



**Thesis: Modelling of Seismic Response of Masonry Structure:
Comparison between Code-Aster and 3Muri**

Master of science program in
CIVIL ENGINEERING (LM-23)

Supervisors

Prof. Rosario Ceravolo

Dr. Gaetano Miraglia

Company

STA DATA SRL

Thesis Candidate

Muhammad Abraiz

s287972

Abstract

In Structure analysis of Masonry structures two separate models are realized normally for global and local verifications using Finite element analysis (FEM). According to NTC-18 local interventions made either to a single part or an element of a structure should not change the global behavior of structure significantly. But there is always a doubt that whether the local intervention will change the global behavior significantly or not. This gave rise to 3Muri project that make use of a single integrated model for making both Local and global verifications on a same single model and use an innovative approach of frame by macro elements (FMA). To support this idea a masonry building of town hall of pizzoli in central ITALY was modelled in 2 different open-source software's: software Salome Meca which implements code aster, which is powerful FEM solver and 3Muri which implements an innovative fast approach of meshing using Frame by Macro elopements elements. Modal analysis was performed in both software's and seismic response of masonry structures was compared in terms of Time period of vibration of structure for first 4 modes of vibrations and ultimate displacement was also measured and compared. when two software's were compared it was observed that there was negligible difference in time period of vibration (about 15% for first 4 modes) obtained from the 3Muri and Salome Meca. Moreover, the difference in time period of vibration between two software's increases for higher modes.

Keywords

Code-aster, 3Muri, Frame by Macro elements (FMA), structure analysis, Finite element analysis (FEM).






I DEDICATE MY WORK TO MY FAMILY AND TO FIELD OF STRUCTURE ENGINEERING
THAT I LIKE THE MOST AND IS VERY PASSIONATE ABOUT.



**Politecnico
di Torino**

.....	1
Abstract	2
Introduction	8
Background	8
Case Study (Building Description)	9
Numerical Modelling of Masonry Structures.....	11
Code Aster and Salome Meca	12
Code-Aster Website http://www.codeaster.org	12
Code-Aster Input and output formats	13
Salome_Meca.....	14
Numerical Model in Salome Meca.....	14
1. Geometrical Model	15
2. Mesh Module	16
3.	20
5.Assign Material properties	58
3MURI	67
Introduction	67
Background of 3Muri Project.....	67
Theoretical Basis of 3-Muri	70
Numerical Model of 3Muri	106
Reference Codes	106
Response Spectrum Definition	107
Modules Inside 3Muri	112
Units and formats	112
Geometrical Model	113
Structure Properties.....	115
Walls.....	115

S287972
Thesis Report

<i>Defining Material Properties for walls</i>	 115
<i>Assign of Material Properties</i>	 127
<i>Openings</i>	 128
Analysis inside 3Muri	 134
Linear Static Analysis	 136
Modal Analysis	 139
Results and Conclusion	 149
Bibliography	 151

Introduction

Code aster is a powerful finite element mechanical solver developed in France can perform both static and dynamic analysis in field of civil Engineering as well. Code aster is implemented in an open-source software known as Salome Meca which has been used here for structure analysis of masonry building in order to compare the approach of Finite element analysis, typical approach which has been wide spread in many commercial software's. Salome Meca can perform both linear and non-linear analysis and considers various non-linear material properties. 3Muri on the other hand is a software that use unique innovative approach of Frame by Macro elements particularly designed for structure analysis of masonry structures developed as result of 3Muri project. 3Muri is made specifically for the linear and non-linear analysis of masonry structure developed in ITALY. There are already some common types of masonry structures and various types of bonds strength that can be realized in 3Muri. Moreover, 3Muri there is also a possibility to reduce the material and mechanical properties for existing structure based on level of knowledge of structure. Muri makes use of an innovative approach of Frame by Macro elements (FMA) an approach made specifically based on experience of most common types of failures that occurred in masonry buildings over last couple of years. This approach of FMA is much faster than FEM and saves the computational effort and cost of analysis. 3Muri make use of equivalent frame approach that has deformable parts (spandrels and piers) that are connected through rigid zones.

Background

The seismic response of masonry building subjected to earthquake are modelled using different modelling strategies in the commercial codes but they follow some simplified assumptions. Particularly for existing masonry structures there are some uncertainties associated with material and mechanical properties of mortar material. However, it's not possible to rely completely on a single modelling strategy for all types of structures. It's possible to carry out the non-linear analysis in commercial finite element software's using the non-linear constitutive laws of materials. However, an innovative and simplified approach of equivalent frame method proposed particularly for the structure analysis of masonry structure is proposed based on the observation of damages that can occurred in last couple of years in masonry structures. It has been observed from practical experience that most of damage is concentrated in spandrels and piers while rigid nodes remain intact.

Equivalent frame approach divides the structure into horizontal and vertical elements known as spandrels and piers where most of damages takes place and these spandrels and piers are connected through rigid nodes. It has been observed that most of damages that has occurred in masonry structures has taken place in these horizontal and vertical panels.

(SARERI, 2017-2018)

Case Study (Building Description)

In order to make a comparison between the code aster and 3Muri a two-story building in masonry was chosen. **2-Story Masonry Building** Located in Central **ITALY** in Northwest of city of **L'aquila, (Abruzzo)** was chosen. The building is an old **School Building** built in **1920**. The building was a regular U-shaped Regular Plan with Regular openings and has Total Area of 770 m² and Total Volume of 5000 m³. Roof of this building was replaced with RC Slab and External Elevator was added later. This building is one of network of building **monitored by Seismic observatory of Structure** (Osservatorio sismico delle strutture – OSS). It's a service of seismic risk and it aims to protect the public service buildings and other important structures and are monitored by using some sensors that record the acceleration as result of earthquake. The *figure 1* shows the original condition back in 1920 while *figure 2* shows the current position of building. that was modelled in code aster and *figure 3* shows the plans of building which was almost symmetrical for all floors and positions of accelerometers installed on the buildings that recorded the accelerations continuously as part of structure health monitoring system.



Figure 1: Town Hall of pizzoli in 1920



Figure 2: Town Hall of pizzoli in 2018

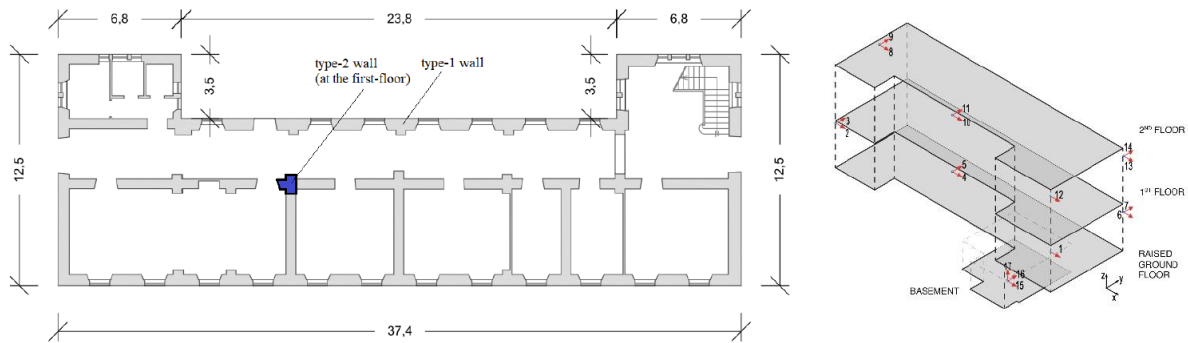


Figure 3: Plan of Building and Position of acetometers installed on building

Numerical Modelling of Masonry Structures

Numerical modelling of masonry structures is a complicated process and there are many simplifications that have to be made because there are many uncertainties associated with material and mechanical properties of materials in masonry structures. These uncertainties are more predominant in case of existing masonry structures. Masonry structures are mostly realized with different types of materials and subjected to decay with passage of time. So, it's always recommended to perform some tests in order to determine the level of knowledge of our structure before performing the structure analysis of an existing masonry building. Moreover, the seismic response of structure also depends on the bond strength of masonry blocks which is not always easy to determine. The most of resistance is provided by compressive strength of masonry blocks while tensile strength is taken equal to zero. However, this assumption can change the seismic response of masonry structure significantly.

Code Aster and Salome Meca

Code_Aster, acronym for **Analysis of Structures and Thermo-mechanics** for Studies and Research from France that takes into account thermal effects and dynamic loading history. It was developed by EDF (Électricité De France) a French electricity utility company.

Code_Aster is a finite element analysis solver, a **FEM Mechanical Solver** Engine and can handle following types of analysis as follows:

- linear statics
- non-linear statics
- dynamic Analysis

Code-aster is an **Open-source software**. open to everyone as everybody can add into its source code. Opposite of it is commercial code not available to everyone.

Code aster is basically a combination of algorithms for Finite element analysis.

Code-Aster Website <http://www.codeaster.org>.

Code_aster have a website which has an open access and all the documents that are required to learn and use the code_aster is available on the website. Moreover, the documents are divided into different categories as can be seen in the *figure 4* below:

S287972 Thesis Report

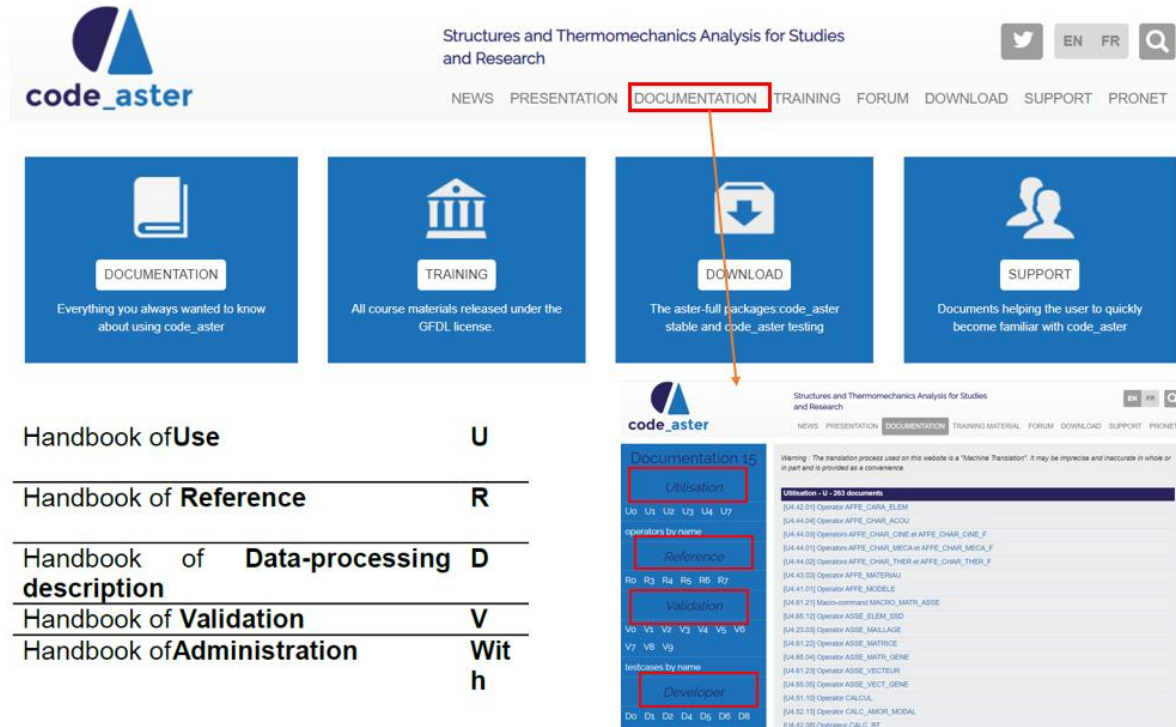


Figure 4: Code_Aster website available documents

Code-Aster Input and output formats

Code aster has various types of input formats and it's written in python. It's possible to give input to code aster in Salome-meca software by using an input file that is written in python or it's possible to gives input in terms of other input file formats such as can be seen in the *figure 5* below:

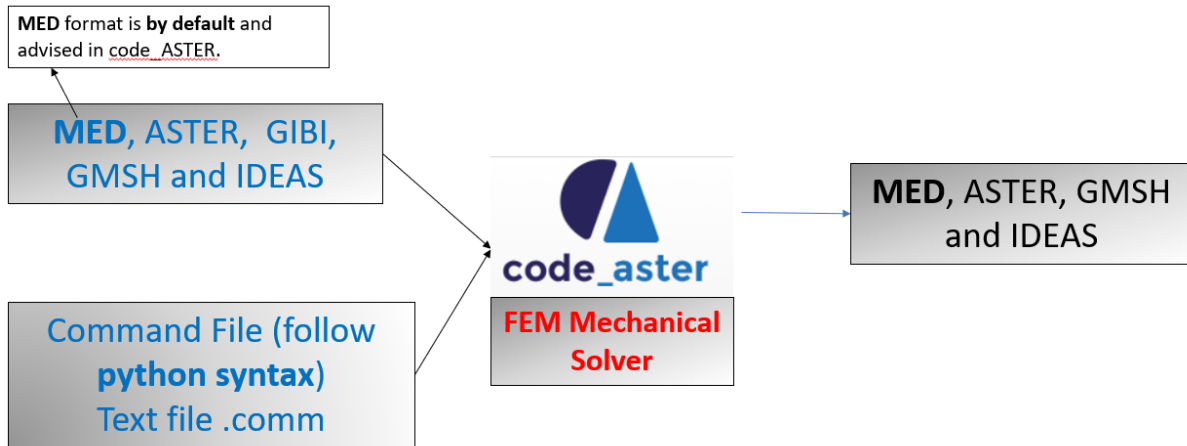


Figure 5: Input and Output File Formats of Code_Aster

However, the. Med format is by default used in code_aster.

Salome_Meca

SALOME is an **open-source software** that provides a generic platform for pre- and post-Processing for numerical simulation Salome-Meca is a bundle which allows to run code-aster in a single GUI. Inside the Salome Meca there is a module of code aster as can be seen in *figure 6* below:

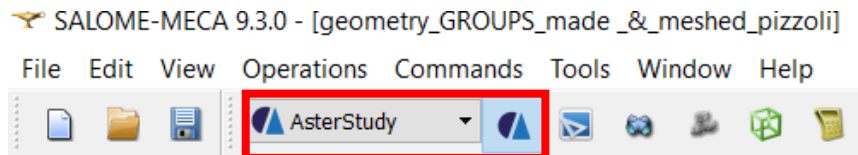


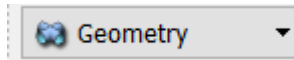
Figure 6: Code_Aster Module inside Salome Meca

Open-source software is open to everyone as everybody can add into its source code. Its source code is available to public and they can view, modify and add into it. Opposite of it is commercial code not available to everyone. Code aster is basically a combination of algorithms for Finite element analysis.

Numerical Model in Salome Meca

A numerical model was realized inside the software completely inside the Salome Meca. It's possible to model directly on Salome Meca or import the input file in .MED file from GMSH. Just the centerlines were modelled 1st then. Building was having regular openings. The geometrical model was having following significant geometric properties:

1. Geometrical Model



The geometrical model realized inside the Salome Meca has following significant features:

- 1074 no of Nodes
- 1609 no of lines
- 588 no of faces/Surfaces
- Modelled just centerlines of walls
- 42 Groups in total
- Regular Openings
- 2 Types of Beams for Elevator Frame and RC Roof.
- Remaining all groups for shells (for walls and slabs).

Groups of nodes and lines were made such that later we will assign the material properties to them. We have to create groups in mesh in order to define boundary conditions and load later on.

Inside Salome Meca it's possible to create groups in mesh in order to define boundary conditions and load later on. Two major types of groups were one for beams as follows:

- 1st elevator all steel beams all line were made a group as Compound group as "Elevator_Steel_Frame_Beam_type1"
 - 2nd RC beams of Roof all line were made a group as Compound group as "Roof_RC_Beam_type2"
 - All other faces were defined as Shell elements.
- 2nd RC beams of Roof all line were made a group as Compound group as "Roof_RC_Beam_type2". All other faces were defined as Shell elements.

Finally geometrical model realized in Salome Meca can be seen as in *figure 7* below:

S287972 Thesis Report

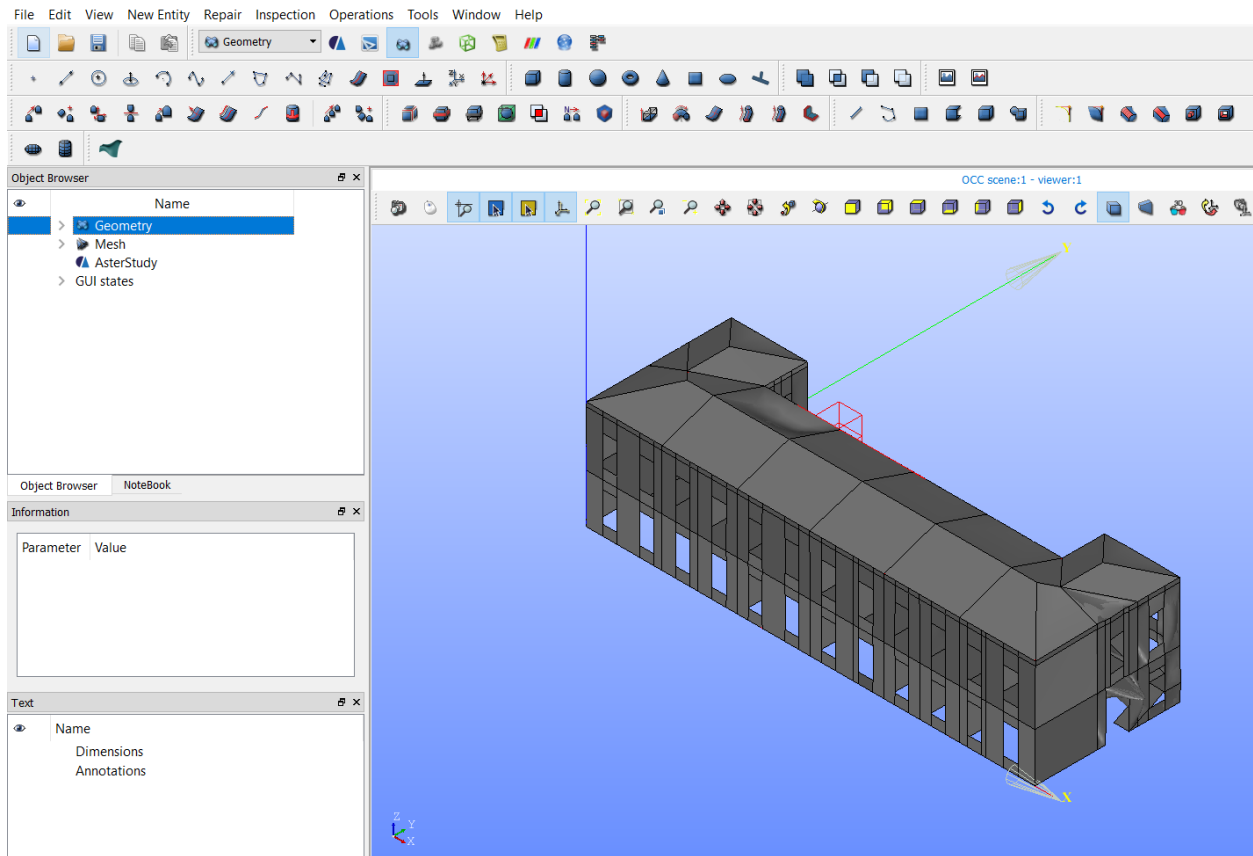
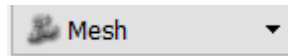


Figure 7: geometrical Model inside Salome Meca

2. Mesh Module



Meshing is subdivision of our structure into elementary objects. Following types of meshing can be done inside Salome Meca:

- point
- edge
- square
- tetrahedron
- triangular

There are many types of meshing that can be defined inside the Salome meca. There were 2 major types of meshes that were created one for beam and other for shell as follows:

([U4.21.01] Operator LIRE_MALLAGE, n.d.)

Meshing for Beams

Automatic length having fineness of 0,25 was assigned to all the beam elements in Hypothesis

Following types of meshes were defined as can be seen in the *figures 8-11* below:

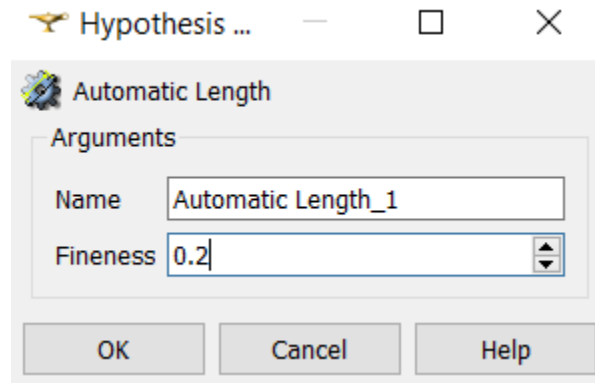


Figure 8: Hypothesis chosen for Beam Meshing

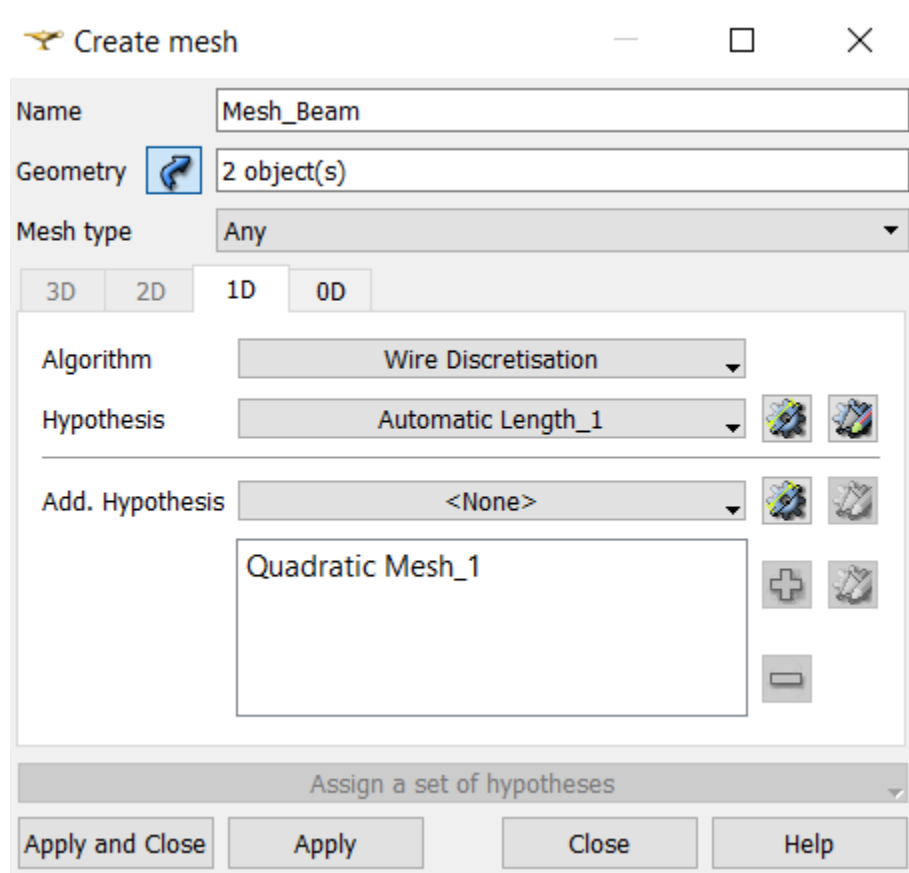


Figure 9: Beam Meshing

Shell Meshing

Walls and slabs were modelled as shell and following and meshing used for shell was having following properties. Shell meshing thickness is taken as 0.2m and no. of layer=1 taken from the as shown below Stairs are modelled as 0.2m thick plate.

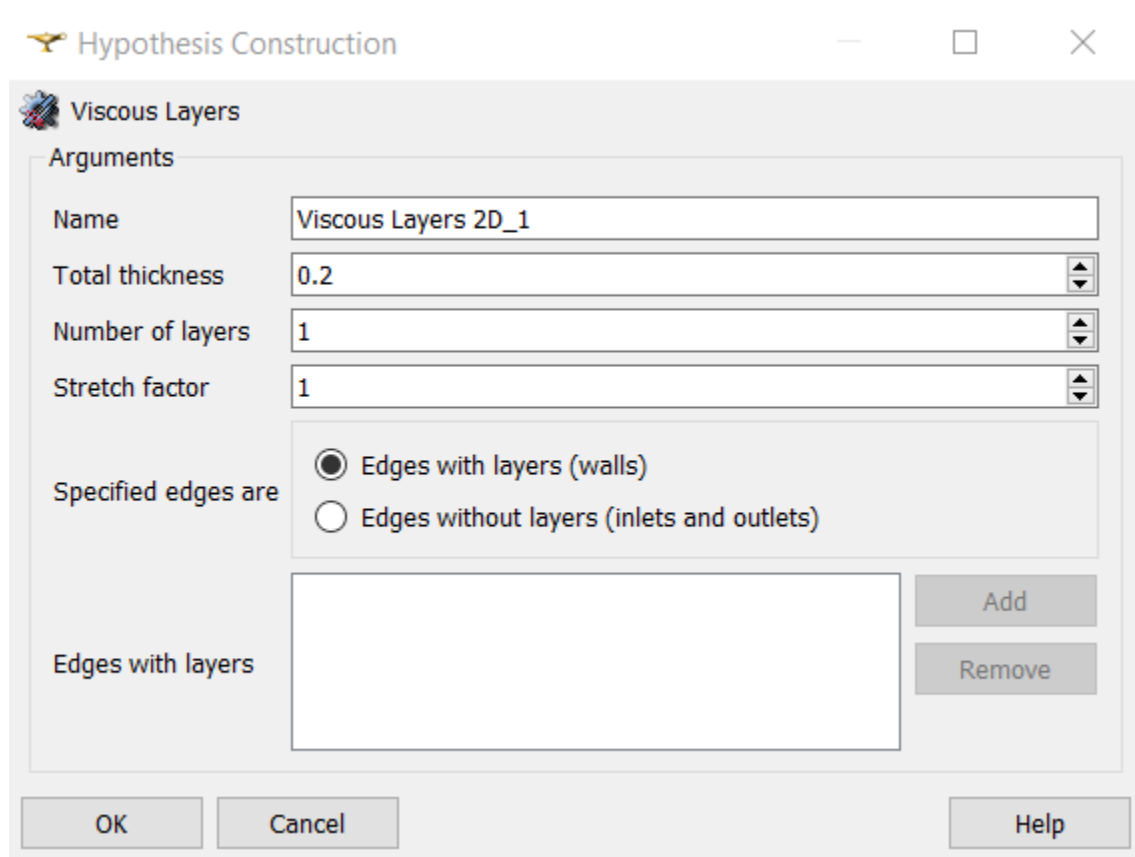


Figure 10: Hypothesis for Stairs for Mesh

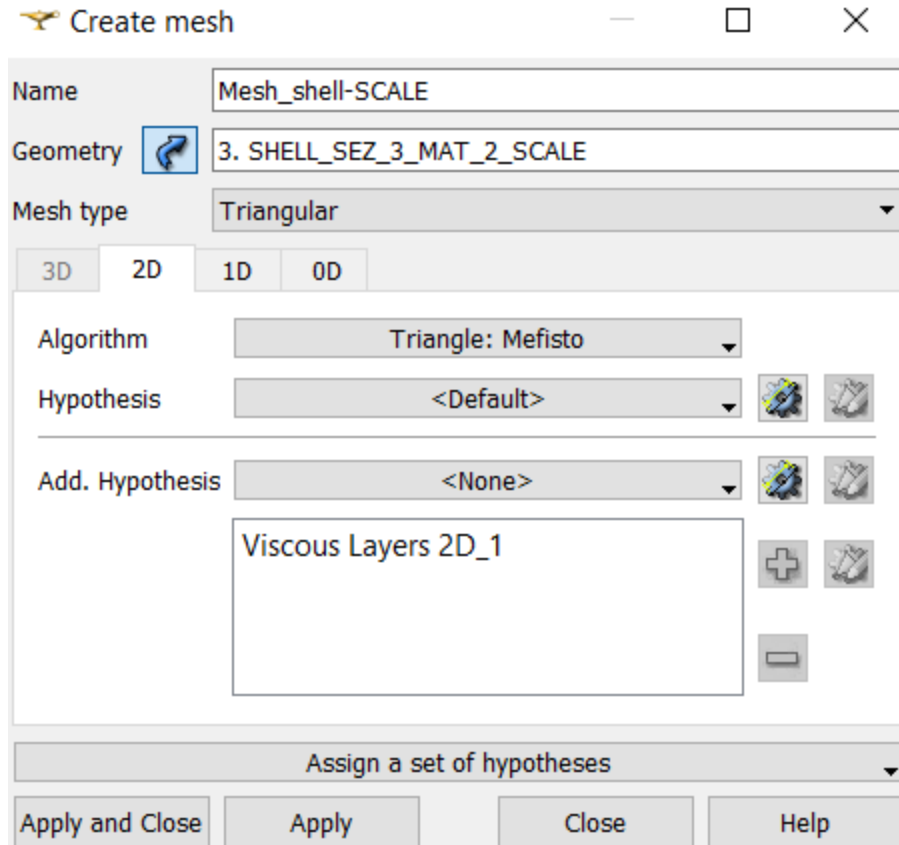


Figure 11: Meshing for Stairs as Shell

In the similar manner the meshing for shells for walls and slabs were defined 1st and then assigned as per the groups made in the geometry module. The *figure 12* below shows the pictures of major two types of meshes defined for Beams and shells as follows:

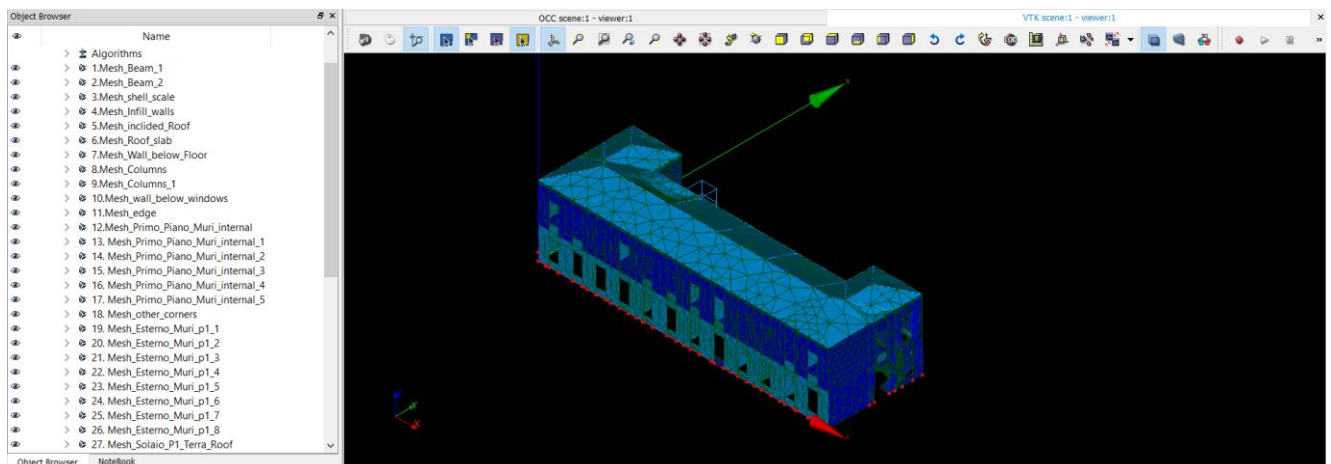


Figure 12: Meshing inside Salome Meca

3.Aster Study

There is complete module inside the Salome meca software that implements code aster. Code aster being the collection of powerful algorithms and has many applications in many fields such as Civil, Mechanical and electrical and its continuously updated and evolved over past 25 years of research.

The following *figure 13* below the steps or stages followed inside the code aster module in order to carry out any types of analysis which has been discussed one by one later:

- Add a stage (we can use a readymade stage using stage assistance of linear static analysis for example)
- Read a mesh
- Define element
- Assign to the group that we have created in mesh
- Command file is created in Aster study

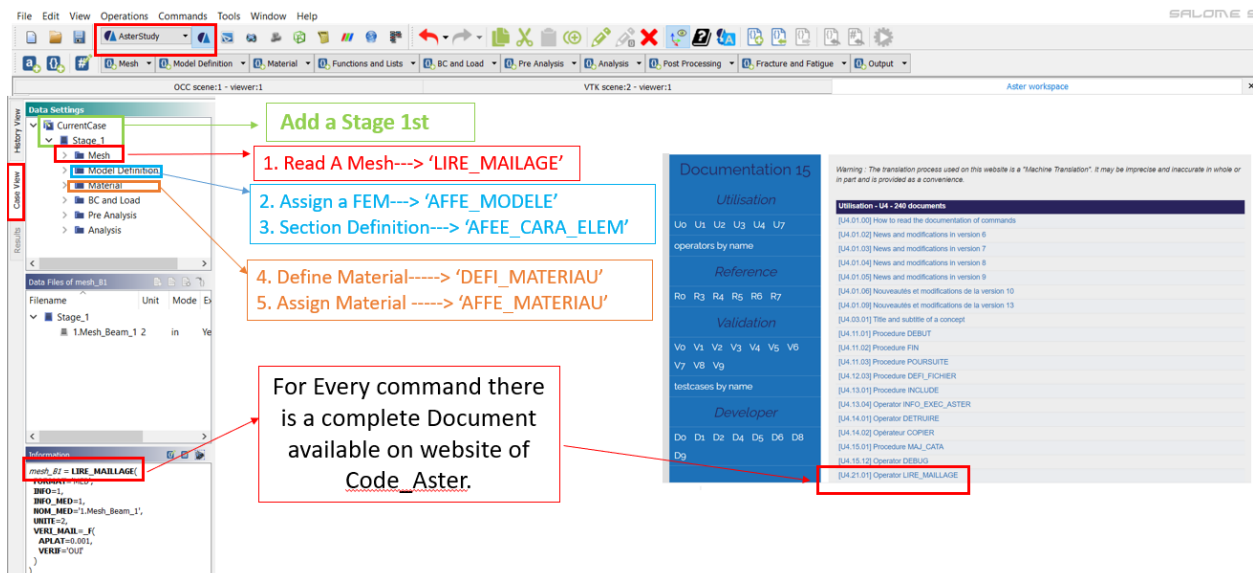


Figure 13: Steps of analysis inside Code aster module

Add a stage

The 1st step to do inside the code aster is to add a stage in case view as I have created stage 1 in the case view and later all the work will be done in this stage.

Reading Mesh

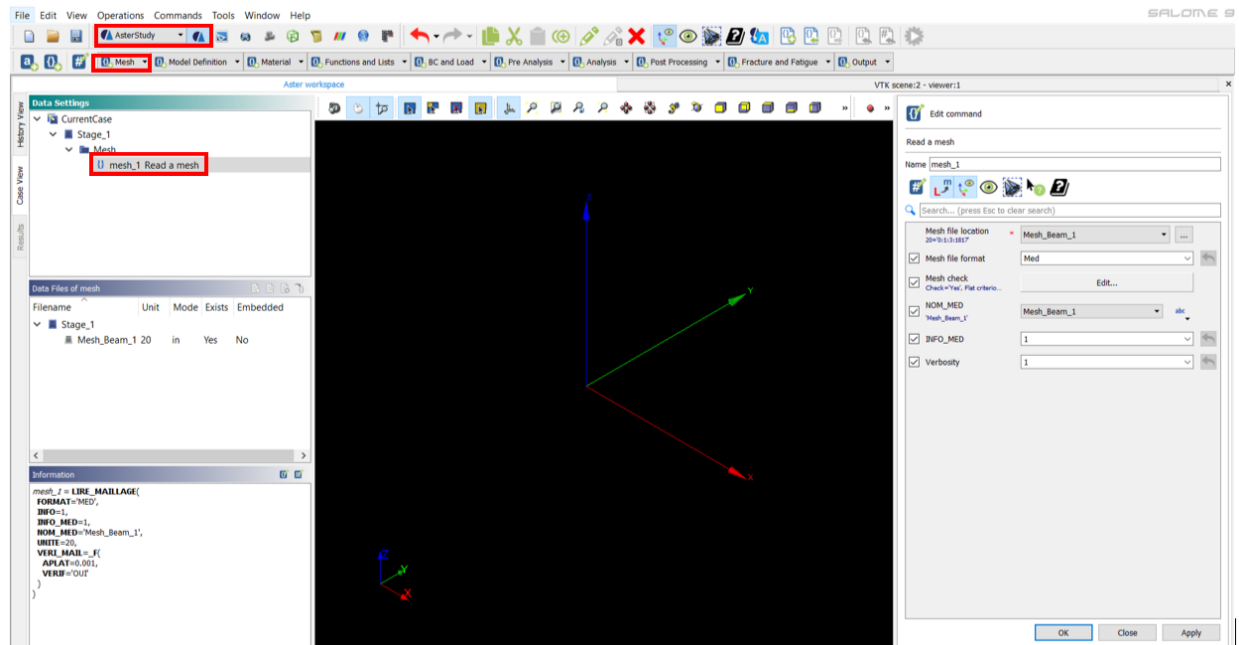


Figure 14: Reading a Mesh

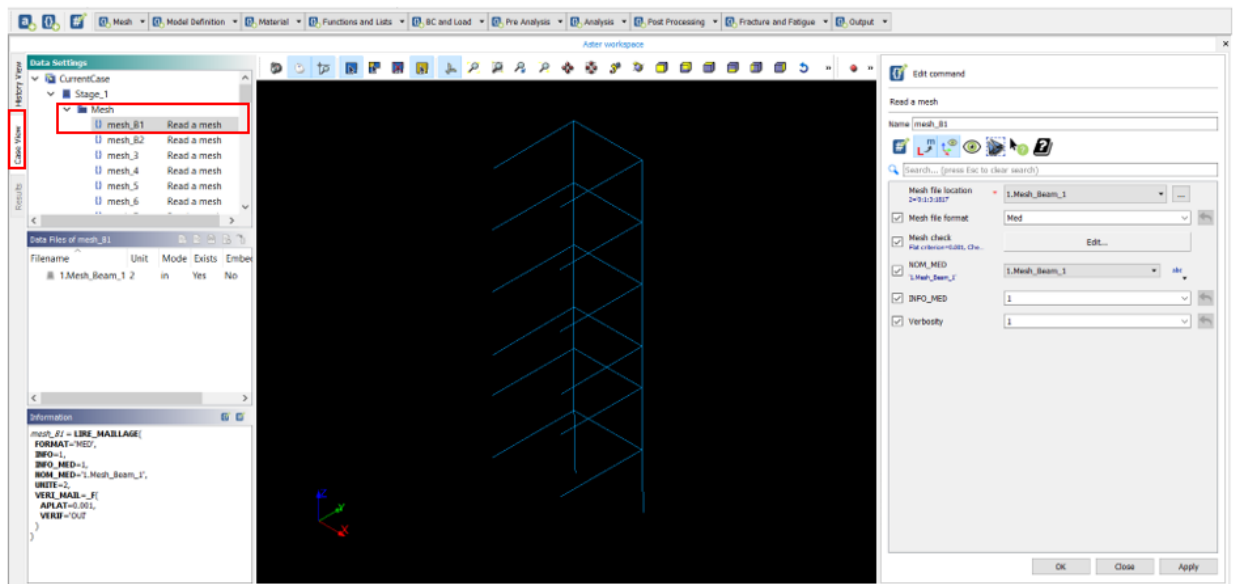


Figure 15: Reading Beam Mesh in code aster

2nd step to do is to read the mesh that we have created in already inside the mesh module. However, it is also import to read a various mesh format such as Gmesh, Gibi and aster.in figure 15 shows the beam mesh that has been read in code aster. In the similar manner all the meshes created for shell walls and slabs were read in code aster module. The *figure 16 below* shows the available possibilities to read a mesh:

([U4.21.01] Operator LIRE_MAILLAGE, n.d.)

