3Muri Project
News Version 14

Summary:

Updates

- **New approach for Static Verifications (Finite Element Analysis - FEA)**
- **New feature: floor loads distribution diagrams**
- **Report - Encirclements**
- **Innovative approach to mass calculation**
- **Report - Summary sheets available**

New modules

- **New modelling tool: OPEN**
- **Kinematic analysis in the plane**
- **Nonlinear kinematic analysis**

Local Verifications

- **Local verification of steel elements (Only for Eurocode)**
- **Dowel bars module (Only for Eurocode)**
- **Arch verification**
- **Local Verifications - Floors.**

Updates

New approach for Static Verifications (Finite Element Analysis – FEA)

In recent years, numerical approaches for seismic verification have made countless leaps forward, particularly due to scientific research and the ever-increasing computing power of PCs.

Unfortunately, static calculation procedures have not undergone significant changes.

If we look at the procedures in the current Italian Legislation, we notice that they contain virtually unchanged approaches from what was published in the 1987 DM.

It seems, therefore, legitimate to ask what limitations are inherent in such outdated procedures and, in particular, whether it is possible to perform more advanced calculations while remaining within the current Standards.

The regulations allow the use of proven validity approaches and, therefore, a finite element calculation (Finite Element Analysis or FEA) will be applied in order to be able to take advantage of the maximum computing power, which modern PCs allow, and allow more accurate results to be achieved.

Technical details

On the FEA Analysis environment, all elements entered in the 3Muri structure model are transformed into 1D (beams and columns) or 2D (wall panels and reinforced concrete partitions) finite elements.

Floors are considered as a load-distributing element on beams, panels and RC walls and do not possess any influence in terms of stiffness.

Materials

Isotropic behavior of linear type.

1D elements

Timoshenko behavior, with generic sections;

Inclusion of disconnections at the ends of the members is allowed for both beams and columns.

2D Elements.

3-node shell elements used for wall panels and RC walls.

Constraints

Constraints entered in the model at foundations are automatically recognized and converted in the mesh environment.

it is possible to enter nodal or line constraints; during calculation all nodes in the line will be constrained.

Loads

Supplementary loads can be introduced after generating the mesh: nodal loads, line loads and surface loads.

Results

Diagrams for 1D elements and color maps for 2D elements on the whole structure;

Deformation drawing over the entire structure.

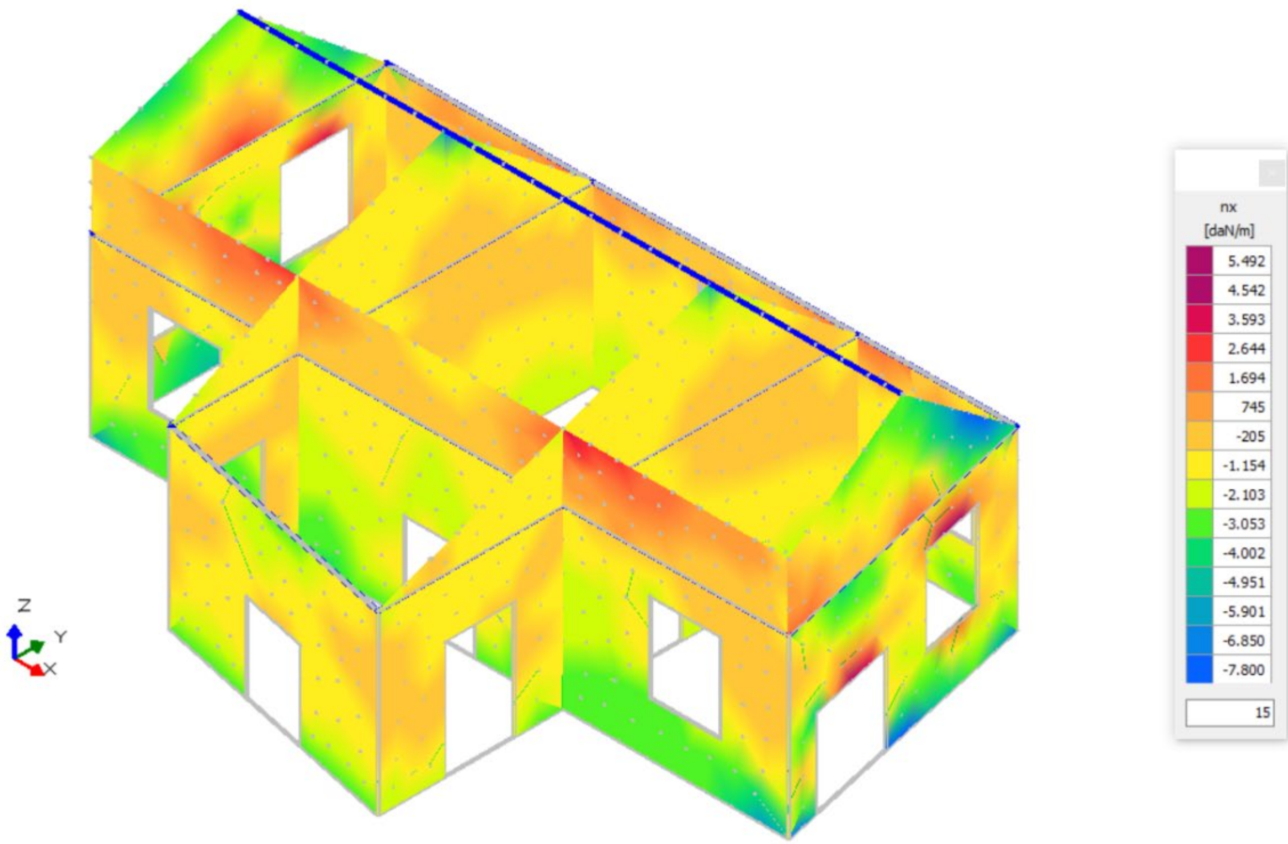
Masonry verification.

Automatic definition of masonry males for walls imputed to levels;

Italian Standards allow for proven verification approaches such as the one given in the Eurocode that will be implemented in the calculation.

Visualization modes.

The environments provide a dual visualization mode, both 3D and plan/prospect.



