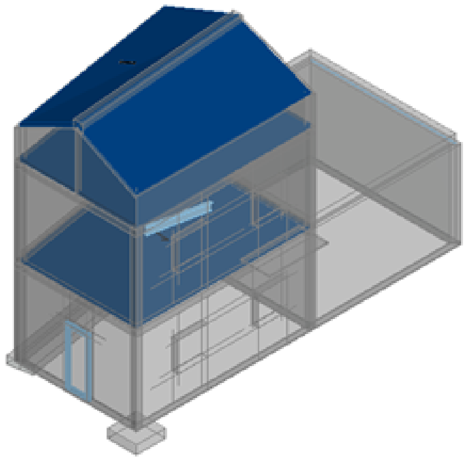
It allows to color the walls exposed to the wind.
The color can be customized by double clicking on the colored rectangle; the default color can be restored by clicking on the icon .

Load variation on slabs

It allows to color the horizontal elements (floors, roof slopes, vaults) with a color scale according to the load acting upon them, according to the combination chosen in the drop-down menu on the right.
The user can then consult the legend by clicking on the icon .

All objects that do not satisfy the active display options are colored in gray and made partially transparent.





OPENGL view: Button that activates / deactivates the OPENGL view within the three-dimensional view of the model.

Cancel selection: Cancels any active selection on 3D elements.

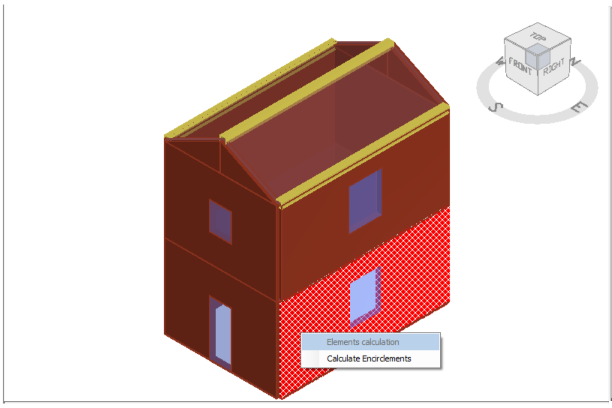
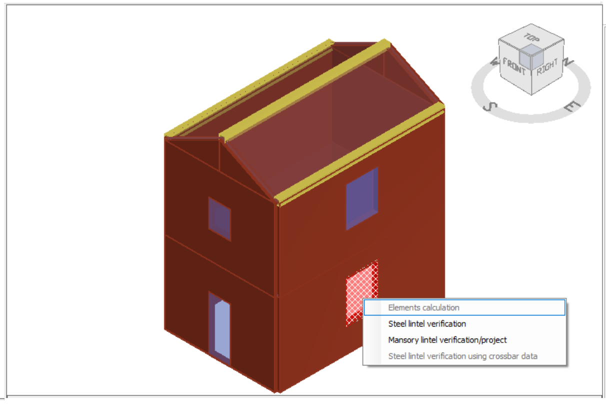
Rotate model: An "Orbit" type CAD command that allows to rotate the model with the left mouse button. Press [Esc] to exit the command.

Element visualization: activates / deactivates the display of the elements: panels, walls, beams, columns, tie rods, floors, roof slopes and foundations.

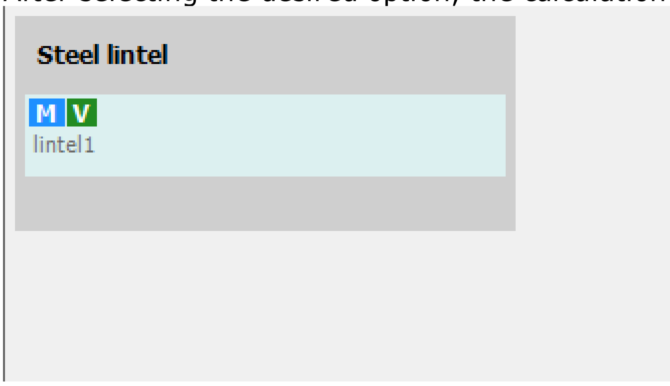
3D Views: Allows you to customize the 3D view.

11.1.2 Model tabs

Each three-dimensional object present within the 3D model is an interactive object. By pressing the right mouse button on each element, it is possible to access the menu that contains all the functions present.

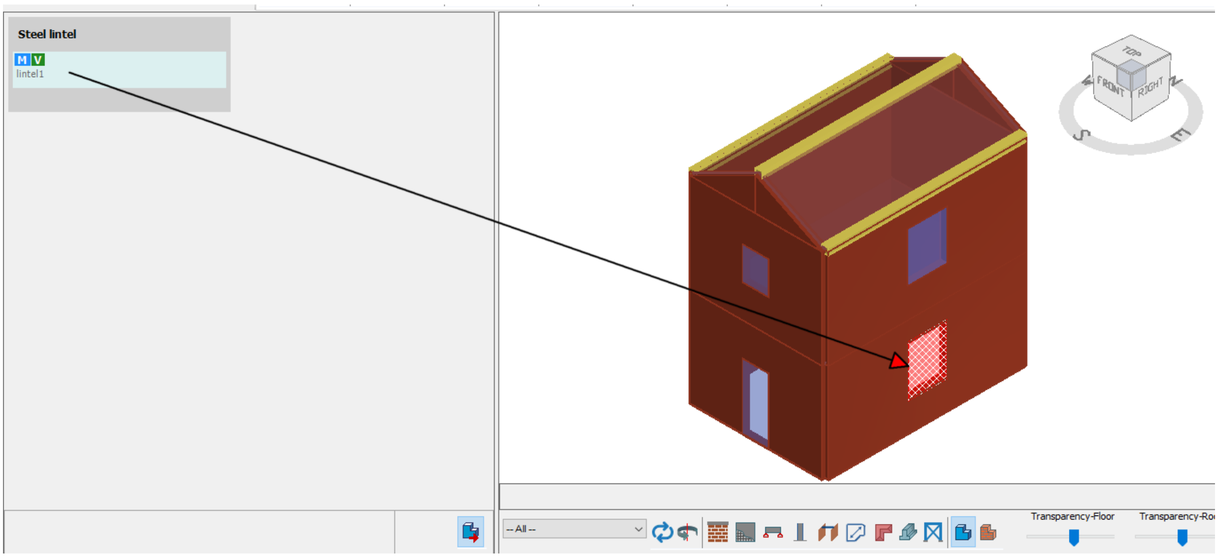
	
Selection of the panel object with its contextual menu	*Selection of the opening object with its contextual menu*

After selecting the desired option, the calculation tab is created on the side:



All the tabs created in this way are distinguished by the information label , which indicates that the tab was created "from the model".

It is always possible to locate the object associated with the calculation sheet by clicking on the relevant sheet. After selection, the object is highlighted and identified within the structure model.



By double clicking on the tab it is possible to view / modify the calculation at any time.

Data input

Using this mode, many data necessary for the calculation are read from the model and automatically inserted as input in the
calculation cards.
Among these are:
- Geometry;
- Materials;
- Rebars;
- Loads / Stresses.

Changing the input data

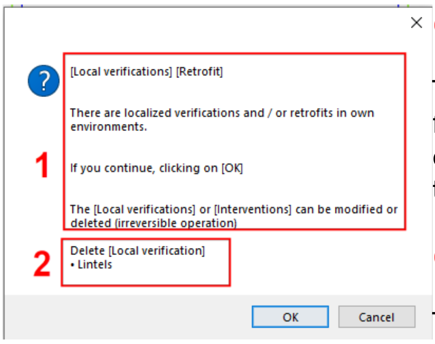
To keep the information consistent between the structure model and the related calculation cards, the modification of the input data
must always be done within the structure environment.
Following the modification, the information contained in the model are updated and consequently also the calculation sheets
associated with the elements.

The operational flow is as follows:

1. Input of the model in the structure environment
2. Once the model is finished, I move on to the "Local verification" environment
3. I create the calculation sheet associated with the element

In the event that it is necessary to modify parameters related to the element to which I have associated a calculation card, I must return to point (1).

The modification of elements to which a calculation card is associated is notified by means of a warning window shown in the following image:



(1)

The first part of the message warns the user that, following changes to the input data in the model, the calculation cards involved and / or the interventions need to be updated.

(2)

The update can be of two types:
- Modify: in this case the new data entered are automatically reported in the calculation cards of the elements affected by the modification. After changing the input data, it will be necessary to recalculate each card.
- Delete: this operation will irreversibly delete the calculation cards involved in the modification.

Under each of the two options are shown the categories of the elements whose calculation cards will be modified /
deleted.

In the case of the image on the side, the message communicates that the calculation card associated with the "lintel" element will be updated following the changes made.

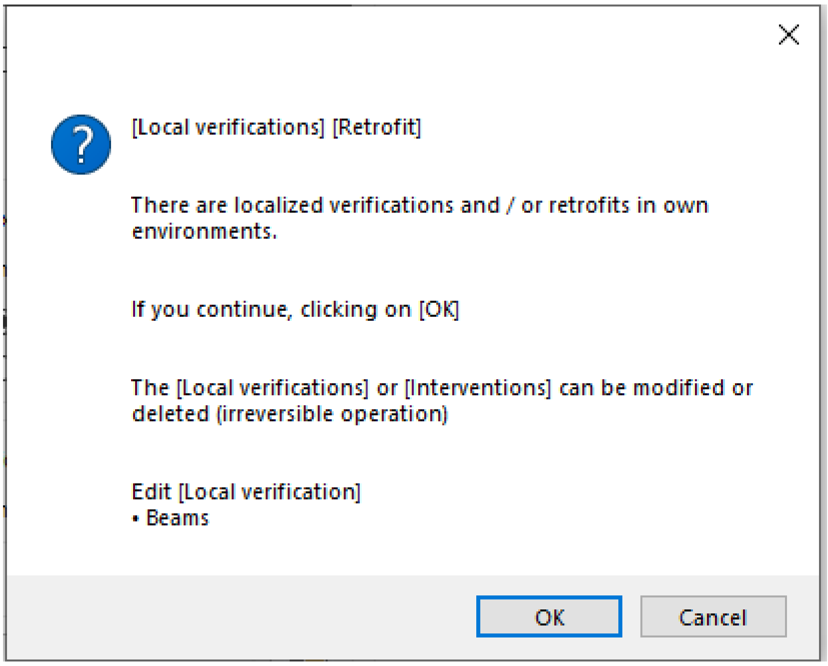
Example:
Suppose we want to calculate a beam inside the model.

After inserting the beam into the model:

- I go into the "Local verification" environment and, by clicking with the right mouse button on the element, the contextual menu associated with the element appears;
- I select the "Calculate beam" option and the program creates the associated calculation card for me;
- I move the calculation sheet from the first to the second column and the program performs the calculation.

Suppose at this point we want to change the section of the beam.

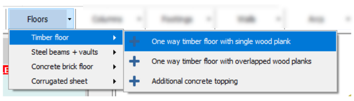
- I go back inside the structure environment and insert the new dimensions of the beam section;
- the following warning window appears, indicating that the previously created calculation card will be updated with the data just entered:



- After confirming the change, I return to the "Local verification" environment and recalculate the element (by moving the tab to the third column).

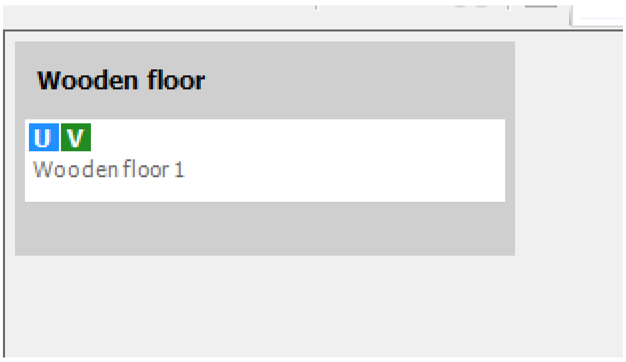
11.1.3 User tabs

To create a "User" tab, simply:



- Choose the category of the element you want to calculate
- Click on the small arrow on the side to view the options available
- Select the desired calculation module

After selecting the desired option, the calculation tab is created on the side:



All the tabs created in this way are distinguished by the information label , which indicates that the tab created is a "user" tab.

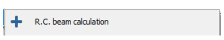
By double clicking on the tab it is possible to view / modify the calculation at any time.

11.2 Beams

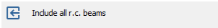
The Beams module allows the verification and / or design of new or existing reinforced concrete beams and joists, with the possibility of customizing loads, reinforcements and exporting the drawings of the latter and FRP and FRCM reinforcements.

11.2.1 Creation of a calculation tab

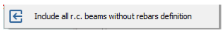
The beam menu has several options:



It allows you to create a new calculation card. By selecting this option, the card will be created as a user card.

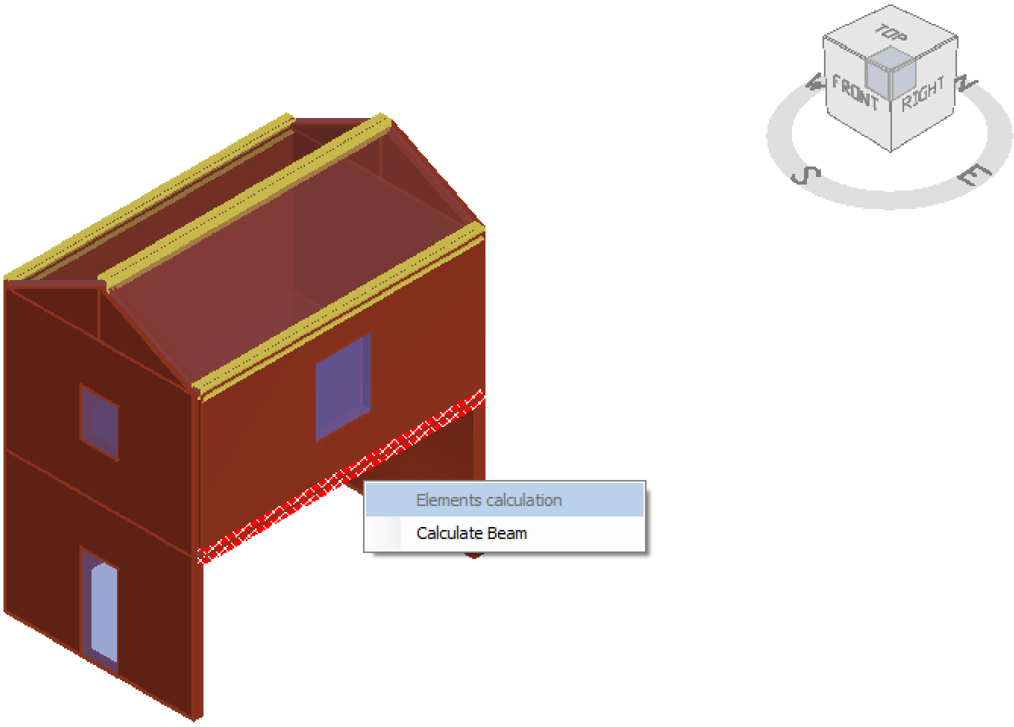


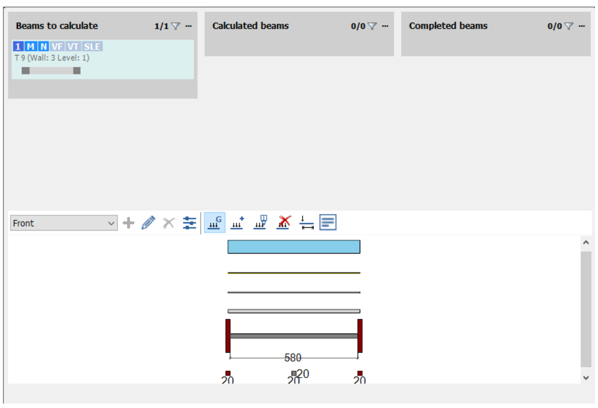
By selecting this option, a calculation card will be created for each beam present in the structure model. The created cards will be model cards.



By selecting this option, a calculation card will be created for each beam without rebars within the structure model. The created cards will be model cards.

It is also possible to create a calculation sheet starting from a beam inside the model. To do this, simply right-click on the "Beam" element present in the model and choose the "Calculate Beam" option.

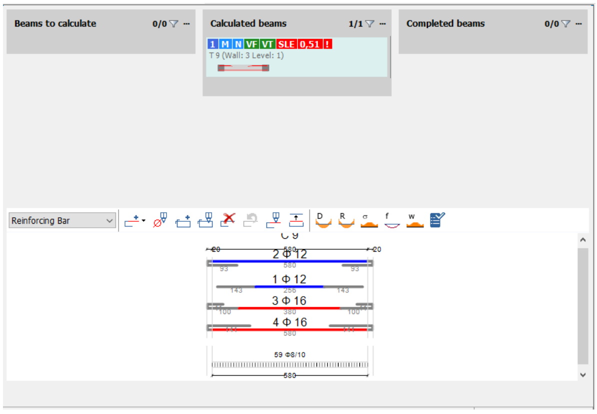




After selecting this option, the calculation window opens and the card is created and placed in the "Beams to be calculated" column.

To calculate the beam, simply drag the card into the "Calculated beams" column.

The information labels on each sheet visually summarize the results of the checks carried out.



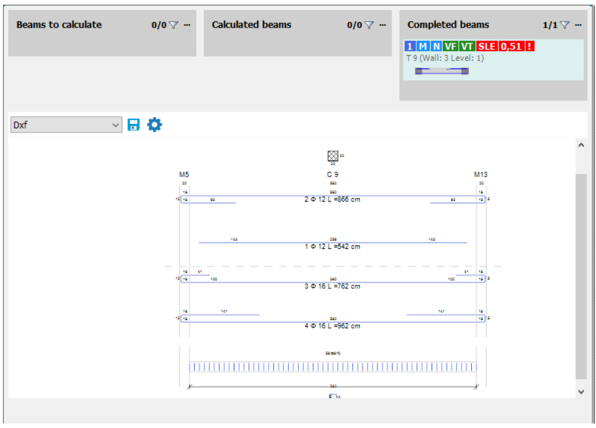
VF Green if the bending check passed. Red otherwise.

VT Green if the shear check is passed. Red otherwise.

SLE Green if the SLE check passed. Red otherwise.

0,51 Safety factor value

! Indicates that there are warnings regarding the calculation made. To view them, simply place the pointer over them.



Once the calculation is finished, by dragging the tab in the "Completed beams" column it is possible to produce the output files of the calculation.

11.2.2 Codes

This section lists the main regulatory references of interest.

11.2.2.1 Normative requirements

This section contains the reference regulations for Italy and Europe.

Italy

The reference legislation is:

- Technical Standards for Construction - D.M. 17 January 2018
- Circular 7 21 January 2019
- Technical standards for constructions in seismic areas - D.M. January 16, 1996

Europe

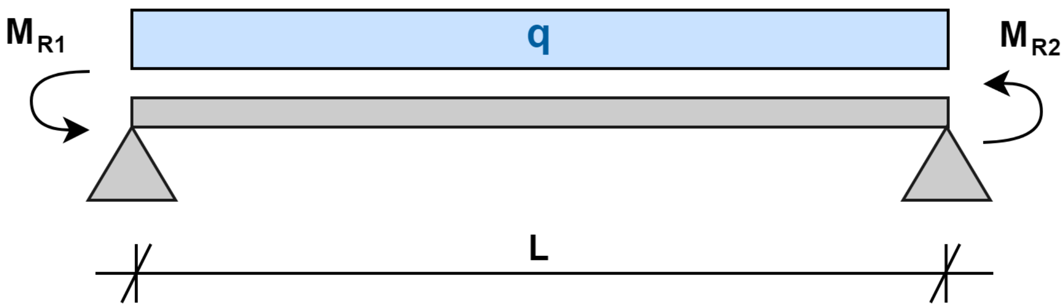
The reference legislation is:

- Eurocode 2 - Design of concrete structures
- Eurocode 8 - Design of structures for seismic resistance

11.2.2.2 Structural calculation type

In the case of non-seismic and non-dissipative designed buildings, it is possible to apply the limit state method according to Chapter 4 of the NTC18.
In the case of designing dissipative buildings, reference must also be made to Chapter 7 and the calculation stresses are evaluated through the resistance hierarchy. The bending moments of calculation, to be used for the dimensioning and verification of beams, are those obtained from the global analysis of the structure for the seismic load combinations, while, in order to exclude the formation of inelastic mechanisms due to shear, the stresses VEd shear to be used for the checks and sizing of the reinforcements are obtained from the equilibrium condition of the beam subject to the action of gravitational loads acting on the beam, hinged at the ends, and of the resisting moments Mb, Rd, 1,2 of the two sections plasticized amplified by the overstrength factor γRd, assumed to be equal, respectively, to 1.20 for structures in CD "A", to 1.10 for structures in CD "B":

$$V_{Ed} = \frac{(M_{R1} + M_{R2})}{l} \cdot \gamma_{Rd} + \frac{ql}{2}$$



In the non-seismic and non-dissipative case, both the bending moments and the calculation cuts, used in the dimensioning and verification of beams, are those obtained from the global analysis of the structure, without applying the resistance hierarchy.

11.2.2.3 Translation of moments

For elements with shear reinforcement the additional tensile force, EC2 recommends that ΔFtd be calculated according to 6.2.3.
For elements without shear reinforcement ΔFtd can be evaluated by translating the bending moment diagram by a distance a l = d according to point 6.2.2 (5). This translation rule can be adopted, alternatively, also for elements with reinforcement for cutting, with: a l = z (cot θ - cot α) / 2, as prescribed also in 4.1.30. of the NTC18.

11.2.2.4 Anchorage calculation

The calculation of the anchorage lengths and the evaluation of their resistance contributions to moment stresses vary according to
the type of work, new project or existing verification, according to the type of calculation, seismic or non-seismic, and possibly the
ductility class of the structure.

- New project;
- Existing verification;
- Resistant moment with null areas;
- Resistant with equivalent areas.

11.2.2.4.1 New project

Dissipative structural behavior

✚ Anchorage of Internal nodes (§7.4.6.2.1 NTC 2018, §8.4.3 EC2, §4.1.2.3.10 NTC 2018)

When it is not possible to avoid anchoring by overlapping the longitudinal reinforcement of the beams in the joints, the following requirements must be respected:
- the bars must be anchored beyond the face opposite to that of intersection with the node, or turned vertically in correspondence with this face, to contain the node;
- the anchorage length of the stretched reinforcement must be calculated in such a way as to develop a tension in the bars equal to 1.25 fyk, and measured starting from a distance of 6 diameters from the face of the pillar towards the inside.

$$1{,}25\, f_{yk} \cdot \frac{\pi\, \emptyset^2}{4} = f_{bd} \cdot \pi\, \emptyset \cdot L_{anchorage}$$

The part of the longitudinal reinforcement of the beam that anchors beyond the node cannot end inside a dissipative zone, but must anchor beyond it;
The part of the longitudinal reinforcement of the beam that anchors in the joint must be placed inside the pillar stirrups.
To prevent the slipping of these reinforcements, the diameter of the non-inclined bars must be ≤ αbL times the height of the column section, being:

$$\alpha_{bL} = \frac{7{,}5 \cdot f_{ctm}}{\gamma_{Rd} \cdot f_{yd}} \cdot \frac{1 + 0{,}8\nu_d}{1 + 0{,}75 k_D \cdot \frac{\rho_{comp}}{\rho}}$$

where:
- vd is the normalized design axial force;
- kD is 1 for CD "A", 2/3 for CD "B";
- γRd is 1.2 for CD "A", 1 for CD "B".

The minimum anchorage length is prescribed by EC2 - 8.4.3:

$$L_{anchorage} = 0{,}25\, \frac{f_{yd} \cdot \emptyset}{f_{bd}}$$

The requirements set out in 4.1.2.3.10 NTC 2018 must also be met:

$$L_{anchorage} = \max(20\phi; 150mm)$$

✚ External Nodes Anchors (§7.4.6.2.1 NTC 2018, §8.4.3 EC2, §4.1.2.3.10 NTC 2018)

When it is not possible to avoid anchoring by overlapping the longitudinal reinforcement of the beams in the joints, the following requirements must be respected:
the bars must be anchored beyond the face opposite to that of intersection with the node, or turned vertically in correspondence with this face, to contain the node;
the anchoring length of the stretched reinforcement must be calculated in such a way as to develop a tension in the bars equal to 1.25 fyk, and measured starting from a distance of 6 diameters from the face of the pillar towards the inside.

$$1{,}25\, f_{yk} \cdot \frac{\pi\, Ø^2}{4} = f_{bd} \cdot \pi\, Ø \cdot L_{anchorage}$$

The part of the longitudinal reinforcement of the beam that anchors beyond the node cannot end inside a dissipative zone, but must anchor beyond it;
The part of the longitudinal reinforcement of the beam that anchors in the joint must be placed inside the pillar brackets.
To prevent the slipping of these reinforcements, the diameter of the non-inclined bars must be ≤ αbL times the height of the column section, being:

$$\alpha_{bL} = \frac{7{,}5 \cdot f_{ctm}}{\gamma_{Rd} \cdot f_{yd}} \cdot (1 + 0{,}8\nu_d)$$

where:
- vd is the normalized design axial force;
- kD is 1 for CD "A", 2/3 for CD "B";
- γRd is 1.2 for CD "A", 1 for CD "B".

If it is not possible to satisfy this limitation, the beam can be extended beyond the column, plates welded to the end of the bars can be used, the bars can be bent for a minimum length equal to 10 times their diameter by placing a special reinforcement behind the folding.

The minimum anchorage length is prescribed by EC2 - 8.4.3:

$$L_{anchorage} = 0{,}25\, \frac{f_{yd} \cdot Ø}{f_{bd}}$$

The requirements set out in 4.1.2.3.10 NTC 2018 must also be met:

$$L_{anchorage} = \max(20\phi; 150mm)$$

Non-dissipative structural behavior

Internal and external node anchors (§4.1.2.3.10 NTC 2018, §8.4.3 EC2)

The minimum anchorage length is prescribed by EC2 - 8.4.3:

$$L_{anchorage} = 0{,}25\, \frac{f_{yd} \cdot Ø}{f_{bd}}$$

The requirements set out in 4.1.2.3.10 NTC 2018 must also be met:

$$L_{anchorage} = \max(20\phi; 150mm)$$

- fbd = fbk/γc
- γc = 1,5
- fbk = 2,25*η*fctk
- η = 1 if O <= 32mm
- η = (132-O)/100 se O > 32mm
- fctk= (fctm) 5% = 0,7*fctm
- fctm= 0,30*fck2/3 for fck<= 50 N/mm2
- fctm= 2,12*ln(1+fcm/10) for fck> 50 N/mm2
- fcm= fck + 8 N/mm2

11.2.2.4.2 Existing verification

NTC and non-seismic seismic calculation

Span anchors

The anchoring length of the stretched reinforcement is considered starting from the end of the iron and is calculated in order to develop a tension in the bars equal to: fyd, for Semiautomatic:

$$f_{yd} \cdot \frac{\pi\, \emptyset^2}{4} = f_{bd} \cdot \pi\, \emptyset \cdot L_{anchorage}$$

$$L_{anchorage} = 0{,}25\, \frac{f_{yd} \cdot \emptyset}{f_{bd}}$$

Permissible sigma stress for simulated tension:

$$\sigma \cdot \frac{\pi \emptyset^2}{4} = f_{bd} \cdot \pi \emptyset \cdot L_{anchorage}$$

$$L_{anchorage} = \sigma \cdot \frac{\emptyset}{4\, f_{bd}}$$

In the case of manual arrangement, the inserted iron includes both the resistant section and the anchoring section, so in this case the anchor is not added, but subtracts from the inserted section of iron.

Internal anchors

See anchors in the span, but the anchoring section starts from the edge of the pillar.

External anchors

Starting from the edge of the pillar proceeding in the direction of the anchor

$$\sigma \cdot \frac{\pi \emptyset^2}{4} = f_{bd} \cdot \pi \emptyset \cdot L_{anchorage}$$

$$L_{anchorage} = \sigma \cdot \frac{\emptyset}{4\, f_{bd}}$$

The stress σ of the reinforcement is that determined at the support.

- If the straight iron is not sufficient, for the upper reinforcement the iron is bent downwards (in this case the anchor length is reduced to 0.7 anchor. If necessary, the iron is bent again after a distance equal to H -2 * concrete cover).
- For the lower reinforcement, on the other hand, the pillar is also raised in addition to H-2 * concrete cover (also in this case with bent iron the total anchor = 0.7 anchor).

- fbd=fbk/γc
- γc=1,5
- fbk=2,25*η*fctk
- η =1 if O <=32mm
- η =(132-O)/100 if O>32mm
- fctk=(fctm)5%=0,7*fctm
- fctm=0,30*fck^(2/3) for fck<=50 N/mm2
- fctm=2,12*ln(1+fcm/10) for fck>50 N/mm2
- fcm=fck+8 N/mm2

11.2.2.4.3 Resistant moment with null areas

The section of anchor iron has a zero contribution in the calculation of the resisting moment and subsequently in the calculation of the stresses in the reinforcement.

11.2.2.4.4 Resistant with equivalent areas

The section of anchor iron contributes to the calculation of the resisting moment, increasingly moving away from the end of the steel.
The Wastlund hypothesis predicts a triangular trend of the adhesion stresses and therefore also of the stresses in the reinforcement.
In fact, following adhesion, the tangential stresses that pass to the concrete generate main tensile and compressive stresses in the conglomerate.

It is not possible to decrease the length of the anchor section. It is possible to increase it, so part of the anchorage is taken into account in the calculation of the resisting moment.

11.2.2.5 Reinforcement calculation

The calculation of the reinforcements using fiber-reinforced materials is carried out in accordance with Circular no. 7 of 21 January 2019, CNR-DT R1 / 2013 and CNR-DT 215/2018.

11.2.2.5.1 Longitudinal reinforcements

ULS Analysis

The calculation procedure is based on the comparison between the design stressing moment Msd and the design resistant moment of the reinforced section Mrd.
Msd ≤ Mrd
The fundamental assumptions on which the ULS analysis of the sections of reinforced concrete is based reinforced with
FRP are as follows:

- preservation of the planarity of the straight sections until failure, so that the diagram of the normal deformations is linear;
- perfect adhesion between the component materials (steel-concrete, FRP-concrete);
- zero tensile strength of concrete;
- constitutive bonds of concrete and steel in compliance with current legislation;
- constitutive bond of the linear elastic fiber-reinforced composite until failure.

Flexural failure is hypothesized to occur when one of the following conditions occurs:

- achievement of maximum plastic deformation in compressed concrete, εcu, as defined by current legislation;
- achievement of a maximum deformation in the FRP reinforcement, εfd.

Given the geometry and materials of the section of the element to be reinforced, the moment of first cracking Mcr is evaluated,
evaluating the average simple tensile strength fctm given by eqs. 11.2.3.a, 11.2.3.b and 11.2.4 - NTC 2018.
The initial deformation of the concrete at the stretched edge is evaluated as:

$$\varepsilon_0 = \frac{M_0 \cdot (h - x_0)}{I_0 \cdot E_c}$$

in which:
-M0 is the initial moment acting in section before the application of the fibers produced by the permanent loads at the SLS;
-x0 is the distance of the neutral axis from the compressed limb;
-I0 is the moment of inertia of the section taking into account any cracking;
-Ec is the elastic modulus of concrete evaluated in accordance with Eq. 11.2.5, NTC 2018.

Given the characteristics of the composite and its methods of application, the maximum design deformation is evaluated: (eq. 4.14 CNR-DT 200 R1 / 2013)

$$\varepsilon_{fd} = \min\{\eta_a\, \varepsilon_{fk} / \gamma_f;\ \varepsilon_{fdd}\}$$

where:

$\varepsilon_{fk} = f_{fk} / E_f$ represents the characteristic deformation at break of the reinforcement;

$\varepsilon_{fdd} = \frac{f_{f\,dd,2}}{E_f} \geq \varepsilon_{sy} - \varepsilon_0$ represents the maximum value attributable to the deformation of the composite during the design so that the intermediate detachment does not occur (eq. 4.7 - CNR-DT 200 R1 / 2013);
in which:

$f_{f\,dd,2} = \frac{k_q}{\gamma_{f,d}} \cdot \sqrt{\frac{E_f}{t_f} \cdot \frac{2 \cdot k_b \cdot k_G}{FC} \sqrt{f_{cm} \cdot f_{ctm}}}$ represents the maximum design stress of the composite (eq. 4.6 - CNR-DT 200 R1 / 2013)

The ultimate limit state resistance for end detachment is evaluated by verifying that the tension of the composite does not exceed the design tension of the reinforcement system for end detachment (eq. 4.4 - CNR-DT 200 R1 / 2013), calculated as:

$$f_{f\,dd} = \frac{1}{\gamma_{f,d}} \cdot \sqrt{\frac{2 \cdot E_f \cdot \Gamma_{Fd}}{t_f}}$$

with $\gamma_{f,d}$ partial coefficient for FRP materials (3.4.1 CNR - DT 200 R1 / 2013), Γ_{Fd} represents the specific fracture energy calculated as (eq. 4.2 - CNR-DT 200 R1 / 2013):

$$\Gamma_{Fd} = \frac{k_b \cdot k_G}{FC} \sqrt{f_{cm} \cdot f_{ctm}}$$

The calculation of the optimal anchoring length is also performed (eq. 4.1 - CNR-DT 200 R1 / 2013):

$$l_{ed} = \max\left\{\frac{1}{\gamma_{Rd} \cdot f_{bd}} \sqrt{\frac{\pi^2 \cdot E_f \cdot t_f \cdot \Gamma_{Fd}}{t_f}}, 200\ mm\right\}$$

In the case of anchorage lengths, lb, less than the optimal one, led, the design voltage must be suitably reduced in accordance with the relationship:

$$f_{f\,dd,rid} = f_{dd} \cdot \frac{l_b}{l_{ed}} \cdot \left(2 - \frac{l_b}{l_{ed}}\right)$$

Once the position of the neutral axis has been determined, the value of the resisting moment MRd can be determined starting from the equation of equilibrium for rotation around the axis passing through the center of gravity of the stretched reinforcement and parallel to the neutral axis (eq. 4.16 - CNR -DT 200 R1 / 2013):

$$M_{Rd} = \frac{1}{\gamma_{Rd}} \cdot \left[\psi \cdot b \cdot x \cdot f_{cd} \cdot (d - \lambda \cdot x) + A_{s2} \cdot \sigma_{s2} \cdot (d - d_2) + A_f \cdot \sigma_f \cdot d_1\right]$$

where the three components represent respectively:
- the contribution of compressed concrete
- the contribution of steel, stretched and compressed
- the contribution of FRP reinforcement

State of the structure at reinforcement

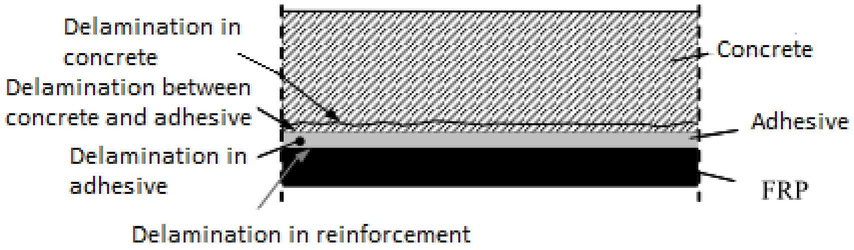
In the hypothesis that the FRP reinforcement is applied on an element subject to a pre-existing stress, which corresponds to an applied moment M_0, the program proceeds to evaluate the initial deformation state considering a different moment of inertia depending on whether M_0 is greater or less than the moment of first crack M_{cr}.

The calculation is performed in the hypothesis of linear elastic behavior of the two materials making up the beam and, in particular, of the inability of the concrete to withstand tensile stresses.

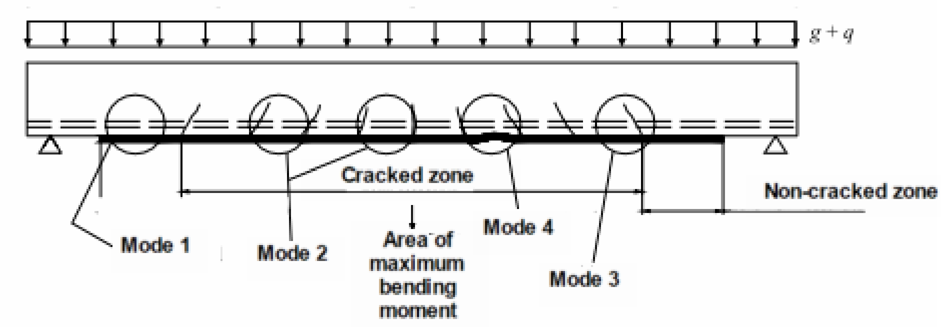
Security checks against delamination

In the reinforcement of reinforced concrete elements using sheets or fabrics of composite material, the role of adhesion between concrete and composite assumes great importance as the mechanism of rupture by delamination (loss of adhesion) is of a fragile type. In the spirit of the criterion of hierarchy of resistances, this crisis mechanism must not precede the collapse by bending or cutting of the reinforced element.

Delamination can occur inside the adhesive, between concrete and adhesive, in the concrete or inside the reinforcement (for example between layers of fabric warped with different inclination angles of the fibers). The program considers that the reinforcement is placed correctly, therefore, since the adhesive resistance is generally much higher than the tensile strength of the concrete, that delamination always takes place inside the latter with the removal of one layer of material.



The modes of collapse by delamination of foils or fabrics used for bending reinforcement can be classified into the following four categories:



- Mode 1 (Delamination of ends);
- Mode 2 (Intermediate delamination, caused by bending cracks in the beam);
- Mode 3 (Delamination caused by diagonal cutting cracks);
- Mode 4 (Delamination caused by irregularities and roughness of the concrete surface).

The program allows the verification of modalities 1 and 2 only, being the ones that occur most frequently in ordinary situations.

✚ ELS Analysis

In a beam reinforced with FRP, stress concentrations (tangential and normal) occur at the interface between concrete and reinforcement, located in correspondence with transverse cracks in the concrete, especially at the ends of the reinforcement. These concentrations can cause cracking of the interface triggering the detachment between the two materials.

It is advisable that, under operating conditions, the opening of the aforementioned cracks should not occur, especially in the presence of loading cycles and freeze / thaw cycles. The competent verification can be performed by a calculation of the interface stresses using linear elastic models.

It must be checked that, at the adhesive-concrete interface, both for the characteristic (or rare) and for the frequent load combination, the "equivalent" shear stress, defined below, is lower than the adhesion resistance between the reinforcement and the substrate of concrete, f bd:

$$\tau_{b,e} \leq f_{bd}$$

The "equivalent" shear stress tb, e can be defined starting from the mean shear stress tm, evaluated at the chord on which the adhesive and concrete interface:

$$\tau_{b,e} = k_{id} \cdot \tau_m$$

where:

-k_{id} is a coefficient (≥ 1) which takes into account the concentration of shear and normal stresses in the terminal zones:

$$k_{id} = \left(k_\sigma^{1.5} + 1.15 \cdot k_\tau^{1.5}\right)^{2/3}$$

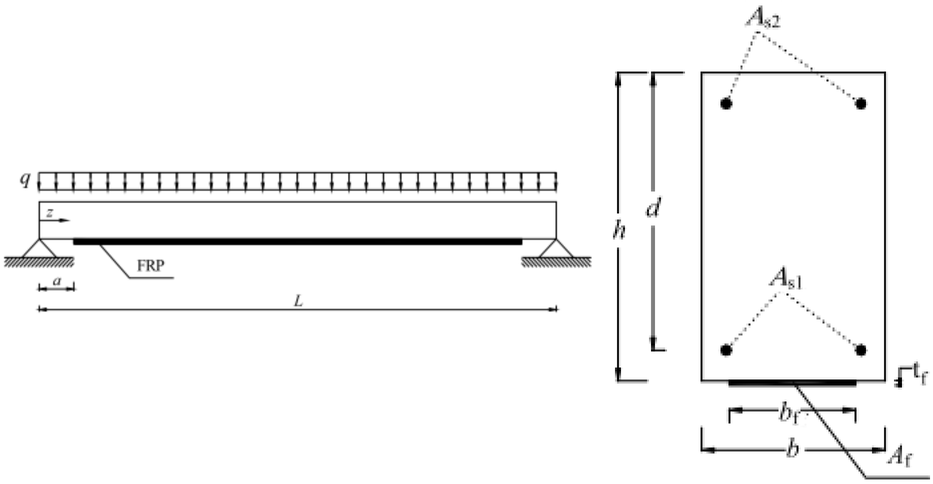
- the coefficients kσ and kτ are respectively valid:

$$k_\sigma = k_\tau \cdot \beta \cdot t_f$$

$$k_\tau = 1 + \alpha \cdot a \cdot \frac{M_{(z=a)}}{V_{(z=a)} \cdot a}$$

- M (z = a) is the moment acting in the interruption section of the reinforcement;

- V (z = a) is the shear acting in the interruption section of the reinforcement, located at a distance z = a from the end of the beam;



- α and β are two elastic constants dependent on the characteristics of the interface and of the FRP reinforcement:

$$\alpha = \sqrt{\frac{K_1}{E_f \cdot t_f}}$$

$$\beta = \left(\frac{b_f \cdot 2.30 \cdot K_1}{4 \cdot E_f \cdot I_f} \right)^{1/4}$$

being E_f , t_f , b_f , I_f and K_1 respectively, the modulus of normal elasticity, the thickness of the FRP reinforcement, its length, the competent moment of inertia (with respect to its own barycentric axis parallel to the length dimension bf) and the angular coefficient of the increasing linear branch of the adhesion bond, assumed to be equal to:

$$K_1 = \frac{1}{t_a/G_a + t_c/G_c}$$

where moreover, respectively, Ga and Gc are the modulus of tangential elasticity of the adhesive and of the concrete, ta is the nominal thickness of the adhesive and tc the effective thickness of the concrete participating in the deformability of the interface (in general it can be assumed tc = 20 ÷ 30 mm);

- tm is the average Jourawski shear stress:

$$\tau_m = \frac{V_{(z=a)} \cdot t_f \cdot (h - x_e)}{I_c/n_f}$$

- x_e and I_c are, respectively, the distance of the neutral axis from the extreme compressed edge and the moment of inertia of the homogenized section, possibly partialized if in the presence of cracking (NOTE: the program considers the cracked section when the moment acting for the operating state considered M (z = a) is higher than the moment of first cracking Mcr);

- nf = Ef / Ec is the homogenization coefficient (with Ec the normal modulus of elasticity of the concrete corresponding to the load combination considered, rare or frequent).

In the presence of a terminal anchorage, made by "U" bandage, the effect of normal tensions for the purposes of verifying the interface can be neglected and, therefore, the coefficient k_σ can be assumed to be equal to zero.

The design strength of the adhesion between reinforcement and concrete, f_{bd}, is a function of the characteristic tensile strength of the concrete, fctm, and is provided by the relationship:

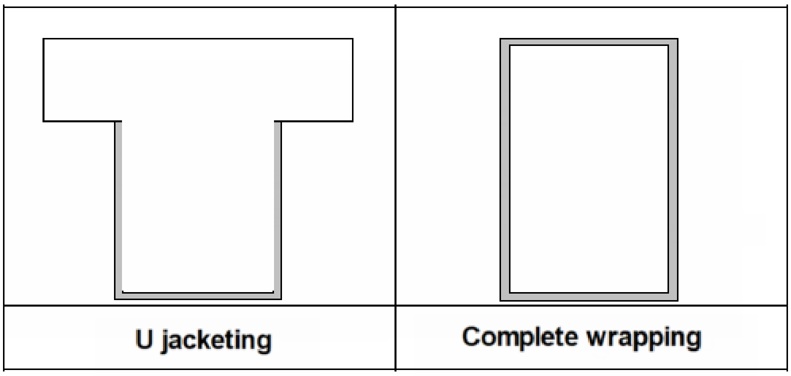
$$f_{bd} = 0.21 \cdot \frac{k_b}{\gamma_b} \cdot \frac{f_{ctm}}{FC}$$

where the partial factor γ_b is 1.0 for the rare load combination, 1.2 for the frequent load combination.

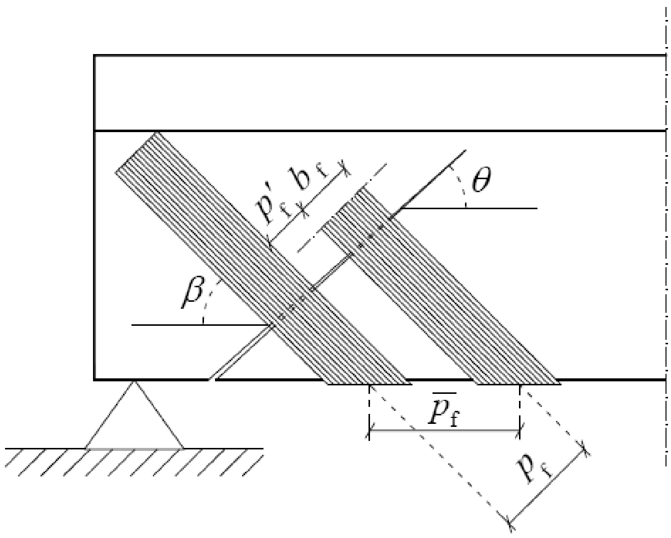
In the calculation of the anchoring stresses, in operating conditions (SLS), it is possible to refer to the state of stress corresponding to the load increase that occurs after the application of the reinforcement.

11.2.2.5.2 Transverse reinforcements

The resistant shear of the reinforced element is given by the expression (CNR DT200 R1 2013 - (4.18)):
Vrd = min {VRd, s + VRd, f, VRd, c}
where VRd, s and VRd, c are, respectively, the contributions of the transverse reinforcement and of the compressed concrete connecting rod, calculated in accordance with current legislation (NTC 18). The contribution VRd, f, on the other hand, is the increase in resistance brought about by the reinforcement. In the event that VRd, c <VRd, s, therefore, the application of the reinforcement cannot produce increases in resistance.
The contribution VRd, f depends, in addition to the characteristics of the materials, on the type of arrangement of the reinforcement which may be of the type (in order of efficiency):
-"U jacketing";
-"complete wrapping".



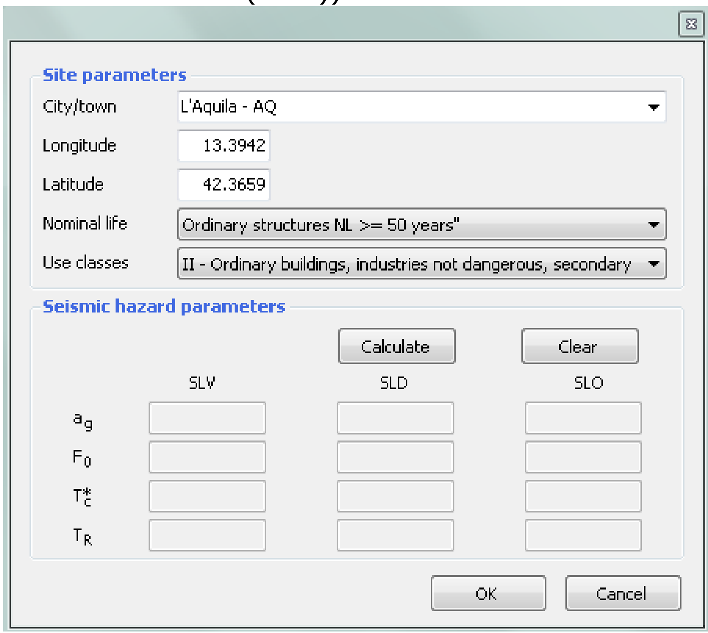
In the U-jacketing reinforcement it is possible to improve the constraint conditions of the free ends of the fabrics, by applying bars, sheets or strips of composite material in these areas. In this case, if the effectiveness of the constraint offered by the aforementioned devices is demonstrated, the behavior of the U-jacketing reinforcement can be considered equivalent to that of the winding reinforcement.
Other factors influencing the contribution VRd, f are the arrangement along the element (continuous or in bands) and the angle of inclination of the fibers with respect to the horizontal axis.



The angle of inclination of the cutting slots θ is to be set equal to 45 °. However, this is in contrast to what is indicated in the NTC 18 where, for the calculation of the shear

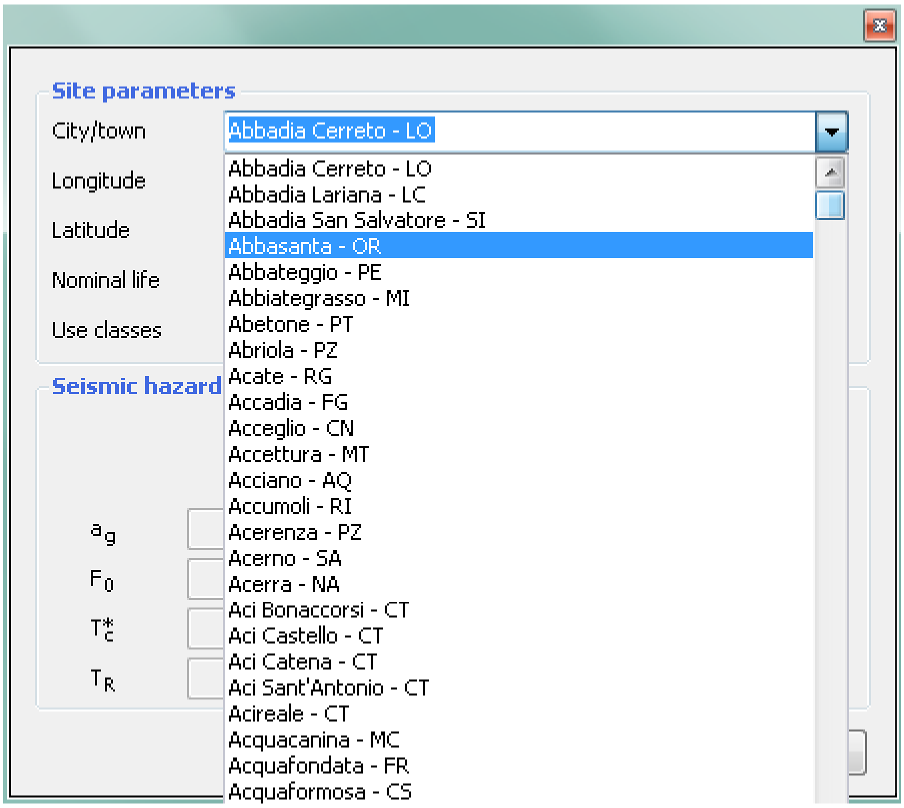
strength, the use of the "variable θ" method is required provided that 1 <= cotg θ <= 2.5. The program therefore leaves the possibility for the user to use the calculation method he deems appropriate.
The contribution of the FRP reinforcement is calculated according to the formula 4.19 of CNR DT200 R1 2013

where:
- d is the useful height of the section;
- tf, bf, pf are the geometric characteristics of the strips;
- ffed is the effective resistance of the reinforcement system, calculated as follows (CNR DT200 R1 2013 - (4.21))



In the case of discontinuous reinforcement systems, the strips of composite material must respect the following limitations:
50 mm ≤ bf ≤ 250 mm
bf ≤ pf ≤ min {0,5 d; 3 bf; bf + 200 mm}

In the case of seismic calculation, the calculation of the shear resistance of the section is carried out according to the indications of paragraph C8.7.2.3.5 of the Circ. 21 January 2019 n. 7 relating to NTC 2018.
The contribution of the FRP reinforcement must be added to the contribution of the stirrups Vw in the equation C8.7.2.8:



11.2.2.6 ζv calculation

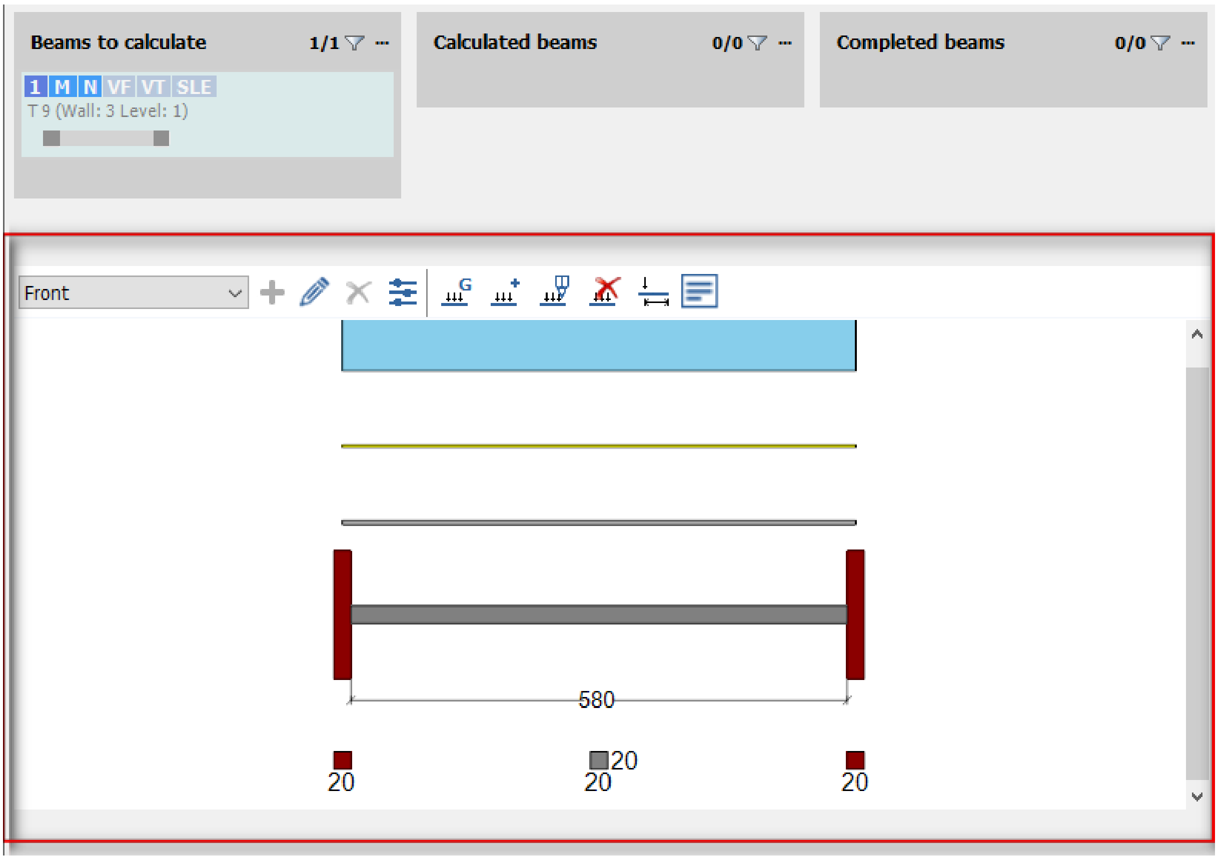
Chapter 8.3 of the NTC 2018 prescribes the methodologies for assessing the safety of existing structures and the introduction of any
restrictions on use in relation to the expected loads.
In particular, for vertical loads: *"The restriction of use can change from portion to portion of the building and, for the i-th portion, it i*
quantified through the ratio ζv, i between the maximum value of the vertical variable overload that can be tolerated by that part of
the construction and the value of the variable vertical overload that would be used in the design of a new construction ".

In the case of beams and floors, the factor ζv is calculated as follows:
1. For each span of the Travata in question, all the design provisions factored to ULS are calculated for variable overloads and permanent loads plus any other variables.
2. For each calculation section of the span, the values of each arrangement are compared with the shear and the resisting moment, purified from the portion of stress carried by the effect of the permanent loads plus any other variables.
3. The minimum ratio obtained corresponds to the design ζv of the single span.
4. The minimum value of all the spans belonging to the girder corresponds to the design ζv value of the girder.

11.2.3 Work environment

11.2.3.1 Front interaction

The "Front" view allows you to interact graphically with the elements using the commands in the top bar.



The command bar has several possible configurations depending on the position of the calculation card within the columns.

11.2.3.1.1 Front configuration

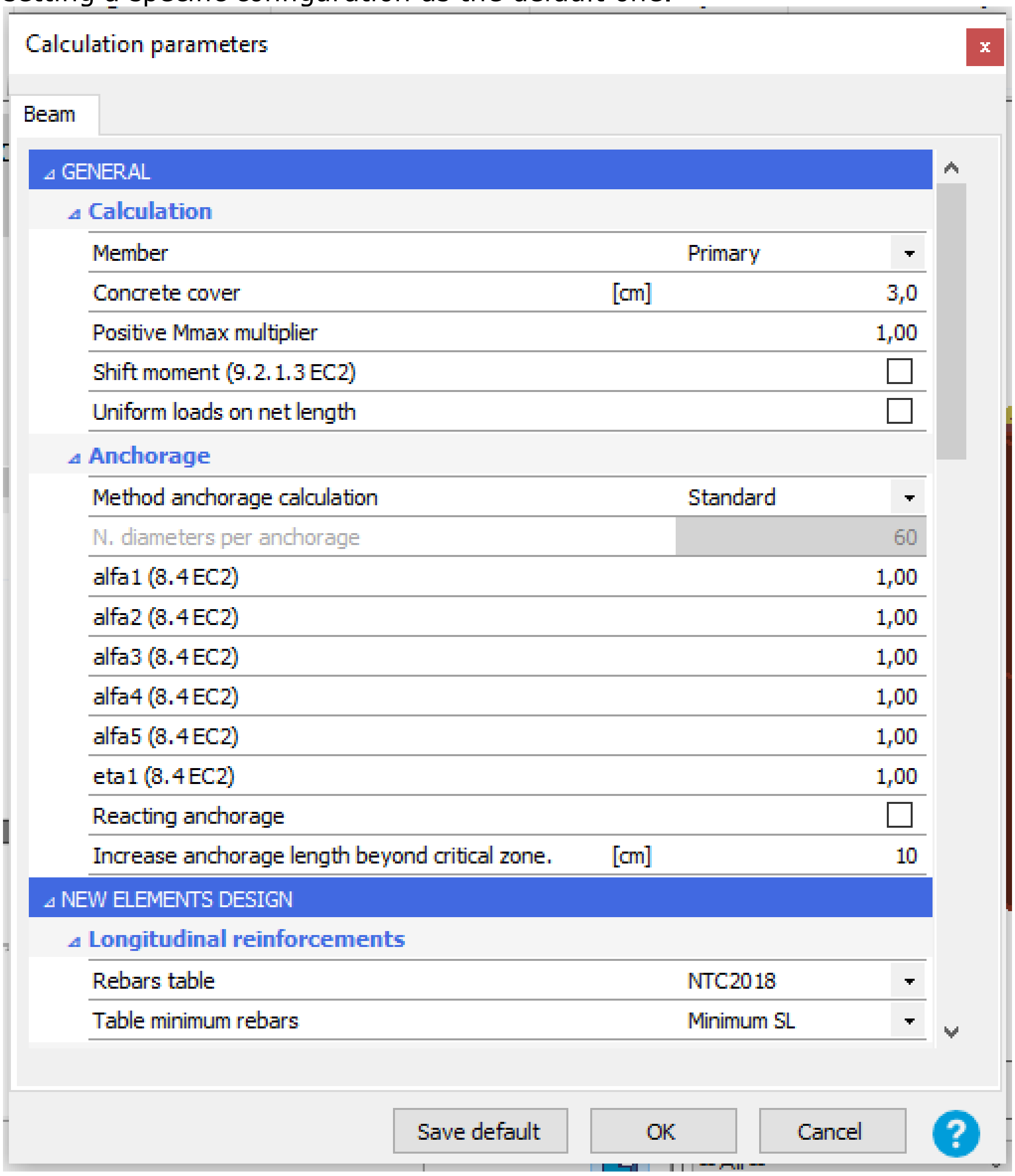
"Front" configuration

It allows to modify geometries, loads and calculation parameters.

- "Add": allows you to add new spans;
- "Modify": allows you to modify the selected span;
- "Delete": allows you to delete the selected spans;
- "Calculation parameters": allows you to set / modify the calculation parameters;
- "Results": allows you to view the results of the calculation.
- "Dead weight": allows you to add the load corresponding to the own weight on all spans;
- "Add load": allows you to add new loads to the selected spans;
- "Edit load": allows you to modify the selected loads;
- "Delete load": allows you to delete the selected loads;
- "Load dimensions": allows you to view / not display the dimensions relating to the loads;
- "Legend of loads": activates the display of the load legend.

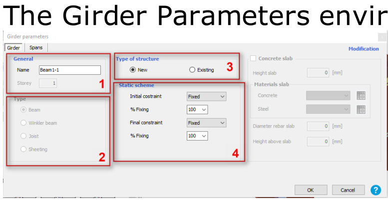
11.2.3.1.1.1 Calculation parameters

By pressing the "Calculation parameters" button it is possible to customize the calculation settings regarding the beams and reinforcements, with the possibility of setting a specific configuration as the default one.



11.2.3.1.1.2 Girder parameters

The Girder Parameters environment provides:

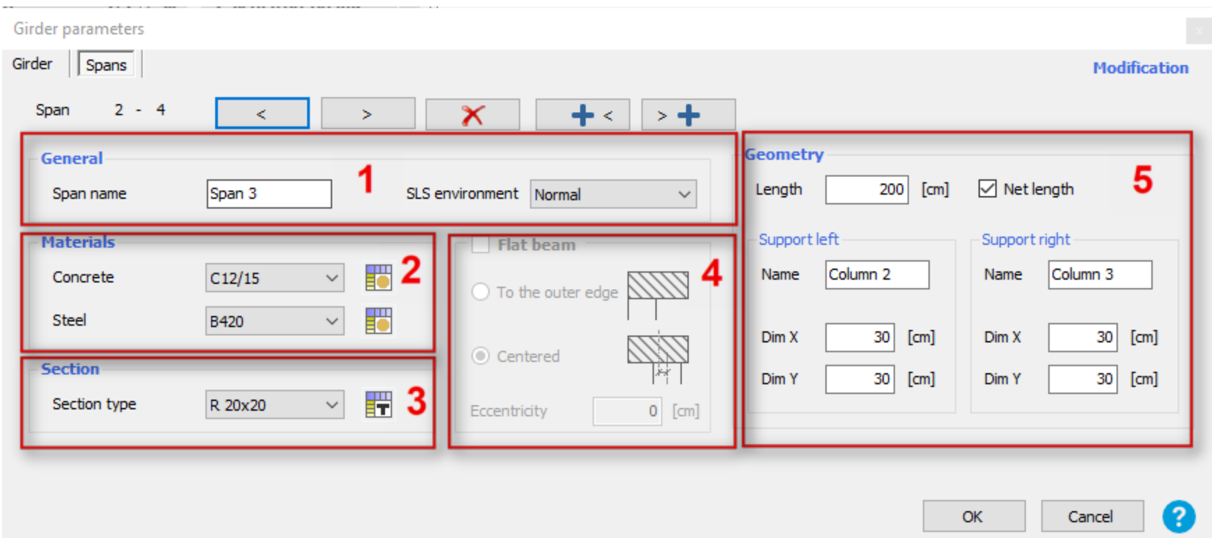


[1] insertion of general data;
[2] type selection (set on beam);
[3] setting the type of structure, with the possibility of choosing between new and existing;
[4] definition of the static scheme, with the possibility of setting the interlocking percentage if the latter is set as a constraint on at least one of the two extremes;

11.2.3.1.1.3 Span parameters

Add Span

After clicking on the "add" button, the following window is displayed on the screen, in which it is possible to add further spans to the girder created.



[1] General

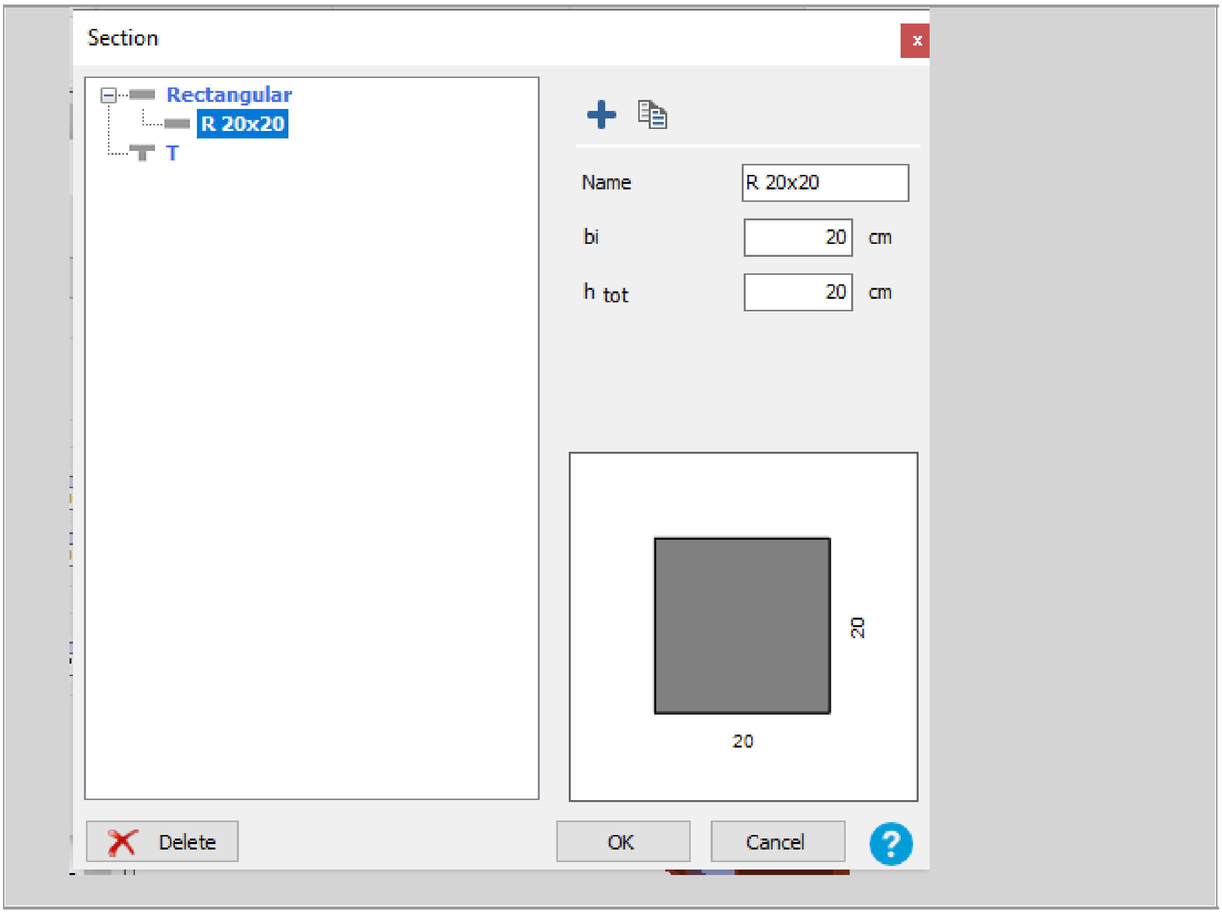
The section provides for the insertion of general data: in particular the name of the span and the aggressiveness of the environment for SLS checks

[2] Materials

The section provides for the definition of materials for concrete and steel, with the possibility of accessing the respective lists.

[3] Section

The menu provides for the definition of the section of the span: it is possible to choose between rectangular and T-shaped sections; the insertion of the geometric data of the section is accessed via .



[4] Flat beam

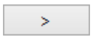
The section provides for the possibility of setting a flat beam, with a choice between reference to the external edge or centered beam; in the latter case the geometric eccentricity data must be entered.

[5] Geometry

The Geometry section provides the insertion of the span length (which will be understood according to the criterion selected in the Girder Parameters environment), the geometric data and the names of the columns at the ends.

Once all the necessary data has been entered, the position of the mouse cursor (without clicking) on one of the two buttons causes the span to be created.
To insert other spans (with the same geometric and mechanical data or different, after modification), simply click on one of the two buttons.

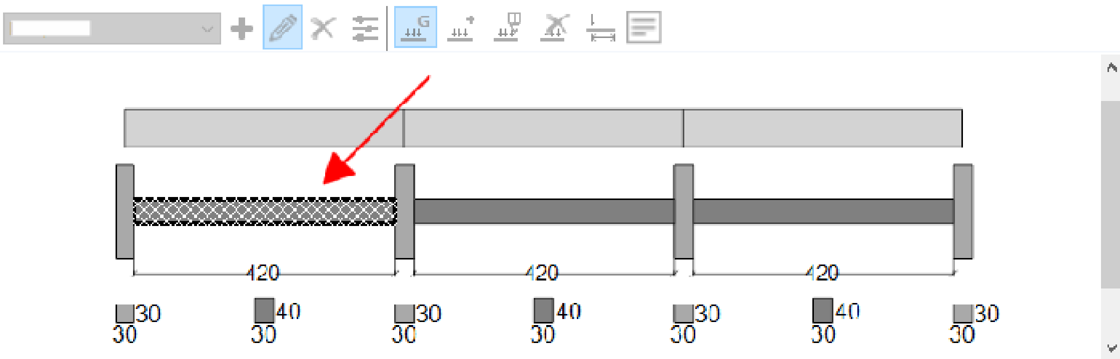
They allow to select the adjacent span (left or right).

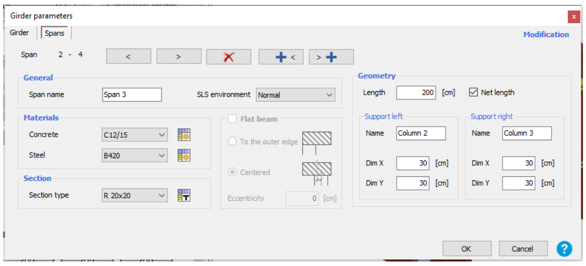



Allows to delete the selected span.

Modify span

It is possible at any time to modify the data entered for the span in question.
To do this, simply click on the "edit" button and select the desired span in the drawing:

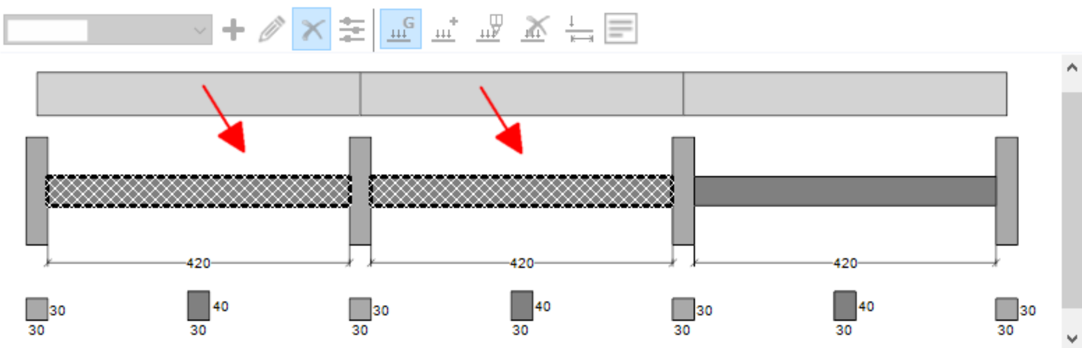




The window that summarizes all the parameters defined for the selected span is displayed.
From here it is possible to edit all the desired parameters and press "OK" to confirm.

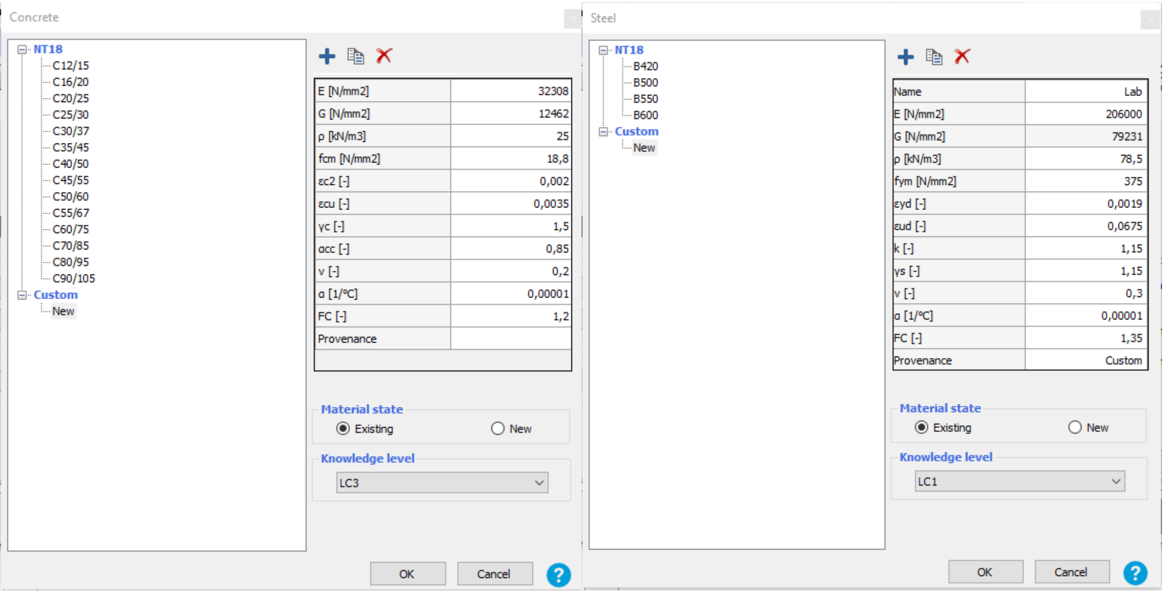
Delete span

It is possible at any time to delete one or more inserted fields.
To do this, simply click on the "delete" button and select the desired spans in the drawing:



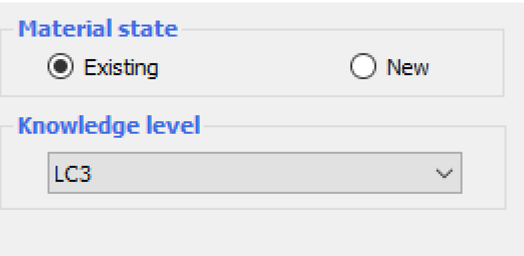
By right-clicking it is possible to confirm the deletion.

The window shows the list of materials for concrete, steel, sheet steel (NT18), reinforcements (NT18) or thermal protection (NT18) present in the work; on the right side of the window, on the other hand, the mechanical data of the selected material are shown.
There is already a list of the most commonly used materials and these cannot be changed.



Each material can be duplicated using the command and the copy will be shown in the custom list, where it can be selected and modified using the data table on the right.

Once the customized list has been selected, it will also be possible to insert a new material (for example if laboratory data on existing materials are available) using the command ; in this case it is also possible to specify that it is existing material and the level of knowledge (LC1, LC2, LC3):



11.2.3.1.1.4 Loads

The Loads environment provides for the definition, modification and insertion of the loads acting on the spans.

Image missing

It is necessary to insert the desired load case through the drop-down menu, with the possibility of quickly entering the self-weight
(G) calculated automatically and accessing the window for defining the load groups.

The type of load includes:

- linear load;
- concentrated load;
- pair on knot;
- surface load.

Once the case and type of load have been set, the button allows you to apply it to the span.
Once applied, the environment for changing the position and value of the load is enabled.

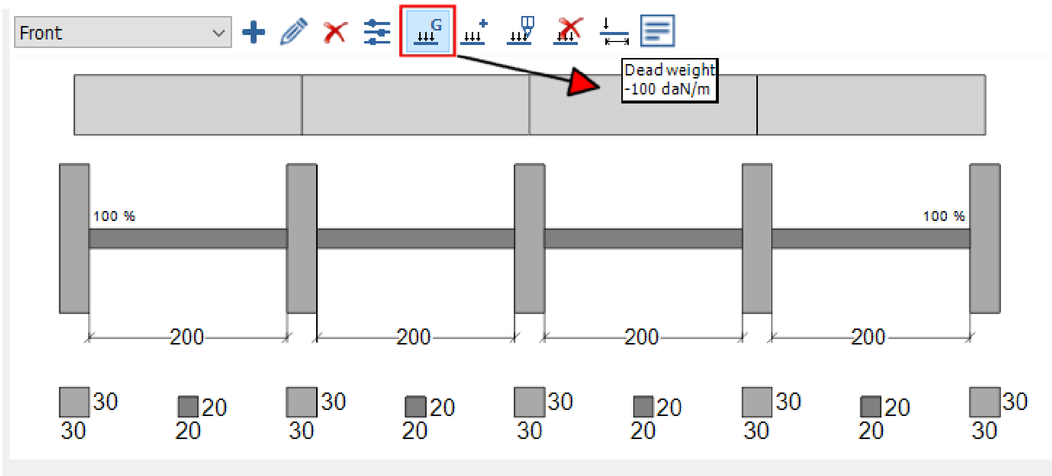
They allow you to apply the selected load to the adjacent span (left or right).

They allow you to select the adjacent span (left or right).

They allow you to select the first or last span.

Allows you to delete the selected span.

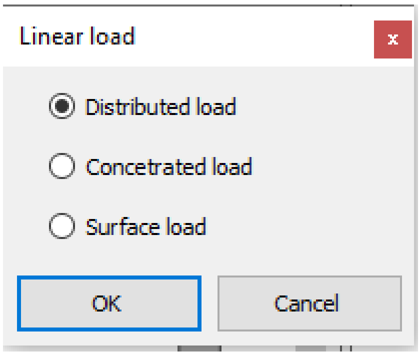
By pressing the "Dead weight" button it is possible to quickly define, on all spans, the self-weight of the element calculated automatically:



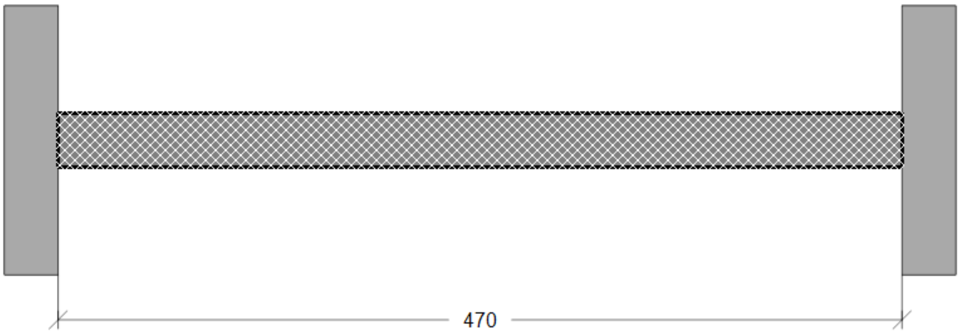
Simply reselect the button to eliminate the load entered due to its own weight.

The addition allows the insertion of a load.

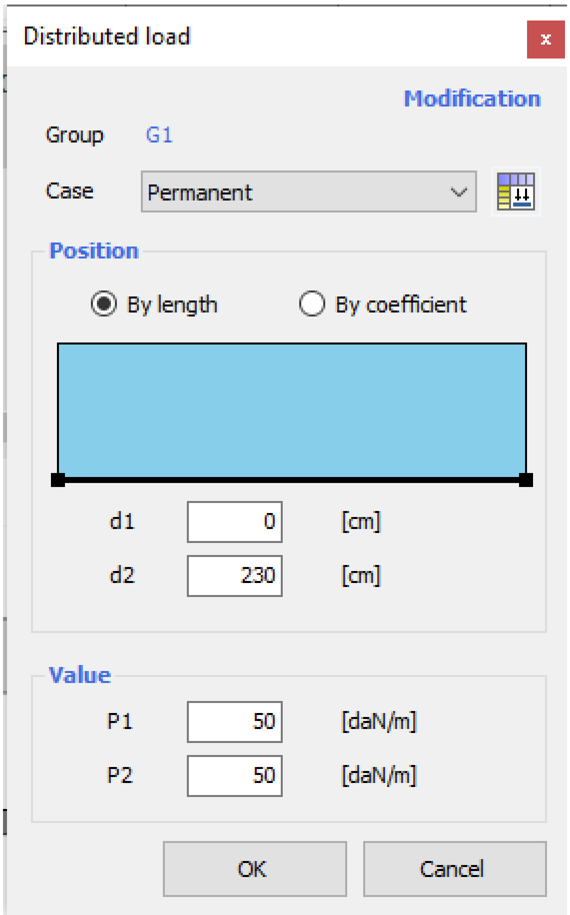
The command first requires the choice of the type of load to be inserted:



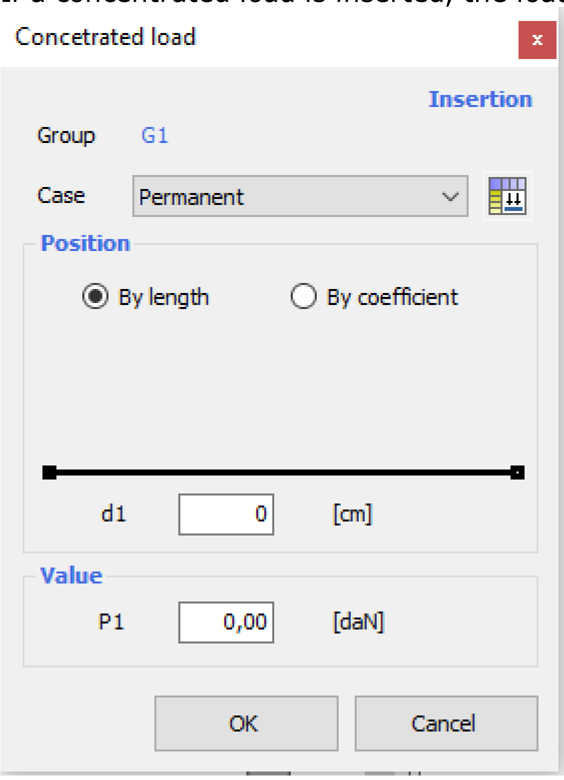
Then it is simply necessary to select the bay to be loaded with a click and confirm the choice by clicking on the right mouse button again:



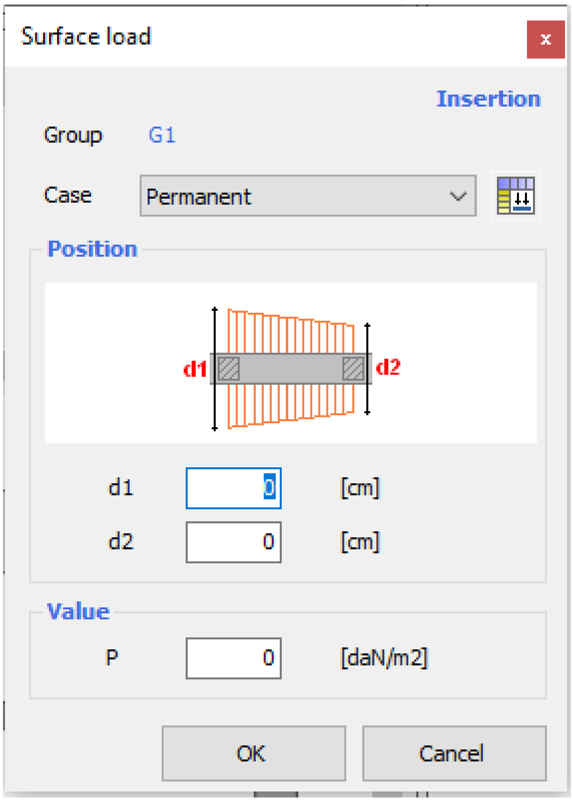
If a distributed load is entered, the load is set through the following menu.



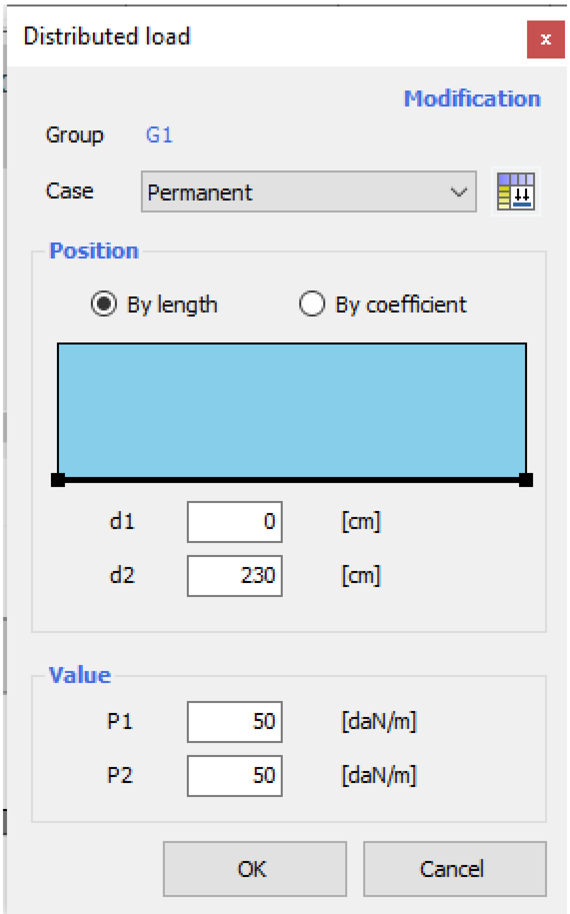
If a concentrated load is inserted, the load is set through the following menu.

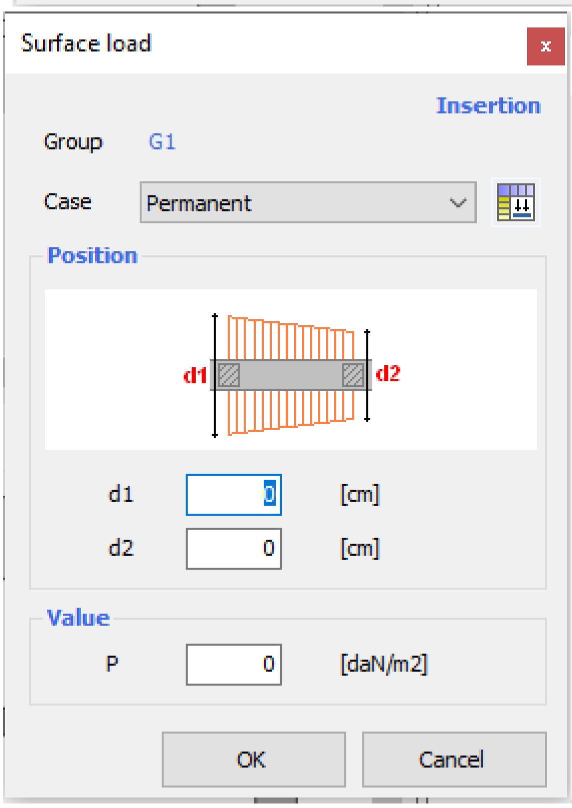


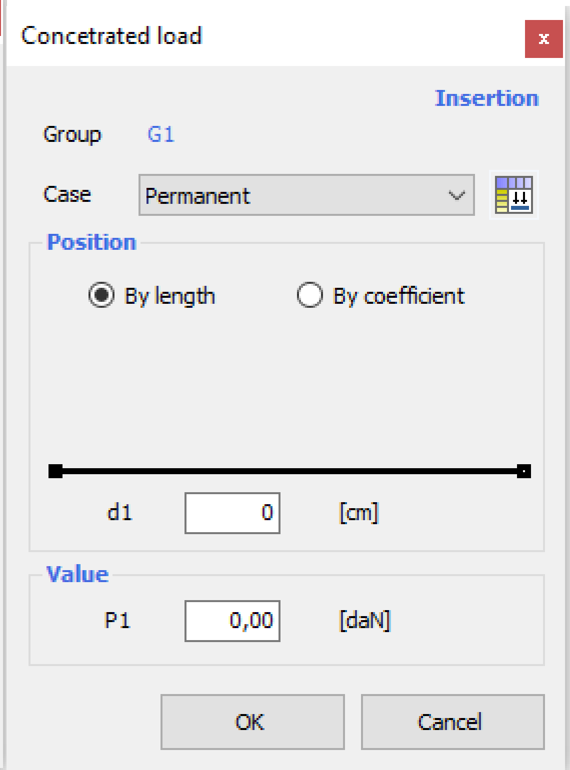
If a surface load is entered, the load is set through the following menu.



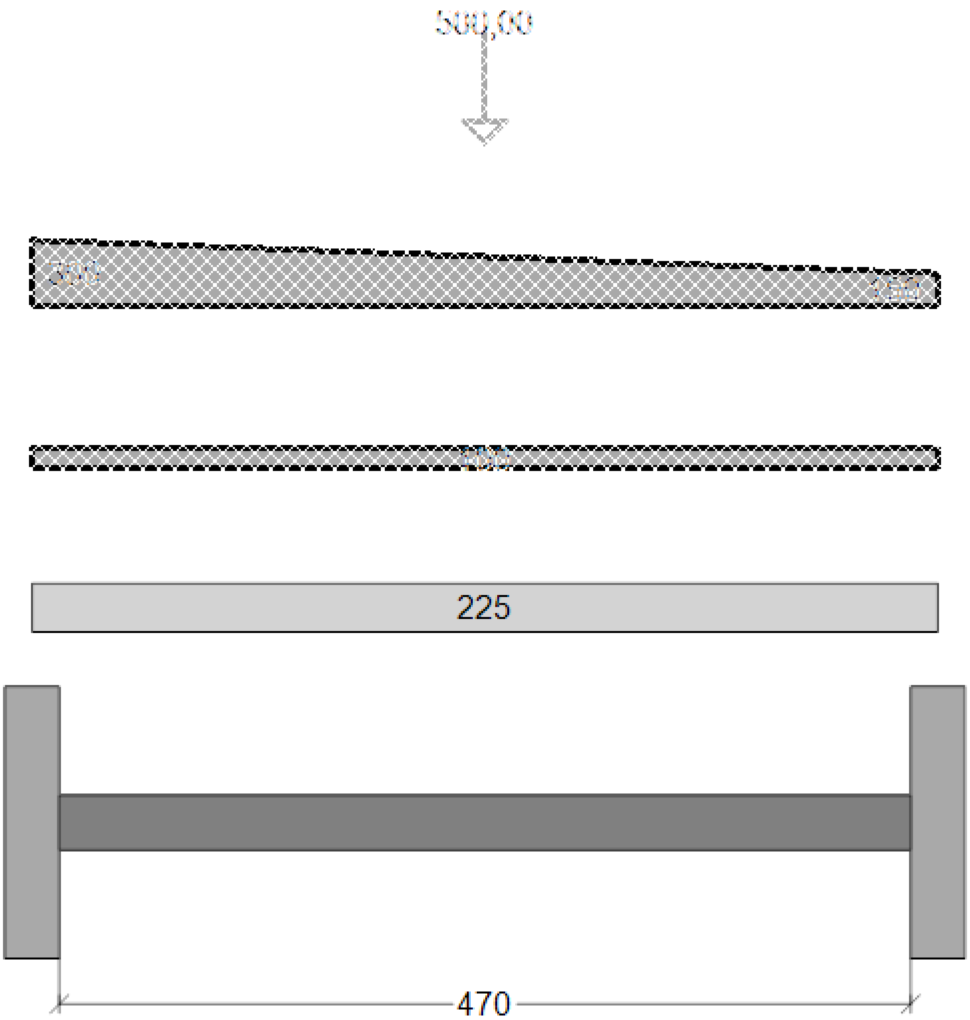
To edit an added load, simply select the load to be modified with a click and set the new parameters using the appropriate menu corresponding to the type of load for which the modification is necessary.







To delete an existing load it is necessary to select the load to be deleted with a click:



And finally confirm the deletion with a right click.

11.2.3.1.2 Reinforcing bar configuration

"Reinforcing bar" configuration

It allows you to insert / modify the reinforcement and graphically display the results of the calculation through diagrams.

 Stresses: Displays the stress diagram.

 Verifications for ULS: Displays the ULS check diagrams.

 Verifications for stresses for SLS: displays the diagrams of the trend of stresses in the case of SLS checks.

 Verifications for strains for SLS: displays the diagrams of the deformation trend in the case of SLS checks.

 Verifications for cracks for SLS: displays the diagrams of the trend of the crack in case of SLS checks.

 Add profile: allows the manual insertion of new longitudinal reinforcing rods or by means of a template; to do this, place the virtual shape on the elevation that is

activated after clicking on the icon; once positioned, a window will open in which the user will have to enter the number and diameter of the reinforcements to be inserted and will have to choose whether to consider the anchoring of the new reinforcement as an increase in the length of the reinforcement or as a reduction of the same; once these parameters have been set, it will be possible to draw the new reinforcements using the appropriate snaps.

- Edit diameters and amount profiles: to modify the diameter or quantity of a profile on the elevation, simply select it and click with the right button; a dialog box will open in which to enter the new parameters.
- Insert stirrups: allows the manual insertion of the stirrups on the beam, with the possibility of customizing the spacing and diameter along the different areas of the beam.
- Edit stirrups: to change the diameter or spacing of the stirrups on the elevation, simply click on them.
- Delete profile: to delete one or more profiles, simply select them and confirm the deletion by clicking with the right button.
- Redo profile: the command is active only if a profile has been previously deleted.
- Stretch reagent length: allows you to stretch the reagent length of a template; it will be sufficient to select it and drag its ends in order to obtain the new desired length.
- Move profile: to move one or more profiles it will be necessary to select them, confirm with the right button and set the movement by clicking on the starting point and the consequent ending point.

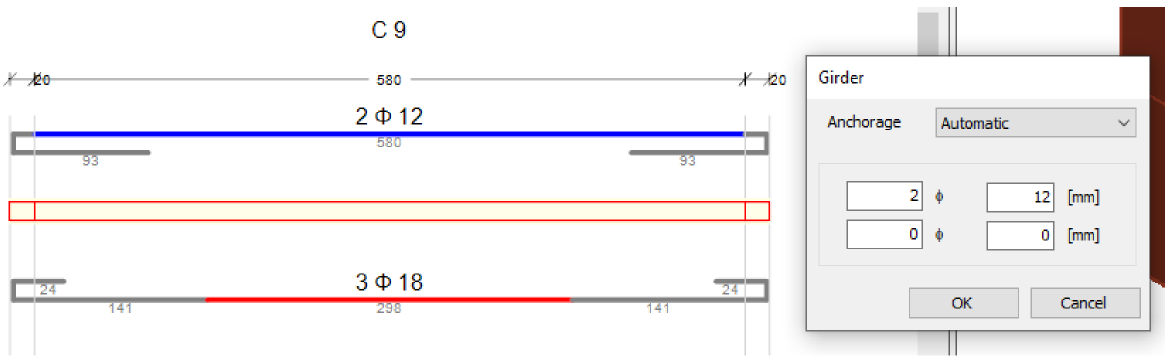
11.2.3.1.2.1 Add profile

Manual addition of profile

Manual addition allows you to insert custom profiles by drawing freehand on the elevation.

The command will require setting the quantity and diameter of the profiles, the type of anchoring (automatic or included) and the positioning of a drawing support profile, whose intersections between the lines can be used as a snap to trace the desired profile .

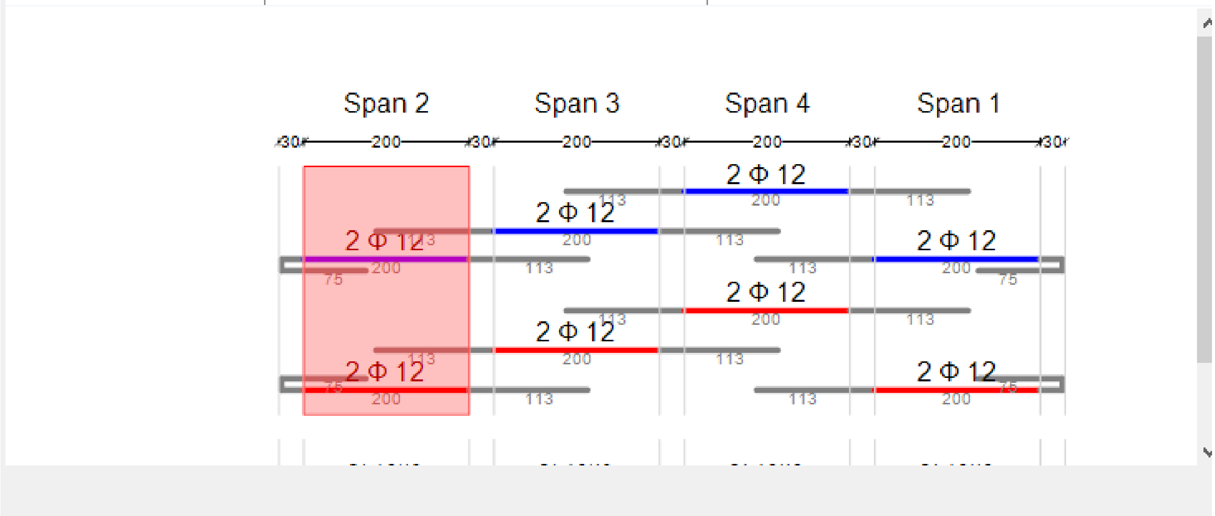
The "automatic" anchor option will add reinforcement sections to the shape drawn manually by the user, treating them as an anchor; the "included" anchor option as a reduction will subtract some sections of reinforcement from the profile drawn manually by the user, treating them as an anchor.

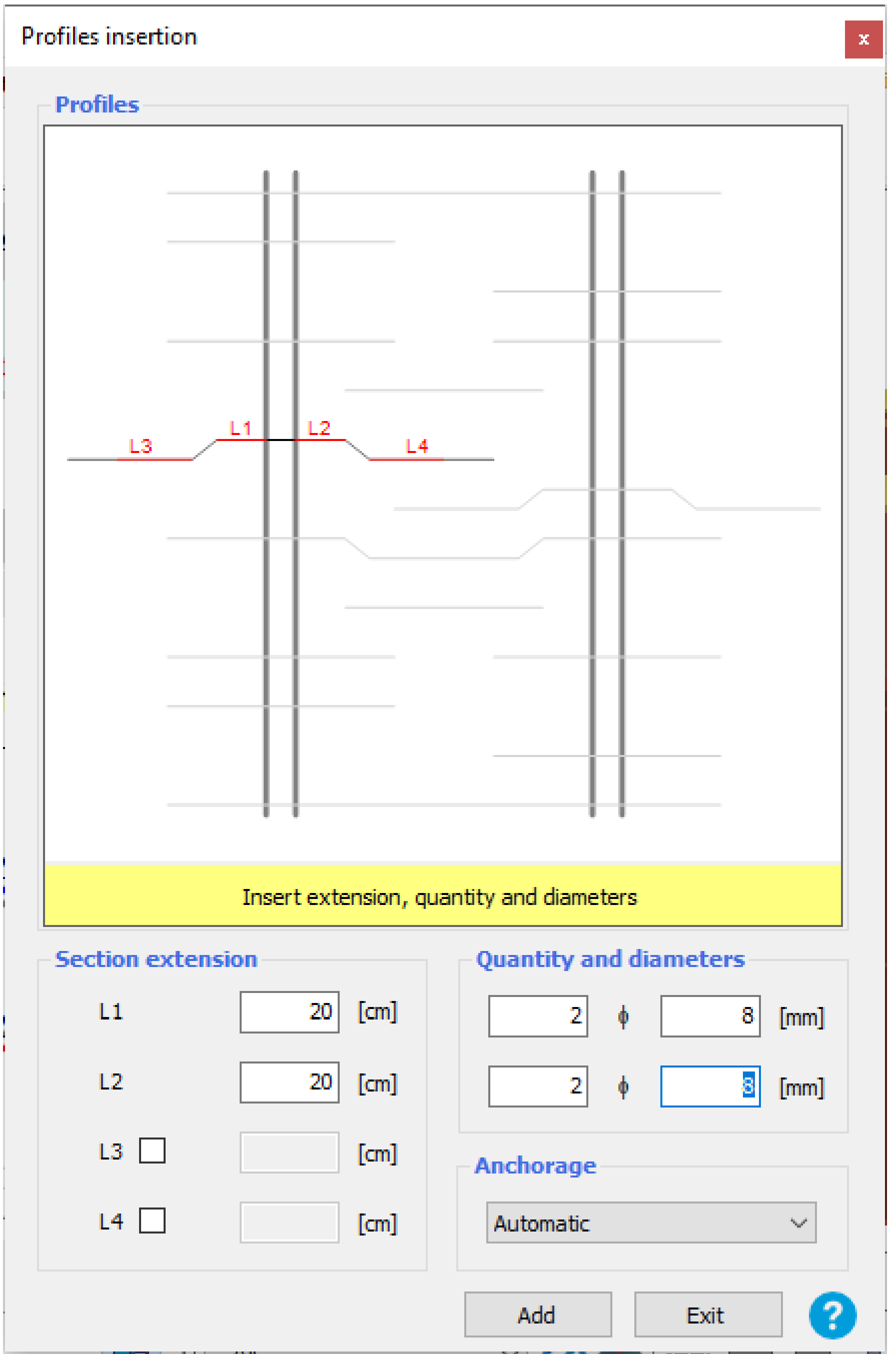


Add profile from template

The addition of a profile from a template allows you to use predefined profile templates, setting their characteristic geometric parameters.

The profiles menu is accessed by selecting the span / s of interest and then clicking with the right button.

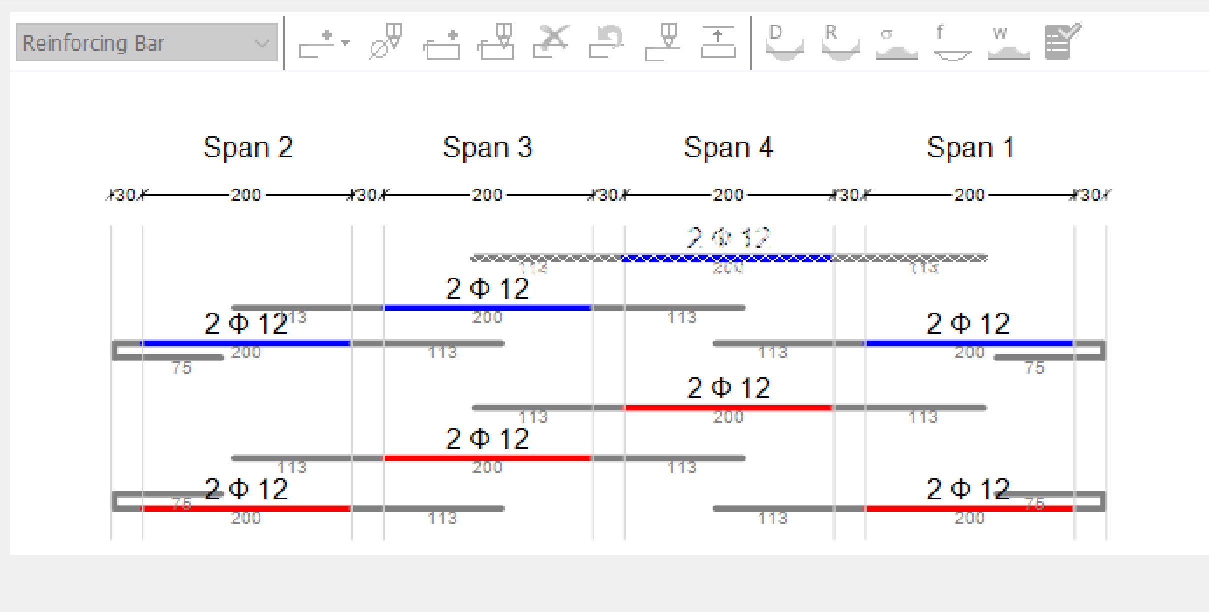




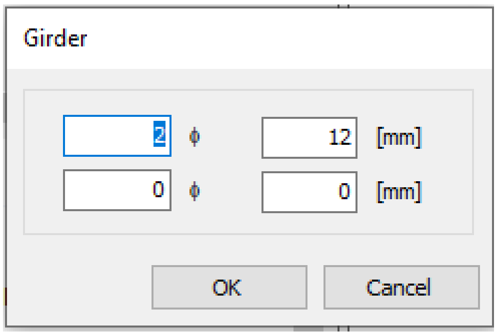
11.2.3.1.2.2 Edit diameters and amount profiles

By clicking on the "Modify diameters / amount profiles" button it is possible to modify the diameter and number of the inserted profiles.

After clicking the button it is sufficient to graphically select the shapes to be modified:

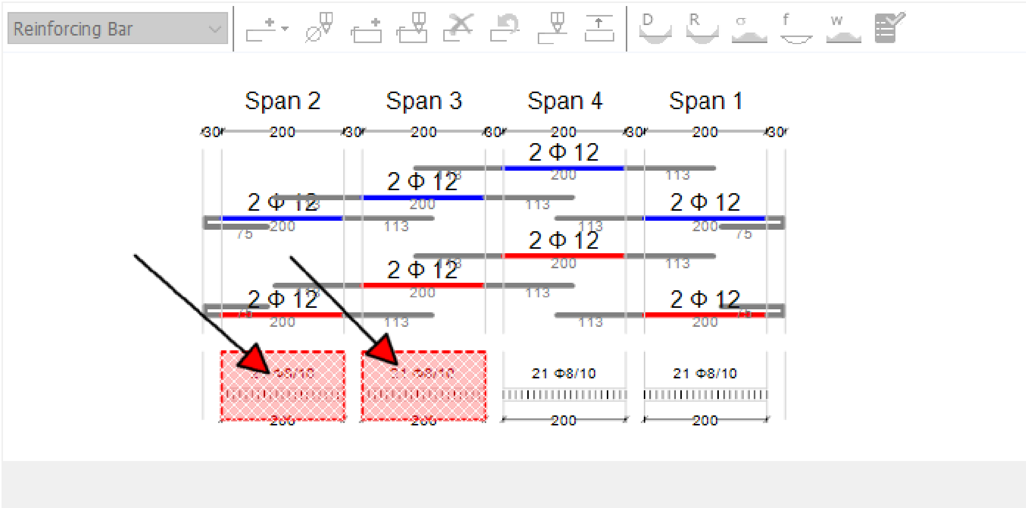


I confirm the selection with the right key.
The window for editing the new parameters is displayed:



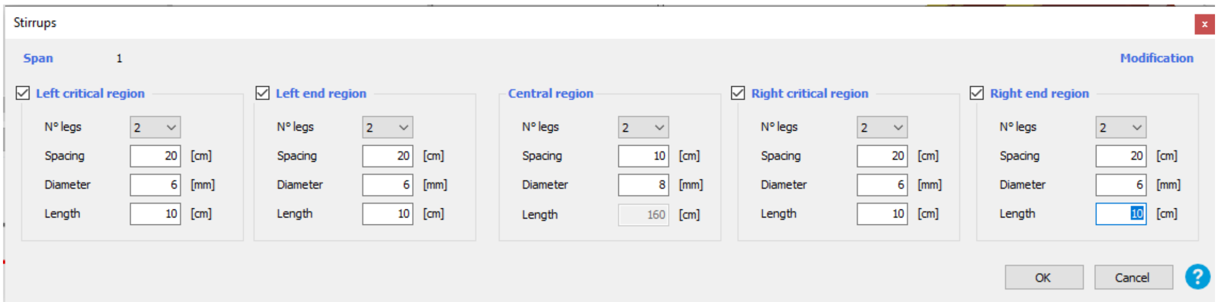
11.2.3.1.2.3 Add/edit strirrups

To insert the stirrups, simply click on the "Insert stirrups" command which, interacting with the drawing, allows you to select the spans on which you want to insert the stirrups.