1. Reading a Mesh ----> 'LIRE_MAILLAGE'(U4.21.01)

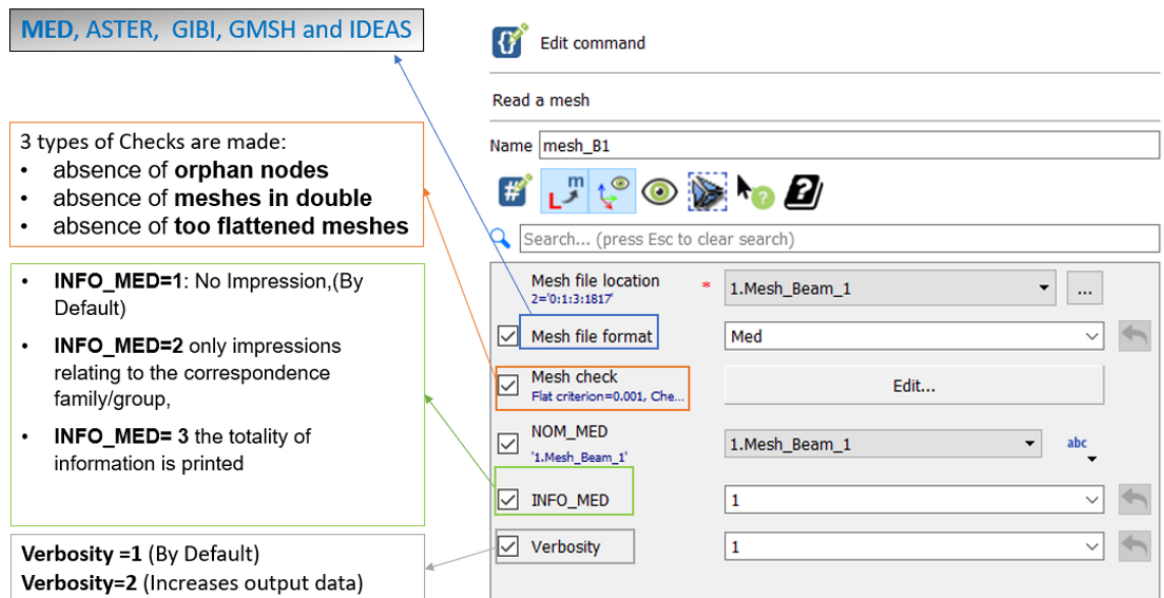


Figure 16: Reading a Mesh

Mesh file Formats

There are many formats in which File can be imported into aster study such as MED, ASTER, GIBI, GMSH and IDEAS. I Imported the mesh from .MED Format The format who means “Modeling and Data exchanges “is a neutral format of data for the data exchanges between computer codes. MED format is by default and advised in code ASTER. The *figure 17 below* shows all the possible file formats that can be imported and exported in code aster.

([U4.21.01] Operator LIRE_MAILLAGE, n.d.)

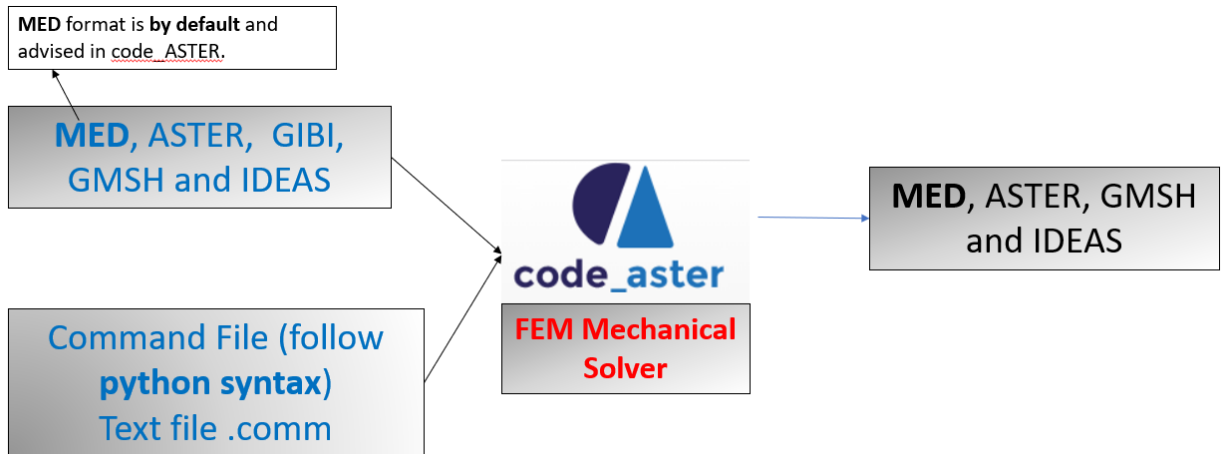


Figure 17: input and output file formats of code_Aster

Mesh Check Mesh Check

Edit command

Read a mesh > **Mesh check**

Name

☒ Check ☒ Yes

☒ Flat criterion 1.12

Figure 18: checks for mesh

The operand **VERI_MAIL** performs three types of checks on the grid:

- absence of orphan nodes (A node is considered as orphan if it does not contain any mesh connectivity).
- absence of mesh "in double",
- absence of too flattened mesh (the flatness of mesh is defined as ratio of shortest and longest of mesh)

By default, these three checks are made by code aster. However, it is also possible to avoid these checks in case if user wants to avoid these checks.

To check the quality standard of the imported grid operand **MACR_INFO_MAIL** is used.

INFO_MED Print information specific on the course of the second reading of the file of grid MED (number of nodes and meshes read again, information on families MED):


INFO_MED=1: No Impression, (By Default)

INFO_MED=2 only impressions relating to the correspondence family/group,

INFO_MED= 3 the totality of information is printed


Verbosity


There are 2 options for verbosity 1 or 2. By default the verbosity is 1 but if we choose 2 it will increase the file output file data.

 Edit command

Read a mesh

Name





Mesh file location
20='0:1:3:1818'

☒ Mesh file format
Check='Yes', Flat criterio...

☒ Mesh check
Check='Yes', Flat criterio...

☒ NOM_MED
'2.Mesh_Beam_2'

☒ INFO_MED

☒ Verbosity

2.Mesh_Beam_2

...

Med

↩

Edit...

2.Mesh_Beam_2

abc

▼


1

↩

1


↩


Figure 19: Reading Mesh of Roof RC Beam

 Edit command

Read a mesh

Name



 Search... (press Esc to clear search)

Mesh file location
3="

<3 undefined>

...

☒ Mesh file format

Med

↕

☒ Mesh check
Check='Yes', Flat criterio...

Edit...

☒ NOM_MED
"

abc

▼

☒ INFO_MED

1

↕

☒ Verbosity

1

↕

OK

Close

Apply

Figure 20: reading Mesh for Walls and slabs

Model Definition (Assign a Finite element)

Basically, there are **3 types** of elements used as follows:

1. **2 node Beam elements** which implies **Timoshenko Theory** used for RC beams and steel beams of elevator Frames
2. **4 node shell elements** with bilinear shape functions used for slabs and masonry walls.
3. A spring element is also used to model the uncertainties of the geometry and materials below the mezzanine floor (boundary conditions).
 - Square Hollow steel beam of 0.18m x 0.18m and 0.004m thick for elevator frame.
 - A RC Beam of depth 0.35m and width 0.30m for Roof.
 - Stairs are modelled as 0.2m thick plate.

Refer to document available on code aster website **Modelazion POU_D_T, POU_D_E, POU_C_T, Bar**

POU_D_T means *Straight* Beam of **Timoshenko** (Refer to Doc U3.11.01) takes into account the effect of transverse shearing. For modelling of beam each node in mesh is subjected to 6 degrees of freedom (three translations along the 3 axes and 3 rotations). The nodes represent the segment of average fiber of a beam.

Finite element	Degrees of freedom (with each node top)					
POU_D_T	DX	DY	DZ	DRX	DRY MARTINI	DRZ
POU_D_E	DX	DY	DZ	DRX	DRY MARTINI	DRZ
BAR	DX	DY	DZ			

The mesh associated with the finite elements of this type of beam is Beam with segment of 2-Nodes.

Modeling	Mesh	Finite element	Remarks
POU_D_T	SEG2	MECA_POU_D_T	
POU_D_E	SEG2	MECA_POU_D_E	
BAR	SEG2	MECA_BARRE	

There is no need to create the mesh discretization for the edge of elements of beam and bar as all the loads applied to elements of beam are taken as direct discretization of mesh.

POU_C_T means *Curved* Beam of **Timoshenko** (Refer to Docs on aster study)

POU_D_E means *Straight* Beam of **Eulerian** (Refer to Docs on aster study)

COQUE means shell in code Aster.

COQUE means shell in code Aster. ([U4.41.01] Operator AFPE_MODELE, n.d.). the *figure 20* shows how to assign a finite element to the mesh:

2. Assign a FEM---> 'AFPE_MODELE' (U4.41.01)

Operands	Contents/significance
ALL	Assignment with the totality of the meshes
GROUP MA	Assignment with a list of groups of meshes

3 types of Phenomenon:

Acoustic

Mechanic

Thermic

- 2-Node Beam Element for Beam -----> **POU_D_T** (U3.11.01)
- 4 Node Shell Element for wall and slab-----> **COQUE_AXIS** (U3.12.02)

Keyword	Modeling
BAR	BAR, CABLE GAINE
CABLE	CABLE, CABLE POULIE
HULL	COQUE_AXIS, COQUE_C_PLAN, COQUE_D_PLAN, DKT, DST, DKQ, DSQ, Q4G, COQUE_3D, DKTG, Q4GG
DISCRETE	DIS T, DIS TR, 2D DIS T, 2D DIS TR
BEAM	POU_D_E, POU_D_T, POU_D TG, POU_D T GD, FLUI STRU, TUYAU 3M, TUYAU 6M, POU_D EM, POU_D TGM
SOLID MASS	3D, AXIS, AXIS FOURIER, C PLAN, D PLAN, TUYAU 3M, TUYAU 6M
GRID	GRILLE EXCENTRE, GRILLE MEMBRANE
MEMBRANE	MEMBRANE
POUTRE FLUI	3D FAISCEAU
MULTI FIBRE	POU_D EM, POU_D TGM
RIGI PARASOL	DIS TR
RIGI_MISS_3D	DIS_T

Edit command

Assign finite element > Finite element assignment > [0]

Name: FEM_B1

Search... (press Esc to clear search)

Exactly One *

☒ Everywhere Yes

☐ Group of element Edit...


Phenomenon: Mechanic

Modelisation: 1 item

[0] POU_D_T









Modeling POU_D_T (Right Beam of Timoshenko)


Figure 20: Assigning a FEM to mesh

 Edit command

Assign finite element > Finite element assignment > [0]

Name



 Search... (press Esc to clear search)

Exactly One *

☐ Everywhere

Yes ▾


☒ Group of element

Edit...

Phenomenon *



Mechanic ▾


Modelisation *

▼ 1 item 

[0] *

POU_D_T ▾





OK

Abort

Cancel


Figure 21:Assigning FEM Beam_1 of Steel elevator Frame Mesh

It's also possible to give Length, temperature and pressure to the FEM model as shown in figure 22 as follows:

The screenshot shows a software interface for assigning characteristic values to a finite element model. At the top, there is a blue icon with a green checkmark and the text "Edit command". Below this is a horizontal line, followed by the text "Assign finite element > Characteristic value". Another horizontal line follows. Below this line is a text input field labeled "Name" containing the text "FEM_B1". Below the input field is a row of icons: a blue icon with a green checkmark, a blue icon with a red 'L' and a green arrow, a blue icon with a green eye, a blue icon with a green question mark, a blue icon with a green question mark, and a blue icon with a green question mark. Below the icons is a search bar with a magnifying glass icon and the text "Search... (press Esc to clear search)". Below the search bar is a table with three rows. Each row has a checkbox, a text label, a text input field, and a dropdown menu. The first row has a checkbox, the label "Length", an input field, and a dropdown menu showing "1.12". The second row has a checkbox, the label "Pressure", an input field, and a dropdown menu showing "1.12". The third row has a checkbox, the label "Temperature", an input field, and a dropdown menu showing "1.12".









Property	Value
<input type="checkbox"/> Length	1.12
<input type="checkbox"/> Pressure	1.12
<input type="checkbox"/> Temperature	1.12


Figure 22: giving properties to FEM

 Edit command

Assign finite element

Name






At Least One *

☐ Substructures ...



Edit...


☒ Finite element ...

▼ 1 item 

[0]

Everywhere='Yes'...






Mesh *

mesh_B1 ...AILLAGE) ▼

☒ Verbosity

1 ▼



☐ Characteristic v...

Edit...

☐ MPI distribution

Edit...

☐ Jacobian check

☒ Yes













Figure 22: Assigning FEM to RC Roof Beams

 Edit command

Assign finite element

Name









At Least One *

☐ Substructures assignement


☒ Finite element assignement

▼ 1 item 

[0]



Mesh *

☒ Verbosity 

☐ Characteristic value

☐ MPI distribution

☐ Jacobian check


☒ Yes 

Figure 24:FEM FOR SHELL

Defining section (Code Aster Document- U4.42.01)

For defining the section properties for beam and shell the following document as shown in figure 25 below:



Figure25: Document for defining section properties for FEM

Defining section for Beam

Defining section of beam 1 of hollow square section of 0.18m x 0.18m having thickness of 0.004m and detail of characteristics of H means length if side if its rectangular is given in the document from code aster website. The figure 26 below shows the section defined for Beam1 inside Salome Meca:

3. Section Definition---> 'AFEE_CARAR_ELEM'(U4.42.01)

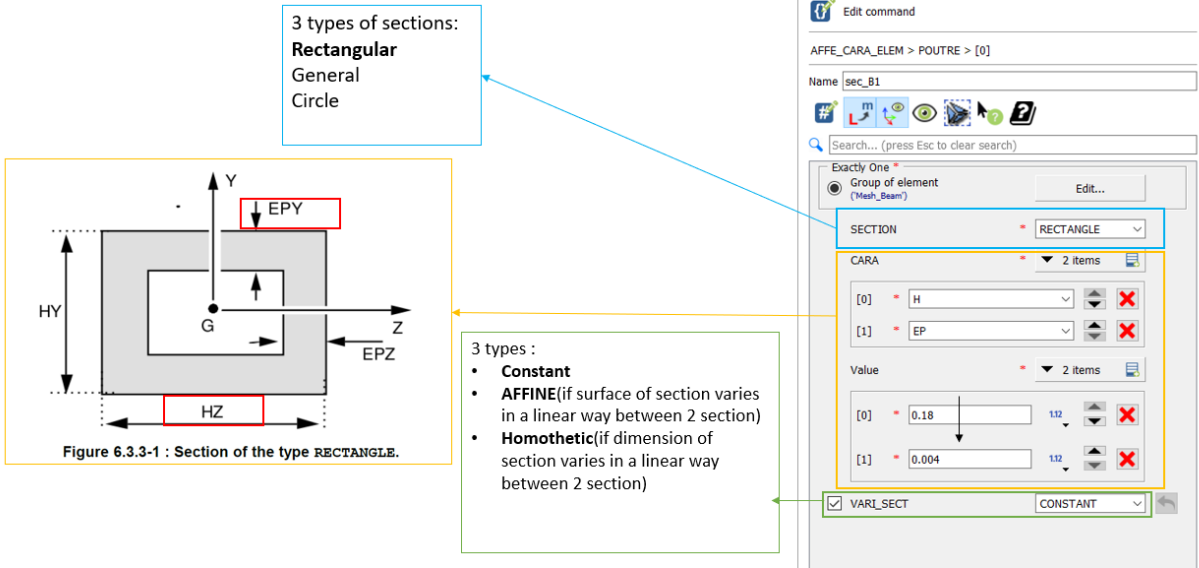


Figure26: section defined for Beam

6.3.3 Operand SECTION = 'RIGHT-ANGLED'

CARA	Significance	Value by default
/ HY	Dimension of the following rectangle GY	Obligatory
/ HZ	Dimension of the following rectangle GZ	Obligatory
/ H	Length of the edge (if the rectangle is square)	Obligatory
/ EPY	Thickness according to GY in the case of a hollow tube	HY / 2
EPZ	Thickness according to GZ in the case of a hollow tube	HZ / 2
/ EP	Thickness along the two axes in the case of a hollow tube	Full tube

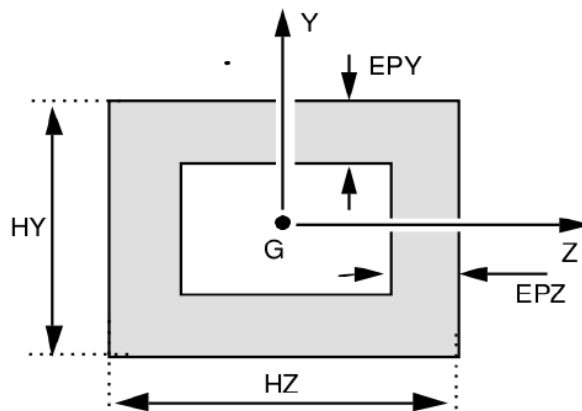


Figure 6.3.3-1 : Section of the type RECTANGLE.

Figure 27: section properties for rectangular section

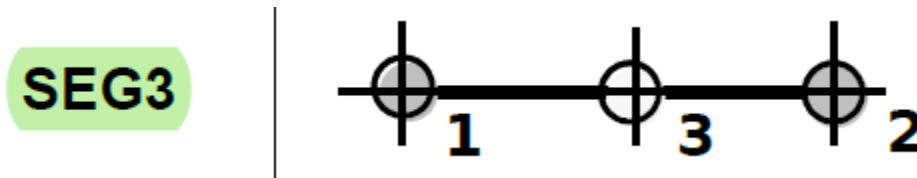
- Vari_section to see if section has constant cross section or not.

Section for shell

Walls and slabs are modelled has shell COQUE_AXIS with 3 degree of freedom two displacements and 1 rotation. The shell element of COQUE_AXIS was used for walls and labs of building as shown below:

Finite element		Degrees of freedom (with each node top)		
MECXSE3		DX	DY	DRZ
Modeling	Mesh	Finite element		Remarks
COQUE_AXIS	SEG3	MECXSE3		

The mesh supported for this kind of shell is SEG-3 as follows:



For application of load there is no need to define separate supports for loads as loads are applicable to direct discretization on mesh support of element in displacement formulation.

Model Definition----→ **AFFE_CARA_ELEM**

The operand used to give the section properties to the already defined elements is “**AFFE_CARA_ELEM**”. This assignment is done in two stages:

First, we select the type of elements concerned with the model such as beam or shell.

For Beam

Secondly the section properties are assigned to the meshes to be affected.

- It is possible to define characteristics of section by operand **"CARA"** and each characteristic is associated with a value given by the operand **"VALE"**.
- It's also possible to give characteristics to section using an operand **"LIRE_TABLE"** but table must follow a standard format.
- Dimensions of sections are given by operands 'CARA' and 'VALE' and code aster automatically calculates the mechanical characteristics such as surface and inertia of defined section.
- The operand **'VERI_SECT'** can be used to define the beam of constant or variable section.

"Refines" mean that the surface of the section varies in a linear way between the two nodes. Dimensions in the direction are constant there (HY , EPY) and that in the direction z vary linearly ($HZ1$, $HZ2$, $EPZ1$, $EPZ2$).

• **"Homothetic"** mean that dimensions of the section vary linearly between the values given to the two nodes end. In this case, the surface of the section evolves in a quadratic way. In the case as of circular hollow sections, so that the section is regarded as homothetic, it is necessary that $EPDEBUT / RDEBUT = EPFIN / RFIN$. In the case of not respect of homothety the solution

- given by *Code_Aster* is approximate [R3.08.01].
- For rectangle section beams it's possible to define following characteristics and assign value to them using the operand 'CARA' and 'VALE':

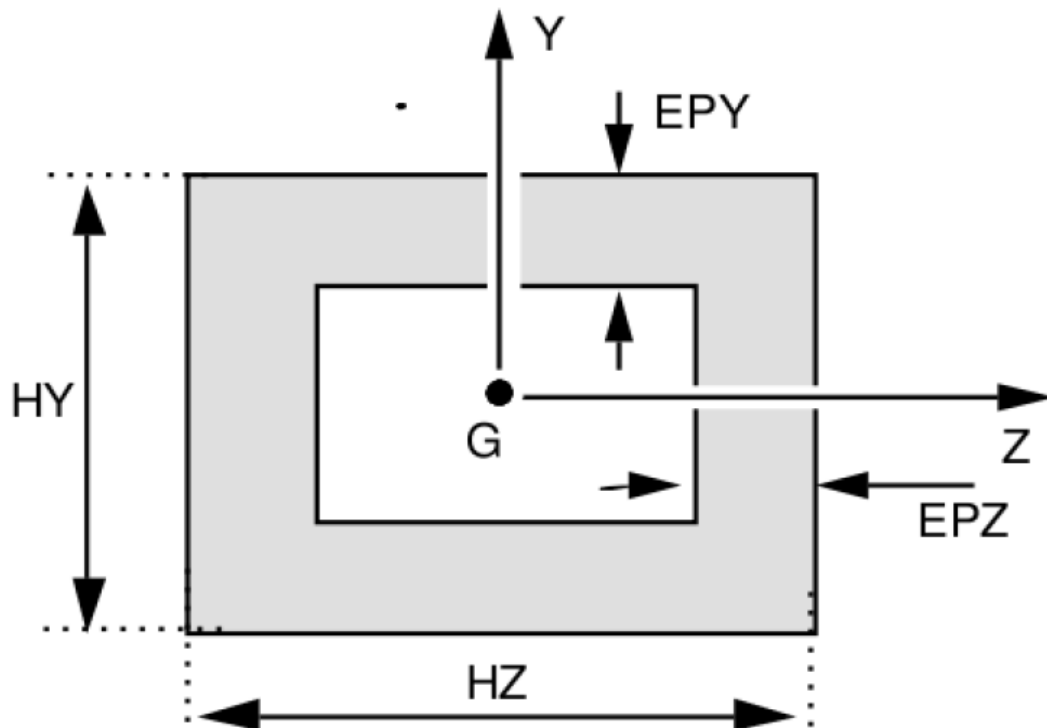


Figure 6.3.3-1 : Section of the type RECTANGLE.


For Shell

For shell following properties can be assigned to the elements:

- Thickness of shell
- The no of layers used in the integration of shell element
- Orientation of local axes









([U4.42.01] Operator AFFE_CARA_ELEM, n.d.)


the figures below show sections defined inside the Salome Meca:

 Edit command

AFFE_CARA_ELEM > POUTRE > [0]

Name



 Search... (press Esc to clear search)

Exactly One *


☒ Group of element
(*Mesh_Beam*)

Edit...

SECTION *


RECTANGLE ▾

CARA *

▼ 2 items 


[0] *

H ▾


▲▼

[1] *

EP ▾

▲▼


Value *

▼ 2 items 

[0] *

0.18


1.12 ▾

▲▼


[1] *

0.004

1.12 ▾

▲▼

☒ VARI_SECT

CONSTANT ▾


OK

Abort

Cancel








Figure 28: Hollow Section properties for Elevator steel Frame


38


 Edit command

AFFE_CARA_ELEM




Name








☒ POUTRE ▼ 1 item 


SECTION='RECT...





☐ BARRE ▶ 0 items 


☐ COQUE ▶ 0 items 


☐ CABLE ▶ 0 items 


☐ DISCRET ▶ 0 items 


☐ DISCRET_2D ▶ 0 items 


☐ MASSIF ▶ 0 items 

☐ GRILLE ▶ 0 items 

☐ MEMBRANE ▶ 0 items 

☐ RIGI_PARASOL ▶ 0 items 

☐ MASS_REP ▶ 0 items 


☐ MULTIFIBRE ▶ 0 items 

Model *

☒ Verbosity









Figure 29: Section_ assigned to FEM of Beam_1


Section properties of Beam 2

 Edit command

AFFE_CARA_ELEM > POUTRE > [0]

Name



Exactly One *

☒ Group of element
(*'2.Mesh_Beam_2'*)

Edit...


SECTION

*

RECTANGLE ▾


CARA

*

▶ 2 items 

Value

*

▶ 2 items 

☐ VARI_SECT


CONSTANT ▾ 

Figure 30:section FOR RC Roof Beams

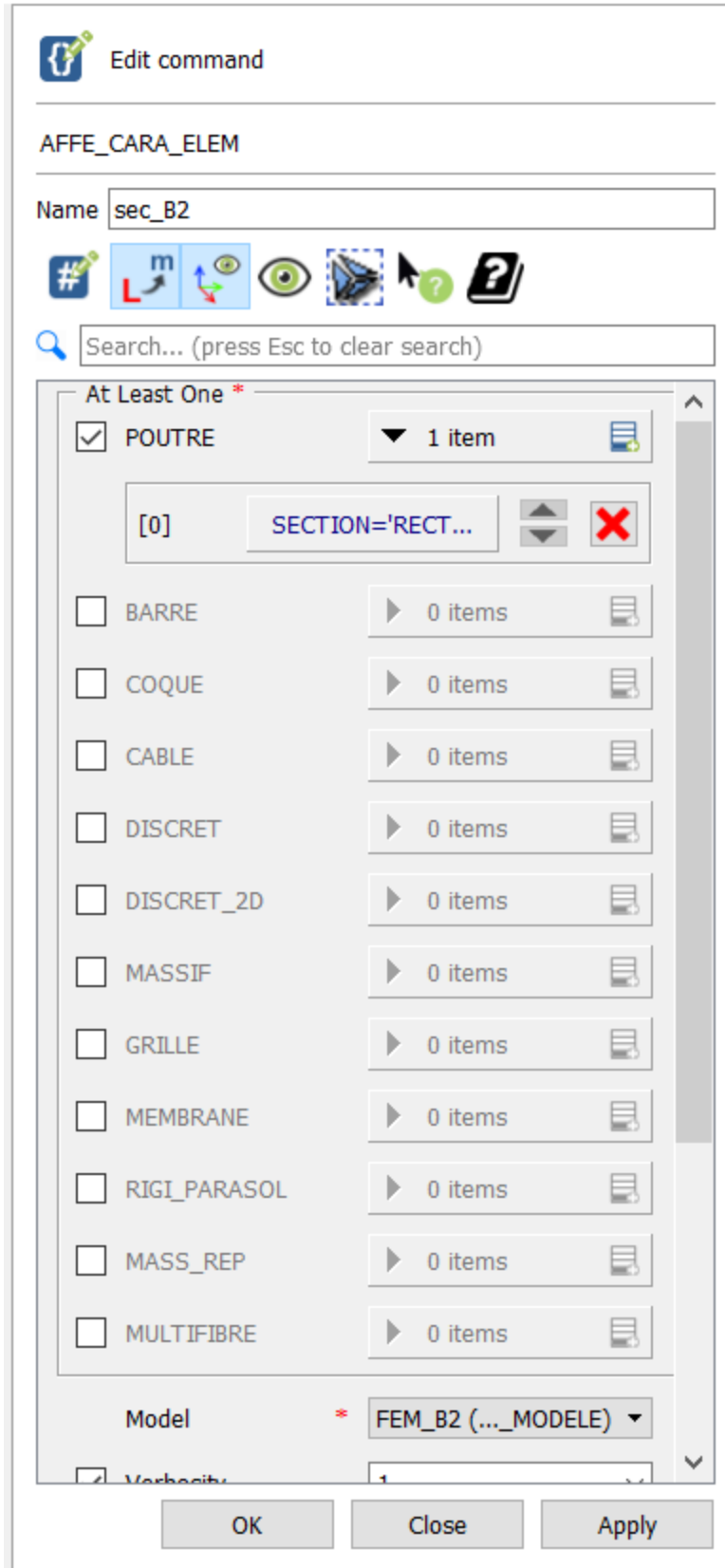











Figure 31 SEC_2 assigned to FEM_B_Beam-2

 Edit command

AFFE_CARA_ELEM > POUTRE > [0]

Name


      










Exactly One *







☐ Group of element

SECTION *

CARA * 

[0]	*	<input type="text" value="HZ"/>			
[1]	*	<input type="text" value="HY"/>			

Value * 

[0]	*	<input type="text" value="0.30"/>	<input type="text" value="1.12"/>			
[1]	*	<input type="text" value="0.35"/>	<input type="text" value="1.12"/>			










☒ VARI_SECT 


Figure 32

 Edit command

AFFE_CARA_ELEM > POUTRE > [0]

Name


      







 Search... (press Esc to clear search)


Exactly One *







☐ Group of element

SECTION *

CARA * 

[0]	*	<input type="text" value="HY"/>			
[1]	*	<input type="text" value="HZ"/>			

Value * 

[0]	*	<input type="text" value="0.35"/>	1.12			
[1]	*	<input type="text" value="0.3"/>	1.12			




☐ VARI_SECT 


Figure 33: section for Beam 2

 Edit command

AFFE_CARA_ELEM > COQUE > [0]

Name





Exactly One *

☒ Group of element
(*3.Mesh_shell_scale*)

Edit...

Exactly One *

☒ EPAIS 1.12

☐ EPAIS_FO <no fonction_s...ule available>

☐ ANGL_REP ▶ 2 items

☐ VECTEUR ▶ 3 items

☐ A_CIS 1.12

☐ COEF_RIGI_DRZ 1.12

☐ COQUE_NCOU 123

☐ EXCENTREMENT 1.12

☐ EXCENTREMEN... <no fonction_s...ule available>

☐ INER_ROTAT


☐ MODI_METRIQUE ☐ No

OK

Abort









Cancel


Figure 34: Section of stairs as shell having thickness of 0.2m

 Edit command
















AFFE_CARA_ELEM

Name



At Least One *


<input type="checkbox"/>	POUTRE	▶ 0 items	
<input type="checkbox"/>	BARRE	▶ 0 items	
<input checked="" type="checkbox"/>	COQUE	▼ 1 item	
<div>[0] <input type="text" value="Group of element..."/>   </div>			
<input type="checkbox"/>	CABLE	▶ 0 items	
<input type="checkbox"/>	DISCRET	▶ 0 items	
<input type="checkbox"/>	DISCRET_2D	▶ 0 items	
<input type="checkbox"/>	MASSIF	▶ 0 items	
<input type="checkbox"/>	GRILLE	▶ 0 items	
<input type="checkbox"/>	MEMBRANE	▶ 0 items	
<input type="checkbox"/>	RIGI_PARASOL	▶ 0 items	
<input type="checkbox"/>	MASS_REP	▶ 0 items	
<input type="checkbox"/>	MULTIFIBRE	▶ 0 items	

Model *

☐ Verbose


OK Close Apply


Figure 35: section assigned to FEM_S1S1S1S1Shell-1 of stairs

 Edit command

AFFE_CARA_ELEM > COQUE > [0]

Name





Exactly One *

☒ Group of element
(4.Mesh_Infill_walls)

Edit...

Exactly One *

☒ EPAIS 1.12

☐ EPAIS_FO <no fonction_s...ule available>

☐ ANGL_REP 2 items

☐ VECTEUR 3 items

☐ A_CIS 0.8333333 1.12

☐ COEF_RIGI_DRZ 1e-05 1.12

☐ COQUE_NCOU 1 123

☐ EXCENTREMENT 1.12

☐ EXCENTREMEN... <no fonction_s...ule available>

☐ INNER_ROTA Yes


☐ MODI_METRIQUE ☐ No

OK

Abort








Cancel


Figure 36:Section for Infill walls

 Edit command
















AFFE_CARA_ELEM

Name



 Search... (press Esc to clear search)

At Least One *

<input type="checkbox"/>	POUTRE	▶ 0 items	
<input type="checkbox"/>	BARRE	▶ 0 items	
<input checked="" type="checkbox"/>	COQUE	▼ 1 item	
<div>[0] <input type="text" value="Group of element..."/>   </div>			
<input type="checkbox"/>	CABLE	▶ 0 items	
<input type="checkbox"/>	DISCRET	▶ 0 items	
<input type="checkbox"/>	DISCRET_2D	▶ 0 items	
<input type="checkbox"/>	MASSIF	▶ 0 items	
<input type="checkbox"/>	GRILLE	▶ 0 items	
<input type="checkbox"/>	MEMBRANE	▶ 0 items	
<input type="checkbox"/>	RIGI_PARASOL	▶ 0 items	
<input type="checkbox"/>	MASS_REP	▶ 0 items	
<input type="checkbox"/>	MULTIFIBRE	▶ 0 items	

Model *


☐ Verbosity

OK

Close


Apply


Figure 37

 Edit command

AFFE_CARA_ELEM > COQUE > [0]

Name





Exactly One *

☒ Group of element


Exactly One *


☒ EPAIS 1.12


☐ EPAIS_FO <no fonction_s...ule available>

☐ ANGL_REP

☐ VECTEUR

☐ A_CIS 1.12 


☐ COEF_RIGI_DRZ 1.12 

☐ COQUE_NCOU 123 

☐ EXCENTREMENT 1.12

☐ EXCENTREMEN... <no fonction_s...ule available>

☐ INER_ROTAT

☐ MODI_METRIQUE ☐ No 

OK

Abort

Cancel

Figure 38: section defined for Inclined Roof

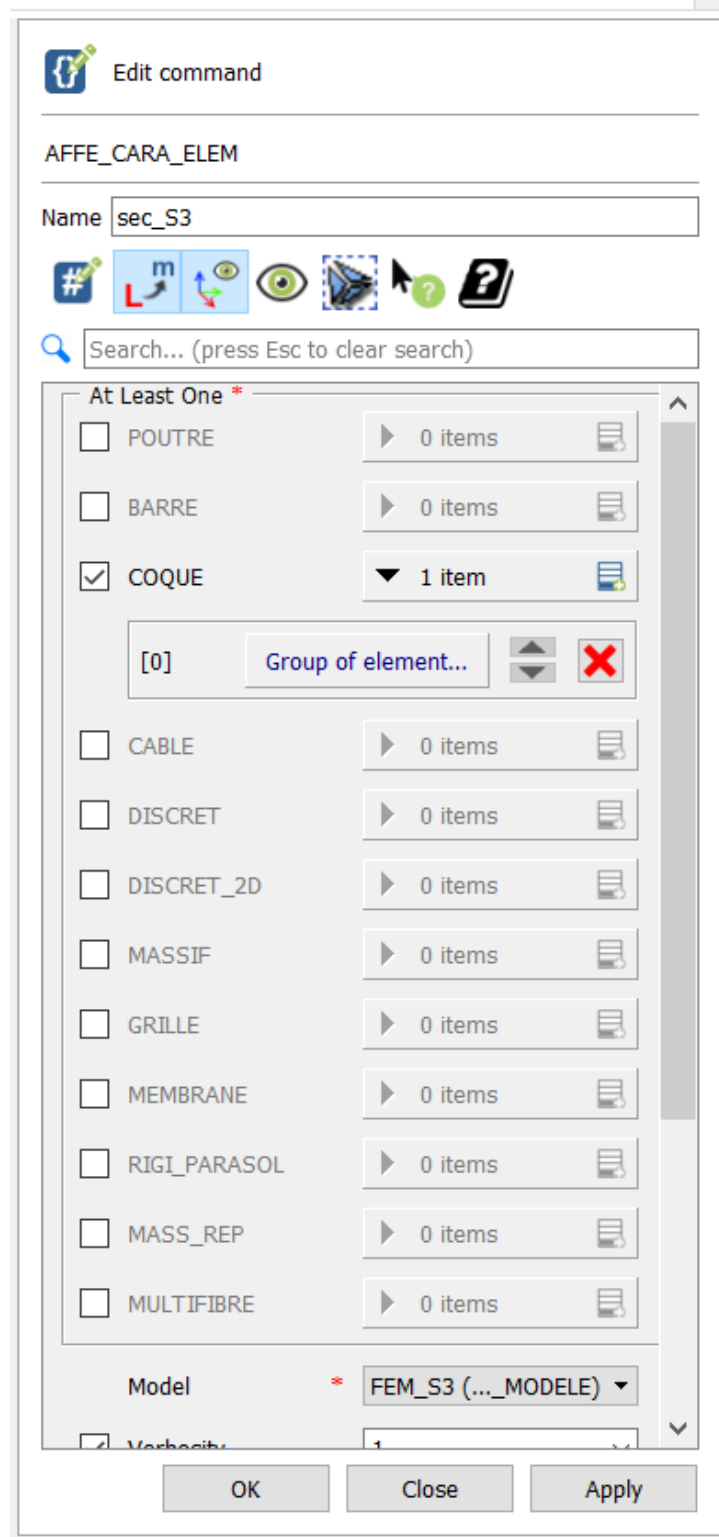


Figure 39: Section assigned to the FEM of inclined Roof

In similar manner section were defined and then assigned to the for all types of walls and slabs in our masonry building inside Salome Meca.

4. Material Properties Definition [\(U4.43.01\)](#)

The operator used to define the material is '**DEFI_MATERIAU**'. There are 26 different types of materials that can be defined in aster code.



Figure 3: Document on code_aster for Material Properties Definition

It's possible to model the hydration and drying of concrete, metallurgical for modelling of steel and modelling of porous environments associated with certain soil mechanics phenomenon.

The keyword '**ELAS**' and '**ELAS_FO**' are used to define constant linear characteristics. All the materials were defined as linear elastic for the modal analysis initially. The following *figure 40* shows the material properties defined for beam type-1:

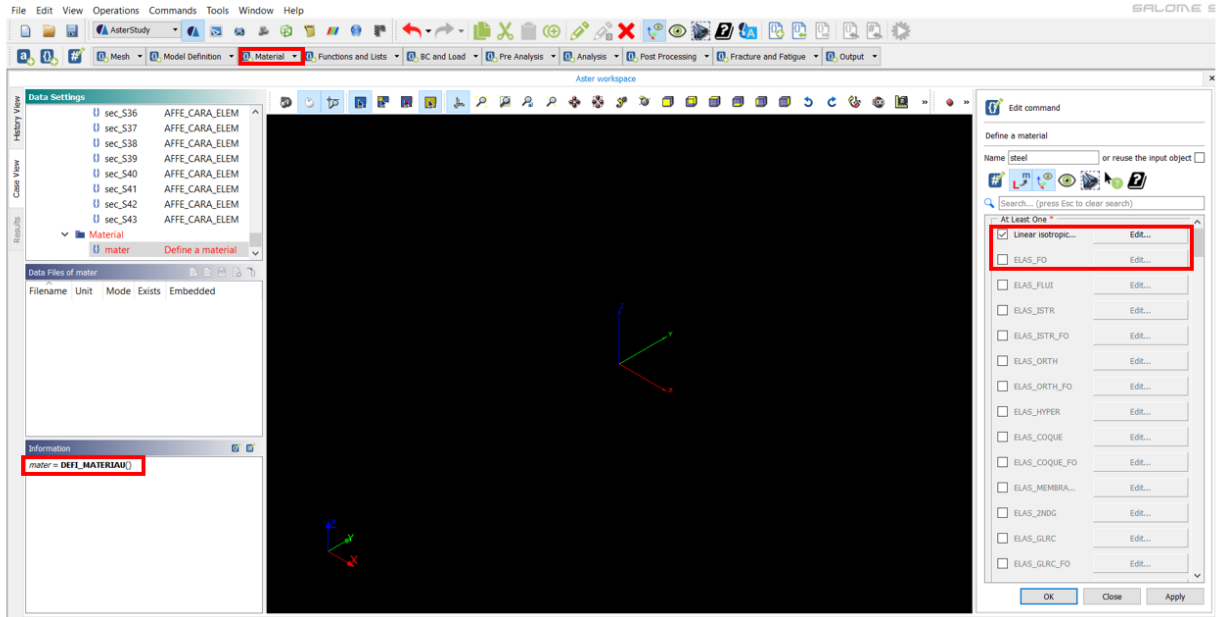


Figure 40: Definition of Material for Beam Type_1

Code aster makes following types of checks on the material properties automatically:

- The operand 'E' is used to define the young modulus of material and it's checked that value of $E \geq 0$.
- The operand 'Nu' is used to define the poison ratio of material and it's checked that value of $0.1 \leq \nu \leq 0.5$.
- The operand 'rho' is used to define the density of material, no check on density of material is defined.
- 11 different types of material were defined: in my model of masonry structure:

([U4.41.01] Operator AFFE_MODELE, n.d.)

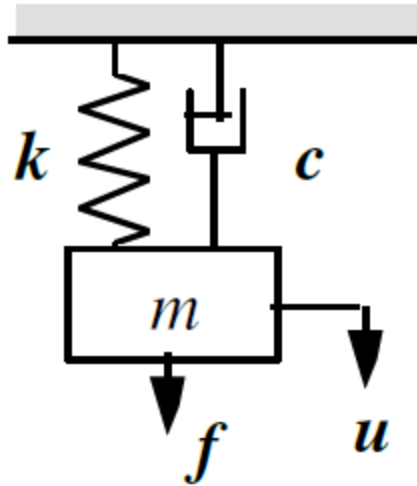
Modeling of damping in dynamics (U2.06.03)

Linear (R5.05.04)

In aster study there are two models of damping available:

- **Viscous damping model** in which the energy dissipated is proportional to the speed of movement

$$E_d = \frac{1}{2} \mathbf{v}^T \mathbf{C} \mathbf{v} = \frac{1}{2} \mathbf{u}^T \mathbf{C} \mathbf{u}$$



The newtons 2nd law is applicable:

$$m \ddot{u} + c \dot{u} + ku = f$$

Where C is the matrix of viscous damping, m is matrix of mass and k is matrix of stiffness.