New feature: floor loads distribution

In professional practice, we increasingly face the problem of validating results.

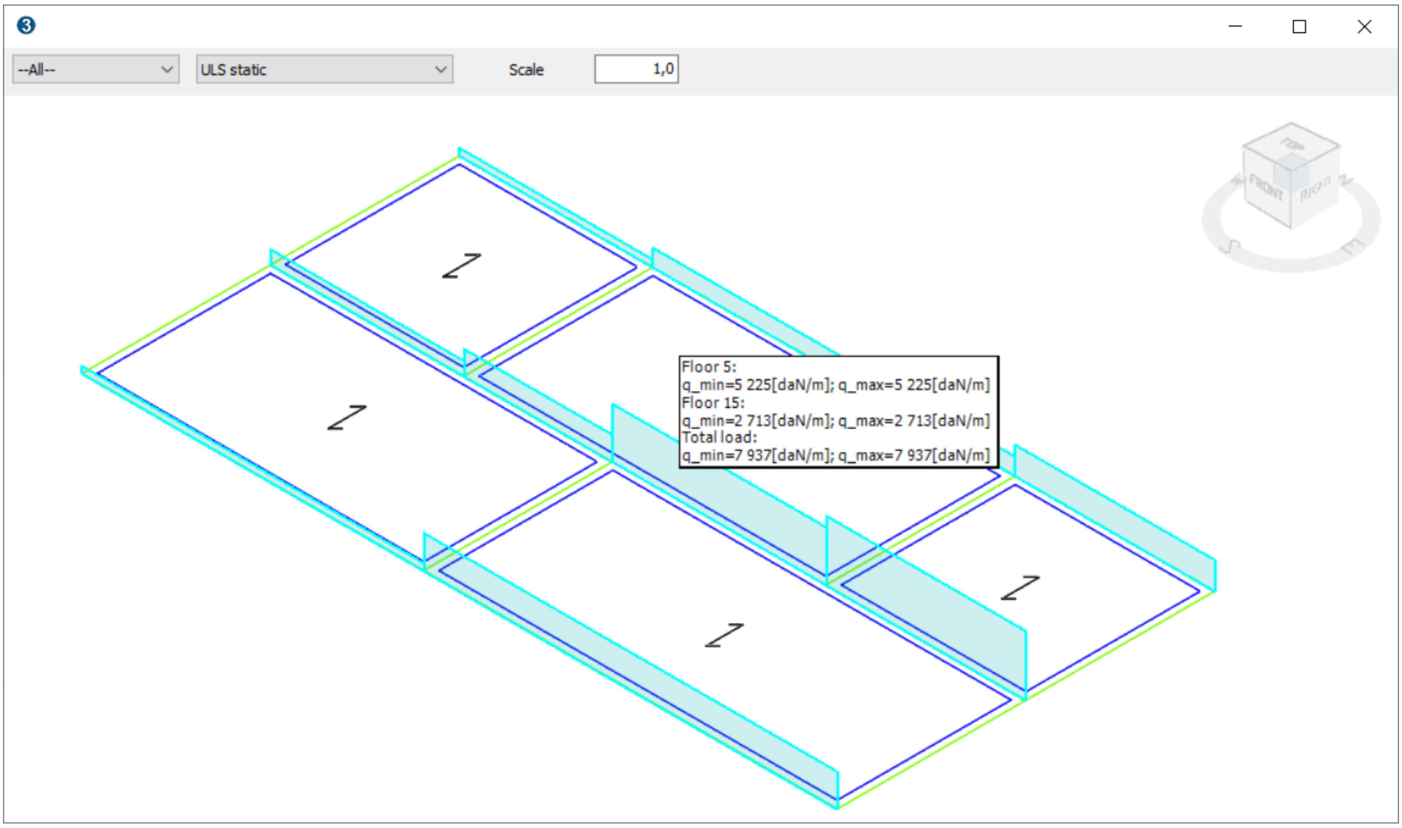
Sometimes , as designers we are required to submit validation calculations along with the report, on other occasions we ourselves feel the need to better understand how the structure "works" or we need to understand whether the model we entered corresponds to what we expect based on structural reality.

Certainly one of the most important aspects is load validation.

The calculation program allows us to enter loads on floors per unit area but, how much of these loads end up on masonry or beams is not always quickly understood since it depends on the areas of influence of the floors for each element.

For this reason, a new feature will make it possible to show the loads acting on structural elements due to the effect of all the slabs acting on them and, if necessary, the effect on one element of each individual slab.

An easily navigable 3-D view of the floor system, with the various diagrams and informative tooltips on the elements, allows for easy querying of the load value.



Report - Encirclements

In the work of verifying, adjusting, and modeling structures, it is essential to have technical documentation that clearly and in detail describes how the structure being modeled is composed.

In daily practice, interventions applied to a structure are as important as the structure itself and must possess in the report the attention they deserve.

In response to this request from the professional world, the visualization of encirclements in the 3Muri report is now available.

Under the "Structure Elements" section of the report, there is now a descriptive table with all the geometric details and mechanical properties of each encirclement included in the model (both steel and reinforced concrete).

Now in the structure elements part it will be possible to display for each hooping information such as:

- Opening ID number and wall where it was inserted
- Dimensions of the opening
- Characteristics of the encirclements (presence of crossbars and posts):
  - Steel encirclements: Type of profile used and its mechanical characteristics.
  - RC encirclements: Section dimensions, materials used, specifications of rebars and stirrups used.

