SeismoStruct

User Manual

2026

Copyright

Copyright © 2002-2026 Seismosoft Ltd. All rights reserved.

SeismoStruct® is a registered trademark of Seismosoft Ltd. Copyright law protects the software and all associated documentation.

No part of this manual may be reproduced or distributed in any form or by any means, without the prior explicit written authorisation from Seismosoft Ltd.:

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

Every effort has been made to ensure that the information contained in this Manual is accurate. Seismosoft is not responsible for printing or clerical errors.

Finally, mention of third-party products is for informational purposes only and constitutes neither an engagement nor a recommendation.

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 [2026] "SeismoStruct 2026 – A computer program for static and dynamic nonlinear analysis of framed structures," available from https://seismosoft.com/.*

Table of Contents

Introduction 10

General 12

System Requirements 12
Installing/Uninstalling the software 12
Opening the software and Registration options 13
Main menu and Toolbar 14

Quick Start 19

Tutorial n.1 – Pushover Analysis of a Two-Storey Building 19
Tutorial n.2 – Pushover Analysis of a Two-Storey Building 49
Tutorial n.3 – Dynamic Time-history Analysis of a Two-Storey Building 54
Tutorial n.4 – Pushover Analysis of a Two-Storey Building 58
Tutorial n.5 – Eigenvalue Analysis of a Two-Storey Building 95
Tutorial n.6 – Dynamic Time-history Analysis of a Two-Storey Building 101
Building Modeller 107
Basic Settings and Structural Configuration 108
Advanced Settings 108
Building Modeller Main Window 114
Insertion of Structural Members 116
Soil Foundation Parameters 127
Steel Column 133
Steel Beam 135
Ramps 144
Inserting Loads 146
Point Loads 146
Linear Loads 149
Inserting Foundation Members 150
Individual Footings 150
Strip Footings 151
Connecting Beams 152
Editing Structural Members 153
View 3D Model 156
Other Building Modeller Functions 156
Saving and Loading Building Modeller Projects 158
Creating SeismoStruct Projects 158
Wizard 159
Structural model and configuration 160
Settings 161
Loading 161
Exporting and Importing SeismoStruct Projects as XML files 162
Exporting and Importing SeismoStruct Projects as OpenSees files 163

Pre-Processor 164

Analysis Types 164
Pre-Processor area 165
Units Selector 166
Editing 167
Editing functions 167
Graphical Input/Generation 169

Node/Element Groups *169*
3D Plot options *171*
Rotating/moving the 3D model *176*
Project Settings 176
General *178*
Analysis *179*
Elements *181*
Constraints *182*
Adaptive Pushover *184*
Eigenvalue *185*
Constitutive Models *187*
Element Subdivision *188*
Response Spectrum Analysis *189*
Cracked/Uncracked Stiffness *190*
Buckling *191*
Convergence Criteria *192*
Global Iterative Strategy *196*
Element Iterative Strategy *198*
Gravity and Mass *200*
Integration Scheme *202*
Damping *204*
Materials 207
Sections 209
Custom Sections 212
Element Classes 214
Structural Geometry 216
Nodes *217*
Element Connectivity *219*
Constraints *230*
Restraints *235*
Loading 236
Tsunami Parameters *236*
Nodal Loads *244*
Loading Phases *249*
Time-history curves *254*
Adaptive pushover parameters *257*
IDA parameters *261*
RSA parameters *261*
Target Displacement 263
Code-based Checks 265
Performance Criteria 269
Model Statistics 272
Analysis Output 272

Processor 276

Post-Processor 281
Post-Processor settings 282
Plot Options 283
Creating an analysis movie 284
Analysis logs 286
Modal/Mass quantities 286
Target Displacement 288
Step output 289
Deformed shape viewer 290
Convergence Problems 293

Action Effects Diagrams 294
Code-based Checks 296
Global response parameters 297
Performance Criteria Checks 301
Element action effects 302
Stress and strain output 307
IDA envelope 310

SeismoStruct Batch Facility 311
Creating new input files with the SPF Creator 311
SeismoBatch 313
Defining the Working Directory in SeismoBatch 314
Running the Analyses from SeismoBatch 315
Extracting Results from SeismoBatch 316

Bibliography 317

Appendix A - Theoretical background and modelling assumptions 328
Geometric nonlinearity 328
Material inelasticity 328
Global and local axes system 331
Nonlinear solution procedure 332

Appendix B - Analysis Types 341
Eigenvalue Analysis 341
Static Analysis (non-variable loading) 342
Static Pushover Analysis 342
Static Adaptive Pushover Analysis 343
Static Time-History Analysis 343
Dynamic Time-History Analysis 344
Incremental Dynamic Analysis – IDA 344
Response Spectrum Analysis – RSA 344
Buckling Analysis 345
Tsunami Nonlinear Analysis 345

Appendix C - Materials 347
Steel materials 347
Concrete materials 355
Other materials 361

Appendix D - Sections 367
One material sections 367
Reinforced concrete sections 384
Jacketed Reinforced concrete sections 401
Composite sections 417
Masonry sections 421

Appendix E – Building Modeller Members 423

Appendix F - Element Classes 486
Beam-Column element types 486
Link element types 514
Mass and Damping element types 534

Appendix G - Response Curves 539

Appendix H – Codes 571

Appendix H1 - EUROCODES 571

Performance Requirements 571
Limit State of Near Collapse (NC) *571*
Limit State of Significant Damage (SD) *571*
Limit State of Damage Limitation (DL) *571*
Information for Structural Assessment 571
KL1: Limited Knowledge *572*
KL2: Normal Knowledge *572*
KL3: Full Knowledge *572*
Confidence Factors *573*
Safety Factors *574*
Capacity Models for Assessment and Checks 574
Deformation Capacity *574*
Bending Moment Capacity *577*
Shear Capacity *577*
Masonry Elements *578*
Capacity Curve *579*
Target Displacement 579

Appendix H2 - ASCE 582
Performance Requirements 582
Performance Level of Operational Level (1-A) *582*
Performance Level of Immediate Occupancy (1-B) *582*
Performance Level of Life Safety (3-C) *582*
Performance Level of Collapse Prevention (5-D) *583*
Information for Structural Assessment 583
Minimum Knowledge *583*
Usual Knowledge *583*
Comprehensive Knowledge *583*
Safety Factors *584*
Capacity Models for Assessment and Checks 584
Deformation Capacity *584*
Bending Moment Capacity *585*
Shear Capacity *585*
Masonry Elements *586*
Capacity Curve 590
Target Displacement 590

Appendix H3 - NTC-18 593
Performance Requirements 593
Limit State of Collapse Prevention (SLC) *593*
Limit State of Life Safety (SLV) *593*
Limit State of Damage Limitation (SLD) *593*
Limit State of Operational Level (SLO) *593*
Information for Structural Assessment 593
KL1: Limited Knowledge *594*
KL2: Adequate Knowledge *594*
KL3: Accurate Knowledge *594*
Confidence Factors *595*
Safety Factors *596*
Capacity Models for Assessment and Checks 596
Deformation Capacity *596*
Bending Moment Capacity *598*
Shear Capacity *598*
Masonry Elements *599*
Capacity Curve 601
Target Displacement 601

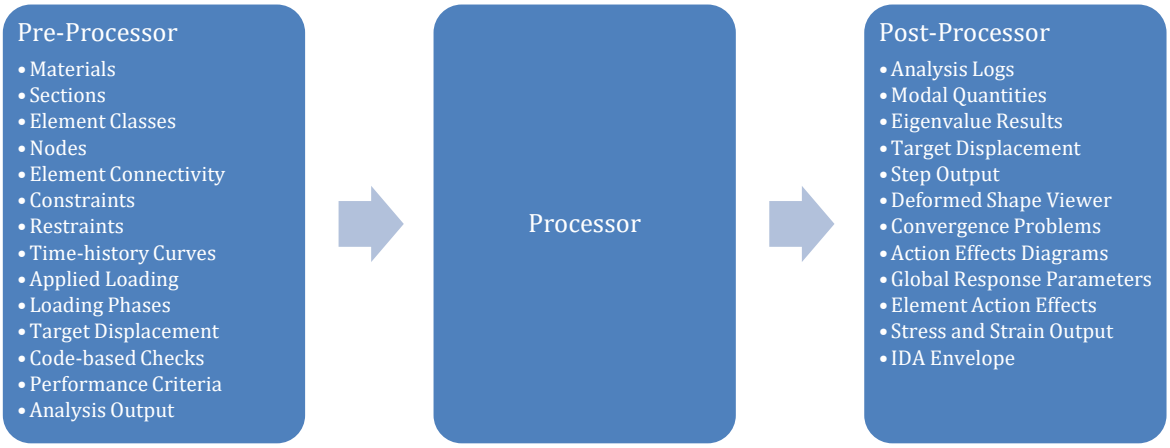
Appendix H4 - KANEPE 603
Performance Requirements 603
Performance Level of Immediate Occupancy (A) 603
Performance Level of Life Safety (B) 603
Performance Level of Collapse Prevention (C) 603
Information for Structural Assessment 604
Tolerable DRL 604
Sufficient DRL 604
High DRL 604
Safety Factors 605
Capacity Models for Assessment and Checks 605
Deformation Capacity 605
Bending Moment Capacity 608
Shear Capacity 608
Masonry Elements 610
Capacity Curve 610
Target Displacement 610

Appendix H5 - TBDY 613
Performance Requirements 613
Performance Level of Continuous Use (KK) 613
Performance Level of Immediate Occupancy (HK) 613
Performance Level of Life Safety (CG) 613
Performance Level of Collapse Prevention (BP) 613
Information for Structural Assessment 614
Limited Knowledge 614
Comprehensive Knowledge 614
Knowledge Factors 614
Safety Factors 614
Capacity Models for Assessment and Checks 615
Deformation Capacity 615
Bending Moment Capacity 616
Shear Capacity 616
Masonry Elements 617
Capacity Curve 617
Target Displacement 617

Introduction

SeismoStruct is a Finite Element package for structural analysis, capable of predicting the large displacement behaviour of space frames under static or dynamic loadings, taking into account both *geometric nonlinearities* and *material inelasticity*.

The software consists of three main modules: a **Pre-Processor**, in which it is possible to define the input data of the structural model, a **Processor**, in which the analysis is carried out, and finally a **Post-Processor** to output the results; all is handled through a **completely visual interface**. No input or configuration files, programming scripts or any other time-consuming and complex text editing are required. The Processor, moreover, 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 offers advanced post-processing facilities, including the ability to custom-format all derived plots and deformed shapes, thus increasing 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. Input data created in spreadsheet programs, such as Microsoft Excel, may be pasted to the SeismoStruct input tables, for easier pre-processing. Conversely, all information visible within the graphical interface of SeismoStruct can be copied to external software applications (e.g. to word processing programs, such as Microsoft Word), including input and output data, high quality graphs, the models' deformed and undeformed shapes and much more.

Finally, with the *Building Modeller* and *Wizard* facility the user can create regular/irregular 2D or 3D models and run all types of analyses on the fly. The whole process takes no more than a few seconds.

Some of the modelling/analysis features of SeismoStruct are listed below:

- Ten different types of analysis, such as dynamic and static time-history, conventional and adaptive pushover, incremental dynamic analysis, eigenvalue, non-variable static loading, response spectrum analysis, buckling analysis and tsunami analysis.
- Twenty material models, such as nonlinear concrete models, high-strength nonlinear concrete model, nonlinear steel models, SMA nonlinear model, etc.
- A large library of 3D elements, such as nonlinear fibre beam-column element, nonlinear truss element, nonlinear infill panel element, nonlinear masonry elements, nonlinear link elements, etc., that may be used with a wide variety of pre-defined steel, concrete and composite section configurations.

- Thirty one hysteretic models, such as linear/bilinear/trilinear kinematic hardening response models, gap-hook models, soil-structure interaction model, Takeda model, Ramberg-Osgood model, etc.
- *Code-based Checks* for elements' Chord Rotation, Shear Capacity and bending moment and for masonry. Six Codes are currently supported, 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).
- Several *Performance Criteria* that allow the user to identify the instants at which different performance limit states (e.g. non-structural damage, structural damage, collapse) are reached. The sequence of cracking, yielding, failure of members throughout the structure can also be, in this manner readily obtained.
- Two different solvers: Skyline solver (Cholesky decomposition, Cuthill-McKee nodes ordering algorithm, Skyline storage format) and the Frontal solver for sparse systems, introduced by Irons [1970] featuring the automatic ordering algorithm proposed by Izzuddin [1991].
- The applied loads may consist of constant or variable forces, displacements and accelerations at the nodes and at the elements. The variable loads can vary proportionally or independently in the pseudo-time or time domain.
- The spread of inelasticity along the member length and across the section depth is explicitly modelled in SeismoStruct allowing for accurate estimation of damage accumulation.
- Numerical stability and accuracy at very high strain levels enabling precise determination of the collapse load of structures.
- SeismoStruct possesses the ability to smartly subdivide the loading increment, whenever convergence problems arise. The level of subdivision depends on the convergence difficulties encountered. When convergence difficulties are overcome, the program automatically increases the loading increment back to its original value.

General

System Requirements

To use **SeismoStruct**, we suggest:

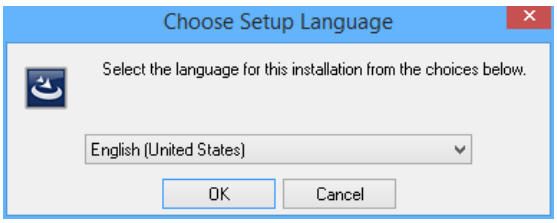
- A PC (or a "virtual machine") with one of the following operating systems: Windows 7 and later versions of Windows (64-bit);
- 8 GB RAM;
- Screen resolution on your computer set to 1366x768 or higher;
- An Internet connection (better if a broadband connection) for the registration of the software.

Installing/Uninstalling the software

Installing the software

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

1. Download the latest version of the program from: https://seismosoft.com/products/
2. Save the application on your computer and launch it. First, you will be asked to select the installation language:



Selection of setup language

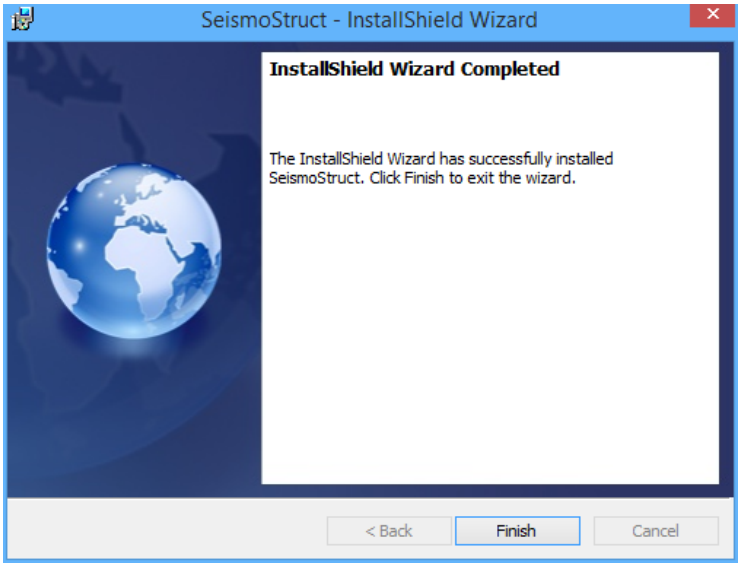
3. After choosing the preferred language from the drop-down menu, click the *OK* button.



Installation wizard (first window)

4. 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.

5. 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.
6. Click the *Install* button and wait until the software is installed.
7. At the end of the procedure, click *Finish* to exit the wizard.



Installation wizard (last window)

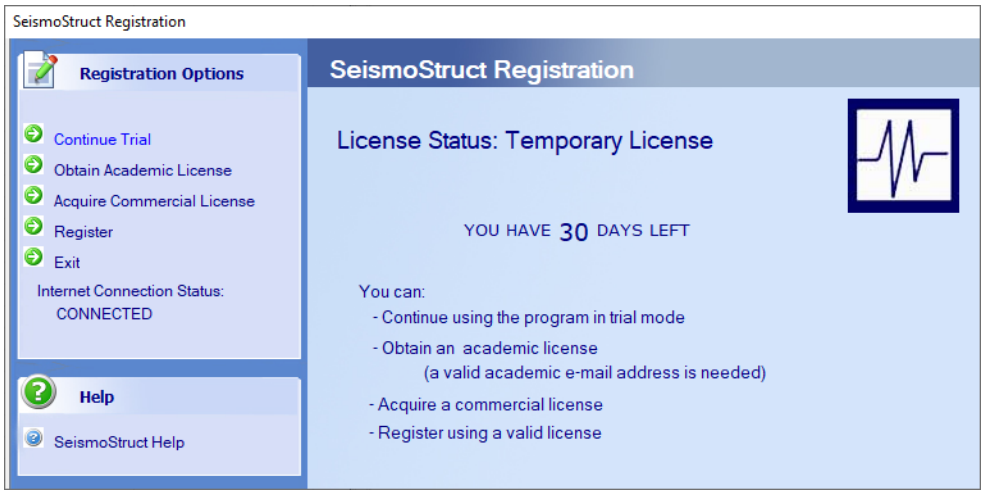
Uninstalling the software

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 SeismoStruct, select *Start > Programs or All Programs > Seismosoft > SeismoStruct 2026 > SeismoStruct 2026*. The following registration's window will appear:



SeismoStruct 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.



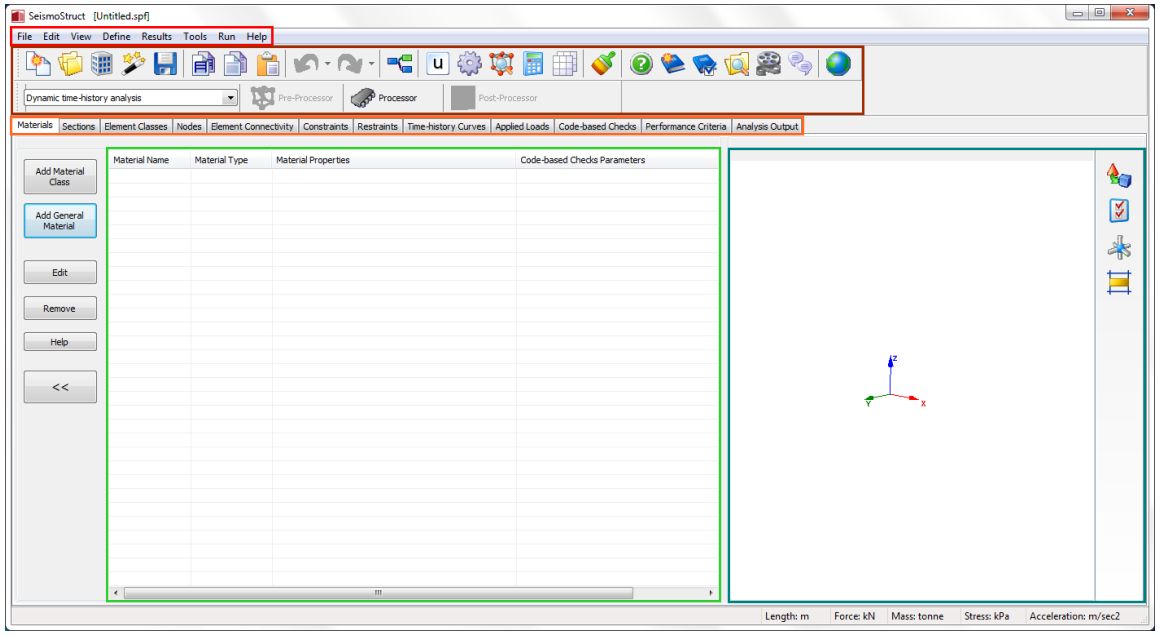
Registration Form

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 2022 and older are not valid in SeismoStruct 2025. Users are thus invited to request a new license.

MAIN MENU AND TOOLBAR

SeismoStruct has a simple and 'easy to understand' user interface. The **main window** of its ***Pre-Processor area***, which is the 'default' program state, is subdivided into the following components:

- **Main menu and toolbar**: at the top of the program window;
- **Modules bar**: below the Main toolbar;
- **Input table**: below the Modules bar;
- **3D Model window and settings bar**: on the right of the program window;
- **Editing bar**: on the left of the program window.



Pre-Processor Area

NOTE: The main menu and toolbar are available in each program state (i.e. Pre-Processor, Processor and Post-Processor). Only the items useful in the current program state (e.g. Pre-Processor) will be selectable; the other ones will be greyed out. Furthermore, additional components will appear depending on the module selected.

Main menu

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

- File
- Edit
- View
- Define
- Results
- Tools
- Run
- 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 SeismoStruct is shown below:

Command	Main menu	Shortcut keys	Toolbar button
File	New	Ctrl+N	
	Open	Ctrl+O	
	Wizard		
	Building Modeller	-	
	Save	Ctrl+S	
	Save as...	-	
	Export to XML File...		
	Import from XML File...		
	Import from OpenSees...		
	Export to OpenSees...		
	Show SPF file structure		
Edit	Undo	Ctrl+Z	
	Redo	Ctrl+R	
	Add to Group		
	Organize Groups	-	
	Sort By Name		
	Sort By Number		
	Copy Selection	Ctrl+C	
	Copy 3D Plot	Ctrl+Alt+C	
	Paste Selection	Ctrl+V	
	Find...	Ctrl+F	
	Find Next	F3	
	Select All	Ctrl+A	
View	Next Properties Module	Ctrl+W	
	Previous Properties Module	Ctrl+Q	
	Model Statistics		
	View Large Icons		
	View Small Icons		
Define	Material properties		
	Section properties		
	Element Classes		
	Structural Nodes		
	Element Connectivity		
	Nodal Constraints		
	Restraints		
	Linear Curves		
	Applied Loading		

Command	Main menu	Shortcut keys	Toolbar button
	Phases		
	Target Displacement		
	Adaptive Parameters		
	Response Spectrum Parameters		
	Capacity Checks		
	Performance Criteria		
	Output		
Results	Analysis Logs		
	Modal Quantities		
	Step Output		
	Deformed Shapes		
	Frame Element Forces		
	Global Response Parameters		
	Member Action Effects		
	Stress and Strain Output		
	IDA Envelope		
Tools	Units Selector	Ctrl+U	
	Create AVI File...		
	Show AVI File...		
	Redraw 3D Plot	-	
	Project Settings.../Post-Processor Settings...	-	
	3D Plot Options	-	
	Deformed Shape Settings	-	
	Calculator	-	
	Open SPF Creator		
	Open SeismoBatch		
Run	Pre-Processor		
	Processor		
	Post-Processor		
Help	SeismoStruct Help	F1	
	Rotate/move the 3D model	-	
	SeismoStruct User Manual		
	SeismoStruct Verification Report		
	Verification Examples		
	Download Sample Files		
	Seismosoft Forum		
	Video Tutorials		
	Send Message to Seismosoft	-	

Command	Main menu	Shortcut keys	Toolbar button
	Seismosoft Website	-	
	Register New License	-	
	About...	-	

Quick Start

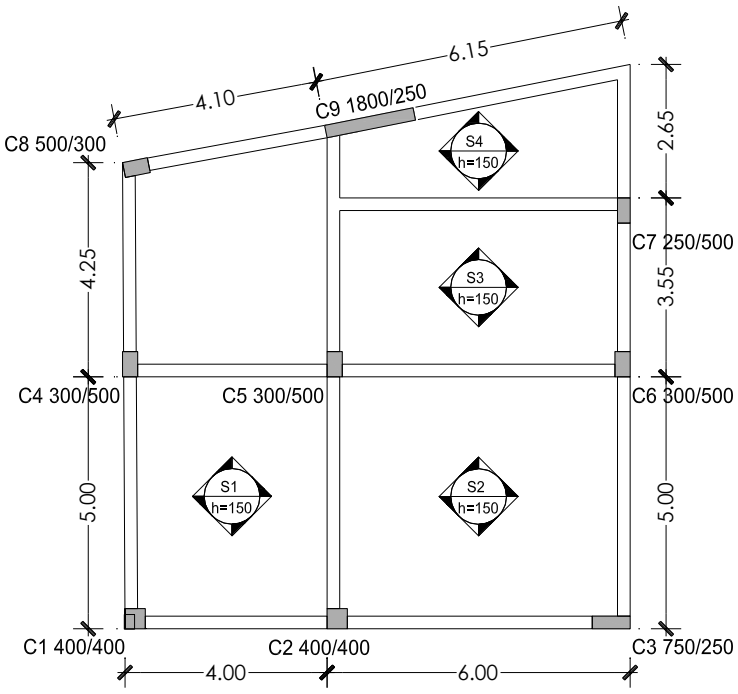
This chapter will walk you through your **first analyses with SeismoStruct**.

SeismoStruct has been designed with both ease-of-use and flexibility in mind. Our goal is to get you run analysis (even the 'troublesome' dynamic time-history analysis) in just some minutes. It is actually much easier to use SeismoStruct than it is to describe. You will see that once you have grasped a few important concepts, the entire process is quite intuitive.

TUTORIAL N.1 – PUSHOVER ANALYSIS OF A TWO-STOREY BUILDING

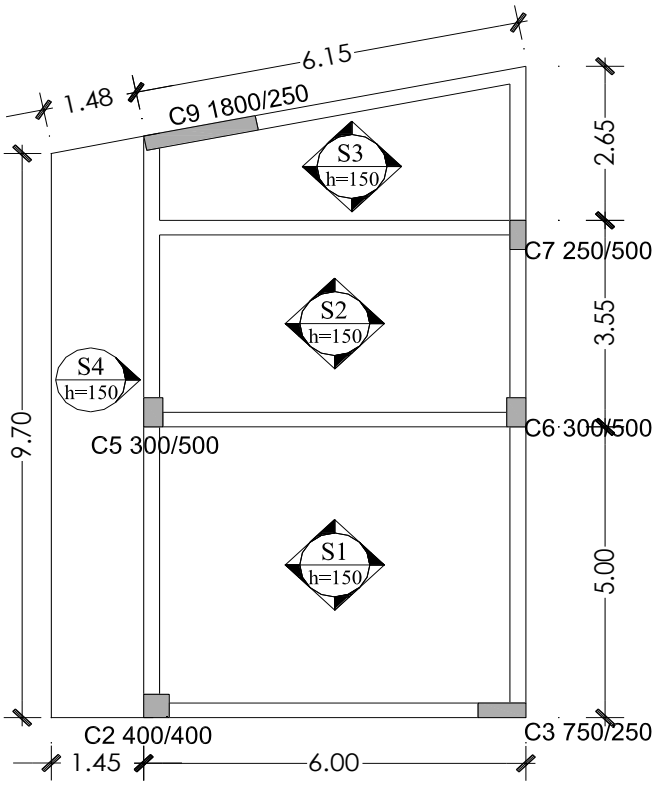
Problem Description

Let us try to model a three dimensional, two-storey reinforced concrete building for which you are asked to run a static pushover analysis (in the X direction). The Building Modeller will be used for 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 website.

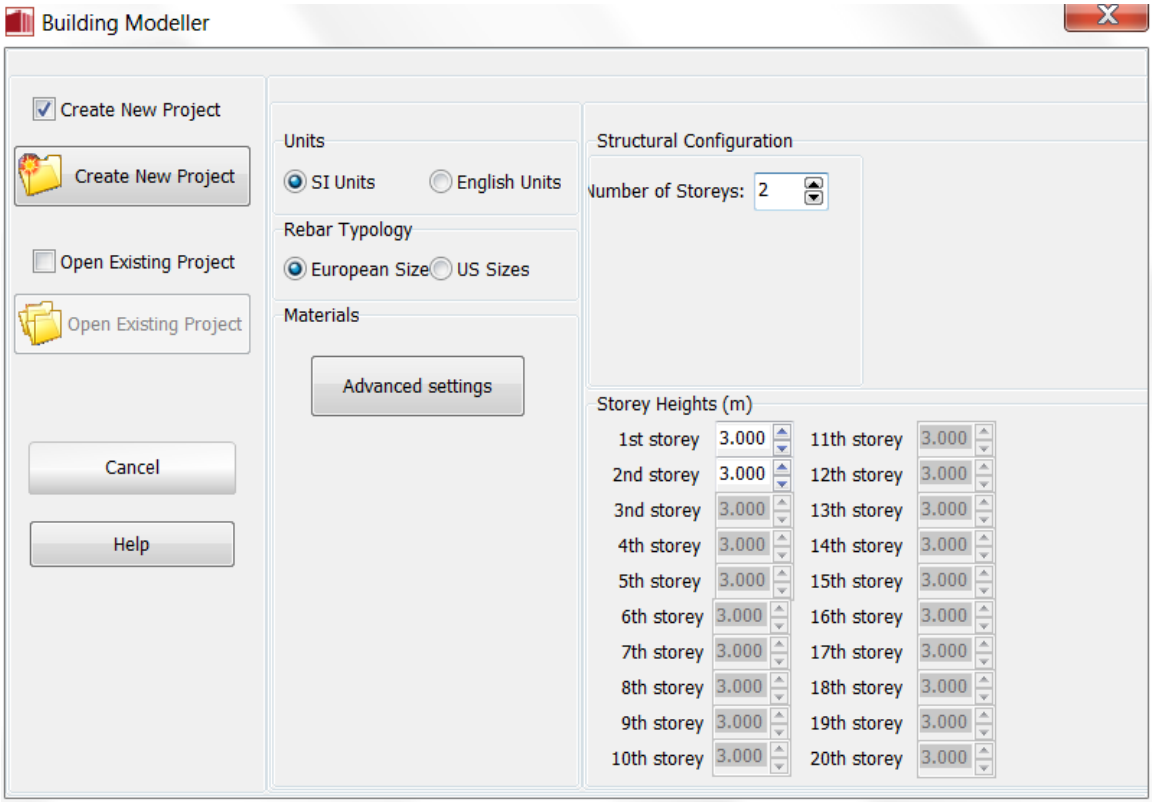


Plan view of 2nd floor of the building

Getting started: a new project

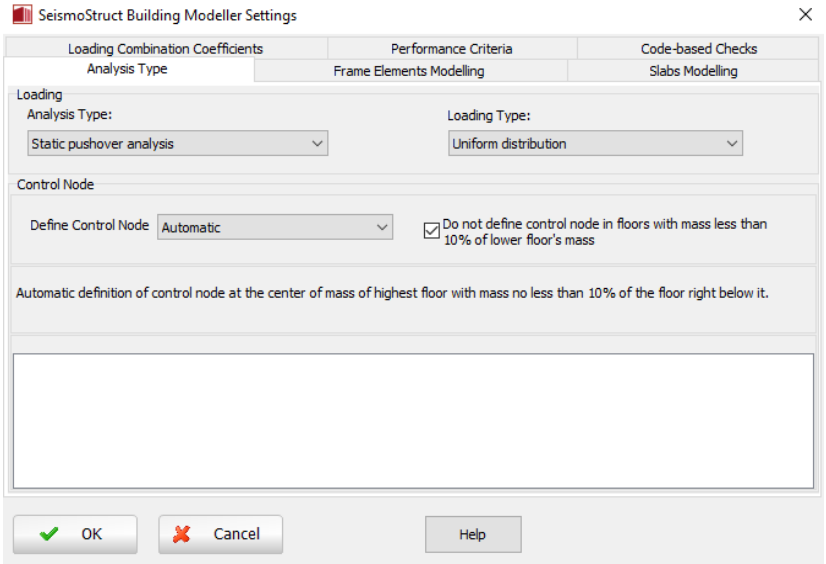
By selecting the *File > Building Modeller...* menu command or clicking on the icon on the toolbar, the Building Modeller initialisation window opens, from which the units, the number of storeys, and the storeys' heights may be selected. Proceed by clicking on the *Create New Building Project button.* For this tutorial the following settings have been chosen:

- SI Units
- European sizes for rebar typology
- 2 Storeys
- Storeys' heights: 3m



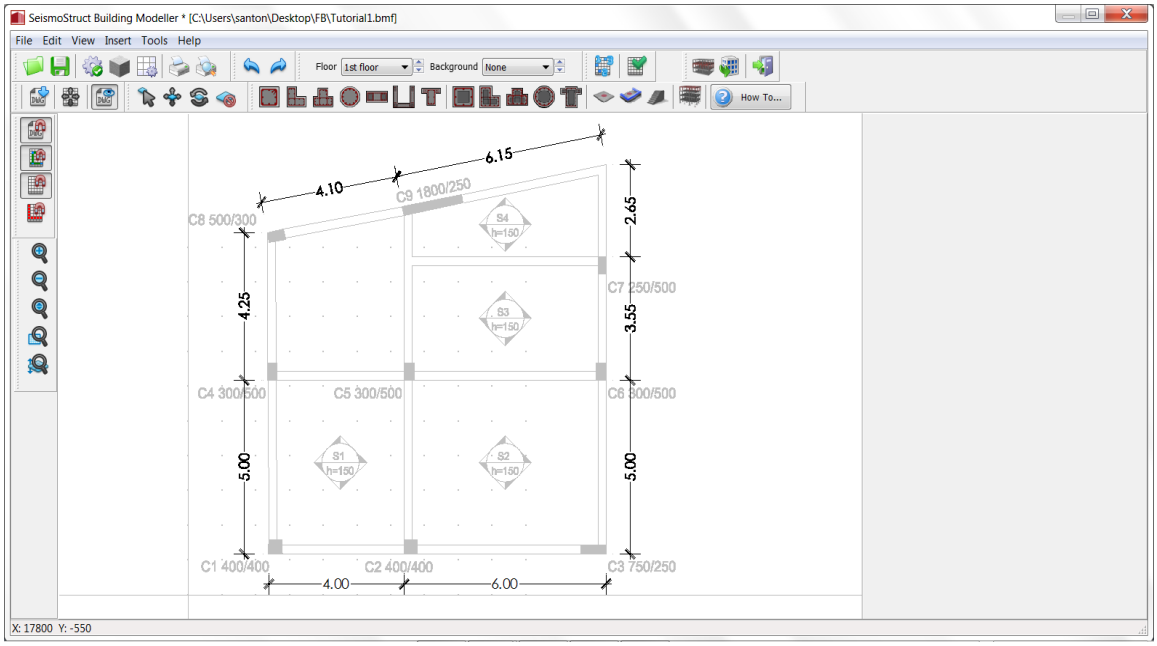
Building Modeller Initialisation module

Click on the *Building Modeller Settings* button and define the Analysis Type (For this tutorial: Static Pushover analysis), the Frame Elements Modelling (Inelastic plastic-hinge force-based frame element for columns/beams, Inelastic force-based frame element for walls and Inelastic displacement-based frame elements for members with length smaller than 1m), the Slabs Modelling (choose to include beam effective widths), the Structural Configuration, the Loading Combination Coefficients and the Performance Criteria checks to be included in the analysis.



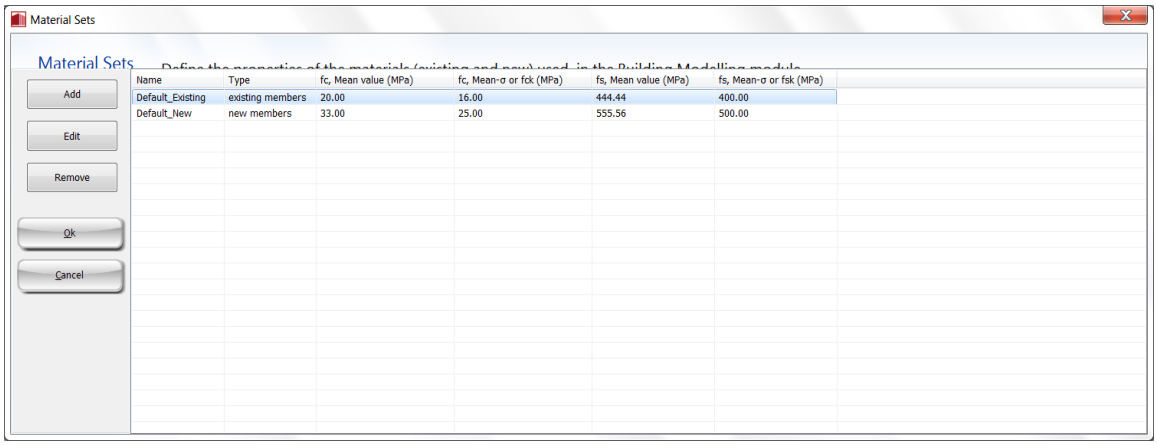
Building Modeller Settings module

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 .

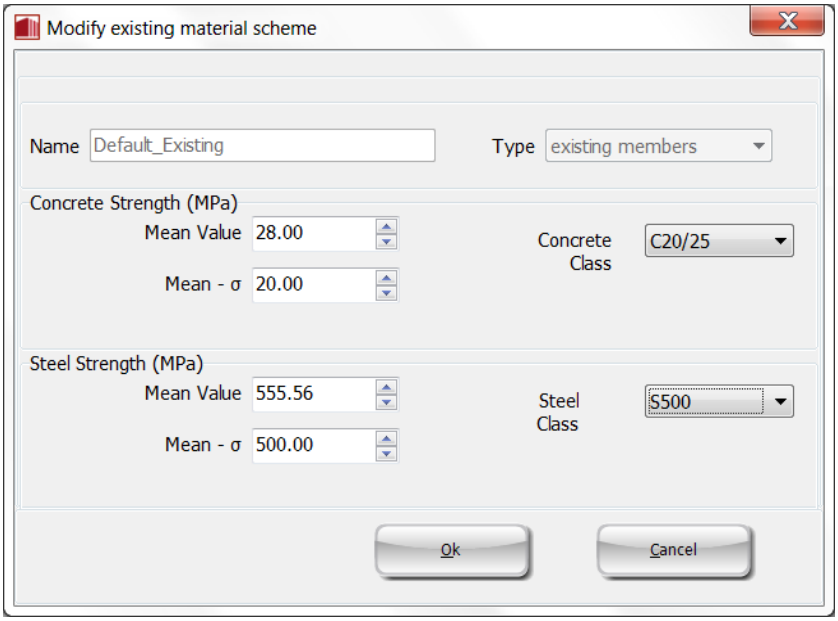


Building Modeller – CAD drawing insertion

Begin inserting the structural members from the main menu (*Insert > Rectangular Column...*) or through the corresponding 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. For this tutorial select to modify the *Default_Existing* material set and assign the C20/25 concrete class and the S500 steel class.

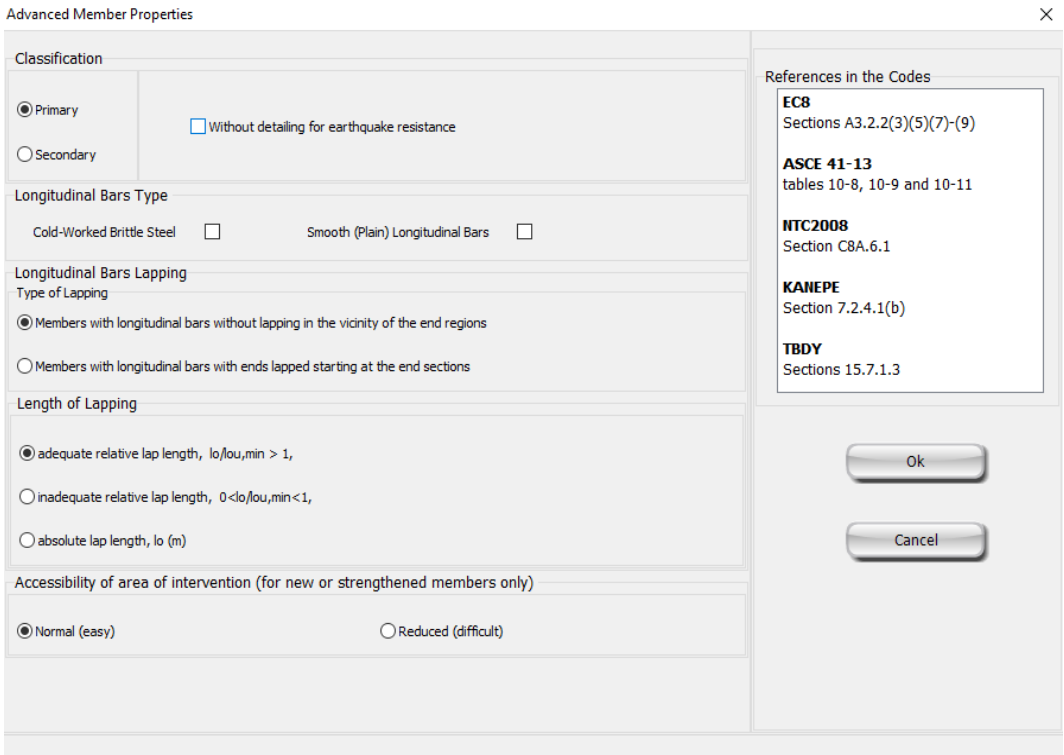


Building Modeller – Material Sets



Building Modeller – Modify Existing Material Scheme

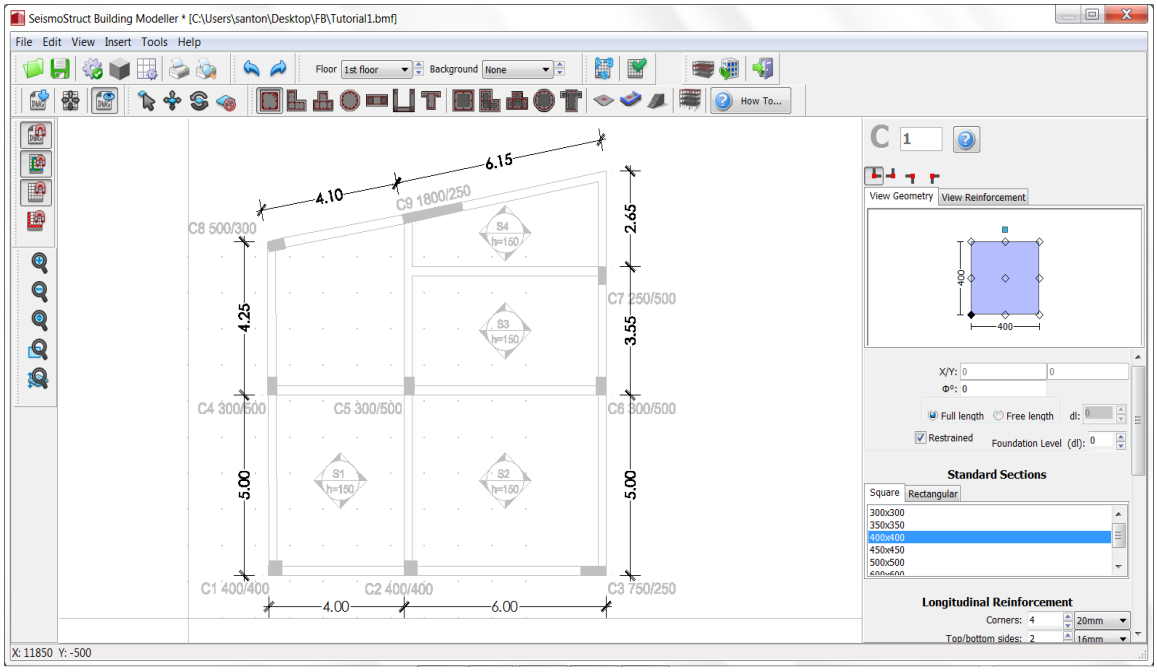
By clicking on the Advanced Member Properties button users may define the settings of the structural member according to the selected Code.



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 0°, 90°, 180° and 270° buttons or by assigning the proper angle on the corresponding of 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:

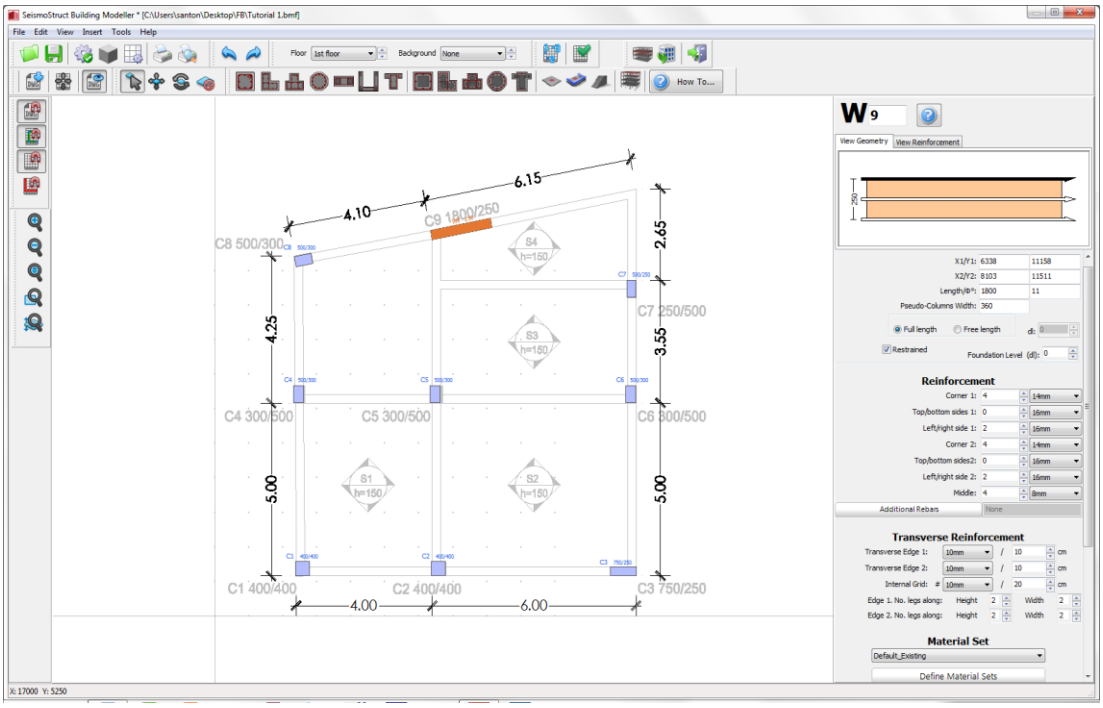
Columns of 1st floor	Height (mm)	Width (mm)	Longitudinal reinforcement	Transverse reinforcement
C1	400	400	4∅18+4∅16	∅10/10
C2	400	400	4∅18+4∅16	∅10/10
C3	750	250	4∅16+8∅14	∅10/10
C4	300	500	6∅18	∅10/10
C5	300	500	6∅18	∅10/10
C6	300	500	6∅20	∅10/10
C7	250	500	4∅20+2∅16	∅10/10
C8	300	500	6∅18	∅10/10
C9	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 u4∅14	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	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	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
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
C8	250	1800	(4∅16+8∅14)+#∅10/20+(4∅8/m^2)	∅10/10

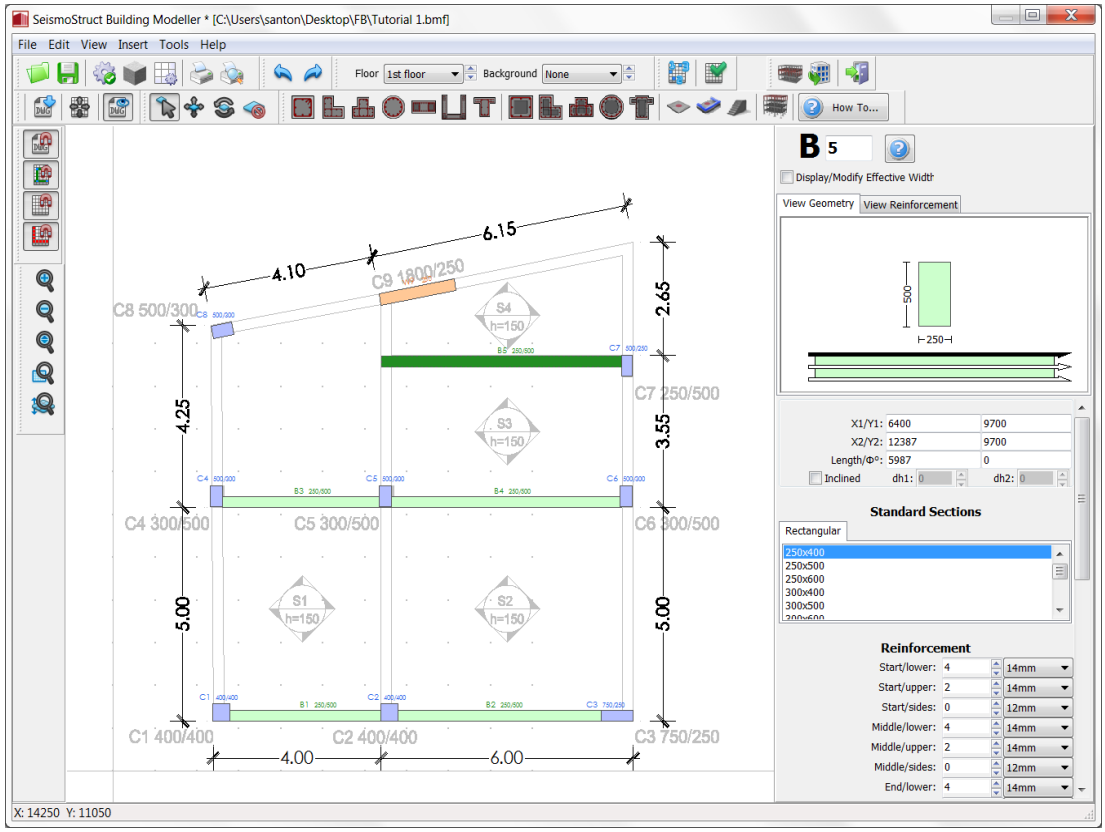
Beams of 2nd Floor	Height (mm)	Width (mm)	Reinforcement at the Start of the beam	Reinforcement at the Middle of the beam	Reinforcement at the End of the beam	Transverse reinforcement
B1	500	250	o4∅16 u4∅14	o2∅14 u4∅14	o4∅16 u4∅14	∅8/10
B2	500	250	o2∅18 u4∅14	o2∅14 u4∅14	o2∅18 u4∅14	∅8/10
B3	500	250	o2∅14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

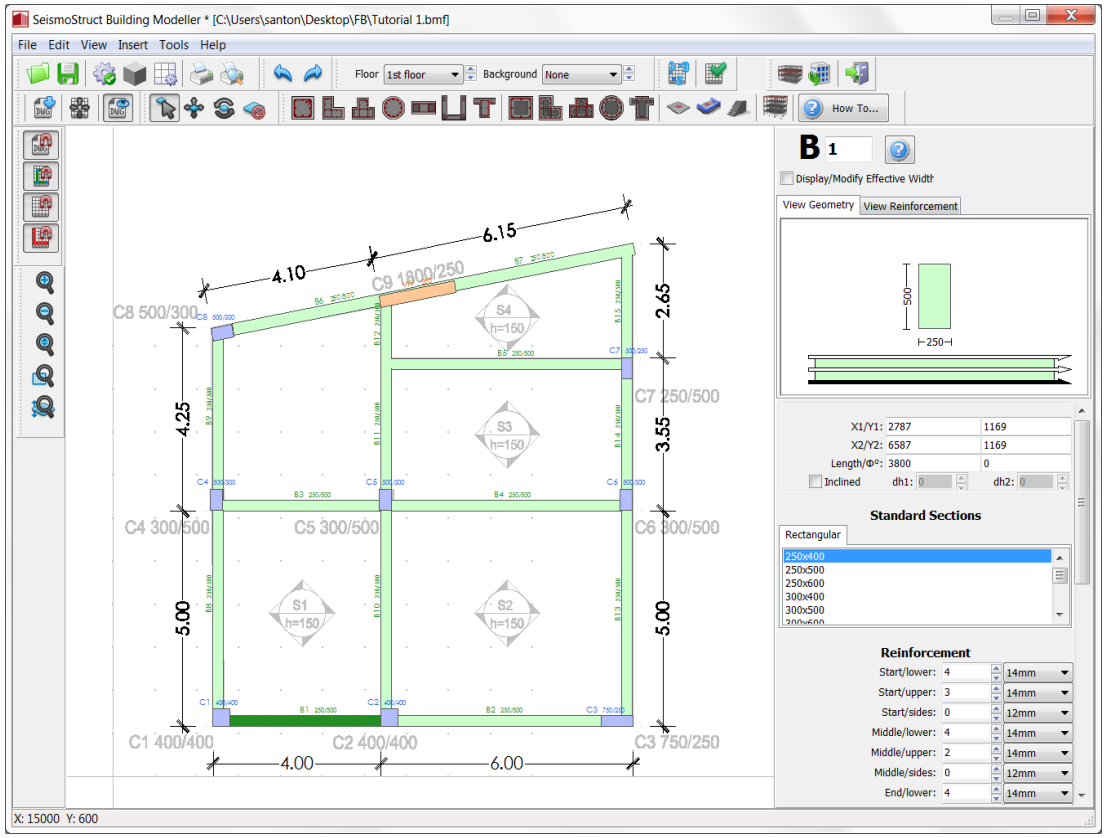


Building Modeller – Wall Element Properties

Insert the beams from the main menu (*Insert* > *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 or the live loads of the slabs (e.g. finishings, infills, etc).

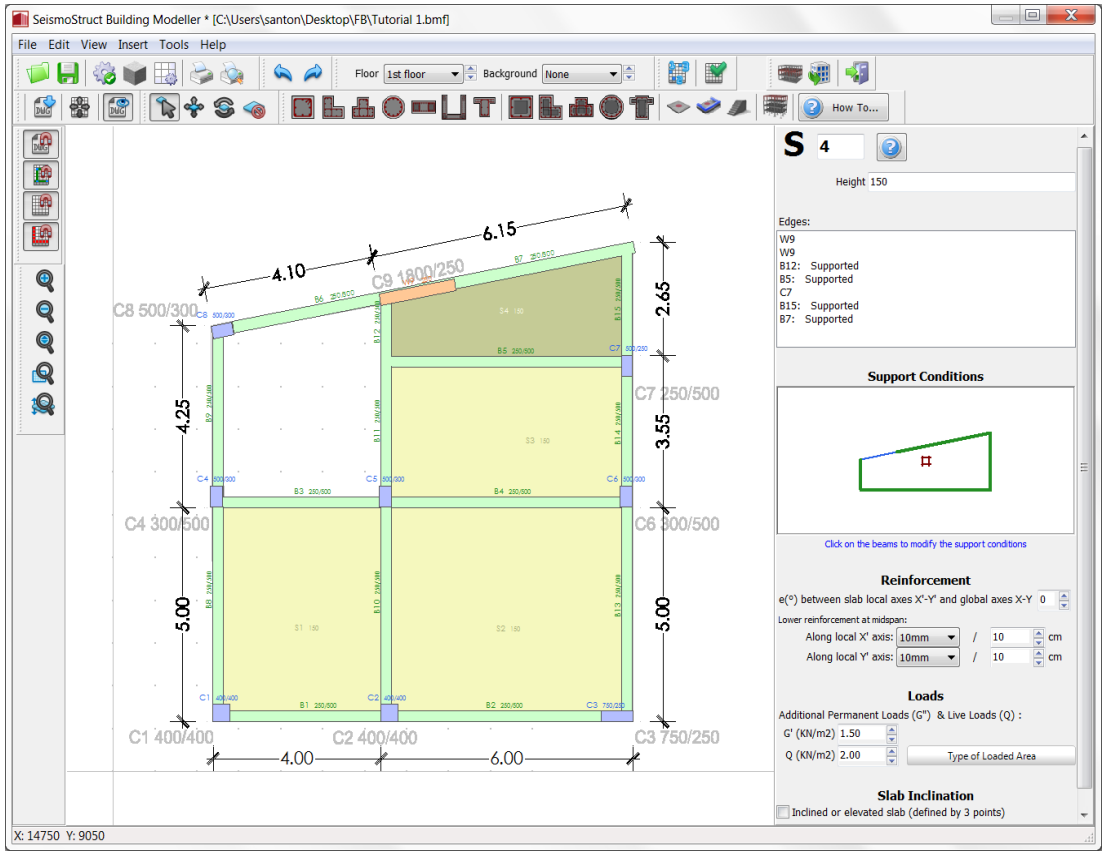


Building Modeller – Insert Elements

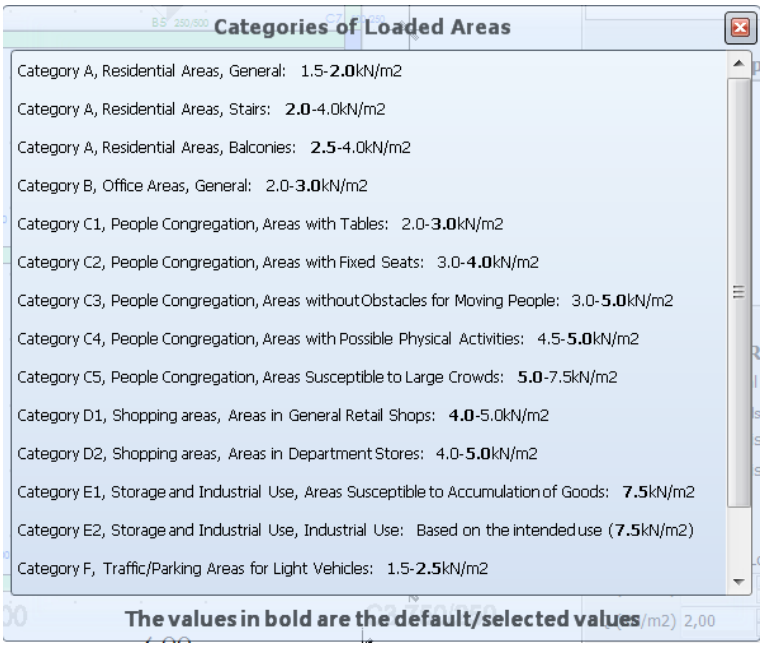


Building Modeller – Beam Element Properties

In order to insert the slabs go to the main menu (*Insert* > *Slab*) or click the toolbar button, assign the slab's properties, which are the section's height, the reinforcement, as well as the 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, which determine the beams where the slab loads are to be distributed, may be modified by just clicking on the corresponding boundaries 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.

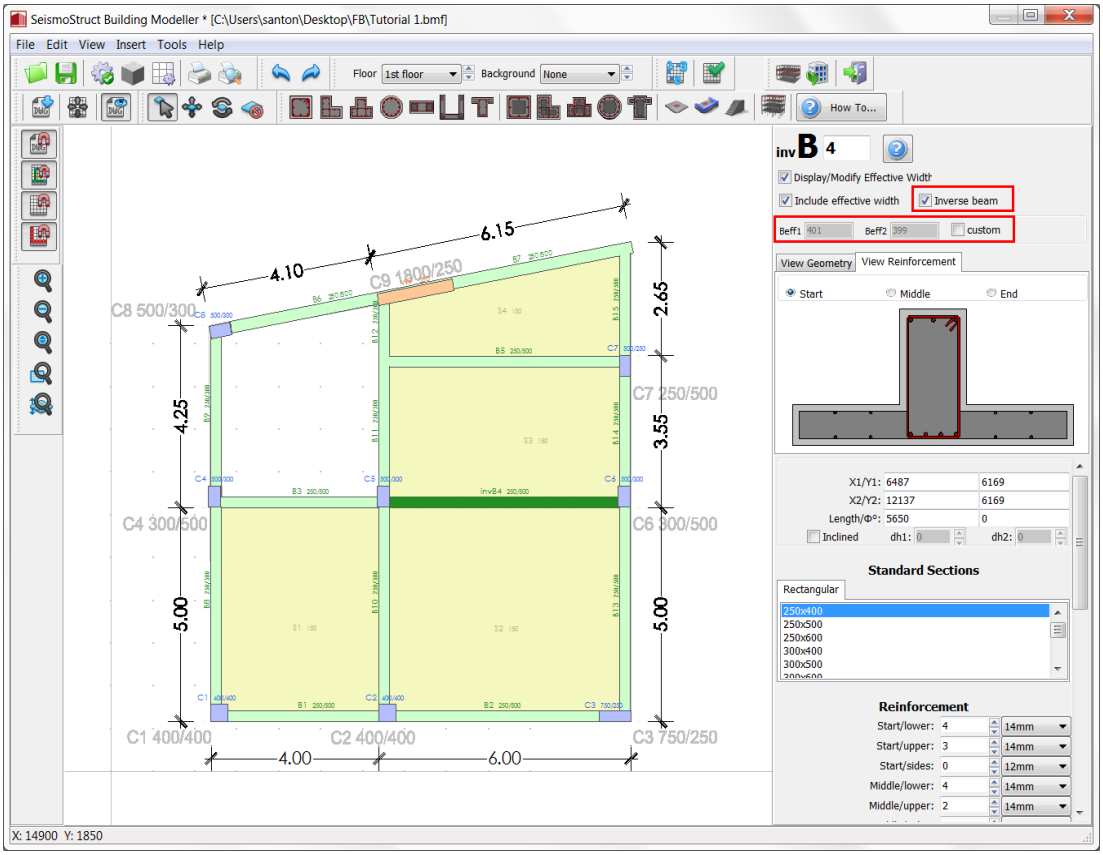


Building Modeller – Slab Element Properties



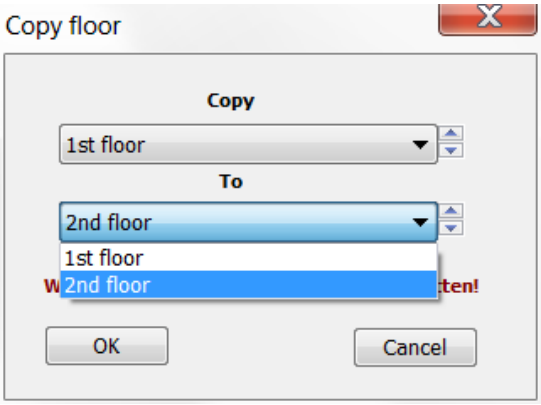
Building Modeller – Categories of Loaded Areas for Slabs

After inserting all the elements you can change the properties of any section 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:



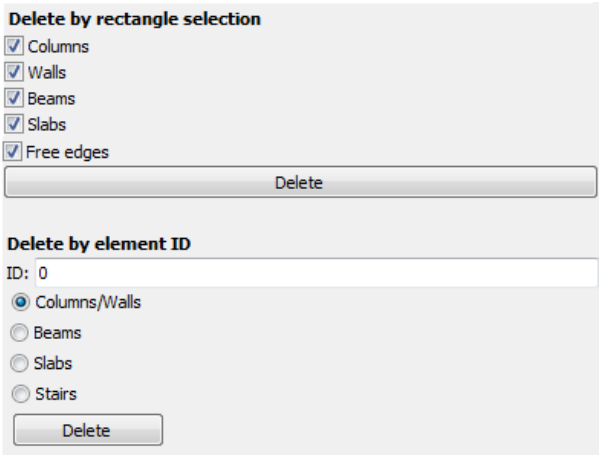
Building Modeller – Beam Element Properties

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



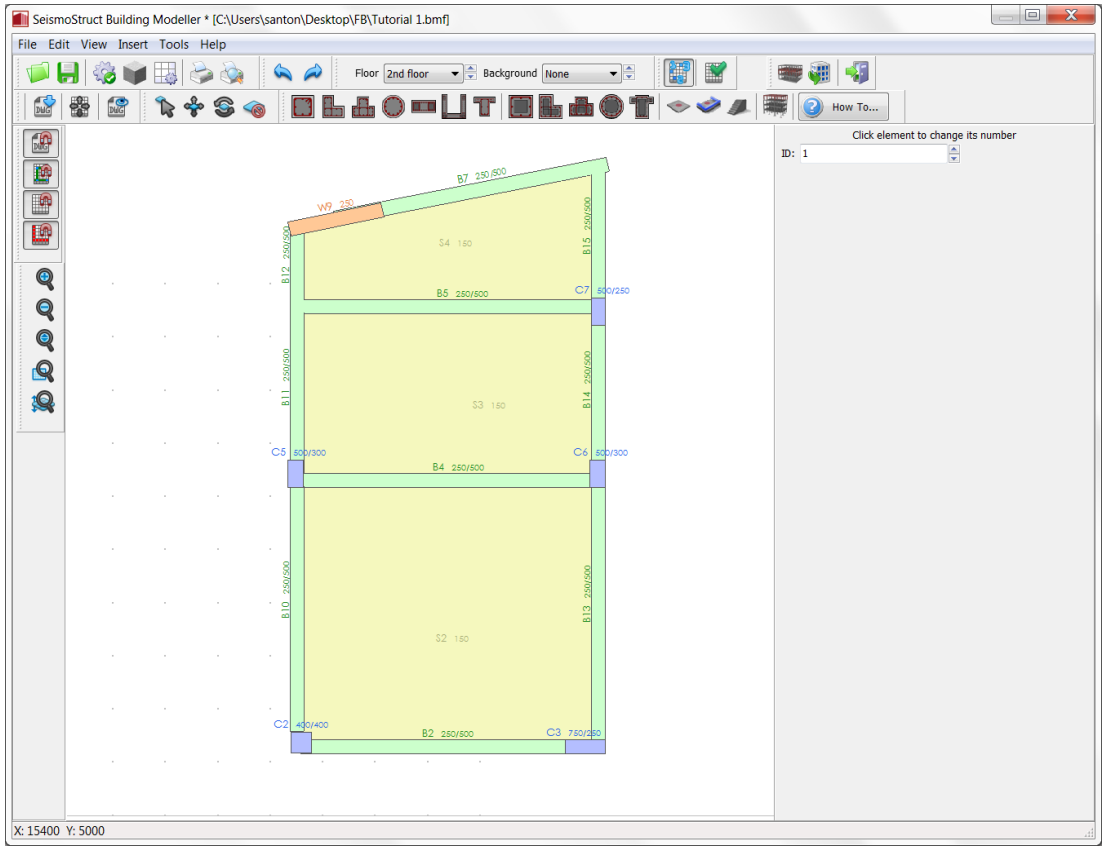
Building Modeller – Copy floor dialogue box

Delete the elements that do not exist 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.



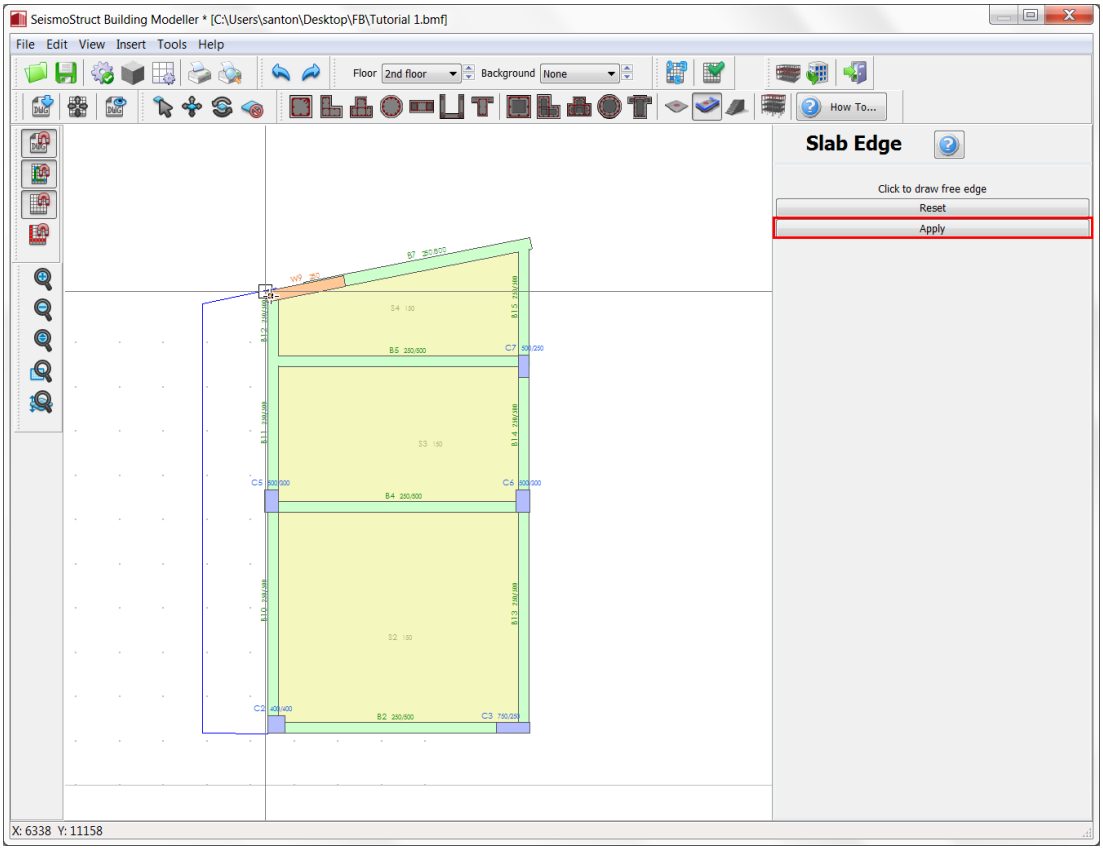
Building Modeller – Delete element dialogue box

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



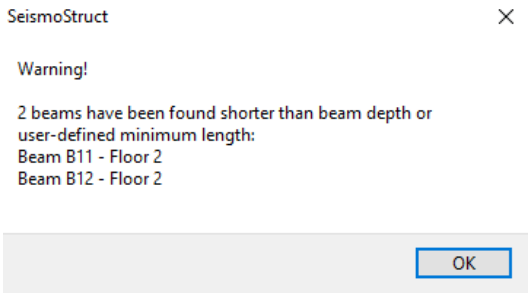
Building Modeller – Renumber elements

Cantilever 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 > 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.



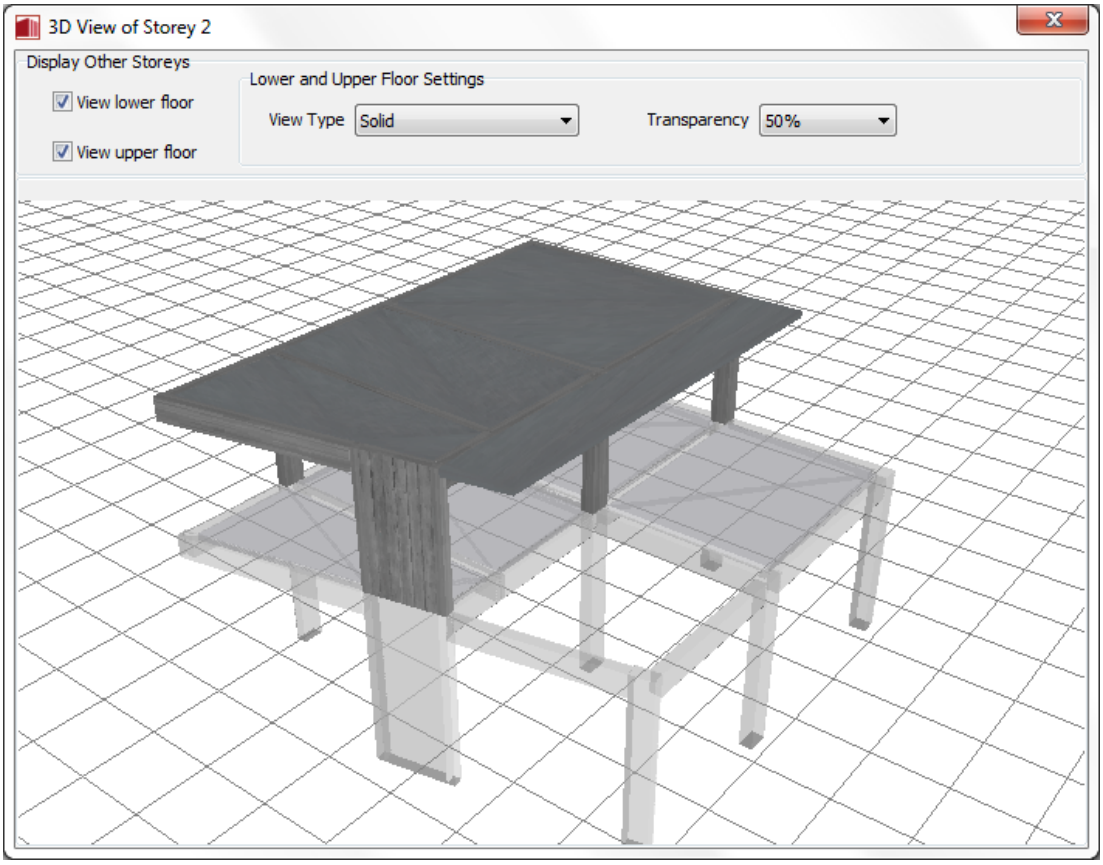
Building Modeller – Add Free Edge

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 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.



Building Modeller – Verify connectivity

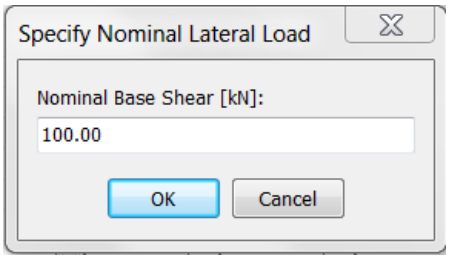
You may also view the 3D model of the current floor to check for its correct definition through the toolbar button.



Building Modeller – View Storey 3D Module

With the building model now fully defined, save the Building Modeller project as a Building Modeller file (with the *.bmf extension, e.g. Tutorial_1.bmf) from the main menu (*File >Save As...*)/ (*File >Save*) or through the corresponding toolbar button. It is noted that this file type is not a SeismoStruct project file (*.spf), hence it can be opened again only from within the Building Modeller.

You are ready to create the new SeismoStruct project. This can be done from the main menu (*File > Exit & Create Project*) or through the corresponding toolbar button. Depending on the analysis type, a new window may appear for the definition of structure's loading; in our case the nominal Base Shear for the pushover analysis should be specified:



Specify Nominal Lateral Load

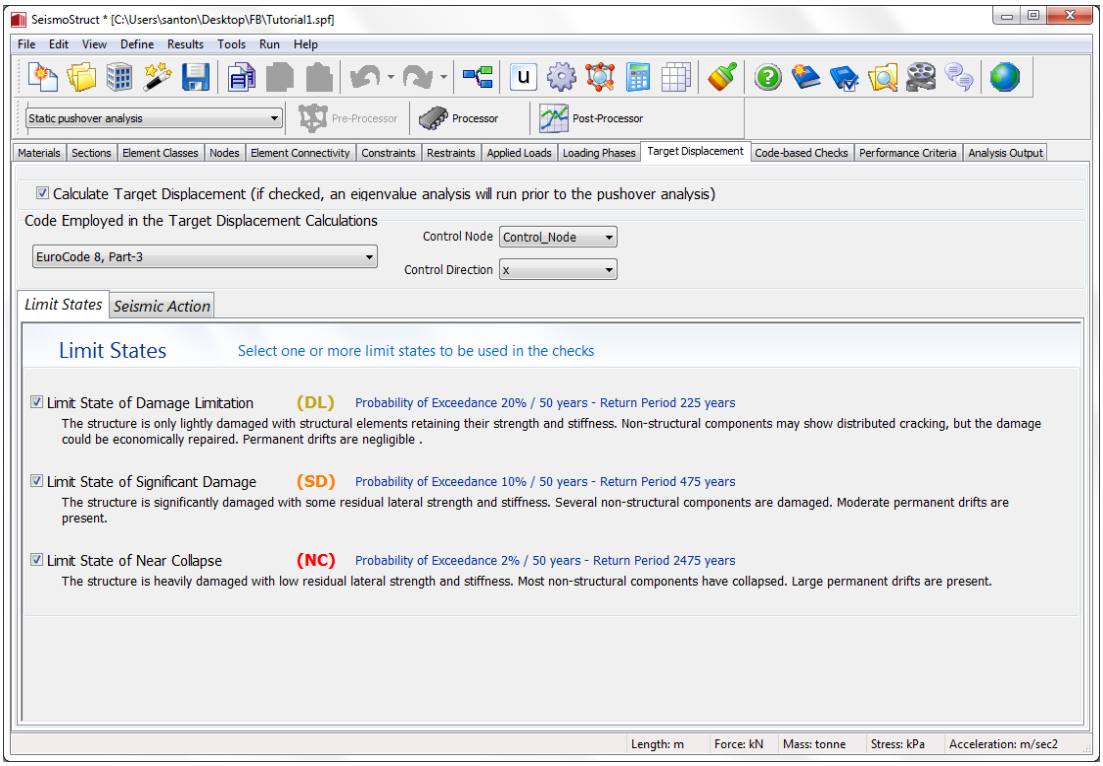
Automatically, the program distributes the lateral loads to the structural nodes according to their concentrated masses.

Pre-processor – Loading Phases

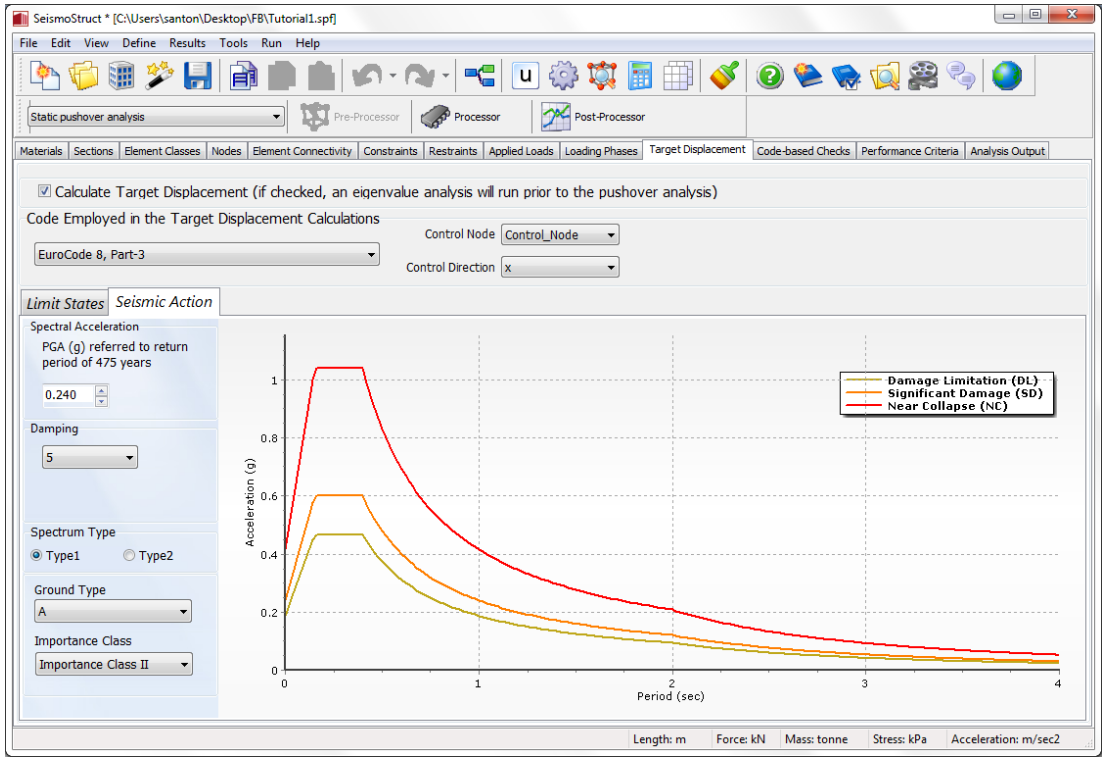
The loading strategy adopted in the pushover analysis is fully defined in the **Loading Phases** module. By default the program define a *Response Control* phase type, a target displacement that corresponds to a 2% total drift ratio, 50 steps of analysis, X direction of loading, and the node in the last floor with the highest applied load value is defined as the controlled node.

Pre-processor – Target Displacement

The employed Code, the control node and direction, the selected Limit Stated or Performance Objectives and the Seismic Action may be easily defined in this module. The default options will be used in this tutorial.



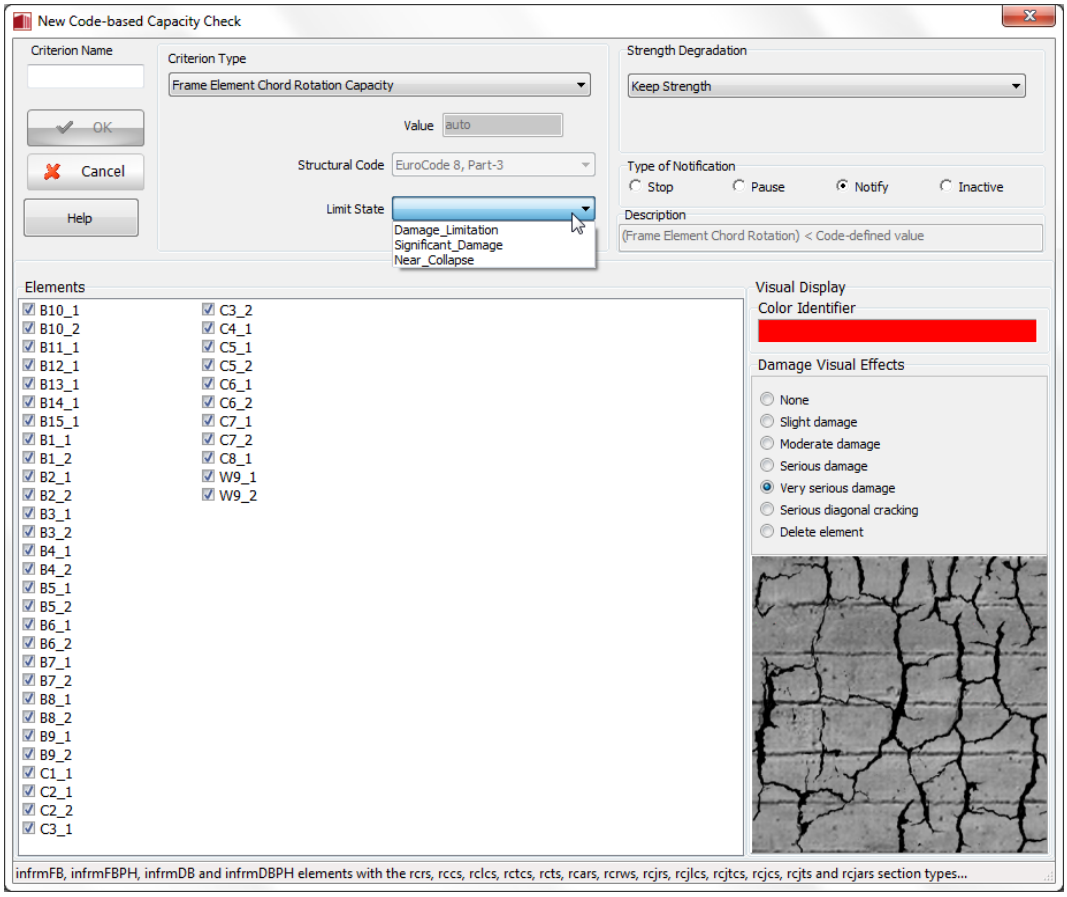
Target Displacement module - Limit States



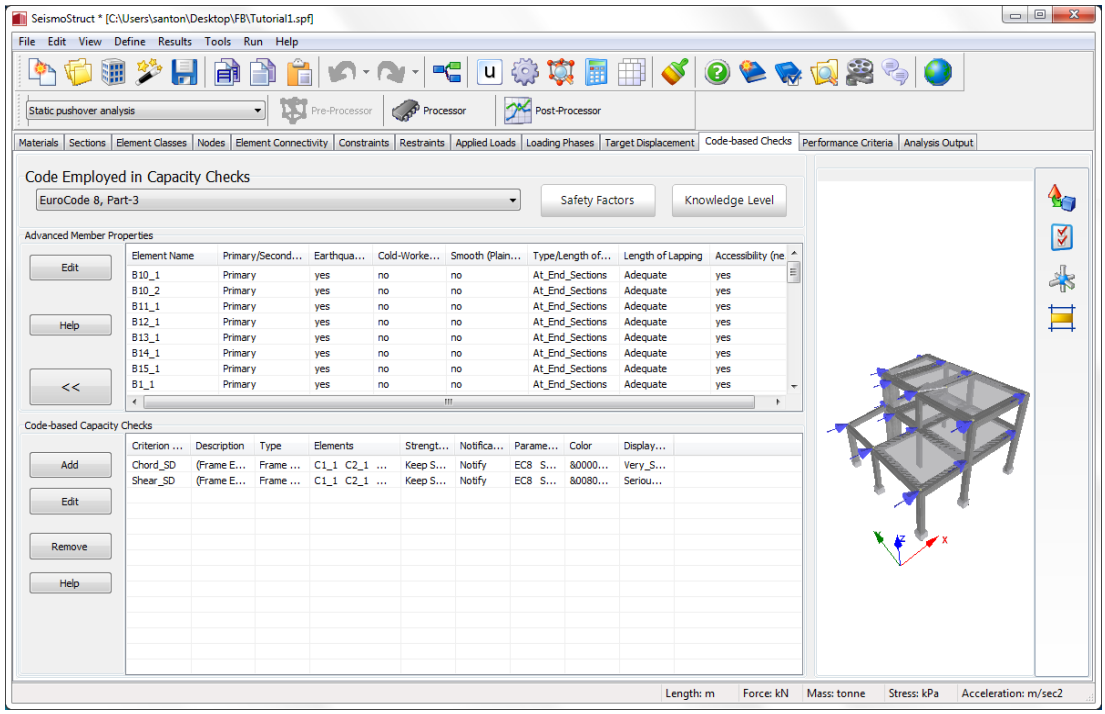
Target Displacement module - Seismic Action

Pre-processor – Code-based Checks

From version 2016 onwards the option to automatically undertake chord-rotation and shear capacity checks is provided. In this module the Code employed for the checks, as well as the Safety Factors and the achieved Knowledge Level may be assigned. The Advanced Member Properties of all the elements, defined in the Building Modeller, may also be visualised and modified in the Code-based Checks module. For the purpose of the current tutorial, the chord rotation and shear capacity check in the significant damage limit state will be introduced.



New Code-based Capacity Check module



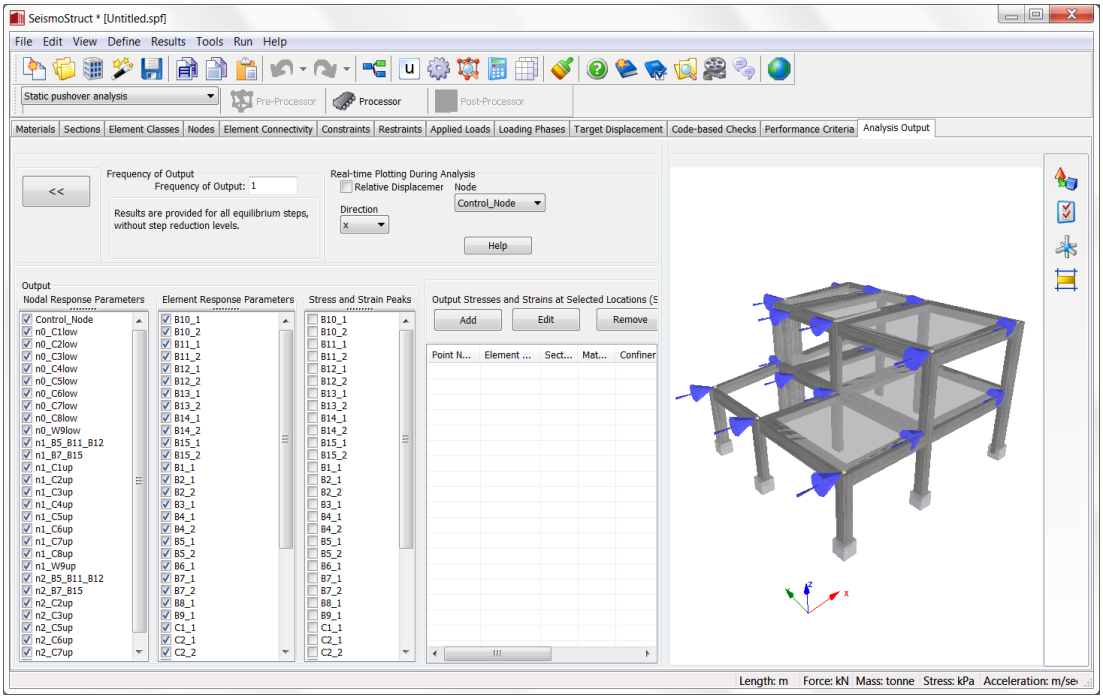
Code-based Checks module

Pre-processor – Performance Criteria

In the Performance Criteria module appear the criteria that were set in the Building Modeller Settings. By default the members' chord rotation and shear capacities are checked.

Pre-processor – Analysis Output

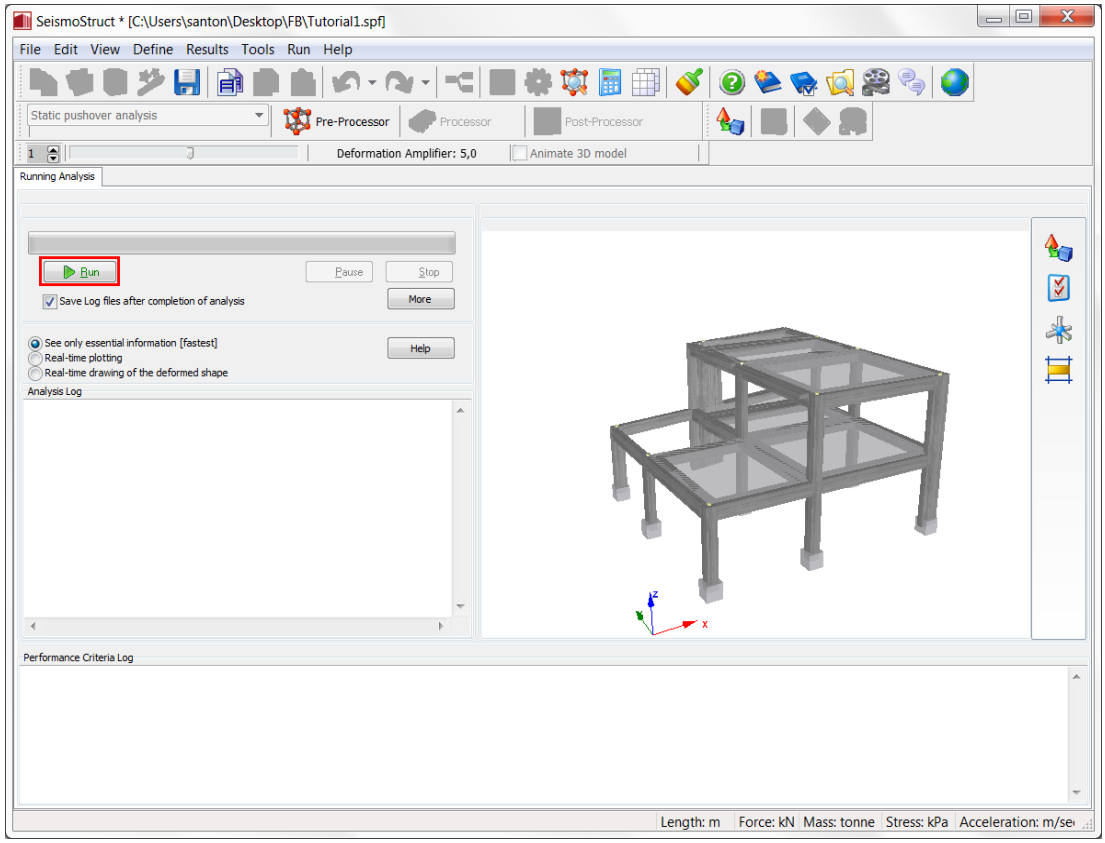
Before accessing to the **Processor** area, you have to set the output preferences in the **Analysis Output** module, as shown below:



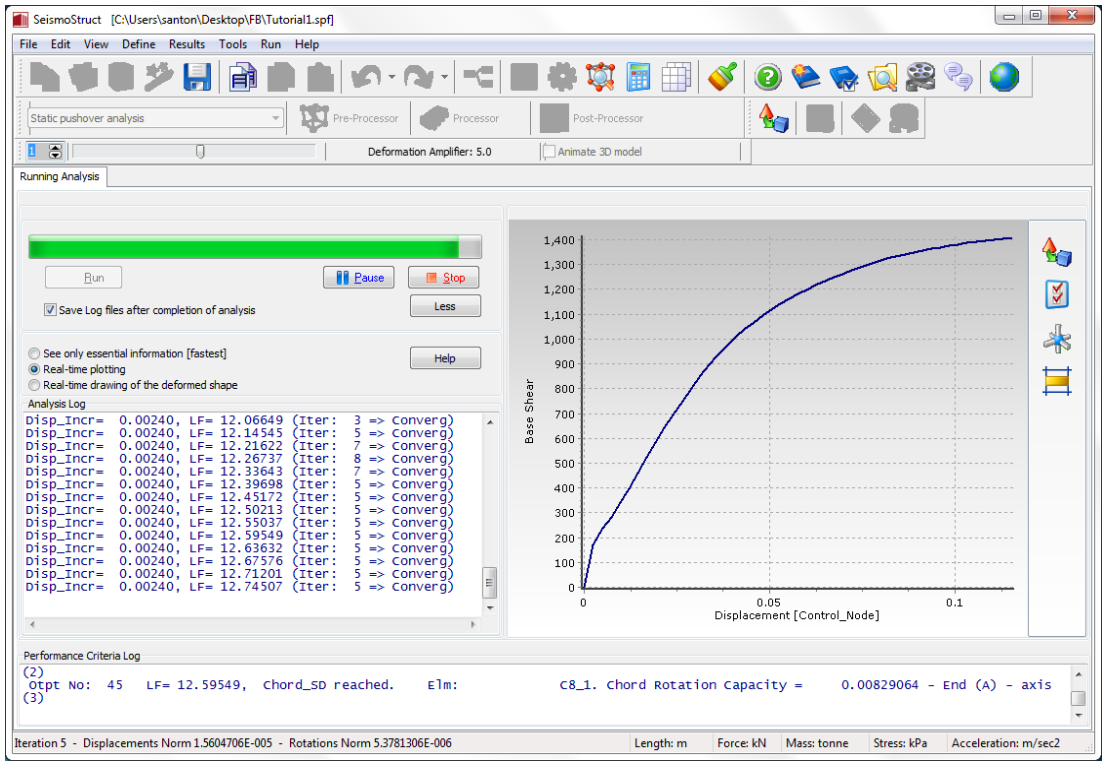
Analysis Output module

Processor

In the **Processor** area you are allowed to start the analysis. Hence, click on the *Run* button.



Processor area



Running the analysis

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

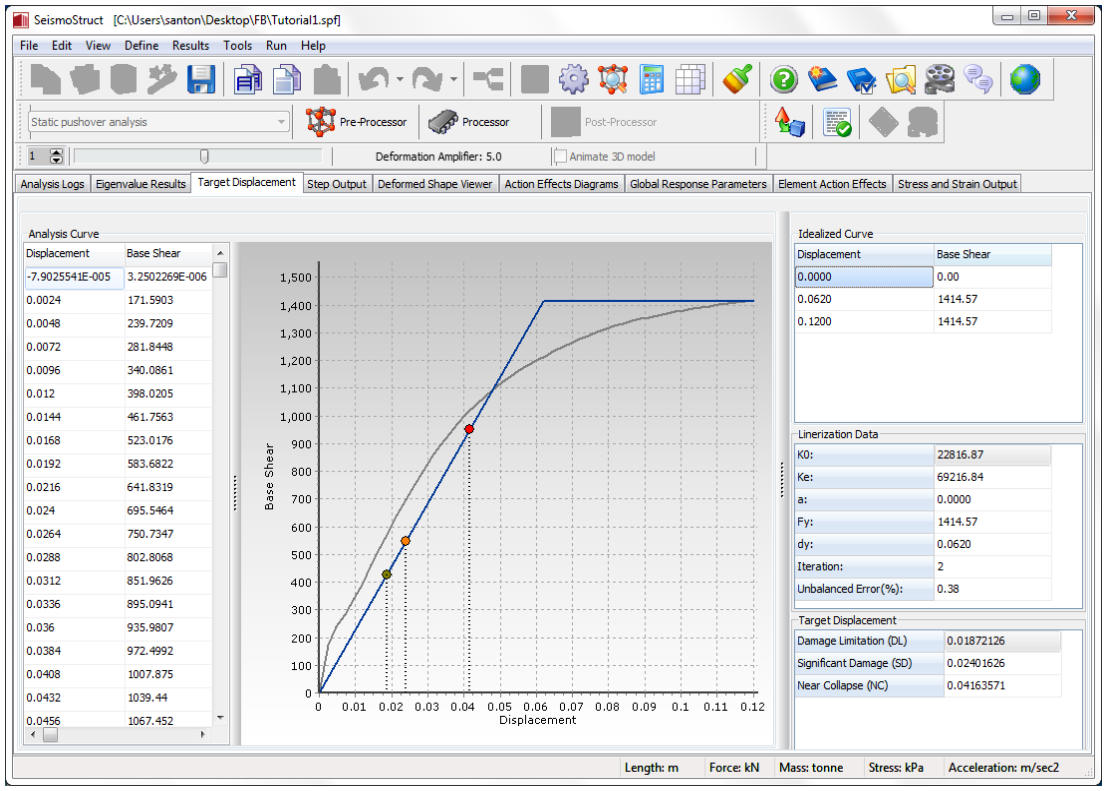
When the analysis has arrived to the end, click on the toolbar button 

or select *Run > Post-Processor* from the main menu.

Post-processor – Target Displacement

The **Post-Processor** area features a series of modules where results can be visualised, in table or graphical format, and then copied into any other Windows application.

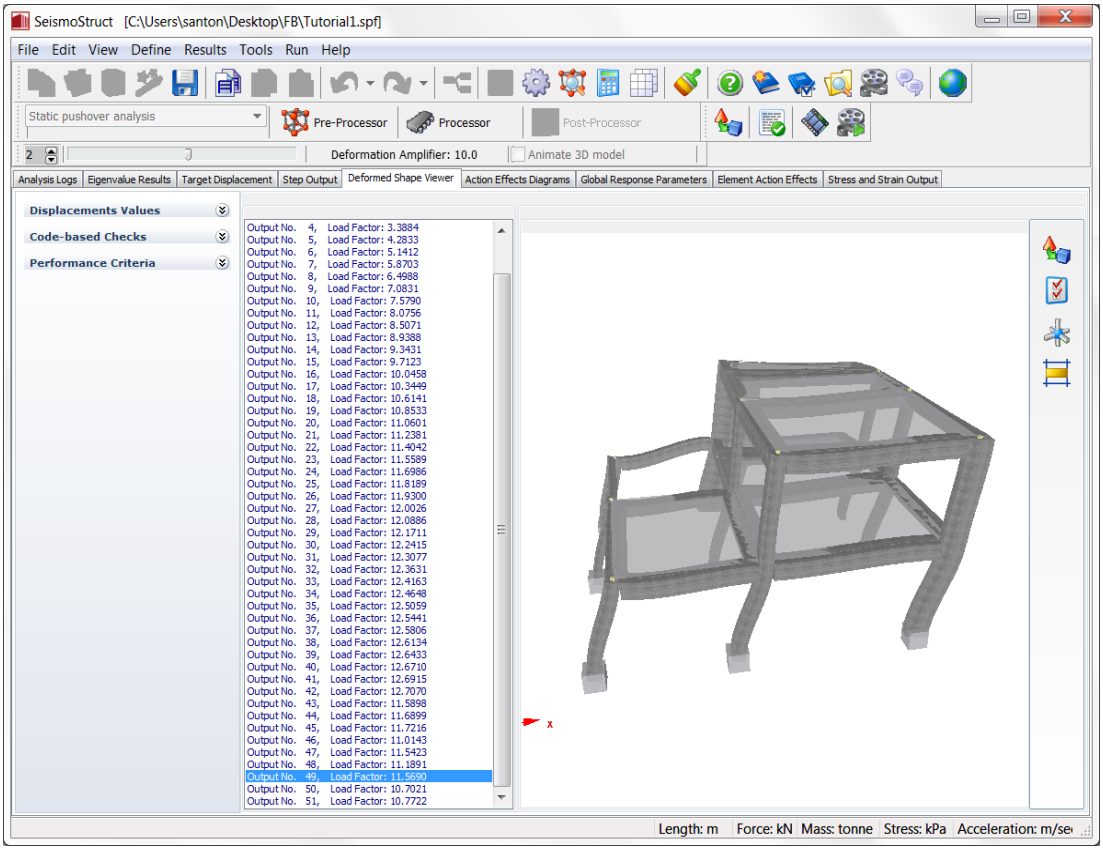
In the **Target Displacement** module you have the possibility of visualising the capacity curves before and after the linearisation, as well as the calculated target displacements for the limit states selected in the Pre-Processor.



Target Displacement module

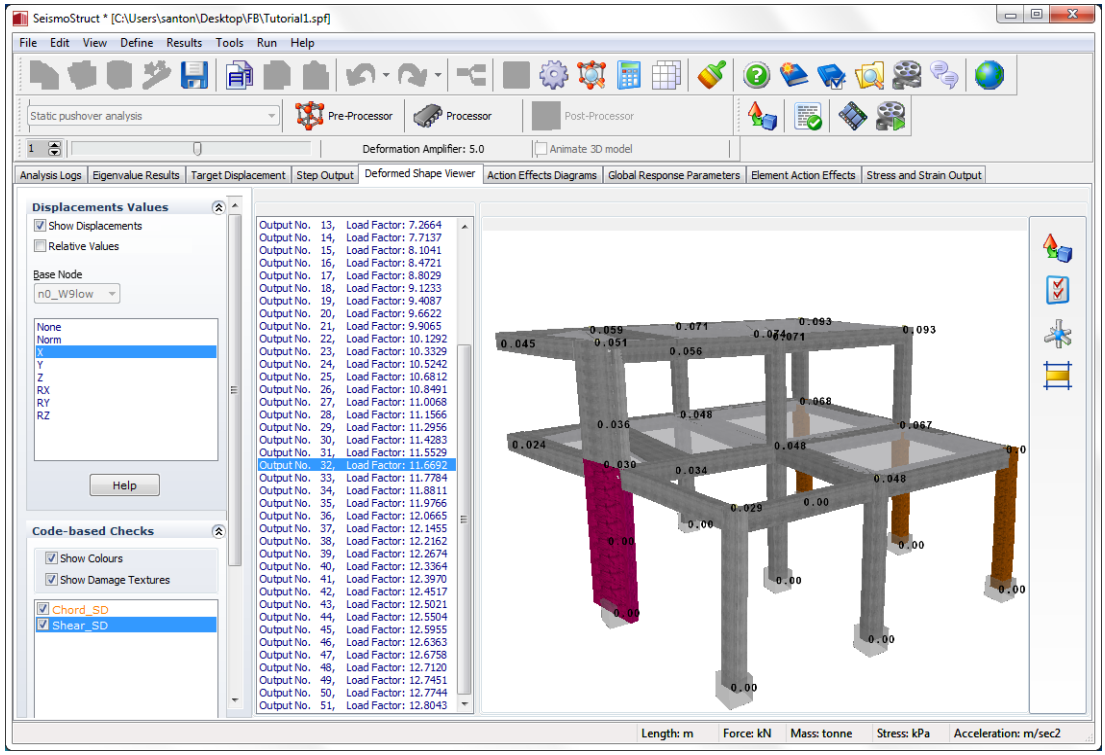
Post-processor – Deformed Shape Viewer

In the **Deformed shape viewer** module you have the possibility of visualising the deformed shape of the model at every step of the analysis. Double-click on the desired output identifier to update the deformed shape view (see figure below).



Deformed Shape Viewer module

It is also possible to visualise the elements that reach a particular Code-based check or performance criterion, which can be done by ticking the corresponding display option and selecting from the lists below the checks or criteria to be displayed. Finally, the displacements values may also be displayed by checking the relevant checkbox.



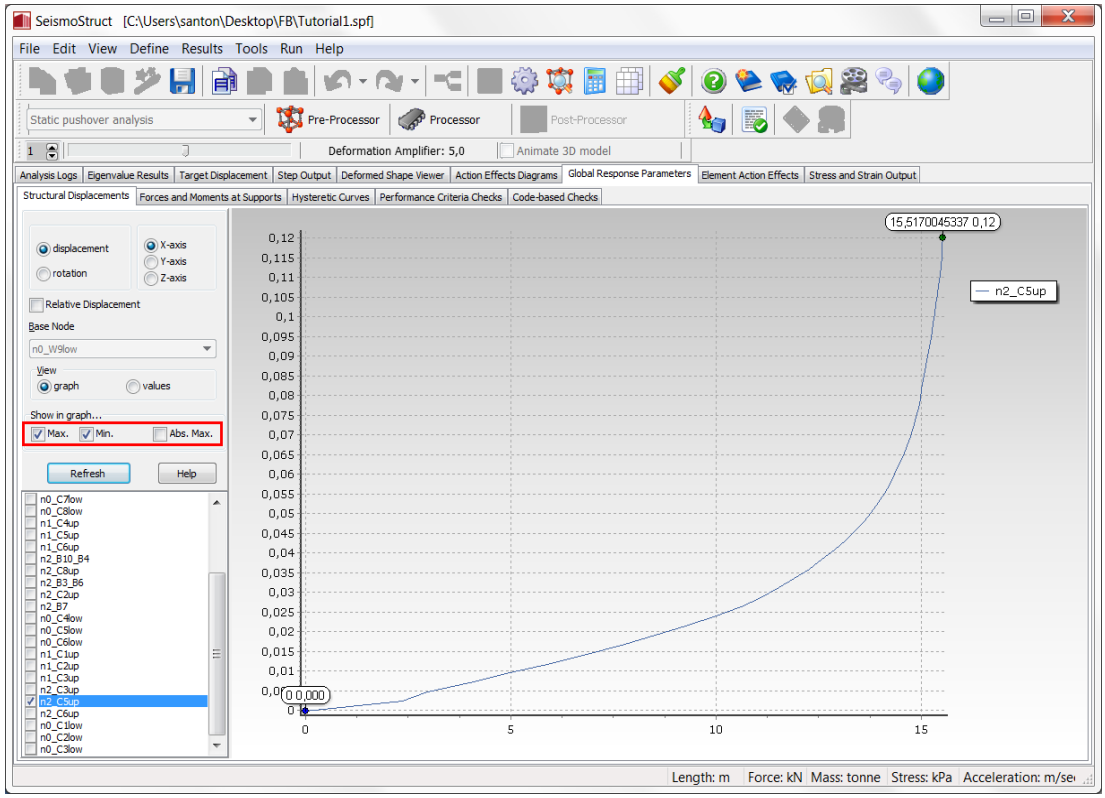
Deformed Shape Viewer module, deformations and Code-based Checks display

Post-processor – Global Response Parameters

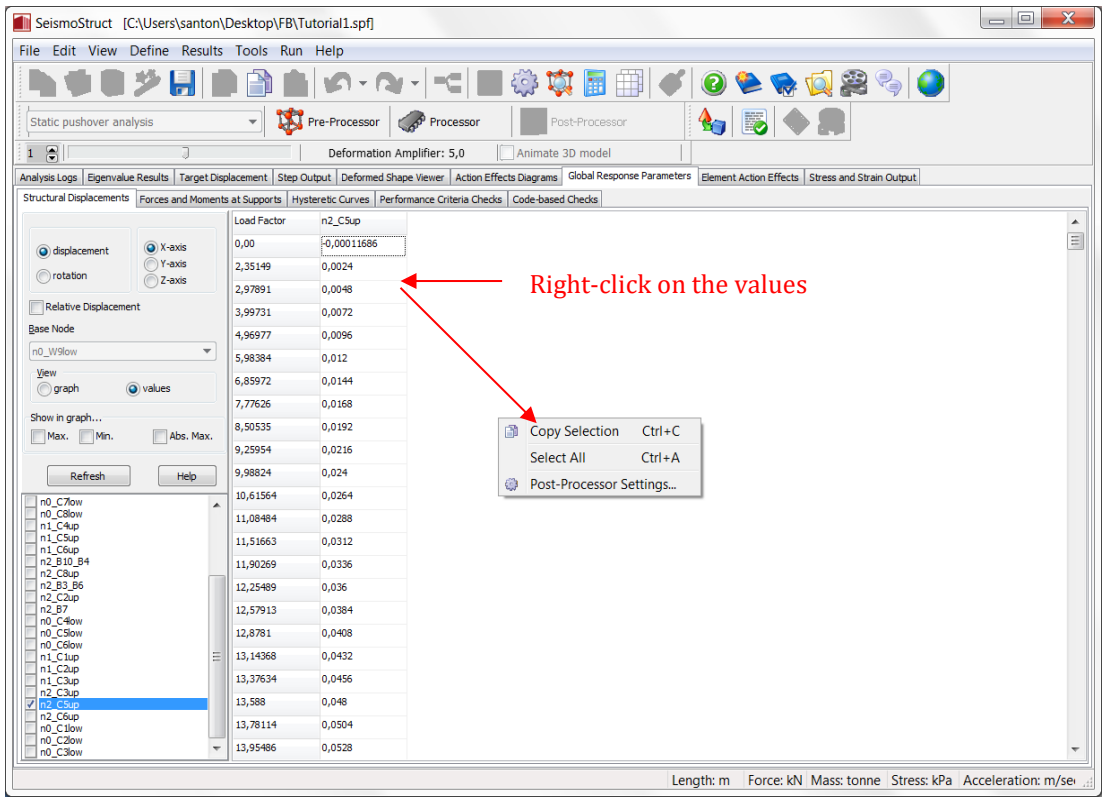
In the **Global Response Parameters** module you can output the following results: (i) structural displacements, (ii) forces and moments at the supports and (iii) hysteretic curves.

First, 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, respectively, *displacement* and *x-axis*, (iii) select the corresponding node from the list (-> n2_C5up) by ticking the box, (iv) choose the results visualisation (*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.

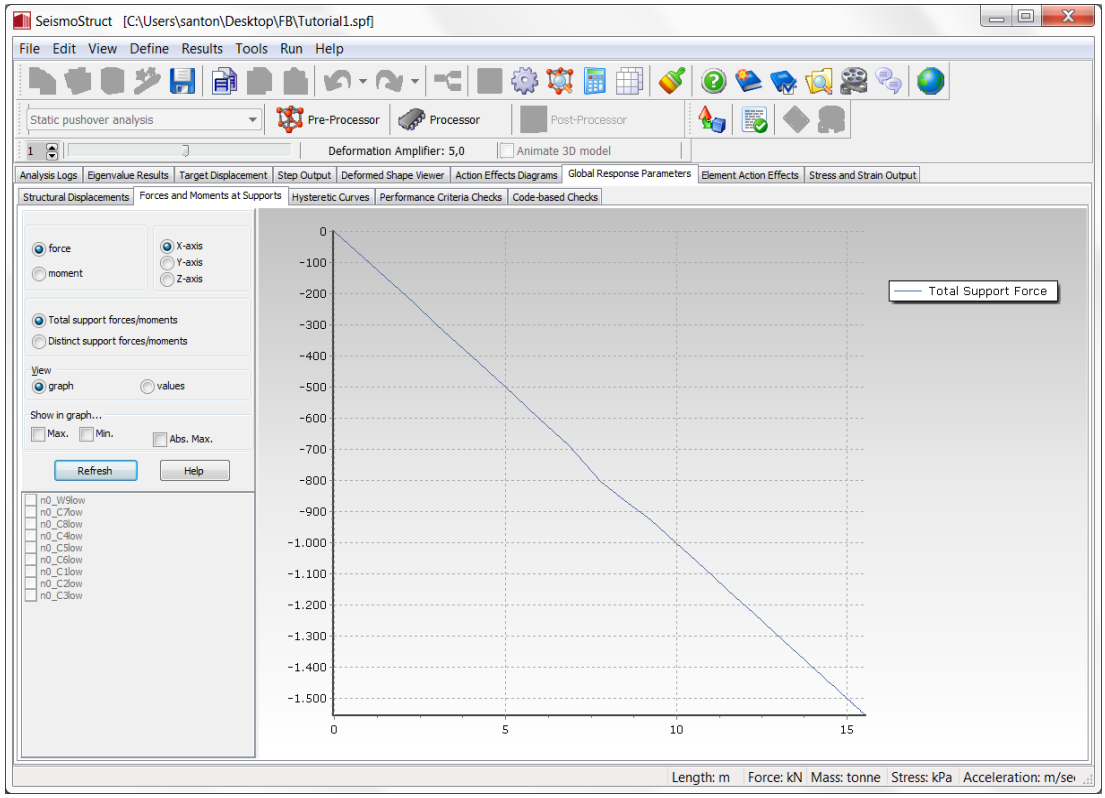


Global Response Parameters Module (Structural Displacements – graph mode)



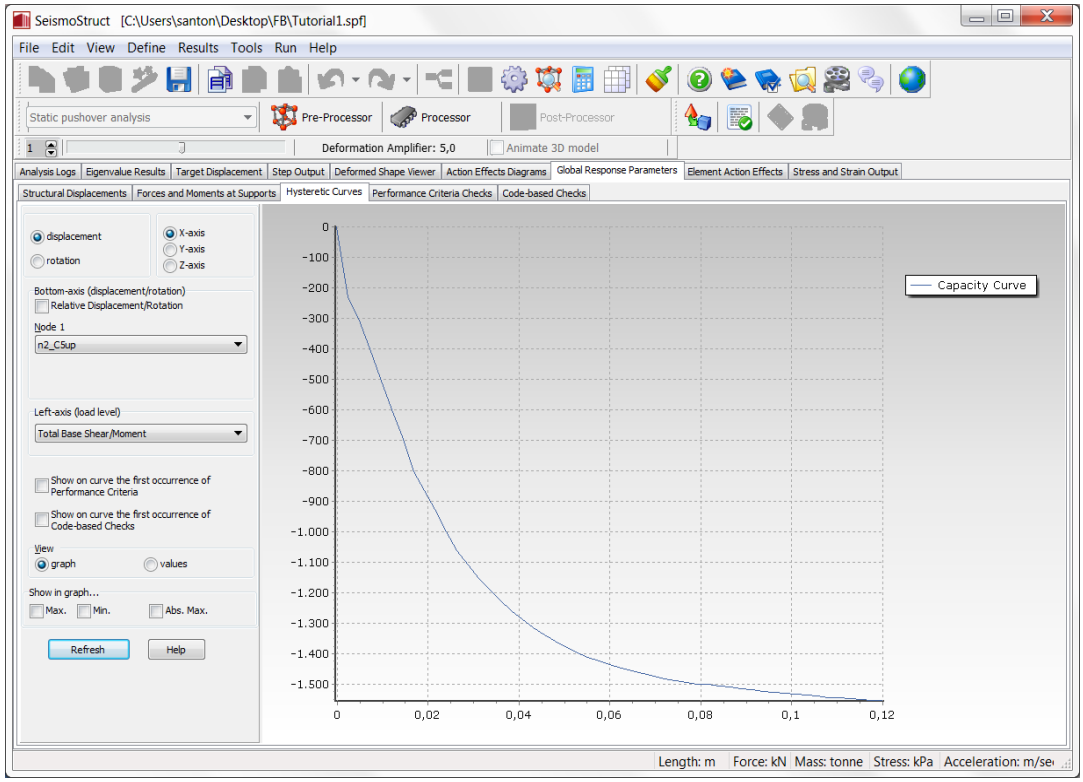
Global Response Parameters Module (Structural Displacements – values mode)

Second, in order to obtain the total support forces (e.g. total base shear), (i) click on the *Forces and Moments at support* tab, (ii) select, respectively, *force* and x-axis and *total support forces/moments*, (iii) choose the results visualisation (*graph* or *values*) and finally (iv) click on the *Refresh* button.



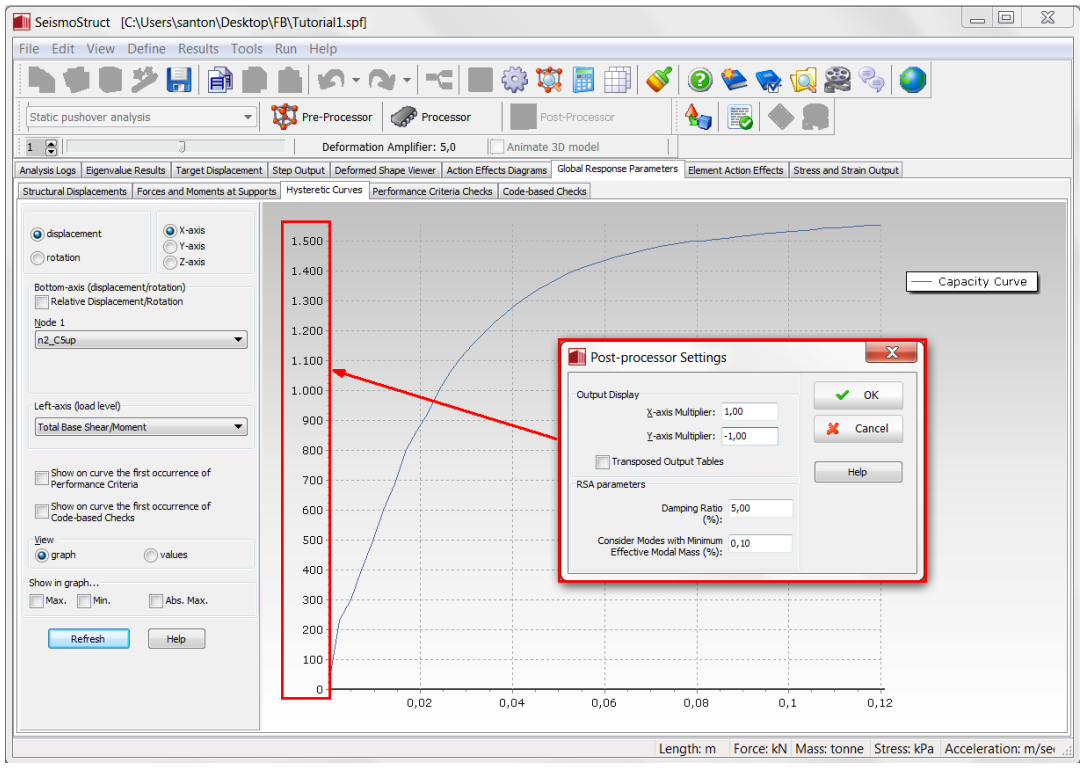
Global Response Parameters Module (Forces and Moments at Supports – graph mode)

Third, in order to plot the capacity curve of your structure (i.e. total base shear vs. top displacement), (i) click on the *Hysteretic Curves* tab, (ii) select, respectively, *displacement* and *x-axis*, (iii) select the corresponding node from the drop-down menu (e.g. n2_C5up) for the bottom-axis, (iv) select the *Total Base Shear/Moment* option for the left-axis, (v) choose the results visualisation (*graph* or *values*) and finally (vi) click on the *Refresh* button.



Global Response Parameters Module (Hysteretic Curves – graph mode)

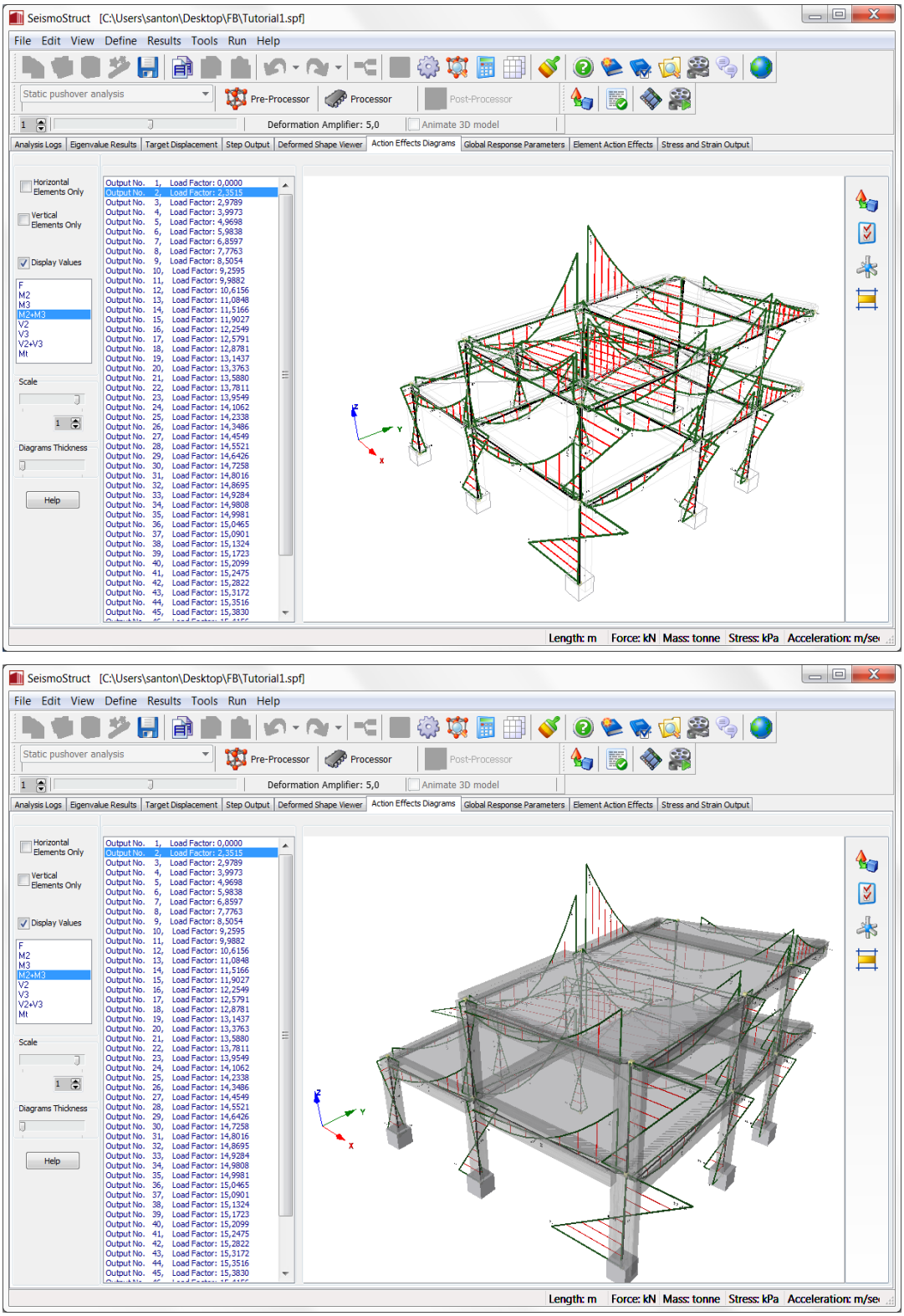
In order to have the shear forces with positive values, (i) right-click on the 3D plot window, (ii) select Post-Processor Settings and (iii) insert the value "-1" as Y-axis multiplier.



Global Response Parameters Module (Hysteretic Curves – graph mode)

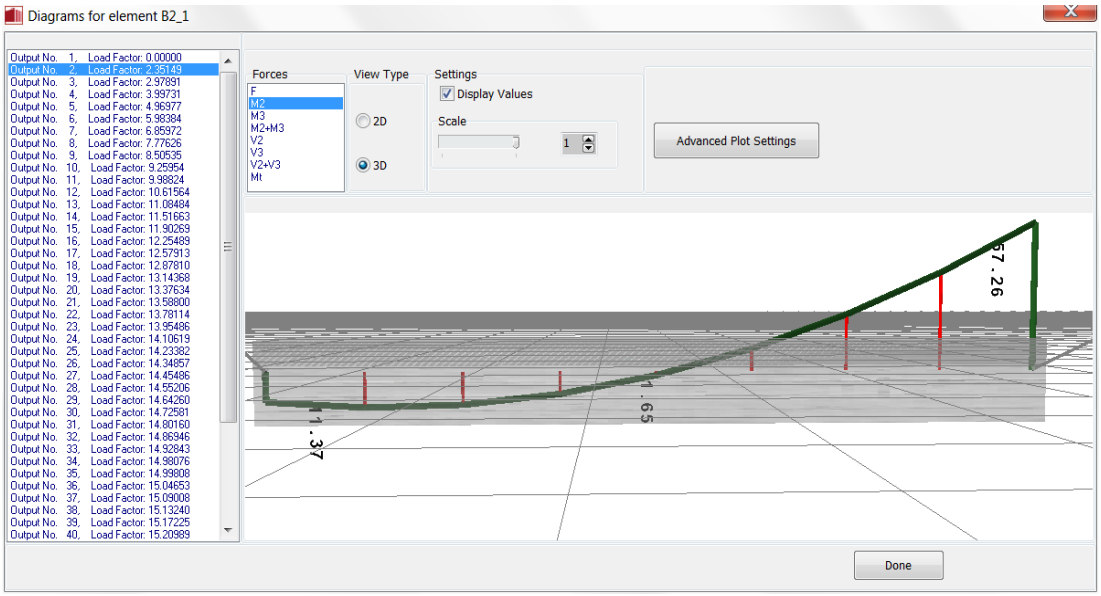
Post-processor –Action Effects Diagrams

In the **Action Effects Diagrams** module, users can visualise the internal forces and moments diagrams for each analysis step. As an example, in the figure below the bending moments diagram is shown:

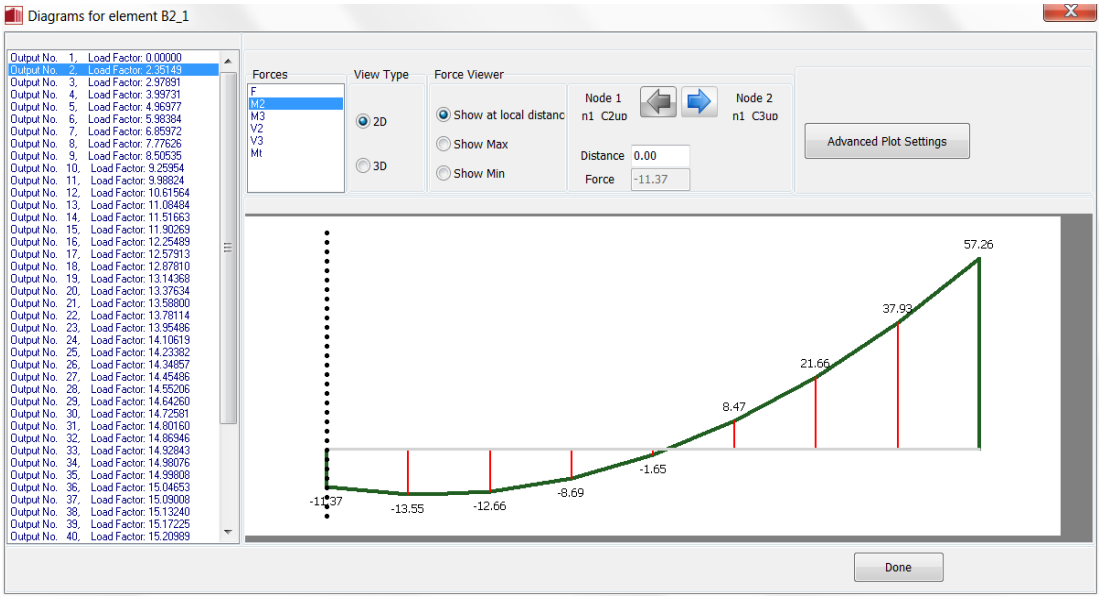


Action Effects Diagrams Module

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



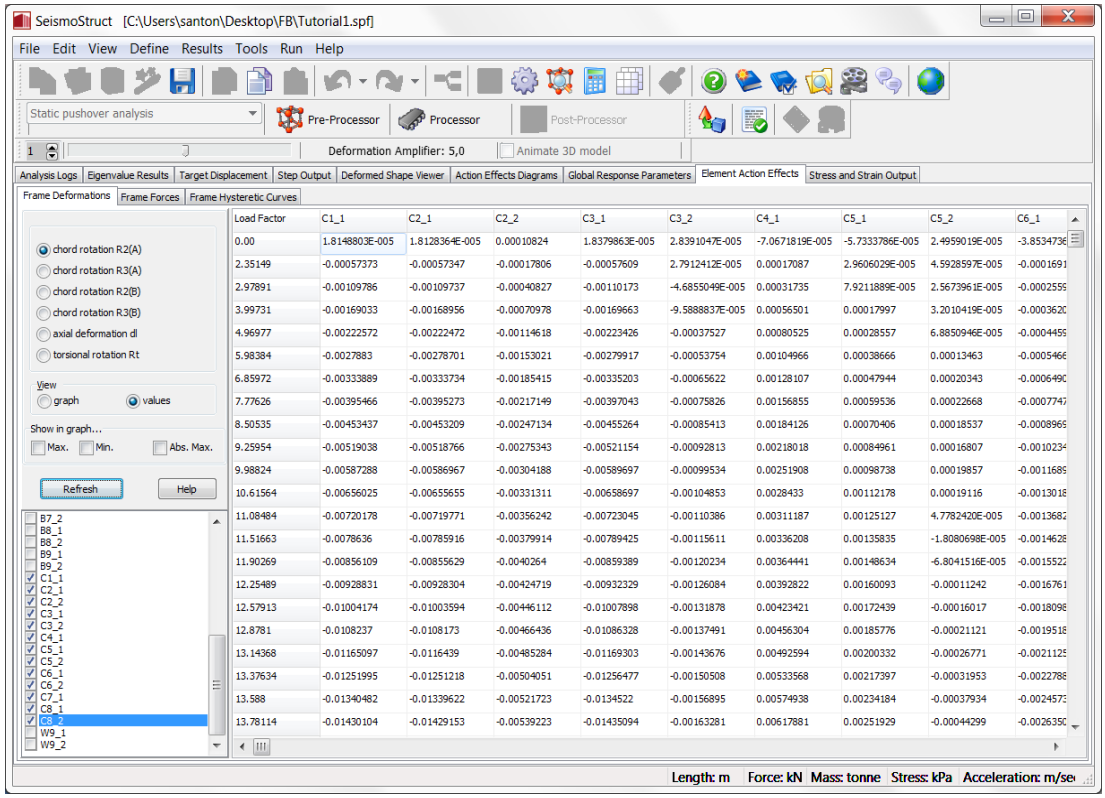
Diagrams for a beam element in 3D



Diagrams for a beam element in 2D

Post-processor – Element Action Effects

The **element chord rotations** and **element shear forces**, which are the main quantities employed in checks from most modern codes (see e.g. Eurocode 8, NTC-18, KANEPE, ASCE 41-23, etc), can be extracted from the *Frame Deformations* and the *Frame Forces* tab windows. Let us start with the former. Since you have employed inelastic force-based frame elements (infrmFB) for defining the structural elements, 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 visualisation (*graph* or *values*) and finally (v) clicking on the *Refresh* button.

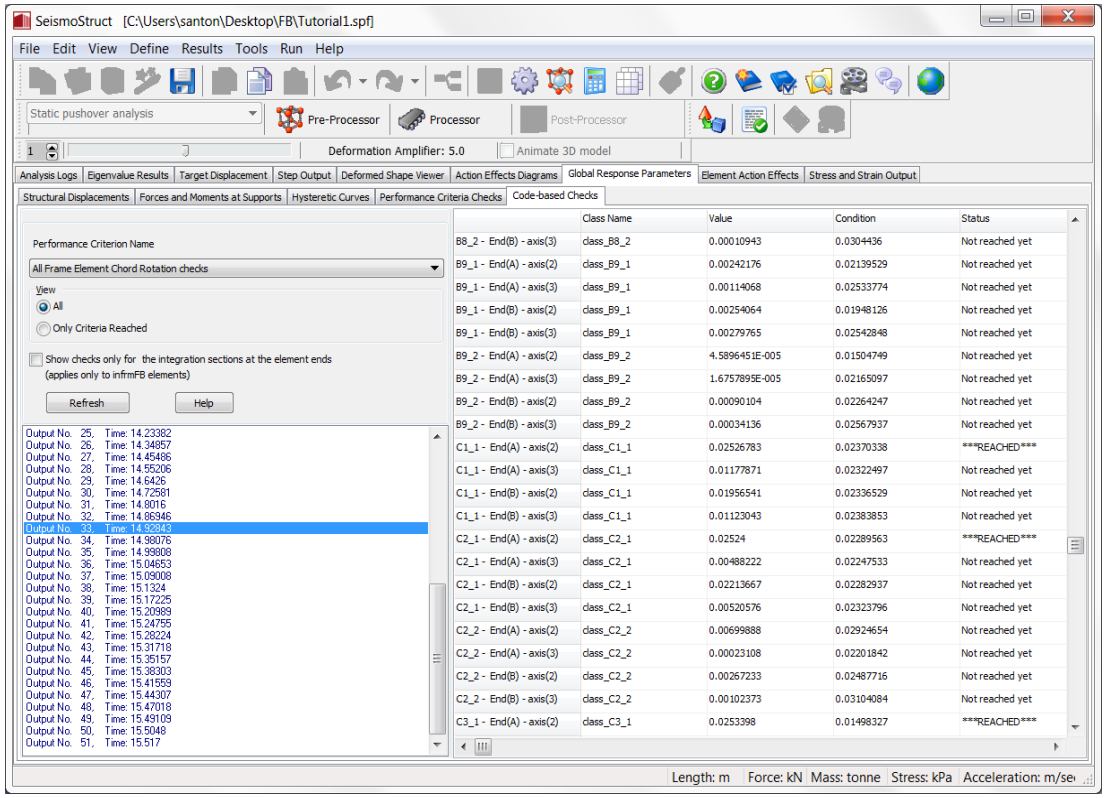


Element Action Effects Module (Frame Deformations – values mode)

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

Post-processor – Code-based Checks and Performance Criteria

In order to avoid the need for users to carry out hand-calculations for the estimation of the capacity of the structural members, SeismoStruct provides the option to automatically undertake chord-rotation and shear capacity checks, according to the expressions defined in the supported Codes (Eurocode 8, ASCE 41-23, NTC-18, KANEPE and TBDY) for the selected limit states. This can be done in the *Code-based Checks* tab of the *Global Response Parameters* page of the Post-Processor.



Code-based Checks

The user may select either a specific Code-based Check or all the defined checks of the same type. Two types of Code-based checks are available, chord rotations and shear capacity checks.

When the user clicks on one of the analysis steps, a list of all the checks for all the structural members appears. The data shown include the demand, the capacity, and whether the particular check has been reached in that particular location, while the results for each integration section and for both local axes, (2) and (3) are provided.

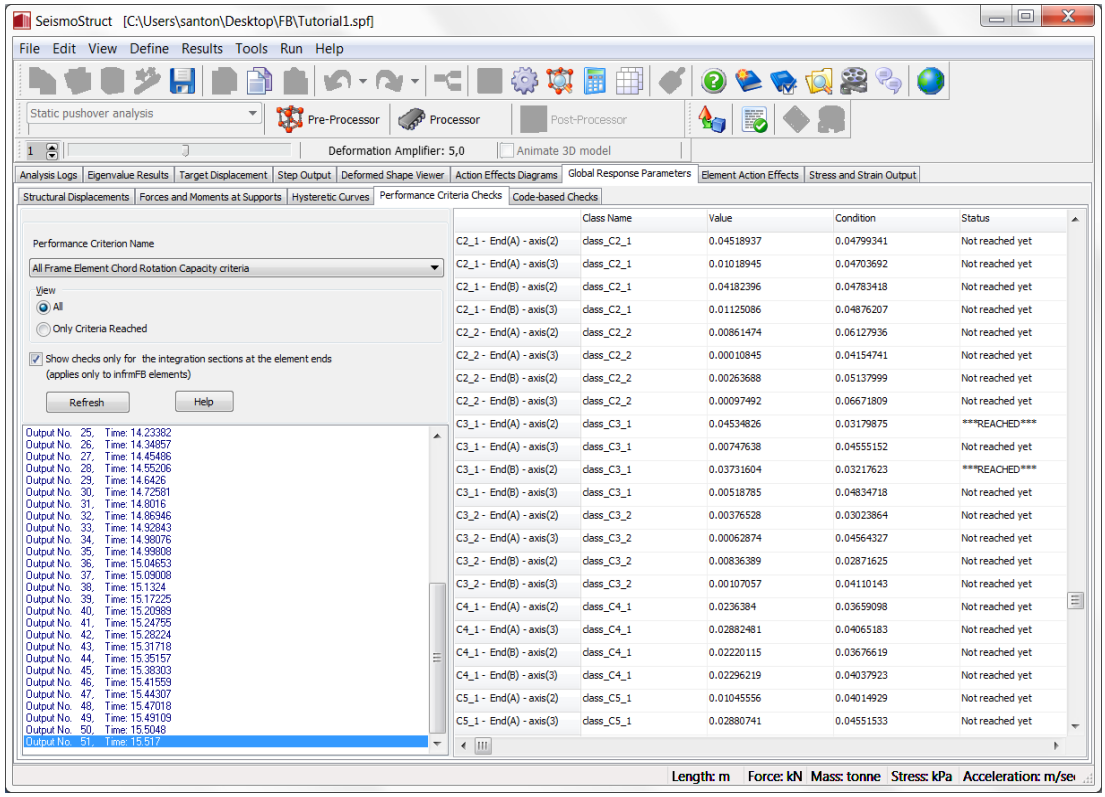
In addition to the code-based checks, users may are capable of setting up performance criteria identifying the instants during the analysis, at which different performance limit states (identified by material strains, section curvature, element chord-rotation and shear values) are expected to be reached. This can be done in the *Performance Criteria Checks* tab of the *Global Response Parameters* page of the Post-Processor. By default the Building Modeller defines two types of criteria (i) a chord-rotation capacity criterion called *chord_rot* and *(ii)* a shear capacity criterion called *shear.*

NOTE: The main difference between the Code-based Checks and the Performance Criteria is that the latter are checks against the 'expected' values of the response quantities, whereas the former follow the conservative assessment methodologies as defined by the corresponding Codes and Standards. Hence, in Code-based Checks the expressions employed for the calculation of the threshold value, when the different performance limit states are reached, employ conservative (e.g. characteristic or nominal) material strengths, and are based on the safety and confidence factors, as specified in the Codes. On the contrary, mean material values and no safety or confidence factors are used in the Performance Criteria calculations.

The user may select the Performance Criterion Name or all the defined criteria of the same type. In SeismoStruct, eight types of performance criteria may be defined (i) Concrete strain of RC or composite sections, (ii) Reinforcement strain of RC or composite sections, (iii) Steel strain of steel or composite sections, (iv) Section curvature, (v) Frame element chord rotation (i.e. whether a specific value has been

reached), (vi) Frame element shear force (i.e. whether a specific value has been reached), (vii) Frame element chord rotation capacity (i.e. whether the capacity has been reached) and (viii) Frame element shear capacity (i.e. whether the capacity been reached).

When the user clicks on one of the analysis steps, a list of all the structural members will appear, with checks of the selected performance criterion in all the integration sections and in both local axes, (2) and (3). The data shown include the demand, the capacity, and whether the particular criterion has been reached in that particular location.



Performance Criteria Checks

> **NOTE: Other performance criteria may be defined either from the Performance Criteria tab of the Building Modeller Settings or from the Performance Criteria page of the Pre-Processor.**

Congratulation, you have finished your first tutorial!

TUTORIAL N.2 – PUSHOVER ANALYSIS OF A TWO-STOREY BUILDING

Problem Description

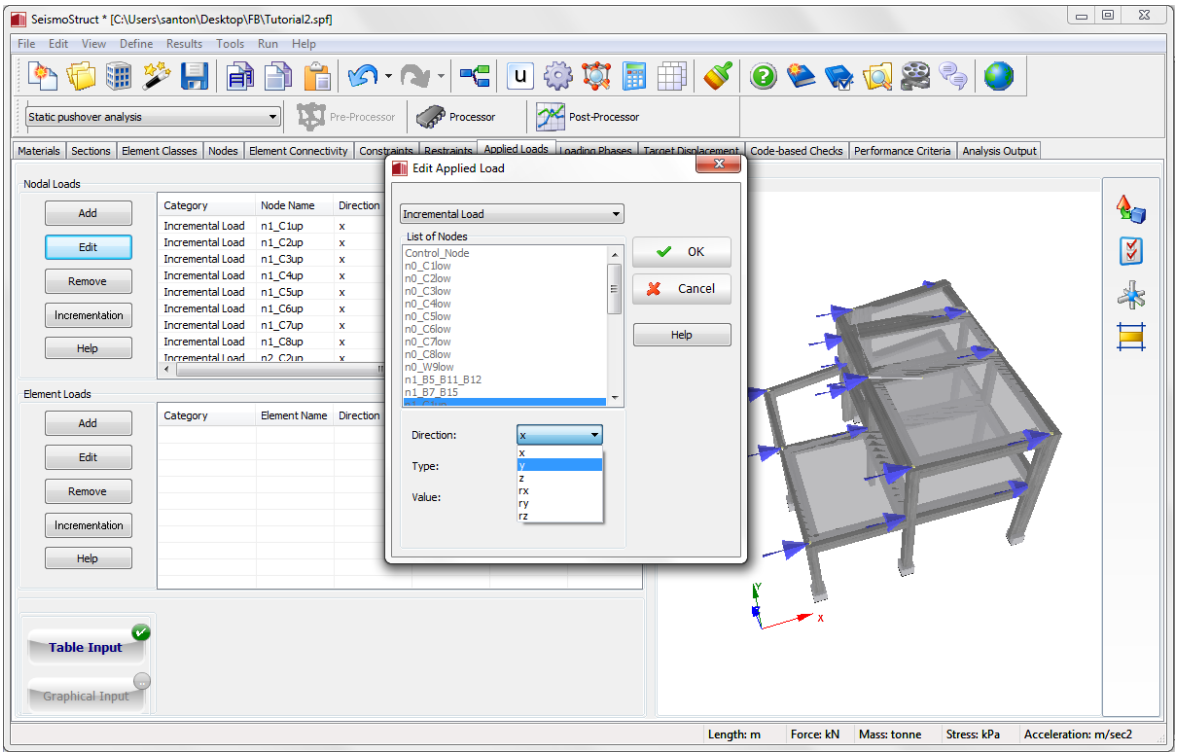
In order to facilitate this second tutorial let us use the model that has already been created in Tutorial n.1.

Getting started: opening an existing project

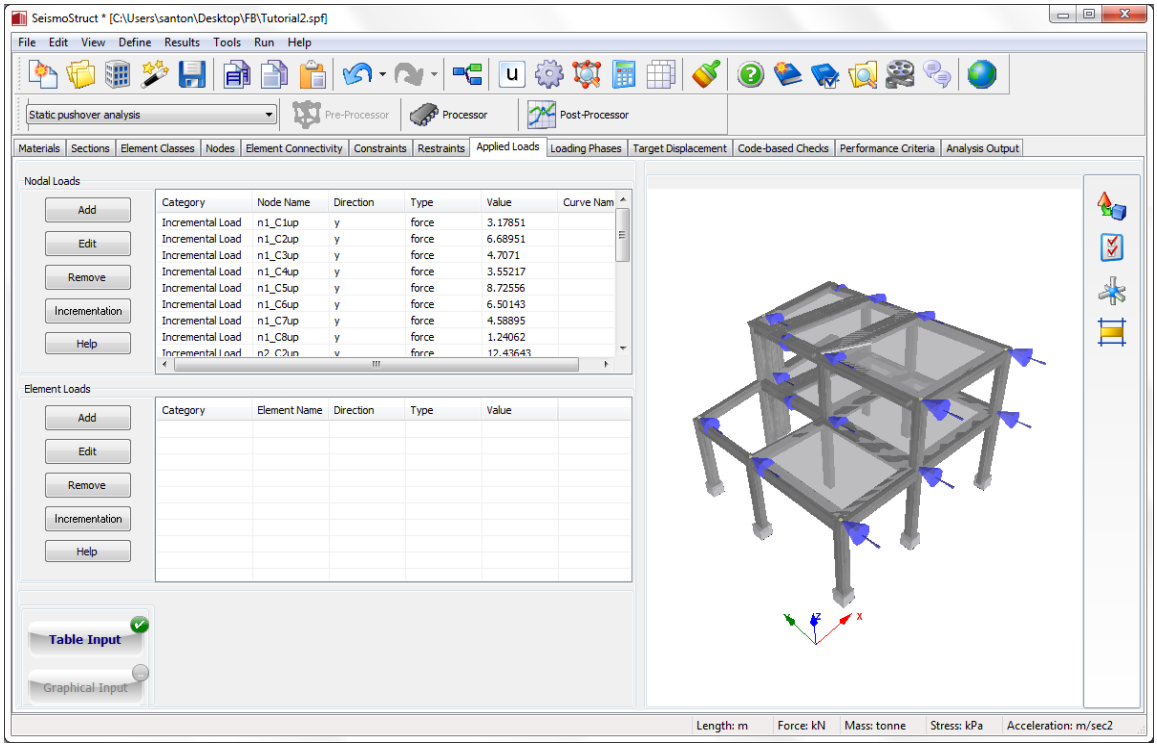
Open again the initial window of the software and, after clicking on  icon on the toolbar, select the previous SeismoStruct project (Tutorial1.spf). Once opened, save the project with a new name through *File > Save as…* menu command.

Pre-Processor - Applied Loads

Select all the Incremental Loads, press the button Edit and change the direction of loads from X to Y, then press OK and the direction of all the loads will have changed.



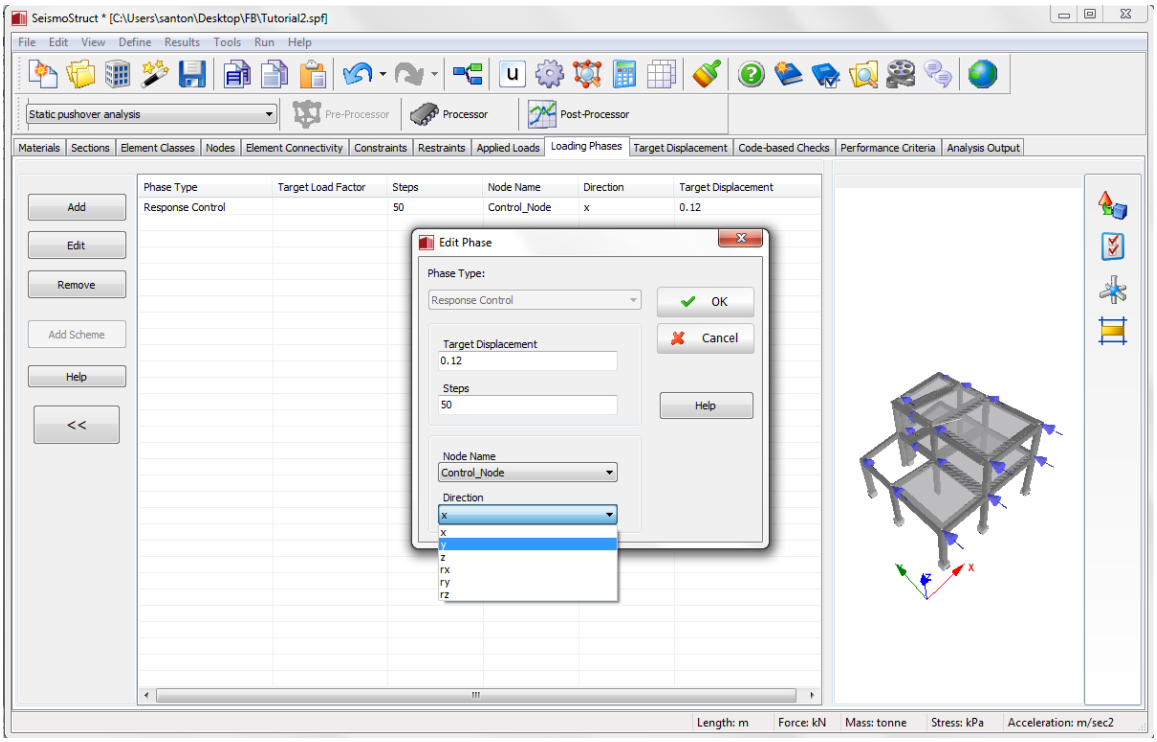
Edit Applied Load



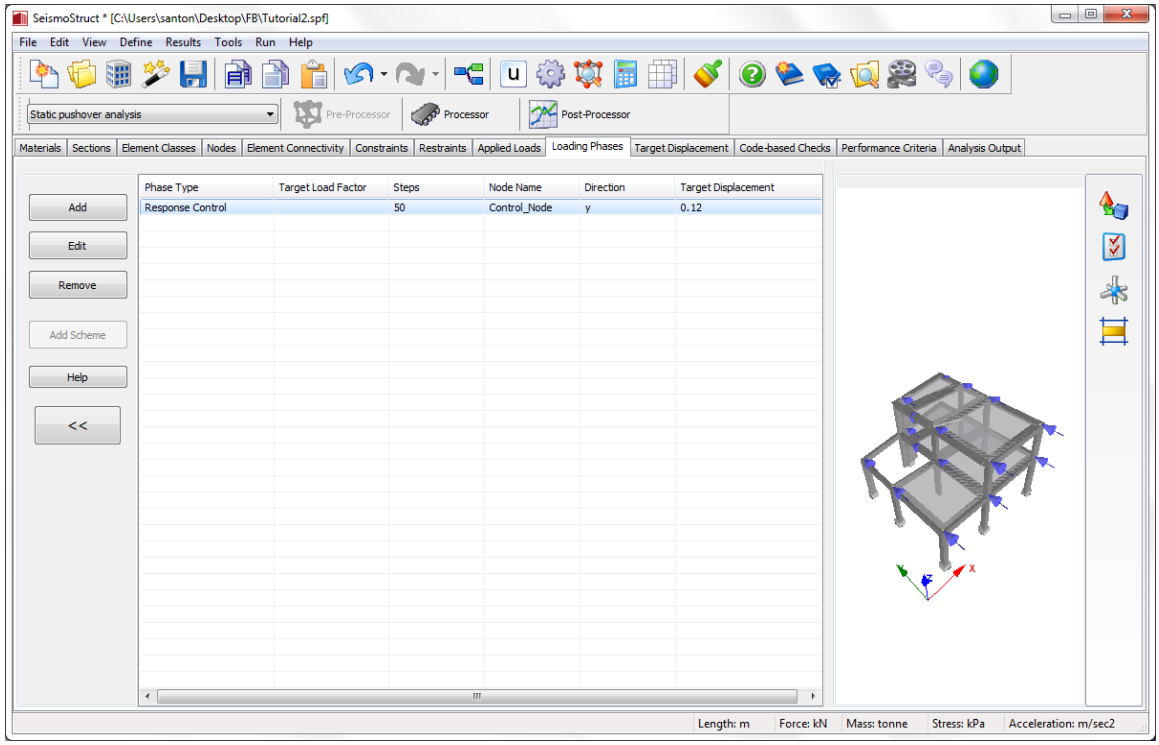
Applied Loads module

Pre-Processor – Loading Phases

Select the phase type, press Edit and change the direction of the Target Displacement from X to Y then press OK.



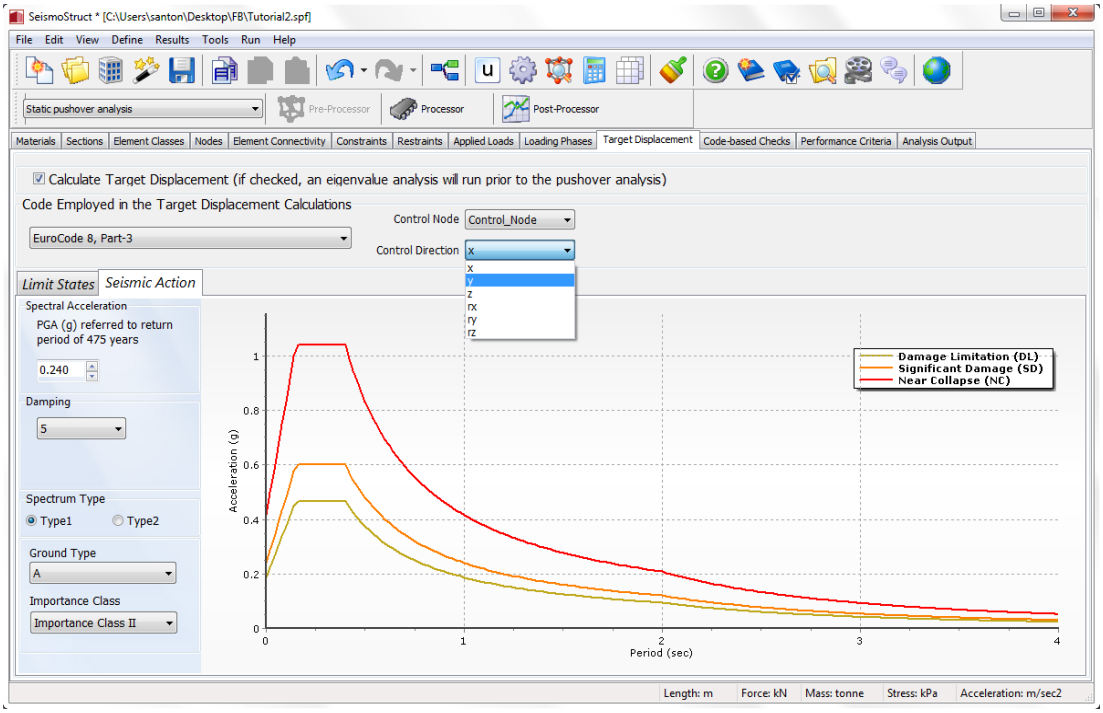
Edit Loading Phase module



Loading Phase module

Pre-Processor – Target Displacement

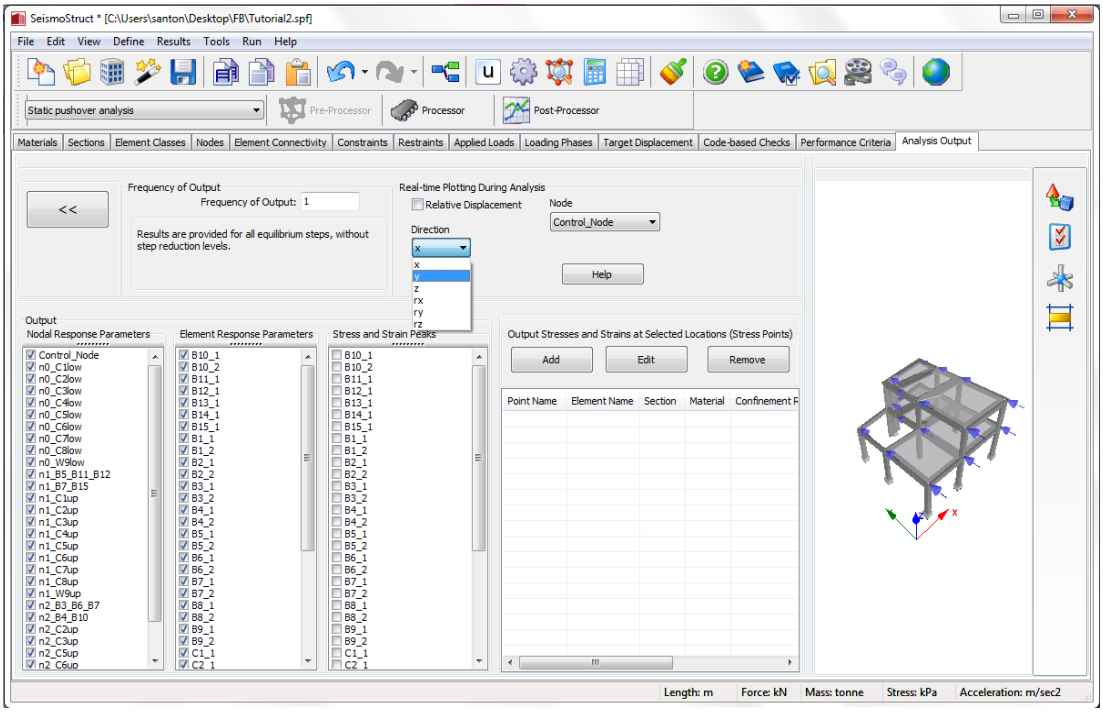
Change the Control Direction in the Target Displacement module from X to Y.



Target Displacement module

Pre-Processor – Analysis Output

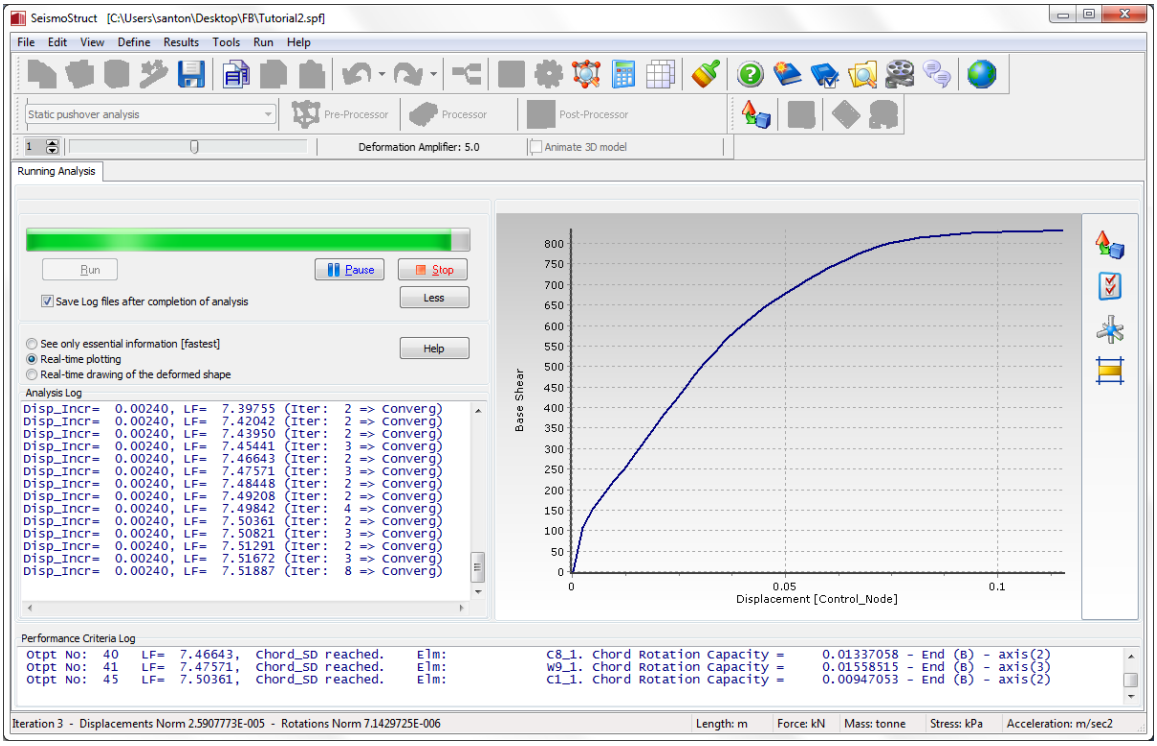
In the Analysis output module change in the Real-time Plotting Analysis the direction from X to Y.



Analysis Output module

Processor

Click on the *Run* button.



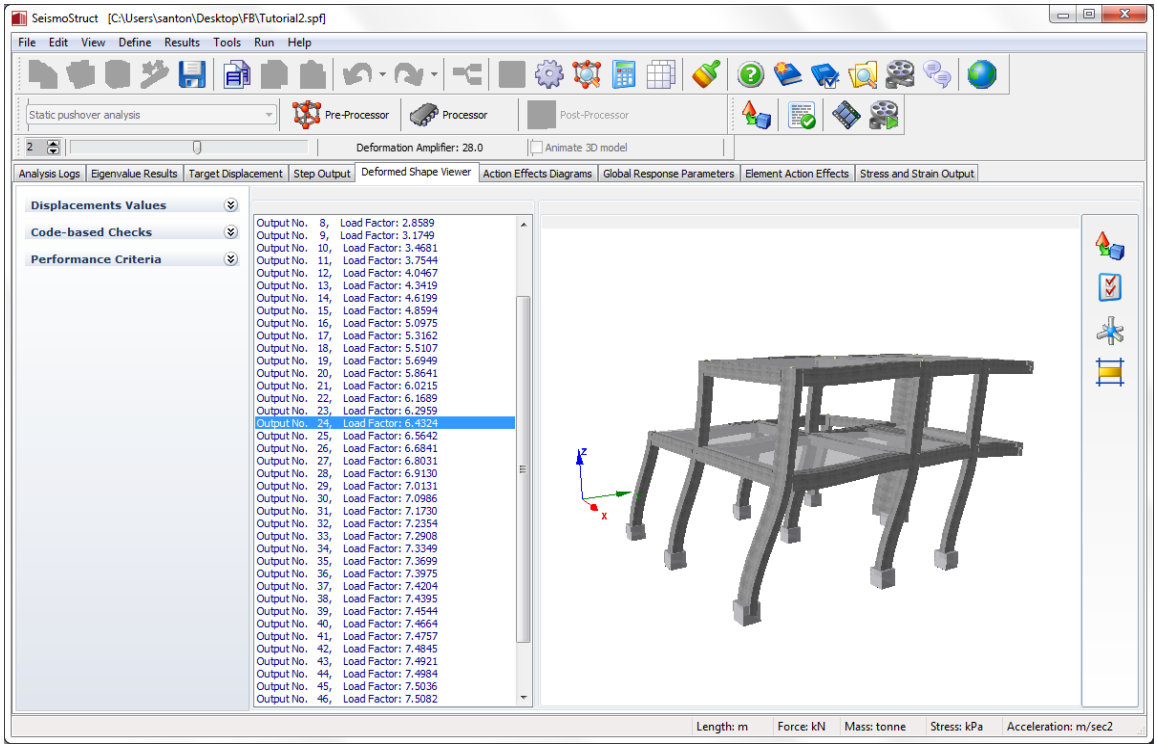
Processor area

When the analysis has arrived to the end, click on the toolbar button 

or select *Run* > *Post-Processor* from the main menu.

Post-Processor

In the **Deformed shape viewer** module you have the possibility of visualising the deformed shape of the model at every step of the analysis. Double-click on the desired output identifier to update the deformed shape view (see figure below).



Deformed Shape Viewer module

TUTORIAL N.3 – DYNAMIC TIME-HISTORY ANALYSIS OF A TWO-STOREY BUILDING

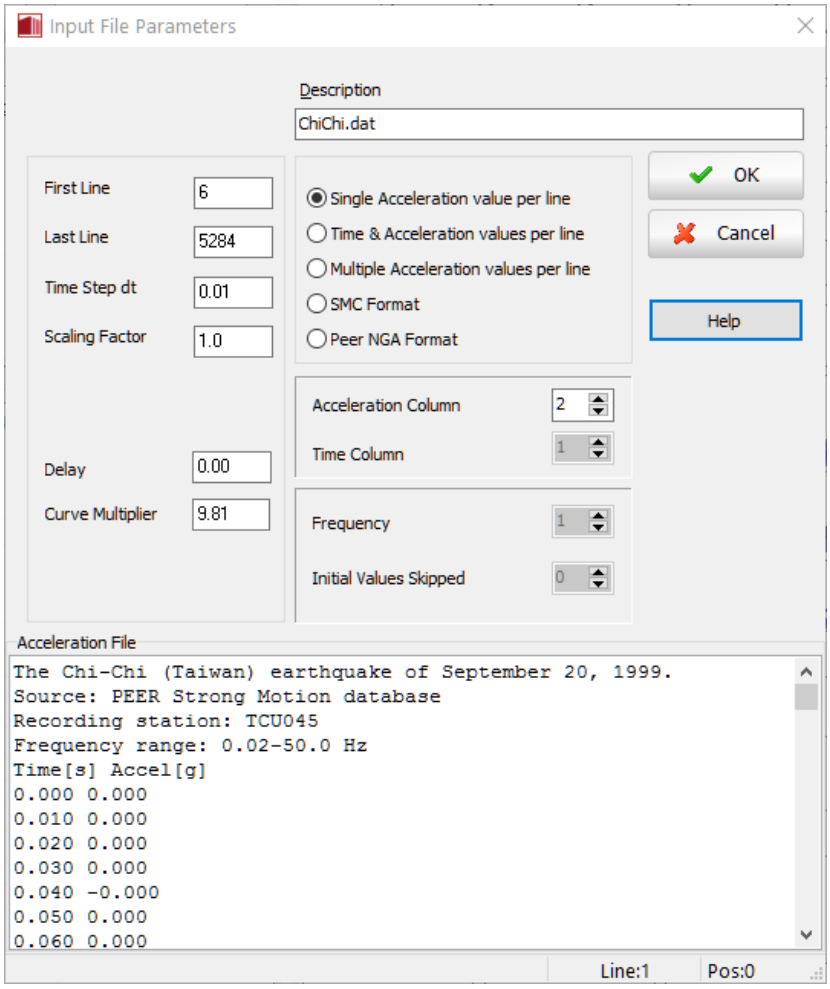
Problem Description

Let us use the model that has already been created in Tutorial n.1 and modified in Tutorial n.2.

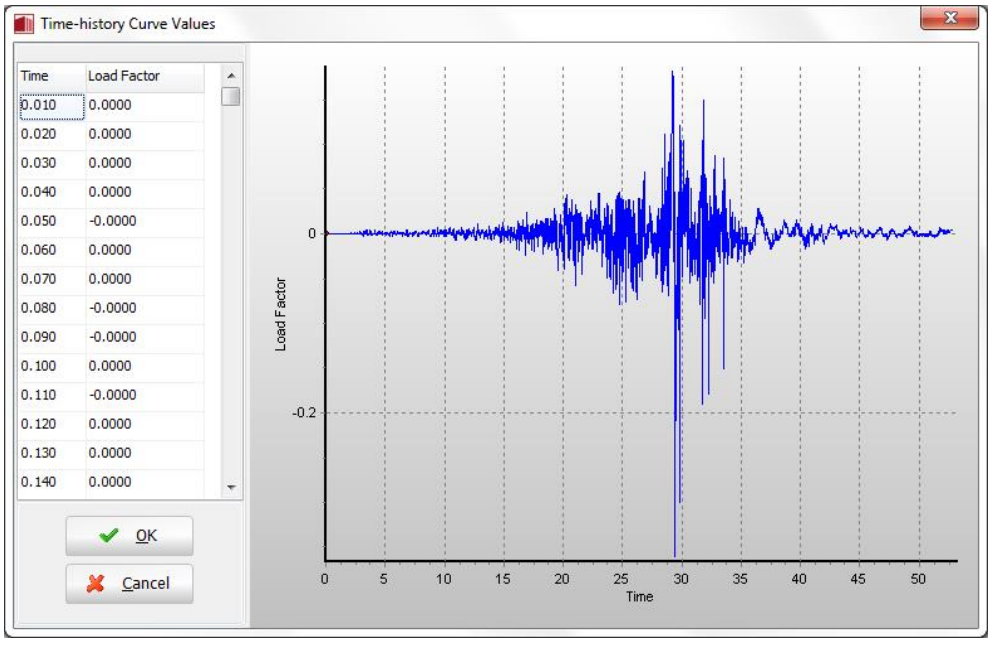
Getting started: opening an existing project

Go to the Building Modeller after clicking on icon on the toolbar, Click on the Open Existing Building Project and select the previous Building Modeller file (Tutorial 1.bmf). Once opened, save the project with a new name through *File > Save as…* menu command. At this point, select the *Dynamic time-history analysis* from the *Building Modeller Settings* and press the Exit and Create project button in order to create the SeismoStruct model. A message for selecting accelerogram appears and you have to

1. Load an accelerogram through the *Select File* button (for simplicity upload one of the curves in the installation folder of the program *(C:\ Program Files\ Seismosoft\ SeismoStruct_2026\ Accelerograms \ ChiChi.dat*);
2. Specify the Curve Multiplier (by default➔ 9.81) and the other Input File Parameters

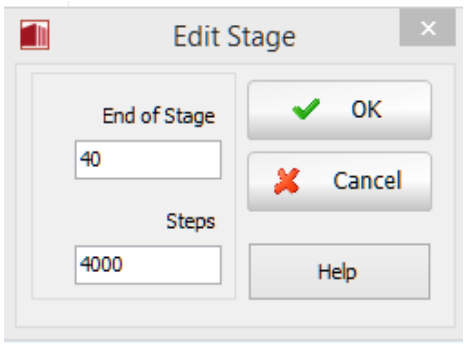


Input File Parameters - Load Curve



Accelerogram - Time-history Curve Values

After loading the curve, you may modify the time-history stages, where the time-step of the analysis can be defined. In the *Time-history stages* section press the *Edit* button, and in the dialog box that opens set (i) the time of the *End of Stage* (which, in this example, is selected 40 sec) and (ii) the number of *steps* (-> 4000).

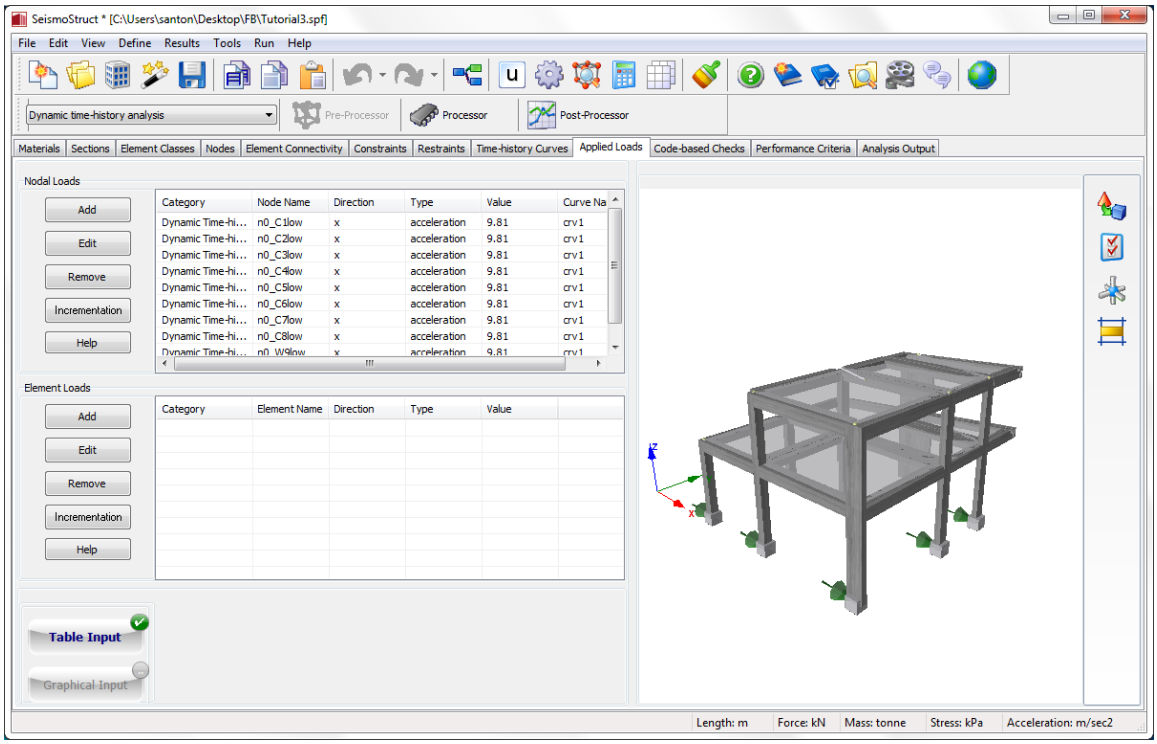


Time-history stage

NOTE: The program computes internally the time step dt. In this case is equal to 40/4000 = 0.01

Pre-processor – Applied Loads

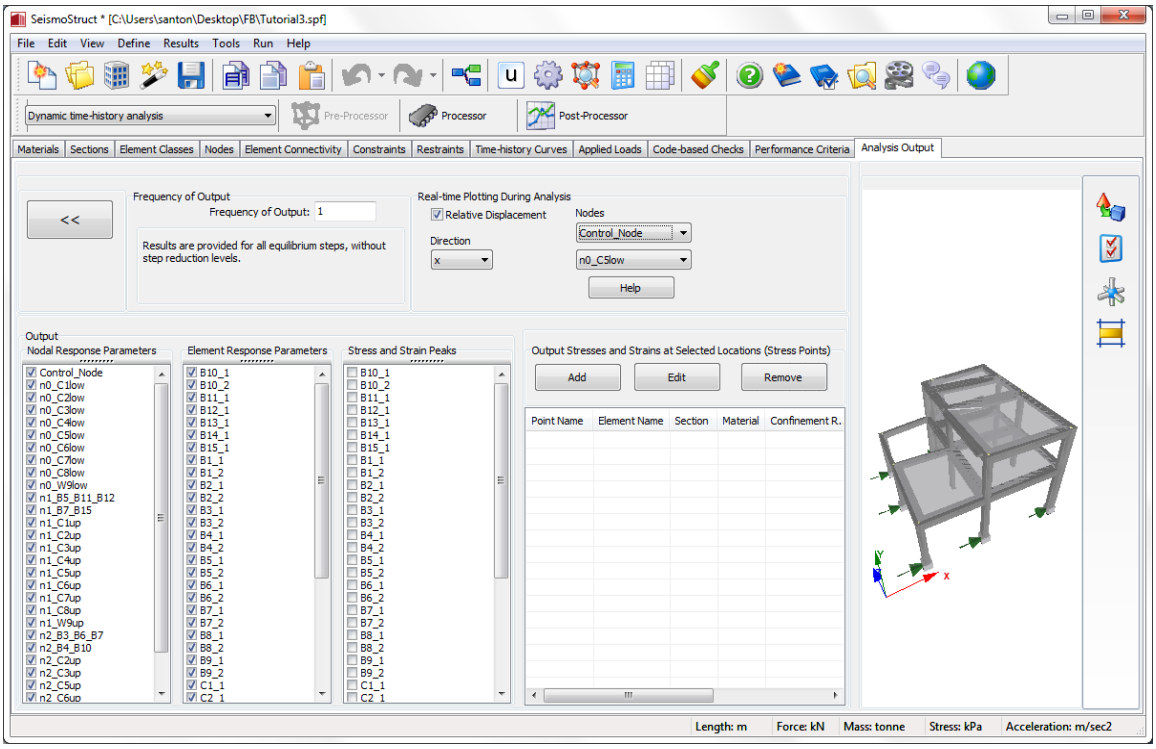
The program automatically applies the Dynamic Time-history load to the ground nodes in Direction X.



Dynamic time-history loads

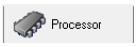
Pre-Processor – Analysis Output

Finally, before entering the **Processor**, you must set your output preferences in the **Analysis Output** module, as shown in the figure below.



Analysis Output module

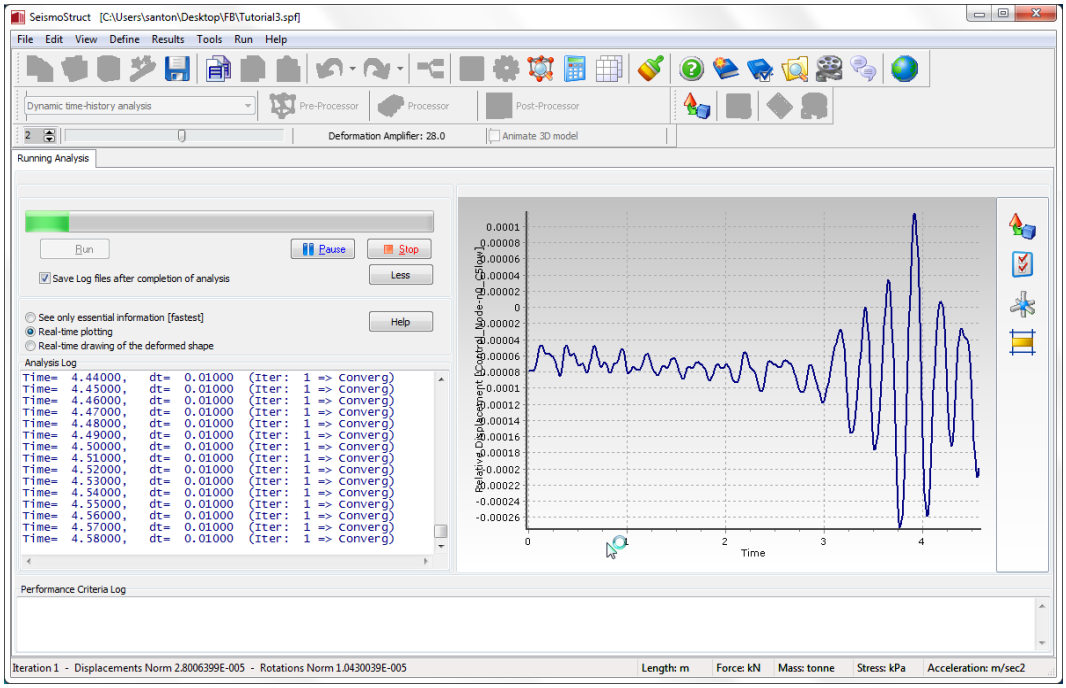
NOTE: Unlike the tutorial 1, in this example we ask to visualise, in the real-time plotting, the total relative displacement of the top node Control_Node with respect to the base node n0_C5low.

At this point you may click on the toolbar button 

or select *Run* > *Processor* from the main menu in order to perform the dynamic time-history analysis.

Processor

Press the *Run* button.



Running the analysis

Once the analysis has arrived to the end, click on the toolbar button 

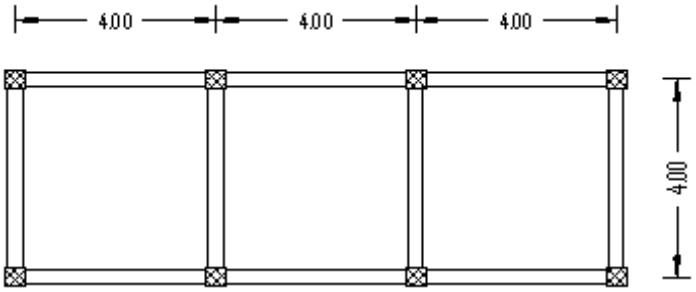
to get the results. As already seen for the Tutorial n.1, in the **Post-Processor** you will be able to see the deformed shape of the structure at each step of dynamic analysis (*Deformed Shape Viewer*) as well as to extract the time-history displacement response of the structure, and so on.

TUTORIAL N.4 – PUSHOVER ANALYSIS OF A TWO-STOREY BUILDING

NOTE: In this Tutorial n.4 you will not use the *Wizard* or *Building Modeller* facility but you will rather create the model entirely yourself, step by step.

Problem Description

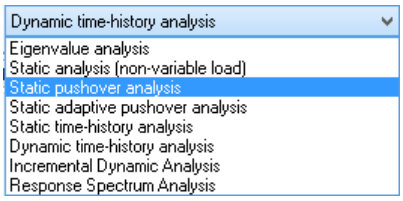
Let us try to model a three dimensional, two-storey reinforced concrete building for which you are asked to run a pushover analysis. Let us assume that the structure is regular, it has three bays and consists of two parallel frames. The bay lengths are 4 meters, the storey heights are 3 meters and the distance between the two frames is 4 meters, as you can see in the pictures below:



Plan view of the building

Getting started: a new project

In order to open SeismoStruct initial window select the *File > New…* menu command or click on icon on the toolbar. Then, first, select *Static pushover analysis* from the drop-down menu at the top left corner on the *Pre-Processor* window (see picture below).



Selection of the analysis type

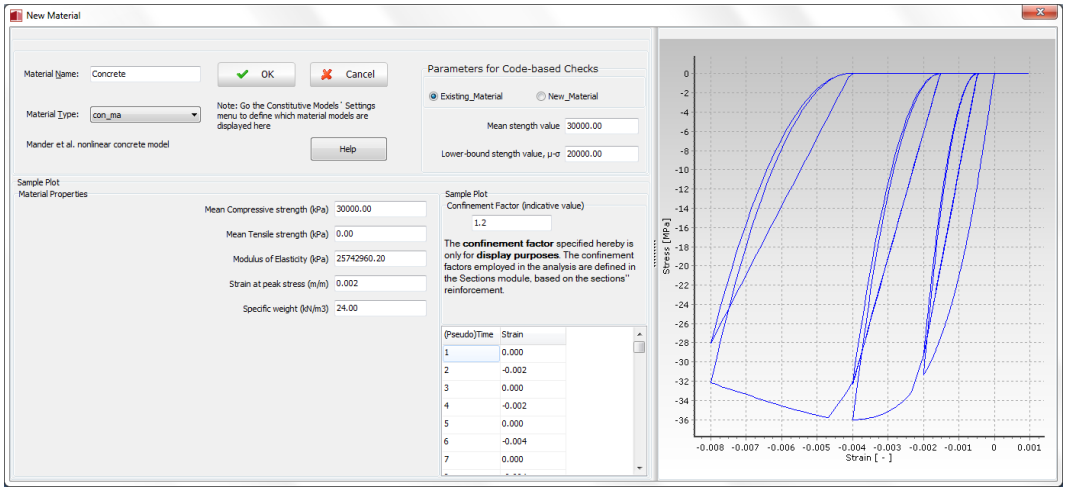
Once the type of analysis has been selected, you can start to create the model.

Pre-Processor – Materials

The **Materials** module is the first module you have to fill in. You have **two options** of inserting a new material: (i) clicking on the *Add Material Class* button in order to select a predefined material class or (ii) clicking on the *Add General Material* button if you are interested in defining all the material parameters.

In the present tutorial three materials are going to be defined in order to fully characterize each element's section. Hence, after selecting the *Add General Material* option (button on the left of the screen), you have to:

1. Assign the material's name (➔ Concrete);
2. Select the material type from the drop-down menu (➔ con_ma);
3. Define the material's properties (➔ default values -> Appendix C - Materials);
4. Define the Parameters needed for the Code-based Checks (➔ Existing_Material).



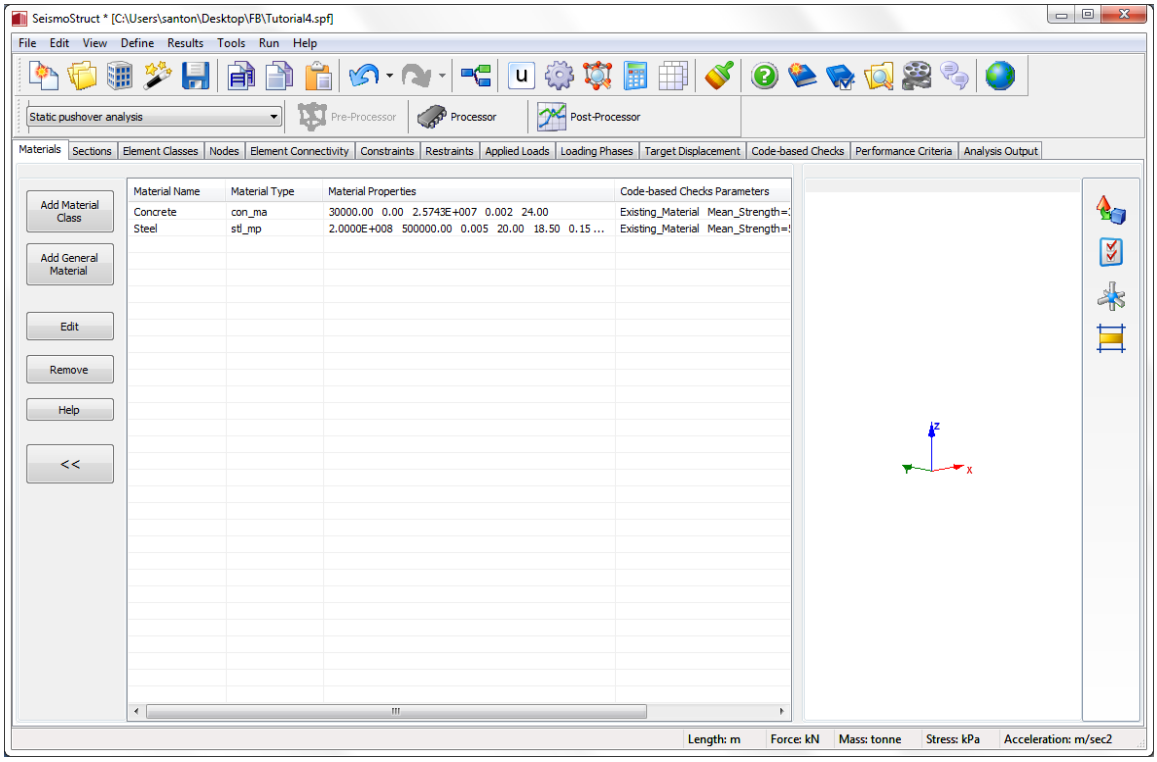
Concrete material

Now you have to repeat the same procedure in order to add the steel material:

1. Assign the material's name (➔ Steel);
2. Select the material type from the drop-down menu (➔ stl_mp);
3. Define the material's properties (➔ default values -> Appendix C - Materials);
4. Define the Parameters for Code-based Checks (➔ Existing_Material).

