SeismoBuild 2025

User Manual

Copyright

Seismosoft Ltd.
Piazza Castello, 19
27100 Pavia (PV) - Italy
e-mail: info@seismosoft.com
website: www.seismosoft.com

How to Cite the use of the Software

In order to acknowledge/reference in any type of publication (scientific papers, technical reports, text books, theses, etc) the use of this software, you should employ an expression of the type: Seismosoft [2025] "SeismoBuild 2025 – A computer program for seismic assessment and retrofitting of RC structures.," available from https://seismosoft.com/.

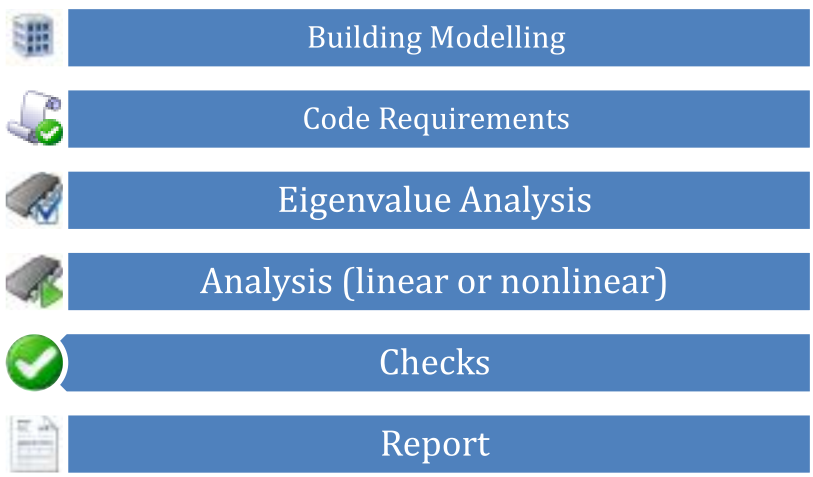
Table of Contents

Introduction

SeismoBuild is a Finite Element package for structural assessment, capable of carrying out all the Code defined checks, taking into account both geometric nonlinearities and material inelasticity.

The software consists of six main modules: the **Building Modelling** module, in which it is possible to define the input data of the structural model, the **Code Requirements** module, whereby the Code-based parameters and options are defined, the **Eigenvalue Analysis** and **Pushover Analysis** modules, where the selected analyses are carried out and their results are obtained, the **Checks** module, in which all the checks for the structural members, according to the selected Code, are carried out and finally the **Report** module to output the results of the building assessment; all is handled through a **completely visual interface**.

With the Building Modeller facility the user can create regular or irregular 3D structural models on the fly. The whole process takes no more than a few minutes. No input or configuration files, programming scripts or any other time-consuming and complex text editing procedures are required. Moreover, the Processor in the Eigenvalue and Pushover analysis modules, features real-time plotting of displacement curves and deformed shape of the structure, together with the possibility of pausing and re-starting the analysis, whilst the Post-Processor, where the results of the analyses are exported, offers advanced post-processing facilities, including the ability to custom-format all derived plots and deformed shapes, thus increasing the productivity of users; it is also possible to create AVI movie files to better illustrate the sequence of structural deformation.



Structure of the software

The software is fully integrated with the Windows environment. All information visible within the graphical interface of SeismoBuild can be copied to external software applications (e.g. to word processing programs, such as Microsoft Word), including output data, high quality graphs, the models' deformed and undeformed shapes and much more.

General

System Requirements

To use **SeismoBuild**, we suggest:

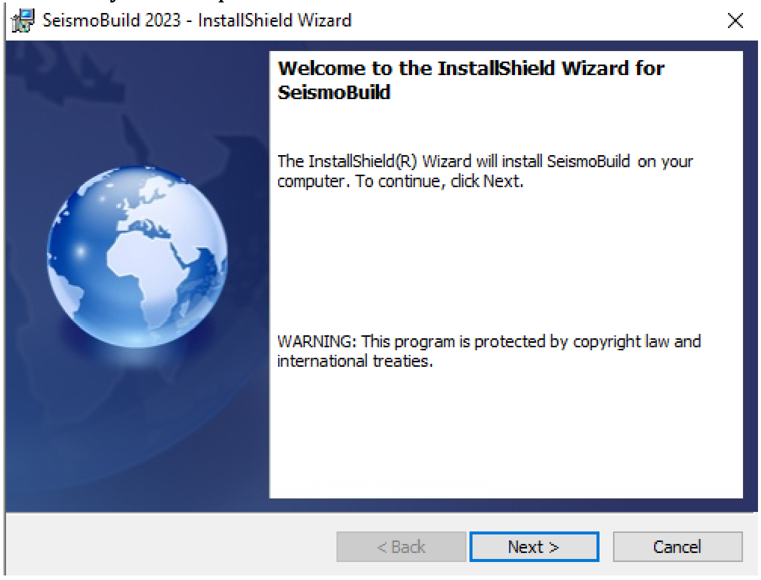
- A PC (or a "virtual machine") with one of the following operating systems: Windows 7 and later versions of Windows (64-bit);
- 8 GB RAM;
- Screen resolution set to 1366x768 pixels or higher;
- An Internet connection (better if a broadband connection) for the registration of the software.

Installing/Uninstalling the software

Installing the software

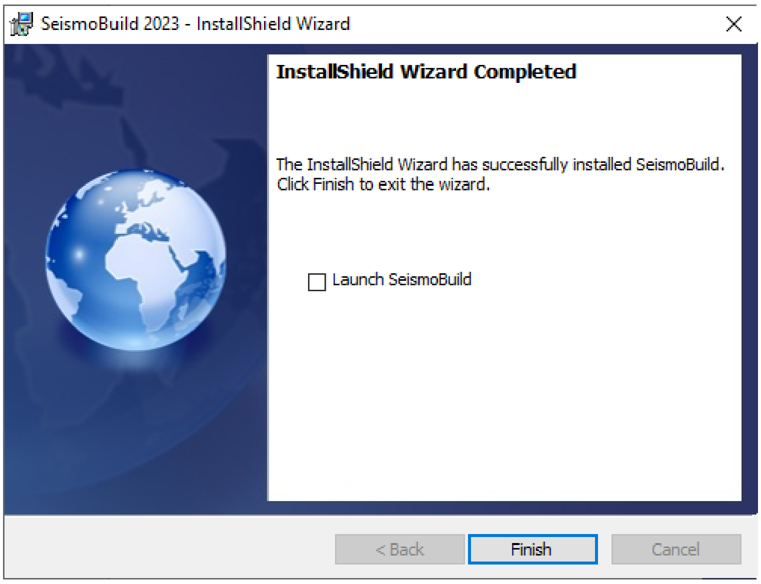
Follow the steps below in order to install **SeismoBuild**:

1. Download the version of the program from: www.seismosoft.com/downloads
2. Save the application on your computer and launch it.



Installation wizard (first window)

3. Click the *Next* button to proceed with the installation. The License Agreement appears on the screen. Please, read it carefully and accept the terms by checking the box.
4. Click the *Next* button. On the next request to select the destination folder, click the *Next* button again to install to the 'default' folder or click the Change button to install to a different one.
5. Click the *Install* button and wait until the software is installed.
6. At the end of the procedure, click *Finish* to exit the wizard.



Installation wizard (last window)

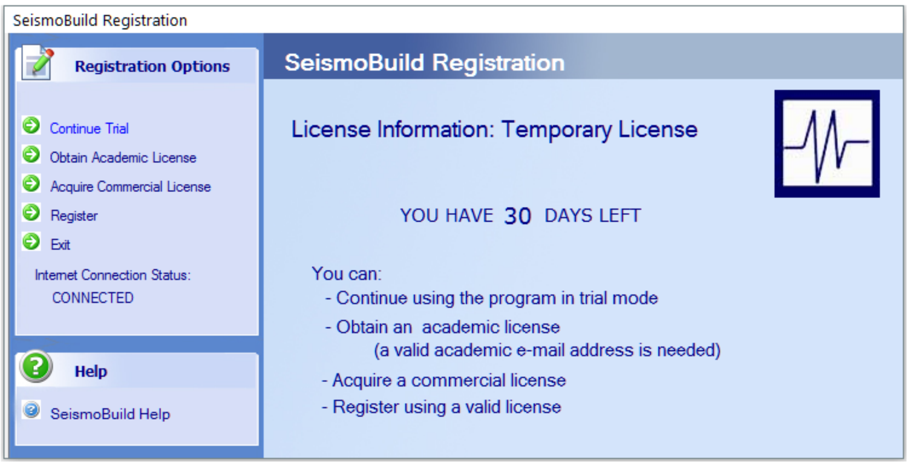
Uninstalling the software

To remove the software from the computer:

1. Open the Start menu.
2. Click Settings.
3. Click System on the Settings menu.
4. Select Apps & features from the left pane.
5. Select the program from the list of all the installed apps.
6. Click the Uninstall button that appears.

OPENING THE SOFTWARE AND REGISTRATION OPTIONS

To launch SeismoBuild, select Start > Programs or All Programs > Seismosoft > SeismoBuild 2025> SeismoBuild 2025. The following registration's window will appear:

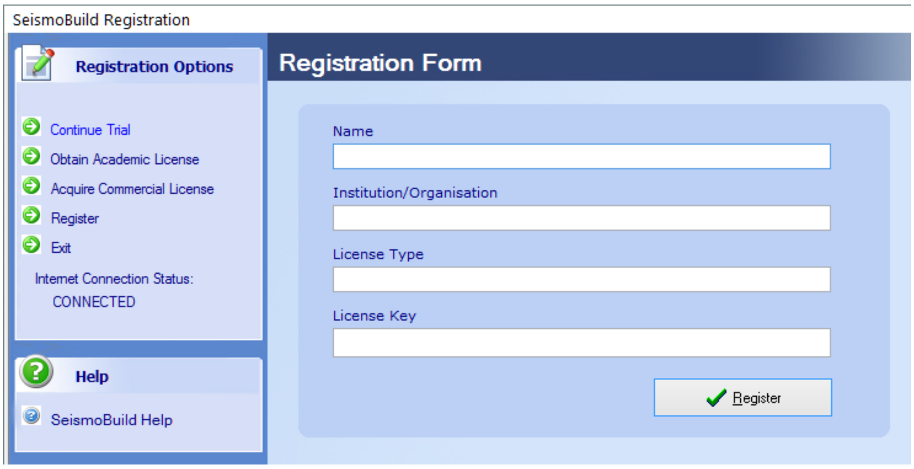


SeismoBuild Registration Window

Before using the software, you must choose one of the following options:

1. Continue using the program in trial mode.
2. Obtain an academic license by providing a valid academic e-mail address.
3. Acquire a commercial license.

NOTE: If you choose option 2 or 3, then you have to register using the provided license.



Registration Form

IMPORTANT: Regarding the license keys please note that, as indicated in the message that appears before the opening of the main window of the program, the licenses of version 2024 and older are not valid in SeismoBuild 2025. Users are thus invited to request a new license.

Quick Start

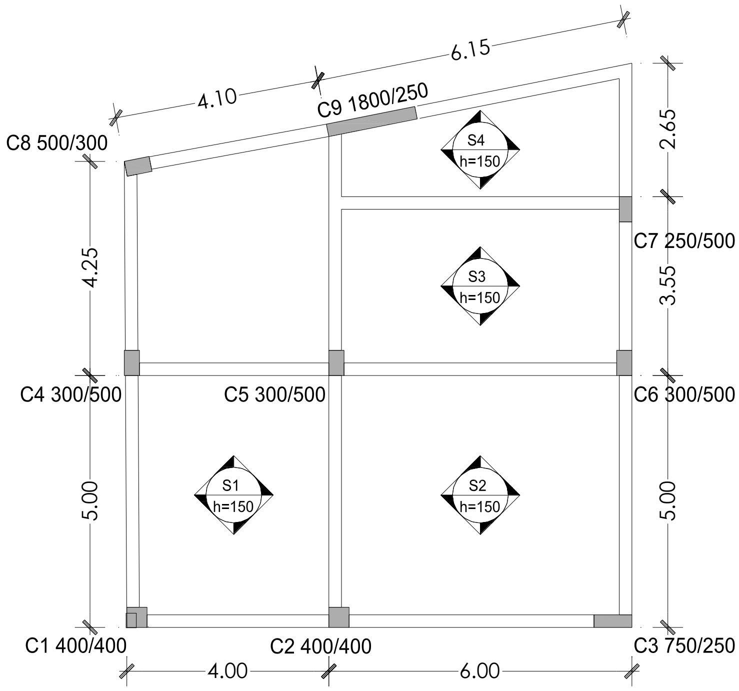
This chapter will walk you through your **first analyses with SeismoBuild**.

SeismoBuild has been designed with both ease-of-use and flexibility in mind. Our goal is to increase the productivity significantly, to the point that the assessment of a multi-storey RC building may be completed within a few minutes, including the creation of the report and the CAD drawings to be submitted to the client. It is actually much easier to use SeismoBuild than it is to describe. You will see that once you have grasped a few important concepts, the entire process is quite intuitive. The model that you will create is packed with features and can simulate efficiently and accurately real structures.

Tutorial N.1 – Assessment of a Two-Storey Building

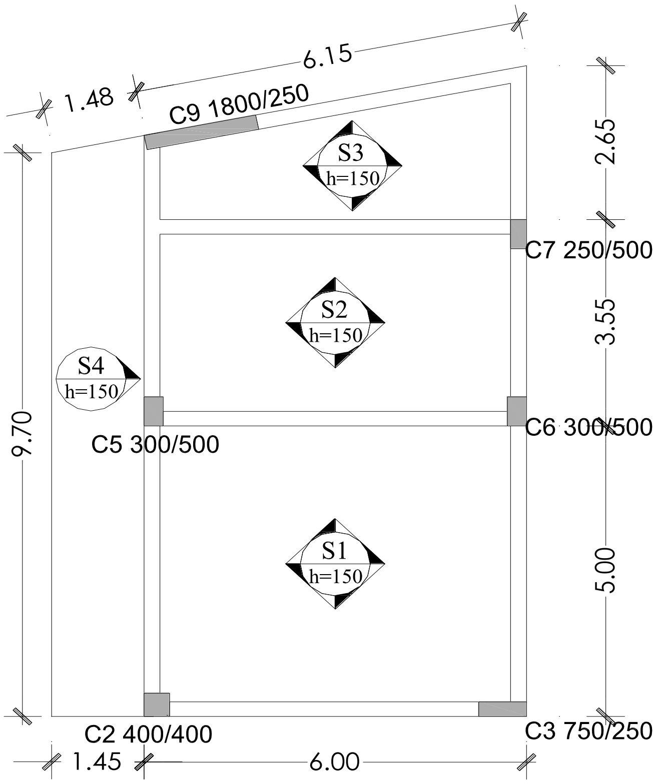
Problem Description

Let us try to model a three dimensional, two-storey reinforced concrete building for which you are asked to assess its capacity according to the Eurocodes. The Building Modeller facility offers a fast and easy definition of the building. The geometry of the first and second floor is shown in the corresponding plan-views below:



Plan view of 1st floor of the building

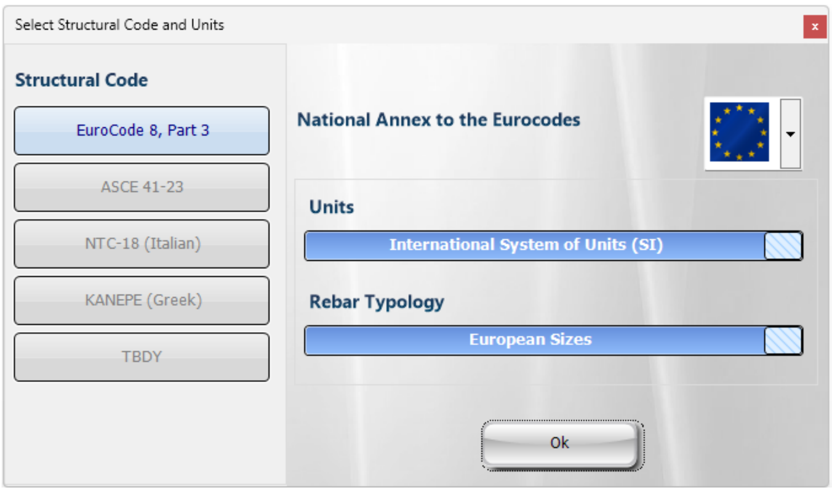
> **NOTE: A movie describing Tutorial N.1 can be found on Seismosoft's YouTube channel.**



Plan view of 2nd floor of the building

Getting started: a new project

When opening SeismoBuild a window appears for selecting the new project's structural Code, units and rebar typology. The available options depending on the edition are Eurocode 8, ASCE 41-23 (American Code for Seismic Evaluation and Retrofit of Existing Buildings), NTC-18 (Italian National Seismic Code), KANEPE (Greek Seismic Interventions Code) and TBDY (Turkish Seismic Evaluation Building Code), SI and Imperial Units, and European sizes and US sizes for the rebar typology.

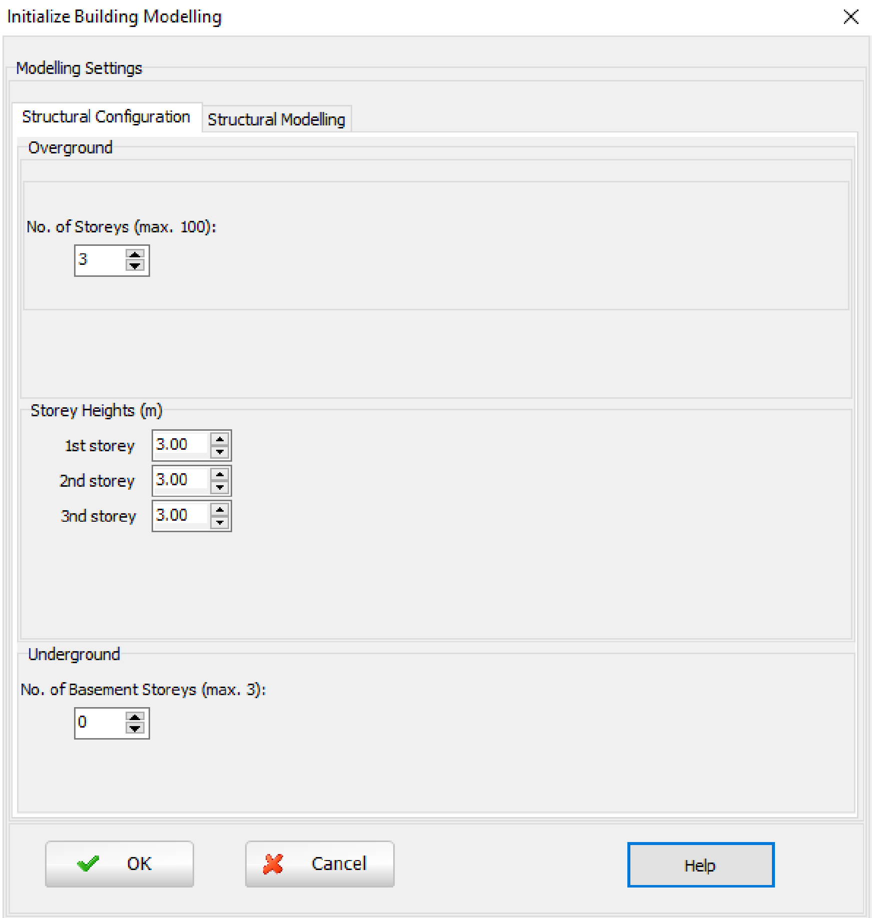


Select Structural Code & Units window

When selecting the Building Modelling module command, the Building Modeller initialisation window opens, from which the number of storeys, the storeys' heights, and other basic settings may be defined.

For this tutorial the following settings have been chosen:

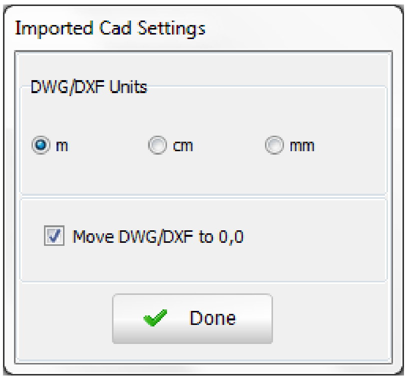
- Eurocode 8, Part 3
- SI Units
- European sizes for rebar typology
- 2 Storeys
- Storeys' heights: 3m
- Do not accept beams with free span less than: 0.1m
- Include beam effective widths



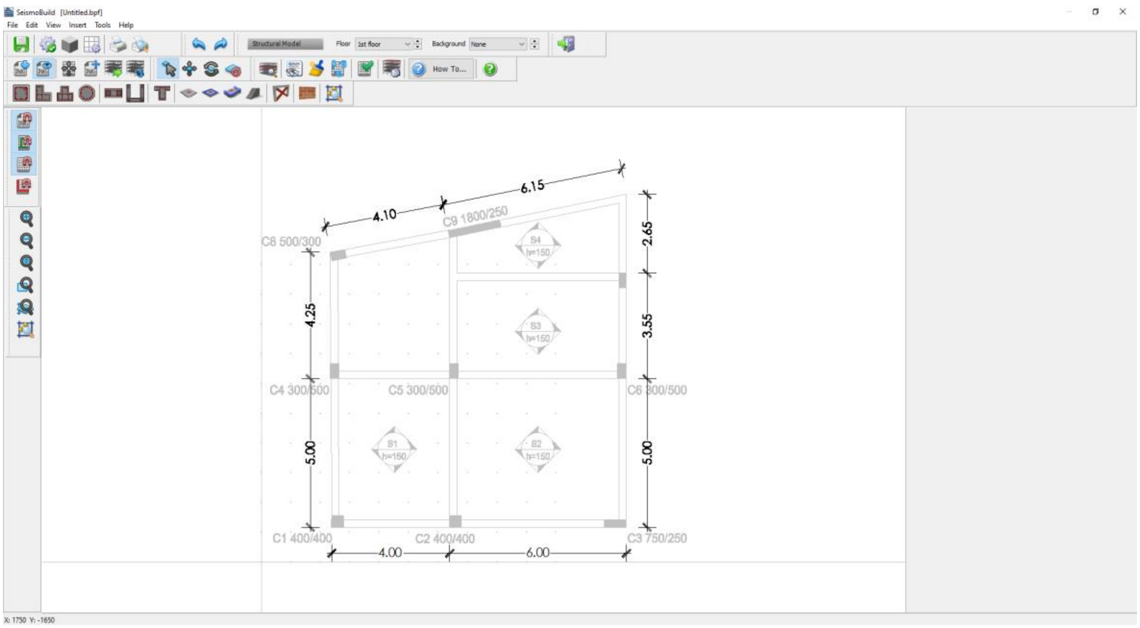
Building Modeller Initialisation module -Structural Configuration

Proceed by clicking OK.

In order to facilitate the definition of the elements' geometry and location, a CAD drawing can be imported from the main menu (*File > Import DWG...*) or through the corresponding toolbar button . A window appears for selecting the DWG/DXF units, and the user is given the possibility to move the drawing to the (0,0) origin of the CAD coordinates.

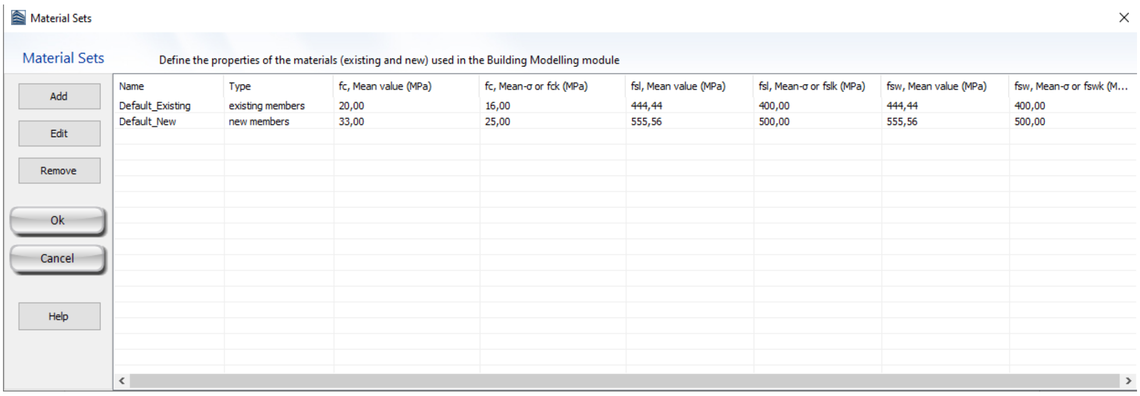


Building Modeller – Imported Cad Settings

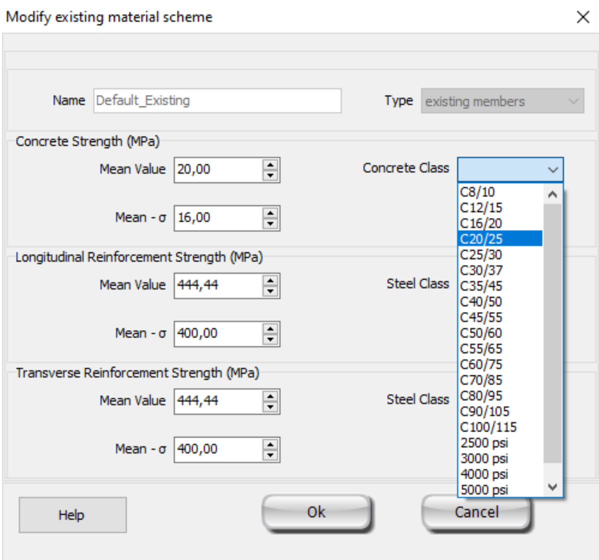


Building Modeller – CAD drawing insertion

Begin inserting the structural members from the main menu (*Insert > Insert Rectangular Column*) or through the toolbar button for rectangular columns. Alternatively, select one of the other available column sections, L-shaped (), T-shaped (), circular column () or their jacketed counterparts. The Properties Window of the column will appear on the right-hand side of the screen and the user can define its geometry, the foundation level, the longitudinal and transverse reinforcement, its material properties, the FRP wrapping and the Code-based settings for structural members. In the material sets module the member's concrete and reinforcement strength values are determined. The material set should be defined for every structural member. By default, there are two material sets in the program, one for the existing members, called *Default_Existing*, which is used in the current tutorial, and one for the new members added for rehabilitation, called *Default_New*. Users may add new material sets or edit the existing ones, but they cannot remove the default material schemes.

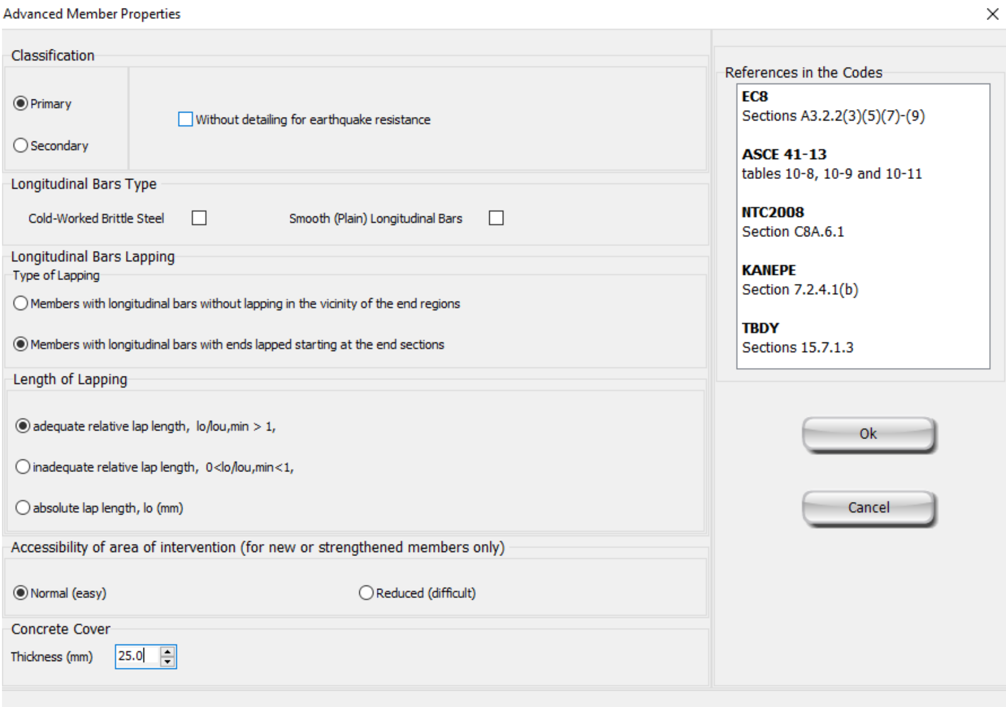


Building Modeller – Material Sets



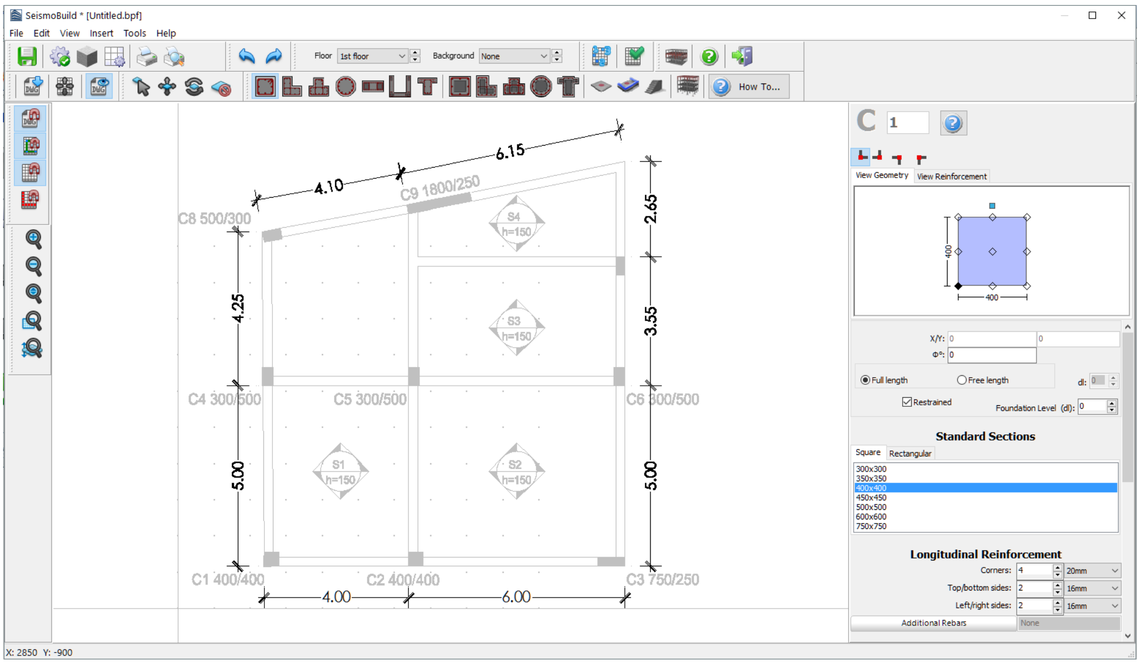
Building Modeller – Modify Existing Material Scheme

By clicking on the *Advanced Member Properties* button users may define the settings of the structural member according to the selected Code.



Building Modeller – Advanced Member Properties

Further, the 'insertion point' of the element can be chosen by clicking on the corner, middle or side points of the section's plot on the Properties Window. You are allowed to change the sections dimensions by clicking on them, whereas the rotation of the column on plan-view can be changed by the 0º, 90º, 180º and 270º buttons or by assigning the proper angle on the corresponding editbox of the Properties Window. Although different foundation levels may be defined for the columns of the first floor, for the purpose of the current tutorial a common foundation level of -1000mm is assigned to all the columns.



Building Modeller – Column Element Properties

The dimensions and the reinforcement of the members (columns, beams and walls) of the first and second floor are shown in the following tables:

Columns of 1st floor	Height (mm)	Width (mm)	Longitudinal reinforcement	Transverse reinforcement
C1	400	400	4∅18+4∅16	∅10/10
C2	400	400	4∅18+4∅16	∅10/10
C3	750	250	4∅16+8∅14	∅10/10
C4	300	500	6∅18	∅10/10
C5	300	500	6∅18	∅10/10
C6	300	500	6∅20	∅10/10
C7	250	500	4∅20+2∅16	∅10/10
C8	300	500	6∅20	∅10/10
W9	250	1800	(4∅16+8∅14)+ #∅10/20+(4∅8/m^2)	∅10/10

Beams of 1st Floor	Height (mm)	Width (mm)	Reinforcement at the Start of the beam	Reinforcement at the Middle of the beam	Reinforcement at the End of the beam	Transverse reinforcement
B1	500	250	o3∅14 u4∅14	o2∅14 u4∅14	o3∅14 u4∅14	∅8/10
B2	500	250	o3∅14 u4∅14	o2∅14 u4v14	o4∅16 u4∅14	∅8/10
B3	500	250	o3∅14 u4∅14	o2∅14 u4∅14	o3∅14 u4∅14	∅8/10
B4	500	250	o3∅14 u4∅14	o2∅14 u4∅14	o2ϕ∅20 u4∅14	∅8/10
B5	500	250	o2∅14 u4∅14	o2∅14 u4∅14	o3ϕ∅14 u4∅14	∅8/10
B6	500	250	o3∅14 u4∅14	o2∅14 u4∅14	o2∅14 u4∅14	∅8/10
B7	500	250	o3∅20 u2∅14	o4∅14 u2∅14	o3∅20 u2∅14	∅8/10
B8	500	250	o3∅14 u4∅14	o2∅14 u4∅14	o2∅14 u4∅14	∅8/10
B9	500	250	o2∅14 u4∅14	o2∅14 u4∅14	o2∅18 u4∅14	∅8/10

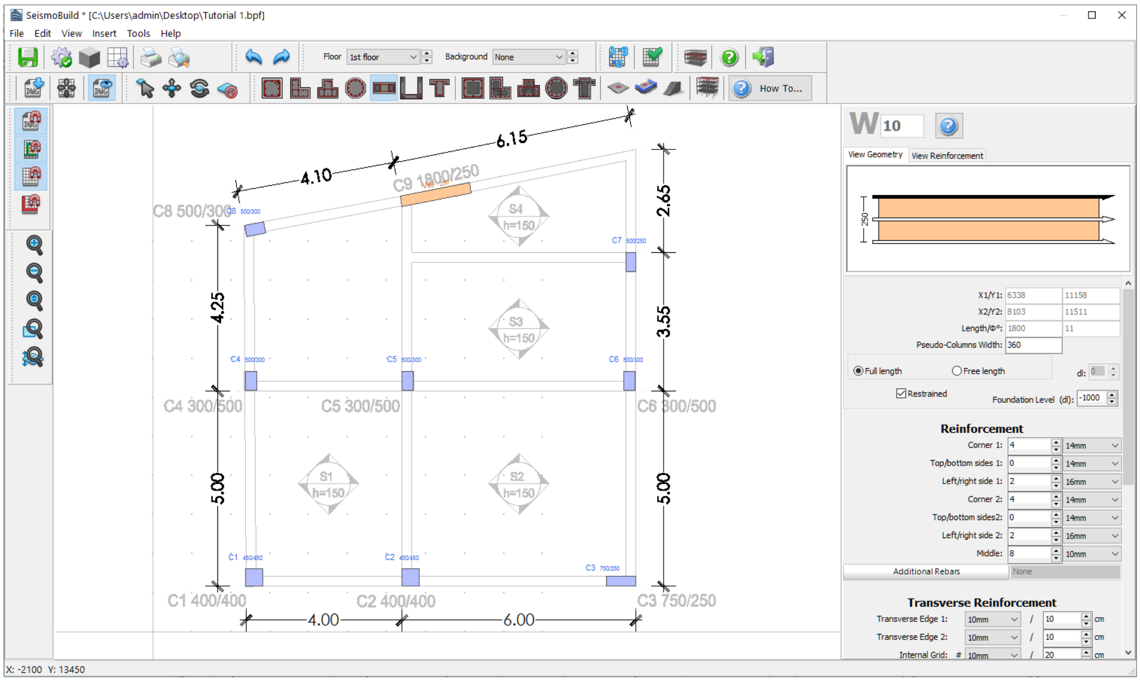
Beams of 1st Floor	Height (mm)	Width (mm)	Reinforcement at the Start of the beam	Reinforcement at the Middle of the beam	Reinforcement at the End of the beam	Transverse reinforcement
B10	500	250	o4∅16 u4∅14	o2∅14 u4∅14	o2∅18 u4∅14	∅8/10
B11	500	250	o2∅18 u4∅14	o2∅14 u4∅14	o2∅14 u4∅14	∅8/10
B12	500	250	o2∅14 u4∅14	o2∅14 u4∅14	o3∅18 u4∅14	∅8/10
B13	500	250	o2∅18 u4∅14	o2∅14 u4∅14	o3∅14 u4∅14	∅8/10
B14	500	250	o2∅18 u4∅16	o2∅16 u4∅16	o2∅16 u4∅16	∅8/10
B15	500	250	o4∅16 u2∅16	o4∅16 u2∅16	o4∅16 u2∅16	∅8/10

columns of 2nd floor	Height (mm)	Width (mm)	Longitudinal reinforcement	Transverse reinforcement
C2	400	400	4∅18+4∅16	∅10/10
C3	750	250	4∅16+8∅14	∅10/10
C5	300	500	4∅20+2∅16	∅10/10
C6	300	500	6∅20	∅10/10
C7	250	500	4∅20+2∅16	∅10/10
W9	250	1800	(4∅16+8∅14)+# ∅10/20+(4∅8/m^2)	∅10/10

Beams of 2nd Floor	Height (mm)	Width (mm)	Reinforcement at the Start of the beam	Reinforcement at the Middle of the beam	Reinforcement at the End of the beam	Transverse reinforcement
B1	500	250	o4∅16 u4∅14	o2∅14 u4∅14	o4∅16 u4∅14	∅8/10
B2	500	250	o2∅18 u4∅14	o2∅14 u4∅14	o2∅18 u4∅14	∅8/10
B3	500	250	o2∅14 u4∅14	o2∅14 u4∅14	o3∅14 u4∅14	∅8/10
B4	500	250	o3∅20 u4∅14	o2∅14 u4∅14	o2∅14 u4∅14	∅8/10
B5	500	250	o2∅18 u4∅14	o2∅14 u4∅14	o3∅14 u4∅14	∅8/10
B6	500	250	o3∅14 u4∅14	o2∅14 u4∅14	o2∅14 u4∅14	∅8/10
B7	500	250	o2∅14 u4∅14	o2∅14 u4∅14	o3∅18 u4∅14	∅8/10

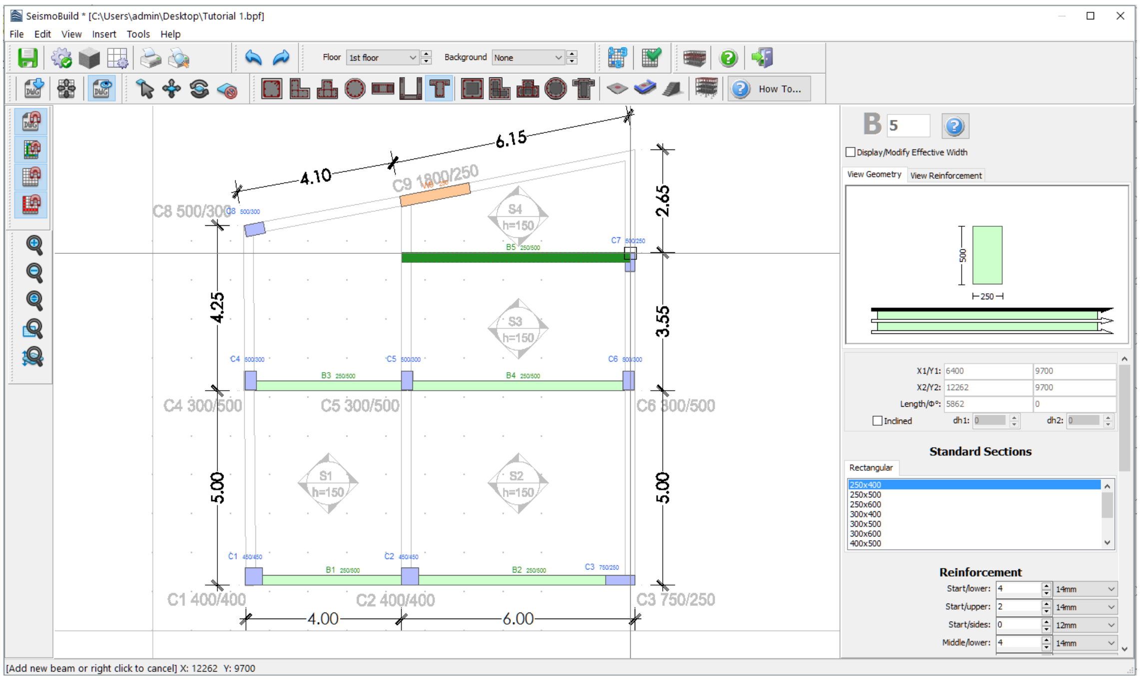
B8	500	250	o3∅14 u4∅14	o2∅14 u4∅14	o3∅14 u4∅14	∅8/10
B9	500	250	o3∅14 u4∅14	o2∅14 u4∅14	o2∅14 u4∅14	∅8/10
B10	500	250	o4∅16 u2∅16	o4∅16 u2∅16	o4∅16 u2∅16	∅8/10

After clicking on the *Insert Wall* button, the Wall's Properties Window appears, where the dimensions, the reinforcement pattern (longitudinal and transverse at the two edges and at the middle), the pseudo-columns' length, the foundation level, the material set, the FRP wrapping and the advanced code-based properties can be defined. Select the insertion line by clicking on any of the three lines on the geometry view (the left is the chosen one in the current example), and insert the structural wall by outlining its two edges on the Main Window.

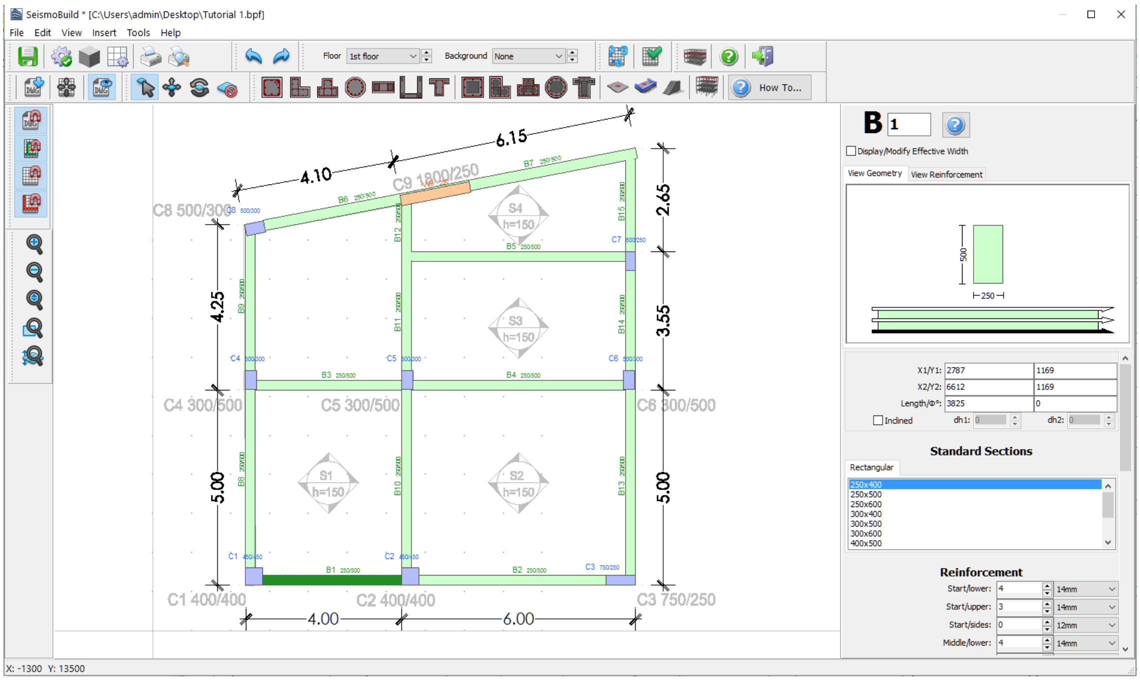


Building Modeller – Wall Element Properties

Insert the beams from the main menu (*Insert > Insert Beam*) or through the corresponding toolbar button, in a similar fashion to the walls. Again, it is possible to easily define the geometry (width and depth), the reinforcement (longitudinal and transverse reinforcement at the start, middle and end sections), the material set, the advanced properties and select the insertion line on the plan view by clicking on the preferred axis (left, centre or right). Additional distributed load may also be defined, which will serve to define any permanent load not associated to the self-weight of the structural system (e.g. finishings, infills, etc).



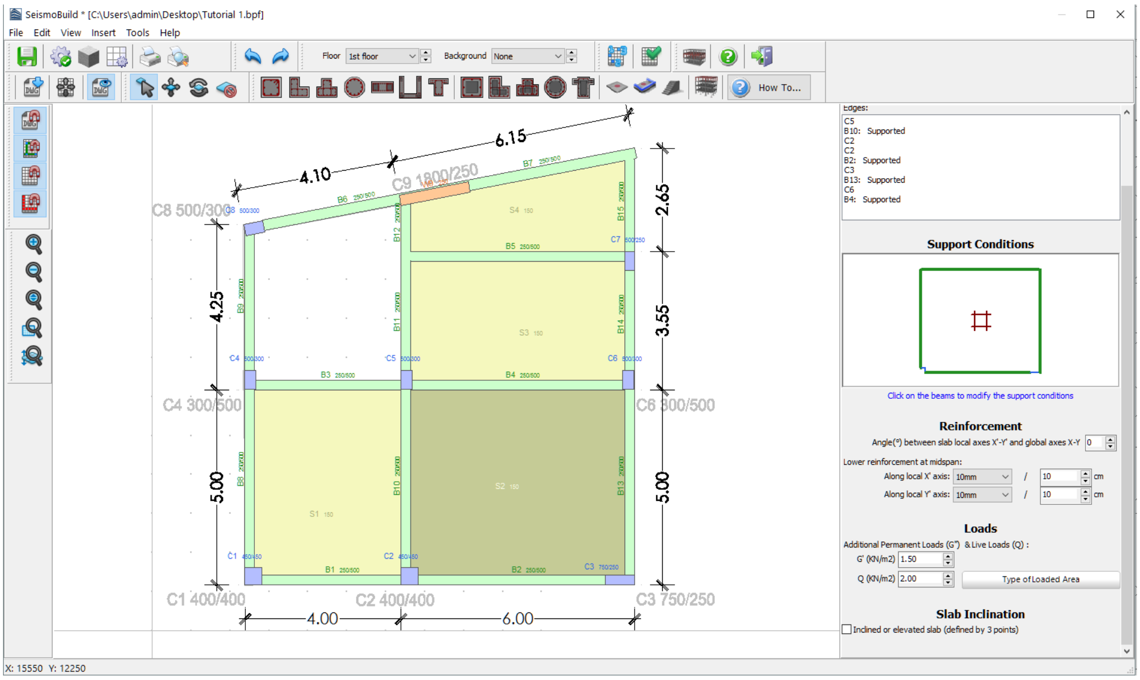
Building Modeller – Insert Elements



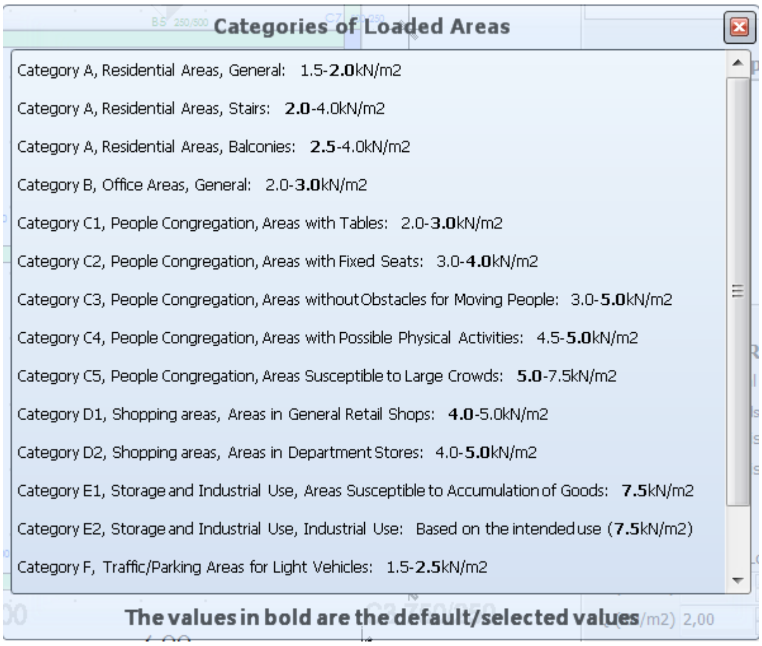
Building Modeller – Beam Element Properties

In order to insert the slabs, go to the main menu (*Insert > Insert Slab*) or click the toolbar button, assign the slab's properties (height, reinforcement, additional permanent and live loads), and click on any closed area surrounded by structural elements (columns, walls and beams). A "Type of Loaded Area" button is available so that the live loads are automatically assigned according to the loading category of the selected Code. It is noted that the self-weight of the slabs is automatically calculated according to the slabs' geometry, materials and specific weight. Once the slab is defined, its support conditions may be modified by just clicking on the corresponding boundaries of the slab's diagram on the Properties

Window. Further, the option of assigning inclined or elevated slabs, by defining the coordinates and the elevation of just three points of the slab, becomes available.



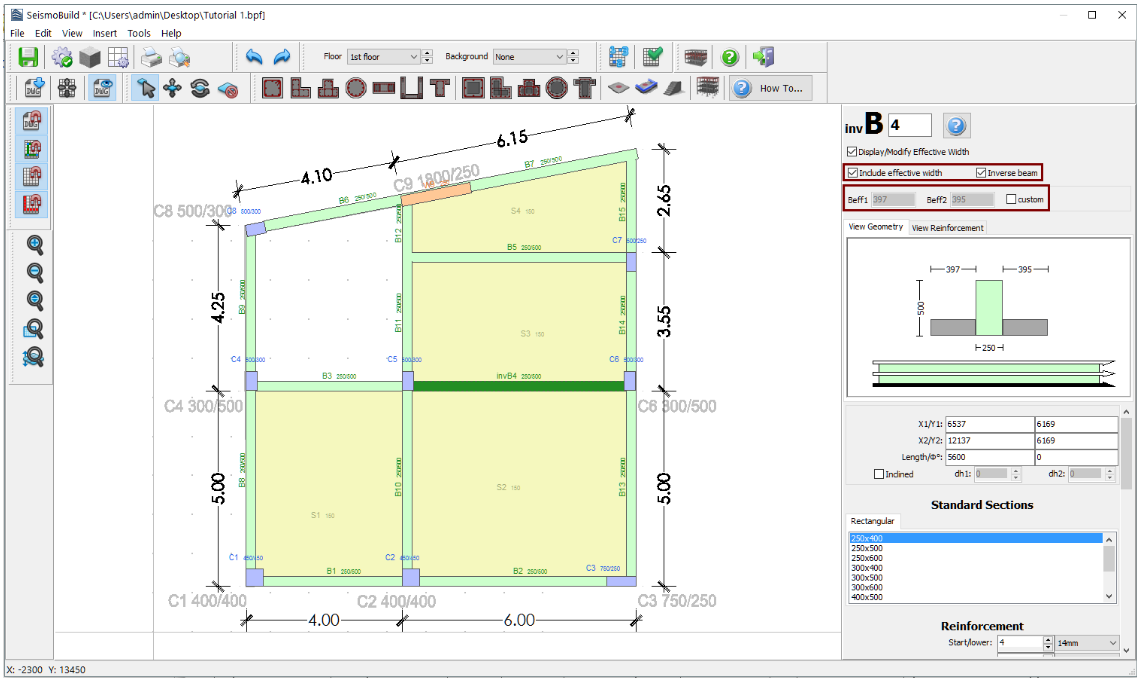
Building Modeller – Slab Element Properties



Building Modeller – Categories of Loaded Areas for Slabs

After inserting all the elements, you can change the properties of any member by clicking on it. In particular, it is noted that, after defining the slabs, you can see the beams' effective width on the beams

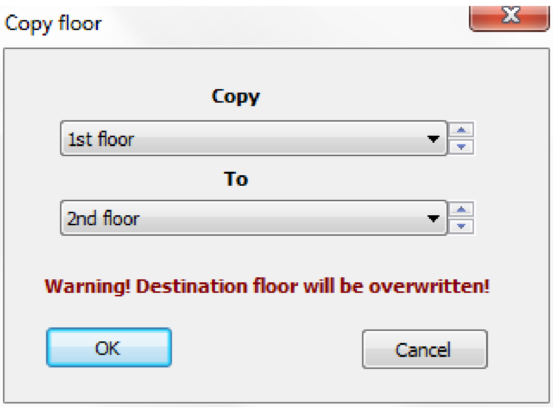
Properties Window; each beam's effective width is automatically calculated, but it can also be changed by the user. Further, inverted beams may also be defined, as shown in the figure below:



Building Modeller – Beam Element Properties

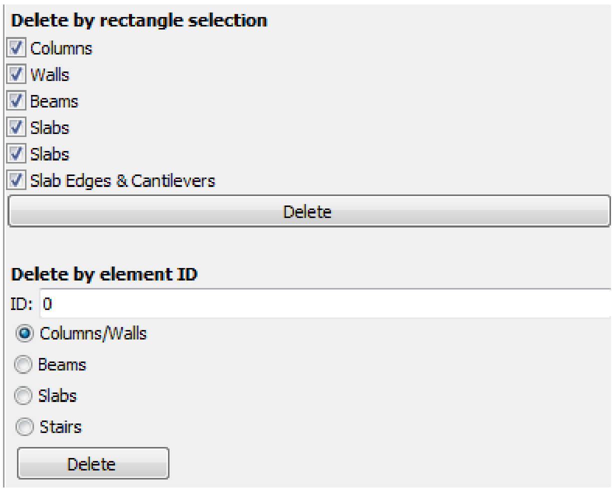
The 2nd floor can automatically be created based on the structural configuration of the 1st one; this may be done from the main menu (*Tools > Copy Floor...*) or through the button.

Now automatically create the 2nd floor based on the already created 1st one from the main menu (*Tools > Copy Floor...*) or through the button.



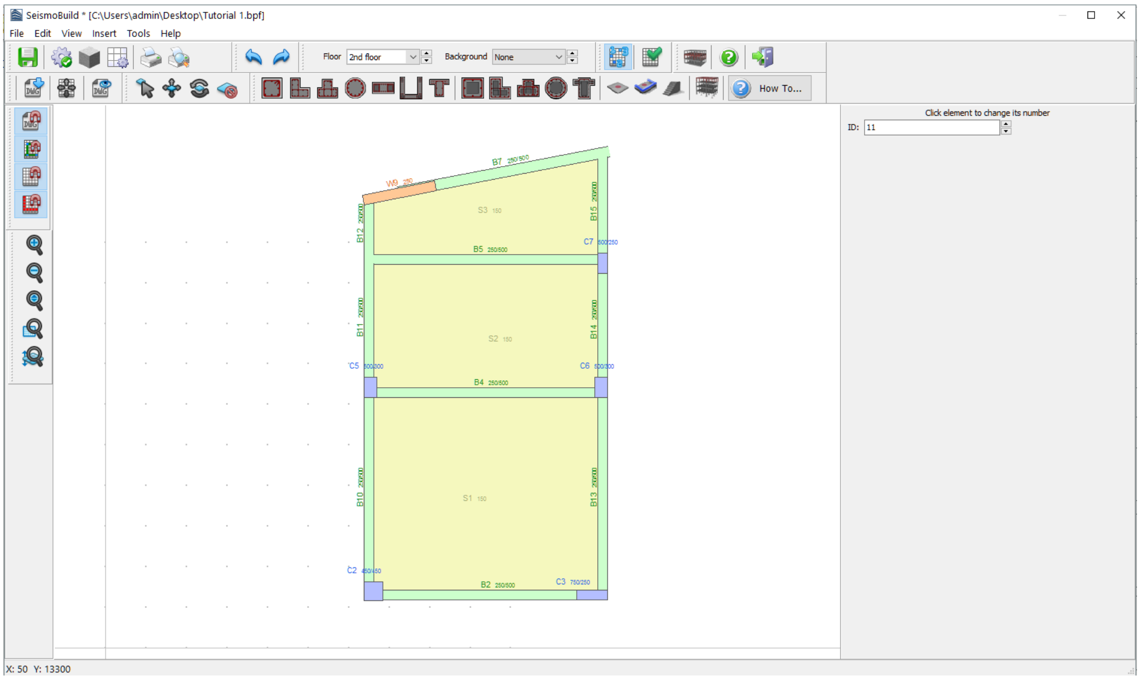
Building Modeller – Copy floor dialogue box

Delete the elements that are not present in the 2nd floor. Users can delete members from the main menu command (*Tools > Delete...*) or through the button, or by selecting a rectangular area on the Main Window and pressing the delete button.



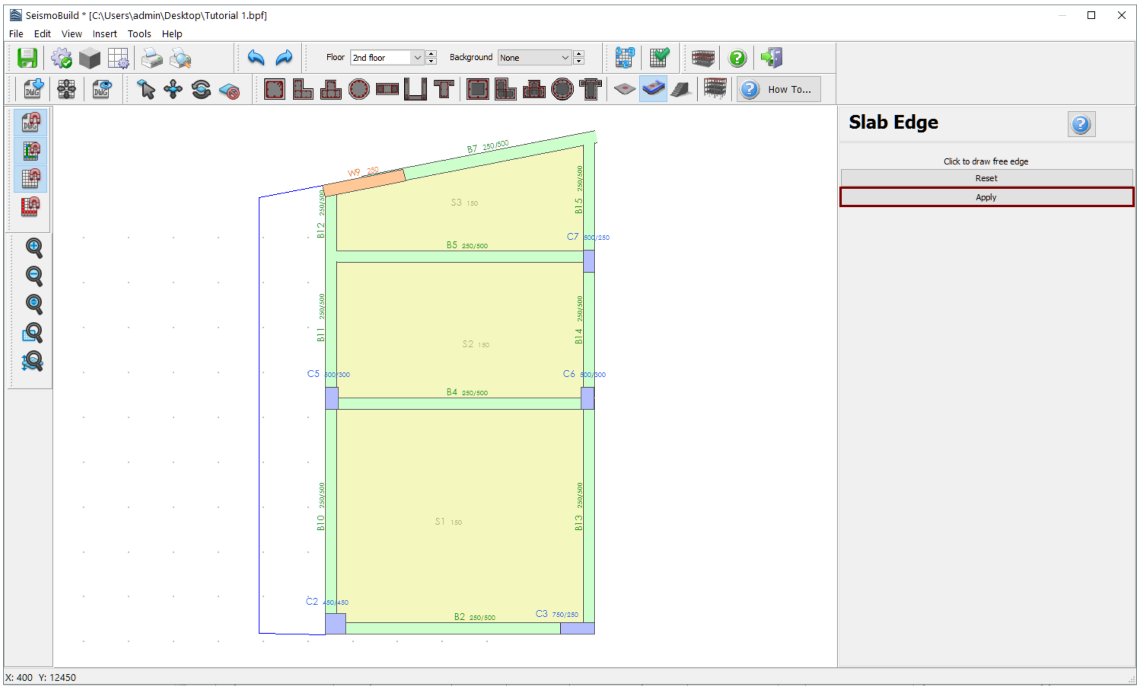
Building Modeller – Delete element dialogue box

Moreover, an option to renumber the structural members is offered from the main menu (*Tools > Renumber Elements*) or through the button. By clicking on a member, the selected number is assigned to it, and the numbering of all other members is changed accordingly.



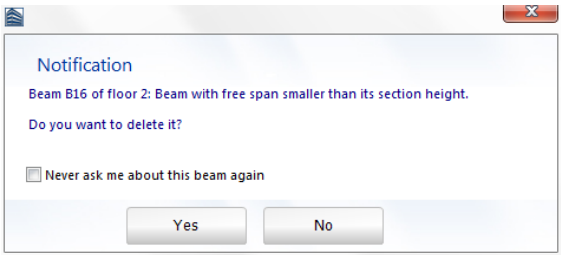
Building Modeller – Renumber elements

Cantilevered slabs can also be considered by the Building Modeller. In order to do so, a Free Edge must be added from the main menu (*Insert > Insert Slab Edges & Cantilevers*) or through the corresponding toolbar button. Once drawn, the Slab Edge is used to outline the shape of the slab. After defining the cantilever's corner points, click the *Apply* button or alternatively click the *Reset* button, if you want to redraw it. After the definition of the free edges that are needed to define a closed area, users can insert a new slab.



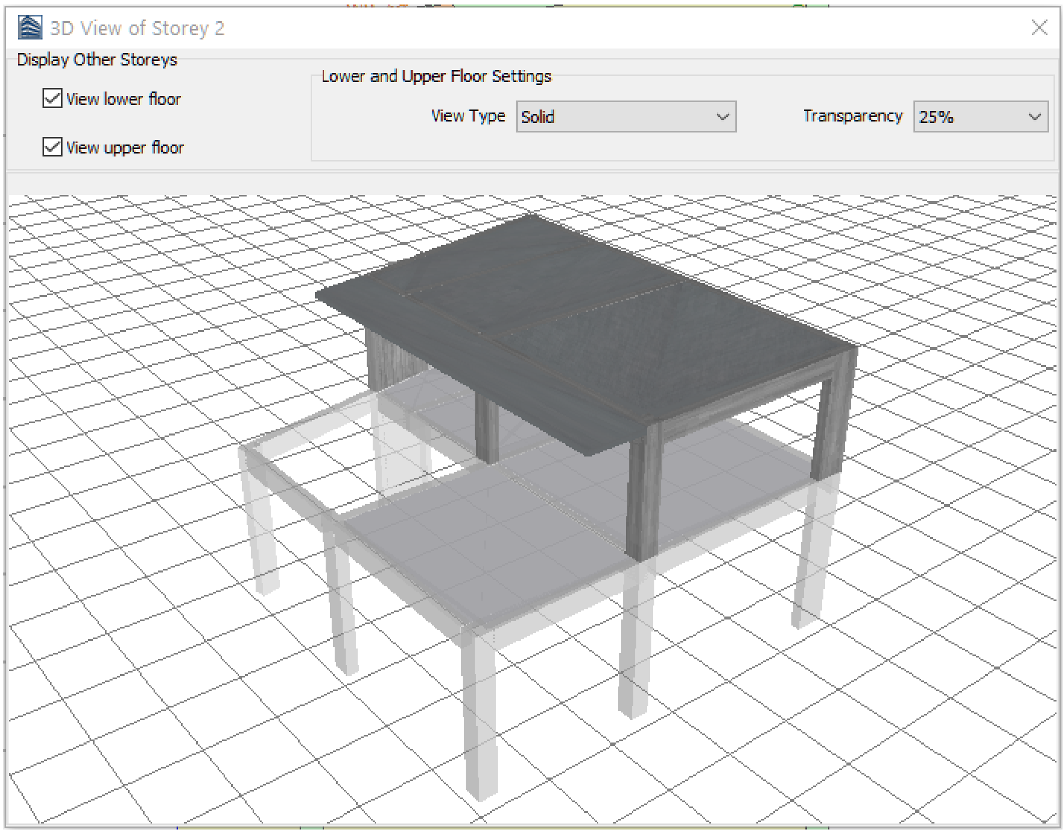
Building Modeller – Add Free Edge

When you create a building model, it is relatively common that one or more very short beams have been created unintentionally, due to graphical reasons (e.g. by extending slightly a beam's end beyond a column edge). For this reason, a check from the main menu (*Tools > Verify Connectivity...*) or through the toolbar button for the existence of any beam with free span smaller than its section height should be carried out. If such beams exist, the following message appears, and the user can select to remove or keep the element.



Building Modeller – Verify connectivity

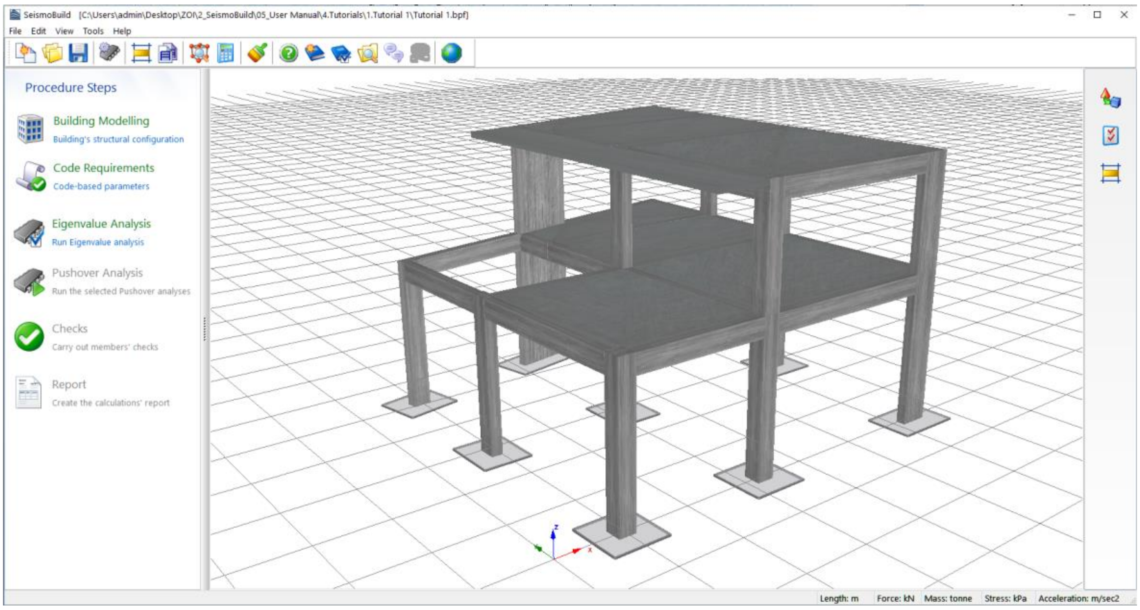
Finally, you may view the 3D model of the current floor to check for its correct definition through the toolbar button.



Building Modeller – View Storey 3D Window

With the building model now fully defined, save the project as a SeismoBuild file (with the *.bpf extension, e.g. Tutorial_1.bpf) from the main menu (*File >Save As...*)/(*File >Save*) or through the toolbar button.

You are ready to go to the SeismoBuild Main Window. This can be done from the main menu (*File > Exit & Create 3D Model*) or through the corresponding toolbar button.



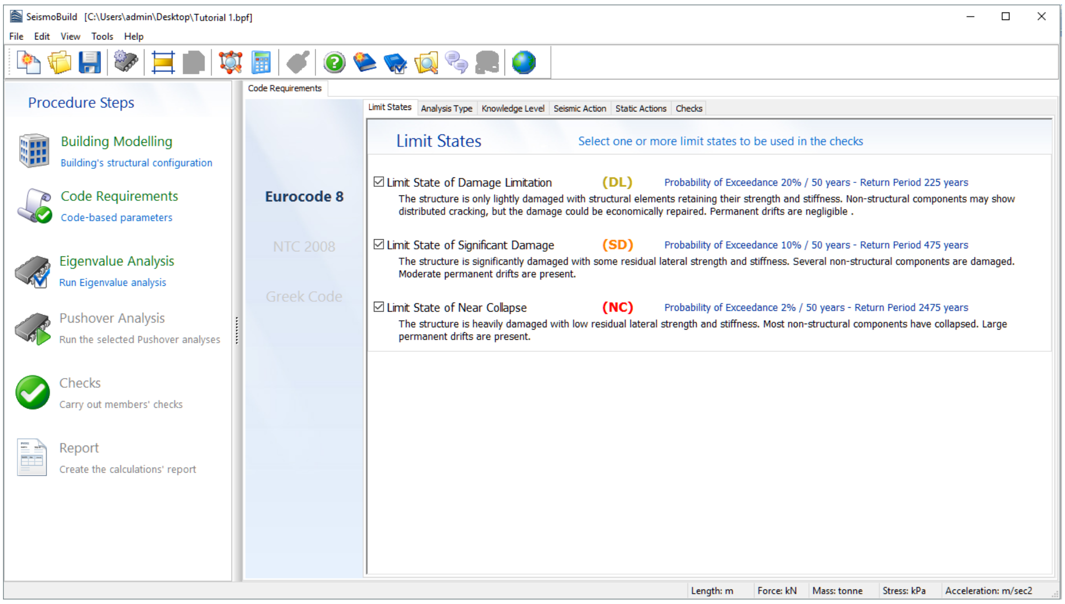
SeismoBuild Main Window

Code Requirements

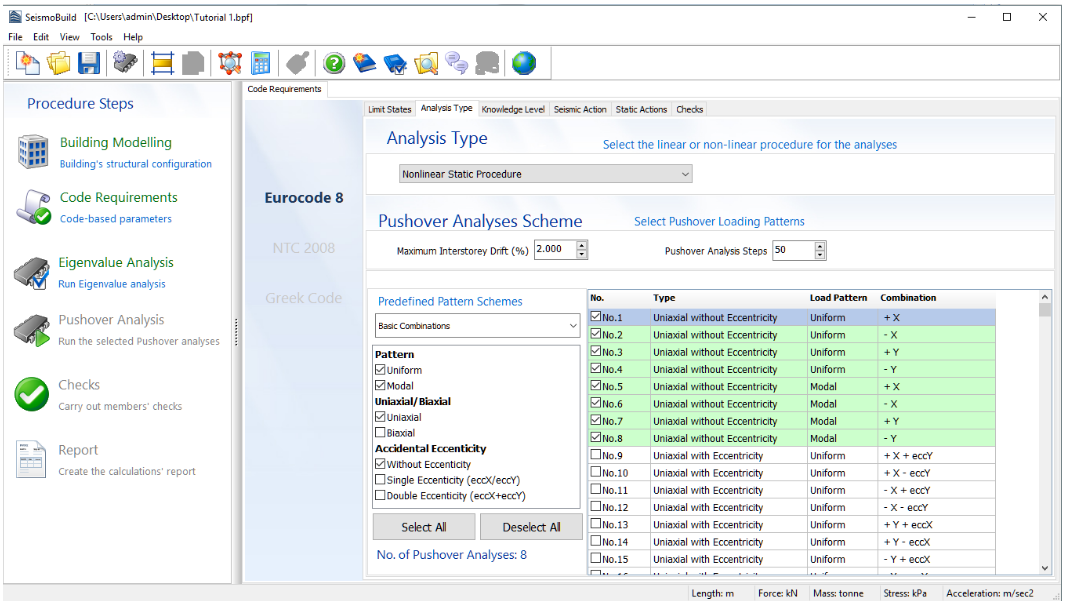
Through the Code requirements module, you are able to define the Code-based parameters and options. The available modules for Eurocode 8 are the Limit States, the Analysis Type, the Knowledge Level, the Seismic Action, the Static Actions and the Checks.

The Code parameters and options are defined for the scope of this tutorial through the following selections:

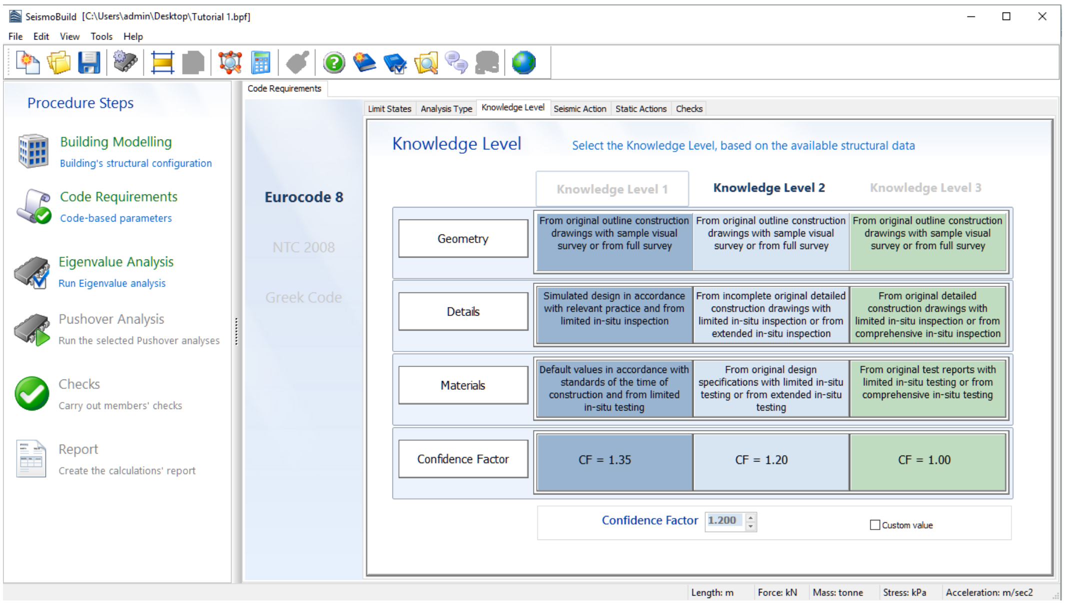
- All the limit states available in EC8 are to be used in the checks, that is the Limit States of Damage limitation, of Significant Damage and of Near Collapse;



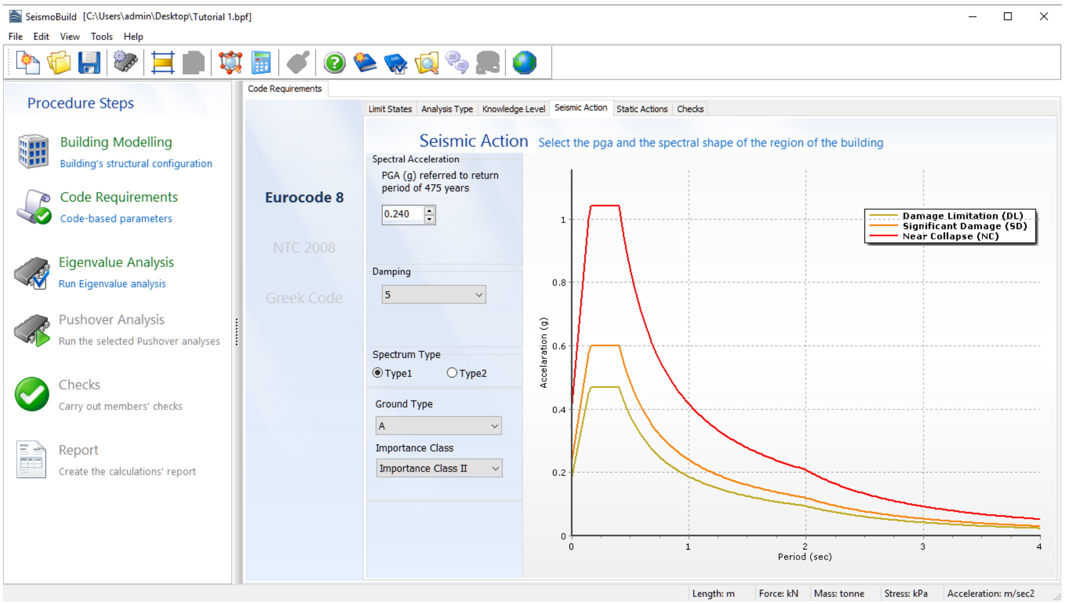
- The Nonlinear Static Procedure type of analysis is selected, with the eight basic loading patterns (uniform or modal uniaxial patterns without eccentricities);


SeismoBuild [C:\Users\admin\Desktop\Tutorial 1.bpf]
File Edit View Tools Help
Procedure Steps
Building Modelling
Building's structural configuration
Code Requirements
Code-based parameters
Eigenvalue Analysis
Run Eigenvalue analysis
Pushover Analysis
Run the selected Pushover analyses
Checks
Carry out members' checks
Report
Create the calculations' report
Code Requirements
Eurocode 8
NTC 2008
Greek Code
Limit States
Analysis Type
Knowledge Level
Seismic Action
Static Actions
Checks
Analysis Type
Select the linear or non-linear procedure for the analyses
Nonlinear Static Procedure
Pushover Analyses Scheme
Select Pushover Loading Patterns
Maximum Interstorey Drift (%) 2.000
Pushover Analysis Steps 50
Predefined Pattern Schemes
Basic Combinations
Pattern
Uniform
Modal
Uniaxial/Biaxial
Uniaxial
Biaxial
Accidental Eccentricity
Without Eccenticity
Single Eccentricity (eccX/eccY)
Double Eccentricity (eccX+eccY)
Select All
Deselect All
No. of Pushover Analyses: 8
No. Type Load Pattern Combination
No.1 Uniaxial without Eccentricity Uniform + X
No.2 Uniaxial without Eccentricity Uniform - X
No.3 Uniaxial without Eccentricity Uniform + Y
No.4 Uniaxial without Eccentricity Uniform - Y
No.5 Uniaxial without Eccentricity Modal + X
No.6 Uniaxial without Eccentricity Modal - X
No.7 Uniaxial without Eccentricity Modal + Y
No.8 Uniaxial without Eccentricity Modal - Y
No.9 Uniaxial with Eccentricity Uniform + X + eccY
No.10 Uniaxial with Eccentricity Uniform + X - eccY
No.11 Uniaxial with Eccentricity Uniform - X + eccY
No.12 Uniaxial with Eccentricity Uniform - X - eccY
No.13 Uniaxial with Eccentricity Uniform + Y + eccX
No.14 Uniaxial with Eccentricity Uniform + Y - eccX
No.15 Uniaxial with Eccentricity Uniform - Y + eccX
Length: m
Force: kN
Mass: tonne
Stress: kPa
Acceleration: m/sec2

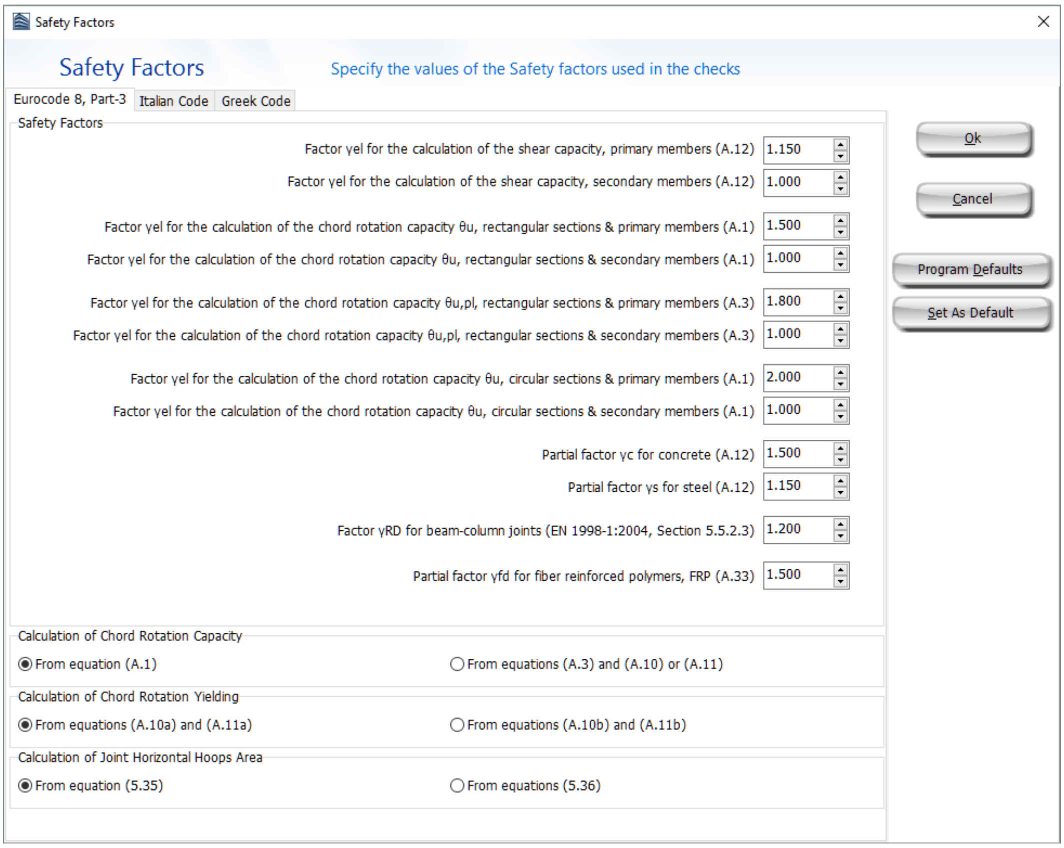
- The Knowledge Level 2 is chosen, with a confidence factor value equal to 1.2;



- The peak ground acceleration is set equal to 0.24g, whereas the default values are employed for the other parameters (5% damping, Type 1 spectral shape, ground type A and Importance class II);

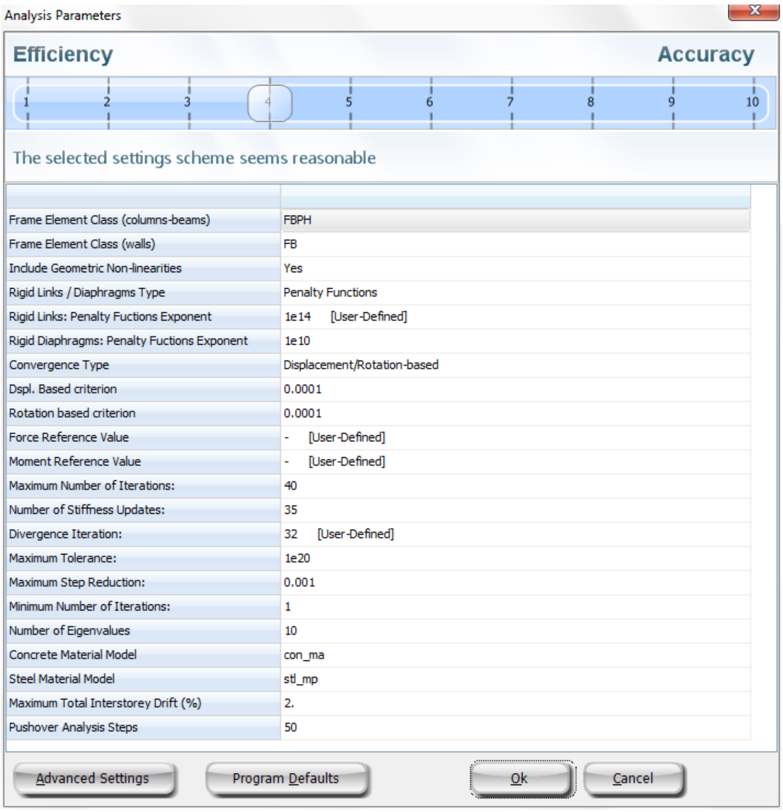


- The Permanent Loads and Live Loads Coefficients are set equal to 1.0 and 0.3, respectively;
- Select all the checks to be carried out, that is Members Chord Rotations checks, Members Shear Forces, Joints Shear Forces, Joints Horizontal Hoops Area and Joints Vertical Reinforcement Area checks. Finally, leave the program default values for all the safety factors.



Analysis & Modelling Parameters

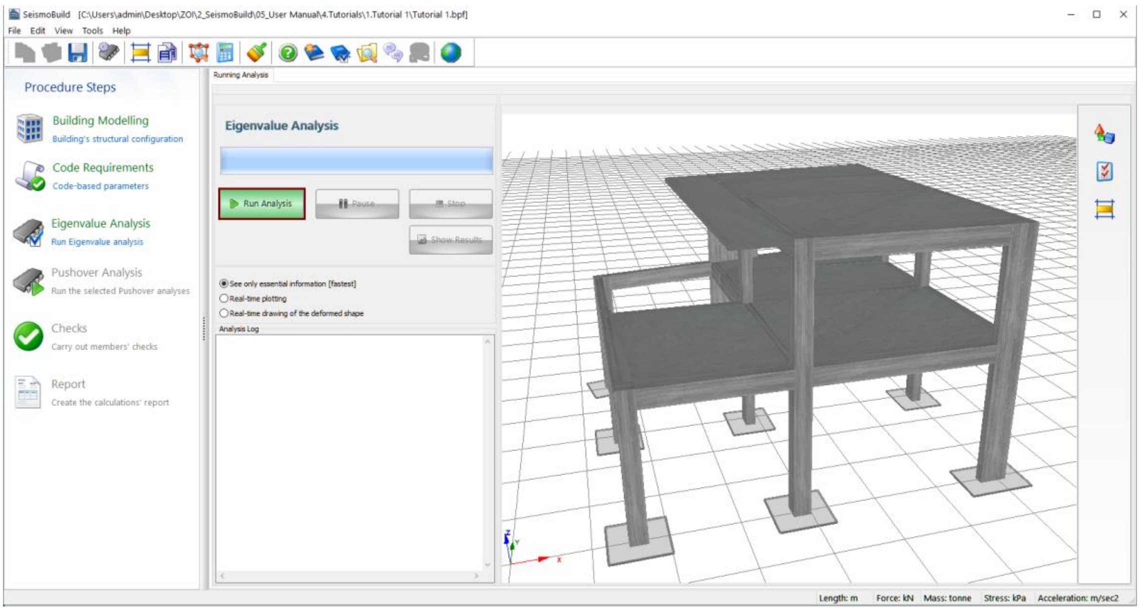
In the Analysis & Modelling Parameters module, accessed through the main menu (*Tools >Analysis & Modelling Parameters*) or through the toolbar button, the parameters specific to the numerical analysis may be defined. Predefined analysis schemes are available for the users' convenience and in order to avoid the introduction of parameter values that may lead to convergence difficulties in the analyses. In the following figure the selected analyses & modelling parameters are shown:



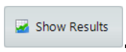
Analysis Parameters module

Eigenvalue Analysis

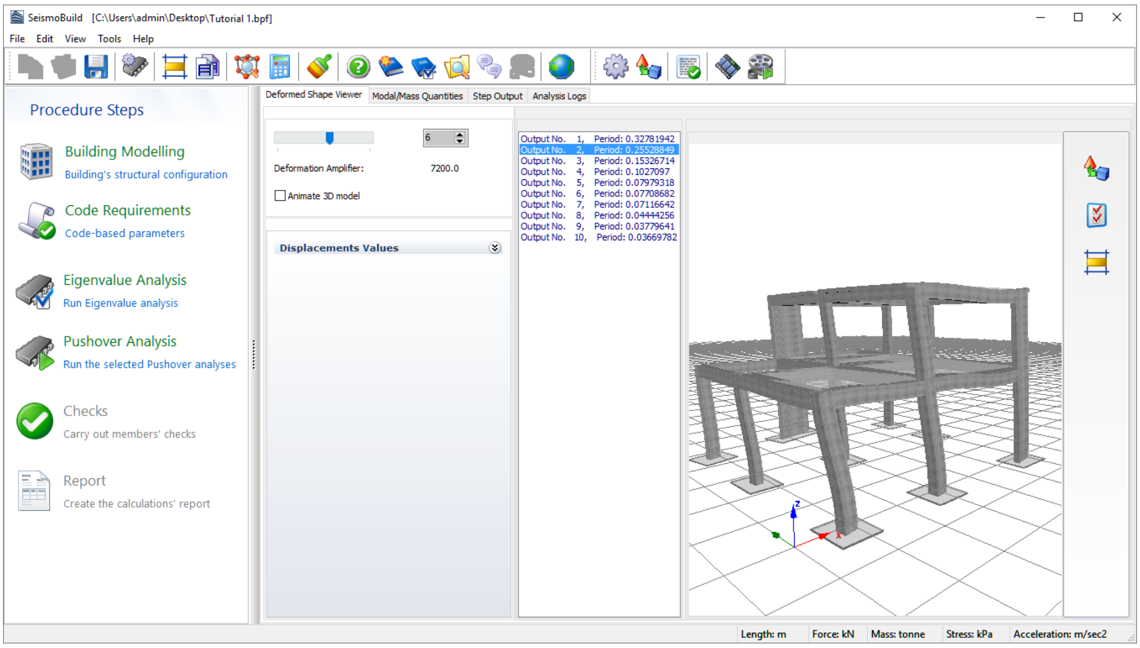
Run the eigenvalue analysis through this module.



Eigenvalue Analysis

After running the analysis, you may see the results by clicking on the *Show Results* button 

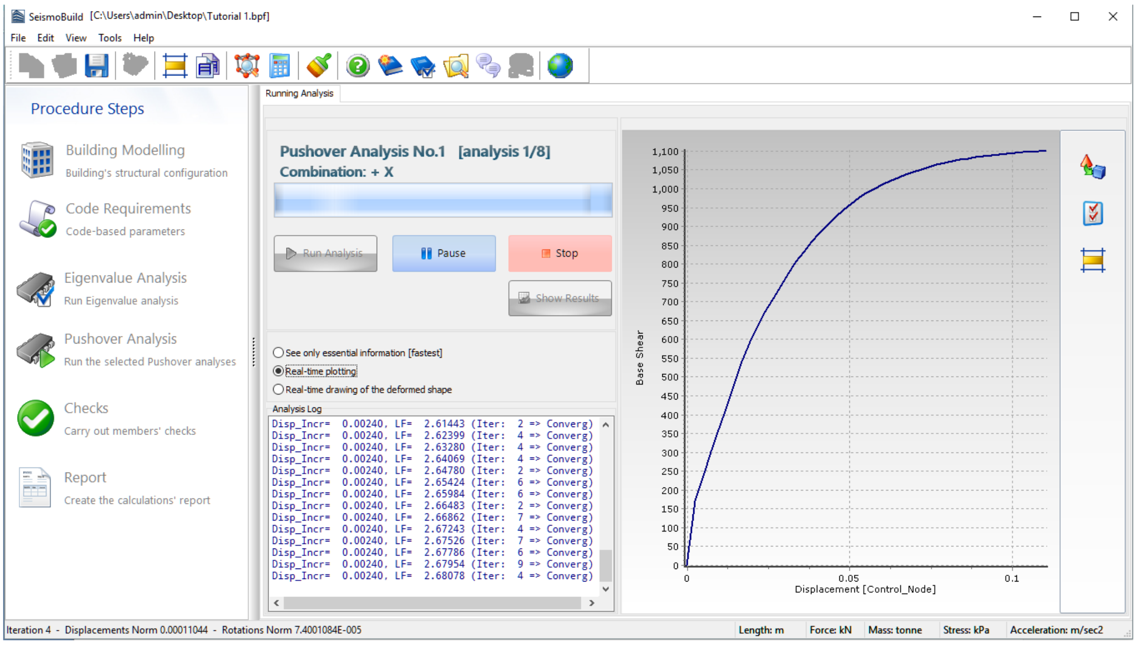
.



Eigenvalue Analysis Results

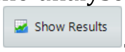
Pushover Analysis

Click on the *Run* button to run all the selected pushover analyses.

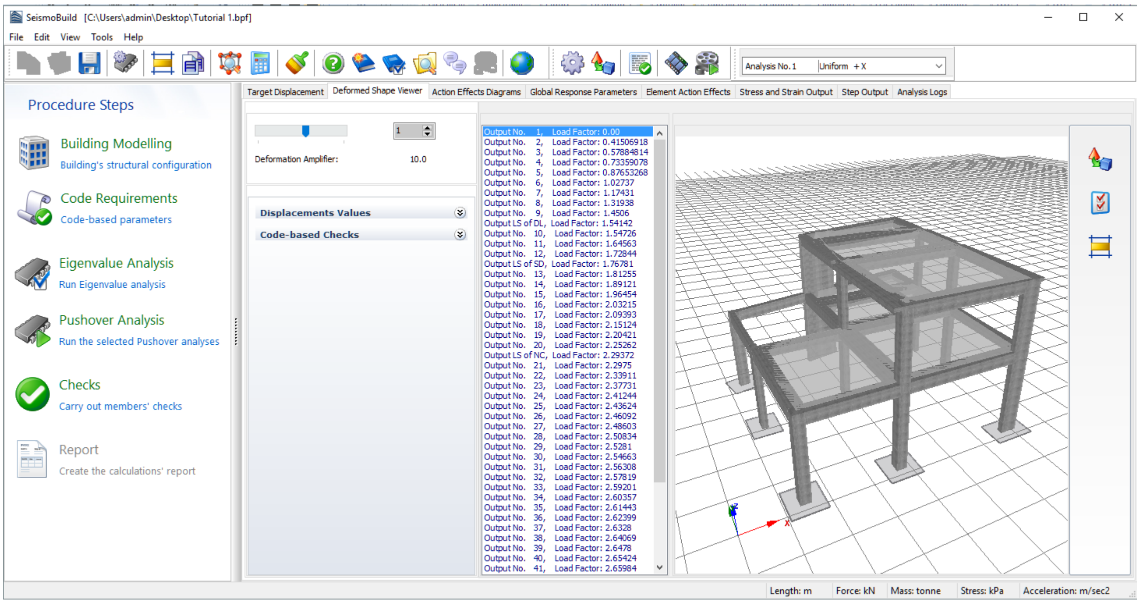


Running the analysis

NOTE: You may choose between three graphical options: (i) see only essential information, (ii) real-time plotting (in this case the Base shear vs. Top displacement capacity curve is shown) and (iii) real-time drawing of the deformed shape. The former is the fastest option.

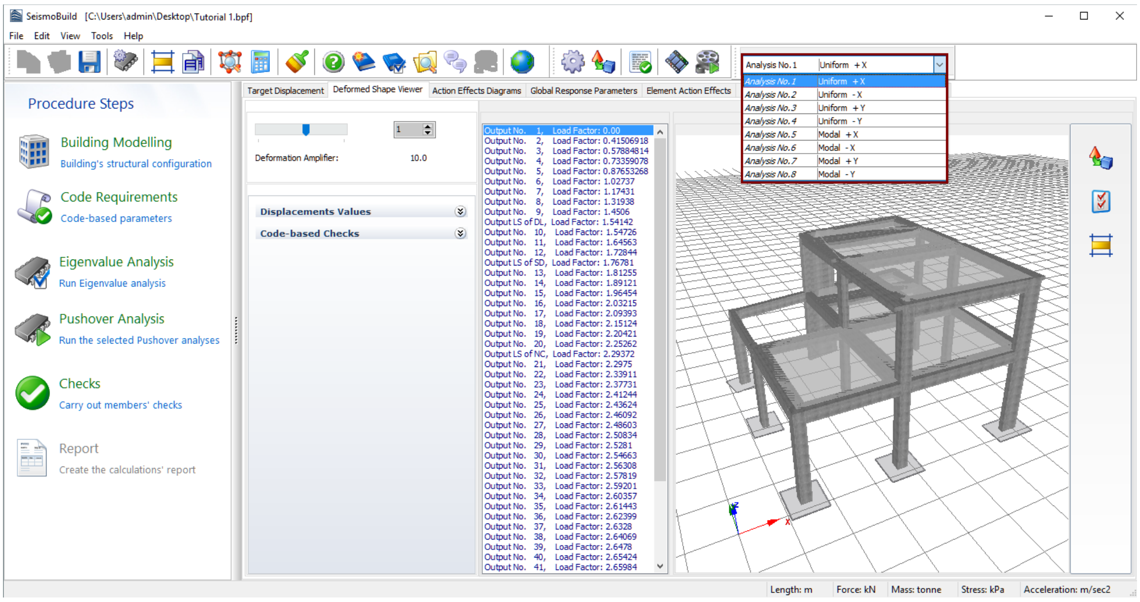
When the analyses have arrived to the end, you may see the results by clicking on the *Show Results* button 

.



Pushover Analysis Results

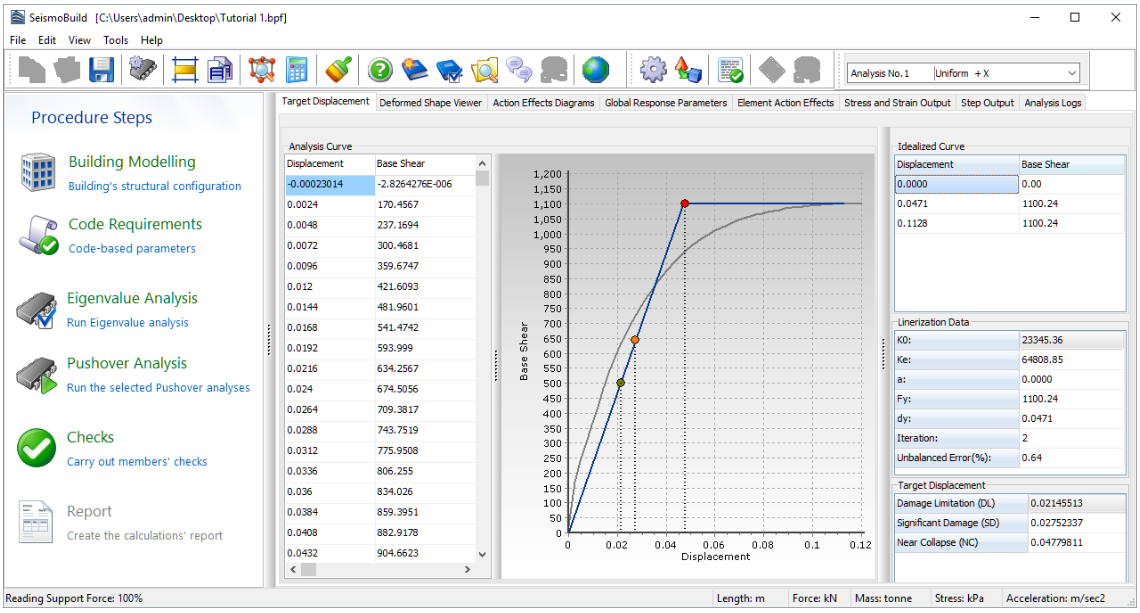
You may choose which analysis results to view from the corresponding drop-down menu on the toolbar.



Select the Pushover Analysis to view

Show Results – Target Displacement

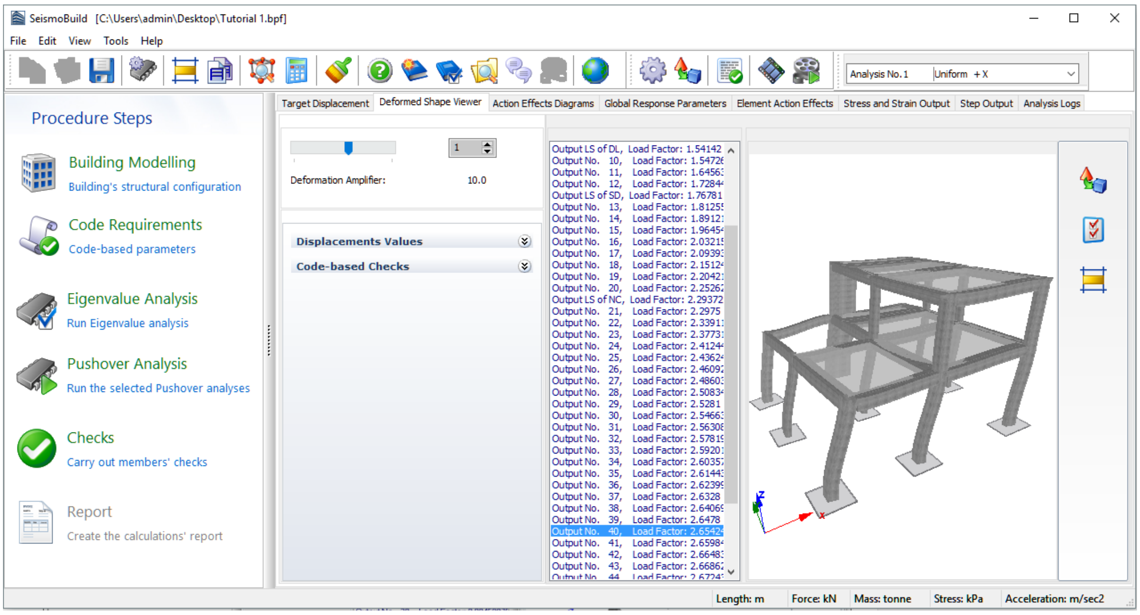
In the **Target Displacement** module, you can visualise the created capacity curve for each pushover analysis, the idealized bi-linear curve, as well as the target displacement values at the considered Limit State– Near Collapse, Significant Damage or Damage Limitation.



Target Displacement

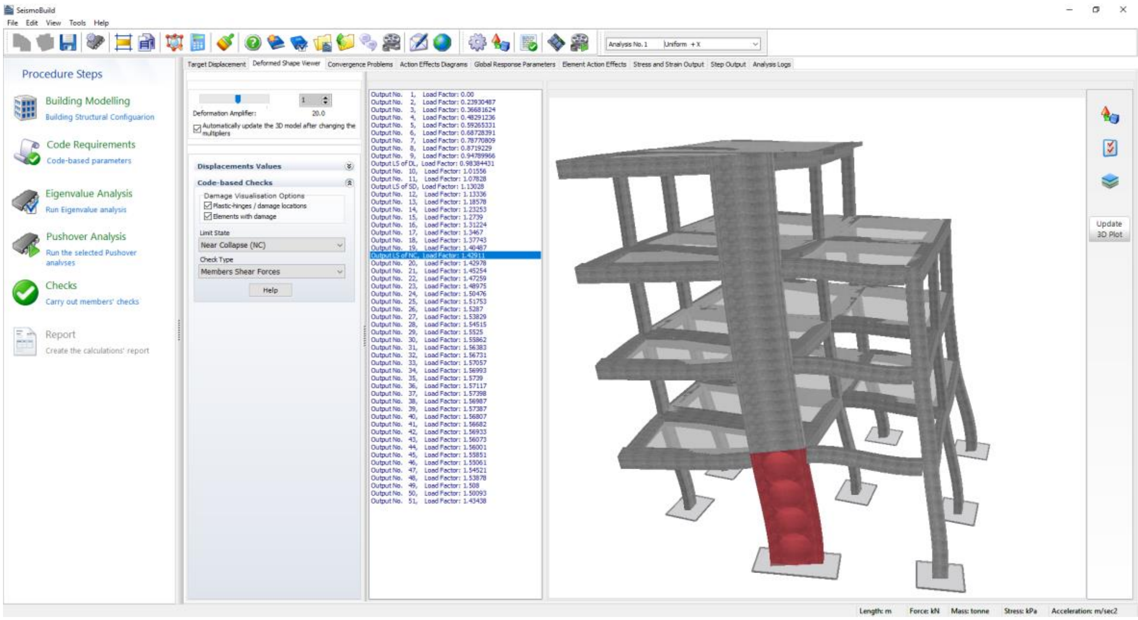
Show Results – Deformed Shape Viewer

In the **Deformed Shape Viewer**, you can visualise the deformed shape of the model at every step of the analysis (click on the desired output identifier to update the deformed shape view.



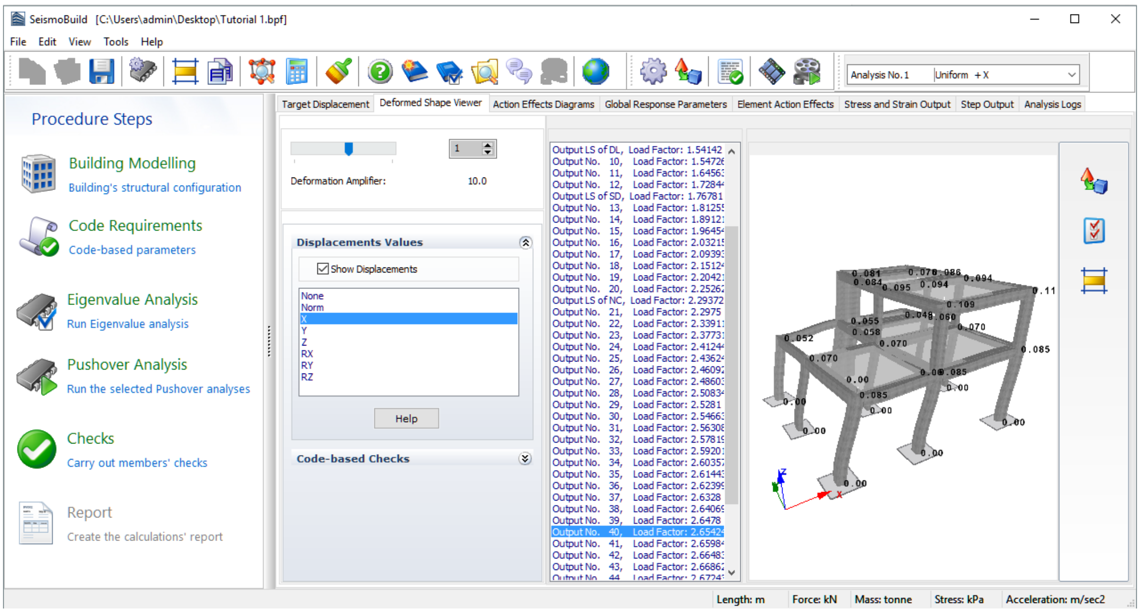
Deformed Shape Viewer module

The elements that have exceeded their capacity at a particular code-based check may be visualised by selecting the Code-based Checks option and selecting whether the plastic hinges/ damage locations will be shown as well as whether the damaged elements will be distinguished through colours.



Deformed Shape Viewer module, Code-based Checks display

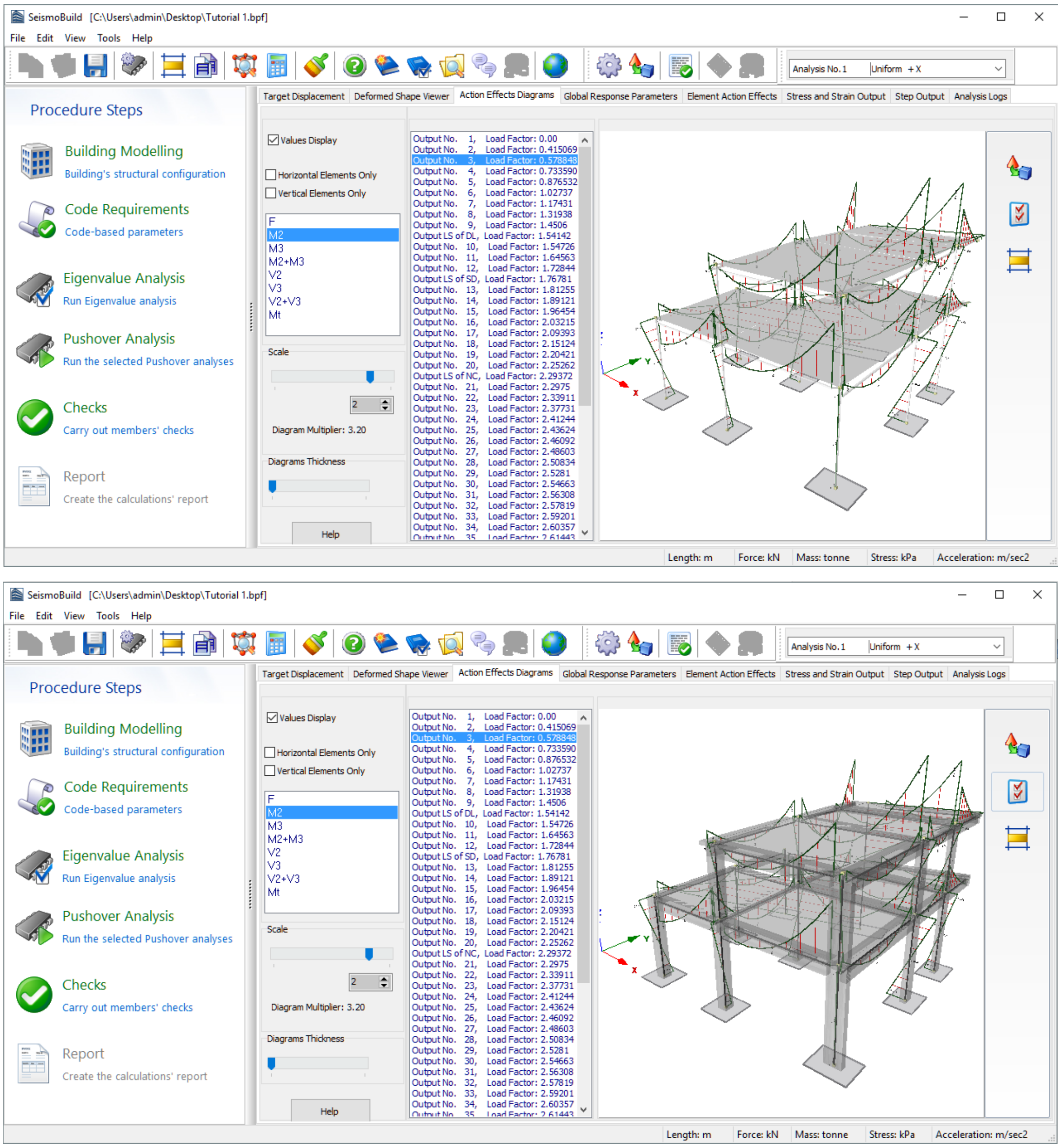
It is also possible to visualise the displacement values by checking the relevant checkbox.



Deformed Shape Viewer module, deformations display

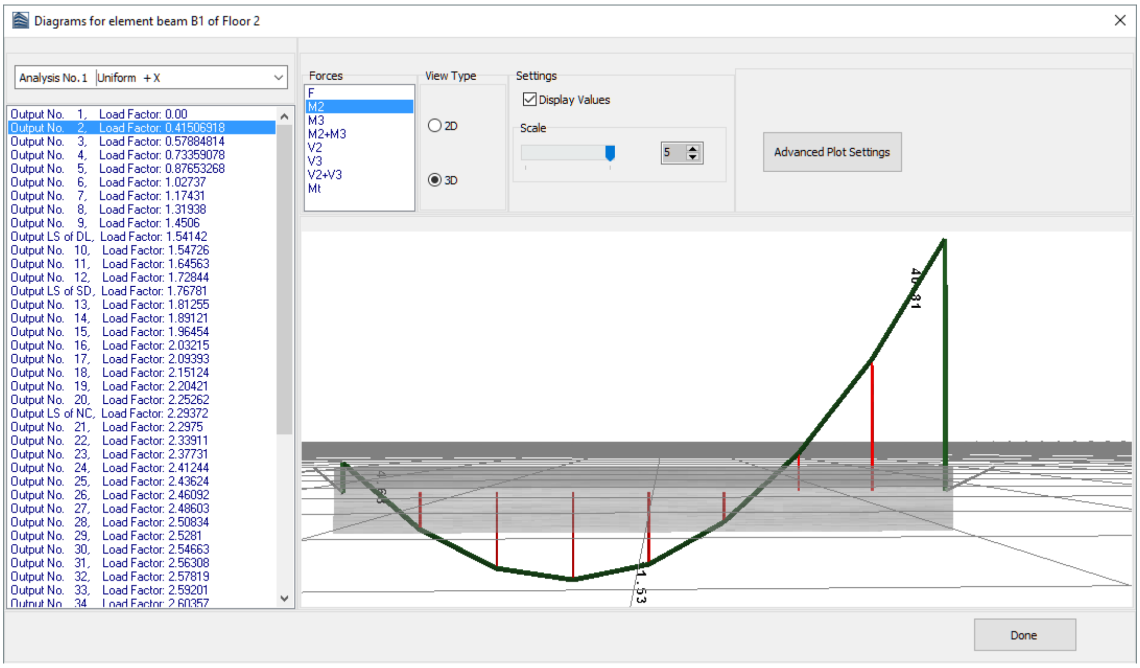
Show Results –Action Effects Diagrams

In the **Action Effects Diagrams** page, you can visualise the internal forces and moments diagrams for each analysis step, as shown below:

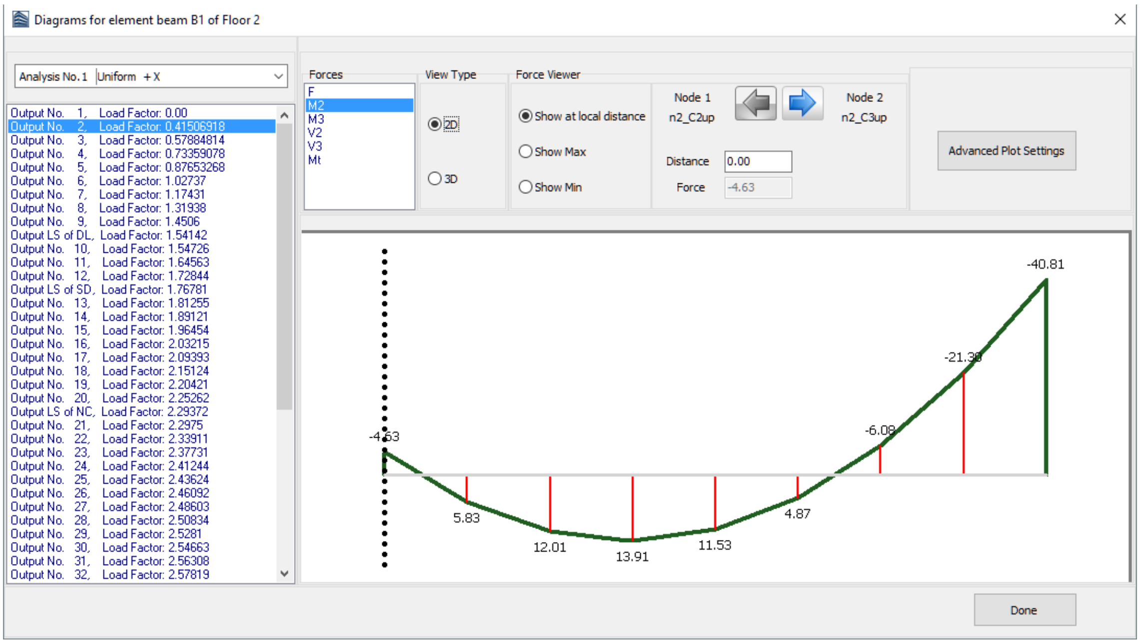


Action Effects Diagrams Module

By double-clicking on any element, you can see its action effects diagrams in 3D or 2D as shown in the figures below:



Diagrams for a beam element in 3D



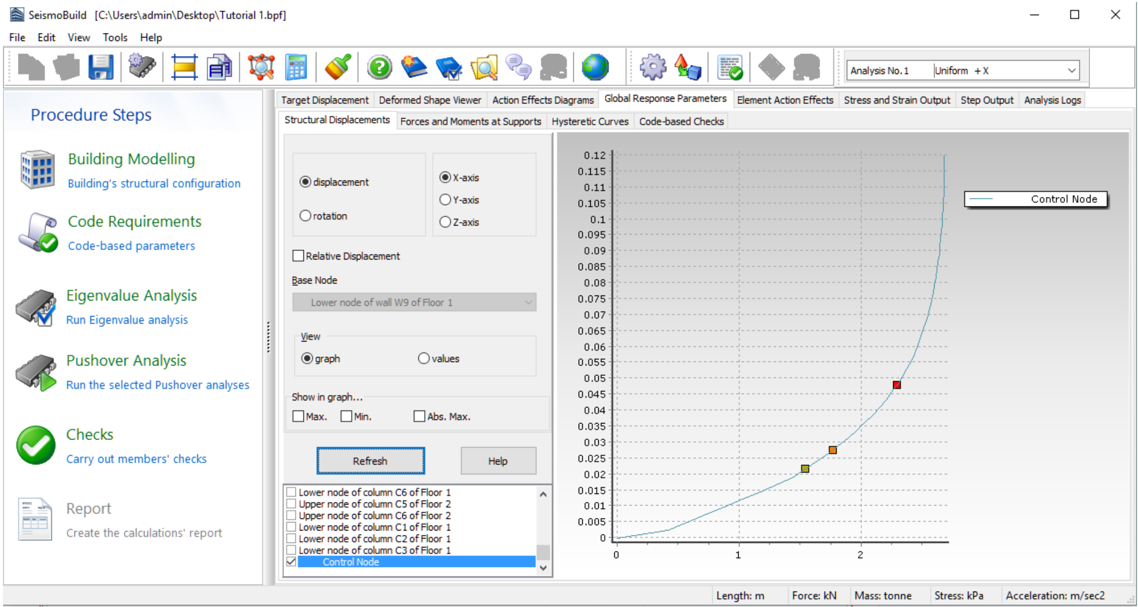
Diagrams for a beam element in 2D

Show Results – Global Response Parameters

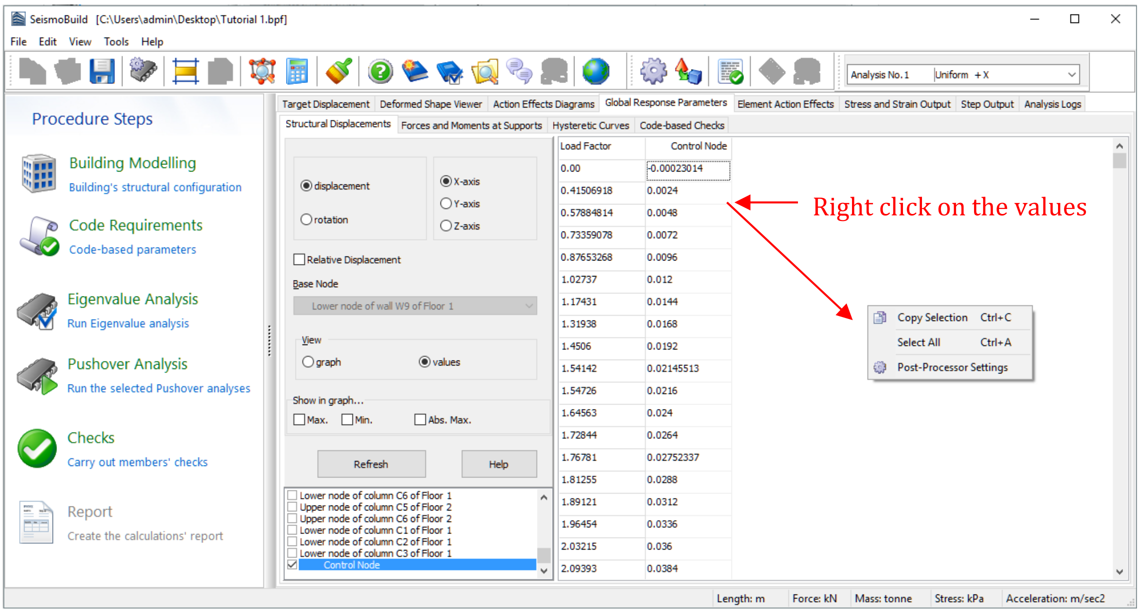
In the **Global Response Parameters** module, you can output the following results: (i) structural displacements, (ii) forces and moments at the supports, (iii) hysteretic curves and (iv) tables for Code-based Checks.

In order to visualise the displacements, in the x direction, of a particular node at the top of the structure, (i) click on the Structural Displacements tab, (ii) select displacement and the x-axis, (iii) select the corresponding node from the list (-> Upper node of column C5 of Floor 2) by ticking the checkbox, (iv) choose the results to visualise (graph or values) and finally (v) click on the *Refresh* button.

NOTE: The results are defined in the global system of coordinates and may be exported in an Excel spreadsheet (or similar) as shown below.

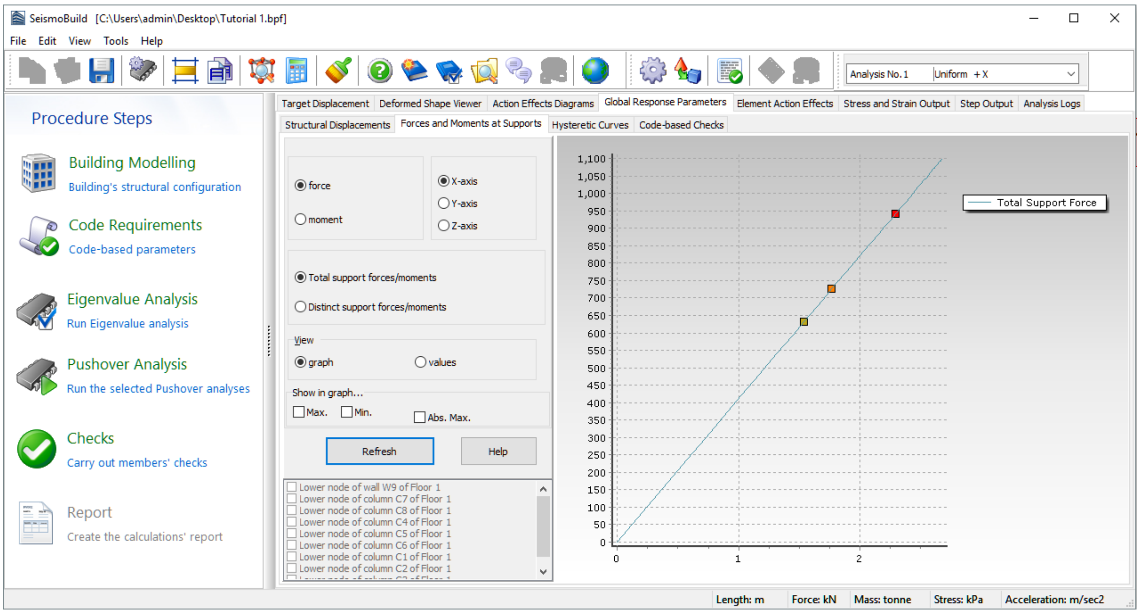


Global Response Parameters Module (Structural Displacements – graph mode)



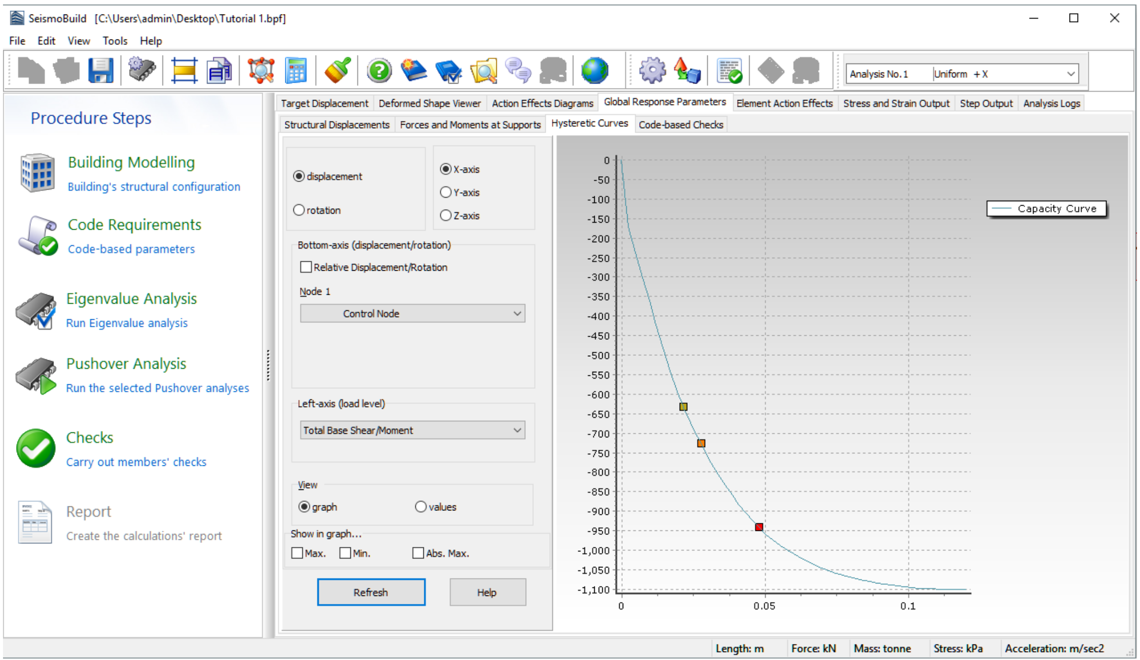
Global Response Parameters Module (Structural Displacements – values mode)

The total support forces (e.g. total base shear) may be obtained by (i) clicking on the Forces and Moments at Support tab, (ii) selecting force and x-axis and total support forces/moments, (iii) choosing the results to visualise (graph or values) and (iv) clicking on the *Refresh* button.



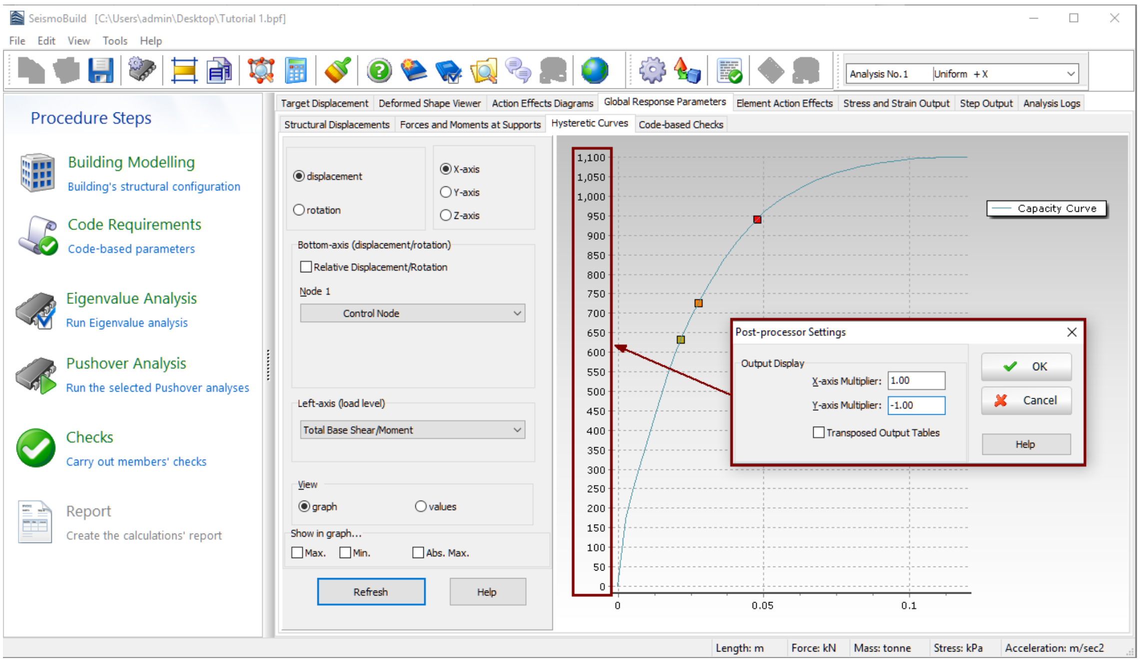
Global Response Parameters Module (Forces and Moments at Supports – graph mode)

Further, the capacity curve of your structure (i.e. total base shear vs. top displacement) can be plotted by (i) clicking on the Hysteretic Curves tab, (ii) selecting displacement and x-axis, (iii) selecting the corresponding node from the drop-down menu (e.g. Control Node), (iv) selecting the Total Base Shear/Moment option, (v) choosing the results to visualise (graph or values) and finally (vi) clicking on the *Refresh* button.



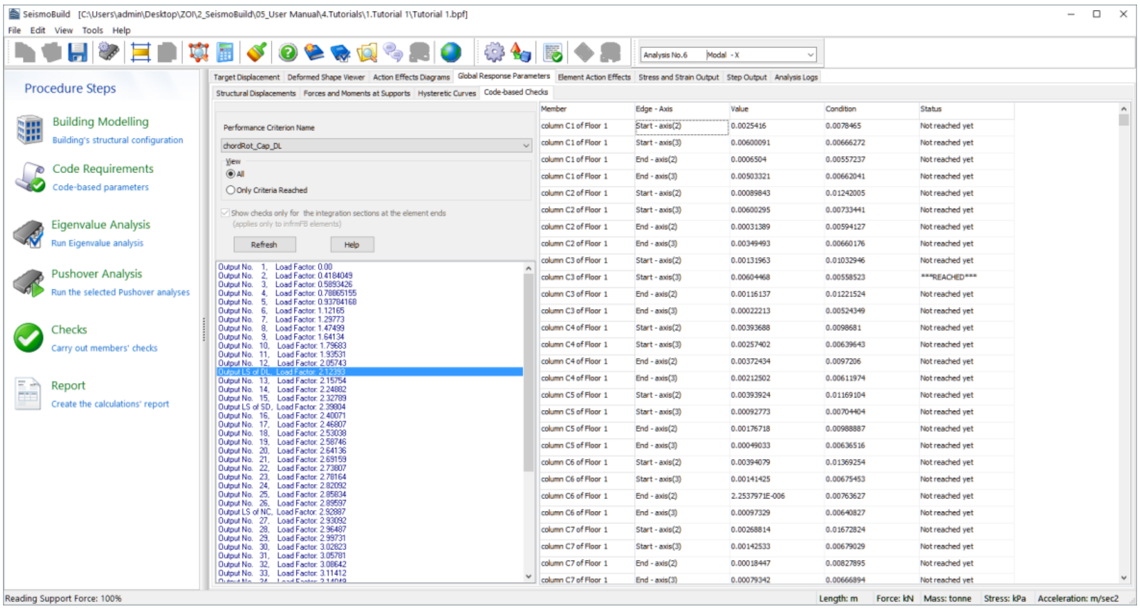
Global Response Parameters Module (Hysteretic Curves – graph mode)

In order to have the shear forces with a positive sign, (i) right-click on the 3D plot window, (ii) select Post-Processor Settings and (iii) insert the value "-1" as Y-axis multiplier.



Global Response Parameters Module (Hysteretic Curves – graph mode)

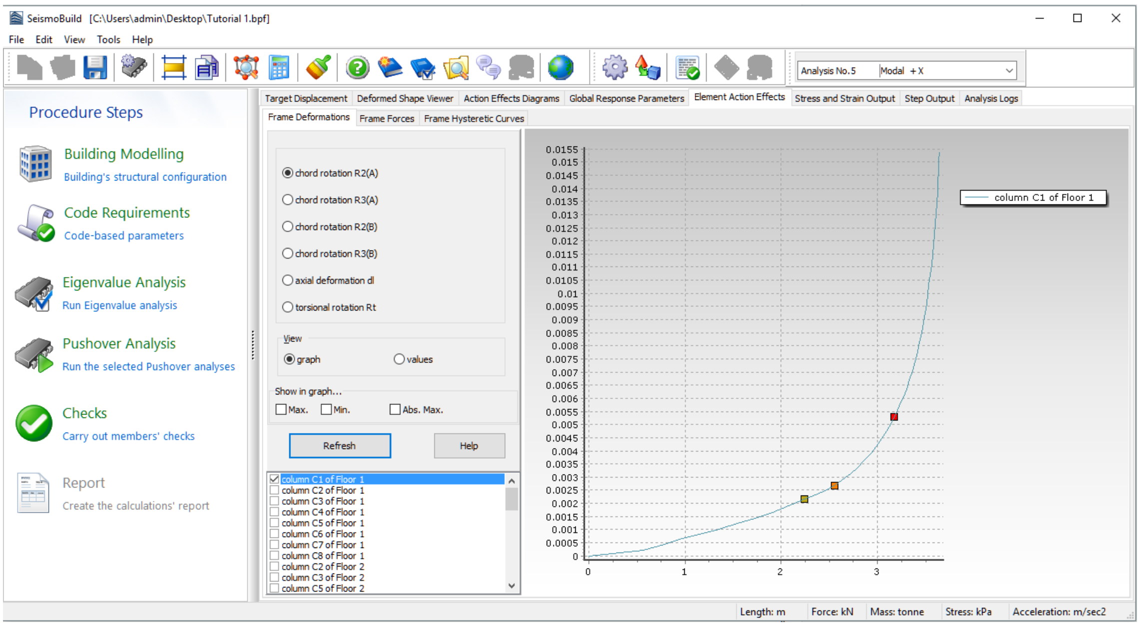
Fourth, in order to visualise the Code-based checks in every step of the analysis of your structure, (i) click on the Code-based Checks tab, (ii) select the chordRot_Cap_DL Code-based check, (iii) select to view All (i.e. even the calculations for the members that have not reached their capacity), (iv) select the Output steps that correspond to the Damage Limitation limit state (i.e. Output LS of DL) and finally (vi) click on the *Refresh* button.



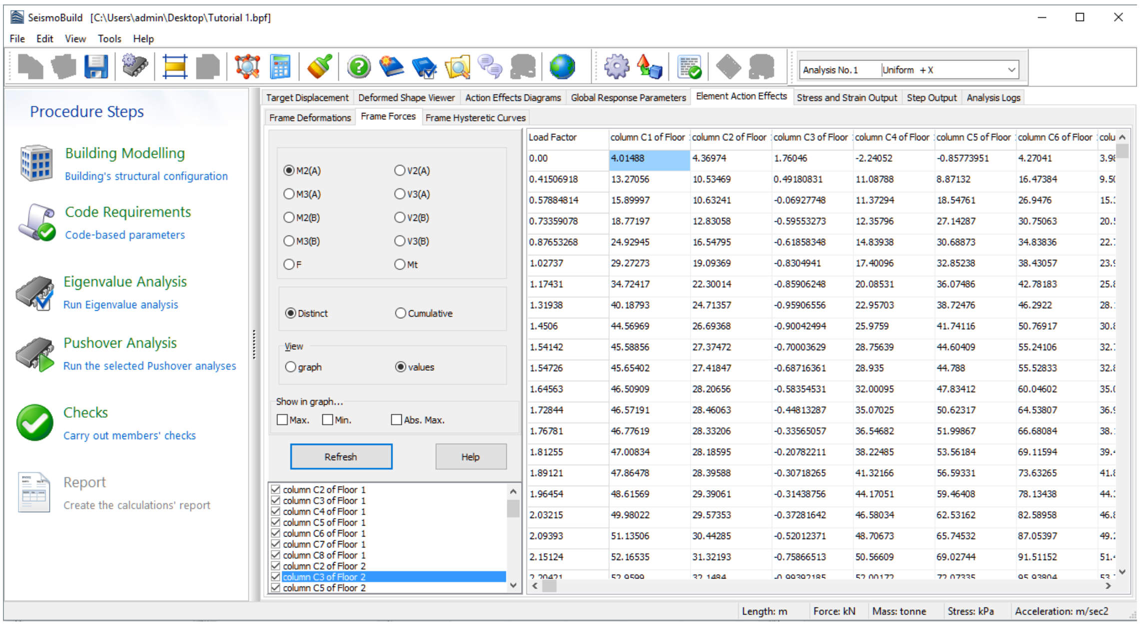
Global Response Parameters Module (Code-based Checks)

Show Results – Element Action Effects

In order to proceed with the seismic verifications prescribed in Eurocode 8 it is necessary to check the **element chord rotations** and **element shear forces.** For this reason, the Frame Deformations and the Frame Forces tab windows may be very useful. The element chord rotations can be directly output by (i) clicking on the Frame Deformations tab, (ii) selecting chord rotation in the direction you are interested in (i.e. R2), (iii) selecting the elements from the list, by ticking the corresponding box, (iv) choosing the results to visualise (graph or values) and finally (v) clicking on the *Refresh* button. The element shear forces can be output by (i) clicking on the Frame Forces tab, (ii) selecting shear in the direction and the section you are interested in (i.e. V2(A)), (iii) selecting the elements from the list, by ticking the corresponding box, (iv) choosing the results to visualise (graph or values) and finally (v) clicking on the *Refresh* button.



Element Action Effects Module (Frame Deformations – graph mode)



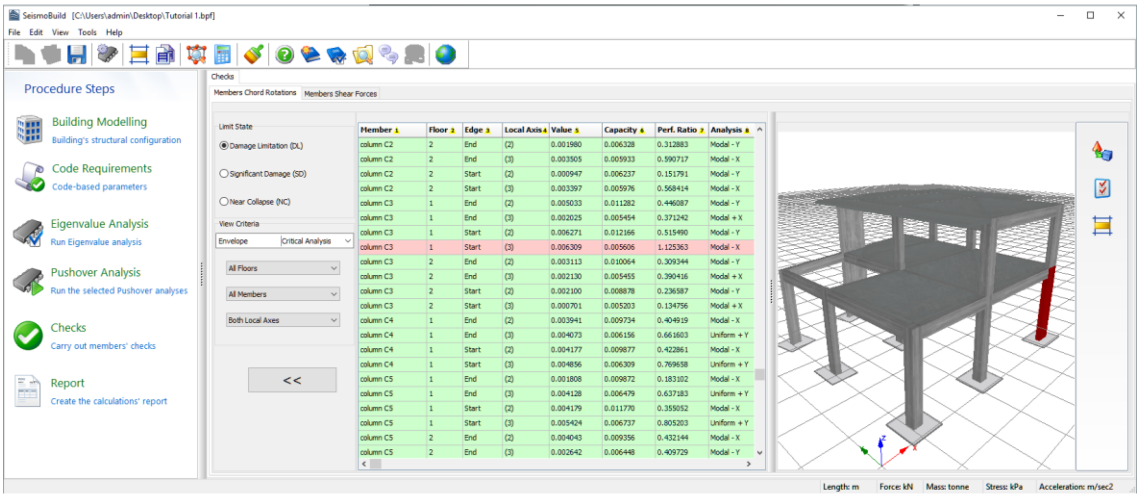
Element Action Effects Module (Frame Forces – values mode)

> **NOTE: The results may be easily exported in an Excel spreadsheet (or similar).**

Checks

SeismoBuild provides the option to automatically undertake chord-rotation and shear checks for structural elements, as well as the necessary beam-column joints checks (shear forces, horizontal hoops area and vertical reinforcement area), according to the expressions defined in the selected Code, and for the selected limit states. This can be visualised in the Checks module of the program's Main Window.

The **Checks** area features a series of pages where the results of the structural members' checks can be visualised, in table and graphical format, and then copied into any other Windows application. Users may select the limit state, as well as the analysis, the floor, the type of members and the local axis to view the results. The elements, where the demand has exceeded the capacity, are displayed in red both in the table and the 3D plot, as it is depicted in the figure below:

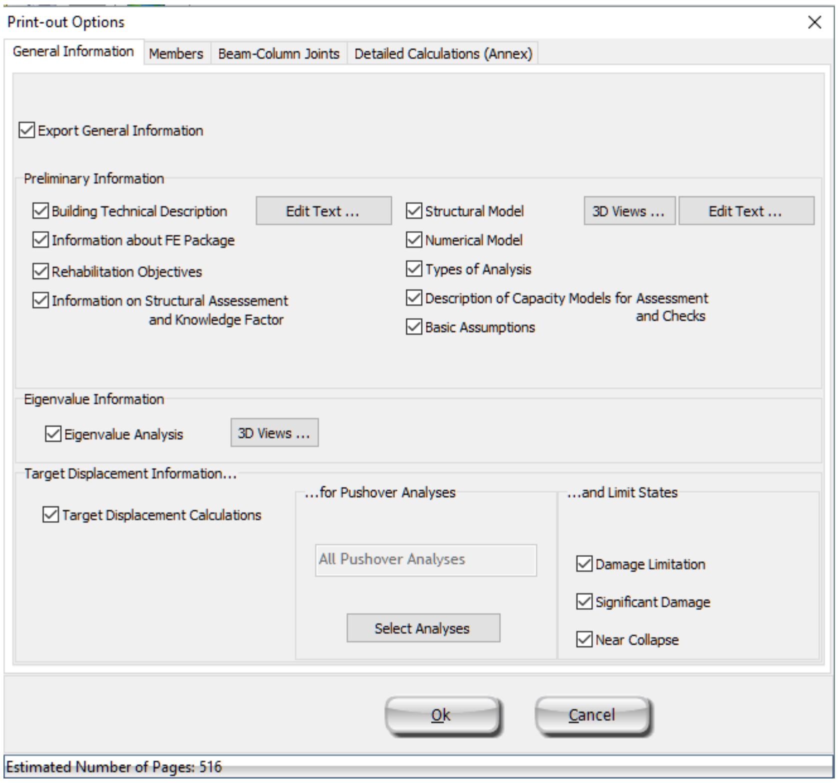


Checks Module (Members Chord Rotations)

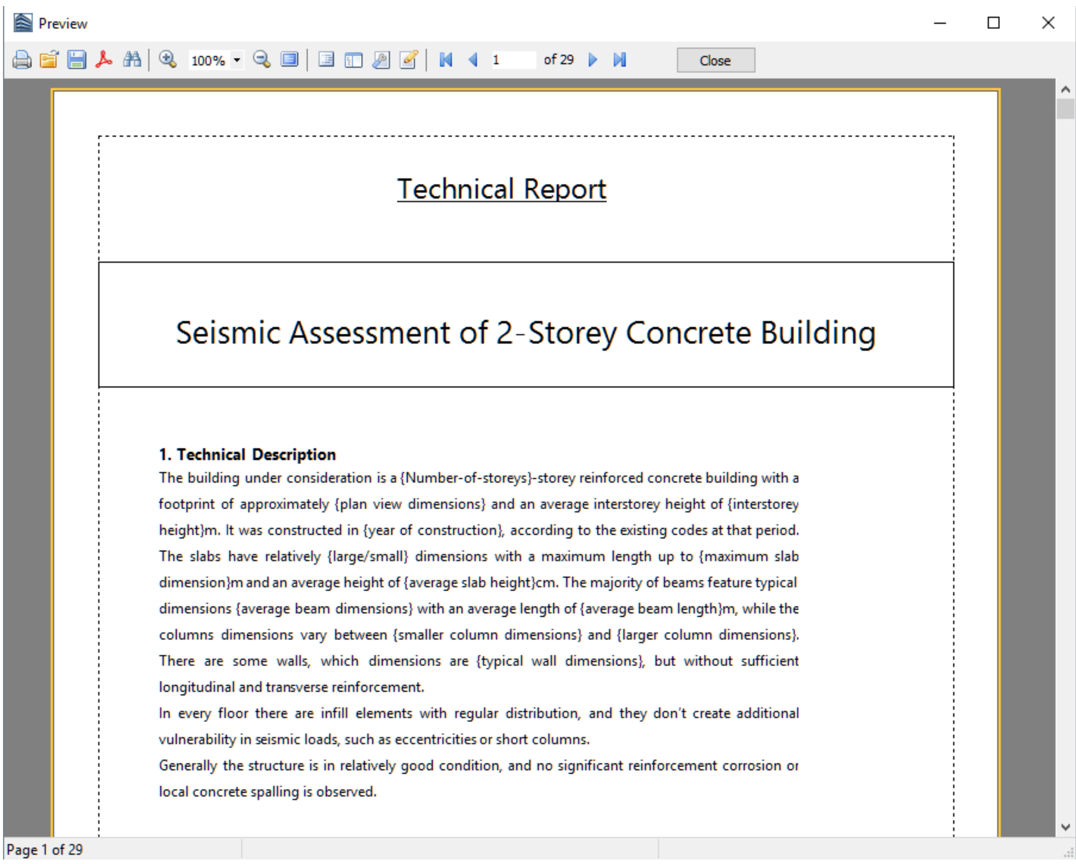
Report

After running the analyses and finishing the checks process, you may create the technical report of the assessment. Once you click on the *Report* button a window will appear in order to define the print output options. Click the *OK* button and the report will automatically be created and shown on screen. The report may be exported in PDF, RTF or HTML file formats, the two latter being editable.

> **NOTE: Creating a report for a typical 4 or 5-storey building may take up to 4-5 minute to complete.**



Print-out Options (General information)



Technical Report

Seismic Assessment of 2-Storey Concrete Building

1. Technical Description

The building under consideration is a {Number-of-storeys}-storey reinforced concrete building with a footprint of approximately {plan view dimensions} and an average interstorey height of {interstorey height}m. It was constructed in {year of construction}, according to the existing codes at that period. The slabs have relatively {large/small} dimensions with a maximum length up to {maximum slab dimension}m and an average height of {average slab height}cm. The majority of beams feature typical dimensions {average beam dimensions} with an average length of {average beam length}m, while the columns dimensions vary between {smaller column dimensions} and {larger column dimensions}. There are some walls, which dimensions are {typical wall dimensions}, but without sufficient longitudinal and transverse reinforcement.

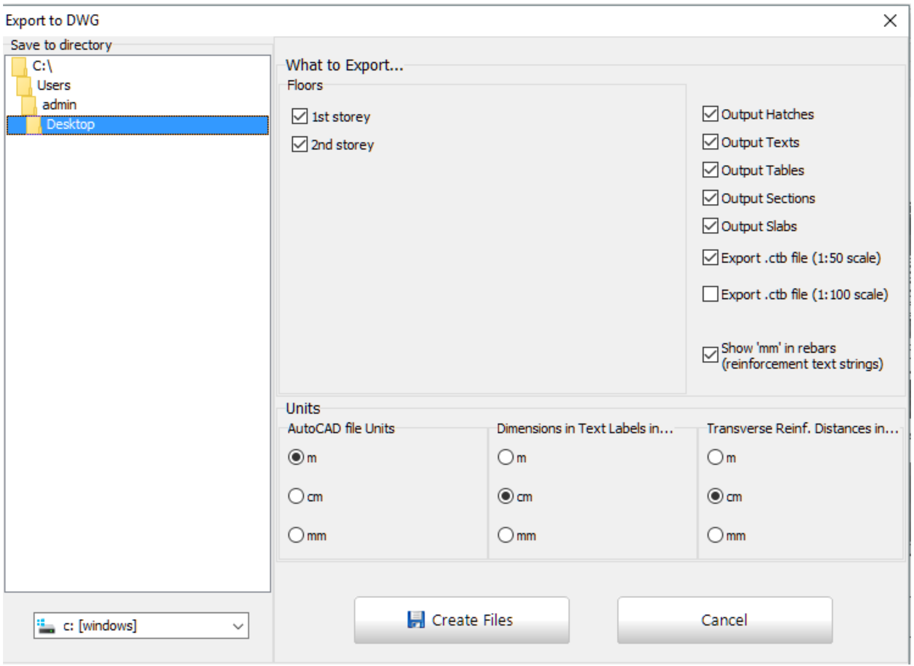
In every floor there are infill elements with regular distribution, and they don't create additional vulnerability in seismic loads, such as eccentricities or short columns.

Generally the structure is in relatively good condition, and no significant reinforcement corrosion or local concrete spalling is observed.

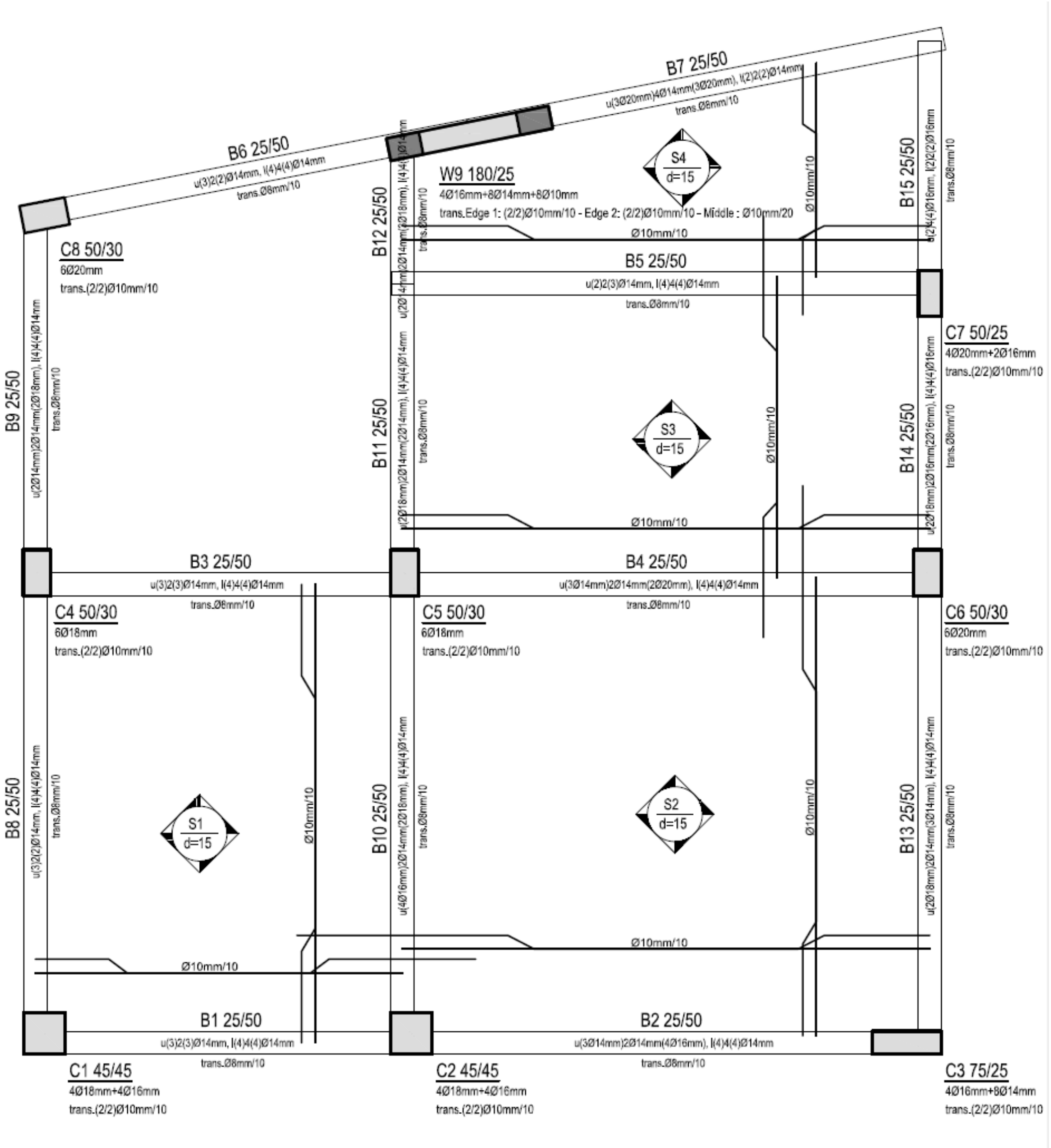
Technical Report

CAD Drawing

Finally, you may export a variety of CAD drawing files of the structural model (plan views, members' cross sections and reinforcement tables), together with specially created *.ctb files that are needed for plotting. It is noted that running the analyses is not a prerequisite for the exportation of the Cad drawing files, and only the introduction of the structural configuration in the Building Modeller is required.



Export to DWG



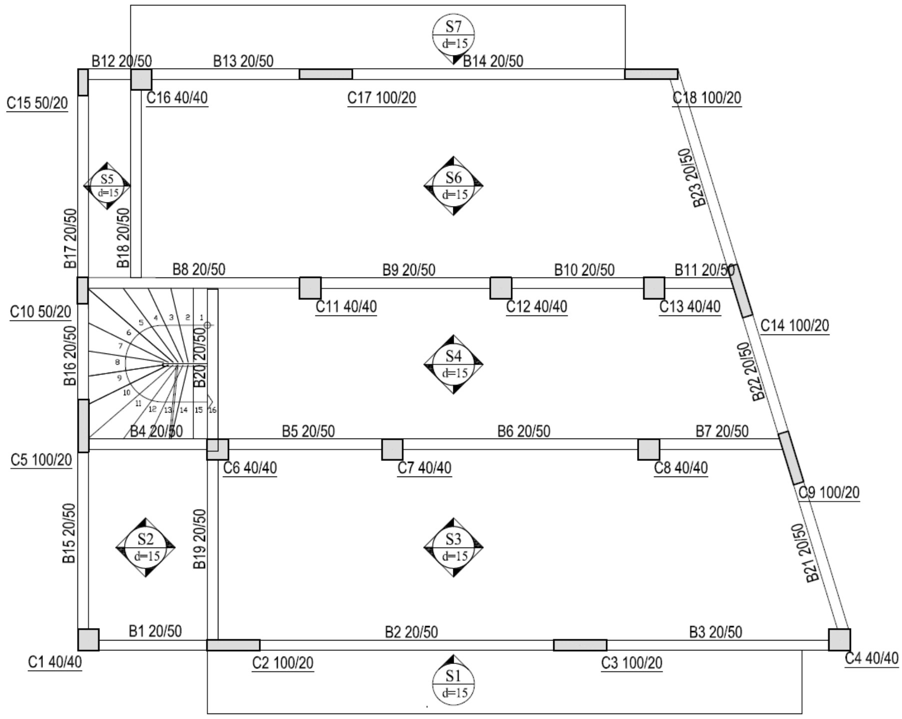
CAD drawing

Congratulation, you have finished your first tutorial!

TUTORIAL N.2 – ASSESSMENT OF A THREE-STOREY BUILDING

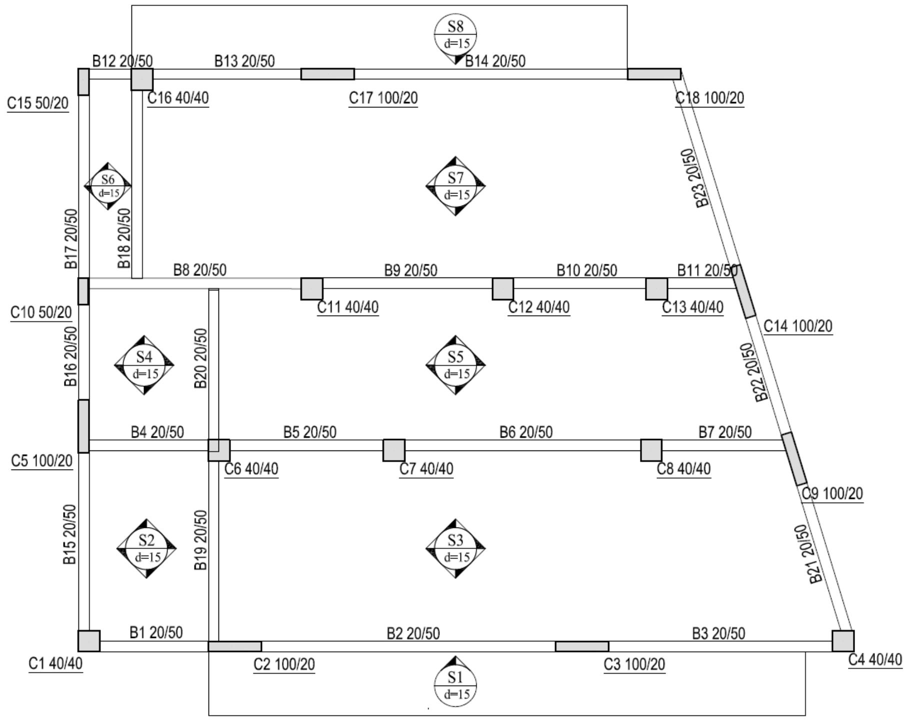
Problem Description

Let us try to model a three dimensional, three-storey reinforced concrete building for which you are asked to assess its capacity according to the Eurocodes. The geometry of all the floors is the same and is shown in the corresponding plan-views below, the only difference being the presence of inclined slabs on the third floor.

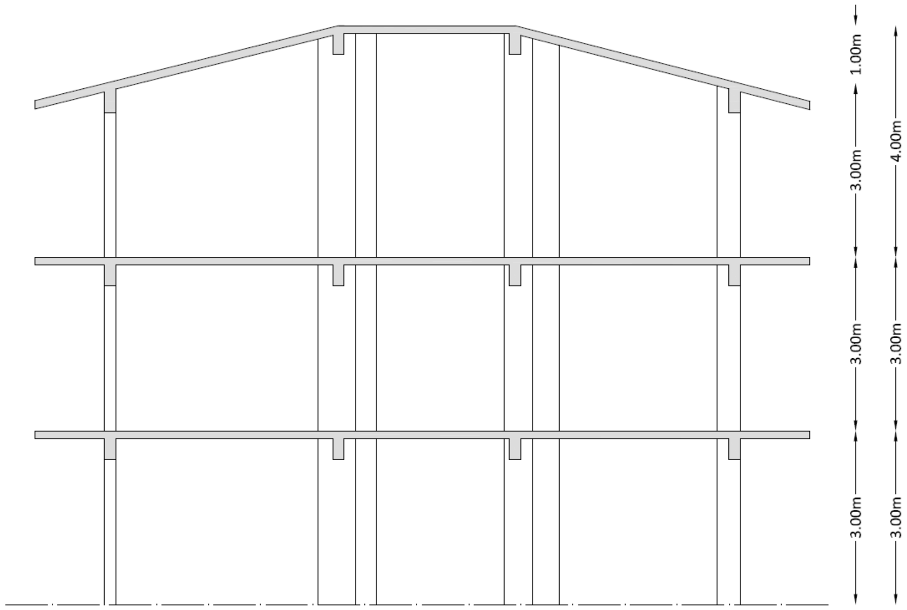


Plan view of the 1st and 2nd floor of the building

NOTE: A movie describing Tutorial N.2 can be found on Seismosoft's YouTube channel.