Encirclements

Opening ID	Wall	Description	Upper crossbar	Lower crossbar	Posts
6	1	h1 [cm] = 90; h2 [cm] = 150; a [cm] = 120	Yes	No	No

Type	Material and Geometry
Crossbars	S 235 (t <= 40mm); IPE 100: Area [cm2] = 10.32: J [cm4] = 171.00: W [cm3] = 34.20

Opening ID	Wall	Description	Upper crossbar	Lower crossbar	Posts
7	1	h1 [cm] = 90; h2 [cm] = 150; a [cm] = 120	Yes	Yes	Yes

Type	Material and Geometry
Posts	S 235 (t <= 40mm); IPE 200; Area [cm2] = 28,48; J [cm4] = 1943,00; W [cm3] = 194,30
Crossbars	S 235 (t <= 40mm); IPE 100: Area [cm2] = 10.32: J [cm4] = 171.00: W [cm3] = 34.20

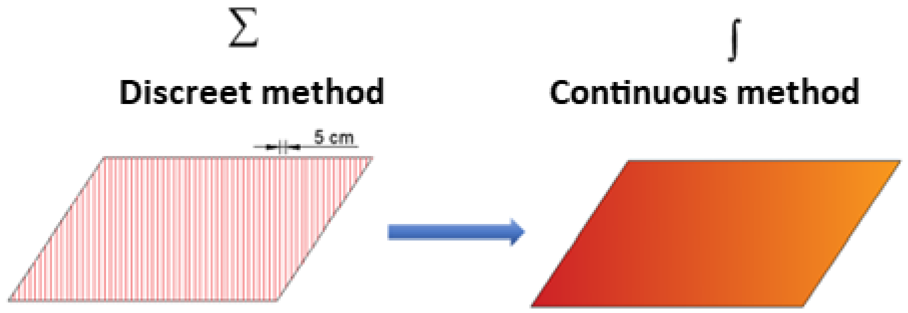
Opening ID	Wall	Description	Upper crossbar	Lower crossbar	Posts
8	1	h1 [cm] = 90; h2 [cm] = 150; a [cm] = 120	Yes	No	No

Type	Material and Geometry	Rebars	Stirrups
Crossbars	C20/25; B450; b [cm] = 10,0; ; h [cm] = 10,0; Area [cm2] = 100,00; J [cm4] = 833,33	Extrados total As [cm2] = 2,26; Intrados total As [cm2] = 2,26; Extrados no. = 2; Intrados no. = 2; Concrete cover [cm] = 3,0	Diameter [mm] = 10; Legs no. = 2; Mid-section spacing [cm] = 10; End spacing [cm] = 10

Innovative approach to mass calculation

The current method of load distribution uses a discrete strip approach parallel to the direction of warping to enable proper load distribution to be found even when floors possess shapes with major irregularities. This approach, in order to provide accurate results, requires that the "strips" possess sufficiently small dimensions, this unfortunately generates significant processing times.

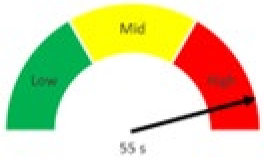
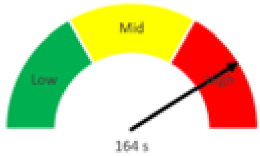
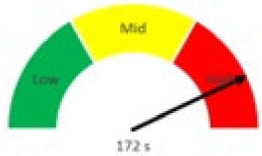
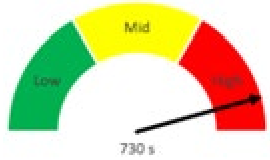
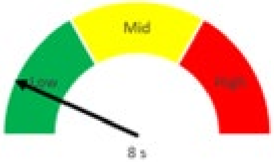
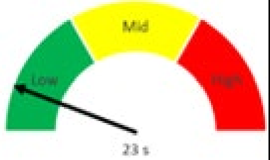
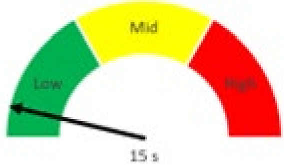
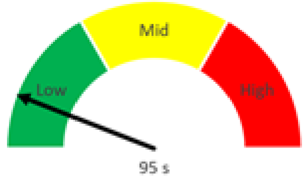
In order to reduce the processing time, the above approach was modified by means of a numerical procedure of continium analysis.



From experiments on the computational code, it appears that in addition to the time advantages, the load distribution is also shown to be more accurate.

To make the magnitude of time savings more concrete, we give below a table of some tests conducted on various models.

	Model 1	**Model 2**	**Model 3**	**Model 4**
	no. floors: 2+roof *no.floors: 5* *floor area: 372 m²* *60% masonry 40% r.c.*	*no. floors: 4+roof* *no.floors: 11* *floor area: 1065 m²* *75% masonry 25% r.c.*	*no. floors: 3+roof* *no.floors: 25* *floor area: 1470 m²* *90% masonry 10% reinforced concrete*	*no. floors: 3* *no.floors: 92* *floor area: 3400 m²* *85% masonry* *10% reinforced concrete 5% steel*

Current algorithm				
New algorithm				
Time saved	**-85%**	**-86%**	**-91%**	**-87%**