I finish the selection with the right mouse text.

The parameter entry window appears on the screen:



I enter the desired parameters and confirm with "OK".

By clicking on the "Edit stirrups" button it is possible to modify the inserted stirrups. The operational flow is identical to that of adding.

11.2.3.1.3 Reinforcement configuration

"Reinforcement" configuration

It allows to insert / modify longitudinal and transversal reinforcements and to graphically display the calculation results through diagrams.

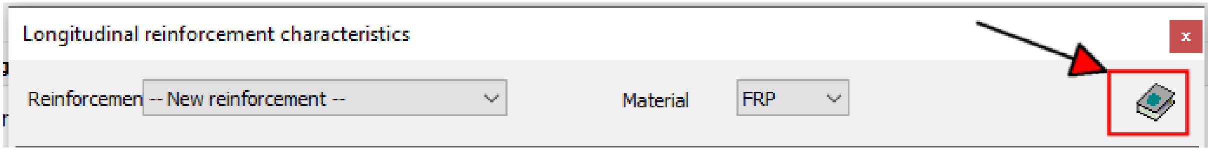
Insert longitudinal reinforcement: allows you to insert longitudinal reinforcements in FRP or FRCM

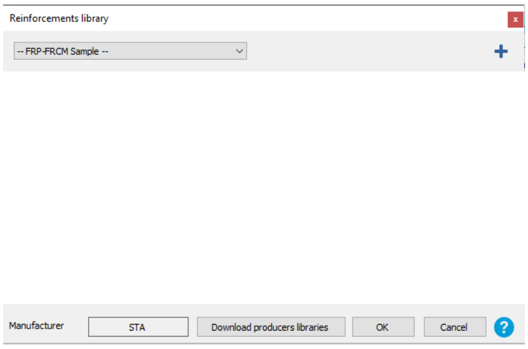
Modify longitudinal reinforcement: allows you to modify a longitudinal reinforcement in FRP or FRCM previously inserted, stretching it or modifying its characteristics

 Delete longitudinal reinforcement: to delete one or more reinforcements, simply select them and confirm the deletion by clicking with the right button.
 Insert transverse reinforcement: allows you to insert transverse reinforcements in FRP or FRCM
 Modify transverse reinforcement: allows you to modify a transverse reinforcement in FRP or FRCM previously inserted, stretching it or modifying its characteristics
 Delete transverse reinforcement: to delete one or more reinforcements, simply select them and confirm the deletion by clicking with the right button.

11.2.3.1.3.1 Reinforcement library

In the upper right part, a button allows you to access the library of possible reinforcements.





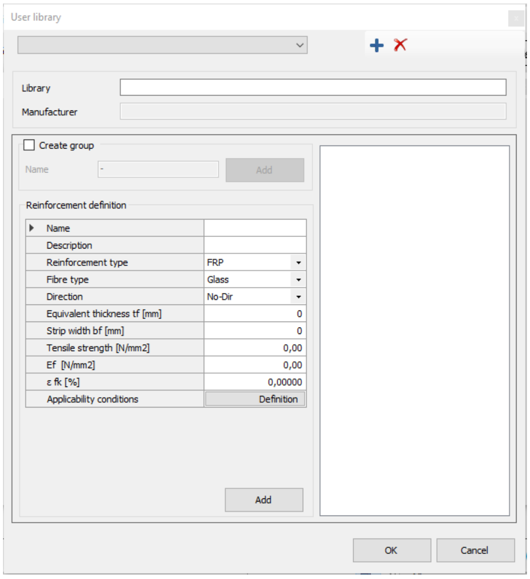
From the drop-down it is possible to view the libraries present and select the desired one.
By checking the "vertical" and / or "transversal" options, it is possible to choose the arrangement of the reinforcement.

Using the "+" symbol it is possible to create a new library to add to the list of those already present.
By clicking on the "Download manufacturer libraries" button you can consult and download the manufacturer libraries.

Once downloaded, the new library will be visible in the dropdown shown above.

If in the design practice we always refer to a specific manufacturer, it is possible to generate your own library that will be available for all future works.

To create a new user library it is necessary to define some parameters:

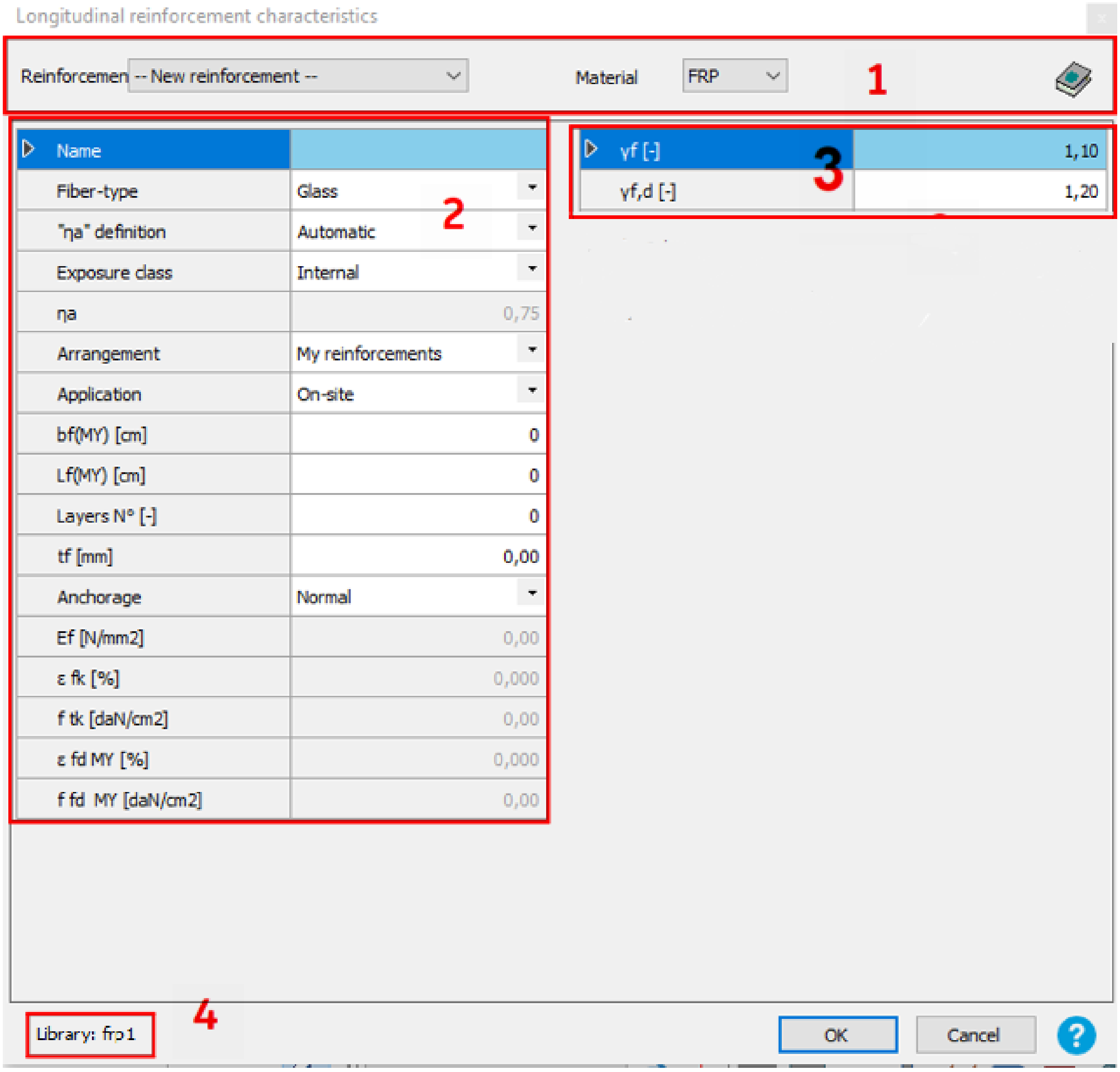


Some of these parameters are descriptive parameters (name, description), others are data necessary for defining the type of reinforcement, the type of fiber, the orientation (uni / bi-directional).

The following data (equivalent thickness, strip width, area, tensile strength, Ef and εfk) are all parameters available from the manufacturers' technical data sheets.

11.2.3.1.3.2 Rinforzi longitudinali

FRP reinforcement:



1
At the top of the previous menu it is possible to:
- insert a reinforcement previously inserted on the column from the list called "Reinforcement";
- select the type of material you intend to use (FRP / FRCM). Specifically, by leaving the FRP session active, all the input required below will comply with the current legislation for FRPs which is CNR-DT 200 R1 / 2013;
- choose the reinforcement fabric from the reinforcement libraries.

2

Name	
Fiber-type	Glass
"ηa" definition	Automatic
Exposure class	Internal
ηa	0,75
Arrangement	My reinforcements
Application	On-site
bf(MY) [cm]	0
Lf(MY) [cm]	0
Layers N° [-]	0
tf [mm]	0,00
Anchorage	Normal
Ef [N/mm2]	0,00
ε fk [%]	0,000
f tk [daN/cm2]	0,00
ε fd MY [%]	0,000
f fd MY [daN/cm2]	0,00

Shows the name of the reinforcement fabric inserted from the library
It reports the type of fiber-resin of the reinforcement inserted from the library
Contains the type of definition of ηa: Automatic / Manual
Exposure class
ηa value
Indicates the arrangement of the reinforcements: My / Mz / My and Mz
Contains the application of On site / Preformed reinforcement
Reinforcement width (MY)
Reinforcement Length (MY)
Number of layers of reinforcement
Thickness of the reinforcement
Type of Anchoring: Normal / Mechanical
Normal modulus of elasticity in the direction of the force
Characteristic deformation
Characteristic tension of the reinforcement
Fiber Calculation Strain at Break (MY)
Calculation resistance of the fiber (MY)

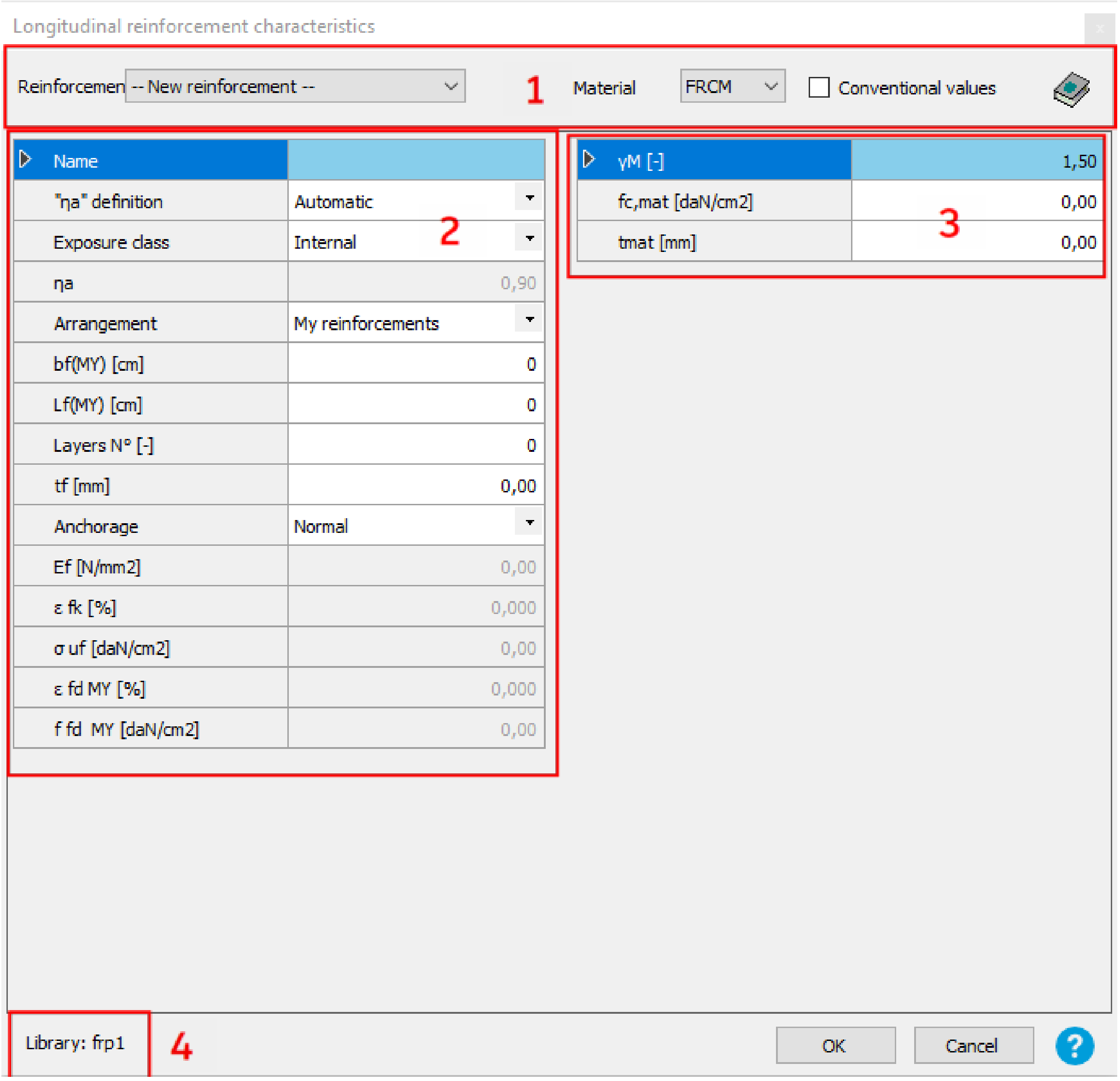
3

γf [-]	1,00
γf,d [-]	1,00

Partial safety factor of the FRP [3.4.1 CNR-DT 200 R1 / 2013]
Safety factor for the detachment of the FRP [3.4.1 CNR-DT 200 R1 / 2013]

4
Indicates the library to which the type of fabric chosen belongs.

+ **FRCM reinforcement:**



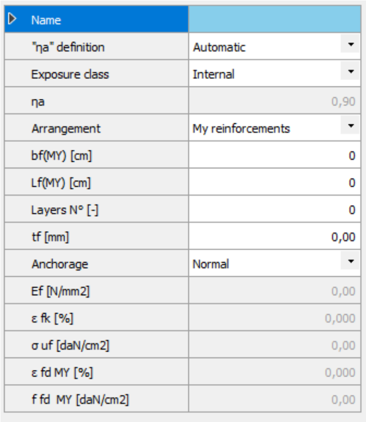
1
At the top of the previous menu it is possible to:

- insert a reinforcement previously inserted on the girder from the list called "Reinforcement";
- select the type of material you intend to use (FRP / FRCM). Specifically, by leaving the FRCM session active, all the input required below will comply with the current legislation for FRCM which is CNR-DT 215/2018;
- choose the reinforcement fabric from the reinforcement libraries;
- choose whether to use conventional values for FRCM fabric.

The use of conventional values refers to what is written in the CNR-DT 215/2018.
In the chapter concerning the calculation of the mechanical characteristics (§3.1) of the aforementioned CNR, reference is made to the concept of "conventional" stresses and deformations which represents the strength of the reinforcement system obtained by detachment tests from conventional supports and as such is dependent on the media type.
The use of the conventional limit deformation and of the competent conventional limit tension, allows to design FRCM reinforcement interventions avoiding *the explicit verification in the comparison of the phenomenon of detachment from the support* or

sliding of the fibers in the matrix at the ends of the reinforcement, otherwise necessary in cases in which this crisis mode is possible.

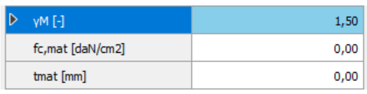
2



Name	
"ηa" definition	Automatic
Exposure class	Internal
ηa	0,90
Arrangement	My reinforcements
bf(MY) [cm]	0
Lf(MY) [cm]	0
Layers N° [-]	0
tf [mm]	0,00
Anchorage	Normal
Ef [N/mm2]	0,00
ε fk [%]	0,000
σ uf [daN/cm2]	0,00
ε fd MY [%]	0,000
f fd MY [daN/cm2]	0,00

Shows the name of the reinforcement fabric inserted from the library
Contains the type of definition of ηa: Automatic / Manual
Exposure class
Value ηa
Indicates the arrangement of the reinforcements: My / Mz / My and Mz
Reinforcement Length (MY)
Reinforcement width (MY)
Number of layers of reinforcement
Thickness of the reinforcement
Type of Anchoring: Normal / Mechanical
Ultimate stress at break characteristic by traction of the dry fabric
Characteristic deformation
Ultimate stress at break characteristic by traction of the dry fabric
Fiber Calculation Strain at Break (MY)
Calculation resistance of the fiber (MY)

3



γM [-]	1,50
fc,mat [daN/cm2]	0,00
tmat [mm]	0,00

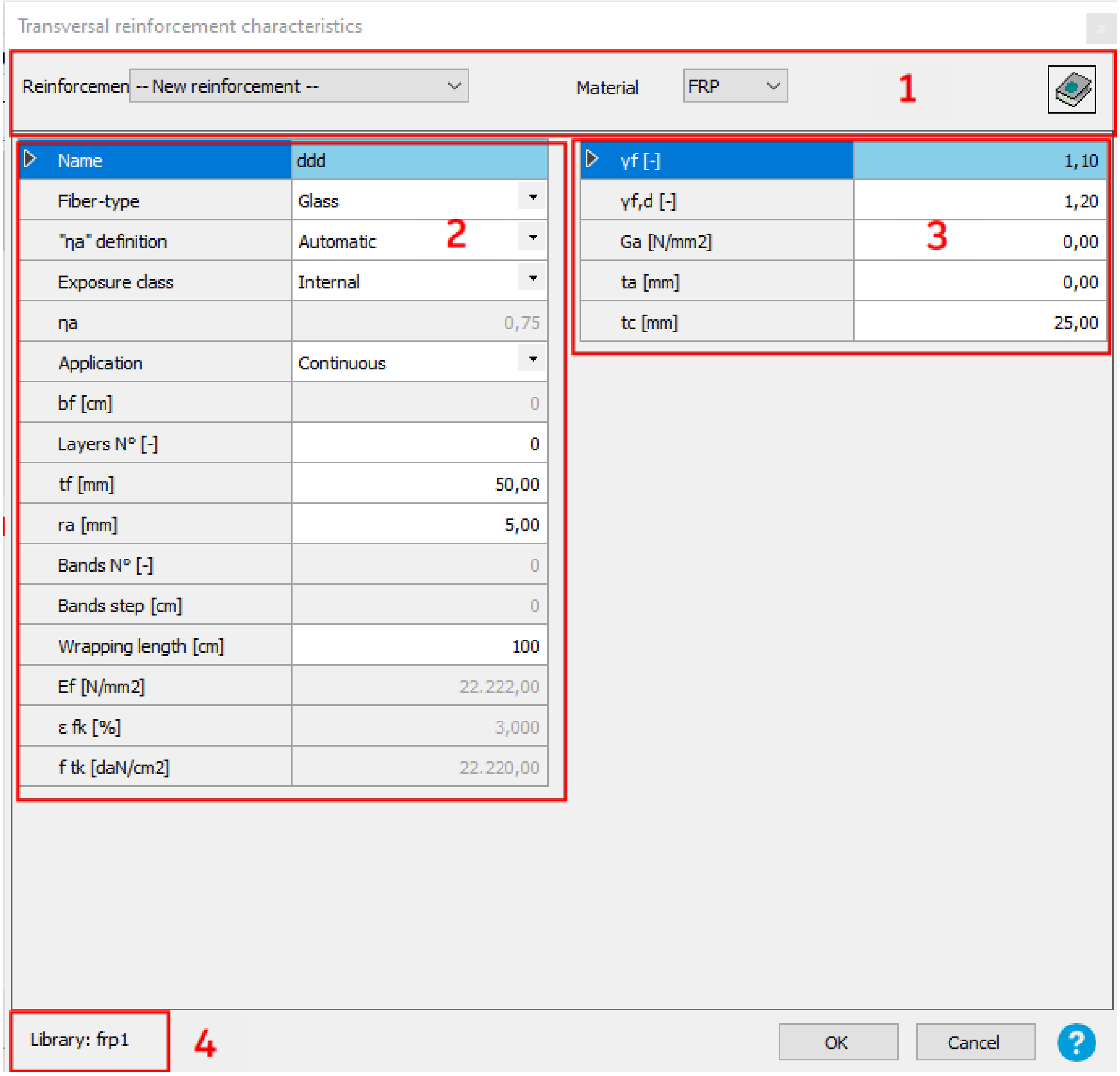
Partial safety coefficient of the FRCM [3.1 CNR-DT 215 2018]
Ultimate stress at break characteristic by traction of the dry fabric
FRCM single layer matrix thickness (1cm)

4

Indicates the library to which the type of fabric chosen belongs.

11.2.3.1.3.3 Transverse reinforcement

FRP reinforcement:



1

At the top of the previous menu it is possible to:

- insert from the list called "Reinforcement" a reinforcement previously inserted on the girder;
- select the type of material you intend to use (FRP / FRCM). Specifically, by leaving the FRP session active, all the input required below will comply with the current legislation for FRPs which is CNR-DT 200 R1 / 2013;
- choose the reinforcement fabric from the reinforcement libraries.

2

Name	ddd
Fiber-type	Glass
"ηa" definition	Automatic
Exposure class	Internal
ηa	0,75
Application	Continuous
bf [cm]	0
Layers N° [-]	0
tf [mm]	50,00
ra [mm]	5,00
Bands N° [-]	0
Bands step [cm]	0
Wrapping length [cm]	100
Ef [N/mm2]	22.222,00
ε fk [%]	3,000
f tk [daN/cm2]	22.220,00

Shows the name of the reinforcement fabric inserted from the library
Reports the fiber type of the reinforcement inserted from the library
na definition (Automatic/Manual)
Exposure class (Internal/External/Aggressive)
na value
Application (Continuous/Discontinuous)
Width of the reinforcement
Number of layers of reinforcement
Thickness of the reinforcement
Rounding radius
Number of bands
Step bands
Wrapping length
Elastic modulus in the direction of the reinforcement
Characteristic deformation of the reinforcement
Characteristic tension of the reinforcement

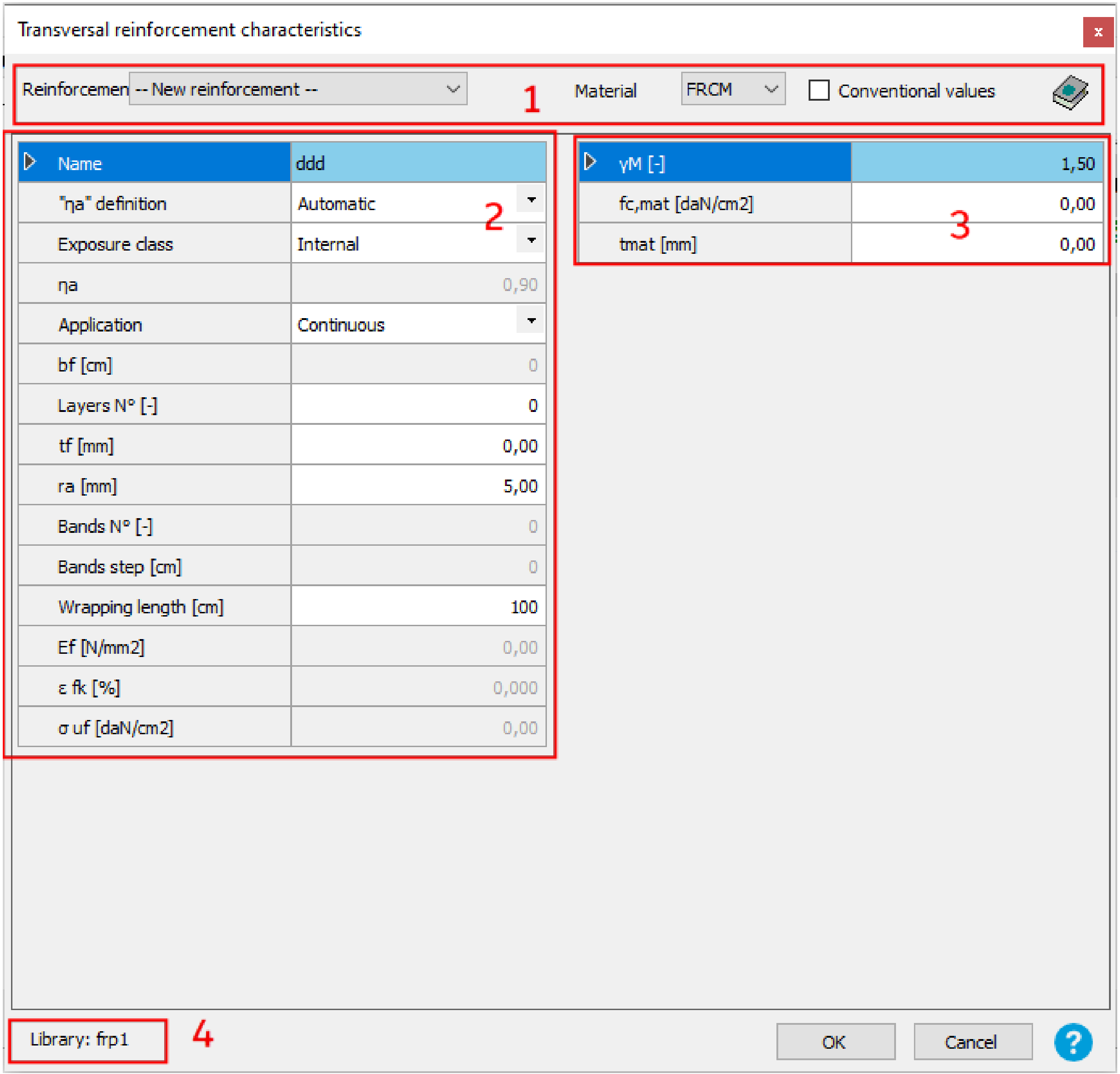
3

γf [-]	1,00
γf,d [-]	1,00
Ga [N/mm2]	0,00
ta [mm]	0
tc [mm]	25

Partial safety factor of the FRP [3.4.1 CNR-DT 200 R1 / 2013]
Safety factor for the detachment of the FRP [3.4.1 CNR-DT 200 R1 / 2013]
Tangential elastic modulus of the adhesive
Nominal thickness of adhesive
Effective thickness of concrete participating in the deformability of the interface

4
Indicates the library to which the type of fabric chosen belongs.

FRCM reinforcement:



1
At the top of the previous menu it is possible to:

- insert a reinforcement previously inserted on the column from the list called "Reinforcement";
- select the type of material you intend to use (FRP / FRCM). Specifically, by leaving the FRCM session active, all the input required below will comply with the current legislation for FRCM which is CNR-DT 215/2018;
- choose the reinforcement fabric from the reinforcement libraries;
- choose whether to use conventional values for FRCM fabric.

The use of conventional values refers to what is written in the CNR-DT 215/2018.
In the chapter concerning the calculation of the mechanical characteristics (§3.1) of the aforementioned CNR, reference is made to
the concept of "conventional" stresses and deformations which represents the strength of the reinforcement system obtained by
detachment tests from conventional supports and as such is dependent on the media type.
The use of the conventional limit deformation and of the competent conventional limit tension, allows to design FRCM reinforcement

interventions avoiding *the explicit verification in the comparison of the phenomenon of detachment from the support* or sliding of the
fibers in the matrix at the ends of the reinforcement, otherwise necessary in cases in which this crisis mode is possible.

2

Name	ddd
"ηa" definition	Automatic
Exposure class	Internal
ηa	0,90
Application	Continuous
bf [cm]	0
Layers N° [-]	0
tf [mm]	0,00
ra [mm]	5,00
Bands N° [-]	0
Bands step [cm]	0
Wrapping length [cm]	100
Ef [N/mm2]	0,00
ε fk [%]	0,000
σ uf [daN/cm2]	0,00

Shows the name of the reinforcement fabric inserted from the library
na definition (Automatic/Manual)
Exposure class (Internal/External/Aggressive)
na value
Report the type of application
Width of the reinforcement
Number of layers of reinforcement
Thickness of the reinforcement
Rounding radius
Number of bands of reinforcement
Step bands of reinforcement
Wrapping length
Elastic modulus in the direction of the reinforcement
Characteristic deformation of the reinforcement
Calculation tension at break of dry fabric

3
Calculation parameters

γM [-]	1,00
fc,mat [N/mm2]	0,00
tmat [mm]	0

Partial safety coefficient of the FRCM [3.1 CNR-DT 215 2018]
Ultimate stress at break characteristic by traction of the dry fabric
FRCM single layer matrix thickness (1cm)

4
Indicates the library to which the type of fabric chosen belongs.

11.2.3.1.4 .dxf setting

"DXF" setting

It allows to export the reinforcement scheme in .dxf format.

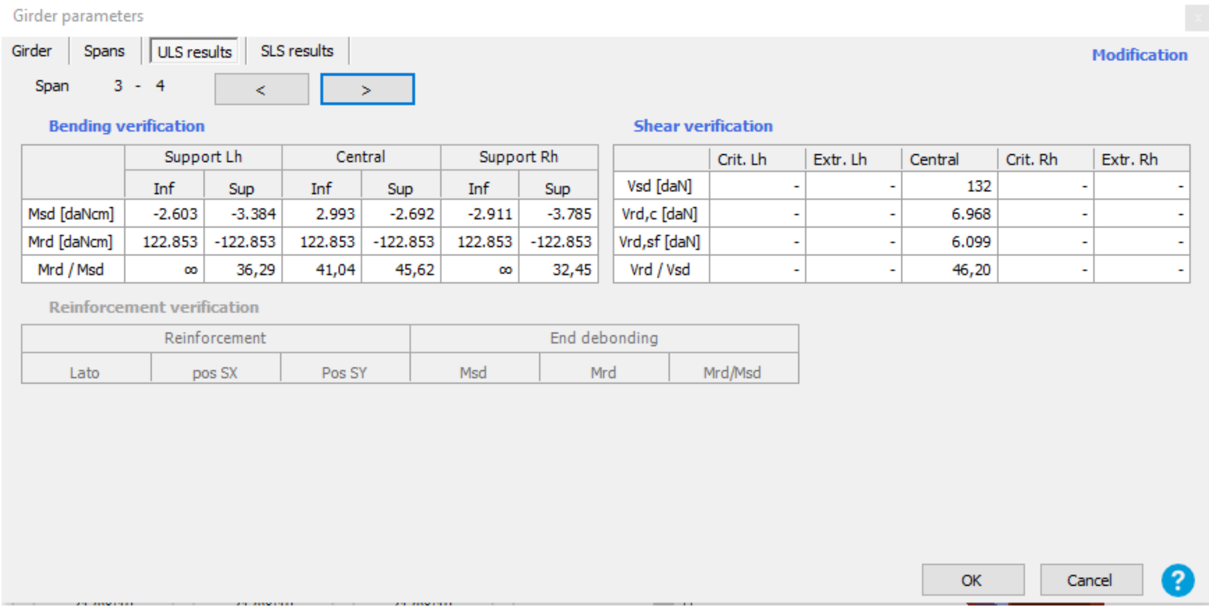
11.2.3.1.5 Results/verification

When the "tab" of the beam is dragged from the "Beams to calculate" column to the "Calculated beams" column, it will be possible to view the results of the calculation.

11.2.3.1.5.1 ULS Results

The ULS Results environment, available when a beam is in the Calculated Beams tab, reports the outcome of ULS checks conducted on the beam.
In particular, the results obtained with regard to the bending and shear verification can

be consulted.
You can consult the results of all the spans by means of the appropriate buttons.



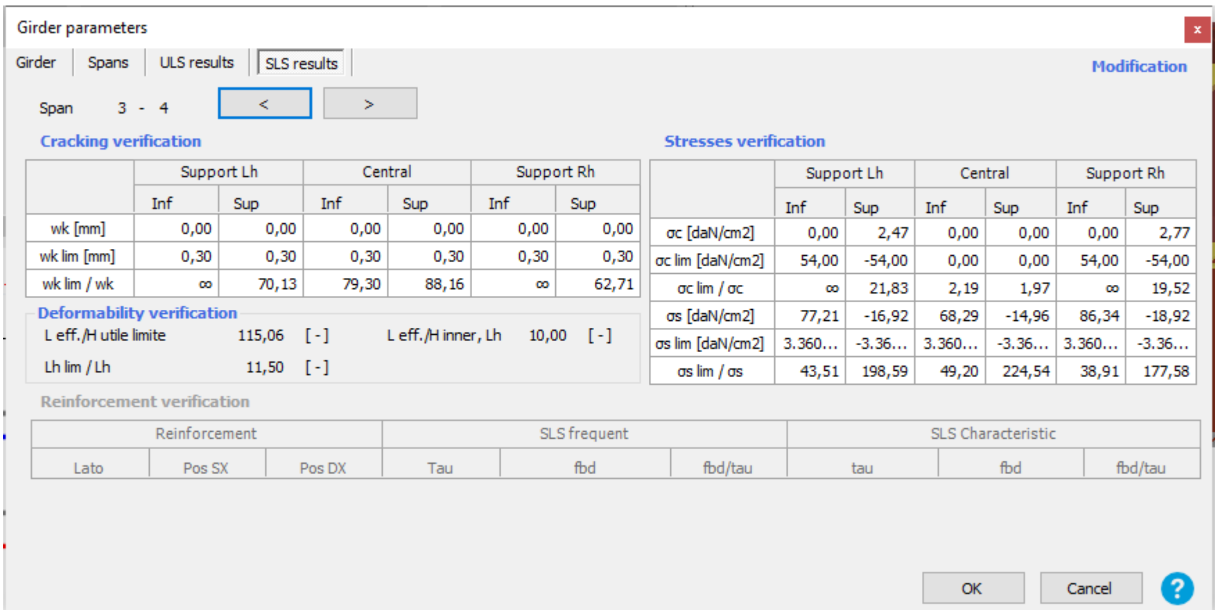
	Support Lh		Central		Support Rh	
	Inf	Sup	Inf	Sup	Inf	Sup
Msd [daNcm]	-2.603	-3.384	2.993	-2.692	-2.911	-3.785
Mrd [daNcm]	122.853	-122.853	122.853	-122.853	122.853	-122.853
Mrd / Msd	∞	36,29	41,04	45,62	∞	32,45

	Crit. Lh	Extr. Lh	Central	Crit. Rh	Extr. Rh
Vsd [daN]	-	-	132	-	-
Vrd,c [daN]	-	-	6.968	-	-
Vrd,sf [daN]	-	-	6.099	-	-
Vrd / Vsd	-	-	46,20	-	-

Reinforcement			End debonding		
Lato	pos SX	Pos SY	Msd	Mrd	Mrd/Msd

11.2.3.1.5.2 SLS Results

The SLS Results environment, available when a beam is in the Calculated Beams tab, reports the outcome of the SLS checks conducted on the beam.
In particular, the results obtained with regard to cracking, deformation and stress tests (on concrete and steel) can be consulted.
You can consult the results of all the spans by means of the appropriate buttons.



	Support Lh		Central		Support Rh	
	Inf	Sup	Inf	Sup	Inf	Sup
wk [mm]	0,00	0,00	0,00	0,00	0,00	0,00
wk lim [mm]	0,30	0,30	0,30	0,30	0,30	0,30
wk lim / wk	∞	70,13	79,30	88,16	∞	62,71

	Support Lh		Central		Support Rh	
	Inf	Sup	Inf	Sup	Inf	Sup
σc [daN/cm2]	0,00	2,47	0,00	0,00	0,00	2,77
σc lim [daN/cm2]	54,00	-54,00	0,00	0,00	54,00	-54,00
σc lim / σc	∞	21,83	2,19	1,97	∞	19,52
σs [daN/cm2]	77,21	-16,92	68,29	-14,96	86,34	-18,92
σs lim [daN/cm2]	3.360...	-3.36...	3.360...	-3.36...	3.360...	-3.36...
σs lim / σs	43,51	198,59	49,20	224,54	38,91	177,58

Reinforcement			SLS frequent			SLS Characteristic		
Lato	Pos SX	Pos DX	Tau	fbd	fbd/tau	tau	fbd	fbd/tau

11.3 Columns

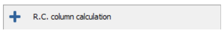
The Column module allows the verification and / or design of new or existing reinforced concrete columns, with the possibility of customizing the stresses and reinforcements and exporting the executive drawings of the latter.

11.3.1 R.C. Column

The R.C. Columns module allows the verification and / or design of new or existing reinforced concrete columns, with the possibility of customizing the stresses and reinforcements and exporting the executive drawings of the latter.

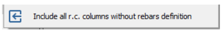
11.3.1.1 Creation of a calculation tab

The menu dedicated to RC columns has several options:



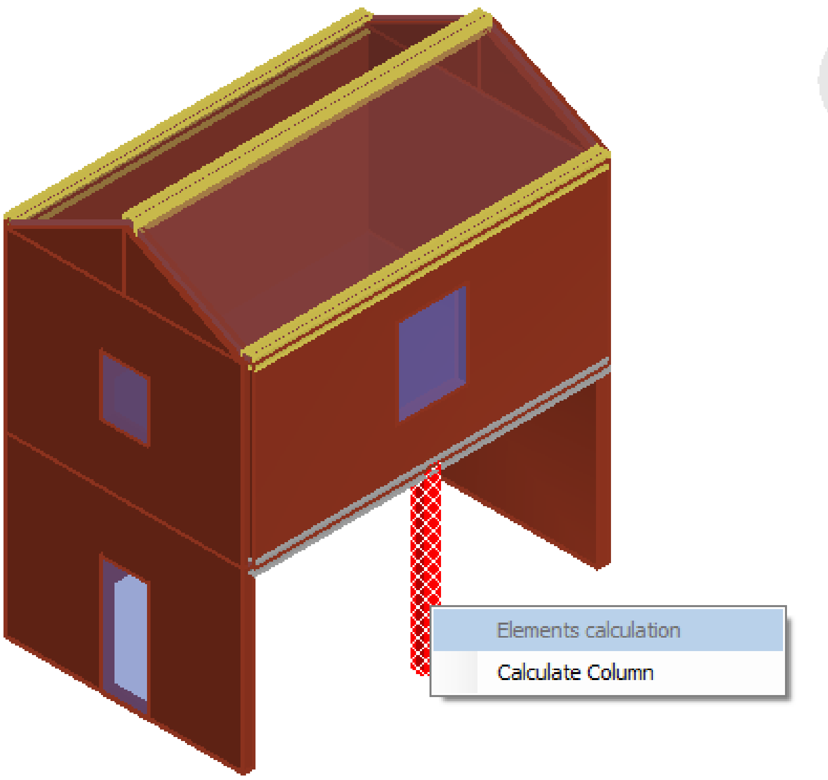
It allows you to create a new calculation tab. By selecting this option, the tab will be created as a user card.

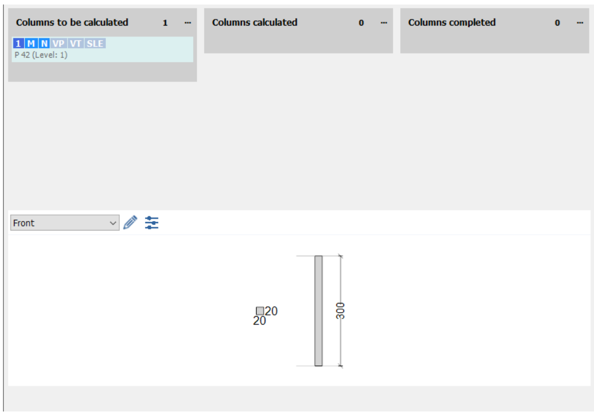
By selecting this option, a calculation tab will be created for each column present in the structure model. The created tab will be model tabs.



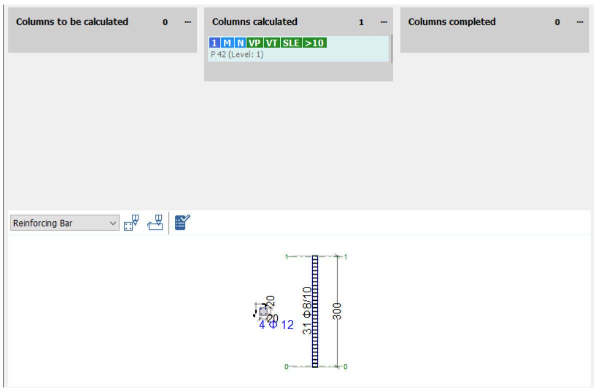
By selecting this option, a calculation tab will be created for each column without reinforcement present in the structure model. The created tabs will be model tabs.

It is also possible to create a calculation tab starting from a column in the model.
To do this, simply right-click on the "Column" element present in the model and choose the "Calculate Column" option.





After selecting this option, the calculation window opens and the form is created and placed in the column "Columns to be calculated".



To calculate the column, simply drag the tab into the "Calculated Columns" column.

The information labels on each sheet visually summarize the results of the checks carried out.

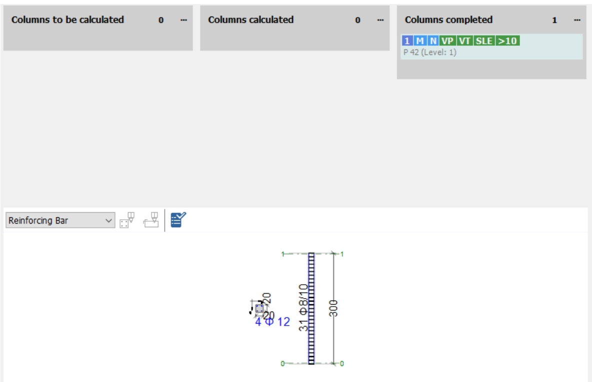
VP Green if the bending test is successful. Red otherwise.

VT Green if the shear check is passed. Red otherwise.

SLE Green if the SLS test passed. Red otherwise.

>10 Value of the safety factor

! Indicates that there are warnings regarding the calculation made. To view them, simply place the pointer over them.



Once the calculation is finished, by dragging the tab in the "Columns completed" column, it is possible to produce the output files of the calculation.

Furthermore, bringing the tab in the "Completed Columns" column will confirm any changes made to the reinforcement and the column in the Structure environment will be updated accordingly.

11.3.1.2 Codes

The reference legislation for the design and verification of pillar elements is the D.M. 17 January 2018.

In particular, the chapters of greatest interest are:

- Chapter 3 - Construction actions
- Chapter 4.1 - Civil and industrial constructions - Concrete constructions
- Chapter 7.4 - Design for seismic actions - Concrete constructions
- Chapter 8 - Existing buildings

In the sections below, clarifications are reported regarding the following topics:

- Calculation stresses according to the resistance hierarchy;
- Resistance checks;
- SLE checks.
- Calculation Zv.

11.3.1.2.1 Calculation stresses according to the resistance hierarchy

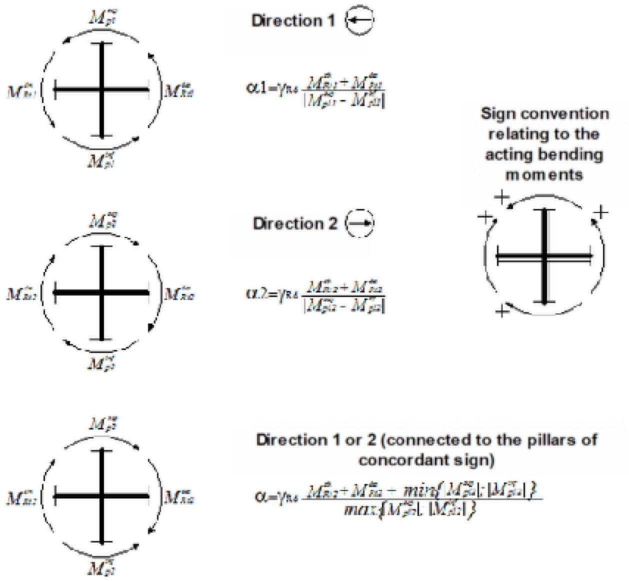
Chapter 7.4.4 of the NTC 2018 prescribes the methods for calculating the stresses acting on the structural elements with the aim of satisfying the requirements of the resistance hierarchy.
In particular, for the pillars (§ 7.4.4.2), the calculation stresses for the strength checks are obtained as follows:

+ Axial bending

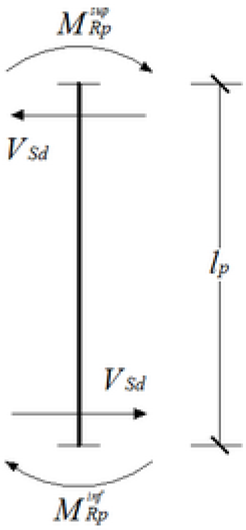
For structures in CD " A " and CD " B ", the bending moments of calculation in the pillars are obtained by multiplying the moments deriving from the analysis by the amplification factor α, which is given by the expression:

$$\alpha = \gamma_{Rd} \frac{\left|\sum M_{Rt}\right|}{\left|\sum M_p\right|}$$

in which γ_{Rd} is equal to 1.3 both for CD " A " and for CD " B ", $\sum M_{Rt}$ is the sum of the resisting moments of the beams converging in a node, having a concordant direction, and $\sum M_p$ is the sum of the moments in the pillars above and below the same node, obtained from the analysis.
In the event that the moments in the pillars are in discordant direction, only the greater value must be placed in the denominator of the factor, while the lower must be added to the resisting moments of the beams.
The amplification factor α must be calculated for both directions of the seismic action, applying the amplification factor α calculated for each direction to the moments calculated in the pillars with the action acting in the same direction.
For the base section of the ground floor pillars, the greater of the moment resulting from the analysis and the moment used for the top section of the pillar is applied.
Amplification factors are not applied to the top sections of the pillars of the top floor.



The value of the calculation moment obtained by applying the above procedure must be associated with the most unfavorable value of the normal stress obtained from the analysis, for each direction of the seismic action.



Shear

The shear stresses in the pillars to be used for the checks and sizing of the reinforcements are obtained from the equilibrium condition of the pillar subject to the action of the resisting moments in the upper M^{s}_{Rp} and lower M^{i}_{Rp} end sections according to the expression:

$$V_{Sd} = \gamma_{Rd} \frac{M^{s}_{Rp} + M^{i}_{Rp}}{l_p}$$

in which γ_{Rd} it results 1.3 if CD "A" and 1.1 if CD "B", l_p is the length of the pillar.

11.3.1.2.2 Resistance checks

Chapter 4.1.2.3 of the NTC 2018 prescribes the ULS resistance checks to be carried out on reinforced concrete members.
In particular, the software provides for the following checks:

Compression

The pure compression verification of the section in reinforced concrete, in the presence of seismic action, is set in the form:

$$N_{Ed} \leq N_{Rd}$$

Chapter 7.4.4.2.1 dictates that

$$N_{Rd} = 0{,}55\, A_c \cdot f_{cd} \quad \text{for} \quad \text{CD "A"}$$

$$N_{Rd} = 0{,}65\, A_c \cdot f_{cd} \quad \text{for} \quad \text{CD "B"}$$

Axial bending

The verification of each section with the deflected deflection is placed in the form:

$$\eta = \left[\left(\frac{M_{Ed,y}}{M_{Rd,y}} \right)^{\beta} + \left(\frac{M_{Ed,z}}{M_{Rd,z}} \right)^{\beta} \right]^{-1} \geq 1$$

dove:

$M_{Ed,y}$ and $M_{Ed,z}$ are the calculation values of the two straight bending components of the action around the y and z axes;
$M_{Rd,y}$ and $M_{Rd,z}$ are the calculation values of the resisting moments of straight bending corresponding to NEd evaluated separately around the y and z axes.

The NTC18 propose to analyze the problem of determining the resistant capacity of sections subject to deflected pressure through a relationship with which to calibrate the coefficient β to be introduced in the equation.
The Monti-Alessandri method (2009) was adopted in RC column verification (2009), who proposed a method for calibrating the β coefficient as a function of the normal stress and the geometric and mechanical characteristics of the section.

Shear

Sections subjected to shear stresses and equipped with transverse reinforcement must, according to Chapter 4.1.2.3.5.2, meet the following requirements:

$$V_{Ed} \leq V_{Rd}$$

where

$$V_{Rd} = \min(V_{Rsd}; V_{Rcd})$$

where:

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

$$V_{Rcd} = 0{,}9 \cdot d \cdot b_w \cdot \alpha_c \cdot \nu \cdot f_{cd} \cdot \frac{(\cot\alpha + \cot\theta)}{1 + \cot^2\theta}$$

where:

- d is the useful height of the section;
- bw is the base of the section;
- αc is an increased coefficient for compressed members (for more details, see §4.1.2.3.5.2 directly);
- Asw is the area of the transverse reinforcement;
- s is the center distance between two consecutive transverse reinforcements;
- α is the angle of inclination of the transverse reinforcement with respect to the axis of the beam;
- v = 0.5;
- θ is the inclination angle of the concrete struts.

11.3.1.2.3 SLS Verification

Chapter 4.1.2.2 of the 2018 NTC prescribes the checks to be carried out against the Service Limit States.
In particular, Chapter 4.1.2.2.5 regulates the maximum compressive stress to which concrete and steel can be subjected:

+ Concrete stress

The maximum compressive stress of the concrete must comply with the following condition:

$\sigma_{c,max} \leq 0{,}60\, f_{ck}$ **for characteristic combination**

$\sigma_{c,max} \leq 0{,}45\, f_{ck}$ **for quasi-permanent combination**

✚ Steel stress

The maximum tension to which the steel can be subjected, due to the characteristic combination, is:

$$\sigma_{s,max} \leq 0{,}8\, f_{yk}$$

11.3.1.2.4 ζv calculation

Chapter 8.3 of the NTC 2018 prescribes the methodologies for assessing the safety of existing structures and the introduction of any restrictions on use in relation to the expected loads.
In particular, for vertical loads: *"The restriction of use can change from portion to portion of the building and, for the i-th portion, it is quantified through the ratio* $\boldsymbol{z}_{v,i}$ *between the maximum value of the vertical variable overload that can be supported by that part of the construction and the value of the variable vertical overload that would be used in the design of a new construction "*.

In the case of columns, the calculation of the $\boldsymbol{z}_v$ factor is managed as follows:

Shear stress

1. For the pillar in question, all the project provisions factored at ULS for variable overloads and permanent loads plus any other variables are calculated.
2. For each calculation section of the pillar in question, the stressing and resistant shear values are compared, purified by the portion of stress carried by the effect of the permanent loads plus any other variables, in the two directions.
3. The minimum ratio obtained corresponds to the shear design ζv of the pillar.

Axial bending

Since the strength of the concrete section subjected to normal stress depends on the latter, for all possible arrangements, the one
that produces the minimum distance, in the interaction domain N-M, from the failure curve is calculated, and the ratio between this is
calculated. 'last is the distance between the $(N,M)_{perm}$ value, produced by the permanent loads, and the failure curve (eg the
residual strength). The values calculated in the two directions are then combined according to the criteria adopted for the pressure
bending in the resistance tests.

11.3.1.2.5 Constructive singularities

This section illustrates the construction details, in terms of geometric and reinforcement limitations, prescribed by the Technical Construction Standards of 2018, with reference both to the general directives (Chap. 4), and to those referring to buildings located in seismic areas. (Chapter 7).

11.3.1.2.5.1 Geometric limitations

The minimum size of the cross section must not be less than 25 cm.
The ratio of the minimum and maximum sides of the cross section should not be less

than 0.3. Otherwise, the element will be assimilated to a load-bearing wall.

In the seismic zone, the length of the critical zone is assumed to be equal to the greater of:

- the height of the section;
- 1/6 of the free height of the pillar;
- 45 cm;
- the free height of the pillar if this is less than 3 times the height of the section.

11.3.1.2.5.2 Longitudinal reinforcements

In the current section of the column the reinforcement must be between:

$$A_{s\min} = 0{,}10\left(N_{Ed}/f_{yd}\right) \quad \text{and in any case not less than} \quad 3‰\, A_c$$

$$A_{s\max} = 4\% A_c$$

with NEd the calculation axial compression force, fyd the yield stress of the reinforcement calculation, A total area of the longitudinal reinforcement and Ac area of the gross section of the pillar.
The bars parallel to the axis must have a diameter greater than or equal to 12 mm.
For the entire length of the pillar, the center distance between the bars must not exceed 30 cm.

On seismic areas, there are further limitations:

- in each section of the beam, unless there are justifications showing that the collapse modes of the section are consistent with the ductility class adopted, the geometric ratio ρ relating to the tensioned reinforcement, regardless of whether the tensioned reinforcement is the one at the upper edge of section As or that at the lower edge of section Ai, must be included within the following limits:

$$\frac{1{,}4}{f_{yk}} < \rho < \rho_{comp} + \frac{3{,}5}{f_{yk}}$$

where:

- ρ is the geometric ratio relative to the stretched reinforcement equal to As / (bh) or Ai / (bh)
- ρcomp is the geometric ratio relating to the compressed reinforcement
- fyk is the characteristic yield stress of steel (in MPa);

- in the critical areas of the beam, moreover, it must be ρcomp = 1/2 ρ and in any case = 0.25 ρ;
- the upper reinforcement, arranged for the negative moment at the ends of the beams, must be contained, for at least 75%, within the width of the core and in any case, for the 'T' or 'L' sections, within a slab band equal respectively to the width of the pillar, or to the width of the pillar increased by 2 times the thickness of the slab on each side of the pillar, depending on whether an orthogonal beam is missing or present in the joint. At least ¼ of the aforementioned reinforcement must be maintained along the entire length of the beam.

11.3.1.2.5.3
Transverse reinforcements

The transverse reinforcements must be placed at a distance between centers not greater than 12 times the minimum diameter of the
bars used for the longitudinal reinforcement, with a maximum of 250 mm.
The diameter of the stirrups must not be less than 6 mm and ¼ of the maximum diameter of the longitudinal bars.

In the event of a seismic event, further limitations are foreseen in the critical areas:

- the bars placed on the corners of the section must be contained by the stirrups;
- at least one out of every two bars, of those arranged on the sides, must be held by internal stirrups or by fastenings;
- unsecured bars must be less than 15 cm from a fixed bar and 20 cm from a fixed bar;
- the diameter of the containment stirrups and ligatures must not be less than 6 mm;
- the spacing of the stirrups equal to the smallest of the following quantities:

- one third and one half of the shorter side of the cross section (CD "" A "" and CD "" B ""),
- 125 cm and 175 cm (CD "" A "" and CD "" B ""),
- 6 and 8 times the diameter of the longitudinal bars that connect (CD ″" A "" and "CD" B "");

$$s \leq \min\left(\frac{1}{3} l_{min}; 125\ cm; 6\emptyset_{long}\right) \quad \textbf{for} \quad \text{CD "A"}$$

$$s \leq \min\left(\frac{1}{2} l_{min}; 175\ cm; 8\emptyset_{long}\right) \quad \textbf{for} \quad \text{CD "B"}$$

- stirrups must be available for a minimum quantity of not less than:

$$\frac{A_{st}}{s} \geq \begin{cases} 0{,}08 \dfrac{f_{cd} b_{st}}{f_{yd}} & \textbf{for} \ \text{CD" A"} \ \textbf{outside the critic zone and for} \ \text{CD" B"} \\ 0{,}12 \dfrac{f_{cd} b_{st}}{f_{yd}} & \textbf{for} \ \text{CD" A"} \end{cases}$$

where Ast is the total area of the arms of the stirrups, bst is the distance between the outermost arms of the stirrups and s is the
spacing of the stirrups.

11.3.1.2.6
Calculation of reinforcements

The calculation of the reinforcements using fiber-reinforced materials is carried out in accordance with Circular no. 7 of 21 January 2019, CNR-DT R1 / 2013 and CNR-DT 215/2018.

11.3.1.2.6.1 Longitudinal reinforcements

✚ULS Analysis

The calculation procedure is based on the comparison between the design stressing moment Msd and the design resistant moment of the reinforced section Mrd.

Msd ≤ Mrd

The fundamental assumptions on which the ULS analysis of the sections of reinforced concrete is based reinforced with
FRP are as follows:

- preservation of the planarity of the straight sections until failure, so that the diagram of the normal deformations is linear;
- perfect adhesion between the component materials (steel-concrete, FRP-concrete);
- zero tensile strength of concrete;
- constitutive bonds of concrete and steel in compliance with current legislation;
- constitutive bond of the linear elastic fiber-reinforced composite until failure.

Flexural failure is hypothesized to occur when one of the following conditions occurs:

- achievement of maximum plastic deformation in compressed concrete, εcu, as defined by current legislation;
- achievement of a maximum deformation in the FRP reinforcement, εfd.

Given the geometry and materials of the section of the element to be reinforced, the moment of first cracking Mcr is evaluated,
evaluating the average simple tensile strength fctm given by eqs. 11.2.3.a, 11.2.3.b and 11.2.4 - NTC 2018.
The initial deformation of the concrete at the stretched edge is evaluated as:

$$\varepsilon_0 = \frac{M_0 \cdot (h - x_0)}{I_0 \cdot E_c}$$

in which:
-M0 is the initial moment acting in section before the application of the fibers produced by the permanent loads at the SLS;
-x0 is the distance of the neutral axis from the compressed limb;
-I0 is the moment of inertia of the section taking into account any cracking;
-Ec is the elastic modulus of concrete evaluated in accordance with Eq. 11.2.5, NTC 2018.

Given the characteristics of the composite and its methods of application, the maximum design deformation is evaluated: (eq. 4.14 CNR-DT 200 R1 / 2013)

$$\varepsilon_{fd} = \min\{\eta_a\ \varepsilon_{fk} / \gamma_f;\ \varepsilon_{fdd}\}$$

where:

$\varepsilon_{fk} = f_{fk} / E_f$ represents the characteristic deformation at break of the reinforcement;

$\varepsilon_{fdd} = \frac{f_{fdd,2}}{E_f} \geq \varepsilon_{sy} - \varepsilon_0$ represents the maximum value attributable to the deformation of the composite during the design so that the intermediate detachment does not occur (eq. 4.7 - CNR-DT 200 R1 / 2013);
in which:

$f_{fdd,2} = \frac{k_q}{\gamma_{f,d}} \cdot \sqrt{\frac{E_f}{t_f} \cdot \frac{2 \cdot k_b \cdot k_G}{FC} \sqrt{f_{cm} \cdot f_{ctm}}}$ represents the maximum design stress of the composite (eq. 4.6 - CNR-DT 200 R1 / 2013)

The ultimate limit state resistance for end detachment is evaluated by verifying that the tension of the composite does not exceed the design tension of the reinforcement system for end detachment (eq. 4.4 - CNR-DT 200 R1 / 2013), calculated as:

$$f_{f\,dd} = \frac{1}{\gamma_{f,d}} \cdot \sqrt{\frac{2 \cdot E_f \cdot \Gamma_{Fd}}{t_f}}$$

with $\gamma_{f,d}$ partial coefficient for FRP materials (3.4.1 CNR - DT 200 R1 / 2013), Γ_{Fd} represents the specific fracture energy calculated as (eq. 4.2 - CNR-DT 200 R1 / 2013):

$$\Gamma_{Fd} = \frac{k_b \cdot k_G}{FC} \sqrt{f_{cm} \cdot f_{ctm}}$$

The calculation of the optimal anchoring length is also performed (eq. 4.1 - CNR-DT 200 R1 / 2013):

$$l_{ed} = \max\left\{\frac{1}{\gamma_{Rd} \cdot f_{bd}} \sqrt{\frac{\pi^2 \cdot E_f \cdot t_f \cdot \Gamma_{Fd}}{t_f}}, 200\ mm\right\}$$

In the case of anchorage lengths, lb, less than the optimal one, led, the design voltage must be suitably reduced in accordance with the relationship:

$$f_{f\,dd,rid} = f_{dd} \cdot \frac{l_b}{l_{ed}} \cdot \left(2 - \frac{l_b}{l_{ed}}\right)$$

Once the position of the neutral axis has been determined, the value of the resisting moment MRd can be determined starting from the equation of equilibrium for rotation around the axis passing through the center of gravity of the stretched reinforcement and parallel to the neutral axis (eq. 4.16 - CNR -DT 200 R1 / 2013):

$$M_{Rd} = \frac{1}{\gamma_{Rd}} \cdot \left[\psi \cdot b \cdot x \cdot f_{cd} \cdot (d - \lambda \cdot x) + A_{s2} \cdot \sigma_{s2} \cdot (d - d_2) + A_f \cdot \sigma_f \cdot d_1\right]$$

where the three components represent respectively:
- the contribution of compressed concrete
- the contribution of steel, stretched and compressed
- the contribution of FRP reinforcement

Design resistance to bending of the element reinforced with FRP in the presence of axial force (axial bending)

The principles and rules of application introduced for bending reinforcement apply (CNR-DT 200 R1 / 2013 § 4.2.2.1) also taking into account the dependence of the design value of the resisting moment of the reinforced section, MRd, on that of the normal stress. soliciting, N_{Sd}.
The application rules introduced for the evaluation of the design resistance to bending apply (CNR-DT 200 R1 / 2013 § 4.2.2.1), with the only variation that in the equilibrium equation for translation along the axis of the pillar must appear normal stressing design effort, N_{Sd}.

State of the structure at reinforcement

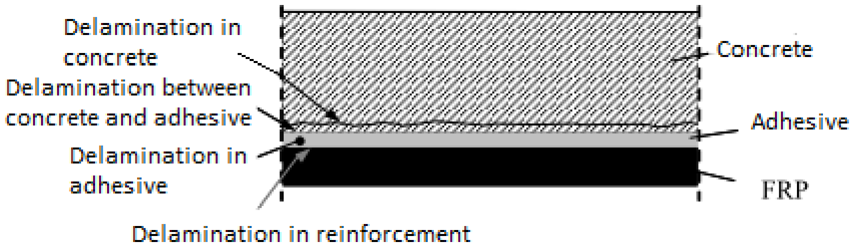
In the hypothesis that the FRP reinforcement is applied on an element subject to a pre-existing stress, which corresponds to an applied moment M_0, the program proceeds to evaluate the initial deformation state considering a different moment of inertia depending on whether M_0 is greater or less than the moment of first crack M_{cr}.

The calculation is performed in the hypothesis of linear elastic behavior of the two materials making up the beam and, in particular, of the inability of the concrete to withstand tensile stresses.

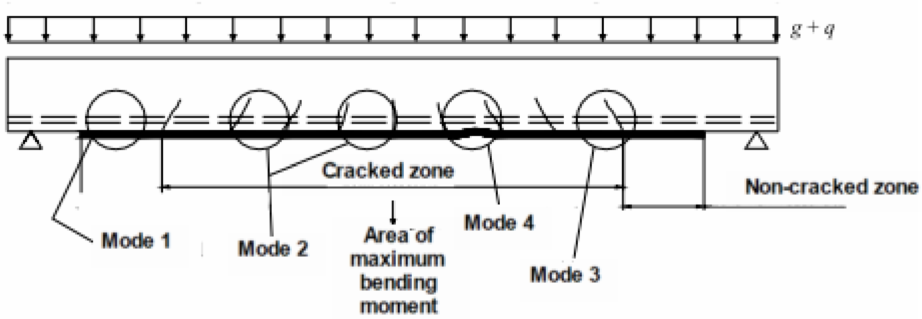
Security checks against delamination

In the reinforcement of reinforced concrete elements using sheets or fabrics of composite material, the role of adhesion between concrete and composite assumes great importance as the mechanism of rupture by delamination (loss of adhesion) is of a fragile type. In the spirit of the criterion of hierarchy of resistances, this crisis mechanism must not precede the collapse by bending or cutting of the reinforced element.

Delamination can occur inside the adhesive, between concrete and adhesive, in the concrete or inside the reinforcement (for example between layers of fabric warped with different inclination angles of the fibers). The program considers that the reinforcement is placed correctly, therefore, since the adhesive resistance is generally much higher than the tensile strength of the concrete, that delamination always takes place inside the latter with the removal of one layer of material.



The modes of collapse by delamination of foils or fabrics used for bending reinforcement can be classified into the following four categories:



- Mode 1 (Delamination of ends);
- Mode 2 (Intermediate delamination, caused by bending cracks in the beam);
- Mode 3 (Delamination caused by diagonal cutting cracks);
- Mode 4 (Delamination caused by irregularities and roughness of the concrete surface).

The program allows the verification of modalities 1 and 2 only, being the ones that occur most frequently in ordinary situations.

✚SLS Analysis

In a beam reinforced with FRP, stress concentrations (tangential and normal) occur at the interface between concrete and reinforcement, located in correspondence with transverse cracks in the concrete, especially at the ends of the reinforcement. These concentrations can cause cracking of the interface triggering the detachment between the two materials.

It is advisable that, under operating conditions, the opening of the aforementioned cracks should not occur, especially in the presence of loading cycles and freeze / thaw cycles. The competent verification can be performed by a calculation of the interface stresses using linear elastic models.

It must be checked that, at the adhesive-concrete interface, both for the characteristic (or rare) and for the frequent load combination, the "equivalent" shear stress, defined below, is lower than the adhesion resistance between the reinforcement and the substrate of concrete, f bd:

$$\tau_{b,e} \leq f_{bd}$$

The "equivalent" shear stress tb, e can be defined starting from the mean shear stress tm, evaluated at the chord on which the adhesive and concrete interface:

$$\tau_{b,e} = k_{id} \cdot \tau_m$$

where:

-k_{id} is a coefficient (≥ 1) which takes into account the concentration of shear and normal stresses in the terminal zones:

$$k_{id} = \left(k_\sigma^{1.5} + 1.15 \cdot k_\tau^{1.5}\right)^{2/3}$$

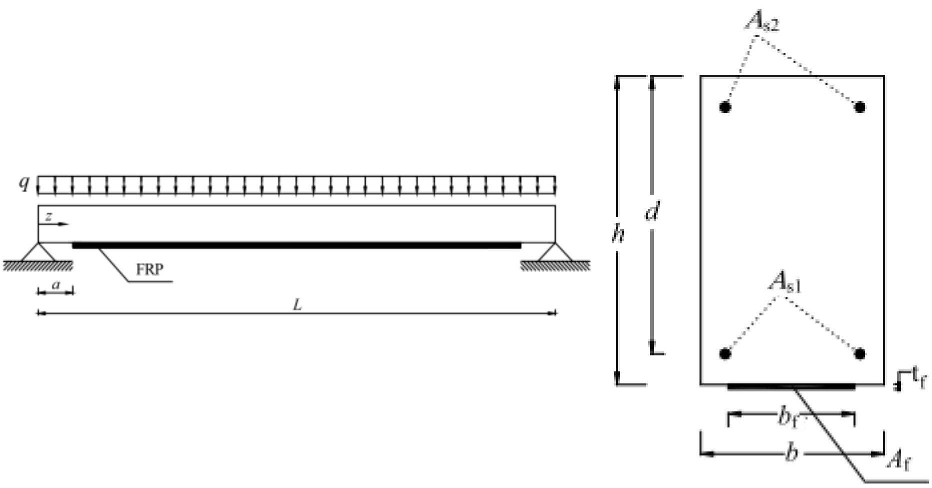
- the coefficients kσ and kτ are respectively valid:

$$k_\sigma = k_\tau \cdot \beta \cdot t_f$$

$$k_\tau = 1 + \alpha \cdot a \cdot \frac{M_{(z=a)}}{V_{(z=a)} \cdot a}$$

- M (z = a) is the moment acting in the interruption section of the reinforcement;

- V (z = a) is the shear acting in the interruption section of the reinforcement, located at a distance z = a from the end of the beam;



- α and β are two elastic constants dependent on the characteristics of the interface and of the FRP reinforcement:

$$\alpha = \sqrt{\frac{K_1}{E_f \cdot t_f}}$$

$$\beta = \left(\frac{b_f \cdot 2.30 \cdot K_1}{4 \cdot E_f \cdot I_f} \right)^{1/4}$$

being E_f , t_f , b_f , I_f and K_1 respectively, the modulus of normal elasticity, the thickness of the FRP reinforcement, its length, the competent moment of inertia (with respect to its own barycentric axis parallel to the length dimension bf) and the angular coefficient of the increasing linear branch of the adhesion bond, assumed to be equal to:

$$K_1 = \frac{1}{t_a/G_a + t_c/G_c}$$

where moreover, respectively, Ga and Gc are the modulus of tangential elasticity of the adhesive and of the concrete, ta is the nominal thickness of the adhesive and tc the effective thickness of the concrete participating in the deformability of the interface (in general it can be assumed tc = 20 ÷ 30 mm);

- tm is the average Jourawski shear stress:

$$\tau_m = \frac{V_{(z=a)} \cdot t_f \cdot (h - x_e)}{I_c/n_f}$$

- x_e and I_c are, respectively, the distance of the neutral axis from the extreme compressed edge and the moment of inertia of the homogenized section, possibly partialized if in the presence of cracking (NOTE: the program considers the cracked section when the moment acting for the operating state considered M (z = a) is higher than the moment of first cracking Mcr);

- nf = Ef / Ec is the homogenization coefficient (with Ec the normal modulus of elasticity of the concrete corresponding to the load combination considered, rare or frequent).

In the presence of a terminal anchorage, made by "U" bandage, the effect of normal tensions for the purposes of verifying the interface can be neglected and, therefore, the coefficient k_σ can be assumed to be equal to zero.

The design strength of the adhesion between reinforcement and concrete, f_{bd}, is a function of the characteristic tensile strength of the concrete, fctm, and is provided by the relationship:

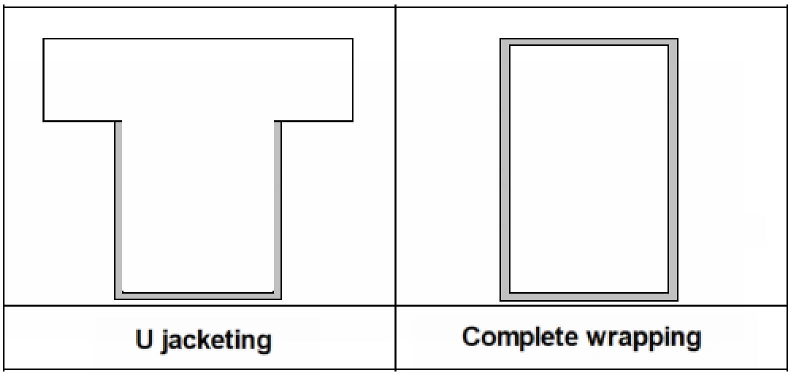
$$f_{bd} = 0.21 \cdot \frac{k_b}{\gamma_b} \cdot \frac{f_{ctm}}{FC}$$

where the partial factor γ_b is 1.0 for the rare load combination, 1.2 for the frequent load combination.

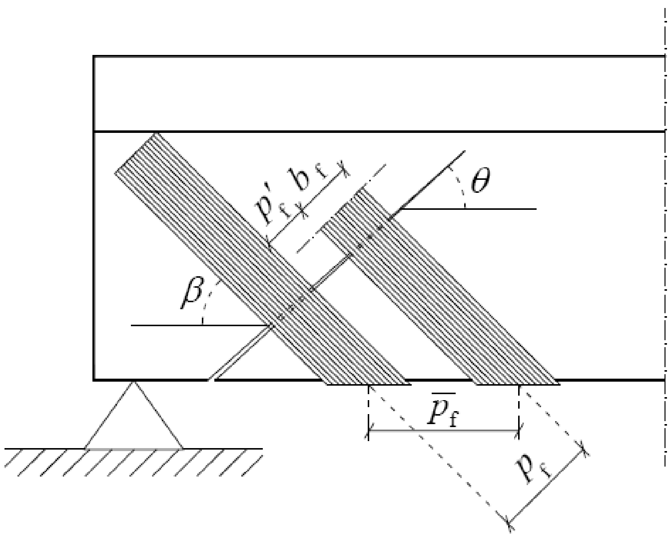
In the calculation of the anchoring stresses, in service conditions (SLS), it is possible to refer to the state of stress corresponding to the load increase that occurs after the application of the reinforcement.

11.3.1.2.6.2 Transverse reinforcements

The resistant shear of the reinforced element is given by the expression (CNR DT200 R1 2013 - (4.18)):
Vrd = min {VRd, s + VRd, f, VRd, c}
where VRd, s and VRd, c are, respectively, the contributions of the transverse reinforcement and of the compressed concrete connecting rod, calculated in accordance with current legislation (NTC 18). The contribution VRd, f, on the other hand, is the increase in resistance brought about by the reinforcement. In the event that VRd, c <VRd, s, therefore, the application of the reinforcement cannot produce increases in resistance.
The contribution VRd, f depends, in addition to the characteristics of the materials, on the type of arrangement of the reinforcement which may be of the type (in order of efficiency):
-"U jacketing";
-"complete wrapping".



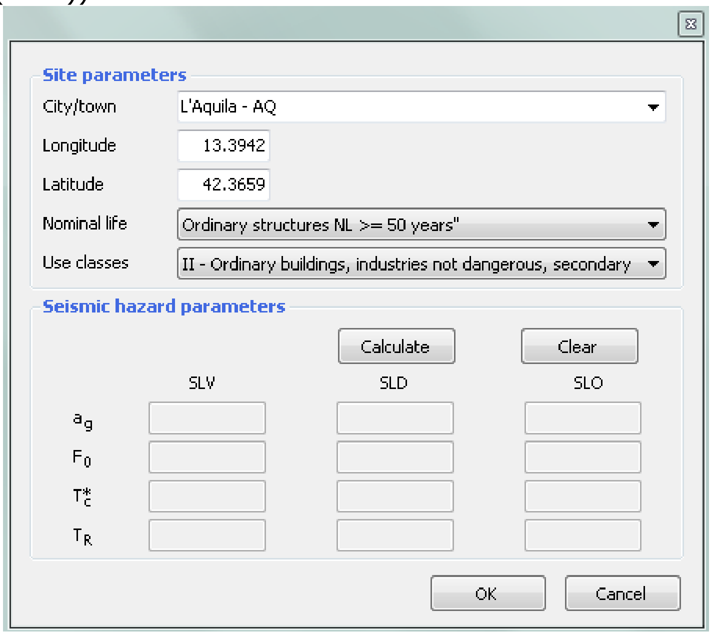
In the U-jacketing reinforcement it is possible to improve the constraint conditions of the free ends of the fabrics, by applying bars, sheets or strips of composite material in these areas. In this case, if the effectiveness of the constraint offered by the aforementioned devices is demonstrated, the behavior of the U-jacketing reinforcement can be considered equivalent to that of the winding reinforcement.
Other factors influencing the contribution VRd, f are the arrangement along the element (continuous or in bands) and the angle of inclination of the fibers with respect to the horizontal axis.



The angle of inclination of the cutting slots θ is to be set equal to 45 °. However, this is in contrast to what is indicated in the NTC 18 where, for the calculation of the shear

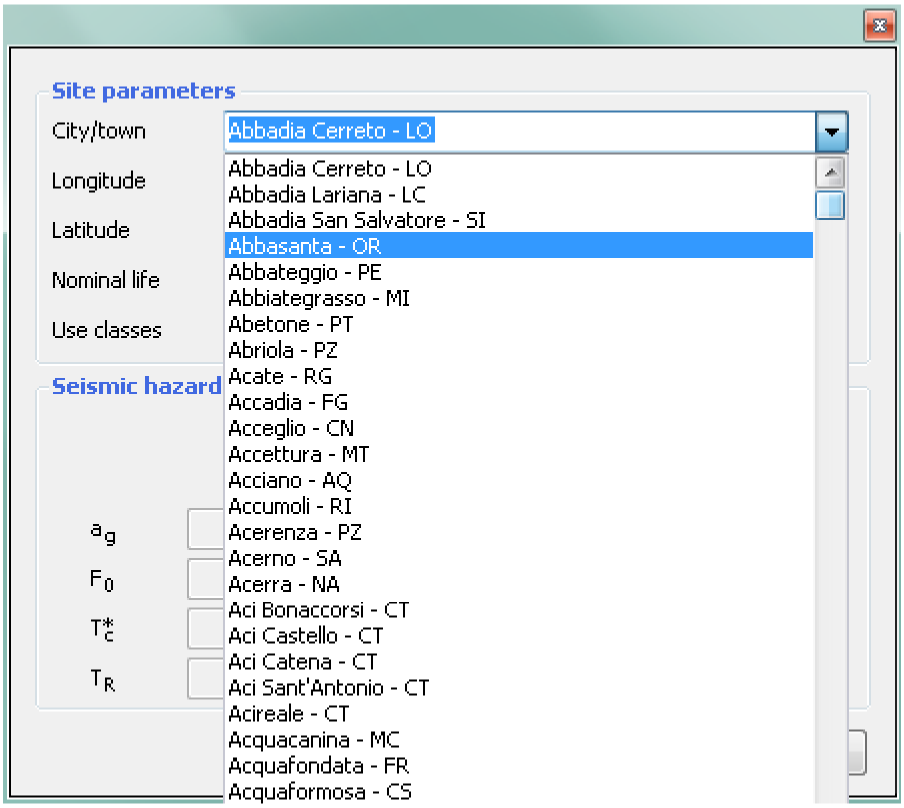
strength, the use of the "variable θ" method is required provided that 1 <= cotg θ <= 2.5. The program therefore leaves the possibility for the user to use the calculation method he deems appropriate.
The contribution of the FRP reinforcement is calculated according to the formula 4.19 of CNR DT200 R1 2013

where:
- d is the useful height of the section;
- tf, bf, pf are the geometric characteristics of the strips;
- ffed is the effective resistance of the reinforcement system, calculated as follows (CNR DT200 R1 2013 - (4.21))



In the case of discontinuous reinforcement systems, the strips of composite material must respect the following limitations:
50 mm ≤ bf ≤ 250 mm
bf ≤ pf ≤ min {0,5 d; 3 bf; bf + 200 mm}

In the case of seismic calculation, the calculation of the shear resistance of the section is carried out according to the indications of paragraph C8.7.2.3.5 of the Circ. 21 January 2019 n. 7 relating to NTC 2018.
The contribution of the FRP reinforcement must be added to the contribution of the stirrups Vw in the equation C8.7.2.8:



Confinement

An adequate confinement of the elements of reinforced concrete can lead to an improvement in the performance of the structural element allowing to increase:

-the ultimate resistance and the corresponding ultimate deformation of elements stressed by centered normal stress or with small eccentricity;

-the ductility and, together with the use of longitudinal reinforcements, the ultimate resistance of elements under bending.

The confinement of elements of reinforced concrete it can be made with FRP fabrics or sheets arranged on the contour so as to form a continuous or discontinuous external bandage.

The increase in compressive strength and the corresponding ultimate deformation of the concrete confined with FRP depend on the applied confinement pressure. The latter is a function of the stiffness of the reinforcement system and the shape of the cross section of the confined element.

In the case of a confined element, its design resistance at centered compression (or with small eccentricity), can be defined as follows. The confined element is verified if the inequality is satisfied:

$$N_{Sd} \leq N_{Rcc,d}$$

where:

-N_{Sd} is the design value of the acting axial action;

-$N_{Rcc,d}$, d is the design value of the resistance of the confined element.

In the absence of instability phenomena, the design resistance NRcc, d is given by the following relation

$$N_{Rcc,d} = \frac{1}{\gamma_{Rd}} \cdot A_c \cdot f_{ccd} + A_s \cdot f_{yd}$$

where:

-γ_{Rd} is the partial coefficient to be assumed equal to 1.10;

-A_C and f_{ccd} are, respectively, the cross-sectional area of the element and the design stress of the confined concrete;

-A_S and f_{ccd} are, respectively, the area and the design voltage of the metal armature present.

The design resistance of confined concrete, fccd, can be evaluated as follows:

$$\frac{f_{ccd}}{f_{cd}} = 1 + 2.6 \left(\frac{f_{1,eff}}{f_{cd}}\right)^{2/3}$$

where:

-f_{cd} is the design stress of unconfined concrete [CNR-DT 200 R1 / 2013 § 3.3.3 (6)];

-$f_{1,eff}$ is the effective confinement pressure.

This effective confinement pressure, depending on the shape of the section and the methods of carrying out the intervention, is provided by the report:

$$f_{1,eff} = k_{eff} \cdot f_1$$

where:

$$f_1 = \frac{1}{2} \cdot \rho_f \cdot E_f \cdot \varepsilon_{fd,rid}$$

where:

-ρ_f is the geometric percentage of reinforcement, Ef is the normal modulus of elasticity of the material in the direction of the fibers, εfd, rid is the reduced deformation of the reinforcement calculation;

-k_{eff} is the efficiency coefficient defined on the standard (CNR-DT 200 R1 / 2013 § 4.5.2.1 (4.34)).

In the calculation, this confinement is effective if the following inequality is verified:

$$f_{1,eff} / f_{cd} > 0.05$$

11.3.1.2.6.3 Jacketing

As reported in C8.7.4.2.1 of Circular 7 of 21 January 2019, the pillars can be fitted with reinforced concrete jacketing to achieve all or some of the following:

- increased vertical bearing capacity;
- increase in bending and / or shear strength;

- increase in deformation capacity;
- improvement of the efficiency of the overlap joints.

The thickness of the "jackets" must be such as to allow the positioning of longitudinal and transversal reinforcements and the creation of an adequate thickness of the concrete cover.

For the purposes of evaluating the strength and deformability of jacketed elements, the following simplification hypotheses are acceptable:
- the jacketed element behaves monolithically, with full adherence between the old and the new concrete;
- the axial load is considered applied only to the pre-existing portion of the element for permanent loads only, to the entire jacketed section for variable loads and for seismic actions;
- the mechanical properties of the concrete of the jacket are considered extended to the entire section if the differences between the two materials are not excessive.

The capacity values to be adopted in the verifications are those calculated with reference to the entire jacketed section in the simplified hypotheses indicated above reduced according to the following expressions:

- capacity in terms of shear strength:

$$\tilde{V}_R = 0.9V_R$$

- capacity in terms of flexural strength:

$$\tilde{M}_y = 0.9M_y$$

- capacity in terms of deformability at yield:

$$\tilde{\theta}_y = 0.9\theta_y$$

- capacity in terms of ultimate deformability:

$$\tilde{\theta}_u = 0.9\theta_u$$

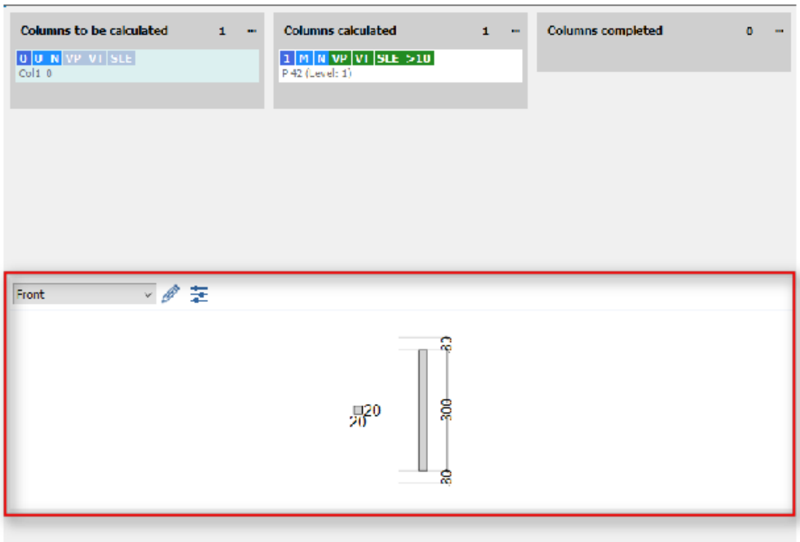
The values to be used for the resistances of the materials are:
a) for the steel of existing structures, the strength obtained as the average between the tests carried out on site and what was obtained from additional sources of information, divided by the appropriate confidence factor in relation to the Level of Knowledge achieved and, only in the calculation of the capacity in terms of shear strength, also divided by the partial coefficient;
b) for the added materials, concrete and steel, the design resistance.

The values to be used for the resistances of the materials in the calculation of the value of the capacity in terms of flexural strength to be used for the evaluation of the shear stress acting on fragile elements / mechanisms are:
c) for the steel of existing structures, the strength obtained as an average between the tests performed on site and from additional sources of information multiplied by the appropriate confidence factor in relation to the level of knowledge achieved;
d) for the added materials, concrete and steel, the characteristic value of the resistance.

11.3.1.3 Work environment

11.3.1.3.1 Front interaction

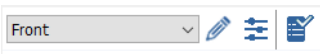
The "Front" view allows you to interact graphically with the elements using the commands in the top bar.



The command bar has several possible configurations depending on the position of the calculation card within the columns.

11.3.1.3.1.1 Front configuration

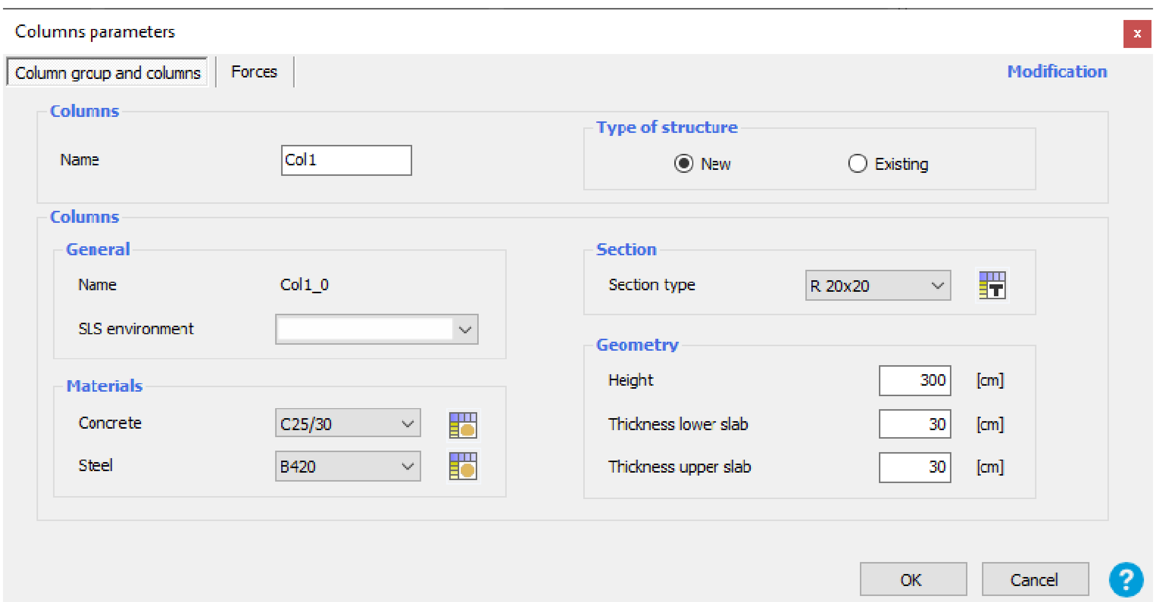
"Front" configuration



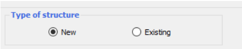
It allows to modify geometries, stresses and calculation parameters.

- "Modification": allows you to modify the selected column;
- "Calculation parameters": allows you to set / modify the calculation parameters;
- "Results": allows you to view the results of the calculation.

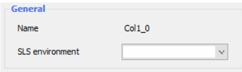
The screen is as follows:



Specifically, the sections are:

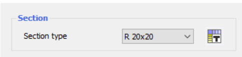


Type of structure
It allows the user to choose the type of structure to design or verify, with the possibility of choosing between new and existing.

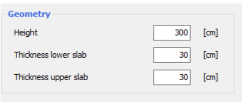


General
Contains the name assigned to the column and the type of SLS environment

Materials
Assignment of the type of concrete and steel, with the possibility of choosing by means of the lists or by accessing the material library.

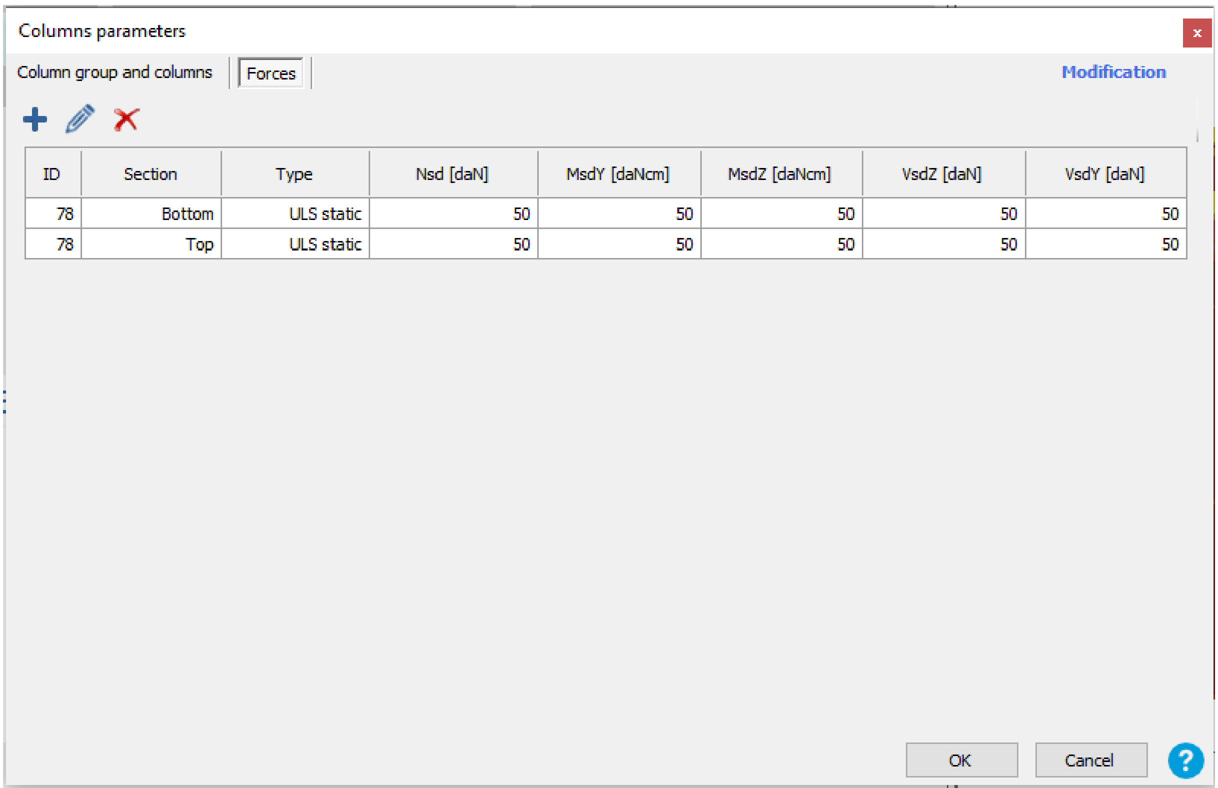


Section
Assignment of the column cross section, with the possibility of choosing a section that has already been used or by accessing the section creation window.

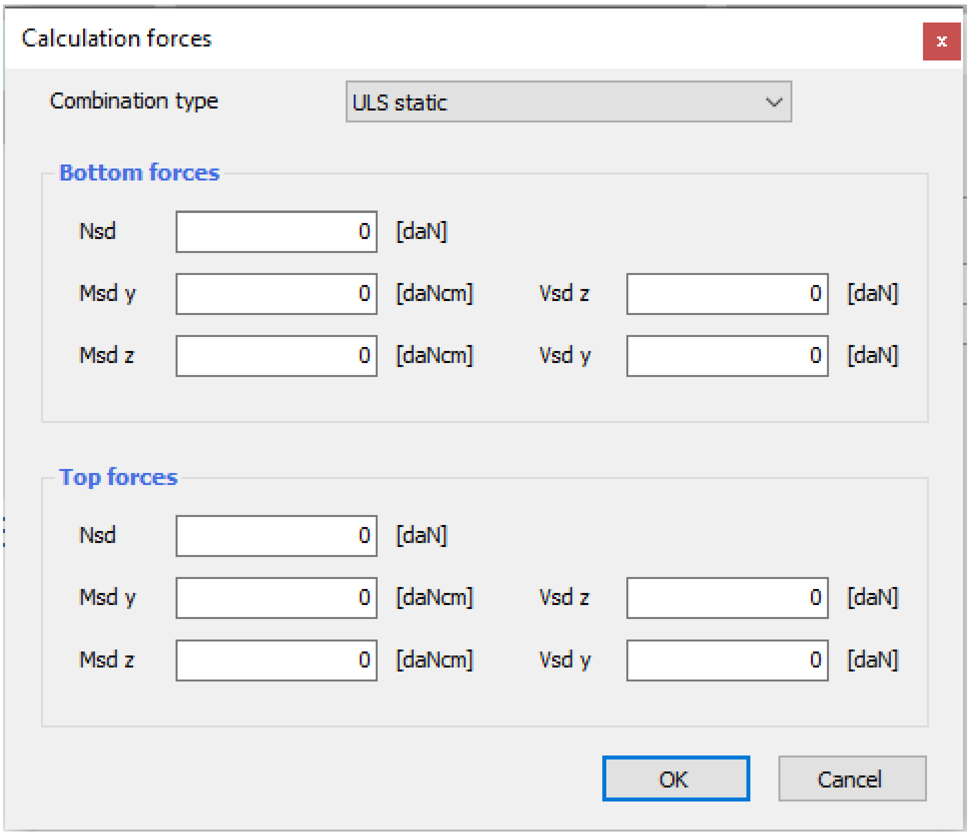


Geometry
It requires the insertion of geometric data regarding the height of the pillar and the floors above and below it.

The screen is as follows:



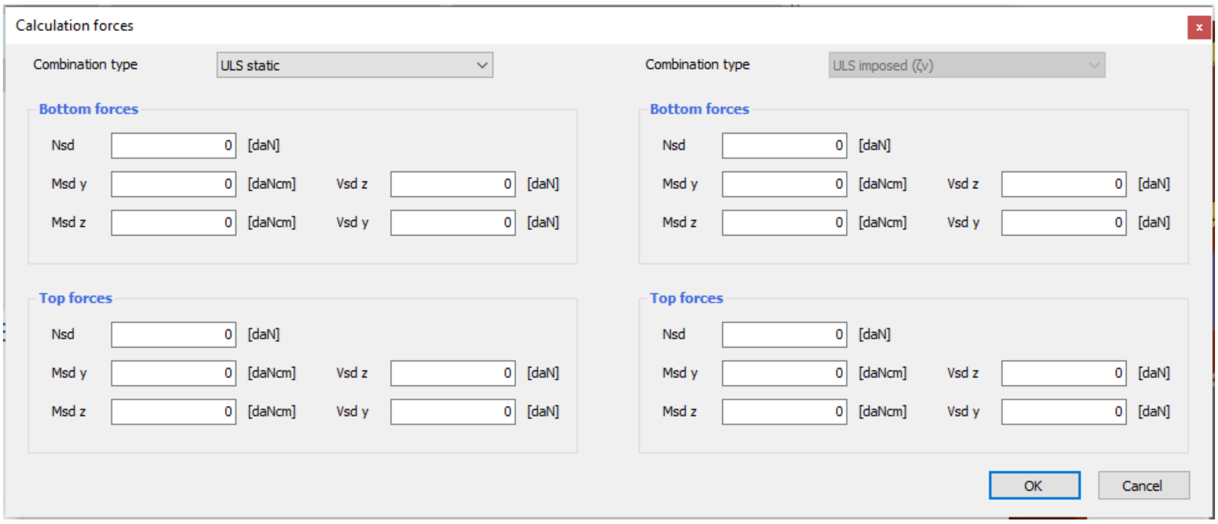
Specifically, the table allows you to insert, modify or delete (using the buttons) combinations of stresses.
In particular, adding a combination will open its definition menu:



The menu requires entering the type of combination and the section on which it acts (base / head).
The stresses to be inserted are:

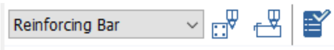
Nsd: design value of the normal effort;
Msd, y: design value of the bending moment that rotates around the local y axis;
Msd, z: design value of the bending moment that rotates around the local z axis;
Vsd, z: design value of the shear acting in the local z direction;
Vsd, y: design value of the shear acting in the local y direction.

In the case of an existing structure, in which the calculation of the parameter zv is required, a relative part of variable overloads will be associated to each static combination on the basis of which the analysis will be carried out. The combinations will be loaded together in the table and will be managed as a single entity, in that case connected to each other.



11.3.1.3.1.2 Reinforcing bar configuration

"Reinforcing bar" configuration

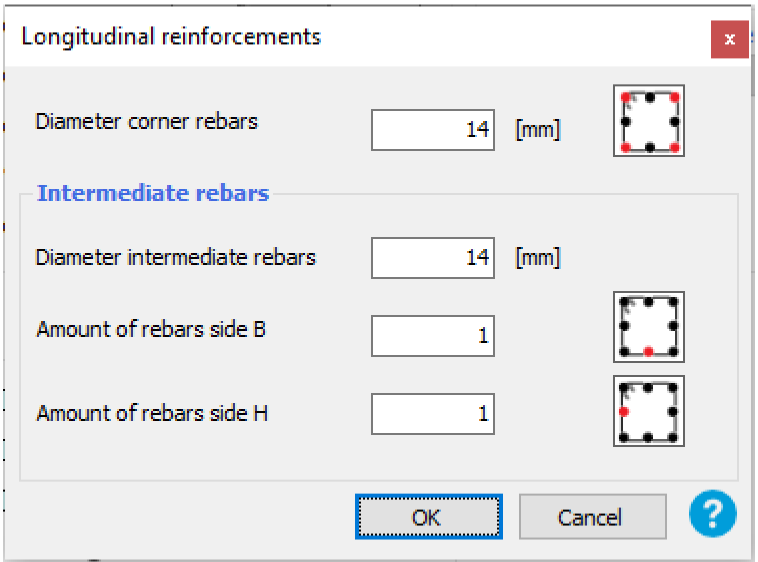


It allows you to insert / modify the reinforcement and view the calculation results.

Edit/insert stirrups: to change the diameter or spacing of the stirrups on the elevation, simply click on them.
Edit / insert longitudinal reinforcement: to modify the diameter or number of longitudinal bars.

Longitudinal reinforcement

The command requires setting the quantity of the bars, along the respective z and y directions, and the diameter of the wall and corner bars through the following menu.

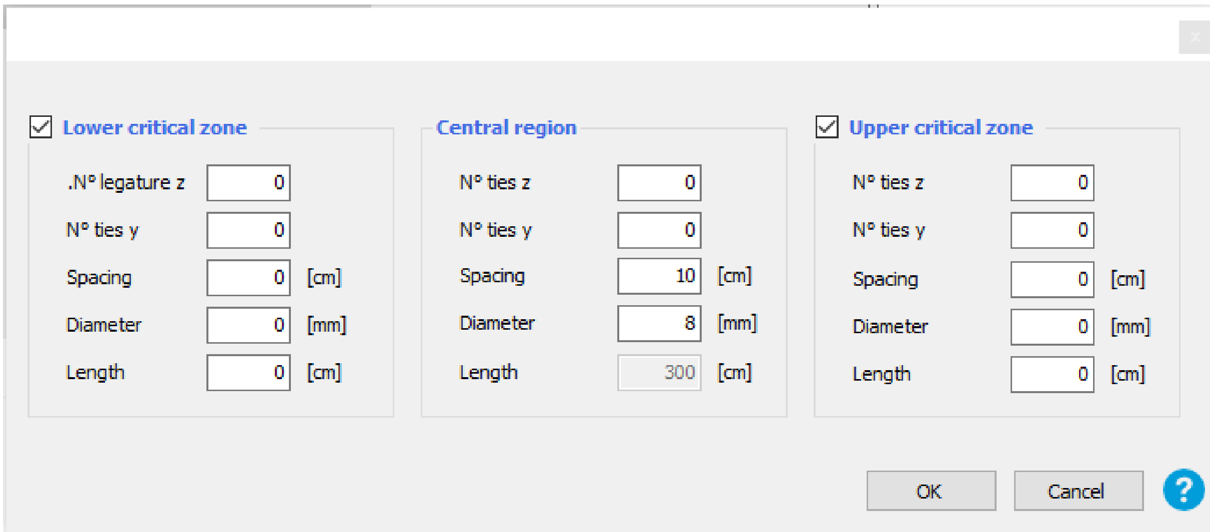


Attention: In the case of columns imported from the model, any changes made to the reinforcement will be confirmed and transmitted to the column in the Structure environment only after having placed the column in the "Completed Columns"

Transversal renforcement

The command requires setting the number of ties in the z and y direction, the spacing and the diameter of the stirrups through the menu that is shown below.

It is possible to choose whether to set the stirrups only in the central area or to insert them also in the lower / upper critical area, possibly also establishing the length of these critical sections.



Attention: In the case of columns imported from the model, any changes made to the reinforcement will be confirmed and transmitted to the column in the Structure environment only after having placed the column in the "Completed Columns"

11.3.1.3.1.3 Reinforcement configuration

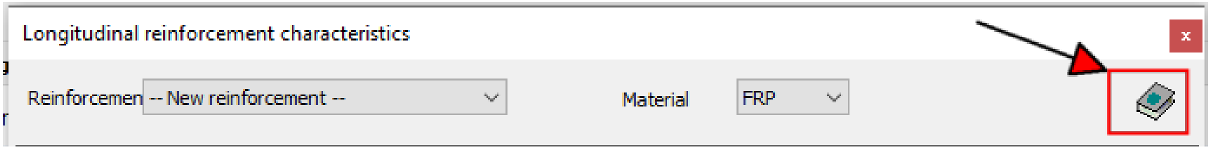
"Reinforcement" configuration

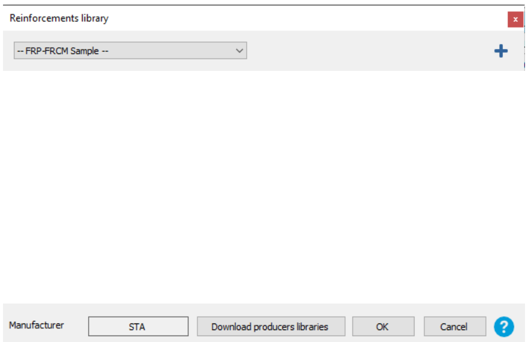
Allows you to insert / modify longitudinal and transverse reinforcements and view the calculation results.

"Results": allows you to view the results of the calculation
"Edit / insert jacketing"
"Delete jacketing/reinforcements"
"Insert longitudinal reinforcement"
"Edit longitudinal reinforcement"
"Delete longitudinal reinforcement"
"Insert jacketing"
"Edit jacketing"

"Remove jacketing"

In the upper right part, a button allows you to access the library of possible reinforcements.





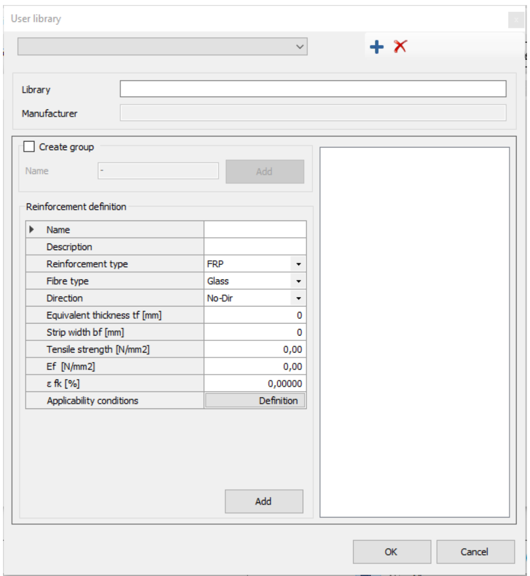
From the drop-down it is possible to view the libraries present and select the desired one.
By checking the "vertical" and / or "transversal" options, it is possible to choose the arrangement of the reinforcement.

Using the "+" symbol it is possible to create a new library to add to the list of those already present.
By clicking on the "Download manufacturer libraries" button you can consult and download the manufacturer libraries.

Once downloaded, the new library will be visible in the dropdown shown above.

If in the design practice we always refer to a specific manufacturer, it is possible to generate your own library that will be available for all future works.