Note that from SeismoStruct v7.0 onwards, there is no longer a need for defining a third material for unconfined concrete, since the user has the possibility to define it through the Sections module, by introducing the transverse reinforcement.

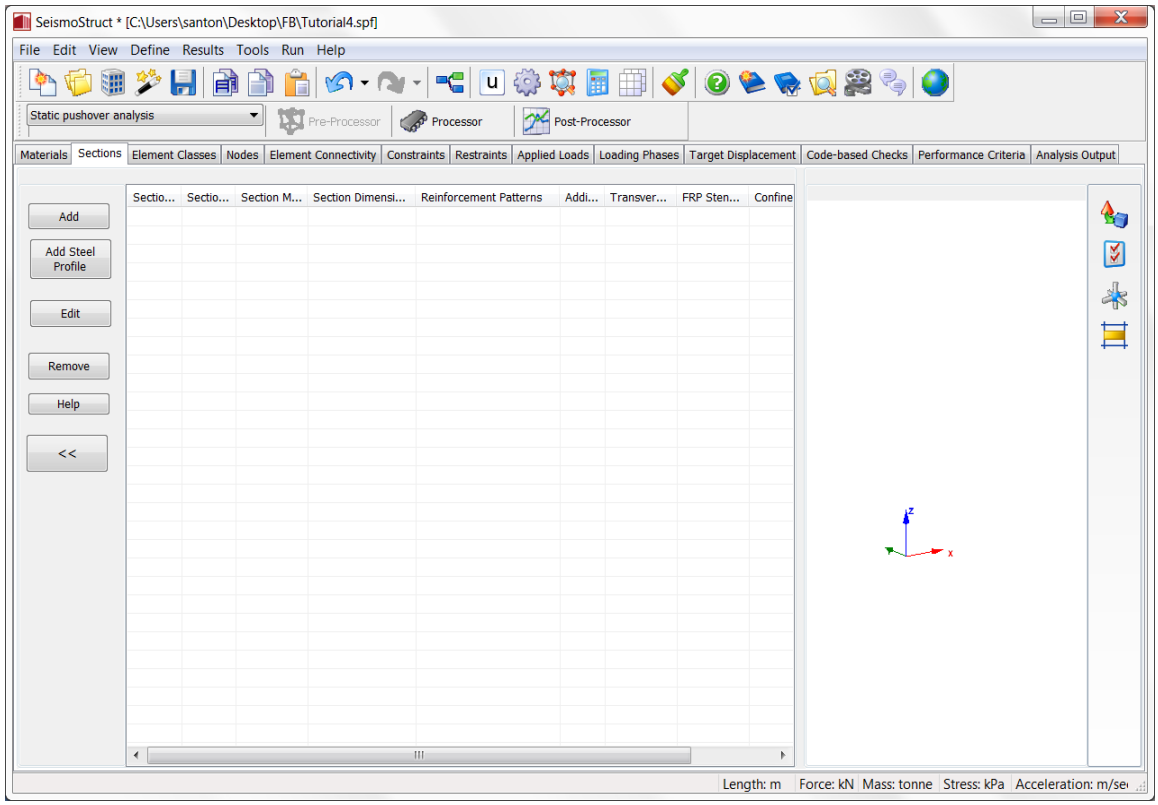
At the end, the **Materials** module will appear as follows:



Materials module

Pre-processor – Sections

Once the materials have been defined, move to the **Sections** module and click on the *Add* button in order to define the sections properties of structural elements.



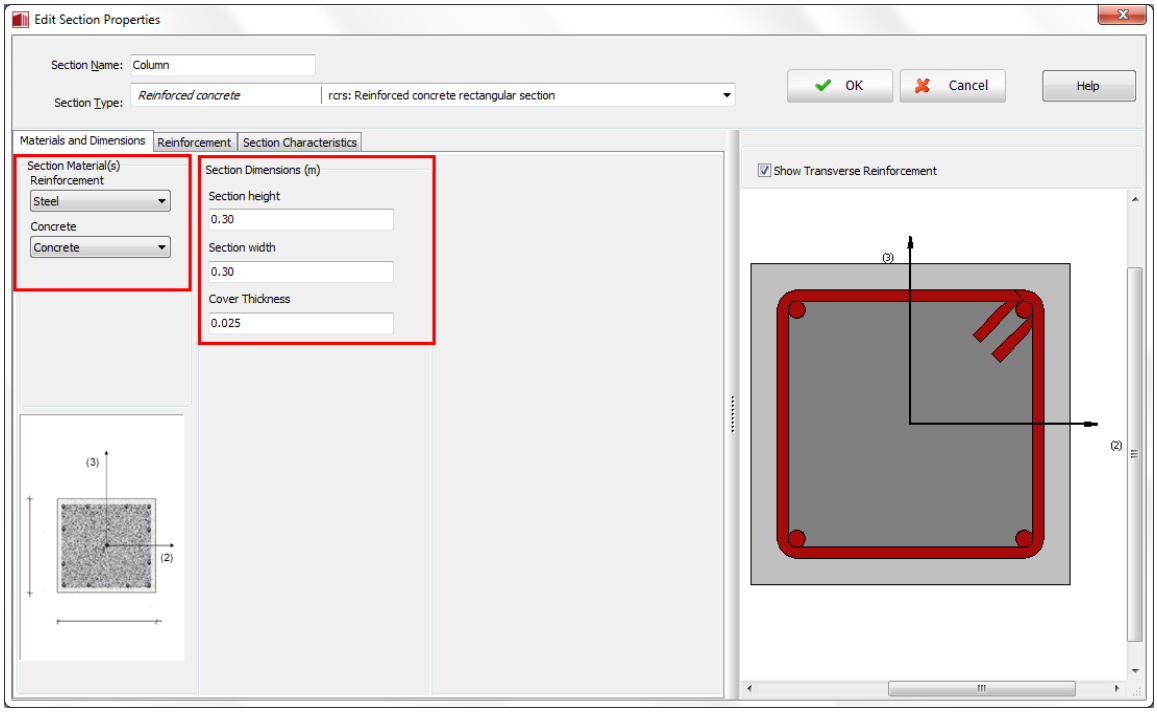
Sections Module

In this example, two different sections will be defined, one for the columns (called Column) and one for the beams (called Beam), by using the same section type (reinforced concrete rectangular section (rcrs)). For each section you have to:

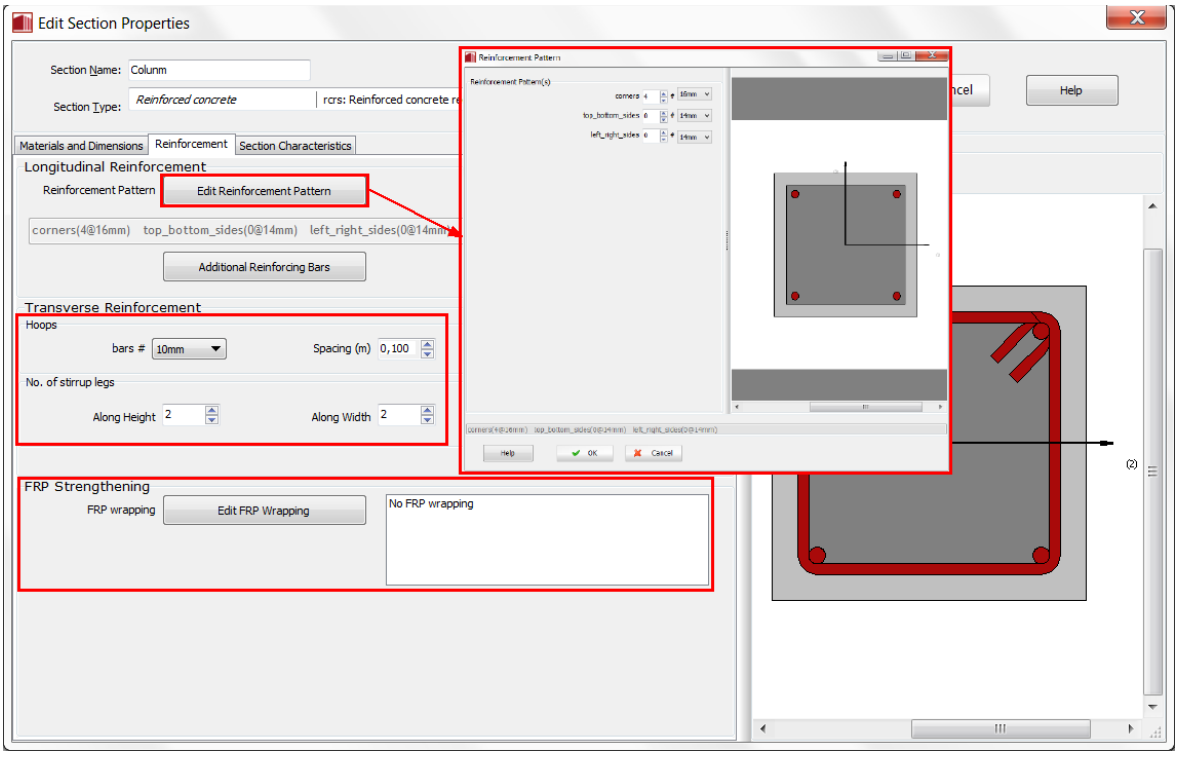
1. Assign the section name;
2. Select the section type from the drop-down menu;
3. Select the section materials from the drop-down menus;
4. Set the section dimensions;
5. Edit the reinforcement pattern;
6. Assign the FRP Wrapping

In the table below the section properties (dimensions and reinforcement) are summarized:

Section Properties	Column values	Beam values
Height	0.3 (m)	0.4
Width	0.3 (m)	0.3
Cover Thickness	0.025 (m)	0.025 (m)
Longitudinal Reinforcement	4 ϕ 16	8 ϕ 16
Transverse Reinforcement	ϕ 10/10	ϕ 10/10
FRP Strengthening	No FRP Wrapping	No FRP Wrapping

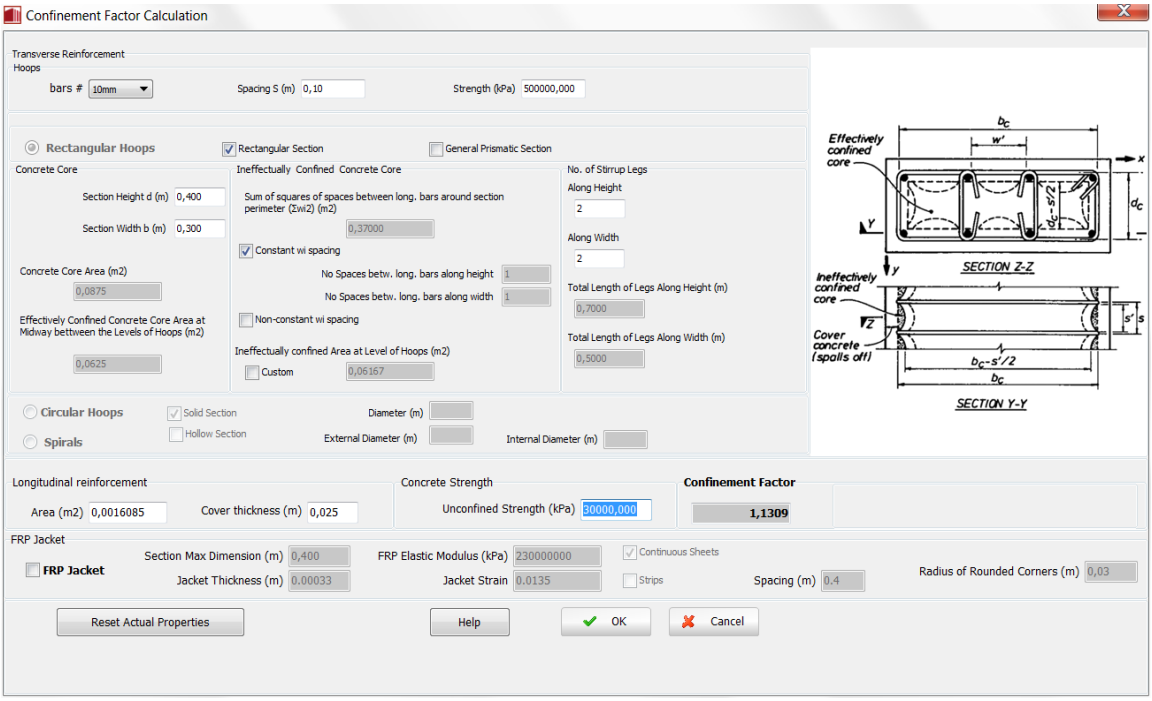


Column section (materials and dimensions)

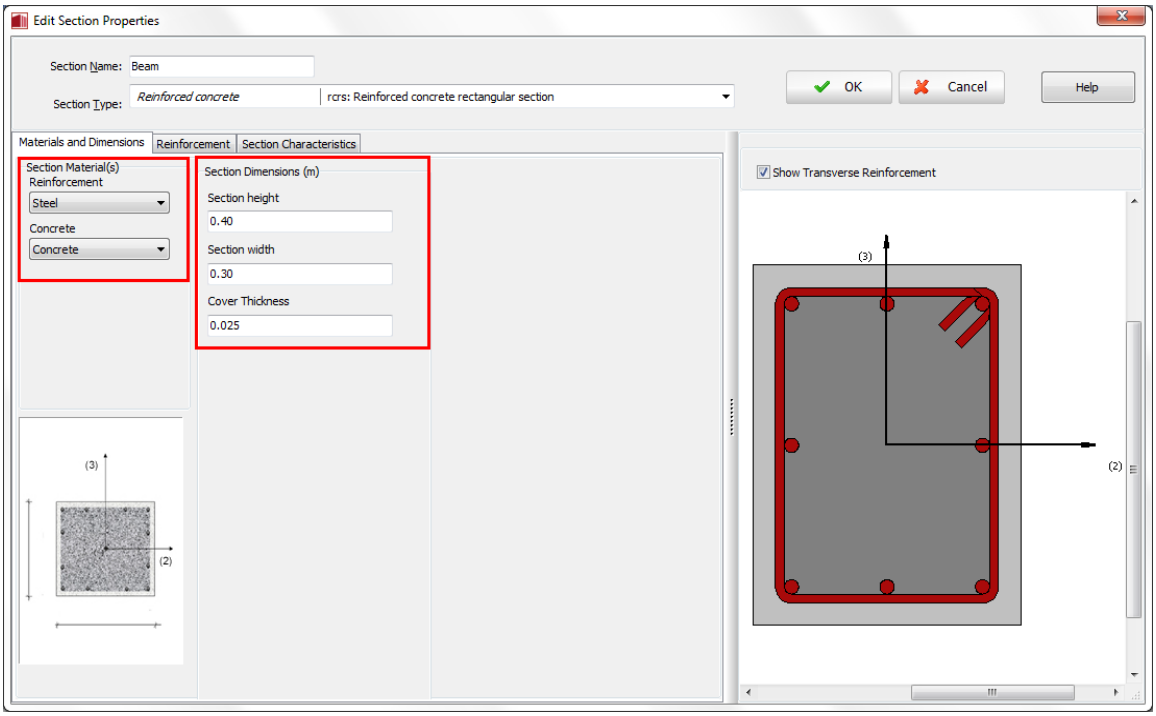


Column section (reinforcement pattern)

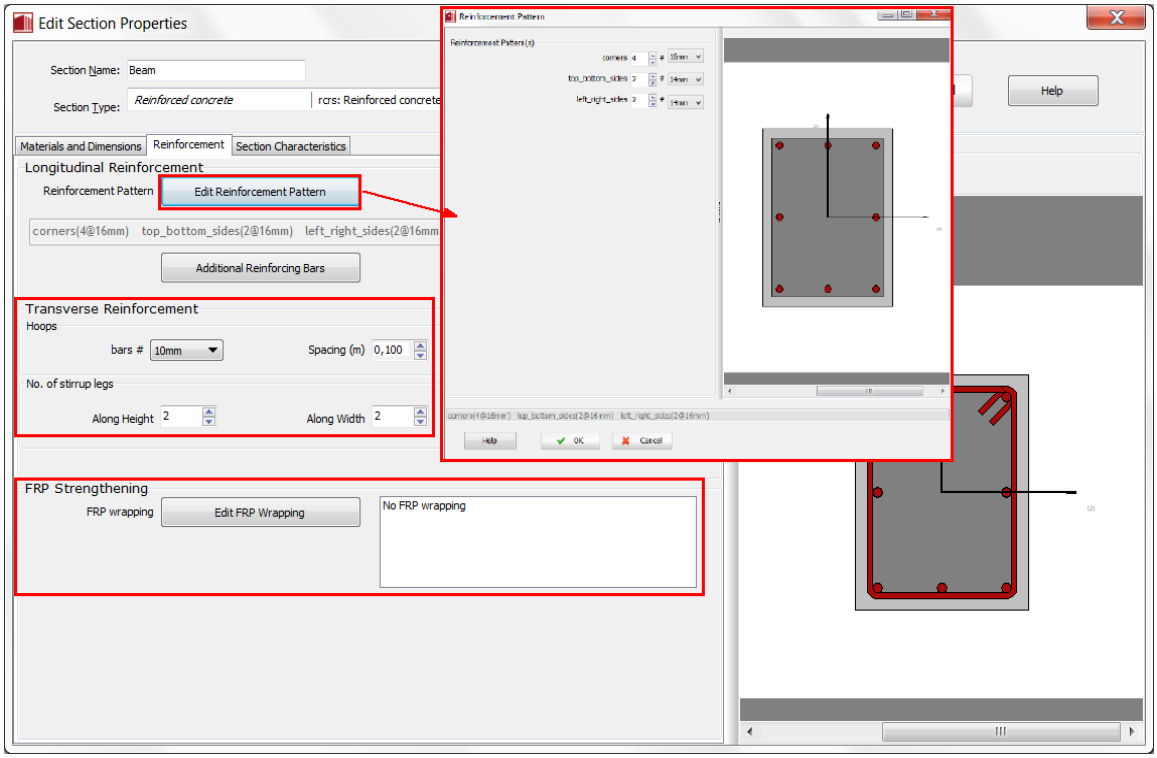
Users can select from Section Characteristics pattern if the confinement factor will be calculated automatically from the transverse reinforcement, or will calculate it with more details.



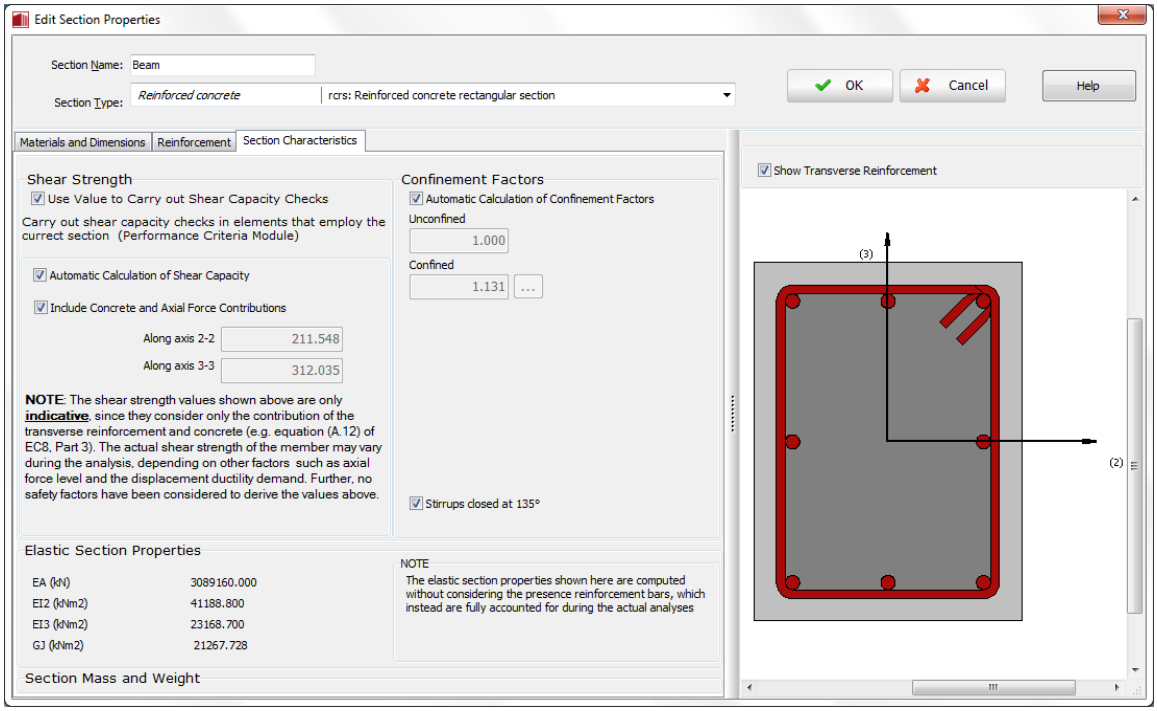
Confinement Factor Calculation pattern



Beam section (materials and dimensions)



Beam section (reinforcement pattern)



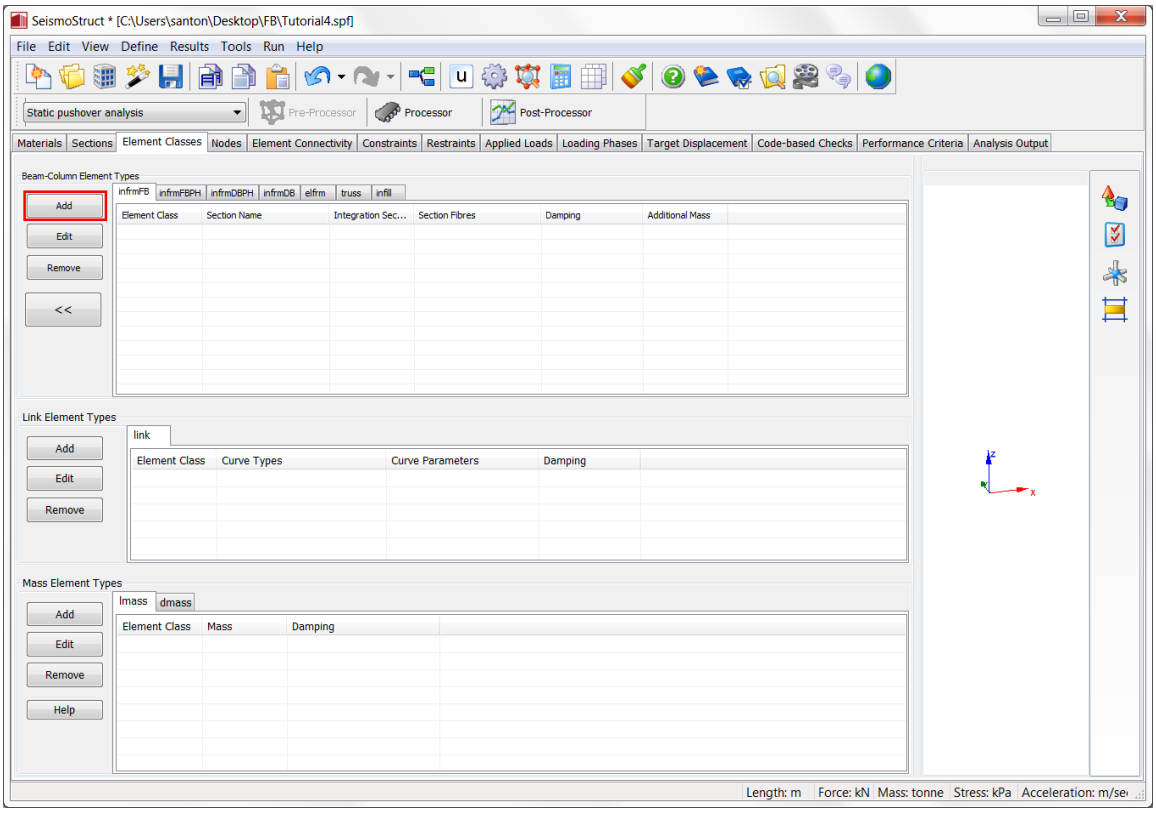
Beam section (Section Characteristics)

NOTE 1: The shear capacity shown in the Sections module is calculated using the expression of EC8-Part 3. It is noted that such value is only indicative, since it considers only the contribution of the transverse reinforcement, but not other factors such as the axial force level or the displacement ductility demand. The Lv ratio of the equation is calculated employing an assumed element length as the minimum of 6*(MaxDim) and 12*(MinDim), where (MaxDim) and (MinDim) are the maximum and minimum section dimension respectively.

NOTE 2: The EA, EI & GJ values shown in this module are merely indicative (i.e not used in the analysis) and calculated using the elastic material properties of the main section material (i.e. concrete in R/C sections). No discretisation of the section in monitoring points takes place in the Pre-Processor (as happens instead during the analysis).

Pre-processor – Element Classes

For each section described above, you have to define an element class in the **Element Classes** module. Hence, click on the *Add* button related to the *Beam-Column Element Types*: a dialogue window will be opened.

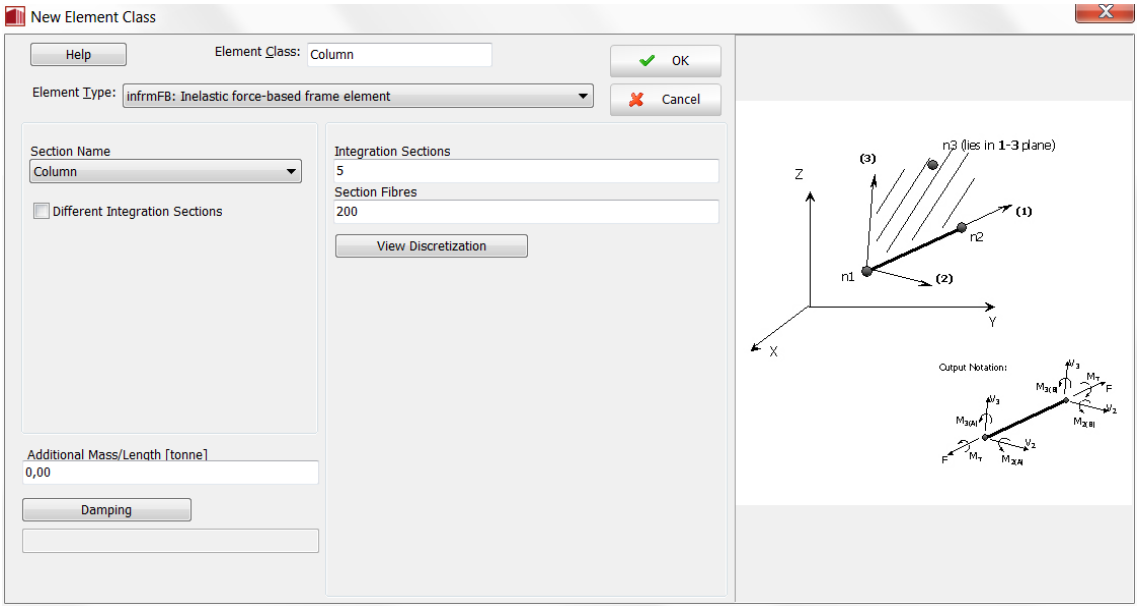


Element Classes module

In the dialogue window you have to:

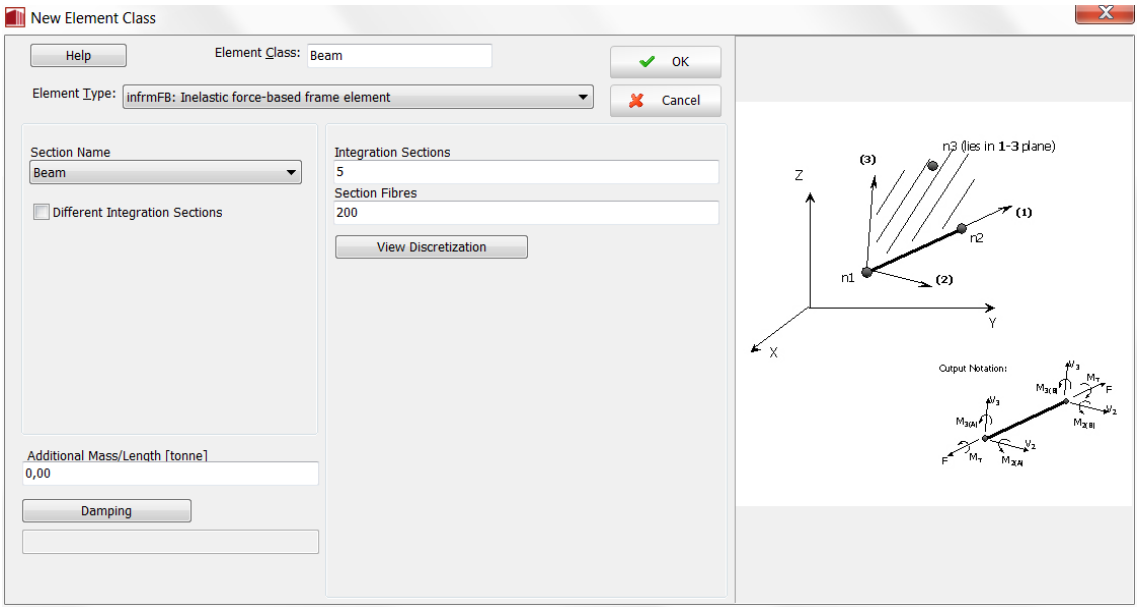
1. Assign a name to the element class (➔ Column);
2. Select the element type from the drop-down menu (➔ infrmFB element);
3. Select the corresponding section name from the drop-down menu (➔ Column);
4. Set the number of integration sections (➔ 5) and section fibres (➔ 200);
5. Assign additional mass/length (➔ No mass/length is assigned);

6. Define the element-specific damping (➔ no element specific damping is applied, which means that the damping defined in the Project Settings will be employed)



Definition of the Element Classes (Column)

Repeat the same procedure in order to create the class for the beam element.

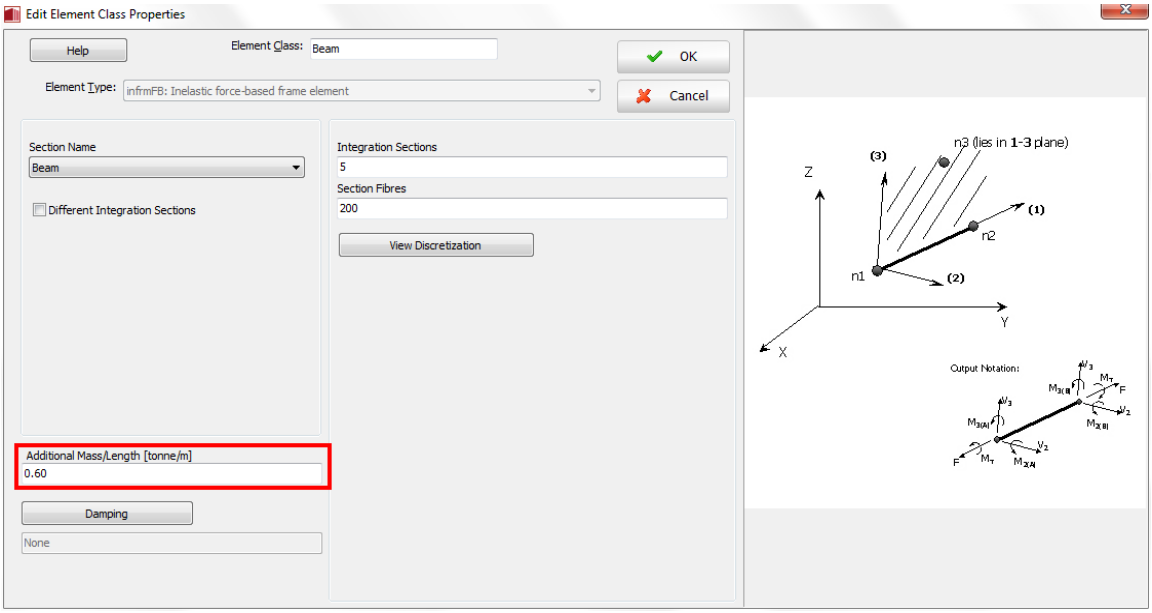


Definition of the Element Classes (Beam)

In order to take into account vertical load acting on the beam elements, you may assign an additional mass/length to the beam element class. For this tutorial let's assume a value of 0.6 tonne/m.

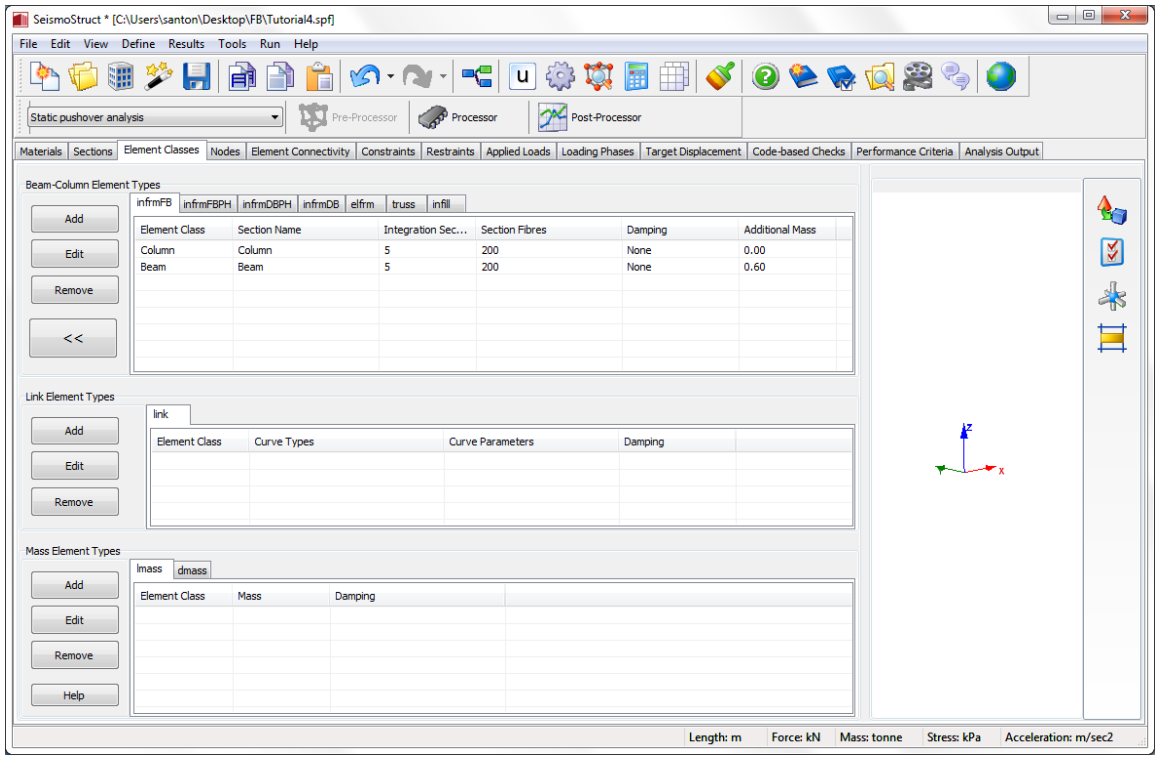
NOTE 1: The additional mass/length will be converted to loads only by checking the *'Loads (ONLY in the gravity direction) are derived from Masses, based on the g value* ' or *'Loads are derived from Masses in any translational direction, according to user-defined coefficients*' option in the Project Settings panel (Project Settings -> Gravity & Mass).

NOTE 2: The additional mass/length may be defined also by using the *distributed mass element* (dmass).



Beam element class (additional mass)

At the end, the **Element Classes** module will appear as follows:



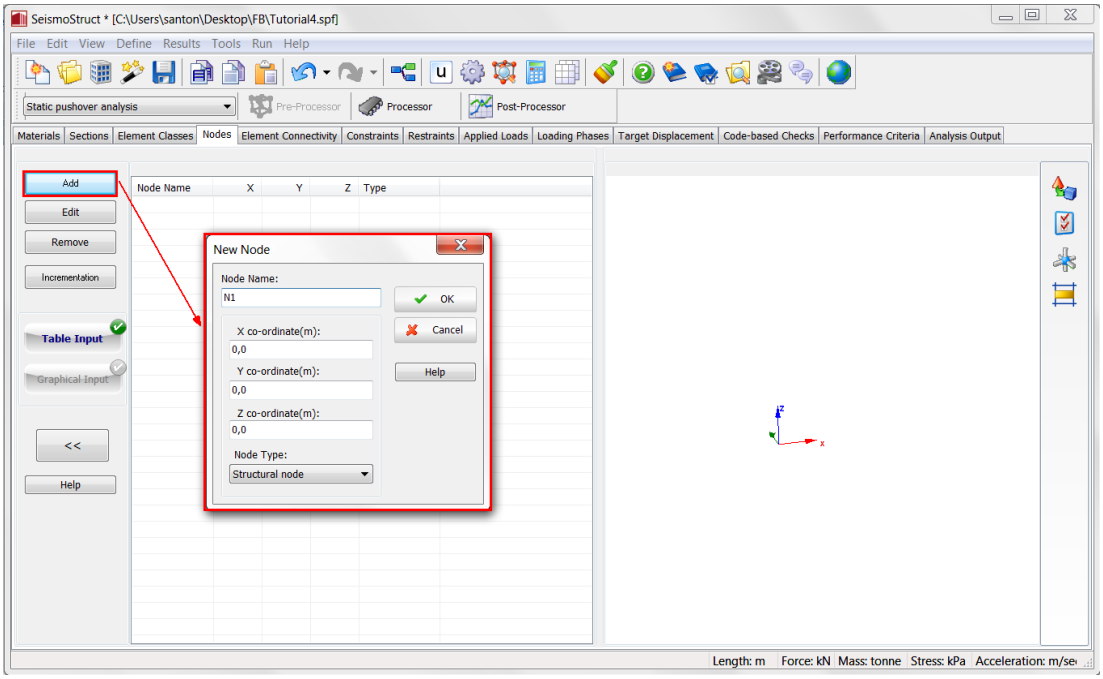
Element Classes module

Pre-processor – Nodes

At this point it is necessary to define the geometry of the structure. Hence, move to the **Nodes** module in order to define the nodes.

> **NOTE: In this tutorial you are going to define just one structural node. The other nodes will be created through the *Incrementation* function.**

The first node you are going to define is a **structural node**. Click on the *Add* button. Then, in the new node dialogue window (i) assign the node name (➔ N1), (ii) introduce the coordinates (➔ x=0, y=0, z=0) and (iii) select the node type from the drop-down menu (➔ structural node).

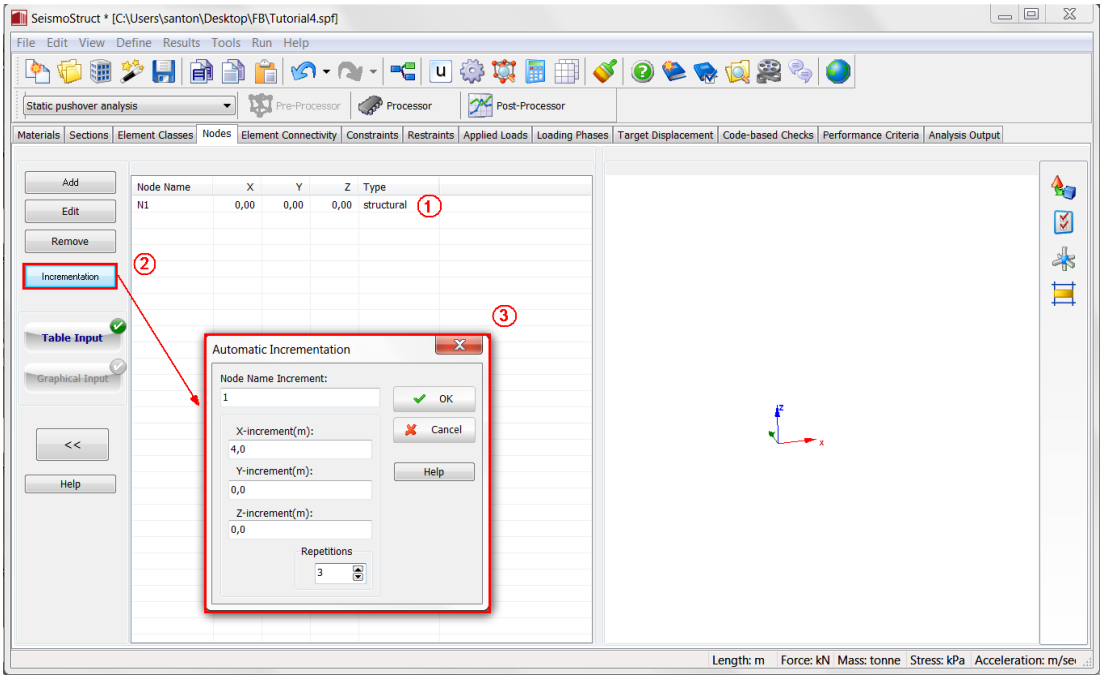


Nodes module and definition of a new node

In order to create the other nodes, you have to:

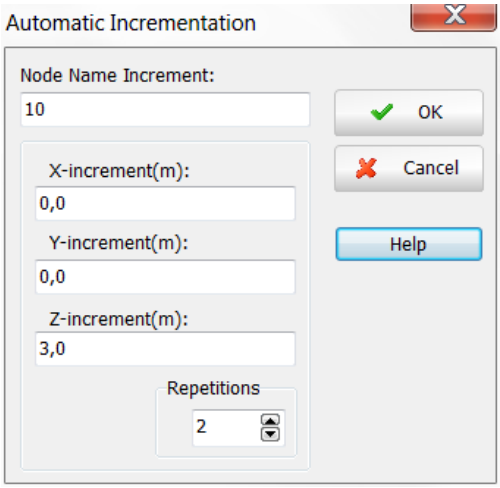
1. Select the node you previously defined;
2. Click on the *Incrementation* button;
3. Assign the node name increment (➔ 1);
4. Introduce the increment (➔ 4) in the right direction (➔ X-increment);
5. Define the number of repetitions (➔ 3).

You will obtain all the base nodes with Y = 0 (see figure below).



Incrementation facility

Now, in order to increment the nodes in Z-direction, (i) select the nodes you previously defined, (ii) click again on the *Incrementation* button, (iii) assign the node name increment (➔ 10), (iv) introduce the increment (➔ 3) in Z-direction, (v) define the number of repetitions (➔ 2).

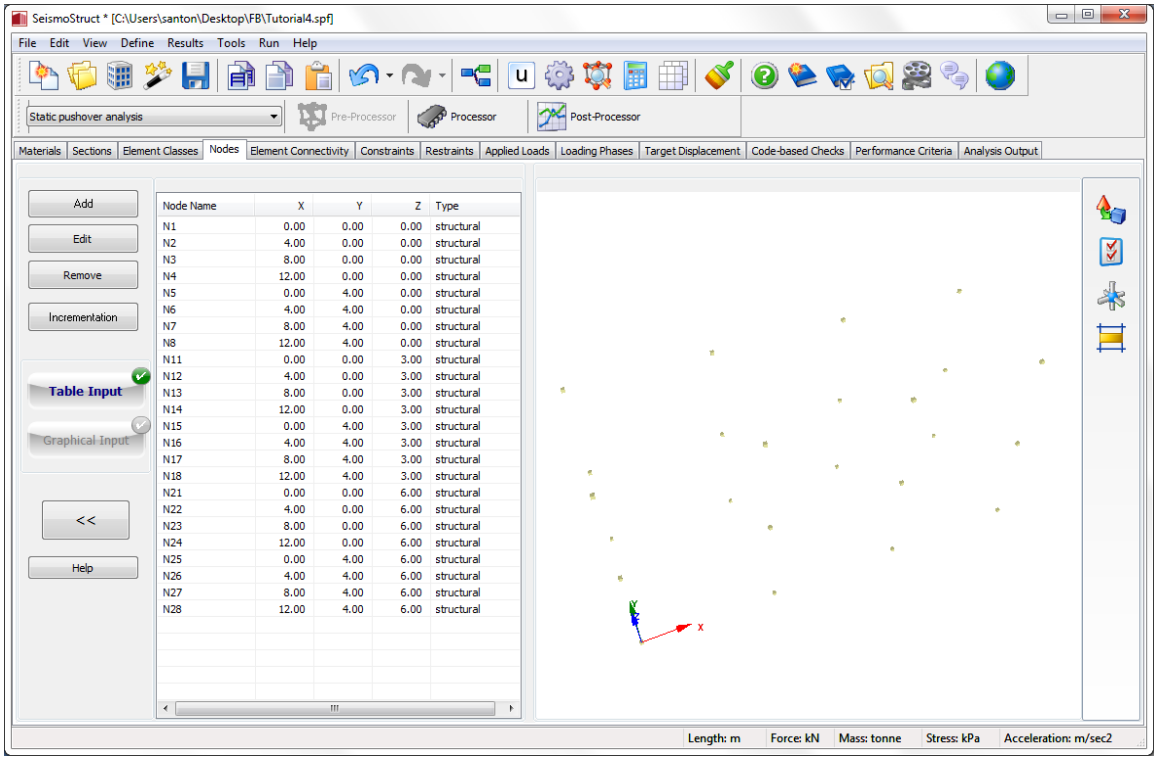


Incrementation in Z-direction

Repeat the steps above in order to define the remaining nodes. In the table below the coordinates of all the structural nodes are summarized:

Node Name	X	Y	Z	Type
N1	0	0	0	structural
N2	4	0	0	structural
N3	8	0	0	structural
N4	12	0	0	structural
N11	0	0	3	structural

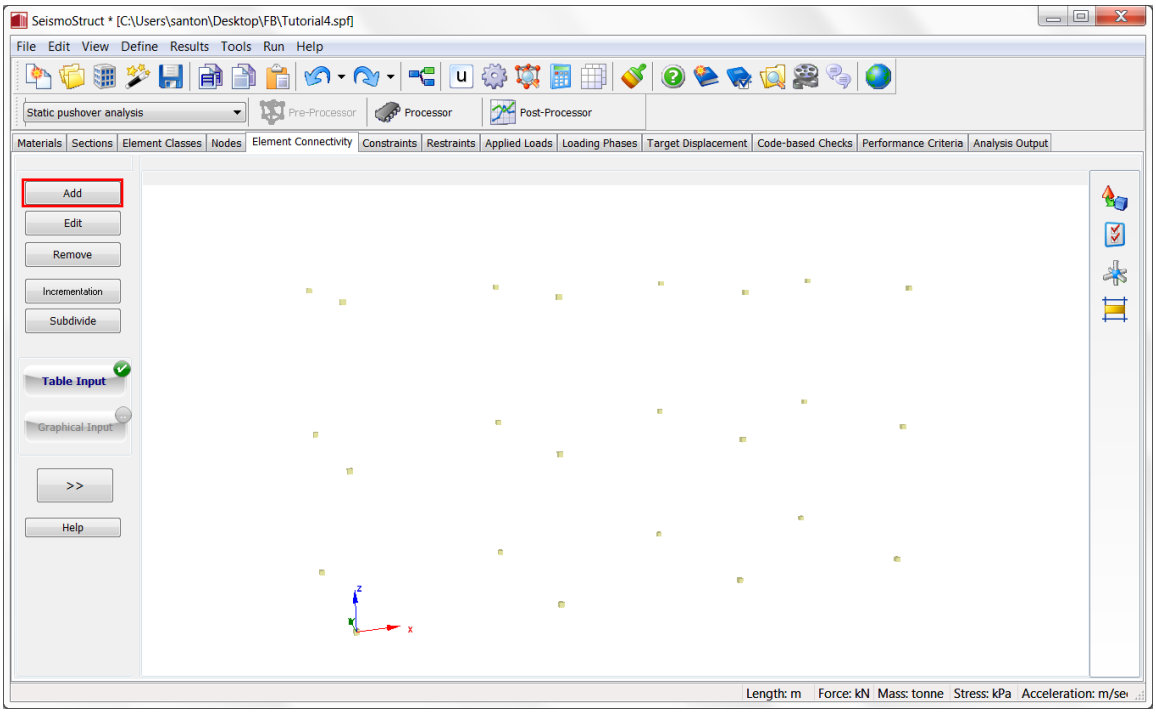
Node Name	X	Y	Z	Type
N12	4	0	3	structural
N13	8	0	3	structural
N14	12	0	3	structural
N21	0	0	6	structural
N22	4	0	6	structural
N23	8	0	6	structural
N24	12	0	6	structural
N5	0	4	0	structural
N6	4	4	0	structural
N7	8	4	0	structural
N8	12	4	0	structural
N15	0	4	3	structural
N16	4	4	3	structural
N17	8	4	3	structural
N18	12	4	3	structural
N25	0	4	6	structural
N26	4	4	6	structural
N27	8	4	6	structural
N28	12	4	6	structural



Structural nodes

Pre-processor – Element Connectivity

Now, move to the **Element Connectivity** module in order to add the structural elements (i.e. columns and beams). The first element you are going to define is a column. Hence, click on the *Add* button.

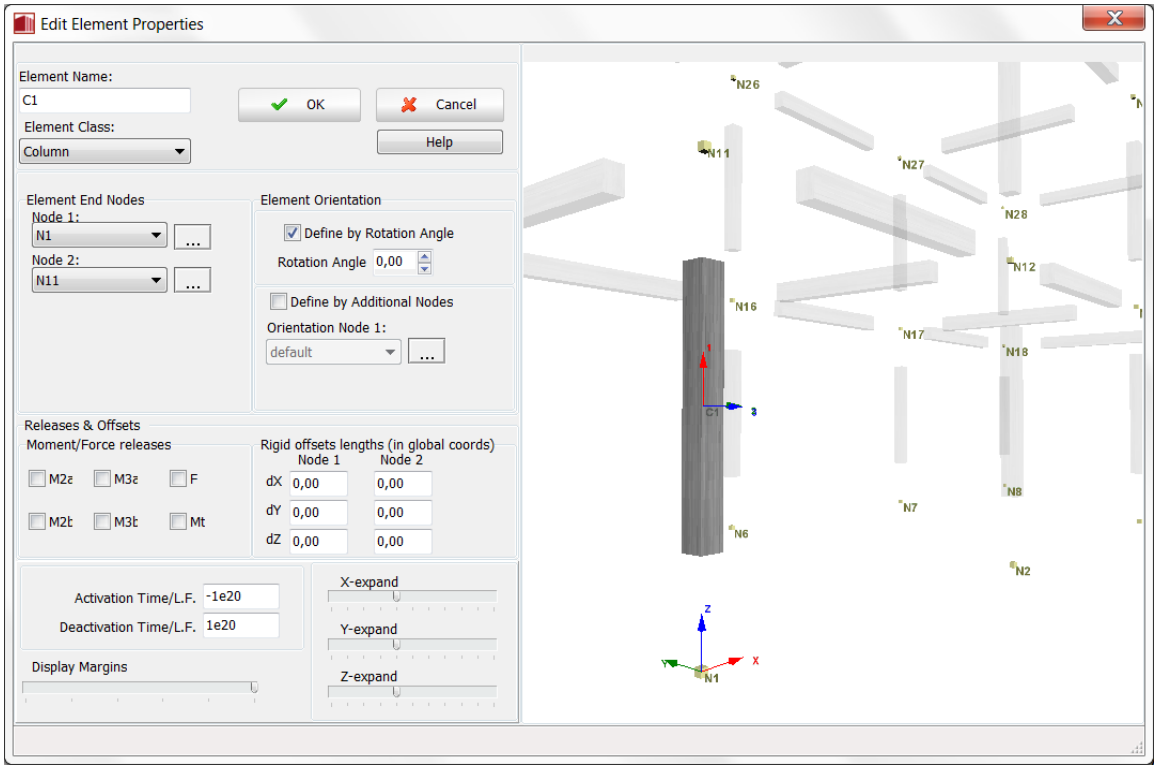


Element Connectivity module

In the new element dialogue window you have to:

1. Assign the element name (➔ C1);
2. Select the element class from the drop-down menu (➔ Column);
3. Select, respectively, the first (structural) node (➔ N1), the second (structural) node (➔ N11) and the orientation of the element (defining a rotation angle equal to 0 ➔ default option), as shown in the figure below.

NOTE: In this tutorial, you will use the *Table Input* instead of the *Graphical Input* mode in order to generate the new elements.



Definition of a new element

Repeat the procedure described above in order to define all the other elements.

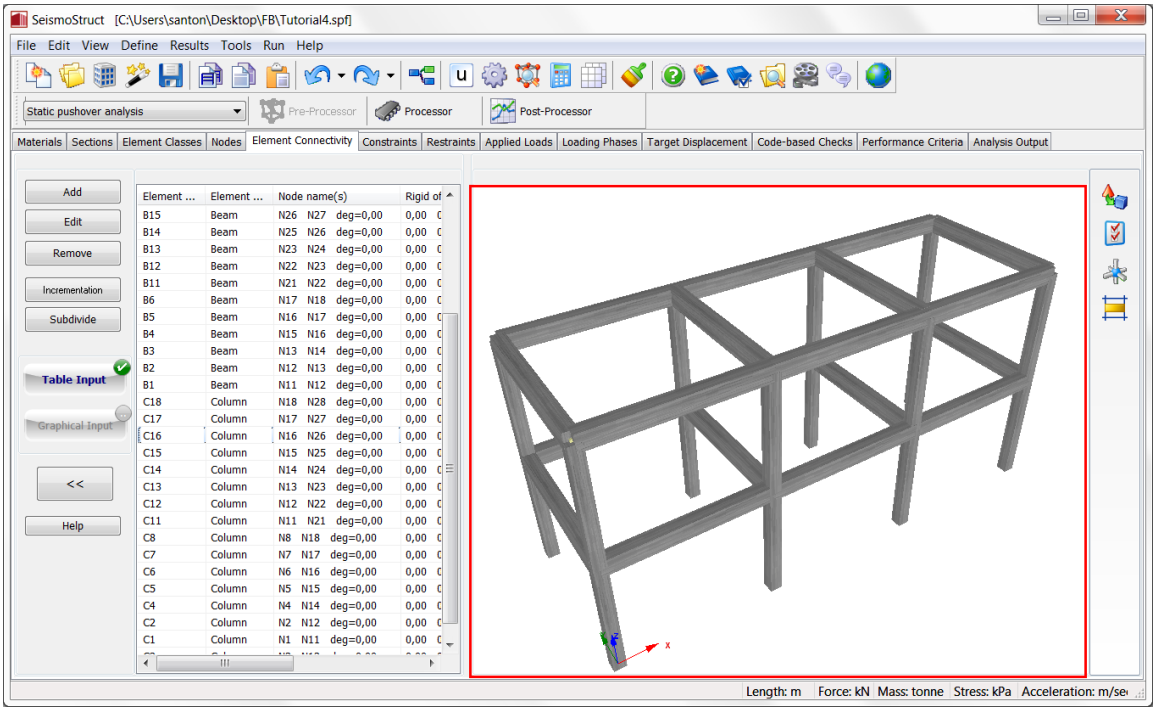
> **NOTE: As in the case of nodes, you may use the *Incrementation* facility in order to generate the new elements.**

In the table below all the elements are summarized:

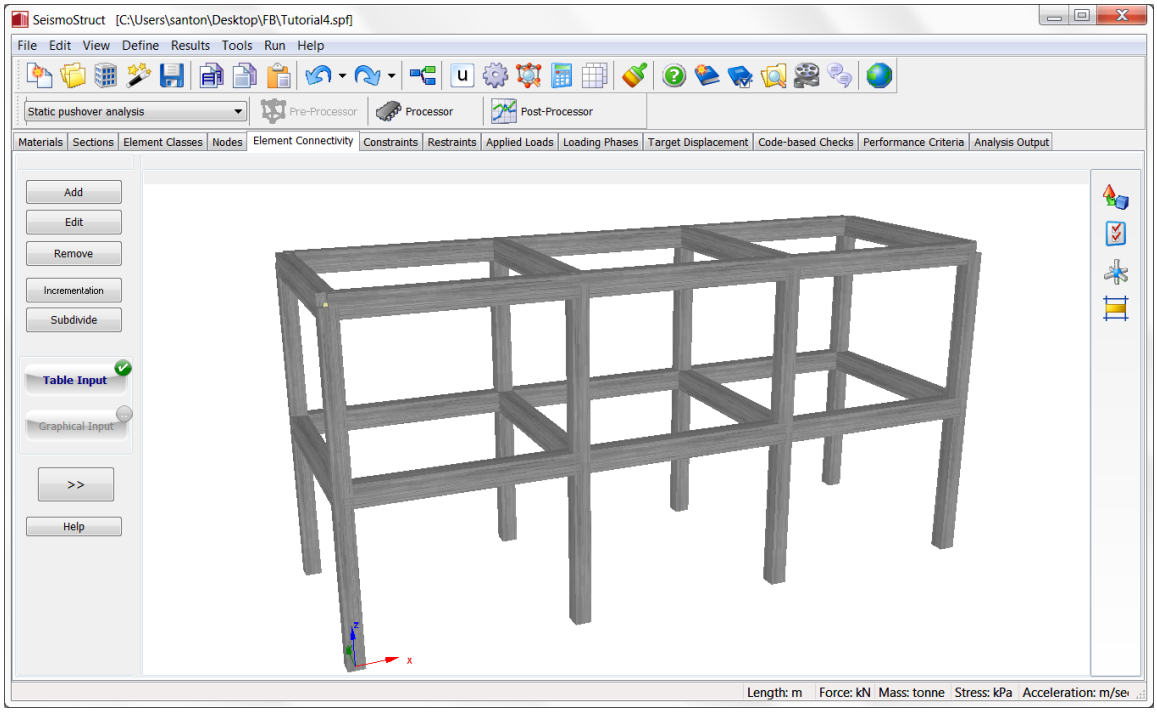
Element Name	Element Class	Nodes
C1	Column	N1 N11 deg=0.0
C2	Column	N2 N12 deg=0.0
C3	Column	N3 N13 deg=0.0
C4	Column	N4 N14 deg=0.0
C5	Column	N5 N15 deg=0.0
C6	Column	N6 N16 deg=0.0
C7	Column	N7 N17 deg=0.0
C8	Column	N8 N18 deg=0.0
C11	Column	N11 N21 deg=0.0
C12	Column	N12 N22 deg=0.0
C13	Column	N13 N23 deg=0.0
C14	Column	N14 N24 deg=0.0
C15	Column	N15 N25 deg=0.0
C16	Column	N16 N26 deg=0.0
C17	Column	N17 N27 deg=0.0
C18	Column	N18 N28 deg=0.0

Element Name	Element Class	Nodes
B1	Beam	N11 N12 deg=0.0
B2	Beam	N12 N13 deg=0.0
B3	Beam	N13 N14 deg=0.0
B4	Beam	N15 N16 deg=0.0
B5	Beam	N16 N17 deg=0.0
B6	Beam	N17 N18 deg=0.0
B11	Beam	N21 N22 deg=0.0
B12	Beam	N22 N23 deg=0.0
B13	Beam	N23 N24 deg=0.0
B14	Beam	N25 N26 deg=0.0
B15	Beam	N26 N27 deg=0.0
B16	Beam	N27 N28 deg=0.0
B7	Beam	N11 N15 deg=0.0
B8	Beam	N12 N16 deg=0.0
B9	Beam	N13 N17 deg=0.0
B10	Beam	N14 N18 deg=0.0
B17	Beam	N21 N25 deg=0.0
B18	Beam	N22 N26 deg=0.0
B19	Beam	N23 N27 deg=0.0
B20	Beam	N24 N28 deg=0.0

At this point, the whole structure has been defined. Now, in the *3D Model* window (on the right of the screen) you can check your model by zooming, rotating, and moving the 3D plot.



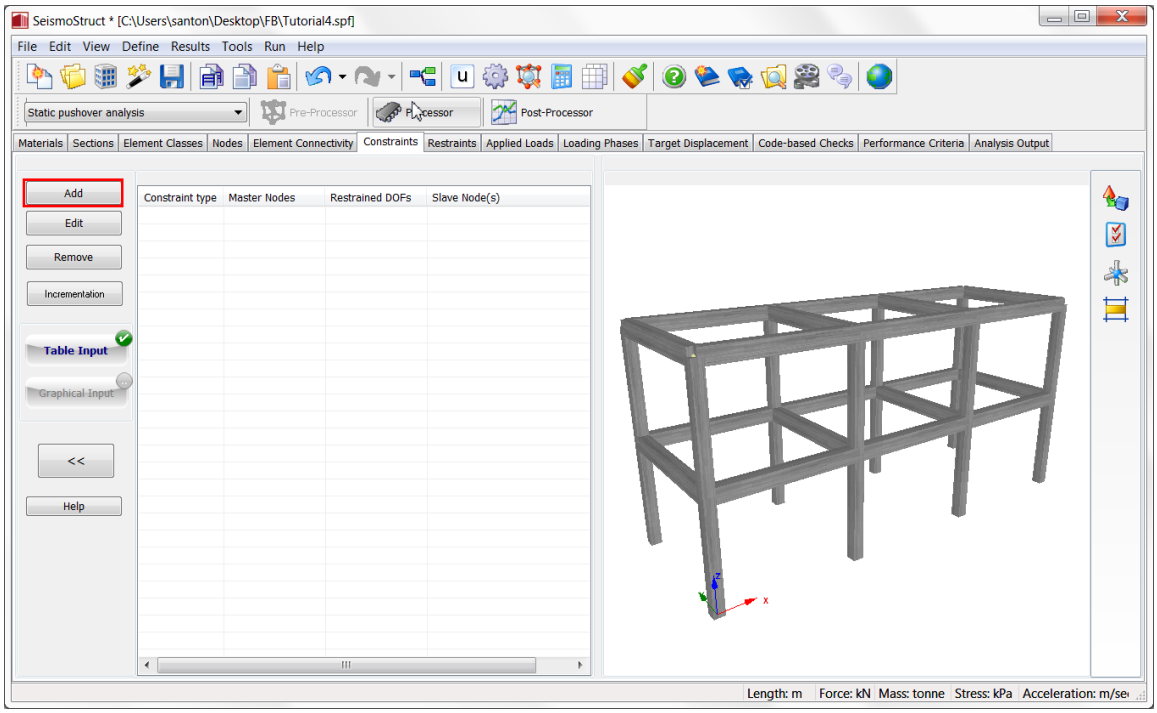
3D Model window



3D Model (full screen)

Pre-processor – Constraints

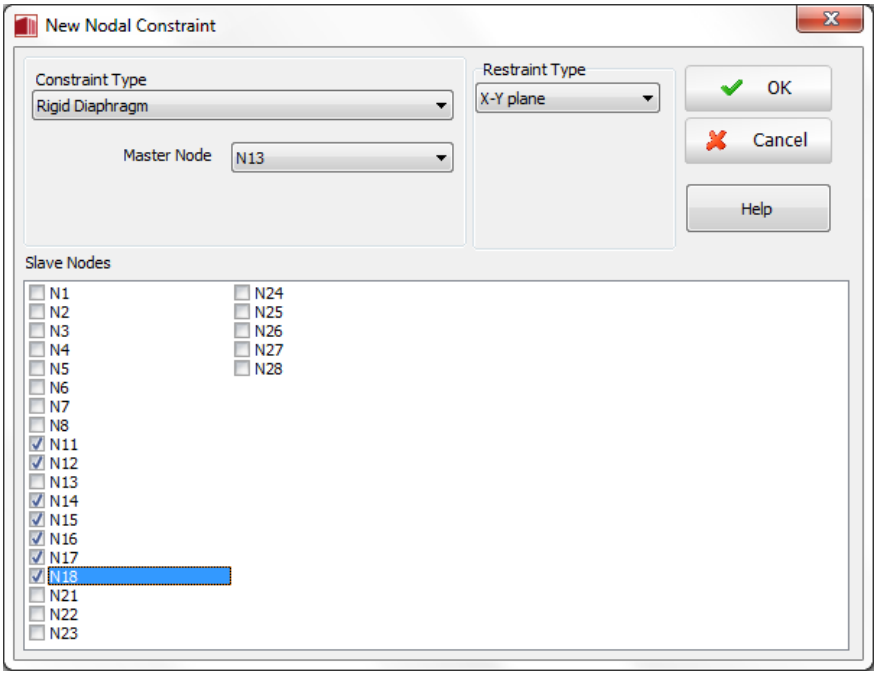
Now you have to define the constraining conditions of the structure. Two rigid diaphragms need to be created. Hence, go to the **Constraints** module and click on the *Add* button.



Constraints module

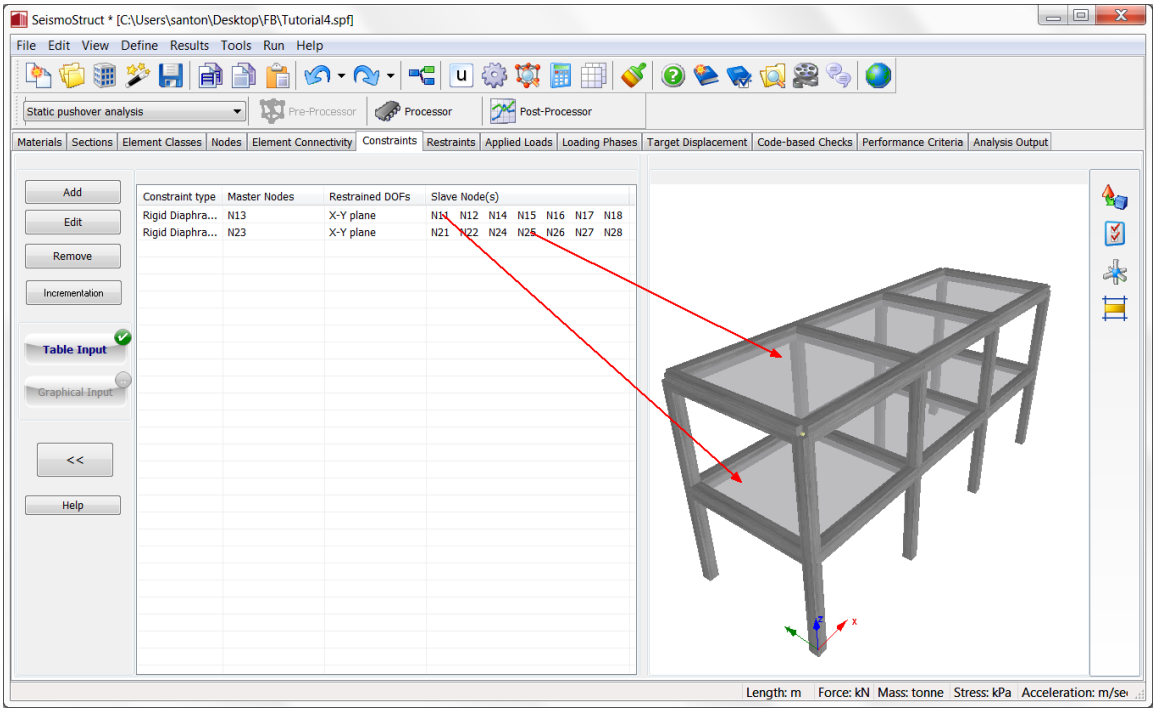
In the new nodal constraint window you have to:

1. Select the constraint type from the drop-down menu (➔ rigid diaphragm);
2. Select the restraint type (➔ X-Y plane);
3. Choose the associated master node from the drop-down menu (➔ N13);
4. Select the slave nodes by ticking the corresponding box.



New Constraints window

Repeat the same procedure in order to define the rigid diaphragm that models the second floor. At the end, the **Constraints** module will appear as follows:

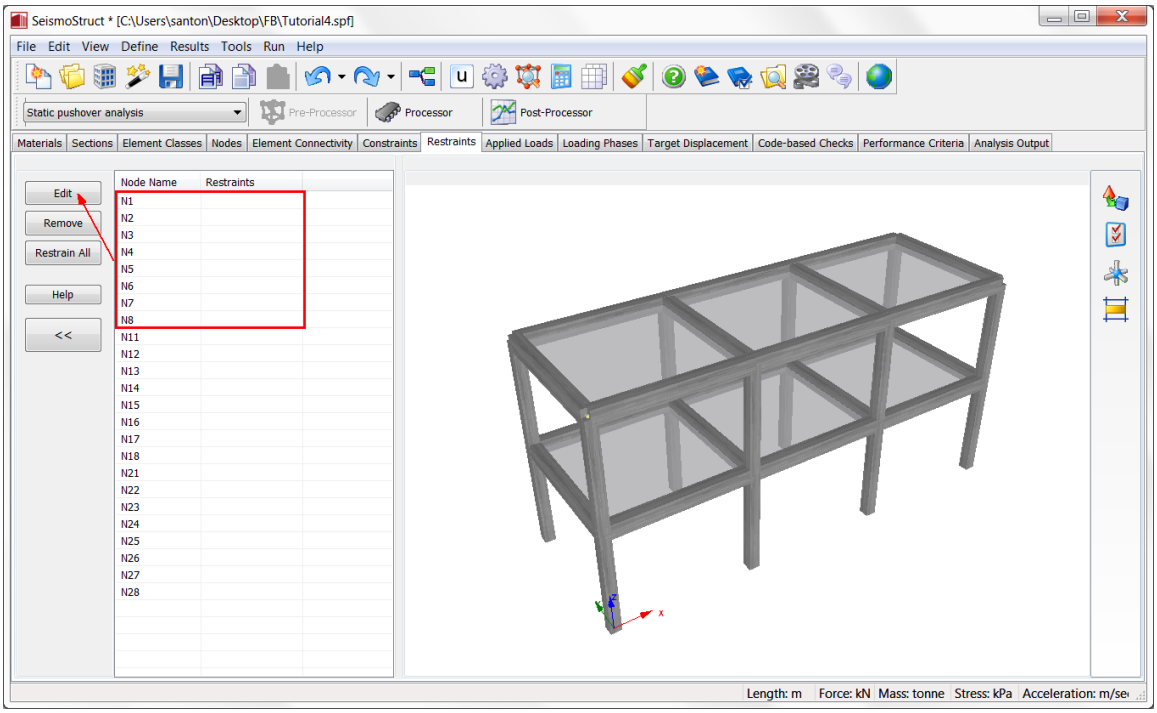


Constraints

NOTE: As in the case of elements, you may use the *Incrementation* facility in order to generate the new rigid diaphragm.

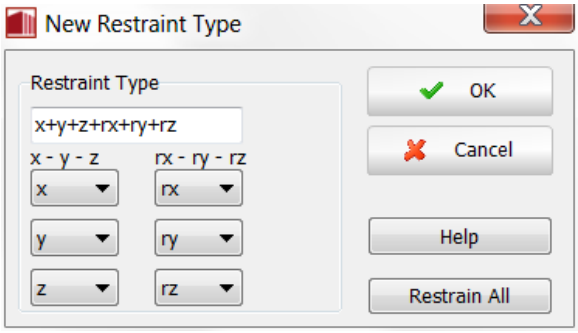
Pre-processor – Restraints

The last step related to the "structural geometry" is the definition of the restraining conditions. In this tutorial you have to fully restrain the base nodes of the structure. To do this, (i) move to the **Restraints** module, (ii) select the nodes you wish to restrain (-> base nodes) and (iii) click on the *Edit* button.



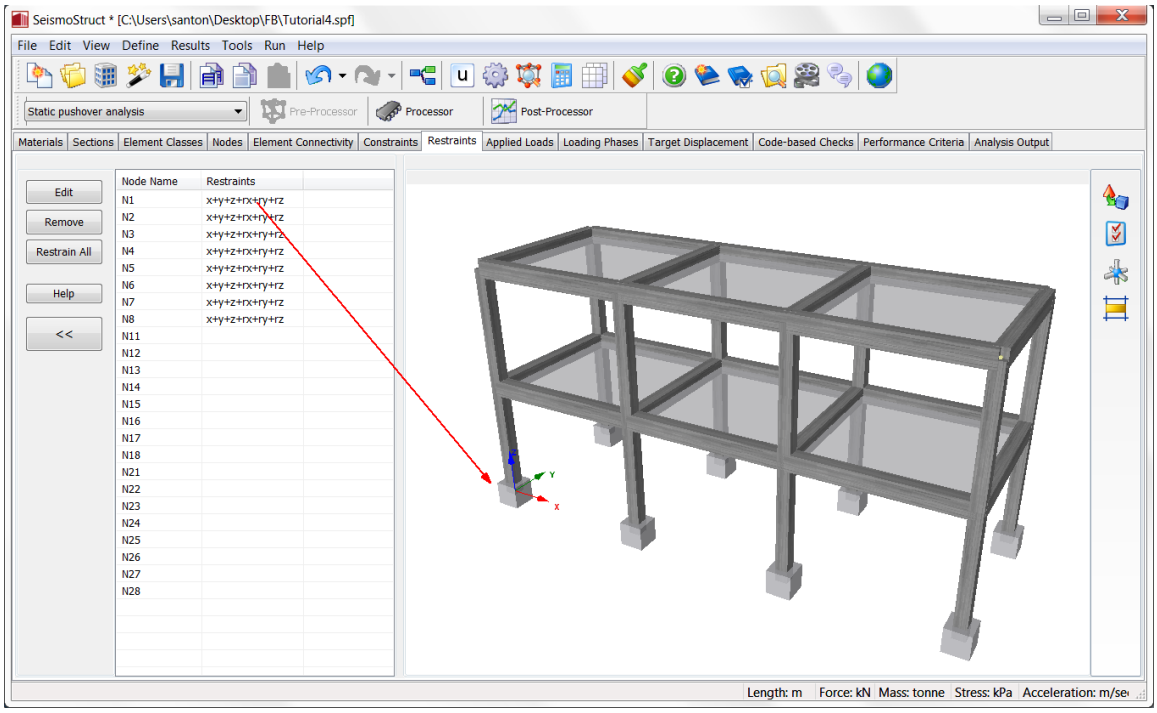
Restraints module

In the new window click on the *Restrain All* button.



New Restraint window

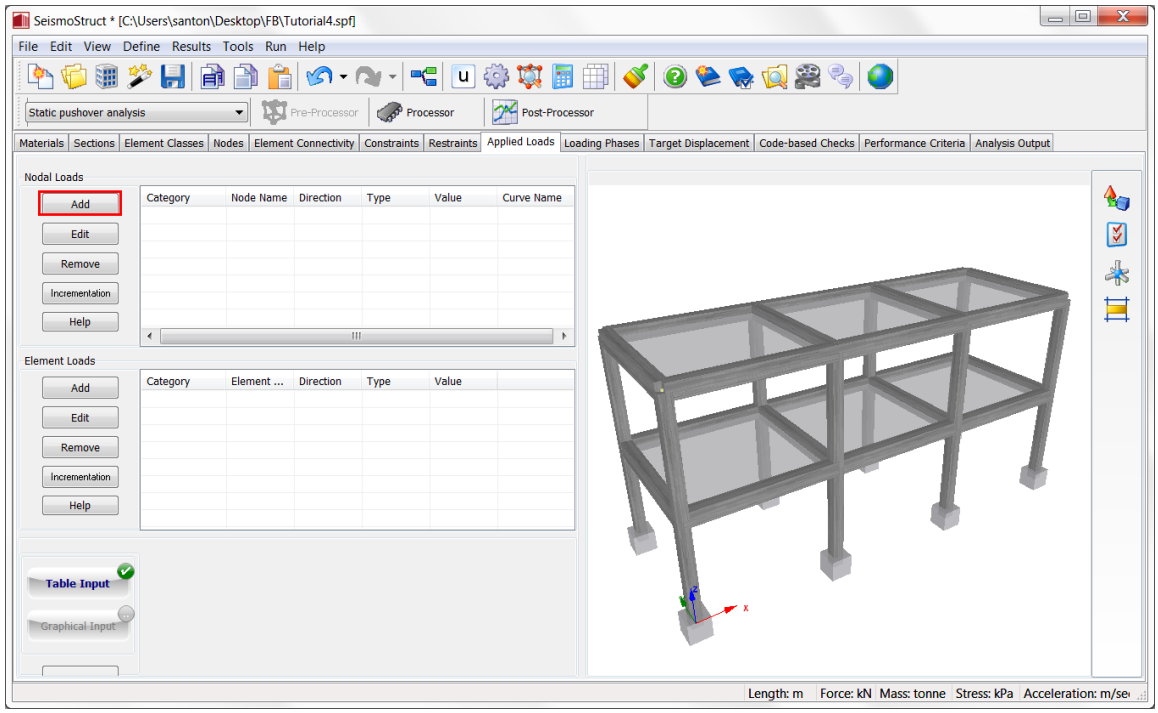
The **Restraints** module will appear as follows:



Restraints

Pre-processor – Applied Loads

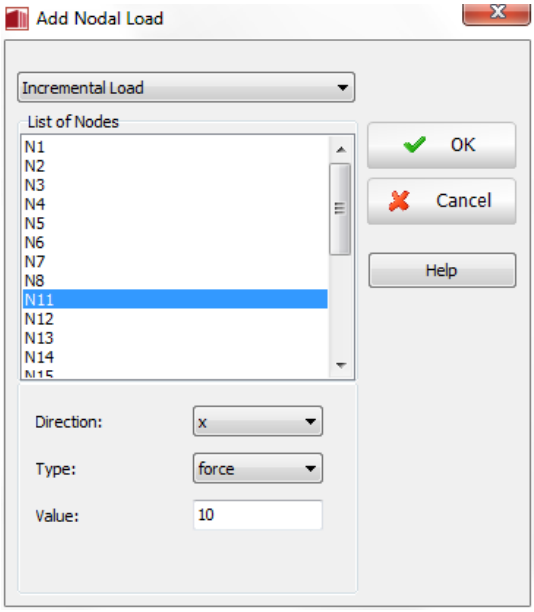
Since a pushover analysis needs to be carried out, you have to apply the appropriated loads (i.e. incremental loads) to the structural model. Hence, go to the **Applied Loads** module and click on the *Add* button for Load Curves.



Applied Loads Module

In the new window you have to:

1. Select the load category from the drop-down menu (➔ Incremental Load);
2. Specify the associated node (➔ N11);
3. Select the load direction from the drop-down menu (➔ X);
4. Select the load type from the drop-down menu (➔ force);
5. Specify the nominal value (➔ 10).



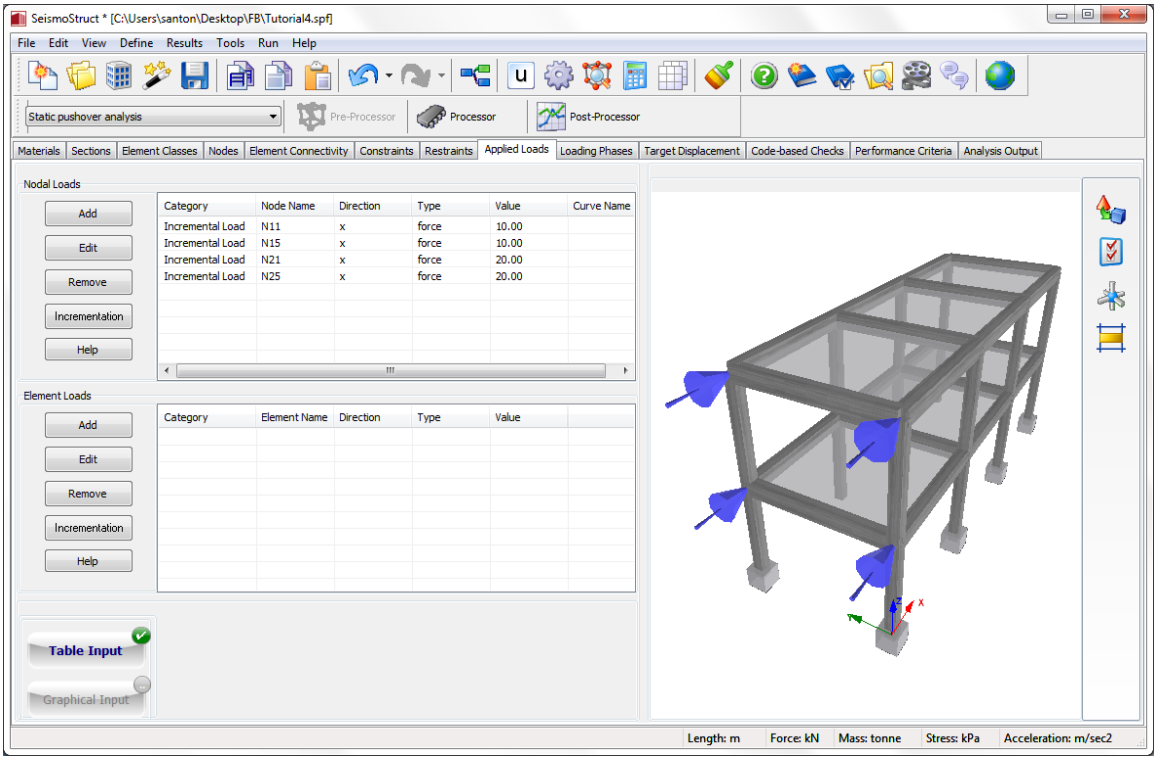
Applied Nodal Load window

Repeat the same procedure in order to apply the other incremental loads.

REMEMBER! The magnitude of a load at any step is given by the product of its nominal value, defined by the user, and the current load factor, which is updated in automatic or user-defined fashion.

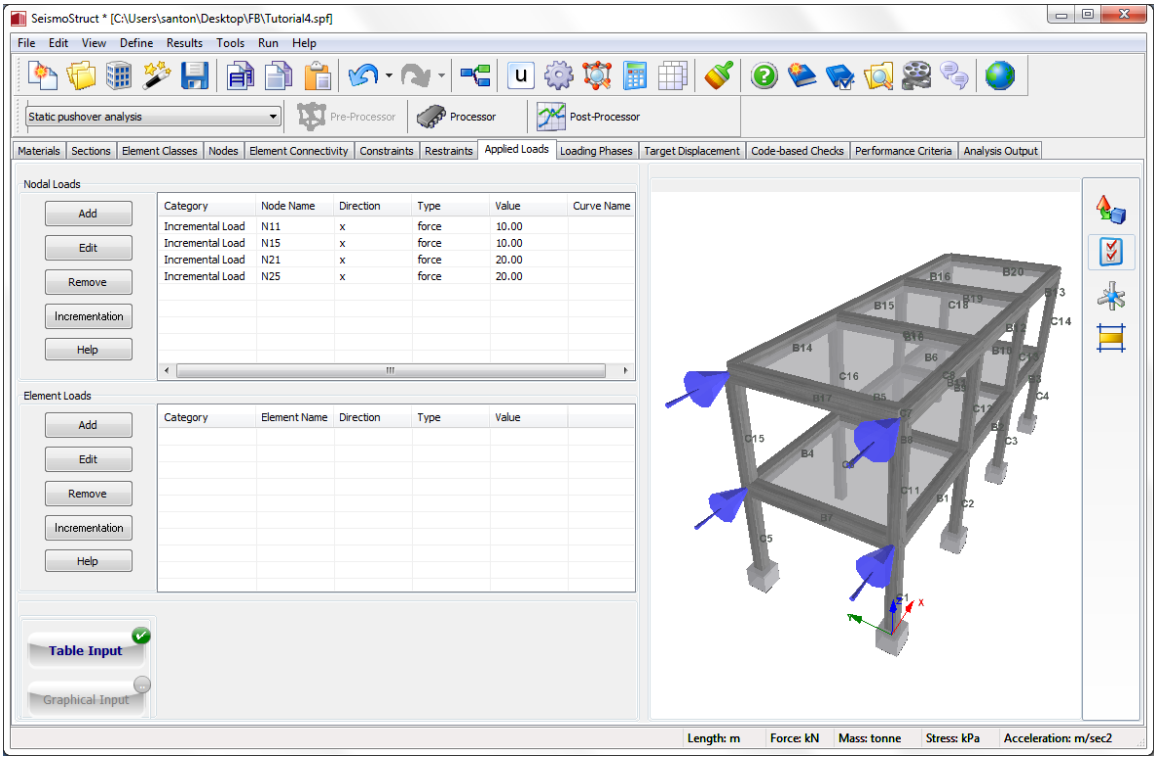
In the table below all the applied loads are summarized:

Category	Node name	Direction	Type	Value
Incremental Load	N11	x	force	10
Incremental Load	N15	x	force	10
Incremental Load	N21	x	force	20
Incremental Load	N25	x	force	20



Incremental Loads

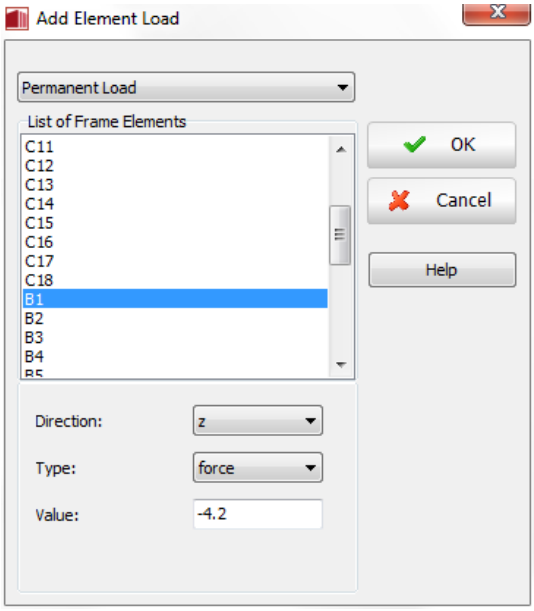
From SeismoStruct v7.0 onwards users can apply distributed load on elements in the **Applied Loads** module by click on the *Add* button for Element Loads.



Applied Loads Module

The user has to:

1. Specify the associated element (➔ B1);
2. Select the load direction from the drop-down menu (➔ Z);
3. Select the load type from the drop-down menu (➔ force);
4. Specify the nominal value (➔ -4.2).

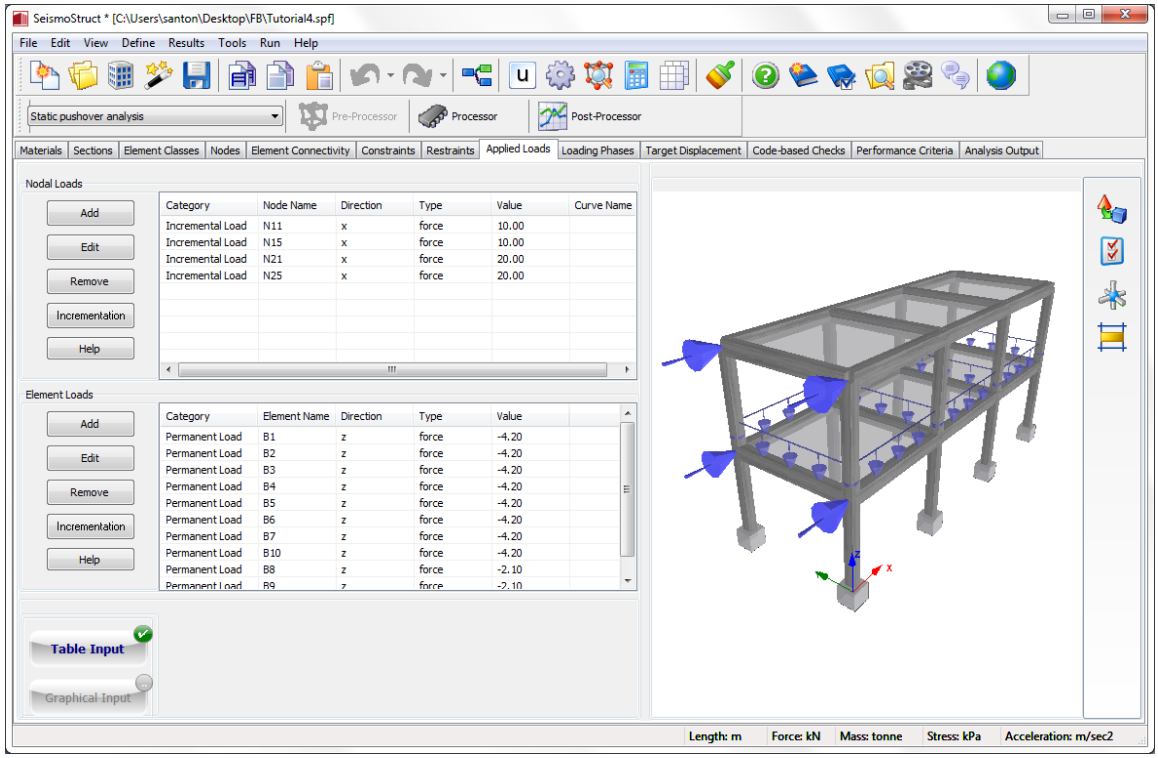


Applied Element Load window

Repeat the same procedure in order to apply the other element loads.

In the table below all the applied loads are summarized:

Element name	Direction	Type	Value
B1	Z	force	-4.2
B2	Z	force	-4.2
B3	Z	force	-4.2
B4	Z	force	-4.2
B5	Z	force	-4.2
B6	Z	force	-4.2
B7	Z	force	-4.2
B10	Z	force	-4.2
B8	Z	force	-2.1
B9	Z	force	-2.1

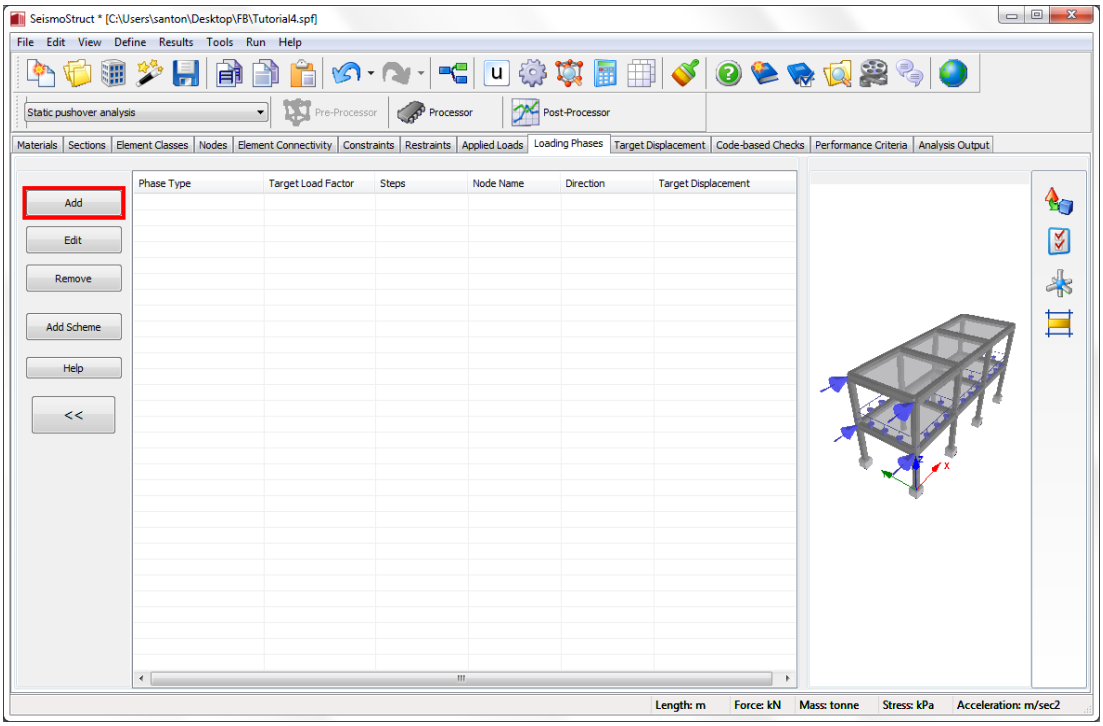


Applied Loads

NOTE: It is recalled that, if it has been selected in the Gravity and Mass settings (Project Settings -> Gravity & Mass) to transform masses to loads, the frame element distributed mass will be transformed to distributed element loads.

Pre-processor – Loading Phases

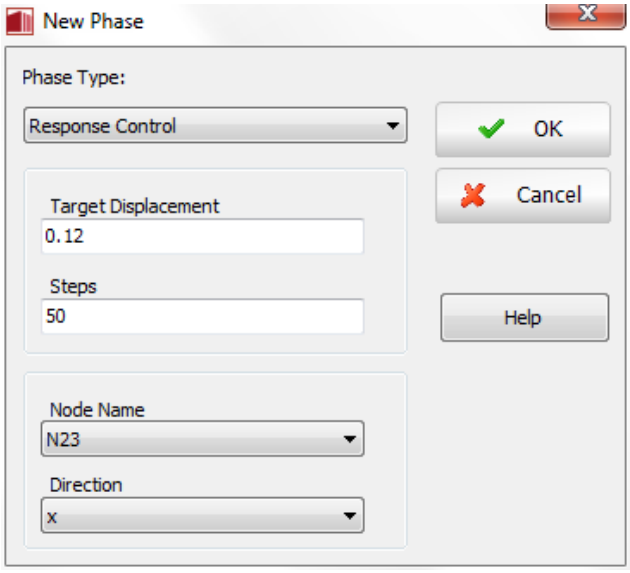
The loading strategy adopted in the pushover analysis is fully defined in the **Loading Phases** module. In this tutorial you are going to define a *Response Control* phase type. Hence, click on the *Add* button.



Loading Phases Module

Then, in the new window, you have to:

1. Select the phase type from the drop-down menu (➔ Response Control);
2. Specify the target displacement (➔ 0.12);
3. Assign the number of steps (➔ default value (50));
4. Select the name of the controlled node from the drop-down menu (➔ N23);
5. Select the direction from the drop-down menu (➔ X).



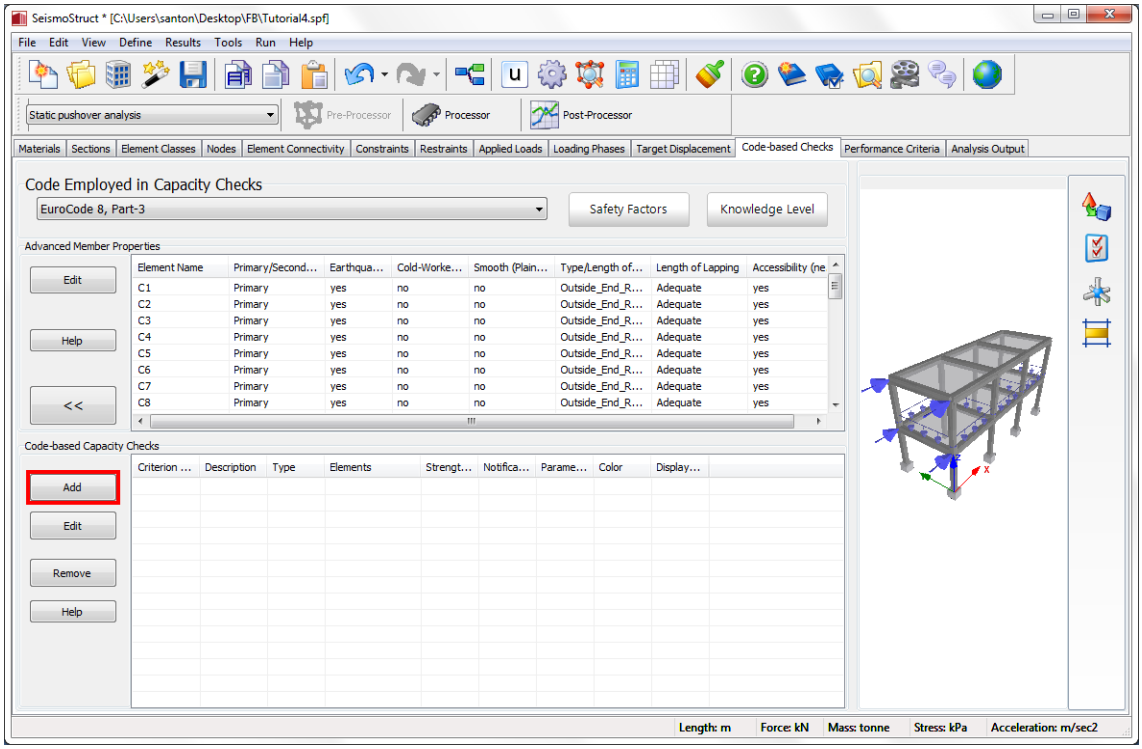
New Phase window

Pre-processor – Target Displacement

In this tutorial we will not select to calculate the Target Displacement.

Pre-processor – Code-based Checks

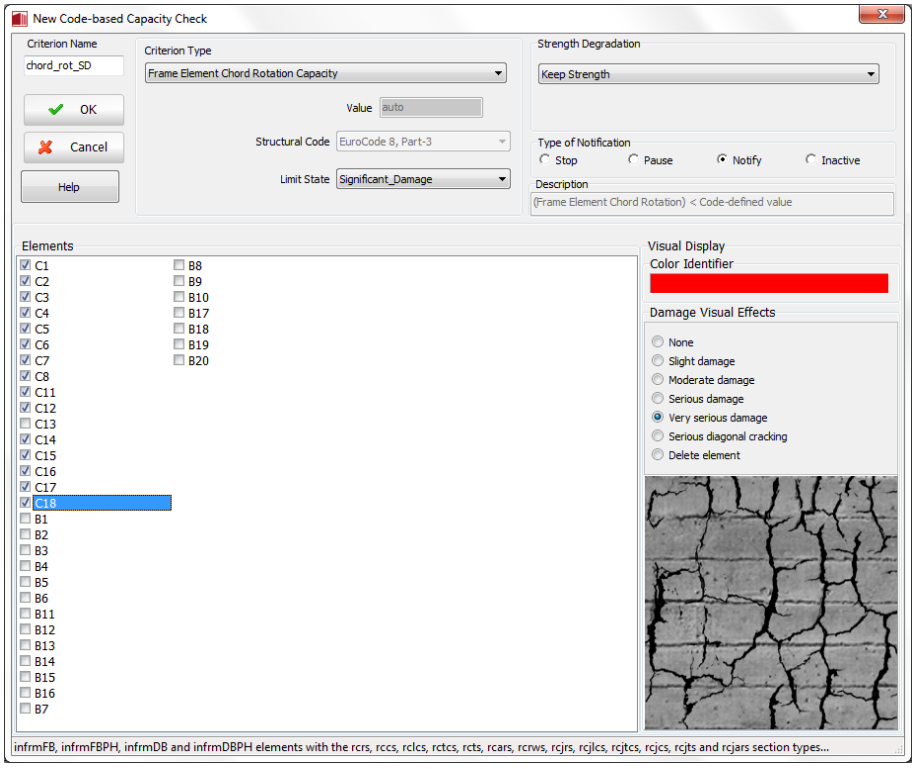
One code-based check related to the chord rotation capacity of all the elements for the Significant Damage limit state will be carried out. In order to do so, you have to go to the **Code-based Checks** module, select the employed Code, define the Safety Factors and the achieved structural Knowledge Level, modify the Advanced Member Properties and click on the *Add* button to add the check.



Code-based checks module

Then, in the new window, you have to:

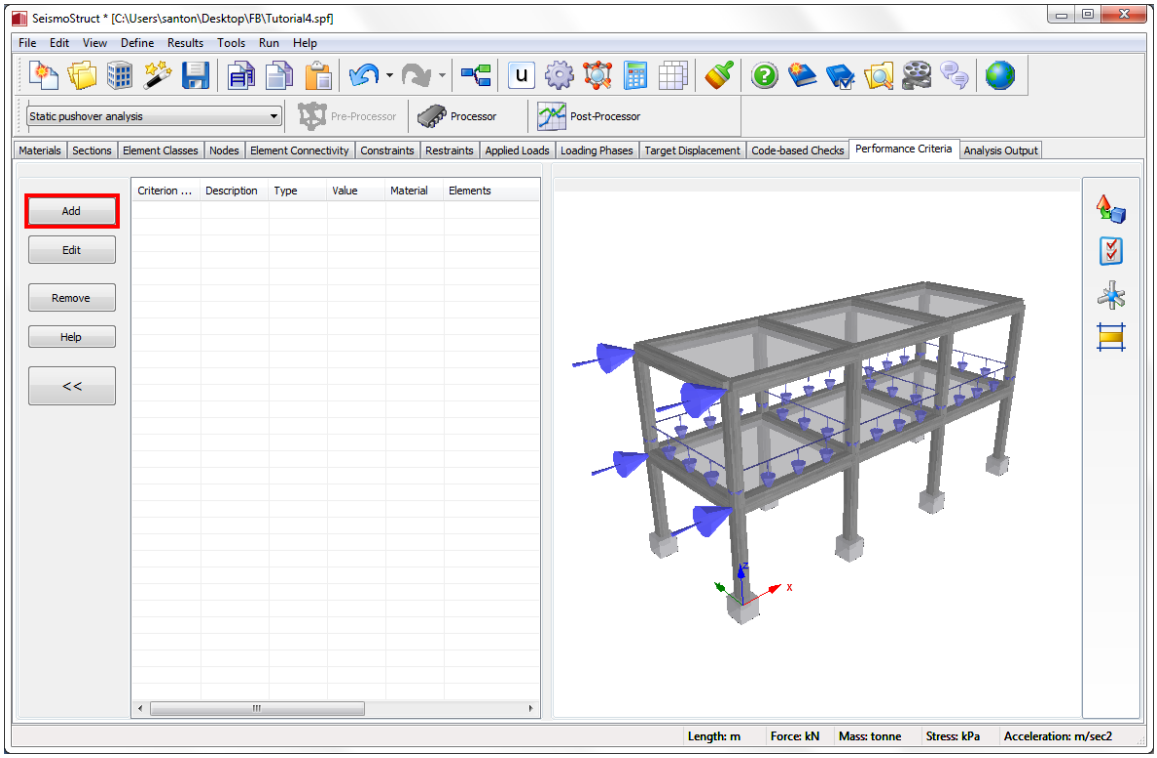
1. Assign the Code-based check name (➔ chord_rot_SD);
2. Select the Code-based check type (➔Frame Element Chord Rotation Capacity) from the drop-down menu;
3. Specify the limit state (➔ Significant_Damage);
4. Select, by ticking the corresponding checkboxes, the elements that will be checked;
5. Define the Strength Degradation upon reach of the check criterion (➔ Keep Strength);
6. Indicate the type of action (➔ Notify);
7. Select the Color Identifier and the Damage Visual effects to enable graphical visualisation in the **Deformed Shape Viewer** module of the Post-Processor.



New Code-based Capacity Check window

Pre-processor – Performance Criteria

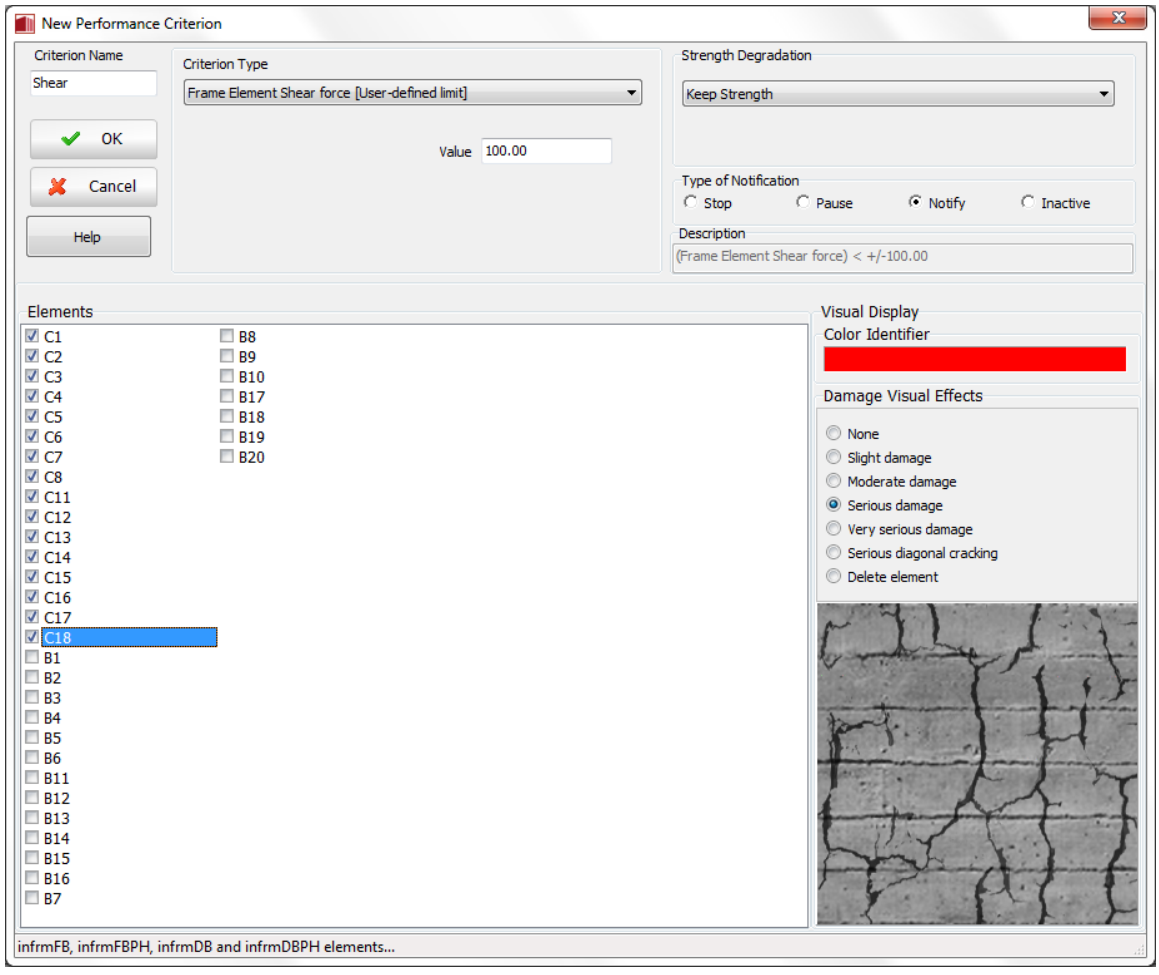
In this tutorial we want to define also a performance criterion related to the shear forces developed in the columns. Hence, you have to go to the **Performance Criteria** module and click on the *Add* button.



Performance Criteria module

Then, in the new window, you have to:

1. Assign the criterion name (➔ Shear);
2. Select the criterion type (➔ frame element shear force [User-defined limit]) from the drop-down menu;
3. Specify the user-defined value at which the criterion is reached (➔ 100);
4. Select, by ticking the corresponding checkboxes, the elements to which the criterion applies to;
5. Define the Strength Degradation upon reach of the criterion (➔ Keep Strength);
6. Indicate the type of action (➔ Notify);
7. Select the Color Identifier and the Damage Visual effects to enable graphical visualisation in the **Deformed Shape Viewer** module of the Post-Processor.

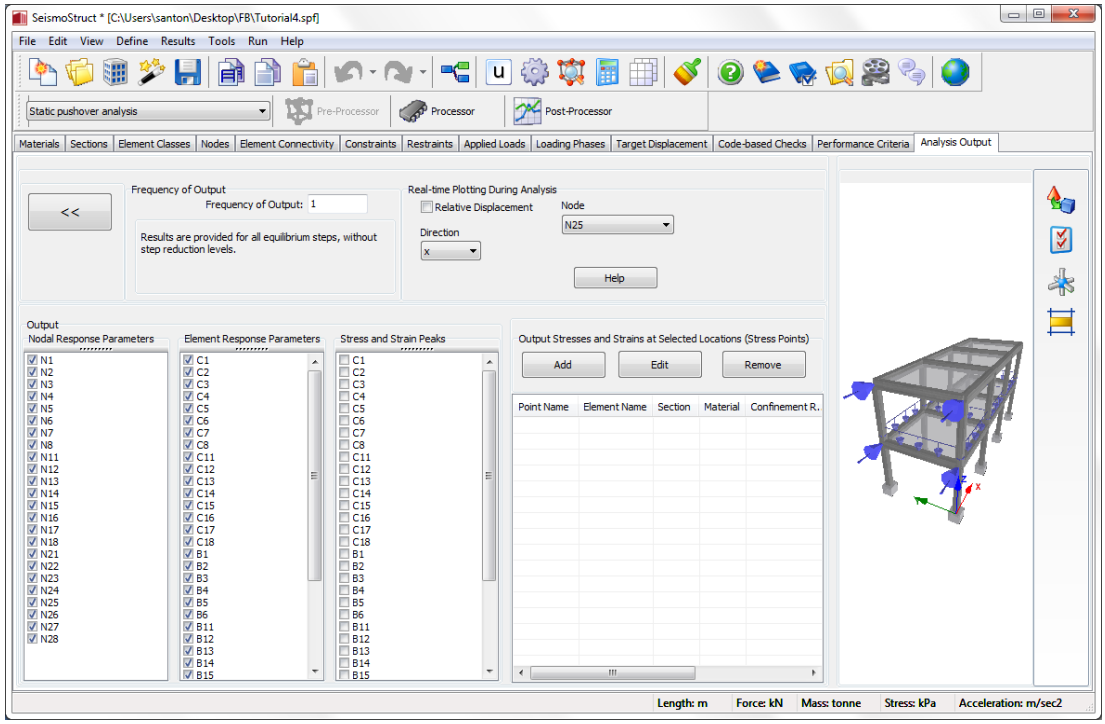


New Performance Criterion window

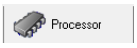
NOTE: It is noted that a large variety of performance criteria may be defined, including criteria on material strains (cracking and spalling of cover concrete, crushing of core concrete, or yielding and fracture of steel), criteria on section curvatures and chord-rotations, and chord rotation and element shear capacity checks.

Pre-processor – Analysis Output

Finally, before accessing to the **Processor** area, you have to set the output preferences in the **Analysis Output** module, as shown below.

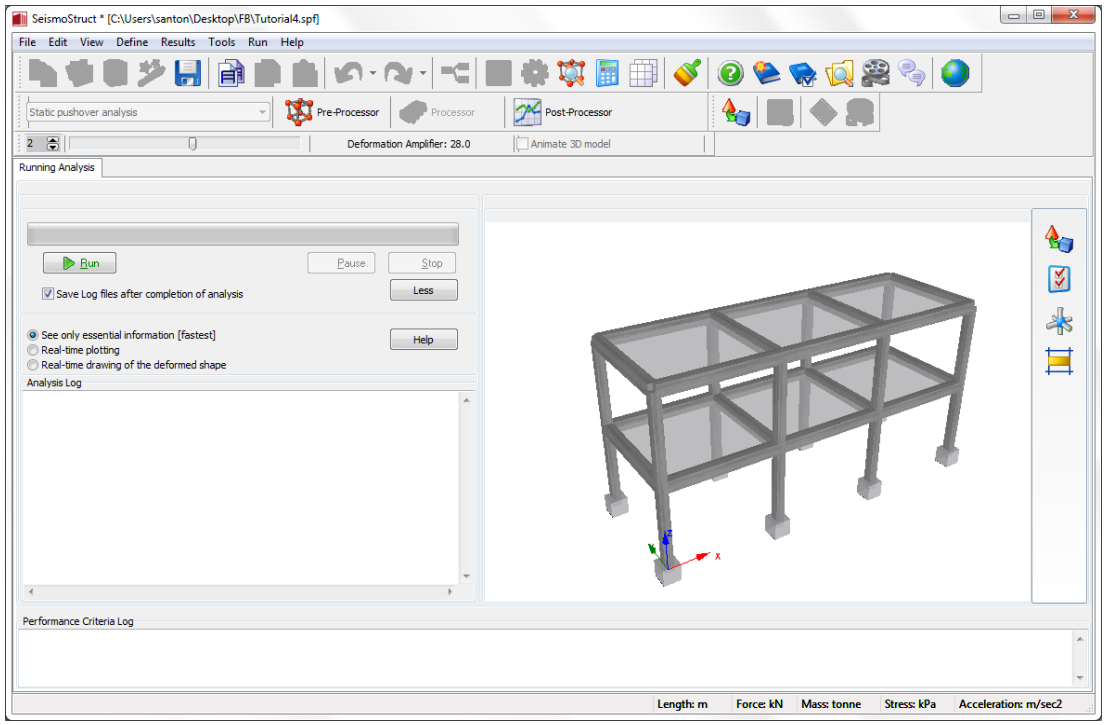


Analysis Output Module

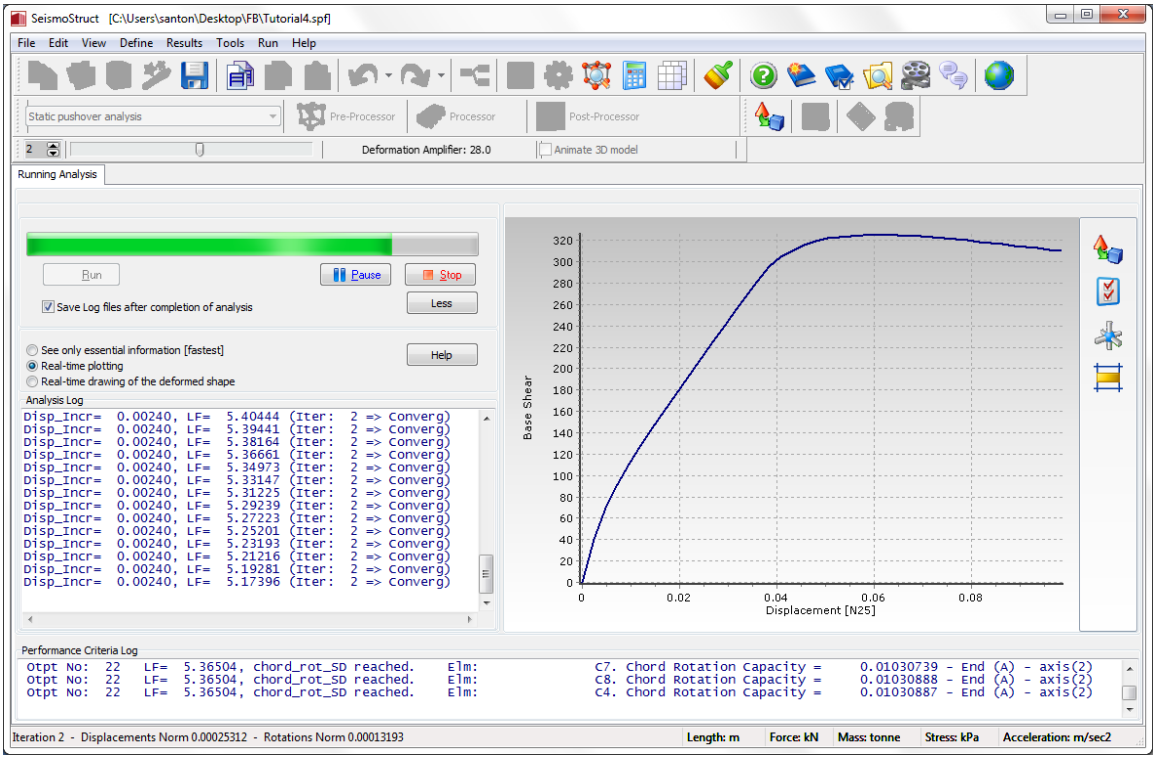
Then, click on the toolbar button  or select *Run* > *Processor* from the main menu.

Processor

In the **Processor** area you are allowed to start the analysis. Hence, click on the *Run* button.



Processor Area



Running the analysis

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

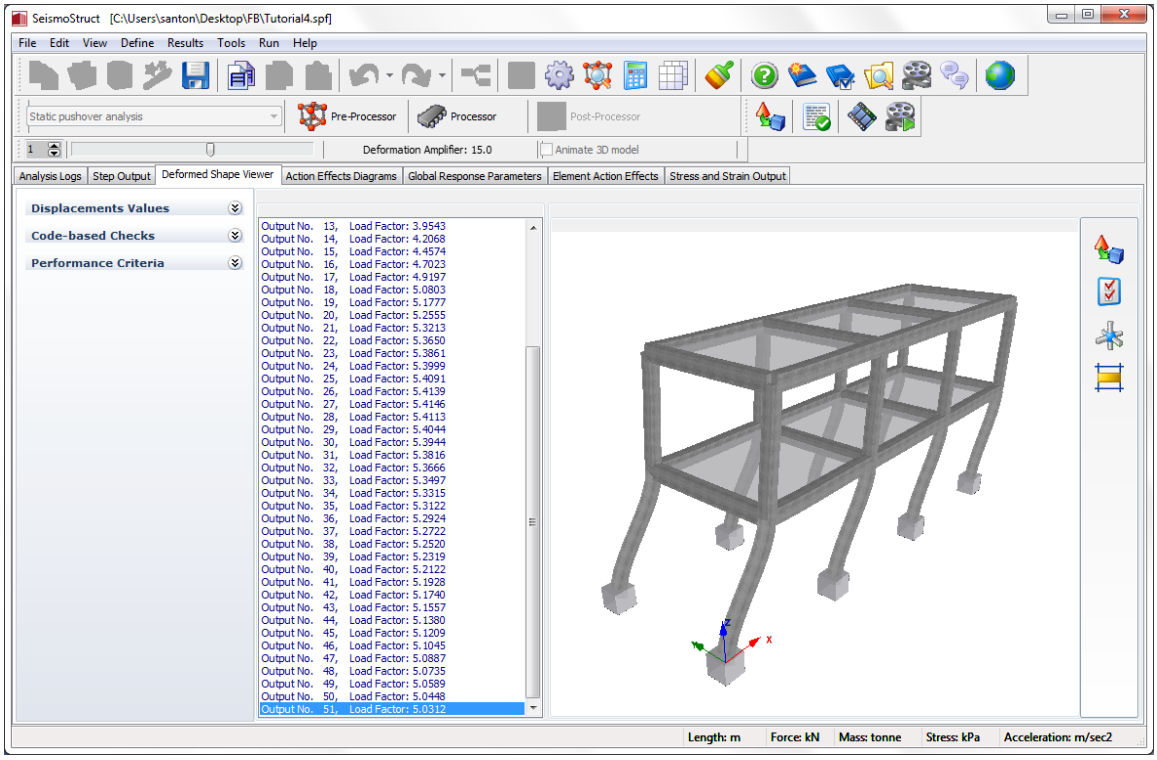
When the analysis has arrived to the end, click on the toolbar button 

or select *Run* > *Post-Processor* from the main menu.

Post-processor – Deformed Shape Viewer

The **Post-Processor** area features a series of modules where results can be visualised, in table or graphical format, and then copied into any other Windows application.

In the **Deformed shape viewer** module you have the possibility of visualising the deformed shape of the model at every step of the analysis. Double-click on the desired output identifier to update the deformed shape view (see figure below).



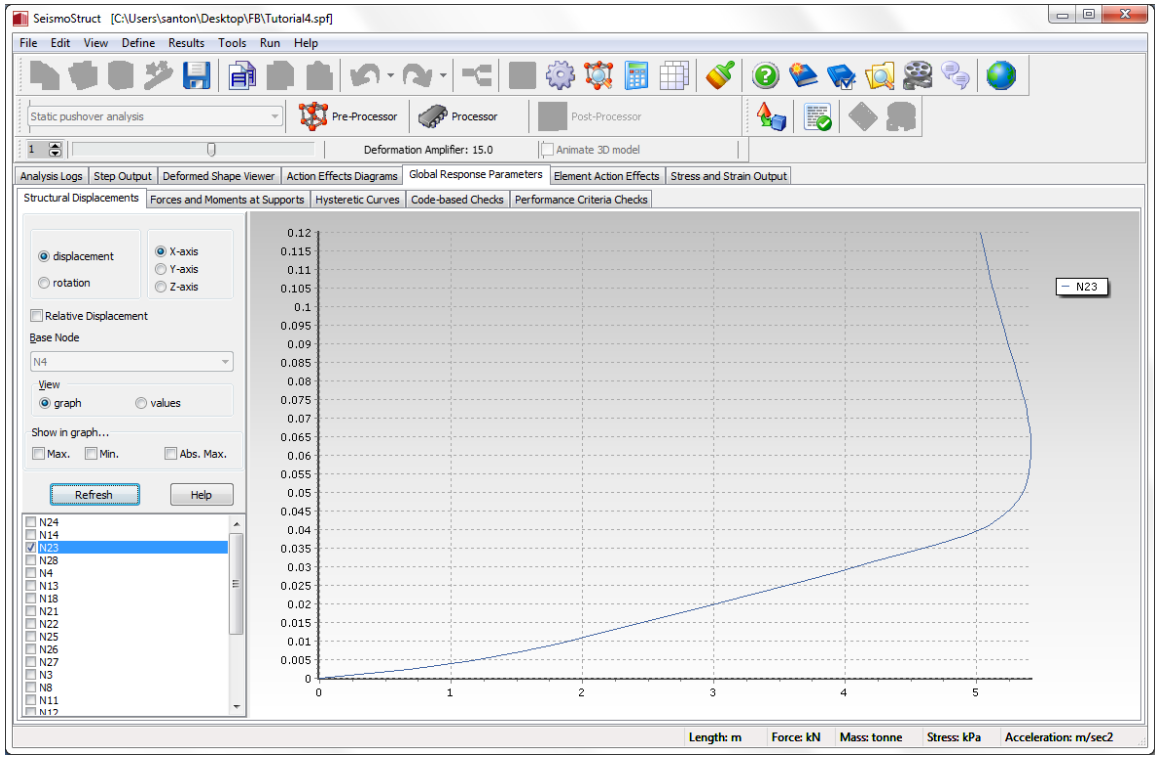
Deformed Shape Viewer module

Post-processor – Global Response Parameters

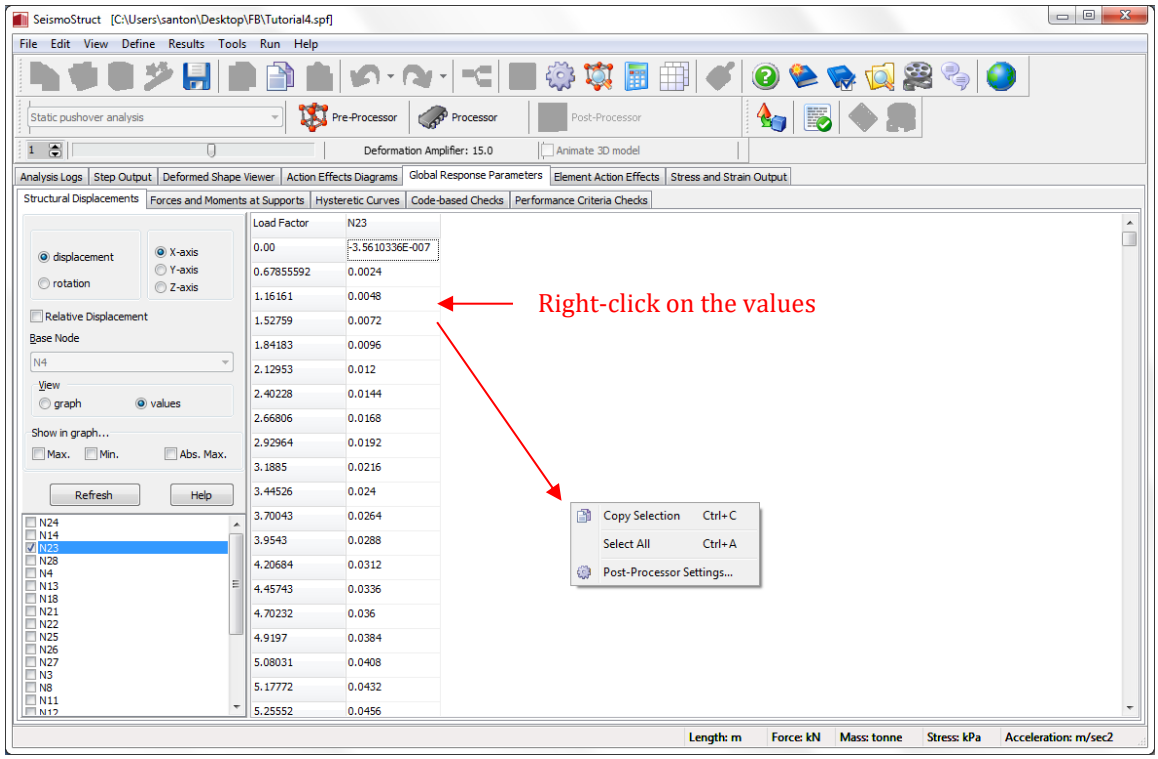
In the **Global Response Parameters** module you can output the following results: (i) structural displacements, (ii) forces and moments at the supports and (iii) hysteretic curves.

First, 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, respectively, *displacement* and *x-axis*, (iii) select the corresponding node from the list (-> N23) by ticking the box, (iv) choose the results visualisation (*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.

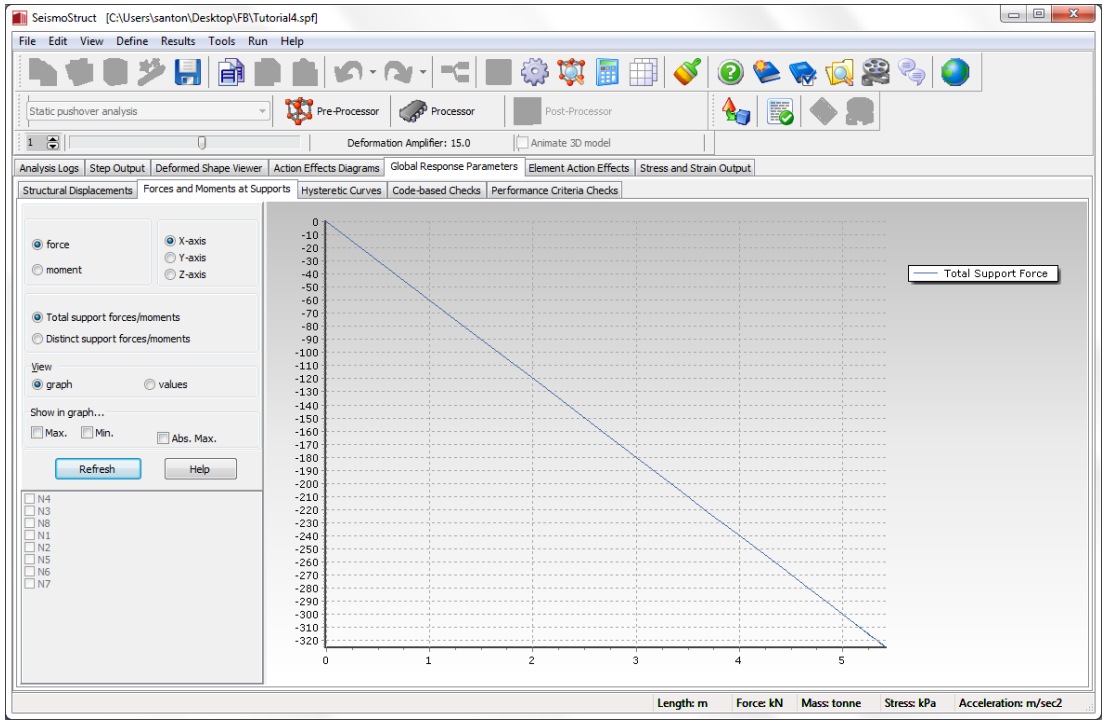


Global Response Parameters Module (Structural Displacements – graph mode)



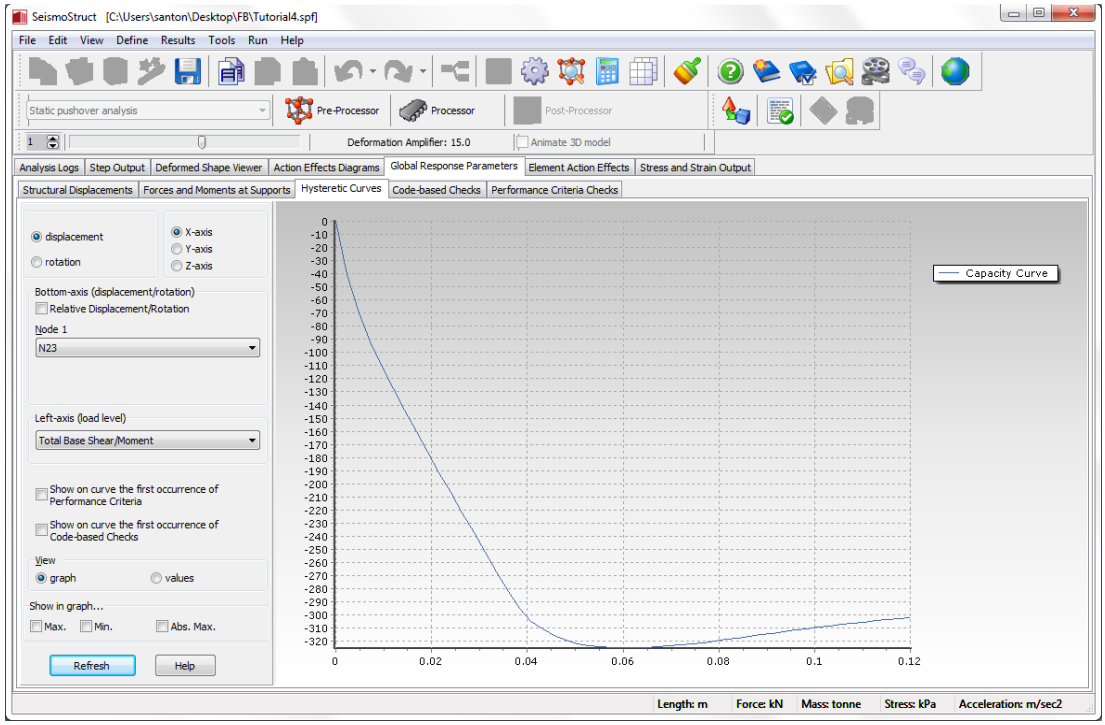
Global Response Parameters Module (Structural Displacements – values mode)

Second, in order to obtain the total support forces (e.g. total base shear), (i) click on the *Forces and Moments at support* tab, (ii) select, respectively, *force* and x-axis and *total support forces/moments*, (iii) choose the results visualisation (*graph* or *values*) and finally (iv) click on the *Refresh* button.



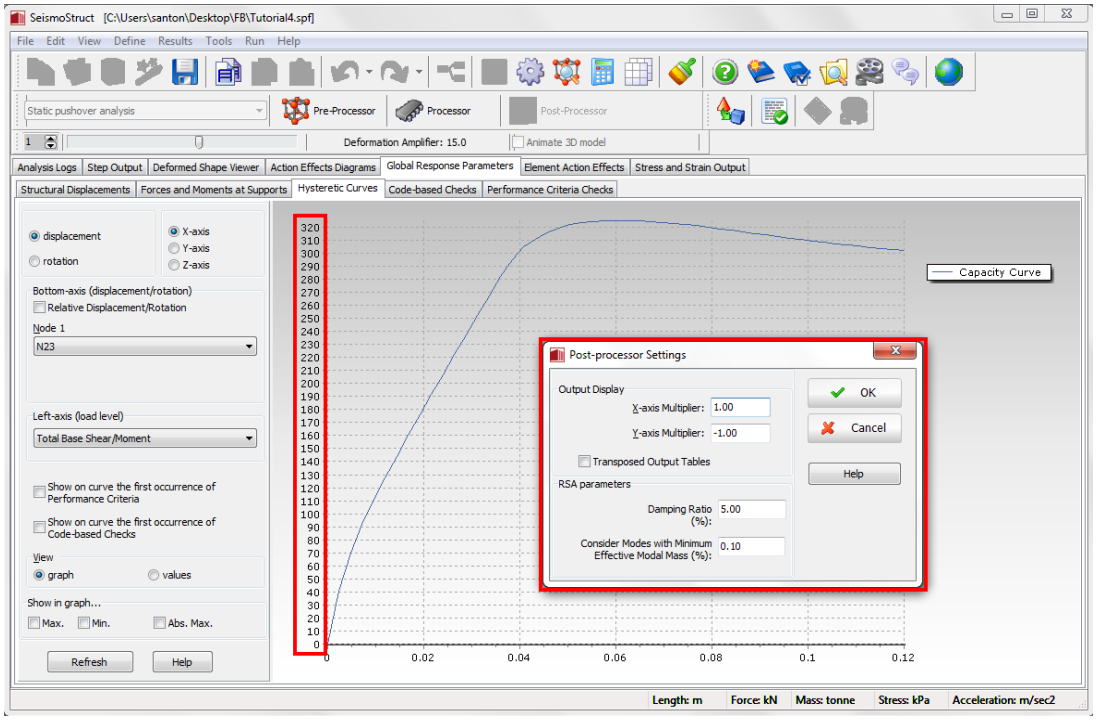
Global Response Parameters Module (Forces and Moments at Supports – graph mode)

Third, in order to plot the capacity curve of your structure (i.e. total base shear vs. top displacement), (i) click on the *Hysteretic Curves* tab, (ii) select, respectively, *displacement* and *x-axis*, (iii) select the corresponding node from the drop-down menu (e.g. N23) for the bottom-axis, (iv) select the *Total Base Shear/Moment* option for the left-axis, (v) choose the results visualisation (*graph* or *values*) and finally (vi) click on the *Refresh* button.



Global Response Parameters Module (Hysteretic Curves – graph mode)

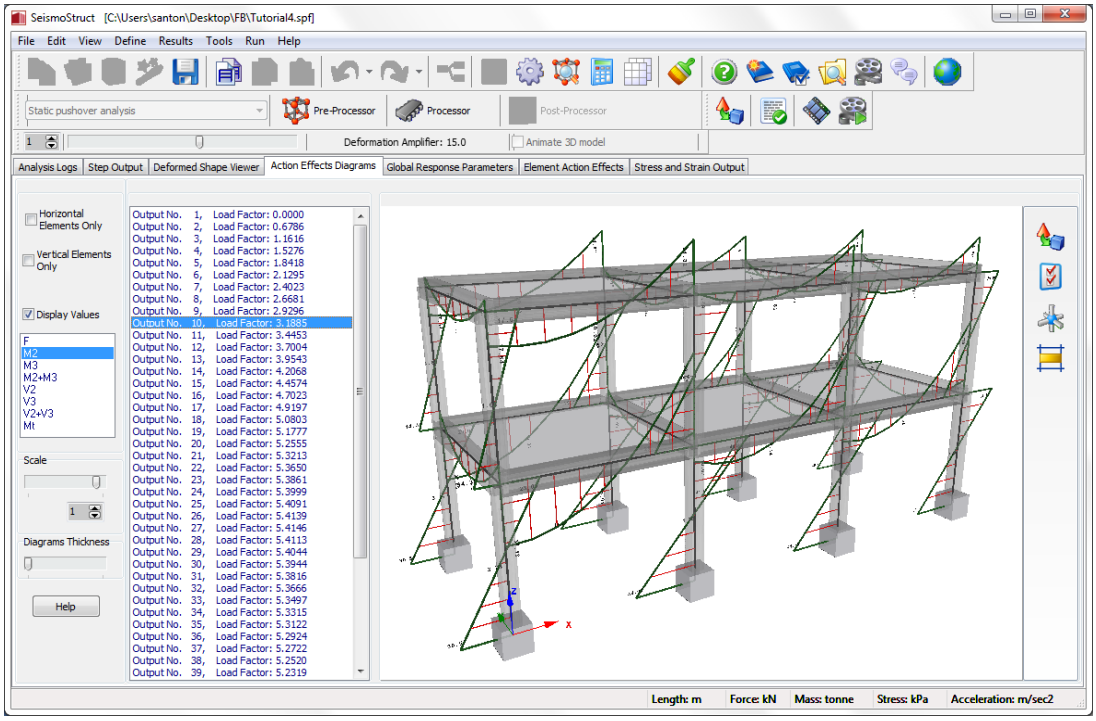
In order to have the shear forces with positive values, (i) right-click on the 3D plot window, (ii) select Post-Processor Settings and (iii) insert the value "-1" as Y-axis multiplier.



Global Response Parameters Module (Hysteretic Curves – graph mode)

Post-processor –Action Effects Diagrams

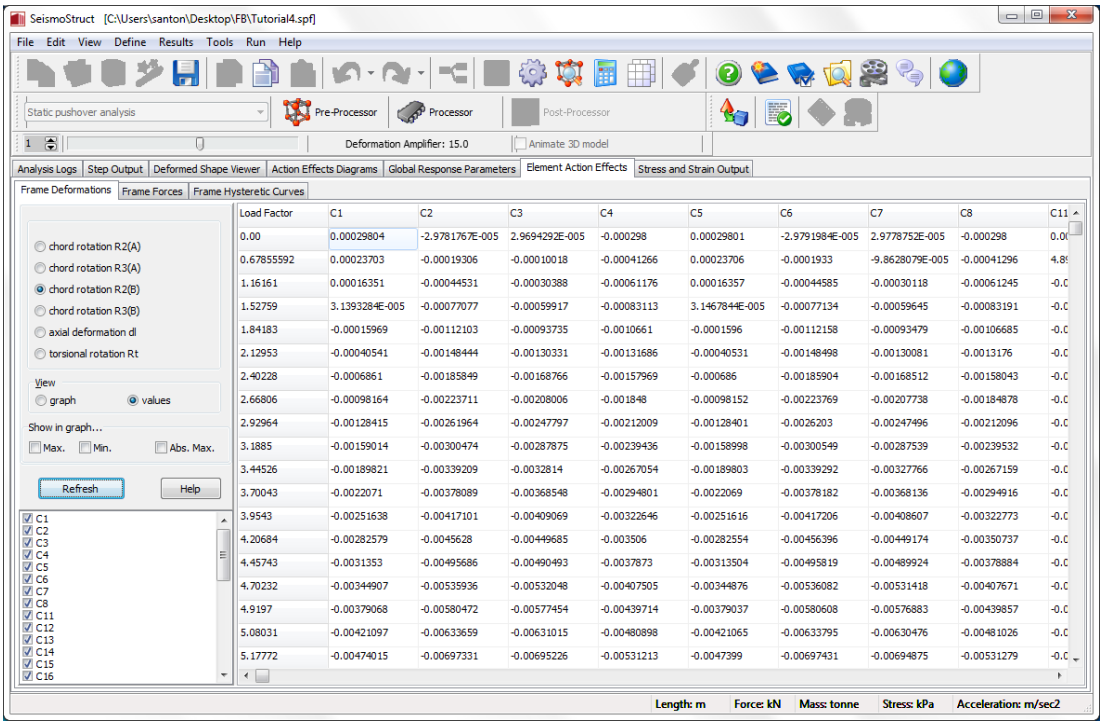
In the **Action Effects Diagrams** module, you can visualise the internal forces and moments diagrams for each analysis step. As an example, in the figure below the moments diagrams are shown:



Action Effects Diagrams Module

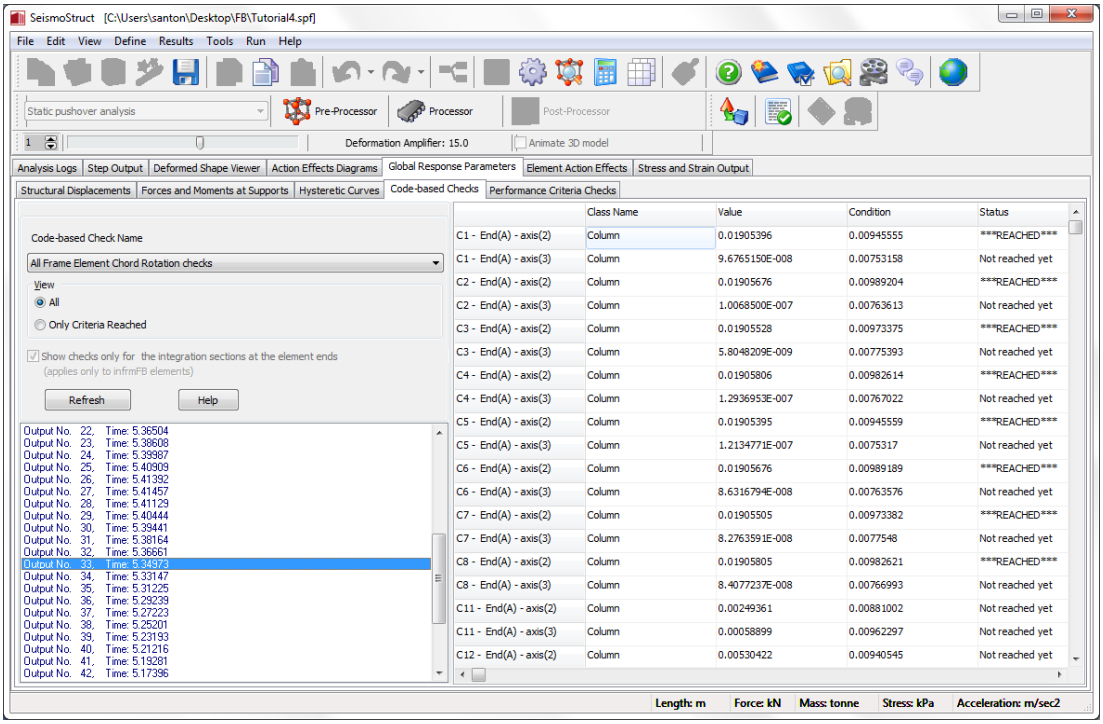
Post-processor – Element Action Effects

In order to proceed with the seismic verifications prescribed in several seismic codes (see e.g. Eurocode 8, KANEPE, ASCE/SEI 7-05, etc) 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. Let us start with the former. Since you have employed inelastic force-based frame elements (infrmFB) for defining the structural elements, 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 visualisation (*graph* or *values*) and finally (v) clicking on the *Refresh* button.



Element Action Effects Module (Frame Deformations – values mode)

In order to avoid the need for users to carry out hand-calculations for the estimation of the capacity of the structural members, SeismoStruct provides the option to automatically undertake chord-rotation and shear capacity checks, according to the expressions defined in the supported Codes (Eurocode 8, ASCE 41-23, NTC-18, KANEPE and TBDY) for the selected limit states. This can be done in the *Code-based Checks* tab of the *Global Response Parameters* page of the Post-Processor.



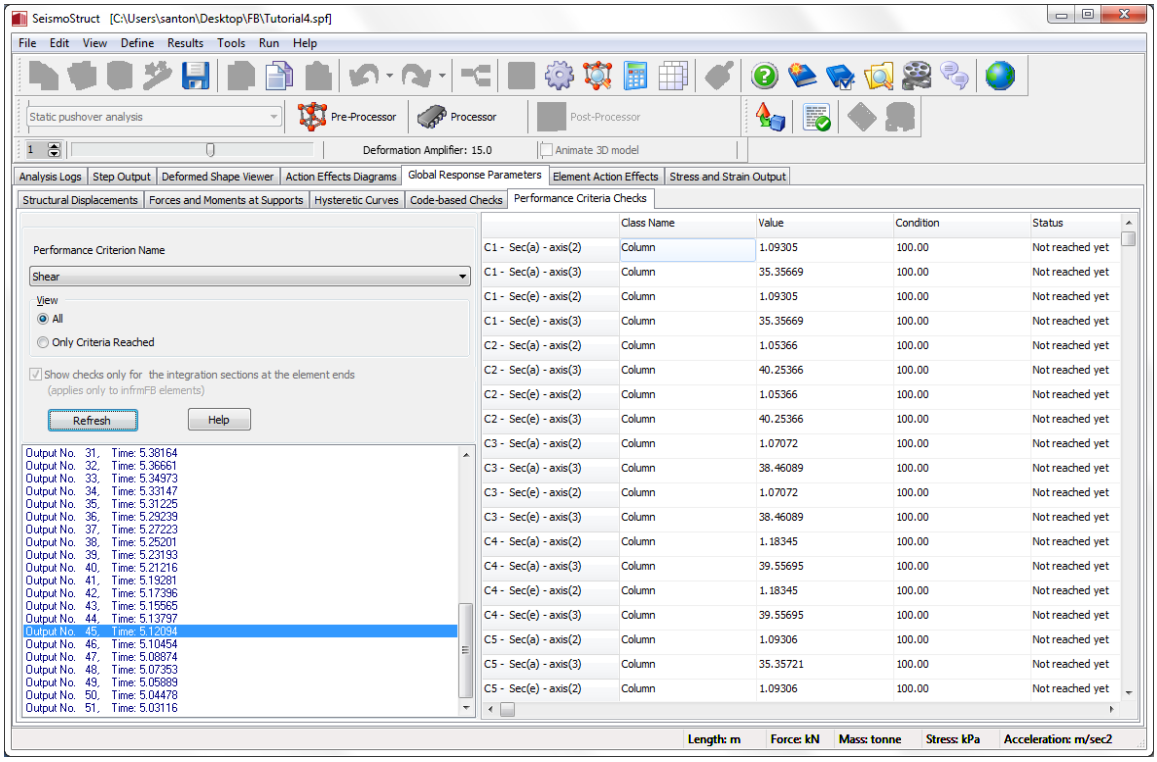
Code-based Checks

The user may select either a specific Code-based Check or all the defined checks of the same type. Two types of Code-based checks are available, chord rotation and shear capacity checks.

When the user clicks on one of the analysis steps, a list of all the checks for all the structural members appears. The data shown include the demand, the capacity, and whether the particular check has been reached in that particular location, while the results for each integration section and for both local axes, (2) and (3) are provided.

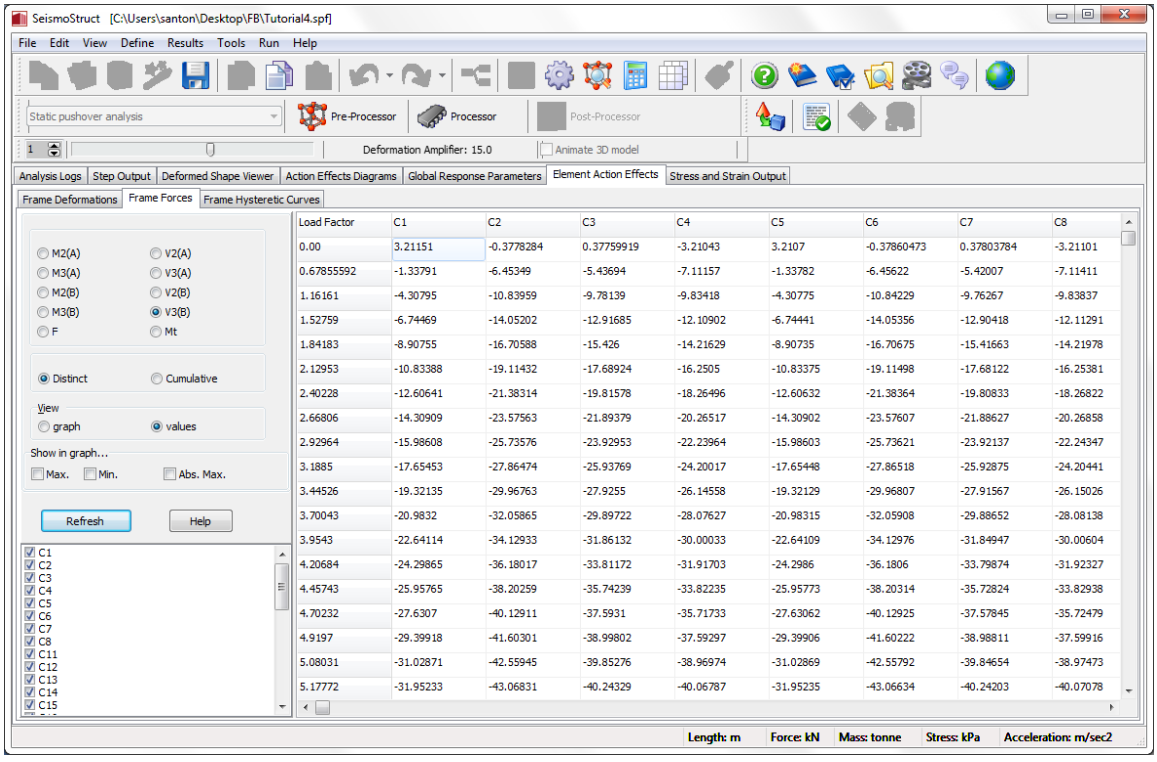
In addition to the code-based checks, users are capable of setting up performance criteria identifying the instants during the analysis, at which different performance limit states (identified by material strains, section curvature, element chord-rotation and shear values) are expected to be reached. This can be done in the *Performance Criteria Checks* tab of the *Global Response Parameters* page of the Post-Processor. By default, the chord rotation and shear capacities criteria are not selected.

NOTE: The main difference between the Code-based Checks and the Performance Criteria is that the latter are checks against the 'expected' values of the response quantities, whereas the former follow the conservative assessment methodologies as defined by the corresponding Codes and Standards. Hence, in Code-based Checks the expressions employed for the calculation of the threshold value, when the different performance limit states are reached, employ conservative (e.g. characteristic or nominal) material strengths, and are based on the safety and confidence factors, as specified in the Codes. On the contrary, mean material values and no safety or confidence factors are used in the Performance Criteria calculations.



Performance Criteria Checks

Finally, in order to visualise the frame element forces (e.g. shear forces), (i) click on the *Frame Forces* tab, (ii) select the force (e.g. V3), (iii) select the elements from the list, by ticking the corresponding box, (iv) choose the results visualisation (*graph* or *values*) and finally (v) clicking on the *Refresh* button.



Element Action Effects Module (Frame Forces – values mode)

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

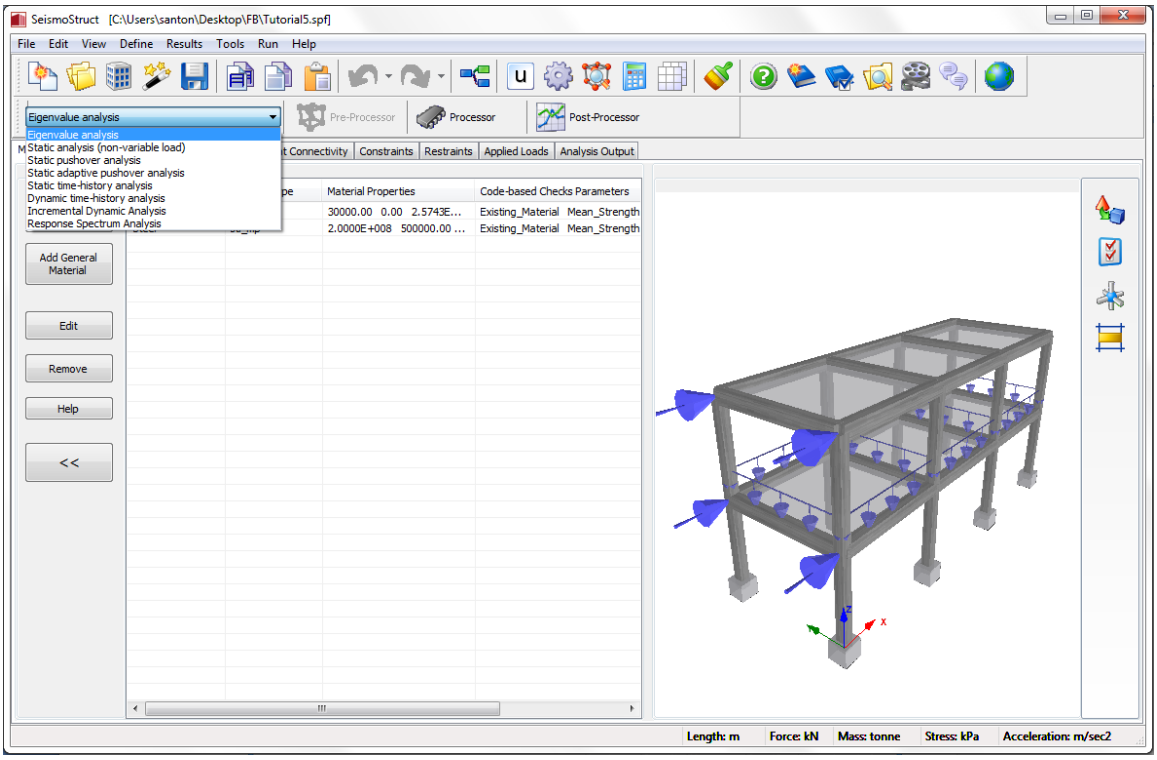
TUTORIAL N.5 – EIGENVALUE ANALYSIS OF A TWO-STOREY BUILDING

Problem Description

Let's use the same model that has already been created in Tutorial 4.

Getting started: opening an existing project

So, in order to start with this new tutorial, (i) open SeismoStruct initial window, (ii) select the previous SeismoStruct project (Tutorial 4.spf) through *File > Open...* menu command or click on icon on the toolbar, (iii) save the project with a new name through *File > Save as...* menu command and then (iv) select the *Eigenvalue analysis* from the drop-down menu at the top left corner in the **Pre-Processor** area.



Selection of the analysis type

NOTE 1: Four modules will disappear (*Loading Phases*, *Target Displacement*, *Code-based Checks* and *Performance Criteria*) with respect to the pushover analysis.

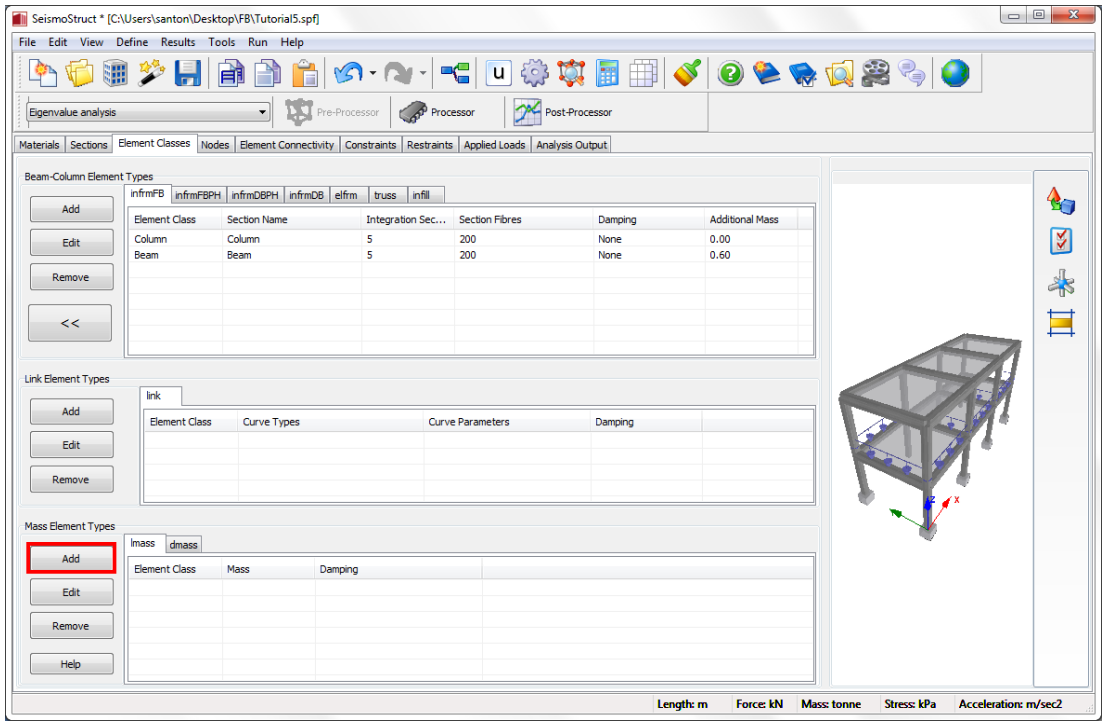
NOTE 2: The existing permanent loads, from Tutorial 4, are not taken into consideration in the eigenvalue analysis, unless the option *Define Mass from both Frame/Mass Elements and Loads* is selected in the *Project Settings > Gravity & Mass* module.

Once the type of analysis has been selected, move to the **Element Classes** module in order to define the mass element types.

IMPORTANT: In the *Material* module the specific weight of each material has been already defined in Tutorial 4 and the software will automatically compute, by default, the element masses from those values (see *Project Settings > Gravity & Mass*).

Pre-processor – Element Classes

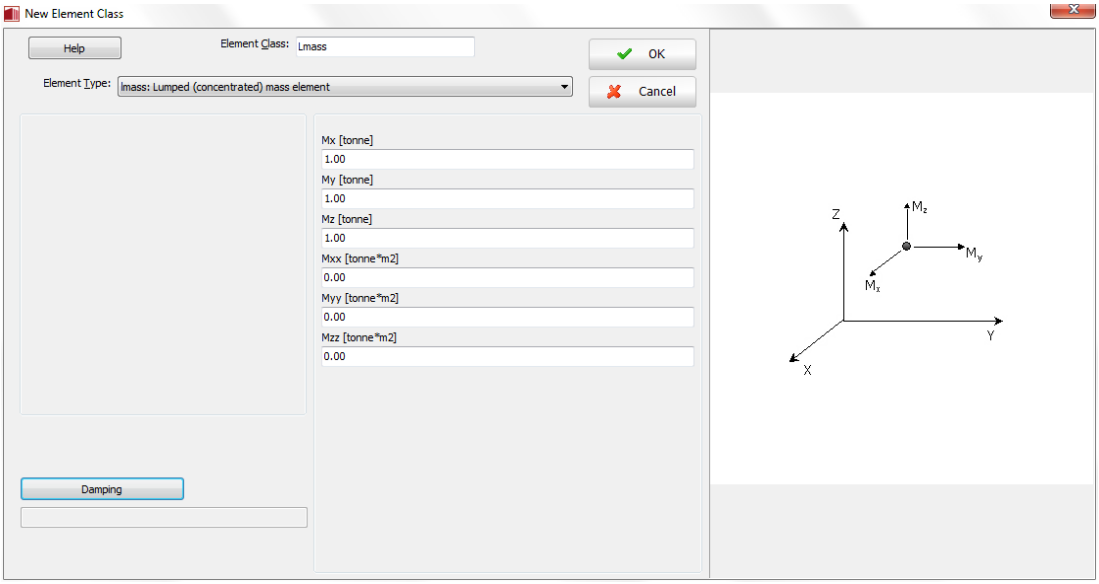
Click on the *Add* button related to the *Mass Element Types*.



Element Classes module

In the dialogue window you have to:

1. Assign the element name (➔ Lmass);
2. Select the element type from the drop-down menu (➔ lmass element);
3. Set the mass value (let's assume 1 ton) in the directions of interest (i.e. translational dir. only);
4. Define an element-specific damping (➔ no element specific damping is applied)



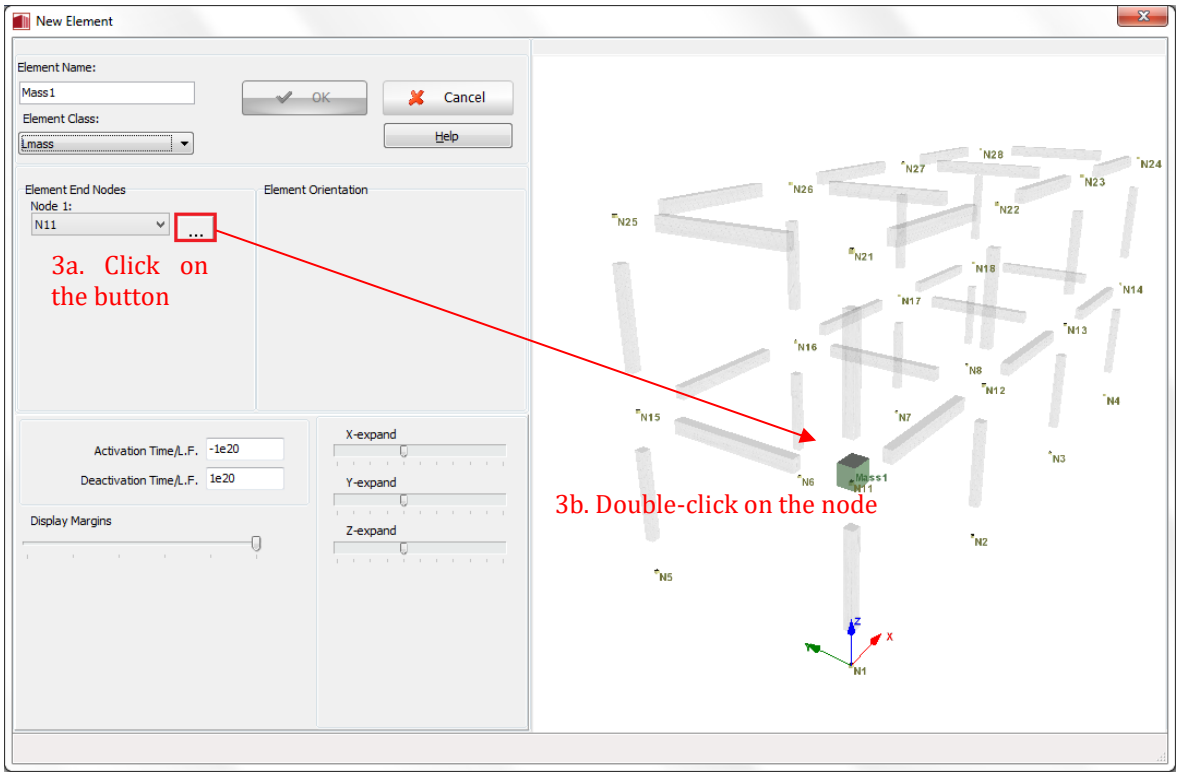
Definition of the Element Classes (Lumped)

Pre-processor – Element Connectivity

Now, move to the **Element Connectivity** module in order to assign the lumped mass element, for example, to the corner nodes of the structure.

Click on the *Add* button. In the new window you have to:

1. Assign the element name (➔ Mass1);
2. Select the element class from the drop-down menu;
3. Select the structural node (see figure below for details).

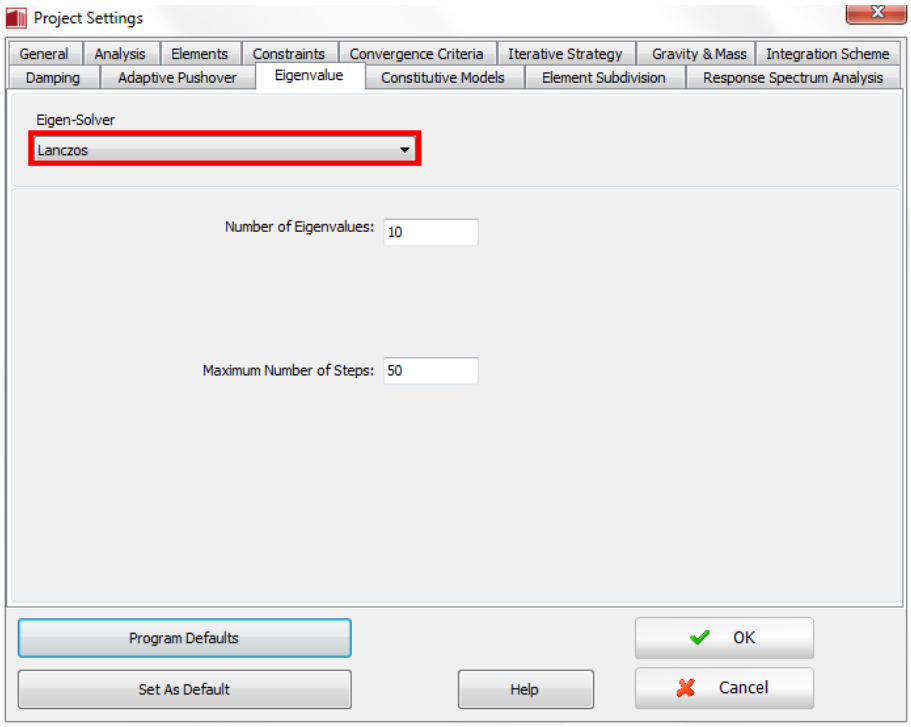


New element window

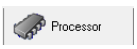
Repeat the procedure described above in order to define all the other lumped mass elements. In the table below all the lumped mass elements are summarized:

Element Name	Element Class	Nodes
Mass1	Lumped	N11
Mass2	Lumped	N14
Mass3	Lumped	N15
Mass4	Lumped	N18
Mass5	Lumped	N21
Mass6	Lumped	N24
Mass7	Lumped	N25
Mass8	Lumped	N28

Before running the analysis, you may choose between two different eigensolvers, the **Lanczos algorithm** or the **Jacobi algorithm with Ritz transformation**, in order to determine the modes of vibration of the structure (*Tools > Project Settings...*). In this tutorial the Lanczos algorithm has been selected.



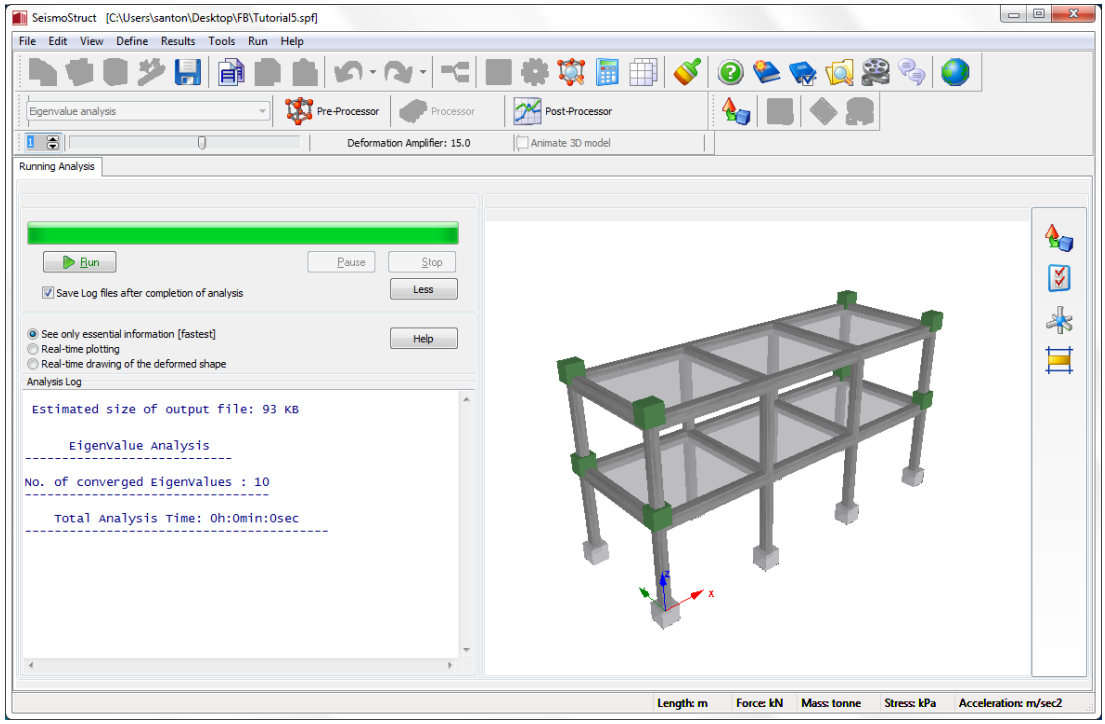
Eigenvalue settings

At this point you may click on the toolbar button 

or select *Run > Processor* from the main menu in order to perform the Eigenvalue analysis.

Processor

Click on the *Run* button.



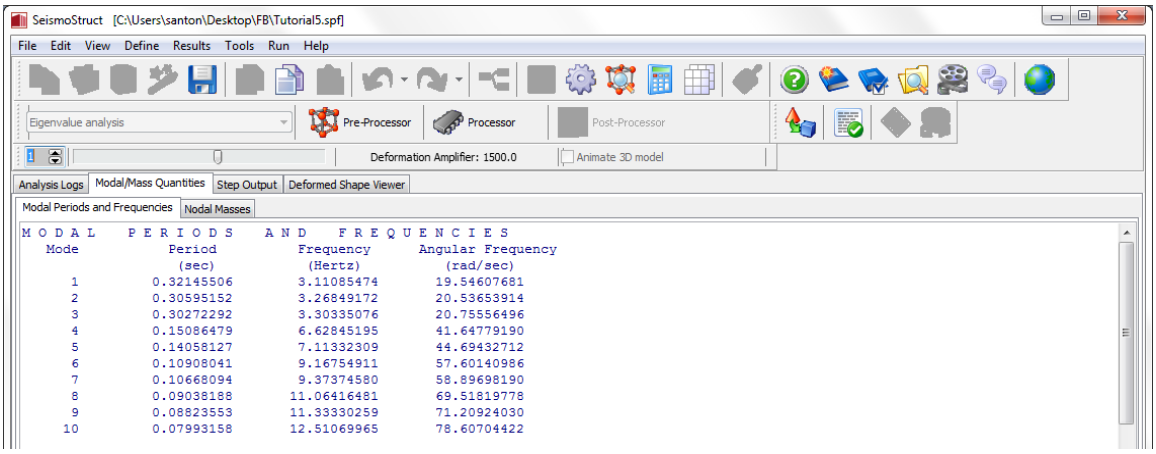
Processor area

When the analysis has arrived to the end, click on the toolbar button 

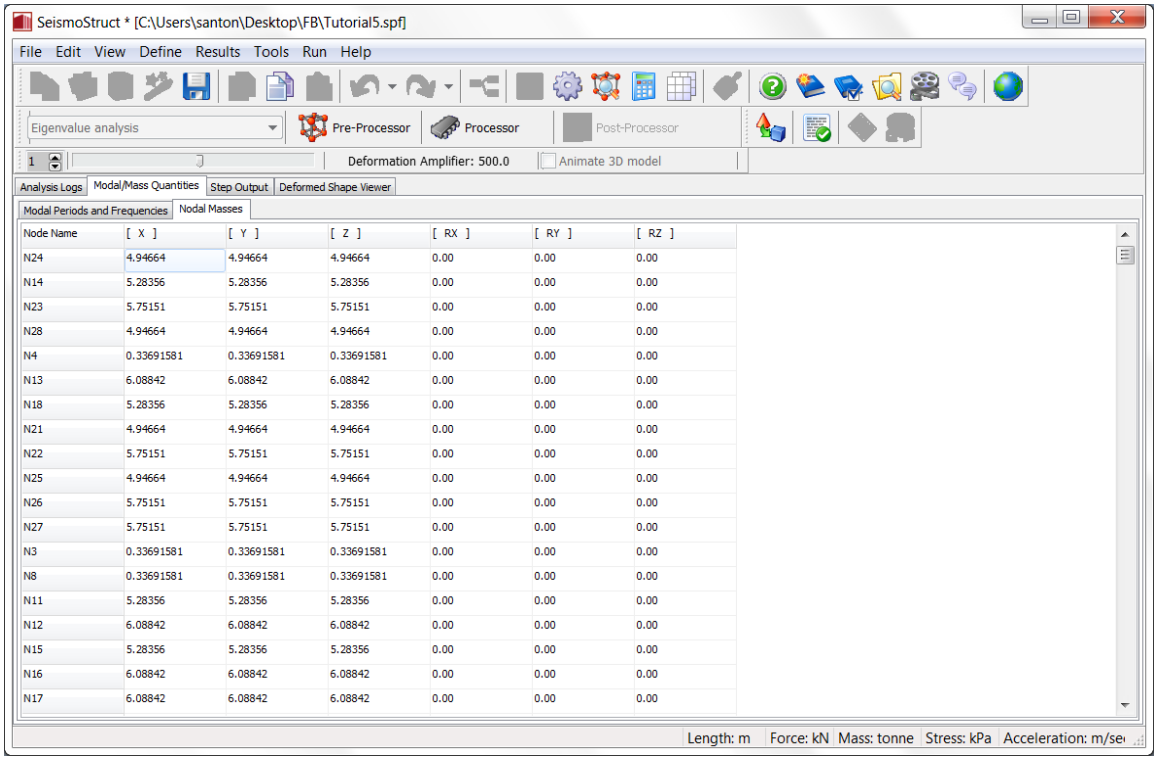
or select *Run* > *Post-Processor* from the main menu.

Post-Processor – Modal/Mass Quantities

In the **Modal/Mass Quantities** module you have the possibility of visualising several eigenvalue data, such as (i) the modal periods and frequencies, (ii) the modal participation factors, (iii) the effective modal masses, (iv) the effective modal mass percentages of your structure, and finally (v) the nodal masses.



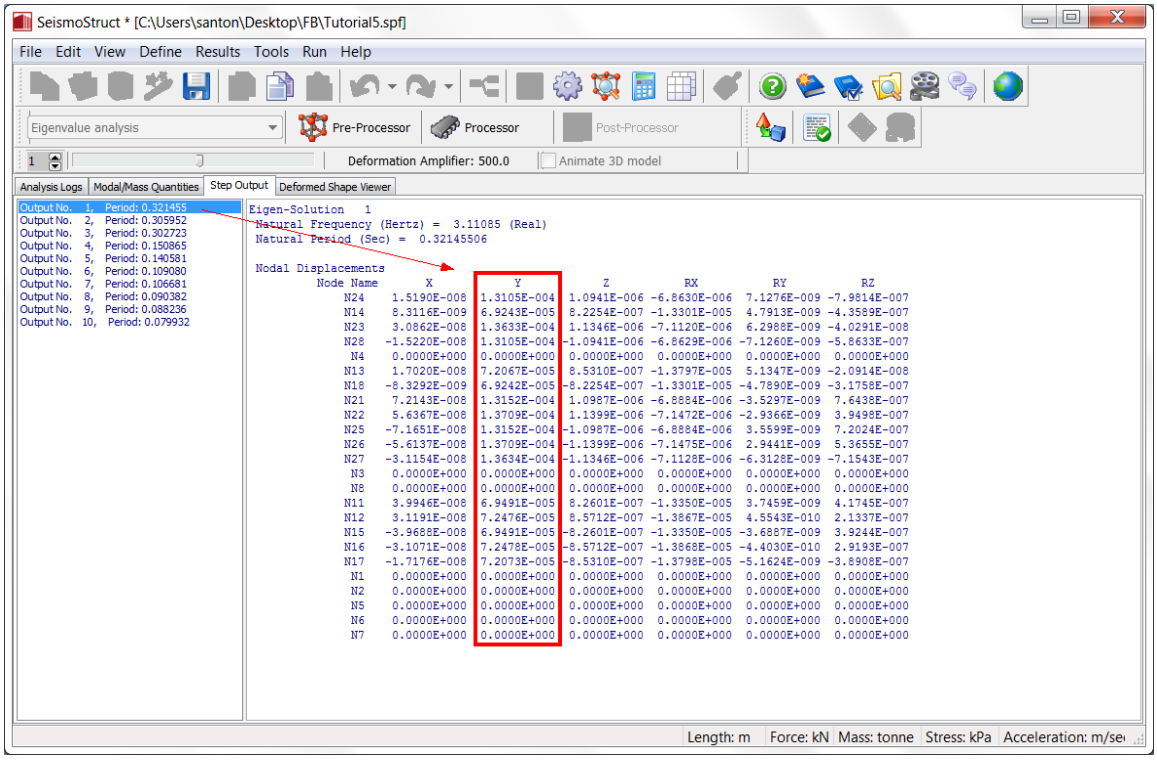
Modal/Mass Quantities Module – Modal Periods and Frequencies



Modal/Mass Quantities Module – Nodal Masses

Post-Processor – Step Output

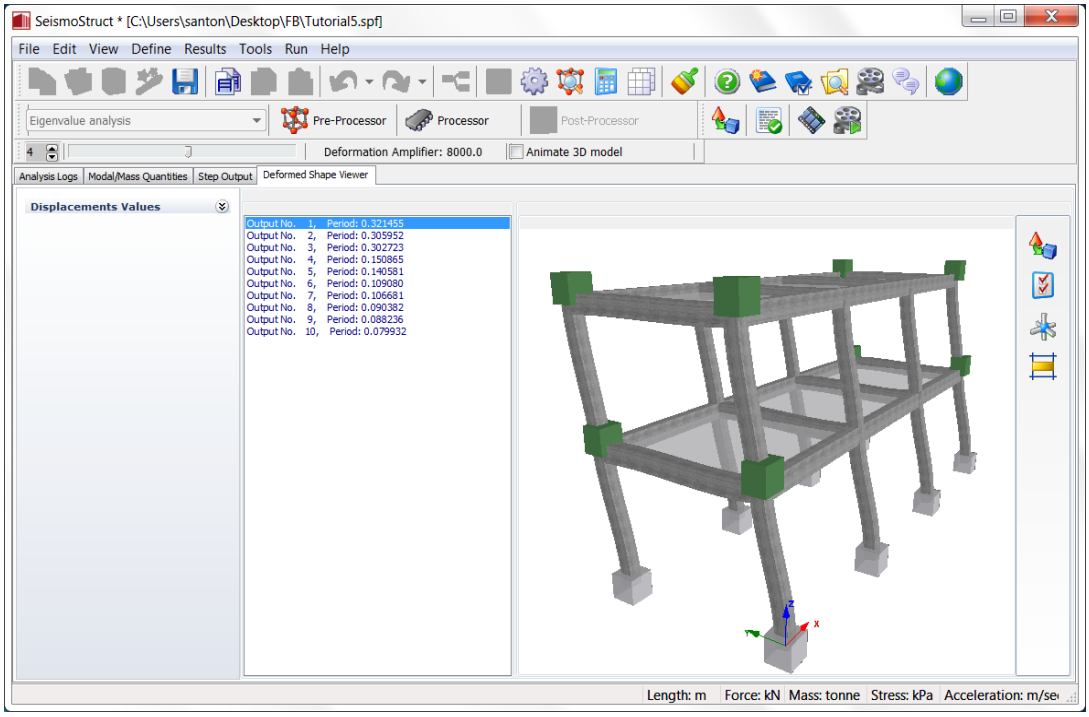
The **Step Output** module provides, for each eigen-solution found by the software, all the nodal displacements.



Step Output module

Post-processor – Deformed Shape Viewer

Finally, as in the previous tutorials, in the **Deformed Shape Viewer** module you have the possibility of visualising the deformed shape of the model at every step of the analysis. Double-click on the desired output identifier to update the deformed shape view (see figure below).



Deformed Shape Viewer Module

In addition, you can also visualise the displacement values by checking the "Displacement Values Display" box (see figure above).

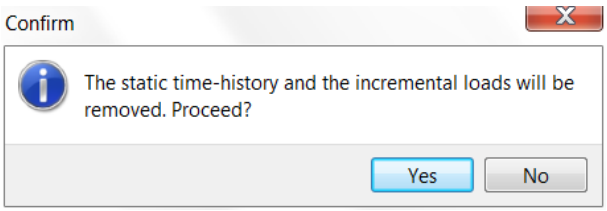
TUTORIAL N.6 – DYNAMIC TIME-HISTORY ANALYSIS OF A TWO-STOREY BUILDING

Problem Description

Also in this case, in order to quicken the procedure, let us use the model that has already been created in Tutorial n.4 and modified in Tutorial n.5.

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 SeismoStruct project (Tutorial 5.spf). Once opened, save the project with a new name through *File > Save as...* menu command. At this point, select the *Dynamic time-history analysis* from the drop-down menu at the top left corner in the **Pre-Processor** area. Since the program kept the incremental loads of tutorial 1 in memory, before proceeding it is required to confirm for their removal (see figure below).

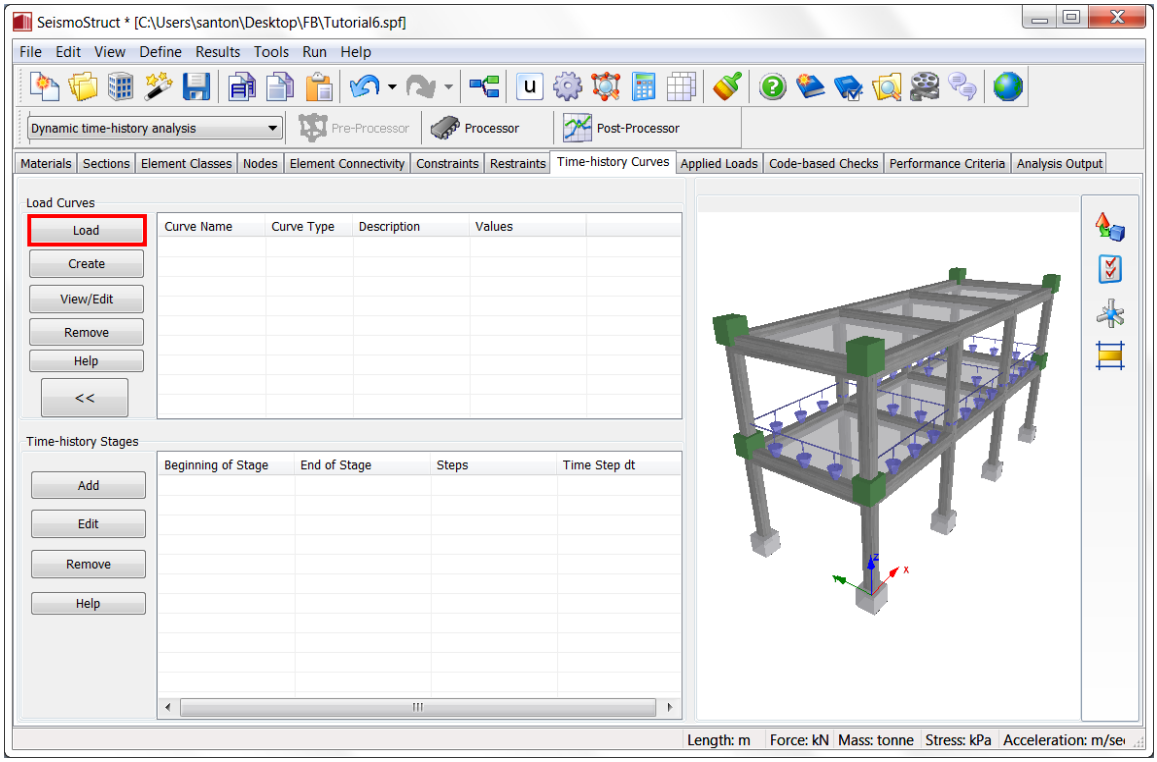


Warning message

After pressing the *Yes* button, go to the **Time-history Curves** module.

Pre-Processor – Time-history Curves

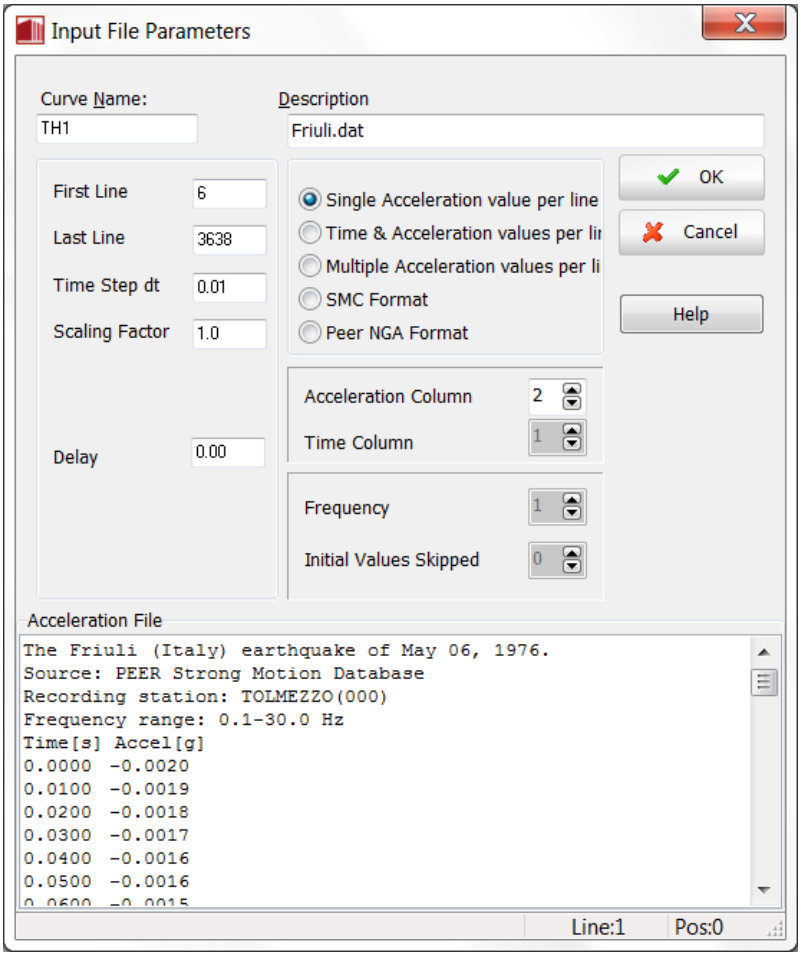
Press the *Load* button of the *Load Curves* section.