Rayleigh method in which viscous damping involves linear combination of mass and stiffness through coefficients α and β is applicable:

$$C = \alpha K + \beta M$$


The exact values of coefficient α and β is difficult to find and can be found by taking the two model damping ratios for each mode:

$$2 \xi = \alpha \omega + \frac{\beta}{\omega}$$

The most optimum value of coefficient α and β are taken to consider the value of modal damping which brings resonance behavior in the structure. There are 2 ways of defining damping of materials:









- Either it's possible to assign the damping to the discrete elements of meshes
Or to define in the material as can be seen in the *figure 41* below:
- **Model of Damping hysteresis** in which the damping is proportional to displacement and force of damping. Damping hysteresis can be used suitably for the structure having viscoelastic behavior.


For transitory and harmonic analysis, the model of viscous damping can be used while for the model of damping hysteresis can only be used for harmonic analysis.

 Edit command

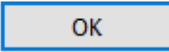
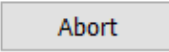
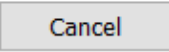
Define a material > Linear isotropic elasticity

Name or reuse the input object ☐

 Search... (press Esc to clear search)

Young's modulus	*	<input type="text" value="210e9"/>	1.12
Poisson's ratio	*	<input type="text" value="0.27"/>	1.12
<input checked="" type="checkbox"/> Density		<input type="text" value="7850"/>	1.12
<input type="checkbox"/> Thermal expansion...		<input type="text"/>	1.12
<input type="checkbox"/> Stiffness damping		<input type="text"/>	1.12
<input type="checkbox"/> Mass damping		<input type="text"/>	1.12
<input type="checkbox"/> Hysteretic damping		<input type="text"/>	1.12
<input type="checkbox"/> Characteristic length		<input type="text"/>	1.12
<input type="checkbox"/> Damping coefficient		<input type="text" value="1.0"/>	1.12

Coefficient α
(AMOR_ALPHA)

Coefficient β
(AMOR_BETA)

Damping
coefficient
hysteresis η
(AMOR_HYST)

damping
coefficient ζ
(COEF_AMOR)

Figure 41: Steel Material defined for Elevator Frame Beams

([U2.06.03] Modeling the mechanical damping, n.d.)

([R5.05.04] Modelling of linear dynamic damping, n.d.)

After that all the materials were defined as can be seen in *figure 42*, it shows like this in the information:

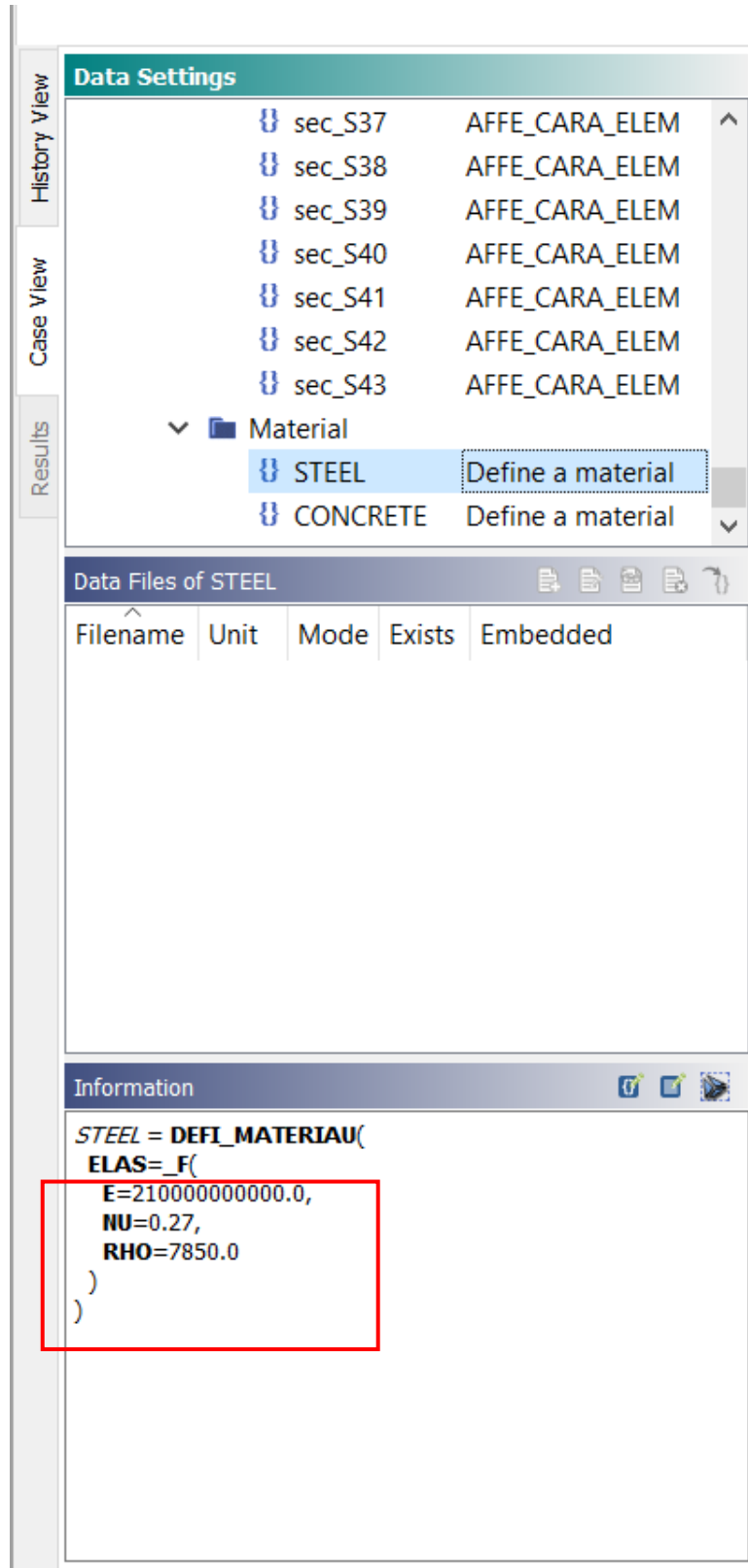











Figure 42

 Edit command

Define a material > Linear isotropic elasticity

Name or reuse the input object ☐



 Search... (press Esc to clear search)

Young's modulus	<input type="text" value="30e9"/>	1.12
Poisson's ratio	<input type="text" value="0.25"/>	1.12
<input checked="" type="checkbox"/> Density	<input type="text" value="2500"/>	1.12
<input type="checkbox"/> Thermal expansion...	<input type="text"/>	1.12
<input type="checkbox"/> Stiffness damping	<input type="text"/>	1.12
<input type="checkbox"/> Mass damping	<input type="text"/>	1.12
<input type="checkbox"/> Hysteretic damping	<input type="text"/>	1.12
<input type="checkbox"/> Characteristic length	<input type="text"/>	1.12
<input type="checkbox"/> Damping coefficient	<input type="text" value="1.0"/>	1.12

Figure 43: Concrete material defined for RC Roof beams

5. Assign Material properties (U4.43.03)

The operand used to give the material properties is “**AFPE_MATERIAU**”

Each material has certain parameter which are in turn associated to number of variables we call these variables as “variables of order”. Each mesh can be affected by 1 type of material.

The operand ‘NOM_VARC’ is used to give properties to a material. following table shows different kind of properties that can be assigned to a material:

TEMP	temperature
GEOM	directions of space
CORR	corrosion of steels
EPSA	unelastic deformation
HYDR	hydration of the concrete
IRRA	Irradiation
M_ACIER	metallurgical phases of steel
M_ZIRC	metallurgical phases of the zircaloy
NEUT1	“neutral” variable 1: the material coefficients of materials according to a parameter “user allows to vary” (see example 3 below)
NEUT2	“neutral” variable 2 (like NEUT1)
NEUT3	“neutral” variable 3 (like NEUT1)
SECH	drying of the concrete
PTOT	Total pressure of fluid in THM (chained resolution)
DIVU	Voluminal deformation in THM (chained resolution)

time varying material properties

- If some material property changes with time, then a keyword ‘EVOL’ must be used with that variable.

1st Define Material give E, v and Density and then assign the material from the material tab:



S287972

Thesis Report

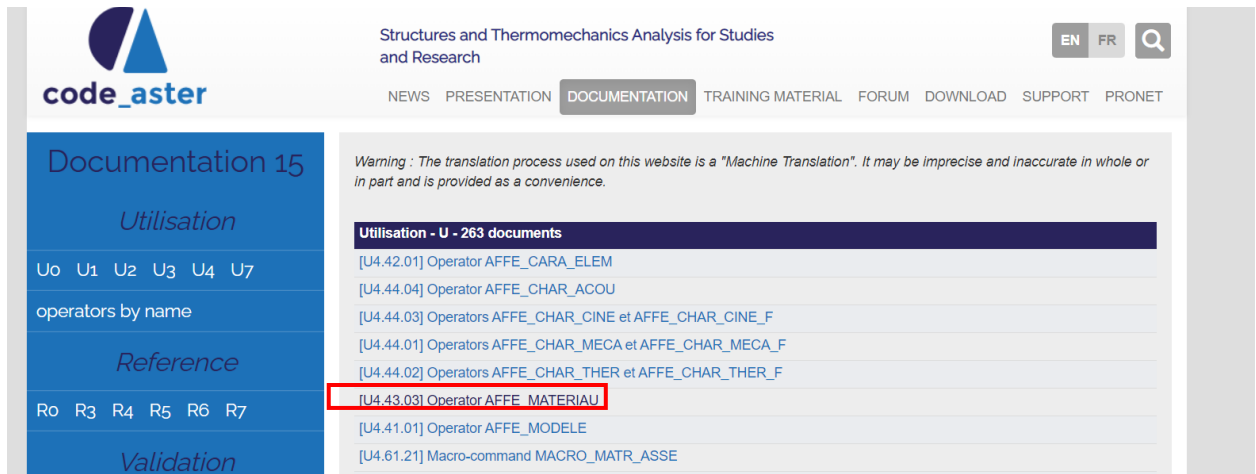


Figure 44: Document on code_aster website for assigning material properties

([U4.43.03] Operator AFFE_MATERIAU, n.d.).

The following figures below shows the material properties that can either be assigned to the mesh or to the finite element that we have already defined previously:

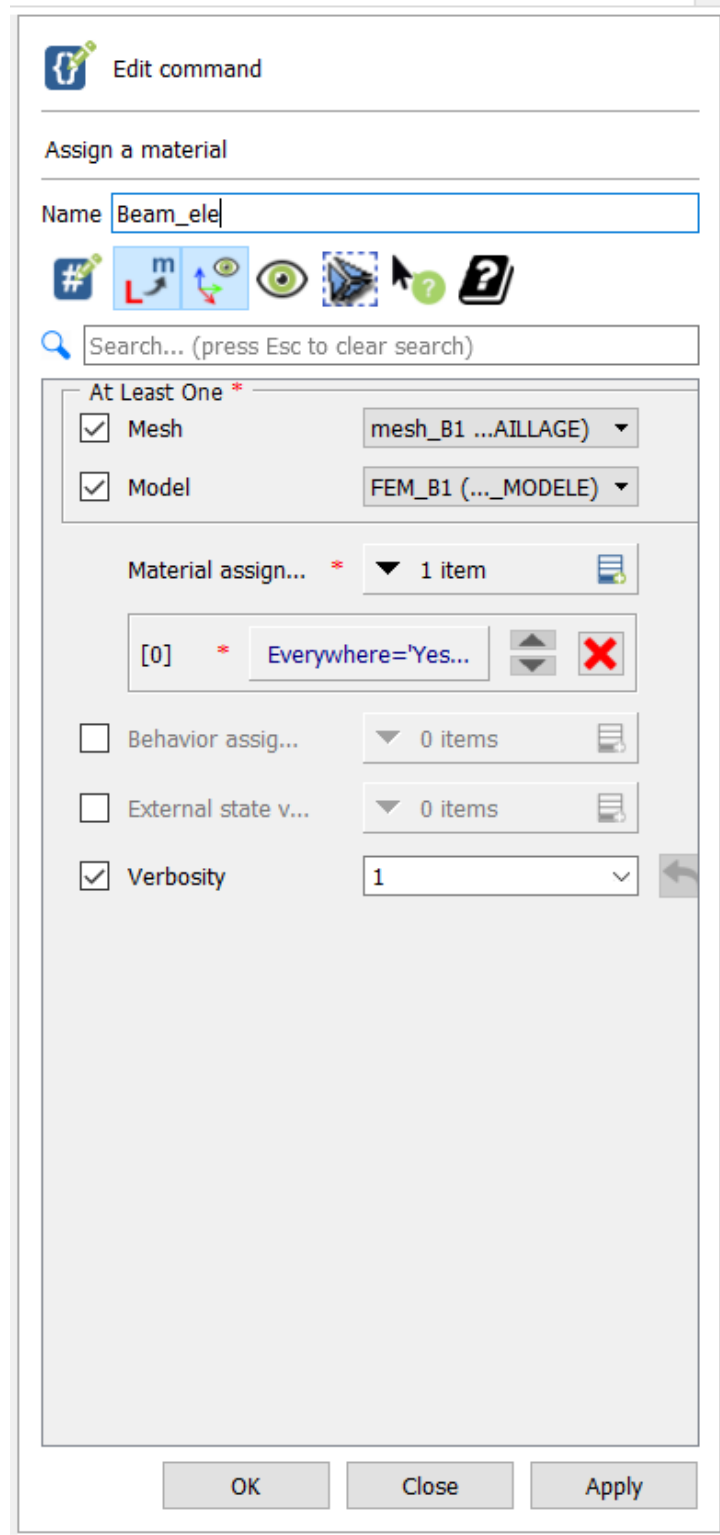


Figure 45: Steel material properties assigned to Elevator Frame Beams

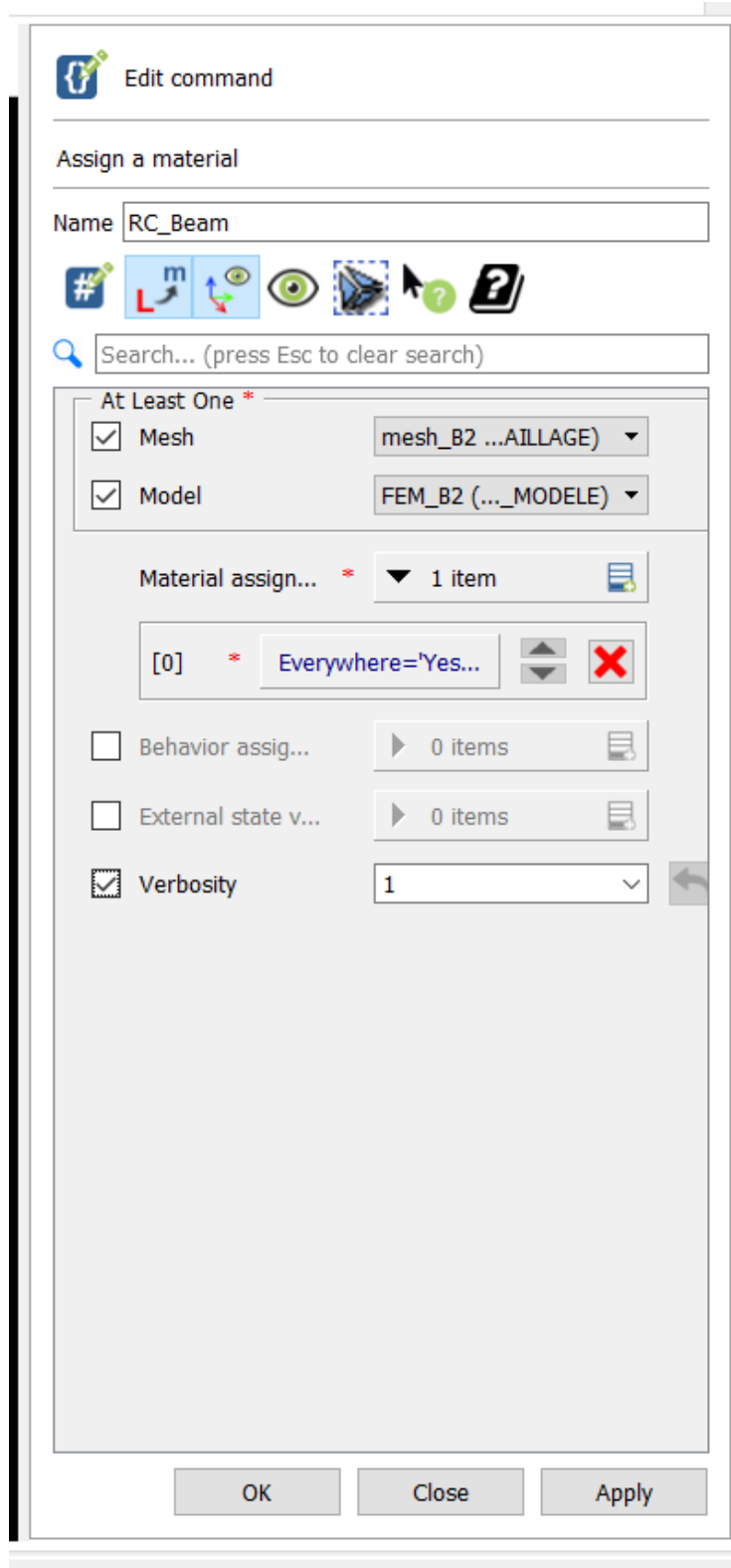


Figure 46: RC Concrete assigned to Roof Beams

6. Boundary Conditions and Loading

All the supports were assigned as fixed for the foundation. In code aster it is Fixed support means is denoted by Castre and Liason means Support. The *figure 47* below shows the fixed supports assigned to the foundation:

Select nodes to whom you want to assign fixed support.

For applying loads Force_

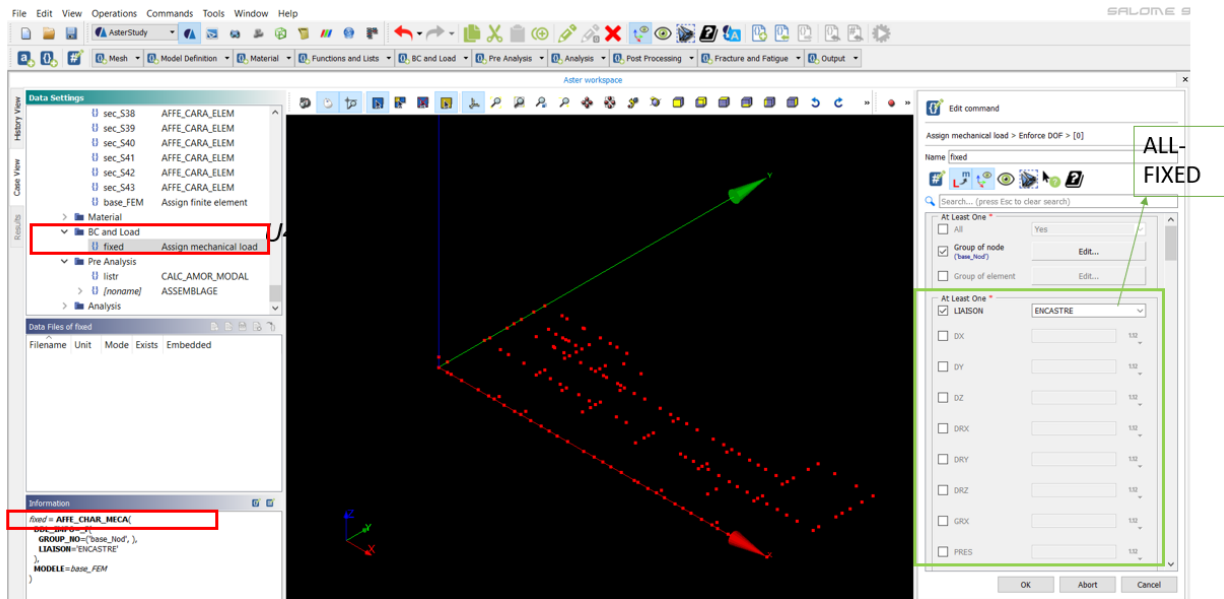


Figure 47: Fixed support assigned to the foundation

([U4.44.01] Operators AFFE_CHAR_MECA et AFFE_CHAR_MECA_F, n.d.)

7. Pre-Analysis

You have to calculate mass and stiffness 1st in order to perform modal analysis in Pre-Analysis module You have to calculate mass and stiffness 1st in order to perform modal analysis in Pre-Analysis module You have to calculate mass and stiffness 1st in order to perform modal analysis in Pre-Analysis module You have to calculate mass and stiffness 1st in order to perform modal analysis in Pre-Analysis module For carrying out the Modal analysis I 1st calculated the mass and stiffness of the system 1st in order to perform modal analysis in Pre-Analysis module the steps followed for calculation of mass and stiffness of system was follows:

Pre-analysis→ Assemblage→ Mass_Meca(for mass matrix) --→ Rigi_Meca (For Stiffness Matrix)

the *figure 48* below shows the steps followed for calculation of mass and stiffness' of the system:

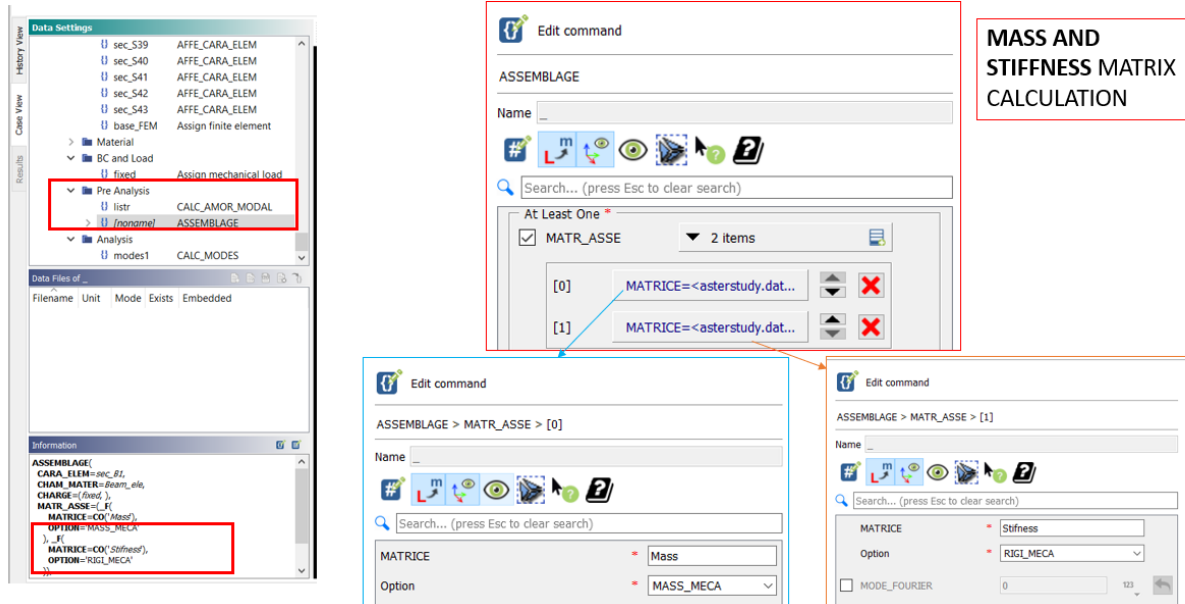


Figure 48: Calculation of Mass and stiffness of structure

8. Analysis Module (Modal analysis)

Calu_Modes is command to perform modal analysis

Choose type of analysis and apply boundary conditions and load if any

There are **26 different types** of materials on the document available on website

The 2 Node beam (Beam-188) has Following Properties:

For every type of material, you have to define these properties:

- As Wooden Roof was replaced by RC Roof so Roof can be modelled as Added mass to take into account in model.

Calu_Modes is command to perform modal analysis inside the code aster.

([U4.52.02] Operator CALC_MODES, n.d.)

1st four modes of vibration were calculated on this model the steps followed in Salome meca are as follows:

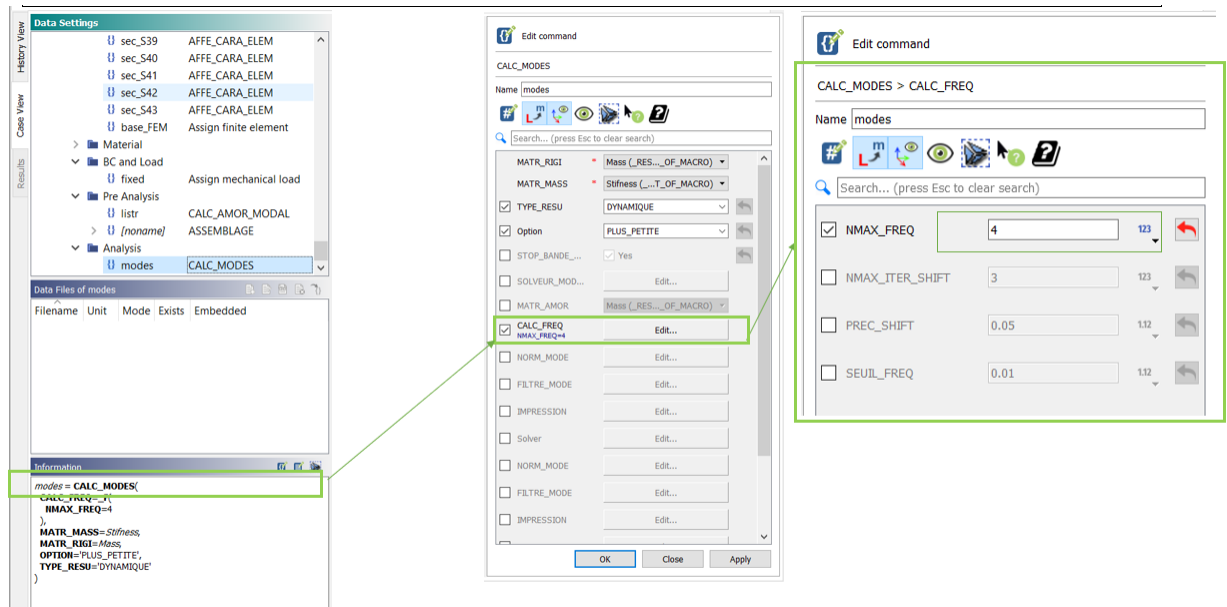


Figure 49: Modal analysis steps inside Salome Meca

After that in order to run analysis, we have to go to history view and run the analysis as can be seen in figure 50 below:

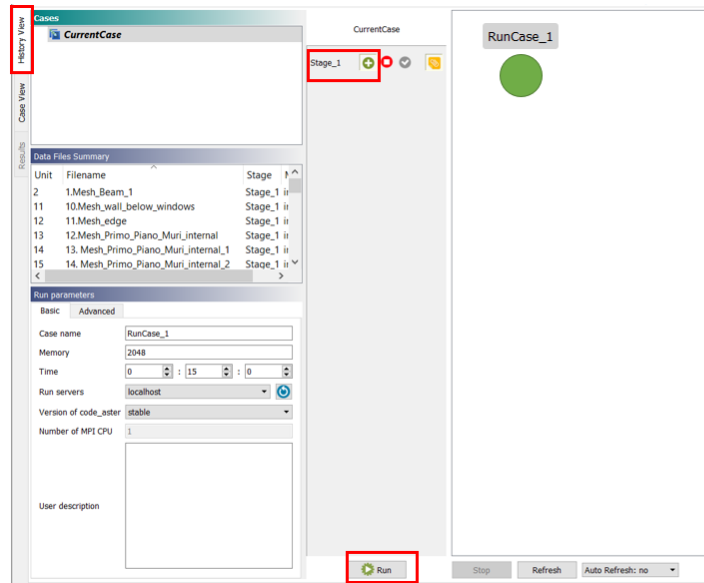


Figure 50: Running Modal analysis inside salome Meca

9. Post processing

After running analysis to view results we have to go to paravis or right click on data file summary → post process.

The results were showed in terms of max displacements and 1st four vibration modes in paravis view inside Salome Meca as can be seen in the *figure 51 and figure 52* below:

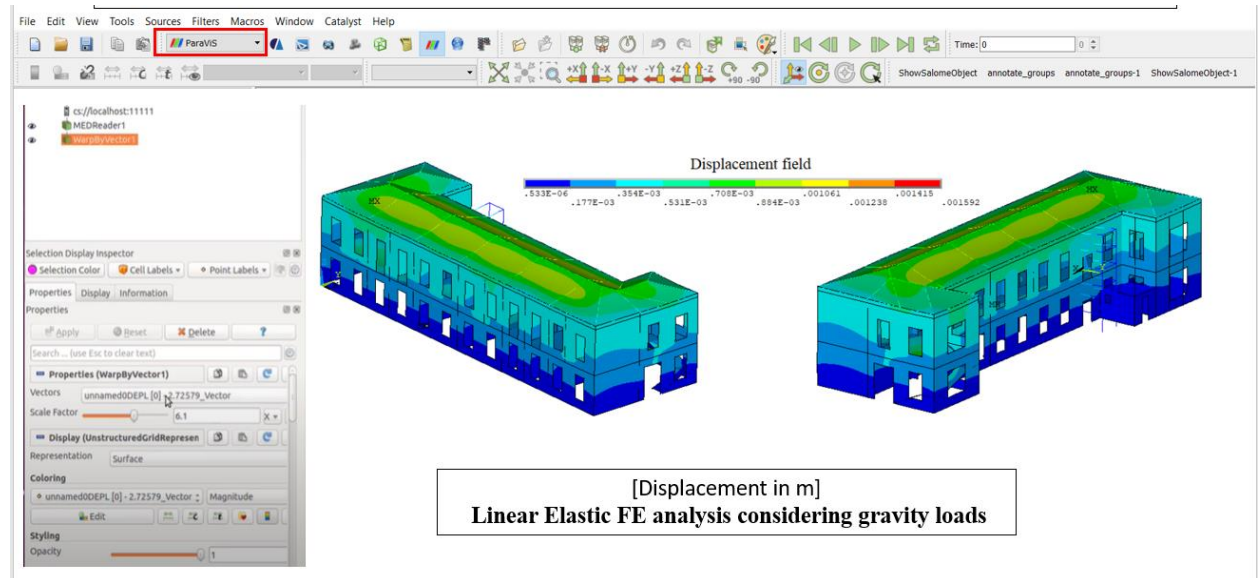


Figure 51: Maximum Displacement of Masonry Building

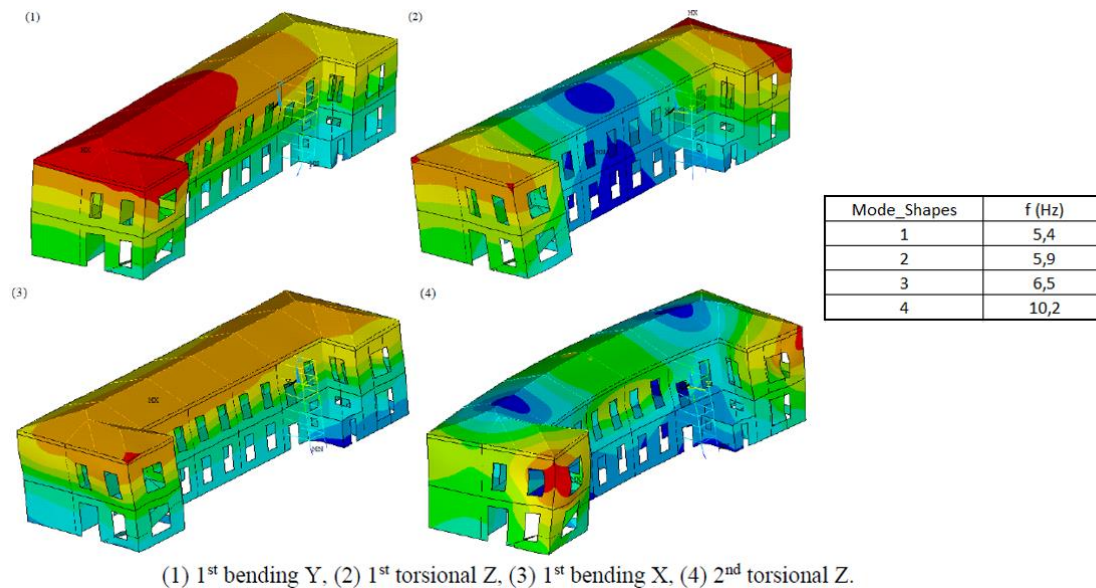


Figure 52: Modes of Vibration of Building

The frequency and Time period of vibration of first 4 modes were measured as can be seen **Table** below:

	Salome Meca	
Mode_Shapes	f [Hz]	T [s]
1	5.4	0.19
2	5.9	0.17
3	6.5	0.15
4	10.2	0.10

Table-1: Salome Meca modes of vibrations

3MURI

Introduction

3Muri is a leader software particularly designed for analysis and design of Masonry and mixed structures. It was developed by **prof. Sergio Lagomarsino** in university of Genova in ITALY. It's possible to carry out the global analysis or part of it in a fast and affordable way. Furthermore, software also identifies the critical areas by performing the sensitivity analysis from seismic point of view and important for retrofitting intervention from seismic point of view. It's also possible to have an estimate of cost of intervention.

3Muri uses following codes:

- **NTC** (norme tecniche per le costruzioni Italian Code)
 - **Eurocode-6 and Eurocode-8**
 - **SIA** (Switzerland code)
- (3Muri - Presentazione breve, 2017)

The software allows to perform structure analysis of masonry structures. The software performs the non-linear pushover analysis specialized for masonry structures. 3-muri can also perform analysis according to Eurocodes EC-6, EC-8 and Italian national code (NTC) and Switzerland and Netherlands national code.

It's possible to model floors with their real stiffness that allows the true distribution of horizontal seismic forces on the walls. There is different kind of library of types of walls that can be modelled and also there is option to model wall with or without reinforcement. It's possible to model reinforced masonry and masonry reinforced with fiber reinforced polymer FRP/FRCM. The mesh used for masonry structures is made of Frame by Macro elements that the software generates automatically by 3Muri software. Using this automatic mesh, it becomes very easy and fast to move from real structure to model.

It's also possible to retrofit using FRP or carbon reinforced polymer (CFRP) in walls that are failing. Using 3-Muri it's also possible to perform pushover analysis of a single wall. In 3-Muri it's also available to reinforce with steel to the walls or floors of masonry structure.

Background of 3Muri Project

The objective of 3Muri project is to have a **single integrated model for both Local and Global analysis** for masonry elements. The types of interventions mostly effect a single element and involve local verification. But sometimes it can also change the global behavior significantly. As it's recommended inside the code NTC-18 that the local interventions should not significantly change the global behavior of structure due to local interventions applied, but there is always an uncertainty associated with this fact and sometimes local interventions change the global behavior significantly. It gave rise to 3Muri project which make it possible to perform both global and local analyses on a single model. It goes without saying that the global verifications

require that there no local failure in our structure. So, there are 2 types of analysis that can be performed on as single model:

- **Global analysis** with Frame by Macro elements
- **Local analysis** (open elements)

So, approach of 3Muri project is to have to use a single integrated model for both local and global verifications as can be seen in *figure 53* below:






NTC 18 – CAP. 8	ELEMENTI SINGOLI	Global model
Types of Intervention	OPEN	FME <small>(Frame by Macro elements)</small>
Local Interventions		
improvements		
Adjustments		

Figure 53: 3Muri Analysis approach

The software divides the systems in 4 major categories as can be seen in *figure 54* below:

- **Simple structures** (in the calculation model is easy to model and in which cause and effect are evidently known)
- **complicated structures** (in which there are difficulties but the model is well defined and well determined, calculation models but there are assembly of simple structures for example a steel structure)
- **complex structures** (Most of Masonry structures lies in this area in which there is uncertainty in the modelling because of damage caused to the structure with the passage of time or there is uncertainty with the level of knowledge we have)
- **Caotico structure** (the cause and effects are unknown).

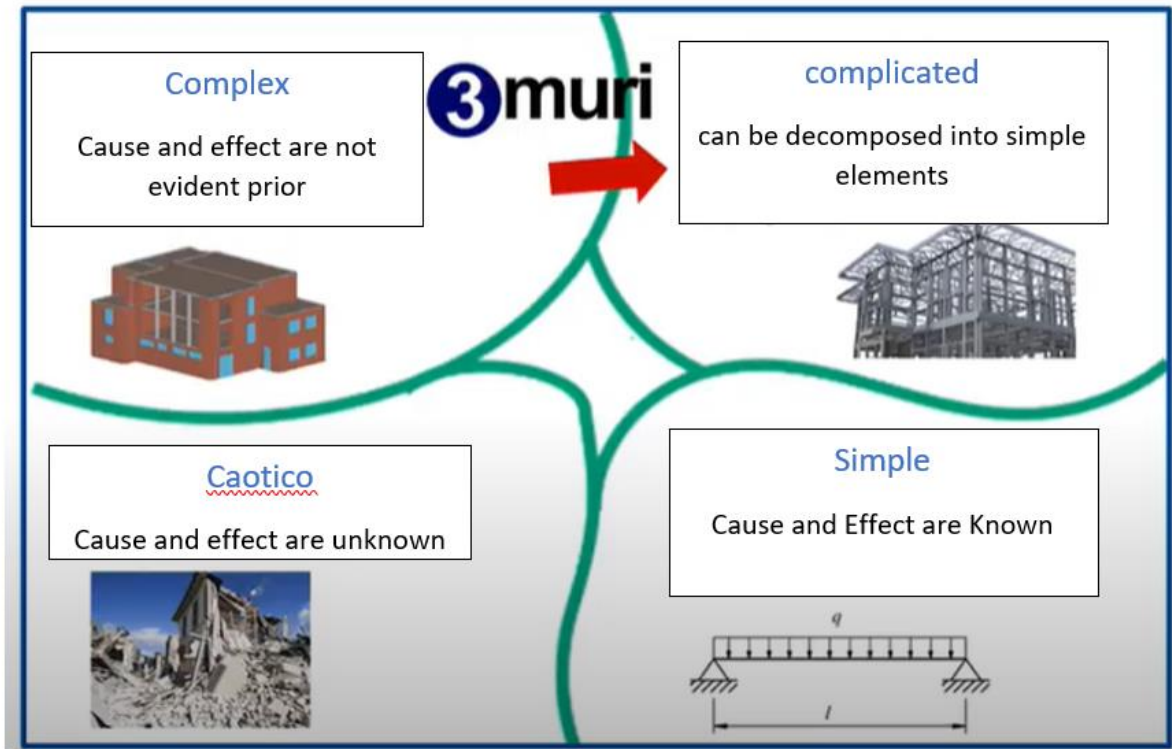


Figure 54: 3Muri categories of systems

Implementation of BIM in 3Muri

In 3Muri it's possible to import the .DXF file and chose the structure centerlines. In this way it's possible to model the structure in a quick way.

As it well known that BIM implementation in project has several advantages as follows:

- It reduces the overall cost of project
 - It eliminates the repetition of operations
 - Increases efficiency in the construction and maintenance phase of building
 - Guarantees integration of data of project
- As can be seen it's possible to integrate the design, construction and maintenance phase of building in a single model as can be seen in the *figure 55* below:

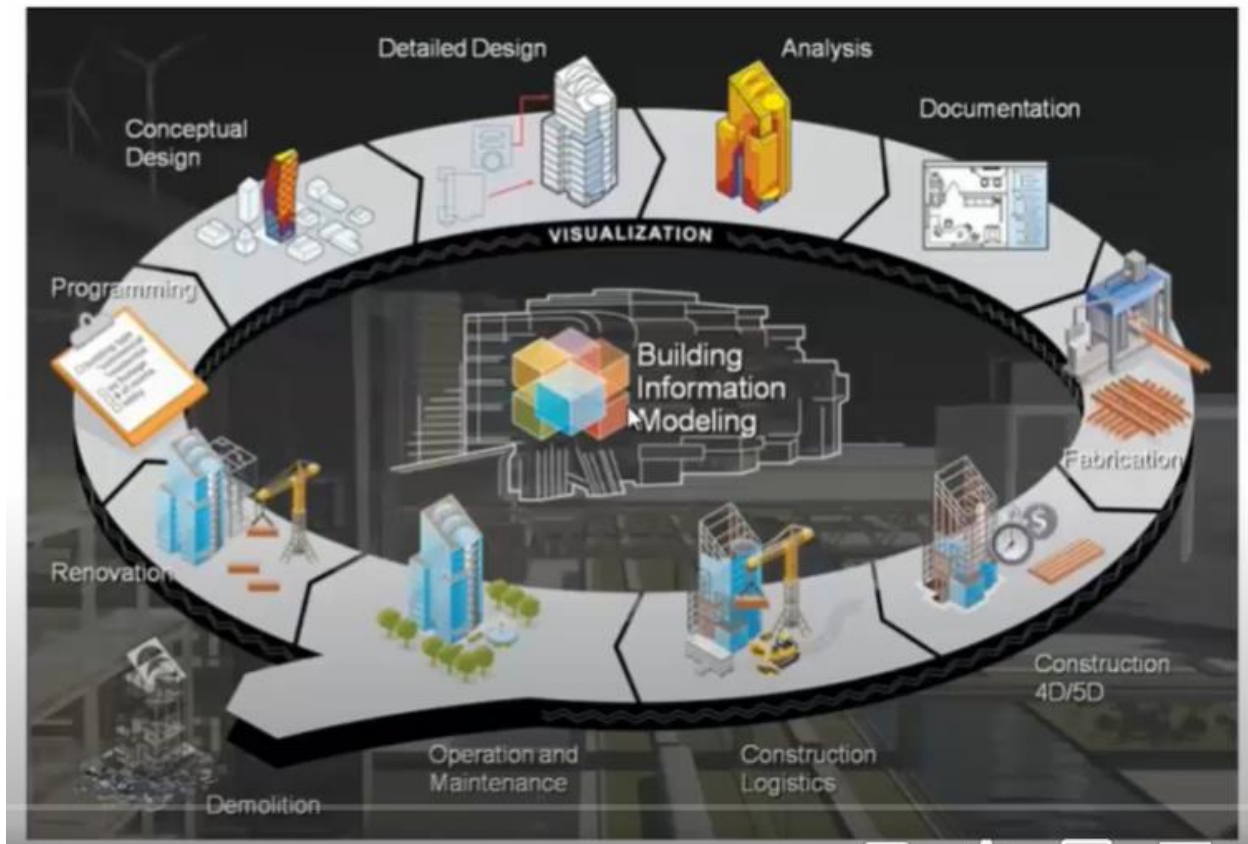


Figure 55: BIM Methodology

The implementation of BIM methodology is needed particularly in building is integration between the architectural model and structural model. The standard format data that is IFC (industry foundation classes) for transfer of among the architectural and structural software's and many other has been implemented in 3Muri.