To create a new user library it is necessary to define some parameters:

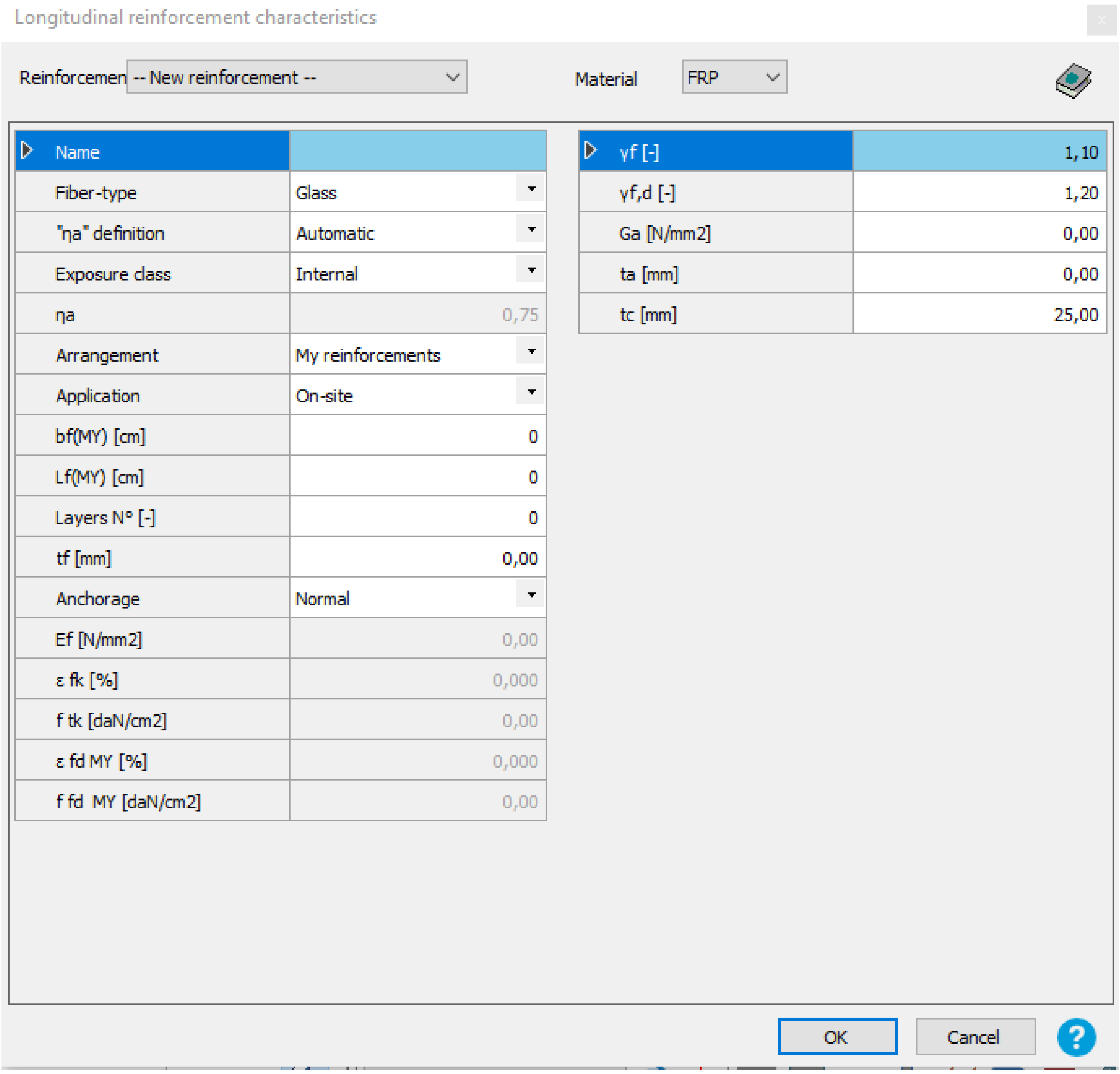


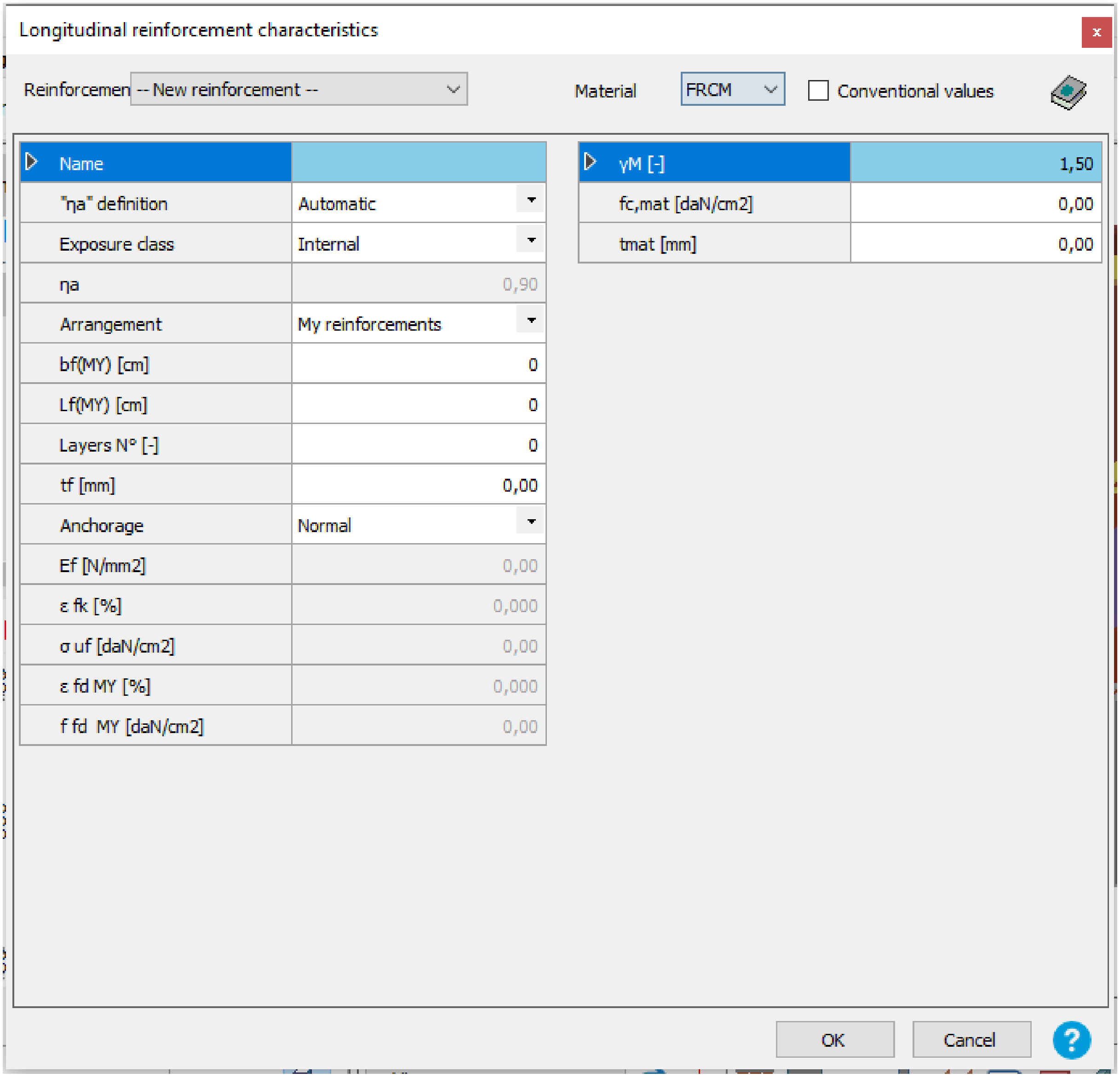
Some of these parameters are descriptive parameters (name, description), others are data necessary for defining the type of reinforcement, the type of fiber, the orientation (uni / bi-directional).

The following data (equivalent thickness, strip width, area, tensile strength, Ef and εfk) are all parameters available from the manufacturers' technical data sheets.

The addition of longitudinal reinforcements is carried out by selecting the parts of the column on which they are to be inserted.

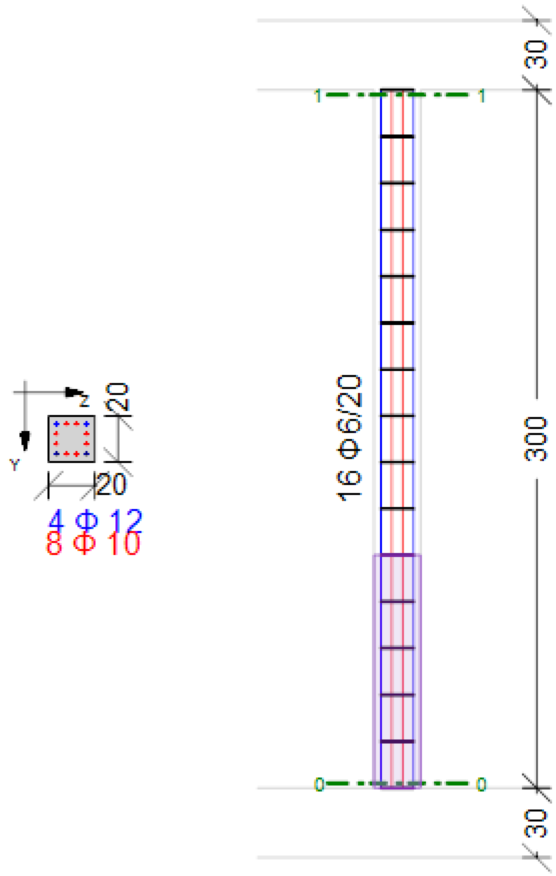
The command will prompt you to set the characteristics of the longitudinal reinforcement fabric: material, arrangement (My, Mz or My and Mz), anchoring (normal or mechanical), No. of layers and geometry for the selected arrangement.
To be able to do this, it is necessary to enter the parameters of the reinforcement that is being inserted, in the following specific menu:



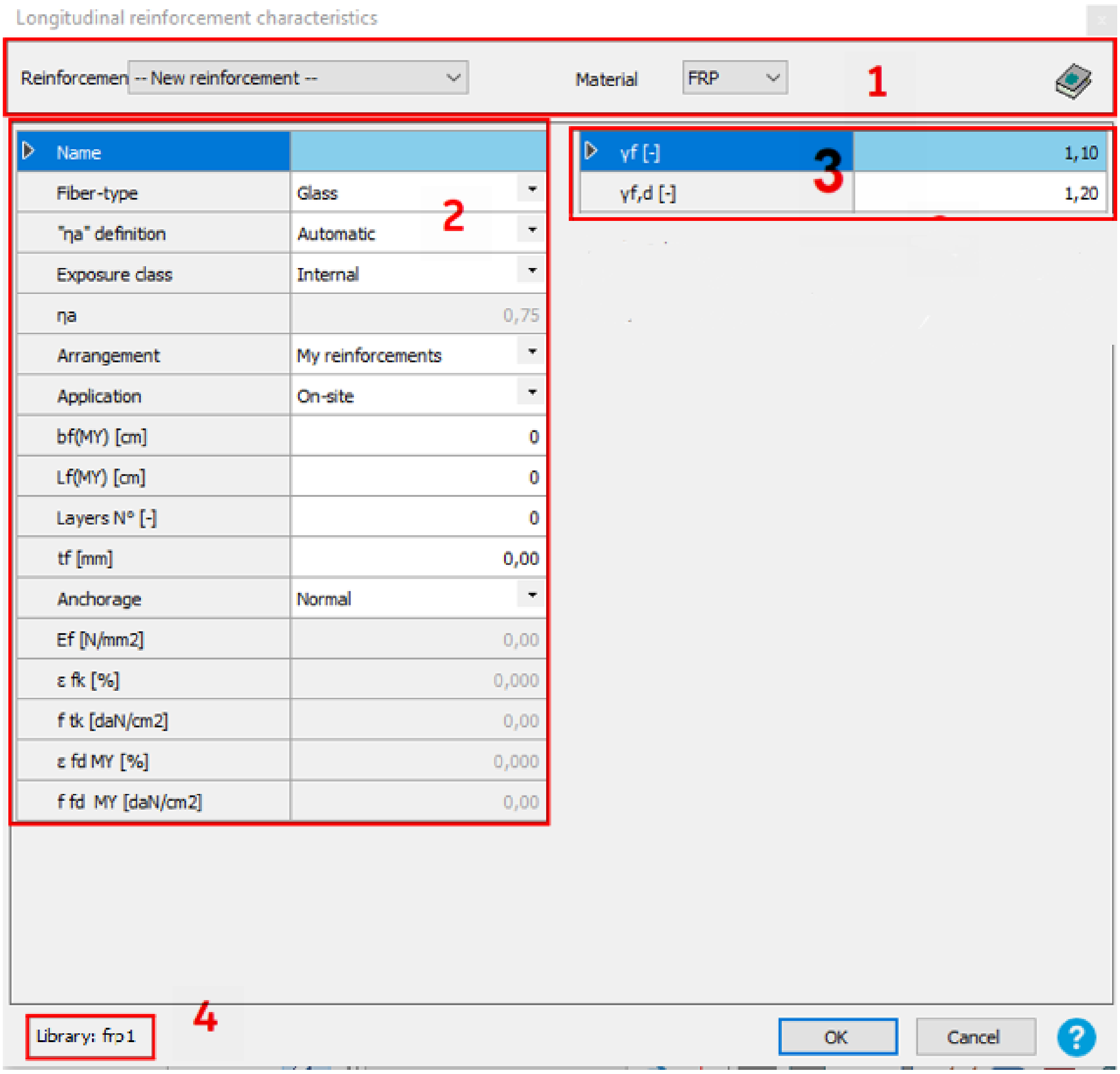


To complete the parameters in the menu illustrated above, it is necessary to refer to the menu of the reinforcement libraries. Through this menu it will be possible to insert a type of fabric from one of the libraries present, or create a new one containing the characteristics of the fabrics that will be applied to the structural element.

Once the modification of the reinforcement parameters has been completed, it will be sufficient to click on the desired parts of the column and confirm the insertion by right-clicking.



FRP reinforcement:

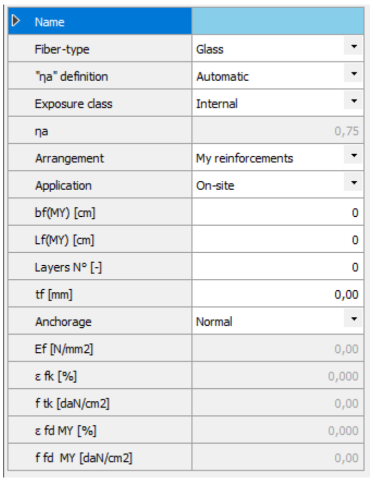


1

At the top of the previous menu it is possible to:

- insert a reinforcement previously inserted on the column from the list called "Reinforcement";
- select the type of material you intend to use (FRP / FRCM). Specifically, by leaving the FRP session active, all the input required below will comply with the current legislation for FRPs which is CNR-DT 200 R1 / 2013;
- choose the reinforcement fabric from the reinforcement libraries.

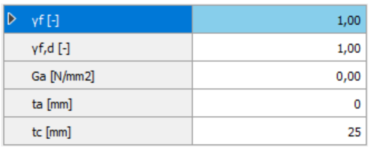
2



Name	
Fiber-type	Glass
"ηa" definition	Automatic
Exposure class	Internal
ηa	0,75
Arrangement	My reinforcements
Application	On-site
bf(MY) [cm]	0
Lf(MY) [cm]	0
Layers N° [-]	0
tf [mm]	0,00
Anchorage	Normal
Ef [N/mm2]	0,00
ε fk [%]	0,000
f tk [daN/cm2]	0,00
ε fd MY [%]	0,000
f fd MY [daN/cm2]	0,00

Shows the name of the reinforcement fabric inserted from the library
It reports the type of fiber-resin of the reinforcement inserted from the library
Contains the type of definition of ηa: Automatic / Manual
Exposure class
ηa value
Indicates the arrangement of the reinforcements: My / Mz / My and Mz
Contains the application of On site / Preformed reinforcement
Reinforcement width (MY)
Reinforcement Length (MY)
Number of layers of reinforcement
Thickness of the reinforcement
Type of Anchoring: Normal / Mechanical
Normal modulus of elasticity in the direction of the force
Characteristic deformation
Characteristic tension of the reinforcement
Fiber Calculation Strain at Break (MY)
Calculation resistance of the fiber (MY)

3

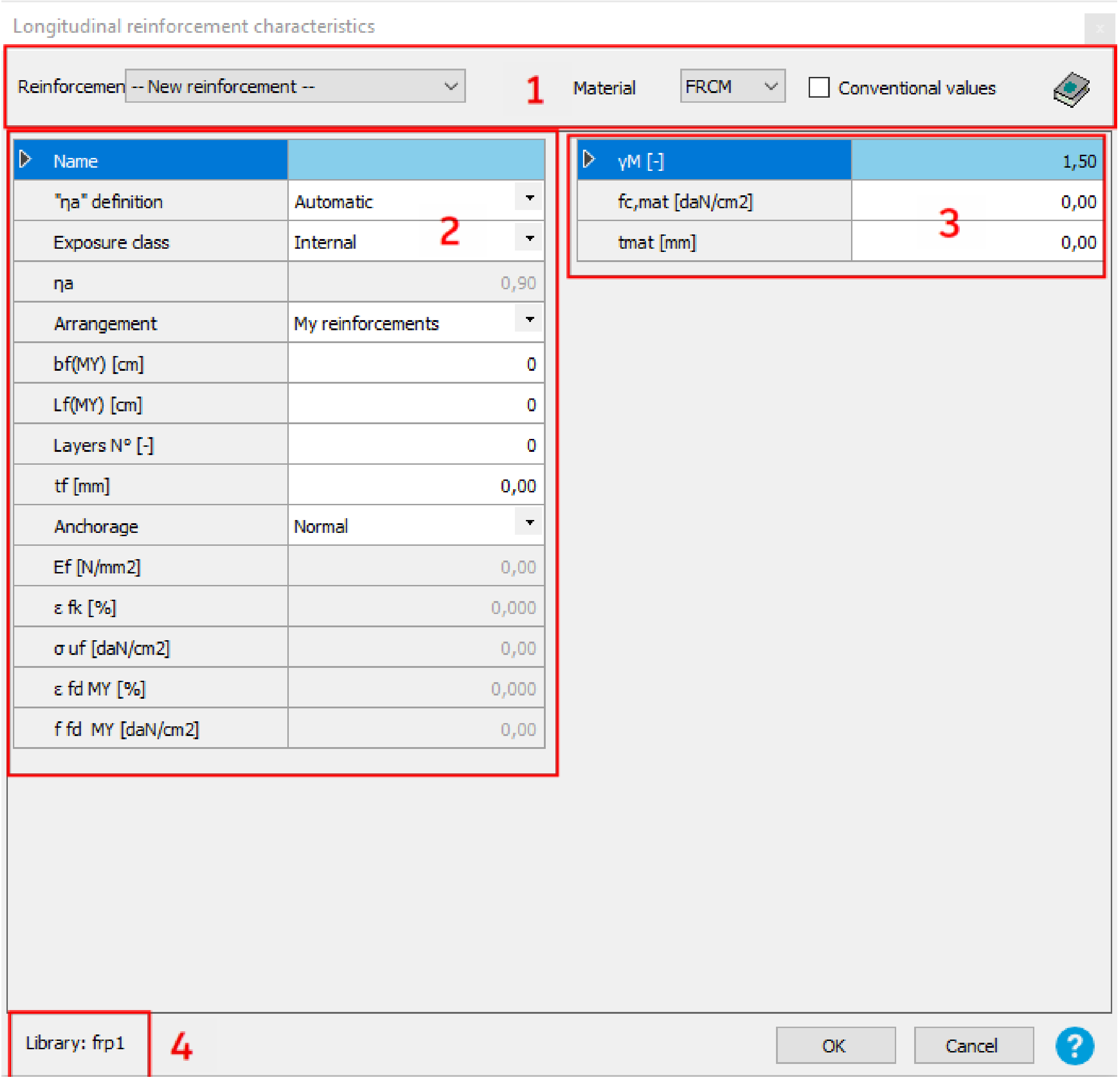


γf [-]	1,00
γf,d [-]	1,00
Ga [N/mm2]	0,00
ta [mm]	0
tc [mm]	25

Partial safety factor of the FRP [3.4.1 CNR-DT 200 R1 / 2013]
Safety factor for the detachment of the FRP [3.4.1 CNR-DT 200 R1 / 2013]
Tangential elastic modulus of the adhesive
Nominal thickness of adhesive
Effective thickness of concrete participating in the deformability of the interface

4
Indicates the library to which the type of fabric chosen belongs.

+ FRCM reinforcement:



1

At the top of the previous menu it is possible to:

- insert a reinforcement previously inserted on the girder from the list called "Reinforcement";
- select the type of material you intend to use (FRP / FRCM). Specifically, by leaving the FRCM session active, all the input required below will comply with the current legislation for FRCM which is CNR-DT 215/2018;
- choose the reinforcement fabric from the reinforcement libraries;
- choose whether to use conventional values for FRCM fabric.

The use of conventional values refers to what is written in the CNR-DT 215/2018.
In the chapter concerning the calculation of the mechanical characteristics (§3.1) of the aforementioned CNR, reference is made to the concept of "conventional" stresses and deformations which represents the strength of the reinforcement system obtained by detachment tests from conventional supports and as such is dependent on the media type.
The use of the conventional limit deformation and of the competent conventional limit tension, allows to design FRCM reinforcement interventions avoiding *the explicit verification in the comparison of the phenomenon of detachment from the support* or

sliding of the fibers in the matrix at the ends of the reinforcement, otherwise necessary in cases in which this crisis mode is possible.

2

Name	
"ηa" definition	Automatic
Exposure class	Internal
ηa	0,90
Arrangement	My reinforcements
bf(MY) [cm]	0
Lf(MY) [cm]	0
Layers N° [-]	0
tf [mm]	0,00
Anchorage	Normal
Ef [N/mm2]	0,00
ε fk [%]	0,000
σ uf [daN/cm2]	0,00
ε fd MY [%]	0,000
f fd MY [daN/cm2]	0,00

Shows the name of the reinforcement fabric inserted from the library
Contains the type of definition of ηa: Automatic / Manual
Exposure class
Value ηa
Indicates the arrangement of the reinforcements: My / Mz / My and Mz
Reinforcement Length (MY)
Reinforcement width (MY)
Number of layers of reinforcement
Thickness of the reinforcement
Type of Anchoring: Normal / Mechanical
Ultimate stress at break characteristic by traction of the dry fabric
Characteristic deformation
Ultimate stress at break characteristic by traction of the dry fabric
Fiber Calculation Strain at Break (MY)
Calculation resistance of the fiber (MY)

3

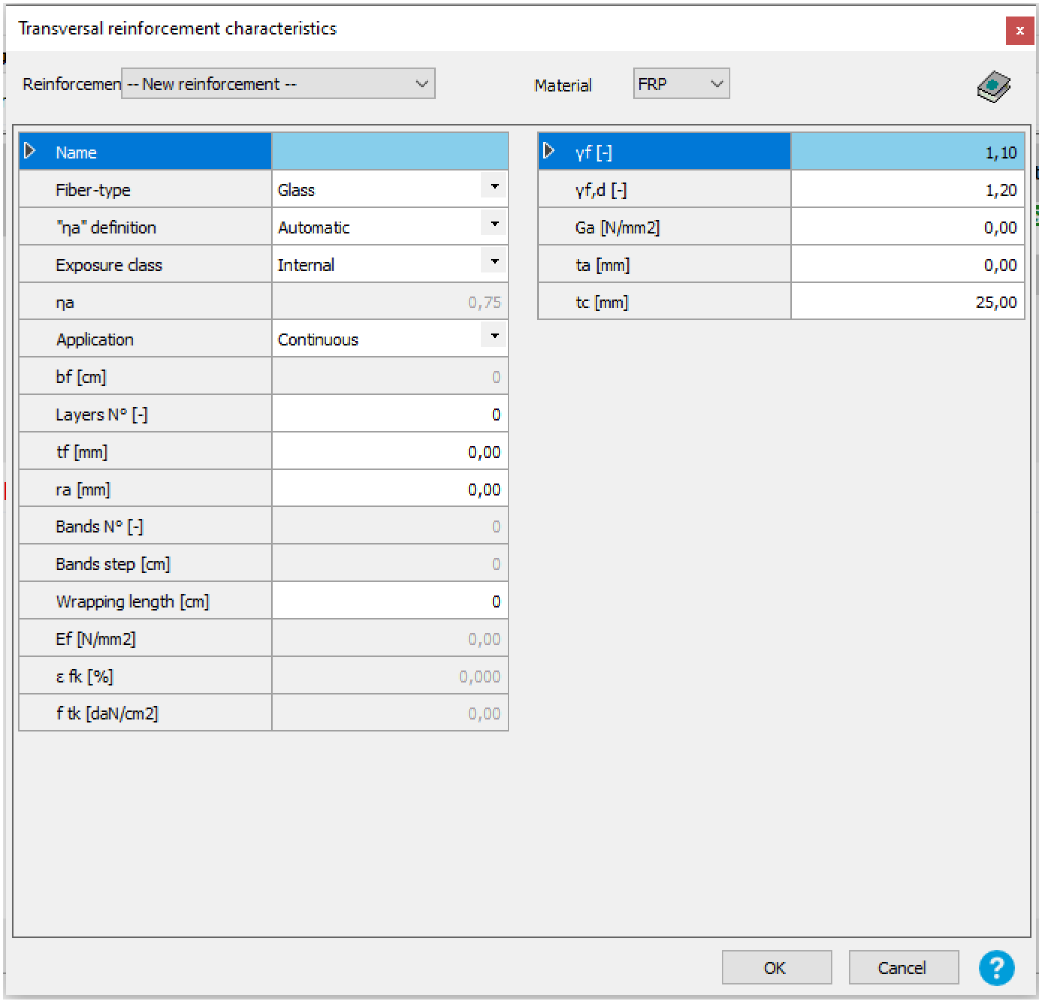
γM [-]	1,50
fc,mat [daN/cm2]	0,00
tmat [mm]	0,00

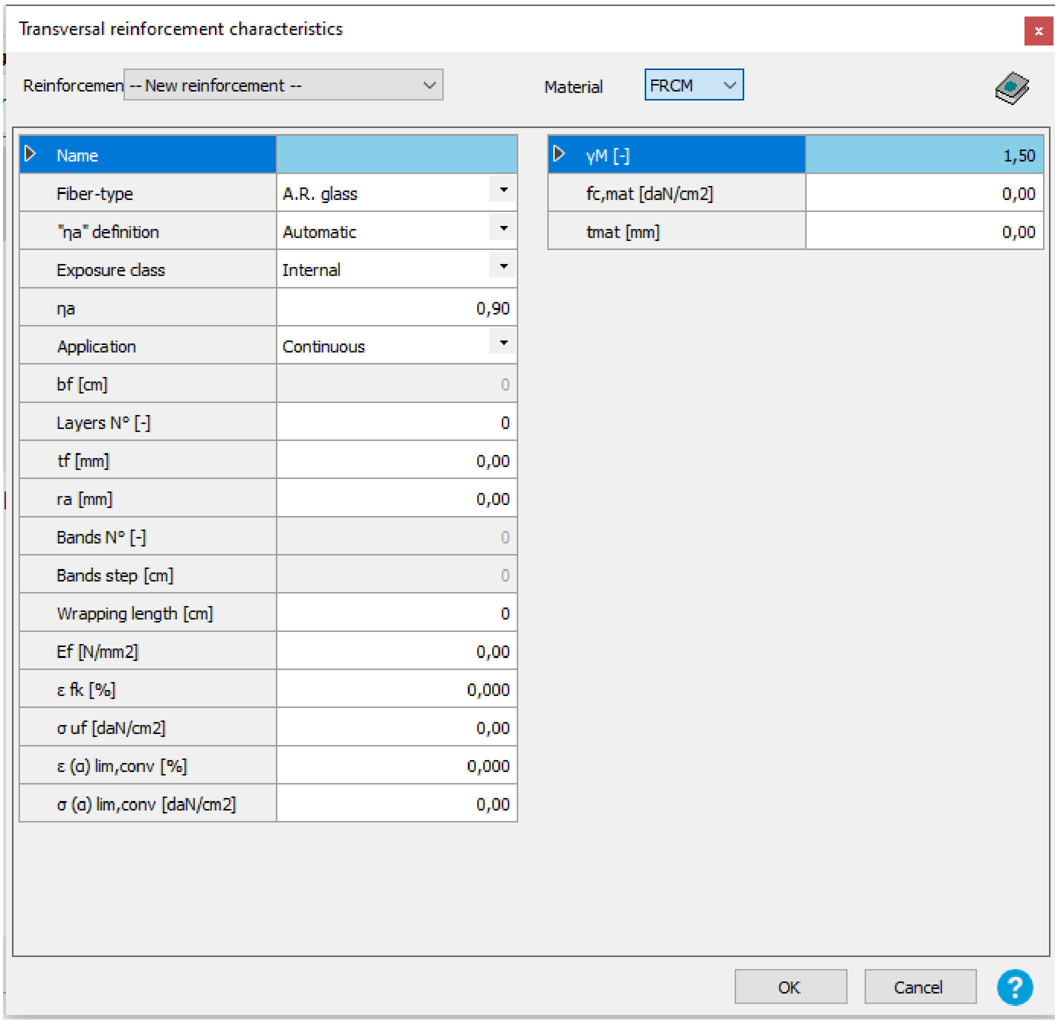
Partial safety coefficient of the FRCM [3.1 CNR-DT 215 2018]
Ultimate stress at break characteristic by traction of the dry fabric
FRCM single layer matrix thickness (1cm)

4
Indicates the library to which the type of fabric chosen belongs.

The addition of transverse reinforcements is carried out by selecting the parts of the column on which they are to be inserted.

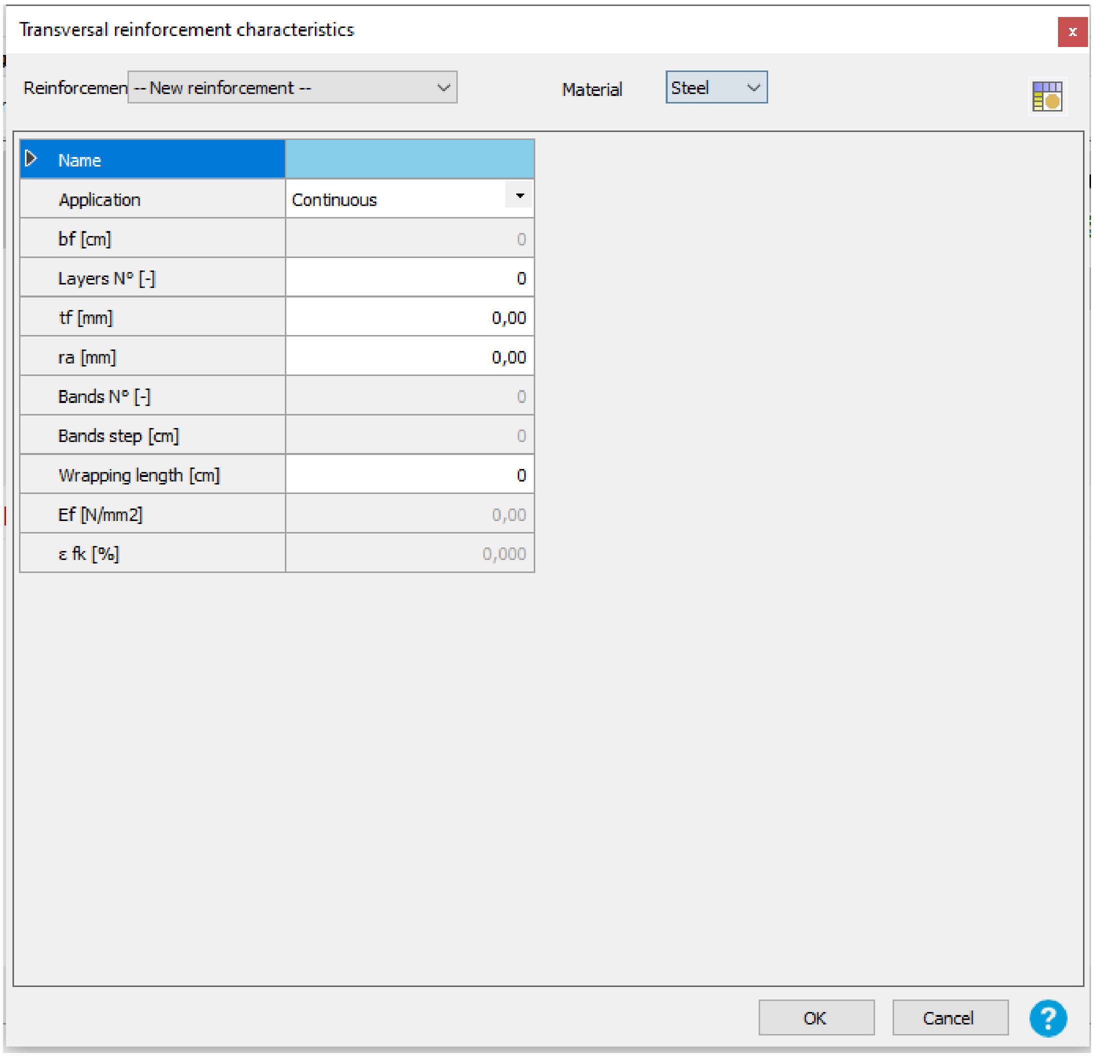
The command will prompt you to set the fabric characteristics of the transverse reinforcement: material, application, No. of layers and geometry for the selected arrangement.
To be able to do this, it is necessary to enter the parameters of the reinforcement that is being inserted, in the following specific menu:


Transversal reinforcement characteristics
Reinforcemen -- New reinforcement --
Material FRP
Name
Fiber-type Glass
"ηa" definition Automatic
Exposure class Internal
ηa 0,75
Application Continuous
bf [cm] 0
Layers N° [-] 0
tf [mm] 0,00
ra [mm] 0,00
Bands N° [-] 0
Bands step [cm] 0
Wrapping length [cm] 0
Ef [N/mm2] 0,00
ε fk [%] 0,000
f tk [daN/cm2] 0,00
γf [-] 1,10
γf,d [-] 1,20
Ga [N/mm2] 0,00
ta [mm] 0,00
tc [mm] 25,00
OK
Cancel



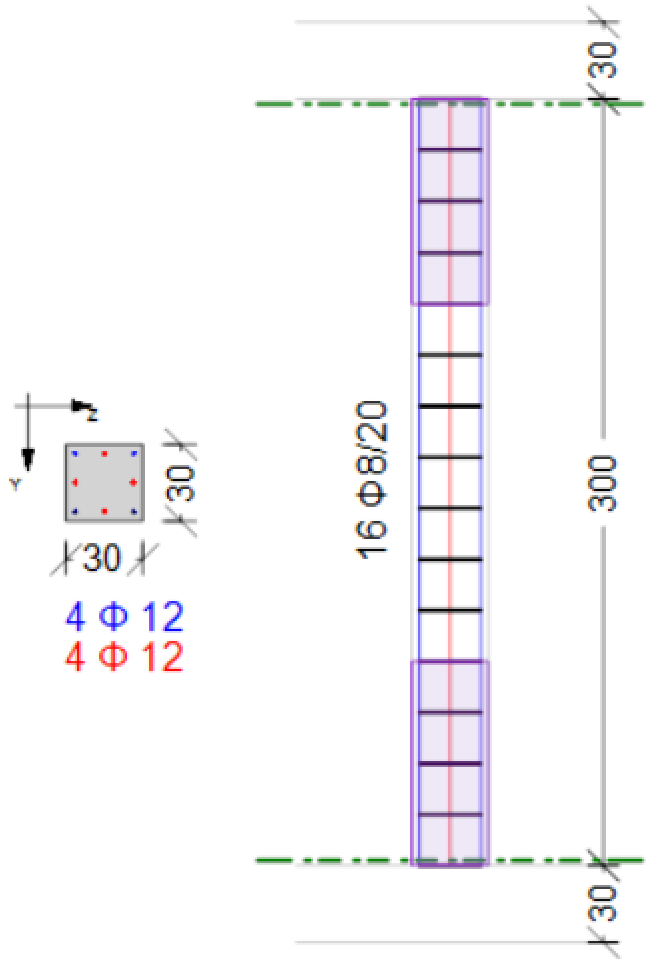
To complete the parameters in the menu illustrated above, it is necessary to refer to the reinforcement libraries menu. Through this menu it will be possible to insert a type of fabric from one of the libraries present, or create a new one containing the characteristics of the fabrics that will be applied to the structural element.

It is possible, always with the same insertion methods, to insert transversal steel reinforcements.
To be able to do this, it is necessary to enter the parameters of the reinforcement that is being inserted, in the following specific menu:

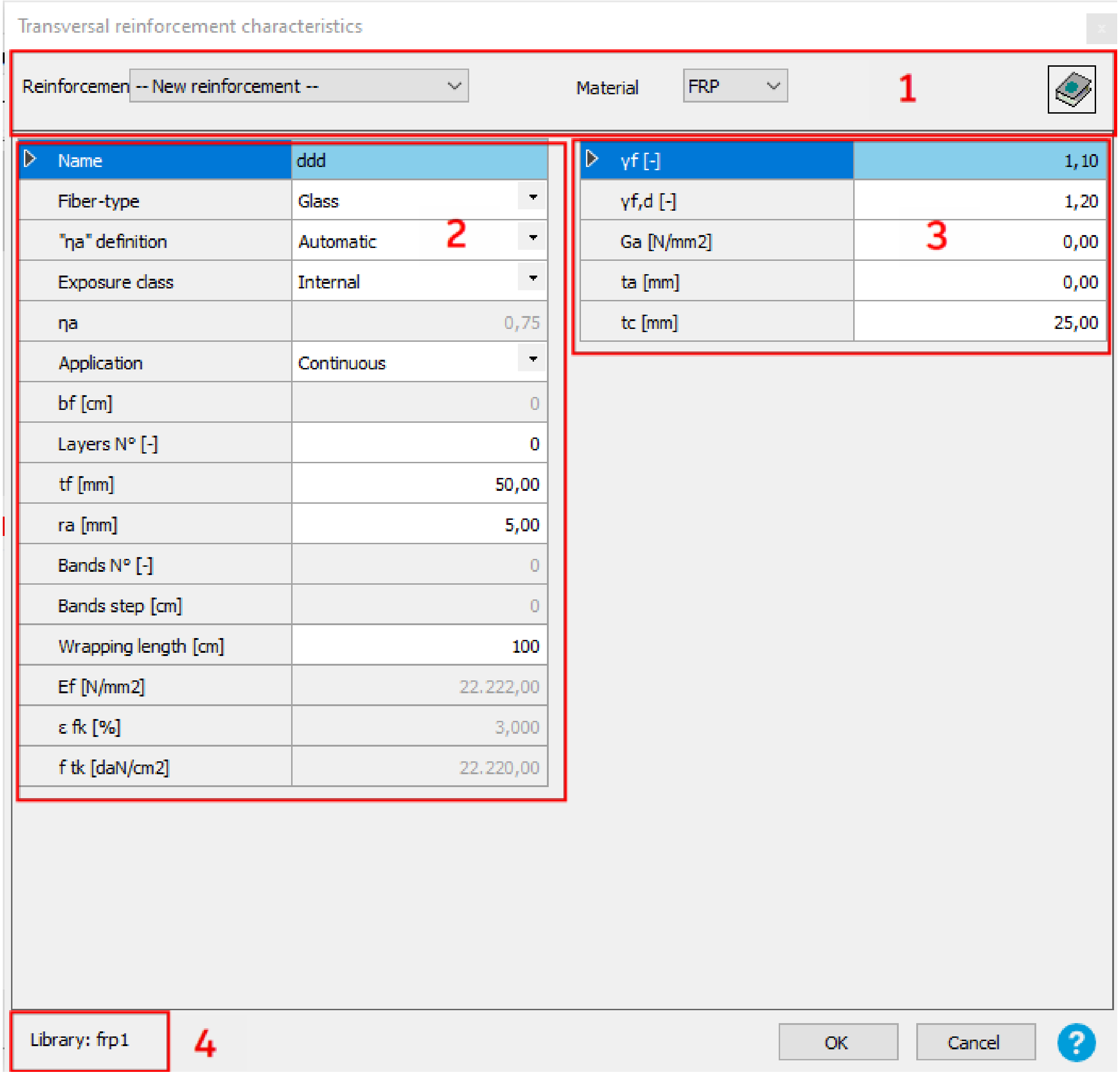


To complete the parameters it is necessary to select the steel material to be used, through the menu that opens with a click on the materials icon.

Once the modification of the reinforcement parameters has been completed, it will be sufficient to click on the desired parts of the column and confirm the insertion by right-clicking.



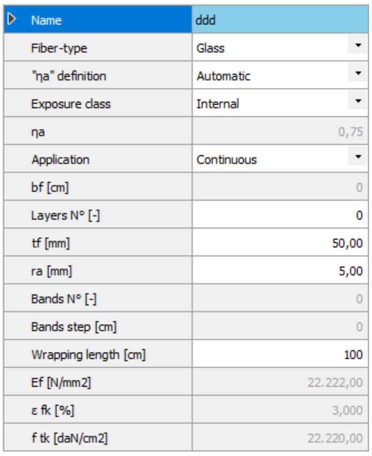
FRP reinforcement:



1
At the top of the previous menu it is possible to:

- insert from the list called "Reinforcement" a reinforcement previously inserted on the girder;
- select the type of material you intend to use (FRP / FRCM). Specifically, by leaving the FRP session active, all the input required below will comply with the current legislation for FRPs which is CNR-DT 200 R1 / 2013;
- choose the reinforcement fabric from the reinforcement libraries.

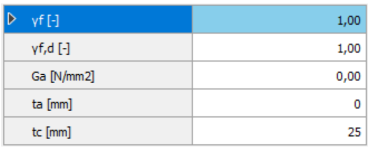
2



Name	ddd
Fiber-type	Glass
"ηa" definition	Automatic
Exposure class	Internal
ηa	0,75
Application	Continuous
bf [cm]	0
Layers N° [-]	0
tf [mm]	50,00
ra [mm]	5,00
Bands N° [-]	0
Bands step [cm]	0
Wrapping length [cm]	100
Ef [N/mm2]	22.222,00
ε fk [%]	3,000
f tk [daN/cm2]	22.220,00

Shows the name of the reinforcement fabric inserted from the library
Reports the fiber type of the reinforcement inserted from the library
na definition (Automatic/Manual)
Exposure class (Internal/External/Aggressive)
na value
Application (Continuous/Discontinuous)
Width of the reinforcement
Number of layers of reinforcement
Thickness of the reinforcement
Rounding radius
Number of bands
Step bands
Wrapping length
Elastic modulus in the direction of the reinforcement
Characteristic deformation of the reinforcement
Characteristic tension of the reinforcement

3

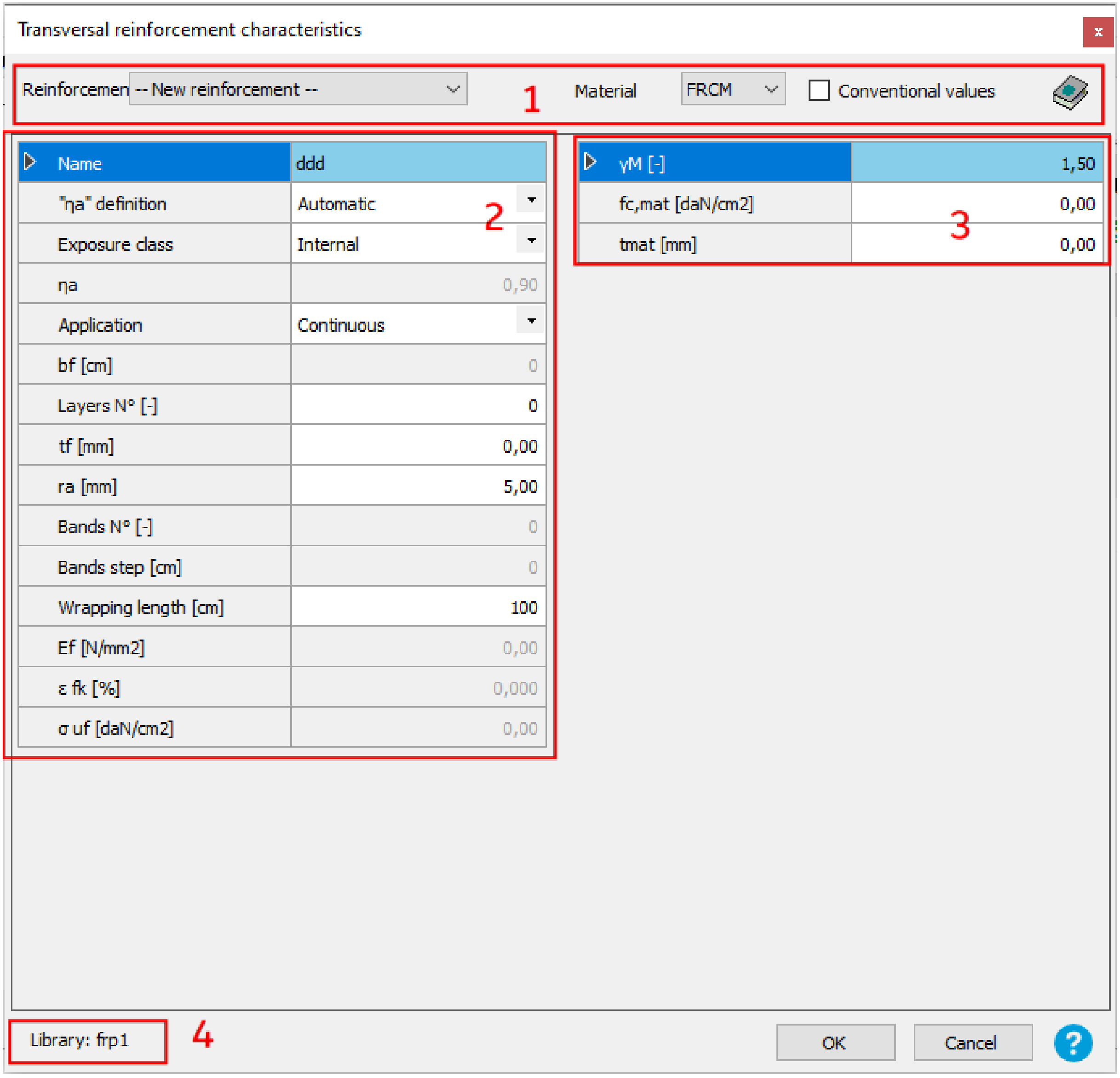


γf [-]	1,00
γf,d [-]	1,00
Ga [N/mm2]	0,00
ta [mm]	0
tc [mm]	25

Partial safety factor of the FRP [3.4.1 CNR-DT 200 R1 / 2013]
Safety factor for the detachment of the FRP [3.4.1 CNR-DT 200 R1 / 2013]
Tangential elastic modulus of the adhesive
Nominal thickness of adhesive
Effective thickness of concrete participating in the deformability of the interface

4
Indicates the library to which the type of fabric chosen belongs.

FRCM reinforcement:



1

At the top of the previous menu it is possible to:

- insert a reinforcement previously inserted on the column from the list called "Reinforcement";
- select the type of material you intend to use (FRP / FRCM). Specifically, by leaving the FRCM session active, all the input required below will comply with the current legislation for FRCM which is CNR-DT 215/2018;
- choose the reinforcement fabric from the reinforcement libraries;
- choose whether to use conventional values for FRCM fabric.

The use of conventional values refers to what is written in the CNR-DT 215/2018.
In the chapter concerning the calculation of the mechanical characteristics (§3.1) of the aforementioned CNR, reference is made to
the concept of "conventional" stresses and deformations which represents the strength of the reinforcement system obtained by
detachment tests from conventional supports and as such is dependent on the media type.
The use of the conventional limit deformation and of the competent conventional limit tension, allows to design FRCM reinforcement

interventions avoiding *the explicit verification in the comparison of the phenomenon of detachment from the support* or sliding of the
fibers in the matrix at the ends of the reinforcement, otherwise necessary in cases in which this crisis mode is possible.

2

Name	ddd
"ηa" definition	Automatic
Exposure class	Internal
ηa	0,90
Application	Continuous
bf [cm]	0
Layers N° [-]	0
tf [mm]	0,00
ra [mm]	5,00
Bands N° [-]	0
Bands step [cm]	0
Wrapping length [cm]	100
Ef [N/mm2]	0,00
ε fk [%]	0,000
σ uf [daN/cm2]	0,00

Shows the name of the reinforcement fabric inserted from the library
na definition (Automatic/Manual)
Exposure class (Internal/External/Aggressive)
na value
Report the type of application
Width of the reinforcement
Number of layers of reinforcement
Thickness of the reinforcement
Rounding radius
Number of bands of reinforcement
Step bands of reinforcement
Wrapping length
Elastic modulus in the direction of the reinforcement
Characteristic deformation of the reinforcement
Calculation tension at break of dry fabric

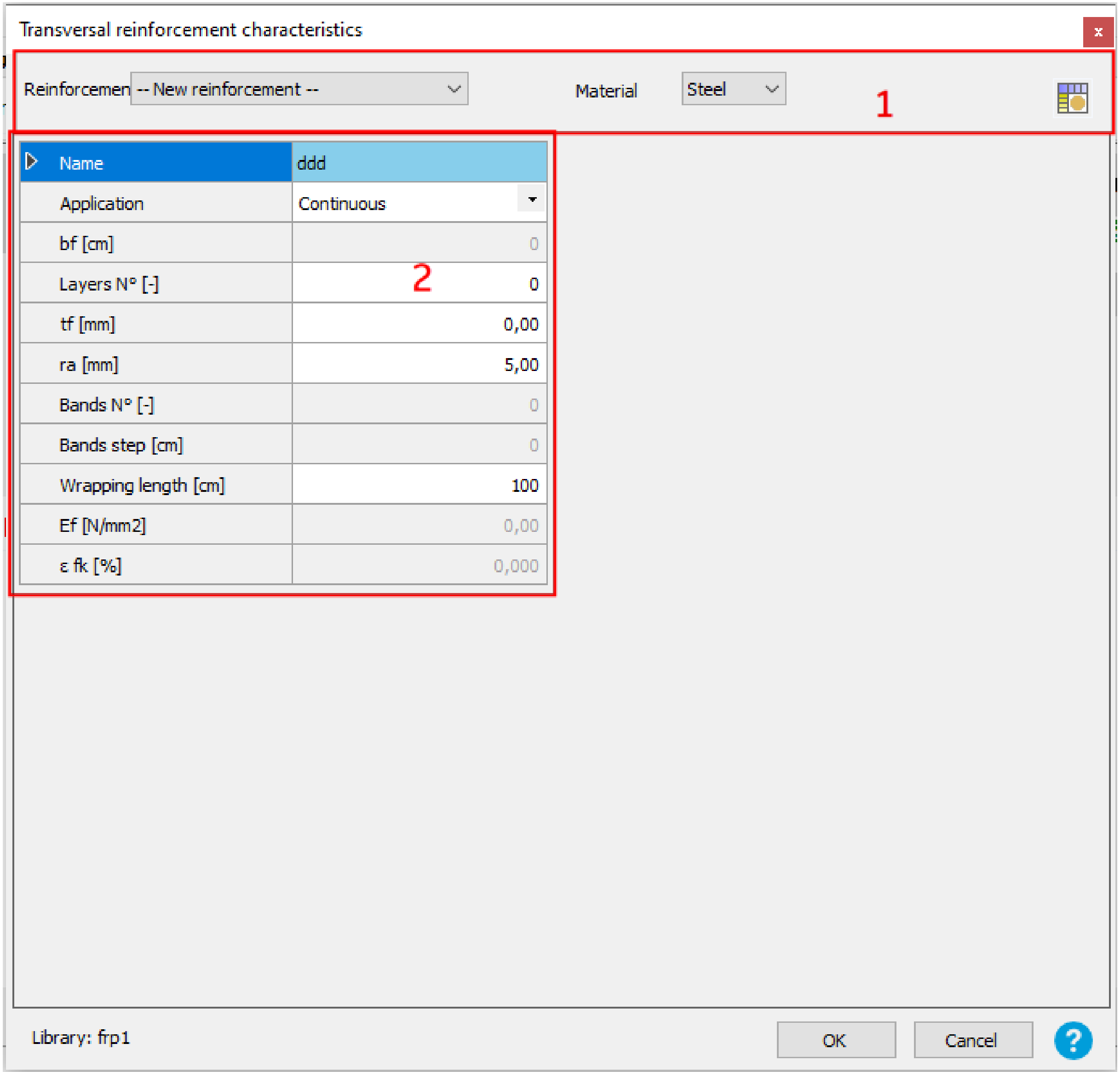
3
Calculation parameters

γM [-]	1,00
fc,mat [N/mm2]	0,00
tmat [mm]	0

Partial safety coefficient of the FRCM [3.1 CNR-DT 215 2018]
Ultimate stress at break characteristic by traction of the dry fabric
FRCM single layer matrix thickness (1cm)

4
Indicates the library to which the type of fabric chosen belongs.

+ **Steel reinforcement:**

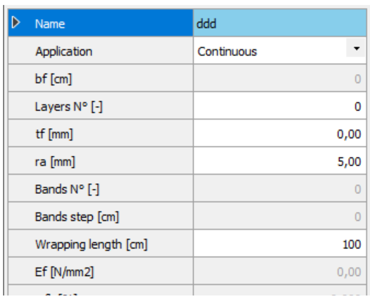


1

At the top of the previous menu it is possible to:

- insert a reinforcement previously inserted on the column from the list called "Reinforcement";
- choose the type of structural steel to be used (UNI reference standards for the quality of steels);

2



Shows the name of the reinforcement fabric inserted from the library
Report the type of application
Width of the reinforcement
Number of layers of reinforcement
Thickness of the reinforcement
Rounding radius
Number of bands of reinforcement
Step bands of reinforcement

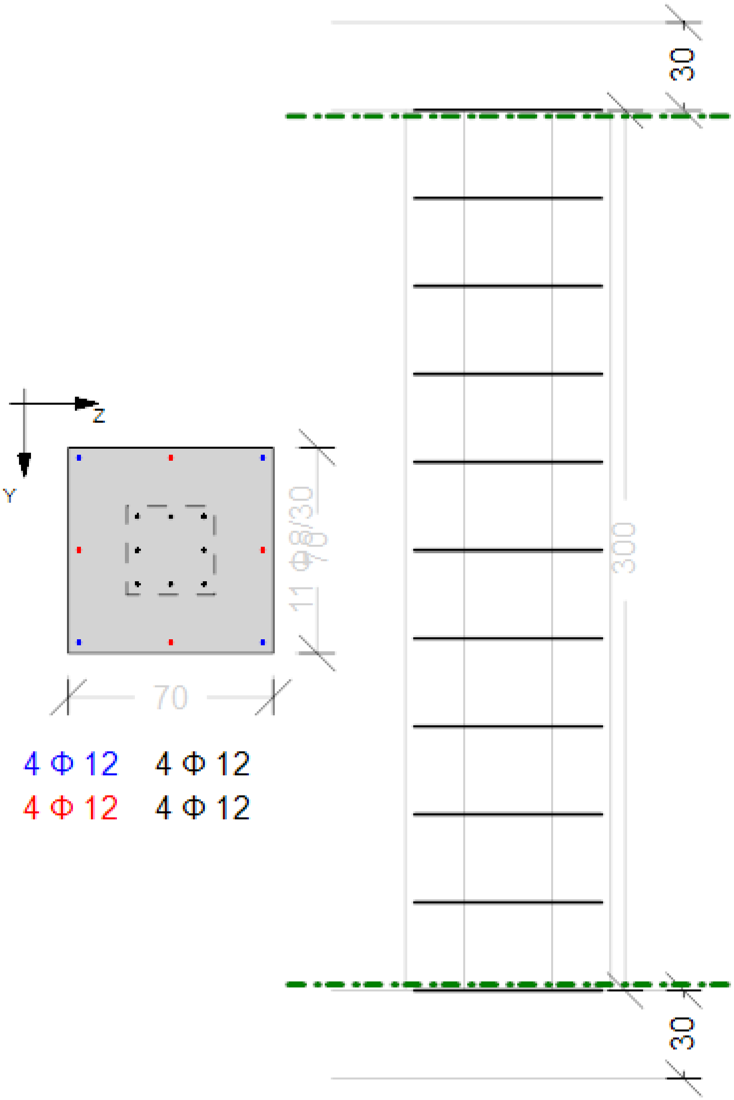
Bandage length
Elastic modulus in the direction of the reinforcement
Calculation tension at break of dry fabric

It is possible to insert / modify the circle by means of the following tools:


Edit / insert jacketing: allows you to insert a jacket in reinforced concrete.

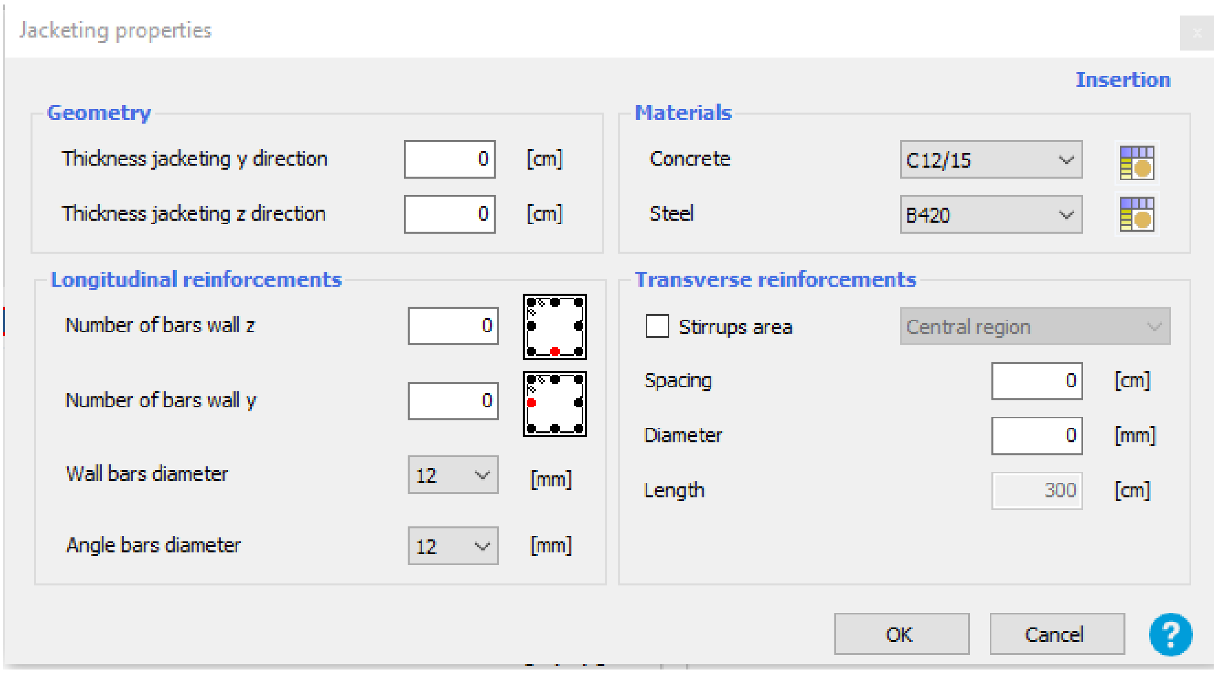

Delete jacketing/reinforcements: to eliminate a reinforcement.

If necessary, the reinforcement that has been inserted is displayed:



For the cladding, the command will require the setting of the following characteristics: geometry (thickness of the jacket in y and z direction), materials (concrete and steel), longitudinal reinforcement (number of wall bars zey, diameter of wall bars and angle) and transverse reinforcement (stirrups area, spacing, diameter and length).

To add a reinforced concrete jacket it is necessary to enter the parameters in the following specific menu.



11.3.1.3.1.4 Results verification

When the column "tab" is dragged from the column "Columns to be calculated" to the column "Calculated columns" it will be possible to view the results of the calculation.

The results of the checks carried out on the pillar in question are reported.
In particular, the checks are as follows:

Normal stress test

Checking by pure compression according to the load combination can be unfavorable, in accordance with what is described in the specific section.

Check with deviated bending

Check with deflection deflection in static and seismic conditions, as described in the specific section.

Verification by shear

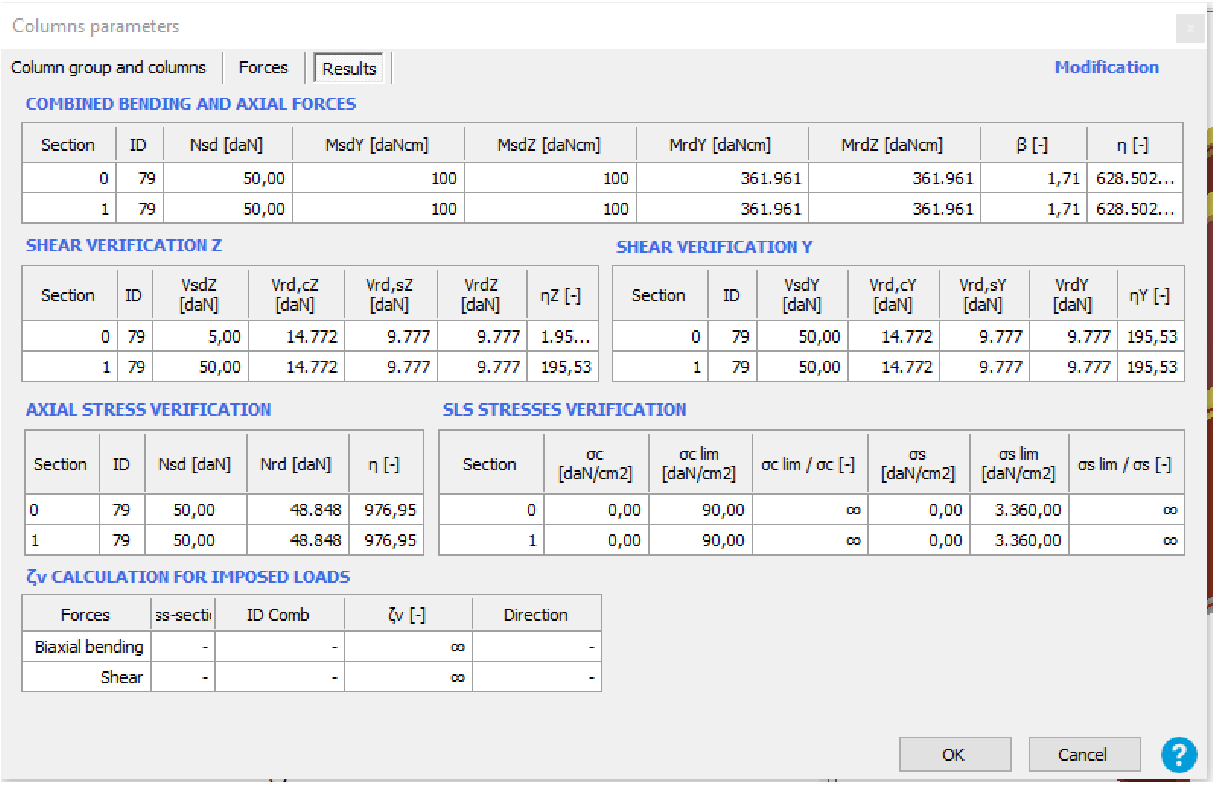
Shear verification performed in static and seismic conditions and according to both local axes, in accordance with what is described in the appropriate section.

Verification of SLS stresses

Verification of the voltages admissible to the SLS, as described in the specific section.

Calculation zv

Calculation of the zv coefficient, as described in the appropriate section.



Section	ID	Nsd [daN]	MsdY [daNcm]	MsdZ [daNcm]	MrdY [daNcm]	MrdZ [daNcm]	β [-]	η [-]
0	79	50,00	100	100	361.961	361.961	1,71	628.502...
1	79	50,00	100	100	361.961	361.961	1,71	628.502...

SHEAR VERIFICATION Z

Section	ID	VsdZ [daN]	Vrd,cZ [daN]	Vrd,sZ [daN]	VrdZ [daN]	ηZ [-]
0	79	5,00	14.772	9.777	9.777	1.95...
1	79	50,00	14.772	9.777	9.777	195,53

SHEAR VERIFICATION Y

Section	ID	VsdY [daN]	Vrd,cY [daN]	Vrd,sY [daN]	VrdY [daN]	ηY [-]
0	79	50,00	14.772	9.777	9.777	195,53
1	79	50,00	14.772	9.777	9.777	195,53

AXIAL STRESS VERIFICATION

Section	ID	Nsd [daN]	Nrd [daN]	η [-]
0	79	50,00	48.848	976,95
1	79	50,00	48.848	976,95

SLS STRESSES VERIFICATION

Section	σc [daN/cm2]	σc lim [daN/cm2]	σc lim / σc [-]	σs [daN/cm2]	σs lim [daN/cm2]	σs lim / σs [-]
0	0,00	90,00	∞	0,00	3.360,00	∞
1	0,00	90,00	∞	0,00	3.360,00	∞

ζv CALCULATION FOR IMPOSED LOADS

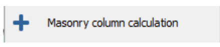
Forces	ss-secti	ID Comb	ζv [-]	Direction
Biaxial bending	-	-	∞	-
Shear	-	-	∞	-

11.3.2 Masonry column

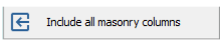
The Masonry column module allows the static verification of new or existing masonry pillars according to the prescriptions of chapter 4.5 of the Ministerial Decree 17 January 2018

11.3.2.1 Creation of a calculation tab

Il menù dedicato ai pilastri in muratura possiede diverse opzioni:

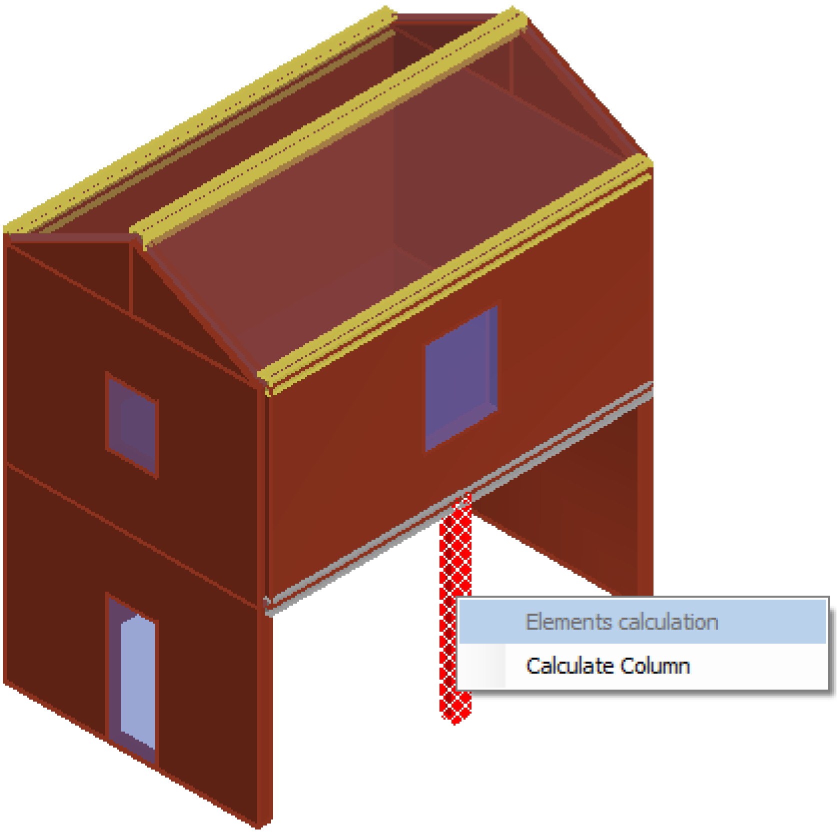


Permette di creare una nuova scheda di calcolo. Selezionando questa opzione, la scheda verrà creata come scheda utente.



Selezionando questa opzione, verrà creata una scheda di calcolo per ogni pilastro presente all'interno del modello della struttura. Le schede create saranno schede da modello.

It is also possible to create a calculation sheet starting from a column in the model. To do this, simply right-click on the "Column" element in the model and choose the "Calculate Masonry Column" option.



11.3.2.2 Code

The checks are carried out according to chapter §4.5.1 of the Ministerial Decree of 17 January 2018. The thickness of the wall element is assumed to be the relative size of the pillar in the direction of calculation:

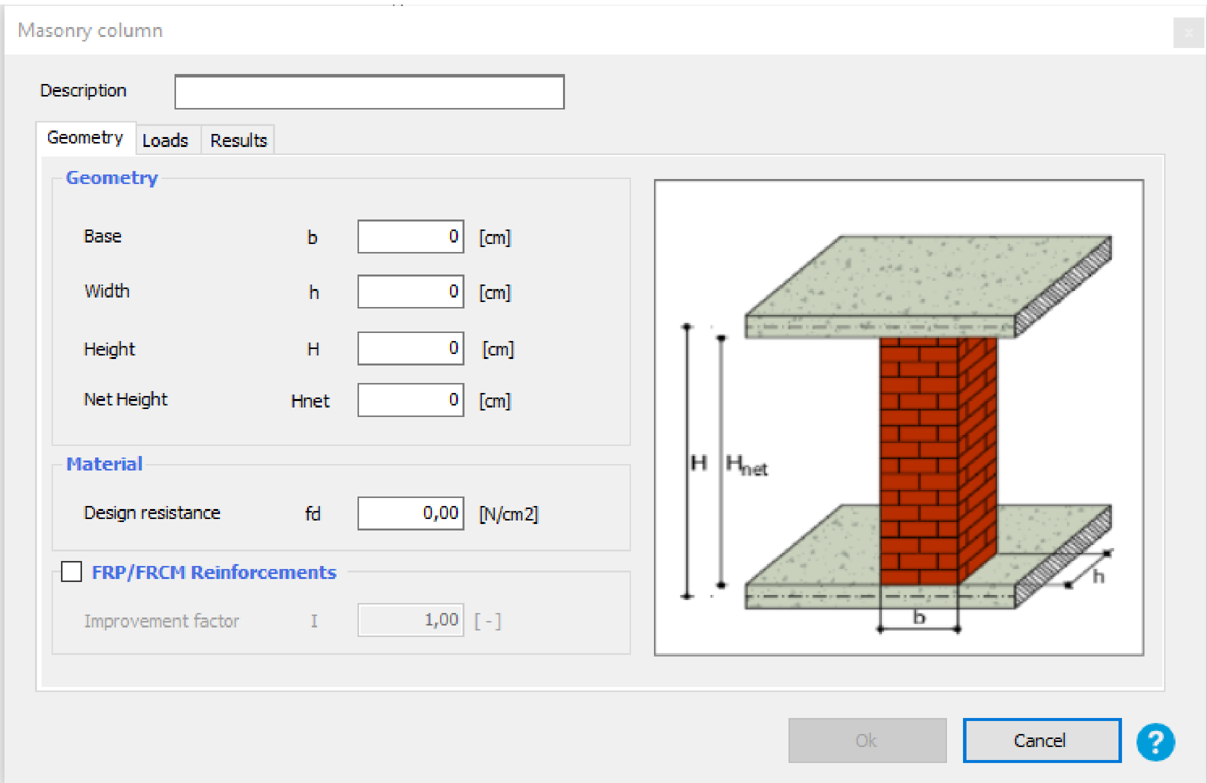
- base b in x direction
- width h in the y direction

Slenderness check: § [4.5.1]	**max(H0,x/b; H0,y/h) ≤20**
	H0, i: free length of wall deflection in direction i equal to ρi • H *b: base of the pillar* *h: width of the pillar* *ρ, i: factor that takes into account the effectiveness of the constraint provided by the orthogonal walls in direction i* *H: internal height of the floor*

Load eccentricity check: § [4.5.11]	**$\max(e_{1,x}/b;\ e_{1,y}/h) \le 0.33$** **$\max(e_{2,x}/b;\ e_{2,y}/h) \le 0.33$**
	$e_1 = \lvert e_s \rvert + \lvert e_a \rvert$; $e_2 = \frac{e_1}{2} + \lvert e_v \rvert$ *eg: total eccentricity of vertical loads;* *ea: eccentricity due to execution tolerances;* *ev: eccentricity due to the wind;*
Verification of vertical loads: § [4.5.6.2]	**$N_d \le \Phi_x f_d A$** **$N_d \le \Phi_y f_d A$**
	Nd: *vertical load computational agent at the base of the wall;* ***A:*** *area of the horizontal section of the wall net of the openings;* ***fd:*** *calculation resistance of the masonry;* ***Φ, i:*** *reduction coefficient of the resistance of the wall in direction i.*

11.3.2.3 Work environment

The main working environment is as follows:

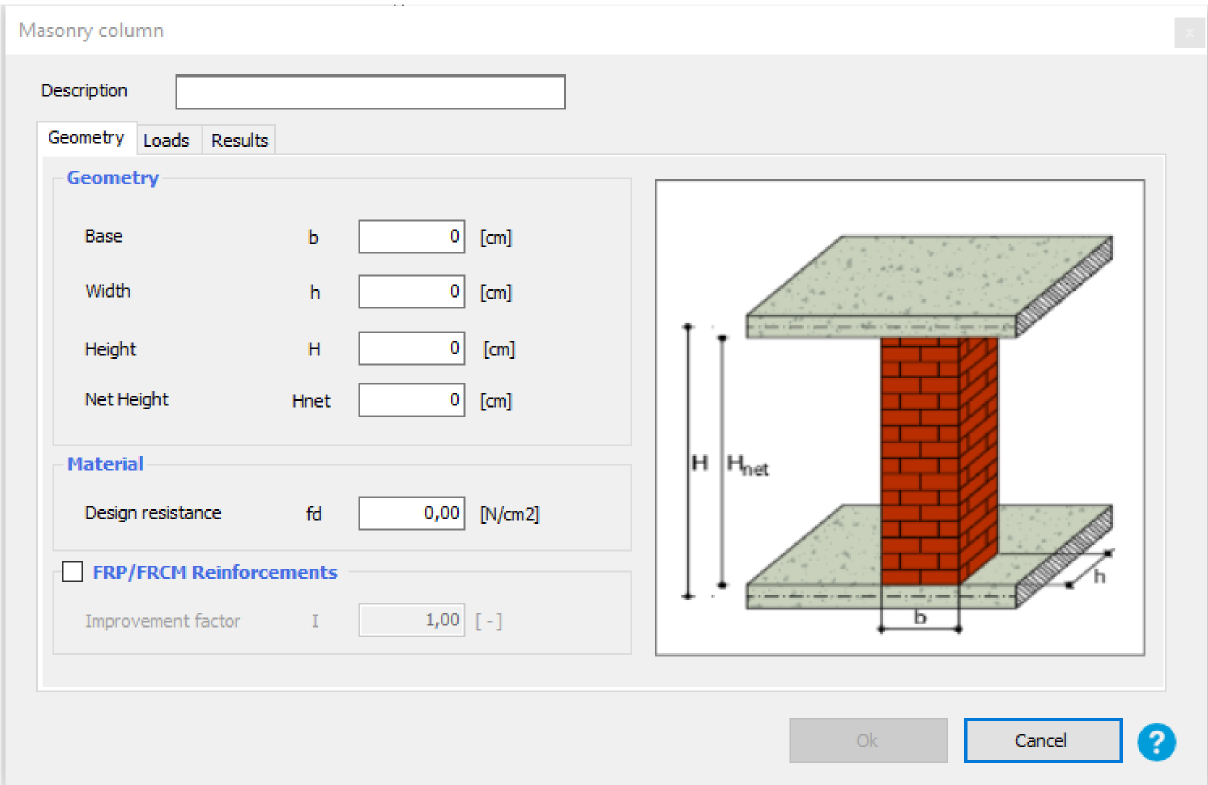


The work environment is divided into 3 parts:

- Geometry
- Loads
- Results

11.3.2.3.1 Geometry

In this section it is possible to define the geometric and material data necessary for the calculation.



Below is a description of the required input data.

Base,b Referred to the x axis of calculation of the element

Width, h Referred to the y axis of calculation of the element

Height, H Referred to the distance between the axes of the lower and upper elements connected to the column

Net Height, H Referred to the free height of the pillar between the elements connected to the pillar

Design resistance, fd Calculation compressive strength of the masonry, equal to:

$fd = fk / \gamma_M$ in the case of a new element

$fd = fm / (\gamma_M FC)$ in the case of an existing element

Where:

fk: characteristic compressive strength of the masonry

fm: average compressive strength of the masonry

γ_M: partial safety factor of the material

FC: material knowledge factor

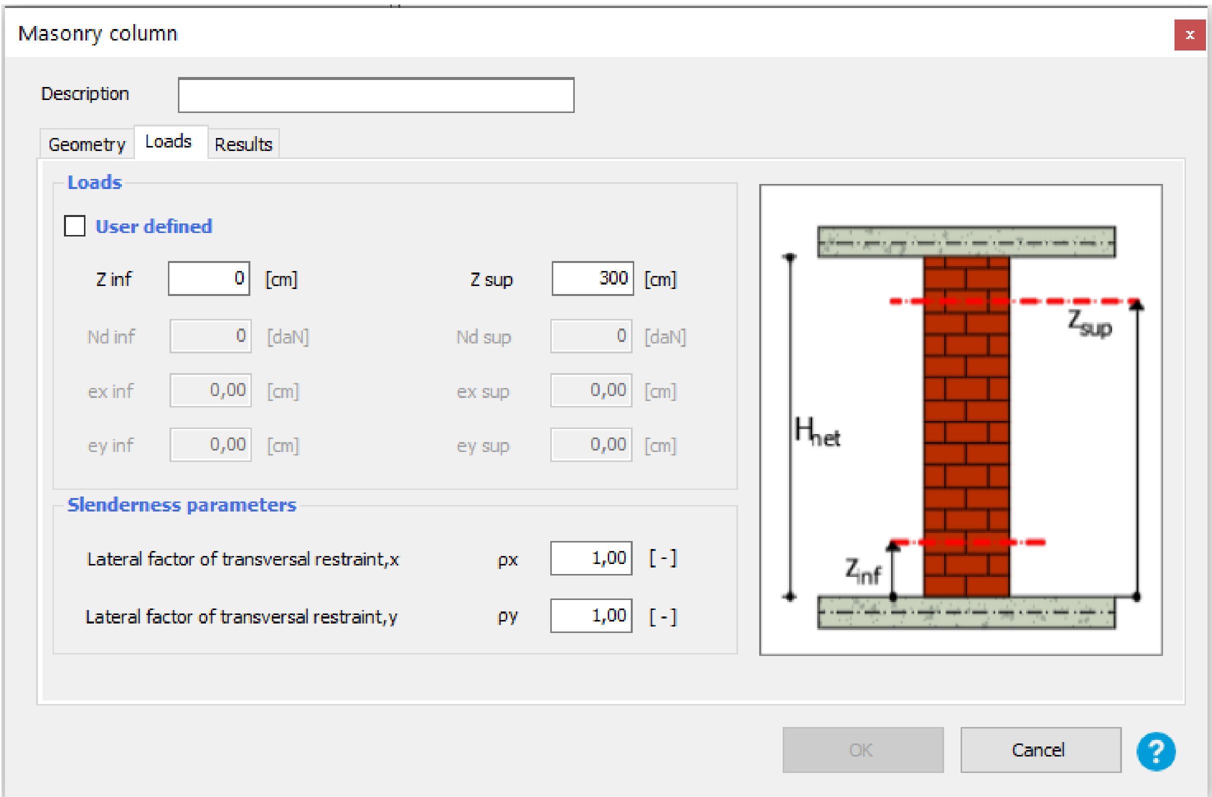
FRP/FRCM Reinforcements It allows to consider the presence of any reinforcement of the material by means of the improvement factor I

Improvement factor, I Multiplication factor of the design resistance fd based on the confinement effect of the reinforcement Calculated according to DT200 R1 / 2013

These data, except for the net height, will not be editable in the case of a model tab as they will derive from the latter.

11.3.2.3.2 Loads

In this section it is possible to define the loads and calculation parameters of the column



Below is a description of the required input data.

User defined
Visible only in the case of a model tab. Allows to manually change the value of the input loads. This will make it no longer possible to choose the Zv parameter in the Results section as it is meaningless.

Z inf
Application height, measured with respect to the net height of the column, of the loads at the lower level

Z sup
Application height, measured with respect to the net height of the column, of the loads at the upper level

Nd inf
Normal design effort, assumed as positive if compressive, at the Zinf level

Nd sup
Normal design effort, assumed as positive if compressive, at the level of Zsup

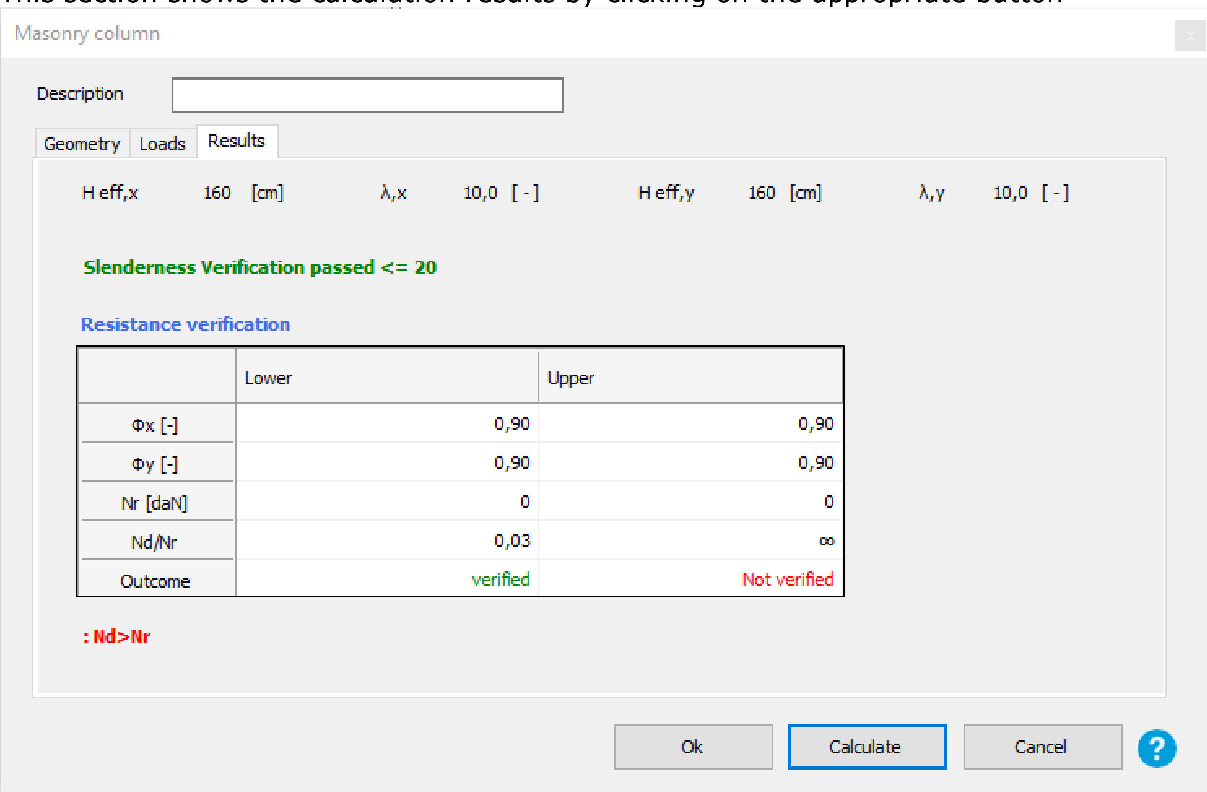
ex inf
Total design eccentricity, referred to the base, due to the vertical loads at the Zinf level

ex sup
Total design eccentricity referred to the base, due to the vertical loads at the Zsup level

ey inf	Total design eccentricity, referred to height, due to vertical loads at the Zinf level
ey sup	Total design eccentricity, referred to height, due to vertical loads at the Zsup level
Lateral factor of transversal restraint, x rx	Factor necessary for the definition of the free length of deflection, calculated according to chapter [4.5.6.2] of NT18
Lateral factor of transversal restraint, y ry	Factor necessary for the definition of the free length of deflection, calculated according to chapter [4.5.6.2] of NT18

11.3.2.3.3 Results

This section shows the calculation results by clicking on the appropriate button



Below is the description of the output data.

H0,i	Free length of column deflection in direction i
l,i	Slenderness in direction i of the pillar
z_v	Visible only if the loads imported from the model are kept and the calculation of zv has been requested in the general static analysis. When changing the selection, it will apply the corresponding overload reduction coefficient zv indicated in § [8.3] of the 2018 NTs, updating the values in the Loads tab
$z_{v,lim}$	Indicates the limit value of zv obtained from the static

analyzes carried out on the global model in order to know which is the acceptable limit value for the remaining elements of the model

Eccentricity checks

This table shows reference values for the eccentricity check for the upper and lower calculation section defined in the Loads tab

- ei: value of the calculation eccentricity (maximum between e1 and e2) obtained as indicated in chapter § [4.5.6.2] of NT18
- in direction i

- ex / b: value of the calculation eccentricity (maximum between e1 and e2) normalized with respect to the base of the pillar

- ey / b: calculation eccentricity value (maximum between e1 and e2) normalized with respect to the column width

Resistance verification

This table shows reference values for the resistance check for the upper and lower calculation section defined in the Loads tab

- Φi: value of the reduction coefficient of the column resistance in direction i

- Nri: reduced normal resistance stress of the abutment in direction i

- Nd: normal design stress acting on the pillar

- Nd / Nri: ratio between normal force acting and resistant design in direction i

11.4 Walls

11.4.1 Openings

11.4.1.1 Introduction

Chapter 8.4 of the Technical Standards for Construction 2018 classifies the interventions into three main families:
- Adjustment intervention
- Improvement Intervention
- Repair or Local intervention

The first two interventions require the designer to carry out an overall check of the structure, while the repair intervention requires a simpler local check.
With reference to what is described in chapter C8.4.1 (Circular n.7 of 21 January 2019)

on the subject of local interventions, it is possible to understand how to distinguish a case of Local Intervention from the other two more complex cases.

[C8.4.1]: "... repairs, reinforcement or replacement of structural elements or parts of them not suitable for the function they must perform (for example beams, architraves, roofs, decks or portions of deck, pillars, wall panels) . [...] The restoration or reinforcement of the existing connections between the single components or between parts of them or the creation of new connections [...]. Finally, the modification of a limited part of the structure (eg. 'opening of a space in a wall, accompanied by appropriate reinforcements), [...] provided that it is demonstrated that the set of interventions does not significantly modify the stiffness, resistance against horizontal actions and deformation capacity of the structure. "

The insertion of a new opening involves variations in the stiffness, strength and ductility of the wall.
The containment of the variation of these quantities allows us to say that the intervention implemented is of a local type.

It is a common practice in the design to size the openings hoops so as to be able to catalog them as a local intervention so as not to have to proceed with the overall verification of the building.

11.4.1.2 Calculation procedure

The calculation procedure is based on the comparison between the as installed and the as designed (introduction of a new opening).
The expression "as installed" means the schematization of the entire wall **before** the opening is made.
The expression "as designed" means the schematization of the entire wall **after** the opening has been made.

The comparison is made with respect to the variations of:

- Resistance of the wall system
- Stiffness of the contained wall system (at the regulatory level there is no limit to this variation, alternative technical documentation suggests ± 15%)
- Deformation work, representative of the overall behavior of the wall

The identification of the stiffness, strength and deformation work of the wall is carried out by calculating the contribution of the individual elements.

The mechanical characteristics of the individual elements can be calculated with the following formulations:

Wall panel

$$Km = \frac{G \cdot l \cdot t}{1.2 \cdot h} \cdot \frac{1}{1 + \frac{1}{1.2} \frac{G}{E} \cdot \left(\frac{h}{l}\right)^2}$$ Stiffness

$$V_{mt} = l \cdot t \cdot \frac{1.5 \cdot \tau_0}{b} \cdot \sqrt{1 + \frac{\sigma_0}{1.5 \cdot \tau_0}}$$ Shear resistance

$$V_{mpf} = \frac{l^2 \cdot t \cdot \sigma_0}{h} \cdot \left(1 - \frac{\sigma_0}{0.85 \cdot f_m}\right)$$ Axial bending resistance

$d_y = \frac{V_t}{K_m}$	Elastic limit displacement assuming Vt=min(V_{mt}, V_{mpf})
$d_u = 0{,}005\ h$	Ultimate displacement of a wall panel that is in crisis due to shear
$d_u = 0{,}01\ h$	Ultimate displacement of a wall panel that is in crisis due to axial bending

In which:
E, G: Elastic modules
h, l, t: height, length and thickness of the panel

Reinforcement frames in reinforced concrete

$K_t = \frac{12 \cdot E \cdot J}{h^3}$	Stiffness

E: elastic modulus of the upright of the steel frame
J: Module of inertia of the section in reinforced concrete
h: height of the upright

Steel frames

$K_t = \frac{12 \cdot E \cdot J}{h^3}$	Stiffness
$V_t = \frac{2 \cdot f_{yk} \cdot W}{\gamma_m \cdot h}$	Resistance
$d_{y,telaio} = \frac{V_{telaio}}{K_{telaio}}$	Elastic limit displacement

E: elastic modulus of the upright of the steel frame
J, W: Modulus of inertia and modulus of resistance
h: height of the upright

Calculation hypothesis

Agent load:

It is assumed that the vertical load acting is discharged entirely on the masonry walls without affecting the uprights of the hoop.

Any partial loading of the uprights can only translate into a reduction of the loads affecting the taps, performed manually by the user.

The loads to be inserted for each single male are to be considered calculated in the half-height section of the male.

Self weight of the masonry:

The self-weight is not calculated automatically but must be entered as a load acting on the tap.

Ultimate wall displacement:

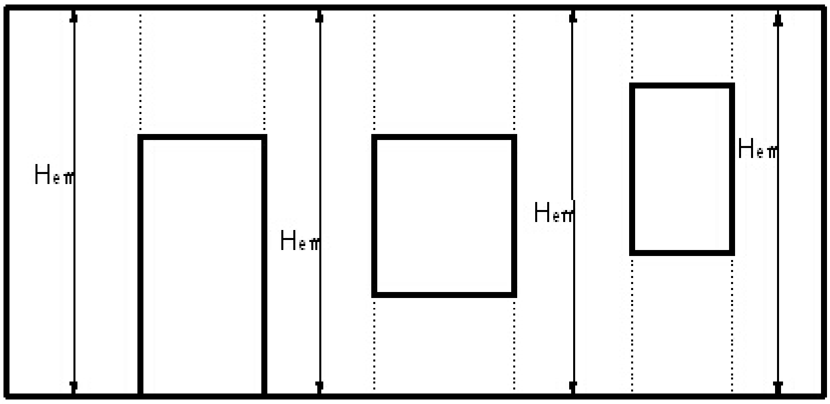
The comparison between the actual state and the project state is carried out until the last least displacement of all the displacements of the panels present is reached, calculated both in the current and adequate conditions.

Panel heights:

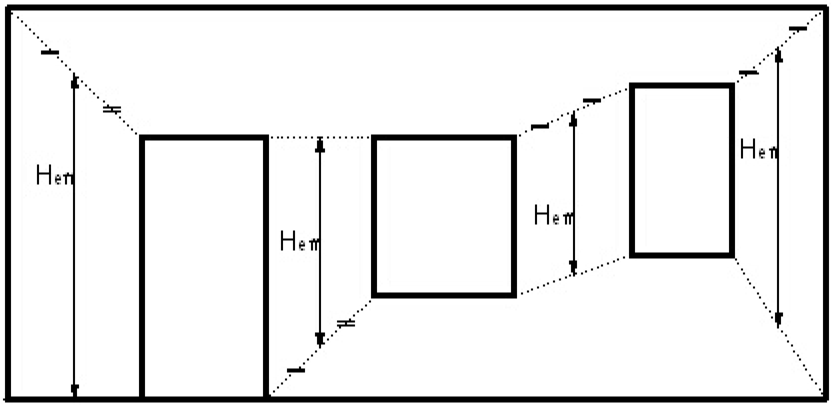
The heights of the panel to be inserted in the input phase are to be understood as "effective heights".

As effective height, we mean the height on which the limit displacement capacity is calculated; this value can be assumed to be equal to the overall height of the wall or distinguished from panel to panel on the basis of how much the contiguous openings can limit the confinement effect.

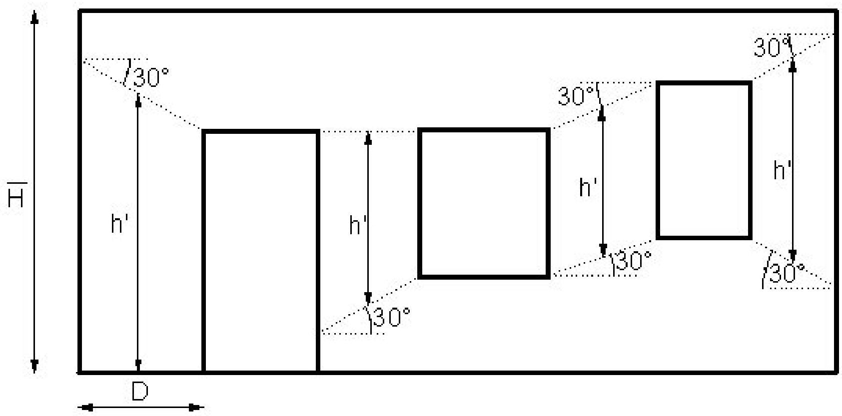
The following figures show some of the most used theories in this schematization.



Heff = Overall height of the wall



Heff = Average height between the heights of the contiguous openings



$$H_{eff} = h' + \tfrac{1}{3} D(\overline{H} - h')/h'$$

Definition of the effective height of the masonry walls (Dolce, 1989).

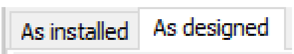
Ultimate displacement of the reinforcement frame:
The yield and ultimate displacements of the frames are calculated using the expressions of the relative rotation capacity reported in Circular no. 7 of 21 January 2019 (expressions (C8.7.2.7a) and (C8.7.2.1) respectively).
Strength of the reinforcement frame:
The resistance of the frame in the horizontal direction T is calculated using the relation:

$$T = {}^{2M}/_{h}$$

In which:
- M is the resistance to limb
- h is the height of the frame
- It is verified that this resistance does not exceed the maximum shear strength assessed according to the indications of the Ministerial Decree of 17.1.2018 (expression (4.1.29)).

11.4.1.3 Work environment

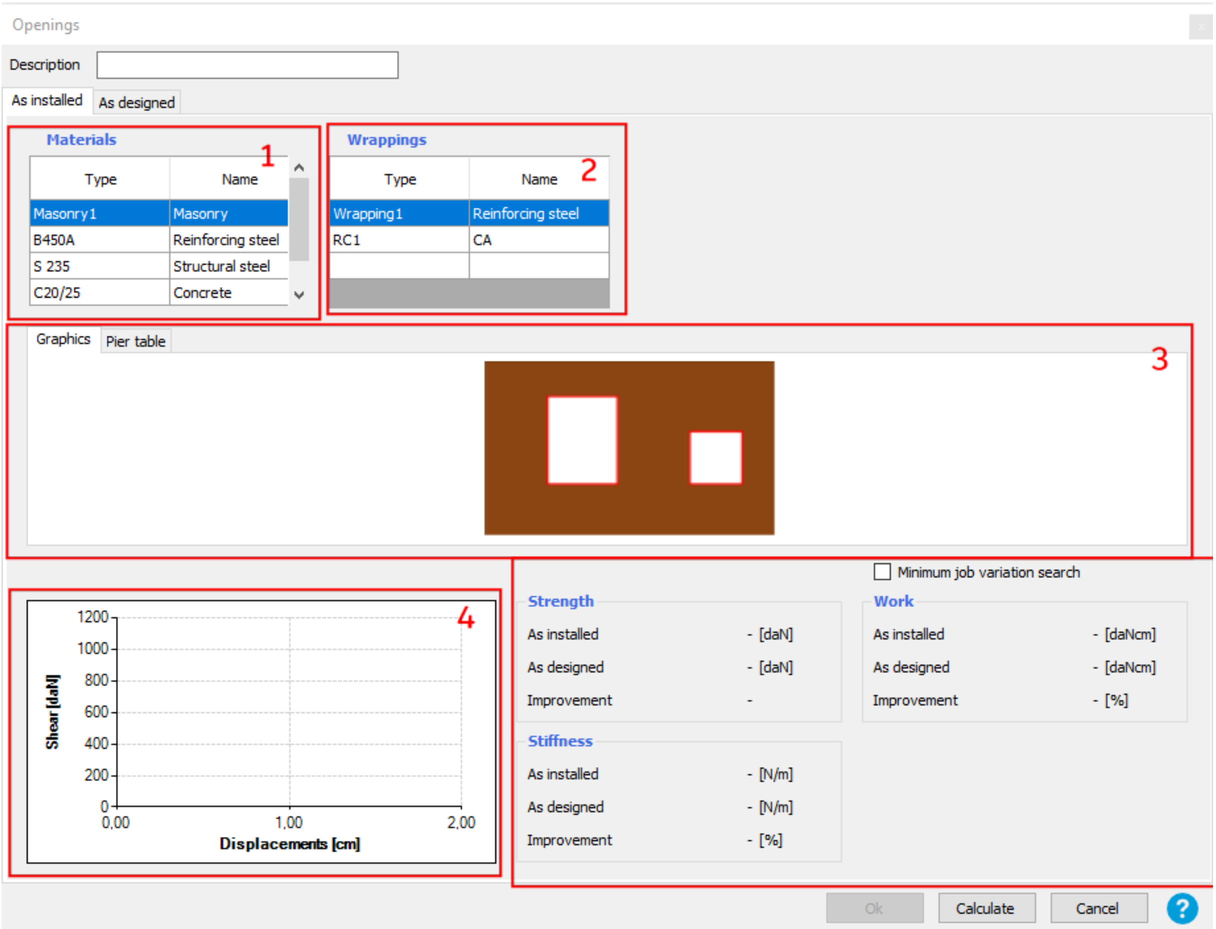
The program window includes two different environments:
As installed: Implementation of the current geometry, prior to the insertion of the openings.
As designed: Implementation of the geometry following the insertion of the openings.



Each of the individual environments is divided into 4 work areas:

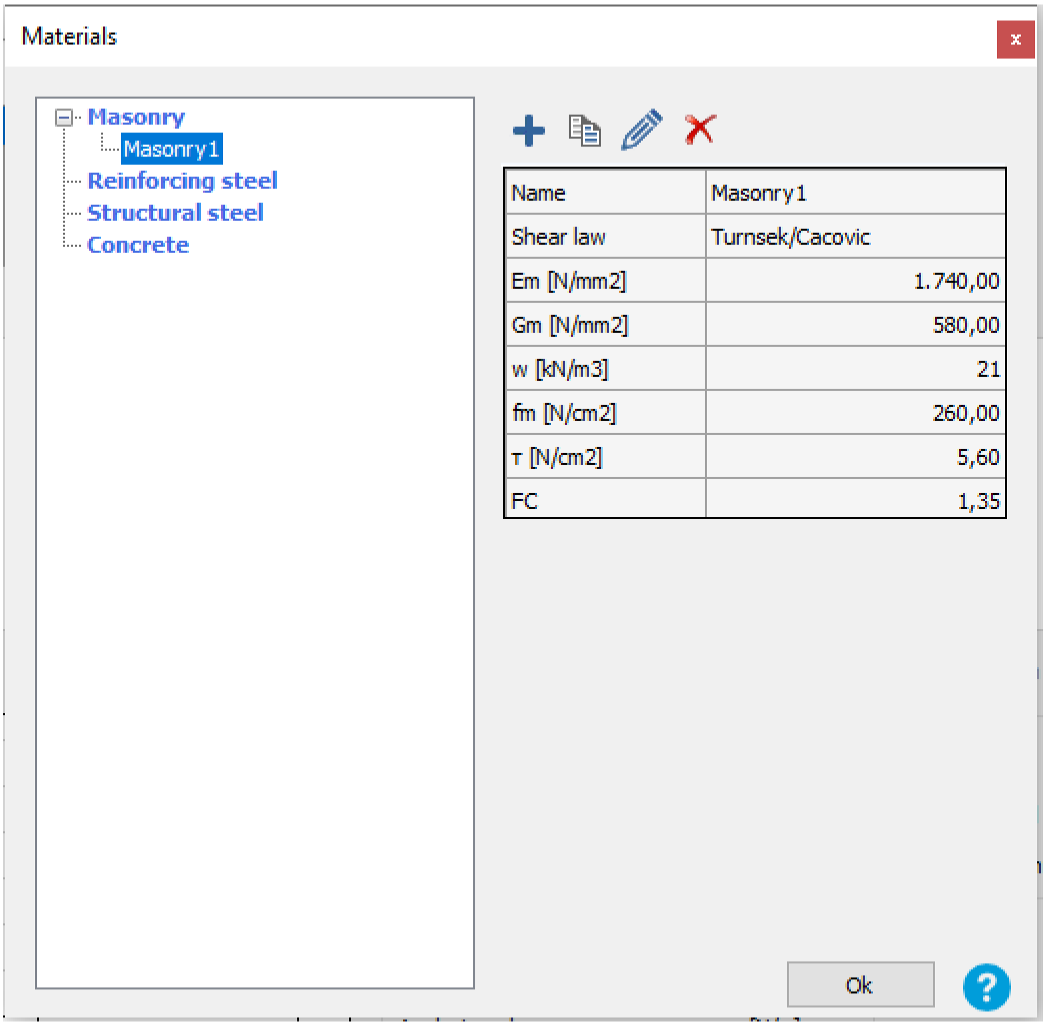
1. Input material characteristics
2. Input characteristics of the hoop
3. Characteristic input of the wall
4. Calculation results and panel capacity diagram.

The individual environments are illustrated in the following figure:

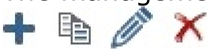


11.4.1.3.1 Input of material properties

In the area for defining the characteristics of the materials, the characteristics of Masonry, Steel and Concrete that will be used in the calculation are defined.
In start mode [U] the user can manually enter the characteristics of the masonry; vice versa, in [M] mode, the masonry material will automatically coincide with that deriving from the calculation model.
The material management screen is as follows:



The management of the added materials can be carried out by means of the icons



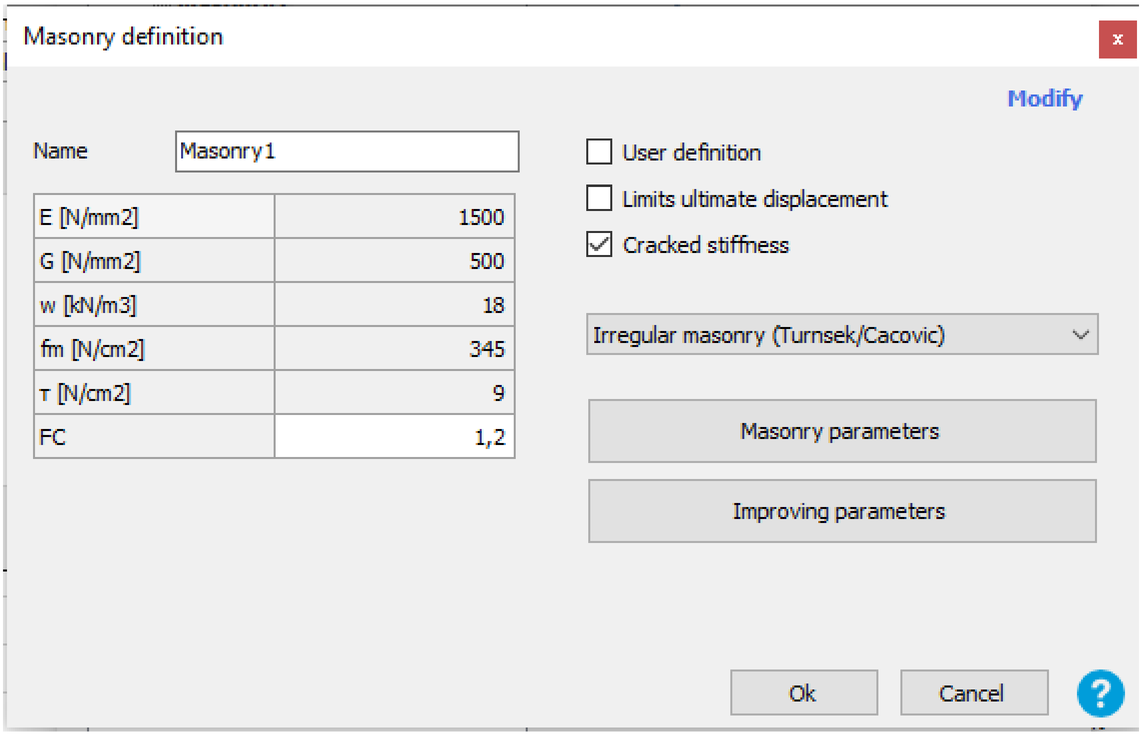
Depending on the material selected, you will enter the screen dedicated to:

- Masonry;
- Reinforcing Steel;
- Structural steel;
- Concrete.

11.4.1.3.1.1 Masonry

If the module is started in stand-alone mode, it will be necessary to define a masonry material.

The masonry material management window is as follows:



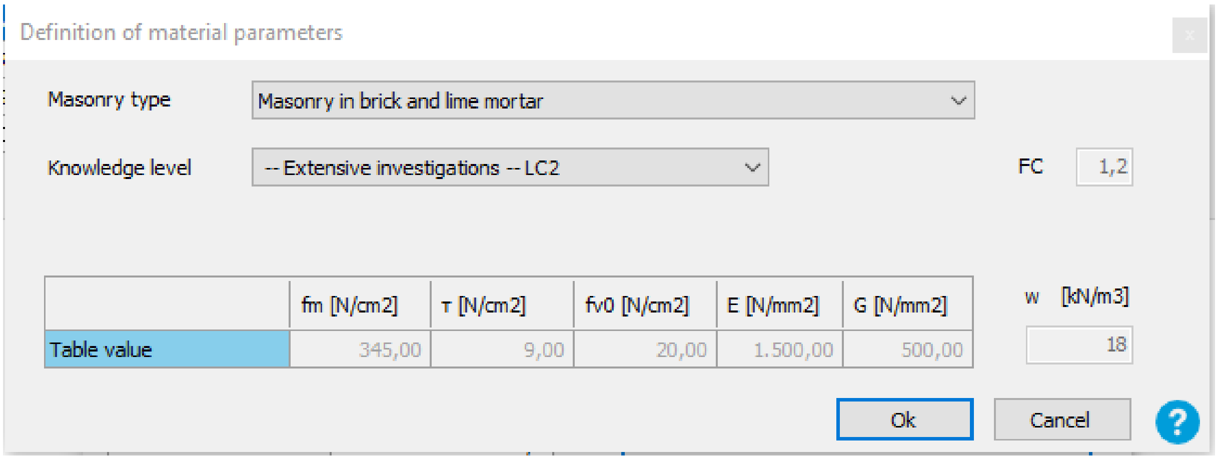
The Limit ultimate displacement option applies the limitation of the ultimate displacement to pressure bending of the C7.8.2.2.1.
This limitation is mandatory for new walls but there is no reference for the existing one.

The possibility to choose between different calculation criteria (Turnsek / Cacovic and Mohr / Coulomb) is conditioned by the choice of the type of masonry in Masonry parameters.

The window for defining the masonry parameters allows the choice of the type of masonry and the level of knowledge.

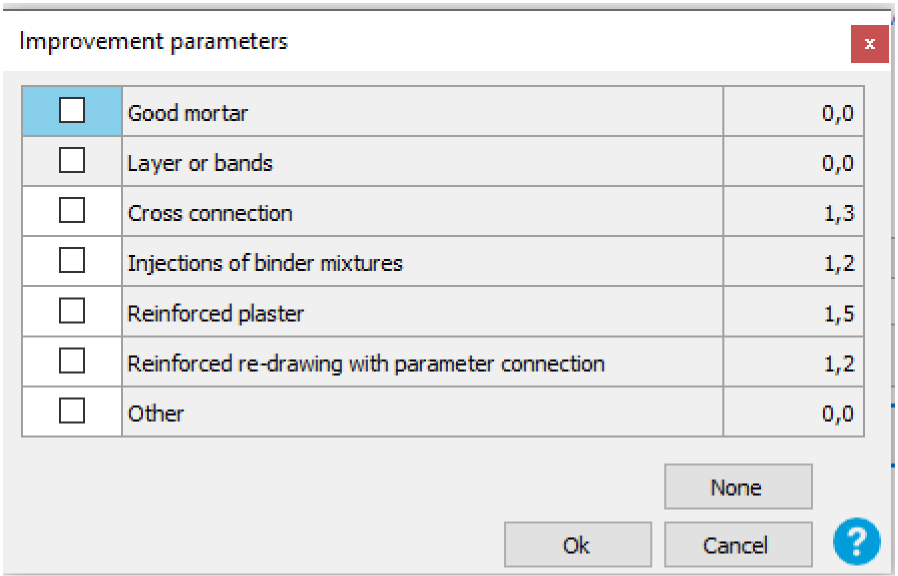
Depending on the combination of the two choices, the table below shows the values of the mechanical characteristics of the project masonry.