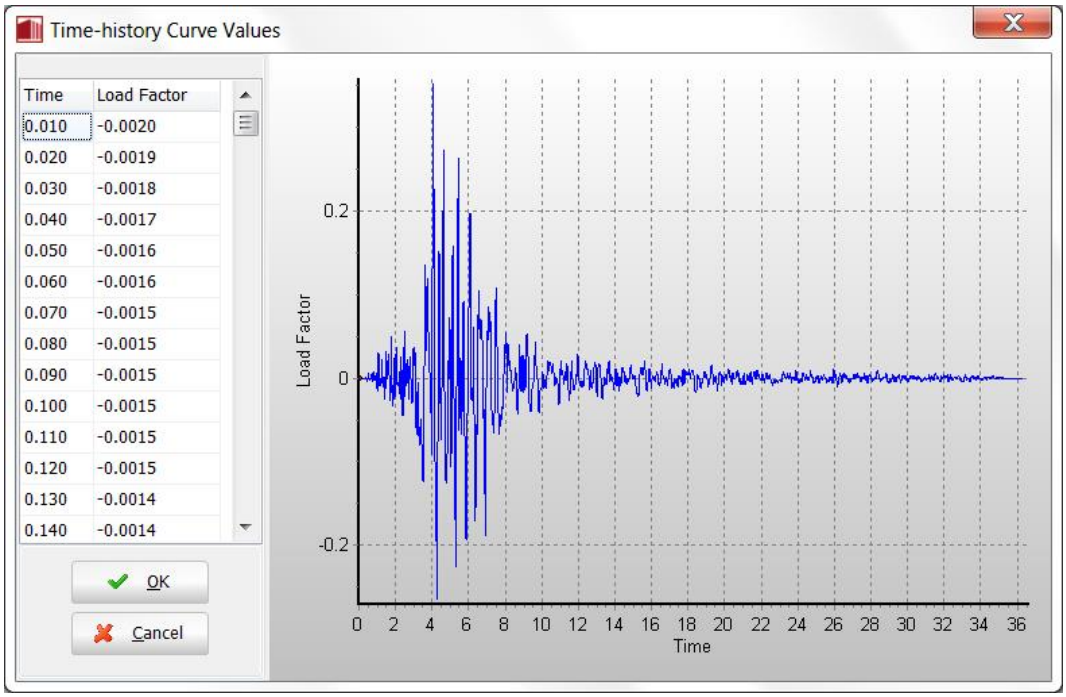
Time-history Curves module

In the new window you have to:

1. Load an accelerogram through the *Select File* button (for simplicity we will upload one of the curves in the installation folder of the program *(C:\ Program Files\ Seismosoft\ SeismoStruct_2026\ Accelerograms \ Friuli.dat*);
2. Assign the curve name (➔ TH1).

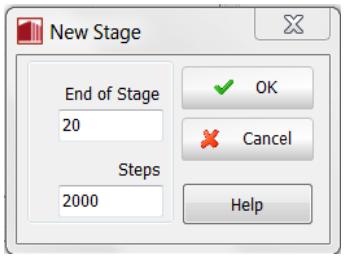


Load Curve - Input File Parameters



Load Curve - Time-history Curve Values

Once loaded the curve, you must define an analysis stage. So, in the *Time-history stages* section press the *Add* button. In the new window, set (i) the time of the *End of Stage* (which, in this example, coincides with the final time of the accelerogram, i.e. 20 sec) and (ii) the number of *steps* (-> 2000).

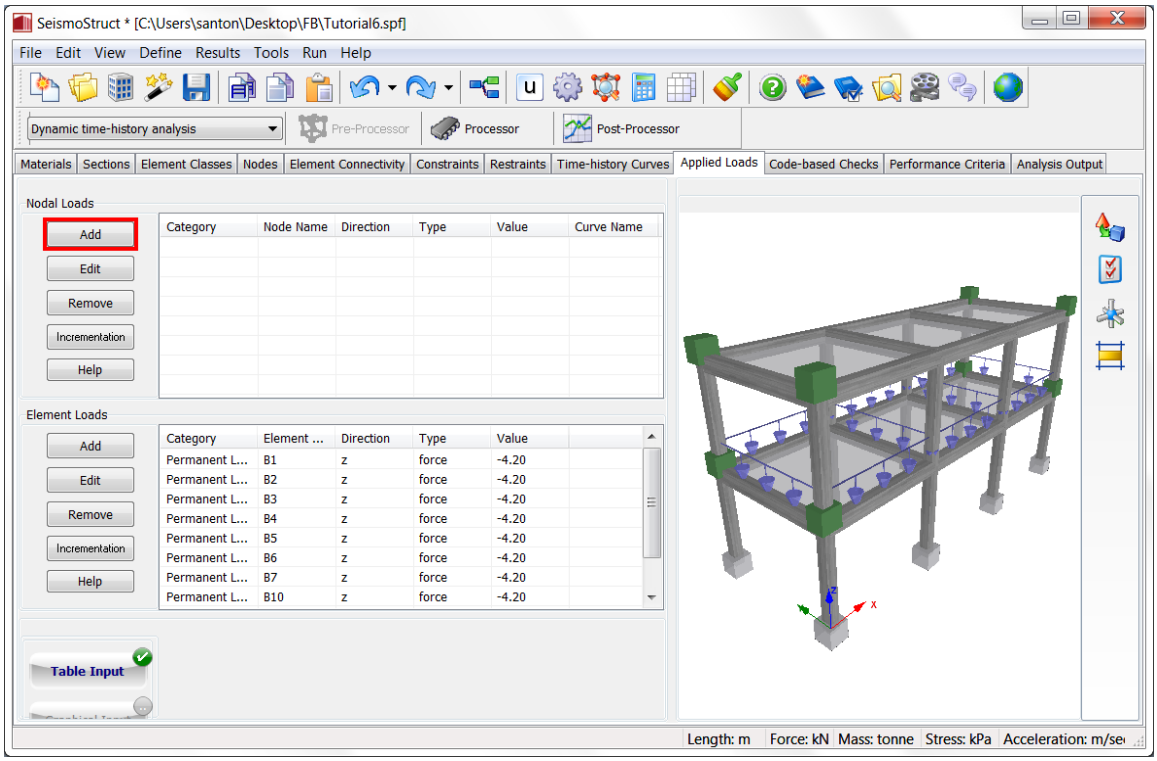


Time-history stage

NOTE: The program computes internally the time step dt. In this case it is equal to 20/2000 = 0.01

Pre-processor – Applied Loads

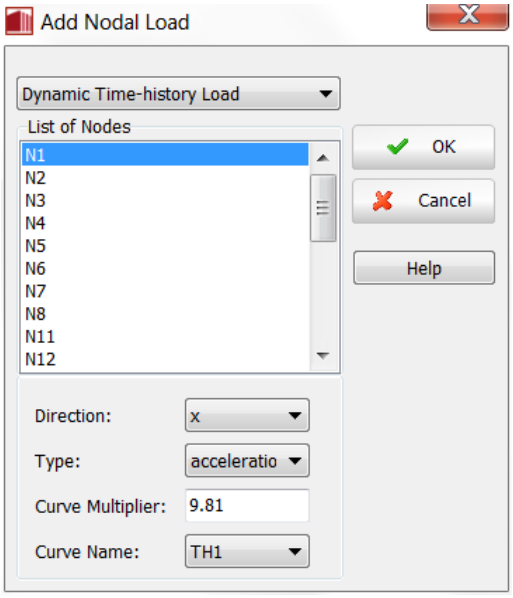
At this point it is necessary to apply the curve to the structural model. So, go to the **Applied Loads** module and click on the *Add* button.



Applied Loads Module

In the new window you have to:

1. Select the load category from the drop-down menu (➔ Dynamic Time-history Load);
2. Specify the associated node (➔ N1);
3. Select the load direction from the drop-down menu (➔ X);
4. Select the load type from the drop-down menu (➔ acceleration);
5. Specify the curve multiplier (➔ 9.81);
6. Select the curve name from the drop-down menu (➔ TH1).

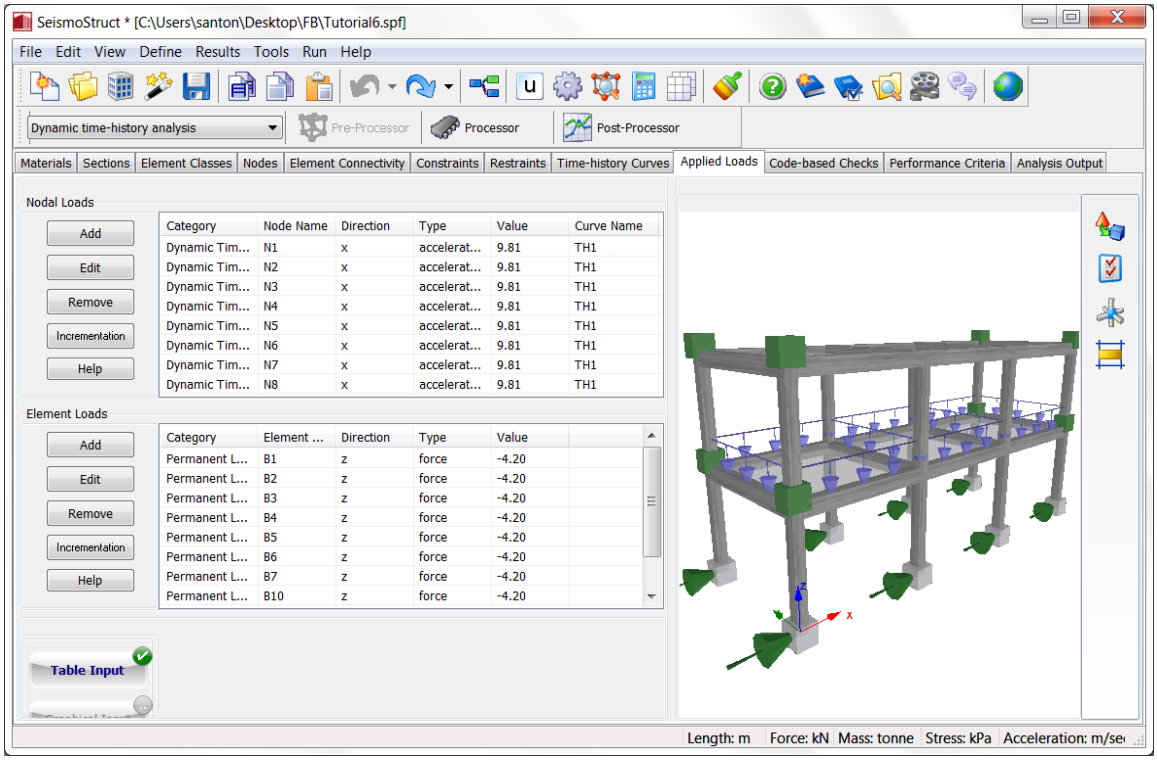


New applied load window

Repeat the same procedure in order to apply the other dynamic time-history loads to the base nodes.

In the table below all the applied loads are summarized:

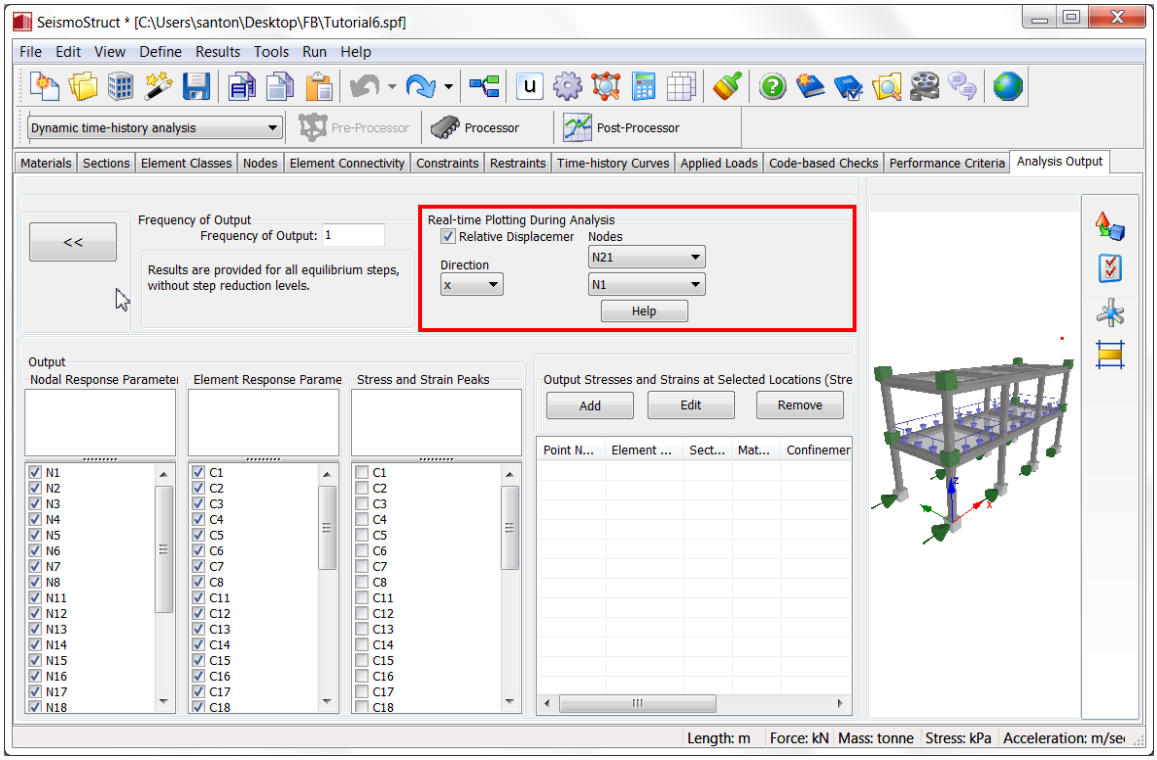
Category	Node name	Direction	Type	Curve multiplier	Curve
Dynamic Time-history Load	N1	X	acceleration	9.81	TH1
Dynamic Time-history Load	N2	X	acceleration	9.81	TH1
Dynamic Time-history Load	N3	X	acceleration	9.81	TH1
Dynamic Time-history Load	N4	X	acceleration	9.81	TH1
Dynamic Time-history Load	N5	X	acceleration	9.81	TH1
Dynamic Time-history Load	N6	X	acceleration	9.81	TH1
Dynamic Time-history Load	N7	X	acceleration	9.81	TH1
Dynamic Time-history Load	N8	X	acceleration	9.81	TH1



Dynamic time-history loads

Pre-Processor – Analysis Output

Finally, before entering the **Processor**, you must set your output preferences in the **Analysis Output** module, as shown in the figure below.



Analysis Output module

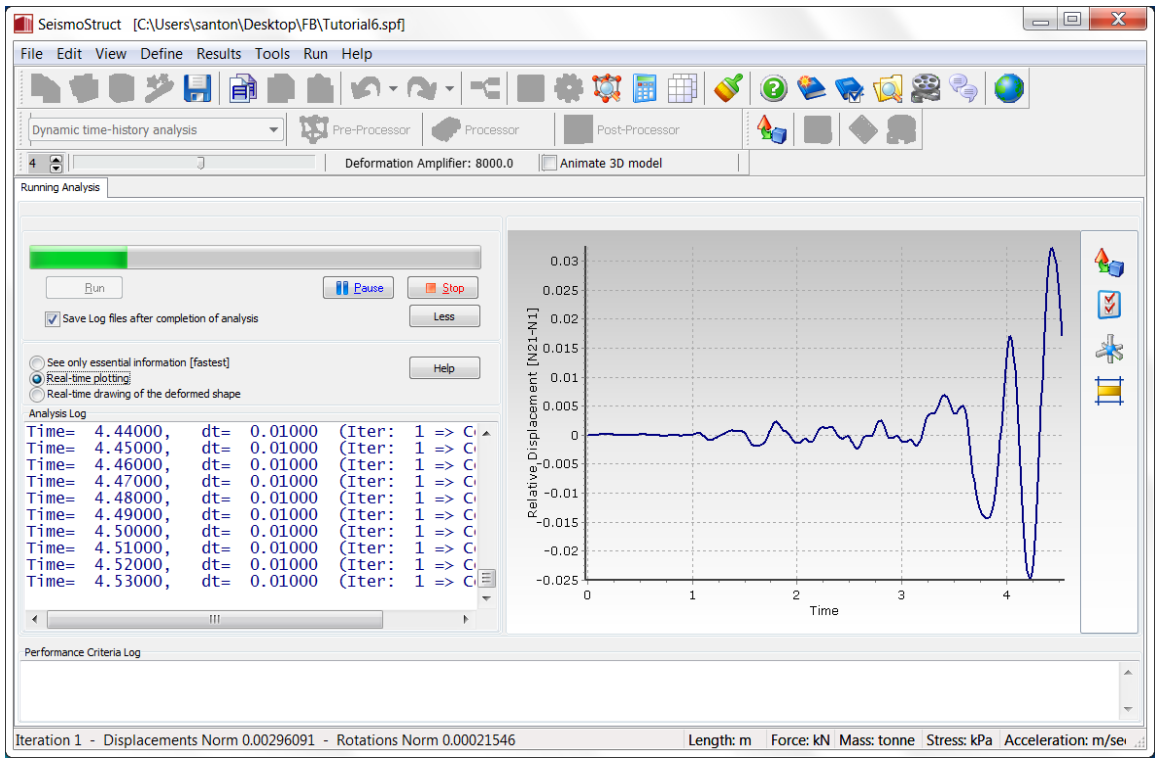
NOTE: Unlike the tutorial 1, in this example we ask to visualize, in the real-time plotting, the total relative displacement of the top node N21 with respect to the base node N1.

At this point you may click on the toolbar button 

or select *Run* > *Processor* from the main menu in order to perform the dynamic time-history analysis.

Processor

Press the *Run* button.



Running the analysis

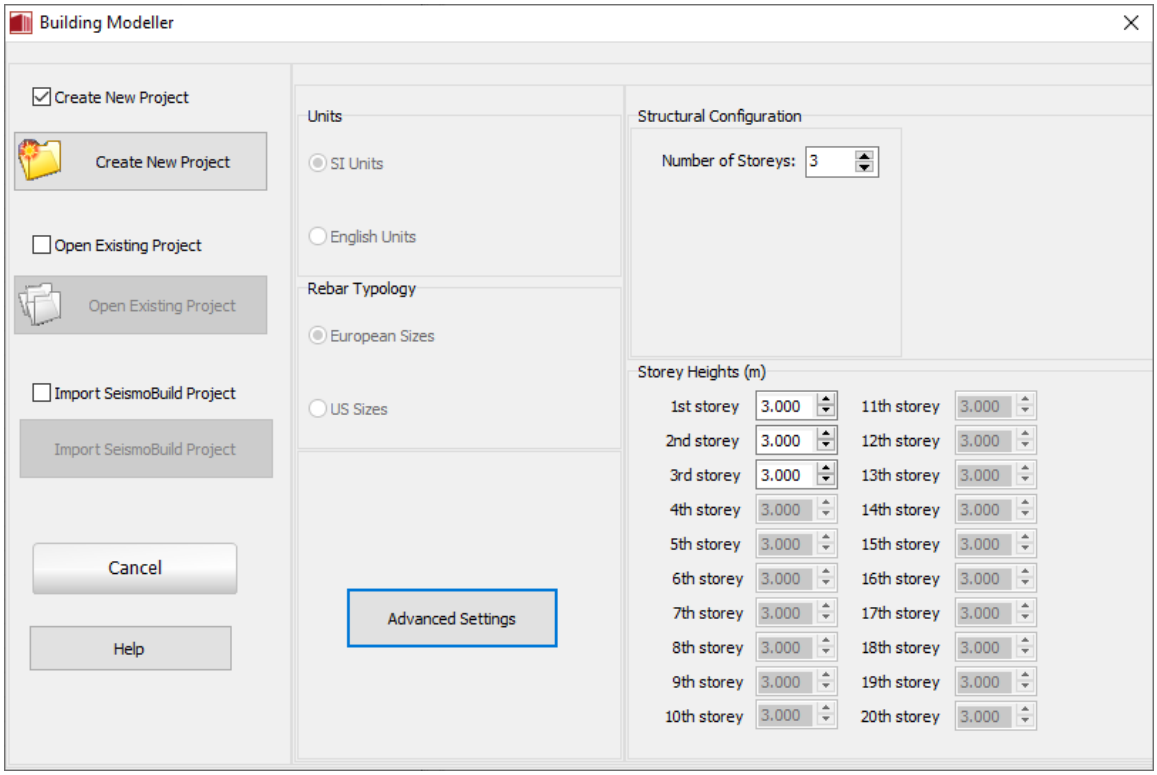
Once the analysis has arrived to the end, click on the toolbar button 

to get the results. As already seen for the previous tutorials, in the **Post-Processor** you will be able to see the deformed shape of the structure at each step of dynamic analysis (*Deformed Shape Viewer*) as well as to extract the time-history displacement response of the structure, and so on.

BUILDING MODELLER

A special modeller facility has been developed and introduced in the program in order 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 main menu (*File* > *Building Modeller...*) or through the corresponding toolbar button  .



Building Modeller Facility window

Basic Settings and Structural Configuration

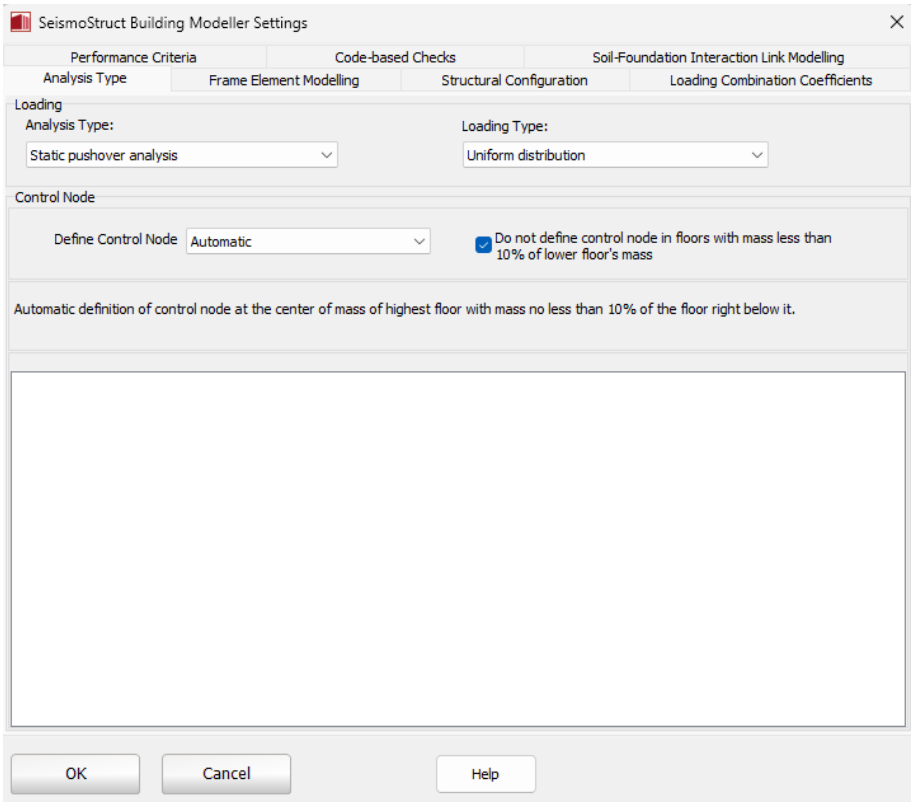
SI units or English units can be selected, as well as European or U.S. sizes in the rebar typology. The number of storeys and their heights are also defined; a number from 1 to 100 storeys, with different heights at each storey and the possibility of applying a common height to a range of storeys, may be selected. Up to three underground floors (basement storeys) and their heights may also be defined.

Advanced Settings

In the Advanced Settings dialog box, accessed by the corresponding button, the following information can be defined:

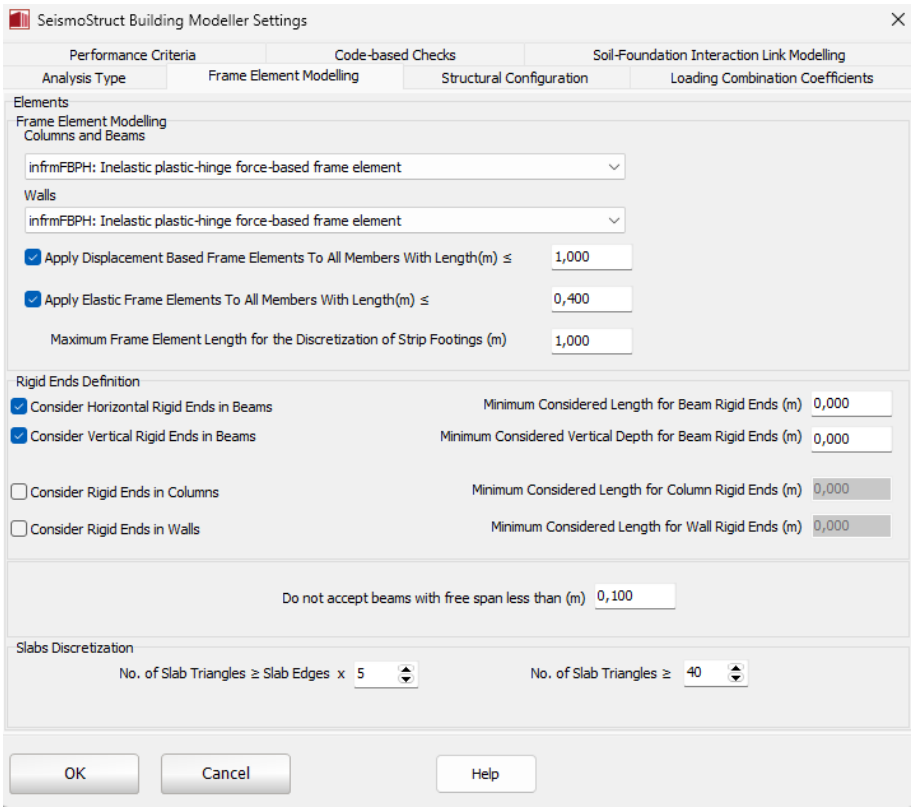
- **Analysis Type**: The type of analysis for which the model will be created. All nine SeismoStruct analyses types are supported.

  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 centre 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).



Advanced Settings module – Analysis Type

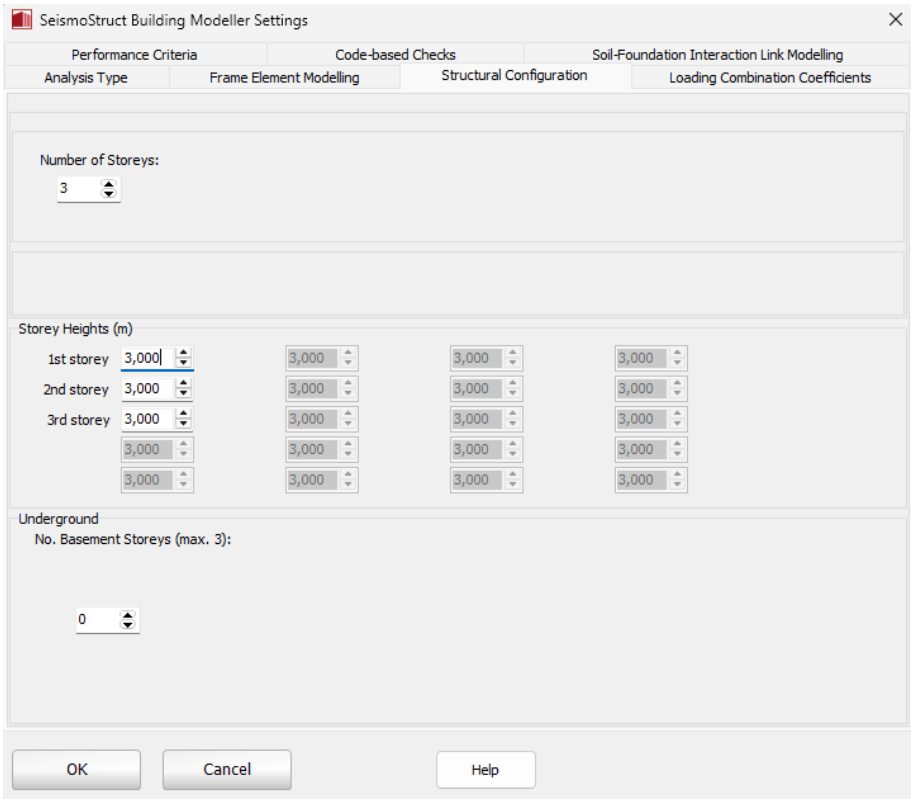
- **Frame Elements Modelling**: The element class to be used to model the structural members is defined herein. Different frame element types may be employed for columns/beams and walls. Further, it is possible to assign the inelastic displacement-based frame element type (infrmDB) to short members, a choice that improves both the accuracy and the stability of the analysis. Users can determine the maximum length of the short members, below which the infrmDB element type is employed (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, a scheme that 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). The choice whether to include or not rigid ends in the beams, columns and walls modelling is also done herein. It is noted that these rigid ends are included in the model, when the length of a member's rigid end is larger than the specified value. The option to reject beams shorter than a defined minimum length is provided to prevent the inadvertent creation of very short beams for graphical reasons (e.g., when a beam end is slightly extended beyond the adjacent column). Finally, the slab discretisation parameters may also be defined here.



Advanced Settings module – Frame Elements Modelling

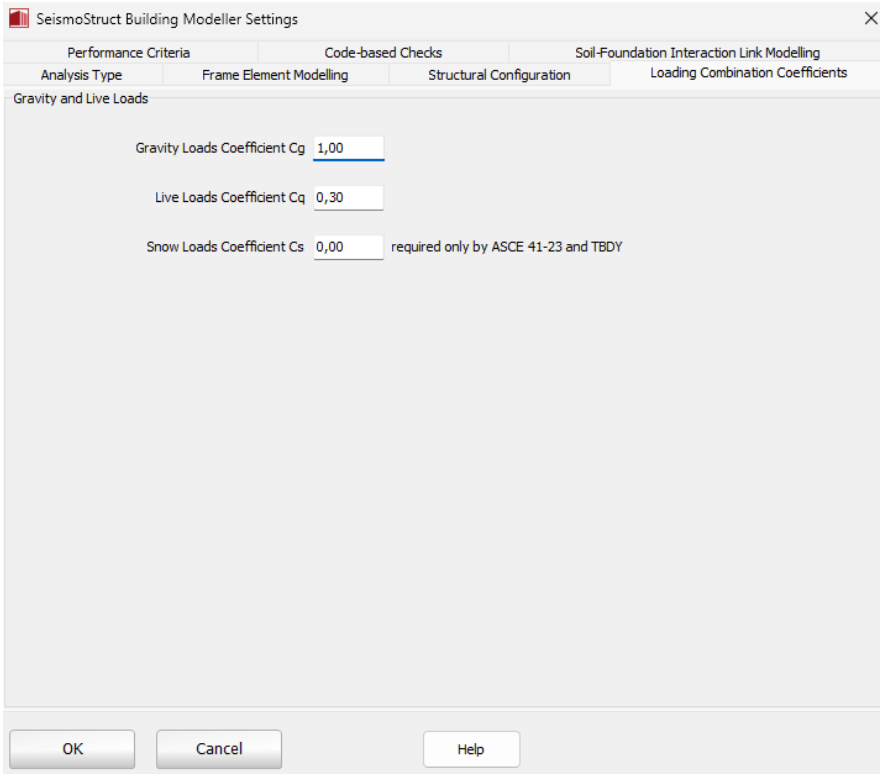
NOTE: Even when no rigid ends are defined by the user, offsets may automatically be introduced to ensure adequate alignment of all structural elements.

- **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.



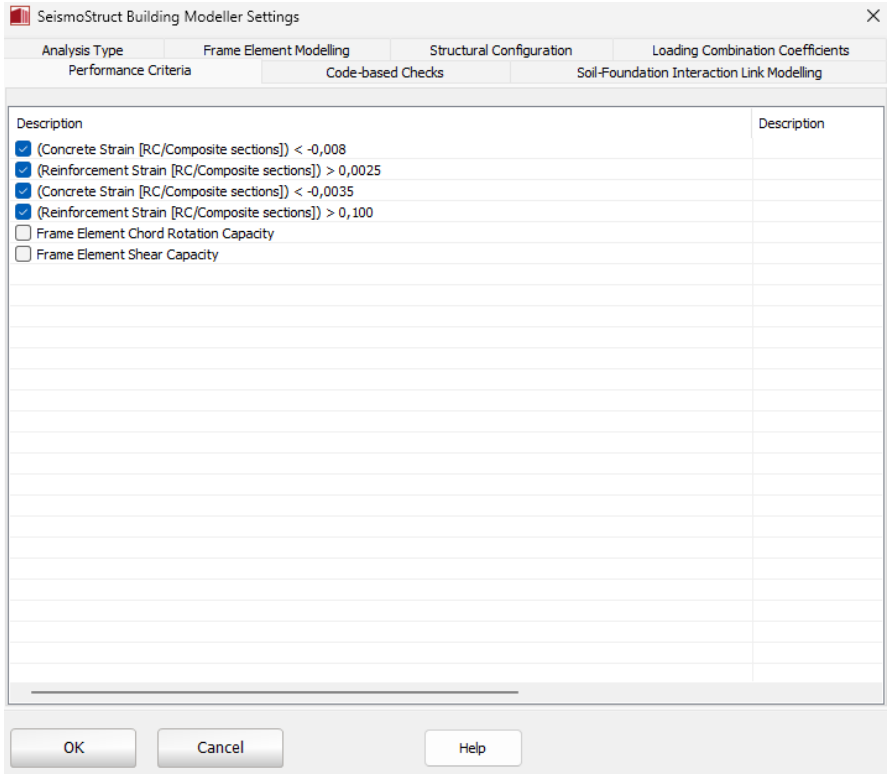
Advanced Settings module – Structural Configuration

- **Loading Combination Coefficients**: The loading combination coefficients for the Seismic Combination (e.g. G+0.3·Q±E) of the slabs' permanent, live and snow (in the case of ASCE 41-23 and TBDY) loads are defined here. The loading of the slabs is defined for each slab separately in the slabs' Properties Window.



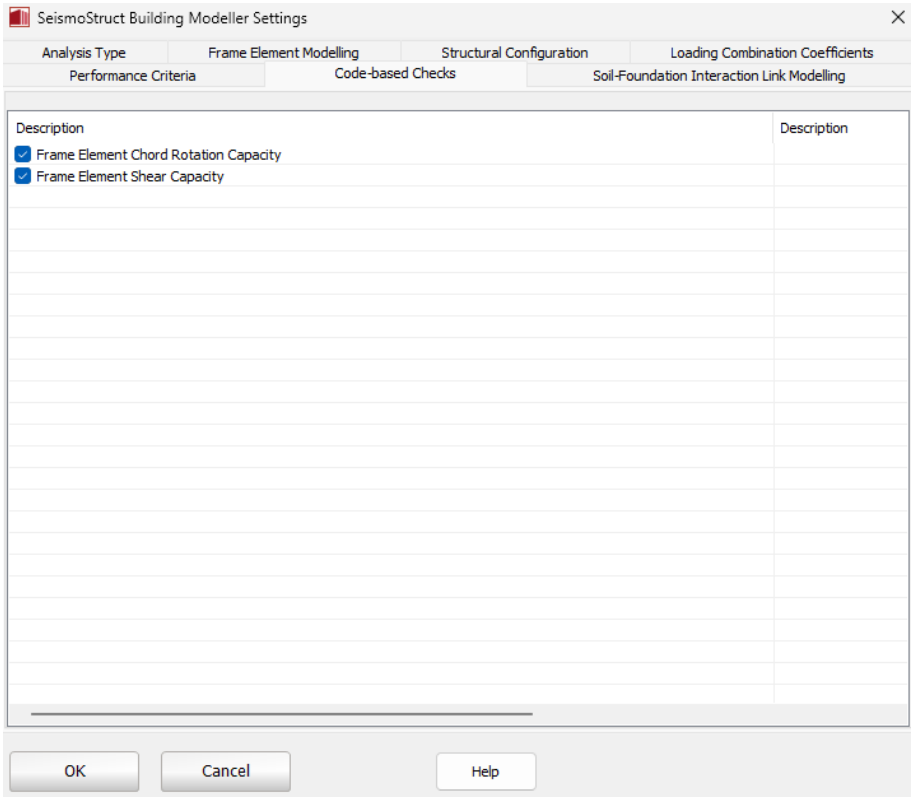
Advanced Settings module – Loading Combination Coefficients

- **Performance Criteria**: Users are able to select which types of performance criteria to include in their analysis. By default, the chord rotation and shear capacities criteria are not selected.



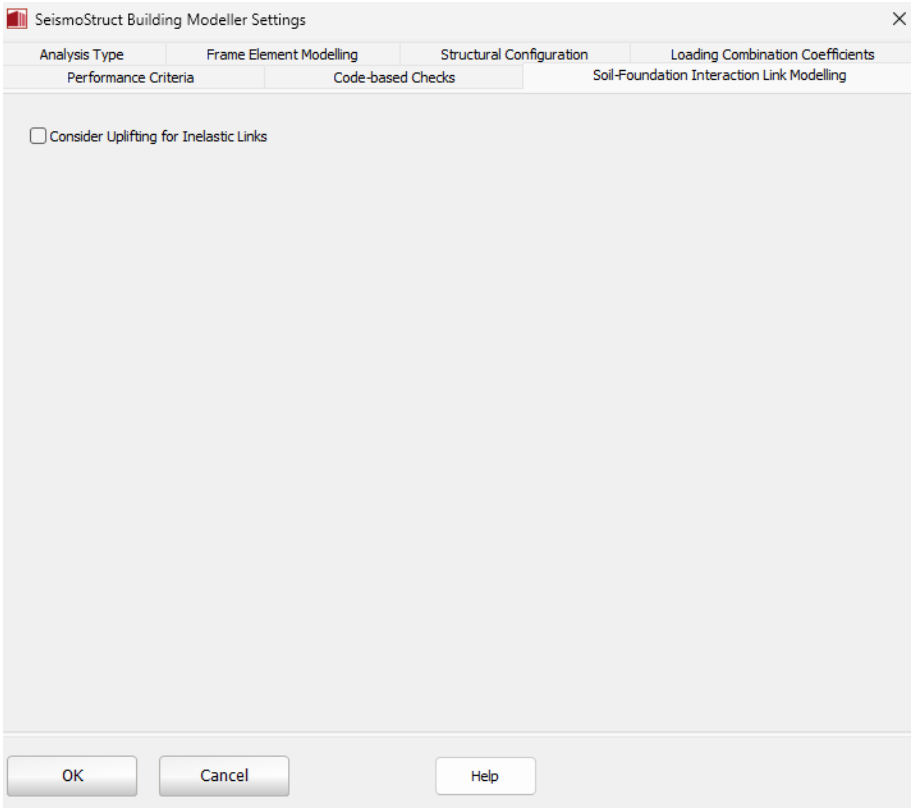
Advanced Settings module – Performance Criteria

- **Code-based Checks**: Users are able to select which types of code-based checks to include in their analysis. By default, both the chord rotation and shear capacities checks are selected.



Advanced Settings module – Code-based Checks

- **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.



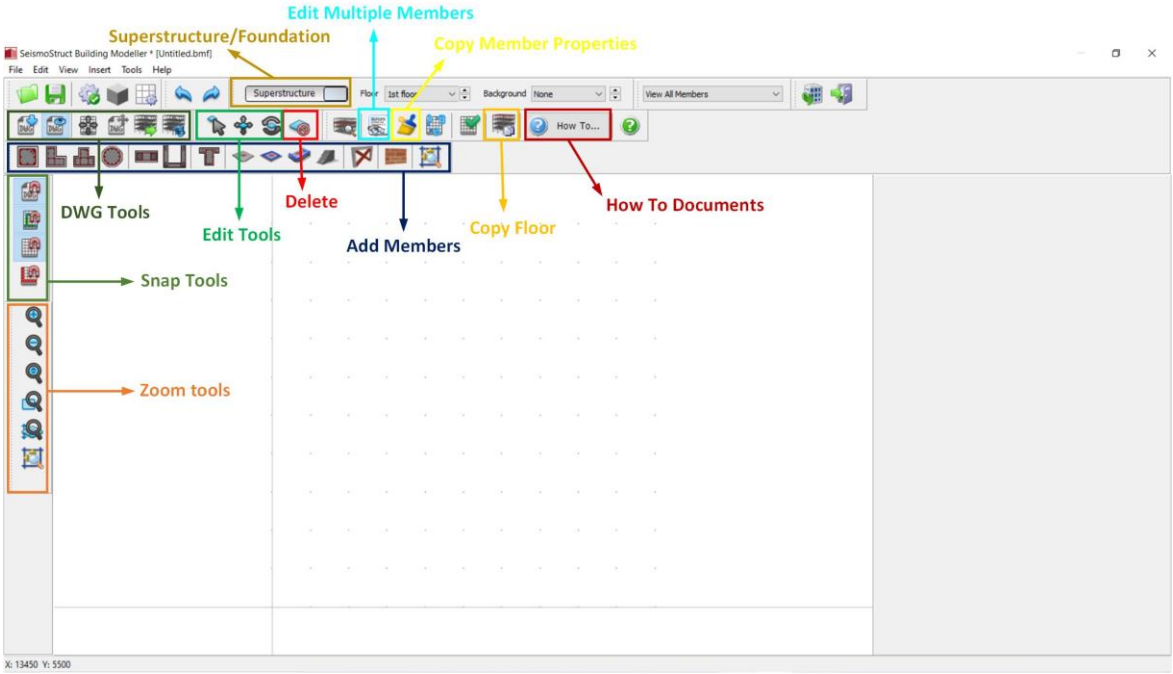
Advanced Settings module – Soil-Foundation Interaction Link Modelling

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

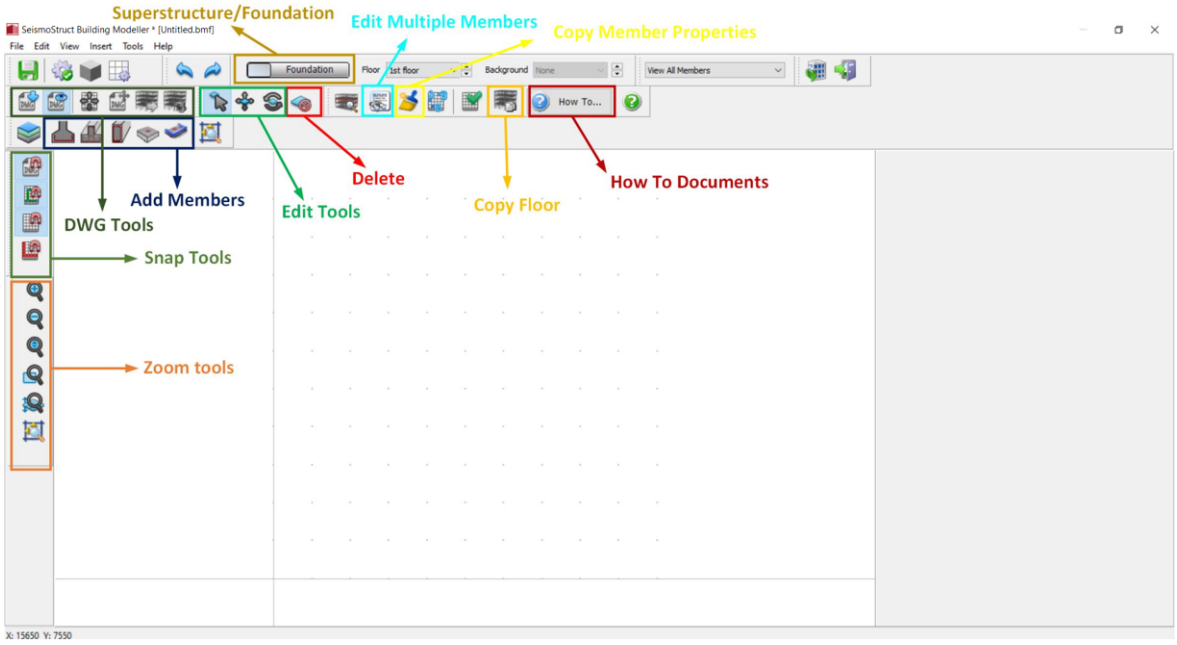
Building Modeller Main Window

After selecting the main settings, users are able to define the geometry of the new building by selecting the *Create New Project* button. The Building Modeller Main Window will appear, 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)

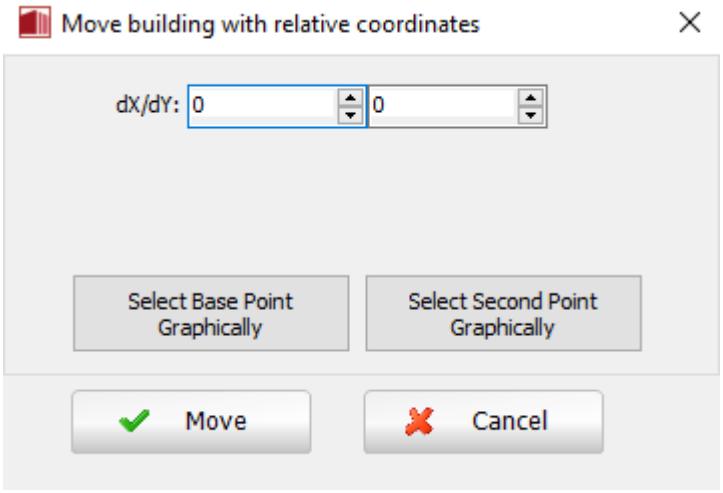


Building Modeller Main Window (Foundation Mode)

The possibility of inserting 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 assign drawing's units and whether to move the DWG/DXF file to 0,0, i.e. to the origin of the coordinates system. Selecting the check-box will move the bottom-left edge of the drawing to the (0,0) coordinates, irrespective of its initial CAD coordinates. Note that the axes origin can be further moved to a different point that might be more suitable with the *Move Axes Center* toolbar button, also accessible from the 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 Menu

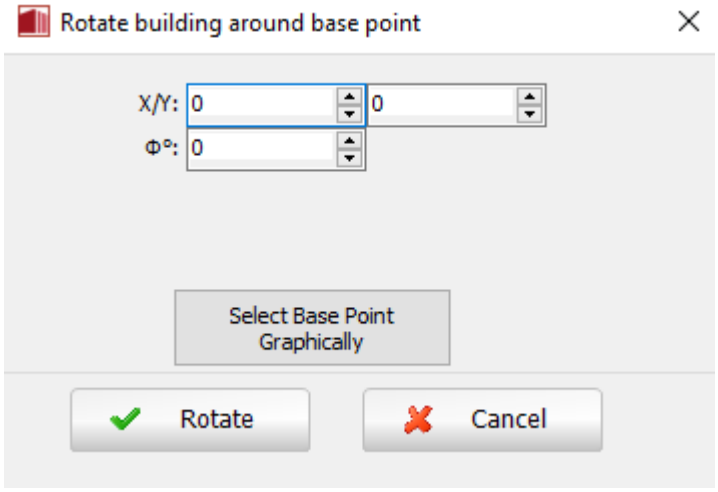
(View > Show/Hide DWG) or through the toolbar button the option whether the CAD drawing will be visible or not is defined.

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 coordinares 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.



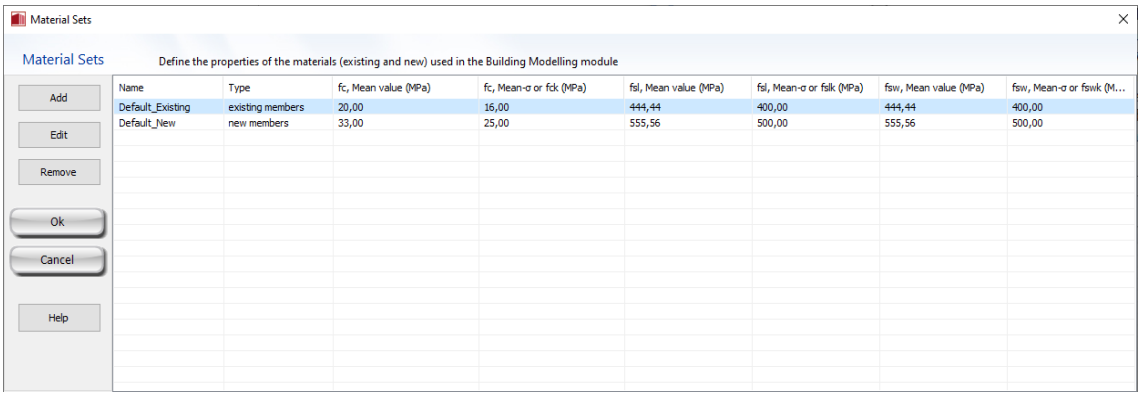
Rotate Building window

Insertion of Structural Members

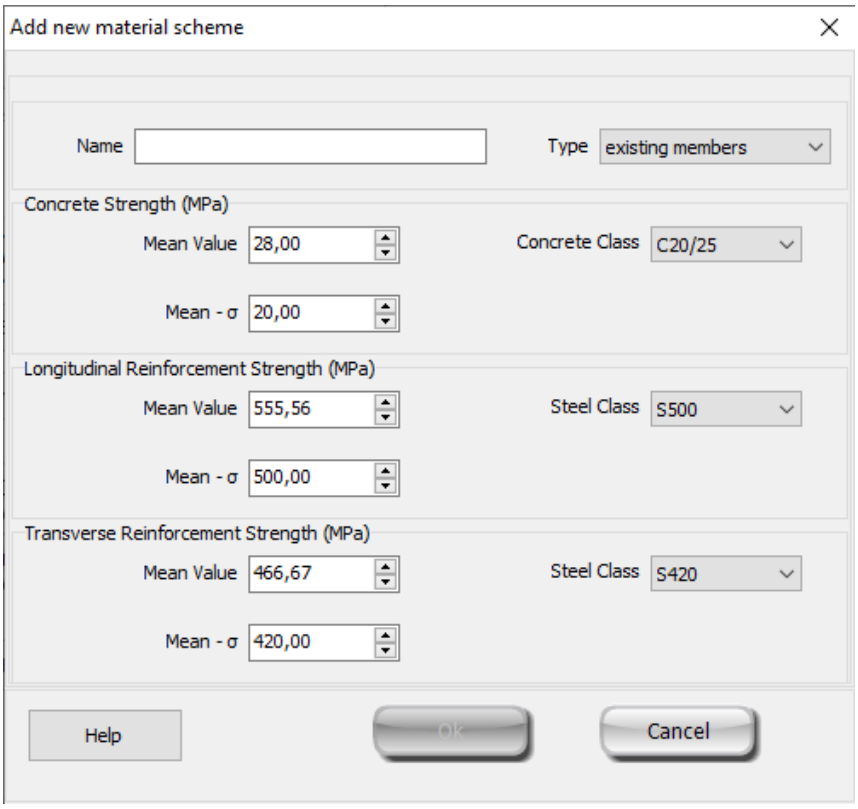
The Material Sets, the Advanced Member Properties, the Modelling Parameters are common to all the sections' properties windows while *FRP Wrapping* is available only for RC members and Jacket is available for RC columns and RC beams. Note that a How-To documents list is introduced for a quick access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

Material Sets

The Material Sets properties can be defined from the 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 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 one 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 several different material strengths are employed in the structural system).



Material Sets Window



Add New Material Scheme

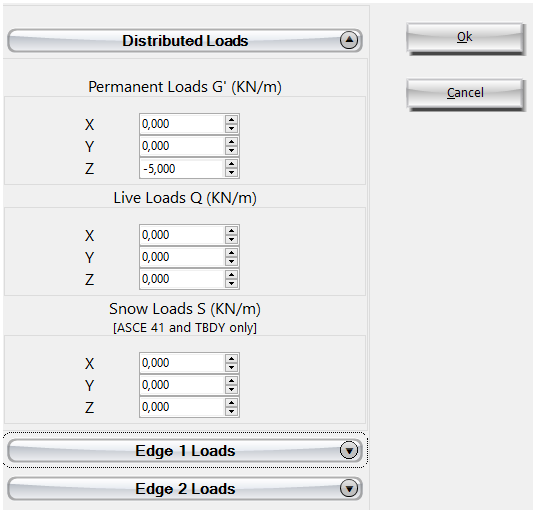
NOTE 1: There is a limit to the number of the defined material schemes equal to 10. The default material sets cannot be removed.

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.

In Steel sections only the characteristic yield strength needs to be specified and whether the member is new or existing.

Member Loading

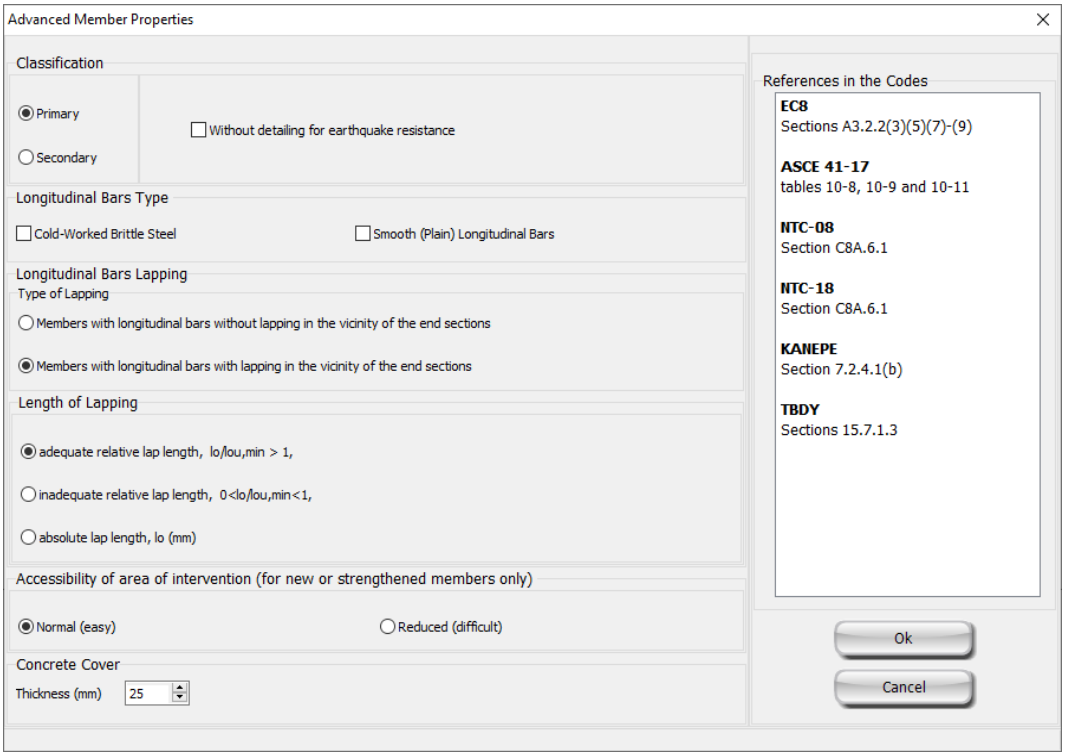
Additional loads can be defined by clicking on the Distributed and Edge Loads button for columns and walls and the More Loads button for beams. Users can define uniform distributed forces along the length of the member in all three translational directions X, Y or Z, and forces or moments in any translational 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

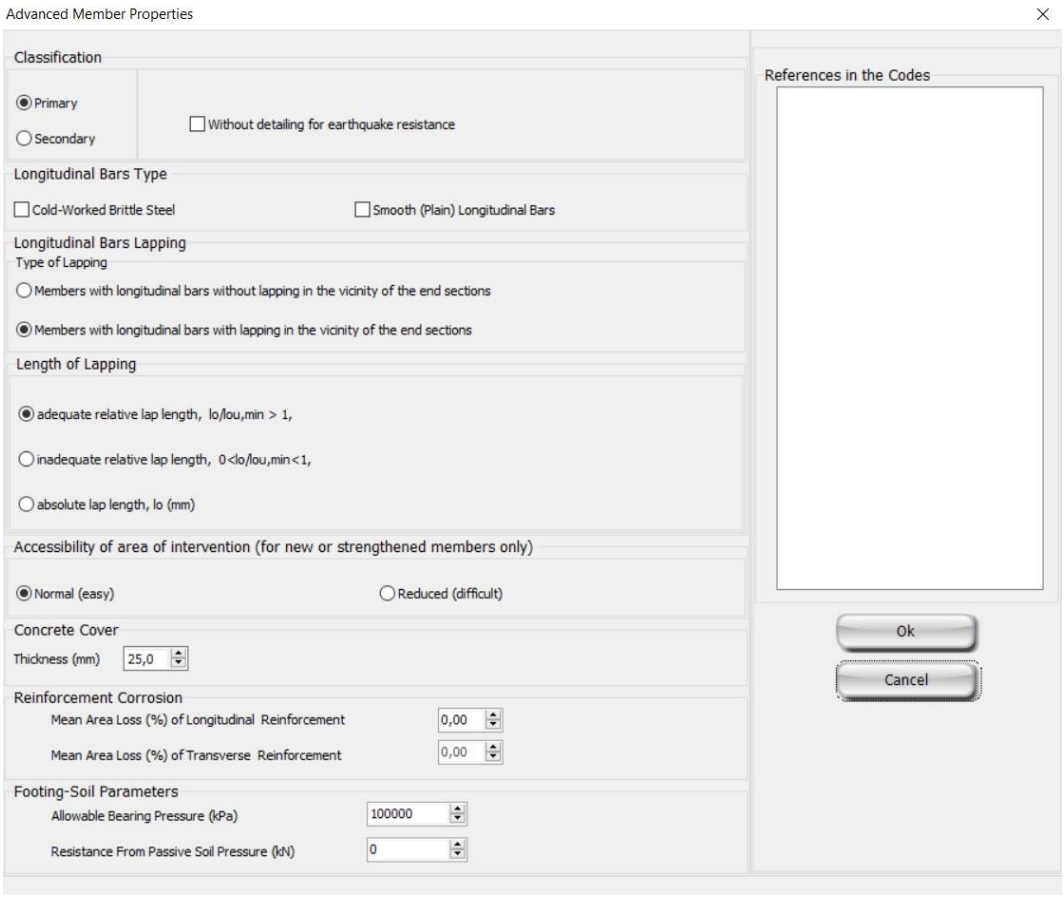
Advanced Member Properties

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 member'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, as well as the accessibility of area of intervention (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).



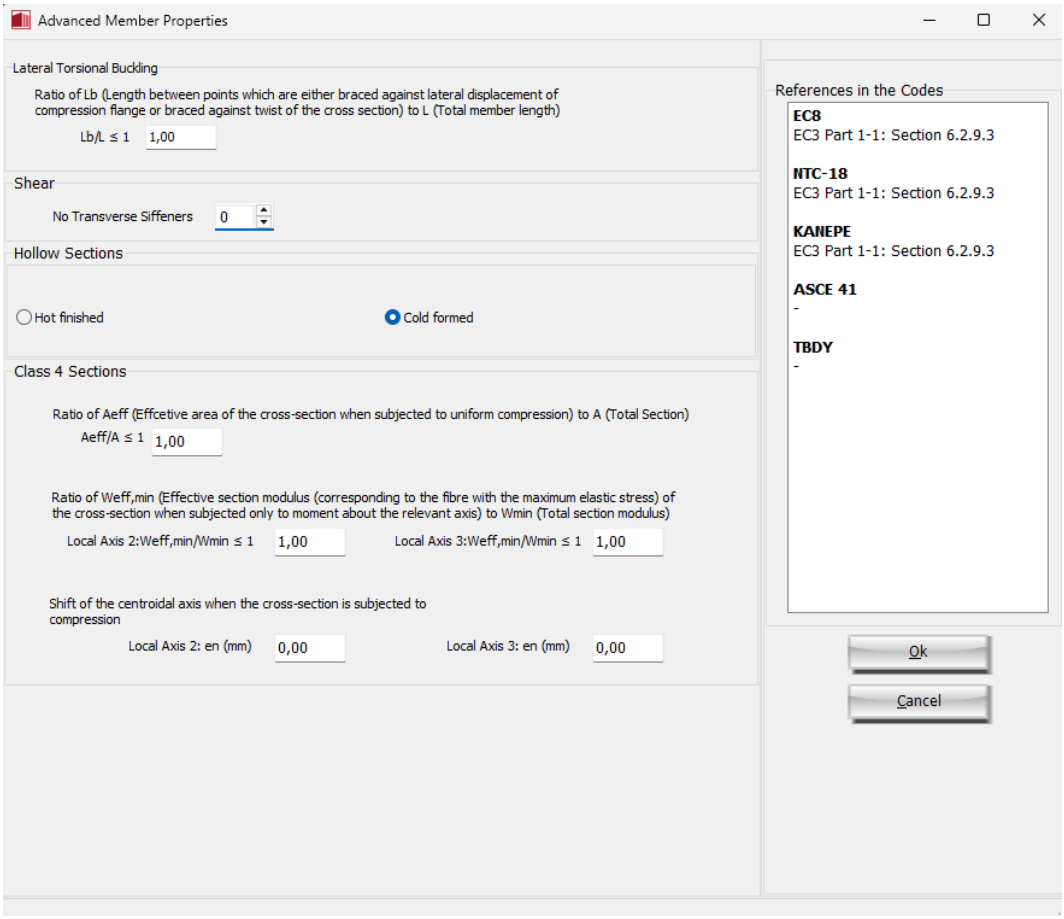
Advanced Member Properties module

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

For steel members, users may define several parameters related to code-based checks, such as the ratio of Lb (the length between points braced against lateral displacement of the compression flange or against twist of the cross-section) to L (the total member length), which must not exceed 1.00. Users may also specify the number of transverse stiffeners and indicate whether hollow sections are hot-finished or cold-formed. For Class 4 sections according to Eurocode, users may define the ratio of the effective area under uniform compression (Aeff) to the total area (A), as well as the ratios of the effective section modulus (Weff,min) to the total section modulus (Wmin) for both local axes 2 and 3. Additionally, users may assign the shift of the centroidal axis under compression (en) for each local axis. A dedicated panel is also provided for referencing the relevant design codes.



Advanced Member Properties module for Steel sections

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 SeismoStruct, 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 SeismoStruct 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 Modelling Parameters tab.

Fourteen material types are available within the Building Modeller of SeismoStruct, 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

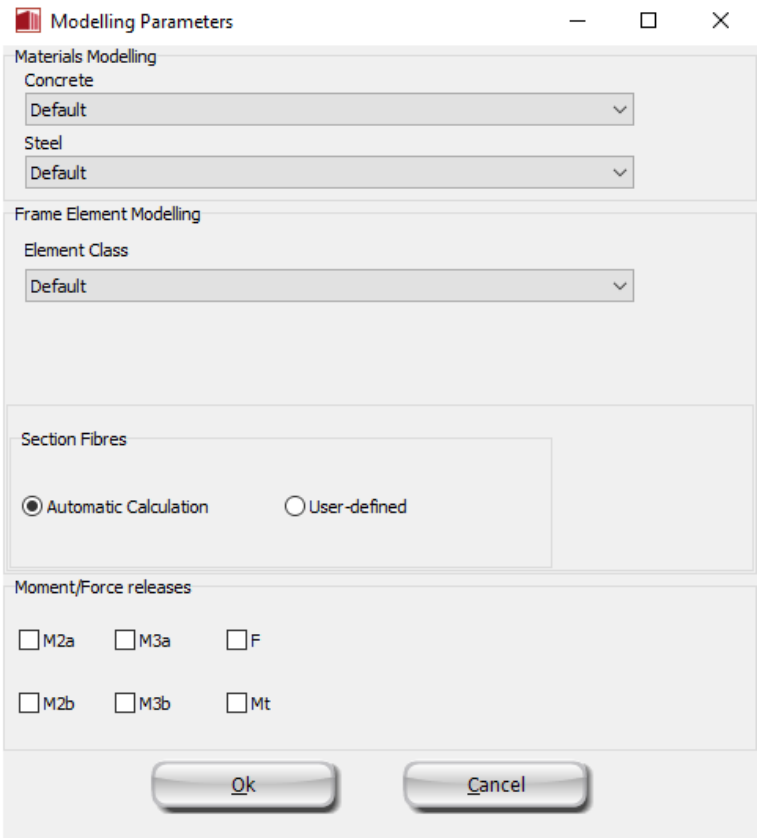
For a comprehensive description of the material types, refer to Appendix C – Materials.

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 1m^2, 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.

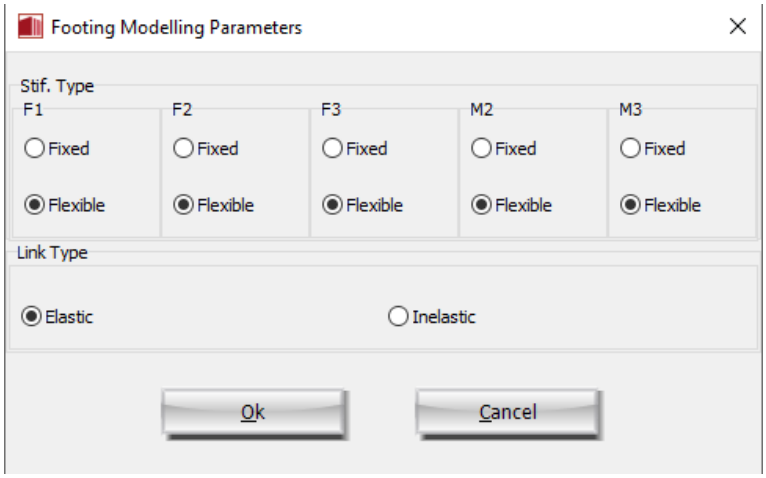
Finally, users may also 'release' one or more of the element degrees of freedom (forces or moments).



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.

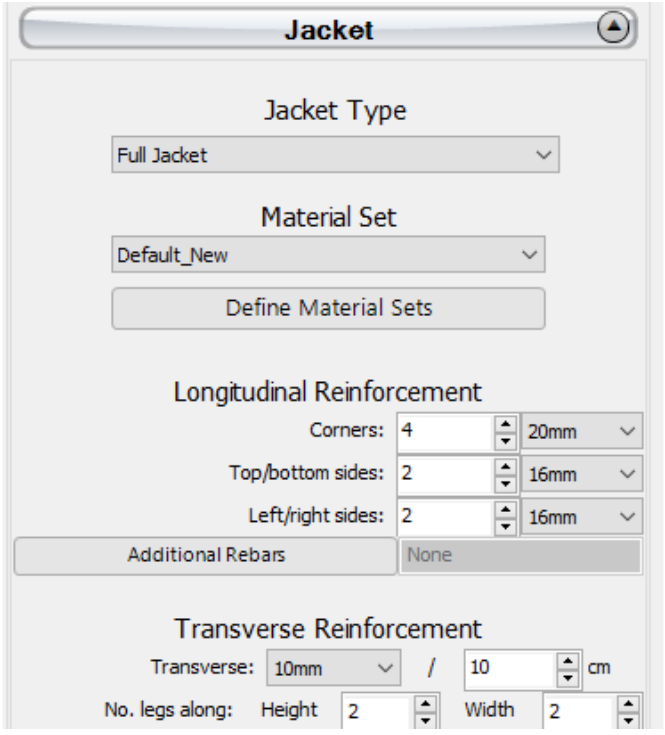


Footing Modelling Parameters

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.

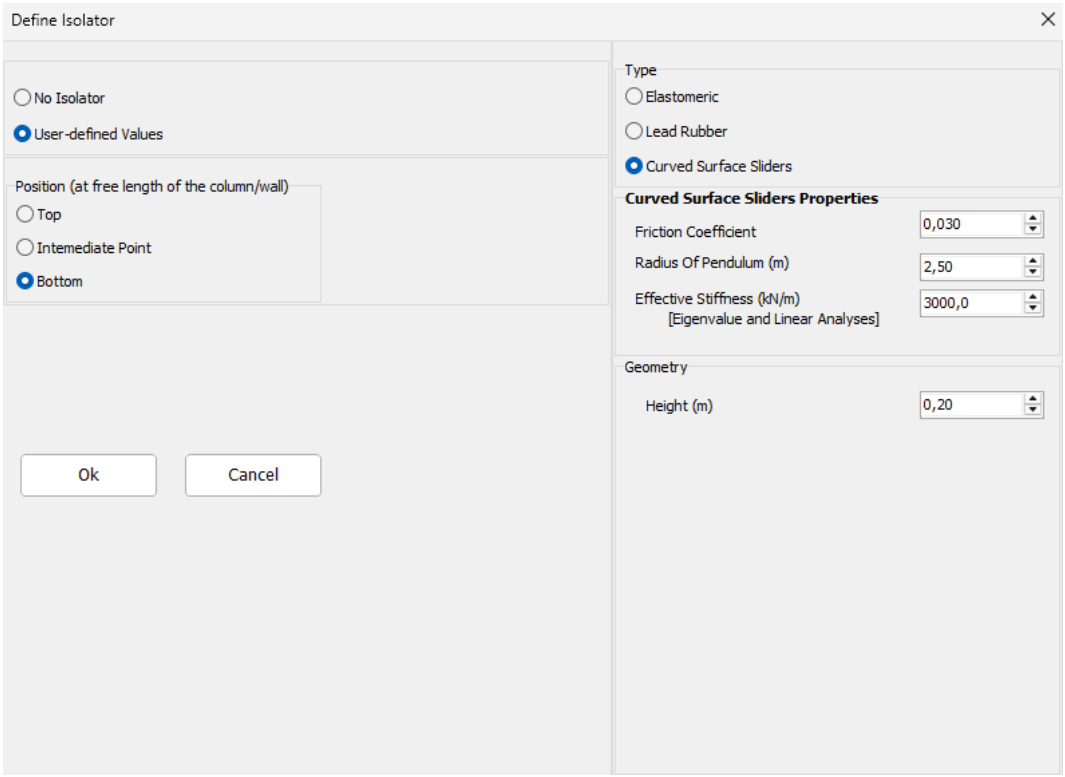
The material set of the jacketed sections, as well as the longitudinal and transverse reinforcement of the jacket may be defined.



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 radium of the pendulum (for curve surface sliders, also known as Friction Pendulum System (FPS)).

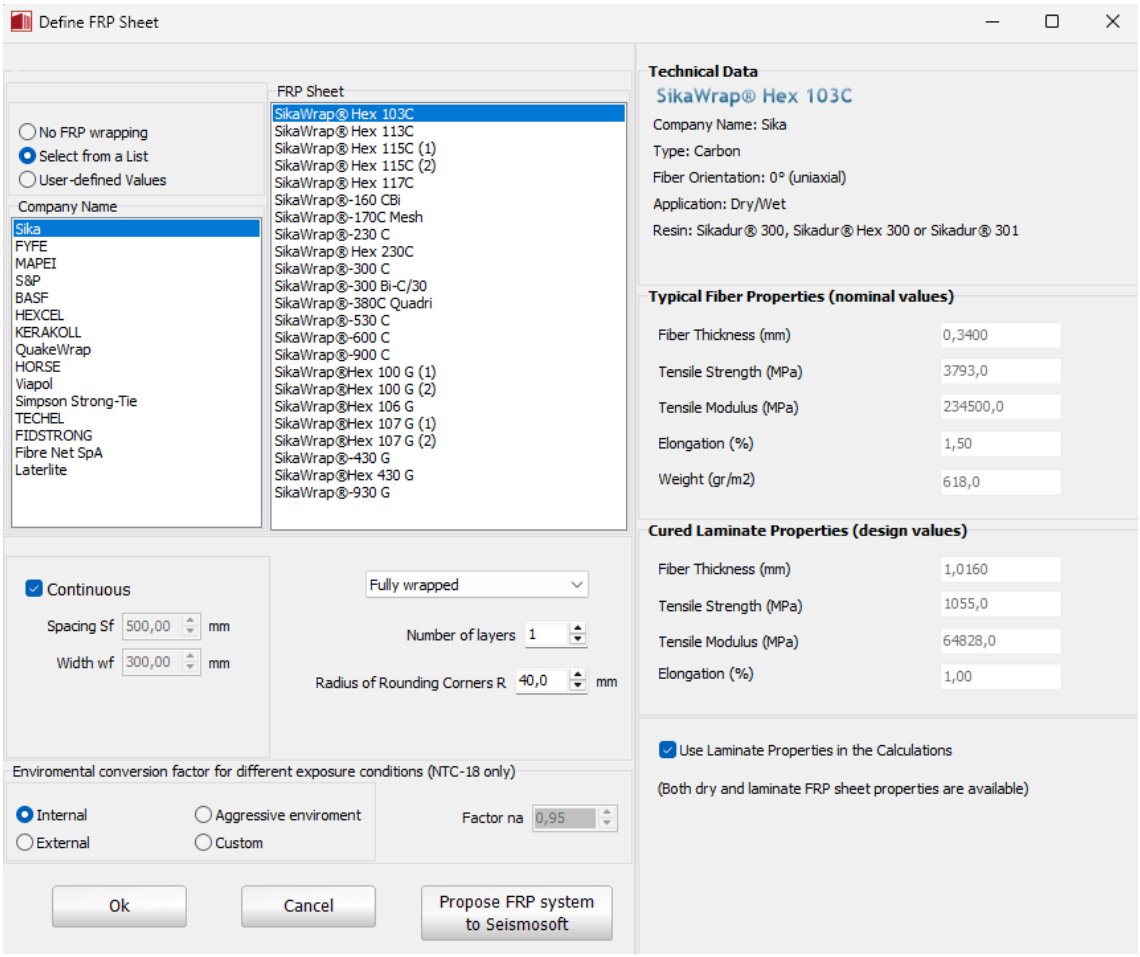


Isolator module

FRP Wrapping

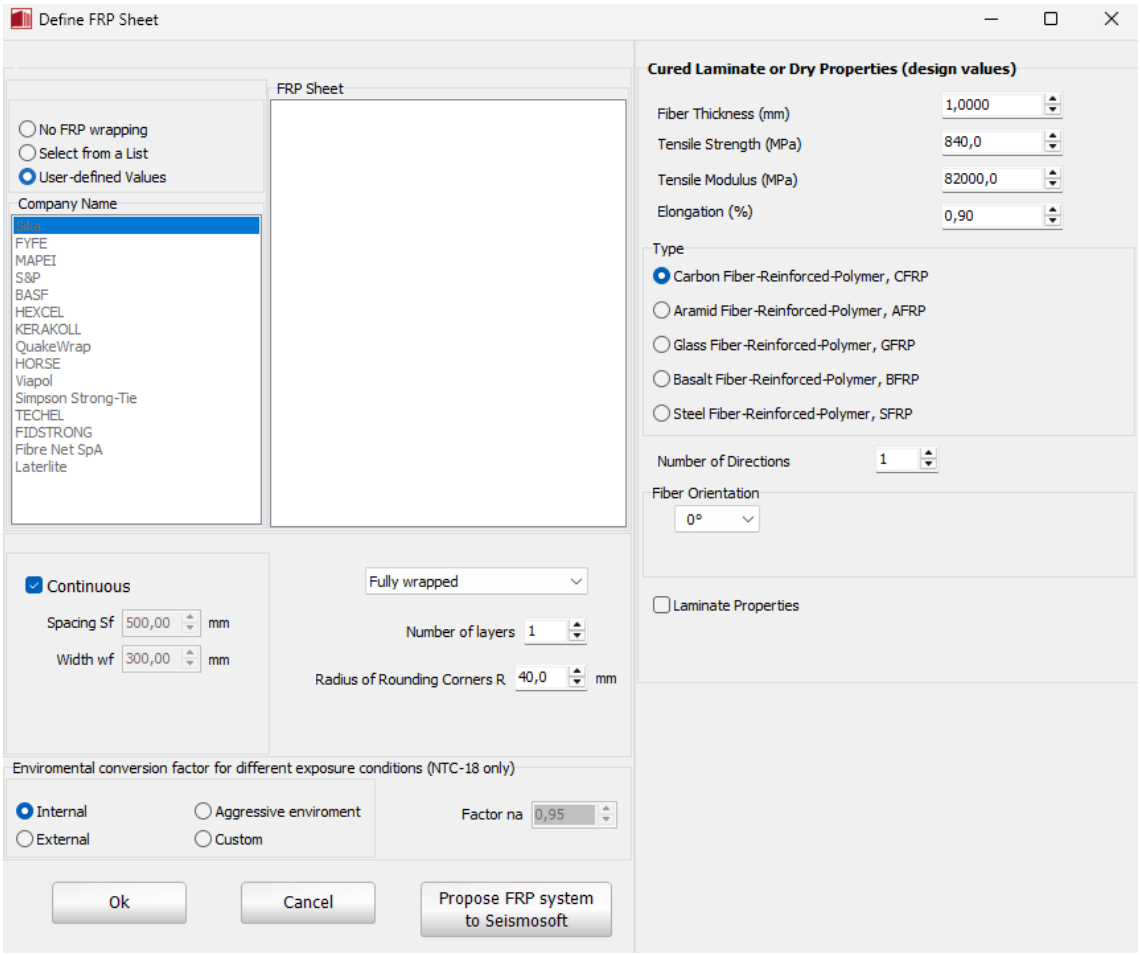
FRP wraps may be assigned to columns, walls and beams through the FRP Wrapping module. Users may select the FRP sheet from a list of the most commonly used products found in the market, or alternatively introduce user-defined values.

The number of applied layers may also be defined, as well as whether the dry or laminate FRP properties are to be used in the calculations. The user may additionally specify whether the FRP wrap is continuous, or define the spacing between the fabrics and the fabric width. Furthermore, the wrapping configuration around the existing member can be selected (for columns only the fully wrapped option is available, while for beams additional options such as 3-sided and 2-sided wrapping are provided, including the use of mechanical anchorage in open ends). Since rectangular cross-sections are considered, the corner rounding radius R, a critical parameter in the application of FRP wraps, may also be specified. Finally, the Environmental Conversion Factor needs to be defined.



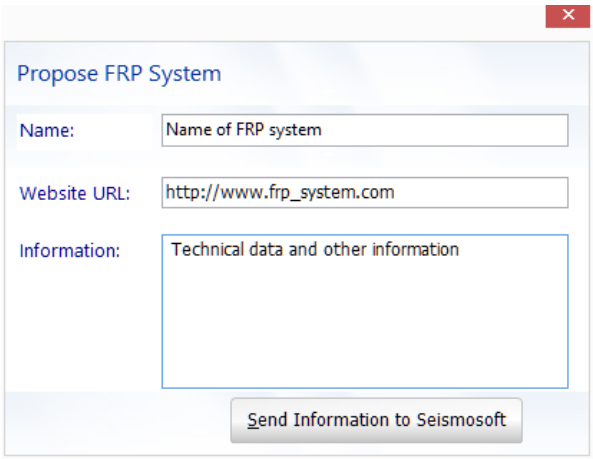
Select from a list module

When users choose to specify user-defined values, the required information includes: the type of FRP sheet (Carbon, Aramid, Glass, Basalt, or Steel fibres), its laminate or dry properties, the number of fibre directions and their orientation relative to the longitudinal direction of the sheet, as well as the wrapping configuration around the existing member. Users must also define whether the FRP wrap is continuous or specify the spacing between the fabrics and the fabric width, the number of layers, and the corner rounding radius R.



User-defined Values module

Finally, FRP systems may be proposed to Seismosoft through the "Propose FRP system to Seismosoft" button, in order to be included in newer releases of the program. 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.



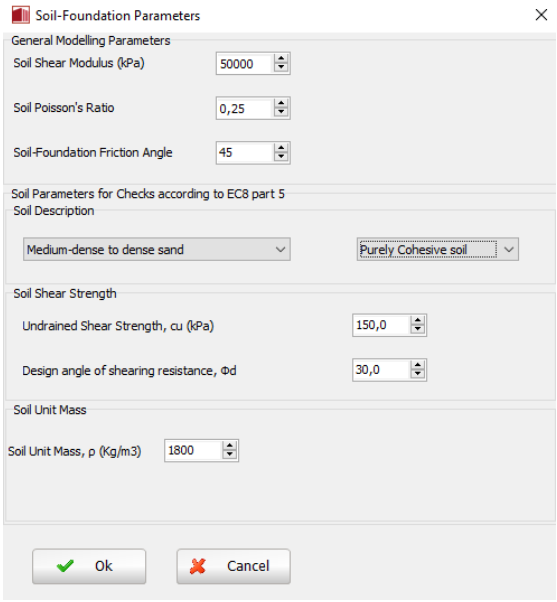
Propose FRP System window

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

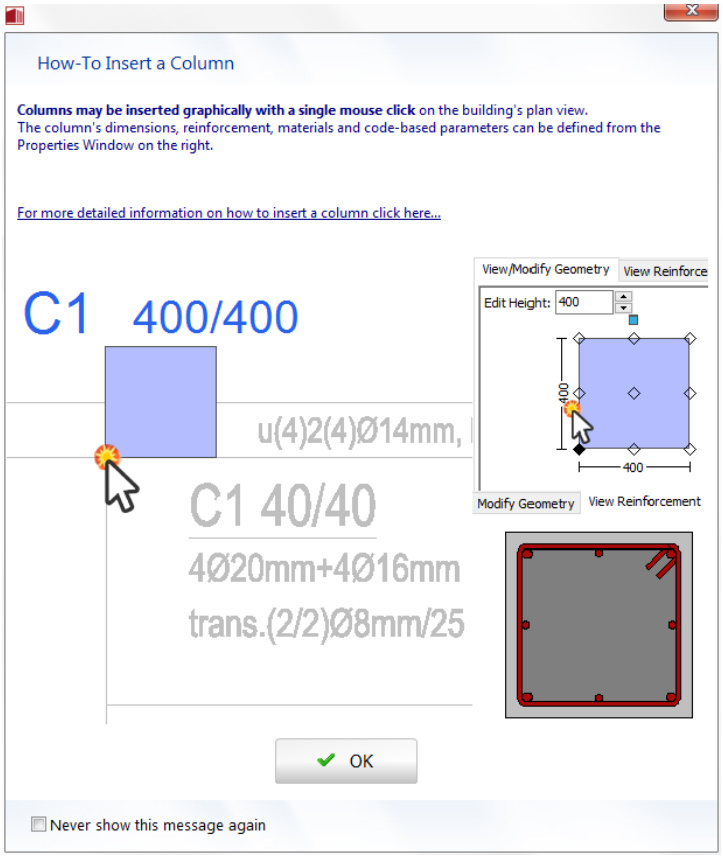
Column Members

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 andwidth)
(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) Reinforcement
(v) Jacket
(vi) Isolator
(vii) FRP Wrapping
(viii) 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:

- 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 discussion about the insertion of columns in the Building Modeller refer to Appendix E – Building Modeller .

Wall Members

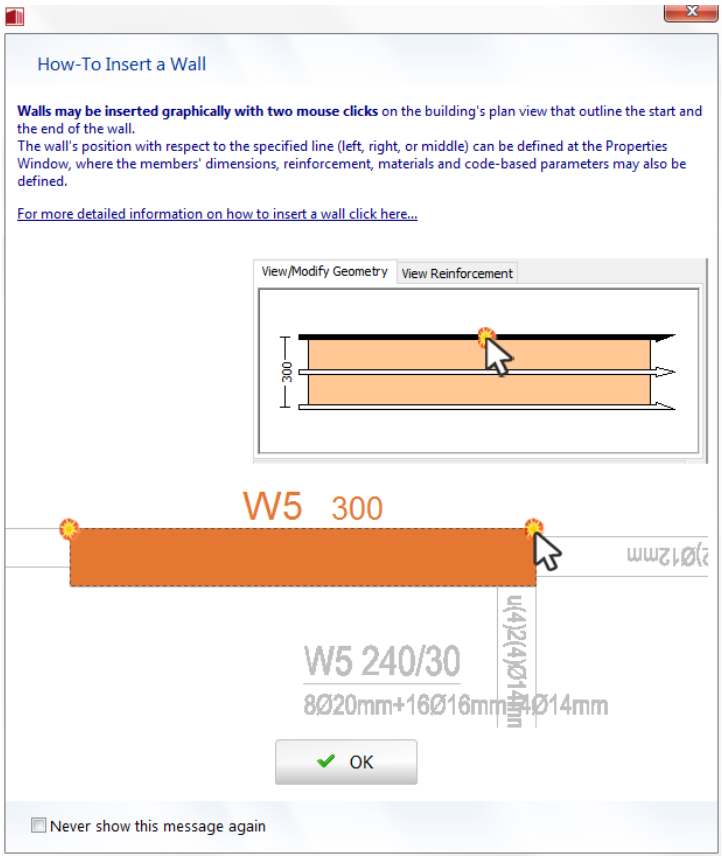
The walls can be inserted from the main menu (Insert >...) or through the corresponding toolbar button. The wall's Properties Window will appear where its properties are explicitly defined in the similar way

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 Building Modeller:

- Wall
- Compound Wall

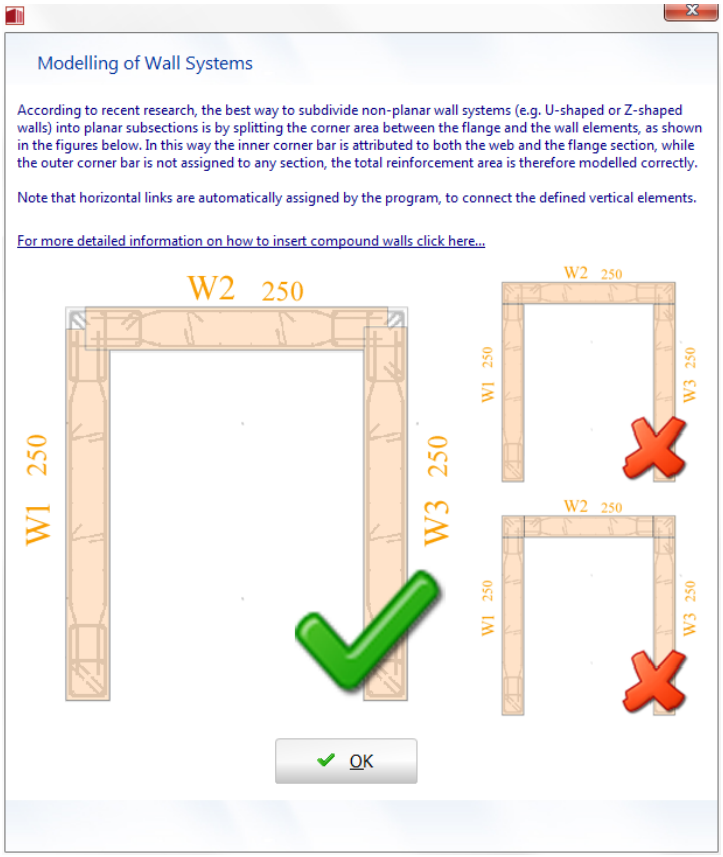
Once the Insert a Wall command is selected, an informative message appears, providing brief information of how to insert a wall.



How-To Insert a Wall window

For a comprehensive discussion about the insertion of walls in the Building Modeller refer to Appendix E – Building Modeller 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 walls. 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.

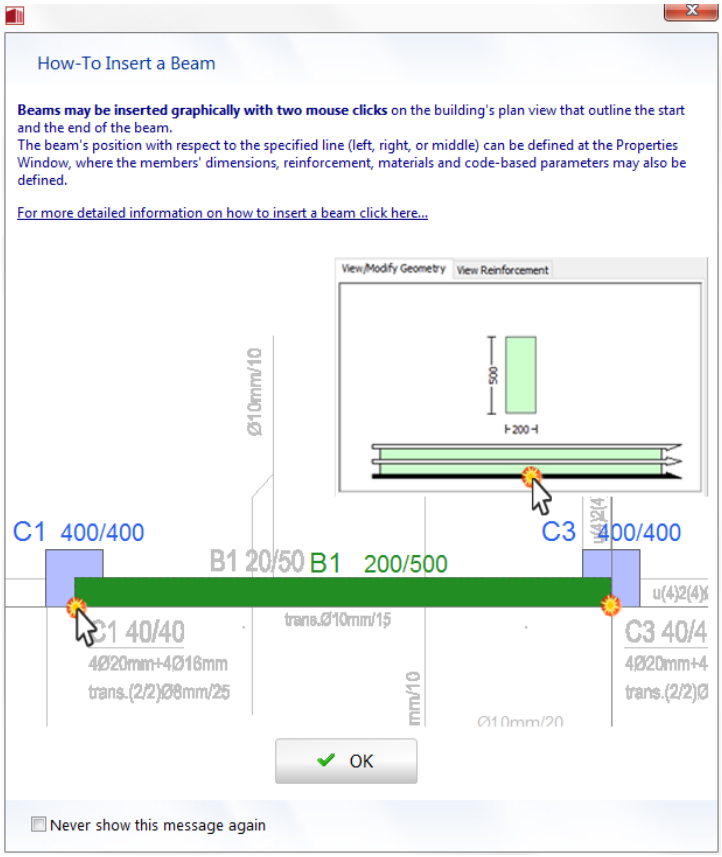
Beam Members

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 not 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 Building Modeller:

- 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.



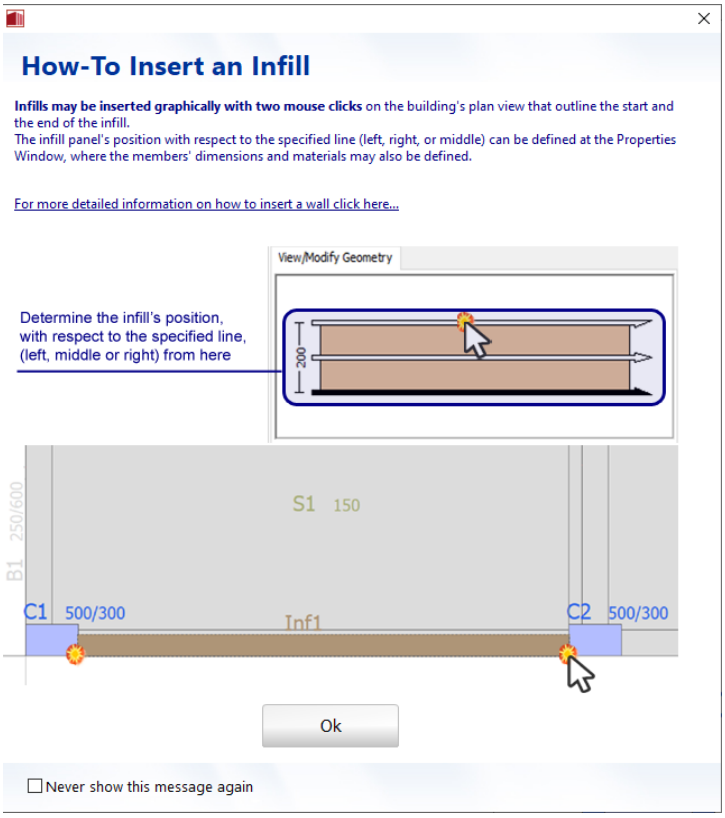
How-To Insert a Beam window

For a comprehensive discussion about the insertion of beams in the Building Modeller refer to Appendix E – Building Modeller .

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

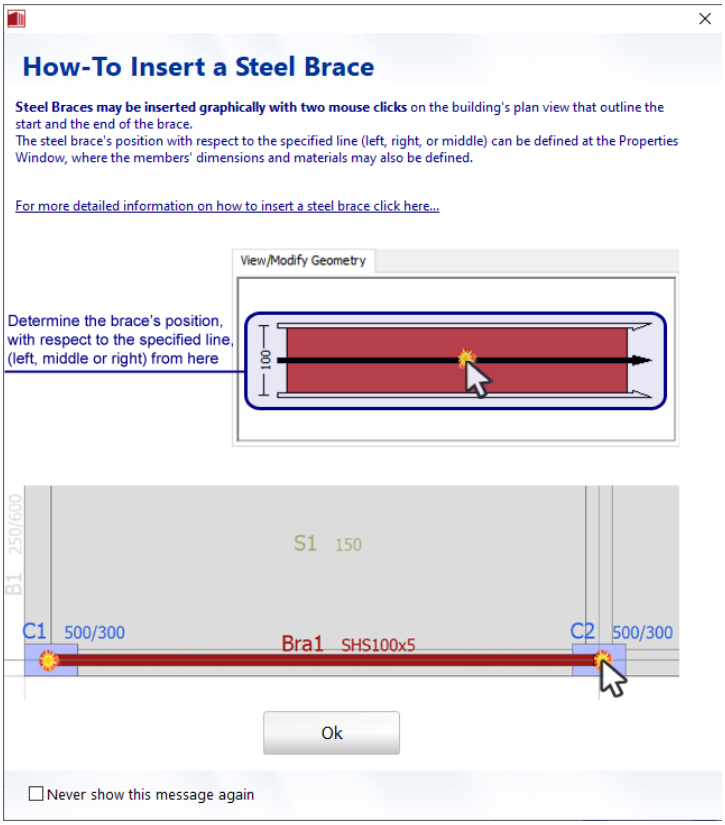
For a comprehensive description about the insertions of infills walls in the Building Modeller refer to Appendix E – Building Modeller.

Steel Braces

The steel braces can be inserted from the main menu (Insert >…) or through the corresponding toolbar button. 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 E – Building Modeller.

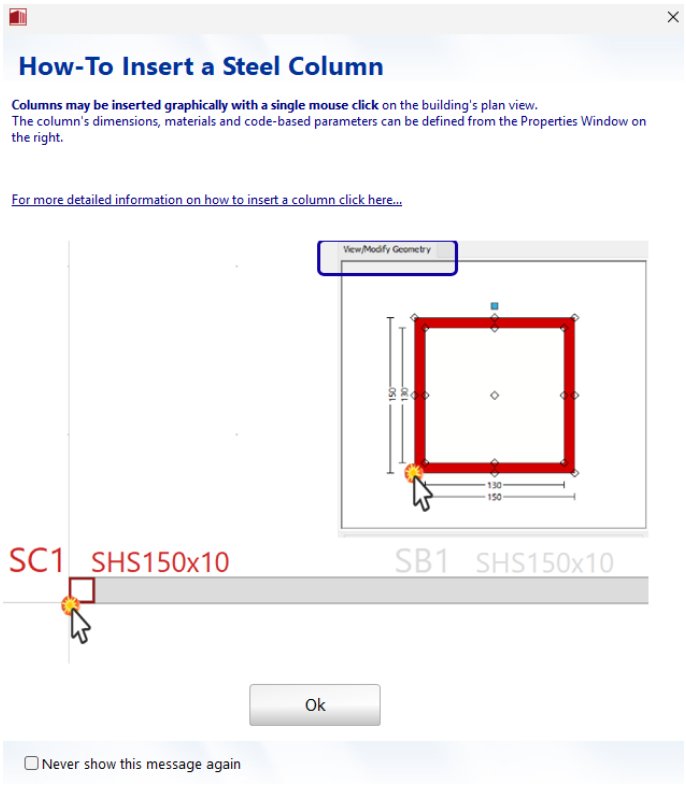
Steel Column

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

(i) Geometry, i.e. the steel section to be used
(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) Isolator
(vii) Advanced Modelling

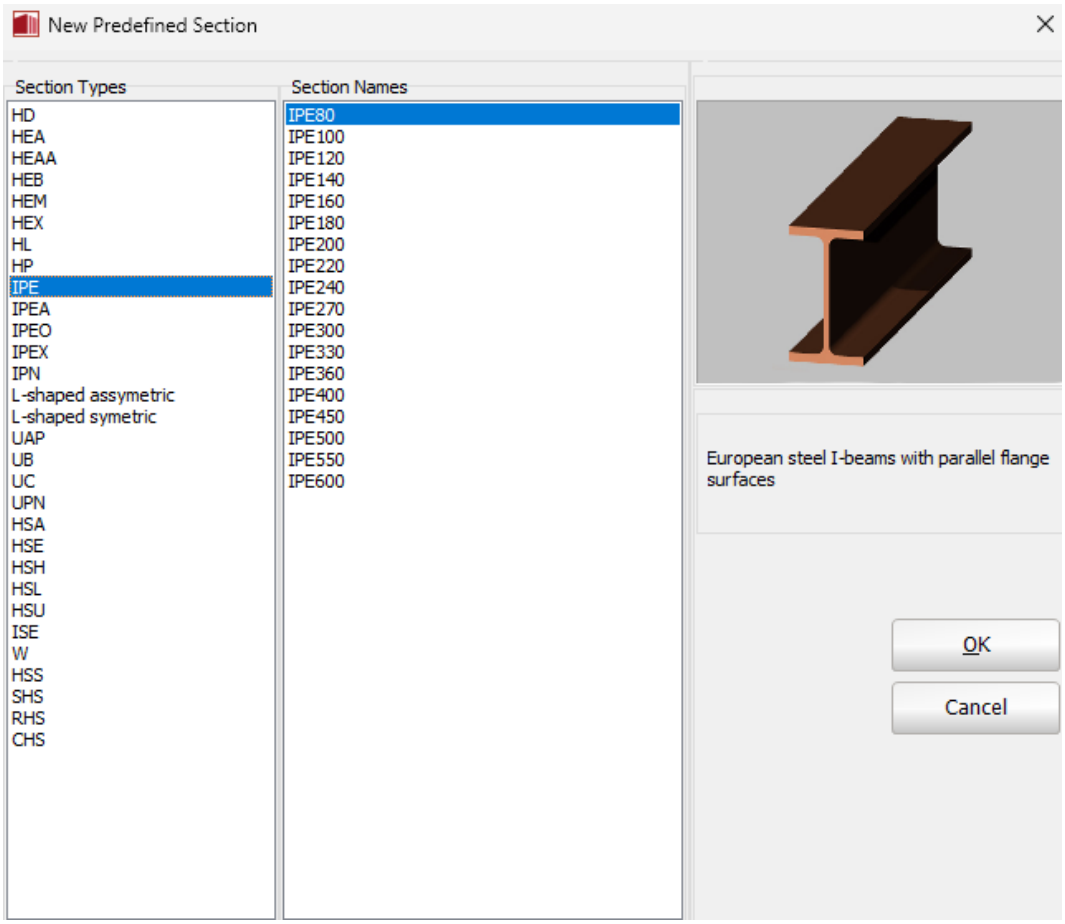
The steel column members may be inserted in the project with a single mouse click.

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



How-To Insert a Steel Column window

A database of the most common steel sections (e.g. HEA, HEB, IPE, etc.) is available, as well as W and HSS sections.



For a comprehensive description about the insertions of steel columns in the Building Modeller refer to Appendix E – Building Modeller.

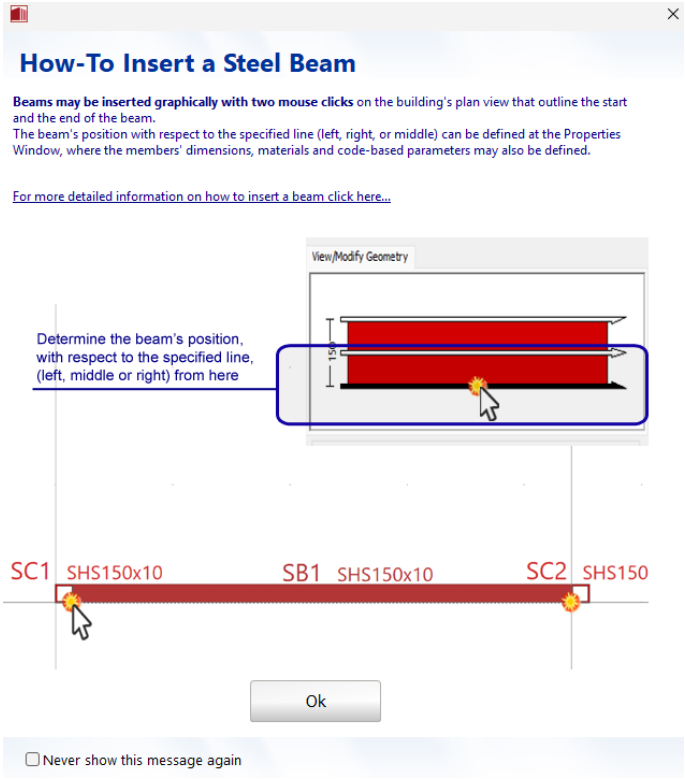
Steel Beam

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

(i) Geometry, i.e. the steel section to be used
(ii) Materials
(iii) Loading
(vii) Advanced Modelling

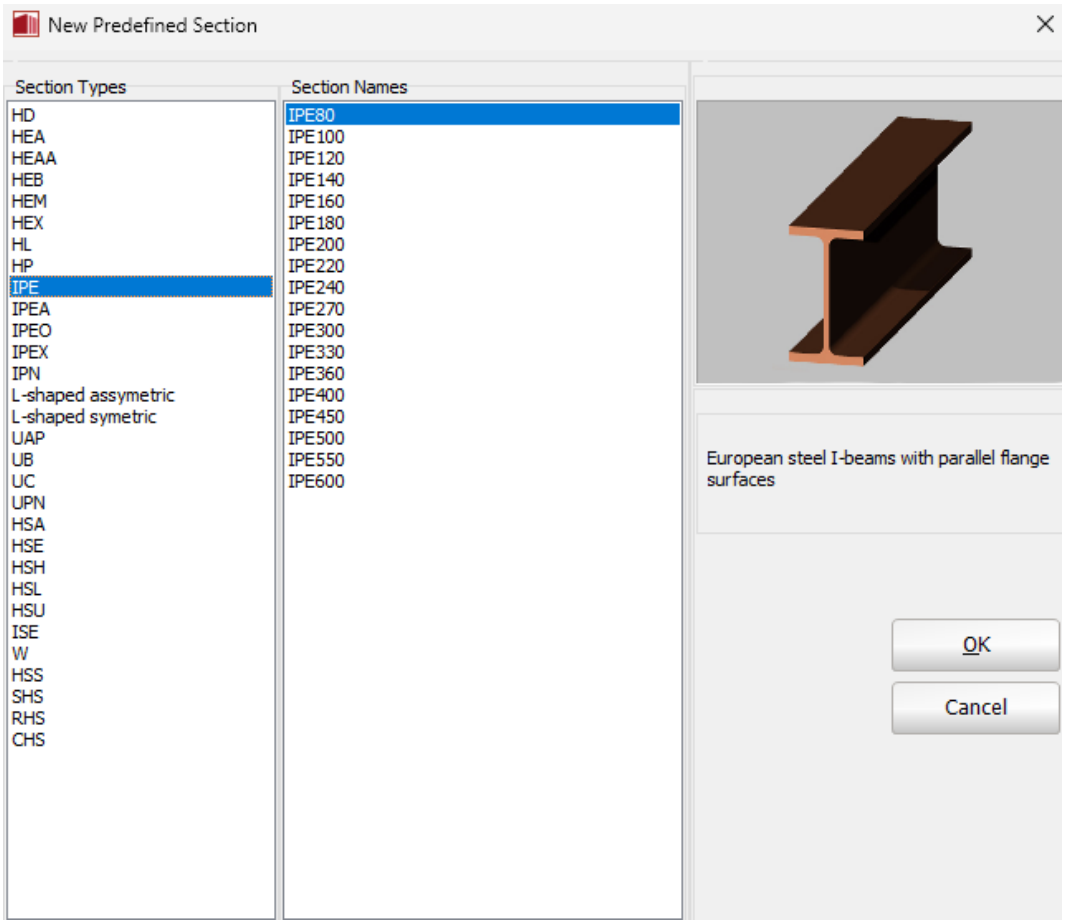
The steel beam members may be inserted in the project by defining their edges with two mouse clicks.

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



How-To Insert a Steel Beam window

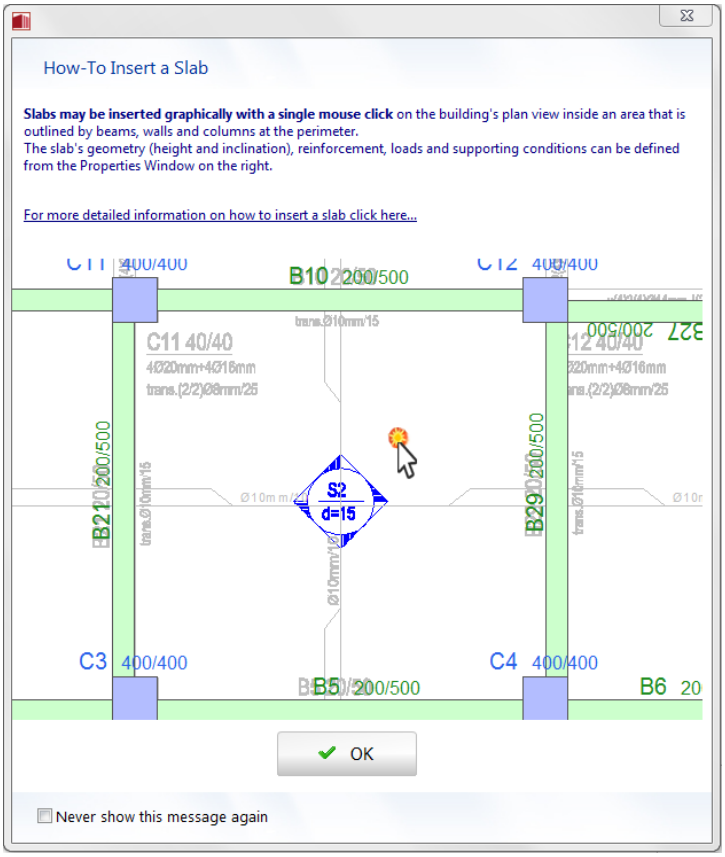
A database of the most common steel sections (e.g. HEA, HEB, IPE, etc.) is available, as well as W and HSS sections.



For a comprehensive description about the insertions of steel beams in the Building Modeller refer to Appendix E – Building Modeller.

Slab

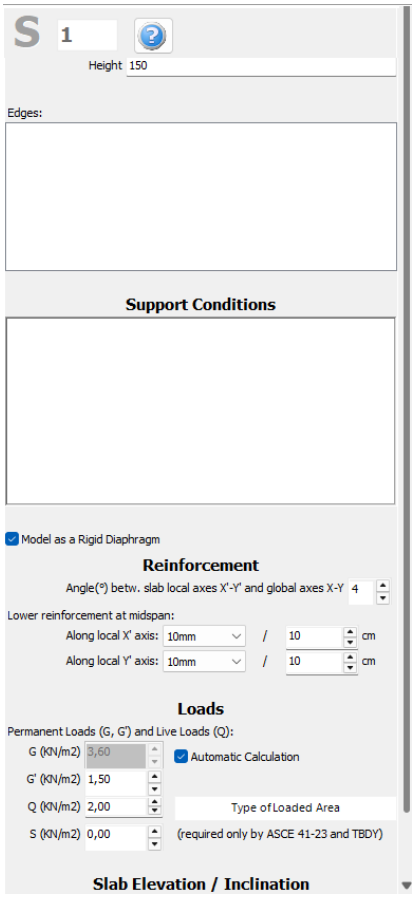
The insertion of slabs can be done through the Menu (*Insert > Slab*) or by clicking the toolbar button. Prior to adding a slab, an informative message appears providing brief information of how to insert a slab.



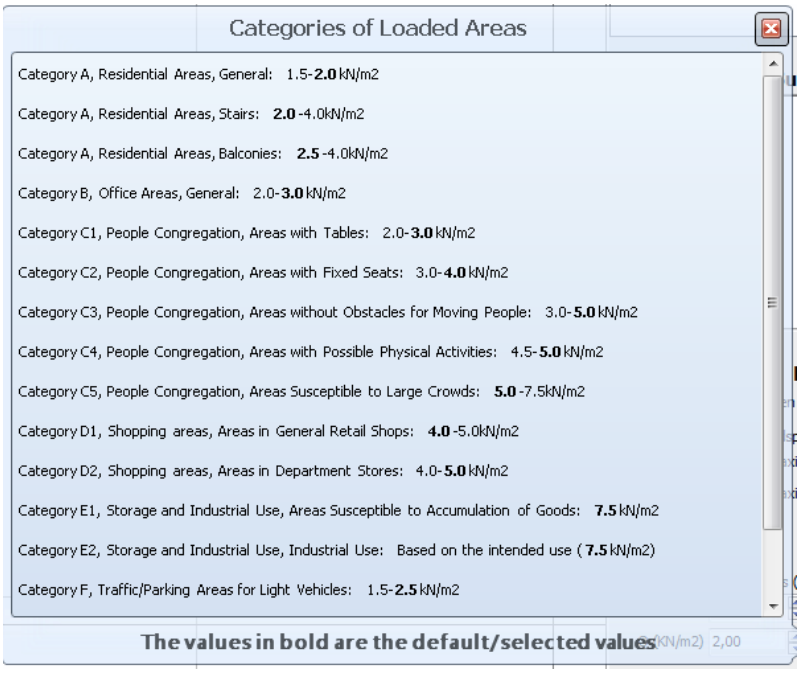
How-To Insert a Slab window

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

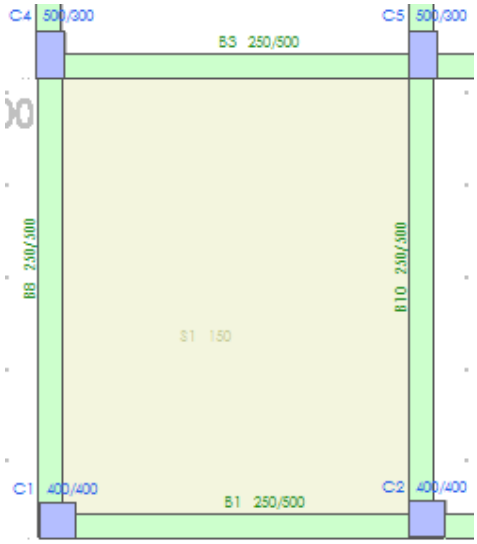
In the slab's Properties Window users can define (i) the section's height, (ii) whether it will be modelled as rigid diaphragm, (iii) the reinforcement and its rotation to the X & Y axes, and (iv) its self weight and the additional permanent, live and snow loads; the latter is required only by ASCE 41-23 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. It is noted that the self-weight of the slabs is automatically calculated and included in the structural model or user-defined value may be assigned.



Slab's Properties Window

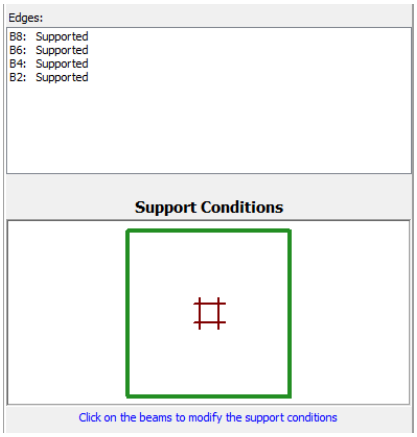


Type of Loaded Area



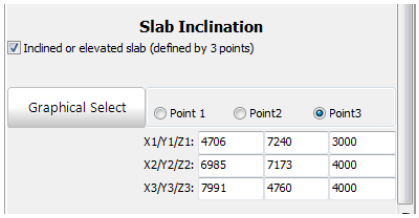
Slab insertion

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 neighboring 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.



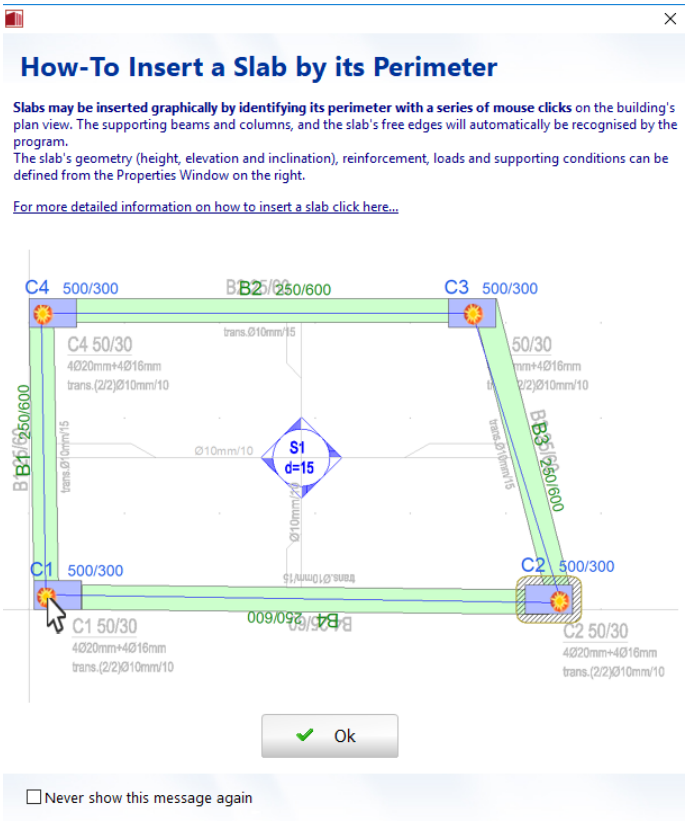
Slab Inclination

NOTE 1: The slab 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.

NOTE 2: The slab modelling is, by default, carried out using rigid diaphragms; therefore, a rigid slab is assumed in the structural configuration, which reflects the typical behaviour of the vast majority of RC buildings. The option to exclude the rigid diaphragm assumption is also available by deselecting the corresponding checkbox. **The slab's loads (self weight, additional gravity and live loads multiplied by the corresponding coefficients in the SeismoStruct Building Modeller Settings module) are transformed to masses, based on the g value, and applied directly to the beams that support the slab.**

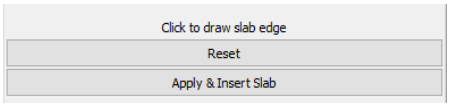
Slab by perimeter

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.



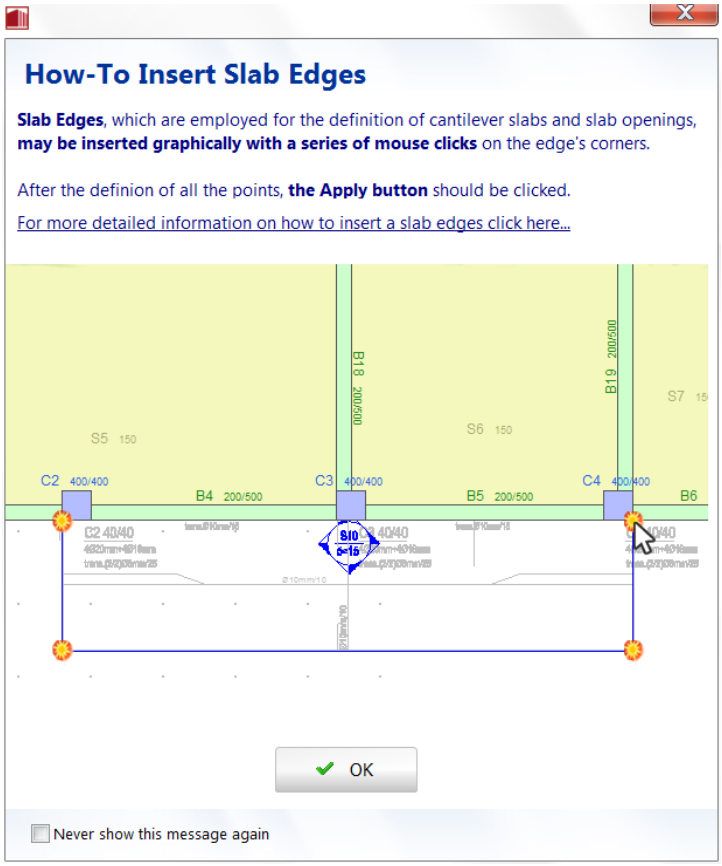
Draw Slab by perimeter

NOTE 1: Slabs are modelled in SeismoStruct 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 with the last one in order to insert the new slab.

Free Edge

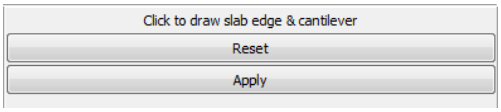
Cantilever slabs can also be defined in the Building Modeller. In order to do so, a Free Edge must be added from the Menu (*Insert > Free edge*) or through the respective toolbar button  . An informative message appears providing brief information of how to insert a Free Edge.



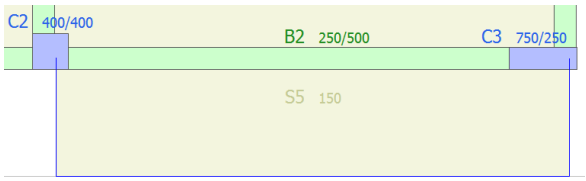
How-To Insert Slab Edges window

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.

NOTE: Slabs are modelled in SeismoStruct 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.



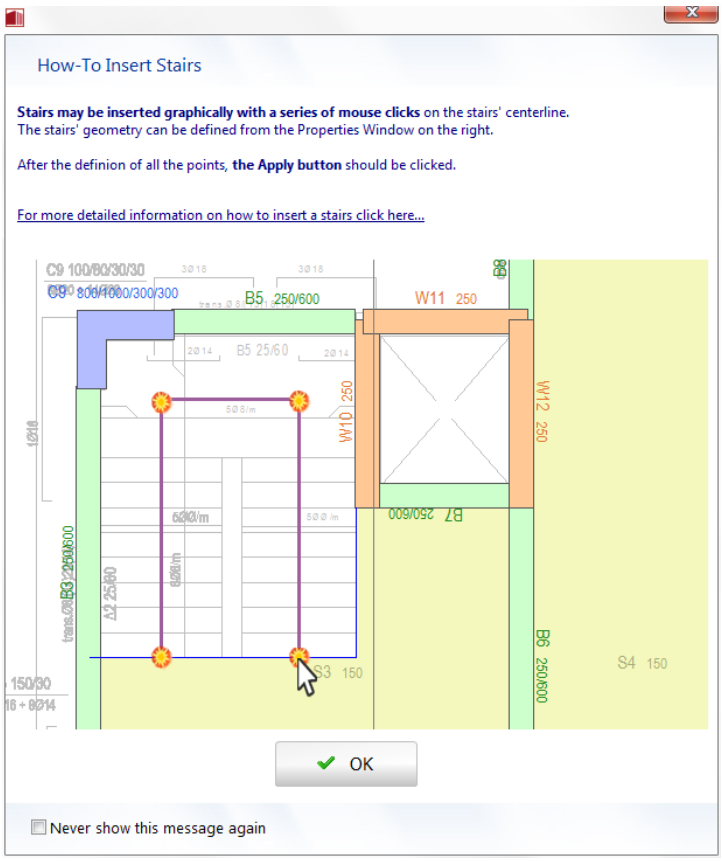
Draw Free Edge



Create a new cantilevered slab

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.

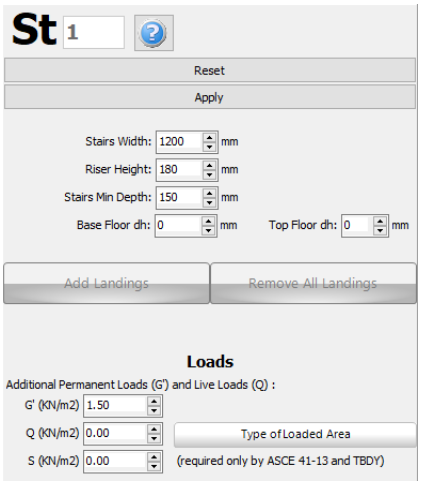


How-To Insert Stairs window

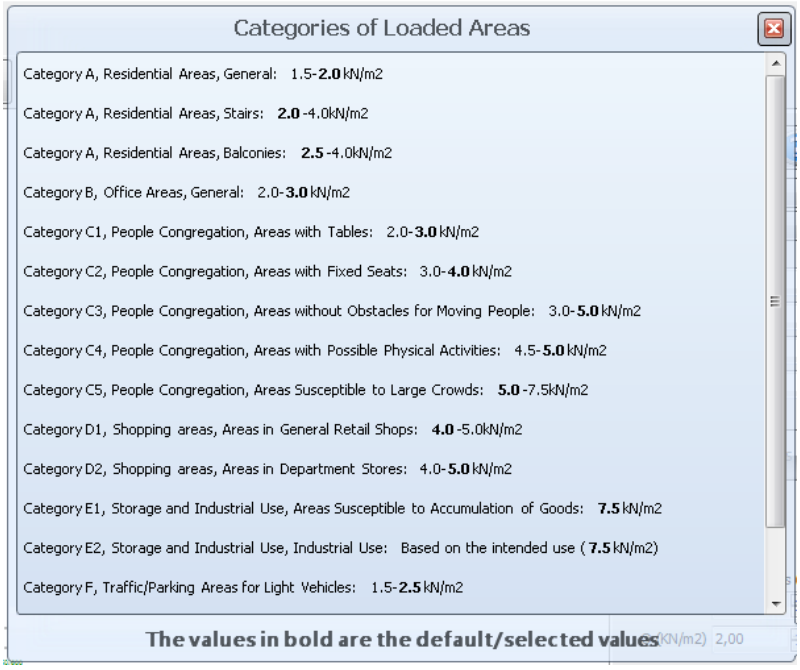
Stairs may be easily defined by specifying their centreline. Landings may be applied through the "Add Landings" button after the insertion of the stairs member in the project. The two ends of the landings need to be specified graphically on the centerline. The defined landings may be removed through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

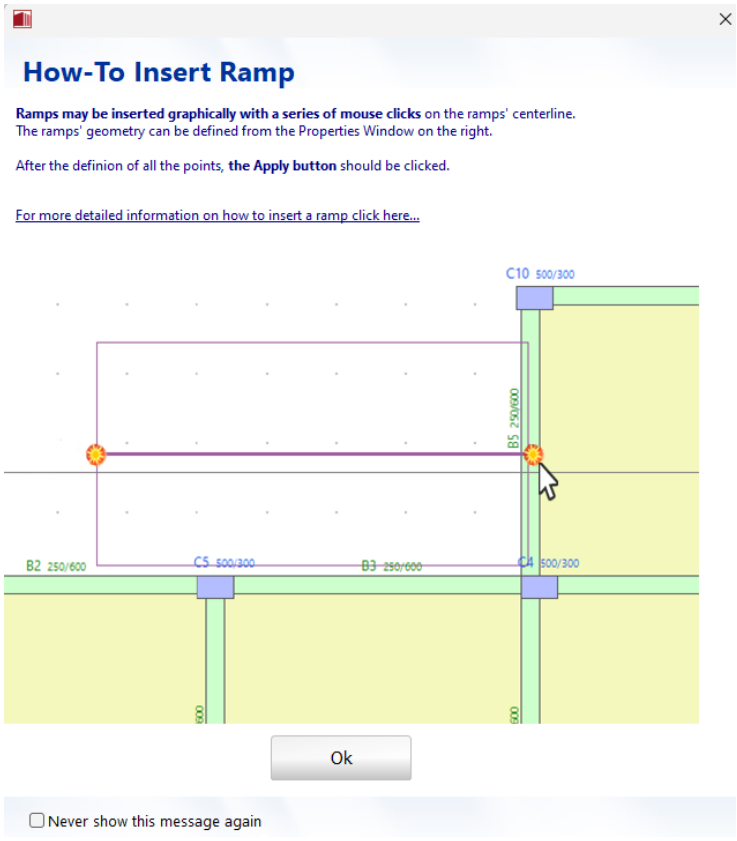


Type of Loaded Area

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

Ramps

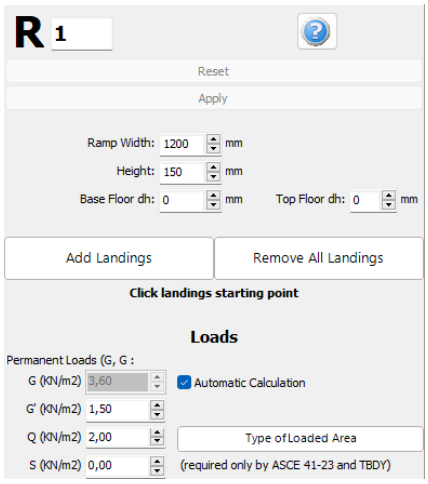
The insertion of ramps can be done through the Menu (*Insert* > *Ramp*) or by clicking the  toolbar button. An informative message appears providing brief information of how to insert Ramp.



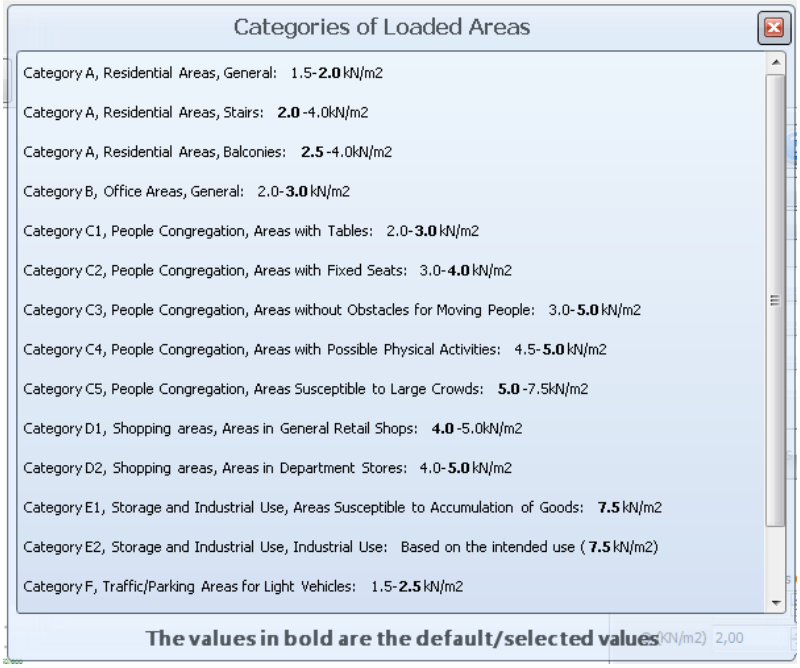
How-To Insert Ramp window

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

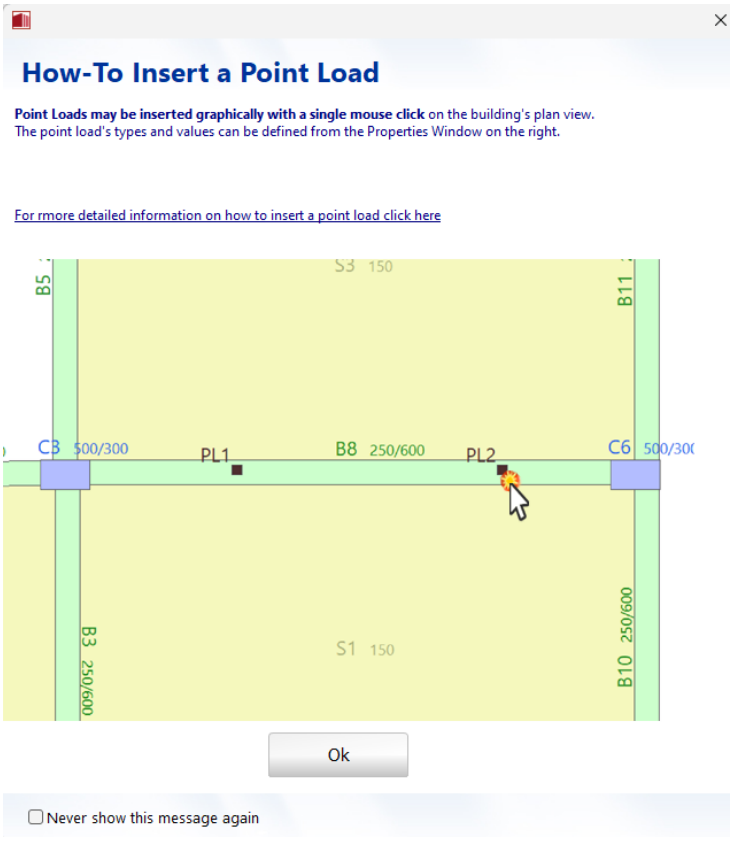
NOTE: Ramps are modelled in SeismoStruct with elastic elements of the specified width and depth.

INSERTING LOADS

In SeismoStruct Building Modeller 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.

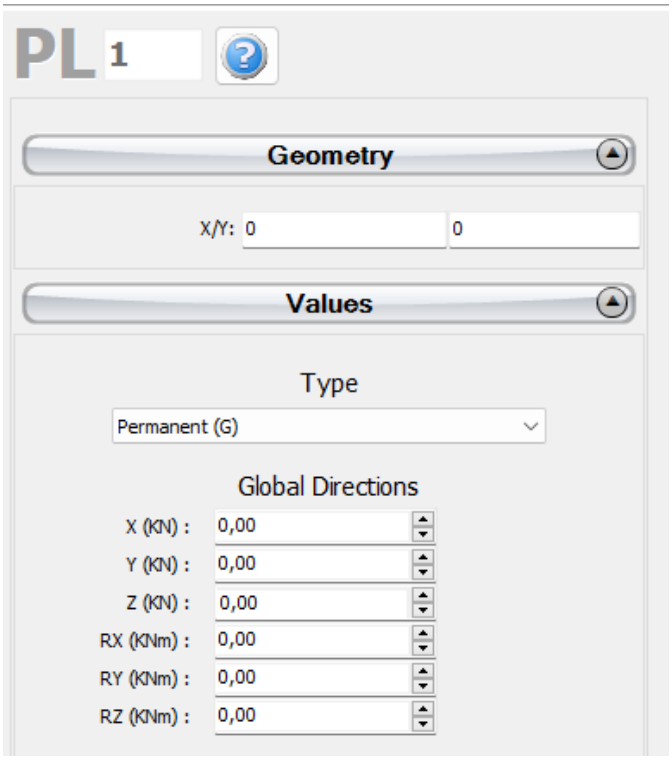
Point Loads

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  ,

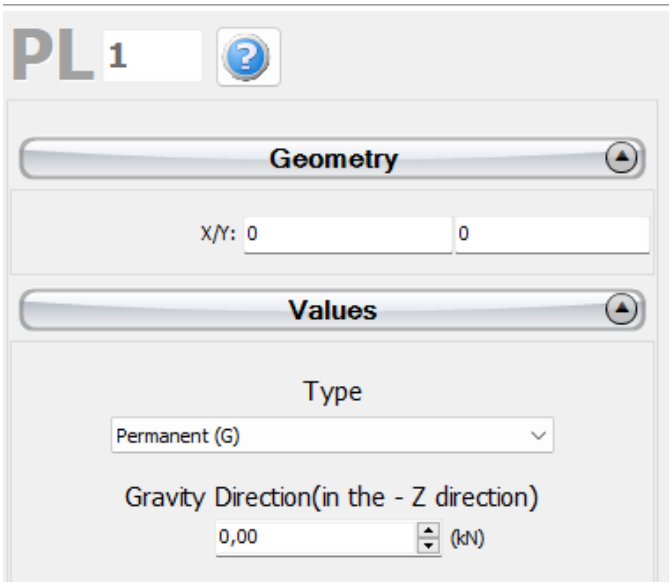


How-To Insert Point Load window

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.

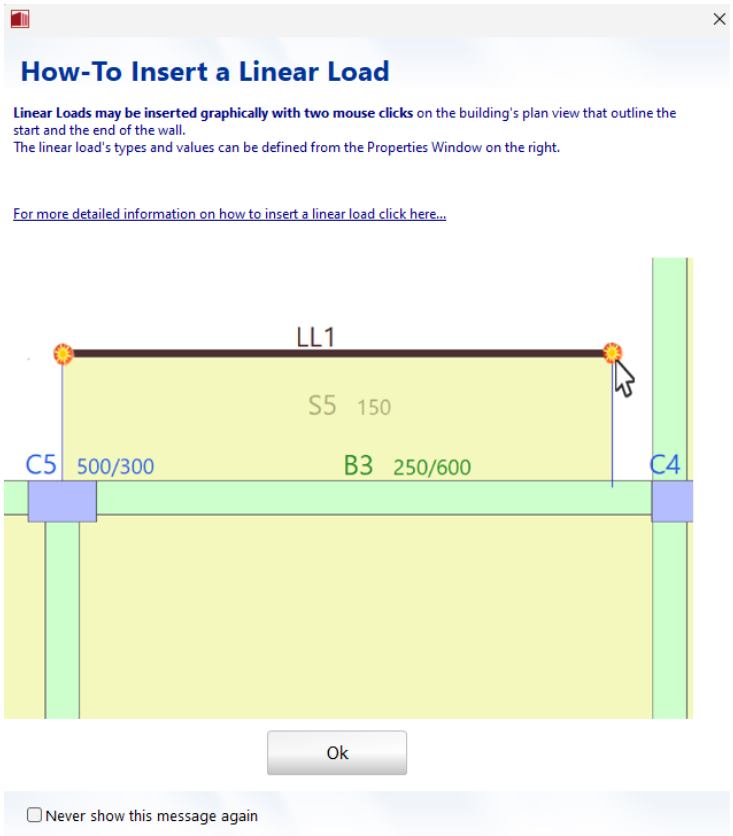
NOTE 1: When a point load is assigned to a beam, the beam is automatically subdivided, resulting in the creation of two members. If the point load is near the start or end of the beam, it is automatically applied at the beam's edge.

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.

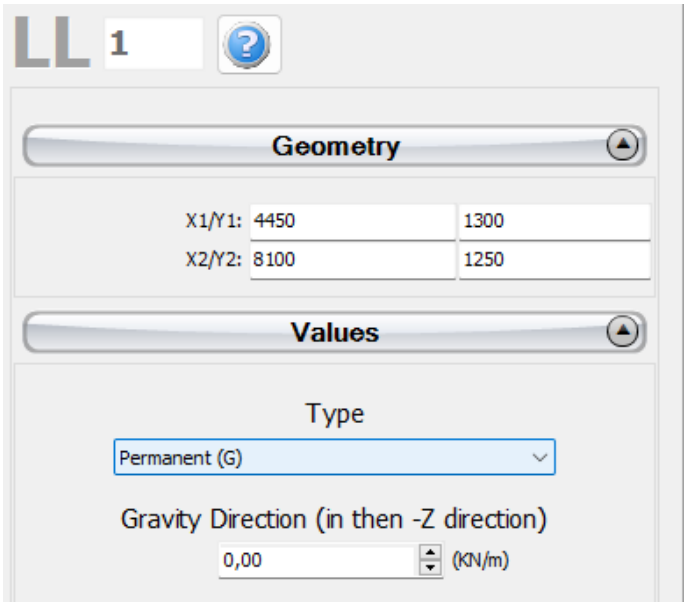
Linear Loads

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, strip footings and raft foundations. 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.

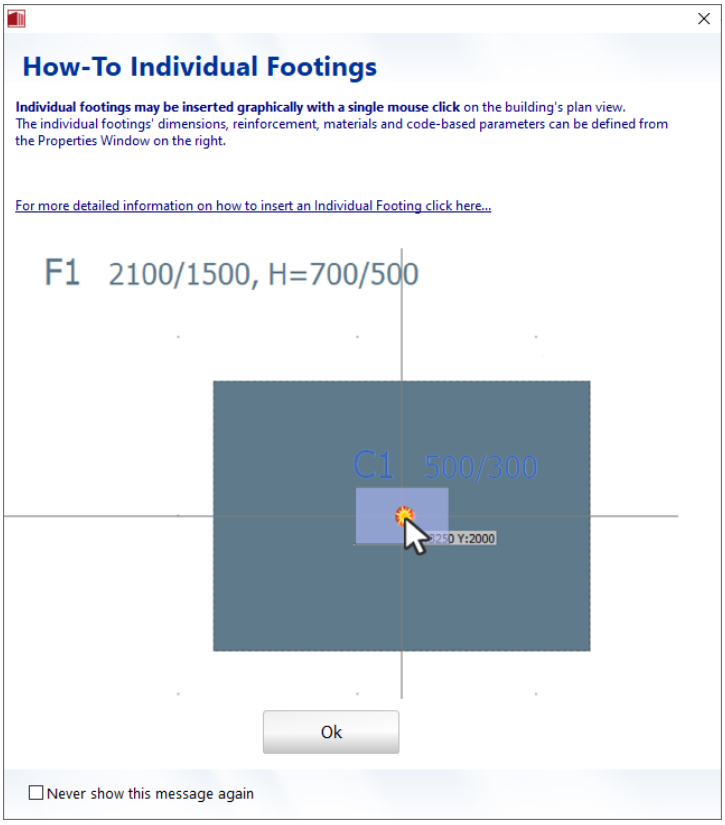
Individual Footings

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 E – Building Modeller.

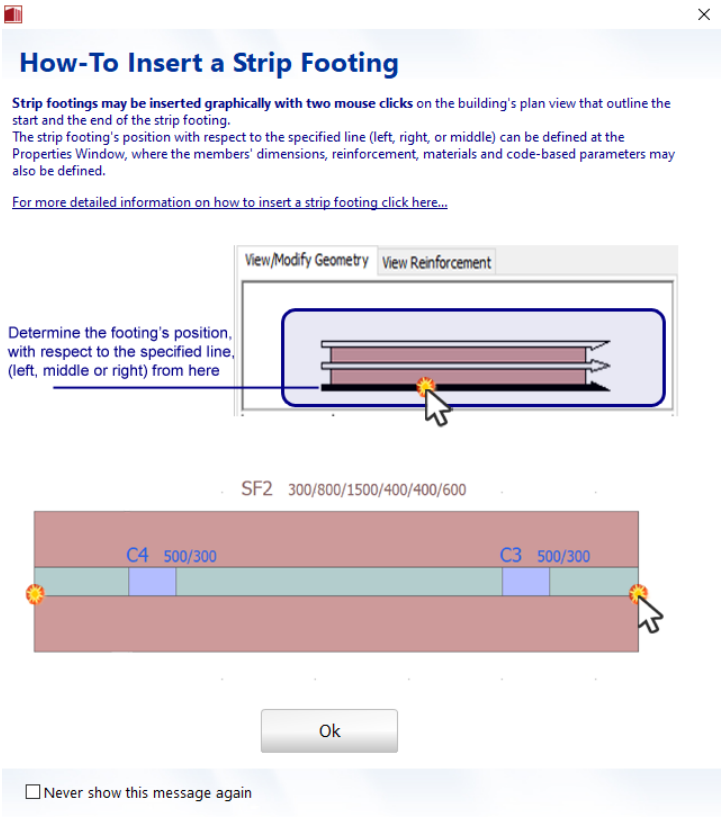
Strip Footings

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, ie 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 E – Building Modeller.

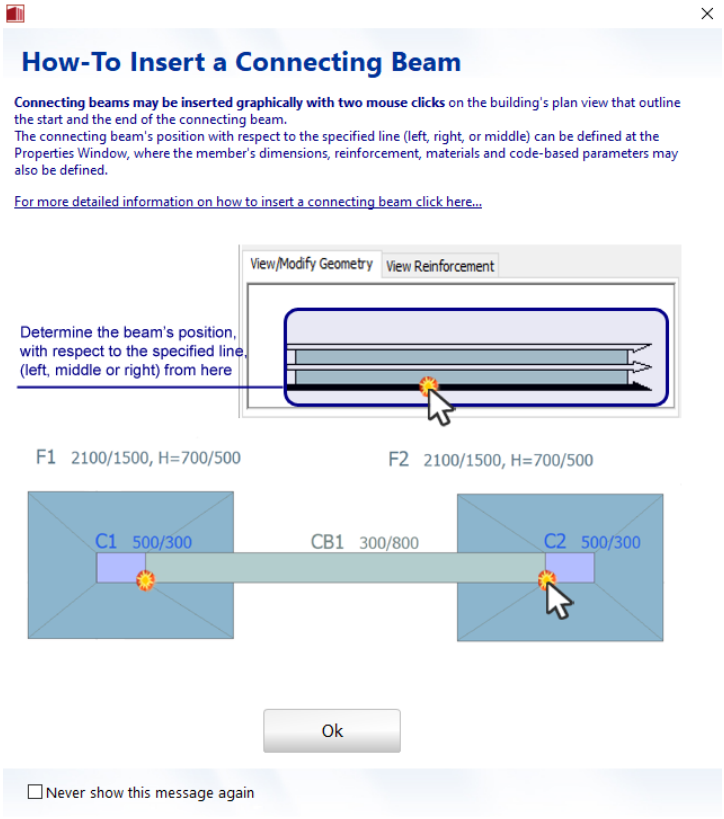
Connecting Beams

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, ie 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.



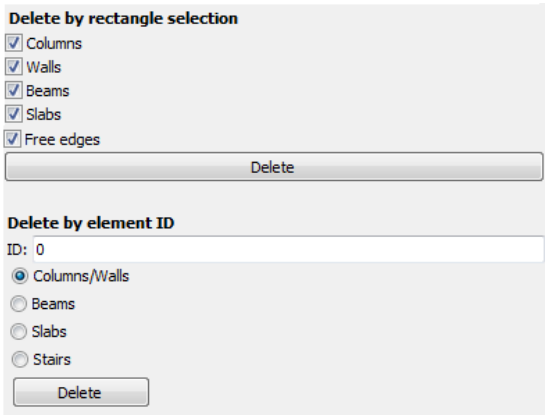
How-To Insert a Connecting Beam window

For a comprehensive description about the insertion of connecting beams in the Building Modeller refer to Appendix E – Building Modeller.

Editing Structural Members

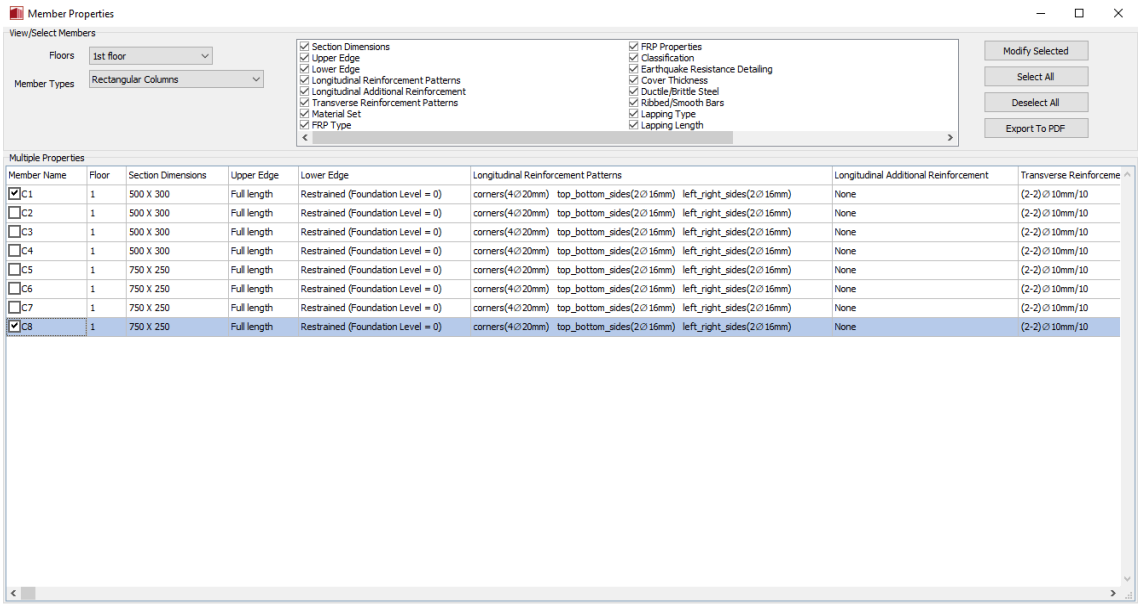
By using the edit tools from main menu (*Tools >...*) or through the corresponding toolbar buttons, users can select ( ) a member to view or change its properties. Further they can move it ( ) to a different location, rotate it ( ) in plan view or delete it ( ).

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.



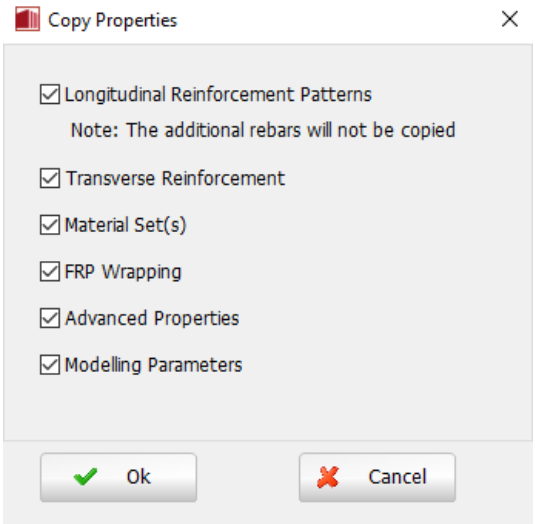
Delete element window

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.

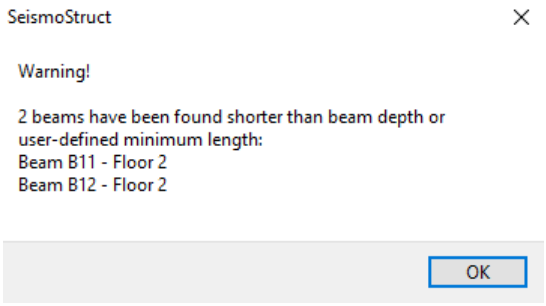


Copy Member Properties window

Moreover, an option to renumber the structural members is offered from the main menu (*Tools > Renumber elements*) or through the corresponding 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 from the main menu (*Tools > Verify Connectivity*) or through the

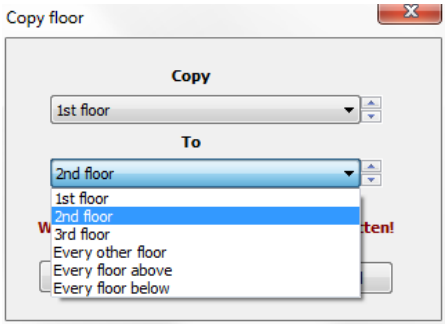
corresponding toolbar button for the existence of any beam with free span smaller than its section height should be carried out. If such beams exist, the following message appears.



Verify connectivity

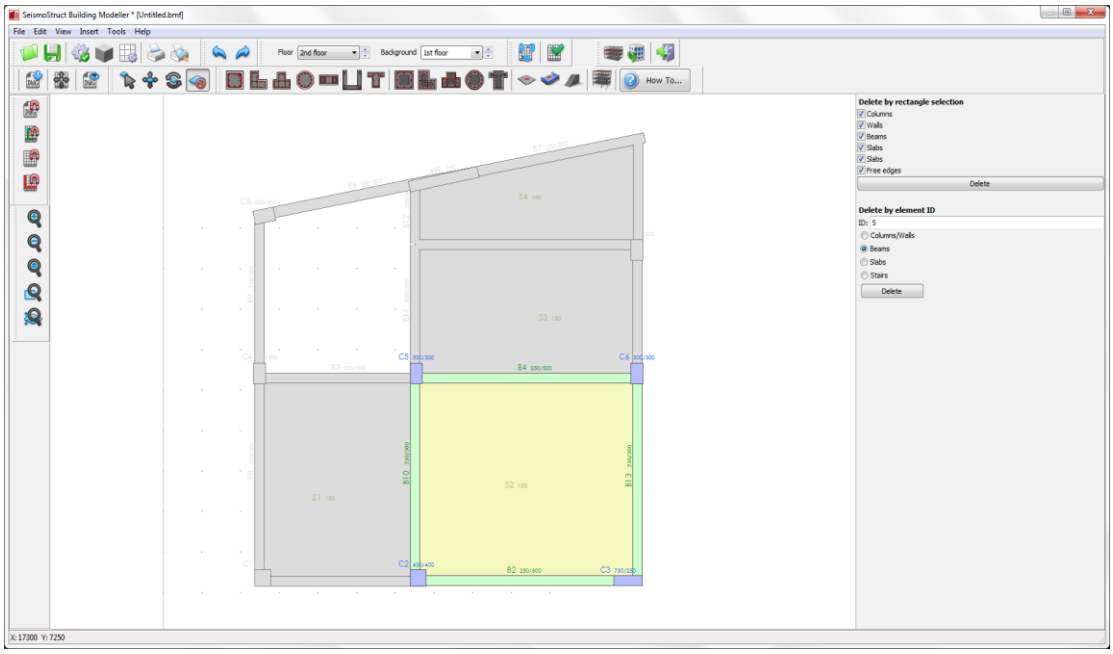
Creating New Storeys

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



Copy floor

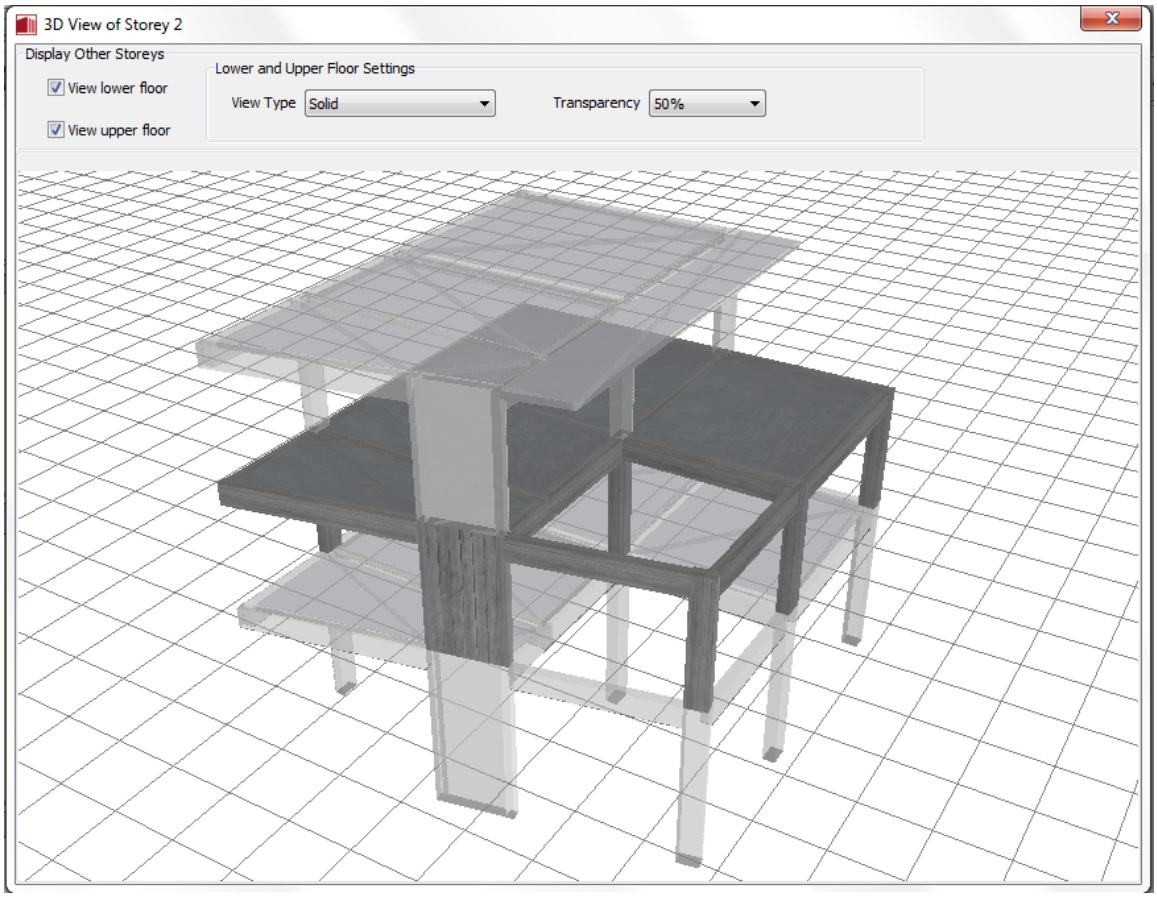
It is noted that users may use the layout of an existing floor as background, in order to easily introduce new members on another storey.



New Floor & Background

View 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

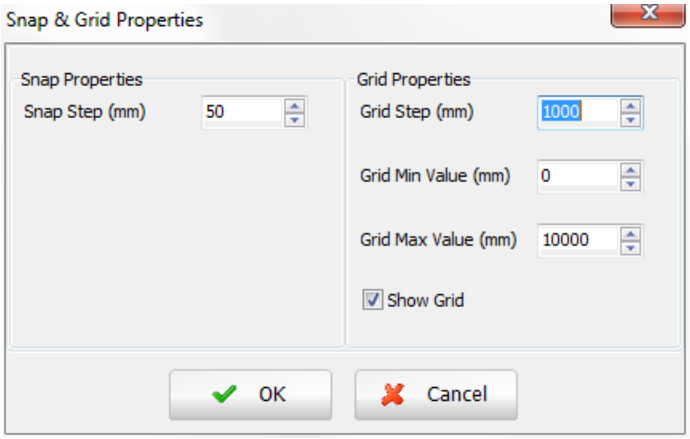
The Building Modeller offers a variety of tools to facilitate the introduction of the structural layout:

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



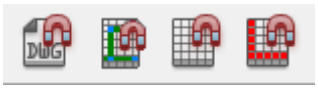
Zoom tools

- 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 member and/or the grid. The grid (step, min and max values) and 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 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 Menu (*View >...*) and through the corresponding toolbar buttons.



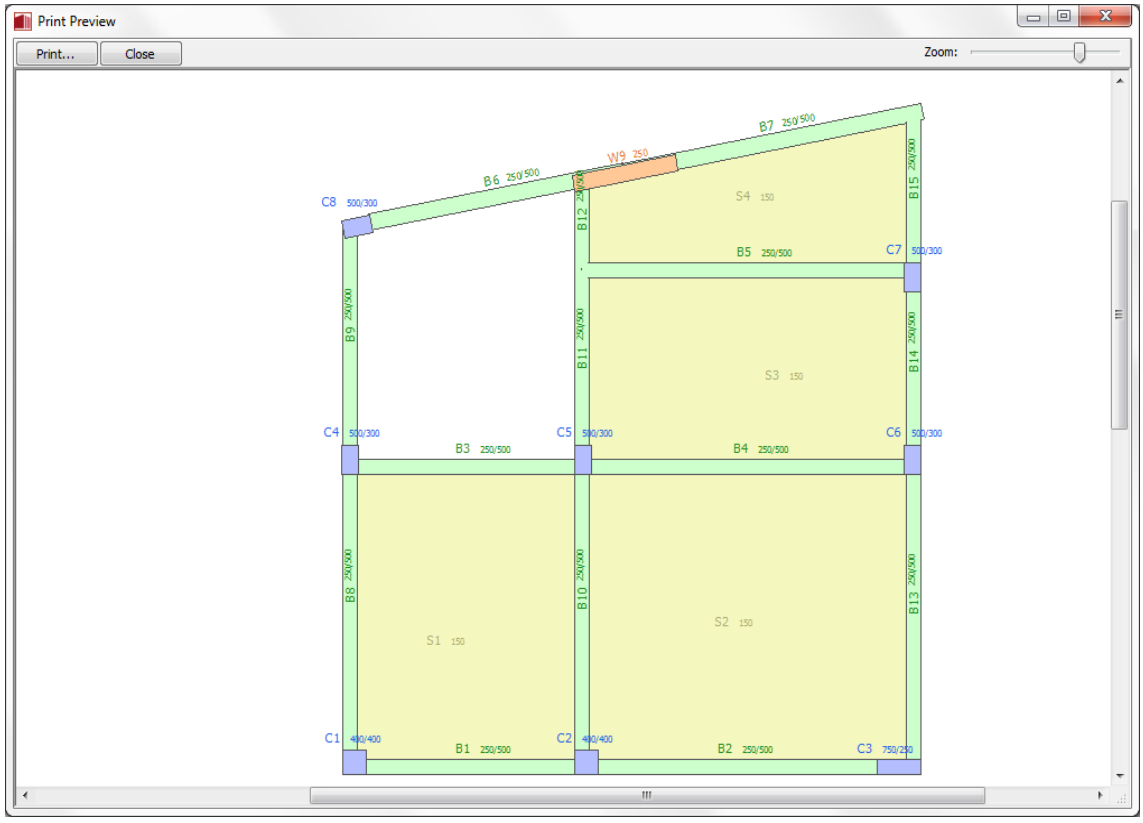
Snap and Ortho tools

- The axes origin of the CAD drawing at the background may be moved from the main menu (*View >Move Axes Center*) or through the corresponding 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 from the main menu (*Edit >Undo*)/ (*Edit >Redo*) or through the corresponding toolbar buttons  .
- The selected plan view can be printed or previewed from the main menu (*File >Print... & File >Print Preview...*) or through the corresponding toolbar buttons  & 



Print preview

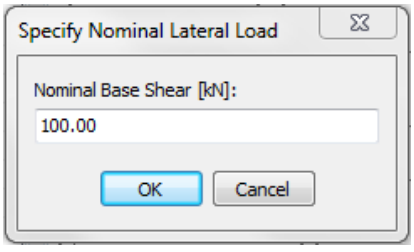
Saving and Loading Building Modeller Projects

The Building Modeller project can be saved as a Building Modeller file (with the *.bmf extension) from the main menu (*File >Save As...*)/ (*File >Save*) or through the corresponding toolbar button  . It is noted that this file type is not a SeismoStruct project file (*.spf), hence it can be opened again only from within the Building Modeller, from the main menu (*File >Open)* or through the corresponding toolbar button  .

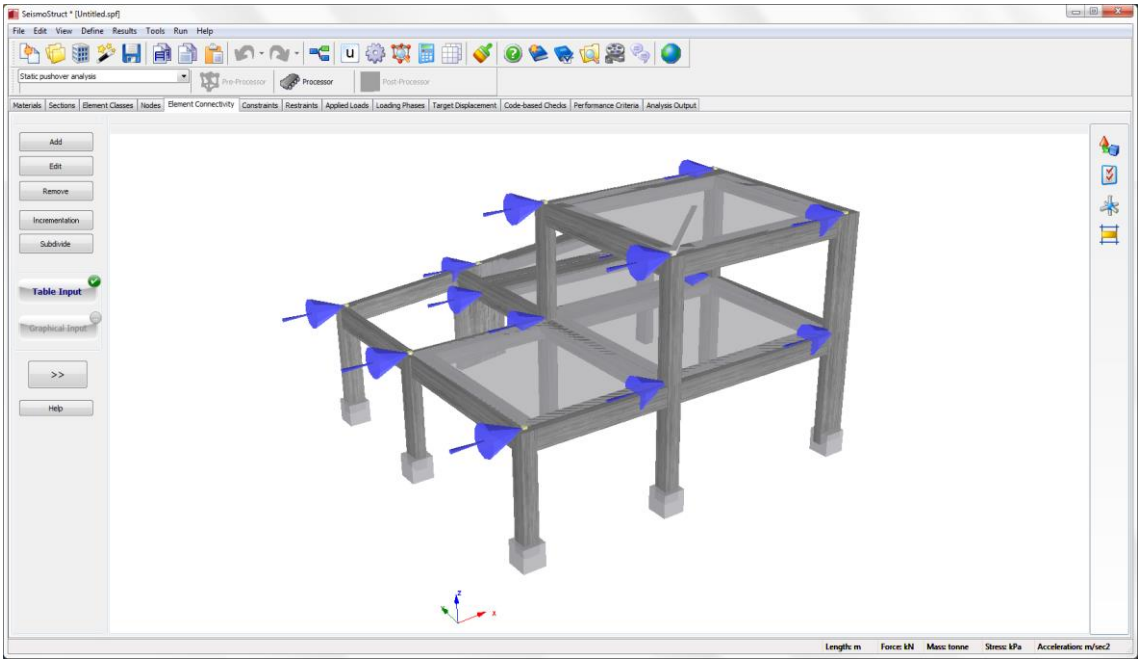
NOTE: SeismoBuild projects (with the *.bpf extension) may be also imported from within the Building Modeller, from the main menu (File >Open) or through the corresponding toolbar button  .

Creating SeismoStruct Projects

A SeismoStruct project is created from the main menu (*File >Exit and Create Project)* or through the corresponding toolbar button  . When this option is selected, a new window appears for the definition of structure's loading, depending on the analysis type.



Specifying the Loading for static pushover analysis



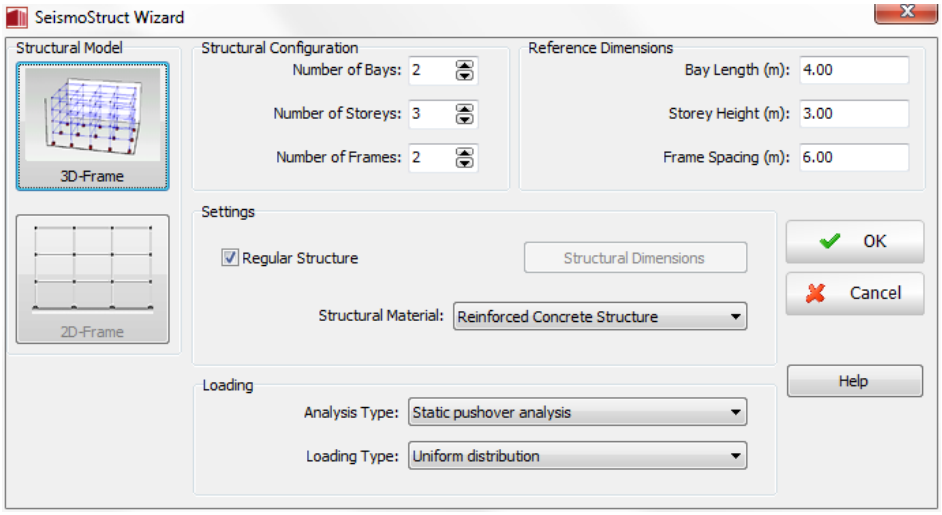
New SeismoStruct model

NOTE: When creating a SeismoStruct 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, through the relevant setting in the Project Settings panel (Project Settings -> Gravity & Mass), i.e. 'Loads (ONLY in the gravity direction) are derived from Masses, based on the g value'.

Finally, an option for exiting the Building Modeller without creating the SeismoStruct project file is offered from the main menu (*File >Exit Without Creating Project)* or through the corresponding toolbar button .

WIZARD

In order to facilitate the creation of frame/building models, a *Wizard* facility has been developed and introduced in the program. The *Wizard* dialog box is accessed from the main menu (*File > Wizard...*) or through the corresponding toolbar button .



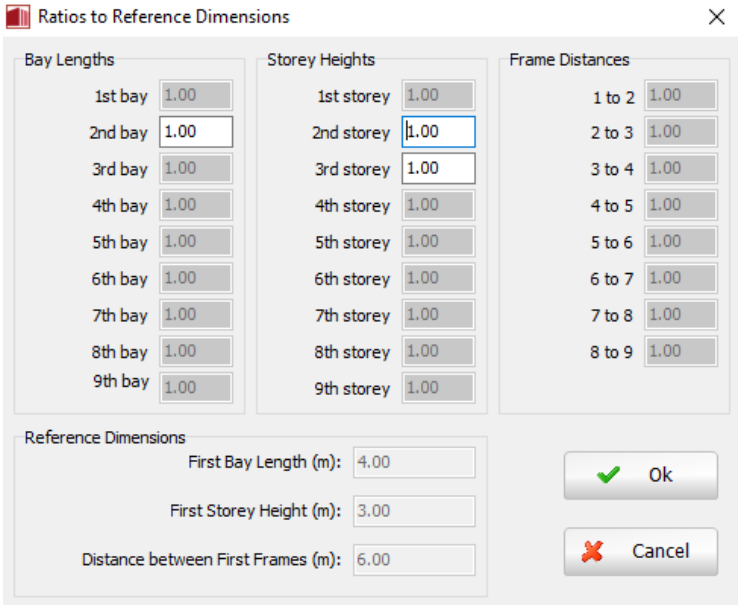
Wizard Facility window

IMPORTANT: New users are strongly advised to use this expeditious model creation facility to get up and running in the minimum amount of time and to gain a quick grasp on the structure and workings of SeismoStruct's project files.

Structural model and configuration

In order to create a building model using the *Wizard*, the user should first decide if he/she intends to create a 2D or 3D structure, after which the number of bays, storeys and frames can be assigned, together with the reference values for bay length, storey height and frame spacing.

If the structure is regular (i.e. all bays have equal length, all storeys feature the same height and all frames are evenly spaced) then the reference dimensions become the actual ones. If, on the other hand, the structure is geometrically irregular, then the *Regular Structure* option should be unchecked so that the user can access the *Structural Dimensions* dialog box, where the actual bay lengths, storey heights and frame distances can be defined. By default, the reference dimensions are adopted.



Structural Dimensions dialog box

Settings

Having defined the structural geometry, the user should now specify if the building is a reinforced concrete or steel structure. The *Wizard* generates structures employing the inelastic force-based plastic-hinge (infrmFBPH) elements type.

NOTE: If the user intends to adopt the other types of inelastic frame elements (infrmFB, infrmDBPH or infrmDB) rather than infrmFBPH, after the model's generation he/she may manually modify the element type in the Element Classes dialog box.

Each frame element generated through the Wizard facility is defined by 'structural' nodes at beam column joints. The names of these nodes are automatically created by following the n111 naming convention: all nodes have a name of the format: "n"+i+j+k, where i is the storey number (starting from the bottom/foundation), j is the column number (starting from the left) and k is the frame number (starting from the front). For instance, n123 would refer to the node on the left column of the model (i=1), in the second frame (j=2) and at the third storey (k=3, third level of nodes). Users should refer to the *Nodes* paragraph for further details on the nodes definition.

The orientation of the frame elements created using the *Wizard* facility is automatically defined by a rotation angle (by default equal to 0). Users should refer to the discussion on *Global and Local Axes Systems* for further details on the element orientation.

Loading

Finally, one of the ten Analysis Types available in SeismoStruct has to be selected, depending on which the following loads and restraining conditions are imposed on the structure:

- Eigenvalue analysis. Self-weight of the structure is considered. No loading is applied.
- Static analysis with non-variable loads. Permanent gravity loads are applied.
- Static pushover analysis. In addition to permanent gravity actions, *Incremental Loads*, consisting of horizontal forces at each storey level, are also applied to the structure in the x-direction. The user has the possibility of choosing between two alternative load distributions (triangular or rectangular/uniform vector shapes) and of defining the nominal base-shear value (usually a value around the expected base shear capacity of the structure is used, though any given value is fine). Refer to *Pre-Processor* > *Applied Loads* > *Loading Phases* for further details on pushover analysis loading characteristics.
- Adaptive static pushover analysis. In addition to permanent gravity actions, *Incremental Loads*, consisting of horizontal displacements at each storey level, are also applied to the structure in the x-direction. Since the load distribution is automatically adapted by the program, the user needs only to specify the nominal displacement load to be used as reference value during the pushover procedure. Refer to *Pre-Processor* > *Applied Loads* > *Adaptive pushover parameters* for further details on adaptive pushover analysis loading characteristics.
- Static time-history analysis. In addition to permanent gravity actions, *Static Time-history Loads* are applied to the top left hand side node of the building, in the x-direction. The user is asked to define the *time-history curve* (a pre-defined standard curve is in any case already provided) and corresponding *curve multiplier* (scaling factor).
- Dynamic time-history analysis. In addition to permanent gravity actions, *Dynamic Time-history Loads* are applied at the foundation nodes of the building, in the x-direction. The user is asked to define the *time-history curve* (usually an accelerogram) and corresponding *curve multiplier* (scaling factor). A number of exemplificative time-history curves (consisting of natural and artificial accelerograms) are pre-installed with the program and can be loaded into the program through the *Select File* command.
- Incremental dynamic analysis. In addition to permanent gravity actions, *Dynamic Time-history Loads* are applied at the foundation nodes of the building, in the x-direction. The user is first asked to define the *Incremental Scaling Factors* (see IDA Parameters) and then needs to enter the *time-history curve* (usually an accelerogram) and corresponding *curve multiplier* (scaling

factor). A number of exemplificative time-history curves (consisting of natural and artificial accelerograms) are pre-installed with the program and can be loaded into the program through the *Select File* command.

- Response spectrum analysis. In addition to permanent gravity actions, static loads are applied to the nodes of each storey level according to the modal shapes. Since the load distribution is automatically adapted by the program, the user needs only to specify the acceleration spectrum data and the loading combinations. A user-defined spectrum can be introduced, or alternatively the time-history curves (consisting of natural and artificial accelerograms) pre-installed with the program can be used through the *Select File* command, and the program creates automatically the spectrum of the selected record.
- Buckling analysis. Permanent gravity loads are applied.
- Tsunami nonlinear analysis. In addition to permanent gravity actions, *Static Time-history Loads* are applied to the left hand side nodes of the building, in the x-direction. The *time-history curves* (pre-defined standard curves) and corresponding *curve multipliers* (scaling factors) are in any case already provided.

NOTE 1: When generating building models, the *Wizard* facility makes use of commonly encountered cross-sections dimensions and detailing, together with standard material properties. Evidently, after the completion of the model, the user may manually modify these input quantities so as to better represent the characteristics of the actual structure that he/she intends to analyse.

NOTE 2: The maximum building size that can be generated with the wizard is 8 bays x 8 storeys x 9 frames. Users who wish to create larger structures, however, can readily do so by employing the *Incrementation* facilities for nodes, elements, constraints and loads.

NOTE 3: To define structural members that are subdivided in more than 4 elements, the model can be wizard-created with 1, 2 or 4 elements per member and then the *Element Subdivision* facility can be employed to further discretise the structural mesh.

NOTE 4: The *Wizard* facility automatically activates the calculation of the Target Displacement in the case of pushover analysis. For further details users may refer to the Target Displacement paragraph.

NOTE 5: The *Wizard* facility automatically generates Code-based Checks. For details on their definition users may refer to the Code-based Checks paragraph.

NOTE 6: The *Wizard* facility automatically generates Performance Criteria checks. For details on their definition users may refer to the Performance Criteria paragraph.

EXPORTING AND IMPORTING SEISMOSTRUCT PROJECTS AS XML FILES

A SeismoStruct 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 included in the SeismoStruct project. An XML file containing the information of a SeismoStruct project can be loaded from the main menu (*File >Import from XML file)* while the information contained in the XML file can be modified directly in the XML file.

EXPORTING AND IMPORTING SEISMOSTRUCT PROJECTS AS OPENSEES FILES

A SeismoStruct project can be exported in the form of an OpenSees file from the main menu (*File >Export to OpenSees…).* When this option is selected, a new window appears for the definition of the name and location of the OpenSees file. The exported OpenSees file will contain all the information included in the SeismoStruct project. An OPenSees file can be loaded from the main menu (*File >Import from OpenSees…)*.

Pre-Processor

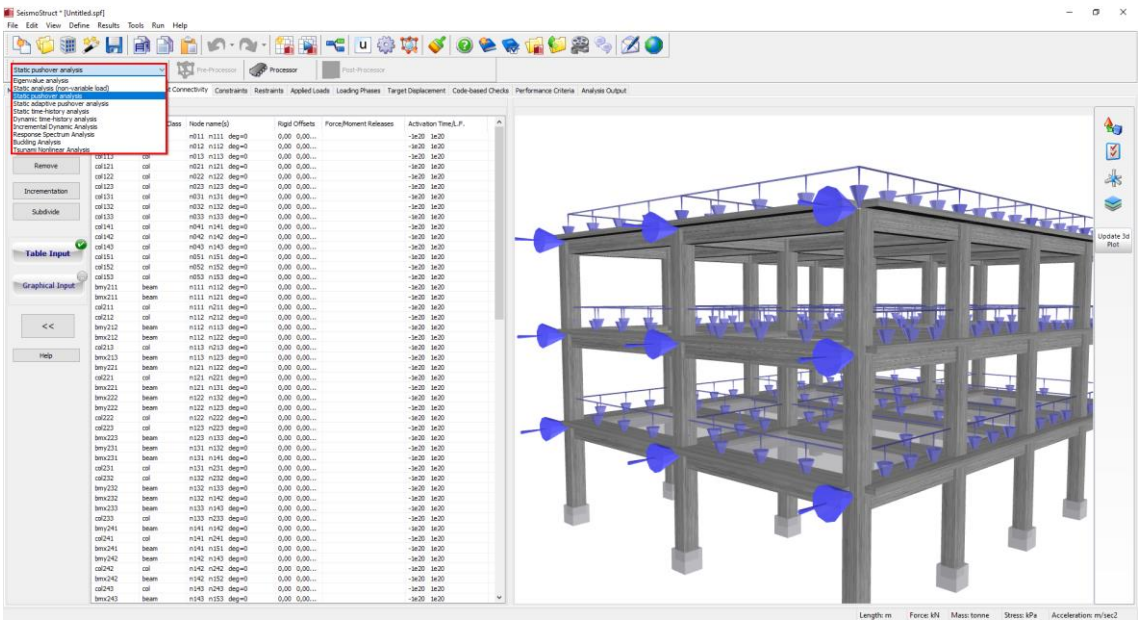
IMPORTANT: Before starting with a new SeismoStruct project, usually it is better to select first an analysis type.

Analysis Types

Currently, ten analysis types are available in the program:

- Eigenvalue analysis
- Static analysis (non-variable load)
- Static pushover analysis
- Static adaptive pushover analysis
- Static time-history analysis
- Dynamic time-history analysis
- Incremental Dynamic Analysis (IDA)
- Response Spectrum Analysis (RSA)
- Buckling Analysis
- Tsunami Nonlinear Analysis

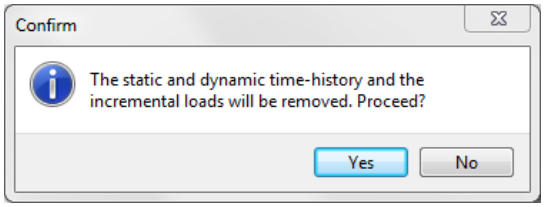
These can be easily selected from the drop-down menu at the top left corner on the **Pre-Processor** window (see picture below)



Selection of the analysis type

Different analysis types present equally diverse modelling requirements (see paragraphs below). Consequently, whereas the frame (elastic and inelastic) and link elements can be used for every analysis type, mass elements (lmass and dmass) are not needed in static analyses (with the exception of static adaptive pushover) and can be used only in dynamic, eigenvalue and adaptive pushover analysis. Moreover, damping elements (dashpt) are only needed in dynamic analysis. Whenever the analysis type is changed, the program automatically attempts to apply the required modifications to the existing

model. For example, if in an already-built dynamic analysis project, the analysis type is changed to static pushover, SeismoStruct will automatically remove the mass and damping elements.



Warning message

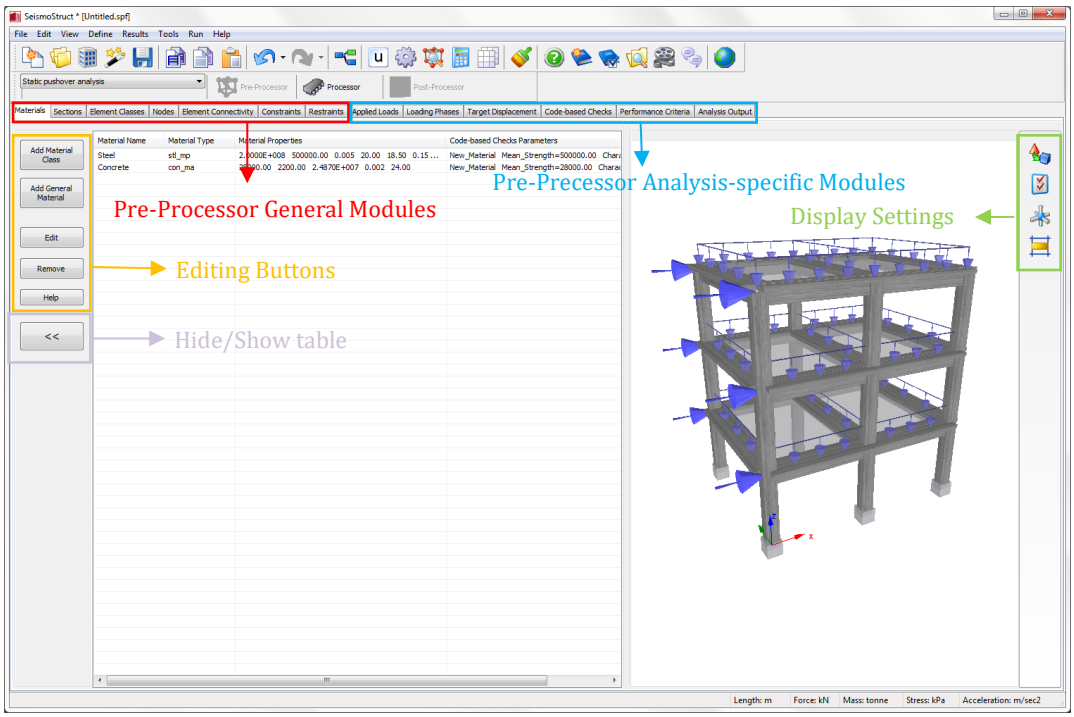
In addition, the different analysis types accept equally diverse types of loading (refer to the Applied Loads paragraph for details (*Pre-Processor > Loading > Applied Loads*)).

For a comprehensive description of the analysis types, refer to Appendix B - Analysis Types.

PRE-PROCESSOR AREA

SeismoStruct projects are created in its **Pre-Processor area**, which features a series of modules that are used in defining the structural model and its loading. These modules can be split into a *general-type of category* (**Materials**, **Sections**, **Element Classes**, **Nodes**, **Element Connectivity**, **Constraints**, **Restraints**, , **Analysis Output**) which apply to all types of analysis (that can be selected through a drop-down menu), and into *analysis-specific modules*, which appear only in some types of analysis (e.g. the **Code-based Checks** and the **Performance Criteria** modules appear to all types of analysis apart from the Eigenvalue analysis, whereas the **Adaptive Parameters** module is available only if the user chooses to run Static Adaptive Pushover Analysis).

In each aforementioned module it is possible to hide the data entry table through the corresponding button (see below) in order to view the 3D rendering of the structural model in 'full-screen' modality.



Pre-Processor Modules

IMPORTANT: All input information required to run an analysis (e.g. structural model, load pattern, output settings, etc.) is saved within a text-based SeismoStruct Project File, distinguishable by its *.spf extension; double-clicking on these files will open SeismoStruct in the Pre-processor area directly.

UNITS SELECTOR

Both SI as well as English units systems can be used in SeismoStruct, with different possible "combinations" being available for each of these two, since users are given the possibility of choosing between the use of two diverse units to define Length and Force quantities; as the units of these two base quantities are changed by the users, the program automatically adjusts the units of the remaining derived entities (Mass, Stress, Acceleration, etc.). Customisation of the Units system is carried out in the *Units Selector* dialog box, accessible from the main menu (*Tools > Units Selector*) or through the corresponding toolbar button  .

Below, please find a summary of the units systems that can be used in SeismoStruct. Note that rotations are always given in radians.

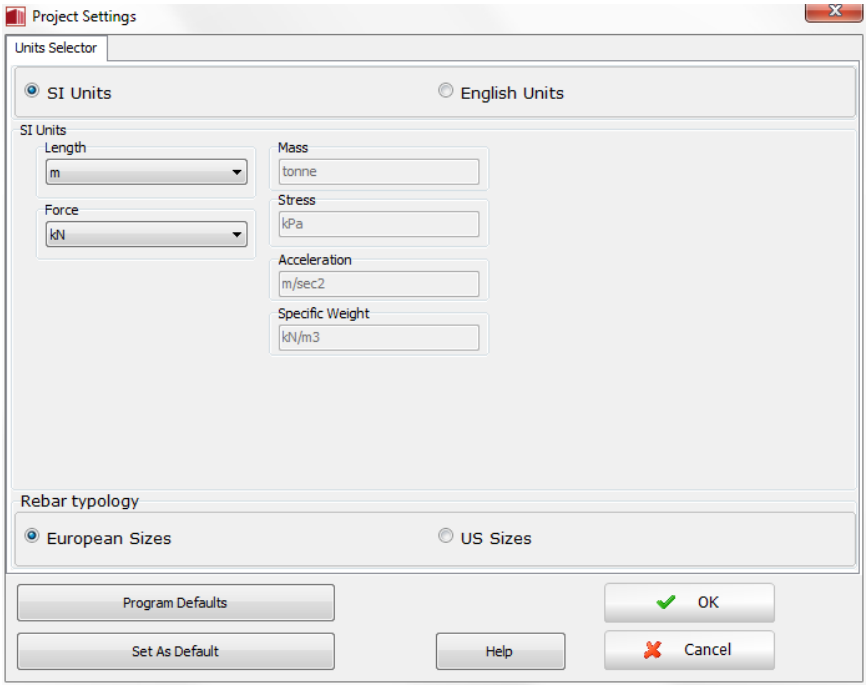
SI Units

Length	Force	Mass	Stress	Acceleration	Specific Weight
mm	N	ton	MPa	(9807) mm/sec2	N/mm3
mm	kN	kton	GPa	(9807) mm/sec2	kN/mm3
m	N	kg	Pa	(9.81) m/s2	N/m3
m	kN	ton	kPa	(9.81) m/s2	kN/m3

English Units

Length	Force	Mass	Stress	Acceleration	Specific Weight
in	lb	lb*sec2/in	psi	(386.1) in/sec2	lb/in3
in	kip	kip*sec2/in	ksi	(386.1) in/sec2	kip/in3
ft	lb	lb*sec2/ft	psf	(32.17) ft/s2	lb/ft3
ft	kip	kip*sec2/ft	ksf	(32.17) ft/s2	kip/ft3

Further, two different rebar typologies may be employed, European and American. It is noted that any combination of units (SI or English) and rebar types (European or American) may be used, for example it is possible to use SI units with American rebars, as it is e.g. customary in Latin American countries.



Units Selector tab window

EDITING

A common set of editing rules and options, which users are strongly advised to consult before embarking on the task of creating a model, apply to all pre-processor modules and are described below.

Editing functions

The majority of SeismoStruct modules feature a spreadsheet where all input parameters are kept and displayed. The data contained in these module tables can be manipulated with the following tools:

Adding new entries

When users click on the *Add* button a dialog box appears, where the properties and characteristics of a new model component (materials, sections, nodes, loads, etc.) can be introduced and fully defined. The procedure is straightforward, since all dialog box entries possess a descriptive text for guidance.

Multiple selection (using the *Control* or *Shift* keys) can be employed to apply a particular restraint or load to more than one node at a time, for as long as the multiple node selection is made before the user opens the *Add* dialogue box. Further, when using drop-down lists with many entries, users can start typing an item's identifier so as to reach it quicker.

NOTE: The identifiers (names) of module entries (materials, sections, nodes, loads, etc.) may be up to 32 characters long and should not contain spaces, #, & and punctuation marks (i.e. "." and ",").

Editing existing entries

If users wish to modify or check the properties of an existing module entry, they can make use of the *Edit* facility, which is accessed either through the *Edit* button, by double-clicking over the table entry of the item that is to be modified or by double-clicking over the corresponding node or element on the 3D plot of the model (the latter for nodes and elements only); an *Edit dialog box* opens, allowing for changes to be applied. Again, multiple selection and editing facility can be employed to modify any given input parameter in a multiple set of nodes, elements, restraints or assigned loads.