Plan view of the 3rd floor of the building



Section plan

Getting started: a new project

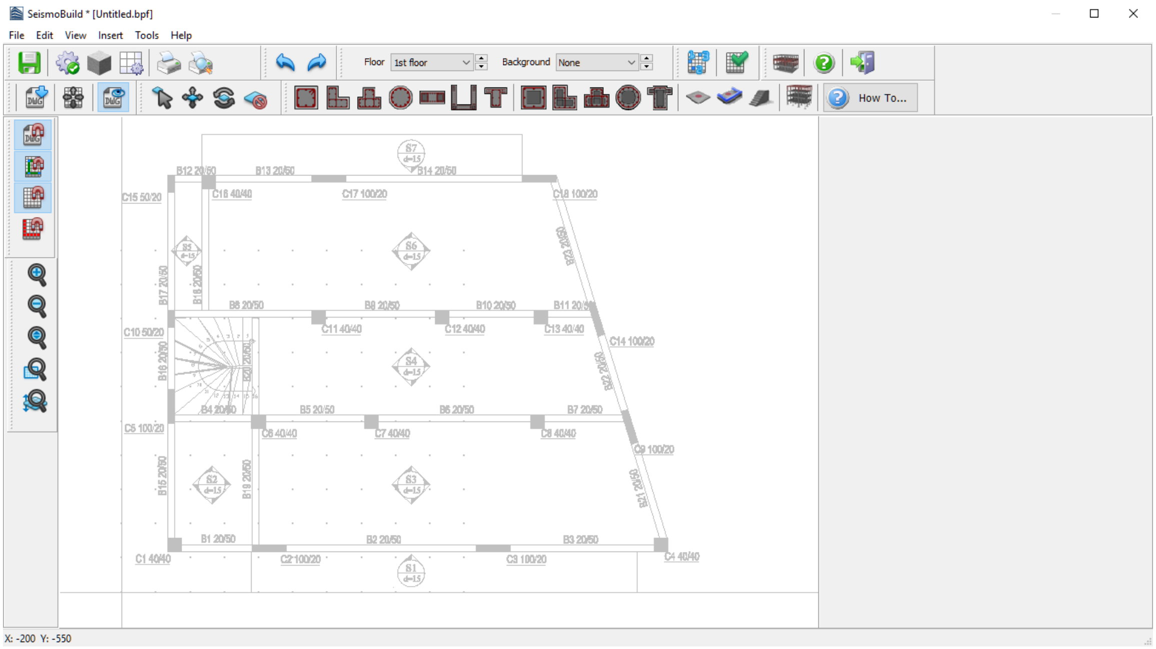
The introduction of structural members is the same with the Tutorial N.1, hence, in the current tutorial only the steps for the definition of the stairs and the inclined slabs will be described.

For this tutorial the following settings have been chosen:

- Eurocode 8, Part 3
- SI Units
- European sizes for rebar typology

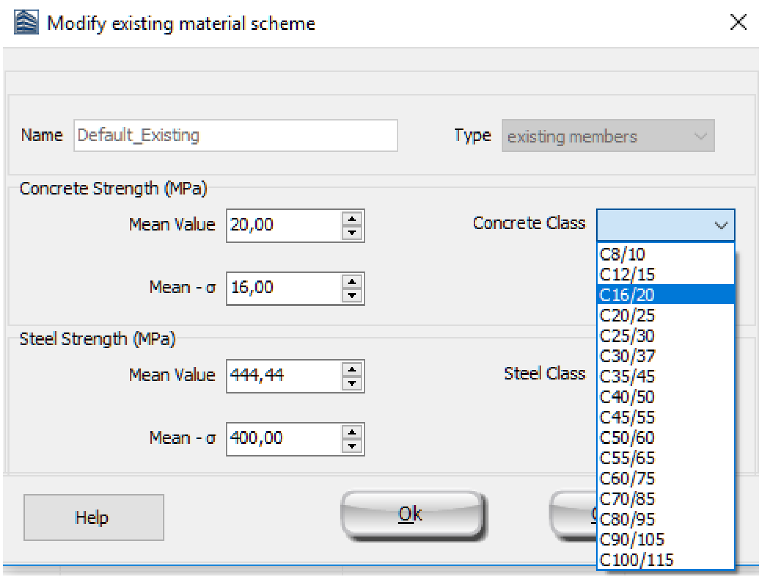
- 3 Storeys
- Storeys' heights: 3m
- Do not accept beams with free span less than: 0.1m
- Include beam effective widths

A CAD drawing is imported as a background to facilitate the definition of the elements' geometry.



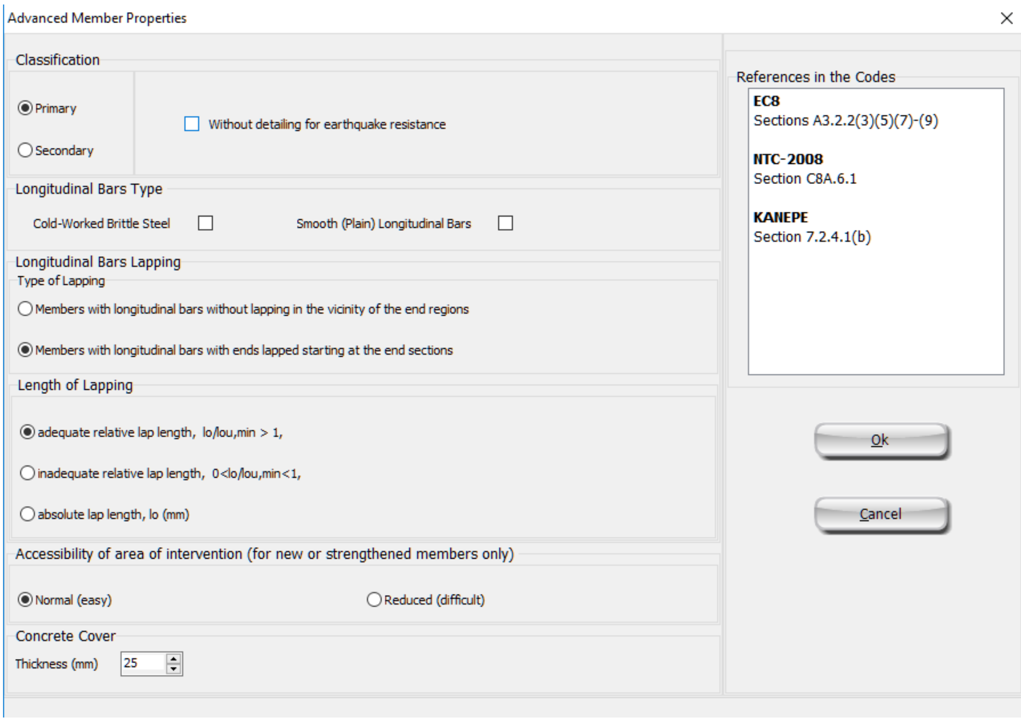
Building Modeller – CAD drawing insertion

In the material sets module the member's concrete and reinforcement strength values are determined. Herein the Default_Existing material set is selected and edited by assigning the C16/20 concrete class and the S400 steel class.



Building Modeller – Modify Existing Material Scheme

By clicking on the *Advanced Member Properties* button users may define the settings for the structural member according to the selected Code. The selected properties for the inserted members are shown in the figure below:



Building Modeller – Advanced Member Properties

The dimensions and the reinforcement of the members (columns and beams) of the typical floor are shown in the following tables:

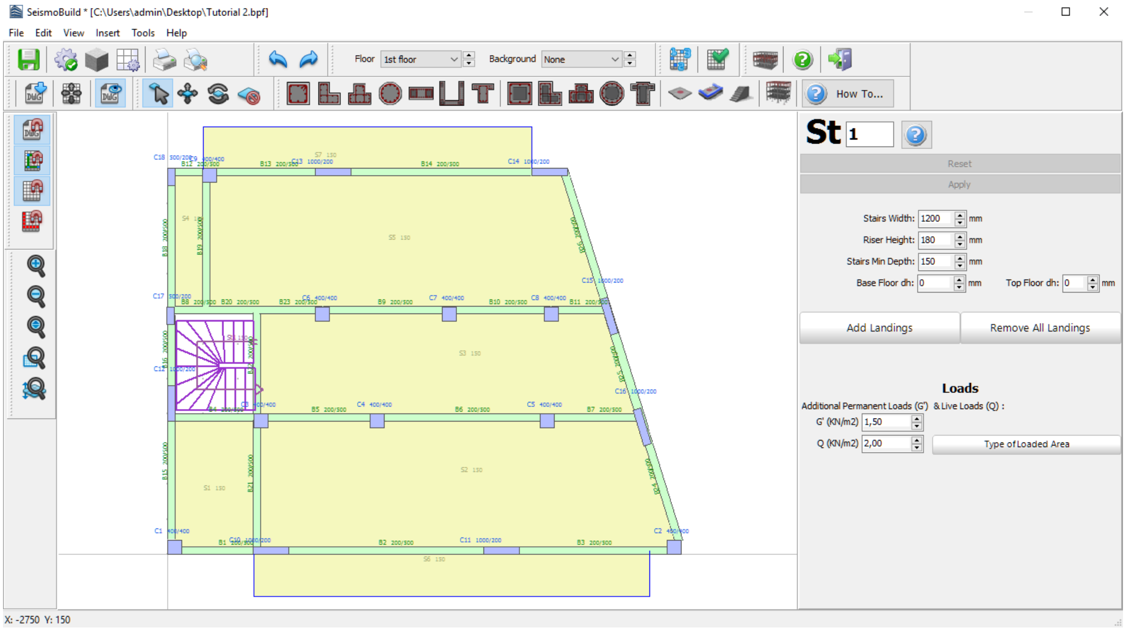
Columns	Height (mm)	Width (mm)	Longitudinal reinforcement	Transverse reinforcement
C1	400	400	4∅16	∅6/25
C2	1000	200	8∅14+8∅8	∅6/25
C3	1000	200	8∅14+8∅8	∅6/25
C4	400	400	4∅16	∅6/25
C5	200	1000	8∅14+8∅8	∅6/25
C6	400	400	4∅18	∅6/25
C7	400	400	4∅18	∅6/25
C8	400	400	4∅18	∅6/25
C9	200	1000	8∅14+8∅8	∅6/25

Columns	Height (mm)	Width (mm)	Longitudinal reinforcement	Transverse reinforcement
C10	200	500	6∅16	∅6/25
C11	400	400	4∅18	∅6/25
C12	400	400	4∅18	∅6/25
C13	400	400	4∅18	∅6/25
C14	200	1000	8∅14+8∅8	∅6/25
C15	200	500	6∅16	∅6/25
C16	400	400	4∅16	∅6/25
C17	1000	200	8∅14+8∅8	∅6/25
C18	1000	200	8∅14+8∅8	∅6/25

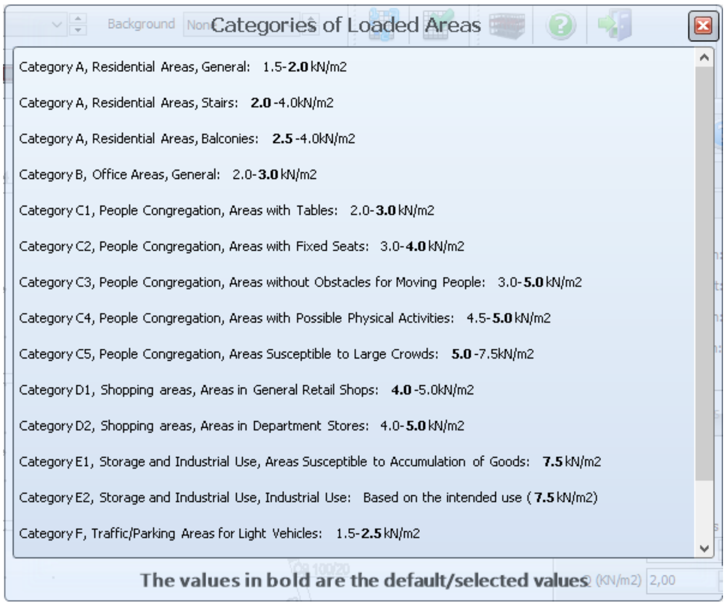
Beams	Height (mm)	Width (mm)	Reinforcement at the Start of the beam	Reinforcement at the Middle of the beam	Reinforcement at the End of the beam	Transverse reinforcement
B1	500	200	o3∅16 u2∅14	o2∅12 u4∅14	o3∅16 u2∅14	∅8/25
B2	500	200	o3∅16 u2∅14	o2∅12 u4∅14	o3∅16 u2∅14	∅8/25
B3	500	200	o3∅16 u2∅14	o2∅12 u4∅14	o3∅16 u2∅14	∅8/25
B4	500	200	o3∅16 u2∅14	o2∅12 u4∅14	o3∅16 u2∅14	∅8/25
B5	500	200	o3∅16 u2∅14	o2∅12 u4∅14	o3∅16 u2∅14	∅8/25
B6	500	200	o3∅16 u2∅14	o2∅12 u4∅14	o3∅16 u2∅14	∅8/25
B7	500	200	o3∅16 u2∅14	o2∅12 u4∅14	o3∅16 u2∅14	∅8/25
B8	500	200	o3∅16 u2∅14	o2∅12 u4∅14	o3∅16 u2∅14	∅8/25
B9	500	200	o3∅16 u2∅14	o2∅12 u4∅14	o3∅16 u2∅14	∅8/25
B10	500	200	o3∅16 u2∅14	o2∅12 u4∅14	o3∅16 u2∅14	∅8/25
B11	500	200	o3∅16 u2∅14	o2∅12 u4∅14	o3∅16 u2∅14	∅8/25

Beams	Height (mm)	Width (mm)	Reinforcement at the Start of the beam	Reinforcement at the Middle of the beam	Reinforcement at the End of the beam	Transverse reinforcement
B12	500	200	o3∅16 u2∅14	o2∅12 u4∅14	o3∅16 u2∅14	∅8/25
B13	500	200	o3∅16 u2∅14	o2∅12 u4∅14	o3∅16 u2∅14	∅8/25
B14	500	200	o3∅16 u2∅14	o2∅12 u4∅14	o3∅16 u2∅14	∅8/25
B15	500	200	o3∅16 u2∅14	o2∅12 u4∅14	o3∅16 u2∅14	∅8/25
B16	500	200	o3∅16 u2∅14	o2∅12 u4∅14	o3∅16 u2∅14	∅8/25

After inserting all the columns and beams you may assign the stairs from the main menu (*Insert > Insert Stairs*) or through the toolbar button. This can be easily done by specifying the centreline and some basic geometric parameters, such us the stairs width, the riser height, the stairs minimum depth, and the elevation differences relatively to the base floor and the top floor, as well as the additional permanent and live loads.

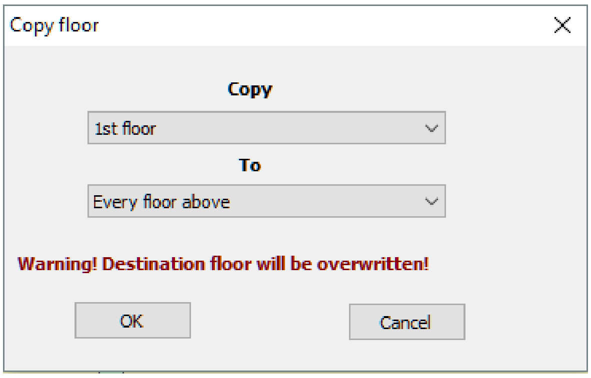


Building Modeller – Stairs Properties



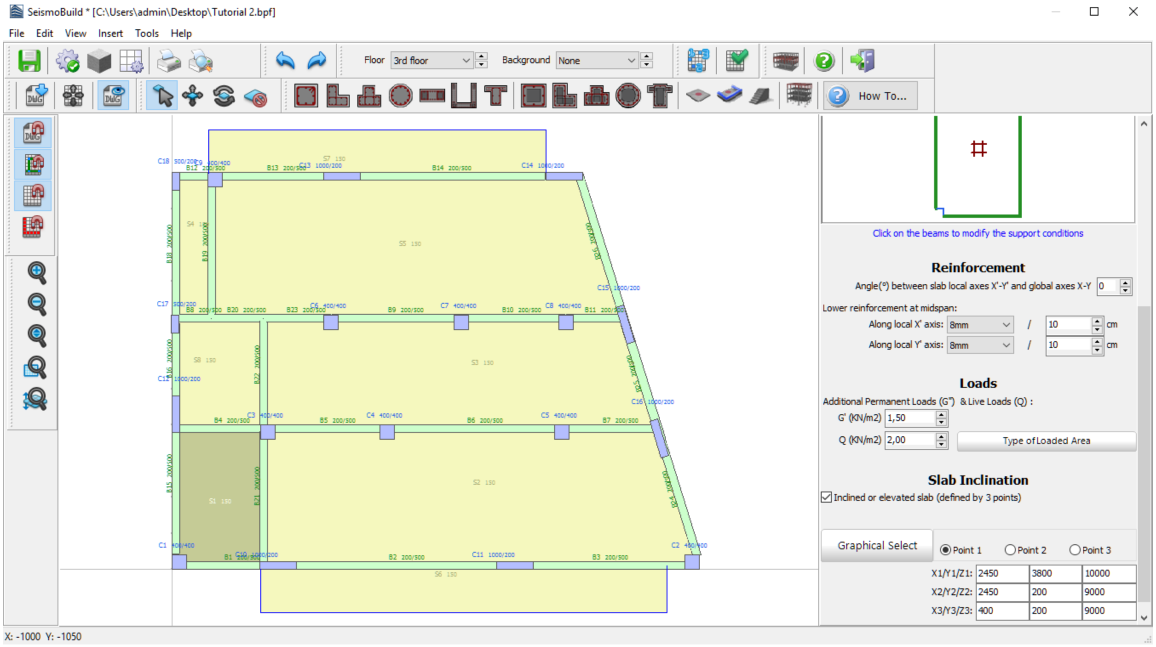
Type of Loaded Area

After inserting all the members of the 1st floor, you may automatically create the 2nd and 3rd floors based on the already created 1st one by using the Copy Floor facility.



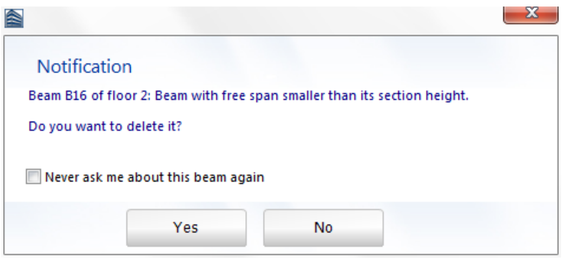
Building Modeller – Copy Floor dialogue box

Delete the elements (e.g. the stairs) that are not present in the 3rd floor and define the inclined slabs. Select the slab that will be modified, click on the "Inclined or elevated slab (defined by 3points)" check-box in the slab's properties window, define graphically the coordinates of the 3 points and assign their elevation.



Building Modeller – Slab Element Properties

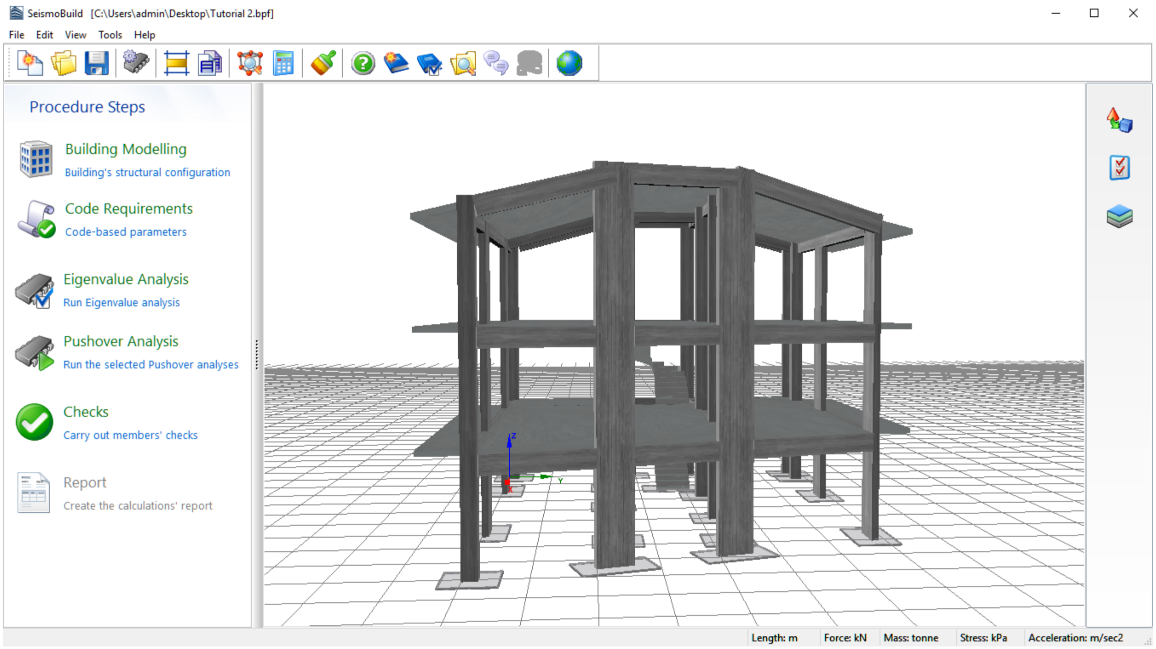
When you create a building model, it is relatively common that one or more very short beams have been created unintentionally, due to graphical reasons (e.g. by extending slightly a beam's end beyond a column edge). For this reason, a check from the main menu (*Tools > Verify Connectivity...*) or through the toolbar button for the existence of any beam with free span smaller than its section height should be carried out. If such beams exist, the following message appears, and the user can select to remove or keep the member.



Building Modeller – Verify Connectivity

With the building model now fully defined, save the project as a SeismoBuild file (with the *.bpf extension, e.g. Tutorial_2.bpf) from the main menu (*File >Save As...*)/(*File >Save*) or through the corresponding toolbar button .

You are ready to go to the SeismoBuild Main Window. This can be done from the main menu (*File > Exit * Create 3D Model*) or through the toolbar button.

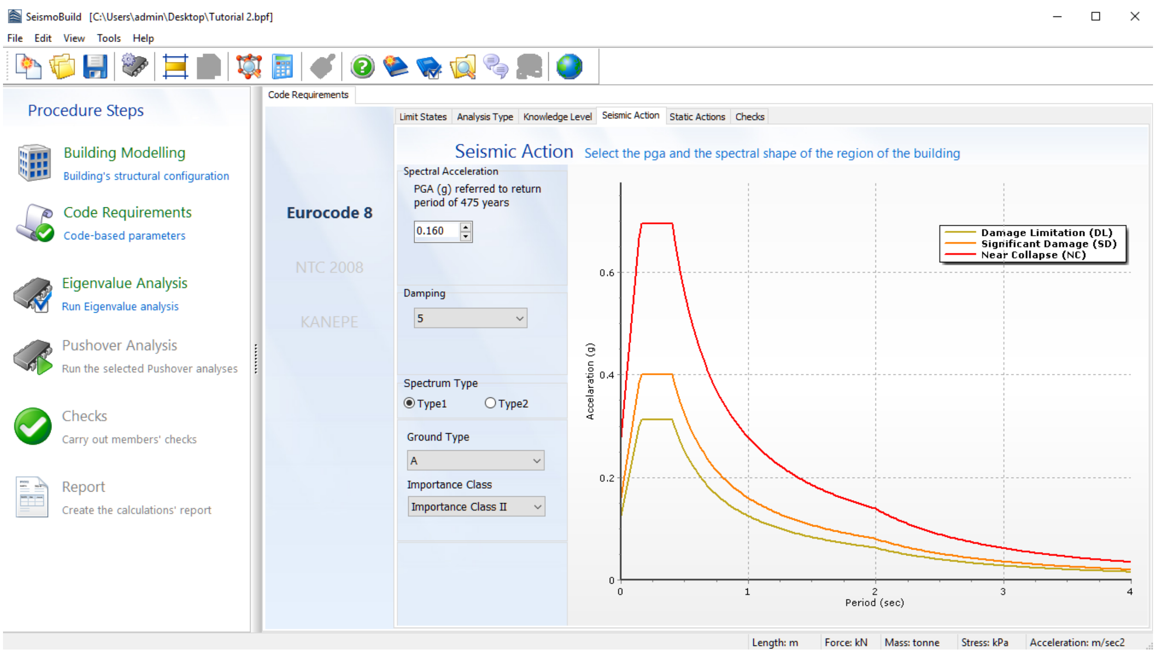


SeismoBuild Main Window

Code Requirements

The Code-based parameters and options are defined as in Tutorial N.1, apart from the seismic action where:

- A peak ground acceleration equal to 0.16g is specified; this acceleration is referred to a return period of 475 years, 5% damping, response spectra Type 1, ground type A and Importance class II;

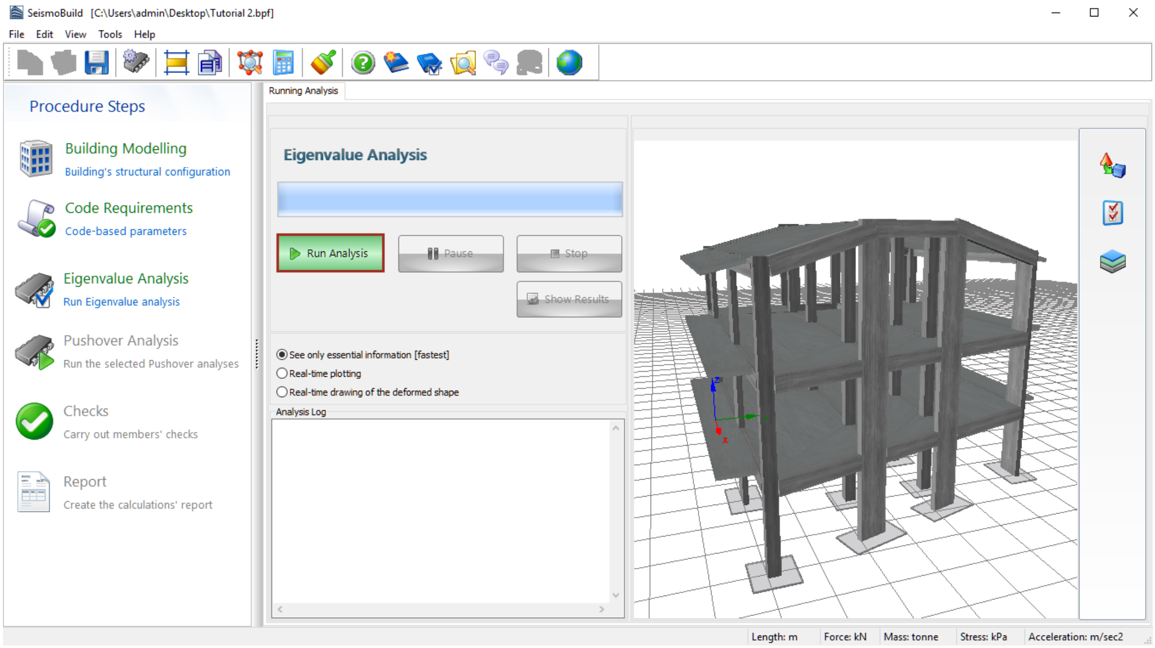


Analysis & Modelling Parameters

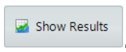
The default predefined settings scheme is employed for the scope of this tutorial.

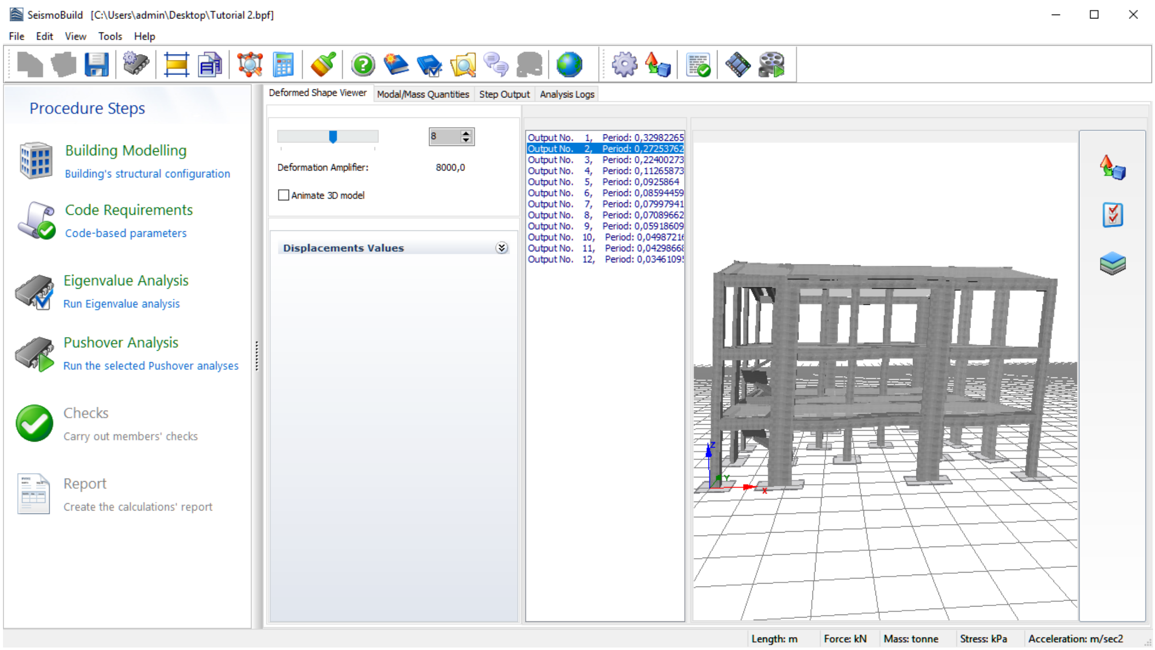
Eigenvalue Analysis

Run the eigenvalue analysis through this module.



Eigenvalue Analysis

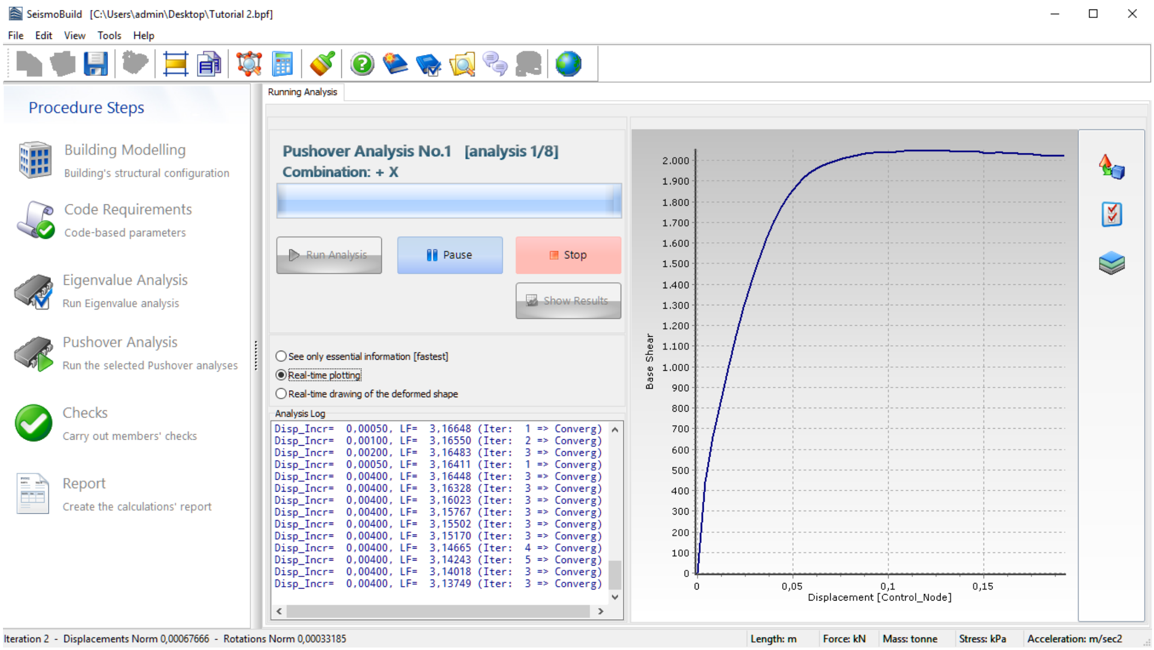
You may see the results after running the analysis by clicking on the *Show Results* button .



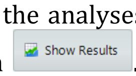
Eigenvalue Analysis Results

Pushover Analysis

Click on the *Run* button to run all the selected pushover analyses.



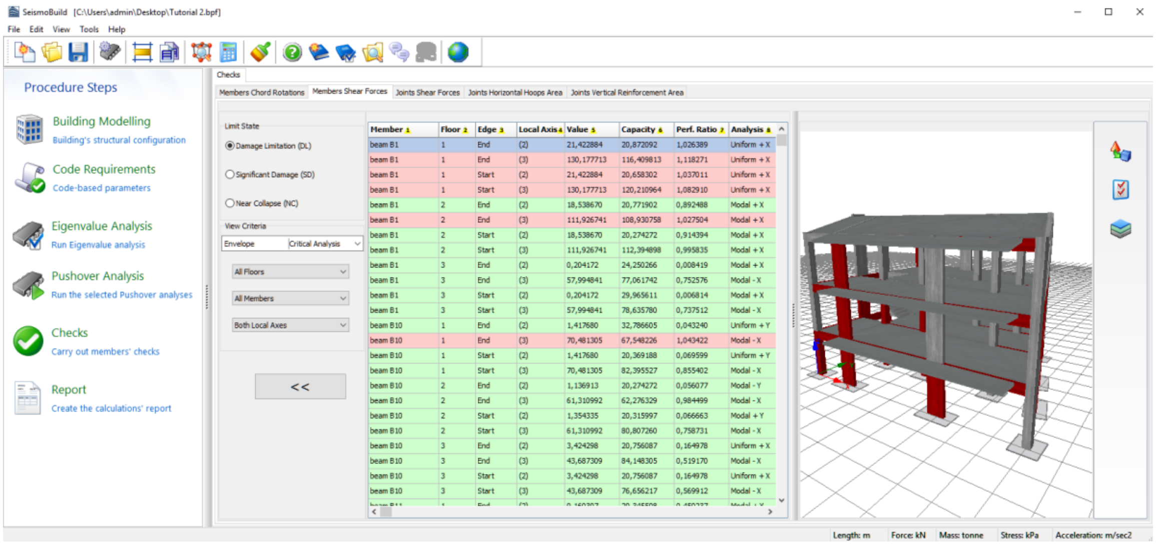
Running the analysis

When the analyses have arrived to the end, you may see the results by clicking on the Show Results button 

. The available modules have been discussed in Tutorial N1.

Checks

The results of the structural members' checks can be visualised in the **Checks** area, in table or graphical format, and then copied into any other Windows application. Users may select the limit state, as well as the analysis, the floor, the type of members and the local axis to view the results. The elements, where the demand has exceeded the capacity, are displayed in red both in the table and the 3D plot, as it is depicted in the figure below:

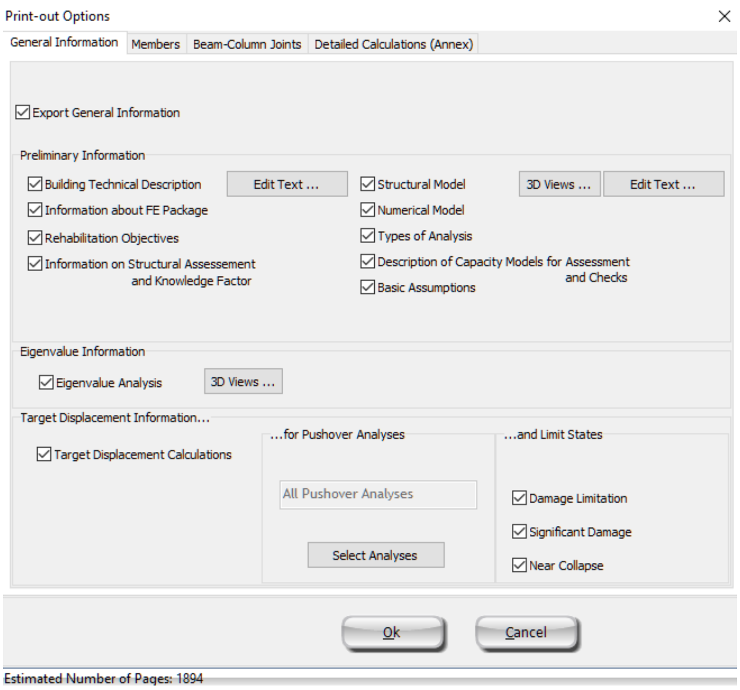


Checks Module (Member Shear Forces)

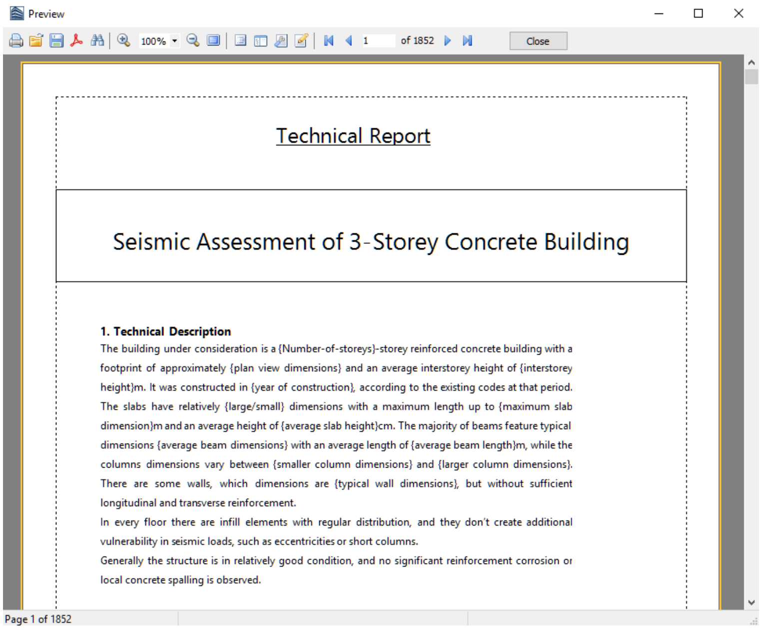
Report

After running the analyses and finishing the checks process, you may create the technical report of the assessment. Once you click on the *Report* button a window will appear in order to define the print output options. Click the *OK* button and the report will automatically be created and shown on screen. The report may be exported in PDF, RTF or HTML file formats, the two latter being editable.

NOTE: Creating a report for a typical 4 or 5-storey building may take up to 4-5 minute to complete.



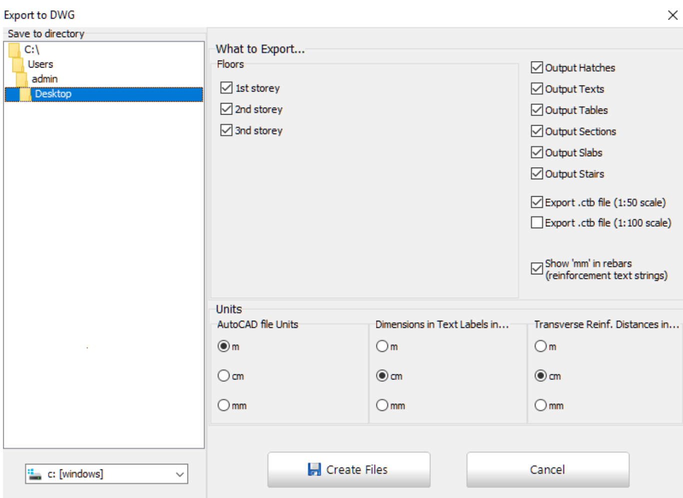
Print-out Options (General information)



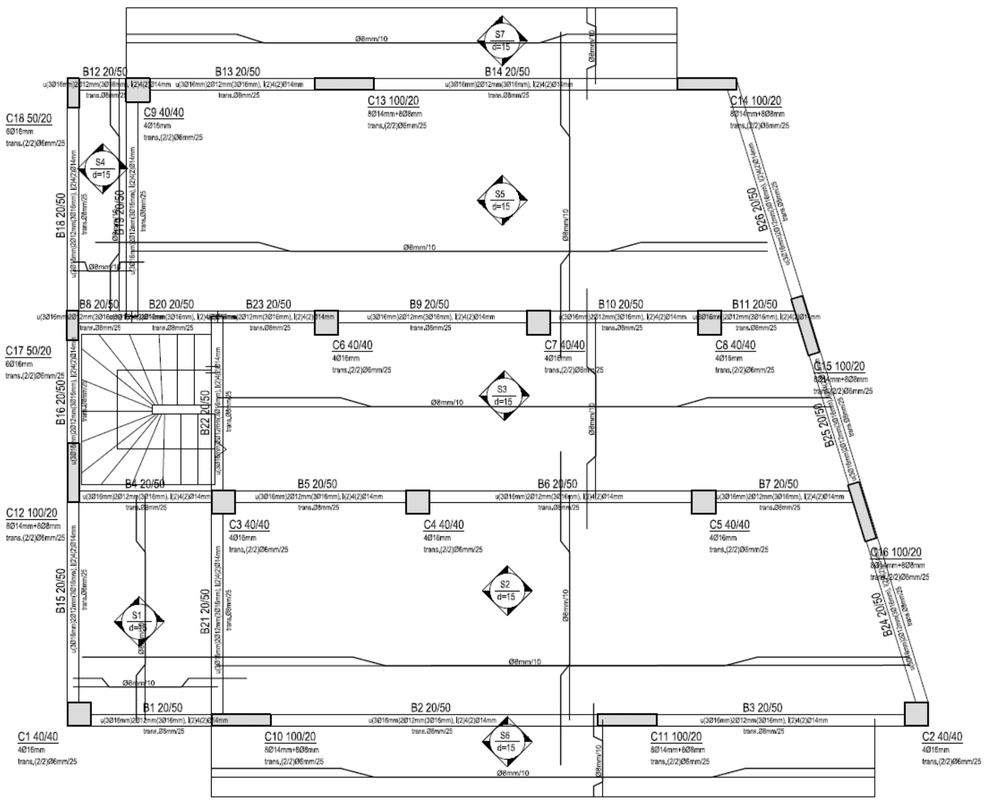
Technical Report

CAD Drawing

Finally, you may export a variety of CAD drawing files of the structural model (plan views, members' cross sections and reinforcement tables), together with specially created *.ctb files that are needed for plotting. It is noted that running the analyses is not a prerequisite for the exportation of the Cad drawing files, and only the introduction of the structural configuration in the Building Modeller is required.



Export to DWG



CAD drawing

TUTORIAL N.3 – REHABILITATION OF A THREE-STOREY BUILDING

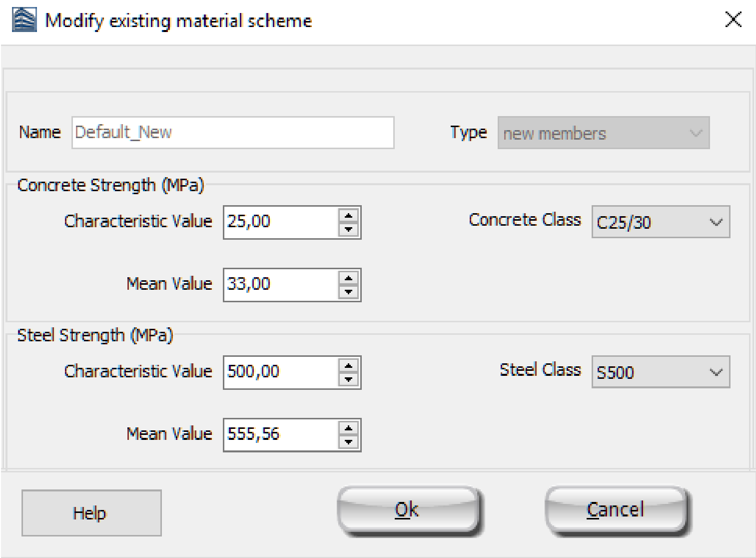
Problem Description

In this third tutorial the model that has already been created in Tutorial N2 will be strengthen with RC jackets. The columns of all the floors will be strengthened, as well as the beams of the first and second floor.

Getting started: opening an existing project

Open again the initial window of the software and, after clicking on icon on the toolbar, select the previous SeismoBuild project (Tutorial_2.bpf). Once opened, save the project with a new name through *File > Save as…* menu command.

In the material sets module the member's concrete and reinforcement strength values are determined. Herein the Default_New material set is selected and edited by assigning the C20/25 concrete class and the S500 steel class.



Building Modeller – Modify Existing Material Scheme

The dimensions and the reinforcement of the jacketed columns of the first floor are shown in the following table:

Columns	Height (mm)	Width (mm)	Internal Longitudinal reinforcement	Internal Transverse reinforcement	External Longitudinal reinforcement	External Transverse reinforcement
C1	600	600	4∅16	∅6/25	12∅20	∅10/10
C2	1200	400	8∅14+8∅8	∅6/25	8∅20+12∅16	∅10/10
C3	1200	400	8∅14+8∅8	∅6/25	8∅20+12∅16	∅10/10
C4	600	600	4∅16	∅6/25	12∅20	∅10/10
C5	400	1200	8∅14+8∅8	∅6/25	26∅20	∅12/10
C6	600	600	4∅18	∅6/25	12∅20	∅10/10
C7	600	600	4∅18	∅6/25	12∅20	∅10/10
C8	600	600	4∅18	∅6/25	12∅20	∅10/10
C9	400	1200	8∅14+8∅8	∅6/25	8∅20+12∅16	∅10/10
C10	400	700	6∅16	∅6/25	10∅20	∅10/10
C11	600	600	4∅18	∅6/25	12∅20	∅10/10

Columns	Height (mm)	Width (mm)	Internal Longitudinal reinforcement	Internal Transverse reinforcement	External Longitudinal reinforcement	External Transverse reinforcement
C12	600	600	4∅18	∅6/25	12∅20	∅10/10
C13	600	600	4∅18	∅6/25	12∅20	∅10/10
C14	400	1200	8∅14+8∅8	∅6/25	8∅20+12∅16	∅10/10
C15	400	700	6∅16	∅6/25	2∅22+8∅20	∅10/10
C16	600	600	4∅16	∅6/25	12∅20	∅10/10
C17	1200	400	8∅14+8∅8	∅6/25	8∅20+12∅16	∅10/10
C18	1200	400	8∅14+8∅8	∅6/25	8∅20+12∅16	∅10/10

The dimensions and the reinforcement of the new/external section of the jacketed beams of the first floor are shown in the following table. It is noted that the reinforcement of the old/internal section of the jacketed beams is the same with that of Tutorial N.2.

Beams	Height (mm)	Width (mm)	External Reinforcement at the Start of the beam	External Reinforcement at the Middle of the beam	External Reinforcement at the End of the beam	External Transverse reinforcement
B1	650	350	o5∅18 u3∅14 s4∅12	o2∅14 u5∅14 s4∅12	o5∅18 u3∅14 s4∅12	∅10/10
B2	650	350	o5∅18 u3∅14 s4∅12	o2∅14 u5∅14 s4∅12	o5∅18 u3∅14 s4∅12	∅10/10
B3	650	350	o5∅18 u3∅14 s4∅12	o2∅14 u5∅14 s4∅12	o5∅18 u3∅14 s4∅12	∅10/10
B4	650	350	o5∅18 u3∅14 s4∅12	o2∅14 u5∅14 s4∅12	o5∅18 u3∅14 s4∅12	∅10/10
B5	650	350	o5∅18 u3∅14 s4∅12	o2∅14 u5∅14 s4∅12	o5∅18 u3∅14 s4∅12	∅10/10
B6	650	350	o5∅18 u3∅14 s4∅12	o2∅14 u5∅14 s4∅12	o5∅18 u3∅14 s4∅12	∅10/10
B7	650	350	o5∅18 u3∅14 s4∅12	o2∅14 u5∅14 s4∅12	o5∅18 u3∅14 s4∅12	∅10/10
B8	650	350	o5∅18 u3∅14 s4∅12	o2∅14 u5∅14 s4∅12	o5∅18 u3∅14 s4∅12	∅10/10

Beams	Height (mm)	Width (mm)	External Reinforcement at the Start of the beam	External Reinforcement at the Middle of the beam	External Reinforcement at the End of the beam	External Transverse reinforcement
B9	650	350	o5∅18 u3∅14 s4∅12	o2∅14 u5∅14 s4∅12	o5∅18 u3∅14 s4∅12	∅10/10
B10	650	350	o5∅18 u3∅14 s4∅12	o2∅14 u5∅14 s4∅12	o5∅18 u3∅14 s4∅12	∅10/10
B11	650	350	o5∅18 u3∅14 s4∅12	o2∅14 u5∅14 s4∅12	o5∅18 u3∅14 s4∅12	∅10/10
B12	650	350	o5∅18 u3∅14 s4∅12	o2∅14 u5∅14 s4∅12	o5∅18 u3∅14 s4∅12	∅10/10
B13	650	350	o5∅18 u3∅14 s4∅12	o2∅14 u5∅14 s4∅12	o5∅18 u3∅14 s4∅12	∅10/10
B14	650	350	o5∅18 u3∅14 s4∅12	o2∅14 u5∅14 s4∅12	o5∅18 u3∅14 s4∅12	∅10/10
B15	650	350	o5∅18 u3∅14 s4∅12	o2∅14 u5∅14 s4∅12	o5∅18 u3∅14 s4∅12	∅10/10
B16	650	350	o5∅18 u3∅14 s4∅12	o2∅14 u5∅14 s4∅12	o5∅18 u3∅14 s4∅12	∅10/10
B17	650	350	o5∅18 u3∅14 s4∅12	o2∅14 u5∅14 s4∅12	o5∅18 u3∅14 s4∅12	∅10/10
B18	650	350	o5∅18 u3∅14 s4∅12	o2∅14 u5∅14 s4∅12	o5∅18 u3∅14 s4∅12	∅10/10
B19	650	350	o5∅18 u3∅14 s4∅12	o2∅14 u5∅14 s4∅12	o5∅18 u3∅14 s4∅12	∅10/10
B20	650	350	o5∅18 u3∅14 s4∅12	o2∅14 u5∅14 s4∅12	o5∅18 u3∅14 s4∅12	∅10/10
B21	650	350	o5∅18 u3∅14 s4∅12	o2∅14 u5∅14 s4∅12	o5∅18 u3∅14 s4∅12	∅10/10
B22	650	350	o5∅18 u3∅14 s4∅12	o2∅14 u5∅14 s4∅12	o5∅18 u3∅14 s4∅12	∅10/10

Beams	Height (mm)	Width (mm)	External Reinforcement at the Start of the beam	External Reinforcement at the Middle of the beam	External Reinforcement at the End of the beam	External Transverse reinforcement
B23	650	350	o5∅18 u3∅14 s4∅12	o2∅14 u5∅14 s4∅12	o5∅18 u3∅14 s4∅12	∅10/10

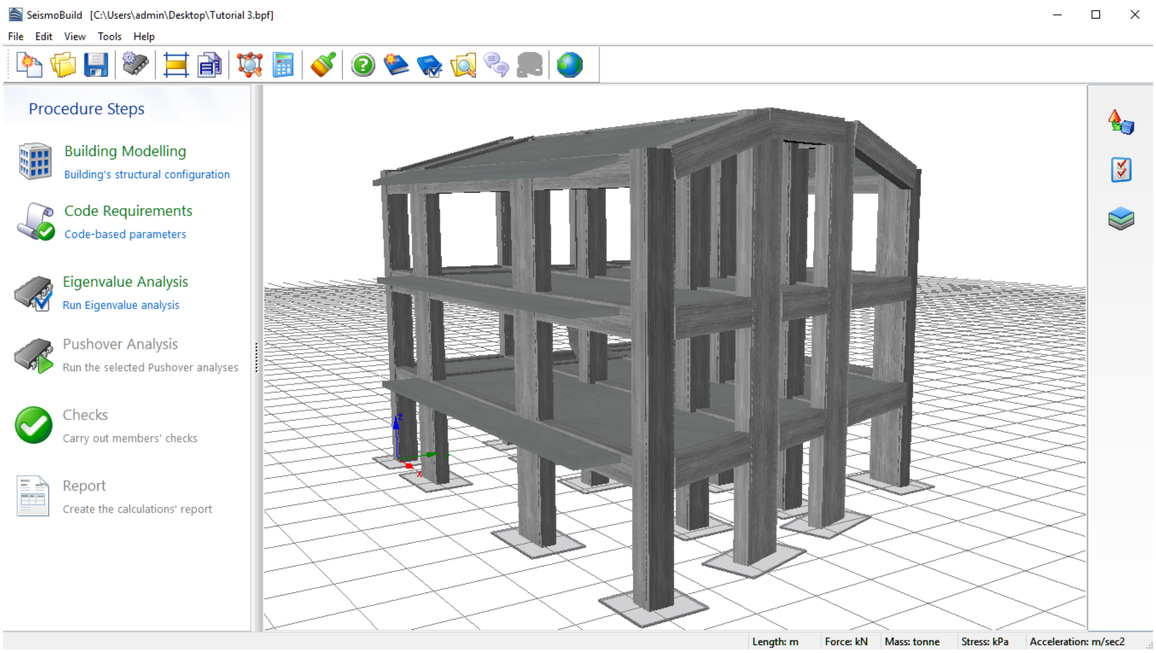
The dimensions and the reinforcement of the jacketed columns of the second and third floors are shown in the following table:

Columns 2nd and 3rd Floors	Height (mm)	Width (mm)	Internal Longitudinal reinforcement	Internal Transverse reinforcement	External Longitudinal reinforcement	External Transverse reinforcement
C1	600	600	4∅16	∅6/25	4∅20+8∅18	∅10/10
C2	1200	400	8∅14+8∅8	∅6/25	8∅20+12∅14	∅10/10
C3	1200	400	8∅14+8∅8	∅6/25	8∅20+12∅14	∅10/10
C4	600	600	4∅16	∅6/25	4∅20+8∅18	∅10/10
C5	400	1200	8∅14+8∅8	∅6/25	8∅20+12∅14	∅10/10
C6	600	600	4∅18	∅6/25	4∅20+8∅18	∅10/10
C7	600	600	4∅18	∅6/25	4∅20+8∅18	∅10/10
C8	600	600	4∅18	∅6/25	4∅20+8∅18	∅10/10
C9	400	1200	8∅14+8∅8	∅6/25	8∅20+12∅14	∅10/10
C10	400	700	6∅16	∅6/25	4∅20+6∅18	∅10/10
C11	600	600	4∅18	∅6/25	4∅20+8∅18	∅10/10
C12	600	600	4∅18	∅6/25	4∅20+8∅18	∅10/10
C13	600	600	4∅18	∅6/25	4∅20+8∅18	∅10/10
C14	400	1200	8∅14+8∅8	∅6/25	8∅20+12∅14	∅10/10
C15	400	700	6∅16	∅6/25	4∅20+6∅18	∅10/10
C16	600	600	4∅16	∅6/25	4∅20+8∅18	∅10/10
C17	1200	400	8∅14+8∅8	∅6/25	8∅20+12∅14	∅10/10
C18	1200	400	8∅14+8∅8	∅6/25	8∅20+12∅14	∅10/10

The dimensions and the reinforcement of the new/external section of the jacketed beams of the second and third floors are the same with those of the first floor.

After inserting all the jacketed elements, you check the building model for one or more very short beams that may have been created unintentionally, due to graphical reasons (e.g. by extending slightly a beam's end beyond a column edge) from the main menu (*Tools > Verify Connectivity...*) or through the respective toolbar button . If such beams exist, the following message appears, and the user can select to remove or keep the element.

You are ready to go to the SeismoBuild Main Window. This can be done from the main menu (*File > Exit & Create 3D Model*) or through the corresponding toolbar button .



SeismoBuild Main Window

Code Requirements

The Code-based parameters and options are defined as in Tutorial N.2.

Analysis & Modelling Parameters

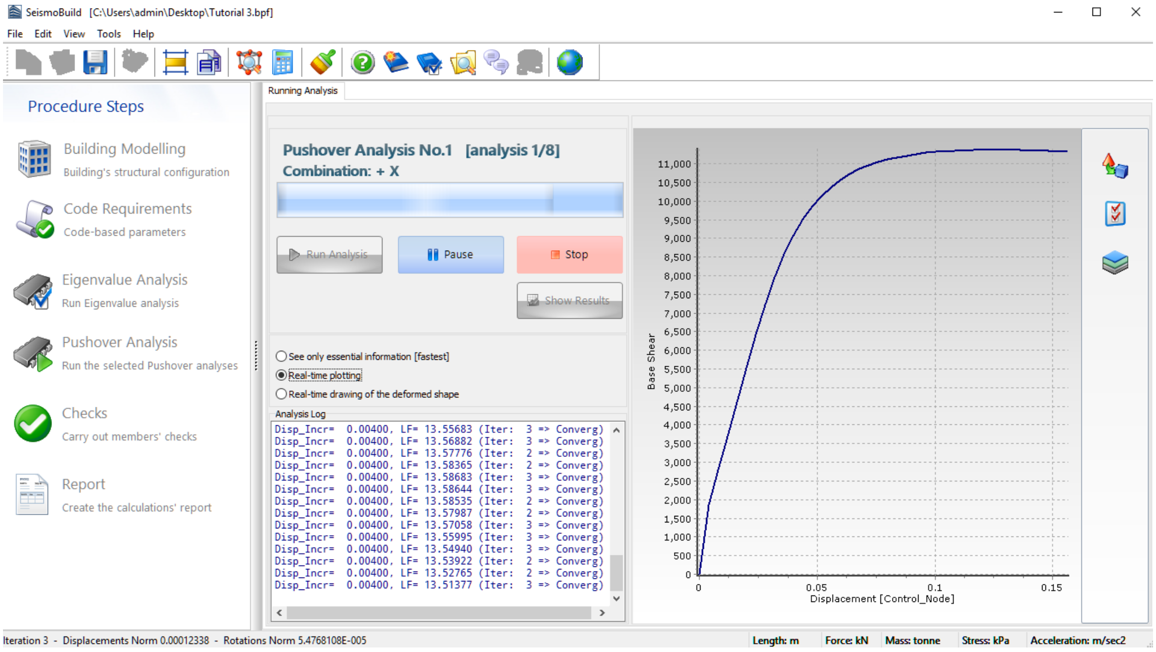
The default predefined settings scheme is employed for the scope of this tutorial.

Eigenvalue Analysis

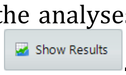
Run the eigenvalue analysis.

Pushover Analysis

Click on the Run button to run all the selected pushover analyses.



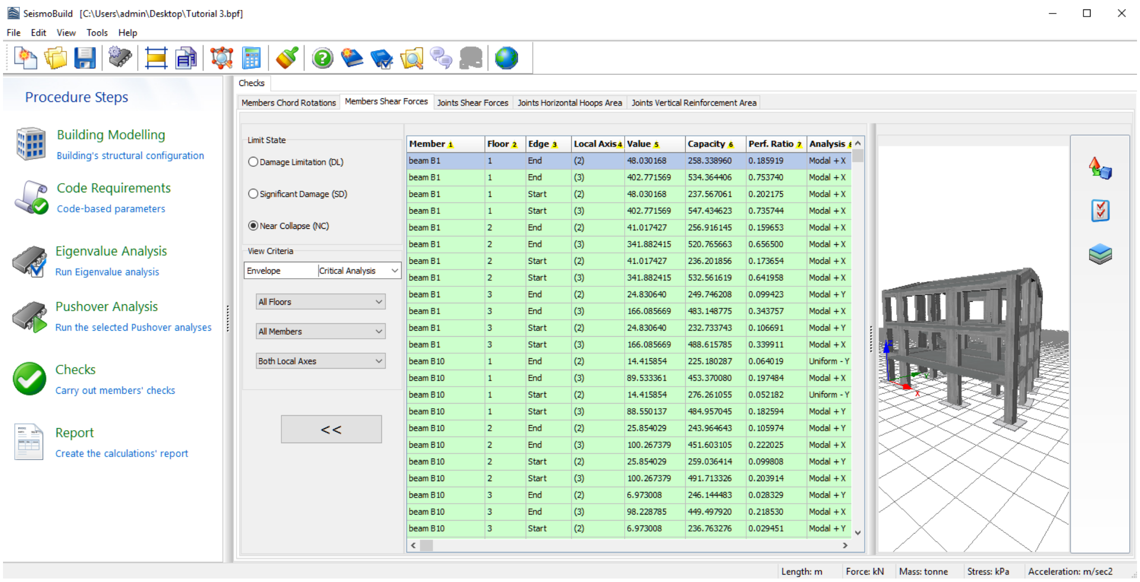
Running the analysis

When the analyses have arrived to the end, you may see the results by clicking on the Show Results button 

. The available modules have been discussed in Tutorial N1.

Checks

SeismoBuild provides the option to automatically undertake chord-rotation and shear checks for structural elements, as well as the beam-column joints checks, according to the expressions defined in the selected Code, herein Eurocode 2 and Eurocode 8, for the selected limit states. The Checks results can be visualised in the Checks module of the 'default' program state, as described in Tutorials 1 and 2.

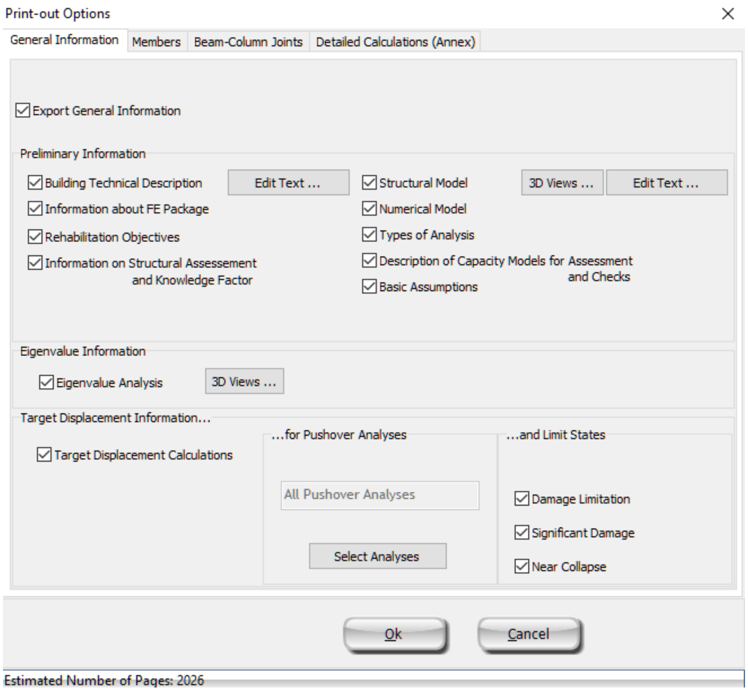


Checks Module (Members Shear Forces)

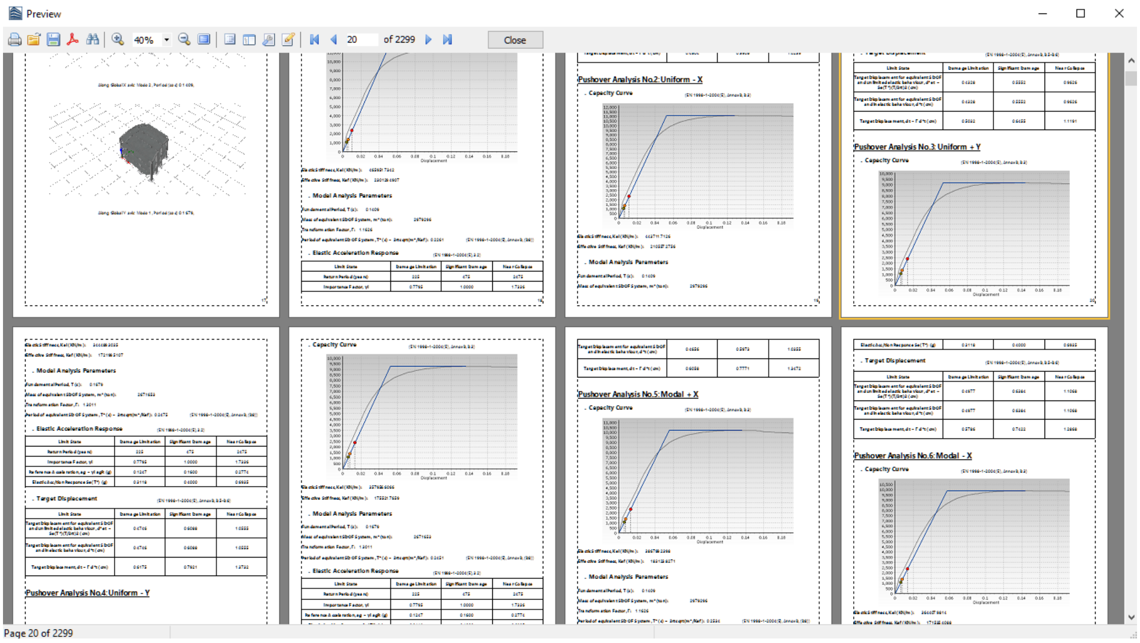
Report

After running the analyses and finishing the checks process, you may create the technical report of the assessment. Once you click on the Report button a window will appear in order to define the print output options. Click the OK button and the report will automatically be created and shown on screen. The report may be exported in PDF, RTF or HTML file formats, the two latter being editable.

NOTE: Creating a report for a typical 4 or 5-storey building may take up to 4-5 minute to complete.



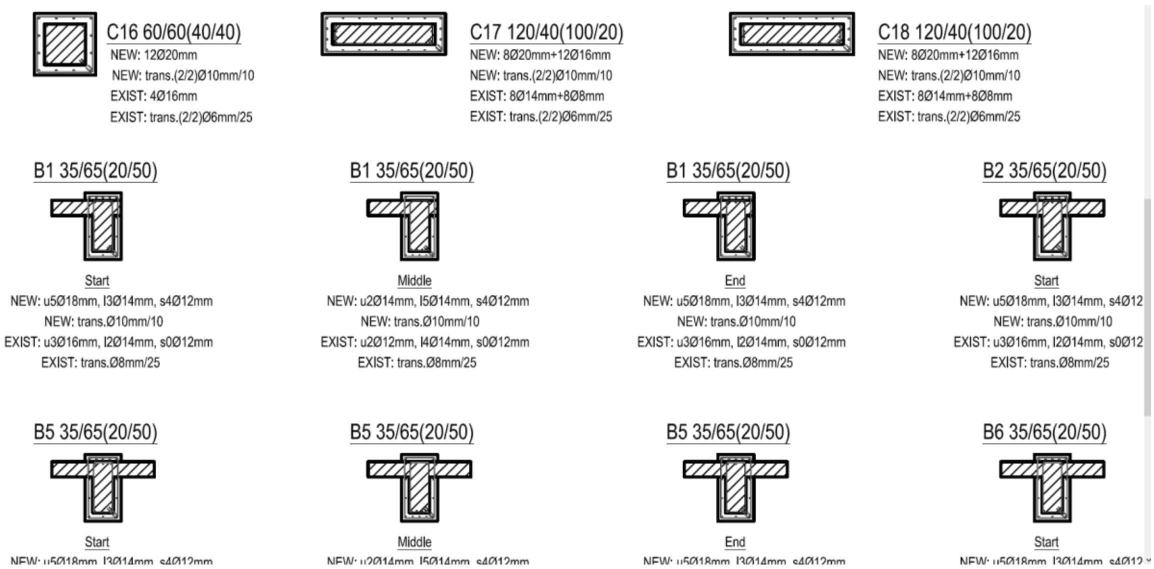
Print-out Options (General information)



Technical Report

CAD Drawing

Finally, you may export a variety of CAD drawing files of the building structural model (plan views, cross sections and reinforcement tables), together with specially created *.ctb files that are needed for plotting. It is noted that running the analyses is not a prerequisite for the exportation of the Cad drawing files.



CAD Drawing

TUTORIAL N.4 – DYNAMIC ANALYSIS OF A THREE-STOREY BUILDING

Problem Description

In this fourth tutorial the model that has already been created in Tutorial N2 will be run with Dynamic Analysis.

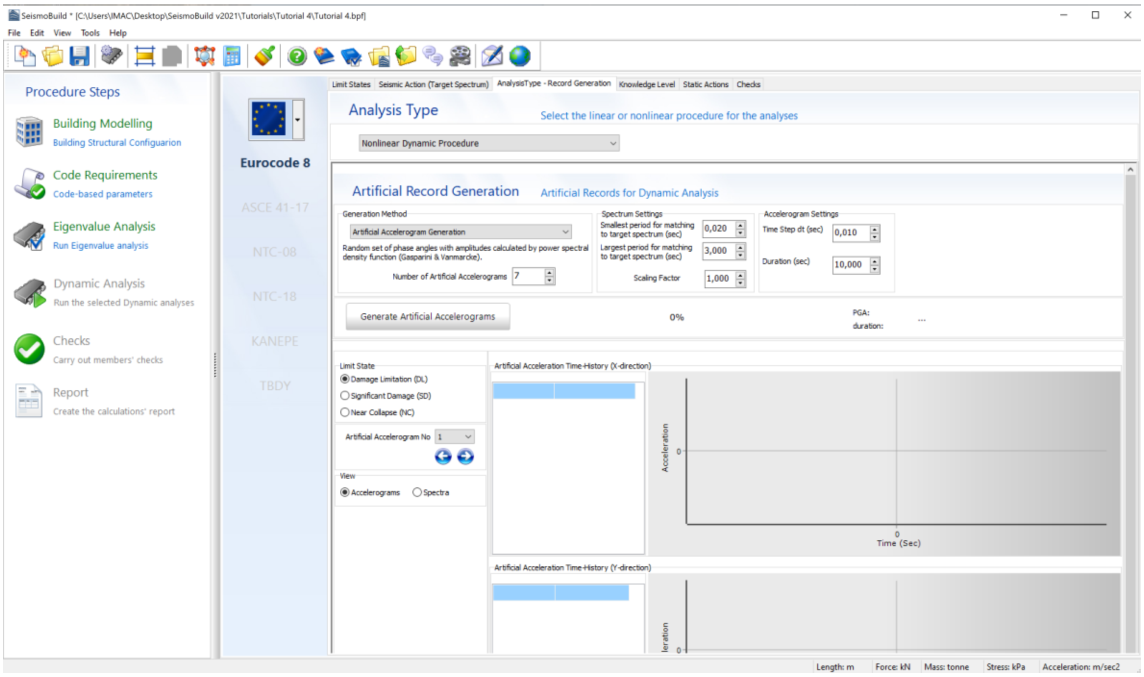
Getting started: opening an existing project

Open again the initial window of the software and, after clicking on icon on the toolbar, select the SeismoBuild project (Tutorial_2.bpf). Once opened, save the project with a new name through *File > Save as…* menu command.

Code Requirements

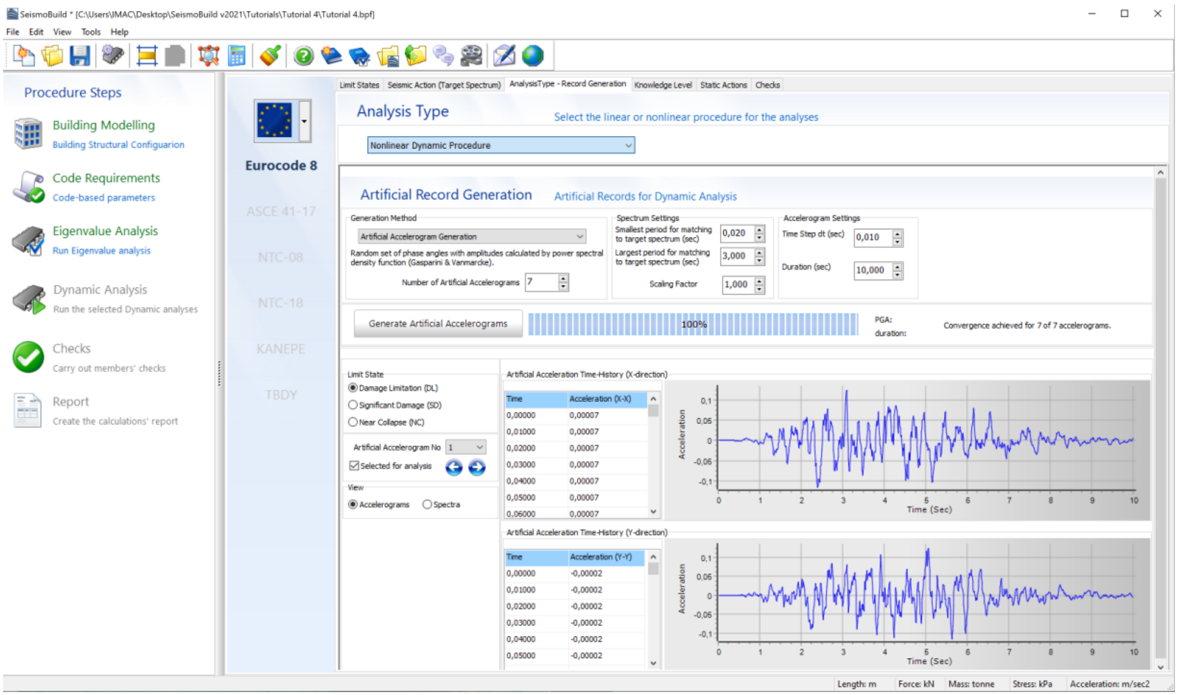
Through the Code requirements module, you are able to define the Analysis Type. The Nonlinear Dynamic Procedure type of analysis is selected.

For this tutorial the artificial accelerogram generation derivation method has been selected for the generation of seven accelerograms with the default options in the spectrum and accelerogram settings.



Analysis Type (Record Generation) – Selected Options

Click the button Generate Artificial Accelerograms.

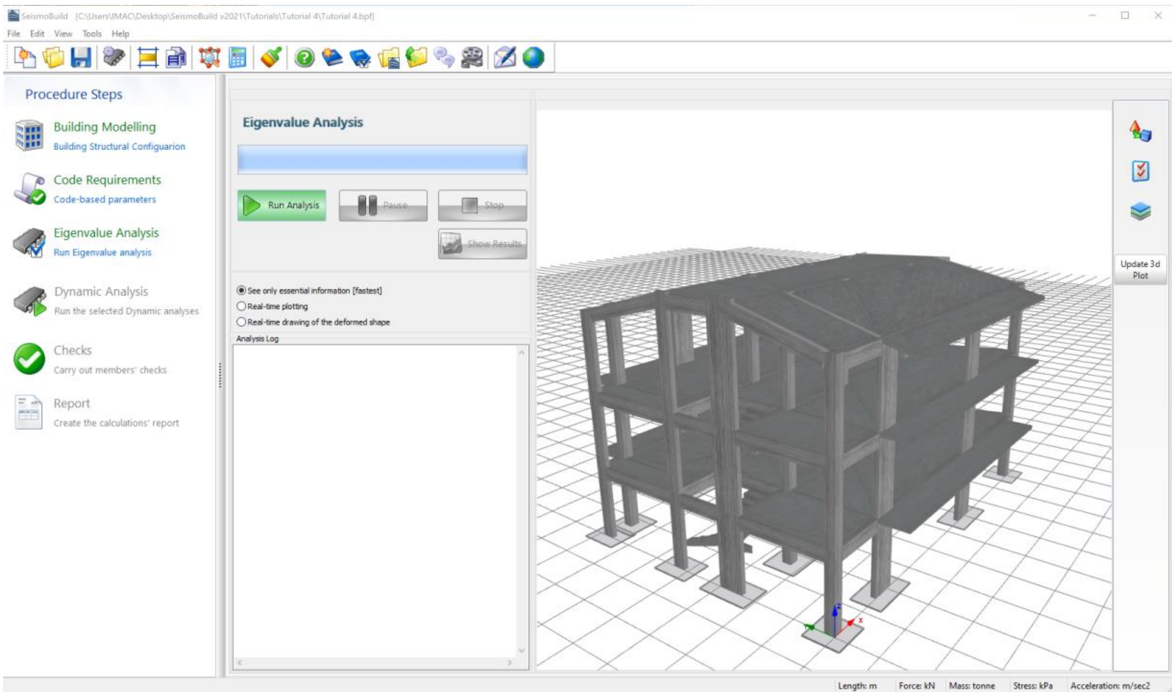


Analysis Type (Record Generation) – Records Generated

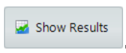
The other Code-based parameters and options are defined as in Tutorial N.2.

Eigenvalue Analysis

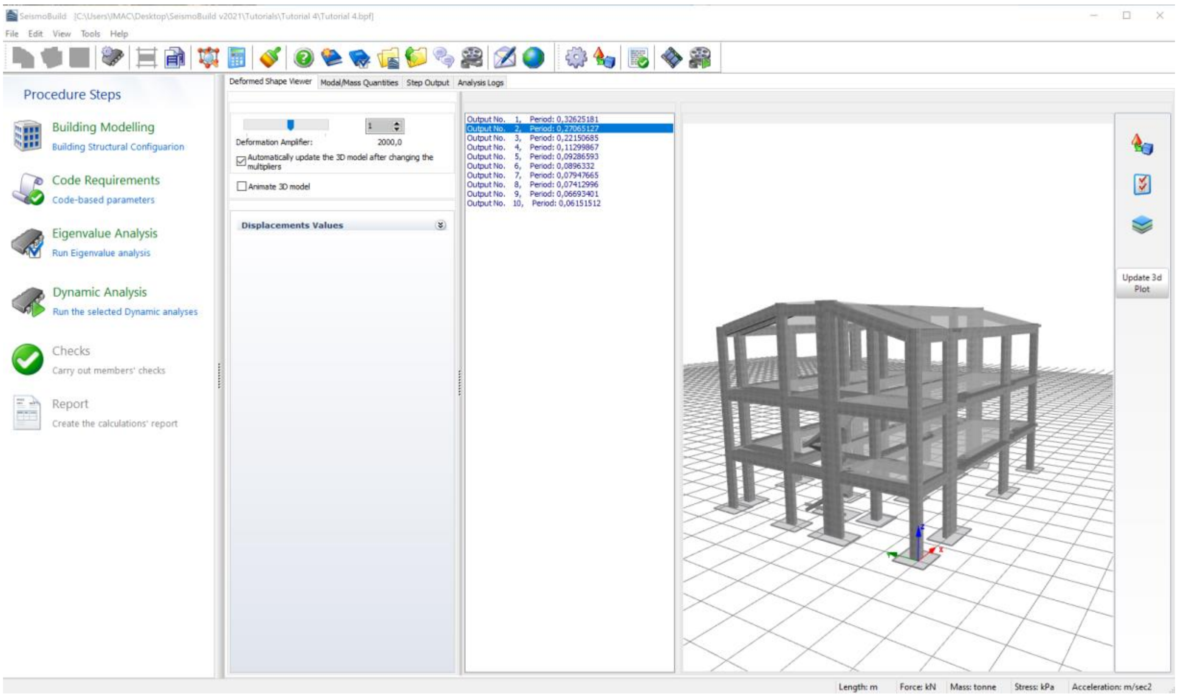
Run the eigenvalue analysis.



Eigenvalue Analysis

You may see the results after running the analysis by clicking on the *Show Results* button 

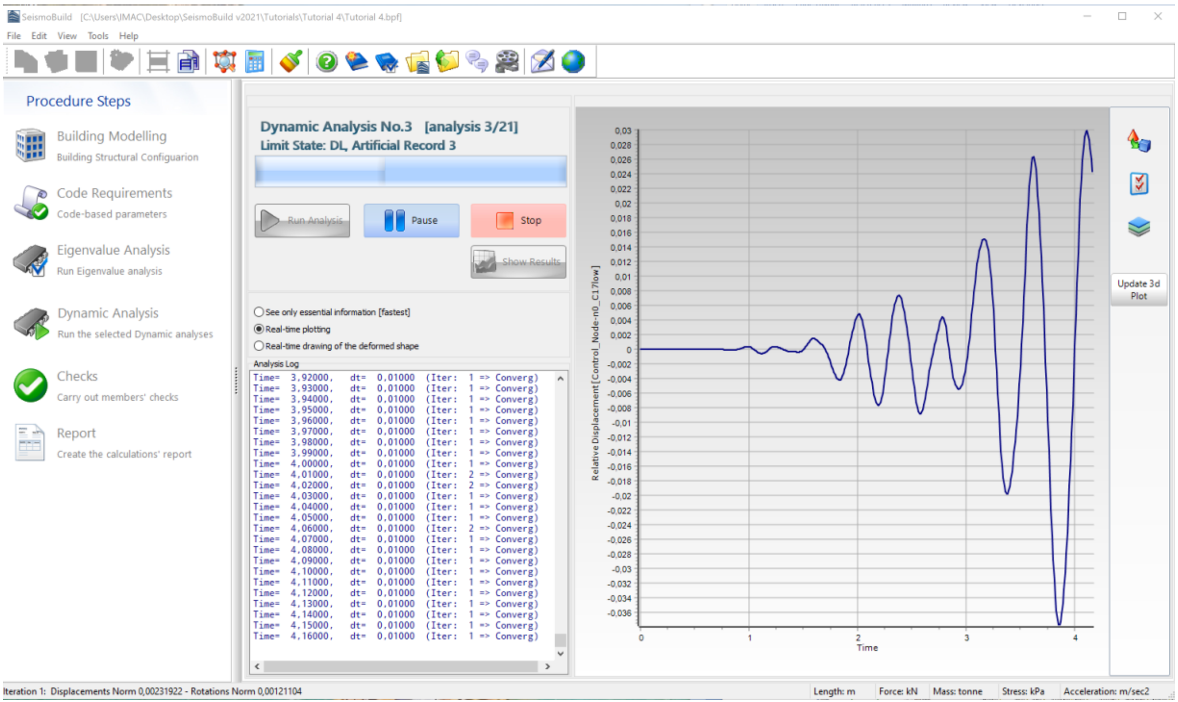
.



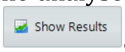
Eigenvalue Analysis Results

Dynamic Analysis

Click on the *Run* button to run all the selected dynamic analyses at once.



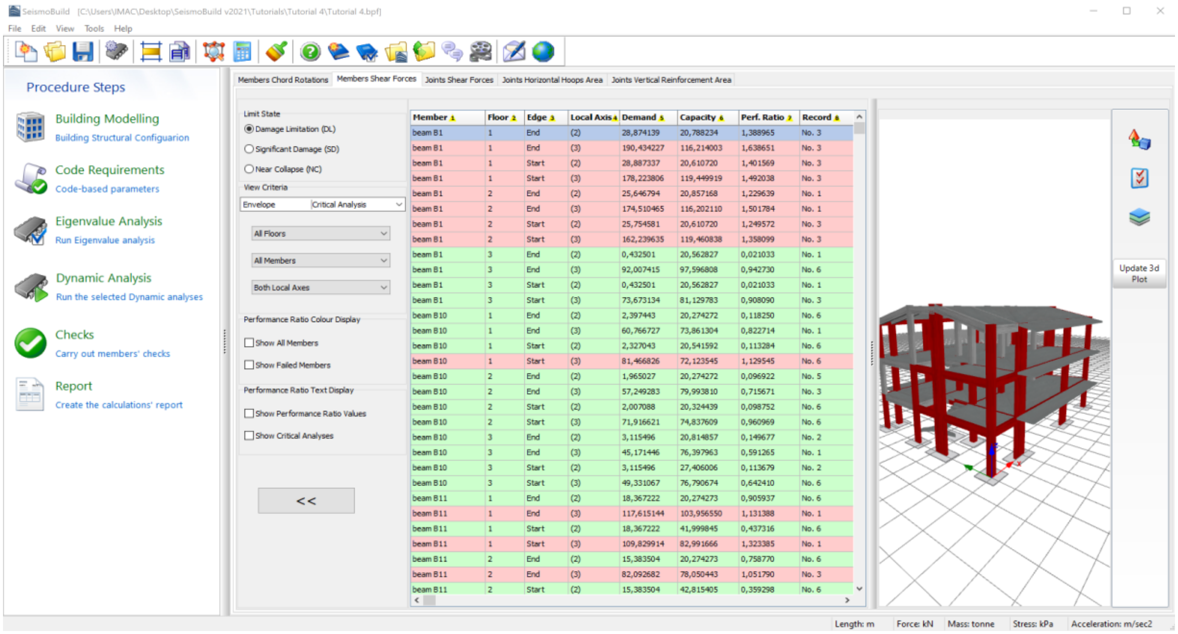
Dynamic Analysis

When the analyses have arrived to the end, you may see the results by clicking on the Show Results button 

. The available modules have been discussed in Tutorial N1.

Checks

SeismoBuild provides the option to automatically undertake chord-rotation and shear checks for structural elements, as well as the beam-column joints checks, according to the expressions defined in the selected Code, herein Eurocode 2 and Eurocode 8, for the selected limit states. The Checks results can be visualised in the Checks module of the 'default' program state, as described in Tutorials 1 and 2.



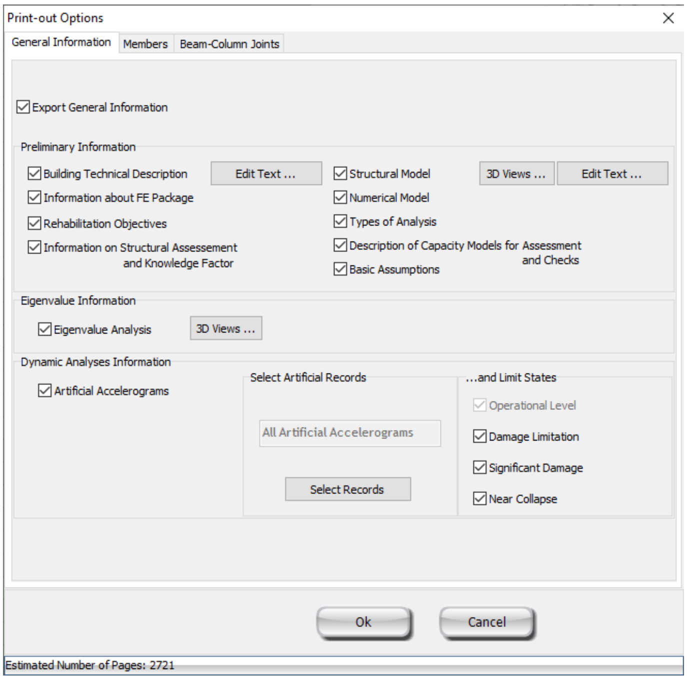
Checks Module (Members Shear Forces)

The results of the structural members' checks can be visualised in table or graphical format, and then copied into any other Windows application. Users may select the limit state, as well as the analysis, the floor, the type of members and the local axis, the envelope and the average performance ratio of the analyses to view the results. The elements, where the demand has exceeded the capacity, are displayed in red both in the table and the 3D plot as shown in the figure above.

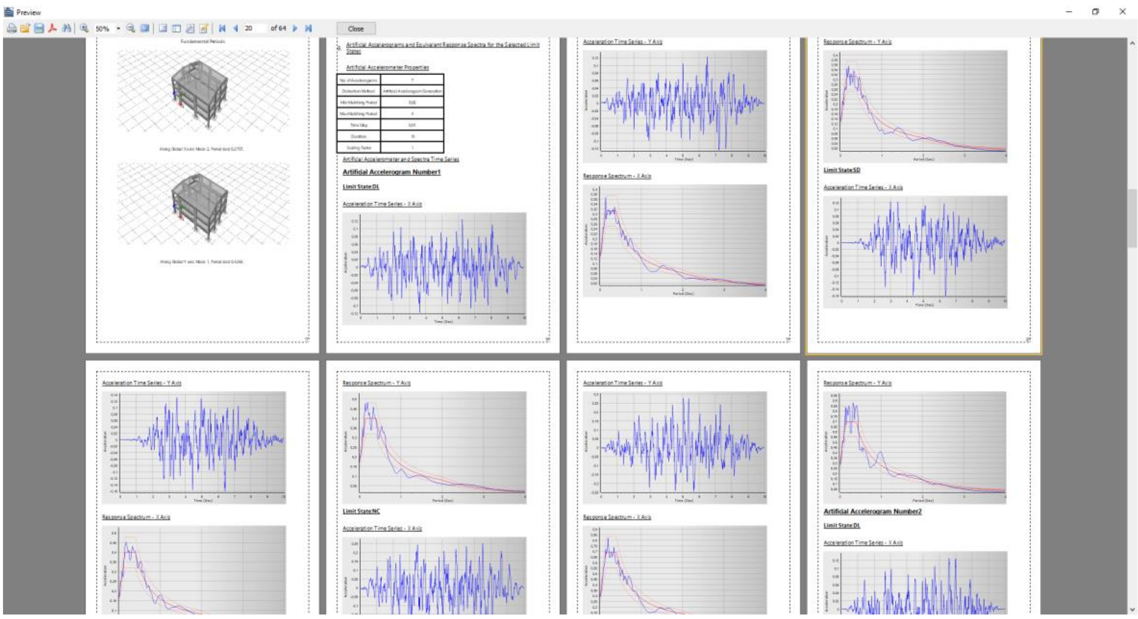
Report

After running the analyses and finishing the checks process, you may create the technical report of the assessment. Once you click on the Report button a window will appear in order to define the print output options. Click the OK button and the report will automatically be created and shown on screen. The report may be exported in PDF, RTF or HTML file formats, the two latter being editable.

NOTE: Creating a report for a typical 4 or 5-storey building may take up to 4-5 minute to complete.



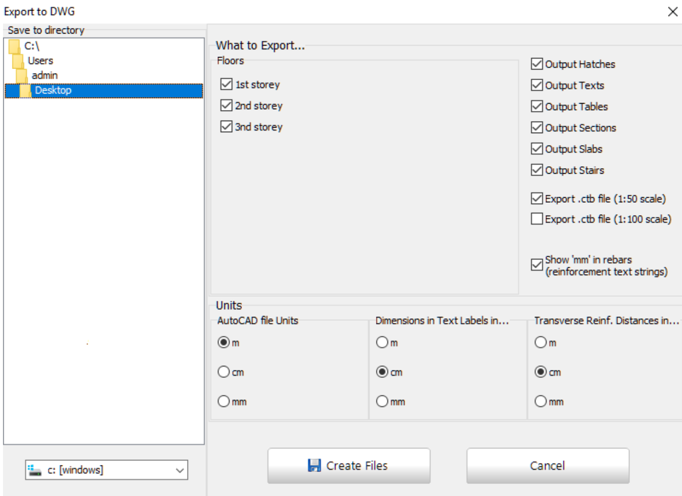
Print-out Options (General information)



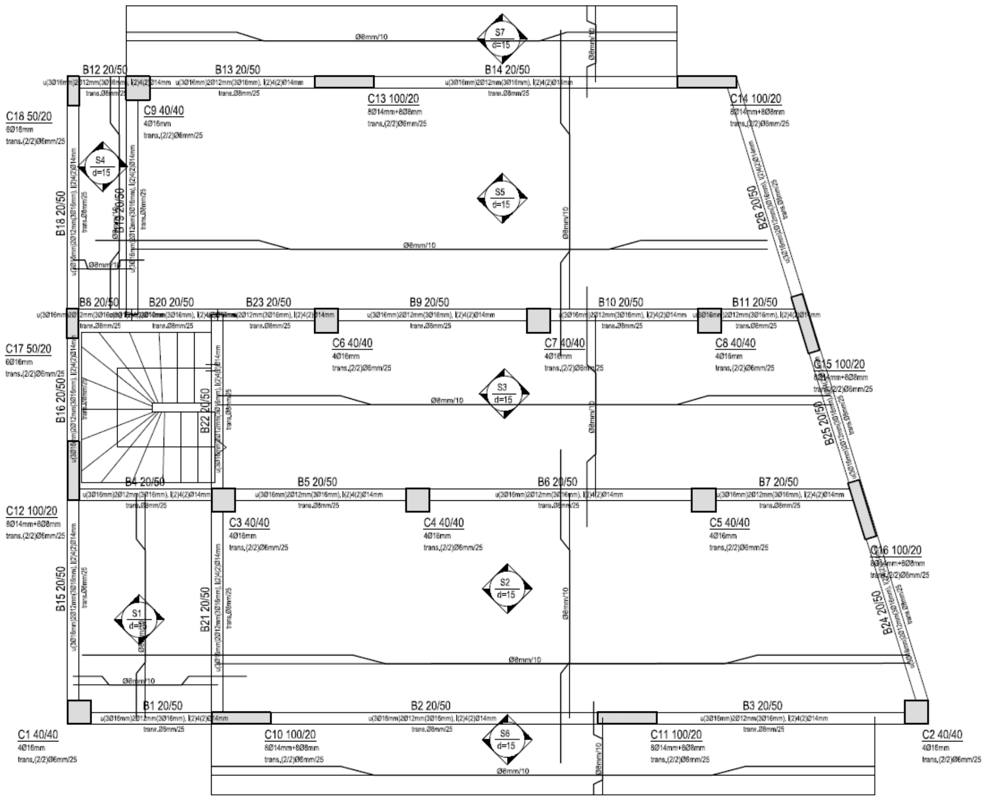
Technical Report

CAD Drawing

Finally, you may export a variety of CAD drawing files of the structural model (plan views, members' cross sections and reinforcement tables), together with specially created *.ctb files that are needed for plotting. It is noted that running the analyses is not a prerequisite for the exportation of the Cad drawing files, and only the introduction of the structural configuration in the Building Modeller is required.



Export to DWG



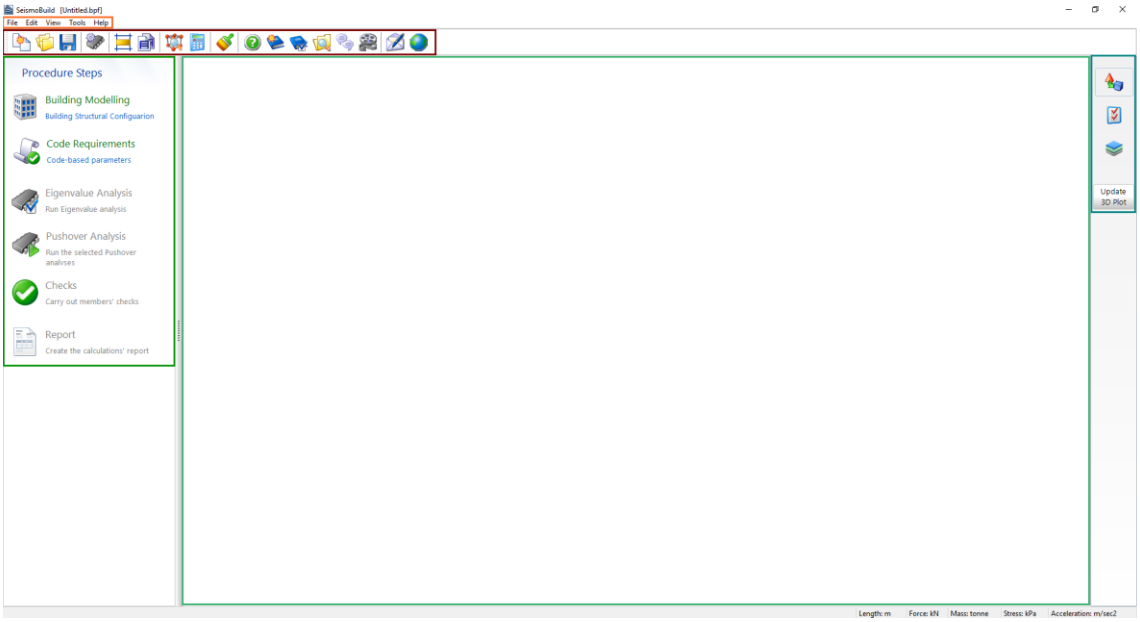
CAD drawing

SeismoBuild Main Menu

MAIN MENU AND TOOLBAR

SeismoBuild has a simple and 'easy to understand' user interface. The program's **Main Window**, is subdivided into the following components:

- **Main menu and toolbar**: at the top of the program window;
- **3D Model window:** on the centre of the screen
- **Settings bar for the 3D Model**: on the right of the program window;
- **Procedure steps list**: on the left of the program window.



Main Window Area

Main menu

The **main menu** is the command menu of the program. It consists of the following sub-menus:

- File
- Edit
- View
- Tools
- Help

Main toolbar

The **main toolbar** provides quick access to frequently used items from the menu.

Main toolbar

An overview of all the commands necessary to run SeismoBuild is shown below:

Command	Main menu	Shortcut keys	Toolbar button
File	New	Ctrl+N	
	Open	Ctrl+O	
	Import from SeismoStruct Building Modeller...	Ctrl+M	
	Import from XML File...		
	Save	Ctrl+S	
	Save as...	-	
	Export Model to SeismoStruct	-	
	Export Model to XML...		
	Export CAD drawings	-	
Edit	Copy 3D Plot	Ctrl+Alt+C	
View	Analysis Parameters		
	Model Statistics		
	View Large Icons		
	View Small Icons		
Tools	FRP Designer		
	3D Plot Options...	-	
	Deformed Shape Settings...	-	
	Export to Text File	-	
	Create AVI File...	-	
	Show AVI File...	-	
	Calculator	-	
Help	SeismoBuild Help	F1	
	SeismoBuild User Manual		
	SeismoBuild and SeismoStruct Verification Reports		
	SeismoBuild Sample Files		
	Seismosoft Forum		
	Video Tutorials		
	Seismosoft Support Webpage		
	Send Message to Seismosoft		
	Seismosoft Website	-	
	Register New License	-	
	Set Language...		

Command	Main menu	Shortcut keys	Toolbar button
	About...	-	

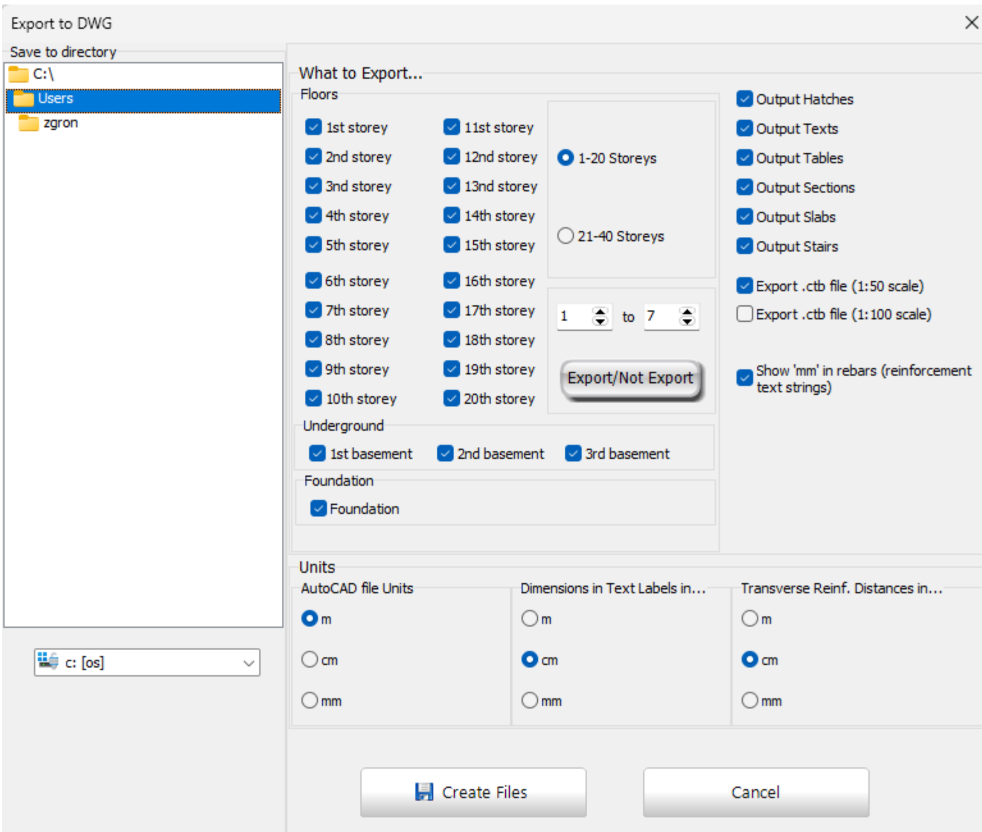
Exporting and Importing SeismoBuild Projects as XML Files

A SeismoBuild project can be exported in the form of an XML file from the main menu (*File >Export to XML file).* When this option is selected, a new window appears for the definition of the name and location of the XML file. The exported XML file will contain all the information defined for the SeismoBuild project in the Building Modeller and the Code Requirements modules, organized in the form of nodes entitled according to the information carried by each node. An XML file containing the information of a SeismoBuild project can be loaded in SeismoBuild from the main menu (*File >Import from XML file)* while the information contained in the XML file and defining the various settings of the project can be modified directly in the XML file.

Export CAD Drawings

A variety of CAD drawing files of the structural model (plan views, cross sections and reinforcement tables) may be quickly created and exported from the main menu (*File > Export CAD drawings*), together with specially created *.ctb files that are needed for plotting.

Users may define the number of exported files (one file per floor) and the information to be included in the CAD file, the units etc.



Export to DWG module

Export Model to SeismoStruct

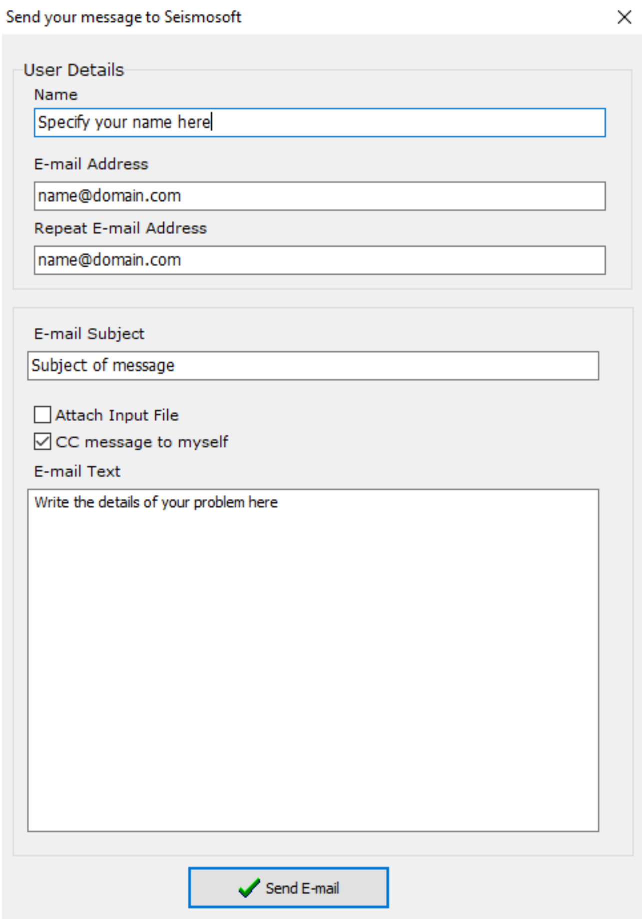
The possibility of exporting SeismoStruct projects from the main menu (*File > Export Data to SeismoStruct*) is available. All the SeismoStruct projects for all the selected analyses (the eigenvalue analysis and all the linear/nonlinear static/ dynamic analyses) will be exported to the folder of the SeismoBuild project.

Import from SeismoStruct Building Modeller

A SeismoStruct Building Modeller project (*.bmf) can be loaded in SeismoBuild from the main menu (*File >Import from SeismoStruct Building Modeller).*

Send Message to Seismosoft

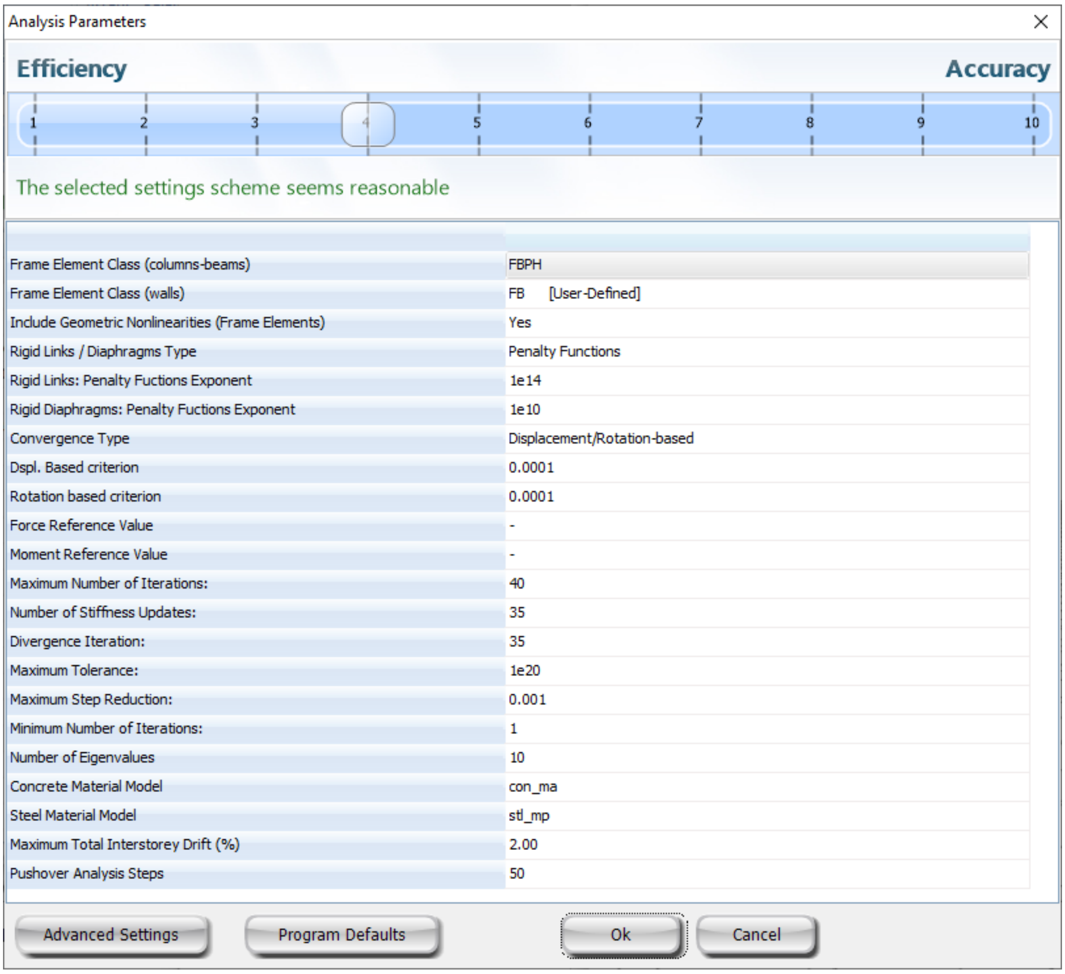
Commercial users may send a message to Seismosoft from the main menu (*Help > Send Message to Seismosoft*) or through the toolbar button. Once the “Attach Input File” checkbox is selected the model is automatically attached and will be sent to the Seismosoft Support group. It is noted that this facility is available in the Commercial Version only.



Send your Message to Seismosoft module

Analysis Parameters

All the parameters required for the nonlinear analytical calculations may be defined from the main menu (*View > Analysis Parameters*) or from the button. Further information regarding the Analysis & Modelling Parameters may be found in the corresponding chapter of this Manual.



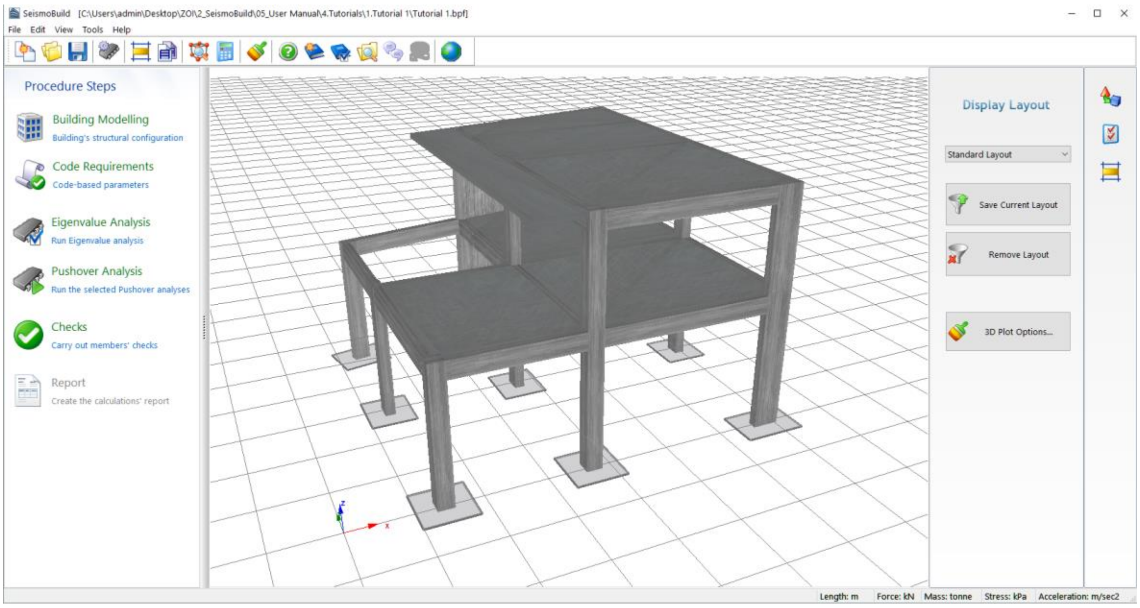
Analysis & Modelling Parameters module

3D PLOT OPTIONS

The 3D Plot settings of the structural model can be adjusted to best meet the user's preferences and requirements.

Display Layout

With this facility, accessible through the button on the right, users can (i) select a pre-defined layout, such as Standard Layout (default) and Structural Model (the latter is particularly useful to visualise internal forces results), (ii) save their personal Display Layouts or (iii) change the 3D Plot Options.



Display Layout

Save Current Layout

Users may wish to save the changes made in the 3D Plot Options. To do so they have to:

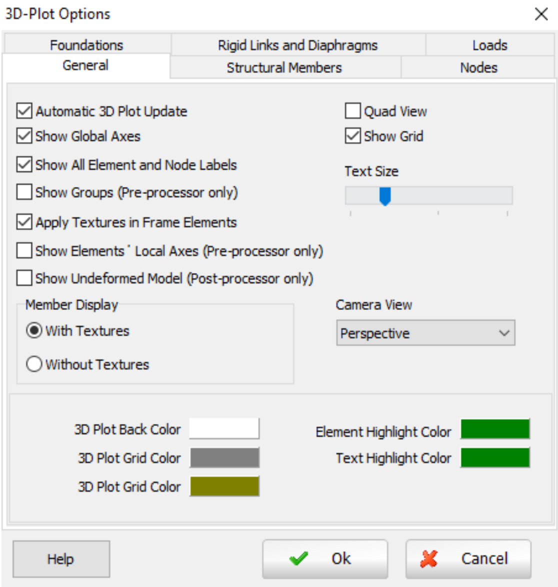
1. Click on the button ;
2. Assign a name to the new layout configuration;
3. Click the OK button to confirm the operation.

The new layout will appear in the corresponding drop-down menu. Further, users may always return to the initial default layout by selecting the Standard Layout option from the drop-down list.

3D Plot Options…

The full range of plotting adjustment parameters, on the other hand, can be found in the 3D Plot Options dialog box, accessible from the main menu (*Tools > 3D Plot Options…*) or through the button.

Within the 3D Plot Options menu, there are a number of submenus from which users can, not only select which model components (nodes, structural members, etc.) to show in the plot, but also change a myriad of settings such as the colour/transparency of elements, the plot axes and background panels, the colour and size of text descriptors, and so on.

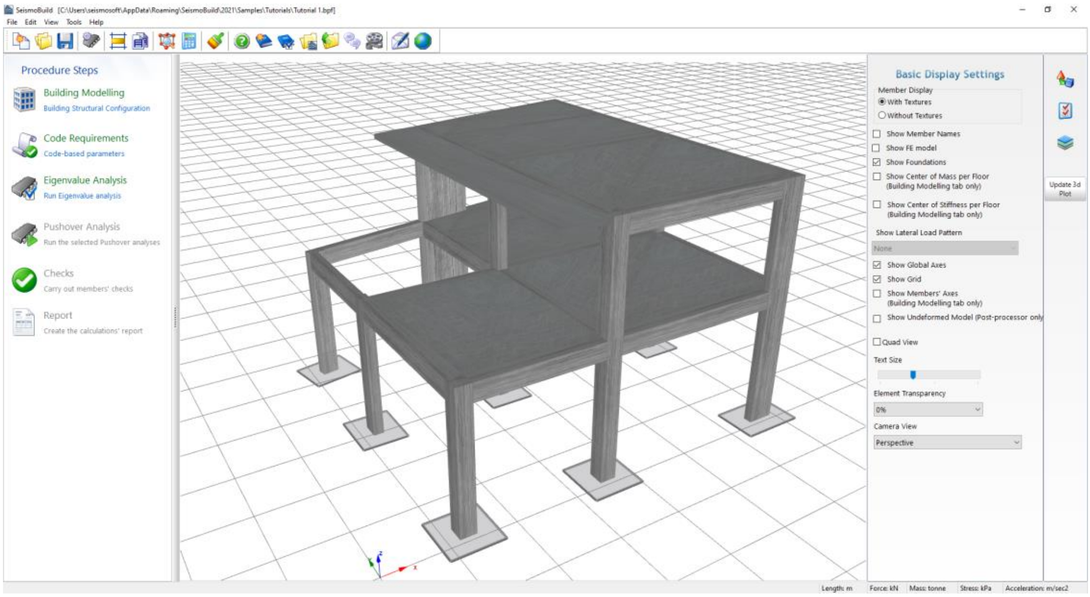


3D Plot Options menu

By default, the 3D Plot is automatically updated. In cases where the structural model is very large (several hundreds of elements) and/or the user is using a laptop running on batteries with a slowed-down CPU (so as to increase the duration of battery), the program takes some seconds to update the view. Hence, it might prove to be more convenient for users to disable this feature (uncheck the *Automatic 3D Plot Update* option in the 3D Plot Options *General* submenu) and thus opt for manual updating instead, carried out with the *Update 3D Plot* command found in the 3D Plot Options on the right of the screen.

Basic Display Settings

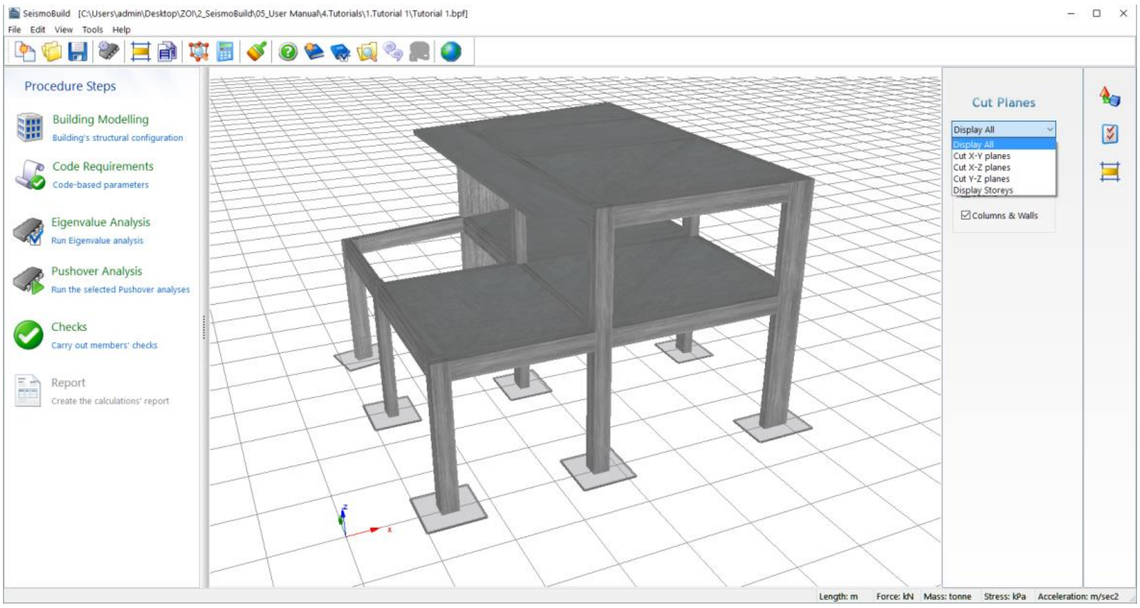
These are a list of settings accessible through the  button on the right, users can tweak the most commonly used plotting features (view type, rendering options, names show, members' axes representation, element transparency, and so on) using the available check-boxes and drop-down menus.



Basic Display Settings

Cut Planes

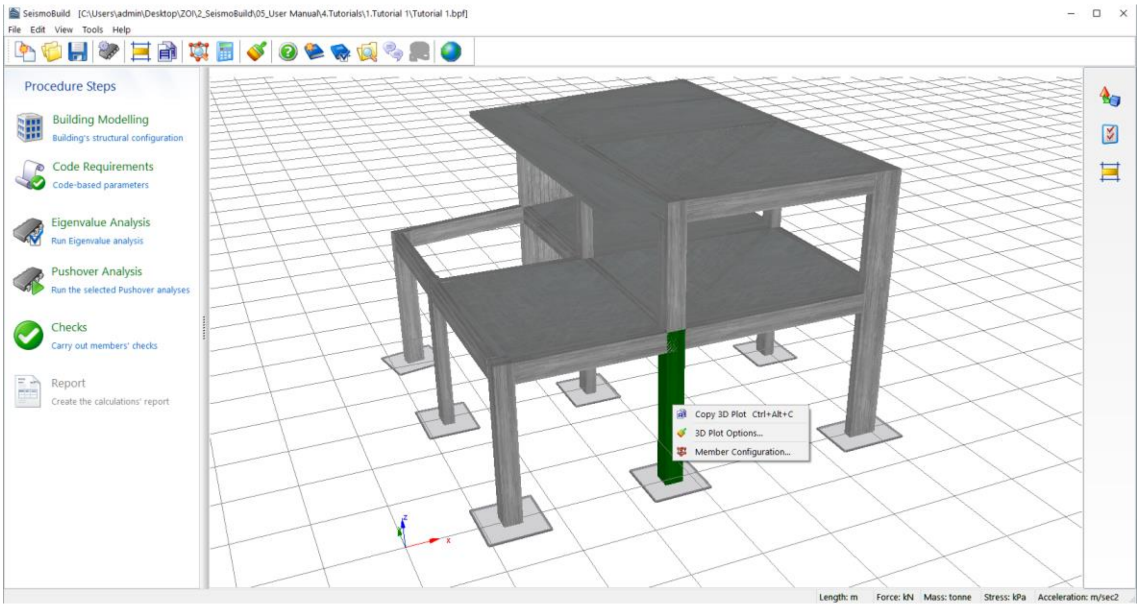
In addition to the previous features, also the Cut Planes option can be activated through the button on the right.



Cut Planes

Additional operations

Users can also quickly zoom, rotate, and move the 3D/2D plot of the structural model, by using either the mouse (highly recommended) or keyboard shortcuts. Further, it is also possible to point&click elements to quickly go to the Building Modeller to view/modify element's properties, or right-click and select "Member Configuration...".


SeismoBuild [C:\Users\admin\Desktop\ZOI\2_SeismoBuild\05_User Manual\4.Tutorials\1.Tutorial 1\Tutorial 1.bpf]
File Edit View Tools Help
Procedure Steps
Building Modelling
Building's structural configuration
Code Requirements
Code-based parameters
Eigenvalue Analysis
Run Eigenvalue analysis
Pushover Analysis
Run the selected Pushover analyses
Checks
Carry out members' checks
Report
Create the calculations' report
Copy 3D Plot Ctrl+Alt+C
3D Plot Options...
Member Configuration...
Length: m
Force: kN
Mass: tonne
Stress: kPa
Acceleration: m/sec2

Building Modeller

A special CAD-based facility is introduced to facilitate the creation of building models. Currently, only reinforced concrete buildings can be created; in subsequent releases of the program steel and composite models will be also supported.

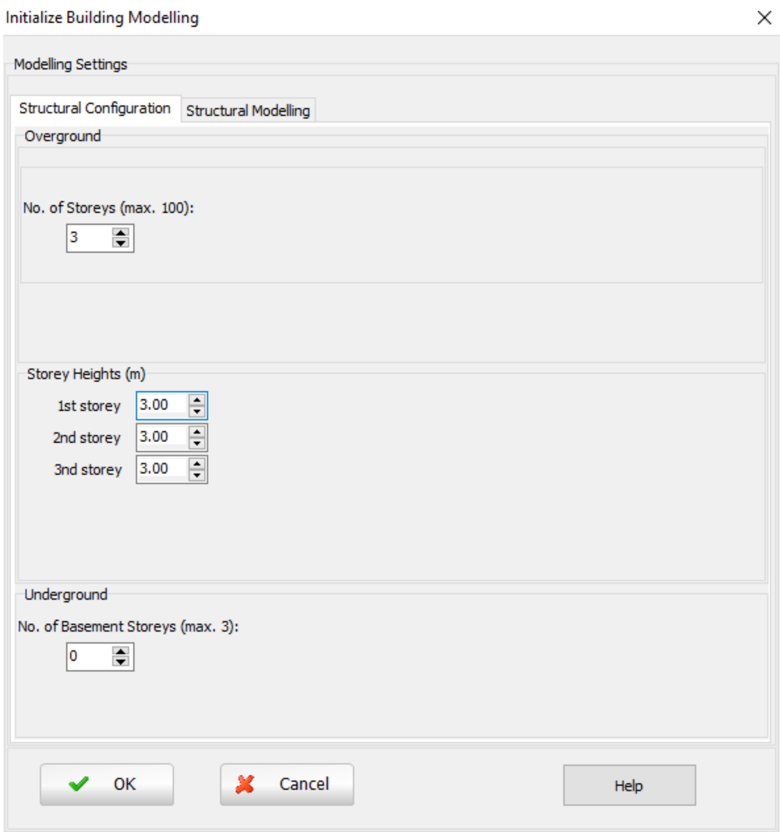
The Building Modeller is accessed from the program Main Window by clicking on the Building Modelling button .

Modelling Settings

Users are able to define the geometry of the new building and the main settings of the model in the Initialize Building Modelling dialog box.

Structural Configuration

In the Structural Configuration tab, the number of storeys and their heights are defined; a number from 1 to 100 storeys, with different heights at each storey and the possibility of applying a common height to a range of storeys, may be selected. Up to three underground floors (basement storeys) and their heights may also be defined. The default selection for this module is 3 storeys with 3.00m height each without basement storeys.



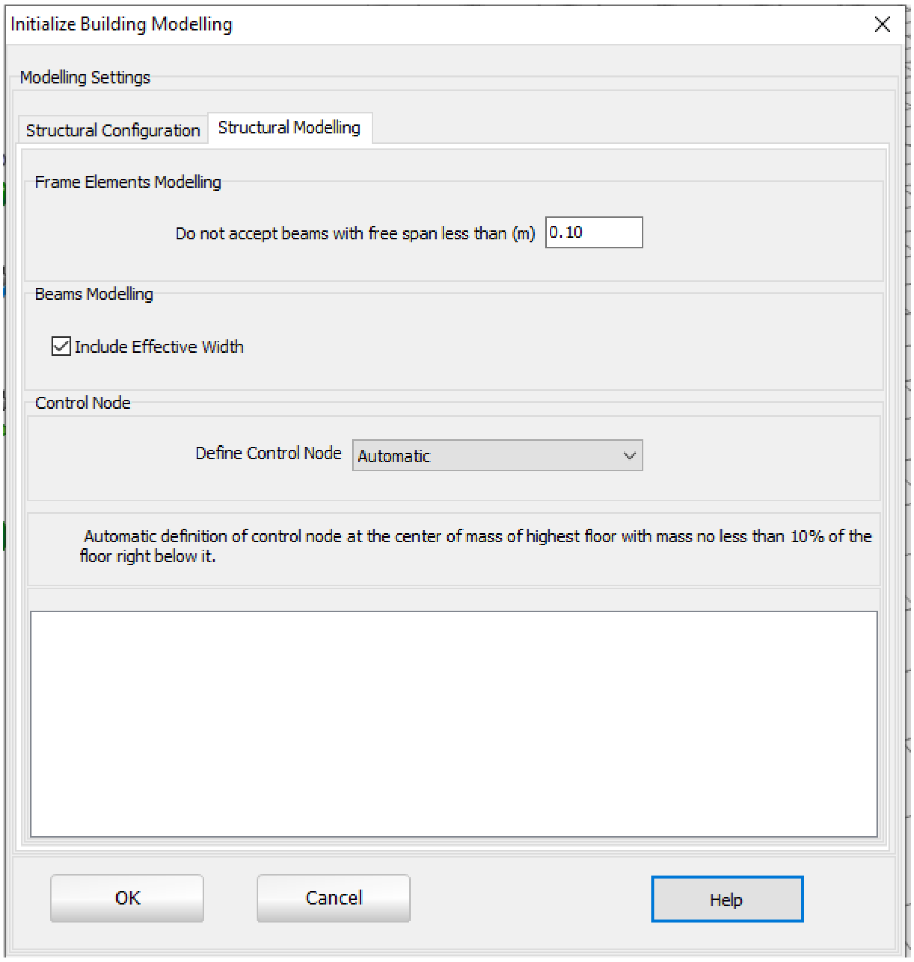
Modelling Settings – Structural Configuration

Structural Modelling

The option of not accepting beams shorter than a specific length is available through the Structural Modelling tab to avoid the creation of very short beams, by mistake (e.g. by extending slightly a beam's edge after the column at its end). The default value for this option is 0.1m.

Users may also decide whether to include the effective slab width in the beams modelling. The choice of considering zero axial forces in shear checks of beam members is also available.

Finally, the definition of the control node is made within this module. Users may select directly the floor of the control node, or alternatively choose the automatic definition, in which the control node is defined at the center of mass of the upper floor or at the floor lower to that (in the case of having a top floor mass less than 10% of the lower floor's), depending on the choice made in the Advanced Settings> Advanced Building Properties.



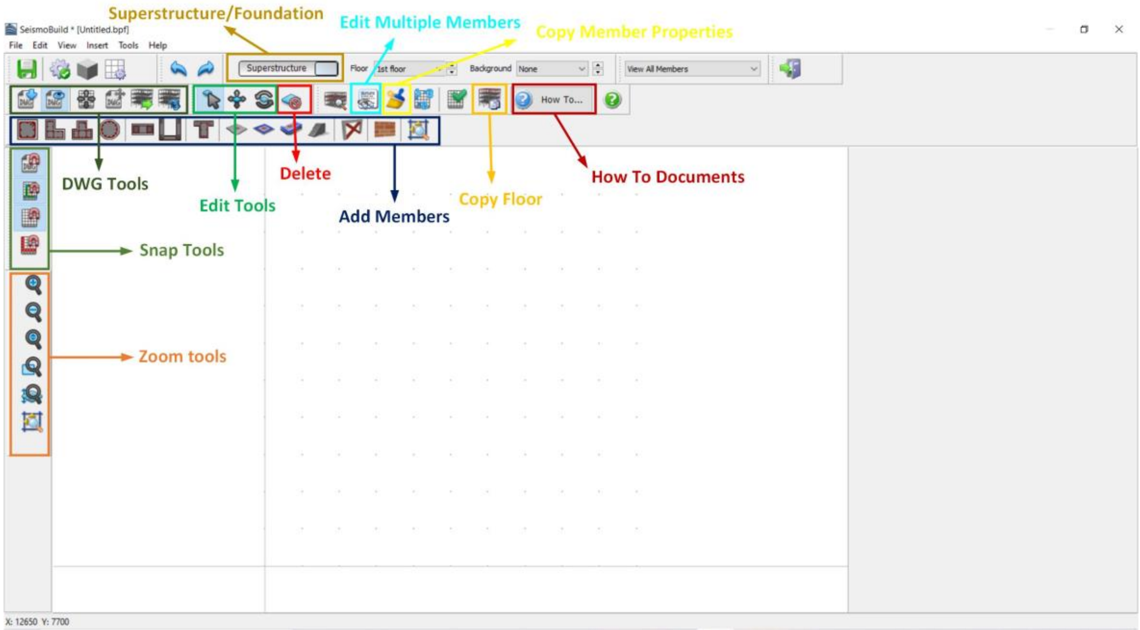
Modelling Settings – Structural Modelling

It is noted that the Building Modeller settings can be changed later through the  toolbar button.

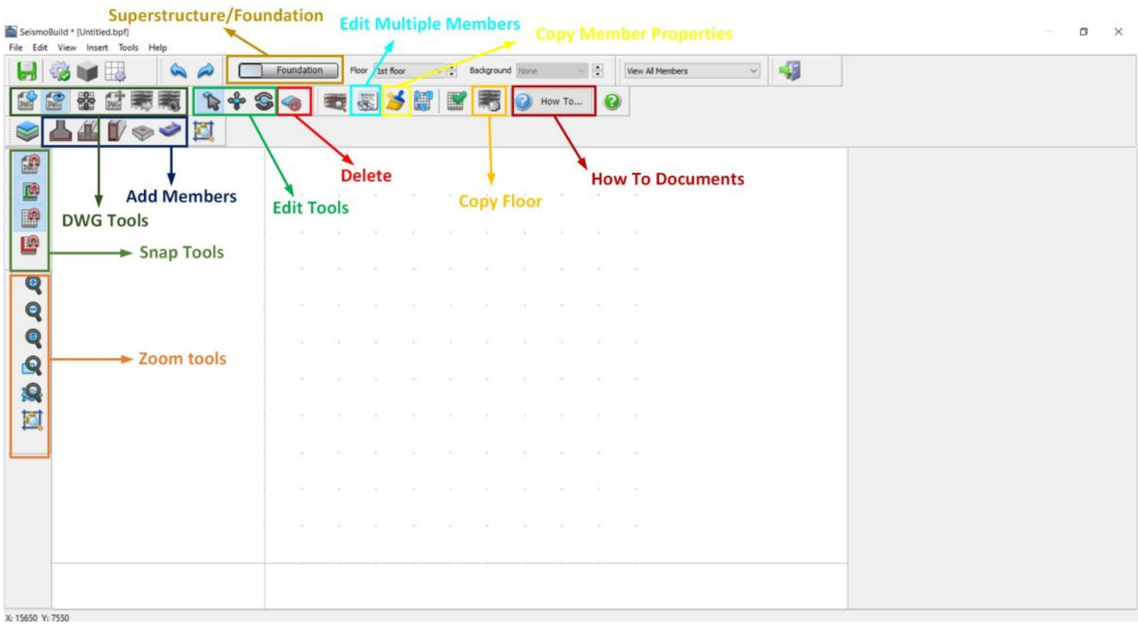
BUILDING MODELLER MAIN WINDOW

After defining the building's main settings, the Building Modeller Main Window appears, as shown in the figure below. The Building Modeller provides two different working modes namely the Superstructure and the Foundation mode. The user can switch between the two modes through the corresponding button in the toolbar.

Users may also select what type of members to view in the Building modeller. The available options are: (i) View Concrete Frame, (ii) View Infills & Steel Braces and (iii) View All Members. When the first option is selected, the infills and steel braces already imported in the model cannot be selected or edited, whereas when the second option is selected the assigned beam members cannot be selected or edited.



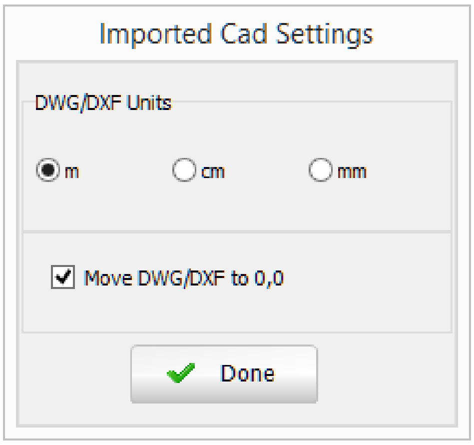
Building Modeller Main Window (Superstructure)



Building Modeller Main Window (Foundation Mode)

INSERTING A BACKGROUND

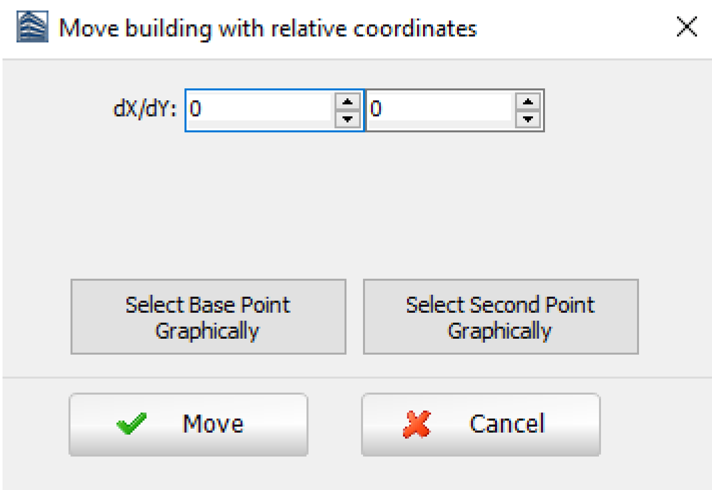
The possibility of inserting as background a CAD drawing is offered from the main menu (*File > Import DWG...*) or through the corresponding toolbar button . Once the drawing is inserted the user is asked to specify drawing's units and to choose whether to move the DWG/DXF file to (0,0), i.e. to the origin of the coordinates system. Selecting this checkbox moves the bottom-left edge of the drawing to the (0,0) coordinates, irrespective of its initial CAD coordinates.



Imported Cad Settings window

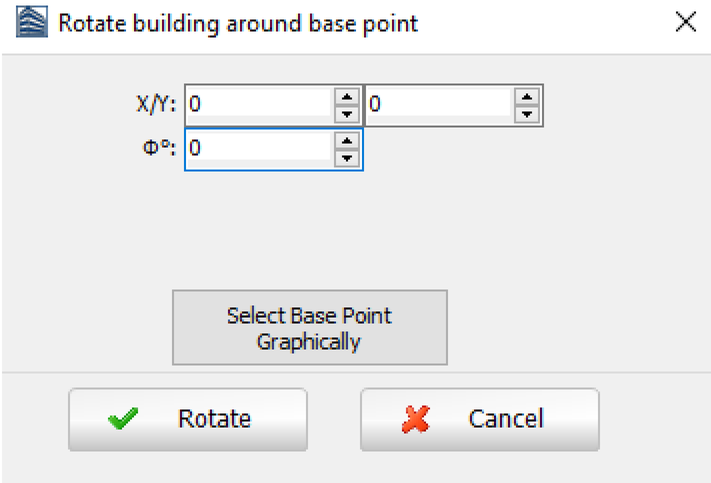
Note that the axes origin can be further moved to a different point that might be more suitable after loading the CAD file with the Move Axes Center ( ) toolbar button, also accessible from the main menu (*View > Move Axes Center*). The option of moving the imported CAD file is also available through the Move DWG ( ) toolbar button or from the main menu (*View > Move DWG*). Further, from the main menu (*View > Show/Hide DWG*) or through the toolbar button is defined whether the CAD drawing will be visible or not.

Users may also move the building in plan view from the main menu (*Tools > Move Building*) or from the corresponding toolbar button by either assigning the relative coordinates or by selecting the base point and the second point graphically.



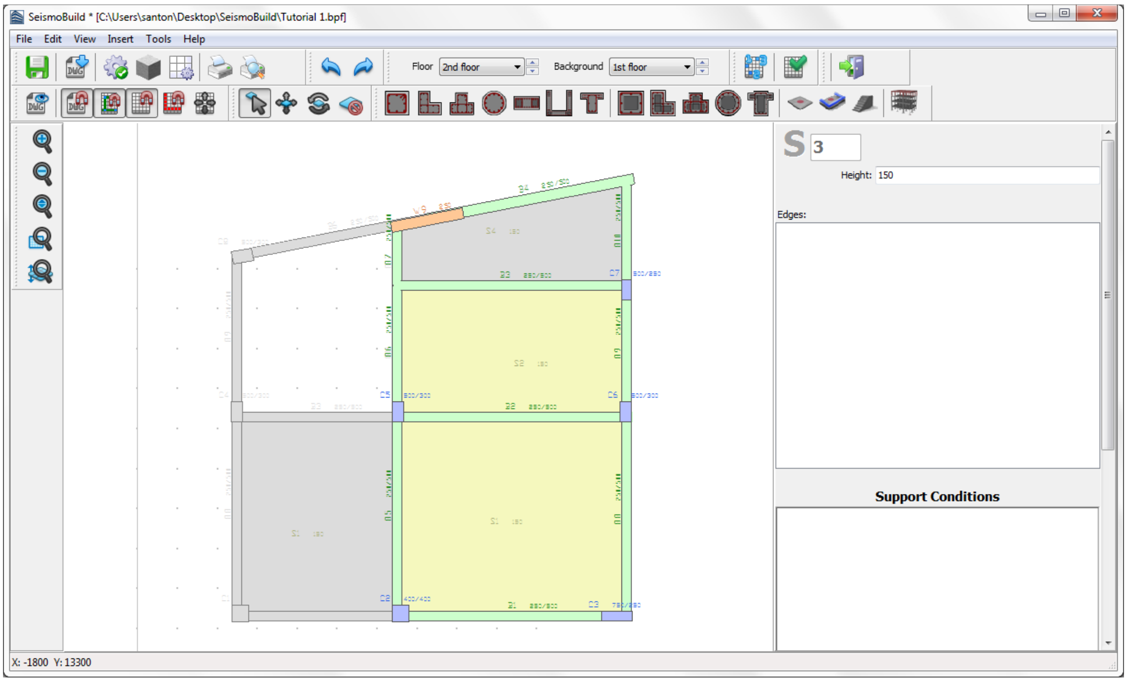
Move Building window

The option of rotating the building in plan view is also available from the main menu (*Tools > Rotate Building*) or from the toolbar button. Users should specify the base point by its coordinates or graphically and assign the rotation angle.



Rotate Building window

Finally, the layout of an existing floor may be used as background in order to easily introduce new members on another storey.



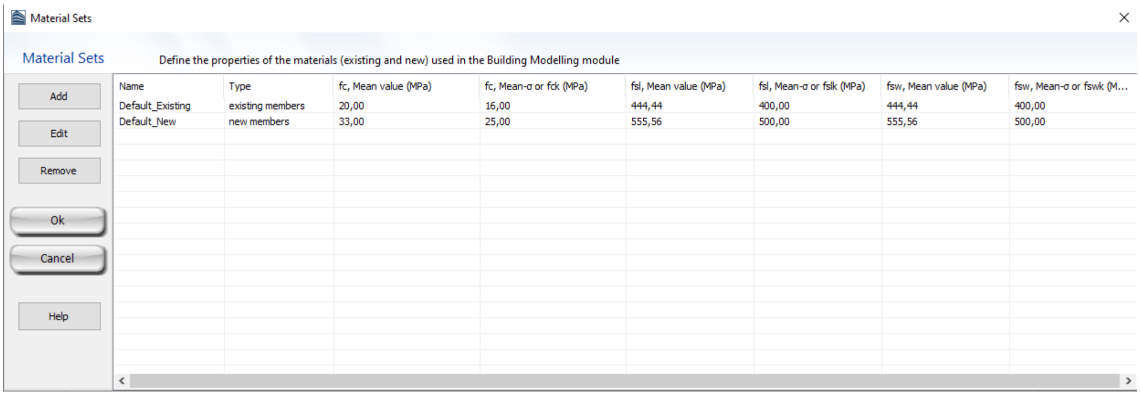
New Floor & Background

Inserting Structural Members

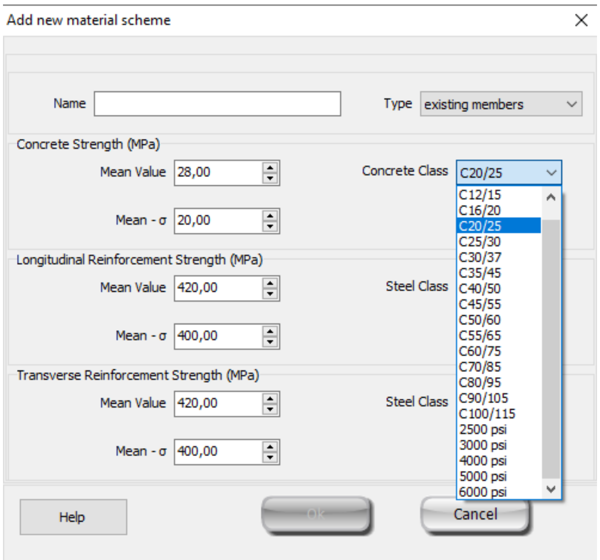
The Material Sets, the member loading, the Advanced Member Properties and the Modelling Parameters are common to all the sections' properties windows while FRP Wrapping is available only for columns and Jacket is available for all members apart from walls. Note that a How-To documents list is introduced for a quick access to all the required information regarding modelling within the Building Modeller.

Material Sets

The material sets properties can be defined from the main menu (*Tools > Define Material Sets*), through the corresponding toolbar button, or through the Define Material Sets button within the member's properties window. The required values for the definition of the materials properties depend on the type of the members, i.e. existing or new members. For existing materials, the mean strength value and the mean strength value minus the standard deviation are required, whereas for new materials the characteristic strength value and the mean strength value should be assigned. By default, there are two material schemes, one for the existing elements and one for the new ones. Users may modify the values of the default sets, but they can also add new material sets to cover the needs of their model (e.g. when different material strengths are employed).



Material Sets Window



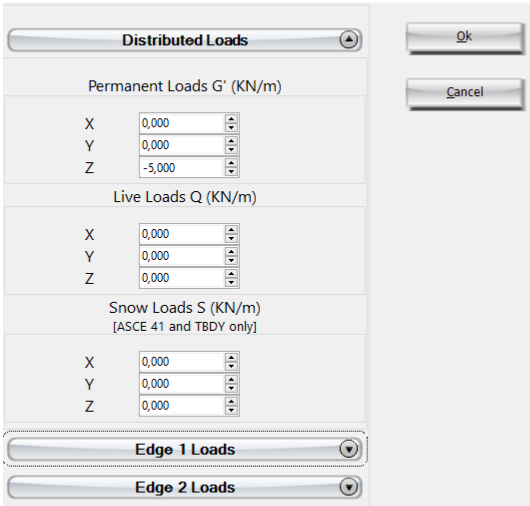
Add New Material Scheme

NOTE 1: There is a limit to the number of the defined material schemes equal to 10. The default material sets cannot be removed.

NOTE 2: The option of applying predefined material strengths, depending on the year of construction of the building, is available when this is allowed from the selected Code.

Member Loading

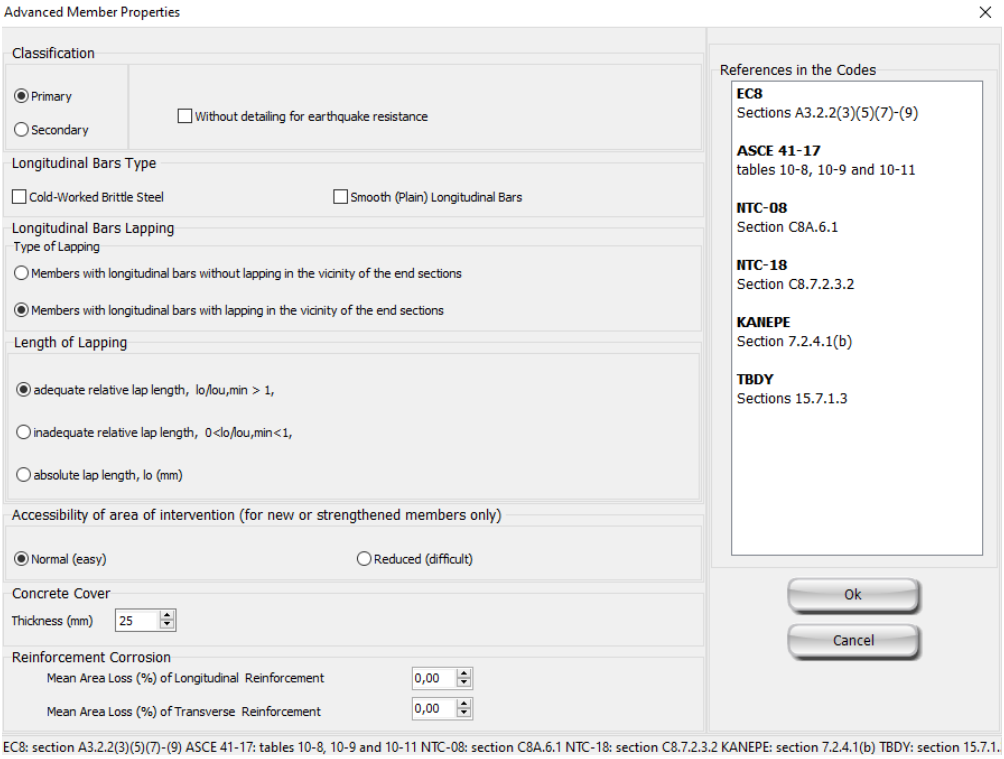
Additional loads can be defined by clicking on the Distributed and Edge Loads button for columns and walls and the More Loads button for beams. Users can define uniform distributed forces along the length of the member in all three translational directions X, Y or Z, and forces or moments in any translational or rotational direction (X, Y, Z, RX, RY or RZ) at either of the two edges of the member. Additional permanent loads G' (not associated with the self-weight of the structure), live Q and snow S loads may be applied, with the latter being applicable only to ASCE 41 and TBDY. By default, all loads are equal to zero.



Distributed and Edge Loads window

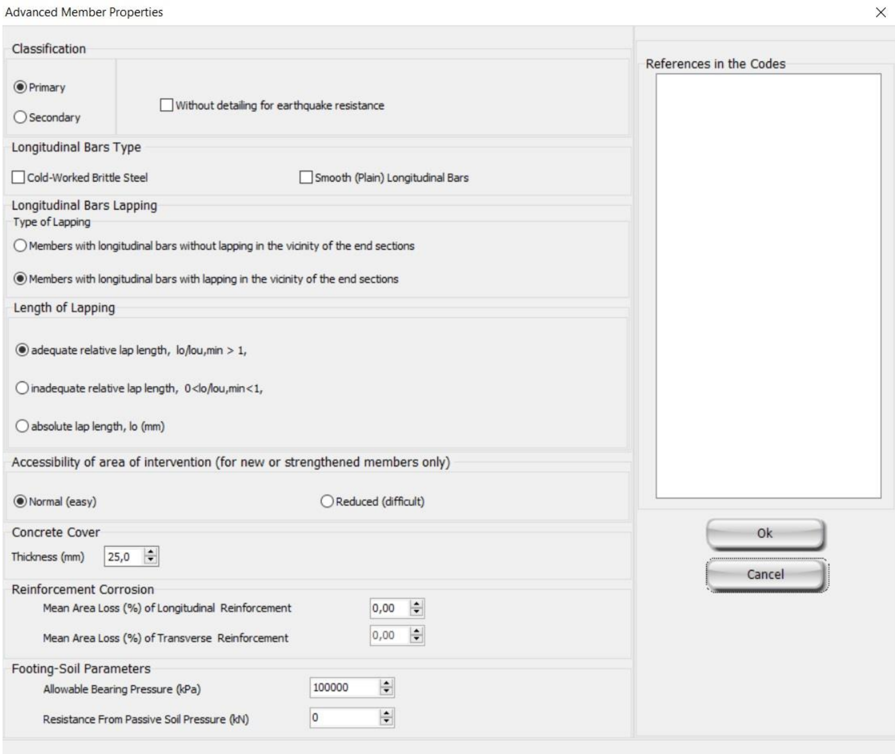
Advanced Member Properties

The member's code-based settings may be defined from the Advanced Member Properties dialog box accessed by the Properties Window. Herein, users may determine the element's classification (i.e. primary or secondary seismic member), whether it is with or without detailing for earthquake resistance, its cover thickness, the type of the longitudinal bars (cold-worked brittle steel and smooth (plain) longitudinal bars may be assigned), the type and length of lapping for the longitudinal bars, the accessibility of area of intervention, as well as the mean percentage of the area loss of longitudinal and transverse reinforcement due to corrosion (the last two are needed for the Greek Seismic interventions Code only). It is noted that the length of lapping may be defined in three ways; (i) the members have adequate relative lap length, compared with the minimum lap length for ultimate deformation (default option); (ii) the members have inadequate relative lap length (the ratio between the applied lap length and the minimum lap length for ultimate deformation should be defined); and (iii) the members have inadequate lap length (the absolute lap length should be assigned).



Advanced Member Properties module

For Foundation Members (i.e. Individual Footings and Strip Footings) two more parameters need to be defined namely the Allowable Bearing Pressure and the Resistance from Passive Soil Pressure.



Footing Modelling Parameters

Modelling Parameters

The member's modelling parameters may be defined from the Modelling Parameters dialog box, accessed by the Properties Window. Herein, users may define the concrete and steel material types and the frame element type that will be used to model the structural member in SeismoBuild, together with other modelling options, such as the number of sections fibres and the assignment of Moment/Force releases.

Materials and frame element types that are to be used within a SeismoBuild project come defined in the Advanced Building Modelling tab of the Advanced Settings module. The choices made in the Advanced Building Modelling tab are the "Default" options within the Member Modelling Parameters tab.

Fourteen material types are available in SeismoBuild, six types for concrete and eight for steel. The complete list of materials is proposed hereafter:

- Mander et al. nonlinear concrete model - con_ma
- Trilinear concrete model - con_tl
- Chang-Mander nonlinear concrete model – con_cm
- Kappos and Konstantinidis nonlinear concrete model - con_hs
- Engineered cementitious composites material – con_ecc
- Kent-Scott-Park concrete model – con_ksp
- Menegotto-Pinto steel model - stl_mp
- Giuffre-Menegotto-Pinto steel model - stl_gmp
- Bilinear steel model - stl_bl
- Bilinear steel model with isotropic strain hardening- stl_bl2
- Ramberg-Osgood steel model - stl_ro
- Dodd-Restrepo steel model – stl_dr
- Monti-Nuti steel model - stl_mn
- Buckling Restrained Steel Brace model – stl_brb

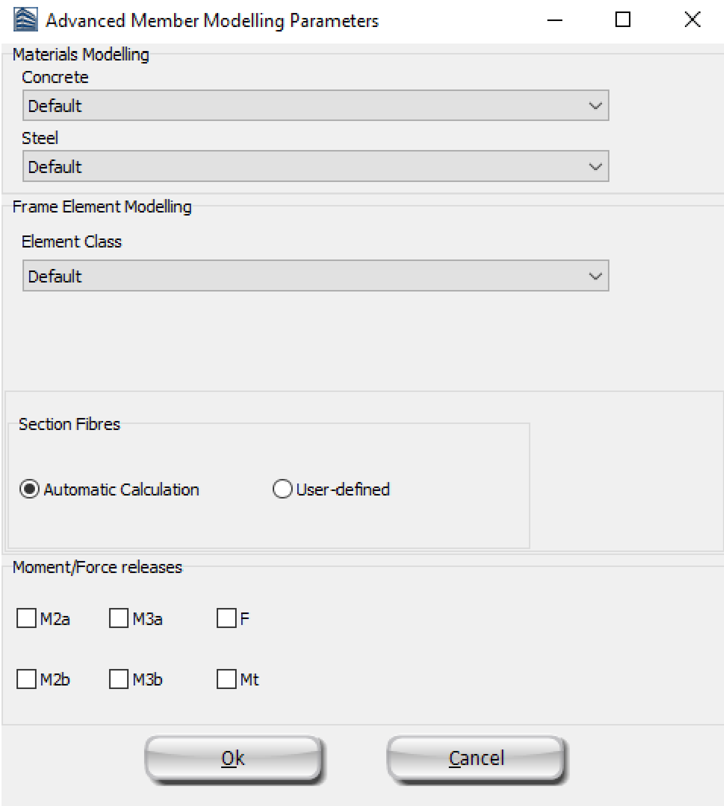
For a comprehensive description of the material types, refer to Appendix C – Materials.

Different frame element types may be employed within the structural members. Users may select between inelastic force-based frame elements (infrmFB), inelastic plastic-hinge force-based frame elements (infrmFBPH), inelastic plastic-hinge displacement-based frame elements (infrmDBPH), inelastic displacement-based frame elements (infrmDB) and elastic frame elements (elfrm). The inelastic displacement-based frame element type (infrmDB) is suggested to be employed for short members, a choice that improves both the accuracy and the stability of the analysis.

NOTE: Code based checks are not executed for the member of the elastic frame element type (elfrm). Hence, this element type may be employed only for special modelling cases, when an elastic member behaviours is expected.

Further, the number of section fibres used in equilibrium computations carried out at each of the element's integration sections needs to be defined. User may assign the number of fibres of their choice or they may select the automatic calculation, according to which 50 fibres are defined for a member's concrete area less than $0.1m^2$ and 200 fibres for a member's concrete area more than 1m2, whereas linear interpolation is executed for the in between values. Each longitudinal reinforcement bar is defined with 1 additional fibre; added to the abovementioned concrete number of fibres.

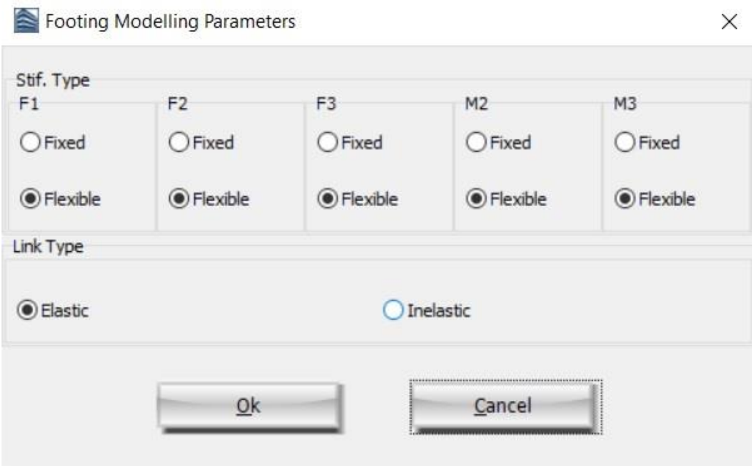
Finally, users may also 'release' one or more of the element degrees of freedom (forces or moments).



Modelling Parameters module

Footing Modelling Parameters

The modelling parameters for the foundation members can be specified in the Advanced Model Parameters for Foundation members. In particular, for Individual Footings and Strip Footings where the connection to the ground is modelled using links the type of Link (Elastic or Inelastic) to be used can be specified and there is also the option to Fix or define as Flexible any of the six degrees of freedom of the link.

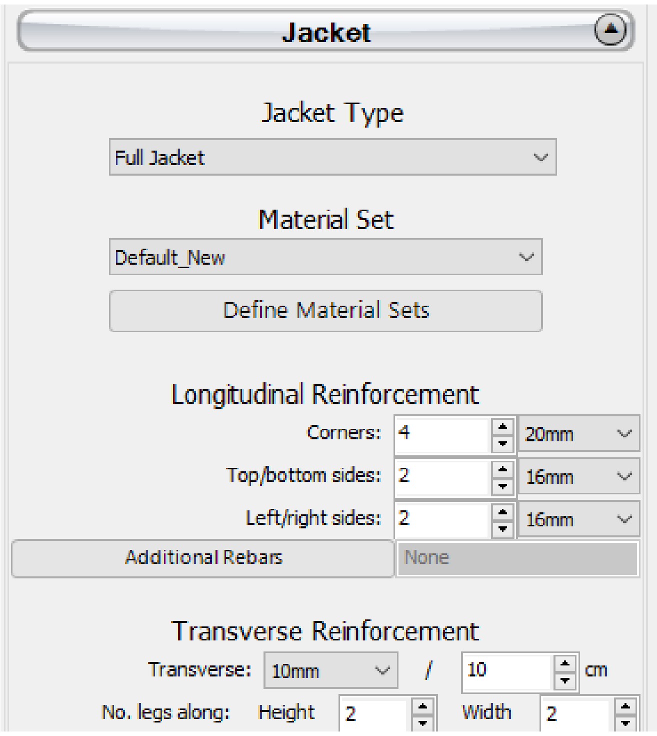


Footing Modelling Parameters

Jacket

Reinforced Concrete Jacket may be assigned to columns and beams through the Jacket module. Users, depending of the section, may select to insert full, 3-sided, 2-sided or 1-sided jacket.