Report - Summary sheets available immediately

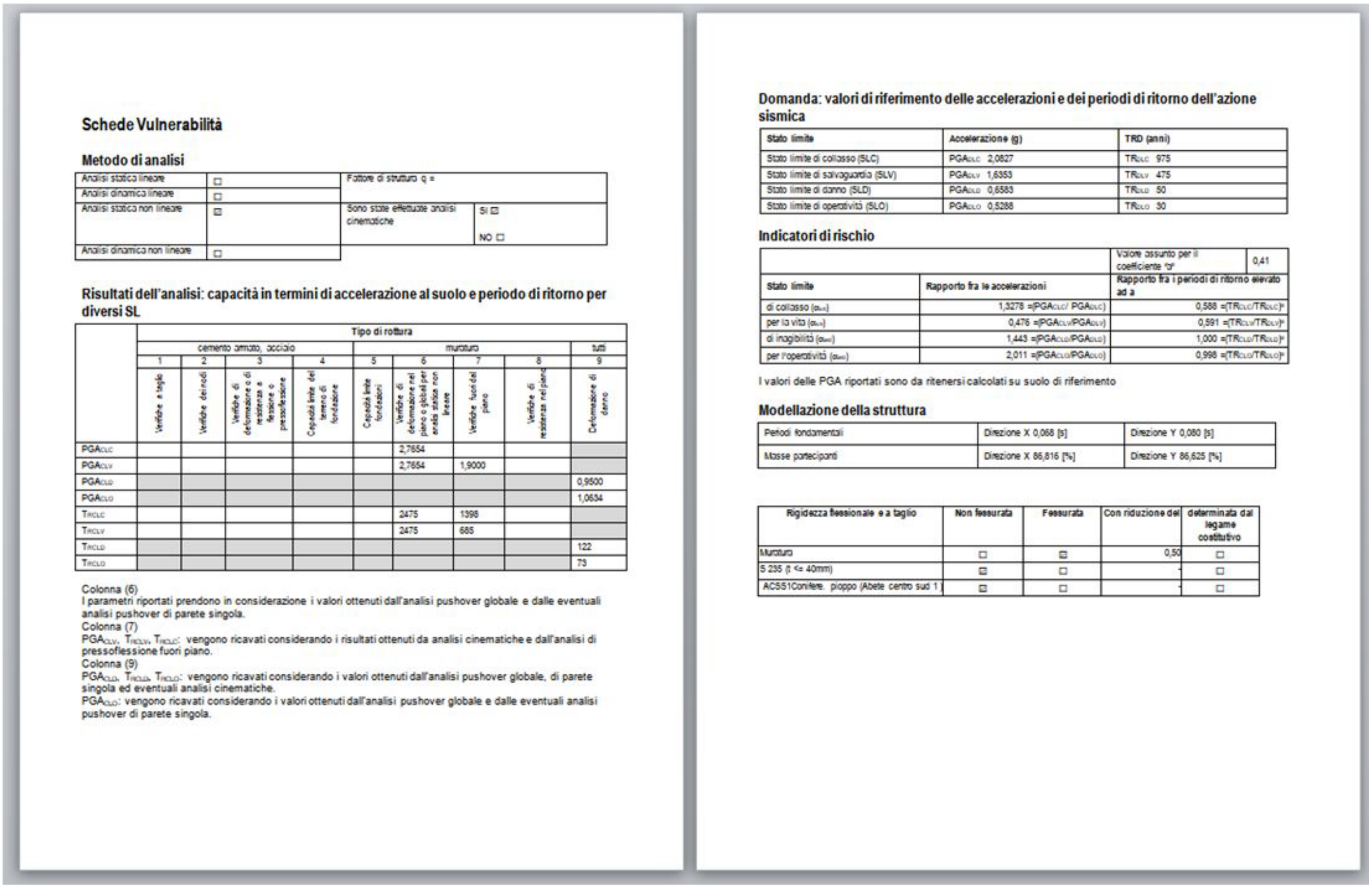
The activity of us designers is made up of research, study and preparation of documents both as drawing tables and drafting of reports or compilation of forms.

Among the forms to be produced upon delivery, there is one that summarizes the outcome of the numerical simulations; to compile it we often have to look for the necessary parameters within the report or in the program.
This search is often cumbersome and occasionally can raise doubts about which are the correct parameters to enter.

We have decided that this research operation must be borne by the program, to allow the designer to devote his time to certainly more significant "research and study" activities in order to produce a quality project.

For this reason, within the calculation report created automatically, some facsimile pages of the aforementioned tables will be available in the queue.

It should be noted that depending on the region, these tables may be slightly different, almost always in terms of layout but almost never in terms of content.



Schede Vulnerabilità

Metodo di analisi

Analisi statica lineare	☐	Fattore di struttura q =	
Analisi dinamica lineare	☐		
Analisi statica non lineare	☒	Sono state effettuate analisi cinematiche	SI ☒ NO ☐
Analisi dinamica non lineare	☐		

Risultati dell'analisi: capacità in termini di accelerazione al suolo e periodo di ritorno per diversi SL

	Tipo di rottura								
	cemento armato, acciaio				muratura				tutti
	1	2	3	4	5	6	7	8	9
	Verifiche a taglio	Verifiche dei nodi	Verifiche di deformazione o di resistenza a flessione o pressoflessione	Capacità limite del terreno di fondazione	Capacità limite fondazioni	Verifiche di deformazione nel piano o globali per analisi statica non lineare	Verifiche fuori del piano	Verifiche di resistenza nel piano	Deformazione di danno
PGA_{CLC}						2,7654			
PGA_{CLV}						2,7654	1,9000		
PGA_{CLD}									0,9500
PGA_{CLO}									1,0634
T_{RCLC}						2475	1398		
T_{RCLV}						2475	685		
T_{RCLD}									122
T_{RCLO}									73

Colonna (6)
I parametri riportati prendono in considerazione i valori ottenuti dall'analisi pushover globale e dalle eventuali analisi pushover di parete singola.
Colonna (7)
PGA_{CLV}, T_{RCLV}, T_{RCLC}: vengono ricavati considerando i risultati ottenuti da analisi cinematiche e dall'analisi di pressoflessione fuori piano.
Colonna (9)
PGA_{CLD}, T_{RCLD}, T_{RCLO}: vengono ricavati considerando i valori ottenuti dall'analisi pushover globale, di parete singola ed eventuali analisi cinematiche.
PGA_{CLO}: vengono ricavati considerando i valori ottenuti dall'analisi pushover globale e dalle eventuali analisi pushover di parete singola.

Domanda: valori di riferimento delle accelerazioni e dei periodi di ritorno dell'azione sismica

Stato limite	Accelerazione (g)	TRD (anni)
Stato limite di collasso (SLC)	PGA_{DLC} 2,0827	TR_{DLC} 975
Stato limite di salvaguardia (SLV)	PGA_{DLV} 1,6353	TR_{DLV} 475
Stato limite di danno (SLD)	PGA_{DLD} 0,6583	TR_{DLD} 50
Stato limite di operatività (SLO)	PGA_{DLO} 0,5288	TR_{DLO} 30

Indicatori di rischio

		Valore assunto per il coefficiente "a"	0,41
Stato limite	Rapporto fra le accelerazioni	Rapporto fra i periodi di ritorno elevato ad a	
di collasso (α)	1,3278 =(PGA_{CLC}/ PGA_{DLC})	0,588 =$(TR_{CLC}/TR_{DLC})^a$	
per la vita (α)	0,476 =(PGA_{CLV}/PGA_{DLV})	0,591 =$(TR_{CLV}/TR_{DLV})^a$	
di inagibilità (α)	1,443 =(PGA_{CLD}/PGA_{DLD})	1,000 =$(TR_{CLD}/TR_{DLD})^a$	
per l'operatività (α)	2,011 =(PGA_{CLO}/PGA_{DLO})	0,998 =$(TR_{CLO}/TR_{DLO})^a$	

I valori delle PGA riportati sono da ritenersi calcolati su suolo di riferimento

Modellazione della struttura

Periodi fondamentali	Direzione X 0,068 [s]	Direzione Y 0,080 [s]
Masse partecipanti	Direzione X 86,816 [%]	Direzione Y 86,625 [%]

Rigidezza flessionale e a taglio	Non fessurata	Fessurata	Con riduzione del	determinata dal legame costitutivo
Muratura	☐	☒	0,50	☐
S 235 (t <= 40mm)	☒	☐	-	☐
AC551Conifere. pioppo (Abete centro sud 1)	☒	☐	-	☐

New modules

New modelling tool: OPEN

In everyday design we have too often been constrained in model definition by the limits of calculation, in truth it is calculation that should serve the structure and not vice versa.