NOTE: In the Nodes, Element Connectivity, Restraints and Applied Loads modules users may select more than one item using the *Ctrl* and *Shift* keys and change particular properties of them at the same time. For example, the user may assign the same X coordinate in several nodes, or the same rotation angle in several frame elements with just one move.

Removing unused entries

Users can remove one or more items by selecting these and clicking the *Remove* button or using the *Delete* key on the keyboard.

Sorting table entries

Clicking on the column headings of each of the modules' tables, allows users to sort its items in ascending (one click) or descending (two clicks) order. For example, if a user clicks on the section names heading, SeismoStruct will sort the sections alphabetically, whilst if nodal x-coordinates heading is clicked instead, the nodes will be sorted according to their x-value. It is noted that by right-clicking on the nodes and elements tables in the respective module, the tables can be sorted by name or by number.

By default, whenever table entries are in number (e.g. 100) or word+number (e.g. nod20) formats, algebraic sorting is carried out, whilst if word format is used (e.g. beam_A) then alphabetical sorting is employed. However, it is nonetheless possible to change this default sorting behaviour through the *Sort by Name* and *Sort by Number* commands, accessible from the *Edit* or table popup menus.

NOTE: Entry sorting is a program-wide feature, meaning that the way in which model components (e.g. nodes, sections, elements, etc.) are sorted in their respective modules, reflects the way these entries appear on all dialogue boxes in the program. For instance, if the user chooses to employ alphabetical sorting of the nodes, then these will appear in alphabetical order in all drop-down menus where nodes are listed, which may, in a given case, ease and speed up their individuation and selection.

Copying and pasting table entries

Users can copy and paste data to and from all module spreadsheets, be it within inside SeismoStruct or in interaction with any other Windows application (e.g. Microsoft Excel, Microsoft Word, etc.). Copying and pasting can be carried out either through the program menu (*Edit > Copy Selection and Edit > Paste Selection*), through the respective toolbar buttons   , through the table popup menu (available with the right-click mouse button) or through the keyboard shortcuts (*Ctrl+C* and *Ctrl+V*).

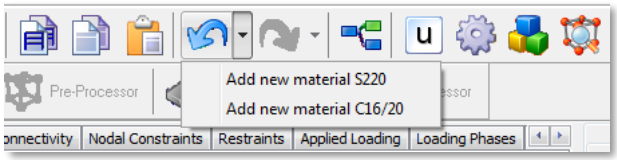
You can use this facility to ease the creation of any model component by copying an already defined module entry and pasting it in the respective module spreadsheet, noting that a star superscript (*) is added at the end of the new entry's name so as to avoid duplications. In addition, users can also create their component listing in a different application (e.g. Microsoft Excel) and then paste into SeismoStruct, for as long as the entries are consistent with the format of the respective module.

Copying 3D plot

Users can also copy, to an external Windows application (e.g. Microsoft Word, Microsoft PowerPoint), the 3D plot of the structural model being created. This is accomplished through the program menu (*Edit > Copy 3D Plot*), through the respective toolbar button  , through the plot popup menu (available with the right-click mouse button) or through a keyboard shortcut (*Ctrl+Alt+C*).

Undoing and redoing operations

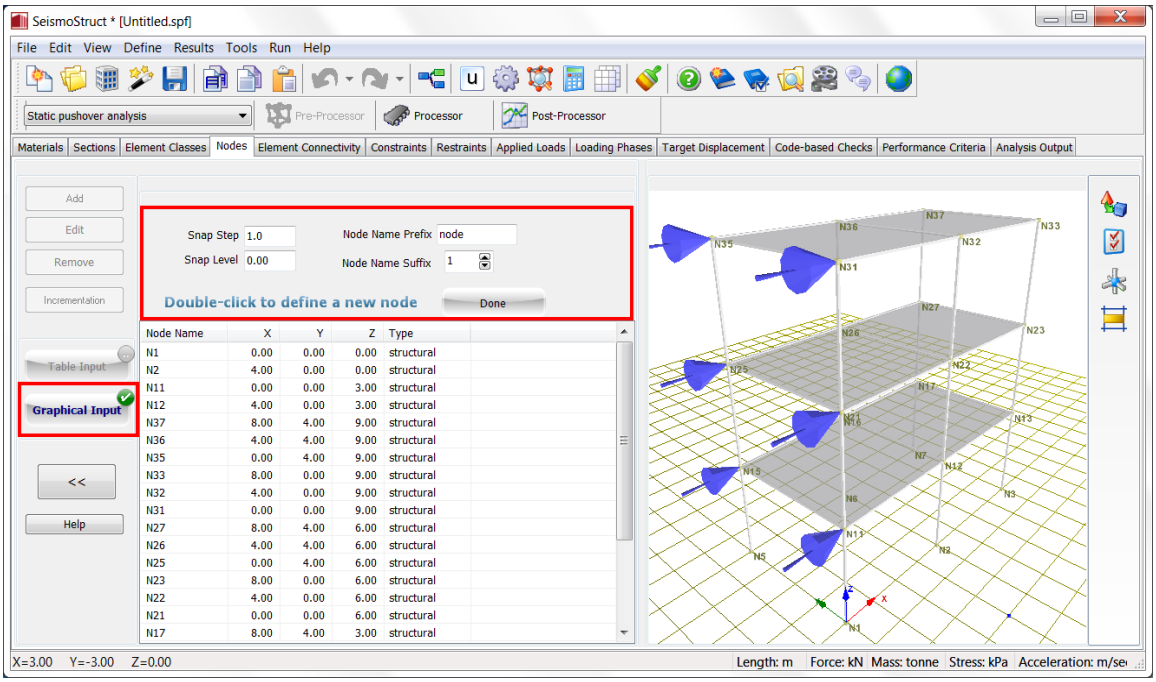
There is an undo-redo facility in SeismoStruct, accessible through the program menu (*Edit > Undo* and *Edit > Redo*) or through the respective toolbar buttons  and  . In addition, through the drop-down menu, multiple operations are also possible.



Undoing and redoing multiple operations

Graphical Input/Generation

In addition to its menu-based model editing facility (and to the *Wizard* and *Building Modeller* facility), structural models can also be generated in a completely graphical manner (Point & Click) through the *Graphical Input* facility, available for **Nodes**, **Element Connectivity, Constraints** and **Loads**, as described in the Structural Geometry paragraph.

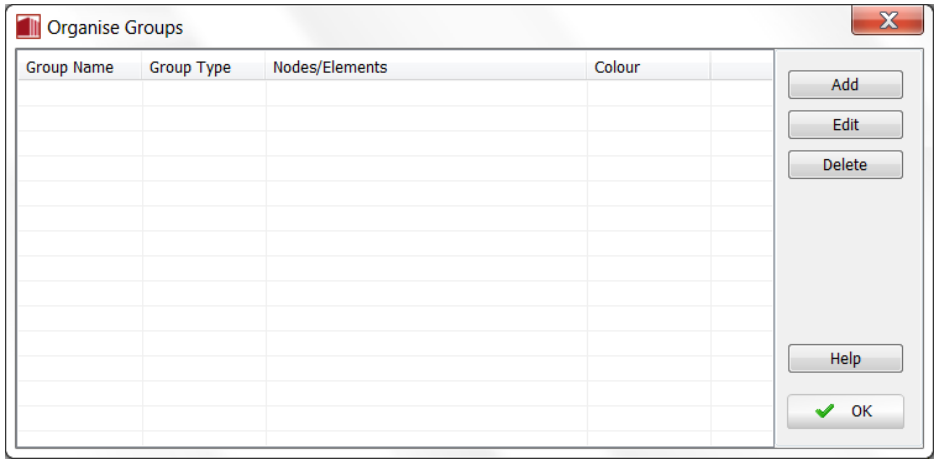


Graphical Input facility for Nodes module

Within this context, users are also advised to take advantage of the presence of *Cut Planes* visualisation facility (see 3D Plot options paragraph), to ease the view and graphical generation of complex 3D models and of the possibility of shrinking/expanding frame elements visualisation, again to facilitate point & click of nodes.

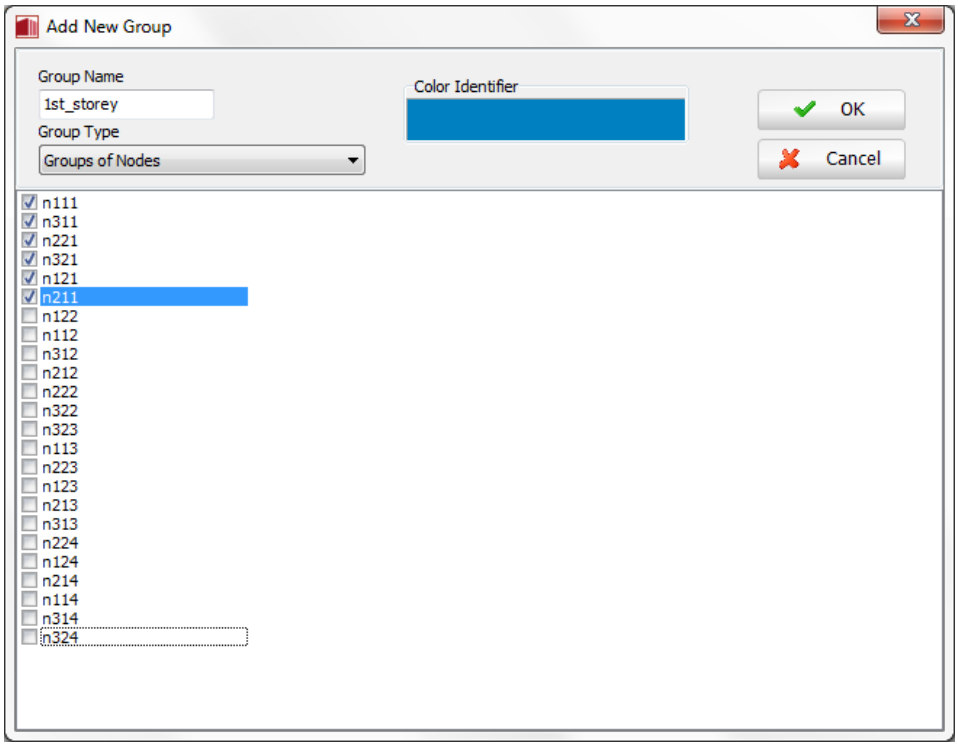
Node/Element Groups

One other power-user facility of SeismoStruct consists on the possibility for the creation of node or element groups. Typically, these nodes/elements feature common characteristics (e.g. they belong to the top storey of a building, they define the deck of a bridge, etc.) and grouping them together serves the purpose of facilitating their individuation and selection in many Pre- and Post-Processing operations. The Groups dialog box is accessed from the main menu (*Edit* > *Organise Groups...*) or through the corresponding toolbar button .

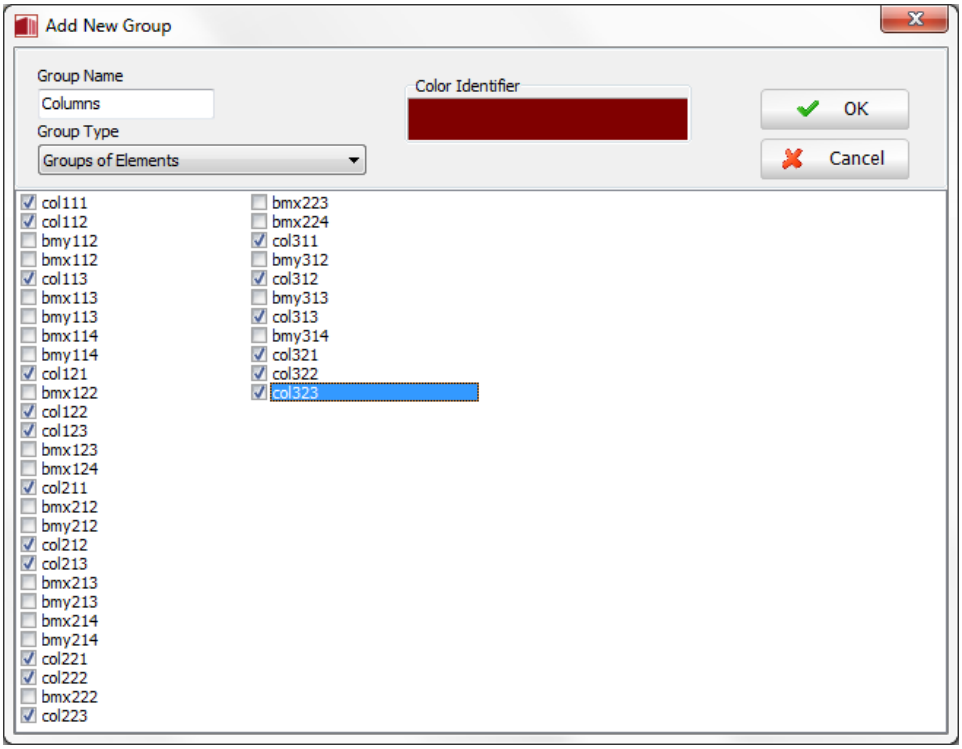


Organize Groups function

Users can add, edit and delete node and element groups using the *Organise Groups* facility, where a list of all nodes and elements used in the current structural model are displayed.



Adding a New Group (nodes)



Adding a New Group (elements)

In addition, users can also use a selection of nodes and elements, made within the **Nodes** and **Element Connectivity** modules respectively, and use the popup menu to add them to a new group. The latter is probably the most effective way of creating a new group, since users can in this way take advantage of the different sorting options to make the selection of nodes/elements of interest significantly faster.

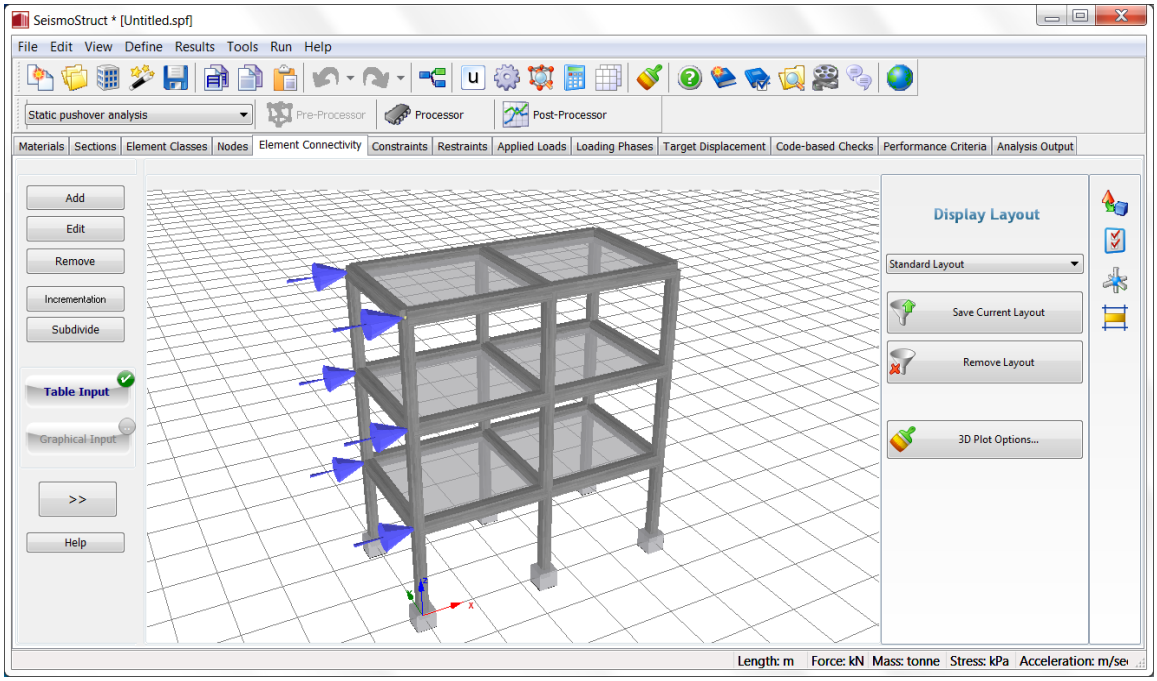
NOTE: The Groups facility is particular useful for selecting nodes and elements to be post-processed, thus reducing the size of output files and speeding up post-processing operations.

3D Plot options

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

Display Layout

Within this pop-up menu, accessible through the toolbar button  , users can (i) select a pre-defined layout, such as *Standard Layout* (default), *Transparent elements* and *Line elements* (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:

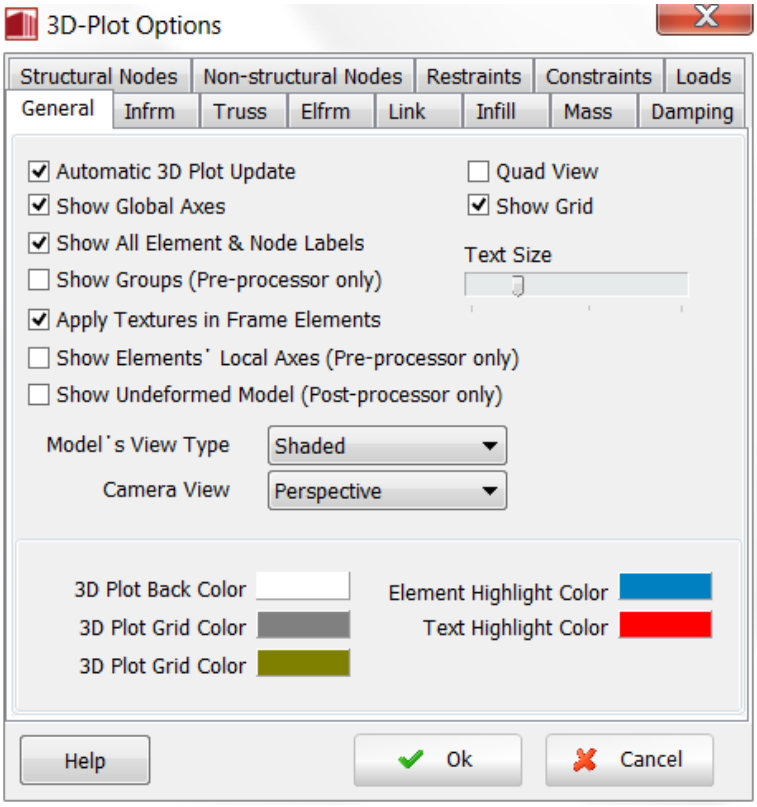
1. Click on the toolbar button ;
2. Assign a name to the new layout configuration;
3. Click the *OK* button to confirm the operation.

The new layout will appear in the drop-down menu located in the toolbar. Further, the user 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 corresponding toolbar button .

Within the *3D Plot Options* menu, there are a number of submenus from which users can not only select which model components (nodes, frame and mass/damping elements, links, 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.

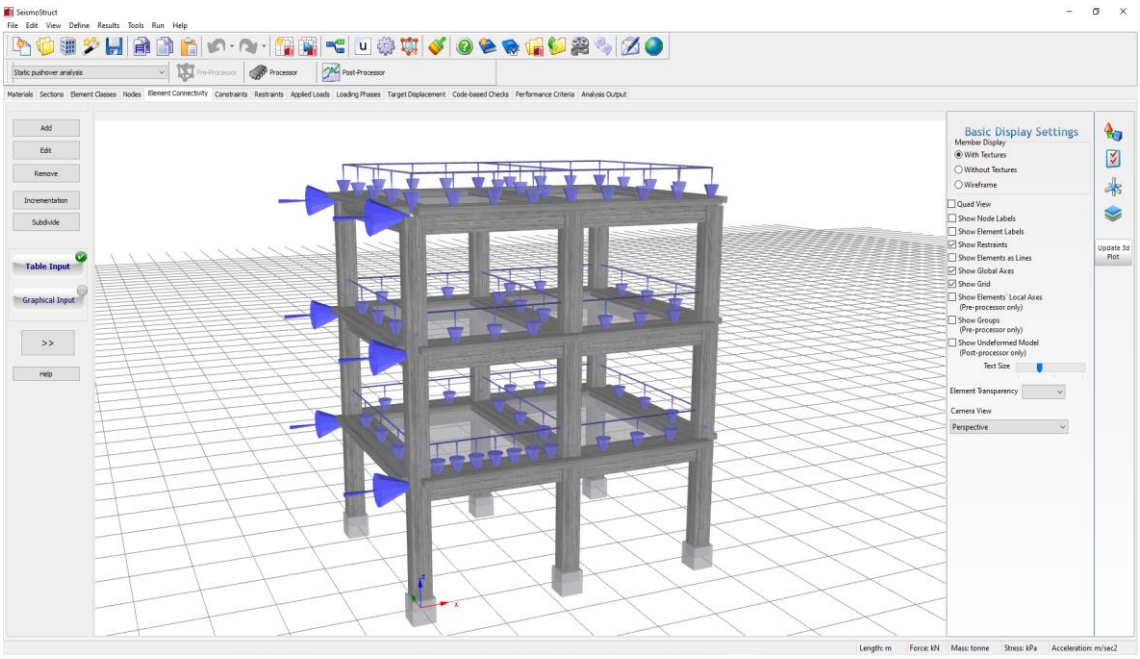


3D Plot Options menu

By default, the 3D Plot is automatically updated, implying that for every input change (e.g. addition of a node or an element), the model plot is refreshed in real-time. This behaviour may be undesirable in cases where the structural model is very large (several hundreds of nodes and 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). In such situations 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 *Redraw 3D Plot* command, found in the *Tools* and popup menus.

Basic Display Settings

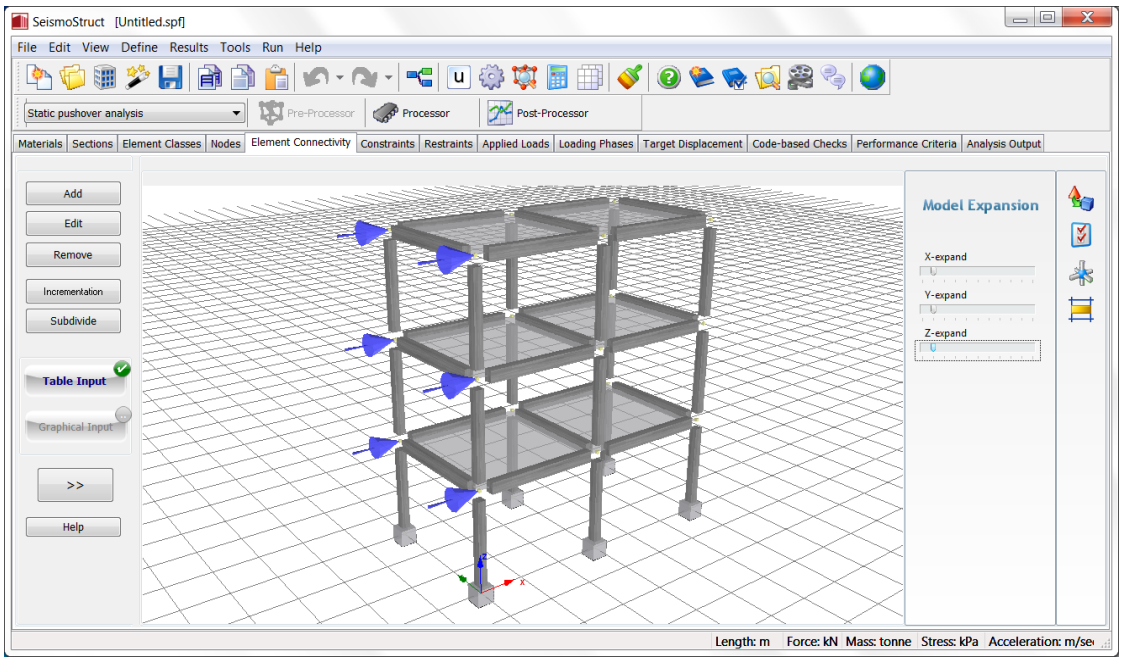
Within this pop-up menu, accessible through the toolbar button , users can tweak the most commonly used plotting features (view type, rendering options, names show, local axes representation, element transparency, and so on) using the available check-boxes and drop-down menus.



Basic Display Settings

Model Expansion

Using this feature, accessible through the toolbar button , the 3D model may be expanded in each global direction (i.e. X, Y and Z) by moving the corresponding cursor.

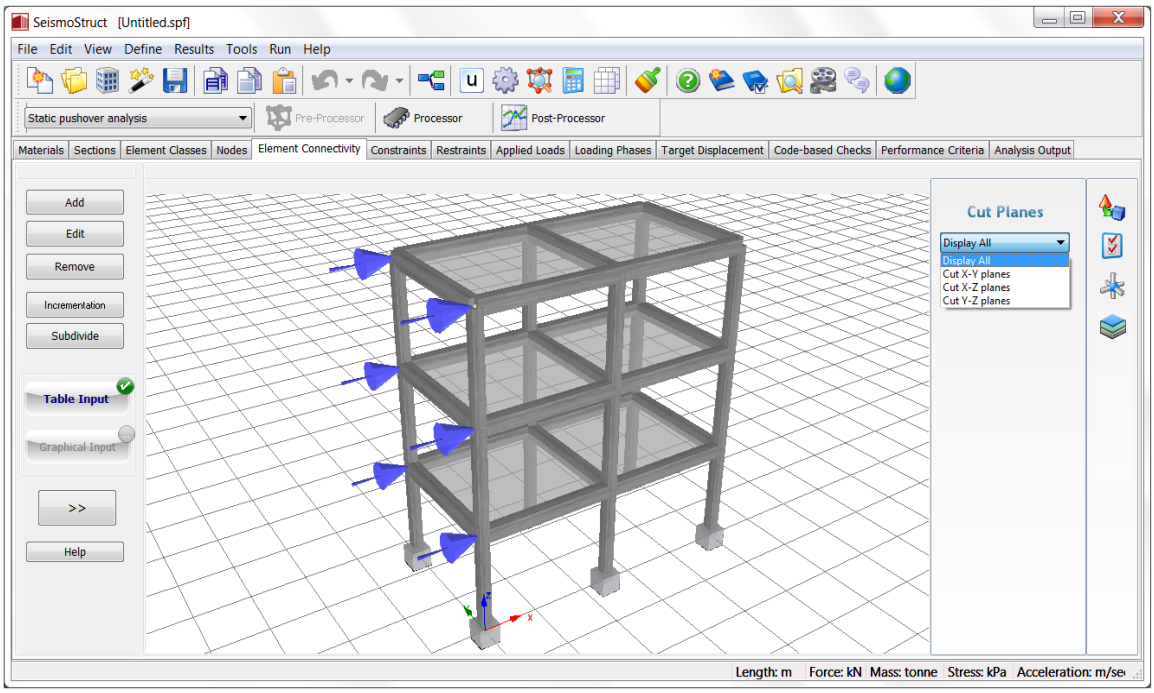


Model Expansion

Cut Planes

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

NOTE: By default the *Display All* option is selected from the drop-down menu.



Cut Planes

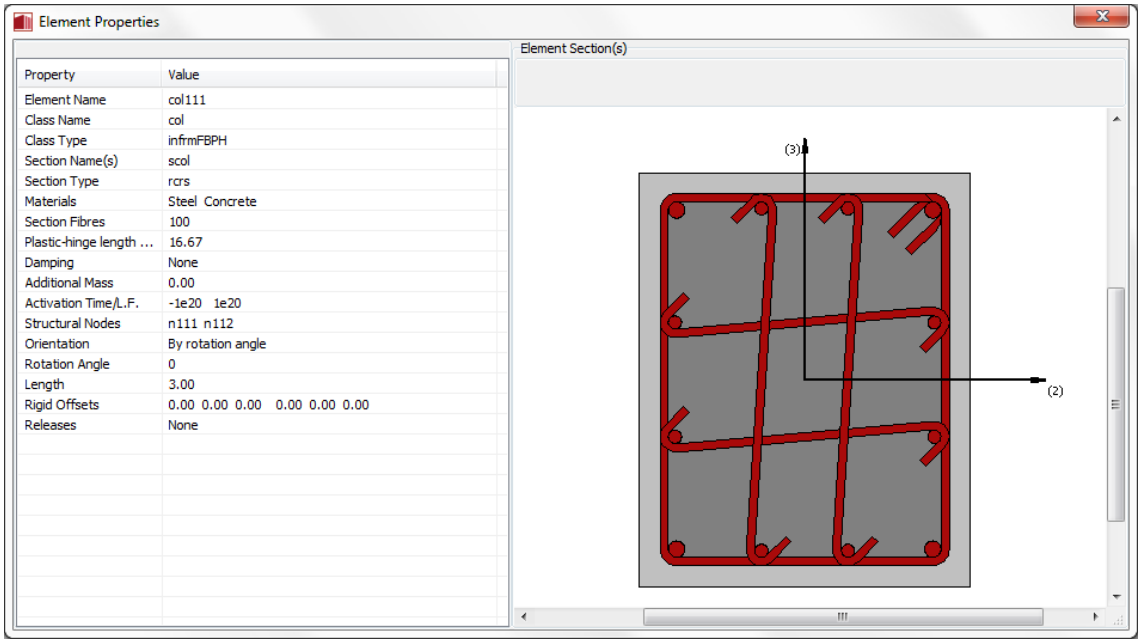
Additional operations

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 nodes and elements, so as to quickly select their corresponding list entry. If, instead, the user chooses to double-click a given node/element, then the corresponding editing dialog box opens.

Finally, by right-clicking on a given element, users can visualise the "summary" of the element properties in a specific dialog box (➔ *Element Properties* from the drop-down menu).

NOTE 1: When users define non-structural nodes with very large coordinates, and then activate visualisation of such nodes, the model will inevitably be zoomed-out to a very small viewing size. To avoid such a scenario, users should (i) bring the non-structural nodes closer to the structure, (ii) disable visualisation of the latter or (iii) zoom-in manually every time the 3D plot is refreshed.

NOTE 2: Activating visualisation of local axes may result in a quite congested 3D model representation, especially when link elements are present, rendering difficult the interpretation/check of local axes' orientation. In such cases, users may simply disable visualisation of some elements (e.g. frame elements) in order to more readily check some others (e.g. links).



Element Properties

Rotating/moving the 3D model

Instruction	Using Keyboard	Using Mouse
Zoom In	press the '*Arrow-up*' key	scroll the mouse-wheel upwards
Zoom Out	press the '*Arrow-down*' key	scroll the mouse-wheel downwards
Rotate Left	press the '*Arrow-left*' key	drag mouse to the left whilst pressing the left mouse-button
Rotate Right	press the '*Arrow-right*' key	drag mouse to the right whilst pressing the left mouse-button
Rotate Up	press the '*Ctrl* + *Arrow-up*' keys	drag mouse upwards whilst pressing the left mouse-button
Rotate Down	press the '*Ctrl* + *Arrow-down*' keys	drag mouse downwards whilst pressing the left mouse-button
Move Left	press the '*Ctrl* + *Arrow-right*' keys	drag mouse to the left whilst pressing the right mouse-button
Move Right	press the '*Ctrl* + *Arrow-left*' keys	drag mouse to the right whilst pressing the right mouse-button
Move Up	press the '*Shift* + *Arrow-down*' keys	drag mouse upwards whilst pressing the right mouse-button
Move Down	press the '*Shift* + *Arrow-up*' keys	drag mouse downwards whilst pressing the right mouse-button

NOTE: If wheel zooming is excessive, then either use the keyboard or adjust your mouse wheel scrolling settings (Windows Control Panel).

PROJECT SETTINGS

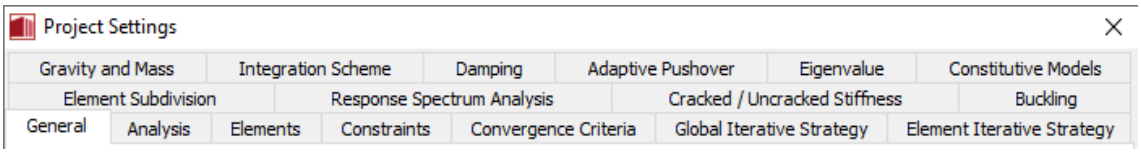
For each SeismoStruct project it is possible to customise both the usability of the program as well as the performance characteristics of analytical proceedings, so as to better suit the needs of any given structural model and/or the preferences of a particular user. This program/project tweaking facility is

available from the *Project Settings* panel, which can be accessed through *Tools > Project Settings...* or through the corresponding toolbar button .

NOTE: Users are advised to always reset the *Project Settings* to its *Program Defaults* after the installation of a new version, since there may be cases where these have not been correctly installed.

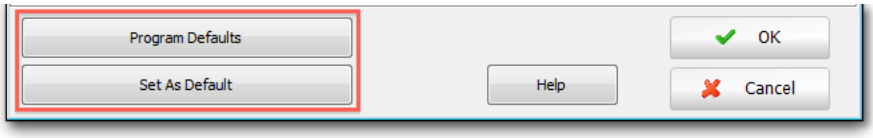
The *Project Settings* panel is subdivided in a number of tab windows, which provide access to different type of settings, as described below:

- General
- Analysis
- Elements
- Constraints
- Adaptive Pushover
- Eigenvalue
- Constitutive Models
- Element Subdivision
- Response Spectrum Analysis
- Cracked/Uncracked Stiffness
- Buckling
- Convergence Criteria
- Global Iterative Strategy
- Element Iterative Strategy
- Gravity and Mass
- Integration Scheme
- Damping



Project settings tab windows

Common to all tab windows are the *Program Defaults* and *Set As Default* options found at the bottom of the *Project 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 Project Settings default values to be modified, since these have been chosen so as to fit the requirements of standard type of analysis and models, 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 likings and preferences.

Binary Output

When activated, the *Binary Output* option will lead to the creation of a binary file (*.srf) containing the output of the entire analysis.

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.

NOTE: At least one type of output, binary or text, should always be selected.

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.

Display Warning Messages

When the *Display Warning Messages* option is activated, users are presented before the beginning of the analysis with warning messages about possible problems that might arise, e.g. convergence difficulties, loads acting on supports, parameters that seem unreasonable etc. The deactivation of the warning messages might prove really useful when running multiple analyses within the SeismoStruct Batch facility. By default this option is active.

NOTE: The warning messages presented before the beginning of the analysis are automatically closed after 2 minutes, if there is no input by the user.

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.

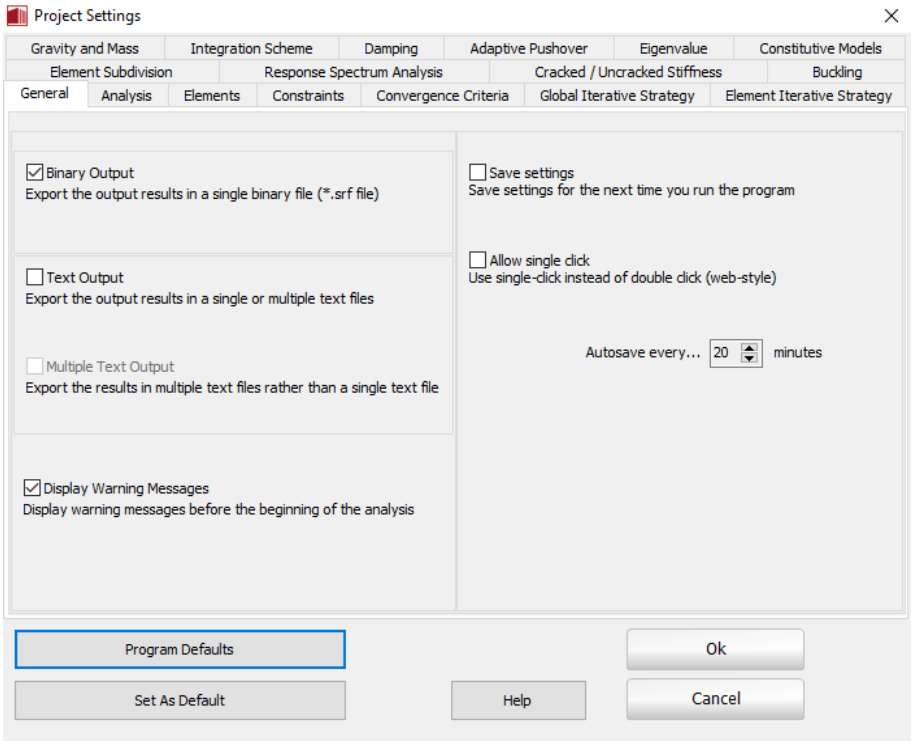
NOTE: Normally, this option is disabled so that the default settings are only changed if explicitly requested by the user (using the *Set as Default* option).

Allow single click

When selected, this option gives the program a web-style single click feel (as opposed to the more common double-click functioning standard).

Autosave every...

So as to protect users against accidental deletion of project files, SeismoStruct automatically creates a backup of the latter at user-specified time intervals (the default is 20 min). The backup files feature a *.bak extension. This facility can be disabled by setting a time interval equal to zero.



General tab window

Analysis

In the *Analysis* tab window some options related to the analysis can be defined. In particular, it is possible to select the solver type, whether to perform eigenvalue analysis at every step in nonlinear dynamic and pushover analysis and to account (or not) for geometric nonlinearities.

Solver

Apart from the linear equation solver, 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 dynamic and 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 Mackayet 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 the 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.

Geometric Nonlinearities

Unchecking this option will disable the geometric nonlinearity formulation described in Appendix A, 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*'.

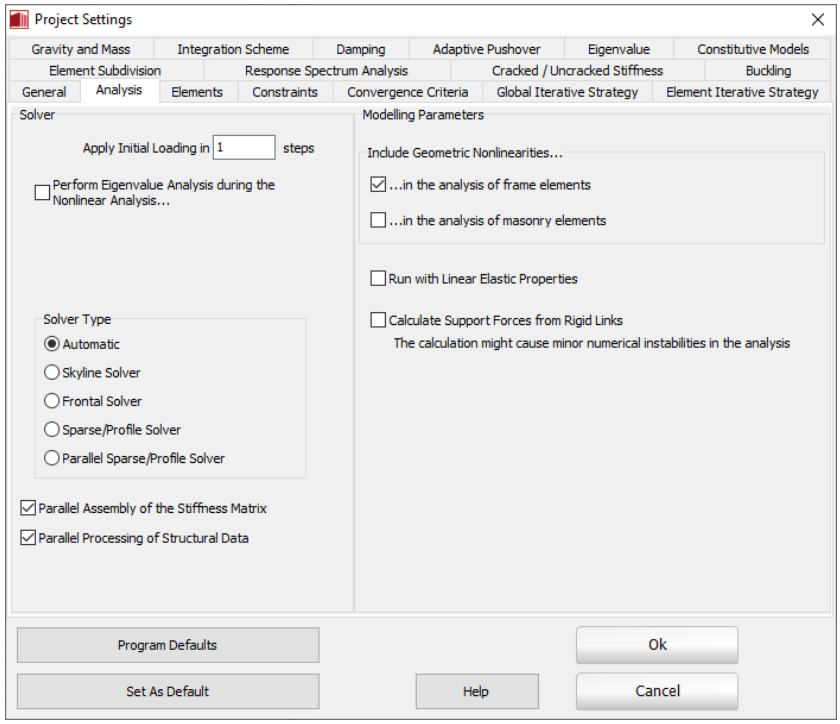
Run with Elastic Linear Properties

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 SeismoStruct for the elastic and 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. Users are therefore advised to disable this option in those cases where obtaining the exact values of internal forces is not of primary importance.

NOTE: *Stress Recovery* option is only of use when distributed loads are defined through the definition of material specific weight or of sectional/element additional mass, but not through the introduction of dmass elements.

Carry out Performance Criteria Checks only at the End Integration Sections

By activating this option users may select to carry out the defined Performance Criteria checks only at the end integration sections of the inelastic force-based element type (infrmFB), which are the locations on the member where checks are typically carried out. In this way, only the useful results are exported, without wasting time in processing the whole output for all the integration sections, and without confusing the user with redundant output.

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

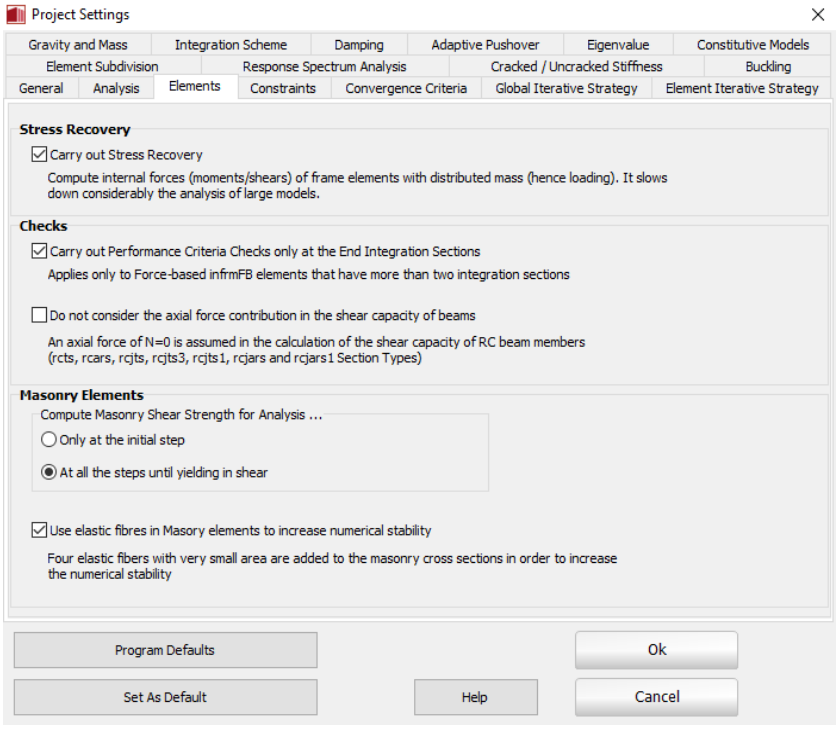
By activating this option the ability to carry out shear checks ignoring the actual axial force applied on the beam member is provided. This feature is particularly important to the shear capacity checks of beams, when the interaction between fibre modelled RC beams and the rigid diaphragm adopted to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

Compute Masonry Shear Strength for Analysis

With this setting users may choose whether to calculate the masonry shear strength (i) only at the initial step or (ii) at all the steps until yielding in shear, i.e. even after reaching of the peak member capacity. The default option is the second, to update the shear strength until yield is reached, which is the best combination of accuracy and stability, since updating the shear strength in the descending branch of the capacity curve may lead to some convergence difficulties without significantly improving the accuracy of the solution.

Use elastic fibres in Masonry elements to increase numerical stability

Increased numerical stability is provided through the addition of very small elastic fibres in masonry elements. These fibres allow for better convergence during the analysis without significantly affecting the element's overall response. By default, this option is active.



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, and have thus been implemented in SeismoStruct.

NOTE: Users are advised to refer to the existing literature [e.g. Cook et al., 1989; Felippa, 2004] for further information on this topic.

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

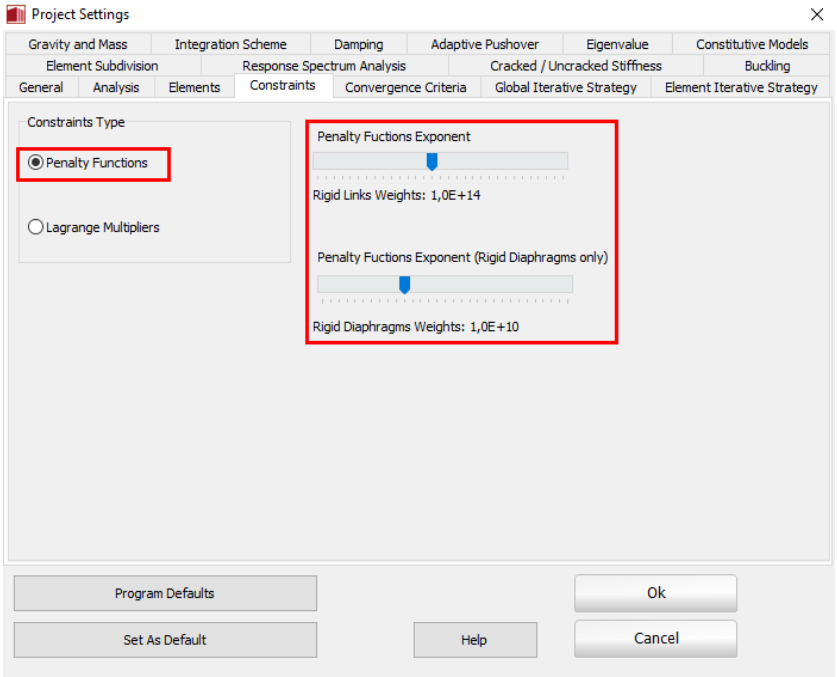
In addition, **Penalty Functions** have the disadvantage that penalty numbers must be chosen in an allowable range (large enough to be effective but not so large as to induce numerical difficulties), and this is not necessarily straightforward [Cook et al., 1989], and may potentially lead to erroneous results.

However, the use of the conceptually superior **Lagrange Multipliers** may slow analyses considerably, and, as such, the **Penalty Functions** are suggested as default in SeismoStruct.

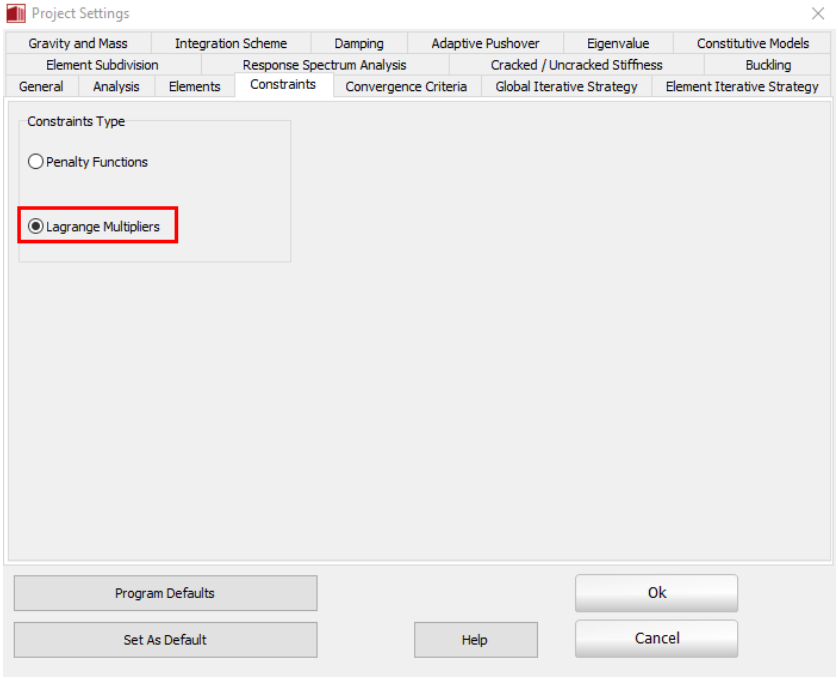
In those cases where the employment of **Lagrange Multipliers** leads to numerical difficulties and users opt for the utilisation of **Penalty Functions**, then the corresponding penalty coefficients, for diaphragm (typically smaller) and rigid links (typically larger) need to be defined; the Penalty Factors are then computed as the product of these penalty coefficients and the highest value found in the stiffness matrix.

It is noted that, contrary to what could perhaps be one's intuition, the use of large values of penalty coefficients is not always required. Indeed, in models where very stiff structural elements already exist, penalty coefficients may need not to be extremely large, since their product by such large values found in the structural stiffness matrix will already lead to a large penalty factor, as shown in the study by Pinho et al. [2008a].

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 SeismoStruct).



Constraints tab window – Penalty Functions



Constraints tab window – Lagrange Multipliers

Adaptive Pushover

In addition to the parameters defined in the **Adaptive Parameters** module, some advanced settings can be selected in this window. These settings are: (i) the **Type of Updating**, (ii) the **Update Frequency** and (iii) the **Modal Combination method**. They are described in detail hereafter.

Type of Updating

This adaptive option defines how the load distribution profile is updated at each analysis step. Four alternatives are available:

- **Total Updating**. The load vector for the current step is obtained through a full substitution of the existing balanced loads (load vector at previous step) by a newly derived load vector, computed as the product between the current total load factor, the current modal scaling vector and the initial user-defined nominal load vector. This updating option is not recommended, since it features limited theoretical support.
- **Incremental Updating**. The load vector for the current step is obtained by adding to the load vector of the previous step (existing balanced loads), a newly derived load vector increment, computed as the product between the current load factor increment, the current modal scaling vector and the initial user-defined nominal load vector. Incremental Updating usually is conceptually sounder than total updating, for which reason it is the default option.
- **Hybrid Updating**. With this third load vector updating option, the possibility of combining the two methods described above, is provided. In this manner, the load vector for the current step is obtained through partial substitution of the existing balanced load vector by a newly derived load vector and by the partial addition of a newly derived load vector increment. The percentage ratios that may lead to an optimum solution, in terms of accuracy and numerical stability, obviously vary according to the model characteristics, the type loading it is subjected to (displacements or forces), and the response spectra used in the determination of the modal scaling vector (if one is being used).
- **Fully Incremental Updating**. The load vector for the current step is obtained by adding to the load vector of the previous step (existing balanced loads), a newly derived load vector increment that reflects the changes in the current modal properties of the structure.

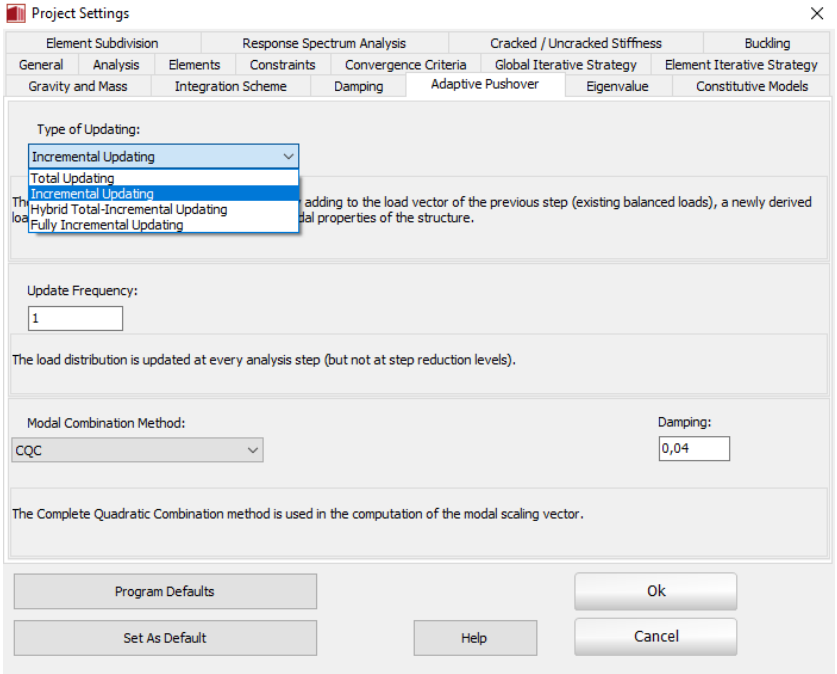
Update Frequency

This parameter defines how and when the modal scaling vector is updated during the analysis. Any integer larger than zero can be used. The default is 1, which means that the load distribution is updated at every analysis step, with the exception of steps where the analysis increment has been reduced due to convergence difficulties (automatic step adjustment). In those cases where a very large number of analysis steps have been defined by the user (i.e. the load is being applied in very small increments), it might be advantageous to use a frequency value that is larger than 1 (i.e. the modal scaling vector does not come updated at every step) so as to reduce the duration of the analysis without loss of accuracy.

Modal Combination method

Three modal combination rules can currently be utilised in the computation of the modal scaling vector, consisting of (i) the well-known **Square Root of the Sum of Squares** (**SRSS**), (ii) the **Complete Quadratic Combination** (**CQC**) and (iii) the **Complete Quadratic Combination with three components** (**CQC3**) methods [see e.g. Clough and Penzien, 1993; Chopra, 1995; Menun and Der Kiureghian 1998]. It is acknowledged that there are conspicuous limitations associated to the use of these always-additive modal combination methods, as discussed by many researchers [e.g. Kunnath, 2004; Lopez, 2004; Antoniou and Pinho, 2004a] and an optimum ideal methodology is yet to be identified. Such limitations, however, may be partially overcome with the employment of Displacement-based Adaptive Pushover, as shown by Antoniou and Pinho [2004b] and Pinho and Antoniou [2005], amongst others.

In addition, users may also employ a **Single-Mode** in the computation of the modal scaling vector, in which case they are asked to define the mode number and corresponding degree of freedom to be used. This may come particularly handy on those situations where the user does not have ways to estimate/represent the expected/design input motion at the site in question, in which case he/she should use DAP-1st mode (for buildings only).



Adaptive Pushover tab window

Eigenvalue

Whenever eigenvalue or adaptive pushover analyses need to be run, 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 is 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 and bridges, 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, sufficiently large to ensure that, for the vast majority of structural configurations, solutions will always be obtained.

NOTE 1: Since the Lanczos algorithm implemented in SeismoStruct may struggle to converge with small models featuring a limited number of degrees of freedom (i.e. 1 to 3), users are advised to instead employ the Jacobi-Ritz option for such cases.

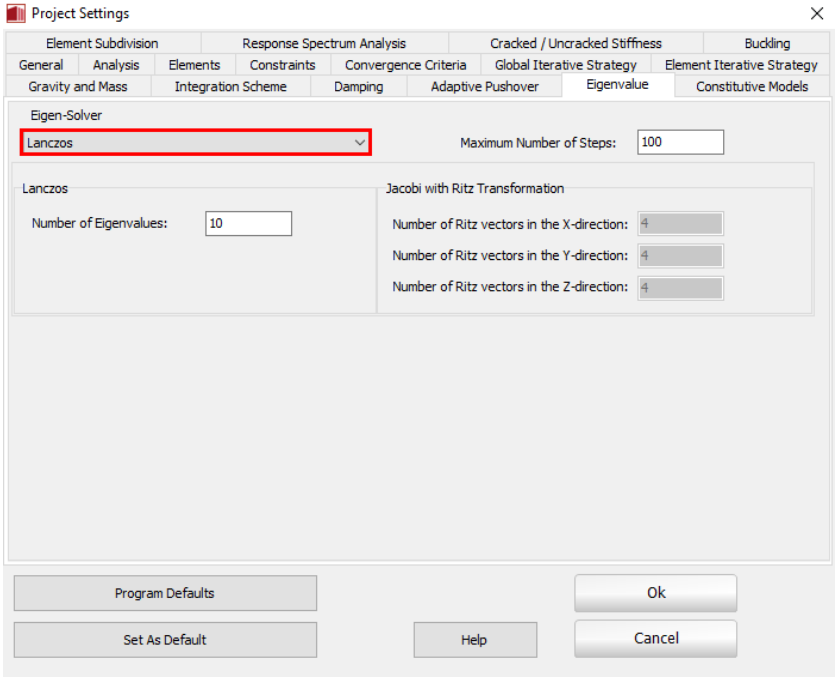
NOTE 2: When running an eigenvalue analysis, user may be presented with a message stating: "*could not re-orthogonalise all Lanczos vectors*", meaning that the Lanczos algorithm could not calculate all or some of the vibration modes of the structure. This behaviour may be observed in either (i) models with assemblage errors (e.g. unconnected nodes/elements) or (ii) complex structural models that feature links/hinges etc. If users have checked carefully their model and found no modelling errors, then they may perhaps try to "simplify" it, by removing its more complex features until the attainment of the eigenvalue solutions. This will enable a better understanding of what might be causing the analysis problems, and thus assist users in deciding on how to proceed. This message typically appears when too many modes are sought, e.g. when 30 modes are asked in a 24 DOF model, or when the eigensolver cannot simply find so many modes (even if DOFs > modes).

Jacobi algorithm with Ritz transformation

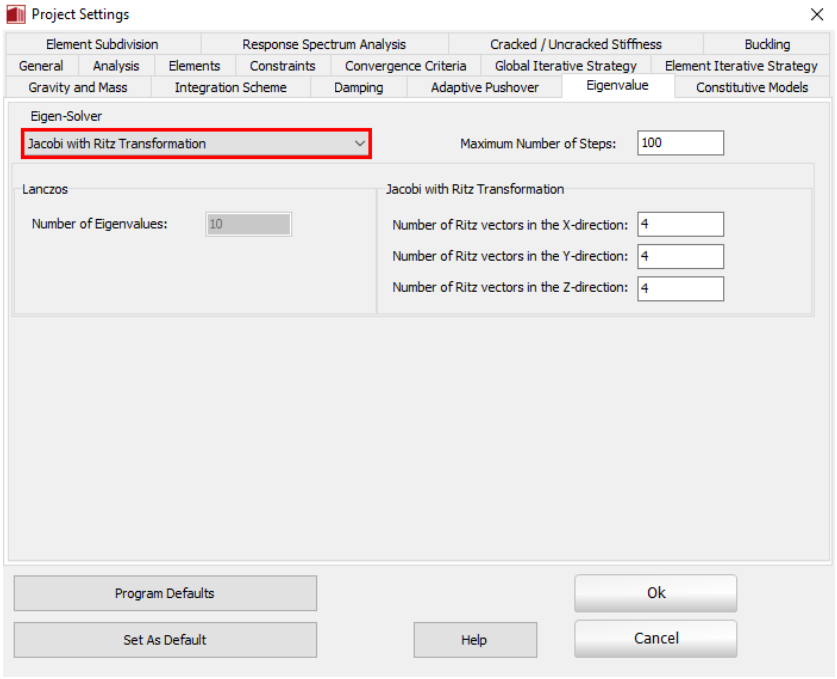
The user may specify:

- **Number of Ritz vectors** (i.e. modes) to be generated in each direction (X, Y and Z). This number cannot exceed the number of dof.
- **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



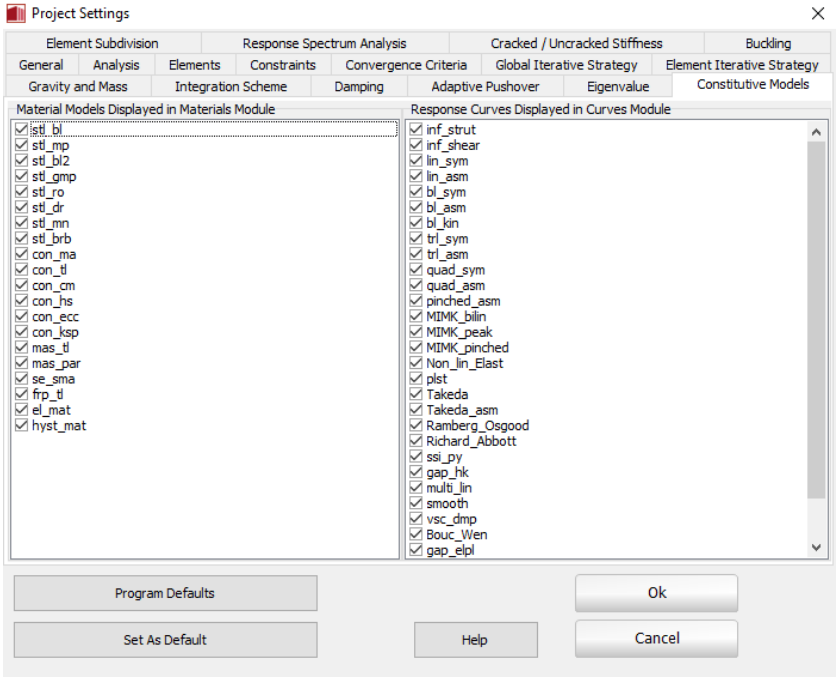
Eigenvalue tab window – Lanczos algorithm



Eigenvalue tab window – Jacobi algorithm

Constitutive Models

Herein, material models and response curves that will be displayed, respectively, in *Materials* module and *Element Classes* module can be activated.

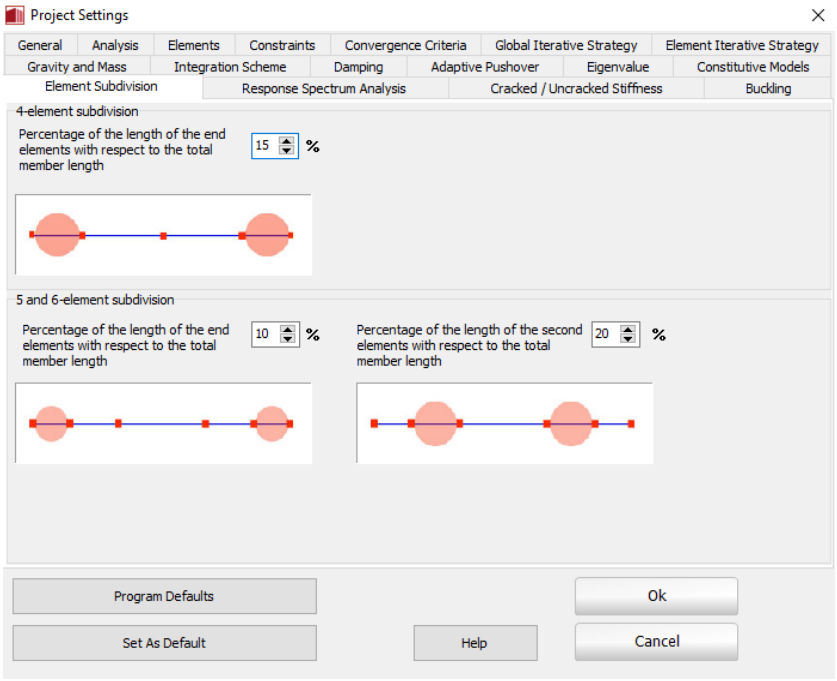


Constitutive Models tab window

NOTE: By default, all material models are selected.

Element Subdivision

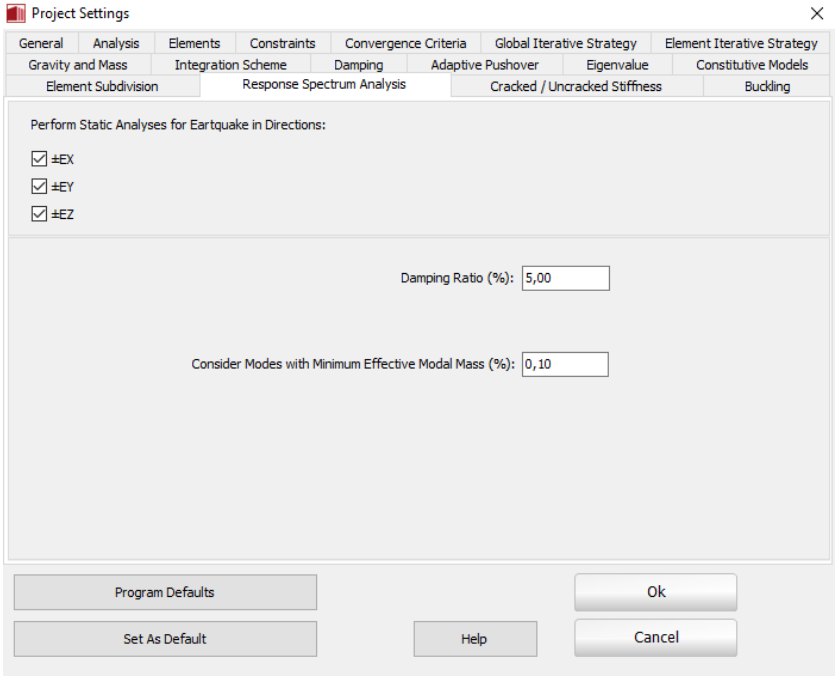
It is possible for users to subdivide existing elements defined in the Element Connectivity module into 2, 4, 5 and 6 smaller components. In that case, it is common for elements at the edge of the member, where material inelasticity usually develops, to be smaller in length so as to more accurately model the eventual formation of plastic hinges. The length of such edge elements can be customised in this menu. If the 4-element subdivision has been selected, the default is for end elements to feature a length that is 15% that of the structural member, thus leading to a member subdivision, in terms of its length, of the type 15%-35%-35%-15%. For the case of the 5- and 6-element subdivision facility, it becomes necessary to establish the length of the new edge components (default is 10% of the initial length of the element) and that of the "second" components (default is 20% of the initial length of the element).



Element Subdivision tab window

Response Spectrum Analysis

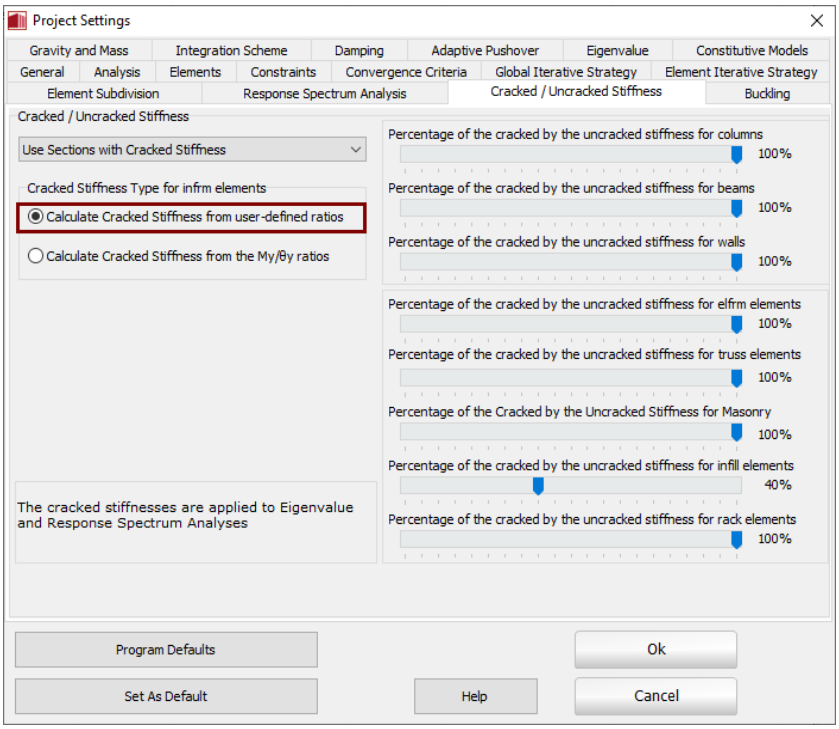
A Response Spectrum Analysis has been added in v7.0 of SeismoStruct. Herein the users can choose in which directions the seismic components will be taken into consideration; by default all the directions, ±EX, ±EY, ±EZ, are selected. Further, the damping ratio of the model and which modes are to be taken into account according to the minimum effective modal mass are defined. The default value of the damping ratio is 5%, whereas a threshold of 0.1% is set for the minimum effective modal mass of the modes to be considered.



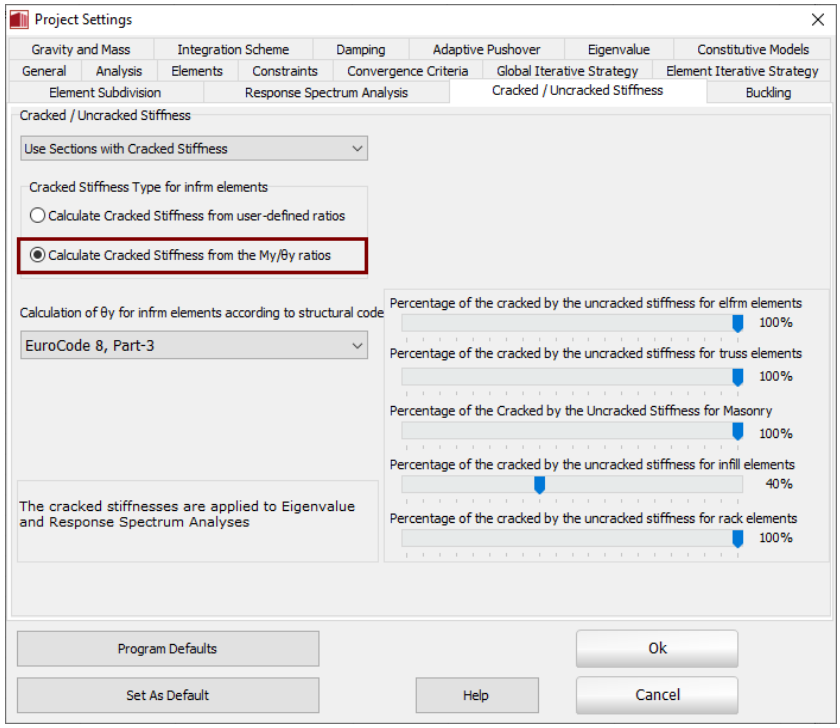
Response Spectrum Analysis tab window

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/Uncracked Stiffness tab window- user-defined ratios



Cracked/Uncracked Stiffness tab window- M_y/θ_y ratios

Buckling

Whenever buckling analysis need to be run an eigenvalue analysis is carried out, in order to determine the modes of vibration of a structure. The **Jacobi algorithm with Ritz transformation** is employed by default, whereas for smaller models the direct inversion of the stiffness matrix (which is stabler) can also be performed.

Jacobi algorithm with Ritz transformation

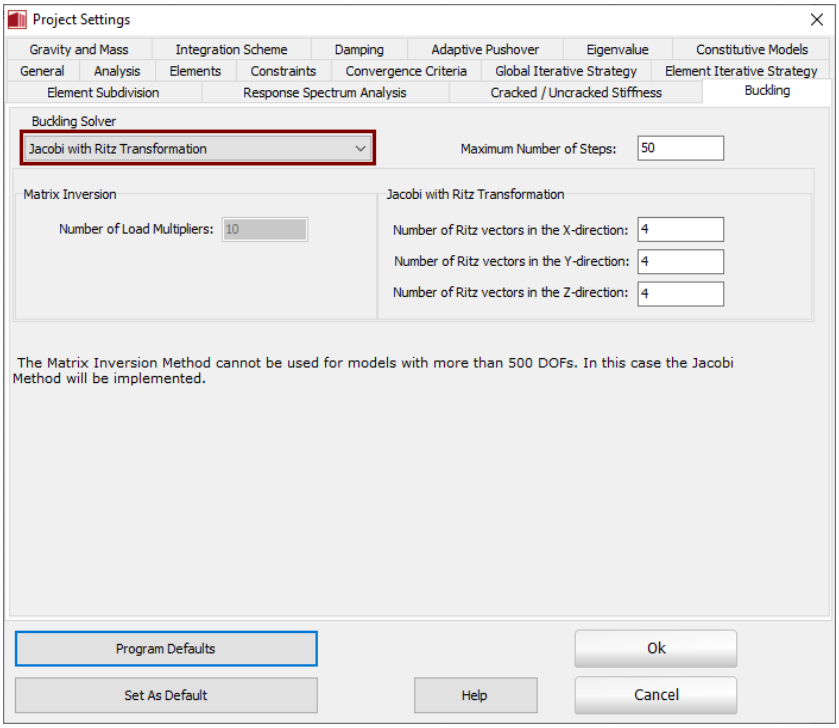
In Jacobi algorithm with Ritz transformation the user may specify:

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

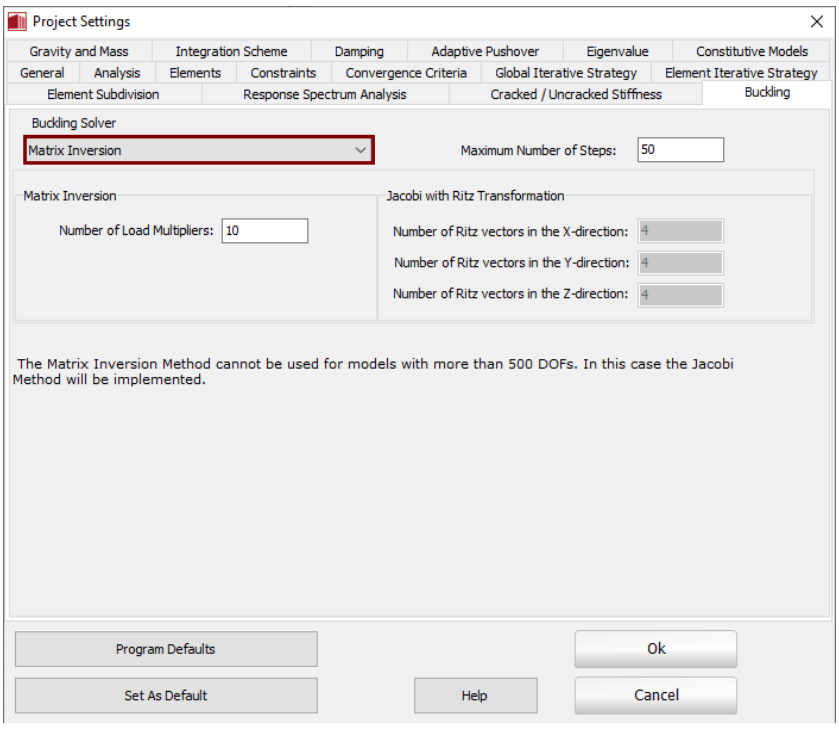
Matrix Inversion

The **Number of load multipliers** should be defined in matrix inversion. The maximum number of load multipliers required by the user. The default value is 10. Users might wish to increase this parameter when analysing very flexible buildings.

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



Buckling tab window - Jacobi with Ritz transformation



Buckling tab window - Matrix Inversion

Convergence Criteria

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

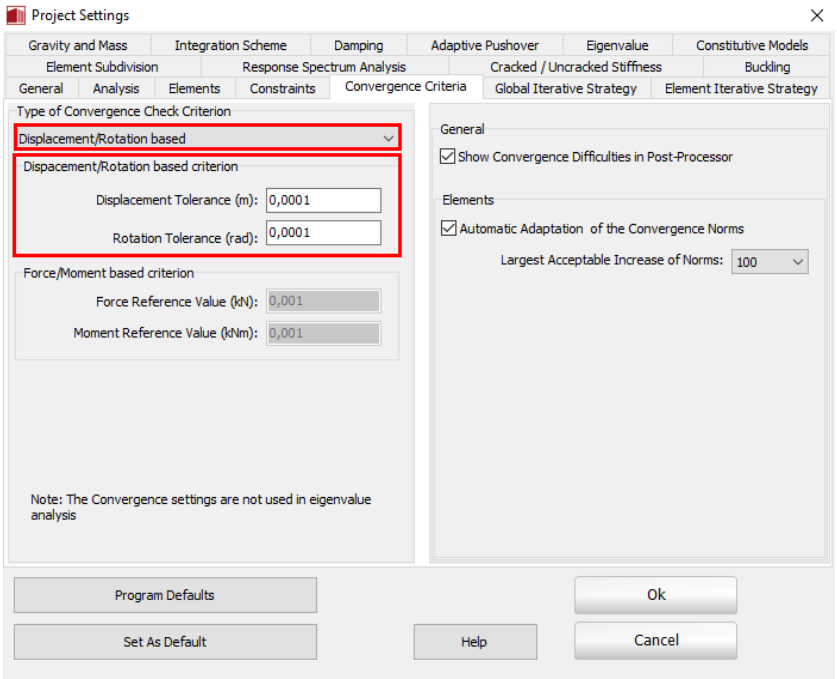
- Displacement/Rotation based

- 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 in SeismoStruct 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, drastic change in loading patterns and 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 SeismoStruct, with a displacement tolerance of 0.1 mm and a rotation tolerance of 1e-4 rad, which lead to precise and stable solutions in the majority of cases.

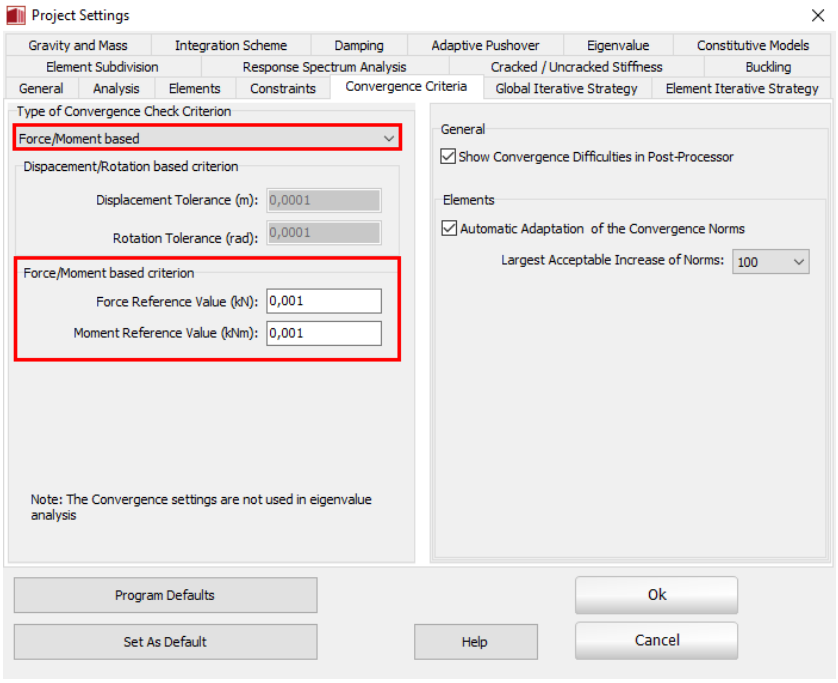


Convergence Criteria tab window – Displacement/Rotation based

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 an RC wall section is used, the stress-strain distribution

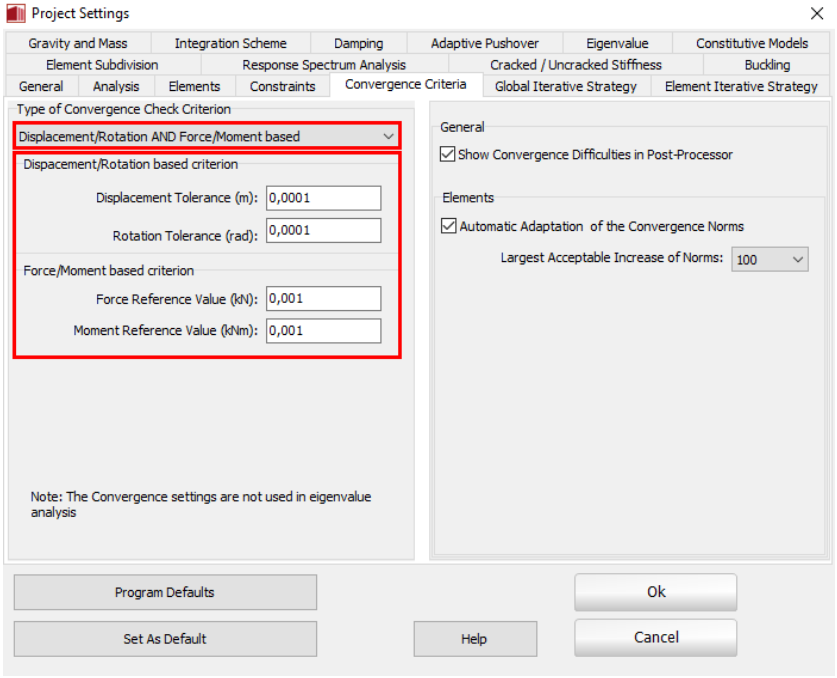
across the section may assume very complex patterns, by virtue of its large width, thus requiring a much higher number of iterations to be fully equilibrated. In such cases, if a force/moment convergence check is not enforced, the response of the structure will result very irregular, with unrealistically abrupt variations of force/moment quantities (e.g. wiggly force-displacement response curve in pushover analysis). As described in Appendix A, a non-dimensional global tolerance is employed in this case, with a default value of 1e-3.



Convergence Criteria tab window – Force/Moment based

Displacement/Rotation AND Force/Moment based

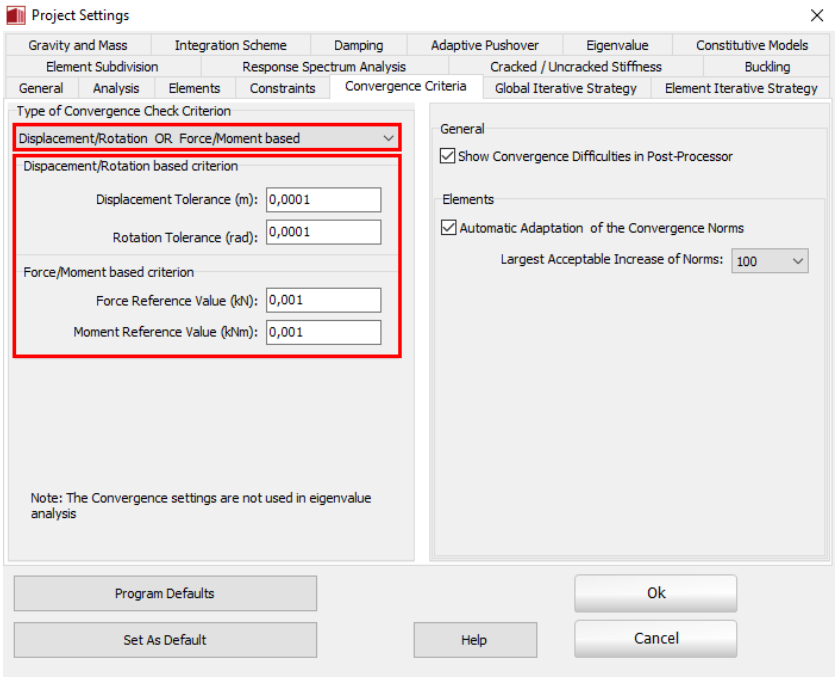
Taking into account the discussion made above, it results clear that maximum accuracy and solution control should be obtained when combining the displacement/rotation and force/moment convergence check criteria. This option, however, is not the default since the force/moment based criterion does, on occasions, create difficulties in models where infinitely stiff/rigid connections are modelled with link elements, as discussed in Appendix A. Still, it is undoubtedly the most stringent convergence and accuracy control criterion available in SeismoStruct, 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.

Elements

If the Automatic Adaptation of the Convergence Norms 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 steps. 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.

NOTE 1: Convergence difficulties in force-based elements are often caused by the employment of a large number of integration sections (e.g. default of 5) together with element discretisation (typically in beams, where the reinforcement details change). In such cases, users should decrease the number of integration sections to 3.

NOTE 2: As discussed in Appendix A, FB formulations can take due account of loads acting along the member, thus avoiding the need for distributed loads to be transformed into equivalent point forces/moments at the end nodes of the element, and for then lengthy stress-recovery to be carried out.

Global Iterative Strategy

In SeismoStruct, 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 problems). The workings and theoretical background of this solution algorithm is described in some detail within the Nonlinear Solution Procedure section in Appendix A, 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** is **40**, which should work well for most practical applications. 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. The same applies when link elements with very low or very high stiffness values are used in the modelling, since such situation often calls for a higher number of iterations to be carried out before structural equilibrium is achieved.

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 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 (typical of models where elements with very high stiffness values are used to model rigid links). A value around 75% of the maximum number of iterations within an increment usually provides a good starting point. The **default** in SeismoStruct is **32**.

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 A for details on this norm) with a pre-defined maximum tolerance (**default** 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 SeismoStruct 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 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

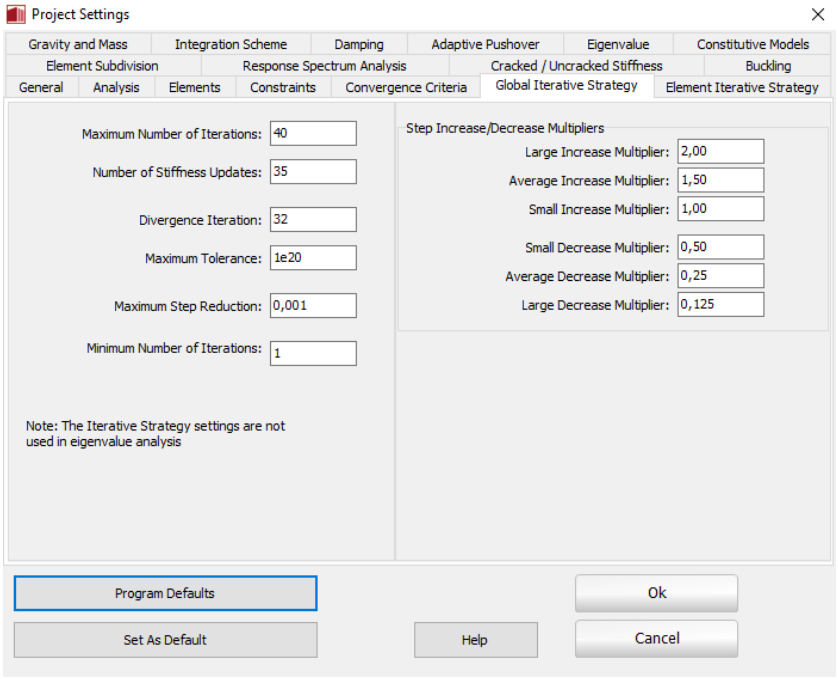
This parameter defines the minimum number of iterations to be performed within each load increment (analysis step). The **default value** is **1**. Through this parameter it is possible to achieve a better convergence when the displacement-based criterion is loose and the force-based very strict (this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

Step Increase/Decrease Multipliers

The automatic stepping algorithm in SeismoStruct 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 A), 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 A), 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 in SeismoStruct 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, drastic change in loading patterns and intensity, 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

In SeismoStruct, all analyses are treated as potentially nonlinear, and therefore an incremental iterative solution is needed.

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 relative 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.

Finally, an additional setting is provided for employing a smoother descending branch in the moment-rotation curve of the plastic hinge, in order to have a smoother and more stable numerically curve. This setting is applied only when users choose the automatic definition of the modelling parameters by the program, but not when they define the a, b and c hinge modelling parameters.

Masonry Element Type

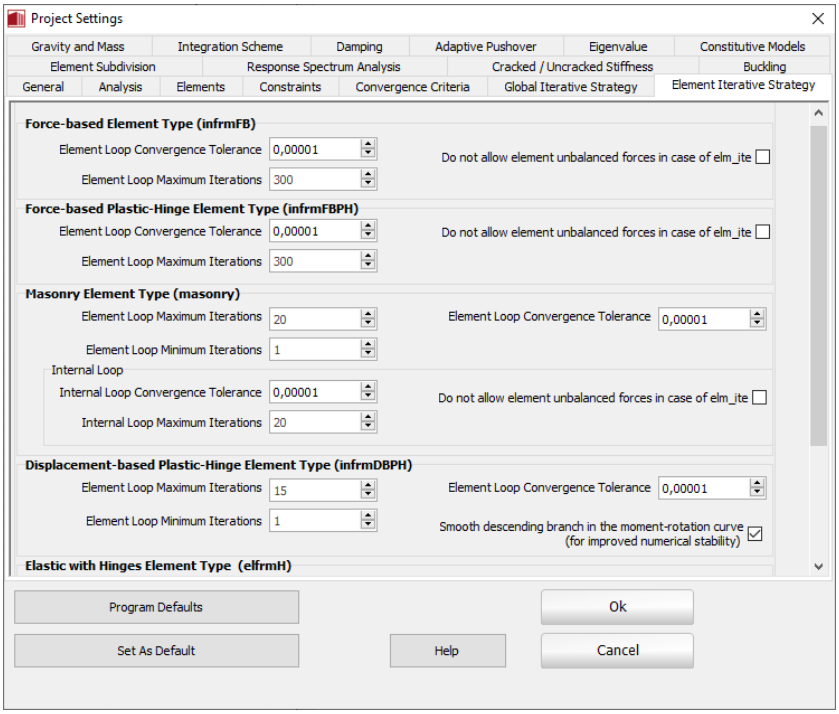
Since the element consists of a force-based element type employed in modelling mainly the bending behaviour of the masonry member (herein called the 'internal sub-element') with two links at the two edges that are employed to simulate the shear behaviour of the member (herein referred to as the 'external links' or the 'link sub-elements'), two internal iterative procedures are required, in order to achieve equilibrium on the element level: one for the internal force-based sub-element, and the second for the assemblage of the three sub-elements, links and frame.

As a result, parameters for both iterative procedures should be provided. The parameters for the internal force-based sub-element are the same with the typical force-based elements, and have the same default values. The parameters for the external loop of the entire element are the maximum and the minimum iterations 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 just a limited number of iterations.

Elastic with Plastic-hinges Element Type & Rack with Plastic-hinges Element Type

Since these elements consist 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

As indicated in the **Materials** module, users have the possibility of defining the materials specific weights, with which the distributed self-mass of the structure can then be calculated. Furthermore, in the **Element Classes** module, additional distributed mass may also be defined, which will serve to define any mass not associated to the self-weight of the structure (e.g. slab, finishings, infills, variable loading, etc). Lumped and distributed mass-only elements can also be defined and then added to the structure in the **Element Connectivity** module, so that users may model mass distributions that cannot be obtained using the aforementioned *Materials/Sections* facilities; e.g. water tank with concentrated mass on top. Finally, in the applied loads module, permanent distributed loads can be applied on the elements in every direction.

Here, it is possible for users to define if and how such mass is to be transformed into loads and which degrees of freedom are to be considered in a dynamic analysis, as well as, if and how mass is to be defined from loads.

Mass Settings

Three options are offered for defining mass in dynamic analysis, IDA and eigenvalue analysis: i) From the Frame Elements, based on the specific weight of their materials and their section's additional mass, as well as the Mass Elements (lmass and dmass), ii) From Loads, point and distributed (the mass is

applied in the gravity direction ONLY, and its value is based on the g value), and iii) From both options (i) and (ii) above, i.e. from both Frame/Mass Elements and Loads. The first option is set by default.

Further, when running dynamic 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). It is also noted that these settings take precedence over the 'mass directions' defined in the lumped/distributed mass elements, that is, if a given distributed mass element should define mass only in the x direction, for instance, but all dofs were to be selected in the Global Mass Directions settings, then even if such element mass contribution to the global Mass matrix of the structure would indeed be considered only in the x direction, the dynamic analysis will nonetheless consider all dofs as active.

NOTE: Analyses of large models featuring distributed mass/loading are inevitably longer than those where lumped masses, and corresponding point loads, are employed to model, in a more simplified fashion, the mass/weight of the structure. If users are not interested in obtaining information on the local stress state of structural elements (e.g. beam moment distribution), but are rather focused only on estimating the overall response of the structure (e.g. roof displacement and base shear), then the employment of a faster lumped mass/force modelling approach may prove to be a better option, with respect to its distributed counterpart.

Gravity Settings

In SeismoStruct loads may be defined in two ways: (i) explicitly in the Applied Loads module, and (ii) indirectly from the transformation of the masses of the structural model to loads.

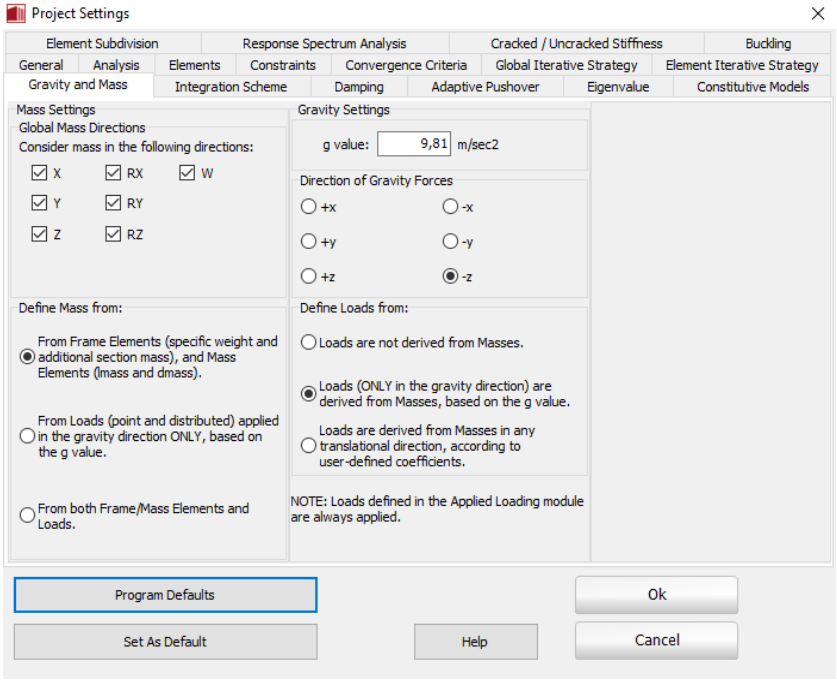
There are three available options for defining Loads from masses: i) Loads are not derived from masses. ii) Loads are derived from masses, based on the g value, but ONLY in the gravity direction, which is the default option, and iii) Loads are derived from masses in any translational direction, according to user-defined coefficients.

NOTE 1: Loads defined in the Applied Loads module are always applied to the structural model, irrespective of the employed option for the masses-to-loads transformations.

NOTE 2: The mass-derived loads are internally transformed into equivalent nodal forces/moments, with the exception of elastic and inelastic frame elements, in which mass-derived loads are distributed along the element.

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

NOTE 3: Stress-recovery (*Project 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 mass, but not through the introduction of dmass elements).



Gravity & Mass tab window

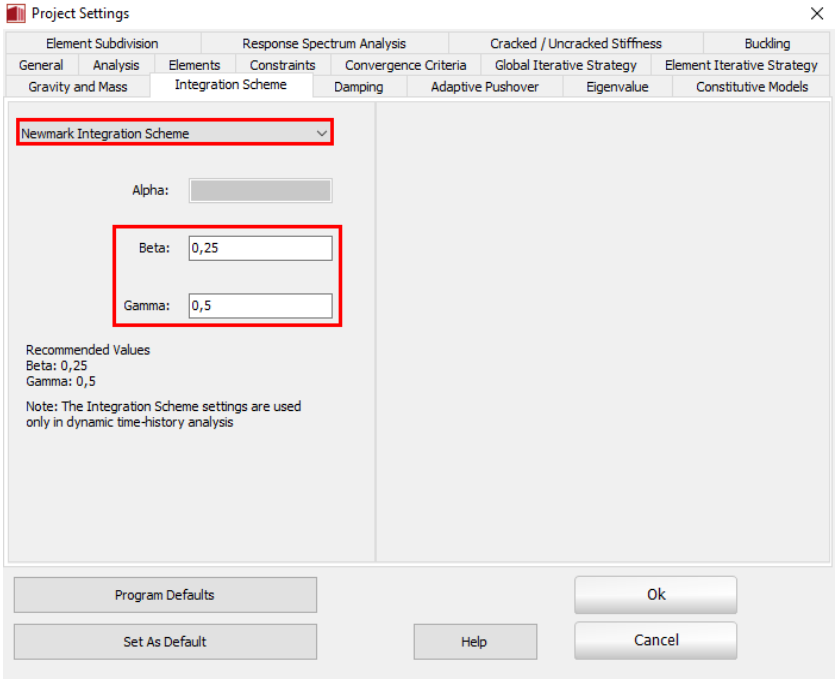
Integration Scheme

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 SeismoStruct, 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

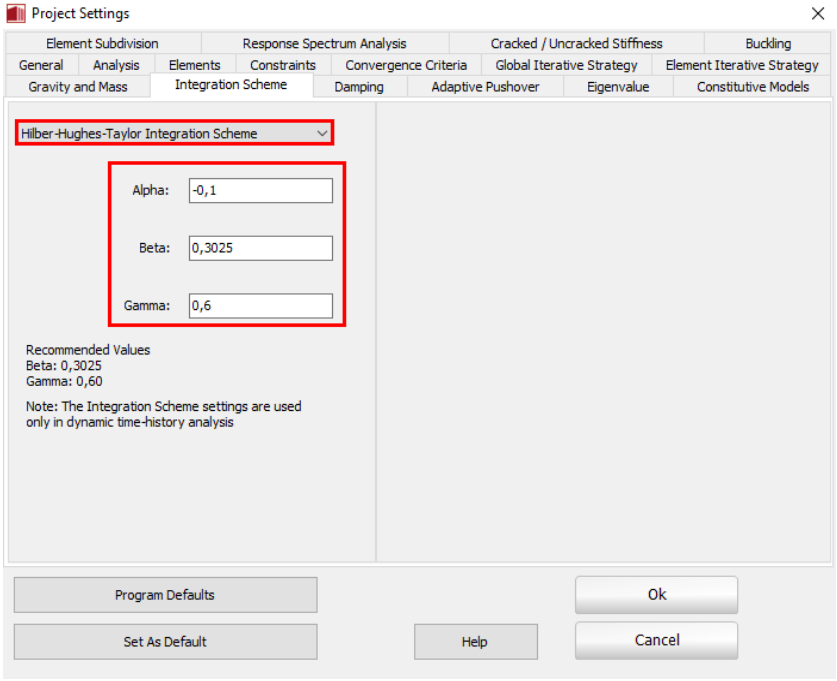
The **Newmark integration scheme** requires the definition of two parameters: beta (β) and gamma (γ). Unconditional stability, independent of time-step used, can be obtained for values of $\beta \geq 0.25(\gamma+0.5)^2$. In addition, if $\gamma=0.5$ is adopted, the integration scheme reduces to the well-known 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 SeismoStruct, 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].

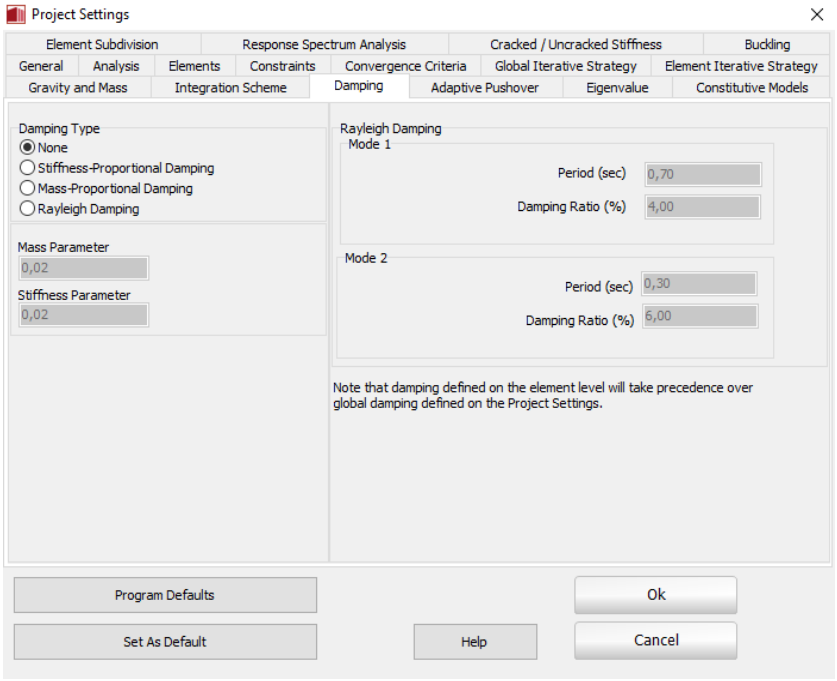
Some disagreement exists amongst the scientific/engineering community with regards to the use of equivalent viscous damping to represent energy dissipation sources that are not explicitly included in the model. Indeed, some authors [e.g. Wilson, 2001] strongly suggest for such equivalent modelling to be avoided altogether, whilst others [Priestley and Grant, 2005; Hall, 2006] advice its employment but not by means of Rayleigh damping, which is proportional to both mass and stiffness, but rather through the use of stiffness-proportional damping only; as discussed by Pegon [1996], Wilson [2001], Abbasi et al. [2004] and Hall [2006], amongst others, if a given structure is "insensitive" to rigid body motion, mass-proportional damping will generate spurious (i.e. unrealistic) energy dissipation. The stiffness-proportional damping modelling approach may then be further subdivided in initial stiffness-

proportional damping and tangent stiffness-proportional damping, the latter having been shown by Priestley and Grant [2005] as the possibly soundest option for common structures.

Nonetheless, even if one would be able to include all sources of energy dissipation within a given finite elements model (and this is definitely always the best option, i.e. to explicitly model infills, dampers, SSI, etc), the introduction of even a very small quantity of equivalent viscous damping might turn out to be very beneficial in terms of the numerical stability of highly inelastic dynamic analyses, given that the viscous damping matrix will have a "stabilising" effect in the system of equations. As such, its use is generally recommended, albeit with small values.

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**.



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):

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

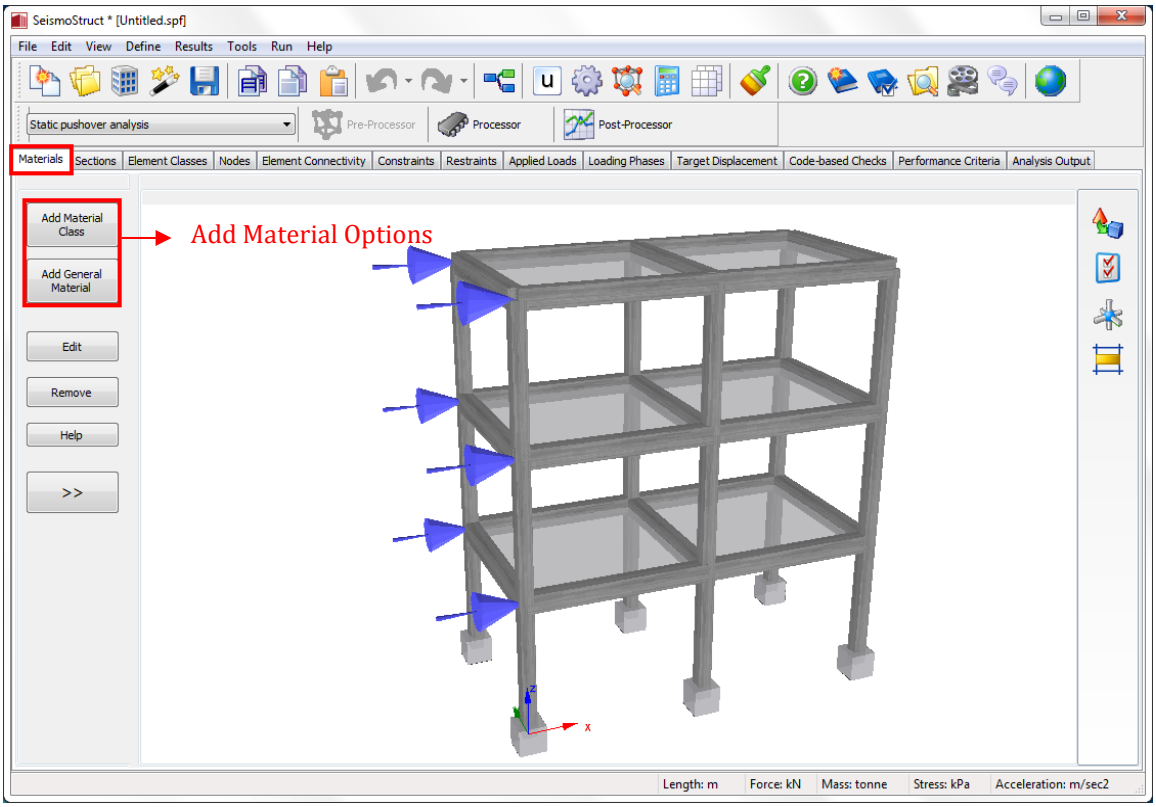
NOTE 1: A relatively large variety of different types of matricial damping exists and is used in different FE codes. These variations may present advantages with respect to traditional Rayleigh damping; e.g. reducing the level of damping that is introduced in higher modes and so on. However, we believe 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.

MATERIALS

Materials that are to be available within a SeismoStruct project come defined in the **Materials** module, where (i) the name (used to identify the material within the project), (ii) the type (listed below), (iii) the mechanical properties (i.e. strength, modulus of elasticity, strain-hardening, etc.) and (iv) the parameters needed for the Code-based Checks (eg. existing or new material) of each particular material can be defined.

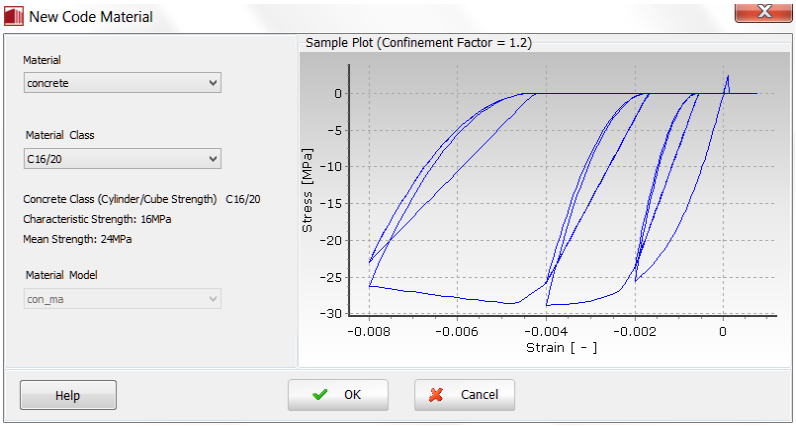


Materials module

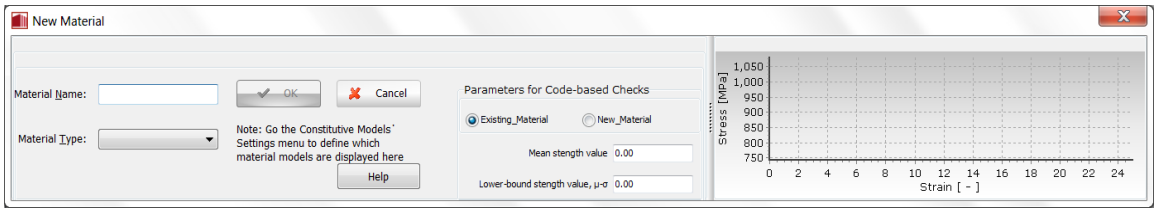
IMPORTANT: Only the material types that have been previously activated in the Constitutive Model tab window (*Tools > Project Settings > Constitutive Model*) will appear in the Materials module.

As anticipated in Tutorial N.1, two options are available for inserting a new material:

1. *Add Material Class*;
2. *Add General Material*.



Materials module – Add Material Class option



Materials module – Add General Material option

Currently, twenty material types are available in SeismoStruct. By default, all the material types may be selected without any changes in the *Project Settings* panel. The complete list of materials is proposed hereafter:

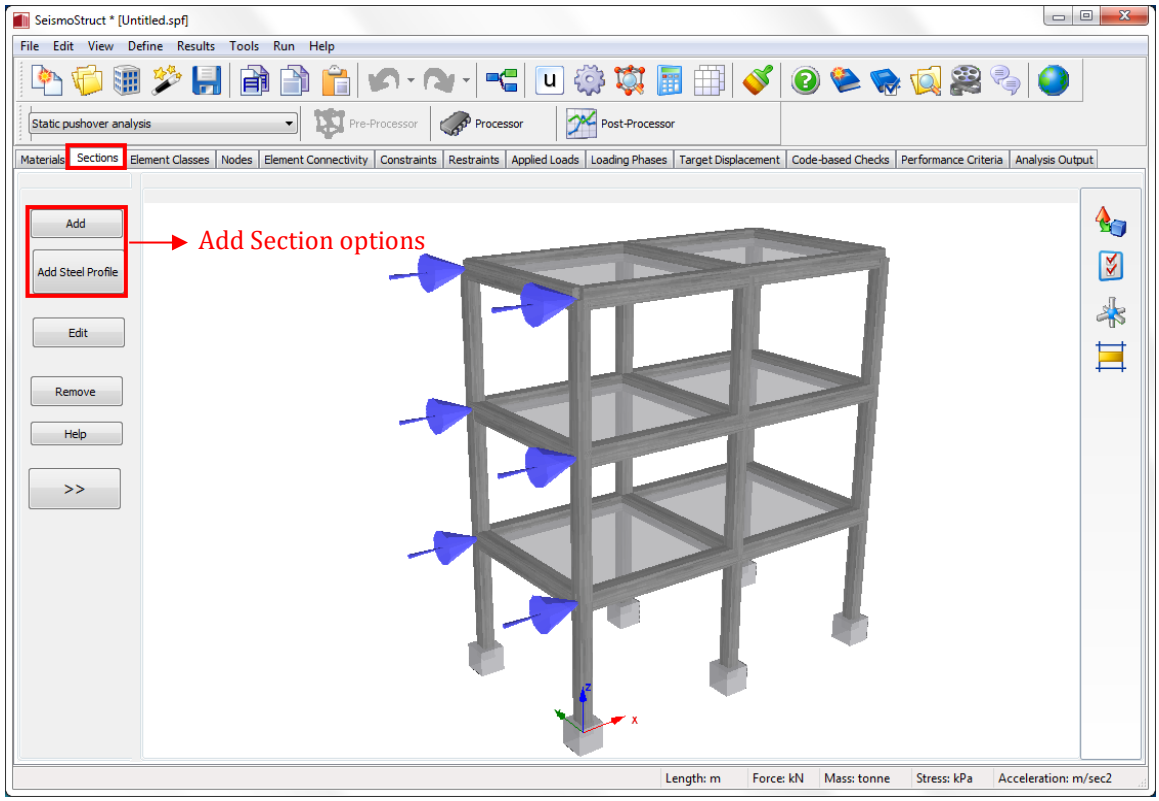
- Bilinear steel model - stl_bl
- Menegotto-Pinto steel model - stl_mp
- Bilinear steel model with isotropic strain hardening- stl_bl2
- Giuffre-Menegotto-Pinto Model with Isotropic Hardening – stl_gmp
- Ramberg-Osgood steel model - stl_ro
- Dodd-Restrepo steel model – stl_dr
- Monti-Nuti steel model - stl_mn
- Buckling Restrained steel brace model – stl_brb
- 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
- Trilinear masonry model - mas_tl
- Parabolic masonry model - mas_par
- Superelastic shape-memory alloys model - se_sma
- Trilinear FRP model - frp_tl
- Elastic material model - el_mat
- Generic Hysteretic material – hyst_mat

NOTE: In SeismoStruct, the Poisson coefficient is assumed as equal to 0.2 for concrete and 0.3 for steel.

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

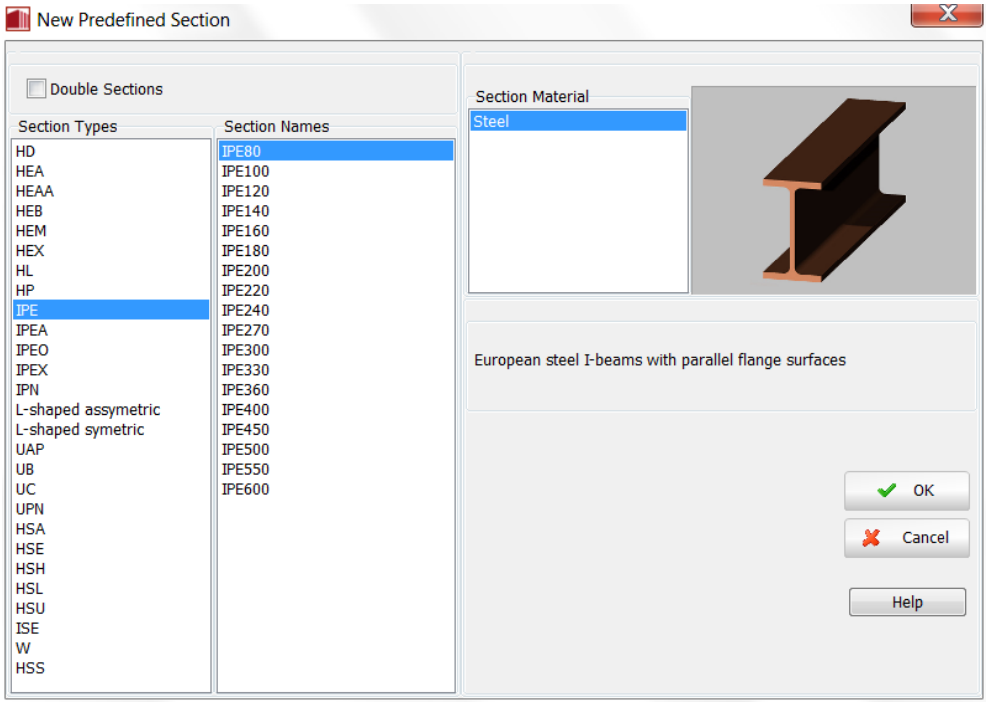
SECTIONS

Cross-sections that are to be available within a SeismoStruct project come defined in the **Sections** module, where (i) the name (used to identify the section within the project), (ii) the type (listed below), (iii) materials (as defined in the Materials module), (iv) dimensions (length, width, etc.) and (v) reinforcement (if supported) can be explicitly defined.



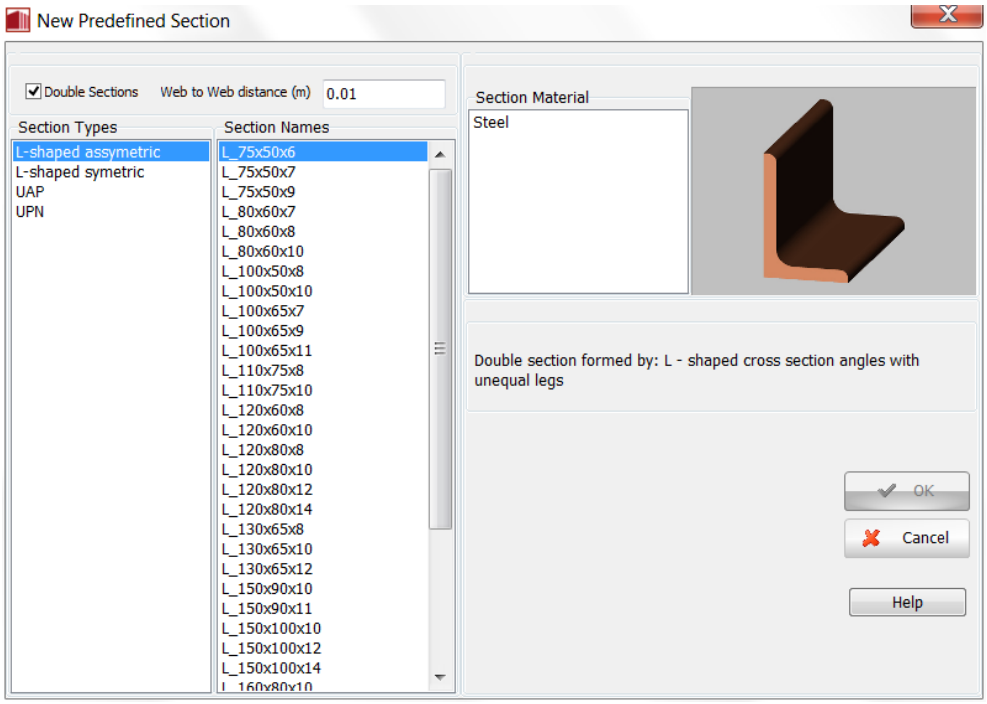
Sections module

SeismoStruct allows also selecting **predefined steel sections** by clicking on the *Add Steel Profile* button. A database of the most common steel sections (e.g. HEA, HEB, IPE, etc.) is available, as well as W and HSS sections which have been introduced with the release of v7.0 of SeismoStruct.



Sections module – Add Steel Profile option

From SeismoStruct v7.0 onwards, it is possible to introduce double steel sections by checking the corresponding checkbox at the New Predefined Section dialog box:



Sections module – Add Steel Profile option

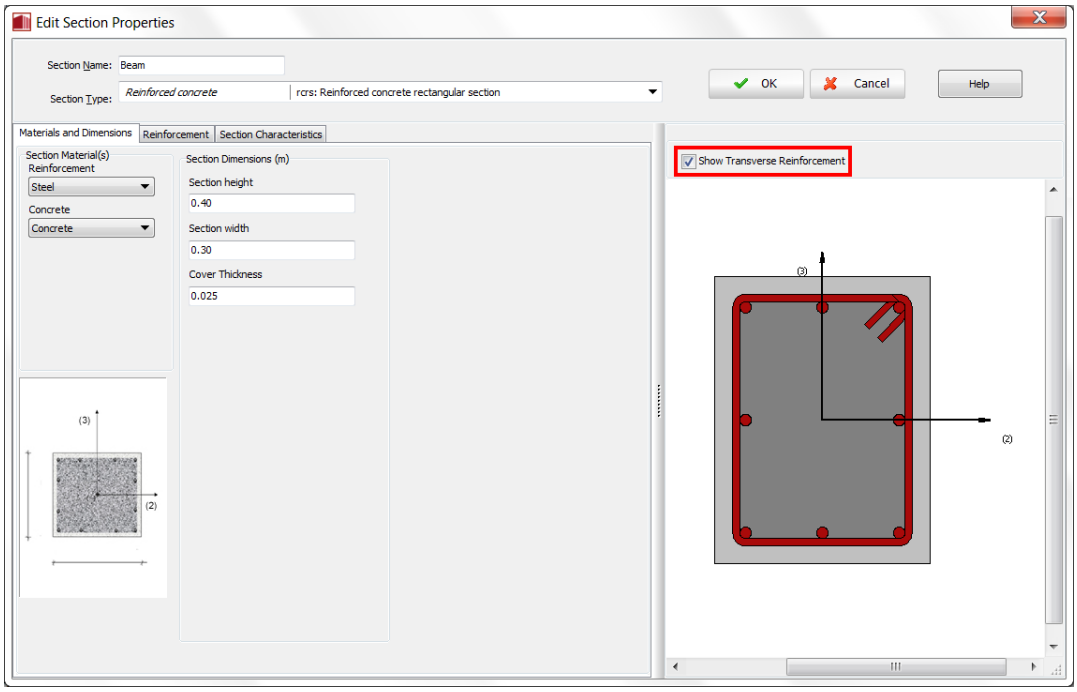
Currently, seventy two section types are available in SeismoStruct. These range from simple single-material solid sections to more complex reinforced concrete and composite sections.

- Rectangular solid section - rss

- Rectangular hollow section - rhs
- Circular solid section - css
- Circular hollow section - chs
- Symmetric I-or T- section - sits
- Asymmetric general-shape section - agss
- Double angle or channel shaped section – dacss
- Double I type1 section – di1
- Double I type2 section – di2
- Double I type1 section with top and bottom plates– di1tbp
- Double I type2 section with top and bottom plates – di2tbp
- Double I type1 section with web plates – di1wp
- Double I type2 section with web plates– di2wp
- Double I type1 section with top, bottom and web plates – di1tbwp
- Double I type2 section with top, bottom and web plates – di2tbwp
- Built up box double channel section – bbdc
- Built up box double channel section with connecting plate - bbdccp
- Built up box double channel section with top and bottom plates - bbdctbp
- Built up box double angle section – bbda
- Built up box double angle section with connecting plate – bbdacp
- I section with top and bottom plates – itbp
- I section with top, bottom and web plates – itbwp
- I section with top plate – itp
- I section with bottom plate – ibp
- I section reinforced with bottom I section – ibri
- I section reinforced with bottom T section – ibrt
- Star section composed from angle sections – sfa
- Double angle back-to-back section – dabtb
- Built up box formed by four angle sections – bbfa
- Double angle section placed along the diagonal – dadg
- Cruciform Section - cfs
- Reinforced concrete rectangular section - rcrs
- Reinforced concrete quadrilateral section - rcqs
- Reinforced concrete rectangular with rounded corners section - rcrrcs
- Reinforced concrete circular section - rccs
- Reinforced concrete Z-shaped column section – rczcs
- Reinforced concrete L-shaped column section – rclcs
- Reinforced concrete T-shaped column section – rctcs
- Reinforced concrete T-section - rcts
- Reinforced concrete asymmetric rectangular section - rcars
- Reinforced concrete rectangular wall section – rcrws
- Reinforced concrete rectangular no pseudo-columns wall section - rcbws
- Reinforced concrete U-shaped wall section - rcuws
- Reinforced concrete Z-shaped wall section - rczws
- Reinforced concrete L-shaped wall section - rclws
- Reinforced concrete rectangular hollow section - rcrhs
- Reinforced concrete rectangular with rounded corners hollow section - rcrrchs
- Reinforced concrete circular hollow section - rcchs
- Reinforced concrete box-girder section - rcbgs
- Reinforced concrete jacketed rectangular section – rcjrs
- Reinforced concrete jacketed rectangular with rounded corners section – rcjrrcs
- Reinforced concrete 3-side jacketed rectangular section – rcjrs3
- Reinforced concrete 2-side jacketed rectangular section – rcjrs2
- Reinforced concrete 1-side jacketed rectangular section – rcjrs1

- Reinforced concrete jacketed circular section – rcjcs
- Reinforced concrete jacketed Z-shaped column section – rcjzcs
- Reinforced concrete jacketed L-shaped column section – rcjlcs
- Reinforced concrete 3-side jacketed L-shaped column section – rcjlcs3
- Reinforced concrete jacketed T-shaped column section – rcjtcs
- Reinforced concrete 3-side jacketed T-shaped column section – rcjtcs3
- Reinforced concrete jacketed T-section – rcjts
- Reinforced concrete 3-side jacketed T-section – rcjts3
- Reinforced concrete 1-side jacketed T-section – rcjts1
- Reinforced concrete jacketed asymmetric rectangular section - rcjars
- Reinforced concrete 1-side jacketed asymmetric rectangular section – rcjars1
- Composite I-section - cpis
- Partially encased composite I-section - pecs
- Fully encased composite I-section - fecs
- Composite rectangular section - crs
- Composite circular section – ccs
- Masonry wall section – mws
- Masonry spandrel section - mss

It is noted that from program version 2016 onwards users are able to visualise the section's shear reinforcement.



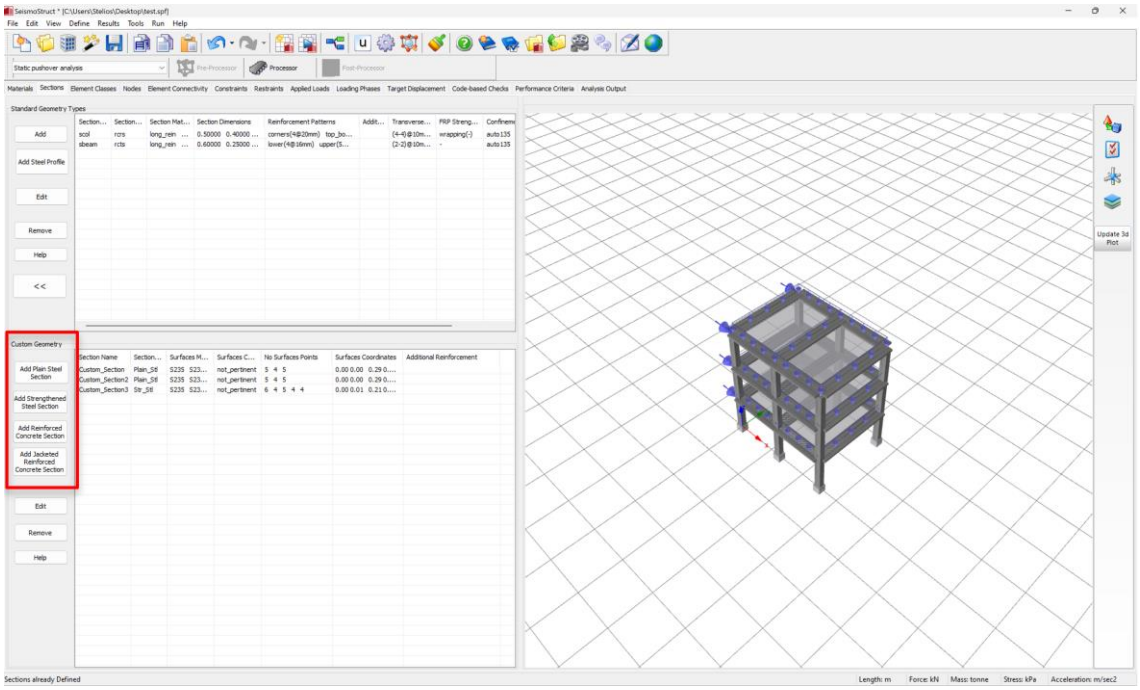
Sections Properties module – Show Transverse Reinforcement

By making use of these section types, the user is able to create up an unlimited number of different cross-sections, which will then be used to define the different element classes of a structural model.

For a comprehensive description of the section types, refer to Appendix D - Sections.

CUSTOM SECTIONS

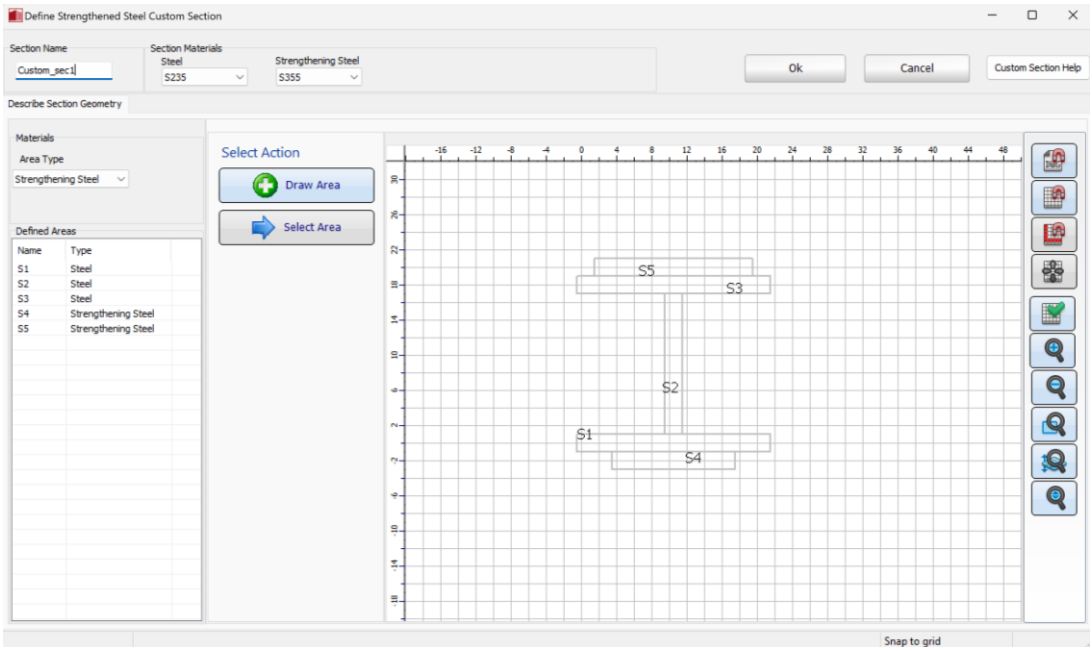
Apart from the standard sections of the SeismoStruct library, custom steel or reinforced concrete sections of any shape can also be defined. Both plain and strengthened variations of the sections are supported.



Custom Sections module

Users can define up to 9 section areas. The areas are specified graphically by their perimeter after clicking on the Draw Area button. The defined areas can be selected and then deleted with the corresponding buttons. The functions and commands at the right side of the screen (snapto existing areas, snap to grid, ortho, pan, zoom in, zoom out, zoom to window and zoom all) can further facilitate the graphical input.

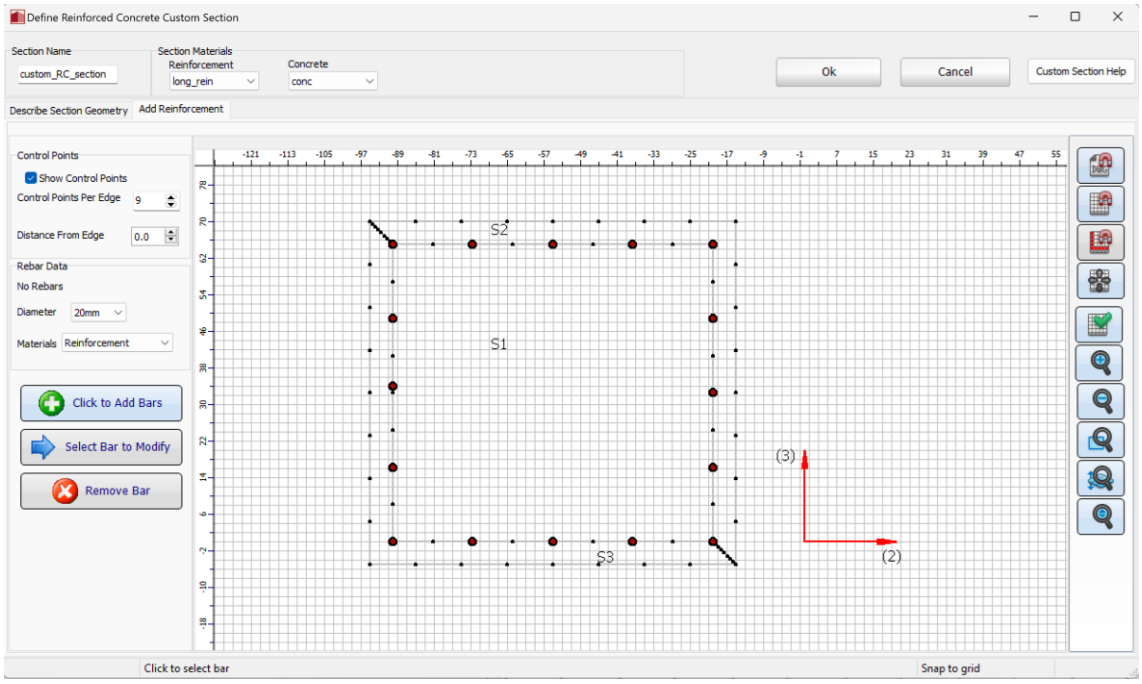
For the plain steel sections one material should be specified, whereas for the strengthened steel sections two materials are required, one for the existing part of the section and one for the strengthening. Users are asked for (i) the name of the section, (ii) the employed materials, and (iii) the type of the different areas of the section (plain or strengthened).



Custom Sections module – Add Strengthened Steel Section option

For the plain reinforced concrete sections two materials should be specified, for the concrete and the reinforcement. For the strengthened (jacketed) sections instead 4 materials are needed, for the concrete and reinforcement of the existing section and for the concrete and reinforcement of the jacket. Different confinement factors may be assigned to the different concrete areas. It is noted that the existing/internal part of the section can be offset, in order to quickly create the cover and the jacket (in the case of strengthened sections), instead of adding manually these areas.

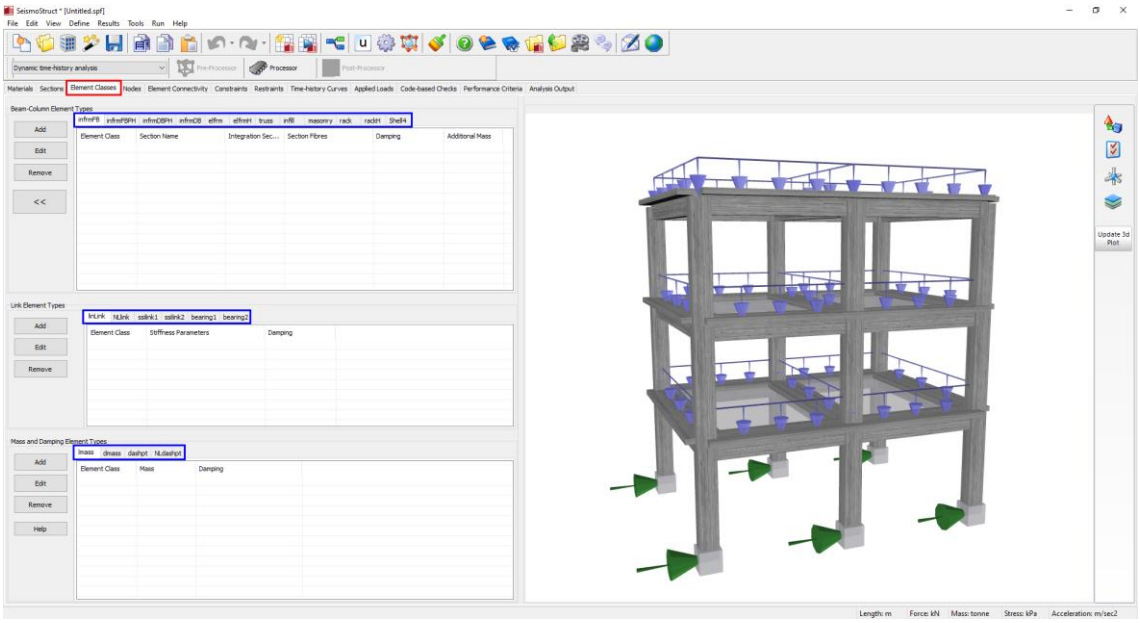
The section reinforcement is specified in the corresponding tab with the help of several control points internally or in the perimeter of the introduced areas. Again the input is done graphically with clicks on the control points.



Custom Sections module – Add Reinforced Concrete Section option

ELEMENT CLASSES

Elements that are to be available within a SeismoStruct project come defined in the **Element Classes** module. Element types are used to define element classes exactly in the same manner that material types were used to define materials or section types were employed to define sections. Hence, just as for the case of materials and sections, in a SeismoStruct project there may exist any given number of different element classes belonging to the same element type (e.g. to model two different columns the user needs to define two different element classes, both appertaining to the same element type - frame elements). The element classes defined in this module are then employed in the **Element Connectivity** module to create the actual elements that form-up the structural model being built.



Element Classes module

Currently, twenty-two element types, divided in three categories (*Beam-column element types*, *Link element types* and *Mass and Damping element types*), are available in SeismoStruct.

- Inelastic frame elements - infrmDB, infrmFB
- Inelastic plastic-hinge frame element – infrmFBPH, infrmDBPH
- Elastic frame element – elfrm
- Elastic frame element with hinges - elfrmH
- Inelastic infill panel element - infill
- Inelastic truss element – truss
- Inelastic masonry frame element – masonry
- Shell element - shell4
- Rack element - rack
- Rack element with hinges - rackH
- Linear link element type - linlink
- Nonlinear link element – NLlink
- Shallow footingsn macro-element – ssilink1
- Pile Foundation macro-element – ssilink2
- Elastomeric Isolator Element (Bouc Wen) – isolator1
- Friction Pendulum Isolator/System – isolator2
- Mass elements - lmass & dmass
- Damping element - dashpt
- Nonlinear damping element - NLdashpt

By making use of these element types, the user is able to create an unlimited number of different elements classes that are not only able to accurately represent intact/repaired structural members (columns, beams, walls, beam-column joints, etc.) and non-structural components (infill panels, energy dissipating devices, inertia masses, etc.) but also allow the modelling of different boundary conditions, such as flexible foundations, seismic isolation, structural gapping/pounding and so on.

NOTE 1: Some element types (e.g. mass and damping elements) cannot be used in certain analysis types (e.g. static analysis) and thus may not always be available in the *Element Classes* module.

NOTE 2: Users may find interesting information/suggestions about the modelling of structural and non-structural components in the NEHRP Seismic Design Technical Brief No. 4 (refer to the Bibliography).

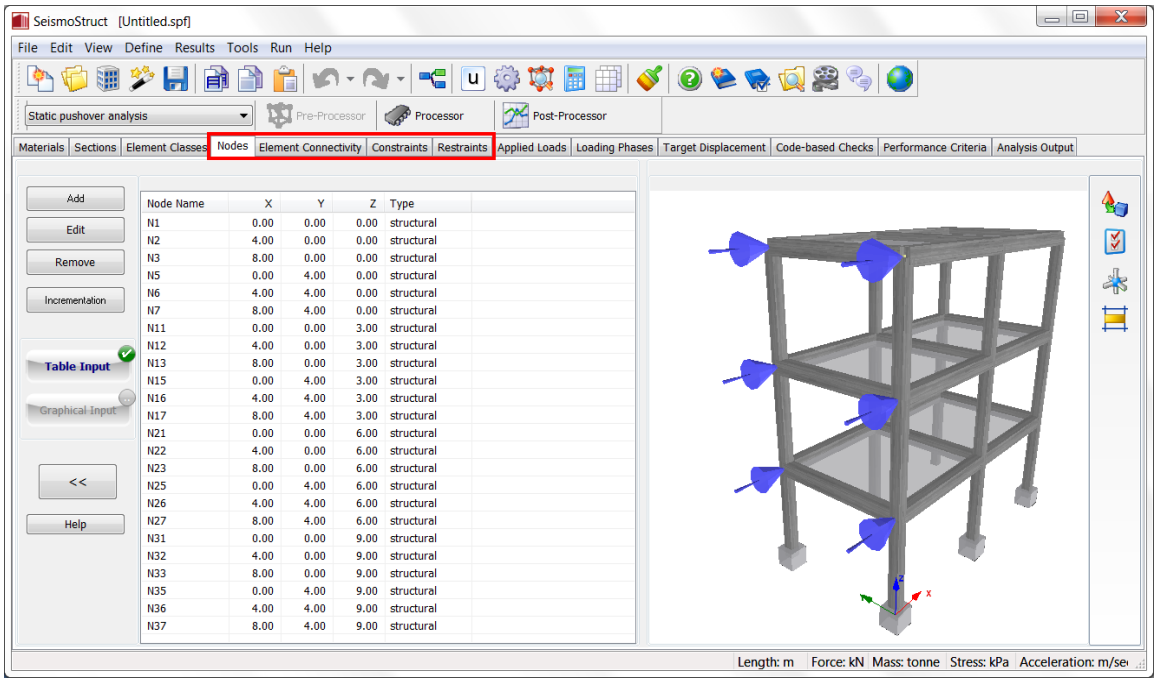
For a comprehensive description of the element types, refer to Appendix E - Element Classes.

STRUCTURAL GEOMETRY

Defining the geometry of the structure being modelled is a ***four-step procedure***. Firstly, all *structural* and *non-structural nodes* are defined, after which *element connectivity* can be stipulated. The process is then concluded with the assignment of structural *restraints*, which fully characterize the structure's boundary conditions. In addition to this, "optional" *Constraints* can be defined.

So, the **structural geometry** is defined through the following modules, which will be described below:

- Nodes
- Element Connectivity
- Constraints
- Restraints



Structural geometry modules

NOTE: An upper bound value of 50000 is set as the maximum number of nodes or elements that can be defined in a SeismoStruct model.

Nodes

Two types of nodes are available in SeismoStruct: **structural** and **non-structural**.

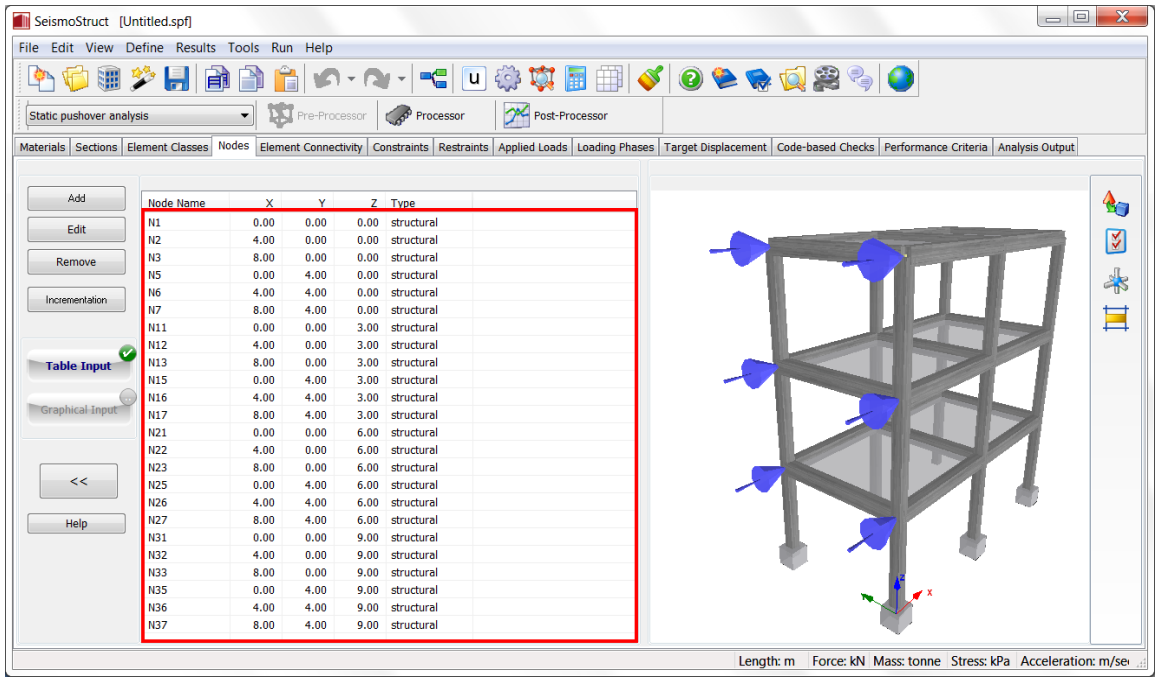
Structural nodes

Are all those nodes to which an element, of whichever type, is attached to. In fact, in SeismoStruct it is not possible to run an analysis of any type if a node that has been defined as "structural" does not feature at least one element connected to it. Put in other words, structural nodes are all those to which degrees-of-freedom are assigned and then included in the assemblage of stiffness matrix and load/displacement vectors.

Non-structural nodes

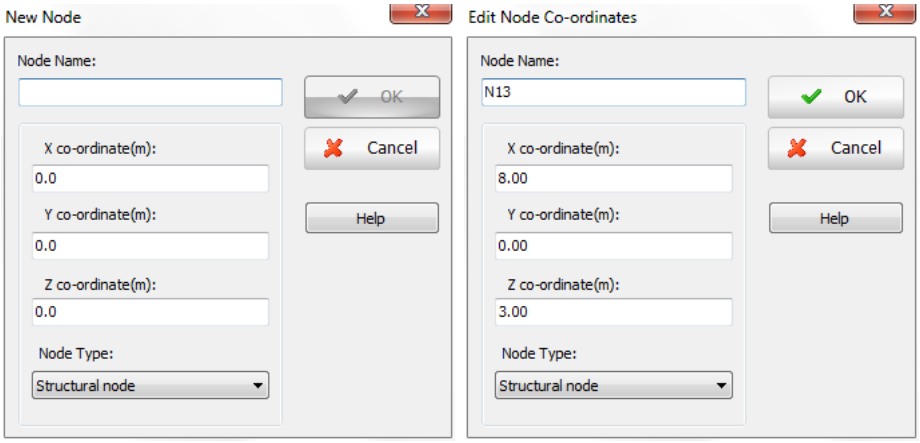
Are nodes that are not to be considered in the solution of the structure but are instead usually needed to define the orientation of local axes of certain types of elements (as described in element connectivity). No elements of any type can be attached to this type of nodes and whilst it is obvious that structural nodes can also be used as a reference point in the definition of these local axes, it usually results much more simple and clear to reserve this role to their non-structural counterparts. The user is referred to the global and local axes systems chapter for a deeper discussion on this subject. By default, non-structural nodes do not result visible on the 3D plot of the model, a condition that can be easily modified through a change in the display settings.

NOTE: When users define non-structural nodes with very large coordinates and then activate visualisation of such nodes, the model will inevitably be zoomed-out to a very small viewing size. To avoid such a scenario, users should (i) bring the non-structural nodes closer to the structure, (ii) disable visualisation of the latter or (iii) zoom-in manually every time the 3D plot is refreshed.



Nodes module

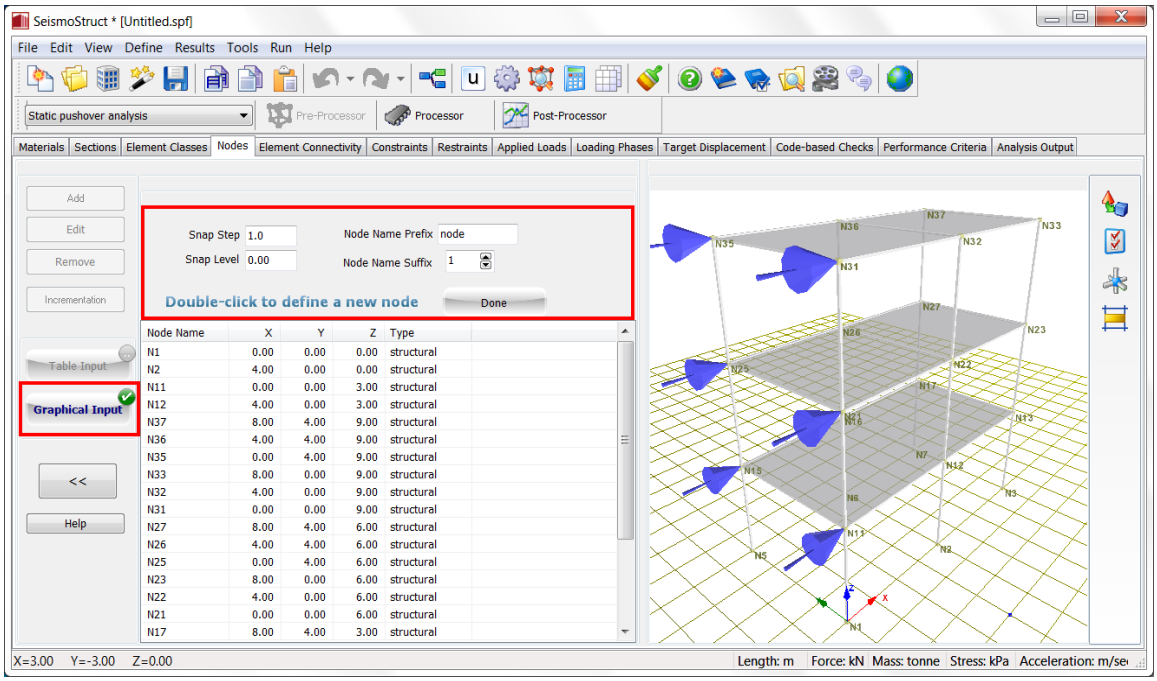
As in all other modules, the user is capable of adding new nodes (also through the *Graphical Input* button) and removing/editing existing selected ones.



Adding/Editing nodes

In the *Graphical Input* mode, the user has to:

1. Select the Snap Level (0 by default);
2. Eventually change the Snap step (1 by default), the Node Name Prefix and Suffix ("node" and "1", respectively by default);
3. Double-click on the grid in order to define the node;
4. Repeat the previous operation until all the nodes have been generated;
5. At the end of the procedure, click *Done* to return to the *Table Input*.



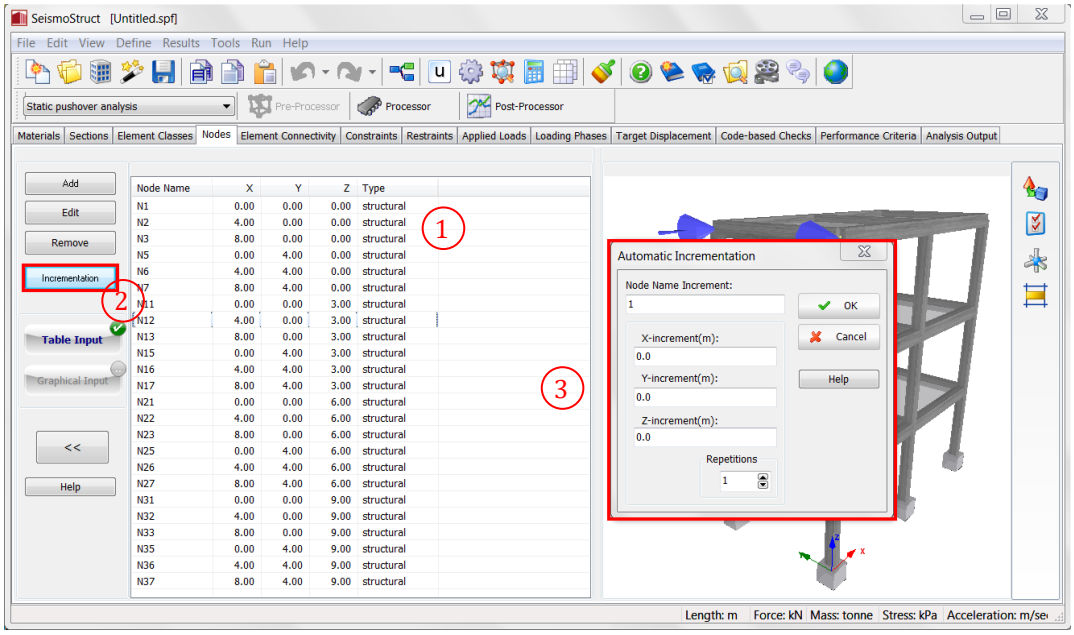
Adding Nodes (Graphical Input facility)

NOTE: An editing feature that might come very useful to users is the ability to change a co-ordinate type of a large number of nodes through a single operation, by making a multiple selection and opening the *Edit* dialog box. This can be very handy, for example, when one needs to change the y-coordinates of all nodes of a frame that is to be moved into a different position in space.

Nodes can be sorted according to their names or their x-, y- or z- coordinates. If the user clicks once on the header of the corresponding column, ascending sorting is adopted, whilst if a second click is employed, the nodes become sorted in descending fashion (see Editing functions for further details on data sorting).

The **Nodes** module features also an *Incrementation* facility with which the user can create new nodes through "repetition" of existing ones. This is done by:

1. Selecting a set of nodes that will serve as the base for the incrementation;
2. Clicking the *Incrementation* button;
3. Specifying the increment in the name and coordinates of the node(s) and finally deciding on the number of "Repetitions" to be carried out.

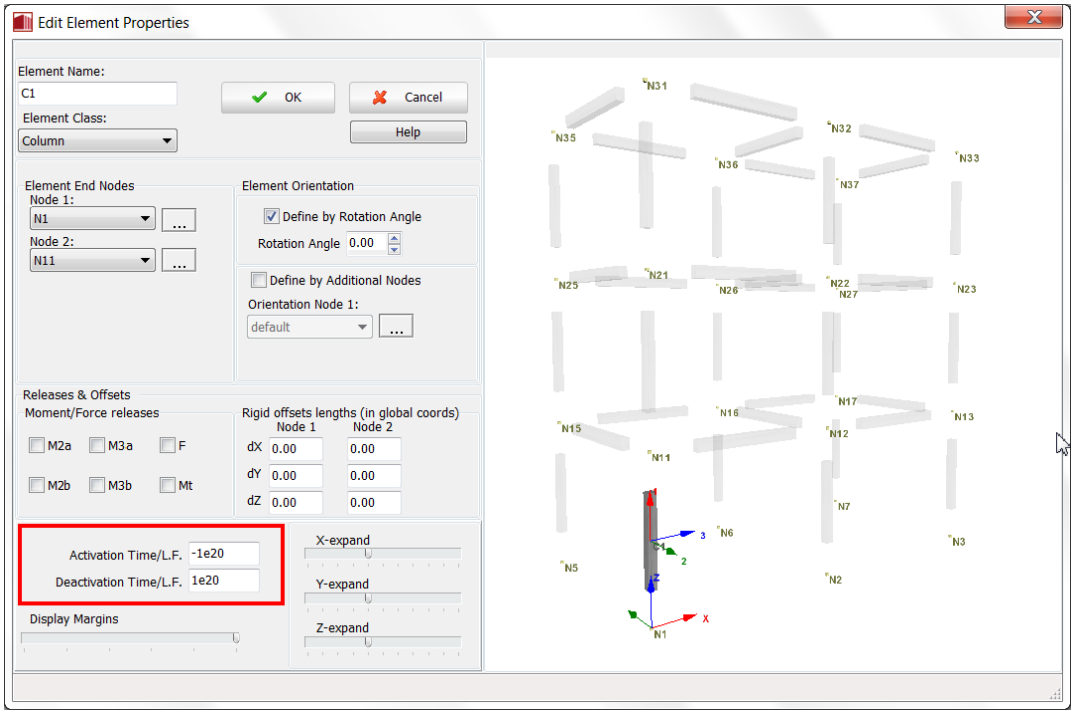


Incrementation facility – Nodes

Element Connectivity

The different elements of the structure are defined in the **Element Connectivity** module, where their name, element class, corresponding nodes, rigid offsets, force/moment releases and eventually activation time/L.F. are identified.

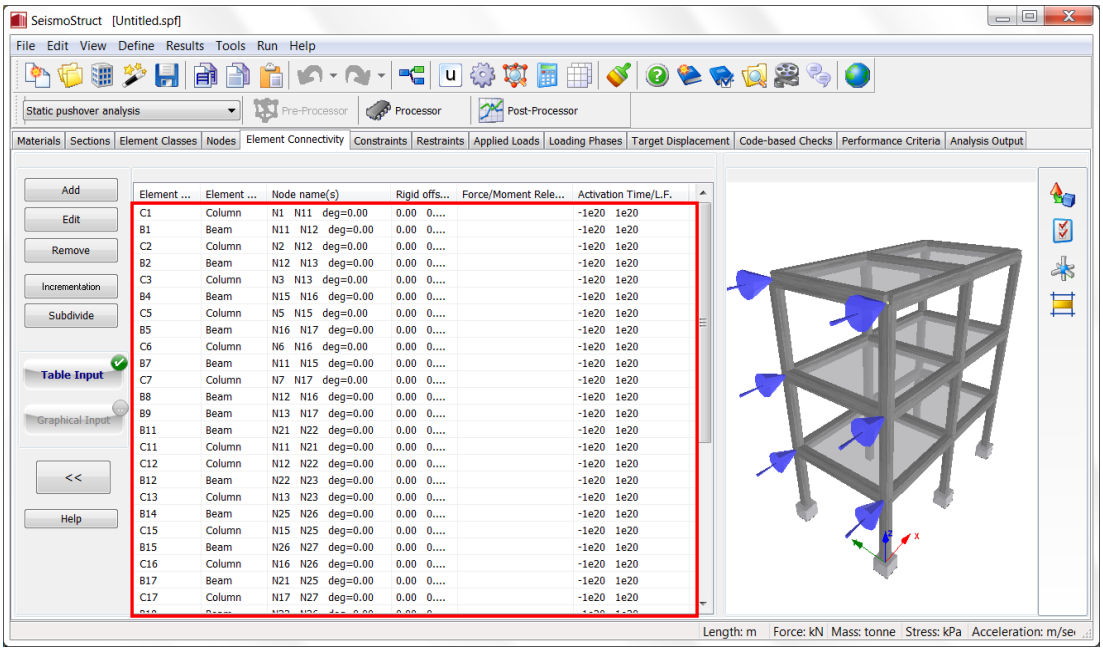
It is noted that the possibility of defining an activation (and deactivation) time/L.F. is provided within each element. The default values are -1e20 for activation (in order to cater for cyclic pushover analysis) and 1e20 for deactivation; this means that the element is activated at the beginning of any analysis and it will not be deactivated.



New Element Connectivity – Activation and deactivation Time/L.F.

As in all other modules, the user is capable of adding new elements (also through the *Graphical Input* button) and removing or editing existing selected elements (see Editing functions).

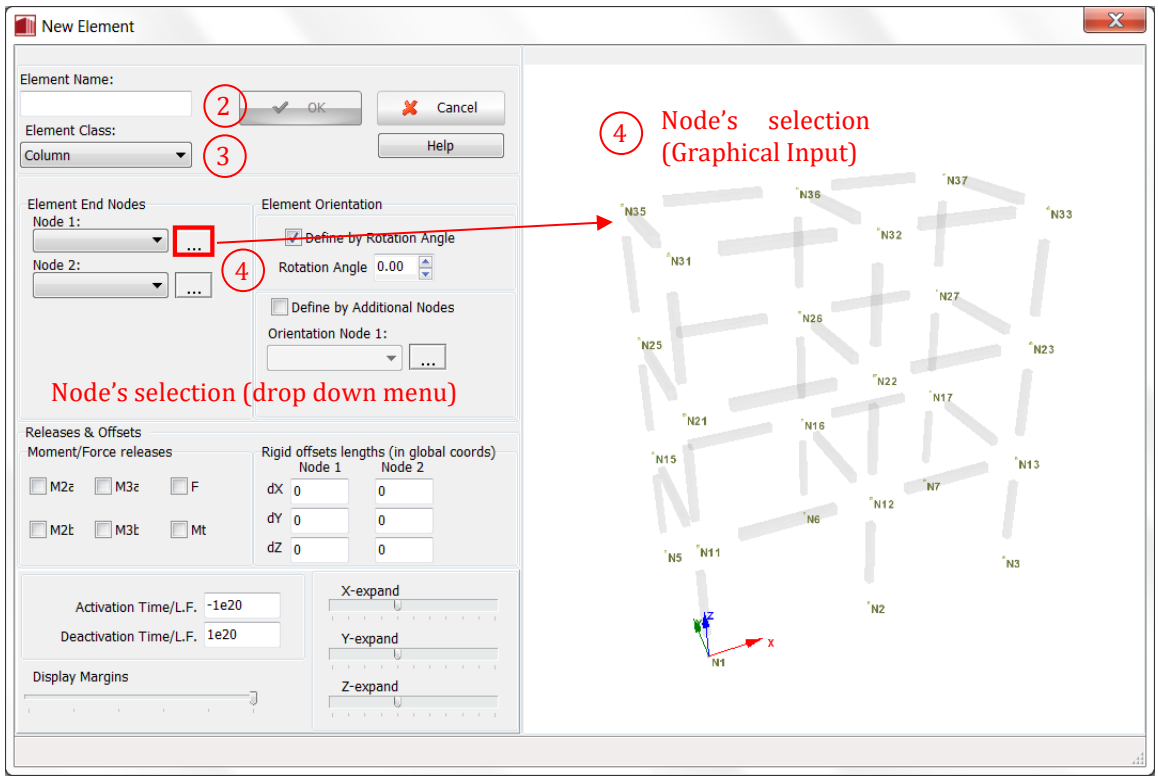
NOTE: Users can also change in a single operation, for instance, the non-structural node used in a large number of elements, again by taking advantage of the multiple selection and editing features.



Element Connectivity module

In order to add a new element in the *Table Input*, the user has to follow the steps listed below:

1. Click the *Add* button;
2. Assign a name;
3. Select the Element Class from the drop-down menu;
4. Select the corresponding nodes using the respective drop-down menus (or graphically);
5. Define the Element Orientation by Rotation Angle or by Additional nodes;
6. Select the Activation and Deactivation time/L.F.



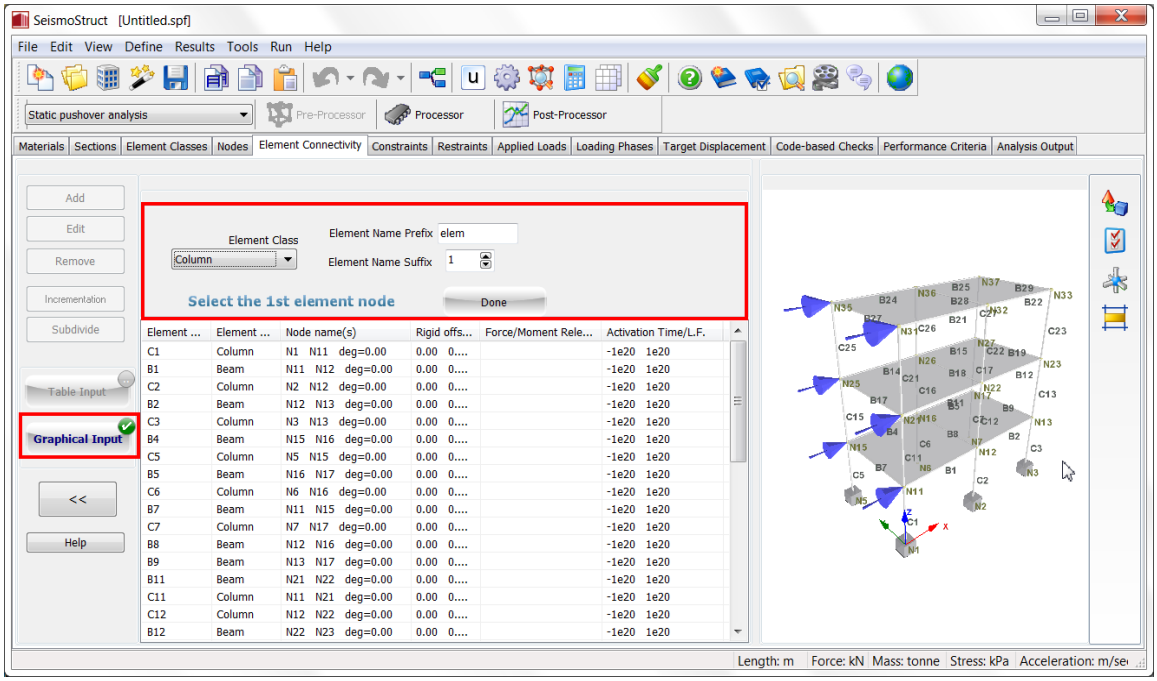
Adding New element (Table Input)

NOTE 1: The number of element nodes, which need to be selected, depends on the Element Class.

NOTE 2: Users may use the 'activation time' feature to exclude gravity loads from retrofitting elements (i.e. by activating retrofitting elements only after the first analysis step, which involves the application of gravity loads).

Otherwise, in order to graphically add a new element in the *Graphical Input* mode, the user has to:

1. Click the *Graphical Input* button;
2. Select the Element Class from the drop-down menu;
3. Double-click in the 'graphical space' to define all the element nodes.

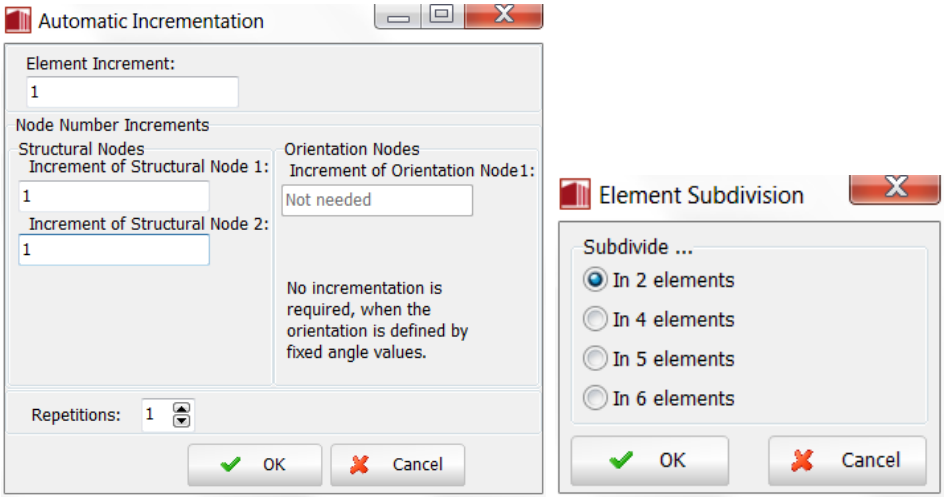


Adding New element (Graphical Input mode)

NOTE: The name of the new element is the concatenation of the element prefix and suffix.

In addition, however, *Incrementation* and *Subdivision* facilities are equally available. As in the case of nodes, *element incrementation* enables the automatic generation of new elements through "repetition" of existing ones. It functions in very much the same manner as the automatic generation of nodes, with the difference that instead of nodal coordinates, it is the names of element nodes that are incremented. This facility obviously requires that element names respect the number (e.g. 100) or word+number (e.g. elm20) formats.

Element subdivision, on the other hand, serves the purpose of providing the user with a tool for easy and fast subdivision of existing frame elements, so as to refine the mesh in localised areas (for instance to increase the accuracy of the analysis in zones of high inelasticity that have been detected only after running a first analysis with a coarser mesh). The creation of the new internal nodes, the generation of the new smaller elements and the updating of element connectivity is all carried out automatically by the program. Users can subdivide existing elements into 2, 4, 5 and 6 smaller components, the length of which is computed as a percentage of the original element's size, as defined in *Project Settings* > *Element Subdivision*.



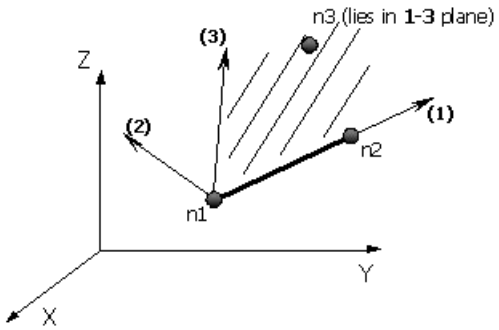
Element Incrementation and Element Subdivision

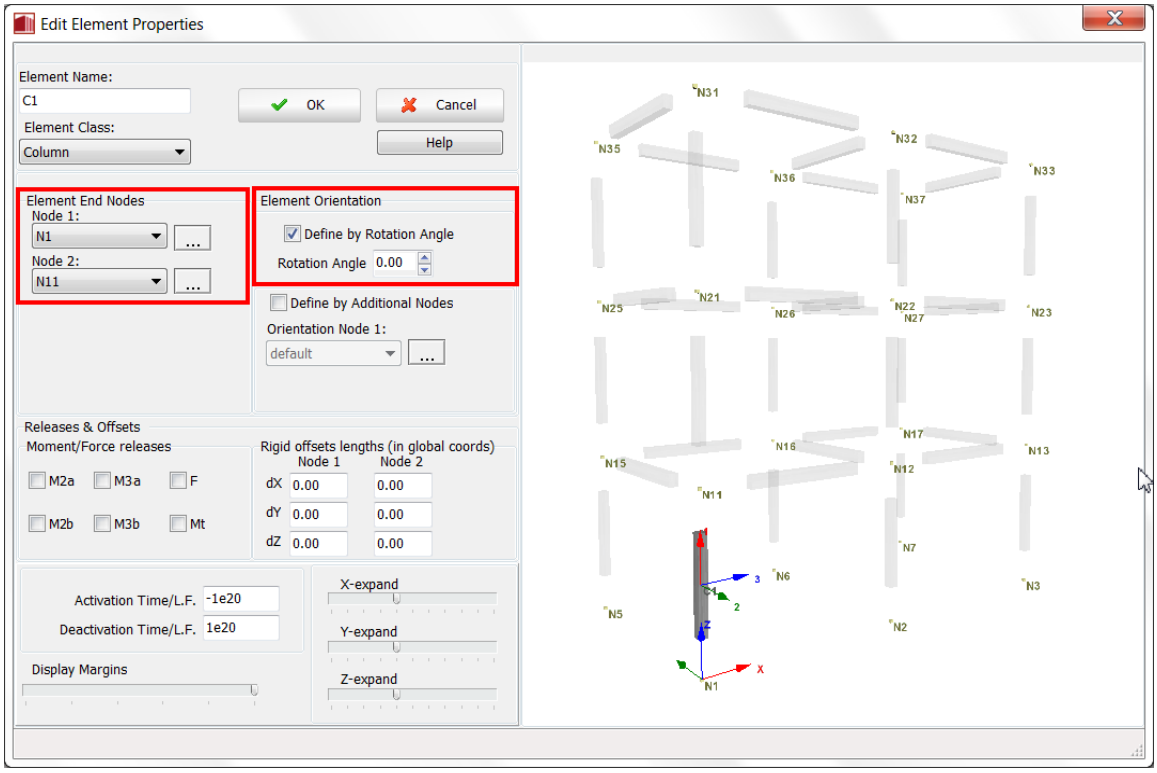
NOTE: Whilst a too course finite element mesh may lead to the impossibility of accurately reproducing certain response shapes/mechanisms, an exaggeratedly mesh refinement may lead to unnecessary long analyses and, in some instances, to less stable solutions. Hence, users are advised to make well balanced and judged decisions on the level of mesh refinement that they decide to introduce, ideally carrying out sensitivity studies in order to define the point of optimum balance between accuracy, numerical stability and analysis' run times.

In what follows, an overview of connectivity requirements for each of the **element types** available in SeismoStruct is given.

Elastic and Inelastic frame elements - infrmFB, infrmDB, infrmFBPH, infrmDBPH, elfrm & elfrmH

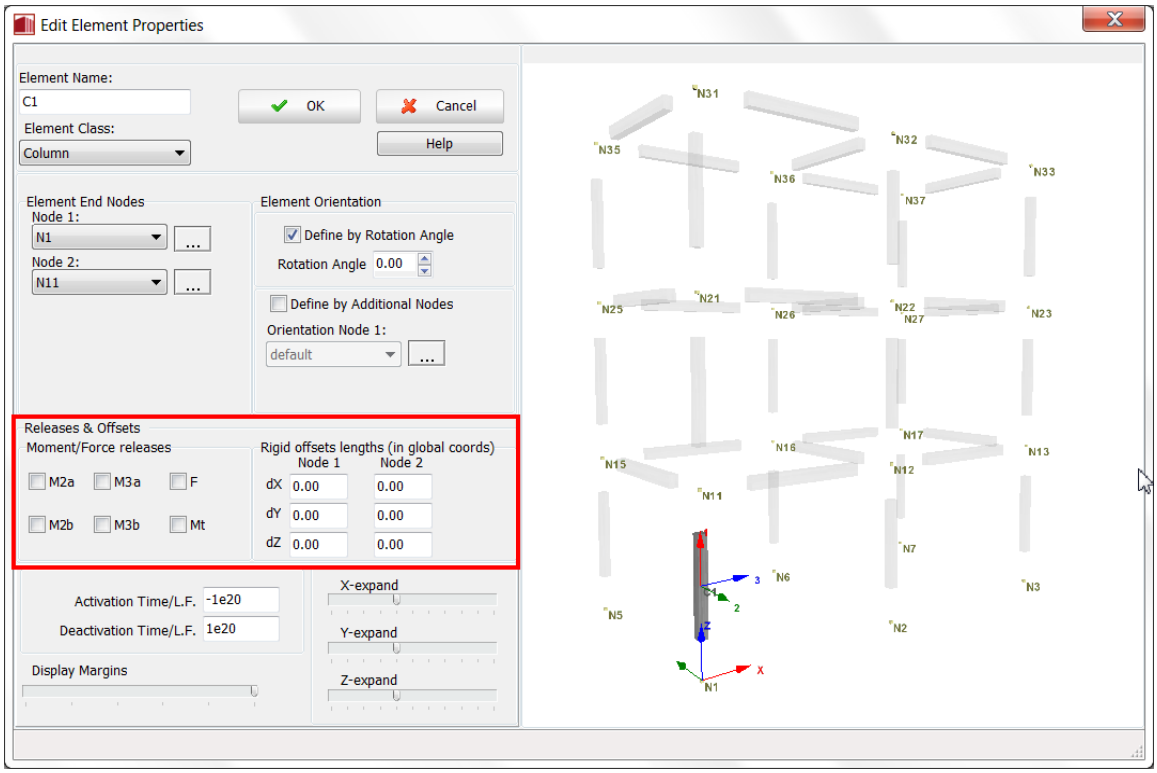
Two nodes need to be defined for these element types, representing the end-nodes of the element, thus defining its length, position in space and direction (local axis 1). A rotation angle or a third node is required so as to define the orientation of the element's cross section (local axes 2 and 3), as described in Global and local axes system.





Edit Element

In addition, for each frame element it is possible to specify *Rigid offsets lengths* (in global coordinates) by assigning a value for dX, dY and dZ to Nodes 1 and 2, respectively. Furthermore, users may also 'release' one or more of the element degrees of freedom (forces or moments) from the joints.



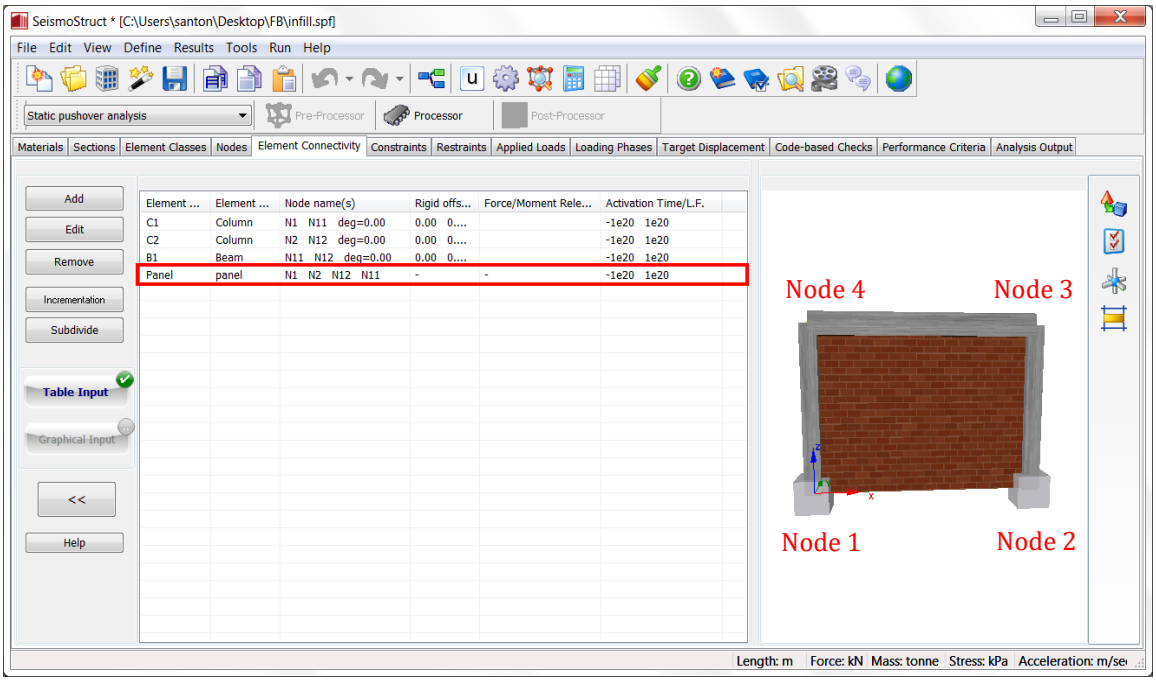
Rigid offsets lengths and Moment/Force releases

NOTE: Moment/force releases are always specified in the element local coordinate system.

Infill panel element - infill

Four nodes need to be defined for this element type. These correspond to the corners of the infill panel, must be entered in anti-clockwise sequence starting from the lower-left-hand corner and must all belong to the same plane.

NOTE: The internal struts 1, 2 and 5 of the panel will then be those connecting its first and third nodes, whilst internal struts 3, 4 and 6 will be made to connect the second and fourth panel corners.



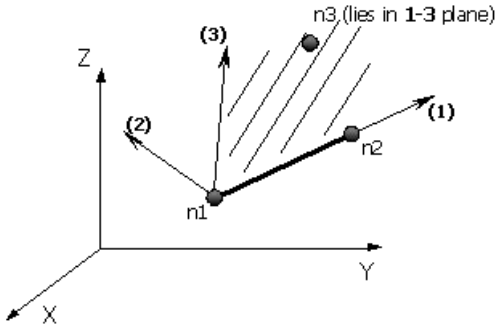
Element Connectivity module – Infill panel element

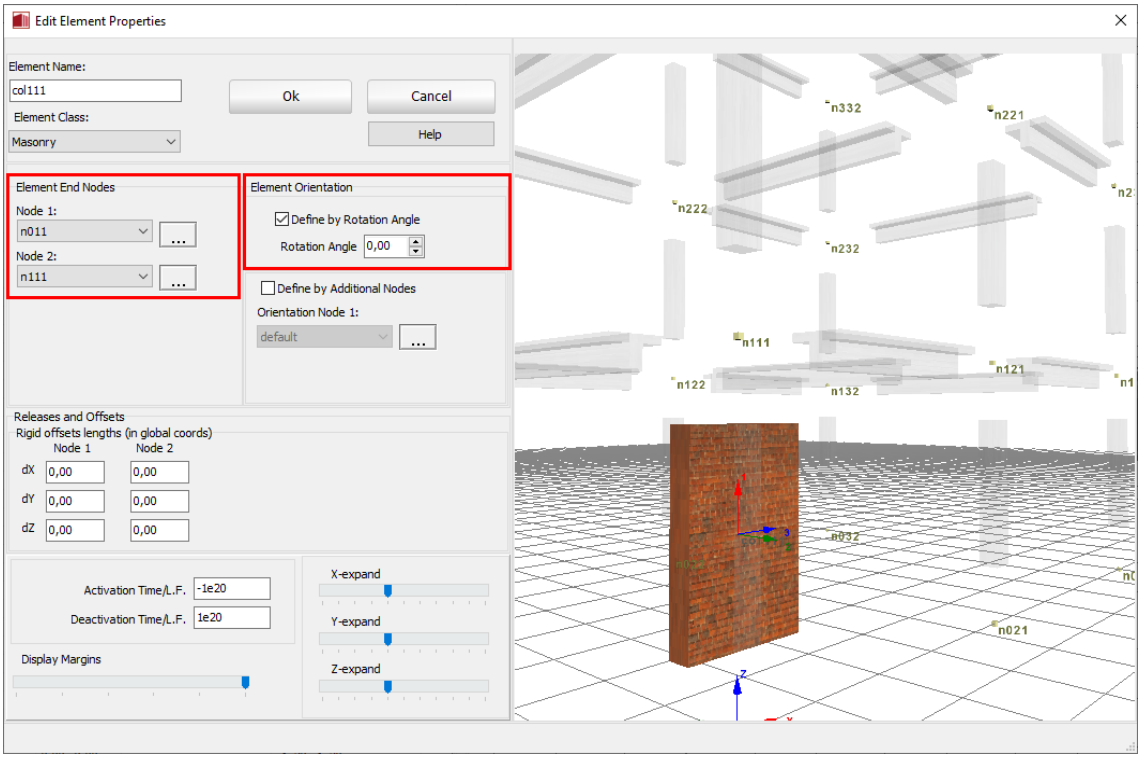
Inelastic truss element - truss

Two nodes need to be defined for this element type, usually corresponding to the extremities of structural members (i.e. one truss element per each structural member), unless there is a need to model element instability, in which case two or more truss elements (including an initial imperfection) per member should be employed.

Masonry element - masonry

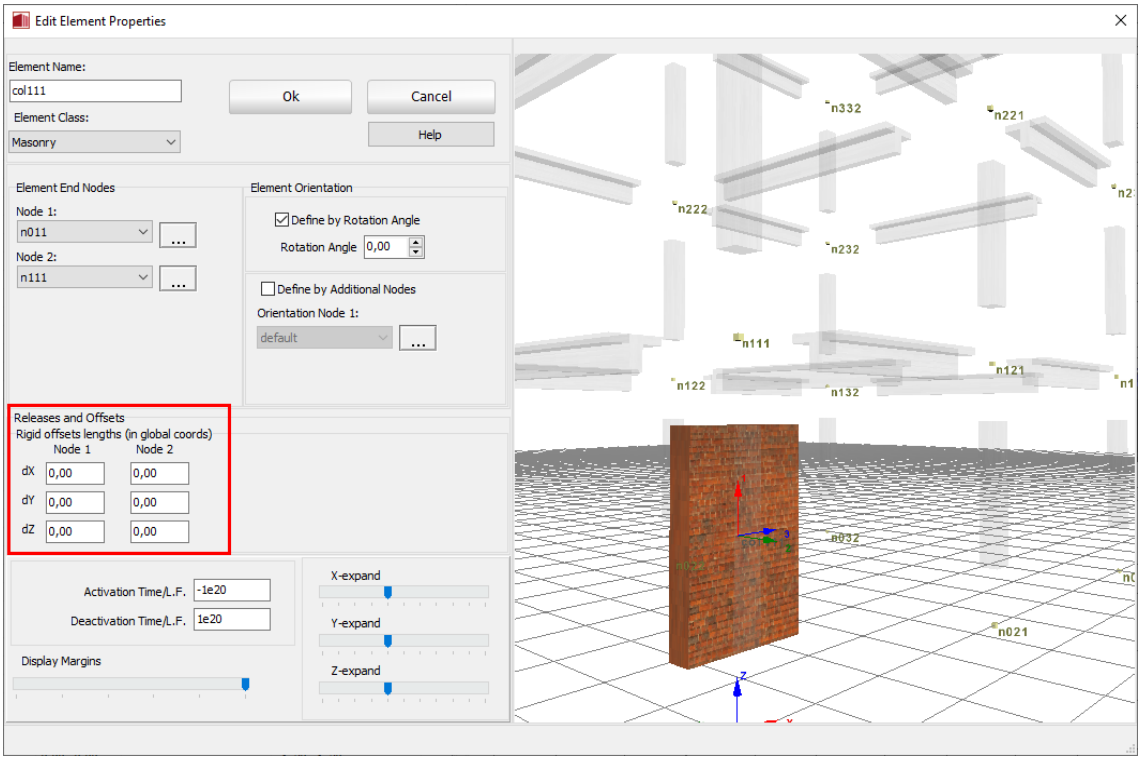
Two nodes need to be defined for these element types, representing the end-nodes of the element, thus defining its length, position in space and direction (local axis 1). A rotation angle or a third node is required so as to define the orientation of the element's cross section (local axes 2 and 3), as described in Global and local axes system.





Edit Element

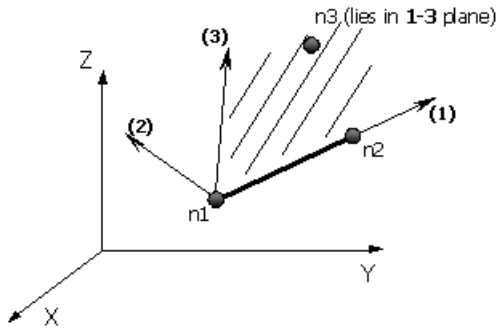
In addition, for each wall element it is possible to specify *Rigid offsets lengths* (in global coordinates) by assigning a value for dX, dY and dZ to Nodes 1 and 2, respectively.

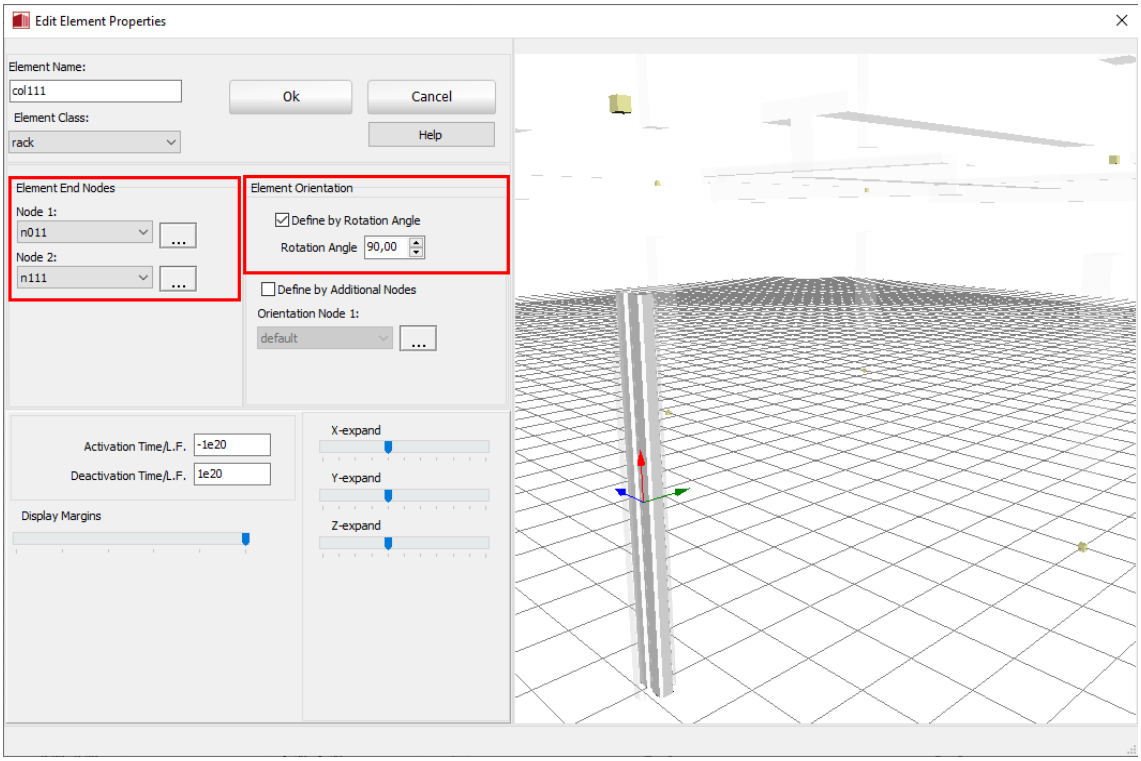


Rigid offsets lengths

Rack elements – rack & rackH

Two nodes need to be defined for these element types, representing the end-nodes of the element, thus defining its length, position in space and direction (local axis 1). A rotation angle or a third node is required so as to define the orientation of the element's cross section (local axes 2 and 3), as described in Global and local axes system.