In the case of Limited investigations (LC1) or Extended investigations (LC2) the program will show the associated Confidence Factor; in the case of selection of the LC3 knowledge level, it will be necessary to enter the number of tests, the method used and the values obtained from the tests carried out, in order to calculate the design values.



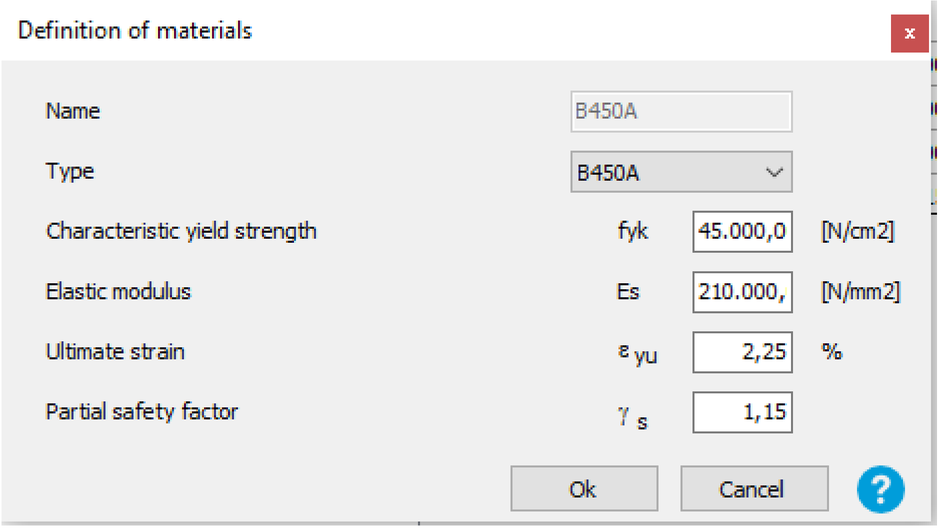
It is possible to set parameters for improving the characteristics of the masonry, on the basis of planned interventions, already carried out, or on the characteristics of the building that are presumed to be favorable.

The screen is as follows:



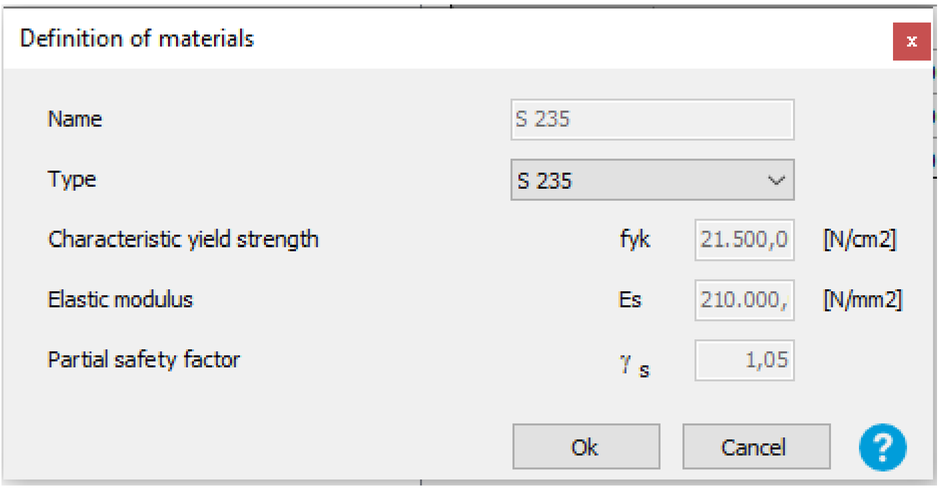
11.4.1.3.1.2 Steel

The steel material management window is as follows:



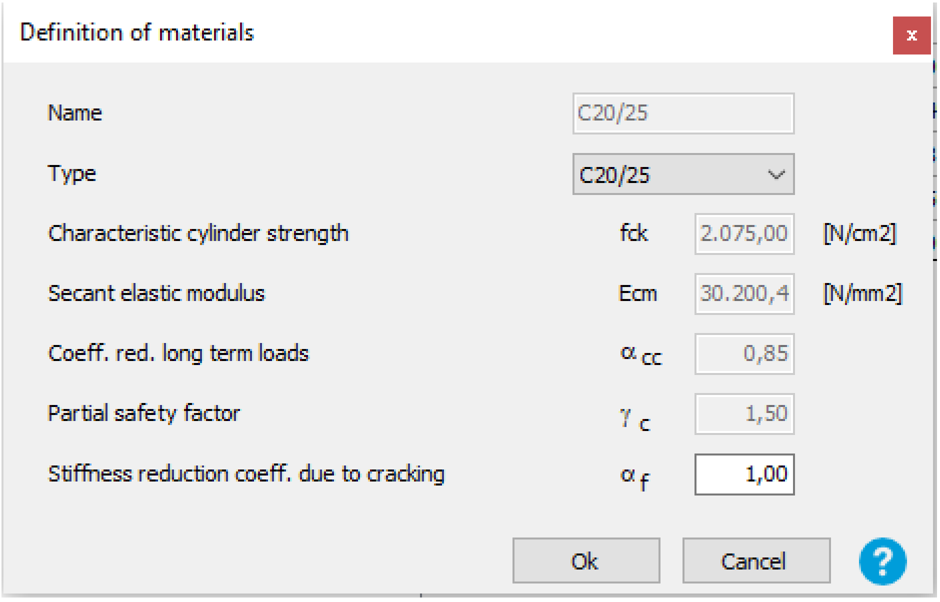
11.4.1.3.1.3 Structural steel

The structural steel material management window is as follows:



11.4.1.3.1.4 Concrete

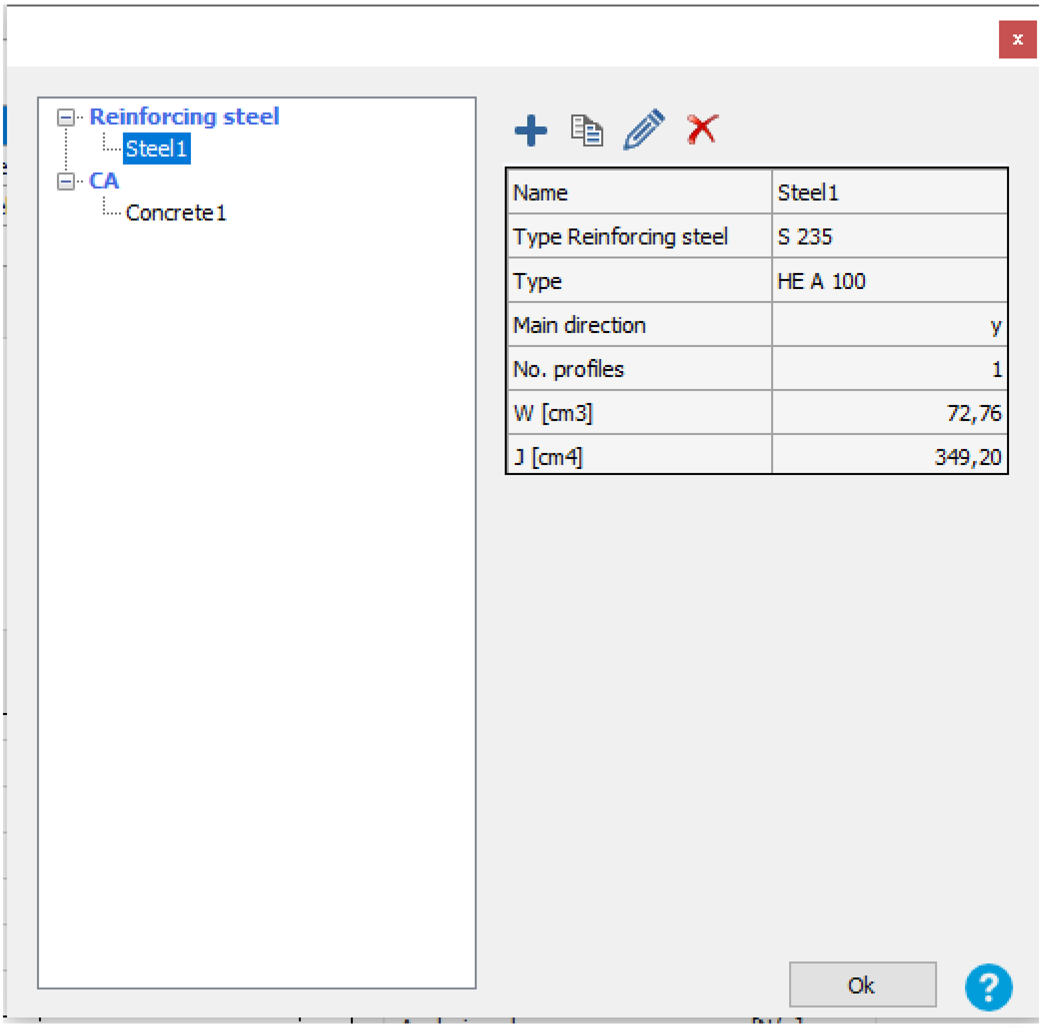
The concrete material management window is as follows:



11.4.1.3.2 Input wrapping properties

In the area for defining the characteristics of the wrappings, the materials, geometry and reinforcement used in the hoop are defined.

The hoop management screen is as follows:



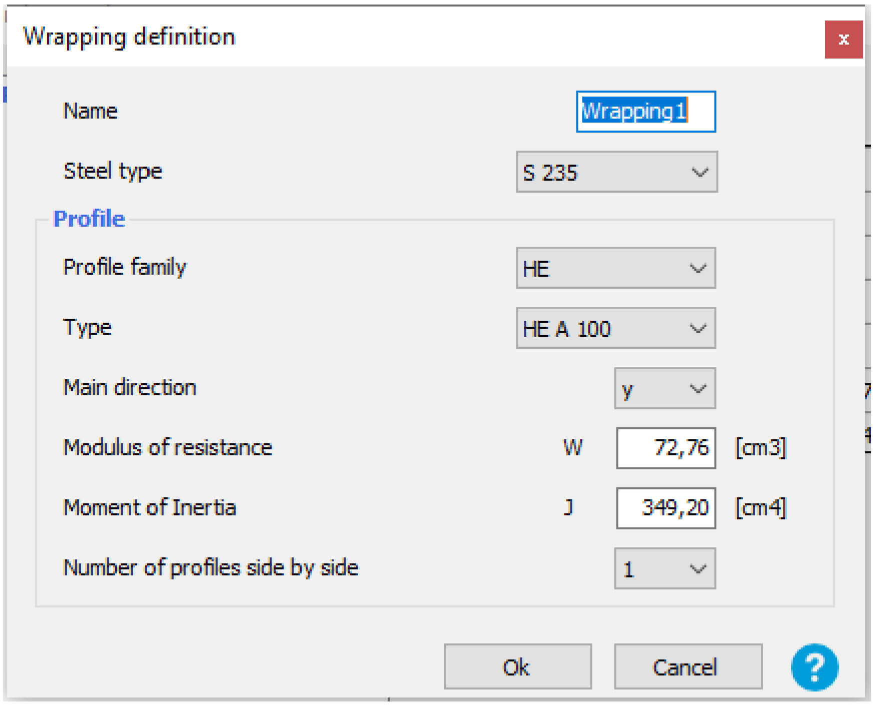
The management of the added circles can be done by means of the icons 

Depending on the selected circle, you will enter the screen dedicated to:

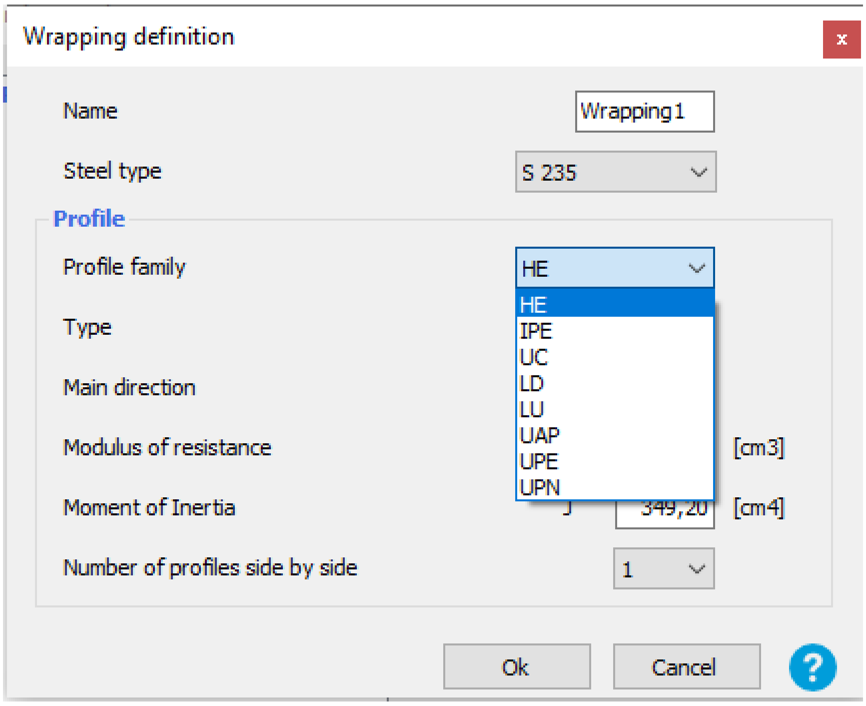
- steel wrappings;
- reinforced concrete wrappings

11.4.1.3.2.1 Definition of steel wrapping

The steel wrapping definition window is as follows:

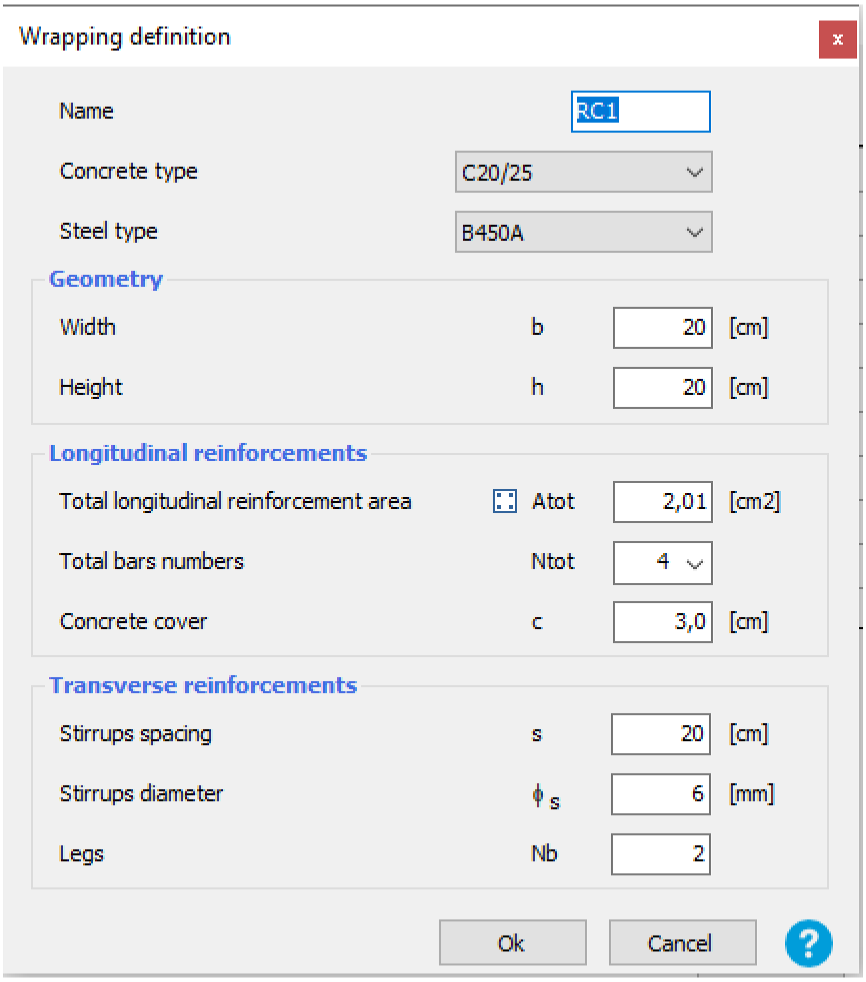


In the list of profiles it is possible to choose, instead of a precise section belonging to the selected family, to let the program design the ring (and then choose its geometry) in such a way as to satisfy all the required checks.



11.4.1.3.2.2 Definition of concrete wrapping

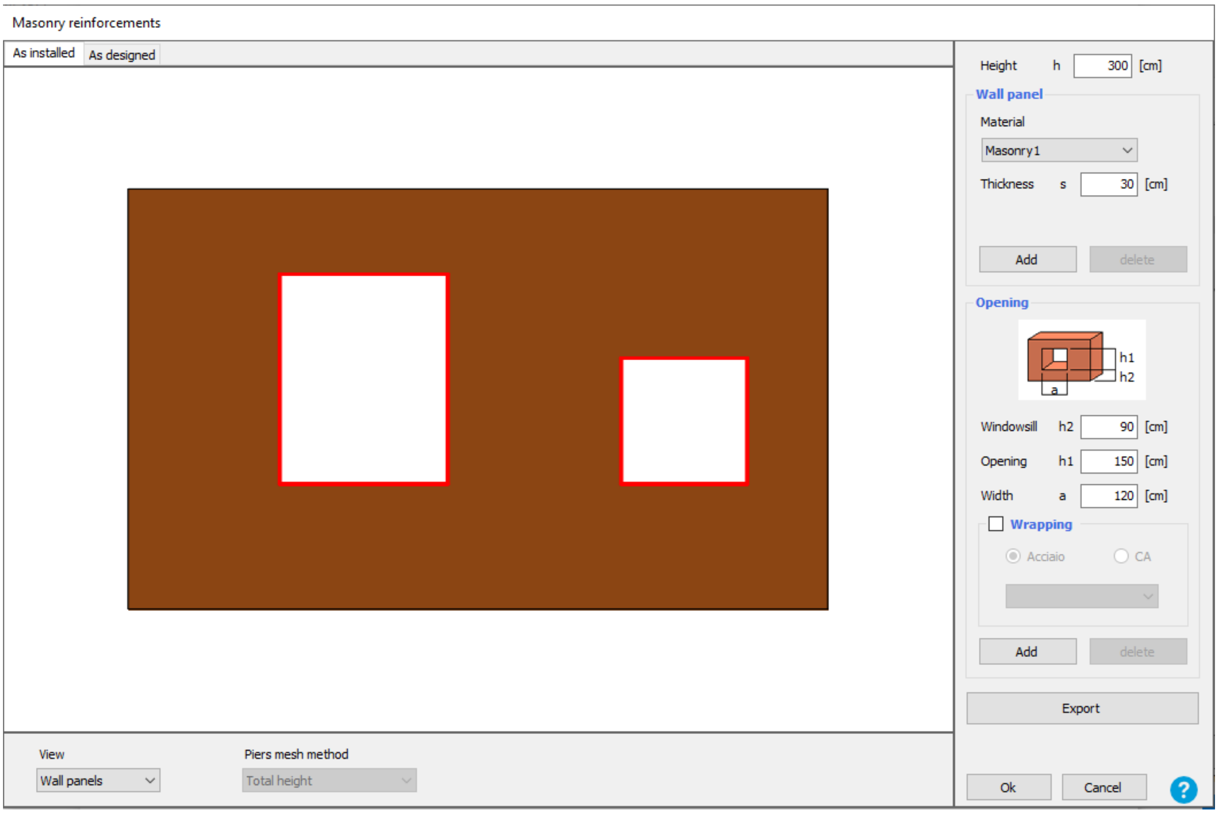
The window for defining the concrete encirclement is as follows:



The longitudinal reinforcement is understood in the calculation as consisting of four corner bars of equal diameter.

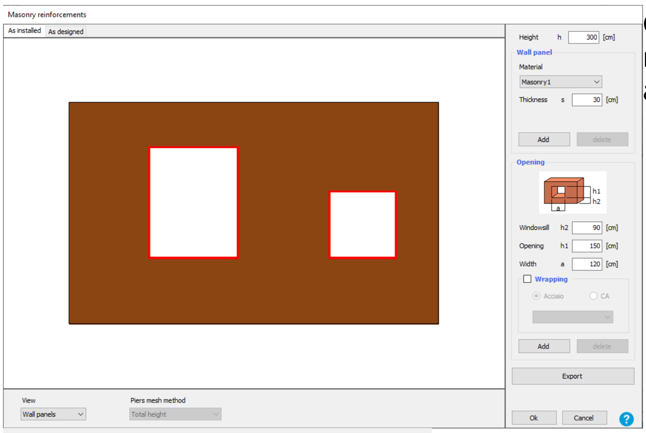
11.4.1.3.3 Input wall properties

To access the area for defining the characteristics of the wall, double click on the image of the wall.

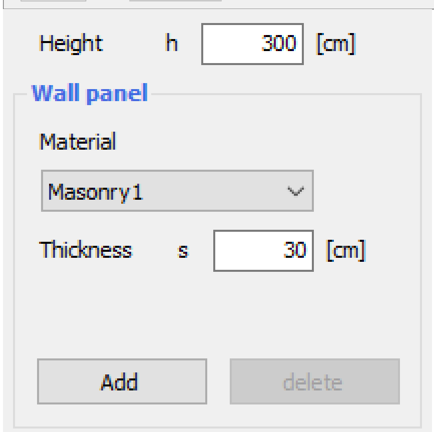


When the environment is opened, an initial panel is proposed that can be modified.

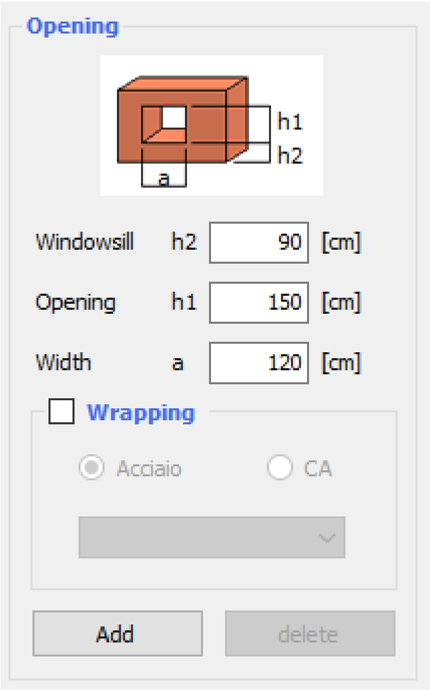
The area for defining the characteristics of the wall is made up as follows:



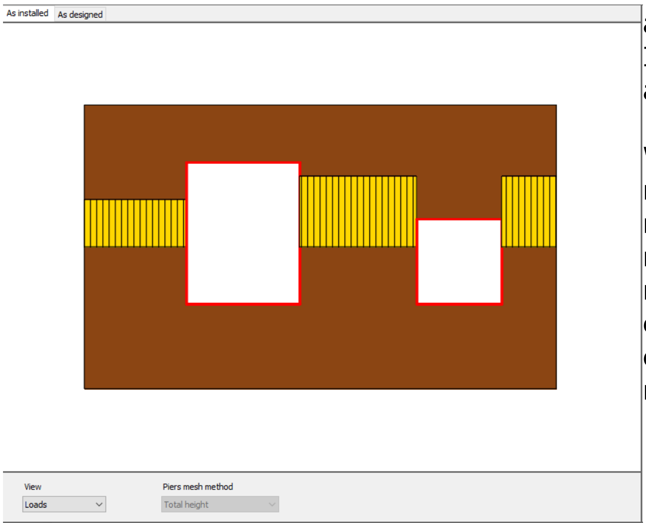
Graphic area, where the wall is represented and it is possible to select or add new elements.



Characteristics of the panels, where it is possible to add a new panel of the indicated characteristics.
By clicking the "<" or ">" buttons the new panels will be added, respectively, to the left or to the right.
By selecting a panel from the graphics area, it is possible to delete it or modify its mechanical and geometric characteristics
The height h can be only one for the whole wall.

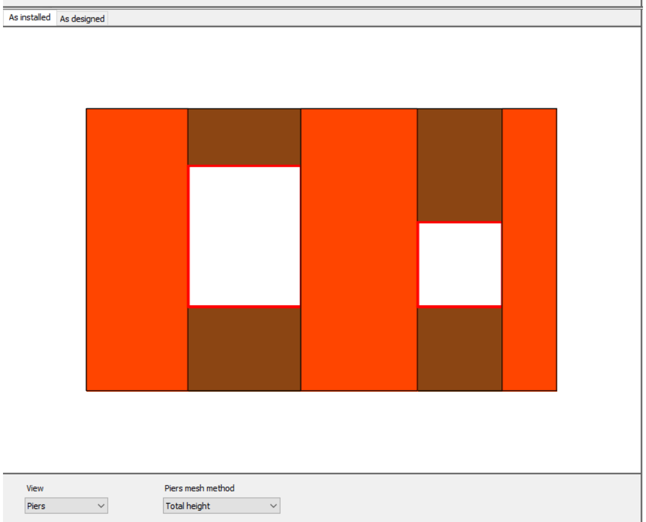


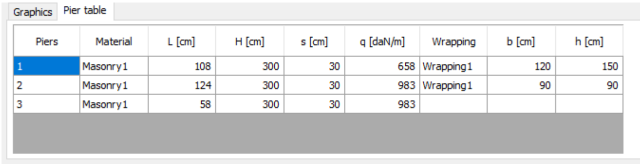
Opening characteristics, where it is possible to add a new opening of the indicated characteristics.
The position of the opening can be defined or modified from the graphics area.
By selecting an opening from the graphics area, it is possible to delete it or modify its characteristics.
Openings with a circle are shown with a red outline.



Display and definition of the calculation mesh, where it is possible to select the calculation method of the masonry walls and display them in the graphics area.
It is also possible to visualize the loads acting on the wall.

When the wall originates from the 3d model, it is not possible to modify the mesh, as it is automatically made on the model analysis environment. For this reason the mesh modification options are enabled only in the case of stand-alone encirclements (not originated from the 3D model)





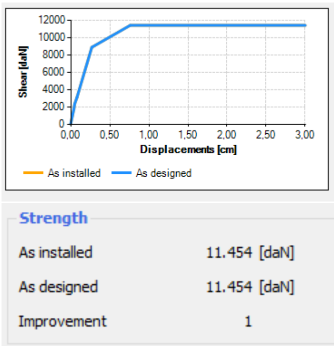
Piers	Material	L [cm]	H [cm]	s [cm]	q [daN/m]	Wrapping	b [cm]	h [cm]
1	Masonry1	108	300	30	658	Wrapping1	120	150
2	Masonry1	124	300	30	983	Wrapping1	90	90
3	Masonry1	58	300	30	983			

Closing the wall definition window, the characteristics of the calculation mesh of the masonry walls and the acting loads are shown in the summary table.

PLEASE NOTE: as long as the characteristics of the project status are not explicitly defined, this is created as a copy of the actual status.
After changing the project status, any further changes to the actual status will no longer be reported in the project status.

11.4.1.3.4 Calculation results

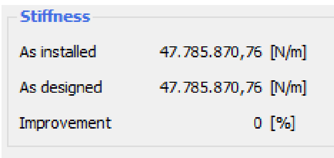
The verification outputs consist of four different outputs that allow you to:



Ensure that the strength of the design state is greater than the strength of the actual state (improvement factor greater than 1)
This check is considered binding upon passing the verification:

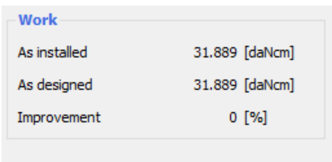
Vsdp / Vsdf> 1

in which:
Vsdp: design status strength
Vsdf: current status strength



Guarantee the containment of the variation of the stiffness between the actual state and the project state.

Normally there is no limit to this variation, alternative technical documentation suggests ± 15%
This check is not considered binding upon passing the verification but a suggestion.
Guarantee the containment of the variation of the work between the state of fact and the state of the project.



There is normally no limit to this variation.
In some practical cases, the insertion of new openings can change the breaking mechanics of the masonry elements, consequently producing even significantly different constitutive bonds of the wall.
In these cases, it is difficult to find standard profiles with characteristics such as to limit the variation of work in a fixed interval, for this reason it is good to try to design the hoops in such a way as to minimize the variation of work regardless of the extent of the same.

11.4.2 Steel lintel

11.4.2.1 Introduction

The Lintel Calculation module allows the verification of a steel lintel in bending and deformation against the stresses induced by the weight of the masonry and any overlying floors.
The profile being tested can be selected from a library of the most common profiles in use or, alternatively, its geometric and mechanical characteristics can be entered manually.

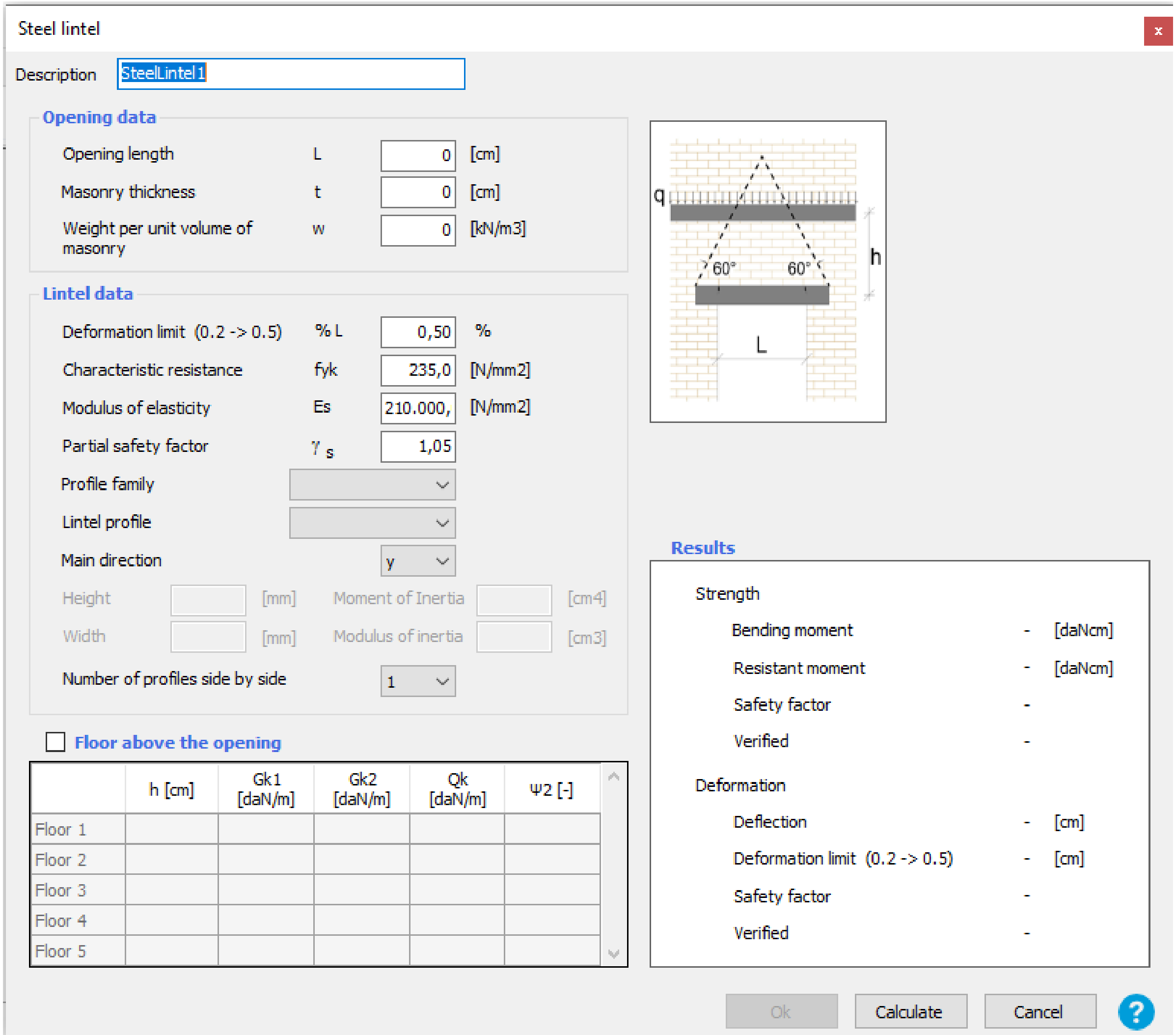
The masonry and any portions of the floor included in an imaginary equilateral triangle, built above the lintel itself, are considered to be weighing on the lintel; the consequent moment stress and deformation are calculated considering the static diagram of a simply supported beam.

11.4.2.2 Calculation procedure

The program considers loads acting on the lintel the weight of the masonry and the portions of any overlying slabs included in an imaginary equilateral triangle built above the lintel and on the side equal to the theoretical span of the lintel itself (equal to the width of the increased opening 5%).
The contribution of the masonry is considered as a triangular distributed load, while the contribution of the possible overlying floor is considered as a uniformly distributed load on the entire lintel, of an intensity equal to the value deriving from the analysis of the loads, reduced by the ratio between the width of the floor included within the aforementioned equilateral triangle and the theoretical span of the opening.
With formulas from construction science, the acting moment and the deflection are then evaluated, compared with the maximum resisting moment - calculated with the pressure-bending formula by setting the maximum stress for calculating the profile of the architrave - and the limit deflection - calculated as an aliquot. of the opening span.
The relationship between the limit values and the acting values return the respective safety coefficients.

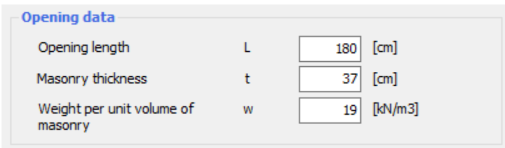
11.4.2.3 Work environment

The working environment is as follows:



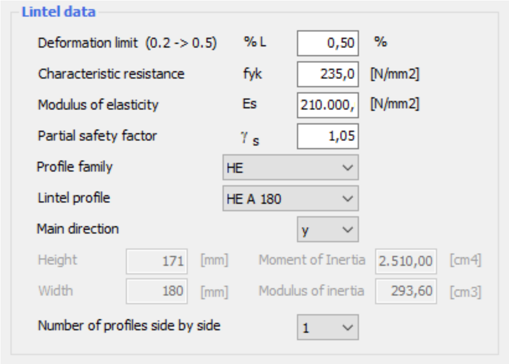
11.4.2.3.1 Main area

The window is divided into four sections.



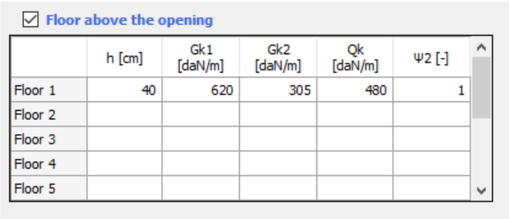
<u>1. Opening and masonry data</u>

Indicate in this section the net span of the opening, the thickness of the masonry and the weight per unit and volume of the masonry itself.



2. Lintel data

In this section it is possible to vary the values of the limit deflection, characteristic resistance, elastic modulus and partial safety coefficient of the lintel.



3. Overlying floor data

In the event that there is a floor above the opening whose weight weighs on the masonry affected by the opening, its effects can be taken into account by activating the relative option.

It is therefore necessary to enter:

- the opening extrados-ceiling intrados distance
- the permanent structural load G1
- the permanent non-structural load G2
- the variable load Q
- the combination coefficient ψ2 (for the calculation of the quasi-permanent combination of loads)

Obviously, only the portions of the load bearing on the masonry affected by the hole will be inserted.

Only the floors included in an imaginary equilateral triangle with a side equal to the theoretical span of the lintel, built above the opening, have effects on the lintel. The program recognizes through the parameter h if, and to what extent, to consider the effect of the floor.

4. Analysis results

In this section, the following results are printed:

- the total stressing moment, including the contribution of the masonry and any overlying floor, Msd;
- the resistant moment Mrd;
- the safety factor given by the ratio between Mrd and Msd;
- the calculation deflection f, due to the actions on the lintel;
- the deflection limit, given by the value entered in section 3;
- the safety factor, given by the ratio between flim and f.

11.4.3 Masonry lintel

11.4.3.1 Introduction

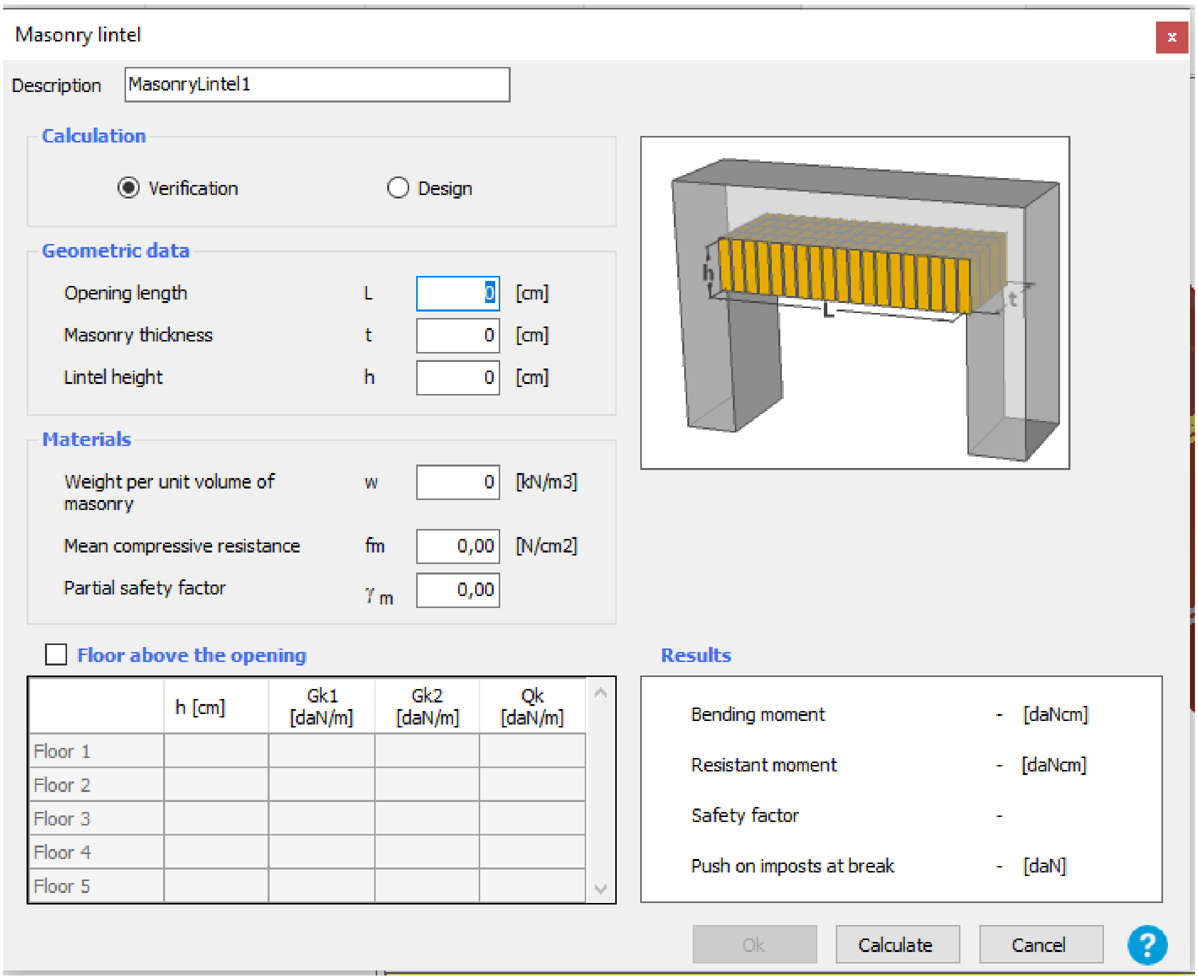
The Masonry Lintel Calculation module allows the verification of a bending lintel against the stresses induced by the weight of the masonry and any overlying floor.
The masonry and any portion of the floor included in an imaginary equilateral triangle, built above the lintel itself, is considered to be weighing on the lintel; the consequent moment stress and deformation are calculated considering the static diagram of a simply supported beam.

11.4.3.2 Calculation procedure

The program considers loads acting on the lintel the weight of the masonry and the portion of any floor above, included in an imaginary equilateral triangle built above the lintel and on the side equal to the theoretical span of the lintel itself (equal to the width of the opening increased by 5%).
The contribution of the masonry is considered as a triangular distributed load, while the contribution of the possible overlying floor is considered as a uniformly distributed load on the entire lintel, of an intensity equal to the value deriving from the analysis of the loads, reduced by the ratio between the width of the floor included within the aforementioned equilateral triangle and the theoretical span of the opening.
With formulas from construction science, the acting moment is then evaluated, compared with the maximum resistant moment.
The relationship between the limit values and the acting values return the respective safety coefficients.
By selecting the design mode, the program returns the required height of the lintel to satisfy the verification.

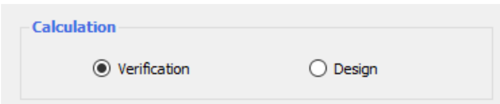
11.4.3.3 Work environment

The working environment is as follows:



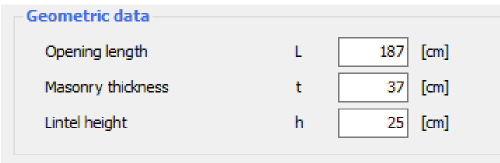
11.4.3.3.1 Main area

The main area is divided into 5 sections.



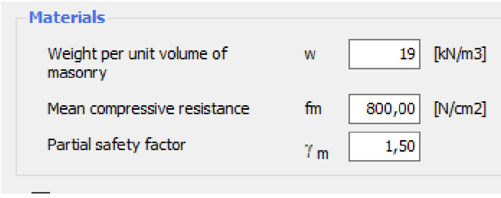
1. Type of calculation

By selecting the design mode, the box for entering the height of the lintel h is blocked, which will be calculated by the program in order to satisfy the limb check.
On the other hand, by selecting the verification mode, the box for entering the height of the lintel *h* will be active and must be filled in by the user.



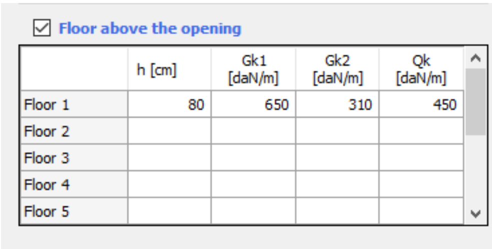
2. Geometric data

In this section indicate the dimensions of the opening and the characteristics of the lintel.



3. Materials

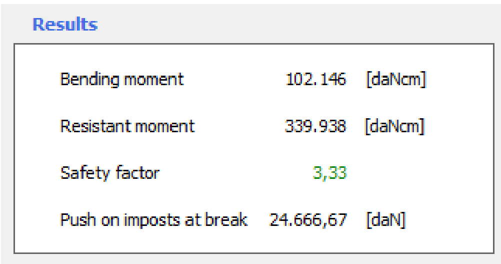
Enter the mechanical characteristics of the masonry.



4. Floor above the opening

In the event that there are floors above the opening whose weight weighs on the masonry affected by the opening, the effects can be taken into account by activating the relative option.
It is therefore necessary to enter:
- the opening extrados-ceiling intrados distance Δ
- the permanent structural load G_1
- the permanent non-structural load G_2
- the variable load Q

Only the portions of the load bearing on the masonry affected by the hole will be inserted.
Only the floors included in an imaginary equilateral triangle with a side equal to the theoretical span of the architrave, built above the opening, have effects on the lintel. The program recognizes through the parameter h if, and to what extent, to consider the effect of the floor.



5. Results of the analysis

In this section, the following results are printed:
- the total stressing moment, including the contribution of the masonry and any overlying floors, $\mathbf{M_{sd}}$;
- the resistant moment $\mathbf{M_{rd}}$;
- the safety factor given by the ratio between M_{rd} and M_{sd};
- the thrust on the shutters, exerted by the architrave due to the arch effect, at the moment of breaking.

11.5 Floors

11.5.1 Steel beams and vaults floors

11.5.1.1 Introduction

The Steel beam+vault floor module performs the verification of floors made with steel profiles type IPE and HE and vaults.
The contribution to the strength of any concrete layer is not taken into account in the calculation.

11.5.1.2 Calculation procedure

The calculation of the strength and deformation of the steel beam is carried out as indicated in the Ministerial Decree 17-01-2018.
The program allows the verification of sections of classes 1, 2 and 3, as defined in paragraph 4.2.3.1. The classification is done automatically by the program itself.
The check for class 1 and 2 sections is performed by default in the plastic field, but it is possible to limit it to the elastic field.
Class 3 sections, on the other hand, can only be calculated in the elastic field.

$$V_{c,Rd} = \frac{A_v \cdot f_{yk}}{\sqrt{3} \cdot \gamma_{M0}}$$

In case of section stressed only in shear, the simple shear check is performed where Av is the shear resistant area calculated according to the expressions reported in paragraph 4.2.4.1.2 of the Ministerial Decree
In the calculation of the resisting moment the reduced yield stress (1-ρ) fyk is considered where ρ:

$$\rho = \left[\frac{2V_{Ed}}{V_{c,Rd}} - 1 \right]^2 \qquad [4.2.31]$$

It can be limited to the elastic range only or extended to the plastic range for class 1 or 2 sections.
The limb check is subordinated to the simple shear check.
The resistant moment is evaluated as:

MV, Rd = min (Mv, Mrd)

where:

- Mv is the reduced resisting moment for [Wpl -1 / (4 nw tw) (r Av2 + NEd2 / (1- r) / fy2)] fd

The safety coefficient for vertical overloads ζv, i is also calculated, defined in paragraph C8.3 of the explanatory circular of NTC18 as the ratio between the maximum value of the variable vertical overload that can be tolerated by the i-th part of the building and the value of the vertical overload variable that would be used in the design of a new building.

The verification is performed against the stresses induced by the ULS combination of loads.

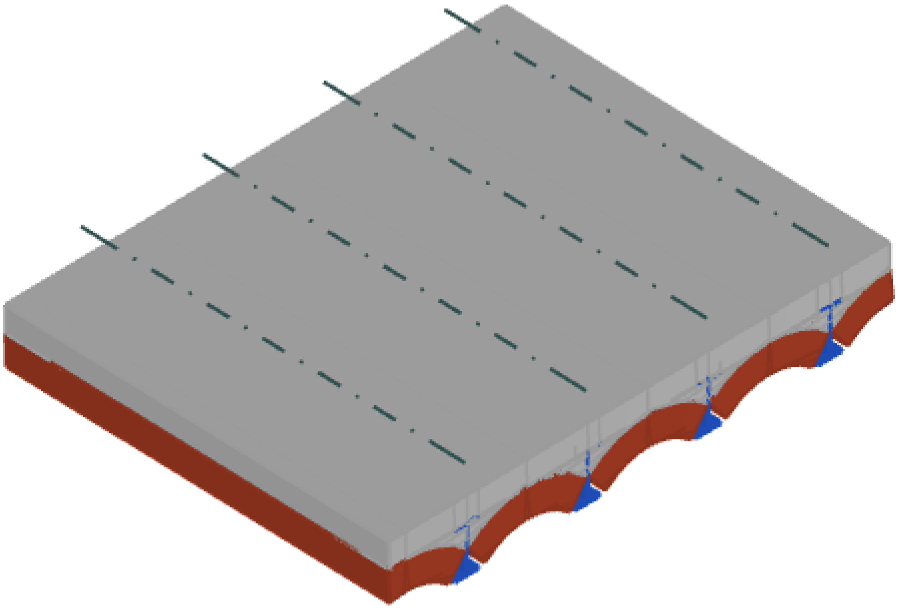
11.5.1.3 Work environment

Access to the verification form can be done in two different ways:

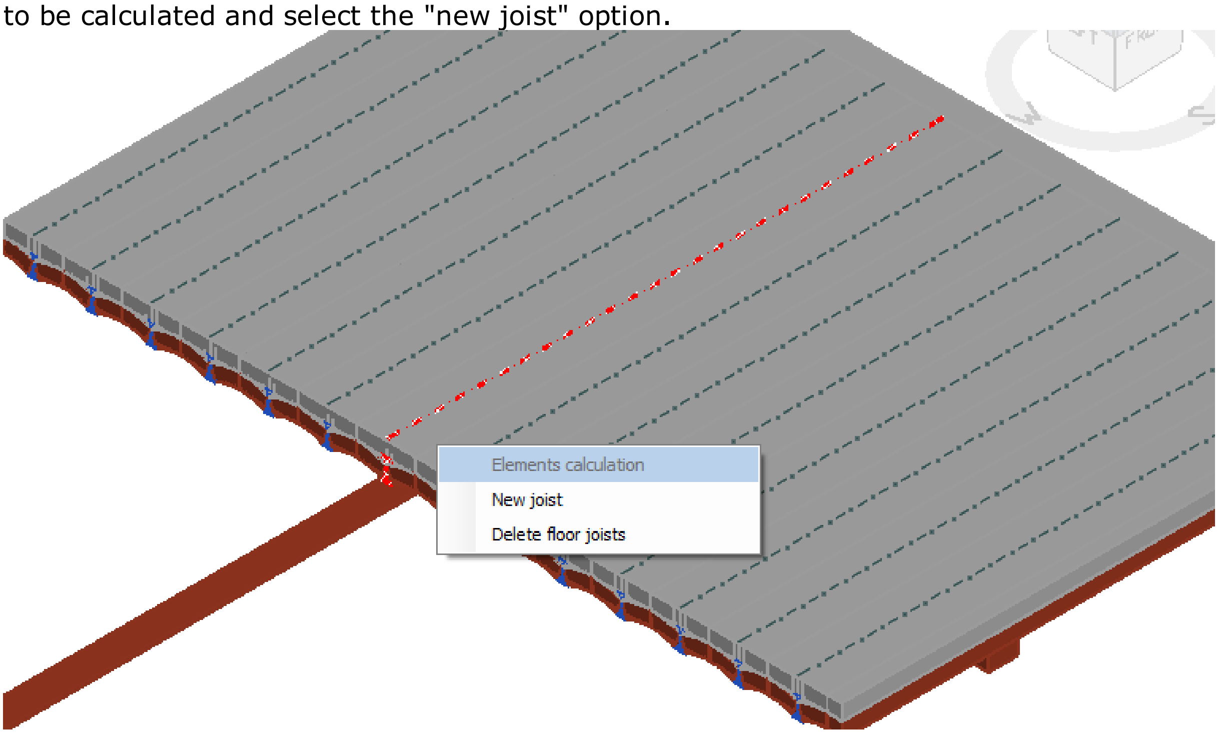
- Using the insert menu, which allows you to conduct a check disconnected from the structural objects present within the 3D model;
- Graphically, by selecting the element of interest in the graphics area.

To be able to interact with the graphic element, just click with the right mouse button on the floor element and choose the option "Joist generation".

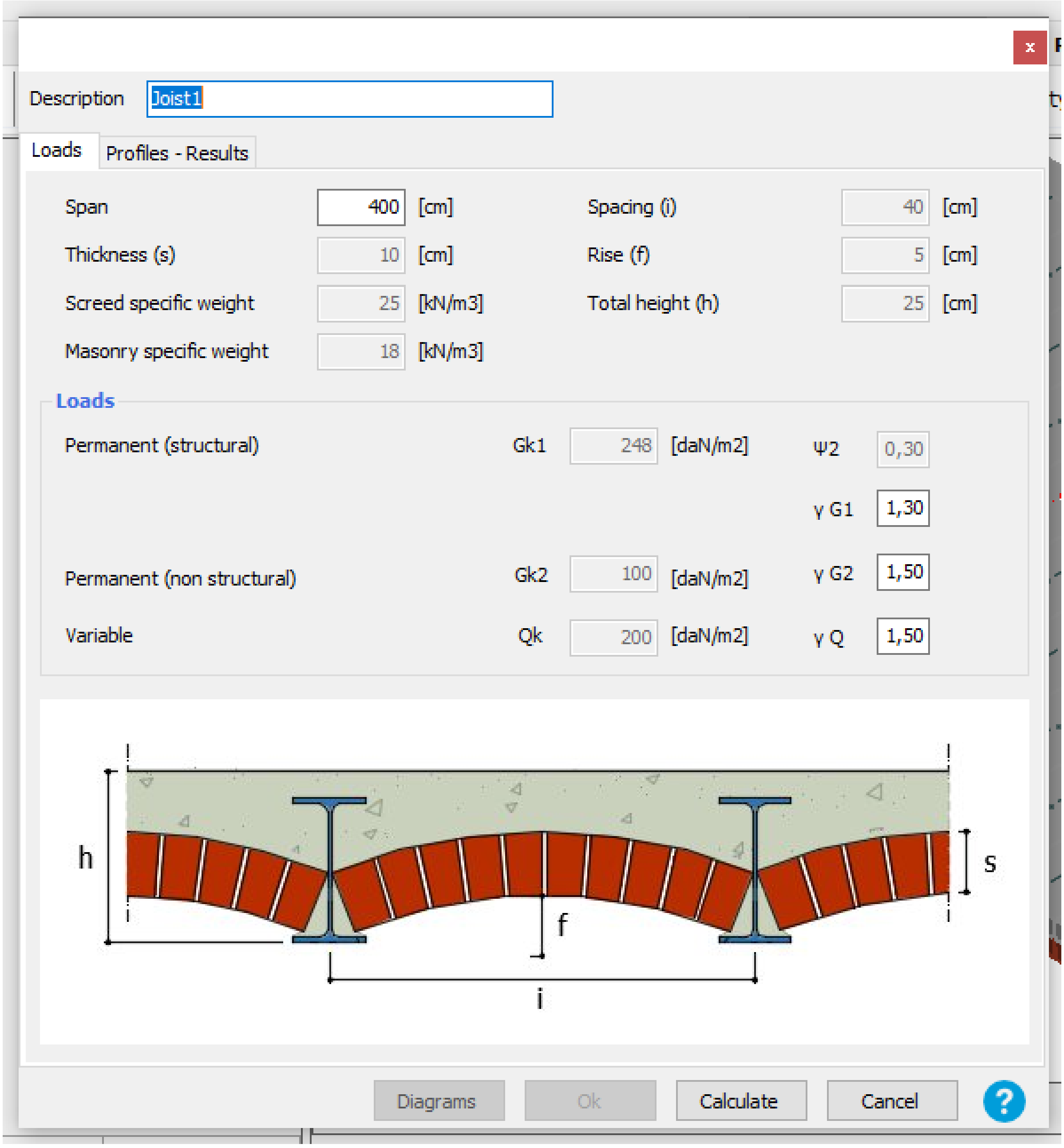
This will draw the slab inside the 3D model.



To continue with the calculation, simply click with the right mouse button on the element to be calculated and select the "new joist" option.



The main working environment is as follows:



The environment is divided into the following two tabs:

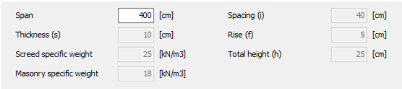
- Loads
- Profiles - Results

11.5.1.3.1 Loads

The tab is divided into the following two sections:

User mode [U] **Model mode [M]**

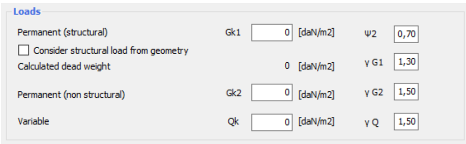
Geometric data



The section provides for the insertion of the main geometric data. The insertion of all the data will allow the automatic calculation of the own weight.
If, on the other hand, the user wants to manually enter the permanent structural loads, the data relating to the span and spacing will suffice.
The light L is to be included in the calculation value (net span).

The geometric data requested are automatically determined by the model, according to the characteristics assigned within the structure environment.

Loads



The section provides for the insertion of the distributed loads acting on the element considered.
It is possible to define permanent structural, permanent non-structural and variable loads.
The user can choose, by selecting the "Consider structural load from geometry" box, to automatically calculate the permanent structural loads on the basis of the geometric data and the materials entered; by unchecking this box, the user can, on the other hand, manually enter the permanent structural loads; in the latter case, the self-weight calculated automatically by the program will not be taken into account.
In the calculation of the combined stresses the variable distributed and concentrated loads

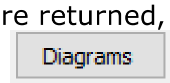
The loads are automatically defined starting from the model, calculated on the basis of the parameters assigned within the structure environment.
You can always manually change the suggested values for the combination coefficients.

are considered simultaneous, therefore they refer to the same combination coefficient.
It is also possible to customize the safety and contemporaneity coefficients for the combination of loads.

11.5.1.3.2 Results

The geometric and mechanical data of the profile are entered in this tab.
The beam section can only be selected from profiles stored in the program library.
Type profiles are available:
- HE A
- HE AA
- HE B
- HE M
- IPE

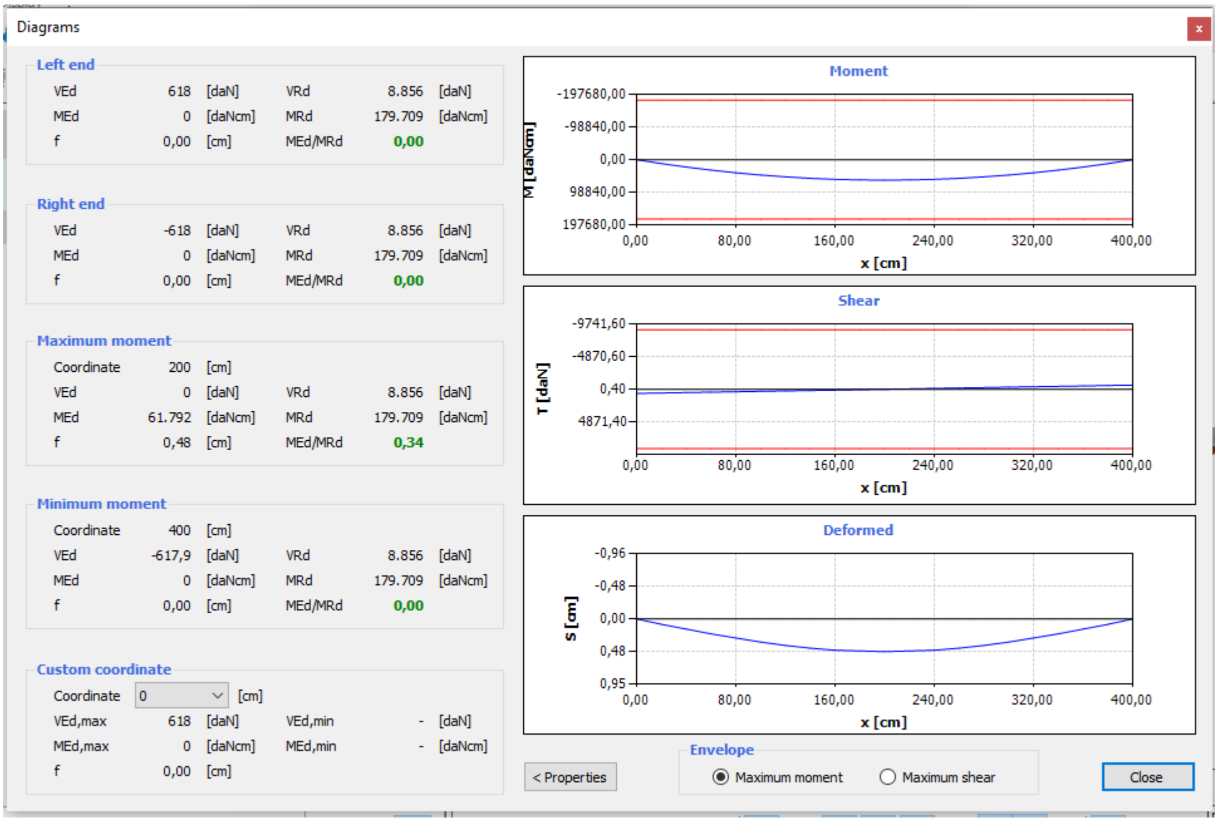
The geometric characteristics and the section scheme are displayed for the selected profile.
In this section it is also possible to choose whether to limit the calculation of the resistance of the section to the elastic range, the mechanical characteristics of the material and the partial safety factor.

Furthermore, the main calculation results are returned, with the possibility of accessing the stress diagrams by means of the button 

.
As regards the strength verification, the results show the stressing moments MEd and resisting moments MRd in the most stressed sections of coordinate x, and the relationship between stress and resistance, representing the efficiency of the section. As illustrated in the Calculation Procedure paragraph, the resisting moment can be decreased by the shear action, therefore the results relating to the envelope of maximum moment and maximum shear in the ULS combination of stresses are reported separately.
As regards the deformation, the most deformed section x, the value of the lowering f and the ratio with the net span L / f are indicated.
More information on the results can be viewed in relation and in the Diagrams.

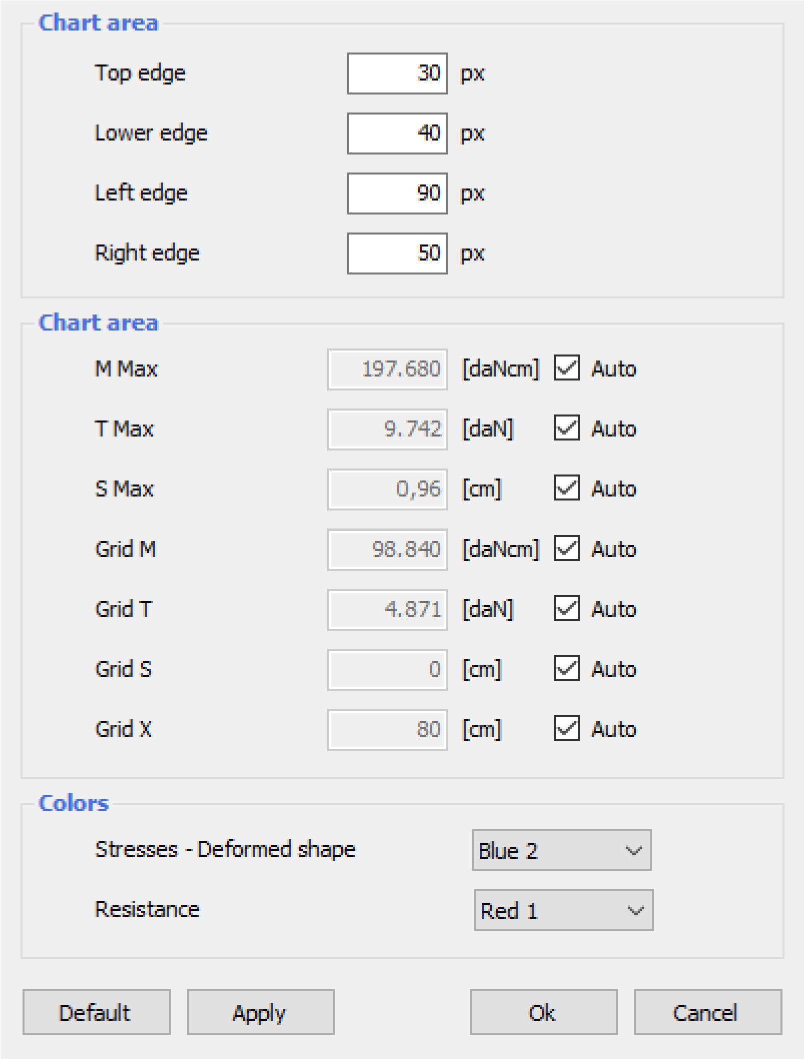
11.5.1.3.2.1 Diagrams

The diagrams area displays moment, shear and deformation diagrams.
The results of the main sections are displayed on the left side of the screen; it is also possible to view the results of a generic section by selecting it in the Custom section box.
It is possible to choose whether to display the results relating to the moment envelope or the maximum shear envelope. However, both results will be reported in relation.



Using the button 

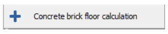
it is possible to customize the graphs in margins, grids and colors.



11.5.2 Concrete brick floor

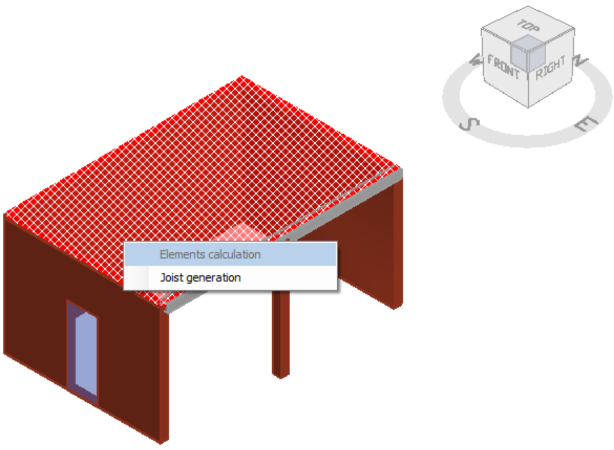
11.5.2.1 Creation of a calculation tab

The menu dedicated to the floors has several options including:

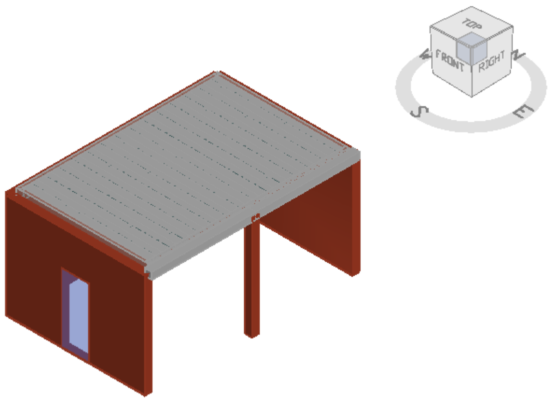


It allows you to create a new calculation tab. By selecting this option, the card will be created as a user card.

It is also possible to create a calculation sheet starting from a slab inside the model and carry out the following steps:

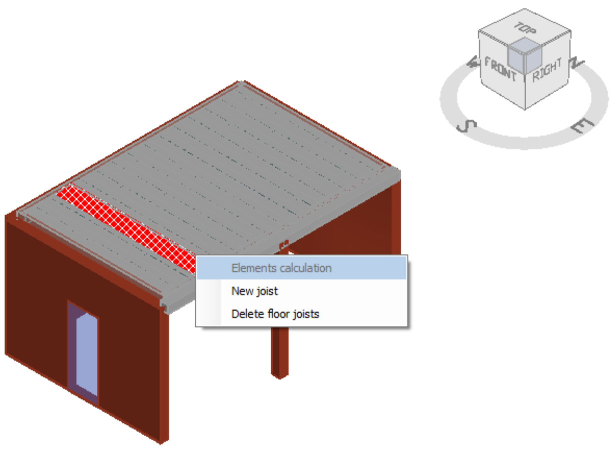


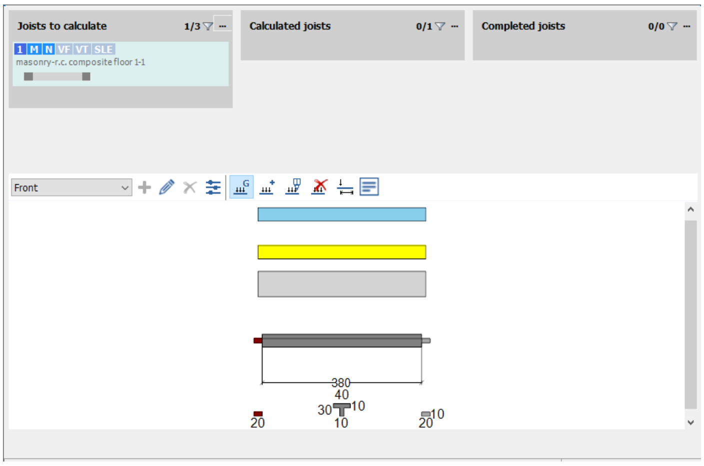
Select the desired floor surface with the right mouse button and select the option "Joist generation" from the menu.
This option allows you to switch from drawing the simple surface of the floor to the three-dimensional drawing of the same.



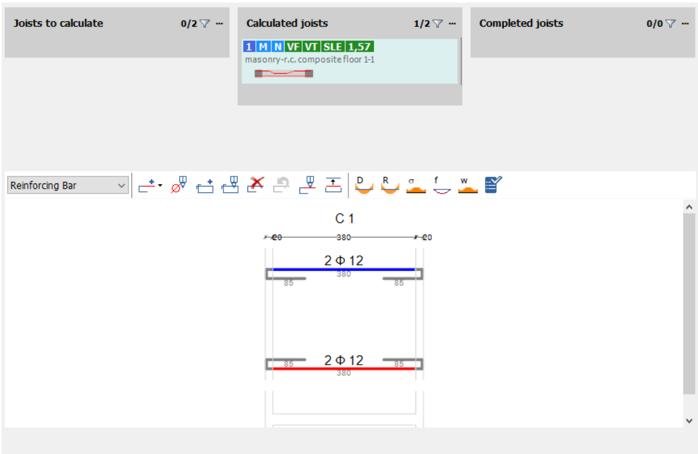
Once the three-dimensional floor is obtained, it is possible to individually select each joist that is part of the floor.
Clicking on the element with the right mouse button, the contextual menu appears from which it is possible to choose the option "New joist".

ATTENTION:
By choosing the "Delete floor joists" option, the three-dimensional drawing of the floor and, with it, all the associated joist cards are deleted.





The calculation window opens and the sheet is created and placed in the "Joists to calculate" column.



To calculate the joist simply drag the tab into the "Calculated joists" column.

The information labels on each sheet visually summarize the results of the checks carried out.

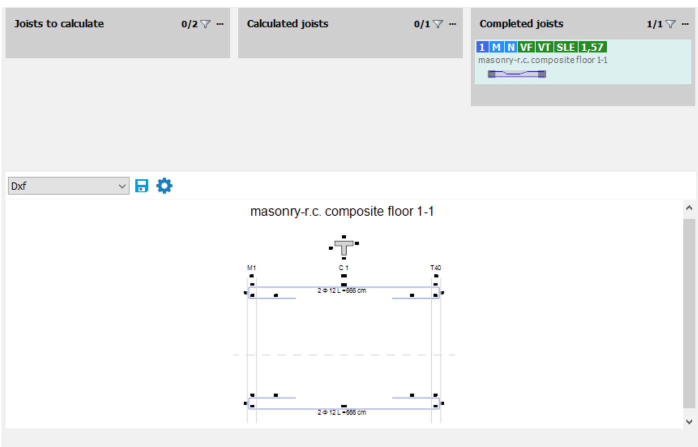
VF Green if the bending check passed. Red otherwise.

VT Green if the shear check is passed. Red otherwise.

SLE Green if the SLS test passed. Red otherwise.

1,02 Value of the safety factor

! Indicates that there are warnings regarding the calculation made. To view them, simply place the pointer over them.



Once the calculation is finished, by dragging the tab in the "Completed joists" column, it is possible to produce the output files of the calculation.

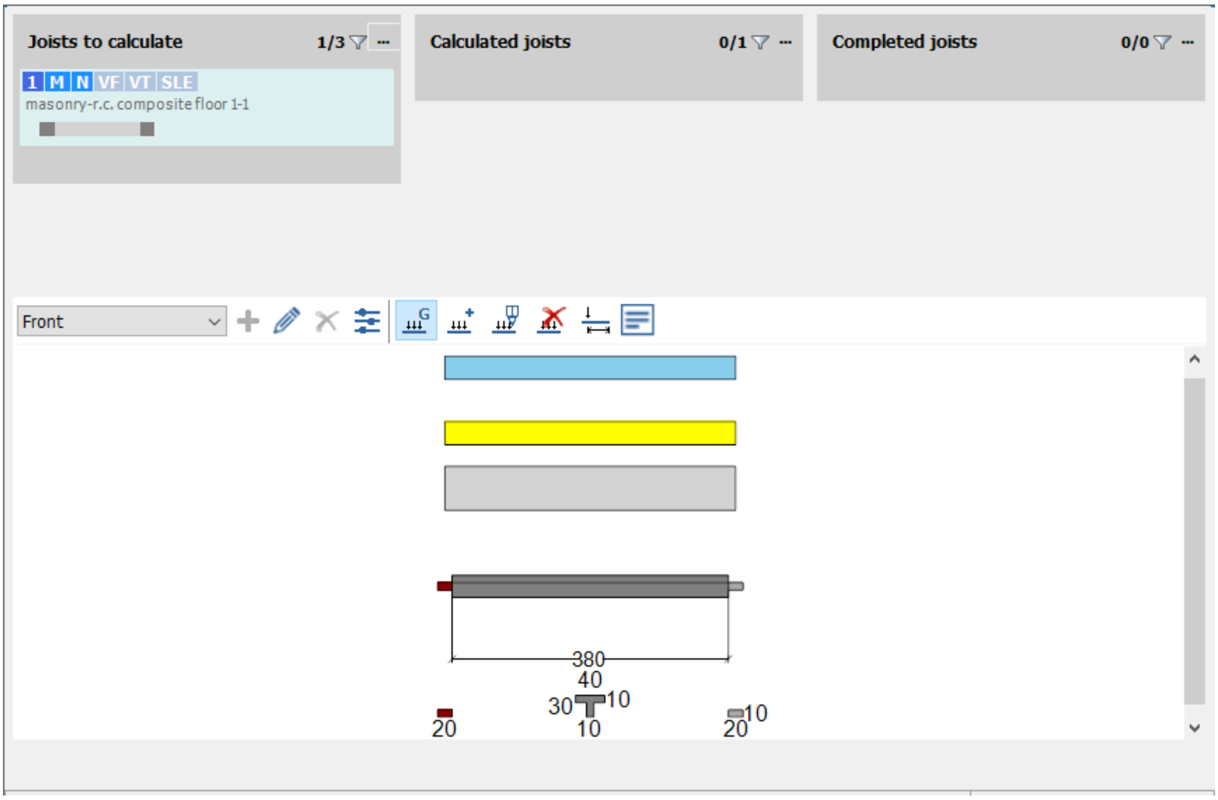
11.5.2.2 Codes

The reference code is the same as reported in the case of beams.

11.5.2.3 Work environment

11.5.2.3.1 Front interaction

The "Front" view allows you to interact graphically with the elements using the commands in the top bar.



The command bar has several possible configurations depending on the position of the calculation card within the columns.

11.5.2.3.1.1 Front configuration

<u>"Front" configuration</u>

It allows to modify geometries, loads and calculation parameters.

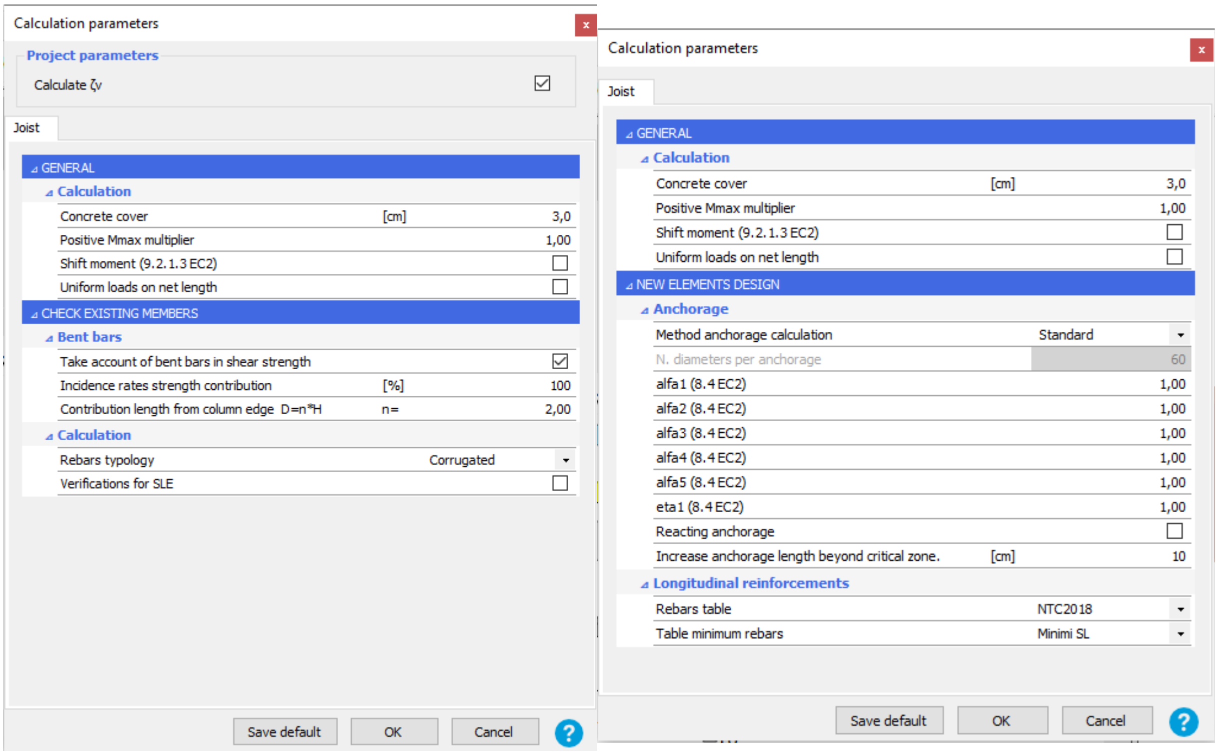
 "Add": allows you to add new spans;

 "Modify": allows you to modify the selected span;

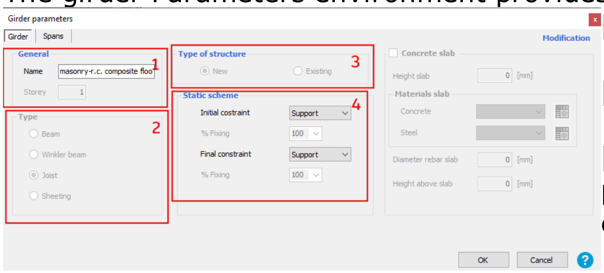
 "Delete": allows you to delete the selected spans;

- "Calculation parameters": allows you to set / modify the calculation parameters;
- "Results": allows you to view the results of the calculation.
- "Dead weight": allows you to add the load corresponding to the own weight on all spans;
- "Add load": allows you to add new loads to the selected spans;
- "Edit load": allows you to modify the selected loads;
- "Delete load": allows you to delete the selected loads;
- "Load dimensions": allows you to view / not display the dimensions relating to the loads;
- "Legend of loads": activates the display of the load legend.

By pressing the "Calculation parameters" button it is possible to customize the calculation settings regarding the beams and reinforcements, with the possibility of setting a specific configuration as the default one.
The possible parameters vary according to the type of structure, new and existing.



The girder Parameters environment provides:



[1] Entry of general data;

[2] Object type set on "joist".

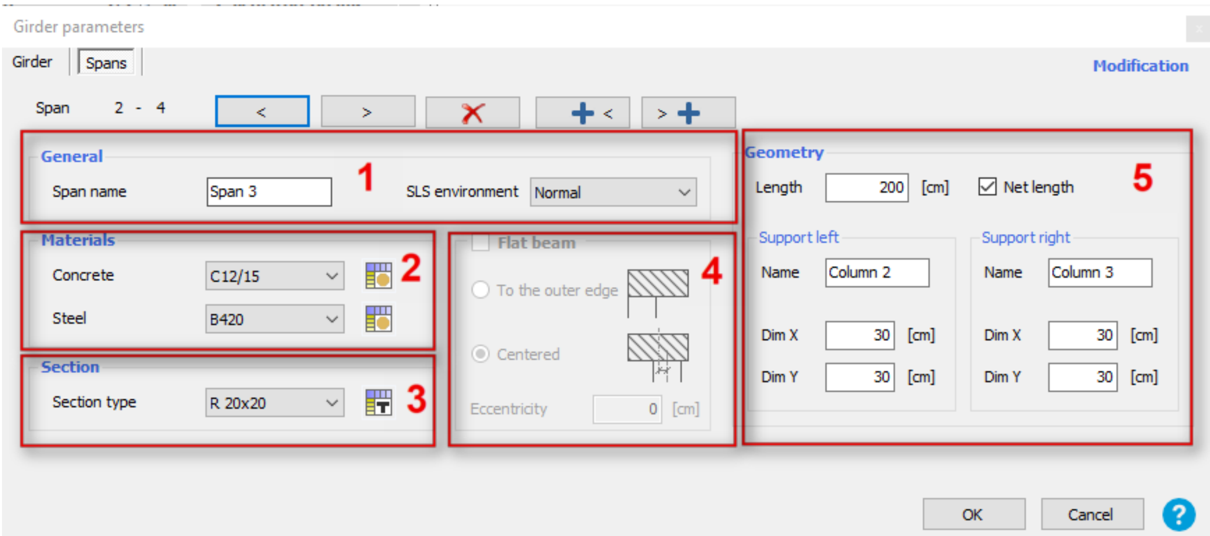
[3] Setting the type of structure, with the possibility of choosing between new and existing;

[4] Definition of the static scheme, with the

possibility of setting the interlocking percentage if the latter is set as a constraint on at least one of the two extremes;

Add span

After clicking on the "add" button, the following window appears on the screen, in which it is possible to add further spans.



[1] General

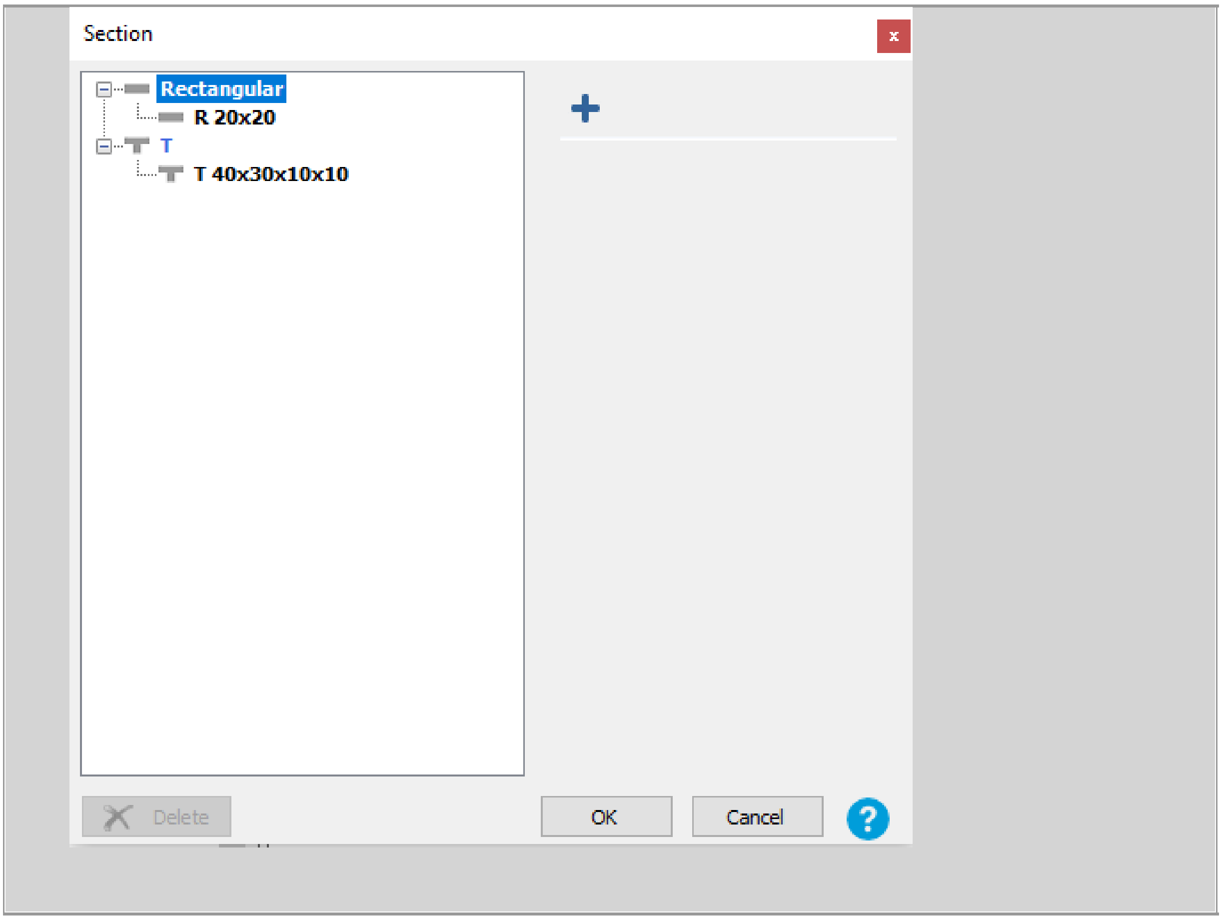
The section provides for the insertion of general data: in particular the name of the span and the aggressiveness of the environment for SLS checks

[2] Materials

The section provides for the definition of materials for concrete and steel, with the possibility of accessing the respective lists.

[3] Sections

The menu provides for the definition of the section of the span: it is possible to choose between rectangular and T-shaped sections; the insertion of the geometric data of the section is accessed via +.



[4] Geometry

The Geometry section provides for the insertion of the span span (which will be understood according to the criterion selected in the Girder Parameters environment), the geometric data and the names of the columns at the ends.

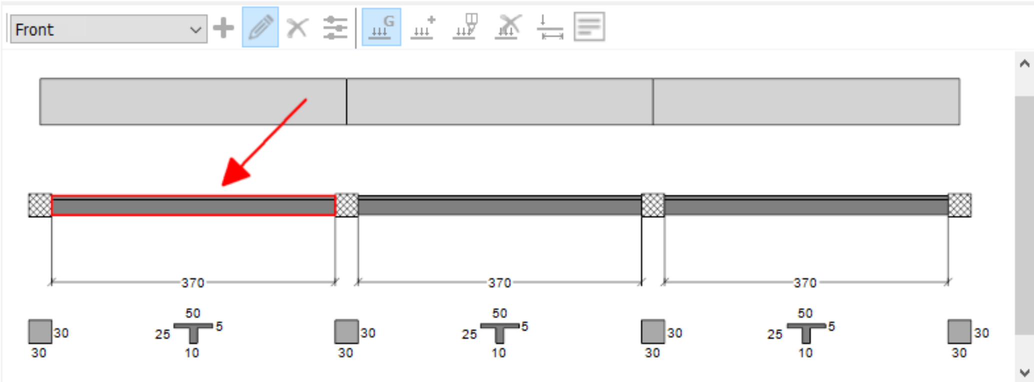
Once all the necessary data has been entered, the position of the mouse cursor (without clicking) on one of the two buttons causes the span to be created.
To insert other spans (with the same geometric and mechanical data or different, after modification), simply click on one of the two buttons.

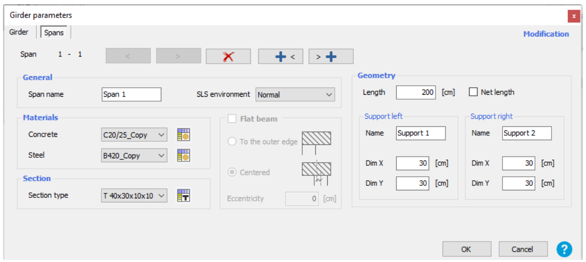
They allow you to select the adjacent span (left or right).

Allows you to delete the selected span.

Modify span

It is possible at any time to modify the data entered for the span in question.
To do this, simply click on the "edit" button and select the desired span in the drawing:

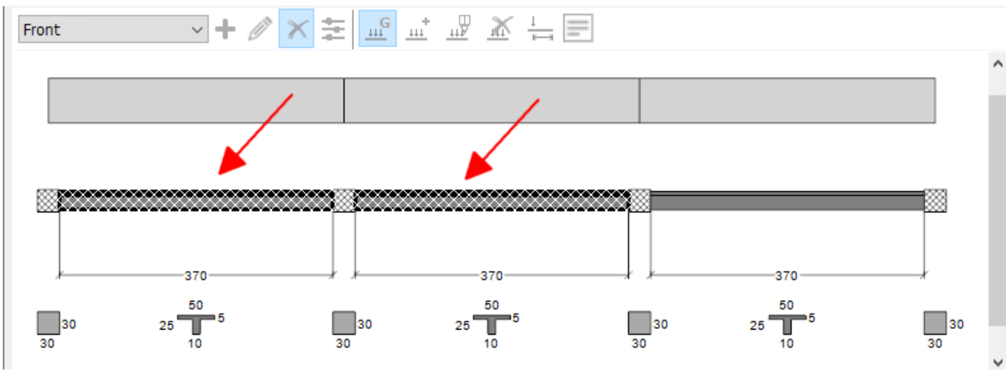




The window that summarizes all the parameters defined for the selected span is displayed.
From here it is possible to edit all the desired parameters and press "OK" to confirm.

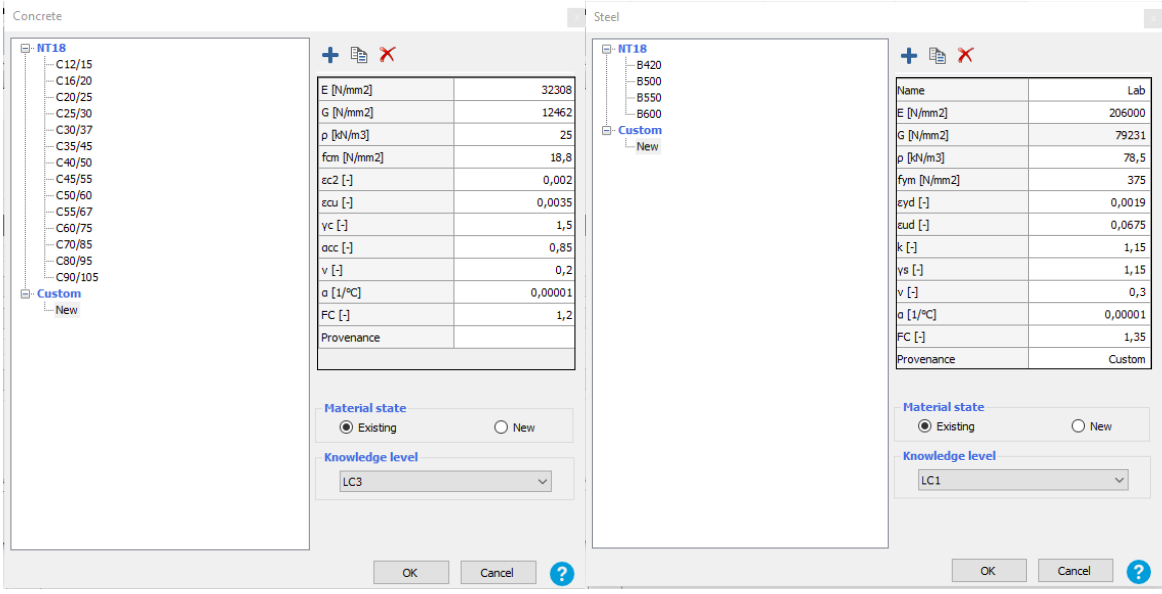
Delete span

It is possible at any time to delete one or more inserted fields.
To do this, simply click on the "delete" button and select the desired spans in the drawing:



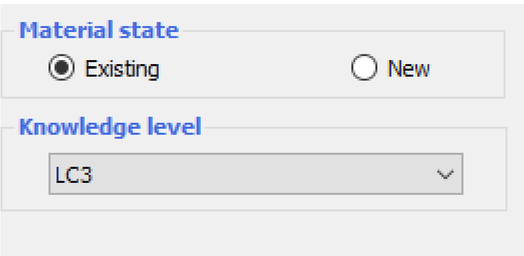
By pressing the right mouse button I confirm the deletion.

The window shows the list of materials for concrete, steel, sheet steel (NT18), reinforcements (NT18) or thermal protection (NT18) present in the work; on the right side of the window, on the other hand, the mechanical data of the selected material are shown.
There is already a list of the most commonly used materials and these cannot be changed.



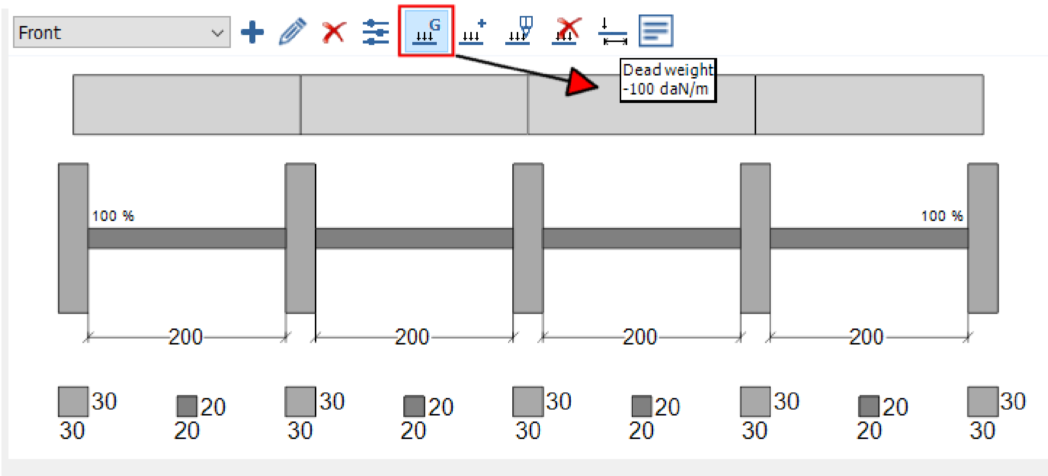
Each material can be duplicated using the command and the copy will be shown in the custom list, where it can be selected and modified using the data table on the right.

Once the customized list has been selected, it will also be possible to insert a new material (for example if laboratory data on existing materials are available) using the command ; in this case it is also possible to specify that it is existing material and the level of knowledge (LC1, LC2, LC3):



The type of load includes:
- linear load;
- concentrated load;
- surface load.

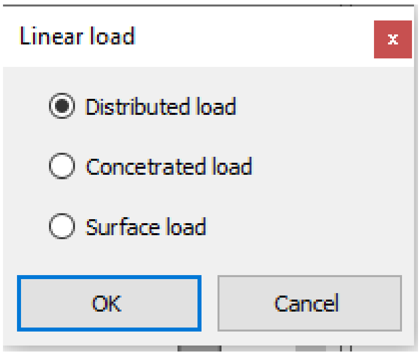
By pressing the "Dead weight" button it is possible to quickly define, on all spans, the self-weight of the element calculated automatically:



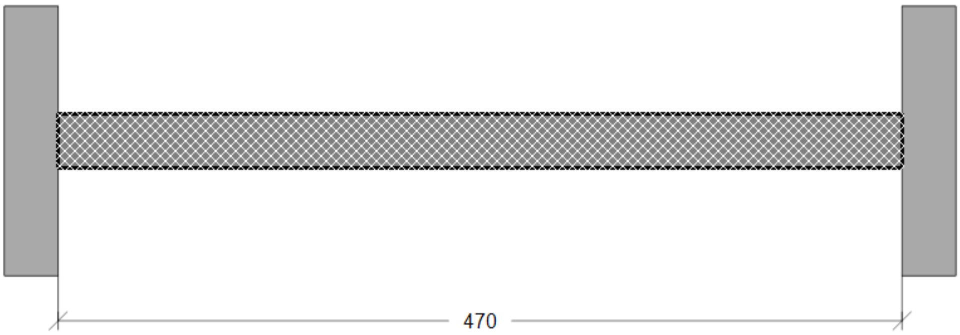
Simply reselect the button to eliminate the load entered due to its own weight.

The addition allows the insertion of a load.

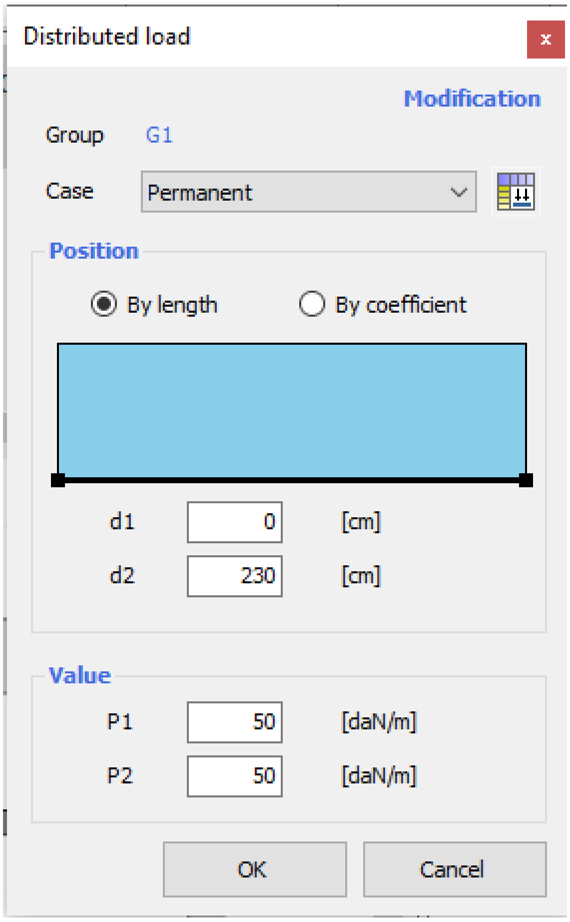
The command first requires the choice of the type of load to be inserted:



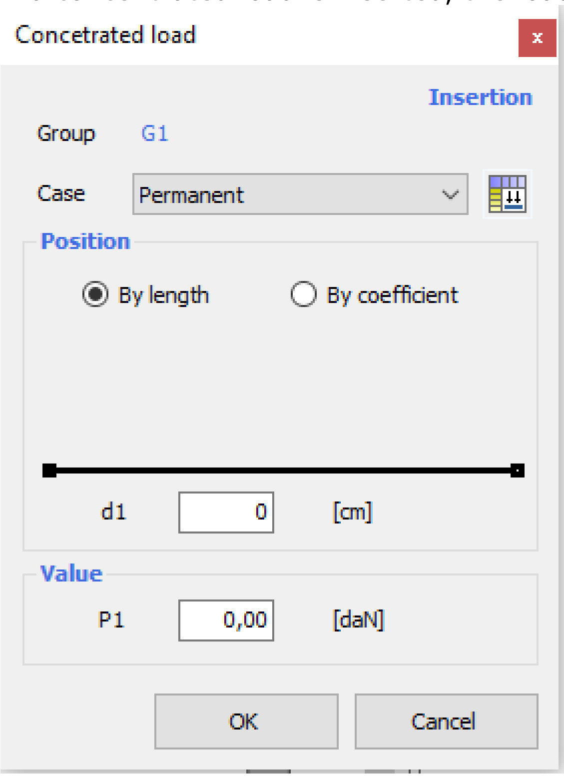
Then it is simply necessary to select the bay to be loaded with a click and confirm the choice by clicking on the right mouse button again:



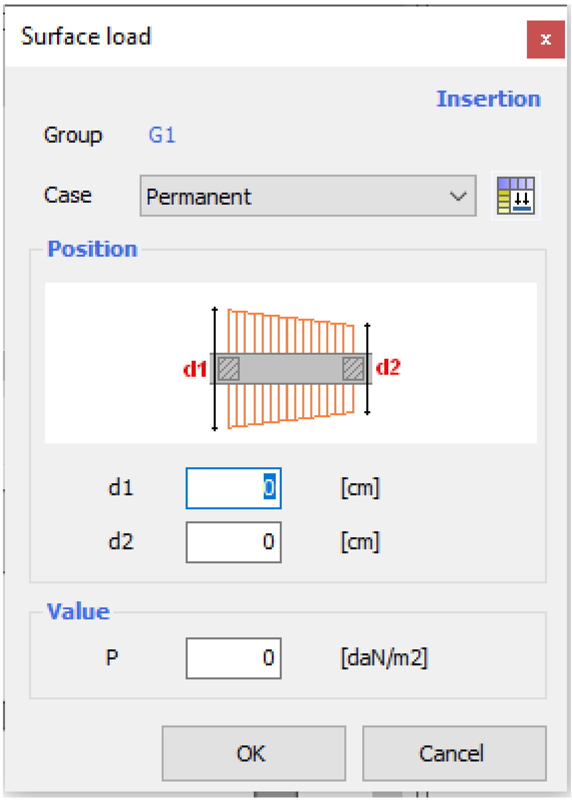
If a distributed load is entered, the load is set through the following menu.



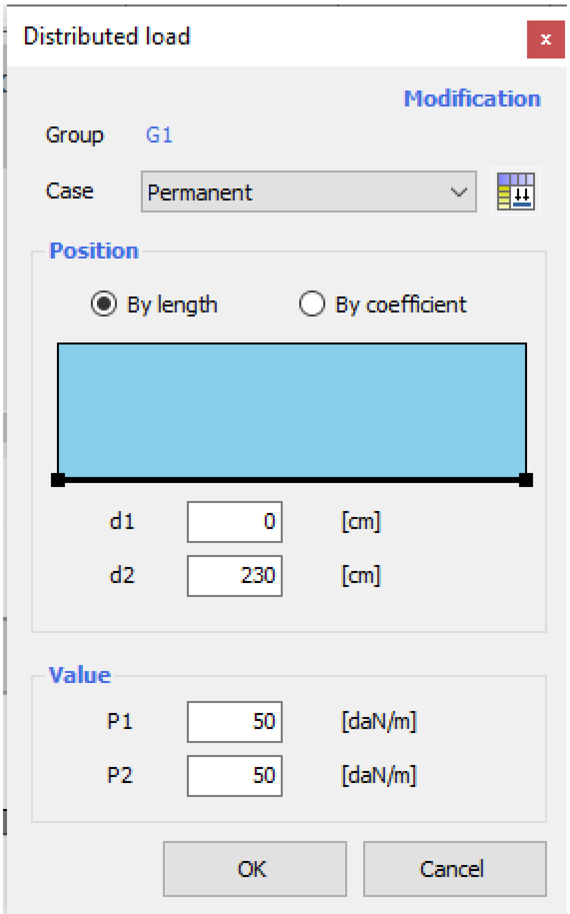
If a concentrated load is inserted, the load is set through the following menu.

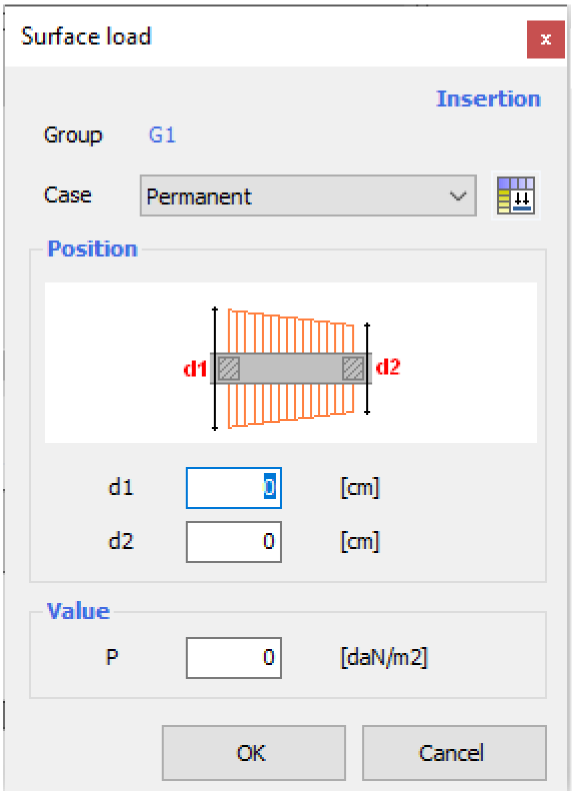


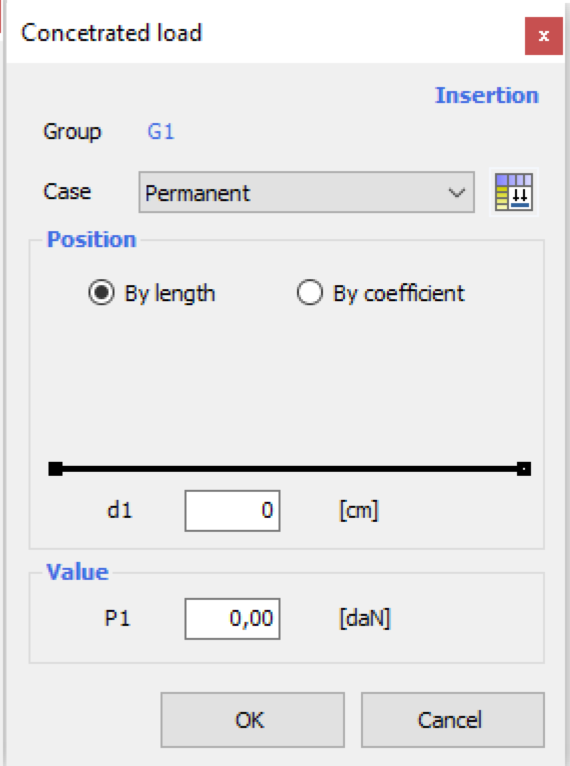
If a surface load is entered, the load is set through the following menu.



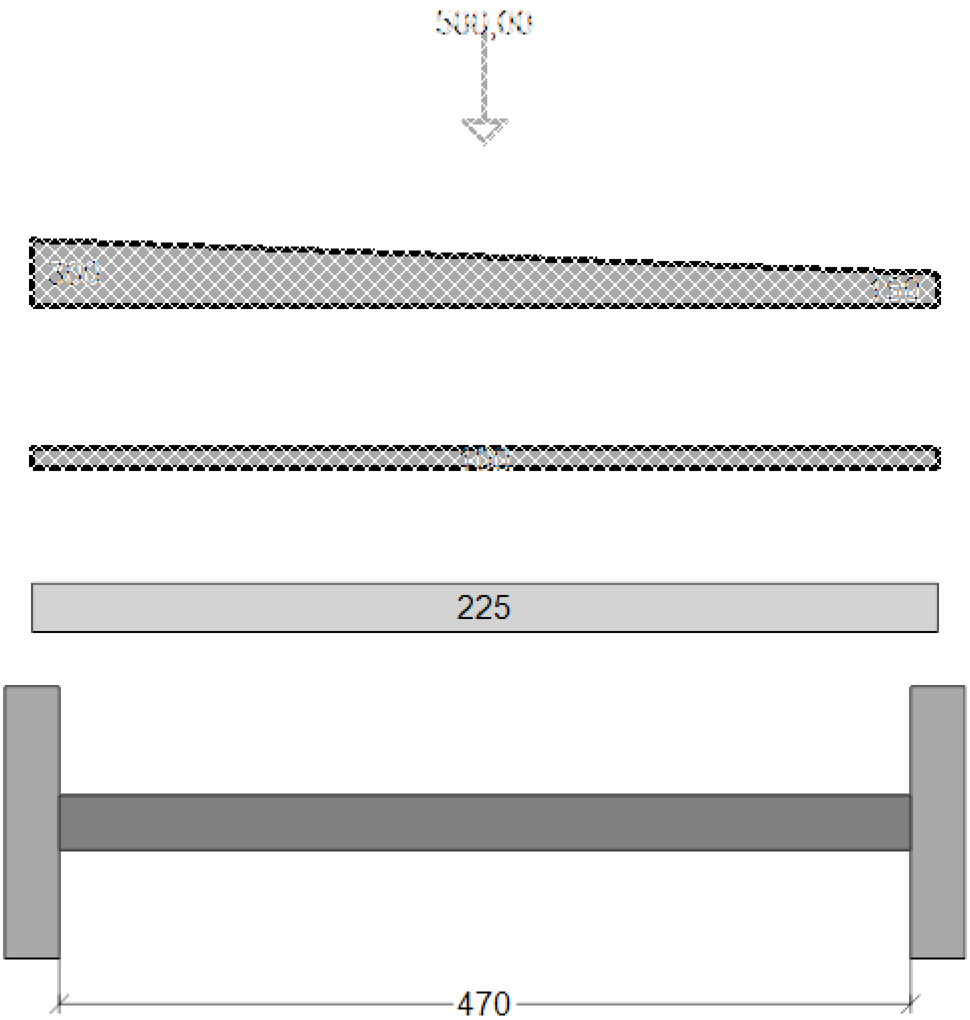
To edit an added load, simply select the load to be modified with a click and set the new parameters using the appropriate menu corresponding to the type of load for which the modification is necessary.