Let us try to think of the Pushover calculation for seismic verifications, the requirement to be able to apply it is the box-like behavior. Very often we find ourselves working with buildings in which the "boxiness" is only partially present, we think of monumental buildings or building aggregates in which the concept of plan depends on the structural unit under consideration.

Can these types of buildings not be calculated, or do they simply need "an eye" to account for their peculiarities?

Obviously these structures must and can be calculated, we must know how to identify the correct approach depending on the portion of the building we are examining. Thus, there will exist parts of the building that can be calculated by pushover because there is boxiness and others, for which local verification such as a kinematic mechanism is necessary.

In order to approach the design as described above, a *modeling tool without geometric constraints* is needed; this will be possible through a new input mode that will complement the existing one. In addition to the modeling for horizontal planes consisting of levels there is a modeling for vertical planes that allows for imputing irregular geometries in elevation and with curved portions.

Obviously, the portions that meet the requirements of boxiness can be calculated by pushover, while the excluded portions can be evaluated by kinematic analysis.

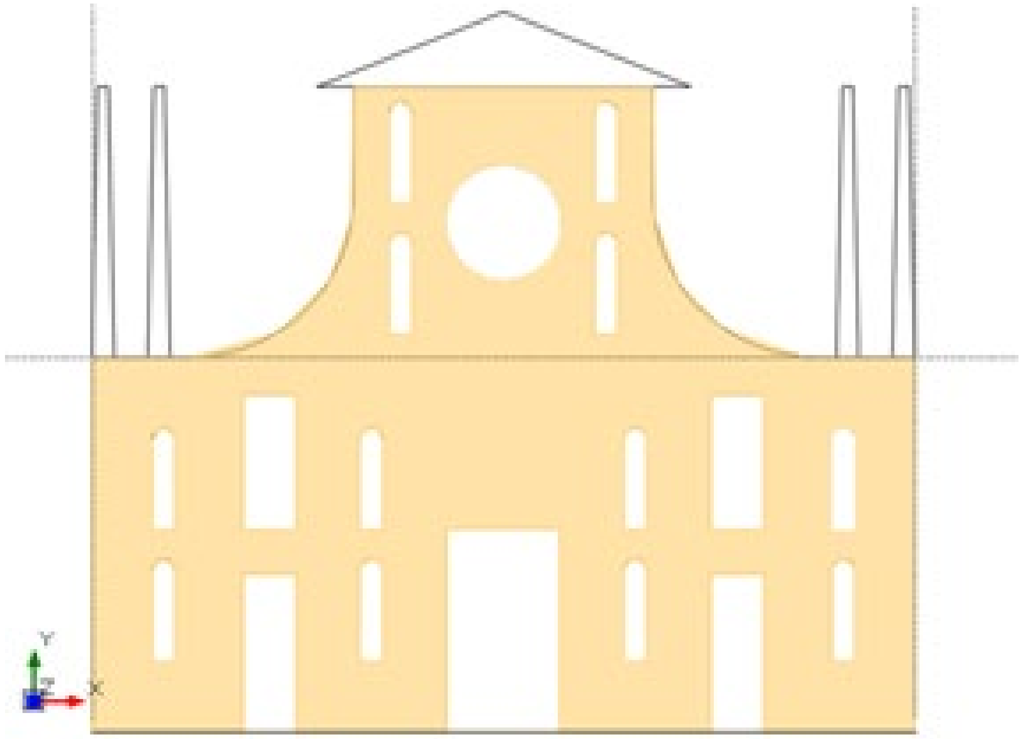


Figure : Example of a wall defined by means of irregular curvilinear elements

---

Kinematic analysis in the plane

A new approach to calculating mechanisms is introduced; until now, only kinematics with "out-of-plane" behavior were examined, but now "in-plane" verification approaches are added.
Porches, arches and curved elements formed by colonnades are usually subjected to this type of failure that can be answered by this calculation method.