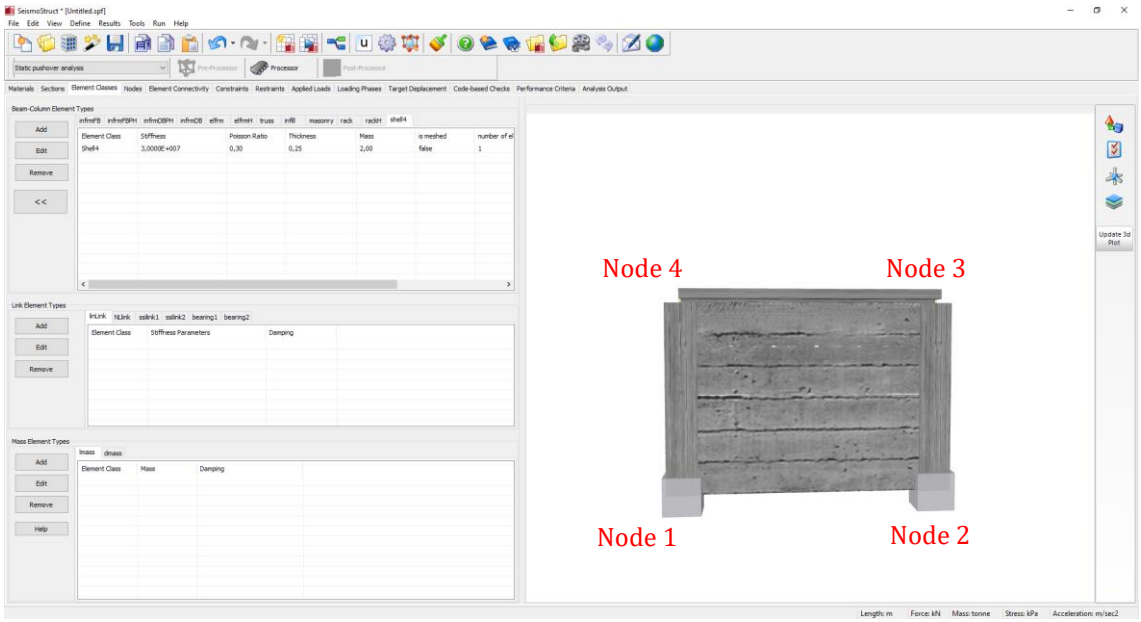




Edit Element

Shell element - shell4

Four nodes need to be defined for this element type. These correspond to the corners of the shell element, must be entered in anti-clockwise sequence starting from the lower-left-hand corner and must all belong to the same plane.

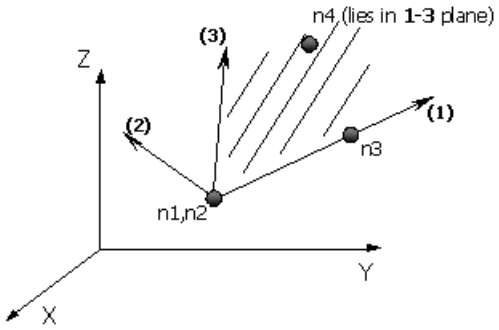


Element Connectivity module – Shell4 element

Link elements linlink, NLlink, ssilink1 & ssilink2, isolator1 & isolator2

Four nodes need to be defined for these element types. The first two are the end-nodes of the element and must be initially coincident since all link elements have an initial length equal to zero. The latter condition implies also that a third node is required to define local axis (1), noting that the orientation of this axis after deformation is determined by its initial orientation and the global rotation of the first node of the element. The fourth node is used to define local axes (2) and (3), following the convention described in global and local axes systems.

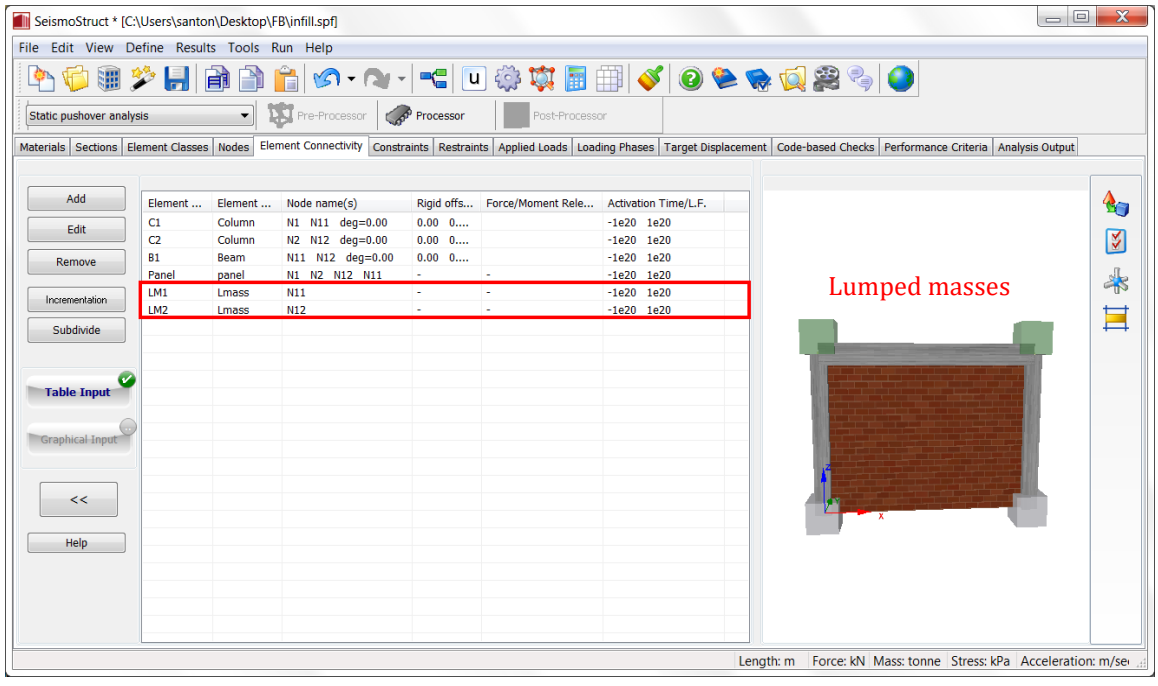
NOTE 1: Instead of the definition of a third and a fourth node, users may simply employ the keyword 'default', which implies that local axis-1 is along the X global axis and local axis-3 is along the Z global axis.



NOTE 2: Users are advised to make use of a non-structural node in the definition of the third and fourth element nodes.

Lumped mass elements - lmass

A single node needs to be defined for this element type.



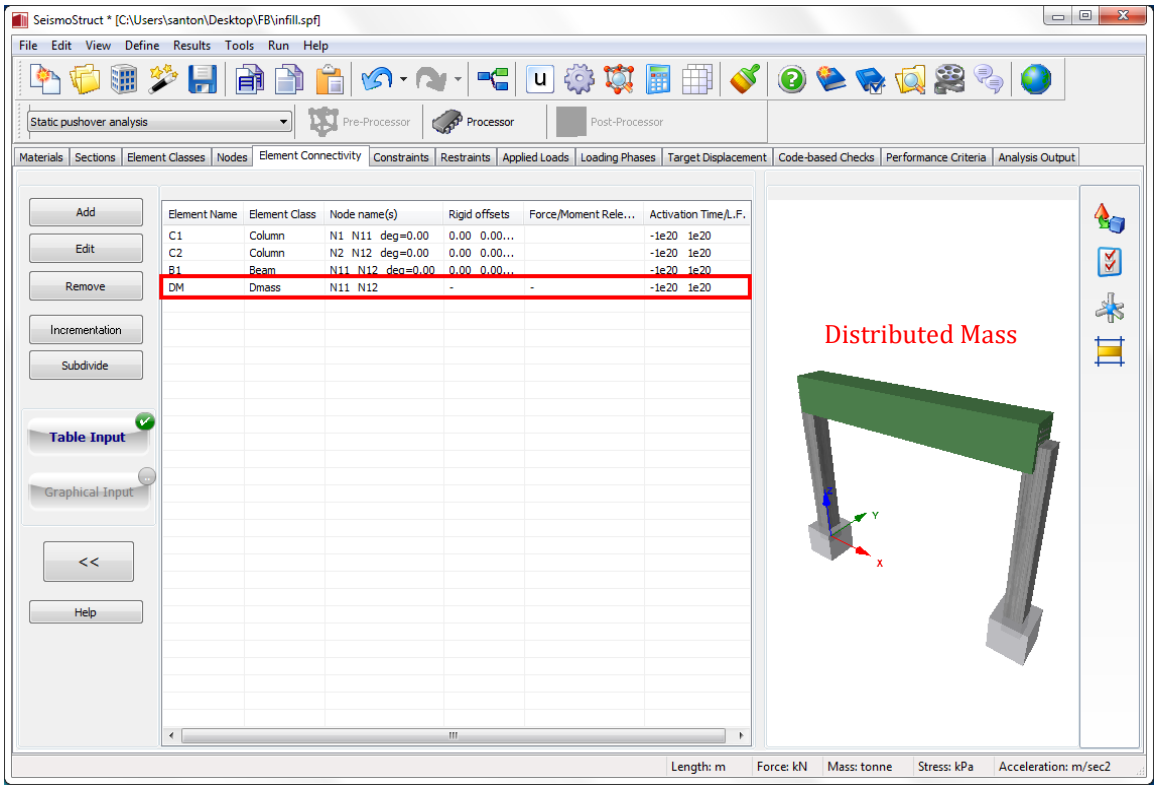
Element Connectivity module – Lumped mass element

In building frames subjected to horizontal excitation, it is customary to assign one lumped element at each beam-column connection, although one element per storey will provide sufficient accuracy for the majority of applications (where vertical excitation and axial beam deformation are negligible).

When analysing bridges, on the other hand, it is common to concentrate deck inertia mass at pier-deck intersection nodes, unless a more rigorous approach is required [e.g. Casarotti and Pinho, 2006].

Distributed mass elements - dmass

Two nodes need to be defined for this element type, usually corresponding to the extremities of structural members (i.e. one dmass element per each column, beam, etc.), unless very large displacements are expected, in which case two or more distributed mass elements per member should be employed.



Element Connectivity module – Distributed mass element

Dashpot damping elements – dashpt & NLdashpt

A single node needs to be defined for these element types (the second node of the dashpot is assumed to be fixed to the ground).

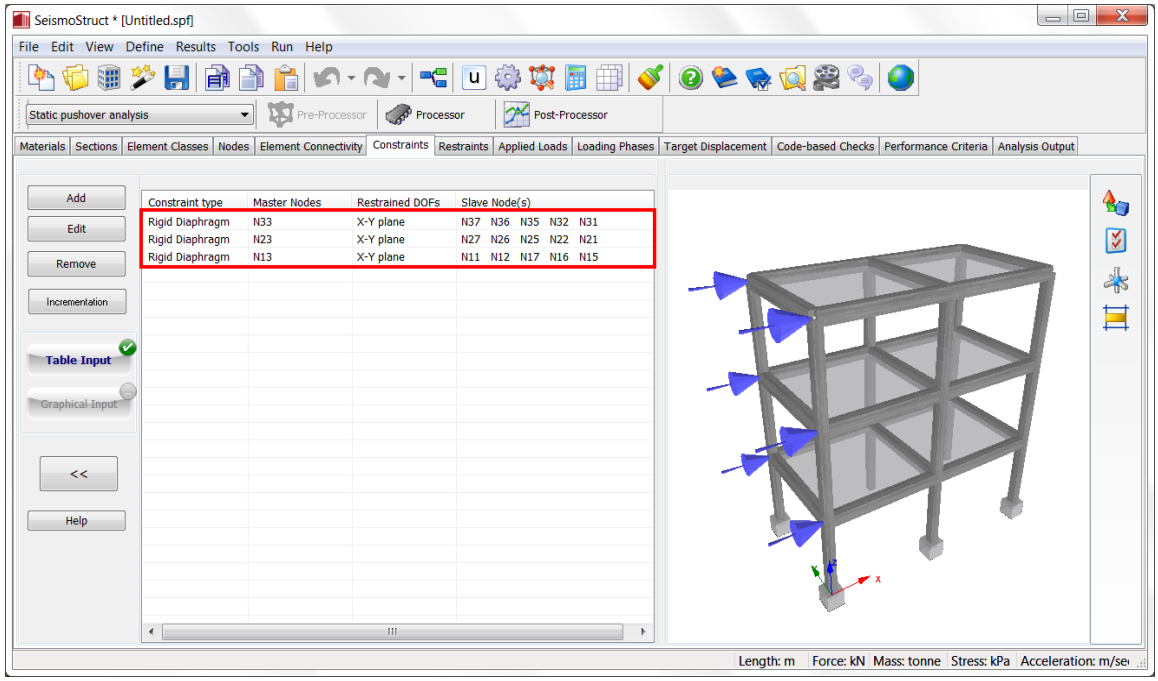
Constraints

The different constraining conditions of the structure are defined in the **Constraints** module, where the constraint type, the associated master node, the restrained DOFs and the slave nodes are identified.

Three different nodal **constraint types** are available in SeismoStruct:

- Rigid Link
- Rigid Diaphragm
- Equal DOF

As in all other modules, the user is capable of adding new conditions (also through the *Graphical Input* button) and removing or editing existing ones (see Editing functions).



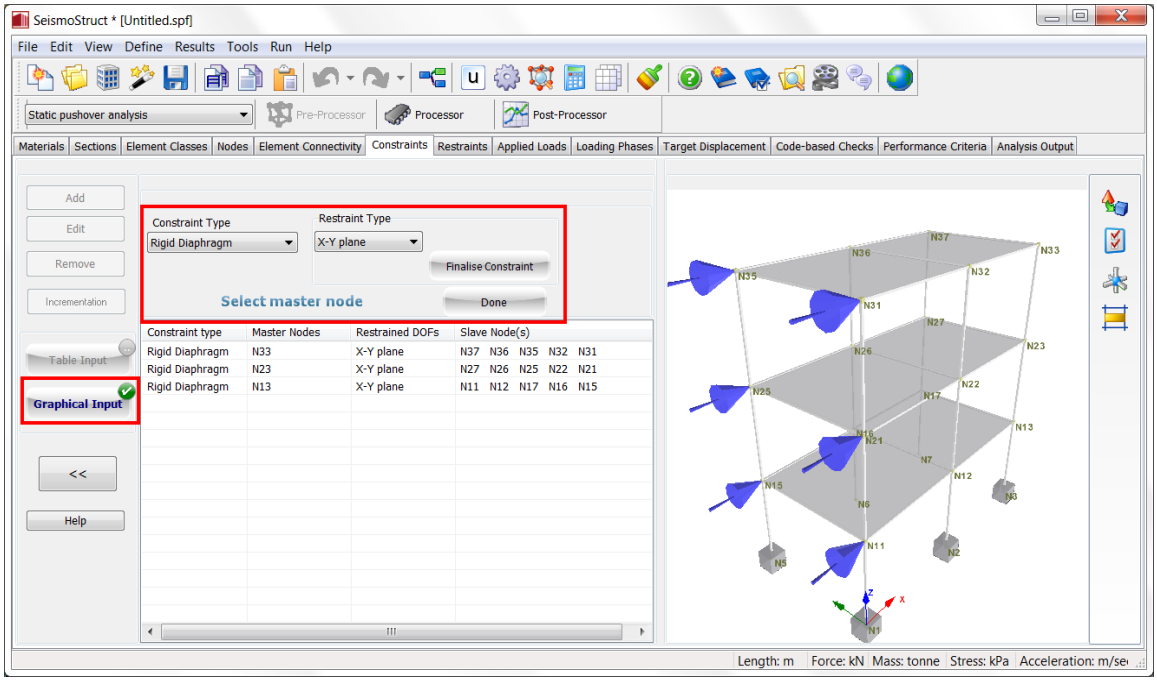
Constraints module

In order to add a new constraint in the *Table Input*, the user has to follow the steps listed below:

1. Click the *Add* button;
2. Select the constraint type from the drop-down menu;
3. Select the restrained DOFs from the drop-down menu(s);
4. Select the master node from the drop-down menu;
5. Select the slave node(s) by checking the corresponding boxes.

Otherwise, in order to graphically add a new constraint in the *Graphical Input* mode, the user has to:

1. Click the *Graphical Input* button;
2. Select the constraint type from the drop-down menu;
3. Select the restrained DOFs from the drop-down menu(s)
4. Double-click to define the master node;
5. Double-click to define the slave node(s);
6. Finally click the *Finalise Constraint* button to complete the process.



Adding New constraint (Graphical Input mode)

In addition, however, *Incrementation facility* is available. As in the case of elements, constraint incrementation enables the automatic generation of new constraints through "repetition" of existing ones. It functions in very much the same manner as the automatic generation of elements, with the difference that in this case only the names of the nodes (master and slave) are incremented. This facility obviously requires that node names respect the number (e.g. 111) or word+number (e.g. n111) formats.

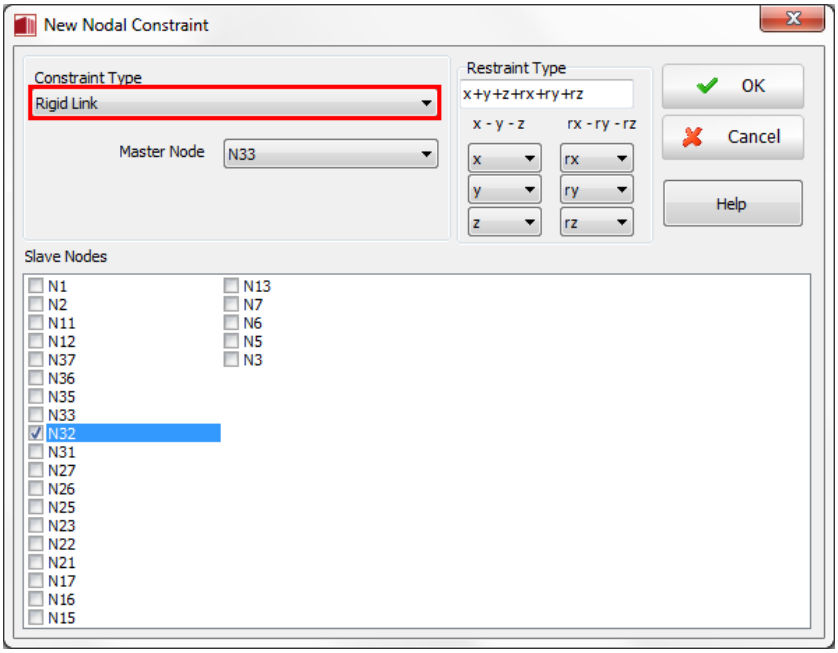
NOTE 1: The application of displacement loads to nodes constrained to displace together may lead to convergence problems (because the applied displacements may be in contrast with the enforced constraint). Amongst many other modelling scenarios, this is particularly relevant when carrying out displacement-based Adaptive Pushover on a 3D model with displacement loads distributed throughout the floor (in such cases either the diaphragm should be eliminated or the displacement loads applied only on the sides of the floor).

NOTE 2: When only two nodes are concerned, from a Finite Elements programming point of view, master and slave nodes are identical; both are "simply" two nodes connected between them. Do refer to the literature for further discussions on this topic [e.g. Cook et al., 1989; Felippa, 2004].

In what follows, an overview of each type is given.

Rigid Link

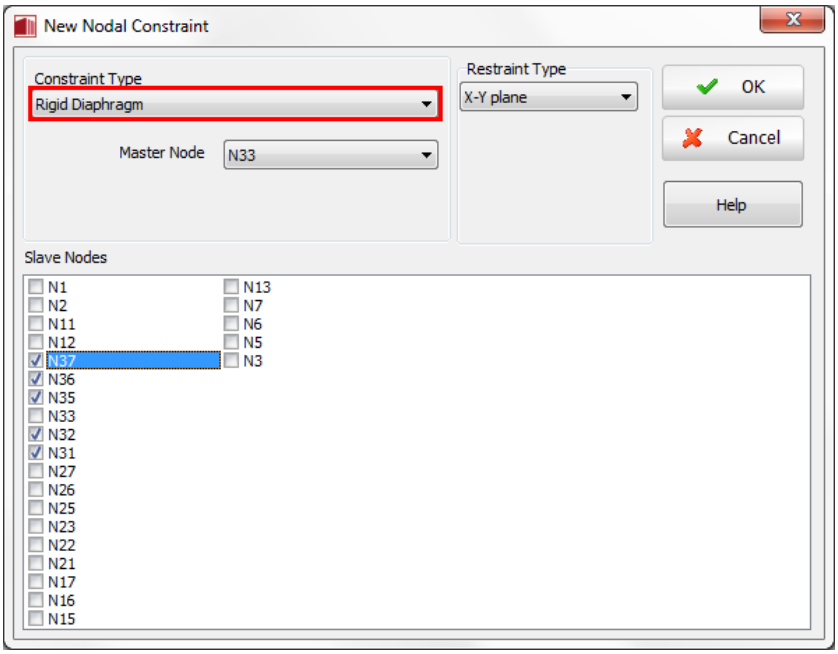
Constrain certain degrees-of-freedom of slave nodes to a master node, by means of a rigid link. In other words, the rotations of the slave node are equal to the rotations of the master node, whilst the translations of the former are computed assuming a rigid lever-arm connection with the latter. Both master and slave nodes need to be defined for this constraint type, and the degrees-of-freedom to be slaved to the master node (restraining conditions) have to be assigned.



Adding New Rigid Link (Table Input)

Rigid Diaphragm

Constrain certain degrees-of-freedom of slave nodes to a master node, by the use of rigid planes (i.e. all constrained nodes will rotate/displace in a given plane maintaining their relative position unvaried, as if they were all connected by rigid lever-arms). As for the previous constraint type, both master and slave nodes need to be defined, with the master node typically corresponding to the baricentre of the diaphragm. Moreover the restraining conditions, in terms of rigid plane connections (X-Y, X-Z and Y-Z plane), need also to be assigned.



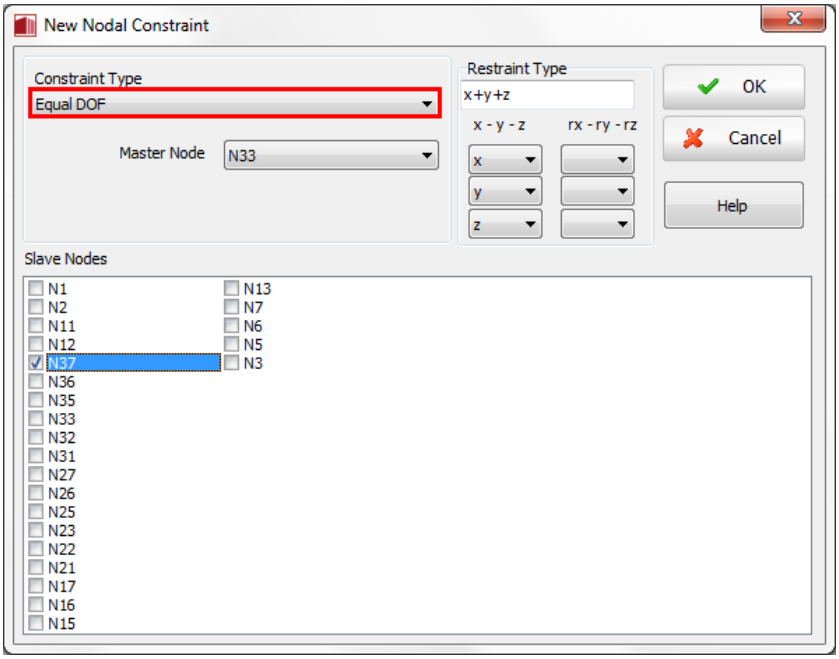
Adding New Rigid Diaphragm (Table Input)

NOTE 1: In general, the diaphragm master node location should correspond to the centre of mass of each floor (it is noted that the location of slab master nodes in Wizard-created 3D models is merely demonstrative and not necessarily correct).

NOTE 2: Constraining all the nodes of a given floor level to a rigid diaphragm may lead to an artificial stiffening/strengthening of the beams, since the latter become prevented from deforming axially (it is recalled that unrestrained nonlinear fibre elements subjected to flexure will deform axially, since the neutral axis is not at the section's baricentre). Users are therefore advised to use great care in the employment of Rigid Diaphragm constraints, carefully selecting the floor nodes that are to be constrained.

Equal DOF

Constrain certain degrees-of-freedom of slave nodes to a master node. Contrary to the Rigid Link constraint, here all constrained dofs (rotations and translations) of master and slave nodes feature the exact same value (i.e. no rigid lever-arm connection exists between them). Both master and slave nodes need to be defined for this constraint type, and the degrees-of-freedom to be slaved to the master node (restraining conditions) have to be assigned.



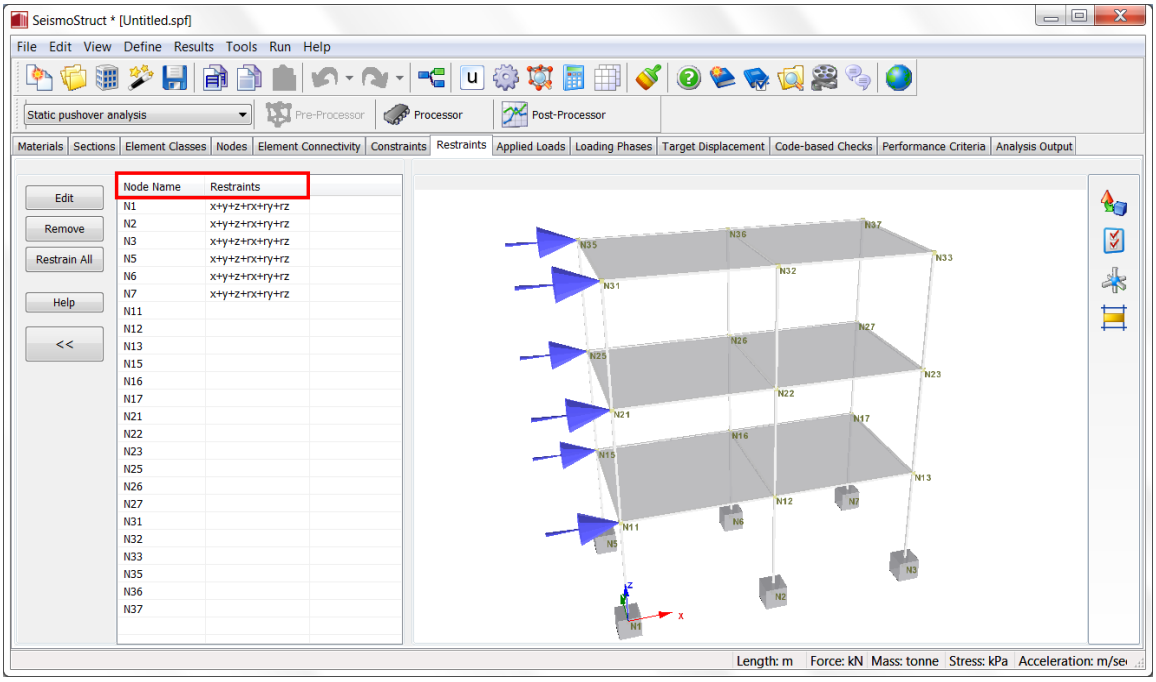
Adding New Equal DOF (Table Input)

NOTE: In previous releases of SeismoStruct, link elements featuring a lin_sym response curve were typically employed to model pinned joints (zero stiffness) and/or Constraints. However, users may now use the Equal DOF facility of this Constrain module to achieve the same objective; e.g. a pin/hinge may be modelled by introducing an 'Equal DOF' constrain defined for translation degrees-of-freedom only.

Restraints

The boundary conditions of a model are defined in the **Restraints** module, where all structural nodes are listed and available for selection and restraining against deformation in any of the six degrees-of-freedom.

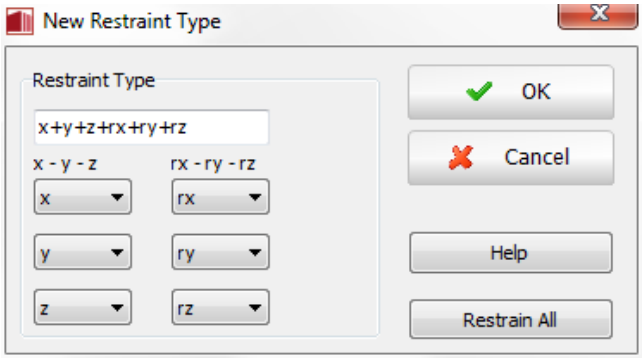
IMPORTANT: Copying & Pasting of data is not possible in this module.



Restraints module

When carrying out 2D analysis, it might be useful to restrain all out-of-plane degrees-of-freedom, so as to minimise running time. Hence, and as an example, for a model defined and responding in the x-z plane (2D models created with the *Wizard* feature are defined in this plane), all nodes should possess y+rx+rz restraining conditions. Note that for this common type of situations (y=0, and y+rx+rz restrained for all the nodes) the y+rx+rz restraints are not shown on the 3D plot, for reasons of clarity.

The modelling of foundation flexibility can be accomplished through the use of link elements, the first structural node of which is restrained in all directions (x+y+z+rx+ry+rz), whilst the second is connected to the structure. Any of the currently available response curves can then be employed to model the elastic or inelastic response of the soil in each of the six degrees-of-freedom.



Edit Restraint window

NOTE: In order to model yield penetration at the base, when present, it suffices to increase the length of the corresponding column element by the adequate amount. Refer to the available literature for indications on how to compute such yield penetration length [e.g. Paulay and Priestley, 1992; Priestley et al., 1996].

LOADING

Once the **structural geometry** has been defined, the users have the possibility of defining the loading applied to the structure through the **Applied Loads** module. Then, a number of additional settings, which vary according to the type of analysis being carried, must be specified in the following modules:

- Tsunami Parameters
- Loading Phases
- Time-history Curves
- Adaptive Parameters
- IDA Parameters
- RSA Parameters

NOTE: Obviously none of these modules will appear when the Eigenvalue analysis is selected.

Tsunami Parameters

With the Tsunami Nonlinear Analysis users can assess the capacity of a structure for Tsunami Loads within the framework of ASCE 7-16. The Tsunami Loads can be calculated either using the ASCE 7-16 relationships or the relationships proposed in Foster et al. [2017] while the analysis procedures are summarized in Petrone et al. [2017] and Baiguera et al. [2019]. More information on the Nonlinear Tsunami analysis can also be found in Appendix B.

For the Tsunami Nonlinear Analysis a number of parameters need to be defined and are summarized below:

Tsunami Force Calculation Method

The user can select whether the Total Tsunami Force at each step is to be calculated using

1. **ASCE 7-16 Method:** Utilizing the relationship 6.1-2 of the 6.10.2.1 paragraph of ASCE 7-16 (ASCE 7-16 chapter 6).

$$F_{dx} = \frac{1}{2}\rho_s I_{tsu} C_d C_{cx} B(hu^2)$$

Where F_{dx} is the Total Tsunami Force, I_{tsu} is the Tsunami Importance Factor, C_d is the drag Coefficient for the building (see Table 6.10-1 of ASCE 7-16 Chapter 6), h is the inundation depth, u is the flow velocity and C_{cx} is a coefficient given by equation 6.10-3 of ASCE 7-16 chapter 6 and B is the width of the structure affected by the Tsunami

2. **Foster et al (2017) Method:** Utilizing equation (1) from Petrone et al. [2017] relationships summarized in the work by Foster et al. [2017].

$$\text{For Fr<Fr}_{rc}\text{: } F/B = sgn(u)\ \tfrac{1}{2} C_D \rho_s u^2 h$$

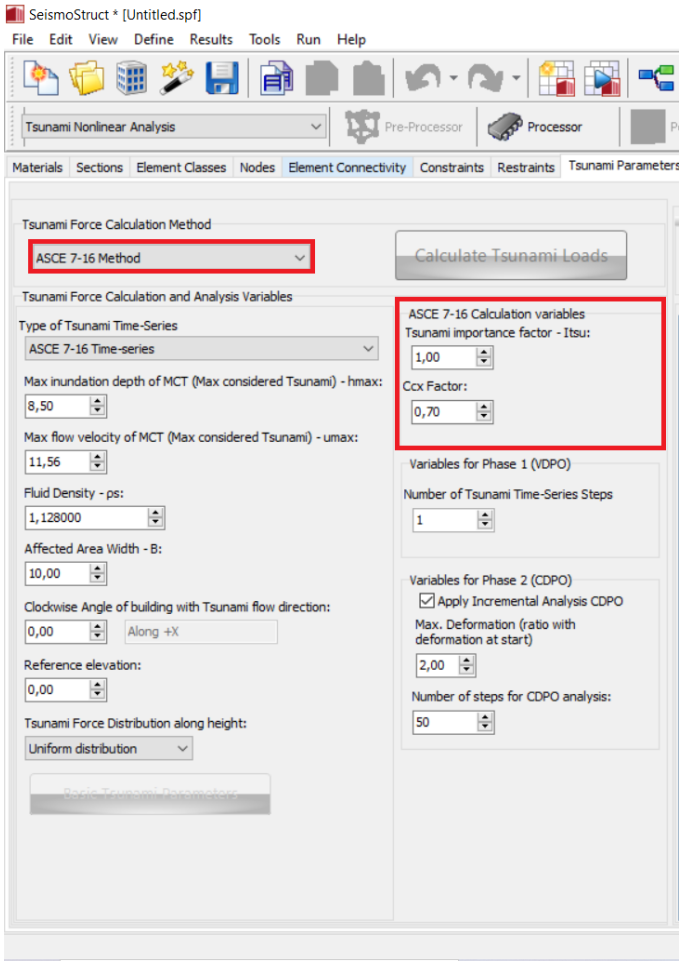
$$\text{For Fr}\geq\text{Fr}_{rc}\text{: } F/B = sgn(u)\ \lambda \rho_s g^{1/3} u^{4/3} h^{4/3}$$

Where Fr is the Froude number, F_{rc} is the Froude number threshold calculated according to the equations in Petrone et al. [2017], C_d is a drag coefficient, h and u are the inundation depth and flow velocity respectively, g is the acceleration of gravity, B is the width of the Tsunami affected area and λ is calculated using eq. (26) from Foster et al. [2017].

For each method a number of method-specific variables need to be defined namely:

Variables for the ASCE 7-16 method:

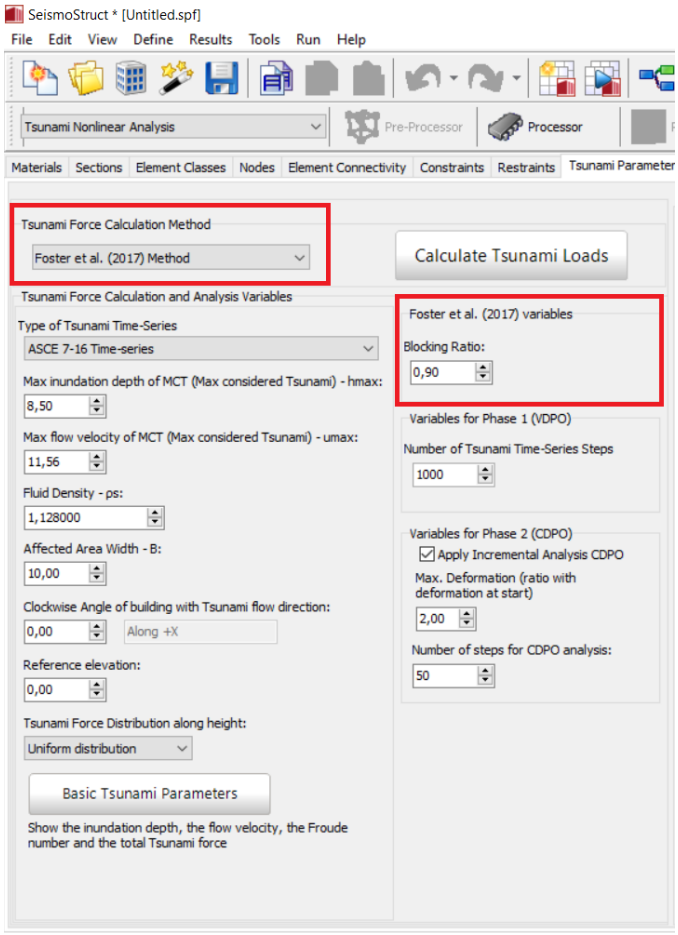
a. Tsunami Importance Factor (I_{tsu}): The Tsunami Importance Factor is taken according to the 6.8.3.2 paragraph of ASCE 7-16, Chapter 6.
b. The C_{cx} factor calculated according to the equation 6.10.2 given in the paragraph 6.10.2.1 of the ASCE 7-16 Chapter 6.



ASCE 7-16 method along with the calculation variables

Variables for the Foster et al. (2017) method:

a. Blocking ratio: The blocking fraction b/w where w is the width of a channel of water flow and b is the width of an obstacle in the way of the channel, a model used in experimental works on Tsunami Modeling [Qi et al., 2014; Foster et al., 2017].

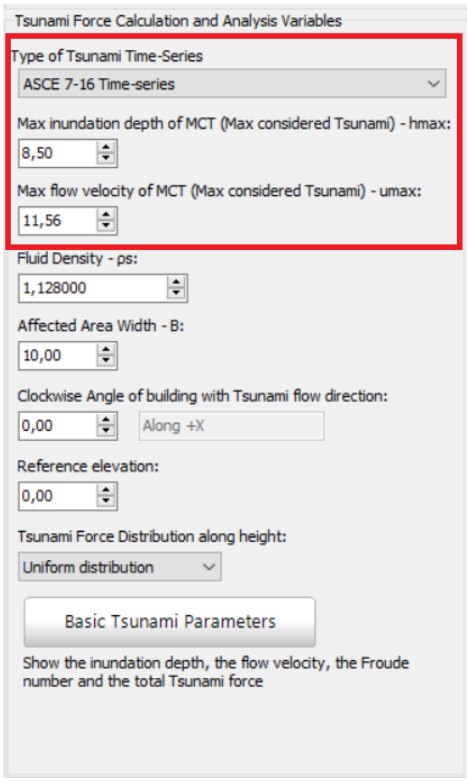


Foster et al. [2017] method along with the calculation variables

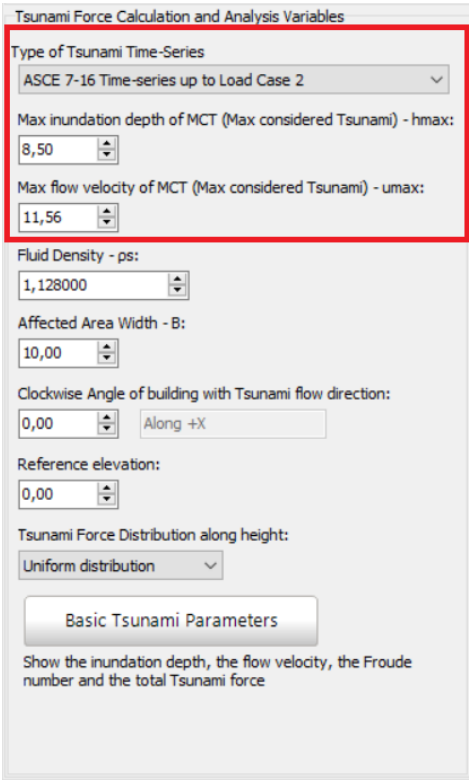
Tsunami force calculation and analysis variables

Type of Tsunami time series: The user can specify the type of depth and velocity time series to be used for the Tsunami modeling and the calculation of the Total Tsunami Force. The user can choose among the following options with each option needing its own variables to be specified.

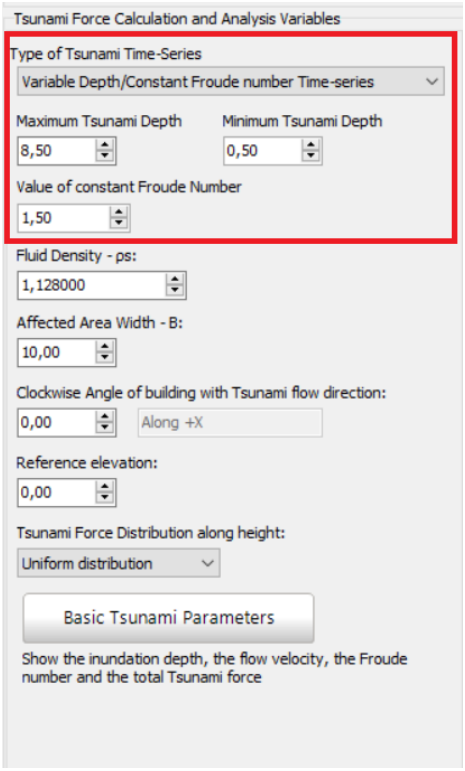
i) ASCE 7-16 Time-Series: Depth and Velocity time series of Figure 6.8-1 of the chapter 6 of ASCE 7-16. In order to fully define the time-series, the Maximum Inundation Depth (hmax) and Flow velocity (umax) of the Maximum Considered Tsunami should be defined
ii) ASCE 7-16 Time-Series up to Load Case 2: The Depth and Velocity time-series of ASCE 7-16 but up to the Load Case 2 point indicated in Figure 6.8-1 of ASCE 7-16 Chapter 6. The variables that need to be defined are again hmax and umax.
iii) Variable Depth/Constant Froude Number Time-Series option: The depth is increasing linearly from a minimum to a maximum depth while the velocity is calculated using a constant Froude number. The needed variables are the Minimum and Maximum Depth and the Constant Froude number
iv) Time-Series with max.depth = building height: This option creates the ASCE 7-16 depth/velocity time-series us to load case 2 and after Load Case 2 the depth continues to increase linearly until it reaches the building height. The hmax and umax need to be defined for the calculation of the ASCE depth and Velocity time-series.



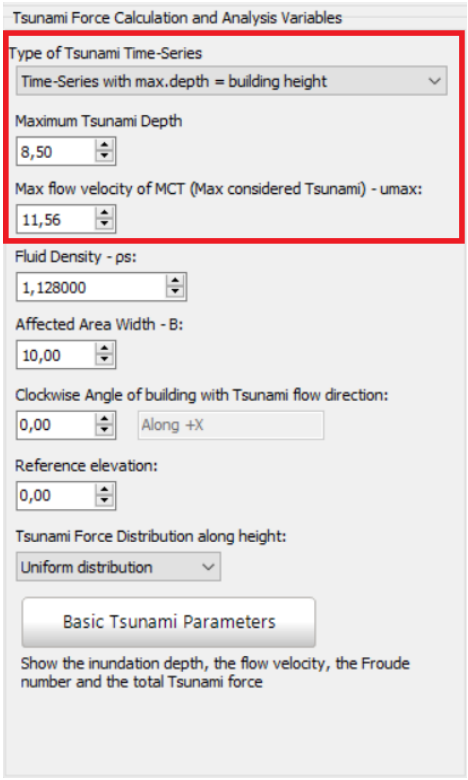
ASCE 7-16 time-series option with the additional needed variables



ASCE 7-16 time-series up to Load Case 2 option with the additional needed variables



Variable Depth/Constant Froude number option with the additional needed variables



Time-Series with max.depth = building height option with the additional needed variables

Fluid Density - ρs

The density of the fluid considered for the calculations of the Tsunami total Force, usually taken equal to the density of sea-water.

Affected Area Width - B

The total width of the structure affected by the tsunami load. For more information the user is directed to chapter 6 of ASCE 7-16 and Foster et al. 2017.

Clockwise Angle of Building with the Tsunami Flow Direction

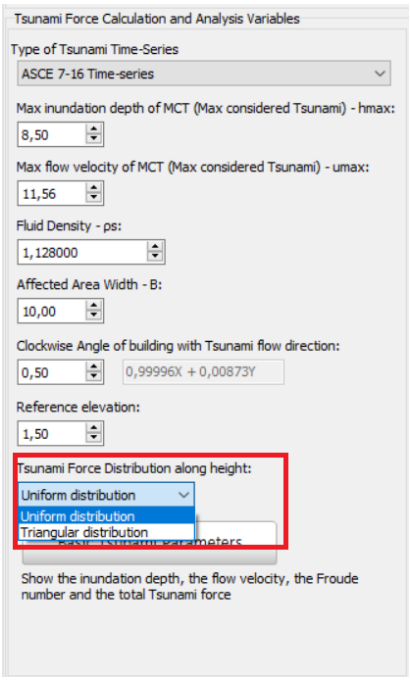
This variable defines the clockwise angle between the Tsunami flow and the X global Axis of Seismostruct. An angle equal to zero indicates that the Tsunami flow direction is along the X global Axis.

Reference Elevation

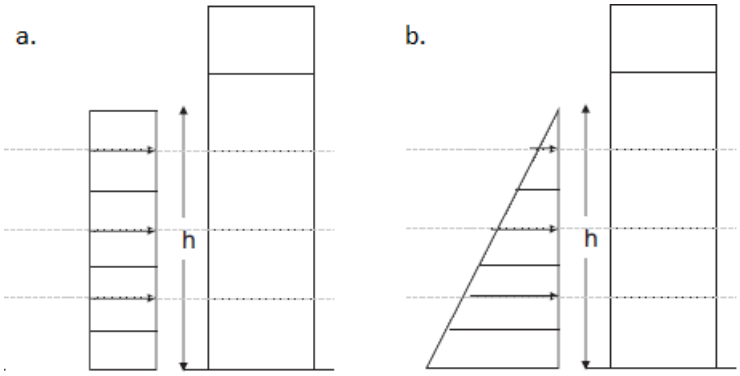
The height difference between the lowest point of the building and the zero inundation depth. A zero value indicates that the point of zero inundation depth coincides with the building lowest node. A negative value indicates that the tsunami zero inundation depth is lower than the building lowest node and a positive value indicates that the lowest node of the building is lower than the zero tsunami inundation depth.

Tsunami force Distribution along height

This option defines the pattern used for the distribution of the Total Tsunami Force along the building height. The Uniform distribution indicates that the Tsunami load is uniformly distributed along each floor affected by the Tsunami while the Triangular distribution follows the distribution pattern (B) indicated in Figure 9 in the work by Petrone et al. [2017].



Tsunami Force Distribution along height options



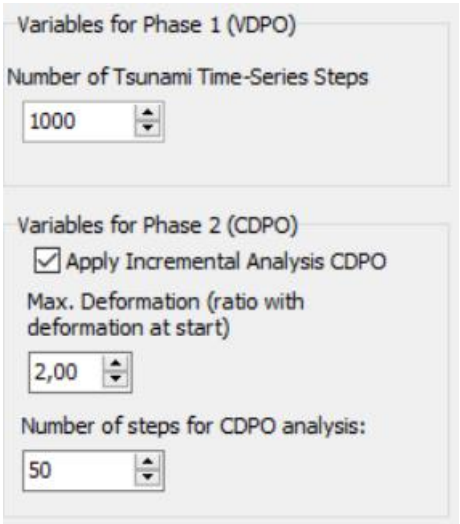
Patterns of Total Tsunami Force distribution along the height of the building: a. Uniform Distibution and b. Triangular distribution after Petrone et al. [2017]

Variables for Phase 1 (VDPO)

The only variable to be defined for the VDPO analysis phase is the number of steps to be used for the VDPO analysis

Variables for Phase 2 (CDPO)

For phase 2 (CDPO) the user needs to define whether the CDPO analysis will be performed or not, the maximum deformation ratio achieved during the CPDO analysis (ratio with the deformation of the control node at the start of the CDPO analysis), and the number of steps in which the CDPO analysis is to be carried out.



Variables to be defined for the VDPO and CDPO phases of the Tsunami Analysis

Tsunami Loaded Nodes

In the table with all the nodes of the building, the user needs to define which nodes are loaded by the Tsunami force. The part of the Total Tsunami force attributed to a certain group of nodes with identical height will be then distributed to the selected nodes belonging to the particular group.

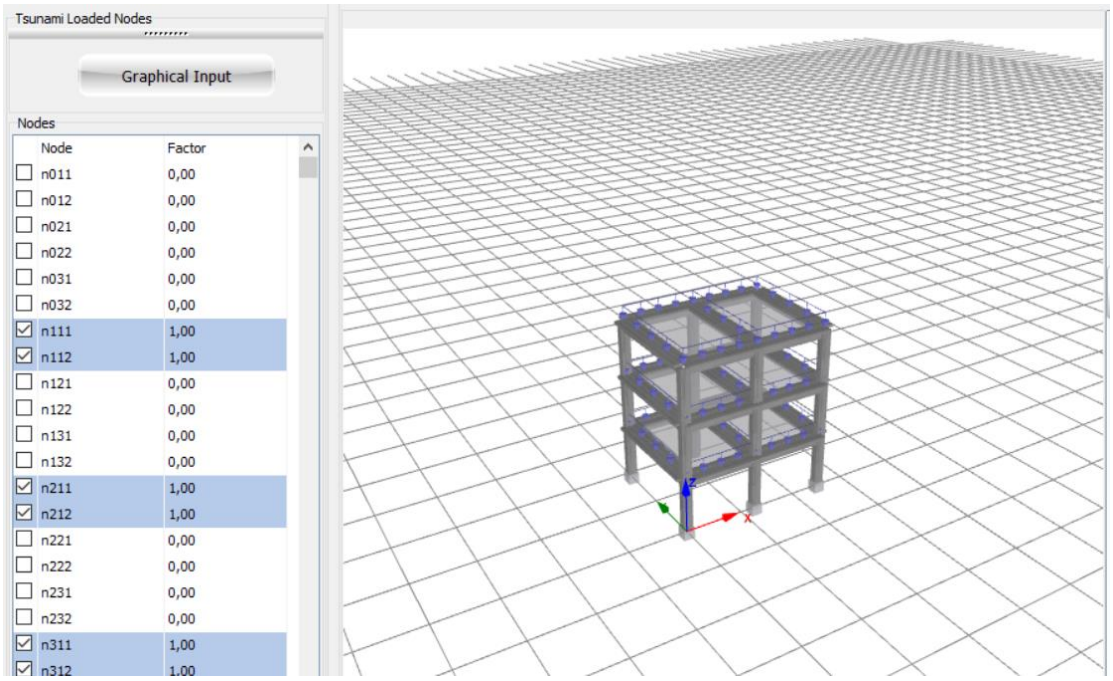
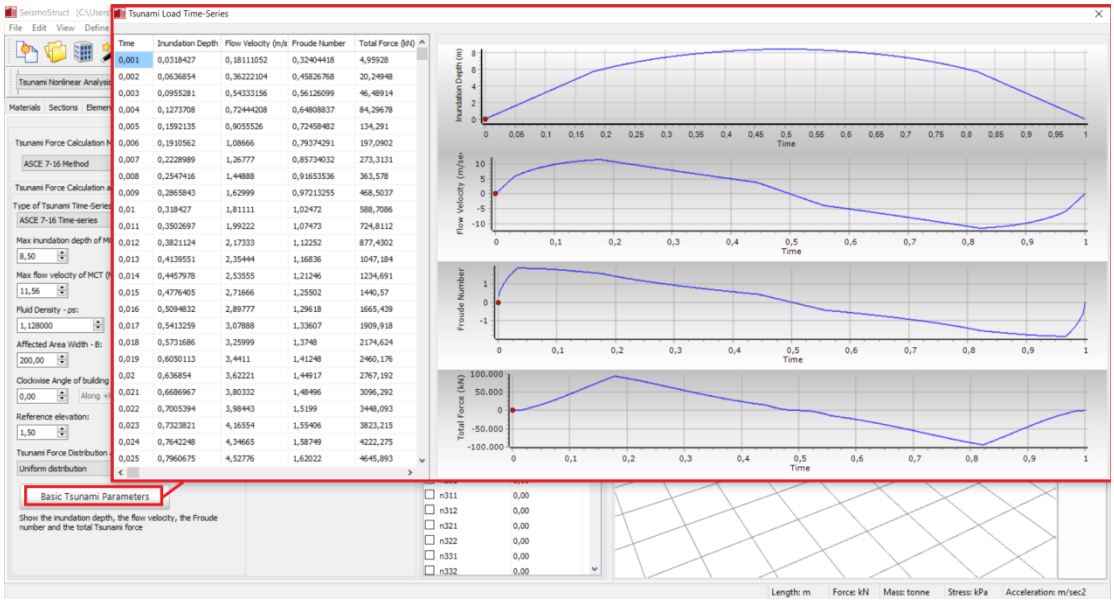


Table with the Tsunami Loaded Nodes

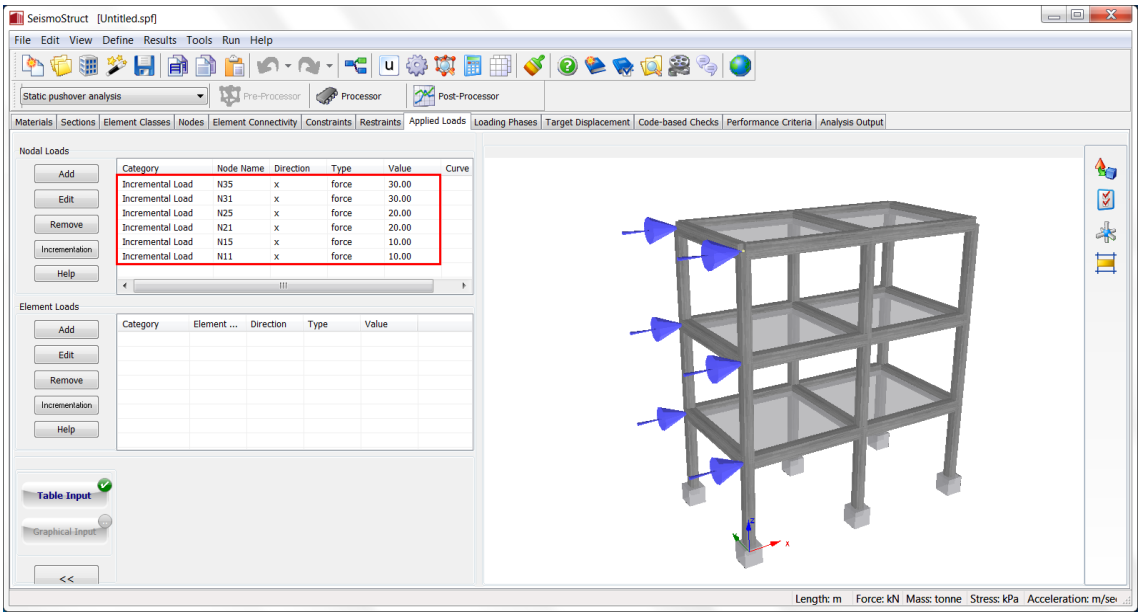
When all the variables are defined the user can select the "Calculate Tsunami Load" button and execute the calculation of the inundation depth/velocity and Tsunami Total Force time-series and distribute the Total Tsunami Force on the Loaded Nodes. The Tsunami Force time-series for the VDPO analysis phase will be introduced in the "Time-history curves" tab and the Static Time-history nodal loads for the VDPO analysis phase will be introduced in the Applied Loads tab of the program. Finally selecting the "Basic Tsunami Parameters" button will open a window presenting the time histories of the main Tsunami Analysis parameters including the inundation depth, the flow velocity, the Froude number and the Tsunami Total Force.



Window with Main Tsunami Load time-Series

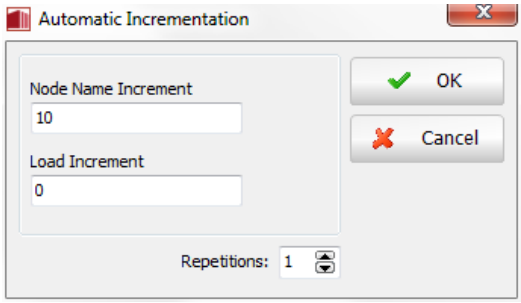
Nodal Loads

In SeismoStruct there are *four nodal load categories* that can be selected. These can be applied to any structural model, either in isolated fashion or in a combined manner, depending on the type of analysis being carried out. Further, it is noteworthy that the term "load", as employed in SeismoStruct, refers to any sort of action that can be applied to a structure, and may thus consist of forces, displacements and/or accelerations.



Applied Loads module

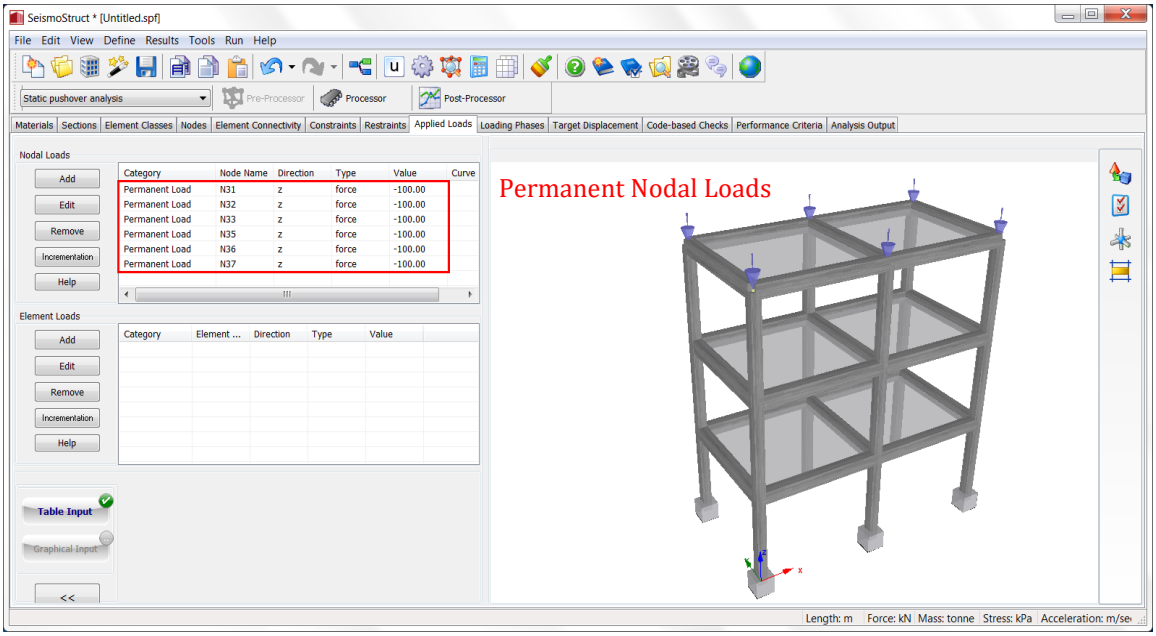
As in all other modules, the user is capable of adding new loads and removing/editing existing ones. In addition, a load *incrementation* facility is also available, so as to enable easier generation of new nodal actions. It functions in very much the same manner as the automatic generation of nodes does; the user defines node name and load value increments, and these are then employed to automatically generate new nodal actions through "repetition" of a selected set of already prescribed loads. This facility requires that node names respect the number (e.g. 100) or word+number (e.g. nod20) formats.



Load Incrementation

Permanent loads (dark blue arrows in rendering plot)

These comprise all static loads that are permanently applied to the structure. They can be forces (e.g. self-weight) or prescribed displacements (e.g. foundation settlement) applied at nodes.



Example of Permanent Loads

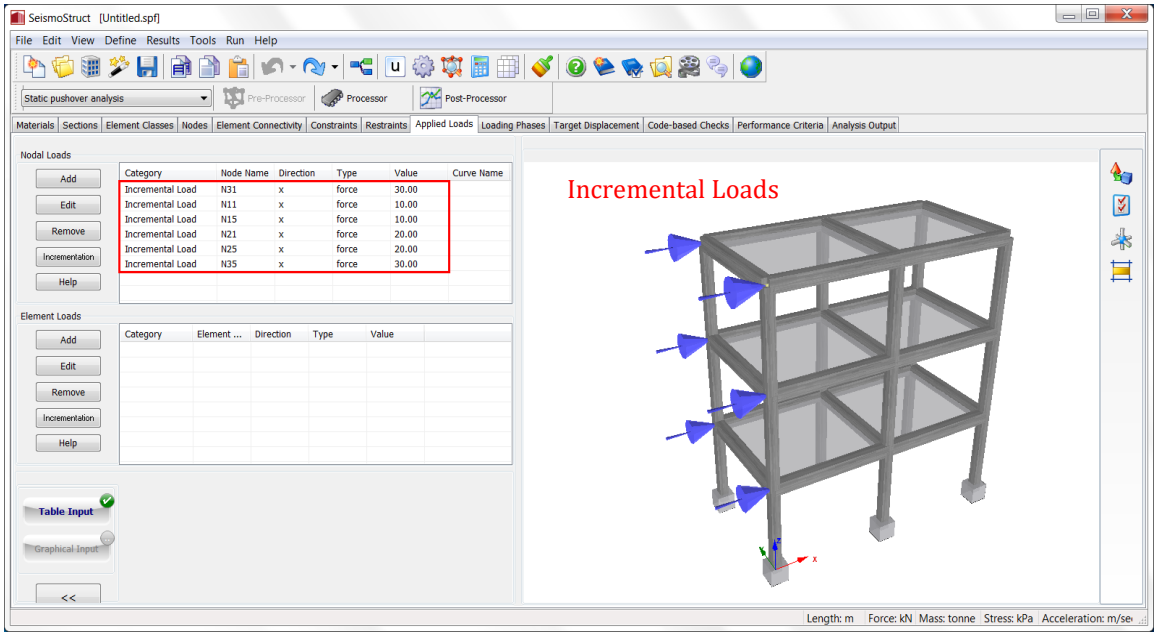
When running an analysis, permanent loads are considered prior to any other type of load, and can be used on all analysis types, with the exception of Eigenvalue analysis, where the permanent loads are only used to derive masses, if a relevant option has been chosen in the Project Settings > Gravity & Mass module.

NOTE 1: Gravity loads should be applied downwards, for which reason they always feature a negative value.

NOTE 2: If it has been selected from the Project Settings -> Gravity & Mass menu that loads are derived from masses (in the gravity direction based on the g value, or in any translational direction, according to user-defined coefficients) and the model already features the presence of masses (defined in the materials, sections or element classes modules), then the program will automatically compute and apply distributed permanent loads.

Incremental loads (light blue arrows in rendering plot)

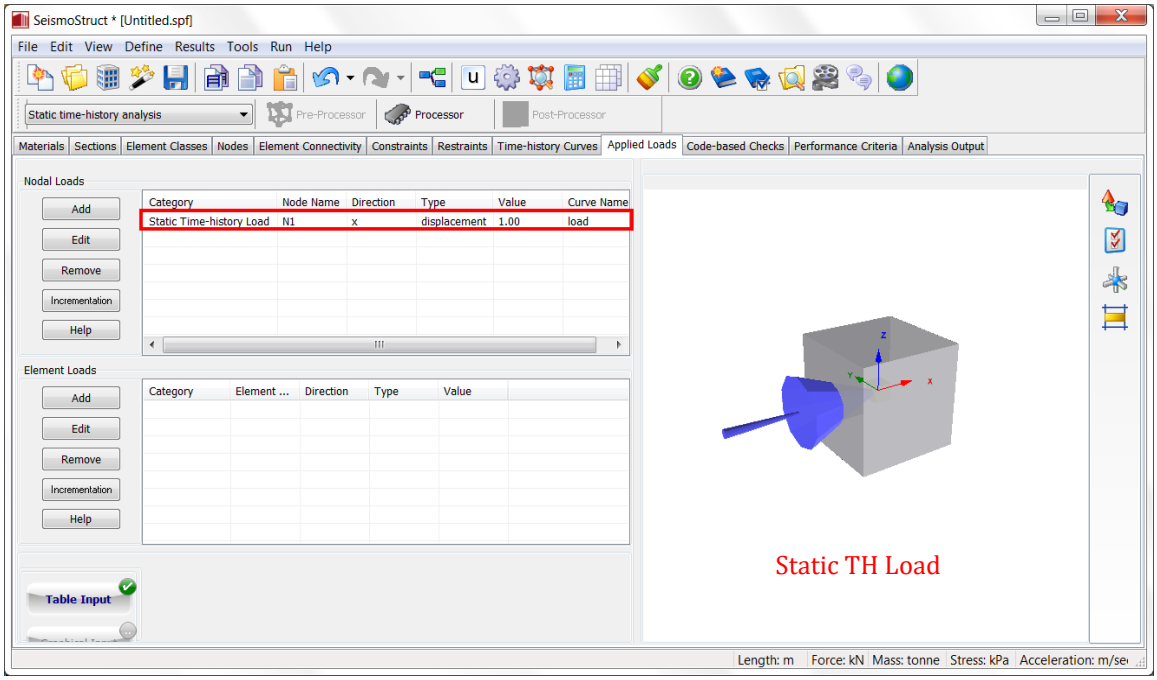
These represent pseudo-static loads (forces or displacements) that are incrementally varied. The magnitude of a load at any step is given by the product of its nominal value, defined by the user, and the current load factor, which is updated in automatic or user-defined fashion. Incremental loads are exclusively employed in pushover type of analyses, generally used to estimate horizontal structural capacity. Both adaptive and non-adaptive load profiles may be used, though the application of Displacements within an adaptive pushover framework stands out as the clearly recommended option [e.g. Antoniou and Pinho, 2004b; Pietra et al., 2006; Pinho et al., 2007].



Example of Incremental Loads

Static time-history loads (light blue arrows in rendering plot)

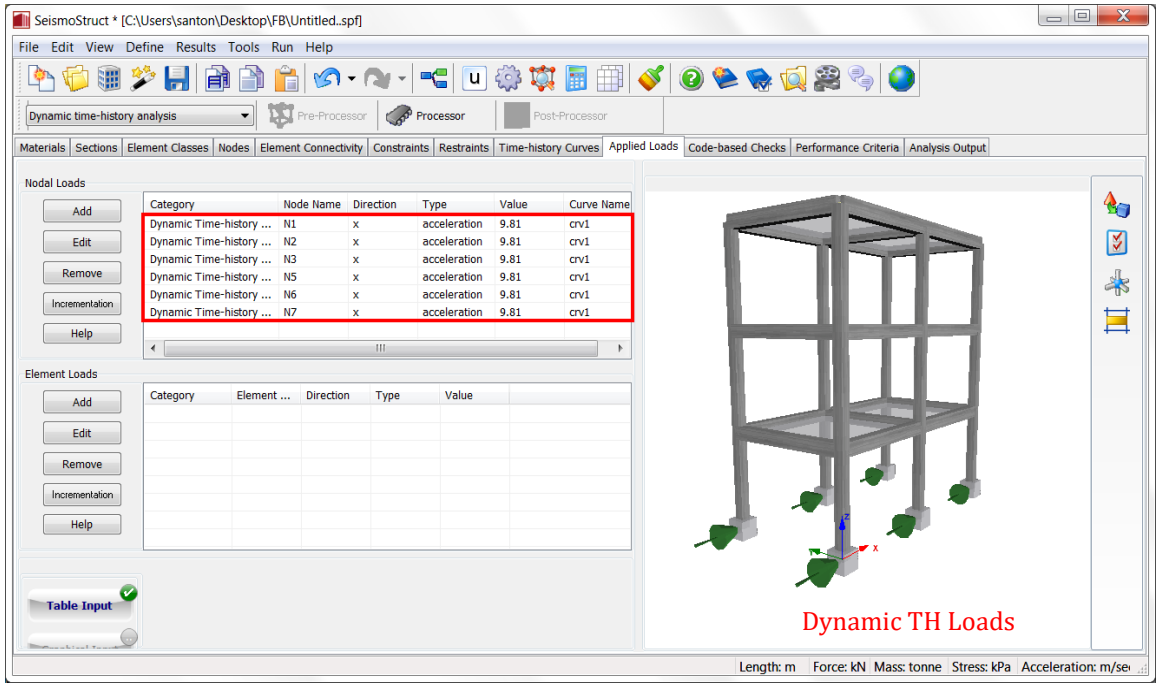
These are static loads (forces and/or displacements) that vary in the pseudo-time domain according to user-defined loading curves. The magnitude of a load at any given time-step is computed as the product between its nominal value, defined by the user, and the variable load factor, characterised by the loading curve. This type of loads is exclusively used in static time history analysis, commonly employed in the modelling of quasi-static testing of structures under various force or displacement patterns (e.g. cyclic loading).



Example of Static Time-history Load

Dynamic time-history loads (green arrows in rendering plots)

These are dynamic loads (accelerations or forces) that vary according to different load curves in the real time domain. The product of their constant nominal value and the variable load factor obtained from its load curve (e.g. accelerogram) at any particular time gives the magnitude of the load applied to the structure.



Example of Dynamic Time-history Loads

These loads can be used in dynamic time history analysis, to reproduce the response of a structure subjected to an earthquake, or in incremental dynamic analysis, to evaluate the horizontal structural capacity of a structure.

NOTE 1: The application of displacement loads to nodes constrained to displace together (e.g. through a rigid link or similar) may lead to convergence problems (because the applied displacements may be in contrast with the enforced constraint).

NOTE 2: With force-based frame element formulations it is possible to explicitly model loads acting along the member, and hence avoid the need for distributed loads to be transformed into equivalent point forces/moments at the end nodes of the element (and then for lengthy stress-recovery to be employed to retrieve accurate member action-effects). However, such feature could not yet be implemented in SeismoStruct.

NOTE 3: Strength and stiffness of infill elements 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). If users wish their infills to resist gravity loads, then they should define the latter as non-initial loads.

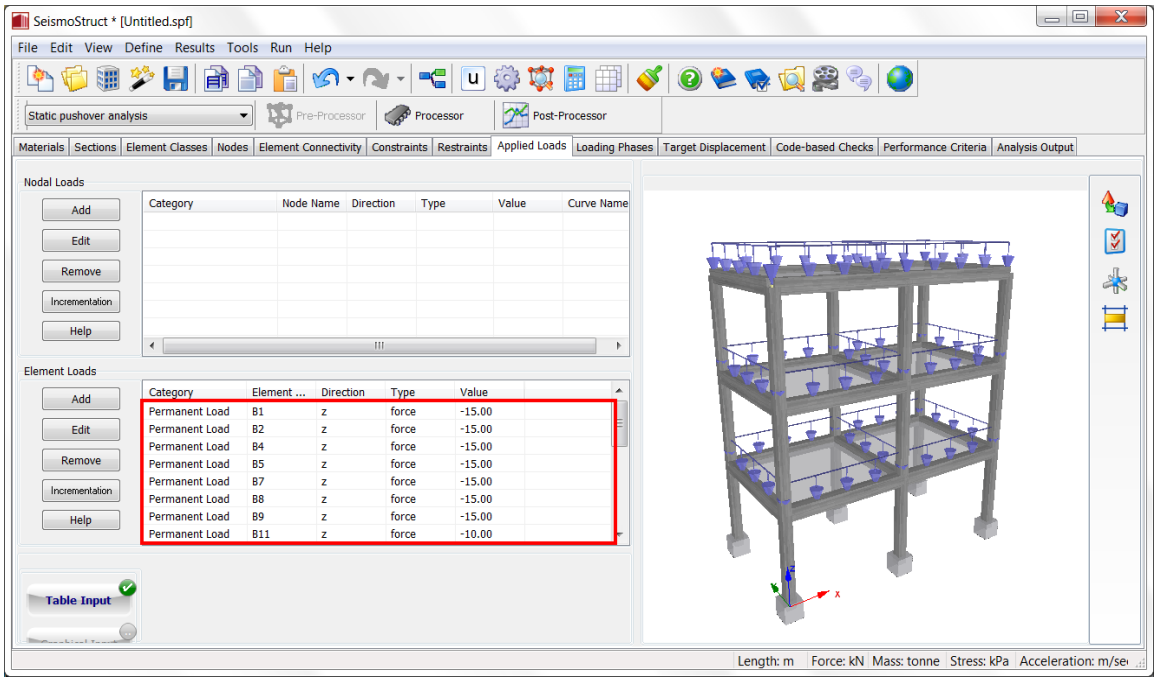
NOTE 4: When assessing the horizontal capacity of non-symmetric structures, users should take care to consider the application of the incremental loads in both directions (i.e. run two pushover analyses) in order to identify the capacity of the structure in both its "weak" and "strong" directions.

NOTE 5: Users who wish to apply loads (including accelerograms) with an angle of incidence different from 90 degrees, can do so by defining such loads in terms of multiple-direction components (x, y, z).

NOTE 6: Explosions may produce three distinct types of loading: (i) air shock wave, which can be considered as an impulsive load, dynamic action or a quasi-static wave depending on its characteristics, (ii) dynamic pressure applied to the structure due to gas expansion and (iii) ground shock wave, which has three types of waves with different velocities and frequencies, namely, compression waves, shear waves and surface waves [Chege and Matalanga, 2000]. Therefore, Permanent, Static time-history and Dynamic time-history loads should be employed when modelling this type of action.

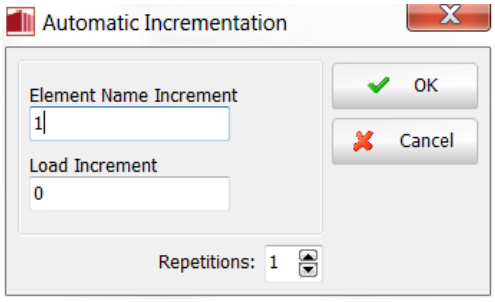
Element (Distributed) Loads

Permanent distributed loads applied along the element's length can be introduced, as shown in the figure below. As pointed out in the Nodal Loads section, permanent loads can be used on all analysis types and are always considered prior to any other type of load.

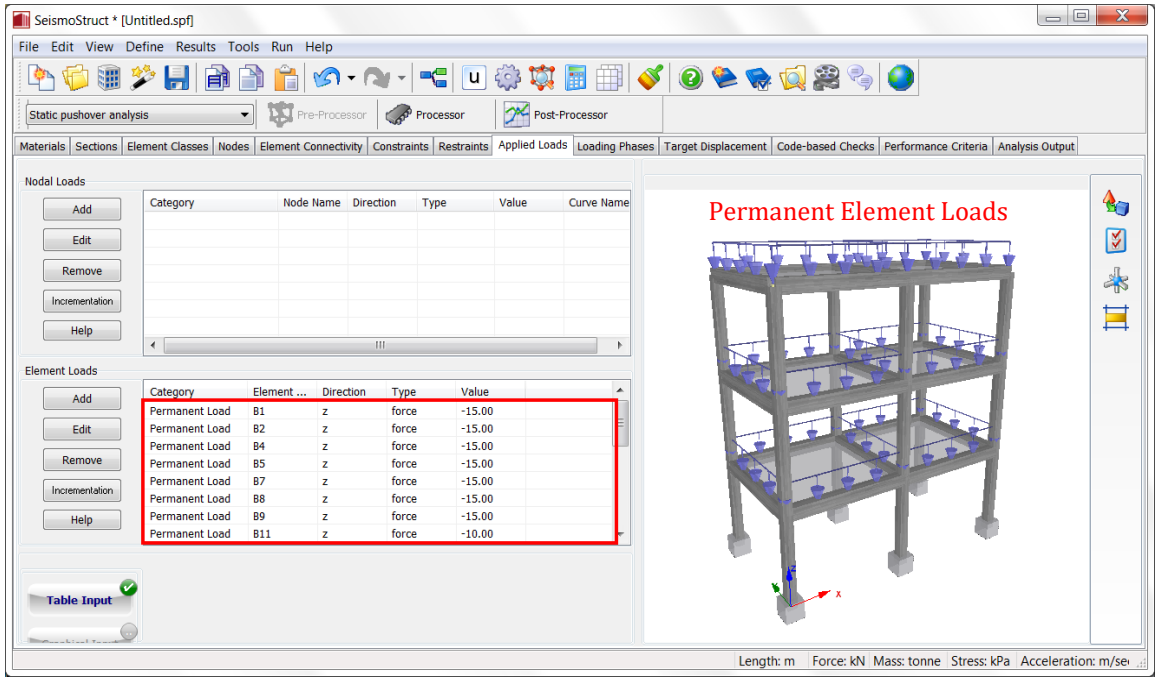


Applied Loads module

As in all other modules, the user is capable of adding new loads and removing/editing existing ones. In addition, a load *incrementation* facility is also available, so as to enable easier generation of new element actions. The user defines element name and load value increments, and these are then employed to automatically generate new element loads through "repetition" of a selected set of already prescribed loads. This facility requires that element names respect the number (e.g. 100) or word+number (e.g. B20) formats.



Load Incrementation



Example of Permanent Loads

NOTE: Gravity loads should be applied downwards, for which reason they always feature a negative value.

Loading Phases

In pushover analysis, the applied loading usually consists of permanent gravity loads in the vertical (z) direction and incremental loads in one or both transversal (x & y) directions. As discussed in *Appendix B > Static pushover analysis*, the magnitude of increment loads P_i at any given analysis step i is given by the product of its nominal value P_0, defined by the user in the **Applied Loads**, and the load factor λ at that step:

$$P_i = \lambda_i P_0$$

The manner in which the load factor λ is incremented throughout the analysis or, in other words, the loading strategy adopted in the pushover analysis, is fully defined in the **Loading Phases** module, where an unlimited number of loading/solution stages can be defined by applying different combinations of the three distinct pushover control types available in SeismoStruct, indicated below.

It is noteworthy that the incremental loading P may consist of forces or displacements, thus enabling for both force- and displacement-based pushover to be carried out. Clearly, for most cases, application of

forces will be preferred to the employment of displacement incremental loads, since constraining the deformation of a structure to a predefined shape may conceal its true response characteristics (e.g. soft-storey), unless the more advanced adaptive pushover analysis type is employed. For this reason, the most common loading strategy in non-adaptive pushover analysis is force-based pushover with response control, described below:

- Load control phase
- Response control phase
- Automatic response control phase

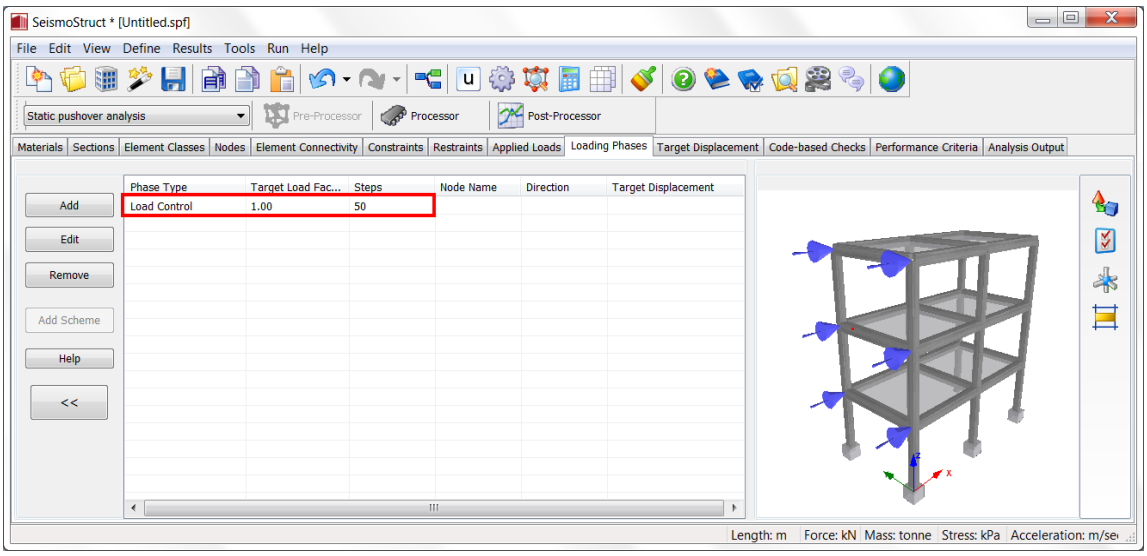
NOTE 1: Users may take advantage of the *Add Scheme* button to apply typical loading phases schemes that will work for the majority of cases. Note, however, that no loading phases should be already defined, in order for this facility to be available.

NOTE 2: It is highlighted again that an unlimited number of loading/solution strategies can be defined, by applying different combination of the three distinct load phase types available. For instance, the user may wish to: (a) apply the pushover loads in two or more load control phases, using a different incremental step for each of those (e.g. larger step in the pre-yield stage, smaller step in the inelastic range), (b) employ several phases to push a 3D model, first in one direction, then in the other, then back in the first one, and so on, (c) carry out cyclic pushover analysis, pushing and pulling the structure in successive cycles (the Static time-history analysis modality is however better tailored for such cases).

NOTE 3: Even in those cases where no permanent loading is present, it might result handy to apply a nil load vector somewhere in the structure, so that the initial permanent loads step is carried out and hence the pushover curve is "forced" to start from the origin, which renders it slightly "more elegant".

Load control phase

In this type of loading/solution scheme, the user defines the target load multiplier (the factor by which all nominal loads, defined in the **Applied Loads** module, are multiplied to get the target loads) and the number of increments in which the target load vector is to be subdivided into, for incremental application.



Example of Loading Phase – Load Control

The load factor λ, therefore, varies between 0 and the target load multiplier value, with an initial step increment $\Delta\lambda_0$ that is equal to the ratio between the target load multiplier and the number of increments. The value of $\Delta\lambda_0$ is changed only when the solution at a particular step fails to converge, in which case the load factor increment is reduced until convergence is reached, after which it tries to return to its initial value (refer to automatic step adjustment for further details). The phase finishes when the target loading is reached or when structural or numerical collapse occurs.

If the user defined the incremental loads as forces, then a force-controlled pushover is carried out, with the load factor being used to scale directly the applied force vector, until the point of peak capacity. If the user wishes also to capture the post-peak softening behaviour of the structure, then a response or automatic response phase needs to be added to the load control one (the program will automatically switch from one phase to the other). This type of loading/solution strategy is employed when the user needs to control directly the manner in which the force vector is incremented and applied to the structure.

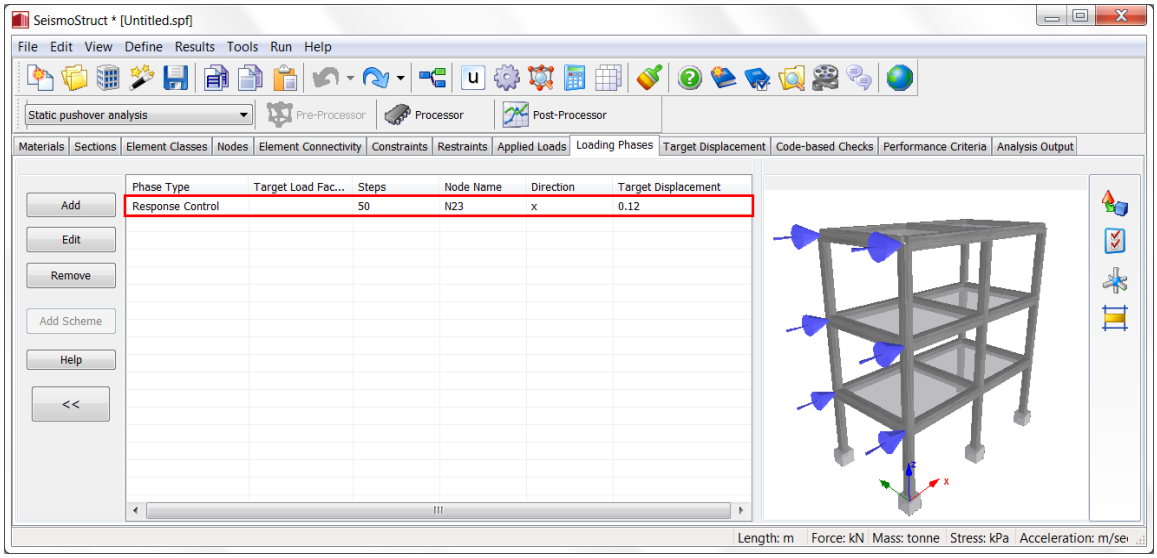
If, on the other hand, the user defined the loads as displacements, then a displacement-controlled pushover is considered instead, with a displacement load vector incrementally applied to the structure. This loading/response strategy is employed when the user wishes to have direct control over the deformed shape of the structure at each stage of the analysis. Its application, however, is usually not recommended, since constraining the deformation of a structure to a predefined shape may conceal its true response characteristics (e.g. soft-storey), unless the more advanced adaptive pushover analysis type is employed.

NOTE 1: When one force-based load control phase (+ one response control phase) is employed, the distribution of force-displacement curve points usually results uneven, with higher density in the pre-peak part, where to relatively large force increments correspond to small displacement steps, and lower point concentration in the post-peak range, where to very small force variations may correspond large deformation jumps. To solve or mitigate such problem a response control phase should be used.

NOTE 2: When the applied incremental loads are displacements, the program will automatically adjust the value of the first increment so that the latter added to the gravity loads-induced displacement equals the initially envisaged target displacement value at the end of the first increment. In other words, if the user wanted, for instance, to impose a 200 mm floor displacement applied in 100 increments, and if the gravity loads would cause a horizontal displacement of 0.04mm, then the displacement load increments would be 1.96, 2.0, 2.0, ..., 2.0. This adjustment will, however, occur only in those cases where the gravity loads-induced displacement is lower than the envisaged first horizontal loads increment; if this condition that does hold (e.g. disp_gravt=2.07, in the example above), then the displacement increments will all be identical and equal to (200-2.07)/100=1.9793, clearly a much less "elegant" figure.

Response control phase

In this type of loading/solution scheme, it is not the load vector that is controlled, as in the load control case, but rather the response of a particular node in the structure. Indeed, when setting a response control phase, the user is requested to define the node and corresponding degree-of-freedom that is to be controlled by the algorithm, together with the target displacement at which the analysis is to be terminated. Moreover, the number of increments, in which the target displacement is to be subdivided into for incremental application, should be specified.



Example of Loading Phase – Response Control

The load factor λ, therefore, is not directly controlled by the user but is instead automatically calculated by the program so that the applied load vector $P_i = \lambda_i P_0$ at a particular increment i corresponds to the attainment of the target displacement at the controlled node at that increment. When the solution at a particular step fails to converge, the initial displacement increment is reduced until convergence is reached, after which it tries to return to its initial value (refer to automatic step adjustment for further details). The phase finishes when the target displacement is reached or when structural or numerical collapse occurs.

With this loading strategy, it is possible to (i) capture irregular response features (e.g. soft-storey), (ii) capture the softening post-peak branch of the response and (iii) obtain an even distribution of force-displacement curve points. For these reasons, this type of loading/solution phase usually constitutes the best option for carrying out non-adaptive pushover analysis.

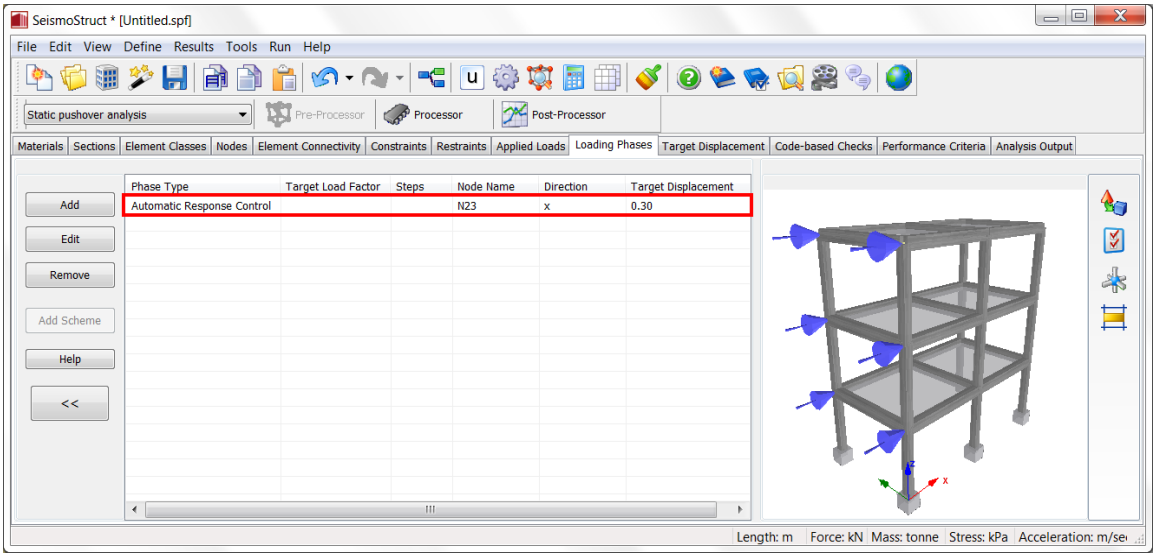
NOTE 1: Response control can be employed in conjunction with displacement incremental loads.

NOTE 2: Response Control does not allow the modelling of snap-back and snap-through response types [e.g. Crisfield, 1991], observed in structures subjected to levels of deformation large enough to cause a shift in their mechanism of deformation and response. For such extreme cases, the employment of Automatic Response Control is required.

NOTE 3: The program will automatically adjust the value of the first increment so that the latter added to the gravity loads-induced displacement equals the initially envisaged target displacement value at the end of the first increment. In other words, if the user wanted, for instance, to impose a 200 mm top floor displacement applied in 100 increments, and if the gravity loads would cause a horizontal displacement of 0.04mm, then the displacement load increments would be 1.96, 2.0, 2.0, ..., 2.0. This adjustment will, however, occur only in those cases where the gravity loads-induced displacement is lower than the envisaged first horizontal loads increment; if this condition that does hold (e.g. disp_gravt=2.07, in the example above), then the displacement increments will all be identical and equal to (200-2.07)/100=1.9793 (clearly a much less "elegant" figure).

Automatic response control phase

This type of loading/solution scheme, adapted from the work of Trueb [1983] and Izzuddin [1991], differs from the response control type only in the fact that it is the program that automatically chooses which nodal degree-of-freedom to control during the analysis and the displacement increment to apply at each analysis step, depending on the convergence characteristics at each analysis step. The user, on the other hand, is asked to define the node, degree-of-freedom and respective target displacement at which the analysis will be completed.



Example of Loading Phase – Automatic Response Control

The program uses the "target degree-of-freedom" as the first control entity for the analysis, changing it whenever another nodal degree-of-freedom with a higher rate of nominal tangential translational response (i.e. larger displacement variation between two consecutive steps) is found. In this manner, it results not only possible for highly geometrically nonlinear snap-back and snap-through responses [e.g. Crisfield, 1991] to be accurately predicted, but also to obtain analyses' solution in the minimum amount of time, rendering this type of loading/solution phase the preferred option for obtaining expeditious and accurate estimations of the force and displacement capacity of structures.

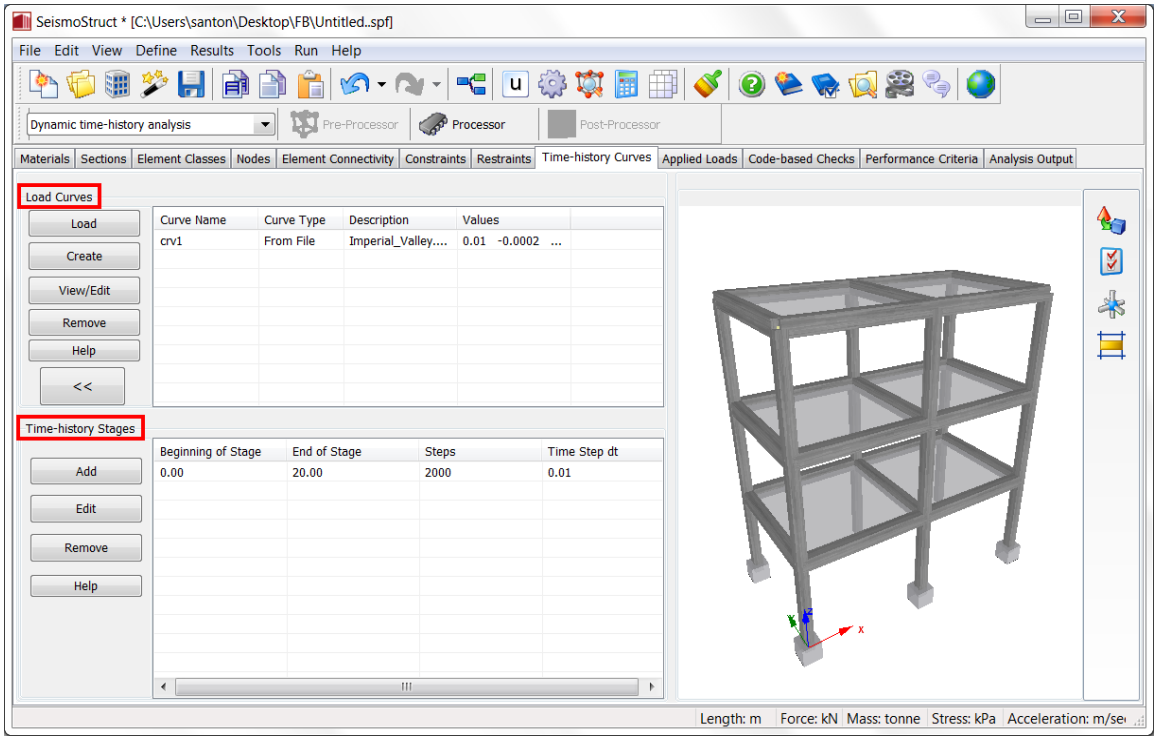
NOTE 1: When carrying out automatic response control pushover analysis on non-symmetric models, it may happen that the program starts applying the load in the 'negative' direction, effectively pulling the structure backwards, rather than pushing it forwards. This occurs when the non-symmetric structure being analysed proves to be more flexible/deformable in 'pulling' rather than 'pushing', a feature that the automatic response algorithm cannot overlook. If users do wish to force the structure to deform in a different direction, then they should start the pushover analysis with load or response control phases, to initiate the deformation in the desired direction, after which they might change to automatic response control, since the already displaced degrees-of-freedom will be inevitably selected as the control ones.

NOTE 2: The automatic reduction and increase of the loading step may, on occasions, cause the force-displacement curve points to result very uneven, for which reason the pushover response curve may not always be visually 'adequate'.

Time-history curves

In both static and dynamic time-history analyses, in addition to permanent loads, structures are subjected to transient loads, which may consist of forces/displacements varying in the pseudo-time domain (static time-history loads) or of accelerations/forces that vary in the real time domain (dynamic time-history loads). Whilst the type, direction, magnitude and application nodes of these loads comes defined in the **Applied Loads** module, their loading pattern, that is, the way in which the loads vary in time (or pseudo-time), is given by the time-history curves, defined in the **Time-history Curves** module. The latter comprises two interrelated sections:

- Load curves
- Time-history stages



Time-History Curves module

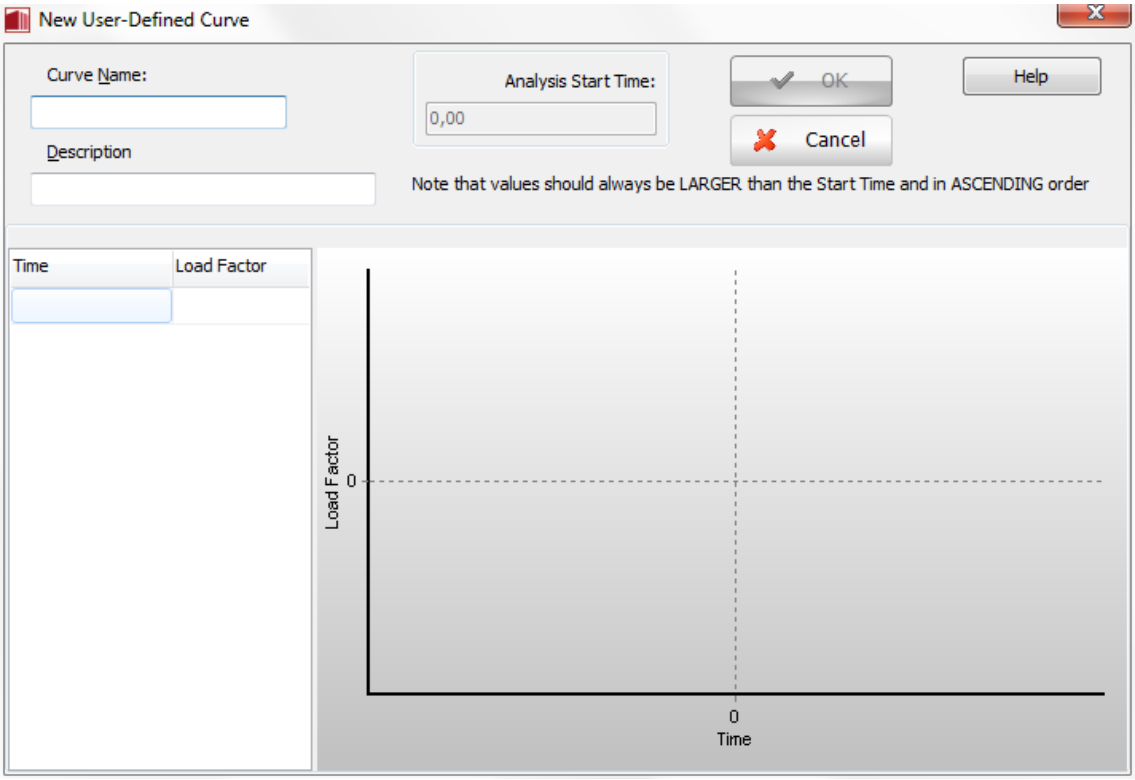
NOTE: Time-history curves provide only the time pattern of the transient loads. Their full absolute magnitude is obtained through the product of time-history ordinates with the Curve Multiplier, defined in the Applied Loads module. This effectively means that time-history curves can be introduced in any given system of units, for as long as a coherent curve multiplier is used (e.g. if an accelerogram is defined in [g] and the system of units adopted by the user requires acceleration values to be defined in mm/sec2, then the corresponding curve multiplier should be 9810).

Load Curves

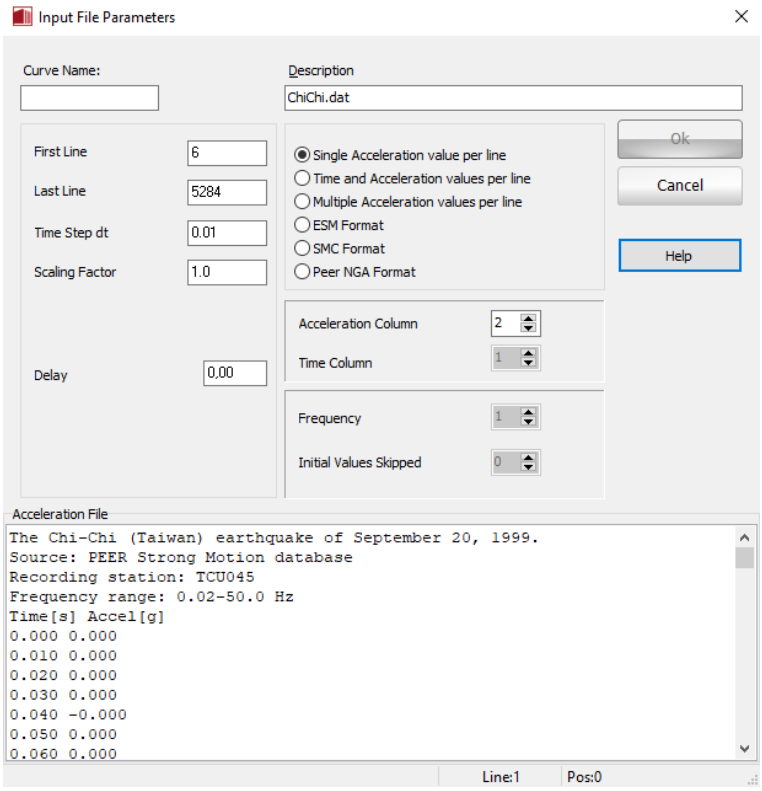
In the **Load Curves** section, the time-history curve is defined either through direct input of the values of time and load pairs (*Create* function) or by reading a text file where the load curve is defined (*Load* function).

IMPORTANT: The text file of the load curve must be in MS-DOS Windows format (i.e. save the file as ANSI (encoding) using the Notepad).

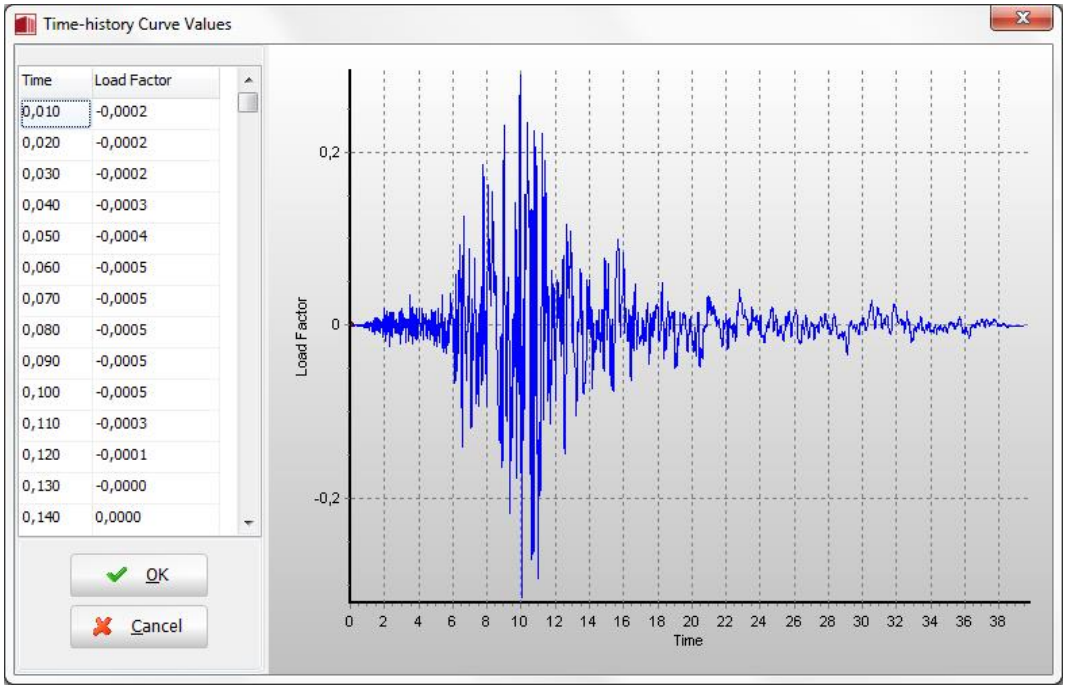
Usually, static time-history analysis is employed to model simple cyclic tests on specimens, in which case the loading curve is fairly simple and users tend to define it directly within SeismoStruct with the *Create* option. In the case of dynamic analysis, on the other hand, the applied curve commonly, though not exclusively (e.g. impact/blast analysis), consists of an accelerogram, with data points found in a text file, which is then loaded into the program with the *Load* option. Nonetheless, any of the two time-history definition options (*Create* and *Load*) can be used for both analysis types.



Load Curves – Create function



Load Curves – Input File Parameters



Load Curves – Time-history Curve Values

The *Analysis Start Time* is the time at which the analysis starts, and is always considered as equal to zero, for which reason all time-history curves must feature time entries larger than 0.0. Further, when time-history curves are to be applied to the structure at different time instants (e.g. asynchronous seismic input, two earthquakes hitting the same structure in succession, etc.), the Delay parameter should be

used to define the time at which a particular time-history, being loaded from a text file, starts being applied to the structure. In other words, there is no need for the user to manually change the time-history data points to introduce a time delay, since the program does it automatically.

Whenever there is some uncertainty with regards to the file loading parameters (time column, acceleration column, first line, last line) to be specified, the user can make use of the *View Text File* facility which permits inspection of the file. After the time-history is loaded, the aforementioned input parameters can still be modified (e.g. if after loading a 5000 lines accelerogram file it is realised that only the first 1000 data points are of interest).

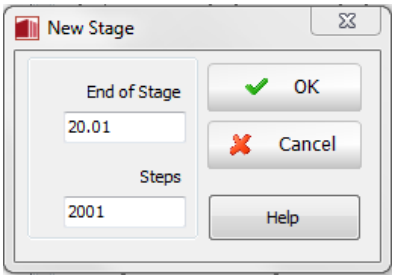
NOTE 1: A maximum number of 260,000 data points may be defined for each curve.

NOTE 2: After loading a time-history curve from a given text file, the latter can be disposed of, since the time-history curve points are saved within the project file itself.

NOTE 3: In order to help users getting started, a set of eight accelerograms, normalised to [g], is provided in the program's installation folder, to where the user is automatically directed whenever he/she presses the Select File button. Users are also referred to online strong-motion databases for access to additional accelerograms.

Time-history Stages

In the **Time-history Stages** section, the user has the possibility of defining up to 20 analysis stages, each of which can be subdivided into a different number of analysis steps, explicitly defined by the user. The program then calculates internally the time-step to be used within a given time-history stage, this being equal to the difference between the end-times of two consecutive time-history stages divided by the number of steps assigned. For the first stage, the difference between its end-time and the *Analysis Start Time* (0.0 secs) is used.



Adding new stage

In the majority of common applications, a single analysis stage is employed. However, there are cases where a user may wish to employ different time-steps at different stages of the analysis (e.g. a free vibration stage is introduced between two successive earthquakes being applied to a given structure or a yield (easy convergence, large time-step can be used) and collapse (difficult convergence, small time-step must be employed) static time-history curves are applied to a model), in which case the possibility of defining more than one analysis stage becomes useful.

Adaptive pushover parameters

In Adaptive pushover, loads are applied to the structure in a manner that is largely similar to the case of conventional pushover. For this reason, users who are interested in using adaptive pushover are strongly advised to first consult the *Loading Phases* section, where the loading application procedure for

conventional pushover is described. The latter should be considered as applicable to the adaptive pushover cases, noting however the following differences:

- In adaptive pushover, it is required that the inertia mass of the structures is modelled so that eigenvalue analysis, employed in the updating of the loading vector, may be carried out. Further, and for the case of force-based adaptive pushover only, it is necessary for the mass to be adequately distributed throughout the nodes where the incremental loads are to be applied, so that the incremental forces (obtained through the product of mass and acceleration) may be calculated. (for displacement-based pushover this is not necessary, given that the displacement profiles are obtained directly from the eigenvalue analyses)
- Although it is permitted to use different nominal values for the loads at different nodes, as in conventional pushover, it is strongly advisable that these incremental loads have equal nominal values (constant load profile) so that the load applied at every node is fully determined by the modal characteristics of the structure and spectral shape used.
- The *Adaptive Load Control* and *Adaptive Response Control* loading/solution procedures are used in substitution of the load control and response control phases. Their input and functionality are identical, noting however that only one adaptive phase (load or response control) can be applied in adaptive pushover, contrary to conventional pushover analysis where more than one load or response control phases may be simultaneously employed. If users wish to switch from *Adaptive Load Control* to *Adaptive Response Control*, or vice-versa, they must first delete whichever of these two phases has already been defined so that the alternative option is made available on the *Add New Phase* dialog box.

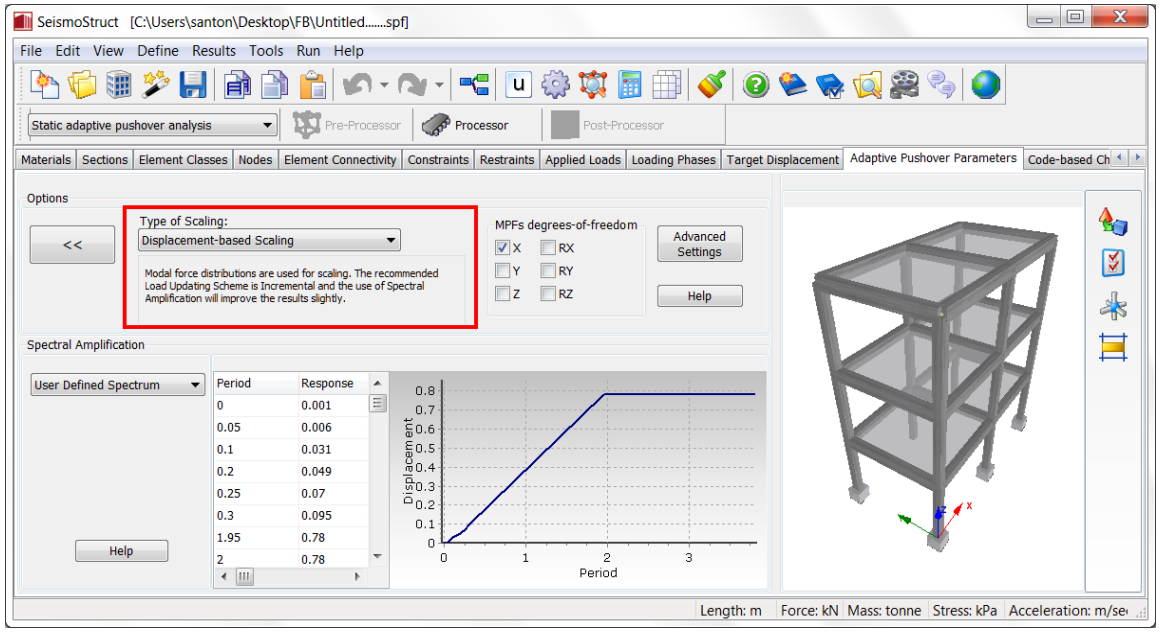
Being an advanced static analysis method, adaptive pushover requires the definition of a number of additional parameters, as included in the **Adaptive Parameters** module. These parameters are:

Type of Scaling

The normalised modal scaling vector, used to determine the shape of the load vector (or load increment vector) at each step, can be obtained using *three distinct types of approaches*:

1. **Force-based Scaling**. Scaling vector reflects the modal force distribution at that step.
2. **Displacement-based Scaling**. Scaling vector reflects the modal displacement distribution at that step.
3. **Interstorey Drift-based Scaling**: scaling vector reflects the modal interstorey drift distribution at that step.

NOTE: The latter cannot be employed in 3D adaptive pushover analyses and requires the nominal lateral displacements to be entered in sequence (the 1st floor load being defined first, followed by the displacement nominal load at level 2, and so on).



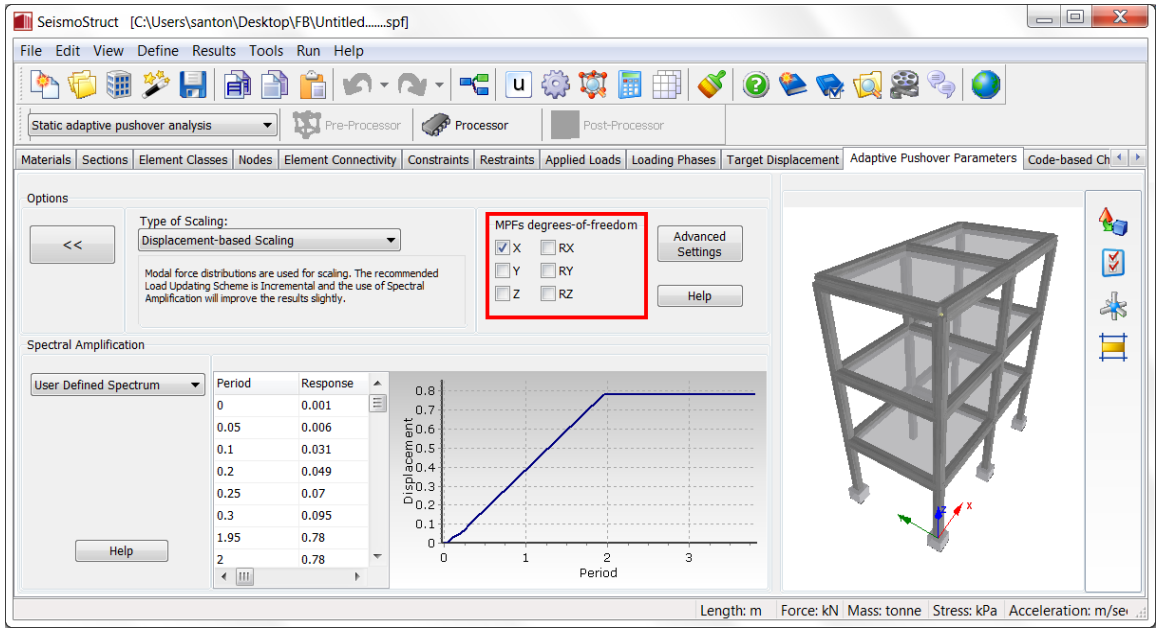
Selection of the type of scaling

MPFs degrees-of-freedom

The user has the possibility of specifying the degrees-of-freedom to be considered in the calculation of the participation factors of the modes (which are then employed in the computation of the modal scaling vector).

For 3D adaptive pushover analysis, it might be convenient for more than one translation degree-of-freedom to be employed (e.g. X & Y) or, instead, for rotation degrees-of-freedom to be used [e.g. Meireles et al., 2006].

In the more common case of 2D analysis, only one translation degree-of-freedom will be chosen, usually X.



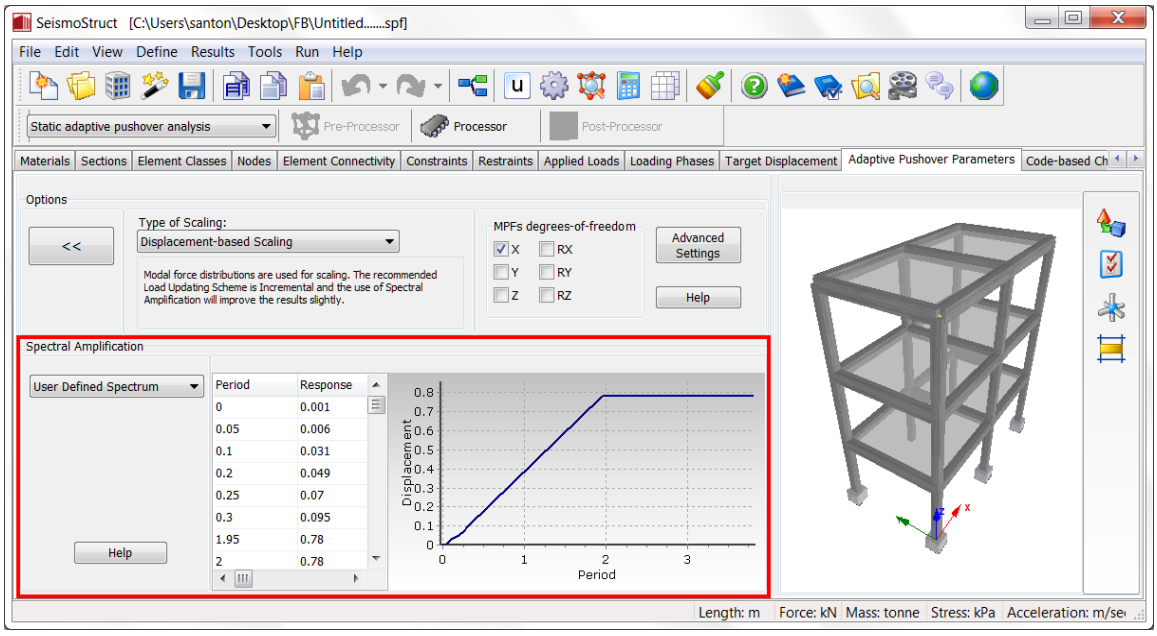
Specification of the MPFs degrees-of-freedom

Spectral Amplification

As previously mentioned, the effect that spectral amplification might have on the combination of the different modal load vector solutions may or may not be taken into account through the choice of one of the three options available within this module:

- **No Spectral Amplification**. The scaling of the load vector distribution profile depends on the modal characteristics of the structure alone, at each particular step.
- **Given Accelerogram**. The user introduces an accelerogram time-history and defines the desired level of viscous damping used by the program to automatically compute an acceleration (when force-based scaling is used) or displacement (when displacement or drift-based scaling is employed) response spectrum (assumed constant throughout the analysis). Note that by default, the resulting response spectrum, as opposed to the accelerogram, is shown to the user. The latter, however, can be visualised through the *Accelerogram* button.
- **User Defined Spectrum**. The pairs of period and response acceleration/displacement values can be directly introduced in an input table by the user. This option is usually employed to introduce code-defined spectra and it is noted that, as in all other SeismoStruct modules, the list of values may be pasted from any other Windows application, as an alternative to direct typing.

NOTE: When running Displacement-based Adaptive Pushover, it is highly recommended, for reasons of accuracy, for Spectral Amplification to be employed. If, for some reason, a user does not have ways to estimate/represent the expected/design input motion at the site in question, then he/she should select Single-Mode analysis in here, so as to run DAP-1st mode (for buildings only).



Spectral Amplification

IMPORTANT: By clicking on the *Advanced Settings* button, the user can define additional parameters to those presented above.

IDA parameters

In Incremental Dynamic Analysis (IDA), structures are subjected to a succession of transient loads, which usually consist of acceleration time-histories of increasing intensity, as described in *Appendix B -> Incremental dynamic analysis*. Therefore, users who are interested in using this type of analysis, are strongly advised to first consult the **Time-history Curves** section, where the loading application procedure for dynamic time-history analysis is described. The latter is fully applicable to IDA cases, noting however that a number of additional parameters, included in the **IDA Parameters** module, need to be defined. These parameters are:

Scaling factors

Each time-history run of an IDA is carried out for a given input motion intensity, defined by the product of the *Scaling Factors* with the accelerogram introduced by the user. Usually, the input motion is incrementally scaled from a low elastic response value up to a large value, corresponding to the attainment of a pre-defined post-yield target limit state.

Fixed and/or variable scaling patterns can be used, either in isolation or in combination. With fixed patterns (*Start-End-Step*), the user defines the start scaling factor, corresponding to the first time-history run, the end scaling factor, corresponding to the last time-history analysis to be carried out, and a scaling factor step which is used to define the evenly spaced intermediate time-history levels. With a variable scaling pattern (*Distinct Scaling Factors*), on the other hand, non-evenly spaced sequences of scaling factors can be used, with the user being required to explicitly define all scaling factors to be considered during the incremental dynamic analysis (unless used in combination with a fixed scaling pattern, in which case only odd non-sequential factors may need to be specified).

Dynamic Pushover Curve

When carrying out Incremental Dynamic Analysis, the user is often interested in obtaining the so-called *Dynamic Pushover Curve* (or IDA envelope), which consists of a plot of peak values of base shear versus maximum values of top, or other, displacement, as obtained in each of the dynamic runs. It is therefore possible to explicitly define which nodes are to be considered in the computation of the maximum relative displacement (difference between the absolute displacement values of the two user-defined nodes, the second of which usually refers to a support node) at each dynamic run.

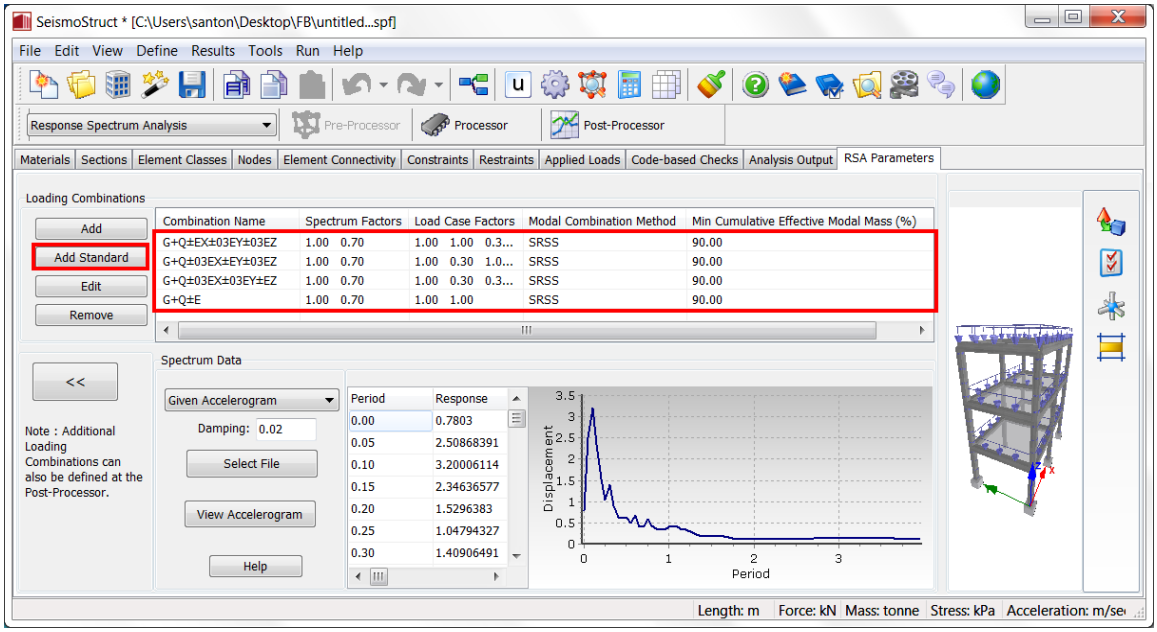
The degree-of-freedom of interest is also explicitly defined by the user, as is the time-window around the maximum drift value within which to find the corresponding peak base shear value (or vice-versa), in case the user is interested in obtaining a curve of corresponding displacement and shear peak values, instead of a curve of not-necessarily correlated pairs of peak displacement and shear values.

NOTE: Usually, the behaviour of structures within their elastic response range can be represented through the use of 2-3 pairs of shear-displacement points, fairly well spaced. In the post-yield region, on the other hand, a finer representation of the dynamic pushover curve may be required. In such cases, users might find useful to employ a combination of both fixed and variable scaling patterns, whereby 2-3 distinct scaling factors are used for the elastic region and then start-end-step range of values is employed for the post-yield response phase.

RSA parameters

Response-spectrum analysis (RSA) is a linear elastic static - (pseudo)dynamic - statistical analysis method which provides the peak values of response quantities, such as forces and deformations, of a structure under seismic excitation, as described in *Appendix B -> Response Spectrum Analysis*.

In RSA users are asked to provide as input the response spectrum and the seismic loading combination(s) for which the RSA will output the results. This spectrum is employed for both the two horizontal (EX, EY) and the vertical (EZ) seismic directions.



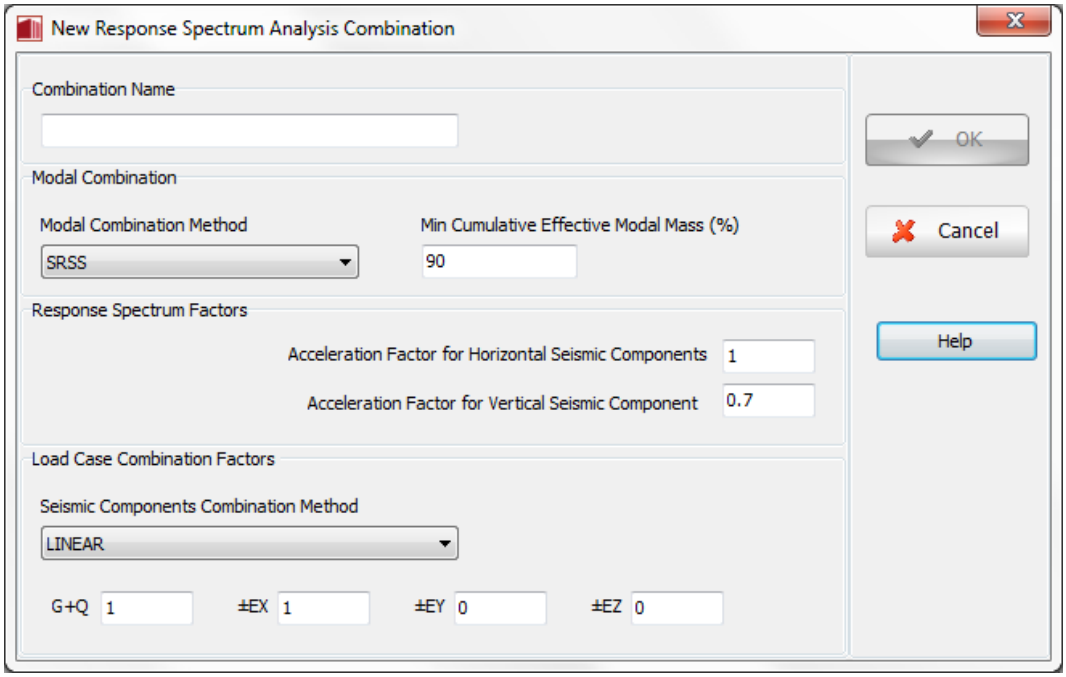
RSA Parameters module

Loading combinations

In the loading combination module different response spectrum factors between horizontal and vertical directions may be defined. The modal combination rule (ABSSUM, SRSS, CQC) should be specified, as well as which modes are to be combined, in terms of accumulation of effective modal mass. User may define a minimum cumulative mass percentage and the program selects the appropriate number of modes that mobilise the largest amount of modal mass, until the target cumulative percentage is reached for every seismic direction.

For each loading case (G, Q, and ±E), users are asked to define the factors for the static gravity or live loading (f_{G+Q}) and the factors of the seismic loading (f_E). Seismic loading directions may be combined linearly ($E = \pm EX \pm EY \pm EZ$) with different factors per direction (f_{EX}, f_{EY}, f_{EZ}) or by the SRSS rule ($E = \pm\sqrt{EX^2+EY^2+EZ^2}$). It is noted that the gravity loads have an explicitly defined algebraic sign, while for the seismic loadings both signs for every direction are taken into account. Consequently, the results of RSA loading combinations in terms of any response quantity are presented as envelopes.

NOTE: Code-defined ready-to-use loading combinations can be defined with the Add Standard Combinations button. The combinations consist of the gravity+live loads, plus 100% of the prescribed seismic forces in one direction and 30% of the prescribed forces in the perpendicular directions, one combination for every seismic direction. Further, a combination of the gravity+live loads plus 100% of the seismic forces is also provided.



Loads combination module

Spectral Data

The response spectrum may be defined directly by the user or may be calculated from a given accelerogram.

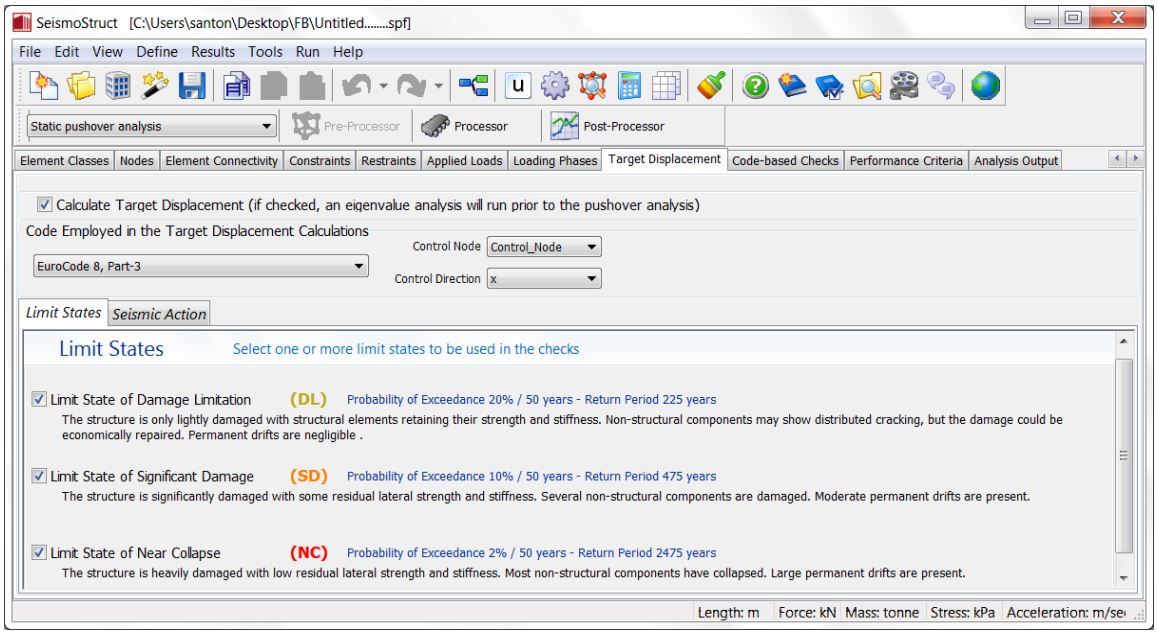
- **Given Accelerogram**. The user introduces an acceleration time-history and defines the desired level of viscous damping to automatically create the spectrum. The resulting response spectrum, as opposed to the accelerogram, is shown to the user. The latter, however, can be visualised through the *Accelerogram* button.
- **User Defined Spectrum.** The pairs of period and response acceleration values can be directly introduced by the user in an input table. This option is usually employed to introduce code-defined spectra and it is noted that, as in all other SeismoStruct modules, the list of values may be pasted from any other Windows application, as an alternative to direct typing.

TARGET DISPLACEMENT

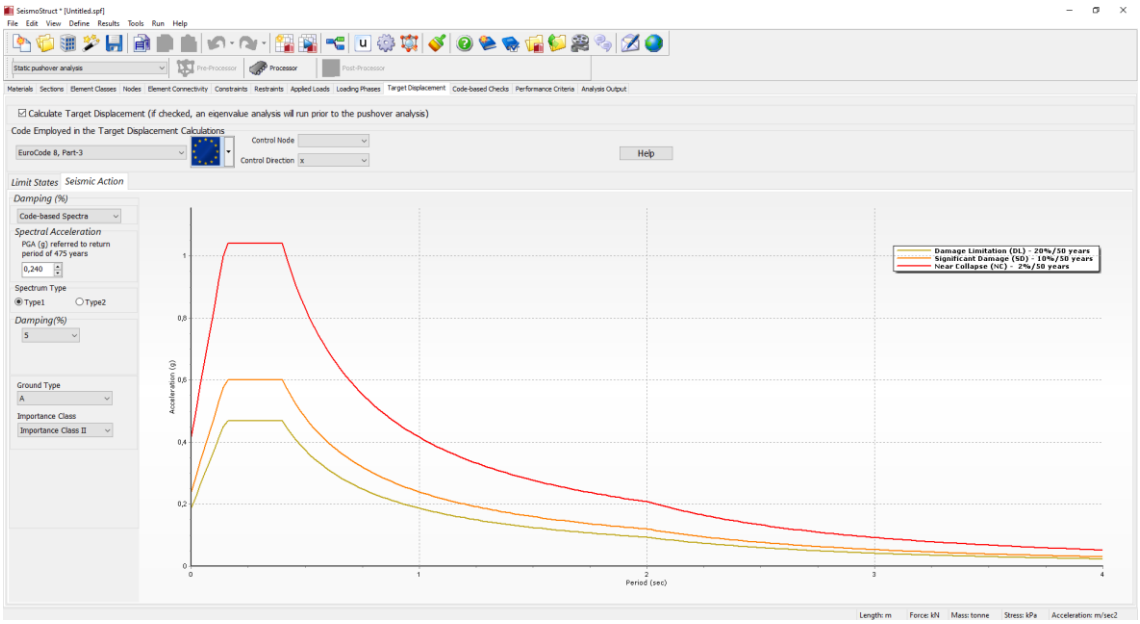
In the case of pushover analysis (conventional or adaptive) users may select the automatic calculation of the target displacement. If the Calculation Target Displacement check-box is selected an Eigenvalue analysis will run prior to the pushover analysis. The parameters below need to be defined in order to calculate the Target Displacement:

1. Code employed; the available options depending on the edition are: Eurocode 8-Part 3 with the majority of National Annexes available, 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). Additional information about the employed Codes may be found in Appendix H – Codes;
2. Control Node and Control Direction; these are automatically assigned if the Building Modeller or the Wizard facility is used;
3. The Limit States (or the Performance Levels in the case of ASCE 41-23 and TBDY and the Performance Objectives in the case of KANEPE), for which the Target Displacement is to be calculated;
4. The elastic response spectrum which 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).

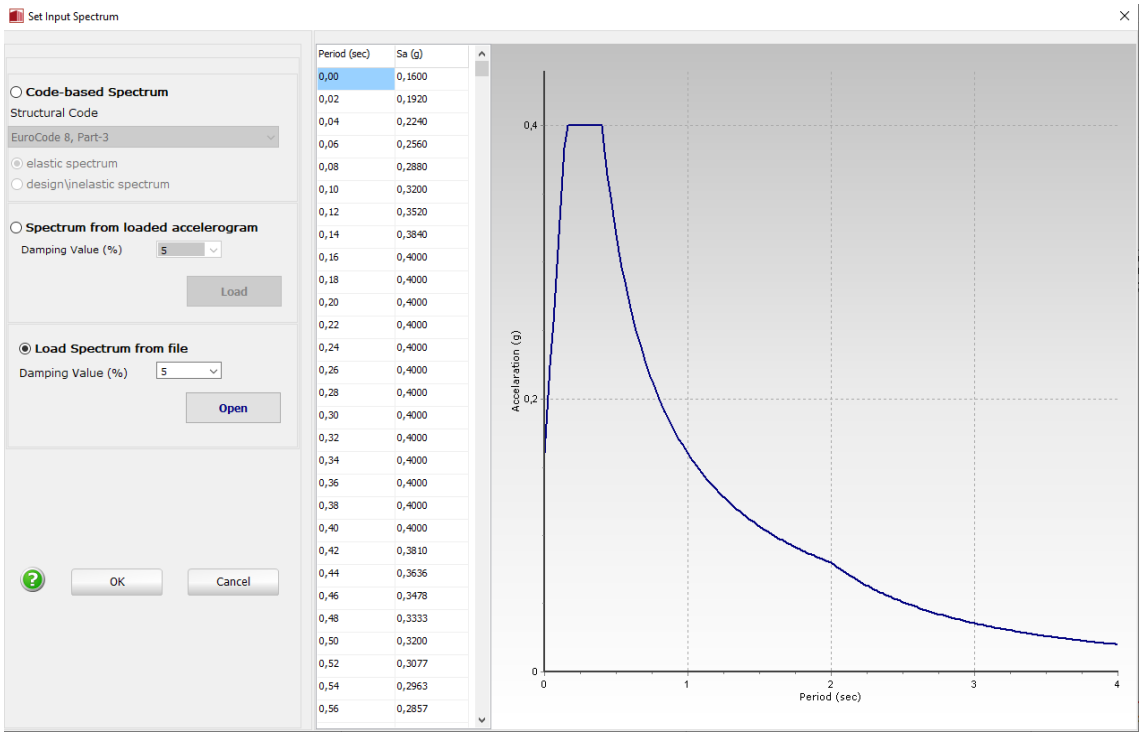
In the case of Code-Based Spectra, users should assign the basic parameters needed for the generation of the spectral shape (i.e. peak ground acceleration, damping, spectrum type, ground type and important class In the case of User Defined Spectra, users can select from a list of 29 spectra defined by various National Codes across the world (Code-Based Spectrum option), they may upload an accelerogram based on which the elastic response spectrum will be calculated (Spectrum from loaded accelerogram option) or they may upload an elastic spectrum from a file (Load Spectrum from file option).



Target Displacement module - Limit States



Target Displacement module - Seismic Action (Code-based Spectra)



Target Displacement module - Seismic Action (User-defined Spectra-Set Input Spectrum)

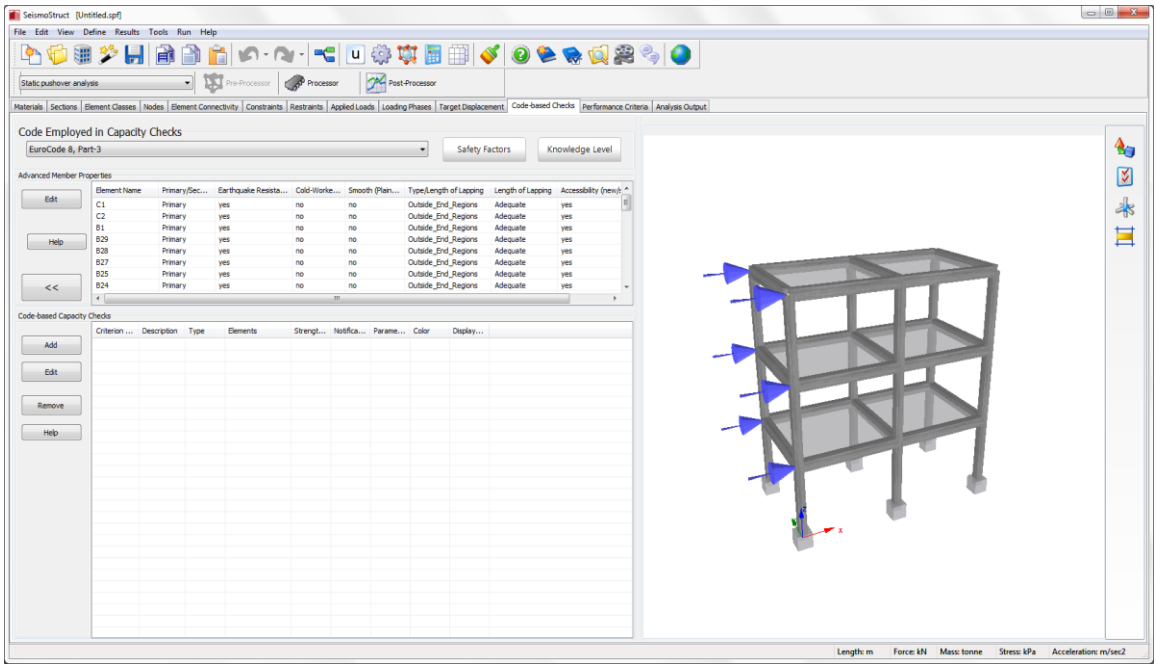
CODE-BASED CHECKS

Herein, the code-based checks to be carried out for the structural members may be selected. Different tabs for Frame Elements and Masonry Elements are available. In order to introduce a code-based check, users need to:

1. Define the Code employed, six options are currently available: Eurocode 8-Part 3 with the majority of National Annexes, 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); additional information about the employed Codes may be found in Appendix H – Codes;
2. Define the values of the Safety Factors and the equations employed in the calculations, when more than one expression are proposed;
3. Select the Knowledge Level that corresponds to the available data on structural configuration;
4. Define the Advanced Member Properties, i.e. all the parameters that characterise the member to be checked, classification (primary or secondary), type and length of lapping, detailing for earthquake resistance etc.;
5. Click the 'Add' button;
6. Introduce the check name;
7. Select the code-based check type (i.e. element chord rotation capacity, element shear capacity or element bending moment capacity for frame elements and masonry shear capacity, compressive force, bending moment or drift for masonry elements) from the drop-down menu;
8. Define the Limit States or the Performance Levels to be used to check the elements;
9. Define the elements to which the check applies to;
10. Define the Strength Degradation of the element, when a given code-based check has been reached. The user can specify the residual strength as a percentage of the capacity, or select to remove the element completely, or to keep it without strength degradation.
11. Define the type of action upon the attainment of each check: (i) stop the analysis and introduce a notification in the analysis log, (ii) pause the analysis and introduce a notification in the analysis log, (iii) leave the analysis undisturbed and introduce a notification in the analysis log, (iv) ignore the occurrence, that is, render the check inactive;

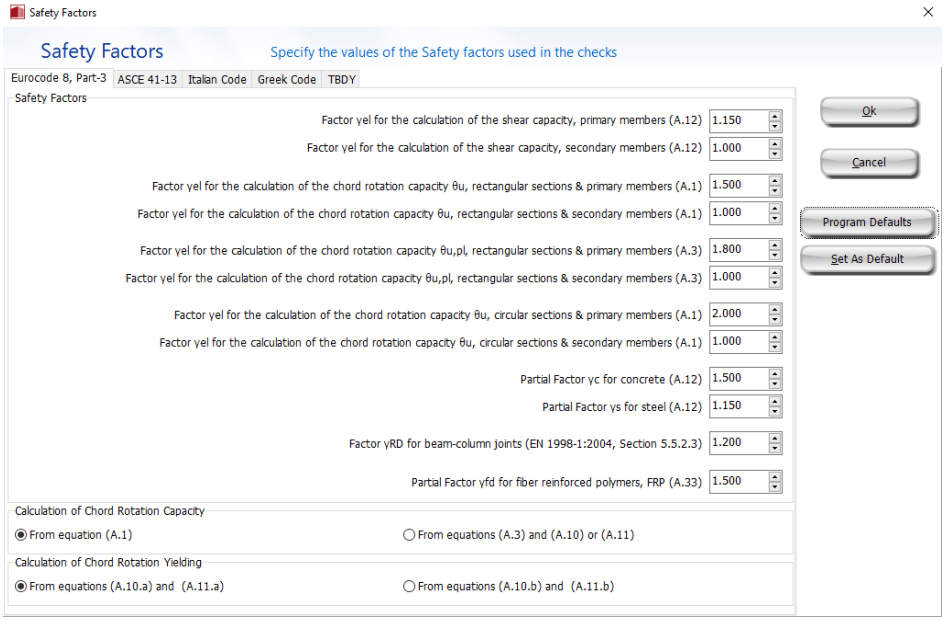
12. Assign a colour to enable graphical visualisation in the **Deformed Shape Viewer** module of the Post-Processor;
13. Select the damage visual effects, in order to enable the graphical visualisation of damage in the **Deformed Shape Viewer** module.

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

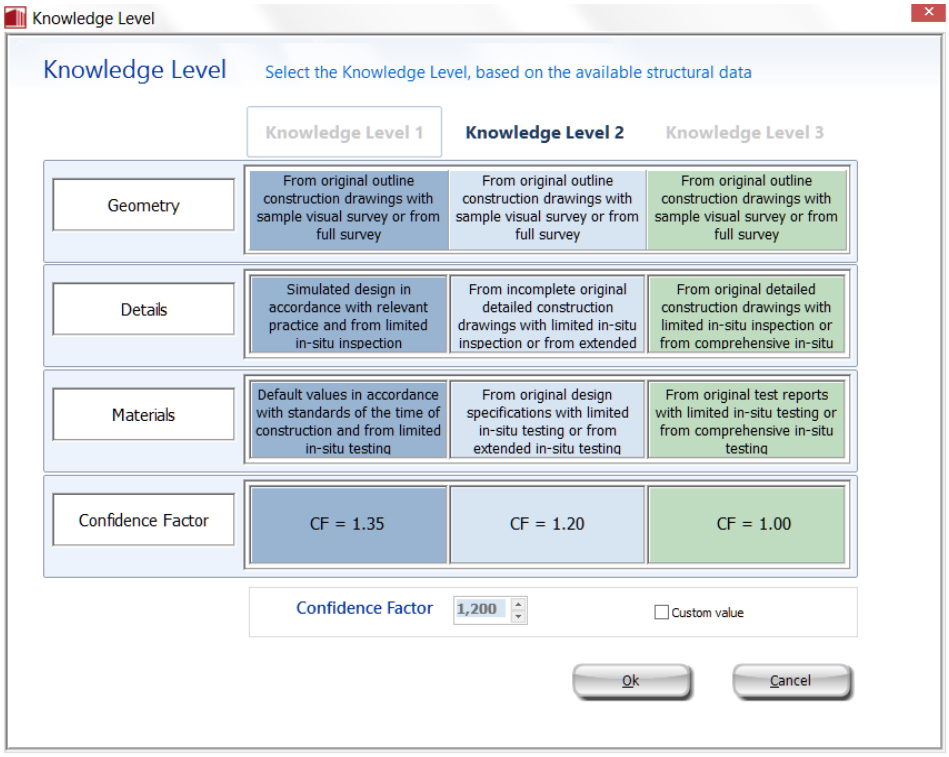


Code-based Checks module

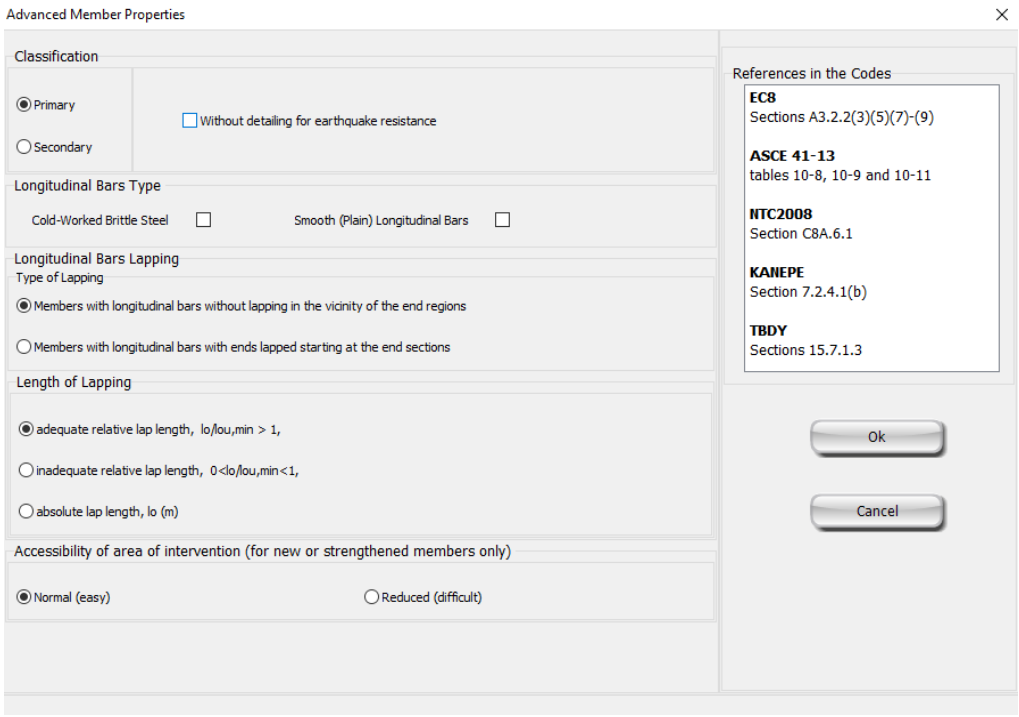
The values of the safety factors and the Code expressions employed may be specified through the dialog box that opens from the corresponding button. It is noted the default values of the safety factors are those defined in Codes.



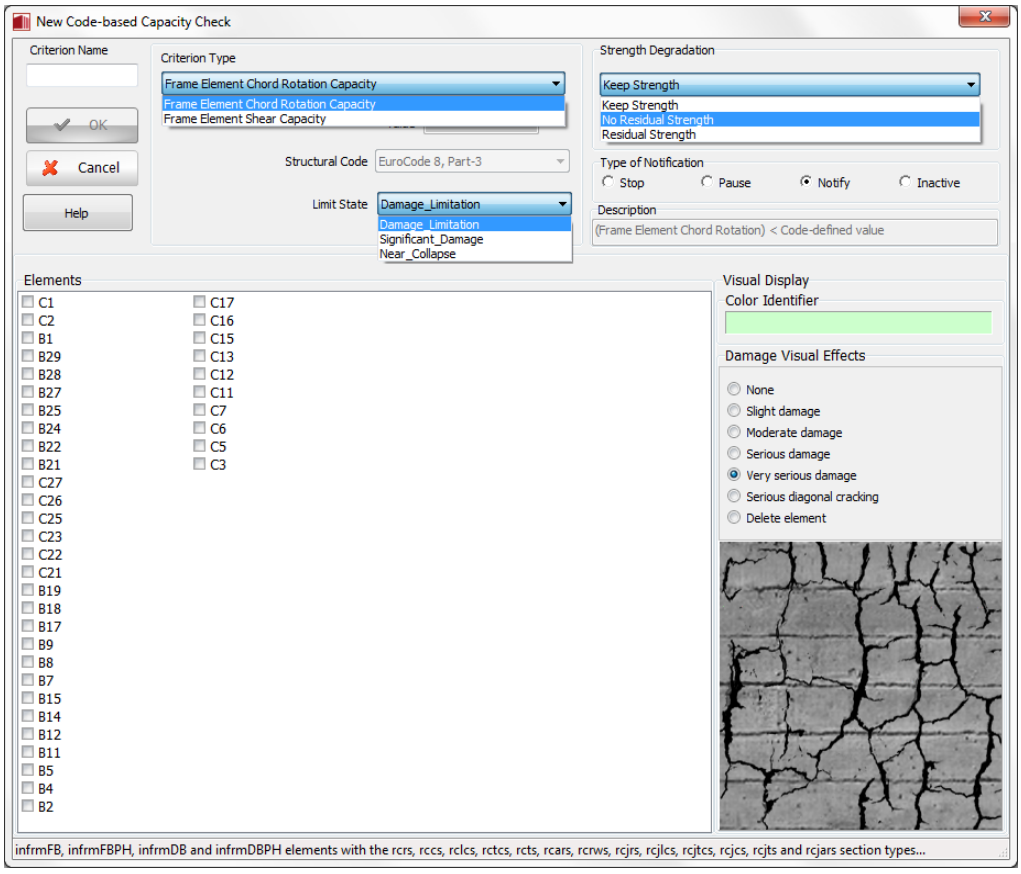
Safety Factors module



Knowledge Level module



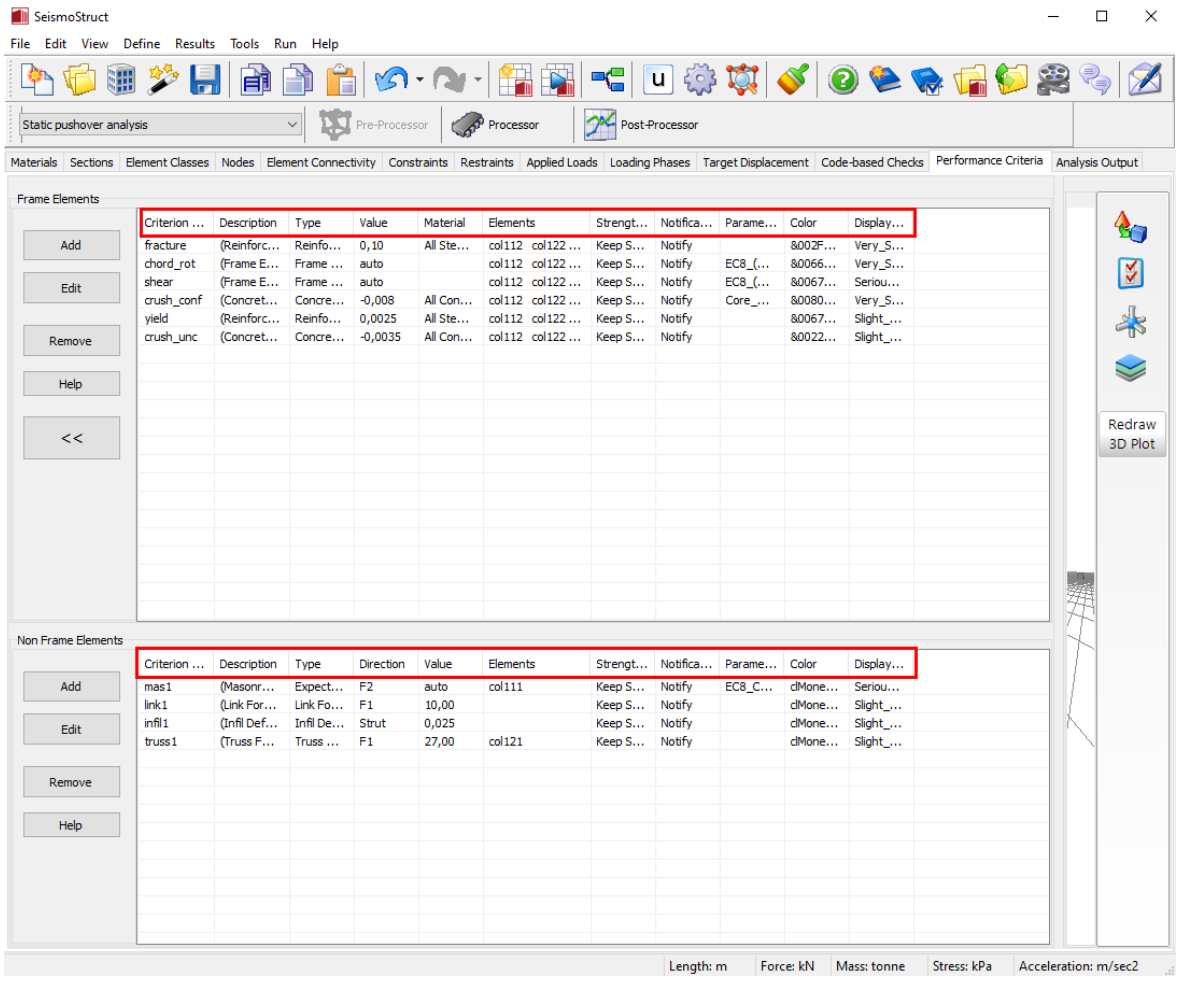
Advanced Member Properties module



Code-based Capacity Checks module

PERFORMANCE CRITERIA

Within the context of performance-based engineering, it is paramount that analysts and engineers are capable of identifying the instants at which different performance limit states (e.g. non-structural damage, structural damage, collapse) are reached. This can be efficiently carried out in SeismoStruct through the definition of **Performance Criteria**, whereby the attainment of a given threshold value of material strain, section curvature, element chord-rotation and/or element shear, element force/moment, element deformation, element drift, etc. during the analysis of a structure is automatically monitored by the program. Different areas for Frame Elements and Non Frame Elements performance criteria definition are available.



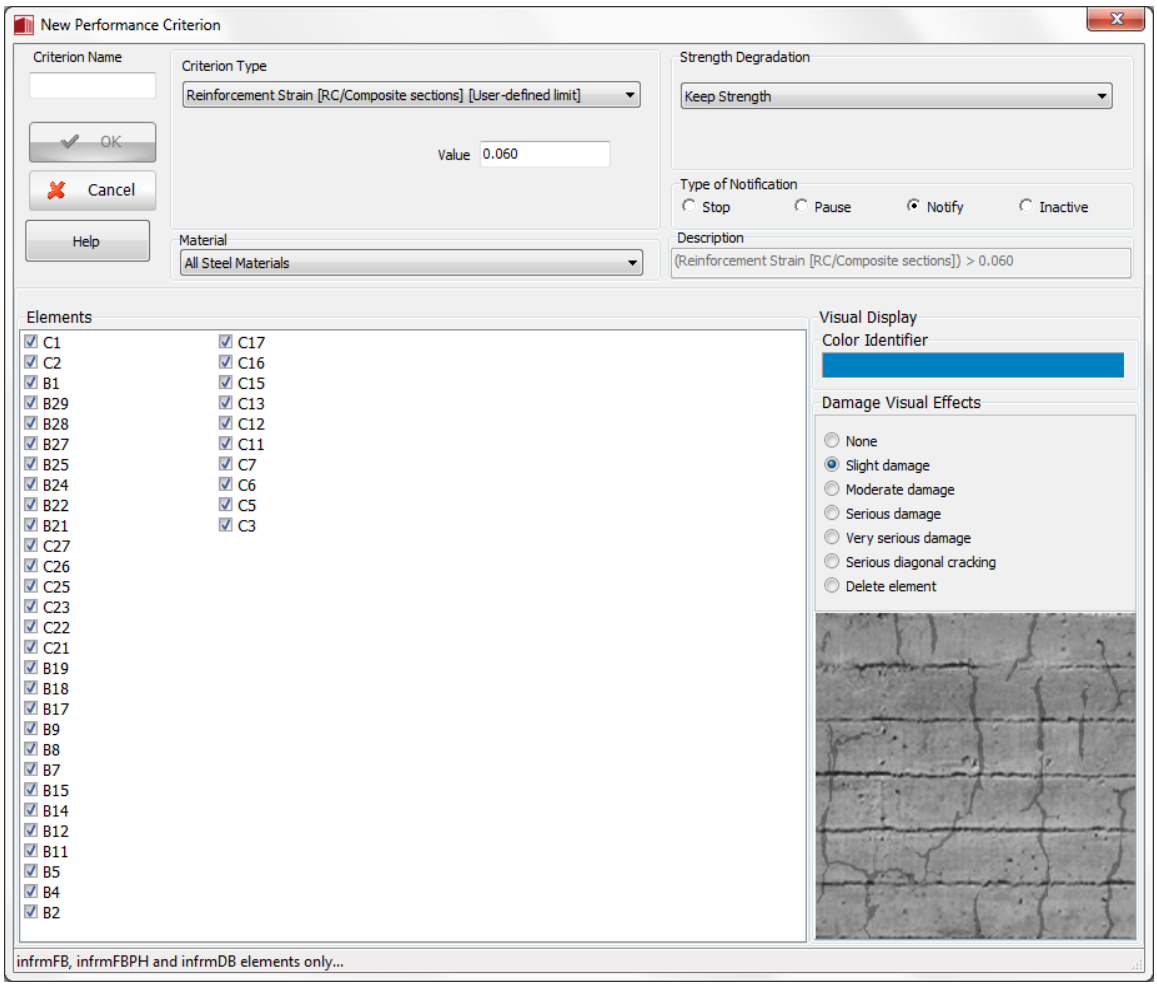
Performance Criteria module

IMPORTANT: Introduction of Performance Criteria checks during the analysis does induce a slight increase in its running time, for obvious reasons.

In order to introduce a given structural performance check, users need to:

1. Define the criterion name;
2. Select the criterion type (i.e. the response quantity to be controlled: material strain, section curvature, element chord-rotation, element shear, element chord rotation capacity or element shear capacity for frame elements and element force/moment, deformation, drift, shear force, compressive force and bending moment for non frame elements depending on the element type) from the drop-down menu;

3. Set the value at which the performance criterion is reached, in the case of criteria with user-defined limit;
4. Select the equation for the calculation of the limit value, in the case of criteria with automatically-defined limits; additional information about the equations used herein may be found in Appendix H – Codes;
5. Define the elements to which the criterion applies to (if a strain criterion has been selected, users have to select a material from the drop-down menu before defining the elements);
6. Define the Strength Degradation of the element, when a given performance criterion has been achieved. The user can specify the residual strength as a percentage of the capacity, or select to remove the element completely, or to keep it without strength degradation.
7. Define the type of action upon the attainment of each criterion: (i) stop the analysis and introduce a notification in the analysis log, (ii) pause the analysis and introduce a notification in the analysis log, (iii) leave the analysis undisturbed and introduce a notification in the analysis log, (iv) ignore the occurrence, that is, render the criterion inactive;
8. Assign a colour to enable graphical visualisation in the **Deformed Shape Viewer** module of the Post-Processor;
9. Select the damage visual effects, in order to enable the graphical visualisation of damage in the **Deformed Shape Viewer**.



Selection of the Criterion Type

NOTE: Users should be careful when defining strength degradation, since such choices may lead to numerical instabilities.

Criterion Type

The type of criteria to be used does clearly depend on the objectives of the user. However, within the context of a fibre-based modelling approach, such as that implemented in SeismoStruct, *material strains* do usually constitute the best parameter for identification of the performance state of a given structure. The available criteria on material strains are:

- **Cracking of structural elements.** It can be detected by checking for (positive) concrete strains larger than the ratio between the tension strength and the initial stiffness of the concrete material. [typical value: +0.0001];
- **Spalling of cover concrete**. It can be recognised by checking for (negative) cover concrete strains larger than the ultimate crushing strain of unconfined concrete material. [typical value: -0.002];
- **Crushing of core concrete**. It can be verified by selecting the "Check the Core Only" check-box and checking for (negative) core concrete strains larger than the ultimate crushing strain of confined concrete material. [typical value: -0.006];
- **Yielding of steel**. It can be identified by checking for (positive) steel strains larger than the ratio between yield strength and modulus of elasticity of the steel material. [typical value: + 0.0025];
- **Fracture of steel.** It can be established by checking for (positive) steel strains larger than the fracture strain. [typical value: +0.060].

Alternatively, or in addition, section curvatures and/or chord-rotations can readily be employed in the verification of a myriad of performance limit states, in which case users should refer to available literature for guidance on curvature/rotation values to be employed [e.g. Priestley, 2003]. Further, it is also feasible to monitor the shear values of frame elements, with the definition of one or more shear threshold values.

Finally, **chord rotation yielding, chord rotation capacity** and element **shear capacity** checks for frame elements can be introduced, whereby the program automatically calculates the capacity of the elements during the analysis, according to the selected equation of the available Codes (Eurocodes, ASCE 41-23, NTC-18, KANEPE and TBDY), and checks it against the corresponding demand. Elements' **Force/moment, displacement** and **Drift** checks for non frame elements can be introduced, whereby the program automatically calculates the capacity of the elements during the analysis, according to the selected equation of the available Codes (Eurocodes, ASCE 41-23 and NTC-18), and checks it against the corresponding demand or user-defined values.

NOTE 1: In the Performance Criteria where only positive values are allowed, the checks are carried out against the absolute value of the response quantity for the demand. Whereas, in the Performance Criteria where both positive and negative values are defined, the check is carried out against the signed value of the response quantity, and different values for the positive and negative values are allowed. In the latter case, if users introduce a positive criterion value, the program will automatically consider a "larger than" performance check. Conversely, if a negative criterion value is defined, the program will automatically activate a "smaller than" performance check.

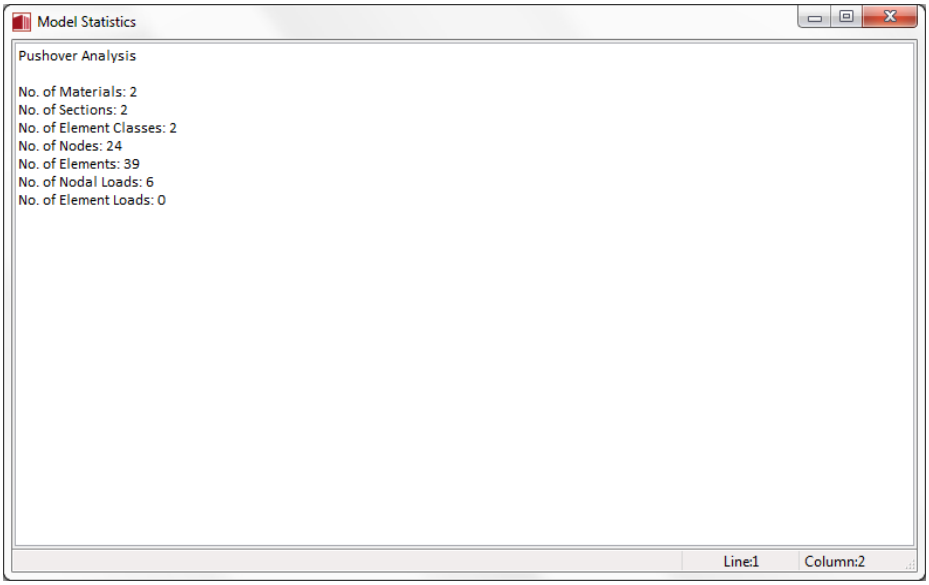
NOTE 2: Strain and curvature performance checks are carried out at the Integration Sections of the selected elements.

NOTE 3: Performance Criteria can only be set to control the response of inelastic frame elements. The latter, however, may always be defined with an elastic material, which effectively means that performance criteria can also be applied to members whose response is elastic.

NOTE 4: Mean material values without safety or confidence factors are used in the automatic calculation of the elements' capacity, i.e. in the case of Performance Criteria with automatically-defined limit.

MODEL STATISTICS

The function 'Model Statistics', available from the program menu (*View > Model Statistics*) or by clicking on , allows users to view a summary of the model input data.

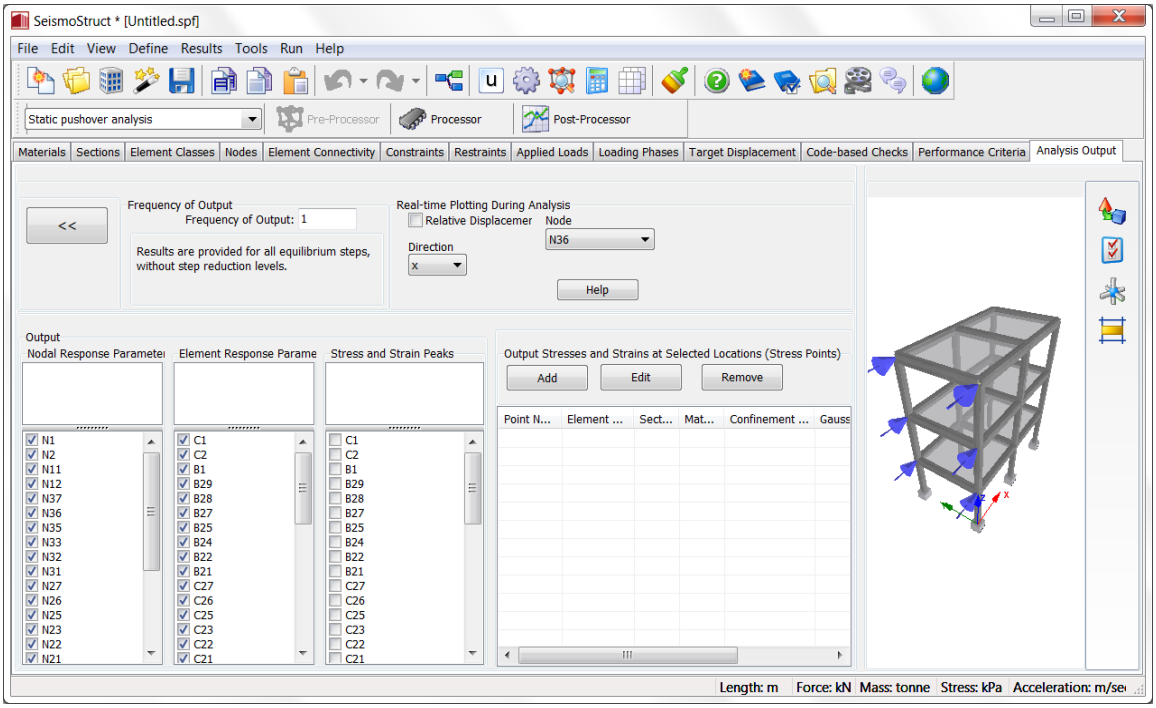


Model Statistics function

ANALYSIS OUTPUT

Being a fibre analysis program, SeismoStruct computes and outputs a very large number of response parameters (e.g. strains, stresses, curvatures, internal member forces, nodal displacements, etc.). This may give rise to two main inconveniencies: (i) user difficulty in post-processing the results and assessing the different levels of performance of the structure and (ii) very large result files (up to 50Mb or more, especially when dynamic analysis is run on large models).

In the majority of cases, users will make use of only a fraction of the wealth of results that can be obtained from SeismoStruct, since it is common for the response of a limited selected number of nodes and/or elements to provide sufficient information on the performance and response of the structure being analysed. Therefore, in the **Analysis Output** module, users are given the possibility to trim down their analysis output to the necessary minimum, thus reducing both hard-drive consumption as well as post-processing time and effort.



Analysis Output module

This can be achieved through the following output settings:

Frequency of Output

If a frequency value equal to zero is adopted, then output is provided at all analysis steps where equilibrium has been reached, including those corresponding to step reduction levels. If a frequency value equal to unity is used instead, then step reduction level output is omitted. This is the default behaviour, since users are usually interested in obtaining results that are in correspondence with the initial number of increments/steps that have been defined in pre-processing. However, if the latter is not the case (e.g. the analysis loading has been split into a very large number of increments just to ease convergence), then a frequency value n larger than unity can be employed, with output being provided at every n equilibrated steps.

Output Nodal Response Parameters

Users can specify the nodes for which output of nodal response parameters (support forces, displacements, velocities and accelerations) will be provided. The user may select all or none of the nodes by right-clicking and choosing *Select All* or *Select None* from the popup menu that appears. Pre-assigned node groups can also be used for easier selection.

NOTE: If not all nodes have been selected for output, the deformed shapes of the structural model cannot be plotted in the Post-Processor.

Output Element Forces Parameters

Users can specify the elements for which output of internal forces (axial/shear forces and bending/torsional moments) will be provided. The user may select all or none of the elements by right-clicking and choosing *Select All* or *Select None* from the popup menu that appears. Pre-assigned element groups can also be used for easier selection.

Output Stress/Strain peaks and Curvature

Users can specify the elements for which output of curvatures and stress/strain peak values (maxima and minima) will be provided (note that such output refers to the Integration Sections of inelastic frame elements). The user may select all or none of the elements by right-clicking and choosing *Select All* or *Select None* from the popup menu that appears. Pre-assigned element groups can also be used for easier selection.

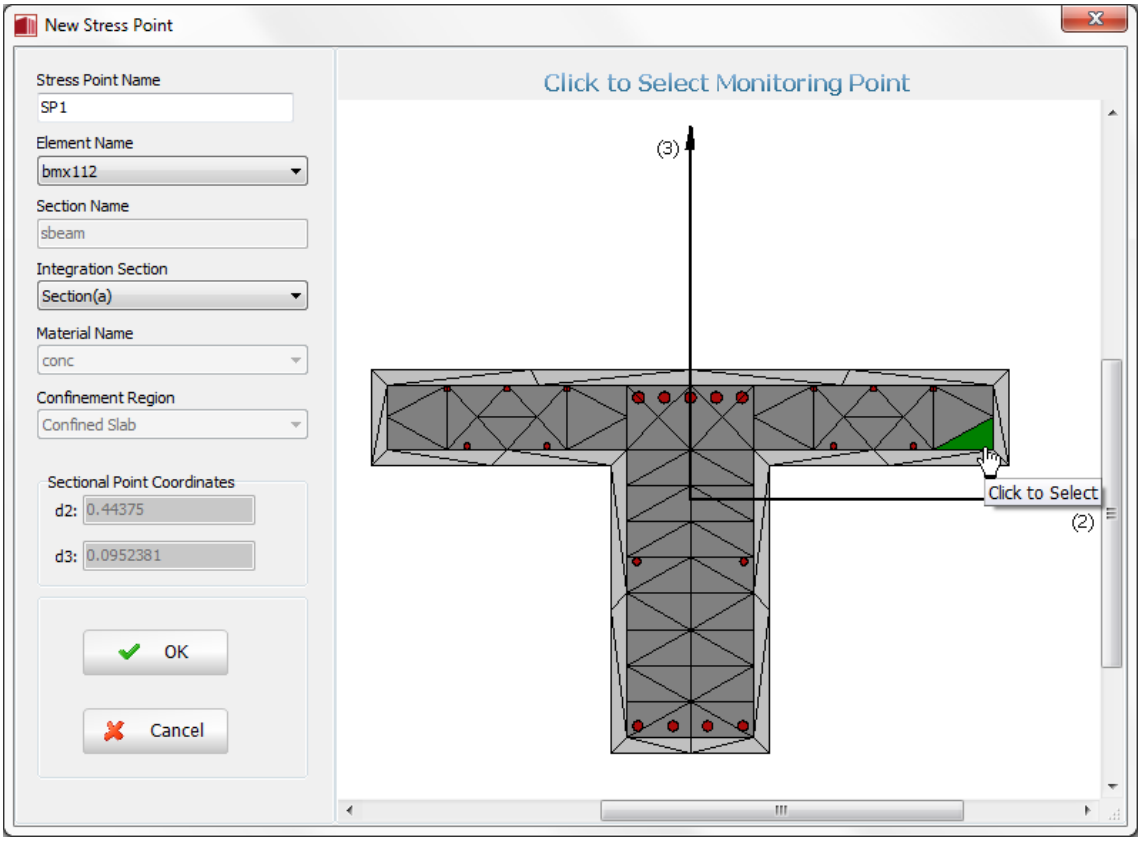
NOTE: This option should be used with care since choosing to output curvature and stress/strain peaks for all elements of a large structure may result in the creation of extremely large (hundreds of Mb) output files.

Output Stress and Strain Values at Selected Locations

If users are interested in following the variation of stress and strain of a particular material, located at a given sectional point in the Integration Sections of inelastic frame elements, then they may define Stress Points.

In order to add a new stress point, the user has to follow the steps listed below:

1. Click the *Add* button;
2. Assign a name;
3. Select the element name from the drop-down menu;
4. Select the integration section from the drop-down menu;
5. Select graphically on the section plot the area to be monitored. The material and the sectional coordinates will be automatically determined by the program.



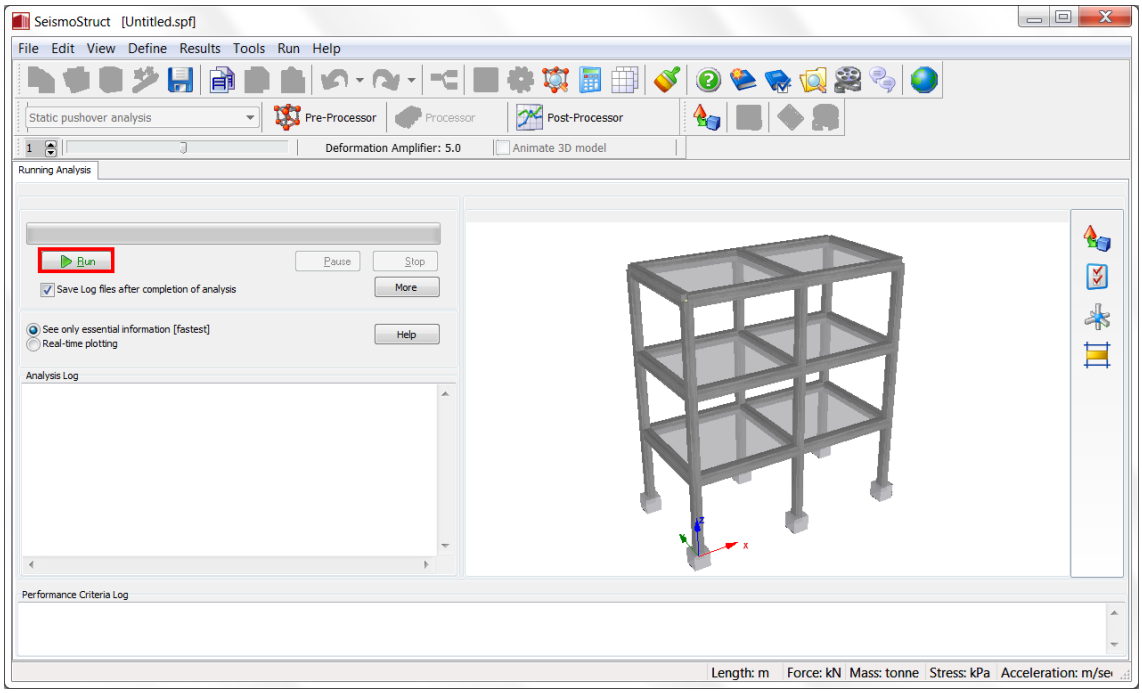
Adding a new stress point

NOTE: In the Output module, there is also the possibility for the user to customise the real-time displacement plotting that is shown during the analysis of a structure, by choosing (i) the node and (ii) degree-of-freedom to be considered. For better visualisation, users are advised to keep the program defaults, which employ the absolute top displacement plotted against base shear for static analysis, and the total drift (difference between top and bottom displacements) plotted against time value for dynamic analysis.

Processor

Having completed the pre-processing phase, the user is then ready to run the analysis. This is carried out in the **Processor** area of SeismoStruct, which is accessible through the corresponding toolbar button 

or by selecting *Run* > *Processor* from the main menu.



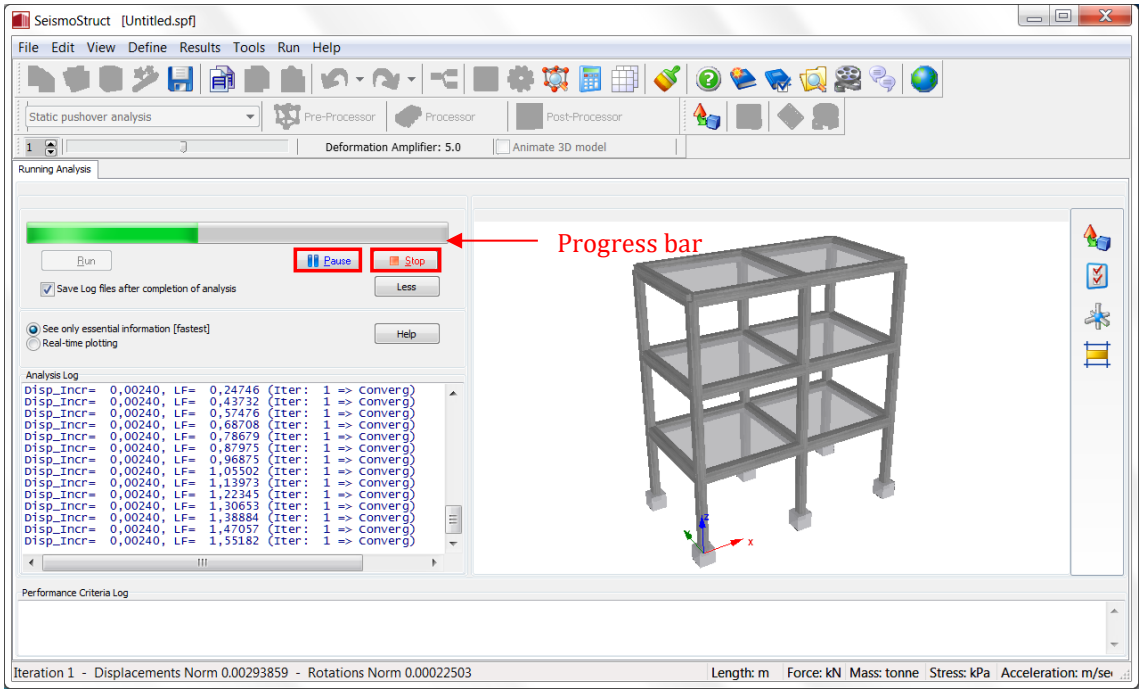
Processor area

NOTE: Simultaneous analysis of multiple models (up to hundreds, the only limit being the computer's physical memory), each of which subjected to similar or diverse loading (e.g. accelerogram), can be accomplished through their definition within the same project file (*.spf). In this manner, significant computing timesaving can be achieved, especially when a large number of simple models (e.g. single DOF cantilevers) are to be analysed, due to the savings in the output of results to the *.srf files. Further, automatic processing of these results can also be obtained through an opportune employment of IDA (with a single load factor).

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 (static analysis), several minutes (time-history analysis) or even hours (time-history analysis of large complex 3D models).

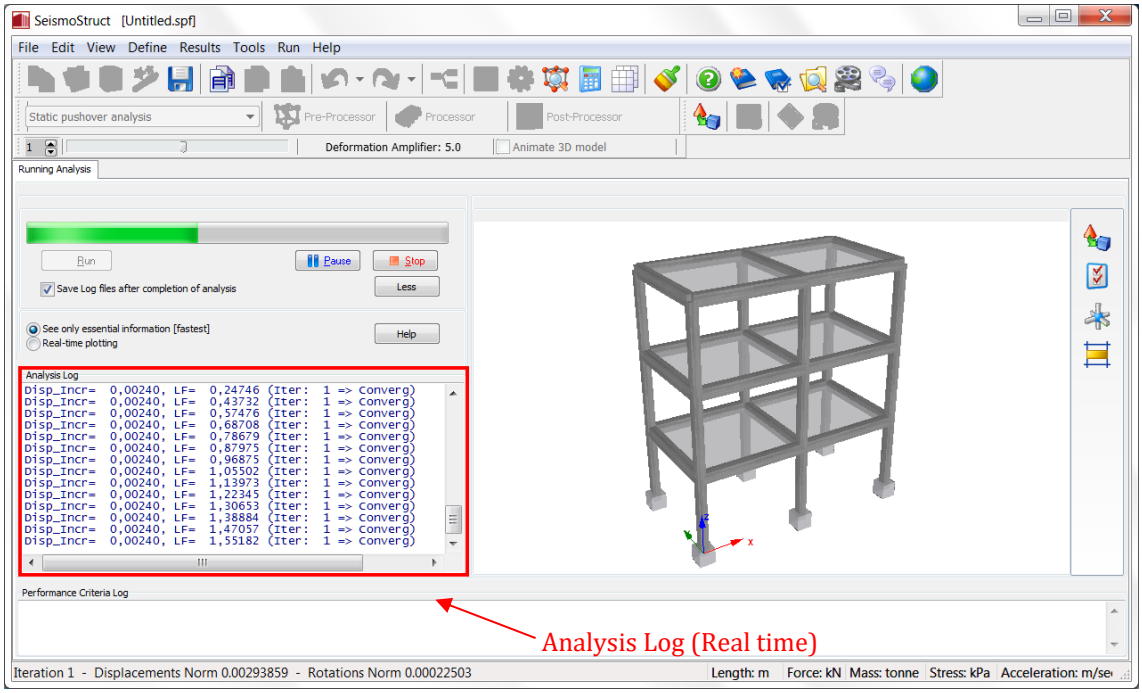
As the analysis is running, a *progress bar* provides the user with a percentage indication of how far has the former advanced to. Users can in this manner quickly assess the waiting time required for the analysis to be completed, and hence quickly plan their subsequent work schedule.

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 a lengthy analysis. If the user presses the *Run* button again, the analysis can be continued.



Progress bar and "Pause"/"Stop" buttons

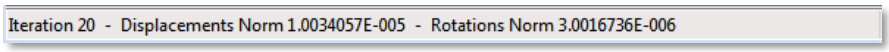
The *Analysis Log* is also shown to the user, in real-time, providing expedient information on the progress of the analysis, loading control and convergence conditions (for each global load increment).



Real-time Analysis Log area

This log is saved on a text file (*.log) that features the same name as the project file and which 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, if the user has specified code-based checks or performance criteria to be checked during the analysis, then the corresponding real-time log is also shown during the analysis and saved to the same *.log file.

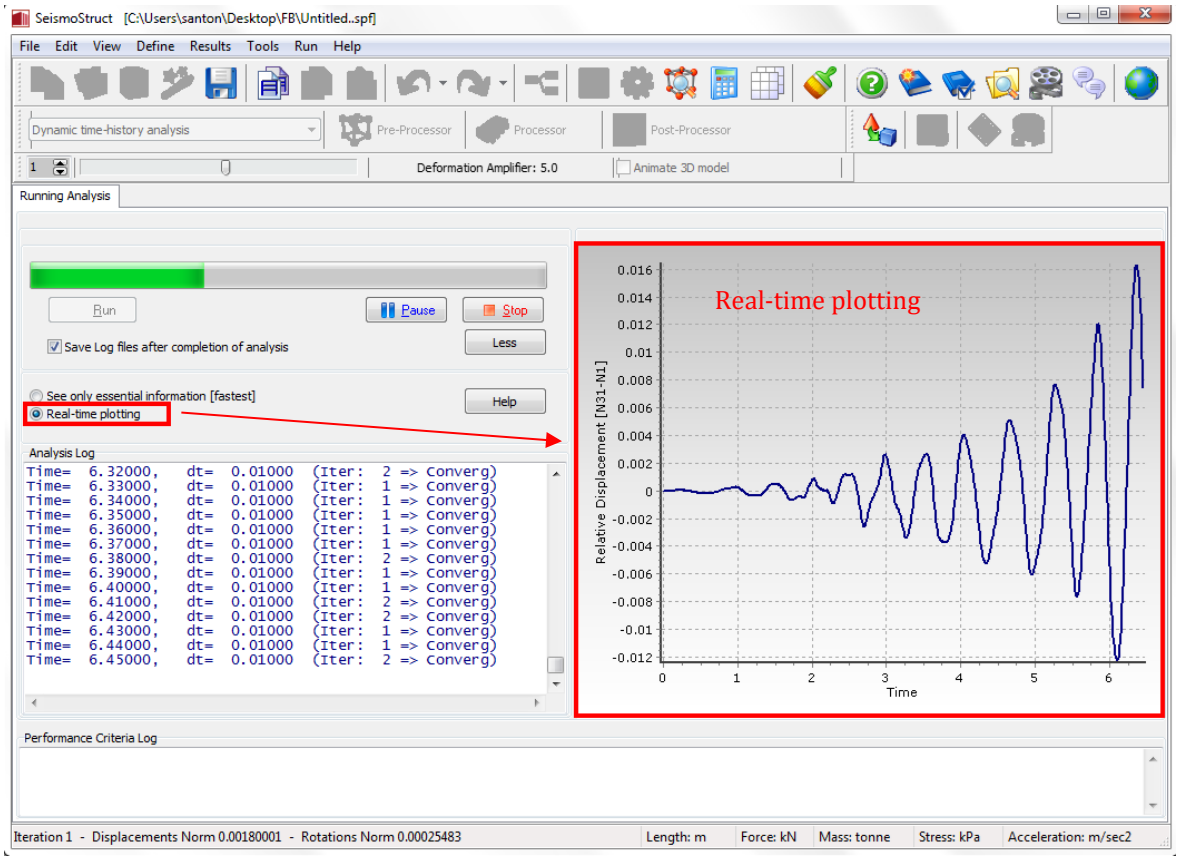
At the bottom of the window, the convergence norms at the end of a given (global) load increment are shown.