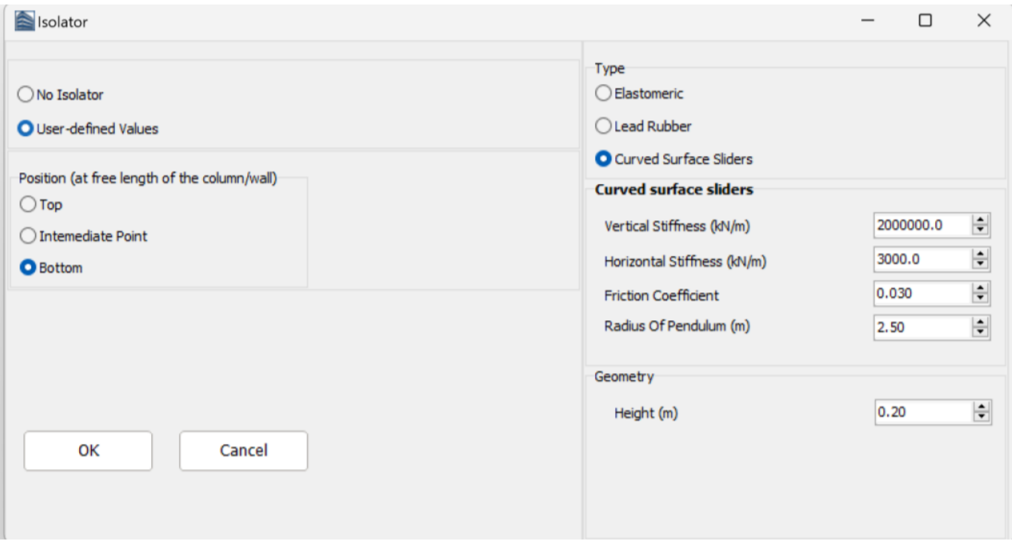
The material set of the jacketed sections, as well as the longitudinal and transverse reinforcement of the jacket may be defined.



Jacket module

Isolator

Isolators can also be added at different locations of the column and wall sections. They are assigned to columns/walls through the Isolator module, where the users may select the geometry (location -bottom, top or intermediate point- and the height of the isolator), its type (elastomeric, lead rubber or curve surface slider) and the isolator parameters: the vertical and horizontal stiffnesses, and the shear yield strength and the strain hardening ratio (for elastomeric and lead rubber isolators) or the friction coefficient and the radius of the pendulum (for curve surface sliders).

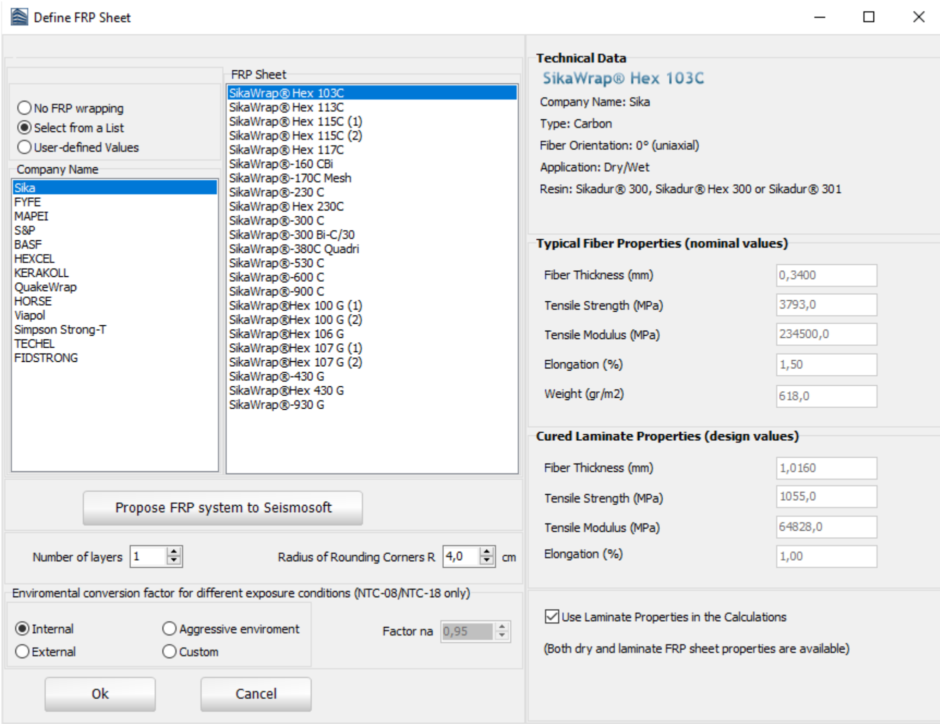


Isolator module

FRP Wrapping

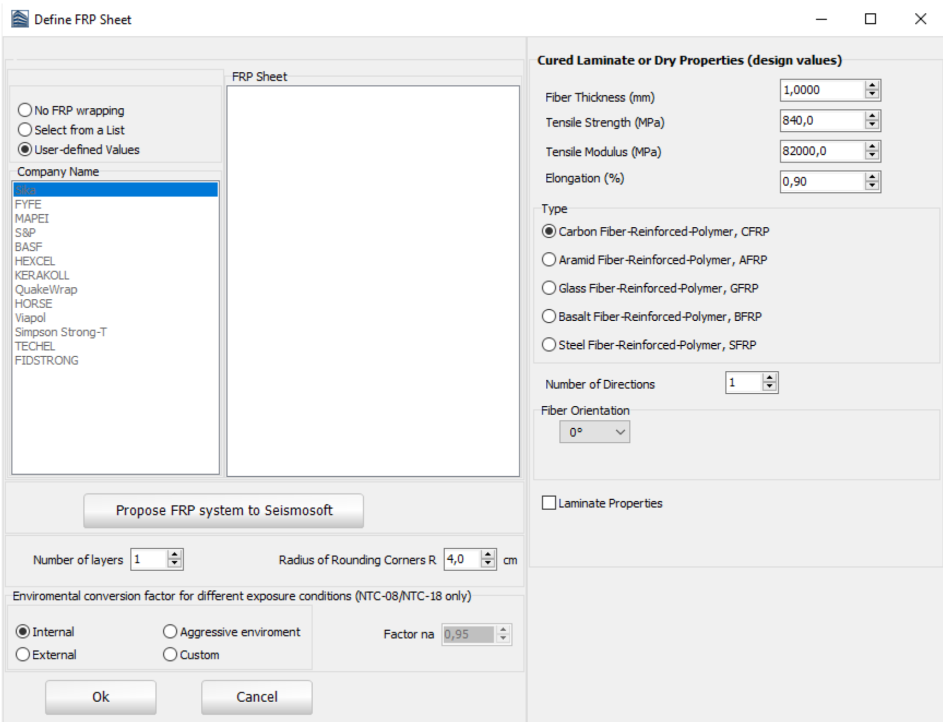
FRP wraps may be assigned to columns through the FRP Wrapping module. Users may select the FRP sheet from a list of the most common products found in the market, or alternatively introduce user-defined values.

The number of applied layers may also be defined, as well as whether the dry or the laminate FRP properties are to be used in the calculations. Finally, for the rectangular cross sections the radius of rounding of the corners R may be specified, a critical parameter in the application of FRP wraps.



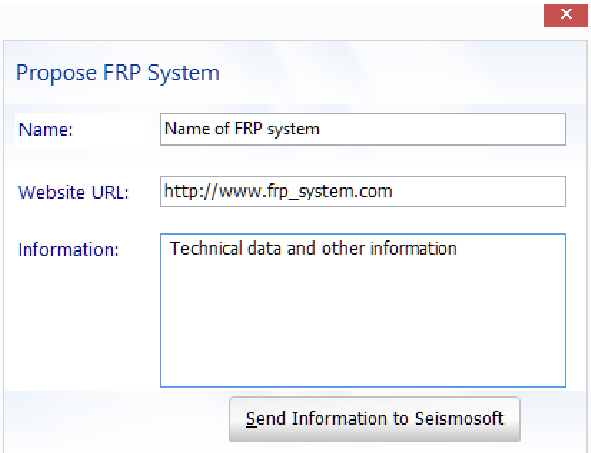
Select from a list module

When users choose to specify user-defined values, the required information is the type of the FRP sheet (Carbon, Aramid, Glass, Basalt or Steel fibres), its laminate or dry properties, the number of direction(s) and the orientation (relatively to the longitudinal direction of the sheet) of the fibres, as well as the number of layers and the radius of rounding corners R.



User-defined Values module

Finally, FRP systems may be proposed to Seismosoft through the "Propose FRP system to Seismosoft" button. Herein, the user is asked to assign the name of the FRP system, the link where information about the product may be found and the technical properties of the FRP sheet.



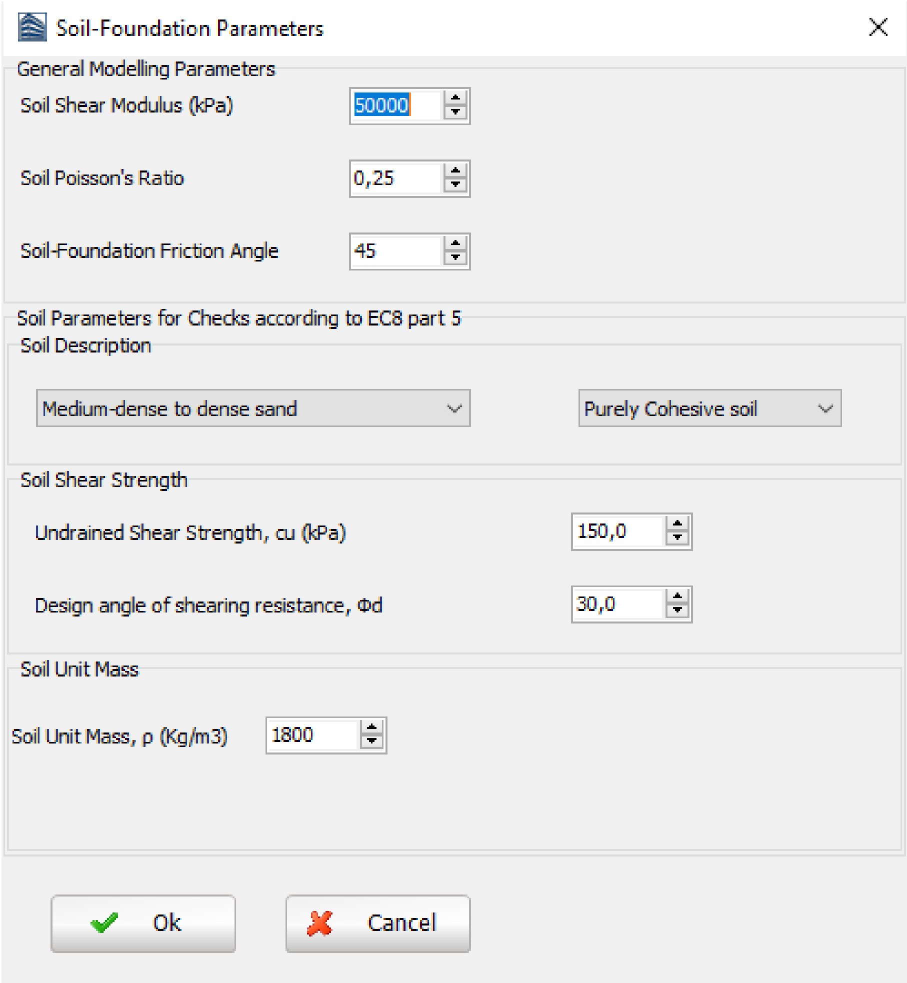
Propose FRP System window

Soil Foundation Parameters

When working in the Foundation Mode the user can define the Soil Foundation Parameters. After selecting the button on the main toolbar the Soil Foundation Parameters will appear, where the properties below can be explicitly defined:

(1) General Modelling Parameters
- Soil Shear Modulus
- Soil Poison's Ratio
- Soil Foundation Friction Angle

(2) Soil Parameters for Code based checks according to EC8 part 5 (i.e. employed in Eurocode, NTC & KANEPE)
- Soil Description
- Soil Shear Strength Parameters that include:
  a. the Undrained Shear Strength, cu, and,
  b. the Design angle of shearing resistance, Φd
- Soil Unit Mass



Define Soil Foundation Parameters Window

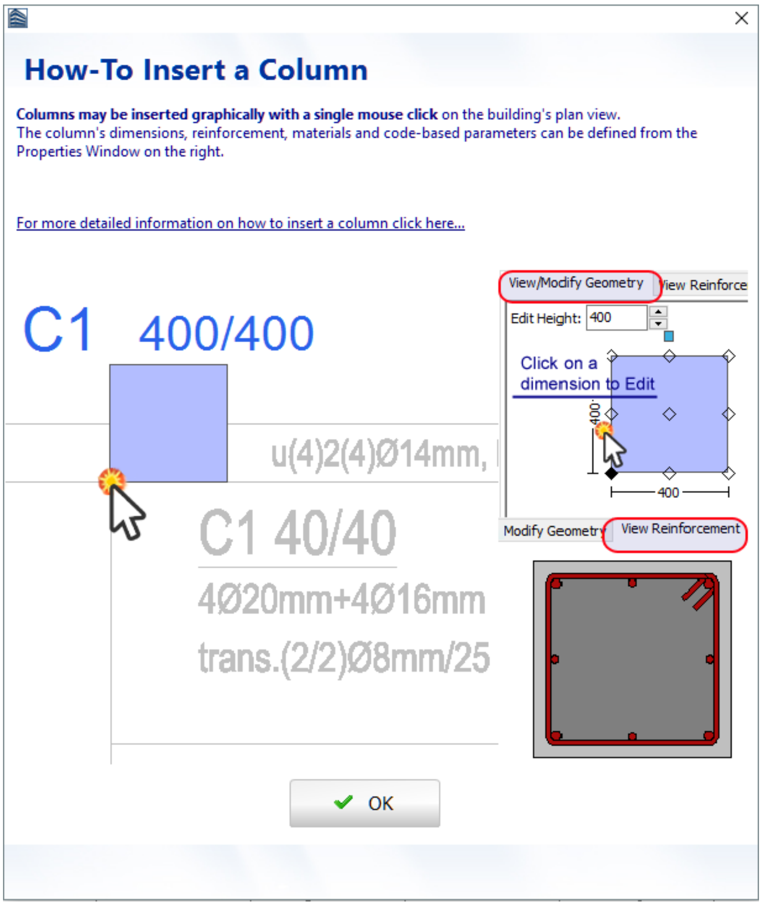
Column Members

The columns can be inserted from the main menu (Insert >...) or through the corresponding toolbar buttons. The column's Properties Window will appear where the properties below can be explicitly defined:

(i) Geometry, i.e. the dimensions (height and width)
(ii) Boundary Conditions, i.e. if it is full length or free length, assigning the length difference in the last case and the foundation level
(iii) Materials
(iv) Loading
(v) Reinforcement
(vi) Jacket
(vii) Isolator
(viii) FRP Wrapping
(vii) Advanced Modelling, which includes the advanced member properties and the modelling parameters

The column members may be inserted in the project with a single mouse click.

Once the Insert a Column command is selected, an informative message appears providing brief information of how to insert a column.



How-To Insert a Column window

Currently, thirteen section types are available in SeismoBuild:

- Rectangular Column
- L-Shaped Column
- T-Shaped Column
- Circular Column
- Rectangular Jacketed Column
- Rectangular 3-sided Jacketed Column
- Rectangular 2-sided Jacketed Column
- Rectangular 1-sided Jacketed Column
- L-Shaped Jacketed Column
- L-Shaped 3-sided Jacketed Column
- T-Shaped Jacketed Column
- T-Shaped 3-sided Jacketed Column
- Circular Jacketed Column

For a comprehensive description about the insertions of columns in the Building Modeller refer to Appendix D - Inserting Structural Members.

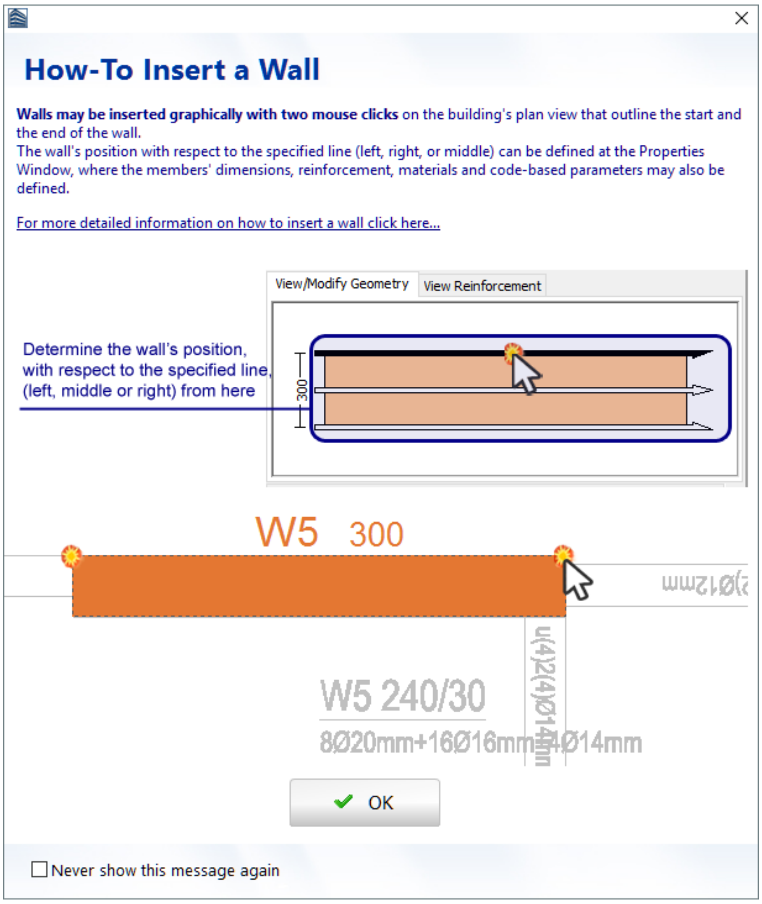
Wall Members

The walls can be inserted from the main menu (Insert >...) or through the corresponding toolbar button. The wall's Properties Window will appear where its properties are explicitly defined in the similar way to the columns. The walls may be inserted in the project by defining their edges; only two mouse clicks are needed.

Currently, the following types are available in SeismoBuild:

- Wall
- Compound Wall

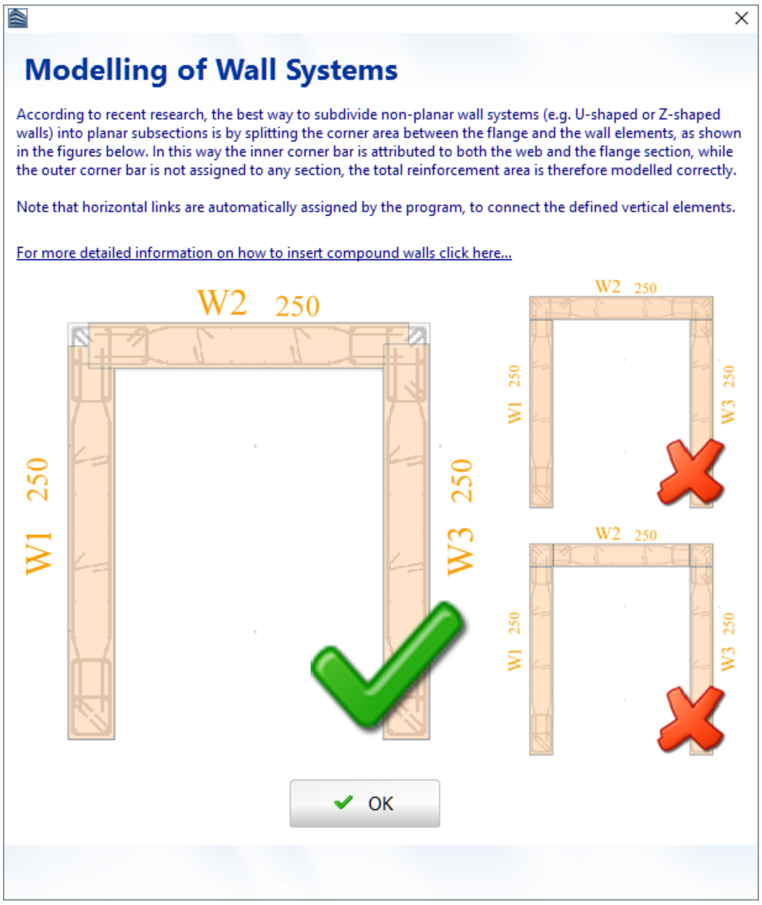
Once the Insert a Wall command is selected, an informative message appears providing brief information of how to insert a wall.



How-To Insert a Wall window

For a comprehensive description about the insertions of walls in the Building Modeller refer to Appendix D - Inserting Structural Members.

If the Insert Compound Wall toolbar button is selected, an informative window will appear proposing the best way to insert compound wall sections. According to recent research, (Beyer K., Dazio A., and Priestley M.J.N. [2008]), the best way to subdivide non-planar wall systems, e.g. U-shaped or Z-shaped walls, into planar subsections is by splitting the corner area between the flange and the wall elements. In this way the inner corner bar is attributed to both the web and the flange section, while the outer bar is not assigned to any section, the total reinforcement area is therefore modelled correctly.



Modelling of Wall Systems message

NOTE: Horizontal links are automatically assigned by the program in order to connect the defined vertical elements.

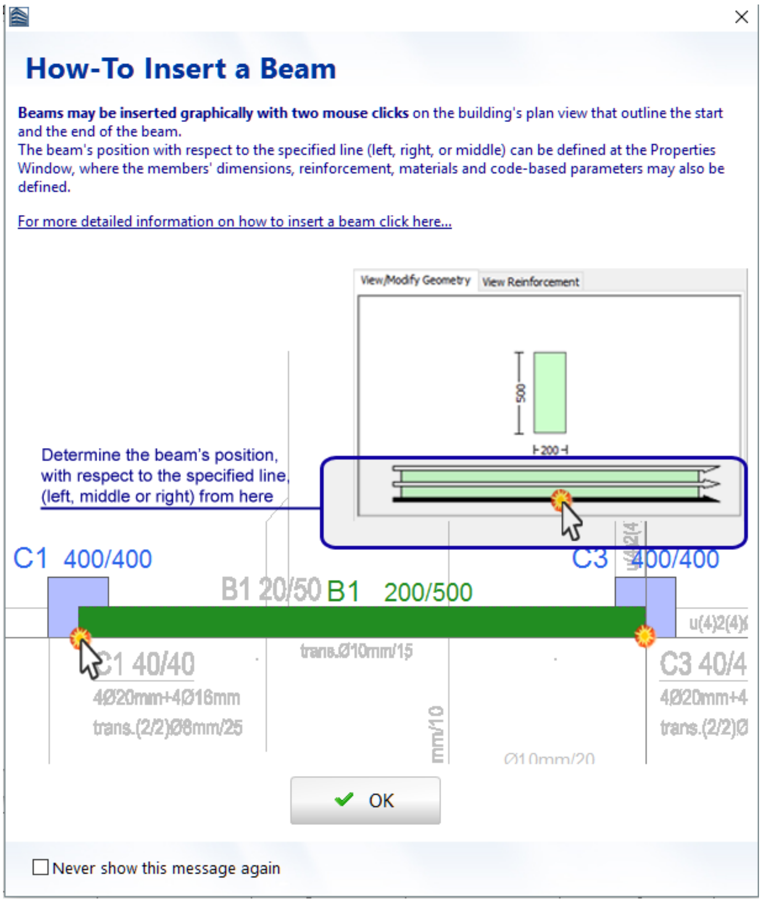
Beam Members

The beams can be inserted from the main menu (Insert >...) or through the corresponding toolbar buttons. Several additional parameters, in addition to those provided for columns, need to be specified for the correct definition of a beam, i.e. whether it is an inclined beam (in this case the height of the two ends should be specified), the additional permanent load and the reinforcement in three integration sections of the beam (in the middle and two edges). Beams may be inserted in the project by defining their edges with two mouse clicks. After assigning the beams and the slabs, the choice of including the effective width and customizing its value, as well as if the beam members will be inversed beams, may be made.

Currently, four types are available in SeismoBuild:

- Beam
- Jacketed Beam
- 3-sided Jacketed Beam
- 1-sided Jacketed Beam

Once the Insert Beam command is selected, an informative message appears providing brief information of how to insert a beam.



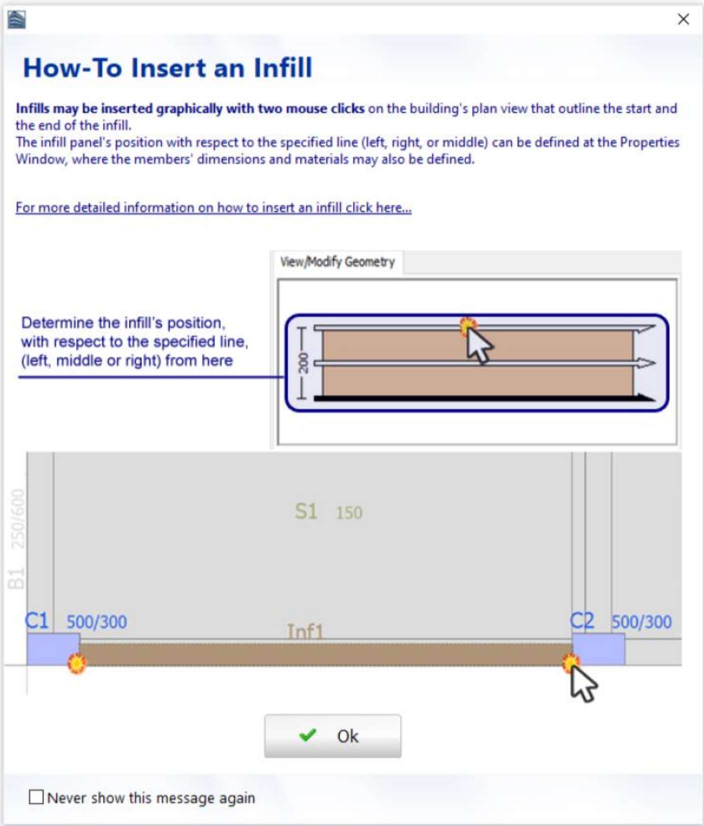
How-To Insert a Beam window

For a comprehensive description about the insertions of beams in the Building Modeller refer to Appendix D - Inserting Structural Members.

Infill Walls

The infills can be inserted from the main menu (Insert >...) or through the corresponding toolbar button. The main parameters that affect the resistance of the wall need to be specified, namely the main geometric (openings percentage and height) and mechanical characteristics of the bricks and the mortal (brick dimensions, mortar thickness, brick compressive strength, mortar compressive strength), as well as the percentage of the openings on the wall and the wall specific weight.

Once the Insert Infill command is selected, an informative message appears providing brief information of how to insert an infill.



How-To Insert an Infill window

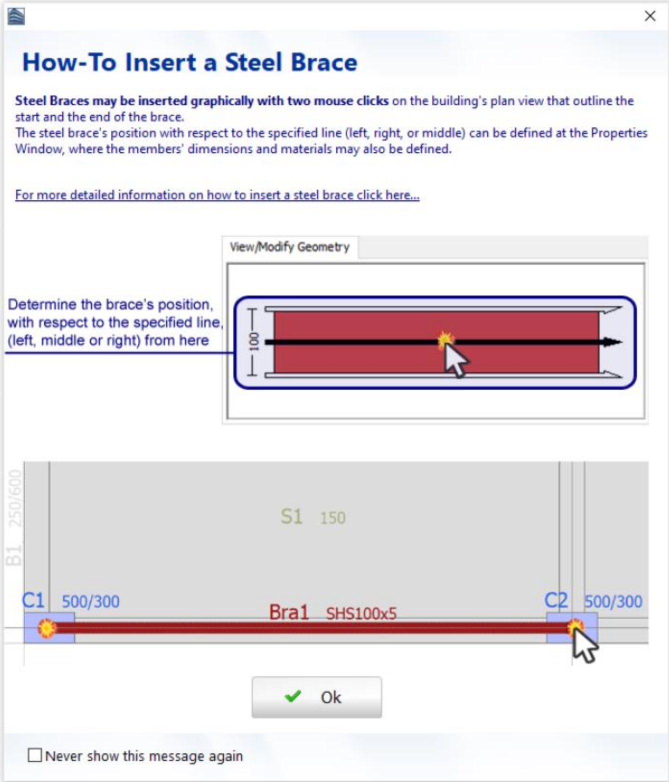
For a comprehensive description about the insertions of infills walls in the Building Modeller refer to Appendix D - Inserting Structural Members.

Steel Braces

The steel braces can be inserted from the main menu (Insert >...) or through the corresponding toolbar buttons. The Properties Window will appear where the following brace's properties can be defined:

- brace type: currently, the following types are supported: (i) X-Brace with connected diagonals, (ii) X-Brace with disconnected diagonals, (iii) diagonal brace, (iv) inverted diagonal brace, (v) V-Brace and (vi) Inverted V-Brace (Chevron Brace)
- The steel section of the brace members
- The yield strength of the brace steel
- The type of connection to the RC frame (pinned or fully fixed)
- The modelling parameters

Once the Insert Steel Brace command is selected, an informative message appears providing brief information of how to insert a brace.

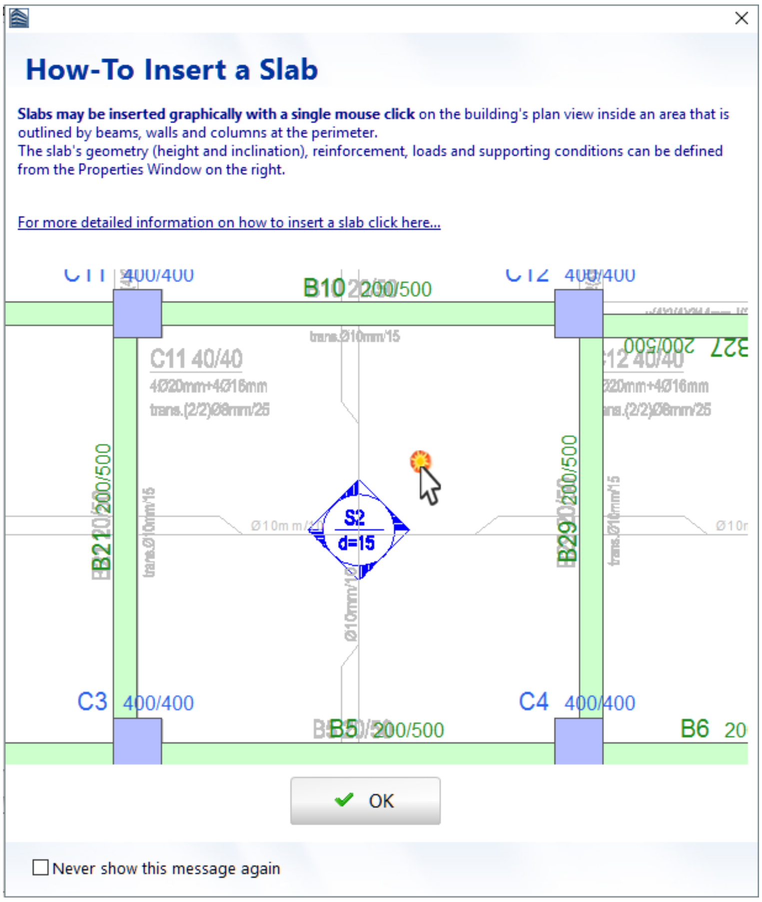


How-To Insert a Steel Brace window

For a comprehensive description about the insertions of steels braces in the Building Modeller refer to Appendix D - Inserting Structural Members.

Slabs

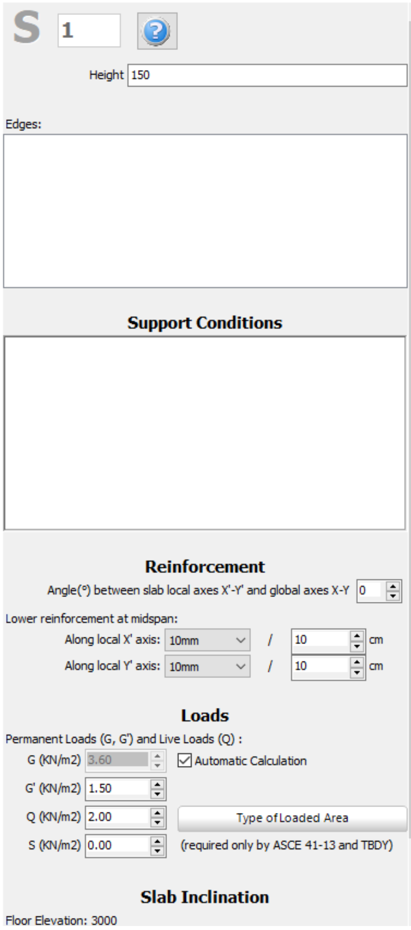
The insertion of slabs can be done through the Menu (*Insert > Insert Slab*) or by clicking the toolbar button. Prior to adding a slab, an informative message appears providing brief information of how to insert a slab.



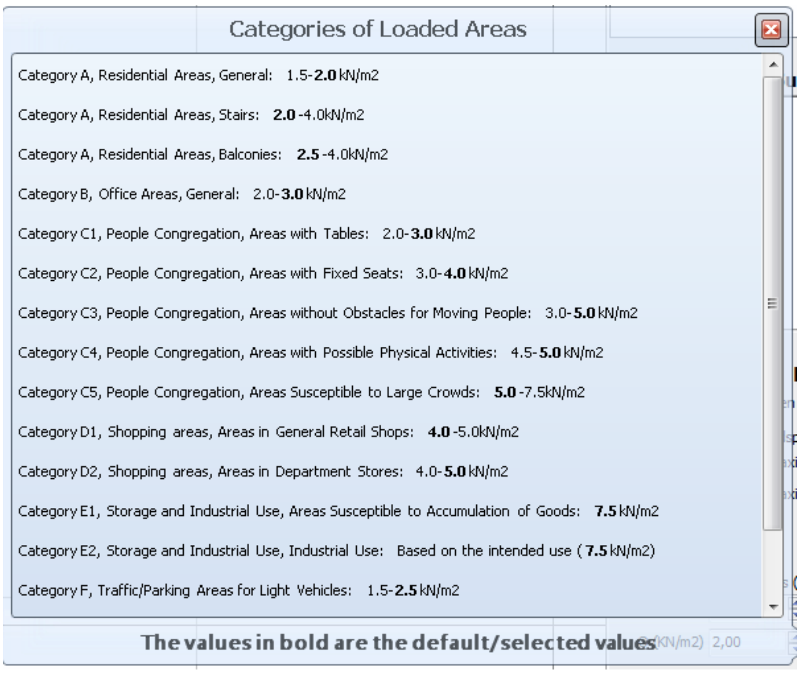
How-To Insert a Slab window

A slab can be defined with a single mouse click on any closed area surrounded by structural members (columns, walls and beams).

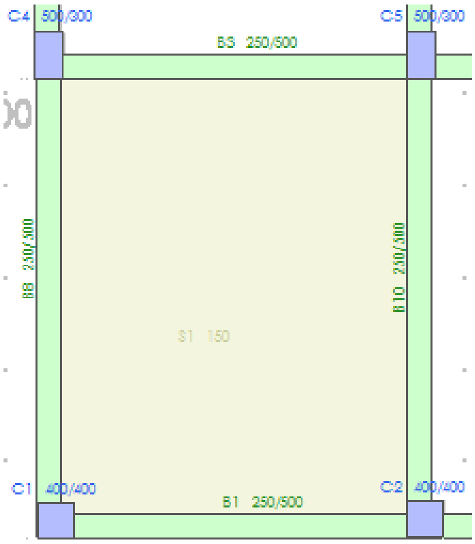
In the slab's Properties Window users can define (i) the section's height, (ii) the reinforcement and its rotation to the X & Y axes, and (iii) its self weight and the additional permanent, live and snow loads; the latter is required only by ASCE 41-23 and TBDY. The self-weight of the slabs may be automatically calculated and included in the structural model or a user-defined value may be used. The slab's live loads are automatically assigned by the program after the user selects the appropriate type of loaded area.



Slab's Properties Window

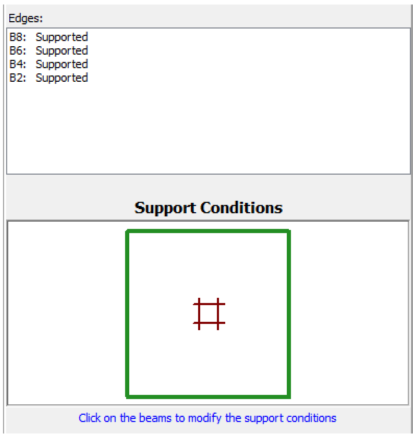


Type of Loaded Area



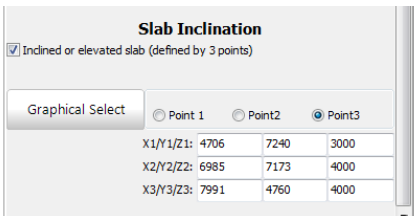
Slab insertion

After defining a slab, users may modify its support conditions, thus adjusting at which beams the slab loads are to be distributed.



Slab Support Conditions

Further the inclination of the slab may be modified, by specifying the slab elevation at three points that can be graphically selected. The neighbouring beams' elevation and column heights are automatically adjusted, whereas the columns are subdivided in shorter members by the program, if this is required, i.e. in the cases where two or more beams are supported by the same column at different levels, thus creating short columns.



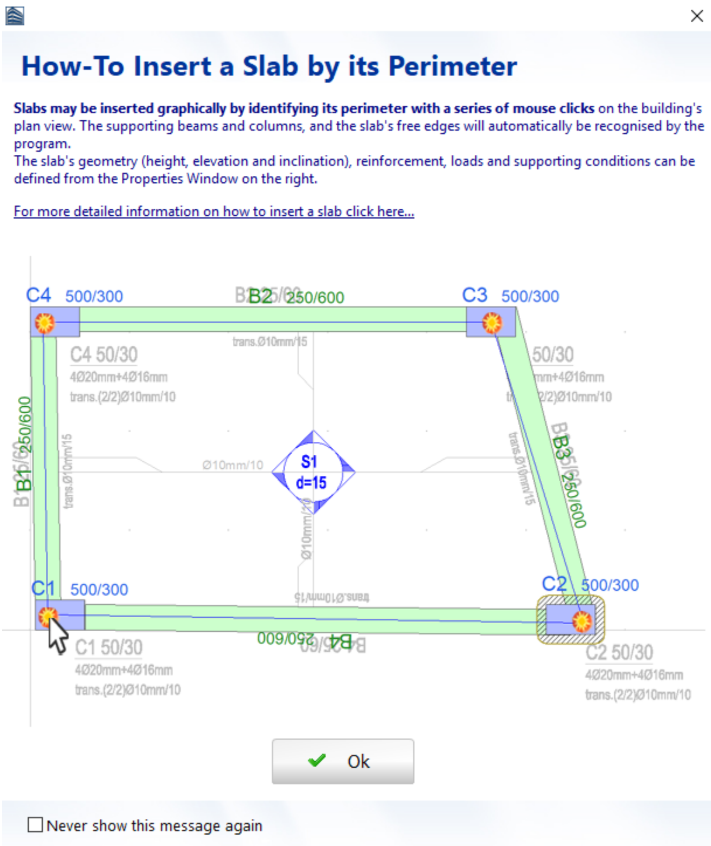
Slab Inclination

NOTE 1: The slab modelling is carried out with rigid diaphragms; hence, a rigid slab is implicitly considered in the structural configuration, which is the case for the vast majority of RC buildings. The slab's loads (self weight, additional gravity and live loads multiplied by the corresponding coefficients in the Static Actions module) are transformed to masses, based on the g value, and applied directly to the beams that support the slab.

NOTE 2: The slab reinforcement is applied at the effective width of the beams at the perimeter of the slab. Obviously, when users select not to include the effective width in the modelling, such reinforcement settings become redundant.

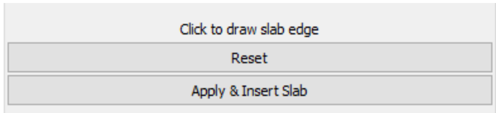
Slab by perimeter

Slabs of any geometry can be defined in the Building Modeller by selecting the *Insert > Insert Slab by perimeter* from the Menu (or through the respective toolbar button). An informative message appears providing brief information of how to insert a Slab by perimeter.



How-To Insert Slab by its Perimeter

After defining the Slab's perimeter by identifying its corners, the "Apply & Insert Slab" button should be clicked. The slab is automatically assigned.



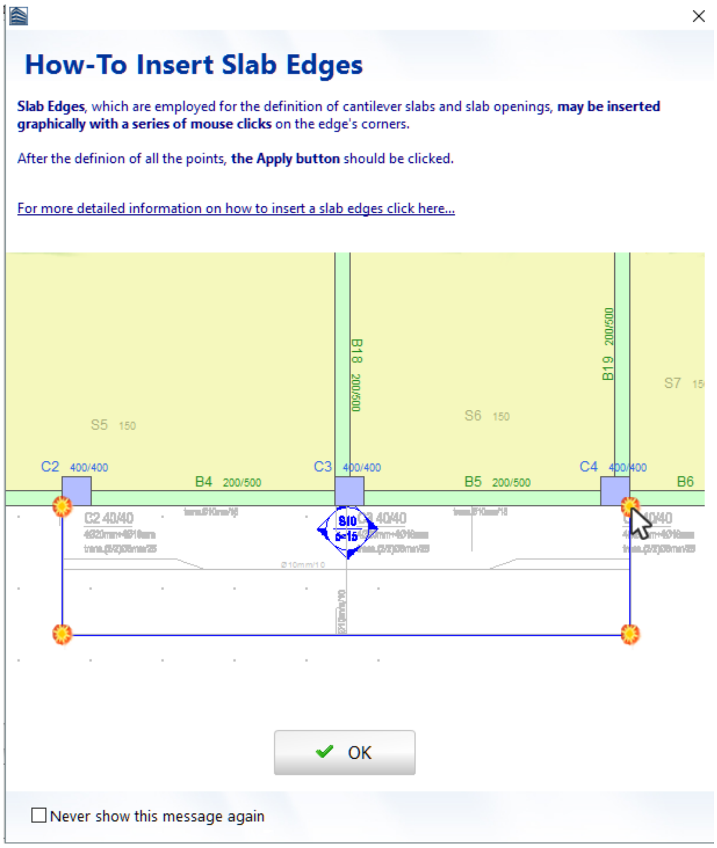
Draw Slab by perimeter

NOTE 1: Slabs are modelled in SeismoBuild as rigid diaphragms that connect the beams, columns and walls in their perimeter and as additional loads applied to the beams. Obviously, in the case of cantilevered slabs no rigid diaphragm is created and a slab is only considered as additional mass on the supporting beam; the additional mass account for the slabs' permanent and live loads.

NOTE 2: When the assigned perimeter does not define a closed area, the first point is automatically connected by the program with the last one in order to assign the new slab.

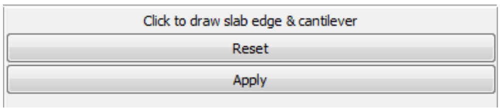
Free Edge

Cantilever slabs can also be defined in the Building Modeller. In order to do so, a Free Edge must be added from the Menu (*Insert > Insert Free edge*) or through the respective toolbar button. An informative message appears providing brief information of how to insert a Free Edge.



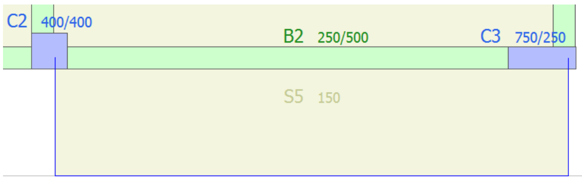
How-To Insert Slab Edges window

After defining the Free Edge's corner points, the “Apply” button should be clicked. Once drawn, the Free Edge is used to outline the shape of the slab.



Draw Free Edge

After the definition of the necessary free edges needed to define a closed area, users can insert a new slab.

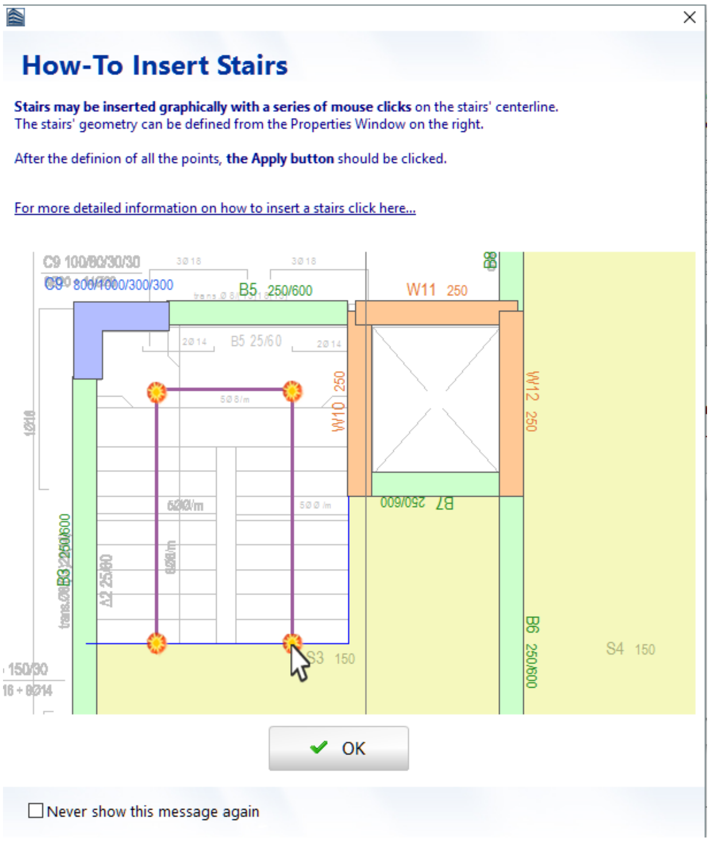


Create a new cantilevered slab

NOTE: Slabs are modelled in SeismoBuild as rigid diaphragms that connect the beams, columns and walls in their perimeter and as additional loads applied to the beams. Obviously, in the case of cantilevered slabs no rigid diaphragm is created and a slab is only considered as additional mass on the supporting beam; the additional mass account for the slabs' permanent and live loads.

Stairs

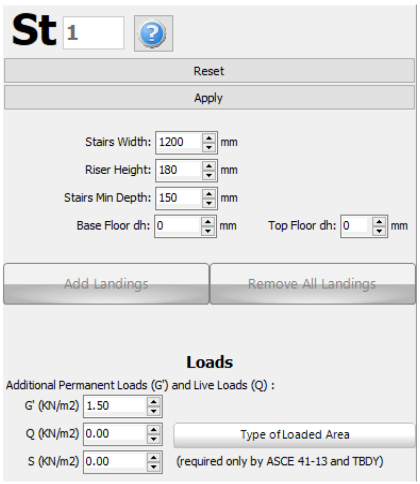
The insertion of stairs can be done through the Menu (*Insert > Stairs*) or by clicking the toolbar button. An informative message appears providing brief information of how to insert Stairs.



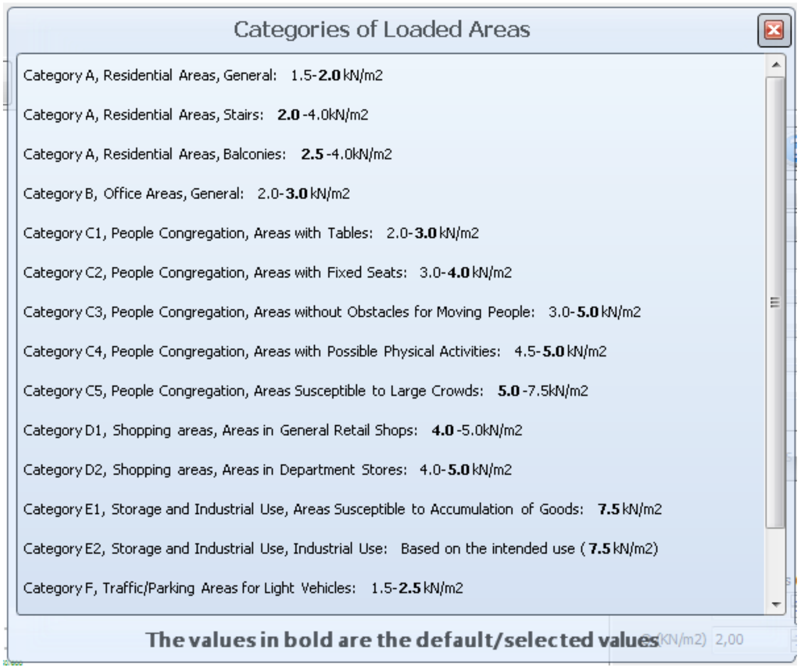
How-To Insert Stairs window

Stairs may be easily defined by specifying their centreline. Landings may be applied through the "Add Landings" button after the insertion of the stairs member in the project. The two ends of the landings need to be specified graphically on the centreline. The defined landings may be removed through the "Remove All Landings" button.

On the Properties Window users can further define the stairs' width, the riser height, the stairs minimum depth, the elevation difference relatively to the base and the top floor level, as well as the self-weight and the additional permanent, live and snow loads; the latter is required only by ASCE 41-23 and TBDY. The self-weight of the stairs may be automatically calculated according to the stairs' geometry, materials and specific weight or a user-defined value may be used.



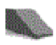
Stairs Properties Window

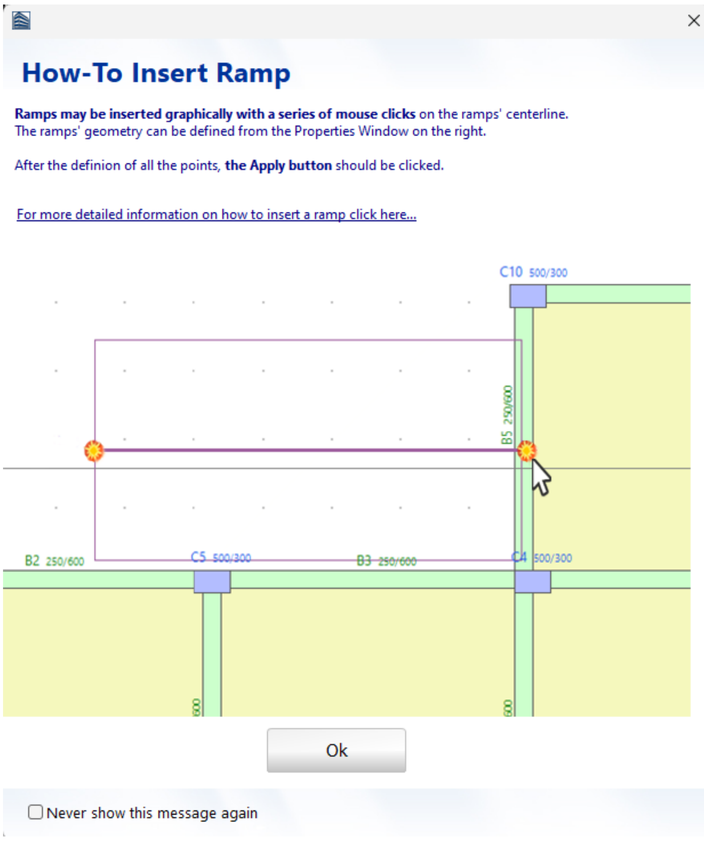


Type of Loaded Area

NOTE: Slabs are modelled in SeismoBuild with elastic elements of the specified width and depth.

Ramps

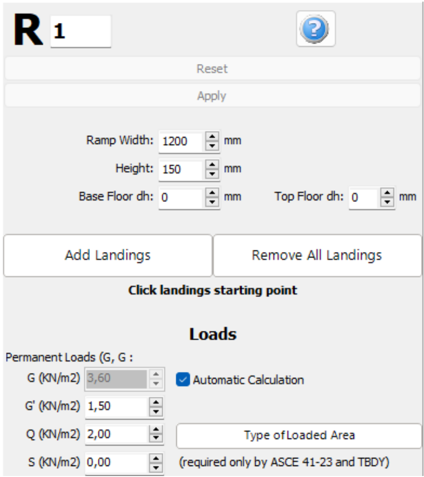
The insertion of ramps can be done through the Menu (*Insert > Ramp*) or by clicking the  toolbar button. An informative message appears providing brief information of how to insert Ramp.



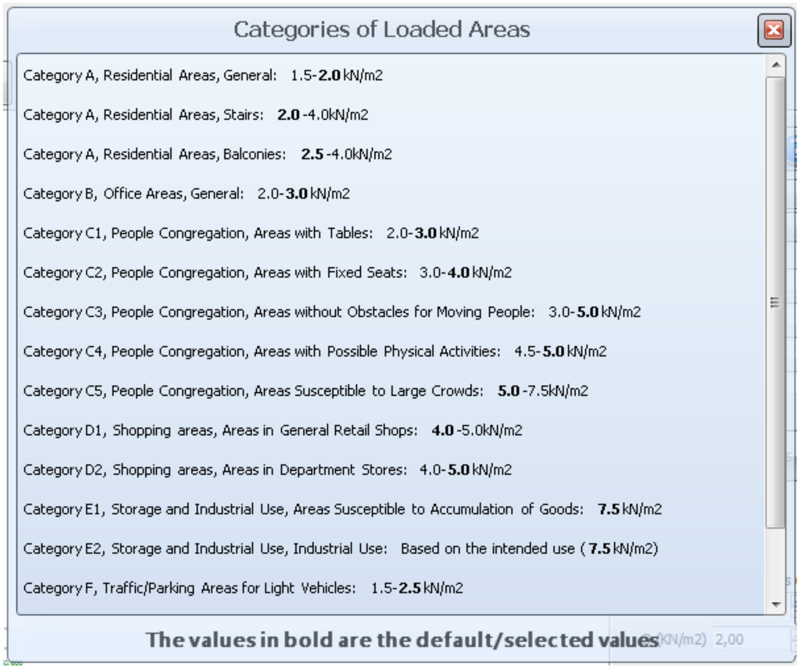
How-To Insert Ramp window

Ramps may be easily defined by specifying their centreline. Landings may be applied through the “Add Landings” button after the insertion of the ramp member in the project. The two ends of the landings need to be specified graphically on the centreline. The defined landings may be removed through the “Remove All Landings” button.

On the Properties Window users can further define the ramp’s width, the height, the elevation difference relatively to the base and the top floor level, as well as the self-weight and the additional permanent, live and snow loads; the latter is required only by ASCE 41-23 and TBDY. The self-weight of the ramp may be automatically calculated according to the ramp’s geometry, materials and specific weight or a user-defined value may be used.



Ramp Properties Window



Type of Loaded Area

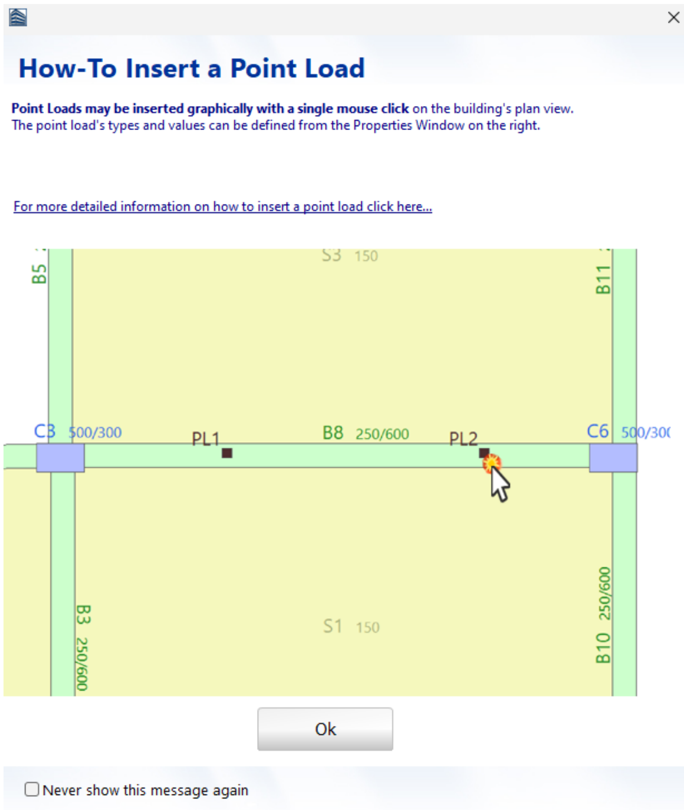
NOTE: Ramps are modelled in SeismoBuild with elastic elements of the specified width and depth.

INSERTING LOADS

In SeismoBuild loads are defined in three ways: (i) from the transformation of the masses of the structural members to loads, based on the g value, which is done automatically by the program, (ii) in the Columns, Walls, Beams, Slabs, Stairs and Ramps Properties Windows or (iii) by clicking the Insert Point Load and Insert Linear Load. For more details in the second way, users may check the corresponding member's description.

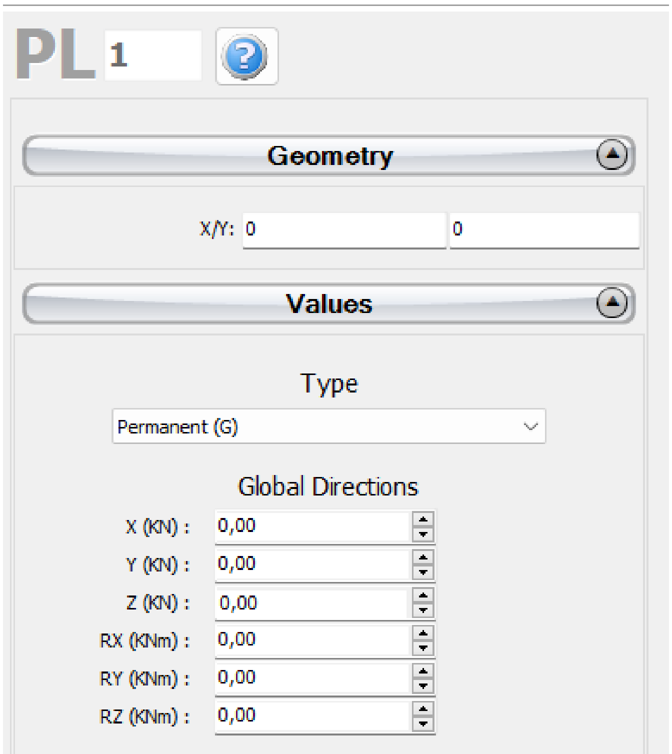
Point Loads

Two types of point loads are available. The Point loads on slabs that may be inserted from the Menu (*Insert > Point Load on Slab*), or through the corresponding toolbar button, and the Point loads on Frame Elements applied on beams, columns and walls that may be inserted from the Menu (*Insert > Point Load on Frame Elements*), or through the corresponding toolbar button,

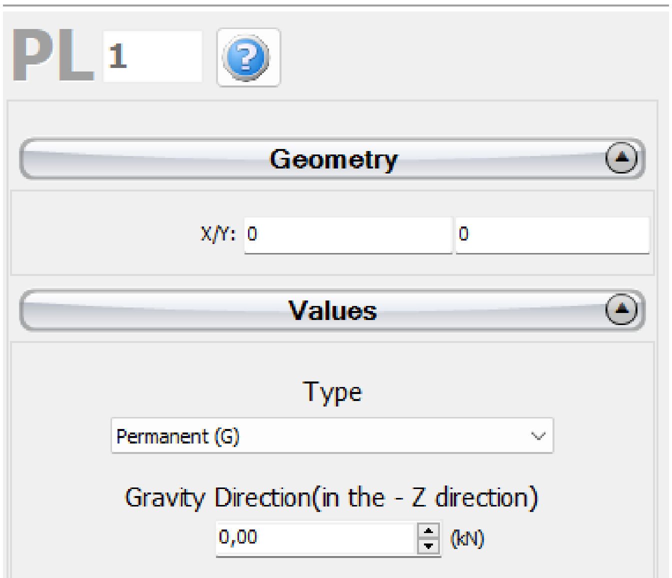


How-To Insert Point Load window

The point loads on slabs may be inserted in the gravity direction only, whereas the point loads on frame elements may be inserted in every direction, i.e. X, Y, Z, RX, RY and RZ. Permanent, live and snow loads may be assigned in both types of point loads.



Point Load on Frame Members Properties Window



Point Load on Slabs Properties Window

After defining all the point load properties, the new point load may be added graphically with a simple mouse click on the building's plan view.

After the insertion of the point load, the coordinates of the applied point and its value can be modified from the its Properties Window.

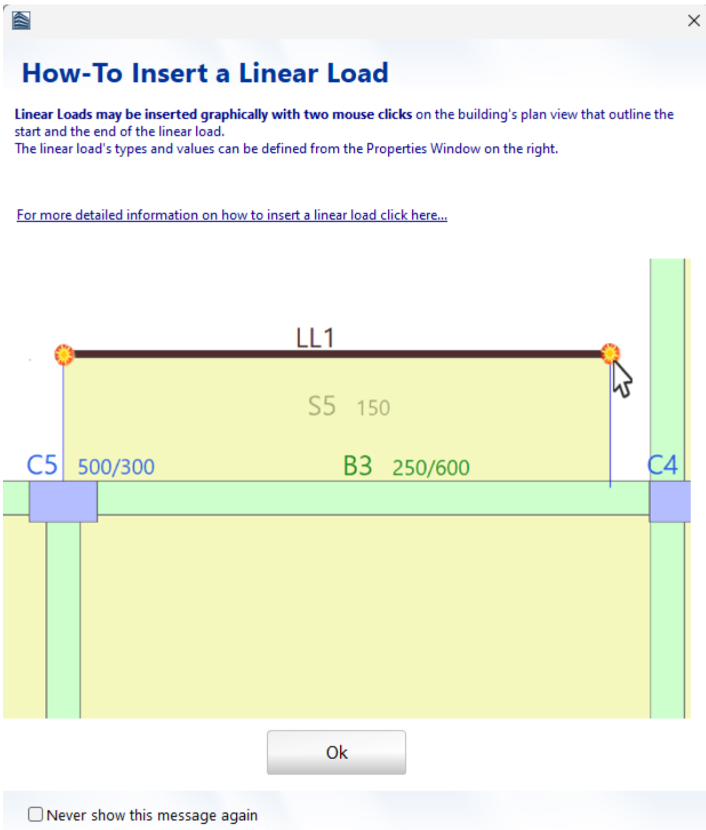
NOTE 1: When a point load is assigned to a beam, the beam is automatically subdivided, resulting in the creation of two members. If the point load is near the start or end of the beam, it is automatically applied at the beam's edge.

NOTE 2: When a point load is assigned to a column or a wall, it is automatically applied to the column's/wall's top edge at the slab's level.

NOTE 3: The slab's point loads are transformed to masses, based on the g value, and they are applied directly to the closest supporting beam according to the slab's discretization.

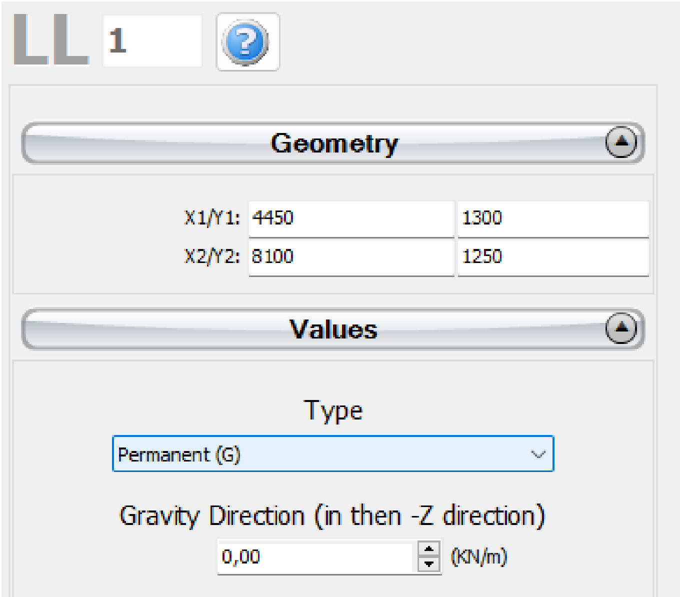
Linear Loads

Linear Loads may be inserted from the Menu (Insert > Linear Load) or through the corresponding toolbar button. An informative message appears providing brief information of how to insert Linear Load.



How-To Insert Linear Load window

Linear loads may be inserted on plan view in every reinforced concrete member and are applied in the gravity direction. Permanent, live and snow loads may be assigned.



Linear Load Properties Window

After defining all the linear load properties, the new linear load may be added graphically with two mouse clicks on the building's plan view that outline the start and the end of the linear load.

After the insertion of the linear load, the coordinates of the start and the end point of the linear load and its value can be modified from the its Properties Window.

NOTE: Linear loads are transformed to masses, based on the g value, and they are applied directly to the closest supporting element.

Inserting Foundation Members

The foundation of the superstructure can be modelled by introducing footings, connecting beams and strip footings. In order to activate the ability to insert foundation members users must select the work in the foundation mode instead of the Superstructure Mode. When the working in the Foundation Mode the vertical members of the superstructure (i.e. Columns, Walls etc) are visible so the user can define their foundation.

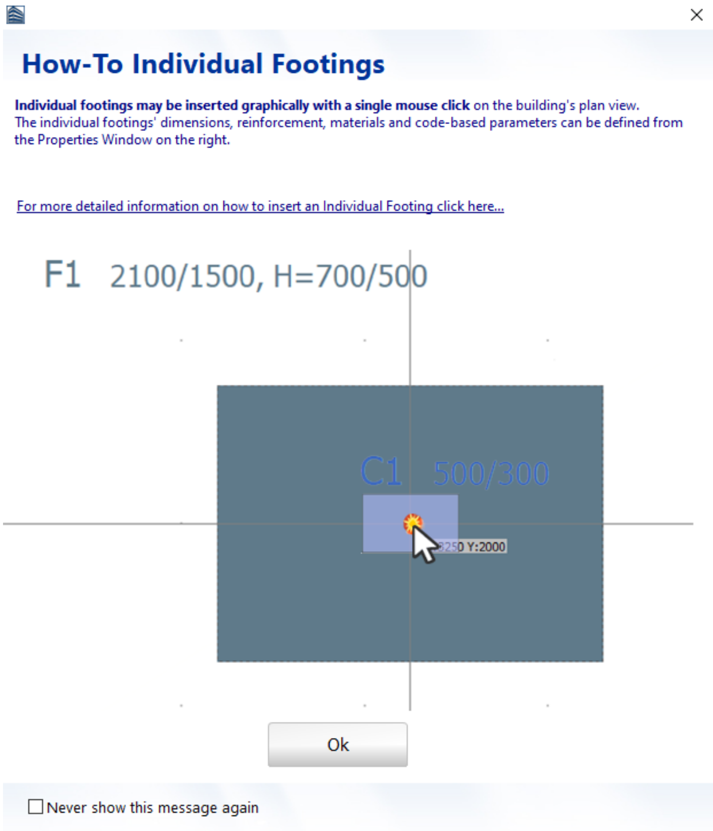
Individual Footings

The individual footings can be inserted by clicking the corresponding toolbar button . The individual footings' Properties Window will appear where the properties below can be explicitly defined:

(i) Geometry, i.e. the dimensions (height and width)
(ii) Materials
(iii) Loading
(iv) Advanced Modelling, which includes the advanced member properties and the modelling parameters

The individual footings members may be inserted in the project with a single mouse click provided that an already inserted column is entirely enclosed by the individual footing in the position where the individual footing is introduced.

Once the Insert an Individual footing command is selected, an informative message appears providing brief information of how to insert an individual footing.



How-To Insert Individual Footings window

For a comprehensive description about the insertion of Individual Footings in the Building Modeller refer to Appendix D - Inserting Structural Members.

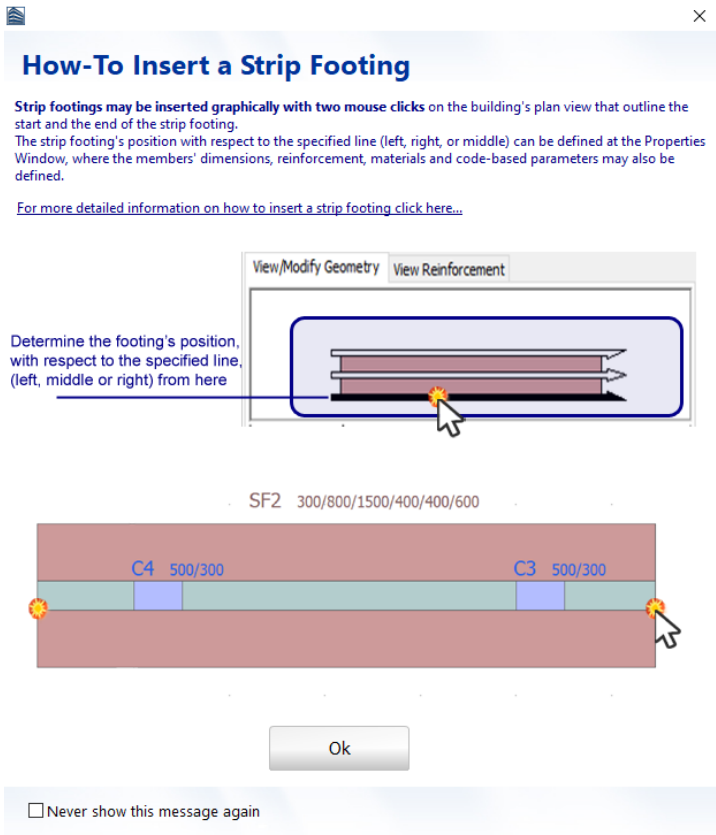
Strip Footings

The strip footings can be inserted by clicking the corresponding toolbar button . The strip footings' Properties Window will appear where the properties below can be explicitly defined:

(i) Geometry, i.e. the dimensions (height and width)
(ii) Materials
(iii) Loading
(iv) Advanced Modelling, which includes the advanced member properties and the modelling parameters

The strip footings members may be inserted in the project with two mouse clicks provided that an already inserted column is entirely enclosed by the strip footing.

Once the Insert a strip footing command is selected, an informative message appears providing brief information of how to insert a strip footing.



How-To Insert a Strip Footing window

For a comprehensive description about the insertion of strip footings in the Building Modeller refer to Appendix D - Inserting Structural Members.

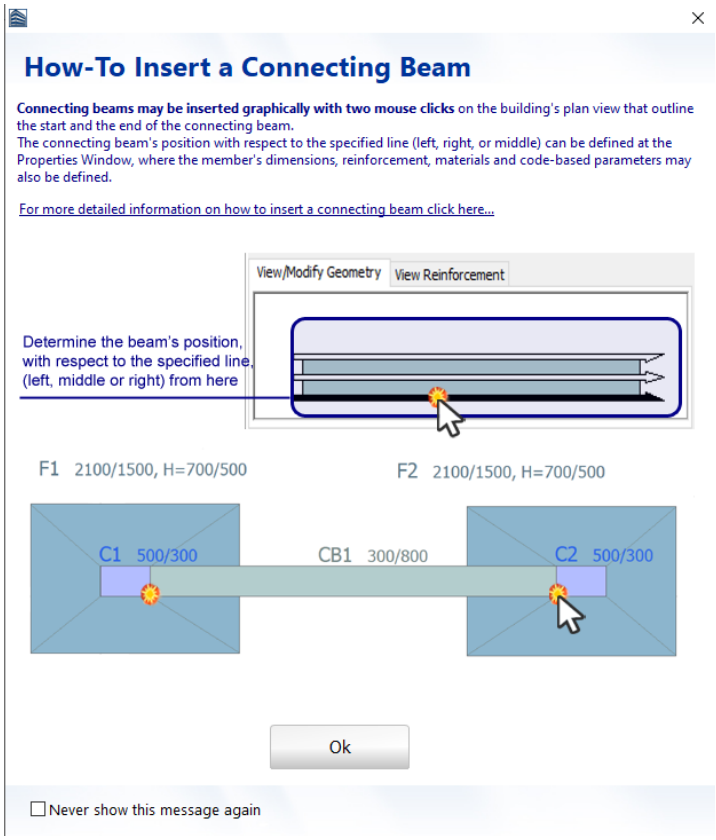
Connecting Beams

The connecting beams can be inserted by clicking the corresponding toolbar button . The connecting beams' Properties Window will appear where the properties below can be explicitly defined:

(i) Geometry, i.e. the dimensions (height and width)
(ii) Materials
(iii) Loading
(iv) Advanced Modelling, which includes the advanced member properties and the modelling parameters

The connecting beams members may be inserted in the project with two mouse clicks.

Once the Insert a connecting beam command is selected, an informative message appears providing brief information of how to insert an individual footing.



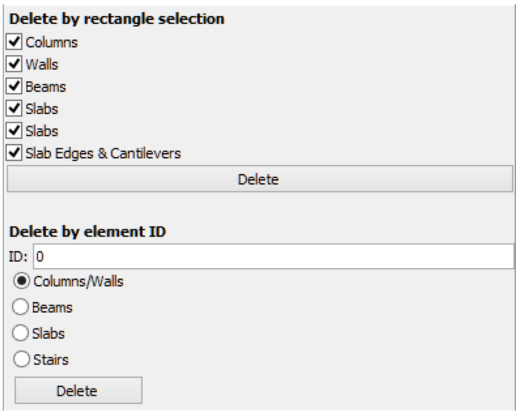
How-To Insert a Connecting Beam window

For a comprehensive description about the insertion of connecting beams in the Building Modeller refer to Appendix D - Inserting Structural Members.

Editing Structural Members

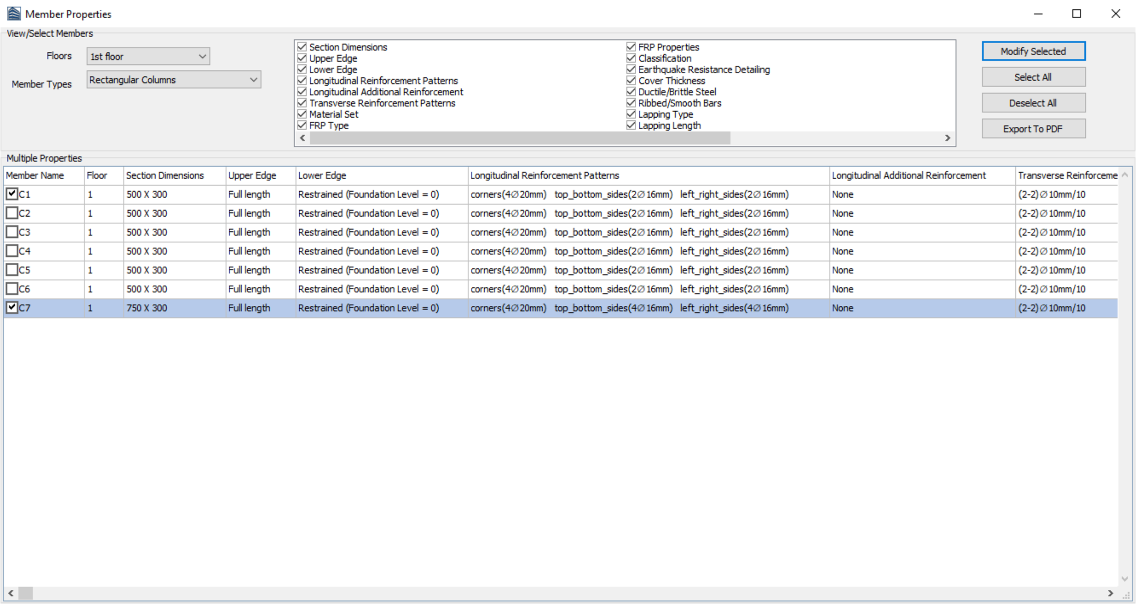
By using the edit tools from the main menu (*Tools >...*) or through the respective toolbar buttons, users can select ( ) a member to view or change its properties. Further they can move it ( ) to a different location, rotate it ( ) in plan view or delete it ( ).

It is noted that there is a number of ways to delete elements: (i) by clicking on the element (ii) by its name or (iii) by selecting a rectangular area on the Main Window.



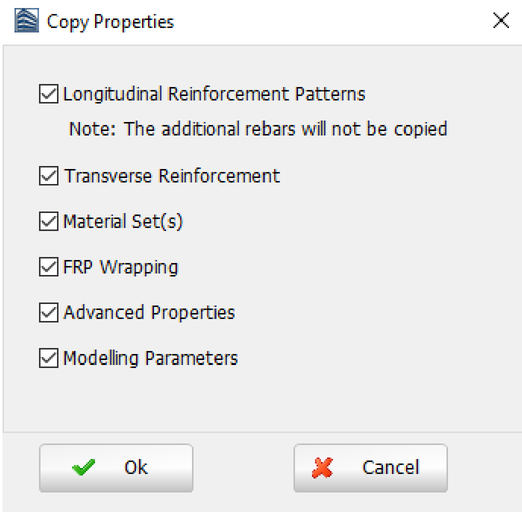
Delete element window

The option of multi-editing structural members is available from the main menu (*Tools > View/Modify Member Properties*) or through the corresponding toolbar button. Users may select multiple members of the same section type and modify their properties at once.



View/Modify Member Properties window

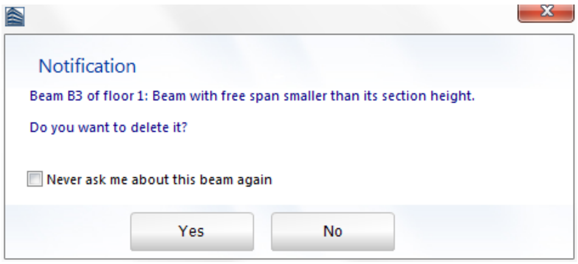
The properties of one member may be applied to others from the main menu (*Tools > Copy Member Properties*) or through the corresponding toolbar button. A window with a list of the properties that will be copied appears after the selection of the member. Users should just click on a member in order to change its properties. It is noted that the additional rebars cannot be copied.



Copy Member Properties window

Moreover, an option to renumber the structural members is offered from the main menu (*Tools > Renumber Elements*) or through the toolbar button. By clicking on a member, the selected number is assigned to it, and the numbering of all other members is changed accordingly.

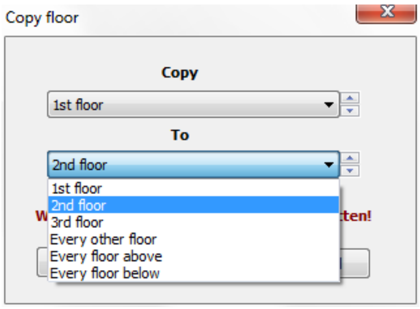
After creating a building model, it is relatively common that one or more very short beams have been created unintentionally, due to graphical reasons (e.g. by extending slightly a beam's end beyond a column edge). For this reason, a check is carried out from the main menu (*Tools > Verify Connectivity...*) or through the corresponding toolbar button for the existence of any beam with free span smaller than its section height should be carried out. If such beams exist, the following message appears, and the user can select to remove or to keep it.



Verify connectivity

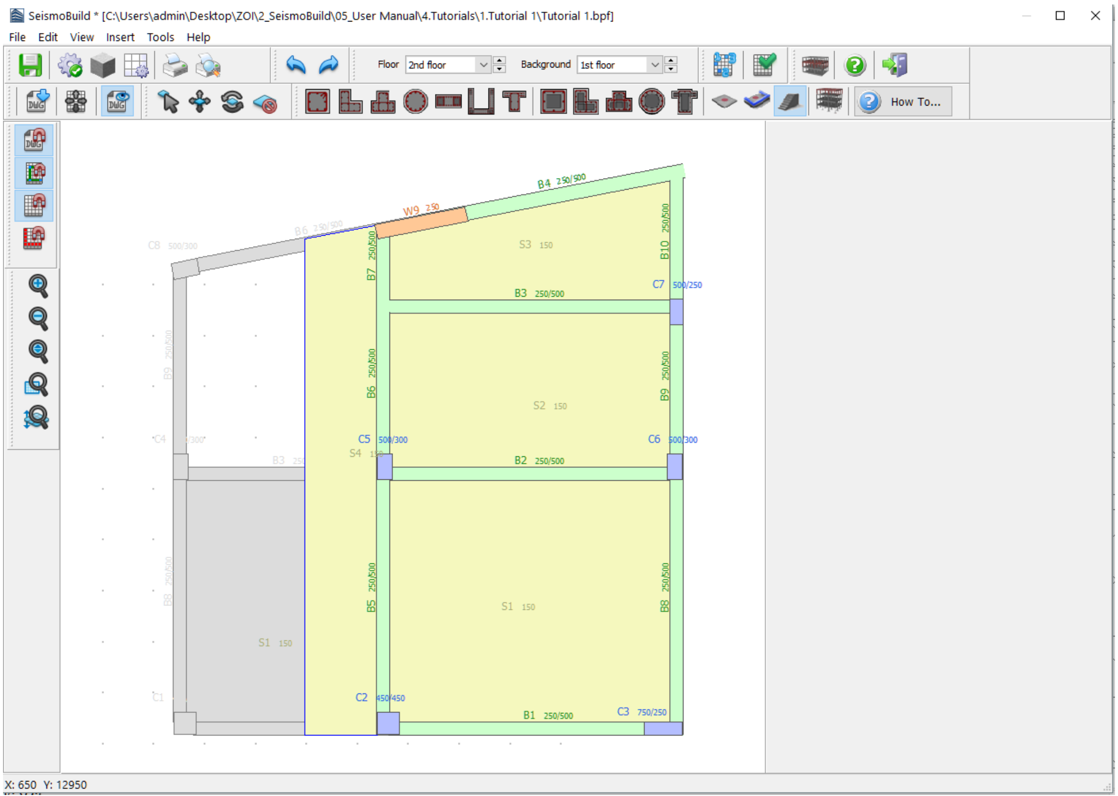
CREATING NEW STOREYS

The possibility of automatically creating new floors, based on the already created ones is offered through the main menu (*Tools > Copy Floor...*) or through the toolbar button.



Copy floor

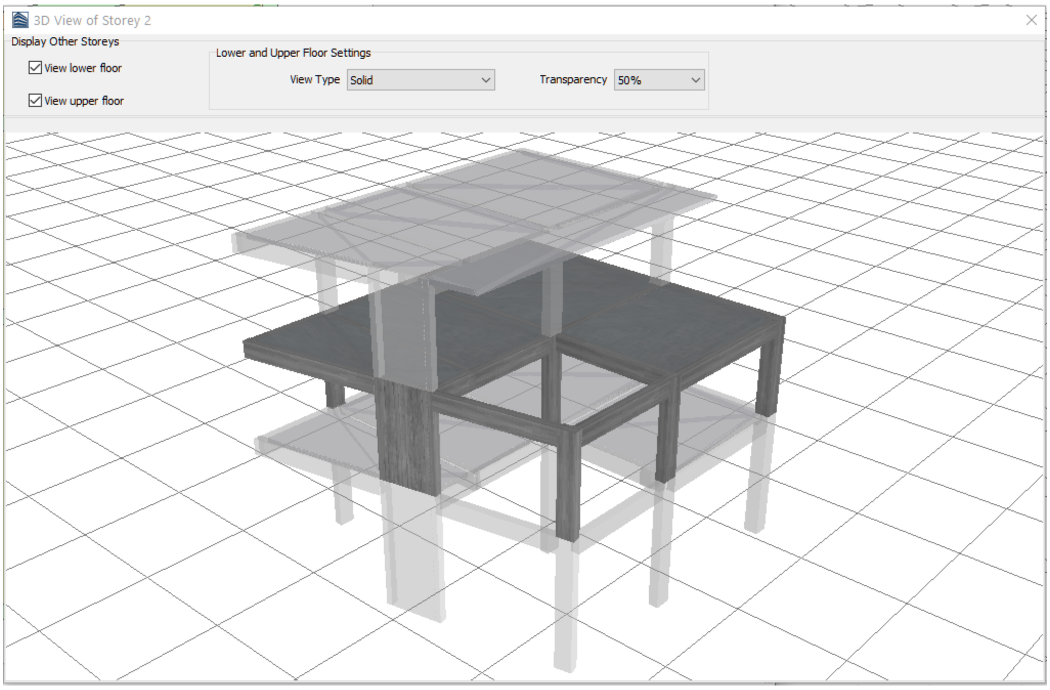
It is noted that users may use the layout of an existing floor as background, in order to easily introduce new members on another storey.



New Floor & Background

VIEW STOREY 3D MODEL

The possibility of viewing the 3D model of the current floor is offered through the main menu (*View > Storey 3D Model...*) or through the toolbar button.

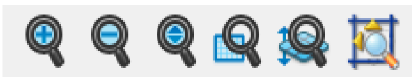


3D View of Storey window

OTHER BUILDING MODELLER FUNCTIONS

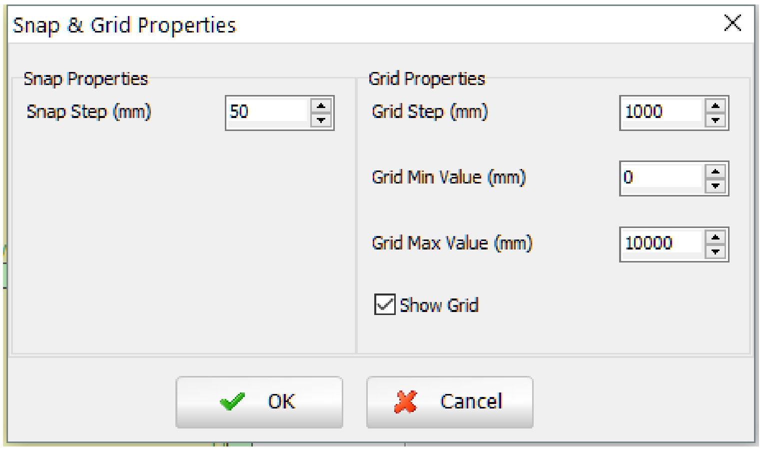
The Building Modeller offers a variety of tools to facilitate the introduction of the structural layout:

- Different zoom tools are available to users (zoom in, zoom out, dynamic zoom, zoom to window, center all and zoom to member). These tools are also available through the respective toolbar buttons or through the main menu (*View >...*).



Zoom tools

- Showing or hiding the CAD drawing as a background image can be done from the main menu (*View > Show/Hide DWG...*) or through the corresponding toolbar button , after it has been loaded with the button .
- Snap tools offer the possibility of snapping to the CAD drawing, the element and/or the grid. The grid properties (step, min and max values) and the snap properties (step), as well as whether the grid will be visualised or not may be defined from the Snap and Grid Properties dialog box accessed by the main menu (*View > Snap & Grid Properties*) or through the  toolbar button.



Snap & Grid Properties

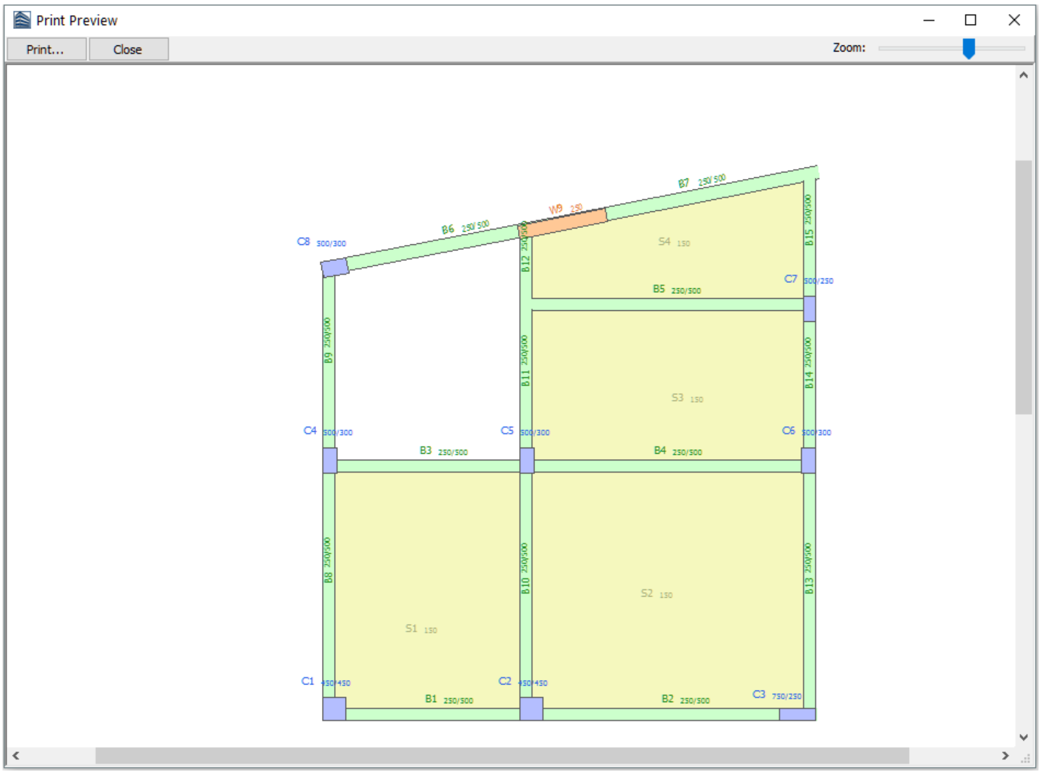
Further, an Ortho facility is provided; Ortho is short for orthogonal, and allows for the introduction of either vertical or horizontal - but not inclined - line (beams or walls) members. Again, all these facilities can be accessed from both the main menu (*View >...*) and through the corresponding toolbar buttons.

Snap and Ortho tools

- The axes origin of the CAD drawing at the background can also be moved from the main menu (*View >Move Axes Centre*) or through the toolbar button.

Move Axes Center

- The building in plan view may also be moved from the main menu (*Tools > Move Building*) or through the toolbar button.
- The option of rotating the building in plan view is available from the main menu (*Tools > Rotate Building*) or from the toolbar button.
- The possibility of undoing and redoing the last operations is offered (*Edit >Undo*)/ (*Edit >Redo*) or through the buttons .
- The selected plan view can be printed or previewed from the main menu (*File >Print... & File >Print Preview...*) or through the respective toolbar buttons &

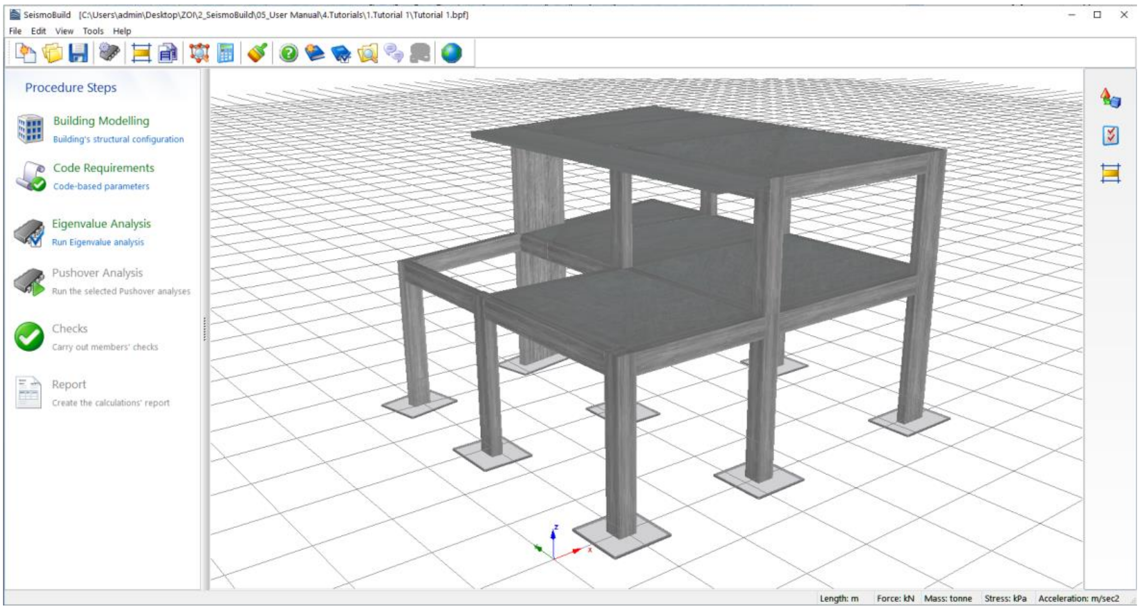


Print Preview

SAVING AND LOADING SEISMOBUILD PROJECTS

The SeismoBuild project (with the *.bpf extension) may be saved from the main menu (*File >Save As...*)/(*File >Save*) or through the toolbar button. These files can be opened again from the Main Window of SeismoBuild (*File >Open*) or through the corresponding toolbar button .

The 3D SeismoBuild model may be visualised by selecting the button or the main menu (*File >Exit and Create 3D Model*).



New SeismoBuild model

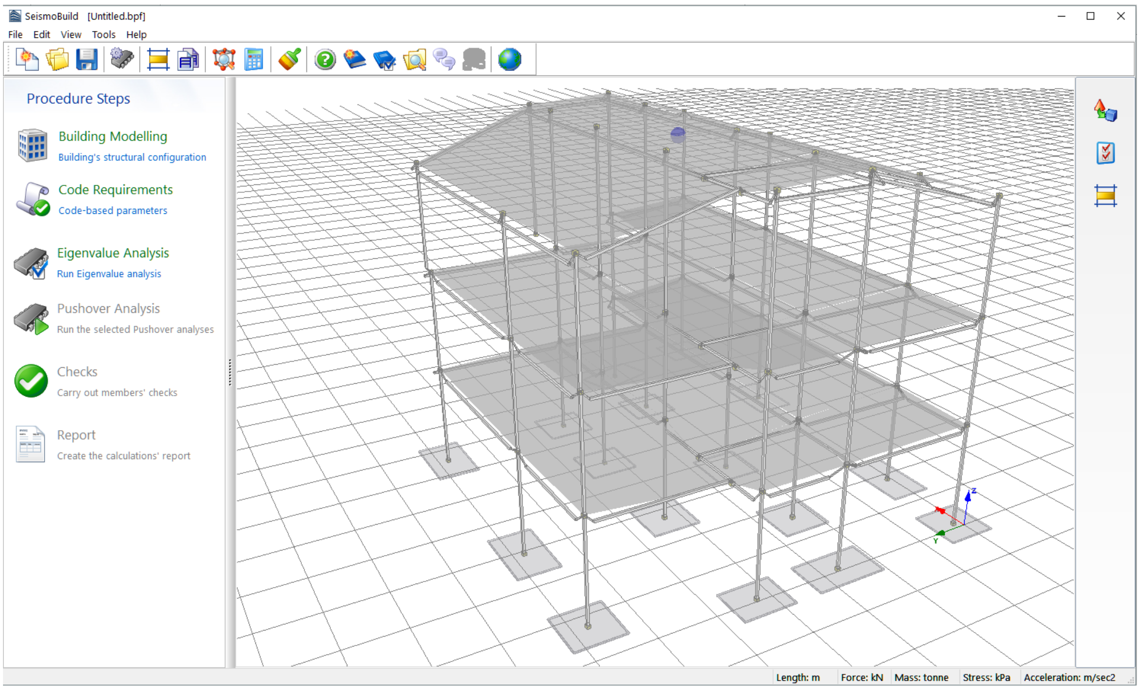
NOTE: When creating a SeismoBuild project file from the Building Modeller, the structural mass is modelled by the material's specific weight, and the sections' additional mass parameters. The former accounts for the mass of the columns, the walls and the beams, while the latter accounts for the mass that corresponds to the slabs' self weight, additional permanent loads and live loads. These defined masses are transformed to gravity loads, based on the g value.

Structural Modelling

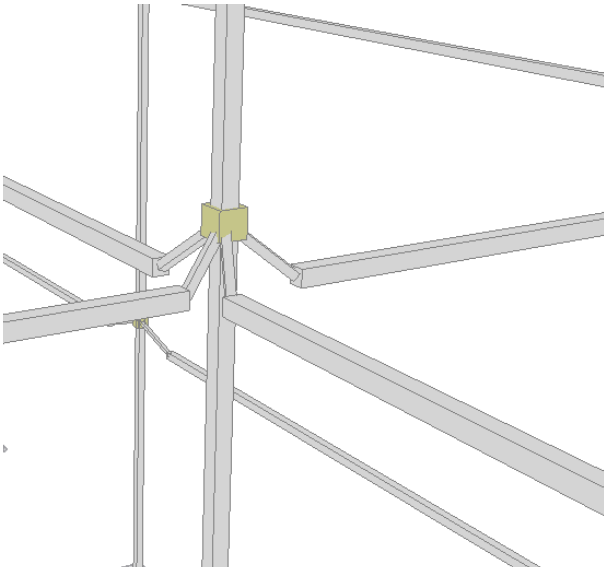
After the model input is defined in the Building Modeller the program automatically creates the structural model according to the following rules:

- The structural members are modelled with fibre-based inelastic frame elements. Both geometric nonlinearities and material inelasticity are considered. Furthermore, the offsets at the beam-column joints are automatically calculated and included in the structural model.
- Fourteen nonlinear material models are available for modelling the concrete and steel behaviour. The Mander et al. [1988] model and the Menegotto-Pinto [1973] model are employed by default for concrete and steel, respectively. A specially created model for high strength concrete is also available.
- Five element types are available, consisting of a combination of force-based and displacement based inelastic elements, with distributed plasticity or lumped plasticity (plastic-hinge elements), as well one elastic element type. In particular the available element types are: (i) Inelastic force-based frame element type - infrmFB, (ii) Inelastic force-based plastic hinge frame element type- infrmFBPH, (iii) Inelastic displacement-based plastic hinge frame element type - infrmDBPH, (iv) Inelastic displacement-based frame element type - infrmDB, and (v) Elastic frame element type –elfrm. The inelastic displacement-based frame element type is generally used for members of small length, e.g. to model short columns.
- With the distributed plasticity formulations (infrmFB and infrmDB), the spread of inelasticity along the member length and across the section depth is explicitly modelled, allowing for the accurate estimation of damage accumulation. This is of particular importance in the modelling of walls at the lower floors and especially at the ground level, where the large bending moments and the distribution of plasticity are not concentrated at the member ends (in which case a plastic-hinge approach would be adequate), but rather along the entire storey height. Considering the dominant role that large shear walls play in the overall structural behaviour, this feature sets apart SeismoBuild from other similar assessment packages that model all the structural members with plastic-hinge element, and thus do not simulate the distribution of inelasticity along the entire height of the ground floor.
- The default element types employed are the lumped plasticity (plastic-hinge) force-based elements infrmFBPH for the columns, beam, and walls. For short elements the distributed plasticity displacement-based is employed, for reasons of improved analysis stability and better convergence.
- Specifically for the beams, different cross section is considered in the element's integration sections, according to the specified reinforcement patterns (longitudinal and transverse) at the start, middle and end of the member. In the columns and the walls one cross-section is employed throughout the length of the member.
- The diaphragmatic action of the slabs is modelled though rigid diaphragms.
- The stairs and ramps are modelled with elastic elements of the specified width and depth.
- The column and beam masses are directly included in the model.
- The slab masses are applied to the supporting beams as additional masses and loads.
- The default option for the location of the control node is automatically determined by the program as the centre of mass of the upper floor or of the floor lower to that (in the case of having a top floor mass less than 10% of the lower floor's). Users may define the control node at a different floor.
- The column elements are considered fixed at the foundation level. Different foundation levels and at different floors may be employed for different columns of the same structure.
- Elastomeric and lead rubber isolators are modelled as isolator1 element type and curve surface sliders are modelled as isolator2 element type.
- Individual Footings are modeled as links with one end fixed to the ground and the other end connected via a rigid link to the column member based on the footing. The link modeling the individual footing can be either elastic or non-elastic

- Strip Footings are modelled using Inelastic Displacement Based Elements. In particular Strip Footings are separated into smaller linear parts each modeled as an Inelastic displacement-based (infrmDB) frame element. At every point where a column or wall is connected to the strip footing a link (either linear or non-linear is used to simulate the connection to the ground. In particular the one end of the link is fixed on the ground and the other end is connected to the strip footing. The column is connected to the strip footing via a rigid link.
- Connecting beams are modelled using Inelastic force-based plastic hinge (infrmFBPH) frame elements.



SeismoBuild Main Window – Show FE Model

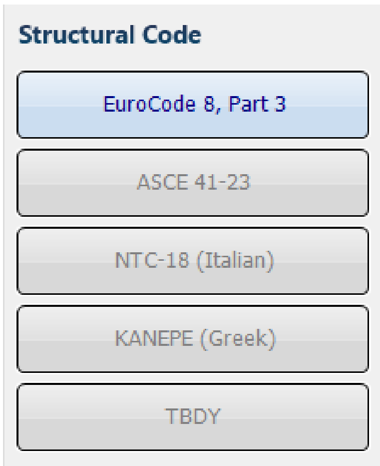


Member offsets

Code Requirements

The Code requirements may be defined through a dialog box accessed by  in the main SeismoBuild window.

The **Code Requirements** area features a series of pages where the necessary settings for the analyses and the checks may be determined according to the selected Code or Standard. The available Codes in the current version of SeismoBuild are the Eurocode 8- Part 3 along with the majority of the available National annexes, the American Code for Seismic Evaluation and Retrofit of Existing Buildings (ASCE 41-23), the Italian National Seismic Code (NTC-18), the Greek Seismic Interventions Code (KANEPE) and the Turkish Seismic Evaluation Building Code (TBDY); once the Code is selected the pages are respectively modified.



Available Codes

The available pages within the Code Requirements area are listed below and will be described in detail in the following paragraphs:

- Limit States / Performance Objectives
- Seismic Action (Target Spectrum)
- Analysis Type (Lateral Load Profile or Record Generation)
- Knowledge Level
- Static Actions
- Interstorey Drift Limits (Only in NTC)
- Target Displacement (Only in KANEPE)
- Checks

NOTE: The available Codes depend on the edition of the SeismoBuild. Users should select the edition with the required Codes.

Limit States

Herein users may define the Limit States or the Performance Levels to be used to check the structure.

In the cases of the Eurocodes, the state of damage in the structure is defined through three limit states, namely Near Collapse (NC), Significant Damage (SD) and Damage Limitation (DL). It is noted that the selection of the limit state(s) that need to be checked in a country may be found in the relevant National Annex. As a result, selection of a National Annex from the available ones will define whether all three Limit States, two or one of them will be employed during the Code-Based Checks.

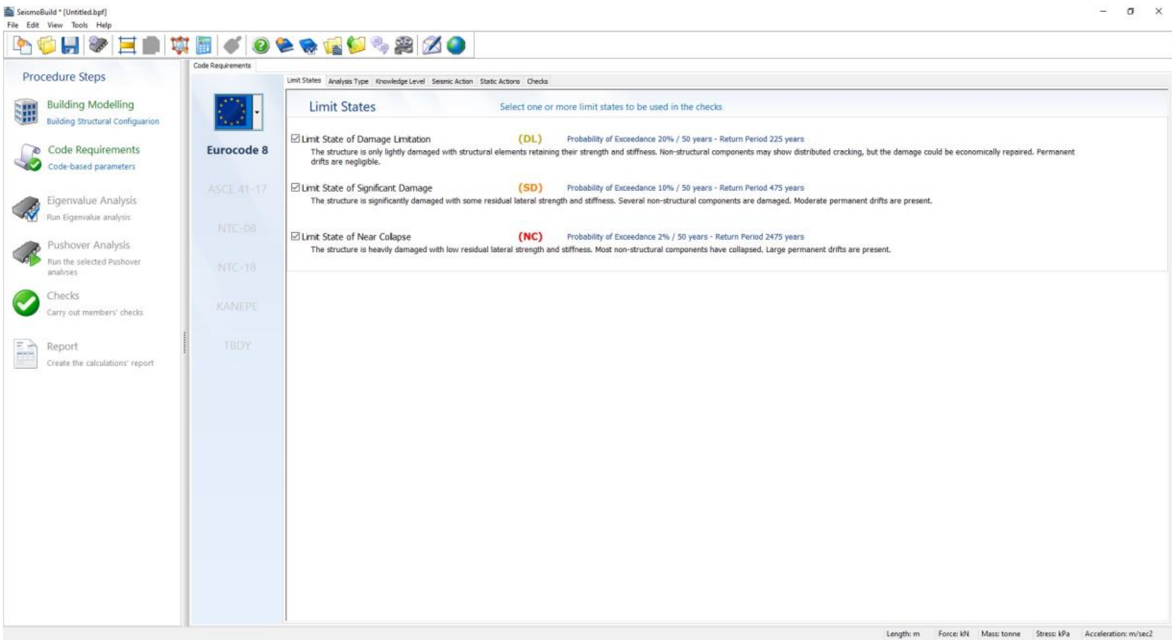
The performance levels employed in ASCE 41-23 consist of combinations of the target building performance levels (Operational Level, Immediate Occupancy, Life Safety and Collapse Prevention) and seismic actions (2%, 5%, 20% and 50% probabilities of exceedance of the seismic action within a conventional life cycle of 50 years).

The corresponding limit states for the Italian Code are four, i.e. limit state of Operational Level (OL), Damage Limitation (DL), Life Safety (LS) and Collapse Prevention (CP). Apart from the considered limit states, users may specify the location of the structure in the Italian territory, the life time of the structure and its importance class, and the parameters needed for the spectra derivation are automatically calculated.

The performance objectives employed in KANEPE consist of combinations of the performance requirements (Immediate Occupancy, Life Safety and Collapse Prevention) and seismic actions (10% and 50% probabilities of exceedance of the seismic action within a conventional life cycle of 50 years).

The performance levels for the TBDY consist of combinations of the target building performance levels (Continuous Use, Immediate Occupancy, Life Safety and Collapse Prevention) and seismic actions (2%, 10%, 50% and 68% probabilities of exceedance of the seismic action within a conventional life cycle of 50 years).

A detailed description on the limit states is available in the corresponding appendix for the selected Code (Appendix A.1 – EUROCODES, Appendix A.2 – ASCE, Appendix A.3 – NTC-18, Appendix A.4 – KANEPE, Appendix A.5 - TBDY).

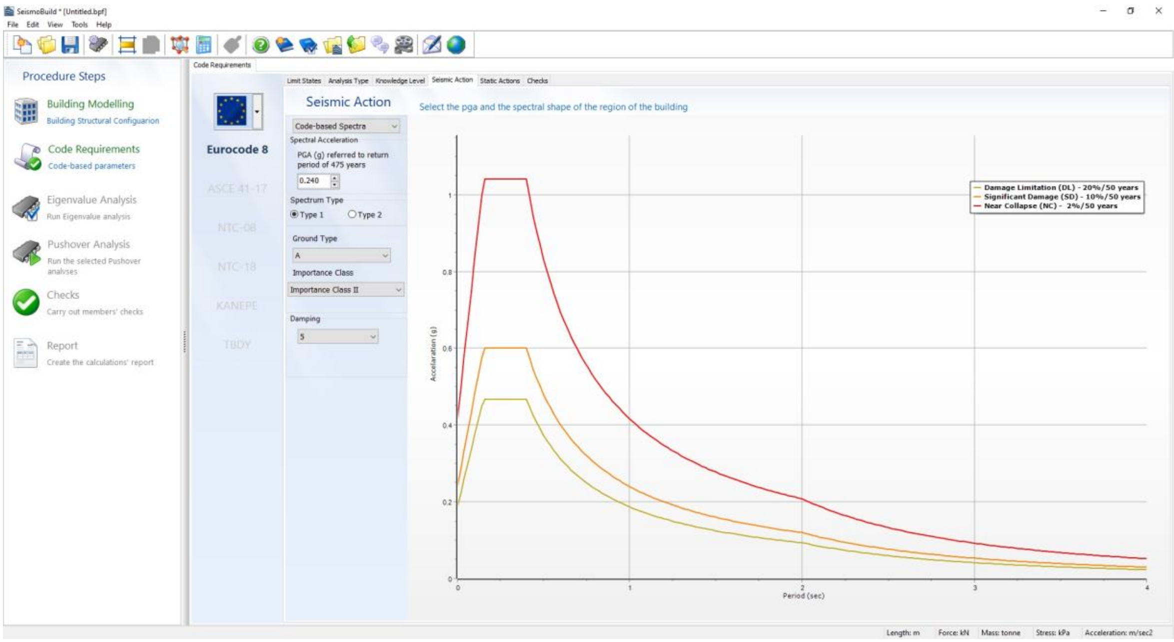


Limit States module

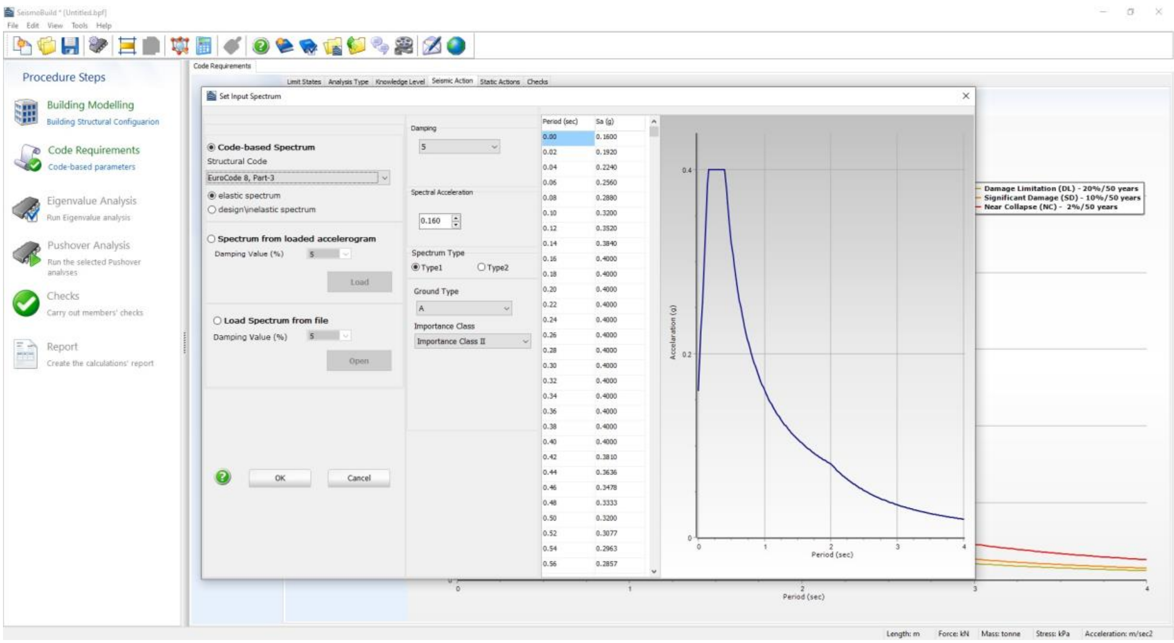
SEISMIC ACTION (TARGET SPECTRUM)

Herein the elastic response spectrum is determined, and scaled to the values of the design ground acceleration established for the different Limit States. The elastic response spectrum can be derived from the code used in the specific project (Code-Based Spectra option) or it can be defined by the user (User –Defined Spectrum option). In the case of Code-Based Spectra, users should assign the basic parameters needed for the generation of the spectral shape (i.e. peak ground acceleration, damping, spectrum type, ground type and important class).For further information you may refer to the corresponding section of the selected Code (Appendix A.1 – EUROCODES, Appendix A.2 – ASCE, Appendix A.3 – NTC-18, Appendix A.4 – KANEPE, Appendix A.5 – TBDY).

In the case of User Defined Spectra, users can select from a list of 29 spectra defined by various National Codes across the world (Code-Based Spetrum option), where again they should supply the basic parameters for the definition of the spectral shape. There is also the option to upload an accelerogram based on which the elastic response spectrum will be calculated (Spectrum from loaded accelerogram option) or to upload an elastic spectrum from a file (Load Spectrum from file option).



Seismic Action module (Code Based Spectra option)



Seismic Action module (User-defined Spectra option)

ANALYSIS TYPE (LATERAL LOAD PROFILE OR RECORD GENERATION)

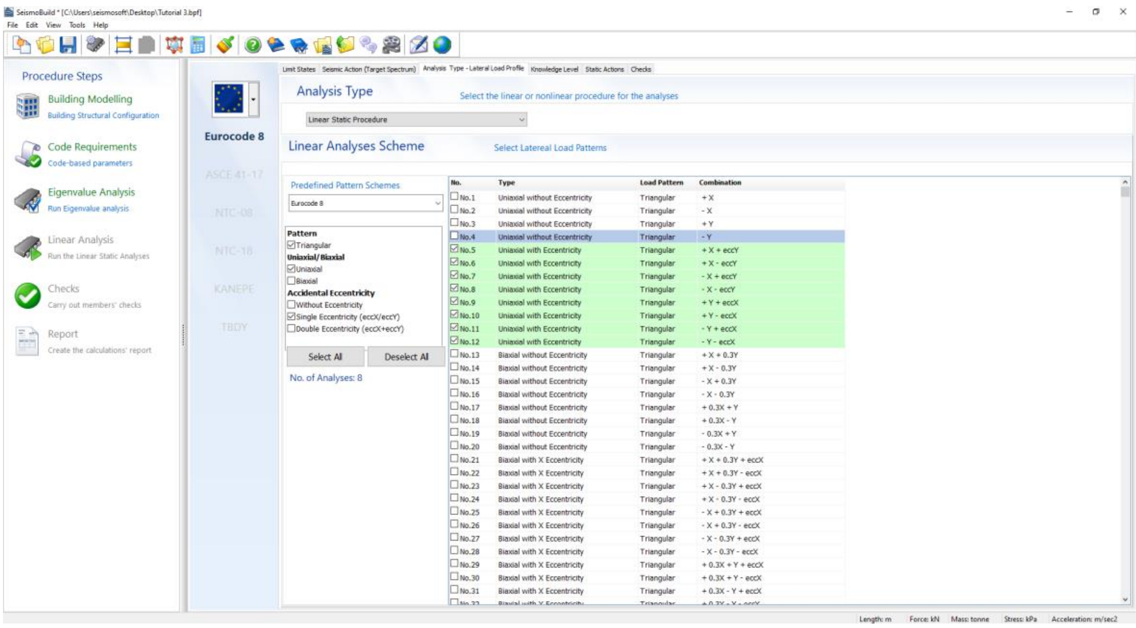
In SeismoBuild both linear and nonlinear methods are available for the structural assessment. Linear Static and Linear Dynamic procedure may be selected, as well as the two most accurate methods in the assessment practice of existing buildings are employed, that is the nonlinear static pushover analysis and the nonlinear time-history dynamic analysis.

Linear Static Procedure

With the Linear Static Procedure (Lateral Force Method with the EC8 naming conventions) a lateral, pseudo-seismic force distribution with a triangular profile is assumed to approximate the earthquake loading. The forces are applied to a linear elastic structural model, in order to calculate the internal forces and the system displacements.

The different load patterns that will be applied to the structure are defined in this module in two ways:

- The first is by selecting one of the pattern schemes as defined by the Codes, i.e. (i) Basic Combinations, (ii) Eurocode 8, (iii) ASCE 41-23, (iv) NTC-18, (v) KANEPE and (vi) TBDY. By choosing one of these schemes the appropriate load patterns will be selected.
- The second way is to choose individual user-defined load patterns from the corresponding checkboxes. Users may decide about the simultaneous or not application of the lateral incremental loads in the two horizontal directions (Uniaxial or Biaxial load patterns) and the existence or not of Single and/or Double Eccentricity.

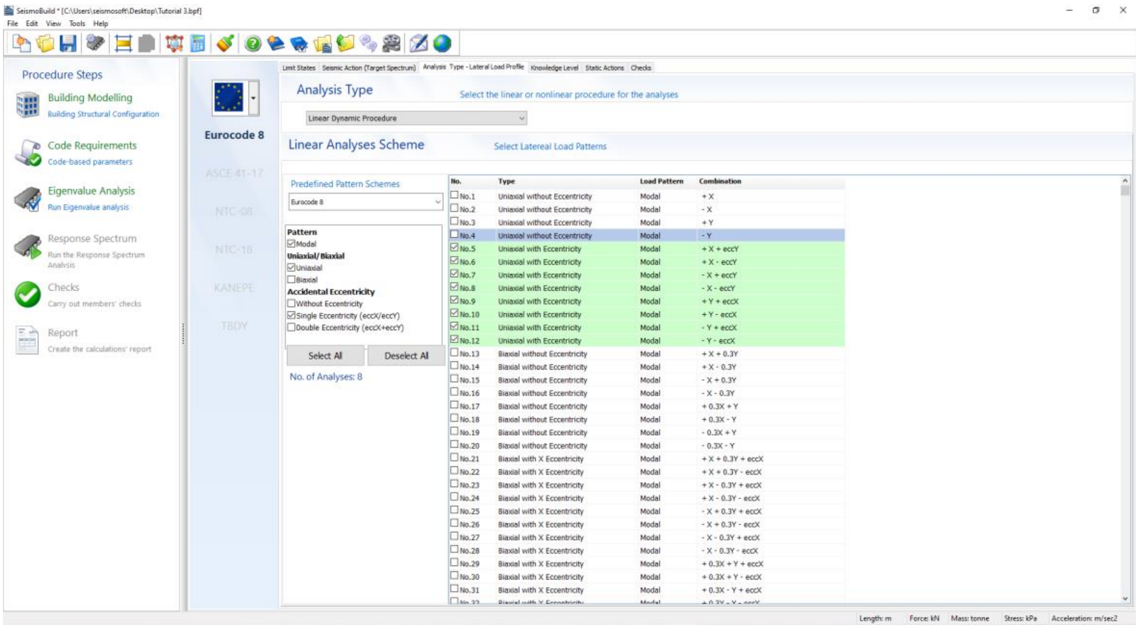


Analysis Type module for the Linear Static Procedure

Users may refer to the General section of the Linear Static Procedure of this Manual for further information about the loading patterns.

Linear Dynamic Procedure

The Linear Dynamic Procedure (Modal Response Spectrum Analysis, according to the EC8 naming conventions) is similar to the LSP, at least as regards the modelling approach. The model is again elastic and there is no stiffness degradation during the analysis. However, the method is somehow more sophisticated, since the profile of the lateral forces is not arbitrary anymore, but rather it is calculated as a combination of the modal contributions of the different modes of vibration of the structure.



Analysis Type module for the Linear Dynamic Procedure

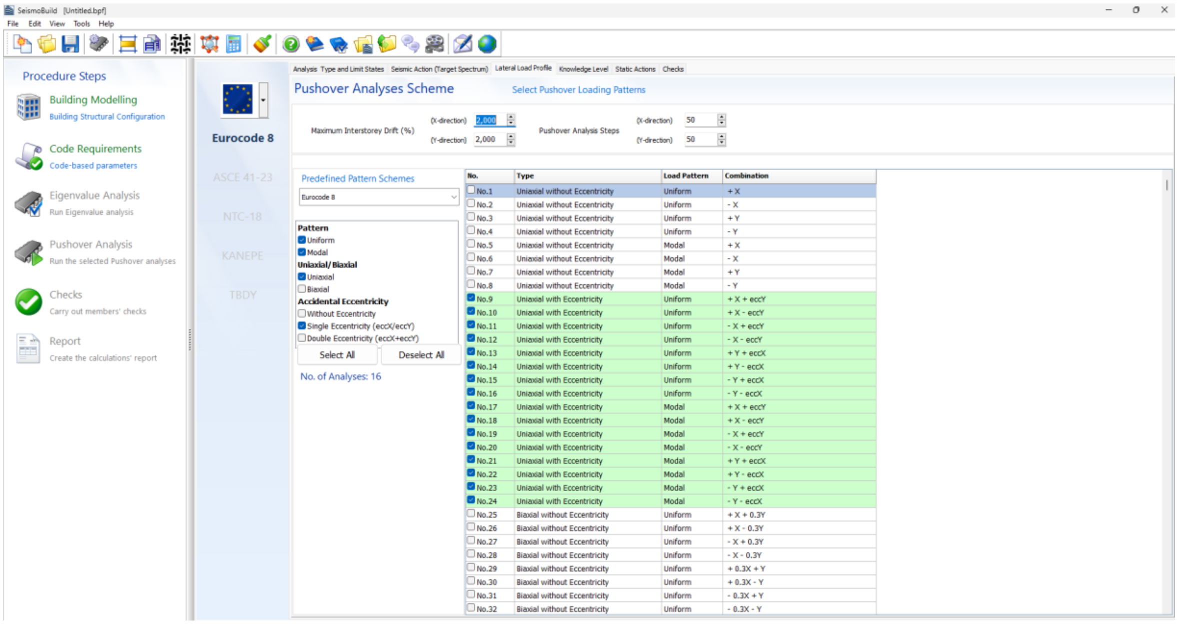
Nonlinear Static Procedure

Pushover analysis is carried out under constant gravity loads and increasing lateral forces, applied at the location of the masses to simulate the inertia forces induced by the seismic action. As the model may account for both geometrical and mechanical nonlinearity, this method can describe the evolution of the expected plastic mechanisms and structural damage.

The different load patterns that will be applied to the structure are defined in this module in two ways:

- The first one is by selecting one of the pattern schemes as defined by the Codes, i.e. (i) Basic Combinations, (ii) Eurocode 8, (iii) ASCE 41-23, (iv) NTC-18, (v) KANEPE and (vi) TBDY. By choosing one of these schemes the appropriate load patterns will be selected.
- The second way is to choose individual user-defined load patterns from the corresponding checkboxes. Users may decide about the vertical distribution of loads (uniform and/or modal patterns), the simultaneous or not application of the lateral incremental loads in the two horizontal directions (Uniaxial or Biaxial load patterns) and the existence or not of Single and/or Double Eccentricity.

The maximum interstorey drift in X and Y direction, as well as the analysis steps in X and Y direction are also defined here.



Analysis Type module for Pushover Analysis

Users may refer to the General section of Linear and Nonlinear Analyses of this Manual for further information about the loading patterns.

Nonlinear Dynamic Analysis

In dynamic time-history analysis a mathematical model directly incorporating the nonlinear load-deformation characteristics of individual components of the building is subjected to earthquake shaking represented by ground motion accelerations.

Instead of the lateral forces distributions that are used in the nonlinear static procedure, an earthquake record is now applied at the foundation level of the building in the form of acceleration time-histories. In SeismoBuild the accelerograms can be either (a) artificial or synthetic records that match the given target spectrum or (b) directly uploaded by the user.

The direct integration of the equations of motion is accomplished using appropriate integration algorithms, such as the numerically dissipative α-integration algorithm (Hilber-Hughes-Taylor HHT scheme) or a special case of the former, the well-known Newmark scheme.

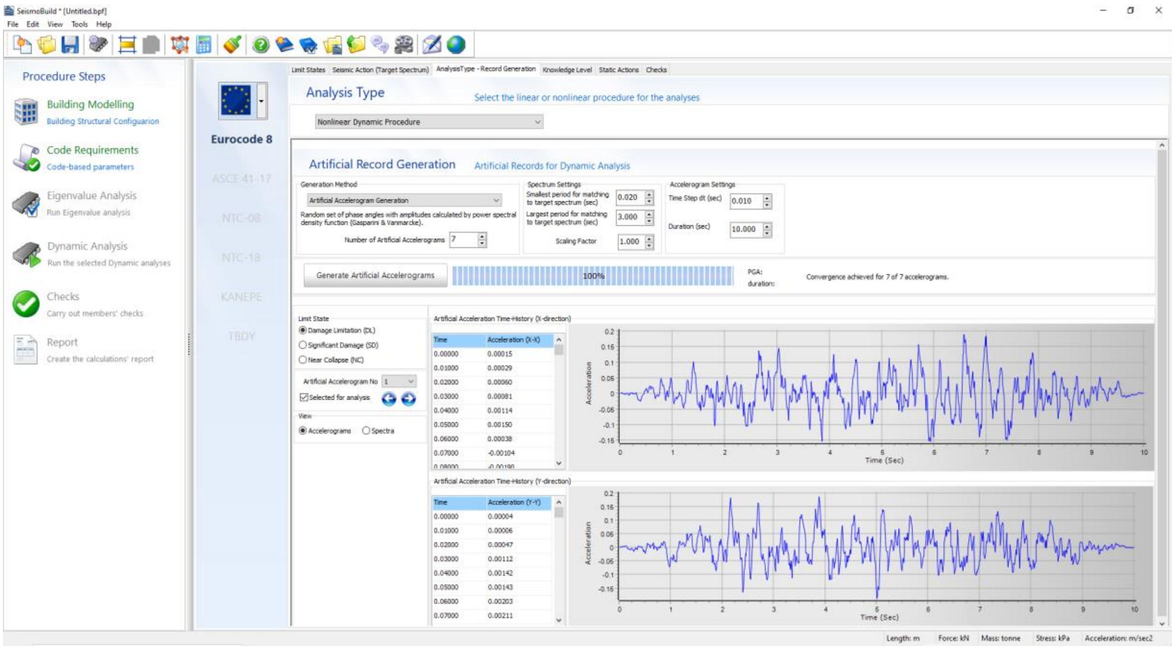
Upon selecting this type of the analysis, the method of generation of the accelerograms to be used in the analysis need to be specified.

If one of three artificial accelerogram generation methods is selected, the Artificial Record Generation module is displayed. The user may select the number of artificial accelerograms to be generated, the target spectrum settings (minimum and maximum period for matching, and the scaling factor for the defined target spectrum), the record settings (time step and duration), as well as the generation algorithm. The three methods currently available in SeismoBuild for the simulation of artificial ground motions:

- Synthetic Accelerogram Generation & Adjustment [Hallodorson & Papageorgiou, 2005]
- Artificial Accelerogram Generation [Gasparini & Vanmarcke,1976], which is the default option
- Artificial Accelerogram Generation & Adjustment

The Artificial Accelerogram Generation and Artificial Accelerogram Generation & Adjustment methods are based on the adaptation of a random process to a target spectrum. The adaptation is based on the frequency content using the Fourier Transformation Method, and the adjustment in the second method is done in frequency domain. Only the target spectrum is required for the generation of an accelerogram in both cases.

On the contrary, for the generation of synthetic accelerograms some basic knowledge of the geotectonic environment and the soil conditions relative to the region/site of interest are required. The artificial accelerogram is defined starting from a synthetic one and adapting its frequency content using the Fourier Transformation Method. This method is able to efficiently provide good results, but it has the disadvantage of additional input other than the target spectral shape (earthquake regime, near or far-field, the expected earthquake magnitude, the distance from the source, and the soil conditions).

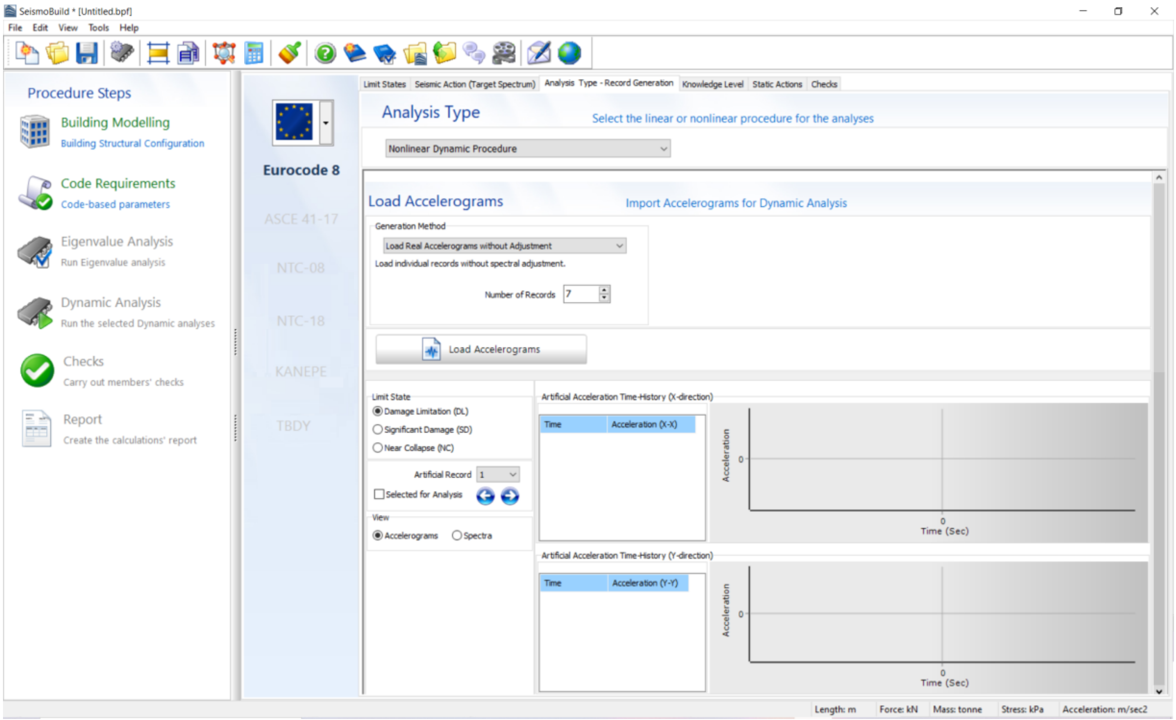


Analysis Type module for Dynamic Analysis with Artificial Record Generation module on display

Instead of selecting one of the three aforementioned artificial accelerogram generation methods, there is the option to load real accelerograms without adjustment. Upon the selection of this fourth option in the Generation Method menu the Load Real Accelerograms module is displayed.

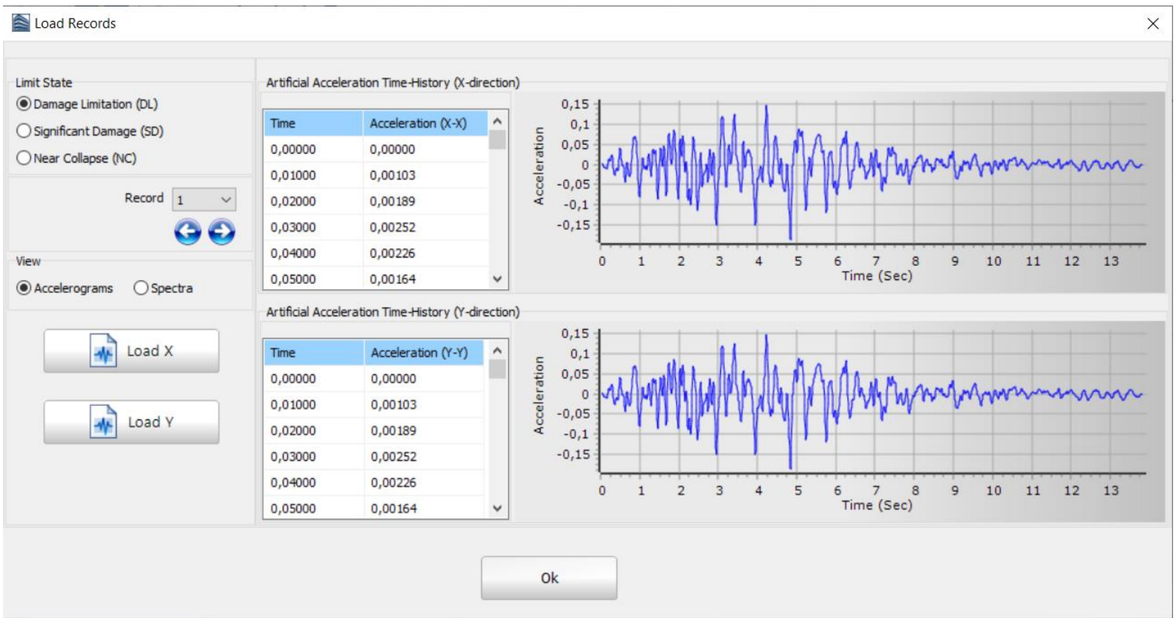
The display of the RotD100 spectrum is also available.

The ability to employ only the X component or only the Y component of accelerogram, when running dynamic time-history analysis in the Nonlinear Dynamic Procedure is also available in all the above-mentioned methods.

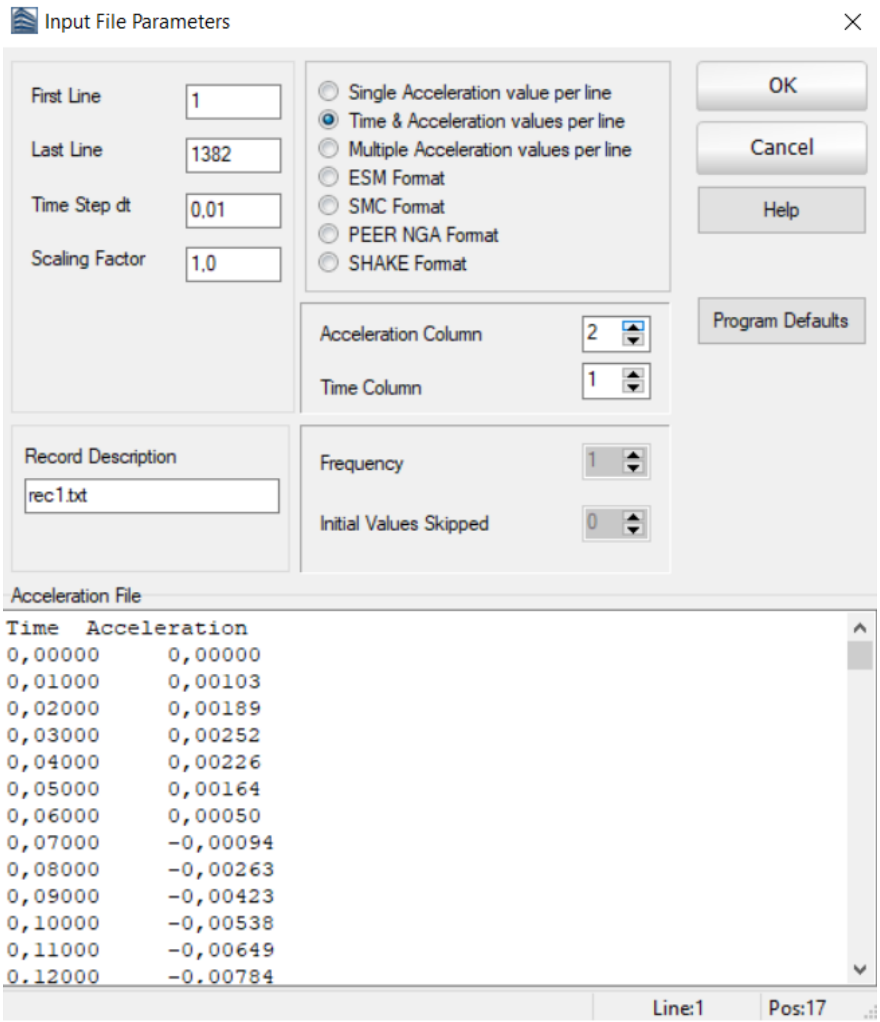


Analysis Type module for Dynamic Analysis with Load Accelerograms Selected

Here the number of records need to be selected and then select Load Accelerograms and in the Load Records window which appears upload the accelerograms to be used in the analysis. Specifically, by selecting Load X and Load Y in the Load Records window the Input File Parameters window opens where the accelerogram file for the corresponding earthquake direction can be selected and uploaded. In total, for each Specified record two accelerograms must be uploaded per Limit State. After uploading an accelerogram no further adjustment can be carried out in SeismoBuild.



Load Records Window



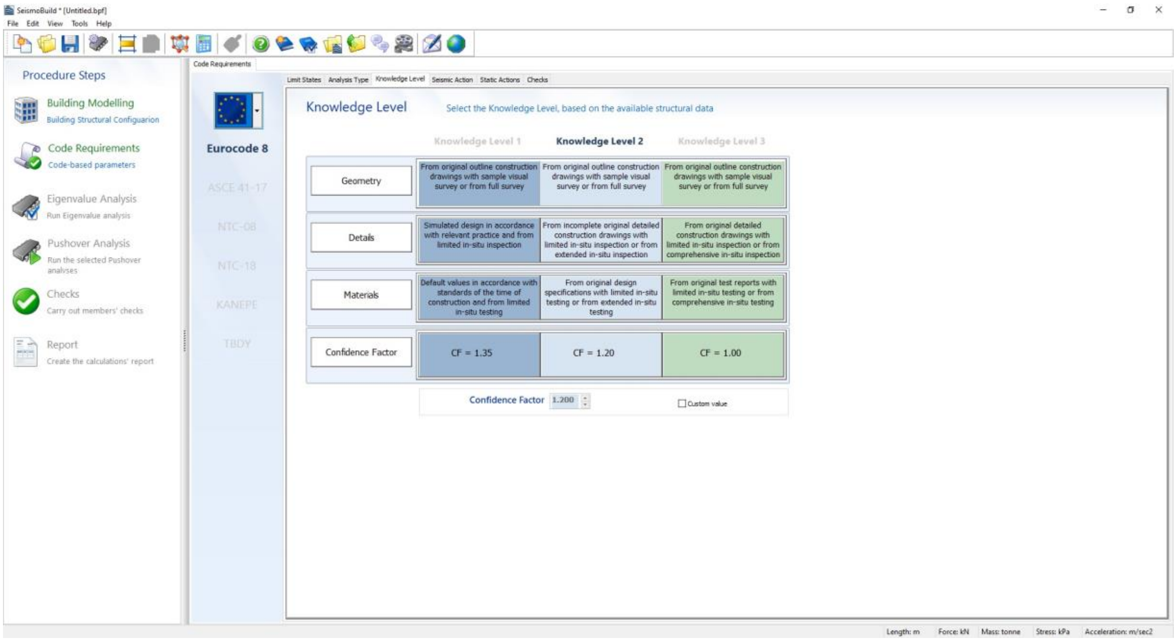
Input File Parameters Window

The display of the RotD100 spectrum is also available.

Users may refer to the General section of Pushover and Dynamic Analysis of this Manual for further information about the loading patterns.

KNOWLEDGE LEVEL

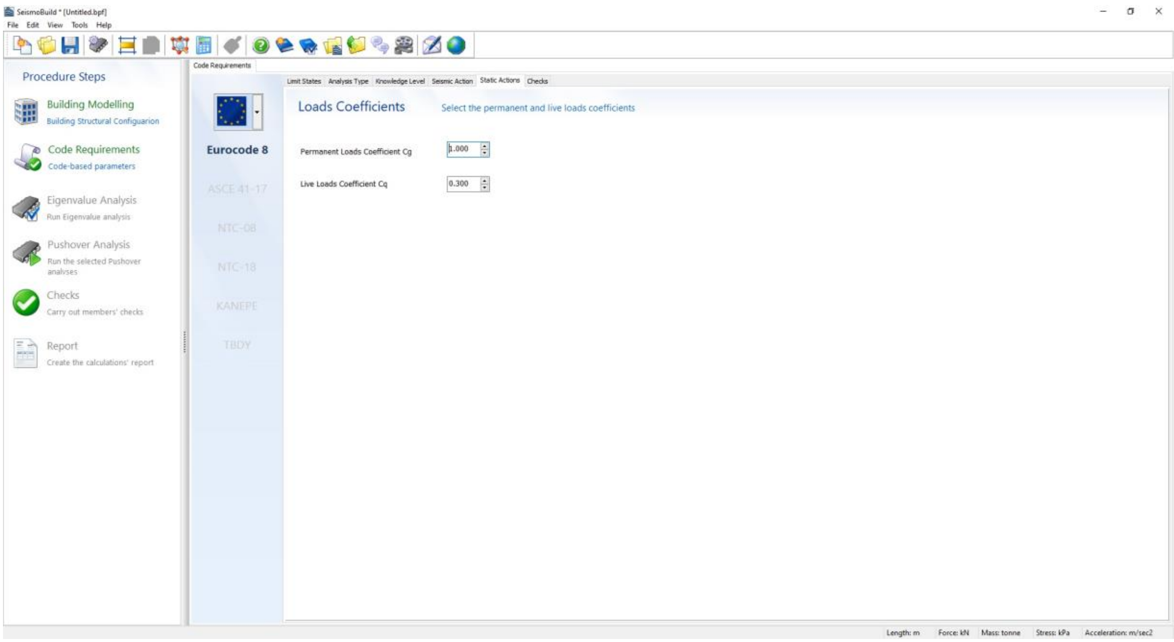
In this module users may select the Knowledge Level that corresponds to the available data on the structural configuration. Three knowledge levels are defined. A more detailed description on the factors determining the achieved knowledge level is available in Appendix A.1 – EUROCODES, Appendix A.2 – ASCE, Appendix A.3 – NTC-18, Appendix A.4 – KANEPE, Appendix A.5 – TBDY. The default values for the confidence factor are those recommended in the corresponding Code, although it is also possible to assign different values by selecting the Custom value checkbox (e.g. according to the country's National Annex for the case of Eurocodes).



Knowledge Level module

STATIC ACTIONS

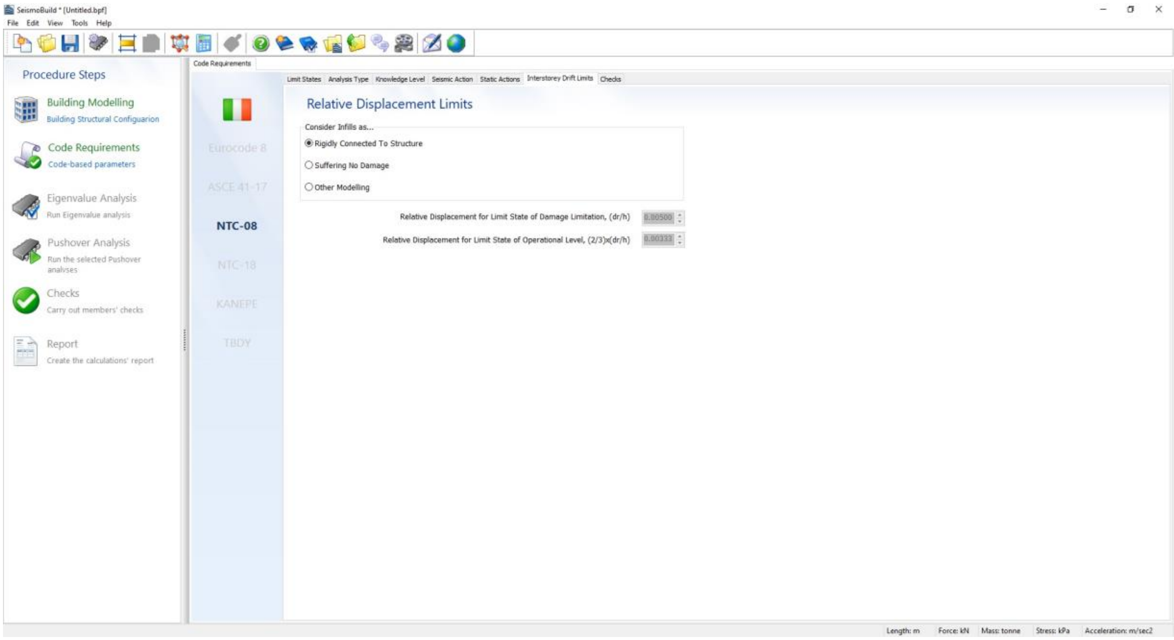
The permanent and live loads coefficients are assigned in this module. As described in the General section of Pushover and Dynamic Analysis, in addition to the introduced vertical loads in nonlinear static analyses the permanent and live loads are applied, which are also used for the definition of the structural mass in the Eigenvalue Analysis. The snow load coefficient is also assigned in the case of ASCE 41-23 or TBDY.



Static Actions module

INTERSTOREY DRIFT LIMITS

In NTC an additional check may be executed for the limit states of Operational Level and Damage Limitation, as described in the Interstorey Drifts section of Appendix A.3 – NTC-18. Users may define through this module whether the infills will be considered as rigidly connected to the structure, or are expected to suffer no damage, in order to specify the target relative deformation at each floor. Alternatively, they can assign user-defined values, by selecting the other modelling option.



Interstorey Drift Limits module

TARGET DISPLACEMENT

Herein the structural type of the building is specified for the proper definition of the C2 factor, taken into consideration in the Target Displacement calculations of KANEPE (see the Target Displacement section of Appendix A.5 – KANEPE.

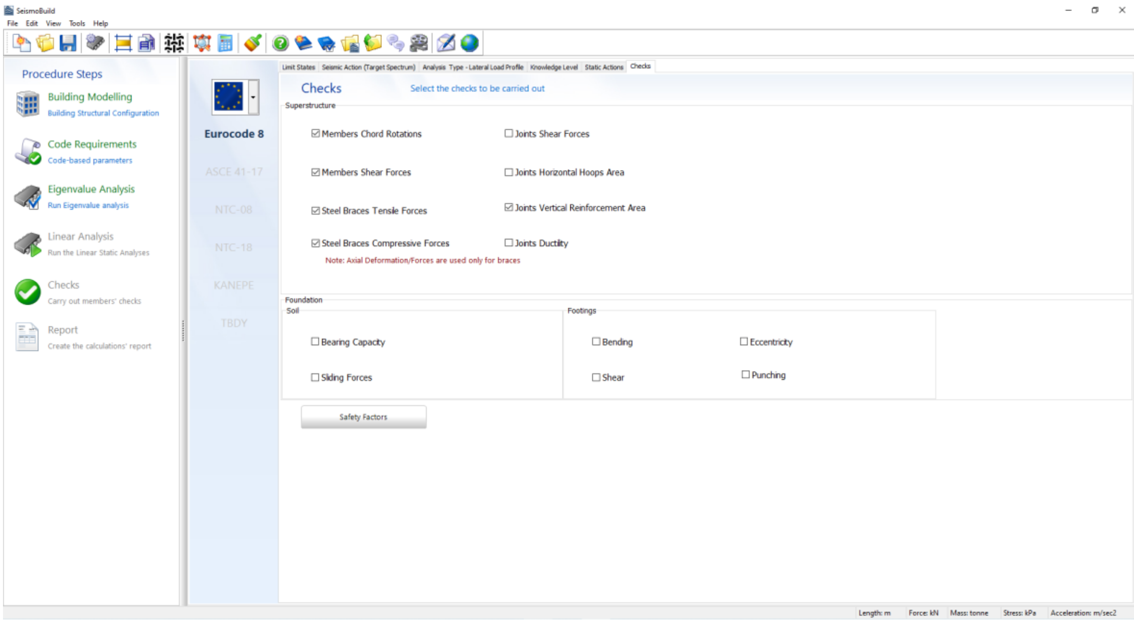
CHECKS

Herein, the checks to be carried out according to the selected Code for structural members, beam-column joints and foundation may be selected. The available checks within all the employed Codes are the following:

- Members Chord Rotations
- Members Bending Moments
- Members Shear Forces
- Members Strains (Only for TBDY)
- Steel Braces Tensile Deformations
- Steel Braces Compressive Deformations
- Steel Braces Tensile Forces
- Steel Braces Compressive Forces
- Joints Shear Forces (Eurocode 8, ASCE 41-23 & TBDY)
- Joints Horizontal Hoops Area (Only for Eurocode 8)

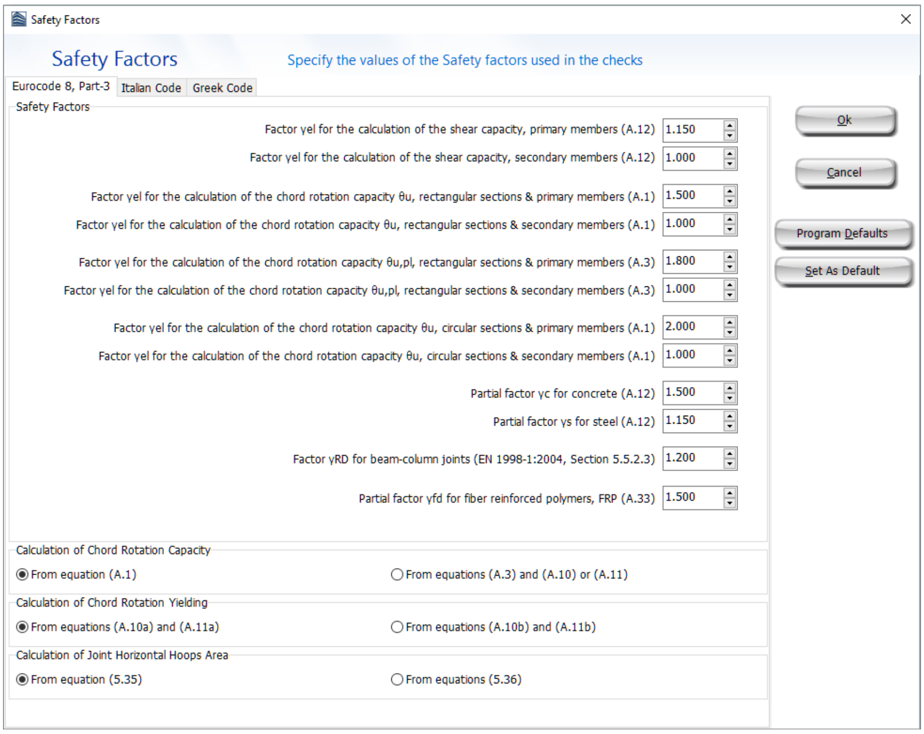
- Joints Vertical Reinforcement Area (Only for Eurocode 8)
- Joints Ductility
- Joints Diagonal Tension (NTC & KANEPE)
- Joints Diagonal Compression (NTC & KANEPE)
- Interstorey Drifts (ASCE 41-23 & NTC)
- Footings Bearing Capacity (Eurocode 8, NTC & KANEPE)
- Footings Sliding Forces (Eurocode 8, NTC & KANEPE)
- Footings Rocking Moment Capacity (ASCE 41-23 & TBDY)
- Footings Rocking Rotation Capacity (ASCE 41-23 & TBDY)
- Footings Bending Capacity
- Footings Shear Capacity
- Footings Punching Capacity
- Footings Eccentricity

A more detailed description on the checks and the equations used in SeismoBuild is available in the Checks and in the Capacity Models for Assessment and Checks of Appendix A.1 – EUROCODES, Appendix A.2 – ASCE, Appendix A.3 – NTC-18, Appendix A.4 – KANEPE, Appendix A.5 – TBDY, respectively.



Checks module

The values of the safety factors used in the checks may be specified through the corresponding button, as well as the employed Code expressions. The program default factors are those defined in the selected Code.



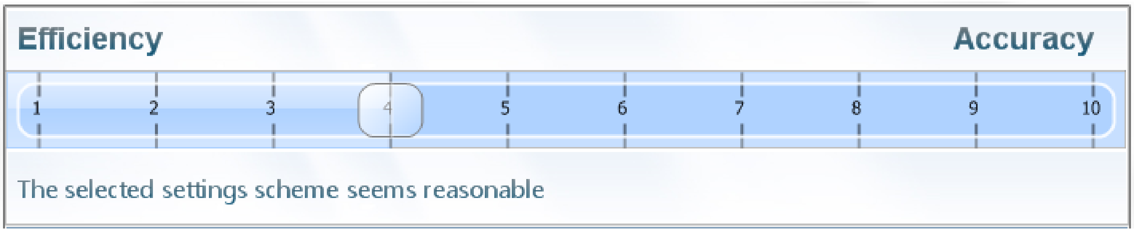
Safety Factors module

Analysis & Modelling Parameters

Users are able to define all the parameters required for the nonlinear analytical calculations within this module, by selecting a predefined settings scheme, by clicking on the Advanced Settings button or double-clicking on a specific value to open the corresponding tab in the Advanced Settings module.

Settings Schemes

Because of the requirement for advanced and specialised knowledge for most of the analysis parameters (e.g. material models, frame element type, convergence criteria tolerances, rigid diaphragm modelling), ten predefined schemes are available that define the more important analysis parameter settings.

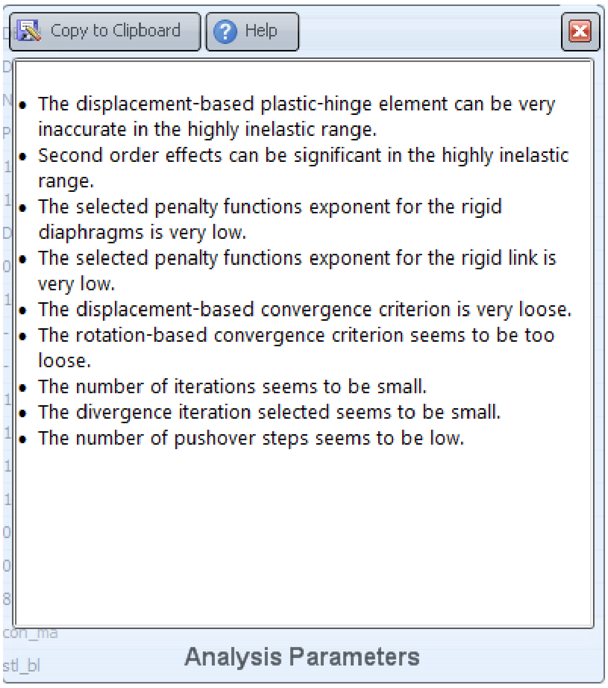


Settings Schemes bar

These predefined Settings Schemes have been chosen so as to fit the requirements of multiple types of analysis and models, leading to optimised solutions in terms of performance efficiency and results accuracy. Depending on a model's particular characteristics and demands, different Settings Schemes might fit at different cases; the program carries out an internal check and a notification message is issued, whenever one or more of the settings do not seem to suit the needs of the specific project. The notification messages that might appear are the following:

- *The selected settings scheme seems reasonable*, which means that with the selected combination users will probably not face convergence difficulties or accuracy problems during the analyses;
- *Strict convergence criteria. Convergence problems might arise*; the program warns for possible convergence problems that might arise because of the strict convergence criteria chosen;
- *Very strict convergence criteria. Convergence problems will probably arise*; the program warns for possible convergence problems that will probably arise because of the very strict convergence criteria chosen;
- *Loose convergence criteria. Accuracy problems might arise*; the program warns for possible accuracy problems that might arise because of the loose convergence criteria chosen;
- *Very loose convergence criteria. Accuracy problems will probably arise*; the program warns for possible accuracy problems that will probably arise because of the very loose convergence criteria chosen.

When a chosen settings scheme does not seem reasonable, a See Why button appears on the right side of the bar. By clicking on this button users are able to see why the selected settings scheme should not be applied to their model, and which specific settings need to be changed to improve them.



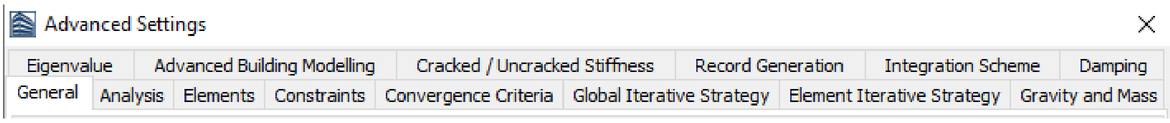
See Why window

ADVANCED SETTINGS

For each SeismoBuild project it is possible to customise both the usability of the program as well as the performance characteristics of analytical proceedings, so as to better suit the needs of any given structural model and/or the preferences of a particular user. This program/project facility is available from the Analysis Parameters module or the Advanced Settings panel, which can be accessed through the Advanced Settings button Advanced Settings.

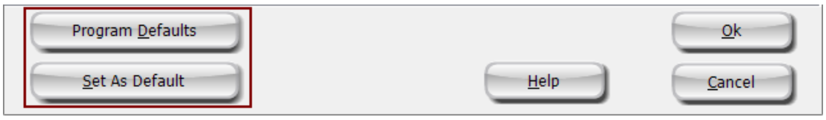
The Advanced Settings panel features a number of tab windows, which provide access to different type of settings, as described below:

- General
- Analysis
- Elements
- Constraints
- Convergence Criteria
- Global Iterative Strategy
- Element Iterative Strategy
- Gravity and Mass
- Eigenvalue
- Advanced Building Modelling
- Cracked/Uncracked Stiffness
- Record Generation
- Integration Scheme
- Damping



Analysis settings tab windows

Common to all tab windows are the “Program Defaults” and “Set As Default” options found at the bottom of the Advanced Settings panel. The “Set As Default” option is employed whenever the user wishes to define new personalised default settings, which will then be used in all new projects subsequently created. The “Program Defaults”, on the other hand, can be used to reload, at any time, the original program defaults, as defined at installation time. Note, however, that the Program Defaults option does not change the default program settings; it simply loads the installation settings in the current project. Hence, if the user has previously personalised the default settings of the program (using the Set As Default option) and then wishes to revert the program default settings back to the original installation defaults, he/she should first load the Program Defaults and then choose the Set As Default option.