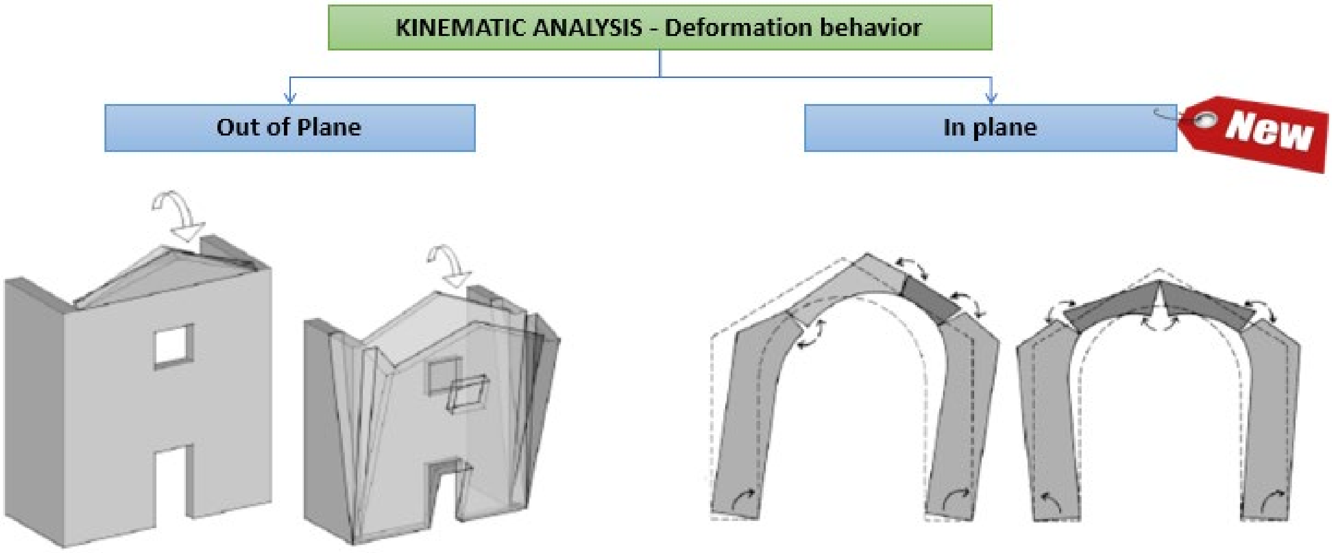


Figure : Diagram of the modes of behavior in the kinematic field

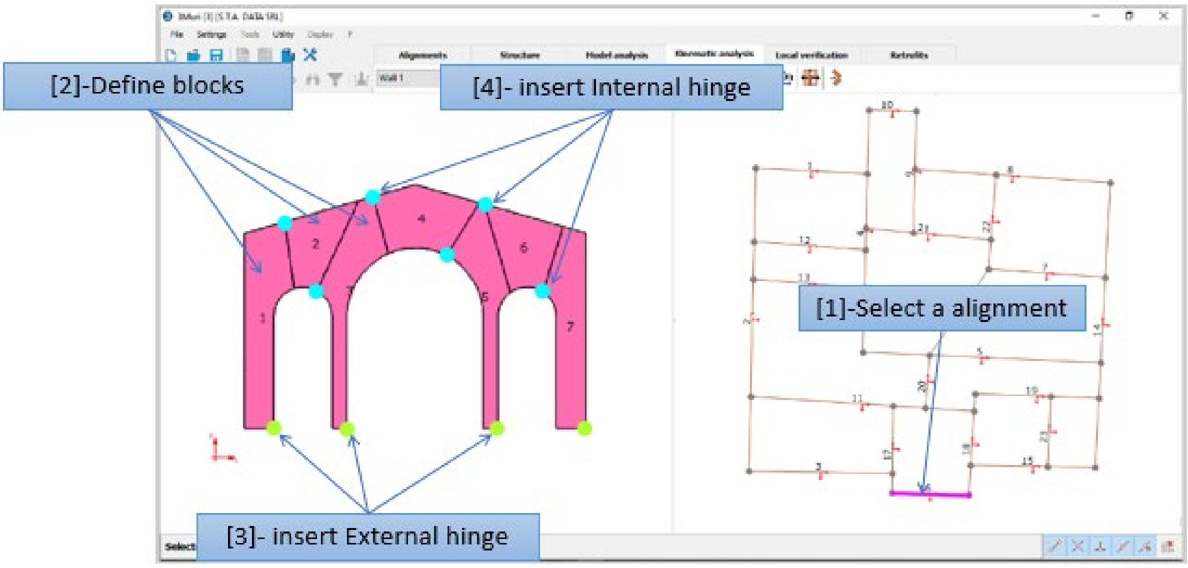


Figure : Modes of input of an in-plane kinematism

Nonlinear kinematic analysis

The linear kinematic analyses currently present are being joined by the nonlinear computational process. The advantages of this type of analysis become particularly evident in all those cases where there are nonlinear elements capable of influencing the mechanism, mainly among them being the inclusion of a tie rod with elasto-plastic behavior that influence the result in the nonlinear field.