Convergence norms

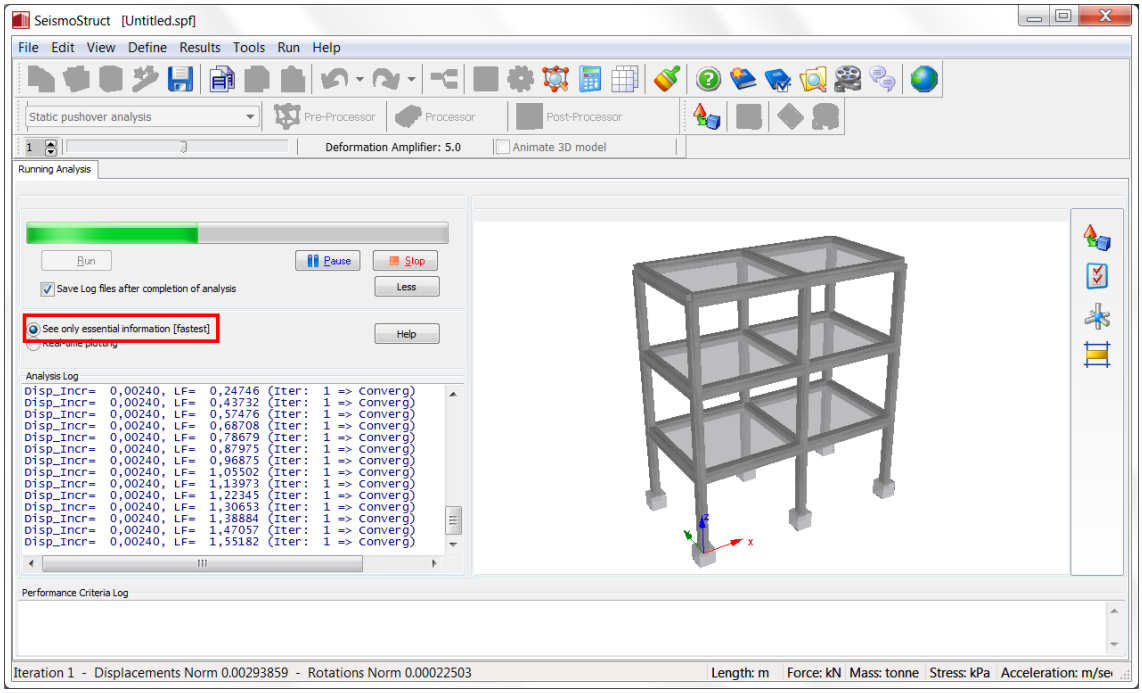
NOTE: As in the case of the Analysis Log described above, this information does not refer to local load increment/iterations of force-based elements mentioned in *Project Settings > Elements*.

Finally, the user has also the option of graphically observing the real-time plotting of a capacity (static pushover) or displacement time-history (time-history analysis) curve of any given node and respective degree-of-freedom, pre-selected in the **Output** module.



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 the real-time plotting, choosing to follow only the analysis logs.



See only essential information option

Furthermore, displaying of the latter can also be disabled (pressing the *Less* button) so as to attain even faster performance (on modern fast computers, however, the difference should be completely negligible).

NOTE 1: Upon start of the analysis, users may be presented with a warning message regarding 'Zero diagonal terms encountered in a give node'. This means that such node is unrestrained in the degrees-of-freedom indicated (i.e. the node is not connected to an element or constraint capable of providing any restrain/stiffness in such dofs), a condition that, if unintended, implies the presence of an error in the assemblage of model. If, instead, such unrestrained nodal dofs have been intentionally introduced, the user may proceed with the analysis, knowing however that numerical convergence difficulties may arise more easily in such cases.

NOTE 2: When running an eigenvalue analysis using Lanczos algorithm, user may be presented with a message stating: "could not re-orthogonalise all Lanczos vectors", meaning that the Lanczos algorithm, currently the eigenvalue solver in SeismoStruct, could not calculate all or some of the vibration modes of the structure. This behaviour may be observed in either (i) models with assemblage errors (e.g. unconnected nodes/elements) or (ii) complex structural models that feature links/hinges etc. If users have checked carefully their model and found no modelling errors, then they may perhaps try to "simplify" it, by removing its more complex features until the attainment of the eigenvalue solutions. This will enable a better understanding of what might be causing the analysis problems, and thus assist users in deciding on how to proceed. This message typically appears when too many modes are sought, e.g. when 30 modes are asked in a 24 DOF model, or when the eigensolver cannot simply find so many modes (even if DOFs > modes).

NOTE 3: The current version of SeismoStruct 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 4: There is a size limitation of the output file in SeismoStruct, the maximum results size that can be opened from the Post-processor is 4GB in 64-bit Windows systems and 3GB in 32-bit Windows systems. In analyses with larger output *.srf files, SeismoStruct is only able to read the results up to that

NOTE 5: Up until now, the development of SeismoStruct has focused primarily on the achievement of ease-of-use and high technical capabilities, with an obvious sacrifice in terms of speed of analysis, something that we hope to address in the future. In the meantime, however, please make sure that your model does not feature an unnecessarily excessive number of elements, section fibres, load increments or iterations, all of which, together with too-stringent *convergence criteria*, contribute to slow analyses.

NOTE 6: When using the less numerically stable Frontal solver, it may happen that analysis stops, at different time-steps. On such occasions, users are advised to change to the default Skyline solver.

Post-Processor

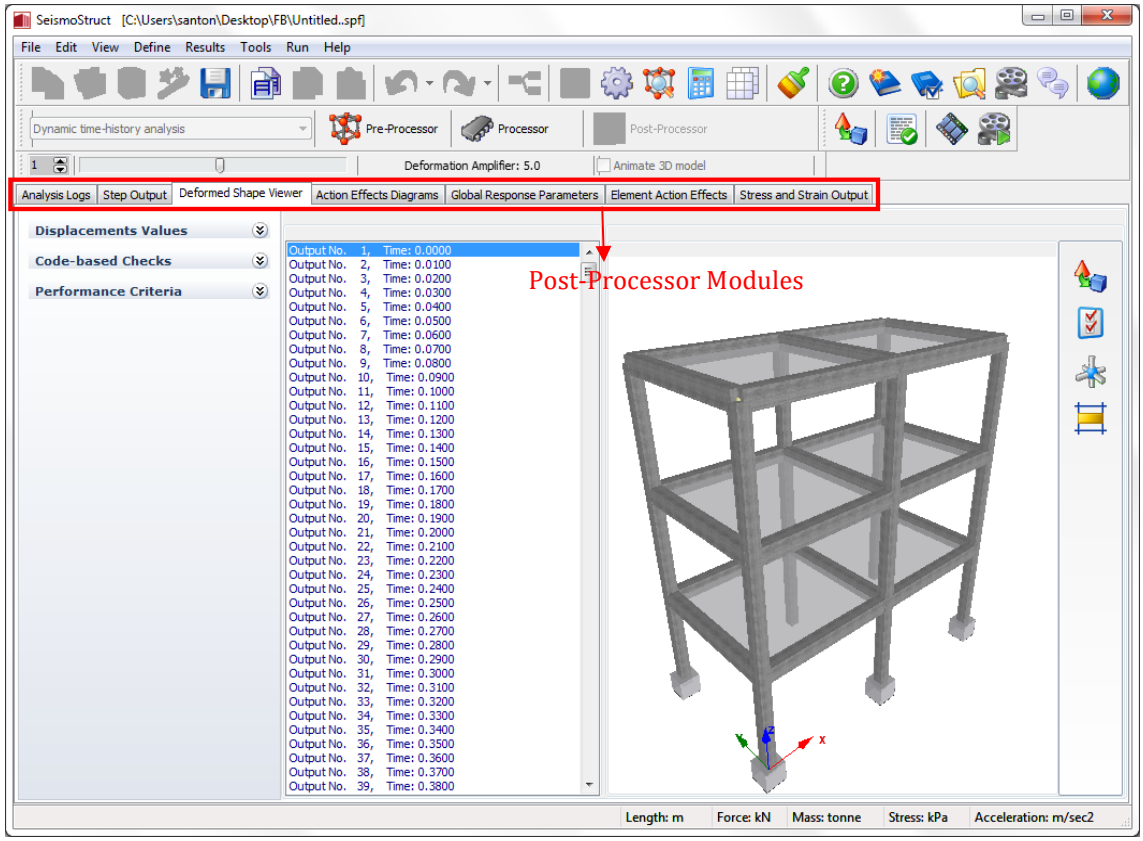
The results of the analysis are saved in a SeismoStruct Results File, distinguishable by its *.srf extension, with the same name as the input project file. Double-clicking on this type of files will open SeismoStruct's Pre-Processor. The **Post-Processor** can then be accessed through the corresponding toolbar button 

or by selecting *Run* > *Post-Processor* from the main menu.

Similarly to its **Pre-Processor** counterpart, the **Post-Processor** area features a series of modules where results from different type of 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). It is noted that a special facility of visualising the maximum, minimum and absolute maximum values in all the plots of the Post-Processor is available.

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

- Analysis Logs
- Modal/Mass Quantities
- Target Displacement
- Step Output
- Deformed Shape Viewer
- ConvergenceProblems
- Action Effects Diagrams
- Code-based Checks
- Global Response Parameters
- Element Action Effects
- Performance Criteria Checks
- Stress and Strain Output
- IDA Envelope Curve



Post-Processor Modules

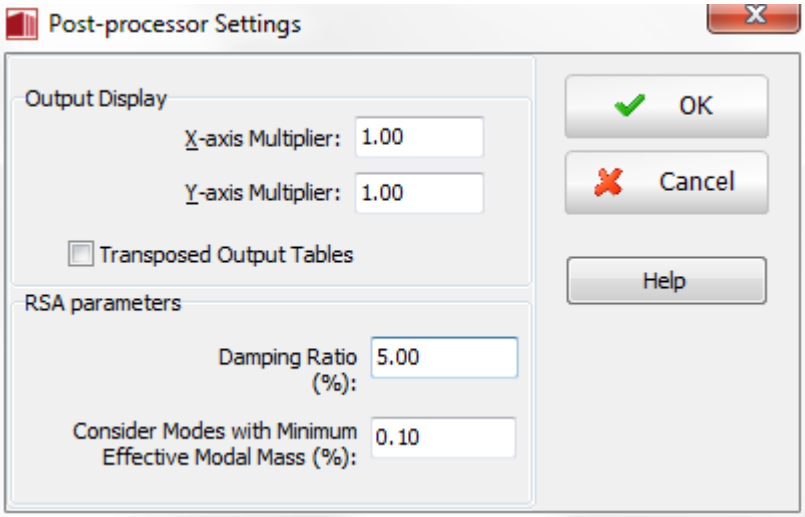
There are some general operations that apply to all the Post-Processor modules. For example, the way in which model components (e.g. nodes, sections, elements, etc.) are sorted in their respective pre-processor modules reflects the way these entries appear on all dialogue boxes in the post-processor. For instance, if the user chooses to employ alphabetical sorting of the nodes, then these will appear in alphabetical order in all drop-down menus where nodes are listed, which may, in a given case, ease and speed up their individuation and selection. An option to sort by name the nodes and elements in the lists of the post-processor is currently available on the right click popup menu.

In addition, when using drop-down lists with many entries, users can start typing an item's identifier so as to reach it quicker.

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 Nmm as the units for moment quantities, users might wish to multiply the corresponding results by 1e-6, so as to obtain moments expressed in kNm instead. Alternatively, and as another example, users might also wish to multiply concrete stress values 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

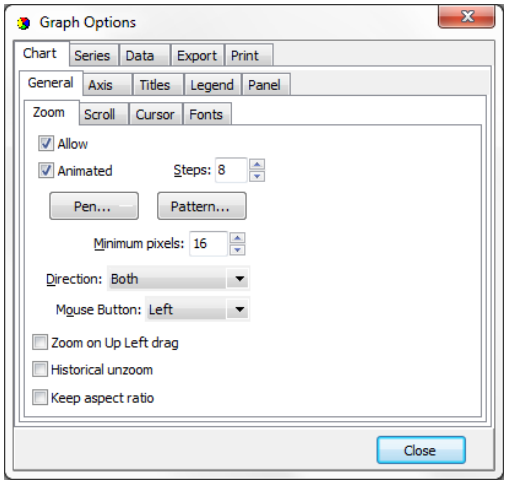
In addition, the *Post-Processing Settings* provide users also with the possibility of transposing the *Output Tables*. This might come very hand in cases where, for instance, a model features several thousands of nodes/elements, which in turn leads to default output tables with an equally very large 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 1048576 rows).

Finally, from the *Post-Processor Settings* the user may change the damping ratio and the minimum effective modal mass of the modes that will be taken into consideration in Response Spectrum Analysis. These two settings, which have initially been defined in the *Pre-Processor Settings,* can also be changed from within the Post-Processor, in order to adapt the loading combinations to specific needs of the users.

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.

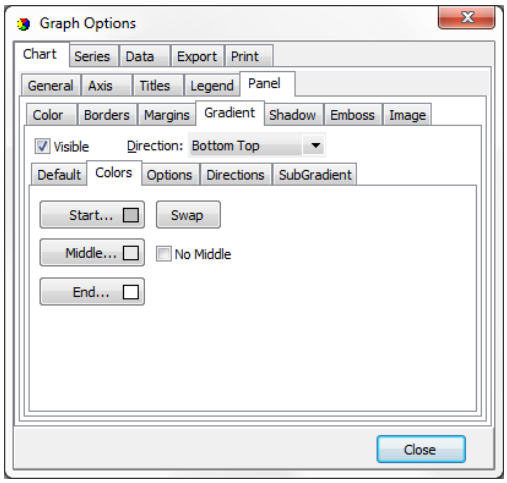
PLOT OPTIONS

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

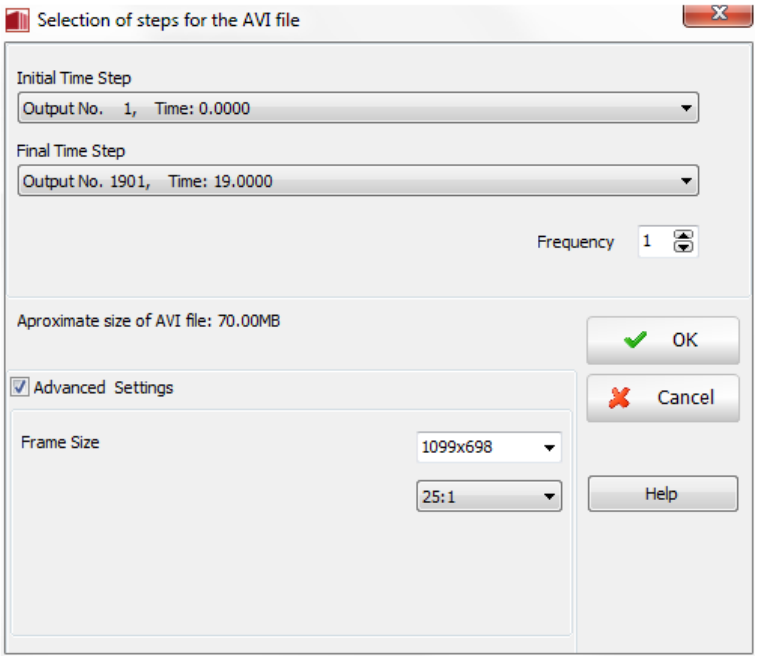
In addition, zooming-in and -out can be done by dragging the mouse on the graph area (a top-left to bottom-right selection zooms in, whereas a bottom-right to top-left selection zooms out).

CREATING AN ANALYSIS MOVIE

SeismoStruct provides users with the possibility of creating animations illustrating the way a particular structure, subjected to a given set of loads, deforms in time (dynamic analysis) or pseudo-time (static analysis). In addition, users can also create 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 .

For the case of static and dynamic 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.

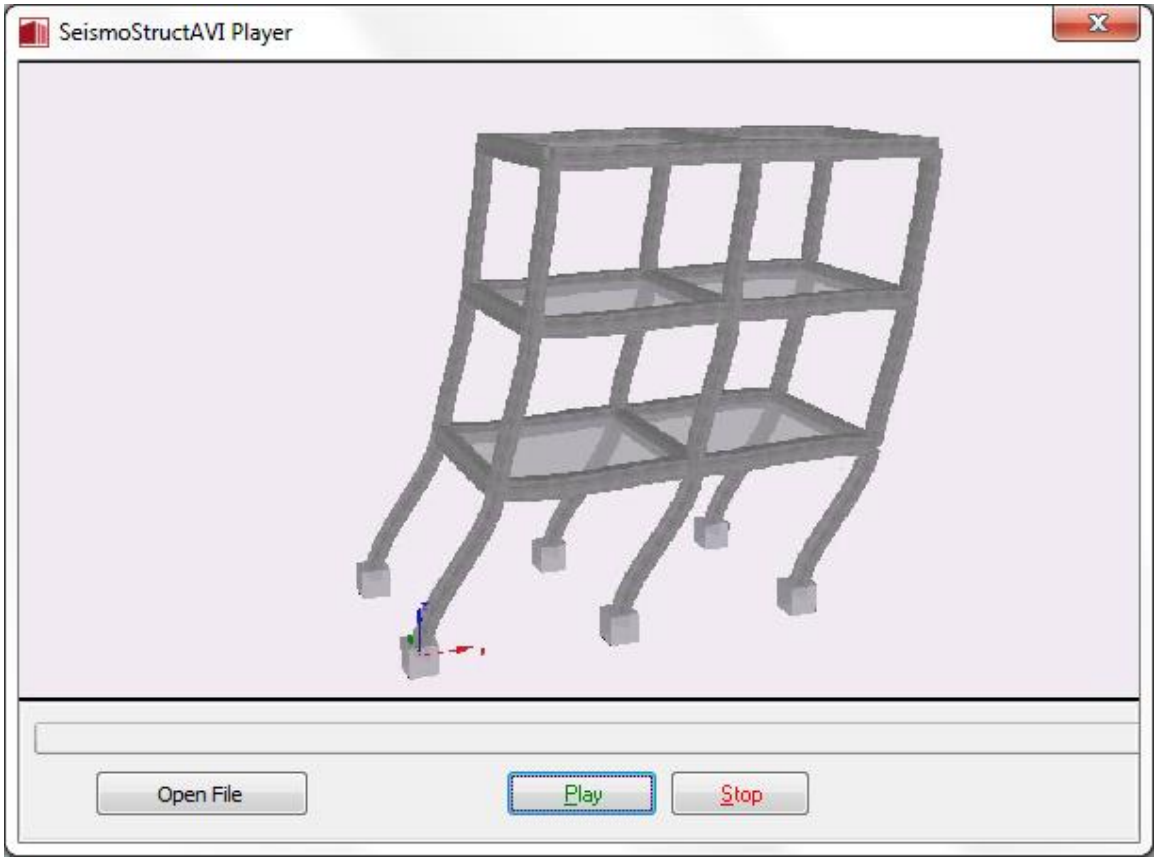


Selection of steps for the AVI file

If, on the other hand, 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.

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 SeismoStruct, accessible from the program main menu (*File > Show AVI file...*) or through the respective toolbar button .



SeismoStruct AVI player

Animations created in SeismoStruct (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.

ANALYSIS LOGS

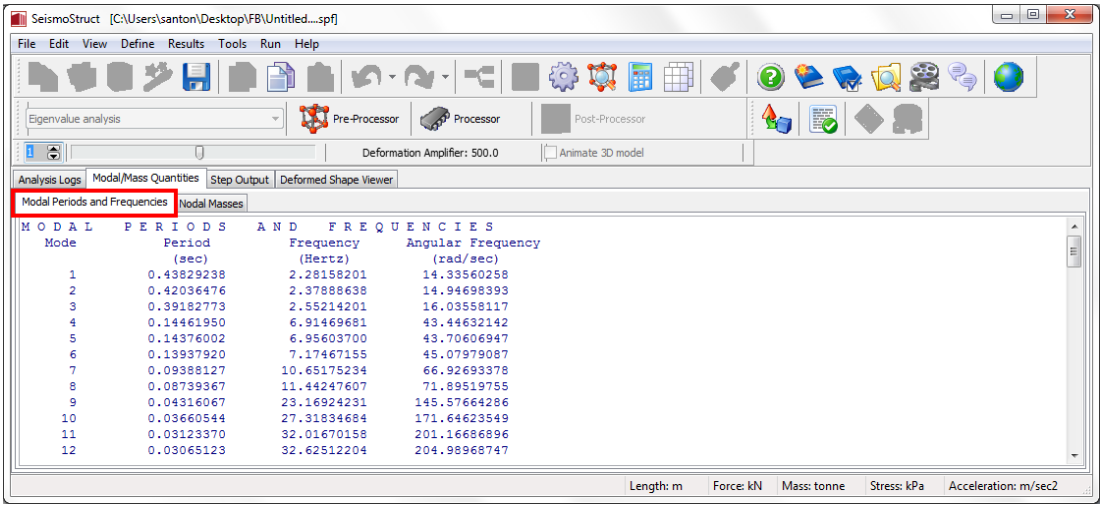
As discussed in the **Processor** area, during any given analysis, a log of its numerical progress and of the performance response of the model is created and saved within the project's log file (*.log). The contents of such file can be visualised in the **Analysis Logs** module and, if required, copied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

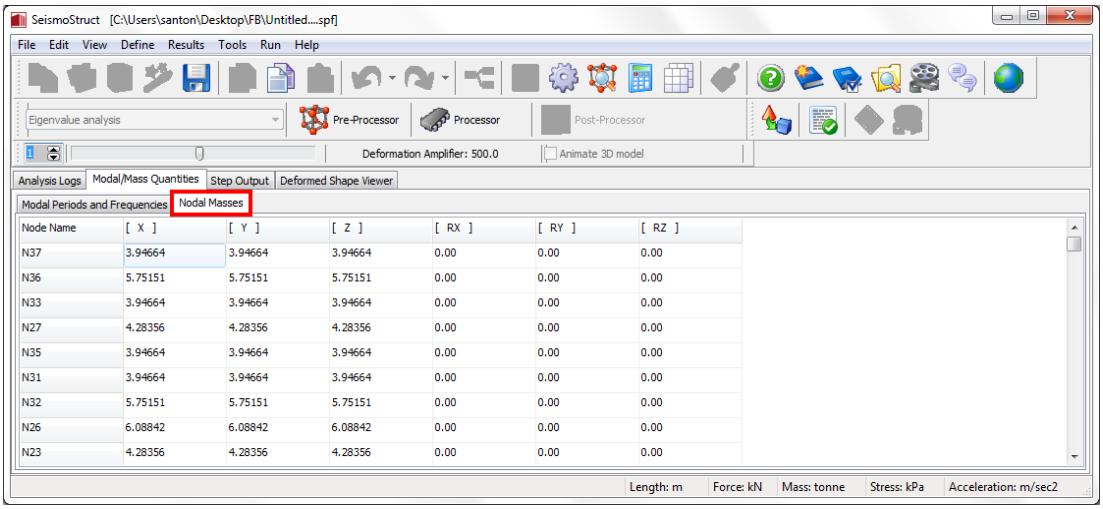
MODAL/MASS QUANTITIES

IMPORTANT: This module is visible only when Eigenvalue or Adaptive Pushover analyses have been carried out. It is also shown with a different name 'Eigenvalue Results', in the case of Pushover analysis when the Target displacement is calculated.

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, SeismoStruct 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 above reason, 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.

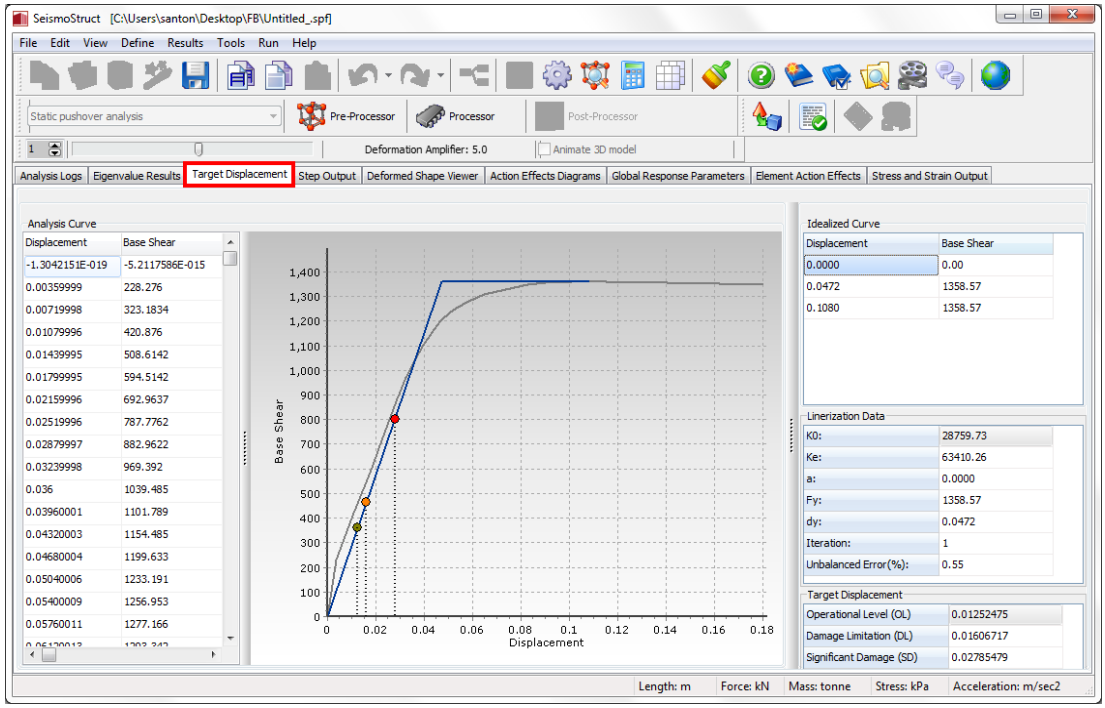
NOTE 2: The mode shapes are normalised so that $\Phi n=1$.

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).

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

TARGET DISPLACEMENT

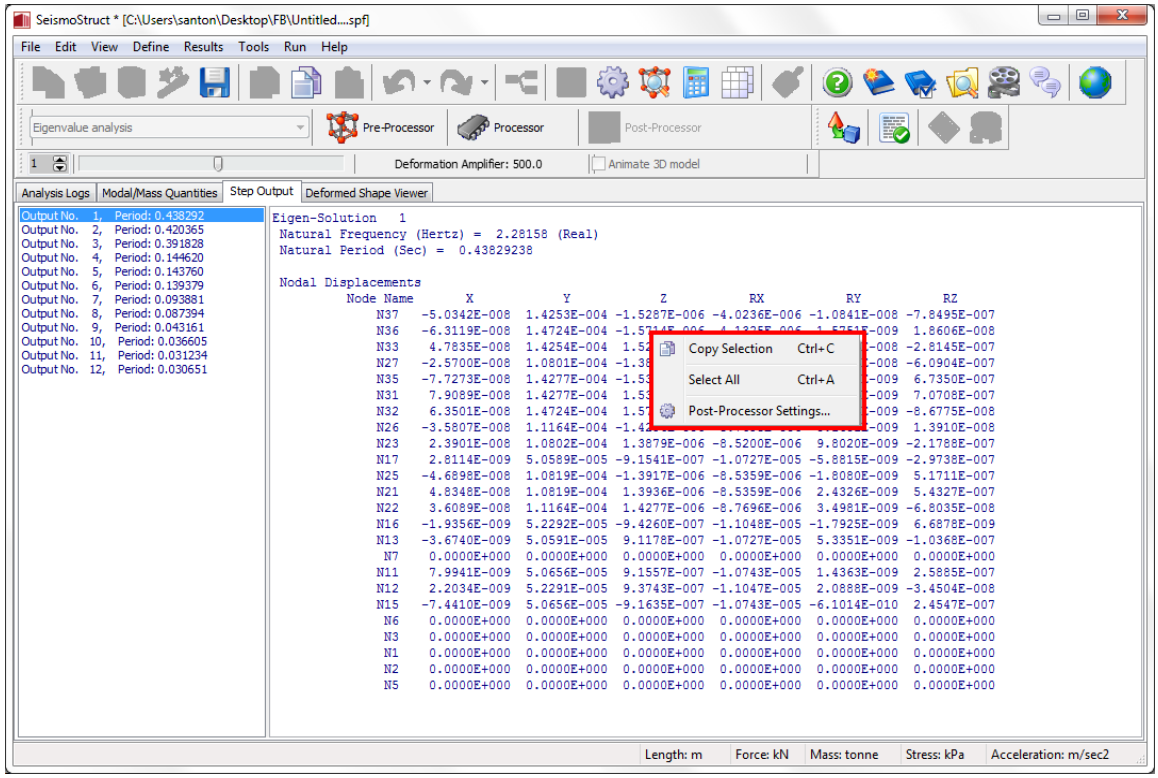
In the Target Displacement module the capacity curves before and after linearisation are shown, together with the calculated target displacements for the selected limit states. Data about linearisation and the target displacement calculation are also provided herein. The linearisation procedure is always carried out according to the methodology proposed by the selected Code. Users may refer to Appendix H – Codes for more information about the calculation of the target displacement.



Target Displacement Module

STEP OUTPUT

This post-processing module applies to all analysis types and 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 SeismoStruct at any given analysis step. 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 .

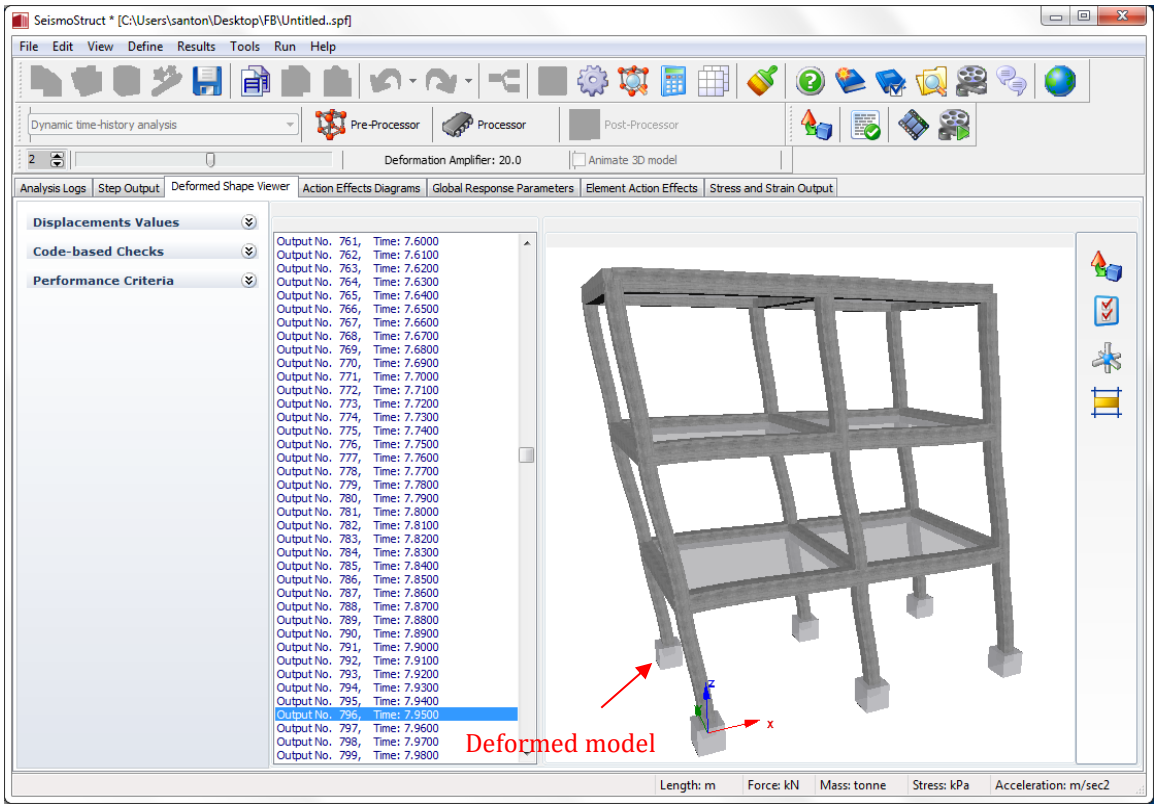
Finally, and as noted in *Project Settings > General*, users may also activate the option of creating, at the end of every analysis, a text file (*.out) containing the output of the entire analysis (as given in this module). This feature may result useful for users, who wish to systematically, rather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

NOTE 1: Step output corresponding to Permanent loads applied at the start of pushover and time-history analysis, refers always to the step where equilibrium has been reached, which usually corresponds to the one single increment/iteration required to balance this type of loads. However, there are occasions (very large permanent loads), where more than one increments/iterations are required to reach structural equilibrium. Users who wish to visualise the interim steps carried out to arrive at the final equilibrated solution of such large initial permanent loads, should run a non-variable static analysis, where such output is given.

NOTE 2: Step output for elastic frame elements (elfrm) is provided always after the output of their inelastic counterparts (infrm, infrmPH), even if the former alphabetically precedes the latter.

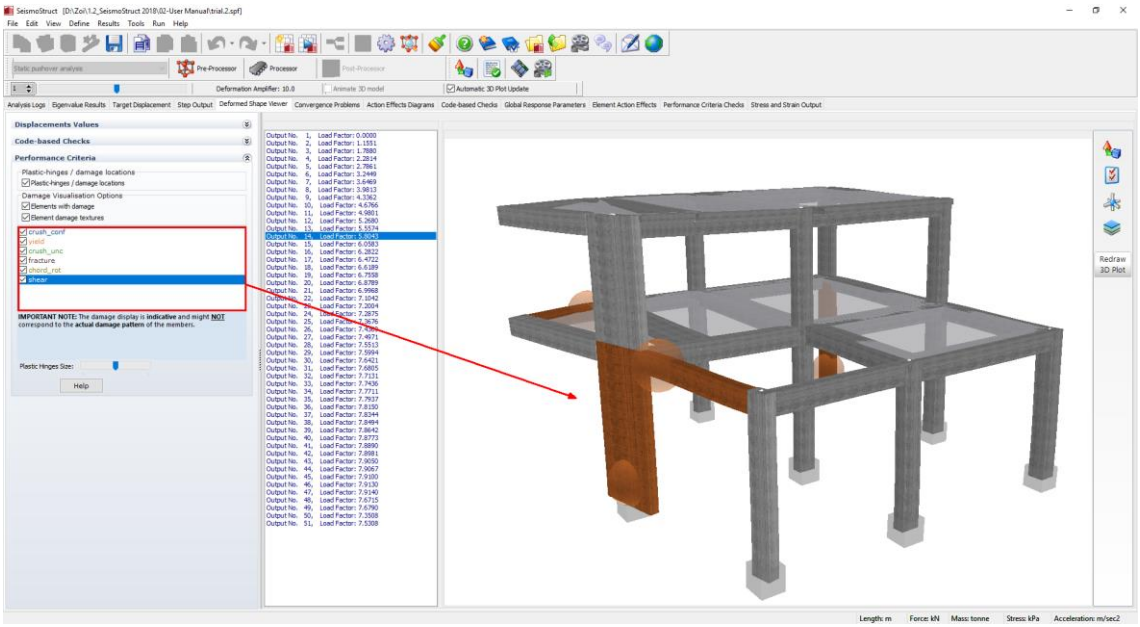
DEFORMED SHAPE VIEWER

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.

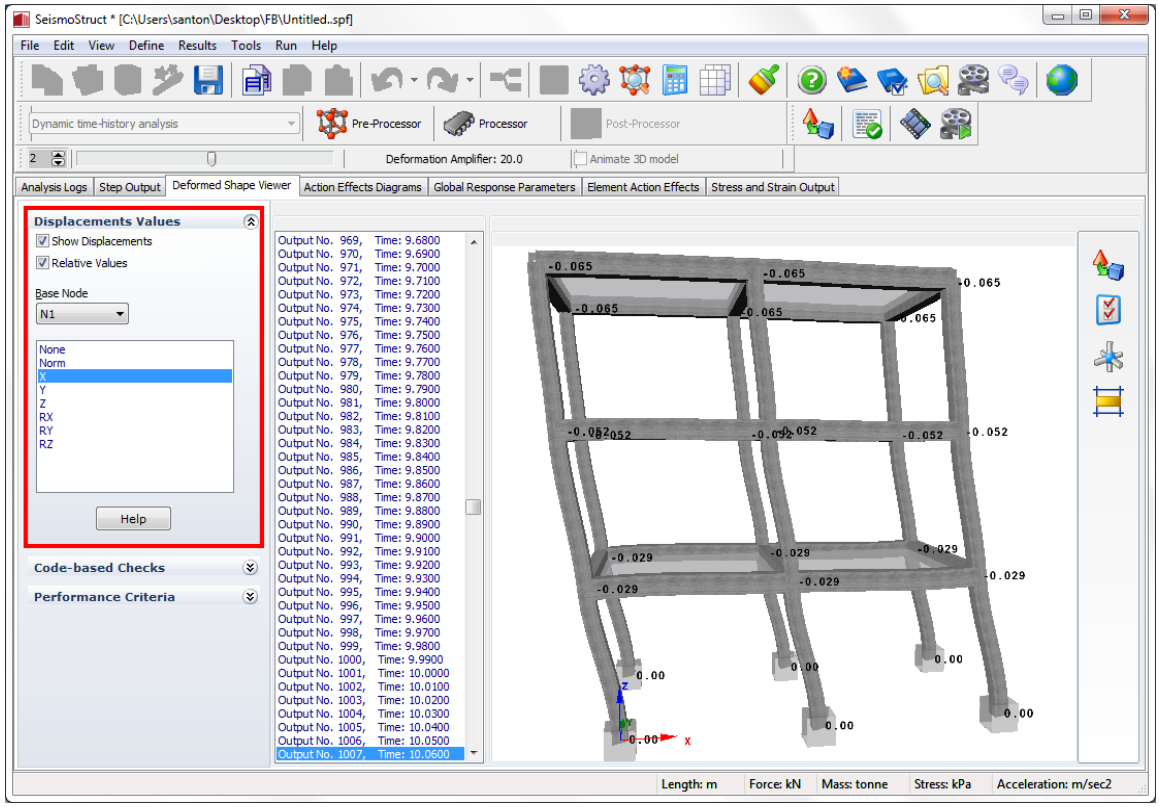


Deformed Shape Viewer

In this module it is also possible to visualise the elements that reach a particular performance criterion. This can be done by choosing the *Performance Criteria* option and selecting if the plastic hinges/ damage locations will be shown, and whether these elements will be distinguished through colours and/or damaged textures. In addition, also the displacements values may be displayed by checking the associated box.

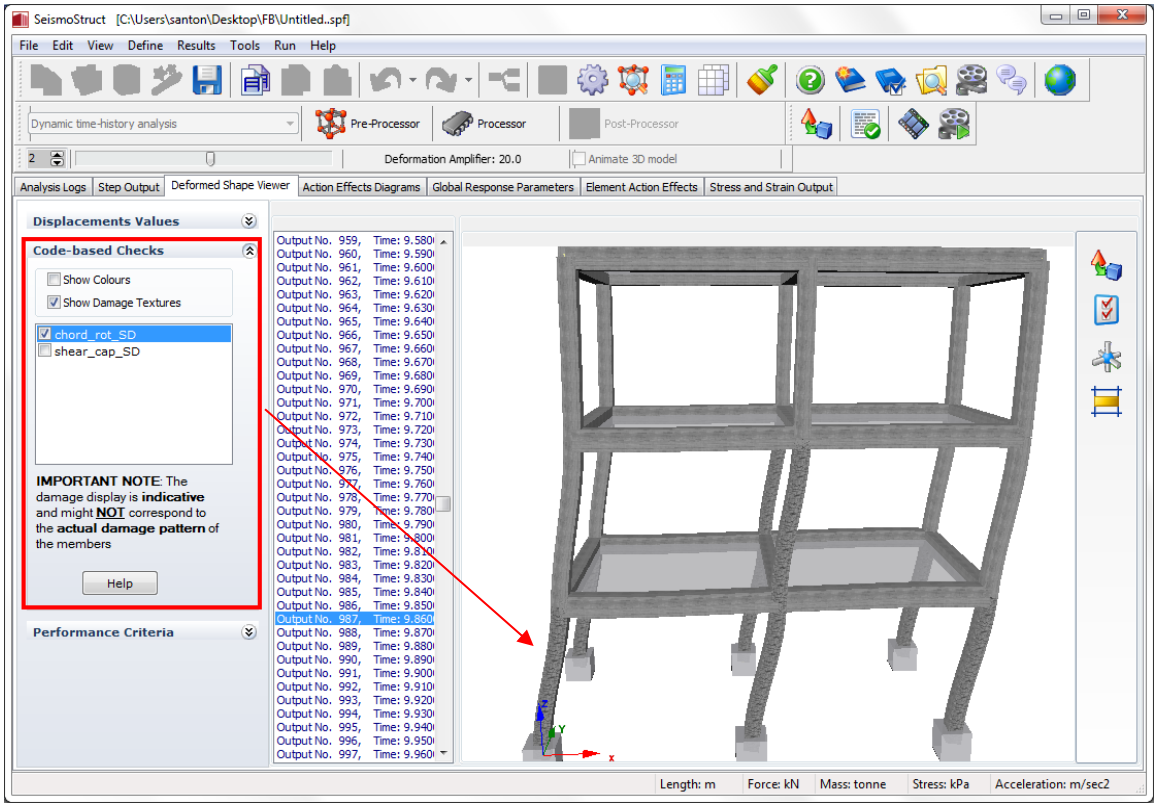


Deformed Shape Viewer – Performance Criteria option



Deformed Shape Viewer – Displacement values option

Finally, the elements that have exceeded their capacity at a particular code-based check may be visualised by choosing the Code-based Checks option and selecting if the plastic hinges/ damage locations will be shown, and whether these elements will be distinguished through colours and/or damaged textures.



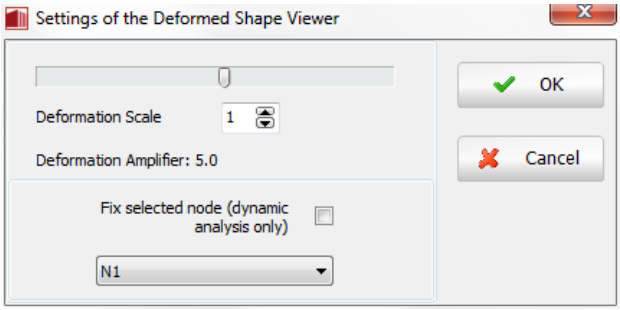
Deformed Shape Viewer – Code-based Checks option

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 *Deformed Shape Multiplier*, available from the right-click popup menu or through the main menu (*Tools > Deformed Shape Settings...*) or through the corresponding toolbar button , to better adapt the plot.

Finally, and in the case of dynamic analysis, it is also useful to check the *Fix selected node* option, so that only the relative displacements of the structure, which are those of interest to engineers, are plotted. The 'selected node' should obviously be a node at the base of the structure in order for this option work; if the *Wizard* facility has been used, the default selected node is N1 (see below).

Moreover, the absolute rigid-body deformation of the structure's foundation nodes (resulting from the double-integration of the acceleration time-history), is usually unrealistically large, since no base-line correction, or other types of filtering, is applied during the integration process, as would be required to obtain sensible results.

IMPORTANT: Users are strongly advised to always make use of this option when post-processing dynamic analysis results.

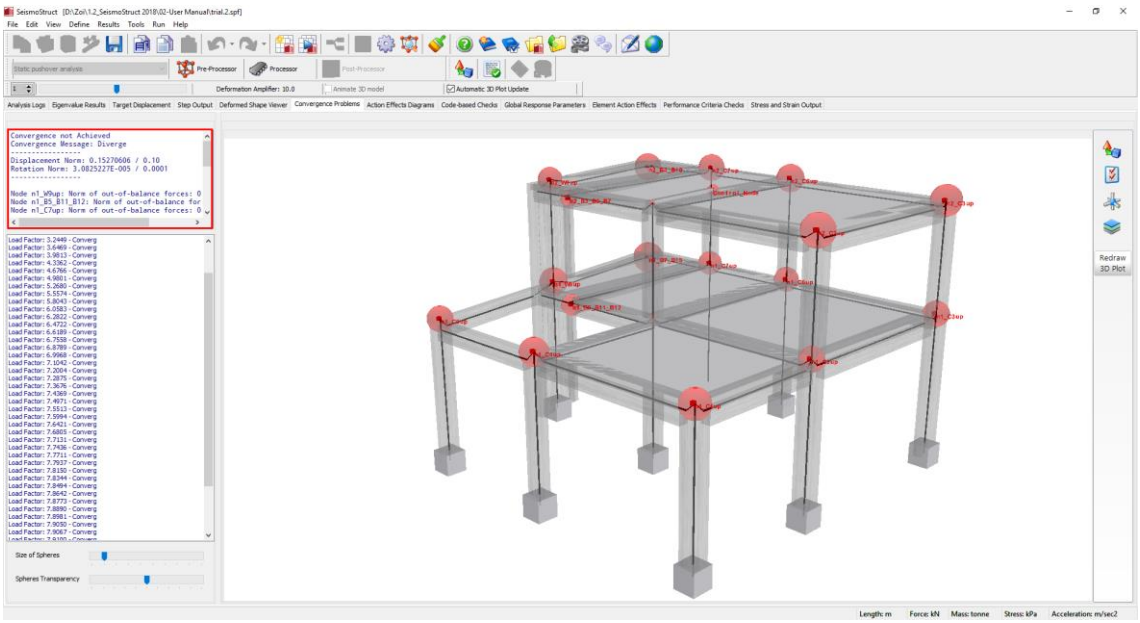


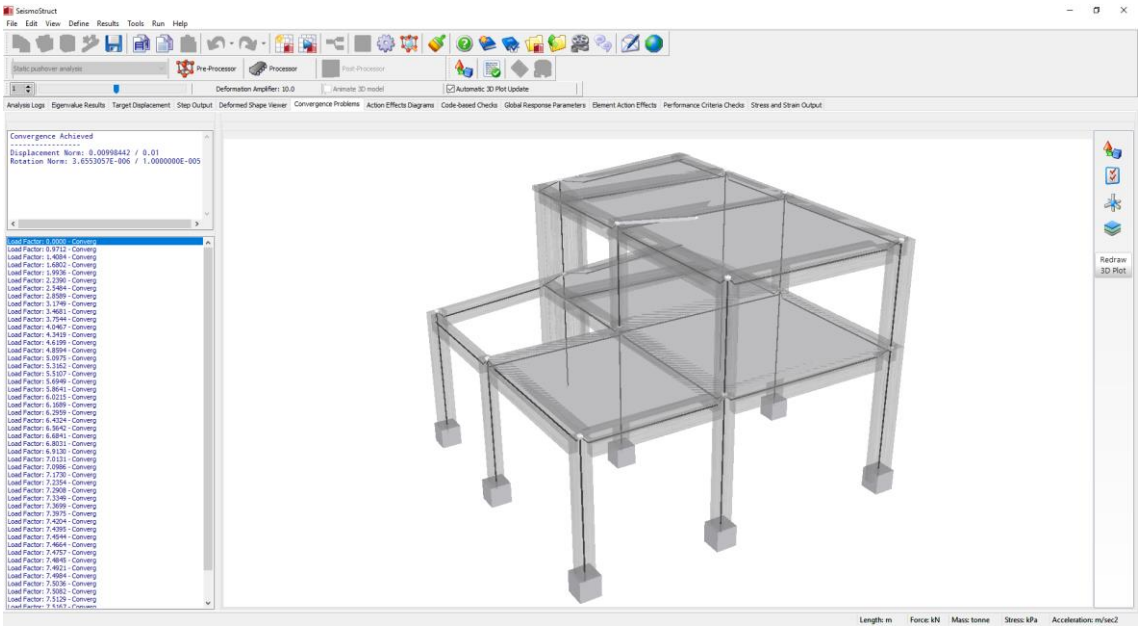
Deformed Shape Settings

NOTE: In order for deformed shape plots to be available, nodal response parameters must have been output for all structural nodes (see Output module), otherwise the Post-Processor will not have sufficient information to compute this type of plots.

CONVERGENCE PROBLEMS

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.



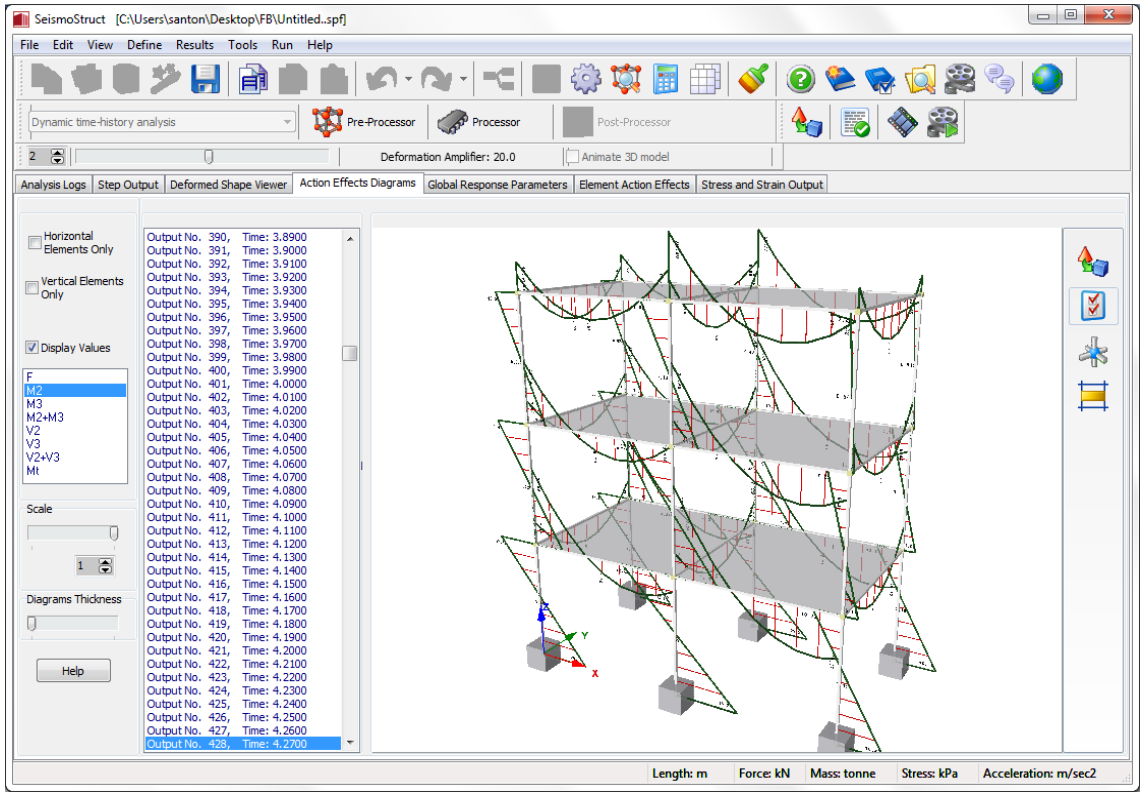


Convergence Problems

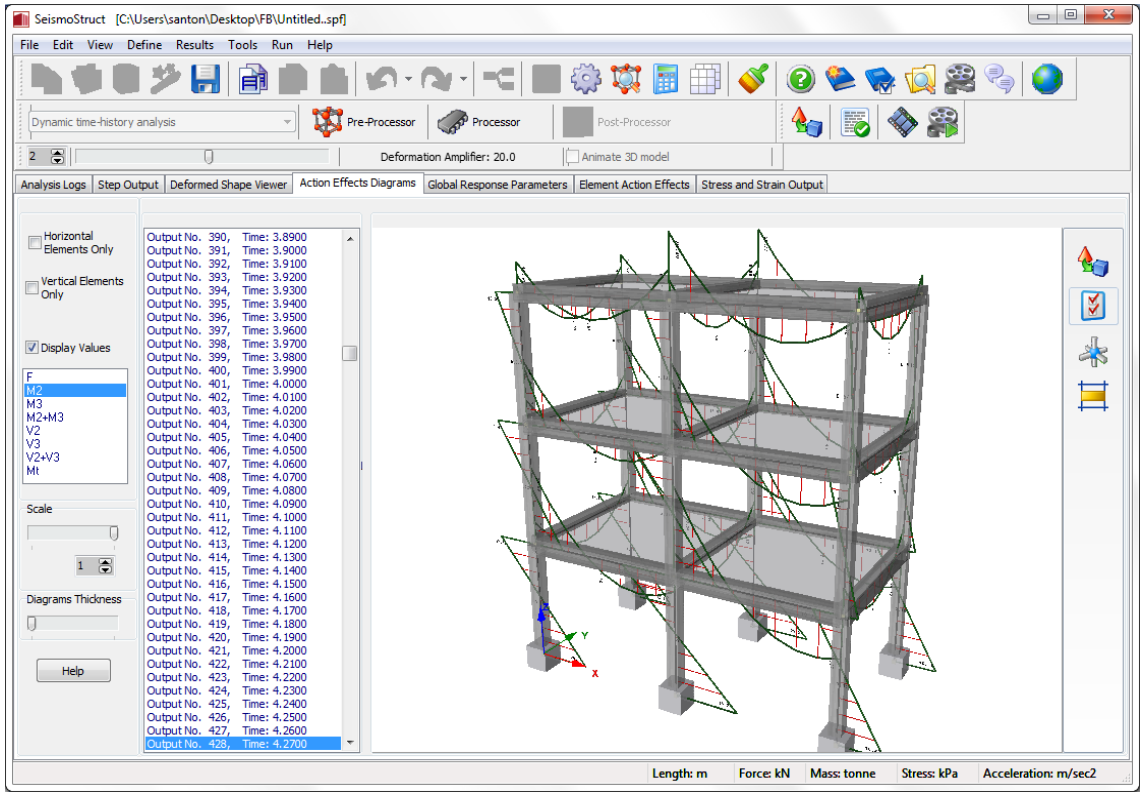
NOTE: Users should activate in Project Settings > Convergence Criteria the option of showing convergence difficulties in Post-Processor in order to be able to view the Convergence Problems tab in the Post-Processor.

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.

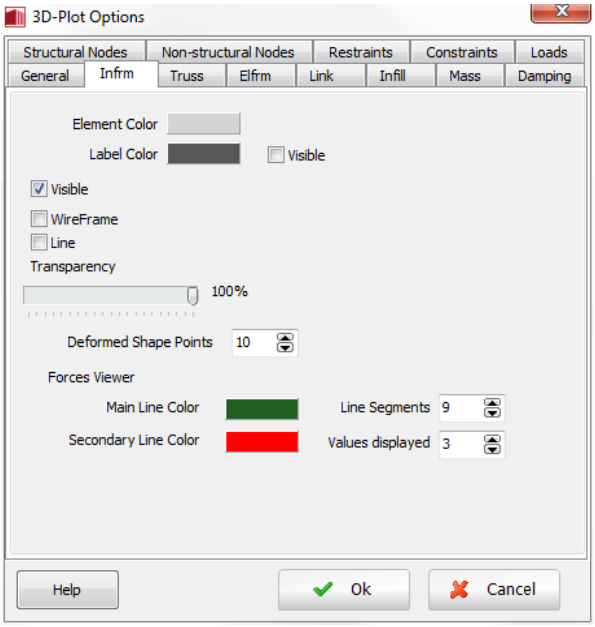


Action Effects Diagrams (Elements as Lines)



Action Effects Diagrams

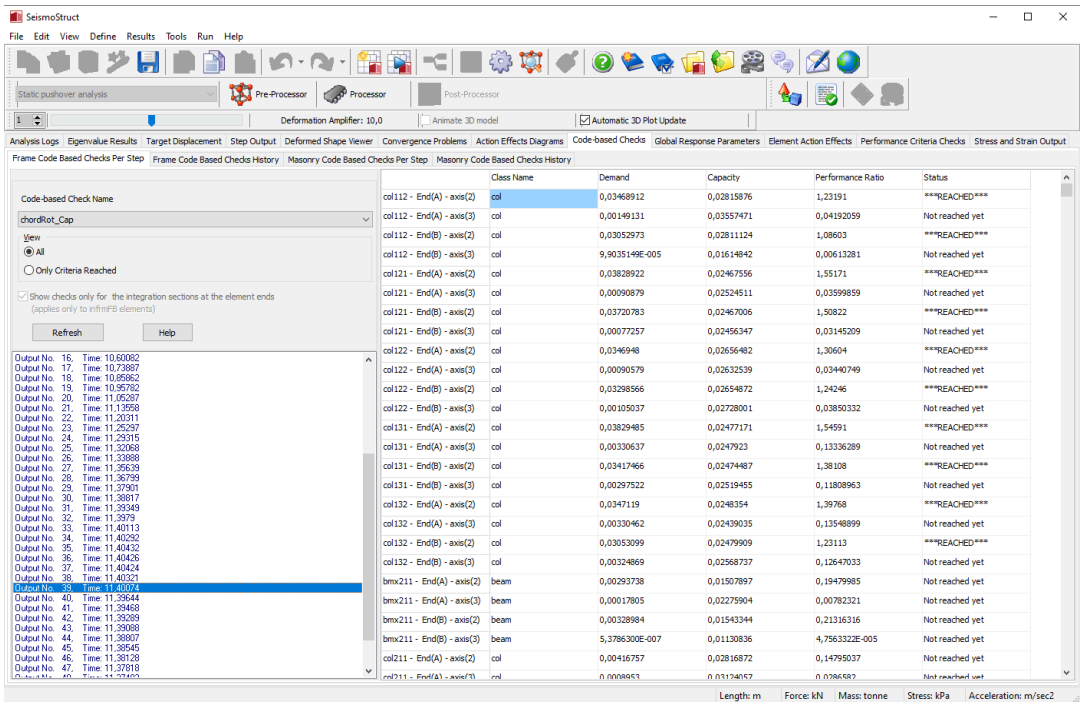
Users may customize the diagrams, through the 'infrm' or 'elfrm' tab in the *3D Plot Options* menu (i.e. main line and secondary line colours, number of sec. lines and number of values).



3D-Plot Options

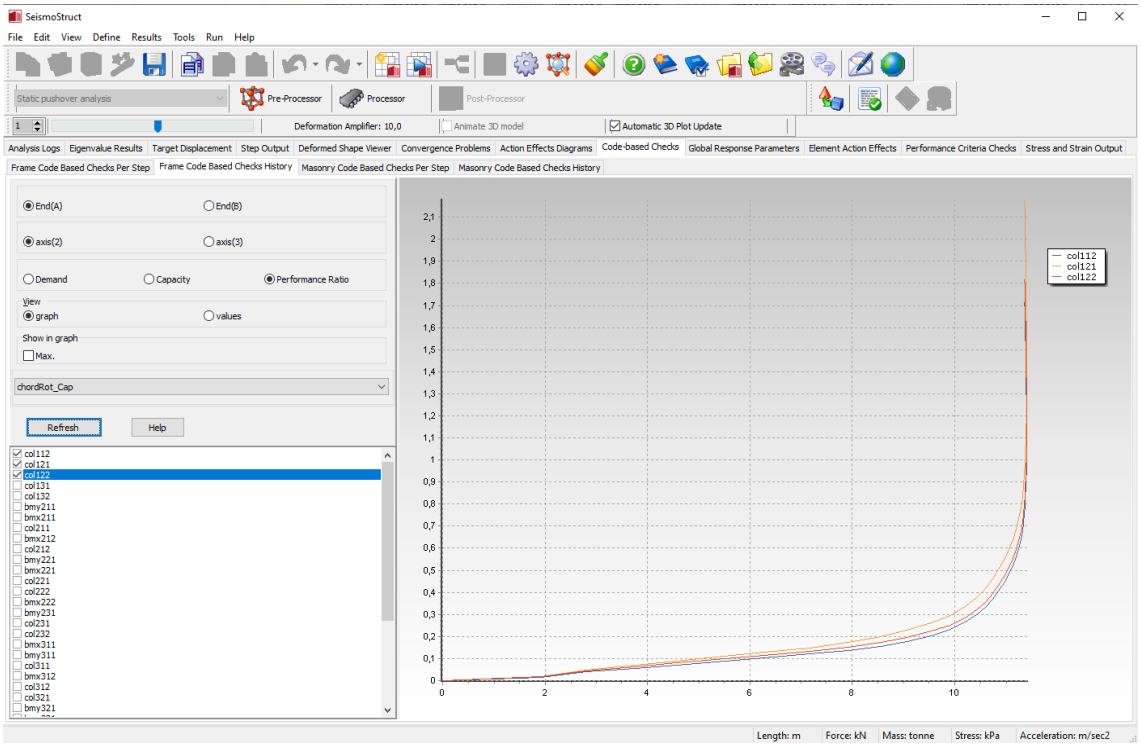
CODE-BASED CHECKS

Here, it is possible for the user to perform the Code-based Checks Per Step. Different tabs for Frame Code-based Checks and masonry Code-based Checks per step are available. First, he/she has to select the code-based check name from the drop-down menu. Then, it is necessary to select the step of the analysis (e.g. a particular limit state). Regarding the view options, the results can be displayed for all the elements or only for those elements that have reached the criterion selected.



Code-based Checks – Code Based Checks Per Step

In addition, the user may extract the Code-based checks history of the structural members. Different tabs for Frame Code-based Checks History and Masonry Code-based Checks History are available. Users have to select the code-based check name from the drop-down menu, then, they should select the element and click the Refresh button. The results can be displayed in the form of a chart or a table. Finally, maximum values can be displayed in the selected chart.



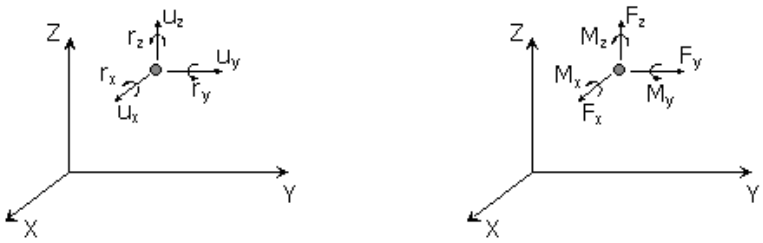
Code-based Checks – Code Based Checks History

GLOBAL RESPONSE PARAMETERS

Depending on the type of analysis and/or the input parameters defined in the Pre-Processor, up to six different kinds of global response parameters results can be output in this module:

1. Structural displacements
2. Forces and Moments at Supports
3. Velocities/Accelerations
4. Total Inertia & Damping Forces
5. Hysteretic curves

Apart from the last two modules, where performance checks are displayed, in 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.

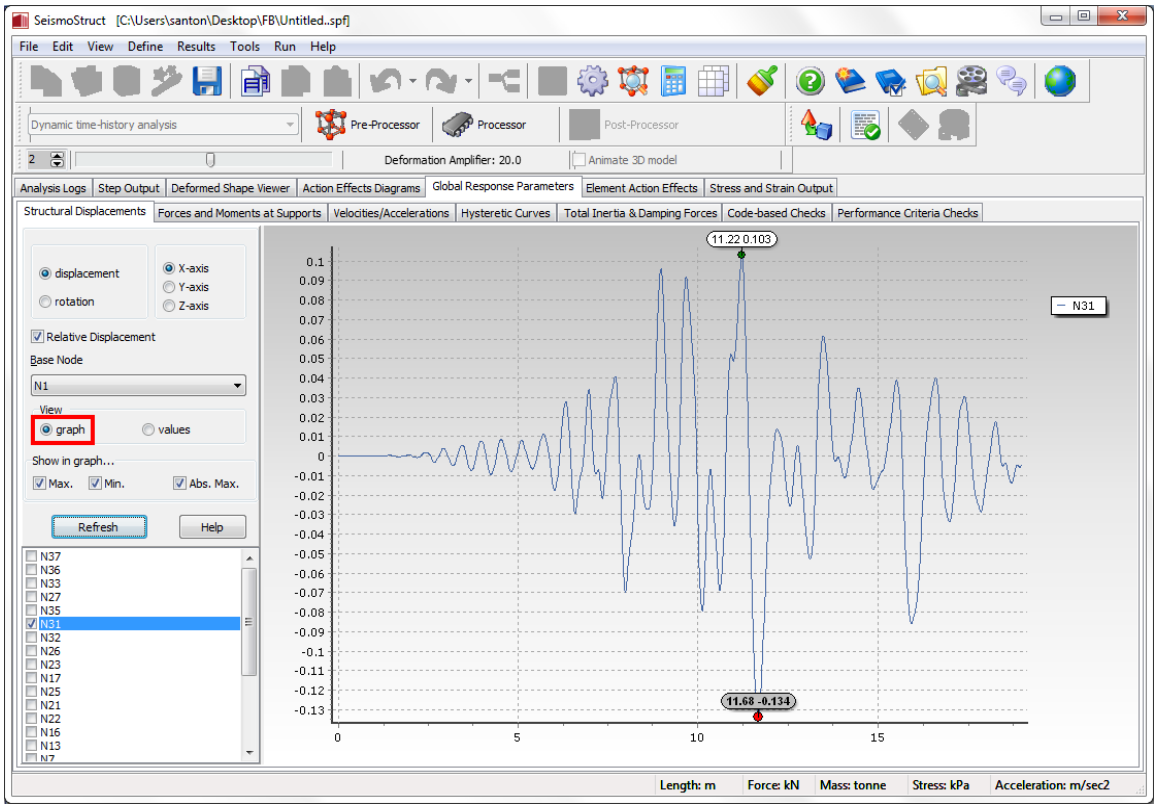


NOTE: The supports reactions should evidently be equal to the internal forces of the base elements that are connected to the foundation nodes. In other words, one would expect the values obtained in *Forces and Moments at Supports* to be identical to those given in the *Element Action Effects* for the elements connected to the foundations. However, some factors may actually lead to differences in these two response parameters: i) 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; ii) in dynamic analyses featuring tangent stiffness proportional equivalent viscous damping, and in some cases only (typically, cantilevers with low/zero axial load), it may happen that differences between elements internal actions and support reactions are observed, due to spurious numerical responses (associated to the fact that the tangent stiffness proportional damping behaves hysteretically and thus may develop damping even for velocities equal to zero); iii) the presence of offsets.

All of these parameters are briefly described hereafter:

Structural displacements

The user can obtain the displacement results of any given number of nodes, relative to one of the six available global degrees-of-freedom. Note that in dynamic analysis it is advisable for relative (with respect to a support), rather than absolute nodal displacements to be plotted. Indeed, due to the unrealistically large rigid body deformation of the foundation nodes (resulting from the uncorrected/unfiltered double-integration of the acceleration time-history), absolute displacements provide little information on the actual structural response characteristics, for which reason they are usually not considered when post-processing dynamic analysis.

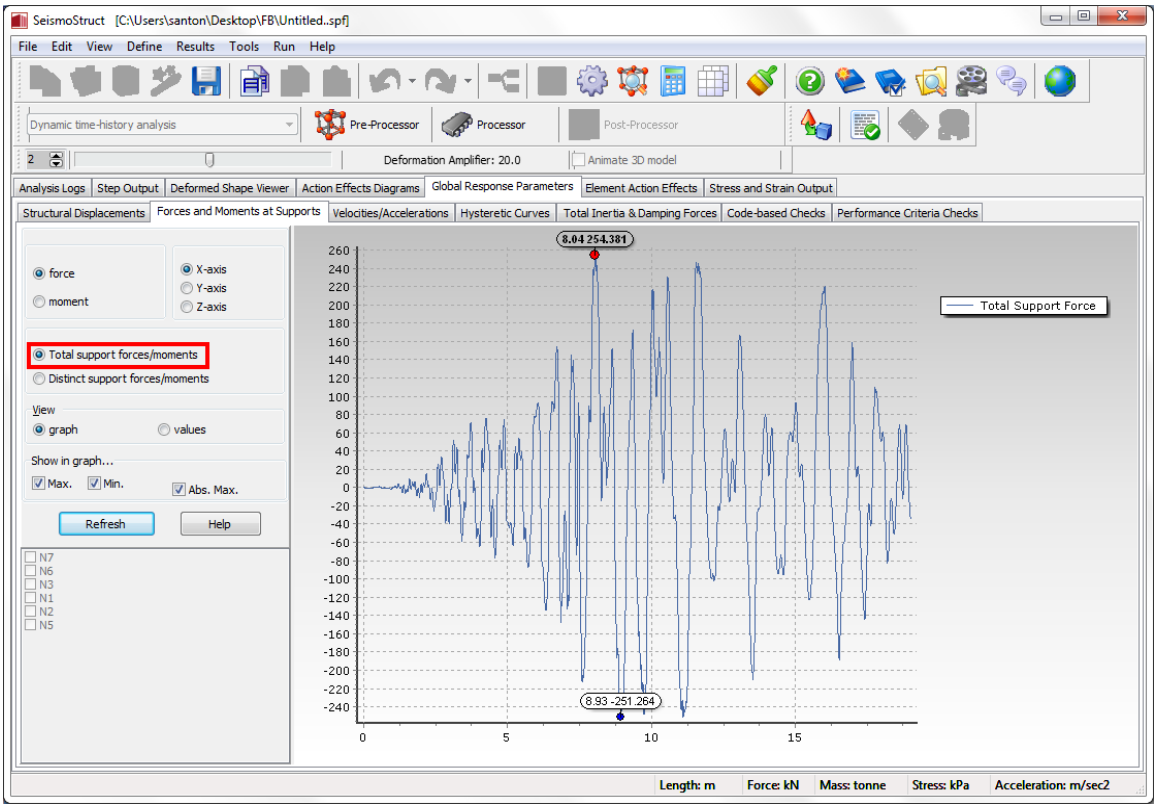


Global Response Parameters – Structural displacements

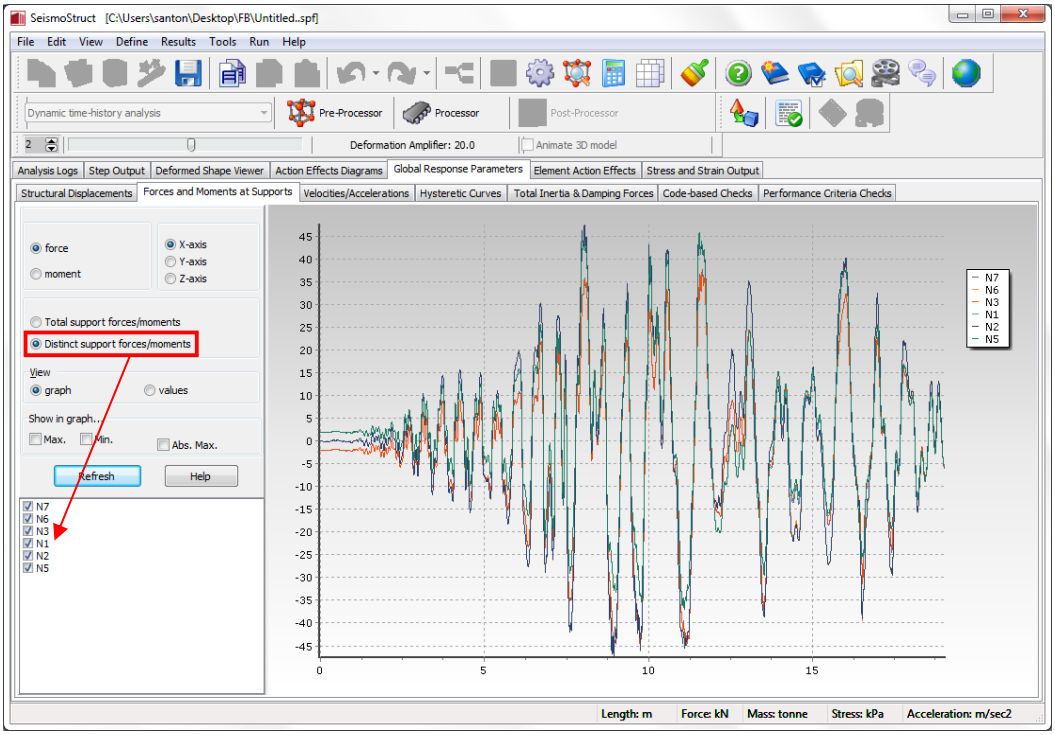
Forces and Moments at Supports

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.

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



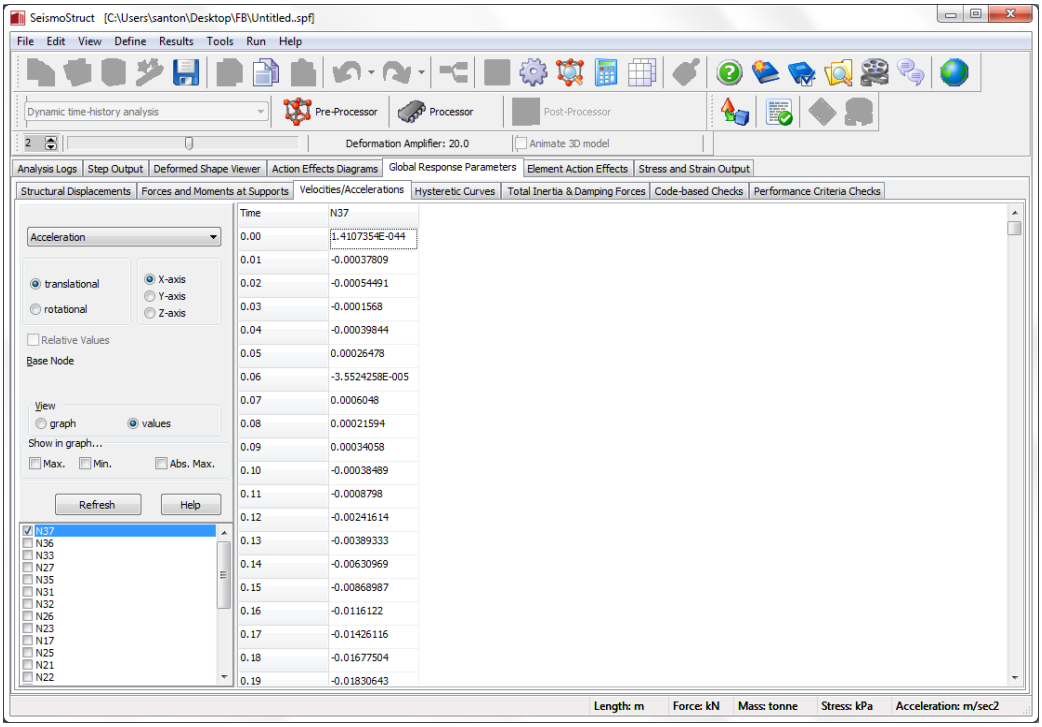
Global Response Parameters – Forces and Moments at Supports (total support)



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

Nodal Accelerations and Velocities

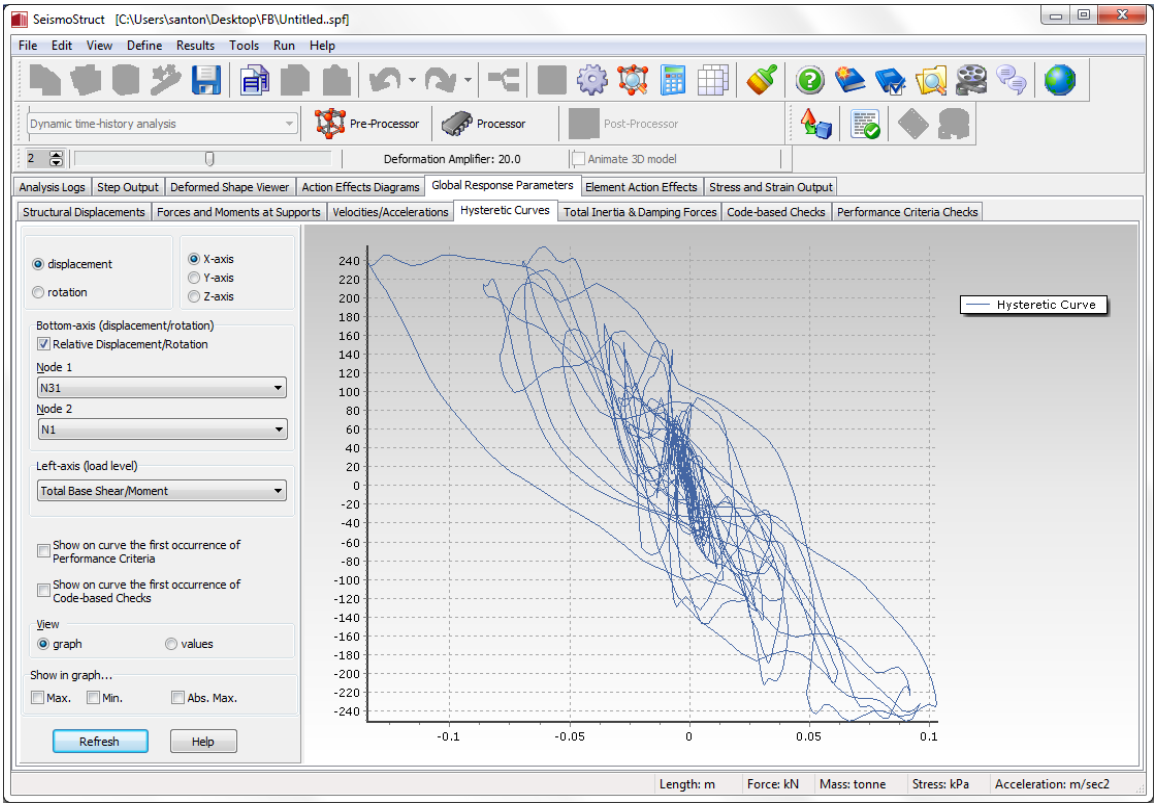
In dynamic time-history analyses, the response nodal accelerations and velocities can be obtained in exactly the same manner as nodal displacements are. The possibility of obtaining relative, as opposed to absolute, quantities is also available. The latter modality is usually adopted when accelerations are selected, whilst the former is usually considered when looking at velocity results.



Global Response Parameters – Accelerations / Velocities

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 analysis, such a plot represents the structure's capacity curve, whilst in time-history analysis this usually reflects the hysteretic response of the model. The possibility for relative displacement output is also available, as this is useful for the case of dynamic analysis post-processing.



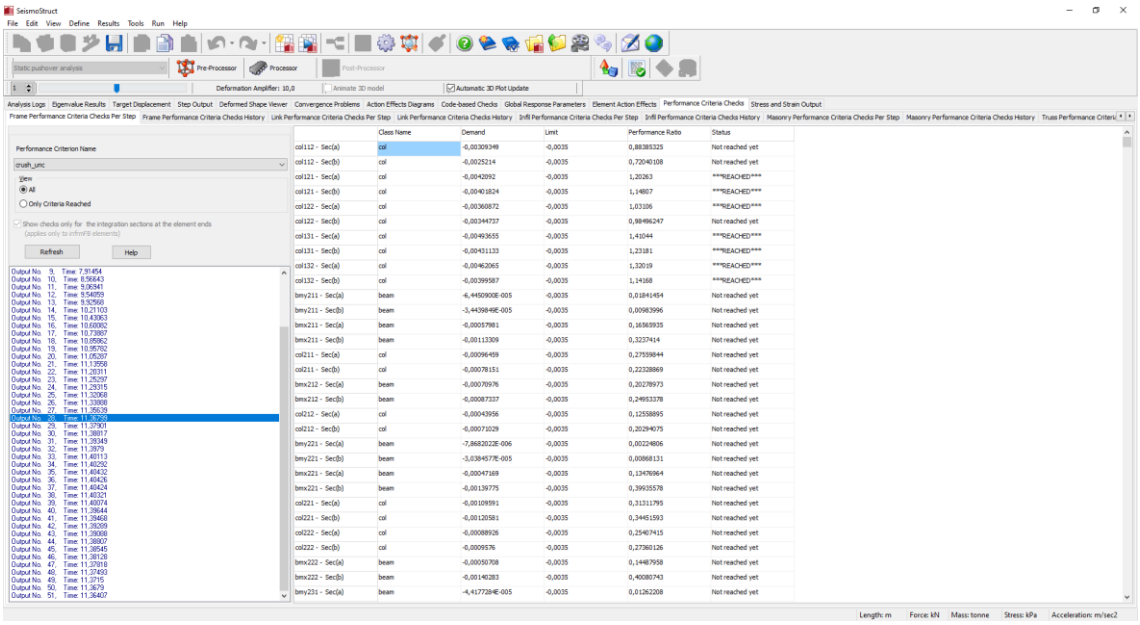
Global Response Parameters – Hysteretic Curves

Total Inertia & Damping Forces

Here, it is possible for the user to obtain the total values of inertia and viscous damping forces mobilised at every given time-step of a dynamic time-history analyses. It is noted that total viscous damping forces (which are the product, at every analysis step, of the damping matrix with the velocity vector) can be computed as the difference between the total internal forces (which are the product, at every analysis step, of the stiffness matrix with the displacement vector) and the total inertia forces (which are the product, at every analysis step, of the mass matrix with the acceleration vector). Evidently, the total internal forces are equal to the Forces and Moments at Supports, given above, and when no viscous damping is defined then the total inertia forces are simply equal to the forces at the supports.

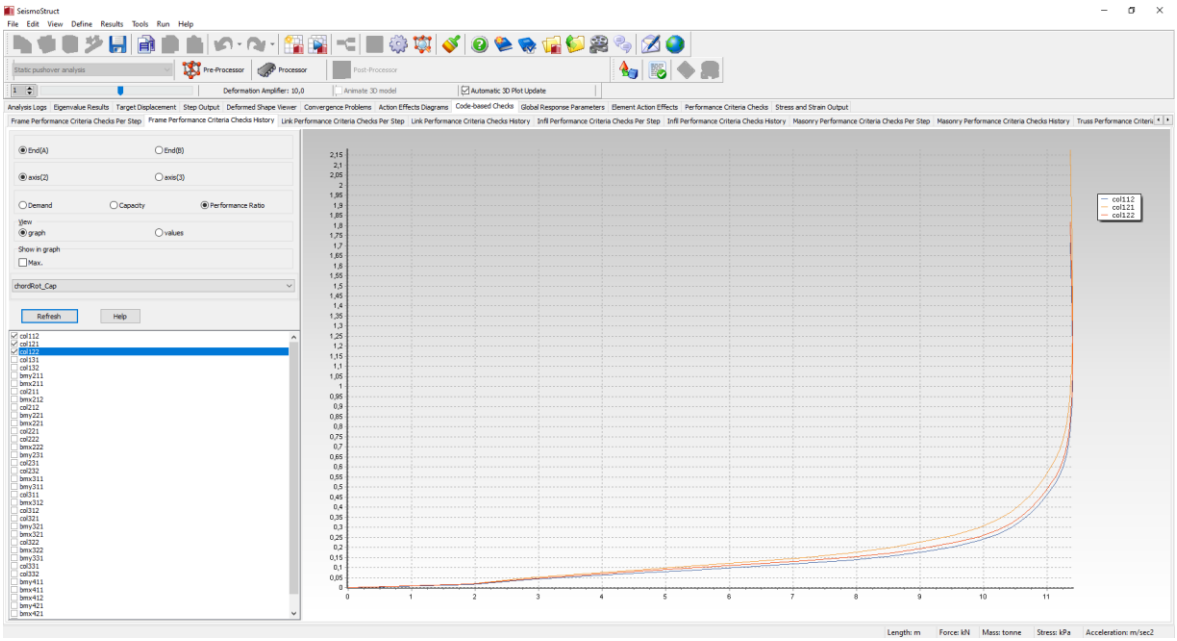
PERFORMANCE CRITERIA CHECKS

Here, it is possible for the user to perform the Performance Criteria Checks Per Step. Different tabs for Frame, Link, Infill, Masonry and Truss Performance criteria Checks Per Step are available. First, he/she has to select the performance criterion name from the drop-down menu. Then, it is necessary to select the step of the analysis (e.g. a particular limit state). Regarding the view options, the results can be displayed for all the elements or only for those elements that have reached the criterion selected.



Performance Criteria Checks - Performance Criteria Checks Per Step

In addition, the user may extract the Performance Criteria Checks History of the structural members. Different tabs for Frame, Link, Infill, Masonry and Truss Performance Criteria Checks History are available. Users have to select the performance criterion name from the drop-down menu, then, they should select the element and click the Refresh button. The results can be displayed in the form of a chart or a table. Finally, maximum values can be displayed in the selected chart.



Performance Criteria Checks - Performance Criteria Checks History

ELEMENT ACTION EFFECTS

Depending on the type of elements employed in the structural model, there can be up to ten kinds of *Element action effects* results (subdivided into four 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 only if one frame element has been employed to represent a given per column or beam, that is, only if there is a one-to-one correspondence between the model and the structure (or some of its elements). Such approach is possible when infrmFB are used, thus allowing the direct employment of element chord rotations in seismic code verifications (see e.g. Eurocode 8, NTC-18, KANEPE, FEMA-356, ATC-40, etc). When the structural member has had to be discretised in two or more frame elements, then users need to post-process nodal displacements/rotation in order to estimate the members chord-rotations [e.g. Mpampatsikos et al. 2008].

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.

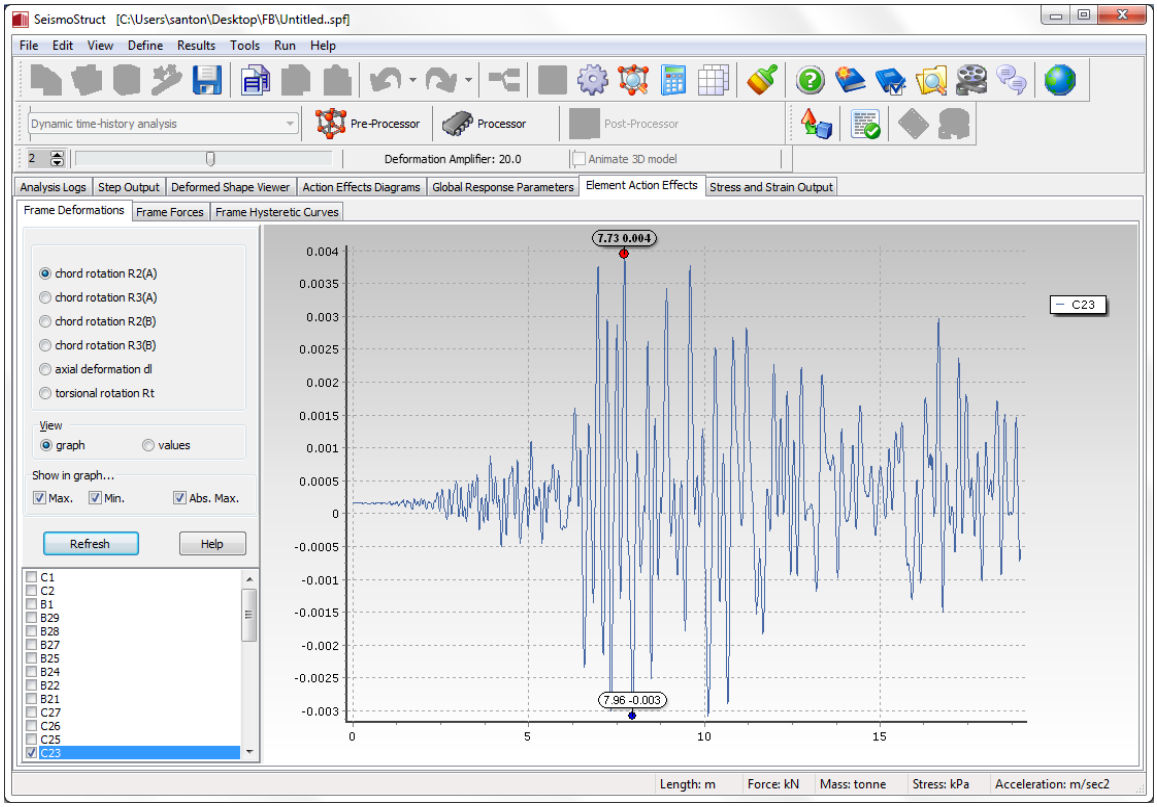
NOTE 4: In principle, the internal forces developed by frame elements during dynamic analysis should not exceed their static capacity, derived through a pushover analysis or hand-calculations. However, some factors may actually lead to differences: i) if cyclic strain hardening of the rebars takes place, then this may lead to higher "dynamic flexural capacities", in particularly for what concerns the comparison with hand-calculations (where strain hardening is normally not accounted for). ii) if equivalent viscous damping is introduced, then the structure/elements may deform less, hence elongate less, developing higher axial load, and thus, again, higher "dynamic flexural capacity". iii) if the elements feature distributed mass, then their bending moment diagram developed during dynamic analysis will differ from its static analysis counterpart, and hence the shear forces cannot really be compared (however, moments still can).

NOTE 5: SeismoStruct 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.

NOTE 6: Since in the modeling of infill panel in SeismoStruct two internal struts are used in each direction, in order to get the total strut infill panel force users need to add the values in two struts.

Frame elements – Deformations

The deformations incurred by inelastic (infrm, infrmPH) and elastic (elfrm) 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 A), the axial deformation and the torsional rotation.



Element Action Effects – Frame Deformations

> **NOTE: Elastic frame elements are always listed after their inelastic counterparts, even if the former alphabetically precedes the latter.**

Frame elements – Forces

The internal forces developed by inelastic (infrm, infrmPH) and elastic (elfrm) 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 A). 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).

> **NOTE: Elastic frame elements are always listed after their inelastic counterparts, even if the former alphabetically precedes the latter.**

Frame elements – Hysteretic Curves

Hysteretic plots of deformation vs. internal forces developed by inelastic (infrm, infrmPH) and elastic (elfrm) frame elements, as computed in their local co-rotational system of reference, are provided.

Truss elements – Forces and Deformations

The axial deformations incurred and axial forces developed by truss elements are provided here, including also the hysteretic plots.

Rack elements – Deformations

The deformations incurred by rack elements, as computed in their local co-rotational system of reference, are provided. The values refer to the deformations, rotations and warp at the end-nodes of each element (referred to as A and B, as indicated in Appendix A), the axial deformation and the torsional rotation.

Rack elements – Forces

The internal forces developed by rack elements, as computed in their local co-rotational system of reference, are provided. The values refer to the internal forces (axial and shear), moments (flexure and torsion) and bi-moments developed at the end-nodes of each element, referred to as A and B (see in Appendix A). 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).

Masonry elements – Deformations

The deformations incurred by masonry elements, as computed in their local co-rotational system of reference, are provided. The values refer to the chord rotations and shear deformation at the end-nodes of each element (referred to as A and B, as indicated in Appendix A), the axial deformation and the torsional rotation.

Masonry elements – Forces

The internal forces developed by masonry 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 A). 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).

Masonry elements – Hysteretic Curves

Hysteretic plots of deformation vs. internal forces developed by masonry elements, as computed in their local co-rotational system of reference, are provided.

Link elements – Deformations

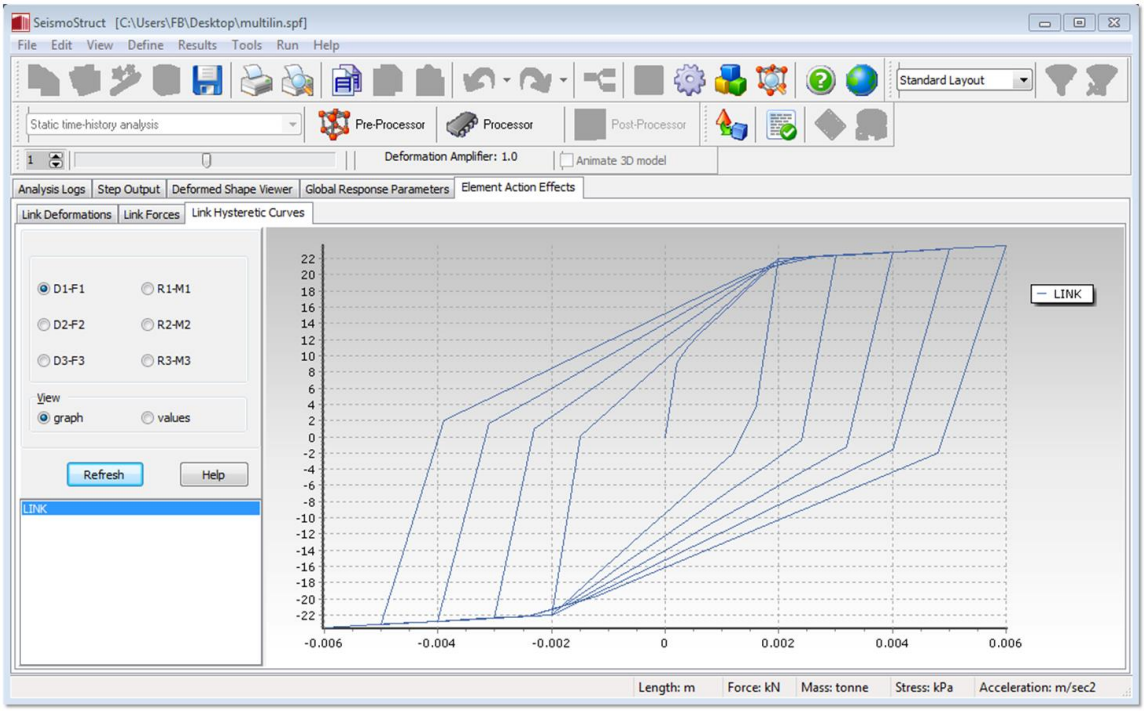
The deformations computed in link elements can be obtained. These consist of three displacements and three rotations, each of which defined with regards to the three local degrees-of-freedom of the link, the definition of which is described in *Pre-Processor* > *Structural Geometry* > *Element Connectivity*.

Link elements – Forces

The internal forces developed in link elements can be obtained. These consist of three forces and three moments, each of which defined with regards to the three local degrees-of-freedom of the link, the definition of which is described in *Pre-Processor* > *Structural Geometry* > *Element Connectivity*.

Link elements – Hysteretic Curves

Hysteretic plots of deformation vs. internal forces developed in link elements, as defined with regards to the three local degrees-of-freedom of the link, the definition of which is described in *Pre-Processor* > *Structural Geometry* > *Element Connectivity*, can be obtained.



Element Action Effects – Link Hysteretic Curves

Infill elements – Deformations

The axial (i.e. diagonal) deformations computed in struts 1 to 4 of the infill element, as well as the shear (i.e. horizontal) displacements measured in struts 5 to 6, are provided here. It is noted that struts 1, 2 and 5 refer to those that connect the first and third nodes of the infill panel (defined in *Pre-Processor* > *Structural Geometry* > *Element Connectivity*), whilst struts 3, 4 and 6 connect the second and the fourth panel corners.

Infill elements – Forces

The axial forces computed in struts 1 to 4 of the infill element, as well as the shears measured in struts 5 to 6, are provided here. It is recalled that, as discussed in *Pre-Processor* > *Element Classes*, the shear struts work only when a given diagonal is in a state of compression, hence the shear forces developed in a strut will always be single-signed (i.e. either always negative or always positive, never both).

Infill elements – Hysteretic Curves

Hysteretic plots of deformation vs. internal forces developed in infill elements are provided here, recalling once again that struts 1, 2 and 5 refer to those that connect the first and third nodes of the infill panel (defined in *Pre-Processor* > *Structural Geometry* > *Element Connectivity*), whilst struts 3, 4 and 6 connect the second and the fourth panel corners.

Shell elements – Deformations

The deformations computed in shell elements can be obtained. These consist of three displacements and three rotations, each of which defined with regards to the three local degrees-of-freedom of the shell, the definition of which is described in *Pre-Processor* > *Structural Geometry* > *Element Connectivity*. When

utilising a meshed shell element the full stiffness matrix with participation of the degrees of freedom of the internal nodes (nodes of the mesh) is factorised using a static condensation procedure. As a result only the deformations of the external four nodes are output.

Shell elements – Forces

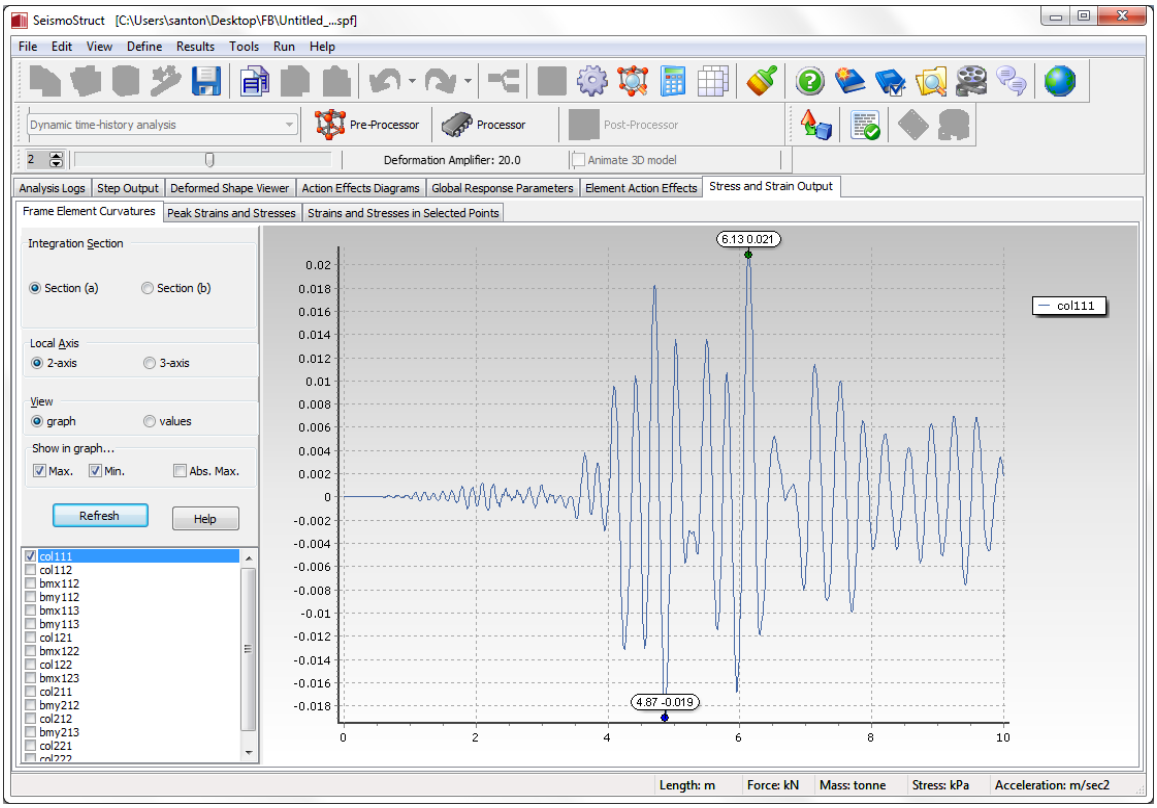
The internal forces developed in shell elements can be obtained. These consist of three forces and three moments, each of which defined with regards to the three local degrees-of-freedom of the link, the definition of which is described in *Pre-Processor* > *Structural Geometry* > *Element Connectivity*. As noted in the case of shell deformations, when using a meshed shell only the forces of the external four nodes of the element are output.

STRESS AND STRAIN OUTPUT

The material response in each of the inelastic frame elements (infrm, infrmPH) employed in the modelling of the structure can be obtained in this module.

Frame Element Curvatures

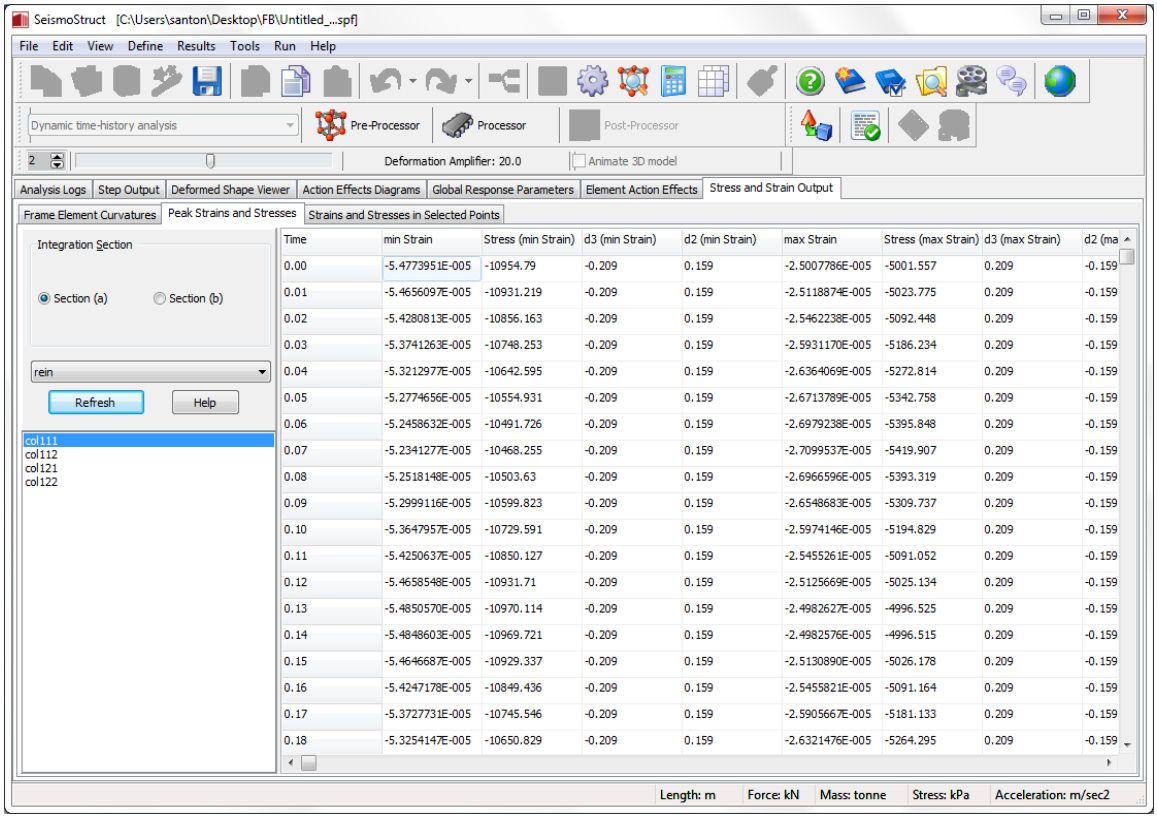
The curvatures of selected elements is provided, for each of the Integration Sections of the element, and with reference to local axes (2) or (3), defined in *Pre-Processor* > *Structural Geometry* > *Element Connectivity*.



Stress and Strain Output – Frame Element Curvatures

Peak Strains and Stresses

The maximum/minimum values of stresses and strains observed in a particular element, as well as the local sectional coordinates where these values occurred, can be obtained. The user has the possibility of selecting the Integration Section and the material type to which these results should refer to.

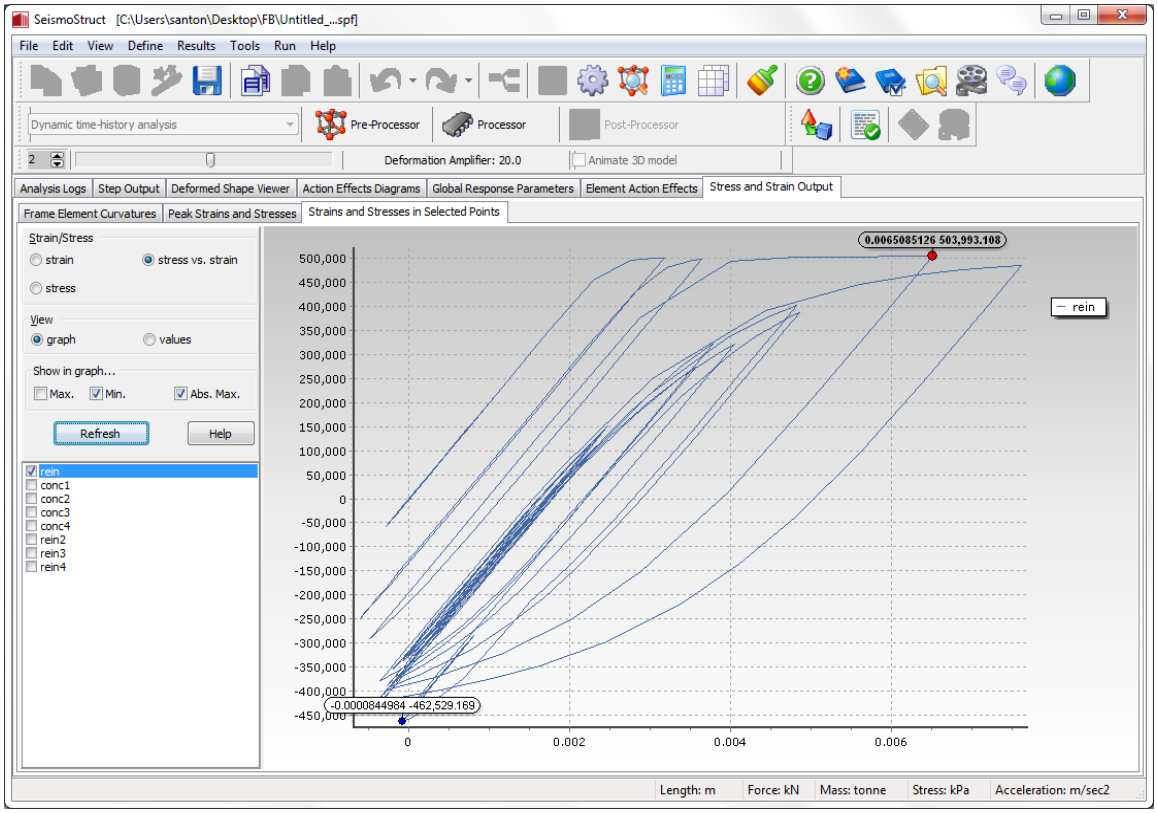


Time	min Strain	Stress (min Strain)	d3 (min Strain)	d2 (min Strain)	max Strain	Stress (max Strain)	d3 (max Strain)	d2 (ma
0.00	-5.4773951E-005	-10954.79	-0.209	0.159	-2.5007786E-005	-5001.557	0.209	-0.159
0.01	-5.4656097E-005	-10931.219	-0.209	0.159	-2.5118874E-005	-5023.775	0.209	-0.159
0.02	-5.4280813E-005	-10856.163	-0.209	0.159	-2.5462238E-005	-5092.448	0.209	-0.159
0.03	-5.3741263E-005	-10748.253	-0.209	0.159	-2.5931170E-005	-5186.234	0.209	-0.159
0.04	-5.3212977E-005	-10642.595	-0.209	0.159	-2.6364069E-005	-5272.814	0.209	-0.159
0.05	-5.2774656E-005	-10554.931	-0.209	0.159	-2.6713789E-005	-5342.758	0.209	-0.159
0.06	-5.2458632E-005	-10491.726	-0.209	0.159	-2.6979238E-005	-5395.848	0.209	-0.159
0.07	-5.2341277E-005	-10468.255	-0.209	0.159	-2.7099537E-005	-5419.907	0.209	-0.159
0.08	-5.2518148E-005	-10503.63	-0.209	0.159	-2.6966596E-005	-5393.319	0.209	-0.159
0.09	-5.2999116E-005	-10599.823	-0.209	0.159	-2.6548683E-005	-5309.737	0.209	-0.159
0.10	-5.3647957E-005	-10729.591	-0.209	0.159	-2.5974146E-005	-5194.829	0.209	-0.159
0.11	-5.4250637E-005	-10850.127	-0.209	0.159	-2.5455261E-005	-5091.052	0.209	-0.159
0.12	-5.4658548E-005	-10931.71	-0.209	0.159	-2.5125669E-005	-5025.134	0.209	-0.159
0.13	-5.4850570E-005	-10970.114	-0.209	0.159	-2.4982627E-005	-4996.525	0.209	-0.159
0.14	-5.4848603E-005	-10969.721	-0.209	0.159	-2.4982576E-005	-4996.515	0.209	-0.159
0.15	-5.4646687E-005	-10929.337	-0.209	0.159	-2.5130890E-005	-5026.178	0.209	-0.159
0.16	-5.4247178E-005	-10849.436	-0.209	0.159	-2.5455821E-005	-5091.164	0.209	-0.159
0.17	-5.3727731E-005	-10745.546	-0.209	0.159	-2.5905667E-005	-5181.133	0.209	-0.159
0.18	-5.3254147E-005	-10650.829	-0.209	0.159	-2.6321476E-005	-5264.295	0.209	-0.159

Stress and Strain Output – Peak Strains and Stresses

Strains and Stresses in Selected Points

For each of the Stress Points defined in the Output module, a complete stress-strain history can be obtained. Plots or tabled results can refer to the variation of stress/strain quantities in time (dynamic analysis) or pseudo-time (static analysis). Alternatively, stress-strain plots can also be created. Note that the material, sectional coordinates, section type and element Integration Section to which these results refer to, are implicit to the definition of each Stress Point, created in *Pre-Processor* > *Analysis Output*.

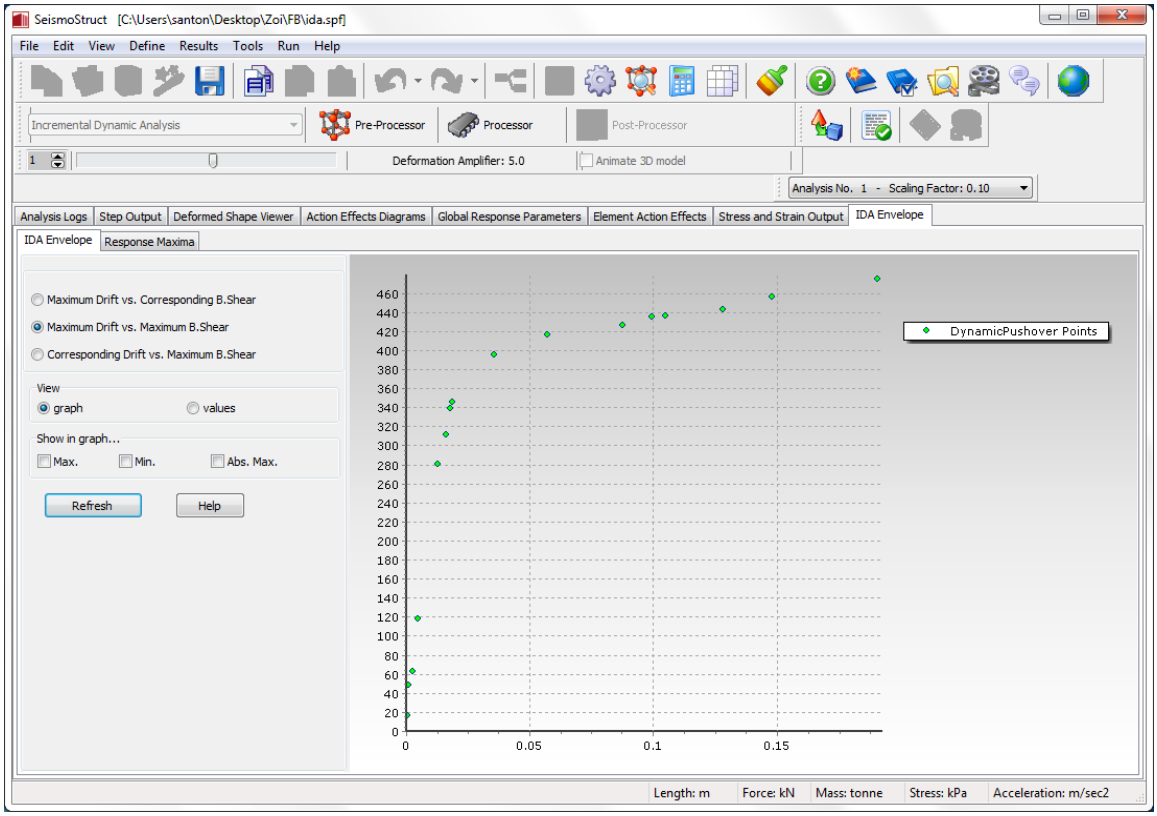


Stress and Strain Output – Strains and Stresses in Selected Points

IDA ENVELOPE

This module is visible when Incremental Dynamic Analysis has been carried out, providing the plot of peak values of base shear versus maximum values of relative displacement (drift) at the node chosen by the user (IDA parameters), as obtained in each of the dynamic runs. It is possible to plot (i) the maximum relative displacement versus the peak base shear value found in a time-window around the maximum drift (Corresponding Base Shear), (ii) the maximum relative displacement versus the maximum base shear value recorded throughout the entire time-history (Maximum Base Shear), or (iii) the maximum base shear versus the peak relative displacement value found in a time-window around the maximum shear (Corresponding Drift). The time-window is specified by the user at the IDA parameters module of the pre-processor.

In addition, it is equally possible for users to obtain in this module the envelopes of a number of additional response quantities, such as displacements, velocities, accelerations, reactions, member deformations and member internal forces.



IDA Envelope – Maximum Drift vs. Maximum B.Shear

SeismoStruct Batch Facility

CREATING NEW INPUT FILES WITH THE SPF CREATOR

A special facility for creating multiple SPF files on-the-fly is available from the main menu (Tools > Open SPF Creator...) or through the corresponding toolbar button  .

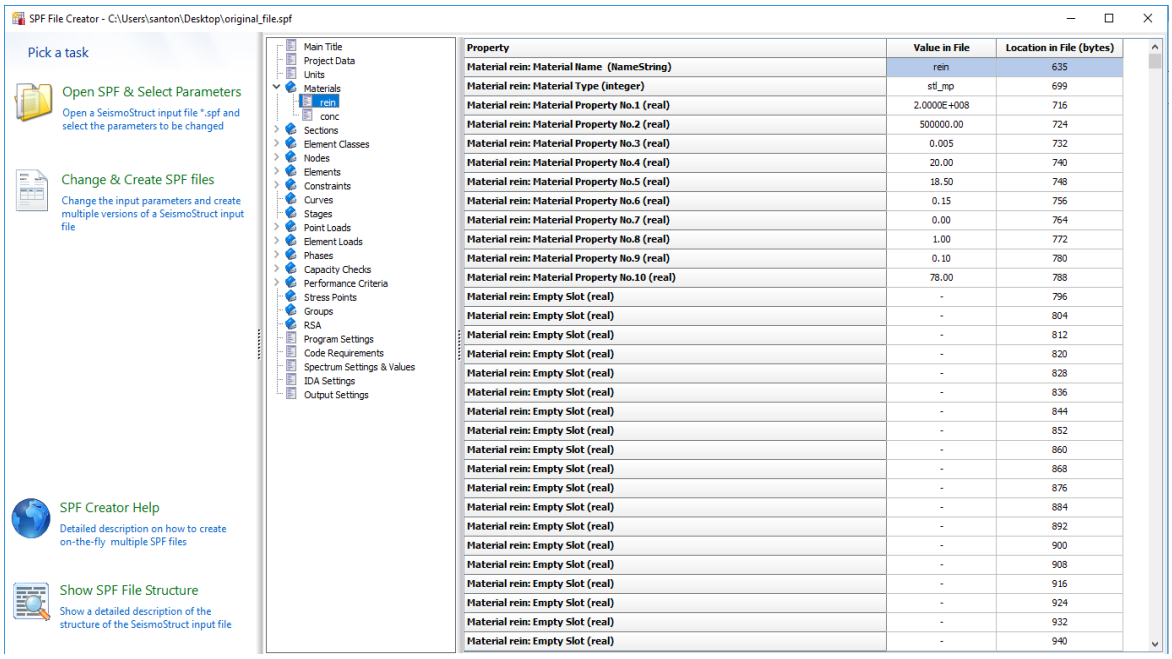
NOTE: SPF Creator has been designed as an independent application and can also be opened from the Windows Start menu without the need to open SeismoStruct.

With the SPF Creator facility users are able to easily and quickly create several new SeismoStruct input files, by adapting an existing one. It is noted that no programming or scripting knowledge is required for these operations, since the program takes care of everything and automatically creates the new files.

Upon opening the program, the SPF Creator Main Window will appear. With the *Open SPF & Select Parameters* button, users may load their base SPF file, which will be used as the template to create all the new SeismoStruct input files.

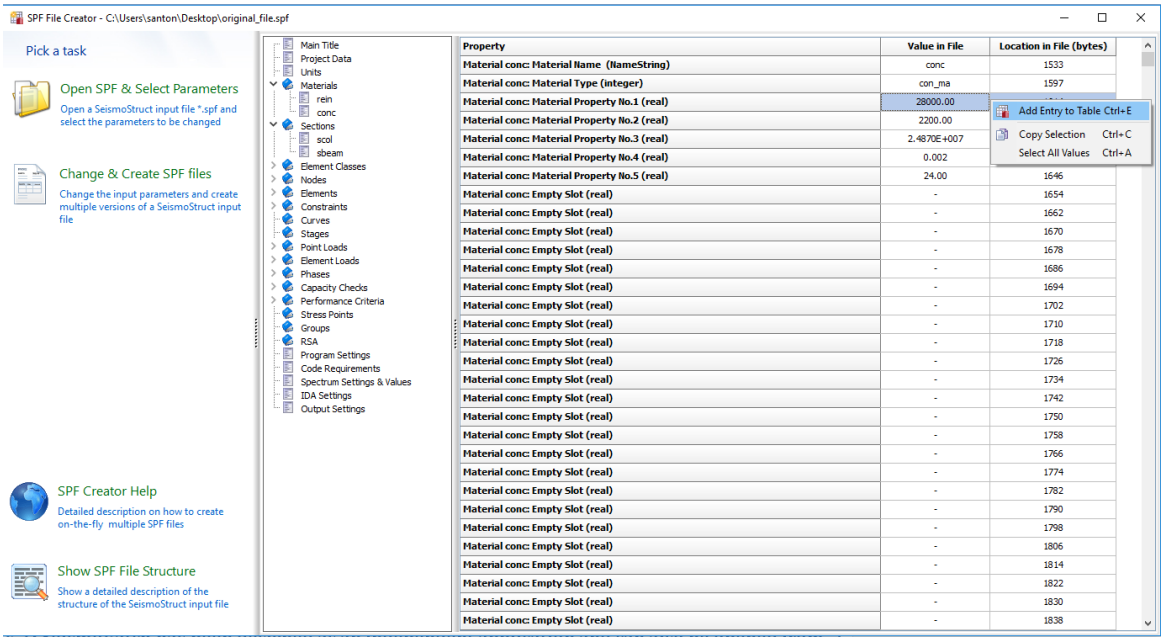
NOTE: SeismoStruct input files are binary files, i.e. non-text files, where all the data structures are stored as a sequence of bytes. A detailed description of the structure of the file format (*.spf) can be found by clicking the *Show SPF File Structure* button.

The structure of the loaded input file is displayed as a tree-view at the left of the screen, starting from the Main Title and the general Project Data, to the *Materials, Sections* and *Element Classes*, through to the *Output Settings*. By selecting each branch of the tree-view all the data of a particular record, which includes properties of the structural model, are displayed at the right of the screen. These properties can then read and modified to create the new SPF files.



SPF Creator Main Window

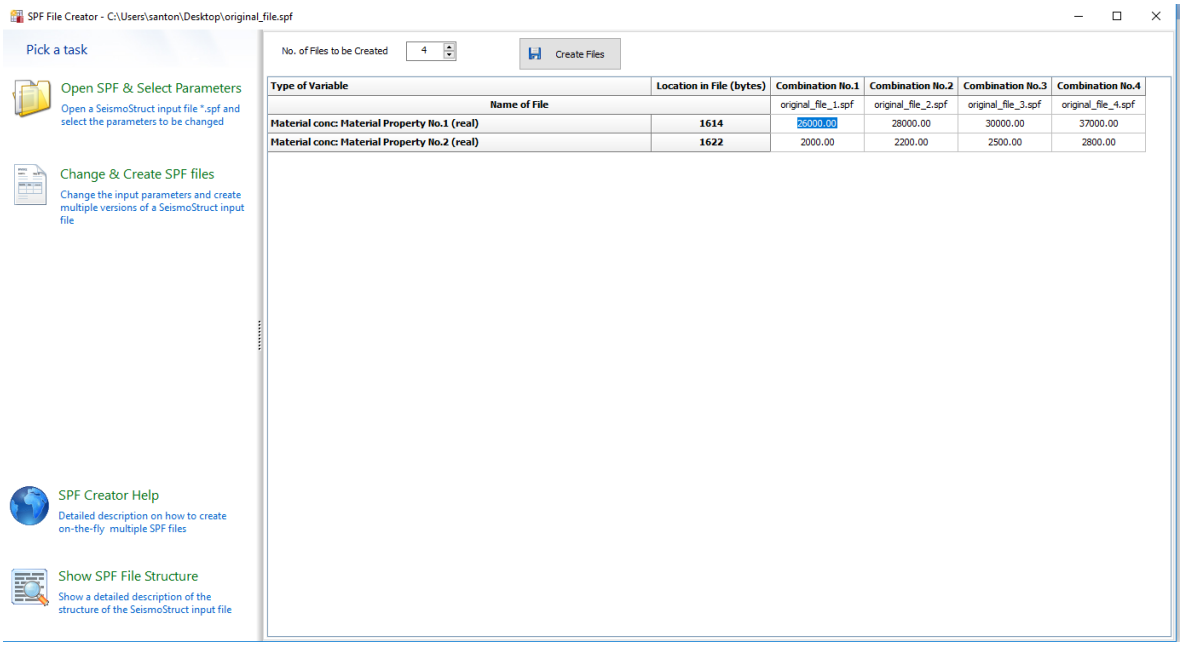
Users may select the properties, for which multiple values are to be introduced, by right-clicking on each value and selecting the *Add Entry to Table* command, as shown in picture below. All the selected properties are added to a new table on the *Change & Create SPF file* tab.



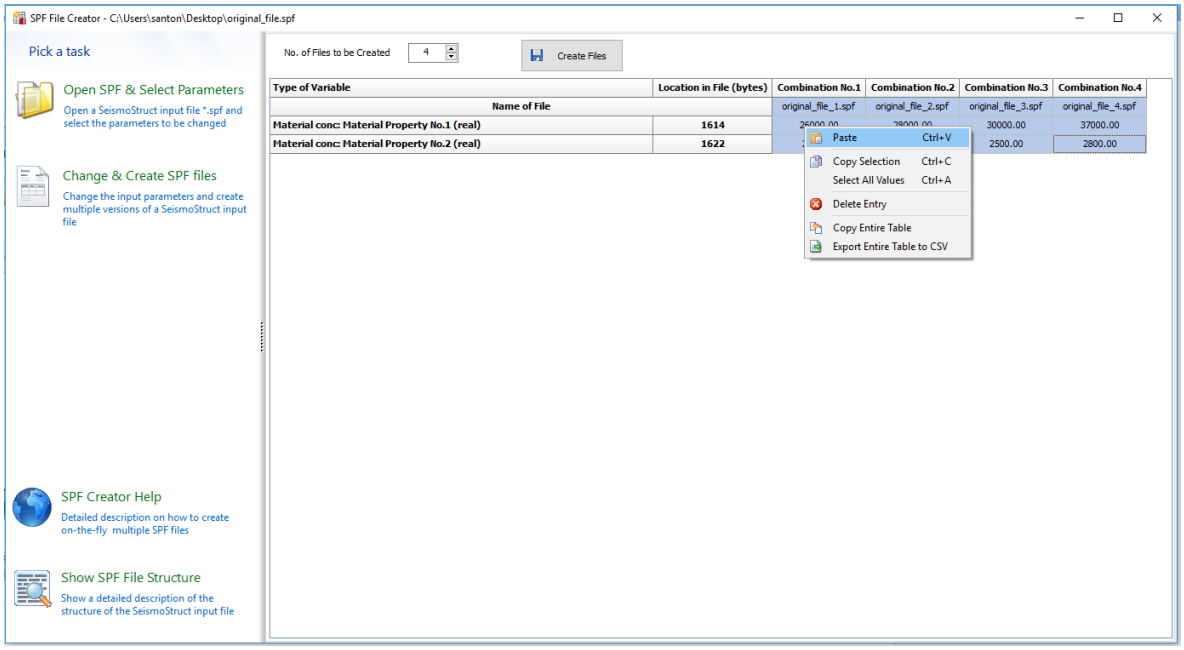
Adding properties of the SPF file to the *Change & Create SPF file* table

In the *Change & Create SPF file* tab the table entries can then be modified either directly on the table (by clicking on each table cell), or by copying and pasting to spreadsheet applications, such as MS Excel. The number of files to be created are determined by the *No. of Files to be Created* parameter.

After selecting the values of the selected properties, users are able to create the new files from the *Create Files* button. The files are created automatically in the folder, where the original file exists.



Direct change of the properties on the *Change & Create SPF file* table



Copying and Pasting data to the *Change & Create SPF file* table by the right-click commands

SeismoBatch

A special batch facility called SeismoBatch has been developed and introduced in SeismoStruct in order to facilitate the automatic execution of numerous analyses in sequence. SeismoBatch is accessed from the main menu (Tools > Open SeismoBatch...) or through the corresponding toolbar button  .

NOTE 1: SeismoBatch has been designed as an independent application and can also be opened from the Windows Start menu without the need to open SeismoStruct.

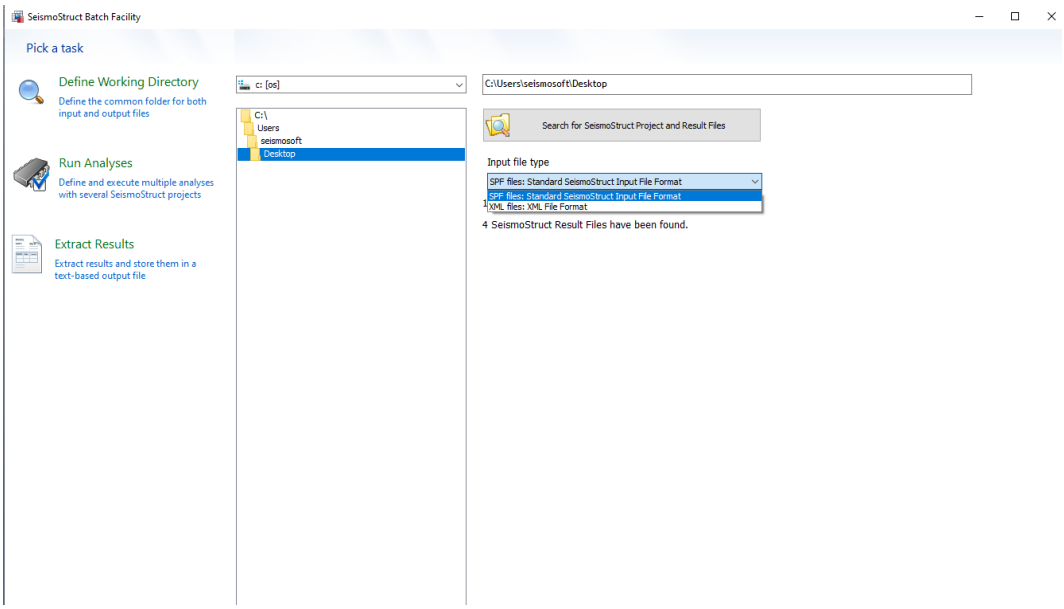
The batch facility is organised in three modules; the first is used to select the working directory, where the SeismoStruct or XML input files are saved, the second is used to run the analyses, while in the third module users may extract the analysis results that they need. Moving from one module to another can be done though the buttons of the menu at the left of the window.

NOTE 2: One very important feature of SeismoBatch is the ability to suppress the warning messages at the beginning of the analysis, in which way the execution is always carried out. This is of particular importance, when a large series of analysis is to be carried out without the presence of the user. Suppressing the warning messages can be done within the General tab of the Projects Settings of SeismoStruct.

NOTE 3: Even if the user chooses to display warning messages at the beginning of the analysis, these are automatically closed after 2 minutes, if there is no input by the user. This is particularly useful in SeismoBatch, since the sequence of the analyses does not stop if a warning message appears.

DEFINING THE WORKING DIRECTORY IN SEISMOBATCH

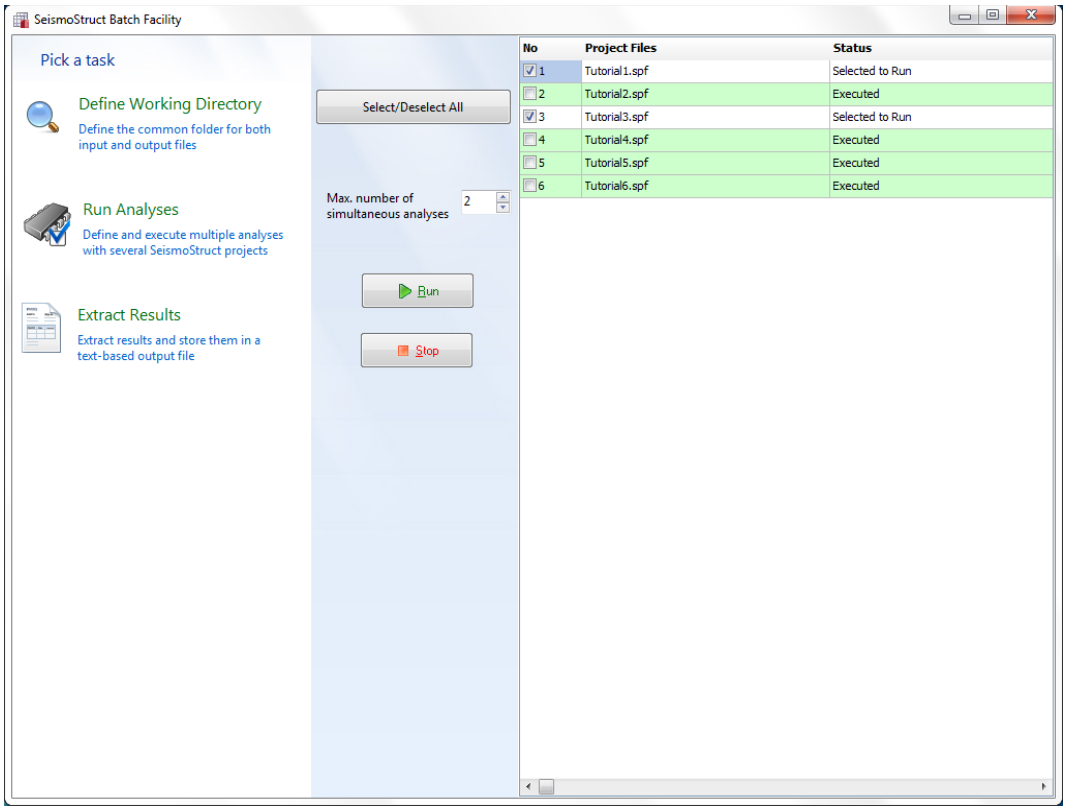
Users may select the working directory from the corresponding folder browser and whether the input file is Standard SeismoStruct or XML file format. When clicking on the *Search for SeismoStruct Project & Results files* button, the program outputs the number of project/XML and results files found on the folder.



Define Working Directory module

RUNNING THE ANALYSES FROM SEISMOBATCH

In this module users may select which of the analyses are to be carried out; this is done by checking or unchecking the checkboxes on the table, where a list of all the *.spf project files is shown. The program automatically selects all the projects, where a results file is not found. On the contrary, when a SeismoStruct results file exists, which means that the analysis has already been executed, the relevant entry is unchecked and identified with a green colour. Further, when a user selects not to run an analysis, although the output file does not exist, the corresponding entry is coloured grey for easier visualisation.



Run analyses module

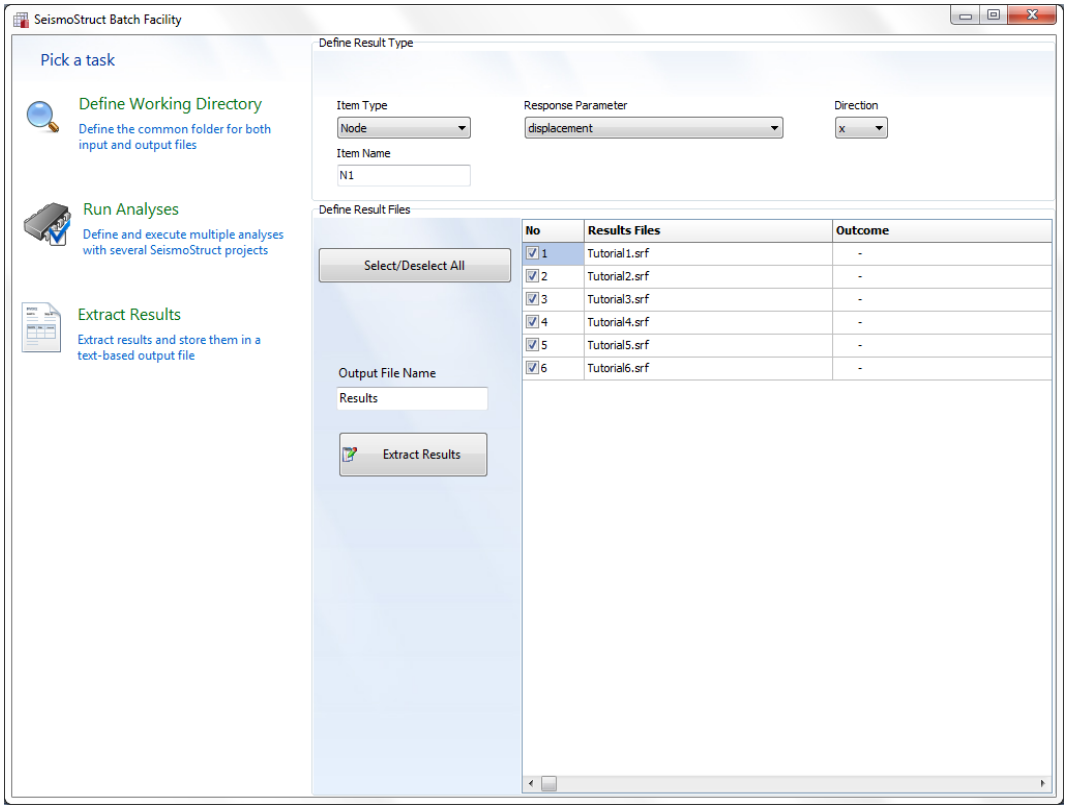
Running the analyses is done by clicking the *Run* button. The analysis that is running at any time is denoted with red, whilst green are coloured the entries of the analyses that have already been executed, and white are the projects that have not been carried out yet. The option of simultaneously executing multiple analyses has been introduced from SeismoBatch 2016 in order to take full advantage of multi-core processors.

NOTE: The analyses carried out with the batch facility employ the SeismoStruct Engine, which is SeismoStruct's solver without the graphical environment. Consequently, the batch analyses run much faster than the ones carried out with SeismoStruct, because no time is spent on the graphical updating of the application and other Windows related functions.

EXTRACTING RESULTS FROM SEISMOBATCH

In this module users may extract simultaneously and for all the selected analyses the analysis results that they need. What should be specified is the type of quantity (i.e. the node or element), the response parameter (e.g. absolute or relative displacement or rotation), and the direction in which the results extraction is to take place. The results will be stored in the output file name, which is a text-based file with the results arranged in columns (one pair of data columns, e.g. time & displacement, for each of the analyses).

The output file then can be opened with any text editor or a spreadsheet application, such as Microsoft Excel. Note that the results from the text-editor can be easily copied and pasted to Excel.



Extract Results module

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.

Abdulridha, A., Palermo, D., Foo, S., & Vecchio, F. J. (2013). Behavior and modeling of superelastic shape memory alloy reinforced concrete beams. Engineering Structures, 49, 893-904.

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 7-16 [2017] American Society of Civil Engineers: Minimum Design Loads and Associated Criteria for Buildings and Other Structures, Chapter 6, “Tsunami Loads and Effects”.

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.

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.

Baiguera M., Rossetto T., Robertson I., and Petrone C. [2019] "Towards a tsunami nonlinear static analysis procedure for the ASCE 7 standard", In Proc. Of SECED Conference, 9-10 September 2019, Greenwich, London, UK.

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.

Bouc, R. [1967]. "Forced vibration of mechanical systems with hysteresis". Proceedings of the Fourth Conference on Nonlinear Oscillation. Prague, Czechoslovakia. p. 315.

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.

Butterfield, R., Gottardi, G. [1994] "A complete three-dimensional failure envelope for shallow footings on sand," Géotechnique, Vol. 44, No. 1, pp. 181-184.

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.

Christopoulos C., Tremblay R., Kim H.-J., Lacerte M. [2008] "Self-Centering Energy Dissipative Bracing System for the Seismic Resistance of Structures: Development and Validation", *Journal of Structural Engineering*, Vol. 134, No. 1, pp. 96-107.

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.

Correia, A.A., Paolucci, R. [2019] "A 3D coupled nonlinear shallow foundation macro-element for seismic soil-structure interaction analysis," submitted to *Earthquake Engineering and Structural Dynamics*.

Correia, A.A., Pecker, A. [2019a] "Failure mechanism for laterally loaded flexible piles based on yield design theory," submitted to *Géotechnique*.

Correia, A.A., Pecker, A. [2019b] "Nonlinear pile-head macro-element for the seismic analysis of structures on flexible piles," submitted to *Bulletin of Earthquake Engineering*.

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.

EC8 – Part 5 [2003] *Eurocode 8: Design of structures for earthquake resistance – Part 5: Foundations, retaining structures and geotechnical aspects*, EN 1998-5:2004, European Committee for Standardization (CEN), Belgium.

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.

Foster A.S.J., Rossetto T., and Allsop W. [2017] “An experimentally validated approach for evaluating tsunami inundation forces on rectangular buildings”, Coastal Engineering, Vol. 128, pp. 44-57.

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.

Gazetas, G. [1991] “Foundation vibrations,” *Foundation Engineering Handbook*, 2nd Ed., Fang, H.Y. (Ed.), pp. 553-593, Van Nostrand Reinhold, New York, USA.

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.

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.”, ACI Structural Journal, Vol. 100, No. 6, pp. 749-757

Hellesland 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.

Ibarra L.F., Medina R.A., Krawinkler H. [2005] "Hysteretic Models that Incorporate Stiffness and Strength Deterioration." Earthquake Engineering and Structural Dynamics, Vol. 34 pp. 1489-1511.

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 Mackay D.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.

Lignos D.G.,Krawinkler H. [2011] "Deterioration Modeling of Steel Components in Support of Collapse Prediction of Steel Moment Frames under Earthquake Loading." Journal of Structural Engineering, Vol. 137, No. 11, pp. 1291-1302.

Lignos, D.G., Krawinkler, H. [2012]. "Development and Utilization of Structural Component Databases for Performance-Based Earthquake Engineering", Journal of Structural Engineering, ASCE.

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.

Mahin SA, Bertero VV.[1976] "Nonlinear seismic response of a coupled wall system." ASCE Journal of the Structural Division Vol. 102:1759–1980.

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.

Muthukumar S. and DesRoches R. [2006] "Elastoplastic A Hertz contact model with non-linear damping for pounding simulation", Earthquake Engineering and Structural Dynamics, Vol. 35, pp. 811-828

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 Engineering 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.

Petrone C., Rossetto T., and Goda K. [2017] "Fragility Assessment of a RC Structure under Tsunami Actions Via Nonlinear Static and Dynamic Analyses", Engineering Structures, Vol. 136, pp. 36–53.

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.

Qi Z.X., Eames I., Johnson E.R. [2014], "Force Acting on a Square Cylinder Fixed in a Free-Surface Channel Flow", Journal of Fluid Mechanics, Vol. 756, pp. 716–727.

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.

Ribeiro F.L.A., Barbosa A.R., Scott M.H., Neves L.C. [2014] "Deterioration Modeling of Steel Moment Resisting Frames Using Finite-Length Plastic Hinge Force-Based Beam-Column Elements." Journal of Structural Engineering, Vol. 141, No. 2, pp. 04014112.

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.

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.

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.

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 - Theoretical background and modelling assumptions

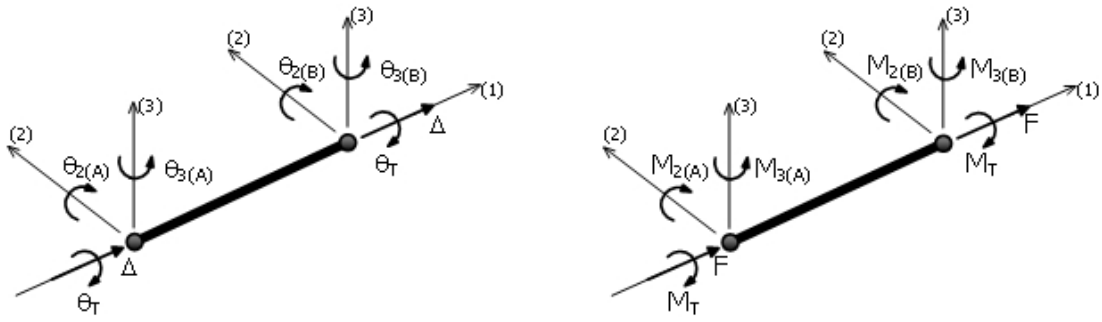
This appendix serves the purpose of providing users with a brief overview of the theoretical foundations and modelling conventions in SeismoStruct, furnishing also pointers to a number of publications where further and deeper explanations and discussion can be found.

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 SeismoStruct, 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 non-linearities 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

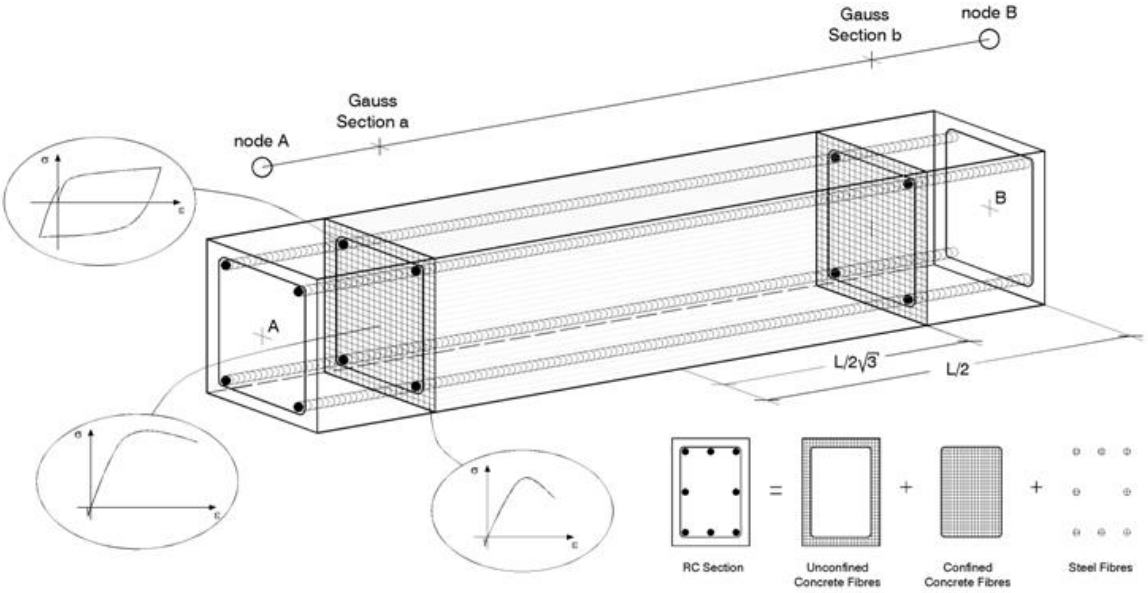
NOTE 1: If a given beam or column is anticipated to experience large deformations relative to the chord connecting its end nodes (i.e. p-delta effects), this effect can be taken into account by using 2-3 elements per member, which is enough for most cases.

NOTE 2: It is noted that when assessing single piers, geometric nonlinearity coupled with a non-baricentrical neutral axis may lead to spurious axial load levels.

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 SeismoStruct, 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 (typically 100-150) 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.



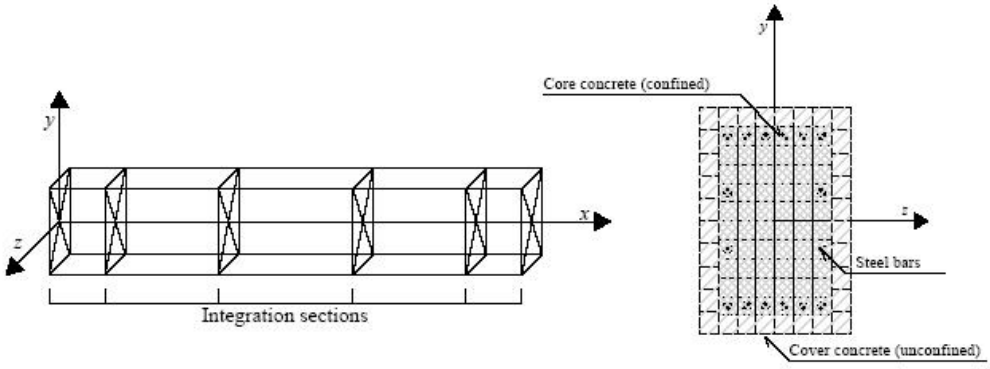
Discretisation of a typical reinforced concrete cross-section

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 minimum number of 3 Gauss-Lobatto integration sections are required to avoid under-integration, however such option will in general not simulate the spread of inelasticity in an acceptable way. Consequently, the suggested minimum number of integration points is 4, although 5-7 IPs are typically used (see figure below). 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 SeismoStruct, both aforementioned DB and FB element formulations are implemented, with the latter being typically recommended, since, as mentioned above, it does not in general 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].

In addition, the use of a single element per structural element gives users the possibility of readily employing element chord-rotations output for seismic code verifications (e.g. Eurocode 8, ASCE/SEI 7-05, etc). Instead, when the structural member has had to be discretised in two or more frame elements (necessarily the case for DB elements), then users need to post-process nodal displacements/rotation in order to estimate the members chord-rotations (e.g. Mpampatsikos et al. [2008]).

Finally, it is noted that, for reasons of higher accuracy, the Gauss quadrature is employed in those cases where two or three integration sections are chosen by the user (it is recalled that for DB elements only the former is possible), whilst Lobatto quadrature is used in those cases where four to ten integration sections are defined. Although users may and should refer to the literature (or to online resources) for further details on such rules, the approximate coordinates along the element's length (measured from its baricentre) of the integration sections is given below:

- 2 integration sections: [-0.577 0.577] x L/2
- 3 integration sections: [-1 0.0 1] x L/2
- 4 integration sections: [-1 -0.447 0.447 1] x L/2
- 5 integration sections: [-1 -0.655 0.0 0.655 1] x L/2
- 6 integration sections: [-1 -0.765 -0.285 0.285 0.765 1] x L/2
- 7 integration sections: [-1 -0.830 -0.469 0.0 0.469 0.830 1] x L/2
- 8 integration sections: [-1 -0.872 -0.592 -0.209 0.209 0.592 0.872 1] x L/2
- 9 integration sections: [-1 -0.900 -0.677 -0.363 0.0 0.363 0.677 0.900 1] x L/2
- 10 integration sections: [-1 -0.920 -0.739 -0.478 -0.165 0.165 0.478 0.739 0.920 1] x L/2

NOTE 1: It is immediate with FB formulations to take into account loads acting along the member, while this is not the case for DB approaches, where distributed loads need to be transformed into equivalent point forces/moments at the end nodes of the element (and then lengthy stress-recovery need to be employed to retrieve accurate member action-effects).

NOTE 2: Should the user wish to, it is possible to adopt a concentrated plasticity approach employing the inelastic displacement-based plastic-hinge element (infrmDBPH), as opposed to the distributed inelasticity modelling philosophy intrinsic to the other beam-column elements of SeismoStruct - for instance the inelastic force-based plastic hinge frame element (infrmFBPH) also concentrates the inelasticity at the two ends of the element, however within a fixed length of the element.

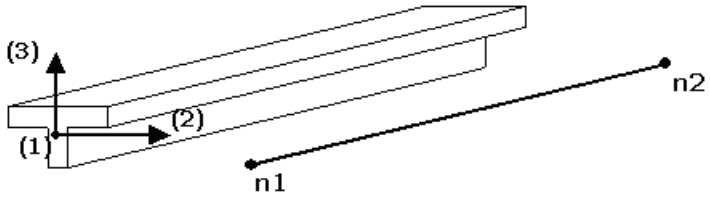
The same modelling effect can be achieved by making use of the elastic beam-column frame element (elfrm) coupled with nonlinear links placed at its end-nodes. Such modelling approach should however be used with care, since accuracy of the analysis may be compromised whenever users are not highly experienced in the calibration of the available response curves, used in the definition of link elements, the uncoupled DOFs nature of which does not also permit the modelling of the necessary moment-axial force interaction curves/surfaces.

NOTE 3: As mentioned above, the distributed inelasticity modelling, on the other hand, requires no modelling experience since all that is required from the user is to introduce the geometrical and material characteristics of structural members (i.e. engineering parameters). Its use is therefore highly recommended and will grant an accurate prediction of the nonlinear response of structures.

NOTE 4: 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 SeismoStruct, 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, SeismoStruct 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 description of SeismoStruct's sections.



Definition of a beam element with a T-section (local direction (1) along the chord axis)

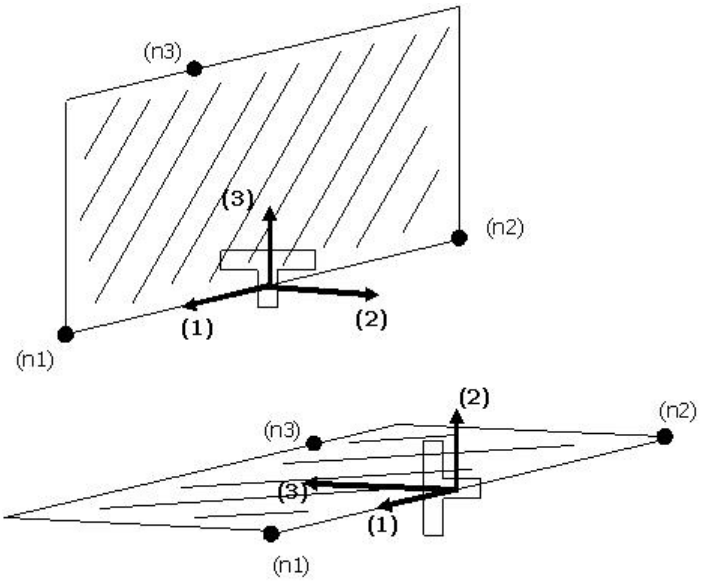
Whilst the orientation of local vector (1) results unambiguously characterised by the line joining the two end-nodes of the element (positive direction is that going from node n1 to node n2), an 'orientation object' is required in order to fully describe the orientation of the two other remaining local axes, and

thus that of the cross-section. From the software version 6 the element's orientation may be achieved through two different ways:

1. by defining a **rotation angle** (default option), which is set equal to 0 by default (models built with the Wizard facility follow this rule), or
2. by defining additional nodes, called **'orientation node'**. If the 'default' object is selected, the element's orientation is automatically computed by the program, otherwise it will depend on the position of the selected node.

NOTE: In general, the rotation angle equal to 0 means that the axis (3) is vertical. The vertical elements (axis (1) is vertical) are a special case, where angle = 0 means that the axis (3) is along the X-direction.

The orientation node allows to define the plane (1-3) in which vector (3) lays in, its direction (perpendicular to axis (1)) and orientation (pointing towards n3), as shown below. Local vector (2) was then automatically obtained through the cross-product of vectors (1) and (3), with positive direction following the so-called right-hand rule.



Orientation of a beam element with a T-section (it depends on the position of (n3))

The vast majority of structures modelled in SeismoStruct are defined in plane frames and feature vertical elements (e.g. rectangular columns, walls) with symmetrical cross-sections and horizontal T-beams that are not symmetrical around their (2) axis. Hence, the selection of the 'default' object as a 'third node' can be very advantageous.

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 SeismoStruct, all analyses (with the obvious exception of eigenvalue procedures and the Response Spectrum Analysis) 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.

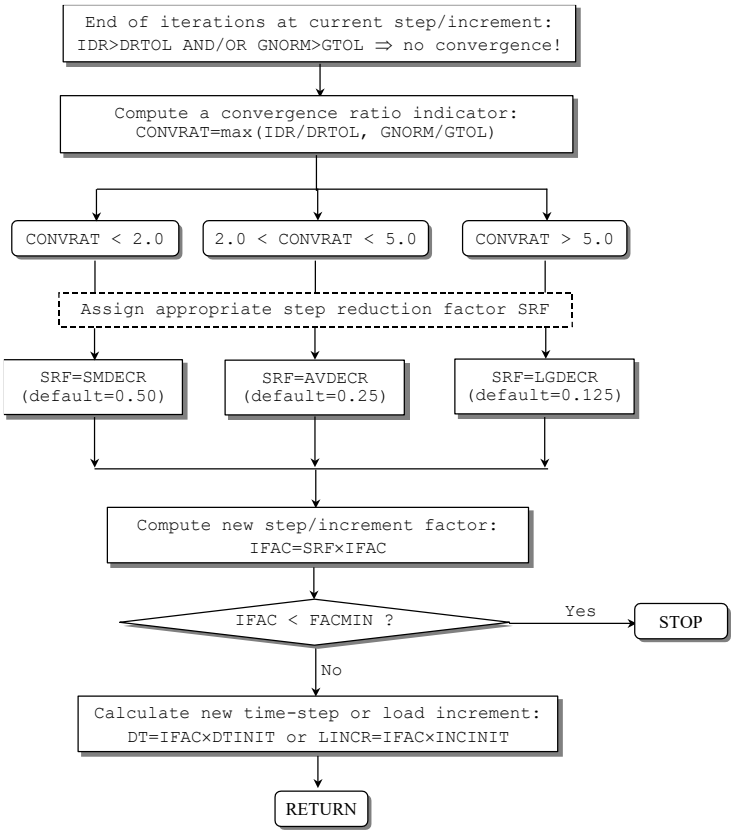
NOTE: Some element types (infrmFB, infrmFBPH, infrmDBPH & masonry) require a number of iterations to be carried in order for internal equilibrium to be reached. In some cases, the latter element loop equilibrium cannot be reached, as signalled by log flag messages elm_inv and elm_ite. Refer to *General* > *Project Settings* > *Element* Iterative Strategy 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.

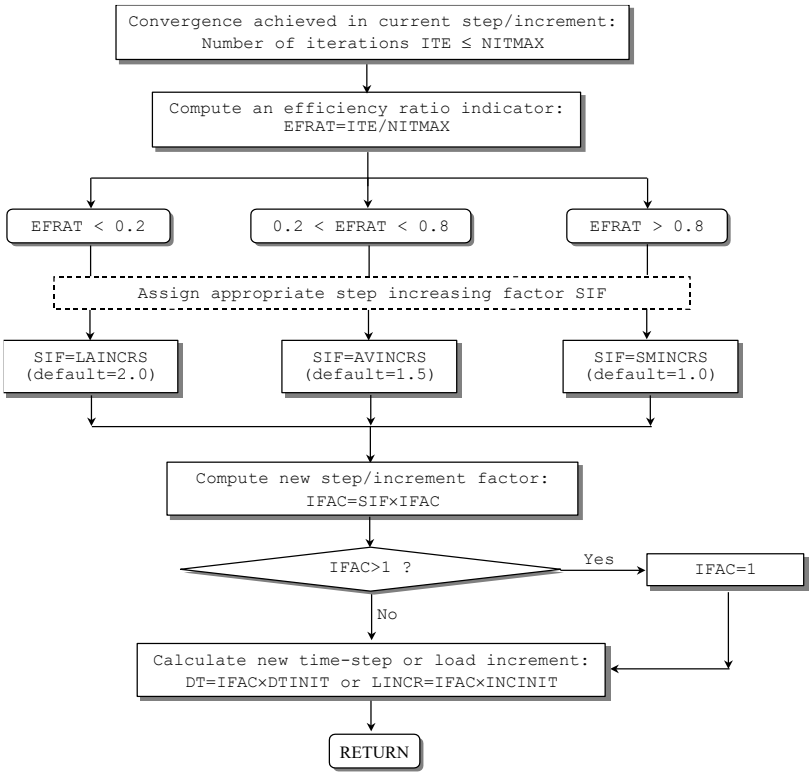
As illustrated below, at the end of a solution step or increment, a convergence ratio indicator (convrat), defined as the maximum of ratios between the achieved and the required displacement/force convergence factors (see convergence), is computed. Then, depending on how far away the analysis was from 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. The only exception to this rule occurs in cases where pushover analysis is carried out using the Automatic Response Control loading/solution algorithm, employed when users are primarily focused on the final solution rather than the load/response path required to arrive at such final equilibrium point.



Convergence criteria

Four different convergence check schemes, which make use of two distinct criteria (displacement/rotation and force/moment based), are available in SeismoStruct for checking the convergence of a solution at the end of every iteration:

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

Herein, the formulation of the two criteria employed in all four schemes is given, whilst the applicability of the latter is discussed elsewhere.

The **displacement/rotation** criterion consists in verifying, for each individual degree-of-freedom of the structure, that the current iterative displacement/rotation is less or equal than a user-specified tolerance. In other words, if and when all values of displacement or rotation that result from the application of the iterative (out-of-balance) load vector are less or equal to the pre-defined displacement/rotation tolerance factors, then the solution is deemed as having converged. This concept can be mathematically expressed in the following manner:

$$\max\left[\left|\frac{\delta\, d_i}{d_{tol}}\right|_{i=1}^{n_d}, \left|\frac{\delta\, \theta_j}{\theta_{tol}}\right|_{j=1}^{n_\theta}\right] \leq 1 \Rightarrow \text{convergence}$$

where,

- δd_i is the iterative displacement at translational degree of freedom i
- $\delta \theta_j$ is the iterative rotation at rotational degree of freedom j
- n_d is the number of translational degrees of freedom
- n_θ is the number of rotational degrees of freedom
- d_{tol} is the displacement tolerance (default = 10^{-2} mm)

- θ_{tol} is the rotation tolerance (default = 10^{-4} rad)

The **force/moment** criterion, on the other hand, comprises the calculation of the Euclidean norm of the iterative out-of-balance load vector (normalised to the incremental loads), and subsequent comparison to a user-defined tolerance factor. It is therefore a global convergence check (convergence is not checked for every individual degree-of-freedom as is done for the displacement/rotation case) that provides an image of the overall state of convergence of the solution, and which can be mathematically described in the following manner:

$$G_{norm} = \frac{\sqrt{\sum_{i=1..n}\left[\frac{G_i}{V_{REF}}\right]^2}}{n} \leq 1 \Rightarrow \text{convergence}$$

where,

- G_{norm} is the Euclidean norm of iterative out-of-balance load vector
- G_i is the iterative out-of-balance load at dof i
- V_{REF} is the reference "tolerance" value for forces (i=0,1,2) and moments (i=3,4,5)
- n is the number of dofs

NOTE: The use of a global, as opposed to local, force/moment criterion is justified with the fact that, in SeismoStruct, it is common for load vectors to feature significant variations in the order of magnitude of forces/moments applied at different degrees-of-freedom of the structure, particularly in the cases where infinitely stiff/rigid connections are modelled with link elements. Hence, the employment of a local criterion, as is done in the case of displacement/rotation criterion, would lead to over-conservative and difficult-to-verify converge checks.

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:

$$itepred = ite + \frac{\log\left(G_{tol} / G_{norm}^{ite}\right)}{\log\left(G_{norm}^{ite} / G_{norm}^{ite-1}\right)}$$

The three checks described above are usually reliable and effective within the scope of applicability of SeismoStruct, 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: Some element types (infrmFB, infrmFBPH, infrDBPH, masonry) 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 *Project Settings > Element Iterative Strategy* menu for further information on this issue.

List of SeismoStruct Convergence and Divergence Flags

Hereby a complete list of the messages that are output by the SeismoStruct 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 (*Project Settings>Global Iterative Strategy*), increase the convergence criteria values (*Project Settings>Convergence Criteria),* employ a less stringent type of convergence check (e.g. *Displacement/Rotation based* only scheme instead of *Displacement/ Rotation based AND Force/Moment based*) or decrease the loading step. The latter can be done in pushover analysis by increasing the number of steps from the *Loading Phases* module, which leads to a decrease of the load factor increment. Similarly, in time-history analysis the time step is reduced by increasing the number of time-steps (*Loading Curves* module> *Time-history Stages*> Number of steps).

NOTE: When rigid constraints (rigid links or rigid diaphragms) have been defined in the model, users are advised not to employ the Force-based convergence criterion, especially when the penalty functions method is used for the modelling of constraints. A Displacement/Rotation based criterion with more stringent displacement and rotation tolerance values may be employed, instead.

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 is 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 (*Project Settings>Global Iterative Strategy*), choose a looser convergence criteria scheme with larger convergence tolerances (*Project Settings>Convergence Criteria),* or decrease the loading step, 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 *Project Settings> Global 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 *Global Iterative Strategy* page of the *Project Settings*, choose looser convergence criteria from the *Convergence Criteria* page, or decrease the loading step.

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, infrmFBPH, infrmDBPH and masonry), without internal equilibrium having been achieved. Users are advised to either increase the number of iterations or increase the convergence tolerance from the *Element Iterative Strategy* tab of the *Project 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 load factor increments or the time-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 *Project Settings*, to decrease the global analysis load step or to increase the global convergence tolerance values.

elm_Tol: This message appears when the maximum tolerance value, as specified in the *Global Iterative Strategy* page of the *Project 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 *Global Iterative Strategy* page of the *Project Settings*. Users are advised to decrease the load increment, 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 the convergence problems that might arise are proposed.

- Apply the automatic adaptation of the norms in the Convergence criteria tab of the program's Project Settings.
- Select to show Convergence problems in the post-processor through the Project 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) and the masonry element types in the *Element Iterative Strategy* tab of the Project Settings.
- Reduce the maximum pushover displacement to 2% in general. This value should not exceed 1.00 or 1.20% for tall buildings and for stiff buildings with large shear walls.
- Assign 50 to 100 pushover analysis steps in the general case. This value should be increased in cases of demanding loading.
- Change the fracture/buckling strain for the steel materials to a very large value (e.g. 1) in the Materials module. When the fracture/buckling strain for the steel materials is reached, the rebar is deactivated and leads to a sudden drop of the building strength and convergence difficulties. There are cases, such as coupling beams, where the deformations are enormous. It is very likely that, if this strain level is reached, the rebars are deactivated (assumed as fractured) and the beams cannot sustain any gravity load and convergence cannot be reached.
- Go to the Element Class module and change the element type of all short elements (e.g. short columns, and coupling beams) from infrmFBPH or infrmFB to infrmDB. It is noted that the infrmDB element type generally provides acceptable accuracy only for short members only. Hence, if applied to short members, this does not affect the analytical results, whilst typically it leads to significantly improvements in the convergence and overall stability of the analysis.
- Increase the Maximum Number of Iterations to 70, the Number of Stiffness Updates to 60 and the Divergence Iterations to 60 in Iterative Strategy tab of the Project Settings.
- Use the elastic frame element type for the coupling beams that cause convergence problems. In such cases the elements' moment releases should released by selecting the relevant checkboxes for the M2a, M3a, M2b and M3b degrees-of–freedom, through the element's Properties window within the Element Connectivity module, 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 Project Settings.
- Increase the rigidity of the rigid diaphragms to 1.0E+13 through the Constraints tab of the Project Settings.
- If the divergence messages of the analysis are mostly Max_Tol or elm_tol, increase the Maximum Tolerance value to 1e40 in the Iterative Strategy tab of the Project Settings.
- Increase the number of fibers for the walls in the Element Class Properties window of the members within the Element Classes module.
- For taller buildings uncheck the Include Geometric Nonlinearities checkbox in the Analysis tab of the Project 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 accidentally not connected to the 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. 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. The introduction of rectangular, rather than T-shaped, beams sometimes leads to more stable solutions, however it should be noted that removing the

slab effective width weakens the beams and it could render lightly reinforced beams unable to sustain the gravity loads, hence leading to new convergence problems.

Appendix B - Analysis Types

In this appendix the available analysis types are described in details.

EIGENVALUE ANALYSIS

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 *Project Settings* menu. Evidently, no loads are to be specified.

Eigenvalue analysis is a purely elastic type of structural analysis, since material properties are taken as constant throughout the entire computation procedure and hence it is natural for elastic frame elements (elfrm) to be employed in the creation of the structural model. As described in *Pre-Processor* > *Element Classes* > *elfrm*, this type of elements do not call for the definition of material or section types, as their inelastic counterparts, being instead fully described by the values of the following sectional mechanical properties: cross-section, moment of inertia, torsional constant, modulus of elasticity and modulus of rigidity [e.g. Pilkey, 1994]. Therefore, an estimate of the vibration period corresponding to the cracked, as opposed to uncracked, state of the structure, can be readily obtained by applying reduction factors to the moment of inertia of beam and column cross-sections, as recommended by Paulay and Priestley [1992], amongst others. These factors may vary from values of 0.3 up to 0.8, depending on the type of member being considered (beam or column), loading characteristics, and structural configuration. Users are advised to refer to the work of Priestley [2003] for a thorough discussion on this matter.

If the user, however, wishes to carry out not only eigenvalue but also other types of analysis, possibly within the inelastic material response range, then he/she might prefer to build only one structural model, employing inelastic rather than elastic frame elements, that will be employed on all 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]. As a consequence, it results impossible for users to directly modify the second moment of area (or moment of inertia) of cross-sections to account for the effects of cracking, for which reason the stiffness reduction of members due to cracking should be instead simulated by changes applied to the modulus of elasticity of the concrete material (e.g. by reducing it by the same factor that one would apply to the moment of inertia of a cross-section).

NOTE 1: The use of inelastic elements in eigenvalue analysis features also the advantage of exempting the user from the onus of (manually) calculating the section mechanical properties of each element type, taking full account of the presence of longitudinal reinforcement bars within the section.

NOTE 2: Concrete confinement will increase the compressive strength of the material, and hence the stiffness of the member, leading thus to shorter periods of vibration.

NOTE 3: When running an eigenvalue analysis using Lanczos algorithm, user may be presented with a message stating: "could not re-orthogonalise all Lanczos vectors", meaning that the Lanczos algorithm, currently the eigenvalue solver in SeismoStruct, could not calculate all or some of the vibration modes of the structure. This behaviour may be observed in either (i) models with assemblage errors (e.g. unconnected nodes/elements) or (ii) complex structural models that feature links/hinges etc. If users have checked carefully their model and found no modelling errors, then they may perhaps try to "simplify" it, by removing its more complex features until the attainment of the eigenvalue solutions. This will enable a better understanding of what might be causing the analysis problems, and thus assist users in deciding on how to proceed. This message typically appears when too many modes are sought, e.g. when 30 modes are asked in a 24 DOF model, or when the eigensolver cannot simply find so many modes (even if DOFs > modes).

STATIC ANALYSIS (NON-VARIABLE LOADING)

This type of analysis is commonly used to model static loads that are permanently applied to the structure (e.g. self-weight, foundation settlement), normally leading to a pre-yield elastic response. If the applied load is such that the structure is forced into a slightly inelastic response, the program performs equilibrium iterations until convergence is reached.

In cases of relatively high nonlinearity, where the solution cannot be found with a single increment, the load is automatically subdivided into smaller steps and an incremental iterative solution is obtained by the program, with no need for user intervention. It is noted, however, that for such cases the use of static pushover analysis is recommended since it will provide the user with greater flexibility in running the analysis and interpreting the results.

STATIC PUSHOVER ANALYSIS

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 applied incremental load P is kept proportional to the pattern of nominal loads (P°) initially defined by the user: $P = \lambda(P°)$. The load factor λ is automatically increased by the program until a user-defined limit, or numerical failure, is reached. For the incrementation of the loading factor, different strategies may be employed, since three types of control are currently available: load, response and automatic response.

Load Control

Refers to the case where the load factor is directly incremented and the global structural displacements are determined at each load factor level.

Response Control

Refers to direct incrementation of the global displacement of one node and the calculation of the loading factor that corresponds to this displacement.

Automatic response Control

Refers to a procedure in which the loading increment is automatically adjusted by SeismoStruct, depending on the convergence conditions at the previous step.

A more detailed description of the three types of control in pushover analysis is given in the Loading Phases paragraph.

NOTE: Conventional pushover analysis features an inherent inability to account for the effects that progressive stiffness degradation, typical in structures subjected to strong earthquake loading, has on the dynamic response characteristics of structures, and thus on the patterns of the equivalent static loads applied during a pushover analysis. Indeed, the fixed nature of the load distribution applied to the structure ignores the potential redistribution of forces during an actual dynamic response, which pushover tries to somehow reproduce. Consequently, the resulting changes in the modal characteristics of the structure (typically period elongation) and consequent variation in dynamic response amplification are not accounted for, which might introduce non-negligible inaccuracies, particularly in those cases where the influence higher mode is, or becomes, significant. These effects can only be accounted for by means of Adaptive Pushover.

Static Adaptive Pushover Analysis

Adaptive pushover analysis is employed in the estimation of the horizontal capacity of a structure, taking full account of the effect that the deformation of the latter and the frequency content of input motion have on its dynamic response characteristics. It may be applied in the assessment of both buildings [e.g. Antoniou et al. 2002; Antoniou and Pinho 2004a; Ferracuti et al. 2009] as well as bridge structures [e.g. Pinho et al. 2007; Casarotti and Pinho 2007; Pinho et al. 2009].

In the adaptive pushover approach, the lateral load distribution is not kept constant but rather continuously updated during the analysis, according to the modal shapes and participation factors derived by eigenvalue analysis carried out at each analysis step. This method is fully multi-modal and accounts for the softening of the structure, its period elongation, and the modification of the inertia forces due to spectral amplification (through the introduction of a site-specific spectrum).

Apart from force distributions, adaptive pushover is also able to efficiently employ deformation profiles [Antoniou and Pinho 2004b; Pinho and Antoniou 2005]. Due to its ability to update the lateral load patterns according to the constantly changing modal properties of the system, it overcomes the intrinsic weaknesses of fixed-pattern displacement pushover and provides a more accurate performance-oriented tool for structural assessment, providing better response estimates than existing conventional methods, especially in cases where strength or stiffness irregularities exist in the structure and/or higher mode effects might play an important role in its dynamic response [e.g. Pietra et al. 2006; Bento et al. 2008; Pinho et al. 2008b].

The adaptive algorithm, as implemented in SeismoStruct, is very flexible and can accept a number of different parameters that suit the specific requirements of each particular project. For example, both SRSS and CQC modal combination methods [e.g. Clough and Penzien, 1993; Chopra, 1995] are supported and the number of modes considered is explicitly defined, whereas users can also chose to update only the increment of loads applied at each step or the total loads already applied throughout the process up to the current point (see Adaptive Parameters).

The load control types available for the case of adaptive pushover are similar, in input and functionality, to those available for conventional pushover; adaptive load control, adaptive response control and automatic response control. For further information, users should refer to the Adaptive Parameters page and consult some of the many publications on this subject that are indicated above.

Static Time-History Analysis

In static time-history analysis, the applied loads (displacement, forces or a combination of both) can vary independently in the pseudo-time domain, according to a prescribed load pattern. The applied load P_i in a nodal position i is given by $P_i = \lambda_i(t)P_i°$, i.e. a function of the time-dependent load factor $\lambda_i(t)$ and the nominal load $P_i°$. This type of analysis is typically used to model static testing of structures under various force or displacement patterns (e.g. cyclic loading).

Dynamic Time-History Analysis

Dynamic analysis is commonly used to predict the nonlinear inelastic response of a structure subjected to earthquake loading (evidently, linear elastic dynamic response can also be modelled for as long as elastic elements and/or low levels of input excitation are considered). The direct integration of the equations of motion is accomplished using the numerically dissipative α-integration algorithm [Hilber et al., 1977] or a special case of the former, the well-known Newmark scheme [Newmark, 1959], with automatic time-step adjustment for optimum accuracy and efficiency (see Automatic adjustment of load increment or time-step).

Modelling of seismic action is achieved by introducing acceleration loading curves (accelerograms) at the supports, noting that different curves can be introduced at each support, thus allowing for representation of asynchronous ground excitation.

In addition, dynamic analysis may also be employed for modelling of pulse loading cases (e.g. blast, impact, etc.), in which case instead of acceleration time-histories at the supports, force pulse functions of any given shape (rectangular, triangular, parabolic, and so on), can be employed to describe the transient loading applied to the appropriate nodes.

Incremental Dynamic Analysis – IDA

In Incremental Dynamic Analysis [Hamburger et al., 2000; Vamvatsikos and Cornell, 2002], the structure is subjected to a series of nonlinear time-history analysis of increasing intensity (e.g. peak ground acceleration is incrementally scaled from a low elastic response value up to the attainment of a pre-defined post-yield target limit state). The peak values of base shear are then plotted against their top displacement counterparts, for each of the dynamic runs, giving rise to the so-called dynamic pushover or IDA envelope curves.

Response Spectrum Analysis – RSA

Response-spectrum analysis (RSA) [e.g. Rosenblueth, 1951; Chopra, 2001; EN 1998-1, 2004] is a linear elastic static-(pseudo)dynamic statistical analysis method which provides the peak values of response quantities, such as forces and deformations, of a structure under seismic excitation. It is called (pseudo)dynamic because the peak response can be estimated directly from the response spectrum for the ground motion by carrying out static analysis 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 all free DOFs of the structure and they represent the contribution from each natural mode of vibration. These equivalent forces are derived per mode as the product of two quantities: (1) the modal inertia force distribution (thus Eigenvalue analysis is needed), and (2) the pseudo-acceleration response per structural mode (obtained from the response spectrum). For each meaningful mode a static analysis is conducted, and then every final peak response quantity is derived by the superposition of the quantities corresponding to each analysis.

It is noted that it is not possible to determine the exact peak value, because, in general, the modal response quantities attain their peaks at different time instants. Approximations are 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 mode shapes. SRSS can be used when periods differ by more than 10%, whilst ABSSUM offers an upper limit of response. This procedure is repeated for each desired seismic direction by using different or same response spectra. RSA is typically considered as useful for design decision-making because it relates structural type-selection to dynamic performance. Structures of shorter period experience greater acceleration, whereas those of longer period experience greater displacement. For design purposes, 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 structure.

Users are asked to provide as input the response spectrum and the seismic loading combination(s) for which the RSA will output the results. The response spectrum may be defined directly or may be

calculated from a given accelerogram. This spectrum is employed for both the two horizontal (EX, EY) and the vertical (EZ) seismic directions. Alternatively, different response spectrum factors between horizontal and vertical directions may be defined in the loading combination module. Further, the modal combination rule (ABSSUM, SRSS, CQC) should be determined, as well as which modes are to be combined, as a function of the target cumulative effective modal mass.

Finally, for each case of loading combination (G+Q±E), users are asked to define the factor of static gravity loading (f_{G+Q}) and the factor of the seismic loading (f_E). Seismic loading directions may be combined linearly ($E = \pm EX \pm EY \pm EZ$) with different factors per direction (f_{EX}, f_{EY}, f_{EZ}) or by the SRSS rule ($E = \pm\sqrt{EX^2+EY^2+EZ^2}$). It should be noted that the gravity loads have an explicitly defined algebraic sign, while for the seismic loadings both signs for every direction are taken into account. Consequently, the results of RSA loading combinations in terms of any response quantity are presented as envelopes.

Buckling Analysis

In general, in order to identify the limit point that recognises the transition from a stable to an unstable structure, an incremental analysis should be performed. The incremental analysis considers both geometric and material nonlinearities. In some cases, i.e. slender steel structures, the stability is governed by the geometric nonlinearities. Hence, neglecting nonlinear material behaviour and assuming the relative distribution of internal force equal at all ratios of the applied load, a buckling analysis can be performed in place of the incremental one. Besides at these two assumptions, the element geometric stiffness matrices are linear functions of their end forces. Hence, these hypothesis permit to write the global stiffness equation in the form of a generalised eigenvalue problem in which the equation of equilibrium at the critical state is

$$[K_E + \lambda_i K_G]d_i = 0$$

K_E is the linear elastic stiffness matrix. K_G is the geometric stiffness matrix which represents the change in stiffness that results from changes in geometry as the applied loading is increased. It is computed for a reference loading pattern P_{ref} which corresponds to the base state of structure with preloads. λ_i is a vector of load factors (eigenvalues) with respect to P_{ref} and d_i is the buckling mode shape (eigenvectors), where i refers to the ith buckling mode. The lowest value of λ_i provides the elastic critical load vector λ_{min} P_{ref}. Commonly, it is easier to solve this kind of problem than to solve an incremental analysis.

Tsunami Nonlinear Analysis

ASCE 7-16 has established a framework for assessing the capacity of structures against Tsunami Loading allowing the use of performance-based criteria and nonlinear methods. A series of papers including the work by Petrone et al. [2017] and Baiguera et al. [2019] have presented a methodology including nonlinear analysis for the tsunami assessment of buildings following the ASCE 7-16 provisions which is implemented in Seismostruct.

For the performance of the analysis pre-defined time-histories of the inundation depth and velocity are used in order to calculate the Total Tsunami Force applied on the building at each time-history step. The time histories to be used can be the ones provided by ASCE 7-16 or different ones available within Seismostruct. The Total Tsunami Force at each step can be calculated using the relationships provided within ASCE 7-16 or in the work by Petrone et al. [2017] and Foster et al. [2017]. Nonlinear Tsunami Analysis can be performed in two phases:

The first phase includes the Variable Depth Pushover (VDPO) Analysis, during which the Total Tsunami Force at each step is distributed along the height and width of the area of the building affected by the Tsunami. The building response is calculated at each step similarly to a non-linear time-series analysis. When the VDPO phase is finished (either at the end of the analysis steps or due to divergence) the analysis can optionally move to the second phase the Constant Depth Pushover (CDPO) Analysis. During the CDPO phase the Tsunami depth is held constant and the lateral loads are increased monotonically until the building reaches a certain displacement defined as a certain percentage of the displacement at the end of the VDPO analysis. The CDPO analysis is similar to a response controlled Static Pushover

Analysis. For a detailed presentation of the procedures during the Nonlinear Tsunami Analysis users are directed to the work by Petrone et al [2017] and Baiguera et al. [2019].

Appendix C - Materials

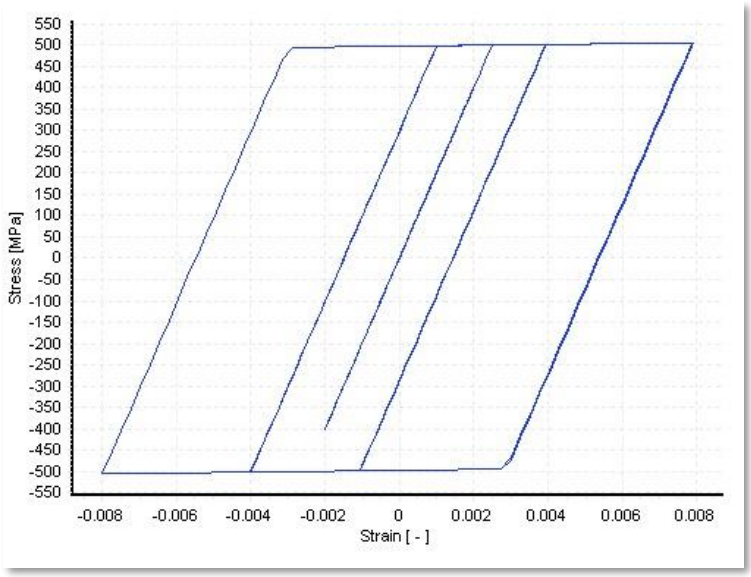
In this appendix the available material types are described in details.

STEEL MATERIALS

Bilinear steel model - stl_bl

NOTE: Due to its very simple and basic formulation, this model is not recommended for the modelling of reinforced concrete members subject to complex loading histories, where significant load reversals might occur. For such cases, models stl_mp and stl_mn should be employed instead.

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 easily identifiable calibrating parameters and by its computational efficiency. It 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.



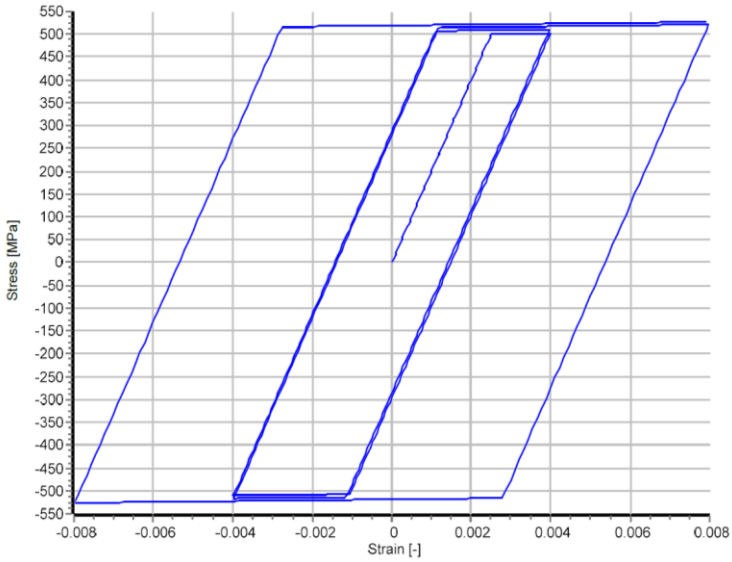
Bilinear steel model

Five model-calibrating parameters must be defined in order to fully describe the mechanical characteristics of the material:

Material Properties	Typical values	Default values
Modulus of elasticity – $\mathbf{E_s}$	2.00E+08 - 2.10E+08 (kPa)	2.00E+08 (kPa)
Yield strength – $\mathbf{f_y}$	230000 - 650000 (kPa)	500000 (kPa)
Strain hardening parameter – $\boldsymbol{\mu}$	0.005 - 0.015 (-)	0.005 (-)
Fracture/buckling strain		0.2 (-)
Specific weight – $\boldsymbol{\gamma}$	78 (kN/m^3)	78 (kN/m^3)

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 non-linear rule.



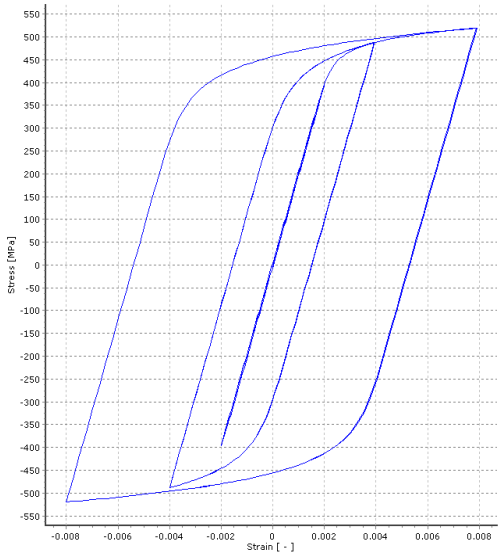
Bilinear steel model with isotropic strain hardening

Nine model-calibrating parameters must be defined in order to fully describe the mechanical characteristics of the material:

Material Properties	Typical values	Default values
Modulus of elasticity – **E_s**	2.00E+08 - 2.10E+08 (kPa)	2.00E+08 (kPa)
Yield strength – **f_y**	230000 - 650000 (kPa)	500000 (kPa)
Strain hardening parameter – **μ**	0 - 0.03 (-)	0.005 (-)
Isotropic Hardening in Compression Parameter – **A1**	-0.5 - 0.5 (-) (set as 0 for no isotropic hardening in compression)	0.1 (-)
Isotropic Hardening in Compression Parameter – **A2**	1 - 7 (-)	7 (-)
Isotropic Hardening in Tension Parameter – **A3**	-0.5 - 0.5 (-) (set as 0 for no isotropic hardening in tension)	0.1 (-)
Isotropic Hardening in Tension Parameter – **A4**	1 - 7 (-)	7 (-)
Fracture / Buckling Strain		0.1 (-)
Specific Weight	78.00 (KN/m³)	78.00 (KN/m³)

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] and Otani [1981].