Program Defaults and Set as Default options

NOTE: For the majority of applications, there is no need for the Advanced Settings default values to be modified, since the available Settings Schemes have been chosen so as to fit the requirements of any building model, leading to optimised solutions in terms of performance efficiency and results accuracy.

General

The General settings provide the possibility of customising the usability of the program to the user's preferences.

Text Output

When activated, the Text Output option will lead to the creation, at the end of every analysis, of a text file (*.out) containing the output of the entire analysis (as given in the Step Output module). This feature may result useful for users who wish to systematically post-process the results using their own custom-made post-processing facility. For occasional access to text output, users are instead advised to use the facilities made available in the Step Output module.

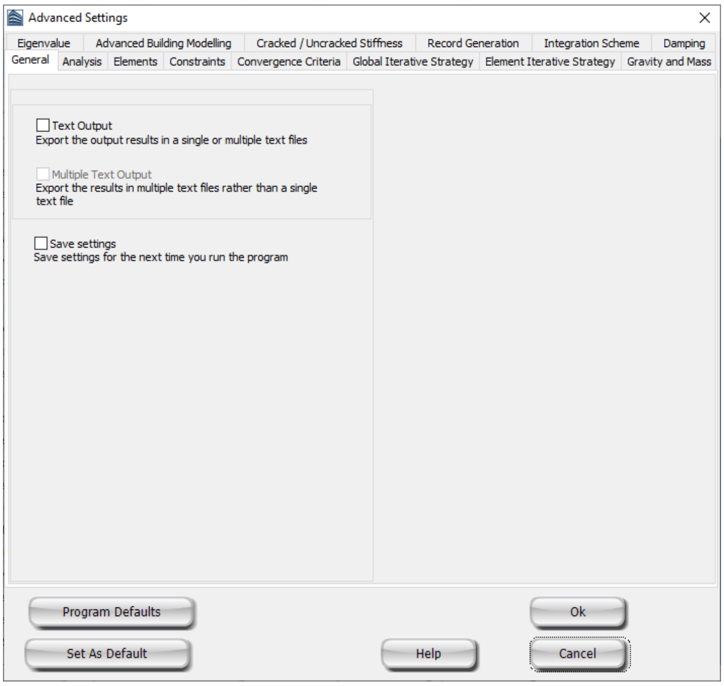
Multiple Text Output

When activated, the Multiple Text Output option will lead to the creation of multiple text files (*.out), rather than a single one. This feature may result useful when large models are going to be analysed.

Save Settings

The Save Settings option is used when the user wishes to always make the current project settings the default settings for every new project that is subsequently created. With this checkbox selected, any change in Project Settings will become a default, without the need for the Set as Default option to be used.

NOTE: Normally, this option is disabled so that the default settings are only changed if explicitly requested by the user (using the Set as Default option).



General tab window

Analysis

In the Analysis tab window, some settings related to the analysis can be defined. In particular, it is possible to select the solver type and whether to account for geometric nonlinearities.

Solver

Apart from the linear equation solver, users, users are able to select whether the initial loading, i.e. structural static loads, will be applied in one or more steps in the nonlinear analysis types. The default option is to apply it in one single step.

Further, the option of executing eigenvalue analysis at every step in nonlinear pushover analysis is available. Users may select to run an eigenvalue analysis at the end of the nonlinear analysis or to perform eigenvalue analysis multiple times during the nonlinear analysis by specifying after how many steps the eigenvalue analysis will be performed.

Users may currently choose between the following different solvers:

- A **Skyline Solver** (Cholesky decomposition, Cuthill-McKee nodes ordering algorithm, Skyline storage format);
- A **Frontal Solver** for sparse systems, introduced by Irons [1970] and featuring the automatic ordering algorithm proposed by Izzuddin [1991].
- A **Sparse/Profile Solver** for sparse systems, introduced by Mackayet al. [1991] and featuring a compact row storage scheme using elimination trees proposed by Liu [1986].

A **Parallel Sparse/Profile Solver** for sparse systems, which is the parallel version of the Mackay et al. algorithm. The method was introduced by Law and Mackay [1992]. Users may select between these four options, or let the program select the most appropriate solver, depending on the characteristics of the structural model. It is noted that generally Sparse/Profile solvers are considerably faster, especially in larger models. In particular, the parallel version is more efficient for larger structural models of 500 nodes and more. In contrast the Skyline method is usually more stable and is capable of accommodating zero diagonal stiffness items.

When the automatic option is selected, which is the default option, the program performs a stability and size check prior to the analysis. If the model is not very small (i.e. smaller than 25 nodes), and if it can run with a Sparse/Profile solver without stability problems, this method is employed, otherwise the Skyline solver is chosen, parallel for more than 1000 nodes or the serial otherwise.

NOTE: Users are obviously advised to refer to the existing literature [e.g. Cook et al. 1989; Zienkiewicz and Taylor 1991; Bathe 1996; Felippa 2004] for further details on these and other direct solvers.

Finally, irrespective of the serial or parallel version of the selected solver, user may select to execute different operations of the structural analysis (initial checks, assembly of stiffness matrix, code-based checks and checks of the performance criteria) in parallel or not. Parallelizing these operations can be significantly faster in larger models, and this is the default option.

Pushover Parameters

The number of the analysis steps of Pushover in X and Y direction is defined by the user in this tab, as well as the maximum interstorey drift of the structure; the default option for the maximum interstorey drift is 2%, whereas the number of pushover analysis steps depends on the selected predefined settings scheme. For the default settings scheme (i.e. N_o 4) the default value is 50, which is reasonable for the majority of cases. Different values may be employed for the analysis steps and the maximum interstorey drift in X and Y direction.

Geometric Nonlinearities

Unchecking this option will disable the geometric nonlinearity formulation described in Appendix B, rendering the analysis linear, from a displacement/rotation viewpoint, which may be particularly useful for users wishing to compare analysis results with hand calculations, for verification purposes. By default, this option is active for frame elements and deactivated for masonry elements.

It is also possible to run the analyses considering the linear elastic properties of materials. In order to do this, user need to check the option '*Run with Linear Elastic Properties*'.

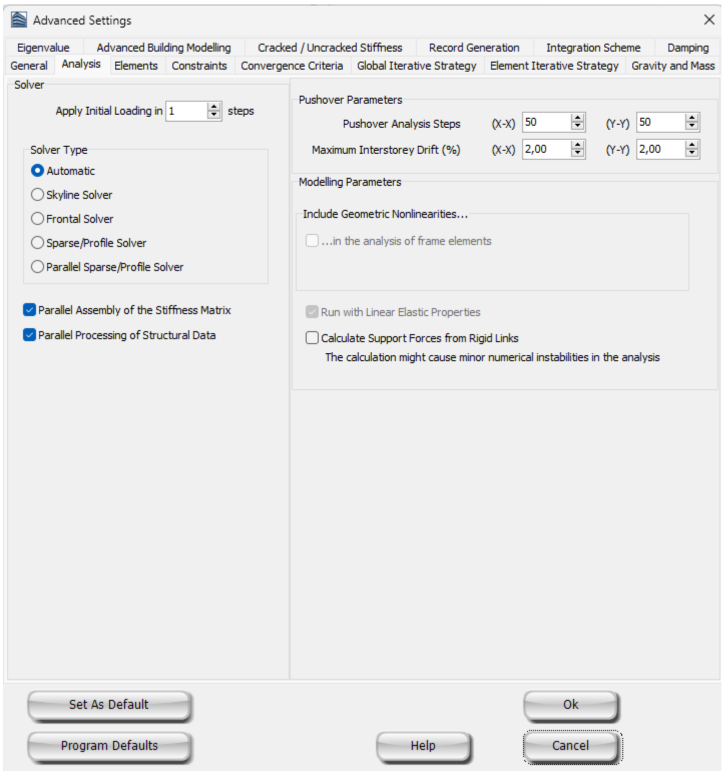
Run with Linear Elastic Properties

Checking this option will disable both material inelasticity and geometric non-linearities, leading to a totally linear, elastic analysis. By default, this option is inactive, with the exception of Response Spectrum Analysis, when it is the default option.

NOTE: When users decide to run an analysis considering the linear elastic properties of materials (see the option described above), they should keep in mind that, if the elements are modelled using RC sections and 'infrm' elements, the infrm elements will account for the reinforcement; on the contrary, if 'elfrm' elements are employed, their properties are calculated using the concrete modulus of elasticity and the section dimensions, thus neglecting the effect of the reinforcement.

Calculate Support Forces from Rigid Links

Checking this option enables the calculation of the support forces in the cases when some DOFs of a constraint (rigid link, rigid diaphragms or equal DOF) are fixed with restraints. By default, this option is inactive, because this calculation can cause minor numerical instabilities.



Analysis tab window

Elements

Herein some settings related to the analysis of frame elements can be defined.

Carry out Stress Recovery

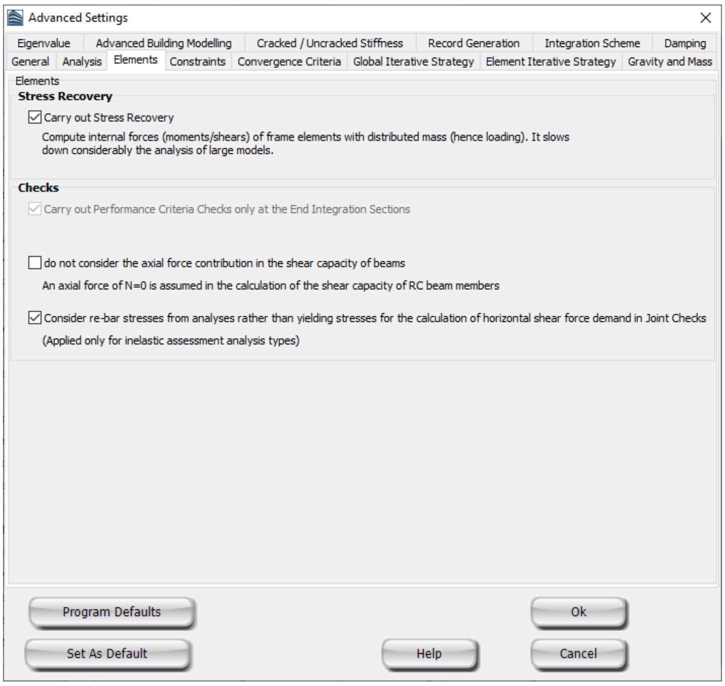
Some beam element formulations, such as those employed in SeismoBuild for the inelastic frame elements, feature the disadvantage that, if the nodal displacement is zero, one then gets also nil strains, stresses, and internal forces (e.g. if one models a fully-clamped beam with a single element, and applies a distributed load, the end moments will come out as zero, which is clearly wrong). To overcome this limitation, it is common for Finite Element programs to use so-called stress-recovery algorithms, which allow one to retrieve the correct internal forces of an element subjected to distributed loading even if its nodes do not displace. It is noted, however, that (i) such algorithms do not cater for the retrieval of the correct values of strains stresses, given that these are characterised by a nonlinear history response, and (ii) will slow down considerably the analyses of large models.

Do not consider the axial force contribution in the shear capacity of beams

By activating this option, the ability to carry out shear checks ignoring the actual axial force applied on the beam member is provided. This feature is particularly important to the shear capacity checks of beams, when the interaction between fibre modelled RC beams and the rigid diaphragm adopted to simulate the concrete slab (a very common configuration in RC buildings) may cause the development of unintended fictitious axial forces in them.

Consider re-bar stresses form analyses rather than yielding stresses for the calculation of horizontal shear force demand in Joints Checks

This is an option employed only in the nonlinear methods of analysis. If this option is checked, the calculations for the horizontal shear force demand in the Joints Checks are carried out employing the actual stresses of the re-bars(as calculated from the nonlinear analyses), rather than the stresses at yield, which are considered in the typical calculations for linear analyses that employ the capacity design philosophy.



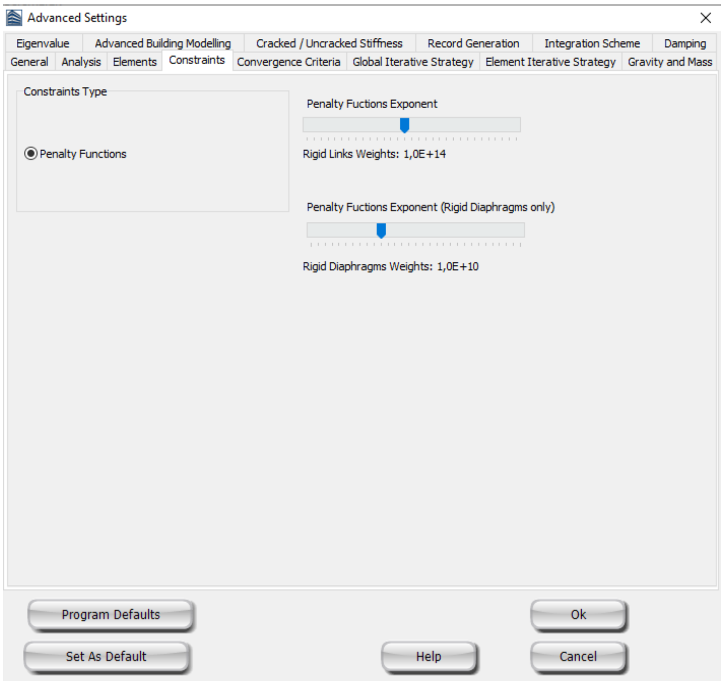
Elements tab window

Constraints

Constraints are typically implemented in structural analysis programs through the use of (i) **Geometrical Transformations**, (ii) **Penalty Functions**, or (iii) **Lagrange Multipliers**. In geometrically nonlinear analysis (large displacement/rotations), however, the first of these three tends to lead to difficulties in numerical convergence, for which reason only the latter two are commonly employed. The second has been implemented in SeismoBuild.

Herein it is simply noted that **Penalty Functions** have the advantage of introducing no new variables (and hence the stiffness matrix does not increase and remains positive definite), hence they do not increase the bandwidth of the structural equations [Cook et al., 1989].

NOTE: Felippa [2004] suggests that the optimum penalty functions value should be the average of the maximum stiffness and the processors precision (1e20, in the case of SeismoBuild).



Constraints tab window

Convergence Criteria

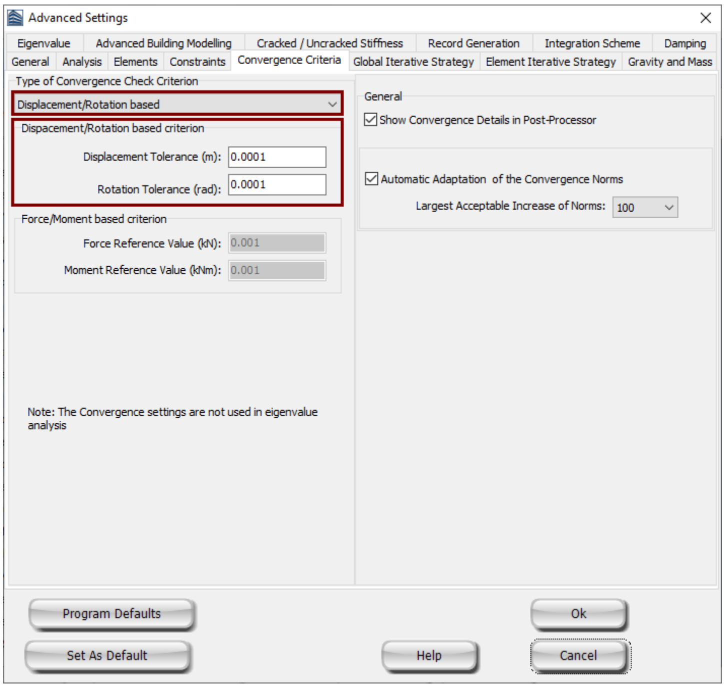
Four different schemes are available in SeismoBuild for checking the convergence of a solution at the end of each iteration:

- Displacement/Rotation based
- Force/Moment based
- Displacement/Rotation AND Force/Moment based
- Displacement/Rotation OR Force/Moment based

NOTE: Users are alerted to the fact that there is no such thing as a set of convergence criteria parameters that will work for every single type of analysis. The default values of the predefined schemes in SeismoBuild will work well for the vast majority of applications, but might need to be tweaked and modified for particularly demanding projects, where strong response irregularities (e.g. large stiffness differentials, buckling of some structural members, drastic change in intensity, etc.) occur. As an example, note that a tighter convergence control may lead to higher numerical stability, by preventing a structure from following a less stable and incorrect response path, but, if too tight, may also render the possibility of achieving convergence almost impossible.

Displacement/Rotation based

Verification, at each individual degree-of-freedom of the structure, that the current iterative displacement/rotation is less or equal than a user-specified tolerance, provides the user with direct control over the degree of precision or, inversely, approximation, adopted in the solution of the problem. In addition, and for the large majority of analyses, such local precision check is also sufficient to guarantee the overall accuracy of the solution obtained. Therefore, this convergence check criterion is the default option in SeismoBuild for the majority of the predefined settings schemes with the default values for displacement and rotation tolerance varying between different settings schemes. For the default settings scheme, which lead to precise and stable solutions in the majority of cases, the values for the displacement and the rotation tolerances are 0.0001 m and 0.0001 rad, respectively.

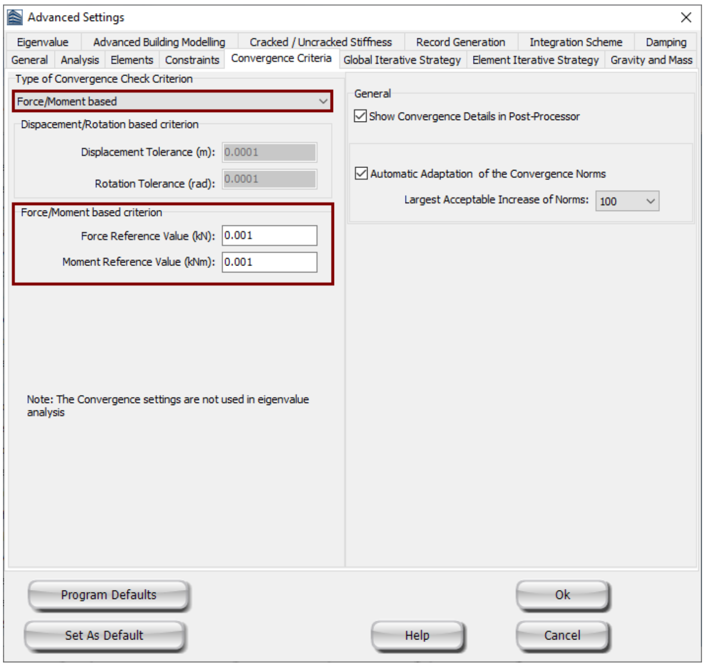


Convergence Criteria tab window – Displacement/Rotation based

Force/Moment based

There are occasions where the use of a displacement/rotation convergence check criterion is not sufficient to guarantee a numerically stable and/or accurate solution, due to the fact that displacement/rotation equilibrium does not guarantee, in such special cases, force/moment balance. This is the typical behaviour, for instance, of simple structural systems (e.g. vertical cantilever), where displacement/rotation convergence is obtained in a few iterations, such is the simplicity of the system and its deformed shape, which however may not be sufficient for the internal forces of the elements to be adequately balanced. Particularly, when a RC wall section is used, the stress-strain distribution across the section may assume very complex patterns, by virtue of its large width, thus requiring a much higher number of iterations to be fully equilibrated. In such cases, if a force/moment convergence check is not enforced, the response of the structure will result very irregular, with unrealistically abrupt variations of force/moment quantities (e.g. wiggly force-displacement response curve in pushover analysis). As described in Appendix B - Theoretical background and modelling assumptions, a non-dimensional global tolerance is employed in this case, with a default value of 0.001

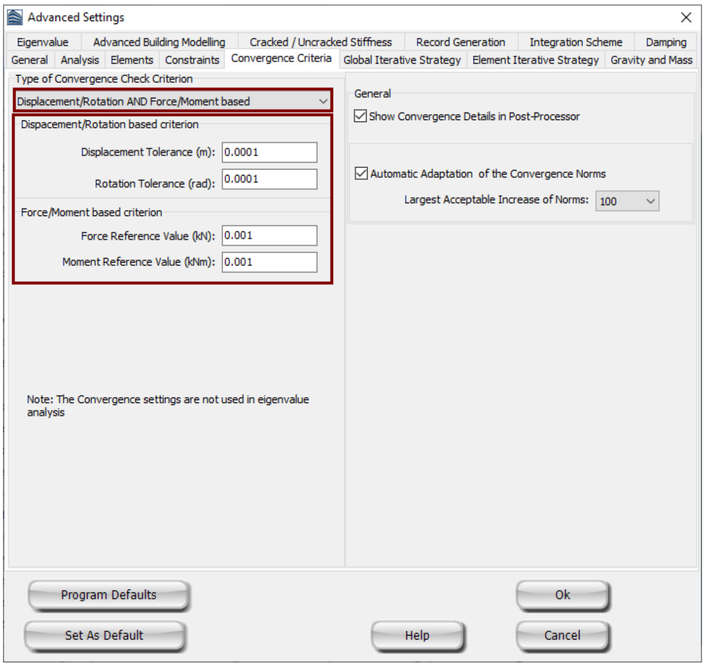
NOTE: Force-based criteria will cause numerical problems and convergence difficulties when used with penalty functions. In these cases the Lagrange Multipliers method should be employed.



Convergence Criteria tab window – Force/Moment based

Displacement/Rotation AND Force/Moment based

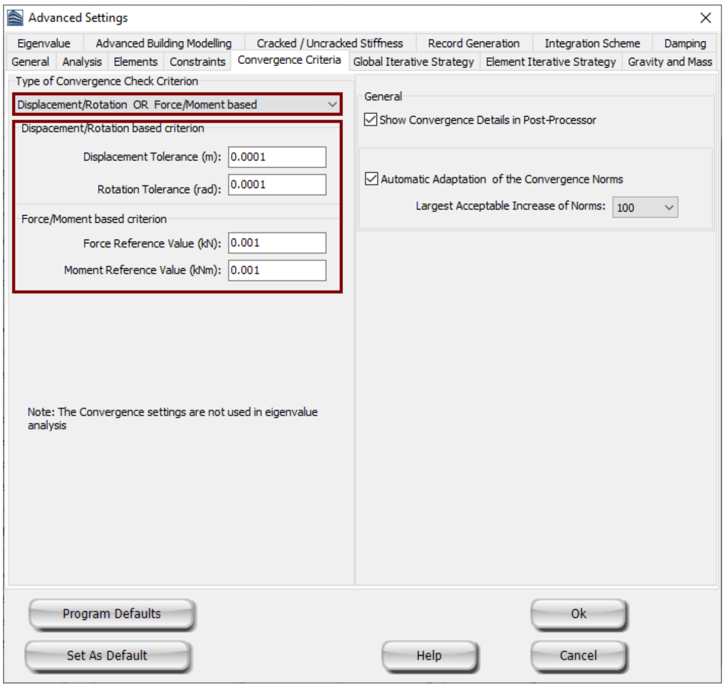
Taking into account the discussion made above, it results clear that maximum accuracy and solution control should be obtained when combining the displacement/rotation and force/moment convergence check criteria. This option, however, is not the default since the force/moment based criterion does, on occasions, create difficulties in models where infinitely stiff/rigid connections are modelled with rigid links, as discussed in Appendix B - Theoretical background and modelling assumptions. Still, it is undoubtedly the most stringent convergence and accuracy control criterion available in SeismoBuild, and experienced users are advised to take advantage of it whenever accuracy is paramount.



Convergence Criteria tab window – Displacement/Rotation AND Force/Moment based

Displacement/Rotation OR Force/Moment based

This last convergence criterion provides users with maximum flexibility as far as analysis stability is concerned, since converge is achieved when one of the two criteria is checked. This option is highly recommended when arriving at a particular final structural solution is the primary objective of the analysis, and accuracy assumes, at least momentarily, a secondary role.



Convergence Criteria tab window – Displacement/Rotation OR Force/Moment based

General

Users may select if the convergence difficulties that might arise during the analysis will be visible in the Post-Processor. The default option is to show the convergence difficulties in the Post-Processor of pushover analysis.

Automatic Adaptation of the Convergence Norms

If this option is selected, in particular steps of the analysis, where convergence is difficult to achieve, the program may smartly increase the defined convergence norms, in order to enable convergence and to allow the program to move to the next step of the analysis. In order not to allow for infinite increase in the value of the convergence norms, a limit is set by the Largest Acceptable Increase of Norms combo box. The default option is to allow for the automatic adaptation of the convergence norm.

Global Iterative Strategy

In SeismoBuild, all analyses are treated as potentially nonlinear, and therefore an incremental iterative solution procedure, whereby loads are applied in pre-defined increments and equilibrated through an iterative procedure, is applied on all cases (with the exception of eigenvalue analyses). The workings and theoretical background of this solution algorithm is described in some detail within the Nonlinear Solution Procedure section in Appendix B - Theoretical background and modelling assumptions, to which users should refer to whenever a deeper understanding of the parameters described herein is sought.

Maximum number of iterations

This parameter defines the maximum number of iterations to be performed within each load increment (analysis step). The **default value** depends on the selected predefined settings scheme, for the default settings scheme, which should work well for most practical applications, this value is equal to **40**. However, whenever structures are subjected to extremely high levels of geometric nonlinearity and/or material inelasticity, it might be necessary for this value to be increased.

Number of stiffness updates

This parameter defines the number of iterations, from the start of the increment, in which the tangent stiffness matrix of the structure is recalculated and updated. It is noteworthy that assigning a value of zero to this parameter effectively means that the modified Newton-Raphson (mNR) procedure is adopted, whilst making it equal to the Number of Iterations transforms the solution procedure into the Newton-Raphson (NR) method.

Usually, the ideal number of stiffness updates lies somewhere in between 50% and 75% of the maximum number of iterations within an increment, providing an optimum balance between the reduction of computation time and stability stemming from the non-updating of the stiffness matrix and the corresponding increase in analysis effort due to the need of further iterations to achieve convergence. The **default value** of this parameter for the default predefined settings scheme, which should work well for most practical applications, is however slightly more conservative, at a value of **35**, leading to the adoption of a hybrid solution procedure between the classic NR and mNR approaches (see also discussion in Incremental Iterative Algorithm).

Divergence iteration

This parameter defines the iteration after which divergence and iteration prediction checks are performed (see divergence and iteration prediction for further details). On all subsequent step iterations, if the solution is found to be diverging or if the predicted number of required iterations for convergence is exceeded, the iterations within the current increment are interrupted, the load increment (or time-step) is reduced and the analysis is restarted from the last point of equilibrium (end of previous increment or analysis step).

Whilst these two checks are usually very useful in avoiding the computation of useless equilibrium iterations in cases where lack of convergence becomes apparent at an early stage within a given loading increment, it is also very difficult, if not impossible, to recommend an ideal value which will work for all types of analysis. Indeed, if the divergence iteration is too low it may not allow highly nonlinear problems to ever converge into a solution, whilst if it is too high it may allow the solution to progress into a numerically spurious mode from which convergence can never be reached. A value around 75% of the maximum number of iterations within an increment usually provides a good starting point. The value for the default predefined settings scheme, which should work well for most practical applications, is set equal to **35**.

Maximum Tolerance

As discussed in Numerical instability, the possibility of the solution becoming numerically unstable is checked at every iteration, right from the start of any given loading increment, by comparing the Euclidean norm of out-of-balance loads (go to Appendix B - Theoretical background and modelling assumptions for details on this norm) with a pre-defined maximum tolerance (for the majority of the predefined settings scheme it is set to **1e20**), several orders of magnitude larger than the applied load vector. If the out-of-balance norm exceeds this tolerance, then the solution is assumed as numerically unstable, iterations within the current increment are interrupted, the load increment (or time-step) is reduced and the analysis is restarted from the last point of equilibrium (end of previous increment or analysis step).

Maximum Step Reduction

Whenever lack of convergence, solution divergence or numerical instability occurs, the automatic stepping algorithm of SeismoBuild imposes a reduction to the load increment or time-step, before the analysis is restarted from the last point of equilibrium (end of previous increment or analysis step). However, in order to prevent ill-behaved analysis (which never reach convergence) to continue on running indefinitely, a maximum step reduction factor is imposed and checked upon after each automatic step reduction. In other words, the new automatically reduced analysis step is confronted with the initial load increment or time-step defined by the user at the start of the analysis, and if the ratio of the former over the latter is smaller than the maximum step reduction value then the analysis is terminated. The **default value**, for the majority of the predefined settings scheme, for this parameter is **0.001**, meaning that if convergence difficulties call for the adoption of an analysis step that is 1000 times smaller than the initial load increment or time-step specified by the user, then the problem is deemed as ill-behaved and the analysis is terminated.

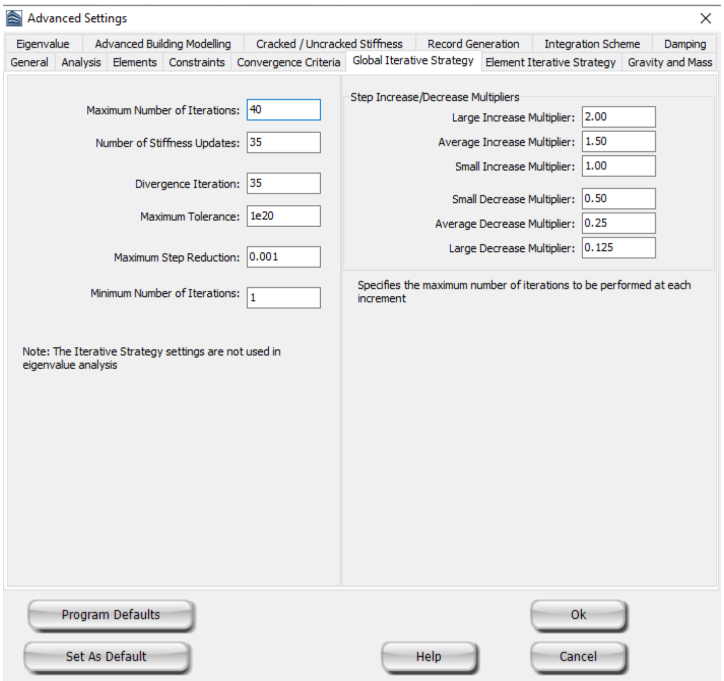
Minimum number of iterations

This parameter defines the minimum number of iterations to be performed within each load increment (analysis step). The **default value**, for the most predefined settings schemes, is **1**. Through this parameter it is possible to achieve a better convergence when the displacement-based criterion is loose and the force-based very strict (this happens in small models in the highly inelastic region).

Step Increase/Decrease Multipliers

The automatic stepping algorithm in SeismoBuild features the possibility of employing adaptive analysis step reductions, which depend on the level of non-convergence verified. When the obtained non-converged solution is very far from convergence, a large step decrease multiplier is used (default = 0.125, i.e. the current analysis increment will be subdivided into 8 equal increments before the analysis is restarted). If, on the other hand, the non-converged solution was very close to convergence, then a small step decrease multiplier is employed (default = 0.5, i.e. the current analysis increment will be subsequently applied in two steps). For intermediate cases, an average step decrease multiplier is utilised instead (default = 0.25, i.e. the current load increment will be split into four equal loads).

Also as described in automatic stepping, once convergence is reached, the load increment or time-step can be gradually increased, up to a size equal to its initial user-specified value. This is carried out through the use of step increasing factors. When the analysis converges in an efficient manner (details in Appendix B - Theoretical background and modelling assumptions), a small step increase multiplier is used (default = 1.0, i.e. the current analysis increment will remain unchanged in subsequent steps). If, on the other hand, the converged solution was obtained in a highly inefficient way (details in Appendix B - Theoretical background and modelling assumptions), then a large step increase multiplier is employed (default = 2.0, i.e. the current load increment will be doubled). For intermediate cases, an average step increase multiplier is utilised instead (default = 1.5, i.e. an increase of 50% will be applied to the current analysis step).



Global Iterative Strategy tab window

NOTE: Users are alerted to the fact that there is no such thing as a set of incremental/iterative parameters that will work for every single type of analysis. The default values of the predefined settings schemes in SeismoBuild will usually work well for the vast majority of applications, but might need to be tweaked and modified for particularly demanding projects, where strong response irregularities (e.g. large stiffness differentials, buckling of some structural members, etc.) occur. As an example, note that a smaller load increment may lead to higher numerical stability, by preventing a structure from following a less stable and incorrect response path, but, if too small, may also render the possibility of achieving convergence almost impossible. Users facing difficulties are advised to consult the Technical Support Forum, where additional guidance and advice is provided.

Element Iterative Strategy

Force-based Element Type / Force-based Plastic-Hinge Elements Type

Individual force-based frame elements require a number of iterations to be carried in order for internal equilibrium to be reached [e.g. Spacone et al. 1996; Neuenhofer and Filippou 1997]. The maximum number of such element loop iterations, together with the corresponding (force) convergence criterion or tolerance, can be defined herein:

- **Element Loop Convergence Tolerance.** The default value is 1e-5 (users may need to relax it to e.g. 1e-4, in case of convergence difficulties)
- **Element Loop Maximum Iterations** (*elm_ite*). The default value is 300 (although this is already a very large value (typically not more than 30 iterations are required to reach convergence), users may need to increase it to 1000 in cases of persistent *elm_ite* error messages)

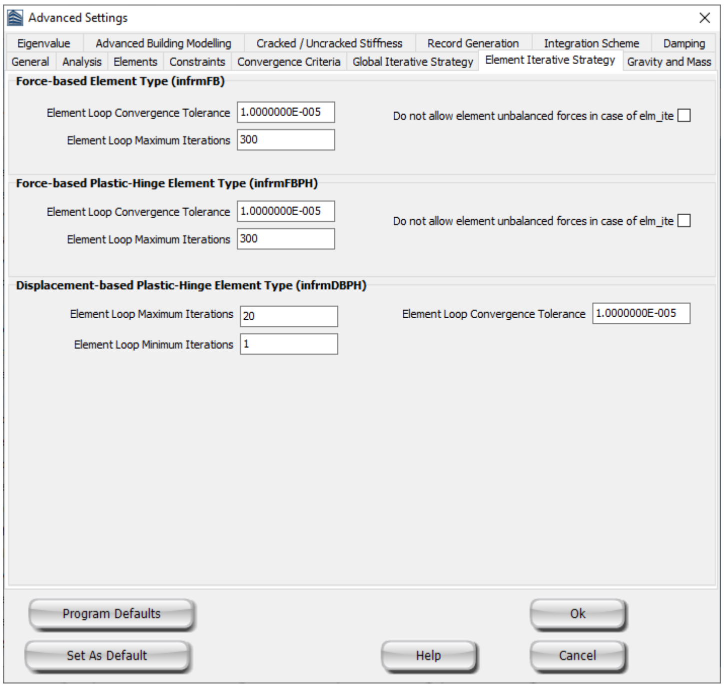
Whilst running an analysis, *elm_inv* and *elm_ite* flag messages may be shown in the analysis log, meaning respectively that the element stiffness matrix could not be inverted or that the maximum allowed number of element loop iterations has been reached. In both cases, the global load increment is subdivided, as described in Appendix A, unless the '*Do not allow element unbalanced forces in case of elm_ite*' option discussed below has been deactivated by the user.

Users are also given the possibility of allowing the element forces to be output and passed on to the global internal forces vector upon reaching the maximum iterations, even if convergence is not achieved. This non-default option may facilitate the convergence of the analysis at global/structure level, since it avoids the subdivision of the load increment (note that the element unbalanced forces are then to be balanced in the subsequent iterations).

Displacement-based Plastic-hinge Element Type

Since the element consists of a series of three sub-elements (two links at the member edges and an elastic frame element in the middle) an iterative procedure is required, in order to achieve internal equilibrium.

The parameters required for the element iterative strategy are the maximum and the minimum iterations allowed, and the value for the convergence norm. It is noted that a relatively small value is given as default for the maximum number of iterations, as it has been observed that typically convergence is achieved within a limited number of iterations. Hence, if convergence is not achieved relatively early, it is highly probable that no convergence will be achieved.



Element Iterative Strategy tab window

Gravity and Mass

The materials' specific weight is automatically defined by the program for the calculation of the distributed self-mass of the structure. More, in the Beams sections, additional distributed load may also be defined, which will serve to define any load not associated to the self-weight of the structure (e.g. finishings, infills, variable loading, etc). Finally, the slab's loads (self weight, additional gravity and live loads) are applied directly to the beams that support the slab.

Here, it is possible for users to define which degrees of freedom are to be considered in the analyses.

Mass Settings

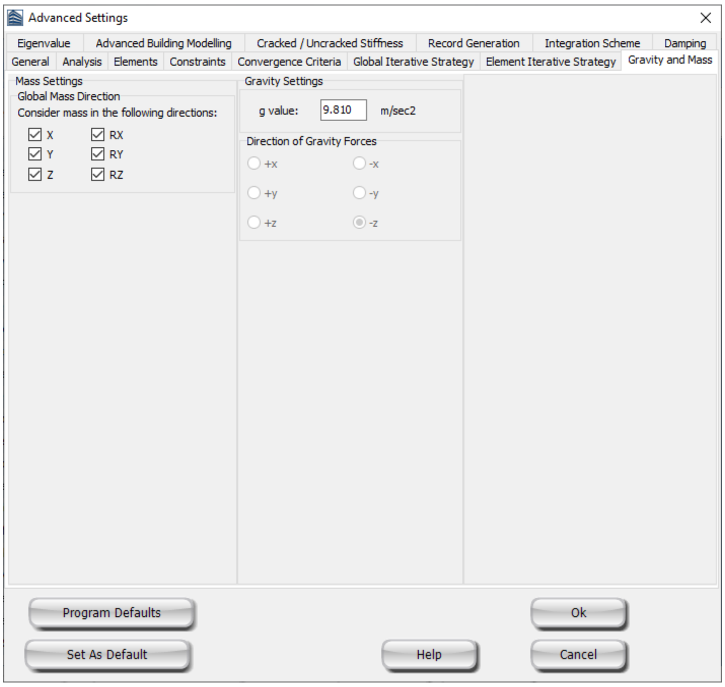
When running analyses, it may sometimes come handy to have the possibility of constraining the dynamic degrees-of-freedom to only a few directions of interest, in order to speed up the analyses or avoid the development of spurious response modes in those directions where the structural mesh was intentionally not adequately devised or refined. This can be done here, by unchecking those dofs that are not of interest (by default, all dofs are activated, i.e. checked).

Gravity Settings

In SeismoBuild loads are defined explicitly in the **Slabs** and **Beams** modules of the Building Modeller.

The user may define the value of acceleration of gravity 'g' (which is to be multiplied by the masses in order to obtain the permanent loads). Clearly, for the vast majority of standard applications, the default value (g=9.81 m/s^2) need not to be modified. The direction of the gravity forces is considered in the -z direction.

NOTE: Stress-recovery (*Advanced Settings > Elements > Carry out Stress Recovery*) may be employed to retrieve correct internal forces when distributed loads are defined (through the definition of material specific weight or of sectional/element additional load).



Gravity & Mass tab window

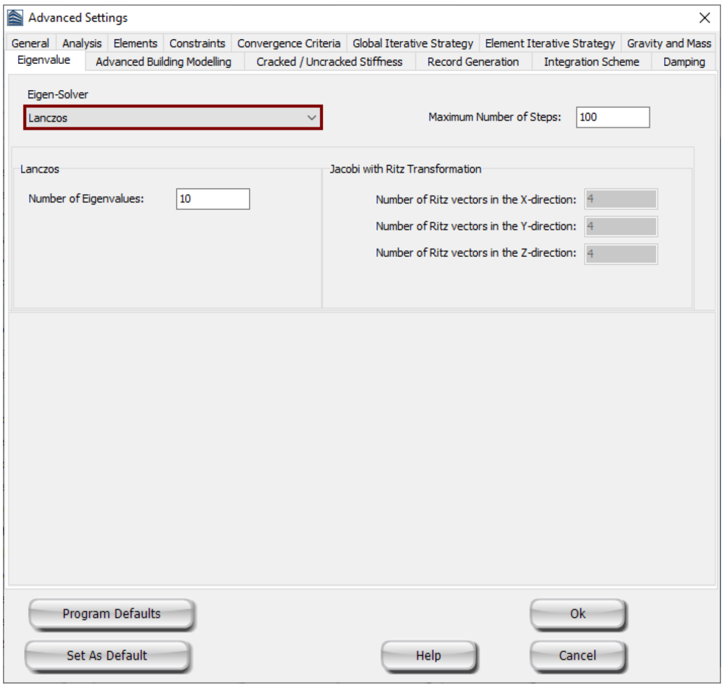
Eigenvalue

Users may choose between two different eigensolvers, the **Lanczos algorithm** presented by Hughes [1987]) or the **Jacobi algorithm with Ritz transformation**, in order to determine the modes of vibration of a structure. When the automatic option is selected the most suitable eigensolver will be used depending on the number of the degrees of freedom of the building. Each algorithm is described in detail hereafter.

Lanczos algorithm

The parameters listed below are used to control the way in which this eigensolver works:

- **Number of eigenvalues.** The maximum number of eigenvalue solutions required by the user. The default value for the default predefined settings scheme is equal to 10, which normally guarantees that, at least for standard structural configurations, all modes of interest are adequately captured. Users might wish to increase this parameter when analysing 3D irregular buildings, where modes of interest might be found beyond the 10th eigensolution.
- **Maximum number of steps.** The maximum number of steps required for convergence to be reached. The default value is 50, for all of the predefined settings schemes, sufficiently large to ensure that, for the vast majority of structural configurations, solutions will always be obtained.



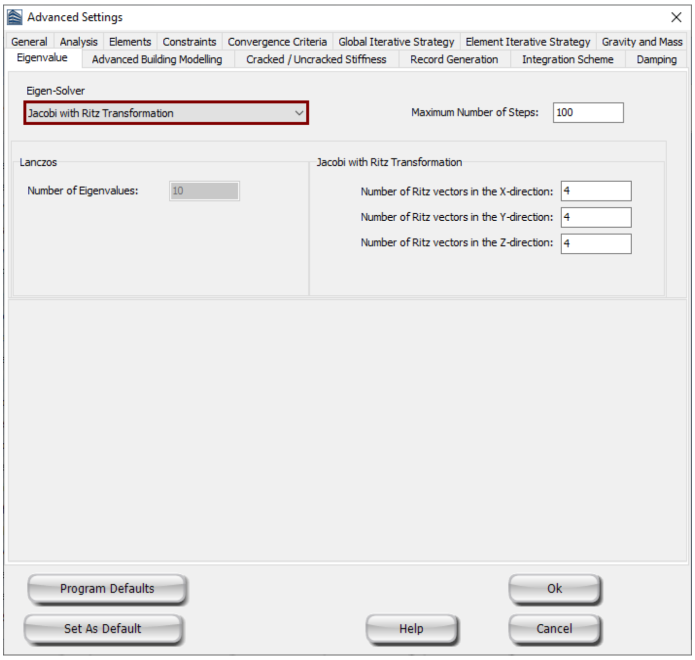
Eigenvalue tab window – Lanczos algorithm

Jacobi algorithm with Ritz transformation

The user may specify:

- **Number of Ritz vectors** (i.e. modes) to be generated in each direction (X, Y and Z)**.** This number cannot exceed the number of the degrees of freedom of the model.
- **Maximum number of steps**. The default value of 50 may, in general, remain unchanged.

NOTE: Users should make sure that the total number of Ritz vectors in the different directions does not exceed the corresponding number of degrees-of-freedom (or of structurally meaningful modes), otherwise unrealistic mode shapes and values will be generated



Eigenvalue tab window – Jacobi algorithm

Advanced Building Modelling

The concrete and steel material types and the frame element types that will be used to model the structural members in SeismoBuild are defined herein, together with other modelling options, such as the modelling of offsets at beam-column joints, the discretisation of the slabs, and the determination of the Control Node.

Materials Modelling

Materials that are to be used within a SeismoBuild project come defined in the Advanced Building Modelling tab. Eight material types are available in SeismoBuild, four types for concrete and four for steel. The complete list of materials is proposed hereafter:

- Mander et al. nonlinear concrete model - con_ma
- Trilinear concrete model - con_tl
- Chang-Mander nonlinear concrete model – con_cm
- Kappos and Konstantinidis nonlinear concrete model - con_hs
- Engineered cementitious composites model – con_ecc
- Kent Scott Park concrete model – con_ksp
- Menegotto-Pinto steel model - stl_mp
- Giuffre-Menegotto-Pinto steel model - stl_gmp
- Bilinear steel model - stl_bl
- Bilinear steel model with isotropic strain hardening- stl_bl2
- Ramberg-Osgood steel model - stl_ro
- Dodd-Restrepo steel model – stl_dr
- Monti-Nuti steel model - stl_mn
- Buckling Restrained Steel Brace model – stl_brb

For a comprehensive description of the material types, refer to Appendix C – Materials.

Frame Elements Modelling

Different frame element types may be employed for columns/beams and walls. Users may select between inelastic force-based frame elements (infrmFB), inelastic plastic-hinge force-based frame elements (infrmFBPH) and inelastic plastic-hinge displacement-based frame elements (infrmDBPH).

Further, it is possible to assign the inelastic displacement-based frame element type (infrmDB) to short members, a choice that improves both the accuracy and the stability of the analysis. Users can determine the maximum length of the short members (1.0m by default). Users can also determine the maximum length of the members, below which the elfrm element type is employed (0.4m by default). The inelastic plastic-hinge force-based frame element, infrmFBPH, is selected for columns/beams and walls in the default predefined settings scheme, which should work well for most practical applications. The maximum frame element length for the discretization of Strip Footings is defined in this tab (1.0m by default).

Rigid Ends Definition

The choice whether to include or not rigid ends at the frame elements to model beam-column joints is also done herein. It is noted that these rigid ends are included in the model, only when the length of a member's rigid end is larger than the specified value, otherwise the beam is connected to the column node directly.

Slabs Discretization

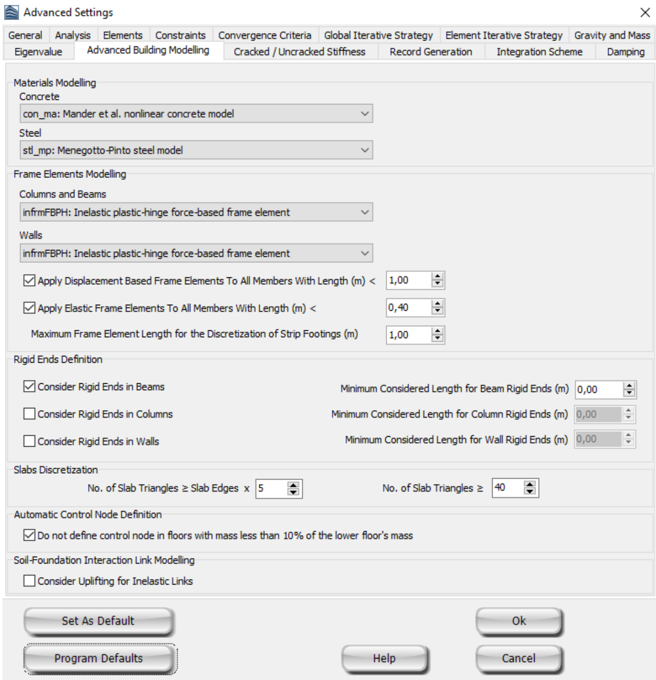
Users may select the number of triangles ,to which the slabs are to be subdivided, so that their weight and mass are appropriately distributed in the supporting beams and columns. This may be done in two ways, either by assigning the exact number of triangles or by providing it as a multiplier of the slabs' edges, which is an indication of the complexity of the slab. Obviously, an increased number of triangles leads to a better and more accurate distribution of loads to vertical members, however it also leads to longer slab analysis.

Control Node Definition

The choice of defining the control node at the upper floor or at the floor lower to that (in the cases of a top floor mass less than 10% of that at the lower floor) is provided.

Soil-Foundation Interaction Link Modelling

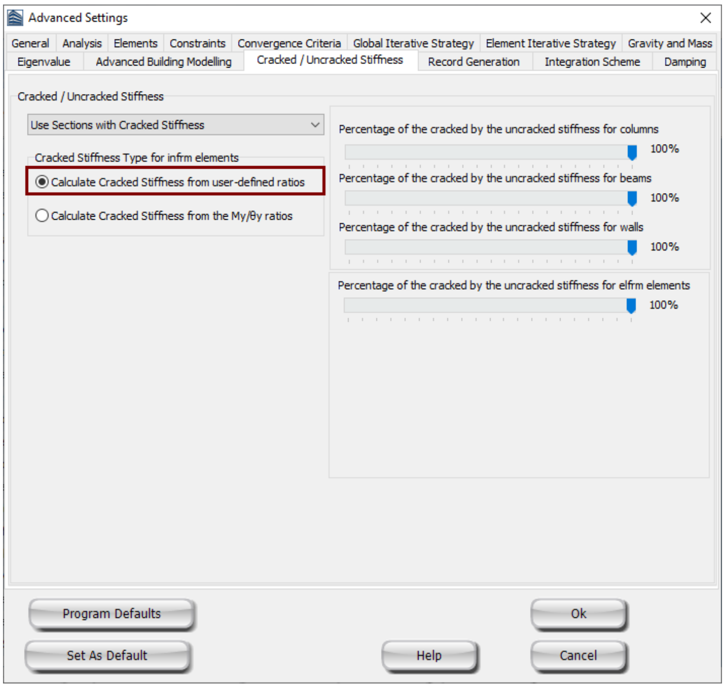
When the Consider Uplifting for Inelastic Links check-box is selected, the foundation link element has zero stiffness during the uplift of the footing.



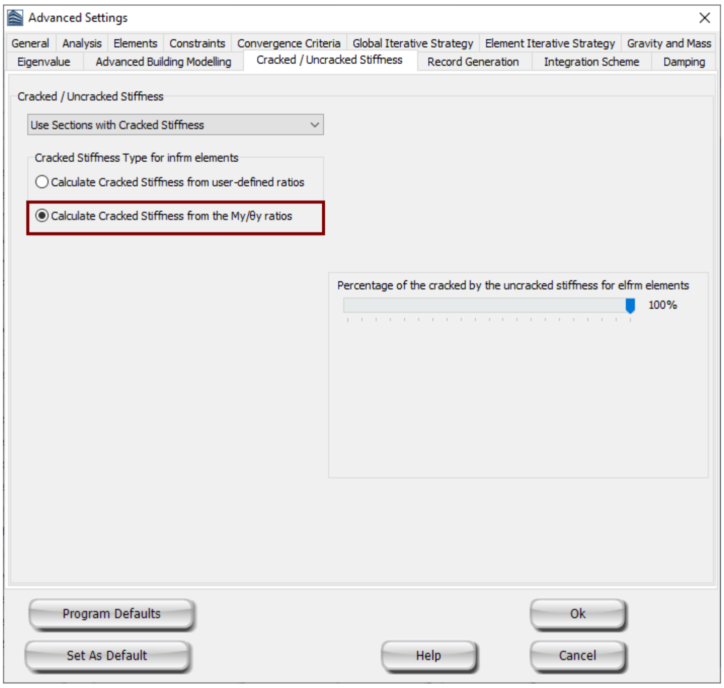
Advanced Building Modelling tab window

Cracked/Uncracked Stiffness

Users may take into account the effect of cracking during the linear analyses, i.e. Eigenvalue and Response Spectrum analyses, by selecting to use sections with cracked stiffness. The cracked stiffness may be defined as a percentage of the corresponding uncracked stiffness, or, in the case of inelastic frame elements only, from the section's M_y/θ_y (bending moment at yield/chord rotation capacity at yield) ratio. In the latter case, users should select the employed Code for the calculation of the chord rotation capacity at yield.



Cracked Stiffness tab window- user-defined ratios

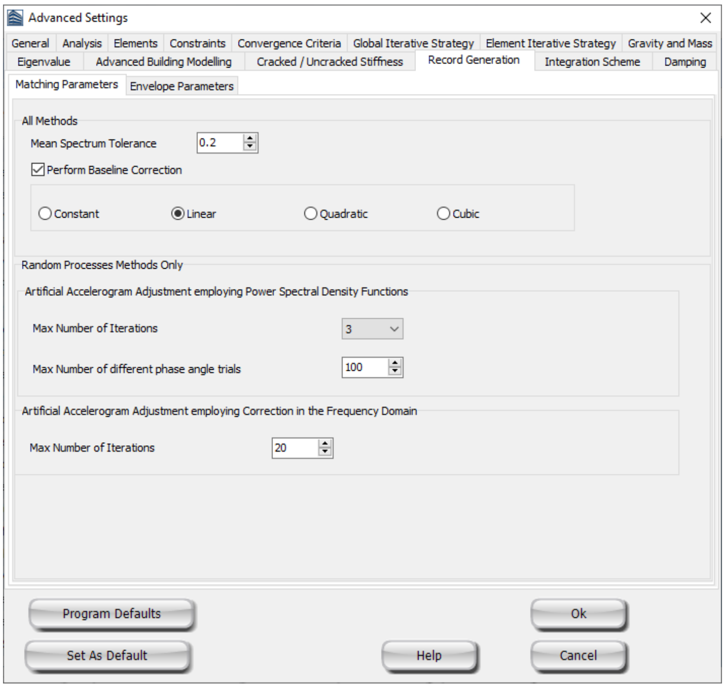


Cracked Stiffness tab window- M_y/θ_y ratios

Record Generation

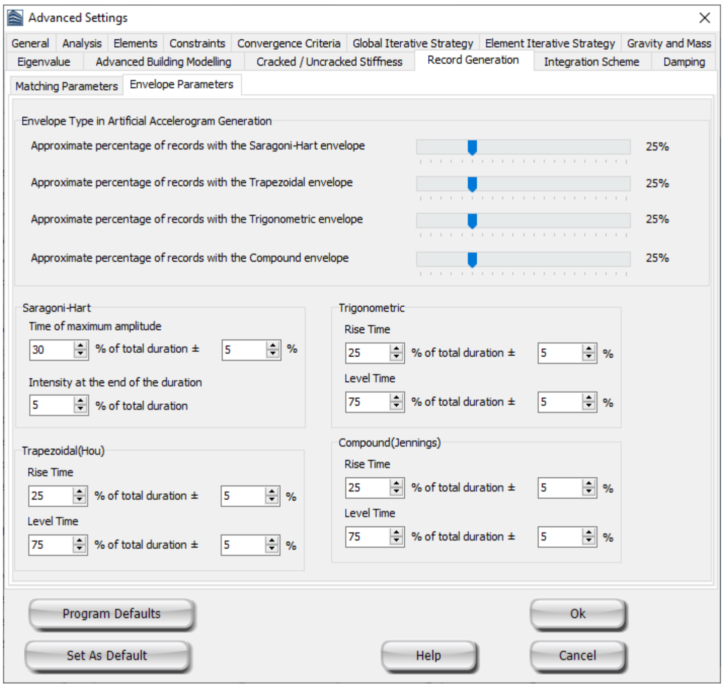
In this section user are able to define the basic settings for the record generation process in nonlinear dynamic analysis. Two pages are available.

In the first page (Matching Parameters) the parameters for the target spectrum matching algorithm are set. The spectrum tolerance (i.e. accepted difference between the spectrum of the generated accelerogram and that of the target spectrum, computed as the mean of the absolute values of positive and negative differences) can be modified here. Users may also decide whether to carry out or not baseline correction in the generation of the artificial accelerograms. By default, a quadratic baseline correction is selected. In addition, users can also customise the parameters used in the random processes methods (Artificial Accelerogram Generation and Artificial Accelerogram Generation & Adjustment), namely: random process starter, maximum number of iterations, maximum number of different phase angle trials, and the number of points used for Power Spectra Density Function (PSDF) calculation. The only setting with practical importance is the Mean Spectrum Tolerance; other parameters need not to be modified, unless there is a specific reason.



Record Generation tab window- Matching Parameters

In the second page (Envelope Parameters) the types of the envelope shapes to be employed by the Artificial Accelerogram Generation and the Artificial Accelerogram Generation & Adjustment methods are set. Users may choose the percentage of each type of envelope in all the records, noting however that a random process is executed internally by the program; hence these setting literally refer to the probability that such an envelope is employed. Finally, the basic settings for the definition of the envelope shapes are also defined herein (time of maximum amplitude, rise and level times and intensity at the end of the duration).



Record Generation tab window- Envelope Parameters

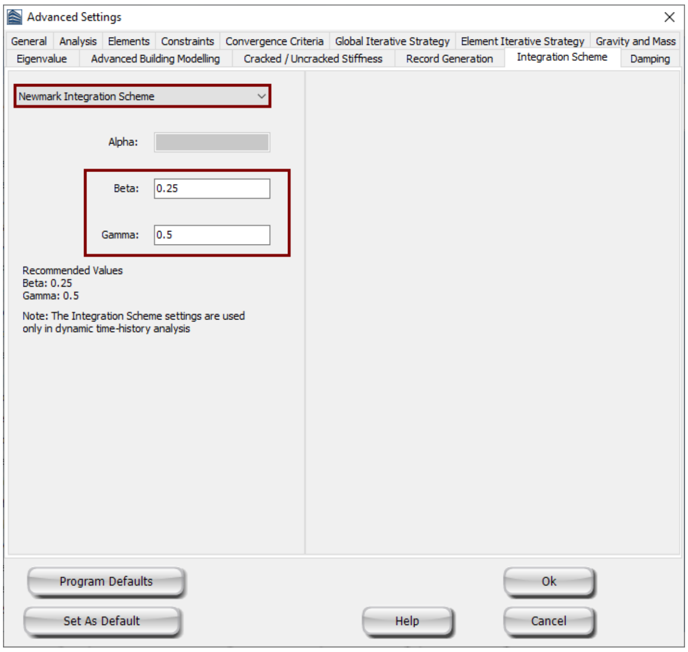
Integration Scheme

In nonlinear dynamic analysis, a numerical direct integration scheme must be employed in order to solve the system of equations of motion [e.g. Clough and Penzien, 1993; Chopra, 1995]. In SeismoBuild, such integration can be carried out by means of two different implicit integration algorithms that the user may choose (i) the **Newmark integration scheme** [Newmark, 1959] or (ii) the **Hilber-Hughes-Taylor integration algorithm** [Hilber et al., 1977].

> **NOTE: Hilber-Hughes-Taylor integration algorithm is the default option.**

Newmark integration scheme

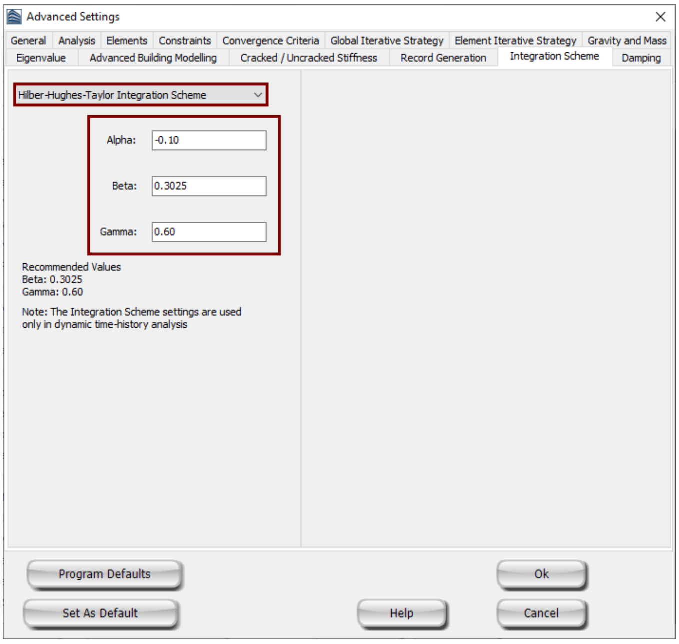
The **Newmark integration scheme** requires the definition of two parameters: beta (β) and gamma (γ). Unconditional stability, independent of time-step used, can be obtained for values of $\beta \geq 0.25(\gamma+0.5)^2$. In addition, if $\gamma=0.5$ is adopted, the integration scheme reduces to the well-known non-dissipative trapezoidal rule, whereby no amplitude numerical damping is introduced, a scenario that may prove to be advantageous on many applications. The default values are therefore $\beta=0.25$ and $\gamma=0.5$.



Integration Scheme tab window - Newmark

Hilber-Hughes-Taylor integration scheme

The **Hilber-Hughes-Taylor algorithm**, on the other hand, calls for the characterisation of an additional parameter alpha (α) used to control the level of numerical dissipation. The latter can play a beneficial role in dynamic analysis, mainly through the reduction of higher spurious modes' contribution to the solution (which typically manifest themselves in the form of very high short-duration peaks in the solution), thus increasing both the accuracy of the results as well numerical stability of the analysis. According to its authors [Hilber et al., 1977], and as confirmed in other studies [e.g. Broderick et al., 1994], optimal solutions, in terms of solution accuracy, analytical stability and numerical damping are obtained for values of $\beta=0.25(1-\alpha)^2$ and $\gamma=0.5-\alpha$, with $-1/3\leq\alpha\leq0$. In SeismoBuild, the default values are $\alpha=-0.1$, $\beta=0.3025$ and $\gamma=0.6$.



Integration Scheme tab window - Hilber-Hughes-Taylor

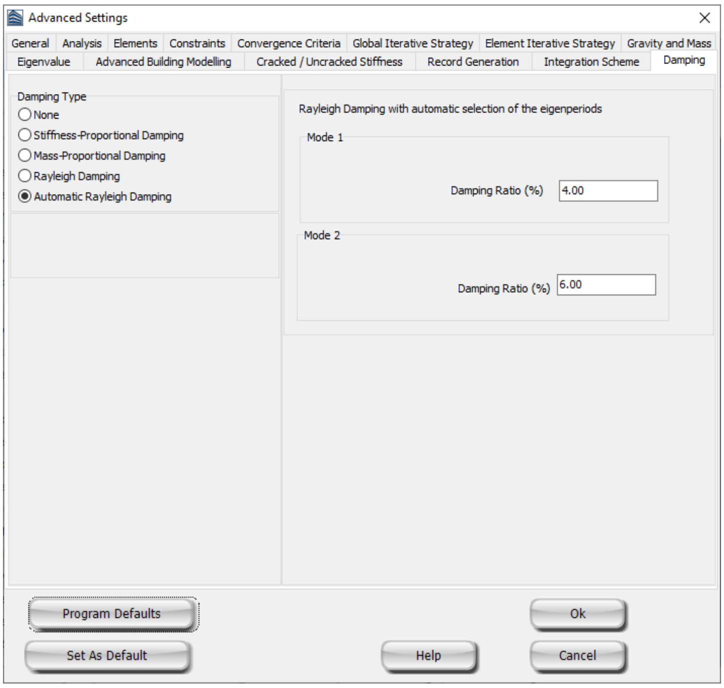
NOTE: For further discussion and clarification on issues of step-by-step solution procedures, explicit vs. implicit methods, stability conditions, numerical damping, and so on, users are strongly advised to refer to available literature, such as the work by Clough and Penzien [1993], Cook et al. [1988] and Hughes [1987], to name but a few.

Damping

In nonlinear dynamic analysis, hysteretic damping, which usually is responsible for the dissipation of the majority of energy introduced by the earthquake action, is already implicitly included within the nonlinear fibre model formulation of the inelastic frame elements or within the nonlinear force-displacement response curve formulation used to characterise the response of link elements. There is, however, a relatively small quantity of non-hysteretic type of damping that is also mobilised during dynamic response of structures, through phenomena such friction between structural and non-structural members, friction in opened concrete cracks, energy radiation through foundation, etc, that might not have been modelled in the analysis. Traditionally, such modest energy dissipation sources have been considered through the use of Rayleigh damping [e.g. Clough and Penzien, 1993; Chopra, 1995] with equivalent viscous damping values (ξ) varying from 1% to 8%, depending on structural type, materials used, non-structural elements, period and magnitude of vibration, mode of vibration being considered, etc [e.g. Wakabayashi, 1986].

In the *Damping* dialog box, the user may therefore choose:

- not to use any viscous damping;
- to employ **stiffness-proportional damping**;
- to introduce **mass-proportional damping;**
- to utilise **Rayleigh damping**;
- to utilise **Rayleigh damping** with the automatic consideration of the values of the first and second periods along the direction of the main excitation. This last option is the program default.



Damping tab window

Stiffness-proportional damping

For **stiffness-proportional damping**, the user is asked to enter the value of the stiffness matrix multiplier (α_K) that he/she intends to use.

Typically, though not exclusively, such value is computed using the following equation:

$$\alpha_K = \frac{T\xi}{\pi}$$

The user is also asked to declare if the damping is proportional to (i) the initial stiffness or (ii) the tangent stiffness.

NOTE 1: The value of the *tangent stiffness-proportional damping* matrix is updated at every load increment, not at every iteration, since the latter would give rise to higher numerical instability and longer run times.

NOTE 2: Should numerical difficulties arise with the use of *tangent stiffness-proportional damping*, the user is then advised to employ *initial stiffness-proportional damping* instead, using however a reduced equivalent viscous damping coefficient, so as to avoid the introduction of exaggeratedly high viscous damping effects. Whilst a 2-3% viscous damping might be a reasonable assumption when analysing a reinforced structure using *tangent stiffness-proportional damping*, a much lower value of 0.5-1% damping should be employed if use is made of its *initial stiffness-proportional damping* counterpart.

Mass-proportional damping

For **mass-proportional damping**, the user is asked to enter the value of the mass matrix multiplier (α_M) that he/she intends to use.

Typically, though not exclusively, such value is computed using the following equation:

$$\alpha_M = \frac{4\pi\xi}{T}$$

Rayleigh damping

For **Rayleigh damping**, the user is asked to enter the period (T) and damping (ξ) values of the first and last modes of interest (herein named as modes 1 and 2).

The mass-proportional (α_M) and stiffness-proportional (α_K) matrices multiplying coefficients are then computed by the program, using the expressions given below, which ensure that true Rayleigh damping is obtained (if arbitrarily defined coefficients would be used, this would imply that matricial rather than Rayleigh damping would be employed):

$$\alpha_M = 4\pi \frac{\xi_1 T_1 - \xi_2 T_2}{T_1^2 - T_2^2} \qquad and \qquad \alpha_K = \frac{T_1 T_2}{\pi} \frac{\xi_2 T_1 - \xi_1 T_2}{T_1^2 - T_2^2}$$

NOTE 1: A relatively large variety of different types of matricial damping exists and is used in different FE codes. These variations may present advantages with respect to traditional Rayleigh damping; e.g. reducing the level of damping that is introduced in higher modes and so on. However, we believe that such level of refinement and versatility is not necessarily required for the majority of analysis, for which reason only the above three viscous damping modalities are featured in the program.

NOTE 2: There is significant scatter in the different proposals regarding the actual values of *equivalent viscous damping* to employ when running dynamic analysis of structures, and the user is advised to investigate this matter thoroughly, in order to arrive at the values that might prove to be more adequate to his/her analyses. Herein, we note simply that the value will depend on the material type (typically higher values are used in concrete, with respect to steel, for instance), structural configuration (e.g. an infilled multi-storey frame may justify higher values with respect to a SDOF bridge bent), deformation level (at low deformation levels it might be justified to employ equivalent viscous damping values that are higher than those used in analyses where buildings are pushed deep into their inelastic range, since in the latter case contribution of non-structural elements is likely to be of lower significance, for instance), modelling strategy (e.g. in fibre modelling cracking is explicitly account for and, as such, it does not need to be somehow represented by means of equivalent viscous damping, as is done instead in plastic hinge modelling using bilinear moment-curvature relationships).

NOTE 3: Damping forces in models featuring elements of very high stiffness (e.g. bridges with stiff abutments, buildings with stiff walls, etc) may become unrealistic - overall damping in a bridge model can introduce significant damping forces, due e.g. to very high stiffness of abutments.

Eigenvalue Analysis

EIGENVALUE PARAMETERS

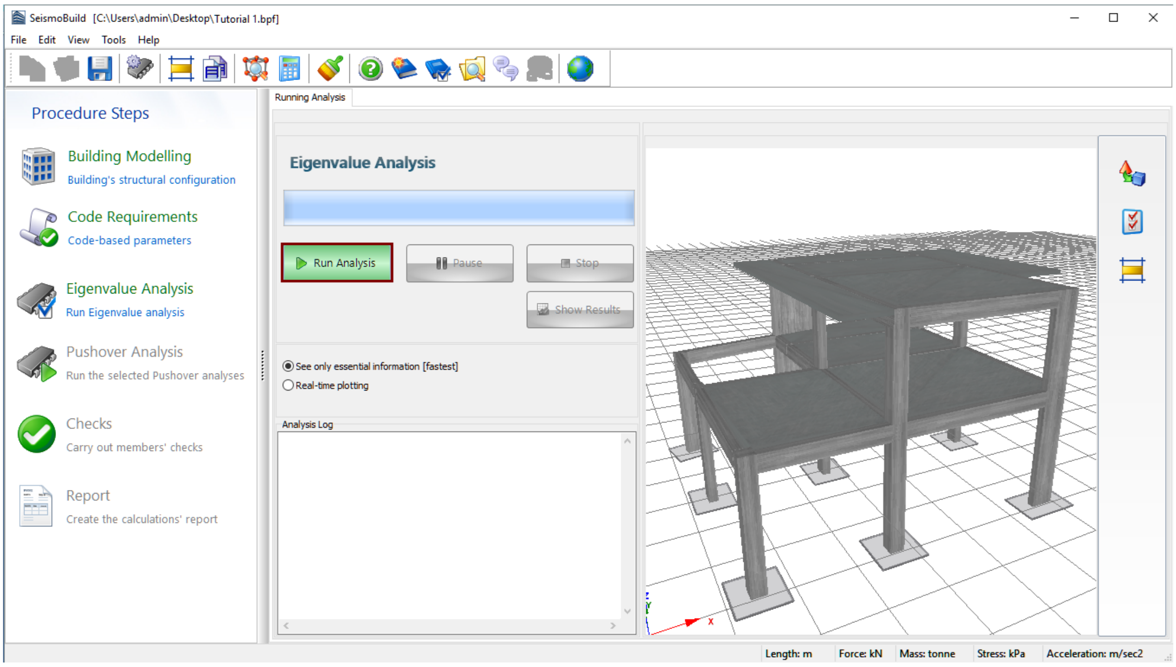
The efficient **Lanczos algorithm** [Hughes, 1987] is used by default for the evaluation of the structural natural frequencies and mode shapes. However, the **Jacobi algorithm with Ritz transformation** may also be chosen by the user in the Advanced Settings module.

Eigenvalue analysis is a purely elastic type of structural analysis, since material properties are taken as constant throughout the entire computation procedure. However, in SeismoBuild inelastic frame elements are used, that will be employed in all the analyses, including the eigenvalue one. Hence, different material and section types are employed in the characterisation of the elements' sectional mechanical properties, which are not defined by the user, but internally determined by the program, using classic formulae that can be found on any book or publication on basics of structural mechanics [e.g. Gere and Timoshenko, 1997; Pilkey, 1994].

NOTE: Concrete confinement will increase the compressive strength of the material, and hence the stiffness of the member, leading thus to shorter periods of vibration.

PROCESSOR

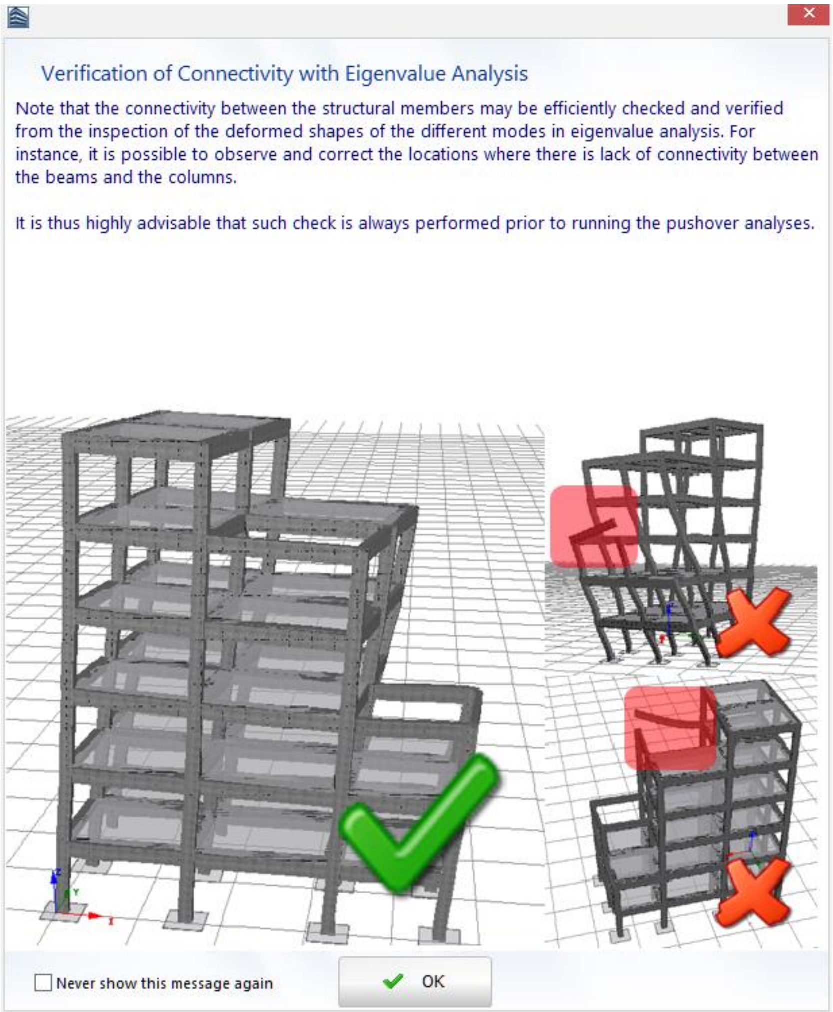
Having defined the eigenvalue analysis' parameters in the Advanced Settings module, the user is then ready to run the analysis. This is carried out in the **Eigenvalue Analysis** area of SeismoBuild, by selecting the Run Analysis button.



Processor area

As the analysis is running, a progress bar provides the user with a percentage indication of how far has the former advanced to. Users can in this manner quickly assess the waiting time required for the analysis to be completed, and hence quickly plan their subsequent work schedule, although, the eigenvalue analysis takes just few seconds to be completed.

When the analysis is completed, an informative message appears suggesting to check the deformed shapes of the different modes for possible connectivity problems between the beams and the columns.



Verification of Connectivity with Eigenvalue Analysis

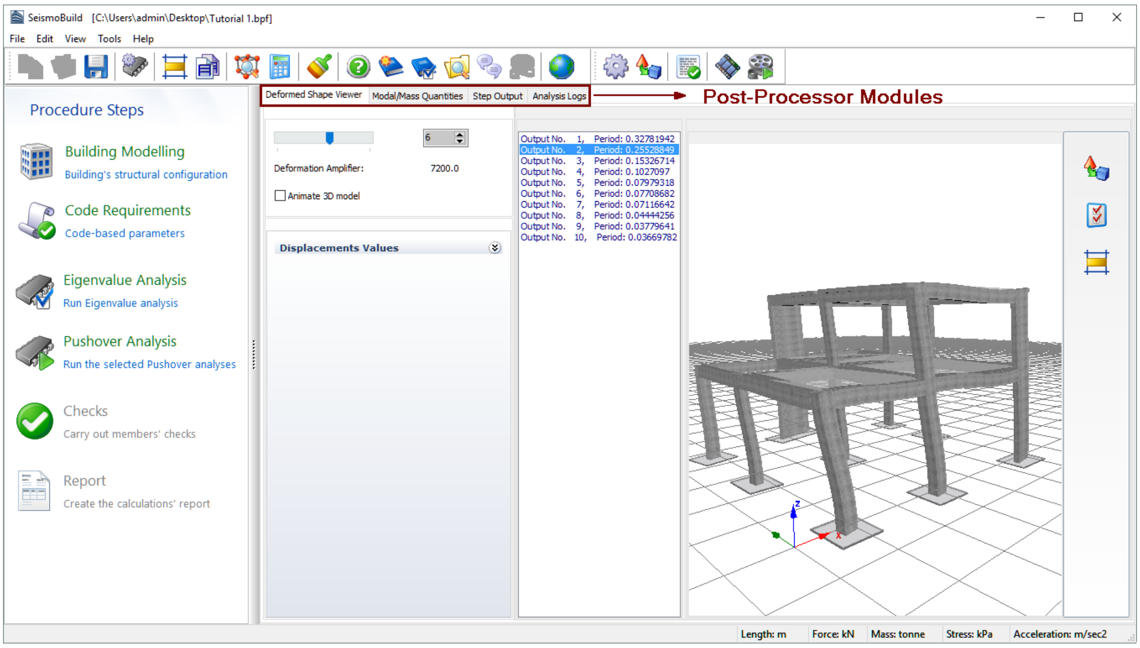
POST-PROCESSOR

After running the Eigenvalue analysis users may go to view the results by clicking the Show Results button. The results of the analysis are saved in a SeismoBuild Results File, distinguishable by its *_Eig.brf extension, with the same name as the input project file.

The Show Results area features a series of modules where results from Eigenvalue analysis can be viewed in table or graphical format, and then copied into any other Windows application (e.g. tabled results can be copied into a spreadsheet like Microsoft Excel, whilst results plots can be copied into a word-processing application, like Microsoft Word).

The available modules are listed below and will be described in the following paragraphs:

- Deformed Shape Viewer
- Modal/Mass Quantities
- Step Output
- Analysis Logs

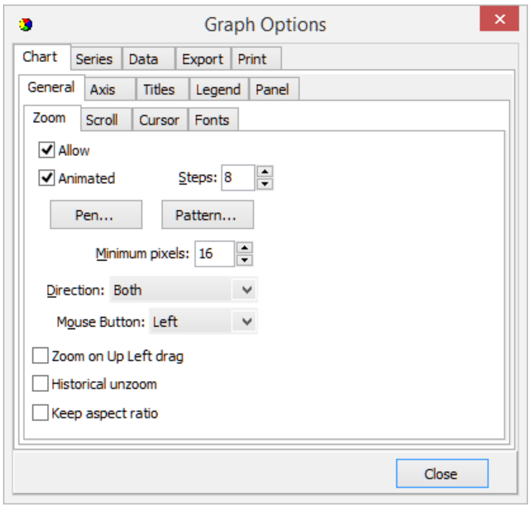


Post-Processor Modules

There are some general operations that apply to all the post -processor modules. For example, the way in which model components (e.g. nodes, sections, elements, etc.) appear on all dialogue boxes in the post-processor.

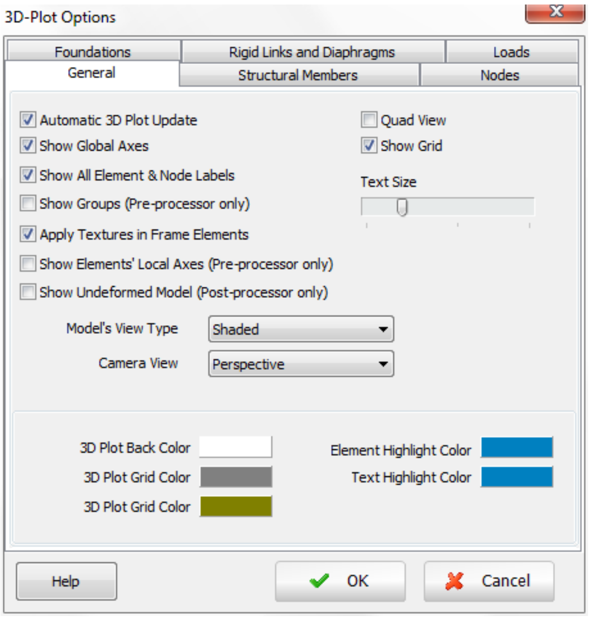
Plot Options

All graphs displayed in the Post-Processor modules can be tweaked and customised using the *Plot Options* facility, available from the main menu (*Tools > Plot Options…*), toolbar button  or right-click popup menu. The user can then change the characteristics of the lines (colour, thickness, style, etc.), the background (colour, gradient), the axes (colour, font size and style of labels etc.) and the titles of the plot. Through the *Save Plot Settings...* and the *Load plot Settings...*, available on the right click popup menu, the plot settings may be saved and retrieved, respectively, to be applied to other plots.



Plot Options – General

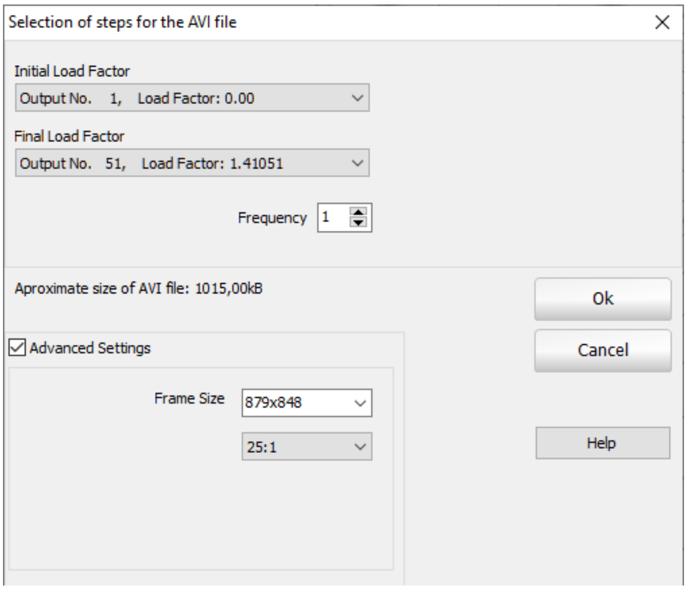
Further, within the 3D Plot Options menu, accessible when the 3D model is visible, there are a number of submenus from which users can not only select which model components (nodes, structural members, etc.) to show in the plot but also change a myriad of settings such as the colour/transparency of elements, the plot axes and background panels, the colour/transparency of load symbols, the colour of text descriptors, and so on.



3D Plot Options – General

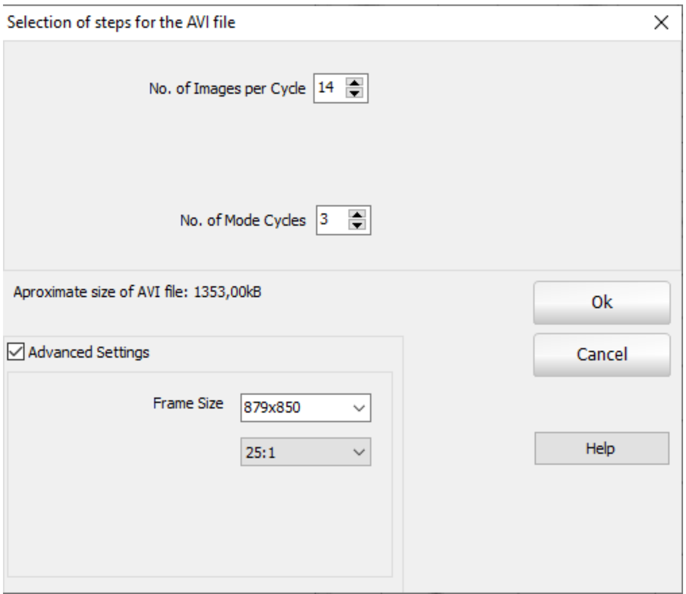
Creating an analysis movie

SeismoBuild provides users with the possibility of creating a movie where the vibration mode of a structure (as obtained from eigenvalue analysis) is animatedly depicted. This facility can be accessed through the program main menu (*Tools > Create AVI File...*) or through the respective toolbar button 
.



Selection of steps for the AVI file

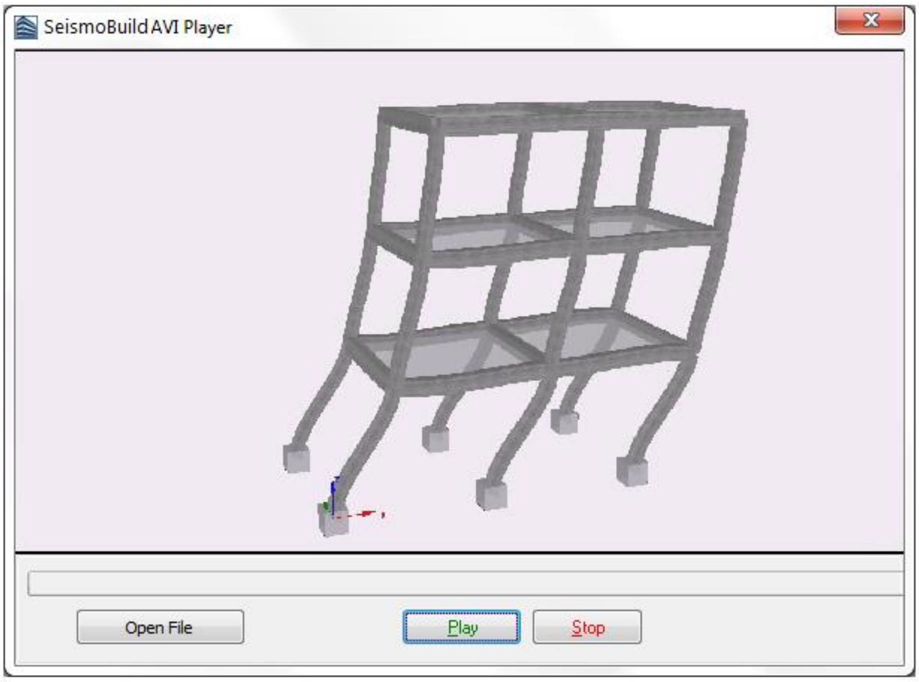
If a user wishes to create a movie illustrating a given vibration mode of a particular structure, then he/she must define the number of mode cycles to be created (i.e. how many times will the modal animation be repeated) and the number of images/frames to be used per cycle. Evidently, the highest the number of interim frames, the smoothest the animation, but also the largest the movie file becomes.



Selection of steps for the AVI file

Before creating the animation, users are advised to customise the 3D Plot to their needs and likings, since these settings will reflect the look and feel of the movie. In particular, it is noted that during movie creation, the axes of the plot are not automatically updated, thus implying that, before initiating the creation process, users should set the axes to their largest needed values. The latter can be done by viewing an output shape where deformations are at their highest.

Once the animation has been created, users can verify its adequacy through the AVI Viewer incorporated in SeismoBuild, accessible from the program main menu (*Tools > Show AVI File...*) or through the respective toolbar button .

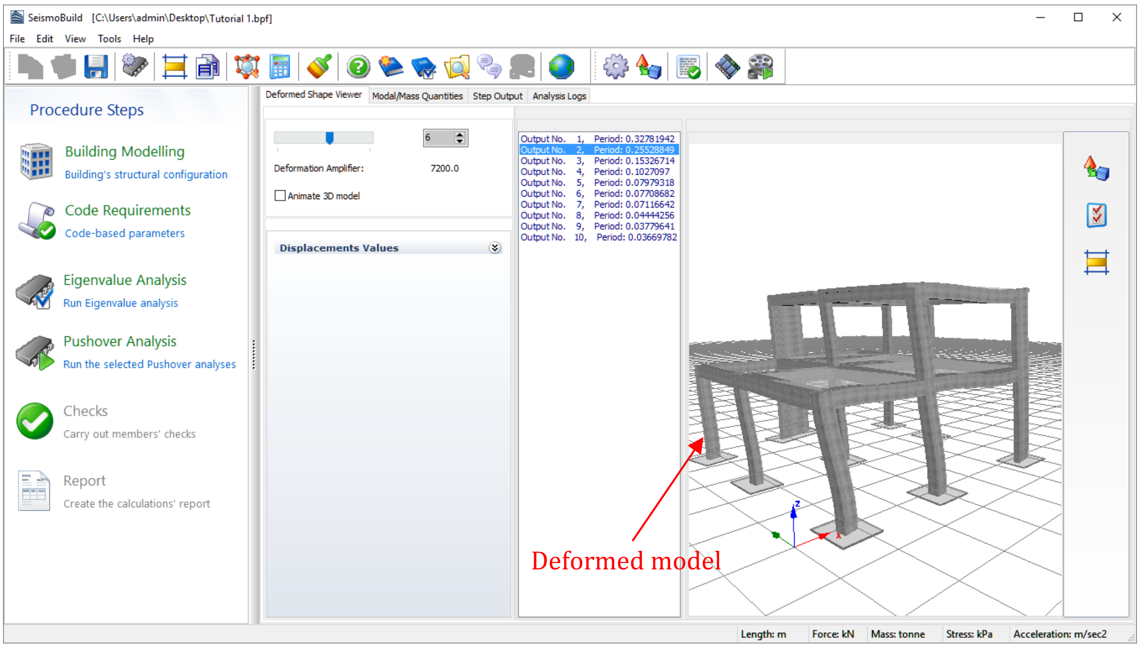


SeismoBuild AVI player

Animations created in SeismoBuild (i.e. AVI movies) can also be opened by other Windows applications such as Windows Media Player or, perhaps more importantly, Microsoft PowerPoint, where they can be used in multimedia presentations.

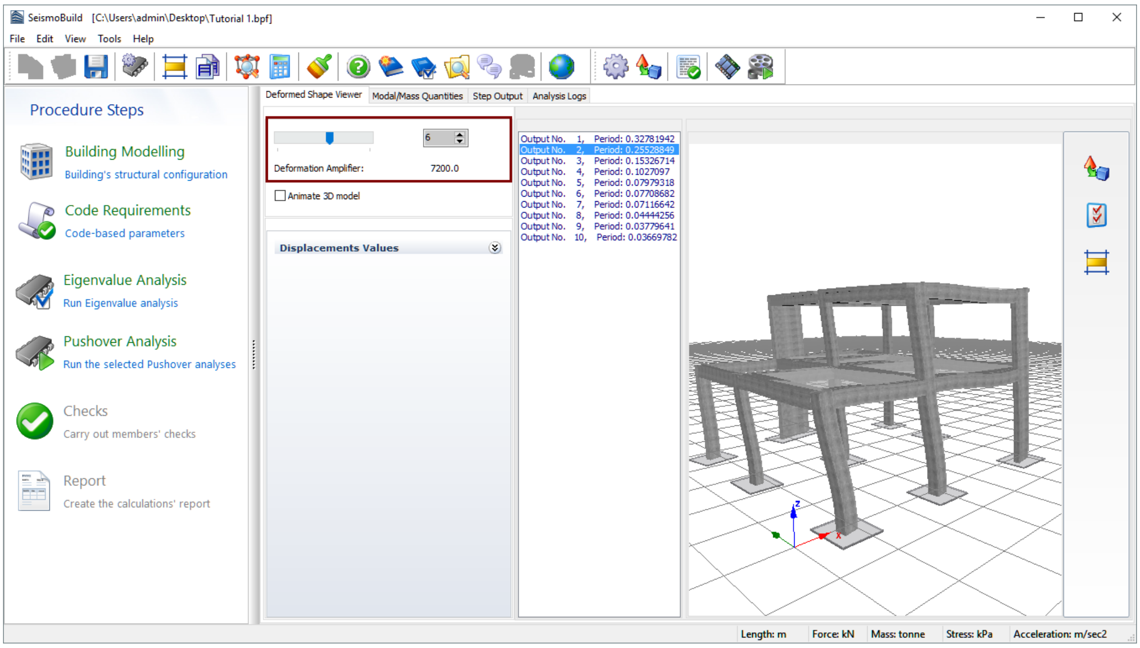
Deformed Shape Viewer

With the Deformed Shape Viewer, users have the possibility of visualising the deformed shape of the model for every period of the model (click on the desired output identifier to update the deformed shape view).

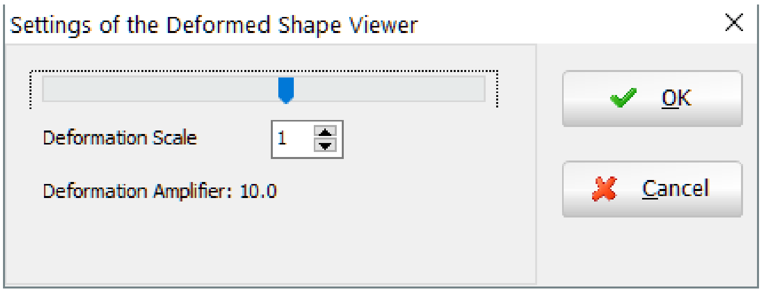


Deformed Shape Viewer

The deformed shape plot can be tweaked and customised using the 3D Plot options and then copied to any Windows application by means of the Copy 3D Plot facility. In addition, and whenever the real-time deformed shape of the structure is difficult to interpret (because displacements are either too large or too small), users can make use of the Deformation Amplifier available on the left of the Deformed Shape Viewer tab or of the Deformed Shape Multiplier, available from the right-click popup menu, the main menu (*Tools > Deformed Shape Settings...*) or through the corresponding toolbar button , to better adapt the plot.



Deformed Shape Viewer-Deformation Amplifier

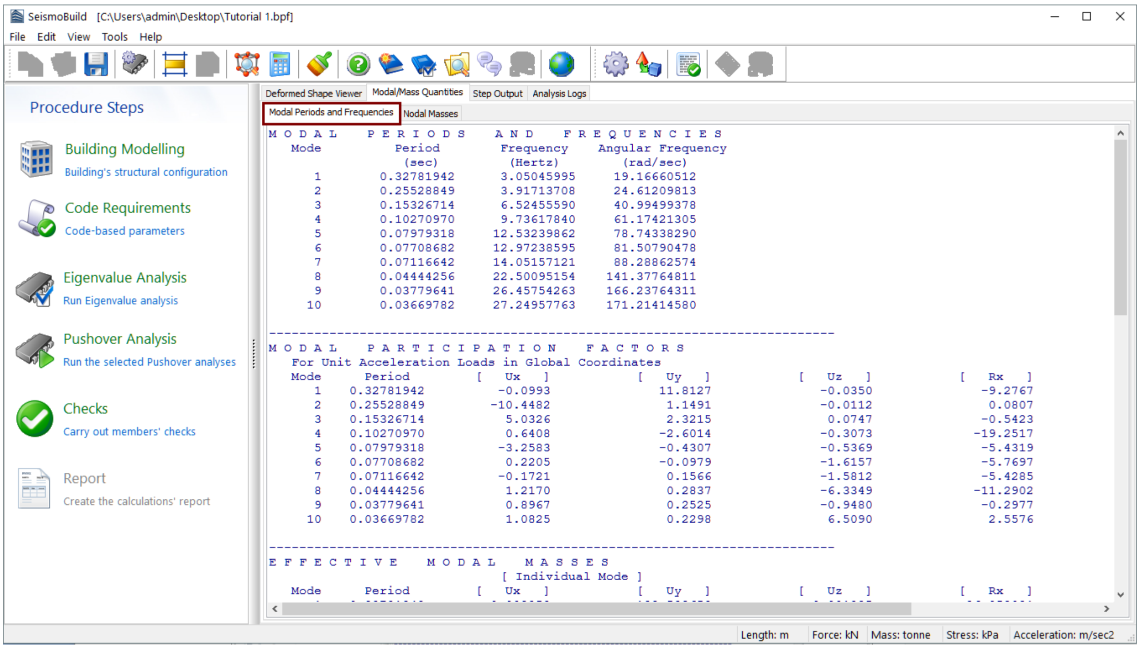


Deformed Shape Settings

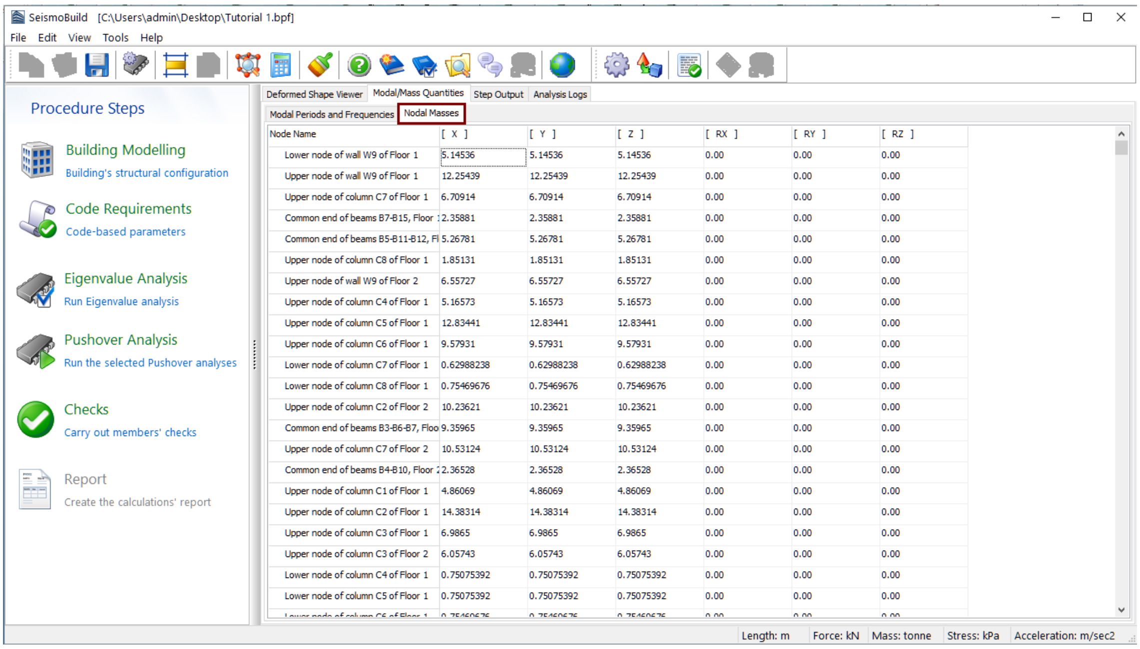
Finally, the vibration mode of a structure (as obtained from eigenvalue analysis) is animatedly depicted through the selection of the Animate 3D model check-box.

Modal/Mass Quantities

The Modal/Mass Quantities module provides a summary of (i) the main eigenvalue results (i.e. the natural period/frequency of vibration of each mode, the modal participation factors and the effective modal masses), and (ii) the nodal masses. These results can be easily copied to a text editor, through the right-click popup menu.



Modal/Mass Quantities Module – Modal Periods and Frequencies



Modal/Mass Quantities Module – Nodal Masses

Regarding the nodal masses, SeismoBuild provides a table in which are summarized the masses of the nodes for each degree of freedom (also for rotation). For a particular node, the rotational mass is computed as the rotational mass defined by the user for that node, plus the translational mass at that node times the square of the distance to the centre of gravity of the model.

The **modal participation factors**, obtained as the **ratio** between the **modal excitation factor** ($L_n=\Phi_n{}^T*M$) and the **generalised mass** ($M_n=\Phi_n{}^T*M*\Phi_n$), provide a measure as to how strongly a given mode n participates in the dynamic response of a structure. However, since mode shapes Φ_n can be normalised in different ways, the absolute magnitude of the modal participation factor has in effect no meaning, and only its relative magnitude with respect to the other participating modes is of significance. [Priestley et al., 1996]

For the reason above, and particularly for the case of buildings subjected to earthquake ground-motion, it is customary for engineers/analysts to use the effective modal mass ($m_{eff,n}=L_n{}^2/M_n$) as a measure of the relative importance that each of the structure's modes has on its dynamic response. Indeed, since $m_{eff,n}$ can be interpreted as the part of the total mass M of the structure that is excited by a given mode n, modes with high values of effective modal mass are likely to contribute significantly to response.

NOTE 1: Users are advised to refer to the available literature [e.g. Clough and Penzien, 1993; Chopra, 1995] for further information on modal analysis and respective parameters.

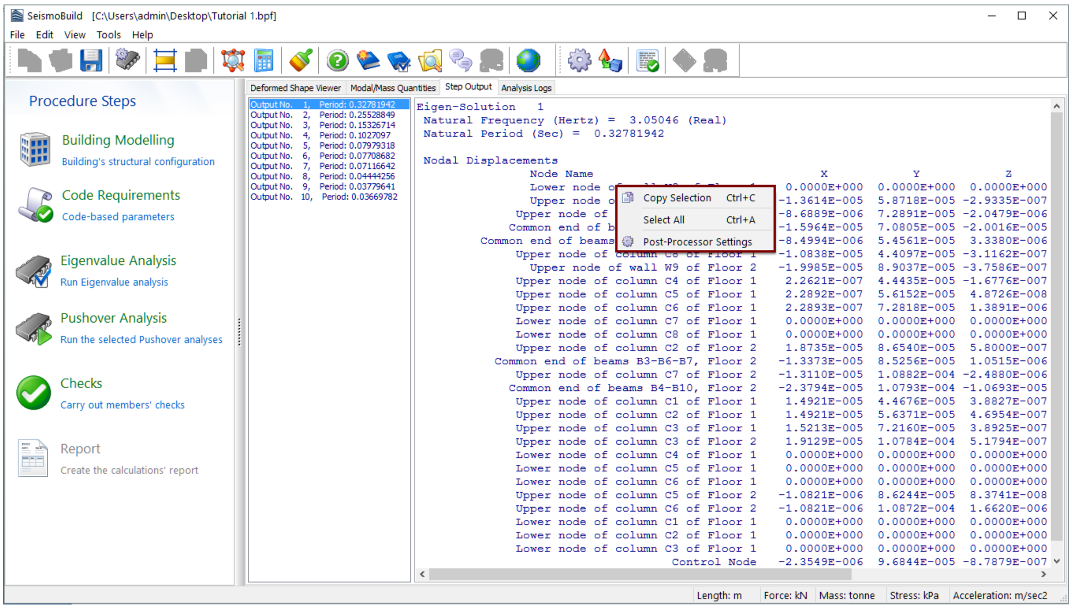
NOTE 2: The mode shapes are normalised so that Φn=1.

NOTE 3: MPFs for rotations are calculated considering a transformation matrix defined as follows (where x_0, y_0, z_0 are the coordinates of the centre of mass), so that the modal excitation factor becomes $L_n=\Phi_n^T*M*T_i$, from which the effective modal mass (as for the translational DOFs).

$$T_i = \begin{bmatrix} 1 & 0 & 0 & 0 & (z-z_0) & -(y-y_0) \\ 0 & 1 & 0 & -(z-z_0) & 0 & (x-x_0) \\ 0 & 0 & 1 & (y-y_0) & -(x-x_0) & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$$

Step Output

This post-processing module provides in text file-type of output all the analytical results (nodal displacements/rotations, natural frequency and natural period) obtained by SeismoBuild. The entire step output, or selected parts of it, can be copied to text editors for further manipulation, using the corresponding menu commands, keyboard shortcuts, toolbar buttons or right-click popup menu.



Step Output

Rather than copying and pasting the contents of this module, users may also choose to simply use the Export to Text File facility, which gives also the possibility of choosing the start and end mode numbers of interest, together with a mode number increment. This useful facility is available from the toolbar button .

Finally, and as noted in *Advanced Settings >General*, users may also activate the option of creating, at the end of every analysis, a text file (*.out) containing the output of the entire analysis (as given in this module). This feature may result useful for users, who wish to systematically, rather than occasionally, post-process the results using their own custom-made post-processing facility.

Analysis Logs

As discussed above, during any given analysis, a log of its numerical progress and of the performance response of the model is created and saved within the project's log file (*_Eig.log). The contents of such file can be visualised in the **Analysis Logs** module and, if required, copied and pasted into any other Windows application.

It is also noted that, since the date and time of the last analysis are saved within the log file, users can refer to this module when such type of information is required.

Linear and Nonlinear Analyses

General

In SeismoBuild all the analytical methods (both linear and nonlinear)that are proposed by the different Standards have been programmed, namely (i) the Linear Static Procedure LSP, (ii) the Linear Dynamic Procedure LDP,(ii) the Nonlinear Static Procedure NSP and (iv) the Nonlinear Dynamic Procedure NDP.

In General, the nonlinear methods are considered numerically more advanced and more accurate in the representation of the earthquake loading. They take explicitly into account the concentration of damage at the weakest locations of the building and the redistribution of forces upon the formation of plastic hinges, considering both material inelasticity and geometric nonlinearities. Furthermore, the nonlinear dynamic method(although more complicated in its application) is considered to be the most accurate method of analysis, since it manages to better represent the dynamic nature of seismic loading with respect to its static counterparts. Consequently, the nonlinear methods they are the ones that are mainly employed for the assessment and strengthening of existing reinforced concrete buildings.

Linear Static Procedure

With the *Linear Static Procedure* (Lateral Force Method with the EC8 naming conventions) a triangular, lateral, pseudo-seismic force distribution that is assumed to approximate the earthquake loading is applied to a linear elastic structural model, in order to calculate the internal forces and the system displacements. These action effects are then compared against the members' capacities for the selected performance level, always in terms of forces, and, if the capacities are larger than the demands, the structure is considered safe.

The fundamental period of vibration of the building for lateral motion in the direction considered is calculated by eigenvalue analysis or with more approximate empirical methods, from which the ordinate of the response spectrum S_a is calculated. The total lateral force is proportional to the spectral acceleration S_a and the building weight W:

$$V = C_1 \times C_2 \times C_m \times S_a \times W \rightarrow in\ ASCE\ 41\ or$$

$$V = \lambda \times S_a \times W \rightarrow in\ Eurocode\ 8$$

C_1, C_2, C_m and $\lambda$are different easily calculated modification factors that are related to higher mode effects, and parameters, such as the expected maximum inelastic displacements, the effect of pinched hysteresis shapes, the stiffness and strength deterioration. This total force is then distributed at each floor level, according to the mass distribution of the building, and the modal shape of the fundamental mode (in EC8) or an inverted triangular distribution (in both ASCE-41 and EC8).

Because of its approximate nature, the linear static procedure is permitted only in cases of very regular, low-rise constructions that sustain limited damage and do not undergo large inelastic deformations. In particular:

(i) The demand to capacity ratios DCR should be small for all structural members. For the brittle failure types, they should be below unity.

(ii) There should be no in-plane strength or stiffness discontinuities & irregularities.

(iii) There should be no out-of-plane strength or stiffness discontinuity & irregularities.

(iv) There should be no weak storey strength or stiffness irregularities.

(v) There should be no torsional strength or stiffness irregularities.

(vi) The fundamental period should not be large.

LINEAR DYNAMIC PROCEDURE

The *Linear Dynamic Procedure* (Modal Response Spectrum Analysis, according to the EC8 naming conventions) is similar to the LSP, at least as regards the modelling approach. The model is again elastic and there is no stiffness degradation during the analysis. However, the method is somehow more sophisticated, since the profile of the lateral forces is not arbitrary anymore, but rather it is calculated as a combination of the modal contributions of the different modes of vibration of the structure. The action effects of the structural members are again compared against the capacities for the selected performance level in terms of forces, and, if the capacities are larger than the demands, the structure is considered safe. The Linear Dynamic Procedure is based on the well-known response-spectrum analysis (RSA) [e.g. Rosenblueth, 1951; Chopra, 1995] and it is the method of analysis that is typically employed for the design of new structures.

The response-spectrum analysis is a pseudo-dynamic method, which is capable of providing the peak values of response quantities, such as forces and deformations, of a structure under seismic excitation with a series of static analyses, rather than time-history dynamic analysis. In this context, the time–acceleration history imposed to the supports of the structure is replaced by the equivalent static forces, which are distributed to the free DOFs of the structure and represent the contribution from each natural mode of vibration. These equivalent forces are derived for each mode of vibration separately as the product of two quantities: (i) the modal inertia force distribution (thus eigenvalue analysis is needed), and (ii) the pseudo-acceleration response per mode (obtained from the 5% damped response spectrum). For each mode of interest, a static analysis is conducted, and then every final peak response quantity is derived by the superposition of the quantities corresponding to the modes.

A sufficient number of modes has to be considered, so that to capture at least 90% of the participating mass of the building in each of two orthogonal principal horizontal directions of the building, thus neglecting only the less significant ways of vibrating in terms of participant mass. EC8 also requires that all modes with more than 5% of the participating mass in any direction should be considered.

Because the peaks in the responses of each mode generally occur at different time instants and rigorous time-history analysis has not been conducted, it is not possible to determine the exact peak values of the response quantities. Therefore, approximations need to be introduced by implementing one of the modal combination (statistical) rules, such as the absolute sum (ABSSUM), square-root-of-sum-of-squares (SRSS) and the complete quadratic combination (CQC). CQC is suggested when periods are closely spaced, with cross-correlation between the modal shapes. SRSS can be used when the periods differ by more than 10%, whilst ABSSUM offers a very safe, upper limit of response.

The same procedure is repeated for each desired seismic direction EX, EY and EZ by using different or the same response spectra. It is usually requested that two or three seismic loading directions (EX, EY, EZ) are to be considered simultaneously, together with the gravity static loads (G+Q) of the structure (the vertical component EZ is mandatory only for the elements, where the vertical vibration is considered critical, e.g. large cantilevers). The seismic loading directions may be combined linearly ($E = \pm EX \pm EY \pm EZ$) with different factors f_{EX}, f_{EY}, f_{EZ} per direction (usually $f_{EX}=f_{EY}=f_{EZ}=1.00$ or 0.30) or by the SRSS rule ($E = \pm\sqrt{EX^2+EY^2+EZ^2}$). The gravity and live loads are defined and added algebraically. Because the seismic loads are taken into account with both signs for every direction, the results of RSA loading combinations in terms of any response quantity are presented as envelopes.

Contrary to the Linear Static Procedure, the Linear Dynamic Procedure is suitable for buildings with larger fundamental period, where higher-mode effects are important. Apart from this, all the recommendations and limitations described for the LSP apply for the LDP as well.

(i) The demand to capacity ratios DCR should be small for all structural members. For the brittle failure types, they should be below unity.

(ii) There should be no in-plane strength or stiffness discontinuities & irregularities.

(iii) There should be no out-of-plane strength or stiffness discontinuity & irregularities.

(iv) There should be no weak storey strength or stiffness irregularities.

(v) There should be no torsional strength or stiffness irregularities.

NONLINEAR STATIC PROCEDURE

Conventional (non-adaptive) pushover analysis is employed in the estimation of the horizontal capacity of structures implying a dynamic response that is not significantly affected by the levels of deformation incurred (i.e. the shape of the horizontal load pattern, which aims at simulating dynamic response, can be assumed as constant).

The introduced vertical loads applied to the 3D model, in addition to the incremental loads, are equal to C_gG+C_qQ, where C_g and C_q are the permanent and live loads coefficients, respectively, defined in the Static Actions tab of the Code Requirements module. It is noted that the snow load is also introduced when it is required, i.e. $C_gG+C_qQ+C_sS$ for ASCE 41-23 and TBDY. The self weight of the beam and column elements is automatically computed according to the materials' specific weight and sections' geometry. The slabs' additional gravity and live loads are automatically introduced as beams' additional mass. Nonlinear static analysis may be applied with two vertical distributions of loads:

(i) a "uniform pattern", which attempts to simulate an inelastic response dominated by a soft-storey mechanism (development of plastic hinges at both top and bottom ends of all columns of a storey, in general the ground floor, which is subjected to highest lateral forces);

(ii) a "modal pattern", proportional to the fundamental elastic translational mode shape.

The incremental loads may be applied in both positive and negative directions. Furthermore, the incremental loads applied in X and in Y direction, may be taken as acting simultaneously by employing both of the following combinations:

I. $\pm F_x \pm 0.30F_y$

II. $\pm 0.30F_x \pm F_y$

With F_x and F_y representing the incremental loads applied in the X and Y direction of the structure, respectively.

Finally, in order to account for uncertainties in the location of masses and in the spatial variation of the seismic motion, the calculated centre of mass at each floor may be considered as being displaced from its nominal location in each direction by an accidental eccentricity equal to 5% of the floor-dimension perpendicular to the direction of the seismic action.

The applied incremental load P is kept proportional to the pattern of nominal loads (P°) defined by default by the program according to Code requirements: $P = \lambda(P°)$. The load factor λ is automatically increased by the program until a Code-defined limit, or numerical failure, is reached. For the incrementation of the loading factor, a displacement control strategy is employed, which refers to direct incrementation of the global displacement of the control node and the calculation of the loading factor that corresponds to this displacement.

NONLINEAR DYNAMIC PROCEDURE

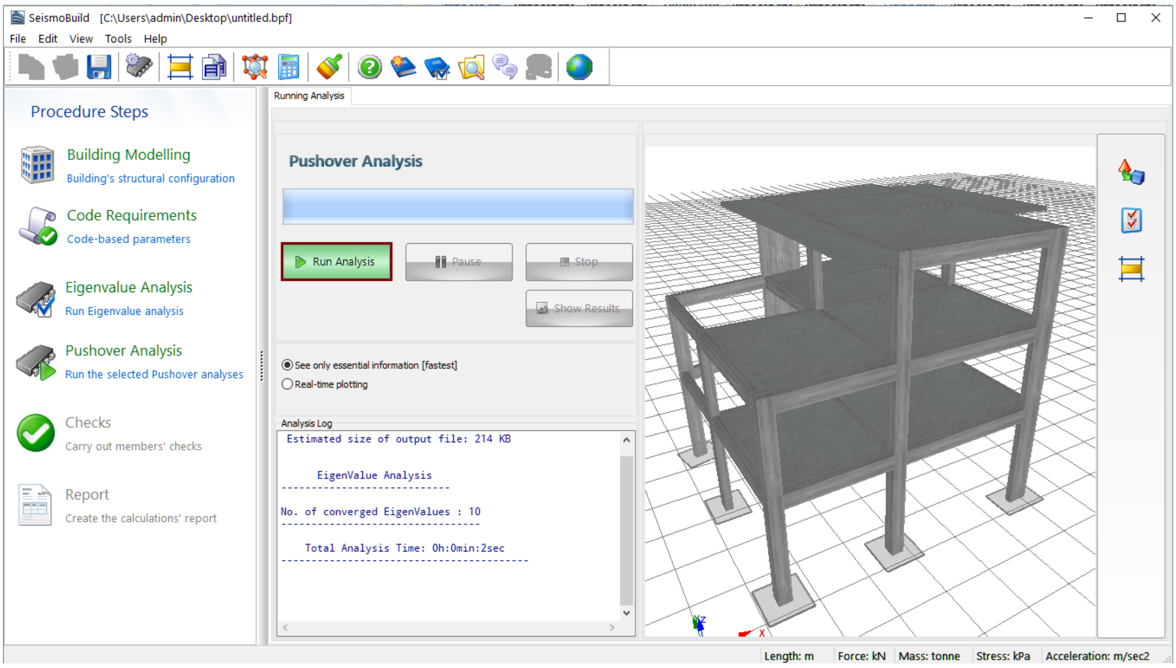
The Nonlinear Dynamic Procedure constitutes a sophisticated approach for examining the inelastic demands produced on a structure by a specific suite of ground motion acceleration time-histories. Being the numerically more advanced method of analysis, it is the most accurate in the representation of the dynamic nature of seismic loading. As nonlinear dynamic analysis involves fewer assumptions than the nonlinear static procedure, it is subject to fewer limitations than nonlinear static procedure. It automatically accounts for higher-mode effects and shifts in inertial load patterns as structural softening occurs. In addition, it provides reliable results even for highly irregular structures, or with irregular seismic action (e.g. near-fault ground motion or loading in 2 or 3 directions simultaneously). As a result, the NDP is the only method that can be used for any structural configuration and any type of loading. In practice, we can analyse with adequate accuracy any structural configuration subjected to any type of seismic action.

Similarly to pushover analysis, the introduced vertical loads applied to the 3D model are equal to C_gG+C_qQ (or $C_gG+C_qQ+C_sS$ for ASCE 41-23 and TBDY). The coefficients C_g, C_q and C_s are the permanent, live and snow loads coefficients defined in the Static Actions tab. The self weight of the beam and column elements is automatically computed according to the materials' specific weight and sections' geometry. The slabs' additional gravity and live loads are automatically introduced as beams' additional mass.

Nonlinear dynamic analysis is performed by applying at the foundation of the building sets of acceleration time-histories. In SeismoBuild the ground motions consist of pairs of orthogonal horizontal ground motion components. Both components are artificial records compatible (for the selected seismic hazard level) with the given target spectrum. In EC8, NTC-18 and KANEPE, when 7 or more record pairs are specified, the average response should be considered; instead, when fewer records are considered, the most unfavourable value of the response quantity among the analyses should be used in the verification checks. Similarly, according to ASCE 41 and TBDY a suite of not less than 11 ground motions shall be selected for each target spectrum and the mean response is checked.

PROCESSOR

After introducing the model's configuration and carrying out the eigenvalue analysis, the user is then ready to run the analysis. This is carried out in the **Linear Analysis Response Spectrum, Pushover Analysis** or **Dynamic Analysis** area of SeismoBuild by selecting the Run Analysis button.

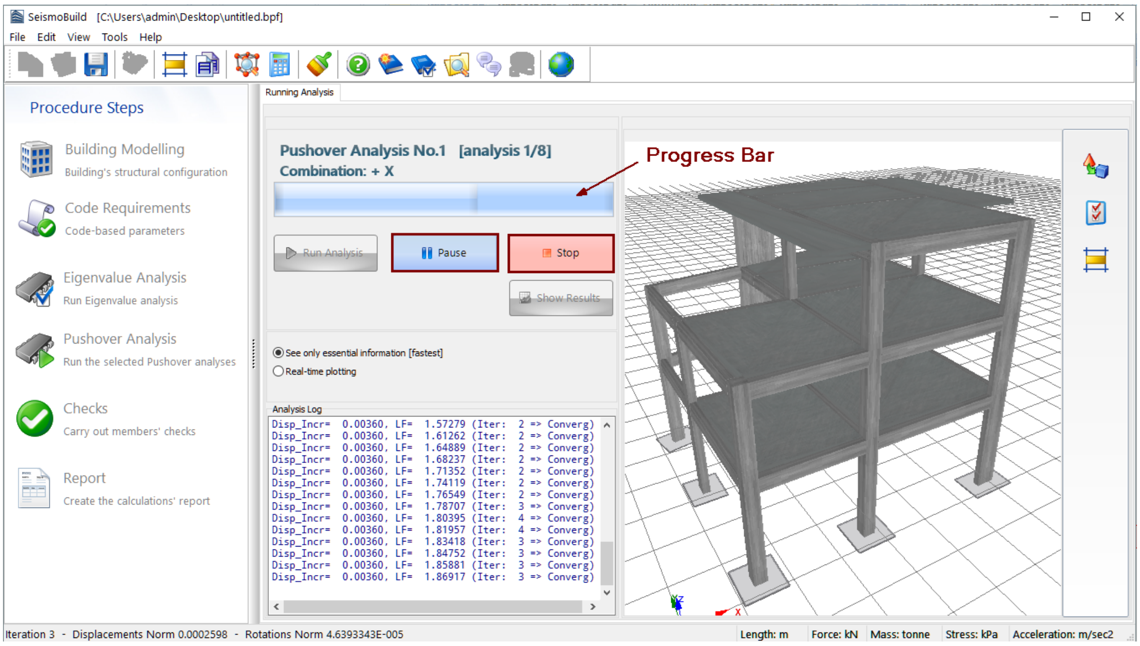


Processor area

Depending on the size of the structure, the selected frame elements type, the applied loads and the processing capacity of the computer being used, the analysis may last some seconds to several minutes.

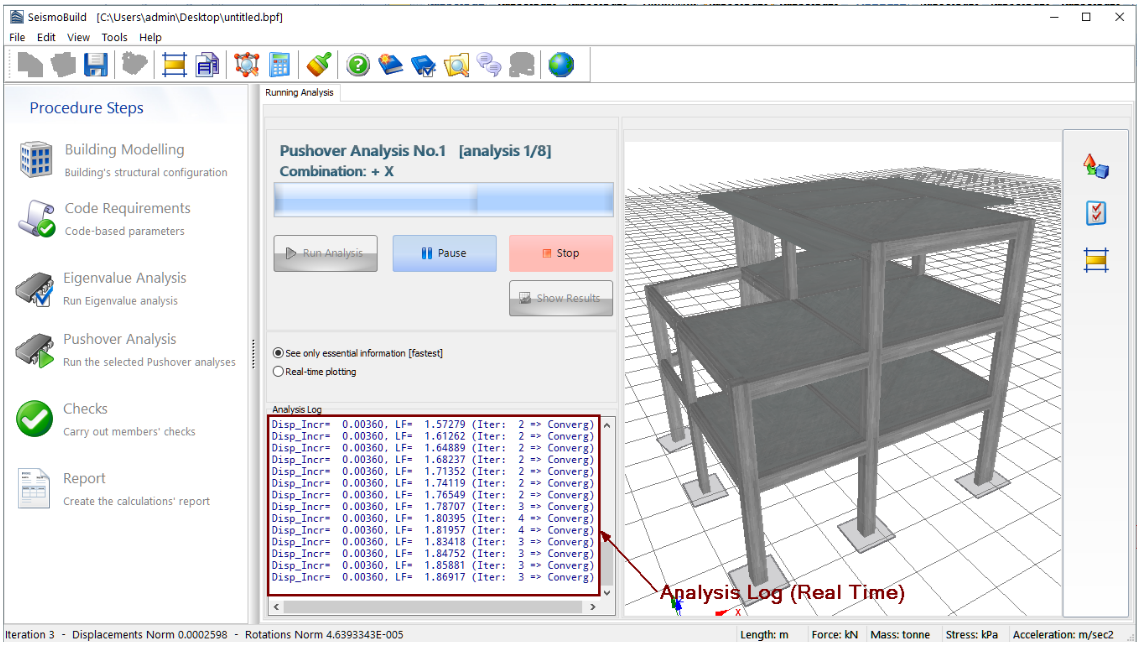
As the analysis is running, a progress bar provides the user with a percentage indication of how far has the former advanced to, and a message placed above the progress bar informs about the number of the analysis that is running. Users can in this manner quickly assess the waiting time required for the analysis to be completed, and hence quickly plan their subsequent work schedule.

The analysis can also be paused, enabling users to (i) momentarily free computing resources so as to carry out an urgent priority task or (ii) check the results obtained up to that point, which may be useful to decide the worthiness of progressing with the other analyses. If the user presses the Run Analysis button again, the analysis can be continued.



Progress bar and "Pause"/"Stop" buttons

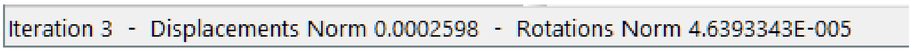
The Analysis Log is also shown to the user, in real-time, providing expedient information on the progress of the analysis, loading control and convergence conditions (for each global load increment).



Real-time Analysis Log area

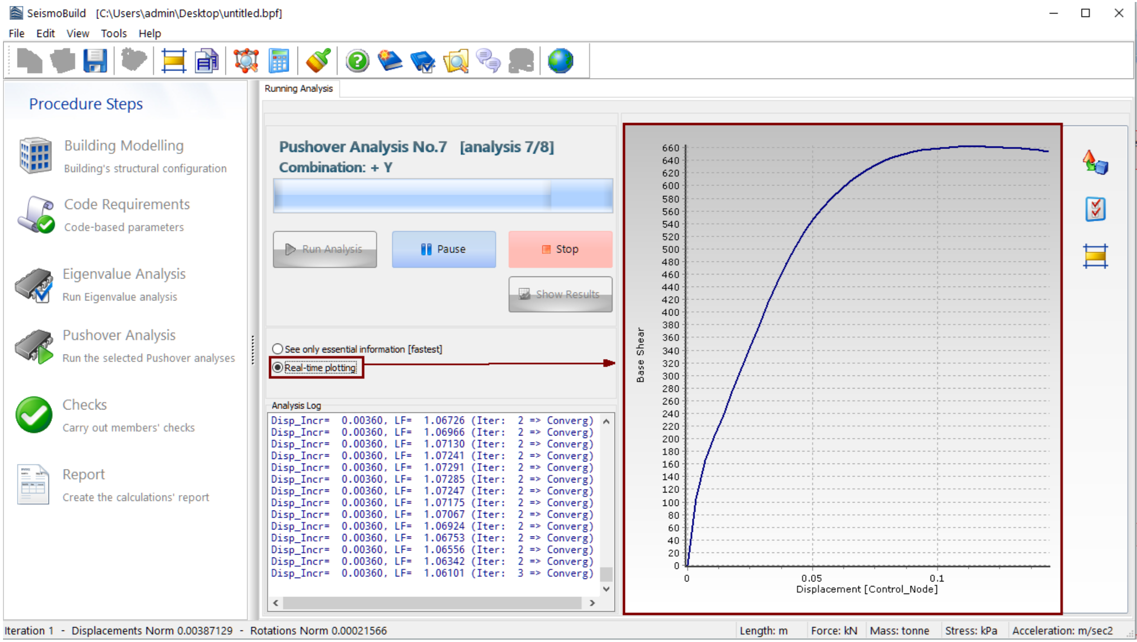
This log is saved on a text file (*_Push(number of analysis).log) that features the same name as the project file, as well as the name and number of the analysis, and indicates the date and time of when the analysis was run (the sort of non-technical information that comes very handy on occasions). In addition, the corresponding real-time log is shown during the analysis and saved to the same *_Push(number of analysis).log file.

At the bottom of the window, the convergence norms at the end of a given (global) load increment are shown.



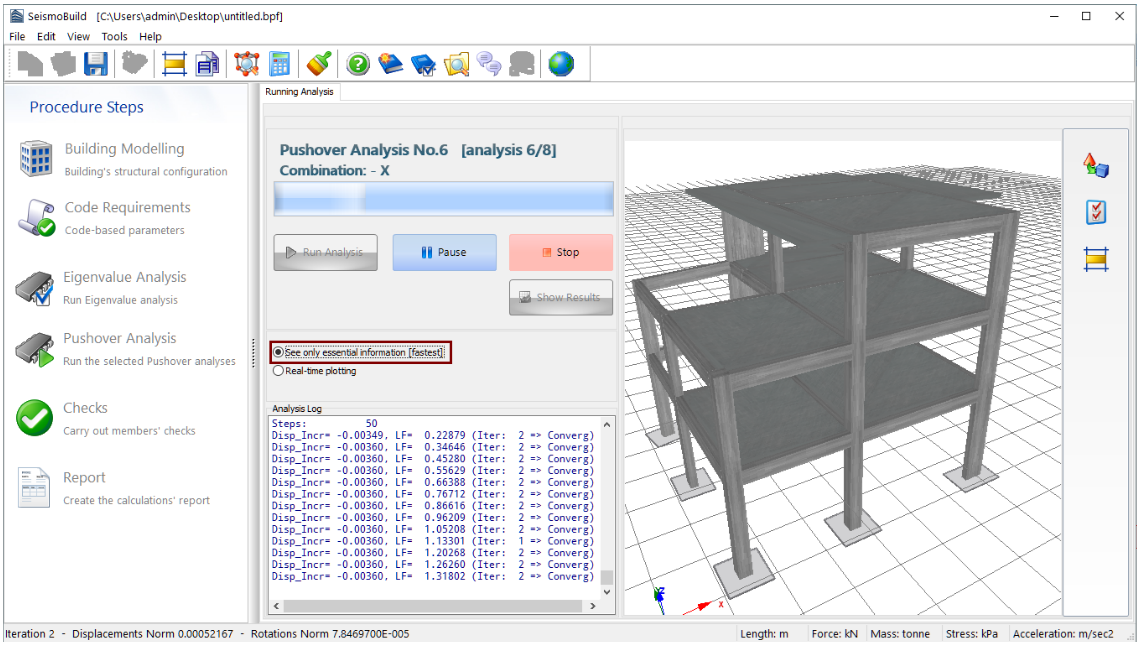
Convergence norms

Further, the user has also the option of graphically observing the real-time plotting of a capacity (static pushover) curve of the control node and respective degree-of-freedom or the top displacement vs. time diagram (dynamic analysis).



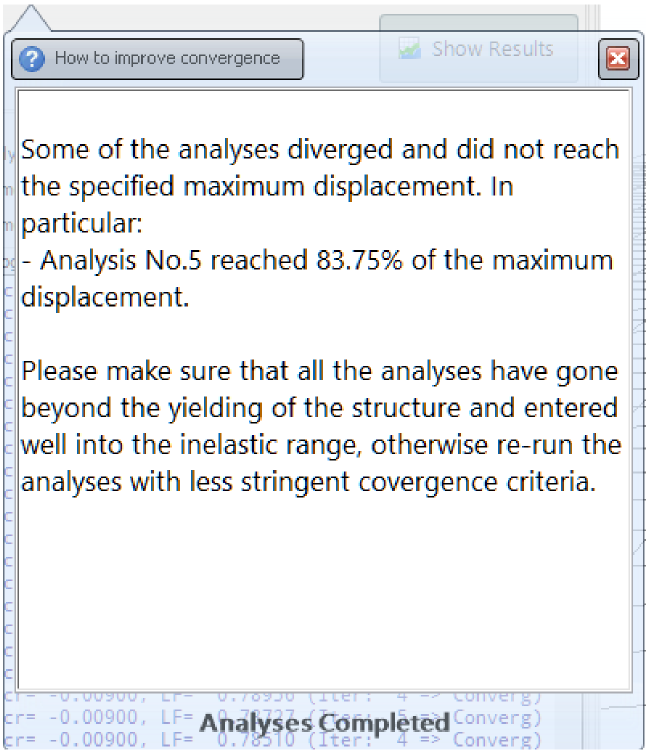
Real-time plotting option

This option, however, might slow down the analysis and increase its running time when used in relatively slow computers, for which reason the user has also the possibility of simply disabling any real-time plotting, choosing to follow only the analysis logs.



See only essential information option

A message appears, when the analyses are completed, that provided information about the execution of the analyses. In particular, the message informs if the whole process was successfully completed, (i.e. all the analyses have reached the specified maximum displacement or all the steps of the defined time histories in dynamic analysis have been completed), or whether the SeismoBuild solver could not execute all the analyses until the end, because of convergence difficulties. In the last case the message informs about the percentage of the maximum displacement reached when the analysis terminated.



Informative message

NOTE 1: Whenever the real-time deformed shape of the structure is difficult to interpret (because displacements are either too large or too small), users can right-click on the plotting window and adjust its respective Deformed Shape Multipliers. The 3D Plot options are also available for further fine-tuning (e.g. on some cases, it may prove handy to fix the graph axis, rather than having them automatically updated by the program). Please refer to the Deformed shape viewer section for further hints and info on real-time visualisation of a model's deformed shape.

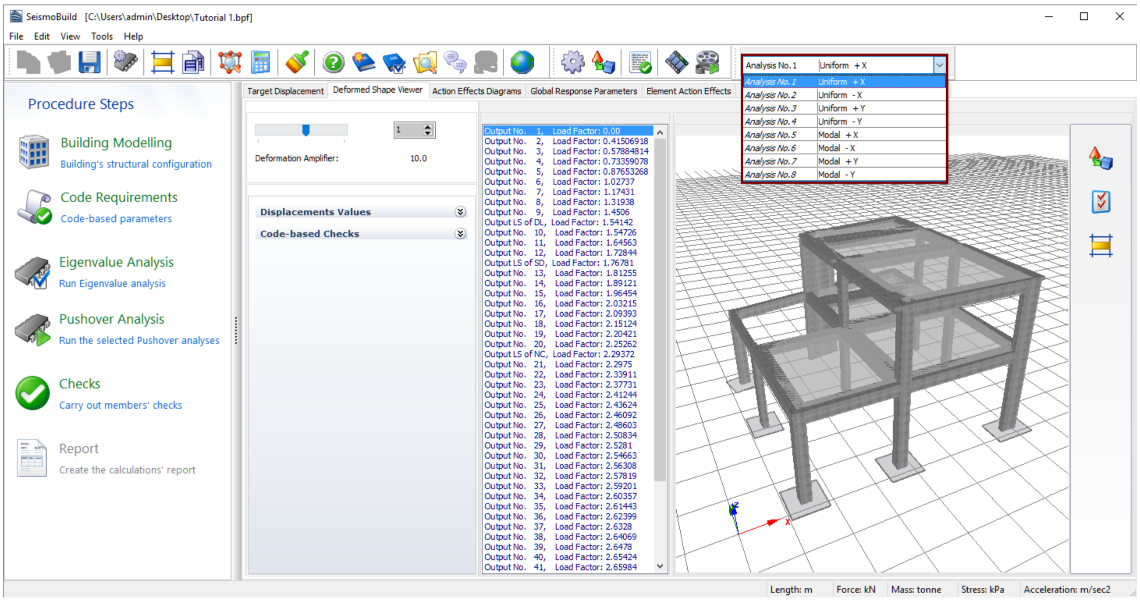
NOTE 2: The current version of SeismoBuild is not capable of taking advantage of multi-processor computing hardware; hence, speed of a single analysis may be increased only by increasing the CPU speed (together with the speeds of the CPU Cache, the Front Side Bus, the RAM modules, the Video RAM, the Hard-Disk (rotation and access)). Having more than one CPU, however, will reduce running times of multiple contemporary analyses, since in such cases "parallel processing" can take place

NOTE 3: There is a RAM limitation in SeismoBuild (4GB in 64-bit Windows systems and 3GB in 32-bit Windows systems).

POST-PROCESSOR

After running the analysis/es users may view the results by clicking the Show Results button. The results of the analysis are saved in a SeismoBuild Results File, and are identified by the *.brf extension, with the same name as the input project file and the analysis.

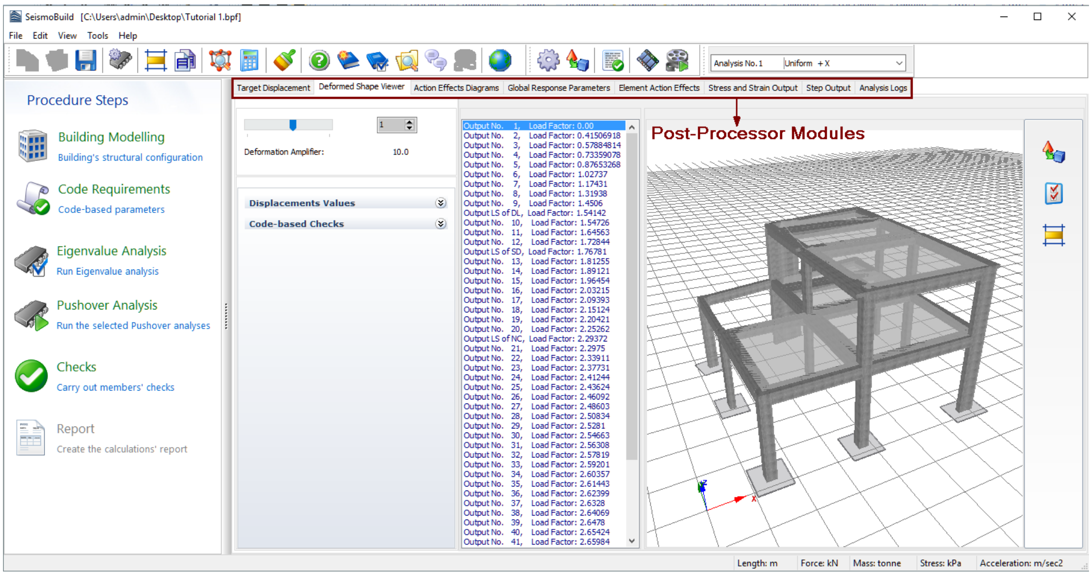
The Show Results area features a series of modules where results from all the analyses can be viewed in table or graphical format, and then copied into any other Windows application (e.g. tabled results can be copied into a spreadsheet like Microsoft Excel, whilst results plots can be copied into a word-processing application, like Microsoft Word). Users may choose which analysis results to view through a drop-down menu available on the top right side of the program.



Selection of the analysis to view

The modules available in the Post-Processor are listed below and will be described in the following paragraphs:

- Target Displacement (available in Pushover analyses results)
- Deformed Shape Viewer
- Convergence Details
- Action Effects Diagrams
- Global Response Parameters
- Element Action Effects
- Step Output
- Analysis Logs



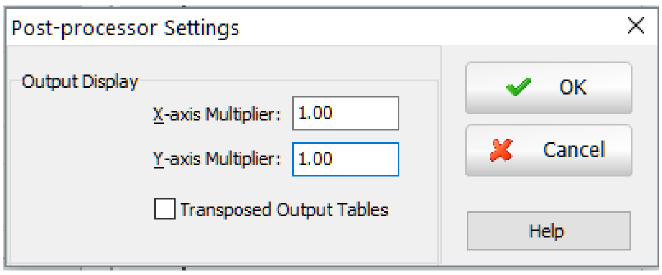
Post-Processor Modules

There are some general operations that apply to all the post -processor modules. For example, the way in which model components (e.g. nodes, sections, elements, etc.) appear on all dialogue boxes in the post-processor.

Post-Processor settings

Often, the possibility of applying a multiplying factor or coefficient to the results comes as very handy. For instance, if the analysis has been carried out using kNm as the units for moment quantities, which are the default units, users might wish to multiply the corresponding results by 1e+06, so as to obtain moments expressed in Nmm instead. Alternatively, and as another example, users might also wish to multiply the curvature values of an element with a factor of -1, so that compression stresses and strains comes plotted in the x-y positive quadrant, as usually presented. Therefore, users are given the possibility to apply multipliers to all quantities being post-processed.

This facility can be accessed through the program menu (*Tools > Post-Processor Settings*), or through the right-click pop-up menu, or through the corresponding toolbar button.



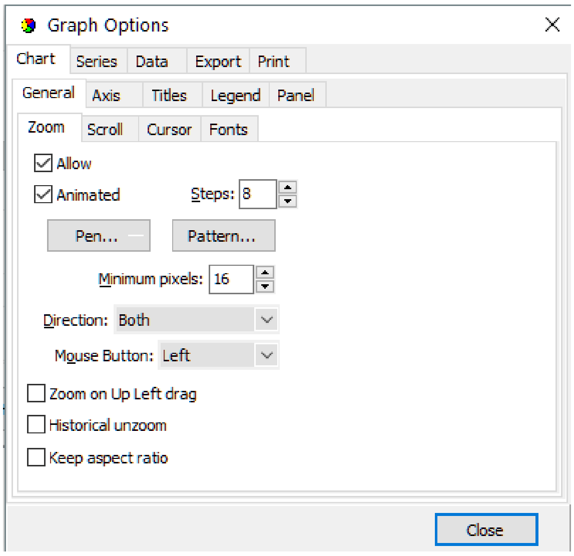
Post-Processor Settings

In addition, the Post-Processor Settings provide users also with the possibility of transposing the Output Tables. This might come very handy in cases where, for instance, a model features several thousands of nodes/elements, which in turn leads to default output tables with an equally very large number of columns, that one may not be able to then copy to spreadsheet applications (e.g. Microsoft Excel) that feature a relatively stringent limit on the number of columns (max = 16384). By transposing the tables, the nodes/elements are then listed in rows, thus overcoming the limitation described above (in general, the aforementioned spreadsheet applications cater for tables with might have up to 1.048.576 rows).

NOTE: The Post-Processor apply to all its modules.. Hence, users should have in mind that if, for instance, they apply a -1 coefficient to the values of total base shear of the structure (plotted as a y-quantity in the hysteretic plots module) then the values of material stresses (plotted as y-quantity in the stress and strain module) will also be modified by this -1 multiplier.

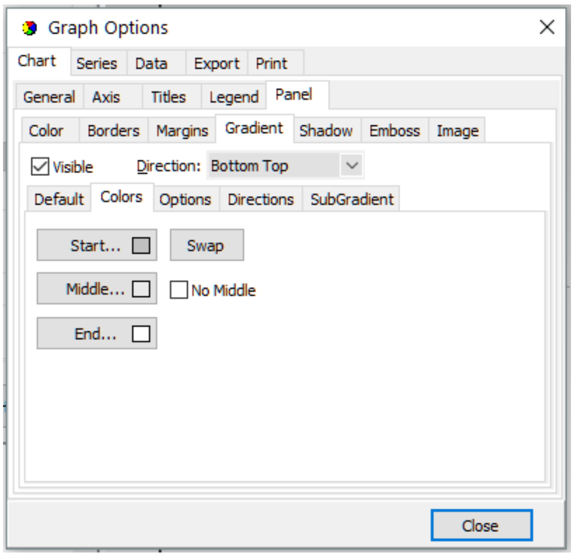
Plot Options

All graphs displayed in the post-processor modules can be tweaked and customised using the Plot Options facility, available from the main menu (*Tools > Plot Options...*), toolbar button or right-click popup menu. The user can then change the characteristics of the lines (colour, thickness, style, etc.), the background (colour, gradient), the axes (colour, font size and style of labels etc.) and the titles of the plot. Through the *Save Plot Settings...* and the *Load Plot Settings...*, available on the right click popup menu, the plot settings may be saved and retrieved, respectively, to be applied to other plots.



Plot Options – General

NOTE: Before copying results plots into other Windows applications, users might wish to remove the plot's background gradient, which looks good on screen but comes out quite badly on printed documents. This can be done easily in the Panel tab of the Plot Options dialog box.



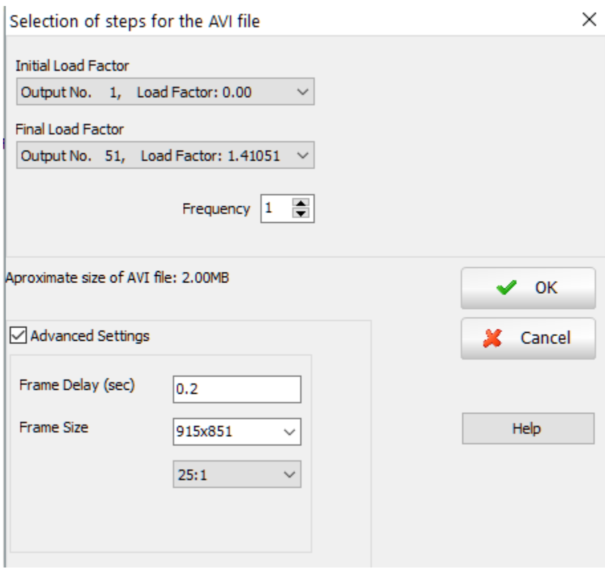
Plot Options – Panel

In addition, zooming-in and -out can be done by dragging the mouse on the graph area (a top-left to bottom-right selection zooms in, whereas a bottom-right to top-left selection zooms out).

Creating an analysis movie

SeismoBuild provides users with the possibility of creating animations illustrating the way a particular structure, subjected to a given set of loads, deforms in pseudo-time (static analysis). This facility can be accessed through the program main menu (*Tools > Create AVI file...*) or through the respective toolbar button .

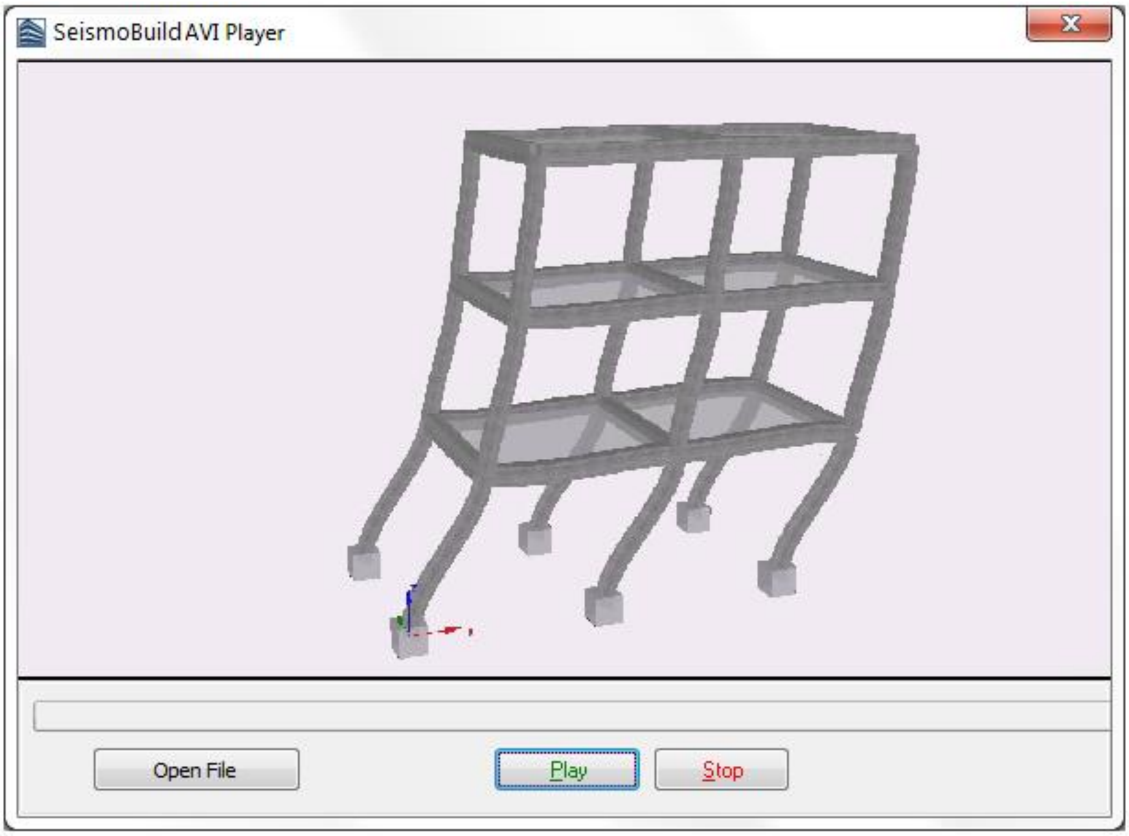
For the case of static analysis animations, users need only to define the name of the movie file to be created (*.avi), the start and end deformed shapes, and the frequency in shape image selection. Evidently, the lower the frequency, the highest number of images will be used in the creation of the movie, and hence the higher the quality (smoothness of the moving sequence), but also the highest the size of the resulting file. The smallest possible frequency value is 1, effectively meaning that all deformed shapes that have been output will be used in the creation of the movie.



Selection of steps for the AVI file

Before creating the animation, users are advised to customise the 3D Plot to their needs and likings, since these settings will reflect the look and feel of the movie. In particular, it is noted that during movie creation, the axes of the plot are not automatically updated, thus implying that, before initiating the creation process, users should set the axes to their largest needed values. The latter can be done either by viewing an output shape where deformations are at their highest, or by manually tweaking the axes characteristics (using the 3D Plot options).

Once the animation has been created, users can verify its adequacy through the AVI Viewer incorporated in SeismoBuild, accessible from the program main menu (*Tools > Show AVI file...*) or through the respective toolbar button .

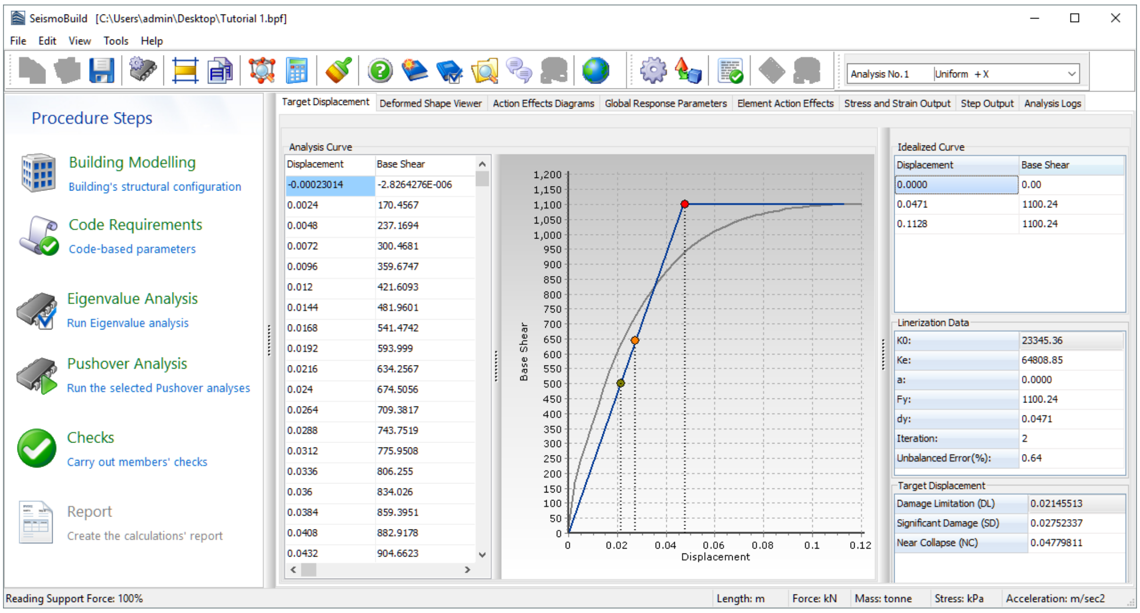


SeismoBuild AVI player

Animations created in SeismoBuild (i.e. AVI movies) can also be opened by other Windows applications such as Windows Media Player or, perhaps more importantly, Microsoft PowerPoint, where they can be used in multimedia presentations.

Target Displacement

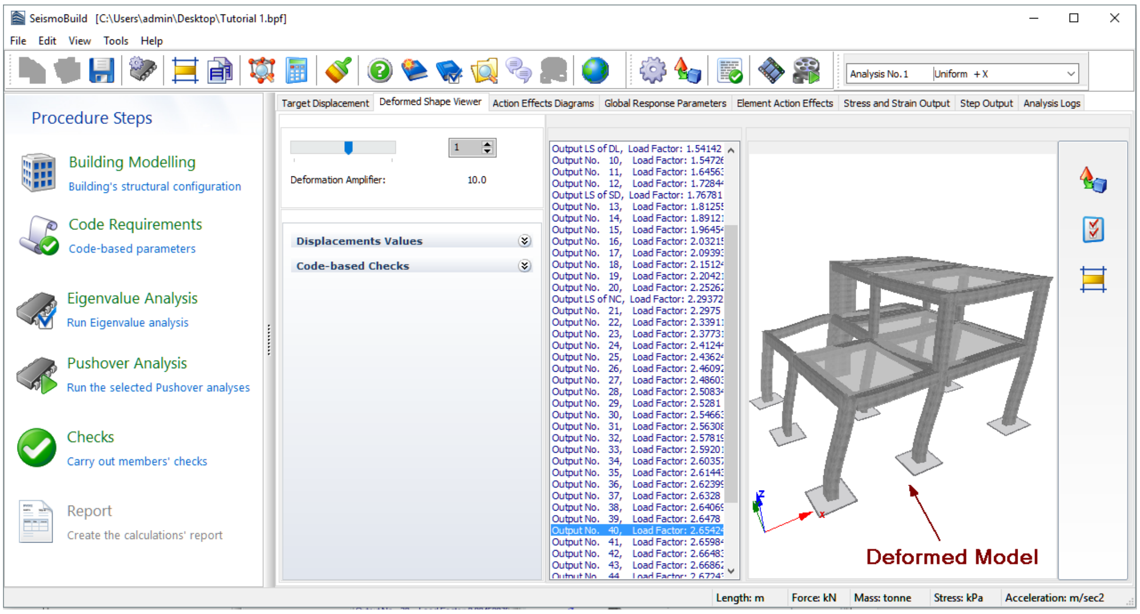
In the Target Displacement module, the capacity curves before and after the linearisation that correspond to the selected pushover analysis are shown, as well as the target displacements for the selected limit states in the Code Requirements window. Data about the linearisation procedure are also provided in this module. Users may refer to Appendix A.1 - EUROCODES, Appendix A.2 – ASCE, Appendix A.3 – NTC-18, Appendix A.4 – KANEPE or Appendix A.5 - TBDY for more information about the calculation of the target displacement.



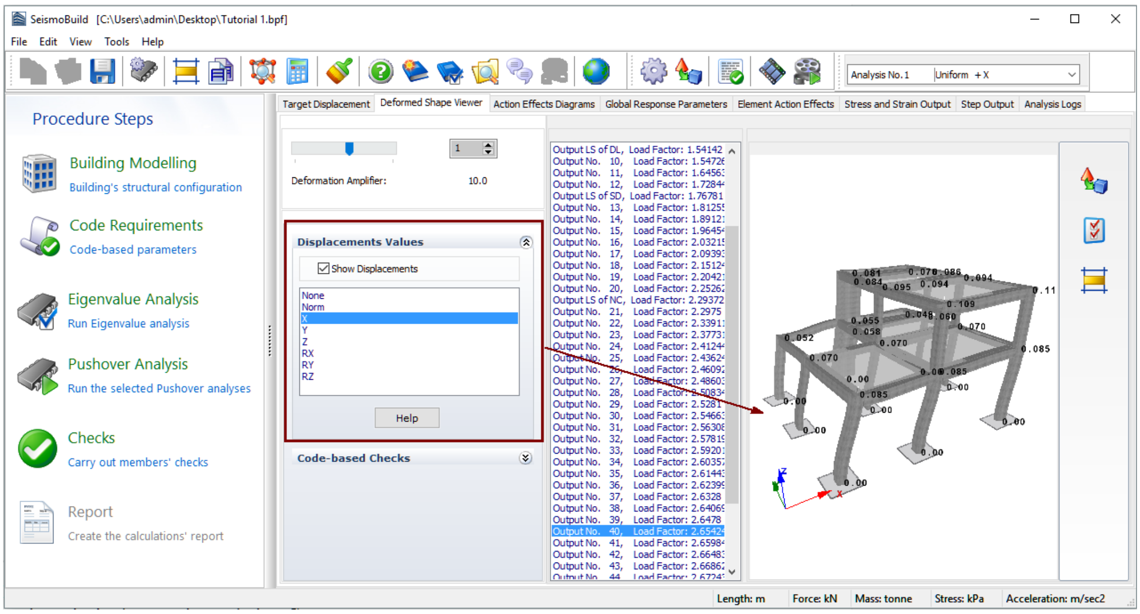
Target Displacement

Deformed Shape Viewer

With the Deformed Shape Viewer, users have the possibility of visualising the deformed shape of the model at every step of the analysis (click on the desired output identifier to update the deformed shape view), thus easily identifying deformation, and eventually collapse, mechanisms.

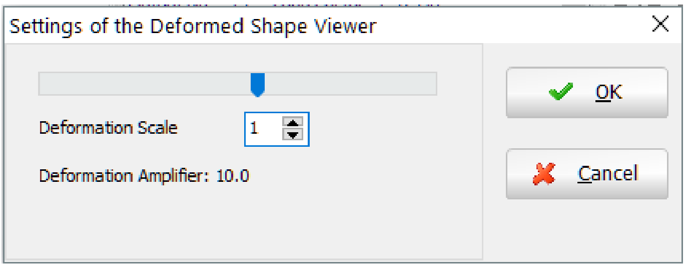


Deformed Shape Viewer



Deformed Shape Viewer – Displacement values display option

The deformed shape plot can be tweaked and customised using the 3D Plot Options and then copied to any Windows application by means of the Copy 3D Plot facility. In addition, and whenever the real-time deformed shape of the structure is difficult to interpret (because displacements are either too large or too small), users can make use of the Deformation Amplifier available on the left side of the Deformed Shape Viewer tab, through the program menu (*Tools > Deformed Shape Settings…*) or through the corresponding popup menu.