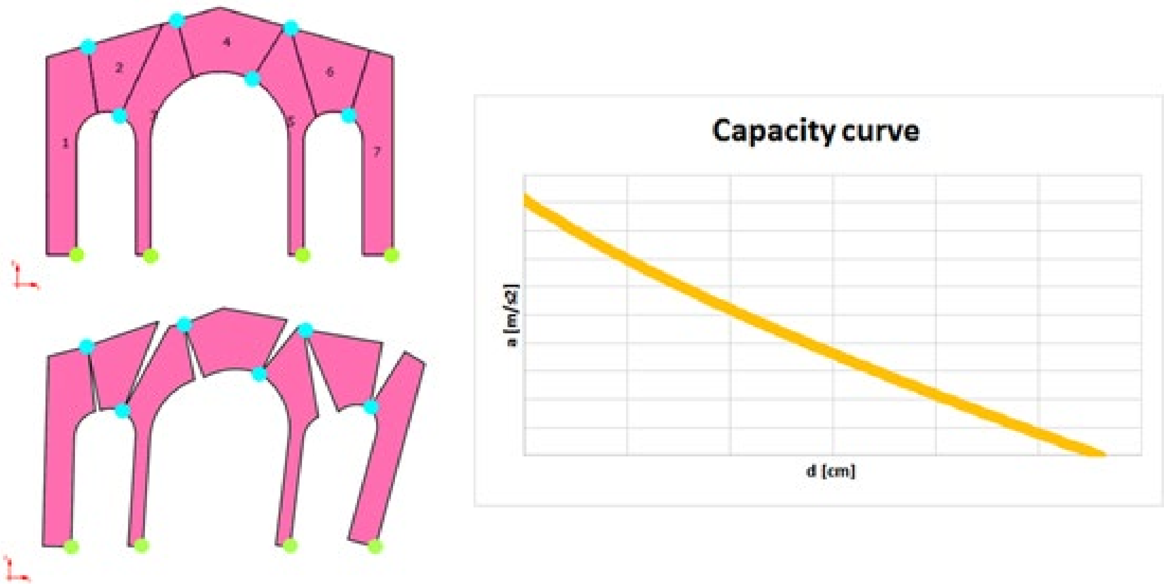


Figure : This figure shows the capacity curve produced by a nonlinear analysis

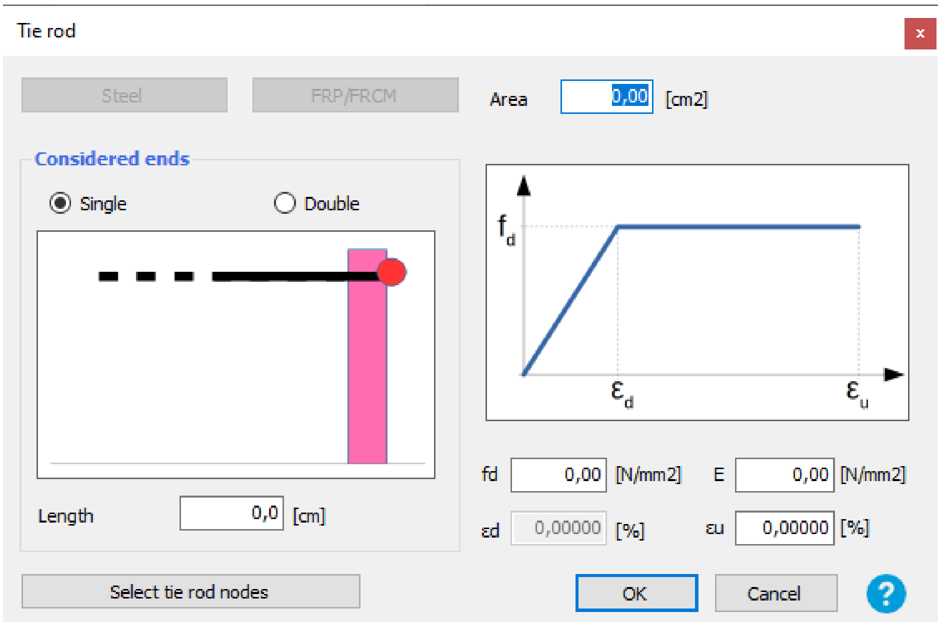


Figure : Ways of inserting a tie rod

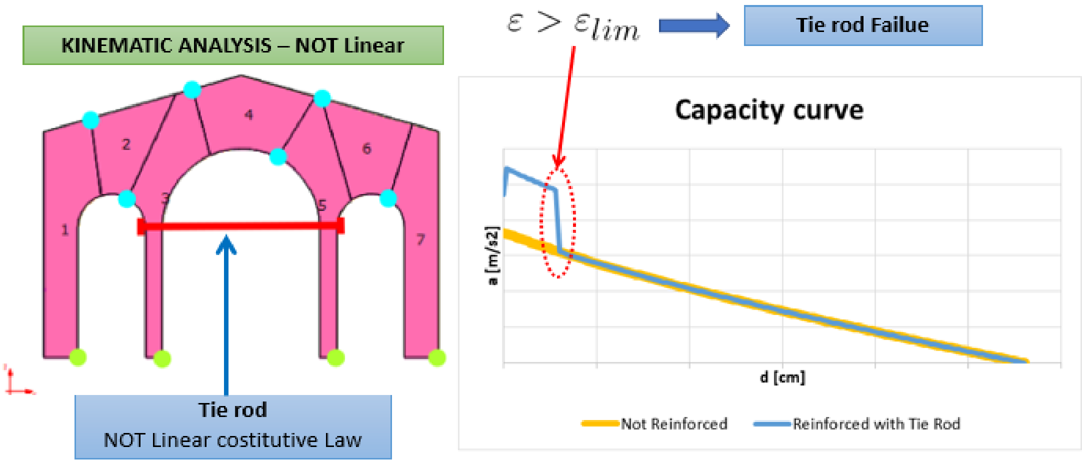


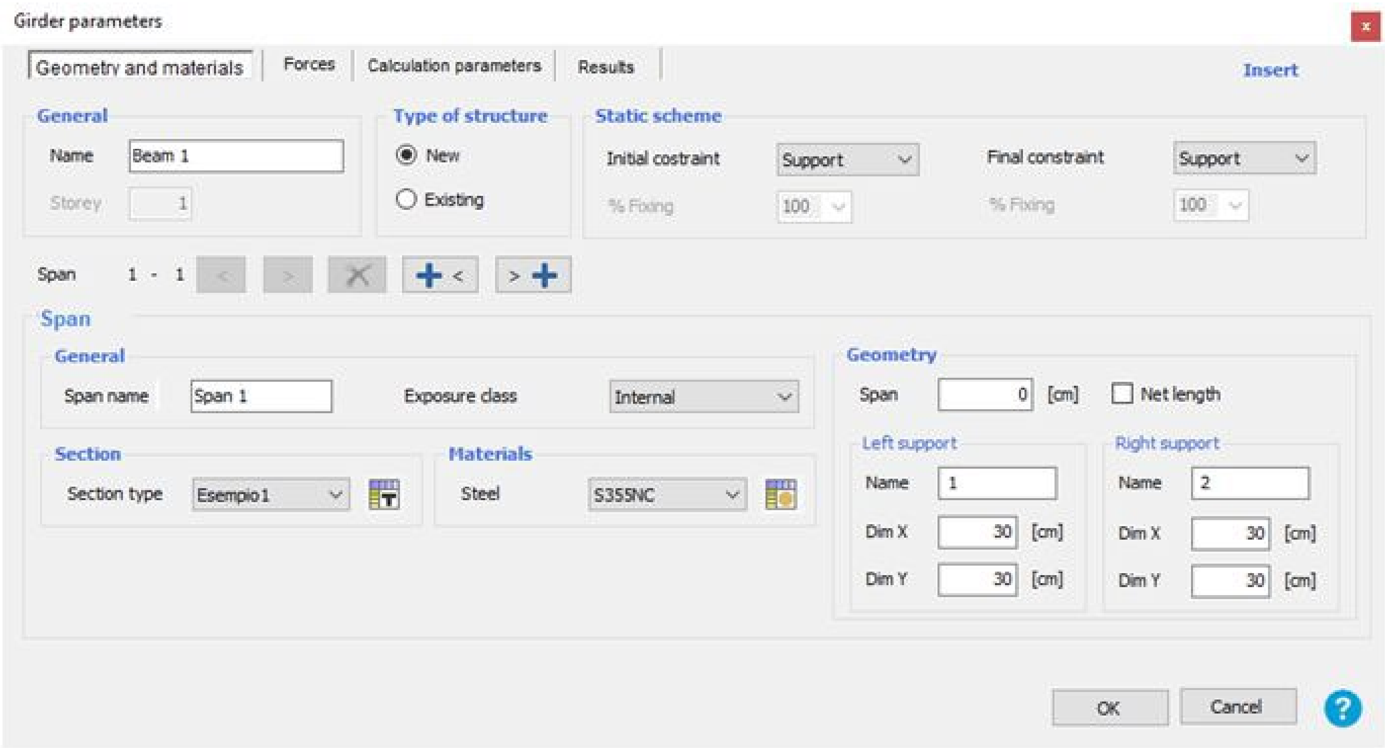
Figure : Benefits of a tie rod in a nonlinear analysis