To delete an existing load it is necessary to select the load to be deleted with a click:



And finally confirm the deletion with a right click.

11.5.2.3.1.2 Reinforcing bar configuration

"Reinforcing bar" configuration

It allows you to insert / modify the reinforcement and graphically display the results of the calculation through diagrams.

 Stresses: Displays the stress diagram.

 Verifications for ULS: Displays the ULS check diagrams.

 Add profile: allows the manual insertion of new longitudinal reinforcing rods or by means of a template; to do this, place the virtual shape on the elevation that is activated after clicking on the icon; once positioned, a window will open in which the user will have to enter the number and diameter of the reinforcements to be inserted and will have to choose whether to consider the anchoring of the new reinforcement as an increase in the length of the reinforcement or as a reduction of the same; once these parameters have been set, it will be possible to draw the new reinforcements using the appropriate snaps.

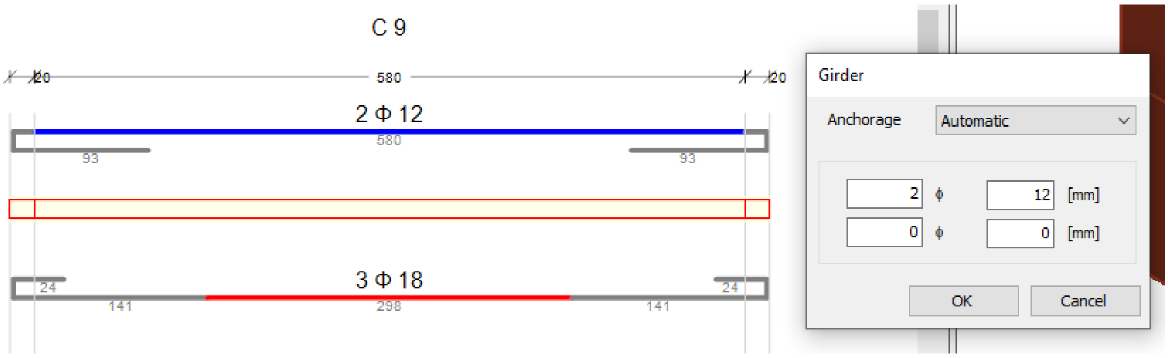
- Edit diameters and amount profiles: to modify the diameter or quantity of a profile on the elevation, simply select it and click with the right button; a dialog box will open in which to enter the new parameters.
- Insert stirrups: allows the manual insertion of the stirrups on the beam, with the possibility of customizing the spacing and diameter along the different areas of the beam.
- Edit stirrups: to change the diameter or spacing of the stirrups on the elevation, simply click on them.
- Delete profile: to delete one or more profiles, simply select them and confirm the deletion by clicking with the right button.
- Redo profile: the command is active only if a profile has been previously deleted.
- Stretch reagent length: allows you to stretch the reagent length of a template; it will be sufficient to select it and drag its ends in order to obtain the new desired length.
- Move profile: to move one or more profiles it will be necessary to select them, confirm with the right button and set the movement by clicking on the starting point and the consequent ending point.

Manual addition of profile

Manual addition allows you to insert custom profiles by drawing freehand on the elevation.

The command will require setting the quantity and diameter of the profiles, the type of anchoring (automatic or included) and the positioning of a drawing support profile, whose intersections between the lines can be used as a snap to trace the desired profile .

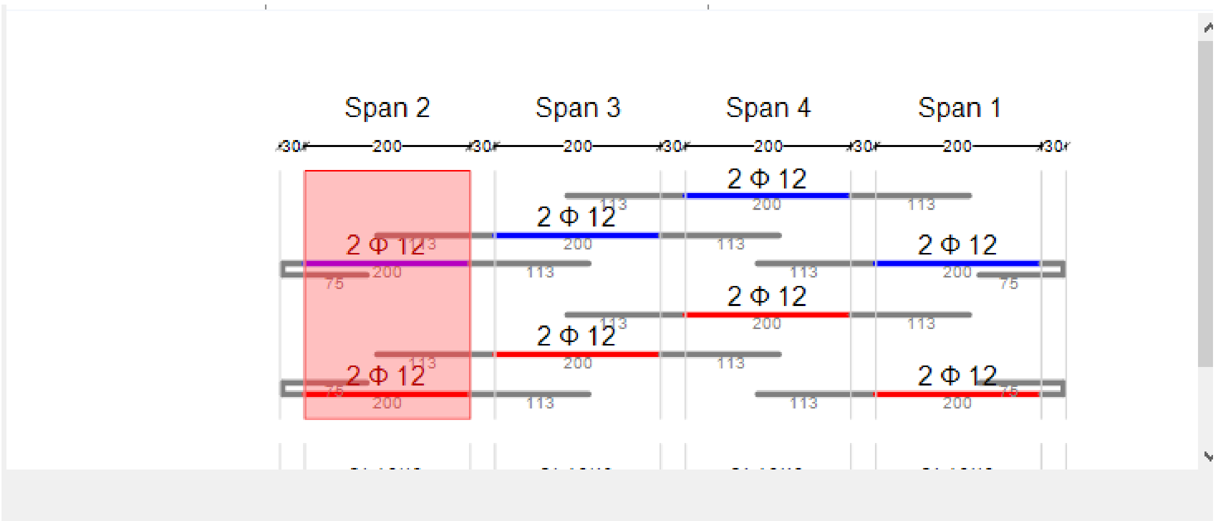
The "automatic" anchor option will add reinforcement sections to the shape drawn manually by the user, treating them as an anchor; the "included" anchor option as a reduction will subtract some sections of reinforcement from the profile drawn manually by the user, treating them as an anchor.

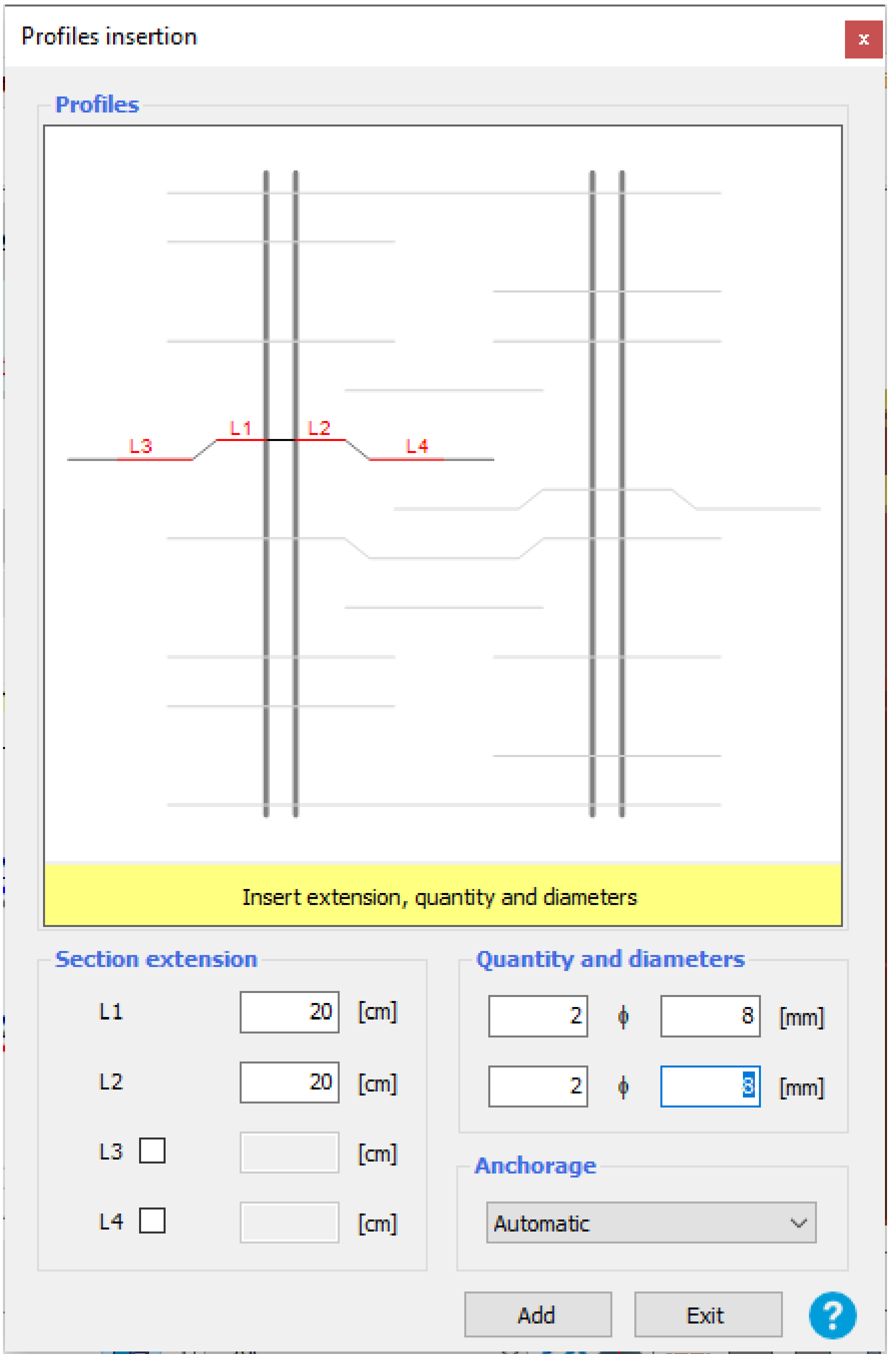


Add profile from template

The addition of a profile from a template allows you to use predefined profile templates, setting their characteristic geometric parameters.

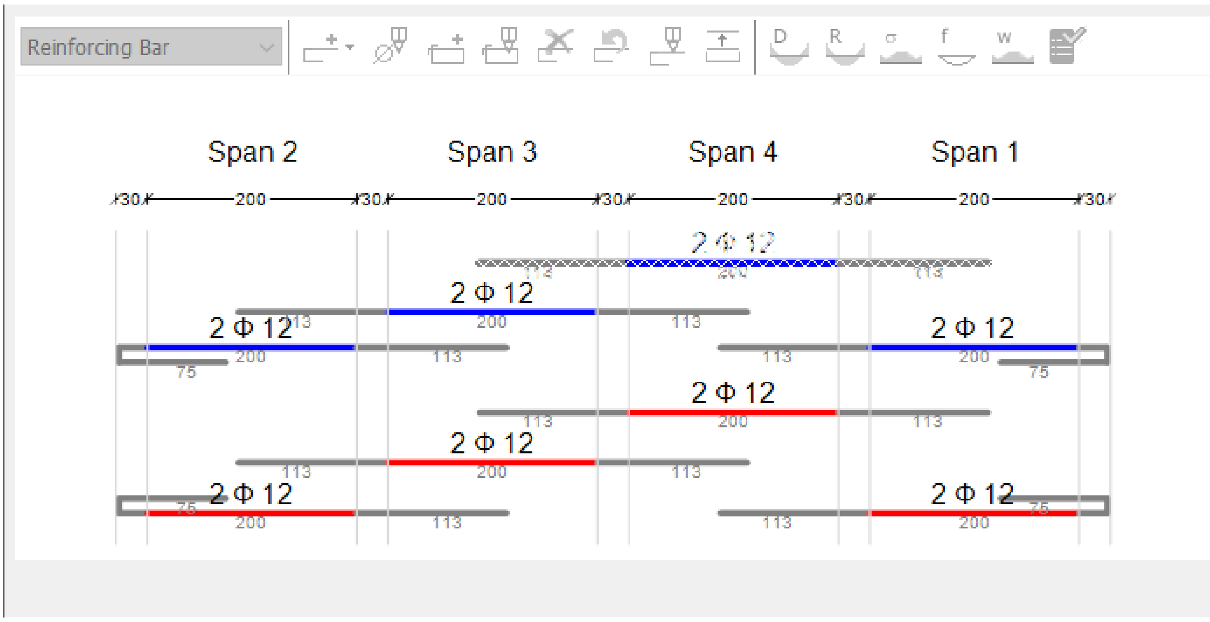
The profiles menu is accessed by selecting the span / s of interest and then clicking with the right button.


Span 2
Span 3
Span 4
Span 1
2 Φ 12
2 Φ 12
2 Φ 12
2 Φ 12
2 Φ 12
2 Φ 12
2 Φ 12
2 Φ 12

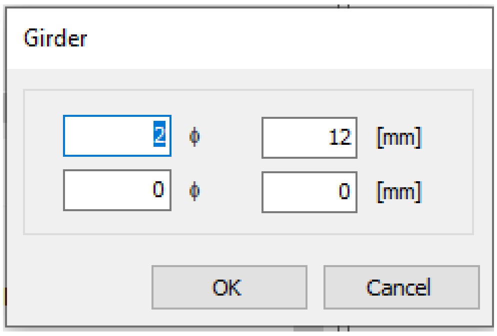


By clicking on the "Modify diameters / amount profiles" button it is possible to modify the diameter and number of the inserted profiles.

After clicking the button it is sufficient to graphically select the shapes to be modified:



I confirm the selection with the right key.
The window for editing the new parameters is displayed:

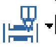


11.5.2.3.1.3 Reinforcement configuration

"Reinforcement" configuration

It allows to insert / modify longitudinal and transversal reinforcements and to graphically display the calculation results through diagrams.

 Insert longitudinal reinforcement: allows you to insert longitudinal reinforcements in FRP or FRCM

 Modify longitudinal reinforcement: allows you to modify a longitudinal reinforcement in FRP or FRCM previously inserted, stretching it or modifying its characteristics

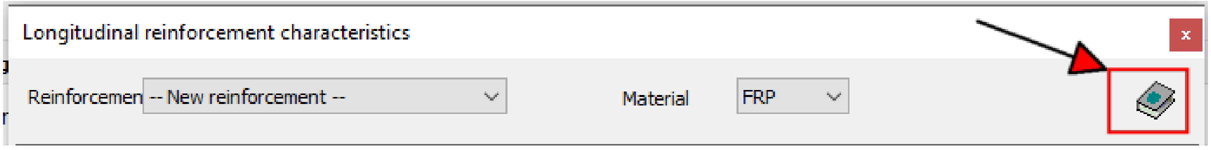
 Delete longitudinal reinforcement: to delete one or more reinforcements, simply select them and confirm the deletion by clicking with the right button.

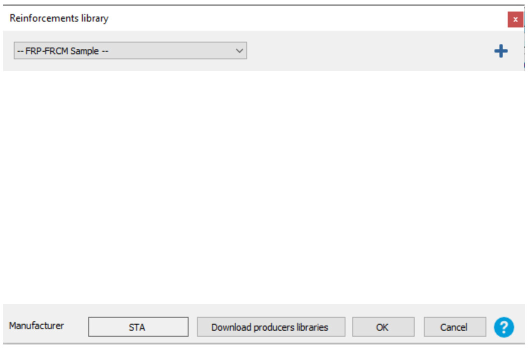
 Insert transverse reinforcement: allows you to insert transverse reinforcements in FRP or FRCM

 Modify transverse reinforcement: allows you to modify a transverse reinforcement in FRP or FRCM previously inserted, stretching it or modifying its characteristics

 Delete transverse reinforcement: to delete one or more reinforcements, simply select them and confirm the deletion by clicking with the right button.

In the upper right part, a button allows you to access the library of possible reinforcements.





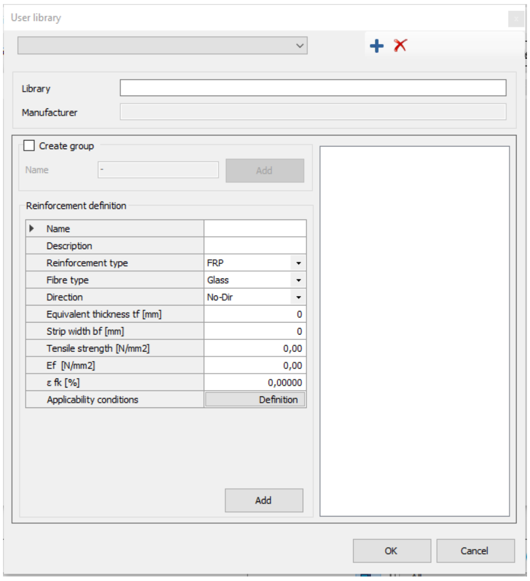
From the drop-down it is possible to view the libraries present and select the desired one.
By checking the "vertical" and / or "transversal" options, it is possible to choose the arrangement of the reinforcement.

Using the "+" symbol it is possible to create a new library to add to the list of those already present.
By clicking on the "Download manufacturer libraries" button you can consult and download the manufacturer libraries.

Once downloaded, the new library will be visible in the dropdown shown above.

If in the design practice we always refer to a specific manufacturer, it is possible to generate your own library that will be available for all future works.

To create a new user library it is necessary to define some parameters:

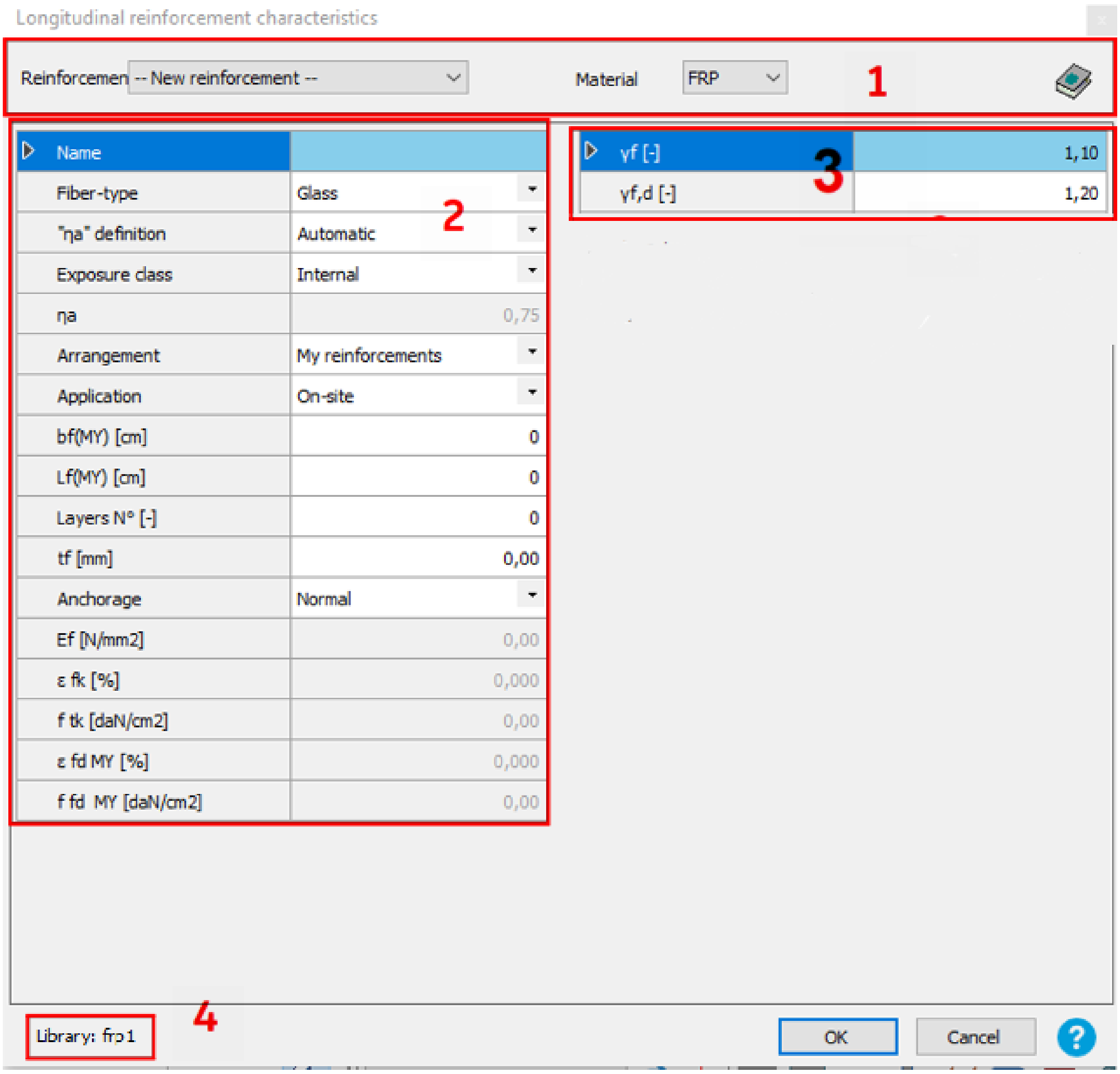


Some of these parameters are descriptive parameters (name, description), others are data necessary for defining the type of reinforcement, the type of fiber, the orientation (uni / bi-directional).

The following data (equivalent thickness, strip width, area, tensile strength, Ef and εfk) are all parameters available from the manufacturers' technical data sheets.

The insertion of FRP / FRCM reinforcements in the case of the joists follows the same normative references and the same insertion methods as in the case of the beams.

FRP reinforcement:



1

At the top of the previous menu it is possible to:

- insert a reinforcement previously inserted on the column from the list called "Reinforcement";
- select the type of material you intend to use (FRP / FRCM). Specifically, by leaving the FRP session active, all the input required below will comply with the current legislation for FRPs which is CNR-DT 200 R1 / 2013;
- choose the reinforcement fabric from the reinforcement libraries.

2

Name		Shows the name of the reinforcement fabric inserted from the library
Fiber-type	Glass	It reports the type of fiber-resin of the reinforcement inserted from the library
"ηa" definition	Automatic	Contains the type of definition of ηa: Automatic / Manual
Exposure class	Internal	Exposure class
ηa	0,75	ηa value
Arrangement	My reinforcements	Indicates the arrangement of the reinforcements: My / Mz / My and Mz
Application	On-site	Contains the application of On site / Preformed reinforcement
bf(MY) [cm]	0	Reinforcement width (MY)
Lf(MY) [cm]	0	Reinforcement Length (MY)
Layers N° [-]	0	Number of layers of reinforcement
tf [mm]	0,00	Thickness of the reinforcement
Anchorage	Normal	Type of Anchoring: Normal / Mechanical
Ef [N/mm2]	0,00	Normal modulus of elasticity in the direction of the force
ε fk [%]	0,000	Characteristic deformation
f tk [daN/cm2]	0,00	Characteristic tension of the reinforcement
ε fd MY [%]	0,000	Fiber Calculation Strain at Break (MY)
f fd MY [daN/cm2]	0,00	Calculation resistance of the fiber (MY)

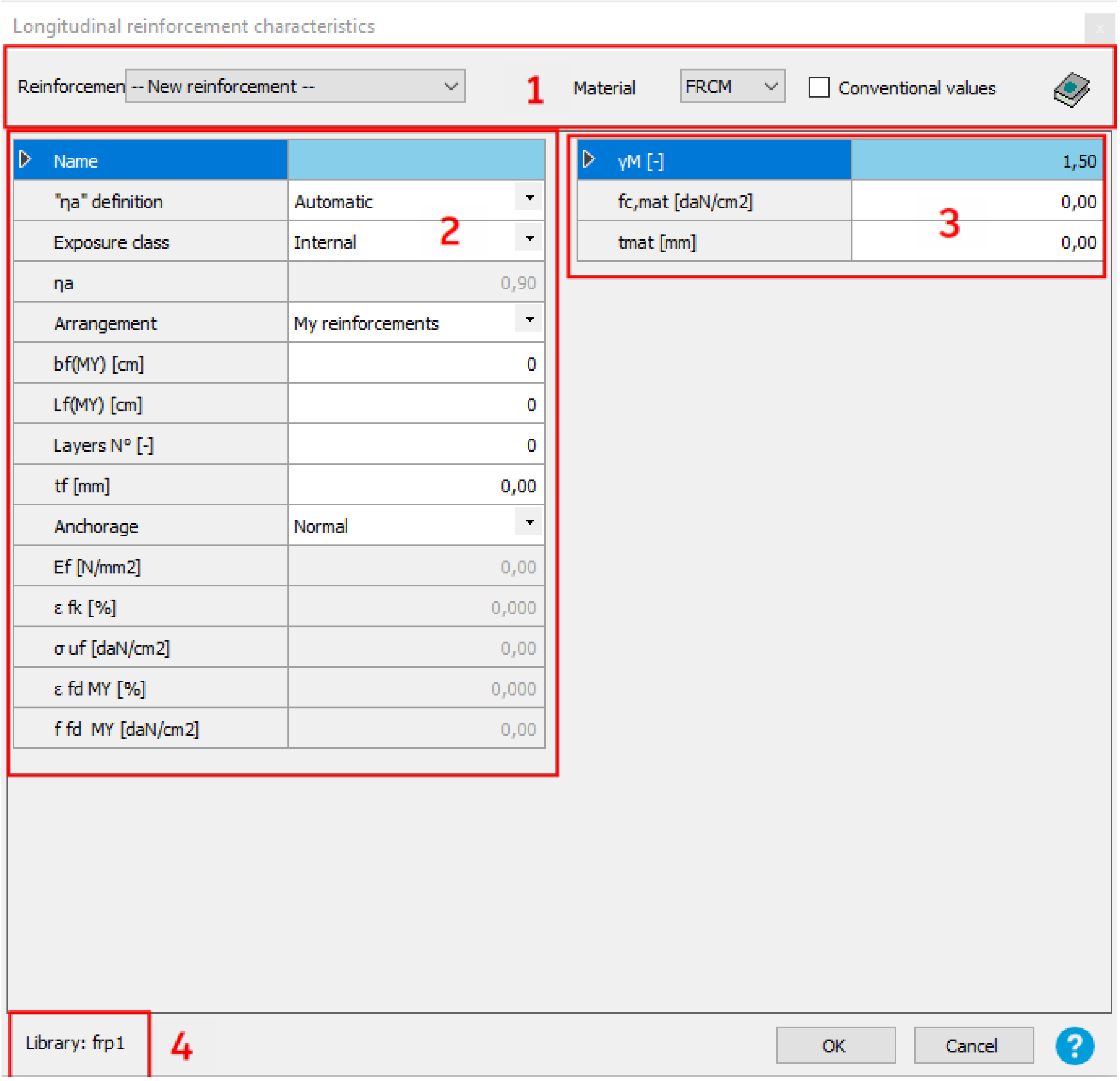
3

γf [-]	1,00	Partial safety factor of the FRP [3.4.1 CNR-DT 200 R1 / 2013]
γf,d [-]	1,00	Safety factor for the detachment of the FRP [3.4.1 CNR-DT 200 R1 / 2013]

4

Indicates the library to which the type of fabric chosen belongs.

+ FRCM reinforcement:



1

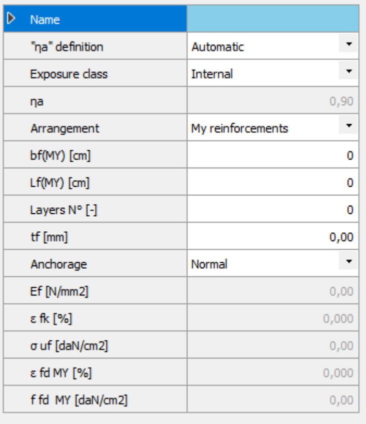
At the top of the previous menu it is possible to:

- insert a reinforcement previously inserted on the girder from the list called "Reinforcement";
- select the type of material you intend to use (FRP / FRCM). Specifically, by leaving the FRCM session active, all the input required below will comply with the current legislation for FRCM which is CNR-DT 215/2018;
- choose the reinforcement fabric from the reinforcement libraries;
- choose whether to use conventional values for FRCM fabric.

The use of conventional values refers to what is written in the CNR-DT 215/2018.
In the chapter concerning the calculation of the mechanical characteristics (§3.1) of the aforementioned CNR, reference is made to the concept of "conventional" stresses and deformations which represents the strength of the reinforcement system obtained by detachment tests from conventional supports and as such is dependent on the media type.
The use of the conventional limit deformation and of the competent conventional limit tension, allows to design FRCM reinforcement interventions avoiding *the explicit verification in the comparison of the phenomenon of detachment from the support* or

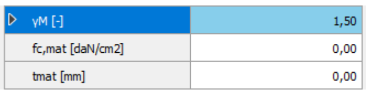
sliding of the fibers in the matrix at the ends of the reinforcement, otherwise necessary in cases in which this crisis mode is possible.

2



Shows the name of the reinforcement fabric inserted from the library
Contains the type of definition of ηa: Automatic / Manual
Exposure class
Value ηa
Indicates the arrangement of the reinforcements: My / Mz / My and Mz
Reinforcement Length (MY)
Reinforcement width (MY)
Number of layers of reinforcement
Thickness of the reinforcement
Type of Anchoring: Normal / Mechanical
Ultimate stress at break characteristic by traction of the dry fabric
Characteristic deformation
Ultimate stress at break characteristic by traction of the dry fabric
Fiber Calculation Strain at Break (MY)
Calculation resistance of the fiber (MY)

3



Partial safety coefficient of the FRCM [3.1 CNR-DT 215 2018]
Ultimate stress at break characteristic by traction of the dry fabric
FRCM single layer matrix thickness (1cm)

4
Indicates the library to which the type of fabric chosen belongs.

11.5.2.3.1.4 .dxf setting

"DXF" setting

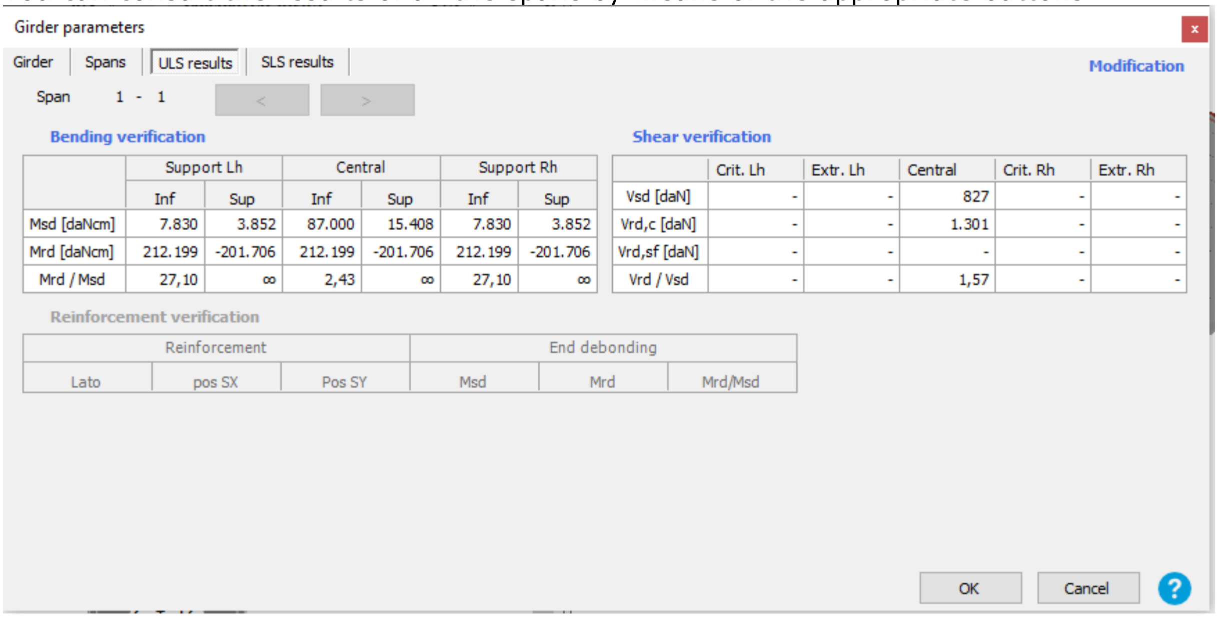
It allows to export the reinforcement scheme in .dxf format.

11.5.2.3.1.5 Results / verification

When the beam "tab" is dragged from the "Joists to calculate" column to the "Calculated joists" column, it will be possible to view the calculation results.

The ULS Results environment, available when a joist is in the Calculated Joists tab, reports the outcome of the ULS checks conducted on the element.
In particular, the results obtained with regard to the bending and shear verification can

be consulted.
You can consult the results of all the spans by means of the appropriate buttons.



Girder parameters

Girder | Spans | ULS results | SLS results — Modification

Span 1 - 1

Bending verification

	Support Lh		Central		Support Rh	
	Inf	Sup	Inf	Sup	Inf	Sup
Msd [daNcm]	7.830	3.852	87.000	15.408	7.830	3.852
Mrd [daNcm]	212.199	-201.706	212.199	-201.706	212.199	-201.706
Mrd / Msd	27,10	∞	2,43	∞	27,10	∞

Shear verification

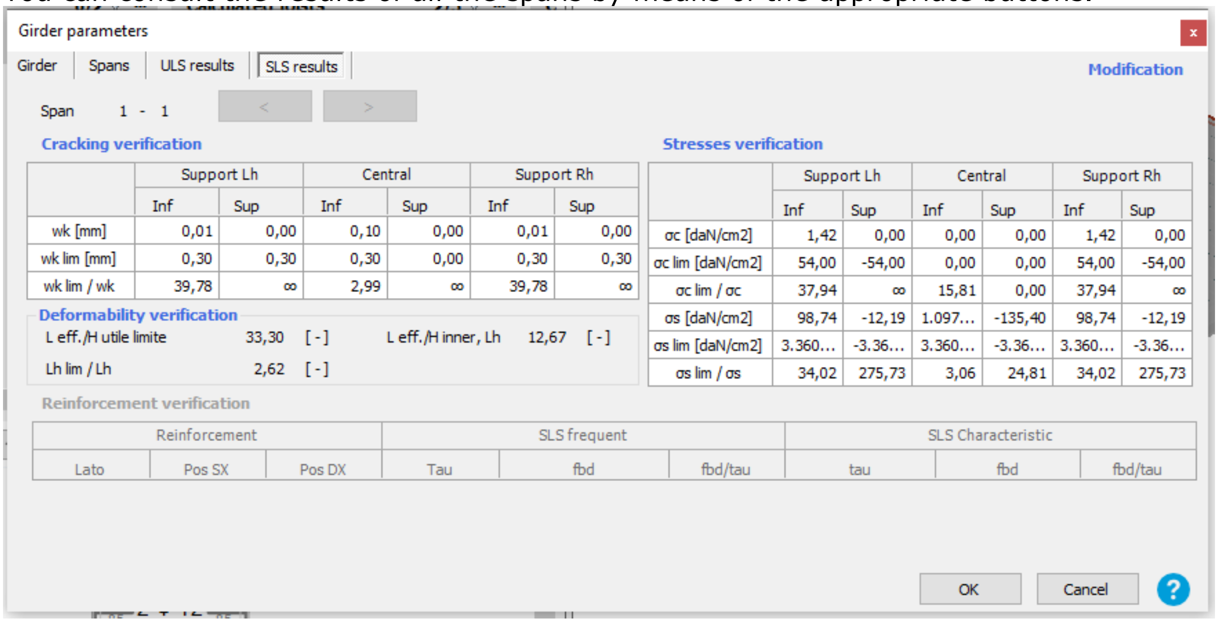
	Crit. Lh	Extr. Lh	Central	Crit. Rh	Extr. Rh
Vsd [daN]	-	-	827	-	-
Vrd,c [daN]	-	-	1.301	-	-
Vrd,sf [daN]	-	-	-	-	-
Vrd / Vsd	-	-	1,57	-	-

Reinforcement verification

Reinforcement			End debonding		
Lato	pos SX	Pos SY	Msd	Mrd	Mrd/Msd

OK | Cancel

The SLS Results environment, available when a beam is in the Calculated Joists tab, reports the outcome of the SLS checks carried out on the element.
In particular, the results obtained with regard to cracking, deformation and stress tests (on concrete and steel) can be consulted.
You can consult the results of all the spans by means of the appropriate buttons.



Girder parameters

Girder | Spans | ULS results | SLS results — Modification

Span 1 - 1

Cracking verification

	Support Lh		Central		Support Rh	
	Inf	Sup	Inf	Sup	Inf	Sup
wk [mm]	0,01	0,00	0,10	0,00	0,01	0,00
wk lim [mm]	0,30	0,30	0,30	0,00	0,30	0,30
wk lim / wk	39,78	∞	2,99	∞	39,78	∞

Deformability verification

L eff./H utile limite 33,30 [-] — L eff./H inner, Lh 12,67 [-]

Lh lim / Lh 2,62 [-]

Stresses verification

	Support Lh		Central		Support Rh	
	Inf	Sup	Inf	Sup	Inf	Sup
σc [daN/cm2]	1,42	0,00	0,00	0,00	1,42	0,00
σc lim [daN/cm2]	54,00	-54,00	0,00	0,00	54,00	-54,00
σc lim / σc	37,94	∞	15,81	0,00	37,94	∞
σs [daN/cm2]	98,74	-12,19	1.097…	-135,40	98,74	-12,19
σs lim [daN/cm2]	3.360…	-3.36…	3.360…	-3.36…	3.360…	-3.36…
σs lim / σs	34,02	275,73	3,06	24,81	34,02	275,73

Reinforcement verification

Reinforcement			SLS frequent			SLS Characteristic		
Lato	Pos SX	Pos DX	Tau	fbd	fbd/tau	tau	fbd	fbd/tau

OK | Cancel

11.5.2.4 ζv calculation

For the brick floor, the calculation method is the same as for the reinforced concrete beams, except for the different method of calculating the resistant contributions.

11.5.3 Corrugated sheet floor

11.5.3.1 Code

11.5.3.1.1 Corrugated sheet

The reference standard for the design and verification of corrugated sheet metal elements is:
- Ministerial Decree 17 January 2018;
- Eurocode 3.

The Italian NTC 2018 standard does not deal with the subject but, for corrugated steel sheets, refers to "standards of proven validity" or the EN 1993-1-3 standard.

In particular, with regard to Eurocode 3, reference is made to chapter 5 of the European standard EN 1993-1-3 with particular attention to point 5.5.3.4 which deals with the specific methodologies for verifying trapezoidal profiles (Trapezoidal sheeting profiles with intermediate stiffeners).

Regarding fire resistance, reference will be made to EN 1993-1-2.

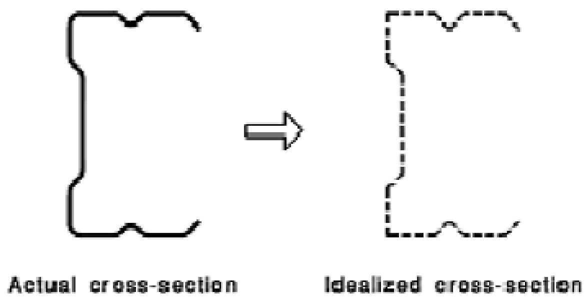
For the evaluation of the shear lag effect and for some aspects of the calculation of the effective section, reference will be made to the EN 1993-1-5 standard.

In the sections below, clarifications are reported regarding the following topics:
- Definition of the effective section
- Calculation criteria
- Fire resistance

11.5.3.1.1.1 Definition of the effective section

Before illustrating the calculation of the effective section, some dimensional constraints reported by the same standards are summarized below.

The influence of the "rounded" edges on the section resistance is not taken into consideration for fillet radii less than 5 times the thickness and the section characteristics are calculated for a set of plane elements connected in the intersection lines of the 'medium axis of the section (sharp corners).



For steel sheets, the effect of intermediate stiffeners on the flanges must be taken into account in the calculation of the effective section in accordance with item 5.5.3.4.2 of EN 1993-1-3.

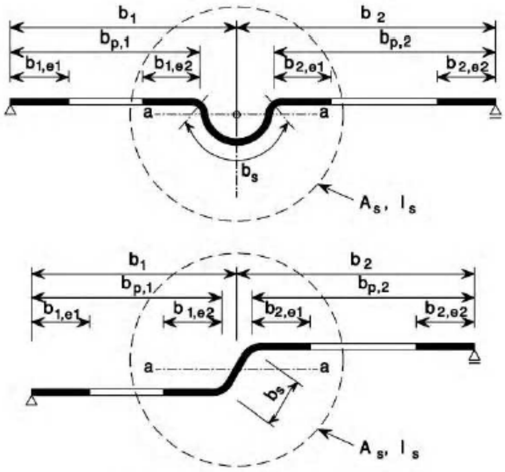
For steel sheets, the effect of intermediate stiffeners along the cores must be taken into account when calculating the effective section in accordance with item 5.5.3.4.3 of EN 1993-1-3

Calculation of the effective section

The procedure for calculating the effective section of the compressed parts is developed in three steps.

- Step I

The initial effective section of the intermediate stiffeners of the flanges and webs is shown below



The effective widths bi, e1; bi, e2 are determined using the width bp between the stiffeners and determining the reduction factor ρ as a function of the dimensionless slenderness λ_p. By way of example:

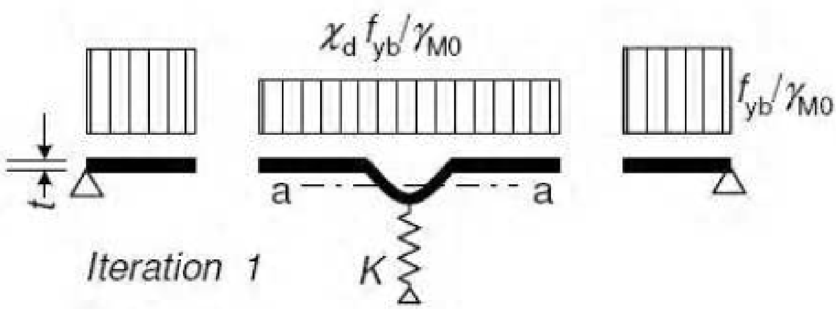
bi, e1 = bi, e2 = ρ × bp1

The effective area of an intermediate stiffener is given by:

As = t (b1, e2 + b2, e1 + bs)

- Step II

The first iteration involves the following stress situation in the compressed flange:



The reduction factor of the stiffening strength X_d is expressed as a function of the dimensionless slenderness λ_d based on the critical stress of flexural-torsional instability σ_{crs}.

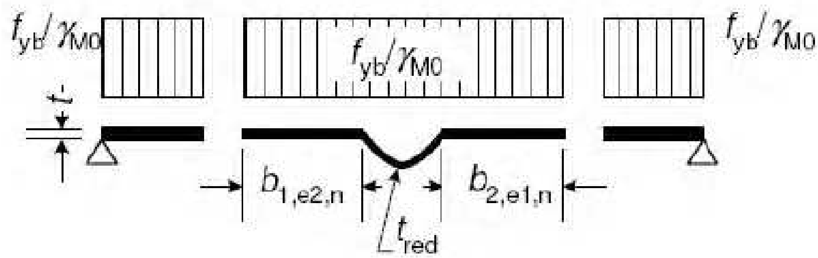
In the case of steel profiles with more than 2 intermediate stiffeners, the effective area of the entire compressed flange is given by:

$A_{eff} = \rho b_e t$

If Xd <1 it can be iteratively redefined starting from a modified value of the reduction factor ρ obtained from a reduced value of the dimensionless slenderness:

λp = λp √ Xd

• Step III:
The state of tension at the nima iteration will be the following:



The reduced effective area of the stiffener is calculated due to the lateral-torsional instability, referring to a compression stress relative to the center of gravity of the stiffener and calculated with reference to the effective stress.

The subsequent iterations to the first can be performed until $X_{d,n} \approx X_{d,n-1}$. The norms suggest a minimum of 2 iterations, convergence is almost always achieved at the third.

Effective section of the cores

The effective widths seff, i adjacent to stiffeners or bends or to the neutral axis are determined starting from an initial value $s_{eff,0}$, 0 obtained by considering the neutral axis of a section with effective area of the flange and gross area of the cores.

The effective sections of the stiffeners of the webs are further reduced by the flexural-torsional buckling factor X_d calculated for a value of the equivalent slenderness λd as a function of the critical buckling stress σcrsa (if the flanges are not stiffened, this critical stress is directly referred to for the calculation of the reduction factor).

Finally, if flanges and webs are stiffened, the method of calculating the factor Xd is still the one described here but referring to a reduced critical stress as a function of the two respective critical stresses.

11.5.3.1.1.2 Calculation criteria

Resistance to bending moment

If the yield point is reached first at the compressed limb ($W_{eff,c}$ /$W_{eff,t}$ <1), the design moment of resistance is given by the following expressions (item 6.1.4 of the EN 1993-1-3 standard):

- if the effective section modulus W_{eff} is less than the gross elastic section modulus W_{el}

$$M_{c,Rd} = W_{eff} f_{yb} / \gamma_{M0}$$

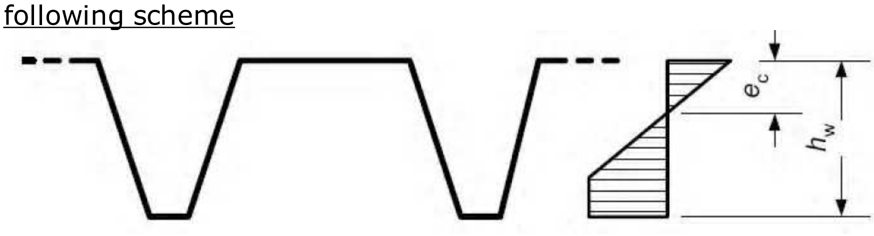
- if the effective section modulus W_{eff} is equal to the gross elastic section modulus W_{el}

$$M_{c,Rd} = f_{yb}\left(W_{el} + (W_{pl} - W_{el})4(1 - \bar{\lambda}_{e\max} / \bar{\lambda}_{e0})\right) / \gamma_{M0} \text{ but not more than } W_{pl} f_{yb} / \gamma_{M0}$$

The calculation of the design moment of resistance reported above is valid for inclination angles of the webs> 60 °; for lower angles the following expression applies:

$M_{c,Rd} = W_{el} f_{ya} / \gamma_{M0}$.

If the yield is reached first at the stretched edge, it is possible to carry out the verification in the elastic-partially plastic range (item 6.1.4 of the EN 1993-1-3 standard). In this case it is possible to define the design moment of resistance with reference to a partially plastic effective modulus ($W_{pp,eff}$) based on a distribution of bilinear stresses in the tense area but linear in the compressed area according to the following scheme



Shear resistance

The design shear strength is given by the following expression in accordance with item 6.1.5 of EN 1993-1-3:

$$V_{b,Rd} = \frac{\frac{h_w}{\sin\phi} t f_{bv}}{\gamma_{M0}}$$

where f_{bv} is tabulated on the reference standard as a function of the relative slenderness λ_w, which depends on the presence or absence of stiffeners along the web.

Resistance to support reaction

The stability check of the cores on the supports is performed taking into account the type of constraint and the depth of support. For steel sheets the reference is to points 6.1.7.3 and 6.1.7.4 of EN 1993-1-3 for non-stiffened and stiffened cores respectively.

The criterion of resistance to support reaction for non-stiffened webs reported below can be applied if the section satisfies the following relationships:

$$h_w / t \leq 200$$

$$r / t \leq 6$$

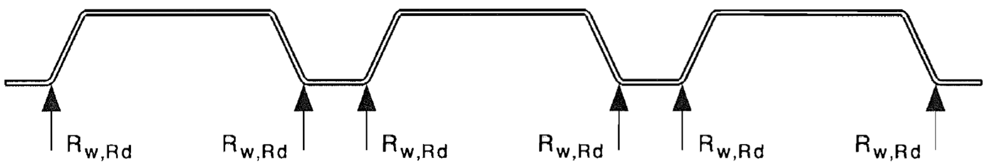
$$45° \leq \phi \leq 90°$$

where:

- hw is the height of the web between the mean lines of the flanges;
- r is the internal radius of the corners;
- Φ is the angle between the flange and the webs.

In this case:

$$R_{w.Rd} = \alpha t^2 \sqrt{f_{yb} E} \left(1 - 0,1\sqrt{r/t}\right)\left[0,5 + \sqrt{0,02\, l_a / t}\right]\left(2,4 + (\phi / 90)^2\right) / \gamma_{M1}$$



where:

- l_a is the support length;
- α is the category factor of the support (category 1 - end support - α = 0.075, category 2 - intermediate support - α = 0.15).

If the cores are stiffened, the resistance to the support reaction is given by the same expression reported above for non-stiffened cores multiplied by a corrective factor ka, s.

Combination moment-shear and moment-reaction of support

The verification for the combined moment-shear effect is as follows:

$$\frac{M_{y,Ed}}{M_{y,Rd}} + \left(1 - \frac{M_{f,Rd}}{M_{pl,Rd}}\right)\left(\frac{2V_{Ed}}{V_{w,Rd}} - 1\right)^2 \leq 1{,}0$$

where:

- $M_{f,Rd}$ is the design plastic moment of resistance of the section consisting of the effective area of the flanges only;
- $M_{pl,Rd}$ is the design plastic resisting moment of the section consisting of the effective area of the flanges and the effective (gross) area of the cores regardless of the their class;
- $V_{w,Rd}$ is the design shear resistance of the cores.

The verification for the combined support moment-reaction effect is as follows

$$M_{Ed} / M_{c,Rd} \leq 1$$

$$F_{Ed} / R_{w,Rd} \leq 1$$

$$\frac{M_{Ed}}{M_{c,Rd}} + \frac{F_{Ed}}{R_{w,Rd}} \leq 1{,}25$$

- $M_{c,Rd}$ is the resisting moment of the section as previously defined
- $R_{w,Rd}$ is the sum of the support reactions of each soul

Deformation

The tests in operation provided for the sheet include the verification of deformation according to the limits set out in section 7 of EN 1993-1-1, depending on the intended use of the floor. In the case of simple corrugated sheet, the inertial properties adopted for the verification can be interpolated between those of the gross section and those of the effective section according to the formula (7.1 EN 1993-1-3):

$$I_{fic} = I_{gr} - \frac{\sigma_{gr}}{\sigma}\left(I_{gr} - I(\sigma_{eff})\right)$$

Where:

- I_{gr} is the moment of inertia of the gross section;
- $\mathbf{s}_{gr}$ is the maximum voltage due to positive moment at the operating limit state, referred to the gross section;
- $I(\mathbf{s}_{eff})$: is the moment of inertia of the effective cross section with the local instability effect, calculated for a maximum stress $\mathbf{s} \geq \mathbf{s}_{gr}$, corresponding to the absolute maximum stress within the calculation length considered ;;

Since this value is variable along the span, the value associated with the maximum bending moment along the span is adopted, as indicated in 7.1 (4). Once the Ific inertia has been defined, the calculation can be performed using linear elastic formulations.

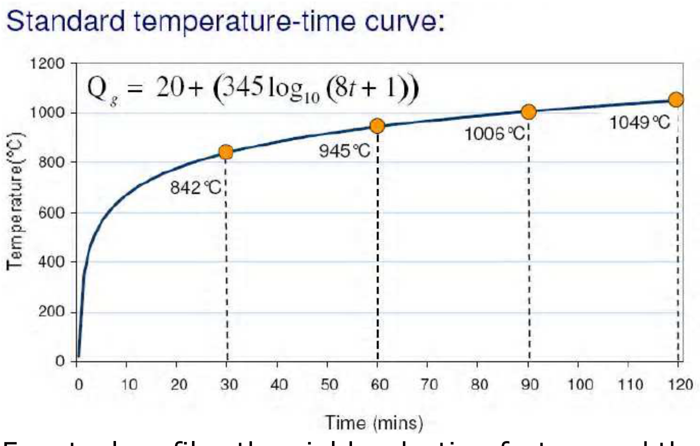
11.5.3.1.1.3 Fire resistance

The design stresses for the verification of fire resistance are given by an exceptional combination of actions with reference to the category of the environment.
They can also be defined starting from the stresses obtained from a fundamental combination (EN 1990) by multiplying the latter by a reduction factor:

$$\eta_{fi} = \frac{G_k + \psi_{fi} Q_{k,1}}{\gamma_G G_k + \gamma_{Q,1} Q_{k,1}}$$

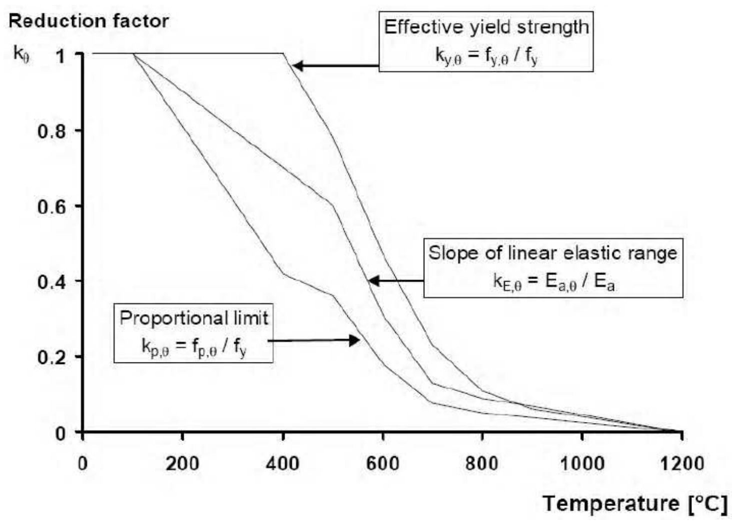
In the expression, the numerator represents the exceptional combination obtained by adding to the permanent loads the variable (operating) load multiplied by a combination factor that depends both on the type of action (frequent or almost permanent) and on the category of the environment in which it is placed the construction.

The standard fire curve used in the calculation is shown below:



For steel profiles the yield reduction factors and the modulus of elasticity are given in

table 3.1 of EN 1993-1-2 for temperatures between 20 ° and 1100 °.
The diagram of the reduction factor as a function of temperature is presented below:



The fire resistance test is performed according to the "R" (resistance) criterion in the resistance domain.
The calculation in the resistance domain consists of determining the resistance (to bending, shear, support reaction and combined) of the profile for the required fire exposure class (REI class).

For protected sections, the type of protection must be defined, i.e. whether in adherence with materials applied with a trowel or spray or with concrete casting or with insulated panels of defined density or with solid or perforated brick walls. The characteristics of the thermal protections (specific weight, conductivity, specific heat and thickness) must be clearly defined with reference either to European or national standards (EN 13381) or to manufacturers' specifications.

In the calculation of the fire resistant moment of steel profiles, two adaptation factors are introduced that take into account the non-uniformity of the temperature on the element.

- k1 = adaptation factor on the cross section of the element
- k2 = adaptation factor along the linear development of the element

For class sections <= 3 the design moment of resistance in fire conditions is given by the following expression:

$$M_{\mathrm{fi,t,Rd}} = k_{\mathrm{y,\theta,max}} M_{\mathrm{Rd}} [\gamma_{\mathrm{M,0}} / \gamma_{\mathrm{M,fi}}] / \kappa_1 \kappa_2$$

where:

- M_{Rd} is the resisting moment calculated for the normal condition;
- $k_{y,q,max}$ is the reduction factor of the yield stress referred to the maximum temperature reached at time t.

For class 4 sections the verification in the resistance domain is satisfied if the temperature of the profile at the time t of exposure to fire is lower than the critical temperature of the material limited to 350 ° C.

The fundamental difference between the two different types of elements is due to the different temperature increase of the profile at time t.

It is possible to perform a calculation considering one of the two possible selected elements.
In the case of the elements protected against fire, the delay in time of the temperature increase of the profile will result in greater safety with regard to fire checks.

For further details on the existing differences, it is possible to consult the EN 1993-1-2 standard.

11.5.3.1.2 Corrugated sheet with slab

The reference standard for the design and verification of corrugated sheet metal elements is:
- Ministerial Decree 17 January 2018;
- Eurocode 4.

As regards the corrugated steel sheets, reference will be made, for use in the casting phase, to the European standard EN 1993-1-3
(Eurocode 3).
The Italian NTC 2018 standard deals with the topic in chapter 4.3.6 "composite slabs with corrugated sheet metal" and, for the
verification of the resistance of the corrugated sheets in the casting phase, refers to the EN 1993-1-3 standard.

As regards the fire resistance of the consolidated slab, reference will be made to the EN 1994-1-2 standard, also cited by the Italian
standard NTC 2018.

In the sections below, clarifications are reported regarding the following topics:
- Calculation criteria for the casting phase
- Calculation criteria for the solidified phase
- Fire resistance

11.5.3.1.2.1 Calculation criteria for the casting phase

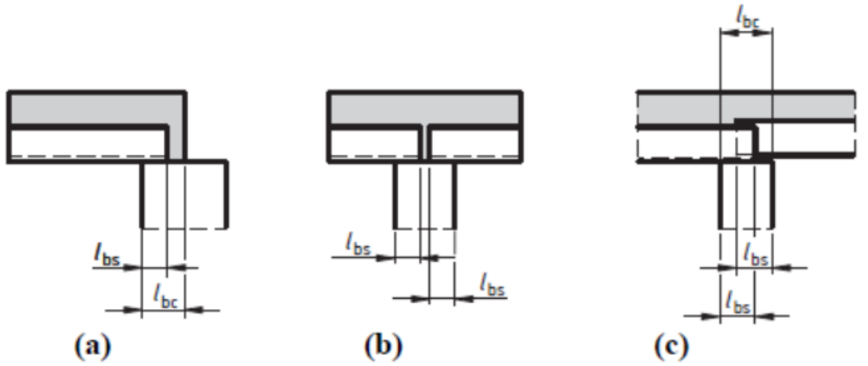
In the initial "casting" phase, the resistant part consists of the corrugated sheet only which can be made continuous with equidistant
intermediate provisional supports.

The calculation procedure of the corrugated sheet in the casting phase is exactly that presented in the chapter dedicated to the
simple corrugated sheet and is therefore performed in compliance with the indications of Eurocode 3 (EN 1993-1-3). Therefore, if you
want to carry out a check during the casting phase, it is possible to create a simple corrugated sheet as a user tab with defined
loads corresponding to the phase considered.

A consideration must be made on the support width which in the case of corrugated sheets for collaborating slabs is treated
differently by the standard EN 1994-1-1 item 9.2.3 (2) fig. 9.3, that is, for the casting phase:
- for simply supported composite slabs as well as for intermediate support of continuous slabs on steel or reinforced concrete support lbs = 50 mm;
- for support on different material lbs = 70 mm.

This limitation is graphically represented in the following figure extracted from the EN 1994-1-1 standard.

– for composite slabs bearing on steel or concrete: l_{bc} = 75 mm and l_{bs} = 50 mm;
– for composite slabs bearing on other materials: l_{bc} = 100 mm and l_{bs} = 70 mm.



11.5.3.1.2.2
Calculation criteria for the consolidated phase

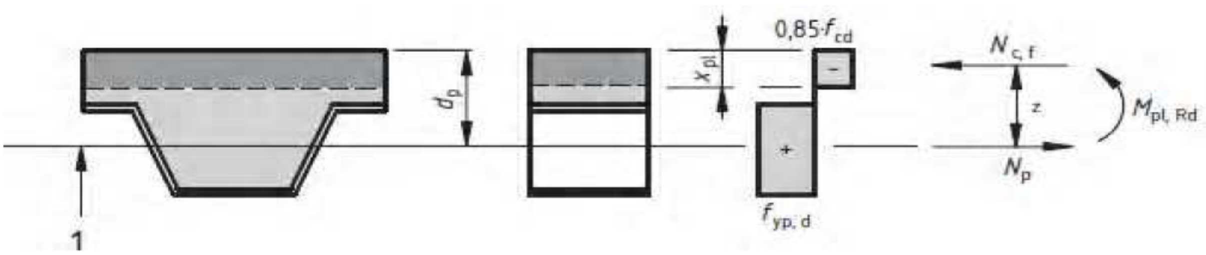
The analysis of the collaborating slab is performed with the linear elastic method for both the S.L.U. that for the S.L.S. (item 9.4.2 EN 1994-1-1).

In the final phase of "mixed slab" the mixed section is considered as a series of simply supported elements (item 9.4.2 (5) EN 1994-1-1) in which the corrugated sheet collaborates with the reinforced concrete slab. and bears not only the permanent loads carried but also the design overload.

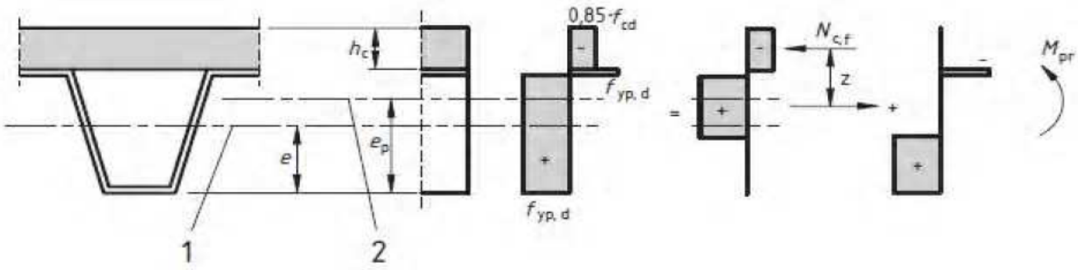
Bending

The calculation of the composite slab is performed based on the position of the plastic neutral axis according to the following diagrams:

- plastic neutral axis above the corrugated sheet:



- plastic neutral axis that cuts the fret:



Key
1 centroidal axis of the profiled steel sheeting
2 plastic neutral axis of the profiled steel sheeting

Note that z represents the distance between the pressure and traction centers of the section with pseudo-plastic behavior, i.e. with limit compression of the concrete equal to 0.85 fcd and limit tension in the steel fret equal to fyp, d.

The design plastic moment of resistance is given by the following expressions:

In the case of a plastic neutral axis above the corrugation:
Mrd = Ncf * z = Np * z

where is it:

z = dp - xpl / 2
Ncf = 0.85 * fcd * Ac (Ac = area of the concrete section between the plastic neutral axis and the interaxis of the frets)
Np = fyp, d * Ap (Ap = area of the steel corrugation included in the step between each corrugation)

In the case of a plastic neutral axis that cuts the corrugation
Mrd = Ncf * z + Mpr
where:

$$z = h - 0{,}5\, h_c - e_p + (e_p - e)\, \frac{N_{cf}}{A_{pe}\, f_{yp,d}}$$

$$M_{pr} = 1{,}25\, M_{pa} \left(1 - \frac{N_{cf}}{A_{pe}\, f_{yp,d}}\right) \leq M_{pa}$$

Mpa = plastic moment of resistance of the steel corrugation only.

Longitudinal shear for collaborating slabs

The resistance to longitudinal shear is evaluated according to three different criteria:
- "m-k" method; item 9.7.3 (2), (4), (5) EN 1994-1-1;
- method of "partial interaction"; item 9.7.3 (7), (8), (9) EN 1994-1-1;
- method of "partial interaction" with end anchors; item 9.7.4.

In the latter case, both the tensile strength of the sheet metal bound to the anchors and the resistance of the same (electro-welded pegs) are evaluated after defining the height, diameter, N ° / fret and distance from the edge of the sheet.

Partial interaction method

The method consists in defining the ultimate resistance shear stress, τu, Rd, which guarantees the non-sliding of the concrete with respect to the steel sheet. This parameter is experimentally defined by large-scale tests and is customized to the shape and material of the corrugated sheet as well as to the characteristics of the concrete (normal or lightened).
The design moment of resistance (MRd) that takes into account the sliding resistance is given by the expression already written for the case of a plastic neutral axis that cuts the fret by replacing Ncf with the value of Nc given by (item 9.7.3 (8)) :
Nc = τu, Rd b Lx <= Ncf (Lx = Sliding length measured from one end)
and replacing z with:

$$z = h - 0{,}5\, x_{\mathrm{pl}} - e_{\mathrm{p}} + \left(e_{\mathrm{p}} - e\right) \frac{N_{\mathrm{c}}}{A_{\mathrm{pe}}\, f_{\mathrm{yp,d}}}$$

The value of Nc can be increased by means of a nominal factor μ * REd if τu, Rd is determined taking into account the greater resistance to longitudinal shear caused by the bearing reaction (generally μ is assumed to be 0.5; item 9.7.3 (9)).
If anchoring systems are provided for the slab at the ends (for example Nelson pegs), a contribution to the resistance to longitudinal shear is given by the resistance traction to S.L.U. of the sheet metal.
For anchoring with Nellson pegs, the previously defined Nc force can be increased by the anchor resistance taken as a minimum value between the shear resistance of the pegs and the tensile strength of the sheet.
The shear strength of the rungs is given by item 6.6.4.2 of EN 1994-1-1.

The applicability of the partial interaction method requires that the behavior of the slab to the longitudinal cut is ductile (item 9.7.3 (3) EN 1994-1-1). This characteristic is considered satisfied when the breaking load exceeds by at least 10% the load that produces a sliding of 0.1 mm at the ends of the prototype. (Item 9.7.3 (3))

M-k method

The longitudinal shear strength can be determined using the m-k method which consists in demonstrating that the maximum design vertical shear VEd does not exceed the shear strength V1, Rd given by:

$$V_{1,\mathrm{Rd}} = \frac{b\, d_{\mathrm{p}}}{\gamma_{\mathrm{VS}}} \left(\frac{m\, A_{\mathrm{p}}}{b\, L_{\mathrm{s}}} + k \right)$$

where:
- b = width of the model;
- Ap = area of the profile section;
- m, k = empirical values in N / mm2 obtained from tests carried out with the m-k method;
- Ls = longitudinal cutting length assumed equal to L / 4 for the test with load uniformly distributed on the span L or distance of the concentrated load from the closest support in the case of a test with a pair of localized loads.

Vertical shear

Verification of resistance to vertical shear must be carried out according to paragraph 9.7.5 of the EN 1994-1-1 standard which however refers to the 1992-1-1 standard,

paragraph 6.2.2.
The vertical shear resistance, given by the expression 6.2.a with a minimum value given by the expression 6.2.b, must be greater than or equal to the design vertical shear (support reaction).

(1) The design value for the shear resistance $V_{Rd,c}$ is given by:

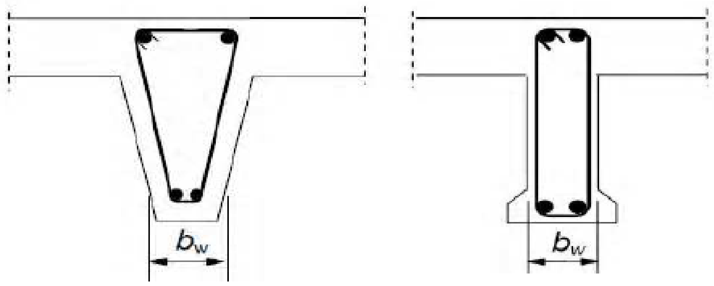
$$V_{Rd,c} = [C_{Rd,c}k(100\,\rho_l f_{ck})^{1/3} + k_1\,\sigma_{cp}]\,b_w d \qquad (6.2.a)$$

with a minimum of

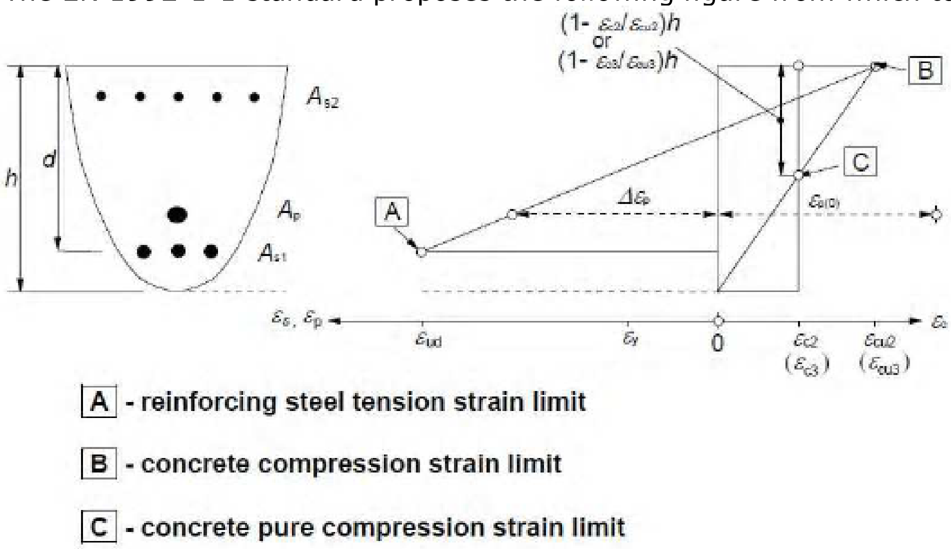
$$V_{Rd,c} = (v_{min} + k_1\sigma_{cp})\,b_w d \qquad (6.2.b)$$

The expression of the resistant cut refers in fact to the standard relating to reinforced concrete structures and must therefore be
adapted to the case of collaborating slabs on corrugated sheet metal. In particular, the meanings of some symbols referring to the
case of corrugated slabs are shown below.

- bw is the minimum width of the stretched area which can reasonably be taken as the width of the lower flange of the corrugation. Paragraph 6.2.3 of EN1992-1-1 proposes this scheme for the definition of bw referred to a reinforced concrete joist:



- d = distance between the slab extrados and the center of gravity of the metal section. The EN 1992-1-1 standard proposes the following figure from which to extrapolate d:



- σ_{cp} can be set equal to 0 in the absence of axial force N_{Ed} of the slab;
- A_{sl} is the area of the steel profile.

Deformation

The deformation check of the consolidated slab must be performed for the almost

permanent or frequent combination with reference to the NTC 2018 item 2.5.3 standard and the EN 1992-1-1 item 7.4.1 standard (4) taking into account the following loads:
a) - permanent load carried (flooring, substrate, false ceiling, fixed installations, etc.);
b) - uniformly distributed variable load reduced for the combination to the SLS considered (frequent or almost permanent);
c) - concentrated load for the removal of any props with a value corresponding to the load share supported in the casting phase.

In the case of a continuous slab on several supports, it is possible to calculate the deformation in the span taking into account not
the inertia of the fully reacting homogeneous section but of an inertia mediated between the inertia of the gross homogeneous
section and the inertia of the cracked section. In this case, a percentage of reinforcement must be provided in the taut area of the
slab and in correspondence with the intermediate supports in order to avoid "excessive cracking", that is: area of the reinforcement>
0.2% of the section of the concrete above the corrugation if the slab is cast with intermediate temporary supports otherwise the
reinforcement area must be> 0.4% of the same concrete section.

11.5.3.1.2.3 Fire resistance

The reference standard for the control of fire resistance is EN 1994-1-2 in which two verification criteria are set out:

- Criterion "I" (insulation) or the insulation capacity between two environments with temperature limitation on the extrados of the slab (on average 140 ° C) called
also criterion of "verification in the temperature domain"
- The "R" (resistance) criterion or the ability to resist for a fixed period of time to a fire exposure defined by a "standard" or parametric fire curve.

This last criterion, of "verification in the domain of resistances", essentially defines the REI class or the resistance time under the
design loads in relation to a fire curve which is normally the standard curve for gas fire.

The stresses to be taken into consideration for the verification of resistance in fire conditions are the same as indicated in the
paragraph relating to simple corrugated sheet metal.

The fundamental parameters for checking the fire resistance of a slab collaborating with corrugated sheet metal are:

- The temperatures of the three parts of the corrugation (lower flange, upper flange and cores) and of any reinforcement bars;
- The reduction factors of the resistances ($k_{y,\theta}$ and $k_{c,\theta}$) and of the modulus of elasticity.

The reduction factors for the characteristics of sheet steel and concrete (both normal and lightened) at high temperatures are given
by the EN 1994-1-2 standard in table 3.2 for steel and in table 3.3 for concrete. For the reinforcement bars, reference will be made
to table 3.4 of the same standard.

The position of the plastic neutral axis can be derived from the following expression (EN1994-1-2 Item 4.3.1 (4)):

$$\sum_{i=1}^{n} A_i k_{y,\theta,i} \left(\frac{f_{y,i}}{\gamma_{M,fi,a}} \right) + \alpha_{slab} \sum_{j=1}^{m} A_j k_{c,\theta,j} \left(\frac{f_{c,j}}{\gamma_{M,fi,c}} \right) = 0$$

By solving and setting Nc, f, fi = Np, fi, the position of the plastic neutral axis is found at the design temperature qa of the individual parts of the sheet:
$x_{pl,fi} = N_{p,fi}$ / (0,85 * b * $f_{c,d}$) where b is the distance between the frets.

For protected slabs, a minimum time of exposure to fire of 30 'and a limit temperature of the steel sheet of 350 ° C is required (item 4.3.3).

For unprotected slabs, the calculation must be performed in accordance with Annex D of the EN 1994-1-2 standard, the most significant parts of which are listed below.

Unprotected slabs

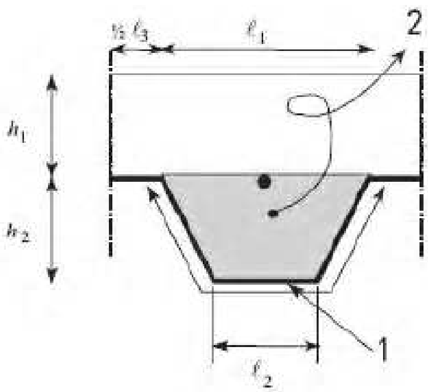
1) Criterion "I"

the fire resistance time with reference to an average temperature of the extrados of the slab of 140 ° C is given by the following expression:

$$t_i = a_0 + a_1 \cdot h_1 + a_2 \cdot \Phi + a_3 \cdot \frac{A}{L_r} + a_4 \cdot \frac{1}{\ell_3} + a_5 \cdot \frac{A}{L_r} \cdot \frac{1}{\ell_3}$$

where:

$$\frac{A}{L_r} = \frac{h_2 \cdot \left(\frac{\ell_1 + \ell_2}{2} \right)}{\ell_2 + 2\sqrt{h_2^2 + \left(\frac{\ell_1 - \ell_2}{2} \right)^2}}$$



1 = Lr ; 2 = A

The coefficients of the expression are shown in the following table:

Table D.1: Coefficients for determination of the fire resistance with respect to thermal insulation

	a_0 [min]	a_1 [min/mm]	a_2 [min]	a_3 [min/mm]	a_4 [mm min]	a_5 [min]
Normal weight concrete	-28,8	1,55	-12,6	0,33	-735	48,0
Lightweight concrete	-79,2	2,18	-2,44	0,56	-542	52,3

The exposure factor of the upper flange is given by:

$$\Phi = \left(\sqrt{h_2^2 + \left(l_3 + \frac{l_1 - l_2}{2} \right)^2} - \sqrt{h_2^2 + \left(\frac{l_1 - l_2}{2} \right)^2} \right) \Big/ l_3$$

2) Criterion "R"

The design moment of resistance for exposure to fire is given by the following expression:

$$M_{fi,t,Rd} = \sum_{i=1}^{n} A_i z_i k_{y,\theta,i} \left(\frac{f_{y,i}}{\gamma_{M,fi}} \right) + \alpha_{slab} \sum_{j=1}^{m} A_j z_j k_{c,\theta,j} \left(\frac{f_{c,j}}{\gamma_{M,fi,c}} \right)$$

where:

- zi, zj is the distance of the center of gravity of the individual parts i (steel), j (concrete) (lower flange, upper flange, cores, reinforcement bars) from the neutral plastic axis of the section;
- the temperature of the steel profile must be calculated for each part (lower flange, upper flange and cores) with the following expression:

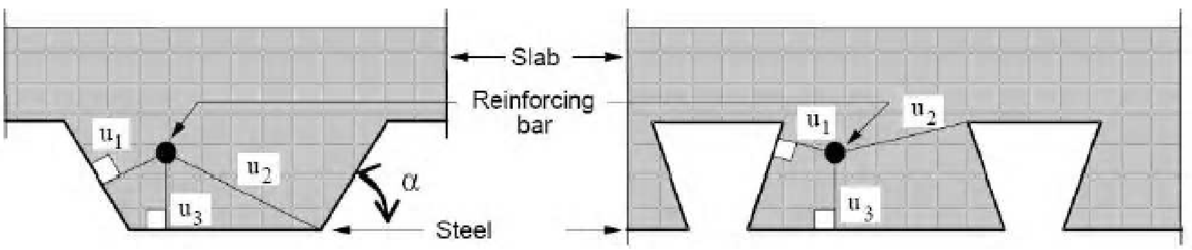
$$\theta_a = b_0 + b_1 \cdot \frac{1}{\ell_3} + b_2 \cdot \frac{A}{L_r} + b_3 \cdot \Phi + b_4 \cdot \Phi^2$$

where:

- the bi factors are given in table D.2 of annex D of the EN1994-1-2 standard according to the type of concrete (normal or lightened), the time of exposure to fire and the steel sheet part considered. Intermediate values can be obtained by linear interpolation;
- the exposure factor was previously defined;
- the temperature of the internal bars is given by the following expression:

$$\theta_s = c_0 + \left(c_1 \cdot \frac{u_3}{h_2} \right) + (c_2 \cdot z) + \left(c_3 \cdot \frac{A}{L_r} \right) + (c_4 \cdot \alpha) + \left(c_5 \cdot \frac{1}{\ell_3} \right)$$

in which z and ui are geometric parameters relating to the position of the bar in the section and a is the inclination of the sheet:



The coefficients are provided by table D.3 of annex D of the standard EN1994-1-2.

The resistance to longitudinal shear must also be verified in fire conditions; consequently the design parameters $\tau_{u,Rd}$ and m-k must also be defined with high temperature tests and the partial interaction diagram must be drawn to check that in each section along the element the moment of resistance is higher than the design moment in that section.

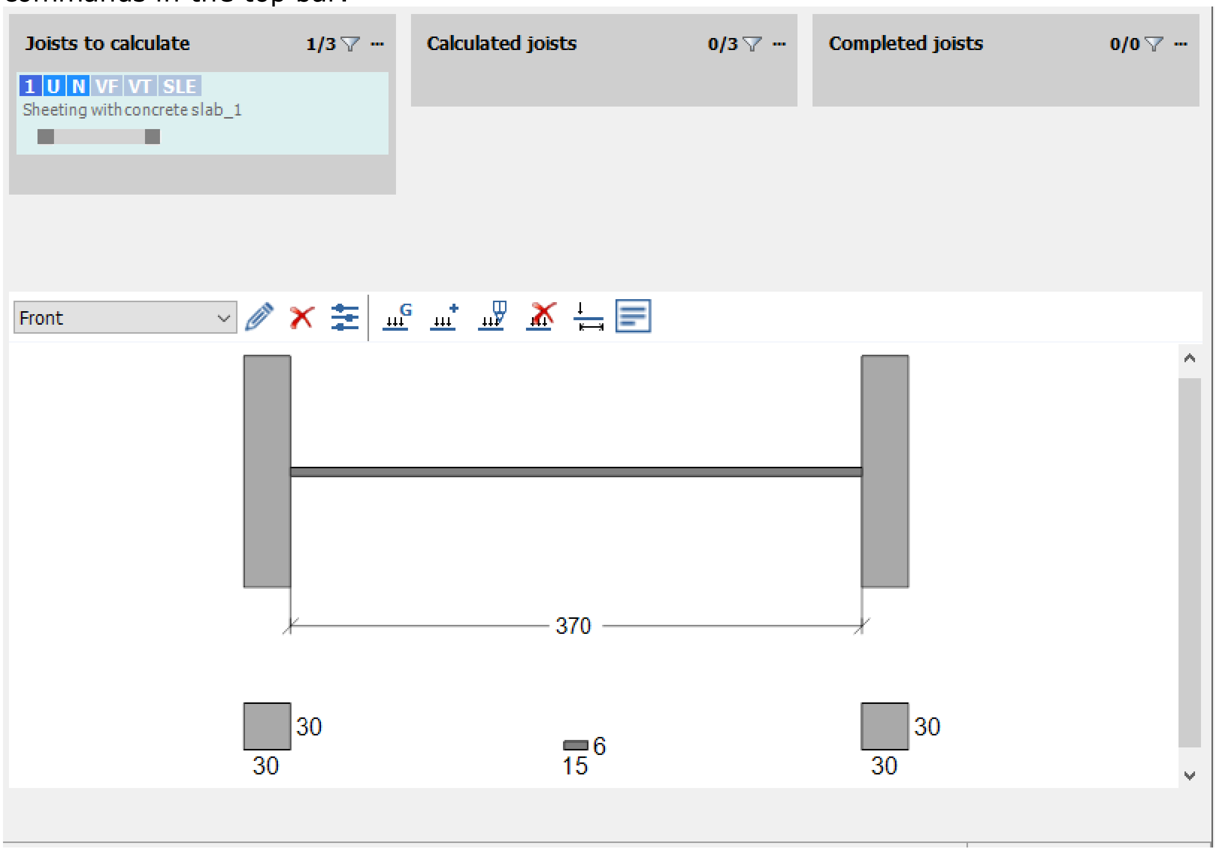
11.5.3.1.3 ζv calculation

For the floor in corrugated sheet and corrugated sheet with collaborating slab, the calculation method is the same as for the reinforced concrete beams, except for the different method of calculating the resistant contributions.

11.5.3.2 Work environment

11.5.3.2.1 Front interaction

The "Front" view allows you to interact graphically with the elements using the commands in the top bar.



The command bar has several possible configurations depending on the position of the calculation card within the columns.

11.5.3.2.1.1 Front configutarion

"Front" configuration

It allows to modify geometries, loads and calculation parameters.

"Modify": allows you to modify the selected span;

"Delete": allows you to delete the selected spans;

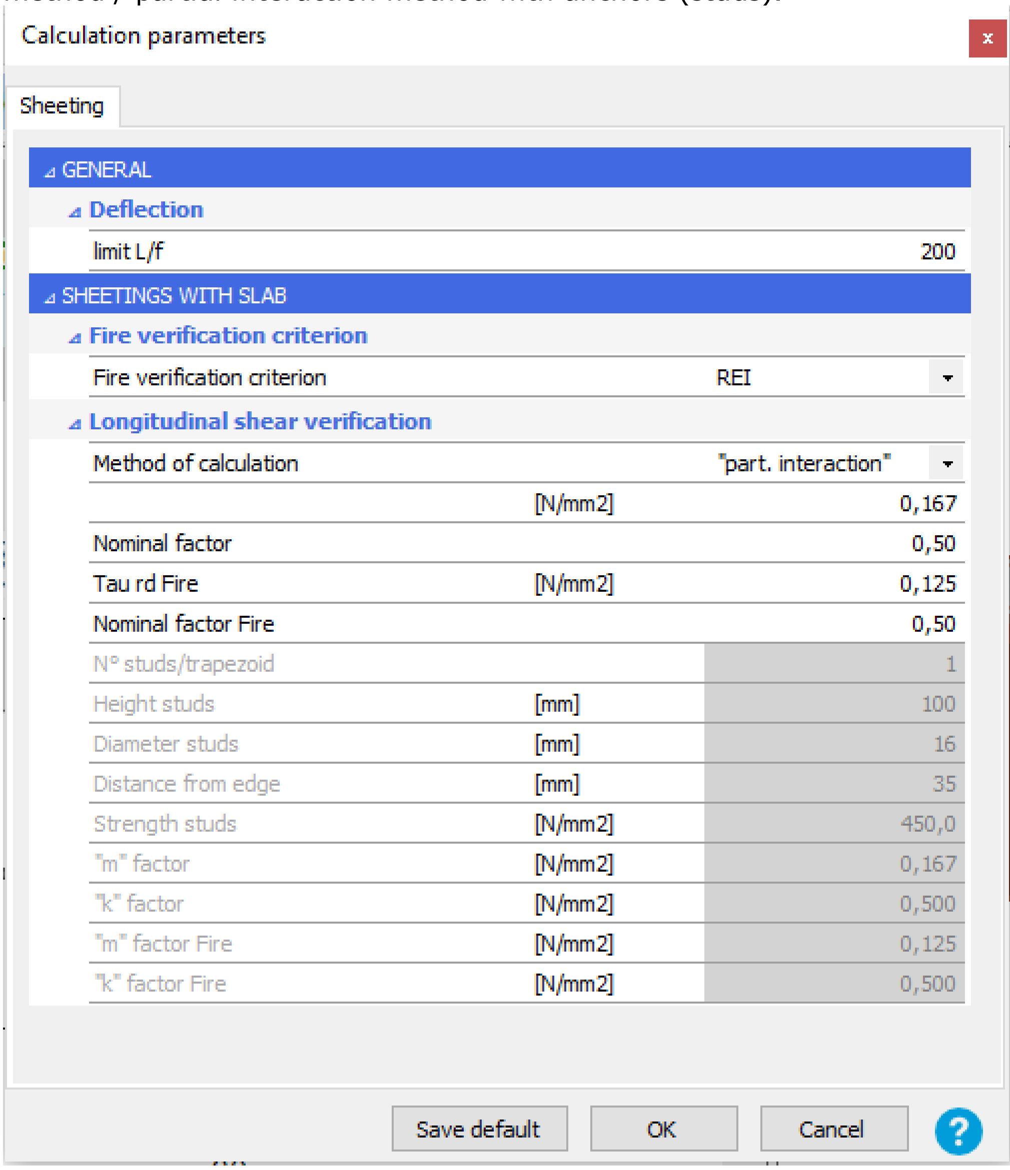
"Calculation parameters": allows you to set / modify the calculation parameters;

"Results": allows you to view the results of the calculation.

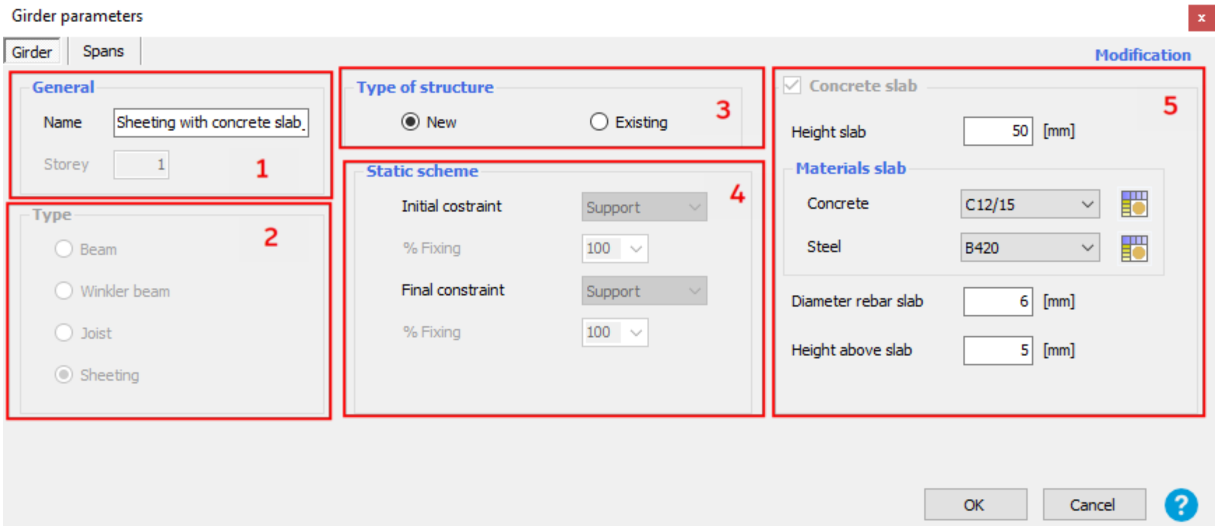
"Dead weight": allows you to add the load corresponding to the own weight on all spans;

"Add load": allows you to add new loads to the selected spans;
"Edit load": allows you to modify the selected loads;
"Delete load": allows you to delete the selected loads;
"Load dimensions": allows you to view / not display the dimensions relating to the loads;
"Legend of loads": activates the display of the load legend.

The calculation settings for simple corrugated sheet slabs and corrugated sheet slabs with slab can be customized. Specifically, the limit of the L / f ratio to be respected in the verification phase can be entered.
In the case of corrugated sheet metal with collaborating slab, it is possible to set:
- fire verification criterion: resistance / insulation;
- longitudinal shear verification calculation method: 'm-k' method / partial interaction method / partial interaction method with anchors (studs).



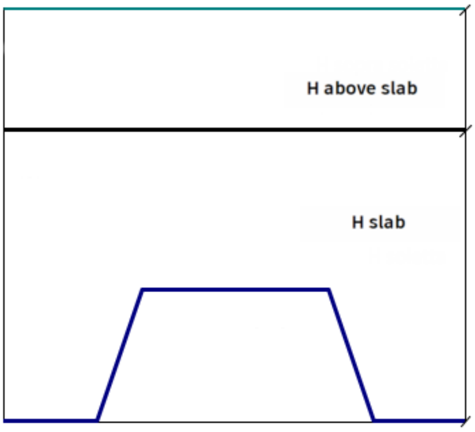
The girder Parameters environment provides:



[1] insertion of general data;
[2] selection of the type of element (set on corrugated sheet);
[3] setting the type of structure, with the possibility of choosing between new and existing;
[4] definition of the static scheme, with the possibility of setting the interlocking percentage if the latter is set as a constraint on at least one of the two extremes;

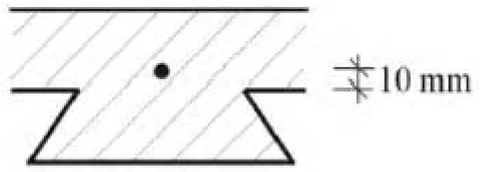
If corrugated sheet is selected as "Type of girder":
[5] definition of the slab (geometry, materials).

The slab data are as follows, illustrated in this explanatory image:



where:

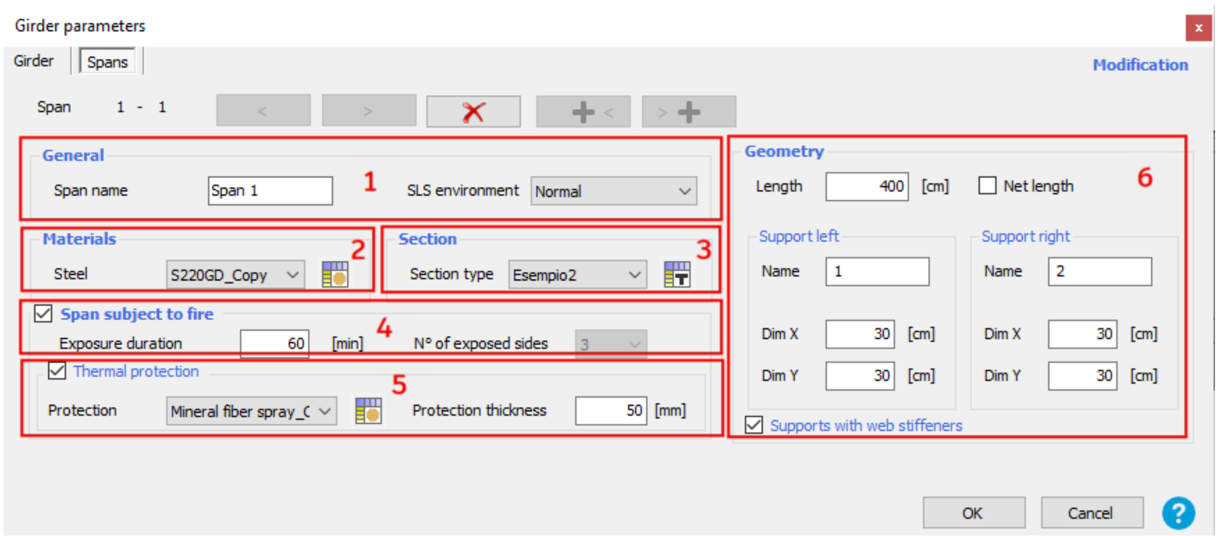
- H slab: height of the slab starting from the lower flange of the sheet metal section;
- Slab materials: these are the materials that characterize the slab of the slab;
- Reinforcement bar diameter: a diameter of the reinforcing bars is required (positioned, as conventionally, 10 mm above the corrugation); to increase the fire resistance of the slab it is recommended to insert at least one steel bar for each corrugation, as specified in the following figure:



if it is not necessary to check the fire calculation, the respective field can be left blank. Therefore, it is specified that the thickness H above the slab will not be considered in subsequent structural calculations.

- H above slab: is the height of the area between the slab and the floor level; the height of the above slab is required, only, in the case in which it is intended to carry out a verification of the fire calculation. If it is not necessary to check the fire calculation, it will be possible to leave the respective field blank; therefore, it is specified that the thickness H above the slab will not be considered in subsequent structural calculations.

The Span environment includes:



[1]General

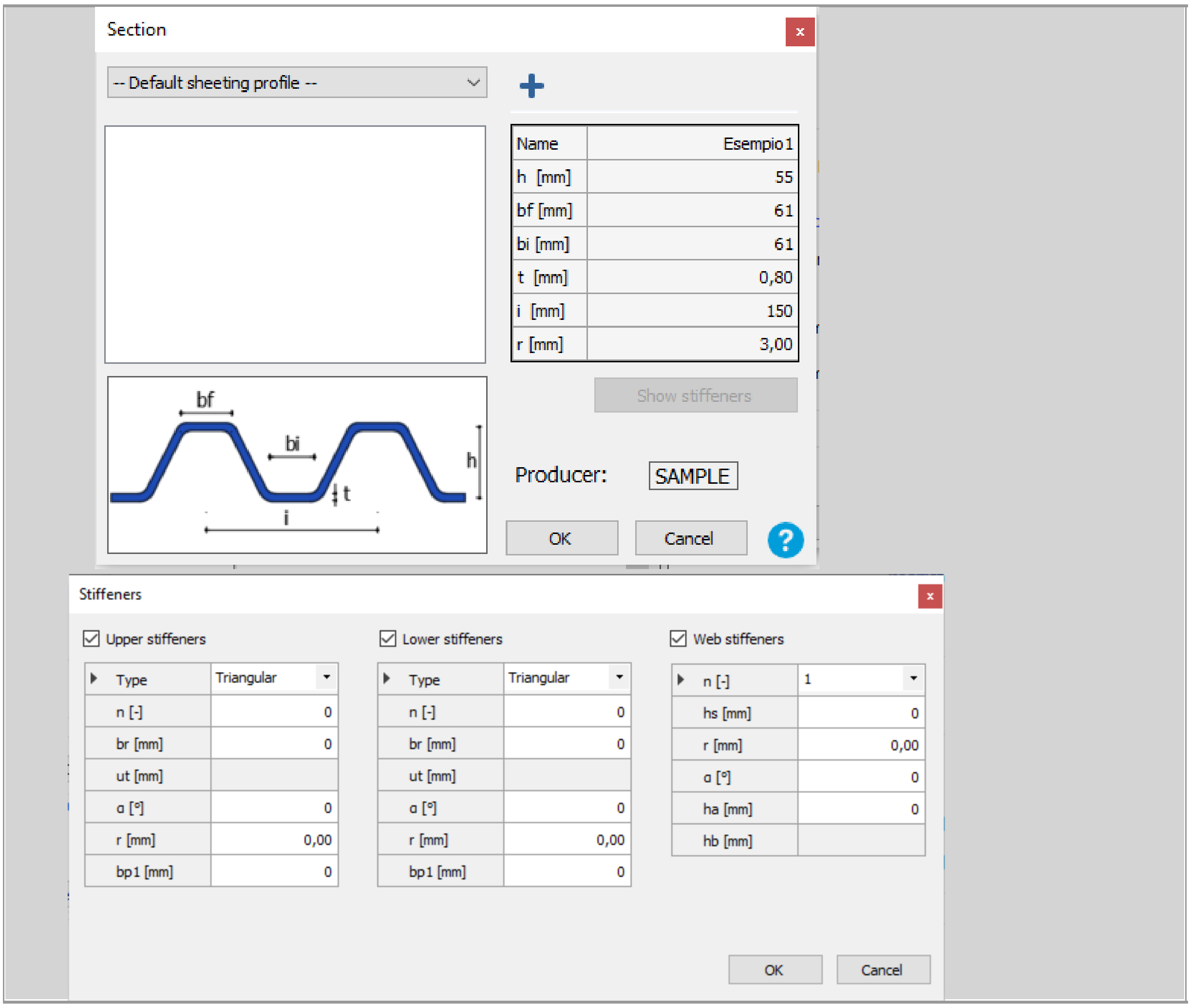
The section provides for the insertion of general data: in particular the name of the span and the aggressiveness of the environment for the SLS checks.

[2]Materials

The section provides for the definition of steel, with the possibility of accessing the respective lists.

[3]Section

The menu provides for the definition of the section of the span: it is possible to choose between a different type of section according to the selected library.
It is possible to add further libraries belonging to different manufacturers with the key +.
In the grid positioned on the right it is possible to view the geometry of the selected section; with a click on the "Stiffeners" button it is possible to view any upper / lower stiffeners and of the web.



[4]Span subject to fire

The section provides for the possibility of setting the calculation for a span subject to fire, after checking the appropriate box; the selection involves setting the exposure time (expressed in [min]) and the number of exposed sides.

[5] Thermal protection

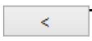
The section provides for the possibility of performing a calculation by adding a thermal protection, after checking the appropriate box; the selection involves setting the type of protection and its thickness.

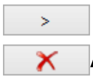
[6]Geometry

The Geometry section provides for the insertion of the span span (which will be understood according to the criterion selected in the Girder Parameters environment), the geometric data and the names of the columns at the ends. Furthermore, it is possible to set by selecting the appropriate box with a check, supports equipped with core stiffening elements.



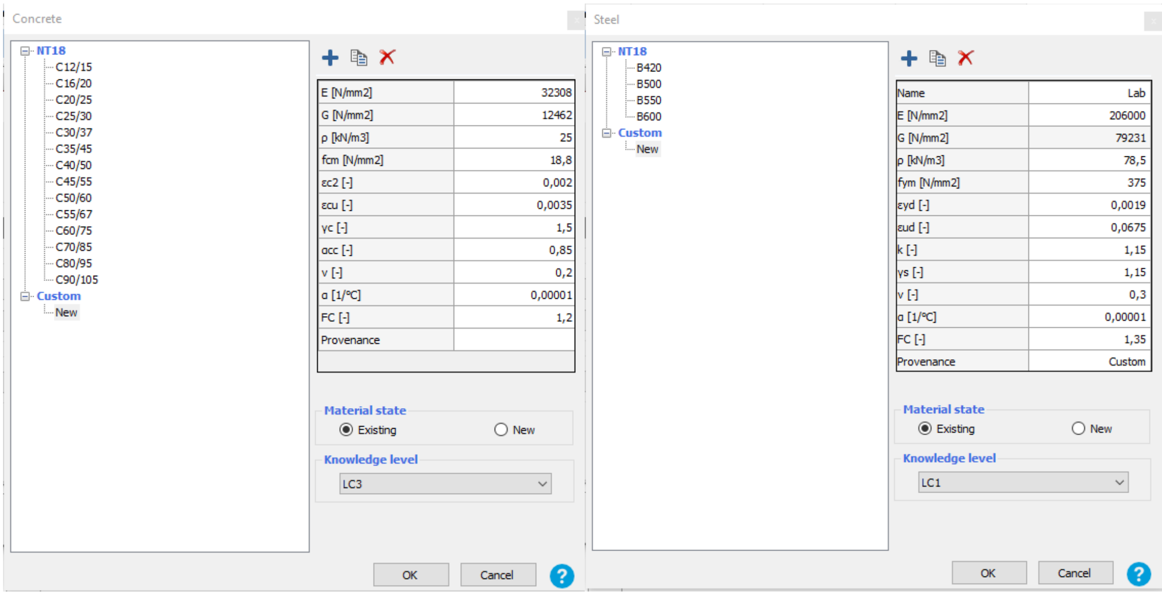
Once all the necessary data has been entered, the position of the mouse cursor (without clicking) on one of the two buttons causes the span to be created.
To insert other spans (with the same geometric and mechanical data or different, after modification), simply click on one of the two buttons.


They allow you to select the adjacent span (left or right).



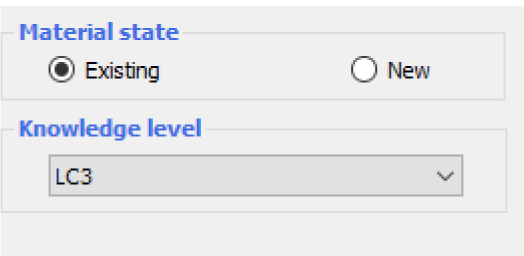
Allows you to delete the selected span.

The window shows the list of materials for concrete, steel, sheet steel (NT18), reinforcements (NT18) or thermal protection (NT18) present in the work; on the right side of the window, on the other hand, the mechanical data of the selected material are shown.
There is already a list of the most commonly used materials and these cannot be changed.

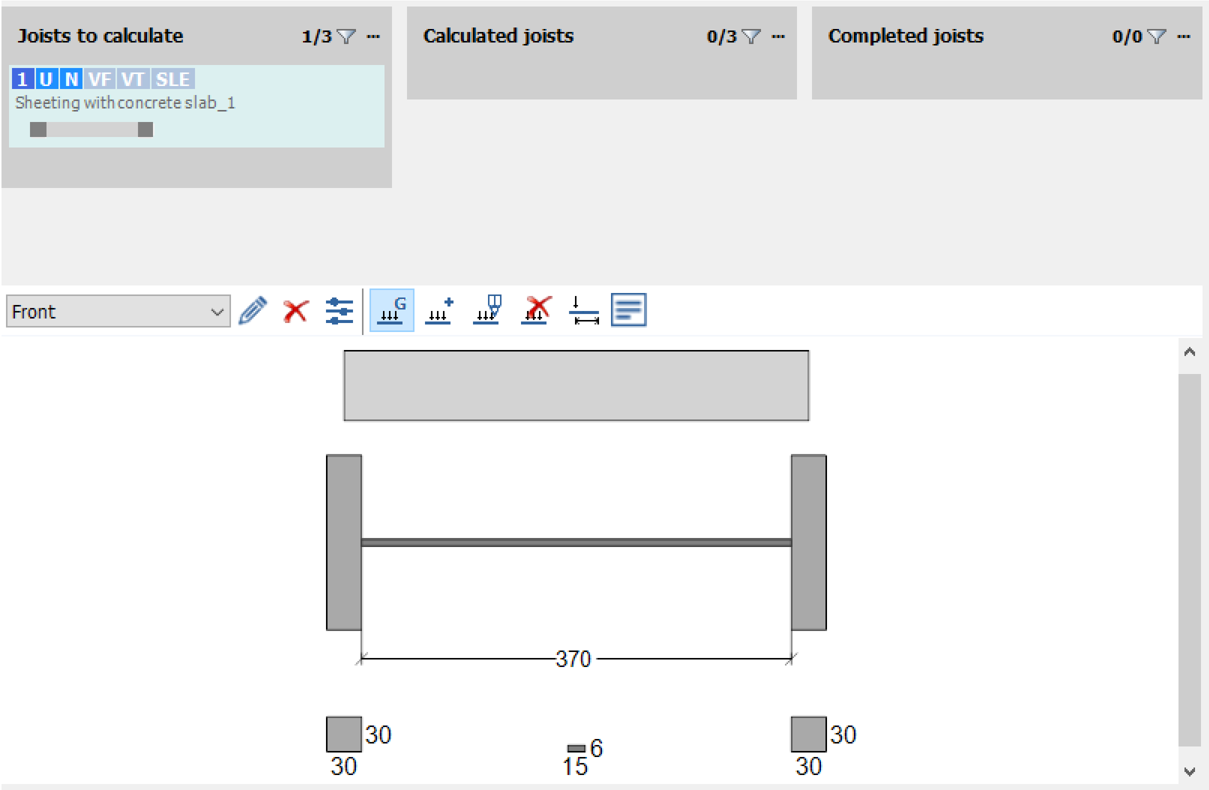


Each material can be duplicated using the command and the copy will be shown in the custom list, where it can be selected and modified using the data table on the right.

Once the customized list has been selected, it will also be possible to insert a new material (for example if laboratory data on existing materials are available) using the command ; in this case it is also possible to specify that it is existing material and the level of knowledge (LC1, LC2, LC3):



The Loads environment provides for the definition, modification and insertion of the loads acting on the spans.



It is necessary to insert the desired load case through the drop-down menu, with the possibility of quickly entering the self-weight
(G) calculated automatically and accessing the window for defining the load groups.

The type of load includes:
- linear load;
- concentrated load;
- pair on knot;
- surface load.

Once the case and type of load have been set, the button allows you to apply it to the span.
Once applied, the environment for changing the position and value of the load is enabled.

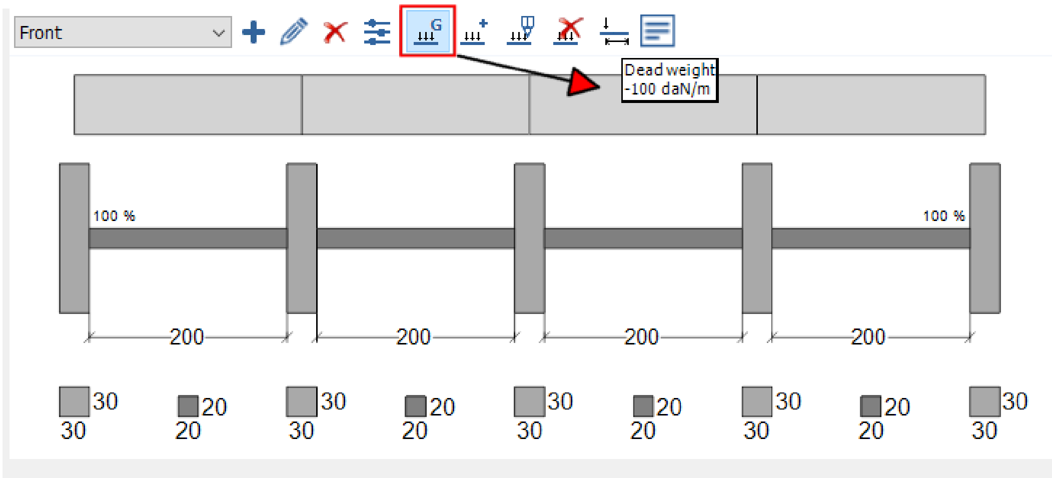
They allow you to apply the selected load to the adjacent span (left or right).