Theoretical Basis of 3-Muri

Types of Analysis

It's possible to perform following types of analysis using 3Muri software:

- **Linear static analysis**
- Linear **Modal analysis**
- Non-linear static **Pushover analysis of Global structure**
- **Pushover of Local mechanism**
- The efficiency of intervention in improved behavior of structure

The output of 3-Muri the report is generated in .rtf format that can be opened by word. The output results can be exported in Excel file as well and it includes all the results of analysis performed step by step.

(3Muri project, n.d.)

There are 2 types of design method used for linear and non-linear analysis in 3-Muri as can be seen in figure 56 below:

- **Linear static analysis** (analysis is done in elastic range considering the behavior factor and verification of each structural element is performed in terms of strength)
- **Non-linear static analysis** (analysis is done considering non-linear constitutive laws and global verification is done in terms of displacement).

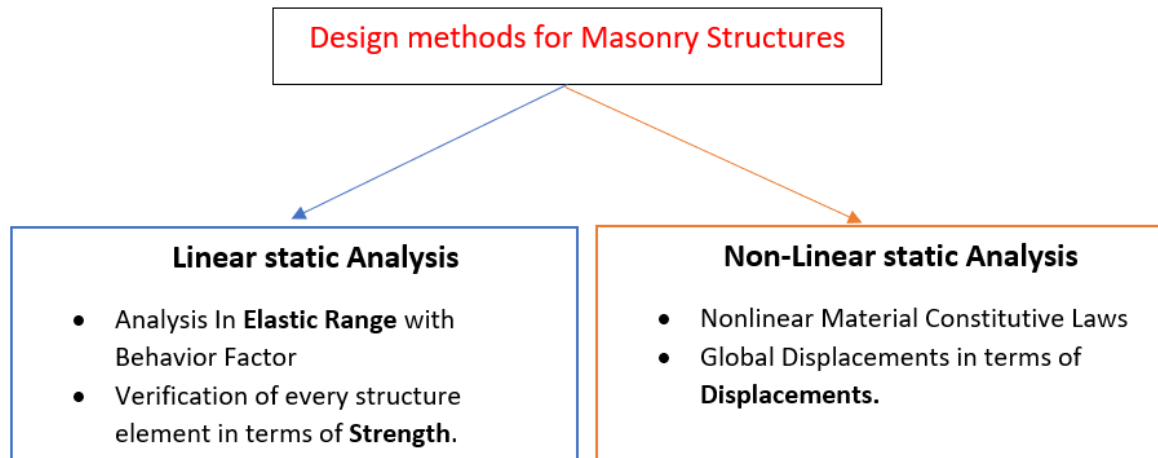


Figure 56: Types of Analysis in 3Muri

Linear static Analysis

This method of analysis is suitable for very regular building and strength of spandrel is so low that it can be considered **cantilever model** of the structure. Verifications are made in terms of Strength in linear static analysis. Behavior factor is automatically calculated by the 3Muri software. Moreover, 3Muri can also perform linear dynamic analysis known as **Modal analysis**.

Non-Linear Static Analysis (pushover analysis)

Equivalent Frame model can be realized for non-linear static analysis. It's based on non-linear constitutive laws and based on macro elements used for generating mesh inside 3-Muri. A non-linear global analysis is performed for equivalent frame model also known as pushover analysis. For global analysis verifications are made in terms of displacement of structure. 3Muri produces pushover analysis in **24 different ways** (considering the combinations of two different main directions X and Y, with and without considering the eccentricities as recommended by codes

and considering uniform and modal distribution of forces) and in each case calculates the vulnerability index. In case of existing masonry, it's very important to consider correct distribution of masses and stiffness during modelling otherwise it can result in underestimation of forces in structure significantly. Therefore, codes suggest to consider the accidental eccentricity of +5 % of direction of floor considered between center of masses and stiffness to take into account this fact. Its fundamental to carry out a preliminary knowledge phase in order to get information about the resisting mechanism of structure and also to know about the material property because maybe the change in type of use of building and change of original plans can change our model significantly. Also, we should consider that the structure has degraded with passage of time specially in case of existing structure.

There are 2 kinds of seismic forces applied uniform and modal; force profile in order to overcome the simplification that the shape of force profile does not change even when structure enters in non-linear field. There is also a possibility to set angle of applied forces.

- **Uniform distribution of Forces** in which applied load has rectangular shape. These applied forces are increased monotonically and does not change their shape. Forces applied are proportional to floor masses.
- **Modal distribution of Forces** in which load distribution is triangular and its increases with the height. Modal forces are proportional to 1st mode of vibration.
(Trapani, politecnico di Torino)

As for masonry material the plastic range is quite larger compared to elastic range which can be taken into account by performing non-linear pushover analysis in 3-muri as can be seen in *figure 57* below.

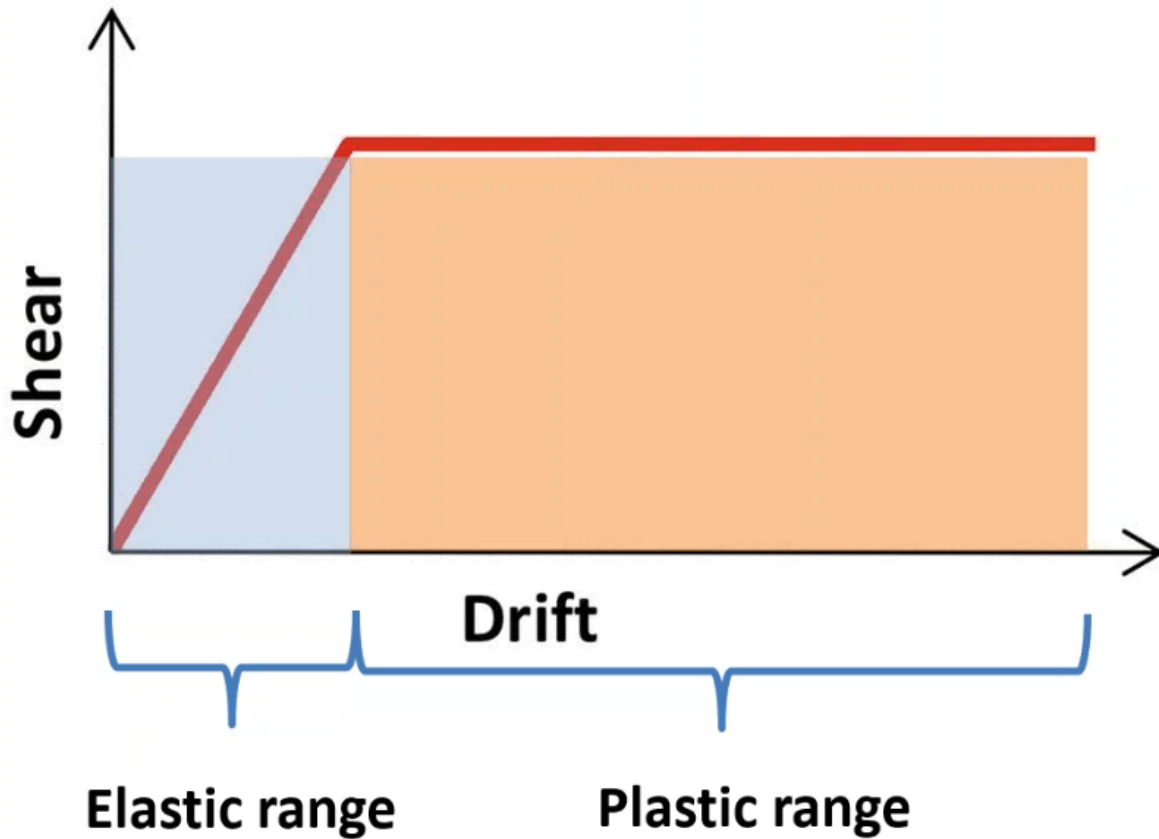


Figure 57: Typical Curve of Push-over Analysis of Masonry material

The meshing in equivalent frame structure is done automatically using Macro elements that consist of Rigid elements and flexible elements (Spandrels and piers). Whereas **Spandrels** are part of masonry between the two openings in vertical direction while **piers** are the parts of masonry between the two openings in horizontal directions while all the remaining nodes are called **Rigid elements** as shown in the *figure 58* below:

Macro-elements method

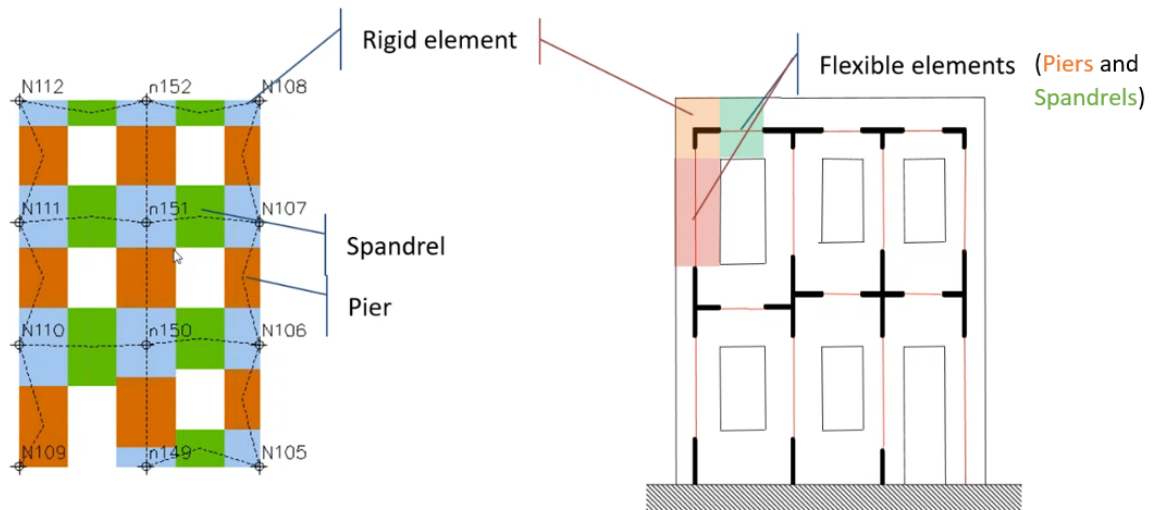


Figure 58: FARME BY MACRO ELEMENTS

It has been seen that almost all the damages in masonry structures occur in the spandrels and piers. This fact is evident by observing the damages that has taken place in masonry structures in the past years. almost all the damages in masonry structures have been took place in the spandrels and piers whereas the rigid part are not damaged by failures. Use of macro elements really save the time and computational effort as compared to time required by frame by microelements. The beams also have their own resistance but are not visible in *figure 58* above behind the spandrels. Concept of equivalent frame is based on a frame with rigid elements at the extremities with flexible elements (spandrels and piers) in-between. so, masonry structure can be modelled as equivalent frame as can be seen in *figure 59* below:

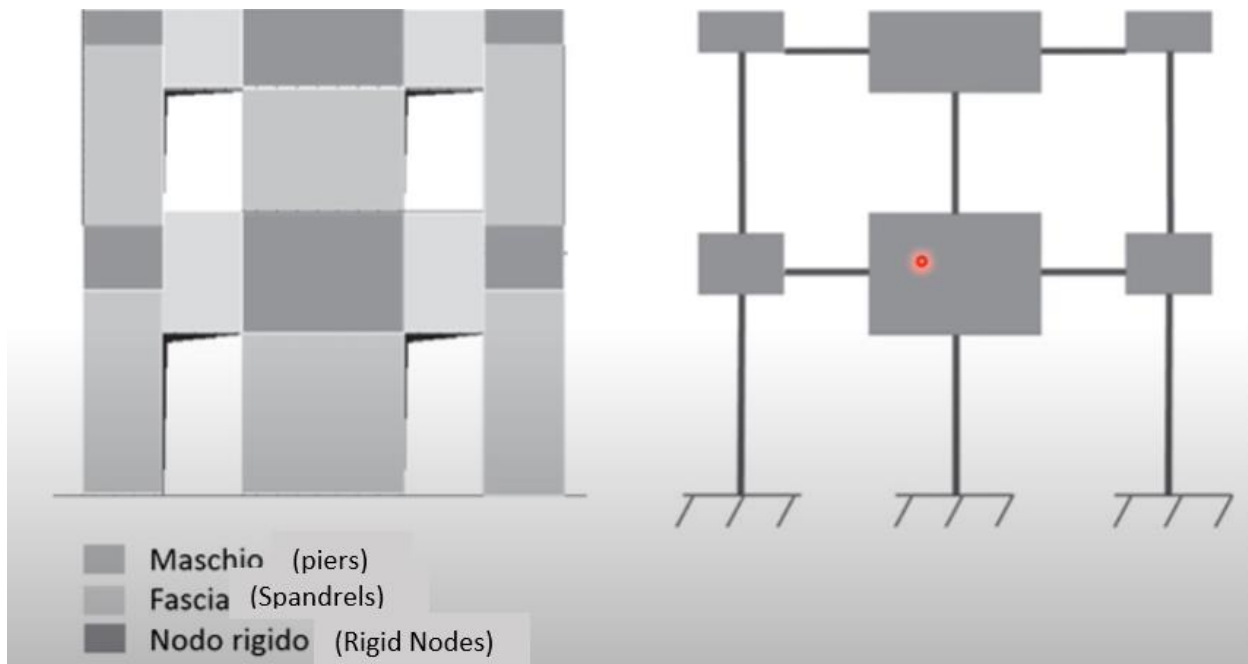


Figure 59: FRAME BY MACRO ELEMENTS

The equivalent frame model can take into account the variation of normal force variation as horizontal force changes its direction. It's possible to update the resistance of element step by step taking into account the interaction of shear and normal forces.

If we observe one of our piers, we can see that it develops shear cracks when earthquake forces are applied in one direction as can be seen in *figure 60* below:

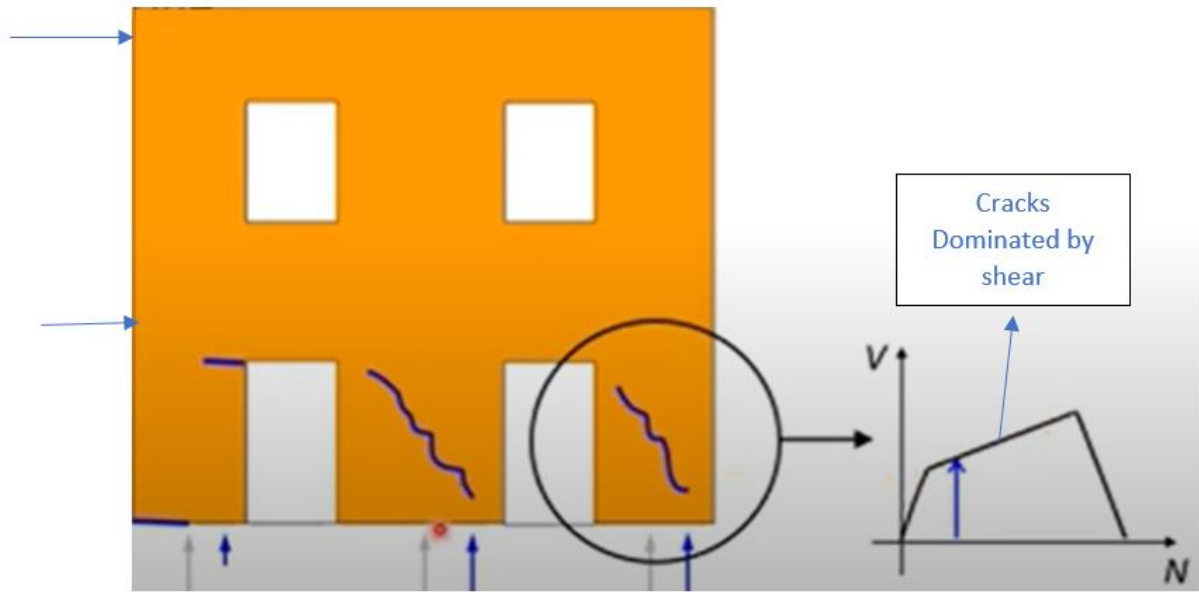


Figure 60: Shear failure Cracks

However, when the earthquake forces are applied in opposite direction, we have flexural cracks on observed pier as can be seen in the *figure 61* below:

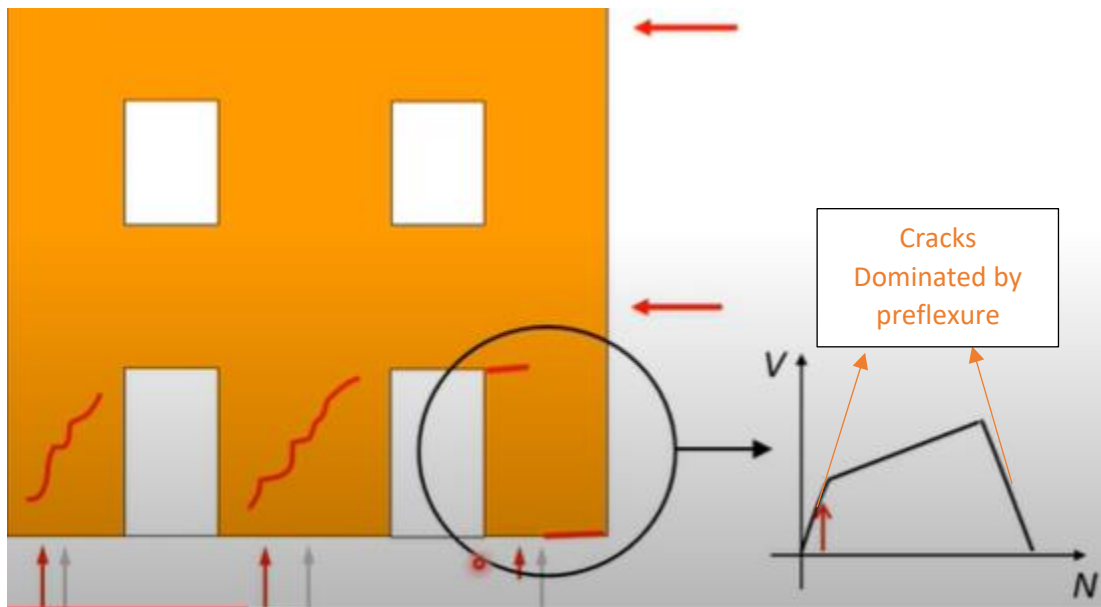


Figure 61: pressure Flexural Cracks

3Muri gives an overall picture as we push the structure and different panels of macro elements as they pass through the elastic phase to plastic phase and then from plastic phase to collapse phase. As all the elements do not enter in their plastic range simultaneously so overall ultimate strength of structure obtained as result of pushover analysis is higher than elastic limit. The *picture 63* below shows the picture as pushover analysis comparing the control node point at the center of mass of top floor to the base shear force.

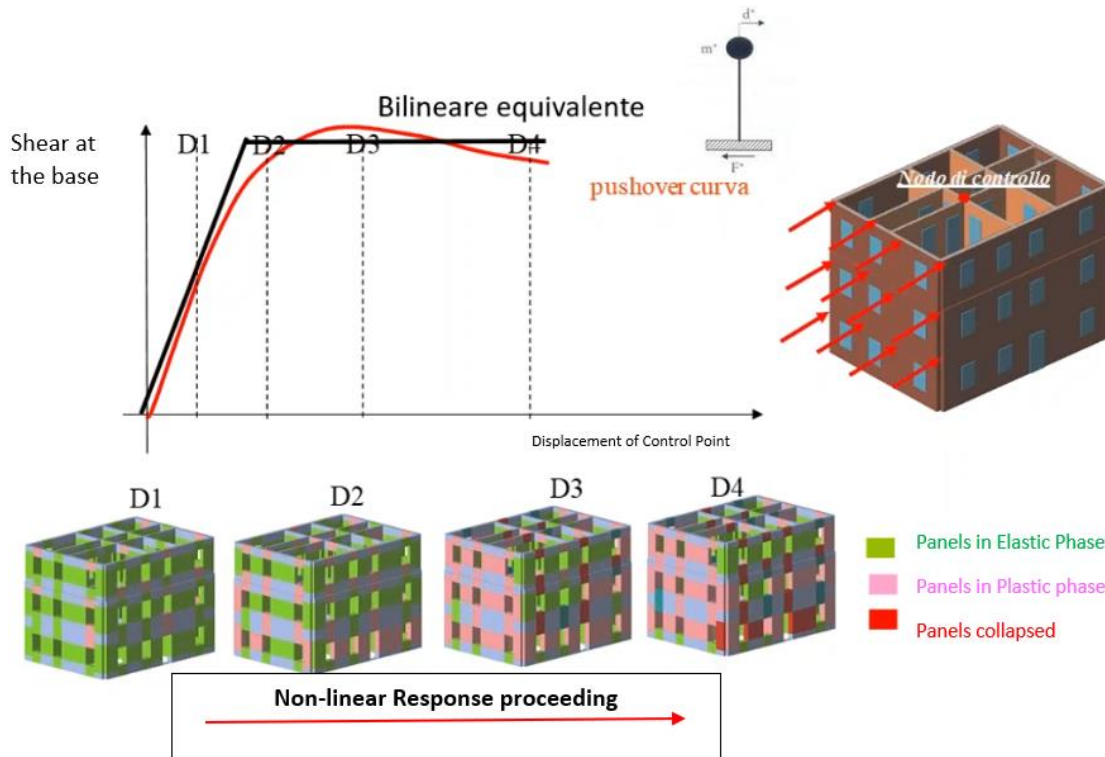


Figure 62: Picture of non-linear Pushover analysis

it is based on the most common types of failures that has been observed over past years of experience as can be seen in figure 63 below:



Figure 63: Real building Shear and Flexural Cracks

The macroelement takes into account this the interaction between the shear and axial force resistance in equivalent frame and update the resistance of element when the opposite direction of earthquake is applied as well.

The lateral displacement of structure is the best measure to know how much our structure has damaged. using pushover analysis, we can determine the lateral displacement caused as result of applied force against the base shear demand which in corresponds to different limit states as can be seen in *figure 64* below. The performance of building depends on how much the building deforms laterally as result of applied external force. These limits help us to define during the design as well as verification phase to consider different limit states associated with different return period hence related to different probability of occurrence:

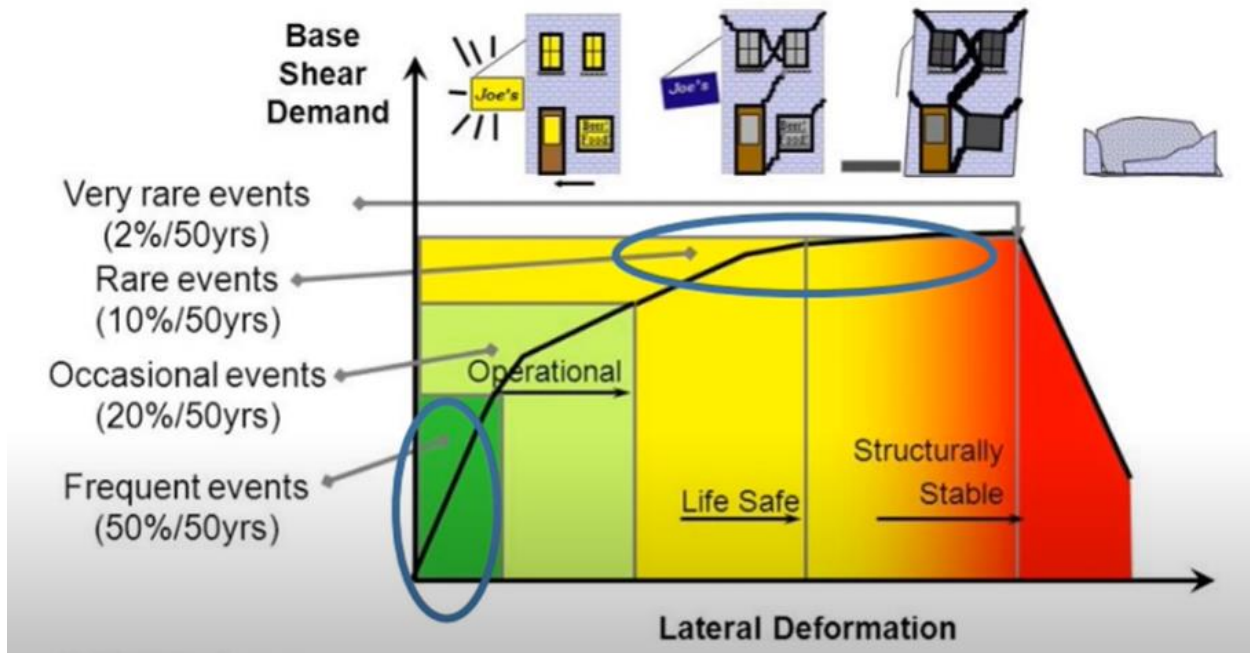


Figure 64: Sequence of limit states as damage

3Muri automatically perform **24 analyses** with changing signs of modal and uniform forces in X and Y directions as suggested by codes. It's more meaningful to present response of a structure to the excitation in terms of displacements rather than forces because displacement is more precisely linked to damage in structure occurred as we displace our structure. In this regard non-linear static analysis (Pushover analysis) play an important role in order to determine the response of structure subjected to predetermined seismic event. Static because the external forces are applied statically and non-linear because the resistant behavior model adopted for structural resistant elements. The pushover analysis can be performed by controlling the either the applied forces or displacements. The pushover analysis compares the max base shear to the max horizontal displacement of a control point as force is applied on the structure. The pushover analysis curve is then converted to capacity curves which present the analysis of hysteresis cycles and can be considered as an indicator of post elastic behavior of structure.

The load applied is proportional to mass and simulates the distribution of inertial forces simulated by earthquake and in agreement to first modal shape. Specially in case of regular buildings the distribution of forces is chosen in such a way that it determines the response of structure 1st in elastic field and then in non-elastic field. The capacity offered by structure is compared with the demand coming from the applied external force.

The energy dissipation effect during the damage of structure that can only be taken into account in non-linear behavior is accounted by reducing the max displacement demand by 20%. Hence the max displacement of the structure in case of seismic event is obtained corresponding to the reduced shear value to take into account non-linearity in the structure.

Types of Failures in Masonry Structures

The two most common types of failures that occur in masonry buildings are as follows:

- Shear failure
- Compression-bending failure

Can be seen as in *figure 65 and 66* below:



Figure 65: Shear Failure in Masonry building



Figure 66: Compression-Bending failure in Masonry Building

This practical observation has led to development of Masonry Micro elements which develops shear cracks in their central parts and at their peripherals develop both compressive and bending stress mechanism. This practical observation has led to development of theoretical Macro elements.

1st of all we have to Avoid the activation of local mechanism in particular, failure by out of plane failure of walls. Walls should develop resistance in plane. In carrying out the global verification it is assumed that the any kind of local failure is avoided even if provided with some suitable devices. The resistance of element (pier or spandrels) will be minimum of following failure mechanism that can occur in an element:

- Bending failure Cracks
- Diagonal shear Cracks
- Sliding shear cracks

The flexural cracks normally take place when the thickness of masonry wall is less below the window. The sliding shear cracks takes place our course or block of masonry wall slips cracks are developed that progressively increase. We must consider the interaction of shear and axial force acting on an element. Shear cracks are developed diagonally. The shear slippage/sliding cracks take place when one course or block slips as can be

seen in picture below. There are different types of failure cracks that can take place in masonry structures as can be seen in *figure 67* below:

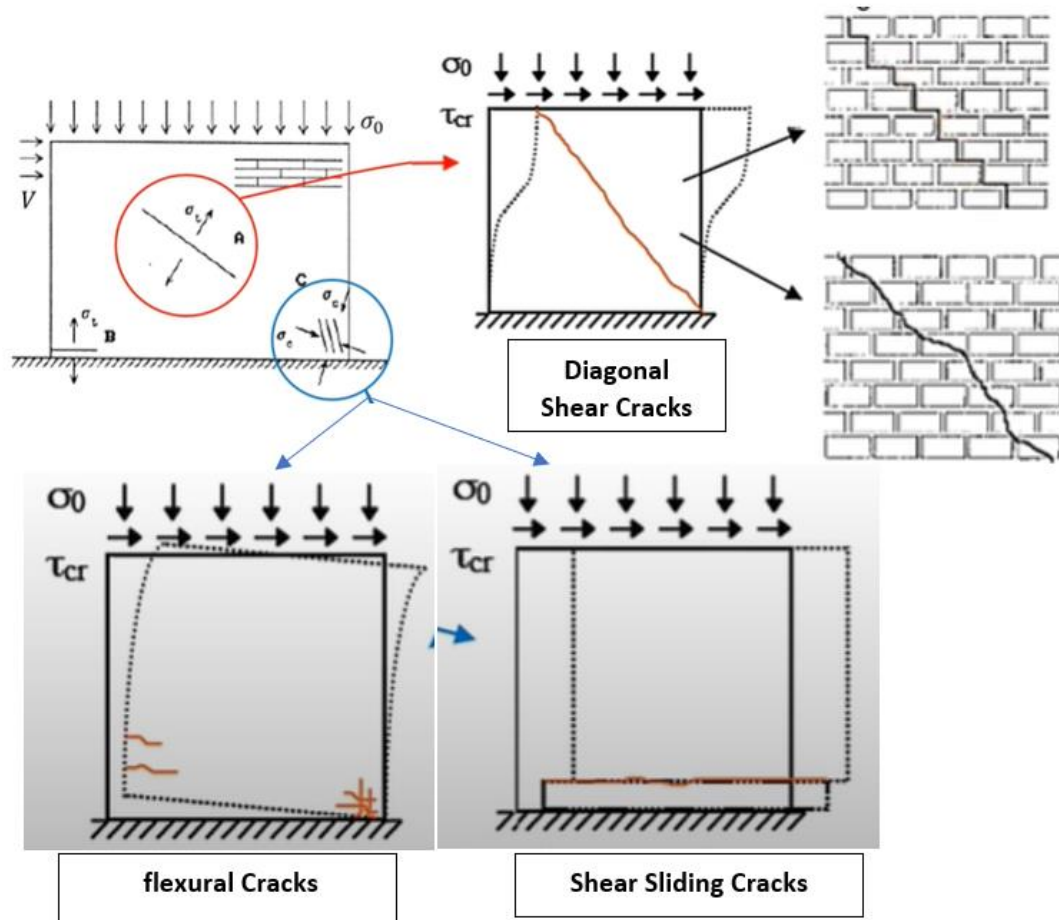


Figure 67:Cracks in Failure Modes in Masonry

the interaction between the shear and Normal force must be taken into count for all three types of failures as shown in *figure 67* below and resistance is given as minimum resistance of shear or flexural resistance of element:

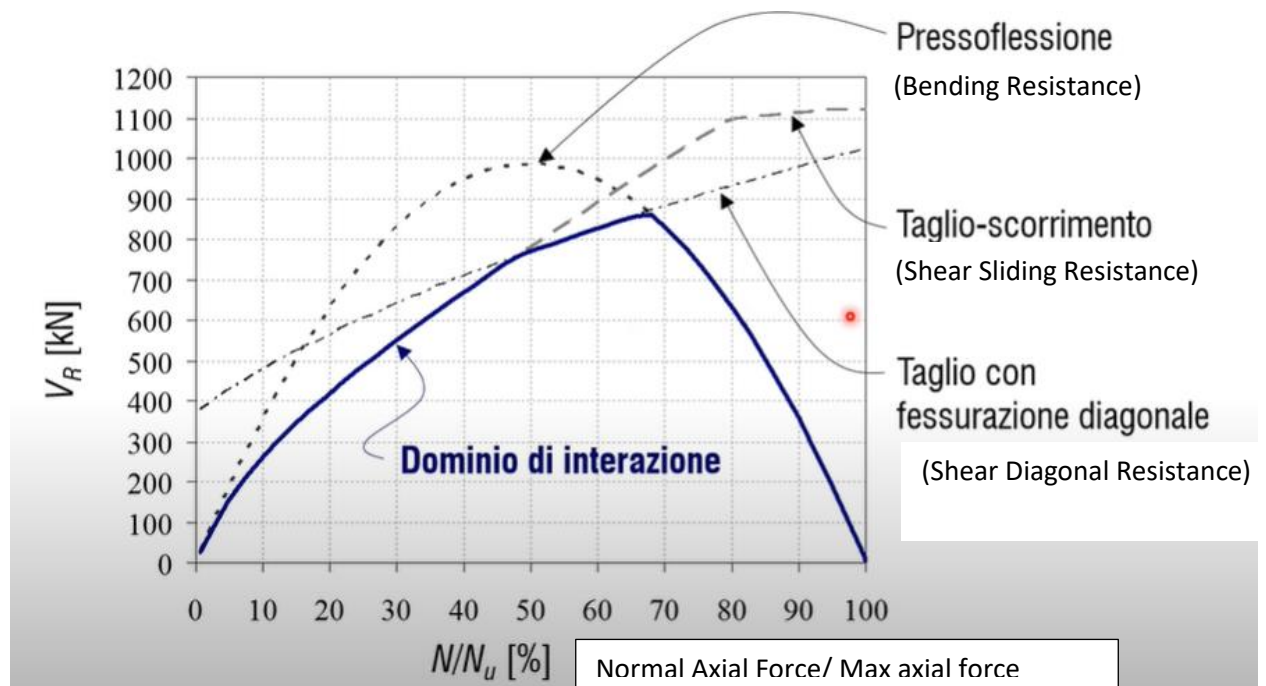


Figure 68: Resistance of section as min resistance

The pushover analysis gives the capacity curve that relates the base shear to the displacement of control node which is usually taken as center of mass of top floor and 3Muri gives suggest some node automatically to be chosen as control nodes having the maximum height form the foundation level automatically. The displacement of structure passes through various limit states and eventually collapses. This displacement must be compared with the displacement demand. The displacement demand can be found from the elastic response spectrum that is based on single degree of freedom system as it oscillates considering the structure as infinitely rigid. To take into account the non-linear behavior the displacement demand is reduced by a behavior factor that takes into account the non-linearity associated with material and of structure.

(STA DATA, 2022)

The theory on which the 3MURI software is based on macro elements considering the constitutive law of material not perfectly elastoplastic that collapses suddenly until reaching the ultimate drift but is based on the step-by-step collapse as can be seen in *figure 69* below:

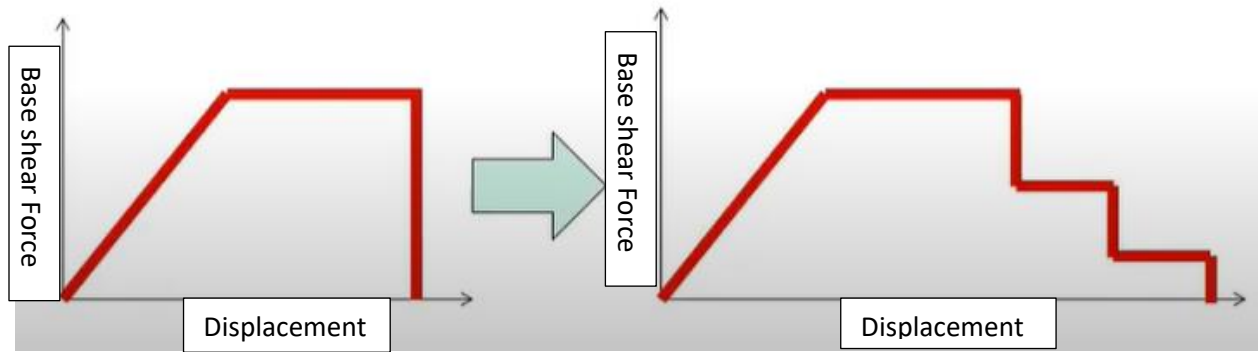


Figure 69: Step by step Collapse proceeding

we can see the sequence of damages as we push our structure. The First part in elastic range in which there are very few damages in the structure. Moving on to plastic range reaching to serviceability limit state (1st limit state governed by people) it's possible to have some damage but structure is still safe for people. Moving on reaching near to ultimate limit state there will be significant damages in structure and need some maintenance for sure. The structure will be pushed until the collapse and here is where the calculation ends. The *figure 70* below shows the sequence of limit states starting from Elastic range to collapse limit state as we push our structure:

(3MURI, 2018)

Damage increase in the wall structures

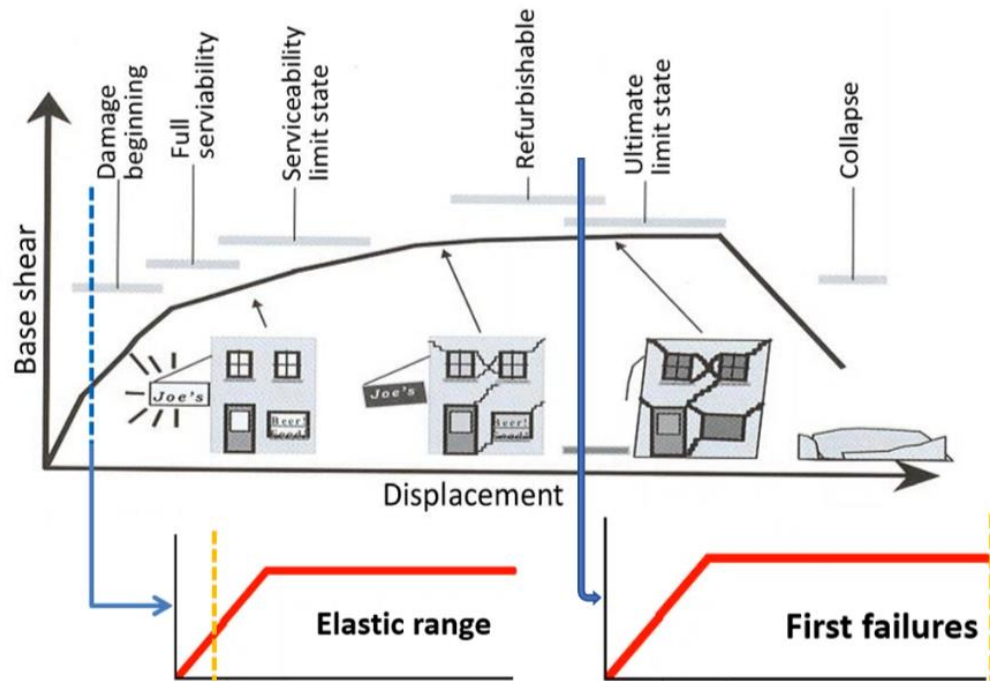


Figure 70: Sequence of limit states of Damages for Push-over analysis

The Macro element collects the axial deformability of and tangential deformability of an element as can be seen in *figure 71* below:

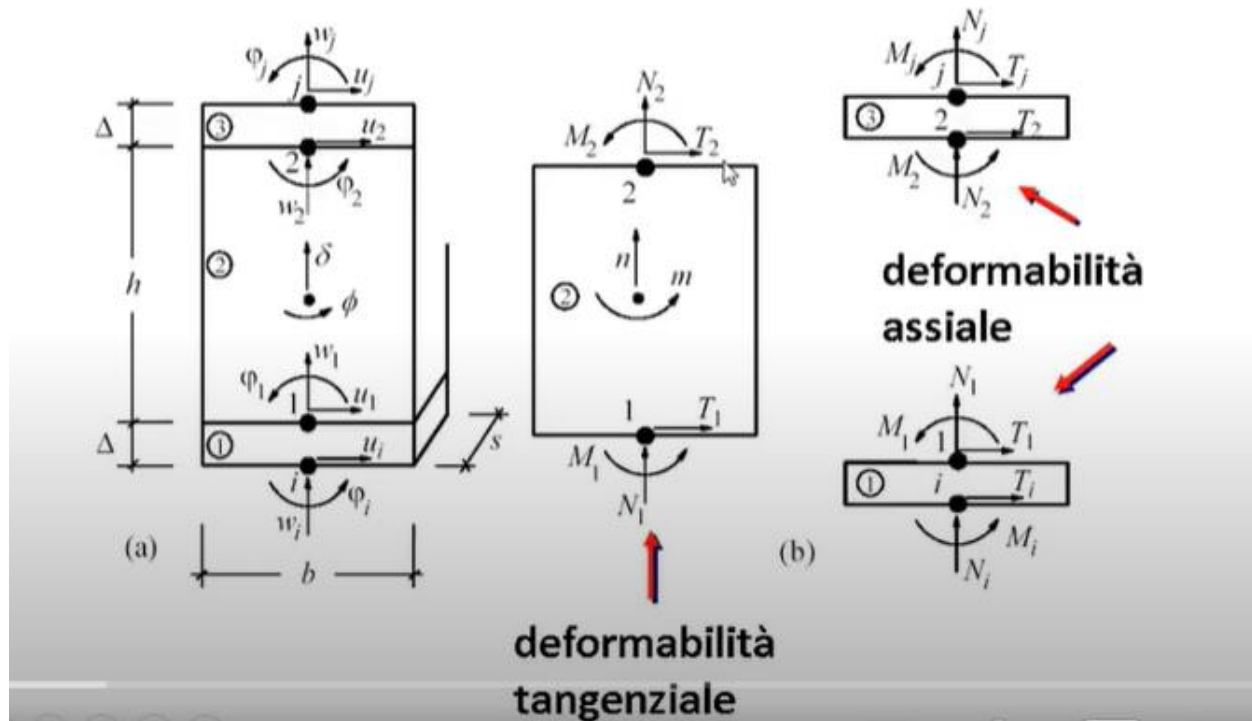


Figure 71: Macro element internal actions

The curve of real pushover analysis passes through these stages as shown in figure 72 below. the maximum strength is little bit higher as not all of structures reaches the elastic limit as we cross the elastic limit. Ultimately, we stop to push our structure when it reaches the collapse limit states.