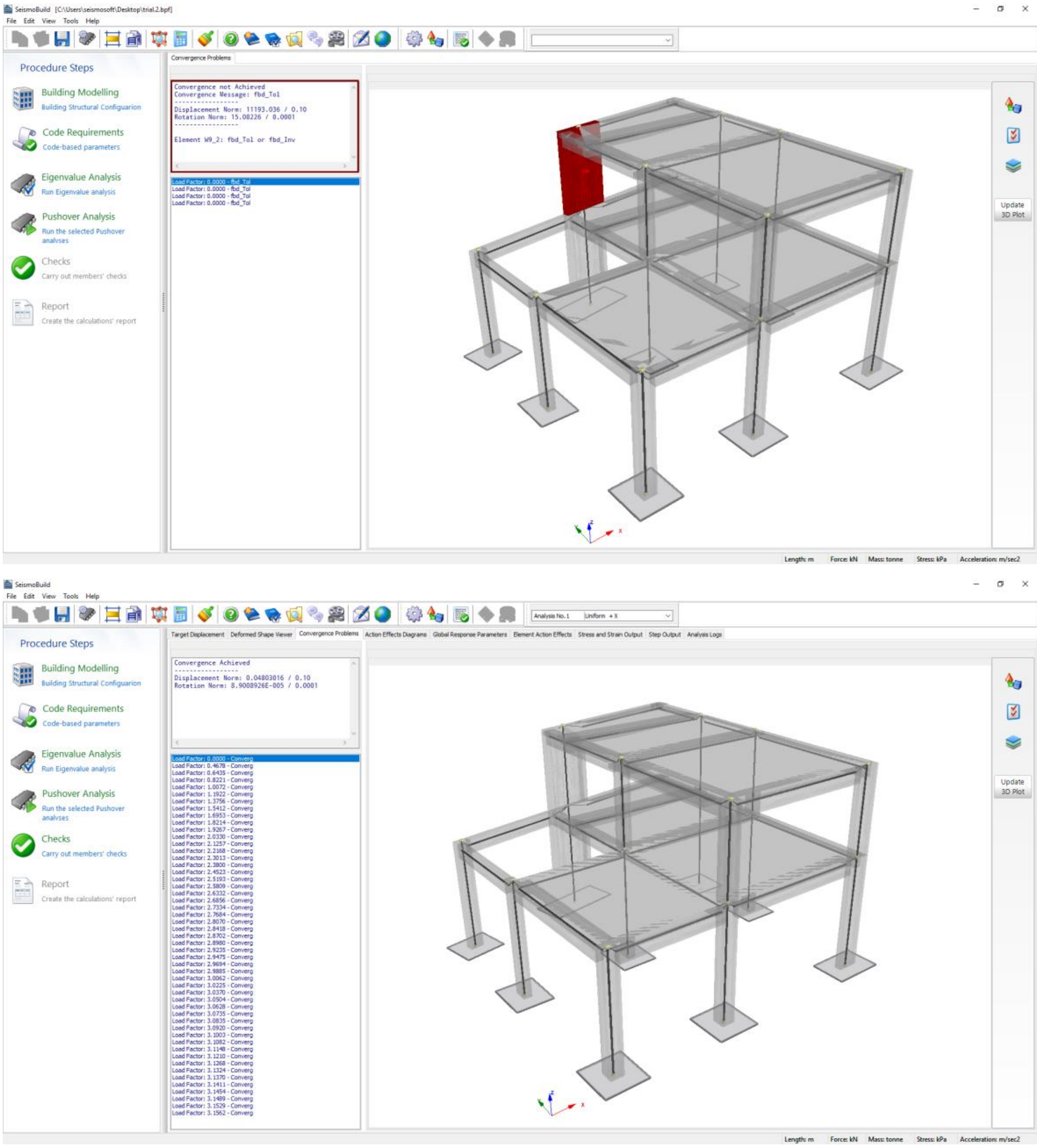


Deformed Shape Settings

The option to automatically update the 3D model after changing the multipliers is available. Users may deactivate this option and update it manually by clicking on the update 3D plot button on the right.

Convergence Details

Whenever convergence problems arise, users may be informed about the elements that cause the diverging solutions. The elements or the locations of the structure, where the convergence problems are caused, are marked in the 3D view format, whereas information about the type of divergence (value of convergence norms and their limits, divergence message and the corresponding elements or nodes) are displayed on the top-left corner of the screen.

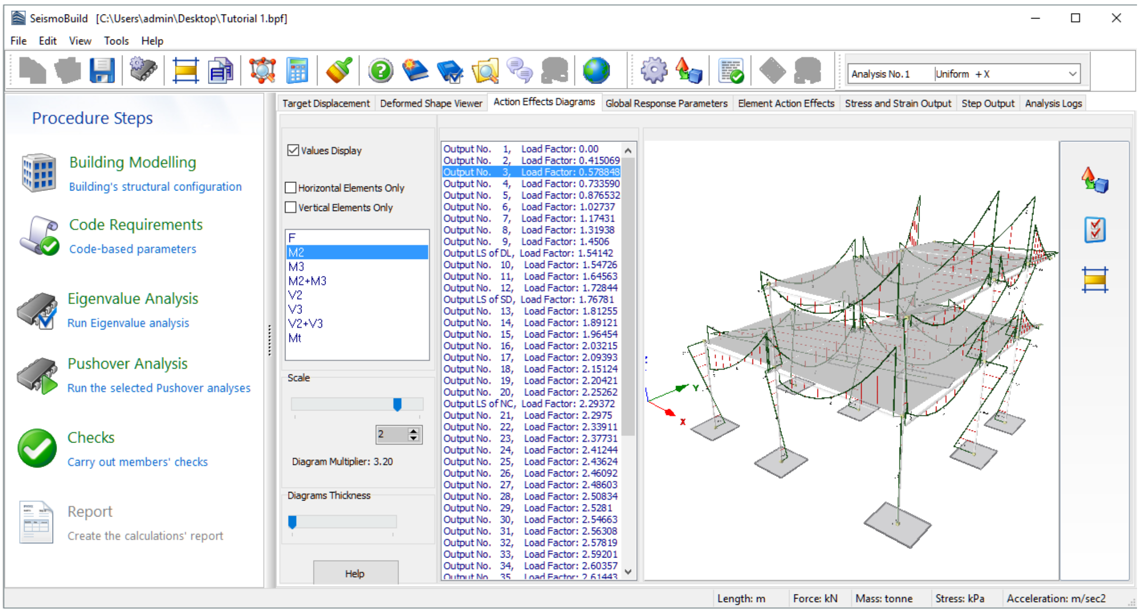


Convergence Problems

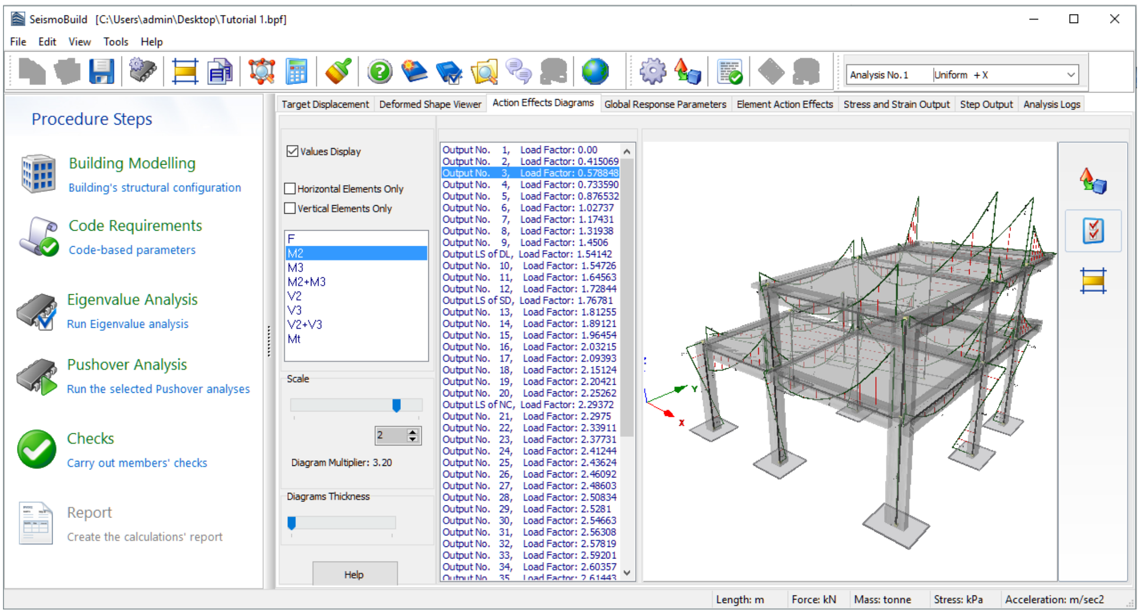
> **NOTE: Users should activate in Advanced Settings > Convergence Criteria the option of showing convergence difficulties in Post-Processor in order to be able to view the Convergence Problems tab in the Post-Processor.**

Action Effects Diagrams

The internal forces (axial and shear) and moments (flexure and torsion) diagrams are provided in the 3D plot view. By default, the diagrams for horizontal and vertical elements are shown in the same plot. If users wish to obtain the diagrams separately (for horizontal or vertical elements only), they have to check the appropriate box. The possibility of scaling the diagrams and the thickness of the lines is also available.

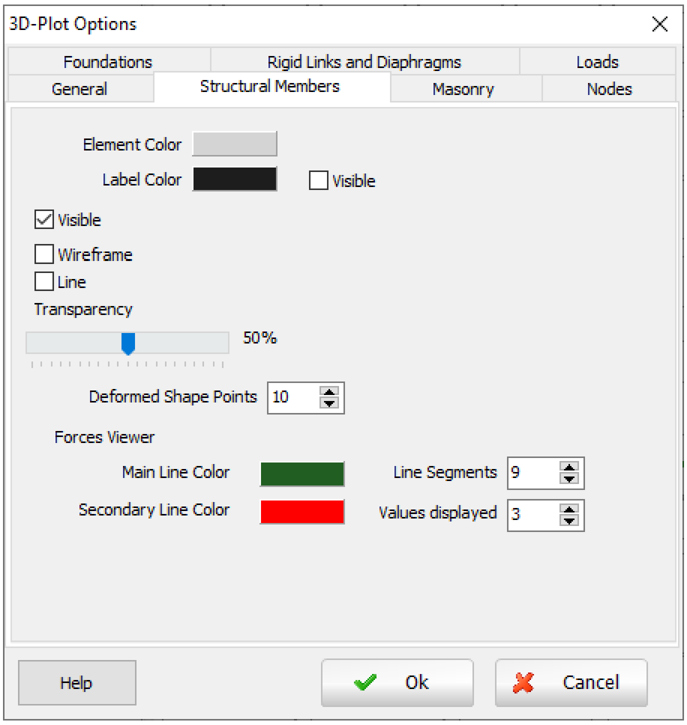


Action Effects Diagrams (Elements as Lines)



Action Effects Diagrams

Users may customize the diagrams aspect by changing the 'Structural Members' settings in the 3D Plot Options menu (i.e. main line and secondary line colours, number of sec. lines and number of values).



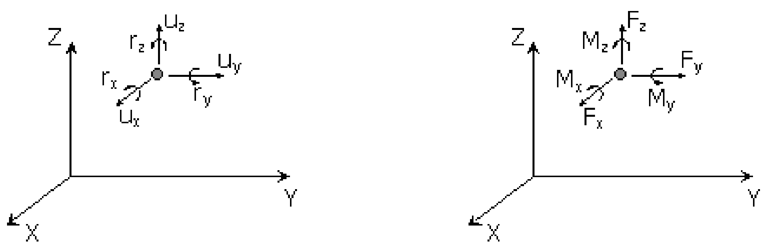
3D-Plot Options-Structural Members

Global Response Parameters

For pushover analysis four different kinds of global response parameters results can be output in this module:

- Structural Displacements
- Forces and Moments at Supports
- Hysteretic Curves
- Code-based Checks

Apart from the latter, all the other results are defined in the global system of coordinates, as illustrated in the figure below, where it is noted that rotation/moment variables defined with regards to a particular axis, refer always to the rotation/moment around, not along, that same axis.

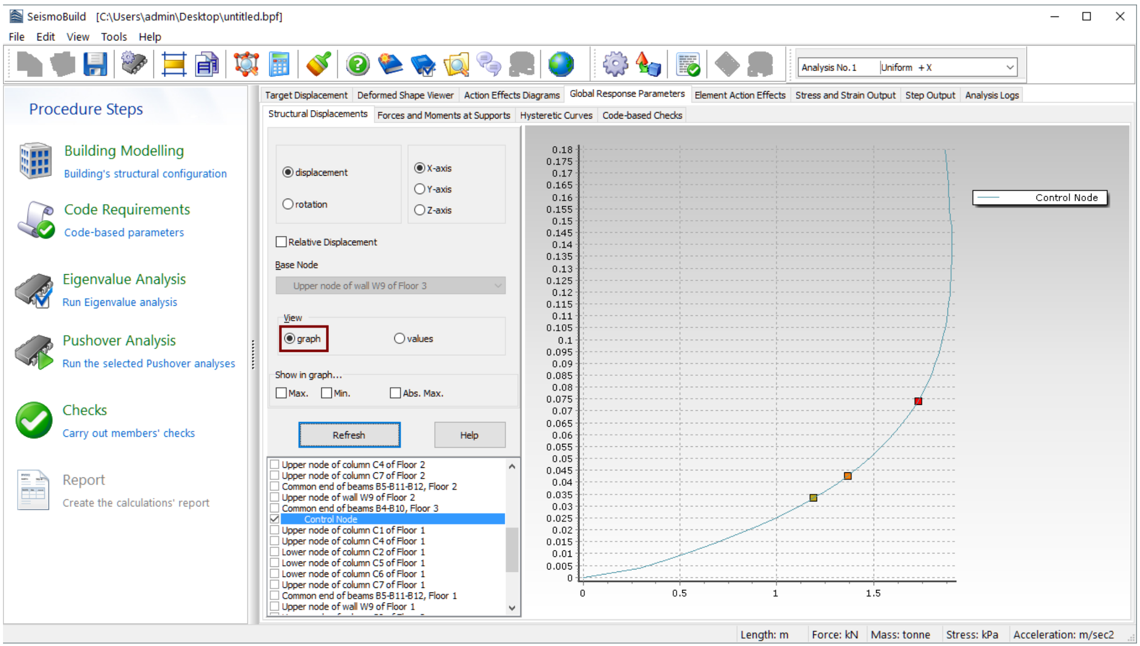


NOTE: The supports reactions should evidently be equal to the internal forces of the base elements that are connected to the foundation nodes. In other words, one would expect the values obtained in *Forces and Moments at Supports* to be identical to those given in the *Element Action Effects* for the elements connected to the foundations. However, some factors may actually lead to differences in these two response parameters, e.g. the member action effects are given in the local reference system of each element, whilst reactions at supports are provided in the global coordinates system. Hence, in those cases where large displacements/rotations are incurred by the structure, differences in element shears and support horizontal reactions may be observed.

All of these parameters are briefly described hereafter:

Structural displacements

The user can obtain the displacement results of any given number of nodes including the control node, for any of the six global degrees-of-freedom. The possibility for relative displacement output is also available, as well as the choice of displaying in the graph the maximum, minimum and absolute maximum values.

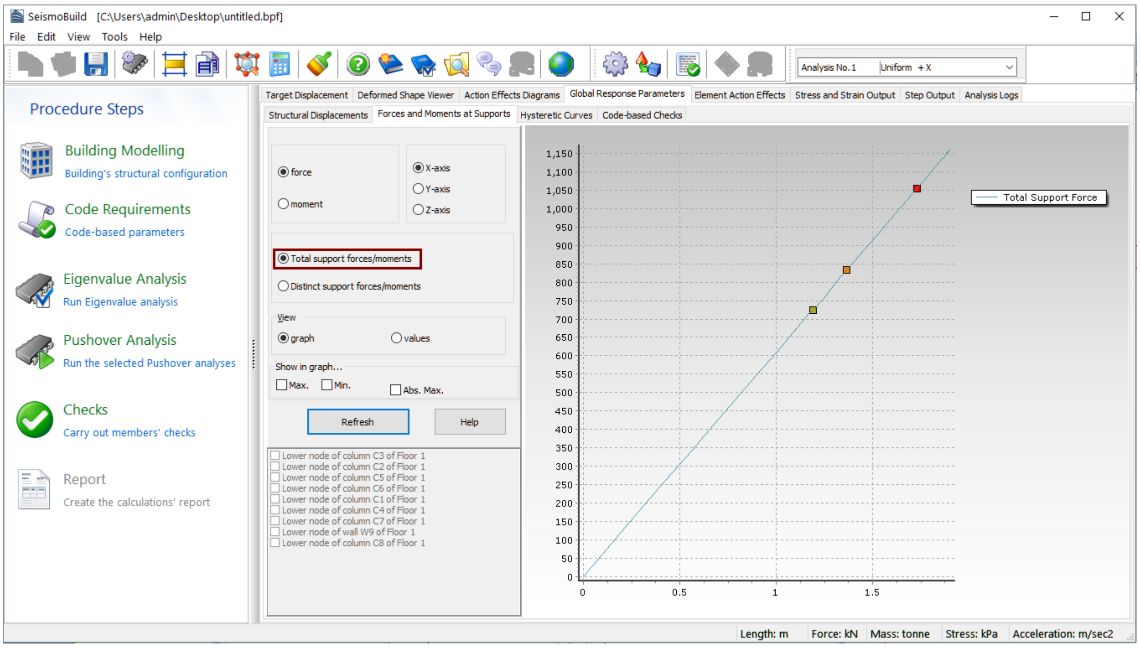


Global Response Parameters – Structural displacements

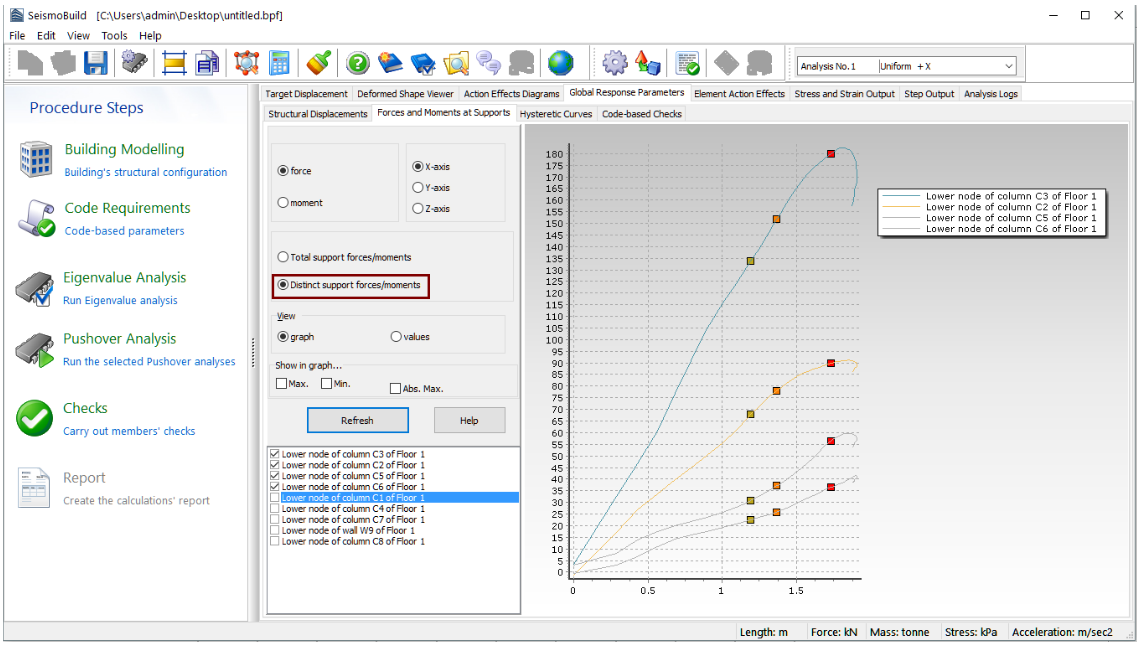
Forces and Moments at Supports

Similarly to the structural deformations, the support forces and moments in every direction can be obtained for all restrained nodes. The possibility for outputting the total support force/moment in the specified direction, instead of individual support values, enables also the computation and plotting of total base shear values, for instance. Finally, the maximum, minimum and absolute maximum values may be displayed on the selected graph.

NOTE: Evidently, the total moment support reaction does not include overturning effects, consisting simply of the sum of moments at the structure's supports.



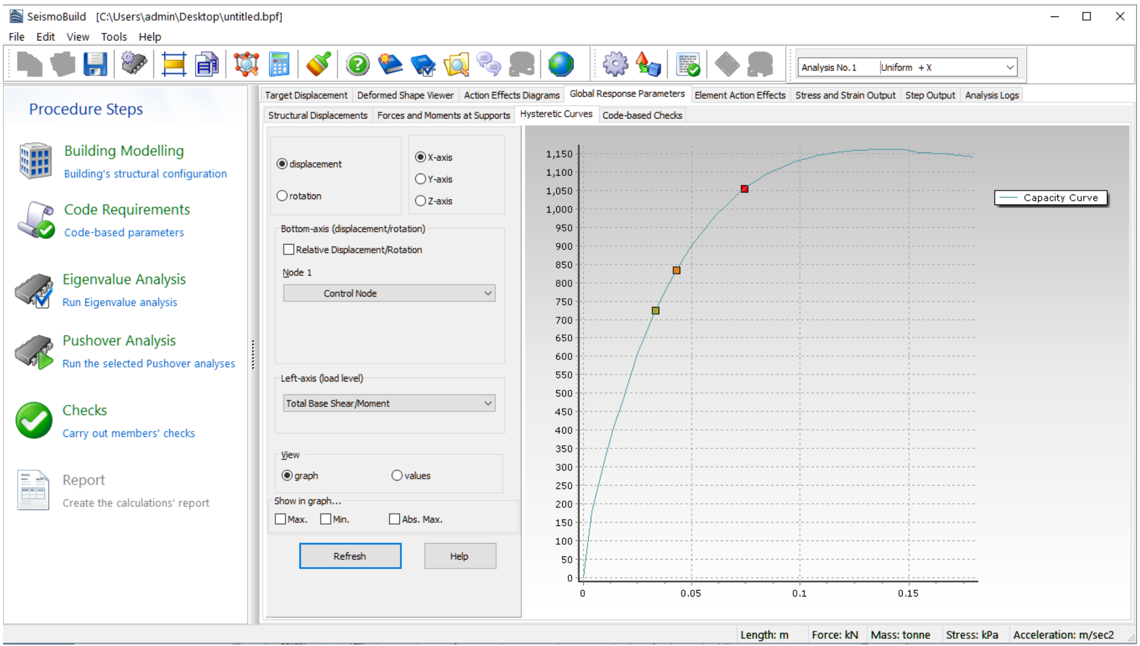
Global Response Parameters – Forces and Moments at Supports (total support)



Global Response Parameters – Forces and Moments at Supports (distinct support)

Hysteretic Curves

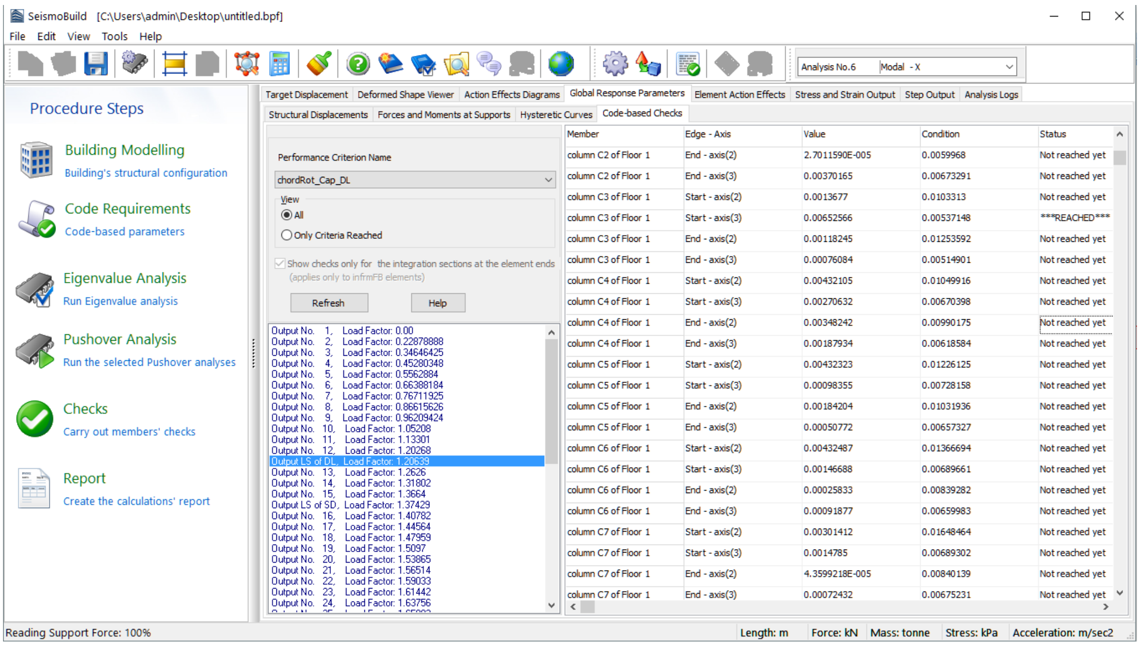
The user is able to specify a translational/rotational global degree-of-freedom to be plotted against the corresponding total base-shear/base-moment or load factor (pushover analysis). In static nonlinear analysis such a plot represents the structure's capacity curve. The possibility for relative displacement output is also available, as well as the choice of displaying in the graph the maximum, minimum and absolute maximum values.



Global Response Parameters – Hysteretic Curves

Code-based Checks

Here, it is possible for the user to perform the Code-based Checks in every step of the analysis. Users may select the name of the check from the relevant drop-down menu, and the step of the analysis, and click on the Refresh button. The results can be displayed for all the structural members, or just for those members that have reached the selected check.



Global Response Parameters – Code-based Checks

Element Action Effects

For the inelastic frame element type employed in the structural model, there can be three kinds of Element action effects results (subdivided into three categories), which are described in detail hereafter.

NOTE 1: Rotational degrees-of-freedom defined with regards to a particular axis, refer always to the rotation around, not along, that same axis. Hence, this is the convention that should be applied in the interpretation of all rotation/moment results obtained in this module.

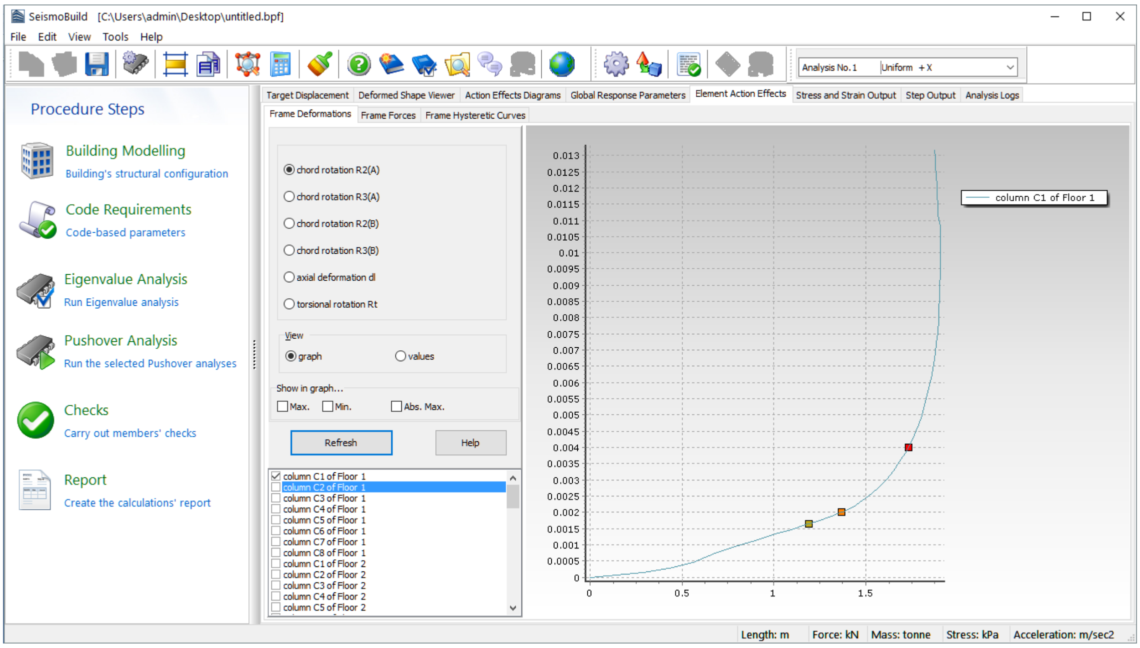
NOTE 2: Element chord-rotations output in this module correspond to structural member chord-rotations, allowing the direct employment of element chord rotations in seismic code verifications (see e.g. Eurocode 8, Italian National Seismic Code NTC-18, Greek Seismic Interventions Code KANEPE, etc).

NOTE 3: Under large displacements, shear forces at base elements might well be different from the corresponding reaction forces at the supports to which such base elements are connected to, since the former are defined in the (heavily rotated) local axis system of the element whilst the latter are defined with respect to the fixed global reference system.

NOTE 4: SeismoBuild does not automatically output dissipated energy values. However, users should be able to readily obtain such quantities through the product/integral of the force-displacement response.

Frame Deformations

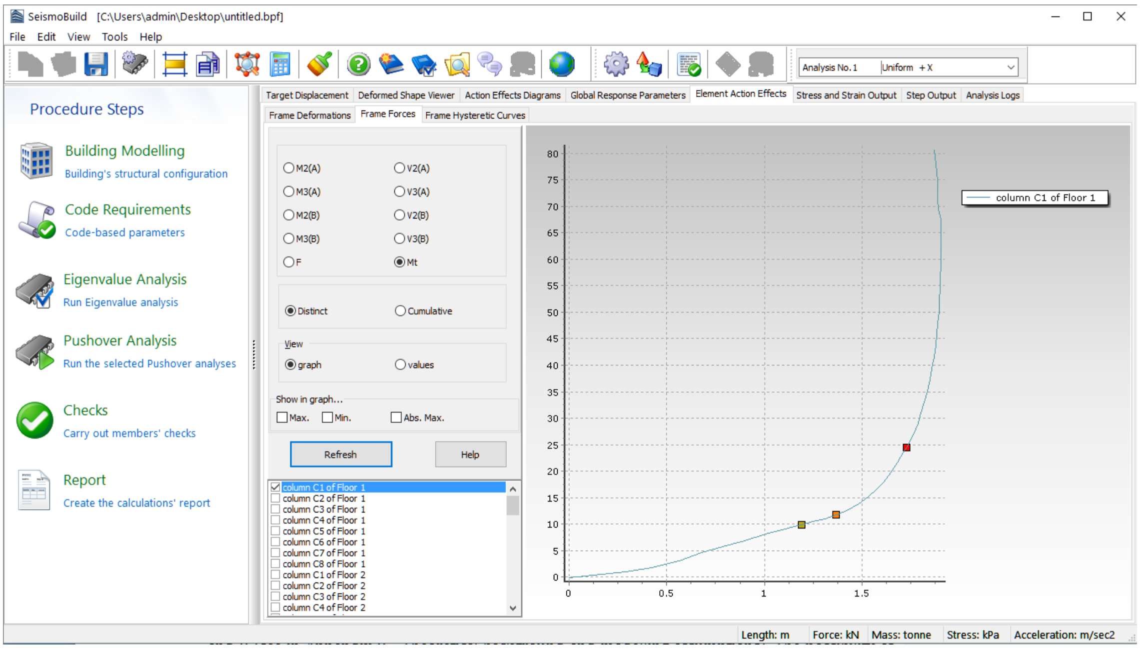
The deformations incurred by inelastic (infrm, infrmPH) frame elements, as computed in their local co-rotational system of reference, are provided. The values refer to the chord rotations at the end-nodes of each element (referred to as A and B, as indicated in Appendix B - Theoretical background and modelling assumptions), the axial deformation and the torsional rotation. Finally, the maximum, minimum and absolute maximum values may be displayed on the selected graph.



Element Action Effects – Frame Deformations

Frame Forces

The internal forces developed by inelastic (infrm, infrmPH) frame elements, as computed in their local co-rotational system of reference, are provided. The values refer to the internal forces (axial and shear) and moments (flexure and torsion) developed at the end-nodes of each element, referred to as A and B (see in Appendix B - Theoretical background and modelling assumptions). The possibility of obtaining the cumulative, rather than the distinct, results of each element can be very handy when a user is interested in adding the response of a number of elements (e.g. obtain the shear at a particular storey, given as the sum of the internal shear forces of the elements at that same level). Further, the maximum, minimum and absolute maximum values may be displayed on the selected graph.



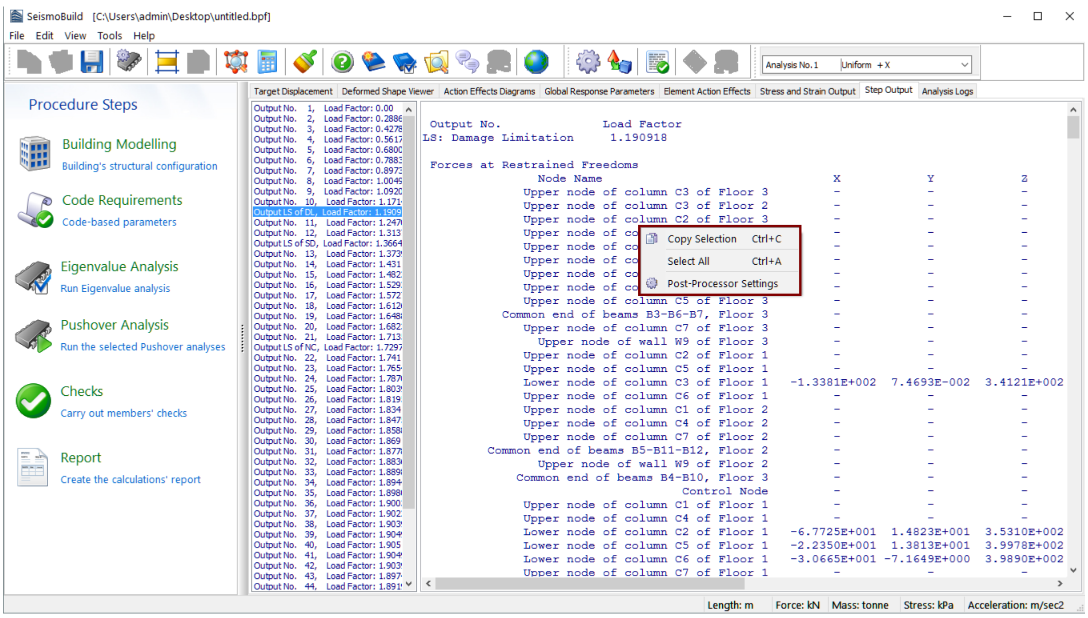
Element Action Effects – Frame Forces

Frame Hysteretic Curves

Hysteretic plots of deformation vs. internal forces developed by inelastic (infrm, infrmPH) frame elements, as computed in their local co-rotational system of reference, are provided. The maximum, minimum and absolute maximum values may be displayed on the selected graph.

Step Output

This post-processing module provides, in text file-type of output, all the analytical results (nodal displacements/rotations, support and element forces/moments, element strains and stresses) obtained by SeismoBuild at any given analysis step plus the analytical results for the selected limit state(s). The entire step output, or selected parts of it, can be copied to text editors for further manipulation, using the corresponding menu commands, keyboard shortcuts, toolbar buttons or right-click popup menu.



Step Output

Rather than copying and pasting the contents of this module, users may also choose to simply use the Export to Text File facility, which gives also the possibility of choosing the start and end output steps of interest, together with a step increment. This useful facility is available from the toolbar button .

Finally, and as noted in Advanced Settings >General, users may also activate the option of creating, at the end of every analysis, a text file (*.out) containing the output of the entire analysis (as given in this module). This feature may result useful for users, who wish to systematically, rather than occasionally, post-process the results using their own custom-made post-processing facility.

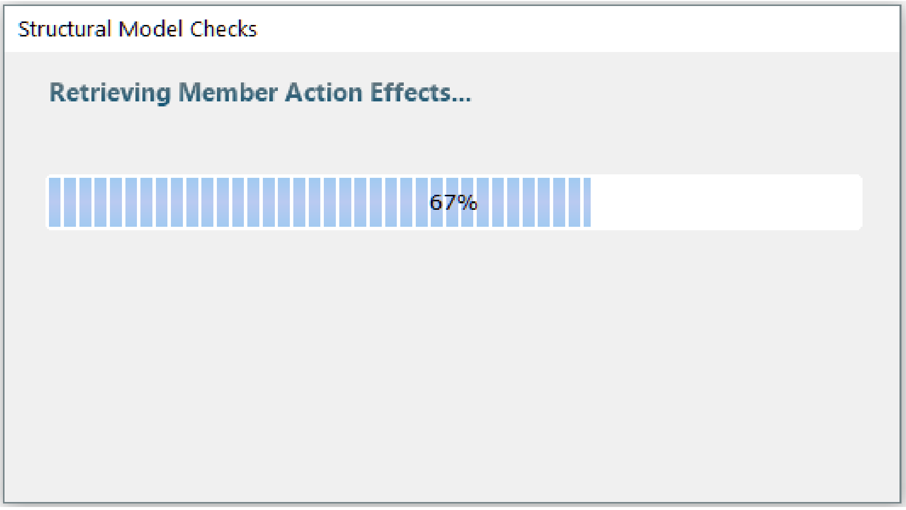
Analysis Logs

As discussed above, during any given analysis, a log of its numerical progress and of the performance response of the model is created and saved within the project's log file (*.log). The contents of such file can be visualised in the **Analysis Logs** module and, if required, copied and pasted into any other Windows application.

It is also noted that, since the date and time of the last analysis are saved within the log file, users can refer to this module when such type of information is required.

Checks

The results of the Code-based checks can be accessed through the corresponding module in the program Main Window. Once the Checks button is selected, an information window appears with a progress bar, providing a percentage indication of how far the structural model checks process has advanced to.



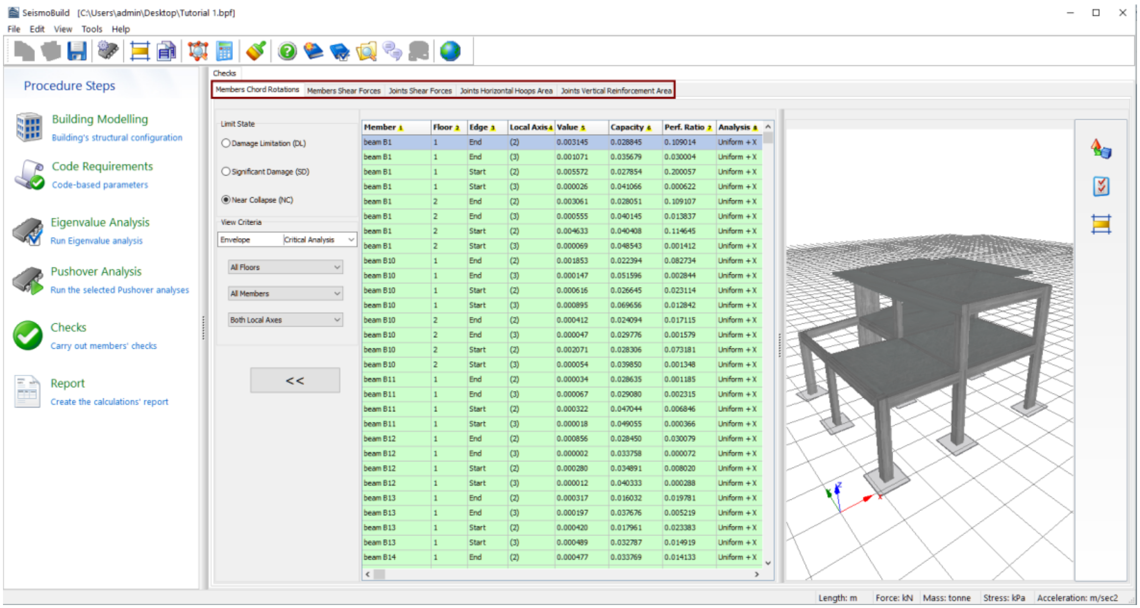
Structural Model Checks informative window

The **Checks** module features a series of tabs where results from different type of checks can be viewed in table or the 3D view format, and then copied into any other Windows applications (i.e. tabled results can be copied into a spreadsheet like Microsoft Excel, whilst results plots can be copied into a word-processing application, like Microsoft Word). Herein, the checks already selected in the Checks module of Code Requirements are shown.

The available tabs, within all the employed Codes, are listed below and will be described in the following paragraphs:

- Members Chord Rotations
- Members Bending Moments
- Members Shear Forces
- Members Strains (TBDY only)
- Members Tensile Deformations
- Members Compressive Deformations
- Members Tensile Forces
- Members Compressive Forces
- Joints Shear Forces (Eurocodes, ASCE 41-23 & TBDY)
- Joints Horizontal Hoops Area (Eurocodes only)
- Joints Vertical Reinforcement Area (Eurocodes only)
- Joints Ductility
- Joints Diagonal Tension (NTC & KANEPE)
- Joints Diagonal Compression (NTC & KANEPE)
- Interstorey Drifts (ASCE 41-23 & NTC)
- PGA ratios (NTC only)
- Seismic Risk Classification (NTC only)

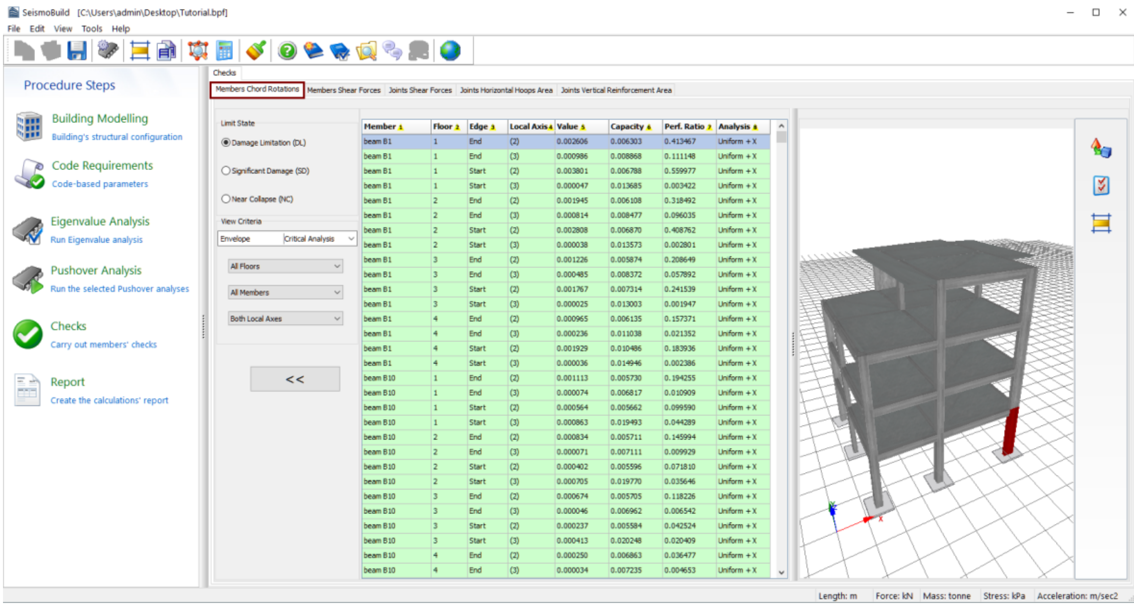
It is noted that in pushover analysis the checks are carried out at the particular step of the analysis that corresponds to the target displacement for the selected Limit State. On the contrary, in dynamic analysis the maxima of the response parameters throughout the time-history are obtained as demand and compared against the components' deformation or strength capacities. If there is exceedance of the capacity, the acceptance criteria are not fulfilled, if there is not, the acceptance criteria are deemed as fulfilled.



Checks Modules

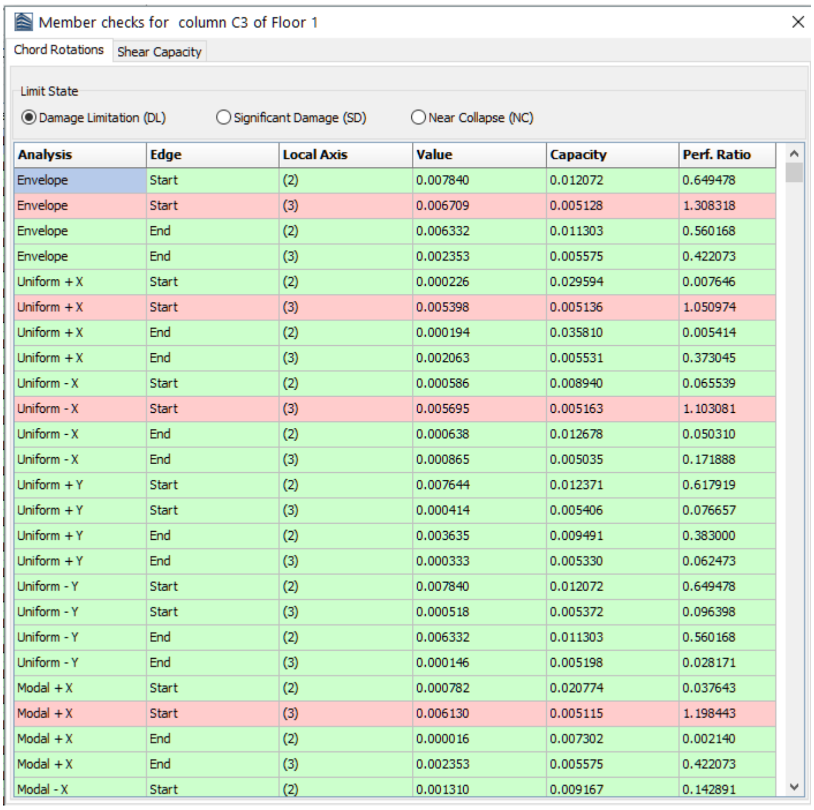
MEMBERS CHORD ROTATIONS

Herein the results of the deformation capacity checks for beams, columns and walls, according to the selected Code, are exported. Users may select for which Limit State, or Performance objective, to view the results and choose filters, which include the determination of the analysis, the floor, the members' type and the local axis. The available limit states are those already selected in the Limit States module in Code Requirements. Further, in the drop-down menu for the analysis selection apart from the executed analyses there is an option called Critical Analysis, which is the envelope of the results of all the analyses. The elements that have exceeded their deformation capacity are marked with red both in the table view and the 3D view format, whereas different visualisation options are available. For instance, users may select to display the members' performance ratios or each member's critical analysis on the 3D plot, or to display the elements with different colours, depending on the value of their performance ratio for the selected limit state.



Member Chord Rotations Module

It is possible to hide the data entry table through the corresponding button in order to view the 3D rendering of the structural model in 'full-screen' modality. By right clicking on an element and selecting the View Members Checks, a window appears with the checks for all the analyses for that particular element.


Member checks for column C3 of Floor 1

Chord Rotations | Shear Capacity

Limit State: Damage Limitation (DL) | Significant Damage (SD) | Near Collapse (NC)

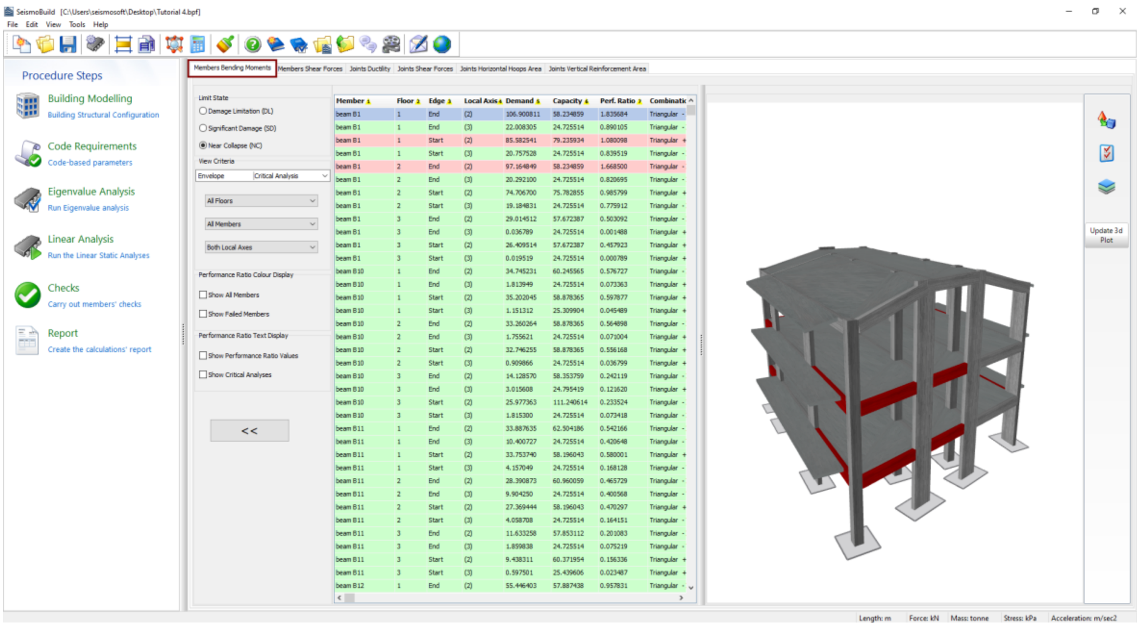
Analysis	Edge	Local Axis	Value	Capacity	Perf. Ratio
Envelope	Start	(2)	0.007840	0.012072	0.649478
Envelope	Start	(3)	0.006709	0.005128	1.308318
Envelope	End	(2)	0.006332	0.011303	0.560168
Envelope	End	(3)	0.002353	0.005575	0.422073
Uniform +X	Start	(2)	0.000226	0.029594	0.007646
Uniform +X	Start	(3)	0.005398	0.005136	1.050974
Uniform +X	End	(2)	0.000194	0.035810	0.005414
Uniform +X	End	(3)	0.002063	0.005531	0.373045
Uniform - X	Start	(2)	0.000586	0.008940	0.065539
Uniform - X	Start	(3)	0.005695	0.005163	1.103081
Uniform - X	End	(2)	0.000638	0.012678	0.050310
Uniform - X	End	(3)	0.000865	0.005035	0.171888
Uniform +Y	Start	(2)	0.007644	0.012371	0.617919
Uniform +Y	Start	(3)	0.000414	0.005406	0.076657
Uniform +Y	End	(2)	0.003635	0.009491	0.383000
Uniform +Y	End	(3)	0.000333	0.005330	0.062473
Uniform - Y	Start	(2)	0.007840	0.012072	0.649478
Uniform - Y	Start	(3)	0.000518	0.005372	0.096398
Uniform - Y	End	(2)	0.006332	0.011303	0.560168
Uniform - Y	End	(3)	0.000146	0.005198	0.028171
Modal +X	Start	(2)	0.000782	0.020774	0.037643
Modal +X	Start	(3)	0.006130	0.005115	1.198443
Modal +X	End	(2)	0.000016	0.007302	0.002140
Modal +X	End	(3)	0.002353	0.005575	0.422073
Modal - X	Start	(2)	0.001310	0.009167	0.142891

Member Checks module

A more detailed description on the members chord rotation checks and the equations used in SeismoBuild is available in the Deformation Capacity section of the respective appendix.

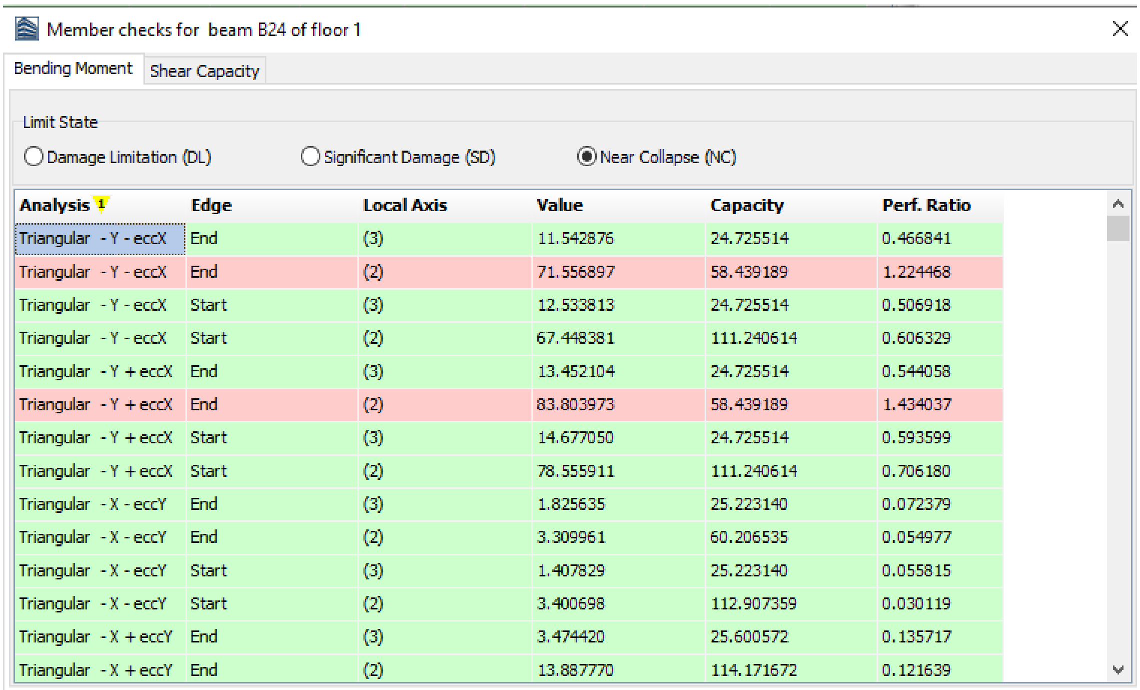
Members Bending Moments

Herein the results of the bending moments checks for beams, columns and walls, according to the selected Code, are exported. Users may select for which Limit State, or Performance objective, to view the results and choose filters, which include the determination of the analysis, the floor, the members' type and the local axis. The available limit states are those already selected in the Limit States module in Code Requirements. Further, in the drop-down menu for the analysis selection apart from the executed analyses there is an option called Critical Analysis, which is the envelope of the results of all the analyses. The elements that have exceeded their bending moment capacity are marked with red both in the table view and the 3D view format, whereas different visualisation options are available. For instance, users may select to display the members' performance ratios or each member's critical analysis on the 3D plot, or to display the elements with different colours, depending on the value of their performance ratio for the selected limit state.



Members Bending Moments Module

It is possible to hide the data entry table through the corresponding button in order to view the 3D rendering of the structural model in 'full-screen' modality. By right clicking on an element and selecting the View Members Checks, a window appears with the checks for all the analyses for that particular element.

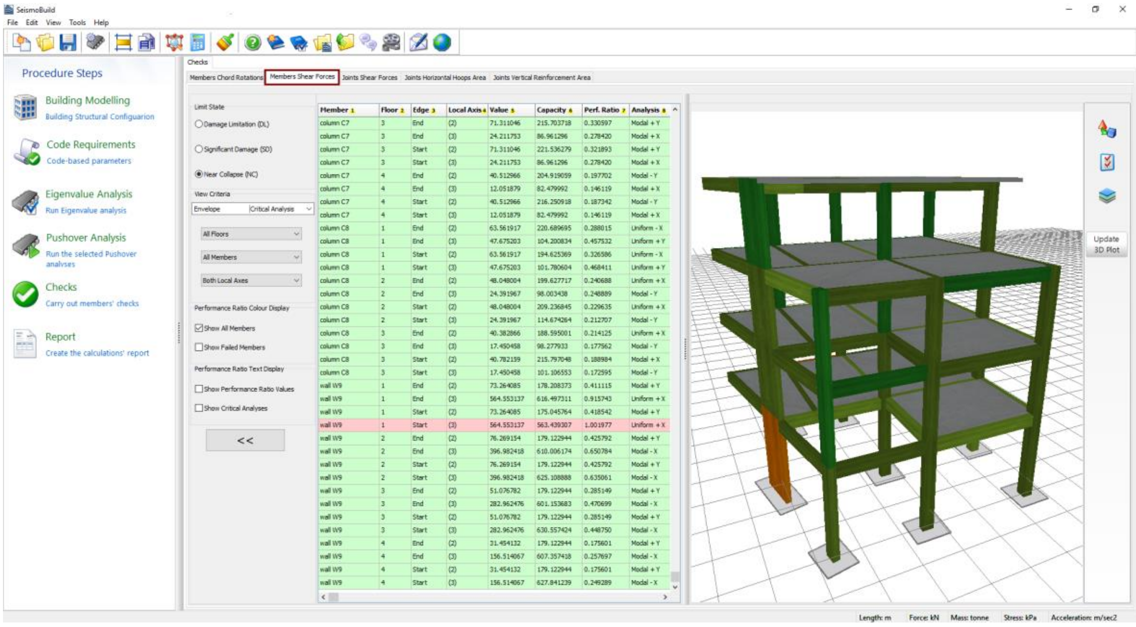


Member Checks module

A more detailed description on the members bending moment checks and the equations used in SeismoBuild is available in the Deformation Capacity section of the respective appendix.

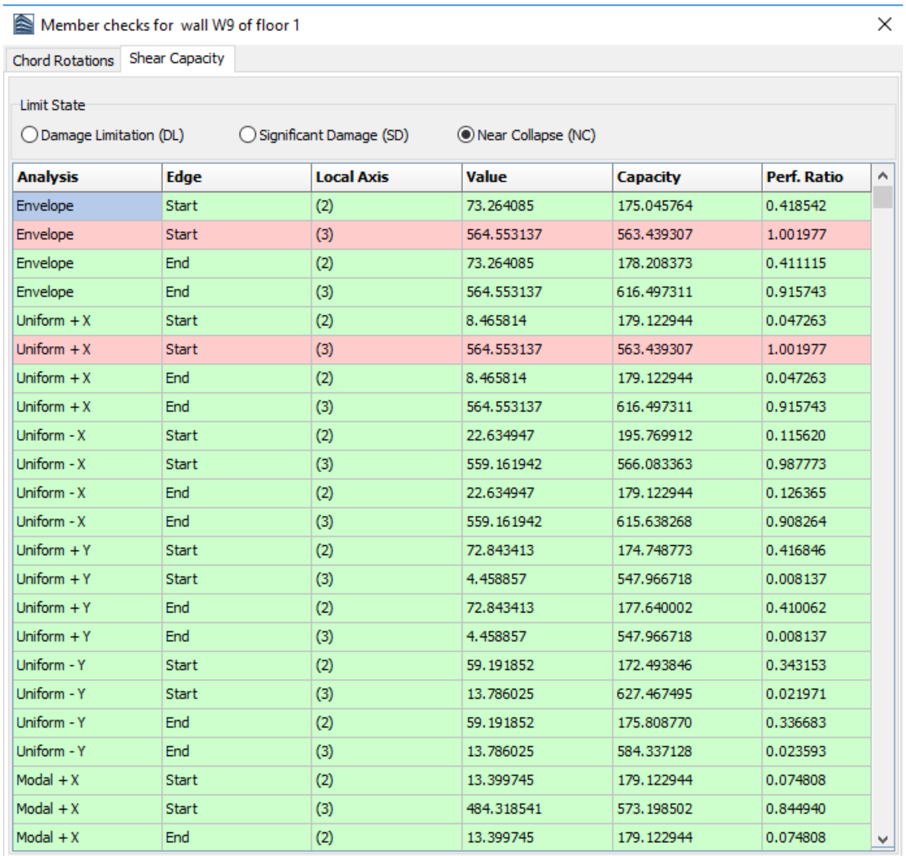
MEMBERS SHEAR FORCES

The results of the shear capacity checks for beams, columns and walls, according to the selected Code, can be visualised in this module. Users may select for which Limit State, or Performance objective, to view the results and choose filters, which include the determination of the analysis, the floor, the members' type and the local axis. The available limit states are those already selected in the Limit States module in Code Requirements. In addition, in the drop-down menu for the analysis selection apart from the executed analyses there is an option called Critical Analysis, which is the envelope of the results of all the analyses. The elements that have exceeded their shear capacity are marked with red both in the table view and the 3D view format, whereas different visualisation options are available. For instance, users may select to display the members' performance ratios or each member's critical analysis on the 3D plot, or to display the elements with different colours, depending on the value of their performance ratio for the selected limit state.



Member Shear Forces Module

It is possible to hide the data entry table through the corresponding button in order to view the 3D rendering of the structural model in 'full-screen' modality. By right clicking on an element and selecting the View Members Checks, a window appears with the checks for all the analyses for that particular element.



Member checks for wall W9 of floor 1

Chord Rotations | Shear Capacity

Limit State

Damage Limitation (DL) | Significant Damage (SD) | Near Collapse (NC)

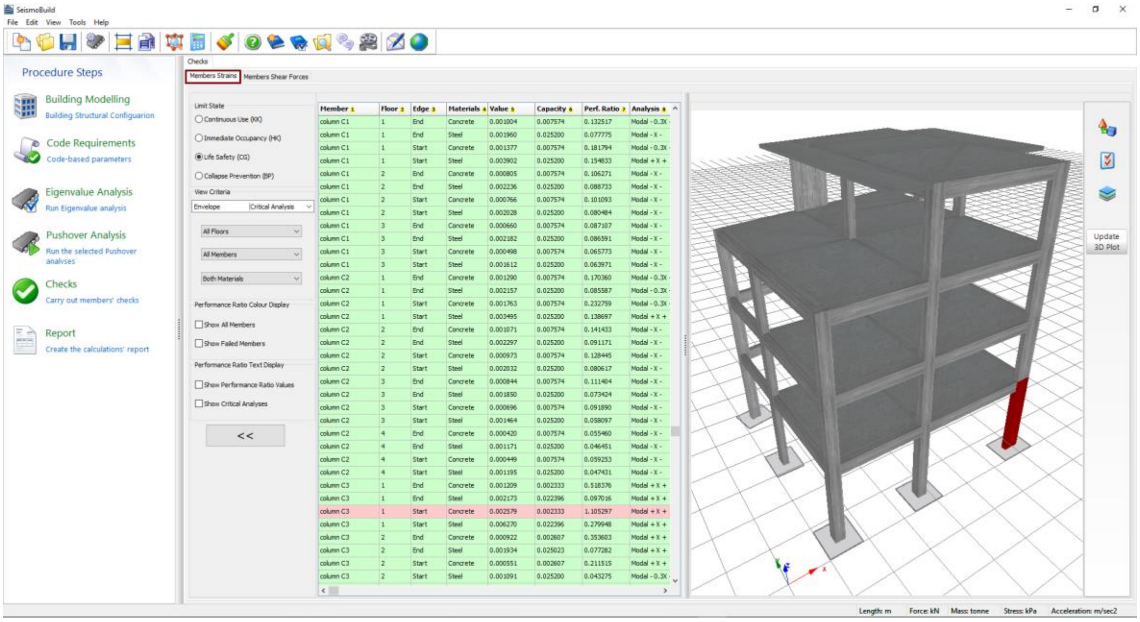
Analysis	Edge	Local Axis	Value	Capacity	Perf. Ratio
Envelope	Start	(2)	73.264085	175.045764	0.418542
Envelope	Start	(3)	564.553137	563.439307	1.001977
Envelope	End	(2)	73.264085	178.208373	0.411115
Envelope	End	(3)	564.553137	616.497311	0.915743
Uniform +X	Start	(2)	8.465814	179.122944	0.047263
Uniform +X	Start	(3)	564.553137	563.439307	1.001977
Uniform +X	End	(2)	8.465814	179.122944	0.047263
Uniform +X	End	(3)	564.553137	616.497311	0.915743
Uniform -X	Start	(2)	22.634947	195.769912	0.115620
Uniform -X	Start	(3)	559.161942	566.083363	0.987773
Uniform -X	End	(2)	22.634947	179.122944	0.126365
Uniform -X	End	(3)	559.161942	615.638268	0.908264
Uniform +Y	Start	(2)	72.843413	174.748773	0.416846
Uniform +Y	Start	(3)	4.458857	547.966718	0.008137
Uniform +Y	End	(2)	72.843413	177.640002	0.410062
Uniform +Y	End	(3)	4.458857	547.966718	0.008137
Uniform -Y	Start	(2)	59.191852	172.493846	0.343153
Uniform -Y	Start	(3)	13.786025	627.467495	0.021971
Uniform -Y	End	(2)	59.191852	175.808770	0.336683
Uniform -Y	End	(3)	13.786025	584.337128	0.023593
Modal +X	Start	(2)	13.399745	179.122944	0.074808
Modal +X	Start	(3)	484.318541	573.198502	0.844940
Modal +X	End	(2)	13.399745	179.122944	0.074808

Member Checks module

A more detailed description on the members shear forces checks and the equations used in SeismoBuild is available in the Shear capacity section of the respective appendix.

MEMBERS STRAINS (TBDY ONLY)

The results of the strains capacity checks for beams, columns and walls, can be visualised in this module (this check applies to TBDY only). Users may select for which Limit State, or Performance objective, to view the results and choose filters, which include the determination of the analysis, the floor, the members' type and the local axis. The available limit states are those already selected in the Limit States module in Code Requirements. In addition, in the drop-down menu for the analysis selection apart from the executed analyses there is an option called Critical Analysis, which is the envelope of the results of all the analyses. The elements that have exceeded their shear capacity are marked with red both in the table view and the 3D view format, whereas different visualisation options are available. For instance, users may select to display the members' performance ratios or each member's critical analysis on the 3D plot, or to display the elements with different colours, depending on the value of their performance ratio for the selected limit state.



Member Strains Module

It is possible to hide the data entry table through the corresponding button in order to view the 3D rendering of the structural model in 'full-screen' modality. By right clicking on an element and selecting the View Members Checks, a window appears with the checks for all the analyses for that particular element.

Member checks for column C3 of floor 1

Members Strains | Shear Capacity

Limit State

○ Continuous Use (KK) ○ Immediate Occupancy (HK) ◉ Life Safety (CG) ○ Collapse Prevention (BP)

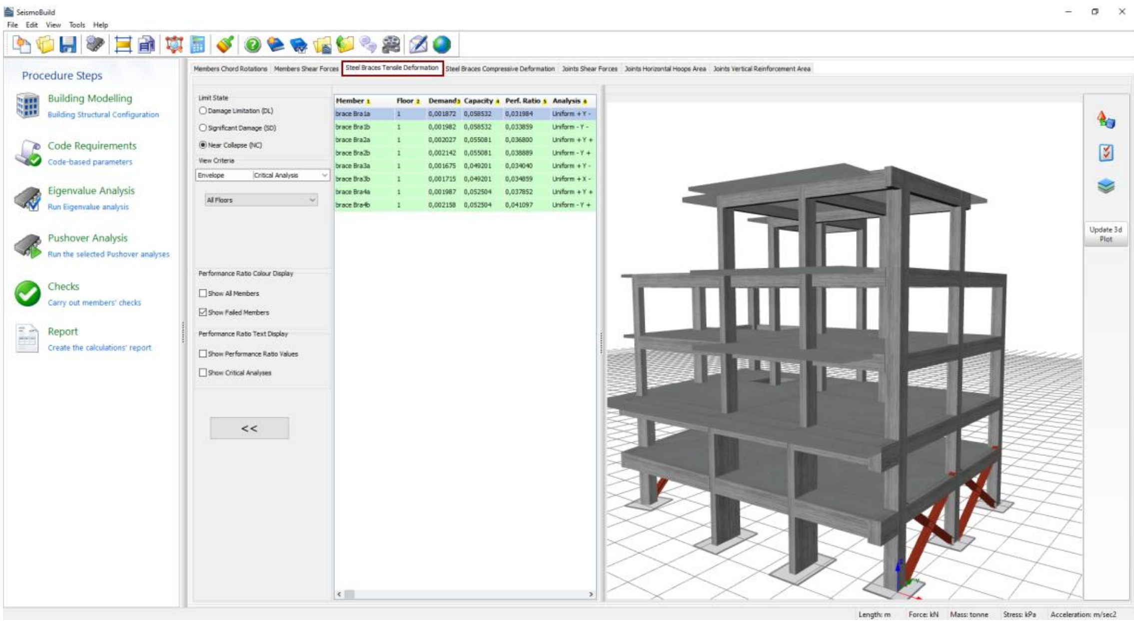
Analysis	Edge	Material	Value	Capacity	Perf. Ratio
Envelope	Start	Concrete	0.002579	0.002333	1.105297
Envelope	Start	Steel	0.006270	0.022396	0.279948
Envelope	End	Concrete	0.001209	0.002333	0.518376
Envelope	End	Steel	0.002173	0.022396	0.097016
Modal +X +0.3Y	Start	Concrete	0.002579	0.002333	1.105297
Modal +X +0.3Y	Start	Steel	0.006270	0.022396	0.279948
Modal +X +0.3Y	End	Concrete	0.001209	0.002333	0.518376
Modal +X +0.3Y	End	Steel	0.002173	0.022396	0.097016
Modal -X -0.3Y	Start	Concrete	0.001991	0.002625	0.758299
Modal -X -0.3Y	Start	Steel	0.006298	0.025200	0.249939
Modal -X -0.3Y	End	Concrete	0.000717	0.002625	0.272999
Modal -X -0.3Y	End	Steel	0.001514	0.025200	0.060061
Modal +0.3X +Y	Start	Concrete	0.001327	0.002625	0.505500
Modal +0.3X +Y	Start	Steel	0.002810	0.025200	0.111511
Modal +0.3X +Y	End	Concrete	0.000775	0.002625	0.295178
Modal +0.3X +Y	End	Steel	0.001570	0.025200	0.062287
Modal -0.3X -Y	Start	Concrete	0.001545	0.002625	0.588584
Modal -0.3X -Y	Start	Steel	0.002933	0.025200	0.116402
Modal -0.3X -Y	End	Concrete	0.000775	0.002625	0.295257
Modal -0.3X -Y	End	Steel	0.001651	0.025200	0.065512

Member Checks module

A more detailed description on the members strains checks and the equations used in SeismoBuild is available in the Members Strains section of the Appendix A.5 - TBDY.

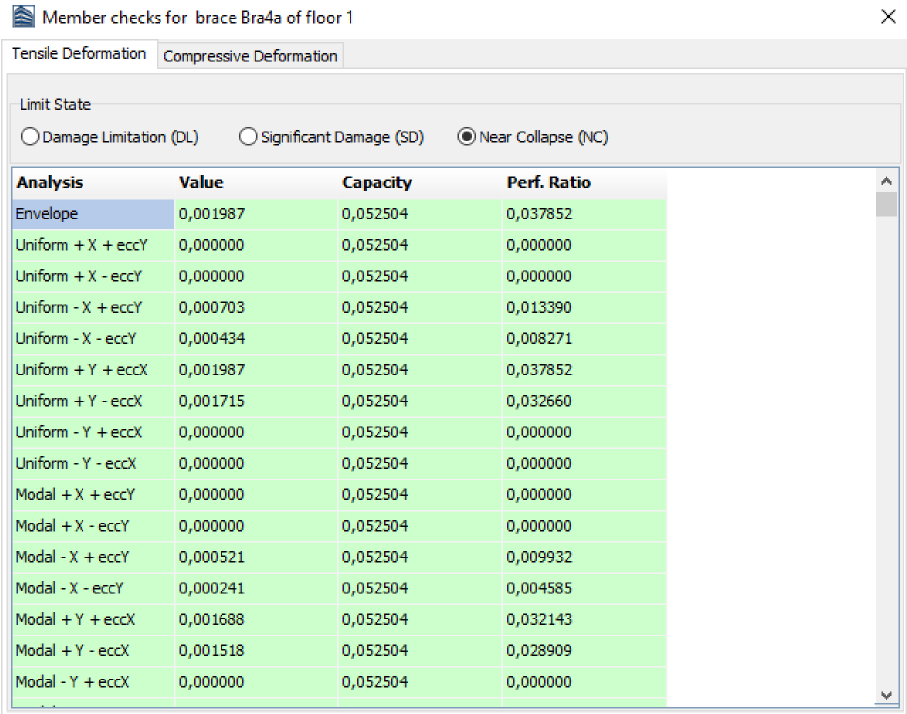
STEEL BRACES TENSILE DEFORMATIONS

The results of the tensile deformations check for steel braces, according to the selected Code, can be visualised in this module. Users may select for which Limit State, or Performance objective, to view the results and choose filters, which include the determination of the analysis and the floor. The available limit states are those already selected in the Limit States module in Code Requirements. In addition, in the drop-down menu for the analysis selection apart from the executed analyses there is an option called Critical Analysis, which is the envelope of the results of all the analyses. The elements that have exceeded their tensile deformation capacity are marked with red both in the table view and the 3D view format, whereas different visualisation options are available. For instance, users may select to display the members' performance ratios or each member's critical analysis on the 3D plot, or to display the elements with different colours, depending on the value of their performance ratio for the selected limit state.



Steel Braces Tensile Deformation Module

It is possible to hide the data entry table through the corresponding button in order to view the 3D rendering of the structural model in 'full-screen' modality. By right clicking on an element and selecting the View All Member Checks, a window appears with the checks for all the analyses for that particular element.



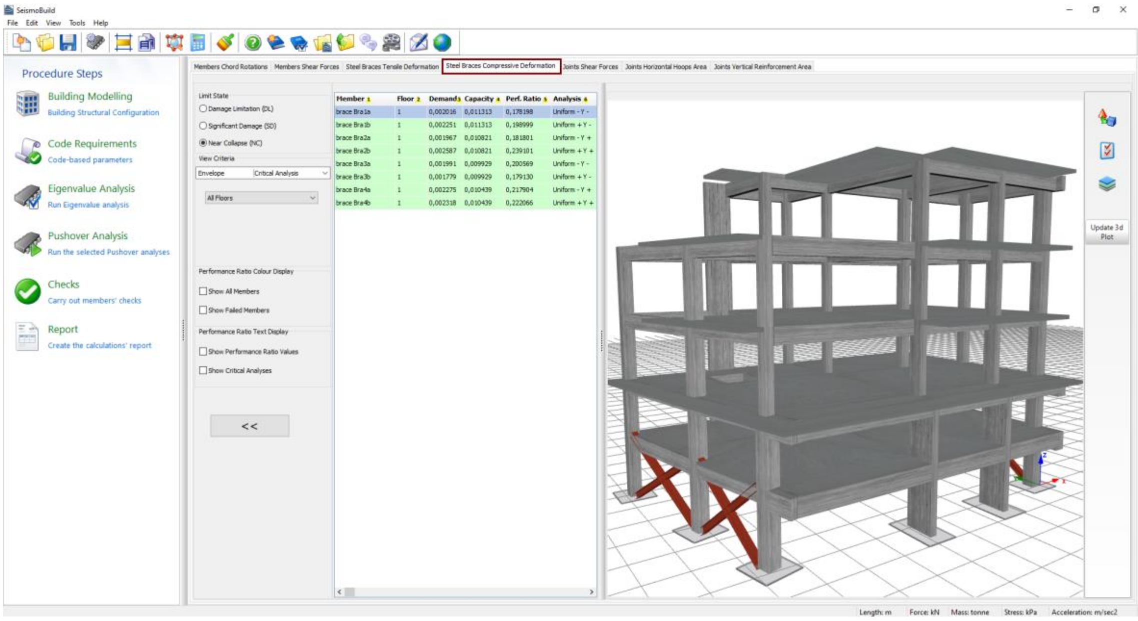
Analysis	Value	Capacity	Perf. Ratio
Envelope	0,001987	0,052504	0,037852
Uniform +X +eccY	0,000000	0,052504	0,000000
Uniform +X -eccY	0,000000	0,052504	0,000000
Uniform -X +eccY	0,000703	0,052504	0,013390
Uniform -X -eccY	0,000434	0,052504	0,008271
Uniform +Y +eccX	0,001987	0,052504	0,037852
Uniform +Y -eccX	0,001715	0,052504	0,032660
Uniform -Y +eccX	0,000000	0,052504	0,000000
Uniform -Y -eccX	0,000000	0,052504	0,000000
Modal +X +eccY	0,000000	0,052504	0,000000
Modal +X -eccY	0,000000	0,052504	0,000000
Modal -X +eccY	0,000521	0,052504	0,009932
Modal -X -eccY	0,000241	0,052504	0,004585
Modal +Y +eccX	0,001688	0,052504	0,032143
Modal +Y -eccX	0,001518	0,052504	0,028909
Modal -Y +eccX	0,000000	0,052504	0,000000

Member Checks module

A more detailed description on the members tensile deformations checks and the equations used in SeismoBuild is available in the Steel Braces Axial Deformations section of the respective appendix.

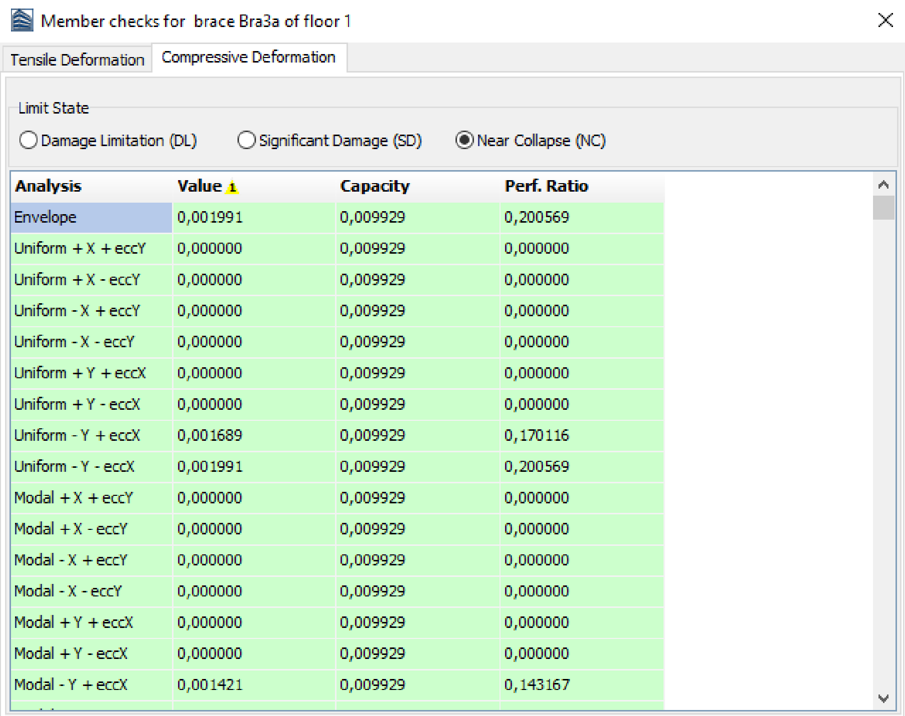
STEEL BRACES COMPRESSIVE DEFORMATIONS

The results of the compressive deformations check for steel braces, according to the selected Code, can be visualised in this module. Users may select for which Limit State, or Performance objective, to view the results and choose filters, which include the determination of the analysis and the floor. The available limit states are those already selected in the Limit States module in Code Requirements. In addition, in the drop-down menu for the analysis selection apart from the executed analyses there is an option called Critical Analysis, which is the envelope of the results of all the analyses. The elements that have exceeded their compressive deformation capacity are marked with red both in the table view and the 3D view format, whereas different visualisation options are available. For instance, users may select to display the members' performance ratios or each member's critical analysis on the 3D plot, or to display the elements with different colours, depending on the value of their performance ratio for the selected limit state.



Steel Braces Compressive Deformation Module

It is possible to hide the data entry table through the corresponding button in order to view the 3D rendering of the structural model in 'full-screen' modality. By right clicking on an element and selecting the View All Member Checks, a window appears with the checks for all the analyses for that particular element.



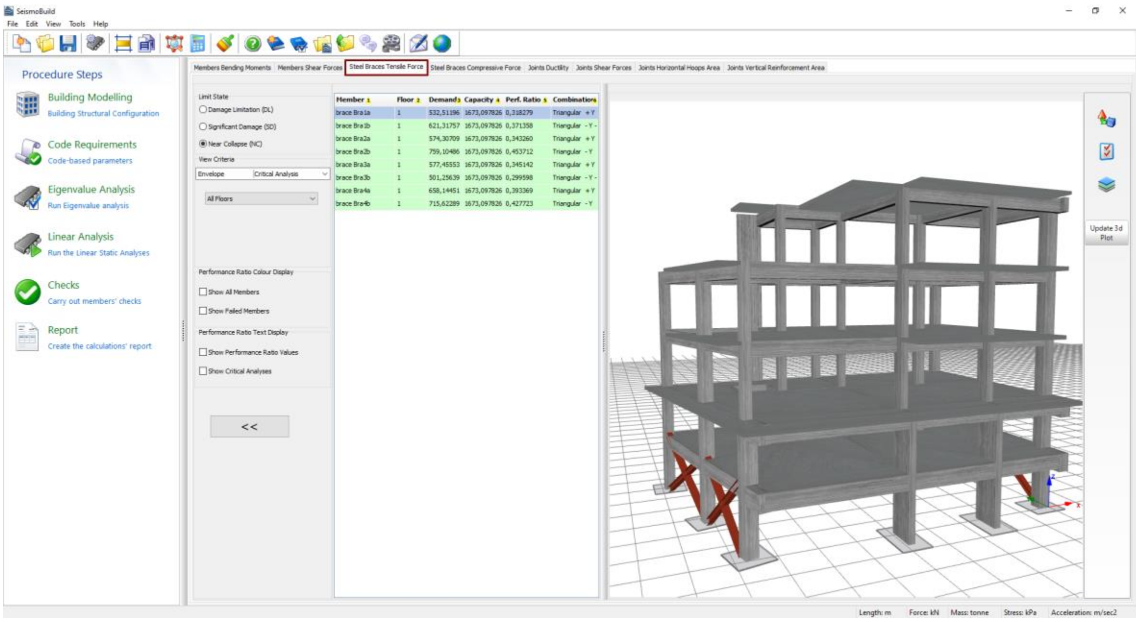
Analysis	Value	Capacity	Perf. Ratio
Envelope	0,001991	0,009929	0,200569
Uniform +X +eccY	0,000000	0,009929	0,000000
Uniform +X -eccY	0,000000	0,009929	0,000000
Uniform -X +eccY	0,000000	0,009929	0,000000
Uniform -X -eccY	0,000000	0,009929	0,000000
Uniform +Y +eccX	0,000000	0,009929	0,000000
Uniform +Y -eccX	0,000000	0,009929	0,000000
Uniform -Y +eccX	0,001689	0,009929	0,170116
Uniform -Y -eccX	0,001991	0,009929	0,200569
Modal +X +eccY	0,000000	0,009929	0,000000
Modal +X -eccY	0,000000	0,009929	0,000000
Modal -X +eccY	0,000000	0,009929	0,000000
Modal -X -eccY	0,000000	0,009929	0,000000
Modal +Y +eccX	0,000000	0,009929	0,000000
Modal +Y -eccX	0,000000	0,009929	0,000000
Modal -Y +eccX	0,001421	0,009929	0,143167

Member Checks module

A more detailed description on the members compressive deformations checks and the equations used in SeismoBuild is available in the Steel Braces Axial Deformations section of the respective appendix.

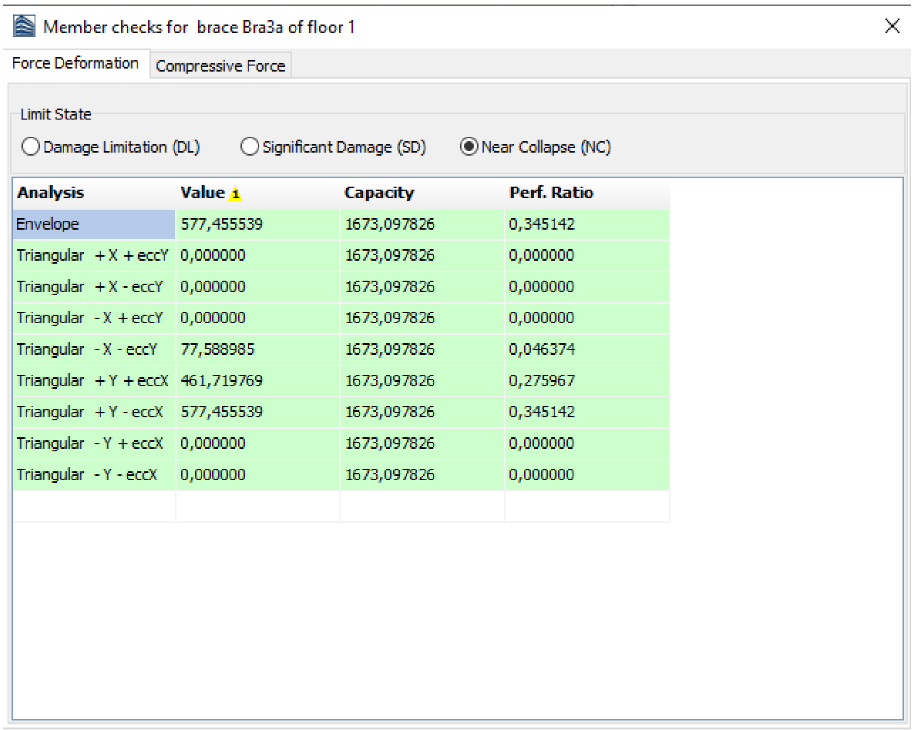
STEEL BRACES TENSILE FORCES

The results of the tensile forces check for steel braces, according to the selected Code, can be visualised in this module. Users may select for which Limit State, or Performance objective, to view the results and choose filters, which include the determination of the analysis and the floor. The available limit states are those already selected in the Limit States module in Code Requirements. In addition, in the drop-down menu for the analysis selection apart from the executed analyses there is an option called Critical Analysis, which is the envelope of the results of all the analyses. The elements that have exceeded their tensile force capacity are marked with red both in the table view and the 3D view format, whereas different visualisation options are available. For instance, users may select to display the members' performance ratios or each member's critical analysis on the 3D plot, or to display the elements with different colours, depending on the value of their performance ratio for the selected limit state.



Steel Braces Tensile Force Module

It is possible to hide the data entry table through the corresponding button in order to view the 3D rendering of the structural model in 'full-screen' modality. By right clicking on an element and selecting the View All Member Checks, a window appears with the checks for all the analyses for that particular element.



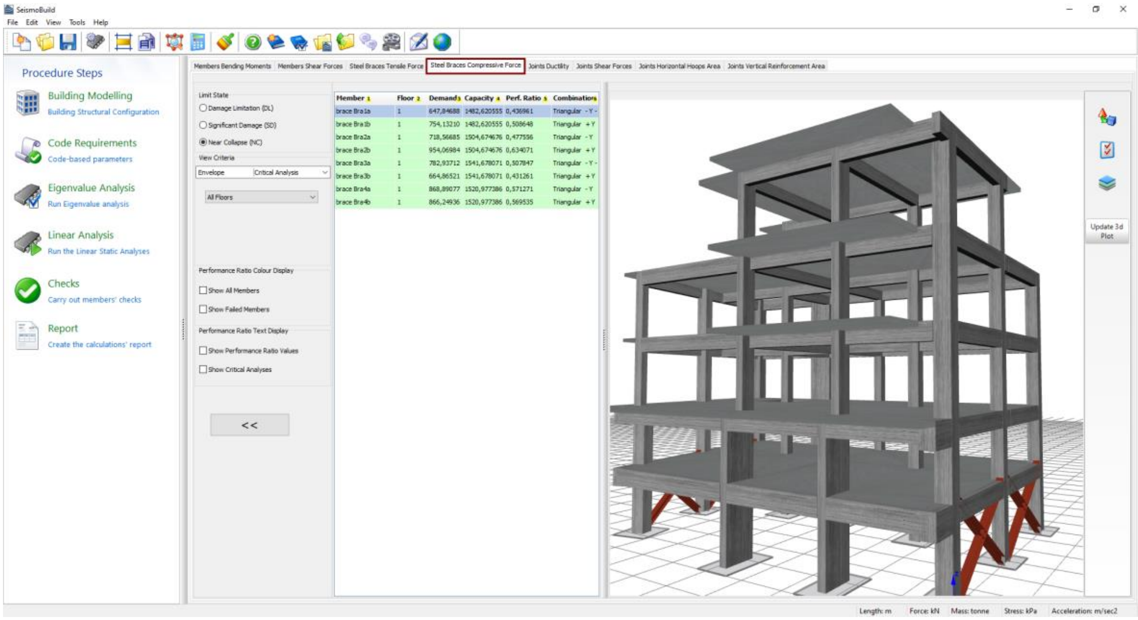
Analysis	Value	Capacity	Perf. Ratio
Envelope	577,455539	1673,097826	0,345142
Triangular +X +eccY	0,000000	1673,097826	0,000000
Triangular +X -eccY	0,000000	1673,097826	0,000000
Triangular -X +eccY	0,000000	1673,097826	0,000000
Triangular -X -eccY	77,588985	1673,097826	0,046374
Triangular +Y +eccX	461,719769	1673,097826	0,275967
Triangular +Y -eccX	577,455539	1673,097826	0,345142
Triangular -Y +eccX	0,000000	1673,097826	0,000000
Triangular -Y -eccX	0,000000	1673,097826	0,000000

Member Checks module

A more detailed description on the members tensile forces checks and the equations used in SeismoBuild is available in the Steel Braces Axial Forces section of the respective appendix.

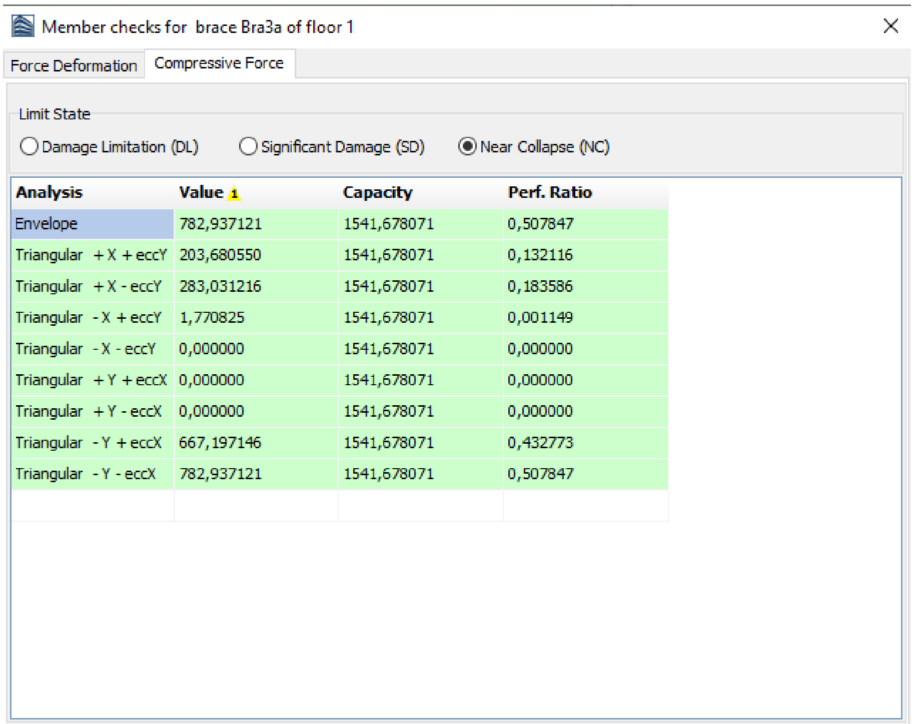
Steel Braces Compressive Forces

The results of the compressive forces check for steel braces, according to the selected Code, can be visualised in this module. Users may select for which Limit State, or Performance objective, to view the results and choose filters, which include the determination of the analysis and the floor. The available limit states are those already selected in the Limit States module in Code Requirements. In addition, in the drop-down menu for the analysis selection apart from the executed analyses there is an option called Critical Analysis, which is the envelope of the results of all the analyses. The elements that have exceeded their compressive force capacity are marked with red both in the table view and the 3D view format, whereas different visualisation options are available. For instance, users may select to display the members' performance ratios or each member's critical analysis on the 3D plot, or to display the elements with different colours, depending on the value of their performance ratio for the selected limit state.



Steel Braces Compressive Force Module

It is possible to hide the data entry table through the corresponding button in order to view the 3D rendering of the structural model in 'full-screen' modality. By right clicking on an element and selecting the View All Member Checks, a window appears with the checks for all the analyses for that particular element.


Member checks for brace Bra3a of floor 1

Force Deformation | Compressive Force

Limit State

Damage Limitation (DL) | Significant Damage (SD) | Near Collapse (NC)

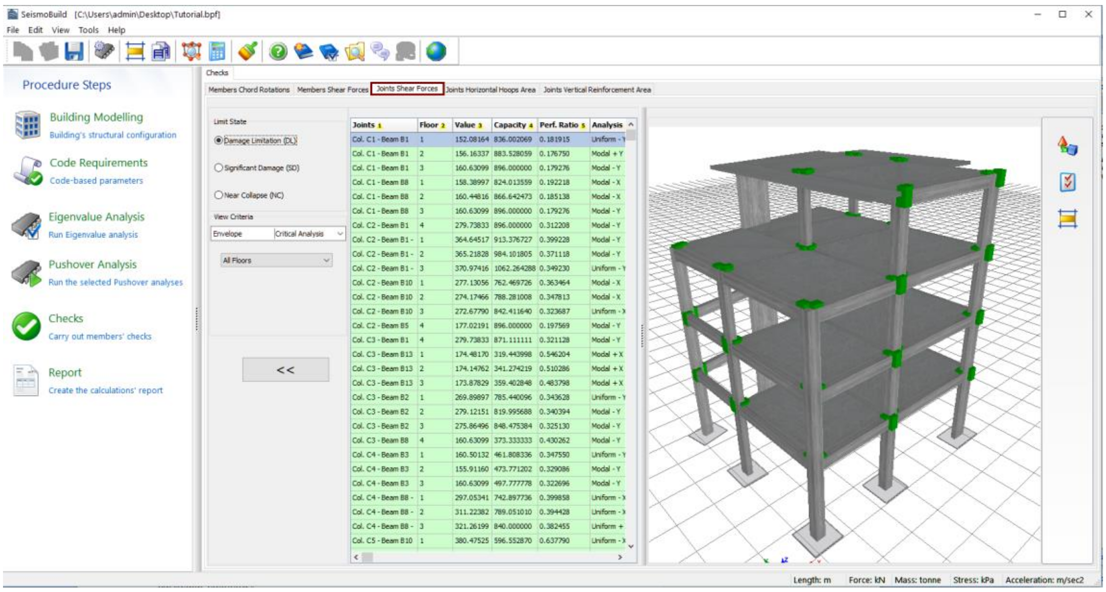
Analysis	Value	Capacity	Perf. Ratio
Envelope	782,937121	1541,678071	0,507847
Triangular +X +eccY	203,680550	1541,678071	0,132116
Triangular +X -eccY	283,031216	1541,678071	0,183586
Triangular -X +eccY	1,770825	1541,678071	0,001149
Triangular -X -eccY	0,000000	1541,678071	0,000000
Triangular +Y +eccX	0,000000	1541,678071	0,000000
Triangular +Y -eccX	0,000000	1541,678071	0,000000
Triangular -Y +eccX	667,197146	1541,678071	0,432773
Triangular -Y -eccX	782,937121	1541,678071	0,507847

Member Checks module

A more detailed description on the members compressive forces checks and the equations used in SeismoBuild is available in the Steel Braces Axial Forces section of the respective appendix.

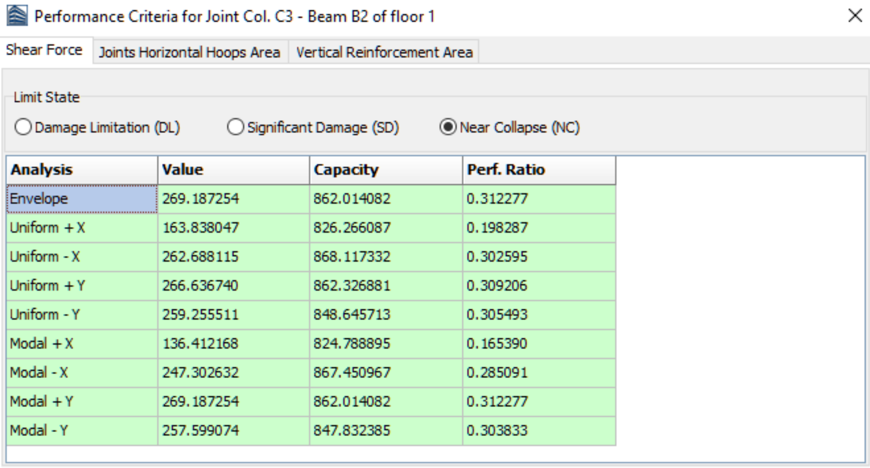
JOINTS SHEAR FORCES (EUROCODES, ASCE 41-23 & TBDY)

Herein the results of the shear forces checks for beam–column joints, according to the selected Code, are exported (this check applies to Eurocodes, ASCE 41-23 and TBDY). Users may select for which Limit State, or Performance objective, to view the results and choose filters, which include the determination of the analysis and the floor. The available limit states are those already selected in the Limit States module in Code Requirements. In addition, in the drop-down menu for the analysis selection apart from the executed analyses there is an option called Critical Analysis, which is the envelope of the results of all the analyses. The elements that have exceeded their shear capacity are marked with red both in the table view and the 3D view format, whereas different visualisation options are available. For instance, users may select to display the joints' performance ratios or each joint's critical analysis on the 3D plot, or to display the joints with different colours, depending on the value of their performance ratio for the selected limit state.



Joints Shear Forces Module

It is possible to hide the data entry table through the corresponding button in order to view the 3D rendering of the structural model in 'full-screen' modality. By right clicking on an element and selecting the View Members Checks, a window appears with the checks for all the analyses for that particular element.



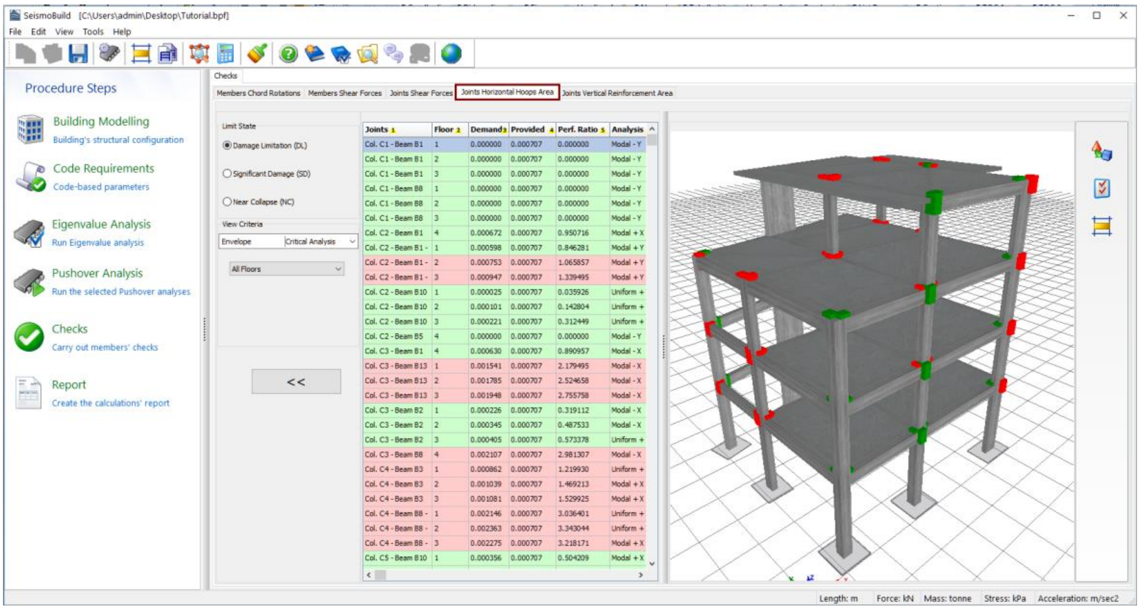
Analysis	Value	Capacity	Perf. Ratio
Envelope	269.187254	862.014082	0.312277
Uniform +X	163.838047	826.266087	0.198287
Uniform -X	262.688115	868.117332	0.302595
Uniform +Y	266.636740	862.326881	0.309206
Uniform -Y	259.255511	848.645713	0.305493
Modal +X	136.412168	824.788895	0.165390
Modal -X	247.302632	867.450967	0.285091
Modal +Y	269.187254	862.014082	0.312277
Modal -Y	257.599074	847.832385	0.303833

Joint Checks module

A more detailed description on the members compressive forces checks and the equations used in SeismoBuild is available in the Joints Shear Forces section of the respective appendix.

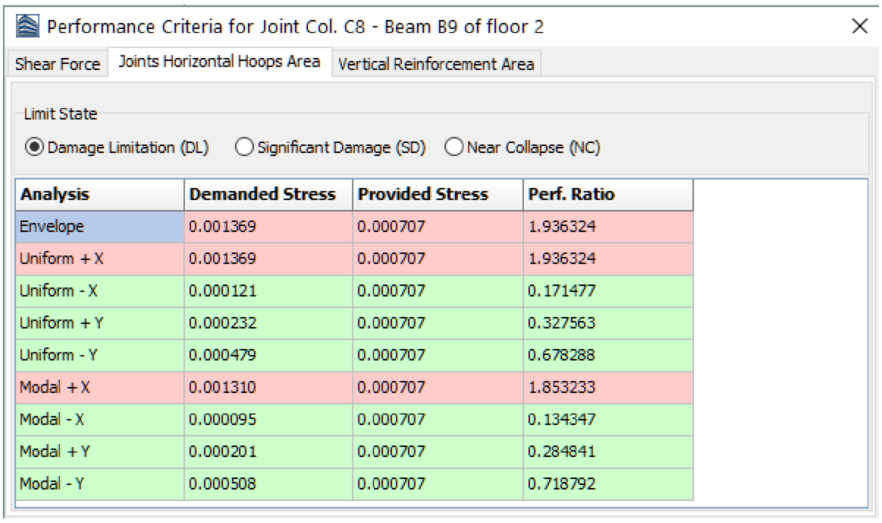
JOINTS HORIZONTAL HOOPS AREA (EUROCODES ONLY)

Herein the results of the horizontal hoops area checks for beam–column joints are exported (this check applies only to Eurocode 8). Users may select for which Limit State to view the results and choose filters, which include the determination of the analysis and the floor. The available limit states are those already selected in the Limit States module in Code Requirements. In addition, in the drop-down menu for the analysis selection apart from the executed analyses there is an option called Critical Analysis, which is the envelope of the results of all the analyses. The elements whose horizontal hoops area is less than the Code defined are marked with red both in the table view and the 3D view format, whereas different visualisation options are available. For instance, users may select to display the joints' performance ratios or each joint's critical analysis on the 3D plot, or to display the joints with different colours, depending on the value of their performance ratio for the selected limit state.



Joints Horizontal Hoops Area Module

It is possible to hide the data entry table through the corresponding button in order to view the 3D rendering of the structural model in 'full-screen' modality. By right clicking on an element and selecting the View Members Checks, a window appears with the checks for all the analyses for that particular element.


Performance Criteria for Joint Col. C8 - Beam B9 of floor 2

Shear Force | Joints Horizontal Hoops Area | Vertical Reinforcement Area

Limit State: Damage Limitation (DL) / Significant Damage (SD) / Near Collapse (NC)

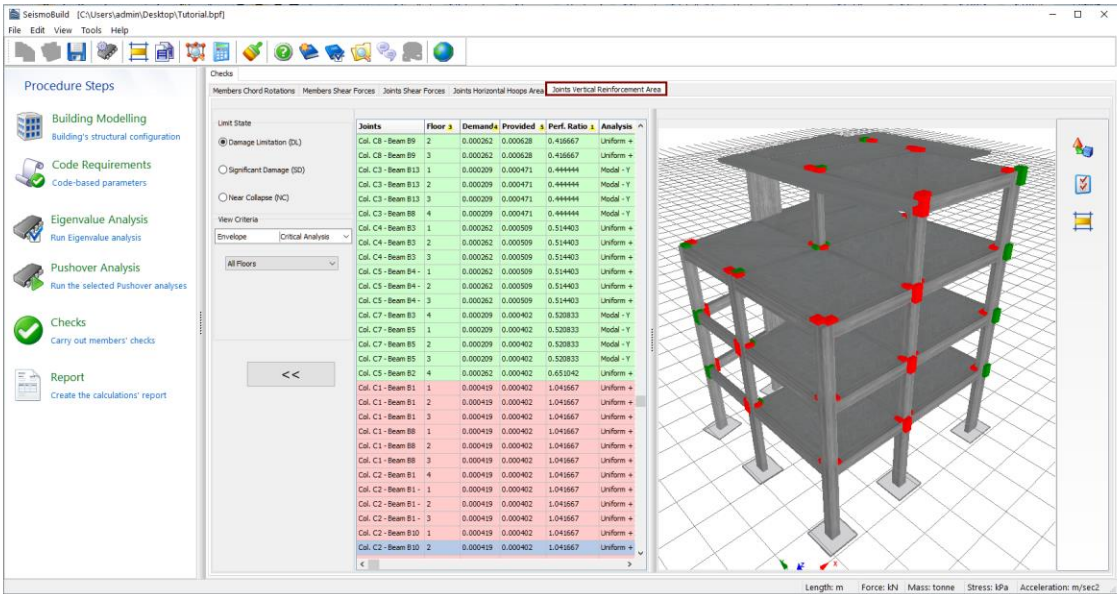
Analysis	Demanded Stress	Provided Stress	Perf. Ratio
Envelope	0.001369	0.000707	1.936324
Uniform +X	0.001369	0.000707	1.936324
Uniform - X	0.000121	0.000707	0.171477
Uniform +Y	0.000232	0.000707	0.327563
Uniform - Y	0.000479	0.000707	0.678288
Modal +X	0.001310	0.000707	1.853233
Modal - X	0.000095	0.000707	0.134347
Modal +Y	0.000201	0.000707	0.284841
Modal - Y	0.000508	0.000707	0.718792

Joint Checks module

A more detailed description on the joints' horizontal hoops area checks and the equations used in SeismoBuild is available Joints Horizontal Hoops Area section of the Appendix A.1-EUROCODES.

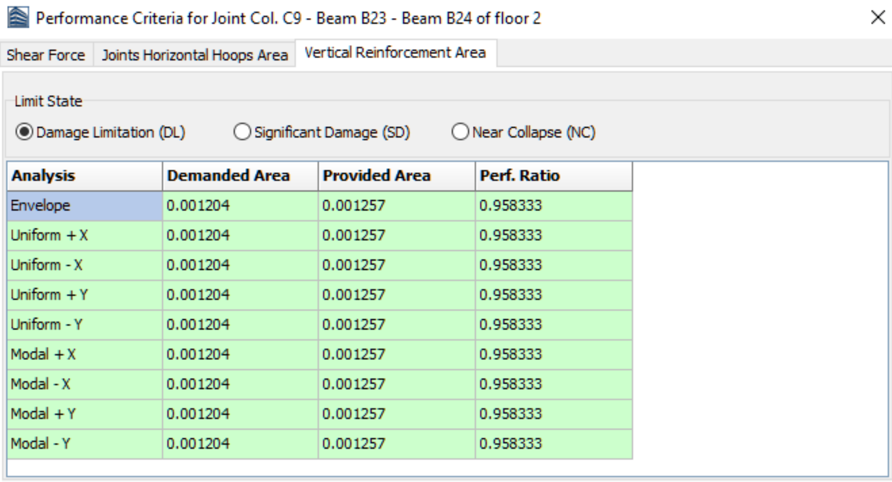
JOINTS VERTICAL REINFORCEMENT AREA (EUROCODES ONLY)

Herein the results of the vertical reinforcement area checks for beam–column joints are presented (this check applies only to Eurocode 8). Users may select for which Limit State to view the results and choose filters, which include the determination of the analysis and the floor. The available limit states are those already selected in the Limit States module in Code Requirements. In addition, in the drop-down menu for the analysis selection apart from the executed analyses there is an option called Critical Analysis, which is the envelope of the results of all the analyses. The elements whose vertical reinforcement area is less than the Code defined are marked with red both in the table view and the 3D view format, whereas different visualisation options are available. For instance, users may select to display the joints' performance ratios or each joint's critical analysis on the 3D plot, or to display the joints with different colours, depending on the value of their performance ratio for the selected limit state.



Joints Vertical Reinforcement Area Module

It is possible to hide the data entry table through the corresponding button in order to view the 3D rendering of the structural model in 'full-screen' modality. By right clicking on an element and selecting the View Members Checks, a window appears with the checks for all the analyses for that particular element.


Performance Criteria for Joint Col. C9 - Beam B23 - Beam B24 of floor 2

Shear Force | Joints Horizontal Hoops Area | Vertical Reinforcement Area

Limit State: Damage Limitation (DL) · Significant Damage (SD) · Near Collapse (NC)

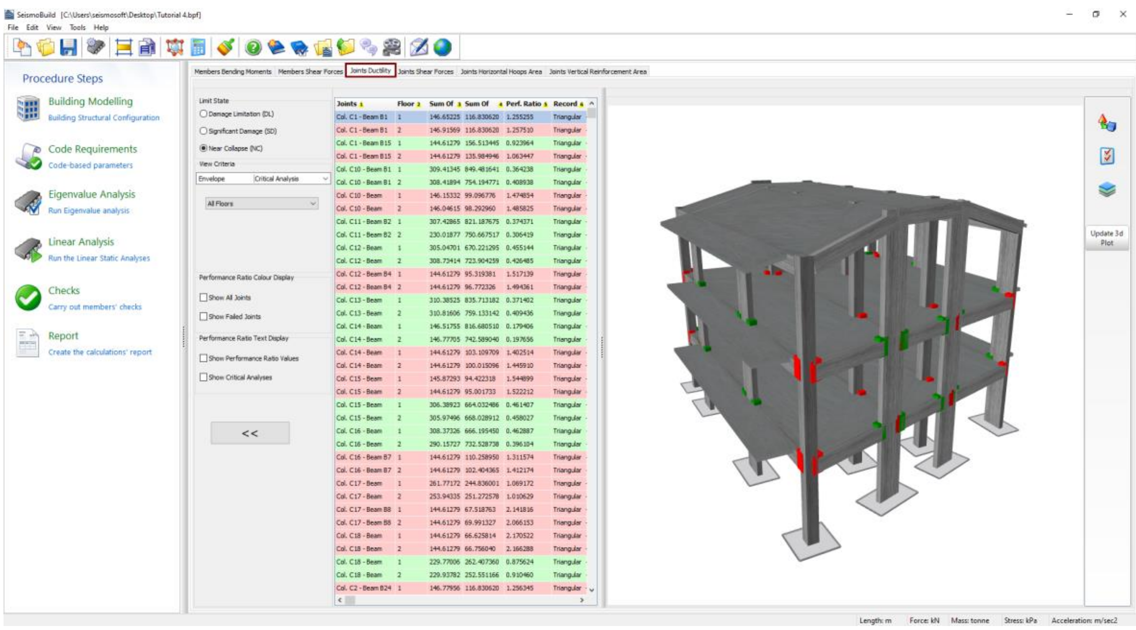
Analysis	Demanded Area	Provided Area	Perf. Ratio
Envelope	0.001204	0.001257	0.958333
Uniform + X	0.001204	0.001257	0.958333
Uniform - X	0.001204	0.001257	0.958333
Uniform + Y	0.001204	0.001257	0.958333
Uniform - Y	0.001204	0.001257	0.958333
Modal + X	0.001204	0.001257	0.958333
Modal - X	0.001204	0.001257	0.958333
Modal + Y	0.001204	0.001257	0.958333
Modal - Y	0.001204	0.001257	0.958333

Joint Checks module

A more detailed description on the joints' vertical reinforcement area checks and the equations used in SeismoBuild is available in the Joints Vertical Reinforcement Area section of the Appendix A.1-EUROCODES.

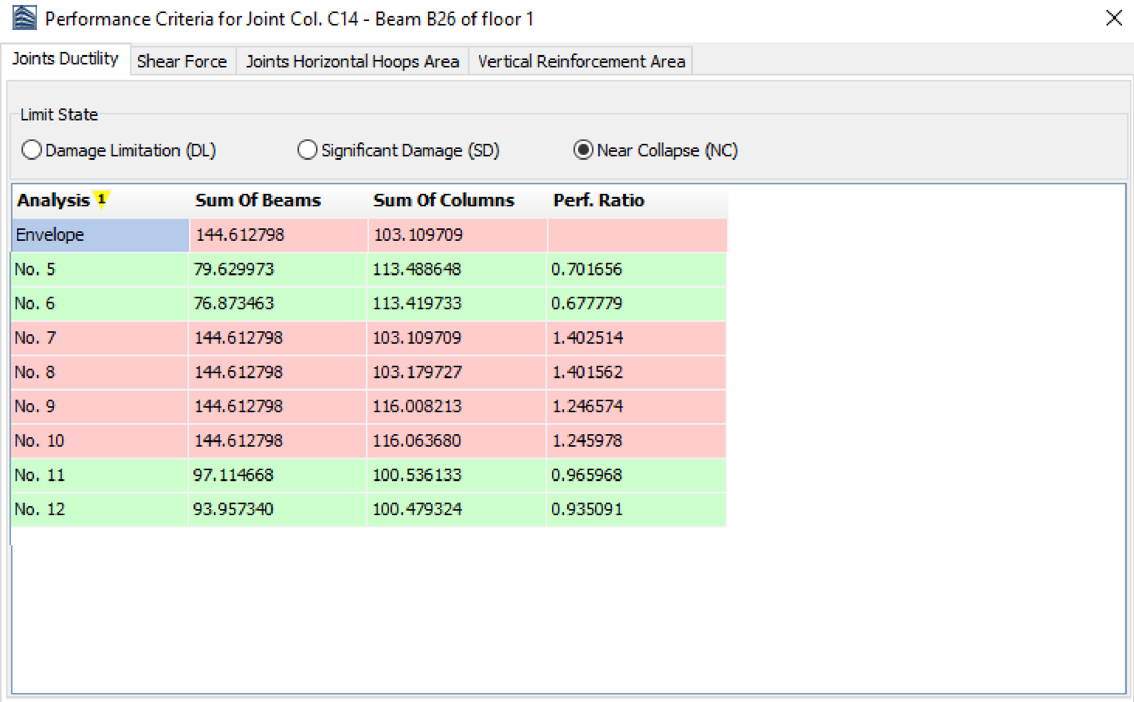
Joints Ductility

Herein the results of the ductility checks for beam–column joints are presented. Users may select for which Limit State to view the results and choose filters, which include the determination of the analysis and the floor. The available limit states are those already selected in the Limit States module in Code Requirements. In addition, in the drop-down menu for the analysis selection apart from the executed analyses there is an option called Critical Analysis, which is the envelope of the results of all the analyses. The elements that reach their joints ductility capacity are marked with red both in the table view and the 3D view format, whereas different visualisation options are available. For instance, users may select to display the joints' performance ratios or each joint's critical analysis on the 3D plot, or to display the joints with different colours, depending on the value of their performance ratio for the selected limit state.



Joints Ductility Module

It is possible to hide the data entry table through the corresponding button in order to view the 3D rendering of the structural model in 'full-screen' modality. By right clicking on an element and selecting the View All Joints Checks, a window appears with the checks for all the analyses for that particular element.



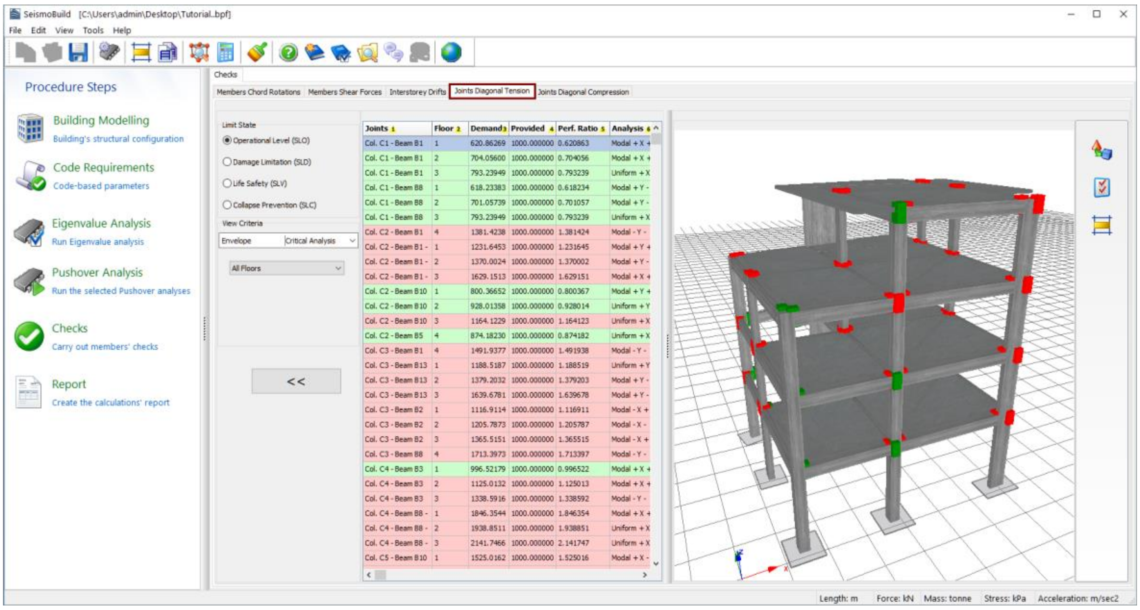
Analysis	Sum Of Beams	Sum Of Columns	Perf. Ratio
Envelope	144.612798	103.109709	
No. 5	79.629973	113.488648	0.701656
No. 6	76.873463	113.419733	0.677779
No. 7	144.612798	103.109709	1.402514
No. 8	144.612798	103.179727	1.401562
No. 9	144.612798	116.008213	1.246574
No. 10	144.612798	116.063680	1.245978
No. 11	97.114668	100.536133	0.965968
No. 12	93.957340	100.479324	0.935091

Joint Checks module

A more detailed description on the joints ductility checks and the equations used in SeismoBuild is available in the Joints Ductility section of the Appendix A.1-EUROCODES.

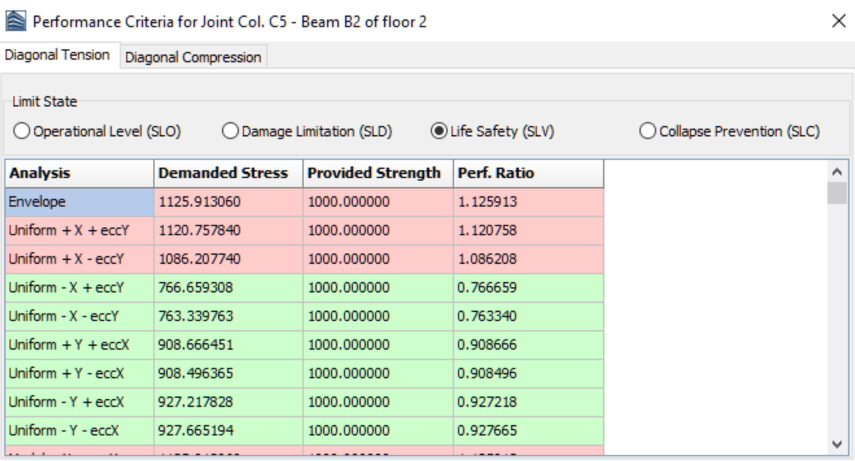
JOINTS DIAGONAL TENSION (NTC & KANEPE)

Herein the results of the diagonal tension checks for beam–column joints are presented (this check applies to NTC and KANEPE). Users may select for which Limit State to view the results as well as the View criteria, which include the determination of the analysis and the floor. The available limit states are those already selected in the Limit States module in Code Requirements. In addition, in the drop-down menu for the analysis selection apart from the executed analyses there is an option called Critical Analysis, which is the envelope of the results of all the analyses. The elements that have exceeded their diagonal tensile stress capacity are marked with red both in the table view and the 3D view format, whereas different visualisation options are available. For instance, users may select to display the joints' performance ratios or each joint's critical analysis on the 3D plot, or to display the joints with different colours, depending on the value of their performance ratio for the selected limit state.



Joints Diagonal Tension Module

It is possible to hide the data entry table through the corresponding button in order to view the 3D rendering of the structural model in 'full-screen' modality. By right clicking on an element and selecting the View All Joints Checks, a window appears with the checks for all the analyses for that particular element.



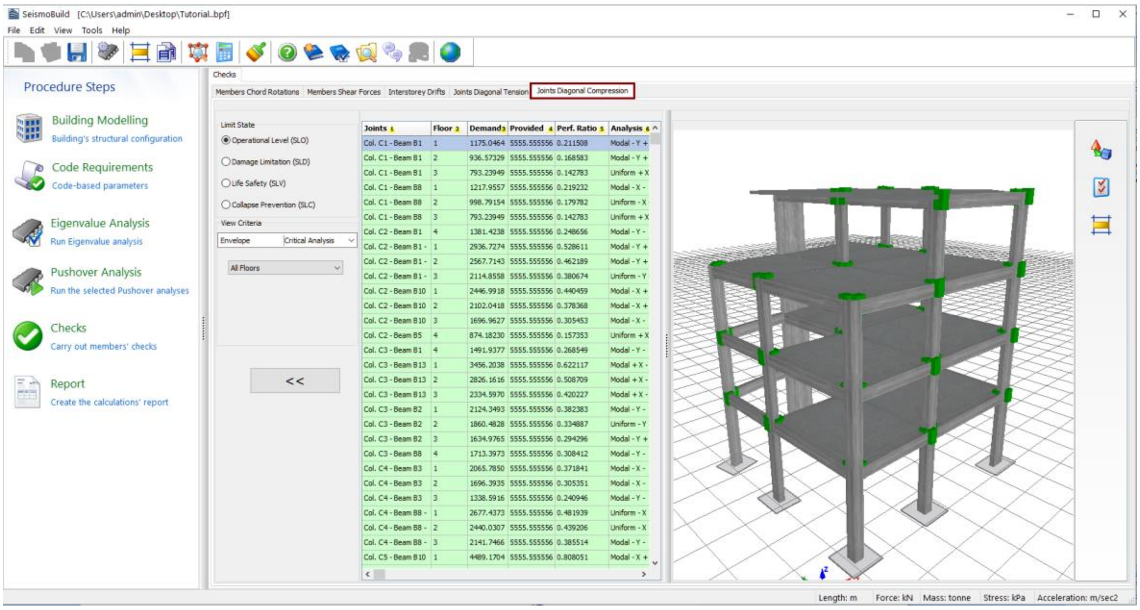
Analysis	Demanded Stress	Provided Strength	Perf. Ratio
Envelope	1125.913060	1000.000000	1.125913
Uniform + X + eccY	1120.757840	1000.000000	1.120758
Uniform + X - eccY	1086.207740	1000.000000	1.086208
Uniform - X + eccY	766.659308	1000.000000	0.766659
Uniform - X - eccY	763.339763	1000.000000	0.763340
Uniform + Y + eccX	908.666451	1000.000000	0.908666
Uniform + Y - eccX	908.496365	1000.000000	0.908496
Uniform - Y + eccX	927.217828	1000.000000	0.927218
Uniform - Y - eccX	927.665194	1000.000000	0.927665

Joint Checks module

A more detailed description on the joints diagonal tension checks and the equations used in SeismoBuild is available in the Joints Diagonal Tension section of the respective appendix.

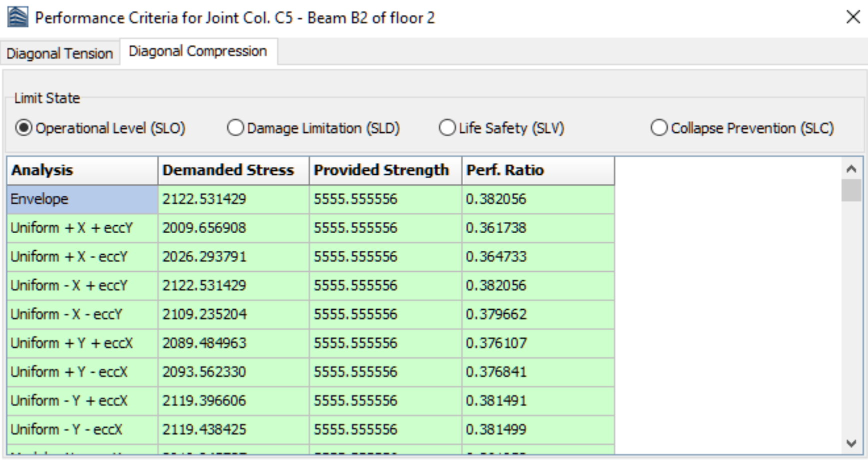
JOINTS DIAGONAL COMPRESSION (NTC & KANEPE)

Herein the results of the diagonal compression checks for beam–column joints are presented (this check applies to NTC and KANEPE). Users may select for which Limit State to view the results as well as the View criteria, which include the determination of the analysis and the floor. The available limit states are those already selected in the Limit States module in Code Requirements. In addition, in the drop-down menu for the analysis selection apart from the executed analyses there is an option called Critical Analysis, which is the envelope of the results of all the analyses. The elements that have exceeded their diagonal compressive stress capacity are marked with red both in the table view and the 3D view format, whereas different visualisation options are available. For instance, users may select to display the joints' performance ratios or each joint's critical analysis on the 3D plot, or to display the joints with different colours, depending on the value of their performance ratio for the selected limit state.



Joints Diagonal Compression Module

It is possible to hide the data entry table through the corresponding button in order to view the 3D rendering of the structural model in 'full-screen' modality. By right clicking on an element and selecting the View All Joints Checks, a window appears with the checks for all the analyses for that particular element.

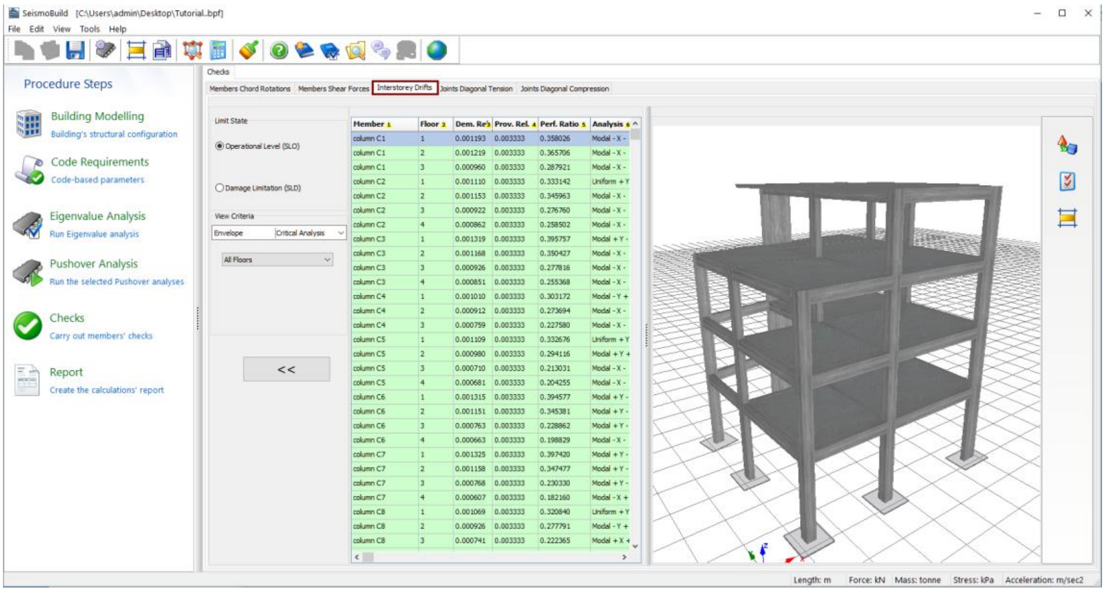


Joint Checks module

A more detailed description on the joints' diagonal compression checks and the equations used in SeismoBuild is available in the Joints Diagonal Compression section of the respective appendix

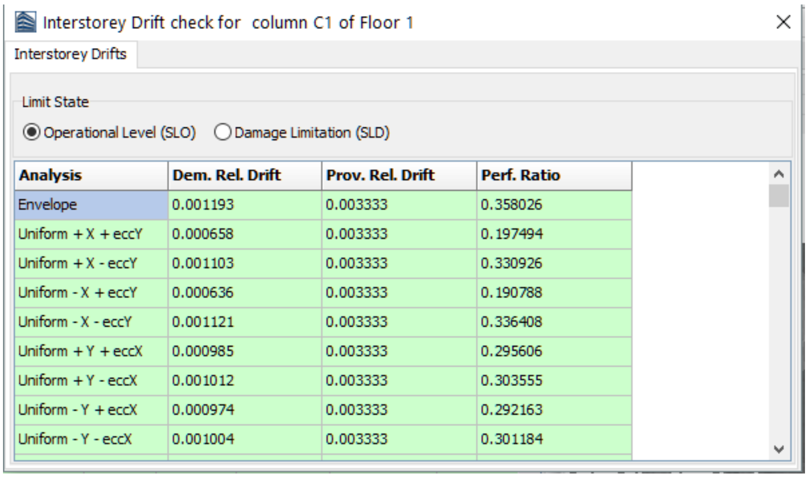
INTERSTOREY DRIFTS (ASCE 41-23 & NTC)

Herein the results of the interstorey drifts checks for columns and walls are presented (this check applies to ASCE 41-23 and NTC). Users may select for which Limit State to view the results, as well as the View criteria, which include the determination of the analysis and the floor. The available limit states are those already selected in the Limit States module in Code Requirements. It is noted that only the walls controlled by shear are checked according to the ASCE 41-23 and only the limit states of Damage Limitation and Operational Level for columns and walls are checked according to the NTC. In addition, in the drop-down menu for the analysis selection apart from the executed analyses there is an option called Critical Analysis, which is the envelope of the results of all the analyses. The elements that have exceeded the Code defined relative drift are marked with red both in the table view and the 3D view format, whereas different visualisation options are available. For instance, users may select to display the members' performance ratios or each member's critical analysis on the 3D plot, or to display the elements with different colours, depending on the value of their performance ratio for the selected limit state.



Interstorey Drifts Module

It is possible to hide the data entry table through the corresponding button in order to view the 3D rendering of the structural model in 'full-screen' modality. By right clicking on an element and selecting the View Member Checks, a window appears with the checks for all the analyses for that particular element.

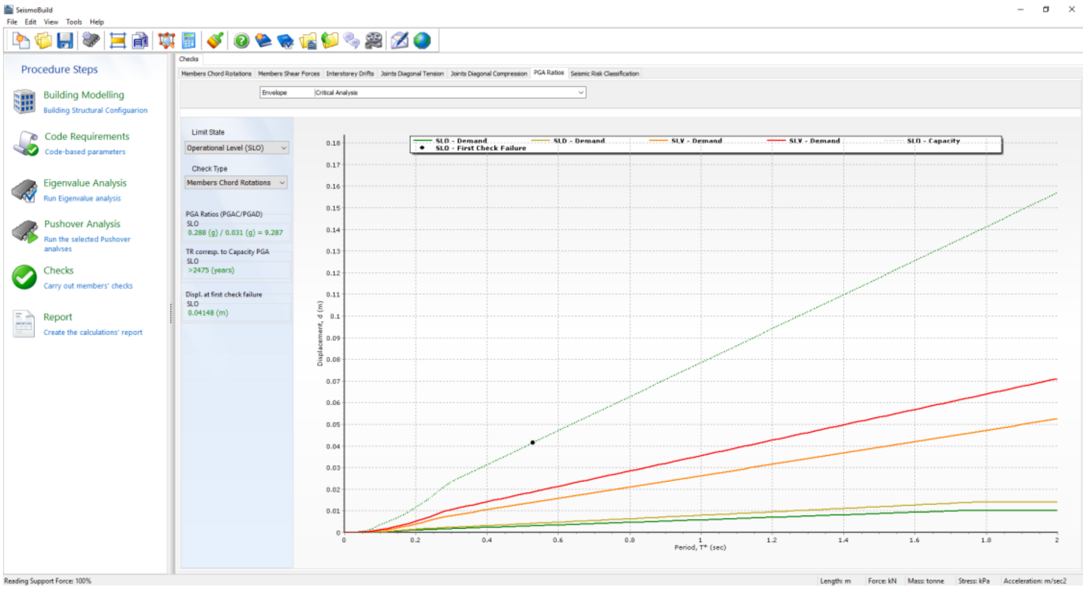


Interstorey Drift Check module

A more detailed description on the interstorey drift checks and the equations used in SeismoBuild is available in the section of the respective appendix.

PGA RATIOS (NTC ONLY)

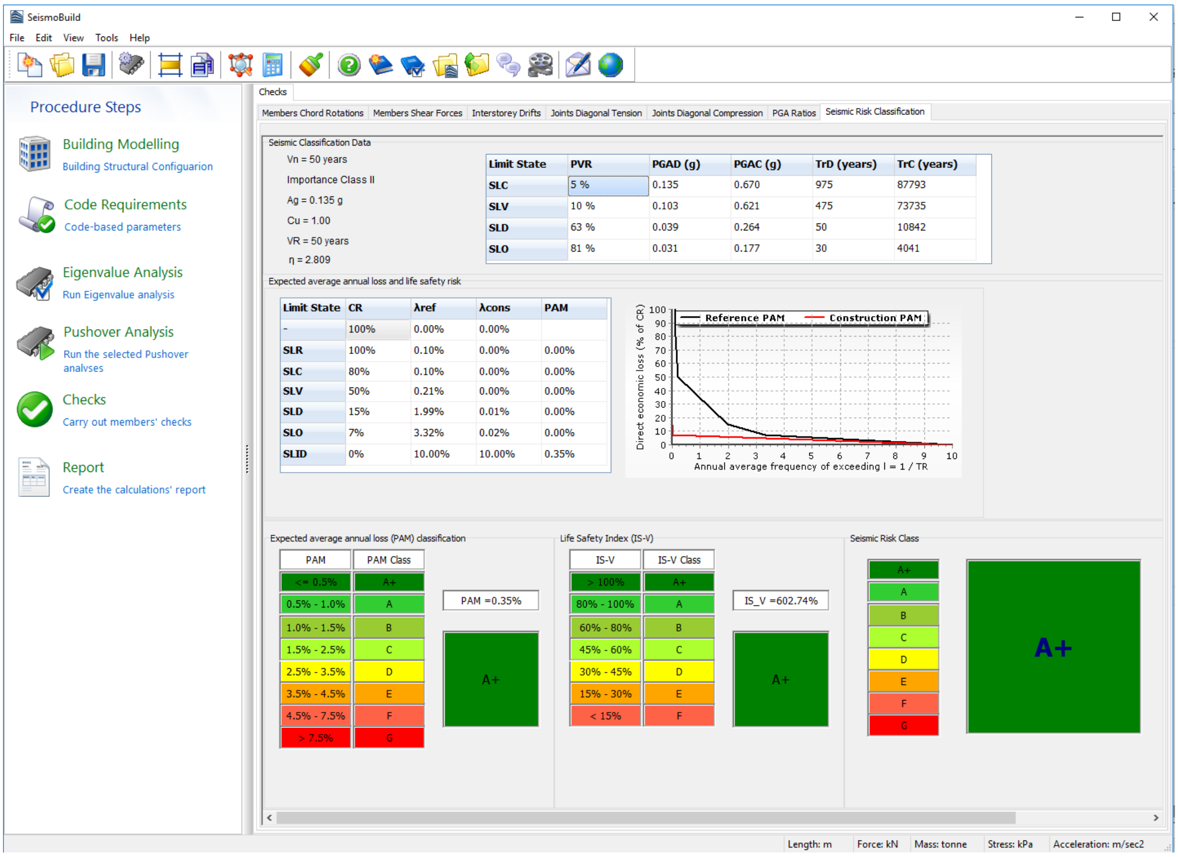
PGA ratios (available only in NTC) provide general aspect of the vulnerability of the structure in terms of peak ground accelerations that a structure can sustain; hence it is a measure of seismic risk. It is defined as the ratio of the PGA Capacity, which corresponds to the displacement on the Capacity Curve, at which the first failure (at any member of the structure) occurs, divided by the PGA Demand, which corresponds to Spectral acceleration specified by the user in the Seismic Action tab. The PGA ratio is calculated for each limit state, check type and pushover analysis. Values larger than unity (displayed in green color) indicate no damages for the selected combination of limit state, check type and analysis, while in case of values smaller than unity (in red color) damage is expected. Values significantly larger than unity indicate a structure, which is able to resist seismic loads quite larger than seismic action required by Code, whereas values close to zero denote a structure with capacity significantly poorer than the capacity required by Code. In the PGA ratios tab the demand and capacity displacement response spectra are also displayed graphically, and the displacement at the first check failure is denoted. The return period that corresponds to the PGA capacity is shown as well.



PGA ratios Module

SEISMIC RISK CLASSIFICATION (NTC ONLY)

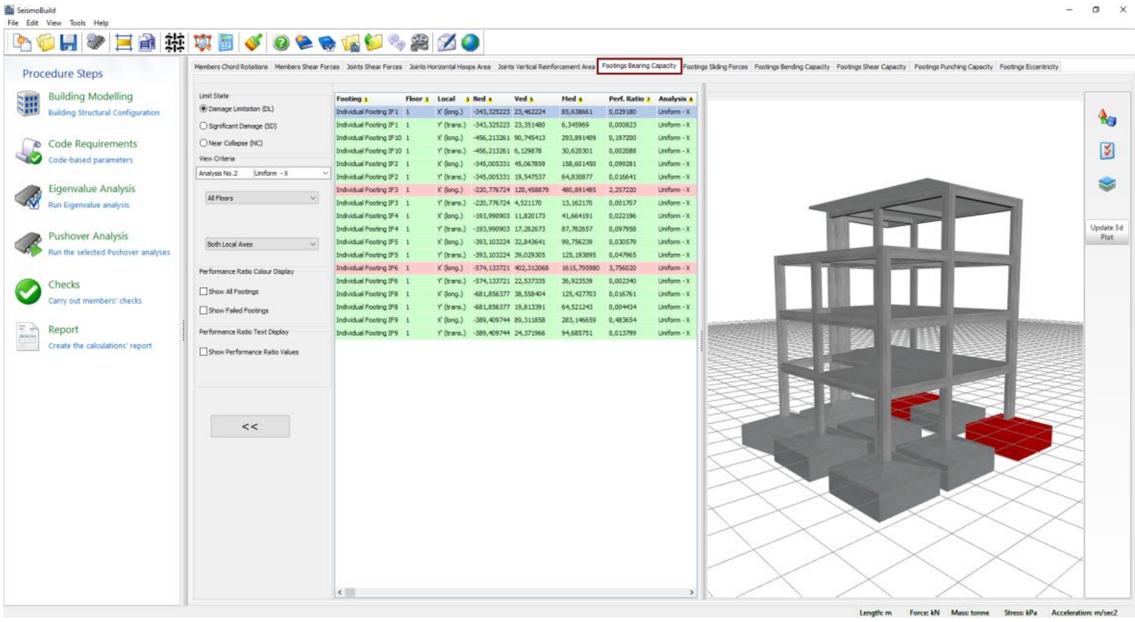
Seismic Risk Classification (specified only in NTC) is an index corresponding to a scale of the seismic hazard, which a building is exposed to, ranging from G (highest seismic risk) to A+ (lowest seismic risk). It is defined as the worst of two partial indices, the expected average annual loss (PAM) and the Life Safety Index (IS-V). For the calculation of the aforementioned indices the PGA ratios for the limit states of Life Safety and Damage Limitation are required, so these limit states should be selected in the Code Requirements/Limit States Module. Selection of the Limit states of Operational Level and Collapse Prevention is not compulsory, however, if these are selected, the procedure will result in a more accurate classification. The lowest calculated values of the PGA ratios for all check type and pushover analyses are considered in the determination of the Seismic Risk Classification. Information about these parameters can be found in the Italian Regulations.



Seismic Risk Classification Module

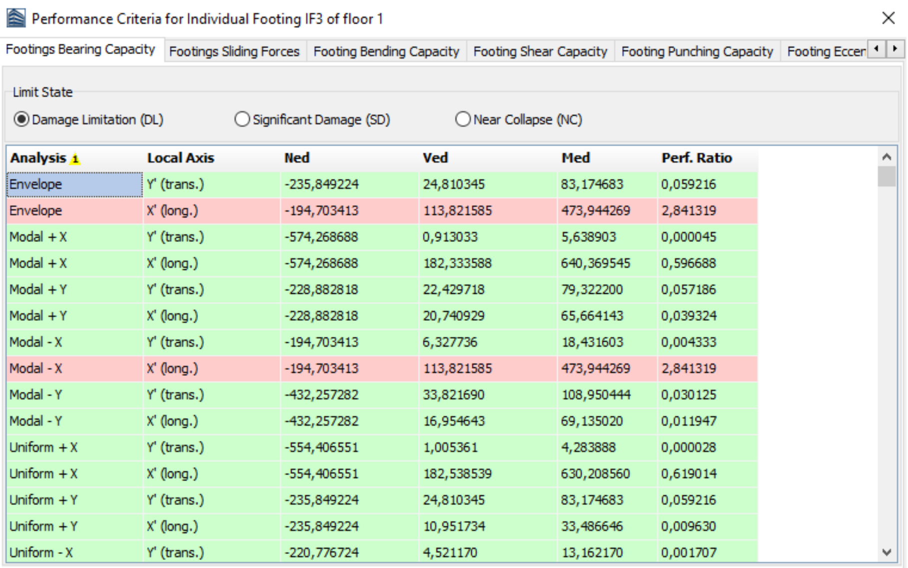
Footings Bearing Capacity (Eurocode 8, NTC & KANEPE)

The results of the bearing capacity checks for individual and strip footings, according to the selected Code, can be visualised in this module. Users may select for which Limit State, or Performance objective, to view the results and choose filters, which include the determination of the analysis, the floor and the local axis. The available limit states are those already selected in the Limit States module in Code Requirements. In addition, in the drop-down menu for the analysis selection apart from the executed analyses there is an option called Critical Analysis, which is the envelope of the results of all the analyses. The elements that have exceeded their bearing capacity are marked with red both in the table view and the 3D view format, whereas different visualisation options are available. For instance, users may select to display the footings' performance ratios or each footing's critical analysis on the 3D plot, or to display the footings with different colours, depending on the value of their performance ratio for the selected limit state.



Footings Bearing Capacity Module

It is possible to hide the data entry table through the corresponding button in order to view the 3D rendering of the structural model in 'full-screen' modality. By right clicking on a footing and selecting the View All Footing Checks, a window appears with the checks for all the analyses for that particular footing.



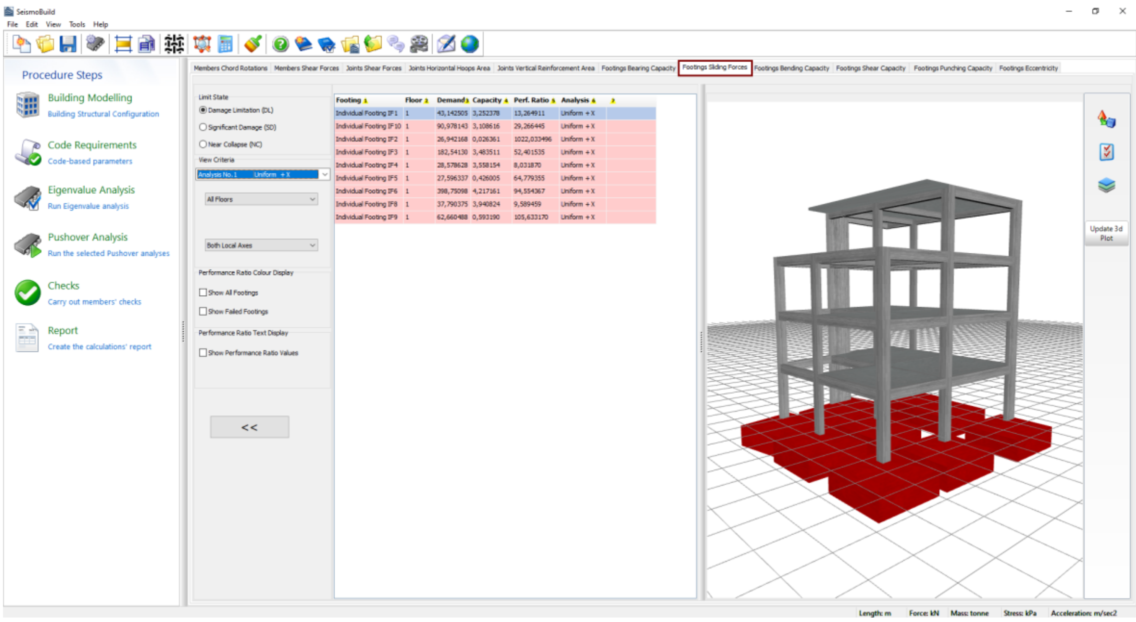
Analysis	Local Axis	Ned	Ved	Med	Perf. Ratio
Envelope	Y' (trans.)	-235,849224	24,810345	83,174683	0,059216
Envelope	X' (long.)	-194,703413	113,821585	473,944269	2,841319
Modal +X	Y' (trans.)	-574,268688	0,913033	5,638903	0,000045
Modal +X	X' (long.)	-574,268688	182,333588	640,369545	0,596688
Modal +Y	Y' (trans.)	-228,882818	22,429718	79,322200	0,057186
Modal +Y	X' (long.)	-228,882818	20,740929	65,664143	0,039324
Modal -X	Y' (trans.)	-194,703413	6,327736	18,431603	0,004333
Modal -X	X' (long.)	-194,703413	113,821585	473,944269	2,841319
Modal -Y	Y' (trans.)	-432,257282	33,821690	108,950444	0,030125
Modal -Y	X' (long.)	-432,257282	16,954643	69,135020	0,011947
Uniform +X	Y' (trans.)	-554,406551	1,005361	4,283888	0,000028
Uniform +X	X' (long.)	-554,406551	182,538539	630,208560	0,619014
Uniform +Y	Y' (trans.)	-235,849224	24,810345	83,174683	0,059216
Uniform +Y	X' (long.)	-235,849224	10,951734	33,486646	0,009630
Uniform -X	Y' (trans.)	-220,776724	4,521170	13,162170	0,001707

Footing Checks module

A more detailed description on the footings bearing capacity checks and the equations used in SeismoBuild is available in the Footings Bearing Capacity section of the respective appendix.

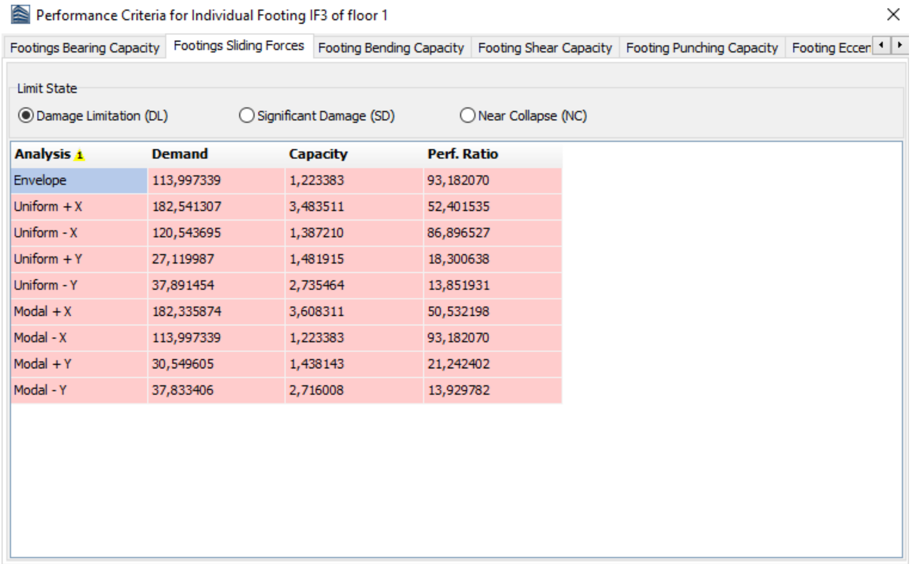
Footings Sliding Forces (Eurocode 8, NTC & KANEPE)

The results of the sliding forces checks for individual and strip footings, according to the selected Code, can be visualised in this module. Users may select for which Limit State, or Performance objective, to view the results and choose filters, which include the determination of the analysis, the floor and the local axis. The available limit states are those already selected in the Limit States module in Code Requirements. In addition, in the drop-down menu for the analysis selection apart from the executed analyses there is an option called Critical Analysis, which is the envelope of the results of all the analyses. The elements with sliding failure are marked with red both in the table view and the 3D view format, whereas different visualisation options are available. For instance, users may select to display the footings' performance ratios or each footing's critical analysis on the 3D plot, or to display the footings with different colours, depending on the value of their performance ratio for the selected limit state.



Footings Sliding Forces Module

It is possible to hide the data entry table through the corresponding button in order to view the 3D rendering of the structural model in 'full-screen' modality. By right clicking on a footing and selecting the View All Footing Checks, a window appears with the checks for all the analyses for that particular footing.



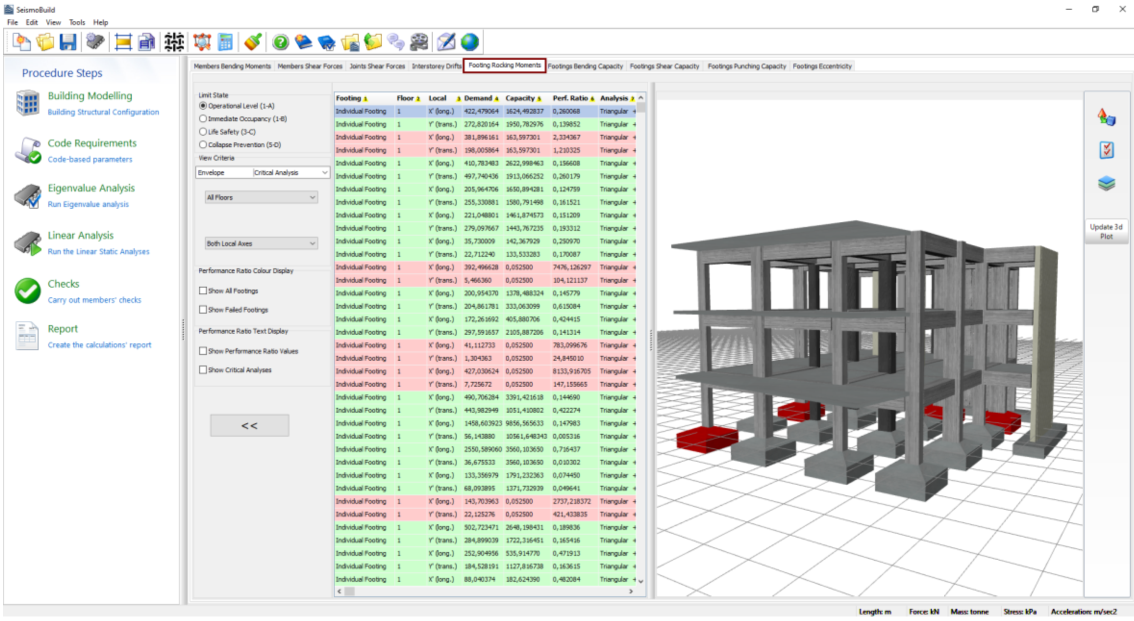
Analysis	Demand	Capacity	Perf. Ratio
Envelope	113,997339	1,223383	93,182070
Uniform +X	182,541307	3,483511	52,401535
Uniform -X	120,543695	1,387210	86,896527
Uniform +Y	27,119987	1,481915	18,300638
Uniform -Y	37,891454	2,735464	13,851931
Modal +X	182,335874	3,608311	50,532198
Modal -X	113,997339	1,223383	93,182070
Modal +Y	30,549605	1,438143	21,242402
Modal -Y	37,833406	2,716008	13,929782

Footing Checks module

A more detailed description on the footings sliding capacity checks and the equations used in SeismoBuild is available in the Footings Sliding forces section of the respective appendix.

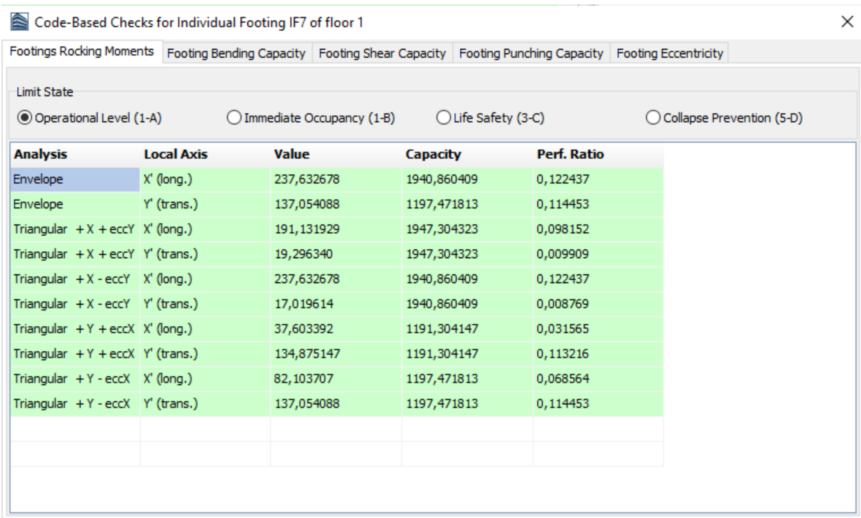
Footings Rocking Moment Capacity (ASCE 41-23 & TBDY)

The results of the rocking moment capacity checks for individual and strip footings, according to the selected Code, can be visualised in this module. Users may select for which Limit State, or Performance objective, to view the results and choose filters, which include the determination of the analysis, the floor and the local axis. The available limit states are those already selected in the Limit States module in Code Requirements. In addition, in the drop-down menu for the analysis selection apart from the executed analyses there is an option called Critical Analysis, which is the envelope of the results of all the analyses. The elements that have exceeded their rocking moment capacity are marked with red both in the table view and the 3D view format, whereas different visualisation options are available. For instance, users may select to display the footings' performance ratios or each footing's critical analysis on the 3D plot, or to display the footings with different colours, depending on the value of their performance ratio for the selected limit state.



Footings Rocking Moment Capacity Module

It is possible to hide the data entry table through the corresponding button in order to view the 3D rendering of the structural model in 'full-screen' modality. By right clicking on a footing and selecting the View All Footing Checks, a window appears with the checks for all the analyses for that particular footing.



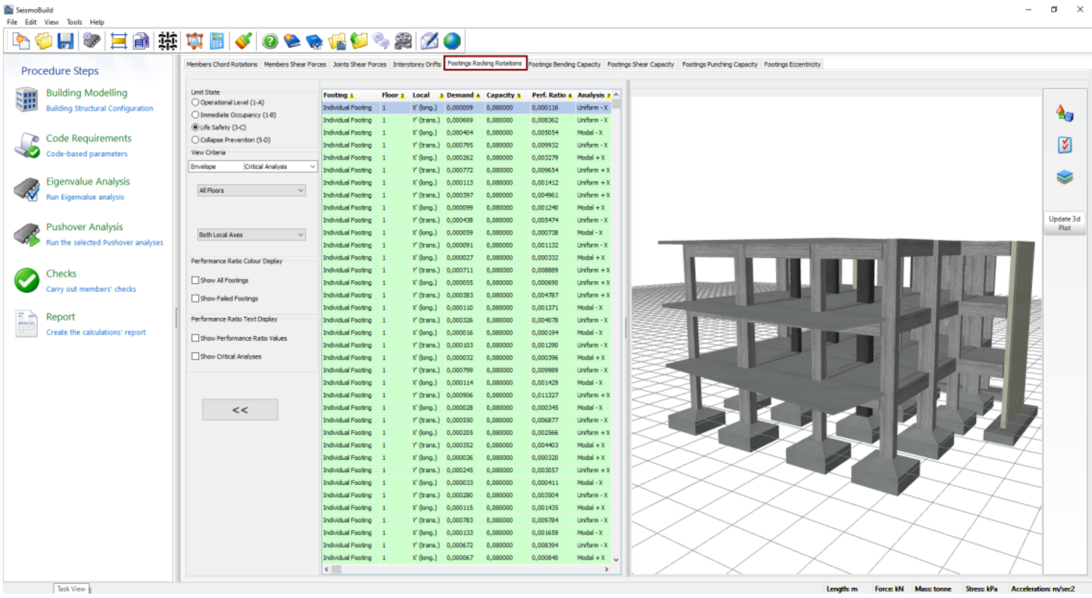
Analysis	Local Axis	Value	Capacity	Perf. Ratio
Envelope	X' (long.)	237,632678	1940,860409	0,122437
Envelope	Y' (trans.)	137,054088	1197,471813	0,114453
Triangular +X +eccY	X' (long.)	191,131929	1947,304323	0,098152
Triangular +X +eccY	Y' (trans.)	19,296340	1947,304323	0,009909
Triangular +X -eccY	X' (long.)	237,632678	1940,860409	0,122437
Triangular +X -eccY	Y' (trans.)	17,019614	1940,860409	0,008769
Triangular +Y +eccX	X' (long.)	37,603392	1191,304147	0,031565
Triangular +Y +eccX	Y' (trans.)	134,875147	1191,304147	0,113216
Triangular +Y -eccX	X' (long.)	82,103707	1197,471813	0,068564
Triangular +Y -eccX	Y' (trans.)	137,054088	1197,471813	0,114453

Footing Checks module

A more detailed description on the footings rocking moment capacity checks and the equations used in SeismoBuild is available in the Footings Rocking Moment Capacity section of the respective appendix.