They allow you to select the adjacent span (left or right).

They allow you to select the first or last span.

Allows you to delete the selected span.

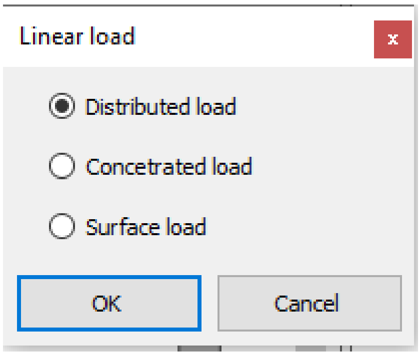
By pressing the "Dead weight" button it is possible to quickly define, on all spans, the self-weight of the element calculated automatically:



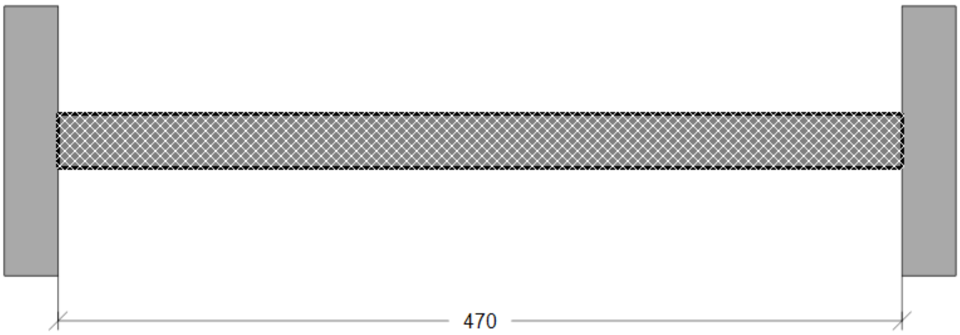
Simply reselect the button to eliminate the load entered due to its own weight.

The addition allows the insertion of a load.

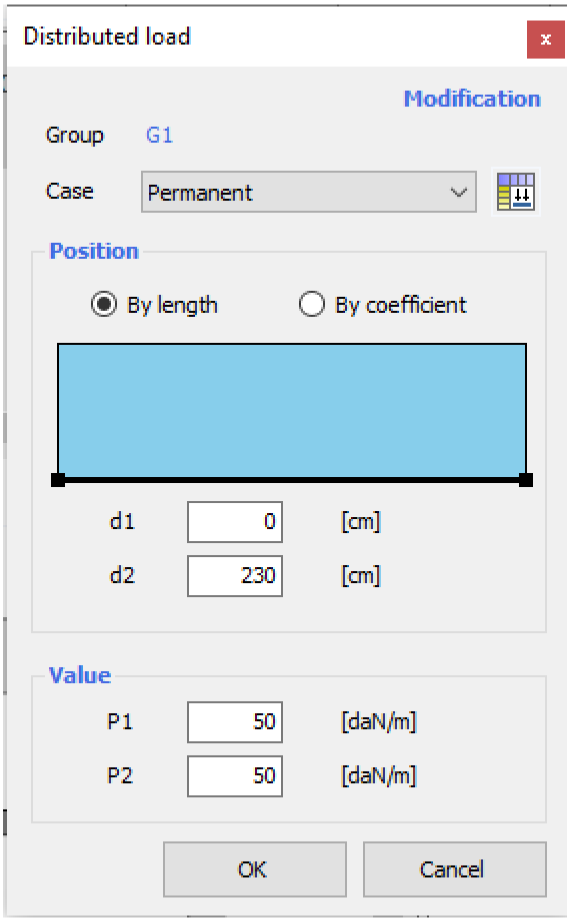
The command first requires the choice of the type of load to be inserted:



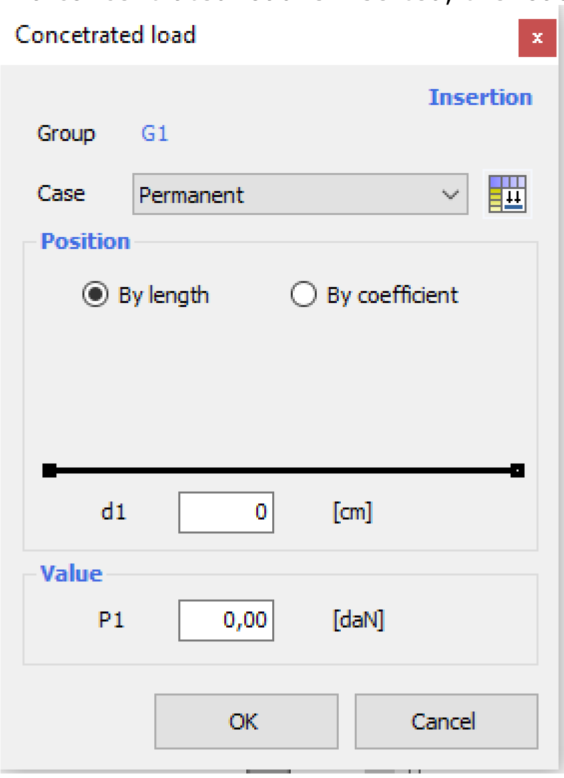
Then it is simply necessary to select the bay to be loaded with a click and confirm the choice by clicking on the right mouse button again:



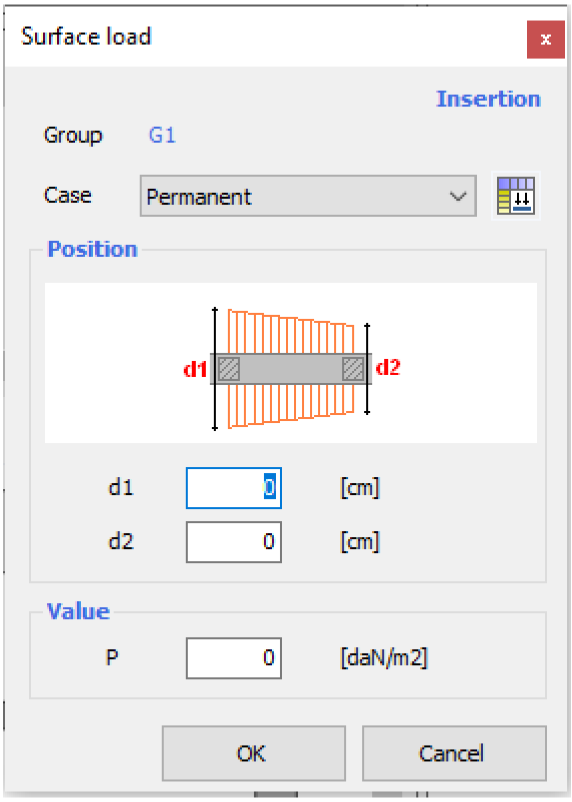
If a distributed load is entered, the load is set through the following menu.



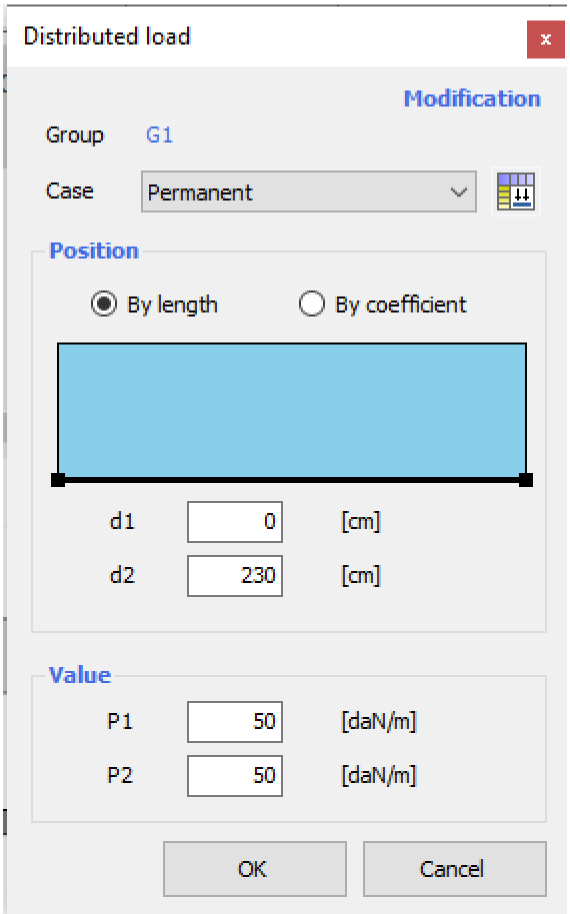
If a concentrated load is inserted, the load is set through the following menu.

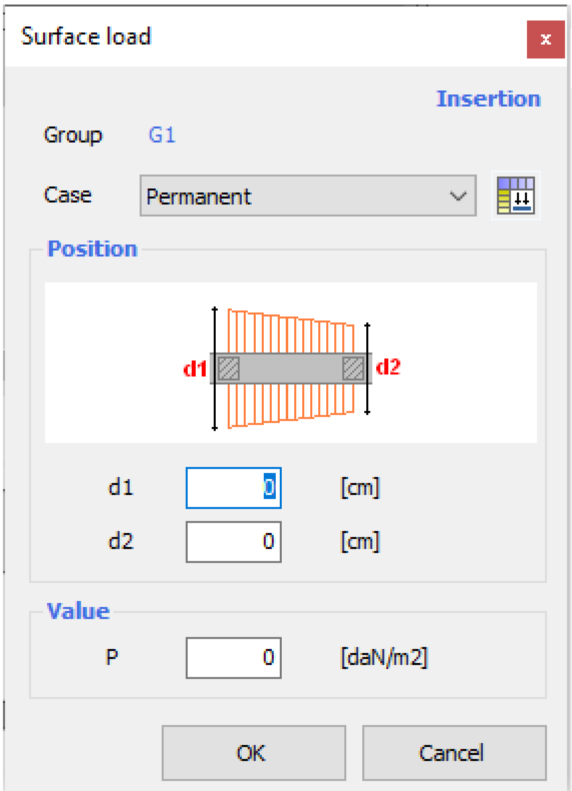


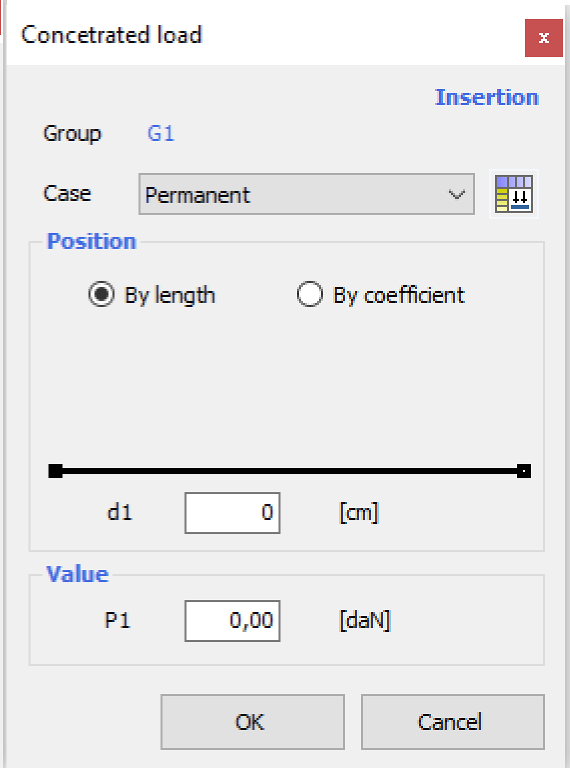
If a surface load is entered, the load is set through the following menu.



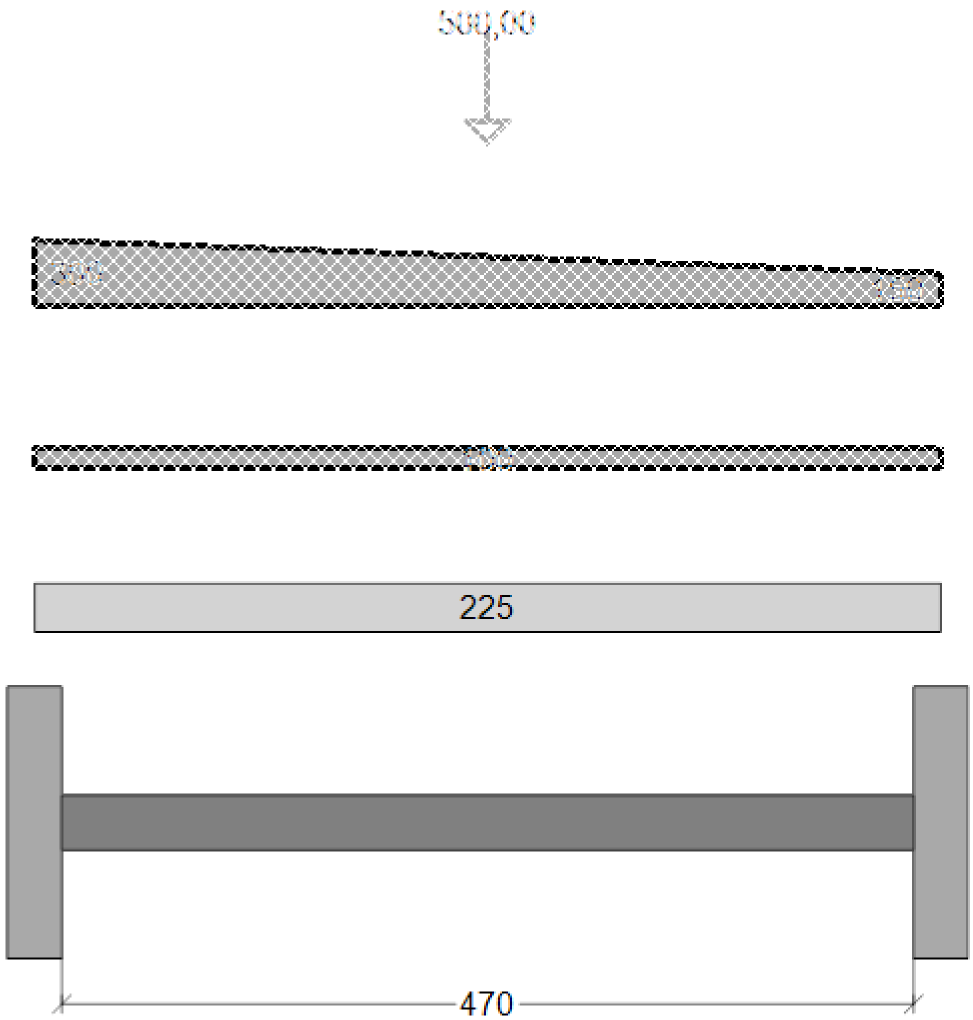
To edit an added load, simply select the load to be modified with a click and set the new parameters using the appropriate menu corresponding to the type of load for which the modification is necessary.







To delete an existing load it is necessary to select the load to be deleted with a click:



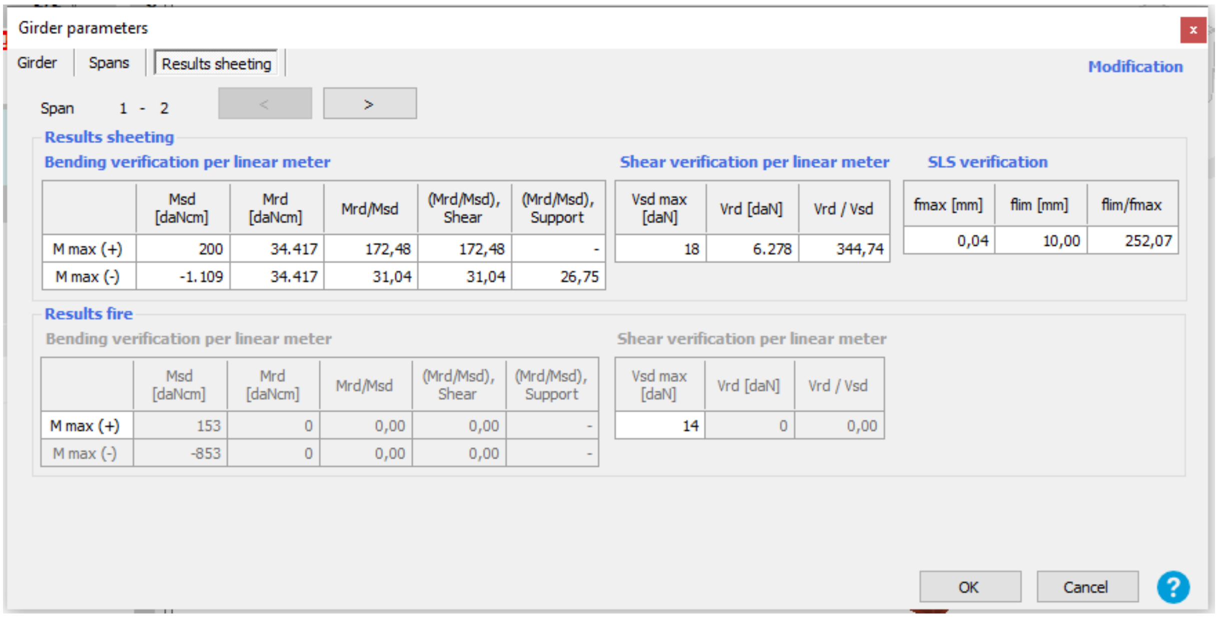
And finally confirm the deletion with a right click.

11.5.3.2.1.2 Results

When the beam "tab" is dragged from the "Joists to calculate" column to the "Calculated joists" column, it will be possible to view the calculation results.

Corrugated sheet

The Sheet metal results environment, available when a beam is in the Calculated sheets tab, reports the outcome of the ULS and SLS checks carried out on the corrugation.
In particular, the results obtained with regard to bending (per linear meter), shear (per linear meter) and deformation can be consulted.
You can navigate on the spans belonging to the girder by means of the appropriate buttons.



Corrugated sheet with collaborating slab

The Sheet Metal Results environment with slab, available when a beam is in the Calculated Sheets tab, reports the outcome of ULS and SLS checks conducted on the corrugated sheet with slab.
In particular, the results obtained can be consulted regarding the verification of bending (per linear meter), transversal cut (per linear meter), longitudinal cut according to the criterion chosen in the calculation and deformation parameters.
Navigation by spans is prevented as the check is carried out on the single span, as illustrated in the paragraph relating to the corrugated sheet with slab in the consolidated phase.

Corrugated sheet

The Fire results environment, available when a sheet is found in the Calculated sheets tab (if a calculation under fire conditions was previously set in the spans tab), reports the result of the fire checks carried out on the sheeting.

In particular, the results obtained with regard to the bending (per linear meter) and shear (per linear meter) verification can be consulted.

You can navigate on the spans belonging to the girder by means of the appropriate buttons.

Corrugated sheet with collaborating slab

The Fire results environment, available when a sheet is found in the Calculated sheets tab (if a calculation under fire conditions has been previously set in the spans tab), reports the result of the fire tests carried out on the corrugated sheet with collaborating slab.

Navigation by spans is prevented as the check is carried out on the single span, as illustrated in the paragraph relating to the corrugated sheet with slab in the consolidated phase.

In this case the displayed results will be different depending on the fire verification criterion selected by the user in the calculation parameters window.

In the event that the chosen calculation criterion is that in the resistance domain, the proposed results will be different, depending on the fire parameters defined under the span tab in the beams to be calculated. If there is no thermal protection, then the bending tests m.l., transversal cut m.l. will be carried out. and longitudinal cut m.l. in fire conditions; if there is a thermal protection, a temperature check will be carried out where the design T must not exceed the critical T value.

If the criterion chosen by the user is the one in the temperature domain, i.e. insulation criterion, an insulation time check will be performed.

11.6 Utility

11.6.1 RC verification

11.6.1.1 Flexional bending

11.6.1.1.1 Introduction

The flexional-bending module allows the calculation of the moment of resistance of a section in reinforced concrete, according to the criteria indicated in the Ministerial Decree 17 January 2018 and Circular 21 January 2019 n.7.
It is possible to choose materials from the library or customized, thus allowing the calculation of existing elements as well.

11.6.1.1.2 Calculation procedure

The program performs three types of checks:
1- bending check of generic sections;
2- straight pressure bending check of generic sections, in which the resisting moment is given by:

$$M_{rd} = M_{rd}(N_{sd})$$

3- deflection test of circular and rectangular sections, where the calculation returns the efficiency coefficient of the section given by:

$$(M_{sd,y} / M_{rd,y})a + (M_{sd,z} / M_{rd,z})^{a}$$

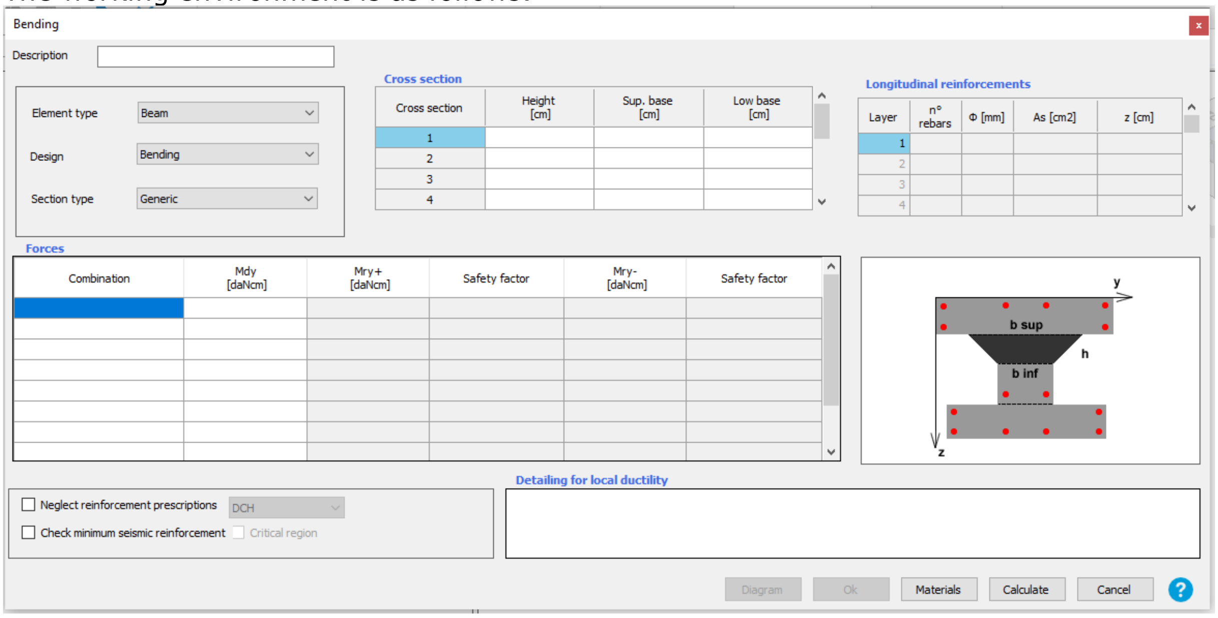
where the coefficient a is calculated with the Monti-Alessandri theory for rectangular sections and equal to 1 for circular sections.

For the bending check, it is possible to include the control of compliance with the construction provisions of the minimum and maximum reinforcement of the beams, prescribed in chapters 4 and 7 of the Ministerial Decree 17 January 2018.
For the bending check, both straight and deviated, it is possible to include the control of compliance with the construction provisions of the minimum and maximum reinforcement of the pillars, prescribed in chapters 4 and 7 of the Ministerial Decree 17 January 2018.

The calculation of the simple bending strength returns the positions of the positive and negative neutral axis, indicated in relation as distances from the upper edge of the tested section.
The calculation of the straight bending produces, in addition to the numerical results, the interaction diagram of the section.
If the soliciting moment indicated by the user does not comply with the minimum values provided for in paragraph 4.1.2.3.4.2 of Ministerial Decree 17-01-18, the program in any case checks with respect to the minimums provided for.

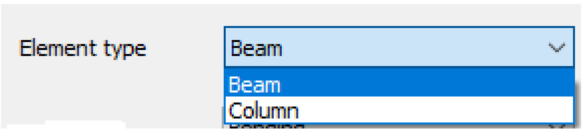
11.6.1.1.3 Work environment

The working environment is as follows:



11.6.1.1.3.1 Main area

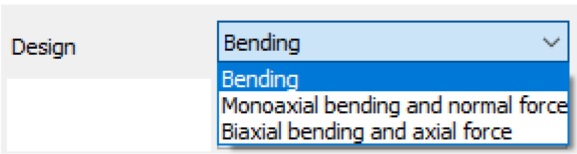
The main input window is divided into 7 sections.



1. Element type

It is possible to choose between:

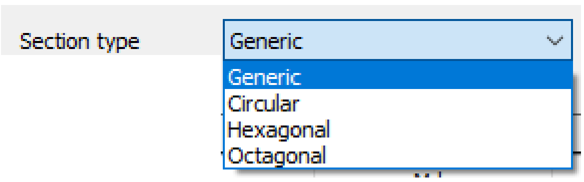
- Beam
- Column



2. Type of verification

It is possible to choose between verification at:

- Simple bending
- Monoaxial bending and normal forces
- Biaxial bending and axial force

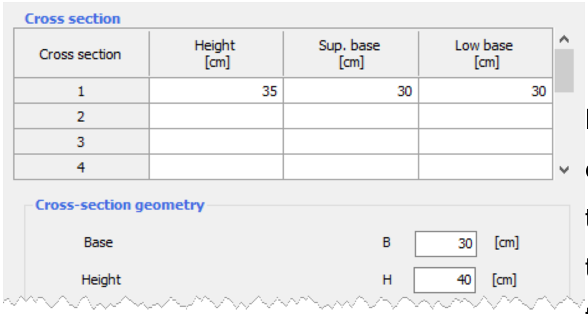


3. Type of section

It is possible to choose between generic or predetermined shape sections, such as: circular, hexagonal and octagonal.

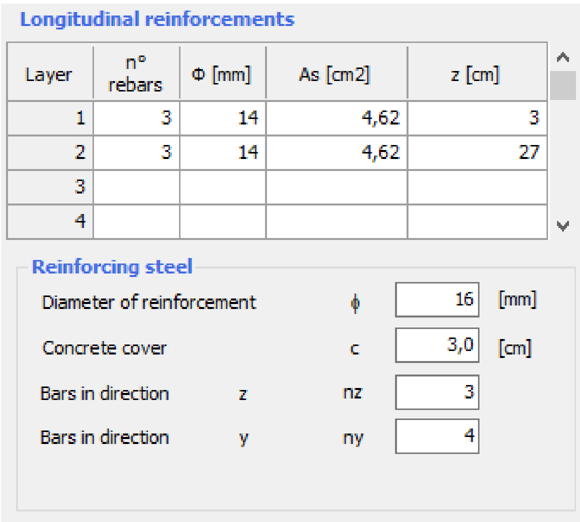
For the biaxial bending verification the choice is limited to the circular and rectangular sections.

4. Section



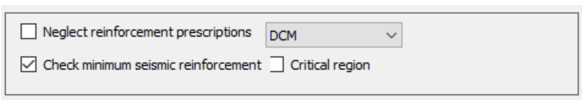
For the generic section, the geometry must be entered as a succession of sections; of each it is necessary to indicate the height, the width of the upper side and the width of the lower side. By completing the grid lines, the **Section Scheme** is automatically updated.

5. Longitudinal reinforcement



For the generic section, the longitudinal reinforcement must be inserted as a succession of layers; indicating the number of bars, the diameter and the position, with respect to the reference system indicated, of each layer. Once the number and diameter have been assigned, the program will automatically fill in the cell with the reinforcement area. By completing the grid lines, the **Section Scheme** is automatically updated.

6. Reinforcing steel check



It is possible to verify that the geometry and reinforcement comply with the constructive requirements of the legislation for the pillars and beams. If a beam is being checked, it will be necessary to to indicate whether the details are being checked for a critical region or not**.**

Forces

Combination	Ned [daN]	Mdy [daNcm]	Mdz [daNcm]	Nr [daN]	Mry [daNcm]	Mrz [daNcm]	Safety factor	Verified

7. Stress and analysis results

Indicate in this table the acting stresses, necessary to carry out the calculation. The "Item No." field is optional. See the specific paragraph to learn more.

Detailing for local ductility

Reinforcing steel respects the limitations for static verifications.
Reinforcing steel respects the limitations for seismic verifications.

8. Construction details

In this area will be reported the messages relating to the satisfaction or not of the construction details for the section in question.

The table for entering the stresses and displaying the results varies with the type of verification selected.
For all verifications, you can optionally enter a name to identify the verification.

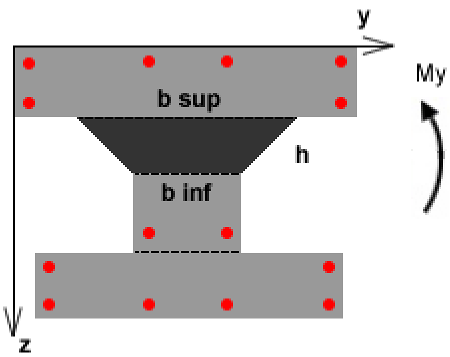
Simple bending verification

The only data necessary for this type of verification is the soliciting moment in the second column Mdy [kN m].

Forces

Combination	Mdy [daNcm]	Mry+ [daNcm]	Safety factor	Mry- [daNcm]	Safety factor
1	20	110.623	2.765,57	1.056.339	-
2	-250	110.623	-	1.056.339	4.225,36

The moment that causes compression at the upper edge of the section and traction at the lower edge is understood to be positive.



The program shows in the third column Mry + [kN m] the positive resistant moment and in the fifth Mry - [kN m] the negative resistant moment.

The safety factor, ratio between the resisting and stressing moments of the same sign, is reported in the safety factor columns:
 - in the fourth column if the soliciting moment is positive;

- in the sixth column if the soliciting moment is negative.

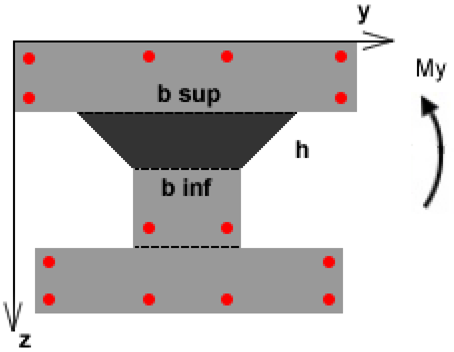
Axial bending verification

For this type of verification it is necessary to indicate the normal stressing force in the second column Nd [kN] and in the third column the stressing moment Mdy [kN m]

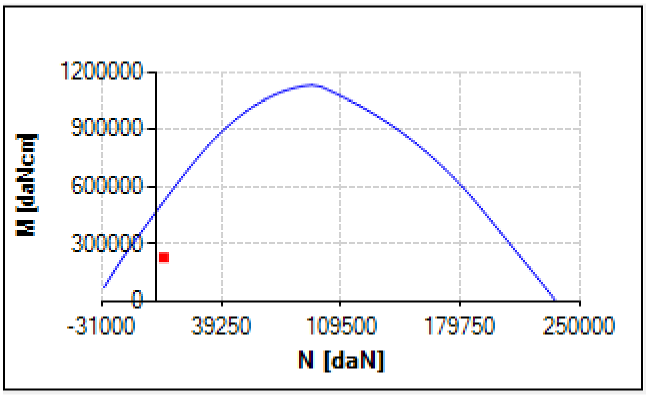
Forces

Combination	Ned [daN]	Mdy [daNcm]	Nr [daN]	Mry [daNcm]	Safety factor
1	10	5000000	162.904,38	758.671	0,15
2	50000	-2500	162.904,38	1.645.820	13,17

The moment is intended around the y axis and is intended as positive if it causes compression at the upper limb of the section and traction at the lower limb.
Normal effort is understood to be positive if it is compressive.



The program shows in the fourth column Nr [kN] the maximum resistant normal stress and in the fifth Mry [kN m] the resistant moment corresponding to the normal stressing stress.
The safety coefficient ratio between resisting moment and stressing is reported in the last column Coeff. Sic.



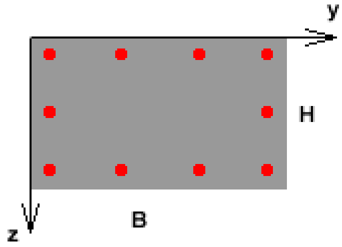
Biaxial bending verification

For this type of verification it is necessary to indicate the normal stressing stress in the second column Nd [kN], while in the third Mdy [kN m] and fourth Mdz [kN m] the stressing moments respectively around the y and z axis.

Forces

Combination	Ned [daN]	Mdy [daNcm]	Mdz [daNcm]	Nr [daN]	Mry [daNcm]	Mrz [daNcm]	Safety factor	Verified
1	10	5000000	100	350.582,01	2.010.272	2.010.272	0,09	No
2	50000	-2500	10	350.582,01	2.447.743	2.447.743	18.779,12	Yes

The moment is intended around the y axis and is intended as positive if it causes compression at the upper limb of the section and traction at the lower limb.
The moment is intended around the z axis and is intended as positive if it causes compression to the left flap of the section and traction to the right flap.
Normal effort is understood to be positive if it is compressive.



The program shows in columns from the fifth to the seventh:
- the normal effort Nr [kN] the maximum resistant normal effort;
- the resisting moment Mry [kN m] around the y axis;
- the resisting moment Mrz [kN m] around the z axis.
The coefficient calculated as:

$$(M_{dy} / M_{ry})^{a} + (M_{dz} / M_{rz})^{a}$$

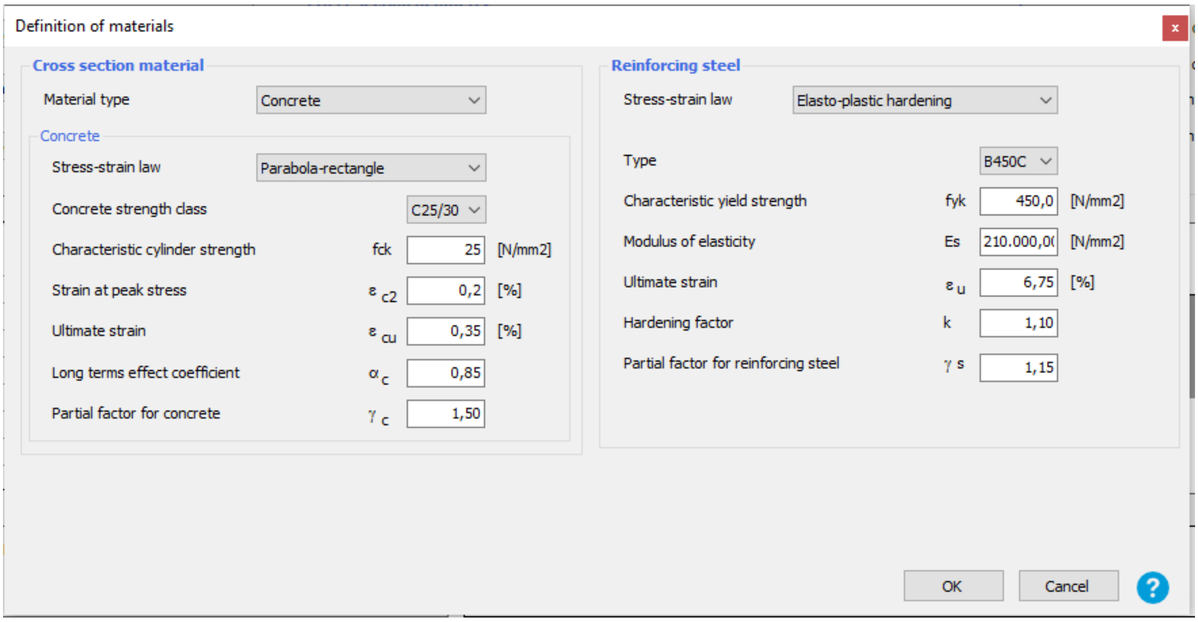
The combination is verified when the coefficient is less than unity.

Forces

Combination	Ned [daN]	Mdy [daNcm]	Mdz [daNcm]	Nr [daN]	Mry [daNcm]	Mrz [daNcm]	Safety factor	Verified
1	10	5000000	100	350.582,01	2.010.272	2.010.272	0,09	No
2	50000	-2500	10	350.582,01	2.447.743	2.447.743	18.779,12	Yes

11.6.1.1.3.2 Definition of materials

This workspace is divided into two sections:



Cross section material:

-Concrete

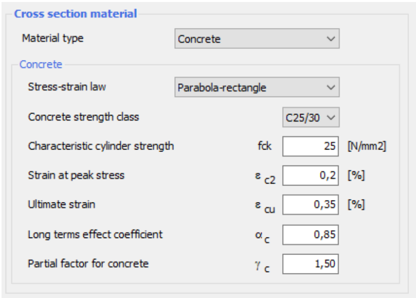
Stress-strain law

It is possible to choose between:

- Parabola-rectangle
- Stress-block
- Bi-linear.

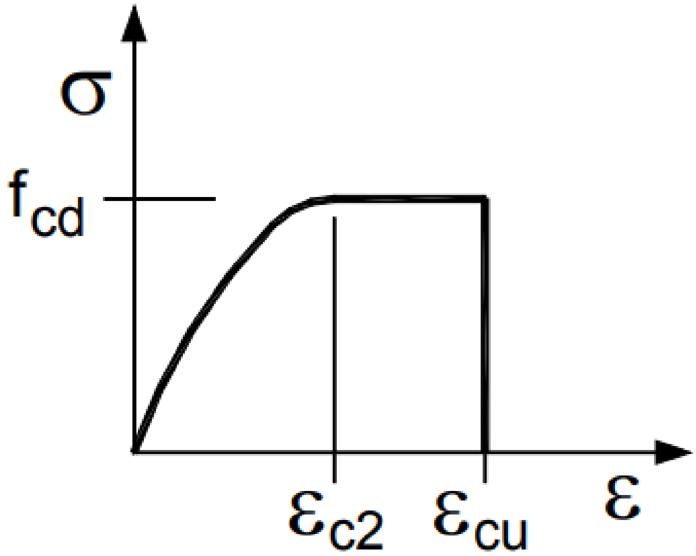
Entering the mechanical properties of the column concrete:

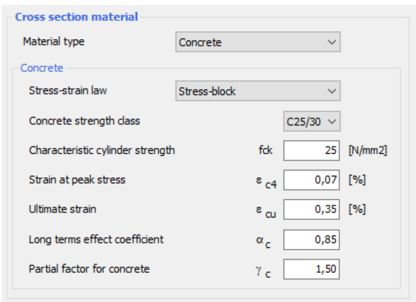
- Concrete strength class
- f_{ck}: characteristic resistance of concrete
- ε_{cu}: ultimate strain of the concrete
- γ_c: partial safety factor of concrete
- α_{cc}: concrete strength reduction coefficient for long-term loads.



Stress-strain law parabola-rectangle:

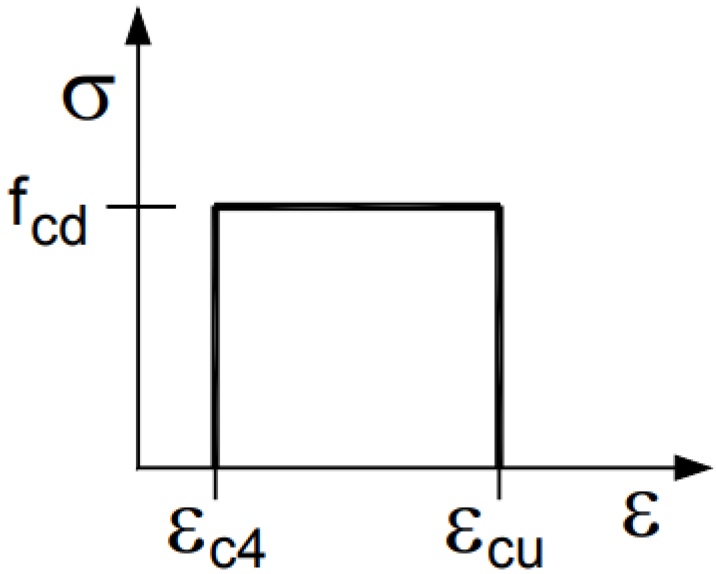
- ε_{c2} : Strain at peak stress

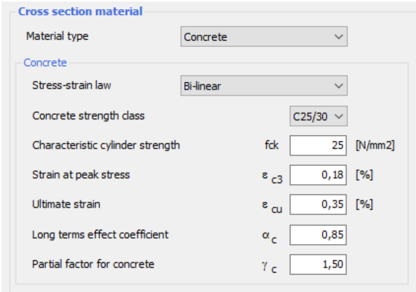




Stress-strain law Stress-block

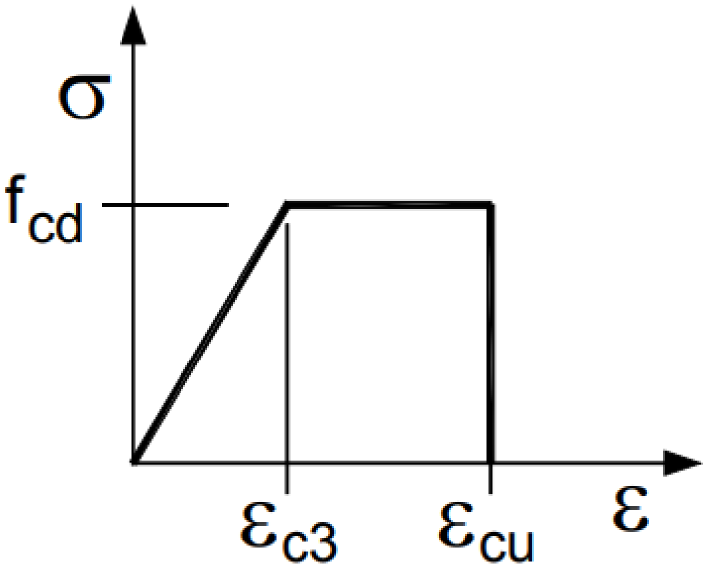
- ε_{c4} : Strain at peak stress





- Stress strain law Bi-linear :

- ε_{c3} : strain at peak stress



-Masonry

Stress-strain law

It is possible to choose between:

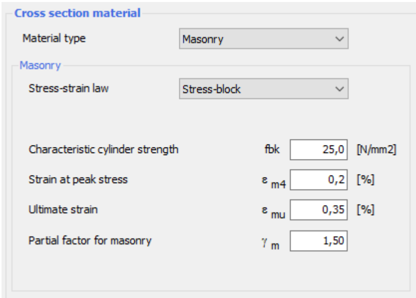
- Stress-block
- Triangle-rectangle.

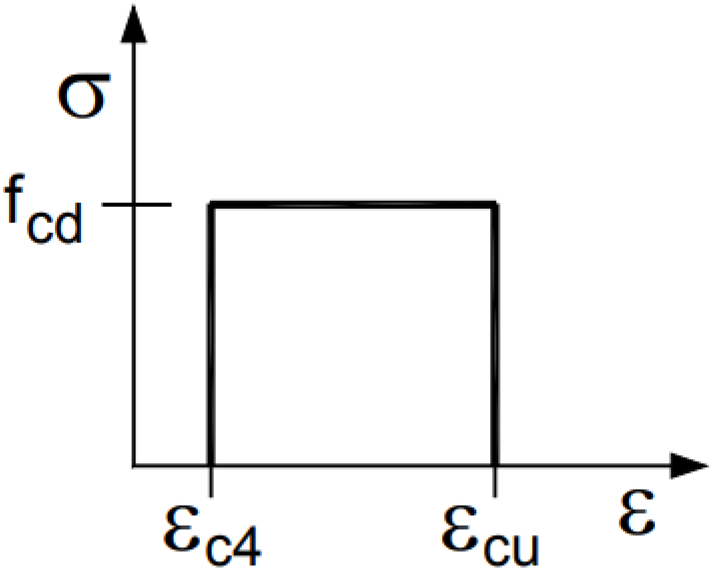
Entering the mechanical properties of the element masonry:

- f_{bk}: characteristic resistance of the masonry
- ε_m: masonry yield deformation
- ε_{mu}: ultimate strain of the masonry
- γ_m: partial safety factor of the masonry

- Stress-strain law Stress-block

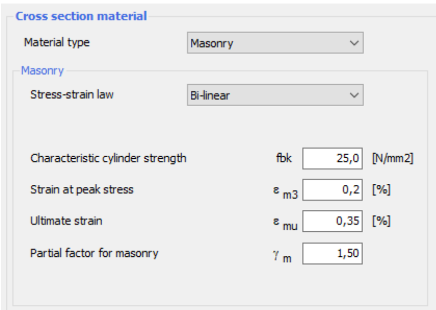
- ε_{m4} : strain at peak stress of masonry

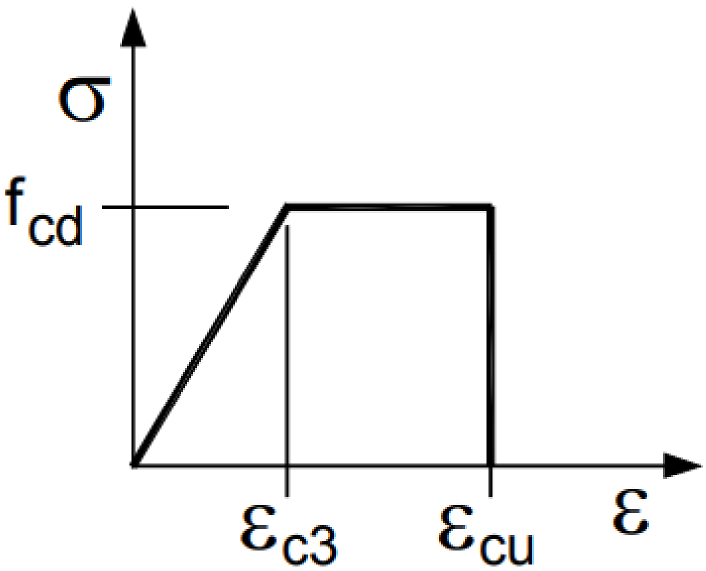




- Stress-strain law Bi-linear:

- ε_{m3} : strain at peak stress of masonry





-Steel:

Stress-strain law

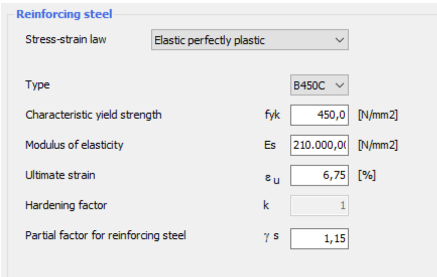
It is possible to choose between:

- Elastic-perfectly plastic
- Elasto-plastic hardening
- Elasto-brittle.

Entering the mechanical properties of steel:

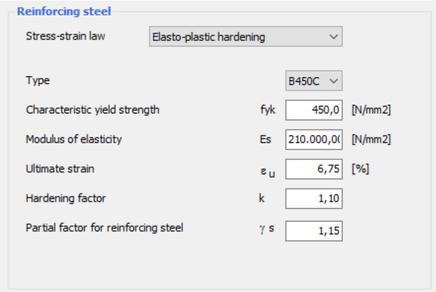
- Type: steel type
- f_{yk}: characteristic resistance of steel
- γ_s: partial safety factor of steel
- ε_u: ultimate strain of steel
- E_s: elastic modulus of steel
- k: plastic hardening coefficient of steel

- Elastic-perfectly plastic

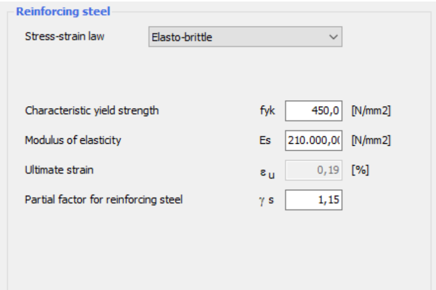


The hardening coefficient k = 1.

- Elasto-plastic hardening



- Elasto-brittle



There is no plasticization, therefore $\varepsilon u = \varepsilon y = fyd / Es$.

11.6.1.2 Shear

11.6.1.2.1 Introduction

The ULS Shear Beam Columns module calculates the value of the maximum shear at ultimate limit state for beams, columns or joists, according to NTC18.
It is possible to check the construction details established by the standard (minimum / maximum reinforcement, bracket steps, etc.), both with seismic and non-seismic criteria.
For elements with transverse reinforcement, the calculation is performed with the method of compressed connecting rods with variable inclination.

11.6.1.2.2 Calculation procedure

The program evaluates the shear strength according to the prescriptions of §4.1.2.3.5 of the D.M. 17 January 2018. In particular in the absence of specific shear reinforcement, it is evaluated according to the arch-tie mechanism, in which the following intervene:

- the compressive strength of concrete;
- the meshing effect of the aggregates;
- the pin effect of the longitudinal armature;
- the possible presence of a compressive stress that increases the contribution offered by the tangential stresses in the concrete of the compressed area.

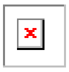
In the presence of shear reinforcement, the resistance is evaluated according to the Ritter-Mörsch lattice mechanism, according to which the element under analysis is schematized as an ideal reticular beam consisting of a compressed and a tense current, completed by compressed connecting rods and rods. The method provides that the inclination of the compressed connecting rods q is variable within the limits:

$$1 \leq \text{ctg } q \leq 2.5$$

The overall strength is given by the minimum between the compressive strength of the compressed connecting rods and the tensile strength of the tie rods, and must in any case be greater than or equal to the strength calculated according to the arch-tie mechanism.

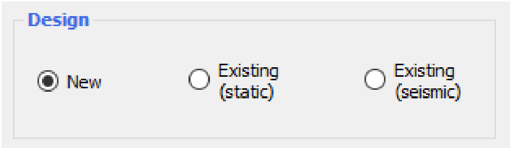
11.6.1.2.3 Work environment

The working environment is as follows:



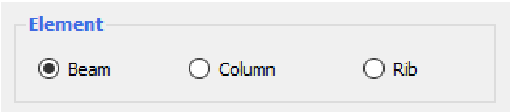
11.6.1.2.3.1 Main area

The main input window is divided into several sections.



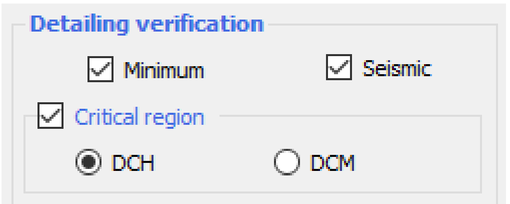
Design type

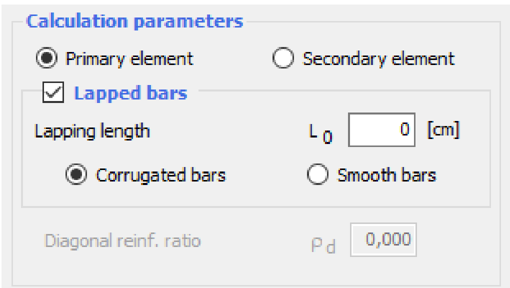
In this section, select the type of design you intend to conduct.



Element

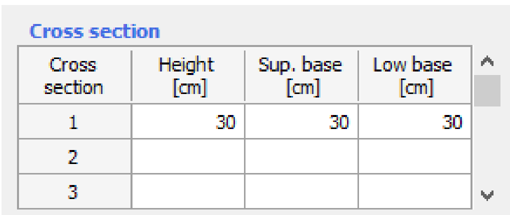
Indicate in this section the element you want to check. Keep in mind that for the verification of the Beam or Column the presence of transverse reinforcement is still necessary even if you have chosen not to verify the construction details. To check an element without transverse reinforcement, use the Rib element.





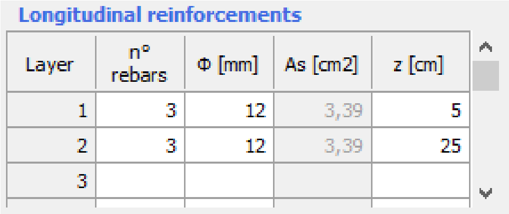
Detailing verification details / Calculation parameters

By selecting the different items available, the program verifies compliance with the construction details such as transverse reinforcement and section dimensions.
If the verification against the actions of the earthquake has been selected, the Calculation parameters section will require the insertion of further data necessary to start the analysis.



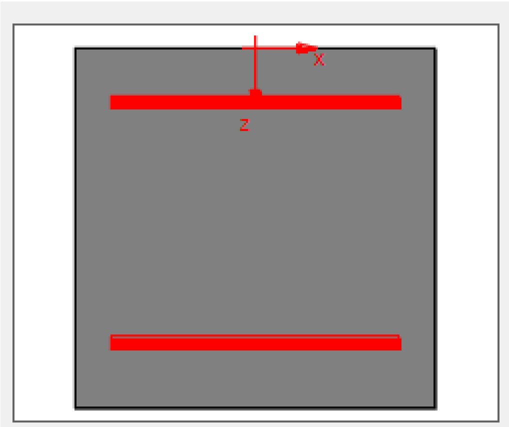
Cross section

The geometry of the section under analysis must be entered as a succession of sections; of each it is necessary to indicate the height, the width of the upper side and the width of the lower side.
As the grid lines are completed, the Section Outline is automatically updated.



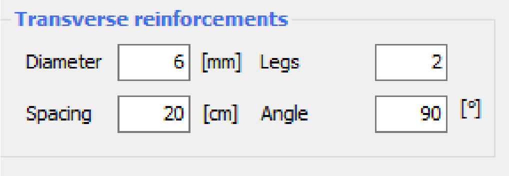
Longitudinal reinforcements

The reinforcement is defined by indicating the number of bars, the diameter and the position, with respect to the reference system indicated, of each layer. Once the number and diameter have been assigned, the program will automatically fill in the cell with the reinforcement area. The latter will be drawn on the screen in order to verify the input.
As the grid lines are completed, the Section Outline is automatically updated.



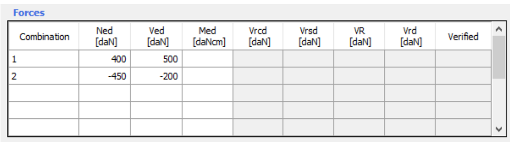
Section diagram

Automatically updated when data is entered in the Section geometry and Longitudinal reinforcement grids, it allows you to quickly check the data entered.



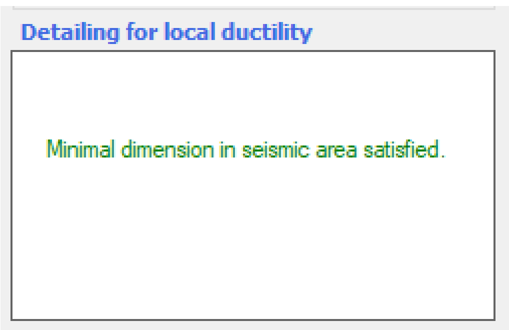
Transverse reinforcements

It is mandatory to complete this section in the case of a Beam or Column verification; on the other hand, it cannot be edited in the case of a Rib verification.
Under "Angle", indicate the angle of the stirrups with respect to the axis of the element under analysis.



Forces

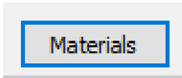
Indicate in this section the project forces for each combination.
The remaining columns will be completed by the program following the start of the calculation.
Normal effort is understood to be positive if it is compressive.



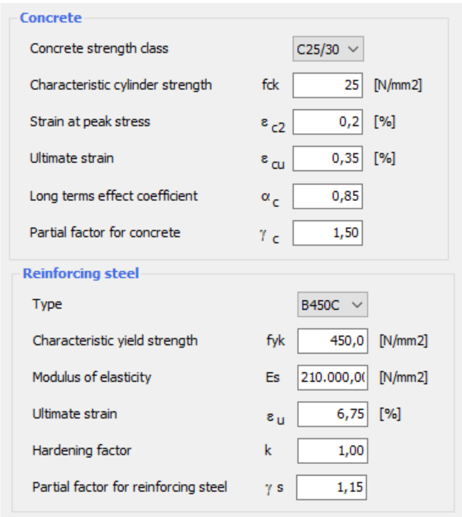
Results of the analysis

This section shows the results of the checks carried out.

11.6.1.2.3.2 Definition of materials

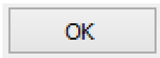
To access this area use the button 

This work area is divided into 2 sections:



The first section is devoted to the mechanical parameters of concrete.
It is possible to select between the types of concrete stored in the program or manually enter the characteristic resistance by selecting the "Other" item in the Concrete class menu.

The first section is dedicated to the mechanical parameters of steel.
It is possible to choose between the types of steel stored in the program or manually enter their characteristics by selecting the "Other" item in the Steel type menu.

To return to the Main Area, press the key 

11.6.1.3 Cracking

The TA cracking beam module calculates the first cracking moment of the beam, a stress for which the concrete no longer has an elastic behavior as it begins to crack.
Furthermore, the module determines the stress state inside the section calculated with the method of admissible stresses.

11.6.1.3.1 Calculation procedure

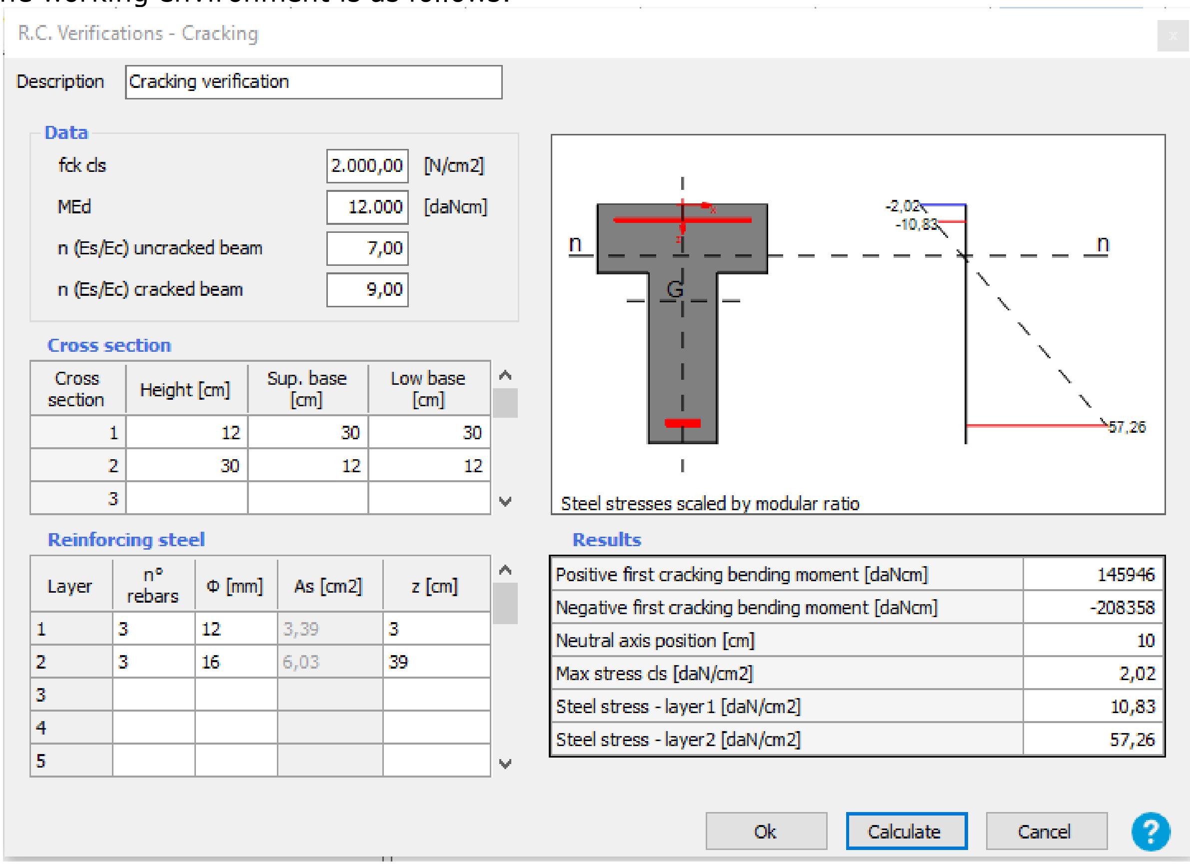
The program considers linear stress-strain bonds for both concrete and steel.
For the calculation of the moment of first cracking, the section under analysis is considered as fully reactive. Having calculated the moment of inertia and the center of gravity of the homogenized section of the concrete, it is possible to evaluate the moment of first cracking with the formula of pressure bending, setting the ultimate tensile stress of the concrete at the end.

$$\sigma = \frac{M}{J_{om}} z \rightarrow M_{cr} = \frac{f_{ctm,fl}}{z} J_{om}$$

The calculation of the stresses in response to a stressing moment is carried out with the method of admissible stresses, considering the tensile strength of the concrete as long as the moment is lower than that of the first crack.

11.6.1.3.2 Work environment

The working environment is as follows:



11.6.1.3.2.1 Main area

The main input window is divided into several sections.

Cross section

Cross section	Height [cm]	Sup. base [cm]	Low base [cm]
1	12	30	30
2	30	12	12
3			

Cross section

The geometry of the section under analysis must be entered as a succession of sections; of each it is necessary to indicate the height, the width of the upper side and the width of the lower side.

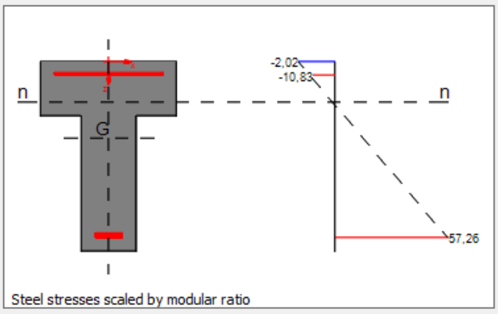
As the grid lines are completed, the Section Outline is automatically updated.

Reinforcing steel

Layer	n° rebars	Φ [mm]	As [cm2]	z [cm]
1	3	12	3,39	3
2	3	16	6,03	39
3				
4				
5				

Reinforcing steel

The longitudinal reinforcement of the section under analysis must be inserted by layers, indicating the number of bars, the diameter and the position, with respect to the reference system indicated, of each layer. Once the number and diameter have been assigned, the program will automatically fill in the cell with the reinforcement area. The latter will be drawn on the screen in order to verify the input.
As the grid lines are completed, the Section Outline is automatically updated.



Section diagram

Automatically updated when data is entered in the Section geometry and Longitudinal reinforcement grids, it allows you to quickly check the data entered.
When the analysis is completed, the section diagram is completed by the indication of the center of gravity G, of the neutral axis n for the indicated moment stress, and the stress diagram.

Data

fck cls	2.000,00	[N/cm2]
MEd	12.000	[daNcm]
n (Es/Ec) uncracked beam	7,00	
n (Es/Ec) cracked beam	9,00	

Characteristics of concrete

In this section indicate the characteristic strength of the concrete fck, the stressing moment for which the stress state (calculated at the admissible stresses) of the beam is sought, and the steel-concrete homogenization coefficients in non-cracked and cracked beam conditions.

Results

Positive first cracking bending moment [daNcm]	145946
Negative first cracking bending moment [daNcm]	-208358
Neutral axis position [cm]	10
Max stress cls [daN/cm2]	2,02
Steel stress - layer1 [daN/cm2]	10,83
Steel stress - layer2 [daN/cm2]	57,26

Results of the analysis

This section shows the positive moment of first cracking (fibers stretched on the lower side of the beam) and negative (fibers stretched on the upper side), the position of the neutral axis, the stresses in the concrete and in the steel layers for the stressing moment. indicated (calculated at allowable stresses).

11.6.2 RC Structure reinforcement

11.6.2.1 Excentric axial force reinforcement

11.6.2.1.1 Introduction

The Column Circles module allows the calculation of the resistance to bending of a reinforced bending column, by means of layers of FRP / FRCM arranged parallel to the main axis of the reinforced element, and by means of steel or FRP / FRCM rims, in

accordance with Circular no. 7 of 21 January 2019, CNR-DT R1 / 2013 and CNR-DT 215/2018.
It is possible to choose between the different configurations of the aforementioned reinforcements.
The materials (concrete, reinforcement and reinforcement steel, FRP) can be chosen from the available libraries or can be customized, thus giving the opportunity to represent the conditions of the current state of the pillar.

11.6.2.1.2 Calculation procedure

The program performs a straight or deviated bending check according to the choice made by the user; in the first case, verification is given by checking that:

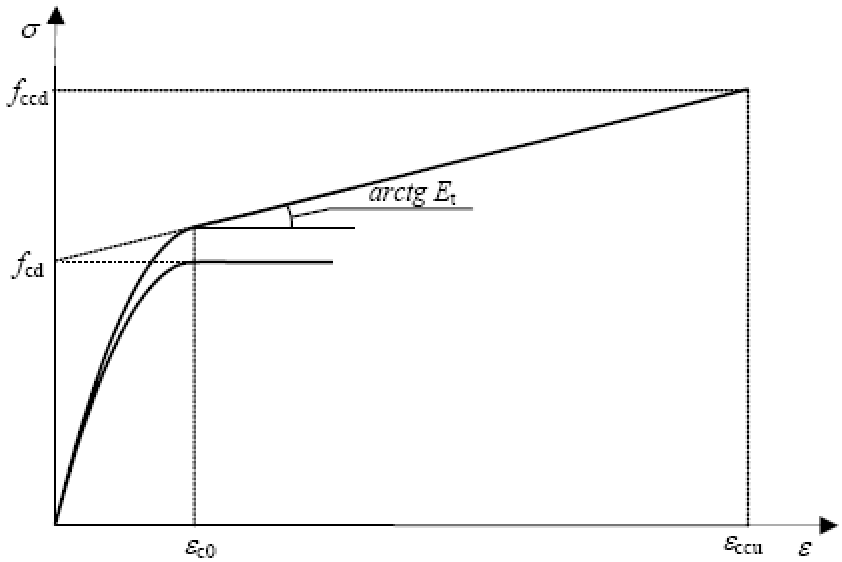
$$M_{rd} = M_{Rd}(N_{Ed}) \leq M_{Ed}$$

while in the second the verification is given by:

$$(M_{Eyd} / M_{Ryd})^{\alpha} + (M_{sd,z} / M_{rd,z})^{\alpha} \leq 1$$

as indicated in §4.1.2.3.4.2 of the Ministerial Decree January 17, 2018.

It is therefore important to underline that, in the case of simple bending, a safety coefficient ≥ 1 must be expected in relation to satisfied checks, while in the case of deviated bending, a safety coefficient ≤ 1 must be expected.
The presence of the hoop, whether made of steel or fiber-reinforced material, produces a confining effect on the concrete, which results in an increase in compressive strength and ultimate deformation at break.



Stress-strain model of confined concrete

The effectiveness of the confinement varies with the shape of the column section (horizontal efficiency, maximum if the sides of the column are the same) and the percentage of longitudinal encirclement (vertical efficiency, maximum in the case of continuous wrapping and decreasing with use of discontinuous bands). In the case of hooping made with FRP, the inclination of the fibers with respect to the axis of the pillar also affects the efficiency of the confinement: it is maximum if the fibers are perpendicular to the longitudinal axis of the pillar.
The longitudinal FRP reinforcement allows to increase the tensile strength of the flap to which it is applied. The limit moment can be reached by breaking the concrete at the

compressed edge or by breaking by traction at the stretched edge.
The activation of the reinforcement action in the nodal zones must be ensured through the adoption of suitable construction solutions. Furthermore, the longitudinal fibers used for the pressure bending reinforcement will be adequately confined in order to avoid their detachment and the expulsion of the support material.

Longitudinal reinforcement

The longitudinal FRP reinforcement reacting to traction causes an improvement in the performance of the section. The limit moment can be reached by breaking the concrete at the compressed edge or by breaking by traction of the fiber at the stretched edge. The activation of the action of the reinforcement in correspondence of the nodal zones will be ensured through the adoption of suitable constructive solutions. In addition, the longitudinal fibers used for the pressure bending reinforcement will be adequately confined in order to avoid the detachment of the same and the expulsion of the support material. Under these conditions, it is assumed that bending failure occurs in conjunction with one of the following conditions:

- achievement of maximum plastic deformation in compressed concrete;
- achievement of a maximum deformation in the FRP reinforcement, ε_{fd} calculated as:

$$\varepsilon_{fd} = min\left\{\eta_a \cdot \frac{\varepsilon_{fk}}{\gamma_f}, \varepsilon_{fdd}\right\}$$

Where:

- ε_{fk} is the characteristic deformation at break of the reinforcement
- γ_f is a partial factor for resistance models (Table 3-1, CNR-DT 200 R1 / 2013)
- η_a is the environmental conversion factor (Table 3-2, CNR-DT 200 R1 / 2013)
- f_{yd} is the maximum deformation for intermediate detachment

Steel wrapping

The confinement effect brought about by the reinforcement manifests itself as an increase in the compressive strength and ultimate deformation of the concrete of the element under analysis. According to Circular no. 7 of 21 January 2019 this effect is assessed according to:

$$f_{cc} = f_c \left[1 + 3{,}7\left(\frac{0{,}5\alpha_n\alpha_s\rho_s f_y}{f_c}\right)^{0{,}86}\right]$$

$$\varepsilon_{cu} = 0{,}0035 + 0{,}5\frac{0{,}5\alpha_n\alpha_s\rho_s f_y}{f_{cc}}$$

where:

$$\alpha_n = 1 - \frac{(b-2R)^2+(h-2R)^2}{3bh}$$

$$\alpha_s = \left(1 - \frac{s-hs}{2b}\right)\left(1 - \frac{s-hs}{2h}\right)$$

With continuous shirt:

$$\rho_s = \frac{2(b+h)t_s}{bh}$$

With discontinuous bands:

$$\rho_s = \frac{2A_s(b+h)}{bhs}$$

represent, respectively, the efficiency factors of confinement in the section, along the element, and the volumetric ratio of transverse reinforcement.

FRP Wrapping

An adequate confinement of the elements in reinforced concrete can determine an improvement in the performance of the structural element, allowing to increase the ultimate strength and the corresponding ultimate deformation of the elements stressed by normal centered stress or with small eccentricity. According to the method proposed by CNR-DT 200 R1 / 2013:

$$N_{Rcc,d} = \frac{1}{\gamma_{Rd}} \cdot A_c \cdot f_{ccd} + A_s \cdot f_{yd}$$

where:

- γ_{Rd} is the partial coefficient to be assumed equal to 1.10
- A_c and f_{ccd} are respectively, the cross-sectional area of the element and the design tension of the confined concrete
- A_s and f_{yd} and are, respectively, the area and the design voltage of the metal reinforcement that may be present

$$f_{ccd} = f_{cd} \cdot \left(1 + 2{,}6 \cdot \left(\frac{f_{l,eff}}{f_{cd}}\right)^{2/3}\right)$$

where:

- f_{cd} is the design stress of the unconfined concrete
- $f_{l,eff}$ is the effective confinement pressure

$$f_{l,eff} = k_{eff} \cdot f_l$$

where:

- k_{eff} is an efficiency coefficient (<= 1)

$$k_{eff} = k_H \cdot k_V \cdot k_\alpha$$

where:

- k_H is a horizontal coefficient of efficiency
- k_V is a vertical efficiency coefficient
- k_α is an efficiency coefficient as a function of the inclination angle of the fibers
- f_l is the confinement pressure

$$f_l = \frac{1}{2} \cdot \rho_f \cdot E_f \cdot \varepsilon_{fd,rid}$$

where:

- ρ_f is the geometric percentage of reinforcement
- E_f is the normal modulus of elasticity of the material in the direction of the fibers
- $\varepsilon_{fd,rid}$ is the reduced calculation deformation of the fiber-reinforced composite

$$\varepsilon_{fd,rid} = min\left\{\frac{\eta_a \cdot \varepsilon_{fk}}{\gamma_f}; 0{,}004\right\}$$

Where η_a and γ_f are, respectively, the environmental conversion factor and the partial coefficient of
fiber-reinforced composite material.

- For rectangular sections

$$\rho_f = \frac{2 \cdot t_f \cdot (b + h) \cdot b_f}{b \cdot h \cdot p_f}$$

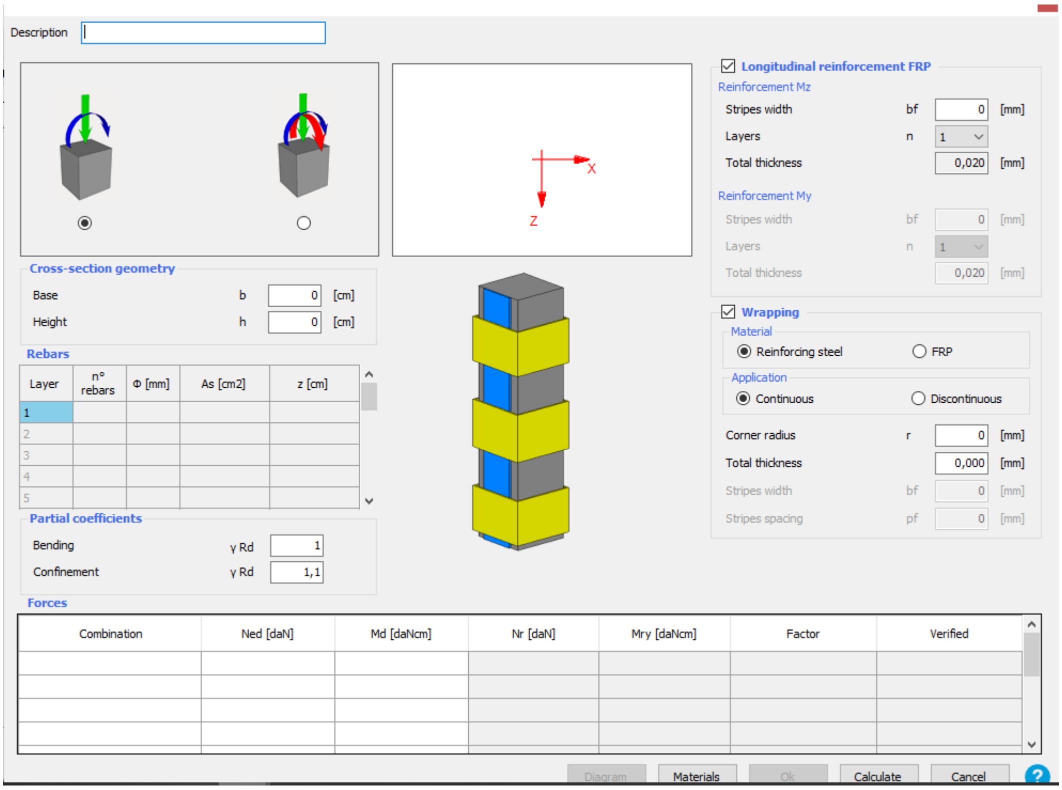
where t_f and b_f are, respectively, the thickness and height of the generic FRP strip, p_f is the spacing
of the strips, while b and h are the transverse dimensions of the rectangular section.

In the case of continuous confinement:

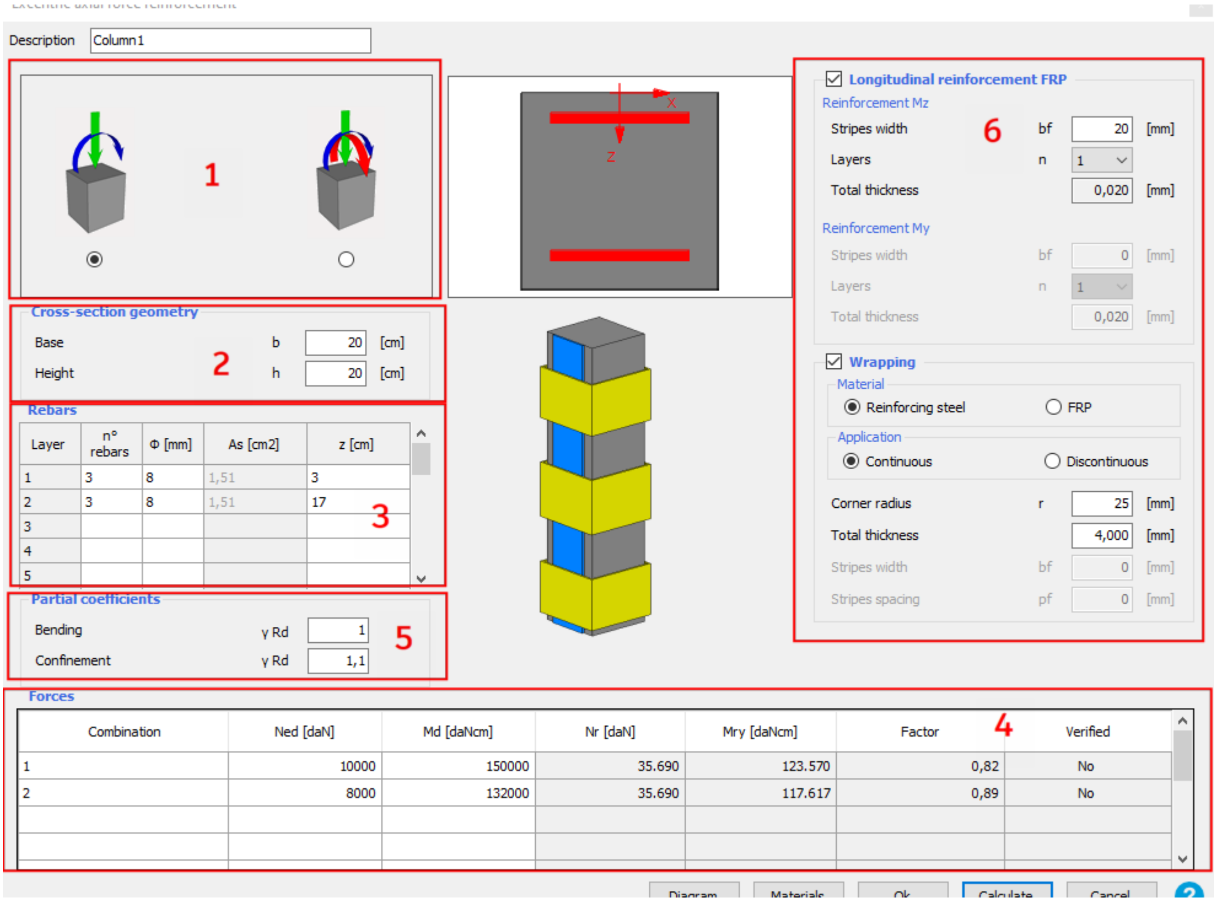
$$\rho_f = \frac{2 \cdot t_f \cdot (b + h)}{b \cdot h}$$

11.6.2.1.3 Work environment

The working environment is as follows:

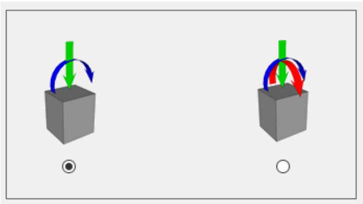


The main area is divided into 6 sections:



1. Selection of the type of verification
2. Column geometry
3. Rebars of the pillar
4. Forces and resulting resistances
5. Definition of the partial coefficients of the mechanisms
6. Features of the reinforcement

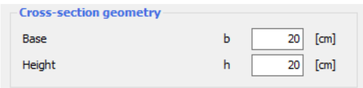
11.6.2.1.3.1 Main area



Selection of the type of verification

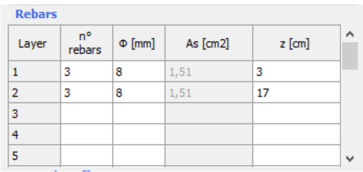
In the first section, select the type of verification to be carried out:

- for the straight bending check click on the first image;
- to check the deflection deflection click on the second.



Column geometry

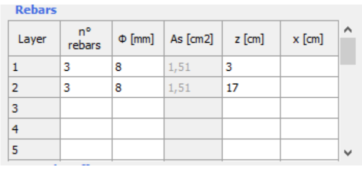
In the second section indicate the dimensions of the pillar.


Rebars

Layer	n° rebars	Φ [mm]	As [cm2]	z [cm]
1	3	8	1,51	3
2	3	8	1,51	17
3				
4				
5				

Straight bending column reinforcement

The reinforcement is defined by indicating the number of bars, the diameter and the position, with respect to the reference system indicated, of each layer. In the case of straight bending, the distance z alone is sufficient for the analysis. Once the number and diameter have been assigned, the program will automatically fill in the cell with the reinforcement area. The latter will be drawn on the screen in order to verify the input.

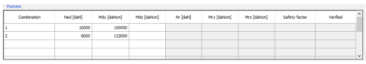


Rebars

Layer	n° rebars	Φ [mm]	As [cm2]	z [cm]	x [cm]
1	3	8	1,51	3	
2	3	8	1,51	17	
3					
4					
5					

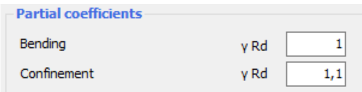
Deflected bending column reinforcement

The reinforcement is defined by indicating the number of bars, the diameter and the position, with respect to the reference system indicated, of each layer. The x indicates the distance from the axis of symmetry of the furthest iron in the series, the program will arrange the internal bars at a constant pitch. If the layer has a single reinforcement bar, this will necessarily be fixed on the symmetry axis. Once the number and diameter have been assigned, the program will automatically fill in the cell with the reinforcement area. The latter will be drawn on the screen in order to verify the input.



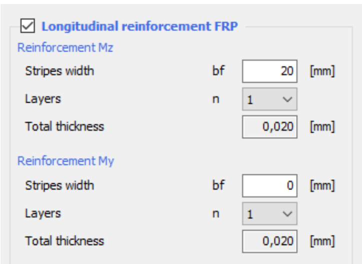
Stress agents and resulting resistances

In the fourth section indicate the name of the combination and the normal moment and effort value.
In the case of deviated bending, the bending moments acting in both directions must be inserted.
The calculation returns the resistant values and the relative safety factors.



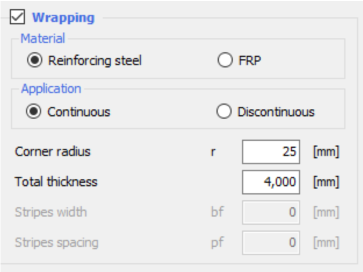
Partial coefficients

In the fifth section indicate the value of the partial coefficients for the bending / pressure bending and confinement mechanism. The default values are indicated in paragraph 3.4.2 CNR-DT R1 / 2013



Reinforcement features - Longitudinal reinforcement

The longitudinal reinforcement is to be made in FRP, whose characteristics are to be defined in the material panel.
In this space it is possible to choose the width and the number of layers to be applied to the element. It is possible to reinforce in different ways in the z and y directions, while along the same direction the reinforcement will be symmetrical.



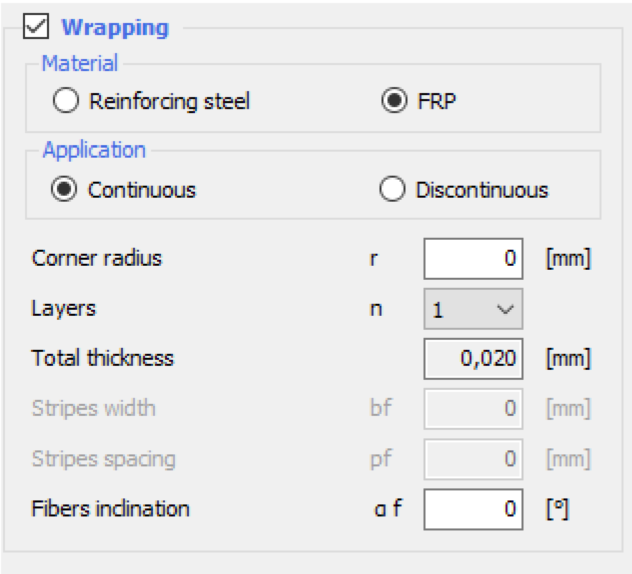
Characteristics of the reinforcement - Wrapping

The wrapping can be made of FRP or steel, the characteristics of which are to be defined in the material panel. It is possible to choose between a continuous or banded shirt (less efficient).
The rounding radius must be at least 20 mm to avoid damage to the composite. For FRP reinforcement, given the anisotropic behavior of the material, it is necessary to enter the angle of inclination of the fibers with respect to the transversal of the element; the maximum yield is given by the zero inclination**.**

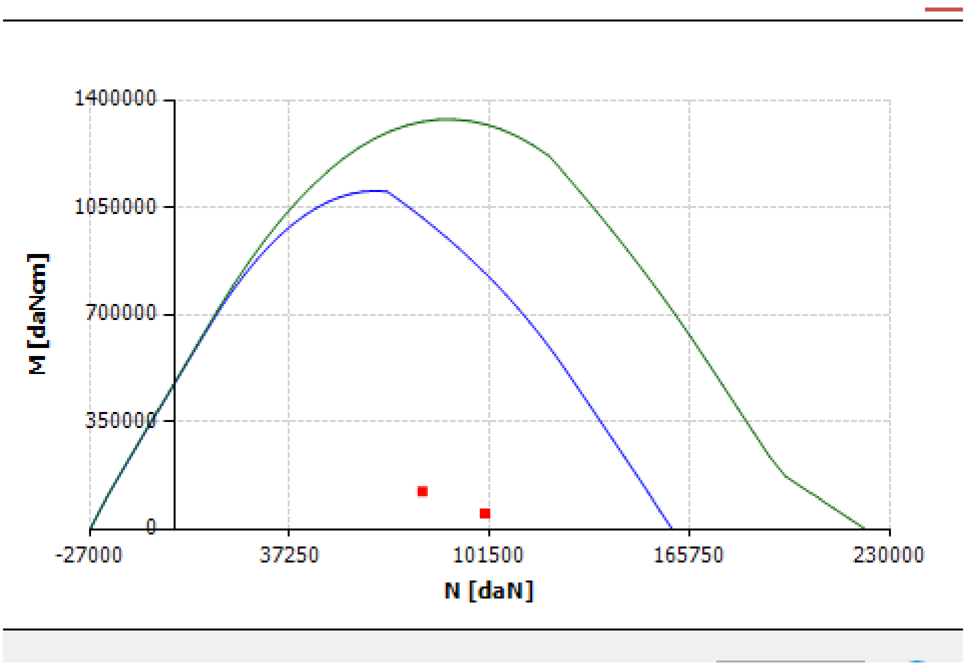
11.6.2.1.3.2 Materials refinition

In the material definition environment it is possible to enter the values to be used in the calculation for:

- Concrete
- Structural rebar
- Reinforcement steel
- FRP material
-



The diagram represents the M-N pairs corresponding to the limit strains.
In the case of a reinforced column, the diagram shows the different limit curve with respect to the same non-reinforced section.



11.6.2.2 Bending reinforcement

11.6.2.2.1 Introduction

The FRP Travi c.a. allows the calculation of the moment of resistance at ULS and performs the verification at the SLS of the interface stresses of a beam with a generic section in reinforced concrete, consolidated using fiber-reinforced materials pursuant to Circular no. 7 of 21 January 2019, CNR-DT R1 / 2013 and CNR-DT 215/2018.

11.6.2.2.2 Calculation procedure

The program calculates the ULS resistant moment of the reinforced section and checks the stresses at the interface between concrete and reinforcement at SLS.

11.6.2.2.2.1 SLU analysis

The calculation procedure is based on the comparison between the design stressing moment Msd and the design resistant moment of the reinforced section M_{rd}.

$$M_{sd} \leq M_{rd}$$

The fundamental assumptions on which the ULS analysis of the sections of reinforced concrete is based reinforced with FRP are the following:

- preservation of the planarity of the straight sections until failure, so that the diagram of the normal deformations is linear;
- perfect adhesion between the component materials (steel-concrete, FRP-concrete);
- zero tensile strength of concrete;
- constitutive bonds of concrete and steel in compliance with current legislation;
- constitutive bond of linear elastic fiber-reinforced composite until failure.

Flexural failure is hypothesized to occur when one of the following conditions occurs:

- achievement of maximum plastic deformation in compressed concrete, ε_{cu}, as defined by current legislation;
- achievement of a maximum deformation in the FRP reinforcement, ε_{fd}.

Given the geometry and materials of the section of the element to be reinforced, the moment of first cracking Mcr is evaluated, evaluating the average simple tensile strength fctm given by eqs. 11.2.3.a, 11.2.3.b and 11.2.4 - NTC 2018.

The initial deformation of the concrete at the stretched edge is evaluated as:

$$\varepsilon_0 = \frac{M_0 \cdot (h - x_0)}{I_0 \cdot E_c}$$

in which:

- M_0 is the initial moment acting in section before the application of the fibers produced by the permanent loads at the SLS;
- x_0 is the distance of the neutral axis from the compressed limb;
- I_0 is the moment of inertia of the section taking into account any cracking;
- E_c is the elastic modulus of concrete evaluated in accordance with Eq. 11.2.5, NTC 2018.

Given the characteristics of the composite and its methods of application, the maximum design deformation is evaluated: (eq. 4.14 CNR-DT 200 R1 / 2013)

$$\varepsilon_{fd} = \min\{\eta_a\ \varepsilon_{fk} / \gamma_f;\ \varepsilon_{fdd}\}$$

where:

$\varepsilon_{fk} = f_{fk} / E_f$ represents the characteristic deformation at break of the reinforcement;

$\varepsilon_{fdd} = \frac{f_{f\,dd,2}}{E_f} \geq \varepsilon_{sy} - \varepsilon_0$ represents the maximum value attributable to the deformation of the composite during the design so that the intermediate detachment does not occur (eq. 4.7 - CNR-DT 200 R1 / 2013);
in which:

$f_{f\,dd,2} = \frac{k_q}{\gamma_{f,d}} \cdot \sqrt{\frac{E_f}{t_f} \cdot \frac{2 \cdot k_b \cdot k_G}{FC} \sqrt{f_{cm} \cdot f_{ctm}}}$ represents the maximum design stress of the composite (eq. 4.6 - CNR-DT 200 R1 / 2013)

The ultimate limit state resistance for end detachment is assessed by verifying that the tension of the composite does not exceed the design tension of the reinforcement system for end detachment (eq. 4.4 - CNR-DT 200 R1 / 2013), calculated as:

$$f_{f\,dd} = \frac{1}{\gamma_{f,d}} \cdot \sqrt{\frac{2 \cdot E_f \cdot \Gamma_{Fd}}{t_f}}$$

with partial coefficient for FRP materials (3.4.1 CNR - DT 200 R1 / 2013), it represents the specific fracture energy calculated as (eq. 4.2 - CNR-DT 200 R1 / 2013):

$$\Gamma_{Fd} = \frac{k_b \cdot k_G}{FC} \sqrt{f_{cm} \cdot f_{ctm}}$$

The calculation of the optimal anchoring length is also performed (eq. 4.1 - CNR-DT 200 R1 / 2013):

$$l_{ed} = \max\left\{\frac{1}{\gamma_{Rd} \cdot f_{bd}} \sqrt{\frac{\pi^2 \cdot E_f \cdot t_f \cdot \Gamma_{Fd}}{t_f}}, 200\ mm\right\}$$

In the case of anchorage lengths, lb, less than the optimal one, led, the design voltage must be suitably reduced in accordance with the relationship:

$$f_{f\ dd,rid} = f_{dd} \cdot \frac{l_b}{l_{ed}} \cdot \left(2 - \frac{l_b}{l_{ed}}\right)$$

Once the position of the neutral axis has been determined, the value of the resisting moment MRd can be determined starting from the equation of equilibrium for rotation around the axis passing through the center of gravity of the stretched reinforcement and parallel to the neutral axis (eq. 4.16 - CNR -DT 200 R1 / 2013):

$$M_{Rd} = \frac{1}{\gamma_{Rd}} \cdot \left[\psi \cdot b \cdot x \cdot f_{cd} \cdot (d - \lambda \cdot x) + A_{s2} \cdot \sigma_{s2} \cdot (d - d_2) + A_f \cdot \sigma_f \cdot d_1\right]$$

where the three components represent respectively:
- the contribution of compressed concrete
- the contribution of steel, stretched and compressed
- the contribution of FRP reinforcement

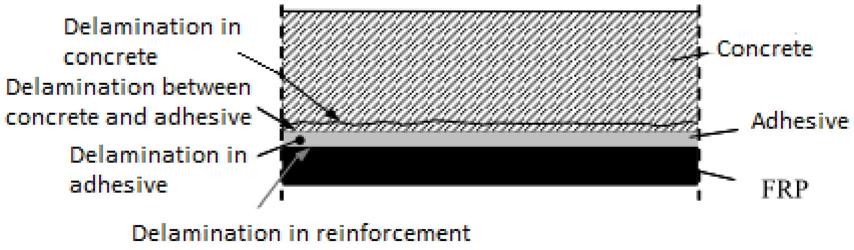
In the hypothesis that the FRP reinforcement is applied on an element subject to a pre-existing stress, which corresponds to an applied moment M_0, the program proceeds to evaluate the initial deformation state considering a different moment of inertia depending on whether M_0 is greater or less than the moment of first crack M_{cr}.

The calculation is performed in the hypothesis of linear elastic behavior of the two materials constituting the beam and, in particular, of the inability of the concrete to withstand tensile stresses.

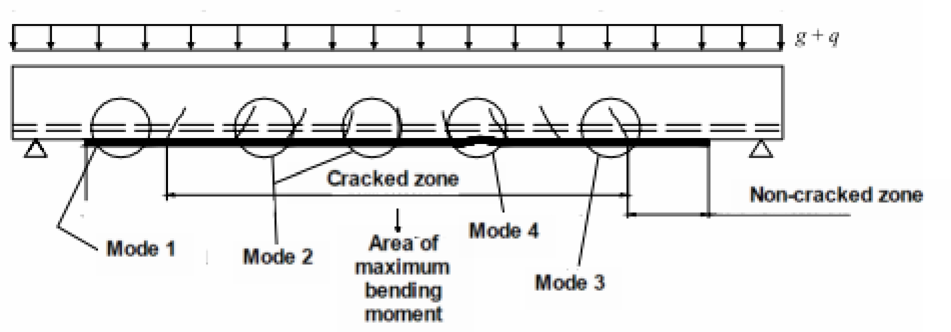
Security checks against delamination

In the reinforcement of reinforced concrete elements using sheets or fabrics of composite material, the role of adhesion between concrete and composite assumes great importance as the mechanism of rupture by delamination (loss of adhesion) is of a fragile type. In the spirit of the criterion of hierarchy of resistances, this crisis mechanism must not precede the collapse by bending or cutting of the reinforced element.

Delamination can occur inside the adhesive, between concrete and adhesive, in the concrete or inside the reinforcement (for example between layers of fabric warped with different inclination angles of the fibers). The program considers that the reinforcement is placed correctly, therefore, since the adhesive resistance is generally much higher than the tensile strength of the concrete, that delamination always takes place inside the latter with the removal of one layer of material.



The modes of collapse by delamination of foils or fabrics used for bending reinforcement can be classified into the following four categories:



- Mode 1 (Delamination of ends);
- Mode 2 (Intermediate delamination, caused by bending cracks in the beam);
- Mode 3 (Delamination caused by diagonal cutting cracks);
- Mode 4 (Delamination caused by irregularities and roughness of the concrete surface).

The program allows the verification of modalities 1 and 2 only, being the ones that occur most frequently in ordinary situations.

11.6.2.2.2.2 SLS analysis

In a beam reinforced with FRP, stress concentrations (tangential and normal) occur at the interface between concrete and reinforcement, located in correspondence with transverse cracks in the concrete, especially at the ends of the reinforcement. These concentrations can cause cracking of the interface triggering the detachment between the two materials.

It is advisable that, in operating conditions, the opening of the aforementioned cracks should not occur, especially in the presence of loading cycles and freeze / thaw cycles. The competent verification can be performed by a calculation of the interface stresses using linear elastic models.

It must be checked that, at the adhesive-concrete interface, both for the characteristic (or rare) and for the frequent load combination, the "equivalent" shear stress, defined below, is lower than the adhesion resistance between the reinforcement and the substrate of concrete, f_{bd}:

$$\tau_{b,e} \leq f_{bd}$$

The "equivalent" shear stress $\tau_{b,e}$ can be defined starting from the mean shear stress τ_m, evaluated at the chord on which the adhesive and concrete interface:

$$\tau_{b,e} = k_{id} \cdot \tau_m$$

where:

- k_{id} is a coefficient (≥ 1) which takes into account the concentration of tangential and normal stresses in the terminal zones:

$$k_{id} = \left(k_\sigma^{1.5} + 1.15 \cdot k_\tau^{1.5}\right)^{2/3}$$

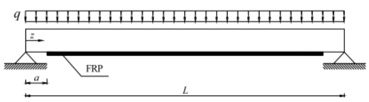
- the coefficients k_σ and k_τ are:

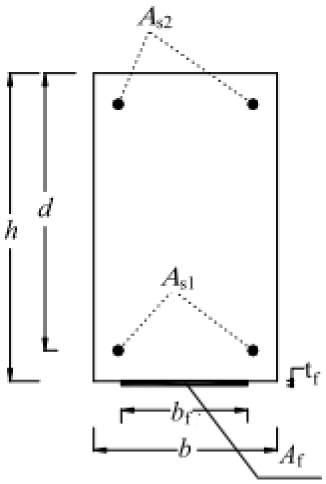
$$k_\sigma = k_\tau \cdot \beta \cdot t_f$$

$$k_\tau = 1 + \alpha \cdot a \cdot \frac{M_{(z=a)}}{V_{(z=a)} \cdot a}$$

- M (z = a) is the moment acting in the interruption section of the reinforcement;

- V (z = a) is the shear acting in the interruption section of the reinforcement, located at a distance z = a from the end of the beam;





- α and β are two elastic constants dependent on the characteristics of the interface and of the FRP reinforcement:

$$\alpha = \sqrt{\frac{K_1}{E_f \cdot t_f}}$$

$$\beta = \left(\frac{b_f \cdot 2.30 \cdot K_1}{4 \cdot E_f \cdot I_f}\right)^{1/4}$$

being E_f , t_f , b_f , I_f and K_1 respectively, the modulus of normal elasticity, the thickness of the FRP reinforcement, its length, the competent moment of inertia (with respect to its own barycentric axis parallel to the length dimension b_f) and the angular coefficient of the increasing linear branch of the adhesion bond, assumed to be equal to:

$$K_1 = \frac{1}{t_a/G_a + t_c/G_c}$$

where moreover, respectively, G_a and G_c are the modulus of tangential elasticity of the adhesive and of the concrete, t_a is the nominal thickness of the adhesive and t_c the effective thickness of the concrete participating in the deformability of the interface (in general it can be assumed tc = 20 ÷ 30 mm);

- τ_m is the Jourawski mean shear stress:

$$\tau_m = \frac{V_{(z=a)} \cdot t_f \cdot (h - x_e)}{I_c/n_f}$$

- x_e and I_c are, respectively, the distance of the neutral axis from the extreme compressed edge and the moment of inertia of the homogenized section, possibly partialized if in the presence of cracking (NOTE: the program considers the cracked section when the moment acting for the operating state considered $M_{(z=a)}$ is higher than the moment of first cracking M_{cr});

- $n_f = E_f/E_c$ is the homogenization coefficient (with E_c the normal modulus of elasticity of the concrete corresponding to the load combination considered, rare or frequent).

In the presence of a terminal anchorage, made by "U" bandage, the effect of normal stresses for the purpose of verifying the interface can be neglected and, therefore, the coefficient k_σ can be assumed to be equal to zero.

The design strength of the adhesion between reinforcement and concrete, f_{bd}, is a function of the characteristic tensile strength of the concrete, f_{ctm}, and is provided by the relationship:

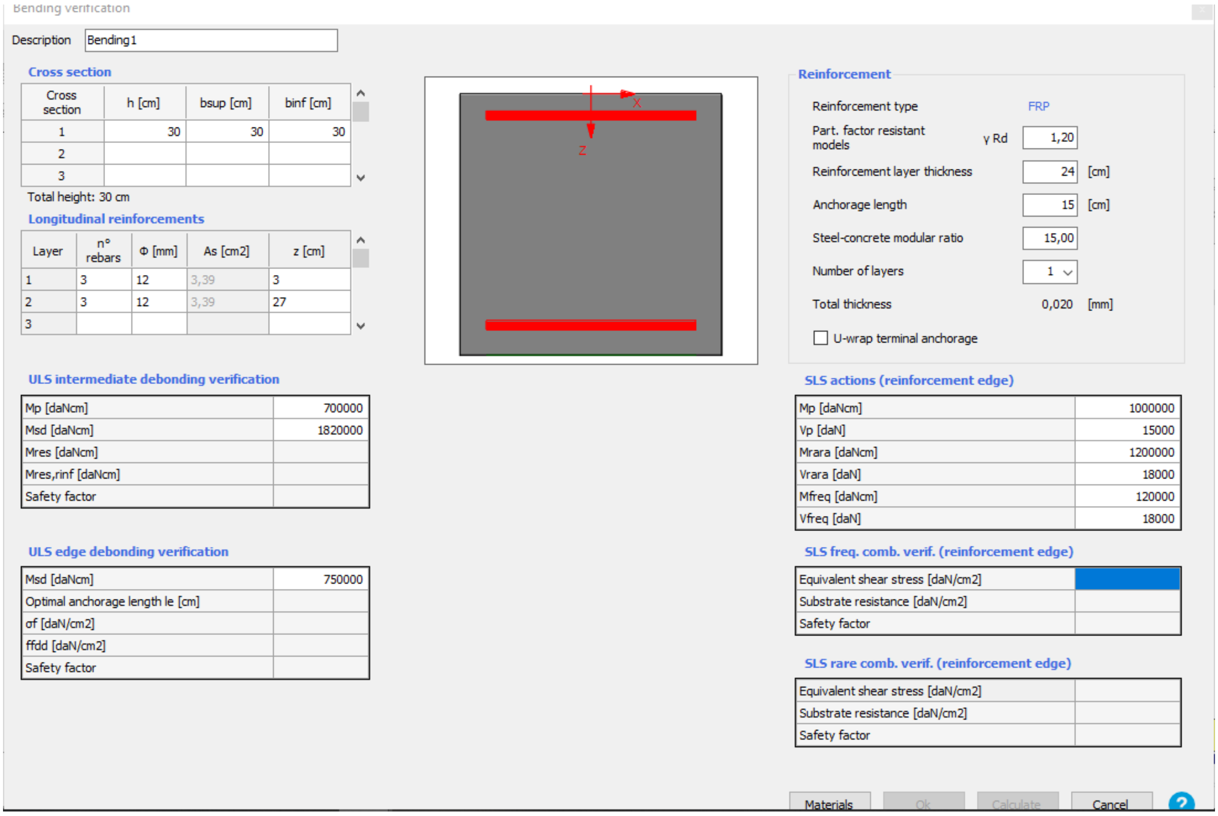
$$f_{bd} = 0.21 \cdot \frac{k_b}{\gamma_b} \cdot \frac{f_{ctm}}{FC}$$

where the partial factor γ_b is 1.0 for the rare load combination, 1.2 for the frequent load combination.

In the calculation of the anchoring stresses, in operating conditions (SLS), it is possible to refer to the state of stress corresponding to the load increase that occurs after the application of the reinforcement.

11.6.2.2.3 Work environment

The working environment is as follows:

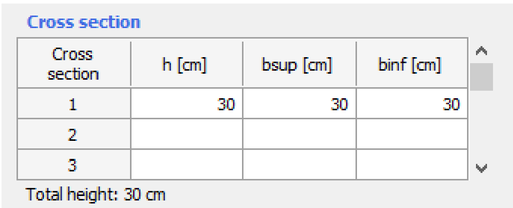


The main work area is divided into 6 sections:

1. Geometria della sezione
2. Longitudinal armor
3. Section diagram
4. Reinforcement geometry
5. ULS check for intermediate posting
6. ULS verification for end detachment
7. Check SLE at the end of the stiffener

11.6.2.2.3.1 Main area

The main input window is divided into seven sections.



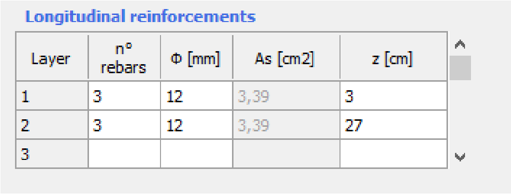
Cross section

Cross section	h [cm]	bsup [cm]	binf [cm]
1	30	30	30
2			
3			

Total height: 30 cm

Section geometry

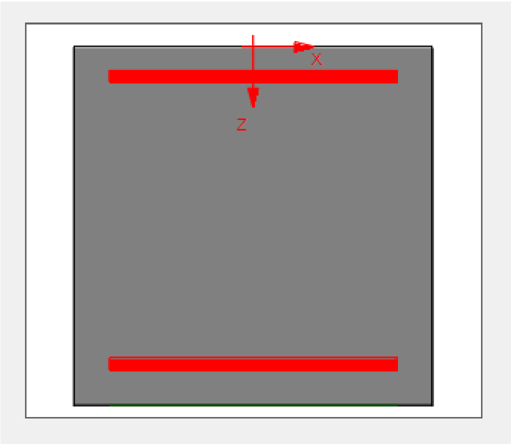
The geometry of the section under analysis must be entered as a succession of sections; of each it is necessary to indicate the height, the width of the upper side and the width of the lower side.
As the grid lines are completed, the Section Outline is automatically updated.



Layer	n° rebars	Φ [mm]	As [cm2]	z [cm]
1	3	12	3,39	3
2	3	12	3,39	27
3				

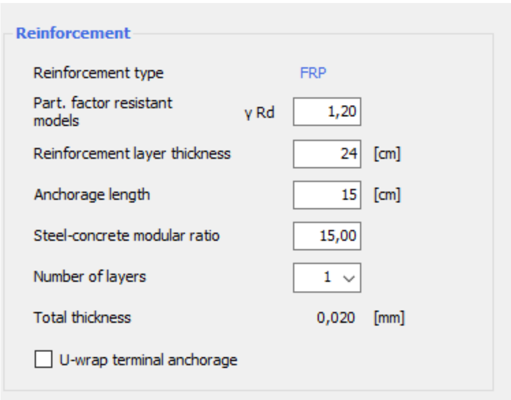
Longitudinal reinforcements

For the rectangular section, the reinforcement is defined by indicating the number of bars, the diameter and the position, with respect to the reference system indicated, of each layer. Once the number and diameter have been assigned, the program will automatically fill in the cell with the reinforcement area.
As the grid lines are completed, the Section Outline is automatically updated.