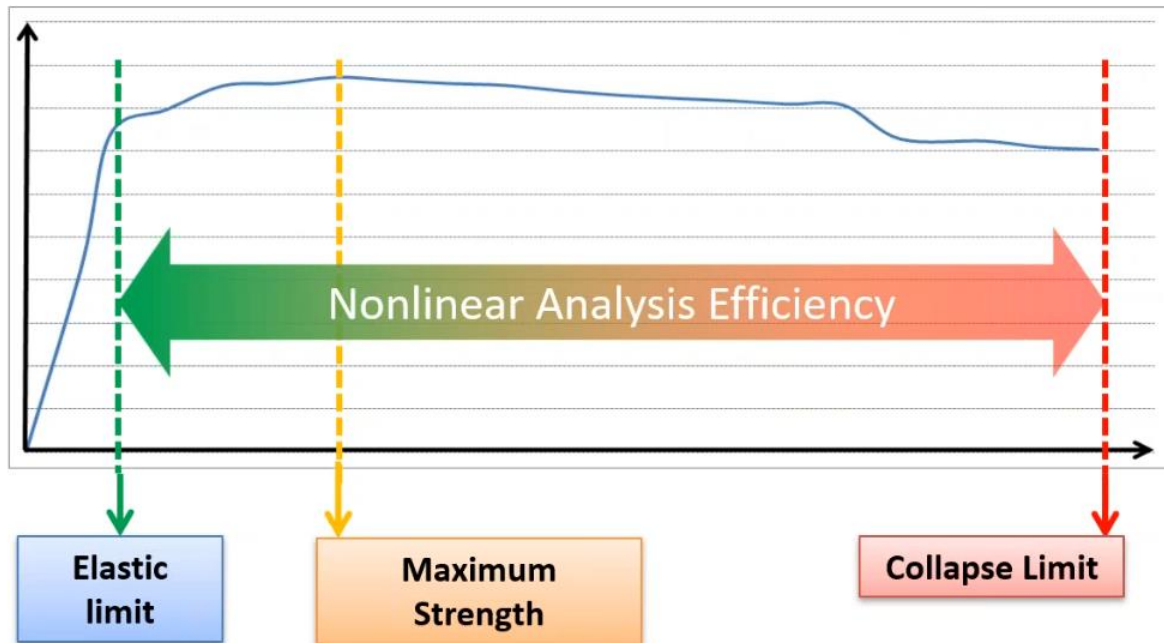


Figure 72:sequence of limit states as damage is progressed

The **Linear static analysis** has the following applicability limits as mentioned in Eurocode-8 as follows:

EN 1998-3: SC.3.2 Linear methods: Static and Multi-modal

- The lateral load resisting walls are regularly arranged in both horizontal directions.
- Walls are continuous along their height.
- The floors possess enough in-plane stiffness and are sufficiently connected to the perimeter walls to assume that they can distribute the inertia forces among the vertical elements as a rigid diaphragm.
- Floors on opposite sides of opposite sides of common wall are at same height.
- Floors on opposite sides of a common wall are at the same height.
- At each floor, the ratio between the lateral in-plane stiffnesses of the stiffest wall and the weakest primary seismic wall, evaluated accounting for the presence of openings, does not exceed 2,5 Linear Static Analysis applicability limits Nonlinear Static Analysis (Pushover) allows you to overcome these limitations
- Spandrel elements included in the model are either made of blocks adequately interlocked to those of the adjacent walls, or have connecting ties. At each floor, the ratio between the lateral in-plane stiffnesses of the stiffest wall and the weakest primary seismic wall, evaluated accounting for the presence of openings, does not exceed 2,5 Linear Static Analysis applicability limits Nonlinear Static Analysis (Pushover) allows you to overcome these limitations.

However in reality these limits are never met and non-linear static analysis allows to overcome these limits.

Types of Materials

Its possible to model different types of material inside 3-Muri software in *figure 73* below and can be seen as can be seen follows:

- Masonary
- Reinforced Masonary
- Steel
- Wood
- Walls reinforced by FRP
- Reinforced Concrete.

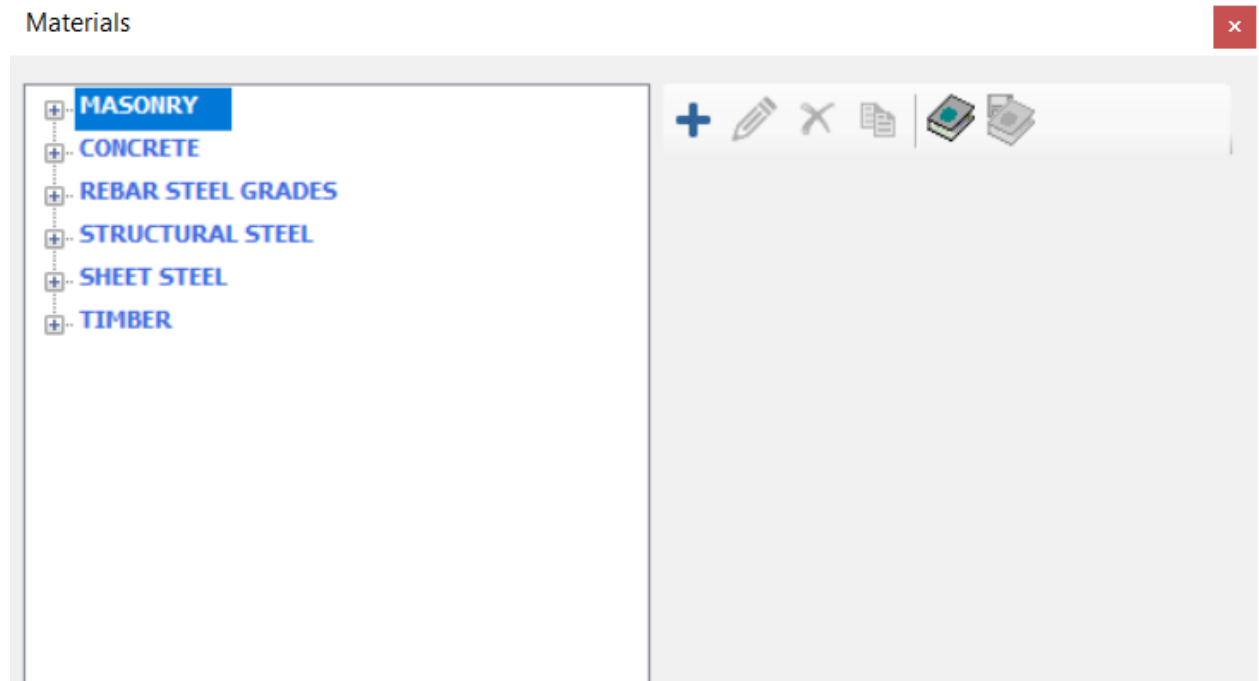


Figure 73: Types of Materials inside 3Muri

Slabs Modelling

To take into account the connection between the slabs and walls, there are basically 3 types of slabs with diaphragmatic that can be modelled as follows:

- Diaphragm with Infinite stiffness
- Diaphragm with finite stiffness
- Diaphragm with negligible stiffness

However, in our model realized inside 3Muri slabs were modelled as rigid diaphragms
Among these slabs with finite stiffness is easiest one to model as it reduces the degree

of freedom to 3 (2 translations and 1 rotation per each floor). However, we must model our slab as such as it takes into account the elastic behavior of slab. We can model our slab as available as membrane or shell as available options in various software's. there are readymade types floors that can be defined inside the 3Muri software as can be seen in the *figure 74* below:

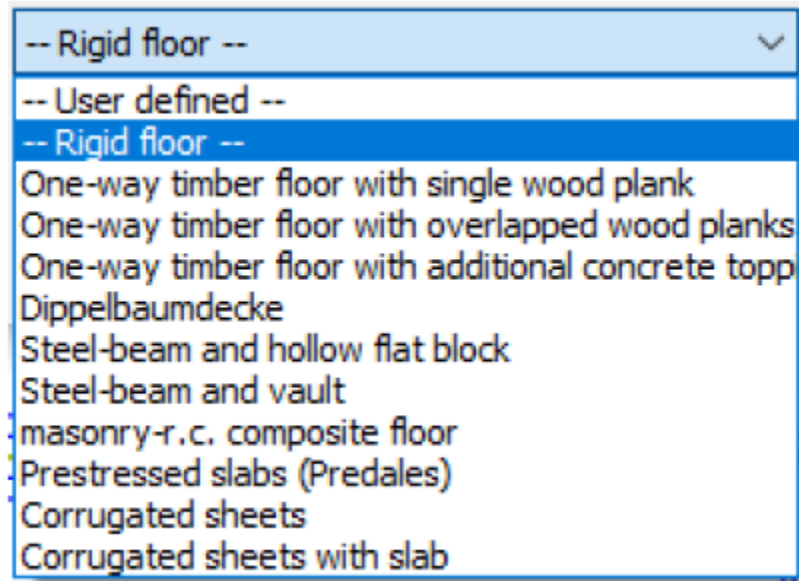
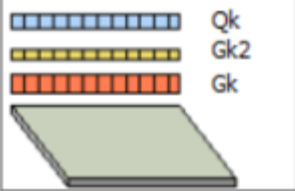


Figure 74: Types of Floors inside 3Muri


Passing from static to dynamic analysis involves a lot of difficulties and complexities. for non-linear dynamic analysis we must take into account the cyclic behavior of masonry walls. Moreover, there when modelling slabs there is an option inside 3Muri to define the support length of slab on walls as can be seen inside *figure 75* below:

Floor

Loads


 Elevation [m]
 Gk [kN/m²]
 Gk2 [kN/m²]
 Qk [kN/m²]

Static verifications


 Support lenght [mm]
 Δ extrados elevation [m]
☒ Leading variable action Δ intrados elevation [m]

NT18 circolare

ψ0 ψ1 ψ2
☐ Category of ζ_v

Type



-- Rigid floor --

Thickness [mm]
 G [kN/m²]
 Ex [kN/m²]
 Ey [kN/m²]
 V

Mass loading

☐ Unidirectional ☒ Bidirectional
 Main direction loading %

Display

Material colour  Texture 


☐ Retrofits 

Figure 75: Definition of support Length of slab on walls

Its possible to model the floors as flexible or with their real stiffness as can be seen in the figure 76 below which changes the distrinution of forces in the wall significantly:

Floors: transmission of horizontal seismic forces

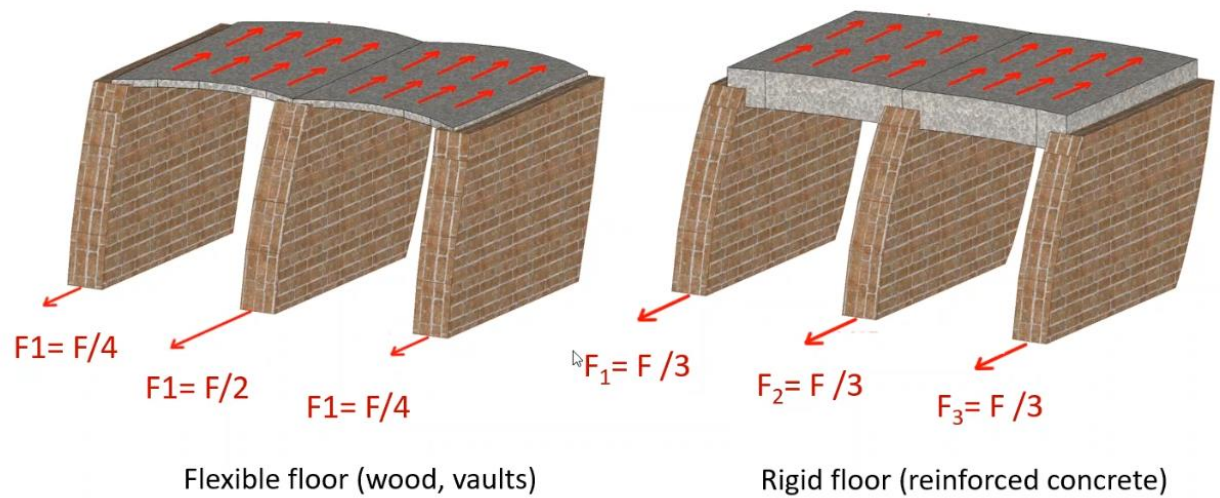


Figure 46

Types of Structure elements defined in 3-Muri

Different types of Floors and vaults can be defined in software 3-Muri as shown in *figure 77* below:

Parametric floors and vaults

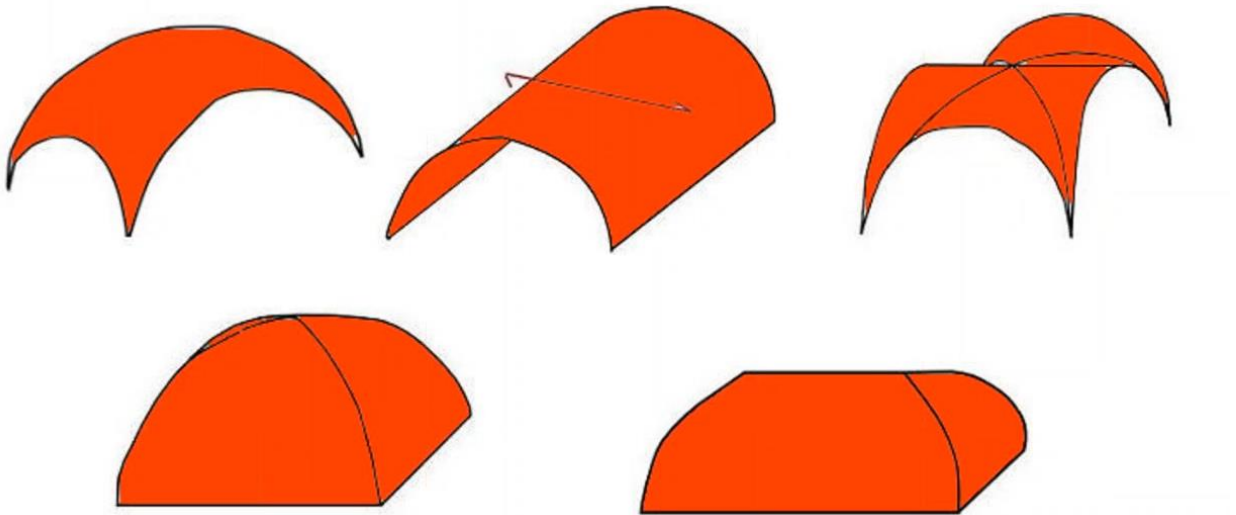


Figure 77: Types of Floors defined in 3-Muri

It's possible to define different types of Walls, Beams and columns as shown in figure 78 below. Moreover, it's also possible to define different types of Beams (Reinforced concrete, wood etc.) with masonry wall as shown in *figure 78* below:

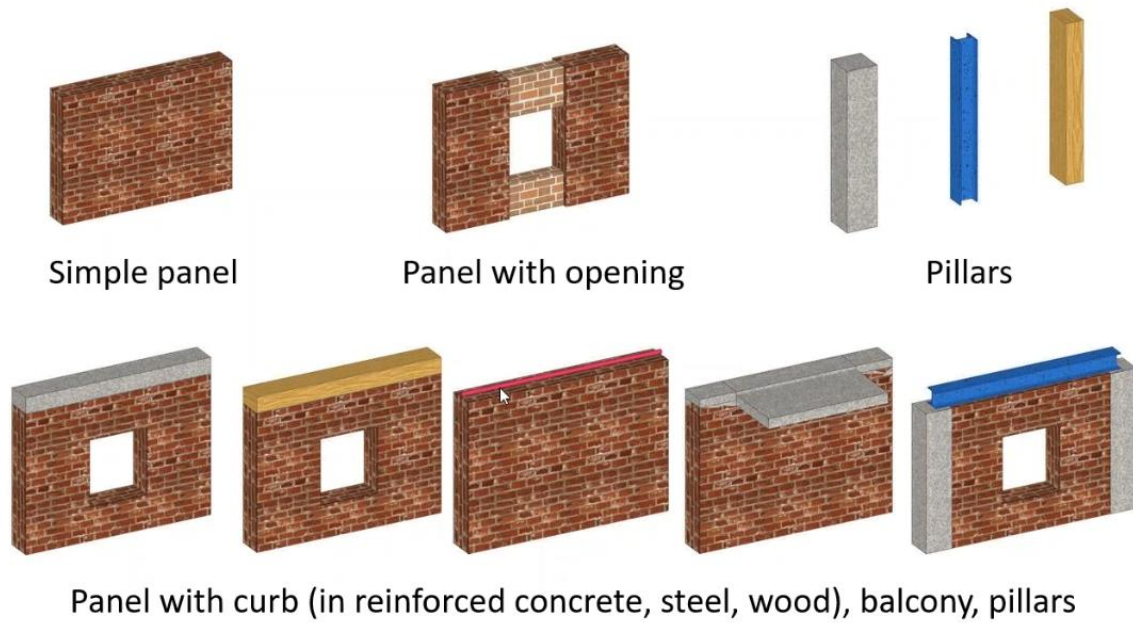


Figure 78: Types of panels in 3MURI

Furthermore, it's also possible to define different types of materials within same masonry wall as shown in *figure 79* below:

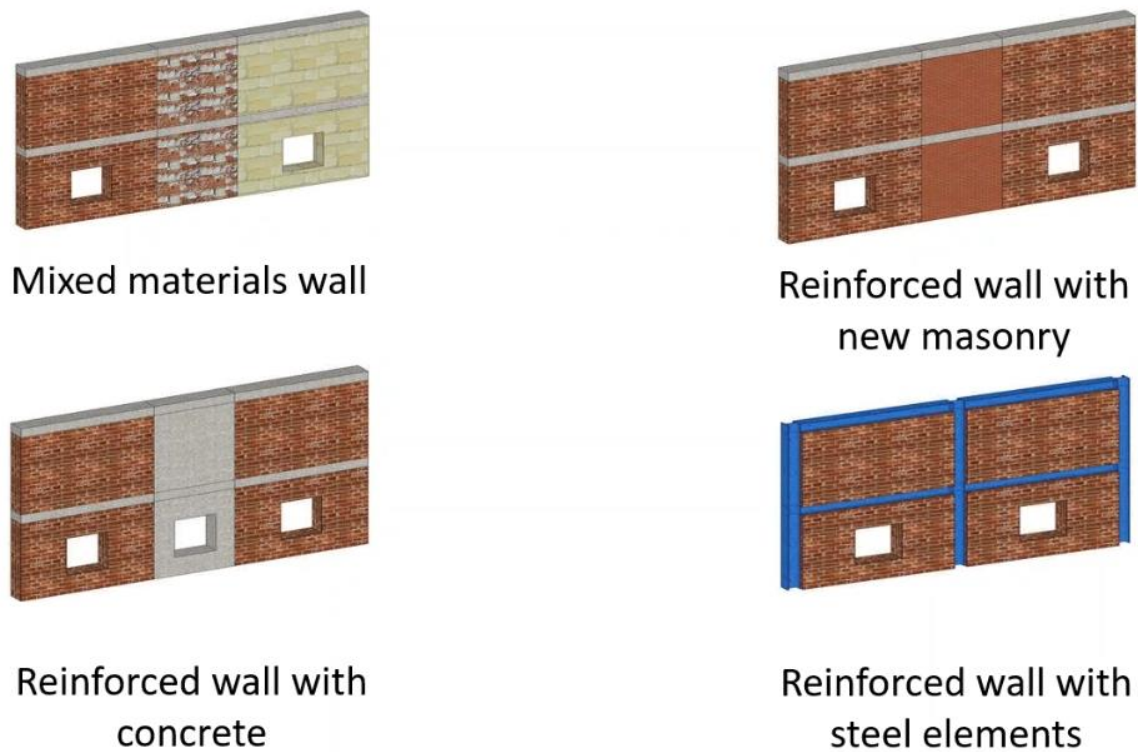


Figure 79: Types of walls inside 3Muri

There is also possibility to insert different kind of reinforcement with steel, FRP and FCRP within the wall as can be seen in the *figure 80* below:

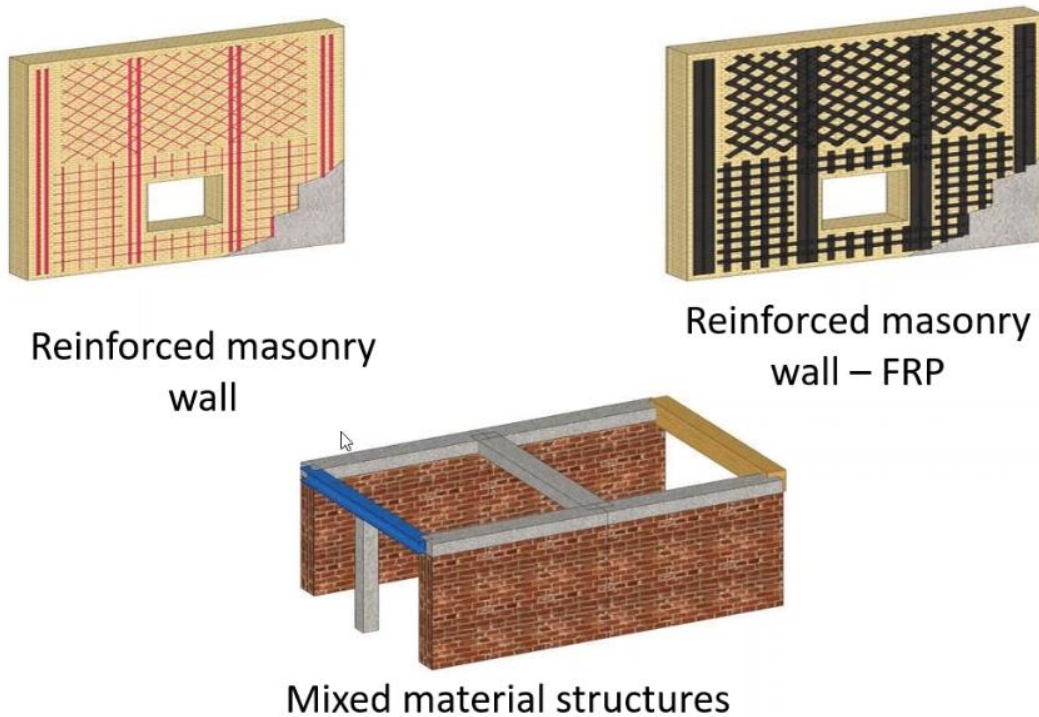


Figure 80: Reinforcement definition inside 3Muri

Foundations

The foundations are not modelled in 3Muri and the superstructure is considered to have a rigid link with the foundation that could be a big approximation for seismic analysis.

Local Mechanism

It's also possible to perform the local mechanism of wall for out of plane failure of wall between two soft floors when the connection between the walls and floors is not good and box types behavior is not sure as can be seen in *figure 81* below:

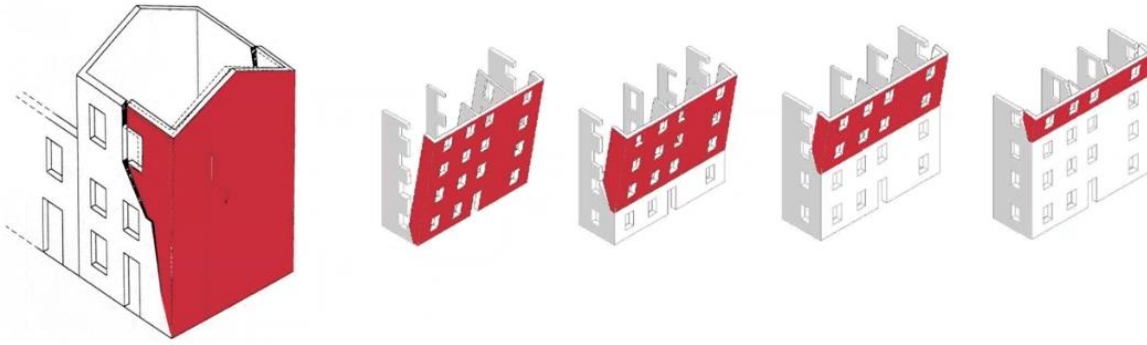


Figure 81: Out of plane Failure of Wall

Choice of Control Point

The choice of control point for performing pushover analysis is important as it effects the reliability of results. As pushover analysis can be done by controlling either the applied force or displacement that is applied step by step. It's recommended to select the node most deformable as less force will be required to achieve that displacement and more accurate will be our results as we can pass through all the breaks of our structure.

Pushover analysis compares the base shear as function of displacement/force applied. We can judge by the shape and value of pushover analysis curve that whether our analysis is correct or not. Generally speaking, if we achieve ultimate displacement du from pushover curve of several cm just an example as 72cm which is obviously not realizable particularly for masonry structure we cannot consider our result to be correct. On the other hand if our ultimate displacement is less than 1cm roughly we can consider our results of pushover analysis. However, the values of ultimate displacement du varies depends on case to case and can varies significantly.

Moreover, the shape of pushover analysis has an initial elastic shape and a plastic stage. When some of elements have reached their ultimate drift limit the resistance is provided by ductility and enter in non-plastic phase.

The **return period** for capacity is done automatically depending on value of acceleration of structure and can have a constant or variable rate of return as can be seen the *figure 82* below:

Vulnerabilità Sismica

	TR_C	TR_D	α_{TR}	PGA_D [m/s ²]	$PGA_C(TR)$ [m/s ²]	$F_0(TR)$	$T^*_C(TR)$	$\alpha_{PGA}(TR)$	PGA_C [m/s ²]	α_{PGA}
SLC	> 2475	975	> 2,538	3,28	4,44	2,46	0,38	1,354	4,96	1,513
SLV	1421	475	2,992	2,56	3,70	2,42	0,37	1,448	3,93	1,535
SLD	576	50	11,520	1,02	2,73	2,37	0,35	2,681	3,12	3,064
SLO	271	30	9,033	0,77	2,08	2,33	0,33	2,697	2,24	2,894

Figure 82: Return Period TR depending on limit state

If we have structure completely underground it's not possible to perform pushover analysis of such structure. We will overestimate the displacement of such structure if structure is totally underground. If we have such structure which has 3 floors above the ground level and have just 1 floor underground it's possible to model all the floors above the ground.

If there is some connection between the two portions of building it must be modelled because it causes the additional constraints to our buildings and behavior of our building would change. If we model the building as isolated it cause significant errors.

The choice of reinforcement chosen for intervention depends on case by case. For example, if our element of structure has lower resistance or have lower ductility. If element have lower resistance, then we can increase the resistance by choosing the steel jacketing for example. Furthermore, it depends on we want to increase our shear or bending resistance. On the other hand, if our element is less ductile, we can increase the ductility by introducing some fiber reinforced bars that increase the ductility of element significantly.

Sensitivity Analysis

It's also possible to perform sensitivity analysis inside 3Muri which allows us to determine which part of structure is more sensitive to change in parameters (such as material parameters) and is helpful in the retrofitting of part of structure. Infact there is a complete module inside the 3Muri for performing the sensitivity analysis and results of sensitivity analysis are helpful while performing the retrofitting interventions.

(3Muri - Masonry buildings analysis, 2022)

Masonry Macro-elements

3Muri uses the concept of frame by macro elements for structure analysis of structures. Macro elements can be considered as a wall or a beam or a column. Use of macro elements for structure analysis saves a lot of time and computational efforts.

for modelling of masonry piers and spandrels a non-linear beam model has been used which initially gives stiffness due to elastic properties and then stiffness degrades in plastic range. This macro elements used in 3muri has following significant properties:

- the initial stiffness of macro element is given by elastic (cracked) properties
- the element follows a bilinear behavior and max value of bending moment and shear is calculated in ultimate limit state.
- The distribution of internal forces takes place according to equilibrium of element.
- For detection damage limit state both global and local damage parameters are considered.
- Stiffness degradation takes place in plastic range
- Ductility control takes place by defining max drift which could be different depending on failure types and is regulated by codes depending on different limit states.
- The element expires at ultimate value of drift and global analysis continues.

The masonry structure is divided into following 3 parts:

The non-linear behavior is activated when one nodal force reaches its maximum value estimated as minimum of following 3 failures can be seen in figure 83 below:

- flexural-rocking
- shear-sliding
- diagonal shear cracking

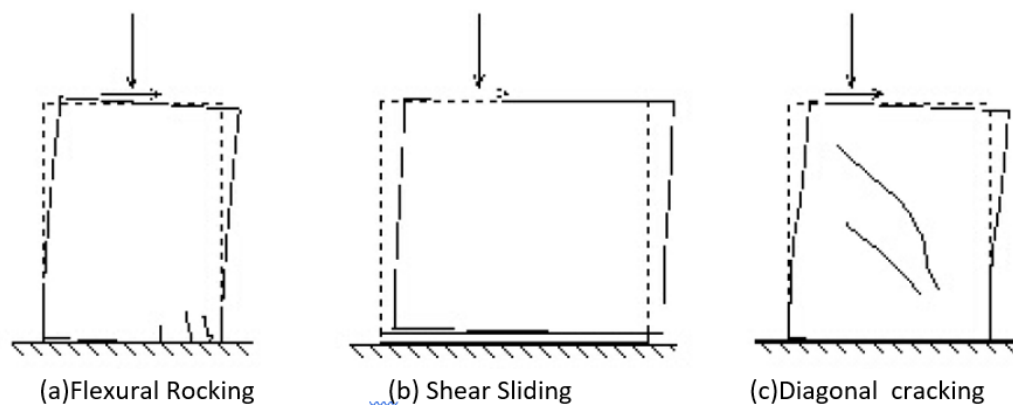


Figure 83: Types of Failures in Masonry Structure

The non-linear analysis push over analysis works on following scheme as shown in *figure 84* below:

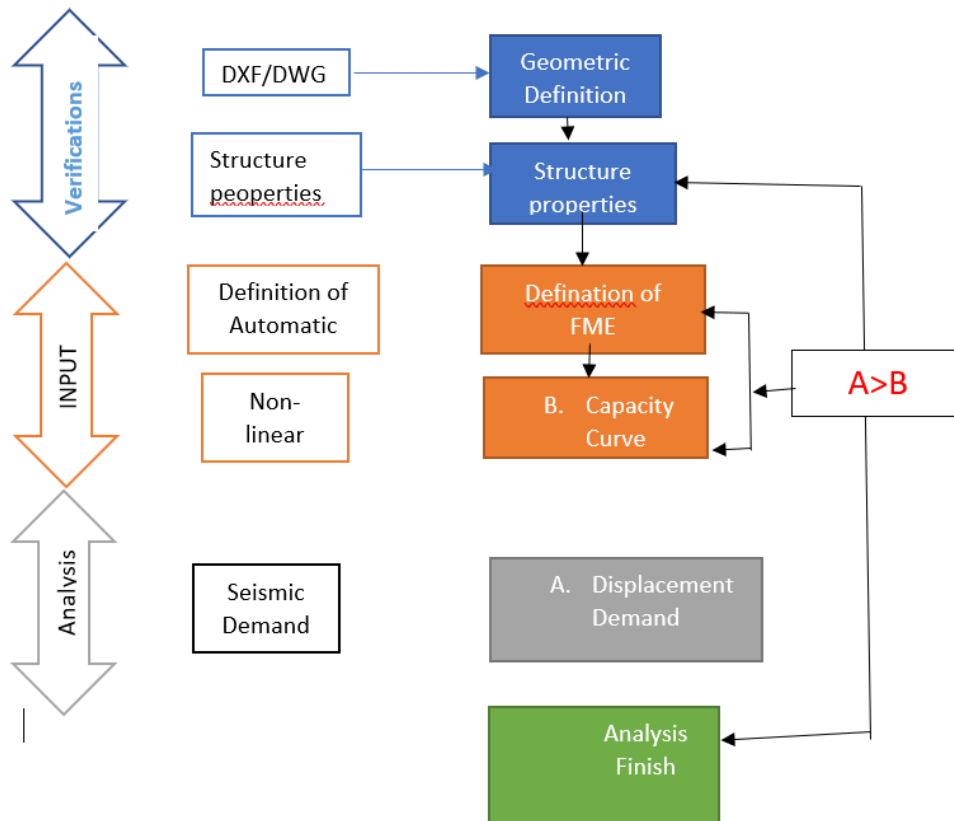


Figure 84: Working scheme of 3Muri for Pushover analysis

Bending: Ultimate moment

The ultimate bending can be calculated based on bending- compression interaction and can be calculated using following expression:

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

Whereas

l is length of panel

t is thickness of panel

ϕ_0 is the average compression strength

N_u is maximum axial compression action of panel taken as $N_u = 0.85 f_m l.t$

f_m is average compressive strength of masonry

the tensile strength is taken as 0 (rectangular stress-block with factor = 0.85)

in existing buildings, the average strength of masonry is divided by confidence factor depending on the level of knowledge we have.

the *figure 85* below shows the interaction of ultimate bending moment and axial compressive action:

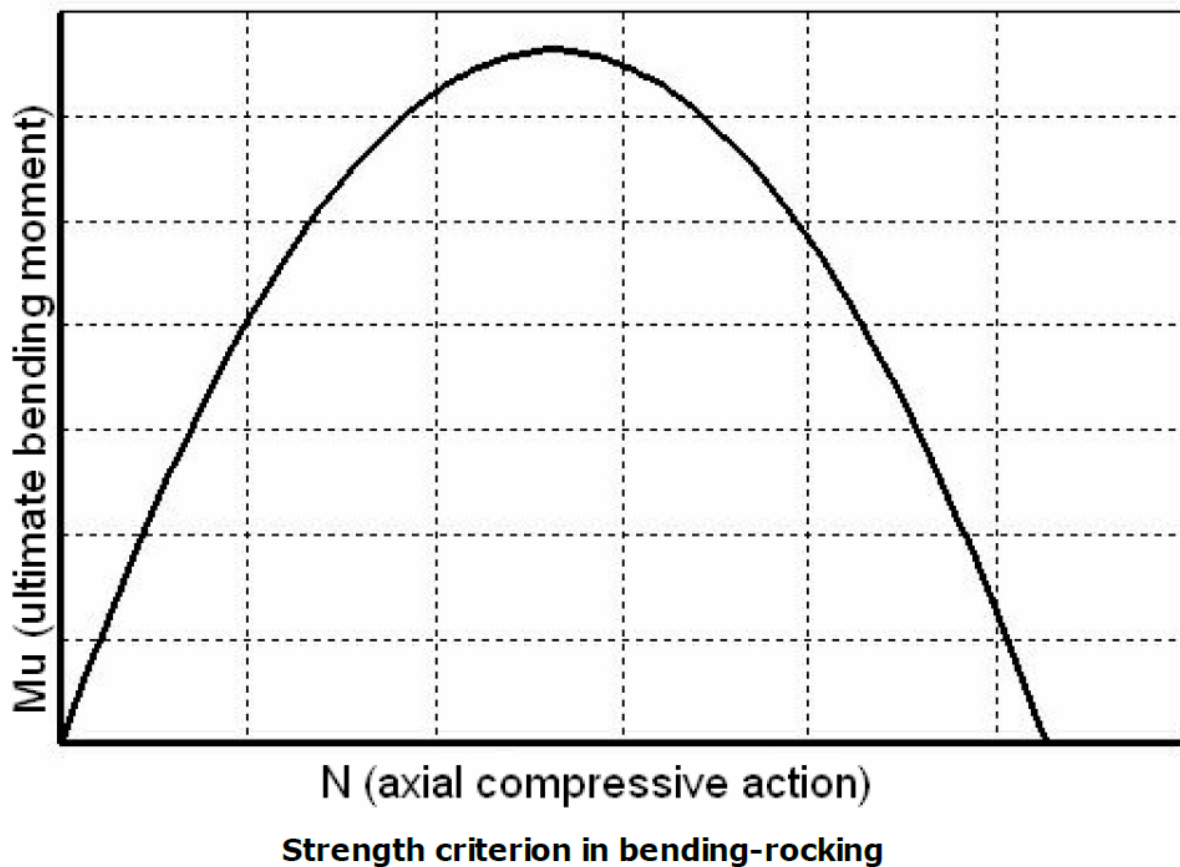


Figure 85: M-N interaction

Shear (Mohr Columb Criteria)

The ultimate shear according to Mohr columb can be found using the following expression:

$$V_u = l' t f_v = l' t (f_w + \mu \sigma_n) = l' t f_w + \mu N$$

Where

l' = length of compressed section of panel normally taken as

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

We refer to effective compressed length when $e=M/N$ exceeds the limit values of $l/6$. However, when $e < l/6$ all the points of section are compressed.

t = thickness of panel

f_v = shear resistance of Masonry that need to be divided by confidence factor depending on level of knowledge we have according to Italian codes

f_{v0} = shear resistance of masonry without compression

m = friction coefficient normally taken as 0,4

σ_n = normal average compressive stress referring to the effective area

The software is used for the structure analysis of both existing and new structures particularly for Masonry structures.

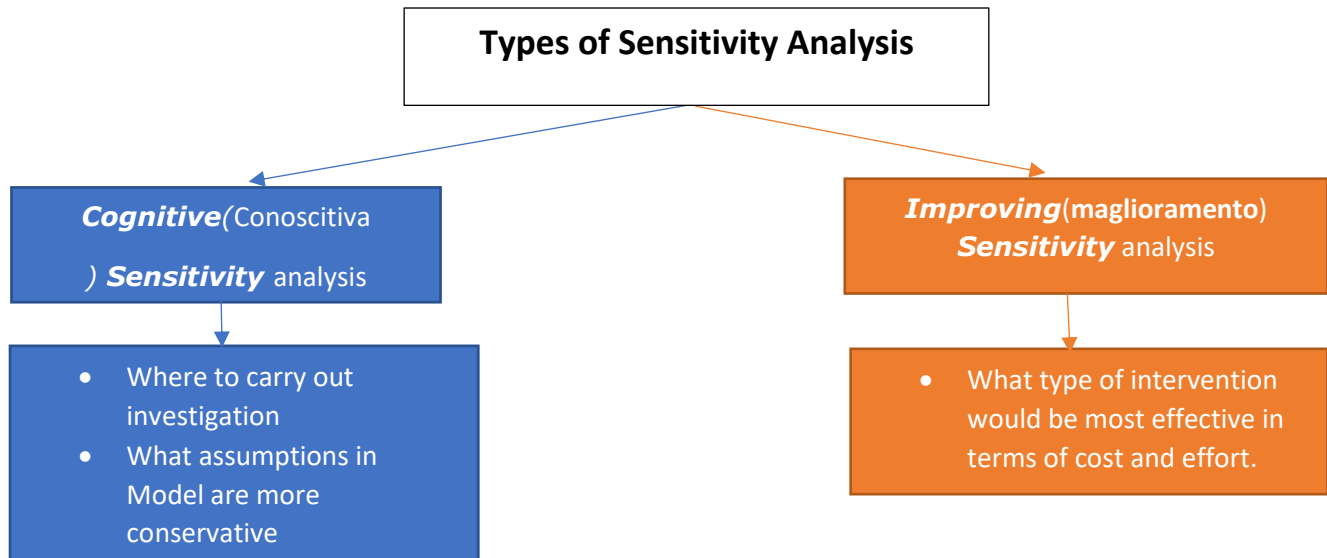
Sensitivity Analysis

We came to know by performing sensitivity analysis that

- What are the model assumptions/parameters are more sensitive in defining our model in case if investigation cannot be carried out
- where I should conduct the investigations as it has to do with the cost so it's important to know where the analysis should be carried out.