Footings Rocking Rotation Capacity (ASCE 41-23 & TBDY)

The results of the rocking rotation capacity checks for individual and strip footings, according to the selected Code, can be visualised in this module. Users may select for which Limit State, or Performance objective, to view the results and choose filters, which include the determination of the analysis, the floor and the local axis. The available limit states are those already selected in the Limit States module in Code Requirements. In addition, in the drop-down menu for the analysis selection apart from the executed analyses there is an option called Critical Analysis, which is the envelope of the results of all the analyses. The elements that have exceeded their rocking rotation capacity are marked with red both in the table view and the 3D view format, whereas different visualisation options are available. For instance, users may select to display the footings' performance ratios or each footing's critical analysis on the 3D plot, or to display the footings with different colours, depending on the value of their performance ratio for the selected limit state.



Footings Rocking Rotation Capacity Module

It is possible to hide the data entry table through the corresponding button in order to view the 3D rendering of the structural model in 'full-screen' modality. By right clicking on a footing and selecting the View All Footing Checks, a window appears with the checks for all the analyses for that particular footing.

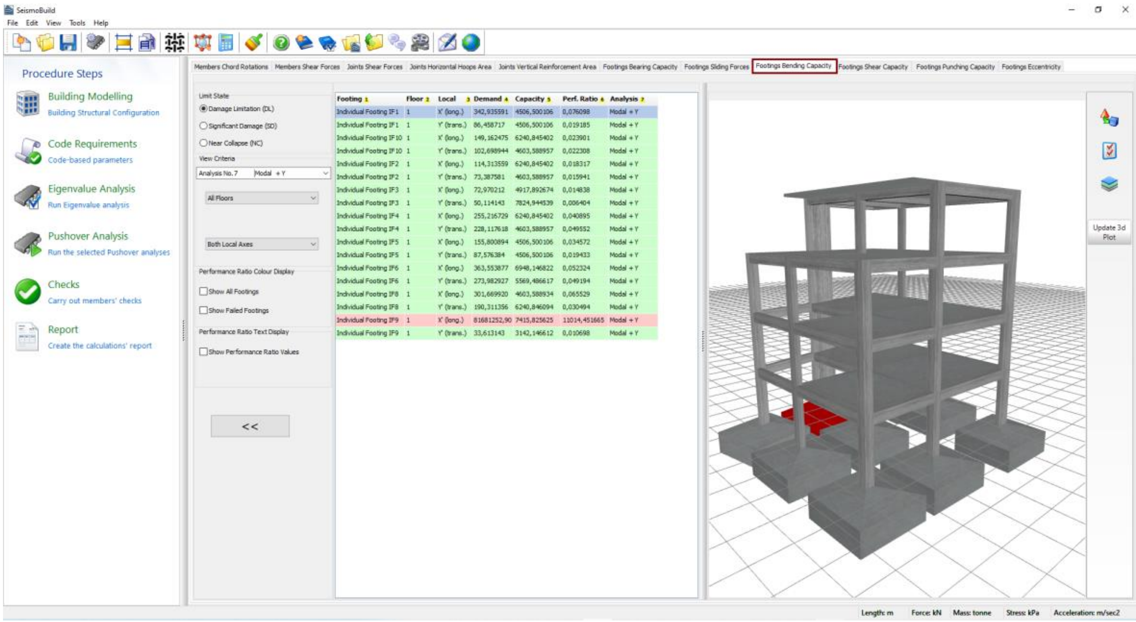
Analysis	Local Axis	Demand	Capacity	Perf. Ratio
Envelope	X' (long.)	0,000115	0,080000	0,001435
Envelope	Y' (trans.)	0,000783	0,080000	0,009784
Uniform + X	X' (long.)	0,000093	0,080000	0,001166
Uniform + X	Y' (trans.)	0,000772	0,080000	0,009649
Uniform - X	X' (long.)	0,000063	0,080000	0,000790
Uniform - X	Y' (trans.)	0,000783	0,080000	0,009784
Modal + X	X' (long.)	0,000115	0,080000	0,001435
Modal + X	Y' (trans.)	0,000742	0,080000	0,009272
Modal - X	X' (long.)	0,000078	0,080000	0,000980
Modal - X	Y' (trans.)	0,000734	0,080000	0,009178

Footing Checks module

A more detailed description on the footings rocking rotation capacity checks and the equations used in SeismoBuild is available in the Footings Rocking Rotation Capacity section of the respective appendix.

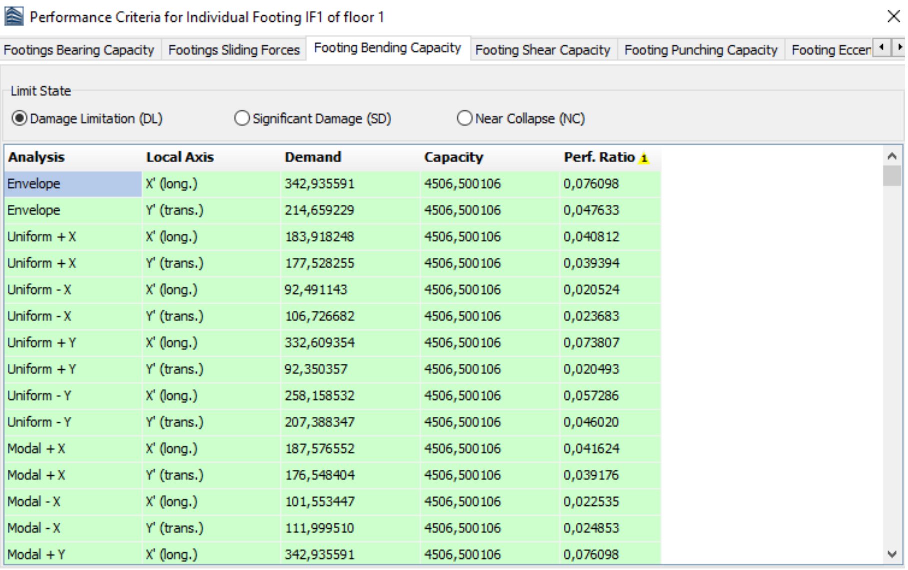
FOOTINGS BENDING CAPACITY

The results of the bending capacity checks for individual and strip footings, according to the selected Code, can be visualised in this module. Users may select for which Limit State, or Performance objective, to view the results and choose filters, which include the determination of the analysis, the floor and the local axis. The available limit states are those already selected in the Limit States module in Code Requirements. In addition, in the drop-down menu for the analysis selection apart from the executed analyses there is an option called Critical Analysis, which is the envelope of the results of all the analyses. The elements that have exceeded their bending capacity are marked with red both in the table view and the 3D view format, whereas different visualisation options are available. For instance, users may select to display the footings' performance ratios or each footing's critical analysis on the 3D plot, or to display the footings with different colours, depending on the value of their performance ratio for the selected limit state.



Footings Bending Capacity Module

It is possible to hide the data entry table through the corresponding button in order to view the 3D rendering of the structural model in 'full-screen' modality. By right clicking on a footing and selecting the View All Footing Checks, a window appears with the checks for all the analyses for that particular footing.



Performance Criteria for Individual Footing IF1 of floor 1

Footings Bearing Capacity | Footings Sliding Forces | Footing Bending Capacity | Footing Shear Capacity | Footing Punching Capacity | Footing Eccen

Limit State

(•) Damage Limitation (DL) ( ) Significant Damage (SD) ( ) Near Collapse (NC)

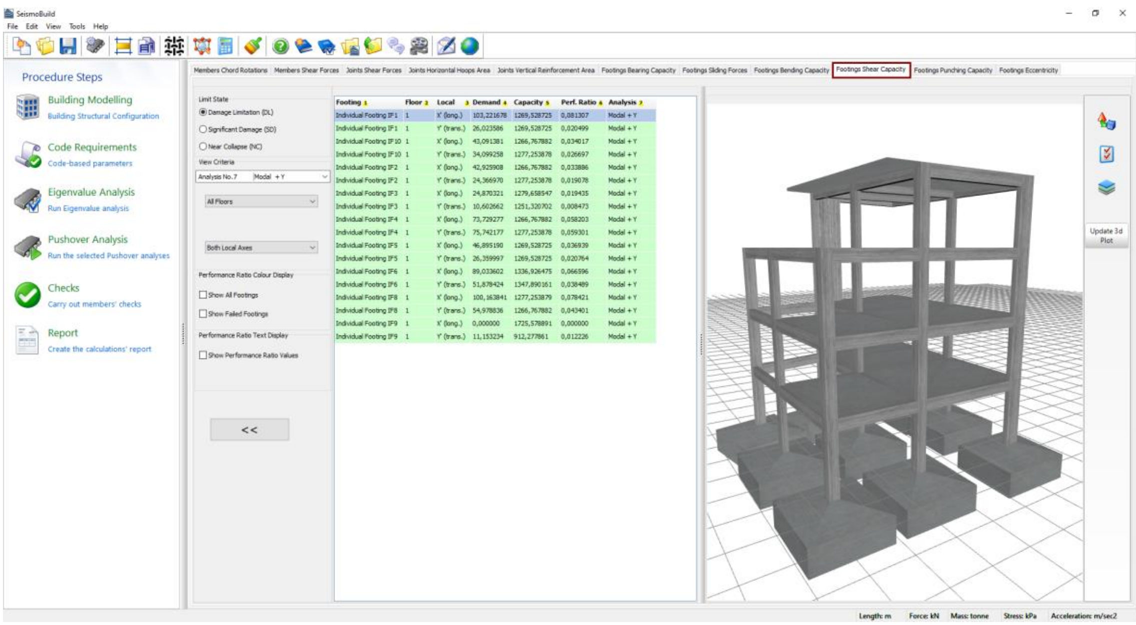
Analysis	Local Axis	Demand	Capacity	Perf. Ratio
Envelope	X' (long.)	342,935591	4506,500106	0,076098
Envelope	Y' (trans.)	214,659229	4506,500106	0,047633
Uniform +X	X' (long.)	183,918248	4506,500106	0,040812
Uniform +X	Y' (trans.)	177,528255	4506,500106	0,039394
Uniform -X	X' (long.)	92,491143	4506,500106	0,020524
Uniform -X	Y' (trans.)	106,726682	4506,500106	0,023683
Uniform +Y	X' (long.)	332,609354	4506,500106	0,073807
Uniform +Y	Y' (trans.)	92,350357	4506,500106	0,020493
Uniform -Y	X' (long.)	258,158532	4506,500106	0,057286
Uniform -Y	Y' (trans.)	207,388347	4506,500106	0,046020
Modal +X	X' (long.)	187,576552	4506,500106	0,041624
Modal +X	Y' (trans.)	176,548404	4506,500106	0,039176
Modal -X	X' (long.)	101,553447	4506,500106	0,022535
Modal -X	Y' (trans.)	111,999510	4506,500106	0,024853
Modal +Y	X' (long.)	342,935591	4506,500106	0,076098

Footing Checks module

A more detailed description on the footings bending capacity checks and the equations used in SeismoBuild is available in the Footings Bending Capacity section of the respective appendix.

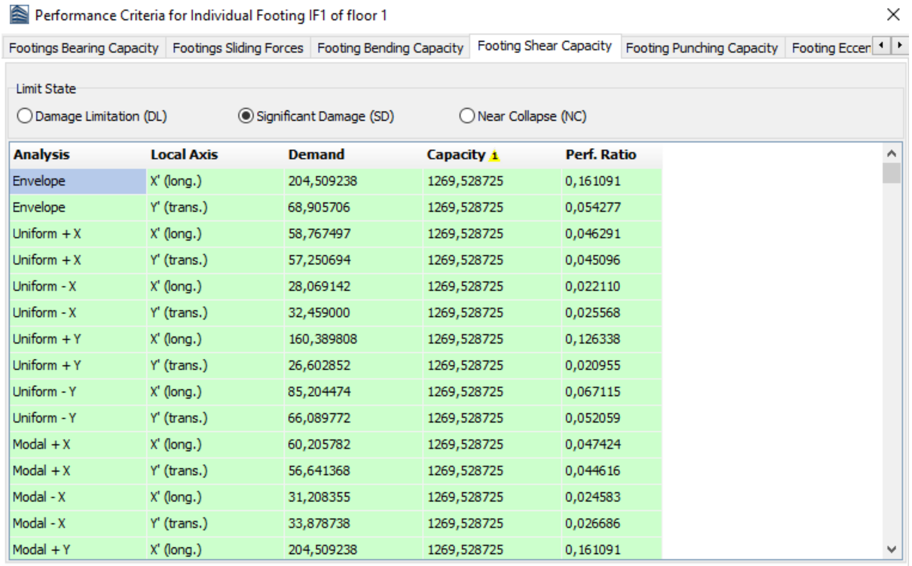
FOOTINGS SHEAR CAPACITY

The results of the shear capacity checks for individual and strip footings, according to the selected Code, can be visualised in this module. Users may select for which Limit State, or Performance objective, to view the results and choose filters, which include the determination of the analysis, the floor and the local axis. The available limit states are those already selected in the Limit States module in Code Requirements. In addition, in the drop-down menu for the analysis selection apart from the executed analyses there is an option called Critical Analysis, which is the envelope of the results of all the analyses. The elements that have exceeded their shear capacity are marked with red both in the table view and the 3D view format, whereas different visualisation options are available. For instance, users may select to display the footings' performance ratios or each footing's critical analysis on the 3D plot, or to display the footings with different colours, depending on the value of their performance ratio for the selected limit state.



Footings Shear Capacity Module

It is possible to hide the data entry table through the corresponding button in order to view the 3D rendering of the structural model in 'full-screen' modality. By right clicking on a footing and selecting the View All Footing Checks, a window appears with the checks for all the analyses for that particular footing.



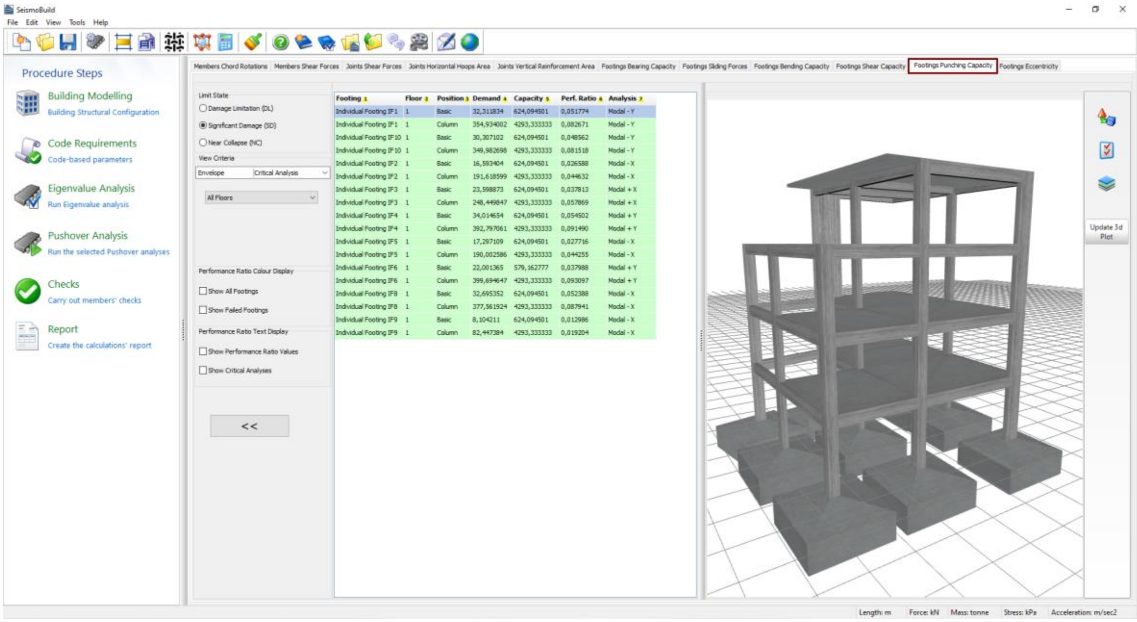
Analysis	Local Axis	Demand	Capacity	Perf. Ratio
Envelope	X' (long.)	204,509238	1269,528725	0,161091
Envelope	Y' (trans.)	68,905706	1269,528725	0,054277
Uniform +X	X' (long.)	58,767497	1269,528725	0,046291
Uniform +X	Y' (trans.)	57,250694	1269,528725	0,045096
Uniform -X	X' (long.)	28,069142	1269,528725	0,022110
Uniform -X	Y' (trans.)	32,459000	1269,528725	0,025568
Uniform +Y	X' (long.)	160,389808	1269,528725	0,126338
Uniform +Y	Y' (trans.)	26,602852	1269,528725	0,020955
Uniform -Y	X' (long.)	85,204474	1269,528725	0,067115
Uniform -Y	Y' (trans.)	66,089772	1269,528725	0,052059
Modal +X	X' (long.)	60,205782	1269,528725	0,047424
Modal +X	Y' (trans.)	56,641368	1269,528725	0,044616
Modal -X	X' (long.)	31,208355	1269,528725	0,024583
Modal -X	Y' (trans.)	33,878738	1269,528725	0,026686
Modal +Y	X' (long.)	204,509238	1269,528725	0,161091

Footing Checks module

A more detailed description on the footings shear capacity checks and the equations used in SeismoBuild is available in the Footings Shear Capacity section of the respective appendix.

Footings Punching Capacity

The results of the punching capacity checks for individual and strip footings, according to the selected Code, can be visualised in this module. Users may select for which Limit State, or Performance objective, to view the results and choose filters, which include the determination of the analysis, the floor and the local axis. The available limit states are those already selected in the Limit States module in Code Requirements. In addition, in the drop-down menu for the analysis selection apart from the executed analyses there is an option called Critical Analysis, which is the envelope of the results of all the analyses. The elements that have exceeded their punching capacity are marked with red both in the table view and the 3D view format, whereas different visualisation options are available. For instance, users may select to display the footings' performance ratios or each footing's critical analysis on the 3D plot, or to display the footings with different colours, depending on the value of their performance ratio for the selected limit state.



Footings Punching Capacity Module

It is possible to hide the data entry table through the corresponding button in order to view the 3D rendering of the structural model in 'full-screen' modality. By right clicking on a footing and selecting the View All Footing Checks, a window appears with the checks for all the analyses for that particular footing.

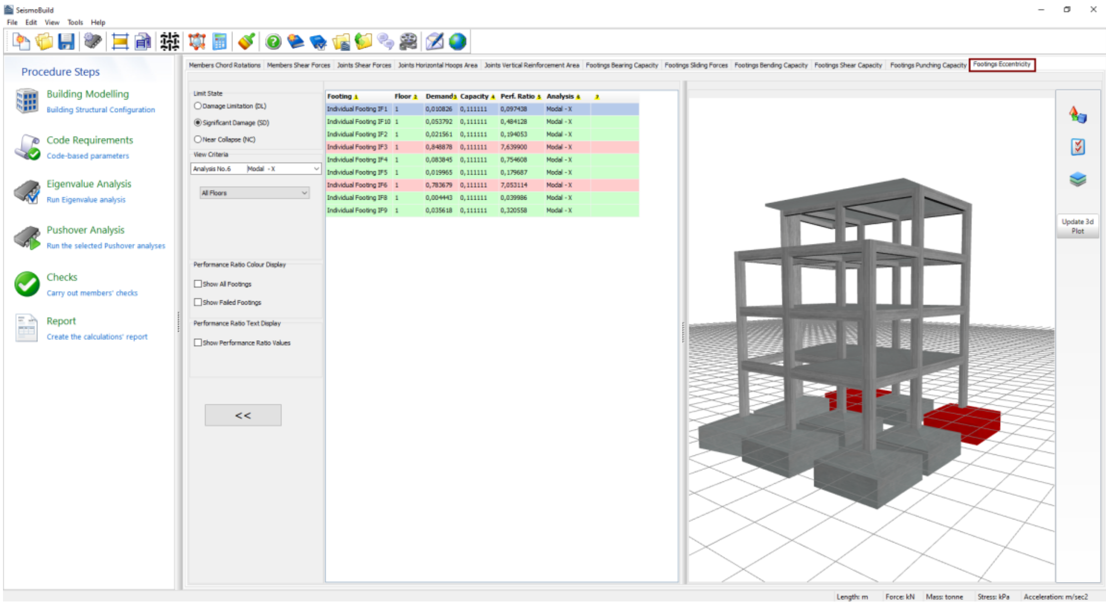
Analysis	Position	Demand	Capacity	Perf. Ratio
Envelope	Column Side	354,934002	4293,333333	0,082671
Envelope	Basic Perimeter	32,311834	624,094501	0,051774
Uniform + X	Column Side	244,079971	4293,333333	0,056851
Uniform + X	Basic Perimeter	22,220107	624,094501	0,035604
Uniform - X	Column Side	131,083324	4293,333333	0,030532
Uniform - X	Basic Perimeter	11,933325	624,094501	0,019121
Uniform + Y	Column Side	105,473579	4293,333333	0,024567
Uniform + Y	Basic Perimeter	9,601911	624,094501	0,015385
Uniform - Y	Column Side	338,335768	4293,333333	0,078805
Uniform - Y	Basic Perimeter	30,800794	624,094501	0,049353
Modal + X	Column Side	240,852121	4293,333333	0,056099
Modal + X	Basic Perimeter	21,926256	624,094501	0,035133
Modal - X	Column Side	137,504067	4293,333333	0,032027
Modal - X	Basic Perimeter	12,517844	624,094501	0,020058
Modal + Y	Column Side	93,035041	4293,333333	0,021670

Footing Checks module

A more detailed description on the footings punching capacity checks and the equations used in SeismoBuild is available in the Footings Punching Capacity section of the respective appendix.

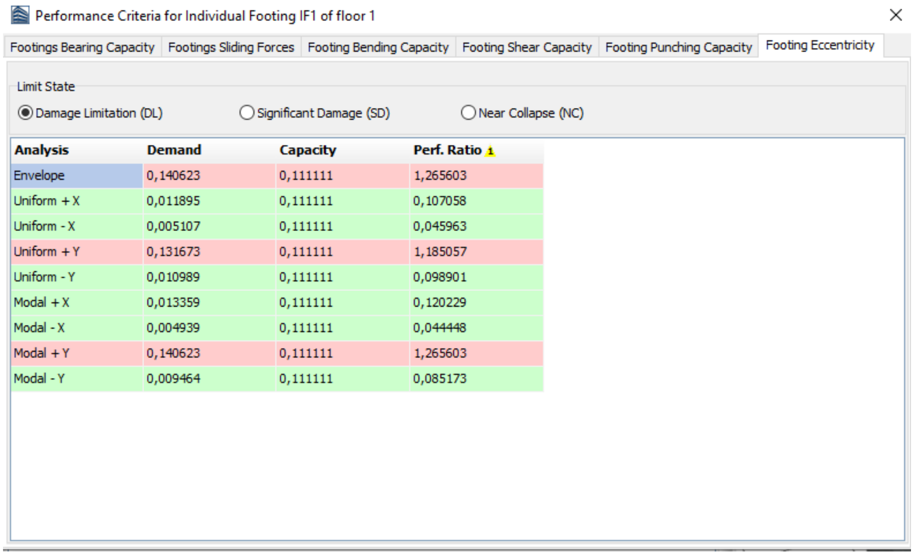
FOOTINGS ECCENTRICITY

The results of the eccentricity of loading checks for individual and strip footings, according to the selected Code, can be visualised in this module. Users may select for which Limit State, or Performance objective, to view the results and choose filters, which include the determination of the analysis, the floor and the local axis. The available limit states are those already selected in the Limit States module in Code Requirements. In addition, in the drop-down menu for the analysis selection apart from the executed analyses there is an option called Critical Analysis, which is the envelope of the results of all the analyses. The elements that have exceeded the acceptable eccentricity of loading according to the selected Code are marked with red both in the table view and the 3D view format, whereas different visualisation options are available. For instance, users may select to display the footings' performance ratios or each footing's critical analysis on the 3D plot, or to display the footings with different colours, depending on the value of their performance ratio for the selected limit state.



Footings Eccentricity Module

It is possible to hide the data entry table through the corresponding button in order to view the 3D rendering of the structural model in 'full-screen' modality. By right clicking on a footing and selecting the View All Footing Checks, a window appears with the checks for all the analyses for that particular footing.



Analysis	Demand	Capacity	Perf. Ratio
Envelope	0,140623	0,111111	1,265603
Uniform +X	0,011895	0,111111	0,107058
Uniform -X	0,005107	0,111111	0,045963
Uniform +Y	0,131673	0,111111	1,185057
Uniform -Y	0,010989	0,111111	0,098901
Modal +X	0,013359	0,111111	0,120229
Modal -X	0,004939	0,111111	0,044448
Modal +Y	0,140623	0,111111	1,265603
Modal -Y	0,009464	0,111111	0,085173

Footing Checks module

A more detailed description on the footings eccentricity checks and the equations used in SeismoBuild is available in the Footings Eccentricity section of the respective appendix.

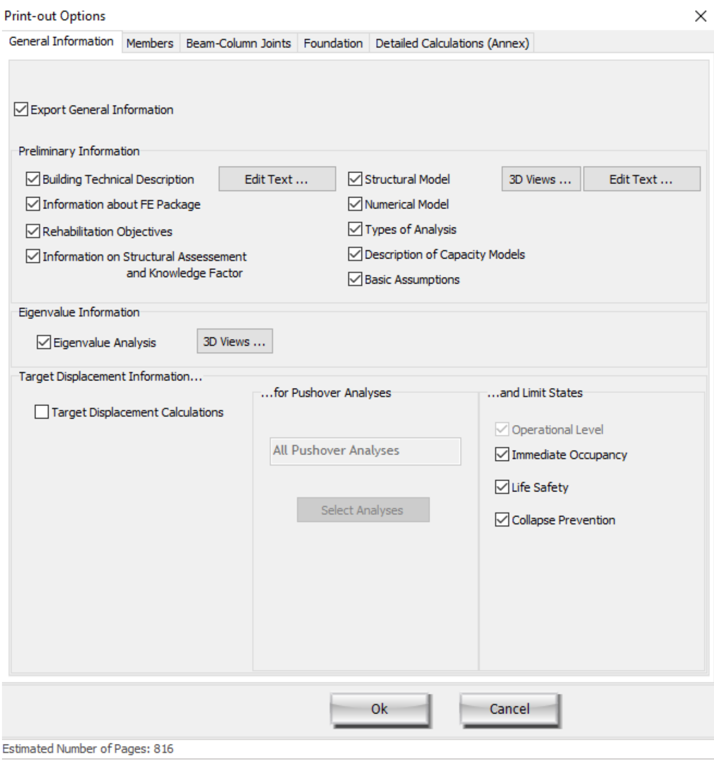
Report

After running all the analyses and the carrying out the Code-based checks, the technical report can be delivered, through the Report module in the program Main Window. Once, the Report button is clicked, a dialog box appears whereby users can determine which information will be included in the report. The available tabs are listed below and will be described in the following paragraphs:

- General Information
- Members
- Beam-Column Joints
- Detailed Calculations (Annex)

General Information

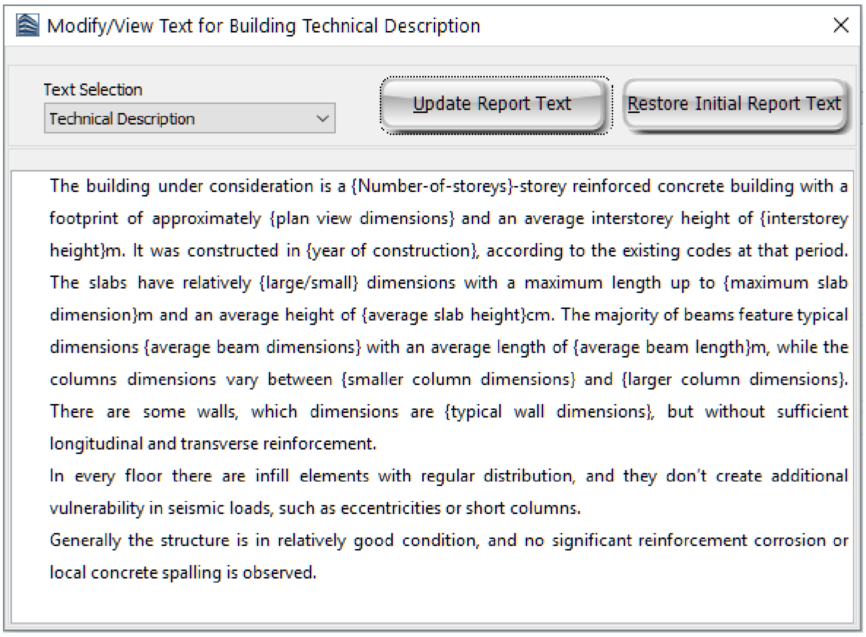
In the General Information module users may determine the general information of the structure that will be included in the technical report. The option of not extracting any general information is available by disabling the Export General Information check-box.



General Information tab

Preliminary Information

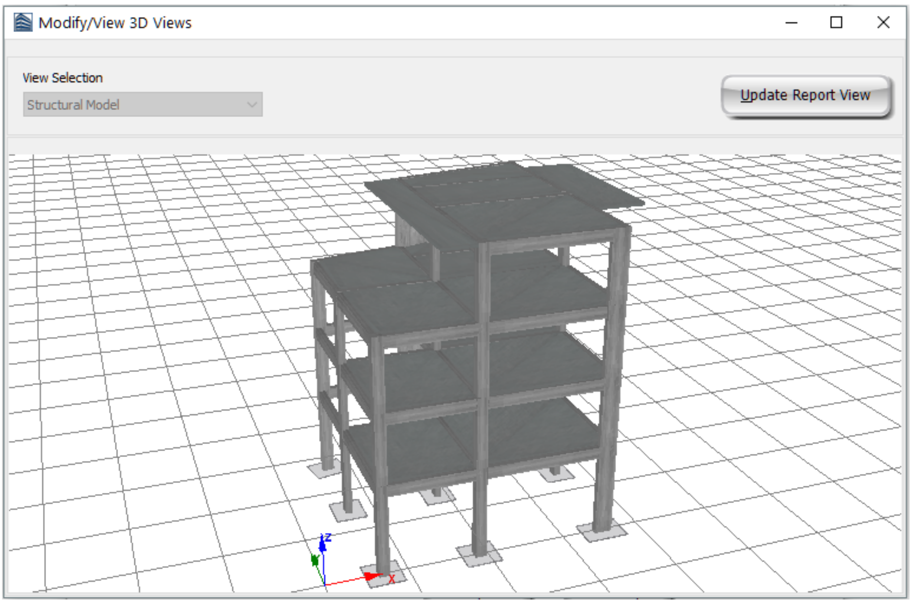
Herein, the information about the program and the Codes that will be included in the report is specified. An effort has been made to include by default the information needed for the case of the most common structures, but there are also numerous options available to the user with data to be included in the report. Further, by selecting the relevant *Edit Text* buttons predefined texts appear providing some guidelines for the technical description of the structure and the software, which can be partially or totally modified according to the users' needs and preferences.



View/Modify Text for Building Technical Description

All the information about the SeismoBuild program and the Codes, such as the rehabilitation objectives, the structural assessment, the knowledge factors, the types of analysis and the capacity models for assessment and checks, as well as the numerical model and the basic assumptions are predefined or automatically assigned and may be exported to the technical report by selecting the corresponding check-box.

The perspective of the 3D plot views of the structure exported in the technical report may be modified once the Update Report View button is selected within the 'View/Modify 3D Views' window, accessible by the *3D Views...* buttons.

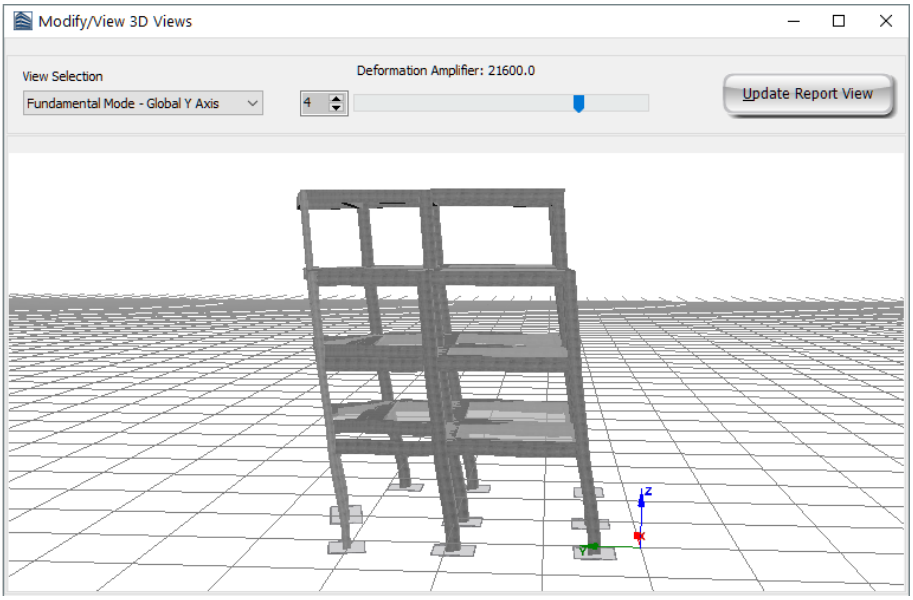


Modify/View 3D Views

NOTE: Users may also modify the text of the technical report if they export it in a *.rtf or *.html file format. The selection of the file format is available in the Preview window.

Eigenvalue Information

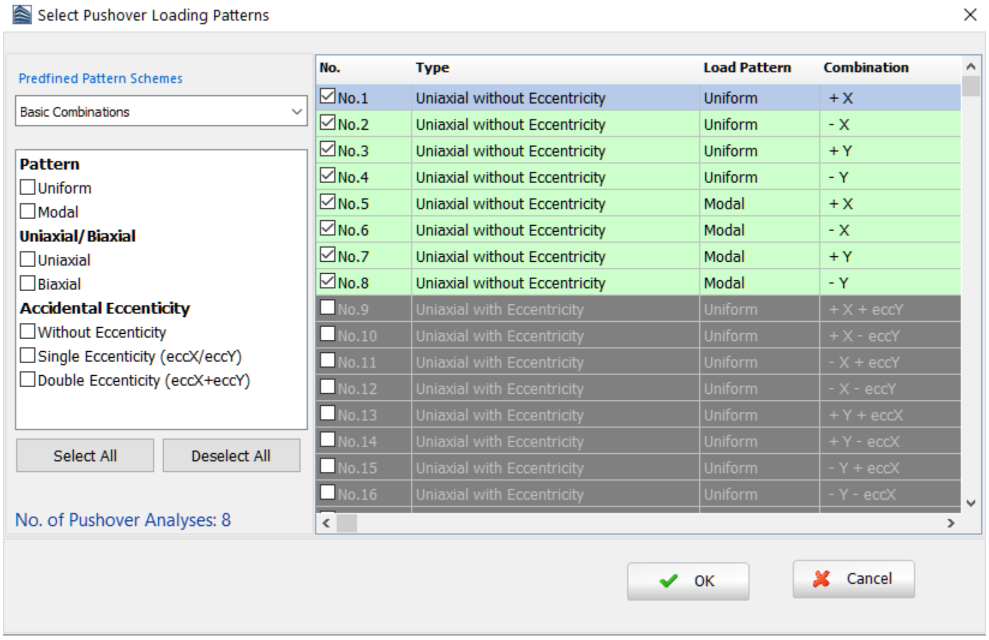
The choice of including the eigenvalue analysis results in the technical report is available in this module. The eigenvalue results that will be exported are the natural period/frequency of vibration of each mode, the modal participation factors, the effective modal masses and their percentages, as well as, two figures for the fundamental periods along global axes X and Y. Users may modify the perspective of these figures through the *3D Views...* button, wherein a scale amplifier is also available in order to change the 3D plot deformations scale.



Modify/View 3D Views

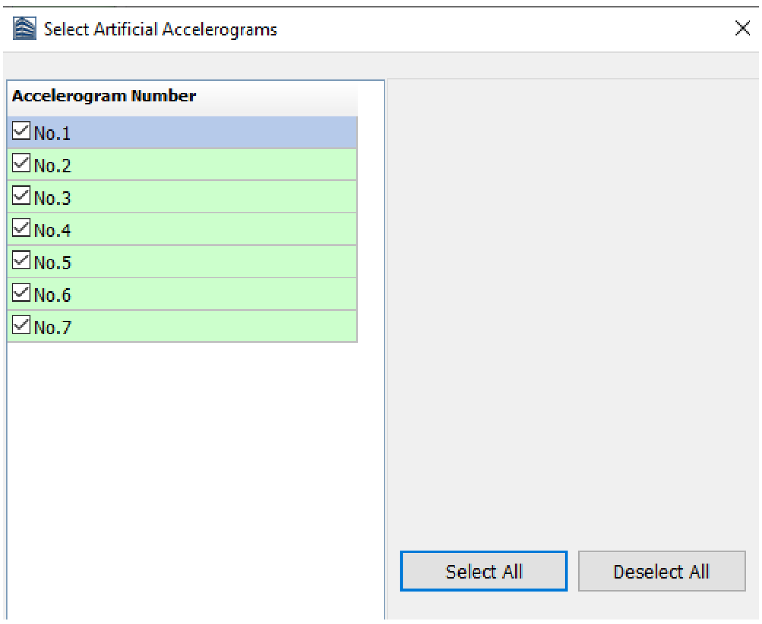
Target Displacement or Dynamic Analysis Information

When the nonlinear static procedure is selected, the capacity curve of the structure before and after the linearisation for the selected pushover analysis, as well as the target displacement for the selected limit states may be exported to the technical report. A detailed description about the Target Displacement calculations is available in the corresponding appendix for the selected Code (Appendix A.1 - EUROCODES, Appendix A.2 – ASCE, Appendix A.3 – NTC-18, Appendix A.4 – KANEPE and Appendix A.5 - TBDY). Users may define, for which analyses the target displacement information will be exported, as well as for which limit states. The default selection is to export the information for all the executed analyses and for all the limit states.



Select Pushover Loading Patterns module

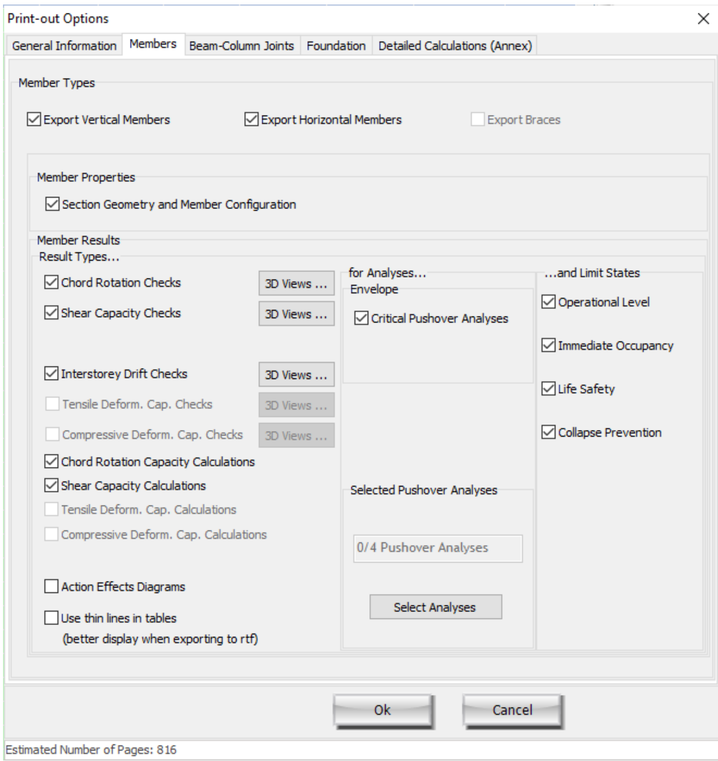
In the case of nonlinear dynamic procedure, the artificial accelerograms for the selected dynamic analysis in X and Y axis, as well as the equivalent response spectra for the selected limit states may be exported to the technical report. Users may define, for which analyses the artificial accelerograms information will be exported, as well as for which limit states. The default selection is to export the information for all the executed analyses and for all the limit states.



Select Artificial Accelerograms module

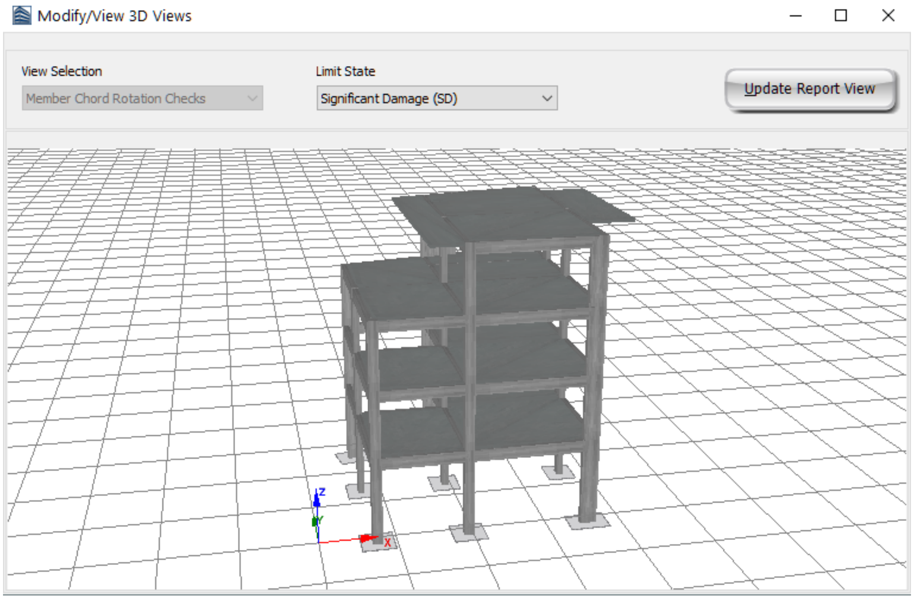
MEMBERS

In the Members module users may determine the information about the structural members checks that will be exported to the technical report. The choice of selecting whether to export just the vertical, the horizontal members or the steel braces, all or none is available from the corresponding check-boxes.



Members tab

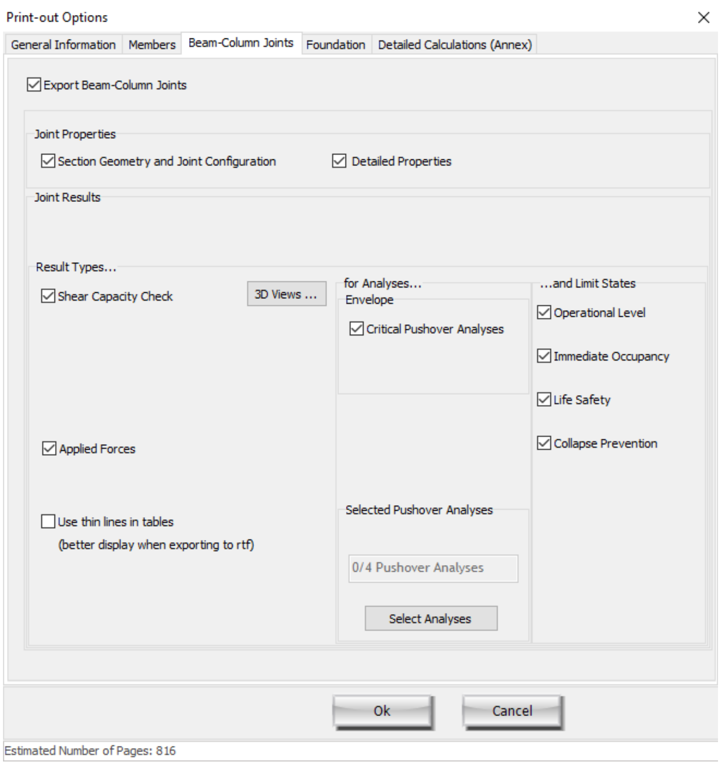
It is noted that the module is active only if the checks for members or steel braces have been carried out. The members' properties may also be selected to be exported. Further, users may choose the members' results that will be included in the technical report, i.e. which of the checks results to be exported, for which analyses and for which limit states. The option of selecting the envelope of the most critical analyses or the average values is available. The latter is available in nonlinear dynamic procedure only. It is noted that the perspective of the 3D plot view is modified in the report, through the *3D Views...* button, once the Update Report View button is selected.



Modify/View 3D Views

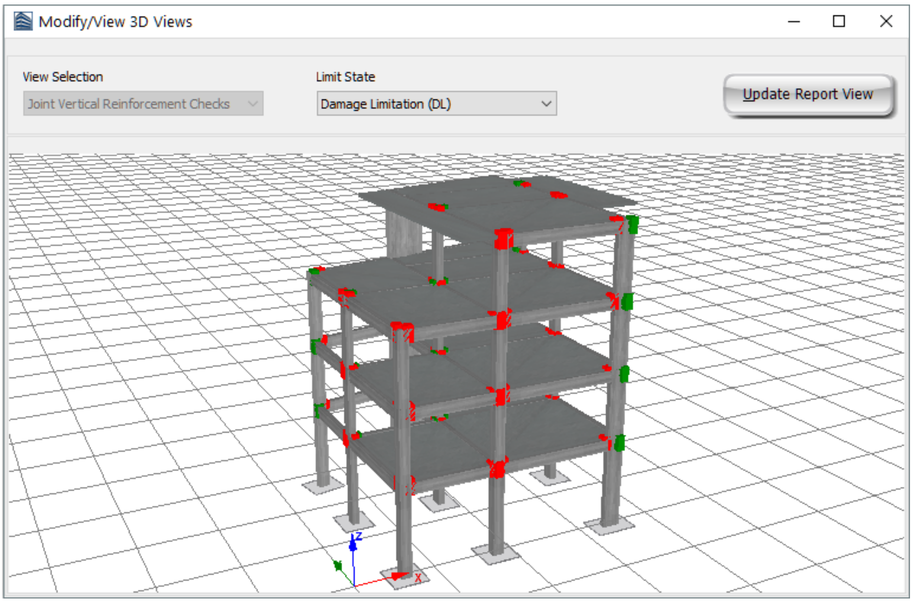
BEAM-COLUMN JOINTS

In the Beam-Column Joints module users may determine the information about the beam-column joint checks that are to be exported in the technical report. It is also possible not to output any information by disabling the Export Beam-Column Joints check-box.



Beam-Column Joints tab

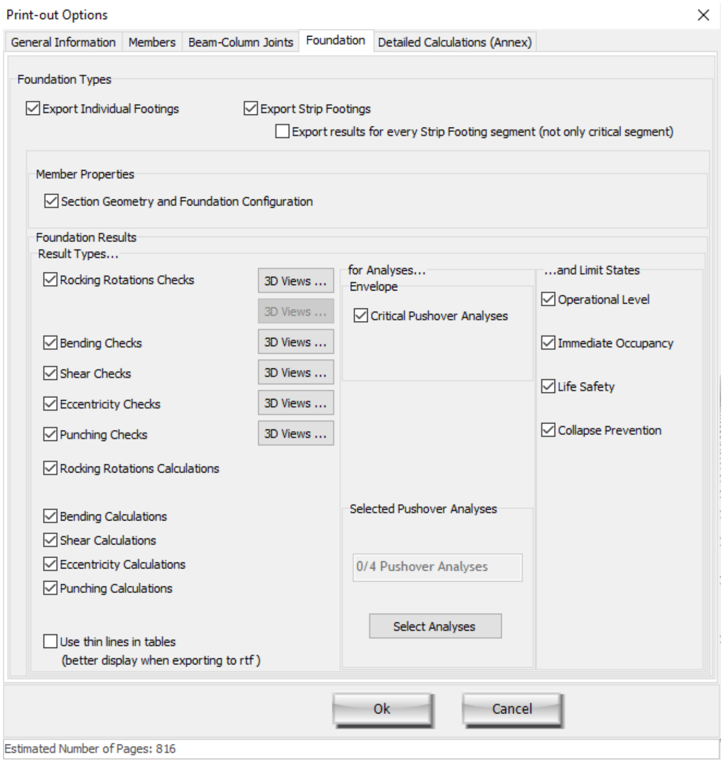
It is noted that the module is active only if one, or more, of the checks regarding the beam-column joints has been carried out. Users may select whether to export just simple or detailed information about the geometry and the configuration of the joint. It is noted that the exported results for the joints may differ depending on the selected Code. A detailed description on the available joints' checks may be found in the corresponding appendix; Appendix A.1 - EUROCODES, Appendix A.2 – ASCE, Appendix A.3 – NTC-18, Appendix A.4 – KANEPE and Appendix A.5 - TBDY. The option of selecting the envelope of the most critical analyses or the average values is available. The latter is available in nonlinear dynamic procedure only. Otherwise, specific analyses may be selected through the Select Analyses button. Finally, the limit states for which the results will be exported, as well as the corresponding 3D Views can be defined. It is noted that the 3D plot view is modified in the report once the Update Report View button is selected.



Modify/View 3D Views

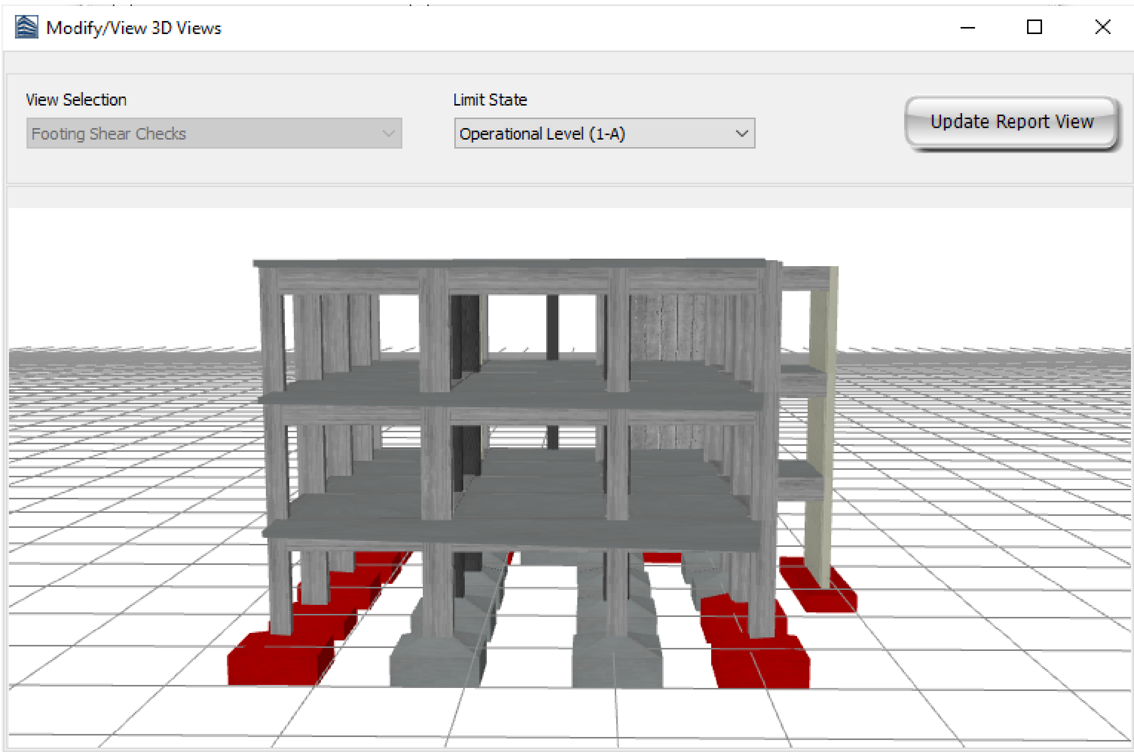
FOUNDATION

In the Foundation module users may determine the information about the foundation checks that are to be exported in the technical report. It is also possible not to output any information by disabling the Export Individual Footings and Export Strip Footings check-boxes.



Foundation tab

It is noted that the module is active only if one, or more, of the checks regarding the foundation has been carried out. Users may select whether to export just simple or detailed information about the geometry and the configuration of the foundation. It is noted that the exported results for the foundation may differ depending on the selected Code. A detailed description on the available foundation checks may be found in the corresponding appendix; Appendix A.1 - EUROCODES, Appendix A.2 – ASCE, Appendix A.3 – NTC-18, Appendix A.4 – KANEPE and Appendix A.5 - TBDY. The option of selecting the envelope of the most critical analyses or the average values is available. The latter is available in nonlinear dynamic procedure only. Otherwise, specific analyses may be selected through the Select Analyses button. Finally, the limit states for which the results will be exported, as well as the corresponding 3D Views can be defined. It is noted that the 3D plot view is modified in the report once the Update Report View button is selected.



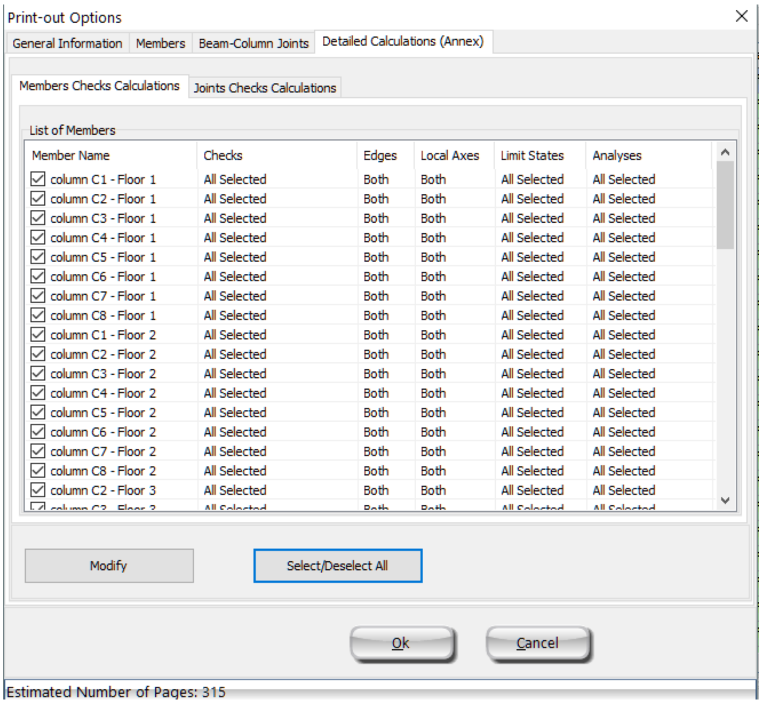
Modify/View 3D Views

Detailed Calculations (Annex)

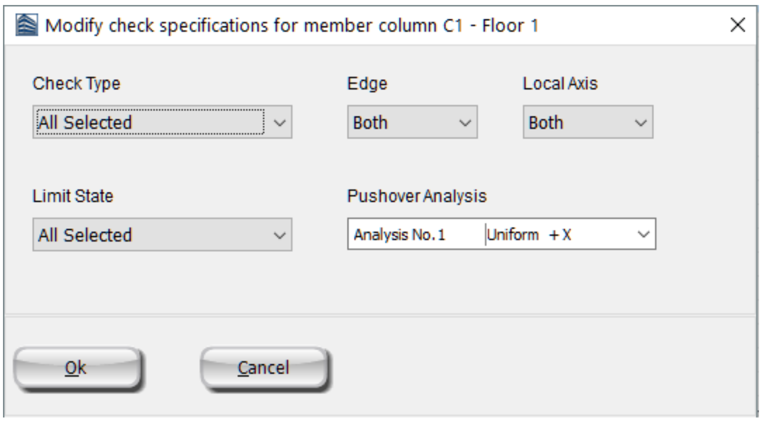
In the Detailed Calculations (Annex) module users may select the members and the beam-column joints, for which detailed checks calculations are to be output in the technical report. This module is available for nonlinear static procedure only.

Members Checks Calculations

The detailed Members' Checks Calculations may be exported to the technical report according to the employed equations, available in the corresponding appendix of the selected Code (Appendix A.1 - EUROCODES, Appendix A.2 – ASCE, Appendix A.3 – NTC-18, Appendix A.4 – KANEPE and Appendix A.5 - TBDY). Users may select the information to export per member, i.e. for which check type, edge, axis, limit state and pushover analysis, by selecting the member and the 'Modify' button. The default selection is to export all the members' checks calculations for the selected members.



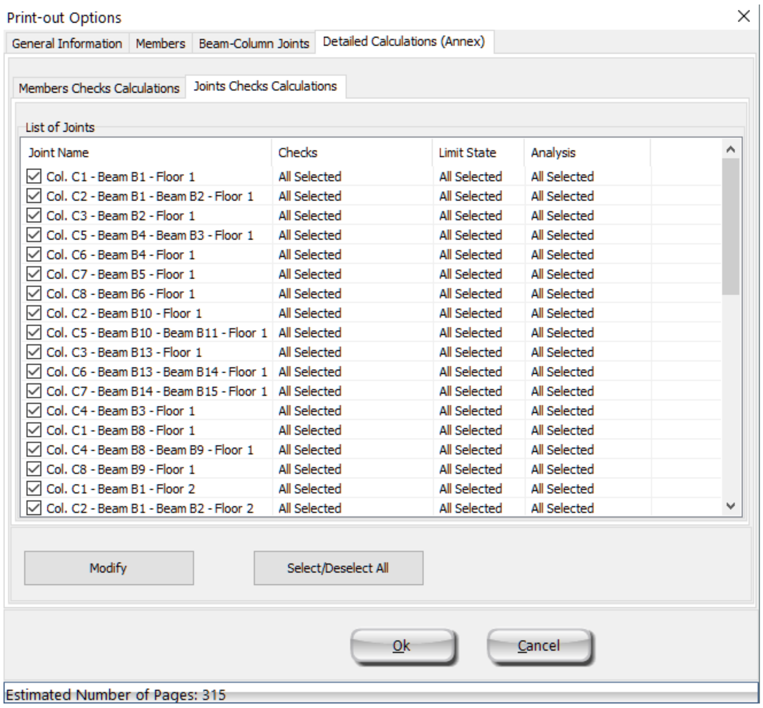
Members Checks Calculations tab



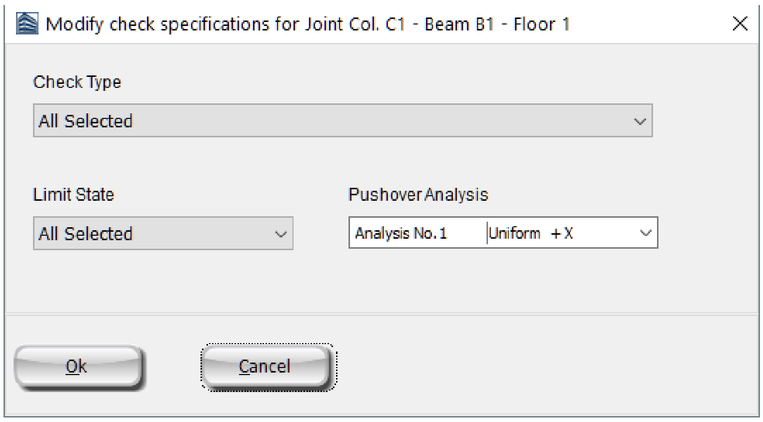
Modify Checks Specifications for Member tab

Joints Checks Calculations

The detailed Joints' Checks Calculations may be exported to the technical report according to the employed equations, available in the corresponding appendix of the selected Code (Appendix A.1 - EUROCODES, Appendix A.2 – ASCE, Appendix A.3 – NTC-18, Appendix A.4 – KANEPE and Appendix A.5 - TBDY). Users may select the information to export per joint, i.e. for which check type, limit state and pushover analysis, by selecting the joint and the 'Modify' button. The default selection is to export all the joints' checks calculations for the selected joints.



Joints Checks Calculations tab



Modify Checks Specifications for Joint tab

FRP Designer

A special program has been introduced in SeismoBuild and it is available from the main menu (Tools > FRP Designer) or through the corresponding toolbar button . FRP Designer provides an efficient solution for designing FRP strengthening of reinforced concrete columns and beams by computing the strength of reinforced concrete members strengthened with FRP laminates.

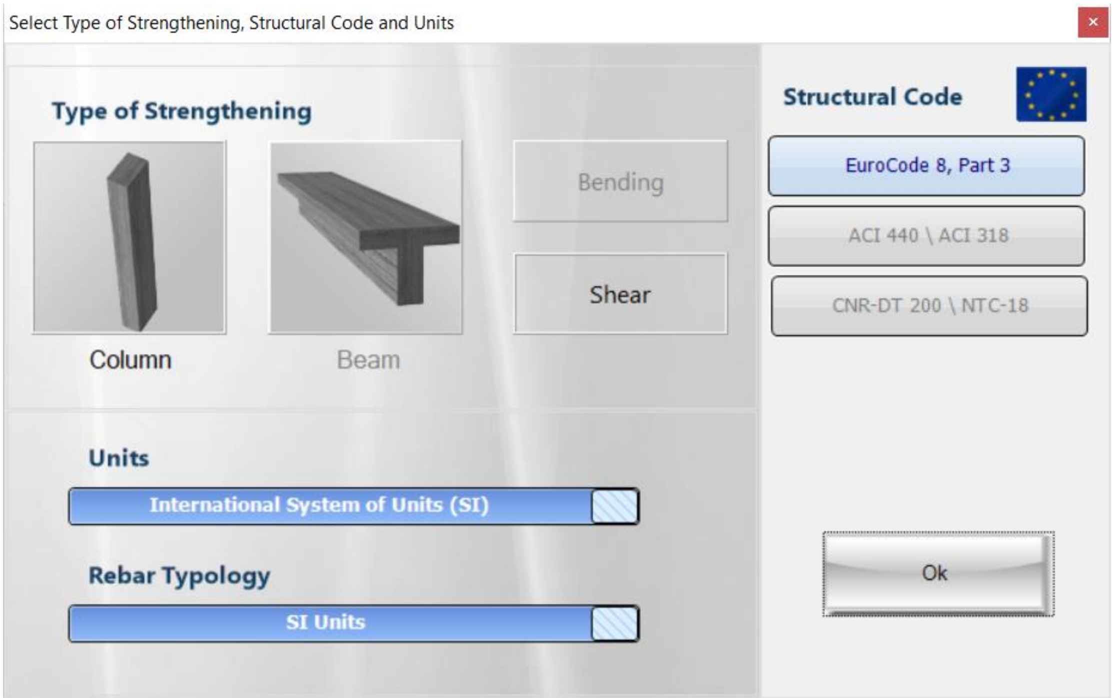
In particular the software computes the bending and shear capacity of reinforced concrete members at first without FRP laminate strengthening and then with strengthening by FRP laminates specified by the user. This way the software provides a tool for designing the strengthening of a reinforced concrete member against bending and shear and also assess the effects of strengthening with various types of FRP laminates.

The properties of the existing members like geometry, reinforcement, materials, loads and code-related parameters can be easily introduced using a user-friendly visual interface. A large variety of commercially available FRP laminates is available in the software while the user can introduce custom FRP laminate types. Parameters associated to the FRP wrapping like Radius of corners, number of FRP layers etc can also be easily introduced in the program.

The bending and shear capacity of an existing member with and without FRP strengthening can be easily computed according to three available codes. The European Eurocode 8 (Part 3) and FIB 90, the American ACI 440/ACI 318 and the Italian CNRT-DT 200 / NTC 18.

FRP Designer also gives the opportunity to create a full report of the capacity calculations of the strengthened members.

Finally, and due to its full integration with the Windows environment, FRP Designer allows for numerical and graphical results to be copied to any Windows application (e.g. MS Excel, MS Word, etc.).



FRP Designer start window

Bibliography

Abbasi V., Daudeville L., Kotronis P., Mazars J. [2004] "Using damage mechanics to model a four story RC framed structure submitted to earthquake loading," Proceedings of the Fifth International Conference on Fracture Mechanics of Concrete Structures, Vol. 2, pp. 823-830.

ACI 318-19 [2019] American Concrete Institute: Building Code Requirements for Structural Concrete (ACI 318-19) and Commentary.

ACI 369.1-22 [2023] American Concrete Institute: Seismic Evaluation and Retrofit of Existing Concrete Buildings-Code and Commentary.

ACI 440.2R-17 [2017] American Concrete Institute: Guide for the Design and Construction of Externally Bonded FRP Systems for Strengthening Concrete Structures.

Ahmad S.H., Shah S.P. [1982] "Stress-strain curves of concrete confined by spiral reinforcement," Journal of the American Concrete Institute, pp. 484-490.

Allotey N.K., El Naggar M.H. [2005a] "Cyclic Normal Force-Displacement Model for Nonlinear Soil-Structure Interaction Analysis: SeismoStruct Implementation," Research Report No. GEOT-02-05, Geotechnical Research Centre, Department of Civil & Environmental Engineering, University of Western Ontario, London, Ontario, Canada.

Allotey N.K., El Naggar M.H. [2005b] "Cyclic soil-structure interaction model for performance-based design," Proceedings of the Satellite Conference on Recent Developments in Earthquake Geotechnical Engineering, TC4 ISSMGE, Osaka, Japan.

Ameny P., Loov R.E., Shrive N.G. [1983] "Prediction of elastic behaviour of masonry," International Journal of Masonry Construction, Vol. 3, No. 1, pp. 1-9.

Annaki M., Lee K.L.L. [1977] "Equivalent uniform cycle concept for soil dynamics," Journal of Geotechnical Engineering Division, ASCE, Vol. 103, No. GT6, pp. 549-564.

ANSI/AISC 342-22 [2022] American Institute of Steel Construction: Seismic Provisions for Evaluation and Retrofit of Existing Structural Steel Buildings.

Anthes R.J. [1997] "Modified rainflow counting keeping the load sequence," International Journal of Fatigue, Vol. 19, No. 7, pp. 529-535.

Alemdar B.N., White D.W. [2005] "Displacement, flexibility, and mixed beam-column finite element formulations for distributed plasticity analysis," Journal of Structural Engineering, Vol. 131, No. 12, pp. 1811-1819.

Antoniou S., Rovithakis A., Pinho R. [2002] "Development and verification of a fully adaptive pushover procedure," Proceedings of the Twelfth European Conference on Earthquake Engineering, London, UK, Paper No. 822.

Antoniou S., Pinho R. [2004a] "Advantages and Limitations of Force-based Adaptive and Non-Adaptive Pushover Procedures," Journal of Earthquake Engineering, Vol. 8, No. 4, pp. 497-522.

Antoniou S., Pinho R. [2004b] "Development and Verification of a Displacement-based Adaptive Pushover Procedure," Journal of Earthquake Engineering, Vol. 8, No. 5, pp. 643-661.

ASCE 41-23 [2023] American Society of Civil Engineers: Seismic Evaluation and Retrofit of Existing Buildings.

Asteris P.G. [2003] "Lateral stiffness of brick masonry infilled plane frames," ASCE Journal of Structural Engineering, Vol. 129, No. 8, pp. 1071-1079.

Asteris PG, Chrysostomou CZ, Giannopoulos IP, Smyrou E. Masonry infilled reinforced concrete frameswith openings. In: COMPDYN 2011: 3rd international conference on computational methods instructural dynamics and earthquake engineering, Corfu, Greece; 26–28 May, 2011.

Atkinson R.H., Amadei B.P., Saeb S., Sture S. [1989] "Response of masonry bed joints in direct shear," ASCE Journal of Structural Division, Vol. 115, No. 9, pp. 2276-2296.

Auricchio F., Sacco E. [1997] "A superelastic shape-memory-alloy beam," Journal of Intelligent Materials and Structures, Vol. 8, pp. 489-501.

Bathe K.J. [1996] "Finite Element Procedures in Engineering Analysis," 2nd Edition, Prentice Hall.

Beyer, K., Dazio, A., and Priestley, M.J.N. [2008] "Seismic design of torsionally eccentric buildings with U-shaped RC walls," ROSE School, Pavia, Italy.

Beyer, K., Dazio, A., and Priestley, M.J.N. [2008] "Inelastic Wide-Column Models for U-Shaped Reinforced Concrete Walls," Journal of Earthquake Engineering 12:Sp1, 1-33.

Beyer, K., Dazio, A., and Priestley, M.J.N.[2008] "Elastic and inelastic wide-column models for RC non rectangular walls," Proceedings of the fortieth World Conference on Earthquake Engineering, Beijing, China.

Benjamin S.T., Williams H.A. [1958] "The behaviour of one-storey brick shear walls," ASCE Journal of Structural Division, Vol. 84, No.ST4, pp. 30.

Bento R., Pinho R., Bhatt C. [2008] "Nonlinear Static Procedures for the seismic assessment of the 3D irregular SPEAR building," Proceedings of the Workshop on Nonlinear Static Methods for Design/Assessment of 3D Structures, Lisbon, Portugal.

Bernuzzi C., Zandonini R., Zanon P. [1996] "Experimental analysis and modelling of semi-rigid steel joints under cyclic reversal loading," Journal of Constructional Steel Research, Vol. 38, No. 2, pp. 95-123.

Bertoldi S.H., Decanini L.D., Gavarini C. [1993] "Telai tamponati soggetti ad azione sismica, un modello semplificato: confronto sperimentale e numerico," (in Italian) Atti del 6° convegno nazionale ANIDIS, Vol. 2, pp. 815-824, Perugia, Italy.

Bertoldi S.H., Decanini L.D., Santini S., Via G. [1994] "Analytical models in infilled frames," Proceedings of the Tenth European Conference on Earthquake Engineering, Vienna, Austria.

Beyer K, Dazio A., Priestley M.J.N. [2008] "Inelastic wide-column models for U-shaped reinforced concrete wall," Journal of Earthquake Engineering, Vol. 12, No. 1, pp. 1-33.

Binda L., Fontana A., Frigerio G. [1988] "Mechanical behaviour of brick masonries derived from unit and mortar characteristics," Proceedings of the Eighth International Brick and Block Masonry Conference, Dublin, Ireland.

Blandon, C.A. [2005] Implementation of an Infill Masonry Model for Seismic Assessment of Existing Buildings, Individual Study, European School for Advanced Studies in Reduction of Seismic Risk (ROSE School), Pavia, Italy.

Broderick B.M., Elnashai A.S., Izzuddin B.A. [1994] "Observations on the effect of numerical dissipation on the nonlinear dynamic response of structural systems," Engineering Structures, Vol. 16, No. 1, pp. 51-62.

Calabrese A., Almeida J.P., Pinho R. [2010] "Numerical issues in distributed inelasticity modelling of RC frame elements for seismic analysis," Journal of Earthquake Engineering, Vol. 14, Special Issue 1, pp. 38-68.

Casarotti C., Pinho R. [2006] "Seismic response of continuous span bridges through fibre-based finite element analysis," Journal of Earthquake Engineering and Engineering Vibration, Vol. 5, No. 1, pp. 119-131.

Casarotti C., Pinho R. [2007] "An Adaptive capacity spectrum method for assessment of bridges subjected to earthquake action," Bulletin of Earthquake Engineering, Vol. 5, No. 3, pp. 377-390.

CEB [1996] RC Frames under Earthquake Loading: State of the Art Report, - Comite Euro-International du Beton, Thomas Telford, London, England.

Celarec D., Dolšek M. [2012] "Practice-oriented probabilistic seismic performance assessment of infilled frames with consideration of shear failure of columns," Earthquake Engineering & Structural Dynamics, Vol. 43, No. 9, pp. 1339-1360.

CEN [2005] European Prestandard ENV 1998: Eurocode 8 - Design provisions for earthquake resistance of structures. Comite Europeen de Normalisation, Brussels.

Chang, G.A., Mander, J.B. [1994] "Seismic Energy Based Fatigue Damage Analysis of Bridge Columns: Part 1 – Evaluation of Seismic Capacity," NCEER Technical Report No. NCEER-94-0006, State University of New York, Buffalo, N.Y.

Chege J.K., Matalanga N. [2000] "NDT Application in Structural Integrity Evaluation of Bomb Blast Affected Buildings," Proceedings of the Fifteenth World Conference on Non-Destructive Testing, Rome, Italy.

Chopra A.K. [1995] Dynamics of Structures: Theory and Applications to Earthquake Engineering, Prentice-Hall.

Chopra A.K. & Goel R.K. [2001] A Modal Pushover Analysis Procedure to Estimate Seismic Demands for Buildings: Theory and Preliminary Evaluation, Pacific Earthquake Engineering Research Center, Technical Report PEER No. 2001/03.

Clough R.W., Johnston S.B. [1966] "Effect of Stiffness Degradation on Earthquake Ductility Requirements" Proceedings, Second Japan National Conference on Earthquake Engineering, 1966, pp.227-232.

Clough R.W., Penzien J. [1994] Dynamics of Structures, 2nd Edition, McGraw Hill.

Constantinou, M.C., Tsopelas, P.C., Kasalanati, A., and Wolff, E. [1999], "Property Modification Factors for Seismic Isolation Bearings," *Report No. MCEER-990012*, Multidisciplinary Center for Earthquake Engineering Research, State University of New York, Buffalo, NY.Cook R.D., Malkus D.S., Plesha M.E. [1989] Concepts and Applications of Finite Elements Analysis, John Wiley & Sons.

Cook R.D., Malkus D.S., Plesha M.E. [1989] Concepts and Applications of Finite Elements Analysis, John Wiley & Sons.

Correia A.A., Virtuoso F.B.E. [2006] "Nonlinear Analysis of Space Frames," Proceedings of the Third European Conference on Computational Mechanics: Solids, Structures and Coupled Problems in Engineering, Mota Soares et al. (Eds.), Lisbon, Portugal.

Cremer C., Pecker A., Davenne L. [2002] "Modelling of non linear dynamic behaviour of a shallow foundation with macro - element," Journal of Earthquake Engineering, Vol. 6, No. 2, pp. 175-212.

Crisafulli F.J. [1997] Seismic Behaviour of Reinforced Concrete Structures with Masonry Infills, PhD Thesis, University of Canterbury, New Zealand.

Crisafulli F.J., Carr A.J., Park R. [2000] "Analytical modelling of infilled frame structures – a general overview," Bulletin of the New Zealand Society for Earthquake Engineering, Vol. 33, No 1.

Crisfield M.A. [1991] Non-linear Finite Element Analysis of Solids and Structures, John Wiley & Sons.

Dawe J.I., Young T.C. [1985] "An investigation of factors influencing the behaviour of masonry infills in steel frames subjected to on-plane shear," Proceedings of the Seventh International Brick Masonry Conference, Melbourne, Australia.

Dawe J. L. and Seah C. K. [1989] "Out-of-plane resistance of concrete masonry infilled panels." Can. J. Civ. Eng., 16(6), 854-856

Decanini L.D., Fantin G.E. [1987] "Modelos Simplificados de la Mamposteria Incluida en Porticos. Características s de Rigidez y Resistencia Lateral en Estado Limite," Jornadas Argentinas de Ingenieria Estructural, Buenos Aires, Argentina, Vol. 2, pp. 817-836. (in Spanish)

Deierlein G.G., Reinhorn A.M., and Willford M.R. [2010] "Nonlinear structural analysis for seismic design," NEHRP Seismic Design Technical Brief No. 4, produced by the NEHRP Consultants Joint Venture, a partnership of the Applied Technology Council and the Consortium of Universities for Research in Earthquake Engineering, for the National Institute of Standards and Technology, Gaithersburg, MD, NIST GCR 10-917-5.

Della Corte G., De Matteis G., Landolfo R. [2000] "Influence of connection modelling on seismic response of moment resisting steel frames," in Moment Resistant Connections of Steel Frames in Seismic Areas: Design and Reliability (ed. F.M. Mazzolani), Chapter 7, E&FN Spon, London, New York.

De Martino A., Faella C., Mazzolani F.M. [1984] "Simulation of Beam-to-Column Joint Behaviour under Cyclic Loads," Construzioni Metalliche, Vol. 6, pp. 346-356.

De Sortis A., Di Pasquale G., Nasini U. [1999] Criteri di Calcolo per la Progettazione degli Interventi – Terremoto in Umbria e Marche del 1997, Servizio Sismico Nazionale, Rome, Italy. (in Italian)

Dionysis Biskinis and Michael N. Fardis [2013] "Stiffness and cyclic deformation capacity of circular RC columns with or wothout lap-splices and FRP wrapping," Bulletin of Earthquake Engineering 11(5)

Dodd L., Restrepo-Posada J. [1995] "Model for Predicting Cyclic Behavior of Reinforcing Steel," Journal of Structural Engineering, Vol. 121, No. 3, pp. 433–445.

Drysdale R.G., Khattab M.M. [1995] "In-plane behaviour of grouted concrete masonry under biaxial tension-compression," American Concrete Institute Journal, Vol. 92, No. 6, pp. 653-664.

Elnashai A.S., Elghazouli A.Y. [1993] "Performance of composite steel/concrete members under earthquake loading, Part I: Analytical model," Earthquake Engineering and Structural Dynamics, Vol. 22, pp. 315-345.

Emori K., Schnobrich W.C. [1978] Analysis of Reinforced Concrete Frame-Wall Structures for Strong Motion Earthquakes, Structural Research Series No. 434, Civil Engineering Studies, University of Illinois at Urbana-Champaign.

EN 1998-1 (2004) (English): Eurocode 8: Design of structures for earthquake resistance – Part 1: General rules, seismic actions and rules for buildings.

EN 1998-3 (English) [2004]: Eurocode 8: Design of structures for earthquake resistance -Part 3: Assessment and retrofitting of buildings.

Felippa C.A. [2001] "Nonlinear Finite Element Methods," Lecture Notes, Centre for Aerospace Structure, College of Engineering, University of Colorado, USA. Available from URL: http://www.colorado.edu/engineering/CAS/courses.d/NFEM.d/Home.html.

Felippa C.A. [2004] "Introduction to Finite Element Methods," Lecture Notes, Centre for Aerospace Structure, College of Engineering, University of Colorado, USA. Available from URL: http://www.colorado.edu/engineering/CAS/courses.d/IFEM.d/Home.html.

Ferracuti B., Savoia M. [2005] "Cyclic behaviour of FRP-wrapped columns under axial and flexural loadings," Proceedings of the International Conference on Fracture, Turin, Italy.

Ferracuti B., Pinho R., Savoia M., Francia R. [2009] "Verification of Displacement-based Adaptive Pushover through multi-ground motion incremental dynamic analyses," Engineering Structures, Vol. 31, pp. 1789-1799.

FIB [2001] "Externally Bonded FRP Reinforcement for RC Structures," FIB Bulletin n. 14, Federation Internationale du Beton, pp. 138.

FIB [2006] Retrofitting of Concrete Structures by Externally Bonded FRPS, with Emphasis on Seismic Applications, FIB Bulletin n. 35, Federation Internationale du Beton, pp. 220.

Filippou F.C., Popov E.P., Bertero V.V. [1983] "Effects of bond deterioration on hysteretic behaviour of reinforced concrete joints," Report EERC 83-19, Earthquake Engineering Research Center, University of California, Berkeley.

Filippou F.C., Fenves G.L. [2004] "Methods of analysis for earthquake-resistant structures", Chapter 6 in 'Earthquake Engineering - From Engineering Seismology to Performance-Based Engineering', eds. Y. Bozorgnia and V.V. Bertero, Cambridge University Press, Cambridge, United Kingdom.

Fiorato A.E., Sozen M.A., Gamble W.L. [1970] An Investigation of the Interaction of Reinforced Concrete Frames with Masonry Filler Walls, Report UILU-ENG-70-100, Department of Civil Engineering, University of Illinois, Urbana-Champaign IL, USA.

Fragiadakis M., Pinho R., Antoniou S. [2008] "Modelling inelastic buckling of reinforcing bars under earthquake loading," in Progress in Computational Dynamics and Earthquake Engineering, Eds. M. Papadrakakis, D.C. Charmpis, N.D. Lagaros and Y. Tsompanakis, A.A. Balkema Publishers – Taylor & Francis, The Netherlands.

Fragiadakis M., Papadrakakis M. [2008] "Modeling, analysis and reliability of seismically excited structures: computational issues," International Journal of Computational Methods, Vol. 5, No. 4, pp. 483-511.

Freitas J.A.T., Almeida J.P.M., Pereira E.M.B.R. [1999] "Non-conventional formulations for the finite element method," Computational Mechanics, Vol. 23, pp. 488-501.

Fugazza D. [2003] Shape-memory Alloy Devices in Earthquake Engineering: Mechanical Properties, Constitutive Modelling and Numerical Simulations, MSc Dissertation, European School for Advanced Studies in Reduction of Seismic Risk (ROSE School), Pavia, Italy.

Gasparini D., Vanmarcke E.H. [1976] Simulated Earthquake Motions Compatible with Prescribed Response Spectra, M.I.T. Department of Civil Engineering Research Report

Gere J.M., Timoshenko S.P. [1997] Mechanics of Materials, 4th Edition.

Giannakas A., Patronis D., Fardis M. [1987] "The influence of the position and the size of openings to the elastic rigidity of infill walls," Proceedings of Eighth Hellenic Concrete Conference, Xanthi-Kavala, Greece. (in Greek)

Giberson, M.F. [1967] “The Response of Nonlinear Multi-Story Structures subjected to Earthquake Excitation,” Doctoral Dissertation, California Institute of Technology, Pasadena, CA., May 1967, 232pp.

Giberson, M.F. [1969] “Two Nonlinear Beams with Definition of Ductility,” Journal of the Structural Division, ASCE, Vol. 95, No. 2, pp. 137-157.

Gostic S., Zarnic R. [1985] "Cyclic lateral response of masonry infilled R/C frames and confined masonry walls," Proceedings of the Eighth North-American Masonry Conference, Austin, Texas, USA.

Halldorsson B., Papageorgiou A.S. [2005] Calibration of the specific barrier model to earthquake of different tectonic regions, Bulletin of the Seismological Society of America, Vol.95, No.4, pp.1276-1300.

Hall J.F. [2006] "Problems encountered from the use (or misuse) of Rayleigh damping," Earthquake Engineering and Structural Dynamics, Vol. 35, No. 5, pp. 525-545.

Hamburger R.O. [1993] "Methodology for seismic capacity evaluation of steel-frame buildings with infill unreinforced masonry," Proceedings of the US National Conference on Earthquake Engineering, Memphis, Tennesse, USA.

Hamburger R.O., Foutch D.A., Cornell C.A. [2000] "Performance basis of guidelines for evaluation, upgrade and design of moment-resisting steel frames," Proceedings of the Twelfth World Conference on Earthquake Engineering, Auckland, New Zealand, paper No. 2543.

Han T.-S., Feenstra P.H., Billington S.L. [2003] "Simulation of Highly Ductile Fiber-Reinforced Cement-Based Composite Components Under Cyclic Loading , Vol. 100, No. 6, pp. 749-757Hellesland J., Scordelis A. [1981] "Analysis of RC bridge columns under imposed deformations," IABSE Colloquium, Delft, pp. 545-559.

Hendry A.W. [1990] Structural Masonry, Macmillan Education Ltd, London, England.

Hilber H.M., Hughes T.J.R., Taylor R.L. [1977] "Improved numerical dissipation for time integration algorithms in structural dynamics," Earthquake Engineering and Structural Dynamics, Vol. 5, No. 3, pp. 283-292.

Holmes M. [1961] "Steel Frames with Brickwork and Concrete Infilling," Proceedings of the Institution of Civil Engineers, Vol. 19, pp. 473-478.

Hughes T.J.R. [1987] The Finite Element Method, Linear Static and Dynamic Finite Element Analysis, Prentice-Hall.

Hyodo M., Yamamoto Y., Sugyiama M. [1994] "Undrained cyclic shear behavior of normally consolidated clay subjected to initial static shear stresses," Soils and Foundations, Vol. 34, No. 4, pp. 1-11.

Irons B.M. [1970] "A frontal solution program for finite element analysis," International Journal for Numerical Methods in Engineering, Vol. 2, pp. 5-32.

Izzuddin B.A. [1991] Nonlinear Dynamic Analysis of Framed Structures, PhD Thesis, Imperial College, University of London, London, UK

Kaldjian M.J. [1967] "Moment-curvature of beams as Ramberg-Osgood functions," Journal of Structural Division, ASCE, Vol. 93, No. ST5, pp. 53-65.

KANEPE [2022] Earthquake Planning and Protection Organisation of Greece: Code for Structural Interventions, 3rd revision.

Kappos A., Konstantinidis D. [1999] "Statistical analysis of confined high strength concrete," Materials and Structures, Vol. 32, pp. 734-748.

Karsan I.D., Jirsa J.O. [1969] "Behavior of concrete under compressive loading.", Journal of the Structural Division, Proceedings of the American Society of Civil Engineers, Vol. 95, Issue 12, pp. 2543-2564

Kent D.C., Park R. [1971] "Flexural members with confined concrete.", Journal of the Structural Division, Proceedings of the American Society of Civil Engineers, Vol. 97, Issue 7, pp. 1969-1990

Kunnath S.K. [2004] "Identification of modal combination for nonlinear static analysis of building structures," Computer-Aided Civil and Infrastructure Engineering, Vol. 19, pp. 246-259.

Law K.H. and MackayD.R.[1992]. A parallel row-oriented sparse solution method for finite element structural analysis. Manuscript NA-92-10. Computer Science Department, Stanford University. August 1992.

Liauw T.C., Lee S.W. [1977] "On the behaviour and the analysis of multi-storey infilled frames subjected to lateral loading," Proceedings of Institution of Civil Engineers, Part 2, Vol. 63, pp. 641-656.

Liauw T.C., Kwan K.H. [1984] "Nonlinear behaviour of non-integral infilled frames," Computer and Structures, Vol. 18, No. 3, pp. 551-560.

Liu, J. W. H. [1986]. A compact row storage scheme for Cholesky factors using elimination trees. ACM Trans. Math. Software 12, 127-148 (1986).

Lopez-Menjivar MA. [2004] Verification of a Displacement-Based Adaptive Pushover Method for Assessment of 2D RC Buildings. PhD Thesis, European School for Advanced Studies in Reduction of Seismic Risk (ROSE School), University of Pavia, Italy.

Mackay, D. R., Law, K. H., and Raefsky, A. [1991] "An implementation of a generalized sparse/profile finite element solution method." Comput. Struct., 41, 723–737.

Madas P. [1993] "Advanced Modelling of Composite Frames Subjected to Earthquake Loading," PhD Thesis, Imperial College, University of London, London, UK.

Madas P. and Elnashai A.S. [1992] "A new passive confinement model for transient analysis of reinforced concrete structures," Earthquake Engineering and Structural Dynamics, Vol. 21, pp. 409-431.

Mainstone R.J. [1971] "On the Stiffnesses and Strength of Infilled Panels," Proceedings of the Institution of Civil Engineers, Supplement IV, pp. 57-90.

Mainstone R.J., Weeks G.A. [1970] "The influence of Bounding Frame on the Racking Stiffness and Strength of Brick Walls," Proceedings of the Second International Brick Masonry Conference, Stoke-on-Trent, United Kingdom.

Mallick D.V., Garg R.P. [1971] "Effect of openings on the lateral stiffness of infilled frames," Proceedings of the Institution of Civil Engineering, Vol. 49, pp. 193-209.

Mander J.B., Priestley M.J.N., Park R. [1988] "Theoretical stress-strain model for confined concrete," Journal of Structural Engineering, Vol. 114, No. 8, pp. 1804-1826.

Mann W., Muller H. [1982] "Failure of shear-stresses masonry - an enlarged theory, tests and application to shear walls, " Proceedings of the British Ceramic Society, Vol. 30, pp. 139-149.

Mari A., Scordelis A. [1984] "Nonlinear geometric material and time dependent analysis of three dimensional reinforced and prestressed concrete frames," SESM Report 82-12, Department of Civil Engineering, University of California, Berkeley.

Martinez-Rueda J.E. [1997] Energy Dissipation Devices for Seismic Upgrading of RC Structures, PhD Thesis, Imperial College, University of London, London, UK.

Martinez-Rueda J.E., Elnashai A.S. [1997] "Confined concrete model under cyclic load," Materials and Structures, Vol. 30, No. 197, pp. 139-147.

Menegotto M., Pinto P.E. [1973] "Method of analysis for cyclically loaded R.C. plane frames including changes in geometry and non-elastic behaviour of elements under combined normal force and bending," Symposium on the Resistance and Ultimate Deformability of Structures Acted on by Well Defined Repeated Loads, International Association for Bridge and Structural Engineering, Zurich, Switzerland, pp. 15-22.

Mirza S.A. [1989] "Parametric study of composite column strength variability," Journal of Construction Steel Research, Vol. 14, pp. 121-137.

Monti G., Nuti C. [1992] "Nonlinear cyclic behaviour of reinforcing bars including buckling," Journal of Structural Engineering, Vol. 118, No. 12, pp. 3268-3284.

Monti, G., Nuti, C., Santini, S. [1996] CYRUS - Cyclic Response of Upgraded Sections, Report No. 96-2, University of Chieti, Italy.

Mosalam K.M., White R.N., Gergely P. [1997] "Static response of infilled frames using quasi-static experimentation," ASCE Journal of Structural Engineering, Vol. 123, No. 11, pp. 1462-1469.

Mpampatsikos V., Nascimbene R., Petrini L. [2008] "A critical review of the R.C. frame existing building assessment procedure according to Eurocode 8 and Italian Seismic Code," Journal of Earthquake Engineering, Vol. 12, Issue SP1, pp. 52-58.

Nagashima T., Sugano S., Kimura H., Ichikawa A. [1992] "Monotonic axial compression tests on ultra high strength concrete tied columns," Proceedings of the Tenth World Conference on Earthquake Engineering, Madrid, Spain, pp. 2983-2988.

Neuenhofer A., Filippou F.C. [1997] "Evaluation of nonlinear frame finite-element models," Journal of Structural Engineering, Vol. 123, No. 7, pp. 958-966.

Newmark N.M. [1959] "A method of computation for structural dynamics," Journal of the Engineering Mechanics Division, ASCE, Vol. 85, No. EM3, pp. 67-94.

Nogueiro P., Simoes da Silva L., Bento R., Simoes R. [2005a] "Numerical implementation and calibration of a hysteretic model with pinching for the cyclic response of steel and composite joints," Proceedings of the Fourth International Conference on Advances in Steel Structures, Shangai, China, Paper no. ISP-45.

Nogueiro P., Simoes da Silva L., Bento R., Simoes R. [2005b] "Influence of joint slippage on the seismic response of steel frames," Proceedings of the EuroSteel Conference on Steel and Composite Structures, Maastricht, Nederlands, Paper no. 314.

NTC 2018, D.M. Infrastrutture Trasporti 17 gennaio 2018 (G.U. 20 febbraio 2018 n. 8 - Suppl. Ord.). ”Norme tecniche per le Costruzioni” (NTC18).

Oran C. [1973] "Tangent stiffness in space frames," Journal of the Structural Division, ASCE, Vol. 99, No. ST6, pp. 987-1001.

Otani, S. [1974] SAKE, A Computer Program for Inelastic Response of R/C Frames to Earthquakes, Report UILU-Eng-74-2029, Civil Engineering Studies, University of Illinois at Urbana-Champaign, USA.

Otani S. [1981] "Hysteresis Models of Reinforced Concrete for earthquake Response Analysis," Journal of Faculty of Engineering, University of Tokyo, Vol. XXXVI, No2, 1981 pp 407-441.

Papia M. [1988] "Analysis of infilled frames using a coupled finite element and boundary element solution scheme," International Journal for Numerical Methods in Engineering, Vol. 26, pp. 731-742.

Park R., Paulay T. [1975] Reinforced Concrete Structures, John Wiley & Sons, New York.

Park Y. J., Wen Y. K., Ang H-S. [1986] “Random Vibration of Hysteretic Systems under Bi- Directional Ground Motions”, *Earthquake Engi neering and Structural Dynamics*, Vol. 14, No 4, pp. 543-557.

Paulay T., Priestley M.J.N. [1992] Seismic Design of Reinforced Concrete and Masonry Buildings, John Wiley & Sons Inc., New York.

Pegon P. [1996] "Derivation of consistent proportional viscous damping matrices," JRC Research Report No I.96.49, Ispra, Italy.

Penelis G.G., Kappos A.J. [1997] Earthquake-resistant Concrete Structures, E & FN Spon, London, UK.

Pietra D., Pinho R. and Antoniou S. [2006] "Verification of displacement-based adaptive pushover for seismic assessment of high-rise steel buildings," Proceedings of the First European Conference on Earthquake Engineering and Seismology, Geneva, Switzerland, Paper no. 956.

Pilkey W.D. [1994] Formulas for Stress, Strain, and Structural Matrices, John Wiley & Sons, New York.

Pinho R., Antoniou S. [2005] "A displacement-based adaptive pushover algorithm for assessment of vertically irregular frames," Proceedings of the Fourth European Workshop on the Seismic Behaviour of Irregular and Complex Structures, Thessaloniki, Greece.

Pinho R., Casarotti C., Antoniou S. [2007] "A comparison of single-run pushover analysis techniques for seismic assessment of bridges," Earthquake Engineering and Structural Dynamics, Vol. 36, No. 10, pp. 1347–1362.

Pinho R., Bhatt C., Antoniou S., Bento R. [2008a] "Modelling of the horizontal slab of a 3D irregular building for nonlinear static assessment," Proceedings of the Fourteenth World Conference on Earthquake Engineering, Beijing, China, Paper no. 05-01-0159.

Pinho R., Marques M., Monteiro R., Casarotti C. [2008b] "Using the Adaptive Capacity Spectrum Method for seismic assessment of irregular frames," Proceedings of the Fifth European Workshop on the Seismic Behaviour of Irregular and Complex Structures, Catania, Italy, Paper no. 21.

Pinho R., Monteiro R., Casarotti C., Delgado R. [2009] "Assessment of continuous span bridges through Nonlinear Static Procedures," Earthquake Spectra, Vol. 25, No. 1, pp. 143-159.

Priestley M.J.N. [2003] Myths and Fallacies in Earthquake Engineering, Revisited. The Mallet Milne Lecture, IUSS Press, Pavia, Italy.

Priestley M.J.N., Grant D.N. [2005] "Viscous damping in seismic design and analysis," Journal of Earthquake Engineering, Vol. 9, Special Issue 1, pp. 229-255.

Priestley M.J.N., Seible F., Calvi G.M. [1996] Seismic Design and Retrofit of Bridges, John Wiley & Sons Inc., New York.

Prota A., Cicco F., Cosenza E. [2009] "Cyclic behavior of smooth steel reinforcing bars: experimental analysis and modeling issues," Journal of Earthquake Engineering, Vol. 13, No. 4, pp. 500–519.

Przemieniecki J.S. [1968] Theory of Matrix Structural Analysis, McGraw Hill.

Ramberg W., Osgood W.R. [1943] Description of Stress-Strain Curves by Three Parameters, National Advisory Committee on Aeronautics, Technical Note 902.

Repapis C. [2000] Study of Different Approaches for Nonlinear Dynamic Analysis of RC Frames, MSc Dissertation, Dept. of Civil Engineering, Imperial College, London, UK.

Richard R.M., Abbott B.J. [1975] "Versatile Elastic Plastic Stress-Strain Formula," Journal of Engineering Mechanics, ASCE, Vol. 101, No. 4, pp. 511-515.

Riddington J.R., Ghazali M.Z. [1988] "Shear strength of masonry walls," Proceedings of the Eighth International Brick and Block Masonry Conference, Dublin, Ireland.

Rosenblueth, E.[1951] *A basis for a Seismic Design*, PhD Thesis, University of Illinois, Urbana, USA..

Sahlin S. [1971] Structural Masonry, Prentice-Hall Inc., New Jersey, USA.

San Bartolome A.[1990] Colleciσn del Ingeniero Civil, Libro No. 4, Colegio de Ingenierios del Peru, Peru. (in Spanish)

Sattar S. and Liel A.B. [2010] "Seismic Performance of Reinforced Concrete Frame Structures with and without Masonry Infill Walls". 9th U.S. National and 10th Canadian Conference on Earthquake Engineering, Toronto, Canada.

Scott B.D., Park R., Priestley M.J.N. [1982] "Stress-strain behaviour of concrete confined by overlapping hoops at low and high strain rates," ACI Journal, Vol. 79, No. 1, pp. 13-27.

Scott M.H., Fenves G.L. [2006] "Plastic hinge integration methods for force-based beam–column elements," ASCE Journal of Structural Engineering, Vol. 132, No. 2, pp. 244-252.

Seed H.B., Idriss I.M., Makdisi F., Banerjee N. [1975] "Representation of irregular stress time-histories by equivalent uniform stress series in liquefaction analysis," Report No. UCB/EERC 75-29, University of California, Berkeley, USA.

Sheikh S.A., Uzumeri S.M. [1982] "Analytical model for concrete confined in tied columns," Journal of the Structural Division, ASCE, Vol. 108, No. ST12, pp. 2703-2722.

Simo J.C., Hughes T.J.R. [1998] “Computational Inelasticity”, Springer-Verlag New York, USA.

Simoes R., Simoes da Silva L., Cruz P. [2001] "Behaviour of end-plate beam-to-column composite joints under cyclic loading," International Journal of Steel and Composite Structures, Vol. 1, No. 3, pp. 355-376.

Sivaselvan M., Reinhorn A.M. [1999] "Hysteretic models for cyclic behavior of deteriorating inelastic structures," Report MCEER-99-0018, MCEER/SUNY/Buffalo.

Sivaselvan M., Reinhorn A.M. [2001] "Hysteretic models for deteriorating inelastic structures," Journal of Engineering Mechanics, ASCE, Vol. 126, No. 6, pp. 633-640, with discussion by Wang and Foliente and closure in Vol. 127, No. 11.

Smyrou E., Blandon C.A., Antoniou S., Pinho R., Crisafulli F. [2011] "Implementation and verification of a masonry panel model for nonlinear dynamic analysis of infilled RC frames," Bullettin of Earthquake Engineering, DOI 10.1007/s10518-011-9262-6.

Spacone E., Ciampi V., Filippou F.C. [1996] "Mixed formulation of nonlinear beam finite element," Computers & Structures, Vol. 58, No. 1, pp. 71-83.

Spoelstra M., Monti G. [1999] "FRP-confined concrete model," Journal of Composites for Construction, ASCE, Vol. 3, pp. 143-150.

Shrive N.G. [1991] "Materials and material properties," in Reinforced and Prestressed Masonry, Longman Scientific and Technical, London, England.

Stafford-Smith B. [1966] "Behaviour of square infilled frames," Proceedings of the American Society of Civil Engineers, Journal of Structural Division, Vol. 92, No. ST1, pp. 381-403.

Stafford-Smith B. and Carter C. [1969]. "A method for the analysis of infilled frames", Proc. Instn. Civ. Engrs., 44, 31–48.

Stockl S., Hofmann P. [1988] "Tests on the shear bond behaviour in the bed-joints of masonry," Proceedings of the Eighth International Brick and Block Masonry Conference, Dublin, Ireland.

Takeda T., Sozen M.A., Nielsen N.N. [1970] "Reinforced concrete response to simulated earthquakes," Journal of Structural Division, ASCE, Vol. 96, No. ST12, pp. 2557-2573.

TBDY [2018]: Turkish Seismic Building Regulations.

Thiruvengadam H. [1980] "On the natural frequencies of infilled frames," Journal of Earthquake Engineering and Structural Dynamics, Vol. 13, pp. 507-526.

Triantafillou T.C. [2006] "Seismic Retrofitting using Externally Bonded Fibre Reinforced Polymers (FRP)," (To appear in Chapter 5 of the fib bulletin "Seismic Assessment & Retrofit of RC Buildings")

Trueb U. [1983] Stability Problems of Elasto-Plastic Plates and Shells by Finite Elements, PhD Thesis, Imperial College, University of London, London.

TS500 [2000]: Requirements for Design and Construction of Reinforced Concrete Structures.

Utku B. [1980] "Stress magnifications in walls with openings," Proceedings of the Seventh World Conference on Earthquake Engineering, Istanbul, Turkey.

Vamvatsikos D., Cornell C.A. [2002] "Incremental dynamic analysis," Earthquake Engineering and Structural Dynamics, Vol. 31, No. 3, pp. 491-514.

Varum H.S.A. [2003] Seismic Assessment, Strengthening and Repair of Existing Buildings, PhD Thesis, University of Aveiro, Portugal.

Wilson E. [2001] Static and Dynamic Analysis of Structures, Computers and Structures Inc, Berkeley, California. (available at URL: http://www.edwilson.org/book/book.htm)

Wakabayashi M. [1986] Design of earthquake-resistant buildings, McGraw-Hill, USA.

Wan Q., Yi W. [1986] "The shear strength of masonry walls under combined stresses," Proceedings of the Fourth Canadian Masonry Symposium, University of New Brunswick, Canada.

Wen Y.K. [1976] "Method for Random Vibration of Hysteretic Systems.", *JourJournal of the Engineering Mechanics Division*, ASCE, Vol. 102, Issue 2, Pg. 249-263.

Wolf J.P. [1994] Foundation Vibration Analysis Using Simple Physical Models, Prentice Hall, New Jersey, USA.

Yassin M.H.M. [1994] Nonlinear analysis of prestressed concrete structures under monotonic and cyclic loads, PhD Thesis, University of California, Berkeley, USA.

Yankelevsky D.Z., Reinhardt H.W. [1989] "Uniaxial behavior of concrete in cyclic tension," Journal of Structural Engineering, ASCE, Vol. 115, No. 1, pp. 166-182.

Zienkiewicz O.C., Taylor R.L. [1991] The Finite Element Method, 4th Edition, McGraw Hill.

Zona A. and Dall'Asta A. [2012] "Elastoplastic model for Steel buckling-restrained braces",Journal of Constructional Steel Research , Vol. 68, pp. 118-125

Appendix A – Codes

Appendix A.1 - EUROCODES

In this appendix the parameters used for the structures assessment according to the Eurocodes (EC8-Part1 and Part3) are presented.

TYPE OF ANALYSIS

Current practice in Europe in structural assessment is regulated by the Eurocode 8: Design of Structures for Earthquake Resistance – Part 1: General rules, seismic action and rules for buildings (CEN, 2005a) and Part 3: Assessment and Retrofitting of Buildings (CEN, 2005b).

According to Eurocode 8 (CEN, 2005b), the seismic actions effects in combination with the effects of the permanent and variable loads are evaluated using one of the following methods:

- Lateral force analysis, subject to limitations specified in EN 1998-1:2004 section 4.3.3.2.1 with the addition of section 4.4.2 of EN 1998-3:2005;
- Modal response spectrum analysis, subject to limitations specified in EN 1998-1:2004 section 4.3.3.3.1 with the addition of the conditions specified in section 4.2 of EN 1998-3:2005;
- Nonlinear static (pushover) analysis, according to sections 4.3.3.4.2.1 of EN1998-1:2004 and 4.4.4 of EN 1998-3:2005;
- Nonlinear time history dynamic analysis, according to the procedure of section 4.3.3.4.3. of EN 1998-1:2004;
- q-factor approach, as described in EN 1998-1:2004 section 4.3.3.2 or 4.3.3.3, as appropriate.

In SeismoBuild the most common method in assessment practice of existing buildings is employed, which is the nonlinear static analysis. It is based on pushover analyses carried out under constant gravity loads and increasing lateral forces, applied at the location of the masses to simulate the inertia forces induced by the seismic action. As the model may account for both geometrical and mechanical nonlinearity, this method can describe the evolution of the expected plastic mechanisms and structural damage.

Each pushover analysis leads to a capacity curve, which is a relationship between the total base shear and the horizontal displacement of a representative point of the structure, termed “control node”. The demand at the considered Limit State – Near Collapse, Significant Damage or Damage Limitation - is determined by the appropriate comparison between the capacity determined by the pushover curve and the demand established as the damped Linear Response Spectrum. To do so, the “control node” displacements are defined in terms of spectral quantities relative to an equivalent single-degree-of-freedom (SDOF) system which is derived from the multi-degree-of-freedom (MDOF) response estimated according to Annex B of EN1998-1:2004.

The structural demand associated with the acquired target displacement shall fulfil the verification criteria defined in Eurocode 8 – Part3 (CEN, 2005b). Accordingly, element’s demand for brittle (shear) and ductile (chord rotation deformation) actions are deemed to comply with limits that take into account: section mechanical properties; element’s bending, shear and axial force interaction; and strength/stiffness degradation associated with the ductility demand and cyclic hysteretic response of reinforced concrete elements, through appropriate material nonlinearity consideration.

Performance Requirements

According to EN1998-3 section 2.1, performance requirements refer to the state of damage in the structure defined through three limit states, namely Near Collapse (NC), Significant Damage (SD) and Damage Limitation (DL).

Limit State of Near Collapse (NC)

The limit state of Near Collapse (NC) may be selected, according to EN 1998-3, where the target state of damage in the structure is near collapse and would probably not survive another earthquake, even of moderate intensity. The structure is heavily damaged with low residual lateral strength and stiffness, although vertical elements are still capable of sustaining vertical loads. Most non-structural components have collapsed and large permanent drifts are present. The appropriate level of protection is achieved by choosing a seismic action with a return period of 2.475 years corresponding to a probability of exceedance of 2% in 50 years.

Limit State of Significant Damage (SD)

The limit state of Significant Damage (SD) may be selected, according to EN 1998-3, where the target state of damage in the structure is significant and can sustain after-shocks of moderate intensity, although it is likely to be uneconomic to repair. Some residual lateral strength and stiffness, and vertical elements are capable of sustaining vertical loads. Non-structural components are damaged, although partitions and infills have not failed out-of-plane. Moderate permanent drifts are present. The appropriate level of protection is achieved by choosing a seismic action with a return period of 475 years corresponding to a probability of exceedance of 10% in 50 years.

Limit State of Damage Limitation (DL)

The limit state of Damage Limitation (DL) may be selected, according to EN 1998-3, where the target state of damage in the structure is insignificant and does not need any repair measures. The structure is only lightly damaged, with structural elements prevented from significant yielding and retaining their strength and stiffness properties. Non-structural components, such as partitions and infills may show distributed cracking, but the damage could be economically repaired. Permanent drifts are negligible. The appropriate level of protection is achieved by choosing a seismic action with a return period of 225 years corresponding to a probability of exceedance of 20% in 50 years.

The Eurocodes National Annexes specify whether to employ all three Limit States, two of them, or just one.

Information for Structural Assessment

In order to choose the admissible type of analysis and the appropriate confidence factor values, the following three knowledge levels are defined:

- KL1: Limited Knowledge
- KL2: Normal Knowledge
- KL3: Full Knowledge

The factors determining the obtained knowledge level are (i) geometry, i.e. the geometrical properties of the structural system and the non-structural elements, e.g. masonry infill panels, that may affect structural response; (ii) details, which include the amount and detailing of reinforcement in reinforced concrete sections, the connection of floor diaphragms to lateral resisting structure, the bond and mortar jointing of masonry and the nature of any reinforcing elements in masonry; and finally (iii) materials, that is the mechanical properties of the constituent materials.

KL1: Limited Knowledge

The limited knowledge level corresponds to a state of knowledge where the overall structural geometry and member sizes are known from survey or from original outline construction drawings used for both the original construction and any subsequent modifications, as well as a sufficient sample of dimensions of both overall geometry and member sizes checked on site. In case of significant discrepancies from the outline construction drawings a fuller dimensional survey is performed. The structural details are not known from detailed construction drawings and are assumed based on simulated design in accordance with usual practice at the time of construction. Limited inspections performed in the most critical elements should prove that the assumptions correspond to the actual situation. Information on the mechanical properties of the construction materials isn't available so default values are assumed in accordance with standards at the time of construction accompanied by limited in-situ testing in the most critical elements.

Structural evaluation based on this state of knowledge is performed through linear analysis methods, either static or dynamic.

KL2: Normal Knowledge

The normal knowledge level corresponds to a state of knowledge where the overall structural geometry and member sizes are known from extended survey or from outline construction drawings used for both the original construction and any subsequent modifications, as well as a sufficient sample of dimensions of both overall geometry and member sizes. The structural details are known from an extended in-situ inspection or from incomplete detailed construction drawings in combination with limited in-situ inspections in the most critical elements, which confirms that the available information corresponds to the actual situation. Information on the mechanical properties of the construction materials is available from extended in-situ testing or from original design specifications and limited in-situ testing.

Structural evaluation based on this state of knowledge is performed through linear or nonlinear analysis methods, either static or dynamic.

KL3: Full Knowledge

The full knowledge level corresponds to a state of knowledge where the overall structural geometry and member sizes are known from a comprehensive survey or from the complete set of outline construction drawings used for both the original construction and subsequent modifications, as well as a sufficient sample of both overall geometry and member sizes checked on site. The structural details are known from comprehensive in-situ inspection or from a complete set of detailed construction drawings in combination with limited in-situ inspections in the most critical elements, which prove that the available information corresponds to the actual situation. Information on the mechanical properties of the construction materials is available from comprehensive in-situ testing or from original test reports and limited in-situ testing.

Structural evaluation based on this state of knowledge is performed through linear or nonlinear analysis methods, either static or dynamic.

Confidence Factors

In the following table of EN1998-3 a summary and recommendations for the confidence factors and the analysis methods are provided for each knowledge level.

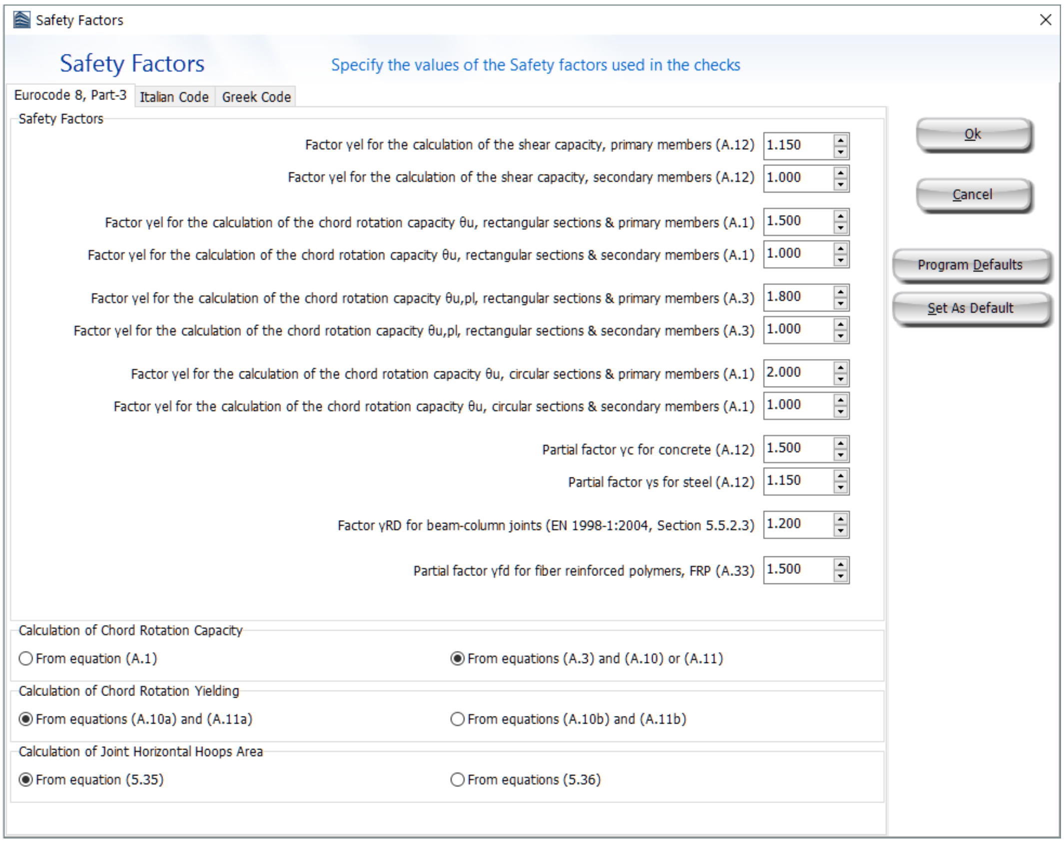
Knowledge Level	Geometry	Details	Materials	Analysis	CF
KL1	From original outline construction drawings with sample **visual** survey **or** from **full** survey	Simulated design in accordance with relevant practice **and** from **limited** in-situ inspection	Default values in accordance with standards of the time of construction **and** from **limited** in-situ testing	LF-MRS	CF_{KL1}
KL2		From incomplete original detailed construction drawings with **limited** in-situ inspection **or** from **extended** in-situ inspection	From original design specifications with **limited** in-situ testing **or** from **extended** in-situ testing	All	CF_{KL2}
KL3		From original detailed construction drawings with **limited** in-situ inspection **or** from **comprehensive** in-situ inspection	From original test reports with **limited** in-situ testing **or** from **comprehensive** in-situ testing	All	CF_{KL3}

NOTE The values ascribed to the confidence factors to be used in a country may be found in its National Annex. The recommended values are CF_{KL1}=1,35, CF_{KL2}=1,20 and CF_{KL3}=1,00.

Table 3.1 of EN 1998-3

Safety Factors

The values of the safety factors and the Code expressions employed may be specified through the dialog box that opens from the corresponding button. It is noted the default values of the safety factors are those defined in Eurocode 8 – Part 3.



Safety Factors module

CAPACITY MODELS FOR ASSESSMENT AND CHECKS

All the member checks (chord rotation capacity and shear capacity) should be carried out for all the elements of every floor, according to Annex A of EN1998-3:2005, considering the members as primary or secondary seismic elements, designated in accordance with the definitions in EN1998-1:2004, 4.2.2(1)P, (2) and (3). Moreover, beam-column joints checks may be employed in order to check (i) the horizontal shear forces acting on the core of the joints; (ii) the joint's horizontal hoops area and (iii) whether adequate vertical reinforcement is provided to the column passing through the joint.

Deformation Capacity

The deformation capacity of beams, columns and walls is defined in terms of the chord rotation θ, that is the angle between the tangent to the axis at the yielding end and the chord connecting that end with the end of the shear span (L_V=M/V=moment/shear at the end section). The chord rotation is also equal to the element drift ratio, which is the deflection at the end of the shear span with respect to the tangent to the axis at the yielding end divided by the shear span.

Deformation capacity of beams and columns is highly influenced by the lack of appropriate seismic resistant detailing in longitudinal reinforcement, as well as by the bars type, that is whether there are smooth bars or/and of cold-worked brittle steel. Inadequate development of splicing along the span (beams) and height (columns) and inadequate embedment into beam-column joints can control the member's response to seismic action, drastically limiting its capacity in respect to the situation in which the reinforcement is considered fully effective. The above limitations to the deformation capacity are taken into consideration.

The value for the chord rotation capacity for the limit state of near collapse (NC) is the value of the total chord rotation capacity (elastic plus inelastic part) at ultimate of concrete members under cyclic loading, which is calculated from the equations (A.1) and (A.3) of EC8: Part 3 (CEN, 2005b):

$$\theta_{um} = \frac{1}{\gamma_{el}} \cdot 0{,}016 \cdot (0{,}3^{v}) \left[\frac{\max(0{,}01;\omega')}{\max(0{,}01;\omega)} f_c\right]^{0{,}225} \cdot \left(\min(9, \frac{L_V}{h})\right)^{0{,}35} 25^{\left(\alpha\rho_{sx}\frac{f_{yw}}{f_c}\right)}(1{,}25^{100\rho_d})$$

(A.1) EC8: Part 3

Where γ_{el} is equal to 1,5 for primary seismic elements and to 1,0 for secondary seismic ones and L_V is the ratio between bending moment, M, and shear force, V. The remaining relevant parameters are defined in section A.3.2.2 of EC8: Part 3.

In walls the value given by the equation (A.1) is multiplied by 0,58.

The total chord rotation capacity at ultimate of concrete members under cyclic loading may be also calculated as the sum of the chord rotation at yielding and the plastic part of the chord rotation capacity calculated from the following expression:

$$\theta_{um}^{pl} = \theta_{um} - \theta_y$$
$$= \frac{1}{\gamma_{el}} \cdot 0{,}0145 \cdot (0{,}25^{v}) \left[\frac{\max(0{,}01;\omega')}{\max(0{,}01;\omega)}\right]^{0{,}3} \cdot f_c^{\,0{,}2} \cdot \left(\min\left(9, \frac{L_V}{h}\right)\right)^{0{,}35} 25^{\left(\alpha\rho_{sx}\frac{f_{yw}}{f_c}\right)}(1{,}275^{100\rho_d})$$

(A.3) EC8: Part 3

Where γ_{el} is equal to 1,8 for primary seismic elements and to 1,0 for secondary seismic ones; the chord rotation at yielding, θ_y, is calculated in accordance with the section A.3.2.4 of EC8: Part 3 and the remaining relevant parameters are defined in section A.3.2.2 of EC8: Part 3.

In walls the value of θ_{um}^{pl} given by the equation (A.3) is multiplied by 0,6.

The chord rotation capacity corresponding to the limit state of significant damage (SD) is assumed to be ¾ of the ultimate chord rotation, calculated from the equations above.

The chord rotation capacity that corresponds to the limit state of damage limitation (DL) is given by the chord rotation at yielding, evaluated as:

For beams and columns:

$$\theta_y = \varphi_y \frac{L_V + \alpha_V z}{3} + 0{,}0014\left(1 + 1{,}5\frac{h}{L_V}\right) + \frac{\varepsilon_y}{d-d'}\frac{d_{bL} f_y}{6\sqrt{f_c}} \qquad \text{(A.10a) EC8: Part 3}$$

For walls or rectangular T- or barbelled section:

$$\theta_y = \varphi_y \frac{L_V + \alpha_V z}{3} + 0{,}0013 + \frac{\varepsilon_y}{d-d'}\frac{d_{bL} f_y}{6\sqrt{f_c}} \qquad \text{(A.11a) EC8: Part 3}$$

Or from alternative and equivalent expressions for beams and columns

$$\theta_y = \varphi_y \frac{L_V + \alpha_V z}{3} + 0{,}0014\left(1 + 1{,}5\frac{h}{L_V}\right) + \varphi_y \frac{d_{bL} f_y}{8\sqrt{f_c}} \qquad \text{(A.10b) EC8: Part 3}$$

For walls or rectangular T- or barbelled section:

$$\theta_y = \varphi_y \frac{L_V + \alpha_V z}{3} + 0{,}0013 + \varphi_y \frac{d_{bL} f_y}{8\sqrt{f_c}} \qquad \text{(A.11b) EC8: Part 3}$$

Where α_V is equal to zero if the yielding bending moment is lower than L_V multiplied by the concrete shear resistance – $V_{R,c}$ - and 1.0 otherwise. $V_{R,c}$ is calculated according to EN1992-1-1:2004 provisions for concrete elements without shear reinforcement. The remaining relevant parameters are defined in section A.3.2.4 of EC8: Part 3.

The yield curvature of the end section is calculated according to the following expression for the sections whose compressive zone is of constant width and for the case that the section's yielding is due to steel yielding.

$$\varphi_y = (1/r)_y = \frac{f_y}{E_s(1-\xi_y)d}$$

If the section yields due to the deformation nonlinearities of the concrete in compression, that is for deformation of the edge compressive fibre larger than $\varepsilon_c \approx 1.8\, f_c/E_c$, then the yield curvature is calculated according to the following expression:

$$\varphi_y = (1/r)_y = \frac{\varepsilon_c}{\xi_y d} \approx \frac{1.8 f_c}{E_c \xi_y d}$$

The lower from the two values above is used for the calculation of the chord rotation capacity.

According to Annex A of EN1998-3 the chord rotation capacity is highly influenced by a number of different factors such as the type of the longitudinal bars. If cold-worked brittle steel is used the plastic part of chord rotation is divided by 2, whereas if smooth (plain) longitudinal bars are applied, section A.3.2.2(5) of Annex A is employed, taking, also, into consideration whether the longitudinal bars are well lapped or not. In case of members with lack of appropriate seismic resistant detailing the values given by expressions (A.1) and (A.3) are divided by 1,2. Moreover, if the deformed (high bond) longitudinal bars have straight ends lapped starting at the end section of the member, the plastic part of chord rotation is calculated with the value of the compression reinforcement ratio, ω', doubled over the value applying outside the lap splice. In addition, in sections where the reinforcement lap length l_o is less than the minimum lap length for ultimate deformation $l_{ou,min}$, the plastic part of the chord rotation capacity, given in (A.3) EC8: Part 3 equation, is multiplied by the ratio $l_o/l_{ou,min}$, for more information about the calculation of $l_{ou,min}$ you may refer to A.3.2.2(4) of Annex A, while the value for chord rotation at yielding, θ_y accounts for the effect of the lapping in accordance with A.3.2.4(3) of Annex A.

In the case of circular column sections, the equations above cannot be employed for the calculation of the elements' chord rotation capacity. In SeismoBuild the equations below suggested by D. Biskinis and M. N. Fardis [2013] are employed for θ_y and θ_u.

$$\theta_y = \varphi_y \frac{L_V + \alpha_V z}{3} + 0.0027\left(1 - \min\left(1; \frac{2}{15}\frac{L_s}{D}\right)\right) + \alpha_{sl}\frac{\varphi_y d_{bL} f_y}{8\sqrt{f_c}}$$

Where f_y and f_c values are in MPa, α_V=1 if V_{Rc}<V_{My}, V_{Rc} is calculated according to Eurocode 2 (CEN 2004), otherwise α_V=0, and α_{sl}=0 if pull-out of the tension bars from their anchorage zone beyond the yielding end is physically impossible, otherwise α_{sl}=1.

$$\theta_u = \left(\theta_y + (\varphi_u - \varphi_y)L_{pl}(1 - 0.5\, L_{pl}/L_s) + \alpha_{sl}\Delta\theta_{u,slip}\right)/\gamma_{el}$$

Where γ_{el} is equal to 2.0 for primary seismic elements and to 1.0 for secondary seismic elements, $\Delta\theta_{u,slip}$ and L_{pl} are calculated according to the following equations:

$$\Delta\theta_{u,slip} = 10 d_{bl}\,(\varphi_u + \varphi_y)/2$$

$$L_{pl} = 0.6D\left[1 + \frac{1}{6}\min\left(9; \frac{L_s}{D}\right)\right]$$

Users are advised to refer to the relevant publications for the definition of the other parameters and further details on the expression.

Concrete Jacketing

The following assumptions are made in order to evaluate the deformation capacities of the jacketed members, according to Annex A of EN1998-3:2005: (i) the jacketed element behaves monolithically, (ii) the full axial load is assumed to act on the jacketed member, disregarding the fact that the axial load is originally applied to the old column, and (iii) the concrete properties of the jacket are assumed to apply over the full section of the element.

The values of the jacketed members for M_y^*, $\theta_y^*$and θ_u^* that are adopted in the capacity verifications depend on the corresponding values calculated under the assumptions above, according to the following equations of Annex A of EN1998-3:2005:

The yield moment:

$$\mathrm{M_y^*} = \mathrm{M_y} \qquad \text{(A.18) EC8: Part 3}$$

The chord rotation at yield:

$$\theta_y^* = 1.05\theta_y \qquad \text{(A.19a) EC8: Part 3}$$

The ultimate chord rotation:

$$\theta_u^* = \theta_u \qquad \text{(A.20) EC8: Part 3}$$

FRP wrapping

The contribution of the FRP wrapping to members' capacity is taken into account, according to Annex A of EN1998-3:2005, as described below:

The effect of FRP wrapping on the members' flexural resistance at yielding is neglected, with the θ_y computed in accordance with A3.2.1(2) to (4).

The total chord rotation capacity and its plastic part for the members of rectangular sections with corners rounded is calculated through the expressions (A.1) and (A.3), respectively, with the exponent of the term due to confinement increased by $\alpha\rho_f f_{f,e}$, where α is the confinement effectiveness factor, ρ_f the FRP ratio parallel to the loading direction and $f_{f,e}$ the effectiveness stress given from the (A.35) equation of EC8: Part 3.

Bending Moment Capacity

The bending moment capacity of beams, columns and walls with prismatic cross section is calculated according to paragraph 4.1 of D. Biskinis and M. N. Fardis (2009), while for the bending moment capacity of circular columns paragraph 4 of Biskinis and M. N. Fardis (2013) is employed.

Shear Capacity

Shear capacity is calculated through the following expression according to Annex A of EN1998-3:2005, as controlled by the stirrups, accounting for the reduction due to the plastic part of ductility demand.

$$\mathrm{V_R} = \frac{1}{\gamma_{el}}\left[\frac{h-x}{2L_V}\min(N; 0{,}55A_c f_c) + \left(1 - 0{,}05\min\left(5; \mu_\Delta^{pl}\right)\right) \cdot \left[0{,}16\max(0{,}5; 100\rho_{tot})\left(1 - 0{,}16\min\left(5; \frac{L_V}{h}\right)\right)\sqrt{f_c}A_c + V_w\right]\right] \qquad \text{(A.12) EC8: Part 3}$$

Where γ_{el} is equal to 1,15 for primary seismic elements and to 1,0 for secondary ones, the other variables are calculated as defined in A.3.3.1 of Annex A of EN1998-3.

The shear strength of a concrete wall is not taken greater than the value corresponding to failure by web crushing, $V_{R,max}$, which under cyclic loading is calculated according to A3.3.1(2) of Annex A of EN1998-3:2005 from the following expression:

$$V_{R,max} = \frac{0,85\left(1-0,06\min\left(5;\mu_{\Delta}^{pl}\right)\right)}{\gamma_{el}}\left(1+1,8\min\left(0,15;\frac{N}{A_c f_c}\right)\right)\left(1+0,25\max(1,75;100\rho_{tot})\right)\cdot\left(1-0,2\min\left(2;\frac{L_V}{h}\right)\right)\sqrt{f_c}\,b_w z \qquad \text{(A.15) EC8: Part 3}$$

If in a concrete column the shear span ratio (L_V/h) at the end section with the maximum of the two end moments is less or equal to 2, the shear strength is not taken greater than the value corresponding to the failure by web crushing along the diagonal of the column after flexural yielding, $V_{R,max}$, which under cyclic loading is calculated according to A3.3.1(3) of Annex A of EN1998-3:2005 from the following expression:

$$V_{R,max} = \frac{{}^{4}/_{7}\left(1-0,02\min\left(5;\mu_{\Delta}^{pl}\right)\right)}{\gamma_{el}}\left(1+1,35\frac{N}{A_c f_c}\right)\left(1+0,45(100\rho_{tot})\right)\sqrt{\min(40;f_c)}\,b_w z\sin 2\delta$$

(A.16) EC8: Part 3

Where δ is the angle between the diagonal and the axis of the column ($\tan\delta = h/2L_V$).

Concrete Jacketing

The following assumptions are made in order to evaluate the strength of the jacketed members, according to Annex A of EN1998-3:2005: (i) the jacketed element behaves monolithically, (ii) the full axial load is assumed to act on the jacketed member, disregarding the fact that the axial load is originally applied to the old column, and (iii) the concrete properties of the jacket are assumed to apply over the full section of the element.

The value for the shear capacity, V_R^*, of the jacketed members that is adopted in the capacity verifications depend on the corresponding value calculated under the assumptions above, according to the following equation of Annex A of EN1998-3:2005:

$$V_R^* = 0.9V_R \qquad \text{(A.17) EC8: Part 3}$$

FRP wrapping

According to section A.4.4.2(9) of Annex A of EN1998-3:2005, in members with their plastic hinge region fully wrapped in an FRP jacket over a length at least equal to the member depth, the cyclic resistance V_R, may be calculated from expression (A.12) of EC8: Part 3 adding in V_w the contribution of the FRP jacket to shear resistance. The contribution of the FRP jacket to V_w is computed through the following expression:

$$V_{w,f} = 0,5\rho_f b_w z f_{u,fd} \qquad \text{(A.33) EC8: Part 3}$$

where ρ_f is the geometric ratio of the FRP, z the length of the internal lever arm and $f_{u,fd}$ the design value of the FRP ultimate strength.

Steel Braces Axial Deformations

Axial deformations of brace members in tension and compression should satisfy the provisions of tables B.2 and B.3 of Annex B of EN1998-3:2005.

Steel Braces Axial Forces

Axial force capacities of brace members in tension and compression should satisfy the provisions of sections 6.2.3 and 6.3 of EN1993-1.

Joints Shear Forces

The diagonal compression induced in the joint by the diagonal strut mechanism shall not exceed the compressive strength of concrete in the presence of transverse tensile strains. EN 1998-1:2004 defines that this requirement is satisfied by means of the subsequent rules:

For interior beam-column joints the following expression should be satisfied:

$$V_{jhd} \leq \eta f_{cd}\sqrt{1-\frac{\nu_d}{\eta}}\, b_j h_{jc} \qquad \text{(5.33) EC8: Part 1}$$

For exterior beam-column joints the corresponding equation is the following:

$$V_{jhd} \leq 80\% \eta f_{cd}\sqrt{1-\frac{\nu_d}{\eta}}\, b_j h_{jc}$$

V_{jhd} is the horizontal shear acting on the core of a joint between primary seismic beams and columns elements and is determined taking into account the most adverse conditions under seismic actions, i.e. capacity design conditions for the beams framing into the joint and the lowest compatible values of shear forces in the other framing elements. The expressions for the horizontal shear force acting on the concrete core of the joints are the following:

For interior beam-column joints:

$$V_{jhd} = \gamma_{Rd}(A_{s1} + A_{s2})f_{yd} - V_C \qquad \text{(5.22) EC8: Part 1}$$

For exterior beam-column joints:

$$V_{jhd} = \gamma_{Rd} A_{s1} f_{yd} - V_C \qquad \text{(5.23) EC8: Part 1}$$

For information about the values in the equations above users may refer to sections 5.5.3.3(2) and 5.5.2.3(2) of EN 1998-1:2004.

The option to consider re-bar stresses from analyses rather than the yielding stresses for the calculation of joints horizontal shear force demand in nonlinear analysis is available in the Elements tab of the Advanced Settings. In this case, the expressions for the horizontal shear force acting on the concrete core of the joints are the following:

For interior beam-column joints:

$$V_{jhd} = (\Sigma A_{1i} \cdot \sigma_{1i} + \Sigma A_{2i} \cdot \sigma_{2i}) - V_C$$

For exterior beam-column joints:

$$V_{jhd} = \Sigma A_{1i} \cdot \sigma_{1i} - V_C$$

Joints Horizontal Hoops Area

According to EN 1998-1:2004, adequate confinement of the joint should be provided, to limit the maximum diagonal tensile stress of concrete. This requirement may be satisfied by providing horizontal hoops calculated from the following expression:

$$\frac{A_{sh} f_{ywd}}{b_j h_{jw}} \geq \frac{\left(\frac{V_{jhd}}{b_j h_{jc}}\right)^2}{f_{ctd} + \nu_d f_{cd}} - f_{ctd} \qquad \text{(5.35) EC8: Part 1}$$

Where A_{sh} is the total area of the horizontal hoops and f_{ctb} is the design value of the tensile strength of concrete. For the definition of the other values users may refer to section 5.5.3.3(3) of EN 1998-1:2004.

Alternatively, the integrity of the joint after diagonal cracking may be ensured by horizontal hoop reinforcement. The total area of horizontal hoops that should be provided in the joint is calculated from the following equations:

For interior joints:

$$A_{sh}f_{ywd} \geq \gamma_{Rd}(A_{s1}+A_{s2})f_{yd}(1-0{,}8\nu_d)$$ (5.36a) EC8: Part 1

For exterior joints:

$$A_{sh}f_{ywd} \geq \gamma_{Rd}A_{s2}f_{yd}(1-0{,}8\nu_d)$$ (5.36b) EC8: Part 1

Where γ_{Rd} is equal to 1,2; for the definition of the other values users may refer to section 5.5.3.3(4) of EN 1998-1:2004.

Joints Vertical Reinforcement Area

Adequate vertical reinforcement of the column passing through the joint should be provided according to section 5.5.3.3(6) of EN 1998-1:2004, so that the following expression is satisfied:

$$A_{sv,i} \geq (2/3)A_{sh}(h_{jc}/h_{jw})$$ (5.37) EC8: Part 1

With $A_{sv,i}$ denoting the total area of the intermediate bars placed in the relevant column faces between corner bars of the column, including bars contributing to the longitudinal reinforcement of columns.

Joints Ductility

Adequate ductility should be possessed by both the structural elements and the structure as a whole according to section 4.4.2.3 of EN 1998-1:2004. In frame buildings with two or more storeys, the following condition should be satisfied at all joints of primary or secondary seismic beams with primary seismic columns:

$$\Sigma MR_c \geq 1{,}3\Sigma MR_b$$ EC8: Part 1 (4.29)

Where ΣMR_c is the sum of the design values of the moments of resistance of the columns framing the joint and ΣMR_b is the sum of the design values of the moments of resistance of the beams framing the joint. The joints ductility check is not employed for the joints of the top level of multistorey buildings according to section 4.4.2.3(6) of EN 1998-1:2004.

Footings Bearing Capacity

Bearing capacity failure is verified under combinations of applied action effects N_{ed}, V_{Ed}, M_{Ed} according to EN 1998-5, 5.4.1.1 (8).

Footings Sliding Forces

Failure by sliding is verified according to EN 1998-5, section 5.4.1.1 (6) by ensuring that sliding force V_{Ed} on the horizontal base does not exceed the following expression:

F_{Rd} +E_{pd}

where

F_{Rd} is the design friction resistance of footings and

E_{pd} is the design lateral resistance arising from earth pressure on the side of the footings.

Footings Bending Capacity

Bending moment capacity check is performed according to EN 1992-1-1. Bending moment demand is calculated by pure stress σ_{net} acting on the horizontal base of footing.

Footings Shear Capacity

Shear capacity check is performed according to EN 1992-1-1. Shear demand is calculated by pure stress σ_{net} acting on the horizontal base of footing.

Footings Punching Capacity

Punching capacity check is carried out as described in EN 1992-1-1, section 6.4.2.

Footings Eccentricity

Eccentricity of loading should not exceed 1/3 of the dimension in each direction of footing according to EN 1997-1, section 6.5.4. Double eccentricity check is verified if the sum of squares of loading eccentricities in 2 horizontal directions is less than 1/9.

Capacity Curve

Each pushover analysis leads to a capacity curve, which is a relationship between the total base shear and the horizontal displacement of a representative point of the structure, termed “control node”, with the values of the control displacement ranging between zero and a maximum value defined by the user, which should correspond to 150% of the target displacement.

Target Displacement

The target displacement is defined as the seismic demand derived from the elastic response spectrum in terms of displacement of an equivalent single-degree-of-freedom system. To define the target displacement of a MDOF system a number of steps have to be followed according to Annex B of EN1998-1.

The following relation between normalized lateral forces F_i and normalized displacements Φ_i is assumed:

$$F_i = m_i \Phi_i$$

Where m_i is the mass in the i-th storey.

Displacements are normalized in such a way that Φ_n=1, where n is the control node, consequently F_n=m_n.

Transformation to an equivalent Single Degree of Freedom (SDOF) system

The mass of an equivalent SDOF system m* is determined as:

$$m^* = \sum m_i \Phi_i = \sum F_i$$

And the transformation factor is given by:

$$\Gamma = \frac{m^*}{\sum m_i {\Phi_i}^2} = \frac{\sum F_i}{\sum \left(\frac{{F_i}^2}{m_i}\right)}$$

The force F* and displacement d* of the equivalent SDOF system are computed as:

$$F^* = \frac{F_b}{\Gamma}$$

$$d^* = \frac{d_n}{\Gamma}$$

Where F_b and d_n are, respectively, the base shear force and the control node displacement of the Multi Degree of Freedom (MDOF) system.

Determination of the idealized elasto-perfectly plastic force-displacement relationship

The yield force F_y^*, which represents also the ultimate strength of the idealized SDOF system, is equal to the base shear force at the formation of the plastic mechanism. The initial stiffness of the idealized system is determined in such a way that the areas under the actual and the idealized force-deformation curves are equal, as shown in the figure B.1 below:

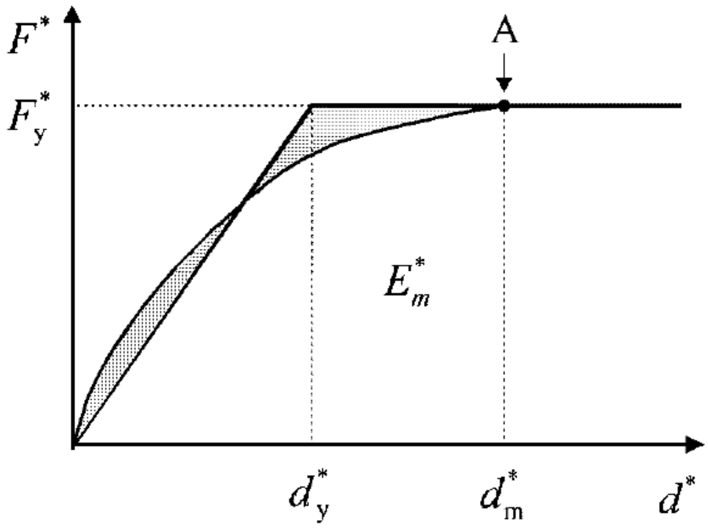


Figure B.1 of EN 1998-1 : Determination of the idealized elasto-perfectly plastic force-displacement relationship

Based on this assumption, the yield displacement on the idealized SDOF system d_y^* is given by:

$$d_y^* = 2\left(d_m^* - \frac{E_m^*}{F_y^*}\right)$$

Where E_m^* is the actual deformation energy up to the formation of the plastic mechanism.

Determination of the period of the idealized equivalent SDOF system

The period T^* of the idealized equivalent SDOF system is determined by:

$$T^* = 2\pi\sqrt{\frac{m^* d_Y^*}{F_Y^*}}$$

Determination of the target displacement for the equivalent SDOF system

The target displacement of the structure with period T* and unlimited elastic behaviour is given by:

$$d_{et}^* = S_e(T^*)\left[\frac{T^*}{2\pi}\right]^2$$

Where $S_e(T^*)$ is the elastic acceleration response spectrum at the period T^*.

For the determination of the target displacement d_t^* for structures in the sort-period range and for structures in the medium and long-period ranges different expressions should be used as indicated below. The corner period between the short- and medium-period range is T_C.

- For $T^*<T_C$ (short period range)

If $F_y^*/m^* \geq S_e(T^*)$, the response is elastic and thus

$$d_t^* = d_{et}^*$$

If $F_y^*/m^* < S_e(T^*)$, the response is nonlinear and

$$d_t^* = \frac{d_{et}^*}{q_u}\left(1 + (q_u - 1)\frac{T_C}{T^*}\right) \geq d_{et}^*$$

Where q_u is the ratio between the acceleration in the structure with unlimited elastic behaviour $S_e(T^*)$ and the structure with limited strength F_y^*/m^*.

$$q_u = \frac{S_e(T^*)m^*}{F_Y^*}$$

d_t^* need not exceed 3 d_{et}^*

- For $T^* \geq T_C$ (medium and long period range)

$$d_t^* = d_{et}^*$$

Determination of target displacement for the MDOF system

The target displacement of the MDOF system is given by:

$$d_t = \Gamma d_t^*$$

Note that the target displacement corresponds to the displacement of the control node.

Appendix A.2 – ASCE

In this appendix the parameters used for the structures assessment according to the American Seismic Evaluation and Retrofit of Existing Buildings Code, ASCE/SEI 41-23, are presented.

Type of Analysis

Current practice in USA is regulated by the ASCE 41-23: Seismic Evaluation and Retrofit of Existing Buildings in combination with ACI 318: Building Code Requirements for Structural Concrete and Commentary, ACI 440.2R-17: Guide for the Design and Construction of Externally Bonded FRP Systems for Strengthening Concrete Structures and ACI 369.1-22: Seismic Evaluation and Retrofit of Existing Concrete Buildings - Code and Commentary.

According to ASCE 41-23, the seismic actions effects in combination with the effects of the permanent and variable loads are evaluated using one of the following methods:

- Linear Static Procedure (LSP) in accordance with section 7.4.1 of ASCE 41-23;
- Linear Dynamic Procedure (LDP) in accordance with section 7.4.2 of ASCE 41-23;
- Nonlinear Static Procedure (NSP) according to section 7.4.3 of ASCE 41-23;
- Nonlinear Dynamic Procedure (NDP) according to section 7.4.4 of ASCE 41-23.

In SeismoBuild the most common method in assessment practice of existing buildings is employed, which is the nonlinear static analysis. It is based on pushover analyses carried out under constant gravity loads and increasing lateral forces, applied at the location of the masses to simulate the inertia forces induced by the seismic action. As the model may account for both geometrical and mechanical nonlinearity, this method can describe the evolution of the expected plastic mechanisms and structural damage.

Each pushover analysis leads to a capacity curve, which is a relationship between the total base shear and the horizontal displacement of a representative point of the structure, termed “control node”. The demand at the considered Performance Level – Operational Level, Immediate Occupancy, Life Safety or Collapse Prevention - is determined by the appropriate comparison between the capacity determined by the pushover curve and the demand established as the damped Linear Response Spectrum. To do so, the “control node” displacements are defined in terms of spectral quantities relative to an equivalent single-degree-of-freedom (SDOF) system which is derived from the multi-degree-of-freedom (MDOF) response estimated according to section 7.4.3.3 of ASCE 41-23.

The structural demand associated with the acquired target displacement shall fulfil the verification criteria defined in ASCE 41-23. Accordingly, element’s demand for brittle (shear) and ductile (plastic hinge or chord rotation) actions are deemed to comply with limits that take into account: section mechanical properties; element’s bending, shear and axial force interaction; and strength/stiffness degradation associated with the ductility demand and cyclic hysteretic response of reinforced concrete elements, through appropriate material nonlinearity consideration.

Performance Requirements

According to ASCE 41-23 section 2.4, the objectives of the assessment or redesign (Table C2-8) consist of combinations of both a performance level and a seismic action, given an "acceptable probability of exceedance within the life cycle of the building" (design earthquake), as shown in Table C2-8 of ASCE 41-23 below.

Seismic Hazard Level	Target Building Performance Levels			
	Operational Performance Level (1-A)	Immediate Occupancy Performance Level (1-B)	Life Safety Performance Level (3-C)	Collapse Prevention Performance Level (5-D)
50%/50years	a	b	c	d
BSE-1E (20%/50 years)	e	f	g	h
BSE-2E (5%/50 years)	i	j	k	l
BSE-2N (2%/50 years)	m	n	o	p

Table C2-8. Performance Objectives

The target building performance levels refer to the state of damage in the structure defined through four limit states, namely Operational Level (1-A), Immediate Occupancy (1-B), Life Safety (3-C) and Collapse Prevention (5-D).

Performance Level of Operational Level (1-A)

The Operational Level (1-A), according to ASCE 41-23, is a condition in which it is expected that damage is insignificant and structure does not need any repair measures. Structural elements are prevented from significant yielding and retaining their strength and stiffness properties. All systems important to normal operation are functional. Non-structural components, such as partitions and infills should not be damaged.

Performance Level of Immediate Occupancy (1-B)

The Immediate Occupancy after the earthquake (1-B), according to ASCE 41-23, is a condition in which it is expected that no building operation is interrupted during and after the design earthquake, with the possible exception of minor importance functions. Structural elements are retaining their strength and stiffness properties. A few hairline cracks may occur in the structure.

Performance Level of Life Safety (3-C)

The Life Safety (3-C), according to ASCE 41-23, is a condition in which moderate damage to the structure is expected to occur during the design earthquake, although it is likely to be uneconomic to repair. Structural elements are retaining some residual strength and stiffness. Non-structural components are damaged, although partitions and infills have not failed out-of-plane. Moderate permanent drifts are present.

Performance Level of Collapse Prevention (5-D)

The Collapse Prevention (5-D), according to ASCE 41-23, is a condition in which severe (non-repairable, in general) damage to the structure is expected during the design earthquake and would probably not survive another earthquake. The structure is heavily damaged with low residual lateral strength and stiffness, although vertical elements are still capable of sustaining vertical loads. Most non-structural components have collapsed and large permanent drifts are present.

The criteria for the selection of the Performance Objectives may be found in ASCE 41-23.

Information for Structural Assessment

In order to choose the admissible type of analysis and the appropriate confidence factor values, the following three knowledge levels are defined:

- Minimum Knowledge
- Usual Knowledge
- Comprehensive Knowledge

The factors determining the obtained data reliability level are the (i) geometry, which is the geometrical properties of the structural system and of non-structural elements, e.g. masonry infill panels, that may affect structural response; (ii) details, which include the amount and detailing of reinforcement in reinforced concrete sections, the connection of floor diaphragms to lateral resisting structure, the bond and mortar jointing of masonry and the nature of any reinforcing elements in masonry; and finally (iii) materials, that is the mechanical properties of the constituent materials.

Minimum Knowledge

The minimum data collection requirements correspond to a state of knowledge where information is obtained from design drawings with sufficient information to analyse component demands and calculate component capacities. The design drawings show the configuration of the gravity load system and seismic-force-resisting system with sufficient details. Information is verified by a visual condition assessment.

In the absence of sufficient information from design drawings, incomplete or non-existent information is supplemented by a comprehensive condition assessment, including destructive and non-destructive investigation. In the absence of material test records and quality assurance reports default material properties are used according to chapter 10 of ASCE 41-23.

Usual Knowledge

The usual data reliability level corresponds to a state of knowledge where information is obtained from design drawings with sufficient information to analyse component demands and calculate component capacities. The design drawings show the configuration of the gravity load system and seismic-force-resisting system with sufficient details. Information is verified by a visual condition assessment.

In the absence of sufficient information from design drawings, incomplete or non-existent information is supplemented by a comprehensive condition assessment, including destructive and non-destructive investigation. In the absence of material test records and quality assurance reports default material properties are used according to chapter 10 of ASCE 41-23.

Comprehensive Knowledge

The comprehensive data reliability level corresponds to a state of knowledge where information is obtained from construction documents including design drawings, specifications, material test records, and quality assurance reports covering original construction and subsequent modifications to the structure. Information is verified by a visual condition assessment.

In cases where construction documents are incomplete, missing information is supplemented by comprehensive condition assessment, including destructive and non-destructive investigation. In the absence of material test records and quality assurance reports, material properties are determined by comprehensive materials testing in accordance to chapter 10 of ASCE 41-23.

Knowledge Factors

In the following table of ASCE 41-23 a summary and recommendations for the confidence factors and the analysis methods are provided for each knowledge level.

Material Strength Specified on Construction Documents	Material Testing Performed	Material Strength Used in Evaluation	Knowledge Factor (κ)
Yes	None	Values specified in the construction documents	0.9
Yes	Usual	The values specified in the construction documents unless the minimum material testing result is less than 85% of the values specified on the construction documents	1.0
Yes	Comprehensive	Values based on the material testing results	1.0
No	None	Default values in Chapters 9 through 12	0.75
No	Usual	The default values in Chapters 9 through 12 unless the minimum material testing result is less than 85% of the default value	1.0
No	Comprehensive	Values based on the material testing results	1.0

Table 6-1 of ASCE

According to Section 6.2.3.2 of ASCE 41-23, where nonlinear procedures are used, data collection consistent with either the usual or comprehensive levels of knowledge shall be performed.

Safety Factors

In ASCE 41-23 the safety factors are directly incorporated in the member's strengths and deformation limits.

In members where the longitudinal spacing of transverse reinforcement does not exceed 75% of the component effective depth measured in the direction of shear, transverse reinforcement shall be assumed 100% effective in resisting shear. Also, in members where the longitudinal spacing of transverse reinforcement exceeds the component effective depth measured in the direction of shear, transverse reinforcement shall be assumed ineffective in resisting shear. Linear interpolation should be executed in other case. Users may decide in the Safety Factors dialog box, whether to enforce such very strict rule or not. For more information on this rule, users may refer to section 4.2.3 of ACI 369.1-22.

Capacity Models for Assessment and Checks

All the member checks (chord rotation capacity and shear capacity) should be carried out for all the elements of every floor, according to Chapter 10 of ASCE 41-23, Chapter 11 of ACI 318-19, Chapter 4 of ACI 369.1-22 and Chapter 11 of ACI 440, taking into account Tables 7-6 and 7-7 of ASCE 41-23. Interstorey drift ratio should be checked for walls controlled by shear. Moreover, beam-column joints checks can be employed in order to check the joint's shear force.

Deformation Capacity

The deformation capacity of beams, columns and walls controlled by flexure is defined in terms of the total chord rotation θ, that is the angle between the tangent to the axis at the yielding end and the chord connecting that end with the end of the shear span (L_V=M/V=moment/shear at the end section). The chord rotation is also equal to the element drift ratio, which is the deflection at the end of the shear span with respect to the tangent to the axis at the yielding end divided by the shear span.

Deformation capacity of beams, columns and walls controlled by flexure is highly influenced by the lack of appropriate seismic resistant detailing in longitudinal reinforcement, as well as whether there are smooth bars. Inadequate development of splicing along the span (beams) and height (columns) and inadequate embedment into beam-column joints can control the members' response to seismic action, drastically limiting its capacity, in respect to the situation in which the reinforcement is considered fully effective. The above limitations to the deformation capacity are taken into consideration.

The total chord rotation capacity at ultimate of concrete members under cyclic loading is calculated as the sum of the chord rotation at yielding and the plastic part of the chord rotation capacity

$$\theta = \theta_y + \theta_p$$

The chord rotation capacity at yield, θ_y, is calculated as described below:

- For beams and columns from the equation (4.29) of D.Biskinis (2007):

  $$\theta_y = \frac{M_y L_s}{3EI_{eff}}$$

  where the effective stiffness value, EI_{eff}, is calculated according to Table 3.1.2.1 of ASCE 41-23.

- For walls from equation (7.4.1.1.1) of ASCE 41-23:

  $$\theta_{yE} = \left(\frac{M_{CyGE}}{(E_{cE}I)_{eff}}\right) l_p \qquad \text{(7.4.1.1.1) ASCE 41-23}$$

The plastic part of the chord rotation capacity is calculated as indicated below:

- For beams according to Table 4.2.2.2.2a of ACI 369.1-22
- For columns according to Tables 4.2.2.2.2b and 4.2.2.2.2c of ACI 369.1-22
- For walls controlled by flexure according to Table 7.4.1.1.1 of ASCE 41-23 and for walls controlled by shear according to Table 7.4.1.1.2 of ASCE 41-23

The deformation capacity of walls controlled by shear is defined in terms of the interstorey drift ratio as indicated in Table 7.4.1.1.3 of ASCE 41-23.

The yield moment capacity is calculated according to the equations of Appendix 7A of KANEPE.

Users are advised to refer to the relevant publications for the definition of the other parameters and further details on the expressions.

FRP wrapping

The contribution of the FRP wrapping to members' capacity is taken into account in the calculation of the yield moment capacity.

Bending Moment Capacity

The bending moment capacity of beams, columns and walls with prismatic cross section is calculated according to paragraph 4.1 of D. Biskinis and M. N. Fardis (2009), while for the bending moment capacity of circular columns paragraph 4 of Biskinis and M. N. Fardis (2013) is employed.

Shear Capacity

The Shear capacity of columns is calculated through the following expression according to section 4.2.3.1 of ACI 369.1-22.

$$V_{Col} = k_{nl} V_{Col0} = k_{nl}\left[\alpha_{Col}\left(\frac{A_v f_{ytL/E} d}{s}\right) + \lambda\left(\frac{6\sqrt{f'_{cL/E}}}{M_{UD}/V_{UD}d}\sqrt{1+\frac{N_{UG}}{6A_g\sqrt{f'_{cL/E}}}}\right)0.8A_g\right] \text{ (lb/in.}^2\text{ units)}$$

(4.2.3.1) ACI 369.1-22

$$V_{Col} = k_{nl}V_{Col0} = k_{nl}\left[\alpha_{Col}\left(\frac{A_v f_{ytL/E} d}{s}\right) + \lambda\left(\frac{0.5\sqrt{f'_{cL/E}}}{M_{UD}/V_{UD}d}\sqrt{1+\frac{N_{UG}}{0.5A_g\sqrt{f'_{cL/E}}}}\right)0.8A_g\right] \text{(Mpa units)}$$

(4.2.3.1 si) ACI 369.1-22

The shear strength of a shear wall is calculated from the following expression:

$$V_n = A_{cv}\left(\alpha_c\lambda\sqrt{f'_c} + \rho_t f_y\right) \qquad \text{(18.10.4.1) ACI 318-19}$$

The value for V_n at any horizontal section for shear in plane of wall shall not be taken greater than $0.83\sqrt{f'_c}hd$ according to section 11.5.4.3 of ACI 318-19.

The shear capacity of beam sections is calculated from the equation (22.5.1.1) of ACI 318-19, with the shear strength provided by the transverse reinforcement computed from equation (22.5.8.5.3) of ACI 318-19 and the shear strength provided by concrete computed by the table 22.5.5.1 of ACI 318-19.

Criteria	V_c		
$A_v \geq A_{v,min}$	Either of:	$\left[0.17\lambda\sqrt{f'_c} + \frac{N_u}{6A_g}\right]b_w d$	(a)
		$\left[0.66\lambda(\rho_w)^{1/3}\sqrt{f'_c} + \frac{N_u}{6A_g}\right]b_w d$	(b)
$A_v < A_{v,min}$	$\left[0.66\lambda_s\lambda(\rho_w)^{1/3}\sqrt{f'_c} + \frac{N_u}{6A_g}\right]b_w d$		(c)

Table 22.5.5.1 of ACI 318-19

Users are advised to refer to the relevant publications for the definition of the other parameters and further details on the expressions.

FRP wrapping

The shear resistance V_n, may be calculated from expression (4.2.3.1) of ACI 369.1-22 for columns or the equation (22.5.1.1) of ACI 318-19 for beams and shear walls adding in V_s the contribution of the FRP jacket to the shear resistance.

The contribution of the FRP jacket to the shear resistance is computed through the following expression multiplied by a reduction factor ψ_f, as described in section 11.4 of ACI 440:

$$V_f = \frac{A_{fv} f_{fe}(\text{sina}+\text{cosa})d_{fv}}{s_f} \qquad \text{(11.4a) ACI 440}$$

where

$$A_{fv} = 2nt_f w_f \qquad \text{(11.4b) ACI 440}$$

and

$$f_{fe} = \varepsilon_{fe}E_f \qquad \text{(11.4d) ACI 440}$$

The total shear strength provided by the sum of the FRP shear reinforcement and the steel shear reinforcement should be limited as indicated in the equation below:

$V_s + V_f \leq 8\sqrt{f_c'}b_w d$ in in-lb units

$V_s + V_f \leq 0.66\sqrt{f_c'}b_w d$ in SI units (11.4.3) ACI 440

Users are advised to refer to the relevant publications for the definition of the other parameters and further details on the expressions.

Steel Braces Axial Deformations

Axial deformations of brace members in tension and compression should satisfy the provisions of table C3.4 of AISC 342-22.

Steel Braces Axial Forces

Axial forces of brace members in tension and compression should satisfy the provisions of table C3.2 of AISC 342-22 and chapters D and E of AISC360-16.

Joints Shear Force

The equation of section 4.2.3.2.2 of ACI 369.1-22 is employed for the calculation of the shear capacity of joints:

$V_J = \lambda\gamma\sqrt{f'_{cL/E}}A_j$ (lb/in.² units) (4.2.3.2.2a) ACI 369.1-22

$V_J = 0.083\lambda\gamma\sqrt{f'_{cL/E}}A_j$ (MPa units) (4.2.3.2.2a.si) ACI 369.1-22

The value for γ is defined in Table 4.2.3.2.2 of ACI 369.1-22.

Users are advised to refer to the relevant publications for the definition of the other parameters and further details on the expressions.