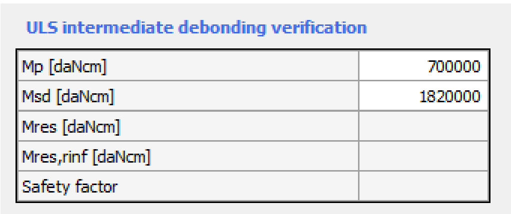
Section diagram

Automatically updated when data is entered in the Section geometry and Longitudinal reinforcement grids, it allows you to quickly check the data entered.



Reinforcement

Indicate in this section the width of the layer of fiber-reinforced material, the anchoring length lb and the number of layers applied to the lower edge of the beam under analysis.
To change the thickness of the FRP layers and its mechanical characteristics, use the materials panel.
Indicate in this section also the steel-concrete homogenization coefficient n in the comparison of permanent actions, used by the program for the calculation of the moment of first cracking and of the deformation state at the moment of application of the reinforcement



Mp [daNcm]	700000
Msd [daNcm]	1820000
Mres [daNcm]	
Mres,rinf [daNcm]	
Safety factor	

ULS intermediate debonding verification

Indicate in this section:

- M_p: moment due to permanent loads in combination SLE;
- M_{sd}:Soliciting moment at ULS.

The following are calculated:

- M_{res}: moment of resistance of the unreinforced section;
- $M_{res,rinf}$: moment of resistance of the reinforced section;

- Safety factor.

ULS edge debonding verification

ULS edge debonding verification	
Msd [daNcm]	750000
Optimal anchorage length le [cm]	
σf [daN/cm2]	
ffdd [daN/cm2]	
Safety factor	

Indicate in this section:
Msd: Stress moment at ULS in the end section.

The following are calculated:

- l_e: optimal anchoring length;
- σ_f: normal stress drawn from the FRP layer;
- f_{fdd}: design resistance to end detachment;
- Safety factor.

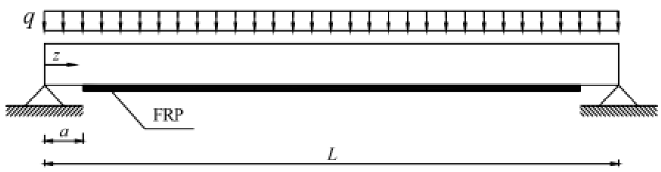
SLS actions (reinforcement edge)

SLS actions (reinforcement edge)	
Mp [daNcm]	1000000
Vp [daN]	15000
Mrara [daNcm]	1200000
Vrara [daN]	18000
Mfreq [daNcm]	120000
Vfreq [daN]	18000

SLS freq. comb. verif. (reinforcement edge)	
Equivalent shear stress [daN/cm2]	
Substrate resistance [daN/cm2]	
Safety factor	

SLS rare comb. verif. (reinforcement edge)	
Equivalent shear stress [daN/cm2]	
Substrate resistance [daN/cm2]	
Safety factor	

Indicate the SLE stresses for verifying the interface stresses in the end section of the reinforcement.



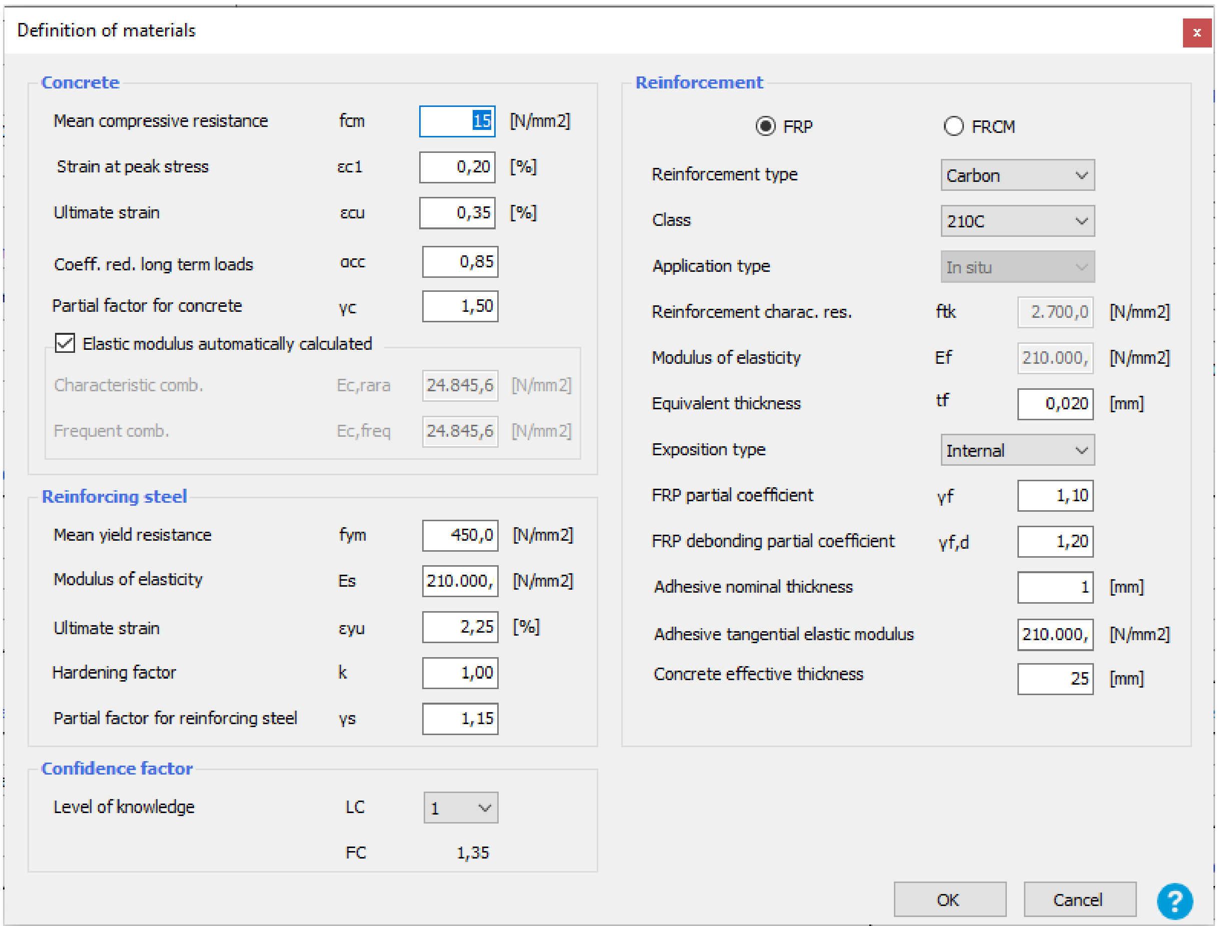
Indicate in this section:

- M_p: permanent moment in the end section of the reinforcement (for SLS checks);
- V_p: permanent cut in the end section of the reinforcement (for SLE checks);
- M_{rara}: rare combination moment in the end section of the reinforcement;
- V_{rara}: - rare combination cut in the end section of the reinforcement;
- M_{freq}: frequent combination moment in the end section of the reinforcement;
- V_{freq}: frequent combination cutting in the end section of the reinforcement.

The following are calculated:

- The equivalent shear stress values;
- The resistance values of the substrate considered;
- The relative safety factors.

11.6.2.2.3.2 Area dettagli materiali



In the material definition environment it is possible to enter the values to be used in the calculation for:

- Concrete
- Reinforcing steel
- FRP / FRCM reinforcement

11.6.2.3 Shear reinforcement

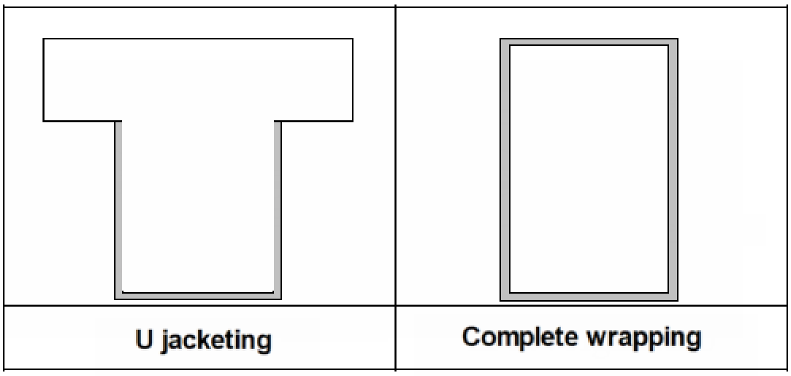
11.6.2.3.1 Introduction

The FRP Shear Reinforcement module allows the calculation of the shear strength of a reinforced concrete element reinforced by wrapping, in accordance with Circ. 21 January 2019 n. 7, CNR-DT R1 / 2013 and CNR-DT 215/2018.

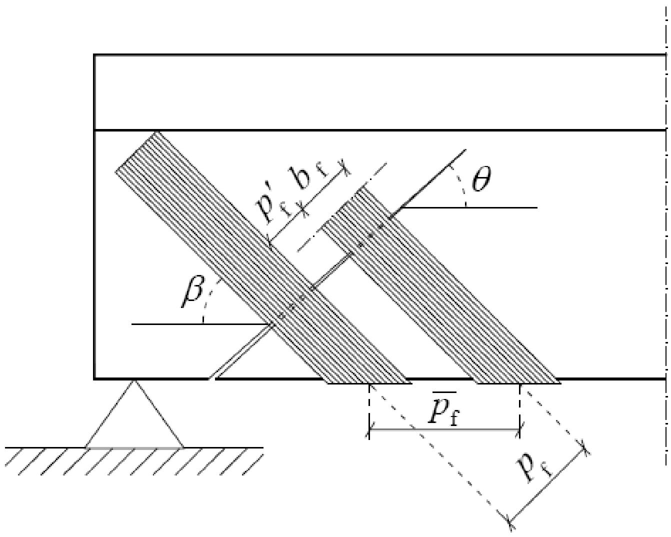
11.6.2.3.2 Calculation procedure

The resistant shear of the reinforced element is given by the expression (CNR DT200 R1 2013 - (4.18)):
Vrd = min {VRd, s + VRd, f, VRd, c}
where VRd, s and VRd, c are, respectively, the contributions of the transverse reinforcement and of the compressed concrete connecting rod, calculated in accordance

with current legislation (NTC 18). The contribution VRd, f, on the other hand, is the increase in resistance brought about by the reinforcement. In the event that VRd, c <VRd, s, therefore, the application of the reinforcement cannot produce increases in resistance.
The contribution VRd, f depends, in addition to the characteristics of the materials, on the type of arrangement of the reinforcement which may be of the type (in order of efficiency):
-"U jacketing";
-"complete wrapping".



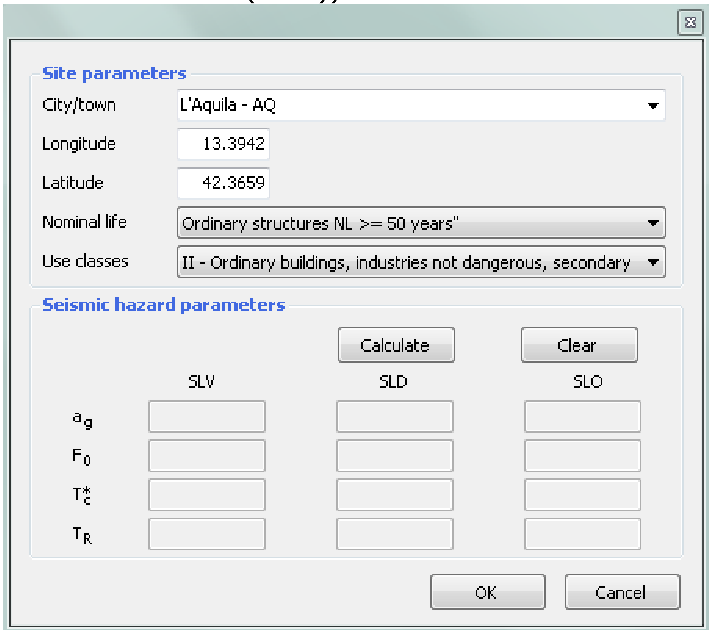
In the U-jacketing reinforcement it is possible to improve the constraint conditions of the free ends of the fabrics, by applying bars, sheets or strips of composite material in these areas. In this case, if the effectiveness of the constraint offered by the aforementioned devices is demonstrated, the behavior of the U-jacketing reinforcement can be considered equivalent to that of the winding reinforcement.
Other factors influencing the contribution VRd, f are the arrangement along the element (continuous or in bands) and the angle of inclination of the fibers with respect to the horizontal axis.



The angle of inclination of the cutting slots θ is to be set equal to 45 °. However, this is in contrast to what is indicated in the NTC 18 where, for the calculation of the shear strength, the use of the "variable θ" method is required provided that 1 <= cotg θ <= 2.5. The program therefore leaves the possibility for the user to use the calculation method he deems appropriate.
The contribution of the FRP reinforcement is calculated according to the formula 4.19 of CNR DT200 R1 2013

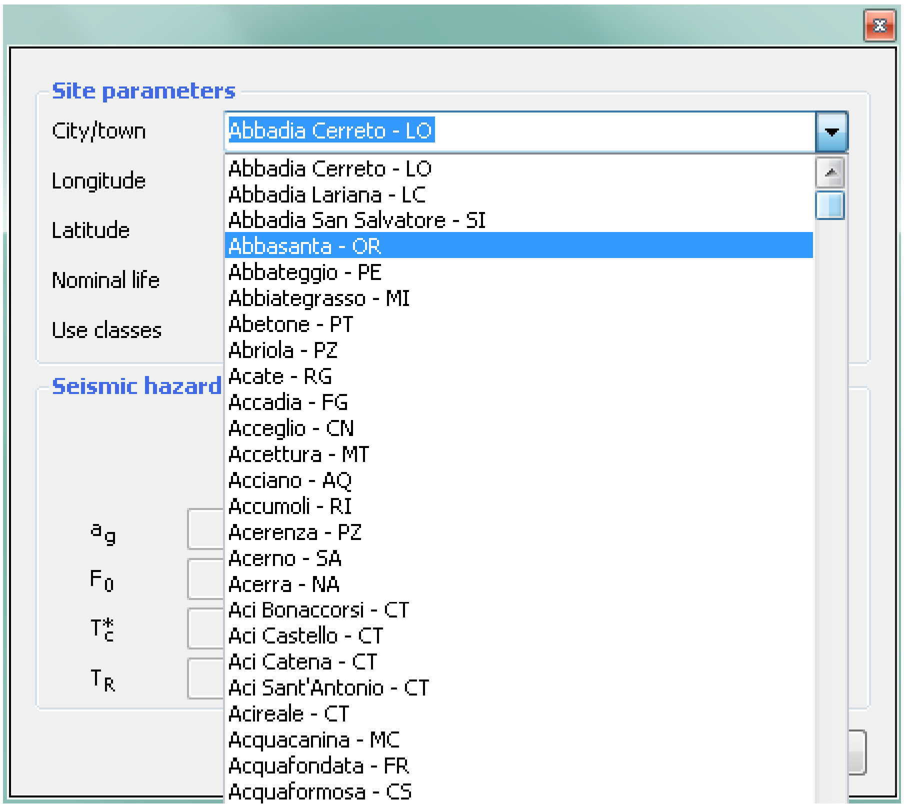
where:

- d is the useful height of the section;
- tf, bf, pf are the geometric characteristics of the strips;
- ffed is the effective resistance of the reinforcement system, calculated as follows (CNR DT200 R1 2013 - (4.21))



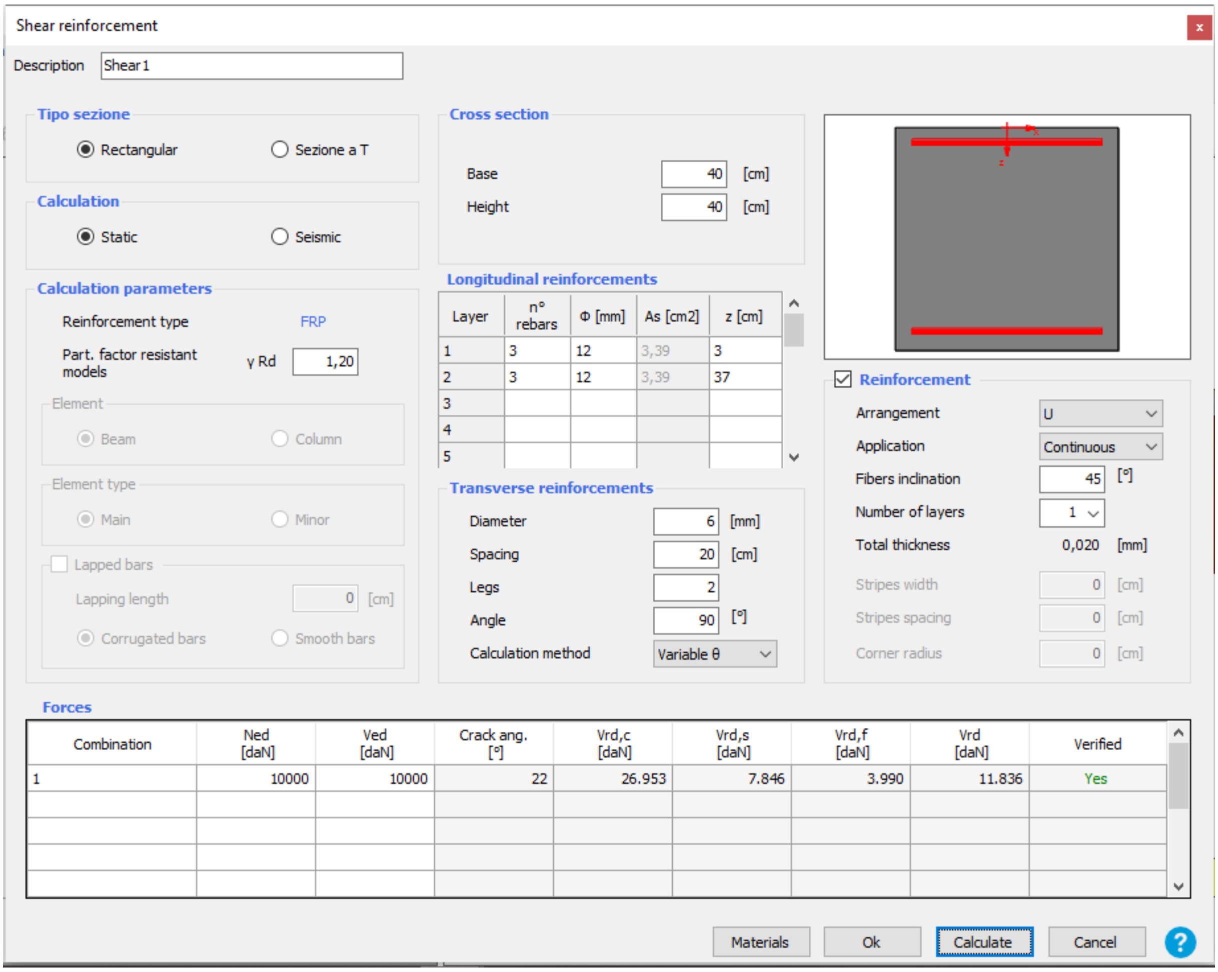
In the case of discontinuous reinforcement systems, the strips of composite material must respect the following limitations:
50 mm ≤ bf ≤ 250 mm
bf ≤ pf ≤ min {0,5 d; 3 bf; bf + 200 mm}

In the case of seismic calculation, the calculation of the shear resistance of the section is carried out according to the indications of paragraph C8.7.2.3.5 of the Circ. 21 January 2019 n. 7 relating to NTC 2018.
The contribution of the FRP reinforcement must be added to the contribution of the stirrups Vw in the equation C8.7.2.8:



11.6.2.3.3 Work environment

The working environment is as follows:

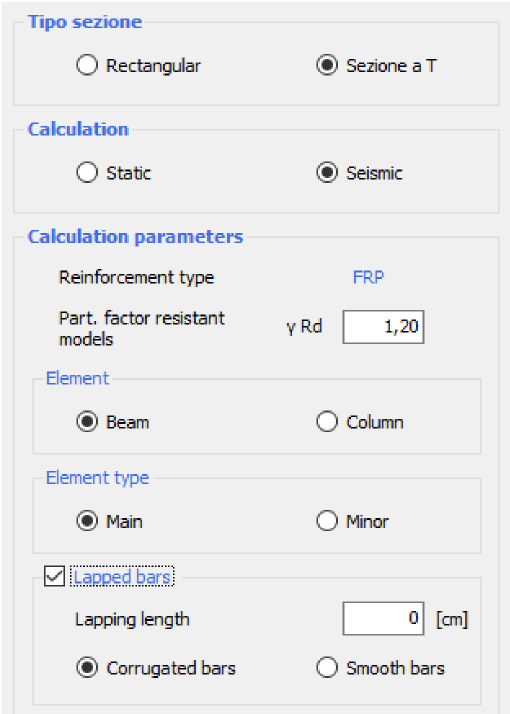


The work environment is made up of a general area divided into 6 sections, and a secondary area relating to materials:

1. Calculation parameters
2. Cross section
3. Longitudinal reinforcements
4. Transverse reinforcements
5. Reinforcement
6. Materials
7. Forces and results

11.6.2.3.3.1 Main area

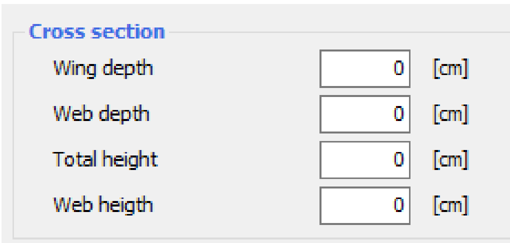
The main area is divided into several sections.



Calculation parameters

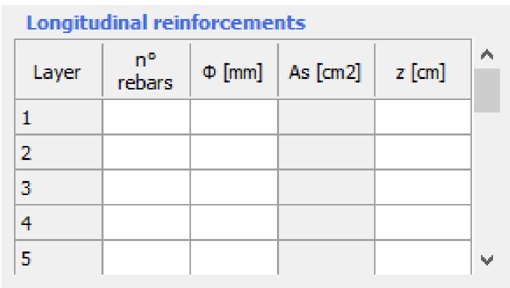
Select the type of section and the calculation to be made: static or seismic.
In case of seismic calculation, the parameters necessary for the calculation of the cyclic shear resistance are enabled (Circ. January 21, 2019 n.7):

- type of structural element;
- detail of the reinforcement (overlapping of the bars, type of bars, diagonal reinforcement present).



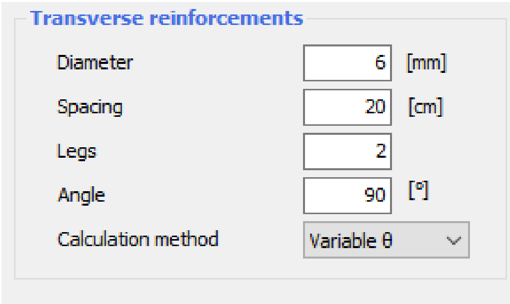
Cross section

When the data is complete, the representation of the section is updated with which it is possible to quickly check the data entered.



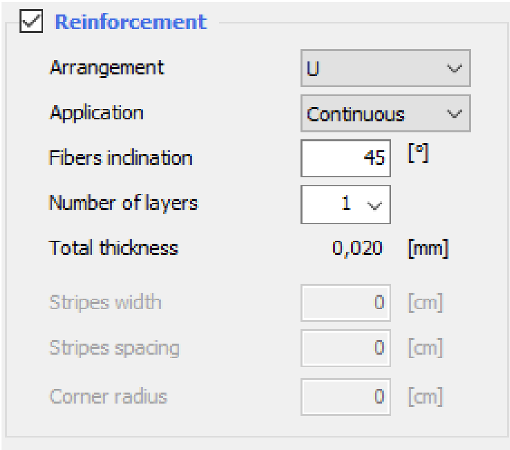
Longitudinal reinforcement

The reinforcement is defined by indicating the number of bars, the diameter and the position, with respect to the reference system indicated, of each layer. Once the number and diameter have been assigned, the program will automatically fill in the cell with the reinforcement area.
As the grid lines are completed, the Section Outline is automatically updated.



Transverse reinforcement

Indicate in this section the transverse reinforcement and the method to be used for calculating the theta angle.
N.B. In the case of variable θ the cotgθ is calculated by equating the resistance on the concrete side and the resistance on the stirrup side, without considering the contribution of the FRP reinforcement.

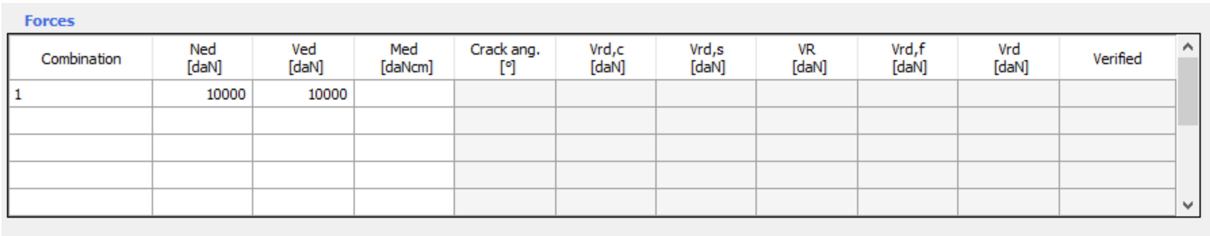


Reinforcement

Indicate in this section information regarding the application of the reinforcement.

Forces and Results

The stresses acting on the section must be entered in this section; after the calculation, the different contributions of shear strength and the overall strength are reported.


Forces

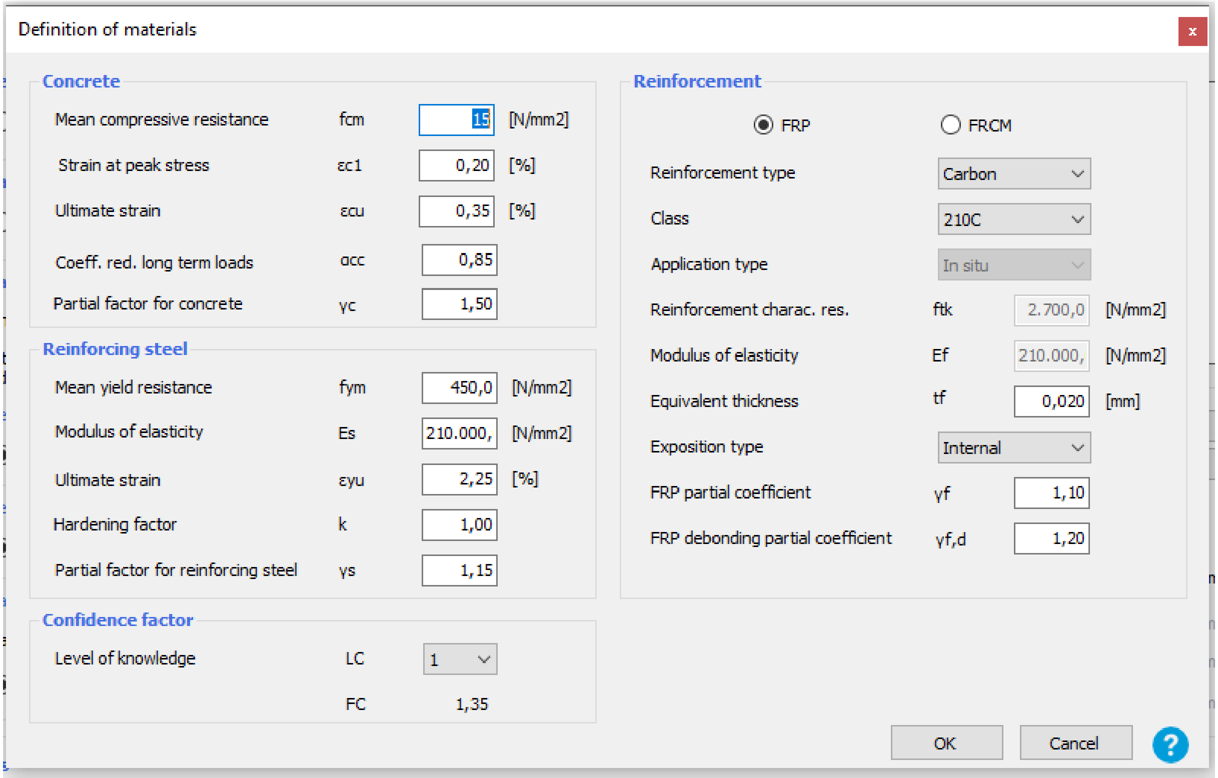
Combination	Ned [daN]	Ved [daN]	Med [daNcm]	Crack ang. [°]	Vrd,c [daN]	Vrd,s [daN]	VR [daN]	Vrd,f [daN]	Vrd [daN]	Verified
1	10000	10000								

11.6.2.3.3.2 Materials

Pressing the "Materials" button at the bottom right opens the window relating to the mechanical characteristics of the materials used.

The mask is divided into 4 sections:
- Concrete;
- Reinforcing Steel;
- Confidence factor;
- FRP/FRCM reinforcement.

In the case of concrete and steel, the mechanical characteristics must be entered manually. As regards the FRP reinforcement, it is possible to select one of the composites present in the library or, alternatively, select the last item in the "Other" drop-down menu to manually enter all the parameters of the material. However, the Type of exposure must be specified.



11.6.3 Geotechnical

11.6.3.1 K-Winkler

11.6.3.1.1 Introduction

A harmonious project is achieved by balancing the state and the internal deformation capacities of the ground, the foundation and the structure.
The oldest ideal elastic model of soil-structure interaction, still in use, is that of Winkler (1867); the soil load plate test measures the Winkler subgrade coefficient as the slope of the initial section of the contact pressure / settlement curve and results:

$$k = \frac{\Delta\sigma_t}{\Delta\delta} \equiv \frac{\sigma_{tamm}}{\delta_{amm}} \leq \frac{\sigma_{tamm}}{1\ in} = \frac{\sigma_{tamm}}{2{,}5cm}$$

The Winkler substrate coefficient k of the contact pressure, has as a unit of measurement 1 kg / cm3 = 10 MPa / m, moreover referred to the allowable settlement for a foundation rather than "elastic" under a plate.
If the average contact pressure of a building, not subject to hydrostatic under-thrust, is for example st = 2.5 kg / cm2 it results k = 1 kg / cm3, naturally for minor settlements the value of k increases and vice versa.

11.6.3.1.2 Calculation procedure

The Winkler constant for vertical loads is evaluated using a methodology suggested by Joseph E. BOWLES on the basis of the bearing capacity (ultimate load) of the foundation according to the following formula:

$$k = \frac{q_{ult}}{\Delta H}$$

where q_{ult} represents the bearing capacity of the foundation and ΔH represents the allowable settlement for a foundation, assumed to be 1 inch.
For the calculation of the ultimate foundation load (q_{ult}), five different methods are taken into consideration:

- Terzaghi (Bowles)
- Terzaghi (Sprangler / Handy)
- Mayerhof
- Hansen
- Vesic

11.6.3.1.2.1 Terzaghi method

Terzaghi proposes the following expression for the calculation of the bearing capacity:

$$q_{ult} = cN_c s_c + qN_q + \frac{B}{2}\gamma N_\gamma s_\gamma$$

where Nq, Nc and Nγ represent the bearing capacity factors and are valid:

$$N_q = \frac{e^{2(0.75\pi - \phi/2)\tan\phi}}{2\cos^2(45 + \phi/2)}$$

$$N_c = (N_q - 1)\cot\phi$$

$$N_\gamma = \frac{\tan\phi}{2}\left(\frac{K_{p\gamma}}{\cos^2\phi} - 1\right)$$

Where:

- sc, sγ are form factor factors and depend on the shape of the foundation.

Specifically, they are worth 1 for ribbon or rectangular foundations and are respectively worth 1.3 and 0.8 for square foundations.

In the calculation with Terzaghi Bowles the passive thrust coefficient Kp used in the calculation of Nγ is obtained by interpolation from known values tabulated as a function of f.

While with the Terzaghi Sprangler / Handy method the value of Nγ is calculated with the following expression:

$$N_\gamma = 1{,}1\,(N_q - 1)\tan(1{,}3\,\varphi)$$

For cohesive soils we consider:

$$q_{ult} = 5{,}14\ c_u\ s_c + q$$

The Terzaghi formula applies to surface foundations with D = B and does not take into account the possible inclination of the foundation and the eccentricity and inclination of the load.

11.6.3.1.2.2 Mayerhof method

Mayerhof for the calculation of the bearing capacity takes into account the possible inclination of the load and suggests the following expression:

$$q_{ult} = cN_c s_c d_c i_c + qN_q s_q d_q i_q + \frac{B}{2}\gamma N_\gamma s_\gamma d_\gamma i_\gamma$$

For cohesive soils (f = 0) we consider:

$$q_{ult} = 5{,}14\ c_u\ s_c d_c i_c + q\ i_q$$

where:

- Nq, Nc and Nγ represent the bearing capacity factors and are valid:

$$N_q = e^{\pi \tan\phi} \tan^2(45 + \frac{\phi}{2})$$

$$N_c = (N_q - 1)\cot\phi$$

$$N_\gamma = (N_q - 1)\tan(1.4\phi)$$

- sc, sq, sγ are the form factors
- dc, dq, dγ are the depth factors
- ic, iq, iγ are the load inclination factors.

f = 0:

Shape factor

$$s_c = 1 + 0.2\,K_p \frac{B}{L}$$

$$s_q = s_\gamma = 1$$

Depth factor

$$d_c = 1 + 0.2\,\sqrt{K_p}\,\frac{D}{B}$$

$$d_q = d_\gamma = 1$$

Load inclination factor

$$i_c = i_q = \left(1 - \frac{\theta°}{90°}\right)$$

$$i_\gamma = 0$$

f > 0

Shape factor

$$s_c = 1 + 0.2\,K_p \frac{B}{L}$$

$$s_q = s_\gamma = 1 + 0.1\,K_p \frac{B}{L}$$

Depth factor

$$d_c = 1 + 0.2\,\sqrt{K_p}\,\frac{D}{B}$$

$$d_q = d_\gamma = 1 + 0.1\sqrt{K_p}\,\frac{D}{B}$$

Load inclination factor

$$i_c = i_q = \left(1 - \frac{\theta°}{90°}\right)$$

$$i_\gamma = \left(1 - \frac{\theta°}{\phi°}\right)^2$$

11.6.3.1.2.3 Hansen method

Hansen, depending on whether one is in the presence of a purely cohesive soil (f = 0) or not, proposes two different expressions for the calculation of the bearing capacity:

$$q_{ult} = cN_c s_c d_c i_c b_c g_c + qN_q s_q d_q i_q b_q g_q + \frac{B}{2}\gamma N_\gamma s_\gamma d_\gamma i_\gamma b_\gamma g_\gamma$$

for cohesive soils:

$$q_{ult} = 5{,}14\,c\,(1 + s_c + d_c - i_c - g_c - b_c) + q$$

The bearing capacity factors Nq, Nc and Nγ are valid:

$$N_q = e^{\pi \tan\phi} \tan^2 (45 + \frac{\phi}{2})$$

$$N_c = (N_q - 1) \cot\phi$$

$$N_\gamma = 1.5 (N_q - 1) \tan\phi$$

- sc, sq, sγ are the form factors
- dc, dq, dγ are the depth factors
- ic, iq, iγ are the load inclination factors
- bc, bq, bγ are the inclination factors of the laying surface
- gc, gq, gγ are factors that take into account the fact that the foundation rests on a sloping ground.

Define the parameter k:

for D / B = 1

$$k = \frac{D}{B}$$

for D / B> 1

$$k = \arctan\frac{D}{B}$$

Shape factor

$s_c = 0.2\frac{B}{L}$

$s_c = 1 + \frac{N_q}{N_c}\frac{B}{L}$

$s_q = 1 + \frac{B}{L}\tan\phi$

$s_\gamma = 1 - 0.4\frac{B}{L}$

Depth factor

$d_c = 0.4\,k$

$d_c = 1 + 0.4\,k$

$d_q = 1 + 2\,\tan\phi\,(1 - \sin\phi)^2\,k$

$d_\gamma = 1$

Load inclination factor

$i_c = i_q = \left(1 - \frac{\theta°}{90°}\right)$

$i_\gamma = 0$

$i_\gamma = \left(1 - \frac{\theta°}{\phi°}\right)^2$

Inclination factor of the laying surface

$b_c = \frac{\eta°}{147°}$

$b_c = 1 - \frac{\eta°}{147°}$

$b_q = b_\gamma = (1 - \eta\ tg\phi)^2$

Slope inclination factor

$g_c = \frac{\beta°}{147°}$

$g_c = 1 - \frac{\beta°}{147°}$

$g_q = g_\gamma = (1 - tg\beta)^2$

11.6.3.1.2.4 Vesic method

Also Vesic, like Hansen, according to whether there is a purely cohesive ground (f = 0) or not, proposes two different expressions for the calculation of the bearing capacity. Only the Nγ factor and the expression of some coefficients change between Hansen and Vesic.

$$q_{ult} = cN_c s_c d_c i_c b_c g_c + qN_q s_q d_q i_q b_q g_q + \frac{B}{2}\gamma N_\gamma s_\gamma d_\gamma i_\gamma b_\gamma g_\gamma$$

for cohesive soils:

$$q_{ult} = 5{,}14\, c\,(1 + s_c + d_c - i_c - g_c - b_c) + q$$

The bearing capacity factors Nq, Nc and Nγ are valid:

$$N_q = e^{\pi \tan\phi} \tan^2(45 + \frac{\phi}{2})$$
$$N_c = (N_q - 1)\, ctg\phi$$
$$N_\gamma = 2(N_q + 1)\, tg\phi$$

- sc, sq, sγ are the form factors
- dc, dq, dγ are the depth factors
- ic, iq, iγ are the load inclination factors
- bc, bq, bγ are the inclination factors of the laying surface
- gc, gq, gγ are factors that take into account the fact that the foundation rests on a sloping ground.

Define the parameter k:

$$k = \frac{D}{B}$$

for D / B = 1

$$k = \arctan\frac{D}{B}$$

for D / B> 1

f = 0 : **f > 0**

Shape factor

$$s_c = 0.2\frac{B}{L}$$

$$s_c = 1 + \frac{N_q}{N_c}\frac{B}{L}$$

$$s_q = 1 + \frac{B}{L}\tan\phi$$

$$s_\gamma = 1 - 0.4\frac{B}{L}$$

Depth factor

$$d_c = 0.4\,k$$

$$d_c = 1 + 0.4\,k$$

$$d_q = 1 + 2\tan\phi\,(1 - \sin\phi)^2\,k$$

$$d_\gamma = 1$$

Load inclination factor

$$i_c = i_q = \left(1 - \frac{\theta°}{90°}\right)$$

$$i_\gamma = 0$$

$$i_\gamma = \left(1 - \frac{\theta°}{\phi°}\right)^2$$

Inclination factor of the laying surface

$$b_c = \frac{\eta°}{147°}$$

$$b_c = 1 - \frac{\eta°}{147°}$$

$$b_q = b_\gamma = (1 - \eta\ tg\phi)^2$$

Slope inclination factor

$$g_c = \frac{\beta°}{147°}$$

$$g_c = 1 - \frac{\beta°}{147°}$$

$$g_q = g_\gamma = (1 - tg\beta)^2$$

Where θ, β and η represent the slope of the load, the slope slope and the slope of the foundation, respectively.

11.6.3.1.2.5
Bearing capacity of foundation

In the presence of stratified soils, if the thickness measured from the foundation plane of the soil layer on which the foundation rests is greater than B, the soil can be considered homogeneous. In the hypothesis that this circumstance is not verified, an equivalent angle of shear resistance f, a cohesion c and a specific weight of the soil γ can be calculated.

The height of the rigid soil wedge is determined H, in which fi is the f relative to the laying surface of the foundation:

$$H = 0{,}5\ tan\left(45 + \frac{\varphi_i}{2}\right)$$

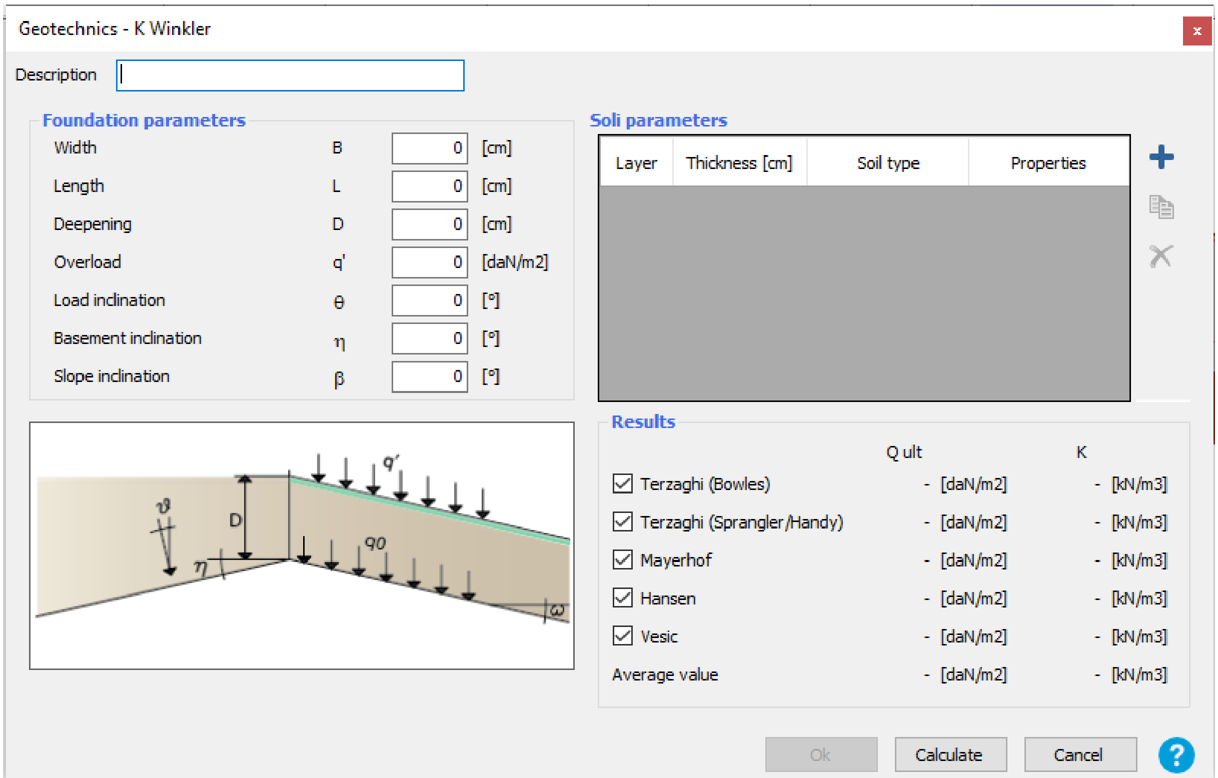
If H> zi (with zi the height of the i-th soil layer and q its height), the value of equivalent f to be used in the calculation of q_{ult} is determined as:

$$\varphi = \frac{\sum_1^{n-1} \varphi_i d_i + \varphi_i (H - z_i)}{H}$$

In a similar way, equivalent c and γ are obtained.

11.6.3.1.3 Work environment

The main working environment is as follows:

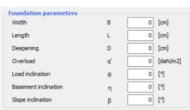


The main work area is divided into 3 sections:

1. Foundation parameters
2. Soil parameters
3. Results

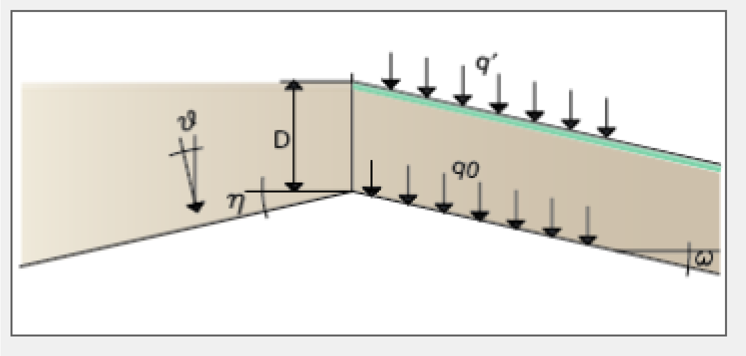
11.6.3.1.3.1 Main area

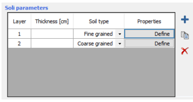
The main input window is divided into two sections:



Foundation parameters

Indicate in this section the length L, the width B and the depth D of the foundation. In addition, the overload q on the sides of the foundation, the inclination angle of the load θ, the inclination of the foundation η and the inclination of the slope β must be entered.

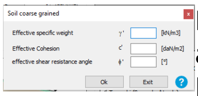




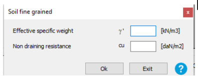
Soil parameters

In this section it is possible to indicate the stratigraphy of the soil from 1 to n layers simply by adding or removing the layers according to the desired stratigraphy.
For each layer it is possible to indicate its thickness and characterize it according to whether it is fine or coarse-grained soil.
Depending on the type of terrain selected, it is possible to insert the parameters γ, f and c in the corresponding window that will open when the button is clicked.

Coarse-grained soil window



Fine-grained soil window



Results

In this section, the following results are printed:
Bearing capacity Qult with the methods of Terzaghi (Bowles), Terzaghi (Sprangler / Handy), Mayerhof, Hansen and Vesic
K by Winkler with the methods of Terzaghi (Bowles), Terzaghi (Sprangler / Handy), Mayerhof, Hansen and Vesic
It is also possible to calculate the average value of Winkler Qult and K bearing capacity between the methods selected

.

11.6.3.2 Piles

11.6.3.2.1 Introduction

The "Piles" module is designed to calculate the value of the base and / or lateral capacity of the type of foundation pile adopted and on the basis of the analysis method chosen. All this is possible through the compilation of some input areas, which take into consideration the type of pole, the analysis method, the calculations of the base and lateral capacity, the characteristics of the ground and finally the tensions and pressures acting on the pole. .
The foundation piles are structural elements capable of transferring the load applied to

their tops, to the deeper and generally more resistant layers of soil. They can be classified in many ways.

The most common are:
-
- based on the material they are made of;
- according to the installation method.

Commonly used materials for posts are concrete, steel and wood.
Regarding the installation method, the poles can be classified through the following categories:
IA: dry bored piles, bored piles with support of the walls by means of mud, piles made with continuous helix and central shaped tube, micropiles (injected at low pressure).
IB: Bored piles with support of the walls of the excavation by means of pipes, fixed piles cast on site.
IIA: Prefabricated poles in reinforced or prestressed concrete, poles in profiles or steel tubes.
IIIA: Injected driven poles.
IIIB: High pressure injected poles; micropiles (injected at high pressure).

Furthermore, due to the effects of driving the poles, the type of soil is of great importance.
Among the vast stratigraphy of the terrain, the following types can be identified:

- Soft clay and mud
- Moderately compact clay
- Silt and loose sand
- Compact to hard clays and compact silts
- Soft white (chalk) limestone
- Moderately thickened sand and gravels
- Alternating and fractured white limestones (chalk)
- Thick to very thickened sand and gravel

11.6.3.2.2 Calculation procedure

The bearing capacity of a pole can be calculated by:
- static formulas
- dynamic formulas
- load tests
- static or dynamic penetometric tests (CPT, Cone Penetration Test)

The bearing capacity of a pole depends on the following factors:
- from the geometry of the problem;
- the shape and type of pole;
- the stratigraphic conditions varying with depth;
- the extent of the load applied;
- from the time elapsed between execution and application of the load.

Furthermore, the bearing capacity of the pole is also influenced by:
- installation method;
- from the interaction between the pole and the ground.

Numerous studies have shown that methods based on static penetometric tests (CPT) are more accurate than other conventional methods. This is mainly due to the fact that the static penetrometric tests provide a continuous profile of the characteristics of the

foundation soils.
The methods of analysis for evaluating the bearing capacity of the pile are then described, which provide the values of the base and lateral load capacity as a function of the type of pile and soil.

11.6.3.2.2.1 Base capacity

The analysis methods implemented for the calculation of the base capacity are:
- Method of Bustamante and Gianeselli (1982)
- Method of De Ruiter and Beringen (1979)
- Method of Almeida et al. (1996)
- Method of Jamiolkowski and Lancellotta (1982)

Also known as the LCPC method, the method is based on the analysis of 197 load tests on piles made in different types of soil.

Required data:

- Pole diameter D;
- Depth of the base of the pole z;
- Point resistance qc measured at depths between 1.5 D above and below the base of the pole;

The value of the basic flow rate qb is obtained from the following relation:

$$q_b = k_c \cdot q_{ca}$$

where:

- k_c is a coefficient of lift depending on the type of terrain
- q_{ca}: mean equivalent value of the resistance at the tip in the section + - 1.5D from the base of the pole.

The method can be applied to poles that are at least 8 times the diameter in length.
The proposed method employs different procedures depending on whether the soils are sandy or clayey.

<u>Sandy soils</u>

Required data:

- Pole diameter D;
- Depth of the base of the pole z;
- Point resistance qc measured at depths between 8D above and 4D below the base of the pole

The value of the basic flow rate qb is obtained from the following relation:

$$q_b = k_{OCR} \cdot (q_{c1}+q_{c2}) / 2$$

where:

- qc1 and qc2 are the average tip resistances calculated, respectively, in the sections 8D above and between 0.7 and 4D below the base of the pole;

- kOCR is a correction coefficient depending on the degree of over-consolidation of the soil.

For the evaluation of kOCR, the program assumes:

- OCR = 1 for: weathered and crushed white limestone (chalk), soft white limestone (chalk);
- OCR = 2-4 for: moderately thickened sand;
- OCR = 6-10 for: thickened to very thickened sand and gravel.

The flow rate calculated with this method will always be a maximum of 15 MPa.

Clayey soils

Required data:
- Point resistance qc measured;
- Correlation coefficient Nk (variable between 15 and 20) between the tip resistance and the undrained shear resistance cu;

The value of the basic flow rate qb is obtained from the following relation:

$$q_b = N_c \cdot c_u$$

where :

- Nc is an assumed coefficient equal to 9
- cu is undrained shear strength calculated as qc / Nk

The method is based on tests carried out on steel piles with a diameter between 0.102 m and 0.812 m and a length / diameter ratio between 27 and 66, driven into clayey soils.

Required data:
- Point resistance qc measured;
- Total vertical tension present in the site σv0;
- Effective vertical tension of the ground σ'v0;
- Net area of the piezocone An;
- Area of the base of the piezocone Ac;
- Neutral pressure measured with a porous filter at the base of the cone u2;

For clays only, Almeida et al. (1996) found a relationship between the tip resistance values for the driven poles and the base range according to the following relationship:

$$q_b = \frac{(q_t - \sigma_{v0})}{k_1}$$

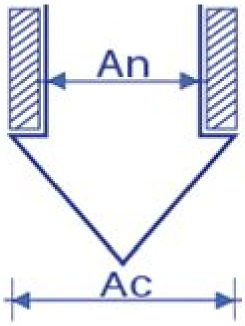
where:

- k1 is a correlation constant calculated as:

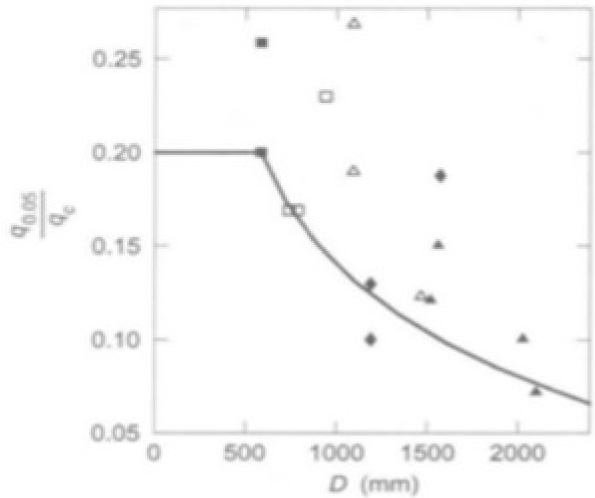
$$k_1 = 12 + 14{,}9 \cdot \log\left(\frac{q_t - \sigma_{v0}}{\sigma'_{v0}}\right)$$

- qt is the tip resistance corrected by the pore pressure, calculated as:

$$q_t = q_c + u_2 \cdot (1 - \frac{A_n}{A_c})$$



Jamiolkowski and Lancellotta, based on the data of 15 load tests, suggest an empirical correlation between diameter D, tip strength q0.05 and base capacity qc as shown in the following graph:



Required data:
- Pole diameter D;
- Point resistance qc understood as the load corresponding to a relative yielding of 5%

11.6.3.2.2.2 Lateral capacity

The methods of analysis implemented for the calculation of the lateral capacity are:
- Method of Bustamante and Gianeselli (1982);
- Method of De Ruiter and Beringen (1979);
- Almeida method (1996);
- De Beer method (1985).

Also known as the LCPC method, the method is based on the analysis of 197 load tests on piles made in different types of soil.

Required data:
- Toe resistance qc;

The value of the lateral flow fs is obtained from the following relation:

$$f_s = \frac{q_c}{\alpha}$$

where:
- α is the adhesion coefficient evaluated according to the type of soil and pile

The method can be applied to poles that are at least 8 times the diameter in length.
The proposed method employs different procedures depending on whether the soils are sandy or clayey.

Sandy soils

Required data:
- Lateral friction fs measured with a CPT test;
- Toe resistance qc;

The lateral flow rate is given by the minimum between:
- 0.12 MPa
- fs
- qc / 300 or qc / 400 depending on whether the pole works, respectively, by compression or traction.

Clayey soils

Required data:
- Point resistance qc measured;
- Correlation coefficient Nk (variable between 15 and 20) between the tip resistance and the undrained shear resistance cu;

The value of the lateral flow fs is obtained from the following relation:

$$f_s = \alpha \cdot c_u$$

where:
- α is 1 for normalconsolidated clays and 0.5 for superconsolidated clays
- cu is undrained shear strength calculated as

$$c_u = q_c / N_k$$

The method is based on tests carried out on steel piles with a diameter between 0.102 m and 0.812 m and a length / diameter ratio between 27 and 66, driven into clayey soils.

Required data:
- Point resistance qc measured;
- Total vertical tension present in the site σv0;
- Net area of the piezocone An;
- Area of the base of the piezocone Ac;
- Neutral pressure measured with a porous filter at the base of the cone u2;
- Nkt cone factor (variable between 10 and 30) used to derive the undrained shear resistance from the tip resistance-

For clays only, Almeida et al. (1996) found a relationship between the tip resistance values and the lateral bearing for the driven poles according to the following relationship:

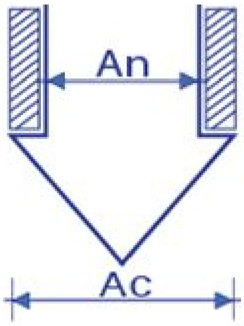
$$f_s = \frac{(q_t - \sigma_{v0})}{k_2}$$

where:
- k2 is a correlation constant calculated as:

$$k_2 = \frac{N_{kt}}{9}$$

- qt is the tip resistance corrected by the pore pressure, calculated as:

$$q_t = q_c + u_2 \cdot (1 - \frac{A_n}{A_c})$$



The experiences gathered by De Beer directly relate the resistance at the tip with the lateral flow rate for the poles driven in sand according to the following relationship:

$$f_s = \frac{q_c}{200} \text{ if } q_c \geq 20 \text{ MPa}$$

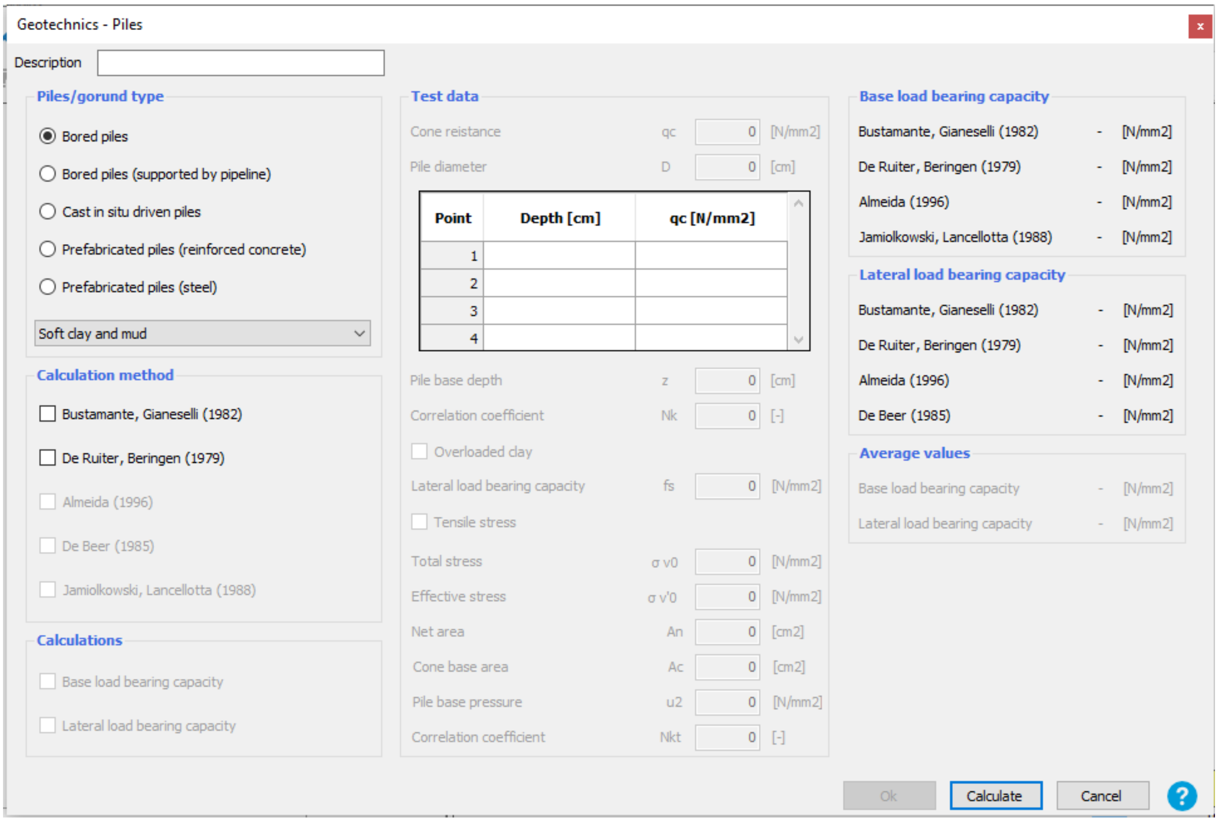
$$f_s = \frac{q_c}{150} \text{ if } q_c \leq 10 \text{ MPa}$$

Where qc is the measured tip resistance.

11.6.3.2.3 Work environment

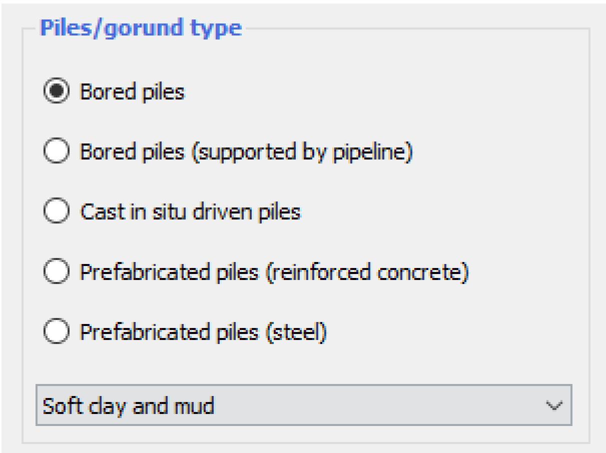
The working environment of the module can be divided into four parts:

- Piles/ ground type;
- Calculation method;
- Test data;
- Results.

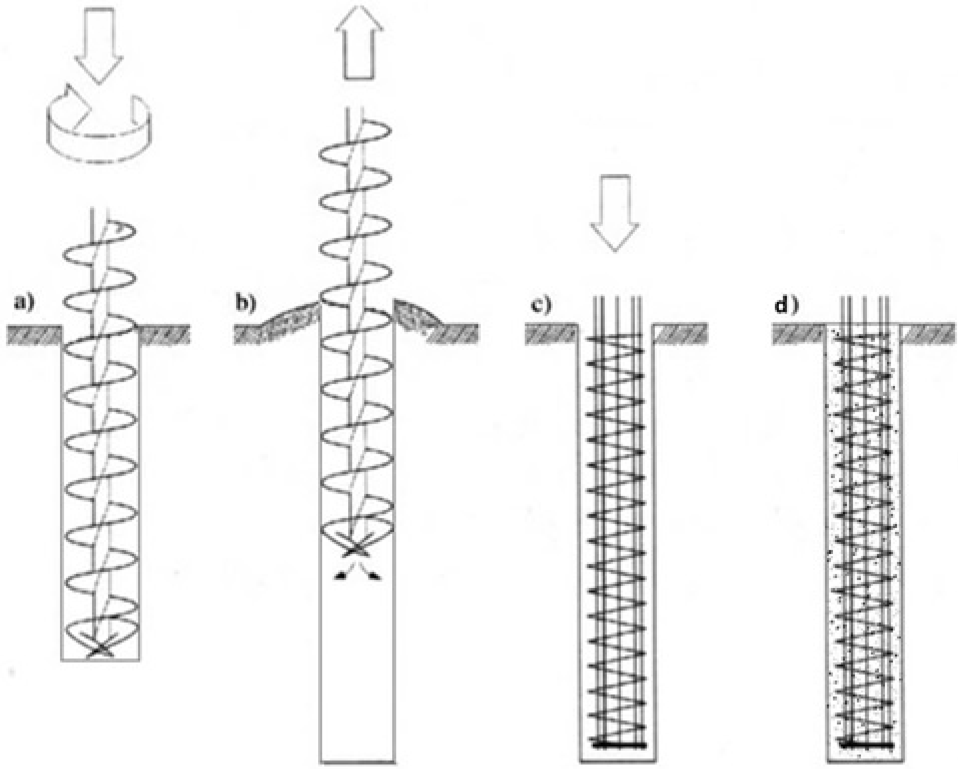


11.6.3.2.3.1 Piles/ground type

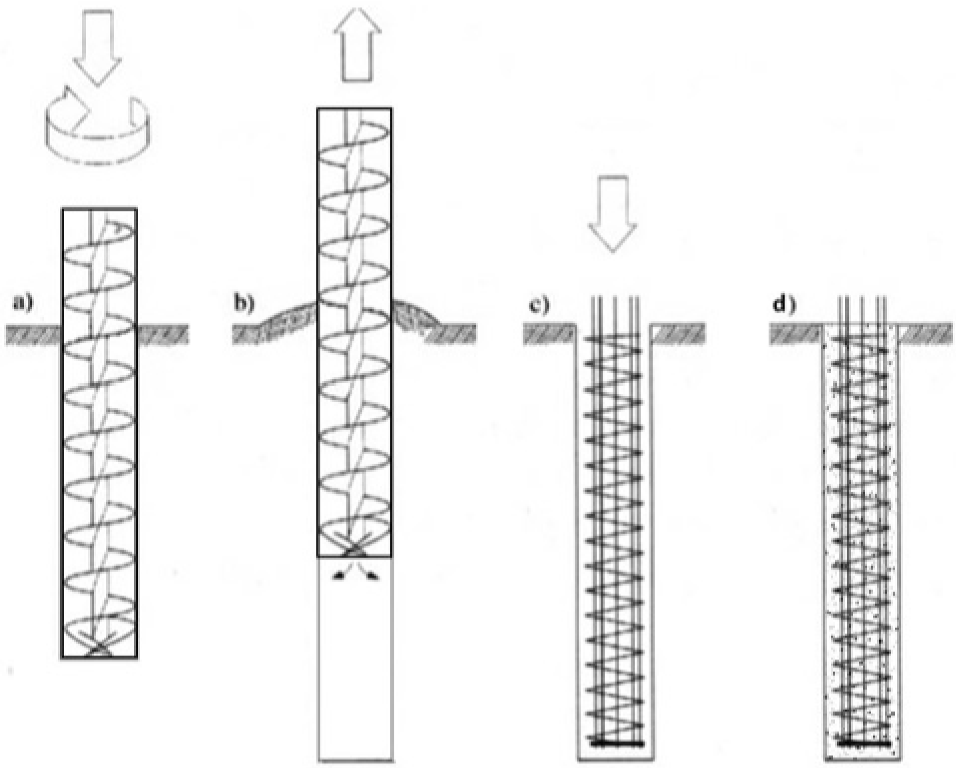
To begin the analysis it is necessary to define the type of pile to be calculated and the land in which it is used.



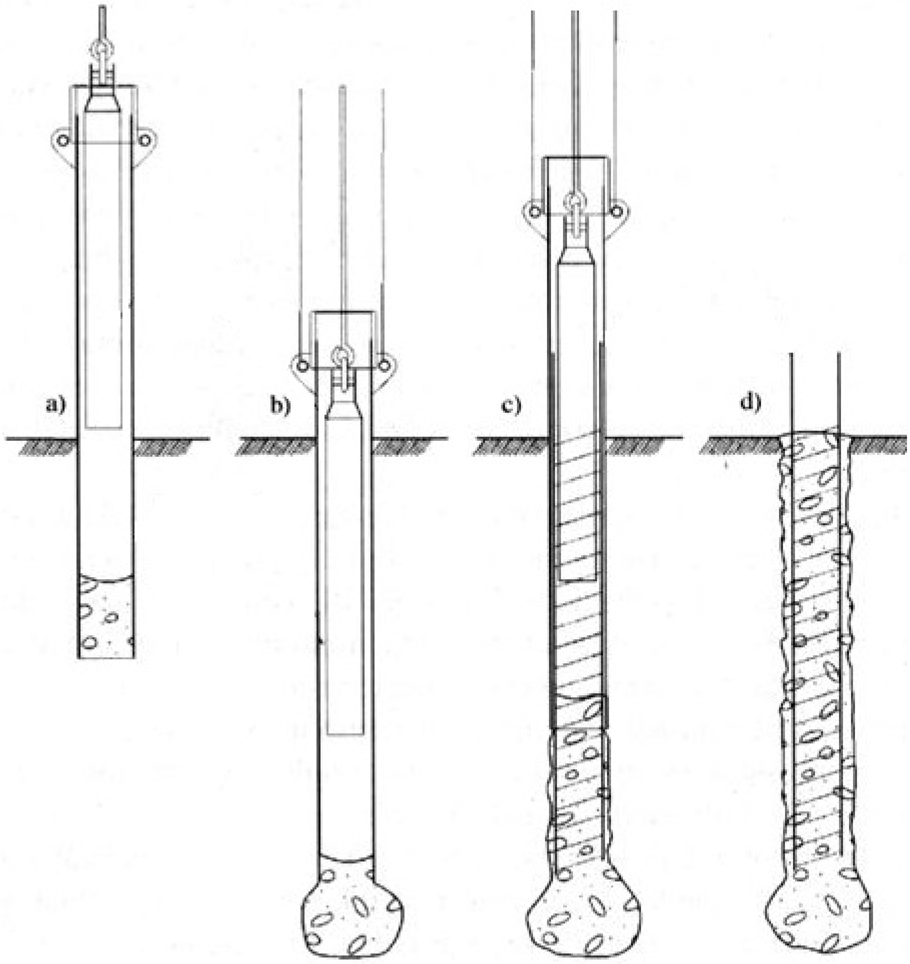
Bored piles



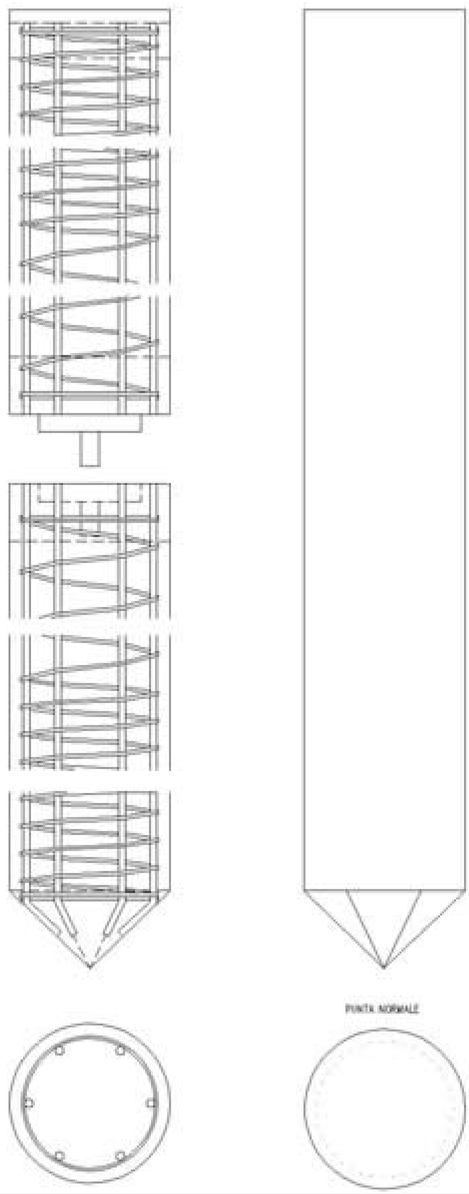
Bored piles with pipe support



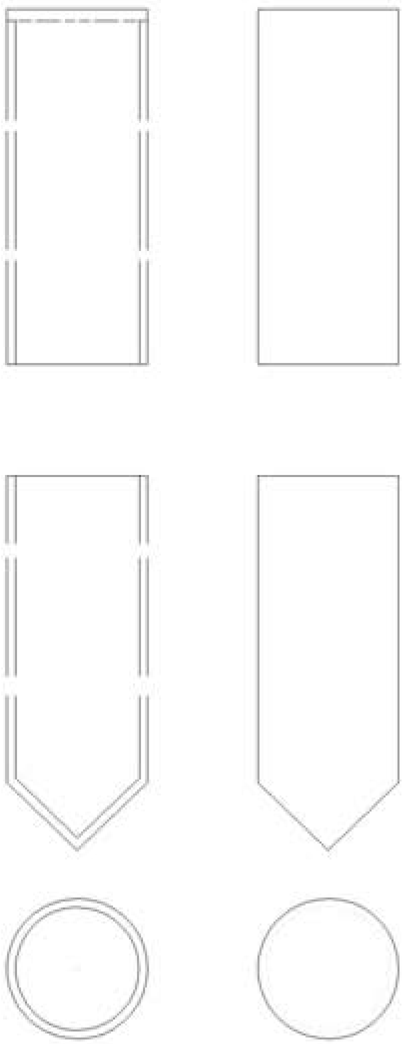
Piles cast on site



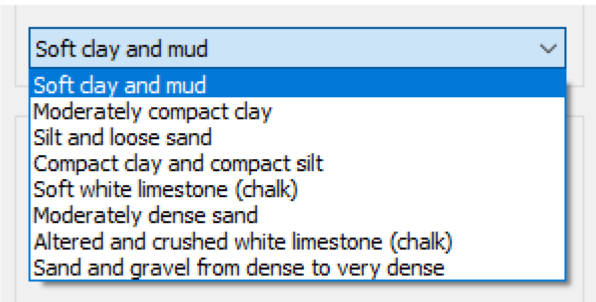
Prefabricated reinforced concrete piles



Prefabricated steel poles

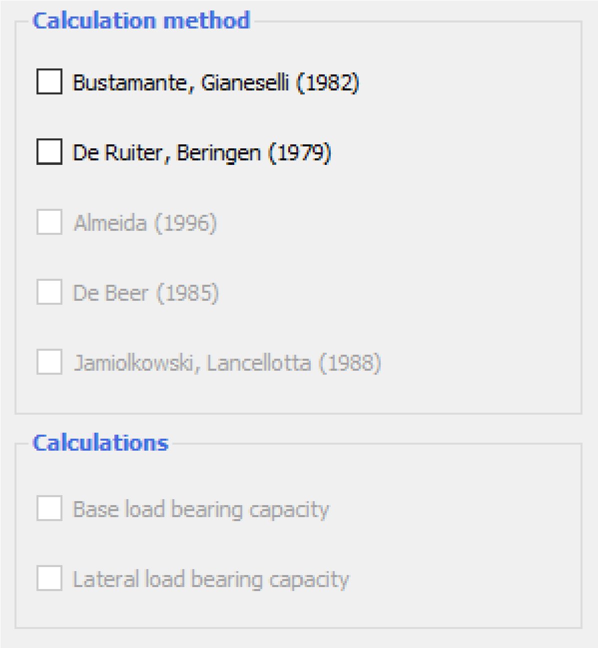


Type of terrain



11.6.3.2.3.2 Calculation method

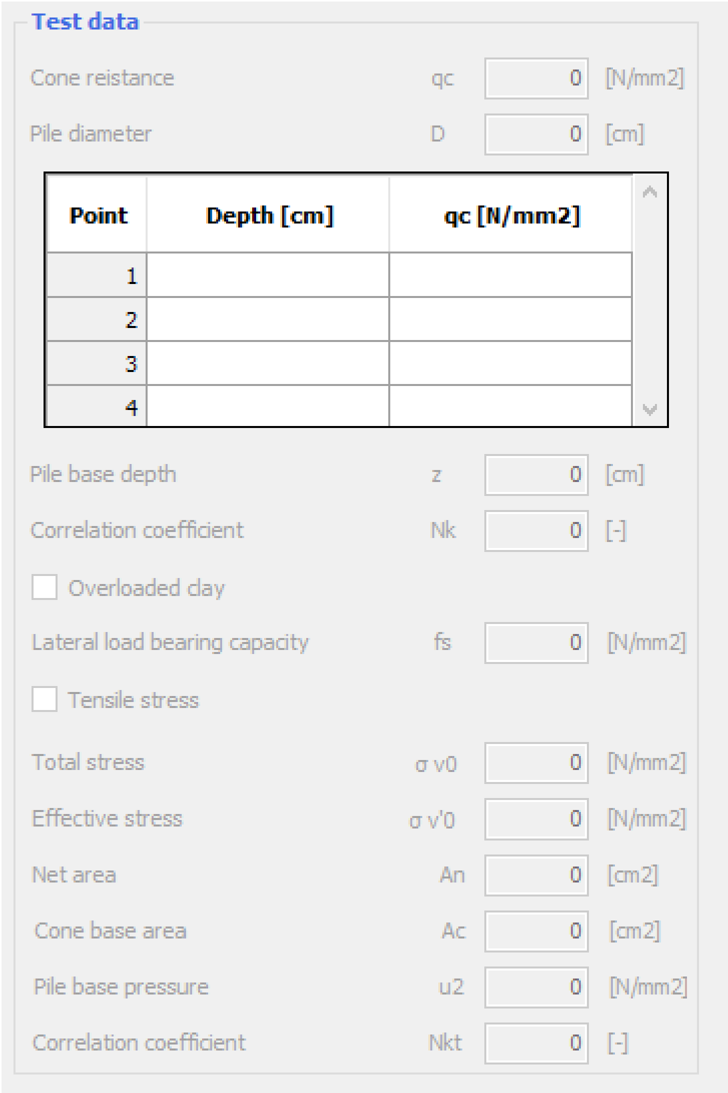
To perform the calculation it is necessary to choose at least one of the calculation methods proposed for the calculation of the flow rate.
Depending on the type of pile and soil selected, different calculation methods can be used. See the chapter on the calculation procedure for more details.
By selecting more than one calculation method, the program will return both the capacity calculated with each method and the average capacity.

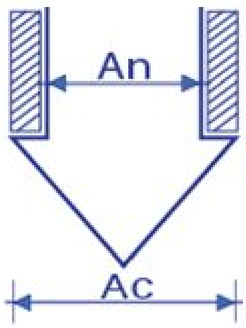


It is also necessary to indicate whether you intend to calculate the basic, lateral or both capacity.

11.6.3.2.3.3 Test data

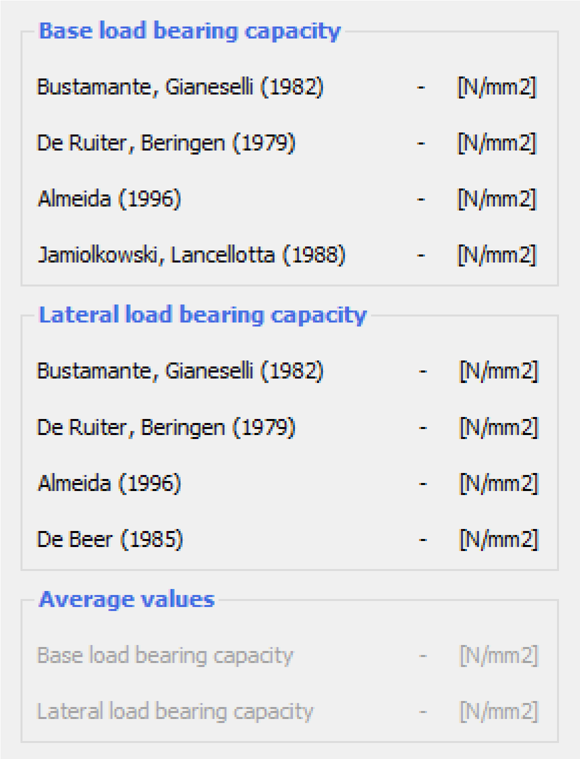
In the section "Test data" it is necessary to enter the characteristics of the pile and the ground required for the application of the calculation methods selected in the Calculation method.
For more details on the necessary data, their meaning and validity limits, refer to the Calculation procedure chapter.





11.6.3.2.3.4 Results

The "Results" section shows the flow rates calculated with each selected method and the average values.



11.6.3.2.4 Bibliography

[1] Mauro Tanzini - "Manuale del Geotecnico", Dario Flaccovio Editore (2010)
[2] R. Lancellotta, J. Cavalera – "Fondazioni", McGraw-Hill (1999)
[3] H.G. Polulos, E.H. Davis – "Pali", Libreria Dario Flaccovio Editrice (1987)

11.6.4 Fire

11.6.4.1 Flexion verification

11.6.4.1.1 Introduction

The Fire Beam module allows thermal analysis and straight bending verification of reinforced concrete sections in fire conditions (REI) through the analytical method as prescribed by the NTC08, EC1991-1-2 and EC1992-1-2 standards.

11.6.4.1.2 Calculation procedure

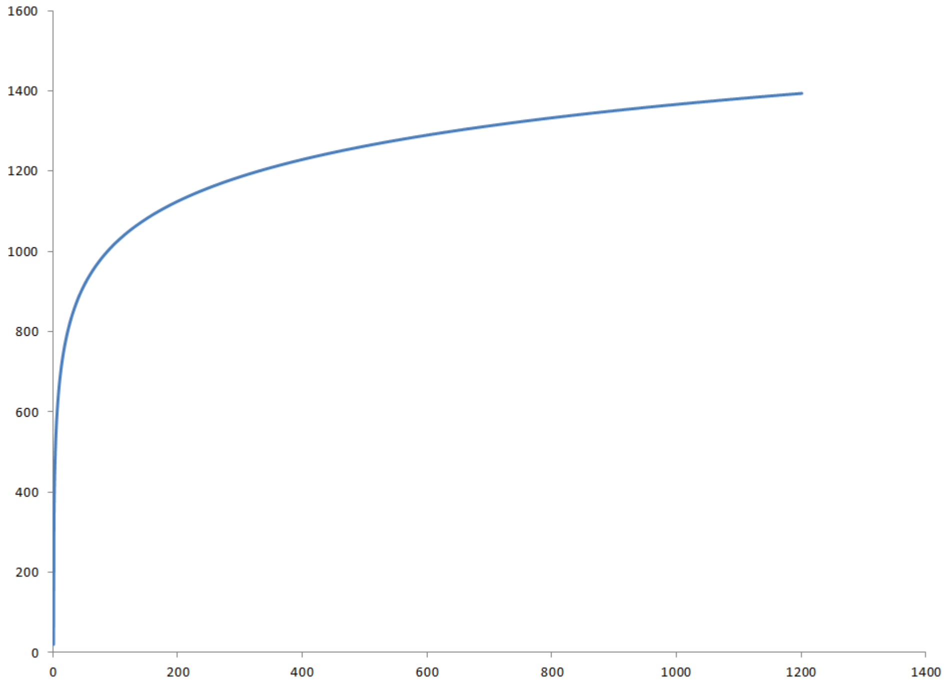
11.6.4.1.2.1 Thermal analysis

<u>FIRE EXPOSURE</u>

Each side can also be partially exposed according to one of the following nominal exposure laws:

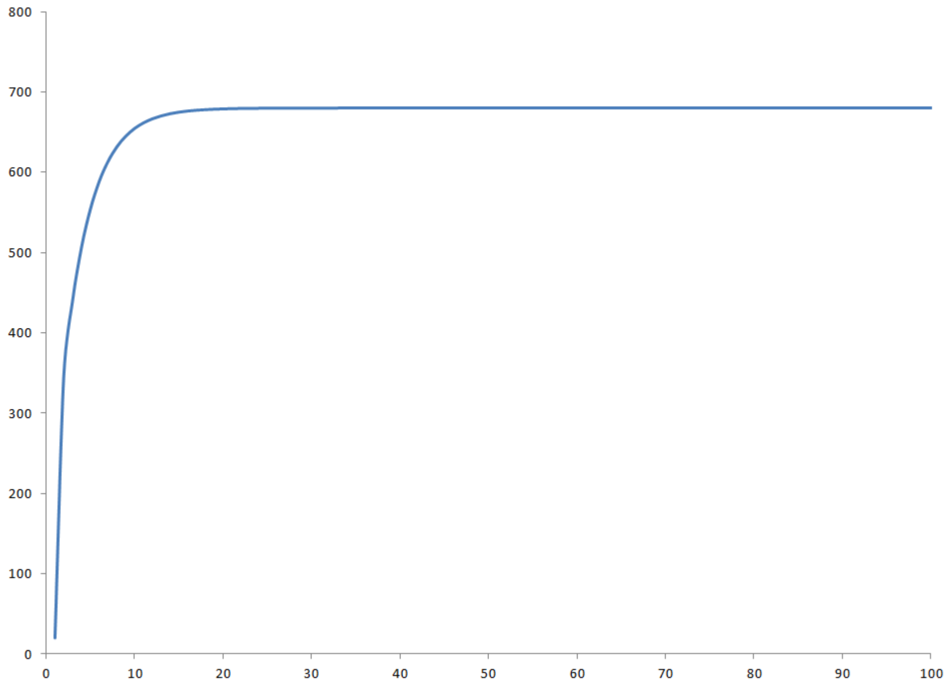
- Normalized fire exposure

$$\Theta_g = 20 + 345 \log_{10} (8\,t + 1)$$



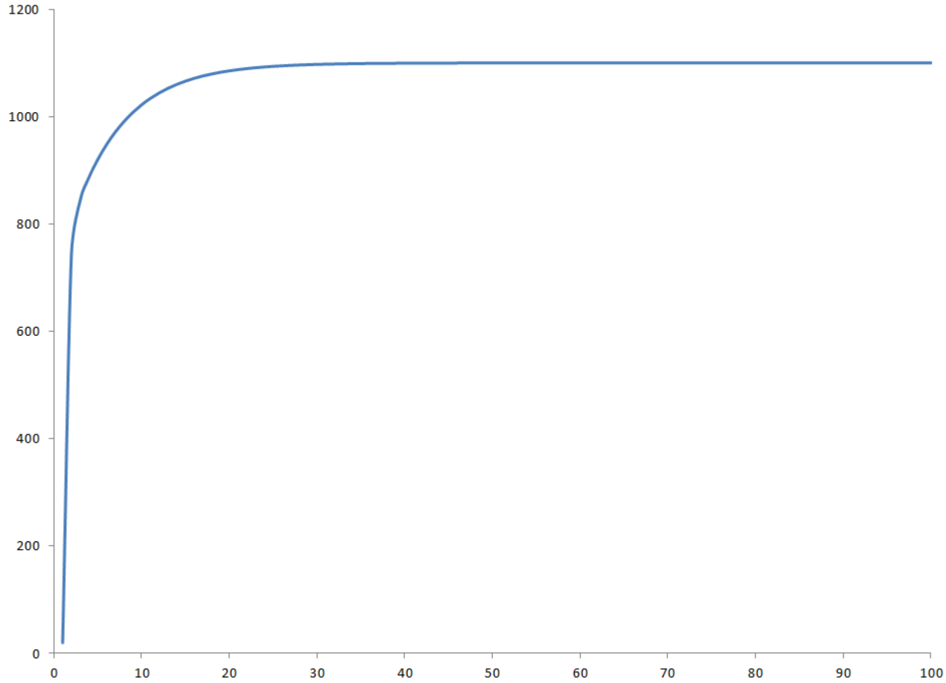
- Exposure from external fire

$$\Theta_g = 660\,(1 - 0{,}687\,e^{-0{,}32\,t} - 0{,}313\,e^{-3{,}8\,t}) + 20$$



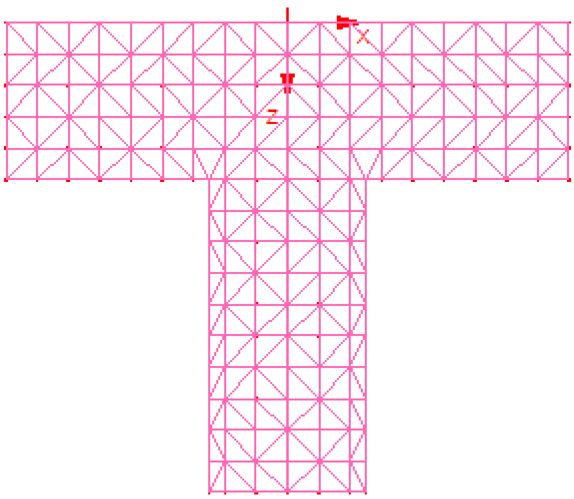
- Fire exposure from hydrocarbons fire

$$\Theta_g = 1\,080\,(1 - 0{,}325\,e^{-0{,}167\,t} - 0{,}675\,e^{-2{,}5\,t}) + 20$$



EXECUTION OF THE MESH

The program automatically performs the subdivision into elementary triangular surfaces (mesh), which the user can customize, to evaluate the temperatures and properties of the material in each node.



THERMAL CALCULATION

The diffusion of heat occurs in a non-stationary (transient) regime, in which the temperature varies not only as a function of space, but also as a function of time. In this case, accumulation or dissipation phenomena and the related physical properties are also taken into consideration. The differential equation governing the phenomenon is the Fourier equation:

$$\text{div}\,(\lambda_c \cdot \text{grad}\,\theta) + w = C_c \cdot \rho_c \cdot \frac{\delta\theta}{\delta t}$$

with initial condition T (0) = T0 = 20 ° C
and boundary condition

$\text{div}\ (\lambda_c \cdot \text{grad}\ \theta)_n = h_{net,d}$

where:

$\dot{h}_{net} = \dot{h}_{net,c} + \dot{h}_{net,r}$

where:

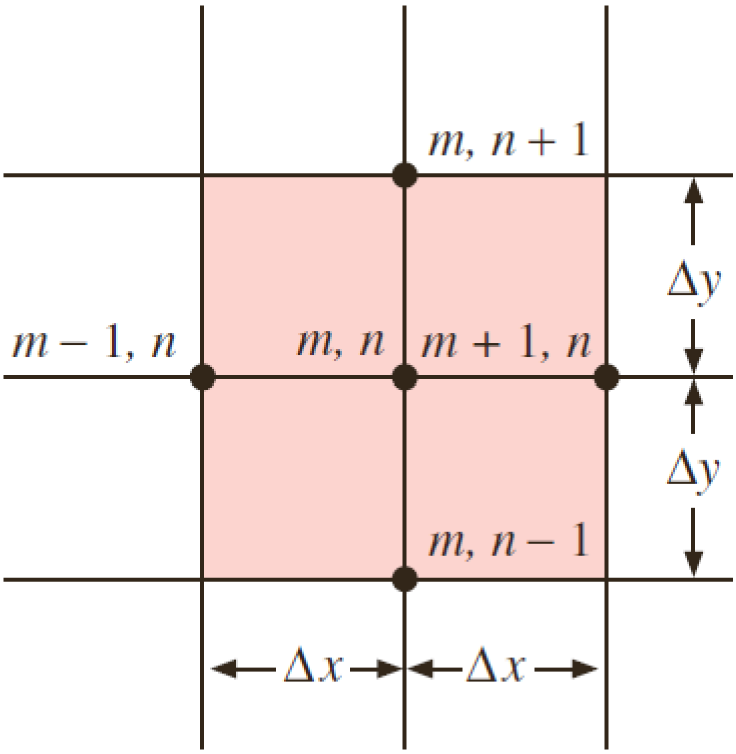
$$\dot{h}_{net,c} = \alpha_c \cdot (\Theta_g - \Theta_m) \qquad [W/m^2] \qquad (3.2)$$

Where:

αc is the coefficient of heat transfer by convection [W/m2K]
Θg is the temperature of the gas in the vicinity of the element exposed to fire [oC]
Θm is the surface temperature of the element [oC]

hnet,r MISSING

Given the non-linearity of the differential equation, a numerical resolution is used, referring to the explicit Euler method:



$$\left.\frac{\partial T}{\partial x}\right]_{m+1/2,n} \approx \frac{T_{m+1,n} - T_{m,n}}{\Delta x}$$

$$\left.\frac{\partial T}{\partial x}\right]_{m-1/2,n} \approx \frac{T_{m,n} - T_{m-1,n}}{\Delta x}$$

$$\left.\frac{\partial T}{\partial y}\right]_{m,n+1/2} \approx \frac{T_{m,n+1} - T_{m,n}}{\Delta y}$$

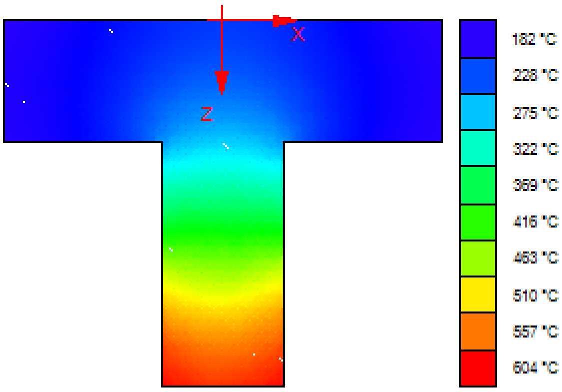
$$\left.\frac{\partial T}{\partial y}\right]_{m,n-1/2} \approx \frac{T_{m,n} - T_{m,n-1}}{\Delta y}$$

$$\left.\frac{\partial^2 T}{\partial x^2}\right]_{m,n} \approx \frac{\left.\frac{\partial T}{\partial x}\right]_{m+1/2,n} - \left.\frac{\partial T}{\partial x}\right]_{m-1/2,n}}{\Delta x} = \frac{T_{m+1,n} + T_{m-1,n} - 2T_{m,n}}{(\Delta x)^2}$$

$$\left.\frac{\partial^2 T}{\partial y^2}\right]_{m,n} \approx \frac{\left.\frac{\partial T}{\partial y}\right]_{m,n+1/2} - \left.\frac{\partial T}{\partial y}\right]_{m,n-1/2}}{\Delta y} = \frac{T_{m,n+1} + T_{m,n-1} - 2T_{m,n}}{(\Delta y)^2}$$

CREATION OF THERMAL MAPPING

The thermal mapping of the reinforced concrete section is determined.



EVALUATION OF NEW CHARACTERISTICS IN MATERIAL FIRE CONDITIONS

The decay of the mechanical characteristics of the materials is evaluated as a function of the temperatures obtained in each node of the mesh.

The results take into account the characteristics of materials and reinforcements (strength of concrete, strength of steel, arrangement and quantity of reinforcements, etc.).

Temperature of the concrete θ	Siliceous aggregates			Calcareous aggregates		
	$f_{c,\theta}/f_{ck}$	$\varepsilon_{c1,\theta}$	$\varepsilon_{cu1,\theta}$	$f_{c,\theta}/f_{ck}$	$\varepsilon_{c1,\theta}$	$\varepsilon_{cu1,\theta}$
[°C]	[-]	[-]	[-]	[-]	[-]	[-]
1	2	3	4	5	6	7
20	1,00	0,0025	0,0200	1,00	0,0025	0,0200
100	1,00	0,0040	0,0225	1,00	0,0040	0,0225
200	0,95	0,0055	0,0250	0,97	0,0055	0,0250
300	0,85	0,0070	0,0275	0,91	0,0070	0,0275
400	0,75	0,0100	0,0300	0,85	0,0100	0,0300
500	0,60	0,0150	0,0325	0,74	0,0150	0,0325
600	0,45	0,0250	0,0350	0,60	0,0250	0,0350
700	0,30	0,0250	0,0375	0,43	0,0250	0,0375
800	0,15	0,0250	0,0400	0,27	0,0250	0,0400
900	0,08	0,0250	0,0425	0,15	0,0250	0,0425
1 000	0,04	0,0250	0,0450	0,06	0,0250	0,0450
1 100	0,01	0,0250	0,0475	0,02	0,0250	0,0475
1 200	0,00	-	-	0,00	-	-

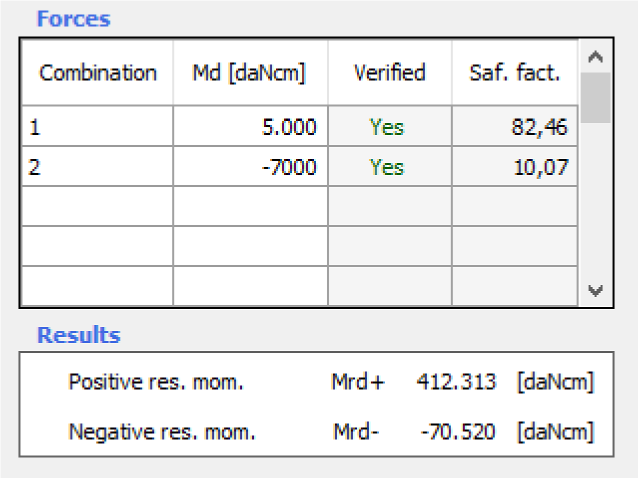
Class N, values for the stress-strain relationship parameters of hot-rolled and cold-drawn reinforcement steel at elevated temperatures

prospetto 3.2a

Steel temperature θ [°C]	$f_{sy,\theta}/f_{yk}$		$f_{sp,\theta}/f_{yk}$		$E_{s,\theta}/E_s$	
	hot rolled	cold rolled	hot rolled	cold rolled	hot rolled	cold rolled
1	2	3	4	5	6	7
20	1,00	1,00	1,00	1,00	1,00	1,00
100	1,00	1,00	1,00	0,96	1,00	1,00
200	1,00	1,00	0,81	0,92	0,90	0,87
300	1,00	1,00	0,61	0,81	0,80	0,72
400	1,00	0,94	0,42	0,63	0,70	0,56
500	0,78	0,67	0,36	0,44	0,60	0,40
600	0,47	0,40	0,18	0,26	0,31	0,24
700	0,23	0,12	0,07	0,08	0,13	0,08
800	0,11	0,11	0,05	0,06	0,09	0,06
900	0,06	0,08	0,04	0,05	0,07	0,05
1 000	0,04	0,05	0,02	0,03	0,04	0,03
1 100	0,02	0,03	0,01	0,02	0,02	0,02
1 200	0,00	0,00	0,00	0,00	0,00	0,00

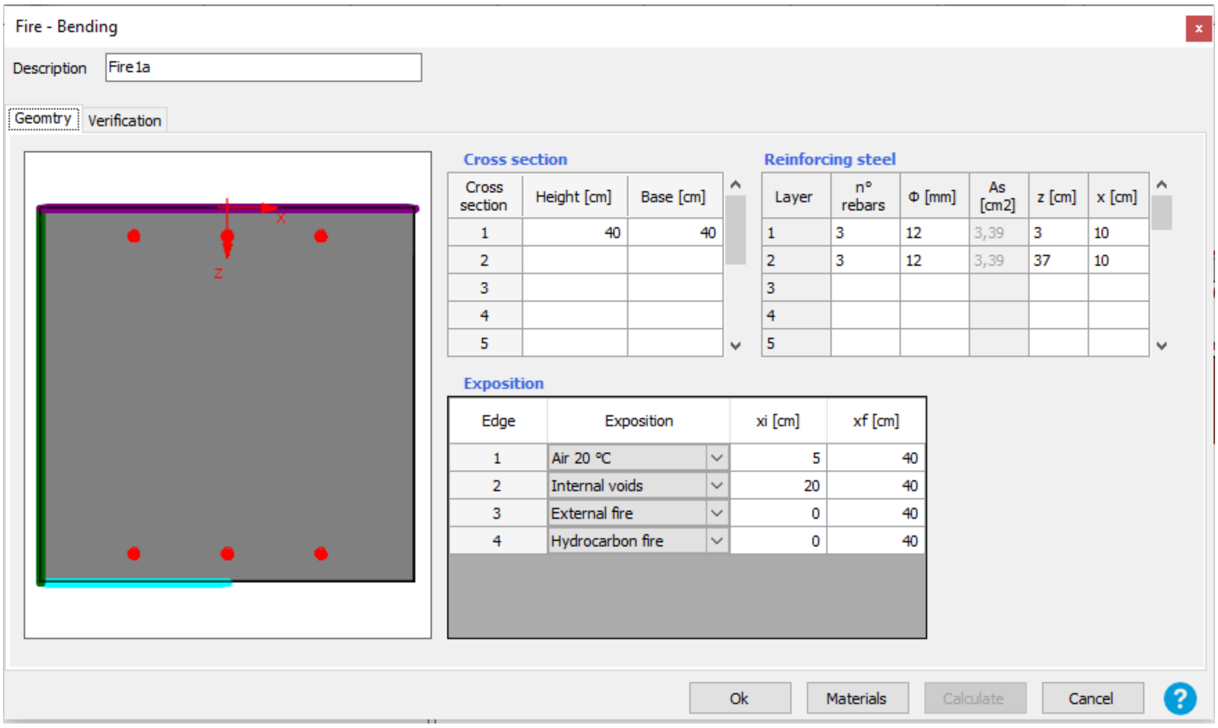
11.6.4.1.2.2 R.C. Verification

Performs ULS verification for straight bending according to EN1992-1-2, in fire conditions, calculating the positive moment of resistance (at the lower edge) and the negative moment of resistance (at the upper edge), explaining the safety coefficients relating to each combination entered in the table.



11.6.4.1.3 Work environment

The working environment is as follows:

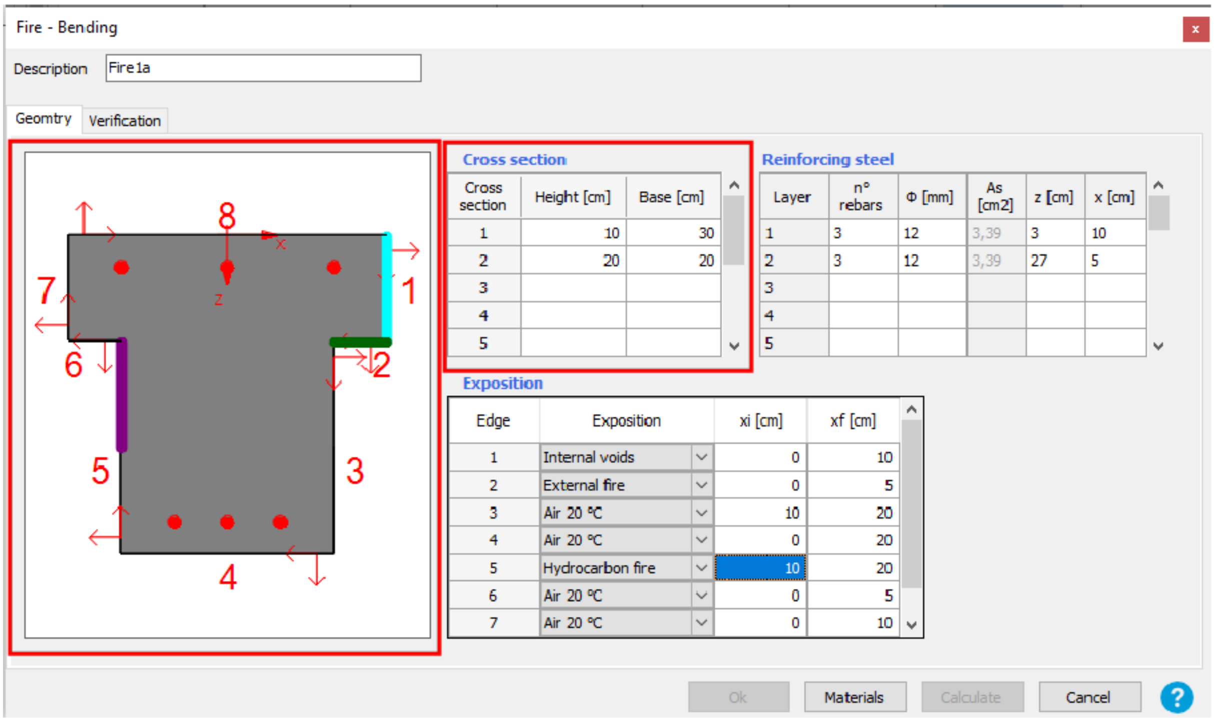


The main screen allows access to the tabs:

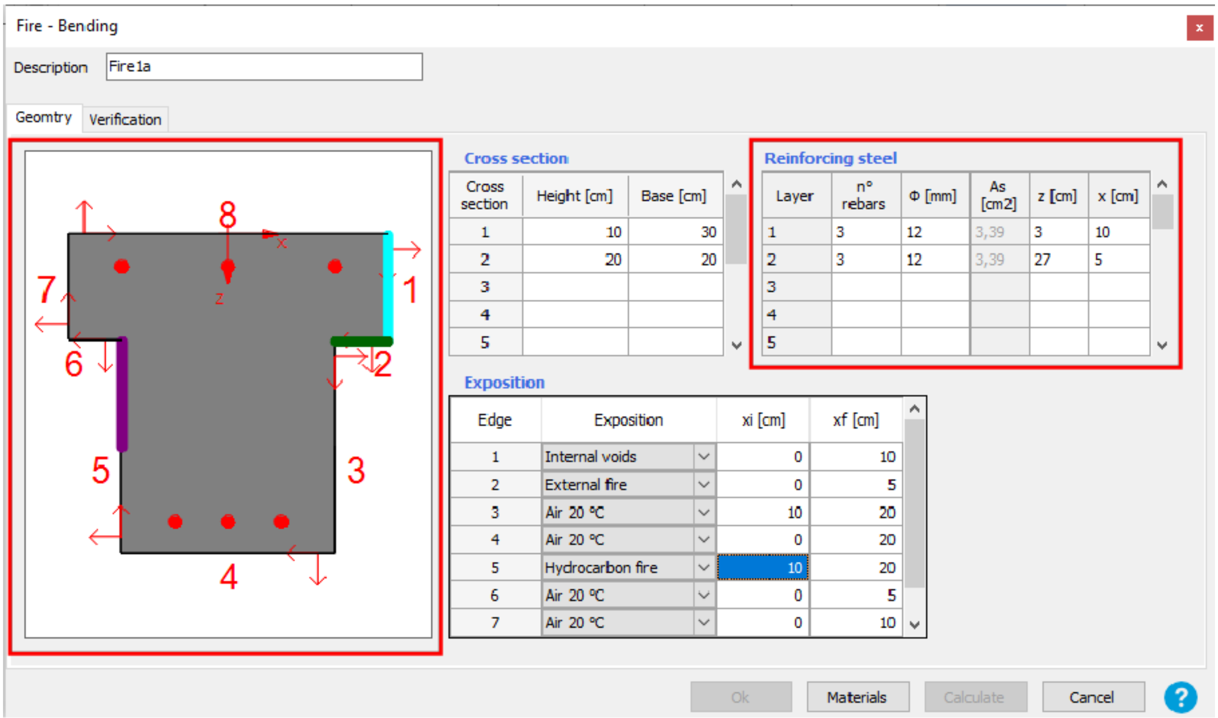
- Geometry;
- Materials;
- Verification.

11.6.4.1.3.1 Section geometry

In the Section Definition Area you define the geometry, reinforcement and exposure of the section to be checked.
The geometry is defined as a series of rectangles whose base and height will be defined.

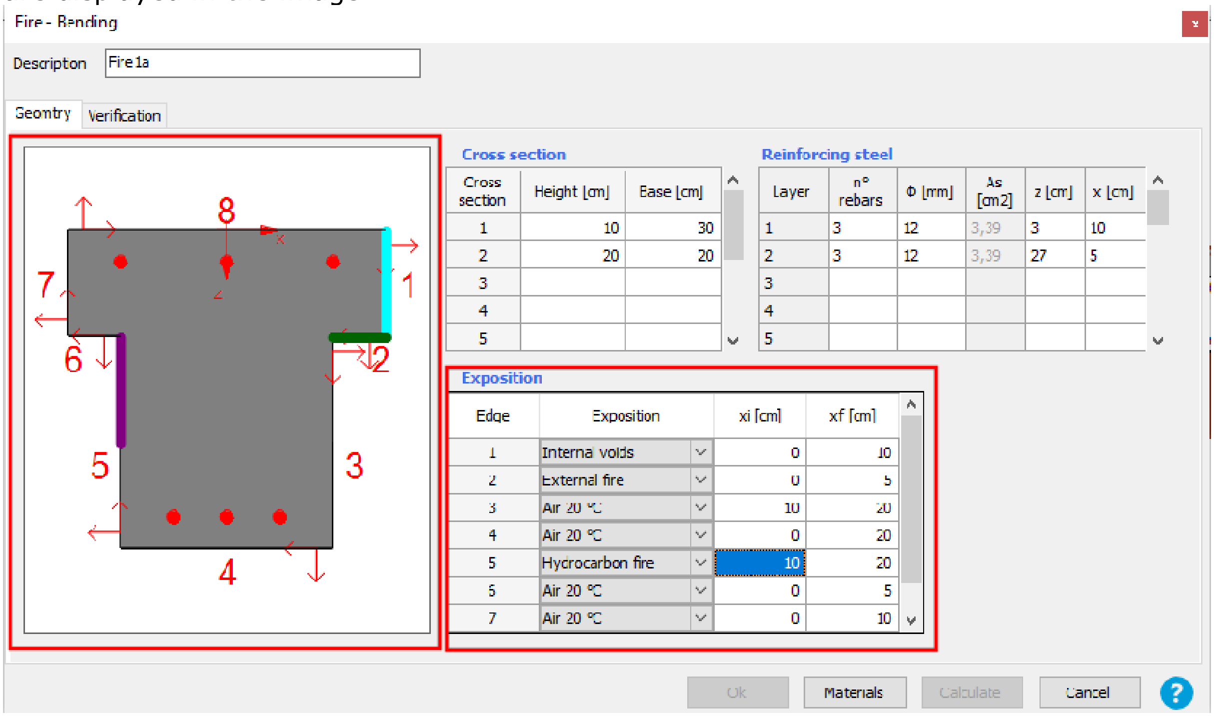


For the rectangular section, the reinforcement is defined by indicating the number of bars, the diameter and the position, with respect to the reference system indicated, of each layer. The x indicates the distance from the axis of symmetry of the furthest iron in the series, the program will arrange the internal bars at a constant pitch. If the layer has a single reinforcement bar, this will necessarily be fixed on the symmetry axis. Once the number and diameter have been assigned, the program will automatically fill in the cell with the reinforcement area.