For example, the experiments done to find the strength of masonry is different from reinforced concrete wall and of steel. So, sensitivity analysis is important tool for carrying out a reasonable analysis. By sensitivity analysis we came to know where we should carry out our investigation in

order to have better level of knowledge about our structure. By performing the sensitivity analysis, we came to what type of intervention will improve the behavior of structure the most:



Stone Masonry has quite unpredictable mechanical properties.

(3MURI, 2018).

Sensitivity analysis is performed to get to know about the structure functioning of structure and to know about where it would be useful to perform the investigation on our structure in order to get a level of knowledge about our structure. As the tests performed for investigation of structure is associated costs so through the results of sensitivity analysis, we can make our investigations cost effective. Moreover, sensitivity analysis also tells us where the intervention would be most effective to make. It's possible to come to know about some investigation that would be least interested to make. As there are uncertainties related to application of loads and also with the material and mechanical properties of material due to multiple reasons. There could be degradation in the strength of material with passage of time and sometimes it's not possible to know all the mechanical properties for example to know about the mechanical properties of reinforcement steel inside the reinforced concrete beam. This kind of analysis reduce the current uncertainty about the model by determining about the importance of parameter in terms of investigation to be make. Sensitivity analysis consist of performing several pushover analyses.

The output of sensitivity analysis is sensitivity index obtained for a series of pushover analysis considering different combination of parameters as follows:

- 1st analysis is performed by considering the mean value of parameters from one group for example a masonry building Group.
- 2nd analysis is performed by taking average values of parameter of 1st group and min and max value of parameter of another group such as Reinforced concrete material group.

Vulnerability index is a ratio of PGA capacity to PGA demand ζ_E :

$$\zeta_E = \frac{PGA_c}{PGA_d} = \frac{a_{g,c} \cdot S}{a_{g,d} \cdot S}$$

in 3Muri vulnerability index is represented by symbol alpha as ratio of PGA of capacity to PGA of demand and is independent of soil type.

$$\alpha_{PGA} = \frac{PGA_C \cdot S}{PGA_D \cdot S}$$

The 3Muri calculates the **vulnerability index** represented by a by 3Muri software which is a ratio of capacity demand over acceleration demand for different limit states (SLV, SLD , SLO) automatically depending on specific return period.

Return Period T_R depends on location of building and depends on three parameters that decide the seismic hazard of building as follows:

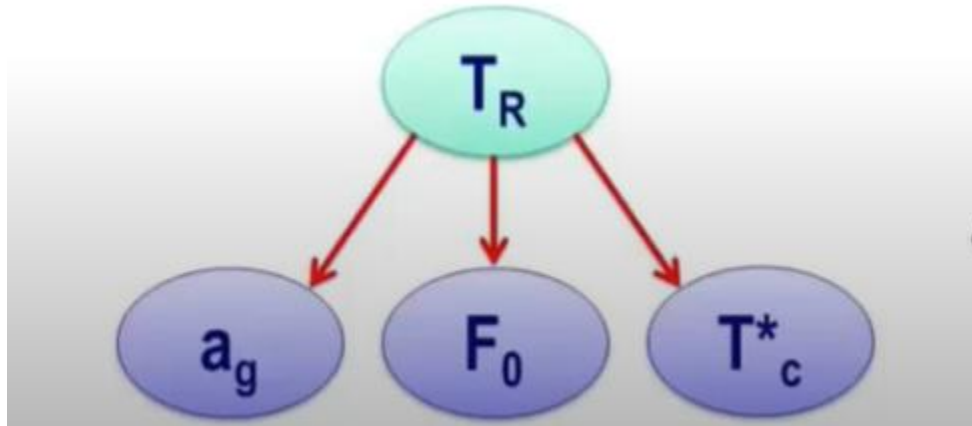


Figure 56

Where PGA_c is capacity peak ground acceleration coming from the pushover analysis and PGA_d is the demand peak ground acceleration that depends on seismic hazard and location of building. S is soil amplification factor that depends on soil type we have.

- when value of vulnerability index $\zeta E > 1$ check is verified as capacity is more than demand.
- However, when the vulnerability index $\zeta E < 1$ the resistance is less than demand and the element/structure has not passed the check and need to be retrofitted by some intervention.

Following series of pushover analysis are performed as follows calculating vulnerability index in each case.

- 1st force distribution applied proportional to the 1st mode shape or triangular distribution of force.
- Seismic action applied in main X and Y direction.
- The action forces applied for positive and negative directions.
- Force applied considering the accidental eccentricity of 5% as max size of building in orthogonal direction to that of building. To be conservative the worst condition of pushover analysis can be considered the one that gives the lowest value of vulnerability index to save some computational efforts.

the 3Muri software shows the output as sensitivity index of analysis in form of a *figure 89* as shown below:

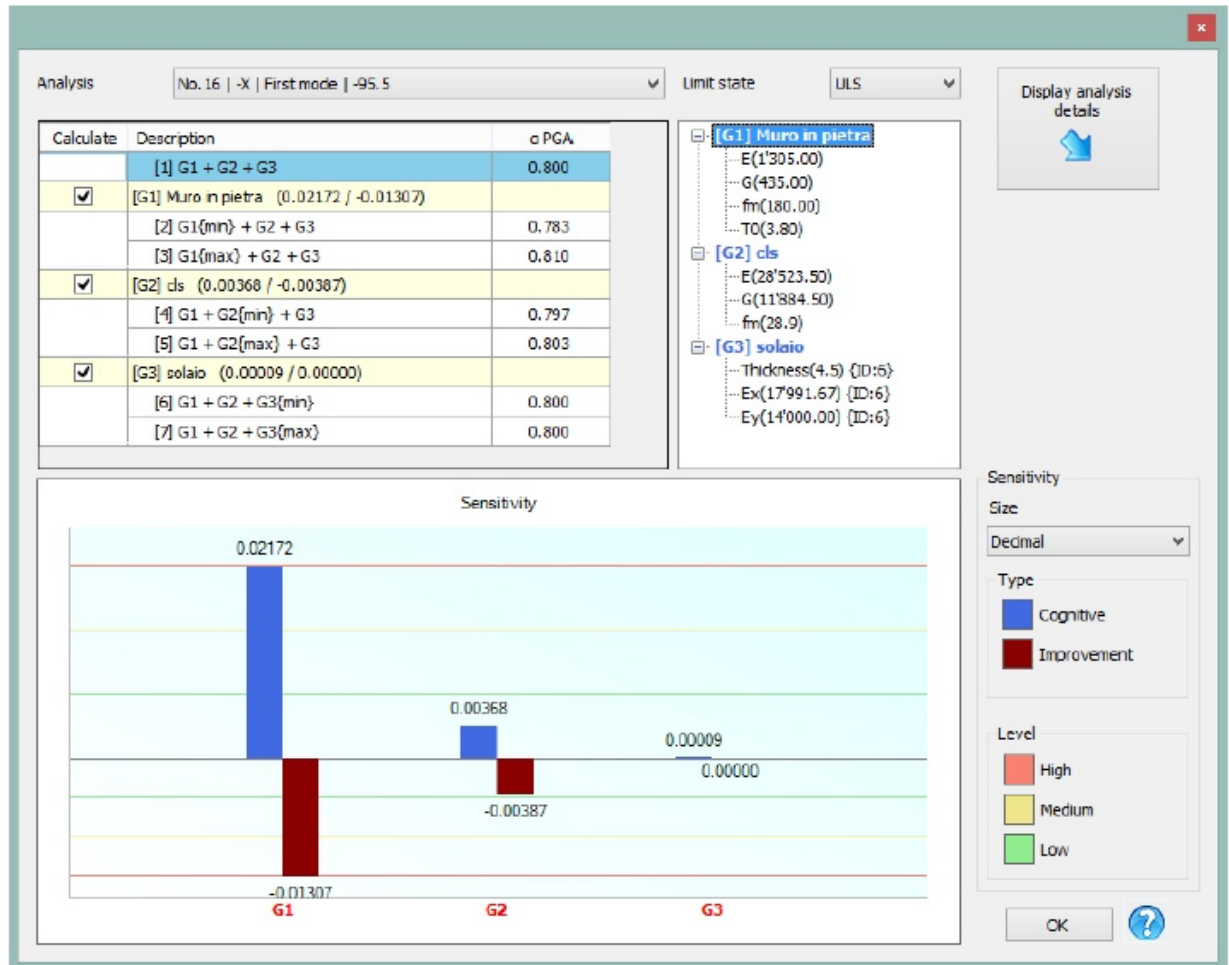


Figure 86: Example of output of Sensitivity analysis in terms of Sensitivity index

It's must be mentioned that the G1, G2 and G3 are the groups based on the different types of materials we have in our building which must not be confused with structural permanent and non-permanent loads. For Example, G1 represents all the group of masonry material, G2 can be taken for reinforced concrete material and G3 for any other types of material we have in our structure.

The analysis represents all pushover analysis result in single diagram as shown below in a figure 90 as an example:

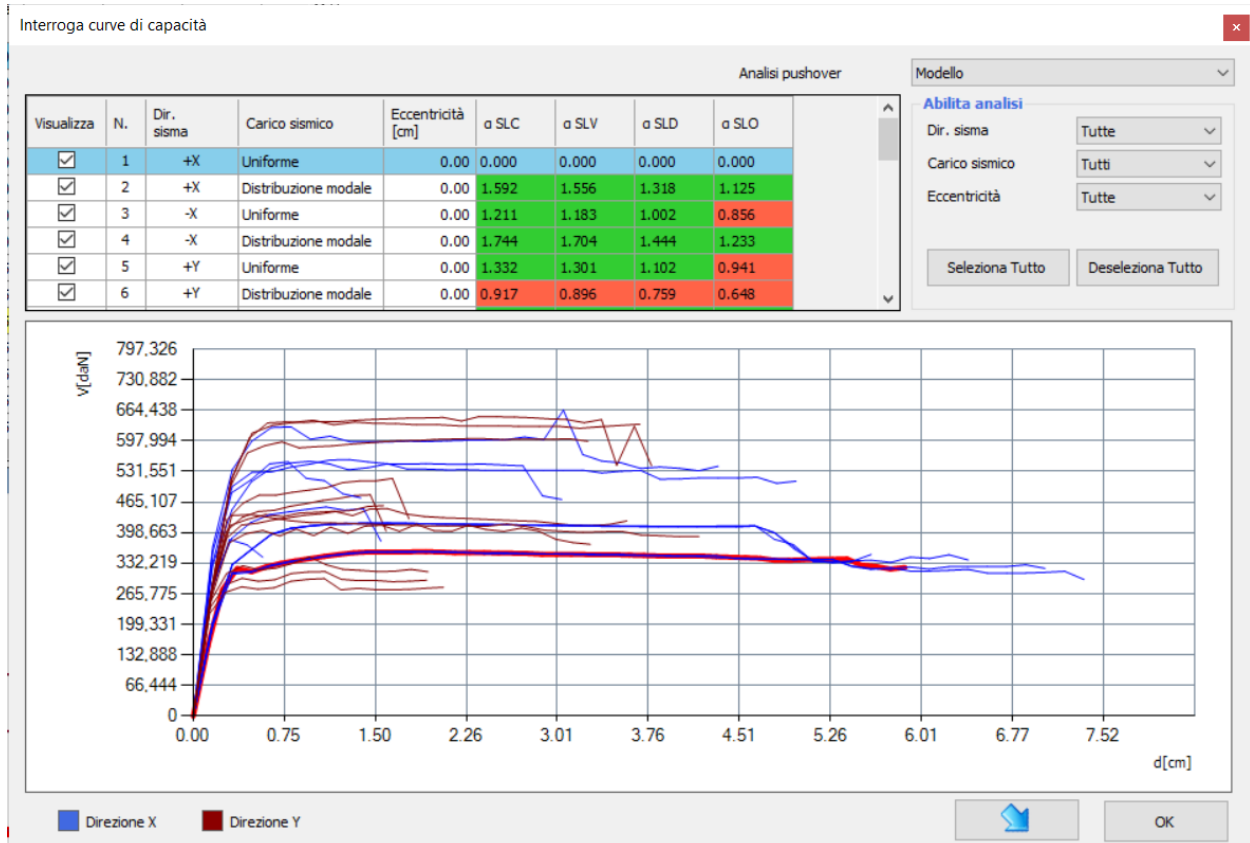


Figure 90: Example of Result of all pushover analysis

(3Muri Project X3 - Version 13.9.0.0 (29/09/2021), 2021, pp. 379-387)

Moreover, it's possible to see the stages of damage caused to the structure as result of applied force is applied.

Furthermore, it's possible to see the stages of damage caused to the structure as result of applied force is applied.

Types of interventions

- Intervention In existing masonry wall
- Introduction of reinforcement in the slab
- Improvement of slab and wall connection

During our model we must consider that there could be uncertainty associated with:

- **Loads applied on our structure** as we can't determine with certainty the exact values of loads such as wind or snow loads
- **Geometrical properties of elements** as for existing buildings if we have not carried out exhaustive investigation, we can have some uncertainty with the design drawings.

Local Verifications

The number of local mechanisms to be verified depends on the designer and he has to make judgment what kind of local failures can be activated and also it depends on type of structure we have. For example, for churches which are important buildings as part of cultural heritage the codes suggest to make 28 different types of local mechanism to be verified reported in the guideline of ministry of heritage.

(3MURI PROJECT, 20)

Types of Verifications

Following types of verifications are made using 3muri software:

- **Global static** verification
- **Out of plane bending** verification
- **Sensitivity analysis**

(3Muri Project X3 - Version 13.9.0.0 (29/09/2021), 2021)

Numerical Model of 3Muri

Reference Codes

There are number of reference codes implemented inside 3Muri such as Italian national code followed for construction, NTC (**Norme tecniche per le costruzioni**), Eurocode and Switzerland codes as can be seen in *figure 91* below. Moreover, it's also possible to choose an existing or new building while starting a new project.

Reference codes inside 3Muri

The screenshot shows a 'Model parameters' dialog box. It has three main sections: 'Building type', 'Selected Code', and 'Model type'. In the 'Building type' section, the 'Existing' radio button is selected and highlighted with a red rectangle. The 'Selected Code' section shows a list of codes with 'Italy (NT18)' highlighted by a red rectangle. The 'Model type' section shows a dropdown menu with 'Current status' selected. At the bottom, there are buttons for 'Parameters', 'OK', 'Cancel', and a help icon.

Flag	Code
Italy	Italy (NT18 circolare)
Italy	Italy (NT18)
Italy	Italy (NT08)
Euro	Euro Code
Euro	Euro Code (NL)
Netherlands	Netherlands (NPR9998)
Netherlands	Netherlands (NPR9998-2018)

Figure 91: standard Code implemented inside 3Muri

There is automatic parameter defined for each of reference codes such as partial safety factors and for materials and some other parameters as can be seen in figure 92 below. However, it's possible to modify them:

For every code there are some by default parameters such as partial safety factor coefficients for structural permanent and variable loads already defined inside the 3Muri software. However, it's possible to modify these parameters as can be seen in the *figure 92 below*:

Parameters library -- Euro Code --	
[1] Materials	
Existing: Drift-shear	0,004
Existing: Drift-Bending	0,008
Existing: FC-LC1	1,35
Existing: FC-LC2	1,2
Existing: FC-LC3	1
New: Drift-shear	0,004
New: Drift-Bending	0,008
Reduction factor for cracked stiffness	2
[2] Static calculation	
γG1	1,35
γG2	1,35
γQ	1,5
Q _{wind}	1,5
Q _{wind}	0,6
Dominant wind load	No
Initial eccentricity coefficient	450
Limit slenderness	27
[3] Misc	
Axis VM: Foundations	Method 2
Safety coefficient bearing capacity	1,4
[1] Bilinear parameters	
Intersection bilinear-pushover	0,7
[2] LS of Near Collapse (NC)	
Limit condition (NC)	Decay
Decay value	0,8
Make use of q* limit	No
q* limit	3
Make use of dt*/det* limit	Yes
dt*/det* limit	3
Displacement reduction factor	1
[3] LS of Significant Damage (SD)	
Limit condition (SD)	By NC
Storey height drift limit (SD)	0,02
Limit value coeff.	0,75

Figure 92: Reference parameters for Eurocode

Response Spectrum Definition

There are 2 options in which response spectrum can be defined inside the 3Muri:

- Parametric (in which its possible to select types of soil and other parameters that define the seismic hazards of site)
- Personalized (in which its possible to develop your own response spectrum by inputting the time period T and pseudo acceleration S_e in m/s^2).

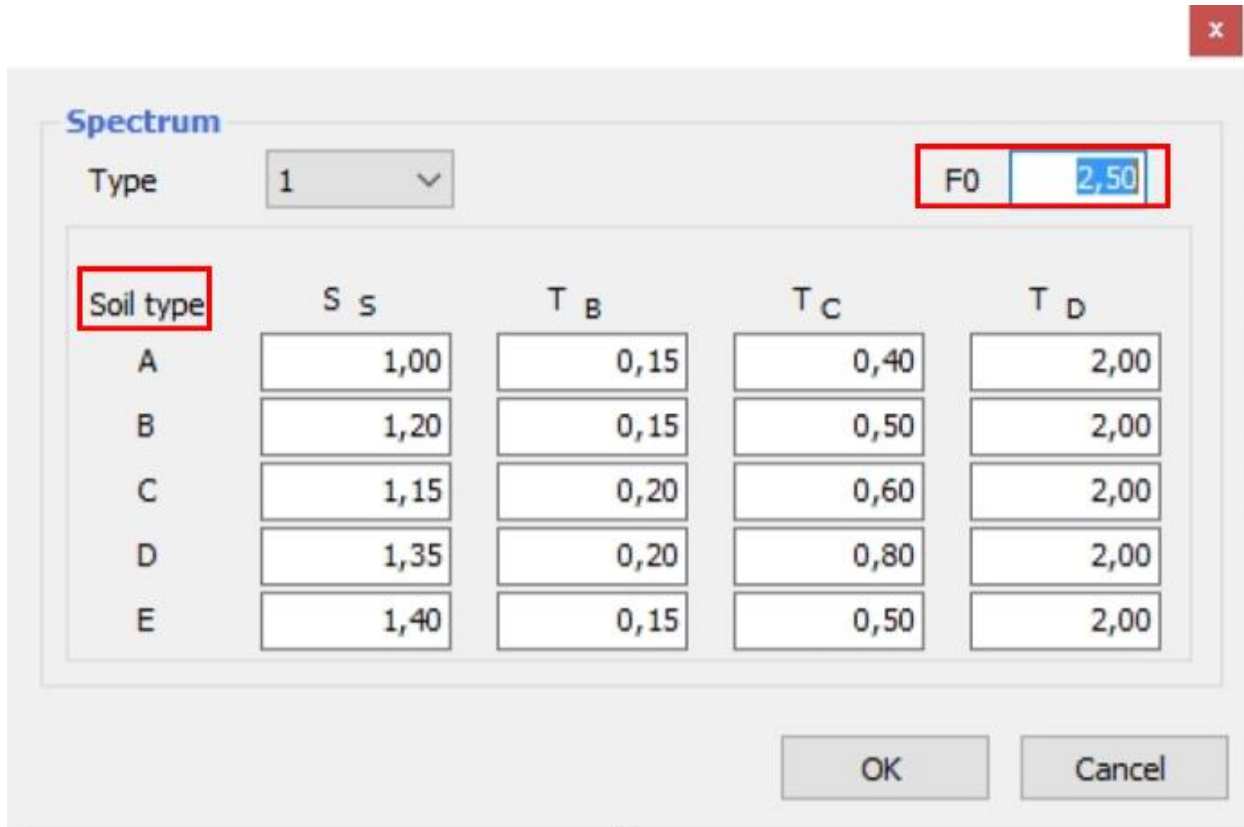
Just with Eurocode it's possible to define the parameters that define the seismic response spectrum parameters such as soil type from A to E and max value of amplification factor usually taken as 2,5 as can be seen in the figure 93 below:

Soil type	S s	T B	T C	T D
A	1,00	0,15	0,40	2,00
B	1,20	0,15	0,50	2,00
C	1,15	0,20	0,60	2,00
D	1,35	0,20	0,80	2,00
E	1,40	0,15	0,50	2,00

Figure 93

By Default, parameters inside Eurocode

Then it's possible also to change the parameters that define the seismic response spectrum by choosing the soil Types from A to E defined as reference in Eurocode, amplification factor F_0 which is normally taken as 2,5 as can be seen in the figure 94 below:



Spectrum

Type: 1

F0: 2,50

Soil type

	S s	T B	T C	T D
A	1,00	0,15	0,40	2,00
B	1,20	0,15	0,50	2,00
C	1,15	0,20	0,60	2,00
D	1,35	0,20	0,80	2,00
E	1,40	0,15	0,50	2,00

OK Cancel

Figure 94: Response spectrum Parameters in Eurocode

On the other hand, using the Italian national code **NTC-18** its directly possible to find the site by choosing the soil type and coordinates of city and seismic parameters will be calculated automatically that define the response spectrum for city of L'Aquila in central ITALY as our building is located there as can be seen in *figure 95* below:

×

Site parameters
City/town
Longitude
Latitude
Nominal life
Use classes

Seismic hazard parameters

	CLS	ULS	DLS	OLS
a_g [m/s ²]	<input type="text" value="3.739"/>	<input type="text" value="2.940"/>	<input type="text" value="1.223"/>	<input type="text" value="0.963"/>
F_0	<input type="text" value="2.43"/>	<input type="text" value="2.38"/>	<input type="text" value="2.32"/>	<input type="text" value="2.34"/>
T_C^* [s]	<input type="text" value="0.37"/>	<input type="text" value="0.36"/>	<input type="text" value="0.29"/>	<input type="text" value="0.28"/>
T_R	<input type="text" value="1462"/>	<input type="text" value="712"/>	<input type="text" value="75"/>	<input type="text" value="45"/>

Figure 95: Seismic demand for L'Aquila City

Carico sismico

×

Forma spettro Parametrica (da reticolo) ▾

Classe suolo B ▾

Diagramma Spettro

Calcola

	SLC	SLV	SLD	SLO
► Verifica	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
$a_g [m/s^2]$	3.28	2.56	1.02	0.77
F_0	2.40	2.36	2.33	2.40
$T^*_C [s]$	0.36	0.35	0.28	0.27
T_R	975.00	475.00	50.00	30.00
S_S	1.08	1.15	1.20	1.20
$T_B [s]$	0.16	0.16	0.13	0.13
$T_C [s]$	0.49	0.47	0.40	0.39
$T_D [s]$	2.94	2.64	2.02	1.92

Categoria topografica T1 ▾

Fattore di amplificazione

S_T 1.0

1.000 ▾

OK
Annulla
?

Figure 96: Seismic Load using NTC-18

Modules Inside 3Muri

There are 4 specific modules dedicated inside 3Muri as follows and can be seen in figure 97 below:

- Alignment
- Structure
- Global Analysis
- Local Mechanism

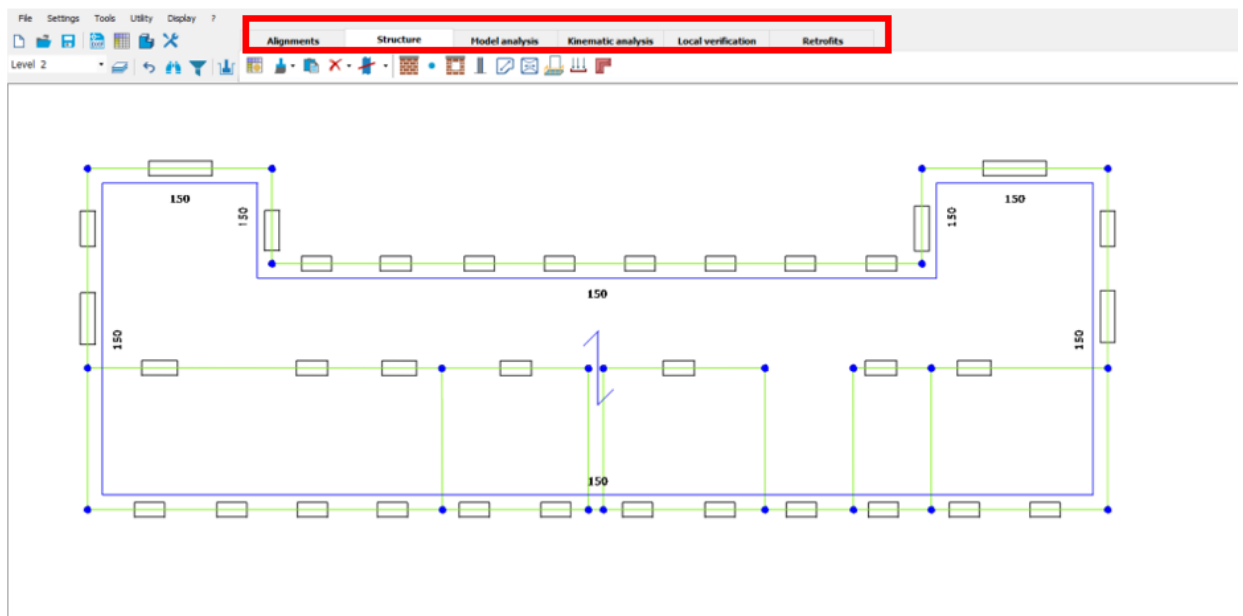


Figure 97: Different Modules inside 3Muri

Units and formats

It's possible to use units of measurement of some standard systems already made inside the 3Muri as can be seen in the figure 98 below:

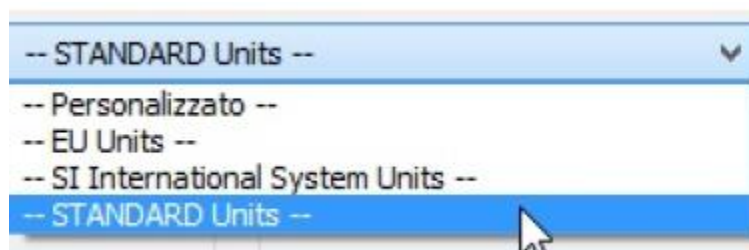


Figure 98: Units setting inside 3Muri

The by default units in the 3Muri software is cm however it's possible to choose any other units of measurements such as m or mm as can be seen in the figure 98 above:

Geometrical Model

It's possible to import the .DXF file directly into the 3Muri software implementing the BIM methodology inside 3Muri as can be seen in the *figure 99* below:

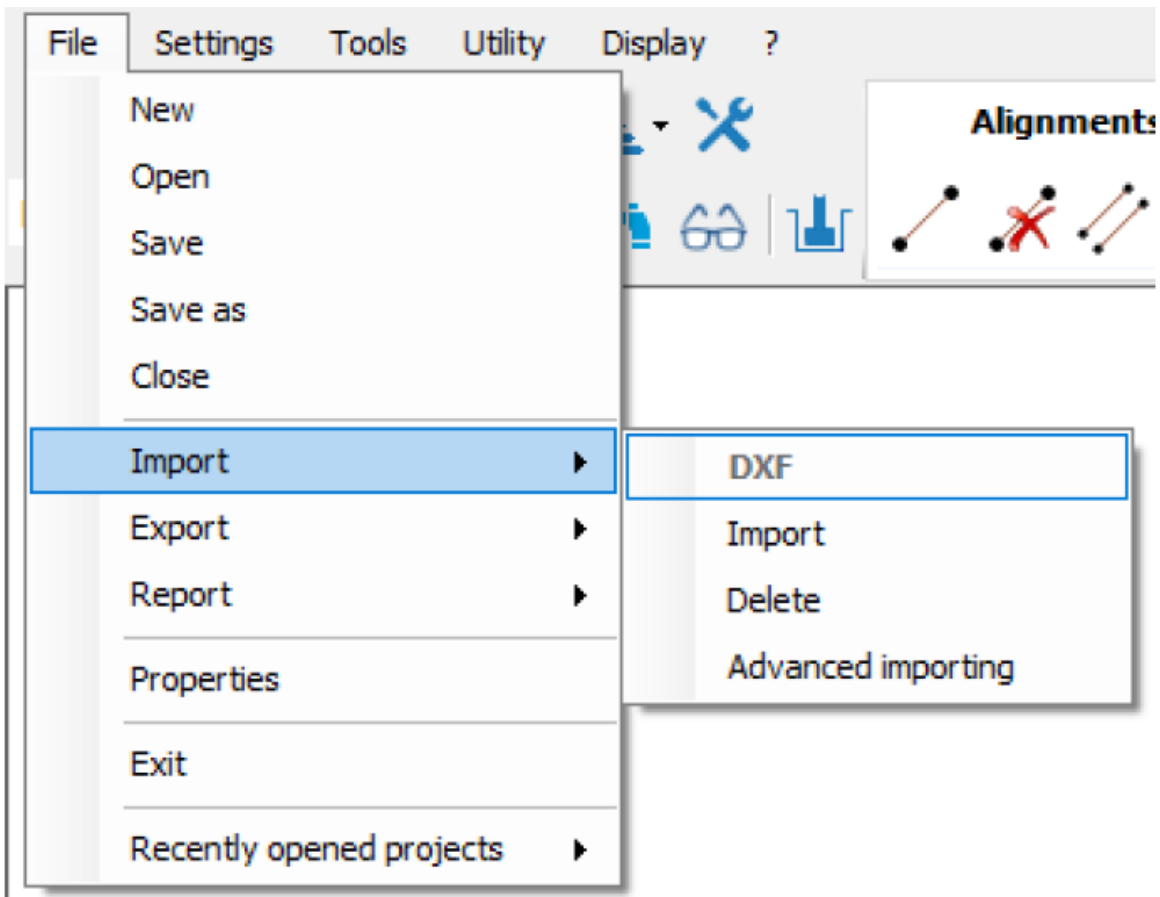


Figure 99: Importing the model inside 3Muri

It's possible to import directly the .DXF file into the 3Muri as I have done and then using the alignment command to draw the walls as I have done and can be seen in the figure 100 below. It's possible to import the .DXF file or .ifc file directly into 3Muri and then using the centerline as a reference it's possible to draw the walls manually.



Figure 100: Importing the .DXF file inside 3Muri

Importing directly the .DXF file into the 3Muri is easiest way to model our structure inside 3Muri and then using the alignment command to draw the walls as I have done and can be seen in the figure 101 below:

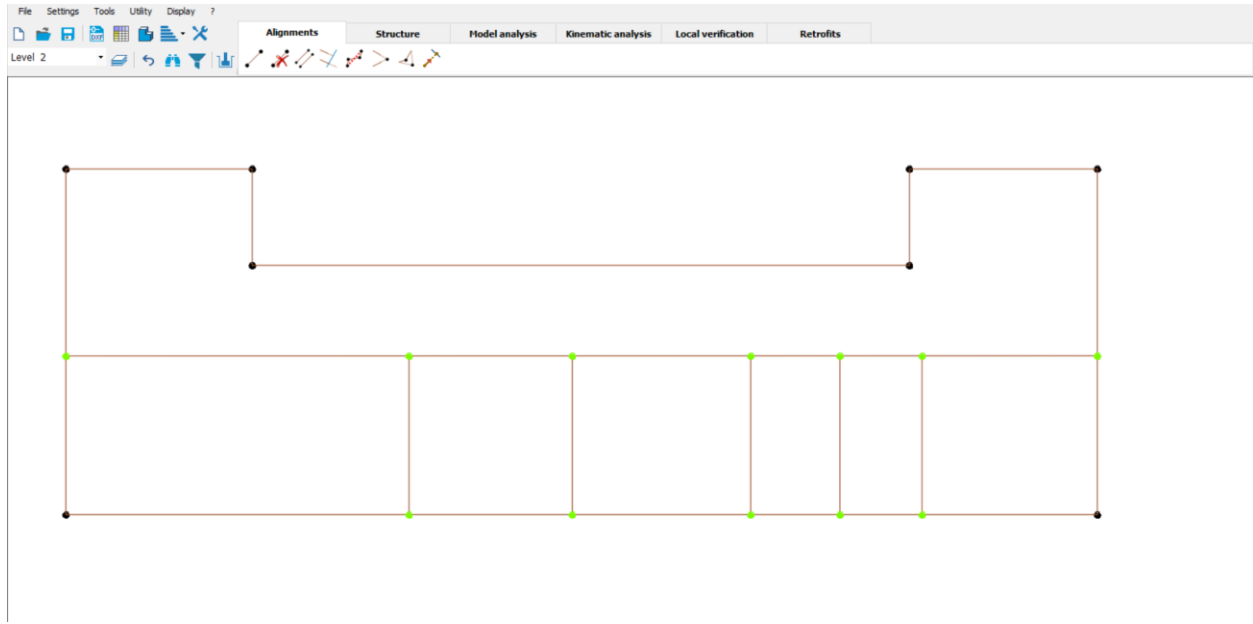


Figure 101: Geometrical model realized with alignment tool

Structure Properties

In structural Tab it's possible to perform following tasks:

- Modelling the openings
- Defining and assigning the material properties of walls and Roofs
- Applying the structural loads on slabs

By following the .DXF file its possible to construct above the model and also the openings.

Walls

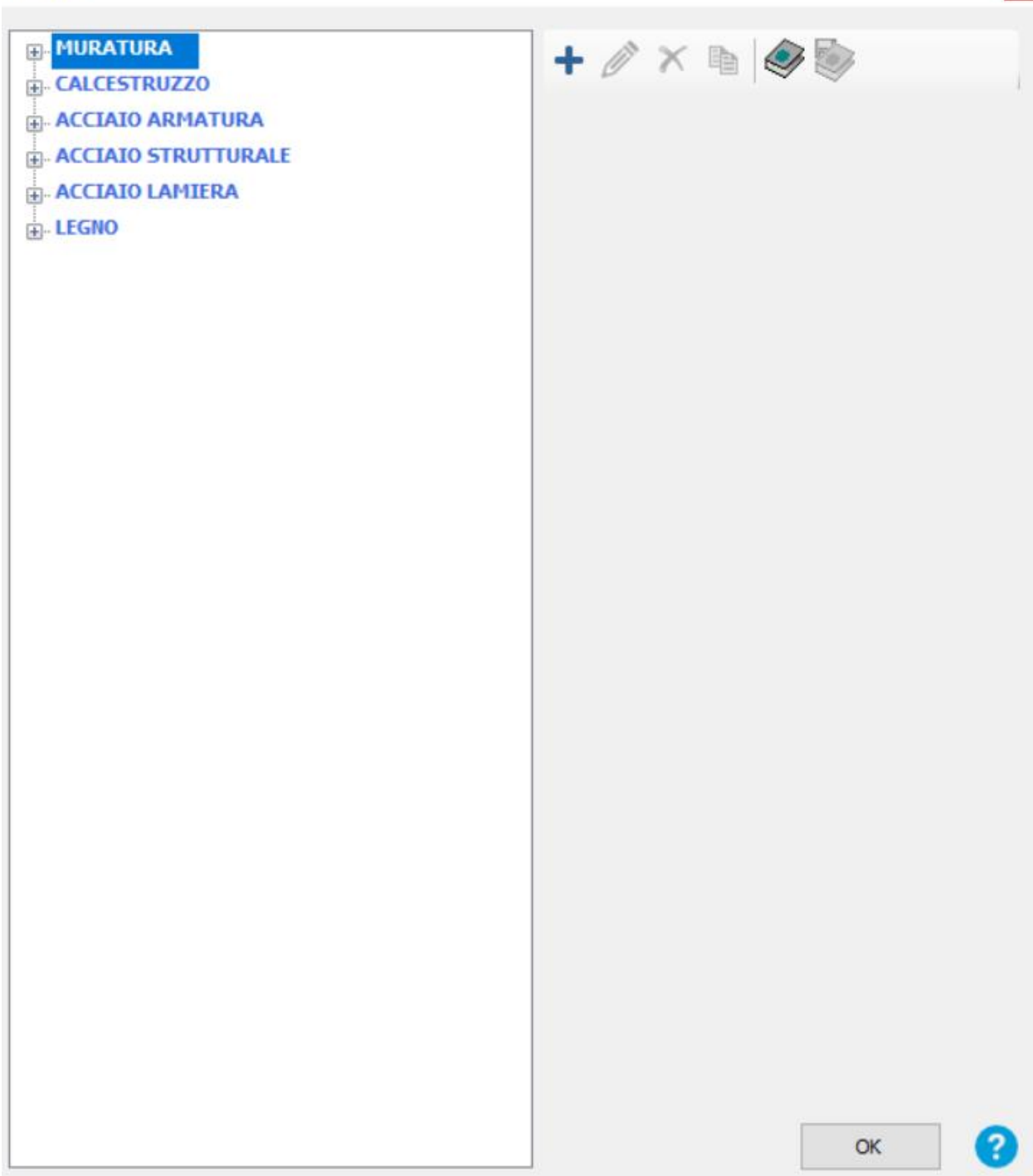


Defining Material Properties for walls

It's possible to define a new material or an existing material inside 3Muri. When a new or existing material is defined the following screens appear as can be seen in figure below:

There are number of types of materials that can be defined inside the 3Muri as can be seen follows in figure below.

Materiali



Moreover, there are already different types of materials of masonry, concrete and steel sections defined in 3Muri and there is a complete library of materials available. Each Material is associated with some mechanical properties. It's also possible to define these material properties manually by user. As there is uncertainty in the material resistance particularly with existing buildings to take into account this fact, level of knowledge is used that reduces the

strength of material by a safety factor depending on level of knowledge we have about our structure as suggested by regulations Eurocode and also by NTC Italian national code .so, with existing material it's possible to define the level of knowledge from 1 to 3 that is associated with different confidence factors we have about our 117structure as defined in Eurocode and NTC-18 as follows:

- Limited information (LC1)
- Extended Information (LC2)
- Exhaustive Information (LC3)

Confidence factor can be inserted while defining the material property as can be seen in the figure 102 below:

The screenshot shows a table of material properties on the left and a 'Knowledge level' dropdown menu on the right. The table has the following data:

E [N/mm ²]	27085
G [N/mm ²]	11285
w [kN/m ³]	25
f _{cm} [N/mm ²]	20
f _{ck} [N/mm ²]	12
α _{cc}	0.85
FC	1.35
γ _c	1.5

The 'Knowledge level' dropdown menu is open, showing the following options:

- Limited information -- LC1
- Limited information -- LC1
- Extended information -- LC2
- Exhaustive information -- LC3
- more --

Figure 102: Confidence Factors depending on Level of Knowledge

The reduction in the material properties for existing buildings is automatically applied by the software depending on chosen level of knowledge. There are 3 levels of knowledge that can be defined and in case of level of knowledge 3 its possible also to input the results of experiments conducted as can be seen in the figure 103 below:

The screenshot shows a form for inputting experimental data. On the left, a text box says 'If working with knowledge level 3'. On the right, a text box says 'The experimental values derived from the tests are requested.' Below this, there is a dropdown menu labeled 'Experimental data' with the value '3' selected. Below the dropdown, there is a 3x4 grid of input fields, all containing the value '0'. At the bottom right of the grid is a button labeled 'Confirm values'. Below the grid, a text box says 'Clicking on "Confirm values" defines the computation values.'

Figure 103: Experiments Results input in case of Level of knowledge 3

The 3 level of knowledge as defined inside NTC-18 for masonry structures are as follows:

Knowledge Level	Geometry (as-built)	Investigations	Tests	Analysis Method	Confidence factor FC	Material Properties (refer to table C.8.5.I) →
KL1 LC1	From original design drawings with random investigations or full surveys. (Historical / critical analysis and geometric survey)	Limited	Limited tests	All	1.35	f (minimum values) τ_0 , f_{v0} (minimum values) E, G (average values)
KL2 LC2		Extended	Extended		1.20	f (average values) τ_0 , f_{v0} (average values) E, G (average values)
KL3 LC3		Exhaustive	Exhaustive		1.00	f τ_0 , f_{v0} E, G (depend on table values modified by results from experimental tests, see C.8.5.4.)