Joints Ductility

Adequate ductility should be possessed by both the structural elements and the structure as a whole according to section 18.7.3 of EN ACI 318-19. In frame buildings with two or more storeys, the following condition should be satisfied at all joints of primary or secondary seismic beams with primary seismic columns:

$\Sigma M_{nc} \geq (6/5)\Sigma M_{nb}$ (18.7.3.2) ACI 318-19

Where ΣM_{nc} is the sum of the design values of the moments of resistance of the columns framing the joint and ΣM_{nb} is the sum of the design values of the moments of resistance of the beams framing the joint. The joints ductility check is not employed for the joints of the top level of multistorey buildings according to section 18.7.3.1 of ACI 318-19.

Footings Rocking Moment Capacity

Rocking moment capacity is verified according to ASCE 41-23, section 8.4.5.2.

Footings Rocking Rotation Capacity

Rocking rotation capacity is verified according to ASCE 41-23, section 8.4.5.3.

Footings Bending Capacity

Bending moment capacity check is performed according to ACI 318-19. Bending moment demand is calculated by pure stress σnet acting on the horizontal base of footing.

Footings Shear Capacity

Shear capacity check is performed according to ACI 318-19, section 22.5.5.1. Shear demand is calculated by pure stress σ_{net} acting on the horizontal base of footing.

Footings Punching Capacity

Punching capacity check is carried out as described in ACI 318-19, section 22.6.5.

Footings Eccentricity

Eccentricity of loading should not exceed 1/3 of the dimension in each direction of footing according to EN 1997-1, section 6.5.4. Double eccentricity check is verified if the sum of squares of loading eccentricities in 2 horizontal directions is less than 1/9.

Capacity Curve

Each pushover analysis leads to a capacity curve, which is a relationship between the total base shear and the horizontal displacement of a representative point of the structure, termed "control node", with the values of the control displacement ranging between zero and a maximum value defined by the user.

Target Displacement

The target displacement δ_t (§ 7.4.3.3 of ASCE 41-23) shall be calculated taking into account all the relevant factors affecting the displacement of a building that responds inelastically. It is permitted to consider the displacement of an elastic single degree of freedom system with a fundamental period equal to the fundamental period of the building that is subjected to the seismic actions, for which the verification is made. An appropriate correction is needed in order to derive the corresponding displacement of the building assumed to be responding as an elastic-perfectly plastic system.

For buildings with rigid diaphragms at each floor level, the target displacement shall be calculated in accordance with equation (7-29) of ASCE 41-23 or by an approved procedure that accounts for the nonlinear response of building.

$$\delta_t = C_0 C_1 C_2 S_\alpha \left(\frac{T_e^2}{4\pi^2}\right) g \qquad \text{(7-29) ASCE 41-23}$$

where S_α is the response spectrum acceleration at the effective fundamental period and damping ratio of the building in the direction under consideration, as calculated in Sections 2.3.1 or 2.3.3 of ASCE 41-23, and C_0, C_1 and C_2 are modification factors that are defined as follows:

C_0: Modification factor that relates the spectral displacement of the equivalent single degree of freedom (SDOF) system with the roof displacement of the building multi degree of freedom (MDOF) system calculated using the appropriate value from Table 7-5.

Number of Stories	Shear Buildings		Other Buildings
	Triangular Load Pattern (1.1, 1.2, 1.3)	Uniform Load Pattern (2.1)	Any Load Pattern
1	1.0	1.0	1.0
2	1.2	1.15	1.2
3	1.2	1.2	1.3

5	1.3	1.2	1.4
10+	1.3	1.2	1.5

Table 7-5 of ASCE 41-23: Values for Modification Factor C_0

C_1: Modification factor to relate expected maximum inelastic displacements to displacements calculated for linear elastic response. For periods less than 0.2s, C1 need not be taken greater than the value at T=0.2s.

C_1=1.0 for T ≥ 1s, and

$$C_1 = 1 + \frac{\mu_{strength} - 1}{\alpha T_e^2} \text{ for } 0.2s \leq T < 1s, \qquad \text{(7-30) ASCE 41-23}$$

where α is the site class factor (is equal to 130 for site class A or B, 90 for site class C and 60 for site class D, E, or F), T_e is the fundamental period of the building in the direction under consideration and $\mu_{strength}$ is the ratio of the elastic strength demand to yield strength coefficient calculated in accordance with equation (7-32) of ASCE 41-23.

C_2: Modification factor to represent the effect of pinched hysteresis shape, cyclic stiffness degradation, and strength deterioration on the maximum displacement response. For periods greater than 0.7, C_2=1.0;

$$C_2 = 1 + \frac{1}{800}\left(\frac{\mu_{strength} - 1}{T_e}\right)^2 \qquad \text{(7-31) ASCE 41-23}$$

Where the strength ratio $\mu_{strength}$ is calculated according to the following equation:

$$\mu_{strength} = \frac{S_\alpha}{V_y/W} C_m \qquad \text{(7-32) ASCE 41-23}$$

C_m is the effective mass factor with values according to Table 7-4 of ASCE 41-23.

Users are advised to refer to the Code for the definition of the other parameters and further details on the expressions.

Determination of the idealized elasto-perfectly plastic force-displacement relationship

The nonlinear force-displacement relationship that relates the base shear with the displacement of the control node shall be replaced by an idealized curve for the determination of the equivalent lateral stiffness K_e and the corresponding yield strength V_y of the building.

It is recommended that the idealized capacity curve (force-displacement relationship) is bilinear, with a slope of the first branch equal to K_e and a slope of the second branch equal $\alpha_1 K_e$. The two lines that compose the bilinear curve can be defined graphically, on the criterion of approximately equal areas of the sections defined above and below the intersection of the actual and the idealized curves (Figure 7-3 of ASCE 41-23).

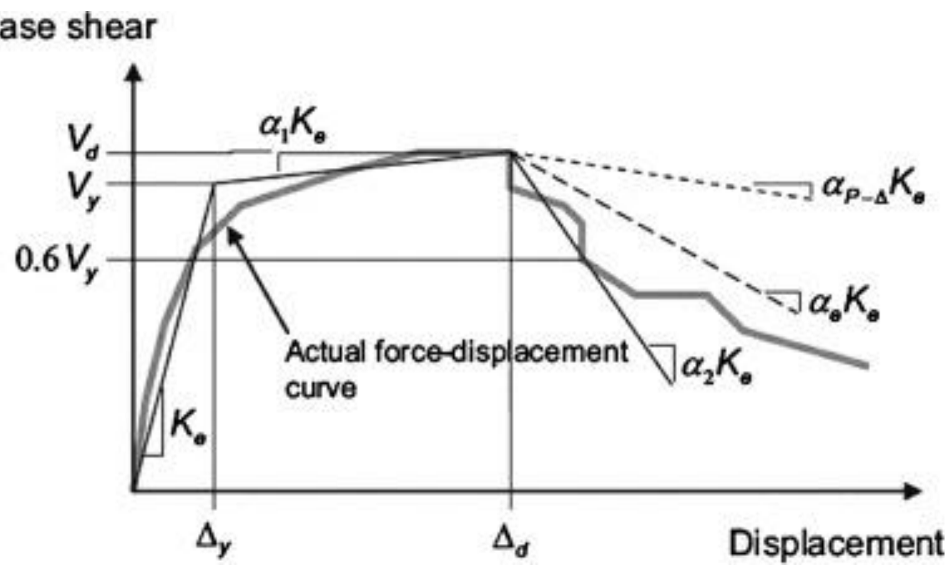


Figure 7-3 of ASCE 41-23 Idealized Force -_Displacement Curve

The equivalent lateral stiffness K_e is determined as the secant stiffness that corresponds to a base shear force equal to the 60% of the effective yield strength V_y, the latter defined by the intersection of the lines above. The normalized inclination (α_1) of the second branch is determined by a straight line passing through the point (V_d, Δ_d) and a point at the intersection with the first line segment such that the areas above and below the actual curve are approximately balanced. (V_d, Δ_d) shall be a point on the actual force-displacement curve at the calculated target displacement, or at the displacement corresponding to the maximum shear, whichever is least.

Determination of the fundamental period

The effective fundamental period in the direction under consideration shall be estimated based on the idealized force-displacement curve.
The value T_e of the effective fundamental period is derived by the following expression:

$$T_e = T_i\sqrt{\frac{K_i}{K_e}} \qquad \text{(7-28) ASCE 41-23}$$

where T_i is the elastic fundamental period in the direction under consideration, and is derived by eigenvalue analysis, K_i is the corresponding elastic lateral stiffness, and K_e is the effective lateral stiffness.

Appendix A.3 – NTC-18

In this appendix the parameters used for the structures assessment according to the Italian National Seismic Code – NTC-18 are presented.

TYPE OF ANALYSIS

Current practice in Italy in structural assessment is regulated by the Italian National Seismic Code – NTC-18.

According to NTC-18, the seismic actions effects in combination with the effects of the permanent and variable loads are evaluated using one of the following methods:

- Linear elastic analysis;
- Plastic analysis;
- Nonlinear analysis.

The nonlinear static analysis is the reference method in assessment practice of existing buildings and is the one employed in SeismoBuild. It is based on pushover analyses carried out under constant gravity loads and increasing lateral forces, applied at the location of the masses to simulate the inertia forces induced by the seismic action. As the model may account for both geometrical and mechanical nonlinearity, this method can describe the evolution of the expected plastic mechanisms and structural damage.

Each pushover analysis leads to a capacity curve, which is a relationship between the total base shear and the horizontal displacement of a representative point of the structure, termed "control node". The demand at the considered Limit State – Operational Level, Damage Limitation, Life Safety or Collapse Prevention - is determined by the appropriate comparison between the capacity determined by the pushover curve and the demand established as the damped Linear Response Spectrum. To do so, the "control node" displacements are defined in terms of spectral quantities relative to an equivalent single-degree-of-freedom (SDOF) system which is derived from the multi-degree-of-freedom (MDOF) response estimated according to C7.3.4 of NTC-18.

The structural demand associated with the acquired target displacement shall fulfil the verification criteria defined in NTC-18. Accordingly, element's demand for brittle (shear) and ductile (chord rotation deformation) actions are deemed to comply with limits that take into account: section mechanical properties; element's bending, shear and axial force interaction; and strength/stiffness degradation associated with the ductility demand and cyclic hysteretic response of reinforced concrete elements, through appropriate material nonlinearity consideration.

PERFORMANCE REQUIREMENTS

According to NTC-18, performance requirements refer to the state of damage in the structure defined through four limit states, namely Operational Level (SLC), Damage Limitation (SLD), Life Safety (SLV) and Collapse Prevention (SLC).

Limit State of Collapse Prevention (SLC)

The limit state of Collapse Prevention (SLC) may be selected, according to NTC-18, where the structure, after the earthquake, undergoes serious cracks and collapses of the non-structural components and equipment and very serious damage of structural components. The building still retains a significant stiffness and resistance against vertical loads and a small safety margin against collapse from horizontal actions. The appropriate level of protection is achieved by choosing a seismic action with a return period of 975 years corresponding to a probability of exceedance of 5% in 50 years.

Limit State of Life Safety (SLV)

The limit state of Life Safety (SLV) may be selected, according to NTC-18, where the building after the earthquake undergoes cracks and collapses of the non-structural components and equipment and significant damage to structural components associated with a significant loss of stiffness against horizontal actions; the construction preserves part of the strength and stiffness for vertical actions and a safety margin against collapse for horizontal seismic actions. The appropriate level of protection is achieved by choosing a seismic action with a return period of 475 years corresponding to a probability of exceedance of 10% in 50 years.

Limit State of Damage Limitation (SLD)

The limit state of Damage Level (SLD) may be selected, according to NTC-18, where the building after the earthquake as a whole, including the structural and non-structural elements, as well as the equipment relevant to its function, has damage that does not compromise significantly the ability of resistance and rigidity against vertical and horizontal actions. The structure remains immediately usable despite the interruption of use of part of the equipment. The appropriate level of protection is achieved by choosing a seismic action with a return period of 50 years corresponding to a probability of exceedance of 63% in 50 years.

Limit State of Operational Level (SLO)

The limit state of Operational Level (SLO) may be selected, according to NTC-18, where the building as a whole, including the structural and non-structural components, as well as the equipment relevant to its function, should not be damaged or interrupted its function after the earthquake. The appropriate level of protection is achieved by choosing a seismic action with a return period of 30 years corresponding to a probability of exceedance of 81% in 50 years.

Information for Structural Assessment

In order to choose the admissible type of analysis and the appropriate confidence factor values, the following three knowledge levels are defined:

- KL1: Limited Knowledge
- KL2: Adequate Knowledge
- KL3: Accurate Knowledge

The factors determining the obtained knowledge level are the (i) geometry, which is the geometrical properties of the structural system and of non-structural elements, e.g. masonry infill panels, that may affect structural response; (ii) details, which include the amount and detailing of reinforcement in reinforced concrete sections, the connection of floor diaphragms to lateral resisting structure, the bond and mortar jointing of masonry and the nature of any reinforcing elements in masonry; and finally (iii) materials, that is the mechanical properties of the constituent materials.

KL1: Limited Knowledge

The limited knowledge level corresponds a state of knowledge where the overall structural geometry and member sizes are known from survey or from original outline construction drawings used for both the original construction and any subsequent modifications, as well as a sufficient sample of dimensions of both overall geometry and member sizes checked on site. In case of significant discrepancies from the outline construction drawings a fuller dimensional survey is performed. The structural details are not known from detailed construction drawings and are assumed based on simulated design in accordance with usual practice at the time of construction. Limited inspections performed in the most critical elements should prove that the assumptions correspond to the actual situation. Information on the mechanical properties of the construction materials isn't available so default values are assumed in accordance with standards at the time of construction accompanied by limited in-situ testing in the most critical elements.

Structural evaluation based on this state of knowledge is performed through linear analysis methods, either static or dynamic.

KL2: Adequate Knowledge

The adequate knowledge level corresponds to a state of knowledge where the overall structural geometry and member sizes are known from extended survey or from outline construction drawings used for both the original construction and any subsequent modifications, as well as a sufficient sample of dimensions of both overall geometry and member sizes. The structural details are known from an extended in-situ inspection or from incomplete detailed construction drawings in combination with limited in-situ inspections in the most critical elements, which confirms that the available information corresponds to the actual situation. Information on the mechanical properties of the construction materials is available from extended in-situ testing or from original design specifications and limited in-situ testing.

Structural evaluation based on this state of knowledge is performed through linear or nonlinear analysis methods, either static or dynamic.

KL3: Accurate Knowledge

The accurate knowledge level corresponds to a state of knowledge where the overall structural geometry and member sizes are known from a comprehensive survey or from the complete set of outline construction drawings used for both the original construction and subsequent modifications, as well as a sufficient sample of both overall geometry and member sizes checked on site. The structural details are known from comprehensive in-situ inspection or from a complete set of detailed construction drawings in combination with limited in-situ inspections in the most critical elements, which prove that the available information corresponds to the actual situation. Information on the mechanical properties of the construction materials is available from comprehensive in-situ testing or from original test reports and limited in-situ testing.

Structural evaluation based on this state of knowledge is performed through linear or nonlinear analysis methods, either static or dynamic.

Confidence Factors

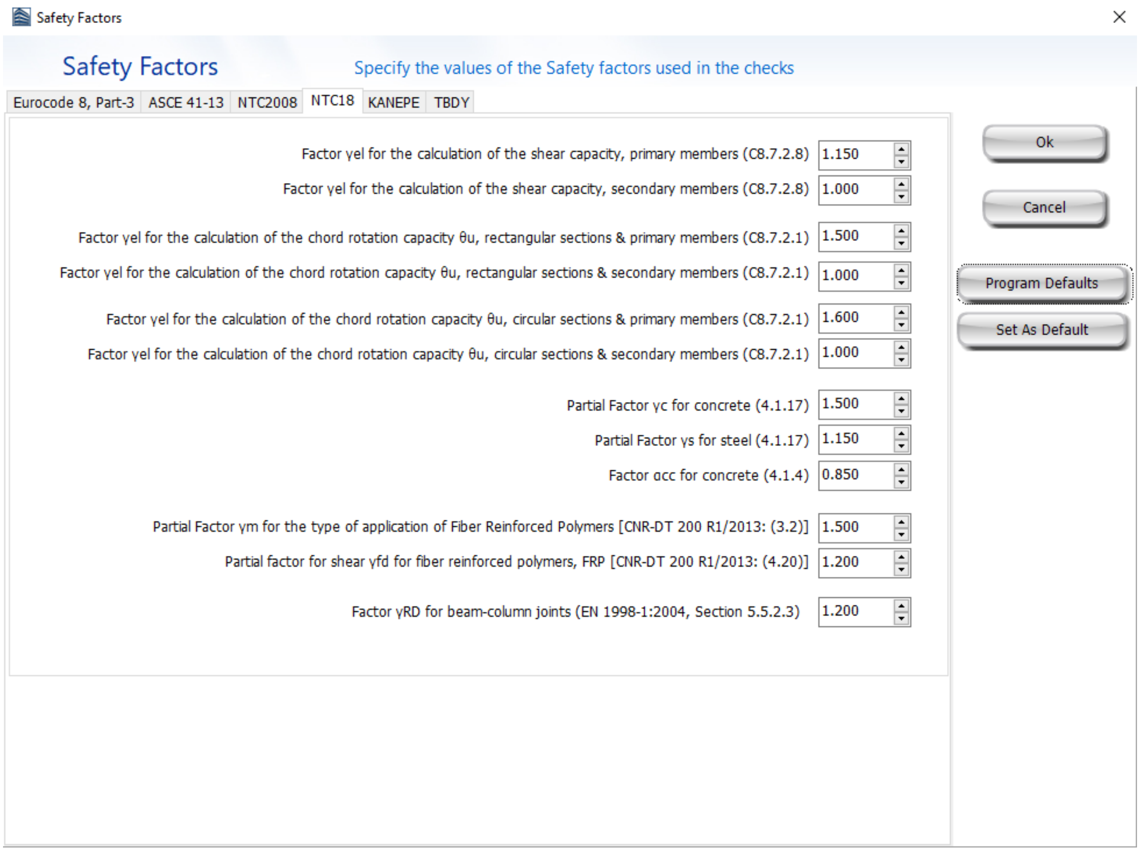
In the following table of section C8.5.4.2 of the commentary of NTC-18 a summary and recommendations for the confidence factors and the analysis methods are provided for each knowledge level.

Level of knowledge	Geometry (structural work)	Structural details	Materials Properties	Methods of analysis	FC
LC1	From original outline construction drawings with sample visual survey or from full survey	Project simulated in compliance with standards era and limited checks in-situ	Usual values for building practice era and limited evidence in-situ	linear analysis static or dynamic	1.35
LC2		Construction drawings incomplete with limited checks situ or extensive checks in-situ	From the specifications original project or by test certificates original with limited evidence in-situ or extensive testing in-situ	All	1.20
LC3		Construction drawings full with limited checks situ or exhaustive testing in-situ	By test certificates original or from original specifications of project with extensive tests in situ or extensive testing in-situ	All	1.00

Table C8.5.IV of commentary of NTC-18- Knowledge Levels as a function of the available information, consequent methods of analysis and allowed values of the confidence factors for reinforced concrete or steel buildings

Safety Factors

The values of the safety factors and the Code expressions employed may be specified through the dialog box that opens from the corresponding button. It is noted the default values of the safety factors are those defined in NTC-18.



Safety Factors module

CAPACITY MODELS FOR ASSESSMENT AND CHECKS

All the member checks (chord rotation capacity and shear capacity) should be carried out for all the elements of every floor, according to section 4.1.2.3.5 of NTC-18 and section C8.7.2.3 of the commentary, considering the members as primary or secondary (section 7.2.3 of NTC-18) seismic elements. Moreover, beam-column joints checks can be employed in order to check (i) the joint's diagonal tension and (ii) the joint's diagonal compression. Finally, interstorey drift checks may be carried out, when needed, for the vertical elements of every floor, according to section 7.3.6.1 of NTC-18.

Deformation Capacity

The deformation capacity of beams, columns and walls is defined in terms of the chord rotation θ, that is the angle between the tangent to the axis at the yielding end and the chord connecting that end with the end of the shear span (L_V=M/V=moment/shear at the end section). The chord rotation is also equal to the element drift ratio, which is the deflection at the end of the shear span with respect to the tangent to the axis at the yielding end divided by the shear span.

Deformation capacity of beams and columns is highly influenced by the lack of appropriate seismic resistant detailing in longitudinal reinforcement, as well as by the bars type, that is whether there are smooth bars. Inadequate development of splicing along the span (beams) and height (columns); and inadequate embedment into beam-column joints can control the members' response to seismic action, drastically limiting its capacity in respect to the situation in which the reinforcement is considered fully effective. The above limitations to the deformation capacity are taken into consideration.

The value for the chord rotation capacity for the limit state of collapse prevention (SLC) is the value of the total chord rotation capacity at ultimate of concrete members under cyclic loading, which is calculated from the following expression:

For beams and columns:

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

(8.7.2.1) commentary of NTC-18

Where γ_{el} is equal to 1,5 for primary seismic elements and to 1,0 for secondary seismic ones; L_V is the ratio between bending moment, M, and shear force, V. The remaining relevant parameters are defined in section C8.7.2.3.2 of the commentary of NTC-18.

For the wall elements the value given in the expression above must be divided by 1.6.

The chord rotation capacity corresponding to the limit state of life safety (SLV) is assumed to be ¾ of the ultimate chord rotation, calculated from the equation above.

The capacity that corresponds to the limit states of operational level (SLO) and of damage limitation (SLD) is given by the chord rotation at yielding, evaluated as:

For beams and columns:

$$\theta_y = \varphi_y \frac{L_V}{3} + 0{,}0013\left(1 + 1{,}5\frac{h}{L_V}\right) + 0{,}13\varphi_y \frac{d_b f_y}{\sqrt{f_c}}$$ (8.7.2.7a) commentary of NTC-18

For walls:

$$\theta_y = \varphi_y \frac{L_V}{3} + 0{,}002\left(1 - 0{,}125\frac{L_V}{h}\right) + 0{,}13\varphi_y \frac{d_b f_y}{\sqrt{f_c}}$$ (8.7.2.7b) commentary of NTC-18

The relevant parameters are defined in section C8.7.2.3.4 of the commentary of NTC-18.

The yield curvature of the end section is calculated according to the following expression for the sections whose compressive zone is of constant width and for the case that the section's yielding is due to steel yielding.

$$\varphi_y = (1/r)_y = \frac{f_y}{E_s(1-\xi_y)d}$$

If the section yields due to the deformation nonlinearities of the concrete in compression, that is for deformation of the edge compressive fibre larger than $\varepsilon_c \approx 1.8\, f_c/E_c$, then the yield curvature is calculated according to the following expression:

$$\varphi_y = (1/r)_y = \frac{\varepsilon_c}{\xi_y d} \approx \frac{1.8 f_c}{E_c \xi_y d}$$

The lower value from the above calculations is used for the calculation of the chord rotation capacity.

According to section C8.7.2.3.2 of the commentary of NTC-18 the chord rotation capacity is highly influenced by a number of different factors such as the type of the longitudinal bars. If smooth (plain) longitudinal bars are applied, the ultimate chord rotation should be multiplied by the factor calculated from equation 8.7.2.4 of the commentary of NTC-18, taking, also, into consideration whether the longitudinal bars are well lapped or not by employing the factor of 8.7.2.3. In case of members with lack of appropriate seismic resistant detailing the ultimate chord rotation capacity is multiplied by 0,85.

In the case of circular column sections, the equations above cannot be employed for the calculation of the elements' chord rotation capacity. In SeismoBuild the equations below suggested by D. Biskinis and M. N. Fardis [2013] are employed for θ_y and θ_u.

$$\theta_y = \varphi_y \frac{L_V + \alpha_V z}{3} + 0.0027\left(1 - \min\left(1; \frac{2}{15}\frac{L_s}{D}\right)\right) + \alpha_{sl}\frac{\varphi_y d_{bL} f_y}{8\sqrt{f_c}}$$

Where f_y and f_c values are in MPa, α_V=1 if V_{Rc}<V_{My}, V_{Rc} is calculated according to Eurocode 2 (CEN 2004), otherwise α_V=0, and α_{sl}=0 if pull-out of the tension bars from their anchorage zone beyond the yielding end is physically impossible, otherwise α_{sl}=1.

$$\theta_u = \left(\theta_y + \left(\varphi_u - \varphi_y\right)L_{pl}\left(1 - 0.5\,L_{pl}/L_s\right) + \alpha_{sl}\Delta\theta_{u,slip}\right)/\gamma_{el}$$

Where γ_{el} is equal to 2.0 for primary seismic elements and to 1.0 for secondary seismic elements, $\Delta\theta_{u,slip}$ and L_{pl} are calculated according to the following equations:

$$\Delta\theta_{u,slip} = 10 d_{bl}\left(\varphi_u + \varphi_y\right)/2$$

$$L_{pl} = 0.6D\left[1 + \frac{1}{6}\min\left(9; \frac{L_s}{D}\right)\right]$$

Users are advised to refer to the relevant publications for the definition of the other parameters and further details on the expression.

Concrete Jacketing

The values of the jacketed members for M_y^*, $\theta_y^*$and θ_u^* that are adopted in the capacity verifications depend on the corresponding values calculated under the requirements of section C8.7.2.3.2 of the commentary of NTC-18, according to the following equations of section C8.7.4.2.1 of the commentary of NTC-18:

The yield moment:

$$M_y^* = 0.9M_y \qquad \text{(8.7.4.2) commentary of NTC-18}$$

The chord rotation at yield:

$$\theta_y^* = 0.9\theta_y \qquad \text{(8.7.4.3) commentary of NTC-18}$$

The ultimate chord rotation:

$$\theta_u^* = \theta_u \qquad \text{(8.7.4.4) commentary of NTC-18}$$

FRP wrapping

The contribution of the FRP wrapping to the members' capacity is taken into account according to Annex A of EN1998-3:2005, as described below:

The effect of FRP wrapping on the members' flexural resistance at yielding, computed in accordance with equations C8.7.4.2.3 of the commentary of NTC-18, is neglected.

The total chord rotation capacity for the members of rectangular sections with corners rounded is calculated through the expression 8.7.2.1 of the commentary of NTC-18, respectively, with the exponent of the term due to confinement increased by $\alpha\rho_f f_{f,e}$, where α is the confinement effectiveness factor, ρ_f the FRP ratio parallel to the loading direction and $f_{f,e}$ the effectiveness stress given from the (A.35) equation of EC8: Part 3.

Bending Moment Capacity

The bending moment capacity of beams, columns and walls with prismatic cross section is calculated according to paragraph 4.1 of D. Biskinis and M. N. Fardis (2009), while for the bending moment capacity of circular columns paragraph 4 of Biskinis and M. N. Fardis (2013) is employed.

Shear Capacity

Shear capacity is calculated through the expressions defined in section 4.1.2.3.5 of NTC-18 and section C.8.7.2.3.5 of the commentary of NTC-18, depending on the value of the ductility demand.

In cases where the value of the ductility demand is less than 1 the shear capacity is calculated from equation 4.1.23 of NTC-18, which corresponds to the elements without taking into consideration the transverse reinforcement:

$$V_{Rd} = \left\{\left[0{,}18 \cdot k \cdot (100 \cdot \rho_1 \cdot f_{ck})^{1/3}/\gamma_c + 0{,}15 \cdot \sigma_{cp}\right] \cdot b_w \cdot d; (v_{min} + 0{,}15 \cdot \sigma_{cp}) \cdot b_w \cdot d\right\} \quad \text{(4.1.23) NTC-18}$$

When the value of the ductility demand is between 1 and 2, then the shear capacity is equal to the maximum value obtained from the equations 4.1.29 of NTC-18 and 8.7.2.8 of the commentary of NTC-18.

The equations 4.1.29 of NTC-18 corresponds to the shear capacity of the elements taking into consideration the transverse reinforcement.

$$V_{Rd} = \min(V_{Rsd}, V_{Rcd}) \quad \text{(4.1.29) NTC-18}$$

where V_{Rsd} is the shear strength that corresponds to the contribution of the shear reinforcement and is calculated according to the equation below:

$$V_{Rsd} = 0{,}9 \cdot d \cdot \frac{A_{sw}}{s} \cdot f_{yd} \cdot (\mathrm{ctg}\alpha + \mathrm{ctg}\theta) \cdot \sin\alpha \quad \text{(4.1.27) NTC-18}$$

and V_{Rcd} is the shear strength that corresponds to the confined concrete core and is calculated according to the following equation:

$$V_{Rcd} = 0{,}9 \cdot d \cdot b_w \cdot \alpha_c \cdot v \cdot f_{cd} \cdot (\mathrm{ctg}\alpha + \mathrm{ctg}\theta)/(1 + \mathrm{ctg}^2\theta) \quad \text{(4.1.28) NTC-18}$$

The equation 8.7.2.8 of the commentary of NTC-18 corresponds to the shear capacity as controlled by the stirrups, accounting for the reduction due to the plastic part of ductility demand.

$$V_R = \frac{1}{\gamma_{el}}\left[\frac{h-x}{2L_V}\min(N; 0{,}55A_c f_c) + \left(1 - 0{,}05\min\left(5; \mu_{\Delta,pl}\right)\right) \cdot \left[0{,}16\max(0{,}5; 100\rho_{tot})\left(1 - 0{,}16\min\left(5; \frac{L_V}{h}\right)\right)\sqrt{f_c}A_c + V_w\right]\right] \quad \text{(8.7.2.8) commentary of NTC-18}$$

where γ_{el} is equal to 1,15 for primary seismic elements and to 1,0 for secondary ones, the other variables are calculated as defined in section C.8.7.2.3.5 of the commentary of NTC-18.

If the value of the ductility demand is greater than 3, the shear capacity is calculated from equation 8.7.2.8 of the commentary of NTC-18.

Finally, linear interpolation should be executed for values of ductility demand between 2 and 3.

Concrete Jacketing

The value for the shear capacity, V_R^*, of the jacketed members that is adopted in the capacity verifications depend on the corresponding value calculated under the assumptions of section C8.7.4.2.1 of the commentary of NTC-18, according to the following equation:

$$V_R^* = 0.9V_R \quad \text{(8.7.4.1) commentary of NTC-18}$$

FRP wrapping

The cyclic resistance V_R, may be calculated from the section 4.1 of NTC-18 adding in V_w the contribution of the FRP jacket to shear resistance. The contribution of the FRP jacket to V_w is computed according to 4.19 equation of CNR-DT 200 R1/2013 in the following form:

$$V_{Rd,f} = \frac{1}{\gamma_{Rd}} \cdot 0.9 \cdot d \cdot f_{fed} \cdot 2 \cdot t_f \cdot (\cot\theta + \cot\beta) \cdot \sin\beta$$

Steel Braces Axial Deformations

Axial deformations of brace members in tension and compression should satisfy, according to Eurocodes, the provisions of tables B.2 and B.3 of Annex B of EN1998-3:2005.

Steel Braces Axial Forces

Axial force capacities of brace members in tension and compression should satisfy, according to Eurocodes, the provisions of sections 6.2.3 and 6.3 of EN1993-1.

Joints Diagonal Tension

According to C8.7.2.3.5 of the commentary of NTC-18 the diagonal tensile stress that can be induced in the joint may be calculated from the following expression:

$$\sigma_{nt} = \left| \frac{N}{2A_g} - \sqrt{\left(\frac{N}{2A_g}\right)^2 + \left(\frac{V_n}{A_g}\right)^2} \right| \leq 0{,}3\sqrt{f_c} \qquad \text{(8.7.2.11) commentary of NTC-18}$$

Joints Diagonal Compression

The diagonal compression induced in the joint by the diagonal strut mechanism shall not exceed the compressive strength of concrete in the presence of transverse tensile strains. NTC-18 indicates the following expression for the calculation of the joints' diagonal compression capacity:

$$\sigma_{nc} = \frac{N}{2A_g} + \sqrt{\left(\frac{N}{2A_g}\right)^2 + \left(\frac{V_n}{A_g}\right)^2} \leq 0{,}5f_c \qquad \text{(8.7.2.12) commentary of NTC-18}$$

For the definition of the values, you may refer to section C8.7.2.3.5 of the commentary of NTC-18.

Joints Ductility

Adequate ductility should be possessed by both the structural elements and the structure as a whole according to section 7.4.4.2.1 of NTC-18. In frame buildings with two or more storeys, the following condition should be satisfied at all joints of primary or secondary seismic beams with primary seismic columns:

$$\Sigma M_{c,Rd} \geq \gamma_{Rd} \Sigma M_{b,Rd} \qquad \text{(7.4.4) NTC-18}$$

Where γ_{Rd} is taken according to table 7.2.I of NTC-18, $\Sigma M_{c,Rd}$ is the sum of the design values of the moments of resistance of the columns framing the joint and $\Sigma M_{b,Rd}$ is the sum of the design values of the moments of resistance of the beams framing the joint.

Interstorey Drifts

According to section 7.3.6.1 of NTC-18, the damage caused from the seismic action should be limited to non-structural elements and should be assured that no structural damage will be caused that would make the structure temporarily unusable.

This target is achieved when the values of interstorey drift, concerning the limit states of operational level (SLO) and damage limitation (SLD), are below the limits indicated in the following table:

	Relative displacement d_r for the Limit State of Damage Limitation	Relative displacement d_r for the Limit State of Operational Level
Infill rigidly connected to the structure and interferes with the deformability of the latter	0,005 h*	2/3 of that for limit state damage limitation
Infill designed, so as not to be damaged from the interstorey drifts d_{rp}, that are caused by the infill's own deformability or by the connections to the structure	$d_r < d_{rp} < 0,01$ h	

Table C8.7.1 of commentary of NTC-18 – Limit values of Interstorey Drifts

Footings Bearing Capacity

Bearing capacity failure is verified under combinations of applied action effects N_{ed}, V_{Ed}, M_{Ed} according to EN 1998-5, 5.4.1.1 (8).

Footings Sliding Forces

Failure by sliding is verified according to EN 1998-5, section 5.4.1.1 (6) by ensuring that sliding force V_{Ed} on the horizontal base does not exceed the following expression:

$F_{Rd} + E_{pd}$

where

F_{Rd} is the design friction resistance of footings and

E_{pd} is the design lateral resistance arising from earth pressure on the side of the footings.

Footings Bending Capacity

Bending moment capacity check is performed according to EN 1992-1-1. Bending moment demand is calculated by pure stress σ_{net} acting on the horizontal base of footing.

Footings Shear Capacity

Shear capacity check is performed according to EN 1992-1-1. Shear demand is calculated by pure stress σ_{net} acting on the horizontal base of footing.

Footings Punching Capacity

Punching capacity check is carried out as described in EN 1992-1-1, section 6.4.2.

Footings Eccentricity

Eccentricity of loading should not exceed 1/3 of the dimension in each direction of footing according to EN 1997-1, section 6.5.4. Double eccentricity check is verified if the sum of squares of loading eccentricities in 2 horizontal directions is less than 1/9.

CAPACITY CURVE

Each pushover analysis leads to a capacity curve, which is a relationship between the total base shear and the horizontal displacement of a representative point of the structure, termed "control node", with the values of the control displacement ranging between zero and a maximum value defined by the user.

TARGET DISPLACEMENT

The target displacement is defined as the seismic demand derived from the elastic response spectrum in terms of displacement of an equivalent single-degree-of-freedom system. To define the target displacement of a MDOF system a number of steps have to be followed according to C7.3.4.2 of NTC-18.

The following relation between normalized lateral forces F_i and normalized displacements Φ_i is assumed:

$F_i = m_i \Phi_i$

Where m_i is the mass in the i-th storey.

Displacements are normalized in such a way that Φ_n=1, where n is the control node, consequently F_n=m_n.

Transformation to an equivalent Single Degree of Freedom (SDOF) system

The transformation factor is given by:

$\Gamma = \frac{\varphi^T M \tau}{\varphi^T M \varphi}$ (C7.3.5) commentary of NTC-18

The vector τ is the vector of deformation corresponding to the earthquake direction considered; the vector φ is the fundamental mode of vibration of the real system normalized placing d_c = 1; and the matrix M is the mass matrix of the real system.

The force F* and displacement d* of the equivalent SDOF system are computed as:

$F^* = F_b/\Gamma$

$d^* = d_c/\Gamma$ (C7.3.4) commentary of NTC-18

Where F_b and d_c are the base shear force at the control node and the displacement of the Multi Degree of Freedom (MDOF) system, respectively.

Determination of the idealized elasto-perfectly plastic force-displacement relationship

The yield force F_y^*, which represents also the ultimate strength of the idealized SDOF system, is equal to the base shear force at the formation of the plastic mechanism. The initial stiffness of the idealized system is determined in such a way that the areas under the actual and the idealized force-deformation curves are equal, as shown in the figure C7.3.1 of NTC-18 below:

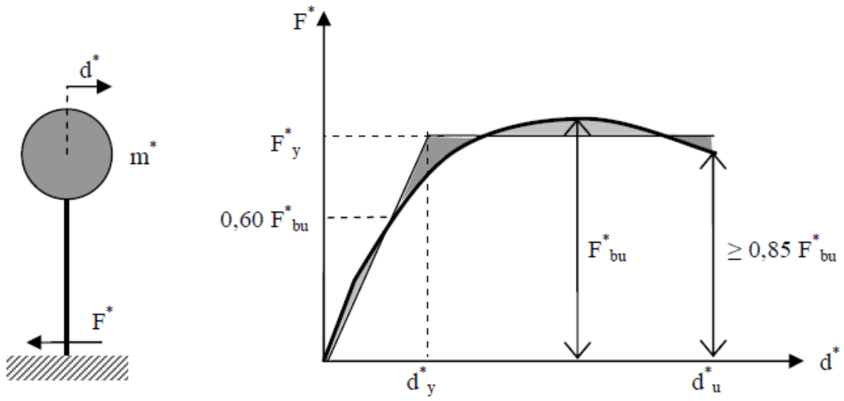


Figure C7.3.1 of commentary of NTC-18: Determination of the idealized elasto-perfectly plastic force-displacement relationship of NTC-18

The capacity curve of the equivalent system must be replaced by an idealized bilinear curve, which has a linear first branch and an elastic, perfectly plastic, second branch (see Fig. C7.3.1). The idealized capacity curve is defined by two points, the first one corresponds to $0{,}6F_{bu}^*$, where $F_{bu}^* = F_{BU}/\Gamma$ is the maximum resistance of the equivalent system and F_{BU} the maximum resistance of the real structural system. The second point corresponds to the yielding strength F_y^*, which can be defined graphically, on the criterion of approximately equal areas of the sections defined above and below the intersection of the real and the idealized curves for the maximum displacement d_u^* that corresponds to a reduction of the resistance $\leq 0{,}15F_{bu}^*$.

Determination of the period of the idealized equivalent SDOF system

The period T^* of the idealized equivalent SDOF system is determined by:

$$T^* = 2\pi\sqrt{\frac{m^*}{k^*}} \qquad \text{(C7.3.6) commentary of NTC-18}$$

Where k^* is the stiffness of the elastic branch of the bilinear curve.

Determination of the target displacement for the equivalent SDOF system

For the determination of the target displacement d^*_{max} for structures in the sort-period range and for structures in the medium and long-period ranges different expressions should be used as indicated below. The corner period between the short- and medium-period ranges is T_C.
The target displacement of the structures with period $T^* \geq T_C$ is given by:

$$d^*_{max} = d^*_{e,max} = S_{De}(T^*) \qquad \text{(C7.3.7) commentary of NTC-18}$$

$$\text{with } S_{De}(T^*) = S_e(T^*)\left[\frac{T^*}{2\pi}\right]^2 \qquad \text{(3.2.10) NTC-18}$$

Where $S_e(T^*)$ is the elastic acceleration response spectrum at the period T^*.
The target displacement of the structures with period $T^* < TC$ is given by:

$$d^*_{max} = \frac{d^*_{e,max}}{q^*}\left[1 + (q^* - 1)\frac{T_C}{T^*}\right] \geq d^*_{e,max} \qquad \text{(C7.3.8) commentary of NTC-18}$$

Where $q^* = S_e(T^*)\, m^*/F_y^*$ is the ratio between the acceleration in the structure with unlimited elastic behaviour, $S_e(T^*)$, and the structure with limited strength F_y^*/m^*.
In cases where $q^* \leq 1$ the $d^*_{max} = d^*_{e,max}$.

Determination of target displacement for the MDOF system

The target displacement of the MDOF system is given by:

$$d_t = \Gamma d^*_{max}$$

Note that the target displacement corresponds to the control node.

Appendix A.4 – KANEPE

In this appendix the parameters used for the structures assessment according to the Greek Seismic Interventions Code KANEPE are presented.

Type of Analysis

Current practice in Greece is regulated by the KANEPE: Code for structural interventions in combination with EAK 2000: Code for seismic resistant structures. Eurocode 8: Design of Structures for Earthquake Resistance – Part 1: General rules, seismic action and rules for buildings (CEN, 2005a) and Part 3: Assessment and Retrofitting of Buildings (CEN, 2005b) may be also applied.

According to KANEPE, for the assessment and redesign of a building, one of the following analysis methods may be used:

- Elastic (equivalent) static analysis, with global (q) or local (m) behaviour or ductility factors, subject to the conditions of section 5.5, regardless of the data reliability level;
- Elastic dynamic analysis with global (q) or local (m) behaviour or ductility factors, subject to the conditions of section 5.6, regardless of the data reliability level;
- Inelastic static analysis, subject to the conditions of section 5.7. In this case, it is recommended to ensure, as a minimum, a "sufficient" data reliability level;
- Inelastic dynamic (response history) analysis, subject to the conditions of section 5.8. In this case, it is again recommended to ensure, as a minimum, a "sufficient" data reliability level;
- In special cases, e.g., when the assessment concerns a significant number of buildings, which it is aimed to determine whether there is, in principle, need for seismic strengthening (and with what priority), or the building to be assessed is of low importance; then, in addition to purely analytical methods, the assessment may be done by empirical methods, subject to the conditions of §2.1.4.1 b(iv).

In SeismoBuild the most common method in assessment practice of existing buildings is employed, which is the nonlinear static analysis. It is based on pushover analyses carried out under constant gravity loads and increasing lateral forces, applied at the location of the masses to simulate the inertia forces induced by the seismic action. As the model may account for both geometrical and mechanical nonlinearity, this method can describe the evolution of the expected plastic mechanisms and structural damage.

Each pushover analysis leads to a capacity curve, which is a relationship between the total base shear and the horizontal displacement of a representative point of the structure, termed "control node". The demand at the considered Performance Objective – Immediate Occupancy, Life Safety or Collapse Prevention - is determined by the appropriate comparison between the capacity determined by the pushover curve and the demand established as the damped Linear Response Spectrum. To do so, the "control node" displacements are defined in terms of spectral quantities relative to an equivalent single-degree-of-freedom (SDOF) system which is derived from the multi-degree-of-freedom (MDOF) response estimated according to section 5.7.4.2 of KANEPE.

The structural demand associated with the acquired target displacement shall fulfil the verification criteria defined in KANEPE. Accordingly, element's demand for brittle (shear) and ductile (chord rotation deformation) actions are deemed to comply with limits that take into account: section mechanical properties; element's bending, shear and axial force interaction; and strength/stiffness degradation associated with the ductility demand and cyclic hysteretic response of reinforced concrete elements, through appropriate material nonlinearity consideration.

Performance Requirements

According to KANEPE section 2.2, the objectives of the assessment or redesign (Table 2.1) consist of combinations of both a performance level and a seismic action, given an "acceptable probability of exceedance within the life cycle of the building" (design earthquake), as shown in Table 2.1 of KANEPE below.

Probability of exceedance of seismic action within a conventional life cycle of 50 years	Target Building Performance Levels		
	A Immediate Occupancy	B Life Safety	C Collapse Prevention
2%	A0	B0	C0
5%	A1+	B1+	C1+
10%	A1	B1	C1
20%	A2+	B2+	C2+
30%	A2	B2	C2
50%	A3+	B3+	C3+
70%	A3	B3	C3
90%	A4+	B4+	C4+
>90%	A4	B4	C4

Table 2.1 Assessment or Redesign objectives of the structure.

The target building performance levels refer to the state of damage in the structure defined through three limit states, namely Immediate Occupancy (A), Life Safety (B) and Collapse Prevention (C).

Performance Level of Immediate Occupancy (A)

The Immediate Occupancy after the earthquake (A), according to KANEPE, is a condition in which it is expected that no building operation is interrupted during and after the design earthquake, with the possible exception of minor importance functions. A few hairline crack may occur in the structure.

Performance Level of Life Safety (B)

The Life Safety (B), according to KANEPE, is a condition in which repairable damage to the structure is expected to occur during the design earthquake, without causing loss or serious injury of people and without substantial damage to personal property or materials that are stored in the building.

Performance Level of Collapse Prevention (C)

The Collapse Prevention (C), according to KANEPE, is a condition in which extensive and serious or severe (non-repairable, in general) damage to the structure is expected during the design earthquake; however, the structure retains its ability to bear the prescribed vertical loads (during and for a period after the earthquake), in any case without other substantial safety factor against total or partial collapse.

The choice of which Performance Objectives will be checked may be found in KANEPE.

Information for Structural Assessment

In order to choose the admissible type of analysis and the appropriate confidence factor values, the following three data reliability levels (DRL) are defined:

- Tolerable DRL

- Sufficient DRL
- High DRL

The factors determining the obtained data reliability level are the (i) geometry, which is the geometrical properties of the structural system and of non-structural elements, e.g. masonry infill panels, that may affect structural response; (ii) details, which include the amount and detailing of reinforcement in reinforced concrete sections, the connection of floor diaphragms to lateral resisting structure, the bond and mortar jointing of masonry and the nature of any reinforcing elements in masonry; and finally (iii) materials, that is the mechanical properties of the constituent materials.

Tolerable DRL

The tolerable data reliability level corresponds to a state of knowledge where the overall structural geometry and member sizes are known from survey or from original outline construction drawings used for both the original construction and any subsequent modifications, as well as a sufficient sample of dimensions of both overall geometry and member sizes checked on site. In case of significant discrepancies from the outline construction drawings a fuller dimensional survey is performed. The structural details are not known from detailed construction drawings and are assumed based on simulated design in accordance with usual practice at the time of construction. Limited inspections performed in the most critical elements should prove that the assumptions correspond to the actual situation. Information on the mechanical properties of the construction materials isn't available so default values are assumed in accordance with standards at the time of construction accompanied by limited in-situ testing in the most critical elements.

Structural evaluation based on this state of knowledge is performed through linear analysis methods, either static or dynamic.

Sufficient DRL

The sufficient data reliability level corresponds to a state of knowledge where the overall structural geometry and member sizes are known from extended survey or from outline construction drawings used for both the original construction and any subsequent modifications, as well as a sufficient sample of dimensions of both overall geometry and member sizes. The structural details are known from an extended in-situ inspection or from incomplete detailed construction drawings in combination with limited in-situ inspections in the most critical elements, which confirms that the available information corresponds to the actual situation. Information on the mechanical properties of the construction materials is available from extended in-situ testing or from original design specifications and limited in-situ testing.

Structural evaluation based on this state of knowledge is performed through linear or nonlinear analysis methods, either static or dynamic.

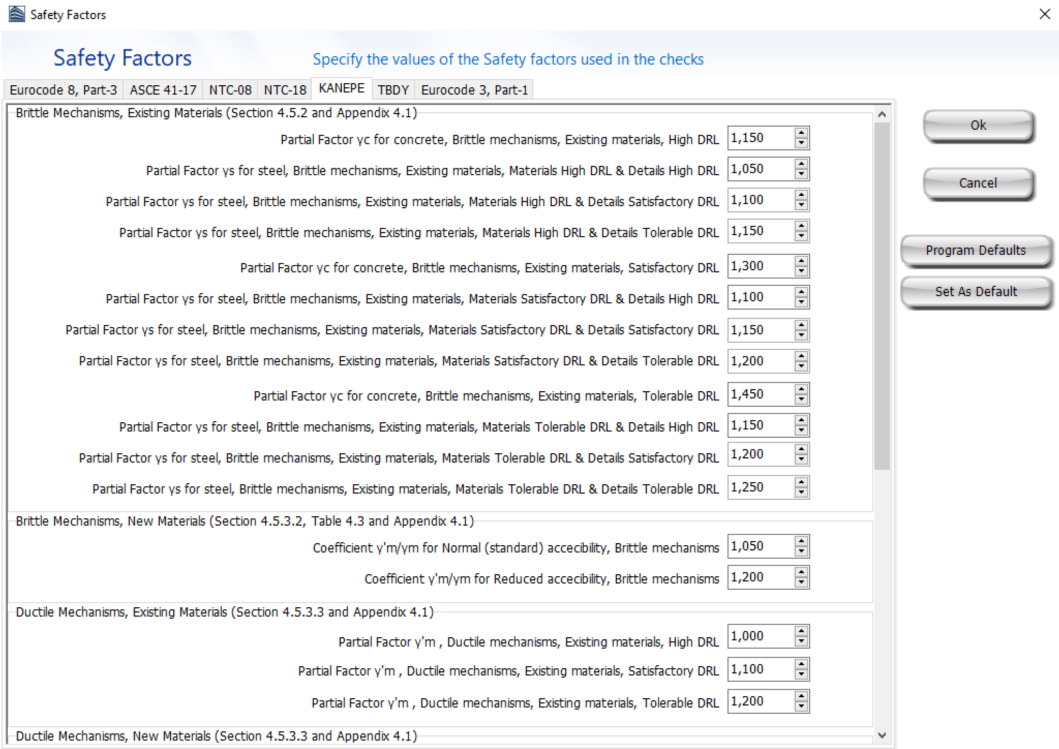
High DRL

The high reliability level corresponds to a state of knowledge where the overall structural geometry and member sizes are known from a comprehensive survey or from the complete set of outline construction drawings used for both the original construction and subsequent modifications, as well as a sufficient sample of both overall geometry and member sizes checked on site. The structural details are known from comprehensive in-situ inspection or from a complete set of detailed construction drawings in combination with limited in-situ inspections in the most critical elements, which prove that the available information corresponds to the actual situation. Information on the mechanical properties of the construction materials is available from comprehensive in-situ testing or from original test reports and limited in-situ testing.

Structural evaluation based on this state of knowledge is performed through linear or nonlinear analysis methods, either static or dynamic.

Safety Factors

The recommended values for the safety and partial factors corresponding to the achieved data reliability level, as defined in section 4.5 of KANEPE, are introduced in the Safety Factors module. Users may edit the assigned values.



Safety Factors module

CAPACITY MODELS FOR ASSESSMENT AND CHECKS

All the member checks (chord rotation capacity and shear capacity) should be carried out for all the elements of every floor, according to sections 7.2.2, 7.2.4 and Appendix 7C of KANEPE, considering the members as primary or secondary seismic elements, designated in accordance with the definitions of section 2.4.3.4 of KANEPE. Moreover, beam-column joints checks can be employed in order to check (i) the joint's diagonal tension and (ii) the joint's diagonal compression.

Deformation Capacity

The deformation capacity of beams, columns and walls is defined in terms of the chord rotation θ, that is the angle between the tangent to the axis at the yielding end and the chord connecting that end with the end of the shear span (L_s=M/V=moment/shear at the end section). The chord rotation is also equal to the element drift ratio, which is the deflection at the end of the shear span with respect to the tangent to the axis at the yielding end divided by the shear span.

Deformation capacity of beams and columns is highly influenced by the lack of appropriate seismic resistant detailing in longitudinal reinforcement, as well as whether there are smooth bars and the accessibility of area of intervention. Inadequate development of splicing along the span (beams) and height (columns); and inadequate embedment into beam-column joints can control the members' response to seismic action, drastically limiting its capacity in respect to the situation in which the reinforcement is considered fully effective. The above limitations to the deformation capacity are taken into consideration.

The value for the chord rotation capacity for the performance level of immediate occupancy (A) is the value of the chord rotation capacity at flexural yield, θ_y, which is calculated from the equations (S.2a) and (S.2b) of KANEPE:

- For beams and columns:

$$\theta_y = \left(1/r\right)_y \frac{L_s + \alpha_V z}{3} + 0{,}0014\left(1 + 1{,}5\frac{h}{L_s}\right) + \frac{(1/r)_y d_b f_y}{8\sqrt{f_c}} \qquad \text{(S.2a) KANEPE}$$

- For walls:

$$\theta_y = \left(1/r\right)_y \frac{L_s + \alpha_V z}{3} + 0{,}0013 + \frac{(1/r)_y d_b f_y}{8\sqrt{f_c}} \qquad \text{(S.2b) KANEPE}$$

Where L_s is the ratio between bending moment, M, and shear force, V; and a_V is equal to 1,0 if the value of the shear force V_{R1}, which causes diagonal cracking of the element, is less than the value of the shear force during flexural yielding $V_{Mu}=M_y/L_s$, or 0 otherwise.

The value for the chord rotation capacity for the performance level of life safety (B) is calculated from the following equation, according to section 9.3.1 of KANEPE.

$$\theta_d = 0{,}5(\theta_y + \theta_{um})/\gamma_{Rd}$$

Where θ_y is calculated according to (S.2a) and (S.2b) equations and θ_{um} according to (S.11a) and (S.11b) equations of KANEPE.

The value for the chord rotation capacity for the performance level of collapse prevention (C) is the mean value of the chord rotation capacity at failure, which is calculated according to equation S.11a of KANEPE, from the following expressions:

- For beams and columns designed and constructed based on post-1985 provisions on seismic design, from:

$$\theta_{um} = 0{,}016 \cdot (0{,}3^{v}) \left[\frac{\max(0{,}01;\omega')}{\max(0{,}01;\omega-\omega')} f_c\right]^{0{,}225} \cdot (\alpha_s)^{0{,}35} 25^{\left(\alpha\rho_s \frac{f_{yw}}{f_c}\right)} (1{,}25^{100\rho_d}) \qquad \text{(S.11a) KANEPE}$$

- For walls with rectangular section, designed and constructed based on post-1985 provisions on seismic design, taking into consideration the paragraph ii) of the commentary of section 7.2.4.1b, from:

$$\theta_{um} = 0{,}01 \cdot (0{,}3^{v}) \left[\frac{\max(0{,}01;\omega')}{\max(0{,}01;\omega-\omega')} f_c\right]^{0{,}225} \cdot (\alpha_s)^{0{,}35} 25^{\left(\alpha\rho_{sx} \frac{f_{yw}}{f_c}\right)} (1{,}25^{100\rho_d}) \qquad \text{(S.11a) KANEPE}$$

The values above are divided by the γ_{Rd} factor, according to section 9.3.1 of KANEPE.

The total chord rotation capacity at ultimate of concrete members under cyclic loading may be also calculated as the sum of the chord rotation at yielding and the plastic part of the chord rotation capacity calculated from the following expression:

- For beams and columns designed and constructed based on post-1985 provisions on seismic design, from:

$$\theta_{um}^{pl} = \theta_u - \theta_y = 0{,}0145 \cdot (0{,}25^{v}) \left[\frac{\max(0{,}01;\omega')}{\max(0{,}01;\omega-\omega')}\right]^{0{,}3} (f_c)^{0{,}2} (\alpha_s)^{0{,}35} 25^{\left(\alpha\rho_s \frac{f_{yw}}{f_c}\right)} (1{,}275^{100\rho_d})$$

(S.11b) KANEPE

- For walls with rectangular section designed and constructed based on post-1985 provisions on seismic design, taking into consideration the paragraph ii) of the commentary of section 7.2.4.1b, from:

$$\theta_{um}^{pl} = \theta_u - \theta_y = 0{,}0087 \cdot (0{,}25^{\nu}) \left[\frac{\max(0{,}01;\omega')}{\max(0{,}01;\omega-\omega')}\right]^{0{,}3} (f_c)^{0{,}2} (\alpha_s)^{0{,}35} 25^{\left(\alpha\rho_s \frac{f_{yw}}{f_c}\right)} (1{,}275^{100\rho_d})$$

(S.11b) KANEPE

For elements with deformed bars designed and constructed according the pre-1985 rules applying in Greece, the values calculated based on equations S.11a and S.11b above need to be divided by 1.2.

The yield curvature of the end section is calculated according to the following equation (Appendix 7A of KANEPE), for the sections whose compressive zone is of constant width and for the case that the section's yielding is due to steel yielding.

$$\varphi_y = (1/r)_y = \frac{f_y}{E_s(1-\xi_y)d} \qquad \text{(A.1) KANEPE}$$

If the section yields due to the deformation nonlinearities of the concrete in compression, that is for deformation of the edge compressive fibre larger than $\varepsilon_c \approx 1.8\, f_c/E_c$, then the yield curvature is calculated according to the following expression, of Appendix 7A of KANEPE:

$$\varphi_y = (1/r)_y = \frac{\varepsilon_c}{\xi_y d} \approx \frac{1.8 f_c}{E_c \xi_y d} \qquad \text{(A.2) KANEPE}$$

The lower from the two values above is used for the calculation of the chord rotation capacity.

According to section 7.2.4 of KANEPE the chord rotation capacity is highly influenced by several factors such as the year of construction. If the structure has been constructed with Regulations before 1985 then the mean values of chord rotation capacity and the plastic part of the mean chord rotation are divided by 1.2. Moreover, if the deformed longitudinal bars have straight ends lapped starting at the end section of the member, the plastic part of chord rotation is calculated with the value of the compression reinforcement ratio, ω', doubled over the value applying outside the lap splice. In addition, in sections where the reinforcement lap length l_b is less than the minimum lap length for ultimate deformation $l_{bu,min}$, the plastic part of the chord rotation capacity is multiplied by the ratio $l_b/l_{bu,min}$ (information about the calculation of $l_{bu,min}$ can be found in section 7.2.4 of KANEPE, while the value for chord rotation at yielding, θ_y accounts for the effect of the lapping in accordance with section 7.2.2 of KANEPE). Further, if smooth (plain) longitudinal bars are applied, the values obtained for ribbed bars are multiplied with a factor equal to 95% and in case of having elements with reinforcement lap length l_b less than 15db, the mean value of the chord rotation at failure is multiplied by a factor available in section 7.2.4 of KANEPE. Finally, reducing factors are employed according to Annex 7ST of KANEPE to account for the reinforcement corrosion.

In the case of circular column sections, the equations above cannot be employed for the calculation of the elements' chord rotation capacity. In SeismoBuild the equations below suggested by D. Biskinis and M. N. Fardis [2013] are employed for θ_y and θ_u.

$$\theta_y = \varphi_y \frac{L_V + \alpha_V z}{3} + 0.0027\left(1 - \min\left(1; \frac{2}{15}\frac{L_s}{D}\right)\right) + \alpha_{sl}\frac{\varphi_y d_{bL} f_y}{8\sqrt{f_c}}$$

Where f_y and f_c values are in MPa, α_V=1 if V_{Rc}<V_{My}, V_{Rc} is calculated according to equation (S.3) of KANEPE, otherwise α_V=0, and α_{sl}=0 if pull-out of the tension bars from their anchorage zone beyond the yielding end is physically impossible, otherwise α_{sl}=1.

$$\theta_u = \left(\theta_y + (\varphi_u - \varphi_y)L_{pl}(1 - 0.5\, L_{pl}/L_s) + \alpha_{sl}\Delta\theta_{u,slip}\right)/\gamma_{el}$$

Where γ_{el} is equal to 2.0 for elements designed with the pre-1985 provisions about seismic design and to 1.0 for elements designed and constructed according to the post-1985 rules applying in Greece, $\Delta\theta_{u,slip}$ and L_{pl} are calculated according to the following equations:

$$\Delta\theta_{u,slip} = 10 d_{bl}\,(\varphi_u + \varphi_y)/2$$

$$L_{pl} = 0.6D\left[1 + \frac{1}{6}\min\left(9; \frac{L_s}{D}\right)\right]$$

Users are advised to refer to the relevant publications for the definition of the other parameters and further details on the expression.

Concrete Jacketing

The values of the jacketed members for M_y^*, $\theta_y^*$and θ_u^* that are adopted in the capacity verifications depend on the corresponding values calculated under the requirements of section 8.2.1 of KANEPE, according to the following equations of section 8.2.1.5(η) of KANEPE:

The yield moment:

$$M_y^* = 0.90M_y$$

The chord rotation at yield:

$$\theta_y^* = 1.25\theta_y$$

The ultimate chord rotation:

$$\theta_u^* = 0.80\theta_u$$

FRP wrapping

The contribution of the FRP wrapping to members' capacity is taken into account, according to Annex A of EN1998-3:2005, as described below:

The effect of FRP wrapping on the members' flexural resistance at yielding, computed in accordance with section 7.2.2, is neglected.

The total chord rotation capacity and its plastic part for the members of rectangular sections with corners rounded is calculated through the expressions (S.11a) and (S.11b), respectively, with the exponent of the term due to confinement increased by $\alpha\rho_f f_{f,e}$, where α is the confinement effectiveness factor, ρ_f the FRP ratio parallel to the loading direction and $f_{f,e}$ the effectiveness stress given from the (A.35) equation of EC8: Part 3.

Bending Moment Capacity

The bending moment capacity of beams, columns and walls with prismatic cross section is calculated according to paragraph 4.1 of D. Biskinis and M. N. Fardis (2009), while for the bending moment capacity of circular columns paragraph 4 of Biskinis and M. N. Fardis (2013) is employed.

Shear Capacity

Shear capacity is calculated through the following expression according to Annex 7C of KANEPE, as controlled by the stirrups, accounting also for the reduction due to the plastic part of ductility demand.

$$V_R = \frac{h-x}{2L_s}\min(N; 0{,}55A_c f_c) + \left(1 - 0{,}05\min\left(5; \mu_\theta^{pl}\right)\right)\left[0{,}16\max(0{,}5; 100\rho_{tot})\,(1 - 0{,}16\min(5; \alpha_s))\sqrt{f_c}A_c + V_w\right] \qquad \text{(C.1) KANEPE}$$

The shear strength of a shear wall may not be taken greater than the value corresponding to failure by web crushing, $V_{R,max}$, which under cyclic loading is calculated according to Annex 7C of KANEPE. from the following expression:

$$V_{R,max} = 0{,}85\left(1 - 0{,}06\min\left(5; \mu_\theta^{pl}\right)\right)\left(1 + 1{,}8\min\left(0{,}15; \frac{N}{A_c f_c}\right)\right)\left(1 + 0{,}25\max(1{,}75; 100\rho_{tot})\right)\cdot\left(1 - 0{,}2\min(2; \alpha_s)\right)\sqrt{f_c}b_w z \qquad \text{(C.4) KANEPE}$$

The shear strength, V_R, of columns with shear ratio $\alpha_s \leq 2.0$ may not be taken greater than the value corresponding to failure by web crushing along the diagonal of the column after flexural yielding, $V_{R,max}$, which under cyclic loading is calculated according to Annex 7C of KANEPE from the following expression:

$$V_{R,max} = {}^4/_7\left(1 - 0{,}02\min\left(5; \mu_\theta^{pl}\right)\right)\left(1 + 1{,}35\frac{N}{A_c f_c}\right)\left(1 + 0{,}45(100\rho_{tot})\right)\sqrt{\min(40; f_c)}\,b_w z \sin 2\delta$$

(C.5) KANEPE

Where δ is the angle between the diagonal and the axis of the column ($\tan\delta = h/2L_s = 0.5/\alpha_s$).

The possibility of sliding at the base or at any part of the section where the longitudinal reinforcement might yield should be examined in walls. The sliding resistance should not be checked in walls that shear failure happens before flexural yielding. The sliding resistance may be calculated from the following equation of Appendix 7C of KANEPE:

$$V_{R,SLS} = V_i + V_f + V_d$$ (C.6) KANEPE

with

$$V_i = \sum A_{si} f_{yi} \cos\varphi$$ (C.7) KANEPE

$$V_f = \min\left(\mu\left[\left(\sum A_{sv} f_{yv} + N\right)\xi + M_y/z\right]; 0.3 f_c A_{compr}\right)$$
(C.8) KANEPE

where ξ is calculated according to equations C.10-C13 of Appendix 7C of KANEPE, and

$$V_d = 1.6\sum A_{sv}\sqrt{f_c f_{yv}} \le \sum A_{sv} f_{yv}/\sqrt{3}$$ (C.9) KANEPE

Users are advised to refer to the relevant publication for the definition of the parameters and further details on the above expressions.

Alternatively, the sliding resistance may be calculated from the following equation of Appendix 7C of KANEPE:

$$V_{R,SLS} = \left(1 - 0{,}025\mu_\theta^{pl}\right)\min\left(\begin{array}{c} 0{,}5\sum\left[A_s f_y(0{,}6\sin\varphi + \cos\varphi)\right] + 0{,}6N + 1{,}1\sum\left[A_s\sqrt{f_c f_y}\sin\varphi\right]; \\ 0{,}2\min\left(0{,}55; 0{,}55\left(\frac{30}{f_c}\right)^{\frac{1}{3}}\right) f_c A_c \end{array}\right)$$

(C.14) KANEPE

Users are advised to refer to the relevant publication for the definition of the parameters and further details on the expression.

The equations (C.1)-(C.3) and (C.4) may be used for walls with shear ratio $\alpha_s \geq 1.0$. For walls with low shear ratio $\alpha_s \leq 1.2$, the shear capacity should be calculated from the following equation of Appendix 7C of KANEPE:

$$V_{R,squat} = V_{si} + V_c$$ (C.15) KANEPE

with

$$V_s = \min\left\{\begin{array}{c} \rho_h b_w \min\left((d - x)/\tan\theta_{cr}, L_s\right) f_{yh} \\ \left(\rho_v b_w \min(L_s \tan\theta_{cr}, d - x) f_{yv} + A_s f_y\right)/\tan\theta_{cr} \end{array}\right\}$$ (C.16) KANEPE

and

$$V_c = (1 + 150\rho)(1 - 0.725\alpha_s)\left(\frac{2}{3}A_c f_{ct}\sqrt{1 + \frac{N}{A_c f_{ct}}}\right)$$ (C.17) KANEPE

Users are advised to refer to the relevant publication for the definition of the parameters and further details on the expressions.

It is noted that reducing factors are employed according to Annex 7ST of KANEPE to account for the reinforcement corrosion.

Concrete Jacketing

The value for the shear capacity, V_R^*, of the jacketed members that is adopted in the capacity verifications depend on the corresponding value calculated under the requirements of section 8.2.1 of KANEPE, according to the following equation of section 8.2.1.5(η) of KANEPE:

$$V_R^* = 0.9V_R$$

FRP wrapping

The cyclic resistance V_R, may be calculated from expression (C.1) of KANEPE adding in V_w the contribution of the FRP jacket to the shear resistance. The contribution of the FRP jacket to V_{jd} is computed through the following expression:

$$V_{jd} = \sigma_{jd}\rho_j b_w h_{j,ef}(\cot\theta + \cot\alpha)\sin^2\alpha \qquad \text{(8.13) KANEPE}$$

where ρ_f is the geometric ratio of the FRP, calculated according to (S8.8) equation of KANEPE.

Steel Braces Axial Deformations

Axial deformations of brace members in tension and compression should satisfy the provisions of section 8.5.5.4 of KANEPE.

Steel Braces Axial Forces

Axial force capacities of brace members in tension and compression should satisfy, according to Eurocodes, the provisions of sections 6.2.3 and 6.3 of EN1993-1.

Joints Diagonal Tension

According to section 7.2.5 of KANEPE, diagonal tension cracking of the core of joints reinforced with horizontal stirrups occurs when the principle tensile stress, i.e. the combination of (i) the mean shear stress τ_j, (ii) the mean vertical normal compressive stress in the joint, $\sigma_c = \nu_{top} f_c$, and (iii) the mean horizontal compressive stress that develops in the core of the joint as a result of the confinement provided by the horizontal stirrups, exceeds the tensile strength of concrete, f_{ct}, i.e.:

$$\tau_j \geq \tau_c = f_{ct}\sqrt{\left(1 + \frac{\rho_{jh} f_{yw}}{f_{ct}}\right)\left(1 + \frac{\nu_{top} f_c}{f_{ct}}\right)} \qquad \text{(4) KANEPE}$$

where $\rho_{jh} = A_{sh}/b_j h_{jb}$ i.e. the total cross section A_{sh} of the horizontal stirrup legs parallel to the vertical plane of the stress τ_j, normalised to the area of the vertical cross section of the joint, $b_j z_b$. For more information you may refer to section 7.2.5 of KANEPE.

Joints Diagonal Compression

According to section 7.2.5 of KANEPE, failure of the core due to diagonal compression occurs if the principle compressive stress exceeds the compressive stress (as reduced by possible transverse tensile deformations) of the concrete. If the mean shear stress in the joint, τ_j, exceeds the value of τ_c given by Eq. (4), then it may be assumed that failure of the joint due to diagonal compression occurs when the value of τ_j exceeds the value:

$$\tau_j \geq \tau_{ju} = n f_c\sqrt{\left(1 - \frac{\nu_{top}}{n}\right)} \qquad \text{(5) KANEPE}$$

Where $n=0.6(1-f_c(MPa)/250)$ the reduction factor of the uniaxial compressive strength due to transverse tensile deformations. If, on the other hand, τ_j is less than τ_c given by expression (4), then it may be assumed that failure of the joint due to diagonal compression occurs when τ_j exceeds the value derived from expression (5) for n=1.

Joints Ductility

Adequate ductility should be possessed by both the structural elements and the structure as a whole according to section 9.3.3 of KANEPE. In frame buildings with two or more storeys, the following condition should be satisfied at all joints of primary or secondary seismic beams with primary seismic columns:

$\Sigma M_{Rc} \geq 1{,}3 \Sigma M_{Rb}$ (S1) ΚΑΝ.ΕΠΕ.

Otherwise, the values of table S 4.4 should be taken for buildings constructed before 1985. Where ΣM_{Rc} is the sum of the design values of the moments of resistance of the columns framing the joint and ΣM_{Rb} is the sum of the design values of the moments of resistance of the beams framing the joint. The joints ductility check is not employed for the joints of the top level of multistorey buildings according to section 9.3.3 of KANEPE.

Footings Bearing Capacity

Bearing capacity failure is verified under combinations of applied action effects N_{ed}, V_{Ed}, M_{Ed} according to EN 1998-5, 5.4.1.1 (8).

Footings Sliding Forces

Failure by sliding is verified according to EN 1998-5, section 5.4.1.1 (6) by ensuring that sliding force V_{Ed} on the horizontal base does not exceed the following expression:

$F_{Rd} + E_{pd}$

where

F_{Rd} is the design friction resistance of footings and

E_{pd} is the design lateral resistance arising from earth pressure on the side of the footings.

Footings Bending Capacity

Bending moment capacity check is performed according to EN 1992-1-1. Bending moment demand is calculated by pure stress σ_{net} acting on the horizontal base of footing.

Footings Shear Capacity

Shear capacity check is performed according to EN 1992-1-1. Shear demand is calculated by pure stress σ_{net} acting on the horizontal base of footing.

Footings Punching Capacity

Punching capacity check is carried out as described in EN 1992-1-1, section 6.4.2.

Footings Eccentricity

Eccentricity of loading should not exceed 1/3 of the dimension in each direction of footing according to EN 1997-1, section 6.5.4. Double eccentricity check is verified if the sum of squares of loading eccentricities in 2 horizontal directions is less than 1/9.

Capacity Curve

Each pushover analysis leads to a capacity curve, which is a relationship between the total base shear and the horizontal displacement of a representative point of the structure, termed “control node”, with the values of the control displacement ranging between zero and a maximum value defined by the user.

TARGET DISPLACEMENT

The target displacement δ_t (§ 5.7.4.2) shall be calculated taking into account all the relevant factors affecting the displacement of a building that responds inelastically. It is permitted to consider the displacement of an elastic single degree of freedom system with a fundamental period equal to the fundamental period of the building that is subjected to the seismic actions for which the verification is made. An appropriate correction is needed in order to derive the corresponding displacement of the building assumed to be responding as an elastic-perfectly plastic system.

If a more accurate method is not used, the target displacement δ_t can be calculated using the following equation and be corrected (where necessary) according to §5.7.4.2 as follows:

$$\delta_t = C_0 \cdot C_1 \cdot C_2 \cdot C_3 \cdot (T_e^2/4\pi^2)S_{e(T)} \qquad \text{(S5.6) KANEPE}$$

where $S_{e(T)}$ is the elastic spectral pseudo-acceleration (derived from the EC8 spectrum) corresponding to the equivalent fundamental period of the structure T_e (the latter calculated using the point of contraflexure in the force-displacement diagram of the system, as defined in equation S5.5 of § 5.7.3.5), and C_0, C_1, C_2 and C_3 are correction factors that are defined as follows:

C_0: Coefficient that relates the spectral displacement of the equivalent elastic system of stiffness K_e (S_d=[$T_e^2/4\pi^2$] $S_{e(T)}$), with the actual displacement δ_t of the top of the structure, which is assumed to be responding as an elasto-plastic system (§ 5.7.3.4). The values of this coefficient can be taken equal to 1.0, 1.2, 1.3, 1.4, 1.5, for a number of storeys equal to 1, 2, 3, 5, and ≥ 10, respectively.

The ratio $C_1=\delta_{inel}/\delta_{el}$ of the maximum inelastic displacement of a building to the corresponding elastic displacement may be obtained from the following relationships:

C_1=1.0 for $T_e \geq T_C$, and

C_1=[1.0+(R-1)T_C/ T_e]/R for $T_e < T_C$,

where T_C is the corner period initiating the descending branch of the response spectrum (EC8-Part1) and $R=V_{el}/V_y$ the ratio of the elastic demand over the yield strength of the structure. This ratio can be estimated from the relationship:

$$R = \frac{S_e/g}{V_y/W} \cdot C_m \qquad \text{(S5.7) KANEPE}$$

where the yield strength V_y is calculated by appropriate bilinearisation of the base shear vs. top displacement relationship of the building, as defined in §5.7.3.4. For simplicity, (and conservatively), the ratio V_y/W in equation can be taken equal to 0.15 for buildings with a dual structural system, and 0.10 for buildings with a pure frame system.

C_2: Coefficient that takes into account the influence of the shape of the hysteresis loop at the maximum displacement. Its values may be obtained from Table S5.1.

Performance level	T=0.1s		T≥T_C	
	Structural Type 1	Structural Type 2	Structural Type 1	Structural Type 2
Immediate Occupancy	1.0	1.0	1.0	1.0
Life Safety	1.3	1.0	1.1	1.0
Collapse Prevention	1.5	1.0	1.2	1.0

Table S5.1 of KANEPE: Values of coefficient C_2

As structural systems of Type 1 are denoted structures with low ductility (e.g. buildings constructed prior to 1985 or buildings whose capacity curve is characterized by an available displacement ductility which is lower than 2), that are expected to have inferior hysteretic behaviour than structures with high ductility which are characterised as Type 2 systems, e.g. buildings constructed after 1985, or buildings whose capacity curve is characterized by an available displacement ductility which is higher than 2. Given the fact that the influence of hysteretic behaviour is greater for higher levels of post-elastic structural response, the values of the coefficient C_2 are conditioned to the performance level.

C_3: Coefficient that takes into account the increase of displacements due to second order (P-Δ) effects. It can be taken equal to $1+5(\theta-0.1)/T_e$, where θ is the interstorey drift sensitivity coefficient (see EC8-Part1). In the common case (for RC and masonry buildings) where $\theta<0.1$, the coefficient is taken equal to $C_3=1.0$.

Determination of the idealized elasto-perfectly plastic force-displacement relationship

The nonlinear force-displacement relationship that relates the base shear with the displacement of the control node shall be replaced by an idealized curve for the determination of the equivalent lateral stiffness K_e and the corresponding yield strength V_y of the building.

It is recommended that the idealized capacity curve (force-displacement relationship) is bilinear, with a slope of the first branch equal to K_e and a slope of the second branch equal αK_e. The two lines that compose the bilinear curve can be defined graphically, on the criterion of approximately equal areas of the sections defined above and below the intersection of the actual and the idealized curves (Figure 5.2 of KANEPE).

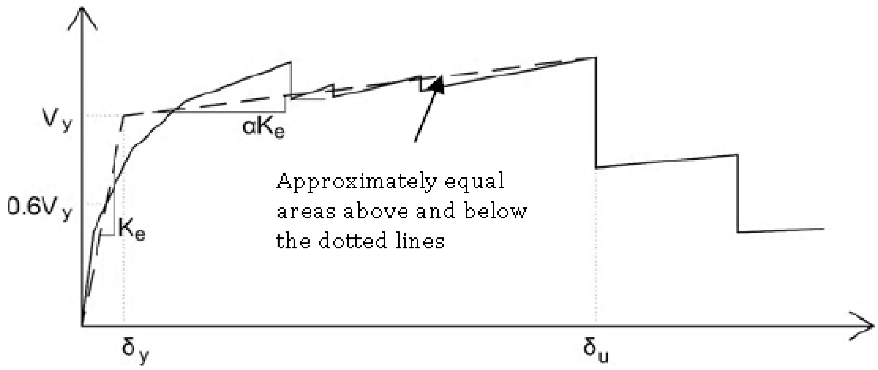


Figure 5.2 of KANEPE Idealization of a (indicative) capacity curve with a bilinear curve

The equivalent lateral stiffness K_e is determined as the secant stiffness that corresponds to a force equal to the 60% of the yielding force V_y, the latter defined by the intersection of the lines above. The normalized inclination (α) of the second branch is determined by a straight line passing through the point of the (actual) nonlinear capacity curve that corresponds to the ultimate displacement (δ_u), beyond which a significant drop of the strength of the structure is observed (Figure 5.2). In any case, the derived value of α must be positive (or zero), but not larger than 0.10 (in order to be compatible with the other assumptions made by the method for estimating the target displacement δ_t, such as the coefficient C_1). The recommended fraction of the resistance reduction is 15%, provided that no primary vertical member has reached failure at this level (in such a case, the bilinearisation of the curve shall be made for the displacement that corresponds to this failure).

Determination of the fundamental period

The equivalent fundamental period in the direction examined shall be estimated based on the idealized capacity curve.
The value T_e of the equivalent fundamental period is derived by the following expression:

$$T_e = T\sqrt{\frac{K_0}{K_e}} \qquad \text{(S5.5) KANEPE}$$

where T is the elastic fundamental period in the direction under examination, and is derived by eigenvalue analysis, K_0 is the corresponding elastic lateral stiffness, and K_e is the equivalent lateral stiffness.

Appendix A.5 – TBDY

In this appendix the parameters used for the structures assessment according to the Turkish Earthquake Building Regulations-TBDY are presented.

Type of Analysis

Current practice in Turkey is regulated by the TBDY: Turkish Seismic Building Regulations in combination with TS500: Requirements for Design and Construction of Reinforced Concrete structures.

According to TBDY, for the assessment and redesign of a building, one of the following analysis methods may be used:

- Equivalent Seismic Load Linear method according to section 4.7 of TBDY;
- Modal Analysis Linear method according to section 4.8 of TBDY;
- Nonlinear static (pushover) method according to section 5.6 of TBDY;
- Nonlinear Time History method according to section 5.7 of TBDY.

In SeismoBuild the most common method in assessment practice of existing buildings is employed, which is the nonlinear static analysis. It is based on pushover analyses carried out under constant gravity loads and increasing lateral forces, applied at the location of the masses to simulate the inertia forces induced by the seismic action. As the model may account for both geometrical and mechanical nonlinearity, this method can describe the evolution of the expected plastic mechanisms and structural damage.

Each pushover analysis leads to a capacity curve, which is a relationship between the total base shear and the horizontal displacement of a representative point of the structure, termed “control node”. The demand at the considered Performance Level – Continuous Use, Immediate Occupancy, Life Safety or Collapse Prevention - is determined by the appropriate comparison between the capacity determined by the pushover curve and the demand established as the damped Linear Response Spectrum. To do so, the “control node” displacements are defined in terms of spectral quantities relative to an equivalent single-degree-of-freedom (SDOF) system which is derived from the multi-degree-of-freedom (MDOF) response estimated according to Appendix 5B of TBDY.

The structural demand associated with the acquired target displacement shall fulfil the verification criteria defined in TBDY. Accordingly, element’s demand for force-controlled (shear) and deformation-controlled (chord rotation or strain deformation) actions are deemed to comply with limits that take into account: section mechanical properties; element’s bending, shear and axial force interaction; and strength/stiffness degradation associated with the ductility demand and cyclic hysteretic response of reinforced concrete elements, through appropriate material nonlinearity consideration.

Performance Requirements

According to TBDY sections 2.2, 3.4 and 3.5, the objectives of the assessment or redesign consist of combinations of both a performance level and a seismic action, given an "acceptable probability of exceedance within the life cycle of the building" (design earthquake), as shown in the table below.

Seismic Hazard Level	Target Building Performance Levels			
	Continuous Use (KK)	Immediate Occupancy (HK)	Life Safety (CG)	Collapse Prevention (BP)
DD-4 (68%/50 years)	a	b	c	d
DD-3 (50%/50 years)	e	f	g	h
DD-2 (10%/50 years)	i	j	k	l
DD-1 (2%/50 years)	m	n	o	p

Building Performance Levels.

The target building performance levels refer to the state of damage in the structure defined through four limit states, namely Continuous Use (KK), Immediate Occupancy (HK), Life Safety (CG) and Collapse Prevention (BP).

Performance Level of Continuous Use (KK)

The Continuous Use after the earthquake (KK), according to TBDY, is a condition in which it is expected that damage is insignificant and structure does not need any repair measures. Structural elements are prevented from significant yielding and retaining their strength and stiffness properties. Non-structural components, such as partitions and infills should not be damaged.

Performance Level of Immediate Occupancy (HK)

The Immediate Occupancy (HK), according to TBDY, is a condition in which it is expected that no building operation is interrupted during and after the design earthquake, with the possible exception of minor importance functions. A few hairline cracks may occur in the structure.

Performance Level of Life Safety (CG)

The Life Safety (CG), according to TBDY, is a condition in which repairable damage to the structure is expected to occur during the design earthquake, without causing loss or serious injury of people and without substantial damage to personal property or materials that are stored in the building.

Performance Level of Collapse Prevention (BP)

The Collapse Prevention (BP), according to TBDY, is a condition in which extensive and serious or severe (non-repairable, in general) damage to the structure is expected during the design earthquake; however, the structure retains its ability to bear the prescribed vertical loads (during and for a period after the earthquake), in any case without other substantial safety factor against total or partial collapse.

The criteria for the selection of the Performance Objectives may be found in TBDY.

Information for Structural Assessment

In order to choose the admissible type of analysis and the appropriate confidence factor values, the following two knowledge levels are defined:

- Limited Knowledge

- Comprehensive Knowledge

The factors determining the obtained data reliability level are the (i) geometry, which is the geometrical properties of the structural system and of non-structural elements, e.g. masonry infill panels, that may affect structural response; (ii) details, which include the amount and detailing of reinforcement in reinforced concrete sections, the connection of floor diaphragms to lateral resisting structure, the bond and mortar jointing of masonry and the nature of any reinforcing elements in masonry; and finally (iii) materials, that is the mechanical properties of the constituent materials.

Limited Knowledge

The limited knowledge level corresponds to a state of knowledge where information is obtained from design drawings with sufficient information to analyse component demands and calculate component capacities. The design drawings show the configuration of the gravity load system and seismic-force-resisting system with sufficient details. Information is verified by a visual condition assessment. In the absence of sufficient information from design drawings, incomplete or non-existent information is supplemented by a comprehensive condition assessment, including destructive and non-destructive investigation.

Comprehensive Knowledge

The comprehensive knowledge level corresponds to a state of knowledge where information is obtained from construction documents including design drawings, specifications, material tests records, and quality assurance reports covering original construction and subsequent modifications to the structure. Information is verified by a visual condition assessment. In cases where construction documents are incomplete, missing information is supplemented by comprehensive condition assessment, including destructive and non-destructive investigation. In the absence of material test records and quality assurance reports, material properties are determined by comprehensive materials testing in accordance to TBDY, Chapter 15.

Knowledge Factors

In the following table of section 15.2.12 of TBDY the confidence factors for each knowledge level are shown.

Level of Knowledge	Knowledge Factor
Limited	0.75
Comprehensive	1.00

Table 15.1 of TBDY – Knowledge Factors

Safety Factors

In TBDY the safety factors are directly incorporated in the member's strengths and deformation limits.

Capacity Models for Assessment and Checks

All the member checks (chord rotation capacity, strain capacity and shear capacity) should be carried out for all the elements of every floor, according to sections 5 and Appendix 15 of TBDY. Moreover, beam-column joints checks can be employed in order to check the joint's shear forces.

Chord Rotation Capacity

The chord rotation capacity of beams, columns and walls, θ, that is the angle between the tangent to the axis at the yielding end and the chord connecting that end with the end of the shear span (L_s=M/V=moment/shear at the end section). The chord rotation is also equal to the element drift ratio, which is the deflection at the end of the shear span with respect to the tangent to the axis at the yielding end divided by the shear span.

Chord rotation capacity of beams and columns is highly influenced by the lack of appropriate seismic resistant detailing in longitudinal reinforcement, as well as whether there are smooth bars. Inadequate development of splicing along the span (beams) and height (columns), and inadequate embedment into beam-column joints can control the members' response to seismic action, drastically limiting its capacity in respect to the situation in which the reinforcement is considered fully effective. The above limitations to the deformation capacity are taken into consideration.

The value for the chord rotation capacity for the performance levels of continuous use (KK) and immediate occupancy (HK) is the value of the chord rotation capacity at flexural yield, θ_y, which is calculated from the equations 5.3 and 5.8b of TBDY:

$$\theta_y = \frac{\varphi_y L_s}{3} + 0{,}0015\eta\left(1 + 1{,}5\frac{h}{L_s}\right) + \frac{\varphi_y d_b f_{ye}}{8\sqrt{f_{ce}}} \quad \text{(5.3) TBDY}$$

Where L_s is the ratio between bending moment, M, and shear force, V; and η is equal to 1,0 for beams and columns and 0,5 for walls.

$$\theta_p^{(HK)} = 0 \quad \text{(5.8b) TBDY}$$

The value for the chord rotation capacity for the performance level of life safety (CG) is calculated from the following equation:

$$\theta^{(CG)} = \theta_y + \theta_p^{(CG)}$$

Where θ_y is calculated according to (5.3) equation and $\theta_p^{(CG)}$ according to (5.7b) equation of TBDY:

$$\theta_p^{(CG)} = 0{,}75\theta_p^{(GO)} \quad \text{(5.7b) TBDY}$$

The value $\theta_p^{(GO)}$ is calculated from the following equation:

$$\theta_p^{(GO)} = \frac{2}{3}\left[(\varphi_u - \varphi_y)L_p\left(1 - 0{,}5\frac{L_p}{L_s}\right)\right] + 4{,}5\varphi_u d_b \quad \text{(5.6) TBDY}$$

The value for the chord rotation capacity for the performance level of collapse prevention (BP) is the value of the chord rotation capacity at failure, which is calculated as the sum of the chord rotation at yielding and the plastic part of the chord rotation capacity, according to equations 5.3 and 5.6 of TBDY.

$$\theta_u = \theta_y + \theta_p$$

The yield curvature of the end section is calculated according to the following equation (Appendix 7A of KANEPE), for the sections whose compressive zone is of constant width and for the case that the section's yielding is due to steel yielding.

$$\varphi_y = (1/r)_y = \frac{f_y}{E_s(1-\xi_y)d} \quad \text{(A.1) KANEPE}$$

If the section yields due to the deformation nonlinearities of the concrete in compression, that is for deformation of the edge compressive fibre larger than $\varepsilon_c \approx 1.8\, f_c/E_c$, then the yield curvature is calculated according to the following expression, of Appendix 7A of KANEPE:

$$\varphi_y = (1/r)_y = \frac{\varepsilon_c}{\xi_y d} \approx \frac{1.8 f_c}{E_c \xi_y d} \quad \text{(A.2) KANEPE}$$

The lower from the two values above is used for the calculation of the chord rotation capacity.

The equations proposed by D. Biskinis [2007] are employed for the calculation of the ultimate curvature at the end section. If the failure is due to steel rupture, then the ultimate curvature is calculated according to the following expression:

$$\varphi_{su} = \frac{\varepsilon_{su}}{(1 - \xi_{su})d}$$

If the section fails by crushing of the extreme concrete fibres, then the ultimate curvature is calculated according to the following expression:

$$\varphi_{cu} = \frac{\varepsilon_{cu}}{\xi_{cu} d}$$

Users are advised to refer to the relevant publications for the definition of the other parameters and further details on the expression.

In the case of circular column sections, the equations above cannot be employed for the calculation of the elements' yield and ultimate curvatures. In SeismoBuild the equations suggested by D. Biskinis and M. N. Fardis [2013] are employed for φ_y and φ_u.

Concrete Jacketing

The values of the jacketed members for θ_y^* and θ_u^* that are adopted in the capacity verifications depend on the corresponding values calculated under the requirements of section 5 of TBDY, according to the following limitations of section 15.10.1 of TBDY:

The chord rotation at yield:

$$\theta_y^* = 0.90\theta_y$$

The ultimate chord rotation:

$$\theta_u^* = 0.90\theta_u$$

FRP wrapping

The contribution of the FRP wrapping to members' capacity is taken into account in the calculation of the yield and ultimate curvature due to concrete failure.

Strains Capacity

The value for the strain capacity for the performance levels of continuous use (KK) and immediate occupancy (HK) is defined by the following equation 5.8a of TBDY:

$$\varepsilon_c = 0.0025 \quad \text{and} \quad \varepsilon_s = 0.0075 \qquad \text{(5.8a) TBDY}$$

The value for the strain capacity for the performance level of life safety (CG) is calculated from the following equation:

$$\varepsilon_c^{(CG)} = 0.75\varepsilon_c^{(GO)} \quad \text{and} \quad \varepsilon_s^{(CG)} = 0.75\varepsilon_s^{(GO)} \qquad \text{(5.7a) TBDY}$$

Where $\varepsilon_c^{(GO)}$ is calculated according to the following equations of TBDY:

For rectangular columns, beams and walls:

$$\varepsilon_c^{(GO)} = 0.0035 + 0.04\sqrt{\omega_{we}} \leq 0.018 \qquad \text{(5.4a) TBDY}$$

For circular columns:

$$\varepsilon_c^{(GO)} = 0.0035 + 0.07\sqrt{\omega_{we}} \leq 0.018 \qquad \text{(5.4b) TBDY}$$

and $\varepsilon_s^{(GO)}$ is calculated according to equation 5.5 of TBDY:

$$\varepsilon_c^{(GO)} = 0.75\varepsilon_{su} \qquad \text{(5.5) TBDY}$$

FRP wrapping

The contribution of the FRP wrapping to members' strain capacity is taken into account according to section 15B.3 of the Appendix 15B of TBDY.

Bending Moment Capacity

The bending moment capacity of beams, columns and walls with prismatic cross section is calculated according to paragraph 4.1 of D. Biskinis and M. N. Fardis (2009), while for the bending moment capacity of circular columns paragraph 4 of Biskinis and M. N. Fardis (2013) is employed.

Shear Capacity

Shear capacity is calculated through the following expression according to Appendix 15B of TBDY, as controlled by the stirrups, accounting also for the increment due to the FRP wrapping.

$$V_r = V_c + V_w + V_f \leq V_{max} \qquad \text{(15B.1) TBDY}$$

where V_c is calculated according to equations 8.1 and 8.4 of TS500:

$$V_{cr} = 0.65 f_{ctd} b_w d \left(1 + \gamma \frac{N_d}{A_c}\right) \qquad \text{(8.1) TS500}$$

$$V_c = 0.8 V_{cr} \qquad \text{(8.4) TS500}$$

The contribution of the transverse reinforcement to the shear capacity is calculated according to equation 8.5 of TS00:

$$V_w = \frac{A_{sw}}{s} f_{ywd} d \qquad \text{(8.5) TS500}$$

The contribution of the FRP jacket to the shear resistance is computed through the following expression:

$$V_f = \frac{2\, n_f\, t_f\, w_f\, E_f\, \varepsilon_f\, d}{s_f} \qquad \text{(15B.2) TBDY}$$

The shear strength of a member may not be taken greater than the value corresponding to failure by web crushing, V_{max}, which is calculated according to the following expression:

$$V_{max} \leq 0.22\, f_{cd} b_w d \qquad \text{(8.7) TS500}$$

Concrete Jacketing

The value for the shear capacity, V_R^*, of the jacketed members that is adopted in the capacity verifications depend on the corresponding value calculated under the requirements of the Appendix 15B of TBDY, according to the following limitations of section 15.10.1 of TBDY:

$$V_R^* = 0.9 V_R$$

FRP wrapping

The contribution of the FRP jacket to the shear resistance is taken into account according to section 15B.3 of the Appendix 15B of TBDY, as shown above.

Steel Braces Axial Deformations

Axial deformations of brace members in tension and compression should satisfy the provisions of table 5C.4 of TBDY.

Steel Braces Axial Forces

Axial forces of brace members in tension and compression should satisfy, according to ASCE, the provisions of table C3.4 of AISC 342-22 and chapters D and E of AISC360-16.

Joints Shear Force

The design shear force of joints is calculated through the following expression according to TBDY:

$$V_e = 1.25\, f_{yk}(A_{s1} + A_{s2}) - V_{kol} \qquad \text{(7.11) TBDY}$$

The option to consider re-bar stresses from analyses rather than the yielding stresses for the calculation of the design shear force of joints in nonlinear analysis is available in the Elements tab of the Advanced Settings. In this case, the expression for the design shear force of joints is the following:

$$V_e = (\Sigma A_{s1i} \cdot \sigma_{1i} + \Sigma A_{s2i} \cdot \sigma_{2i}) - V_{kol}$$

The value for the design shear force of joints should be less than their shear capacity as shown below. Two different expressions are employed, according to section 7.5 of TBDY, depending on whether the joints are with or without confinement from transverse reinforcement, as defined in figure 7.10 of TBDY.

For joints with confinement from transverse reinforcement:

$$V_e \le 1.7 b_j h \sqrt{f_{ck}} \qquad \text{(7.12) TBDY}$$

For joints without confinement from transverse reinforcement:

$$V_e \le 1.0 b_j h \sqrt{f_{ck}} \qquad \text{(7.13) TBDY}$$

Joints Ductility

Adequate ductility should be possessed by both the structural elements and the structure as a whole according to section 18.7.3 of EN ACI 318-19. In frame buildings with two or more storeys, the following condition should be satisfied at all joints of primary or secondary seismic beams with primary seismic columns:

$$\Sigma M_{nc} \ge (6/5)\Sigma M_{nb} \qquad \text{(18.7.3.2) ACI 318-19}$$

Where ΣMnc is the sum of the design values of the moments of resistance of the columns framing the joint and ΣMnb is the sum of the design values of the moments of resistance of the beams framing the joint. The joints ductility check is not employed for the joints of the top level of multistorey buildings according to section 18.7.3.1 of ACI 318-19.

Footings Rocking Moment Capacity

Rocking moment capacity is verified according to ASCE 41-23, section 8.4.5.2.

Footings Rocking Rotation Capacity

Rocking rotation capacity is verified according to ASCE 41-23, section 8.4.5.3.

Footings Bending Capacity

Bending moment capacity check is performed according to ACI 318-19. Bending moment demand is calculated by pure stress σnet acting on the horizontal base of footing.

Footings Shear Capacity

Shear capacity check is performed according to ACI 318-19, section 22.5.5.1. Shear demand is calculated by pure stress σ_{net} acting on the horizontal base of footing.

Footings Punching Capacity

Punching capacity check is carried out as described in ACI 318-19, section 22.6.5.

Footings Eccentricity

Eccentricity of loading should not exceed 1/3 of the dimension in each direction of footing according to EN 1997-1, section 6.5.4. Double eccentricity check is verified if the sum of squares of loading eccentricities in 2 horizontal directions is less than 1/9.

CAPACITY CURVE

Each pushover analysis leads to a capacity curve, which is a relationship between the total base shear and the horizontal displacement of a representative point of the structure, termed "control node", with the values of the control displacement ranging between zero and a maximum value defined by the user.

TARGET DISPLACEMENT

The target displacement is defined as the seismic demand derived from the elastic response spectrum in terms of displacement of an equivalent single-degree-of-freedom system. To define the target displacement of a MDOF system a number of steps have to be followed according to Appendix 5B of TBDY.

The target displacement shall be calculated in accordance with equation (5B.12) of TBDY.

$$d_{1,max}^{(X)} = S_{di}(T_1) \qquad \text{(5B.12) TBDY}$$

where

$$S_{di}(T_1) = C_R S_{de}(T_1) \qquad \text{(5B.13) TBDY}$$

$S_{de}(T_1)$ is the elastic spectral pseudo-acceleration corresponding to the equivalent fundamental period of the structure T_1 and C_R is a modification factor to relate expected maximum inelastic displacements to displacements calculated for linear elastic response. The value for C_R is computed according to the equation below:

$$C_R = \frac{\mu(R_y, T_1)}{R_y} \qquad \text{(5B.14) TBDY}$$

where R_y is the yield reduction factor calculated from the following equation:

$$R_y = \frac{f_e}{f_y} = \frac{S_{ae}(T_1)}{\alpha_{y1}} \qquad \text{(5B.15) TBDY}$$

The equation (5B.14) for the modification factor C_R takes the following form by using the equations (5B.16) of TBDY:

$$C_R = 1 \text{ for } T_1 > T_B \qquad \text{(5B.17a) TBDY}$$

$$C_R = \frac{1 + (R_y - 1)\frac{T_B}{T_1}}{R_y} \text{ for } T_1 \leq T_B \qquad \text{(5B.17b) TBDY}$$

Determination of the idealized elasto-perfectly plastic force-displacement relationship

The yield force f_y, which represents also the ultimate strength of the idealized SDOF system, is equal to the base shear force at the formation of the plastic mechanism. The initial stiffness of the idealized system is determined in such a way that the areas under the actual and the idealized force-deformation curves are equal.

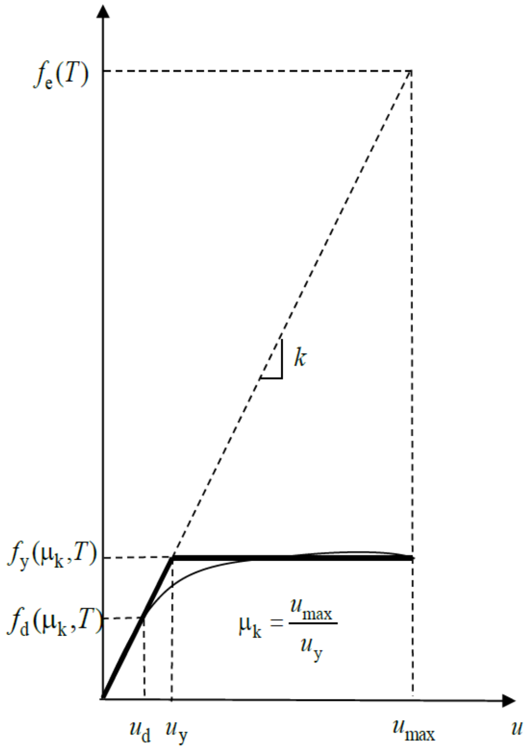


Figure 4A.1 of TBDY Determination of the idealized elasto-perfectly plastic force-displacement relationship

Appendix B - Theoretical background and modelling assumptions

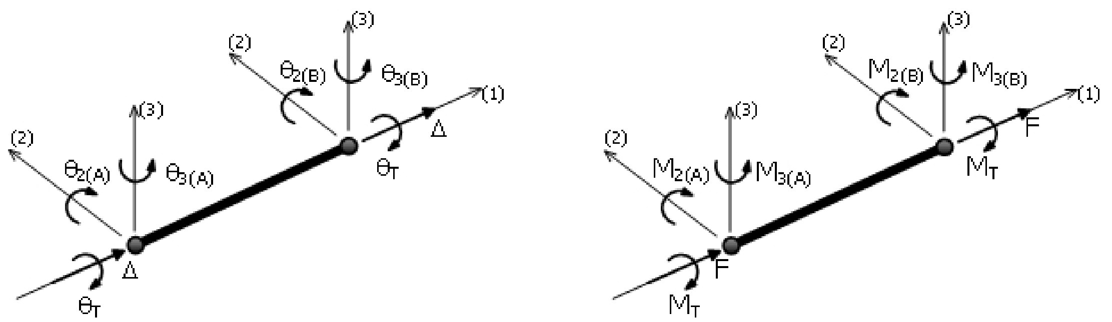
This appendix serves the purpose of providing users with a brief overview of the theoretical foundations and modelling conventions in SeismoBuild, furnishing also pointers to a number of publications where further and deeper explanations and discussion can be found.

GEOMETRIC NONLINEARITY

Large displacements/rotations and large independent deformations relative to the frame element's chord (also known as P-Delta effects) are taken into account in SeismoBuild, through the employment of a total co-rotational formulation developed and implemented by Correia and Virtuoso [2006].

The implemented total co-rotational formulation is based on an exact description of the kinematic transformations associated with large displacements and three-dimensional rotations of the beam-column member. This leads to the correct definition of the element's independent deformations and forces, as well as to the natural definition of the effects of geometrical nonlinearities on the stiffness matrix.

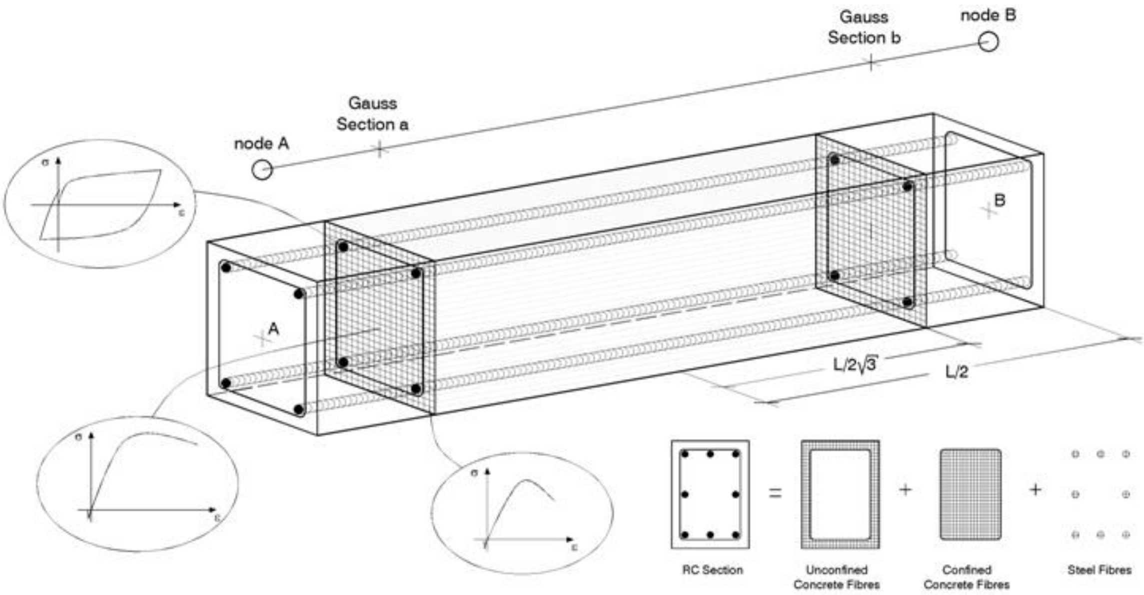
The implementation of this formulation considers, without losing its generality, small deformations relative to the element's chord, notwithstanding the presence of large nodal displacements and rotations. In the local chord system of the beam-column element, six basic displacement degrees-of-freedom ($\theta_{2(A)}$, $\theta_{3(A)}$, $\theta_{2(B)}$, $\theta_{3(B)}$, Δ, θ_T) and corresponding element internal forces ($M_{2(A)}$, $M_{3(A)}$, $M_{2(B)}$, $M_{3(B)}$, F, M_T) are defined, as shown in the figure below:



Local chord system of the beam-column element

MATERIAL INELASTICITY

Distributed inelasticity elements are becoming widely employed in earthquake engineering applications, either for research or professional engineering purposes. Whilst their advantages in relation to the simpler lumped-plasticity models, together with a concise description of their historical evolution and discussion of existing limitations, can be found in e.g. Filippou and Fenves [2004] or Fragiadakis and Papadrakakis [2008], here it is simply noted that distributed inelasticity elements do not require (not necessarily straightforward) calibration of empirical response parameters against the response of an actual or ideal frame element under idealized loading conditions, as is instead needed for concentrated-plasticity phenomenological models. In SeismoBuild, use is made of the so-called fibre approach to represent the cross-section behaviour, where each fibre is associated with a uniaxial stress-strain relationship; the sectional stress-strain state of beam-column elements is then obtained through the integration of the nonlinear uniaxial stress-strain response of the individual fibres (by default 150) in which the section has been subdivided (the discretisation of a typical reinforced concrete cross-section is depicted, as an example, in the figure below). Such models feature additional assets, which can be summarized as: no requirement of a prior moment-curvature analysis of members; no need to introduce any element hysteretic response (as it is implicitly defined by the material constitutive models); direct modelling of axial load-bending moment interaction (both on strength and stiffness); straightforward representation of biaxial loading, and interaction between flexural strength in orthogonal directions.

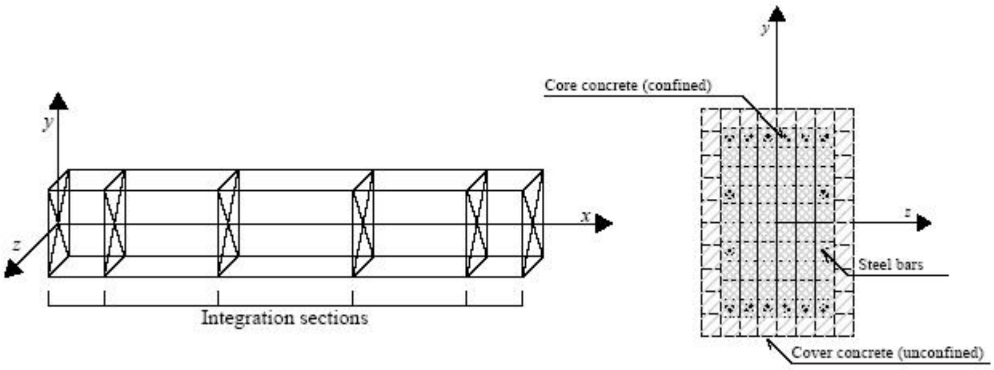


Discretisation of a typical reinforced concrete cross-section

Distributed inelasticity frame elements can be implemented with two different finite elements (FE) formulations: the classical displacement-based (DB) ones [e.g. Hellesland and Scordelis 1981; Mari and Scordelis 1984], and the more recent force-based (FB) formulations [e.g. Spacone et al. 1996; Neuenhofer and Filippou 1997].

In a DB approach the displacement field is imposed, whilst in a FB element equilibrium is strictly satisfied and no restraints are placed to the development of inelastic deformations throughout the member; see e.g. Alemdar and White [2005] and Freitas et al. [1999] for further discussion. In the DB case, displacement shape functions are used, corresponding for instance to a linear variation of curvature along the element.

In contrast, in a FB approach, a linear moment variation is imposed, i.e. the dual of previously referred linear variation of curvature. For linear elastic material behaviour, the two approaches obviously produce the same results, provided that only nodal forces act on the element. On the contrary, in case of material inelasticity, imposing a displacement field does not enable to capture the real deformed shape since the curvature field can be, in a general case, highly nonlinear. In this situation, with a DB formulation a refined discretisation (meshing) of the structural element (typically 4-5 elements per structural member) is required for the computation of nodal forces/displacements, in order to accept the assumption of a linear curvature field inside each of the sub-domains. Still, in the latter case users are not advised to rely on the values of computed sectional curvatures and individual fibre stress-strain states. Instead, a FB formulation is always exact, since it does not depend on the assumed sectional constitutive behaviour. In fact, it does not restrain in any way the displacement field of the element. In this sense this formulation can be regarded as always "exact", the only approximation being introduced by the discrete number of the controlling sections along the element that are used for the numerical integration. A number of 4 Gauss-Lobatto integration sections are required to avoid under-integration, option which will in general simulate the spread of inelasticity in an acceptable way. Such feature enables to model each structural member with a single FE element, therefore allowing a one-to-one correspondence between structural members (beams and columns) and model elements. In other words, no meshing is theoretically required within each element, even if the cross section is not constant. This is because the force field is always exact, regardless of the level of inelasticity.



Gauss-Lobatto integration sections

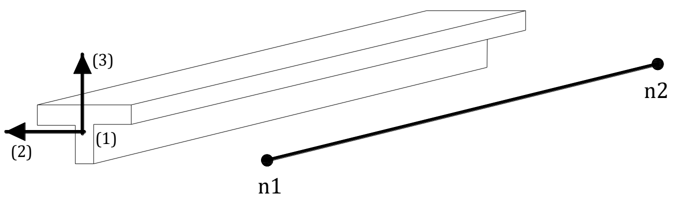
In SeismoBuild, both aforementioned DB and FB element formulations are implemented, with the latter being typically recommended, since, as mentioned above, it does not in generally call for element discretisation, thus leading to considerably smaller models, with respect to when DB elements are used, and thus much faster analyses, notwithstanding the heavier element equilibrium calculations. An exception to this non-discretisation rule arises when localisation issues are expected, in which case special cautions/measures are needed, as discussed in Calabrese et al. [2010].

Finally, it is noted that, for reasons of higher accuracy, the Lobatto quadrature is used. The approximate coordinates along the element's length (measured from its barycentre) of the integration sections are for 4 integration sections: [-1 -0.447 0.447 1] x L/2.

NOTE: Users are also invited to read the NEHRP Seismic Design Technical Brief No. 4 (i.e. Deierlein G.G., Reinhorn A.M., and Willford M.R. [2010]), in which the nonlinear modelling is well covered.

GLOBAL AND LOCAL AXES SYSTEM

In SeismoBuild, a fixed X-Y-Z global axis system is in place, used to define length (X), depth (Y) and height (Z) of all structural models. In addition, and being a 3D modelling program, SeismoBuild requires also that local 1-2-3 coordinate systems are assigned to all structural elements, so that their orientation in space is known. By convention, local direction (1) refers to the chord axis of the element, whilst axes (2) and (3) define the plane of the cross-section and its orientation. Although there are no constraints imposed on the definition of local axes (2) and (3), it is common for users to associate axis (2) to the "weak direction" of the member and to link axis (3) to the "strong direction" of the element, as illustrated below, where a beam is schematically represented. This is the convention also adopted in the illustrative drawings employed in the Modify/View additional reinforcement window available in the Building Modeller sections' Properties Window.



Definition of a beam element with a T-section (local direction (1) along the chord axis)

Whilst the orientation of local vector (1) results unambiguously characterised by the line joining the two end-nodes of the element (positive direction is that going from node n1 to node n2), the so-called right-hand rule is employed in order to fully describe the orientation of the two other remaining local axes, and thus that of the cross-section.

NOTE: For column members the orientation of the elements is automatically defined by the program. The default orientation for the local vector (1) is from the lower to the upper floor, i.e. the node n1 is the node of the lower floor and the node n2 is the node of the floor above.

NONLINEAR SOLUTION PROCEDURE

True structural behaviour is inherently nonlinear, characterised by non-proportional variation of displacements with loading, particularly in the presence of large displacements or material nonlinearities. Hence, in SeismoBuild, all analyses (with the obvious exception of eigenvalue procedure) are treated as potentially nonlinear, implying the use of an incremental iterative solution procedure whereby loads are applied in pre-defined increments, equilibrated through an iterative procedure.

Incremental iterative algorithm

The solution algorithm is fairly flexible since it allows the employment of Newton-Raphson (NR), modified Newton-Raphson (mNR) or NR-mNR hybrid solution procedures. It is clear that the computational savings in the formation, assembly and reduction of the stiffness matrix during the iterative process can be significant when using the mNR instead of the NR procedures. However, more iterations are often required with the mNR, thus leading in some cases to an excessive computational effort. For this reason, the hybrid approach, whereby the stiffness matrix is updated only in the first few iterations of a load increment, does usually lead to an optimum scenario.

The iterative procedure follows the conventional schemes employed in nonlinear analysis, whereby the internal forces corresponding to a displacement increment are computed and convergence is checked. If no convergence is achieved, then the out-of-balance forces (difference between applied load vector and equilibrated internal forces) are applied to the structure, and the new displacement increment is computed. Such loop proceeds until convergence has been achieved (log flag message equal to Converg) or the maximum number of iterations, specified by the user, has been reached (log flag message equal to Max_Ite).

For further discussion and clarifications on the algorithms described above, users are strongly advised to refer to available literature, such as the work by Cook et al. [1988], Crisfield [1991], Zienkiewicz and Taylor [1991], Bathe [1996] and Felippa [2002], to name but a few.

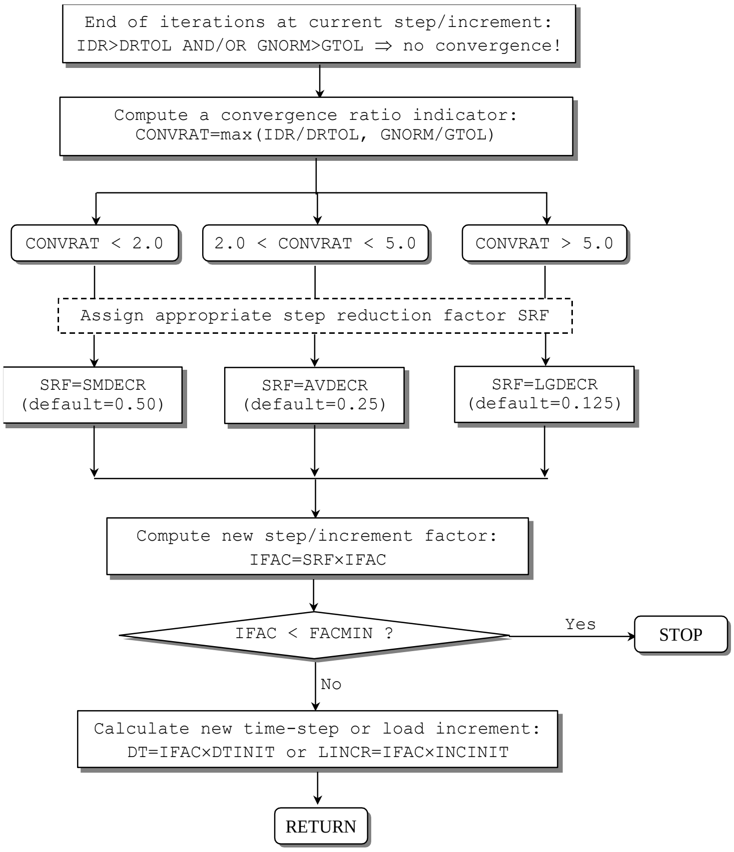
NOTE: Individual force-based frame elements require a number of iterations to be carried in order for internal equilibrium to be reached. In some cases, the latter element loop equilibrium cannot be reached, as signalled by log flag messages elm_inv and elm_ite. Refer to *Analysis Parameters > Analysis Settings > Elements* menu for further information on this issue.

Automatic adjustment of load increment or time-step

As discussed in the previous paragraph, for each increment, several iterations are carried out until convergence is achieved. If convergence is not reached within the specified maximum number of iterations, the load increment (or time-step) is reduced and the analysis is restarted from the last point of equilibrium (end of previous increment or time-step). This step reduction, however, is not constant but rather adapted to the level of non-convergence verified.

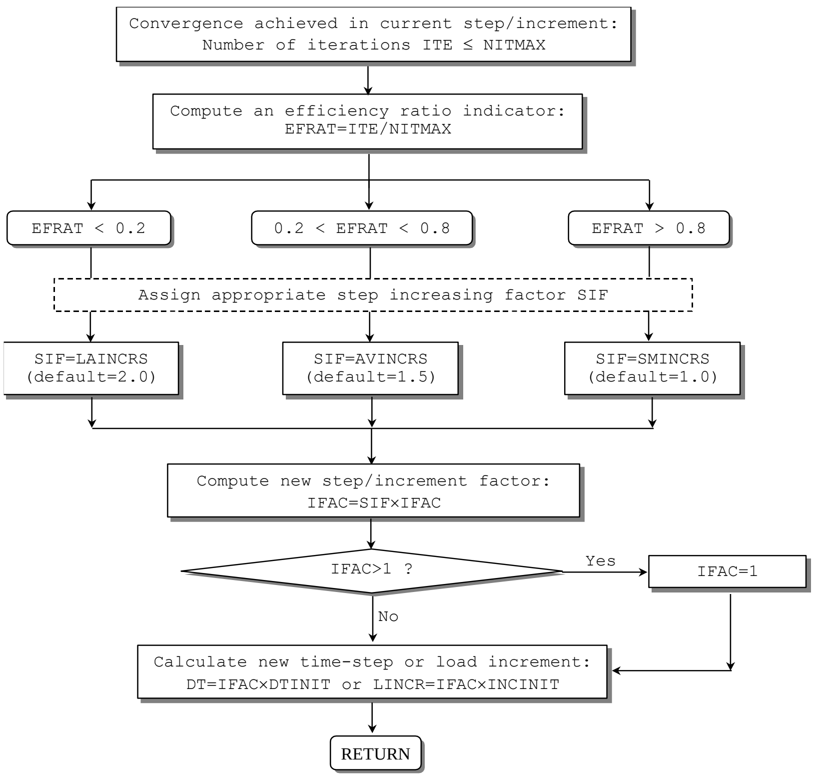
As illustrated below, at the end of a solution step or increment, a convergence ratio indicator (convrat), defined as the maximum of ratios between the achieved and the required displacement/force convergence factors (see convergence), is computed. Then, depending on how far away the analysis was from reaching convergence (convrat = 1.0), a small, average or large step reduction factor (srf) is adopted and employed in the calculation of the new step factor (ifac). The product between the latter and the initial time-step or load increment, defined by the user at the start of the analysis, yields the reduced analysis step to be used in the subsequent increment.

It is however noteworthy that, in order to prevent ill-defined analysis (which never reach convergence) to continue on running indefinitely, a user-defined lower limit for the step factor (facmin) is imposed and checked upon. If ifac results smaller than facmin then the analysis is terminated.



To minimise duration of analyses, it is fundamental that once convergence is reached, the load increment or time-step can be gradually increased. For this reason, an efficiency ratio indicator (efrat), defined as the ratio between the number of iterations carried out (ite) to reach convergence and the maximum number of iterations that were allowed (nitmax), is calculated. Depending on how far the analysis was from 'efficiency' (efrat > 0.8), a small, average or large step increasing factor (sif) is adopted and employed in the calculation of the new step factor (ifac). The product between the latter and the initial time-step or load increment, defined by the user at the start of the analysis, yields the augmented analysis step to be used in the subsequent increment.

It is however noteworthy that the step factor is upper-bounded by a value of 1, so as to ensure that the time step or load increment do not become larger than its initial counterpart, defined by the user at the start of the analysis.



Numerical instability, divergence and iteration prediction

In addition to the convergence verification, at the end of an iterative step three other solution checks may be carried out; numerical instability, solution divergence and iteration prediction. These criteria, all of a force/moment nature, serve the purpose of avoiding the computation of useless equilibrium iterations in cases where it is apparent that convergence will not be reached, thus minimising the duration of the analysis.

Numerical instability

The possibility of the solution becoming numerically unstable is checked at every iteration by comparing the Euclidean norm of out-of-balance loads, G_{norm}, with a pre-defined maximum tolerance (default=1.0E+20), several orders of magnitude larger than the applied load vector. If G_{norm} exceeds this tolerance, then the solution is assumed as being numerically unstable and iterations within the current increment are interrupted, with a log flag message equal to Max_Tol.

On occasions, very unstable models lead to the sudden development of out-of-balance forces that are several orders of magnitude larger than the maximum tolerance value. This in turn creates a so-called Solution Problem (i.e. the analysis crashes, albeit in a "clean manner"), and iterations within the current increment are interrupted, with a log flag message equal to Sol_Prb.

Solution divergence

Divergence of the solution is checked by comparing the value of G_{norm} obtained in the current iteration with that obtained in the previous one. If G_{norm} has increased, then it is assumed that the solution is diverging and iterations within the current increment are interrupted, with a log flag message equal to Diverge.

Iteration prediction

Finally, a logarithmic convergence rate check is also carried out, so as to try to predict the number of iterations (itepred) required for convergence to be achieved. If itepred is larger than the maximum number of iterations specified by the user, then it is assumed that the solution will not achieve convergence and iterations within the current increment are interrupted, with a log flag message equal to Prd_Ite.

The following equation is used to compute the value of itepred, noting that ite represents the current number of iterations and G_{tol} is the force/moment tolerance:

$$\text{itepred} = \text{ite} + \frac{\log\left(\text{G}_{\text{tol}} / \text{G}_{\text{norm}}^{\text{ite}}\right)}{\log\left(\text{G}_{\text{norm}}^{\text{ite}} / \text{G}_{\text{norm}}^{\text{ite}-1}\right)}$$

The three checks described above are usually reliable and effective within the scope of applicability of SeismoBuild, for as long as the divergence and iteration prediction check is not carried out during the first iterations of an increment when the solution might not yet be stable enough. This issue is discussed in further detail in the iterative strategy section, where all user-defined parameters related to these criteria are described.

NOTE: Individual force-based frame elements require a number of iterations to be carried in order for internal equilibrium to be reached. In some cases, the latter element loop equilibrium cannot be reached, as signalled by log flag messages elm_inv and elm_ite. Refer to *Analysis Settings > Elements* menu for further information on this issue.

List of SeismoBuild Convergence and Divergence Flags

Hereby a complete list of the messages that are output by the SeismoBuild solver in the case if divergence is provided, together with possible measures that the user can take, in order to make the analysis converge.

Converg: This message means that the analysis has converged in the current loading step, and is proceeding to the next step.

Max_Ite: This message is output, if the maximum number of iterations has been reached in the current loading step, and convergence has not been achieved yet. In such cases, either increase the maximum number of iterations (*Analysis Settings>Iterative Strategy*), increase the convergence criteria values (*Analysis Settings>Convergence Criteria*) or employ a less stringent type of convergence check (e.g. Displacement/Rotation based only scheme instead of Displacement/ Rotation based AND Force/Moment based).

Prd_Ite: This flag is similar to the Max_Ite message, the difference being that the solver does not wait until the maximum number iterations have been reached. Instead, it makes a prediction of the number of iterations that are expected to be needed for convergence, based on how the iterative solution is converging (i.e. size of out-of-balance forces, and how fast the convergence tolerance is being reached). If the predicted iterations are larger than the maximum iterations specified by the user, the Prd_Ite flag is output and the analysis diverges. In such cases, either increase the Maximum number of iterations (*Analysis Settings>Iterative Strategy*), choose a looser convergence criteria scheme with larger convergence tolerances (*Analysis Settings>Convergence Criteria*), or increase the analysis steps, as with the Max_Ite message. It is noted that Prd_Ite is the most common divergence flag.

Diverge: This flag is output when the iterative process in the current step is diverging, instead of converging to the solution. It is noted that the check for diverging solutions is always carried out after the Divergence Iteration that is specified by the user in Analysis Settings>Iterative Strategy. This is done because in general the solution procedures are unstable at the initial 3-4 steps, before they get stable and gradually converge to the solution. Users are advised to either increase the Divergence Iteration and the Maximum number of iterations from the Iterative Strategy page of the Analysis Settings, choose looser convergence criteria from the Convergence Criteria page, or increase the analysis steps.

elm_Ite: This message appears when the maximum number of iterations is reached in the internal element loop of the elements that require iterations on the element level (infrmFB and infrmFBPH), without internal equilibrium having been achieved. Users are advised to either increase the number of iterations or increase the convergence tolerance from the Elements tab of the Analysis Settings. Alternatively, the 'Do not allow element unbalanced forces in case of elm_ite' option may be unchecked. Finally, measure on the global level may be taken, for instance the analysis load step can be decreased (by increasing the analysis steps), and the global convergence criteria can be increased. Users are advised to refer to the specific documentation [e.g. Spacone et al. 1996; Neuenhofer and Filippou 1997] for a better understanding of the internal loops of the force-based elements.

elm_Inv: This message appears when the stiffness matrix of an element that employs internal iterations cannot be inverted during the internal element loops. In such cases, users are advised to increase the elements' convergence tolerance from the Element Iterative Strategy tab of the Analysis Settings, to increase the analysis steps or the global convergence tolerance values.

elm_Tol: This message appears when the maximum tolerance value, as specified in the Iterative Strategy page of the Analysis Settings, has been exceeded during the internal element loops of the force-based elements. Similar actions with the elm_Inv flag should be taken.

Max_Tol: This flag signifies solutions that become very unstable numerically with out-of-balance forces larger than the Maximum Tolerance (default=1.0E+20) that is specified in the Iterative Strategy page of the Analysis Settings. Users are advised to increase the analysis steps, or to adopt looser convergence criteria. Alternatively, the Maximum Tolerance may be increased, but its value should never exceed values of 1.0E+35 or 1.0E+40, whilst it is noted that in very few cases the latter will lead to stable solutions. If the Max_Tol message appears in first 2-3 steps of the analysis, or at the application of the initial loads, an eigenvalue analysis should be run, in order to confirm that all the members of the model are correctly connected to each other.

Sol_Prb: This message means that a solution of the analysis equations in the current iteration could not be found. There are numerous reasons for this behaviour, such as extreme values of out-of-balance forces or zero diagonal stiffness values. Similar measures to those suggested for the case of Max_Tol flag should be taken.

Tips to Solve Convergence Problems

Hereby a number of steps to follow for solving convergence problems are proposed. Users are advised to:

- Apply the automatic adaptation of the norms in the Convergence criteria tab of the program's Advanced Settings (Analysis Parameters>Advanced Settings).
- Select to show Convergence problems in the post-processor through the Analysis Parameters>Advanced Settings> Convergence criteria tab. The visualisation of the locations of the structure (elements or nodes), where the convergence difficulties arise, provides significant feedback for the identification of the reasons for divergence (e.g. under-reinforced beams that cannot sustain the gravity loads, elements with very high deformations demand, such as short columns or coupling beams, etc.).
- Uncheck the 'Do not allow unbalanced forces in case of elm_Ite' for both the force-based (infrmFB & infrmFBPH) element types in the Element Iterative Strategy tab of the Advanced Settings.
- Reduce the Maximum Interstorey Drift value in the Analysis tab of the Code Requirements. This value should not exceed 1.00 or 1.20% for tall buildings and for stiff buildings with large shear walls.
- Assign 100 pushover analysis steps in the Analysis tab of the Code Requirements. This value should be further increased in the cases, where significant loading is expected.
- Select the 'Apply Displacement Based Frame Elements To All Members With Length (m) <' in the Advanced Building Modelling tab of the program's Advanced Settings (Analysis

Parameters>Advanced Settings), in order to use the infrmDB element type for short members. This change typically leads to improved convergence.

- Increase the maximum number of iterations to 70, the number of stiffness updates to 60 and the divergence iteration to 60 in the Iterative Strategy tab of the Advanced Settings (Analysis Parameters>Advanced Settings).
- Use the elastic frame element type for the coupling beams that cause convergence problems. In such cases the elements' moment releases should be released by selecting the relevant checkboxes for the M2a, M3a, M2b and M3b degrees-of–freedom, through the Advanced Member Modelling Parameters of the member in the Building Modeller, in order to account for the formation of plastic hinges at the ends of the coupling beams.
- Increase the values of the convergence norms from the Convergence Criteria tab of the program's Advanced Settings (Analysis Parameters>Advanced Settings).
- Increase the rigidity of the rigid diaphragms to 1.0E+13 through the Constraints tab of the Advanced Settings.
- Select the Control Node to be in the side of the building with the larger deformation demand, through the Structural Modelling tab of the Building Modelling Settings inside the Building Modeller.
- If the divergence messages of the analysis are mostly Max_Tol or fbd_tol, increase the Maximum Tolerance value to 1e40 in the Iterative Strategy tab of the Advanced Settings (Analysis Parameters>Advanced Settings).
- Increase the number of fibres for the walls in the Modelling Parameters of the members inside the Building Modeller.
- For taller buildings uncheck the Include Geometric Nonlinearities checkbox in the Analysis tab of the Advanced Settings.

Moreover:

- users are advised to check the last or the 2-3 last steps of the analysis with convergence problems in order to understand and resolve the reasons for divergence. In such cases the Convergence Problems page of the post-processor should be advised. Furthermore, running an Eigenvalue analysis with the same model might offer valuable insight to the problem (e.g. identify a beam that is close to, but unconnected, to an adjacent column, and behaves as a cantilever, not being able to sustain the gravity load);
- it is noted that elements that cause divergence problems are not necessarily the ones that withstand significant loading. They are the ones that at the current step face increased tangential change of the deformation state/internal force re-distribution. Hence, sometimes failed elements can increase significantly the load sustained by adjacent elements, thus leading them to convergence difficulties, contrary to the failed elements themselves, which converge easily;
- the removal of the effective width of beams should also be considered by unchecking the 'Include Effective Width' checkbox in the Structural Modelling tab of the Building Modelling Settings inside the Building Modeller.

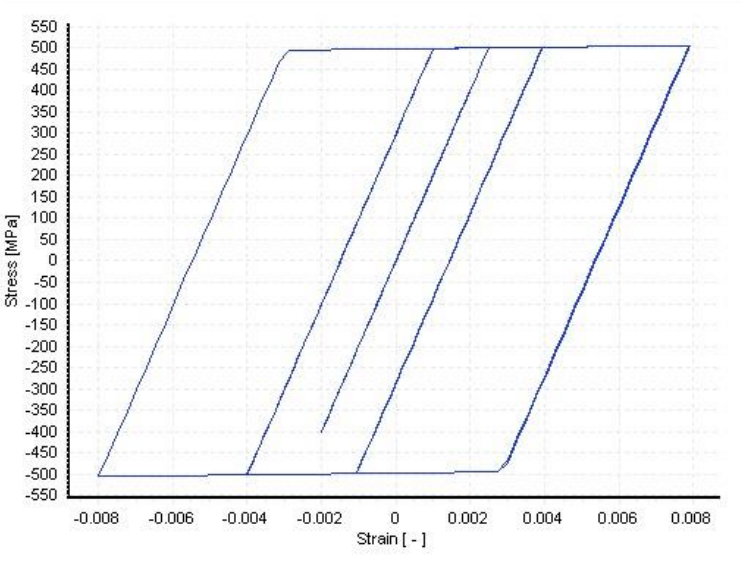
Appendix C – Materials

In this appendix the available material types are described in details.

STEEL MATERIALS

Bilinear steel model - stl_bl

This is a uniaxial bilinear stress-strain model with kinematic strain hardening, whereby the elastic range remains constant throughout the various loading stages, and the kinematic hardening rule for the yield surface is assumed as a linear function of the increment of plastic strain. This simple model is also characterised by its computational efficiency and can be used in the modelling of both steel structures, where mild steel is usually employed, as well as reinforced concrete models, where worked steel is commonly utilised. As discussed by Prota et al. [2009], with the correct calibration, this model, initially developed with ribbed reinforcement bars in mind, can also be employed for the modelling of smooth rebars, often found in existing structures.

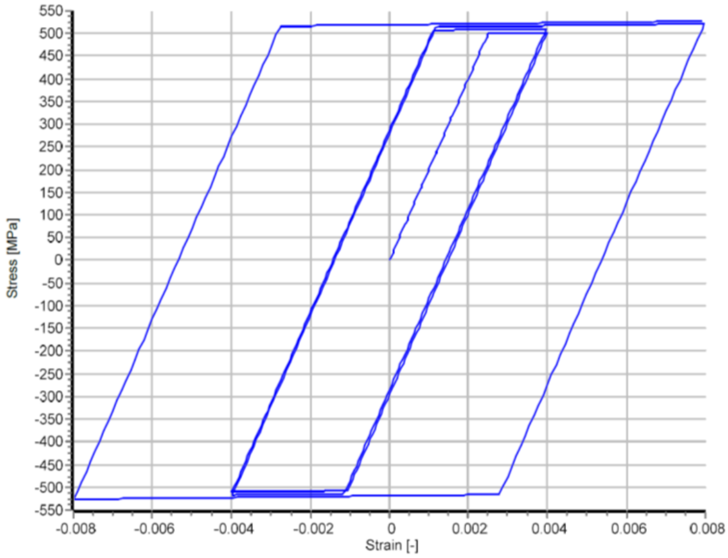


Bilinear steel model

Users have to assign the mean value of steel strength and the mean value minus the standard deviation or the characteristic value in order to describe the mechanical characteristics of the existing or the new material respectively.

Bilinear steel model with isotropic strain hardening- stl_bl2

This is a uniaxial bilinear stress-strain model characterized by a linear kinematic hardening rule and an optional feature of isotropic hardening which is described by a nonlinear rule.

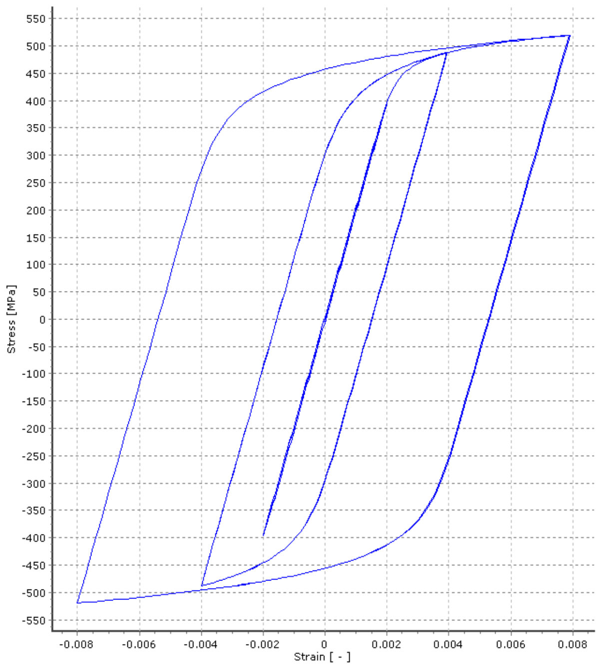


Bilinear steel model with isotropic strain hardening

Users have to assign the mean value of steel strength and the mean value minus the standard deviation or the characteristic value in order to describe the mechanical characteristics of the existing or the new material respectively.

Ramberg-Osgood steel model - stl_ro

This is the Ramberg-Osgood stress-strain model [Ramberg and Osgood, 1943], as described in the work of Kaldjian [1967]. It has been initially programmed by Otani [1981].

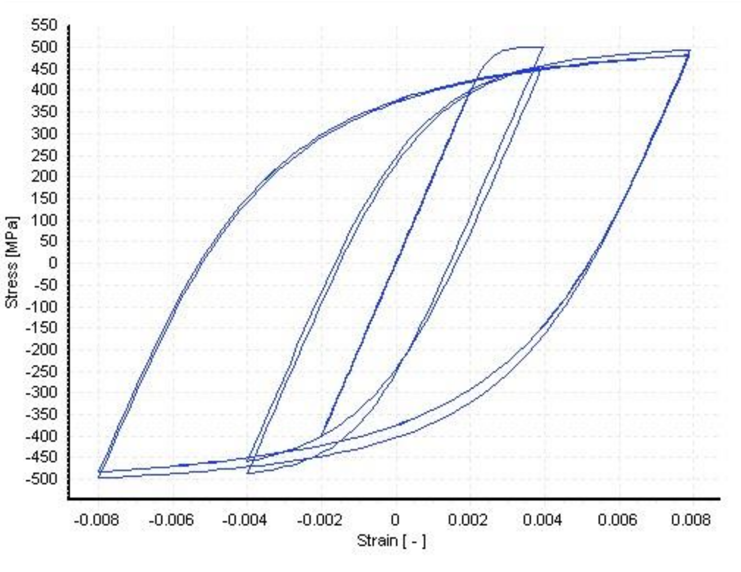


Ramberg-Osgood steel model

Users have to assign the mean value of steel strength and the mean value minus the standard deviation or the characteristic value in order to describe the mechanical characteristics of the existing or the new material respectively.

Menegotto-Pinto steel model - stl_mp

This is a uniaxial steel model initially programmed by Yassin [1994] based on a simple, yet efficient, stress-strain relationship proposed by Menegotto and Pinto [1973], coupled with the isotropic hardening rules proposed by Filippou et al. [1983]. The current implementation follows that carried out by Monti et al. [1996]. An additional memory rule proposed by Fragiadakis et al. [2008] is also introduced, for higher numerical stability/accuracy under transient seismic loading. Its employment should be confined to the modelling of reinforced concrete structures, particularly those subjected to complex loading histories, where significant load reversals might occur.

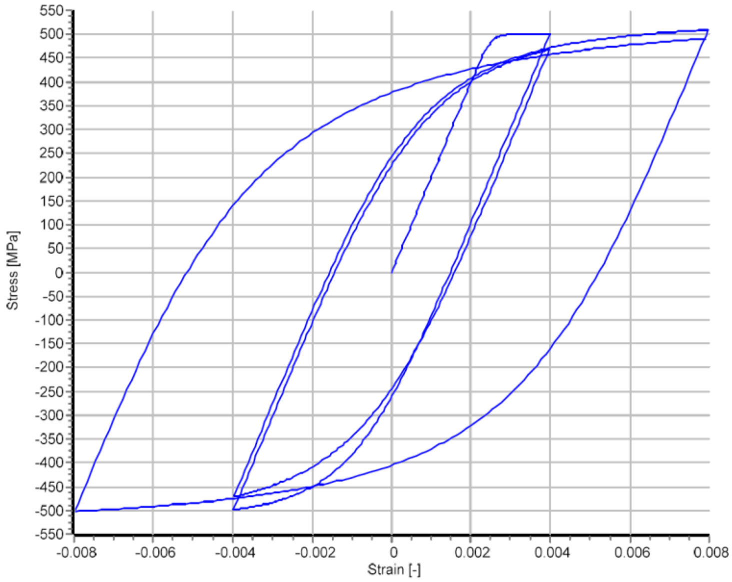


Menegotto-Pinto steel model

Users have to assign the mean value of steel strength and the mean value minus the standard deviation or the characteristic value in order to describe the mechanical characteristics of the existing or the new material respectively.

Giuffre-Menegotto-Pinto Model with Isotropic Hardening – stl_gmp

This is a uniaxial Giuffre-Menegotto-Pinto Material with optional isotropic hardening described by a nonlinear rule. The transition from elastic to plastic behaviour is described by the Giuffre-Menegotto-Pinto Model. The material model was described in full detail by Filippou et al. [1983]. The material should be mainly utilized for the modelling of the behaviour of reinforcing steel in reinforced concrete structures, especially in the case when load reversals occur.

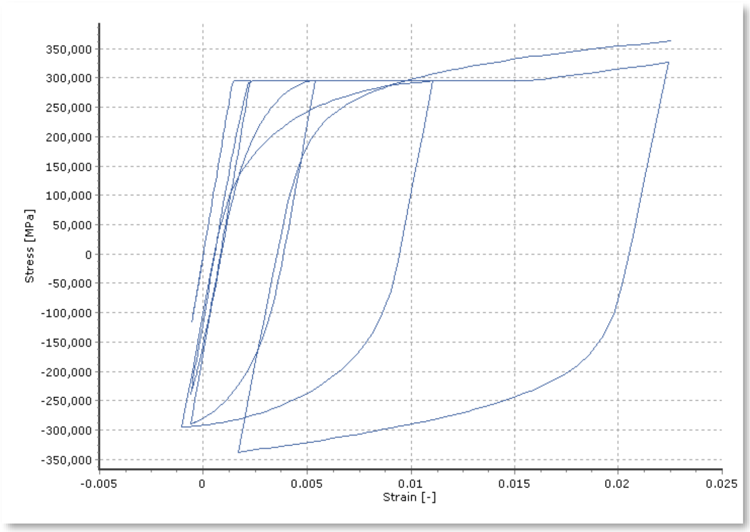


Giuffre-Menegotto-Pinto Model with Isotropic Hardening

Users have to assign the mean value of steel strength and the mean value minus the standard deviation or the characteristic value in order to describe the mechanical characteristics of the existing or the new material respectively.

Dodd-Restrepo steel model - stl_dr

This is a uniaxial steel model initially programmed by Dodd and Restrepo [1995]. It considers the reduction of the unloading modulus with the plastic strain, whilst the reduction of the ultimate tensile strain is taken solely as a function of the maximum compressive strain, when the number of cycles is small enough to ignore the effects of low-cycle fatigue.

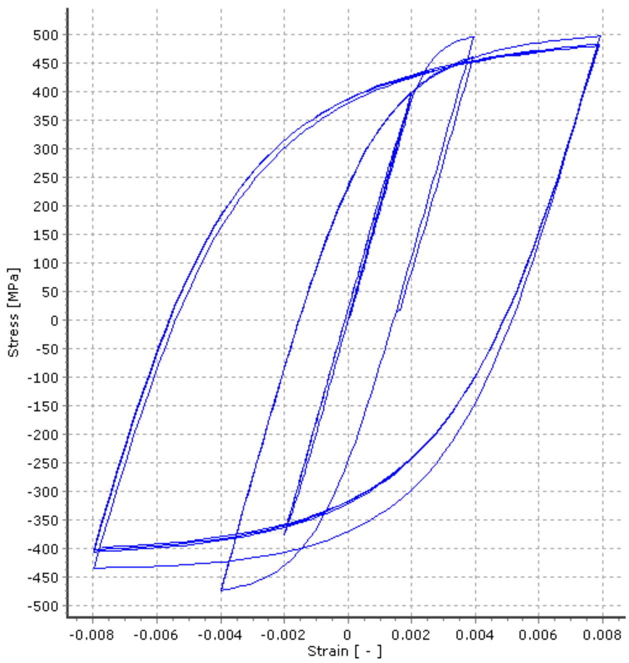


Dodd-Restrepo steel model

Users have to assign the mean value of steel strength and the mean value minus the standard deviation or the characteristic value in order to describe the mechanical characteristics of the existing or the new material respectively.

Monti-Nuti steel model - stl_mn

This is a uniaxial steel model initially programmed by Monti et al. [1996], which is able to describe the post-elastic buckling behaviour of reinforcing bars under compression. It uses the Menegotto and Pinto [1973] stress-strain relationship together with the isotropic hardening rules proposed by Filippou et al. [1983] and the buckling rules proposed by Monti and Nuti [1992]. An additional memory rule proposed by Fragiadakis et al. [2008] is also introduced, for higher numerical stability/accuracy under transient seismic loading. Its employment should be confined to the modelling of reinforced concrete members where buckling of reinforcement might occur (e.g. columns under severe cyclic loading). Further, as discussed by Prota et al. [2009], with the correct calibration, this model, initially developed with ribbed reinforcement bars in mind, can also be employed for the modelling of smooth rebars, often found in existing structures.

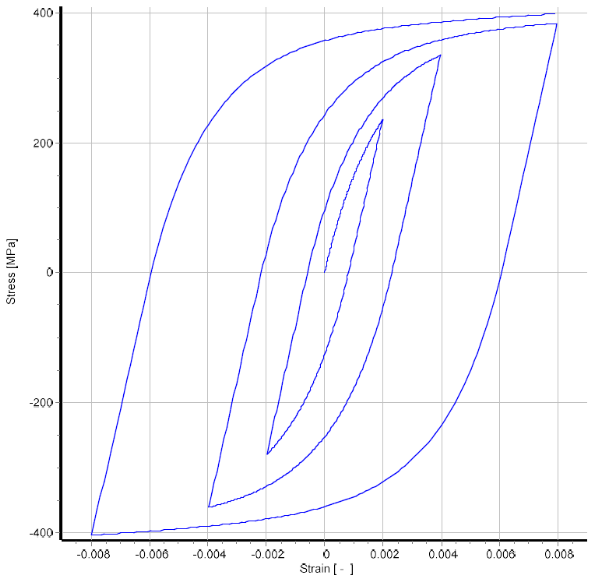


Monti-Nuti steel model

Users have to assign the mean value of steel strength and the mean value minus the standard deviation or the characteristic value in order to describe the mechanical characteristics of the existing or the new material respectively.

Buckling Restrained steel brace model - stl_brb

Stl_BRB is a uniaxial steel material model describing the behaviour of steel in Bucking Restrained Braces. The model has been presented by Zona et al. [2012]

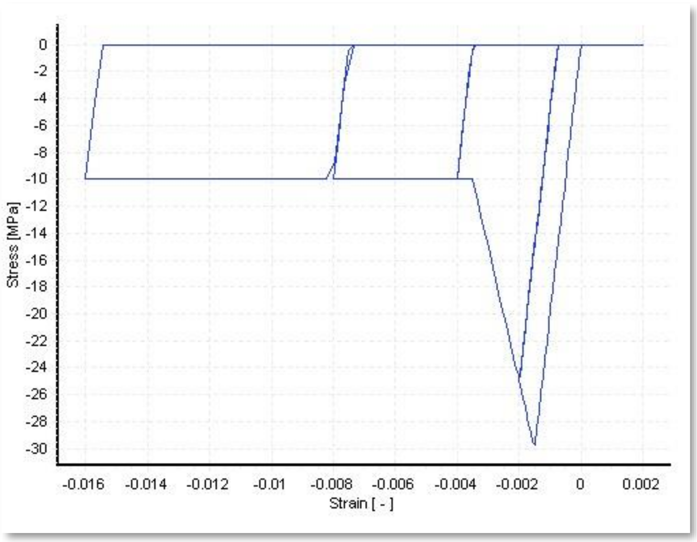


Buckling restrained steel braces material

CONCRETE MATERIALS

Trilinear concrete model - con_tl

This is a simplified uniaxial trilinear concrete model that assumes no resistance to tension and features a residual strength plateau.

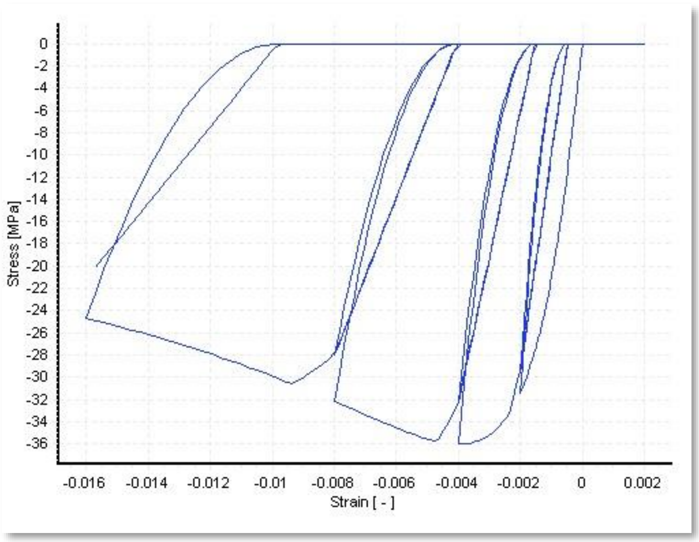


Trilinear concrete model

Users have to assign the mean value of concrete strength and the mean value minus the standard deviation or the characteristic value in order to describe the mechanical characteristics of the existing or the new material respectively.

Mander et al. nonlinear concrete model - con_ma

This is a uniaxial nonlinear constant confinement model, initially programmed by Madas [1993], that follows the constitutive relationship proposed by Mander et al. [1988] and the cyclic rules proposed by Martinez-Rueda and Elnashai [1997]. The confinement effects provided by the lateral transverse reinforcement are incorporated through the rules proposed by Mander et al. [1988] whereby constant confining pressure is assumed throughout the entire stress-strain range.



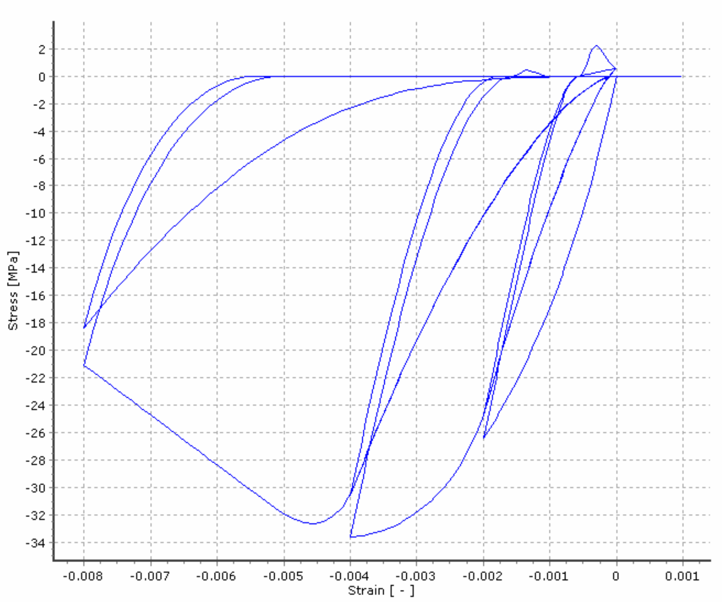
Mander et al. nonlinear concrete model

Users have to assign the mean value of concrete strength and the mean value minus the standard deviation or the characteristic value in order to describe the mechanical characteristics of the existing or the new material respectively.

NOTE: The confinement factor employed by this material type is a constant confinement factor. It is defined as the ratio between the confined and unconfined compressive stress of the concrete, and used to scale up the stress-strain relationship throughout the entire strain range. Although it may be computed through the use of any confinement model available in the literature [e.g. Ahmad and Shah, 1982; Sheikh and Uzumeri, 1982; Eurocode 8, 2004; Penelis and Kappos, 1997], the Mander et al. [1989] is used by the program. Its value usually fluctuates between the values of 1.0 and 2.0 for reinforced concrete.

Chang-Mander nonlinear concrete model - con_cm

It is the implementation of Chang & Mander's [Chang & Mander, 1994] concrete model, which puts particular emphasis on the transition of the stress-strain relation upon crack opening and closure, contrary to other similar models that assume sudden crack closure with rapid change in section modulus. The concrete in tension is modelled with a cyclic behaviour similar to that in compression, and the model envelopes for compression and tension have control on the slope of the stress-strain behaviour at the origin, and the shape of both the ascending and descending (i.e., pre-peak and post-peak) branches of the stress-strain behaviour.



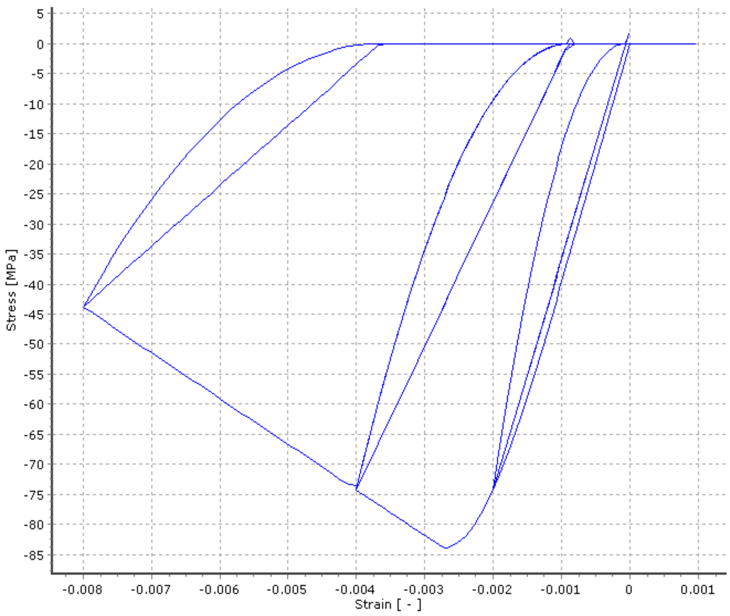
Chang-Mander nonlinear concrete model

Users have to assign the mean value of concrete strength and the mean value minus the standard deviation or the characteristic value in order to describe the mechanical characteristics of the existing or the new material respectively.

Kappos and Konstantinidis nonlinear concrete model - con_hs

NOTE: The need for a special-purpose high-strength concrete model raises from the fact that this type of concrete features a stress-strain response that differs quite significantly from its normal strength counterpart, particularly in what concerns the post-peak behaviour, which tends to be considerably less ductile.

This is a uniaxial nonlinear constant confinement for high-strength concrete model, developed and initially programmed by Kappos and Konstantinidis [1999]. It follows the constitutive relationship proposed by Nagashima et al. [1992] and has been statistically calibrated to fit a very wide range of experimental data. The confinement effects provided by the lateral transverse reinforcement are incorporated through the modified Sheikh and Uzumeri [1982] factor (i.e. confinement effectiveness coefficient), assuming that a constant confining pressure is applied throughout the entire stress-strain range.

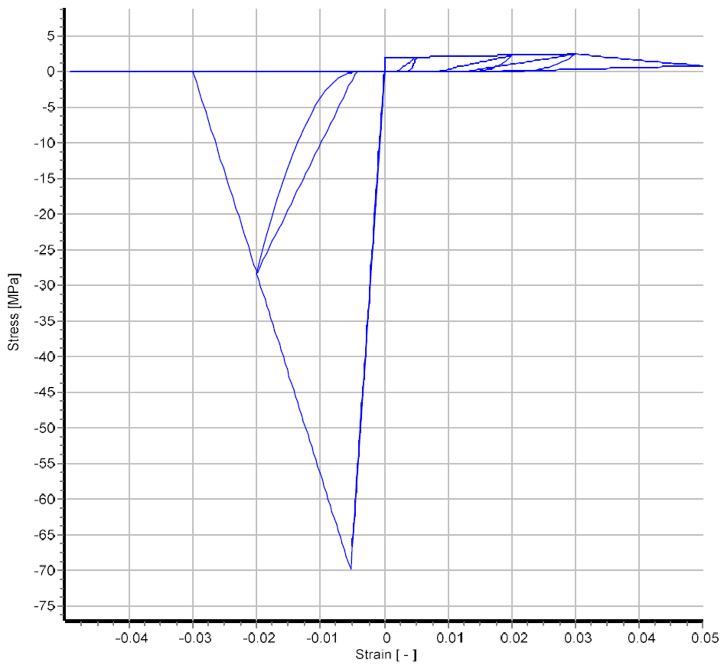


Kappos and Konstantinidis nonlinear concrete model

Users have to assign the mean value of concrete strength and the mean value minus the standard deviation or the characteristic value in order to describe the mechanical characteristics of the existing or the new material respectively.

Engineered cementitious composites material – con_ecc

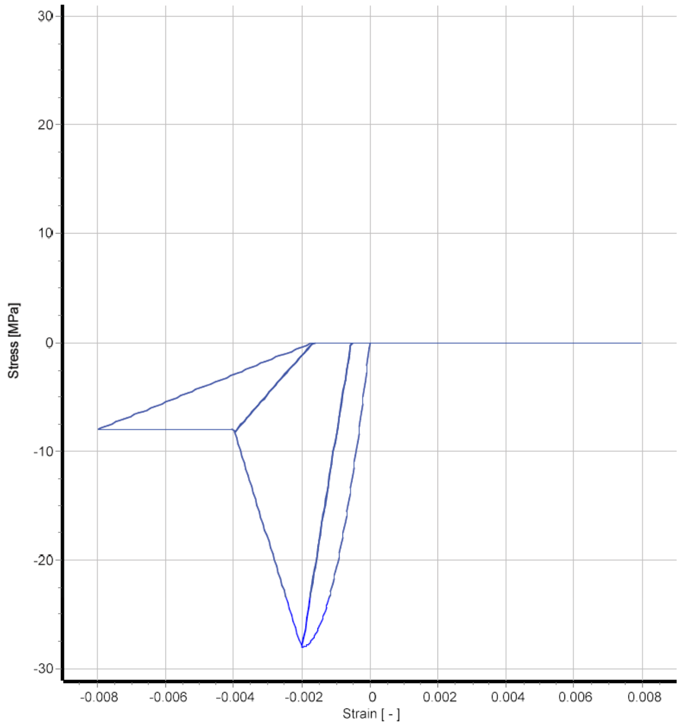
Con_ecc is a uniaxial generic material modeling the behavior of ductile fiber-reinforced cement-based composites as described by Han et al. [2003]. The model needs 13 variables for its definition.



Ductile fibre – reinforced cement based composites model

Kent-Scott-Park concrete model – con_ksp

The con_ksp is a simplified uniaxial concrete model with a stress-strain relationship described by Kent and Park [1971] and a cyclic behaviour proposed by Karsan and Jirsa [1969]. The model is characterized by zero tensile strength. Five variables are needed for the definition of the model.