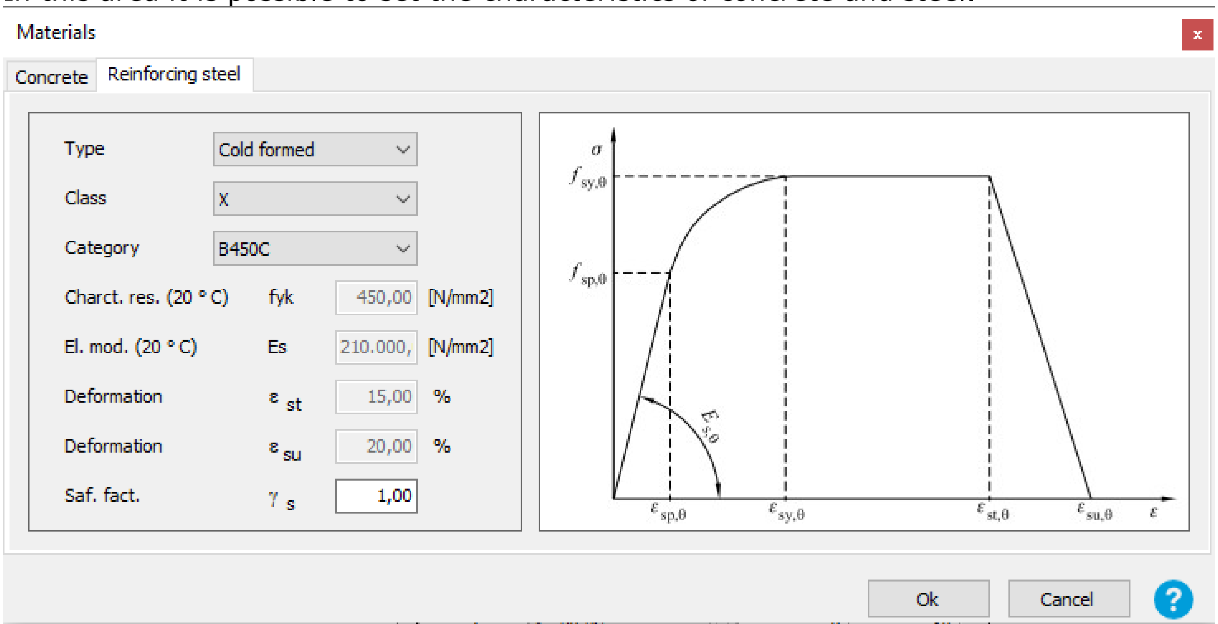
The exposure is defined on each side according to the local reference system. It is possible to define a partial exposure on each side by modifying the parameters xi, starting point, and xf, ending point, with respect to the local reference system.

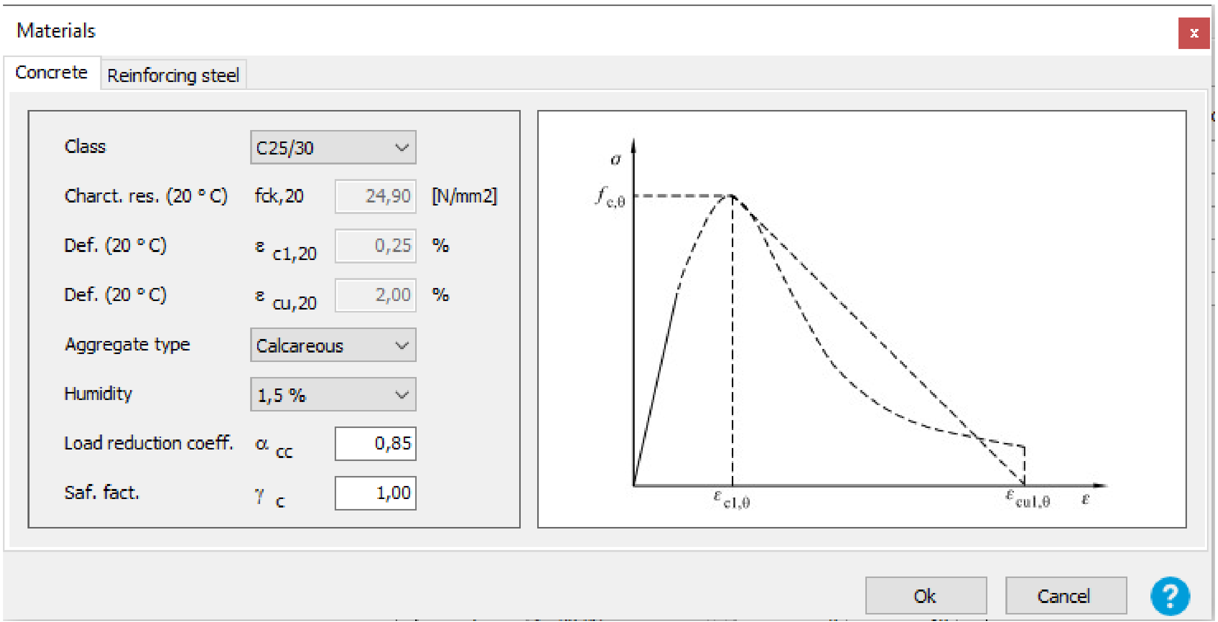
When the exposure table is active, the numbering of the sides and the reference systems are displayed in the image.



11.6.4.1.3.2 Materials

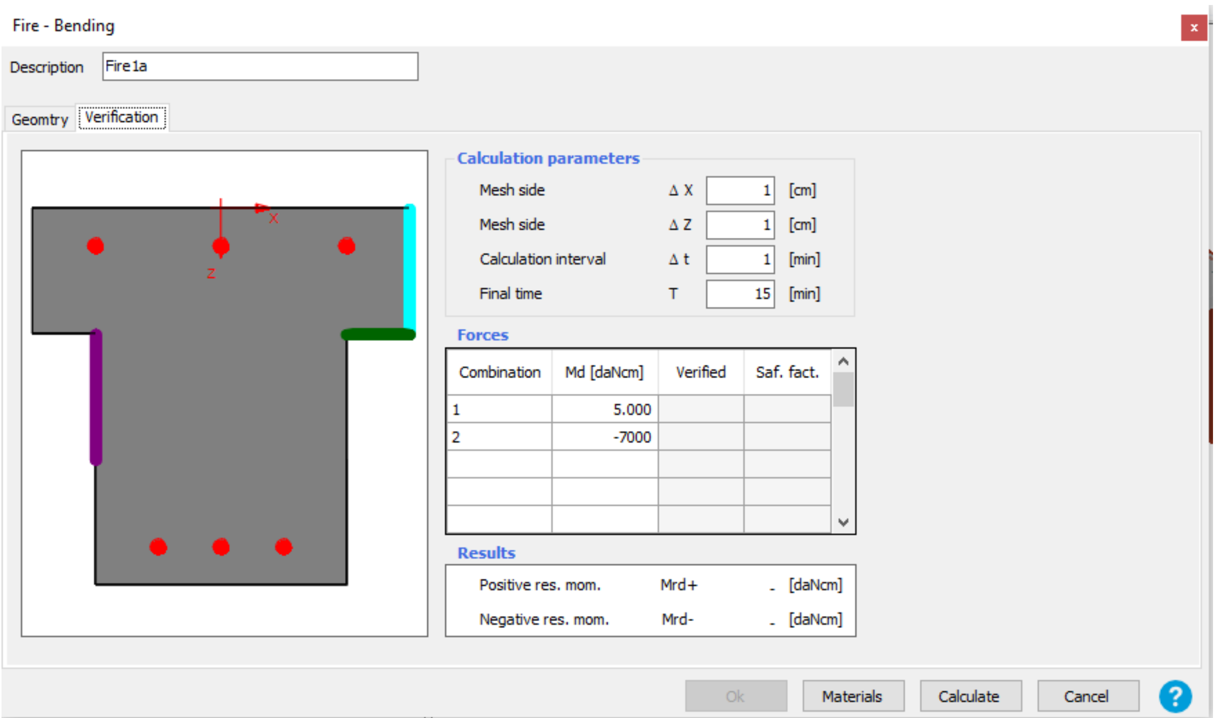
It is possible to access the Materials Area by clicking on the appropriate button.
In this area it is possible to set the characteristics of concrete and steel.



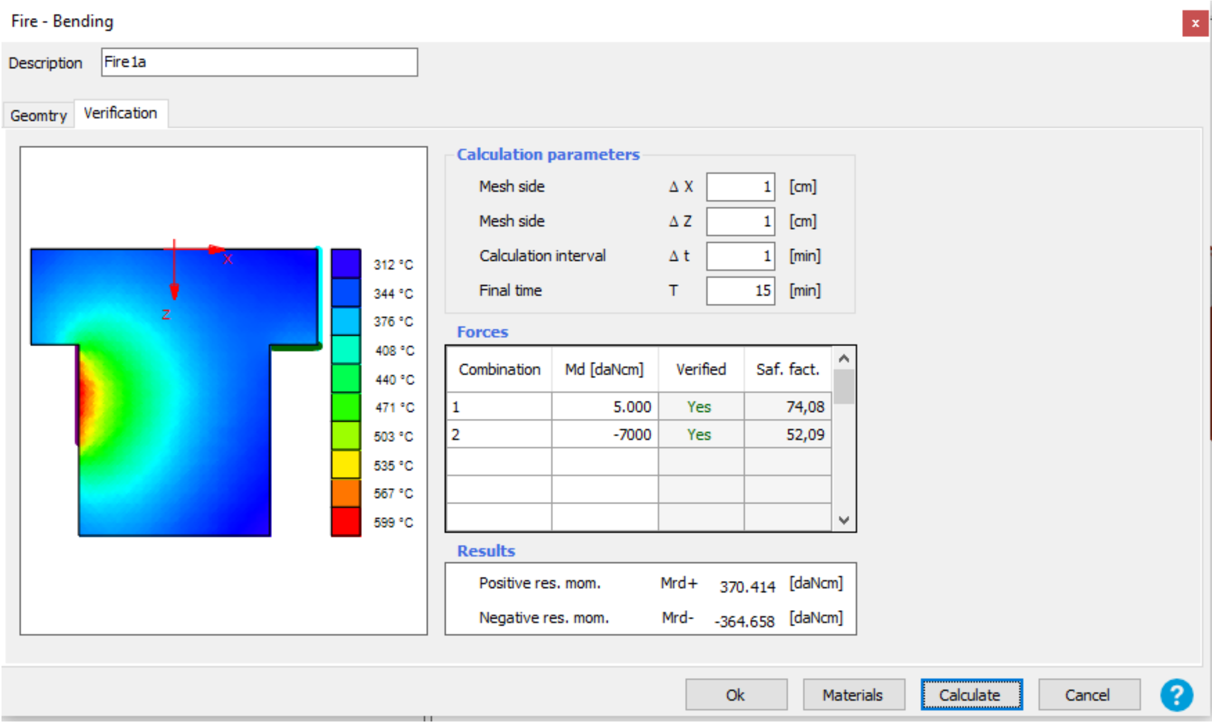


11.6.4.1.3.3 Verification

In the Verification Area, the calculation parameters are defined, such as:
- dimensions of the DX and DZ computational mesh
- calculation time interval Dt
- final calculation time T



With the Calculate button, the application calculates the positive and negative moment of resistance of the section and checks the combinations entered in the table according to EN1992-1-2, providing the relative safety coefficient.



11.6.4.1.4 Bibliography

[1] EC1991-1-2: Eurocode 1 - Actions on structures - Part 1-2: Actions in general - Actions on structures exposed to fire
[2] EC1992-1-2: Eurocode 2 - Design of concrete structures - Part 1-2: General rules - Structural fire design
[3] Ministerial Decree January 17, 2018 - Approval of the new technical standards for construction
[3] Circular 21 January 2019, n. 7 - Instructions for the application of the "Technical standards for constructions"
[4] J.Taler, P. Duda - Solving Direct and Inverse Heat Conduction Problems - Springer
[5] J. A. Purkiss - Fire Safety engineering - Design of structures - BH

11.6.4.2 Bending verification

11.6.4.2.1 Introduction

The Fire Pillars module allows the thermal analysis and the deflection test of reinforced concrete sections in fire conditions (REI) through the analytical method as prescribed by the NTC18, EC1991-1-2 and EC1992-1-2 standards.

11.6.4.2.2 Calculation procedure

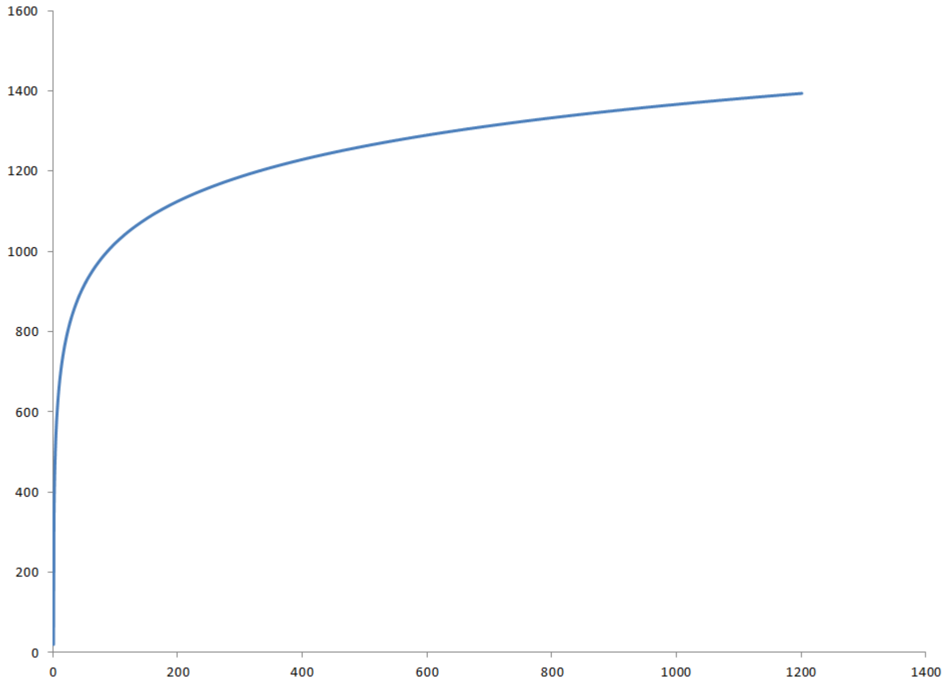
11.6.4.2.2.1 Thermal analysis

<u>FIRE EXPOSURE</u>

Each side can also be partially exposed according to one of the following nominal exposure laws:

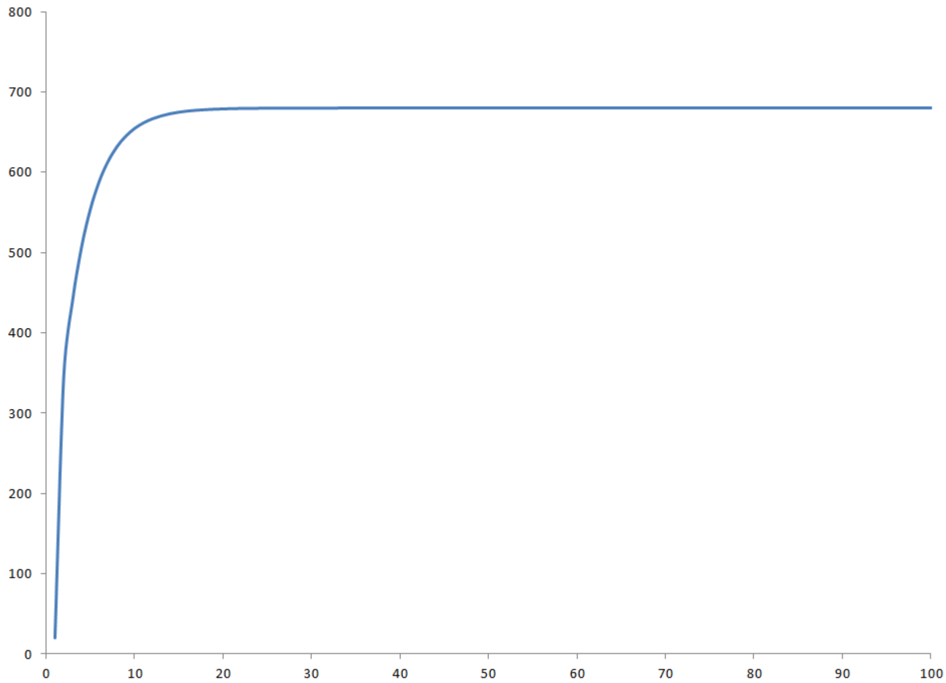
- Normalized fire exposure

$$\Theta_g = 20 + 345 \log_{10} (8\,t + 1)$$



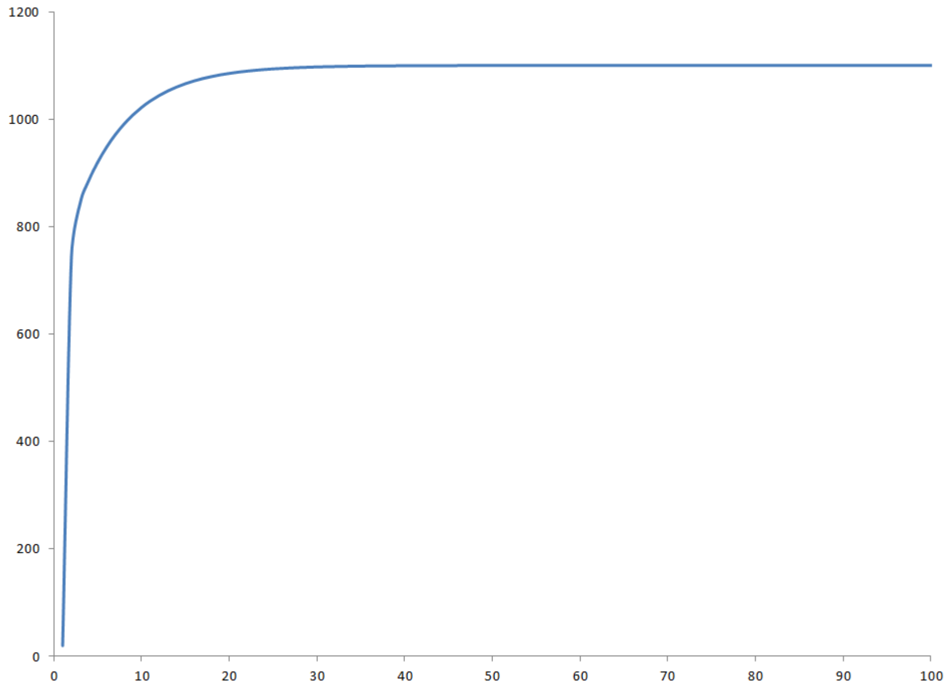
- Exposure from external fire

$$\Theta_g = 660\,(1 - 0{,}687\,e^{-0{,}32\,t} - 0{,}313\,e^{-3{,}8\,t}) + 20$$



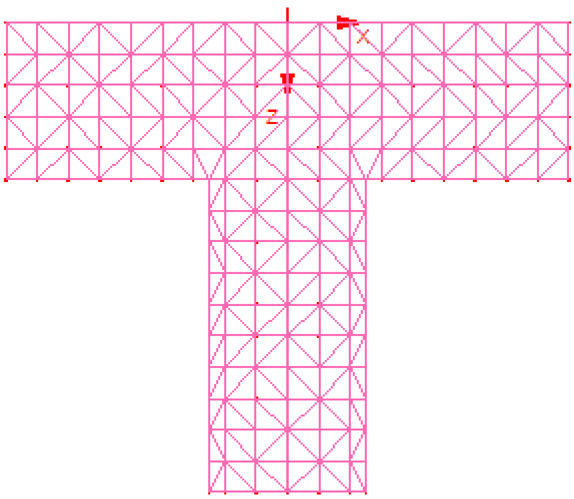
- Fire exposure from hydrocarbons fire

$$\Theta_g = 1\,080\,(1 - 0{,}325\,e^{-0{,}167\,t} - 0{,}675\,e^{-2{,}5\,t}) + 20$$



EXECUTION OF THE MESH

The program automatically performs the subdivision into elementary triangular surfaces (mesh), which the user can customize, to evaluate the temperatures and properties of the material in each node.



THERMAL CALCULATION

The diffusion of heat occurs in a non-stationary (transient) regime, in which the temperature varies not only as a function of space, but also as a function of time. In this case, accumulation or dissipation phenomena and the related physical properties are also taken into consideration. The differential equation governing the phenomenon is the Fourier equation:

$$\mathrm{div}\ (\lambda_c \cdot \mathrm{grad}\ \theta) + w = C_c \cdot \rho_c \cdot \frac{\delta \theta}{\delta t}$$

with initial condition T (0) = T0 = 20 ° C
and boundary condition

$$\text{div}\,(\lambda_c \cdot \text{grad}\,\theta)_n = h_{net,d}$$

where:

$$\dot{h}_{net} = \dot{h}_{net,c} + \dot{h}_{net,r}$$

where:

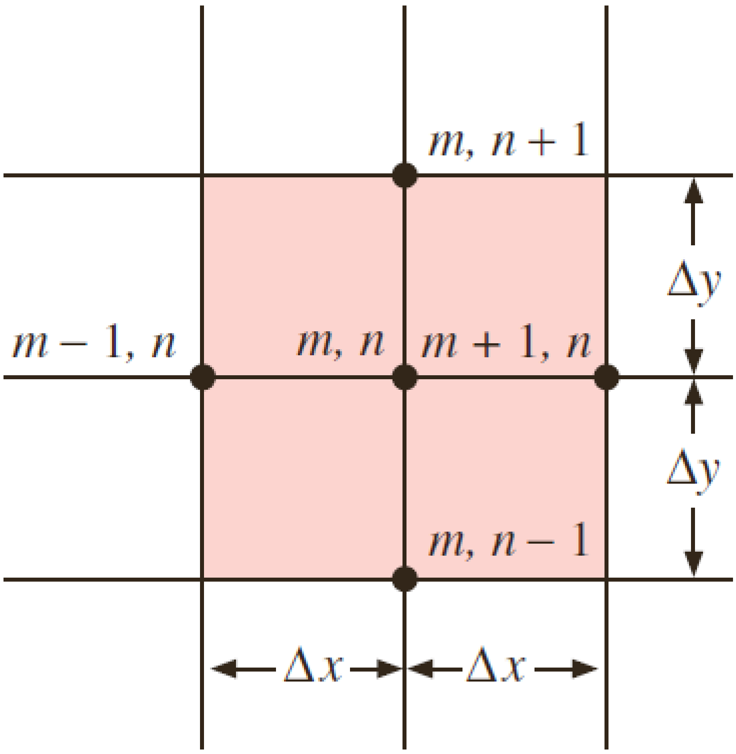
$$\dot{h}_{net,c} = \alpha_c \cdot (\Theta_g - \Theta_m) \qquad [W/m^2] \qquad (3.2)$$

Where:

αc is the coefficient of heat transfer by convection [W/m2K]
Θg is the temperature of the gas in the vicinity of the element exposed to fire [oC]
Θm is the surface temperature of the element [oC]

hnet,r MISSING

Given the non-linearity of the differential equation, a numerical resolution is used, referring to the explicit Euler method:



$$\left.\frac{\partial T}{\partial x}\right]_{m+1/2,n} \approx \frac{T_{m+1,n} - T_{m,n}}{\Delta x}$$

$$\left.\frac{\partial T}{\partial x}\right]_{m-1/2,n} \approx \frac{T_{m,n} - T_{m-1,n}}{\Delta x}$$

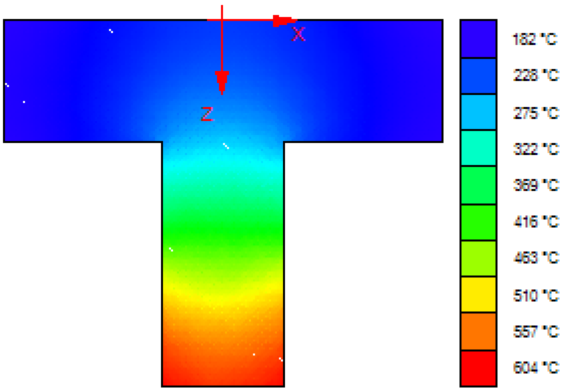
$$\left.\frac{\partial T}{\partial y}\right]_{m,n+1/2} \approx \frac{T_{m,n+1} - T_{m,n}}{\Delta y}$$

$$\left.\frac{\partial T}{\partial y}\right]_{m,n-1/2} \approx \frac{T_{m,n} - T_{m,n-1}}{\Delta y}$$

$$\left.\frac{\partial^2 T}{\partial x^2}\right]_{m,n} \approx \frac{\left.\frac{\partial T}{\partial x}\right]_{m+1/2,n} - \left.\frac{\partial T}{\partial x}\right]_{m-1/2,n}}{\Delta x} = \frac{T_{m+1,n} + T_{m-1,n} - 2T_{m,n}}{(\Delta x)^2}$$

$$\left.\frac{\partial^2 T}{\partial y^2}\right]_{m,n} \approx \frac{\left.\frac{\partial T}{\partial y}\right]_{m,n+1/2} - \left.\frac{\partial T}{\partial y}\right]_{m,n-1/2}}{\Delta y} = \frac{T_{m,n+1} + T_{m,n-1} - 2T_{m,n}}{(\Delta y)^2}$$

CREATION OF THERMAL MAPPING
The thermal mapping of the reinforced concrete section is determined.



EVALUATION OF NEW CHARACTERISTICS IN MATERIAL FIRE CONDITIONS
The decay of the mechanical characteristics of the materials is evaluated as a function of the temperatures obtained in each node of the mesh.
The results take into account the characteristics of materials and reinforcements (strength of concrete, strength of steel, arrangement and quantity of reinforcements, etc.).

Temperature of the concrete θ	Siliceous aggregates			Calcareous aggregates		
	$f_{c,\theta}/f_{ck}$	$\varepsilon_{c1,\theta}$	$\varepsilon_{cu1,\theta}$	$f_{c,\theta}/f_{ck}$	$\varepsilon_{c1,\theta}$	$\varepsilon_{cu1,\theta}$
[°C]	[-]	[-]	[-]	[-]	[-]	[-]
1	2	3	4	5	6	7
20	1,00	0,0025	0,0200	1,00	0,0025	0,0200
100	1,00	0,0040	0,0225	1,00	0,0040	0,0225
200	0,95	0,0055	0,0250	0,97	0,0055	0,0250
300	0,85	0,0070	0,0275	0,91	0,0070	0,0275
400	0,75	0,0100	0,0300	0,85	0,0100	0,0300
500	0,60	0,0150	0,0325	0,74	0,0150	0,0325
600	0,45	0,0250	0,0350	0,60	0,0250	0,0350
700	0,30	0,0250	0,0375	0,43	0,0250	0,0375
800	0,15	0,0250	0,0400	0,27	0,0250	0,0400
900	0,08	0,0250	0,0425	0,15	0,0250	0,0425
1 000	0,04	0,0250	0,0450	0,06	0,0250	0,0450
1 100	0,01	0,0250	0,0475	0,02	0,0250	0,0475
1 200	0,00	-	-	0,00	-	-

Class N, values for the stress-strain relationship parameters of hot-rolled and cold-drawn reinforcement steel at elevated temperatures

prospetto 3.2a

Steel temperature θ [°C]	$f_{sy,\theta}/f_{yk}$		$f_{sp,\theta}/f_{yk}$		$E_{s,\theta}/E_s$	
	hot rolled	cold rolled	hot rolled	cold rolled	hot rolled	cold rolled
1	2	3	4	5	6	7
20	1,00	1,00	1,00	1,00	1,00	1,00
100	1,00	1,00	1,00	0,96	1,00	1,00
200	1,00	1,00	0,81	0,92	0,90	0,87
300	1,00	1,00	0,61	0,81	0,80	0,72
400	1,00	0,94	0,42	0,63	0,70	0,56
500	0,78	0,67	0,36	0,44	0,60	0,40
600	0,47	0,40	0,18	0,26	0,31	0,24
700	0,23	0,12	0,07	0,08	0,13	0,08
800	0,11	0,11	0,05	0,06	0,09	0,06
900	0,06	0,08	0,04	0,05	0,07	0,05
1 000	0,04	0,05	0,02	0,03	0,04	0,03
1 100	0,02	0,03	0,01	0,02	0,02	0,02
1 200	0,00	0,00	0,00	0,00	0,00	0,00

11.6.4.2.2.2 R.C. verification

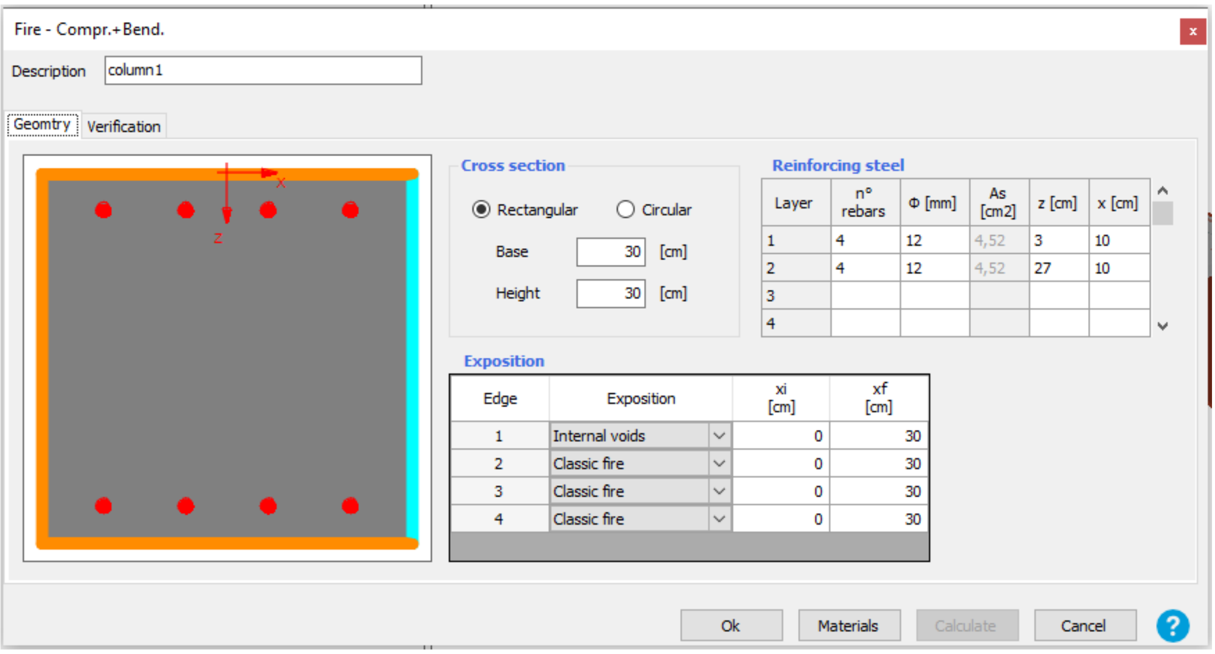
Performs the ULS verification for deflected bending according to EN1992-1-2, in fire conditions, calculating the resisting moment in the main directions, explaining the safety coefficients relating to each combination included in the table.

Forces

Combination	Nd [daN]	Mdx [daNcm]	Mdz [daNcm]	Mrx [daNcm]	Mrz [daNcm]	Verified	Saf. fact.
1	1.000,00	50.000	20.000	237.404	222.678	Yes	4,75
2	-50.000,00	-100.000	20.000	15.407	222.678	No	-∞

11.6.4.2.3 Work environment

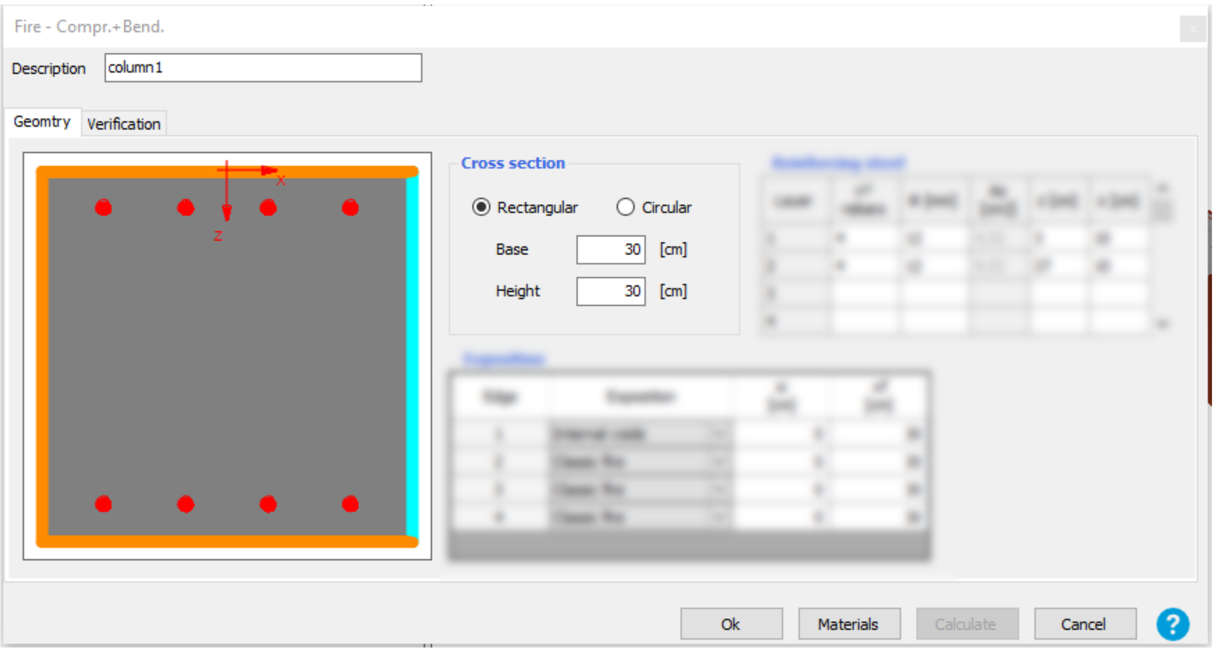
The working environment is as follows:

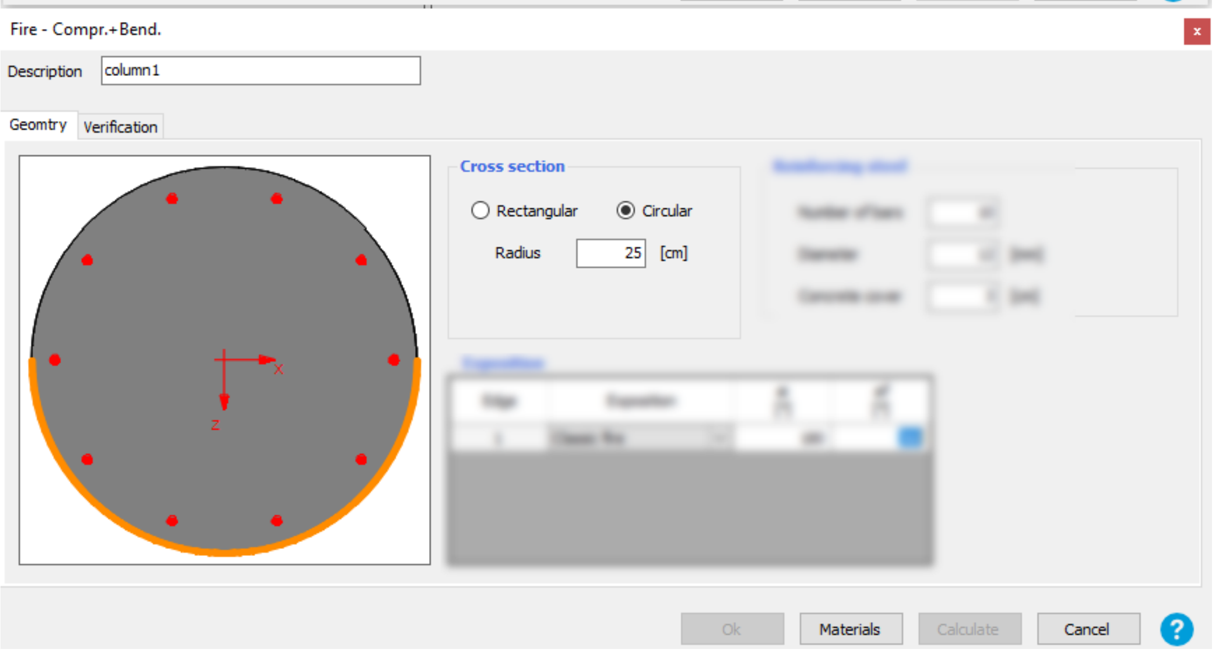


The latter is divided into three sections:
- Section geometry;
- Materials;
- Verification.

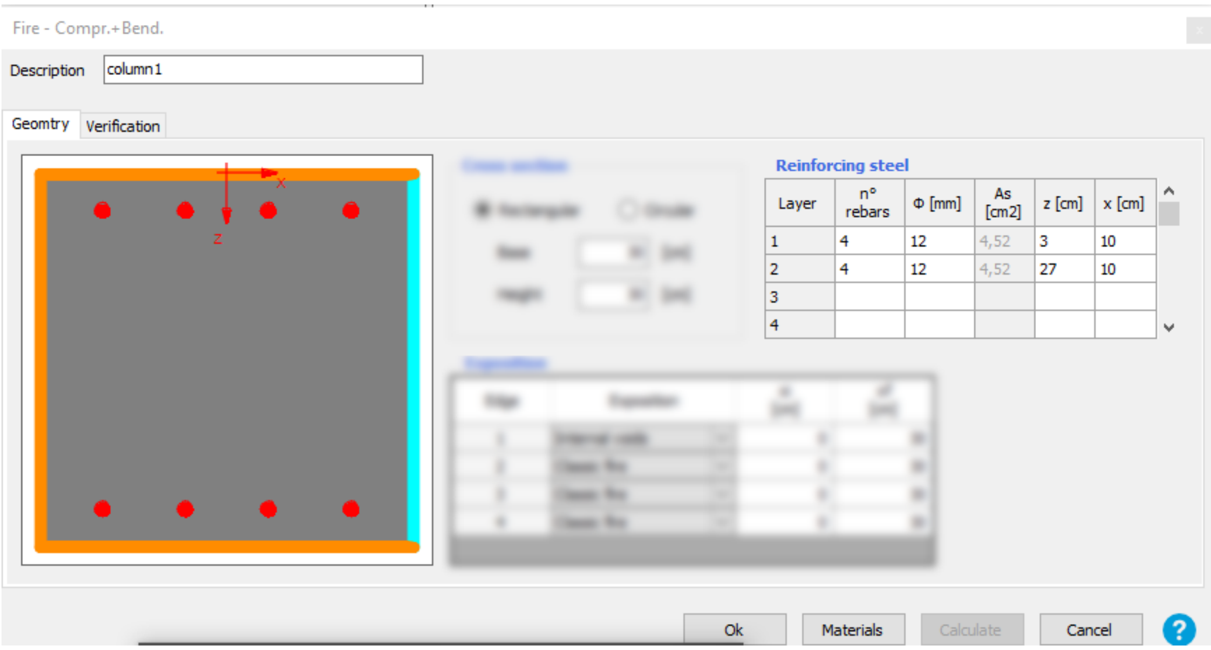
11.6.4.2.3.1 Section geometry

In the Section Definition Area you define the geometry, reinforcement and exposure of the section to be checked.
The geometry is defined as a rectangle or circle.

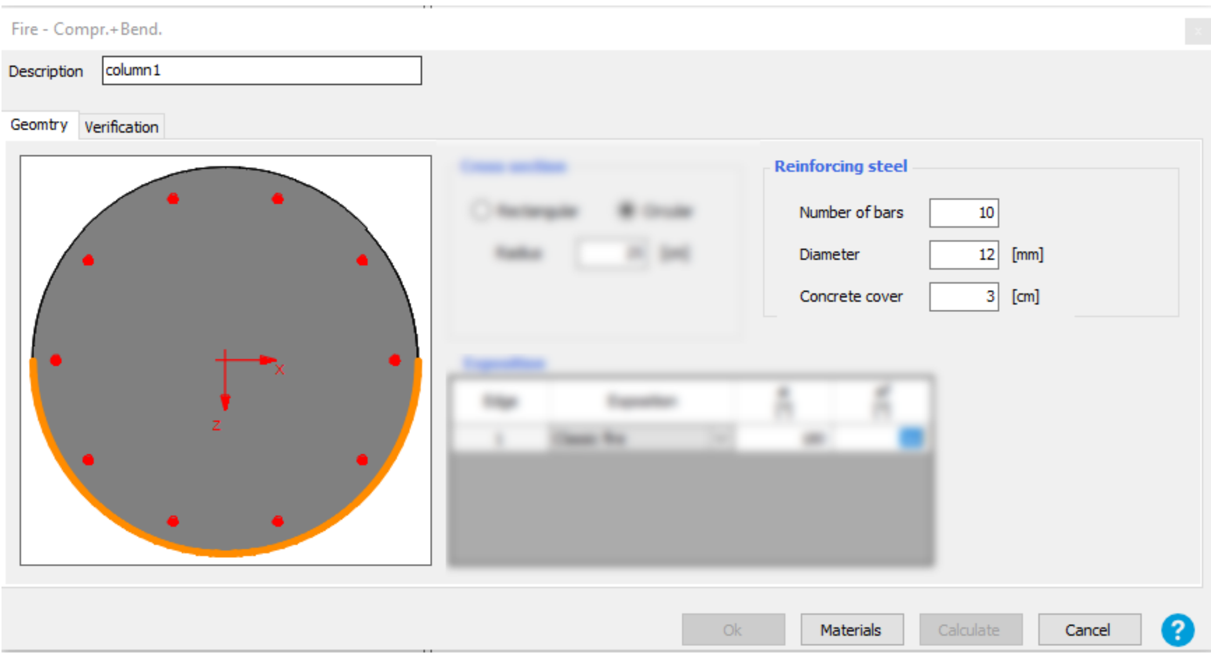




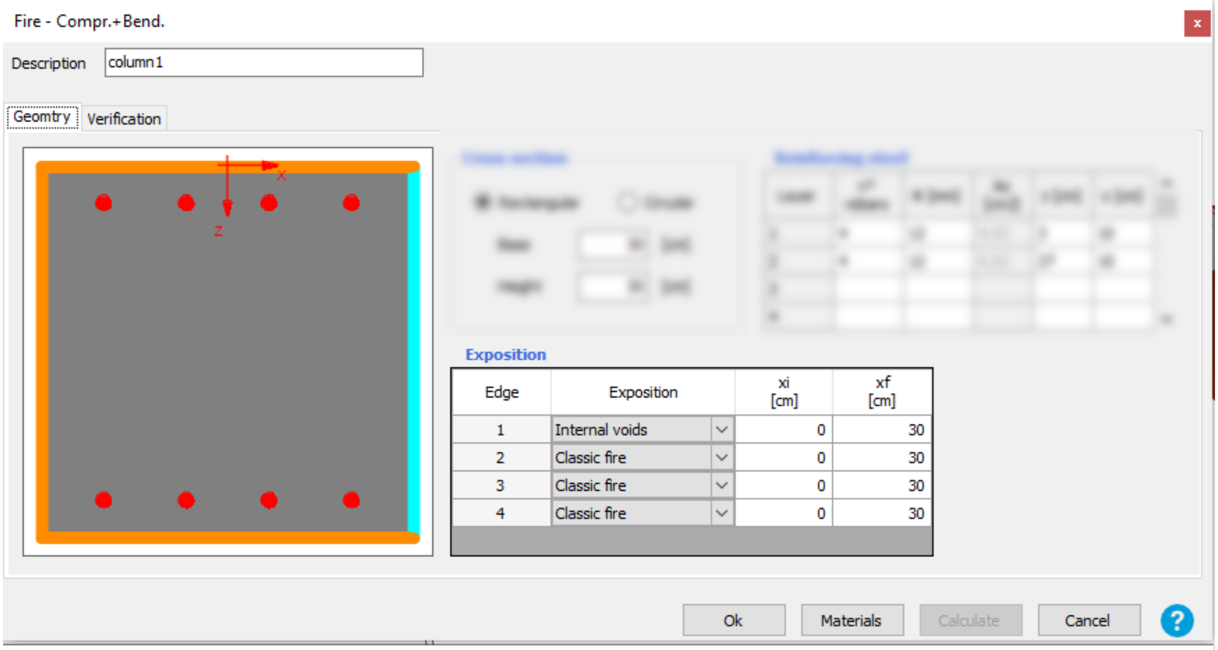
For the rectangular section, the reinforcement is defined by indicating the number of bars, the diameter and the position, with respect to the reference system indicated, of each layer. The x indicates the distance from the axis of symmetry of the furthest iron in the series, the program will arrange the internal bars at a constant pitch. In the event that the layer has a single reinforcing bar, this will necessarily be fixed on the axis of symmetry. Once the number and diameter have been assigned, the program will automatically fill in the cell with the reinforcement area.

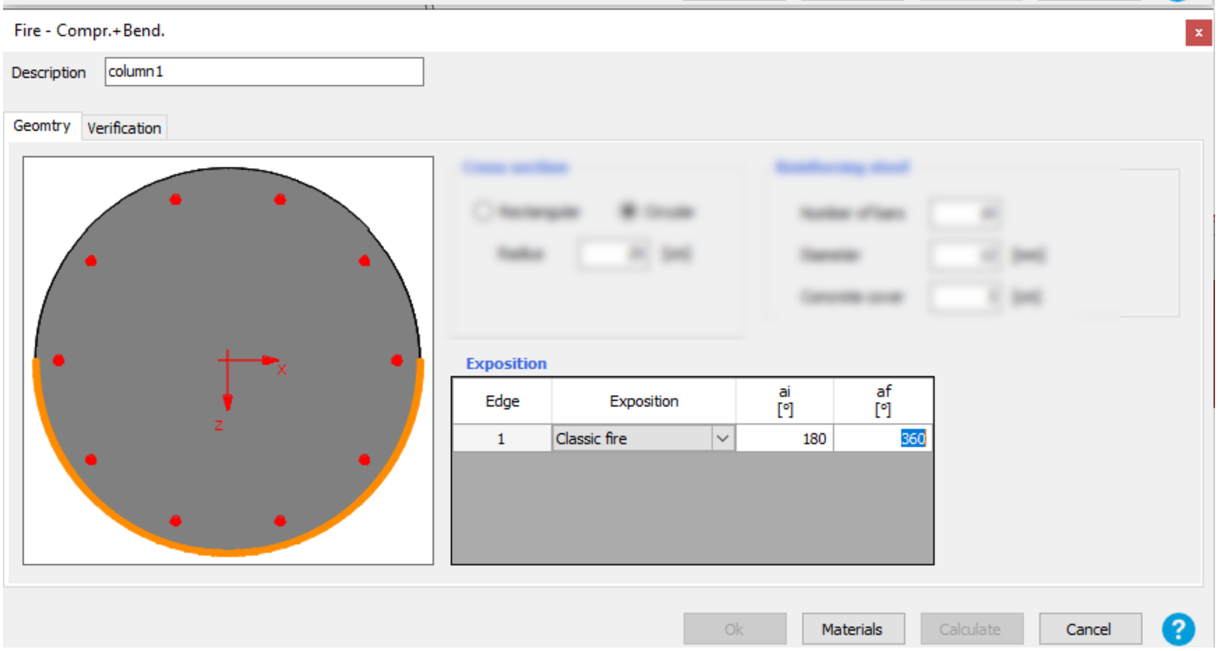


For the circular section the reinforcement is defined by indicating the diameter, number and concrete cover.



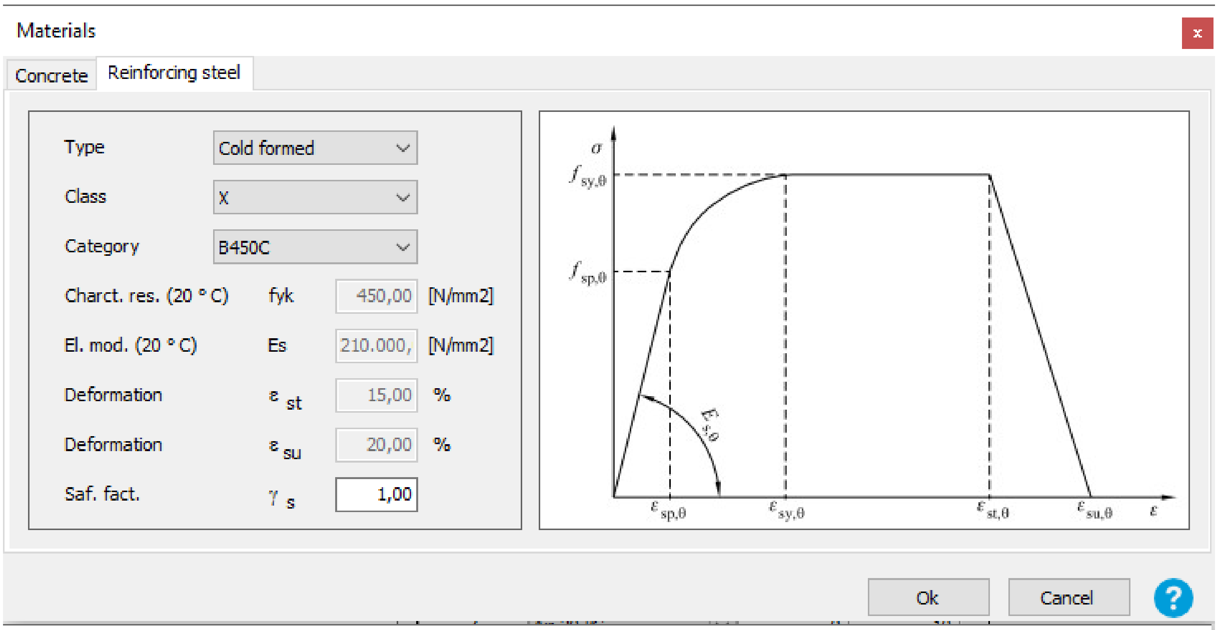
The exposure is defined on each side according to the local reference system. It is possible to define a partial exposure on each side by modifying the parameters xi, starting point, and xf, ending point, with respect to the local reference system for the rectangular column and the parameters ai, starting angle, and af, ending angle , relative to the x axis for the circular column.
When the exposure table is active, the numbering of the sides and the reference systems are displayed in the image.

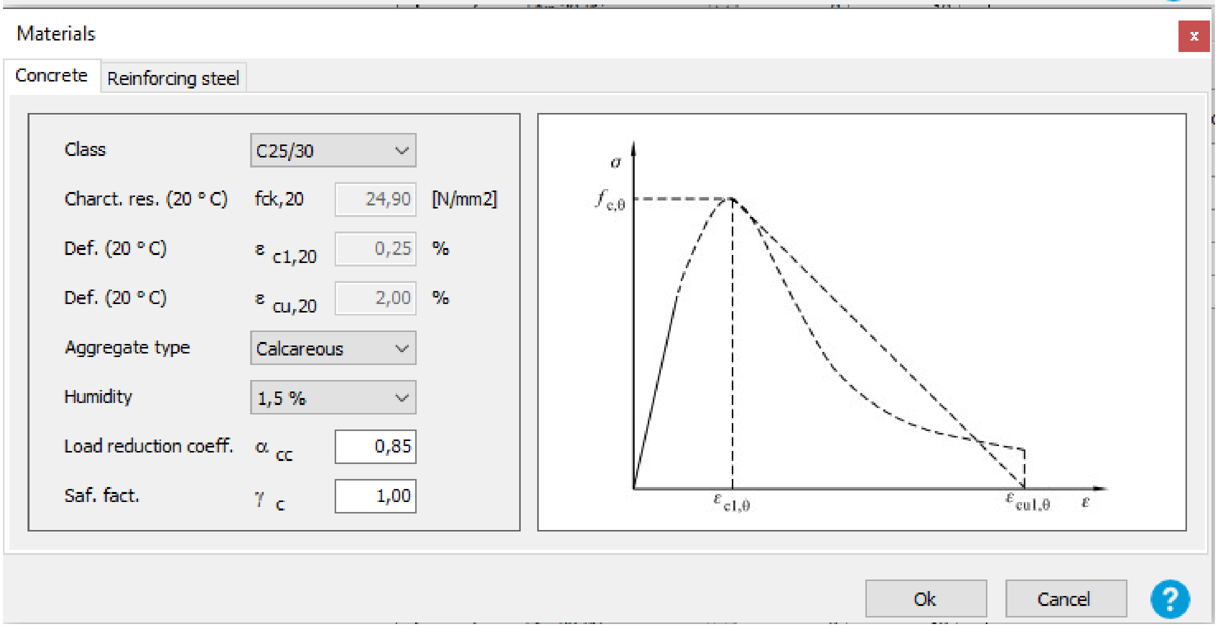




11.6.4.2.3.2 Materials

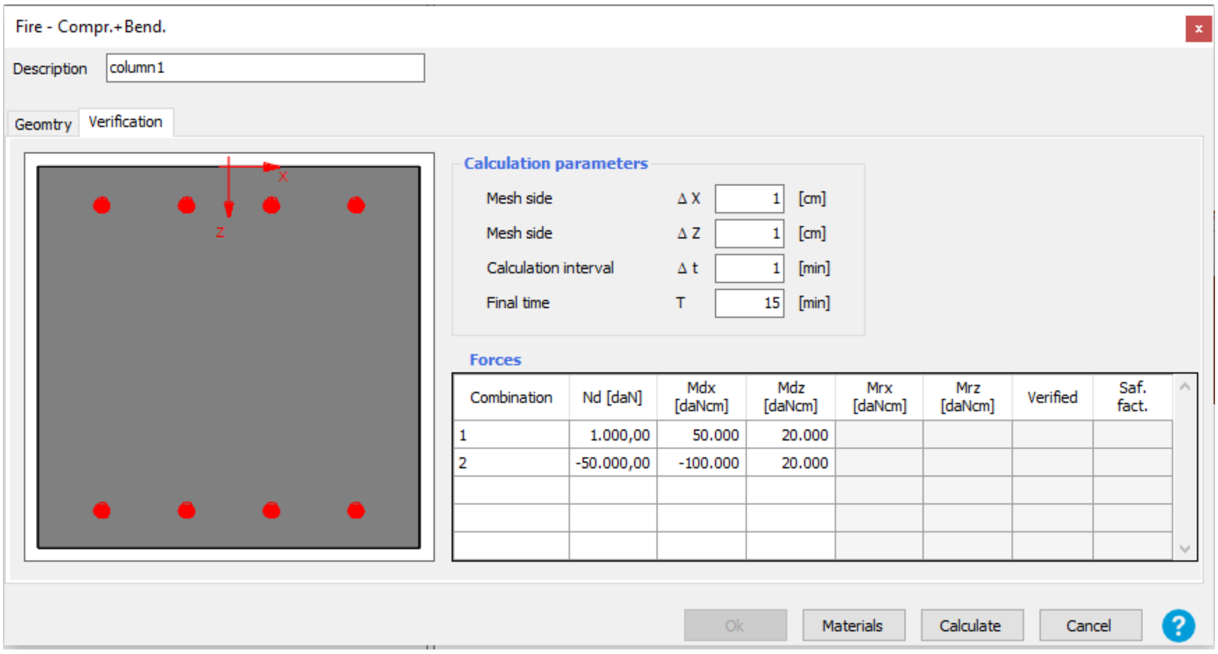
It is possible to access the Materials Area by clicking on the appropriate button.
In this area it is possible to set the characteristics of concrete and steel.



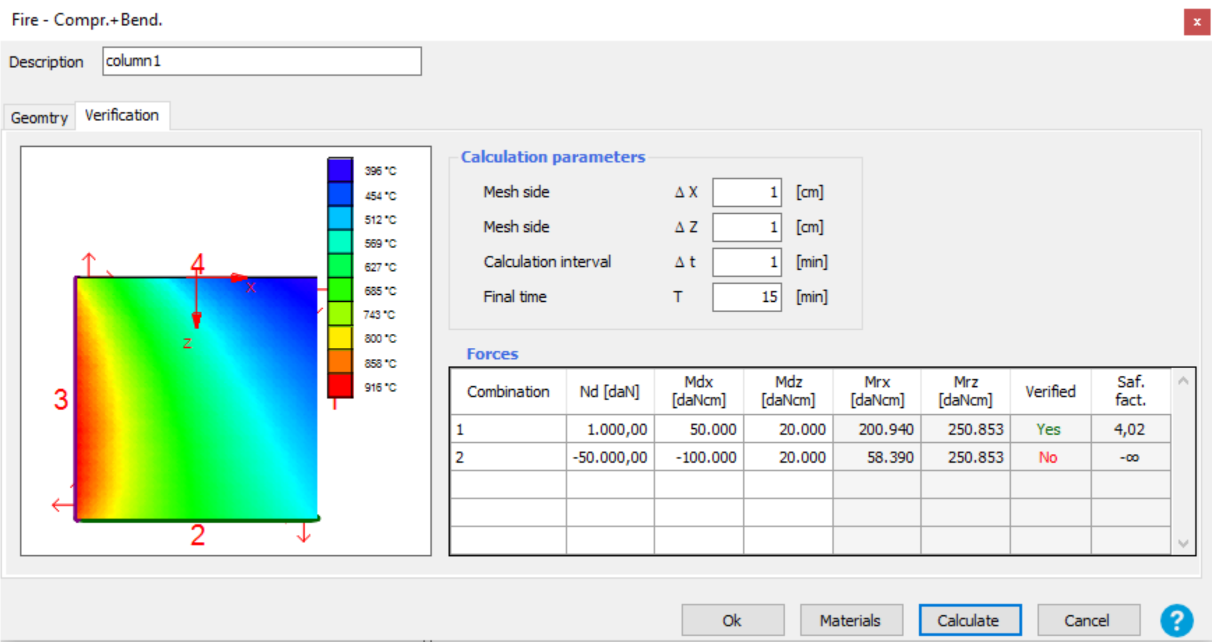


11.6.4.2.3.3 Verification

In the Verification Area, the calculation parameters are defined, such as:
- dimensions of the DX and DZ computational mesh
- calculation time interval Dt
- final calculation time T



With the Calculate button, the application calculates the resisting moment in the main directions of the section and checks the combinations entered in the table according to EN1992-1-2, providing the relative safety coefficient.



11.6.4.2.4 Bibliography

[1] EC1991-1-2: Eurocode 1 - Actions on structures - Part 1-2: Actions in general - Actions on structures exposed to fire
[2] EC1992-1-2: Eurocode 2 - Design of concrete structures - Part 1-2: General rules - Structural fire design
[3] Ministerial Decree January 17, 2018 - Approval of the new technical standards for construction
[3] Circular 21 January 2019, n. 7 - Instructions for the application of the "Technical standards for constructions"
[4] J.Taler, P. Duda - Solving Direct and Inverse Heat Conduction Problems - Springer

[5] J. A. Purkiss - Fire Safety engineering - Design of structures - BH

11.6.5 RC Beam, Steel, Timber

11.6.5.1 Introduction

The Beam module performs stress calculation and verification of reinforced concrete, timber and steel beams. The stresses are obtained from the combinations provided for by the regulations of the loads inserted.

In particular, combinations made in accordance with the Ministerial Decree of 17 January 2018 (NTC), with the Eurocode or customized are envisaged.

It is possible to define the constraint conditions, apply distributed and concentrated loads, distinguishing permanent, variable and seismic.

11.6.5.2 Calculation procedure

The procedure differs according to the type of calculation selected. It is indeed possible to carry out

- Calculation of stresses;
- Verification of reinforced concrete beam;
- Verification steel beam;
- Verification wooden beam;

For each type of element tested, the safety coefficient for vertical overloads ζv, i is calculated, defined in paragraph C8.3 of the explanatory circular of NTC18 as the ratio between the maximum value of the variable vertical overload that can be tolerated by the i-th part of the construction and the value of the variable vertical overload that would be used in the design of a new construction.

11.6.5.2.1 Loads calculation

The load calculation returns the moment and shear envelopes in the fundamental and seismic combinations, resulting from the inserted loads.

It is possible to insert loads of the following type:
- distributed constant or linearly variable, possibly acting on only a portion of the beam;
- concentrate along the beam;
- couples applied to the nodes.

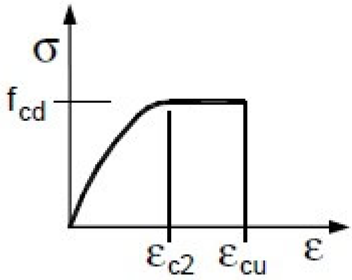
These loads can be distinguished between permanent, variable and seismic.
All inserted variable loads are considered to be simultaneous.

11.6.5.2.2 Verification of RC beam

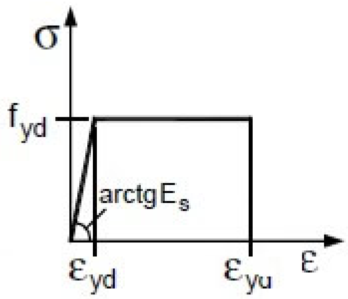
The calculation of the strengths of the reinforced concrete beam is carried out as indicated in the Ministerial Decree 17-01-2018.

In particular, the following constitutive bonds are adopted:

- rectangle parabola for concrete



- elastic - perfectly plastic for steel



The resisting moment is obtained by solving the equation:

$$0 = \psi \cdot b \cdot x \cdot f_{cd} + A_{s2} \cdot \sigma_{s2} - A_{s1} \cdot f_{yd}$$

where:
- x is the position of the neutral axis;
- f_{cd} is the calculation resistance of the concrete;
- y is a dimensionless coefficient representing the intensity of the concrete compression resultant;
- A_{s1} and A_{s2} are, respectively, the area of tense and compressed iron;
- f_{yd} is the calculation stress of the steel;

The calculation of the resistant shear is possible both in the absence and in the presence of a specific shear reinforcement.
In particular, in the absence of shear reinforcement, it is evaluated according to the arch-tie mechanism in which the following intervene:
- the compressive strength of concrete;
- the meshing effect of the aggregates;
- the pin effect of the longitudinal reinforcement.

In the presence of shear reinforcement, the resistance is evaluated according to the Ritter-Mörsch lattice mechanism, according to which the element being analyzed is schematized as an ideal reticular beam consisting of a compressed and a tense current, completed by compressed connecting rods and rods. The method provides that the inclination of the compressed connecting rods θ is variable within the limits:

$1 \leq \text{cotg}\ \theta \leq 2.5$

The overall strength is given by the minimum between the compressive strength of the compressed connecting rods and the tensile strength of the tie rods.

11.6.5.2.3 Verification of steel beam

The calculation of the strength and deformation of the steel beam is carried out as indicated in the Ministerial Decree 17-01-2018.

The program allows the verification of sections of classes 1, 2 and 3, as defined in paragraph 4.2.3.1. The classification is done automatically by the program itself.
The check for class 1 and 2 sections is performed by default in the plastic field, but it is possible to set it in the elastic field. Class 3 sections, on the other hand, cannot be calculated in the plastic field.

In the case of a section stressed only in shear, a simple shear check is performed

$$V_{c,Rd} = \frac{A_v \cdot f_{yk}}{\sqrt{3} \cdot \gamma_{M0}}$$

where:

- Av is the shear resistant area calculated according to the expressions reported in paragraph 4.2.4.1.2 of the Ministerial Decree

In the calculation of the resisting moment the reduced yield stress (1-r) f_{yk} is considered where r

$$\rho = \left[\frac{2V_{Ed}}{V_{c,Rd}} - 1\right]^2 \qquad (4.2.32)$$

It can be limited to the elastic range only or extended to the plastic range for class 1 or 2 sections.
The limb check is subordinated to the simple shear check.
The resistant moment is evaluated as:

$$M_{V.\ Rd} = \min\,(M_v, M_{rd})$$

where:

- Mv is the reduced resisting moment for
  [Wpl -1/(4 nw tw) (r Av2+ NEd2/(1- r)/ fy2)] fd

11.6.5.2.4 Verification of wood beam

The calculation of the strength and deformation of the wooden beam is carried out as indicated in the Ministerial Decree 17-01-2018.
The internal stresses are calculated in the hypothesis of conservation of the plane sections and of a linear relationship between stresses and deformations up to failure.
The calculation values of a material property is calculated using the relationship:

$$X_d = \frac{k_{mod} X_k}{\gamma_M}$$

where:

- X_k is the characteristic value of the property of the material;
- γ_m is the relative partial safety factor;
- k_{mod} is a correction coefficient that takes into account the effect of both the duration of the load and the humidity of the structure.

The characteristic value $f_{m,k}$ of bending strength can be increased with the coefficient k_h

respectively equal to:

- for solid timber sections where the largest cross-sectional dimension is less than 150mm

$$k_h = \min\left\{\left(\frac{150}{h}\right)^{0,2} ; 1,3\right\}$$

- for glulam sections where the largest dimension of the cross section is less than 600 mm

$$k_h = \min\left\{\left(\frac{600}{h}\right)^{0,1} ; 1,1\right\}$$

For the bending check the condition must be satisfied:

$$\frac{\sigma_{m,z,d}}{f_{m,z,d}} \leq 1$$

where:
- $\sigma_{m,d}$ is the calculation stress deriving from the combination under consideration.

The resistant shear is evaluated with Jourawski's theory.

Since the calculation resistances are variable with the duration of the applied load (due to the contribution of the k_{mod} coefficient),
the program performs the ULS checks against the seismic combination (verification of the instantaneous load) and fundamental, the
latter in the presence of variable loads and at the same time (verification of the load of the duration specified by the user) and in the
presence of only the permanent ones (verification of the permanent load).
The deformation is calculated in instantaneous and deferred conditions.
The instantaneous deformation is calculated using the average values of the elastic modulus of the beam in the rare load
combination.

The final deformation, sum of the instantaneous and the deferred deformation, is calculated with the simplified approach:

$$u_{fin} = u_{in} + u_{dif}$$

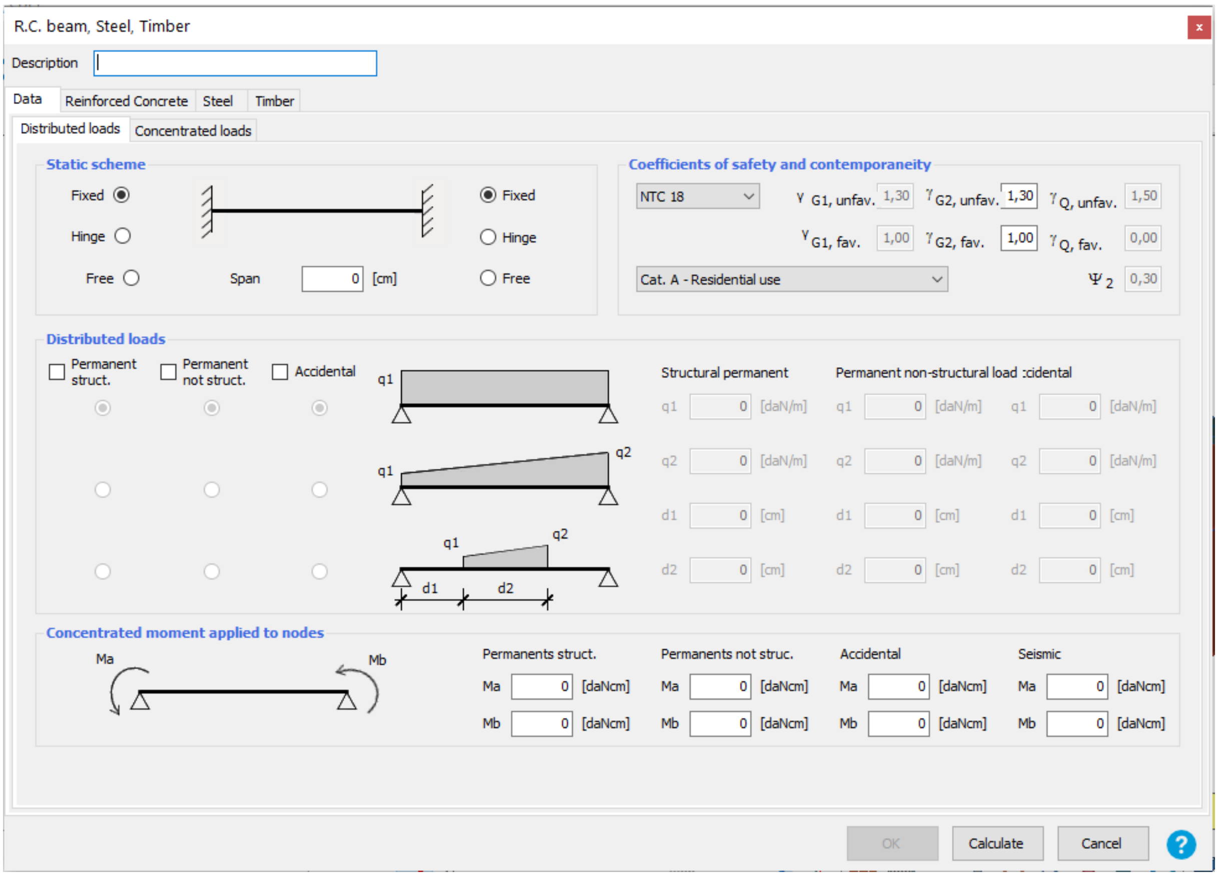
where:
- u_{in} is the initial instantaneous deformation
- u_{dif} is the deferred deformation, calculated as:

$$u_{dif} = u'_{in}\, k_{def}$$

where:
- u'_{in} is the instantaneous deformation calculated with reference to the quasi-permanent load combination
- k_{def} is a coefficient that takes into account the rheological behavior of the wood.

11.6.5.3 Work environment

The main working environment is the following:



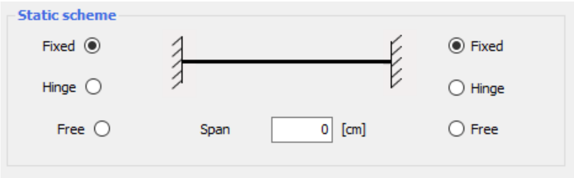
The main work area is divided into 4 tabs:

1. Data
2. Reinforced concrete
3. Steel
4. Wood

11.6.5.3.1 Data tab

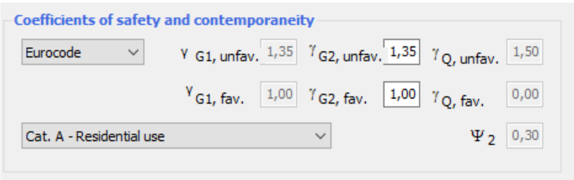
The Data tab is divided into five sections, respectively relating to static diagram, safety coefficients and load combination, distributed loads, concentrated loads and moments applied to the nodes.

This tab is divided into two tabs, respectively dedicated to the assignment of distributed and concentrated loads.



Static scheme

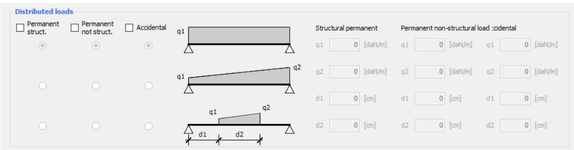
This section must indicate the static diagram and the calculation span of the beam. It is not possible to insert a labile static scheme.



Safety and load combination coefficients

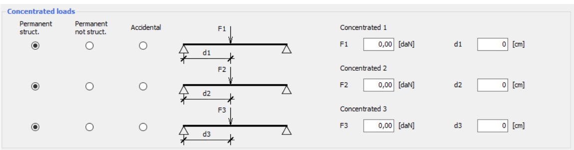
It is possible to select among the coefficients proposed by the legislation or to insert personal values.

Since the variable loads are considered to be all simultaneous, only a value of y2 is used for the calculation of the seismic combination.



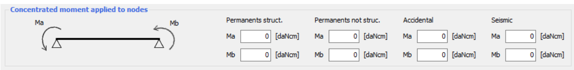
Distributed loads

It is possible to define a permanent distributed load and a variable one, constant or linearly variable and acting on the entire length of the beam or a portion of it.



Concentrated loads

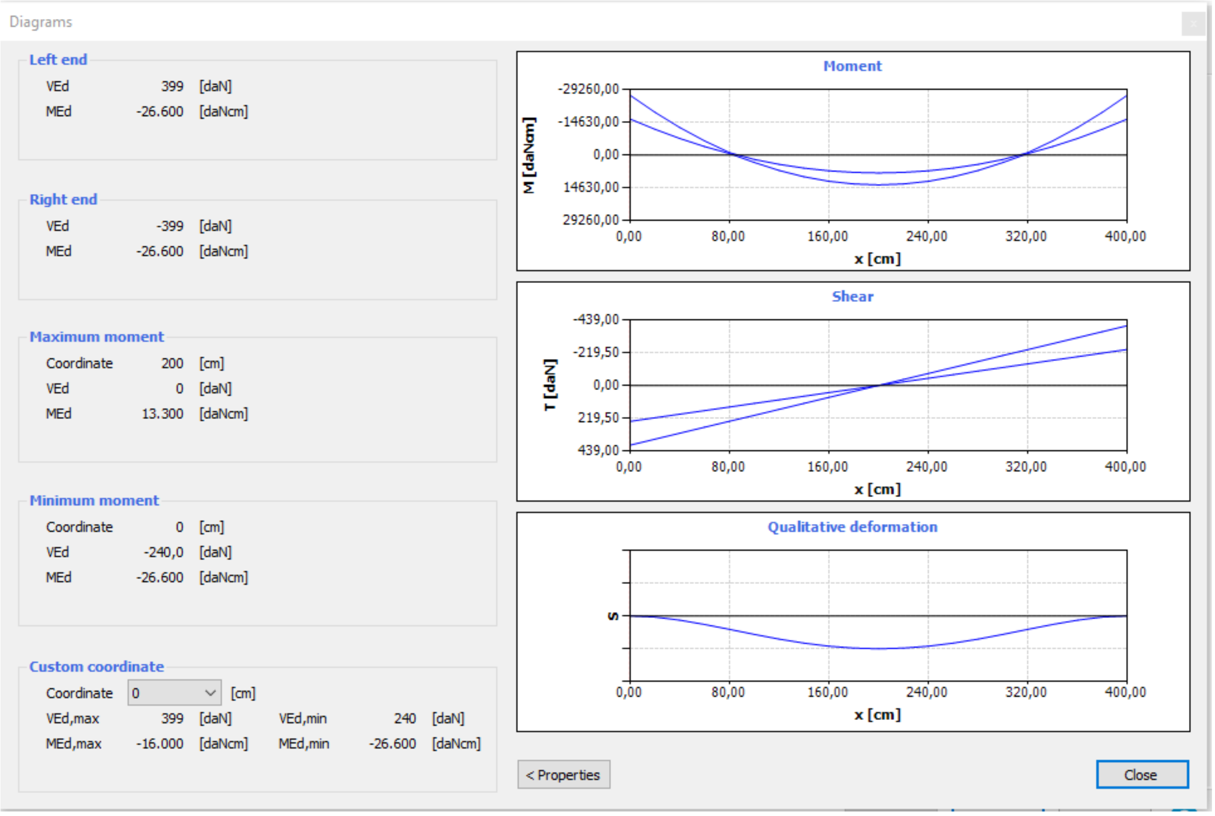
It is possible to define up to 3 concentrated loads, whether they are permanent or variable.



Concentrated moments applied to nodes

It is possible to define moments at the ends of the beam, of permanent, variable or seismic type.

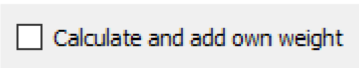
By performing the calculation in the Data tab, the program calculates the envelope of the moment and shear stresses in the fundamental and seismic combination and the qualitative elastic deformation.



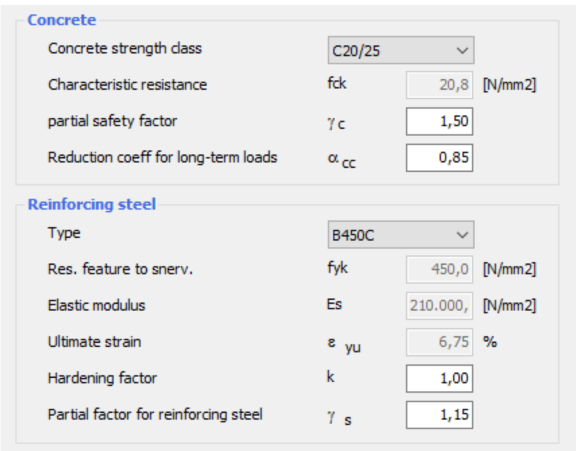
11.6.5.3.2 Reinforced concrete tab

The Reinforced Concrete tab allows the verification of a reinforced concrete beam. with the loads indicated in the Data tab.

It is possible to have the program calculate the self-weight of the defined section and add it to the loads by ticking the appropriate option.



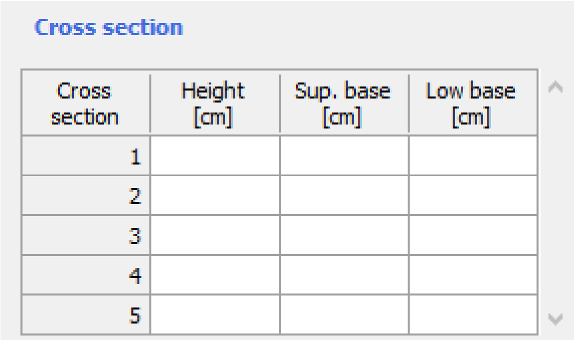
The tab is divided into several parts:



Characteristics of materials

In this section it is possible to define the characteristics and the partial safety coefficients of the materials used

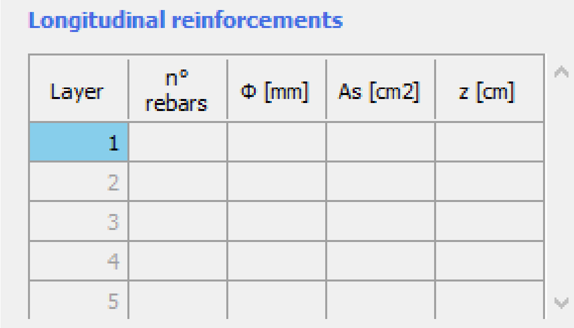
Section geometry



The geometry of the section under analysis must be entered as a succession of sections; of each it is necessary to indicate the height, the width of the upper side and the width of the lower side.

By completing the grid lines, the section diagram is automatically updated.

Longitudinal reinforcements

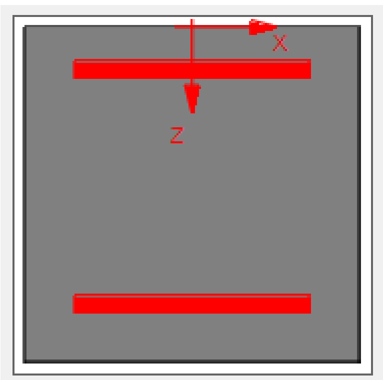


The longitudinal reinforcement of the section under analysis must be inserted by series, indicating the number of bars, the diameter and the position, with respect to the reference system indicated, of each layer. Once the number and diameter have been assigned, the program will automatically fill in the cell with the reinforcement area. The latter will be drawn on the screen in order to verify the input.

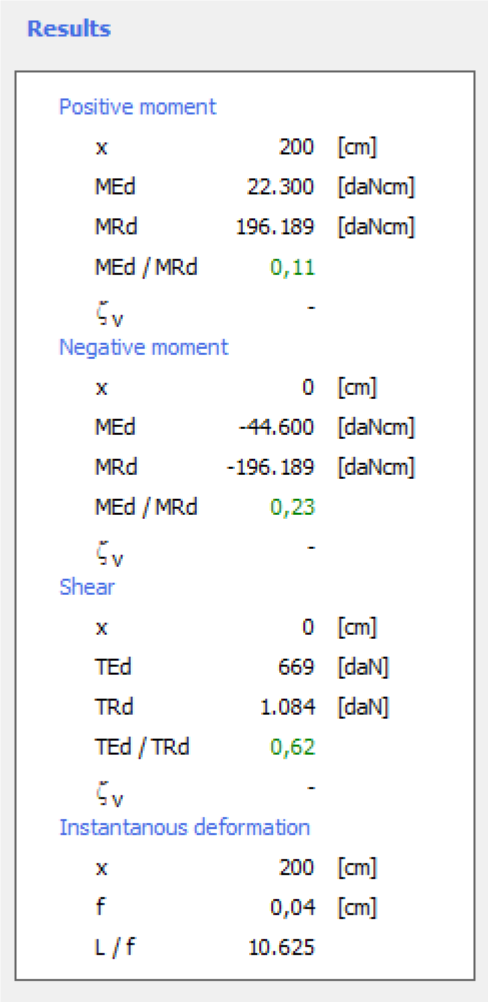
By completing the grid lines, the section diagram is automatically updated.

Transverse reinforcements

Section diagram



Automatically updated when data is entered in the Section geometry and Longitudinal reinforcement grids, it allows you to quickly check the data entered.



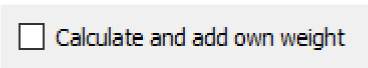
Results

This section shows the main results of the analysis:

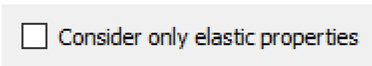
- Positive moment verification;
- Negative time check;
- Verification by shear;
- Maximum deformation.

11.6.5.3.3 Steel tab

The Steel tab allows the verification of a steel beam with the loads indicated in the Data tab.
It is possible to have the program calculate the self-weight of the defined section and add it to the loads by ticking the appropriate option.



It is possible to calculate the resistances limited to the elastic range by ticking the appropriate option.

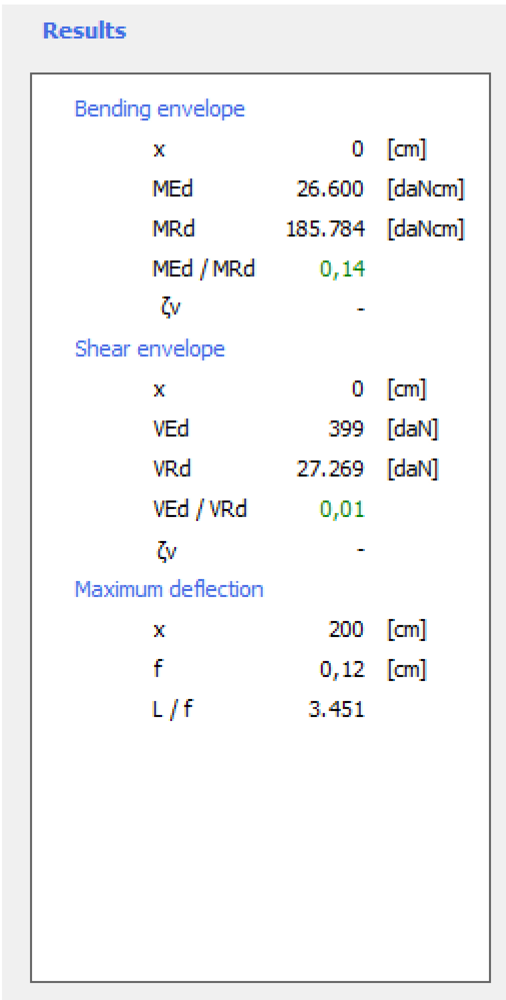


The card can be divided into two parts:

Characteristics of the profile

In this section you can define the characteristics and the partial safety coefficients of the material used and select

the type of profile adopted.
It is not possible to calculate with profiles other than those available in the program library.

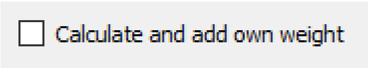


Results

This section shows the results of the moment check in the most stressed sections, for the maximum moment and maximum shear envelopes. The maximum deformation is also indicated.

11.6.5.3.4 Wood tab

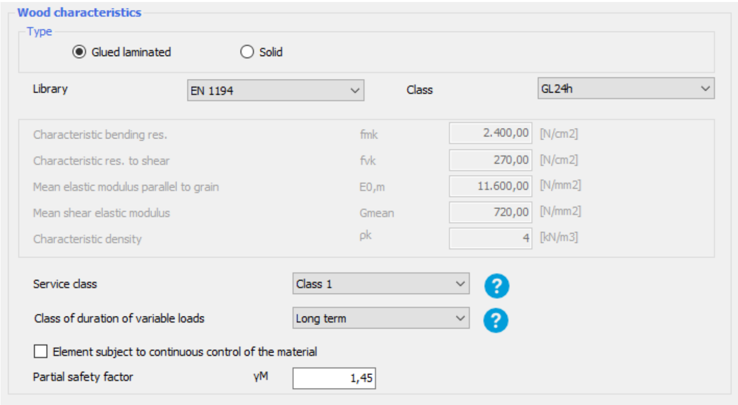
The Wood tab allows the verification of a wooden beam against the loads indicated in the Data tab.
It is possible to have the program calculate the self-weight of the defined section and add it to the loads by ticking the appropriate option.



The card can be divided into three parts:

Cross section definition

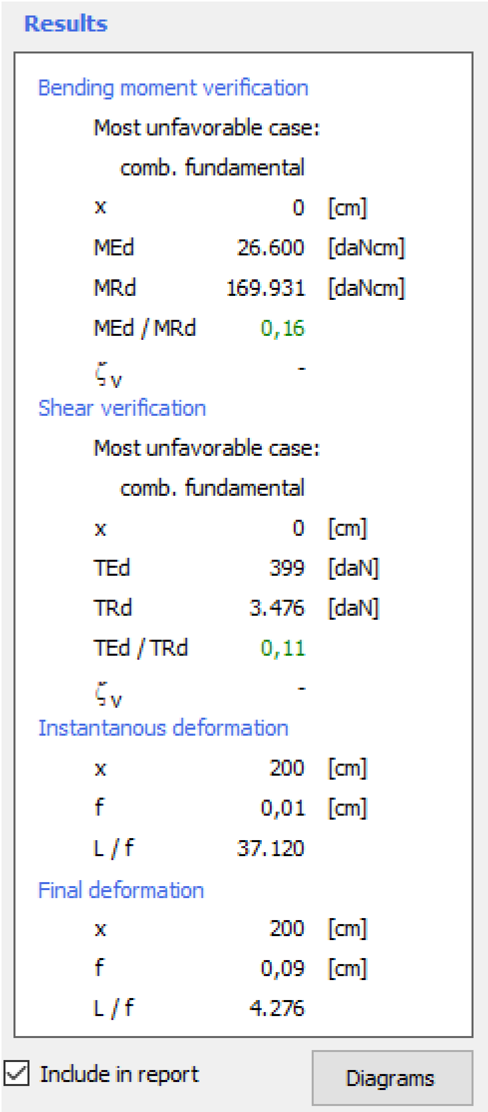
In this section you can define the shape, rectangular or circular, and the dimensions of the section.



Characteristics of the material

In this section it is possible to define the characteristics of the wood used, the partial safety coefficient, the service class and duration of the variable loads.

By clicking on the buttons ? it is possible to view extracts of the reference standard to help in the choice of parameters.



Results

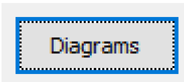
This section shows the main results of the analysis:

- Check at maximum moment;
- Verification by shear;
- For both checks, the case to which it refers is indicated (refer to the calculation procedure for further explanations).

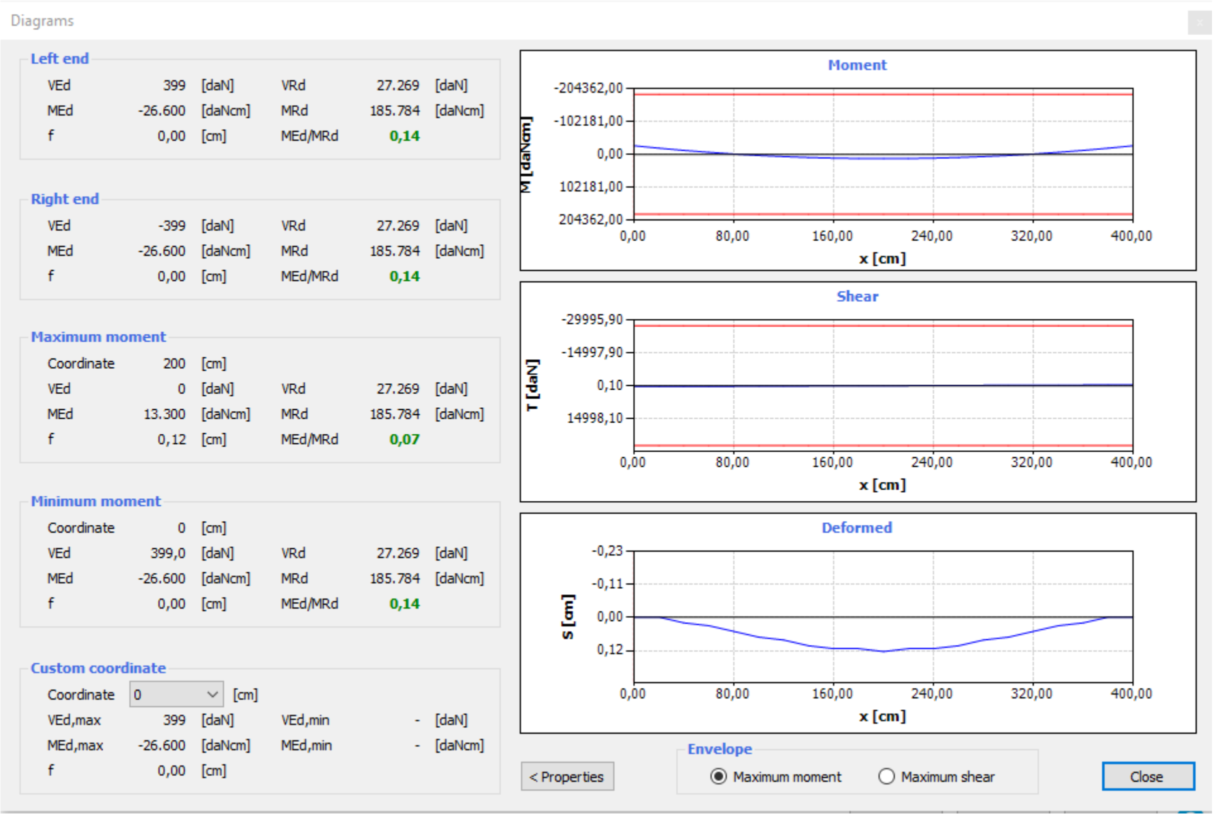
They also report:

- Maximum instantaneous deformation;
- Maximum final deformation.

11.6.5.3.5 Graphics area

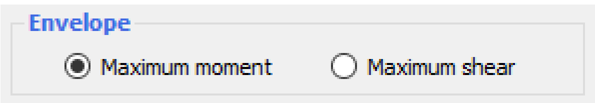
The graphic area is displayed at the end of the calculation performed from the Data tab or when it is accessed via the button 

in the tabs of the different materials.

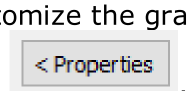


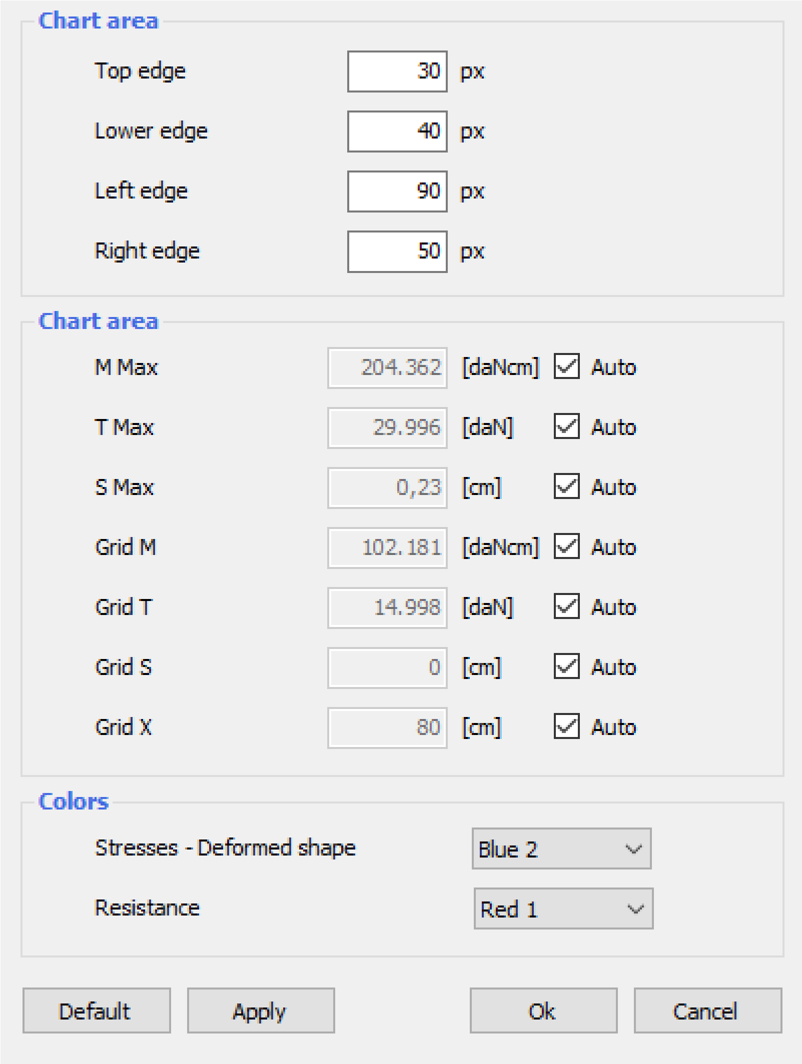
The area shows on the right the diagrams of the stresses and resistances resulting from the analysis, and on the left the respective numerical values.
In the Custom coordinate box it is possible to select query any calculation section, identifying the same also on the graphs.

For the verification of steel beams it is possible to choose whether to display the moment envelope or the maximum shear envelope using the appropriate button.



However, both possibilities will be reported in the report.

If you want to customize the graphs, you can access their properties through the appropriate button  , where you can change the maximum values, grid, margins and colors of the diagrams.



11.6.5.4 Bibliography

[1] Ministerial Decree January 17, 2018 - Approval of the new technical standards for construction
[2] Circular 21 January 2019, n. 7 - Instructions for the application of the "Technical standards for constructions"
[3] EN 1993-1-1 (Eurocode 3)
[4] EN 1998-4: 2006 (Eurocode 8)

11.6.6 Dowel bars

11.6.6.1 Introduction

The Dowel bars module provides the verification of these elements, widely used in structural reinforcement interventions, which have the purpose of connecting two structural parts of the building (for example the floor with the perimeter wall) ensuring the resistance to sliding suitable to absorb the shear stresses that are generated.

The dowel bar retrofit is relatively simple, inexpensive and minimally invasive.

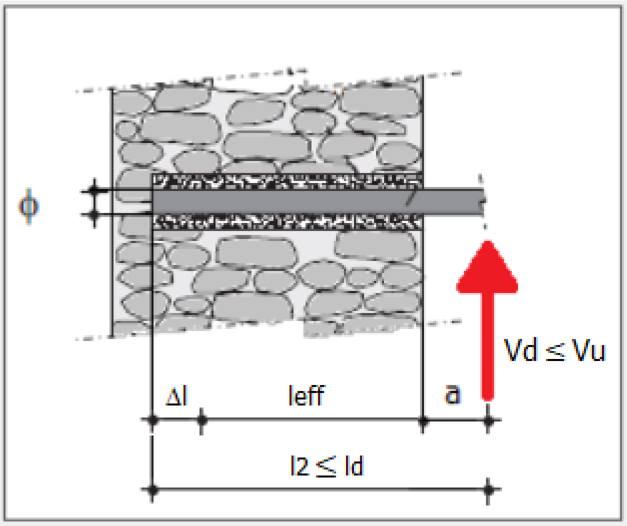
11.6.6.2 Calculation procedure

The dowel bars are steel bars with a circular section, which are fixed into the structural elements to be connected. Known the geometric dimensions of the dowel bar and the stressing shear force perpendicular to its axis, the designer must determine the resistant shear. The latter dimension is a function of the type of constraint of the dowel bar (free end to rotate or constrained end), its diameter, its length, the material of which it is formed and the bearing resistance of the masonry.

Description of the calculation procedure

The designer, after having determined whether the dowel bar head is free to rotate or not (depending on the geometry of the structural elements to be connected), introduces hypotheses on its mechanical behavior (infinitely long dowel bar, short dowel bar, very short dowel bar). On the basis of these assumptions, the shear strength and the effective length are calculated, which must be guaranteed so that the shear strength calculated following the behavior hypothesis can be fully achieved. If the real length of the dowel bar is greater than or equal to the effective length, the behavior hypothesis is correct. Otherwise it is necessary to recalculate the shear strength by changing the behavior hypothesis and then adopting a different formula.

The formulas used to determine the shear strength and effective length of the dowel bar as a function of the type of constraint of the head and of the behavioral hypotheses are illustrated below.



The values used in the calculation are the following:

σrif: bearing stress of the masonry

σrif: yield stress of the dowel bar

ϕ: diameter of the dowel bar

ld: real length of the dowel bar

a: distance between the stressing shear force and the masonry

l_{eff}: effective length of the dowel bar

V_d: shear demand

V_u: ultimate shear strength

CASE 1: End free to rotate

<u>- Hypothesis of infinitely long dowel bar</u>

The effective length is equal to:

$$\ell_{eff} = \phi\left[-\frac{a}{\phi} + \sqrt{\left(\frac{a}{\phi}\right)^2 + \frac{1}{3}\frac{\sigma_y}{\sigma_{rif}}}\right]$$

(1)

The ultimate shear resistance of the dowel bar is calculated as:

$$V_u = \sigma_{rif}\phi^2\left[\sqrt{\left(\frac{a}{\phi}\right)^2 + \frac{1}{3}\frac{\sigma_y}{\sigma_{rif}}} - \frac{a}{\phi}\right]$$

(2)

Please note:

$$\Delta\ell = \phi\sqrt{\frac{2}{3}\frac{\sigma_y}{\sigma_{rif}}}$$

the minimum total length is equal to:

$$\ell_2 = \ell_{eff} + \Delta\ell + a$$

For the long dowel bar hypothesis to be valid, it must be verified that:

$$\ell_d \geq \ell_2$$

Prudentially, in the pre-design phase, for each type of material mentioned above, it can be assumed:

$$\ell_d \geq 8\phi + a.$$

<u>-Short dowel bar hypothesis</u>

In the event that the hypothesis of a long dowel bar is not correct, the effective length is assumed to be equal to:

$$\ell_{eff} = \ell_a \left[\sqrt{2 + 2(a/\ell_a)^2} - (1 + a/\ell_a) \right]$$

(3)

where $l_a = l_d - a$.

Consequently, the ultimate shear strength is:

$$V_u = \sigma_{rif} \phi \ell_{eff} = \sigma_{rif} \phi^2 \left\{ \frac{\ell_a}{\phi} \left[\sqrt{2 \left[1 + \left(\frac{a}{\ell_a} \right)^2 \right]} - (1 + a/\ell_a) \right] \right\}$$

(4)

CASE 2: constrained end

- Hypothesis of infinitely long dowel bar

By introducing this hypothesis, we obtain:

$$\ell_{eff} = \phi \left[\sqrt{\left(\frac{a}{\phi} \right)^2 + \frac{2}{3} \frac{\sigma_y}{\sigma_{rif}}} - \frac{a}{\phi} \right]$$

(5)

$$V_u = \sigma_{rif} \phi^2 \left[\sqrt{\left(\frac{a}{\phi} \right)^2 + \frac{2}{3} \frac{\sigma_y}{\sigma_{rif}}} - \frac{a}{\phi} \right]$$

(6)

$$\ell_2 = \phi \left[\sqrt{\left(\frac{a}{\phi} \right)^2 + \frac{2}{3} \frac{\sigma_y}{\sigma_{rif}}} + \sqrt{\frac{2}{3} \frac{\sigma_y}{\sigma_{rif}}} \right]$$

Similarly to the previous case, in order for the long dowel bar hypothesis to be valid, it must be verified that:

$$\ell_d \geq \ell_2$$

Otherwise, the hypothesis of a long dowel bar is inconsistent and it is necessary to formulate the hypothesis of a short dowel bar.

-Short dowel bar hypothesis

By introducing this hypothesis, we obtain:

$$\ell_{eff} = \ell_a \left[\sqrt{(1+2a/\ell_a)^2 + 1 + \frac{2}{3}\frac{\sigma_y}{\sigma_{rif}}\left(\frac{\phi}{\ell_a}\right)^2} - (1+2a/\ell_a) \right]$$

(7)

$$V_u = \sigma_{rif} \times \phi^2 \left\{ \frac{\ell_a}{\phi} \left[\sqrt{\left(1+\frac{2a}{\ell_a}\right)^2 + \frac{2}{3}\frac{\sigma_y}{\sigma_{rif}}\left(\frac{\phi}{\ell_a}\right)^2 + 1} - \left(1+\frac{2a}{\ell_a}\right) \right] \right\}$$

(8)

where $l_a = l_d - a$.

- Very short dowel bar hypothesis

In the case of a very short dowel bar, the collapse occurs by bearing without forming a plastic hinge, and consequently.

$$V_u = \sigma_{rif}\,\phi(\ell_d - a)$$

(9)

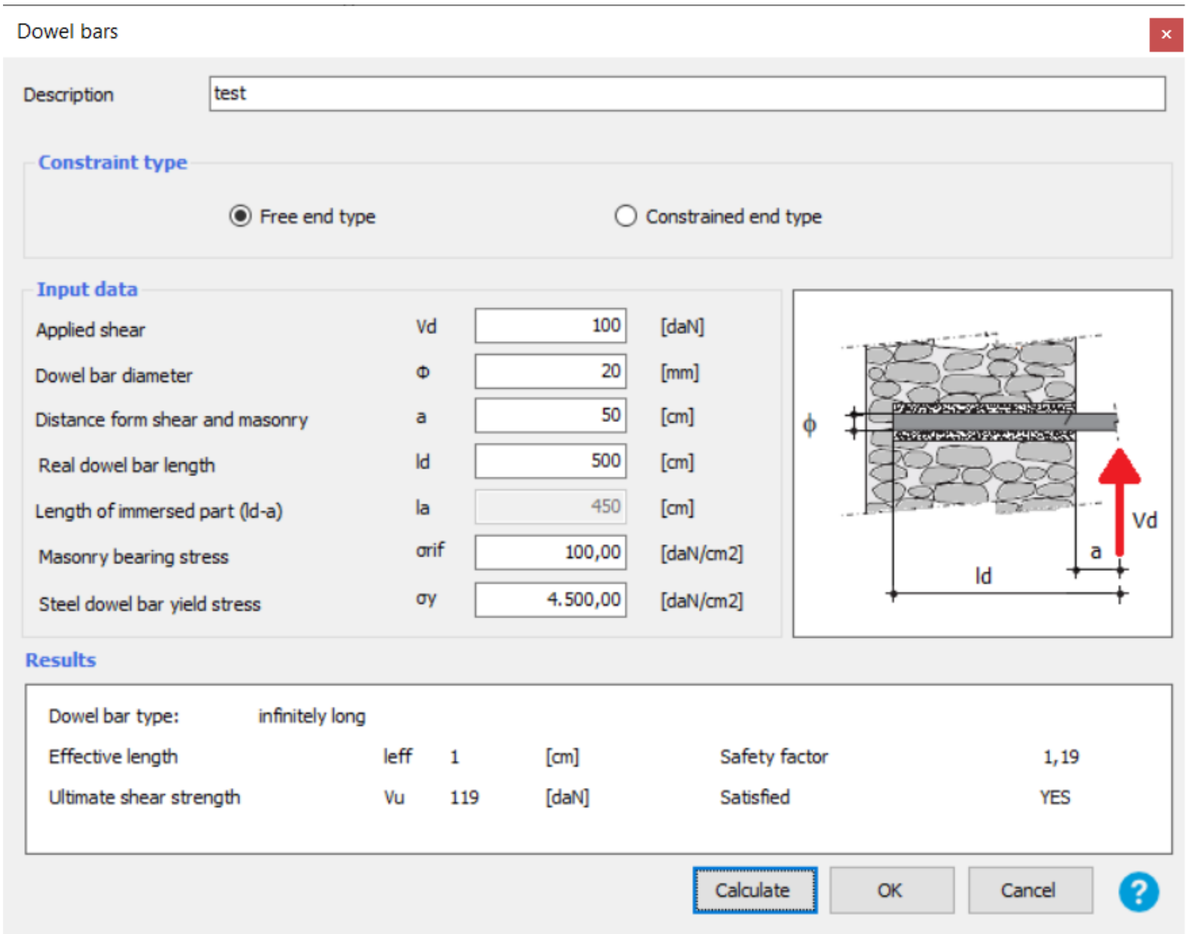
For this hypothesis to be valid, the length l1 defined as:

$$\ell_1 = \phi\sqrt{\left(\frac{a}{\phi}\right)^2 + \frac{1}{3}\frac{\sigma_y}{\sigma_{rif}}}$$

must be greater than or equal to the length ld.

11.6.6.3 Work environment

The work environment is as follows:



The window is divided into three sections:

- the upper part is dedicated to the type of constraint (free end or constrained end);
- input data, where the geometric and mechanical characteristics of the dowel bar and of the masonry are entered;
- calculation results which indicate the type of dowel bar from which the designer can deduce which formulas are used in the calculation, which, with reference to the paragraph "calculation procedure", can be summarized into the following table:

Type of dowel bar	End free to rotate	Constrained end
Infinitely long	Eq. (1) , Eq. (2)	Eq. (5) , Eq. (6)
Short	Eq. (3) , Eq. (4)	Eq. (7) , Eq. (8)
Very short	-	Eq. (9)

In the case of a free to rotate end, the very short dowel bar loses practical interest and is therefore not considered, while in the case of a constrained end the effective length is not defined.

Finally, the effective length, the ultimate shear strength and the safety factor evaluated as Vu / Vd are shown.

11.6.6.4 Bibliography

[1] Giuriani E., Consolidation of historic buildings, UTET Technical Sciences (2012)

[2] Gelfi P., Giuriani E., Theoretical model of the constitutive link for rung connections, Postgraduate course for reinforced concrete constructions, Italcementi S.p.A. - Bergamo (1987)