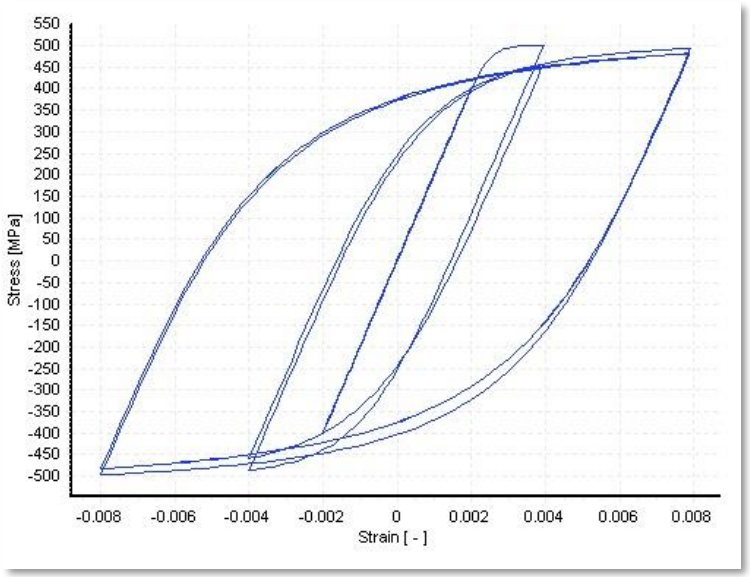
Ramberg-Osgood steel model

Six model-calibrating parameters must be defined in order to fully describe the mechanical characteristics of the material:

Material Properties	Typical values	Default values
Yield strength – $\mathbf{f_y}$	230000 - 650000 (kPa)	500000.00 (kPa)
Yield strain		0.0025
Ramberg Osgood Parameter - $\boldsymbol{\gamma}$	10.00- 30.00 (-)	20.00 (-)
Convergence Limit for the Newton-Raphson procedure – $\boldsymbol{\beta_1}$	1.00E-02 - 1.00E-04 (-)	0.001 (-)
Fracture / Buckling strain		0.10 (-)
Specific Weight	78.00 (KN/m^3)	78.00 (KN/m^3)

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. 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.



Menegotto-Pinto steel model

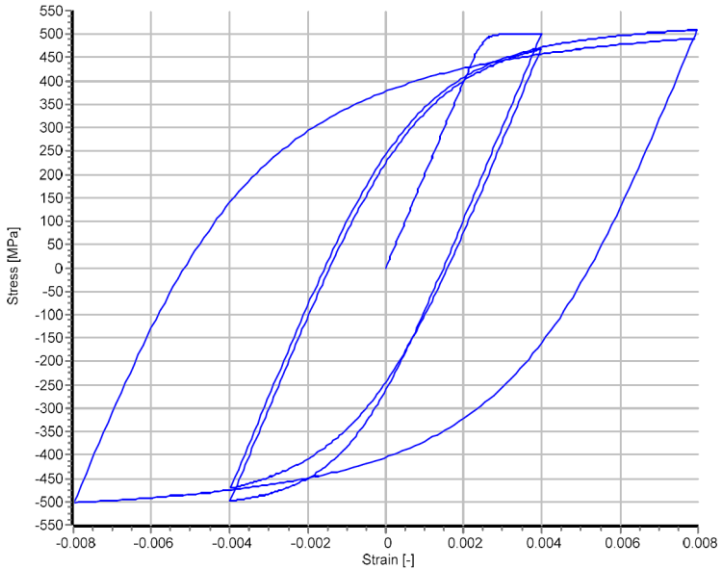
Ten model-calibrating parameters must be defined in order to fully describe the mechanical characteristics of the material:

Material Properties	Typical values	Default values
Modulus of elasticity – **E_s**	2.00E+08 - 2.10E+08 (kPa)	2.00E+08 (kPa)
Yield strength – **f_y**	230000 - 650000 (kPa)	500000 (kPa)
Strain hardening parameter – **μ**	0.005 - 0.015 (-)	0.005 (-)
Transition curve initial shape parameter – **R_0**	20 (-)	20 (-)
Transition curve shape calibrating coefficient – **A1**	18.5 (-)	18.5 (-)
Transition curve shape calibrating coefficient – **A2**	0.05 - 0.15 (-)	0.15 (-)
Isotropic hardening calibrating coefficient – **A3**	0.01 – 0.025 (-)	0 (-)
Isotropic hardening calibrating coefficient – **A4**	2 - 7 (-)	1 (-)
Fracture/buckling strain		0.1 (-)
Specific weight – **γ**	78 (kN/m^3)	78 (kN/m^3)

NOTE: It is possible to assign a negative value to parameter A3 in order to artificially introduce softening in the response of a structural element featuring this material model. In such cases, however, users should check the results carefully, since this material model was not initially devised with such feature in mind.

Giuffre-Menegotto-Pinto Model with Isotropic Hardening – stl_gmp

This is a uniaxial Giuffre-Menegotto-Pinto Material with optional isotropic hardening described by a non-linear 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.



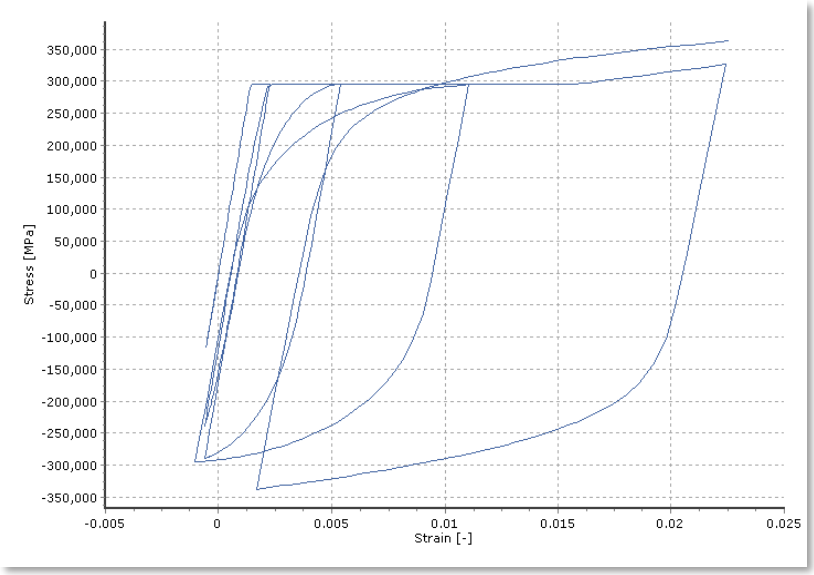
Giuffre-Menegotto-Pinto Model with Isotropic Hardening

Thirteen model-calibrating parameters must be defined in order to fully describe the mechanical characteristics of the material:

Material Properties	Typical values	Default values
Modulus of elasticity – E_s	2.00E+08 - 2.10E+08 (kPa)	2.00E+08 (kPa)
Yield strength – f_y	230000 - 650000 (kPa)	500000 (kPa)
Strain hardening parameter – μ	0 - 0.03 (-)	0.005 (-)
Initial Value of Curvature Parameter– R_0	10 - 20 (-)	20 (-)
Curvature Degradation Parameter– R_1	0 - 0.98 (-) (Recommended value is 0.925)	0.925 (-)
Curvature Degradation Parameter– R_2	Recommened value is 0.15(-)	0.15 (-)
Isotropic Hardening in Compression Parameter – **A1**	-0.5 - 0.5 (-) (set as 0 for no isotropic hardening in compression)	0.1 (-)
Isotropic Hardening in Compression Parameter – **A2**	1 - 7 (-)	7 (-)
Isotropic Hardening in Tension Parameter – **A3**	-0.5 - 0.5 (-) (set as 0 for no isotropic hardening in tension)	0.1 (-)
Isotropic Hardening in Tension Parameter – **A4**	1 - 7 (-)	7 (-)
Initial Stress –σ_0		0 (kPa)
Fracture / Buckling Strain		0.1 (-)
Specific Weight	78.00 (KN/m^3)	78.00 (KN/m^3)

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

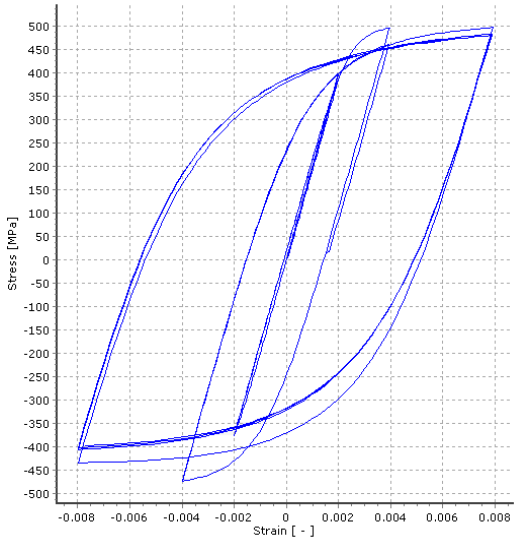
Eight model-calibrating parameters must be defined in order to fully describe the mechanical characteristics of the material:

Material Properties	Typical values	Default values
Modulus of elasticity – E_s	2.00E+08 - 2.10E+08 (kPa)	2.10E+08 (kPa)
Yield strength – f_y	230000 - 650000 (kPa)	295400 (kPa)
Stress at peak load		479500 (kPa)
Strain at initiation of strain hardening curve	(-)	0.016 (-)
Strain at peak load	(-)	0.207 (-)
Strain of the intermediate point of the strain hardening curve	(-)	0.053 (-)
Stress of the intermediate point of the strain hardening curve	(-)	420000 (kPa)
Specific weight	78 (kN/m^3)	78 (kN/m^3)

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.



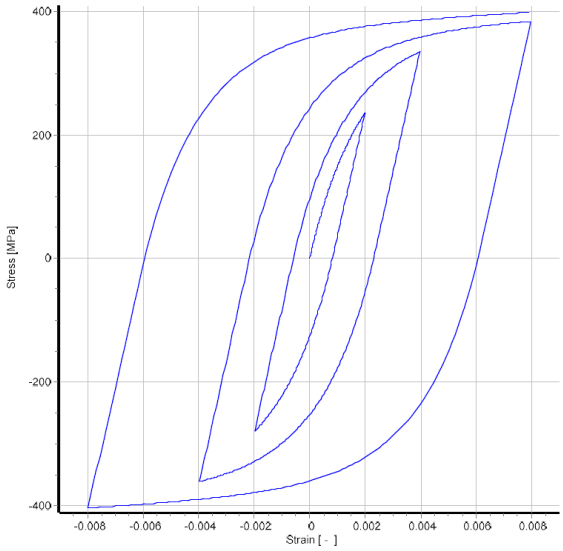
Monti-Nuti steel model

Ten model-calibrating parameters must be defined in order to fully describe the mechanical characteristics of the material:

Material Properties	Typical values	Default values
Modulus of elasticity – $\mathbf{E_s}$	2.00E+08 - 2.10E+08 (kPa)	2.00E+08 (kPa)
Yield strength – $\mathbf{f_y}$	230000 - 650000 (kPa)	500000 (kPa)
Strain hardening parameter – $\boldsymbol{\mu}$	0.005 - 0.015 (-)	0.005 (-)
Transition curve initial shape parameter – $\mathbf{R_0}$	20 (-)	20 (-)
Transition curve shape calibrating coefficient – **A1**	18.5 (-)	18.5 (-)
Transition curve shape calibrating coefficient – **A2**	0.05 - 0.15 (-)	0.15 (-)
Kinematic/isotropic weighing coefficient – **P**	Close to 0.9 (-)	0.9 (-)
Spurious unloading corrective parameter – **r**	2.5 - 5 (%)	2.5 (%)
Fracture strain	-	0.1 (-)
Specific weight – $\boldsymbol{\gamma}$	78 (kN/m^3)	78 (kN/m^3)

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

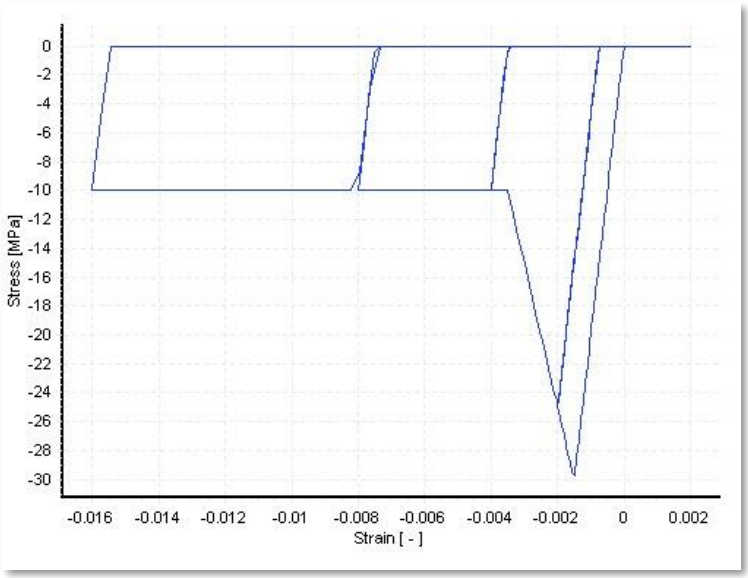
Eleven parameters are needed in order to describe the mechanical characteristics of the material.

Material Properties	Typical values	Default values
Modulus of elasticity – $\mathbf{E_s}$	2.00E+08 - 2.10E+08 (kPa)	2.00E+08 (kPa)
Yield strength – $\mathbf{f_y}$	260000 - 420000 (kPa)	37000 (kPa)
Maximum yield force for fully saturated isotropic hardening for Tension – $\mathbf{f_{y,max\,T}}$	400000 - 600000 (kPa)	492000 (kPa)
Constant controlling elastic to plastic transition for tension– $\boldsymbol{\alpha_{,T}}$	0.2-1.2	0.9
Hardening ratio for tension – $\mathbf{b_{,T}}$	0.01-0.04	0.01
Constant controlling isotropic hardening for tension – $\boldsymbol{\delta_{,T}}$	0.1-0.8	0.2
Maximum yield force for fully saturated isotropic hardening for compression – $\mathbf{f_{y,max\,C}}$	400000 - 600000 (kPa)	581000
Constant controlling elastic to plastic transition for compression – $\boldsymbol{\alpha_{,C}}$	0.2-1.2	0.9
Hardening ratio for compression – $\mathbf{b_{,C}}$	0.01-0.04	0.01
Constant controlling isotropic hardening for compression – $\boldsymbol{\delta_{,C}}$	0.1-0.8	0.2
Specific weight – $\boldsymbol{\gamma}$	78 (kN/m^3)	78 (kN/m^3)

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

Five model-calibrating parameters must be defined in order to fully describe the mechanical characteristics of the material:

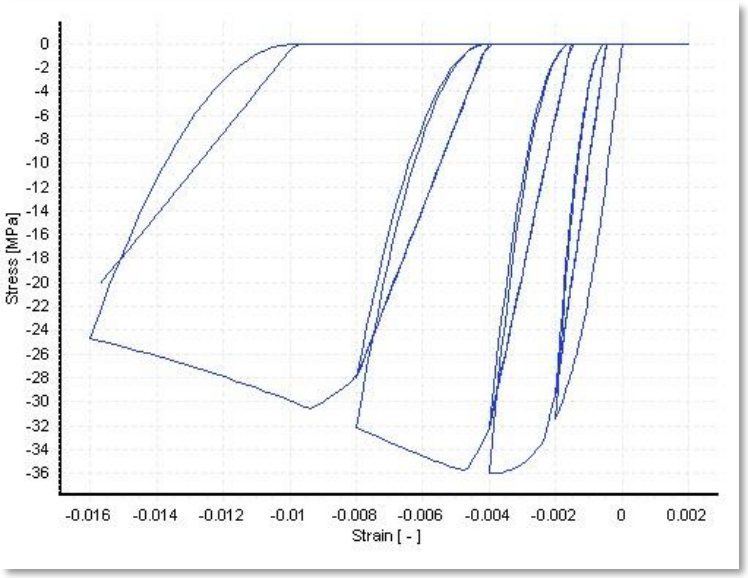
Material Properties	Typical values	Default values
Mean Compressive strength – f_{c1}	15000 - 45000 (kPa)	28000 (kPa)
Initial stiffness – E_1	1.50E+07 - 3.00E+07 (kPa)	2.00E+07 (kPa)
Post-peak stiffness – E_2	-5.00E+06 - -3.00E+07 (kPa)	-1.00E+07 (kPa)
Residual strength – f_{c2}	5000 - 15000 (kPa)	5000 (kPa)
Specific weight – γ	24 (kN/m^3)	24 (kN/m^3)

NOTE 1: Values of compressive strength capacity obtained through testing of concrete cubes are usually 25 to 10 percent higher than their cylinder counterparts, for cylinder concrete strengths of 15 to 50 MPa, respectively.

NOTE 2: Some researchers [e.g. Scott et al., 1982] have suggested that the influence of the high strain rates expected under seismic loading (0.0167/sec) on the stress-strain behaviour of the core concrete can be accounted for by adjusting the results of tests conducted at usual strain rates (0.0000033/sec); the adjustment could consist simply of applying a multiplying factor of 1.25 to the peak stress, the strain at the peak stress, and the slope of the post-yield falling branch. Mander et al. [1989] also present methods by which strain rate effects can be incorporated into the model, although the basic formulae, implemented here, do not include the effect.

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

Five model-calibrating parameters must be defined in order to fully describe the mechanical characteristics of the material:

Material Properties	Typical values	Default values
Mean Compressive strength – f_c	15000 - 45000 (kPa)	28000 (kPa)
Mean Tensile strength – f_t	2000 - 3000 (kPa)	2200 (kPa)
Modulus of Elasticity – E_c	18000 - 30000 (MPa)	24870 (MPa)
Strain at peak stress – ε_c	0.002 - 0.0022 (m/m)	0.002 (m/m)
Specific weight – γ	24 (kN/m^3)	24 (kN/m^3)

NOTE 1: Values of compressive strength capacity obtained through testing of concrete cubes are usually 25 to 10 percent higher than their cylinder counterparts, for cylinder concrete strengths of 15 to 50 MPa, respectively.

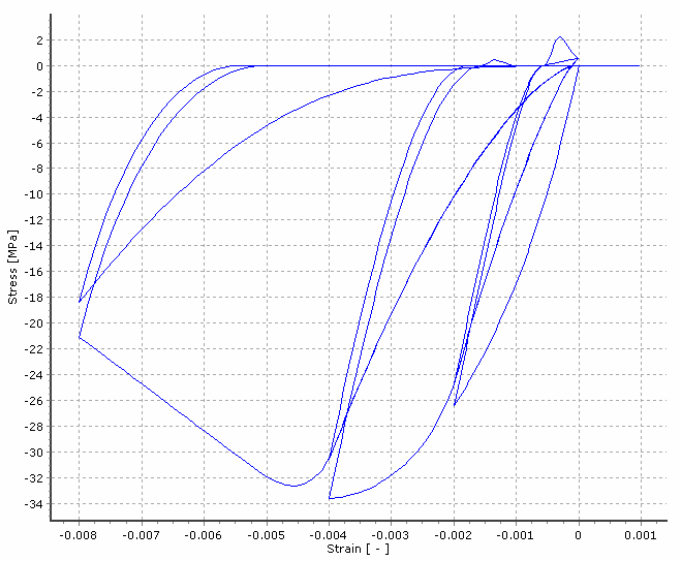
NOTE 2: Some researchers [e.g. Scott et al., 1982] have suggested that the influence of the high strain rates expected under seismic loading (0.0167/sec) on the stress-strain behaviour of the core concrete can be accounted for by adjusting the results of tests conducted at usual strain rates (0.0000033/sec); the adjustment could consist simply of applying a multiplying factor of 1.25 to the peak stress, the strain at the peak stress, and the slope of the post-yield falling branch. Mander et al. [1989] also present methods by which strain rate effects can be incorporated into the model, although the basic formulae, implemented here, do not include the effect.

NOTE 3: On occasions, depending on the structural model and applied loading, crack opening may introduce numerical instabilities in the analyses. If, on some of those instances, the user is interested in predicting, for example, the top displacement of a building (i.e. global response) rather than accurately reproducing the local response of elements and sections (e.g. section curvatures), then tensile resistance may be simply ignored altogether (i.e. f_t = 0 MPa), and in this way stability of the analysis will most certainly be achieved in easier fashion.

NOTE 4: 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 both in the Sections and in the Confinement Factor Calculation module. Its value usually fluctuates between the values of 1.0 and 2.0 for reinforced concrete members and between 1.5 and 4.0 for steel-concrete composite members.

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

Eight model-calibrating parameters must be defined in order to fully describe the mechanical characteristics of the material:

Material Properties	Typical values	Default values
Mean Compressive strength – $\mathbf{f_c}$	15000 - 45000 (kPa)	28000 (kPa)
Mean Tensile strength – $\mathbf{f_t}$	-	2200 (kPa)
Modulus of Elasticity – $\mathbf{E_c}$	10000 - 30000 (MPa)	22960 (MPa)

Material Properties	Typical values	Default values
Strain at peak compressive stress – ε_c	0.002 - 0.0022 (m/m)	0.002 (m/m)
Strain at peak tensile stress – ε_t	0.0002 - 0.00022 (m/m)	0.0002 (m/m)
Nondimensional critical compressive strain – r	1.1 - 1.5 [-]	1.3 [-]
Nondimensional critical tensile strain - x_{cr}	1-5 [-]	3 [-]
Specific weight – γ	24 (kN/m^3)	24 (kN/m^3)

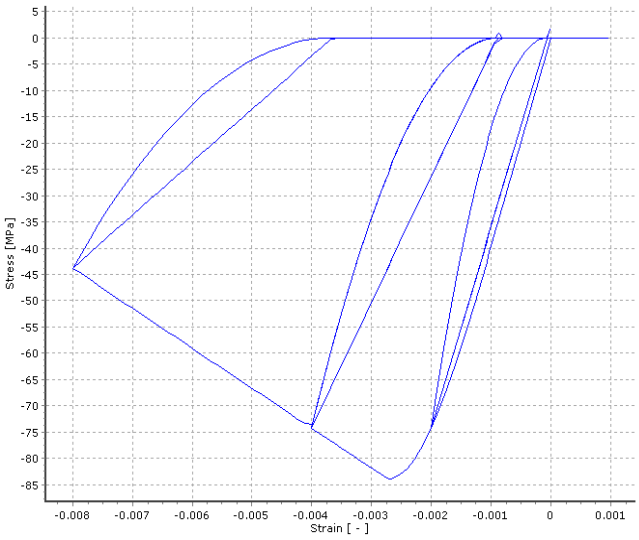
NOTE 1: Values of compressive strength capacity obtained through testing of concrete cubes are usually 25 to 10 percent higher than their cylinder counterparts, for cylinder concrete strengths of 15 to 50 MPa, respectively.

NOTE 2: The non-dimensional critical strain values, r for compression and x_{cr} for tension, determine the shape of the descending branches of the curve. Users are advised to view the plot on the New Material dialog box, in order to understand how the curve changes with different values of the parameters.

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

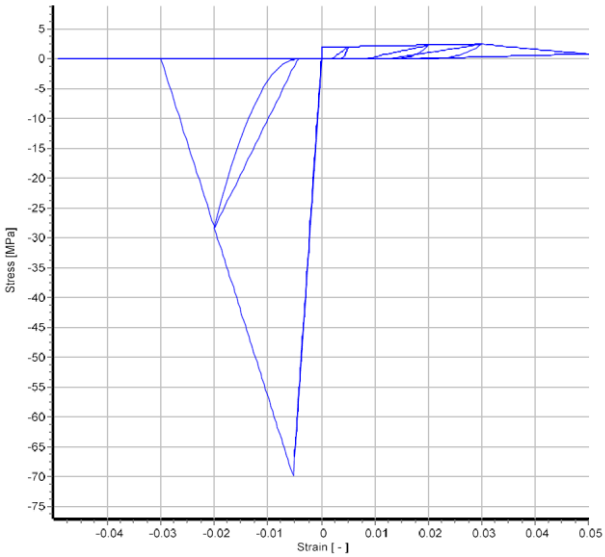
Four model-calibrating parameters must be defined in order to fully describe the mechanical characteristics of the material:

Material Properties	Typical values	Default values
Mean Compressive strength – $\mathbf{f_c}$	50000 - 120000 (kPa)	78000 (kPa)
Mean Tensile strength – $\mathbf{f_t}$	-	4600 (kPa)
Modulus of Elasticity - $\mathbf{E_c}$	35000 – 45000 (MPa)	40742 (MPa)
Specific weight – $\boldsymbol{\gamma}$	24 (kN/m³)	24 (kN/m³)

NOTE: On occasions, depending on the structural model and applied loading, crack opening may introduce numerical instabilities in the analyses. If, on some of those instances, the user is interested in predicting, for example, the top displacement of a building (i.e. global response) rather than accurately reproducing the local response of elements and sections (e.g. section curvatures), then tensile resistance may be simply ignored altogether (i.e. f_t = 0 MPa), and in this way stability of the analysis will most certainly be achieved in easier fashion.

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.

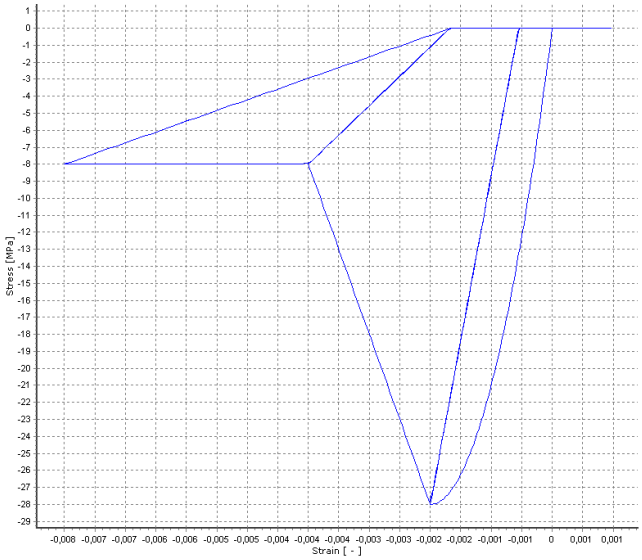


Ductile fiber – reinforced cement based composites model

Material Properties	Typical values	Default values
Cracking stress – $\sigma_{t,0}$	2.00E+03 – 4.00E+03 (kPa)	2.00E+03 (kPa)
Cracking strain – $\varepsilon_{t,0}$	0.000015 – 0.00025	0.00015
Peak tensile stress – $\sigma_{t,p}$	2.50E+03 – 6.00E+03 (kPa)	2.50E+03 (kPa)
Strain at peak tensile stress – $\varepsilon_{t,p}$	0.03-0.037	0.03
Ultimate tensile strain – $\varepsilon_{t,u}$	0.06	0.06
Compressive strength – $\sigma_{c,p}$	70.00E+03 – 80.00E+03 (kPa)	70.00E+03 (kPa)
Strain at compressive strength– $\varepsilon_{c,p}$	0.005-0.006	0.00525
Ultimate compressive strain – $\varepsilon_{c,u}$	0.01-0.03	0.03
Power for tensile unloading curve – a_t	≥1	5
Power for compressive unloading curve – a_c	≥1	2
Factor for tensile unloading curve – **bt**	>0	0.4
Factor for compressive unloading curve – **bc**	>0	0.3
Specific weight – γ	24 (kN/m^3)	24 (kN/m^3)

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.



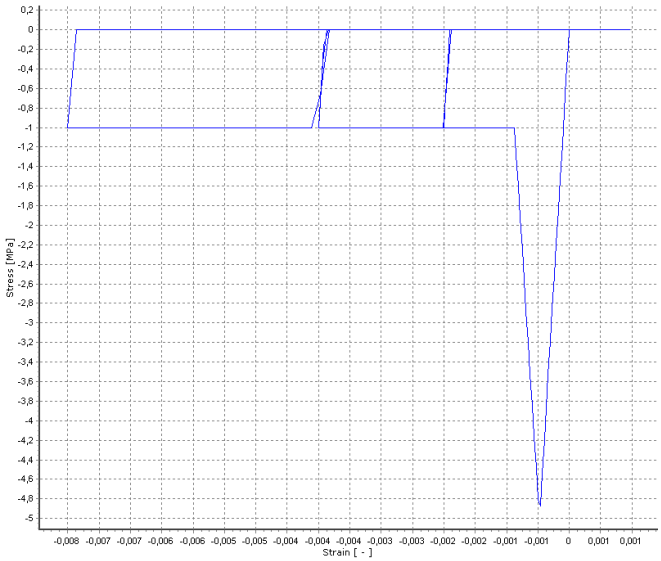
Kent – Scott – Park concrete model

Material Properties	Typical values	Default values
Compressive strength – σ_c	15000 - 45000 (kPa)	28000 (kPa)
Strain at compressive strength – ε_c	0.002 - 0.0022 [-]	0.002
Residual strength – σ_c	5000 - 15000 (kPa)	8000 (kPa)
Strain at residual strength– ε_c	-	0.004
Specific weight - γ	24 (kN/m³)	24 (kN/m³)

OTHER MATERIALS

Trilinear masonry model - mas_tl

This is a simplified uniaxial trilinear material model that assumes no resistance to tension and features a residual strength plateau.



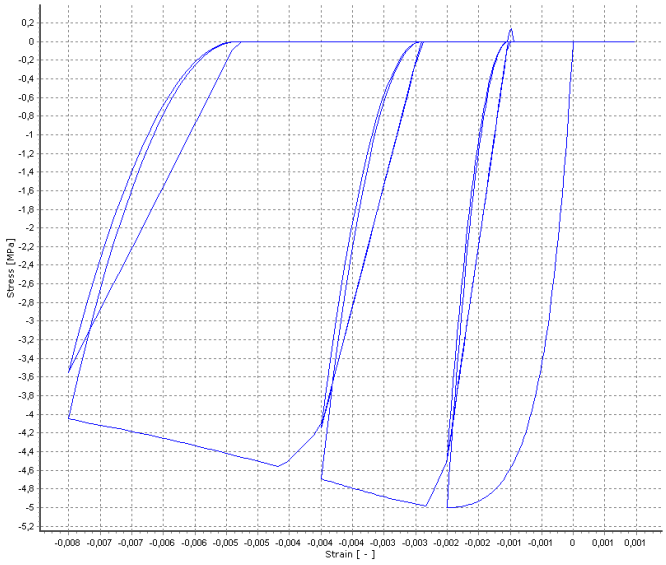
Trilinear masonry model

Five model-calibrating parameters must be defined in order to fully describe the mechanical characteristics of the material:

Material Properties	Typical values	Default values
Mean Compressive strength – f_{c1}	1000 - 10000 (kPa)	5000 (kPa)
Initial stiffness – E_1	5.0E+06 - 2.00E+07 (kPa)	1.05E+07 (kPa)
Poisson Ratio	0.10-0.30 (-)	0.20 (-)
Post-peak stiffness – E_2	-2.50E+06 - -3.00E+07 (kPa)	-1.00E+07 (kPa)
Residual strength – f_{c2}	500 - 5000 (kPa)	1000 (kPa)
Specific weight – γ	24 (kN/m^3)	24 (kN/m^3)

Parabolic masonry model - mas_par

This is a uniaxial nonlinear material model for masonry that is based on the hysteretic rules of the con_ma typical constant confinement concrete model.



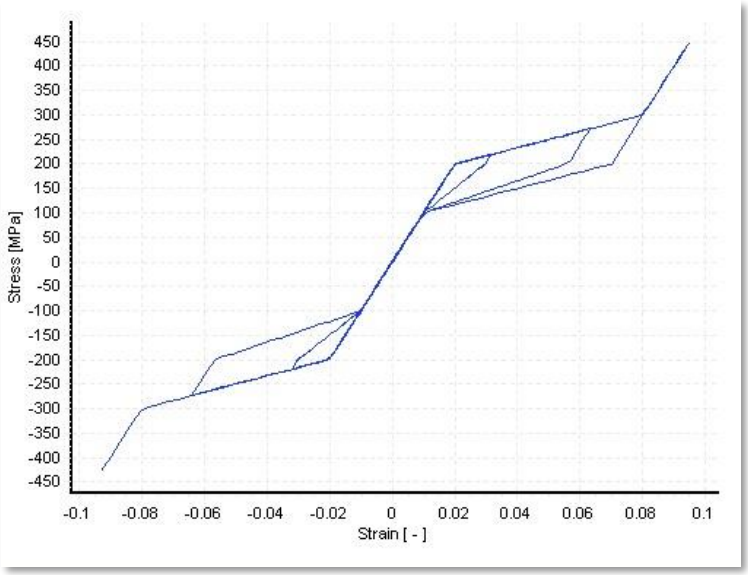
Parabolic Masonry model

Seven model-calibrating parameters must be defined in order to fully describe the mechanical characteristics of the material:

Material Properties	Typical values	Default values
Mean Compressive strength – f_c	1000 - 10000 (kPa)	3500 (kPa)
Mean Tensile strength – f_t	0 - 2000 (kPa)	150 (kPa)
Modulus of Elasticity – E_c	5000 - 20000 (MPa)	1.05E+07 (kPa)
Poisson Ratio	0.10-0.30 (-)	0.20 (-)
Strain at peak stress – ε_c	0.002 - 0.0022 (m/m)	0.002 (m/m)
Descending branch parameter	0.50 - 10 (-)	1.00 (-)
Specific weight – γ	24 (kN/m^3)	24 (kN/m^3)

Superelastic shape-memory alloys model - se_sma

This is a uniaxial model for superelastic shape-memory alloys (SMAs), programmed by Fugazza [2003], and that follows the constitutive relationship proposed by Auricchio and Sacco [1997]. The model assumes a constant stiffness for both the fully austenitic and fully martensitic behaviour, and is also rate-independent.



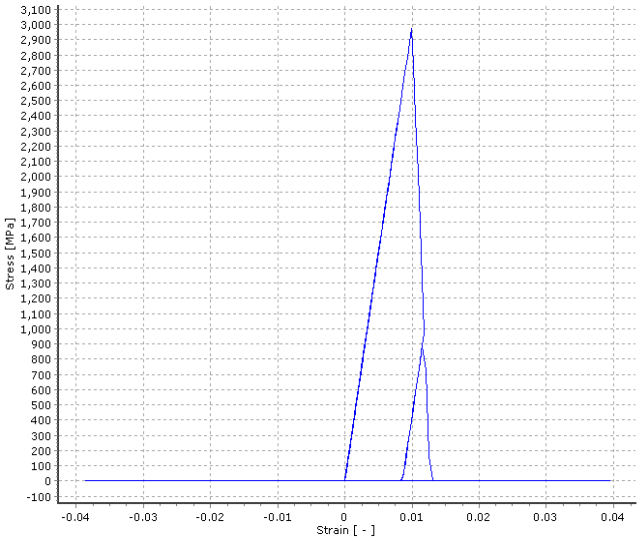
Superlelastic shape-memory alloys model

Seven model-calibrating parameters, the values of which can be obtained from simple uniaxial tests performed on SMA elements (wires or bars, typically), must be defined in order to fully describe the mechanical characteristics of the material:

Material Properties	Typical values	Default values
Modulus of elasticity - **E**	1.00E+07 - 8.00E+07 (kPa)	1.00E+07 (kPa)
Austenite-to-martensite starting stress - $\sigma_{s\text{-}AS}$	200000 - 600000 (kPa)	200000 (kPa)
Austenite-to-martensite finishing stress - $\sigma_{f\text{-}AS}$	300000 - 700000 (kPa)	300000 (kPa)
Martensite-to-austenite starting stress - $\sigma_{s\text{-}SA}$	600000 - 200000 (kPa)	200000 (kPa)
Martensite-to-austenite finishing stress - $\sigma_{f\text{-}SA}$	500000 - 100000 (kPa)	100000 (kPa)
Superelastic plateau strain length - ε_L	4 - 8 (%)	5 (%)
Specific weight – γ	65 (kN/m^3)	65 (kN/m^3)

Trilinear FRP model - frp_tl

This is a simplified uniaxial trilinear FRP model that assumes no resistance in compression.



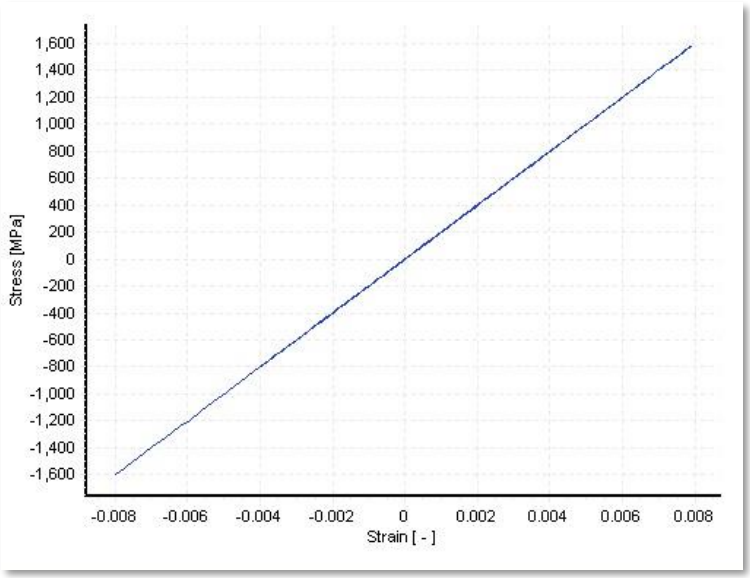
Trilinear FRP model

Four model-calibrating parameters must be defined in order to fully describe the mechanical characteristics of the material:

Material Properties	Typical values	Default values
Tensile strength - f_t	2.10E+06 - 4.80E+06 (kPa) (carbon-based fibres) 1.90E+06 - 4.80E+06 (kPa) (glass fibres) 3.50E+06 - 4.10E+06 (kPa) (aramid fibres)	3.00E+06 (kPa)
Initial stiffness - E_1	2.15E+08 - 7.00E+08 (kPa) (carbon-based fibres) 7.00E+07 - 9.00E+07 (kPa) (glass fibres) 7.00E+07 - 1.30E+08 (kPa) (aramid fibres)	3.00E+08 (kPa)
Post-peak stiffness - E_2	-	-5.00E+08 (kPa)
Specific weight – γ	18 (kN/m^3)	18 (kN/m^3)

Elastic material model - el_mat

This is a simplified uniaxial elastic material model with symmetric behaviour in tension and compression.



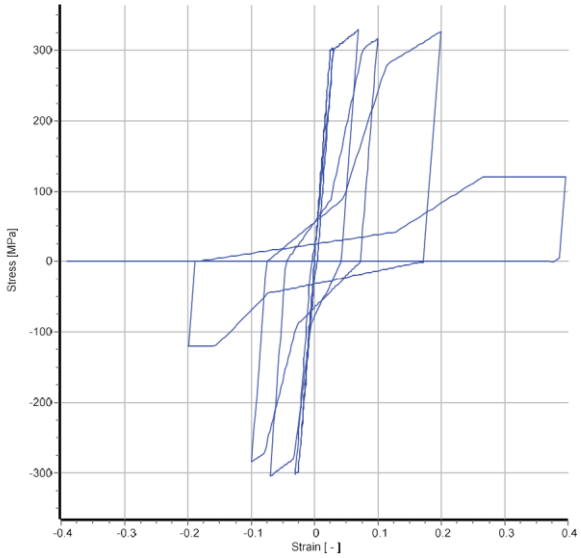
Elastic material model

Two model-calibrating parameters must be defined in order to fully describe the mechanical characteristics of the material:

Material Properties	Typical values	Default values
Modulus of elasticity - E_s	-	2.00E+08 (kPa)
Specific weight – γ	20 (kN/m^3)	20 (kN/m^3)

Generic Hysteretic material - hyst_mat

Hyst_mat is a generic uniaxial material model characterised by Pinching effect (controlled by a pinching factor with values from 0 for no pinching to 1 for high pinching) and by four deterioration modes including strength deterioration, peak stress deterioration, reloading and unloading stiffness deterioration. All deterioration modes are controlled by the deterioration factor (with values from 0 for no deterioration to 1 for high deterioration). An initial backbone curve consisting of the yielding point, the peak stress point and the reidual strength is initialy defined and deteriorates after each unloading incident.



Generic Hysteretic material model

Twelve parameters are needed in order to describe the mechanical characteristics of the material. Pairs of stress strain defining the backbone curve must be given in ascending order

Material Properties	Typical values	Default values
Modulus of Elasticity - E_s	-	2.00E+08 (kPa)
Yield Stress in positive direction – $\sigma_{y,pos}$	-	300E+03 (kPa)
Yield Stress in negative direction – $\sigma_{y,neg}$	-	300E+03 (kPa)
Peak Strain in positive direction – $\varepsilon_{peak,pos}$	-	300
Peak Stress in positive direction – $\sigma_{peak,pos}$	-	500 (kPa)
Peak Strain in negative direction – $\varepsilon_{peak,neg}$	-	300
Peak Stress in negative direction – $\sigma_{peak,neg}$	-	500 (kPa)
Residual Strength in positive direction – $\sigma_{Res,pos}$	-	120 (kPa)
Residual Strength in negative direction – $\sigma_{Res,neg}$	-	120 (kPa)
Pinching factor – **FPinch**	0-1	0.5
Deterioration factor – **FDet**	0-1	0.5
Specific weight - γ	-	24 (kN/m^3)

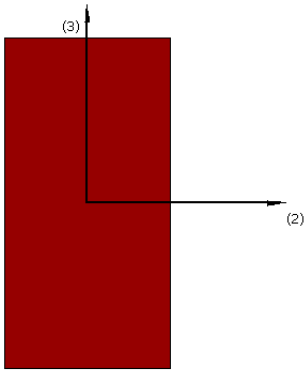
Appendix D - Sections

In this appendix the available section types are described in details.

ONE MATERIAL SECTIONS

Rectangular solid section - rss

This is a section frequently adopted for the modelling of rectangular members in steel structures.



Materials and Dimensions

Only one material (steel/ concrete) needs to be defined.

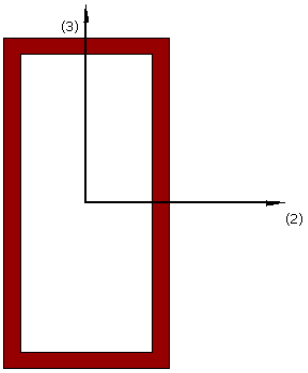
The required dimensions are as follows:

- Section width. The default value is 0.1 m
- Section height/depth. The default value is 0.2 m

NOTE: From SeismoStruct 2016 onwards it is possible to apply concrete material models to the rss

Rectangular hollow section - rhs

This is a section frequently adopted for the modelling of rectangular hollow members in steel structures.



Materials and Dimensions

Only one material (steel) needs to be defined.

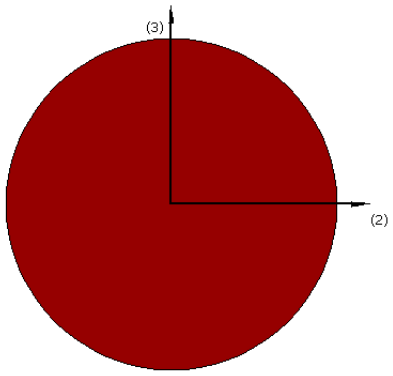
The required dimensions are as follows:

- Section width. The default value is 0.1 m
- Section height/depth. The default value is 0.2 m
- Section thickness. The default value is 0.01 m

NOTE: Users may use this section to model the retrofitting of a RC rectangular member with longitudinally-oriented steel. To do this, first create a material model featuring the properties of the retrofitting material and then use an rhs-section element with internal height/depth that equals that of the original element's section, a thickness corresponding to the thickness of the retrofitting layer, and connect this new retrofitting element to the same nodes to which the existing element is connected to.

Circular solid section - css

This is a section frequently adopted for the modelling of circular members in steel structures.



Materials and Dimensions

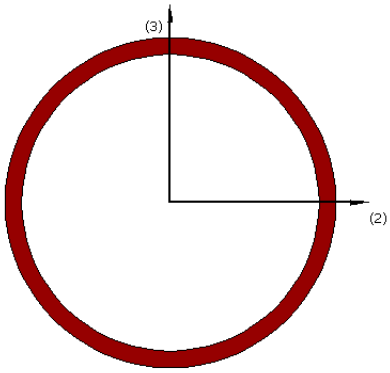
Only one material (steel) needs to be defined.

The required dimension is as follows:

Section diameter. The default value is 0.2 m

Circular hollow section - chs

This is a section frequently adopted for the modelling of circular hollow members in steel structures.



Materials and Dimensions

Only one material (steel) needs to be defined.

The required dimensions are as follows:

Section diameter. The default value is 0.2 m

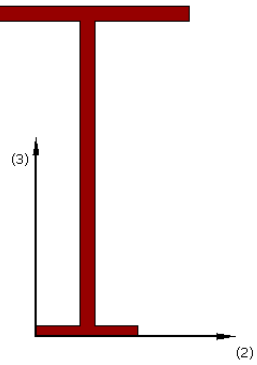
Section thickness. The default value is 0.01 m

NOTE: Users may use this section to model the retrofitting of a RC circular member with longitudinally-oriented steel. To do this, first create a material model featuring the properties of the retrofitting material and then use a chs-section element with internal diameter that equals that of the original element's section, a thickness corresponding to the thickness of the retrofitting layer, and connect this new retrofitting element to the same nodes to which the existing element is connected to.

Symmetric I- or T-section - sits

This is a section frequently adopted for the modelling of I- or T-shaped steel profiles.

NOTE 1: A T-section can be obtained by assigning identical values to bottom flange width and web thickness.



Materials and Dimensions

Only one material (steel) needs to be defined.

The required dimensions are as follows:

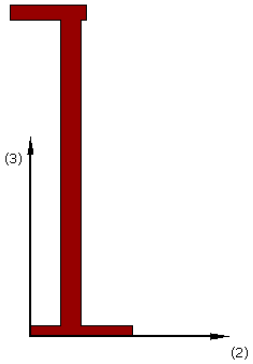
- Bottom flange width. The default value is 0.1 m
- Bottom flange thickness. The default value is 0.01 m
- Top flange width. The default value is 0.2 m
- Top flange thickness. The default value is 0.015 m
- Web height. The default value is 0.3 m
- Web thickness. The default value is 0.015 m

NOTE 2: Users may use an I-section to model the retrofitting of a RC rectangular member with longitudinally-oriented steel or FRP layers applied on the two opposite sides of the section. To do this, first create a material model featuring the properties of the retrofitting material and then use an I-section element with web height that equals that of the original element's section, a web thickness that is approximately zero, flange width/thickness dimensions corresponding to the width/thickness of the retrofitting layer, and connect this new retrofitting element to the same nodes to which the existing element is connected to. Evidently, for those cases where the fibres are placed only on one side (e.g. retrofitting of beams) a T-shaped section can be used.

Asymmetric general-shape section - agss

This is a section frequently adopted for the modelling of non-standard shape steel profiles.

NOTE: A C-shaped section can be obtained by defining zero-length bottom and top flange eccentricities. An L-shaped section, on the other hand, can be obtained by assigning identical values to top flange width and web thickness (together with bottom and top flange eccentricities equal to zero).



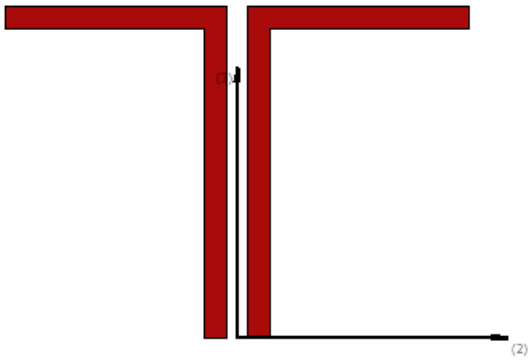
Materials and Dimensions

Only one material (steel) needs to be defined.

The required dimensions are as follows:

- Bottom flange width. The default value is 0.1 m
- Bottom flange thickness. The default value is 0.01 m
- Top flange width. The default value is 0.075 m
- Top flange thickness. The default value is 0.015 m
- Web height. The default value is 0.3 m
- Web thickness. The default value is 0.02 m
- Bottom flange eccentricity. The default value is 0.03 m
- Top flange eccentricity. The default value is 0.05 m

Double angle or channel shaped section - dacss



Materials and Dimensions

Only one material (steel) needs to be defined.

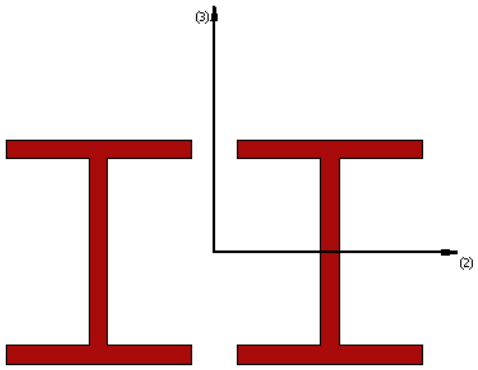
The required dimensions are as follows:

- Bottom flange width. The default value is 0.01 m
- Bottom flange thickness. The default value is 0.01 m
- Top flange width. The default value is 0.1 m
- Top flange thickness. The default value is 0.01 m
- Web height. The default value is 0.13 m

- Web thickness. The default value is 0.01 m
- Web to web distance. The default value is 0.01 m

Double I type 1 section – di1

This is a section frequently adopted for the modelling of double I steel profiles with distance between the flanges.



Materials and Dimensions

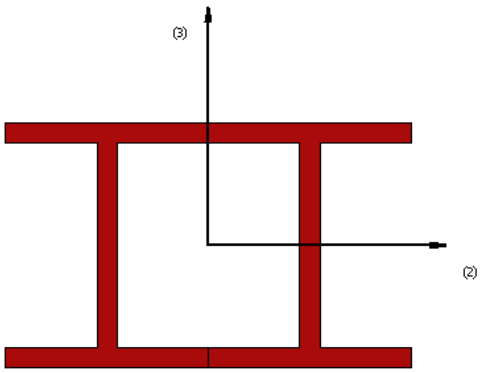
Only one material (steel) needs to be defined.

The required dimensions are as follows:

- Flange width. The default value is 0.2 m
- Flange thickness. The default value is 0.02 m
- Web height. The default value is 0.2 m
- Web thickness. The default value is 0.02 m
- Flange to Flange distance. The default value is 0.05 m

Double I type 2 section – di2

This is a section frequently adopted for the modelling of double I steel profiles without distance between the flanges.



Materials and Dimensions

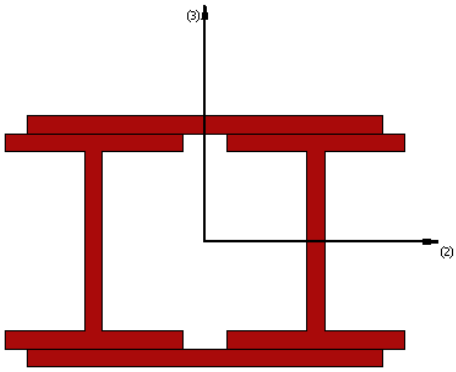
Only one material (steel) needs to be defined.

The required dimensions are as follows:

- Flange width. The default value is 0.2 m
- Flange thickness. The default value is 0.02 m
- Web height. The default value is 0.2 m
- Web thickness. The default value is 0.02 m

Double I type 1 section with top and bottom plates – di1tbp

This is a section frequently adopted for the modelling of double I steel profiles with distance between the flanges with top and bottom plates.



Materials and Dimensions

Two different materials (steel) can be defined:

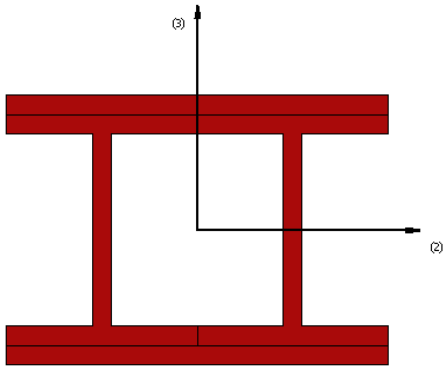
- Steel profile (I Sections),
- Steel profile (Plates).

The required dimensions are as follows:

- Flange width. The default value is 0.2 m
- Flange thickness. The default value is 0.02 m
- Web height. The default value is 0.2 m
- Web thickness. The default value is 0.02 m
- Flange to Flange distance. The default value is 0.05 m
- Reinforcing Plate Width. The default value is 0.40 m
- Reinforcing Plate Thickness. The default value is 0.02 m

Double I type 2 section with top and bottom plates – di2tbp

This is a section frequently adopted for the modelling of double I steel profiles without distance between the flanges with top and bottom plates.



Materials and Dimensions

Two different materials (steel) can be defined:

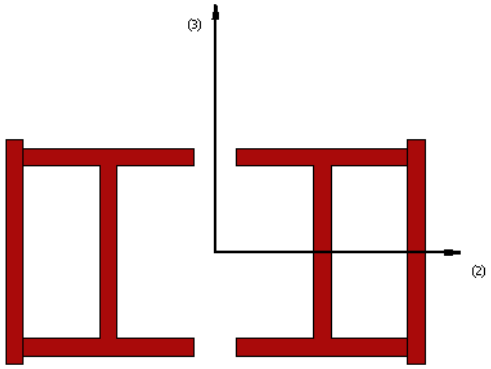
- Steel profile (I Sections),
- Steel profile (Plates).

The required dimensions are as follows:

- Flange width. The default value is 0.2 m
- Flange thickness. The default value is 0.02 m
- Web height. The default value is 0.2 m
- Web thickness. The default value is 0.02 m
- Reinforcing Plate Width. The default value is 0.40 m
- Reinforcing Plate Thickness. The default value is 0.02 m

Double I type 1 section with web plates – di1wp

This is a section frequently adopted for the modelling of double I steel profiles with distance between the flanges with web plates.



Materials and Dimensions

Two different materials (steel) can be defined:

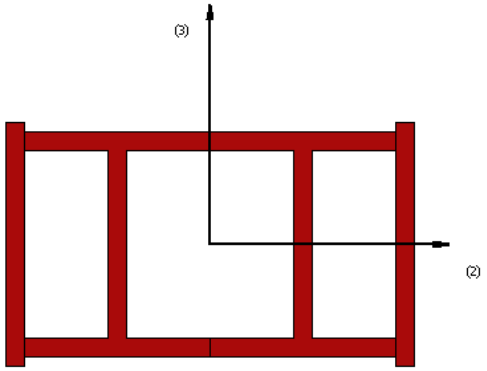
- Steel profile (I Sections),
- Steel profile (Plates).

The required dimensions are as follows:

- Flange width. The default value is 0.2 m
- Flange thickness. The default value is 0.02 m
- Web height. The default value is 0.2 m
- Web thickness. The default value is 0.02 m
- Flange to Flange distance. The default value is 0.05 m
- Reinforcing Plate Width. The default value is 0.26 m
- Reinforcing Plate Thickness. The default value is 0.02 m

Double I type 2 section with web plates – di2wp

This is a section frequently adopted for the modelling of double I steel profiles without distance between the flanges with web plates.



Materials and Dimensions

Two different materials (steel) can be defined:

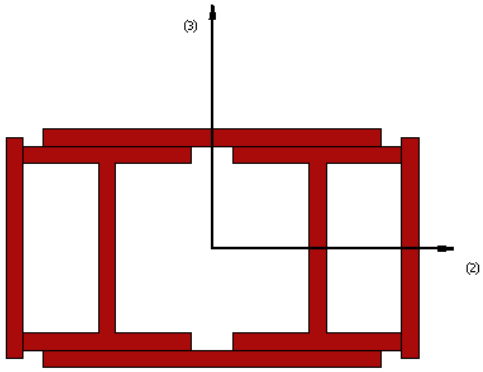
- Steel profile (I Sections),
- Steel profile (Plates).

The required dimensions are as follows:

- Flange width. The default value is 0.2 m
- Flange thickness. The default value is 0.02 m
- Web height. The default value is 0.2 m
- Web thickness. The default value is 0.02 m
- Reinforcing Plate Width. The default value is 0.26 m
- Reinforcing Plate Thickness. The default value is 0.02 m

Double I type 1 section with top, bottom and web plates – di1tbwp

This is a section frequently adopted for the modelling of double I steel profiles with distance between the flanges with top, bottom and web plates.



Materials and Dimensions

Two different materials (steel) can be defined:

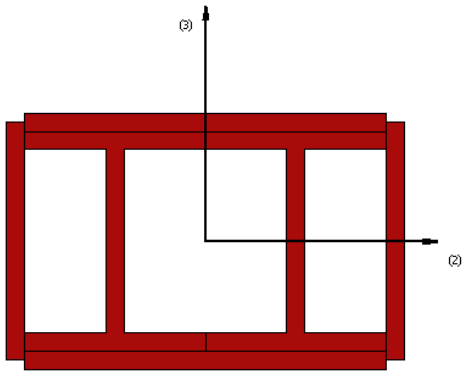
- Steel profile (I Sections),
- Steel profile (Plates).

The required dimensions are as follows:

- Flange width. The default value is 0.2 m
- Flange thickness. The default value is 0.02 m
- Web height. The default value is 0.2 m
- Web thickness. The default value is 0.02 m
- Flange to Flange distance. The default value is 0.05 m
- Top-Bottom Plate Width. The default value is 0.4 m
- Top-Bottom Plate Thickness. The default value is 0.02 m
- Web Plate Width. The default value is 0.26 m
- Web Plate Thickness. The default value is 0.02 m

Double I type 2 section with top, bottom and web plates – di2tbwp

This is a section frequently adopted for the modelling of double I steel profiles without distance between the flanges with top, bottom and web plates.



Materials and Dimensions

Two different materials (steel) can be defined:

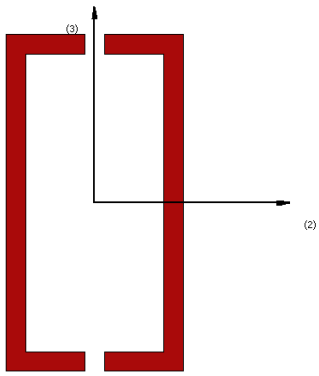
- Steel profile (I Sections),
- Steel profile (Plates).

The required dimensions are as follows:

- Flange width. The default value is 0.2 m
- Flange thickness. The default value is 0.02 m
- Web height. The default value is 0.2 m
- Web thickness. The default value is 0.02 m
- Top-Bottom Plate Width. The default value is 0.4 m
- Top-Bottom Plate Thickness. The default value is 0.02 m
- Web Plate Width. The default value is 0.26 m
- Web Plate Thickness. The default value is 0.02 m

Built up box double channel section – bbdc

This is a section frequently adopted for the modelling of double C channel steel profiles.



Materials and Dimensions

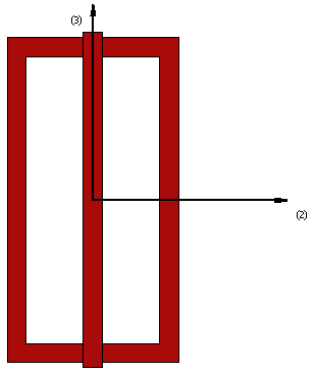
Only one material (steel) needs to be defined.

The required dimensions are as follows:

- Flange width. The default value is 0.08 m
- Flange thickness. The default value is 0.02 m
- Web height. The default value is 0.3 m
- Web thickness. The default value is 0.02 m
- Flange to Flange distance. The default value is 0.02 m

Built up box double channel section with connecting plate – bbdccp

This is a section frequently adopted for the modelling of double C channel steel profiles with connecting plate.



Materials and Dimensions

Two different materials (steel) can be defined:

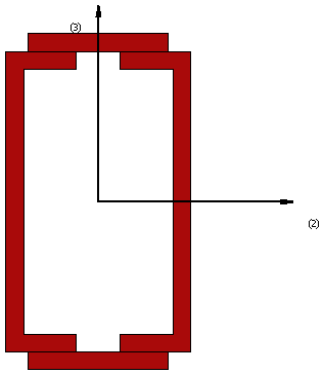
- Steel profile (Channel Sections),
- Steel profile (Plate).

The required dimensions are as follows:

- Flange width. The default value is 0.08 m
- Flange thickness. The default value is 0.02 m
- Web height. The default value is 0.3 m
- Web thickness. The default value is 0.02 m
- Reinforcing Plate Width. The default value is 0.35 m
- Reinforcing Plate Thickness. The default value is 0.02 m

Built up box double channel section with top and bottom plates – bbdctbp

This is a section frequently adopted for the modelling of double C channel steel profiles with top and bottom plates.



Materials and Dimensions

Two different materials (steel) can be defined:

- Steel profile (Channel Sections),
- Steel profile (Plates).

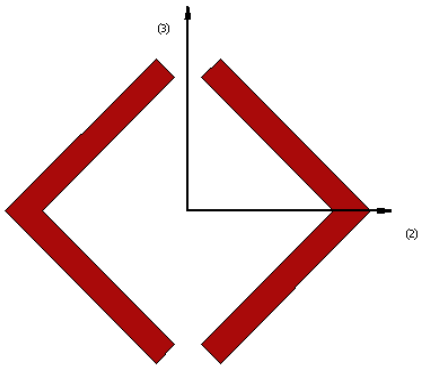
The required dimensions are as follows:

- Flange width. The default value is 0.08 m

- Flange thickness. The default value is 0.02 m
- Web height. The default value is 0.3 m
- Web thickness. The default value is 0.02 m
- Flange to Flange distance. The default value is 0.05 m
- Reinforcing Plate Width. The default value is 0.16 m
- Reinforcing Plate Thickness. The default value is 0.02 m

Built up box double angle section – bbda

This is a section frequently adopted for the modelling of double angle steel profiles.



Materials and Dimensions

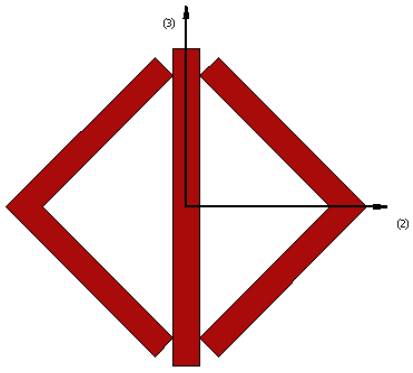
Only one material (steel) needs to be defined.

The required dimensions are as follows:

- Flange width. The default value is 0.08 m
- Flange thickness. The default value is 0.01 m
- Flange to Flange distance. The default value is 0.01 m

Built up box double angle section with connecting plate – bbdacp

This is a section frequently adopted for the modelling of double angle steel profiles with connecting plate.



Materials and Dimensions

Two different materials (steel) can be defined:

- Steel profile (Angle Sections),
- Steel profile (Plate).

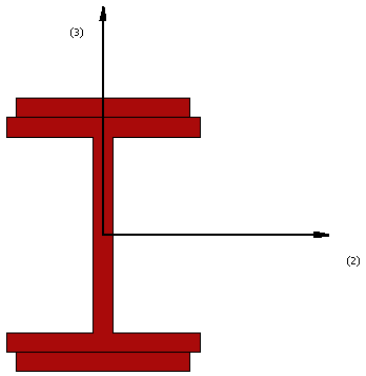
The required dimensions are as follows:

- Flange width. The default value is 0.08 m

- Flange thickness. The default value is 0.01 m
- Reinforcing Plate Width. The default value is 0.12 m
- Reinforcing Plate Thickness. The default value is 0.01 m

I section with top and bottom plates – itbp

This is a section frequently adopted for the modelling of I-shaped steel profiles with top and bottom plates.



Materials and Dimensions

Two different materials (steel) can be defined:

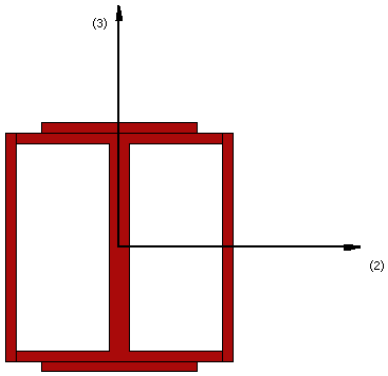
- Steel profile (I Section),
- Steel profile (Plates).

The required dimensions are as follows:

- Flange width. The default value is 0.2 m
- Flange thickness. The default value is 0.02 m
- Web width. The default value is 0.2 m
- Web thickness. The default value is 0.02 m
- Reinforcing Plate Width. The default value is 0.18 m
- Reinforcing Plate Thickness. The default value is 0.02 m

I section with top, bottom and web plates – itbwp

This is a section frequently adopted for the modelling of I-shaped steel profiles with top, bottom and web plates.



Materials and Dimensions

Two different materials (steel) can be defined:

- Steel profile (I Section),

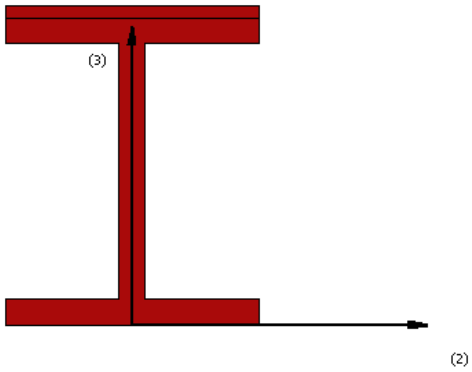
- Steel profile (Plates).

The required dimensions are as follows:

- Flange width. The default value is 0.2 m
- Flange thickness. The default value is 0.01 m
- Web width. The default value is 0.2 m
- Web thickness. The default value is 0.02 m
- Top-Bottom Plate Width. The default value is 0.15 m
- Top-Bottom Plate Thickness. The default value is 0.01 m
- Web Plate Width. The default value is 0.22 m
- Web Plate Thickness. The default value is 0.01 m

I section with top plate – itp

This is a section frequently adopted for the modelling of I-shaped steel profiles with top plate.



Materials and Dimensions

Two different materials (steel) can be defined:

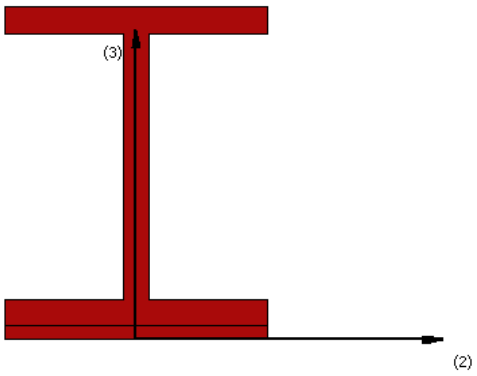
- Steel profile (I Section),
- Steel profile (Plate).

The required dimensions are as follows:

- Flange width. The default value is 0.2 m
- Flange thickness. The default value is 0.02 m
- Web width. The default value is 0.2 m
- Web thickness. The default value is 0.02 m
- Top Plate Width. The default value is 0.2 m
- Top Plate Thickness. The default value is 0.01 m

I section with bottom plate – ibp

This is a section frequently adopted for the modelling of I-shaped steel profiles with bottom plate.



Materials and Dimensions

Two different materials (steel) can be defined:

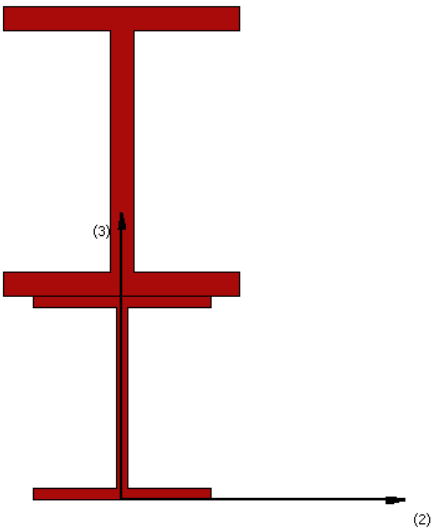
- Steel profile (I Section),
- Steel profile (Plate).

The required dimensions are as follows:

- Flange width. The default value is 0.2 m
- Flange thickness. The default value is 0.02 m
- Web width. The default value is 0.2 m
- Web thickness. The default value is 0.02 m
- Bottom Plate Width. The default value is 0.2 m
- Bottom Plate Thickness. The default value is 0.01 m

I section reinforced with bottom I section – ibri

This is a section frequently adopted for the modelling of I-shaped steel profiles reinforced with I-shaped steel profiles.



Materials and Dimensions

Two different materials (steel) can be defined:

- Steel profile (Upper I Section),

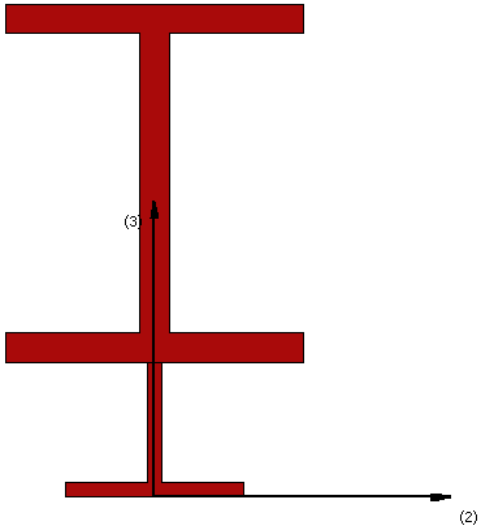
- Steel profile (Lower I Section).

The required dimensions are as follows:

- Upper Section Flange Width. The default value is 0.2 m
- Upper Section Flange Thickness. The default value is 0.02 m
- Upper Section Web Width. The default value is 0.2 m
- Upper Section Web thickness. The default value is 0.02 m
- Lower Section Flange Width. The default value is 0.15 m
- Lower Section Flange Thickness. The default value is 0.01 m
- Lower Section Web Width. The default value is 0.15 m
- Lower Section Web thickness. The default value is 0.01 m

I section reinforced with bottom T section – ibrt

This is a section frequently adopted for the modelling of I-shaped steel profiles reinforced with T-shaped steel profiles.



Materials and Dimensions

Two different materials (steel) can be defined:

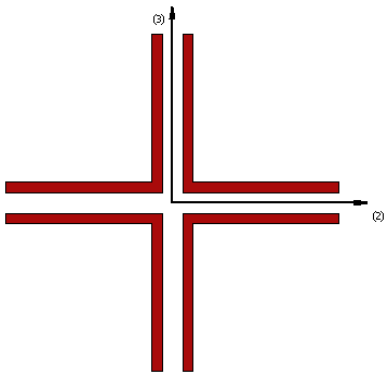
- Steel profile (Upper I Section),
- Steel profile (Lower T Section).

The required dimensions are as follows:

- I Section Flange Width. The default value is 0.2 m
- I Section Flange Thickness. The default value is 0.02 m
- I Section Web Width. The default value is 0.2 m
- I Section Web thickness. The default value is 0.02 m
- T Section Flange Width. The default value is 0.12 m
- T Section Flange Thickness. The default value is 0.01 m
- T Section Web Width. The default value is 0.08 m
- T Section Web thickness. The default value is 0.01 m

Star section composed from angle sections – sfa

This is a section frequently adopted for the modelling of star section steel profiles.



Materials and Dimensions

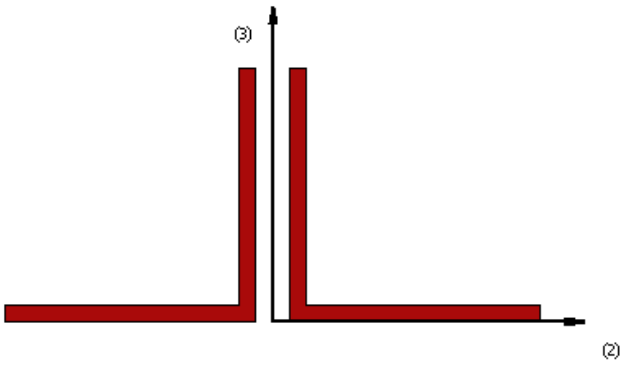
Only one material (steel) needs to be defined.

The required dimensions are as follows:

- Flange width. The default value is 0.15 m
- Flange thickness. The default value is 0.01 m
- Flange to Flange distance. The default value is 0.02 m

Double angle back-to-back section – dabtb

This is a section frequently adopted for the modelling of double angle back-to-back section steel profiles.



Materials and Dimensions

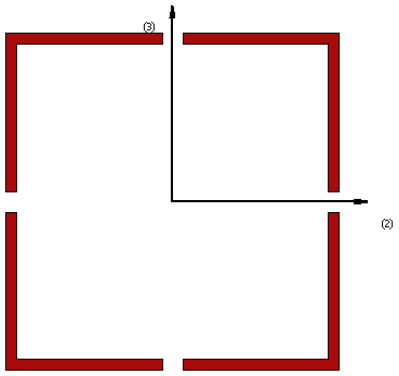
Only one material (steel) needs to be defined.

The required dimensions are as follows:

- Flange width. The default value is 0.15 m
- Flange thickness. The default value is 0.01 m
- Flange to Flange distance. The default value is 0.02 m

Built up box formed by four angle sections– bbfa

This is a section frequently adopted for the modelling of box formed by four angle sections steel profiles.



Materials and Dimensions

Only one material (steel) needs to be defined.

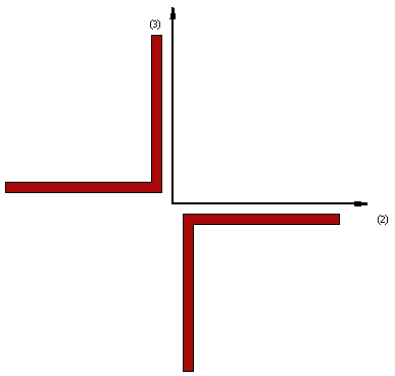
The required dimensions are as follows:

- Flange width. The default value is 0.15 m
- Flange thickness. The default value is 0.01 m
- Flange to Flange distance. The default value is 0.02 m

NOTE: Users may use a built up box formed by four angle sections to model the retrofitting of a RC rectangular member with longitudinally-oriented steel or FRP layers applied on the two opposite sides of the section. To do this, first create a material model featuring the properties of the retrofitting material and then use a built up box formed by four angle sections element that equals that of the original element's section, flange thickness dimensions corresponding to the thickness of the retrofitting layer, and connect this new retrofitting element to the same nodes to which the existing element is connected to.

Double angle section placed along the diagonal – dadg

This is a section frequently adopted for the modelling of double angle placed along the diagonal sections steel profiles.



Materials and Dimensions

Only one material (steel) needs to be defined.

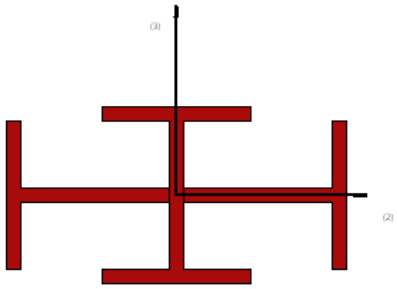
The required dimensions are as follows:

- Flange width. The default value is 0.15 m

- Flange thickness. The default value is 0.01 m
- Flange to Flange distance. The default value is 0.02 m

Cruciform Section – cfs

This is a section frequently adopted for the modelling an I section with two T sections connected symmetrically on its web forming a cross section with flanges.



Materials and Dimensions

Two different materials (steel) can be defined:

- Steel profile (I Section),
- Steel profile (Side T Sections).

The required dimensions are as follows:

- I Section Web Height. The default value is 0.2 m
- I Section Web Thickness. The default value is 0.02 m
- I Section Flange Width. The default value is 0.2 m
- I Section Flange Thickness. The default value is 0.02 m
- T Section Web Height. The default value is 0.2 m
- T Section Web Thickness. The default value is 0.02 m
- T Section Flange Width. The default value is 0.2 m
- T Section Flange Thickness. The default value is 0.02 m

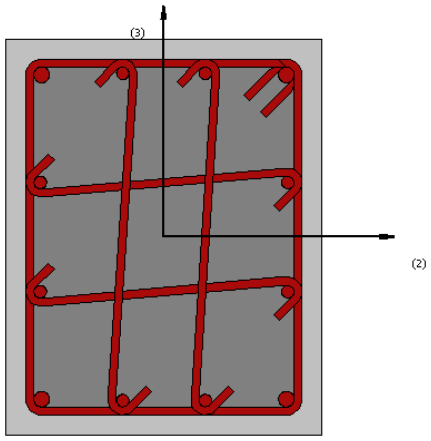
REINFORCED CONCRETE SECTIONS

NOTE 1: The confined concrete region is automatically computed by the program using the R/C cover thickness defined in the section's module (the default value is 2.5 cm).

NOTE 2: All rebars must be located within the confined concrete region.

Reinforced concrete rectangular section - rcrs

This is a section frequently adopted for the modelling of reinforced concrete rectangular columns. The use of this section to model wide columns or structural walls of any shape is also feasible, for as long as rigid links/arms featuring half of the column's/wall's width are used to connect the column's/wall's frame element to adjacent structural members, in order for the rigid body motion of the wide column/wall, and its influence on such connected structural elements, to be adequately modelled.



Materials and Dimensions

Three different materials can be defined:

- Longitudinal Reinforcement,
- Transverse Reinforcement,
- Concrete.

The required dimensions are as follows:

- Section height. The default value is 0.5 m
- Section width. The default value is 0.4 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

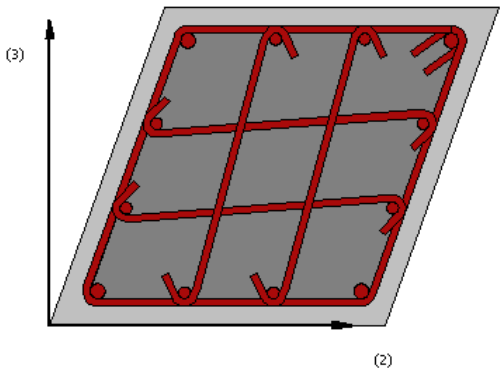
Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them. Diamond stirrups may be also added.

FRP Strengthening

FRP wrapping may be introduced by selecting the FRP Wrap from a list of the most common products found in the market, or by introducing user defined values.

Reinforced concrete quadrilateral section - rcqs

This is a section frequently adopted for the modelling of reinforced concrete quadrilateral columns.



Materials and Dimensions

Three different materials can be defined:

- Longitudinal Reinforcement,
- Transverse Reinforcement,
- Concrete.

The required dimensions are as follows:

- Side 1. The default value is 0.45 m
- Side 2. The default value is 0.45 m
- Side 3. The default value is 0.45 m
- Side 4. The default value is 0.45 m
- Angle. The default value is 70°
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them.

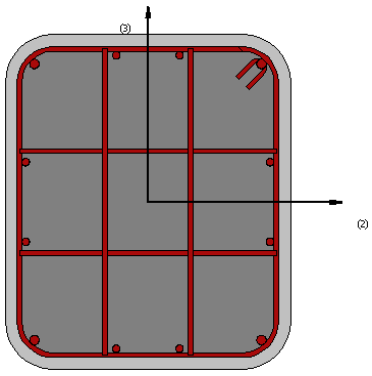
FRP Strengthening

FRP wrapping may be introduced by selecting the FRP Wrap from a list of the most common products found in the market, or by introducing user defined values.

NOTE: Re-bar distances d3 and d2 are to be measured from the bottom left corner of the section.

Reinforced concrete rectangular with rounded corners section - rcrrcs

This is a section frequently adopted for the modelling of reinforced concrete rectangular with rounded corners columns.



Materials and Dimensions

Three different materials can be defined:

- Longitudinal Reinforcement,
- Transverse Reinforcement,
- Concrete.

The required dimensions are as follows:

- Section height. The default value is 0.7 m
- Section width. The default value is 0.6 m
- Rounding corners radius. The default value is 0.1 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

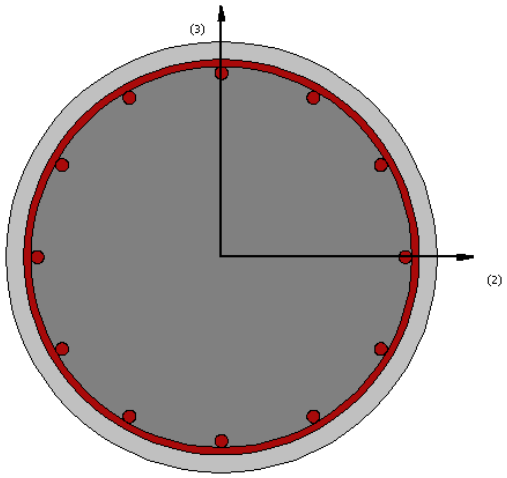
Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them. Diamond stirrups may be also added.

FRP Strengthening

FRP wrapping may be introduced by selecting the FRP Wrap from a list of the most common products found in the market, or by introducing user defined values.

Reinforced concrete circular section - rccs

This is a section frequently adopted for the modelling of reinforced concrete circular columns.



Materials and Dimensions

Three different materials can be defined:

- Longitudinal Reinforcement,
- Transverse Reinforcement,
- Concrete.

The required dimension is as follows:

- Section diameter. The default value is 0.6 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

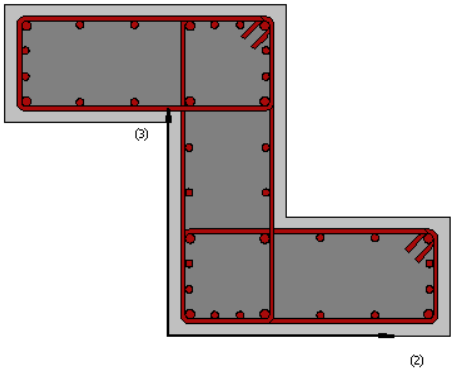
Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and distance between them, as well as to select a circular or spiral arrangement.

FRP Strengthening

FRP wrapping may be introduced by selecting the FRP Wrap from a list of the most common products found in the market, or by introducing user defined values.

Reinforced concrete Z-shaped column section – rczcs

This is a section frequently adopted for the modelling of reinforced concrete Z-shaped columns.



Materials and Dimensions

Three different materials can be defined:

- Longitudinal Reinforcement,
- Transverse Reinforcement,
- Concrete.

The required dimensions are as follows:

- Bottom flange width. The default value is 0.6 m
- Bottom flange thickness. The default value is 0.25 m
- Top flange width. The default value is 0.6 m
- Top flange thickness. The default value is 0.25 m
- Web height. The default value is 0.7 m
- Web thickness. The default value is 0.25 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them.

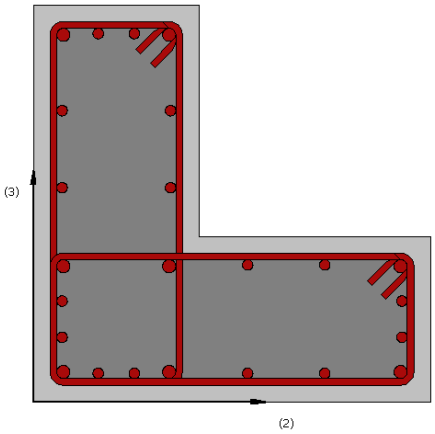
FRP Strengthening

FRP wrapping may be introduced by selecting the FRP Wrap from a list of the most common products found in the market, or by introducing user defined values.

NOTE: Re-bar distance d3 is to be measured from the bottom of the section.

Reinforced concrete L-shaped column section - rclcs

This is a section frequently adopted for the modelling of reinforced concrete L-shaped columns.



Materials and Dimensions

Three different materials can be defined:

- Longitudinal Reinforcement,
- Transverse Reinforcement,
- Concrete.

The required dimensions are as follows:

- Side 1 width. The default value is 0.6 m
- Side 2 width. The default value is 0.6 m
- Side 1 thickness. The default value is 0.25 m
- Side 2 thickness. The default value is 0.25 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them.

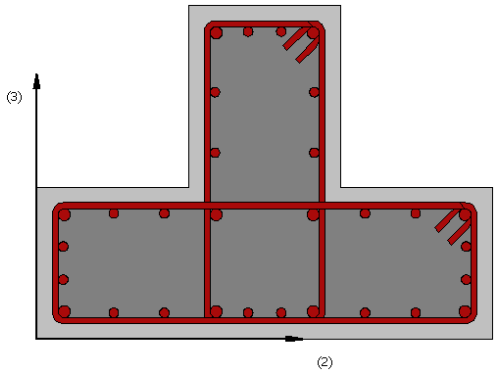
NOTE : Re-bar distance d3 is to be measured from the bottom of the section.

FRP Strengthening

FRP wrapping may be introduced by selecting the FRP Wrap from a list of the most common products found in the market, or by introducing user defined values.

Reinforced concrete T-shaped column section - rctcs

This is a section frequently adopted for the modelling of reinforced concrete T-shaped columns.



Materials and Dimensions

Three different materials can be defined:

- Longitudinal Reinforcement,
- Transverse Reinforcement,
- Concrete.

The required dimensions are as follows:

- Flange width. The default value is 0.75 m
- Flange height. The default value is 0.25 m
- Web width. The default value is 0.25 m
- Web height. The default value is 0.30 m
- Web eccentricity. The default value is 0.25 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them.

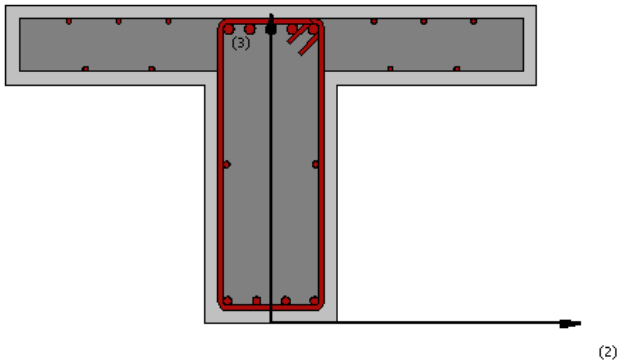
NOTE: Re-bar distance d3 is to be measured from the bottom of the section.

FRP Strengthening

FRP wrapping may be introduced by selecting the FRP Wrap from a list of the most common products found in the market, or by introducing user defined values.

Reinforced concrete T-section - rcts

This is a section frequently adopted for the modelling of reinforced concrete beams, with T-, L- or rectangular shapes (to model the L-section it is necessary to define a null beam eccentricity, whilst to model the latter users should define identical values for slab and beam widths).



Materials and Dimensions

Three different materials can be defined:

- Longitudinal Reinforcement,
- Transverse Reinforcement,
- Concrete.

The required dimensions are as follows:

- Beam height. The default value is 0.6 m
- Beam width. The default value is 0.25 m
- Slab effective width. The default value is 1 m
- Slab 1 thickness. The default value is 0.15 m
- Slab 2 thickness. The default value is 0.15 m
- Beam eccentricity. The default value is 0.375 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

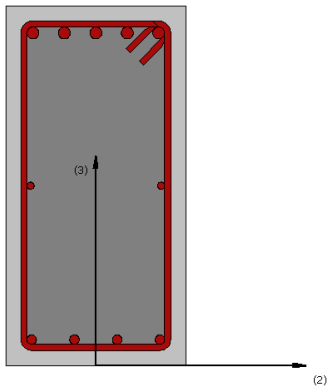
Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them.

NOTE 1: Re-bar distance d3 is to be measured from the bottom of the section.

NOTE 2: From version 6 it is possible to define asymmetric flanges thicknesses (see above).

Reinforced concrete asymmetric rectangular section - rcars

This is a section frequently adopted for the modelling of reinforced concrete rectangular beams.



Materials and Dimensions

Three different materials can be defined:

- Longitudinal Reinforcement,
- Transverse Reinforcement,
- Concrete.

The required dimensions are as follows:

- Section height. The default value is 0.6 m
- Section width. The default value is 0.3 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

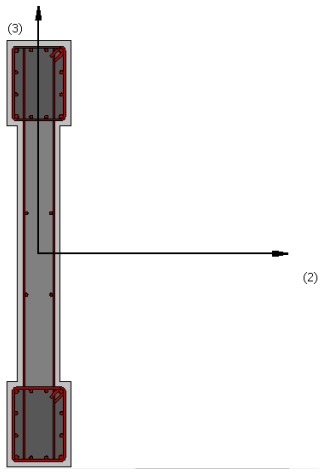
Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them.

NOTE: Re-bar distance d3 is to be measured from the bottom of the section.

Reinforced concrete rectangular wall section - rcrws

This is a section that can be adopted in the modelling of reinforced concrete walls of any shape. Rigid links/arms featuring half of the wall's width need to be used to connect the wall's frame element to adjacent structural members, in order for the rigid body motion of the wall, and its influence on such connected structural elements, to be adequately modelled.

IMPORTANT: Users are warmly advised to read the work of Beyer et al. [2008] for further guidance on this topic, especially when interested in using this cross-section to model L- or U-shaped walls.



Materials and Dimensions

Three different materials can be defined:

- Longitudinal Reinforcement,
- Transverse Reinforcement,
- Concrete.

The required dimensions are as follows:

- Wall width. The default value is 2 m
- Thickness of section edges. The default value is 0.3 m
- Width of section edges. The default value is 0.4 m
- Thickness of section core. The default value is 0.2 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

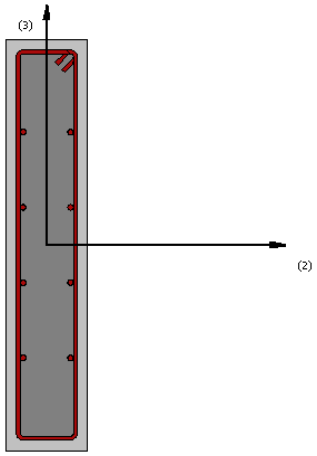
Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them. Diamond stirrups at the section's edges may be also added.

FRP Strengthening

FRP wrapping may be introduced by selecting the FRP Wrap from a list of the most common products found in the market, or by introducing user defined values.

Reinforced concrete rectangular no pseudo-columns wall section - rcbws

This is a section that can be adopted in the modelling of reinforced concrete walls without pseudo-columns.



Materials and Dimensions

Three different materials can be defined:

- Longitudinal Reinforcement,
- Transverse Reinforcement,
- Concrete.

The required dimensions are as follows:

- Wall width. The default value is 1 m
- Thickness of section core. The default value is 0.2 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them.

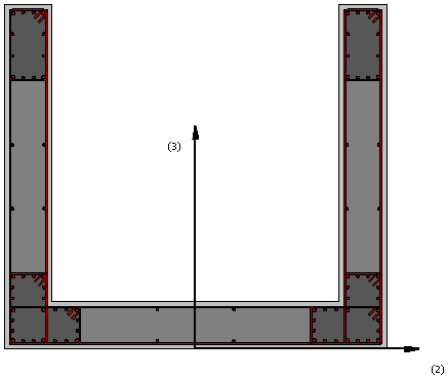
FRP Strengthening

FRP wrapping may be introduced by selecting the FRP Wrap from a list of the most common products found in the market, or by introducing user defined values.

Reinforced concrete U-shaped wall section - rcuws

This is a section that can be adopted in the modelling of isolated U-shaped reinforced concrete walls subjected to orthogonal seismic loading. If the wall finds itself inside a given building, then appropriate rigid links/arms need to be introduced in order for the rigid body motion of the wall, and its influence on connected structural elements, to be adequately modelled. For non-orthogonal (i.e. diagonal) loading, the use of this section should be avoided, and assemblage of properly connected rectangular wall sections (rcrws, rcrs) is instead strongly advised.

IMPORTANT: Users are warmly advised to read the work of Beyer et al. [2008] for further guidance on this topic.



Materials and Dimensions

Three different materials can be defined:

- Longitudinal Reinforcement,
- Transverse Reinforcement,
- Concrete.

The required dimensions are as follows:

- Back side width. The default value is 2 m
- Side 1 width. The default value is 1.8 m
- Side 2 width. The default value is 1.8 m
- Back side thickness. The default value is 0.25 m
- Side 1 thickness. The default value is 0.25 m
- Side 2 thickness. The default value is 0.25 m
- Width of back side confined edges. The default value is 0.4 m
- Width of side 1 confined edges. The default value is 0.4 m
- Width of side 2 confined edges. The default value is 0.4 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

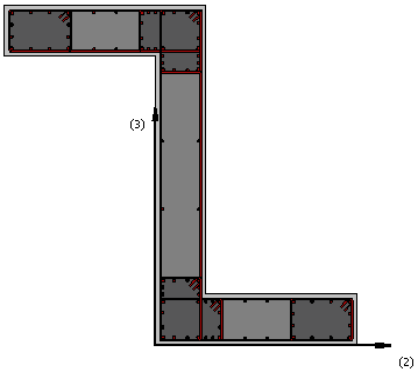
1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them. Diamond stirrups at the section's rectangular confined edges may be also added.

NOTE: Re-bar distance d3 is to be measured from the bottom of the section.

Reinforced concrete Z-shaped wall section - rczws

This is a section that can be adopted in the modelling of isolated Z-shaped reinforced concrete walls.



Materials and Dimensions

Three different materials can be defined:

- Longitudinal Reinforcement,
- Transverse Reinforcement,
- Concrete.

The required dimensions are as follows:

- Web height. The default value is 2 m
- Side 1 width. The default value is 1.2 m
- Side 2 width. The default value is 1.2 m
- Web thickness. The default value is 0.3 m
- Side 1 thickness. The default value is 0.3 m
- Side 2 thickness. The default value is 0.3 m
- Width of web confined edges. The default value is 0.4 m
- Width of side 1 confined edges. The default value is 0.4 m
- Width of side 2 confined edges. The default value is 0.4 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them. Diamond stirrups at the section's rectangular confined edges may be also added.

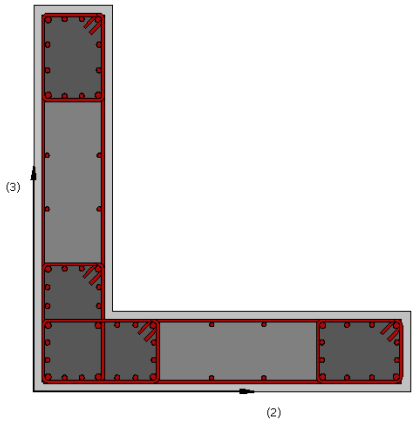
NOTE: Re-bar distance d3 is to be measured from the bottom of the section.

Reinforced concrete L-shaped wall section - rclws

This is a section that can be adopted in the modelling of isolated L-shaped reinforced concrete walls subjected to orthogonal seismic loading. If the wall finds itself inside a given building, then appropriate rigid links/arms need to be introduced in order for the rigid body motion of the wall, and its influence on connected structural elements, to be adequately modelled. For non-orthogonal (i.e. diagonal) loading,

the use of this section should be avoided, and assemblage of properly connected rectangular wall sections (rcrws, rcrs) is instead strongly advised.

IMPORTANT: Users are warmly advised to read the work of Beyer et al. [2008] for further guidance on this topic.



Materials and Dimensions

Three different materials can be defined:

- Longitudinal Reinforcement,
- Transverse Reinforcement,
- Concrete.

The required dimensions are as follows:

- Side 1 width. The default value is 1.2 m
- Side 2 width. The default value is 1.2 m
- Side 1 thickness. The default value is 0.25 m
- Side 2 thickness. The default value is 0.25 m
- Confined width 1 of central region. The default value is 0.4 m
- Confined width 2 of central region. The default value is 0.4 m
- Confined width of edge 1. The default value is 0.3 m
- Confined width of edge 2. The default value is 0.3 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

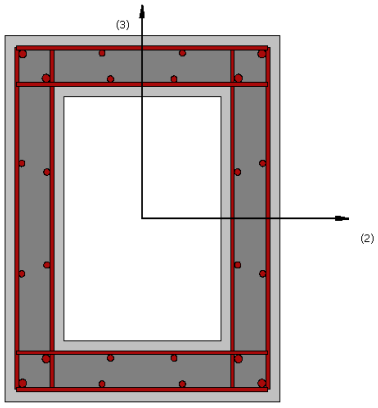
1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them. Diamond stirrups at the section's rectangular confined edges may be also added.

NOTE: Re-bar distances d3 and d2 are to be measured from the bottom left corner of the section.

Reinforced concrete rectangular hollow section - rcrhs

This is a section frequently adopted for the modelling of rectangular hollow piers, in reinforced concrete bridges.



Materials and Dimensions

Three different materials can be defined:

- Longitudinal Reinforcement,
- Transverse Reinforcement,
- Concrete.

The required dimensions are as follows:

- Outer section height. The default value is 0.9 m
- Inner section height. The default value is 0.6 m
- Outer section width. The default value is 0.7 m
- Inner section width. The default value is 0.4 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

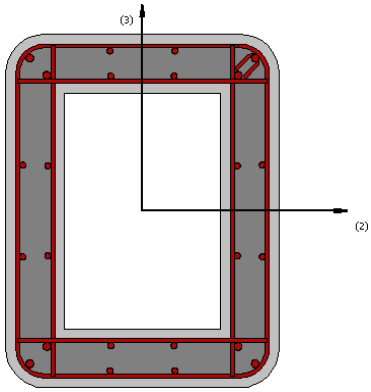
Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them.

Reinforced concrete rectangular with rounded corners hollow section - rcrrchs

This is a section frequently adopted for the modelling of rectangular hollow piers with rounded corners, in reinforced concrete bridges.



Materials and Dimensions

Three different materials can be defined:

- Longitudinal Reinforcement,
- Transverse Reinforcement,
- Concrete.

The required dimensions are as follows:

- Outer section height. The default value is 0.9 m
- Inner section height. The default value is 0.6 m
- Outer section width. The default value is 0.7 m
- Inner section width. The default value is 0.4 m
- Rounding corners radius. The default value is 0.1 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

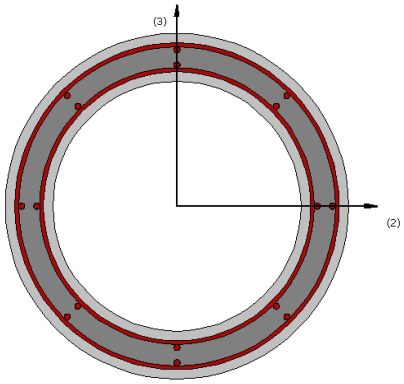
Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them.

Reinforced concrete circular hollow section - rcchs

This is a section frequently adopted for the modelling of circular hollow piers, in reinforced concrete bridges.



Materials and Dimensions

Three different materials can be defined:

- Longitudinal Reinforcement,
- Transverse Reinforcement,
- Concrete.

The required dimensions are as follows:

- Outer section diameter. The default value is 0.9 m
- Inner section diameter. The default value is 0.65 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

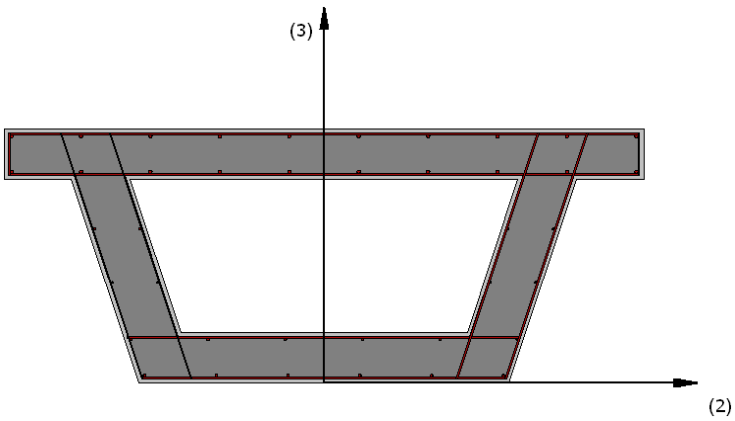
Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and distance between them, as well as to select a circular or spiral arrangement.

Reinforced concrete box-girder section - rcbgs

This is a section frequently adopted for the modelling of hollow-core concrete girders.



Materials and Dimensions

Three different materials can be defined:

- Longitudinal Reinforcement,
- Transverse Reinforcement,
- Concrete.

The required dimensions are as follows:

- Height under flanges. The default value is 1.2 m
- Top width. The default value is 3 m
- Base width. The default value is 2.2 m
- Web thickness. The default value is 0.35m
- Top thickness. The default value is 0.3 m
- Base thickness. The default value is 0.3 m
- Flange width. The default value is 0.4 m
- Flange height. The default value is 0.3 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

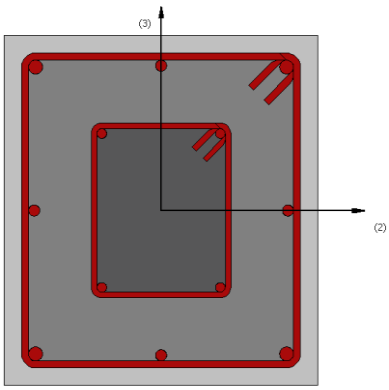
Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them.

NOTE: Re-bar distance d3 is to be measured from the bottom of the section.

JACKETED REINFORCED CONCRETE SECTIONS

Reinforced concrete jacketed rectangular section - rcjrs

This is a section frequently adopted for the modelling of rectangular columns that have been retrofitted by means of reinforced concrete jacketing (steel- or FRP-wrapping can be modelled using the existing RC sections). The possibility of defining different confinement levels for the internal (pre-existing) and the external (new) concrete materials is available.



Materials and Dimensions

Six different materials can be defined:

- External Longitudinal Reinforcement,
- Internal Longitudinal Reinforcement,
- External Transverse Reinforcement,
- Internal Transverse Reinforcement,
- Concrete jacket,
- Concrete core.

The required dimensions are as follows:

- External height. The default value is 0.5 m
- Internal height. The default value is 0.25 m
- External width. The default value is 0.45 m
- Internal width. The default value is 0.2 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

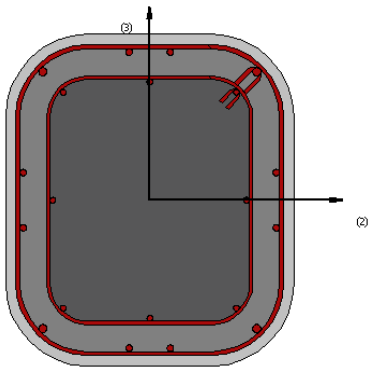
Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them. Diamond stirrups may be also added.

FRP Strengthening

FRP wrapping may be introduced by selecting the FRP Wrap from a list of the most common products found in the market, or by introducing user defined values.

Reinforced concrete jacketed rectangular with rounded corners section – rcjrrcs

This is a section frequently adopted for the modelling of rectangular with rounded corners columns that have been retrofitted by means of reinforced concrete jacketing (steel- or FRP-wrapping can be modelled using the existing RC sections). The possibility of defining different confinement levels for the internal (pre-existing) and the external (new) concrete materials is available.



Materials and Dimensions

Six different materials can be defined:

- External Longitudinal Reinforcement,
- Internal Longitudinal Reinforcement,
- External Transverse Reinforcement,

- Internal Transverse Reinforcement,
- Concrete jacket,
- Concrete core.

The required dimensions are as follows:

- External height. The default value is 0.8 m
- External width. The default value is 0.7 m
- Internal rounding corner radius. The default value is 0.1 m
- Jacket thickness. The default value is 0.1 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

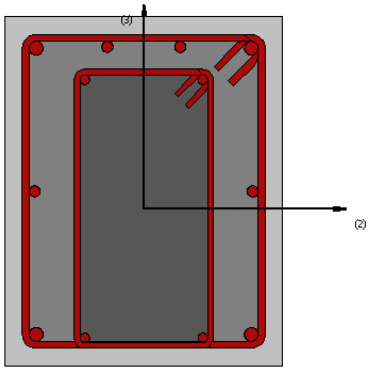
Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them. Diamond stirrups may be also added.

FRP Strengthening

FRP wrapping may be introduced by selecting the FRP Wrap from a list of the most common products found in the market, or by introducing user defined values.

Reinforced concrete 3-side jacketed rectangular section – rcjrs3

This is a section frequently adopted for the modelling of rectangular columns that have been retrofitted in three sides by means of reinforced concrete jacketing (steel- or FRP-wrapping can be modelled using the existing RC sections). The possibility of defining different confinement levels for the internal (pre-existing) and the external (new) concrete materials is available.



Materials and Dimensions

Six different materials can be defined:

- External Longitudinal Reinforcement,
- Internal Longitudinal Reinforcement,
- External Transverse Reinforcement,
- Internal Transverse Reinforcement,
- Concrete jacket,
- Concrete core.

The required dimensions are as follows:

- External height. The default value is 0.5 m

- Internal height. The default value is 0.4 m
- External width. The default value is 0.4 m
- Internal width. The default value is 0.2 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

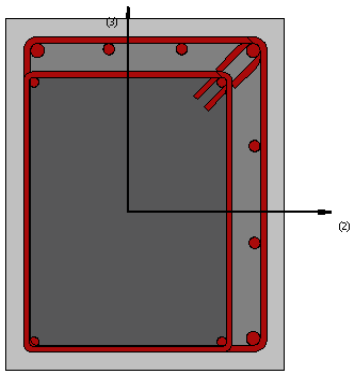
Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them. Diamond stirrups may be also added.

FRP Strengthening

FRP wrapping may be introduced by selecting the FRP Wrap from a list of the most common products found in the market, or by introducing user defined values.

Reinforced concrete 2-side jacketed rectangular section – rcjrs2

This is a section frequently adopted for the modelling of rectangular columns that have been retrofitted in two sides by means of reinforced concrete jacketing (steel- or FRP-wrapping can be modelled using the existing RC sections). The possibility of defining different confinement levels for the internal (pre-existing) and the external (new) concrete materials is available.



Materials and Dimensions

Six different materials can be defined:

- External Longitudinal Reinforcement,
- Internal Longitudinal Reinforcement,
- External Transverse Reinforcement,
- Internal Transverse Reinforcement,
- Concrete jacket,
- Concrete core.

The required dimensions are as follows:

- External height. The default value is 0.5 m
- Internal height. The default value is 0.4 m
- External width. The default value is 0.4 m
- Internal width. The default value is 0.3 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

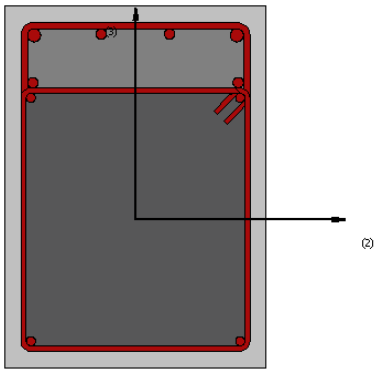
Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them. Diamond stirrups may be also added.

FRP Strengthening

FRP wrapping may be introduced by selecting the FRP Wrap from a list of the most common products found in the market, or by introducing user defined values.

Reinforced concrete 1-side jacketed rectangular section – rcjrs1

This is a section frequently adopted for the modelling of rectangular columns that have been retrofitted in one side by means of reinforced concrete jacketing (steel- or FRP-wrapping can be modelled using the existing RC sections). The possibility of defining different confinement levels for the internal (pre-existing) and the external (new) concrete materials is available.



Materials and Dimensions

Six different materials can be defined:

- External Longitudinal Reinforcement,
- Internal Longitudinal Reinforcement,
- External Transverse Reinforcement,
- Internal Transverse Reinforcement,
- Concrete jacket,
- Concrete core.

The required dimensions are as follows:

- External height. The default value is 0.55 m
- Internal height. The default value is 0.4 m
- Section width. The default value is 0.4 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

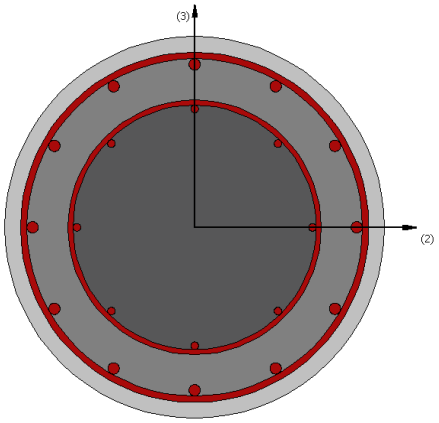
Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them. Diamond stirrups may be also added.

FRP Strengthening

FRP wrapping may be introduced by selecting the FRP Wrap from a list of the most common products found in the market, or by introducing user defined values.

Reinforced concrete jacketed circular section - rcjcs

This is a section frequently adopted for the modelling of circular columns that have been retrofitted by means of reinforced concrete jacketing (steel- or FRP-wrapping can be modelled using the existing RC sections). The possibility of defining different confinement levels for the internal (pre-existing) and the external (new) concrete materials is available.



Materials and Dimensions

Six different materials can be defined:

- External Longitudinal Reinforcement,
- Internal Longitudinal Reinforcement,
- External Transverse Reinforcement,
- Internal Transverse Reinforcement,
- Concrete jacket,
- Concrete core.

The required dimensions are as follows:

- External Diameter. The default value is 0.6 m
- Internal Diameter. The default value is 0.4 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

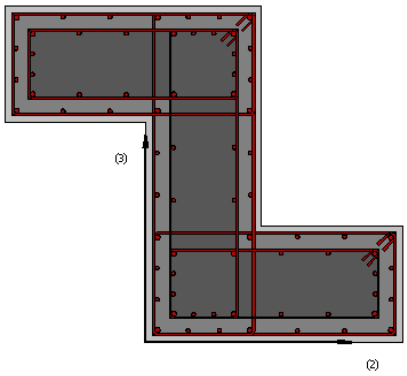
Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and distance between them, as well as to select a circular or spiral arrangement.

FRP Strengthening

FRP wrapping may be introduced by selecting the FRP Wrap from a list of the most common products found in the market, or by introducing user defined values.

Reinforced concrete jacketed Z-shaped column section – rcjzcs

This is a section frequently adopted for the modelling of Z-shaped columns that have been retrofitted by means of reinforced concrete jacketing (steel- or FRP-wrapping can be modelled using the existing RC sections). The possibility of defining different confinement levels for the internal (pre-existing) and the external (new) concrete materials is available.



Materials and Dimensions

Six different materials can be defined:

- External Longitudinal Reinforcement,
- Internal Longitudinal Reinforcement,
- External Transverse Reinforcement,
- Internal Transverse Reinforcement,
- Concrete jacket,
- Concrete core.

The required dimensions are as follows:

- Bottom flange width. The default value is 1 m
- Bottom flange thickness. The default value is 0.45 m
- Top flange width. The default value is 1 m
- Top flange thickness. The default value is 0.45 m
- Web height. The default value is 1.3 m
- Web thickness. The default value is 0.45 m
- Jacket thickness. The default value is 0.1 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them.

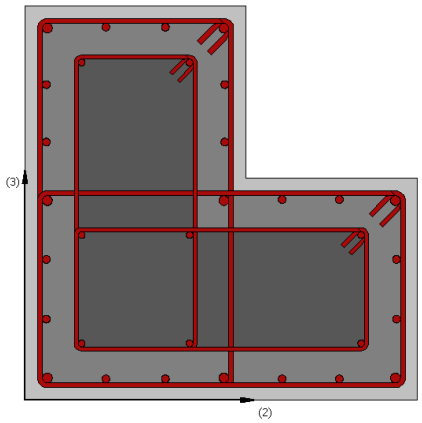
NOTE: Re-bar distance d3 is to be measured from the bottom of the section.

FRP Strengthening

FRP wrapping may be introduced by selecting the FRP Wrap from a list of the most common products found in the market, or by introducing user defined values.

Reinforced concrete jacketed L-shaped column section - rcjlcs

This is a section frequently adopted for the modelling of L-shaped columns that have been retrofitted by means of reinforced concrete jacketing (steel- or FRP-wrapping can be modelled using the existing RC sections). The possibility of defining different confinement levels for the internal (pre-existing) and the external (new) concrete materials is available.



Materials and Dimensions

Six different materials can be defined:

- External Longitudinal Reinforcement,
- Internal Longitudinal Reinforcement,
- External Transverse Reinforcement,
- Internal Transverse Reinforcement,
- Concrete jacket,
- Concrete core.

The required dimensions are as follows:

- Side 1 width. The default value is 0.8 m
- Side 2 width. The default value is 0.8 m
- Side 1 thickness. The default value is 0.45 m
- Side 2 thickness. The default value is 0.45 m
- Jacket thickness. The default value is 0.1 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them.

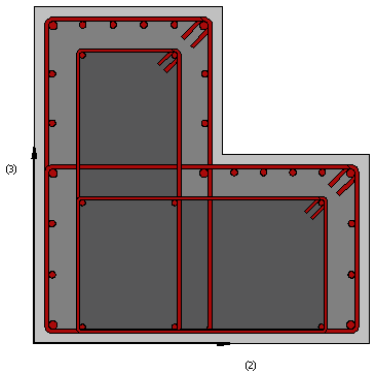
NOTE: Re-bar distances d3 and d2 are to be measured from the bottom left corner of the section.

FRP Strengthening

FRP wrapping may be introduced by selecting the FRP Wrap from a list of the most common products found in the market, or by introducing user defined values.

Reinforced concrete 3-side jacketed L-shaped column section – rcjlcs3

This is a section frequently adopted for the modelling of L-shaped columns that have been retrofitted in three sides by means of reinforced concrete jacketing (steel- or FRP-wrapping can be modelled using the existing RC sections). The possibility of defining different confinement levels for the internal (pre-existing) and the external (new) concrete materials is available.



Materials and Dimensions

Six different materials can be defined:

- External Longitudinal Reinforcement,
- Internal Longitudinal Reinforcement,
- External Transverse Reinforcement,
- Internal Transverse Reinforcement,
- Concrete jacket,
- Concrete core.

The required dimensions are as follows:

- Side 1 width. The default value is 0.8 m
- Side 2 width. The default value is 0.8 m
- Side 1 thickness. The default value is 0.45 m
- Side 2 thickness. The default value is 0.45 m
- Jacket thickness. The default value is 0.1 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them.

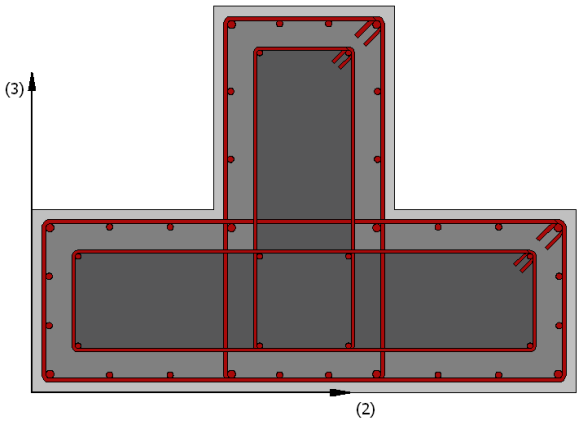
NOTE: Re-bar distances d3 and d2 are to be measured from the bottom left corner of the section.

FRP Strengthening

FRP wrapping may be introduced by selecting the FRP Wrap from a list of the most common products found in the market, or by introducing user defined values.

Reinforced concrete jacketed T-shaped column section - rcjtcs

This is a section frequently adopted for the modelling of T-shaped columns that have been retrofitted by means of reinforced concrete jacketing (steel- or FRP-wrapping can be modelled using the existing RC sections). The possibility of defining different confinement levels for the internal (pre-existing) and the external (new) concrete materials is available.



Materials and Dimensions

Six different materials can be defined:

- External Longitudinal Reinforcement,
- Internal Longitudinal Reinforcement,
- External Transverse Reinforcement,
- Internal Transverse Reinforcement,
- Concrete jacket,
- Concrete core.

The required dimensions are as follows:

- Flange width. The default value is 1.35 m
- Flange height. The default value is 0.45 m
- Web width. The default value is 0.45 m
- Web height. The default value is 0.5 m
- Web eccentricity. The default value is 0.45 m
- Jacket thickness. The default value is 0.1 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them.

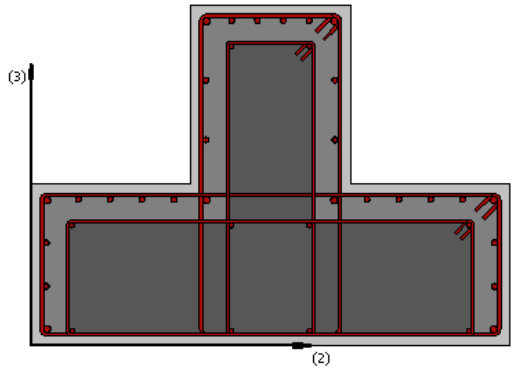
NOTE: Re-bar distances d3 and d2 are to be measured from the bottom left corner of the section.

FRP Strengthening

FRP wrapping may be introduced by selecting the FRP Wrap from a list of the most common products found in the market, or by introducing user defined values.

Reinforced concrete 3-side jacketed T-shaped column section – rcjtcs3

This is a section frequently adopted for the modelling of T-shaped columns that have been retrofitted in three sides by means of reinforced concrete jacketing (steel- or FRP-wrapping can be modelled using the existing RC sections). The possibility of defining different confinement levels for the internal (pre-existing) and the external (new) concrete materials is available.



Materials and Dimensions

Six different materials can be defined:

- External Longitudinal Reinforcement,
- Internal Longitudinal Reinforcement,
- External Transverse Reinforcement,
- Internal Transverse Reinforcement,
- Concrete jacket,
- Concrete core.

The required dimensions are as follows:

- Flange width. The default value is 1.35 m
- Flange height. The default value is 0.45 m
- Web width. The default value is 0.45 m
- Web height. The default value is 0.5 m
- Web eccentricity. The default value is 0.45 m
- Jacket thickness. The default value is 0.1 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them.

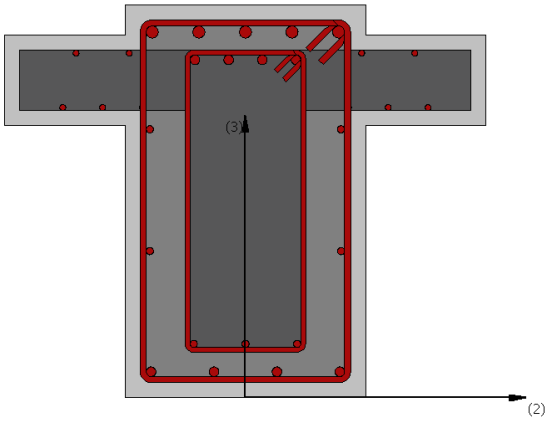
NOTE: Re-bar distances d3 and d2 are to be measured from the bottom left corner of the section.

FRP Strengthening

FRP wrapping may be introduced by selecting the FRP Wrap from a list of the most common products found in the market, or by introducing user defined values.

Reinforced concrete jacketed T-section - rcjts

This is a section frequently adopted for the modelling of reinforced concrete beams, with T-, L- or rectangular shapes (to model the L-section it is necessary to define a null beam eccentricity, whilst to model the latter users should define identical values for slab and beam widths) that have been retrofitted by means of reinforced concrete jacketing (steel- or FRP-wrapping can be modelled using the existing RC sections). The possibility of defining different confinement levels for the internal (pre-existing) and the external (new) concrete materials is available.



Materials and Dimensions

Six different materials can be defined:

- External Longitudinal Reinforcement,
- Internal Longitudinal Reinforcement,
- External Transverse Reinforcement,
- Internal Transverse Reinforcement,
- Concrete jacket,
- Concrete core.

The required dimensions are as follows:

- External Beam height. The default value is 0.65 m
- External Beam width. The default value is 0.4 m
- External Slab height. The default value is 0.05 m
- Internal Beam height. The default value is 0.5 m
- Internal Beam width. The default value is 0.2 m
- Slab effective width. The default value is 0.8 m
- Slab 1 thickness. The default value is 0.15 m
- Slab 2 thickness. The default value is 0.15 m
- Beam eccentricity. The default value is 0.3 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

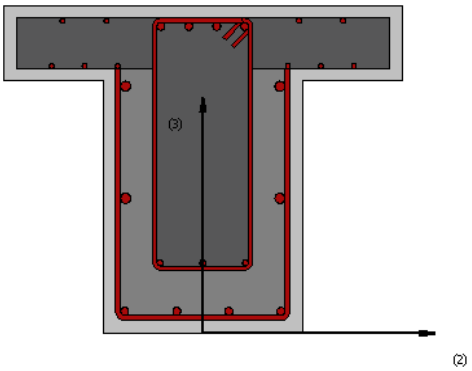
1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them.

NOTE: Re-bar distance d3 is to be measured from the bottom of the section.

Reinforced concrete 3-side jacketed T-section – rcjts3

This is a section frequently adopted for the modelling of reinforced concrete beams, with T-, L- or rectangular shapes (to model the L-section it is necessary to define a null beam eccentricity, whilst to model the latter users should define identical values for slab and beam widths) that have been retrofitted in three sides by means of reinforced concrete jacketing (steel- or FRP-wrapping can be modelled using the existing RC sections). The possibility of defining different confinement levels for the internal (pre-existing) and the external (new) concrete materials is available.



Materials and Dimensions

Six different materials can be defined:

- External Longitudinal Reinforcement,
- Internal Longitudinal Reinforcement,
- External Transverse Reinforcement,
- Internal Transverse Reinforcement,
- Concrete jacket,
- Concrete core.

The required dimensions are as follows:

- External Beam height. The default value is 0.65 m
- External Beam width. The default value is 0.4 m
- Internal Beam height. The default value is 0.5 m
- Internal Beam width. The default value is 0.2 m
- Slab effective width. The default value is 0.8 m
- Slab 1 thickness. The default value is 0.15 m
- Slab 2 thickness. The default value is 0.15 m
- Beam eccentricity. The default value is 0.3 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

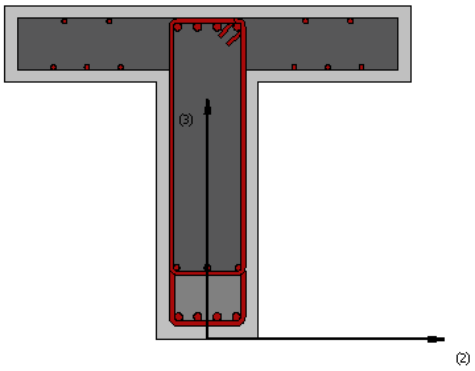
1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them.

NOTE: Re-bar distance d3 is to be measured from the bottom of the section.

Reinforced concrete 1-side jacketed T-section – rcjts1

This is a section frequently adopted for the modelling of reinforced concrete beams, with T-, L- or rectangular shapes (to model the L-section it is necessary to define a null beam eccentricity, whilst to model the latter users should define identical values for slab and beam widths) that have been retrofitted in one side by means of reinforced concrete jacketing (steel- or FRP-wrapping can be modelled using the existing RC sections). The possibility of defining different confinement levels for the internal (pre-existing) and the external (new) concrete materials is available.



Materials and Dimensions

Six different materials can be defined:

- External Longitudinal Reinforcement,
- Internal Longitudinal Reinforcement,
- External Transverse Reinforcement,
- Internal Transverse Reinforcement,
- Concrete jacket,
- Concrete core.

The required dimensions are as follows:

- External Beam height. The default value is 0.65 m
- Internal Beam height. The default value is 0.5 m
- Beam width. The default value is 0.2 m
- Slab effective width. The default value is 0.8 m
- Slab 1 thickness. The default value is 0.15 m
- Slab 2 thickness. The default value is 0.15 m
- Beam eccentricity. The default value is 0.3 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

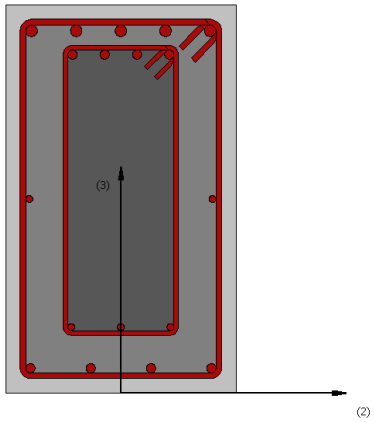
1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them.

NOTE: Re-bar distance d3 is to be measured from the bottom of the section.

Reinforced concrete jacketed asymmetric rectangular section - rcjars

This is a section frequently adopted for the modelling of reinforced concrete rectangular beams that have been retrofitted by means of reinforced concrete jacketing (steel- or FRP-wrapping can be modelled using the existing RC sections). The possibility of defining different confinement levels for the internal (pre-existing) and the external (new) concrete materials is available.



Six different materials can be defined:

- External Longitudinal Reinforcement,
- Internal Longitudinal Reinforcement,
- External Transverse Reinforcement,
- Internal Transverse Reinforcement,
- Concrete jacket,
- Concrete core.

The required dimensions are as follows:

- External height. The default value is 0.67 m
- External width. The default value is 0.4 m
- External upper flange thickness. The default value is 0.07 m
- Internal height. The default value is 0.5 m
- Internal width. The default value is 0.2 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

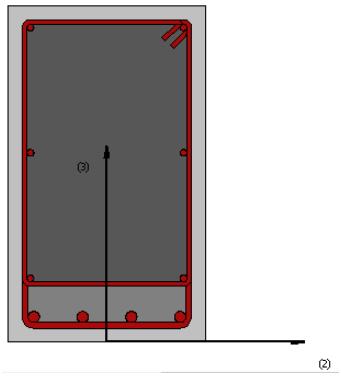
1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them.

NOTE: Re-bar distance d3 is to be measured from the bottom of the section.

Reinforced concrete 1-side jacketed asymmetric rectangular section – rcjars1

This is a section frequently adopted for the modelling of reinforced concrete rectangular beams that have been retrofitted in one side by means of reinforced concrete jacketing (steel- or FRP-wrapping can be modelled using the existing RC sections). The possibility of defining different confinement levels for the internal (pre-existing) and the external (new) concrete materials is available.



Six different materials can be defined:

- External Longitudinal Reinforcement,
- Internal Longitudinal Reinforcement,
- External Transverse Reinforcement,
- Internal Transverse Reinforcement,
- Concrete jacket,
- Concrete core.

The required dimensions are as follows:

- External height. The default value is 0.6 m
- Internal height. The default value is 0.5 m
- Section width. The default value is 0.35 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them.

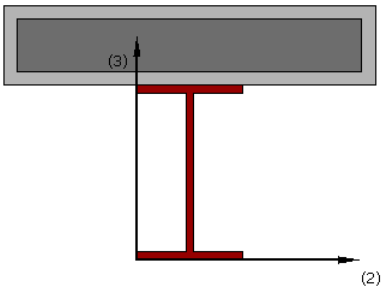
NOTE: Re-bar distance d3 is to be measured from the bottom of the section.

COMPOSITE SECTIONS

Composite I-section - cpis

This is a section frequently adopted for the modelling of simply-supported composite beams.

NOTE: The reinforcement in the concrete slab is currently not modelled; hence the section will have a reduced negative moment resistance capacity.



Materials and Dimensions

Two different materials can be defined:

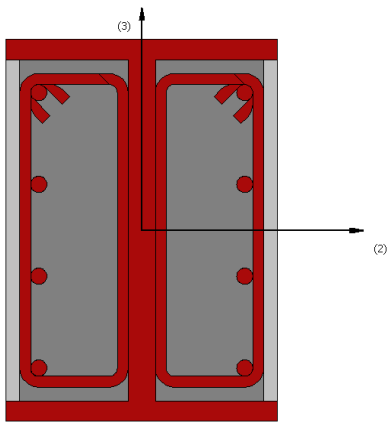
- Steel profile,
- Concrete.

The required dimensions are as follows:

- Bottom flange width. The default value is 0.1 m
- Bottom flange thickness. The default value is 0.01 m
- Top flange width. The default value is 0.2 m
- Top flange thickness. The default value is 0.015 m
- Web height. The default value is 0.3 m
- Web thickness. The default value is 0.015 m
- Slab effective width. The default value is 1 m
- Confined width in slab. The default value is 0.95 m
- Slab thickness. The default value is 0.15 m
- Confined thickness in slab. The default value is 0.1 m

Partially encased composite I-section - pecs

This is a section frequently adopted for the modelling of composite columns.



Materials and Dimensions

Four different materials can be defined:

- Longitudinal Reinforcement,
- Transverse Reinforcement,
- Steel profile,
- Concrete.

The required dimensions are as follows:

- Flange width. The default value is 0.2 m
- Flange thickness. The default value is 0.015 m
- Web height. The default value is 0.25 m
- Web thickness. The default value is 0.02 m
- Unconfined concrete thickness. The default value is 0.01 m

Reinforcement

Longitudinal reinforcement bars can be defined in two different ways:

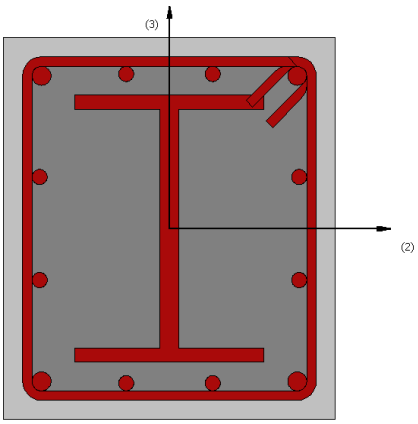
1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them.

NOTE: A parabolic curve has been assumed to represent the boundary between fully and partially confined concrete areas. Its depth may be conservatively estimated as 20% of the profile's flange width. More rigorous estimation procedures, however, can be found in the work of Mirza [1989] or Elnashai and Elghazouli [1993], amongst others.

Fully encased composite I-section - fecs

This is a section frequently adopted for the modelling of composite columns.



Materials and Dimensions

Four different materials can be defined:

- Longitudinal Reinforcement,
- Transverse Reinforcement,
- Steel profile,
- Concrete.

The required dimensions are as follows:

- Flange width. The default value is 0.2 m
- Flange thickness. The default value is 0.015 m
- Web height. The default value is 0.25 m
- Web thickness. The default value is 0.02 m
- Stirrup width. The default value is 0.31 m
- Section width. The default value is 0.35 m
- Stirrup height. The default value is 0.36 m
- Section height. The default value is 0.4 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

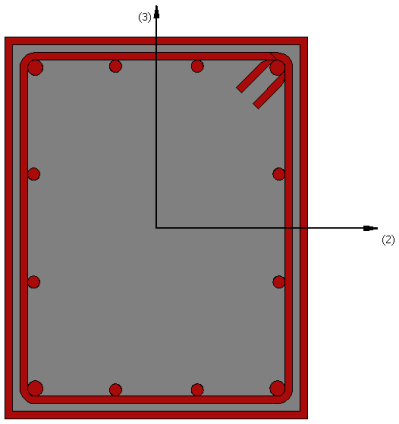
Longitudinal reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Further, the transverse reinforcement may be specified, in terms of the stirrups' diameters and legs, as well as the distance between them.

Composite rectangular section - crs

This is a section frequently adopted for the modelling of composite columns.



Materials and Dimensions

Four different materials can be defined:

- Longitudinal Reinforcement,
- Transverse Reinforcement,
- Steel tube,
- Concrete.

The required dimensions are as follows:

- Section height. The default value is 0.5 m
- Section width. The default value is 0.4 m
- Steel Thickness. The default value is 0.01 m

Reinforcement

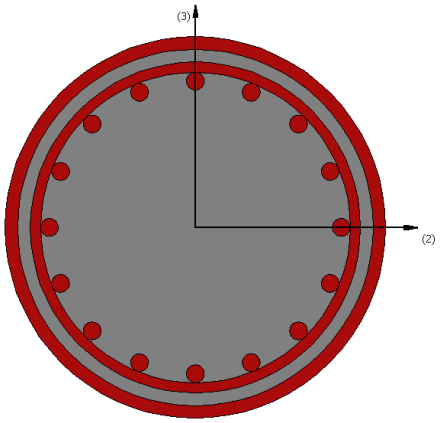
Reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;

2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Composite circular section - ccs

This is a section frequently adopted for the modelling of composite columns.



Materials and Dimensions

Four different materials can be defined:

- Longitudinal Reinforcement,
- Transverse Reinforcement,
- Steel tube,
- Concrete.

The required dimensions are as follows:

- Section Diameter. The default value is 0.3 m
- Steel Thickness. The default value is 0.01 m

Reinforcement

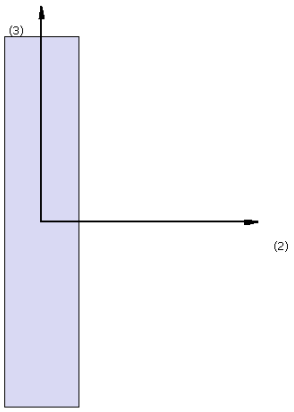
Reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

MASONRY SECTIONS

Masonry wall section - mws

This is a section frequently adopted for the modelling of masonry members.



Materials and Dimensions

Three different materials can be defined:

- Longitudinal Reinforcement,
- Transverse Reinforcement,
- Masonry

The required dimensions are as follows:

- Wall width (in-plane dimension). The default value is 2.0 m
- Wall thickness (out-of-plane dimension). The default value is 0.4 m
- Cover Thickness. The default value is 0.025 m

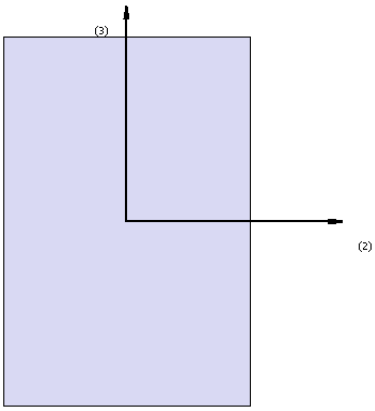
Reinforcement

Reinforcement bars can be defined in two different ways:

1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Masonry spandrel section - mss

This is a section frequently adopted for the modelling of coupling beams bertween two wall elements.



Materials and Dimensions

Three different materials can be defined:

- Longitudinal Reinforcement,
- Transverse Reinforcement,
- Masonry

The required dimensions are as follows:

- Spandrel height (in-plane dimension). The default value is 0.6 m
- Spandrel thickness (out-of-plane dimension). The default value is 0.4 m
- Cover Thickness. The default value is 0.025 m

Reinforcement

Reinforcement bars can be defined in two different ways:

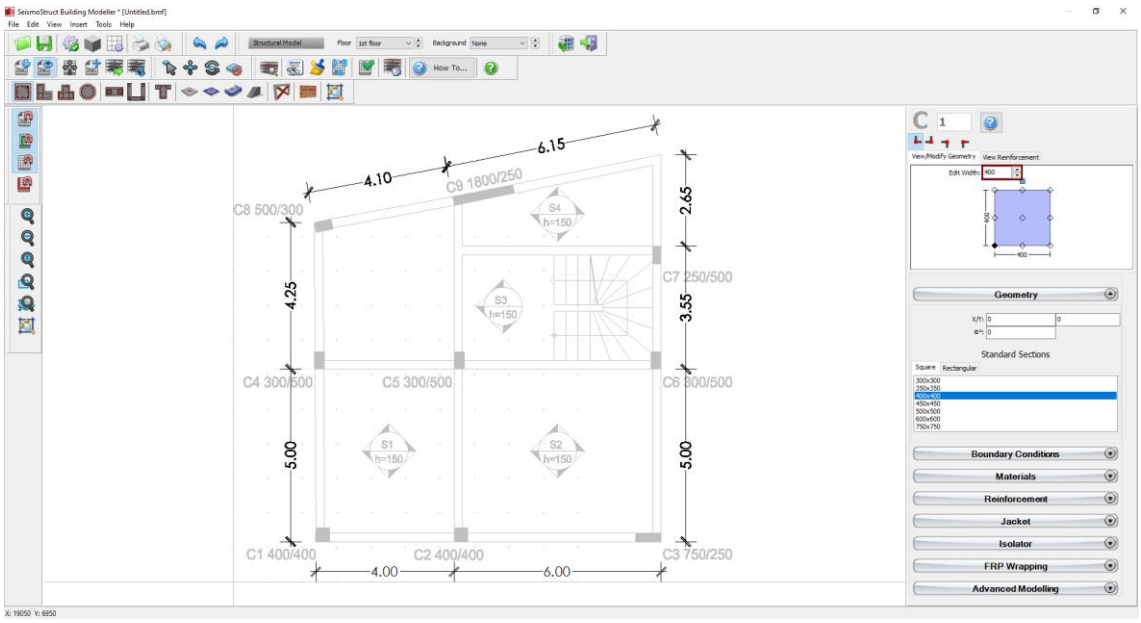
1. By editing the reinforcement pattern;
2. By entering the respective area and sectional coordinates (the latter being defined in the local coordinate system of the section).

Appendix E – Building Modeller Members

In this appendix the available section types are described in details.

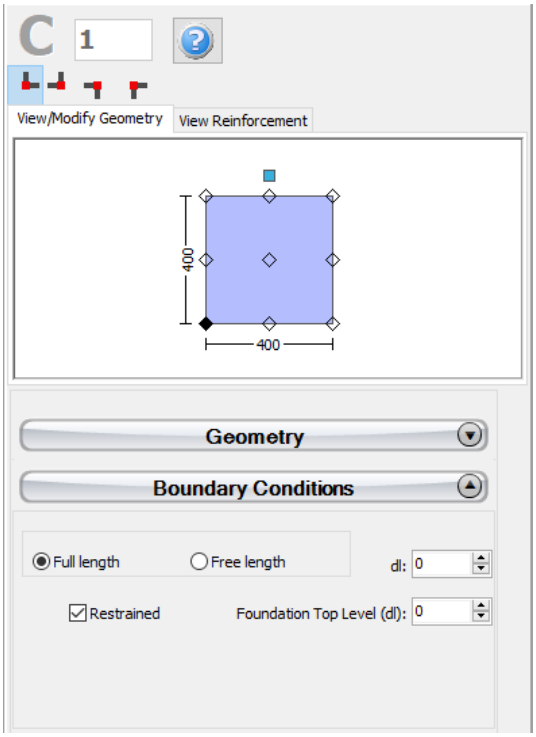
Rectangular Column

Rectangular columns may be inserted from the main menu (*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).



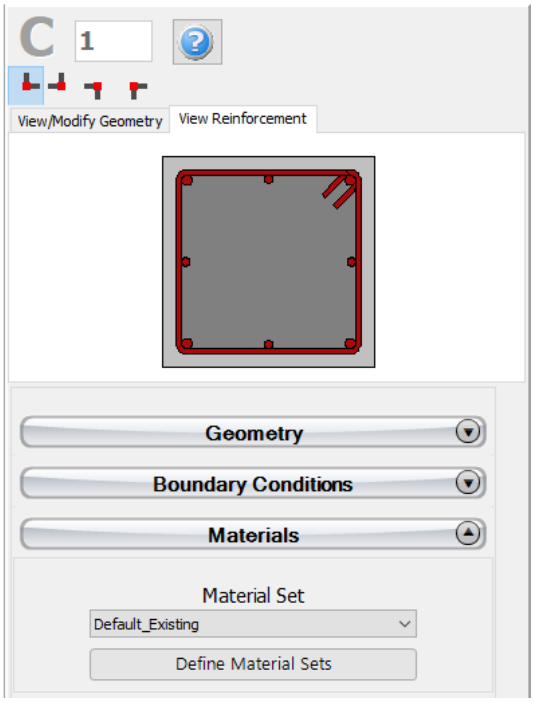
Edit sections dimensions

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.



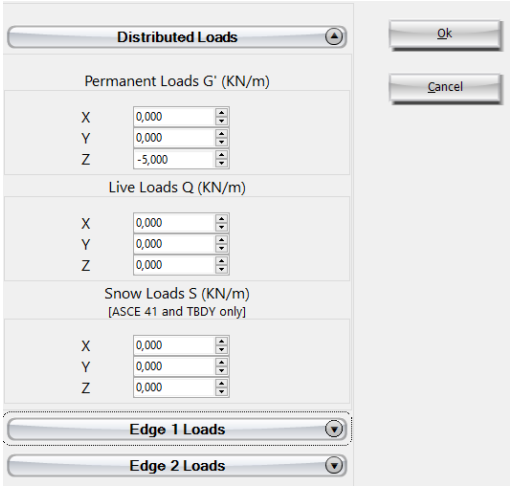
Boundary Conditions

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.



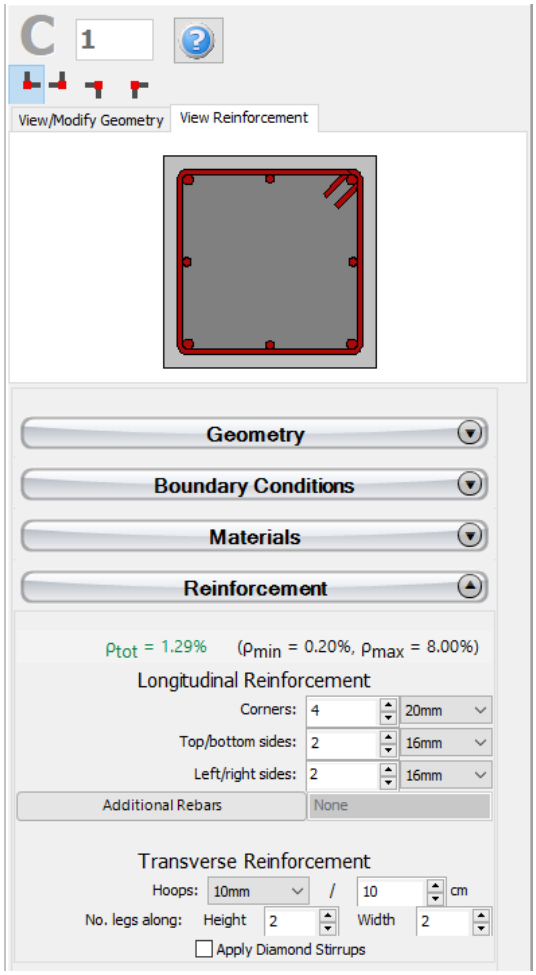
Materials

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.



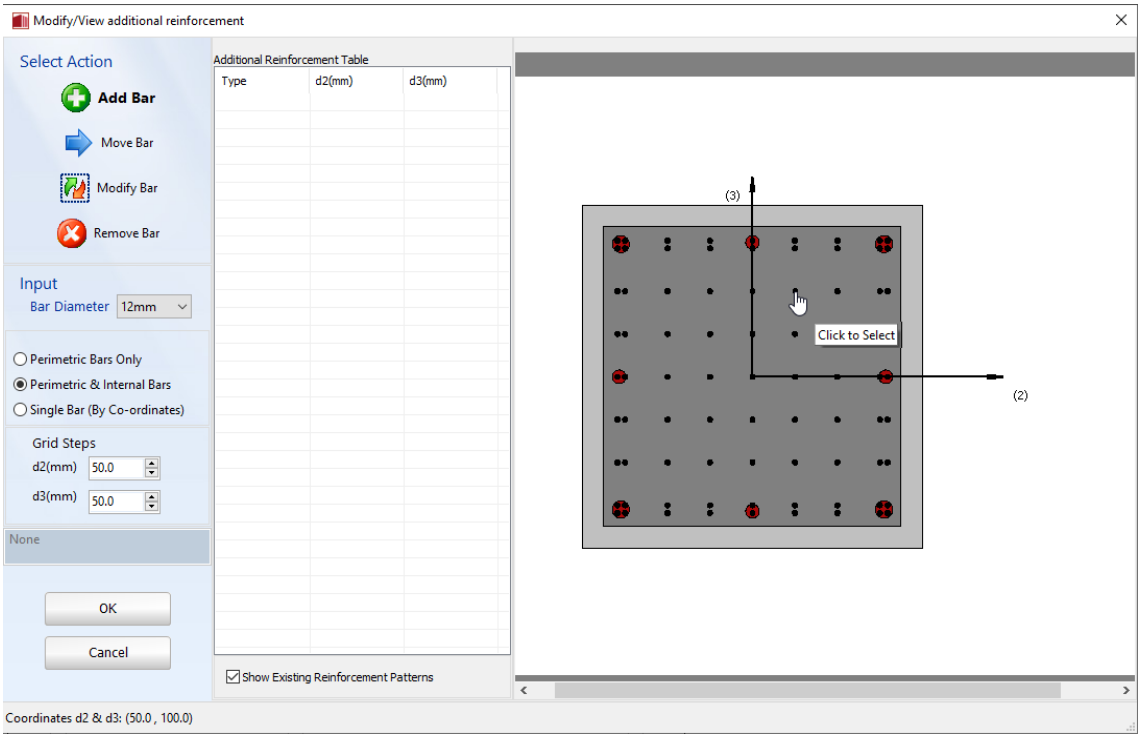
Distributed and Edge Loads window

Further, the longitudinal and transverse reinforcement may be defined by editing the relevant reinforcement pattern controls.



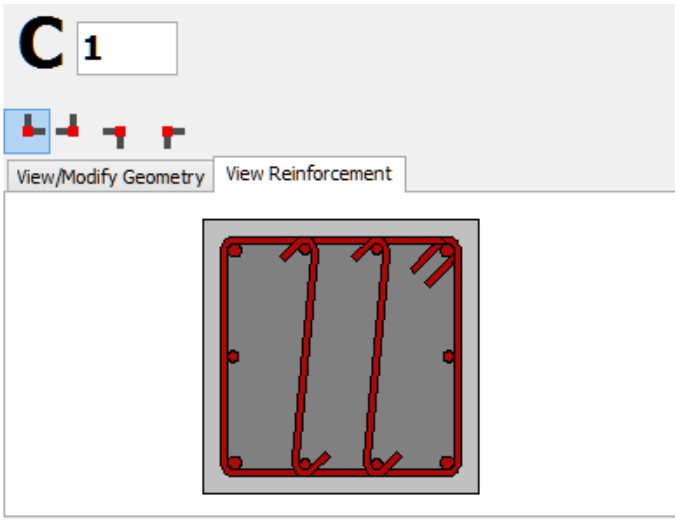
Reinforcement Pattern

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:



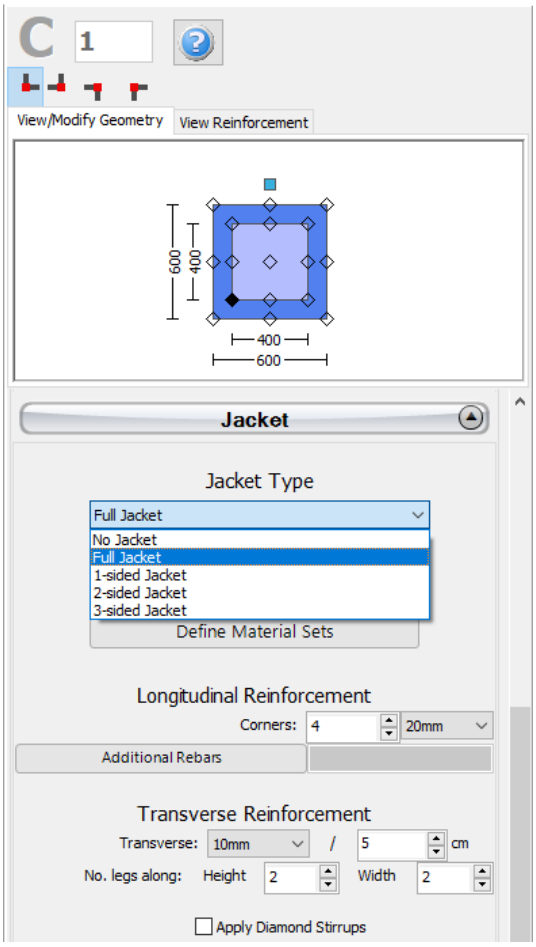
Modify/View additional reinforcement window

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.



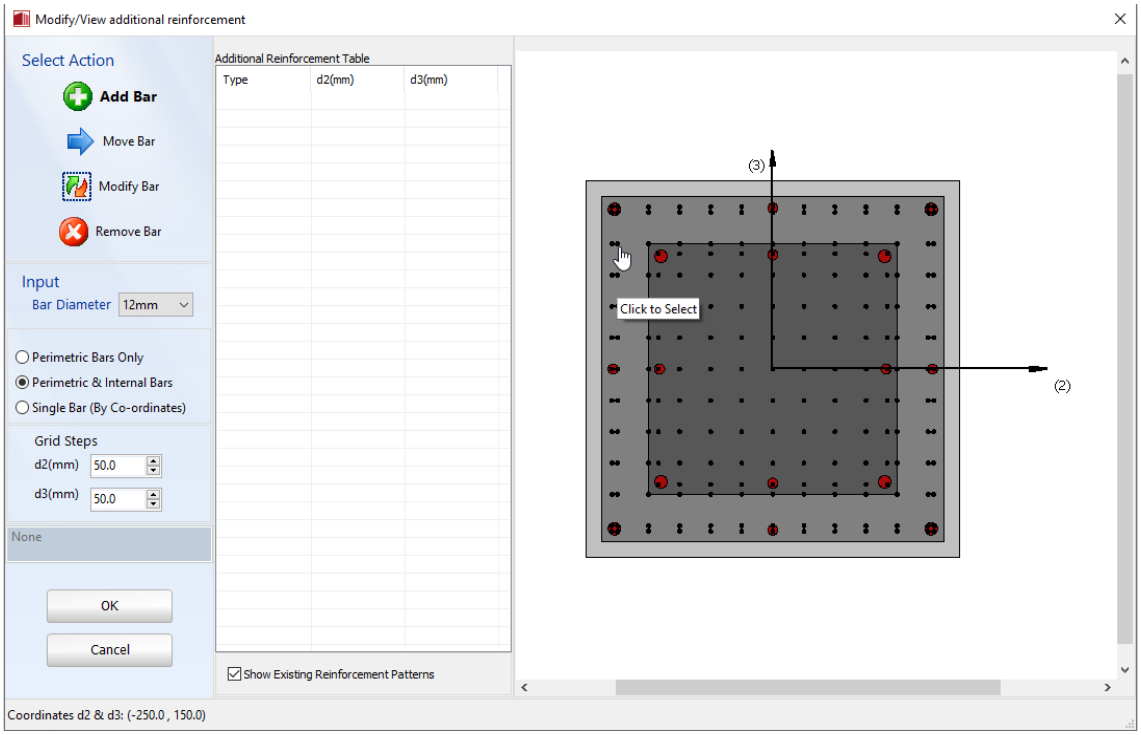
Reinforcement View

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

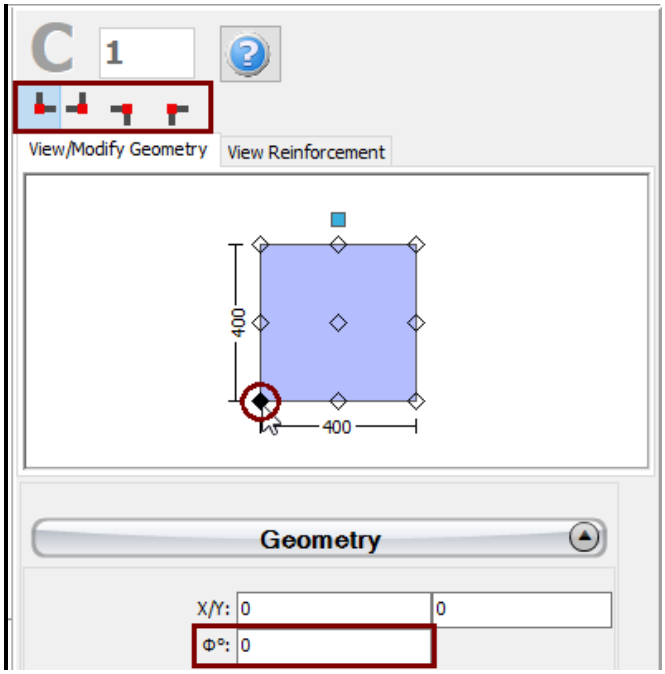
Isolators can also be added at different locations of the column. They are assigned through the Isolator module, where the users may select the geometry (location -bottom, top or intermediate point- and the height of the isolator), its type (elastomeric, lead rubber or curve surface slider) and the isolator parameters: the vertical and horizontal stiffnesses, and the shear yield strength and the strain hardening ratio (for elastomeric and lead rubber isolators, also known as Friction Pendulum System (FPS), which are modelled as isolator1 element type) or the friction coefficient and the radium 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.

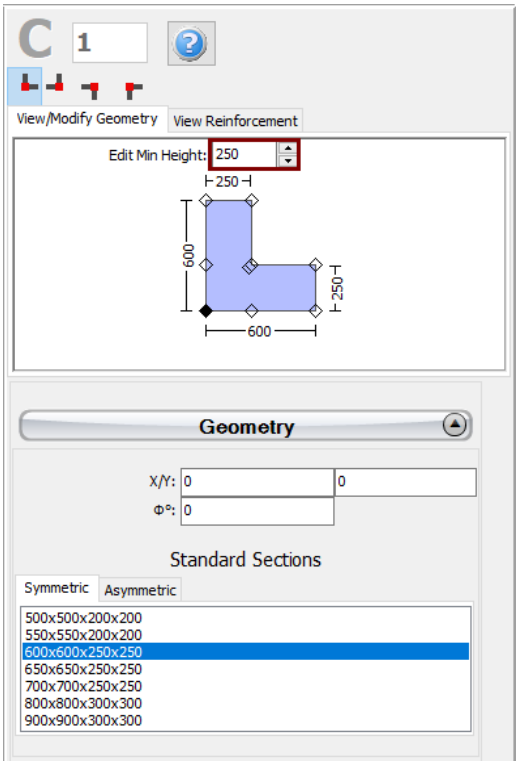
After defining all the section's properties, the new member may be added with a simple click on the Building Modeller Main Window. The location of the section that corresponds to the insertion point (i.e. the mouse click), and rotation of the section on plan view may be selected from the Member Properties window.



Selecting the insertion point and rotate the section's plan view

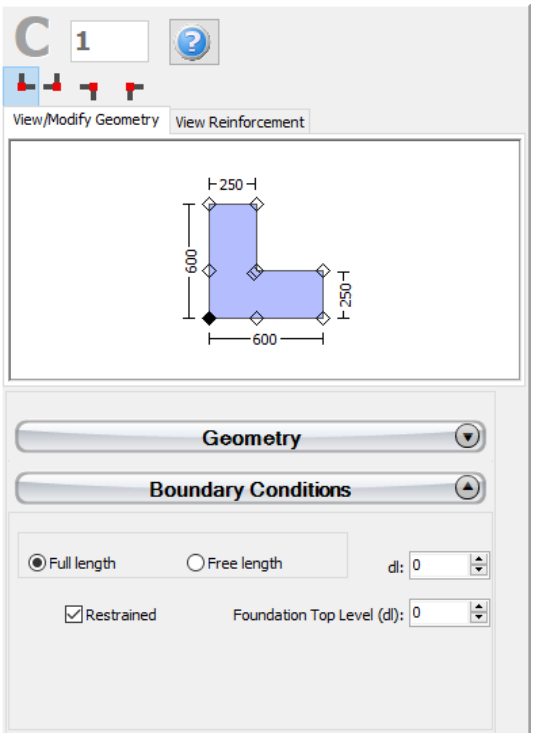
L-Shaped Column

L-Shaped columns can be inserted from the main menu (*Insert > 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).



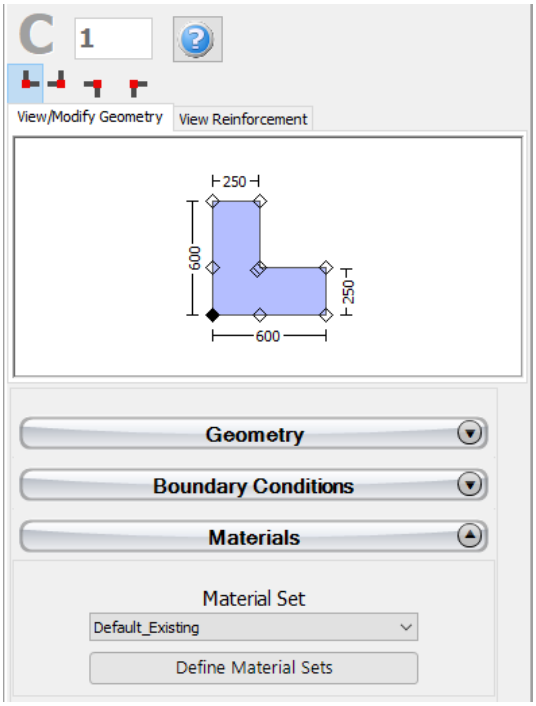
Edit sections dimensions

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.



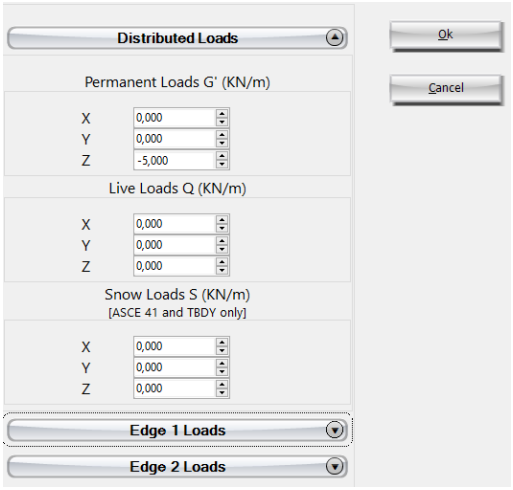
Boundary Conditions

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.



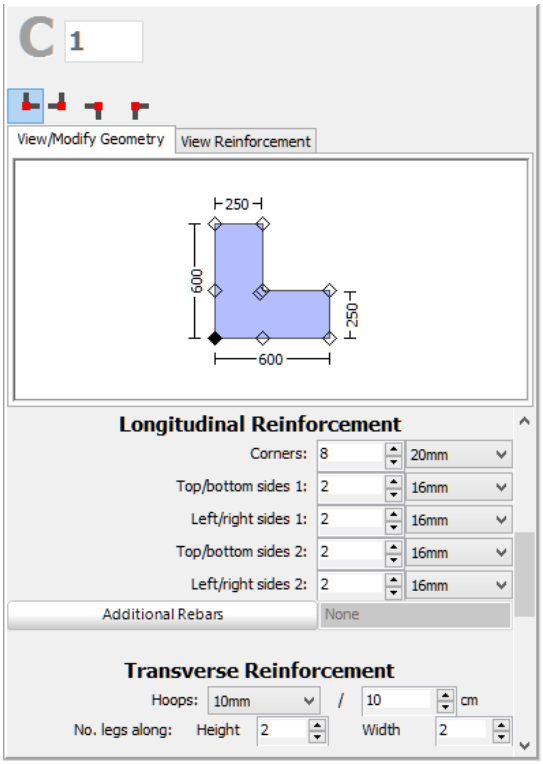
Materials

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.



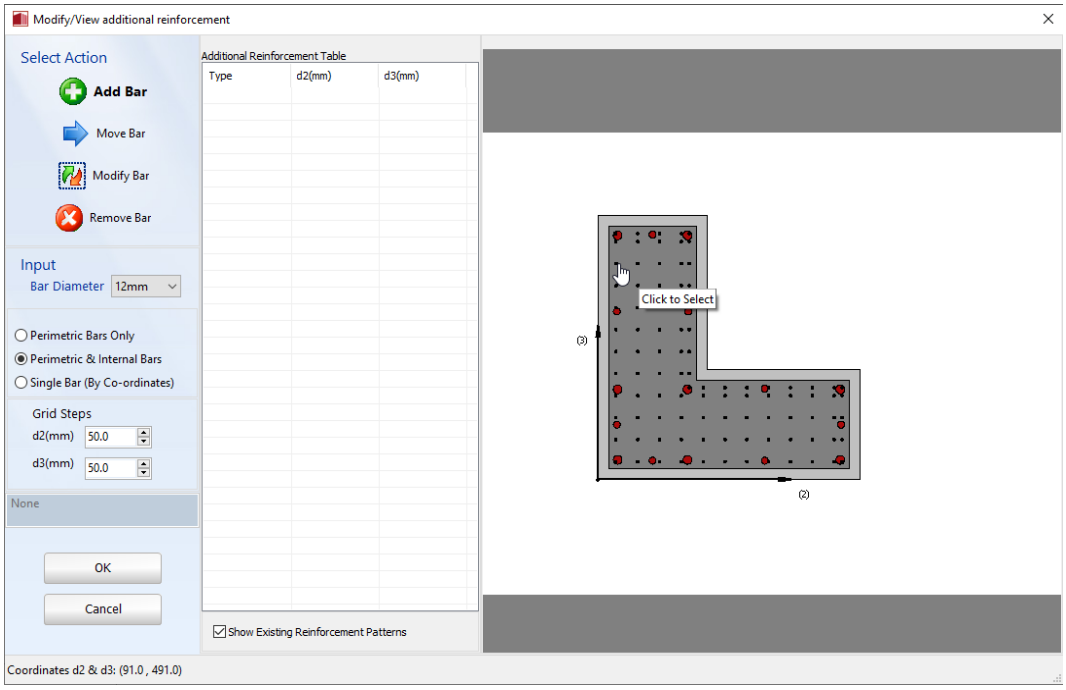
Distributed and Edge Loads window

Further, the longitudinal and transverse reinforcement may be defined by editing the relevant reinforcement pattern controls.



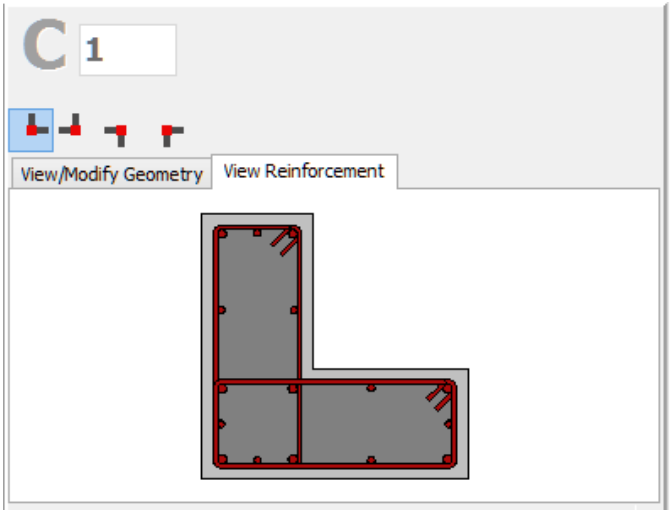
Reinforcement Pattern

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:



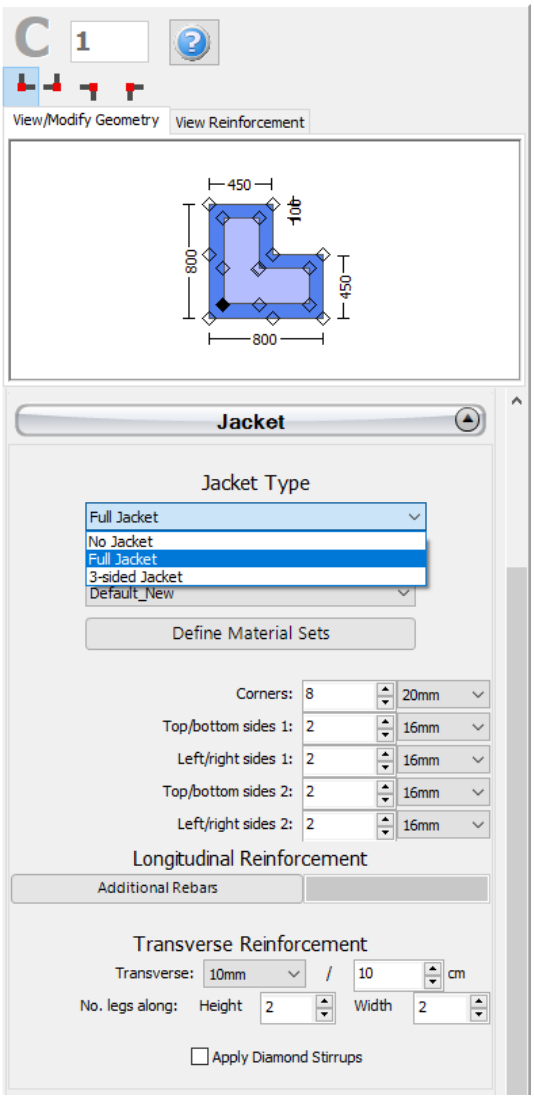
Modify/View additional reinforcement window

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.



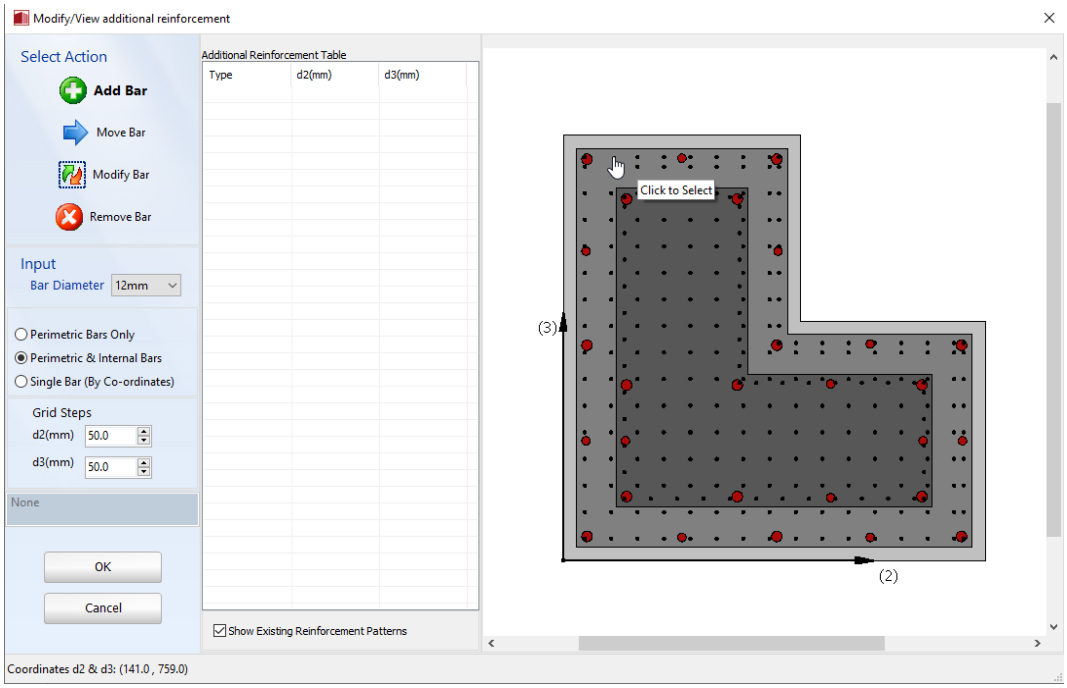
Reinforcement View

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

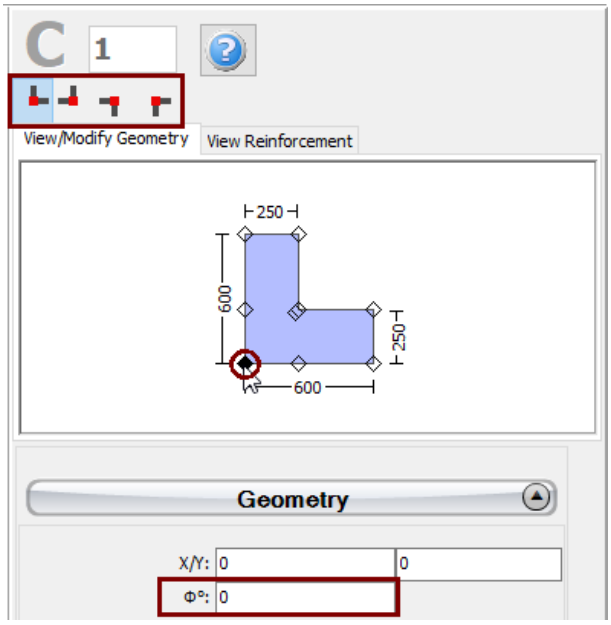
Isolators can also be added at different locations of the column. They are assigned through the Isolator module, where the users may select the geometry (location -bottom, top or intermediate point- and the height of the isolator), its type (elastomeric, lead rubber or curve surface slider) and the isolator parameters: the vertical and horizontal stiffnesses, and the shear yield strength and the strain hardening ratio (for elastomeric and lead rubber isolators, also known as Friction Pendulum System (FPS), which are modelled as isolator1 element type) or the friction coefficient and the radium 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.

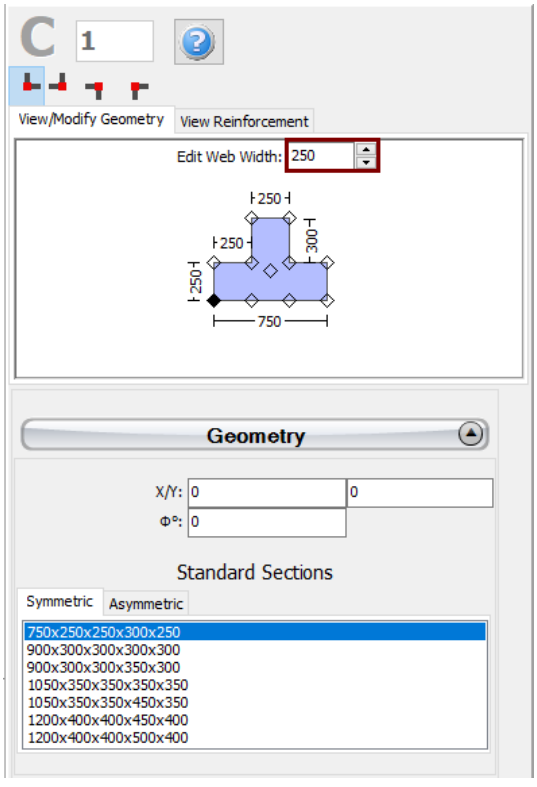
After defining all the section's properties, the new member may be added with a simple click on the Building Modeller Main Window. The location of the section that corresponds to the insertion point (i.e. the mouse click), and rotation of the section on plan view may be selected from the Member Properties window.



Selecting the insertion point and rotate the section's plan view

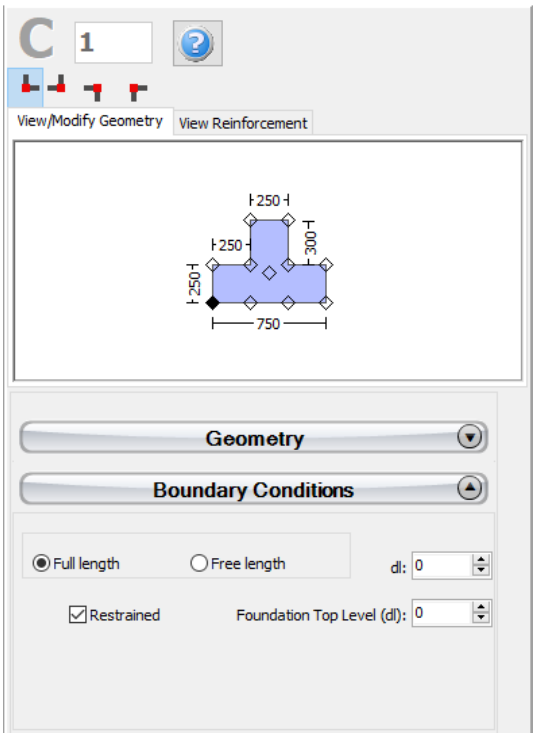
T-Shaped Column

T-Shaped columns may be inserted from the main menu (*Insert > 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).



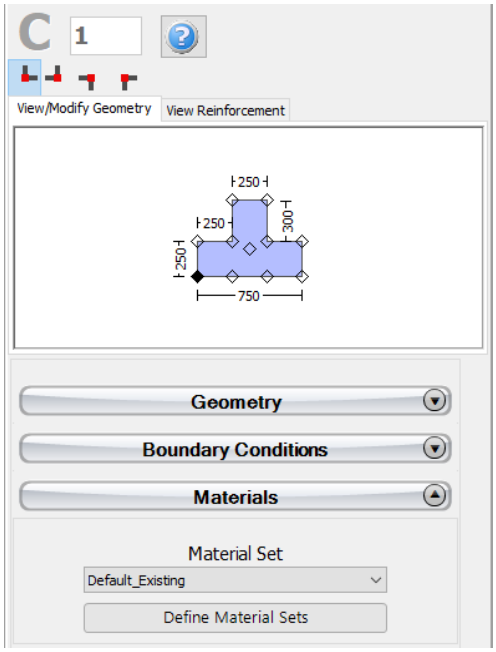
Edit sections dimensions

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.



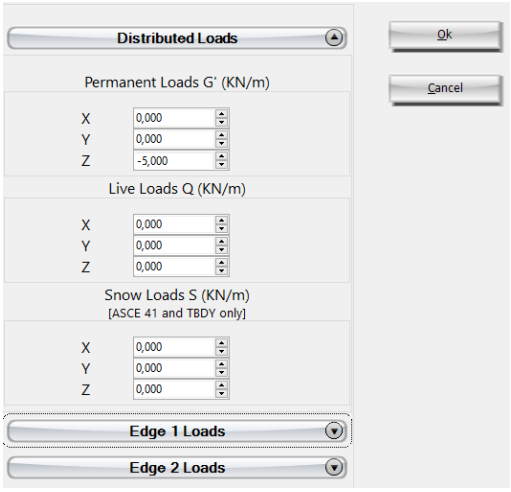
Boundary Conditions

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.



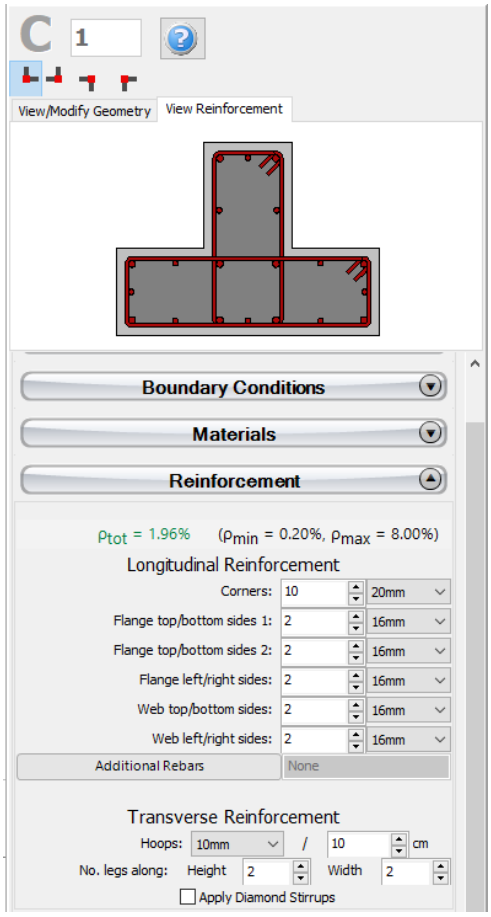
Materials

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.



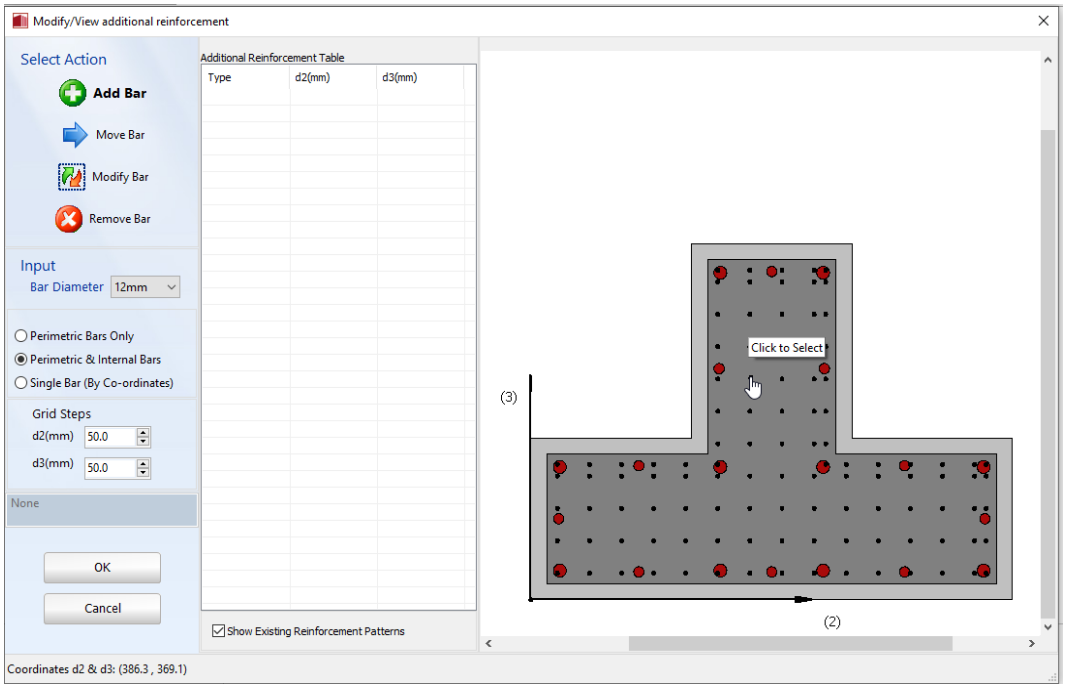
Distributed and Edge Loads window

Further, the longitudinal and transverse reinforcement may be defined by editing the relevant reinforcement pattern controls.



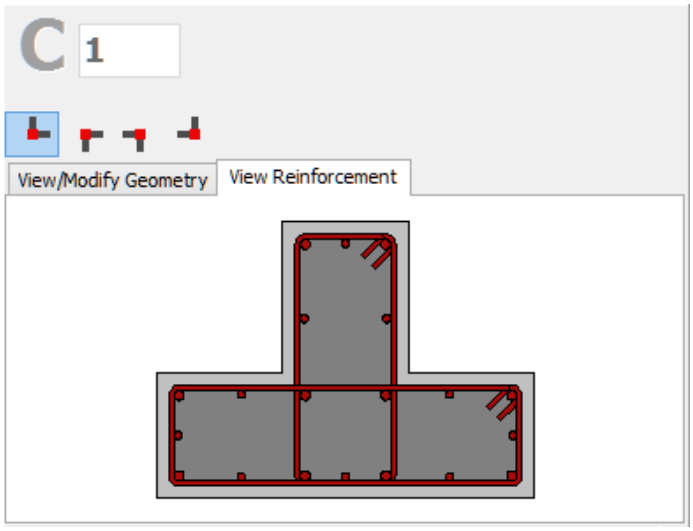
Reinforcement Pattern

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:



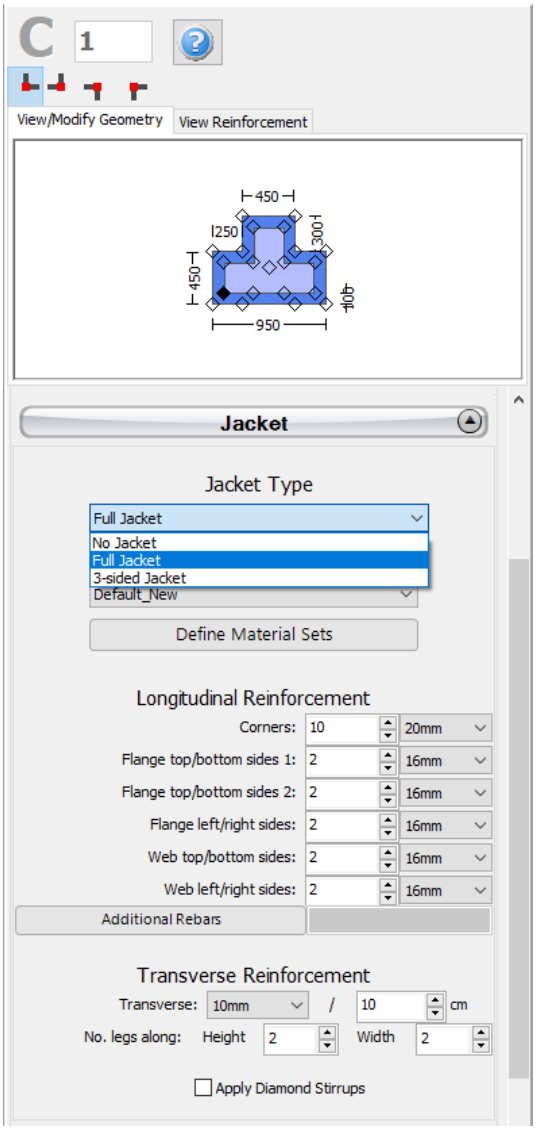
Modify/View additional reinforcement window

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.



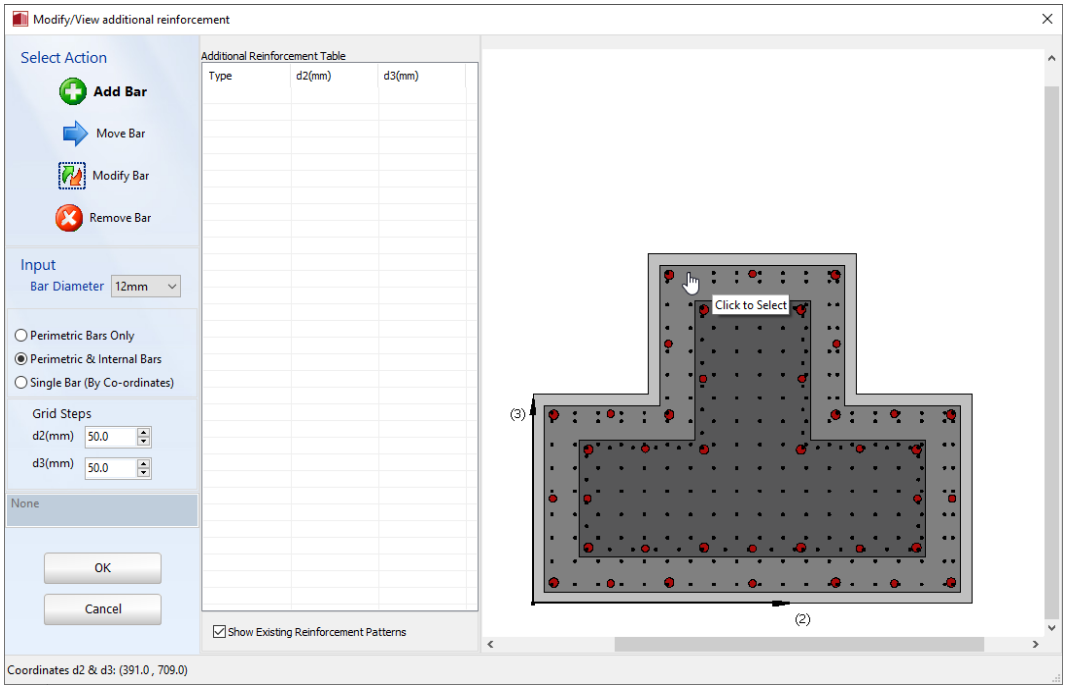
Reinforcement View

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

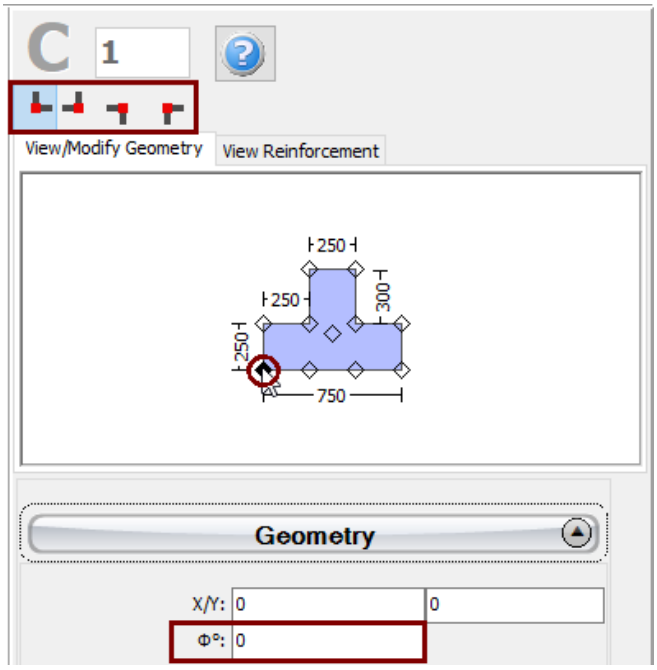
Isolators can also be added at different locations of the column. They are assigned through the Isolator module, where the users may select the geometry (location -bottom, top or intermediate point- and the height of the isolator), its type (elastomeric, lead rubber or curve surface slider) and the isolator parameters: the vertical and horizontal stiffnesses, and the shear yield strength and the strain hardening ratio (for elastomeric and lead rubber isolators, also known as Friction Pendulum System (FPS), which are modelled as isolator1 element type) or the friction coefficient and the radium 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.