Local verifications

Local verification of steel elements (Only for Eurocode)

It is well known to all how the panorama of existing masonry buildings is actually made up of "mixed" buildings with predominantly walls but, what makes the building "mixed" is the simultaneous presence of structural elements in different materials (c.a., steel, wood, etc ...) commonly known as rod elements (beams and pillars).
In addition to the verification of reinforced concrete elements, already present in previous versions, the verification of steel rods is added.
The stresses in the sections of the element are automatically extracted from the global model for their structural verification. The stresses in question are always exposed to the user in order to allow for any manual modification in order to manage any special cases that the designer must face in practice.
For steel rods the software performs:

- Classification of sections,
- Calculation of the effective sections if they fall into Class 4,
- Strength checks by combination of normal stress, shear and bending moment in the main directions,
- Verification of instability for pressure and lateral flexion.



Dowel bars module (Only for Eurocode)

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 stresses of shear that are generated.

The dowel technique is relatively simple, inexpensive and minimally invasive.

Arch verification

The Arch module allows limit analysis with a static and kinematic approach for arched structures of generic shape, and also fiber-reinforced with FRP / FRCM, pursuant to Circular no. 7 of 21 January 2019, of the LLPP Guidelines of 29 May 2019, of the CNR-DT 200 R1 / 2013 and of the CNR-DT 215/2018.

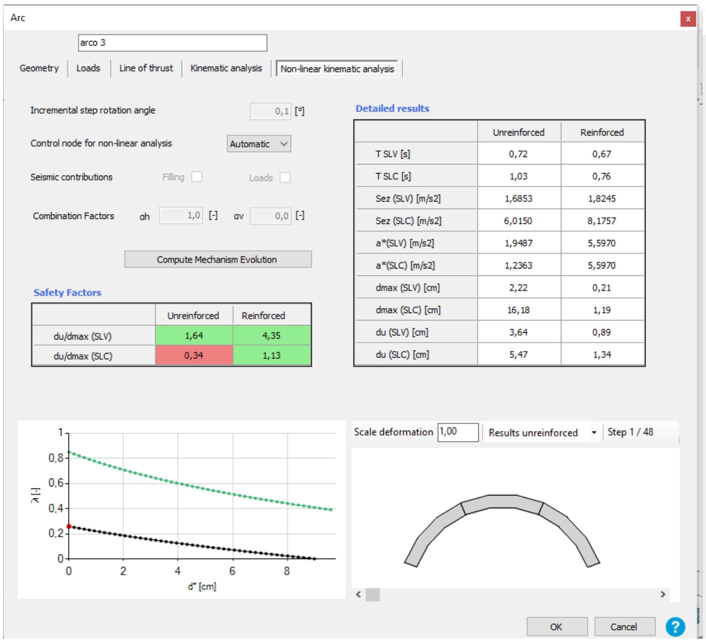
The calculation using the kinematic approach can be performed in the presence or absence of seismic forces.

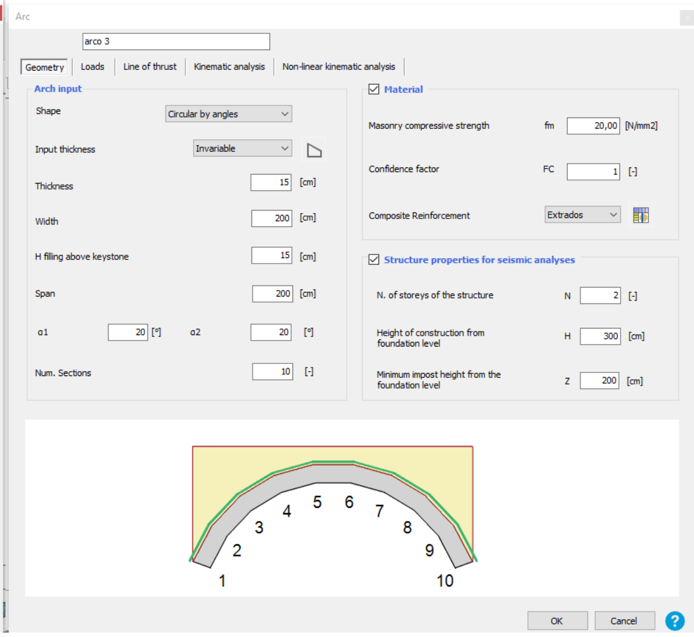
The general approach of the static method is the tracing of a funicular, which if it falls within the edges of the arch indicates its safety status.

The general principle underlying the kinematic analysis is that the structure is supposed to be three times hyperstatic and that, therefore, the collapse occurs when four degrees of freedom are added (with consequent transformation of the structure into a kinematism).

The module contains:

- Calculation of the pressure curve (non-reinforced arches)
- Checks of the structure following the static analysis
- Calculation of the kinematic chain under static loads (non-reinforced and reinforced arches)
- Calculation of the kinematic chain under seismic loads (non-reinforced and reinforced arches)
- Non-linear calculation of the kinematic chain (evolution of the collapse mechanism)
- Verifications of the structure following the kinematic analysis.





Local Verifications - Floors.

The recent implementation of the "OPEN" module, to allow modeling of walls with irregular geometric conformation to better adhere to the design needs of professionals, undergoes further integration.

Now, local verification modules for floors are also available for floors that interact directly with irregular walls modeled in "vertical plane" mode.