Figure 104: Level of knowledges for Masonry Structures

Definition of Materials

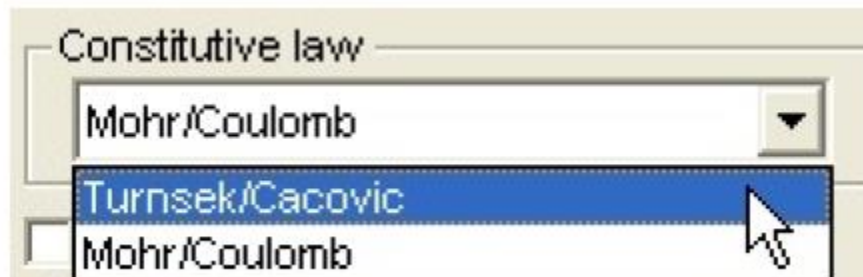
There are various types of materials that can be defined inside the 3muri software. Various types of masonry such as stone and clay are available, various sections of steel sections and all classes of concrete are available already inside the 3Muri. Moreover, it's possible to define the section of its own inside software and save it for future use.

Types of Bonds of Masonry

There are 2 types of constitutive laws that take into account the bonds that can be defined for the **shear bond of masonry blocks** as follows:

- **Mohr-Coulomb Bond Criteria**
- **Turnsek Cacovic Bond Criteria**

Mohr-Coulomb Bond Criteria involves presents sliding type failure and it is recommended to use for new masonry structures. On the other hand, **Turnsek cacovic** model is recommended to be used for **existing masonry** buildings and it involves diagonal shear failure.



Furthermore, it's also possible to improve the material properties by defining the improvement parameters as suggested in the codes. For Masonry Material Stone Masonry has been defined with level of knowledge 3 associated with the confidence factor of 1 as can be seen in *figure* below:

For Masonry Following material properties can be defined as follows:

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.

Following types of masonry materials can be defined inside 3Muri software:

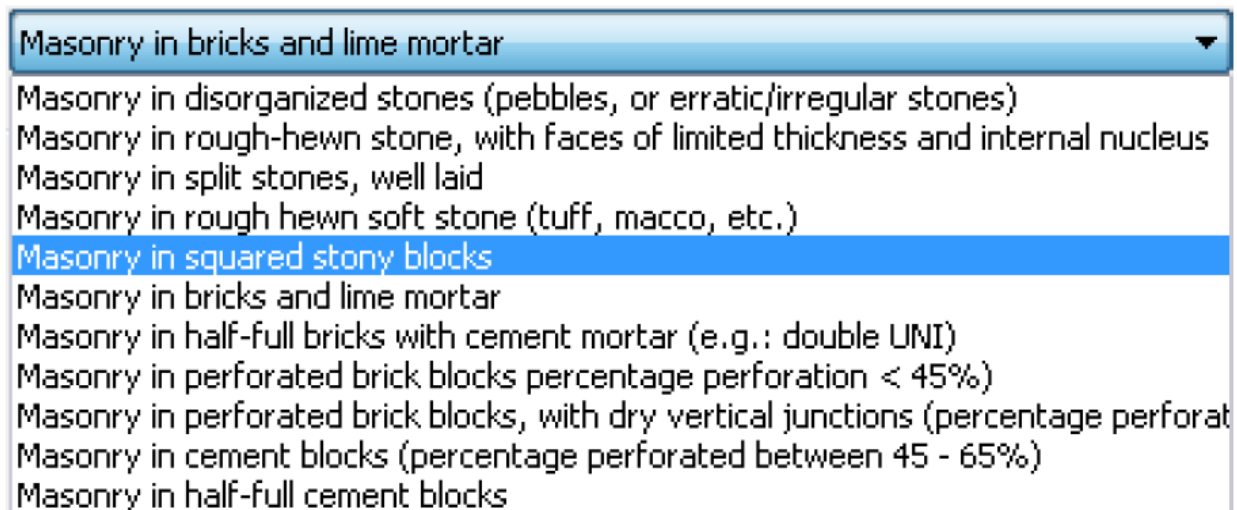


Figure 105: Types of Masonry Materials inside 3Muri

Following material properties can be defined for concrete both for new or existing concrete material as follows:

PARAMETER	EXISTING	NEW	DESCRIPTION
E	✓	✓	Longitudinal modulus of elasticity
G	✓	✓	Tangential modulus of elasticity
W	✓	✓	Specific weight
f_{cm}	✓	✓	Medium compressive strength
f_{ck}	✓	✓	Characteristic compressive strength
α_{cc}	✓	✓	Coefficient that takes into account long term effects on the compressive strength and of unfavorable effects resulting from the way the load is applied.
FC	✓	✗	Confidence factor
γ_c	✓	✓	Safety factor of the resistance of the material to be used in order to perform the static verifications.

Figure 106

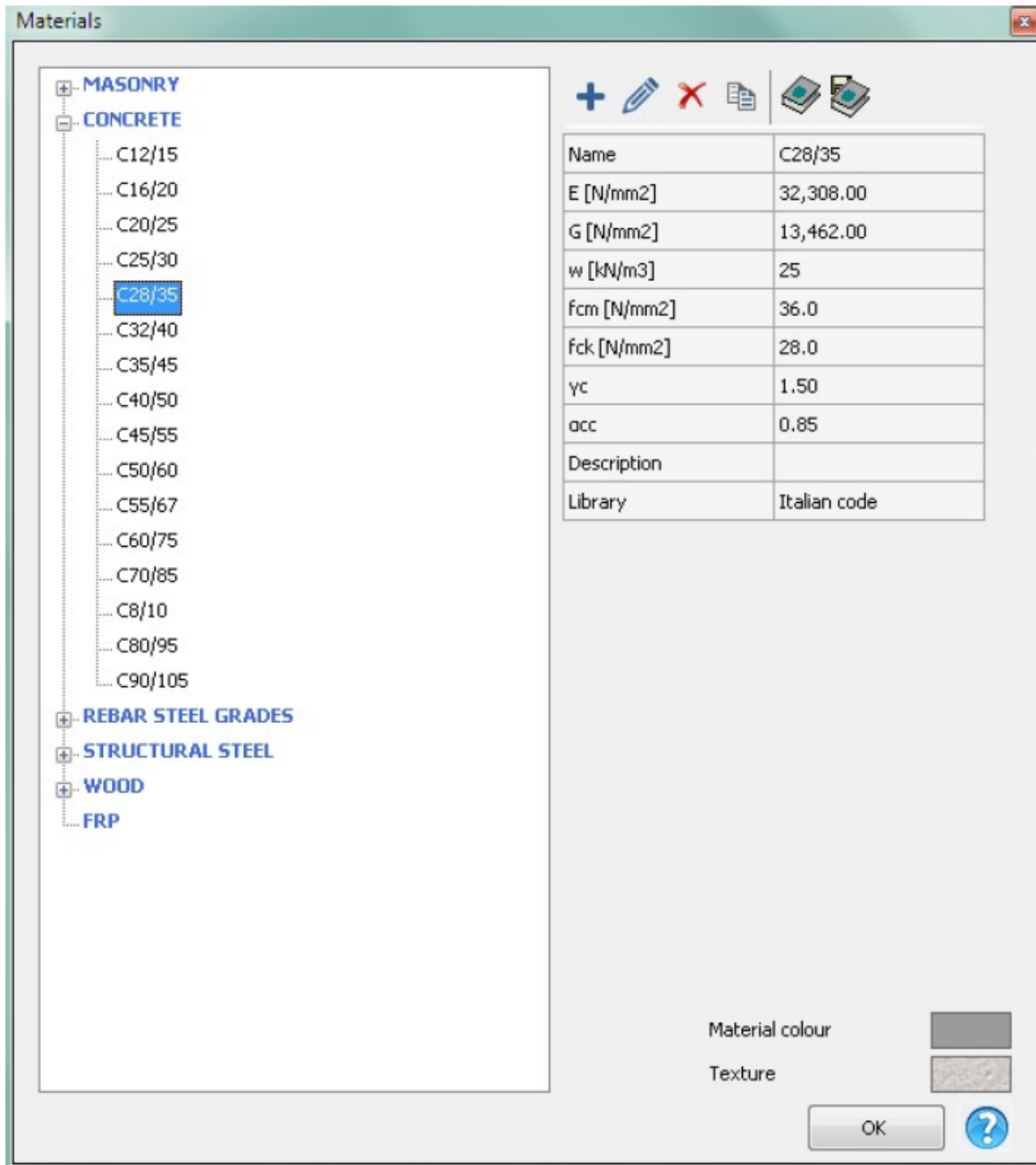


Figure 107: types of Materials defined Inside 3Muri

For my model of town hall of Pezzoli there were two types of masonry materials were defined as there were some tests performed on this building. In particular double flat jack tests was performed on town

hall of Pezzoli to find the mechanical properties of types of masonry we have. It was observed that the elastic modulus of masonry structure stone masonry was quite different than the reference values we have from the NTC-18. However, following two types of masonry were defined for modelling the walls as can be seen in the figure 108 below:

Tipologia di muratura	f_m (N/cm ²)	τ_0 (N/cm ²)	E (N/mm ²)	G (N/mm ²)	w (kN/m ³)
	Min-max	min-max	min-max	min-max	
Muratura in pietrame disordinata (ciottoli, pietre erratiche e irregolari)	100 180	2,0 3,2	690 1050	230 350	19
Muratura a conci sbozzati, con paramento di limitato spessore e nucleo interno	200 300	3,5 5,1	1020 1440	340 480	20
Muratura in pietre a spacco con buona tessitura	260 380	5,6 7,4	1500 1980	500 660	21
Muratura a conci di pietra tenera (tufo, calcarenite, ecc.)	140 240	2,8 4,2	900 1260	300 420	16
Muratura a blocchi lapidei squadriati	600 800	9,0 12,0	2400 3200	780 940	22
Muratura in mattoni pieni e malta di calce	240 400	6,0 9,2	1200 1800	400 600	18
Muratura in mattoni semipieni con malta cementizia (es.: doppio UNI foratura $\leq 40\%$)	500 800	24 32	3500 5600	875 1400	15
Muratura in blocchi laterizi semipieni (perc. foratura < 45%)	400 600	30,0 40,0	3600 5400	1080 1620	12
Muratura in blocchi laterizi semipieni, con giunti verticali a secco (perc. foratura < 45%)	300 400	10,0 13,0	2700 3600	810 1080	11
Muratura in blocchi di calcestruzzo o argilla espansa (perc. foratura tra 45% e 65%)	150 200	9,5 12,5	1200 1600	300 400	12
Muratura in blocchi di calcestruzzo semipieni (foratura < 45%)	300 440	18,0 24,0	2400 3520	600 880	14

Figure 108: Mechanical Properties of Masonry walls

S287972
Thesis Report

Tipologia di muratura	Malta buona	Giunti sottili (<10 mm)	Ricorsi o listature	Connessione trasversale	Nucleo scadente e/o ampio	Iniezione di Miscela leganti	Infonaco armato ¹
Muratura in pietrame disordinata (ciottoli, pietre erratiche e irregolari)	1,5	-	1,3	1,5	0,9	2	2,5
Muratura a conci sbazzati, con paramento di limitato spessore e nucleo interno	1,4	1,2	1,2	1,5	0,8	1,7	2
Muratura in pietra a spacco con buona tessitura	1,3	-	1,1	1,3	0,8	1,5	1,5
Muratura a conci di pietra tenera (tufo, calcarenite, ecc.)	1,5	1,5	-	1,5	0,9	1,7	2
Muratura a blocchi lapidei squadriati	1,2	1,2	-	1,2	0,7	1,2	1,2
Muratura in mattoni pieni e malta di calce	1,5	1,5	-	1,3	0,7	1,5	1,5

Tabella C8A.2.2 della Circolare 617/2009)

Figure 109: types of Masonry walls improvement factors

Proprietà materiale

Tipo

Nome

Masonry type 1

Colore materiale

Texture

E [N/mm²]

Eh [N/mm²]

G [N/mm²]

w [kN/m³]

f_m [N/cm²]

f_{hm} [N/cm²]

f_k [N/cm²]

τ [N/cm²]

FC

γ_m

Maschio drift taglio

Maschio drift flessione

Fascia drift taglio

Fascia drift flessione

Fascia accoppiata drift

3385.72

3385.72

1244

22

840

630

588

12.6

1

3

0.005

0.01

0.015

0.015

0.02

Condizione del materiale

Esistente

Nuovo

☒

Rigidità fessurata

☐

Malta scadente (resistenza < 0.7 [N/mm²])

☐

Definizione utente

☐

Limita spostamento ultimo

Tipo legame

Fascia

Degrado di resistenza a un valore residuo

Taglio

Muratura irregolare (Turnsek/Cacovic)

☐

Considera contributo a trazione fascia

Parametri muratura

Parametri di miglioramento

Definisci materiale multistrato

Descrizione

Libreria

Italia (NT18 circolare)

Normativa

OK

Annulla

Figure 110: Mechanical properties of Type-1 Masonry

The values of elastic modulus used for stone masonry were used as derived from the tests. While the compressive and shear strength of were taken as average values taken from the codes based on the types of masonry we have and average value was taken. the value of shear Modulus G was taken as calculated from young modulus E considering the linear elastic relationship between E and G and considering the poison ratio value of 0,3 as follows:

124

$$G = E / [2(1 + \nu)]$$

The level of knowledge used for defining the mechanical properties of was taken as. the figure 111 below shows the mechanical properties defined for type-1 Masonry were as follows:

Definizione parametri materiali

Tipo muratura: Muratura a blocchi lapidei squadrati

Livello conoscenza: -- Esaustive indagini -- LC3 FC: 1

Dati sperimentali: 1 Metodo prova: Martinetto piatto doppio

	f_m [N/cm ²]	τ [N/cm ²]	f_{v0} [N/cm ²]	E [N/mm ²]	G [N/mm ²]	w [kN/m ³]
Valore calcolato	700.00	10.50	23.00	2,821.43	1,036.67	22
Minimo tabellare	580.00	9.00	18.00	2,400.00	800.00	
Massimo tabellare	820.00	12.00	28.00	3,300.00	1,100.00	
k	2.00	1.00	0.00	1.50	1.00	
Prova 1	0.00	0.00	0.00	0.00	0.00	

Conferma valori

Normativa OK Annulla

Figure 111: Type-1 Masonry Material Properties

In similar manner the mechanical properties of 2nd type of masonry material were defined which was Brick Masonry and has following mechanical properties:

Definizione parametri materiali

	f_m [N/cm ²]	τ [N/cm ²]	f_{v0} [N/cm ²]	E [N/mm ²]	G [N/mm ²]	w [kN/m ³]
Valore calcolato	522.50	10.00	20.00	3,214.29	1,320.00	18
Minimo tabellare	260.00	5.00	13.00	1,200.00	400.00	
Massimo tabellare	430.00	13.00	27.00	1,800.00	600.00	
k	2.00	1.00	0.00	1.50	1.00	
Prova 1	0.00	0.00	0.00	0.00	0.00	

Figure112: Mechanical Properties of type-2 Masonry

Table below elaborates all the mechanical properties used for the modelling of walls. There all the external walls were modelled as stone masonry (type-1) while the internal walls were modelled as brick Masonry (type-2).

Mechanical Properties							
wall Types	Masonry Types	E[N/mm ²]	Improvemet factor	f _m (compressive strength) [N/cm ²]	E [pa]	poisson Ratio, v	G [pa]
Masonry type-1	muratura in blocchi lapidei squadrati	4707	1.2	600-800	2.80E+09	0.3	1.08E+09
Masonry type-2	muratura in mattoni pieni	2800	1.5	240-400	4.50E+09	0.3	1.73E+09

TABLE-2 Mechanical properties of Masonry walls

Modeling Assumptions

While modelling the walls its assumed that there is no resistance of walls out of plane. Also, out of plane resistance of floors is taken as null for purpose of safety. Special Care should be taken in modelling of slab. If we overestimate the stiffness if slab, it will make over structure more rigid overall and it will reduce the ductility of structure and eventually will result in the lower displacements of structure than reality. After Creating 1 floor it's possible to Duplicate the floor for upper levels if building is symmetric in elevation. Moreover, it's also possible to model different kind of Flat or Flaps or vaults like.

Walls-----→ Stone Masonry

Roof and Stairs-----→ RC Roof

External Elevator-----→ Steel



Assign of Material Properties

It's possible to make masonry panels and masonry panels with reinforced tie beams, masonry panels with steel or wood beams. Table Moreover, it's possible to change the thickness of masonry walls above or below the windows as can be seen in *figure 113* below. Furth more the thickness of wall below the window was defined differently as 41cm than the wall thickness.

Definizione caratteristiche

The screenshot shows the 'Definizione caratteristiche' (Define characteristics) dialog box for a 'Pannello murario' (Masonry panel). The dialog has several tabs at the top: 'Pannello murario' (selected), 'Pannello + Cordolo C.A.', 'Pannello + Trave Acciaio / Legno', and 'Pannello + Catena'. Below these are sub-tabs: 'Setto C.A.', 'Trave C.A.', 'Trave Acciaio / Legno', 'Catena', and 'Nessuna definizione'. The 'Pannello murario' tab is active, showing the following settings:

- Quota:** 850 [cm]
- Altezza:** 450 [cm]
- Spessore:** 60.0 [cm]
- Tipo architrave:** Calcestruzzo (dropdown)
- Verifiche statiche:** Eccentricità: 0.0 [cm], ☒ Esposto al vento
- Materiale:** Masonry_type_1 (dropdown)
- ☐ Muratura armata/rinforzi
- ☒ Sotto/Sopra finestra
 - SottoFinestra:** Spessore: 41.0 [cm], Masonry_type_1 (dropdown)
 - SopraFinestra:** Spessore: 60.0 [cm], Masonry_type_1 (dropdown)
- ☐ Non unire durante la mesh

At the bottom, there are checkboxes for 'Non caricabile da solaio', 'Caratteristiche fondazione', and 'Interventi', followed by 'OK' and 'Annulla' buttons.

Figure 73: Assignment of Material properties to external wall of 2nd Floor

Definizione caratteristiche

Figure 114: Assignment of material properties to Internal Walls of 2nd Floor

The table below summarizes the thickness of external and internal walls modelled inside 3Muri for the both 2 stories as follows:

WALLS Thicknesses		
Floor No.	wall Type	Average Wall Thickness [cm]
Raised Ground Floor	External	63
	Internal	54
1st Floor	External	60
	Internal	50

Openings

It's possible to create the openings such as doors and windows in a fast manner inside the 3Muri using the .dxf File. It's possible to create the openings using the Centrepont of opening or by selecting the 2 edge points of the opening.

Openings Height[m]	
Windows	
Bottom Height of External Windows	0.85
Height of External Windows	2.45
Bottom Height of internal Windows	0.85
Height of internal Windows	1
Doors	
Height of External Doors	3.3
Height of internal Doors	2.4

TABLE-3: Openings Detail

Floor Slabs

Slabs can be modelled with various typologies starting from the fully rigid floors to floors having no stiffness and fully deformable. However, I decided to model the slab as **rigid diaphragms** as there replaced with RC Roofs later. For rigid floor there would be no requirement on the mechanical properties of slab and floor will be modelled as Rigid.

Giving the input geometrical properties of slab, software automatically calculates the stiffness of Slab, structural permanent loads G_1 and other mechanical properties of slab. Then it's also possible to assign additional non-structural loads G_2 .

Solaio

×

Carichi




Quota [cm]

Gk1 [daN/m²]

Gk2 [daN/m²]

Qk [daN/m²]

Verifiche statiche



Lungh. appoggio [cm]

Δ quota estradosso [cm]

Δ quota intradosso [cm]

☒ Carico dominante

NT18 circolare

ψ₀
 ψ₁
 ψ₂

☐ Categoria ζ_v

Tipo

-- Impalcato rigido --



Spessore	<input type="text" value="0.0"/> [cm]
G	<input type="text" value="0.00"/> [N/mm ²]
Ex	<input type="text" value="0.00"/> [N/mm ²]
Ey	<input type="text" value="0.00"/> [N/mm ²]
V	<input type="text" value="0.00"/>

Scarico masse

☐ Monodirezionale
 ☒ Bidirezionale

Scarico direzione principale %

Visualizzazione

Colore materiale 
 Texture 

☐ Interventi

?

Figure 115: Loads on the Floors

There are various types of Masonry, Reinforced concrete and floors mixed with different material types that can be defined inside the 3Muri software as follows:

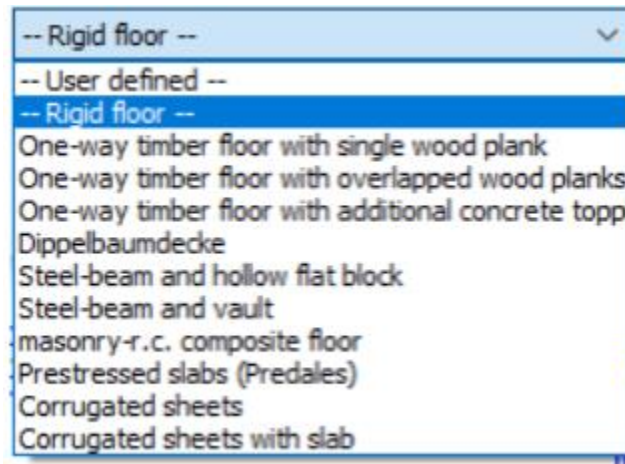


Figure 116: Various Floor Typologies inside 3Muri

The structure model realized with walls and opening can be seen in 117 below:

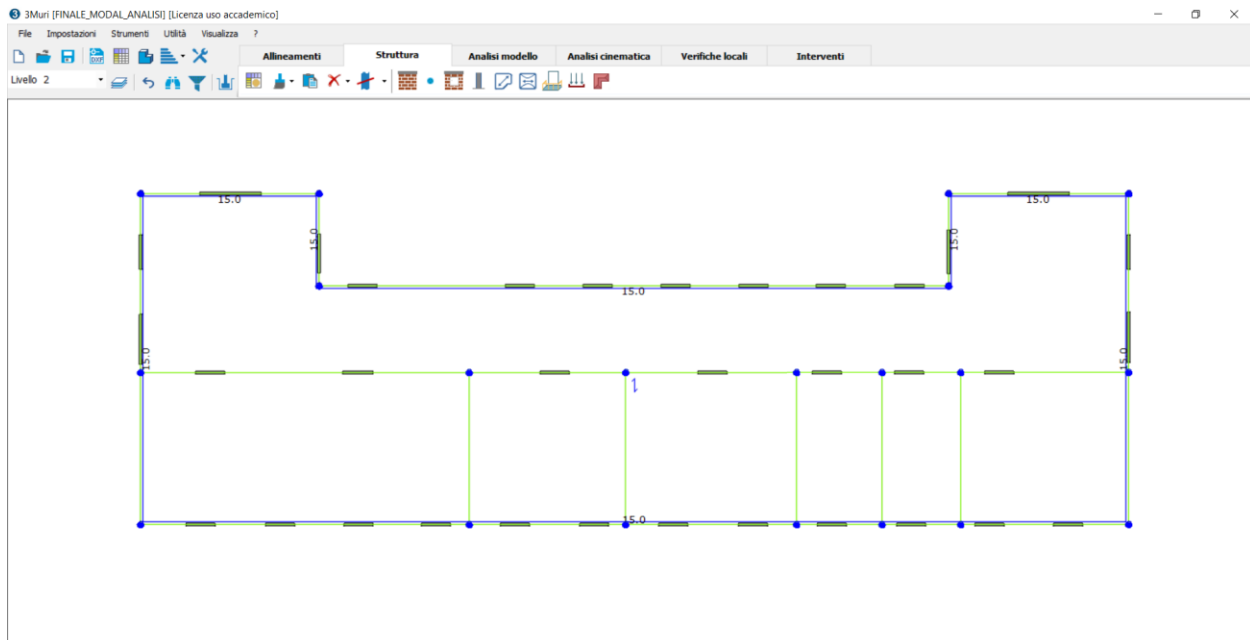


Figure 117: Structural Model of Town Hall of pizzoli inside 3Muri

Duplication of Floors

It's possible to duplicate the floor for upper levels very easily as more or less our structure was having similar mechanical and material properties. Moreover, there is also a possibility to change the height of upper floor if duplicated. Depending on the floor type selected it's

possible to. Roofs can be modelled as structural or non-structural elements, Structural when the roof has significant stiffness for example when we have reinforced concrete slab, non-structural when slab has negligible stiffness for example if we have wooden roof. Also, if we have masonry roof whose stiffness is significant and it would not be suitable to model as rigid. If we model the roof as nonstructural element software will treat the structure same as without roof. After modelling the roof Inside the different levels of each floor it's possible to select whether it's covered by a roof or not and whether the roof is structural or not.

Figure 8:3D Model inside 3Muri

Gestione livelli

Livello	Visibile	Descrizione	Altezza [cm]	Quota [cm]	Q vento [daN/m2]	Tetto
1	<input checked="" type="checkbox"/>	Livello 1	400	400	0	<input type="checkbox"/>
2	<input checked="" type="checkbox"/>	Livello 2	450	850	0	<input type="checkbox"/>

Altezza nuovo livello: 450 [cm] Vista 3D ☒ Mostra nel 3D i livelli spenti in grigio

Tipo tetto:
☒ Non strutturale
☐ Strutturale

Nuovo (sopra)
Elimina
Duplica (sopra)
Attiva livello
Elimina tetto
OK ?

Figure 118

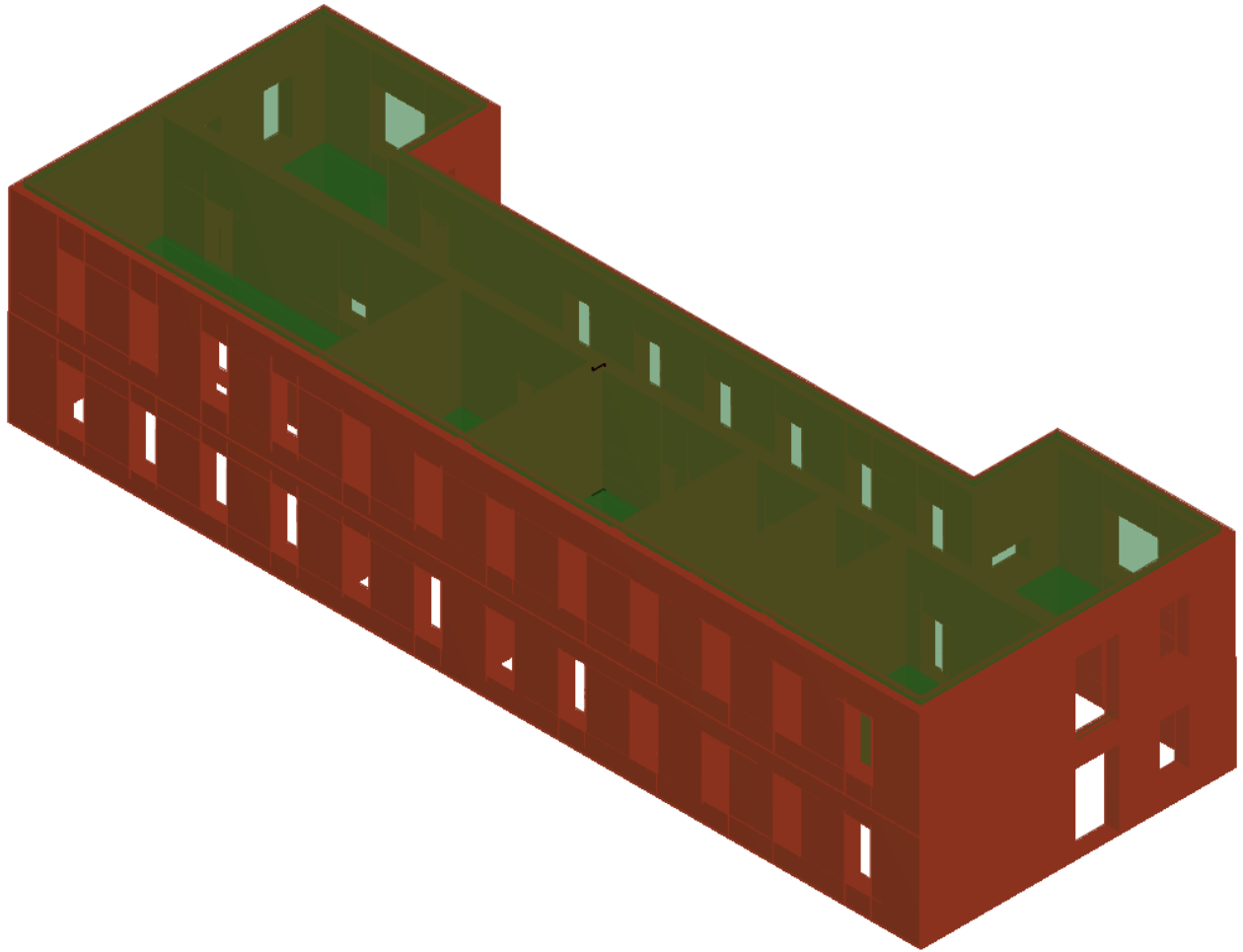


Figure 119: 3D Model inside 3Muri

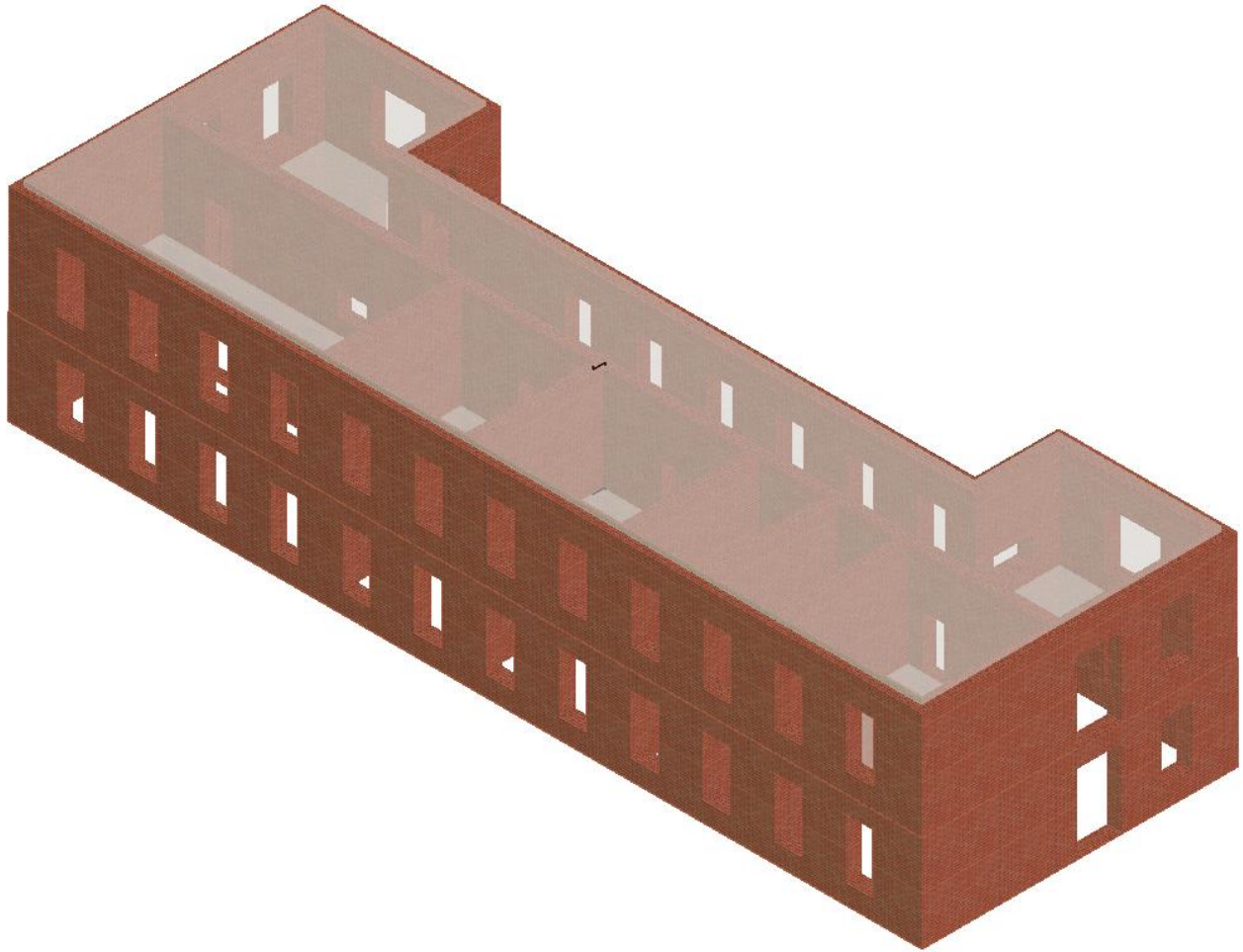


Figure 120: 3D model inside 3Muri

Analysis inside 3Muri

the software 3Muri automatically generates the mesh dividing the masonry wall into 3 elements:

- Piers in Orange color
- Spandrels beams in Green Color

S287972
Thesis Report

- Rigid Nodes in Blue Color
- Yellow points on the bottom fixed in foundation

This division can be seen in the picture below:

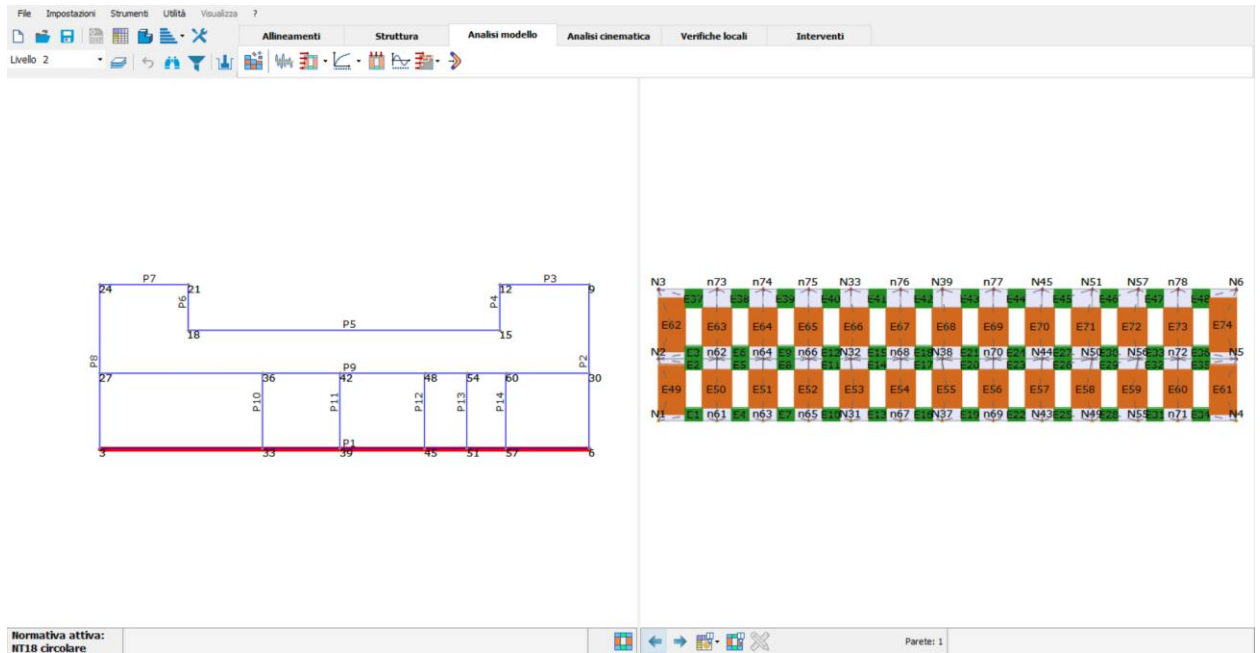


Figure 121: Wall-1 automatic Mesh generated by 3Muri

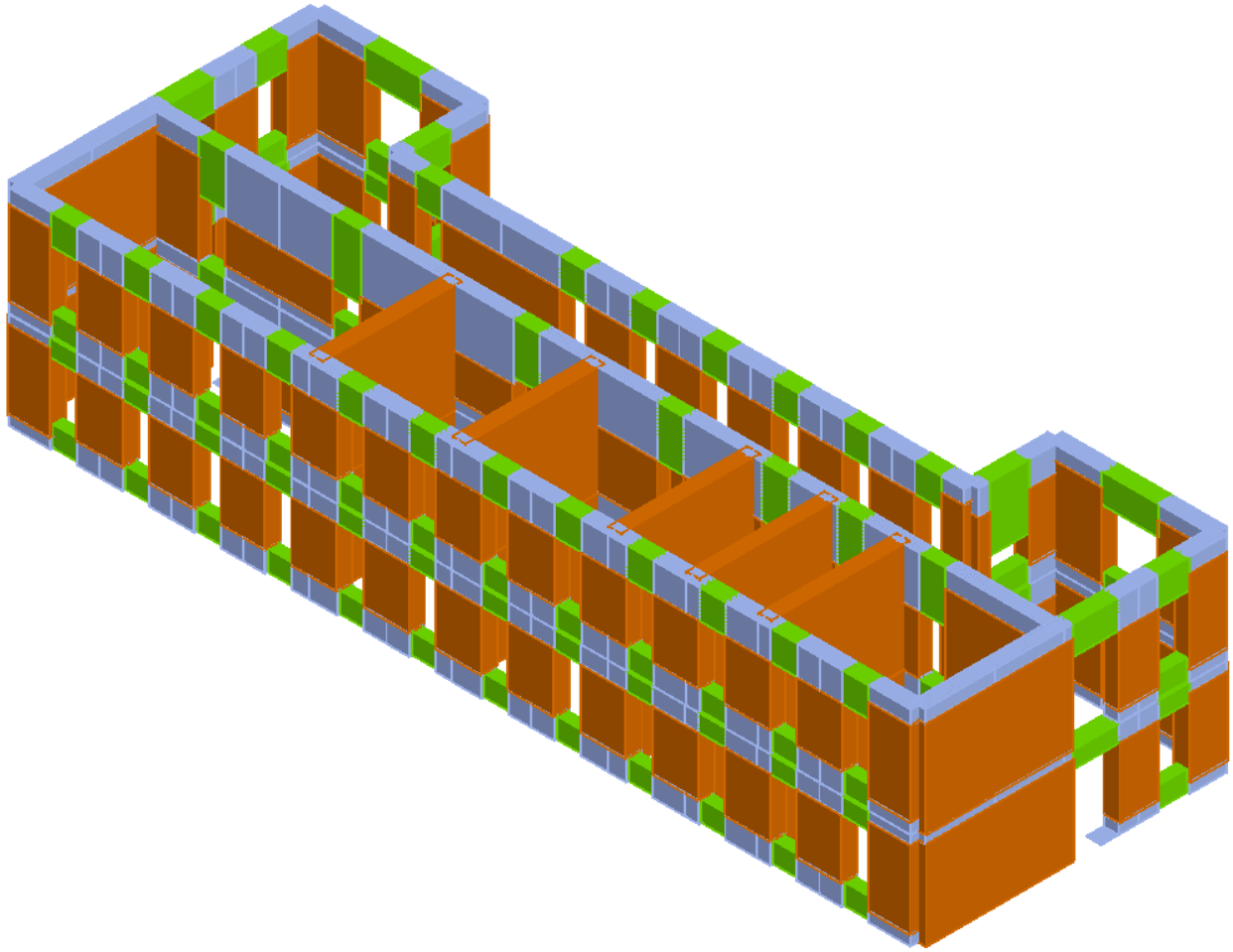


Figure 122: Automatic Mesh Generation by 3Muri

Linear Static Analysis

1st linear analysis was performed which software performs automatically based on the criteria of code selected in our case as Eurocode. It was observed that there was no failure in the wall depending on under static loads as can be seen in the *figure 123* below. Infact the walls were working around their 60% capacity.

Muratura						Verifiche a carico verticale
Parete	Maschi rotti	Nd/Nr Max	h0/t Max	e1/t Max	e2/t Max	
2	3	0.60	7.50	0.413	0.169	
8	2	0.51	7.50	0.427	0.164	
3	1	0.24	7.50	0.339	0.123	
4	0	0.26	7.50	0.240	0.098	
5	0	0.20	7.50	0.244	0.090	
6	0	0.14	7.50	0.038	0.038	
1	0	0.28	7.50	0.330	0.140	
7	0	0.21	7.50	0.318	0.111	
9	0	0.27	9.00	0.107	0.049	
10	0	0.17	9.00	0.163	0.058	
11	0	0.15	9.00	0.060	0.045	
12	0	0.16	9.00	0.151	0.052	
13	0	0.15	9.00	0.058	0.045	
14	0	0.17	9.00	0.156	0.055	

Figure 123: Linear Static Analysis of all Masonry Walls

The figure 124 below compares the axial demand of walls to the axial load carrying capacity of wall under static loads for wall/pier-1. In similar it was possible to compare the results for each wall.