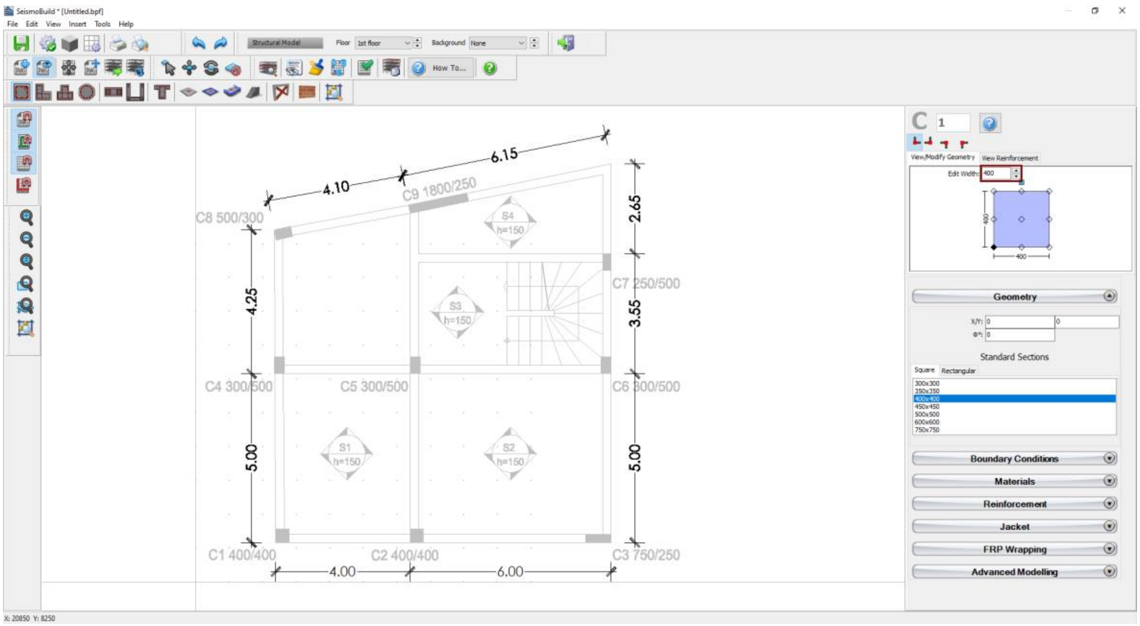
Kent – Scott – Park concrete model

Appendix D - Inserting Structural Members

In this appendix the section types available in SeismoBuild are described in details.

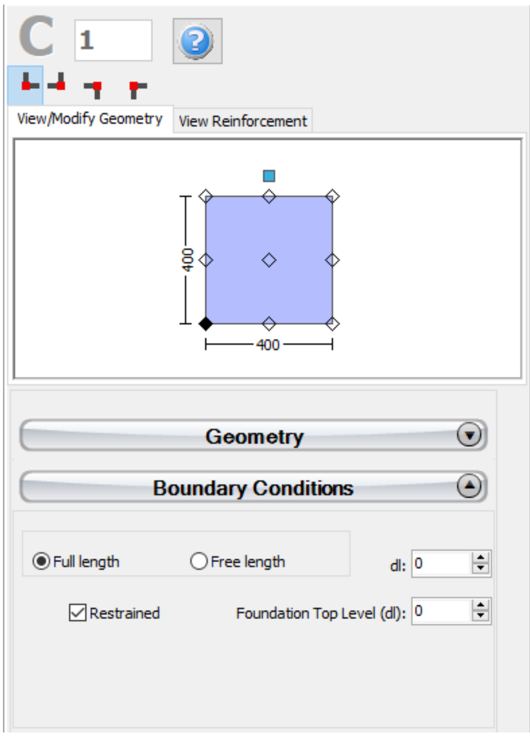
Rectangular Column (Plain and Jacketed)

Rectangular columns may be inserted from the main menu (*Insert > Insert Rectangular Column*) or through the corresponding toolbar button . On the Properties Window that appears users can adapt the section's dimensions either in the View/Modify Geometry window or by selecting one section from the predefined standard sections (square or rectangular).



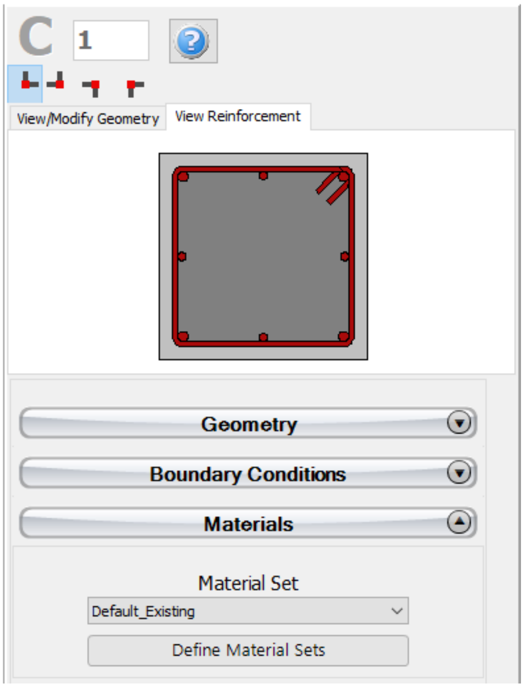
Edit sections dimensions

It is possible to define a column height different from the general storey height, through the selection of the Free length radio button and the assignment of a different length. If, on the other hand, the Full length radio button is selected then the member has the same height with the storey height. In addition, the foundation level of the column may be adapted, thus providing the possibility to the user to define different foundation levels.



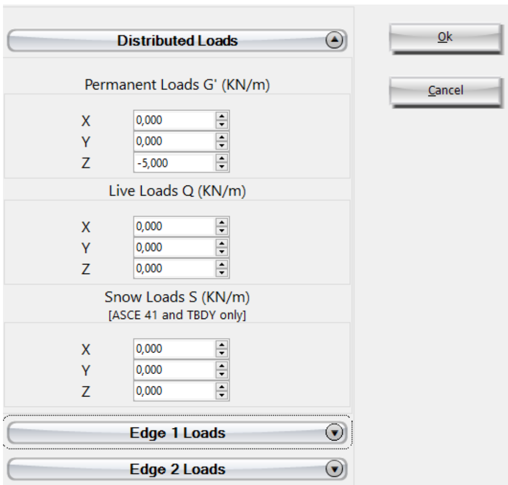
Boundary Conditions

The material set properties can be defined from the main menu (*Tools > Define Material Sets*), through the corresponding toolbar  button, or through the Define Material Sets button within the member's Properties Window. The required values for the definition of the materials properties depend on the type of the members, i.e. existing or new members. By default, there are two material schemes, one for the existing elements and one for the new ones.



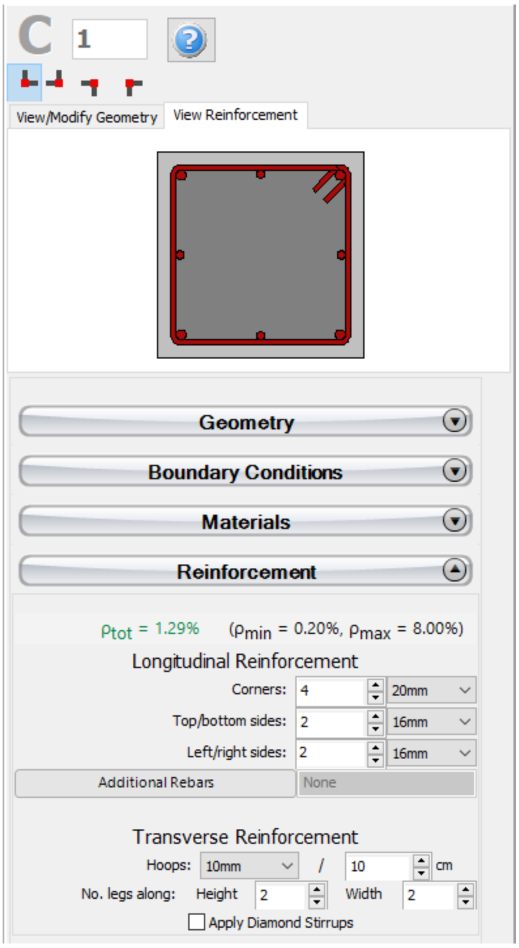
Materials

Additional loads can be defined by clicking on the Distributed and Edge Loads button. Users can define uniform distributed forces along the length of the member in all three translational directions X, Y or Z, and forces or moments in any translational or rotational direction (X, Y, Z, RX, RY or RZ) at either of the two edges of the member (bottom or top). Additional permanent loads G' (not associated with the self-weight of the structure), live Q and snow S loads may be applied, with the latter being applicable only to ASCE 41 and TBDY. By default, all loads are equal to zero.



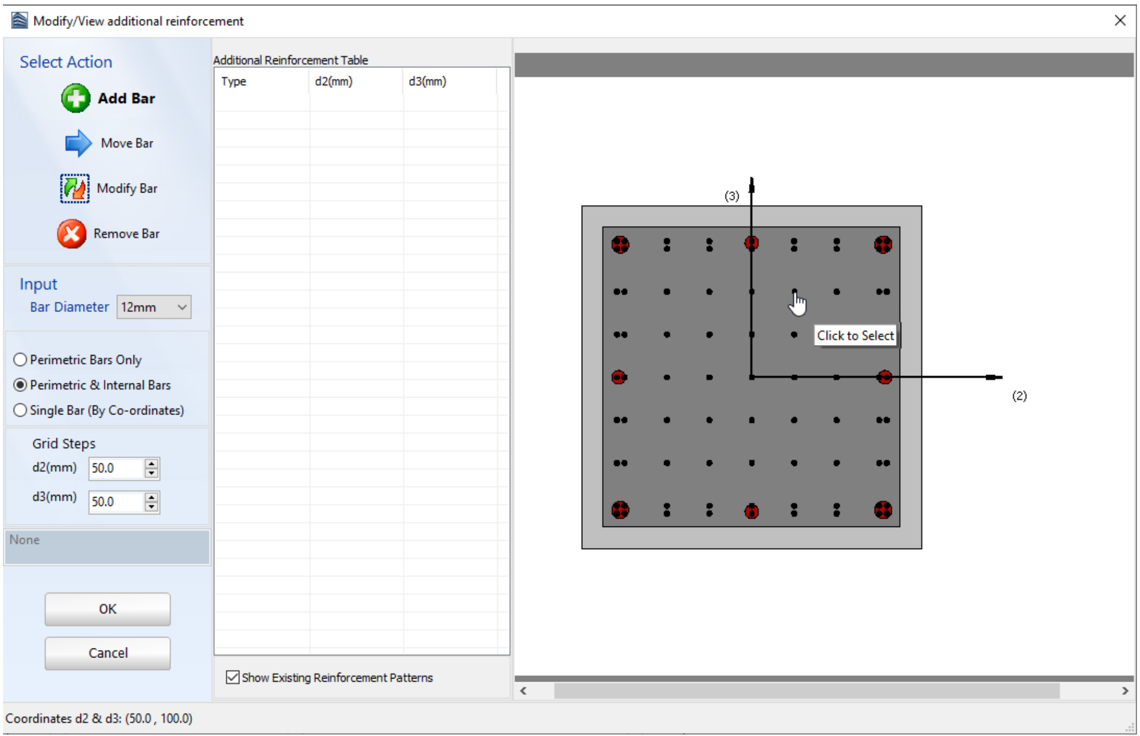
Distributed and Edge Loads window

Further, the longitudinal and transverse reinforcement may be defined by editing the relevant reinforcement pattern controls.



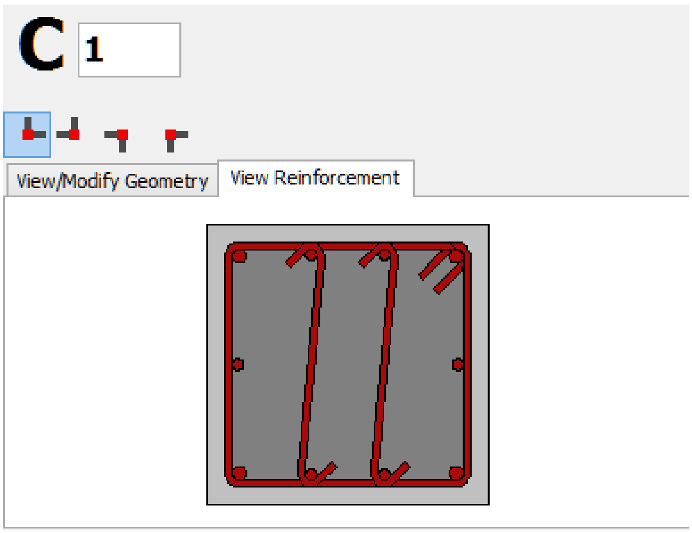
Reinforcement Pattern

Adding single longitudinal reinforcement bars may also be carried out through the corresponding Additional Rebars module, where additional reinforcement may be introduced graphically as shown in the following figure:



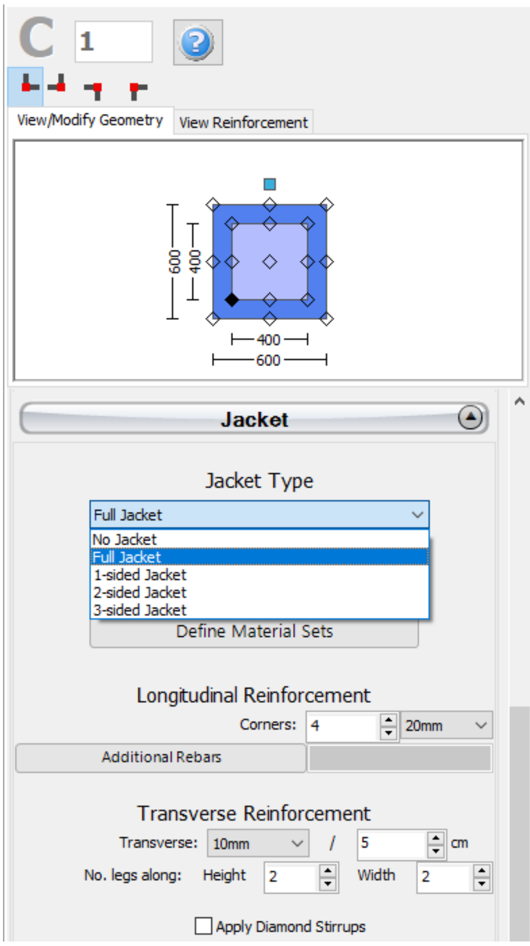
Modify/View additional reinforcement window

On the Properties Window users may choose between the View Reinforcement, where the reinforcement of the section is displayed (longitudinal and transverse), and the View/Modify Geometry, where the section's dimensions may be viewed and modified.



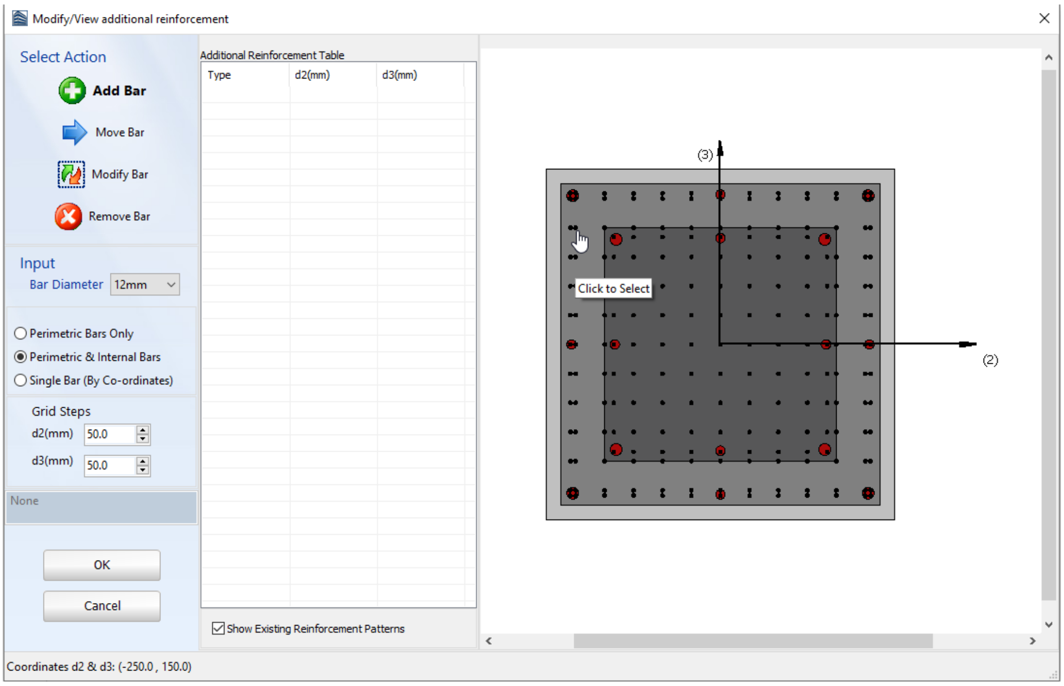
Reinforcement View

Jackets may be applied to the section in the Jacket area by selecting the jacket type, i.e. whether it is full jacketed, 1-sided, 2-sided or 3-sided jacket, and assigning the material set and the longitudinal and transverse reinforcement of the jacket.



Jacket

Adding single longitudinal reinforcement bars to the jacket can also be carried out through the corresponding Additional Rebars module, where additional reinforcement may be introduced graphically to both the existing and the new part of the section, as shown in the following figure:



Modify/View additional reinforcement window

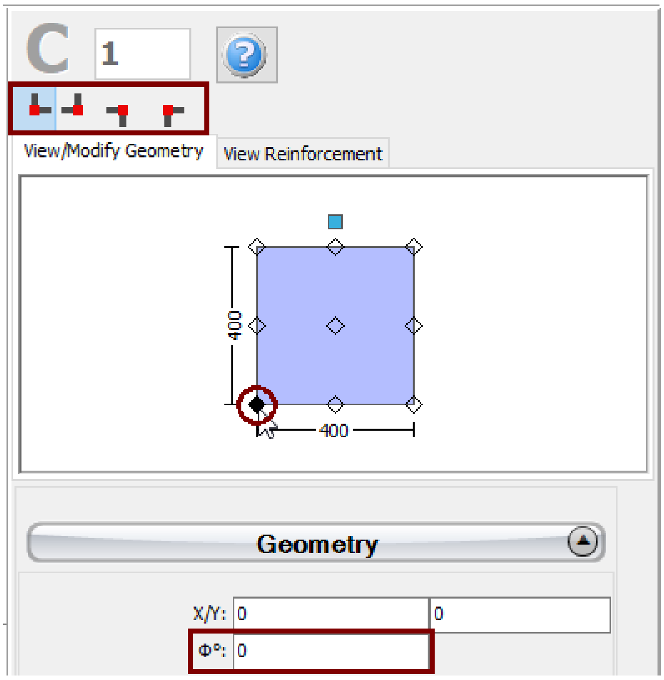
Isolators can also be added at different locations of the column. They are assigned through the Isolator module, where the users may select the geometry (location -bottom, top or intermediate point- and the height of the isolator), its type (elastomeric, lead rubber or curve surface slider) and the isolator parameters: the vertical and horizontal stiffnesses, and the shear yield strength and the strain hardening ratio (for elastomeric and lead rubber isolators, which are modelled as isolator1 element type) or the friction coefficient and the radius of the pendulum (for curve surface sliders, which are modelled as isolator2 element type).

Further, FRP wraps may be assigned to column elements through the FRP Wrapping module, where the users may select the FRP wrap from a list of the most common products found in the market, or introduce user defined values.

In the Advanced Modelling area, the code-based settings of the structural member can be defined through the Advanced Member Properties dialog box that opens from the corresponding button. The member's modelling parameters may be also defined from the Modelling Parameters dialog box, accessed by the corresponding button.

NOTE: When the section is jacketed, in the *Advanced Member Properties* module users should take decisions on the parameters, so as to account for the entire section, i.e. for both the existing and the new parts.

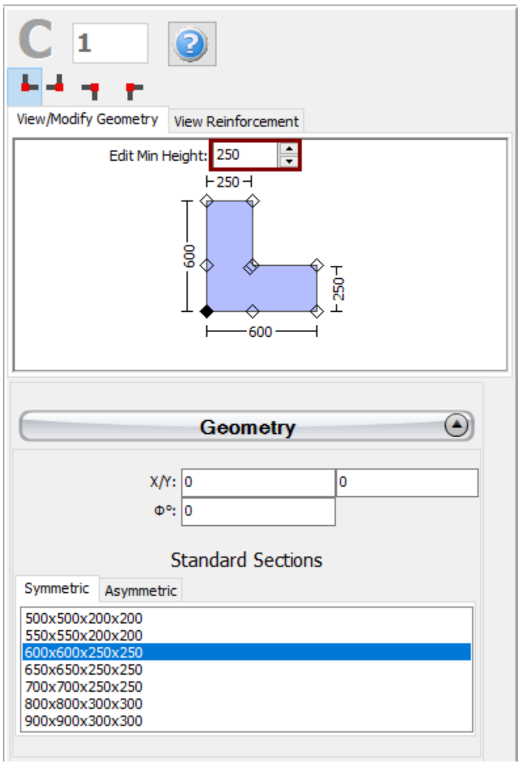
After defining all the section's properties, the new member may be added with a simple click on the Building Modeller Main Window. The location of the section that corresponds to the insertion point (i.e. the mouse click), and rotation of the section on plan view may be selected from the Member Properties window.



Selecting the insertion point and rotate the section's plan view

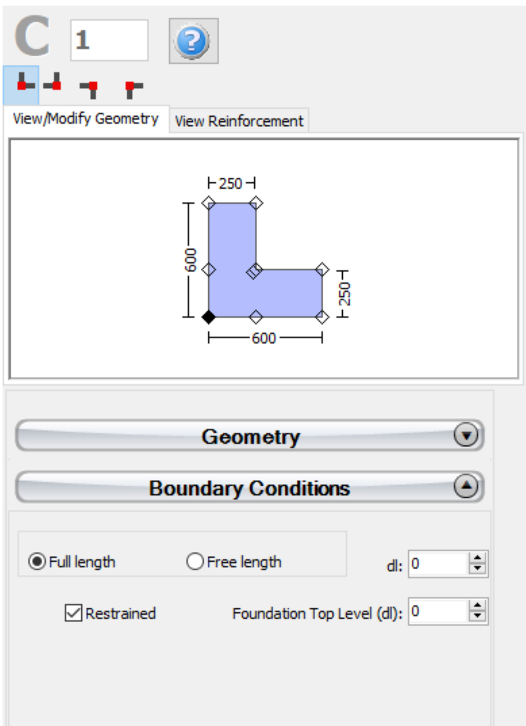
L-Shaped Column

L-Shaped columns can be inserted from the main menu (*Insert > Insert L-Shaped Column*) or through the corresponding toolbar button . On the Properties Window that appears users can adapt the section's dimensions either in the View/Modify Geometry window or by selecting one section from the predefined standard sections (symmetric or asymmetric).



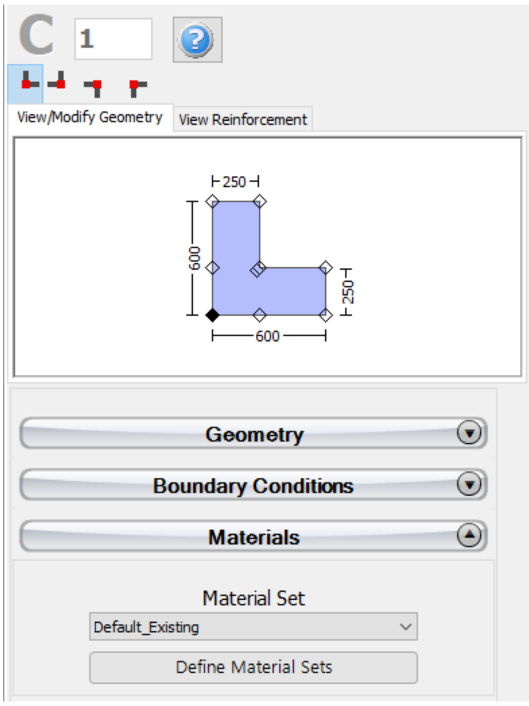
Edit sections dimensions

It is possible to define a column height different from the general storey height, through the selection of the Free length radio button and the assignment of a different length. If, on the other hand, the Full length radio button is selected then the member has the same height with the storey height. In addition, the foundation level of the column may be adapted, thus providing the possibility to the user to define different foundation levels.



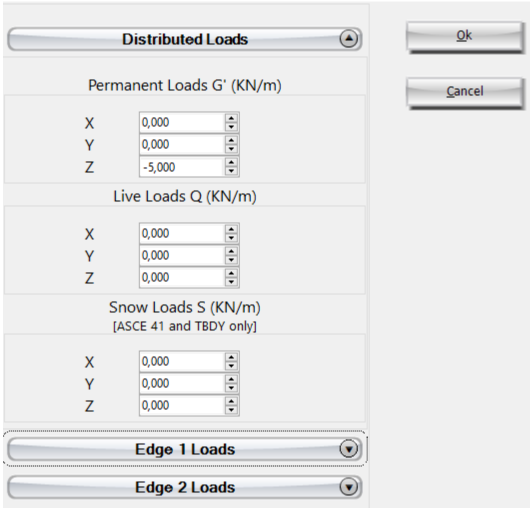
Boundary Conditions

The material set properties can be defined from the main menu (*Tools > Define Material Sets*), through the corresponding toolbar button, or through the Define Material Sets button within the member's Properties Window. The required values for the definition of the materials properties depend on the type of the members, i.e. existing or new members. By default, there are two material schemes, one for the existing elements and one for the new ones.



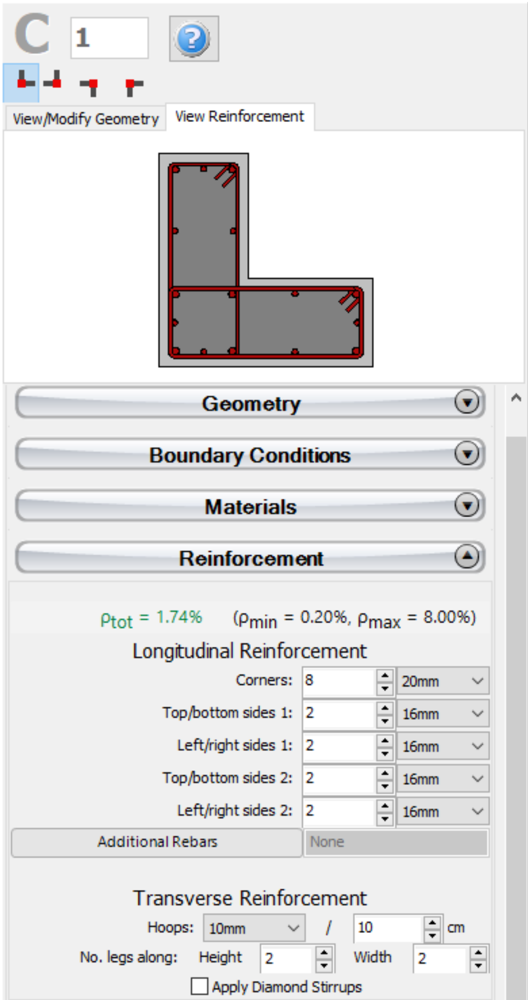
Materials

Additional loads can be defined by clicking on the Distributed and Edge Loads button. Users can define uniform distributed forces along the length of the member in all three translational directions X, Y or Z, and forces or moments in any translational or rotational direction (X, Y, Z, RX, RY or RZ) at either of the two edges of the member (bottom or top). Additional permanent loads G' (not associated with the self-weight of the structure), live Q and snow S loads may be applied, with the latter being applicable only to ASCE 41 and TBDY. By default, all loads are equal to zero.



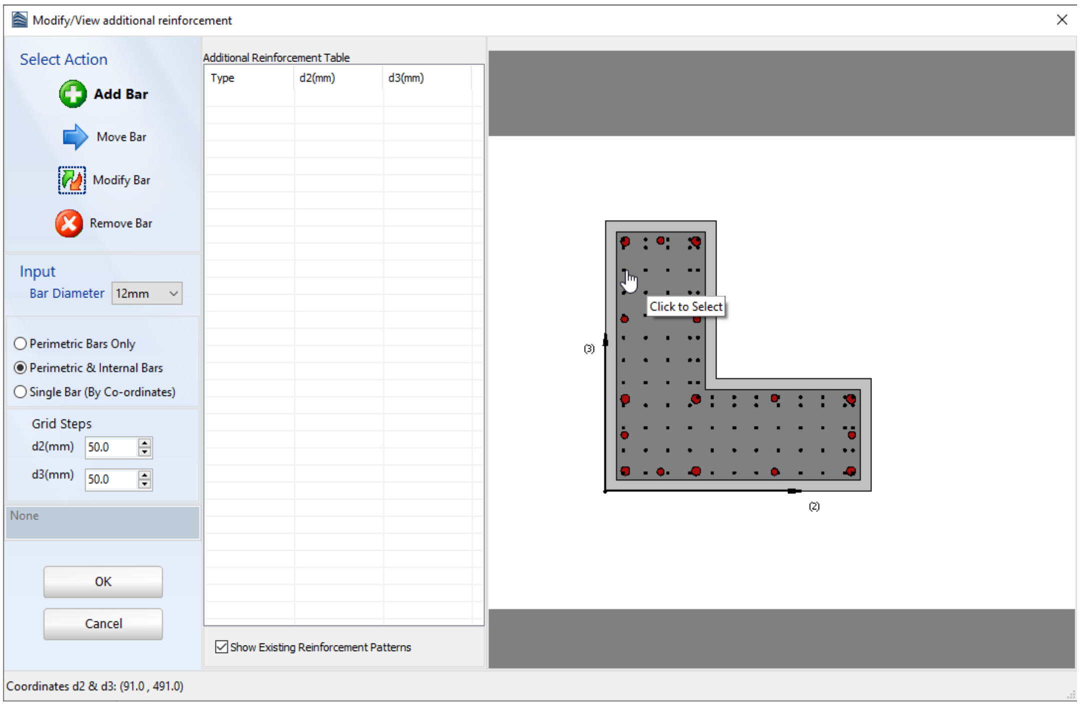
Distributed and Edge Loads window

Further, the longitudinal and transverse reinforcement may be defined by editing the relevant reinforcement pattern controls.



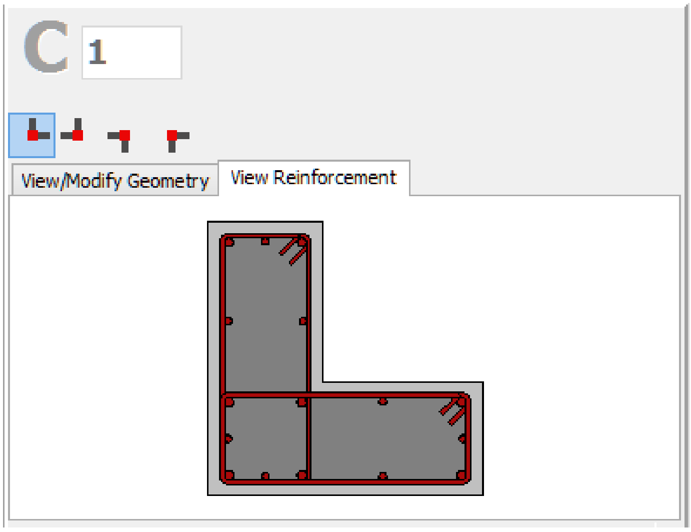
Reinforcement Pattern

Adding single longitudinal reinforcement bars may also be carried out through the corresponding Additional Rebars module, where additional reinforcement may be introduced graphically as shown in the following figure:



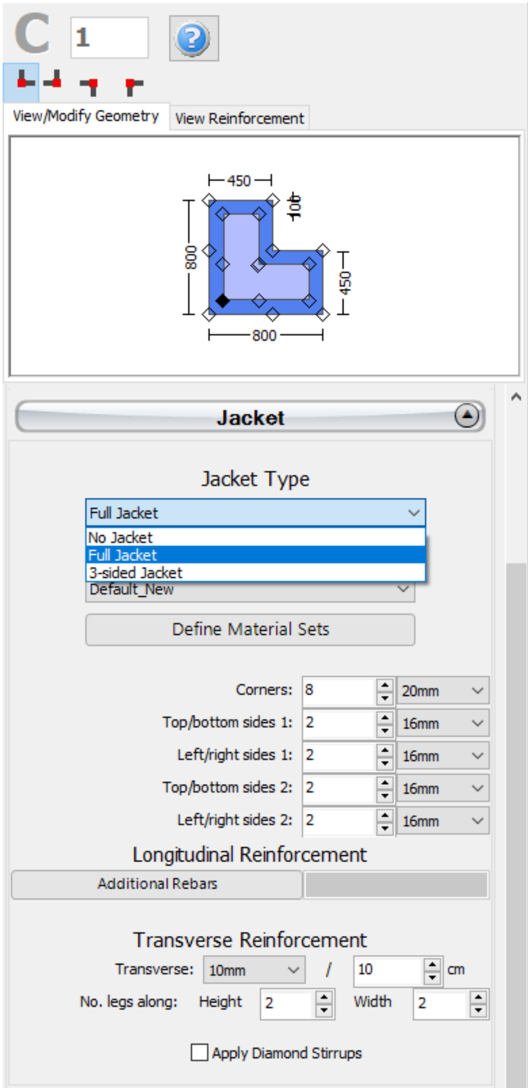
Modify/View additional reinforcement window

On the Properties Window users may choose between the View Reinforcement, where the reinforcement of the section is displayed (longitudinal and transverse), and the View/Modify Geometry, where the section's dimensions may be viewed and modified.



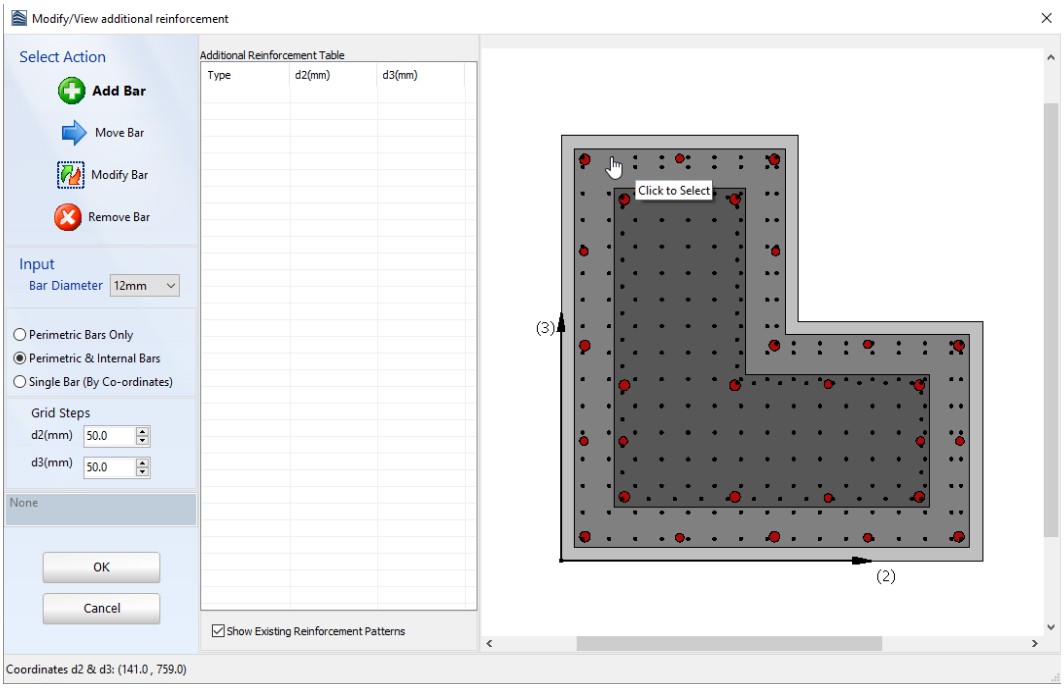
Reinforcement View

Jackets may be applied to the section in the Jacket area by selecting the jacket type, i.e. whether it is full jacketed or 3-sided jacket, and assigning the material set and the longitudinal and transverse reinforcement of the jacket.



Jacket

Adding single longitudinal reinforcement bars to the jacket can also be carried out through the corresponding Additional Rebars module, where additional reinforcement may be introduced graphically to both the existing and the new part of the section, as shown in the following figure:



Modify/View additional reinforcement window

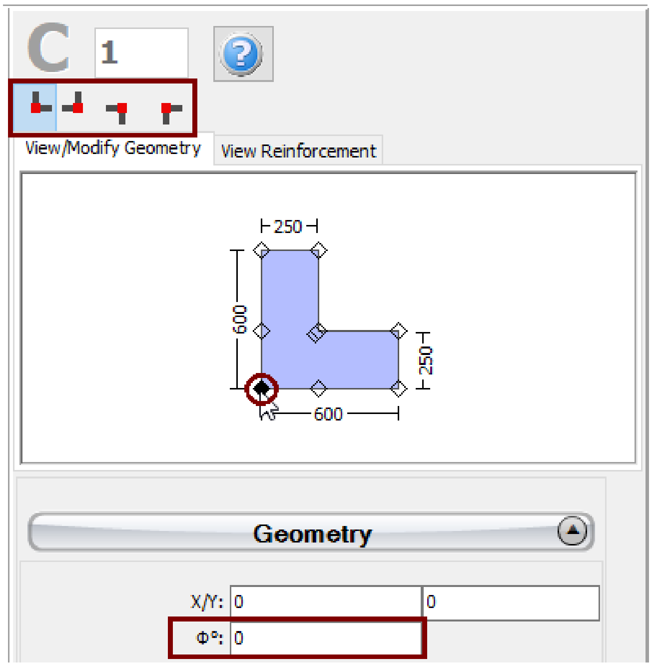
Isolators can also be added at different locations of the column. They are assigned through the Isolator module, where the users may select the geometry (location -bottom, top or intermediate point- and the height of the isolator), its type (elastomeric, lead rubber or curve surface slider) and the isolator parameters: the vertical and horizontal stiffnesses, and the shear yield strength and the strain hardening ratio (for elastomeric and lead rubber isolators, which are modelled as isolator1 element type) or the friction coefficient and the radius of the pendulum (for curve surface sliders, which are modelled as isolator2 element type).

Further, FRP wraps may be assigned to column elements through the FRP Wrapping module, where the users may select the FRP wrap from a list of the most common products found in the market, or introduce user defined values.

In the Advanced Modelling area, the code-based settings of the structural member can be defined through the Advanced Member Properties dialog box that opens from the corresponding button. The member's modelling parameters may be also defined from the Modelling Parameters dialog box, accessed by the corresponding button.

NOTE: When the section is jacketed, in the *Advanced Member Properties* module users should take decisions on the parameters, so as to account for the entire section, i.e. for both the existing and the new parts.

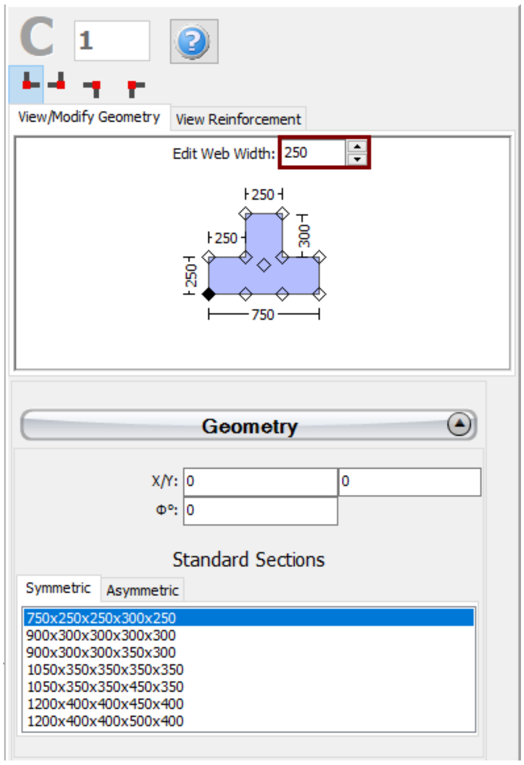
After defining all the section's properties, the new member may be added with a simple click on the Building Modeller Main Window. The location of the section that corresponds to the insertion point (i.e. the mouse click), and rotation of the section on plan view may be selected from the Member Properties window.



Selecting the insertion point and rotate the section's plan view

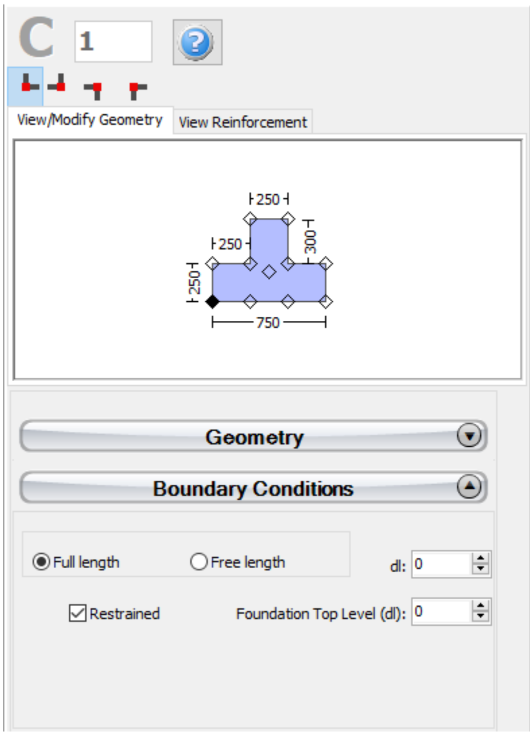
T-Shaped Column

T-Shaped columns may be inserted from the main menu (*Insert > Insert T-Shaped Column*) or through the corresponding toolbar button . On the Properties Window that appears users can adapt the section's dimensions either in the View/Modify Geometry window or by selecting one section from the predefined standard sections (symmetric or asymmetric).



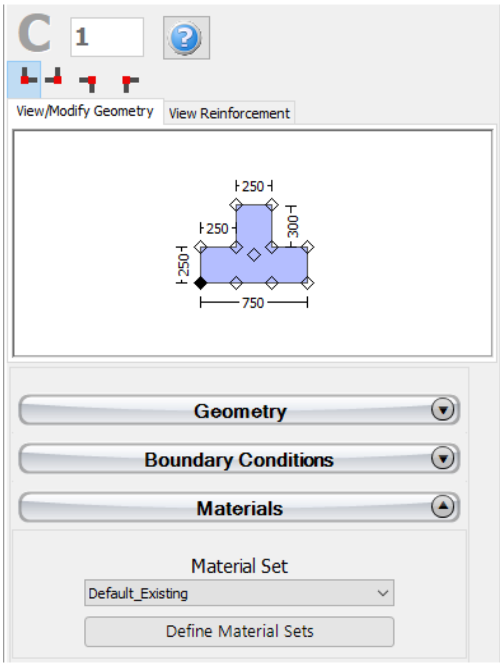
Edit sections dimensions

It is possible to define a column height different from the general storey height, through the selection of the Free length radio button and the assignment of a different length. If, on the other hand, the Full length radio button is selected then the member has the same height with the storey height. In addition, the foundation level of the column may be adapted, thus providing the possibility to the user to define different foundation levels.



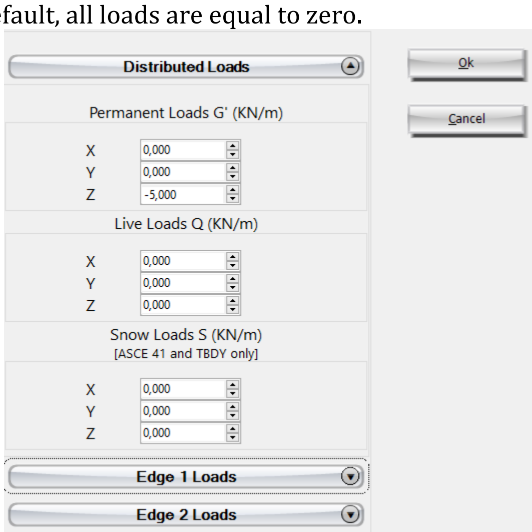
Boundary Conditions

The material set properties can be defined from the main menu (*Tools > Define Material Sets*), through the corresponding toolbar  button, or through the Define Material Sets button within the member's Properties Window. The required values for the definition of the materials properties depend on the type of the members, i.e. existing or new members. By default, there are two material schemes, one for the existing elements and one for the new ones.



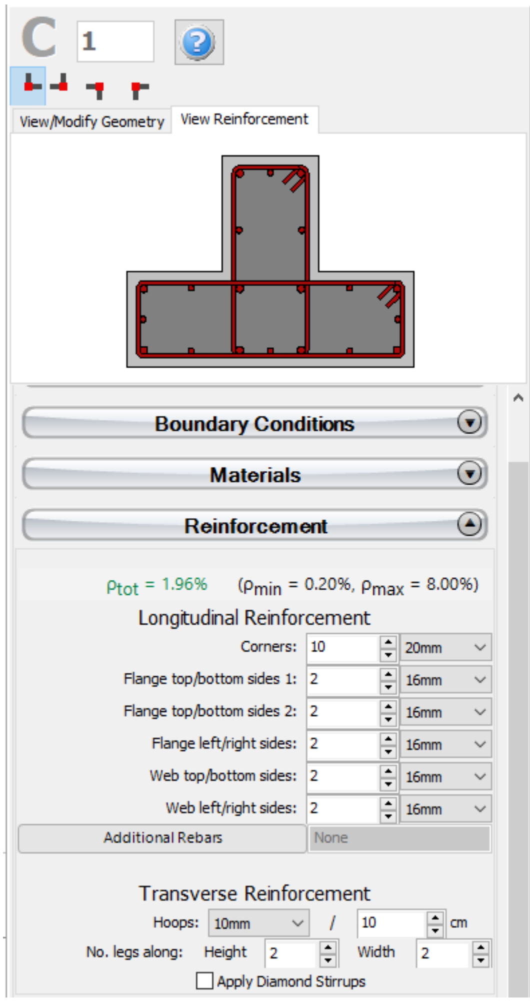
Materials

Additional loads can be defined by clicking on the Distributed and Edge Loads button. Users can define uniform distributed forces along the length of the member in all three translational directions X, Y or Z, and forces or moments in any translational or rotational direction (X, Y, Z, RX, RY or RZ) at either of the two edges of the member (bottom or top). Additional permanent loads G' (not associated with the self-weight of the structure), live Q and snow S loads may be applied, with the latter being applicable only to ASCE 41 and TBDY. By default, all loads are equal to zero.



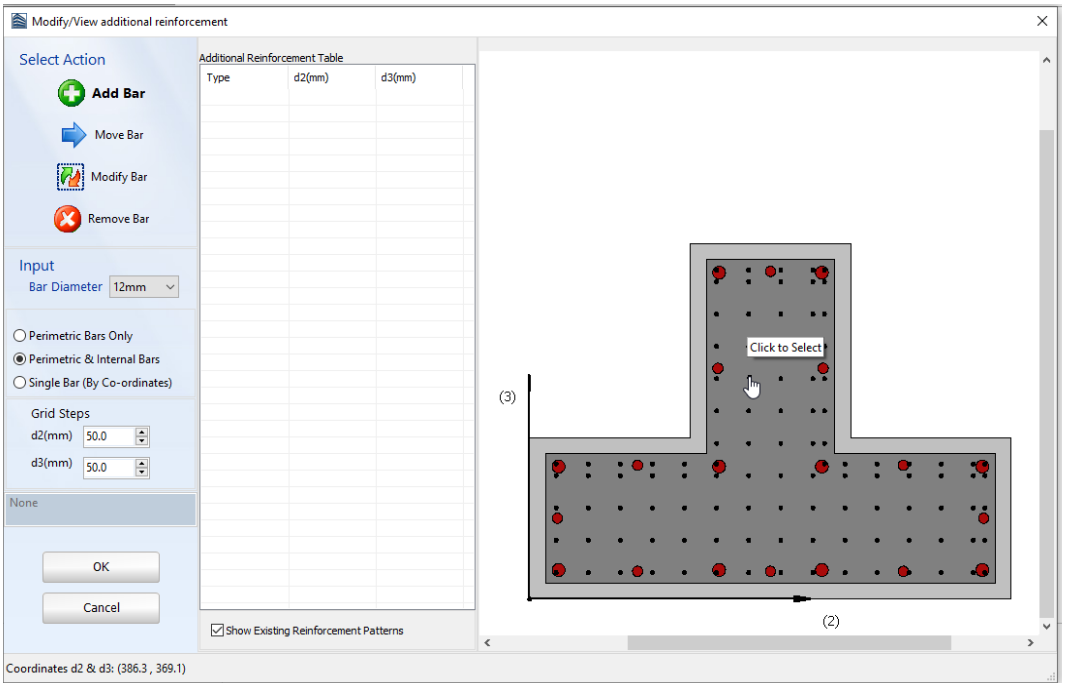
Distributed and Edge Loads window

Further, the longitudinal and transverse reinforcement may be defined by editing the relevant reinforcement pattern controls.



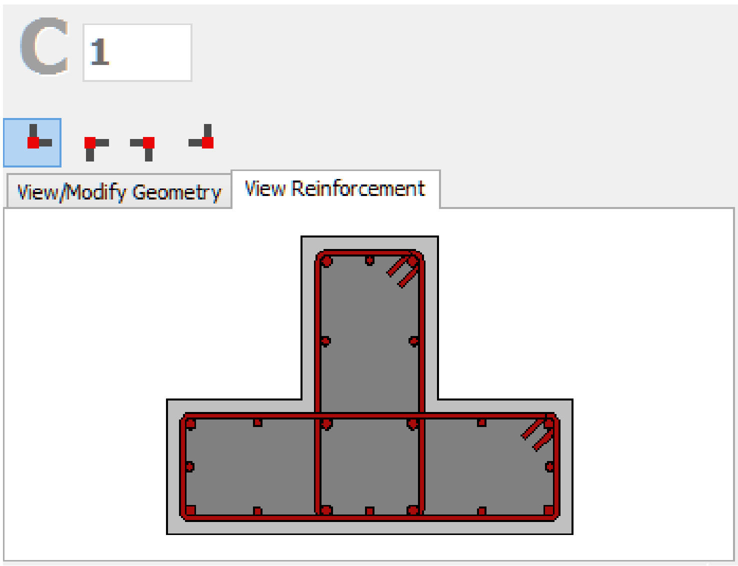
Reinforcement Pattern

Adding single longitudinal reinforcement bars may also be carried out through the corresponding Additional Rebars module, where additional reinforcement may be introduced graphically as shown in the following figure:



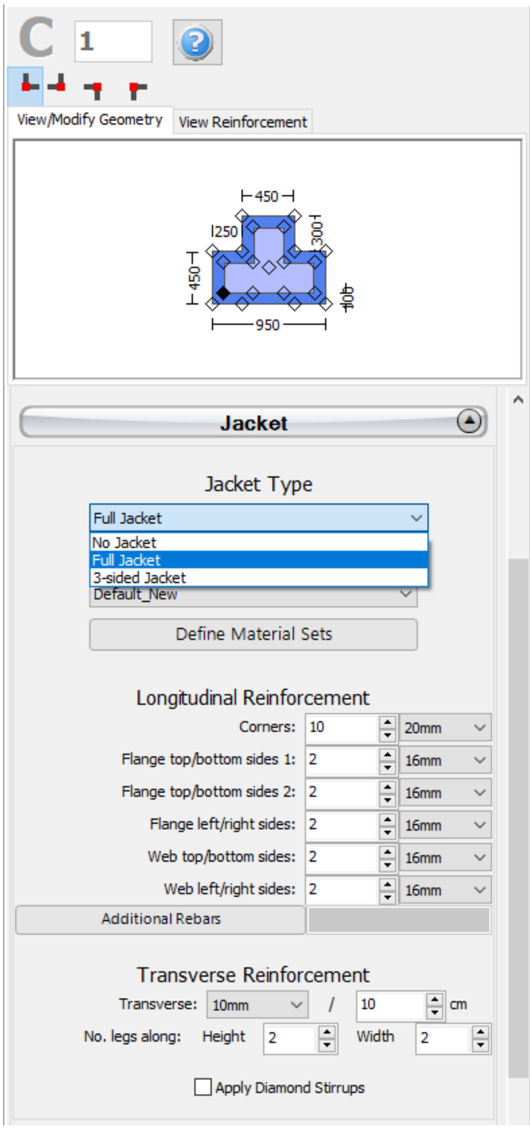
Modify/View additional reinforcement window

On the Properties Window users may choose between the View Reinforcement, where the reinforcement of the section is displayed (longitudinal and transverse), and the View/Modify Geometry, where the section's dimensions may be viewed and modified.



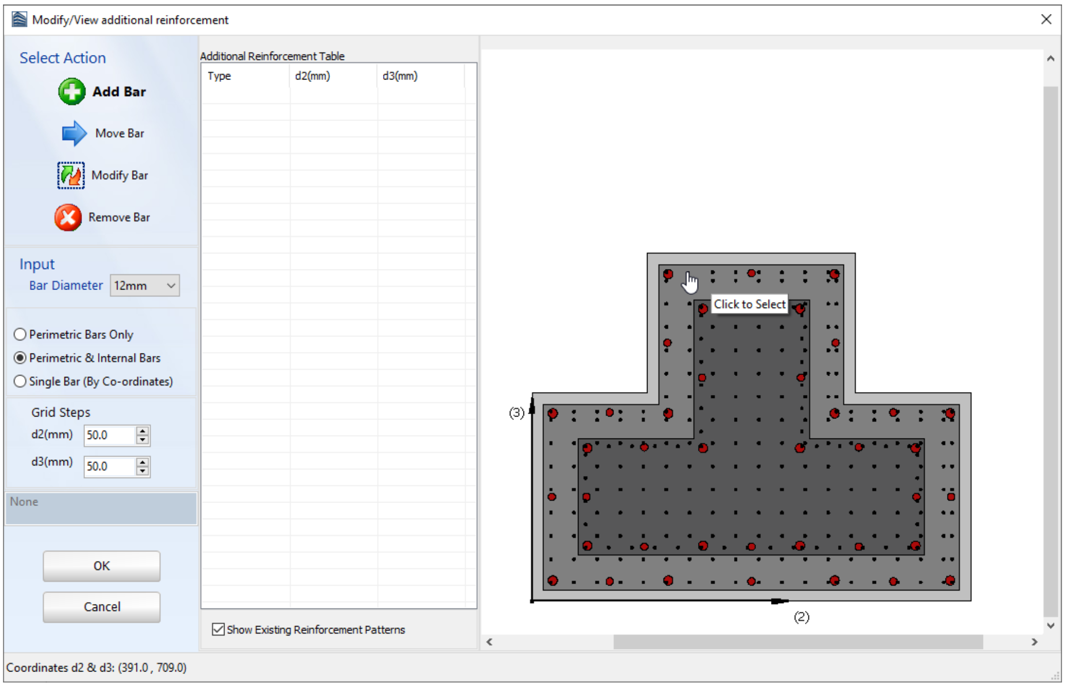
Reinforcement View

Jackets may be applied to the section in the Jacket area by selecting the jacket type, i.e. whether it is full jacketed or 3-sided jacket, and assigning the material set and the longitudinal and transverse reinforcement of the jacket.



Jacket

Adding single longitudinal reinforcement bars to the jacket can also be carried out through the corresponding Additional Rebars module, where additional reinforcement may be introduced graphically to both the existing and the new part of the section, as shown in the following figure:



Modify/View additional reinforcement window

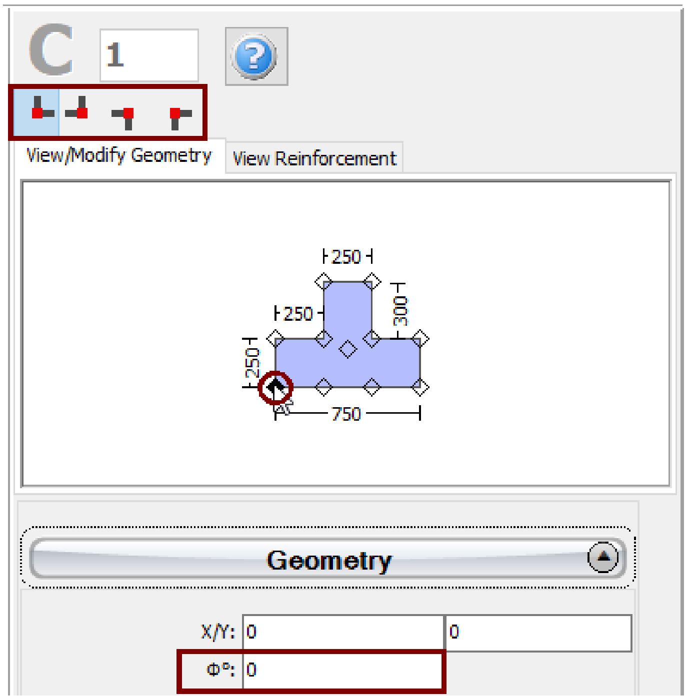
Isolators can also be added at different locations of the column. They are assigned through the Isolator module, where the users may select the geometry (location -bottom, top or intermediate point- and the height of the isolator), its type (elastomeric, lead rubber or curve surface slider) and the isolator parameters: the vertical and horizontal stiffnesses, and the shear yield strength and the strain hardening ratio (for elastomeric and lead rubber isolators, which are modelled as isolator1 element type) or the friction coefficient and the radius of the pendulum (for curve surface sliders, which are modelled as isolator2 element type).

Further, FRP wraps may be assigned to column elements through the FRP Wrapping module, where the users may select the FRP wrap from a list of the most common products found in the market, or introduce user defined values.

In the Advanced Modelling area, the code-based settings of the structural member can be defined through the Advanced Member Properties dialog box that opens from the corresponding button. The member's modelling parameters may be also defined from the Modelling Parameters dialog box, accessed by the corresponding button.

NOTE: When the section is jacketed, in the *Advanced Member Properties* module users should take decisions on the parameters, so as to account for the entire section, i.e. for both the existing and the new parts.

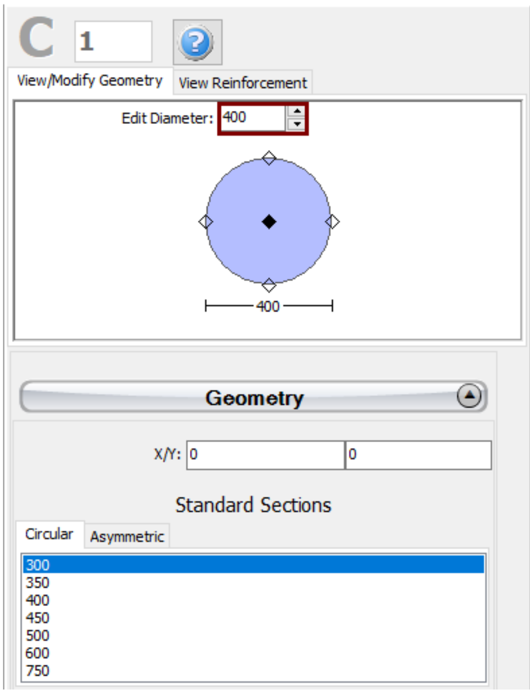
After defining all the section's properties, the new member may be added with a simple click on the Building Modeller Main Window. The location of the section that corresponds to the insertion point (i.e. the mouse click), and rotation of the section on plan view may be selected from the Member Properties window.



Selecting the insertion point and rotate the section's plan view

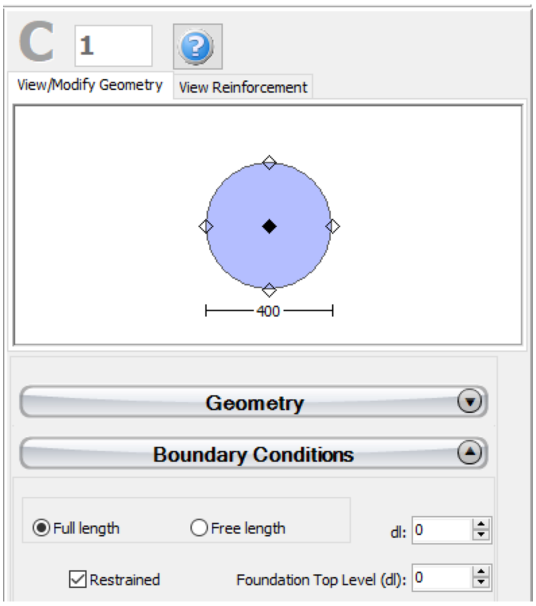
Circular Column

Circular columns may be inserted from the main menu (*Insert > Insert Circular Column*) or through the corresponding toolbar button . On the Properties Window that appears users can adapt the section's dimensions either in the View/Modify Geometry window or by selecting one section from the predefined standard sections.



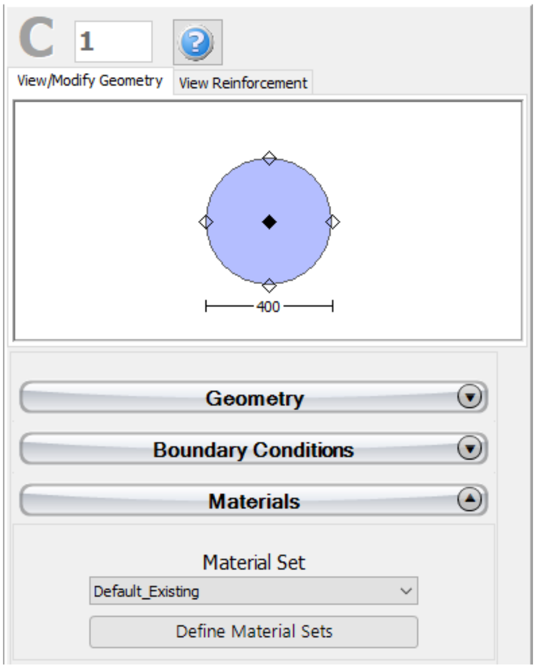
Edit sections dimensions

It is possible to define a column height different from the general storey height, through the selection of the Free length radio button and the assignment of a different length. If, on the other hand, the Full length radio button is selected then the member has the same height with the storey height. In addition, the foundation level of the column may be adapted, thus providing the possibility to the user to define different foundation levels.



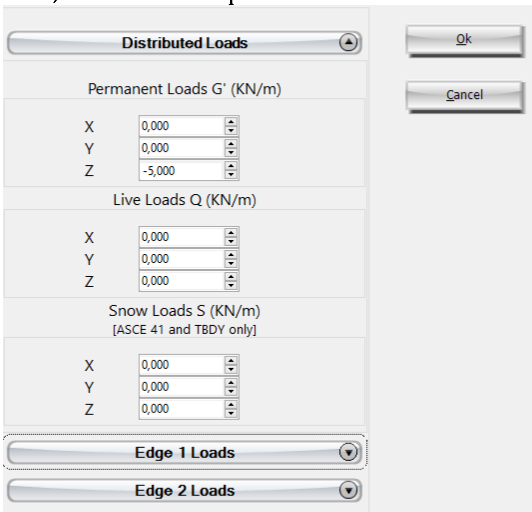
Boundary Conditions

The material set properties can be defined from the main menu (*Tools > Define Material Sets*), through the corresponding toolbar button, or through the Define Material Sets button within the member's Properties Window. The required values for the definition of the materials properties depend on the type of the members, i.e. existing or new members. By default, there are two material schemes, one for the existing elements and one for the new ones.



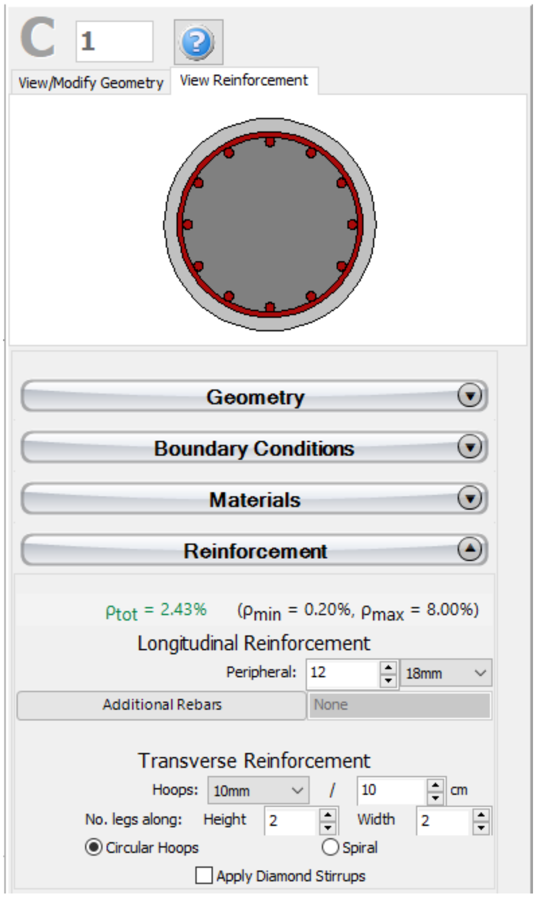
Materials

Additional loads can be defined by clicking on the Distributed and Edge Loads button. Users can define uniform distributed forces along the length of the member in all three translational directions X, Y or Z, and forces or moments in any translational or rotational direction (X, Y, Z, RX, RY or RZ) at either of the two edges of the member (bottom or top). Additional permanent loads G' (not associated with the self-weight of the structure), live Q and snow S loads may be applied, with the latter being applicable only to ASCE 41 and TBDY. By default, all loads are equal to zero.



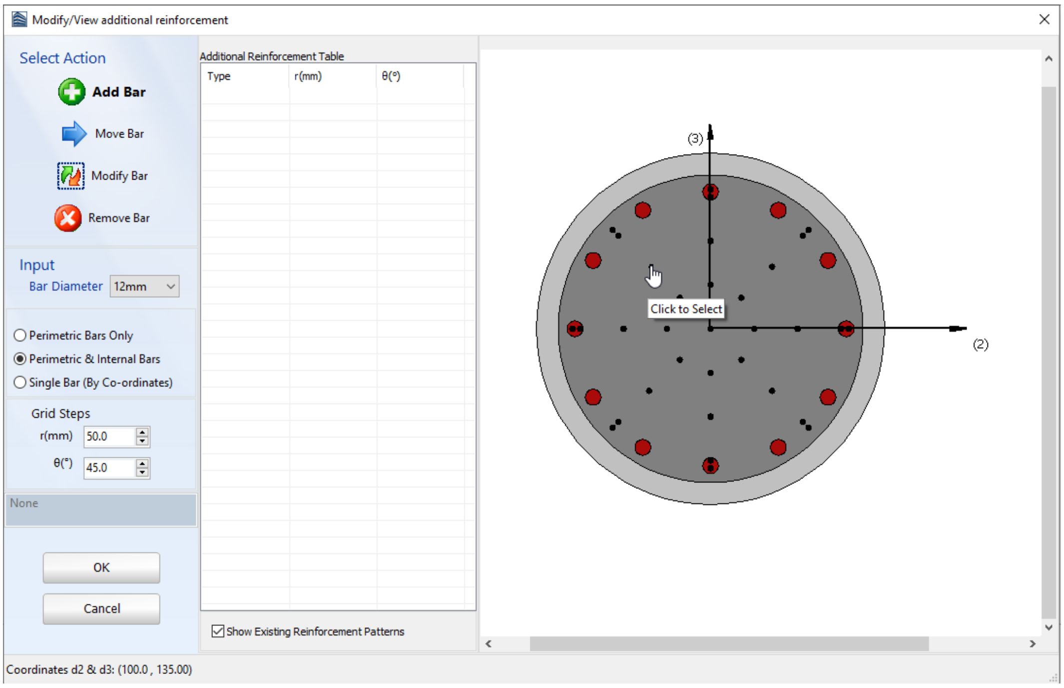
Distributed and Edge Loads window

Further, the longitudinal and transverse reinforcement may be defined by editing the relevant reinforcement pattern controls.



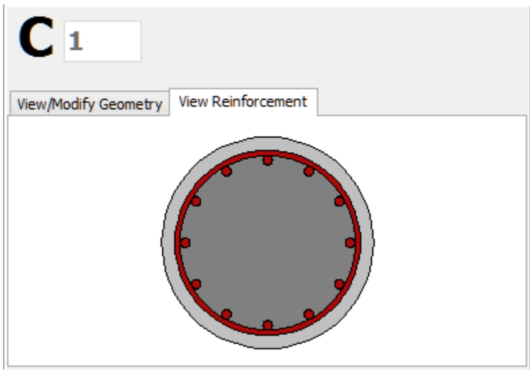
Reinforcement Pattern

Adding single longitudinal reinforcement bars may also be carried out through the corresponding Additional Rebars module, where additional reinforcement may be introduced graphically as shown in the following figure:



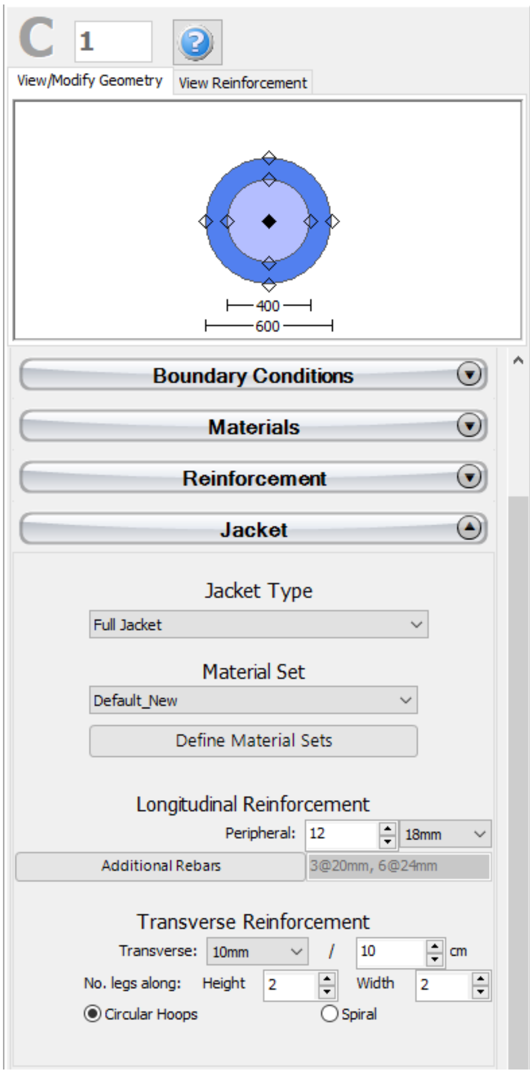
Modify/View additional reinforcement window

On the Properties Window users may choose between the View Reinforcement, where the reinforcement of the section is displayed (longitudinal and transverse), and the View/Modify Geometry, where the section's dimensions may be viewed and modified.



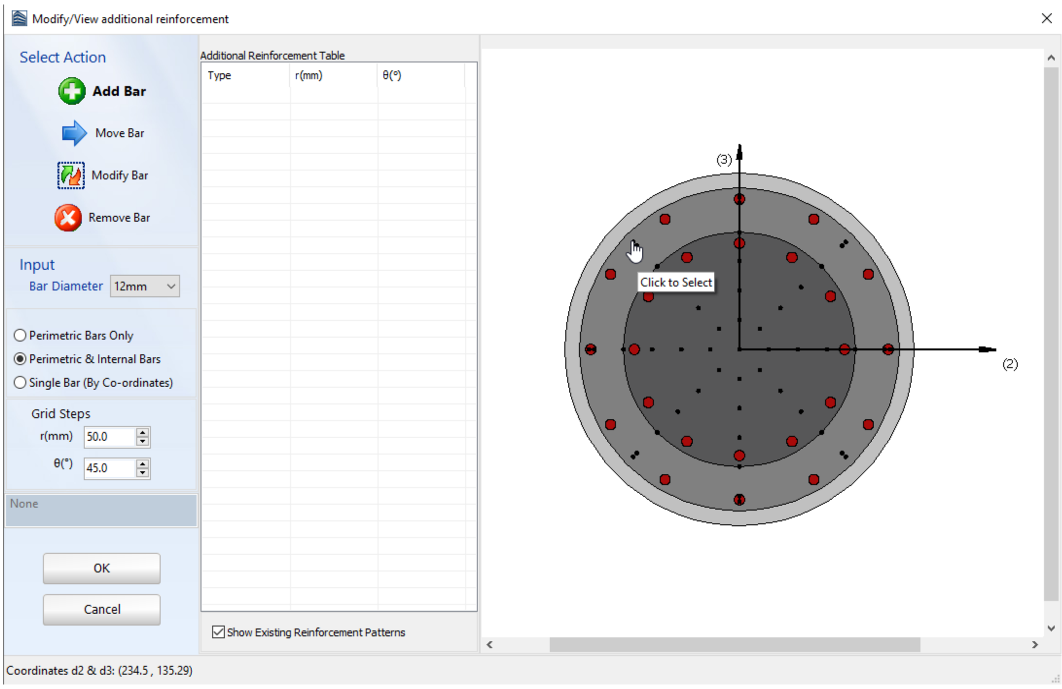
Reinforcement View

Jacket may be applied to the section in the Jacket area by selecting the full jacket type and assigning the material set and the longitudinal and transverse reinforcement of the jacket.



Jacket

Adding single longitudinal reinforcement bars to the jacket can also be carried out through the corresponding Additional Rebars module, where additional reinforcement may be introduced graphically to both the existing and the new part of the section, as shown in the following figure:



Modify/View additional reinforcement window

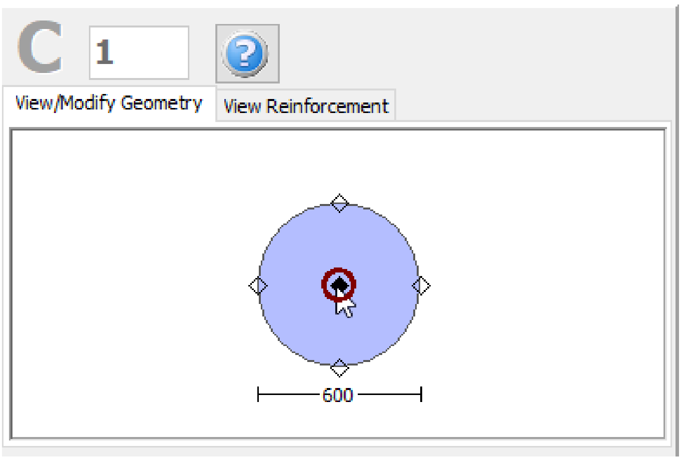
Isolators can also be added at different locations of the column. They are assigned through the Isolator module, where the users may select the geometry (location -bottom, top or intermediate point- and the height of the isolator), its type (elastomeric, lead rubber or curve surface slider) and the isolator parameters: the vertical and horizontal stiffnesses, and the shear yield strength and the strain hardening ratio (for elastomeric and lead rubber isolators, which are modelled as isolator1 element type) or the friction coefficient and the radius of the pendulum (for curve surface sliders, which are modelled as isolator2 element type).

Further, FRP wraps may be assigned to column elements through the FRP Wrapping module, where the users may select the FRP wrap from a list of the most common products found in the market, or introduce user defined values.

In the Advanced Modelling area, the code-based settings of the structural member can be defined through the Advanced Member Properties dialog box that opens from the corresponding button. The member's modelling parameters may be also defined from the Modelling Parameters dialog box, accessed by the corresponding button.

NOTE: When the section is jacketed, in the *Advanced Member Properties* module users should take decisions on the parameters, so as to account for the entire section, i.e. for both the existing and the new parts.

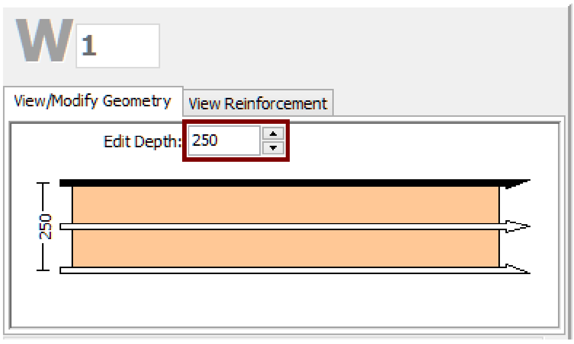
After defining all the section's properties, the new member may be added with a simple click on the Building Modeller Main Window. The location of the section that corresponds to the insertion point (i.e. the mouse click), and rotation of the section on plan view may be selected from the Member Properties window.



Selecting the insertion point

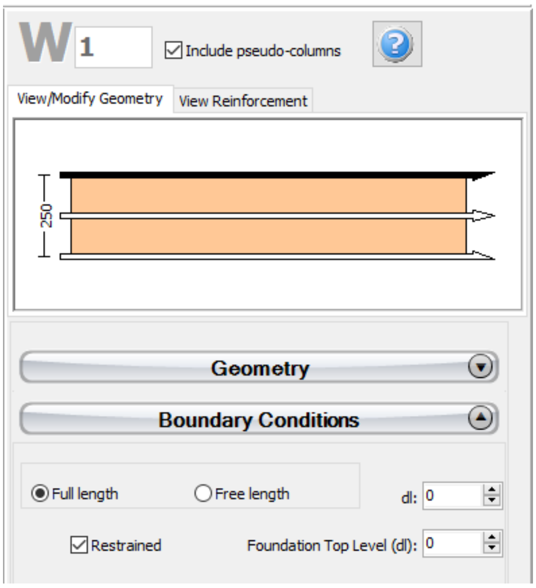
Wall

Walls may be added from the main menu (*Insert > Insert Wall*) or the corresponding toolbar button . On the Properties Window that appears users can adapt the section's width dimension in the View/Modify Geometry window, whereas its length is graphically defined with its insertion by specifying two points, start and end. Initially, the pseudo-columns width is automatically estimated as one fifth (1/5) of the total wall's length with a maximum value equal to 600 mm. After the insertion of the wall, it can be modified from the wall's Properties Window.



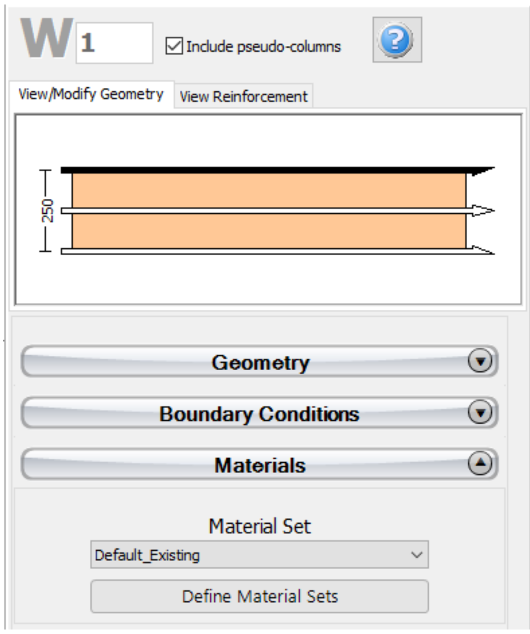
Edit sections dimensions

It is possible to define a wall height different from the general storey height, through the selection of the Free length radio button and the assignment of a different length. If, on the other hand, the Full length radio button is selected then the member has the same height with the storey height. In addition, the foundation level of the column may be adapted, thus providing the possibility to the user to define different foundation levels.



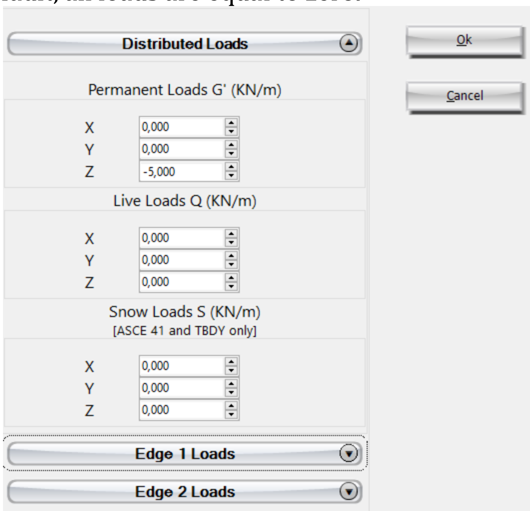
Boundary Conditions

The material set properties can be defined from the main menu (*Tools > Define Material Sets*), through the corresponding toolbar button, or through the Define Material Sets button within the member's Properties Window. The required values for the definition of the materials properties depend on the type of the members, i.e. existing or new members. By default, there are two material schemes, one for the existing elements and one for the new ones.



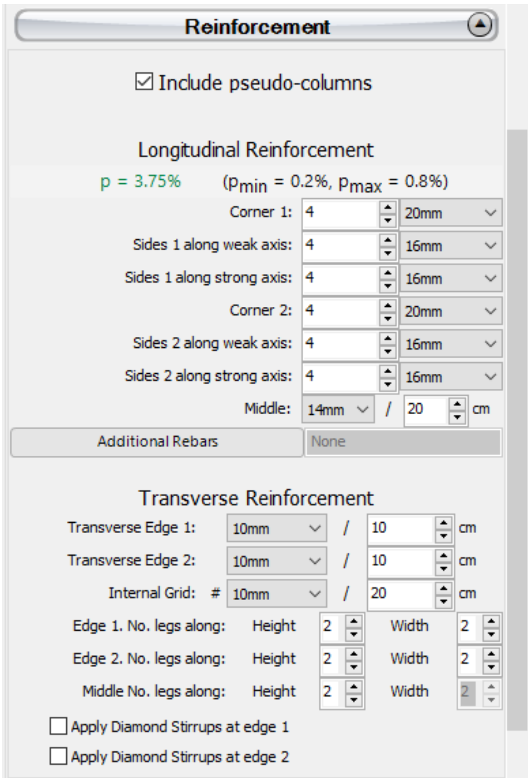
Materials

Additional loads can be defined by clicking on the Distributed and Edge Loads button. Users can define uniform distributed forces along the length of the member in all three translational directions X, Y or Z, and forces or moments in any translational or rotational direction (X, Y, Z, RX, RY or RZ) at either of the two edges of the member (bottom or top). Additional permanent loads G' (not associated with the self-weight of the structure), live Q and snow S loads may be applied, with the latter being applicable only to ASCE 41 and TBDY. By default, all loads are equal to zero.



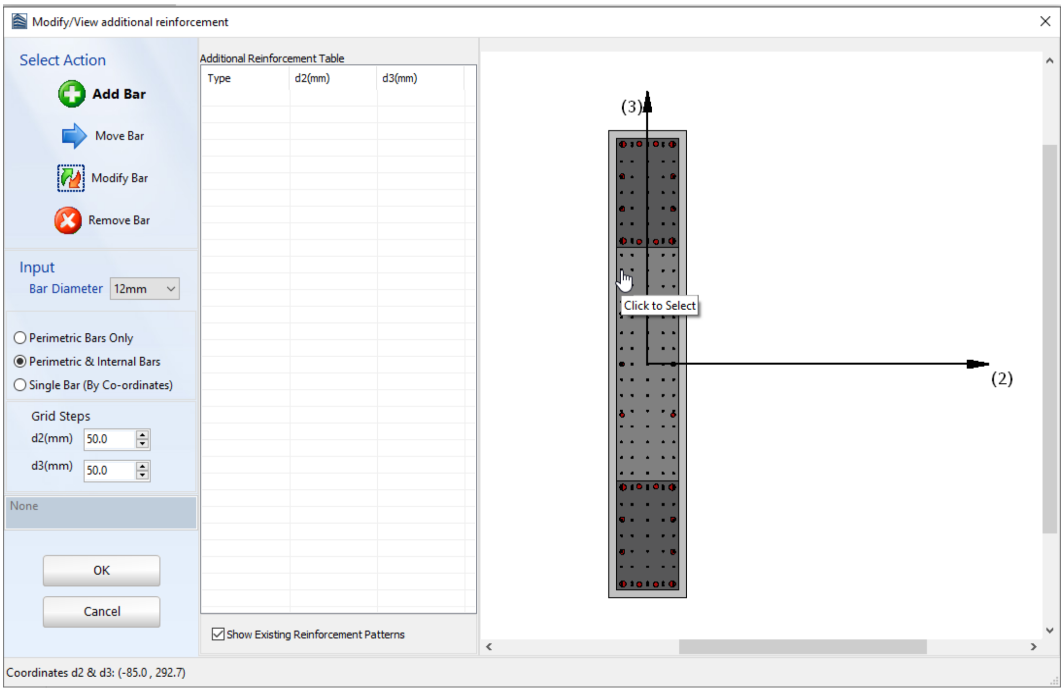
Distributed and Edge Loads window

Further, the option to include pseudo-columns is available is available in the reinforcement area and the longitudinal and transverse reinforcement may be defined by editing the relevant reinforcement pattern controls.



Reinforcement Pattern

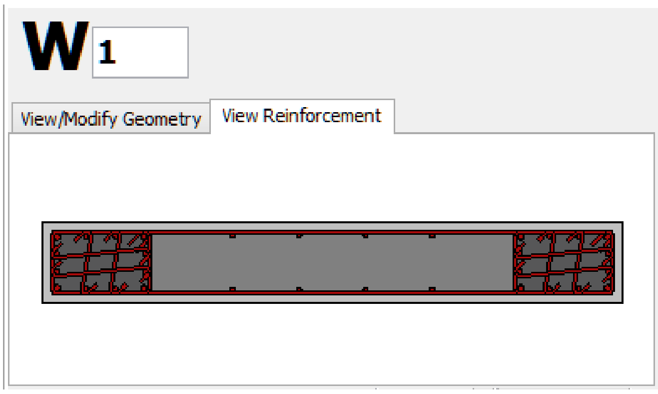
Adding single longitudinal reinforcement bars may also be carried out through the corresponding Additional Rebars module, where additional reinforcement may be introduced graphically as shown in the following figure:



Modify/View additional reinforcement window

NOTE: In order to add longitudinal reinforcement bars through the Additional Rebars module, users should first insert the wall section in the model, so as to have completely defined the wall's dimensions, that is wall's total length and pseudo-columns width.

On the Properties Window users may choose between the View Reinforcement, where the reinforcement of the section is displayed (longitudinal and transverse), and the View/Modify Geometry, where the section's dimensions may be viewed and modified.



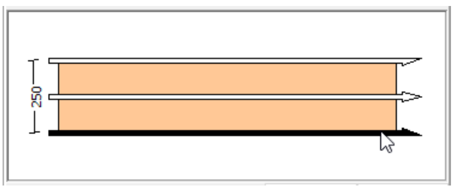
Reinforcement View

Isolators can also be added at different locations of the wall. They are assigned through the Isolator module, where the users may select the geometry (location -bottom, top or intermediate point- and the height of the isolator), its type (elastomeric, lead rubber or curve surface slider) and the isolator parameters: the vertical and horizontal stiffnesses, and the shear yield strength and the strain hardening ratio (for elastomeric and lead rubber isolators, which are modelled as isolator1 element type) or the friction coefficient and the radius of the pendulum (for curve surface sliders, which are modelled as isolator2 element type).

Further, FRP wraps may be assigned to column elements through the FRP Wrapping module, where the users may select the FRP wrap from a list of the most common products found in the market, or introduce user defined values.

In the Advanced Modelling area, the code-based settings of the structural member can be defined through the Advanced Member Properties dialog box that opens from the corresponding button. The member's modelling parameters may be also defined from the Modelling Parameters dialog box, accessed by the corresponding button.

Contrary to the column's definition, where a simple click is adequate to define the member, in wall sections two points should be outlined on the Building Modeller Window. The inserting line can lie at the centre or at either of the two sides of the wall; this can be determined by clicking on any of the three lines on the View/Modify Geometry window (the black line is the selected option).

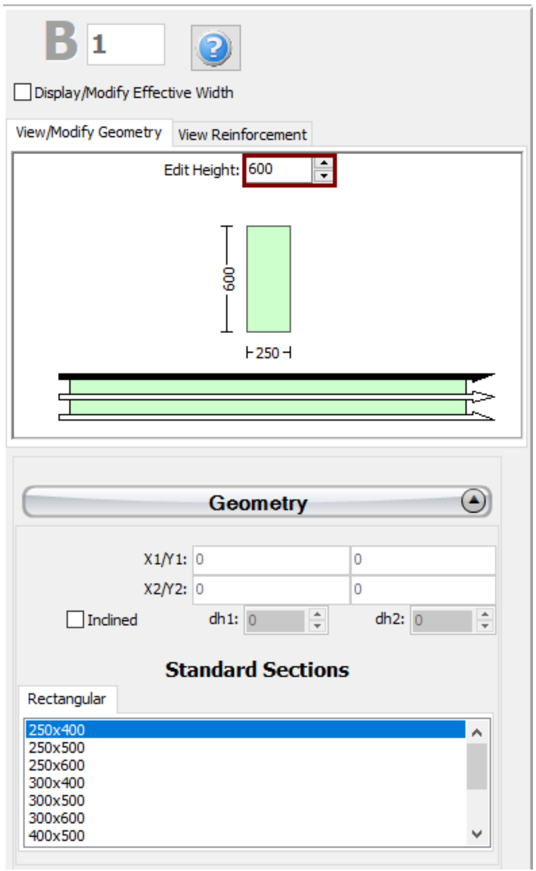


Select the inserting line

Insertion of wall

Beam

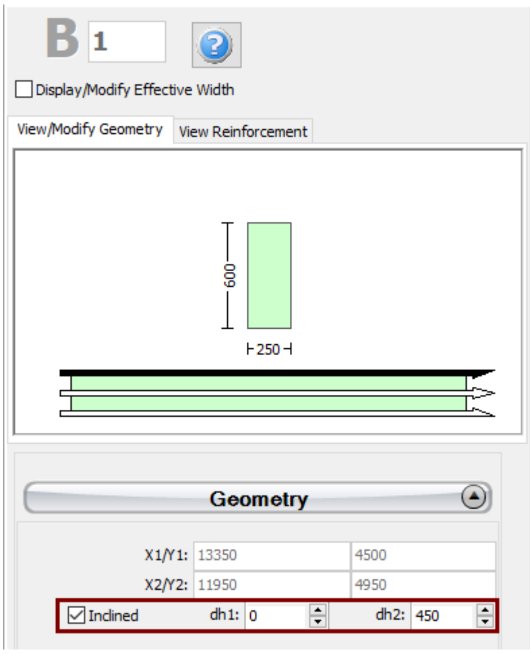
Beams may be inserted from the main menu (*Insert > Insert Beam...*) or through the corresponding toolbar button . On the Properties Window that appears users can adapt the section's dimensions either in the View/Modify Geometry window or by selecting one section from the predefined standard sections.



Edit sections dimensions

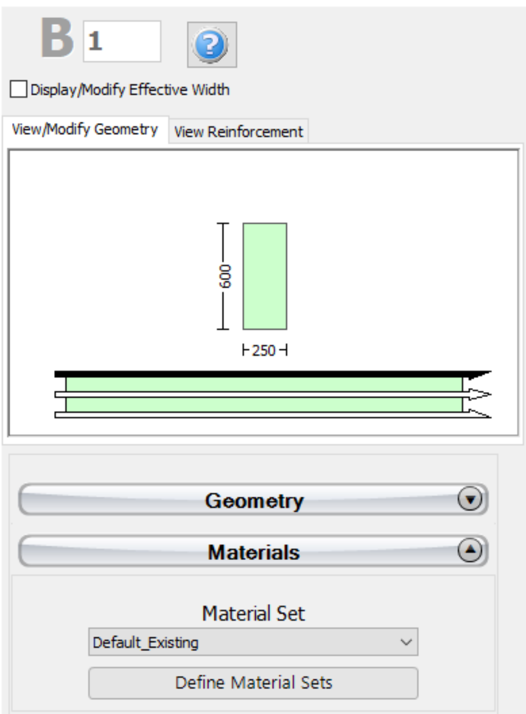
Inclined beams may be efficiently modelled by specifying the elevation differences of the two beam ends relatively to the storey height. The height of the supporting columns is then automatically adapted.

NOTE 1: In the case of beams being supported by the same column at different heights, the program automatically subdivides the column member, so that to simulate effectively the short column that is generated.



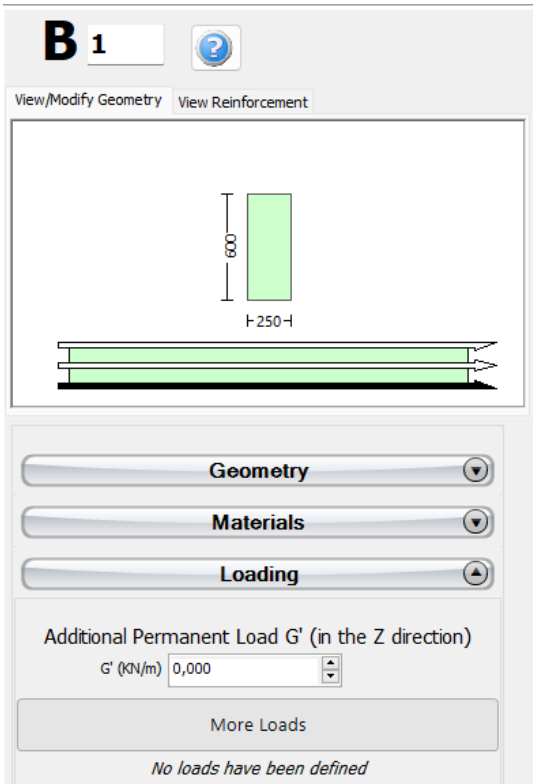
Inclined beam

The material set properties can be defined from the main menu (*Tools > Define Material Sets*), through the corresponding toolbar button, or through the Define Material Sets button within the member's Properties Window. The required values for the definition of the materials properties depend on the type of the members, i.e. existing or new members. By default, there are two material schemes, one for the existing elements and one for the new ones.



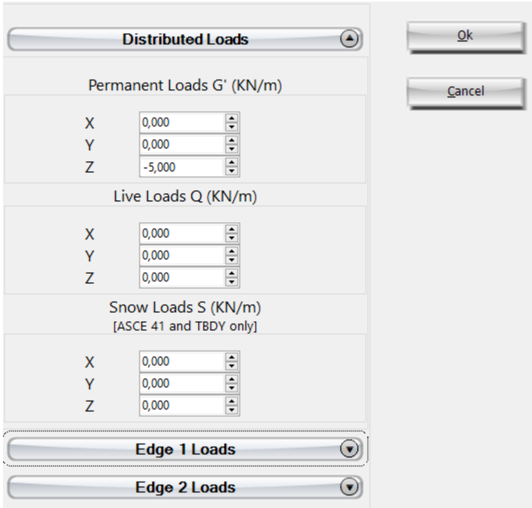
Materials

In the beam's sections module, additional permanent distributed load may also be assigned, which will serve to define any load not associated to the self-weight of the structure (e.g. finishings, infills, variable loading, etc).



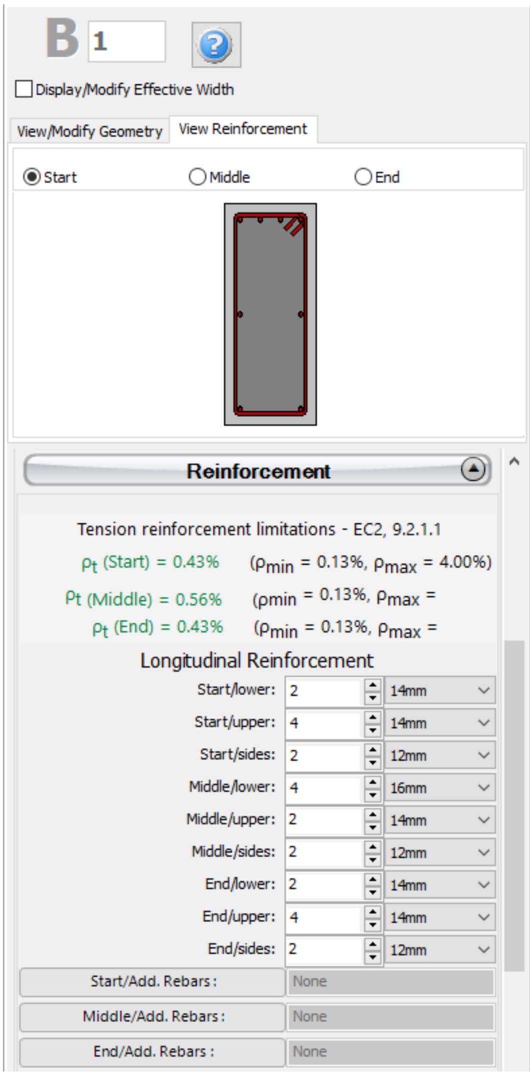
Loading

Also loads can be defined by clicking on the More Loads button. Users can define uniform distributed forces along the length of the member in all three translational directions X, Y or Z, and forces or moments in any translational or rotational direction (X, Y, Z, RX, RY or RZ) at either of the two edges of the member (start or end). Additional permanent loads G' (not associated with the self-weight of the structure), live Q and snow S loads may be applied, with the latter being applicable only to ASCE 41 and TBDY. By default, all loads are equal to zero.



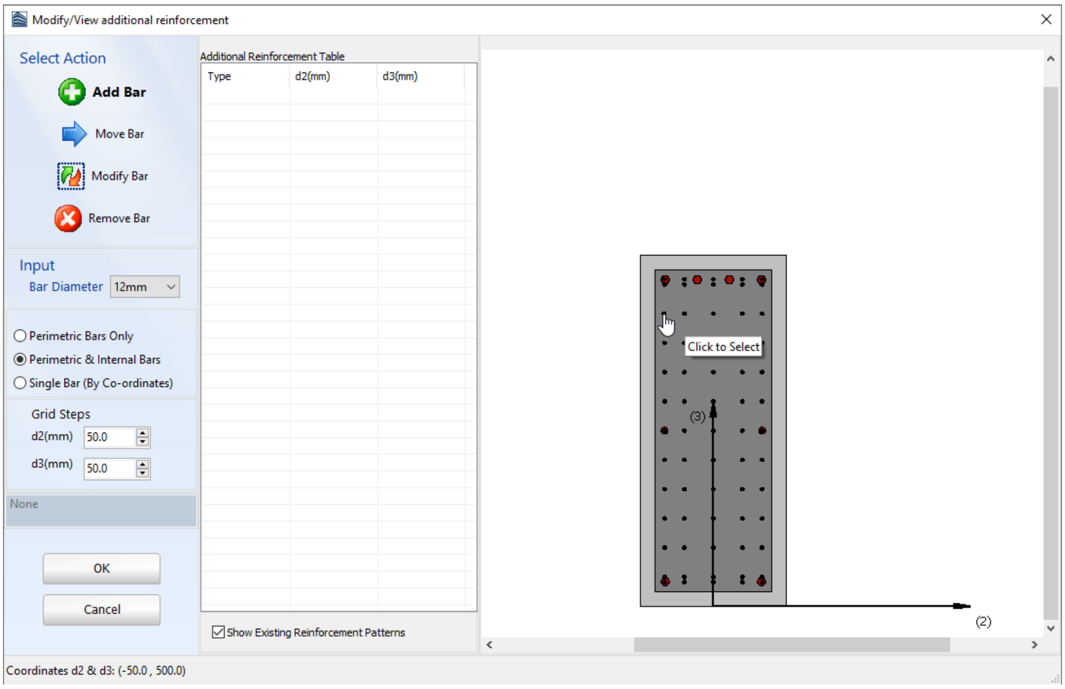
Distributed and Edge Loads window

Further, the longitudinal and transverse reinforcement may be assigned through the relevant reinforcement pattern controls. Different reinforcement patterns may be defined at the middle and at the two edges of the beam.



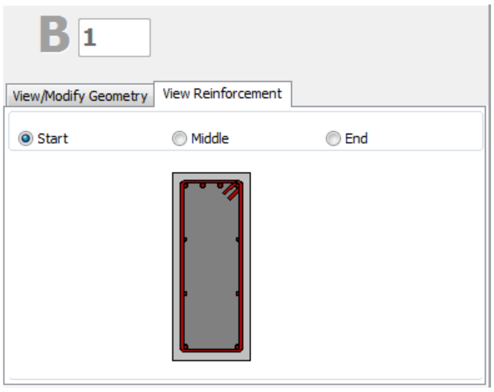
Reinforcement Pattern

Adding single longitudinal reinforcement bars may also be carried out through the corresponding Additional Rebars modules, where additional reinforcement can be introduced graphically as shown in the following figure:

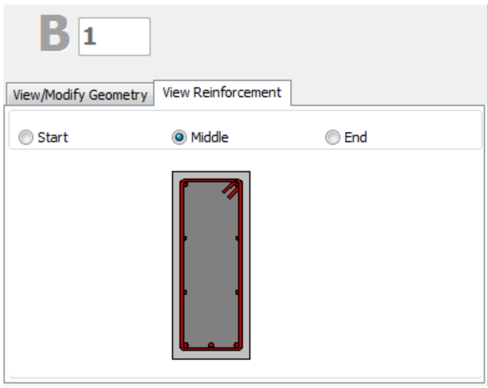


Modify/View additional reinforcement window

On the Properties Window users may choose between the View Reinforcement, where the reinforcement of the start, middle and end sections is displayed (longitudinal and transverse), and the View/Modify Geometry, where the section's dimensions may be viewed and modified.

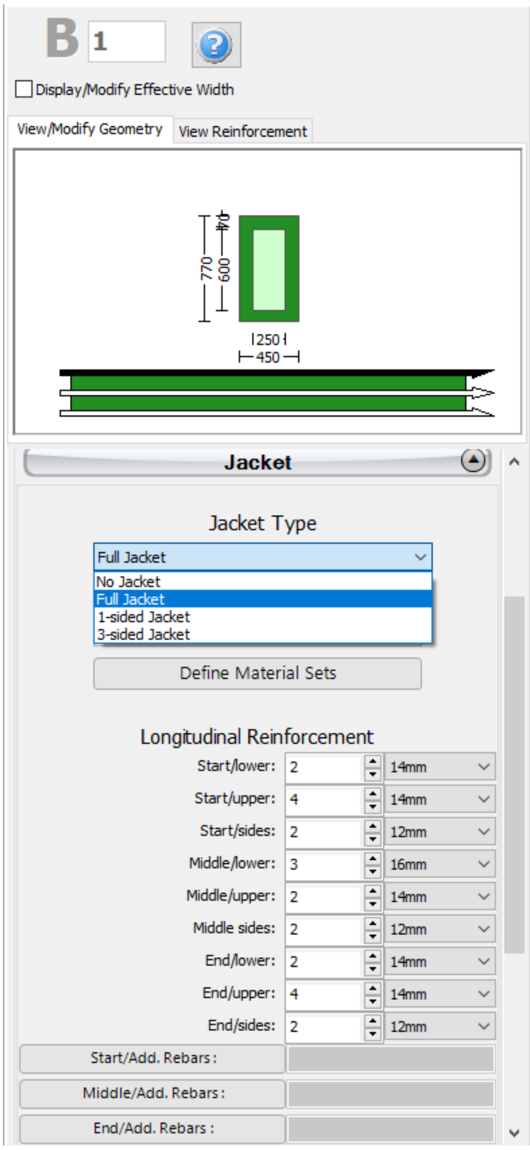


Beam's start section reinforcement



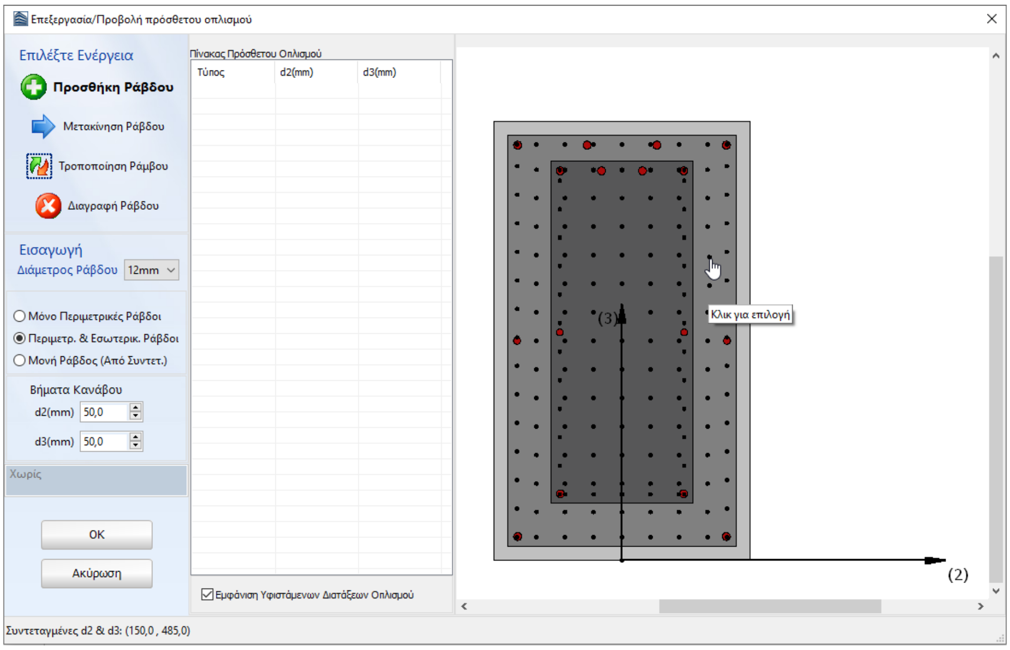
Beam's middle section reinforcement

Jackets may be applied to the section in the Jacket area by selecting the jacket type, i.e. whether it is full jacketed, 3-sided or 1-sided jacket, and assigning the material set and the longitudinal and transverse reinforcement of the jacket.



Jacket

Adding single longitudinal reinforcement bars to the jacket can also be carried out through the corresponding Additional Rebars module, where additional reinforcement may be introduced graphically to both the existing and the new part of the section, as shown in the following figure:

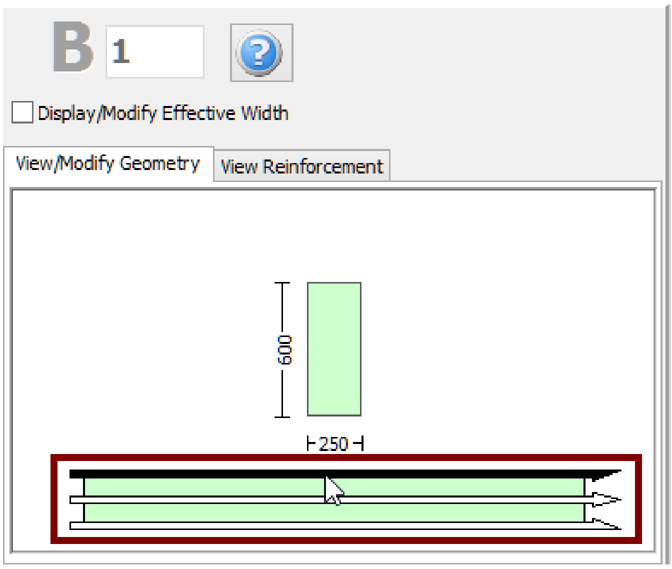


Modify/View additional reinforcement window

In the Advanced Modelling area, the code-based settings of the structural member can be defined through the Advanced Member Properties dialog box that opens from the corresponding button. The member's modelling parameters may be also defined from the Modelling Parameters dialog box, accessed by the corresponding button.

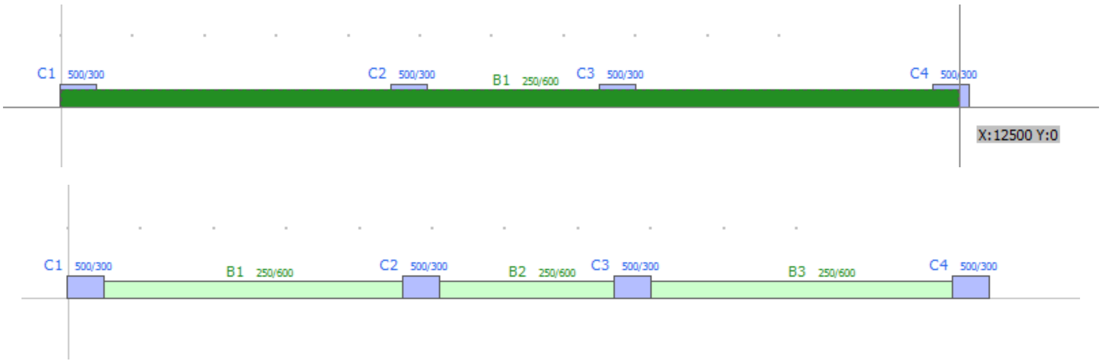
NOTE 2: When the section is jacketed, in the *Advanced Member Properties* module users should take decisions on the parameters, so as to account for the entire section, i.e. for both the existing and the new parts.

In a similar fashion to the walls, for beam's definition two points should be outlined on the Main Window. The inserting line can lie at the centre or at either of the two sides of the beam; this can be determined by clicking on any of the three lines on the View/Modify Geometry (the black line is the selected option).



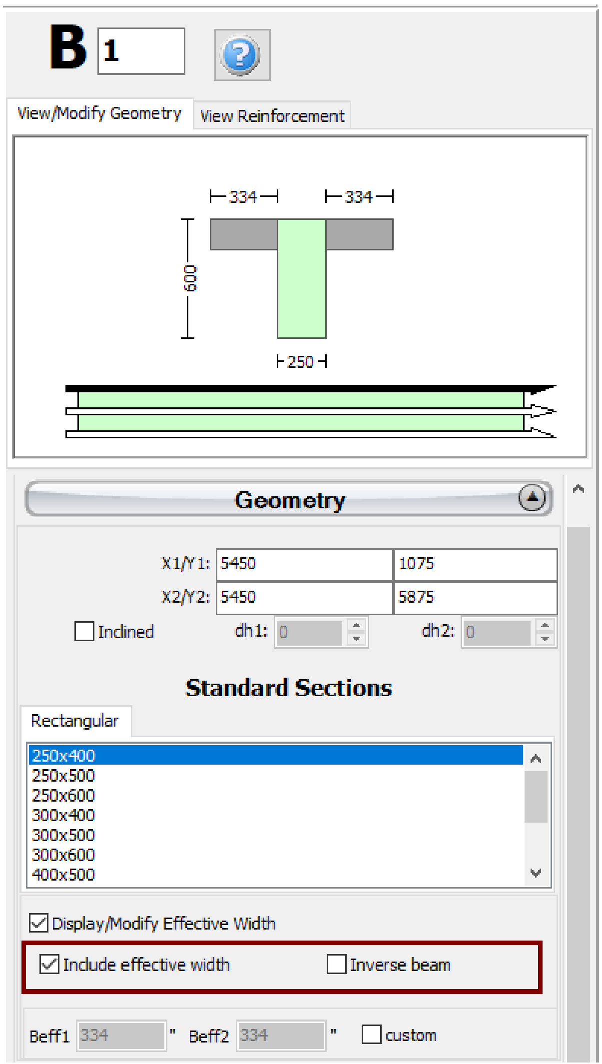
Selecting the insertion point

When an assigned beam intersects an existing column or wall, it is automatically subdivided and two members are thus created. Consequently, several beams may be defined in a row with just two clicks.



Inserting 3 beams in one move

After the definition of the slabs, two additional options may appear on the Geometry area of the beams Properties Window: (i) select whether to include or not the beam's effective width in the calculations and (ii) select whether the beam is inverted or not. The effective width is automatically calculated by the program, but it can also be modified by the user.



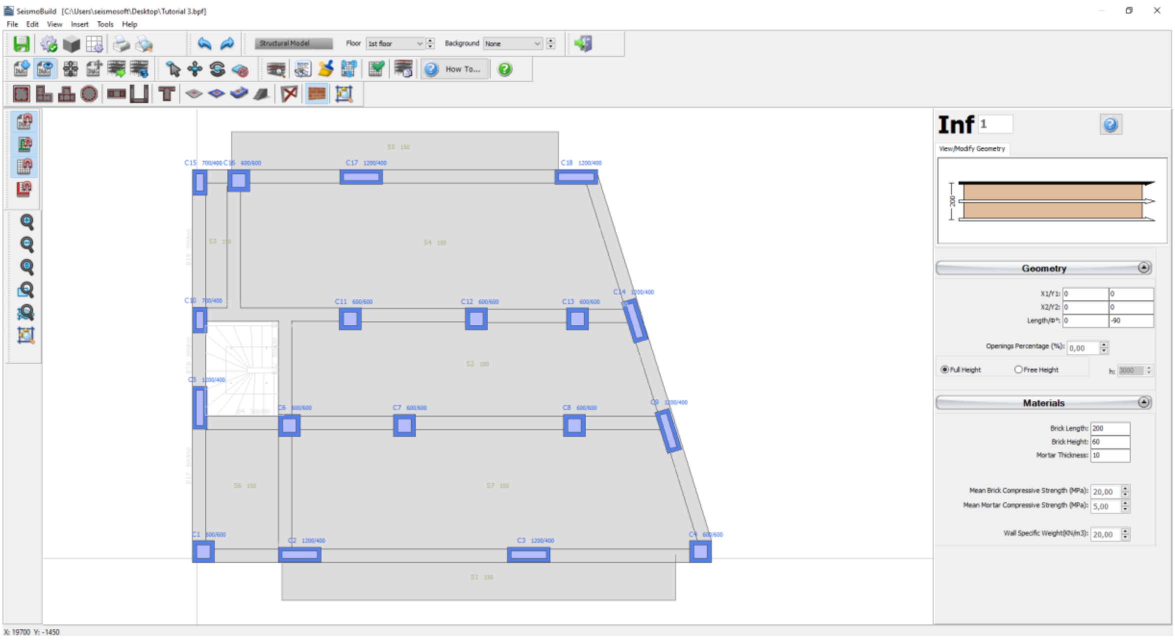
Beam's Properties Window (after the definition of slabs)

Infill Walls

Infill walls may be inserted from the main menu (*Insert > Insert Infill*) or through the corresponding toolbar button . On the Properties Window that appears, users can adapt the main parameters that affect the strength and stiffness of the new infill.

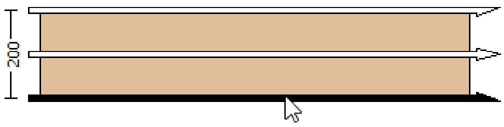
(i) The brick dimensions, length and height
(ii) The mortar thickness
(iii) The brick and mortar compressive strengths
(iv) The percentage of the openings on the wall and
(v) The wall specific weight.

It is possible to define an infill height different from the general storey height, through the selection of the Free Height radio button and the assignment of its height. When this option is selected the neighbouring columns are automatically subdivided in shorter members by the program. If, on the other hand, the Full Height radio button is selected then the member has the same height with the storey height.

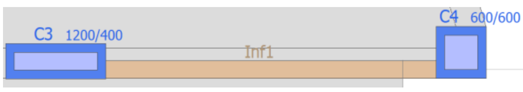


Edit infill parameters

Similarly to the walls and the beams, two points should be outlined on the Building Modeller Window with the mouse. The inserting line can lie at the centre or at either of the two sides of the infill; this can be determined by clicking on any of the three lines on the View/Modify Geometry window (the black line is the selected option).



Select the inserting line

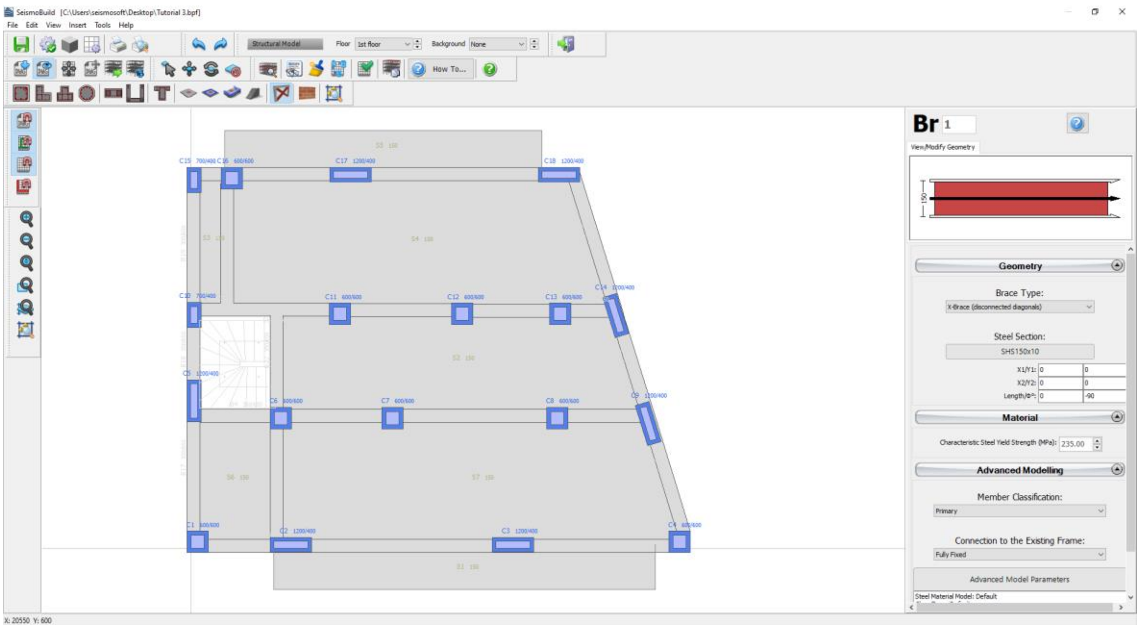


Insertion of the infill

Steel Braces

Steel braces may be inserted from the main menu (*Insert > Insert Steel Brace*) or through the corresponding toolbar button. On the Properties Window that appears, users can adapt the main parameters that affect the strength and stiffness of the new brace.

(i) The type of the brace: currently the following types are supported: (i) X-Brace with connected diagonals, (ii) X-Brace with disconnected diagonals, (iii) diagonal brace, (iv) inverted diagonal brace, (v) V-Brace and (vi) Inverted V-Brace (Chevron Brace).
(ii) The steel section of the brace members
(iii) The yield strength of the brace steel
(iv) The type of connection to the concrete frame (pinned or fully fixed)
(v) The modelling parameters



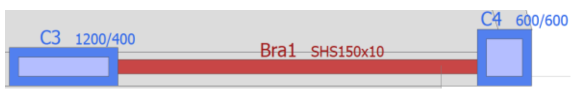
Edit steel brace parameters

Similarly to the walls and the beams, two points should be outlined on the Building Modeller Window with the mouse. The inserting line can lie at the centre or at either of the two sides of the brace; this can be determined by clicking on any of the three lines on the View/Modify Geometry window (the black line is the selected option).

Contrary to the other structural members that are defined with two mouse clicks, in steel braces users do not need to specify the brace width, since this is automatically considered from the selected steel section.

It is noted that the beams, which lie under the V-Brace and over the inverted V-Brace, are automatically subdivided by the program.

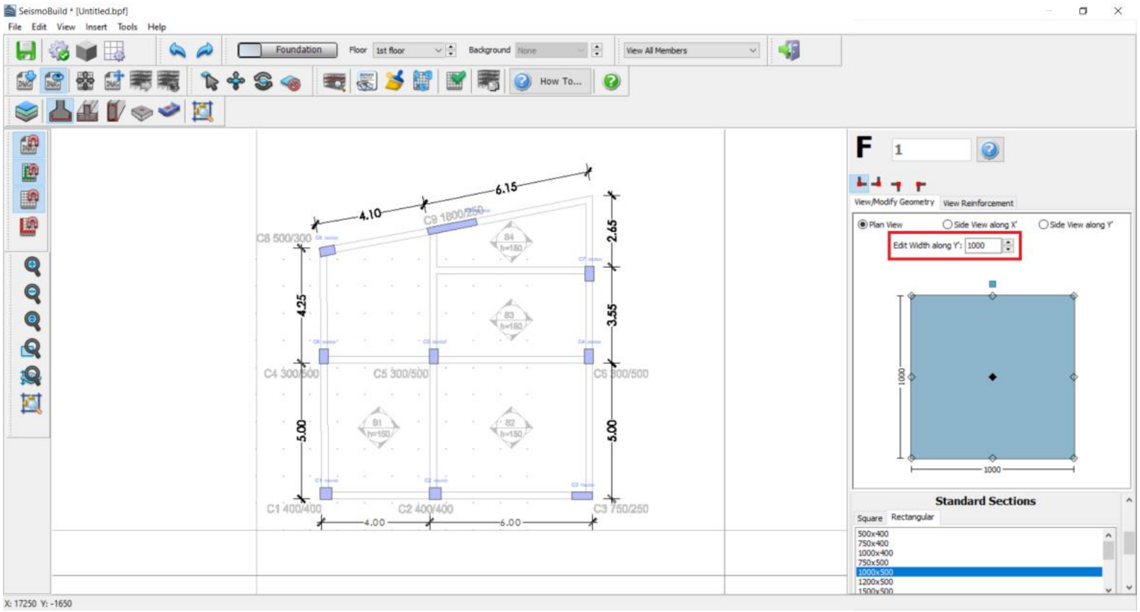
Select the inserting line



Insertion of the Steel Brace

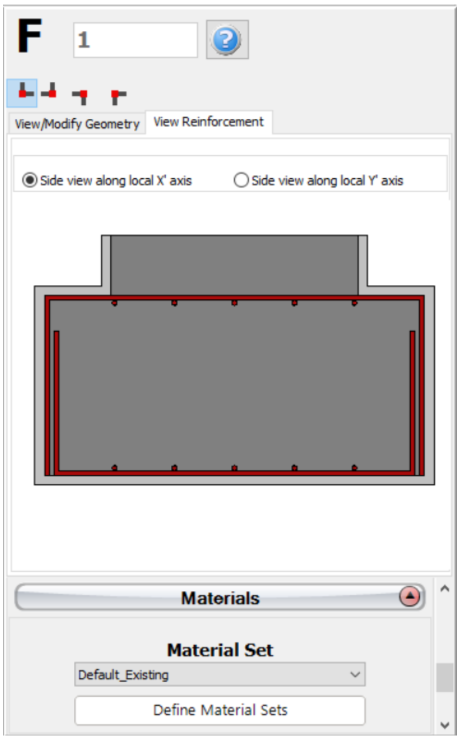
Individual Footings

Individual Footings may be inserted through the corresponding toolbar button . On the Properties Window that appears users can adapt the footing's dimensions either in the View/Modify Geometry window or by selecting one section from the predefined standard sections (square or rectangular).



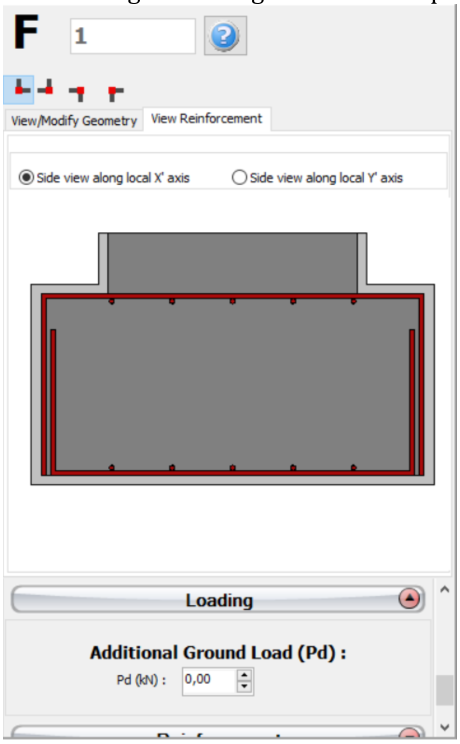
Edit footing's dimensions

The material set properties can be defined from the main menu (*Tools > Define Material Sets*), through the corresponding toolbar  button, or through the Define Material Sets button within the member's Properties Window. The required values for the definition of the materials properties depend on the type of the members, i.e. existing or new members. By default, there are two material schemes, one for the existing elements and one for the new ones.



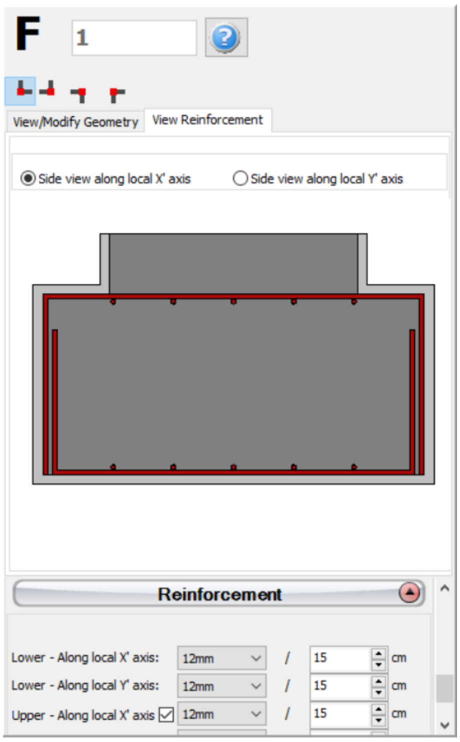
Materials

Additional loading to the Individual Footing from the ground can be specified in the loading module.



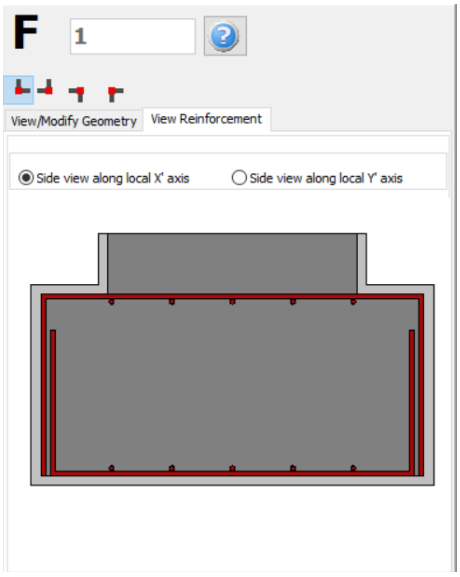
Loading

Further, the reinforcement of the lower and upper part of the individual footing in the two reinforcing directions may be defined by editing the relevant reinforcement pattern controls.



Reinforcement Pattern

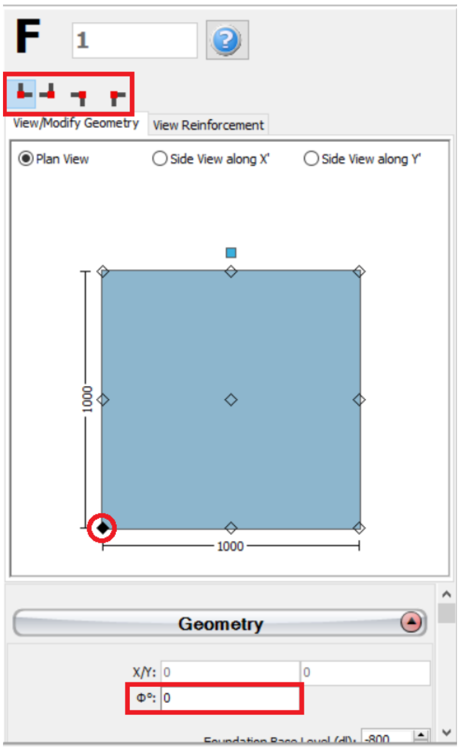
On the Properties Window users may choose between the View Reinforcement, where the reinforcement of the section is displayed, and the View/Modify Geometry, where the footing's dimensions may be viewed and modified.



Reinforcement View

In the Advanced Modelling area, the code-based settings of the footing can be defined through the Advanced Member Properties dialog box that opens from the corresponding button. The footing's modelling parameters may be also defined from the Modelling Parameters dialog box, accessed by the corresponding button.

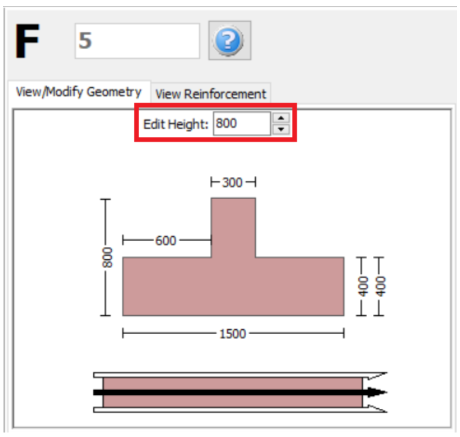
After defining all the properties, of the individual footing the new member may be added with a simple click on the Building Modeller Main Window. The location of the footing that corresponds to the insertion point (i.e. the mouse click), and rotation of the member on plan view may be selected from the Member Properties window.



Selecting the insertion point and rotate the individual footing's plan view

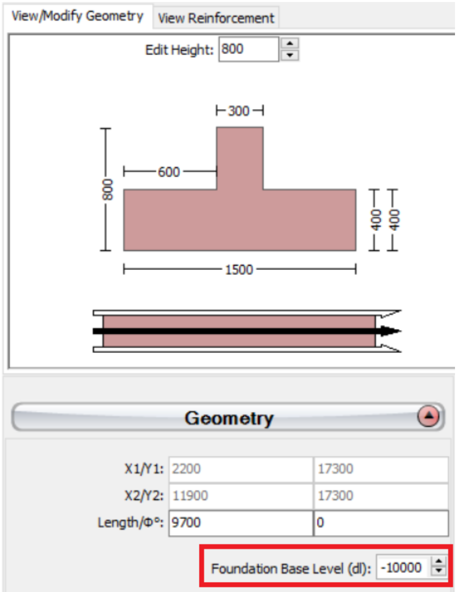
Strip Footing

Strip Footings may be added from the corresponding toolbar button . On the Properties Window that appears users can adapt the section's dimensions in the View/Modify Geometry window, whereas its length is graphically defined with its insertion by specifying two points, start and end.



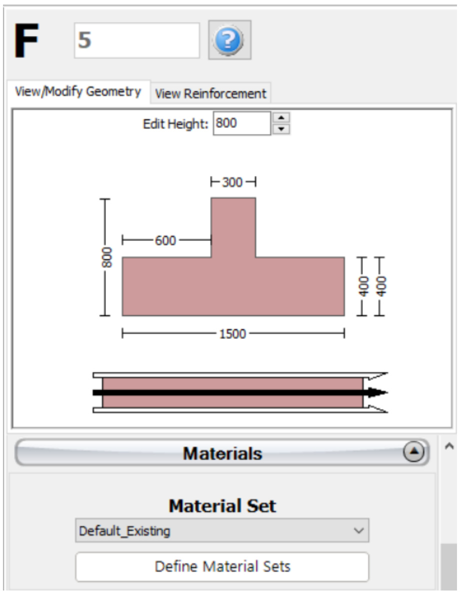
Edit sections dimensions

The base level of the strip foundation may be adapted relatively to the foundation level of the building, in order to define a different foundation level for a specific strip footing.



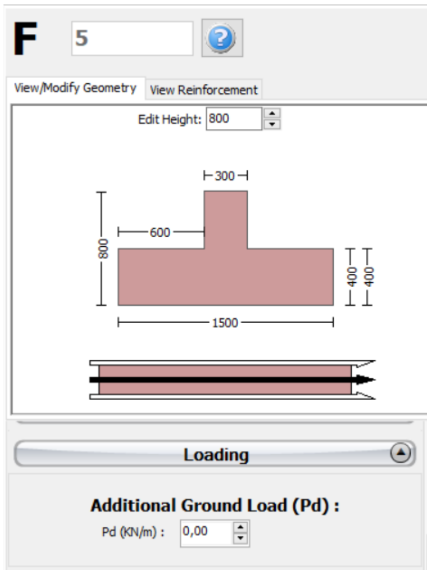
Foundation Base Level

The material set properties can be defined from the main menu (*Tools > Define Material Sets*), through the corresponding toolbar  button, or through the Define Material Sets button within the member's Properties Window. The required values for the definition of the materials properties depend on the type of the members, i.e. existing or new members. By default, there are two material schemes, one for the existing elements and one for the new ones.



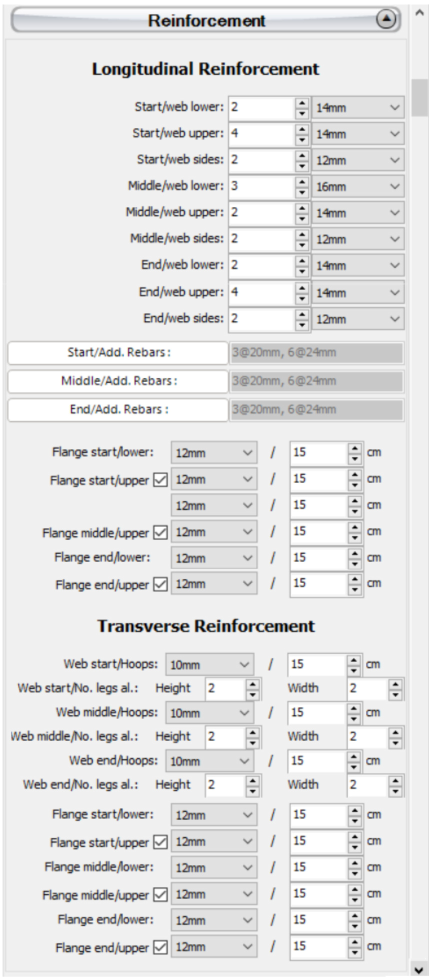
Materials

Additional distributed load may also be assigned in the Loading area, which will serve to define any load from the ground to the strip footing.



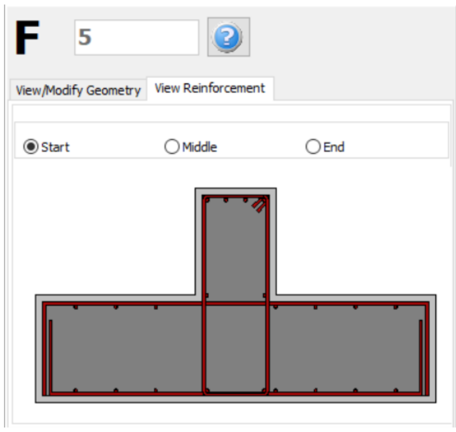
Loading

Further, the longitudinal and transverse reinforcement may be defined by editing the relevant reinforcement pattern controls in the reinforcement area.



Reinforcement Pattern

On the Properties Window users may choose between the View Reinforcement, where the reinforcement of the start, middle and end sections is displayed (longitudinal and transverse), and the View/Modify Geometry, where the section's dimensions may be viewed and modified.

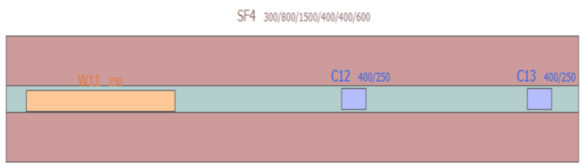


Reinforcement View

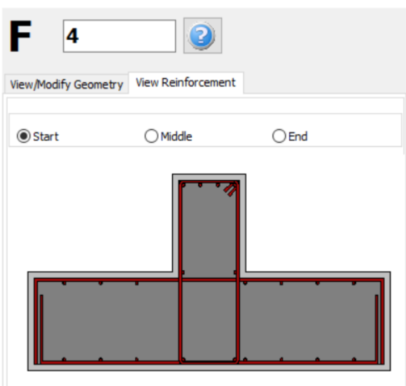
In the Advanced Modelling area, the code-based settings of the structural member can be defined through the Advanced Member Properties dialog box that opens from the corresponding button. The member's modelling parameters may be also defined from the Modelling Parameters dialog box, accessed by the corresponding button.

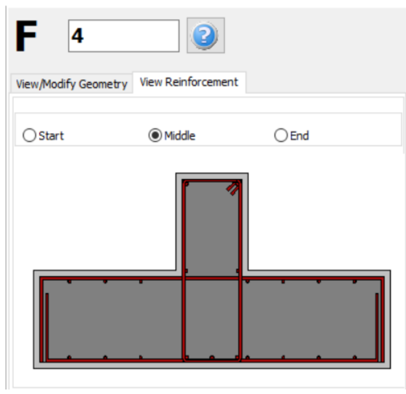
Contrary to the Individual Footings definition, where a simple click is adequate to define the member, in Strip Footing sections two points should be outlined on the Building Modeller Window. The inserting line can lie at the centre or at either of the two sides of the wall; this can be determined by clicking on any of the three lines on the View/Modify Geometry window (the black line is the selected option).

Select the inserting line



Insertion of strip footing

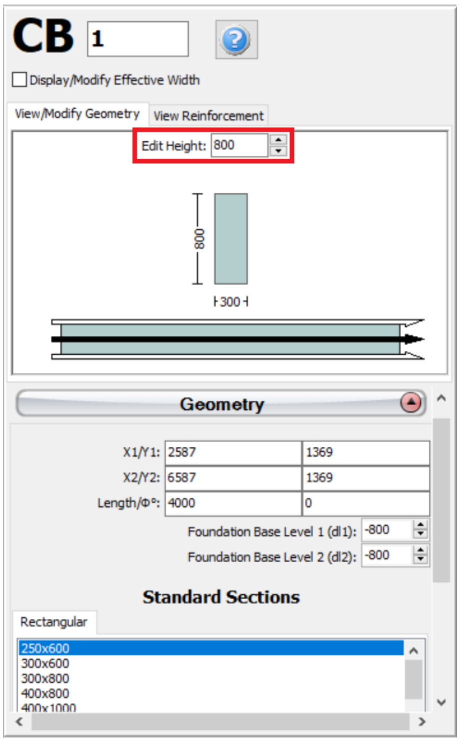




Strip Footing's start section reinforcement **Strip Footing's middle section reinforcement**

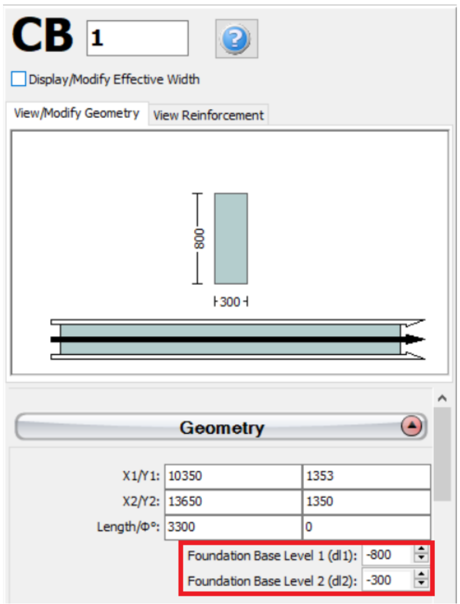
Connecting Beam

Connecting beams may be inserted through the corresponding toolbar button . On the Properties Window that appears users can adapt the section's dimensions either in the View/Modify Geometry window or by selecting one section from the predefined standard sections.



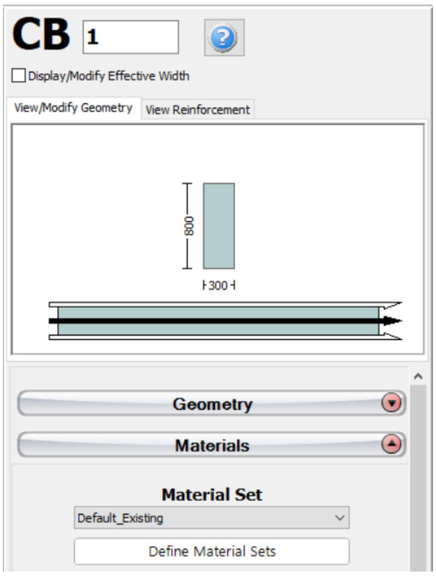
Edit sections dimensions

Inclined connecting beams may be efficiently modelled by specifying the elevation differences of the two connecting beam ends relatively to the foundation base level.



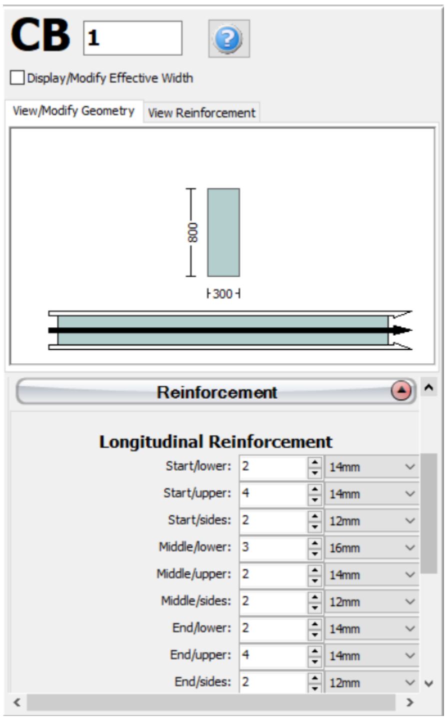
Inclined connecting beam

The material set properties can be defined from the main menu (*Tools > Define Material Sets*), through the corresponding toolbar button, or through the Define Material Sets button within the member's Properties Window. The required values for the definition of the materials properties depend on the type of the members, i.e. existing or new members. By default, there are two material schemes, one for the existing elements and one for the new ones.



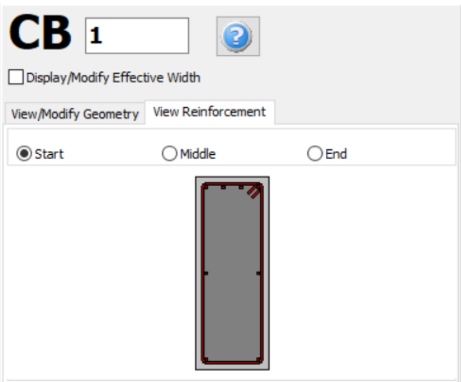
Materials

Further, the longitudinal and transverse reinforcement may be assigned through the relevant reinforcement pattern controls. Different reinforcement patterns may be defined at the middle and at the two edges of the beam.

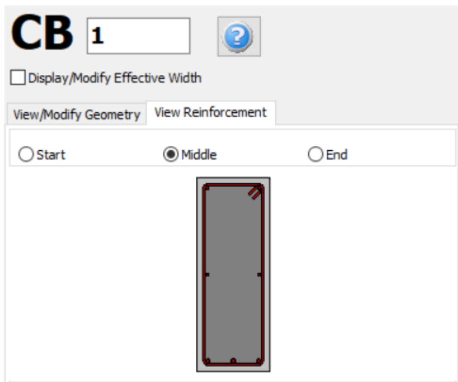


Reinforcement Pattern

On the Properties Window users may choose between the View Reinforcement, where the reinforcement of the start, middle and end sections is displayed (longitudinal and transverse), and the View/Modify Geometry, where the section's dimensions may be viewed and modified.



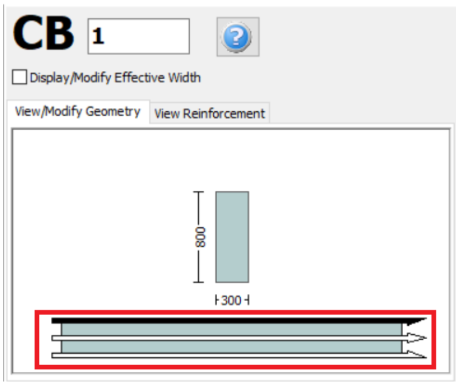
Connecting Beam's start section reinforcement



Connecting Beam's middle section reinforcement

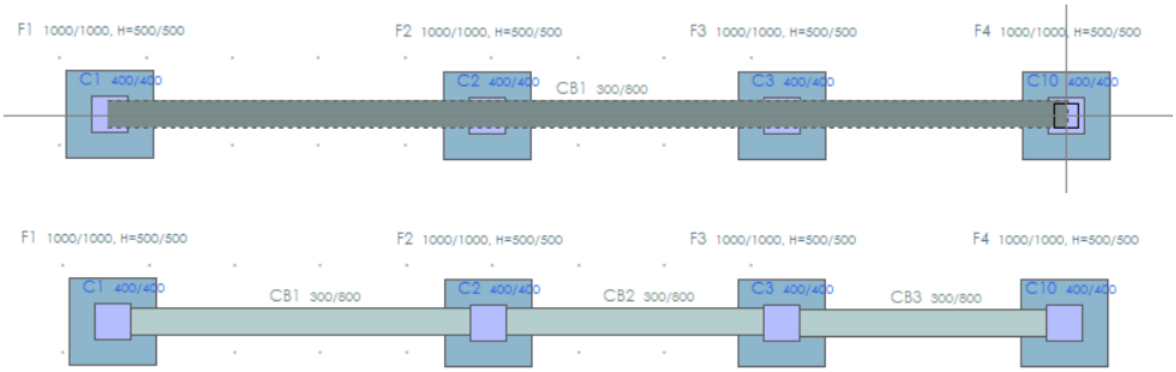
In the Advanced Modelling area, the code-based settings of the structural member can be defined through the Advanced Member Properties dialog box that opens from the corresponding button. The member's modelling parameters may be also defined from the Modelling Parameters dialog box, accessed by the corresponding button.

In a similar fashion to the beams, for connecting beam's definition two points should be outlined on the Main Window. The inserting line can lie at the centre or at either of the two sides of the connecting beam; this can be determined by clicking on any of the three lines on the View/Modify Geometry (the black line is the selected option).



Selecting the insertion point

When an assigned connecting beam intersects an existing column or wall, it is automatically subdivided and two members are thus created. Consequently, several beams may be defined in a row with just two clicks.



Inserting 3 connecting beams in one move

Appendix E - Element Classes

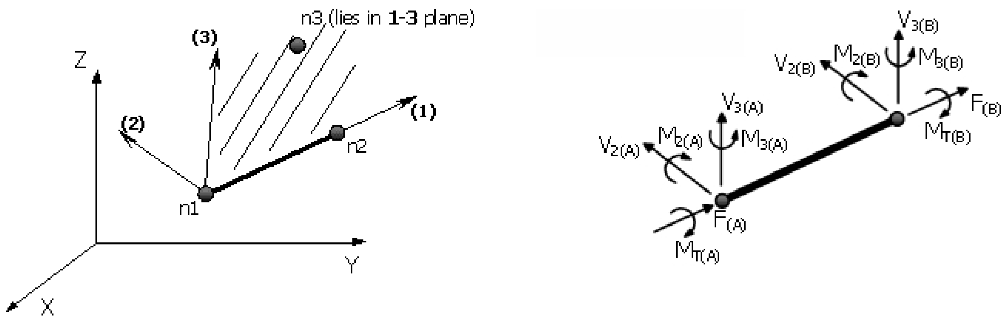
In this appendix the available element types are described in details.

Inelastic force-based frame element type - infrmFB

This is the force-based 3D beam-column element type capable of modelling members of space frames with geometric and material nonlinearities. As described in the Material inelasticity paragraph, the sectional stress-strain state of beam-column elements is obtained through the integration of the nonlinear uniaxial material response of the individual fibres in which the section has been subdivided, fully accounting for the spread of inelasticity along the member length and across the section depth.

Element infrmFB is the most accurate among the four inelastic frame element types of SeismoBuild, since it is capable of capturing the inelastic behaviour along the entire length of a structural member, even when employing a single element per member. Hence, its use allows for very high accuracy in the analytical results, while giving users the possibility of readily employing element chord-rotations output for seismic code verifications (e.g. Eurocode 8, ASCE/SEI 41-23, etc.). The number of section fibres used in equilibrium computations carried out at each of the four element's integration sections is by default equal to 150.

Local axes and output notation are defined in the figure below. Refer to the discussion on global and local frame axes in Appendix B for a detailed description on the determination of the orientation of the local element axis system.



Local Axes and Output Notation for infrmFB elements

Inelastic force-based plastic hinge frame element type - infrmFBPH

This is the plastic-hinge counterpart to the infrmFB element, featuring a similar distributed inelasticity displacement- and forced-based formulation, but concentrating such inelasticity within a fixed length of the element, as proposed by Scott and Fenves [2006].

The advantages of such formulation are not only a reduced analysis time (since fibre integration is carried out for the two member's end sections only), but also a full control/calibration of the plastic hinge length (or spread of inelasticity), which allows the overcoming of localisation issues, as discussed e.g. in Calabrese et al. [2010].

The number of section fibres used in equilibrium computations carried out at the element's end sections is equal to 150. This number of section fibres is sufficient to guarantee an adequate reproduction of the stress-strain distribution across the element's cross-section.

In addition, the plastic hinge length is defined as a percentage of the total element's length, the default percentage in SeismoBuild is set equal to 16.67%.

Local axes and output notation are the same as the infrmFB elements.

Inelastic displacement-based plastic hinge frame element type - infrmDBPH

This is a displacement-based plastic-hinge 3D beam-column element with concentrated plasticity at the two element's ends. It is a typical one-component Giberson model [Giberson, 1967], which consists of one elastic girder and four nonlinear rotational springs attached at the two element's ends in both the 2nd and the 3rd local axis.

All nonlinear deformations of the element are lumped in these rotational springs, whereas the rest of the member remains elastic. The moment-rotation curves in the two local axes at each end are independent. This is obviously a simplification with respect to the force-based plastic hinge element, where inelastic deformations spread over a finite region at the ends of the girder and the behaviour in the two local axes is correlated. However, this lack of accurate modelling is compensated by increased stability and significantly shorter analysis times.

Geometric nonlinearities, specifically force –displacement relationships, are modelled in the DBPH formulation by four nonlinear rotational springs at the element's ends featuring a hysteretic curve based on the Modified Ibarra Medina Krawinkler (MIMK) deterioration curve deterioration curve with bilinear hysteretic rules. The hysteretic curve parameters are calculated automatically on the basis of ASCE 41-23 provisions, hence users need only to specify the member's section. Local axes and output notation are the same as with the other frame element types.

Inelastic displacement-based frame element type - infrmDB

This is the displacement-based 3D beam-column element type capable of modelling members of space frames with geometric and material nonlinearities. As described in the Material inelasticity paragraph, the sectional stress-strain state is obtained through the integration of the nonlinear uniaxial material response of the individual fibres in which the section has been subdivided, fully accounting for the spread of inelasticity along the member length and across the section depth.

The displacement-based formulation follows a standard FE approach [e.g. Hellesland and Scordelis 1981; Mari and Scordelis 1984], where the element deformations are interpolated from an approximate displacement field, before the PVD is used to form the element equilibrium relationship. The DB formulation features two integration sections per element, and the Gauss quadrature is employed for higher accuracy.

In order to approximate nonlinear element response, constant axial deformation and linear curvature distribution are enforced along the element length, which is exact only for prismatic linear elastic elements. Consequently, infrmDB should be employed with members of small length, for which reason infrmDB elements are used in SeismoBuild only to model short columns and beams.

Similarly to the force-based elements, the number of section fibres used in equilibrium computations carried out at each of the element's integration sections is equal to 150.

Local axes and output notation are the same as with the other frame element types.

Elastic frame element - elfrm

There are cases where the use of an inelastic frame element is not required (e.g. members subjected to low levels of excitation and thus responding within their elastic range). For such cases, the employment of an elastic linear frame element might be desirable, for which reason element type *elfrm* has been developed and implemented in SeismoBuild.

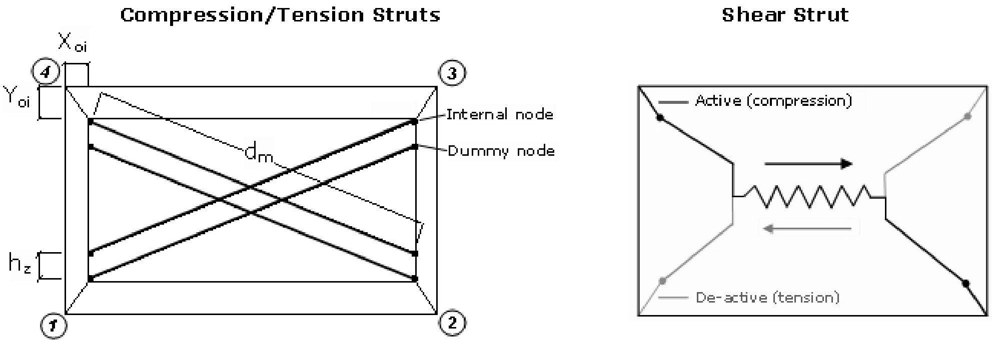
NOTE: In the elfrm element, P-delta effects as well as large displacement/rotation effects are duly taken into account.

Local axes and output notation are the same as infrmDB and infrmFB elements.

Inelastic infill panel element type - infill

In SeismoBuild infills are modelled with the four-node masonry panel element that has been proposed by Crisafulli [1997] for the modelling of the nonlinear response of infill panels in framed structures.

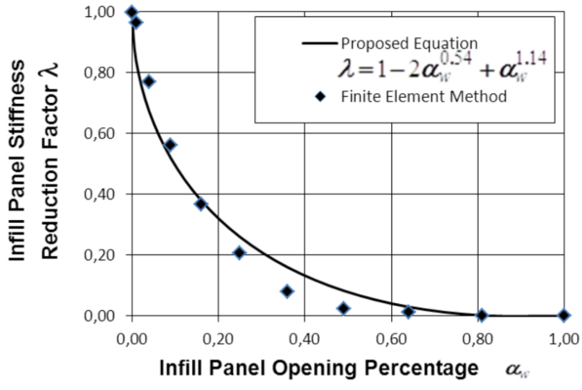
Every panel is represented by six struts; each diagonal direction features two parallel struts to carry axial loads across two opposite diagonal corners and a third one to carry the shear from the top to the bottom of the panel. This latter strut only acts across the diagonal that is on compression, hence its "activation" depends on the deformation of the panel. The diagonal struts employ a specialised hysteretic model that considers the gradual degradation of the infill with increased deformation and loading cycles, while the shear strut uses a dedicated bilinear hysteresis rule. The implementation of the model in SeismoBuild supports the gradual decrease in the area of the diagonal struts with increased deformation and damage, as well as out-of-plane failures for large out-of-plane drifts.



Compression/tension and shear structs in the infill model of SeismoBuild

The effect of the openings of the infill are considered though the following expression proposed by Asteris et al. [2011]:

$\lambda = 1 - 2\alpha_w^{0.54} + \alpha_w^{1.14}$



Infill panel stiffness reduction factor in relation to the openings' percentage

NOTE: In the nonlinear analysis types of SeismoBuild , the strength and stiffness of the infills are introduced after the application of the initial loads, so that the former do not resist to gravity loads (which are normally absorbed by the surrounding frame, erected first).

Linear Link element type - linlink

These are the 3D link elements with uncoupled axial, shear and moment actions that are user in SeismoBuild to model the elastic footings and in the modelling of the elastic strip footings. All six local Degrees of Freedom of the element (F1, F2, F3, M1, M2, M3) follow a linear behaviour.

Nonlinear Link element type - NLlink

These are the 3D link elements with uncoupled axial, shear and moment actions that are used in SeismoBuild to model the inelastic footings and in the modelling of the inelastic strip footings.

The link elements connect two initially coincident structural nodes and require the definition of an independent force-displacement (or moment-rotation) response curve for each of its local six degrees-of-freedom (F1, F2, F3, M1, M2, M3).

Elastomeric Isolator Element (Bouc Wen) - isolator1

Isolator 1 Elements are 3D elements with zero length used to model the behaviour of elastomeric isolators used in Seismic Isolation Applications. Isolator 1 Elements have coupled plasticity properties for the two shear directions (axes 2 and 3 in the local coordinate system of the isolator 1 element) while they are characterised by linear elastic behaviour for the remaining four deformation types. The behaviour in the shear directions is based on the hysteretic behaviour proposed by Wen [1976] and Park et al. [1986].

Friction Pendulum Isolator/System element type – isolator2

Isolator 2 Elements are 3D elements with zero length used to model the behaviour of single friction pendulum isolators used in Seismic Isolation Applications. Isolator 2 Elements have coupled plasticity properties for the two shear directions (axes 2 and 3 in the local coordinate system of the element) while they are characterised by linear elastic behaviour for the remaining four deformation types. The friction model described by Constantinou et al. [1999] is utilised for calculating the friction coefficient of the friction pendulum isolator sliding surface. The friction coefficient is calculated according to the following equation:

$$\mu = ffast\_1 - (ffast\_1 - fslow\_1)exp(-rate\_1\left|v\right|)$$

where ffast_1 and fslow_1 are the isolator friction coefficients at fast and slow velocities respectively, v is the isolator velocity and rate_1 is the rate controlling the transition from low to high velocities.

The Isolator 2 element behaves elastically in the shear directions, with a stiffness equal to the elastic stiffness provided by the user, until the yielding limit defined by the yield strength which is calculated according to the following equation

$$Q_{yield} = \mu P$$

where P is the total vertical load on the isolator. Plastic deformations after the yielding point are computed using a Return-Mapping Algorithm as described for hardening models by Simo and Hughes [1998]. The post-yielding stiffness is equal to P/R where R is the radius of curvature of the friction pendulum and P is the total vertical load on the isolator.