S287972 Thesis Report

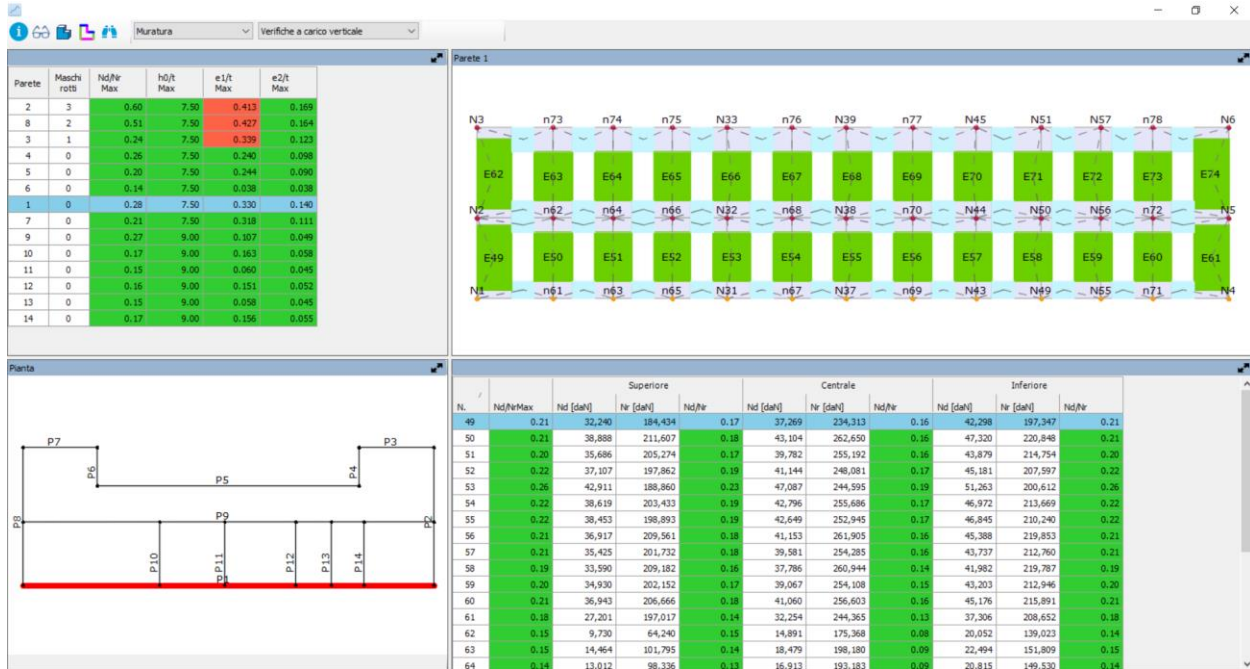


Figure 124: Linear Static Analysis of wall-1

The orange nodes at the bottom shows that the base of building immersed inside the foundation and is fixed and have no displacement as can be seen in the figure 125 below. Actually, inside 3Muri it's not possible to model foundation.

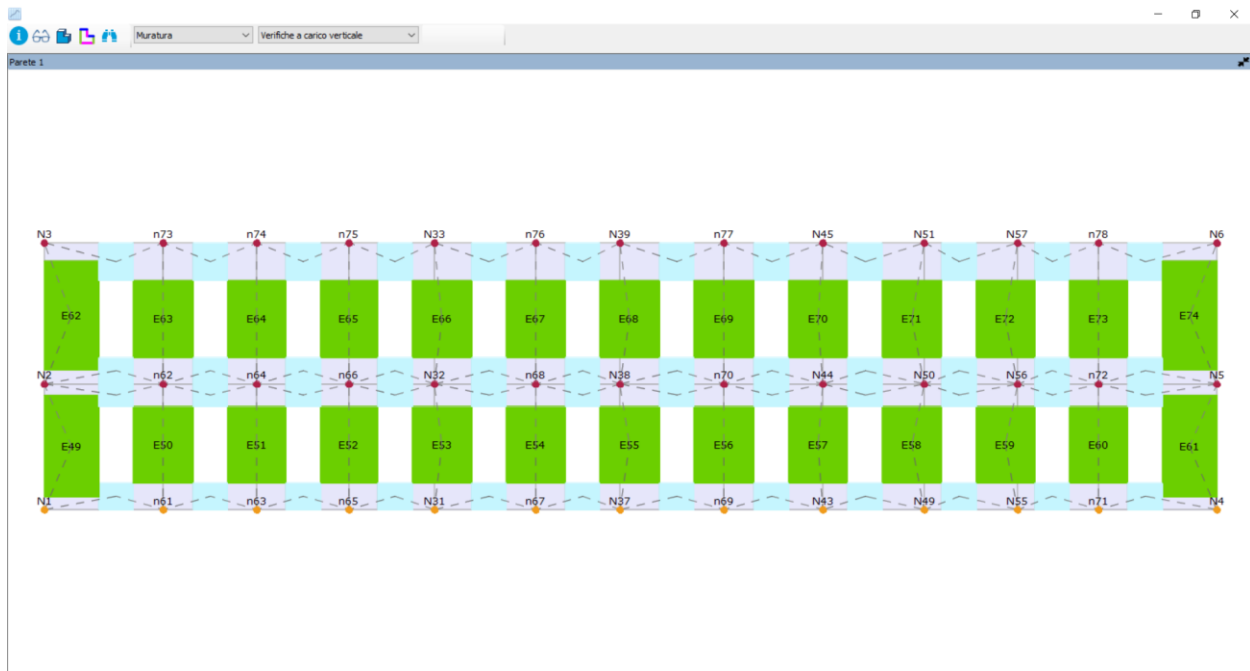
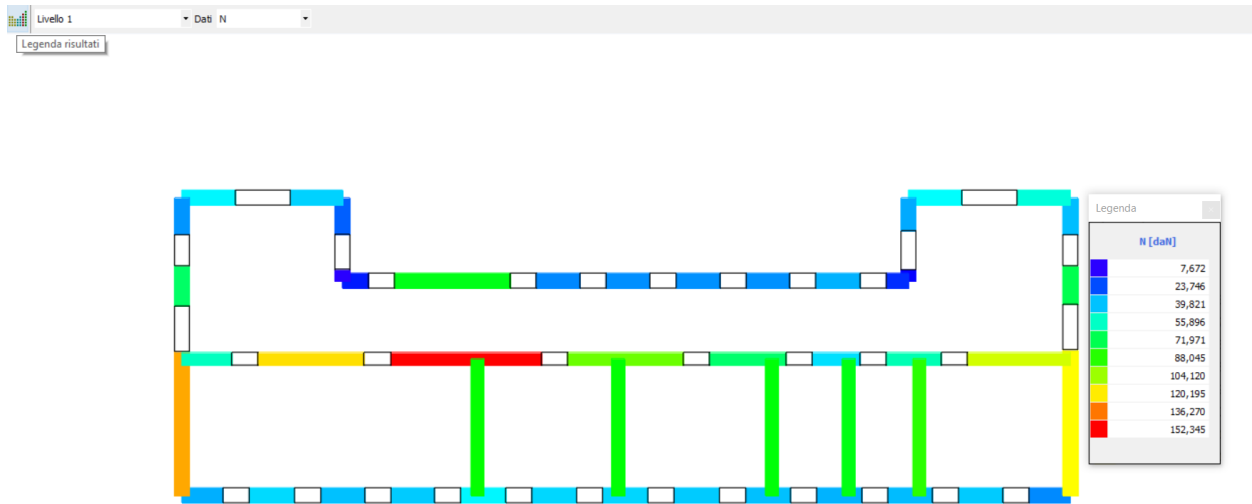


Figure 125: Linear Static Analysis Results All Satisfied of wall-1

The figure 126 below shows the maximum axial force developed inside the wall. The maximum axial force of 137kN was developed in the load bearing central wall as can be seen in the *figure* below:



Modal Analysis

3Muri can also perform modal analysis which gives the picture of vibration of structure at various frequencies of vibration. Just first 4 modes of vibration were calculated that contributed around 93% mass in X-direction and 99% in y-direction as can be seen in *figure 127* below:

S287972

Thesis Report

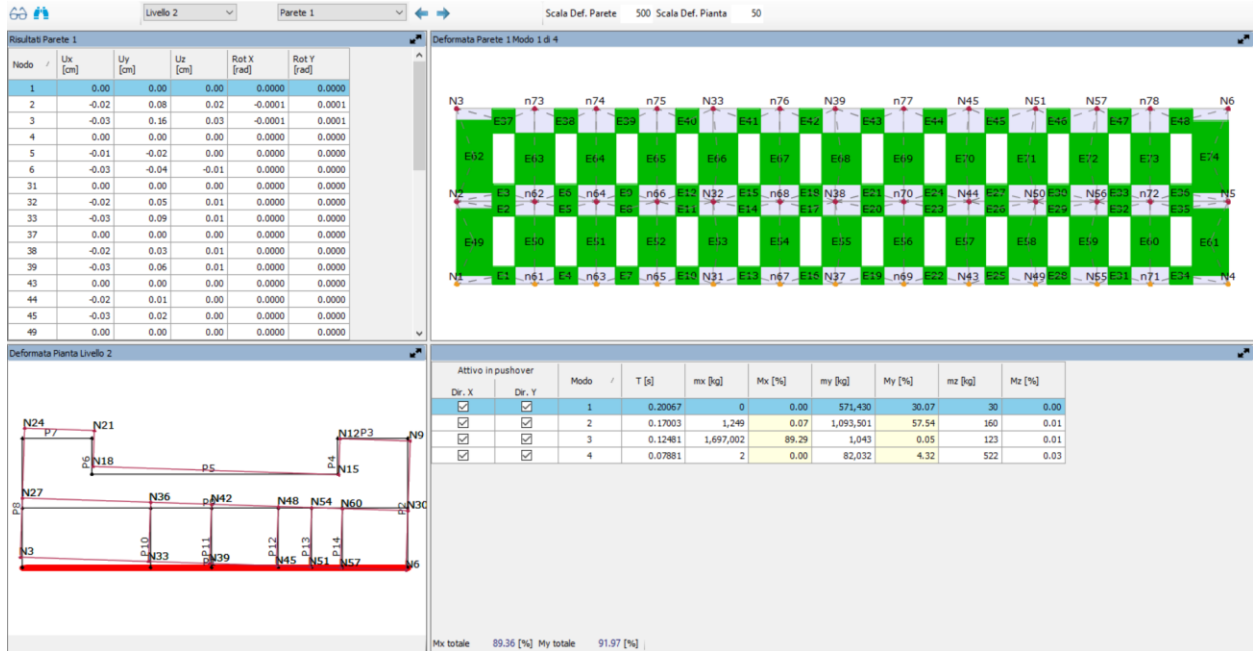


Figure 127: Mode-1 Modal analysis Result

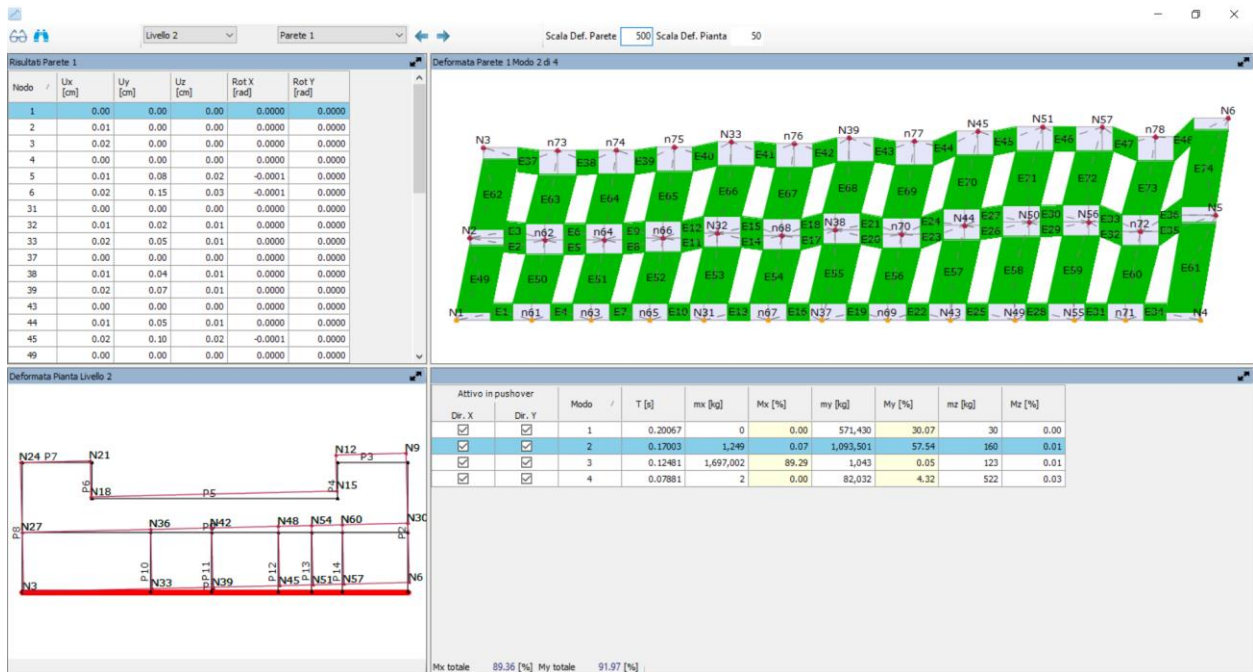


Figure 128: Mode-2 of vibration

S287972 Thesis Report

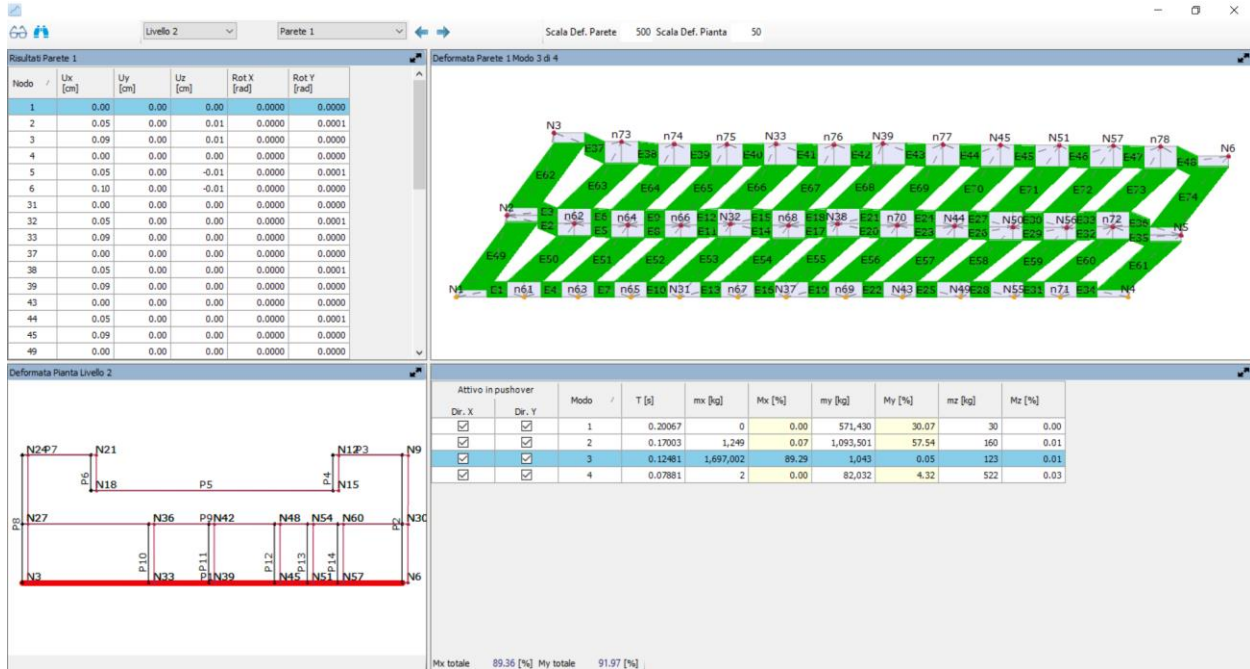


Figure 129: Mode-3 of Vibration

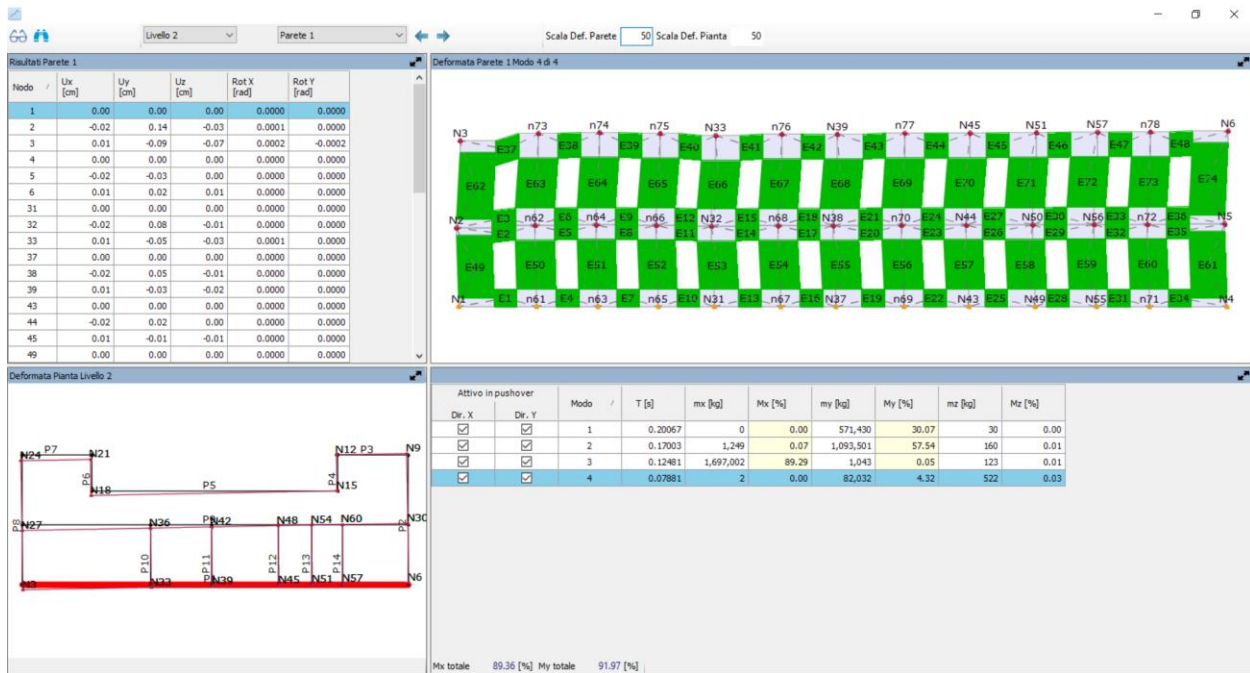


Figure 130: Mode shape-4 of vibration

the table gives the results of modal analysis as follows:

Mode: Mode number of vibration associated to a frequency of vibration

T[s]: Time period of Vibration

mx[kg]:Modal participation mass in kG X-direction

Mx[%]:Modal participation mass %age in X-direction

my[kg]: Modal participation mass in kG Y-direction

My[%]:Modal participation mass %age in Y-direction

mz[kg]:Modal mass Participating in direction Z

Mz[%]:Percentage of participating mass in direction Z

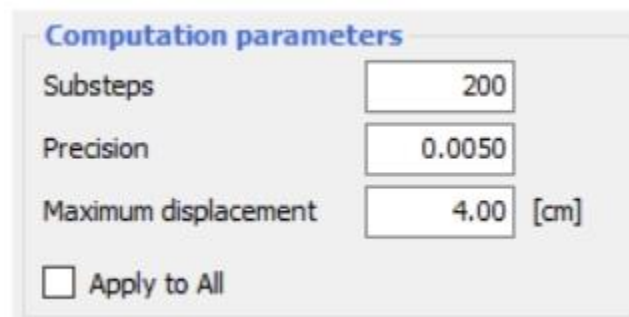
Attivo in pushover		Modo	T [s]	mx [kg]	Mx [%]	my [kg]	My [%]	mz [kg]	Mz [%]
Dir. X	Dir. Y								
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1	0.20067	0	0.00	571,430	30.07	30	0.00
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	2	0.17003	1,249	0.07	1,093,501	57.54	160	0.01
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	3	0.12481	1,697,002	89.29	1,043	0.05	123	0.01
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	4	0.07881	2	0.00	82,032	4.32	522	0.03
<div>Mx totale 89.36 [%] My totale 91.97 [%]</div>									

Figure 131: Modal Analysis Vibration Modes

The results showed that mode-2 was giving highest %age of mass in y-direction about 57% while mode-3 was having highest %age of modal mass of about 89% which satisfies the minimum requirement of 85% of modal mass to be considered.

Pushover Analysis

3Muri performs 24 pushover analysis in X and Y and with combination of two types of distribution of loads (Uniform and Modal) and with different eccentricities of ± 5 . These distributions of forces are proportional to mass and to 1st vibration mode. Inside 3Muri it's possible to perform pushover analysis on a single wall or globally. Using the NTC-18 it's possible to set the seismic demand that will be used in the defining the seismic demand of site where our building is located. In our case as town hall of pizzolli was located in L'Aquila the software automatically calculates the response spectrum using the parameters as can be seen in *figure 132* below. While setting the seismic demand parameters it's possible to select the parametric response spectrum or custom response spectrum. It's possible to select types of soil from A to E and corner period T_c , soil amplification factor S and importance factor as mentioned in Eurocode. These parameters define the response spectrum used for defining the seismic demand in terms of d_{max} . Which later will be used to calculate the vulnerability index as ratio of PGA capacity coming from pushover analysis to seismic demand PGA demand calculated from the response spectrum. The choice of control node and Earthquake directions has to be given as input by the user.



Computation parameters	
Substeps	200
Precision	0.0050
Maximum displacement	4.00 [cm]
<input type="checkbox"/> Apply to All	

Seismic load ×

Spectrum Shape Parametric ▼

Spectrum Diagram

	NC	SD	DL
Verification	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
► a_{gR} [m/s ²]	1	0.8	0.6
Soil type	A ▼	A	A
S	1.00	1.00	1.00
T_B [s]	0.15	0.15	0.15
T_C [s]	0.40	0.40	0.40
T_D [s]	2.00	2.00	2.00

Importance Factor 1.00

Load default OK Cancel ?

Figure 132: Seismic Demand for Pushover analysis

The output of pushover analysis is measured in terms of vulnerability index which should be greater than 1 in order to pushover analysis to be verified. If vulnerability index is greater than 1 it means our pushover analysis has satisfied however if our vulnerability index is less 1 it means our pushover analysis has not passed. than After performing Global analysis it's also possible to perform local analysis on walls or even on more than 1 walls. The software automatically calculates 24 pushover analysis considering different possibilities of load profiles of eccentricity and calculates the **vulnerability index for 3 limit states** for a specific reference return period as follows:

- Life Limit state (SLV)
- Damage Limit state (SLD)
- Operational Limit state (SLO)

Dmax is displacement demand required by the structure depending on seismic demand and response spectrum where Du is ultimate displacement offered by the structure and obtained after performing pushover analysis on our structure. If **Du > Dmax** the

verification of pushover analysis is verified and on contrary the verification of pushover analysis is not verified.

Moreover, the software highlights the pushover analysis which are more critical in yellow. And it possible to sees the pushover analysis capacity curve and bilinear curve of that critical push over analysis.

Control Node Selection

The choice of Control node for pushover analysis is very important and choice of control node can significantly change the ultimate displacement. Control node is the node whose displacement is measured against the maximum shear at the base and is usually taken as center of mass of top floor. the control node should be chosen which is most deformable. The more deformable node we take as our control node the more accurate our results will be. By noticing the ultimate displacement D_u of our structure it's possible to see our pushover analysis was performed correctly or not. If we get very high ultimate displacement of our control node, we obviously need to check our model. The ultimate displacement will even vary if we choose different position of control node within same floor. Moreover, the deformability of floor also depends on the type of slab we have. The software automatically gives the option to select some control nodes while setting parameters for pushover analysis as can be seen in *figure 133* below. Moreover, using average node displacement, the whole building is displacement according to displacement of control node. 3Muri gives the options to choose and suggest some nodes having the highest elevation from the ground level. 24 Pushover analysis was run choosing the control node near the center of mass of top floor and giving displacement of 7cm as can be seen in *figure 133* below:

Analisi

Nodo di controllo

Livello: [2] Livello 2 Nodo: 42

Spostamento: Spostamenti medi del livello selezionato

N.	Calcola analisi	Dir. sisma	Carico sismico	Eccentricità [cm]
1	<input checked="" type="checkbox"/>	+X	Uniforme	0.0
2	<input checked="" type="checkbox"/>	+X	Distribuzione modale	0.0
3	<input checked="" type="checkbox"/>	-X	Uniforme	0.0
4	<input checked="" type="checkbox"/>	-X	Distribuzione modale	0.0
5	<input checked="" type="checkbox"/>	+Y	Uniforme	0.0
6	<input checked="" type="checkbox"/>	+Y	Distribuzione modale	0.0
7	<input checked="" type="checkbox"/>	-Y	Uniforme	0.0
8	<input checked="" type="checkbox"/>	-Y	Distribuzione modale	0.0
9	<input checked="" type="checkbox"/>	+X	Uniforme	62.5
10	<input checked="" type="checkbox"/>	+X	Uniforme	-62.5
11	<input checked="" type="checkbox"/>	+X	Distribuzione modale	62.5
12	<input checked="" type="checkbox"/>	+X	Distribuzione modale	-62.5
13	<input checked="" type="checkbox"/>	-X	Uniforme	62.5
14	<input checked="" type="checkbox"/>	-X	Uniforme	-62.5
15	<input checked="" type="checkbox"/>	-X	Distribuzione modale	62.5
16	<input checked="" type="checkbox"/>	-X	Distribuzione modale	-62.5
17	<input checked="" type="checkbox"/>	+Y	Uniforme	187.0
18	<input checked="" type="checkbox"/>	+Y	Uniforme	-187.0
19	<input checked="" type="checkbox"/>	+Y	Distribuzione modale	187.0
20	<input checked="" type="checkbox"/>	+Y	Distribuzione modale	-187.0
21	<input checked="" type="checkbox"/>	-Y	Uniforme	187.0
22	<input checked="" type="checkbox"/>	-Y	Uniforme	-187.0
23	<input checked="" type="checkbox"/>	-Y	Distribuzione modale	187.0
24	<input checked="" type="checkbox"/>	-Y	Distribuzione modale	-187.0

☐ Definizione per angolo sisma

Angolo: 0 [°]

Aggiungi -- Nessuno --

Incrementa

Elimina

Dati generali

Piano Campagna: 0.0000 [cm]

Step critico: 100

Precisione p.p.: 0.0050

Parametri di calcolo

☐ Applica a tutte

Sottopassi: 50

Tolleranza: 0.0050

Spostamento max: 6.00 [cm]

Eccentricità: [] [cm]

Abilita analisi

Dir. sisma: [v]

Carico sismico: [v]

Eccentricità: [v]

Seleziona Tutto Deseleziona Tutto

Carico sismico

☐ Proporzionale forze statiche

☒ Distribuzione modale

☒ Esegui controllo deformabilità angolare

OK ?

Figure 133: Pushover analysis Input Selection of control node and Maximum Imposed displacement

S287972
Thesis Report

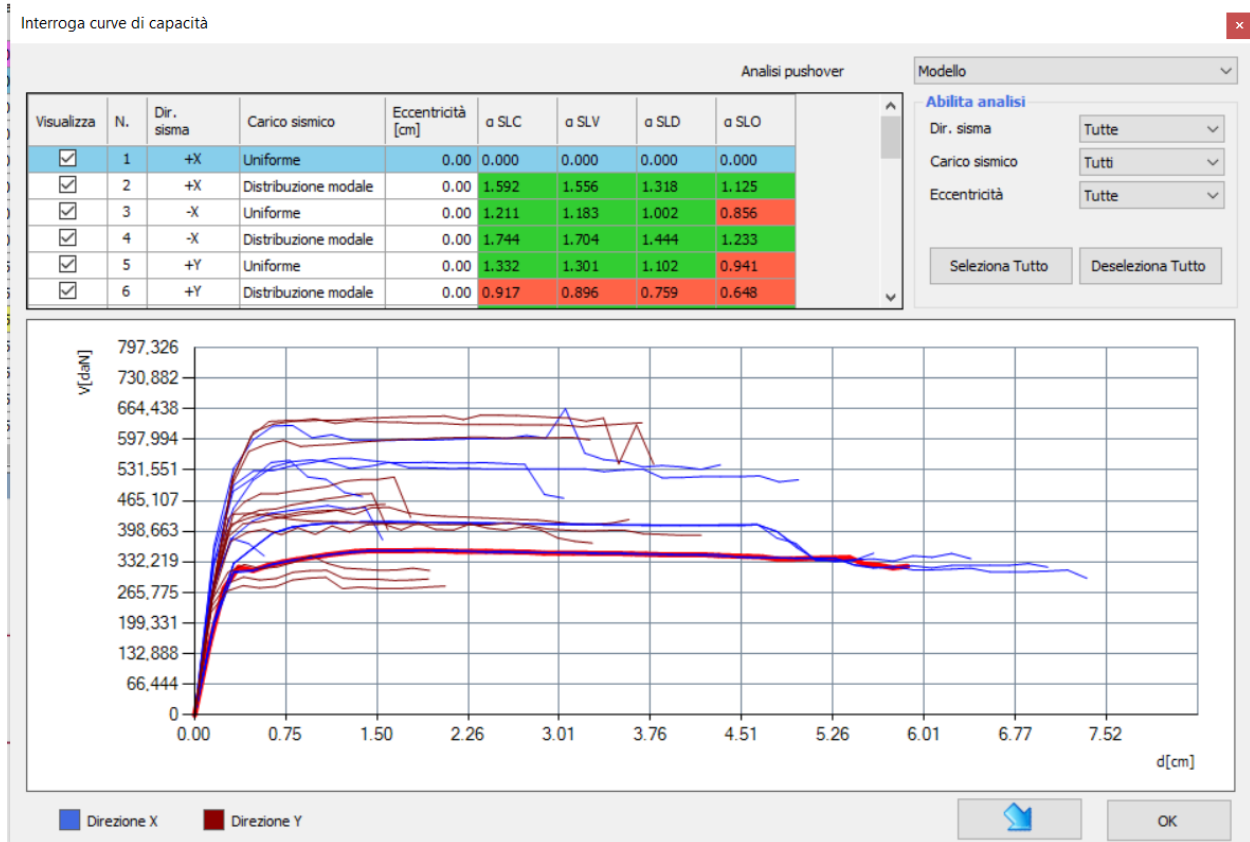


Figure 134: Pushover Analysis Capacity Curves

Analisi								
	N.	Dir. sisma	Carico sismico	Eccentricità [cm]	α SLC	α SLV	α SLD	α SLO
	1	+X	Uniforme	0.00	0.000	0.000	0.000	0.000
	2	+X	Distribuzione modale	0.00	1.592	1.556	1.318	1.125
	3	-X	Uniforme	0.00	1.211	1.183	1.002	0.856
	4	-X	Distribuzione modale	0.00	1.744	1.704	1.444	1.233
	5	+Y	Uniforme	0.00	1.332	1.301	1.102	0.941
	6	+Y	Distribuzione modale	0.00	0.917	0.896	0.759	0.648
	7	-Y	Uniforme	0.00	1.961	1.917	1.624	1.386
	8	-Y	Distribuzione modale	0.00	1.173	1.224	1.169	0.998
	9	+X	Uniforme	62.54	1.012	0.989	0.838	0.715
	10	+X	Uniforme	-62.54	1.000	0.977	0.828	0.707
!	11	+X	Distribuzione modale	62.54	0.502	0.541	0.883	0.754
	12	+X	Distribuzione modale	-62.54	1.085	1.143	1.266	1.081
	13	-X	Uniforme	62.54	1.191	1.164	0.986	0.842
	14	-X	Uniforme	-62.54	1.209	1.182	1.001	0.855
	15	-X	Distribuzione modale	62.54	1.076	1.121	1.068	0.912
	16	-X	Distribuzione modale	-62.54	1.571	1.535	1.300	1.110
	17	+Y	Uniforme	187.04	1.271	1.242	1.052	0.898
	18	+Y	Uniforme	-187.04	1.238	1.210	1.025	0.875
	19	+Y	Distribuzione modale	187.04	0.988	0.966	0.818	0.699
	20	+Y	Distribuzione modale	-187.04	0.863	0.843	0.714	0.610
	21	-Y	Uniforme	187.04	1.949	1.905	1.614	1.378
	22	-Y	Uniforme	-187.04	1.836	1.794	1.520	1.297
	23	-Y	Distribuzione modale	187.04	1.315	1.370	1.255	1.072
	24	-Y	Distribuzione modale	-187.04	1.145	1.193	1.114	0.952

Figure 135: 24 Pushover analysis output

Results and Conclusion

By making a comparison between Modal analysis performed in both software's Salome Meca and 3Muri the time period of vibration of first 4 modes were compared between two software's and it was observed that **there was negligible difference in time period of vibration (about 15% for first 4 modes) obtained from the 3Muri and Salome Meca. Moreover, the difference in time period of vibration increases for higher modes** as can be seen in **table-4 below**. As 3muri is a software specifically designed for structure analysis of masonry structure so 3Muri is able to consider the bond strength among the masonry block elements and support length of roof on supporting walls that cannot be taken into account in typical finite element software like Salome Meca. Moreover, the generation of mesh and modelling is must easier and faster inside 3Muri compared to Salome Meca as 3Muri has materials library and tools specially designed for modelling of masonry structures.

Mode Shapes	Salome Meca		3Muri		%age difference in Time period [T]
	f [Hz]	T [s]	f [Hz]	T [s]	%age ΔT [s]
1	5.4	0.19	5.00	0.2	7.41
2	5.9	0.17	5.88	0.17	0.30
3	6.5	0.15	8.33	0.12	-28.21
4	10.2	0.10	14.29	0.07	-40.06
	%age Average Difference in T				-15.14

TABLE-4

The *figure 136* below shows a comparison between Salome meca (FEM software) and 3Muri (FMA software) in terms of frequency and time period of vibrations of first 4 modes:

Figure 9: Comparison of Modal Analysis Results between Salome Meca and Code Aster

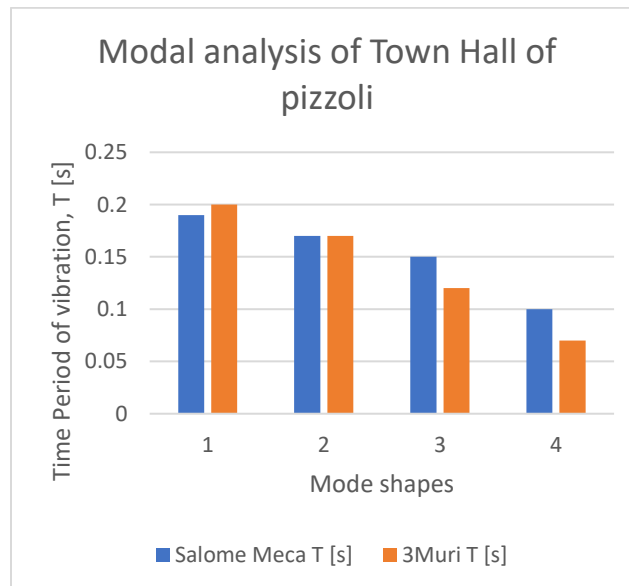


Figure 136: Comparison of Time period of Vibration of Salome Meca vs 3Muri

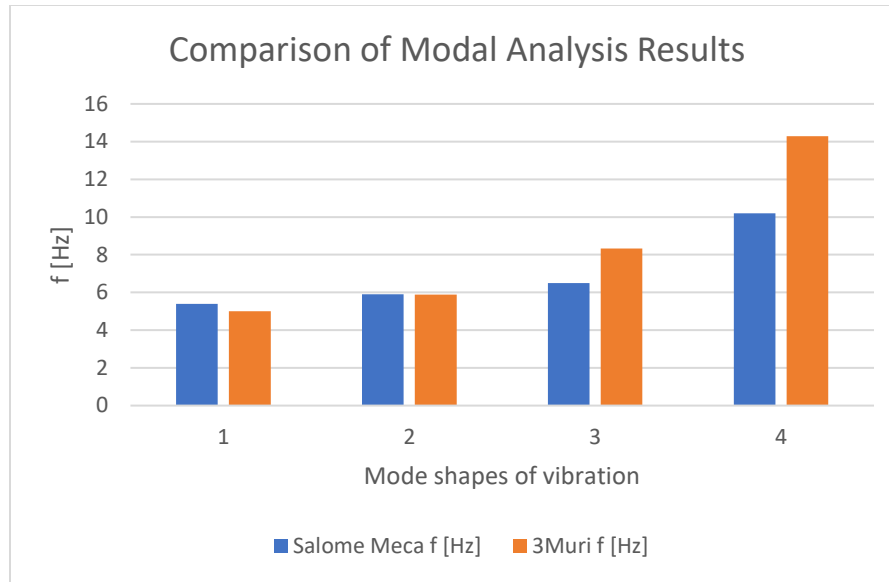


Figure 137: Comparison of f of vibration [s]

Bibliography

- [R5.05.04] *Modelling of linear dynamic damping*. (n.d.). Retrieved from code_aster: https://code-aster.org/V2/doc/default/en/man_r/r5/r5.05.04.pdf
- [U2.06.03] *Modeling the mechanical damping*. (n.d.). Retrieved from code_aster: https://code-aster.org/V2/doc/default/en/man_u/u2/u2.06.03.pdf
- [U4.21.01] *Operator LIRE_MAILLAGE*. (n.d.). Retrieved from code_aster: https://code-aster.org/V2/doc/default/en/man_u/u4/u4.21.01.pdf
- [U4.41.01] *Operator AFFE_MODELE*. (n.d.). Retrieved from code_aster: https://code-aster.org/V2/doc/default/en/man_u/u4/u4.41.01.pdf
- [U4.41.01] *Operator AFFE_MODELE*. (n.d.). Retrieved from code_aster: https://code-aster.org/V2/doc/default/en/man_u/u4/u4.41.01.pdf
- [U4.42.01] *Operator AFFE_CARA_ELEM*. (n.d.). Retrieved from code_aster: https://code-aster.org/V2/doc/default/en/man_u/u4/u4.42.01.pdf
- [U4.43.03] *Operator AFFE_MATERIAU*. (n.d.). Retrieved from code_aster: https://code-aster.org/V2/doc/default/en/man_u/u4/u4.43.03.pdf
- [U4.44.01] *Operators AFFE_CHAR_MECA et AFFE_CHAR_MECA_F*. (n.d.). Retrieved from code_aster: https://code-aster.org/V2/doc/default/en/man_u/u4/u4.44.01.pdf
- [U4.52.02] *Operator CALC_MODES*. (n.d.). Retrieved from code_aster: https://code-aster.org/V2/doc/default/en/man_u/u4/u4.52.02.pdf
- 3Muri - *Masonry buildings analysis*. (2022, JUNE 30). Retrieved from STADATA: <https://www.stadata.com/en/3muri-masonry-buildings-analysis/>
- 3Muri - *Presentazione breve*. (2017, JAN 31). Retrieved from STADATA: <https://www.stadata.com/en/3muri-project-video/?playlist=54a8004&video=4df682e>
- 3MURI. (2018, JULY 20). Retrieved from STA DATA: <https://www.stadata.com/la-relazione-di-calcolo-strutturale-secondo-ntc18-parte-1/>
- 3MURI. (2018, JULY 20). Retrieved from STA DATA: https://www.youtube.com/watch?v=Rbyb3W_f8Sk&list=PLb6RdtWtnvp2K6_pt7EKd-u0O2D92Pohi&index=19&ab_channel=S.T.A.Data
- 3Muri project. (n.d.). Retrieved from STA DATA: https://www.youtube.com/watch?v=cZ6Snj2_sog&ab_channel=AdriaBIM
- 3MURI PROJECT. (20, July 2018). Retrieved from STA DATA: https://www.youtube.com/watch?v=8IR6rsO7Kaw&list=PLb6RdtWtnvp2K6_pt7EKd-u0O2D92Pohi&index=20&ab_channel=S.T.A.Data
- 3Muri Project X3 - Version 13.9.0.0 (29/09/2021). (2021, 09 29). Retrieved from STA DATA STRUCTURAL SOFTWARE: <https://www.servizi.stadata.com/download/?lang=en&sw=3muri%20project>

SARERI, G. I. (2017-2018). *MODELLI NUMERICI PER L'ANALISI SISMICA*. Padova.

STA DATA. (2022, july 13). Retrieved from 3Muri:

https://www.youtube.com/watch?v=ZB3wmly9fJY&ab_channel=S.T.A.Data

Trapani, F. d. (politecnico di Torino). *NONLINEAR STATIC ANALYSIS*. Torino.