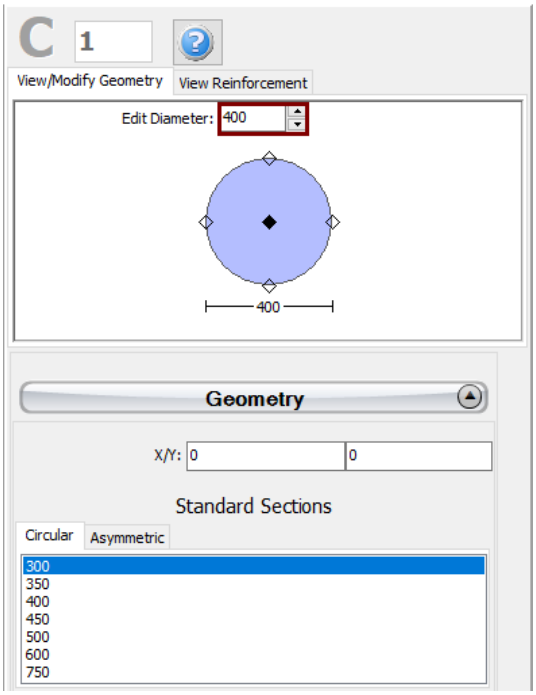
After defining all the section's properties, the new member may be added with a simple click on the Building Modeller Main Window. The location of the section that corresponds to the insertion point (i.e. the mouse click), and rotation of the section on plan view may be selected from the Member Properties window.



Selecting the insertion point and rotate the section's plan view

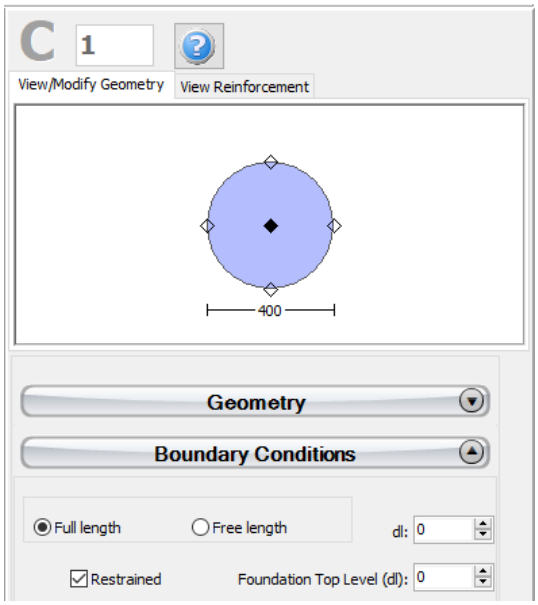
Circular Column

Circular columns may be inserted from the main menu (*Insert* > *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.



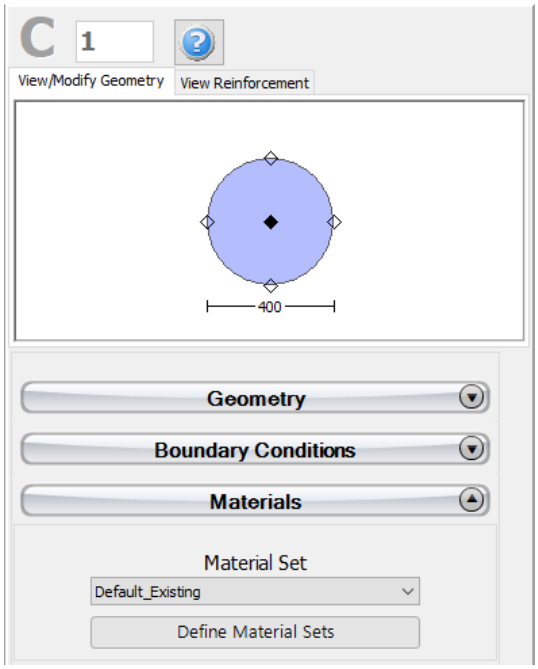
Edit sections dimensions

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.



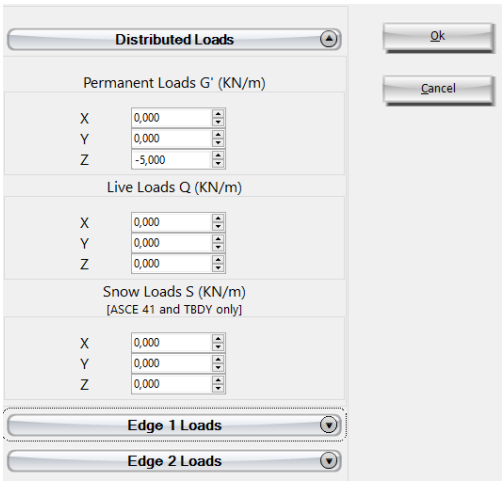
Boundary Conditions

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.



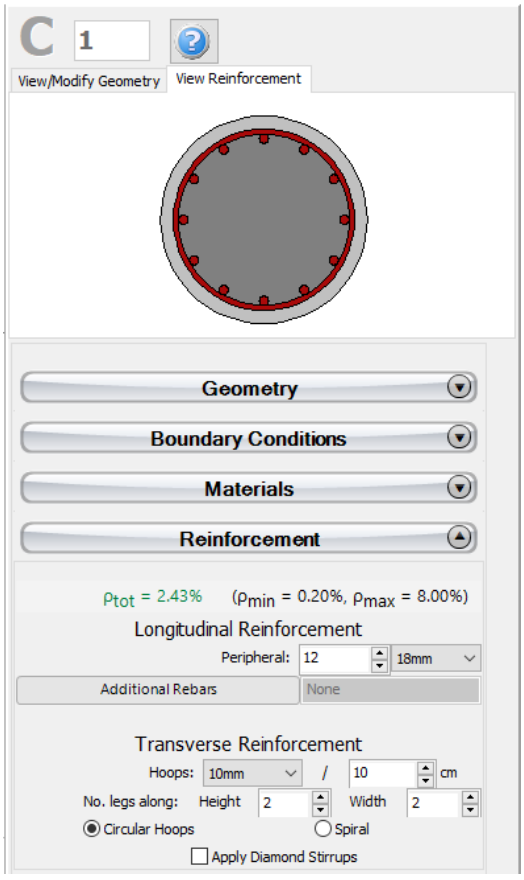
Materials

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.



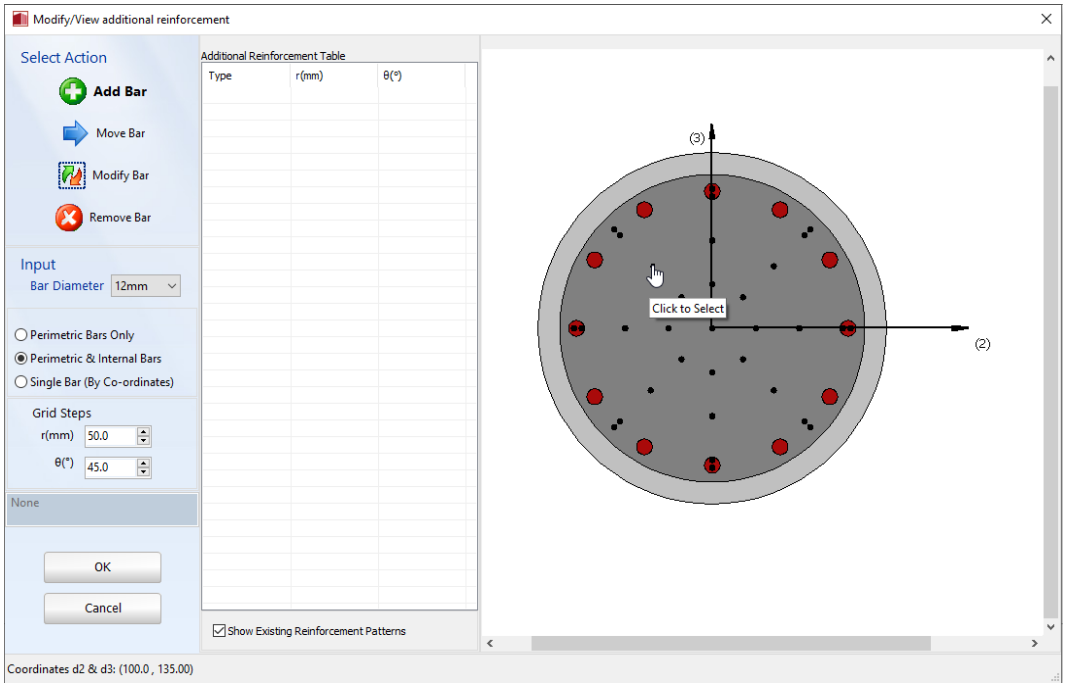
Distributed and Edge Loads window

Further, the longitudinal and transverse reinforcement may be defined by editing the relevant reinforcement pattern controls.



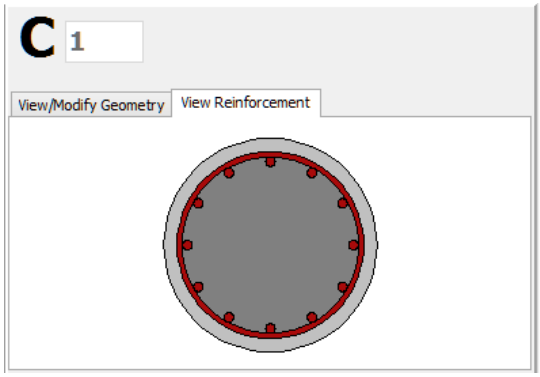
Reinforcement Pattern

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:



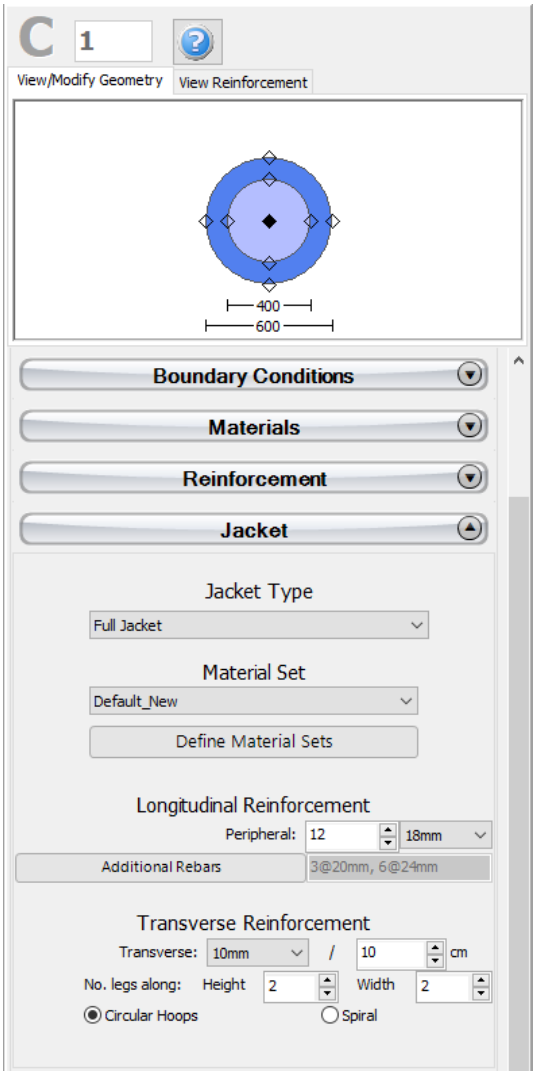
Modify/View additional reinforcement window

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.



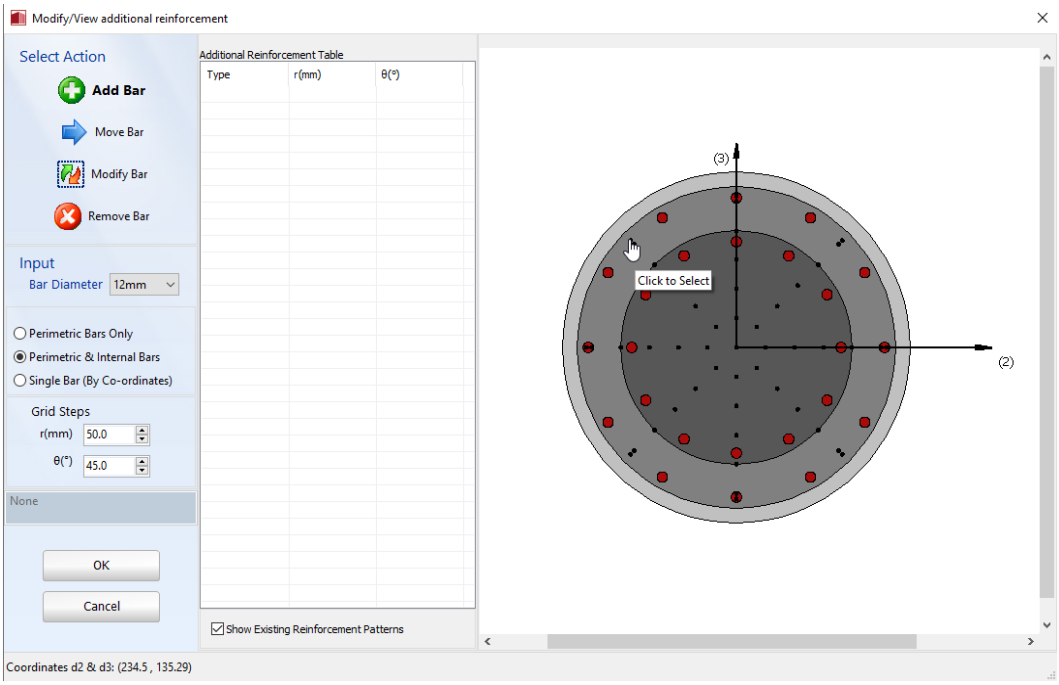
Reinforcement View

Jackets 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

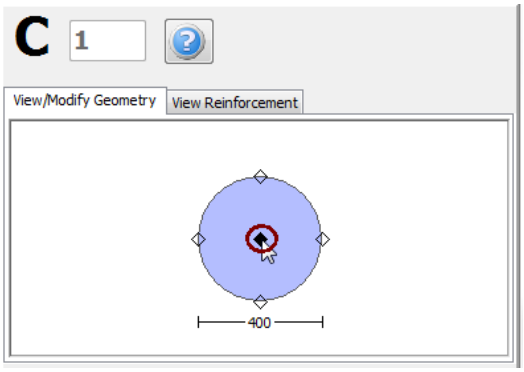
Isolators can also be added at different locations of the column. They are assigned through the Isolator module, where the users may select the geometry (location -bottom, top or intermediate point- and the height of the isolator), its type (elastomeric, lead rubber or curve surface slider) and the isolator parameters: the vertical and horizontal stiffnesses, and the shear yield strength and the strain hardening ratio (for elastomeric and lead rubber isolators, also known as Friction Pendulum System (FPS), which are modelled as isolator1 element type) or the friction coefficient and the radium 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.

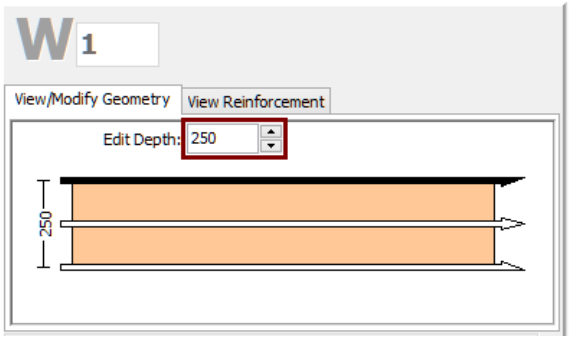
After defining all the section's properties, the new member may be added with a simple click on the Building Modeller Main Window. The location of the section that corresponds to the insertion point (i.e. the mouse click), and rotation of the section on plan view may be selected from the Member Properties window.



Selecting the insertion point

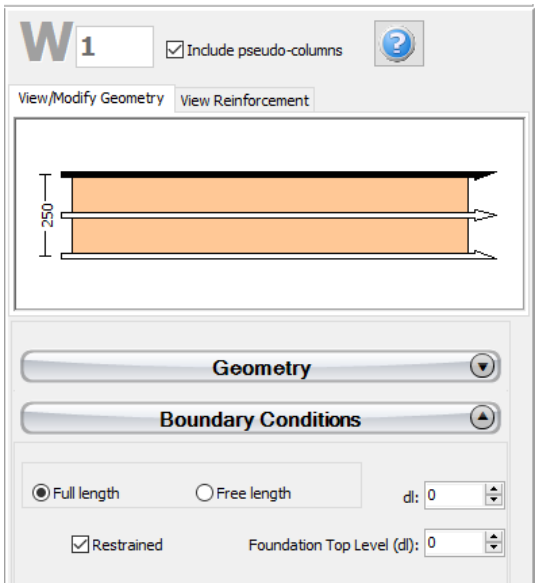
Wall

Walls may be added from the main menu (*Insert* > *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.



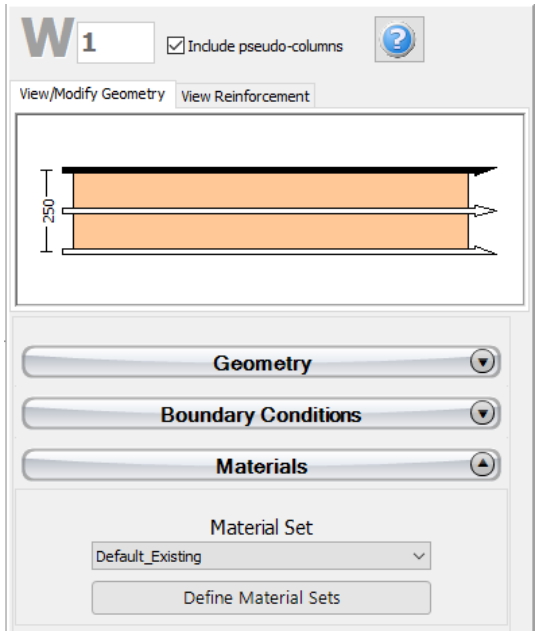
Edit sections dimensions

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.



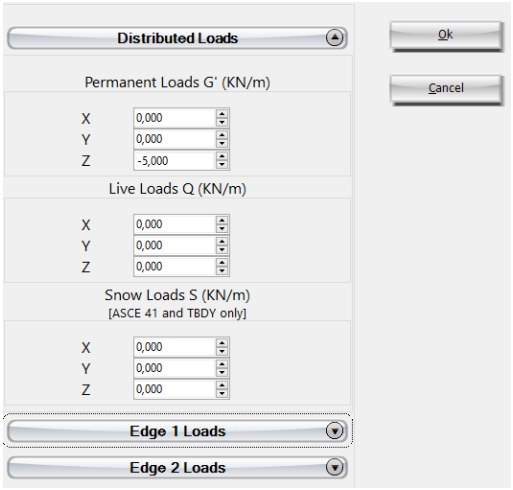
Boundary Conditions

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.



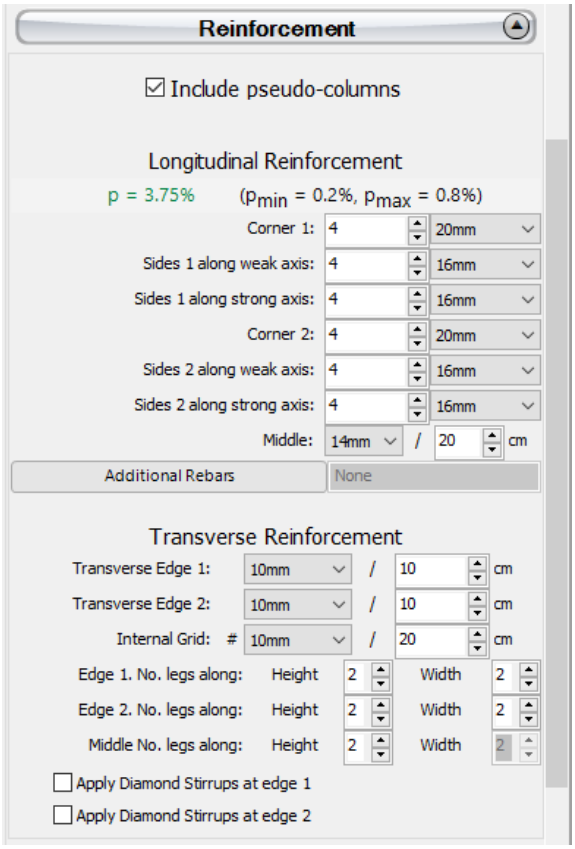
Materials

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.



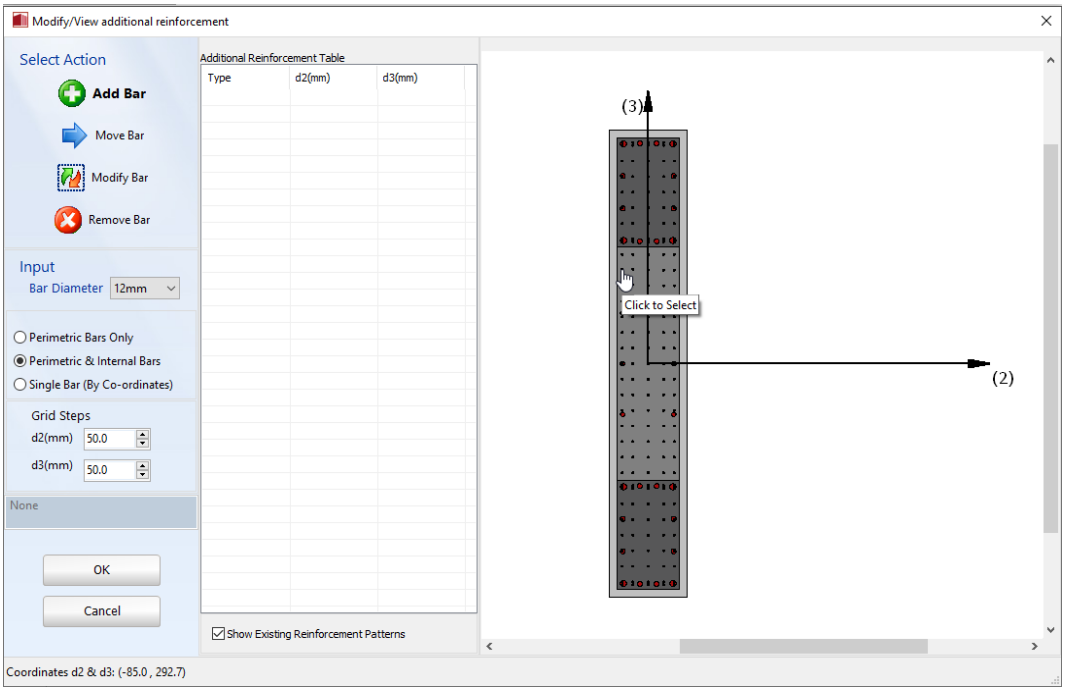
Distributed and Edge Loads window

Further, the option to include pseudo-columns is available in the reinforcement area and the longitudinal and transverse reinforcement may be defined by editing the relevant reinforcement pattern controls.



Reinforcement Pattern

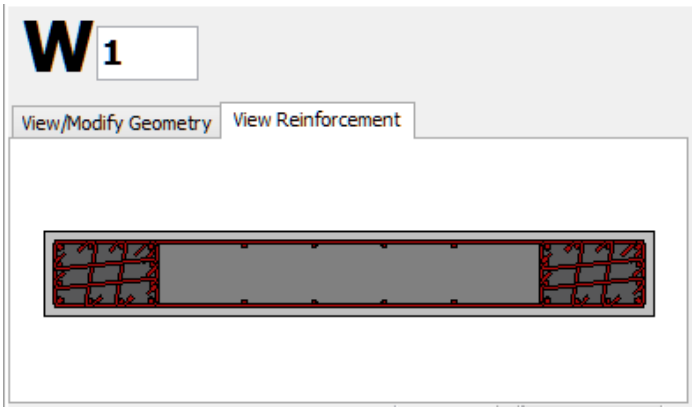
Adding single longitudinal reinforcement bars may also be carried out through the corresponding Additional Rebars module, where additional reinforcement may be introduced graphically as shown in the following figure:



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.



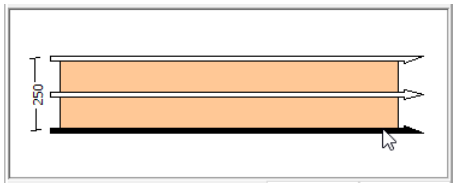
Reinforcement View

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 radium of the pendulum (for curve surface sliders, also known as Friction Pendulum System (FPS), which are modelled as isolator2 element type).

Further, FRP wraps may be assigned to wall 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 columns 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).



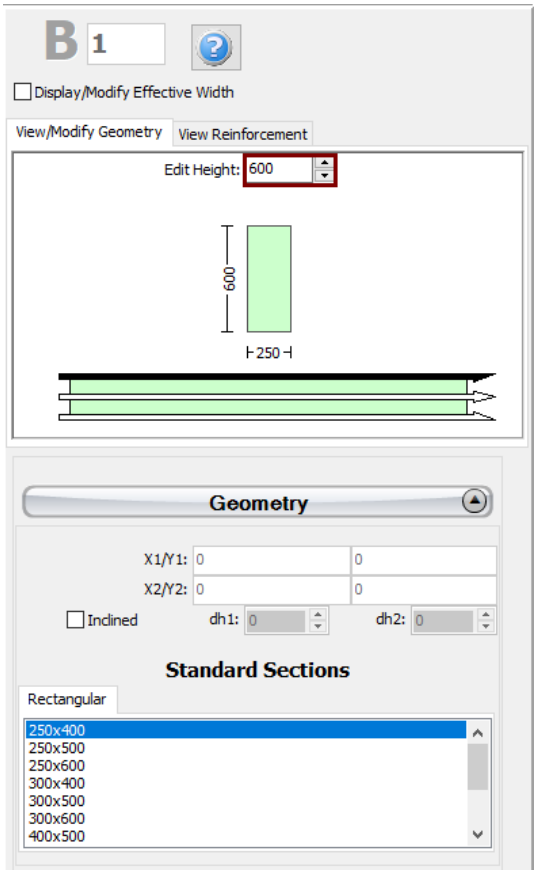
Select the inserting line



Insertion of wall

Beam

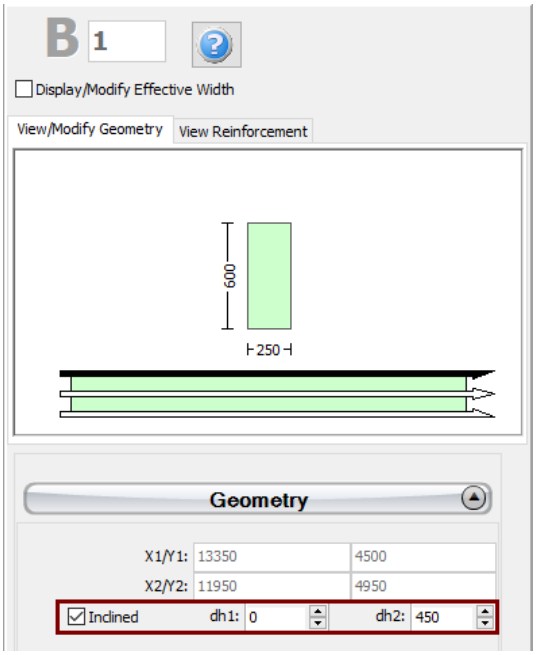
Beams may be inserted from the main menu (*Insert > Beam...*) or through the corresponding toolbar button . On the Properties Window that appears users can adapt the section's dimensions either in the View/Modify Geometry window or by selecting one section from the predefined standard sections.



Edit sections dimensions

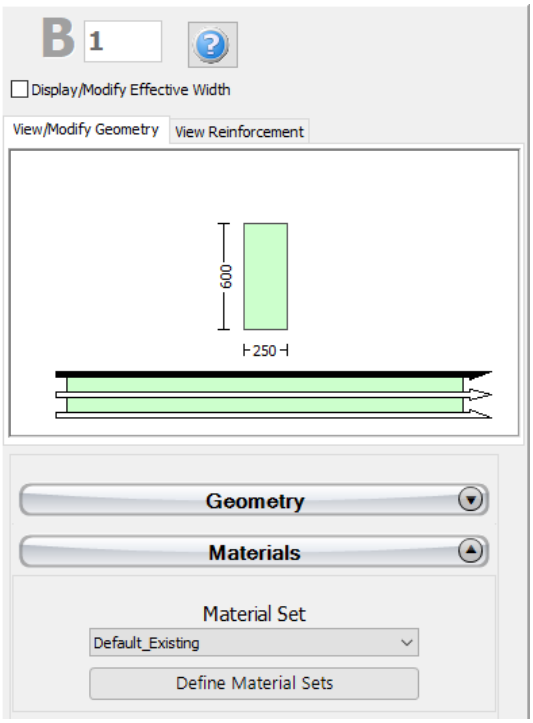
Inclined beams may be efficiently modelled by specifying the elevation differences of the two beam ends relatively to the storey height. The height of the supporting columns is then automatically adapted.

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.



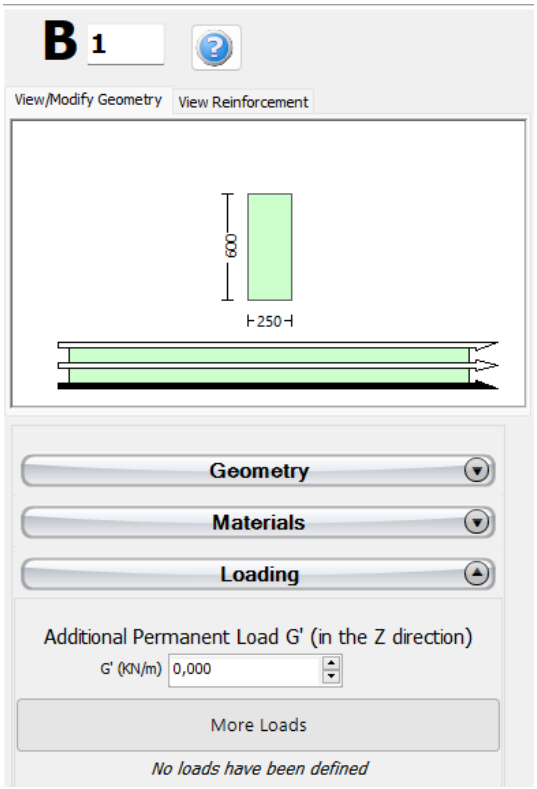
Inclined beam

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.



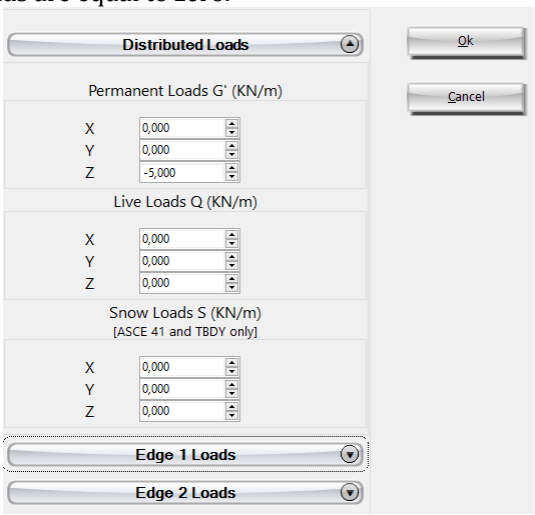
Materials

In the beams 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).



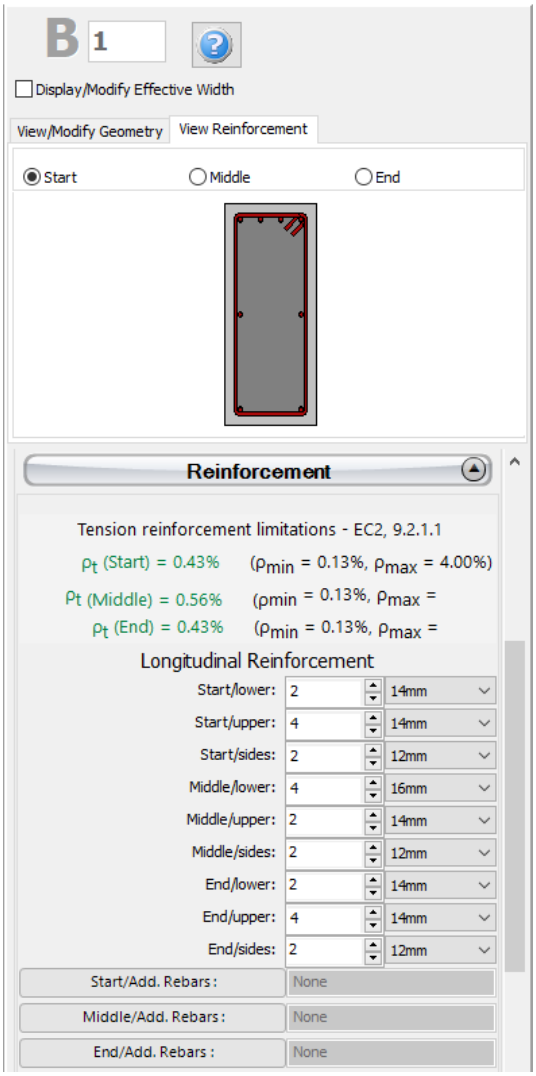
Loading

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.



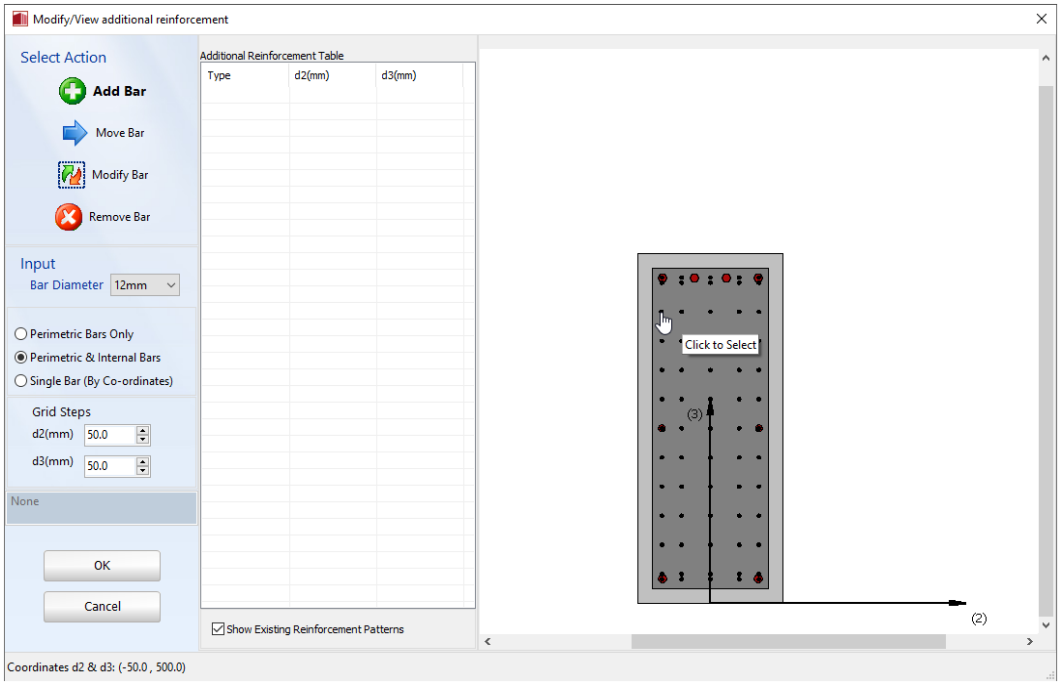
Distributed and Edge Loads window

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.



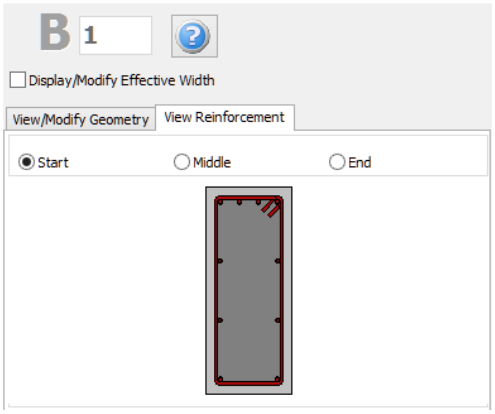
Reinforcement Pattern

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:

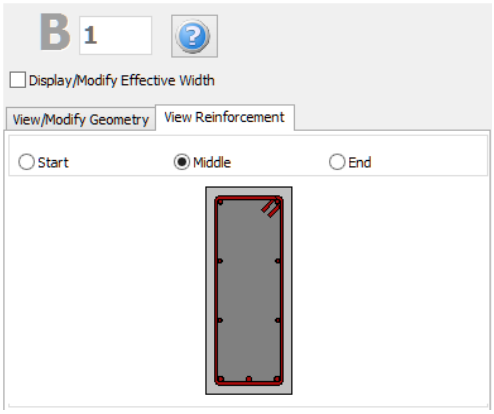


Modify/View additional reinforcement window

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.

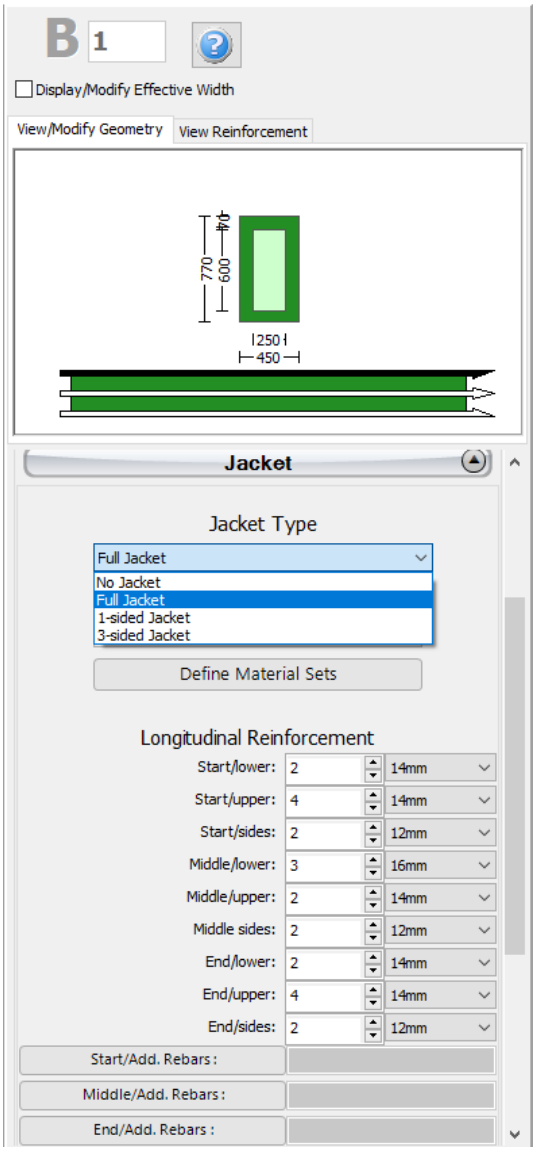


Beam's start section reinforcement



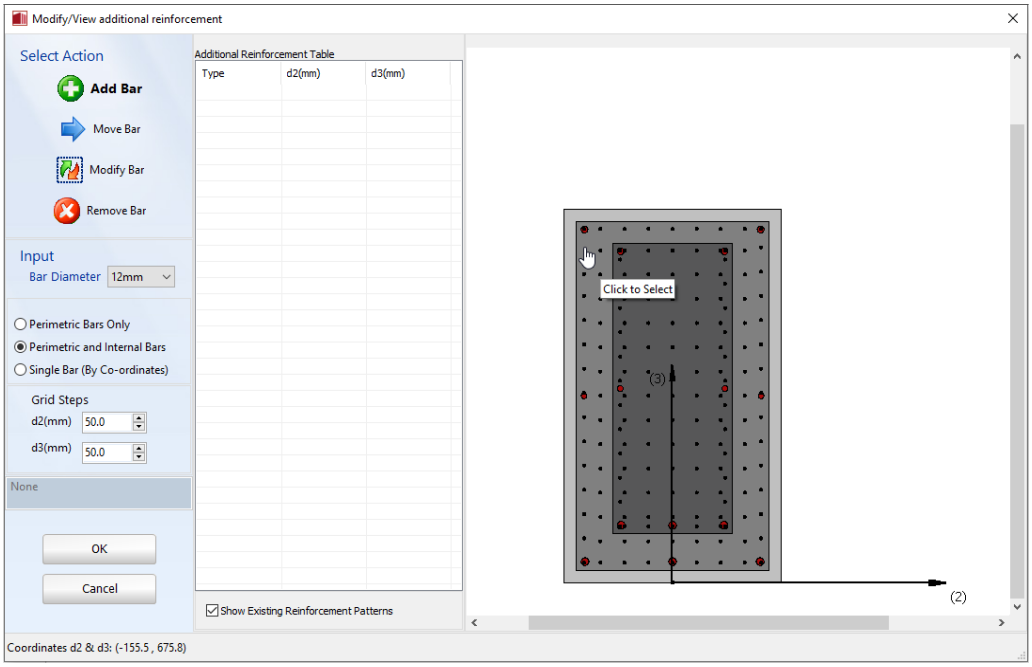
Beam's middle section reinforcement

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:



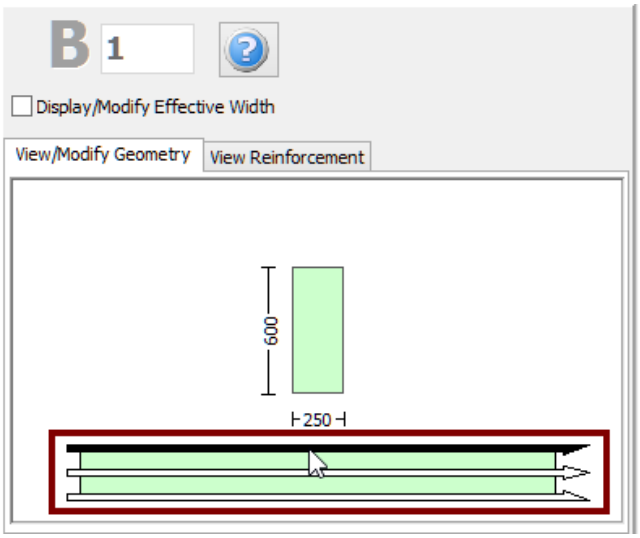
Modify/View additional reinforcement window

Further, FRP wraps may be assigned to beam 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.

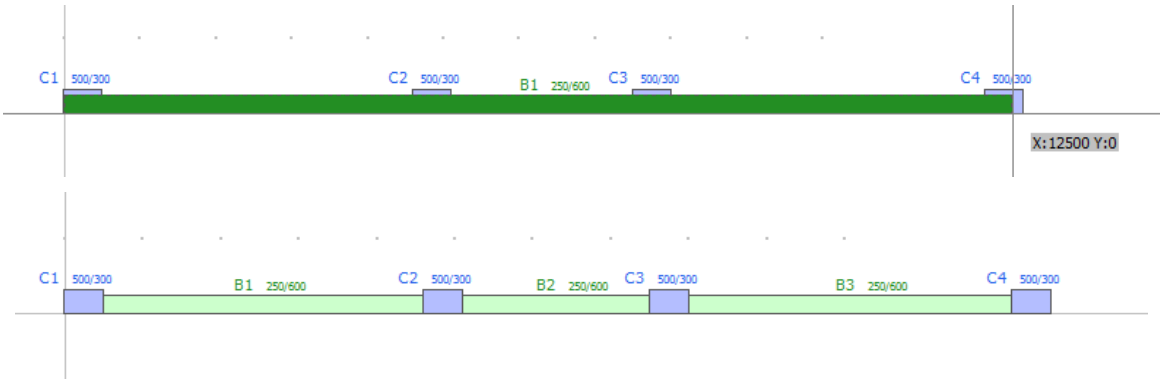
NOTE 2: When the section is jacketed, in the *Advanced Member Properties* module users should take decisions on the parameters, so as to account for the entire section, i.e. for both the existing and the new parts.

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).



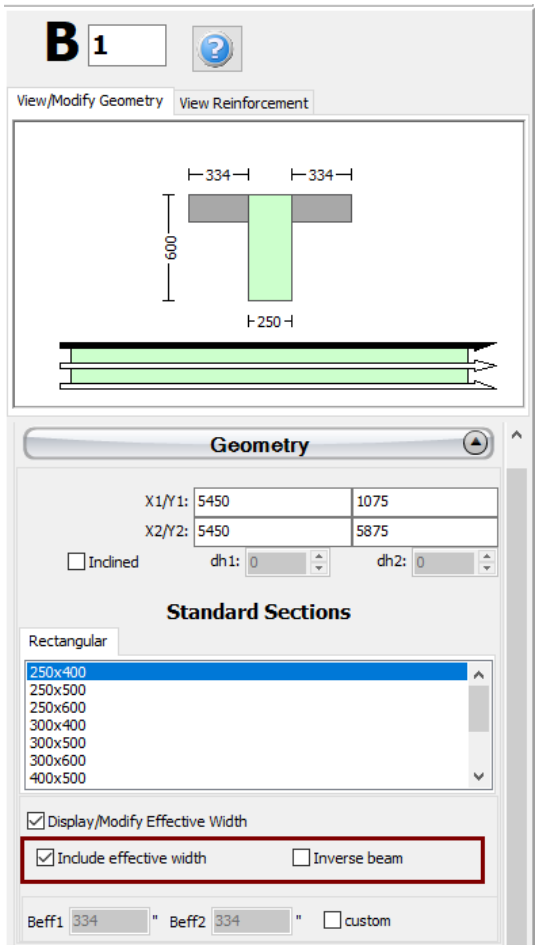
Selecting the insertion line

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.



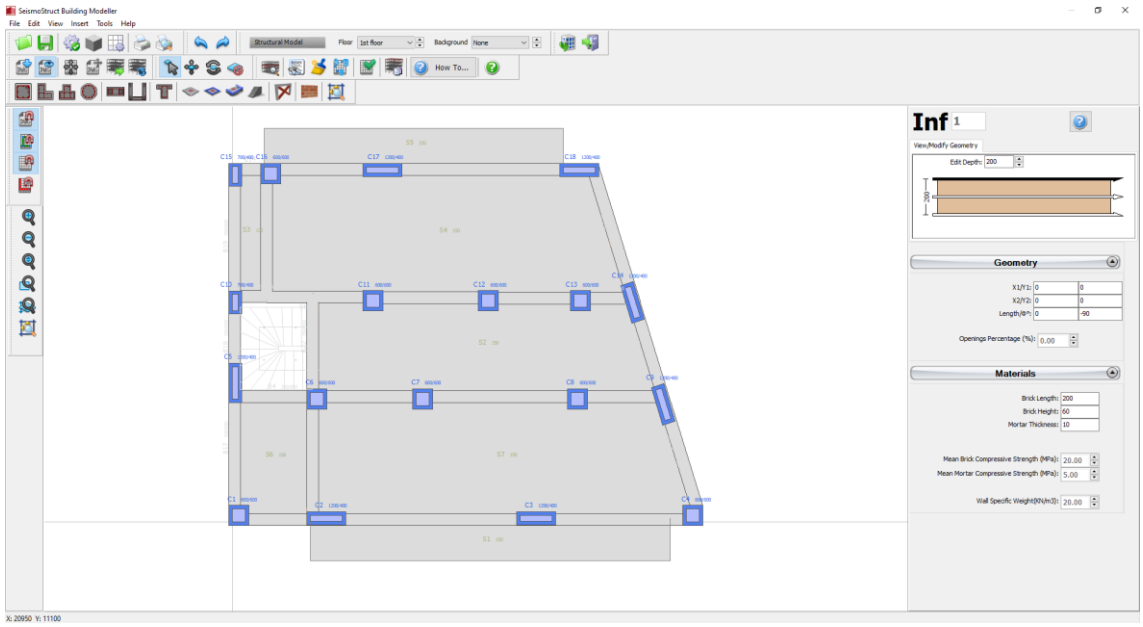
Beam's Properties Window (after the definition of slabs)

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.

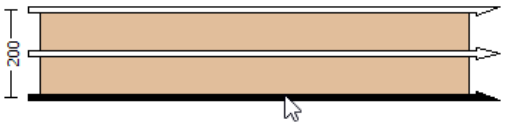
(i) The brick dimensions, length and height
(ii) The mortar thickness
(iii) The brick and mortar compressive strengths
(iv) The percentage of the openings on the wall and
(v) The wall specific weight.

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.

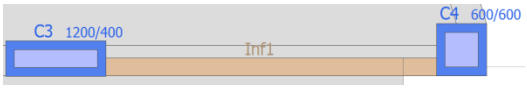


Edit infill parameters

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

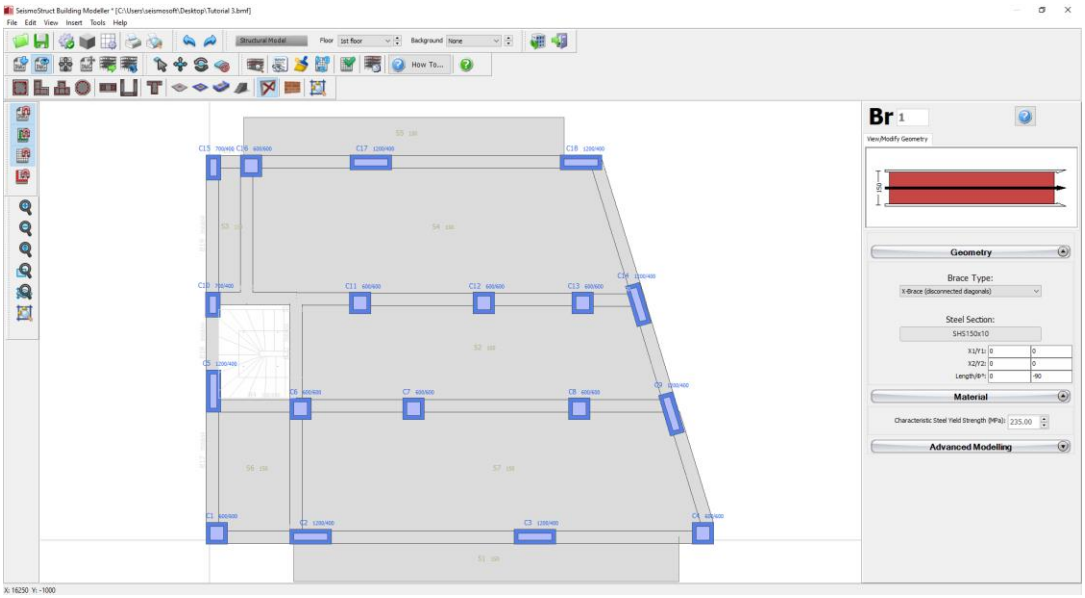


Insertion of the infill

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.



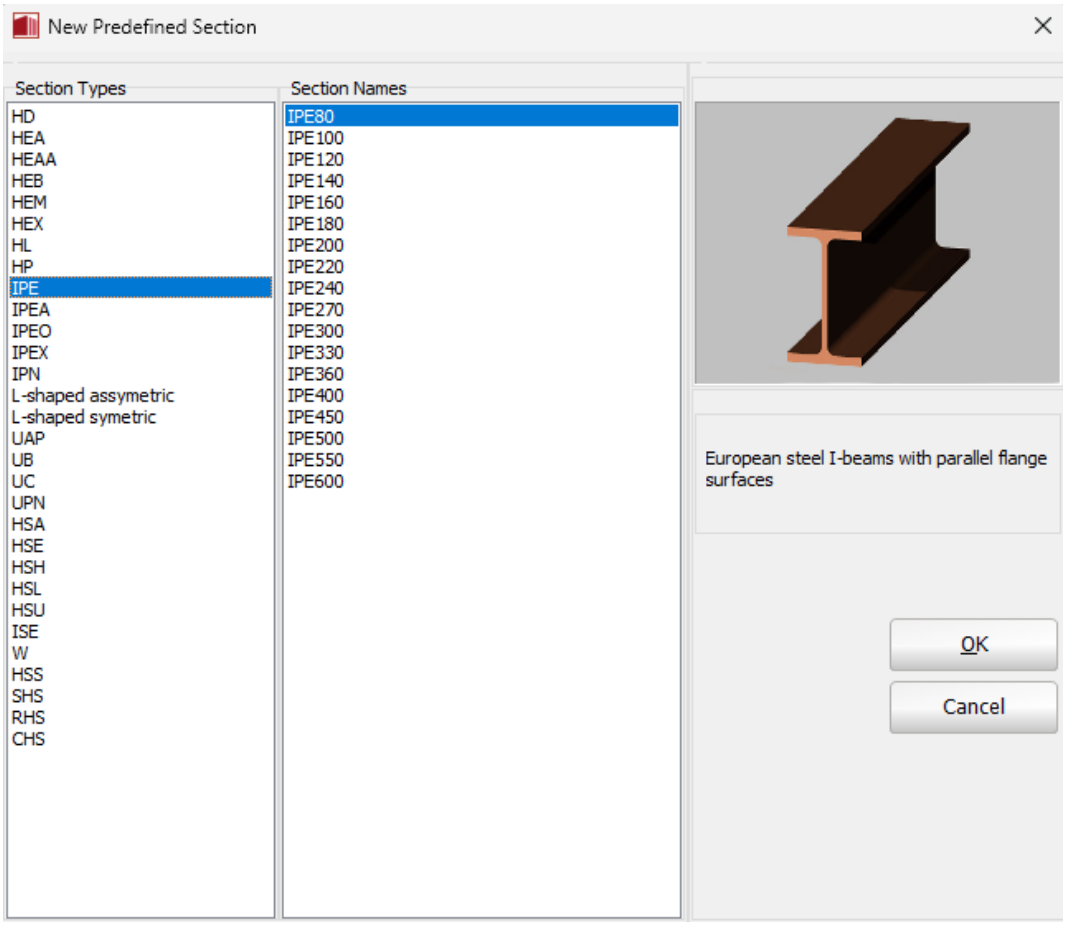
Select the inserting line



Insertion of the Steel Brace

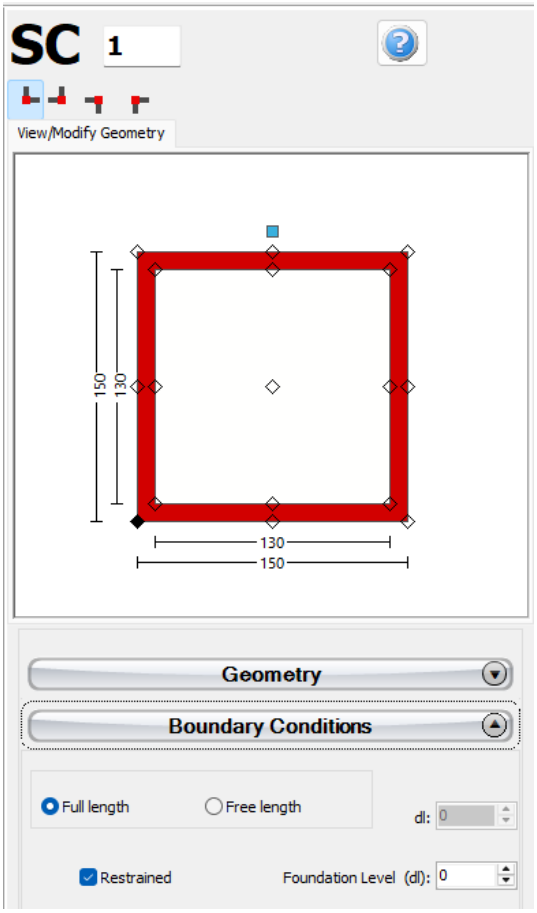
Steel Column

Steel columns may be inserted from the main menu (*Insert > Steel Column*) or through the corresponding toolbar button . On the Properties Window that appears users select one section from the predefined standard sections.



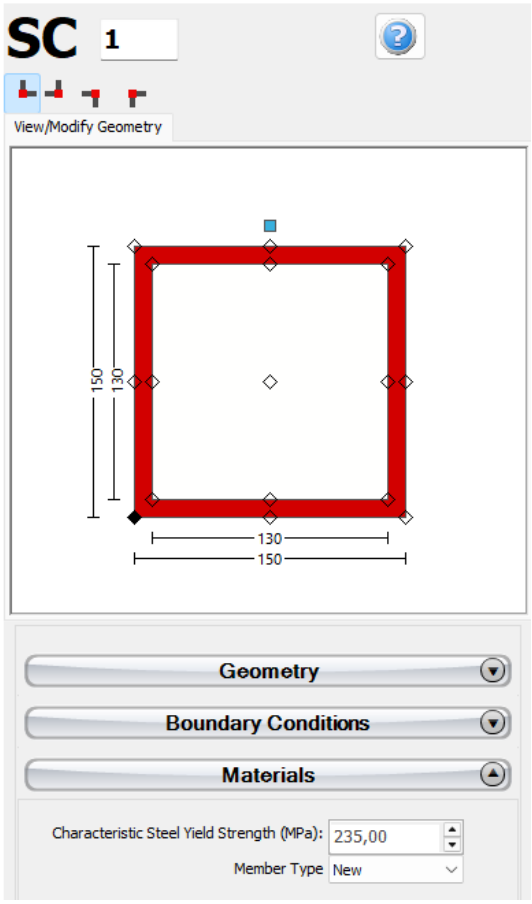
Predefined sections

It is possible to define a steel 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 steel column may be adapted, thus providing the possibility to the user to define different foundation levels.



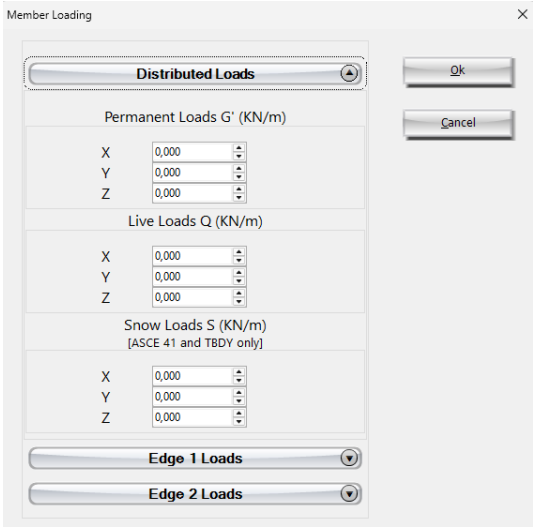
Boundary Conditions

The steel's properties can be defined within the member's Properties Window. The characteristic steel yield strength and the type of the members, i.e. existing or new member, should be defined.



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.

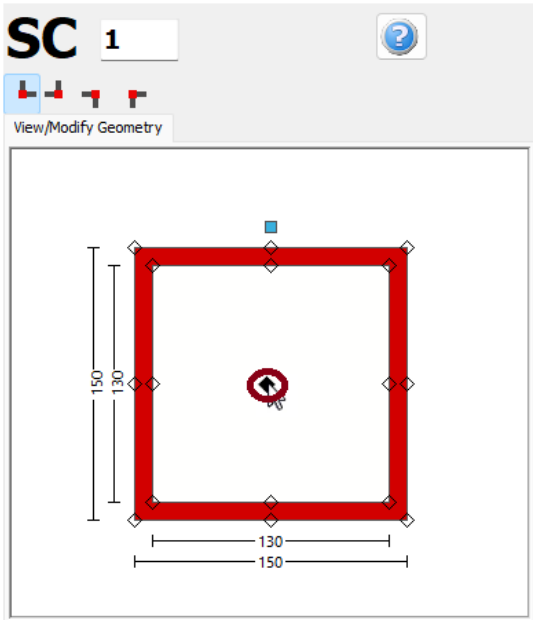


Distributed and Edge Loads window

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, also known as Friction Pendulum System (FPS), which are modelled as isolator2 element type).

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.

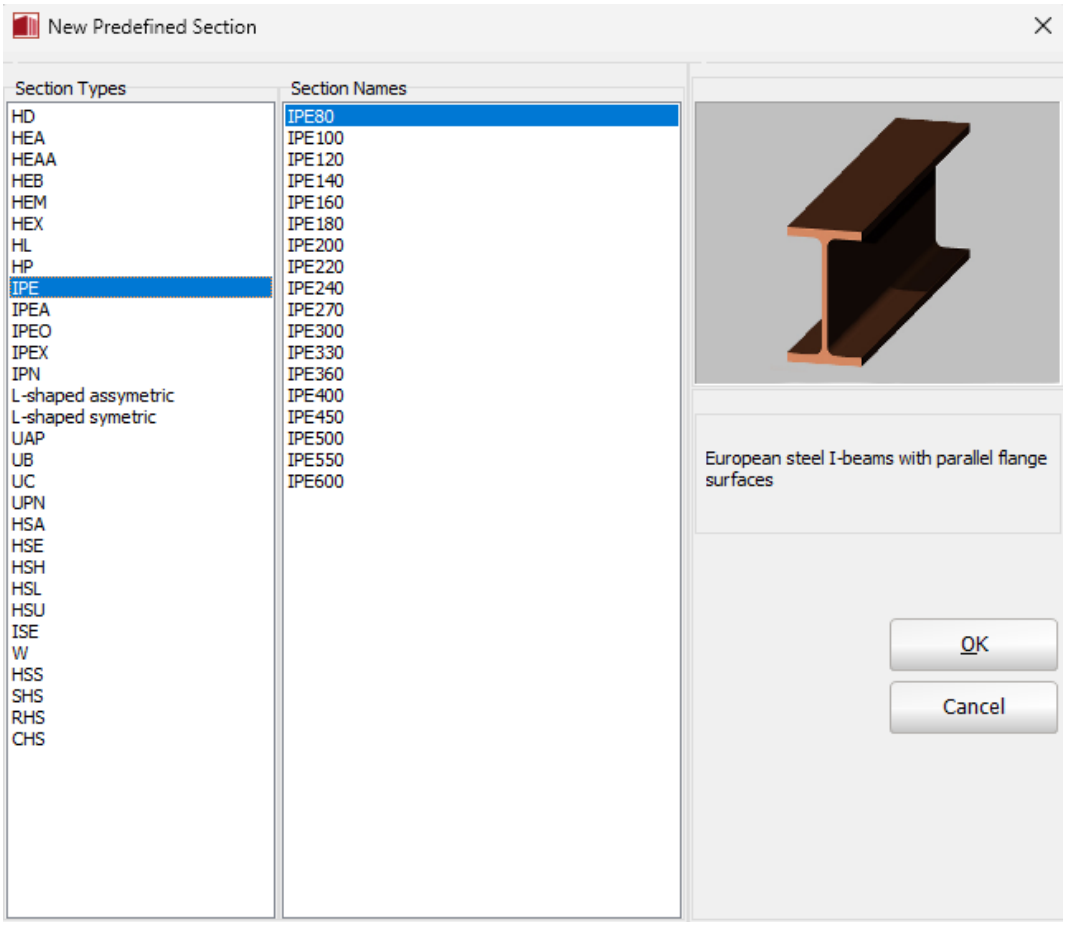
After defining all the section's properties, the new member may be added with a simple click on the Building Modeller Main Window. The location of the section that corresponds to the insertion point (i.e. the mouse click), and rotation of the section on plan view may be selected from the Member Properties window.



Selecting the insertion point

Steel Beam

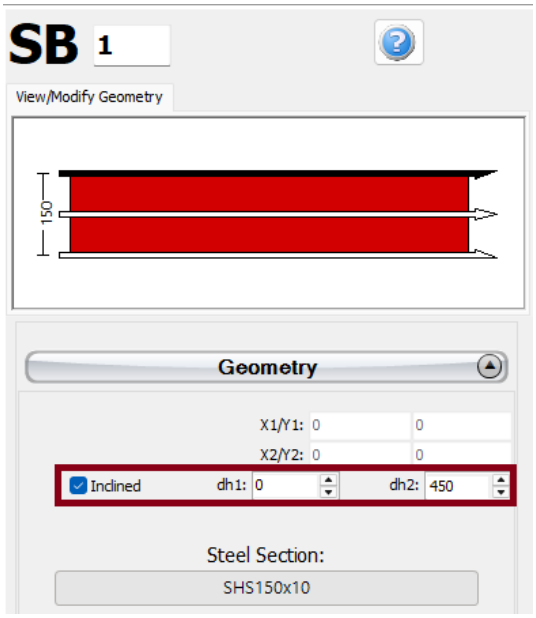
Steel beams may be inserted from the main menu (*Insert > Steel Beam*) or through the corresponding toolbar button  . On the Properties Window that appears users select one section from the predefined standard sections.



Predefined sections

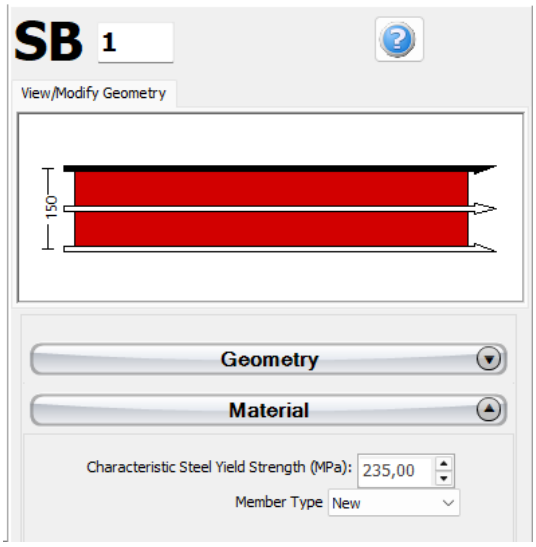
Inclined beams may be efficiently modelled by specifying the elevation differences of the two beam ends relatively to the storey height. The height of the supporting columns is then automatically adapted.

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.



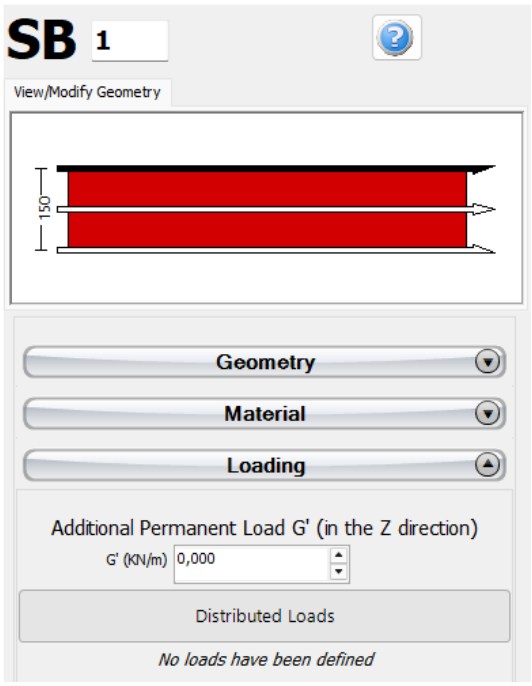
Inclined beam

The steel's properties can be defined within the member's Properties Window. The characteristic steel yield strength and the type of the members, i.e. existing or new member, should be defined.



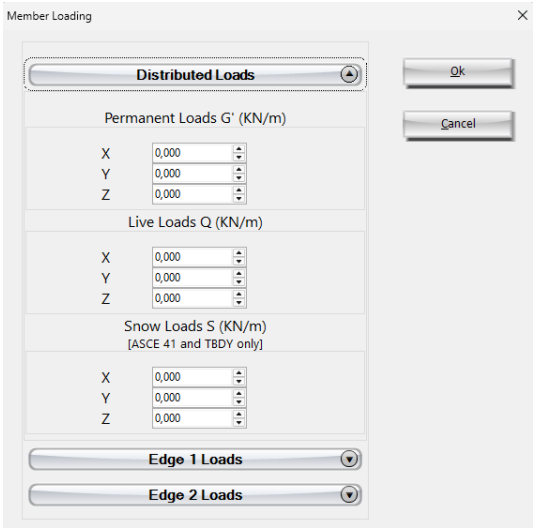
Material

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

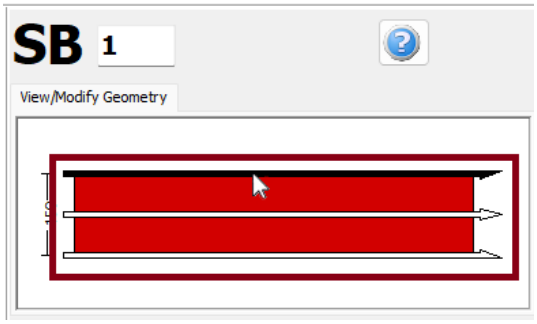
Also loads can be defined by clicking on the More Loads button. Users can define uniform distributed forces along the length of the member in all three translational directions X, Y or Z, and forces or moments in any translational or rotational direction (X, Y, Z, RX, RY or RZ) at either of the two edges 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

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.

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).

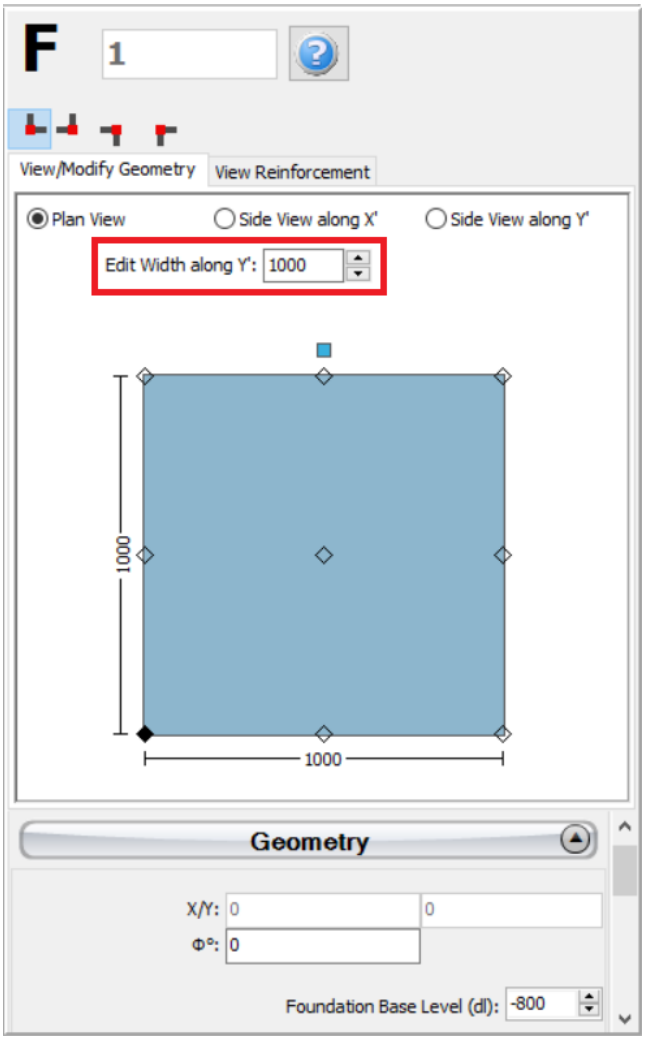


Selecting the insertion line

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.

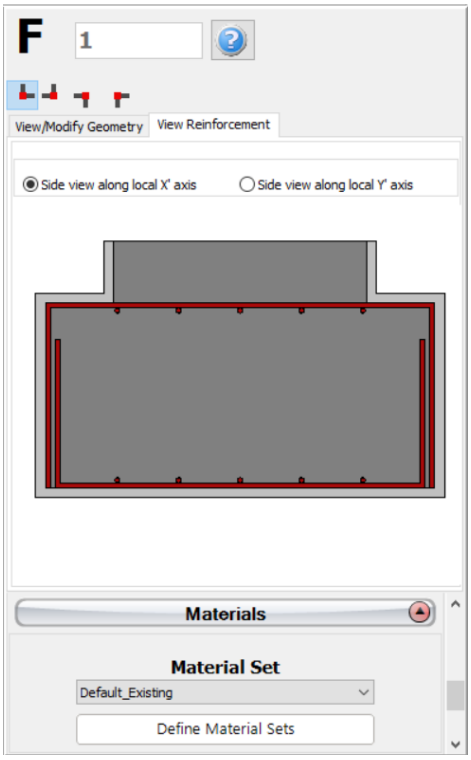
Individual Footings

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).



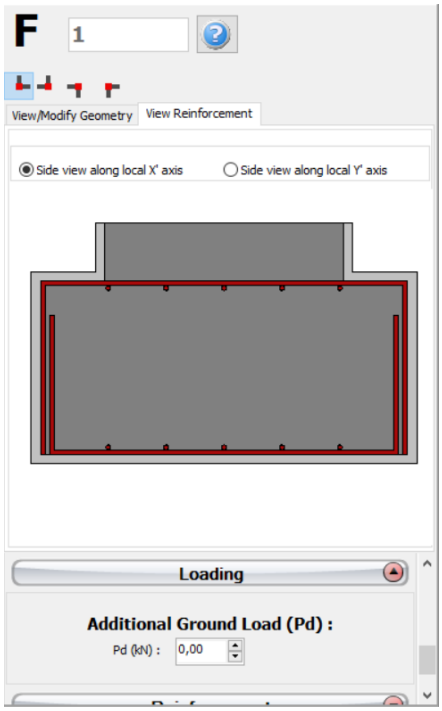
Edit footing's dimensions

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.



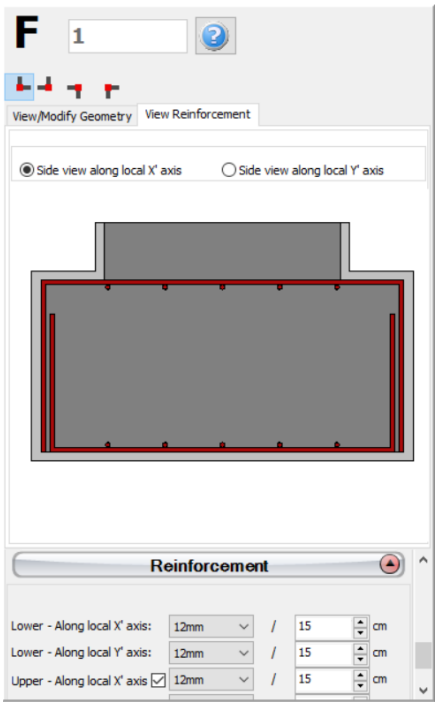
Materials

Additional loading to the Individual Footing from the ground can be specified in the loading module.



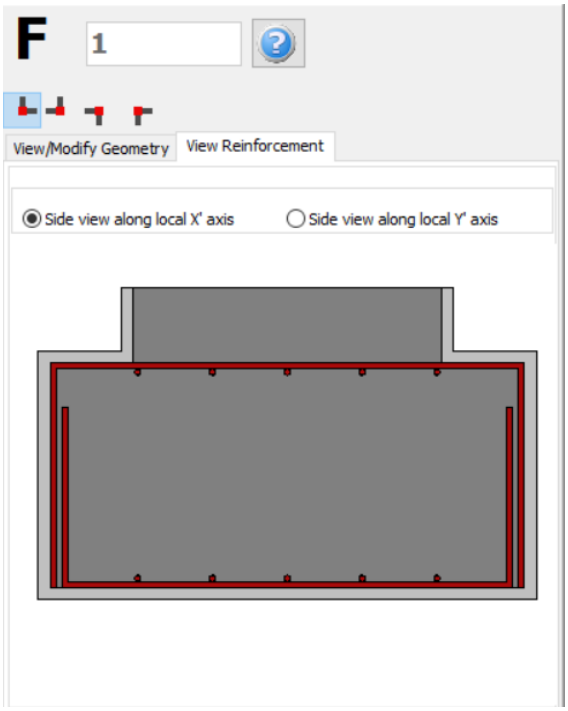
Loading

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.

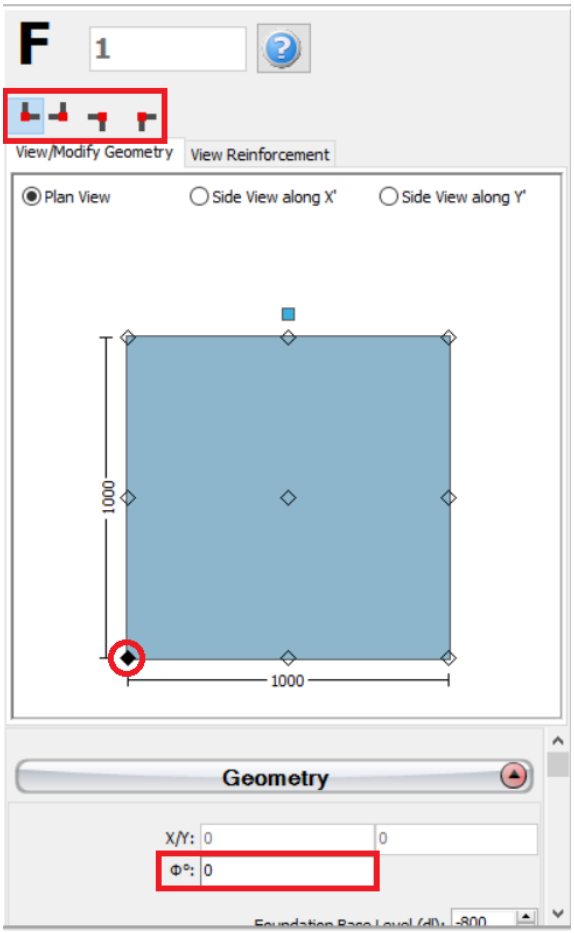


Reinforcement View

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.

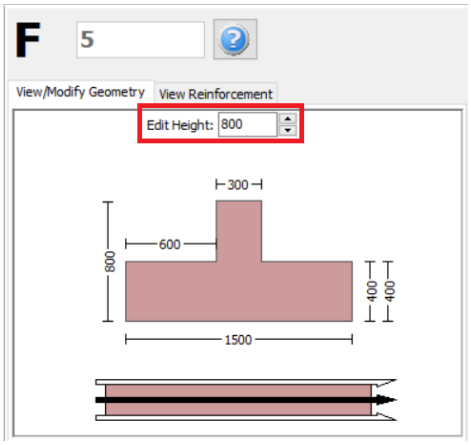
After defining all the properties, of the individual footing the new member may be added with a simple click on the Building Modeller Main Window. The location of the footing that corresponds to the insertion point (i.e. the mouse click), and rotation of the member on plan view may be selected from the Member Properties window.



Selecting the insertion point and rotate the individual footing's plan view

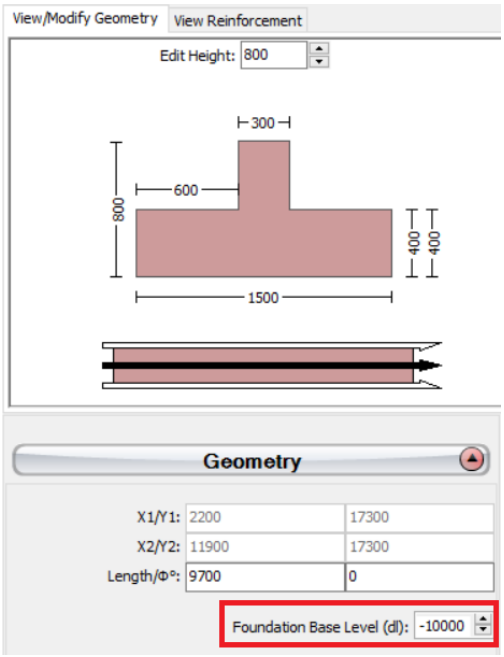
Strip Footing

Strip Footings may be added from the main menu (*Insert > Insert Wall*) or 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.



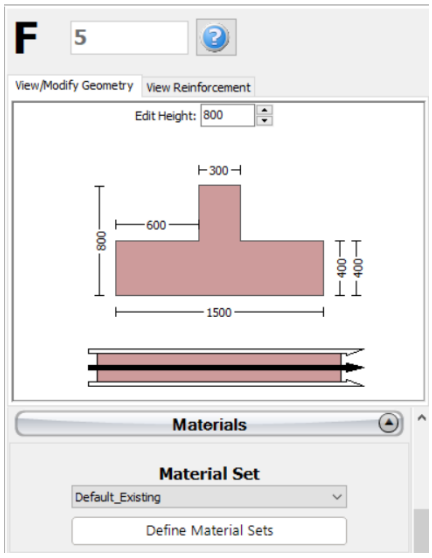
Edit sections dimensions

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.



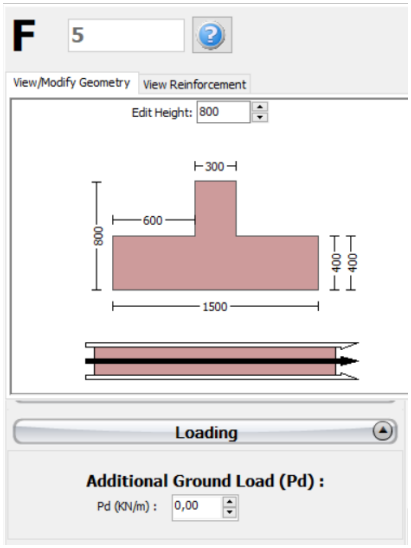
Foundation Base Level

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.



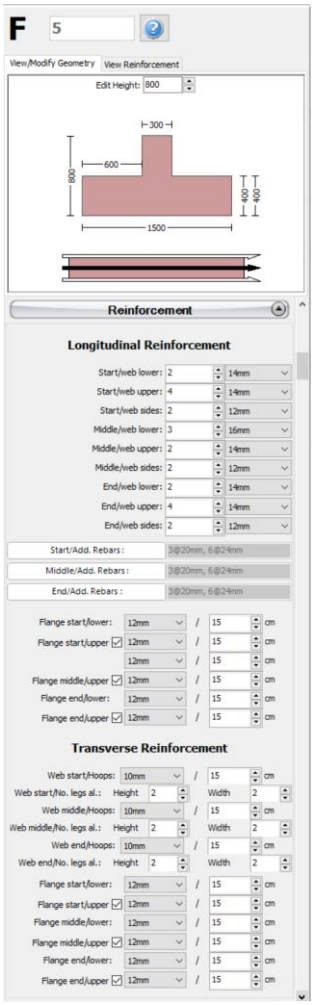
Materials

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.



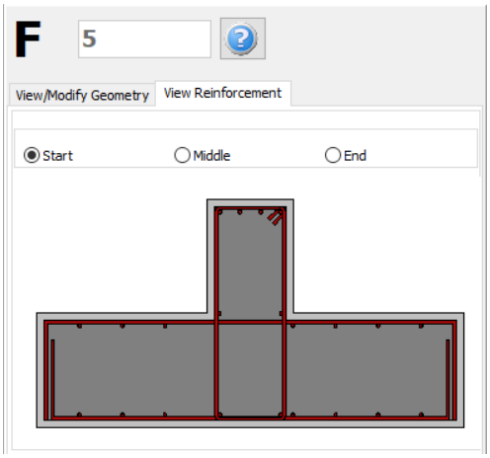
Loading

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 section is displayed (longitudinal and transverse), and the View/Modify Geometry, where the section's dimensions may be viewed and modified.



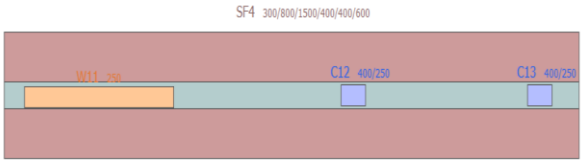
Reinforcement View

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).

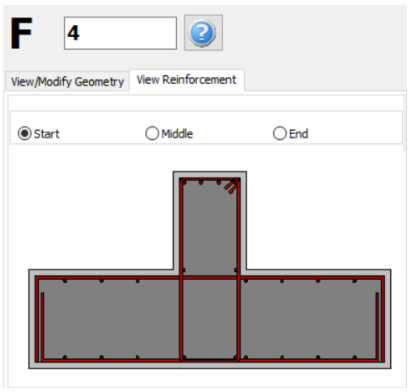


Select the inserting line

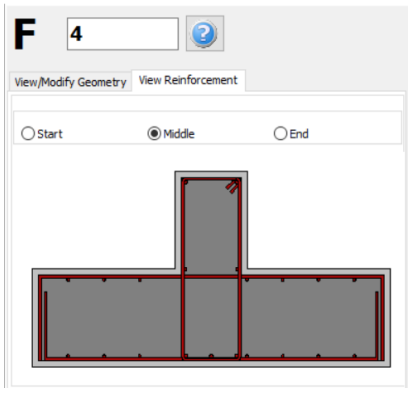


Insertion of strip footing

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.



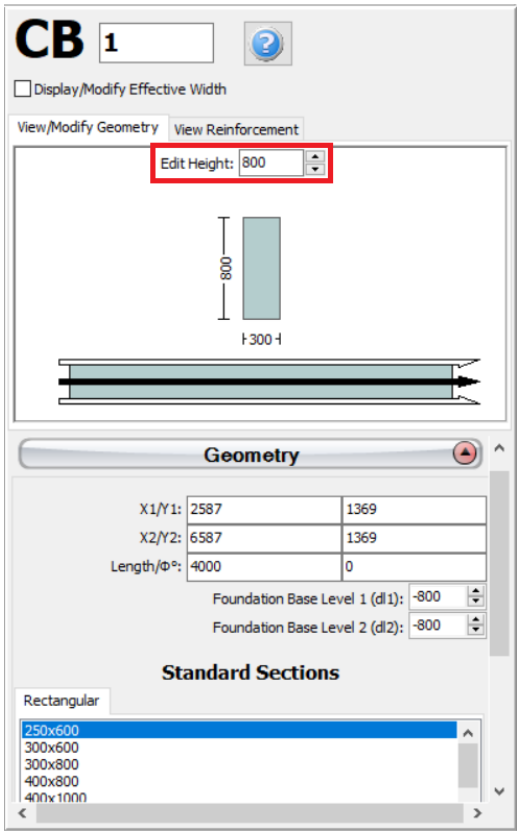
Strip Footing's start section reinforcement



Strip Footing's middle section reinforcement

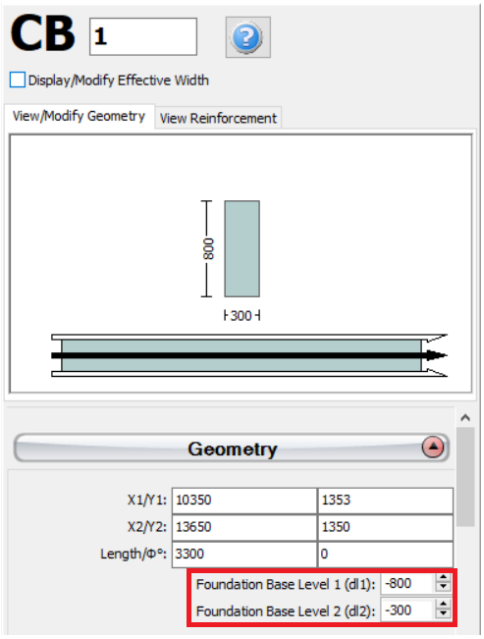
Connecting Beam

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.



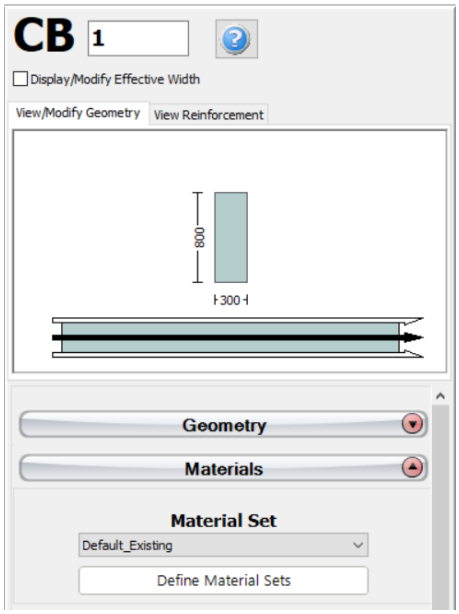
Edit sections dimensions

Inclined connecting beams may be efficiently modelled by specifying the elevation differences of the two connecting beam ends relatively to the foundation base level.



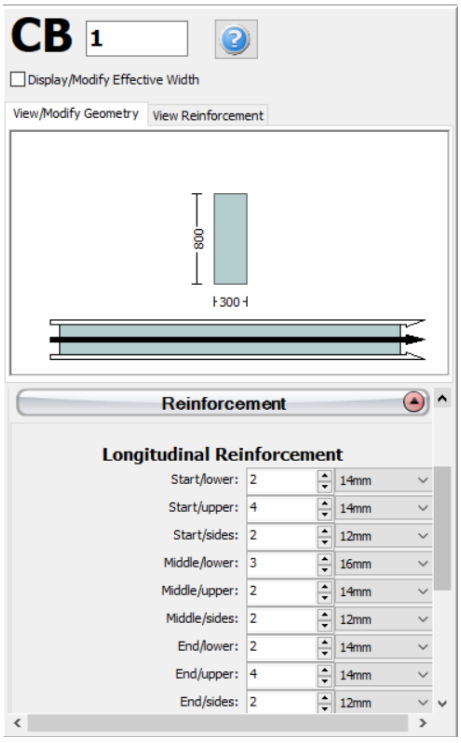
Inclined connecting beam

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.



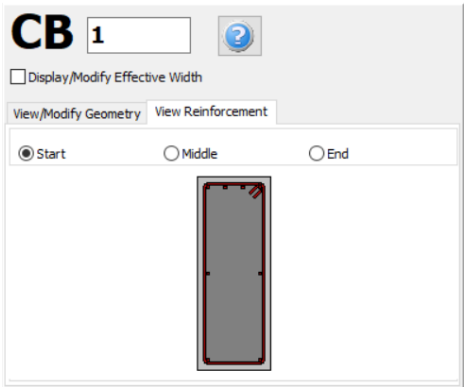
Materials

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.

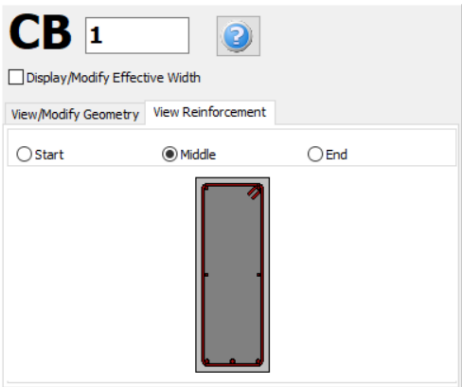


Reinforcement Pattern

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.



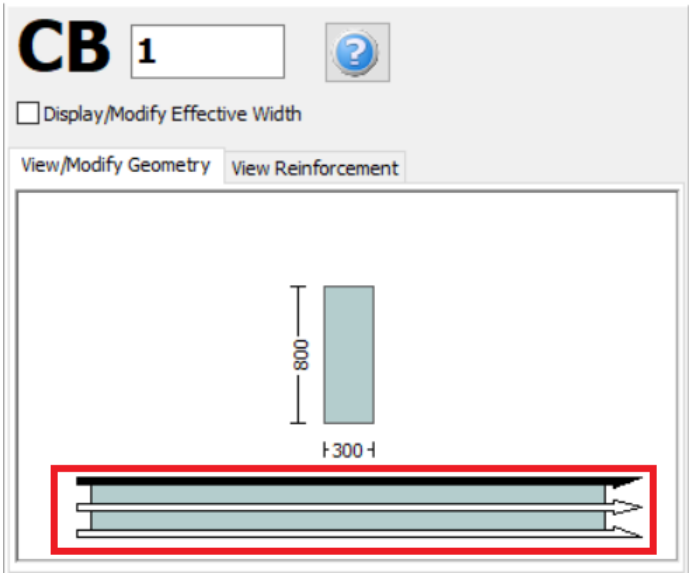
Connecting Beam's start section reinforcement



Connecting Beam's middle section reinforcement

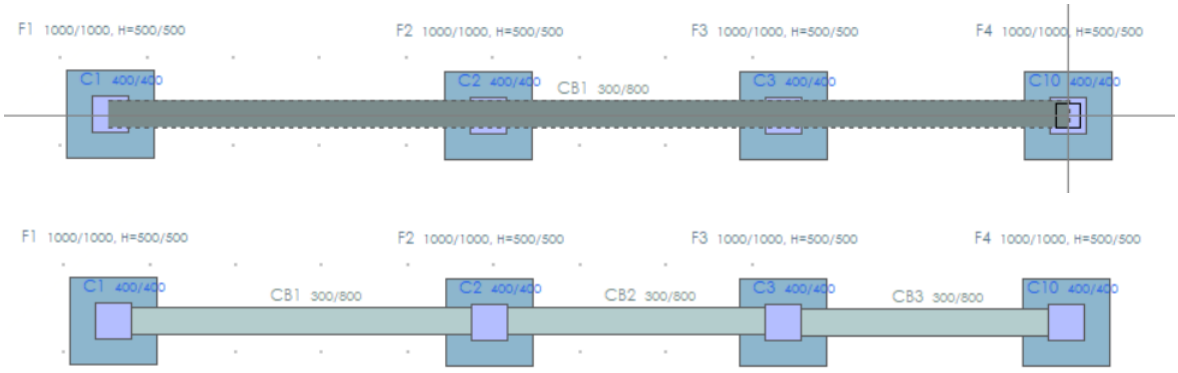
In the Advanced Modelling area, the code-based settings of the structural member can be defined through the Advanced Member Properties dialog box that opens from the corresponding button. The member's modelling parameters may be also defined from the Modelling Parameters dialog box, accessed by the corresponding 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 F - Element Classes

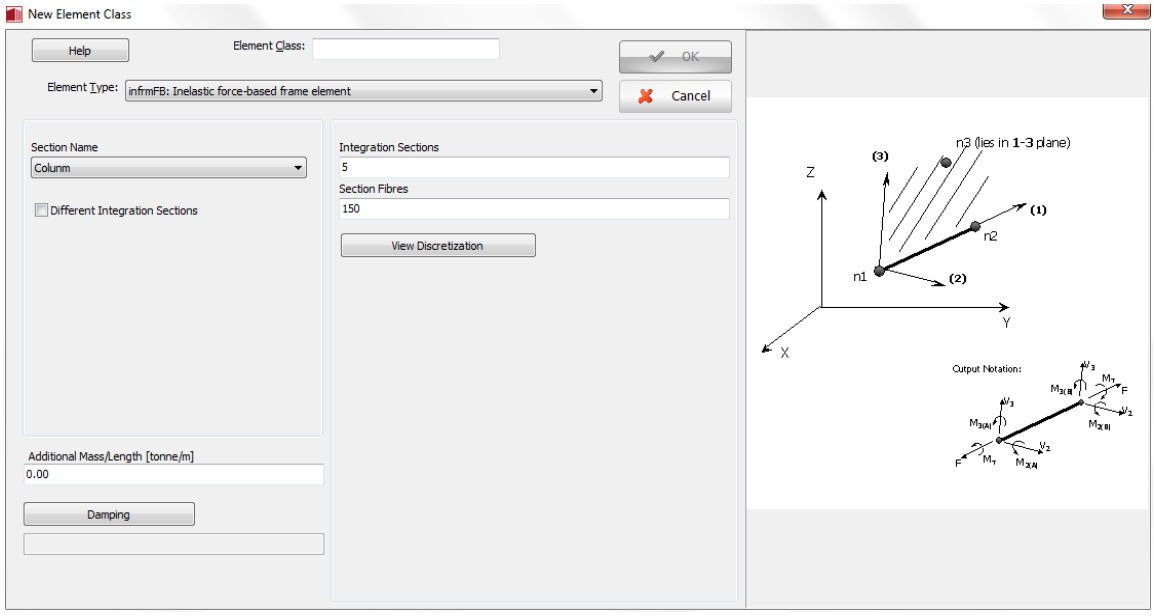
In this appendix the available element types are described in details.

BEAM-COLUMN ELEMENT TYPES

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.

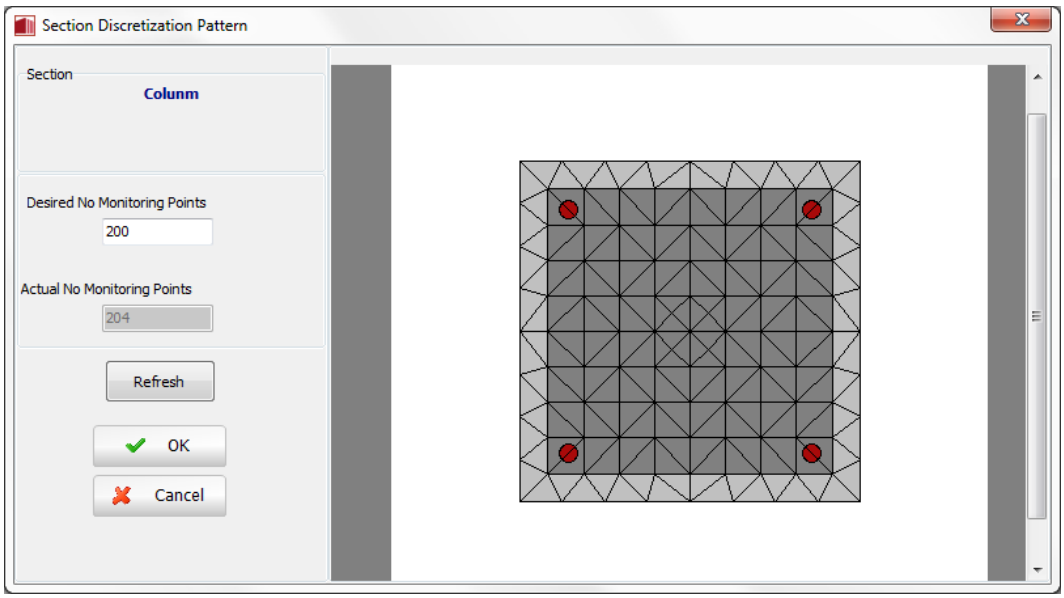
Element *infrmFB* is the most accurate among the four inelastic frame element types of SeismoStruct, since it is capable of capturing the inelastic behaviour along the entire length of a structural member, even when employing a single element per member. Hence, its use allows for very high accuracy in the analytical results, while giving users the possibility of readily employing element chord-rotations output for seismic code verifications (e.g. Eurocode 8, KANEPE, ASCE/SEI 7-05, etc.). The number of section fibres used in equilibrium computations carried out at each of the element's integration sections needs to be defined. User can click the *View Discretization* button in order to visualise the section triangulation (see figure below).



Definition of a new infrmFB element

The ideal number of section fibres, sufficient to guarantee an adequate reproduction of the stress-strain distribution across the element's cross-section, varies with the shape and material characteristics of the latter, depending also on the degree of inelasticity to which the element will be forced to. As a crude rule of thumb, users may consider that single-material sections will usually be adequately represented by 100 fibres, whilst more complicated sections, subjected to high levels of inelasticity, will normally call for the employment of 200 fibres or more. However, and clearly, only a sensitivity study carried out by the user on a case-by-case basis can unequivocally establish the optimum number of section fibres.

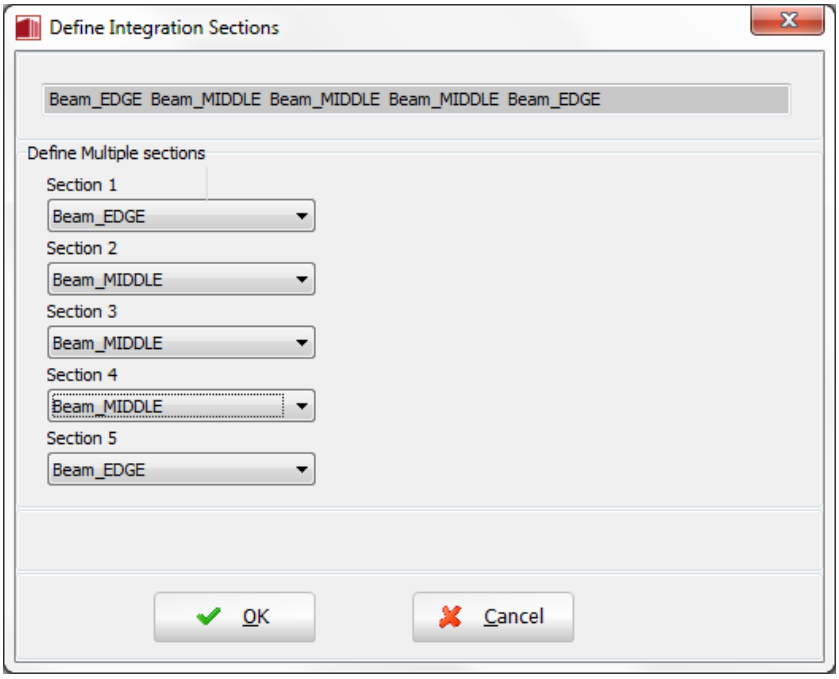
In the *Section Discretization Pattern* dialog box the software provides the desired and the actual (after the section discretisation has been performed, employing triangulation procedures) number of monitoring points. By clicking on the *Refresh* button it is possible to update the view of the section discretisation.



Section triangulation

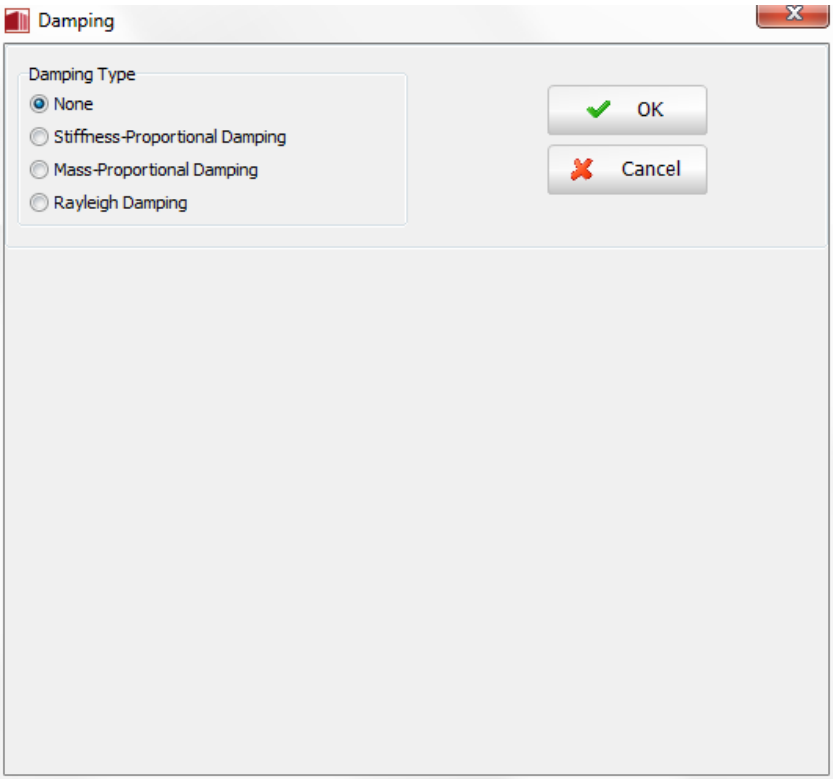
In addition, the number of integration sections needs to be defined. A number between 4 and 7 integration sections will typically be adopted, though users are warmly invited to search the bibliography [e.g. Papadrakakis 2008; Calabrese et al. 2010] for further guidance on this matter (it is recalled that the location of such integration sections across the element's length are indicated in Material Inelasticity). In particular it is noted that up to 7 integration sections may be needed to accurately model hardening response, but, on the other hand, 4 or 5 integration sections may be advisable when it is foreseen that the elements will reach their softening response range.

NOTE: Instead of discretizing the elements to represent the changes in reinforcement details (see above), it is possible to use one single infrmFB element per member and then define multiple sections. It is noted that these sections may differ only in the reinforcement (i.e. section type, dimensions and materials have to be the same).



Multiple sections

In this element's dialog box it is also possible to define an element-specific damping, as opposed to the global damping defined in *General* > *Project Settings* > *Damping*. To do so, users need simply to press the *Damping* button and then select the type of damping that better suits the element in question (users should refer to the Damping menu for a discussion on the different types of damping available and hints on which might the better options).

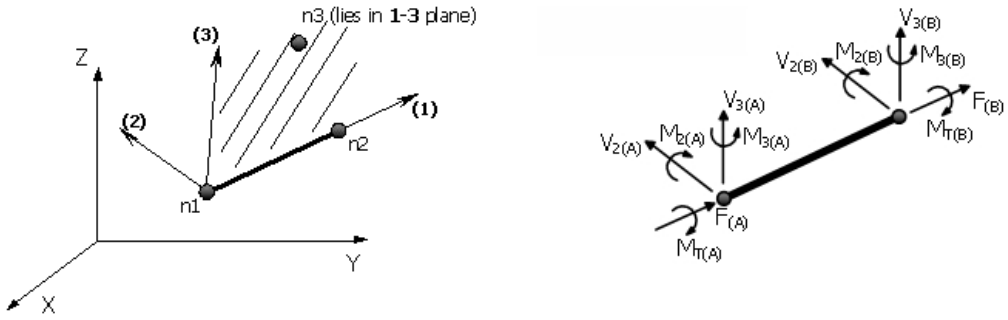


Definition of an element-specific damping

IMPORTANT: Damping defined at element level takes precedence over global damping, that is, the "globally-computed" damping matrix coefficients that are associated to the degrees-of-freedom of a given element will be replaced by coefficients that will have been calculated through the multiplication of the mass matrix of the element by a mass-proportional parameter, or through the multiplication of the element stiffness matrix by a stiffness-proportional parameter, or through the calculation of an element damping Rayleigh matrix.

NOTE: If Rayleigh damping is defined at element level, using varied coefficients from one element to the other, or with respect to those employed in the global damping settings, then non-classical Rayleigh damping is being modelled, since Rayleigh damping requires uniform damping definition.

Local axes and output notation are defined in the figure below. Refer to the discussion on global and local frame axes in Appendix A for a detailed description on the determination of the orientation of the local element axis system.



Local Axes and Output Notation for infrmFB elements

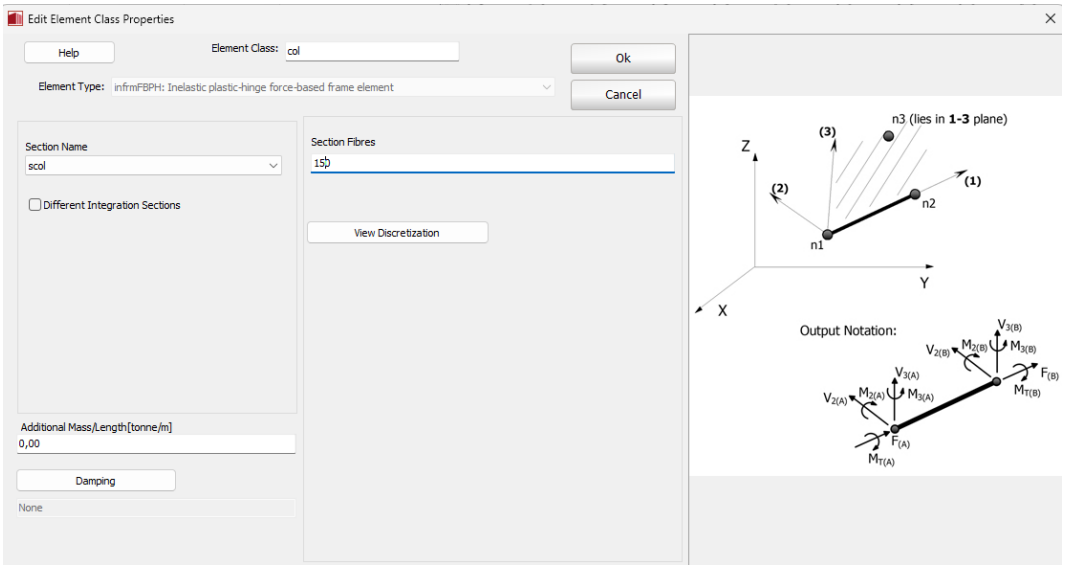
Inelastic force-based plastic hinge frame element type - infrmFBPH

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 end section 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].

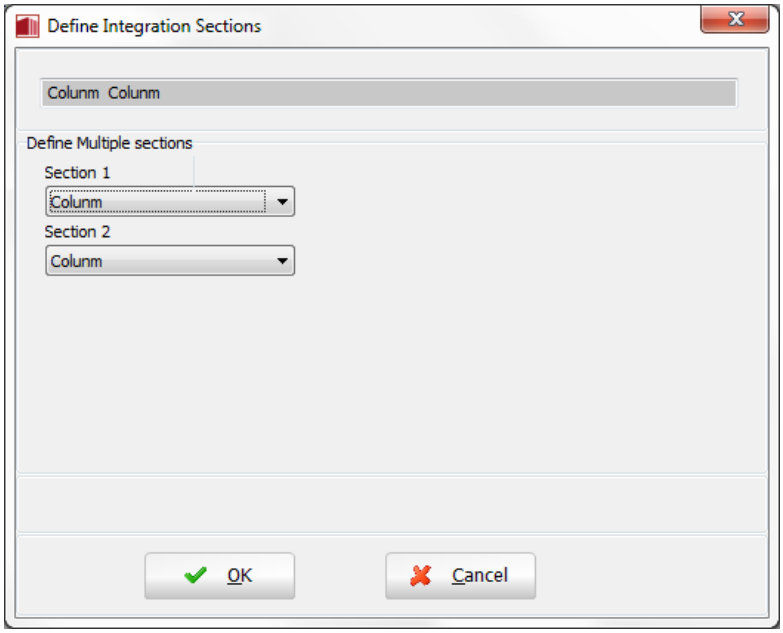
The number of section fibres used in equilibrium computations carried out at the element's end sections needs to be defined. The ideal number of section fibres, sufficient to guarantee an adequate reproduction of the stress-strain distribution across the element's cross-section, varies with the shape and material characteristics of the latter, depending also on the degree of inelasticity to which the element will be forced to. As a crude rule of thumb, users may consider that single-material sections will usually be adequately represented by 100 fibres, whilst more complicated sections, subjected to high levels of inelasticity, will normally call for the employment of 200 fibres or more. However, and clearly, only a sensitivity study carried out by the user on a case-by-case basis can unequivocally establish the optimum number of section fibres.

The plastic hinge is set equal to 16.67%[e.g. Scott and Fenves 2006, Papadrakakis 2008, Calabrese et al. 2010] for guidance.



Definition of a new infrmFBPH element

Similarly to infrmFB, changes in reinforcement details can be achieved with the use of a single infrmFBPH element per member, when multiple sections have been defined. It is noted that these sections may differ only in the reinforcement (i.e. section type, dimensions and materials have to be the same).



Multiple sections for the infrmFBPH element

In this element's dialog box it is also possible to define an element-specific damping, as opposed to the global damping defined in *General* > *Project Settings* > *Damping*. To do so, users need simply to press the *Damping* button and then select the type of damping that better suits the element in question (users should refer to the Damping menu for a discussion on the different types of damping available and hints on which might the better options).

IMPORTANT: Damping defined at element level takes precedence over global damping, that is, the "globally-computed" damping matrix coefficients that are associated to the degrees-of-freedom of a given element will be replaced by coefficients that will have been calculated through the multiplication of the mass matrix of the element by a mass-proportional parameter, or through the multiplication of the element stiffness matrix by a stiffness-proportional parameter, or through the calculation of an element damping Rayleigh matrix.

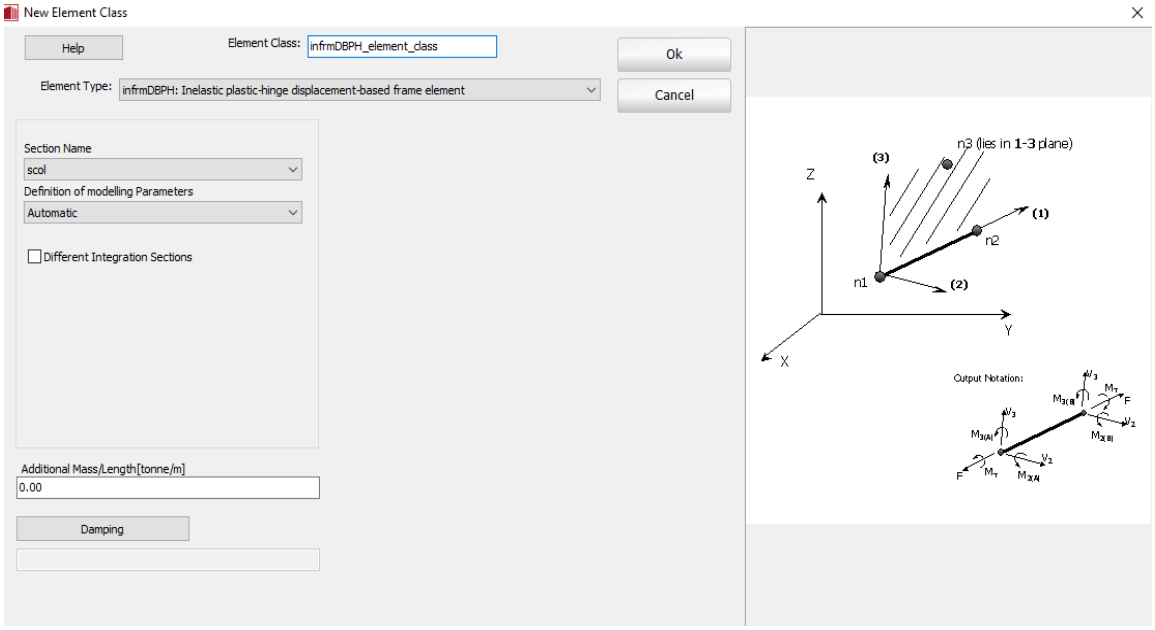
NOTE: If Rayleigh damping is defined at element level, using varied coefficients from one element to the other, or with respect to those employed in the global damping settings, then non-classical Rayleigh damping is being modelled, since Rayleigh damping requires uniform damping definition.

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. All nonlinear deformations of the element are lumped in these rotational springs, whereas the rest of the member remains elastic. The DBPH formulation is capable of modelling the geometric nonlinearities.

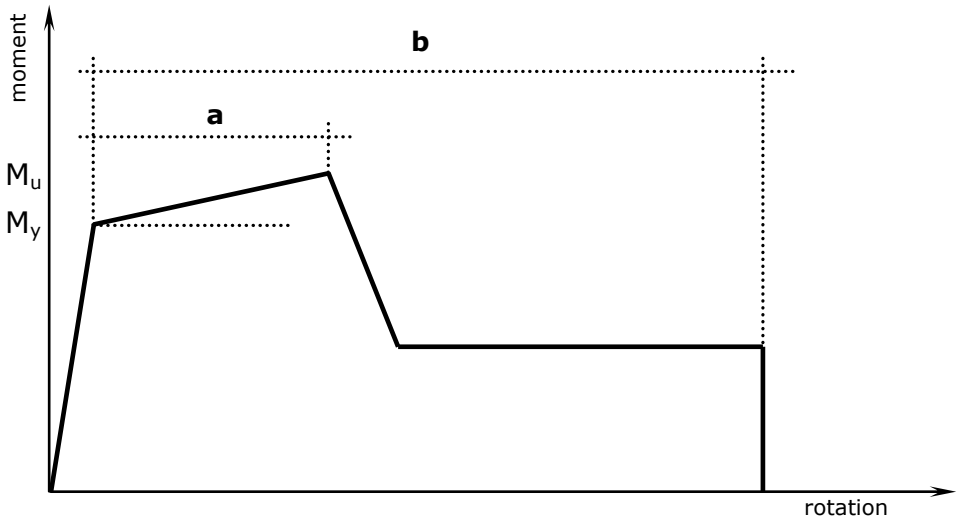
The element consists of three sub-elements, the two links at the member edges, which model the plastic rotational deformations around the 2nd and the 3rd local axes, and an elastic frame element in the middle, which models the part of the member that remains elastic. The sub-elements are connected in series, and an iterative procedure is required, in order to achieve internal element equilibrium.



Definition of a new infrmDBPH element

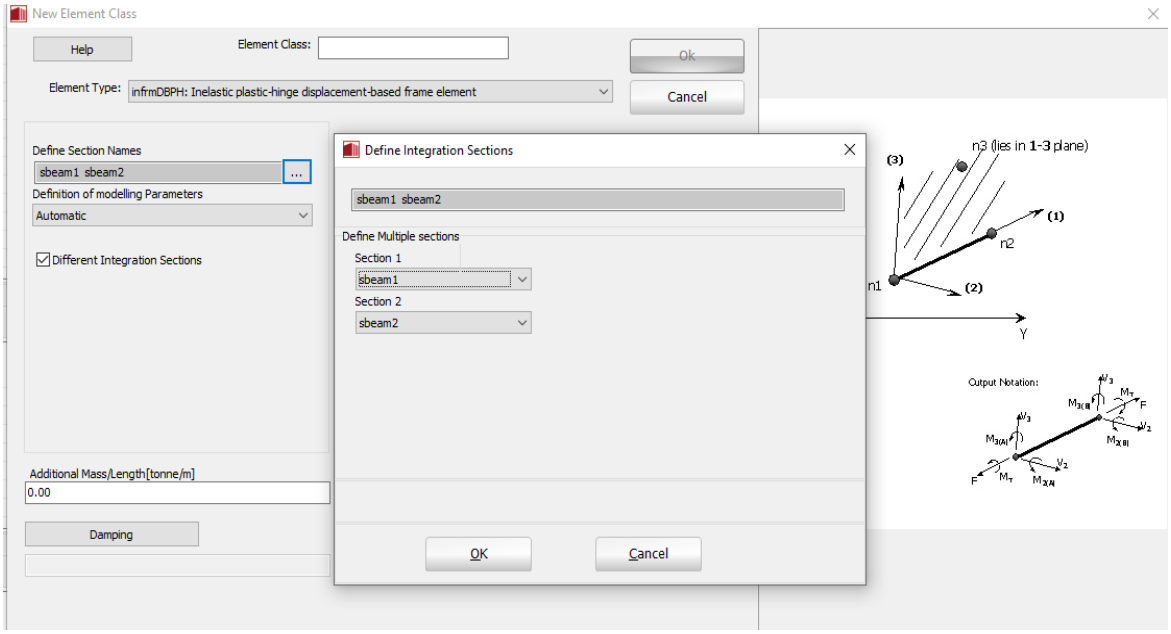
The force-displacements relationships of the four nonlinear rotational springs at the element's ends feature a hysteretic curve that is based on SeismoStruct's built-in MIMK_bilin nonlinear curve (Modified Ibarra-Medina-Krawinkler deterioration curve with bilinear hysteretic rules). The yield and ultimate bending moment (M_y and M_u respectively), as well as the deformation at yield d_y are calculated from a moment-curvature section analysis after the application of the initial loads on the structure, taking into account the axial load imposed on the element. Instead, the plastic rotation capacity **a** and the rotation at ultimate **b** are estimated directly from ASCE 41-23, and in particular from Tables 10-7 (for beams),

10-8 & 10-9 (for columns) and 10-19 (for walls); see also ASCE 41-23, Figure 10-1 and the figure below for more details.



Plastic rotation capacity a and rotation at ultimate b, according to ASCE 41-23

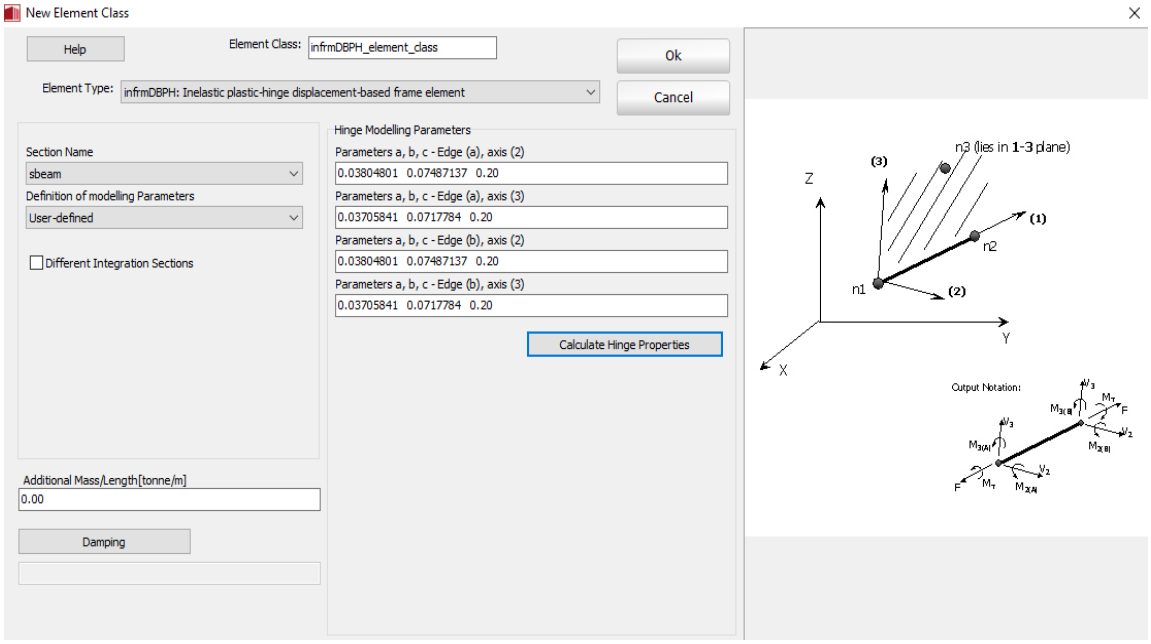
The moment-rotation curves in the two local axes at each end are independent. This is obviously a simplification with respect to the force-based plastic hinge element, where inelastic deformations spread over a finite region at the ends of the girder and the behaviour in the two local axes is correlated. However, this lack of accurate modelling is compensated shorter analysis times. Since the hysteretic curve parameters are automatically calculated by the program, users need only specify the member's section. Similarly to the infrmFB and infrmFBPH elements, the changes in reinforcement details within the same member can be achieved with the use of multiple sections per element.



Definition of a new infrmDBPH element with multiple sections

Apart from the automatic calculation of the plastic hinge properties, SeismoStruct also allows the input of a, b and c values (c being the ratio of the residual to the ultimate strength) that are different from the calculated ones. This can be achieved, if in the Definition of Modelling Parameters drop-down menu the User-defined option is selected. Users are than allowed to freely introduce the plastic hinge modelling

parameters. Note however that even in the case of user defined parameters a section should be specified, mainly for display purposes (e.g. to show the member in the 3D plot).



Definition of a new infrmDBPH element with user defined plastic hinge parameters

Finally, by clicking the Calculate Hinge Properties button a new dialog box appears that provides assistance in the determination of these parameters, based on several parameters such as the ratio of the area of the distributed longitudinal reinforcement to the gross concrete area ρ_l, the ratio of the area of the distributed transverse reinforcement to the gross concrete area ρ_t, the gross section area A_g, the expected yield strength of the longitudinal and transverse reinforcing steel f_{ylE} & f_{ytE}, the expected compressive strength of concrete f'_{cE}, the member axial force N_{ud}, whether the member's shear reinforcement is conforming or not, whether the member is controlled by flexure or shear etc. These calculations are carried out according to the ASCE 41-23 procedures.

It is also possible to define an element-specific damping, as opposed to the global damping defined in *General* > *Project Settings* > *Damping*, as described in the sections on infrmFB and infrmFBPH. Finally, local axes and output notation are the same with the force-based elements.

Inelastic displacement-based frame element type - infrmDB

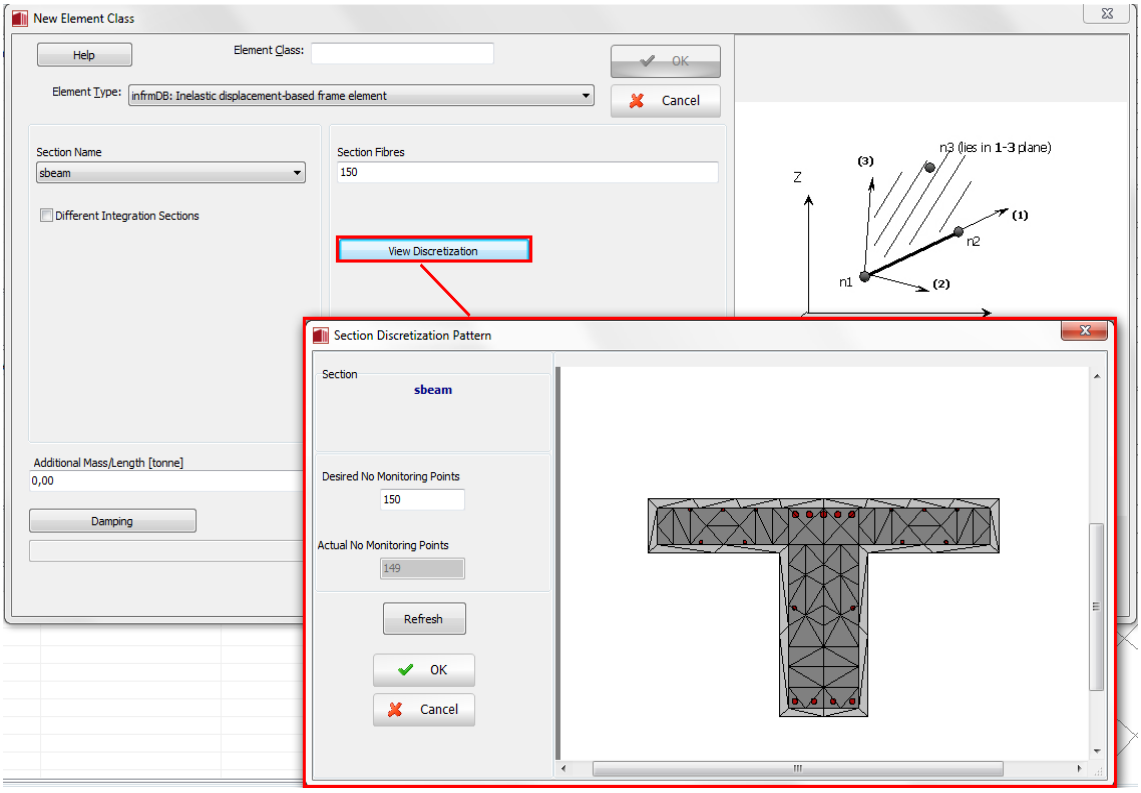
This is the displacement-based 3D beam-column element type capable of modelling members of space frames with geometric and material nonlinearities. As described in the Material inelasticity paragraph, the sectional stress-strain state is obtained through the integration of the nonlinear uniaxial material 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, leading to the need for a mesh refinement, in order to achieve good accuracy in the case of higher order distributions of deformations. Typically, 4 to 6 elements per structural member need to be defined, hence users need to

post-process nodal displacements/rotation in order to estimate the members chord-rotations [e.g. Mpampatsikos et al. 2008].

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 needs to be defined. Users can click the *View Discretization* button in order to visualise the section triangulation (see figure below).



Definition of a new infrmDB element and section discretisation

Similarly to infrmFB and infrmFBPH, multiple sections can be employed to define changes in reinforcement details between the two integration sections. Further, the element-specific damping can also be defined by the *Damping* dialog box (users should refer to the Damping menu for a discussion on the different types of damping available and hints on which might the better options).

Local axes and output notation are defined as with the other frame element types.

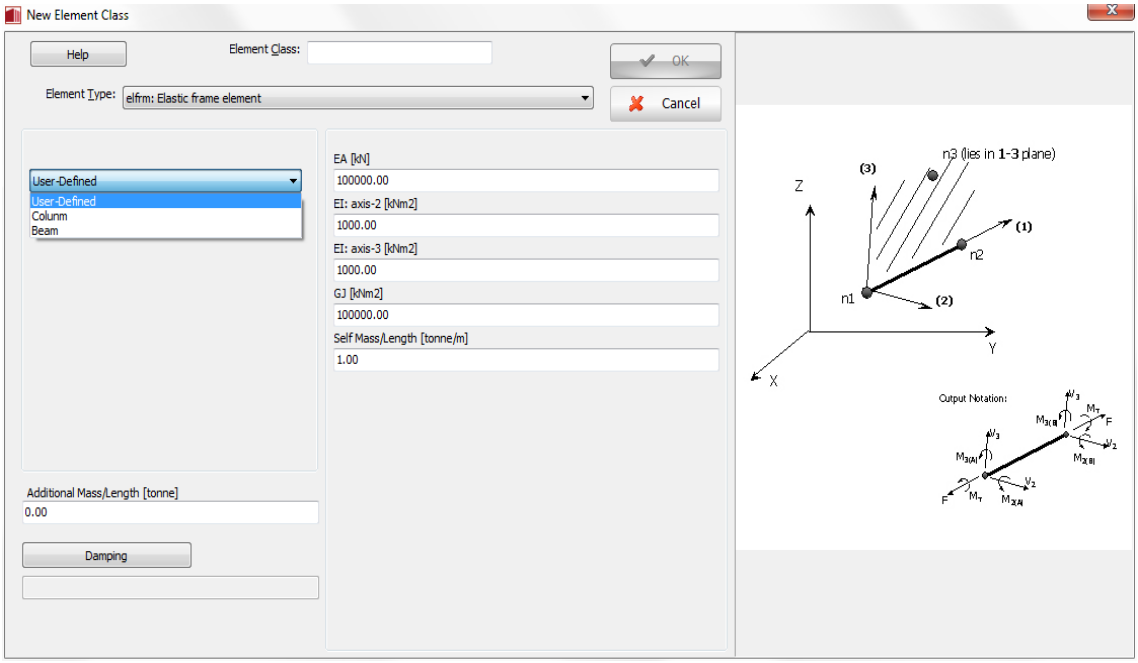
Elastic frame element - elfrm

There are cases where the use of an inelastic frame element is not required (e.g. eigenvalue analysis, structures subjected to low levels of excitation and thus responding within their elastic range, dynamic response of a bridge deck, etc.). 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 SeismoStruct.

In order to fully characterise this type of element, users are asked to either specify an already created section (for which the program will then automatically compute all the necessary elastic mechanical properties) or to instead specify here custom values of EA, EI_2, EI_3 and GJ, where E is the modulus of elasticity, A is the cross-section area and I_2 and I_3 are the moments of inertia (or second moments of area) around local axes (2) and (3). The torsional constant is represented by J (which should not be mistaken with the polar moment of inertia), whilst G stands for the modulus of rigidity, obtained as $G=E/(2(1+\nu))$, where ν is the Poisson's ratio [e.g. see Pilkey, 1994].

The stiffness matrix of the elfrm element, as defined in the local chord system, is:

$$\frac{1}{L}\begin{bmatrix} 4EI_2 & 0 & 2EI_2 & 0 & 0 & 0 \\ 0 & 4EI_3 & 0 & 2EI_3 & 0 & 0 \\ 2EI_2 & 0 & 4EI_2 & 0 & 0 & 0 \\ 0 & 2EI_3 & 0 & 4EI_3 & 0 & 0 \\ 0 & 0 & 0 & 0 & EA & 0 \\ 0 & 0 & 0 & 0 & 0 & GJ \end{bmatrix}$$



Definition of a new elfrm element

In this element's dialog box it is also possible to define an element-specific damping, as opposed to the global damping defined in *General* > *Project Settings* > *Damping*. To do so, users need simply to press the *Damping* button and then select the type of damping that better suits the element in question (users should refer to the Damping menu for a discussion on the different types of damping available and hints on which might the better options).

IMPORTANT: Damping defined at element level takes precedence over global damping, that is, the "globally-computed" damping matrix coefficients that are associated to the degrees-of-freedom of a given element will be replaced by coefficients that will have been calculated through the multiplication of the mass matrix of the element by a mass-proportional parameter, or through the multiplication of the element stiffness matrix by a stiffness-proportional parameter, or through the calculation of an element damping Rayleigh matrix.

NOTE 1: In the elfrm element, P-delta effects as well as large displacement/rotation effects are duly taken into account.

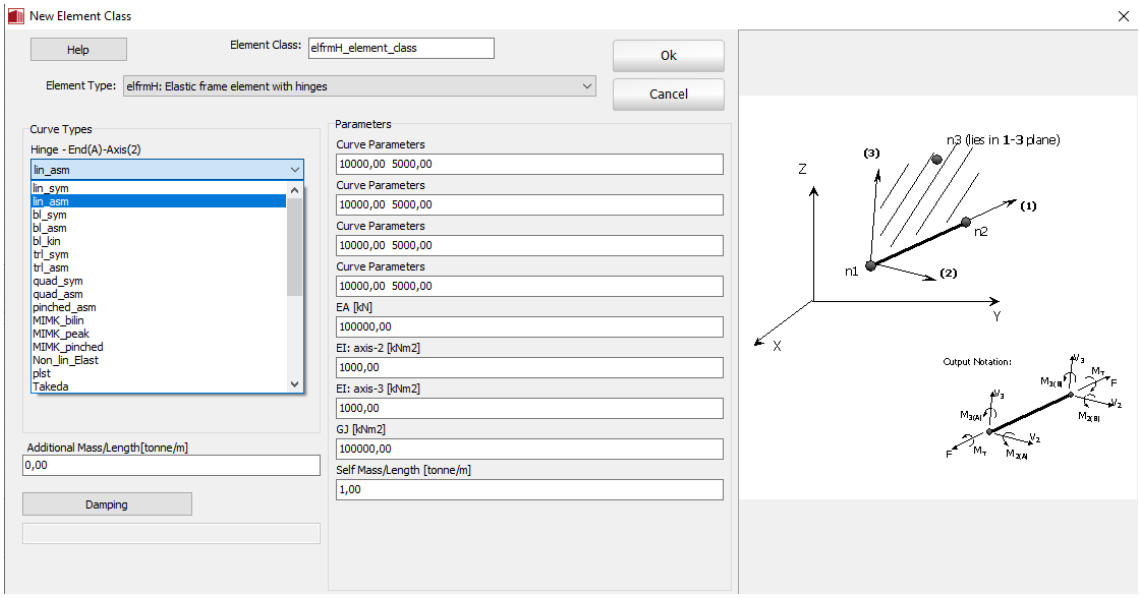
NOTE 2: If Rayleigh damping is defined at element level, using varied coefficients from one element to the other, or with respect to those employed in the global damping settings, then non-classical Rayleigh damping is being modelled, since Rayleigh damping requires uniform damping definition.

Local axes and output notation are the same as infrmDB and infrmFB elements.

Elastic frame element with hinges - elfrmH

This is an elastic linear frame element with two rotational springs at the two ends. Nonlinear behavior is modeled through lumped plasticity at the ends of the element while the rest of the element is elastic. The plastic rotational deformations around the 2nd and the 3rd local axes are modelled using moment-rotation response curves.

As a result the user needs to define the response curves modelling the moment-rotation relationship of the two rotational degrees of freedom at the two edges of the element.



Definition of a new elfrmH element

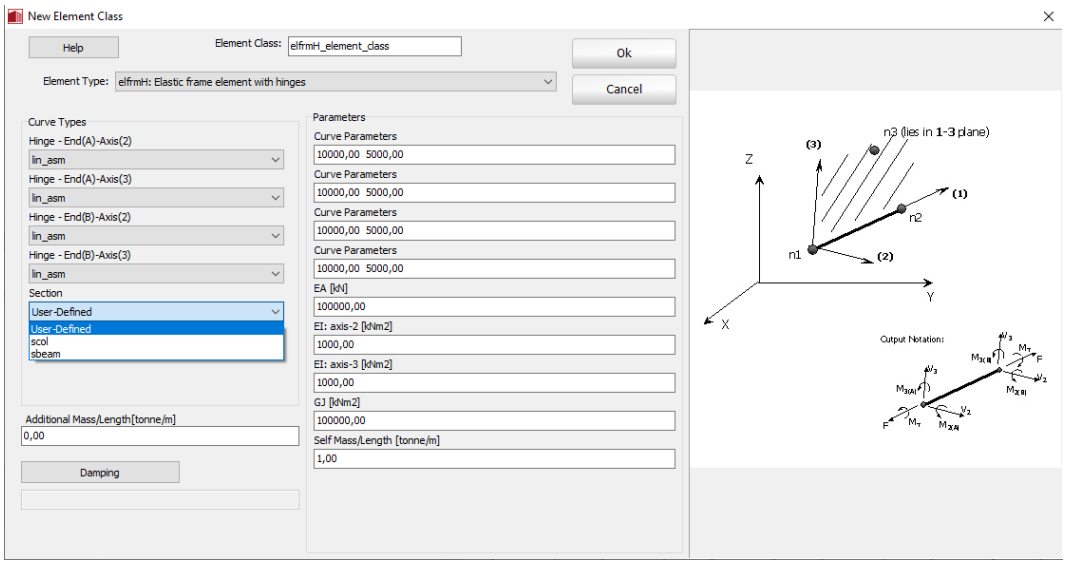
Currently, twenty-nine response curves are available, selectable from within the Element Class dialog box, whenever a rackh element type is selected.

- Linear symmetric curve - lin_sym
- Linear asymmetric curve - lin_asm
- Bilinear symmetric curve - bl_sym
- Bilinear asymmetric curve - bl_asm
- Bilinear kinematic hardening curve - bl_kin
- Trilinear symmetric curve - trl_sym
- Trilinear asymmetric curve - trl_asm
- Quadrilinear symmetric curve - quad_sym
- Quadrilinear asymmetric curve - quad_asm
- Pinched asymmetric curve - pinched_asm
- Modified Ibarra-Medina-Krawinkler Deterioration curve with Bilinear Hysteretic Response – MIMK_bilin
- Modified Ibarra-Medina-Krawinkler Deterioration Model with Peak-Oriented Hysteretic Response – MIMK_peak
- Modified Ibarra-Medina-Krawinkler Deterioration Model with Pinched Hysteretic Response – MIMK_Pinched
- Nonlinear elastic curve - Non_lin_Elast
- Plastic curve – plst
- Simplified bilinear Takeda curve –Takeda
- Asymmetric bilinear Takeda curve –Takeda_asm
- Ramberg Osgood curve - Ramberg_Osgood
- Modified Richard-Abbott curve - Richard_Abbott

- Soil-structure interaction curve - ssi_py
- Gap-hook curve - gap_hk
- Multi-linear curve – multi_lin
- Smooth curve – smooth
- Viscous Damper – vsc_dmp
- Bouc Wen curve - Bouc_Wen
- Elastic – Perfectly plastic Gap curve - gap_elpl
- Impact response curve - pound_hz
- Self Centering Brace response curve - scb
- Generic Hysteretic Curve - gen_hyst

For a comprehensive description of the available response curves associated to the elastic frame element with hinges element refer to Appendix G.

In order to fully characterise this type of element, users are asked to either specify an already created section (for which the program will then automatically compute all the necessary elastic mechanical properties) or to instead specify here custom values of EA, EI_2, EI_3 and GJ, where E is the modulus of elasticity, A is the cross-section area and I_2 and I_3 are the moments of inertia (or second moments of area) around local axes (2) and (3). The torsional constant is represented by J (which should not be mistaken with the polar moment of inertia), whilst G stands for the modulus of rigidity, obtained as $G=E/(2(1+\nu))$, where ν is the Poisson's ratio [e.g. see Pilkey, 1994].

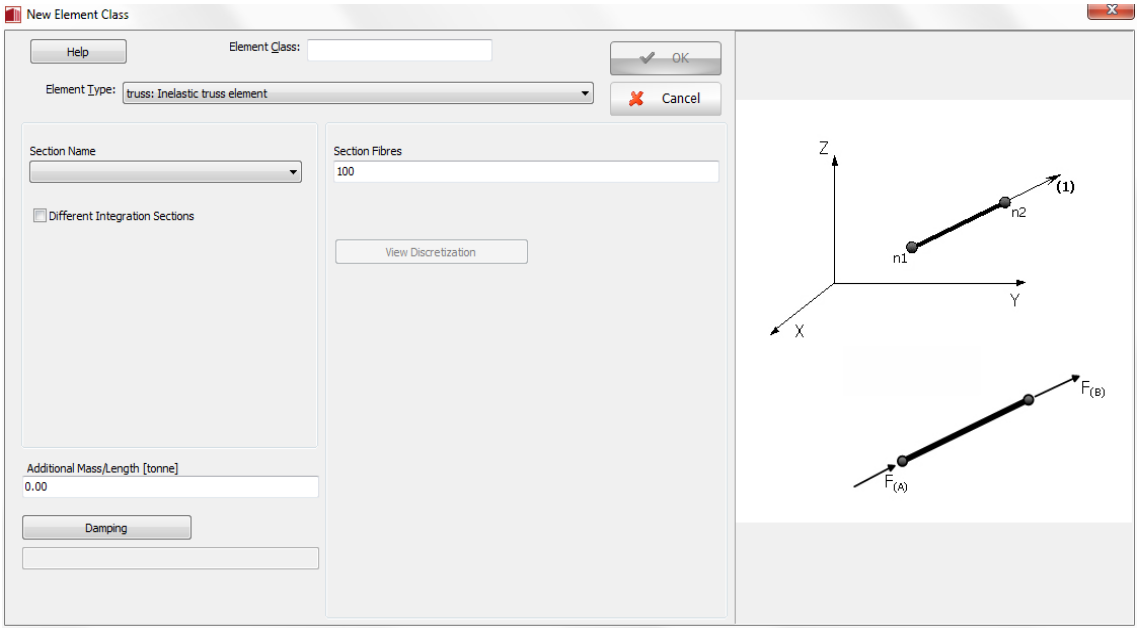


Definition of a new elfrmh element

It is also possible to define an element-specific damping, as opposed to the global damping defined in *General>Project Settings>Damping*, as described in the sections of elfrm. Finally, local axes and output notation are the same with the elfrm element.

Inelastic truss element - truss

The inelastic *truss* element might come particularly handy in those cases where there is a need to introduce members that work in their axial direction only (e.g. horizontal or vertical braces, steel trusses, etc.). In order to fully characterise this type of element class users need only to select a cross-section and specify the number of fibres in which the latter is to be subdivided. The stiffness matrix of this element is made up of a single term EA, updated at every step of the analysis.



Definition of a new truss element

In this element's dialog box it is also possible to define an element-specific damping, as opposed to the global damping defined in *General* > *Project Settings* > *Damping*. To do so, users need simply to press the *Damping* button and then select the type of damping that better suits the element in question (users should refer to the Damping menu for a discussion on the different types of damping available and hints on which might the better options).

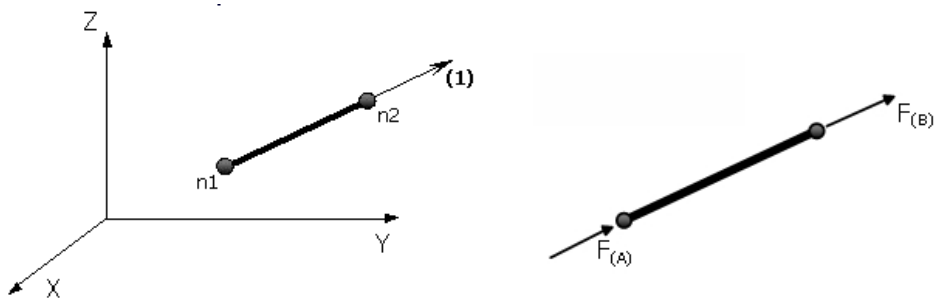
IMPORTANT: Damping defined at element level takes precedence over global damping, that is, the "globally-computed" damping matrix coefficients that are associated to the degrees-of-freedom of a given element will be replaced by coefficients that will have been calculated through the multiplication of the mass matrix of the element by a mass-proportional parameter, or through the multiplication of the element stiffness matrix by a stiffness-proportional parameter, or through the calculation of an element damping Rayleigh matrix.

NOTE 1: Given that no flexure will be present in the element, a much-reduced number of fibres, with respect to the case of infrm elements, needs to be employed in order to warrant accurate results.

NOTE 2: Modelling a rigid floor diaphragm using pinned crossed struts may give rise to unrealistically high axial forces in floor beams. In order to avoid this, one may think of introducing a coincident elfrm element featuring infinite axial stiffness and connected to link elements that would only transmit axial load. In this way, the very rigid element would absorb the axial load, whilst the rotations (hence moments) would be transmitted to the original beam elements.

NOTE 3: If Rayleigh damping is defined at element level, using varied coefficients from one element to the other, or with respect to those employed in the global damping settings, then non-classical Rayleigh damping is being modelled, since Rayleigh damping requires uniform damping definition.

Local axes and output notation are defined in the figure below:

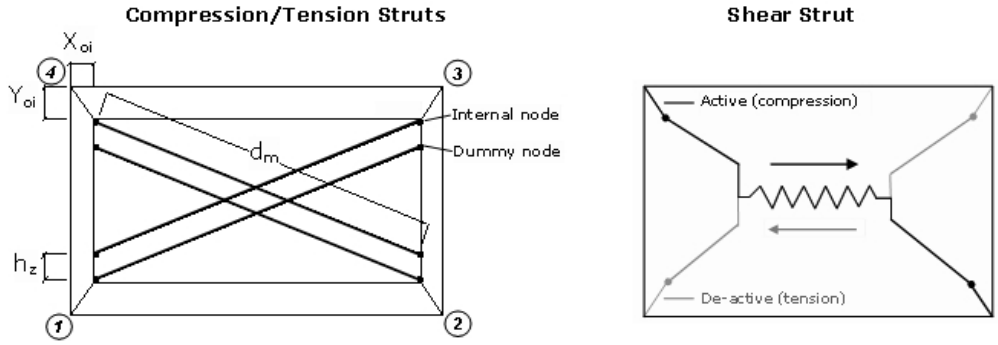


Local Axes and Output Notation

Inelastic infill panel element type - infill

A four-node masonry panel element, developed and initially programmed by Crisafulli [1997] and implemented in SeismoStruct by Blandon [2005], for the modelling of the nonlinear response of infill panels in framed structures. Each panel is represented by six strut members; 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 axial load struts use the masonry strut hysteresis model, while the shear strut uses a dedicated bilinear hysteresis rule.

Also as can be observed in the figure below, four internal nodes are employed to account for the actual points of contact between the frame and the infill panel (i.e. to account for the width and height of the columns and beams, respectively), whilst four dummy nodes are introduced with the objective of accounting for the contact length between the frame and the infill panel. All the internal forces are transformed to the exterior four nodes (which, as noted here, need to be defined in anti-clockwise sequence) where the element is connected to the frame.



NOTE: Although the inelastic infill panel elements have been created for the modelling of the nonlinear response of infill panels in framed structures (as stated above), they might also be employed for the modelling of slabs with a specific stiffness contribution and inelastic behaviour pattern, e.g. by strategically placing these elements and/or inelastic truss elements with properties set to mimic the desired behaviour (as suggested in the Seismosoft Forum).

In order to fully characterise this type of element, the following needs to be defined:

Strut Curve Parameters

Employed in the definition of the masonry strut hysteresis model, which is modelled with the *inf_strut response curve*.

Curve Properties	Typical values	Default values
Initial Young modulus – E_m	$400f_{m\theta}$ - $1000\ f_{m\theta}$ (kPa)	1600000 (kPa)
Compressive strength – $f_{m\theta}$	(see Help System)	1000 (kPa)
Tensile strength – f_t	-	0 (kPa)
Strain at maximum stress – ε_m	0.001 - 0.005 (m/m)	0.0012 (m/m)
Ultimate strain – ε_u	-	0.024 (m/m)
Closing strain – ε_{cl}	0 - 0.003 (m/m)	0.004 (m/m)
Strut area reduction strain – ε_1	0.0003 - 0.0008 (m/m)	0.0006 (m/m)
Residual strut area strain – ε_2	0.0006 - 0.016 (m/m)	0.001 (m/m)
Starting unload. stiffness factor – γ_{un}	1.5 - 2.5 (-)	1.5 (-)
Strain reloading factor – α_{re}	0.2 - 0.4 (-)	0.2 (-)
Strain inflection factor – α_{ch}	0.1 - 0.7 (-)	0.7 (-)
Complete unloading strain factor – β_a	1.5 - 2.0 (-)	1.5 (-)
Stress inflection factor – β_{ch}	0.5 - 0.9 (-)	0.9 (-)
Zero stress stiffness factor – γ_{plu}		1 (-)
Reloading stiffness factor – γ_{pr}		1.5 (-)
Plastic unloading stiffness factor – e_{x1}		3 (-)
Repeated cycle strain factor – e_{x2}		1.4 (-)

Shear Curve Parameters

Employed in the definition of the masonry strut hysteresis model, which is modelled with the *inf_shear response curve*.

Curve Properties	Typical values	Default values
Shear bond strength – τ_0	300 - 600 (kPa) (Hendry, 1990) 100 - 1500 (kPa) (Paulay and Priestley, 1992) 100 - 700 (kPa) (Shrive, 1991)	300 (kPa)
Friction coefficient – μ	0.1 - 1.2 (-)	0.7 (-)
Maximum shear strength – τ_{MAX}	-	600 (kPa)
Reduction shear factor – α_S	1.4 - 1.65 (-)	1.5 (-)

Infill Panel Thickness (t)

It may be considered as equal to the width of the panel bricks alone (e.g. 12 cm), or include also the contribution of the plaster (e.g. 12+2x1.5=15 cm).

Out-of-plane failure drift

Introduced in percentage of storey height, it dictates the de-activation of the element, i.e. once the panel, not the frame, reaches a given out-of-plane drift, the panel no longer contributes to the structure's resistance nor stiffness, since it is assumed that it has failed by means of an out-of-plane failure mechanism.

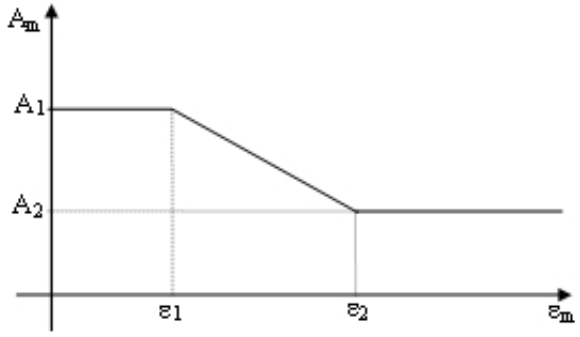
NOTE: Acceleration-triggered de-activation has not been introduced, because it could result very sensitive to high frequency and/or spurious acceleration modes. However, a workaround is nonetheless suggested in note 5, below.

Strut Area 1 (A1)

It is defined as the product of the panel thickness and the equivalent width of the strut (b_w), which normally varies between 10% and 40% of the diagonal of the infill panel (d_m), as concluded by many researchers based on experimental data and analytical results. Indeed, there are numerous empirical expressions, featuring varying degrees of complexity, that have been proposed by different authors [e.g. Holmes, 1961; Stafford-Smith, 1962; Stafford-Smith and Carter, 1969; Mainstone and Weeks, 1970; Mainstone, 1971; Liauw and Kwan, 1984; Decanini and Fantin, 1986; Paulay and Priestley, 1992], and to which the user may refer to for guidance. These have been summarised in the work of Smyrou [2006], where the pragmatic proposals of Holmes [1961] or Paulay and Priestley [1992] of simply assuming a value of b_w which is respectively equal to 1/3 or 1/4 of d_m is suggested as a possible expedite and not necessarily inexact manner of estimating the value of this parameter.

Strut Area 2 (A2)

Introduced as percentage of A1, it aims at accounting for the fact that due to cracking of the infill panel, the contact length between the frame and the infill decreases as the lateral and consequently the axial displacement increases, affecting thus the area of equivalent strut. It is assumed that the area varies linearly as function of the axial strain (see Figure below), with the two strains between which this variation takes place being defined as input parameters of the masonry strut hysteresis model.



Equivalent contact length (h_z)

Introduced as percentage of the vertical height of the panel, effectively yielding the distance between the internal and dummy nodes, and used so as to somehow take due account of the contact length between the frame and the infill panel. Reasonable results seem to be obtained for values of 1/3 to 1/2 of the actual contact length (z), defined by Stafford-Smith [1966] as equal to $0.5\pi\lambda^{-1}$, where λ is a dimensionless relative stiffness parameter computed by the Equation given below, in which E_m is the Elastic Modulus of the masonry, t_w is the thickness of the panel, ϑ is the angle of the diagonal strut with respect to the beams, E_cI_c is the bending stiffness of the columns, and h_w is the height of the infill panel.

$$\lambda = h\sqrt[4]{\frac{E_m t_w \sin(2\vartheta)}{4E_c I_c h_w}}$$

Horizontal and Vertical offsets (X_{oi} and Y_{oi})

Introduced as percentage of the horizontal and vertical dimensions of the panel, they obviously represent the reduction of the latter due to the depth of the frame members. In other words, these parameters provide the distance between the external corner nodes and the internal ones.

Proportion of stiffness assigned to shear (γ_S)

It represents the proportion of the panel stiffness (computed internally by the program) that should be assigned to the shear spring (typically, a value ranging between 0.2 and 0.60 is adopted). In other words, the strut stiffness (K_A) and the shear stiffness (K_S) are computed as follows:

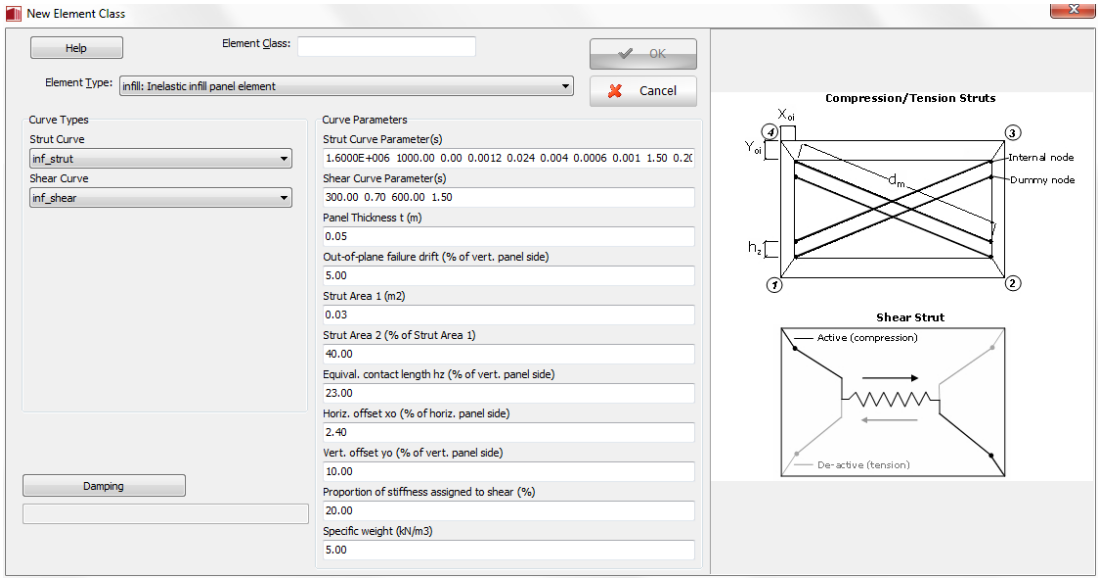
$$K_A = (1-\gamma_s)\frac{A_{ms}E_m}{2d_m} \qquad and \qquad K_S = (\gamma_s)\frac{A_{ms}E_m}{d_m}\cos^2\vartheta$$

Specific weight (γ)

It represents the volumetric weight of the panel (it is recalled that no section, hence no material, is assigned to this element, for which reason the self-weight must be defined here). Default value is 20 kN/m^3.

In this element's dialog box it is also possible to define an element-specific damping, as opposed to the global damping defined in *General* > *Project Settings* > *Damping*. To do so, users need simply to press the *Damping* button and then select the type of damping that better suits the element in question (users should refer to the Damping menu for a discussion on the different types of damping available and hints on which might the better options).

IMPORTANT: Damping defined at element level takes precedence over global damping, that is, the "globally-computed" damping matrix coefficients that are associated to the degrees-of-freedom of a given element will be replaced by coefficients that will have been calculated through the multiplication of the mass matrix of the element by a mass-proportional parameter, or through the multiplication of the element stiffness matrix by a stiffness-proportional parameter, or through the calculation of an element damping Rayleigh matrix.



Definition of a new infill element

NOTE 1: This model (with its struts configuration) is capable of describing only the commonest of modes of failure, since a model that would account for all types of masonry failure would not be practical due to the appreciable level of complexity and uncertainty involved. Users are strongly advised to consult the publications of Crisafulli et al. [2000] and Smyrou et al. [2006] for further details on this model.

NOTE 2: 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). If users wish their infills to resist gravity loads, then they should define the latter as non-initial loads.

NOTE 3: In very refined models, users may wish to introduce link elements between the frame and infill panel nodes, in order to taken into account the fact that the infills are commonly not rigidly connected to the surrounding frames.

NOTE 4: Users may also want to check for values of out-of-plane acceleration exceeding a certain threshold limit that may be inducing out-of-plane failure of the panel.

NOTE 5: The presence of openings in infill panels constitutes an important uncertainty in the evaluation of the behaviour of infilled frames. Several researchers [e.g. Benjamin and Williams, 1958; Fiorato et al., 1970; Mallick and Garg, 1971; Liauw and Lee, 1977; Utku, 1980; Dawe and Young, 1985; Thiruvengadam, 1985; Giannakas et al., 1987; Papia, 1988; Dawe and Seah, 1989; Hamburger, 1993; Bertoldi et al., 1994; CEB, 1996; Mosalam et al., 1997; Gostic and Zarnic, 1999; De Sortis et al., 1999; Asteris, 2003] have investigated the influence that different configurations of openings (in terms of size and location) might have on strength and stiffness. Unfortunately, though somewhat understandably given the large number of variables and uncertainties involved, agreement on this topic has not yet been reached; the above-listed publications have all lead to diverse quantitative conclusions and recommendations. Users will therefore need to resort to their own engineering judgement and experience, coupled with a thorough consultation of the literature on this topic (a small percentage of it has been listed above), in order to decide on how the presence of openings in the structure being studied should be taken into account. As an expedite recommendation, we might perhaps suggest that the effect of openings on the response of an infilled frame can be pragmatically taken into account by reducing the value of the Strut Area (A1), and hence of the panel's stiffness, in proportion to the area of the opening with respect to the panel. That is, as shown by Smyrou et al. [2006], if a given infill panel features openings of 15% to 30% with respect to the area of the panel, good response predictions might be obtained by reducing the value of A1 (i.e. its stiffness) by a value that varies between 30% and 50%. As far as the strength of the infill panel is concerned, and given the extremely varied nature of the observations made on this issue by past researchers, we would perhaps suggest that, in the absence of good evidence otherwise, users should not change its value to take into account the presence of standard openings (i.e. openings that are not larger than 30% of the area of the infill panel).

NOTE 6: Users are also warmly advised to read the publication of Celarec and Dolšek [2012] in which the effects of masonry infills on the shear demand and failure of columns, for the case when reinforced concrete frames with such infills are modelled by means of simplified nonlinear models that are not capable of the direct simulation of these effects, have been investigated.

Masonry element type - masonry

This element is combination of a 3D, force-based, plastic hinge element type employed in modelling mainly the bending behaviour of the masonry member (herein mentioned as the 'internal sub-element') with two links at the two edges that are employed to simulate the shear behaviour of the member (herein referred to as the 'external links' or the 'link sub-elements'). The internal sub-element and the external links are connected in series, ensuring equilibrium in bending moment and shear force. The only 'active' degrees-of-freedom of the link sub-elements are the two translational ones in the shear directions (in-plane and out-of-plane), whilst the other four DOFs (axial and 3 rotational) remain perfectly rigid links. Both masonry walls and spandrels can be accurately modelled with such configuration.

The shear DOFs of the link sub-elements feature a hysteretic curve that is based on SeismoStruct's built-in MIMK_pinched nonlinear curve (Modified Ibarra-Medina-Krawinkler deterioration curve with bilinear hysteretic rules and pinching), according to a phenomenological law that describes the shear

behaviour of the entire member. Simultaneously, in the internal sub-element the fiber-section modelling allows for a relatively accurate description of the coupled axial-flexural behaviour. 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 (as described in the Material inelasticity section).

The determination of the shear strength of the member is crucial in the model accuracy, and is automatically carried out by the model, based on the masonry material properties, the dimensions of the member, and the selected Structural Code. The following expressions are employed for the calculation of the member shear capacity (it is noted that different equations are employed in the different Standards).

- In Eurcode 8, the failure mechanism of bed joint sliding is employed:
  $V_f = f_{vd} D' t$ (C.2) EC8: Part 3

- In ASCE 41-23 and for unreinforced masonry URM walls, the lower value of the bed-joint sliding strength and the diagonal tension strength is used, according to the following expressions:

  $Q_{CE} = V_{bjs1} = v_{me} A_n$ (11-10) ASCE 41-23

  $Q_{CL} = V_{dt} = f'_{dt} A_n \beta \sqrt{1 + \frac{f_\alpha}{f'_{dt}}}$ (11-13) ASCE 41-23

- In ASCE 41-23 and for unreinforced masonry URM spandrels, the lower value of the bed-joint strength and the diagonal tension strength is used, according to the following expressions:

  $V_{s1} = \frac{2}{3}(c_{bj} + \mu_f p_{sp}) h_{sp} b_{sp}$ (11-18) ASCE 41-23

  $V_{s2} = f'_{dt} \beta_{sp} h_{sp} b_{sp} \sqrt{1 + \frac{p_{sp}}{f'_{dt}}}$ (11-19) ASCE 41-23

- In ASCE 41-17 and for masonry walls or spandrels with reinforcement, the following expression is employed:

  $V_n = (V_{nm} + V_{ns}) \gamma_g$ (Equation 9-21) TMS 402

- In NTC-18 and for unreinforced masonry URM, different expressions are used for the different material types: (i) in masonry with blocks the Mohr-Coulomb criterion is employed, (ii) in masonry with bricks or regular stones, the strength assumes the lower value of the Turnšek-Čačovič and the Mann-Müller criteria and (iii) for masonry with irregular stones the Turnšek-Čačovič expression is employed.

  Mohr-Coulomb (§7.8.2.2.2 NTC-18 and §6.2 EC6): $V_t = f_{vd} l' t$

  Turnšek-Čačovič (§C8.7.1.3.1.1 Commentary of NTC-18):

  $$V_t = \frac{1.5\tau_{0d} l t}{b} \sqrt{1 + \frac{\sigma_0}{1.5\tau_{0d}}} = \frac{f_{td} l t}{b} \sqrt{1 + \frac{\sigma_0}{f_{td}}}$$

  Mann-Müller (§C8.7.1.3.1.1 Commentary of NTC2018): $V_t = \frac{lt}{b}\left(\frac{f_{v0d}}{1+\mu\phi} + \frac{\mu}{1+\mu\phi}\sigma_0\right) \leq V_{t,lim}$

- In NTC-18 and for reinforced masonry RM, the shear capacity is calculated according to §7.8.3.2.2 NTC-18:
  $V_t = V_{t,M} + V_{t,S}$ (7.8.7) NTC-18
  where:
  $V_{t,M} = d \cdot t \cdot f_{vd}$ (7.8.8) NTC-18

and

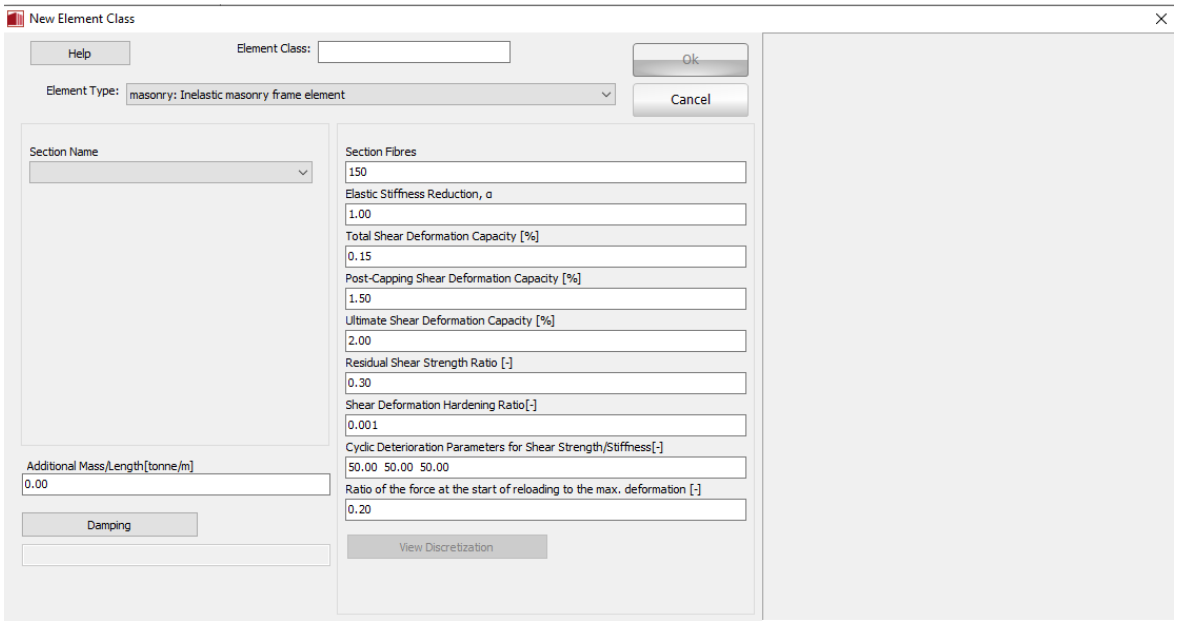
$$V_{t,S} = (0{,}6 \cdot d \cdot A_{sw} \cdot f_{yd})/s \qquad (7.8.9)\ \text{NTC-18}$$

Note that in for the checks according to KANEPE the expressions of EC8 are employed. Similarly, with TBDY the ASCE 41 equations are used. For a complete description of the employed expressions in each Standard, users may refer to the Appendices H1 to H6.

Two special masonry material models may be used with the masonry element type, mas_par and mas_tl. The former is based on the con_ma concrete material model, whilst the latter is a simpler multi-linear model with degradation and residual strength. Both models feature parameters, such as the compressive, the shear and the tensile strength, which are employed in the determination of the member shear strength. Similarly, there are two masonry specific section types in SeismoStruct, one for walls mws, and one for spandrels mss. Both unreinforced URM and reinforced RM masonry members can be effectively be modelled with the proposed features.

NOTE: The masonry element type can only be used with the mas_par and the mas_tl material models. Similarly, only the special masonry section types mws and mss can be employed. The reason for this, is that these can store parameters that are used for the automatic calculation of the member shear strength.

Users may choose whether to calculate the masonry shear strength (i) only at the initial step, (ii) at all the steps until yielding in shear or (iii) at every step, i.e. even after reaching the peak member capacity. The default option is the second, that is to update the shear strength until yield, which is the best combination of accuracy and stability, since updating the shear strength in the descending branch of the capacity curve may lead to convergence difficulties without significantly improving the accuracy of the solution.

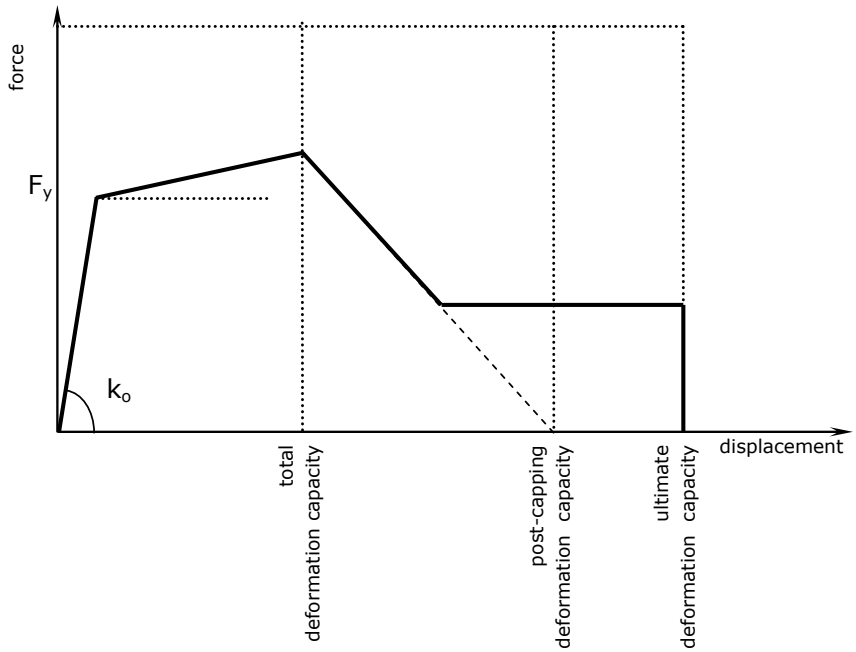


Definition of a new masonry element

The parameters required for the full definition of the element properties are the following (see figure above):

- The number of section fibres used in equilibrium computations carried out at each of the integration sections of the internal sub-element.
- The elastic stiffness reduction α is the reduction of the elastic uncracked stiffness of the shear force-deformation curve that is employed in the calculations

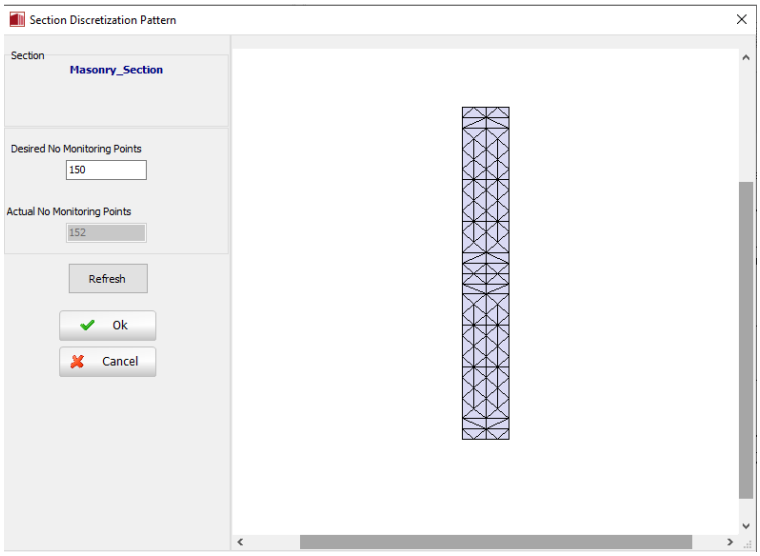
- The total shear deformation capacity, which is the ultimate deformation capacity of the member $d_{tot}=d_{yield}+d_{plastic}$. It is noted that the deformation at yield d_{yield} is directly calculated by the program from the elastic shear stiffness and the yield strength.
- The post-capping shear deformation capacity: this is the deformation level, at which the extrapolation of the descending branch of the shear force-deformation curve reaches the zero axis.
- The ultimate shear deformation capacity: this is the deformation level, after which there is no residual strength
- The residual shear strength ratio is the ratio between the maximum strength (at the total deformation capacity level) and the residual strength
- The shear deformation hardening ratio is the ratio between the elastic and the plastic branches of the shear force-deformation curve
- The cyclic deterioration parameters for the shear strength and stiffness are the following three parameters: (i) the cyclic deterioration parameter for strength deterioration and accelerated reloading deterioration – Λs, Λα, (ii) the cyclic deterioration parameter for unloading stiffness deterioration – ΛK, and (iii) the cyclic deterioration parameter for post-capping strength deterioration– Λc. For all parameters, the smaller the factor, the larger the imposed deterioration on the curve, however note that a zero value leads to no deterioration. For a complete description of the parameters refer to the documentation of the MIMK_bilin curve
- Ratio between force at start of reloading to the force corresponding to the maximum experienced deformation for positive and negative loading directions.



Shear force-deformation curve employed in the masonry model

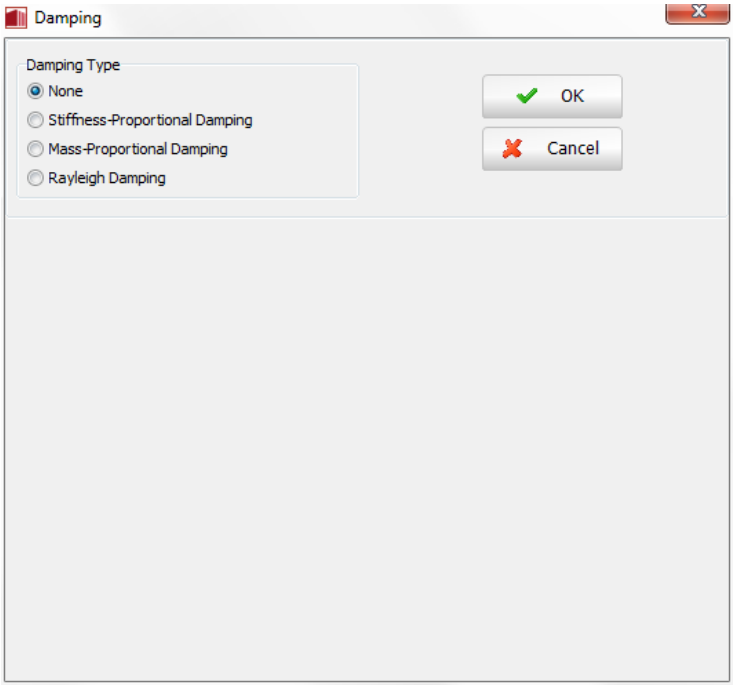
Regarding the section fibres, the ideal number, sufficient to guarantee an adequate reproduction of the stress-strain distribution across the element's cross-section, varies with the shape and material characteristics of the latter, and the degree of inelasticity to which the element will be forced to. As a crude rule of thumb, users may consider that usually 100 fibres should be adequate.

In the *Section Discretization Pattern* dialog box the software provides the desired and the actual (after the section discretization has been performed, employing triangulation procedures) number of monitoring points that will be employed in the analysis. By clicking on the *Refresh* button, it is possible to update the view of the section discretization.



Section discretization to fibres

In this element's dialog box it is also possible to define an element-specific damping, as opposed to the global damping defined in *General* > *Project Settings* > *Damping*. To do so, users need simply to press the *Damping* button and then select the type of damping that better suits the element in question (users should refer to the Damping menu for a discussion on the different types of damping available and hints on which might the better options).

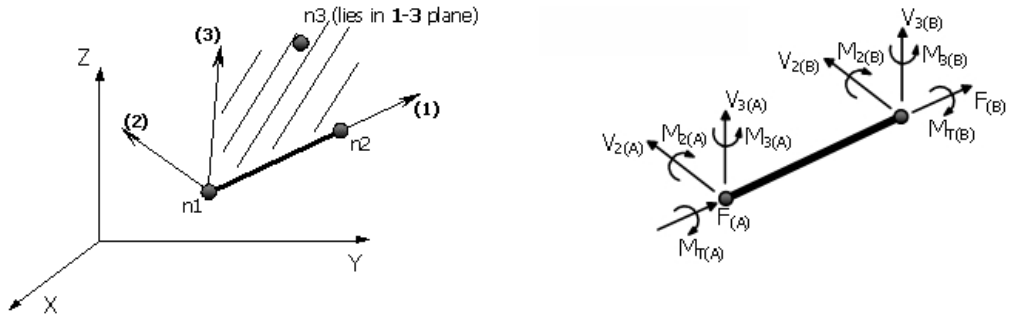


Definition of an element-specific damping

IMPORTANT: Damping defined at element level takes precedence over global damping, that is, the "globally-computed" damping matrix coefficients that are associated to the degrees-of-freedom of a given element will be replaced by coefficients that will have been calculated through the multiplication of the mass matrix of the element by a mass-proportional parameter, or through the multiplication of the element stiffness matrix by a stiffness-proportional parameter, or through the calculation of an element damping Rayleigh matrix.

NOTE: If Rayleigh damping is defined at element level, using varied coefficients from one element to the other, or with respect to those employed in the global damping settings, then non-classical Rayleigh damping is being modelled, since Rayleigh damping requires uniform damping definition.

Local axes and output notation are defined in the figure below. Refer to the discussion on global and local frame axes in Appendix A for a detailed description on the determination of the orientation of the local element axis system.

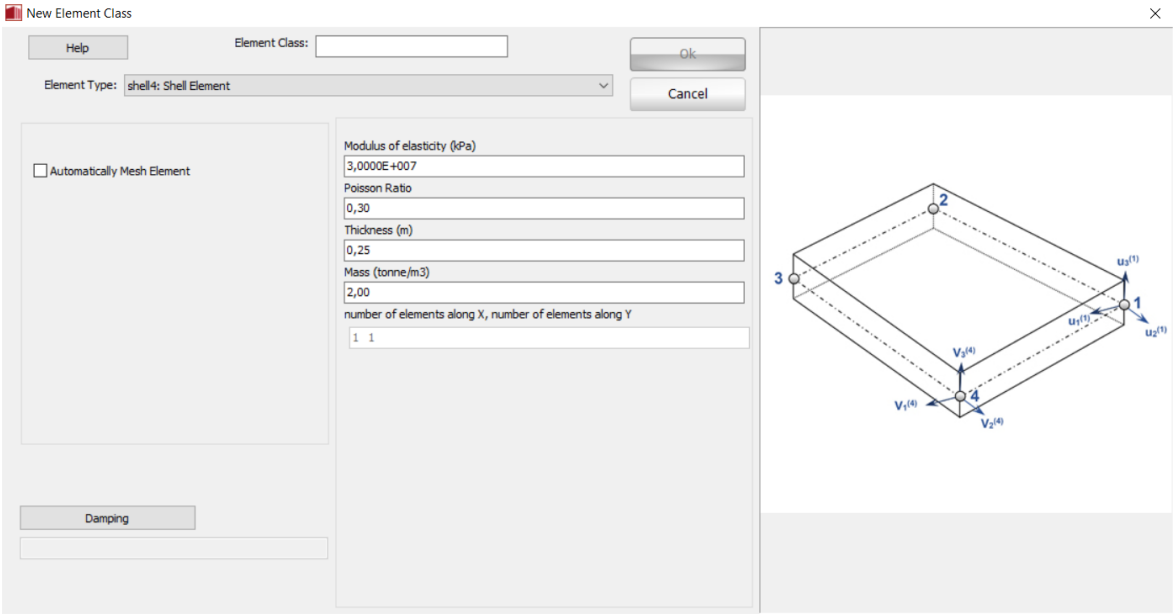


Local Axes and Output Notation for masonry elements

Shell element type – shell4

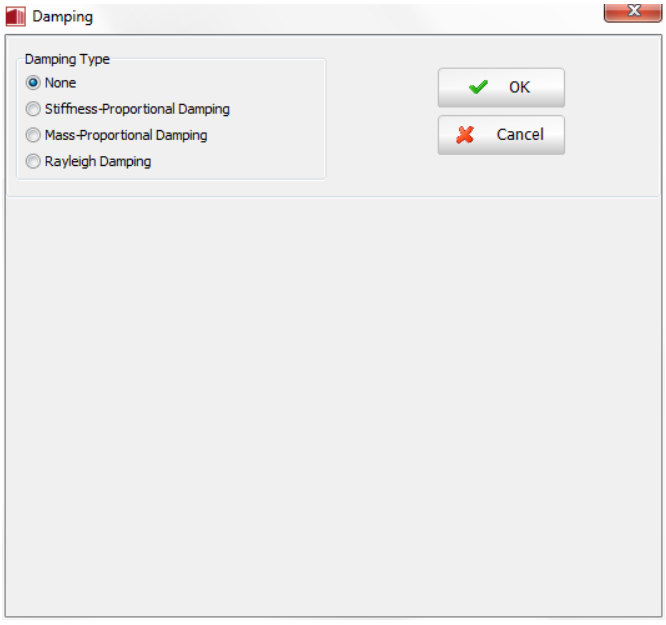
This element is a 3D Shell element with 4 nodes (quadrilateral shell element). It follows the mixed-interpolated tensorial components (MITC) element formulation as described by Dvorkin and Bathe (1984). The element combines plate-bending and membrane behaviour. The shell element can be used for the modelling of structural elements like floors/plates (applicable both for thin and thick plates) and walls and in general surfaces carrying in-plane and out-of-plane loading. The element is elastic with six degrees of freedom per node.

In order to fully define the shell4 element the user needs to specify the modulus of elasticity the Poisson Ratio, the thickness and the distributed mass of the element. There is also the option to discretise an element by checking the corresponding option and defining into how many sub-elements the element will be discretised in each direction.



Definition of a new shell4 element

In this element's dialog box it is also possible to define an element-specific damping, as opposed to the global damping defined in *General>Project Settings>Damping*. To do so, users need simply to press the *Damping* button and then select the type of damping that better suits the element in question (users should refer to the Damping menu for a discussion on the different types of damping available and hints on which might the better options).

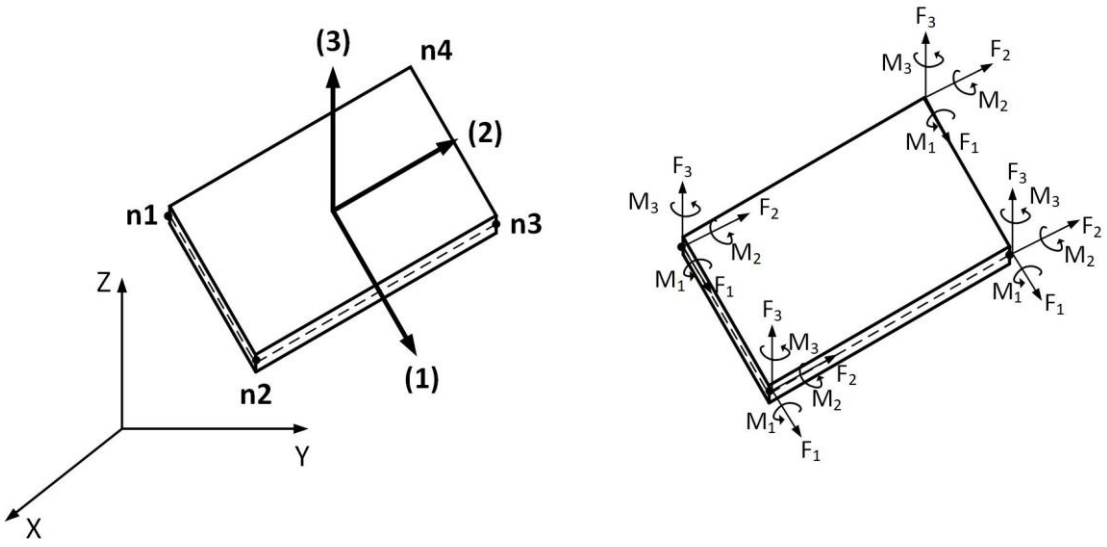


Definition of an element-specific damping

IMPORTANT: Damping defined at element level takes precedence over global damping, that is, the "globally-computed" damping matrix coefficients that are associated to the degrees-of-freedom of a given element will be replaced by coefficients that will have been calculated through the multiplication of the mass matrix of the element by a mass-proportional parameter, or through the multiplication of the element stiffness matrix by a stiffness-proportional parameter, or through the calculation of an

NOTE: If Rayleigh damping is defined at element level, using varied coefficients from one element to the other, or with respect to those employed in the global damping settings, then non-classical Rayleigh damping is being modelled, classing Rayleigh damping requires uniform damping definition.

Local axes and output notation are defined in the figure below. In general local axis (1) is aligned with the node 1 – node 2 direction, local axis (2) is perpendicular to axis (1) in the element plane and local axis (3) is perpendicular to the plane formed by axis (1) and axis (2).



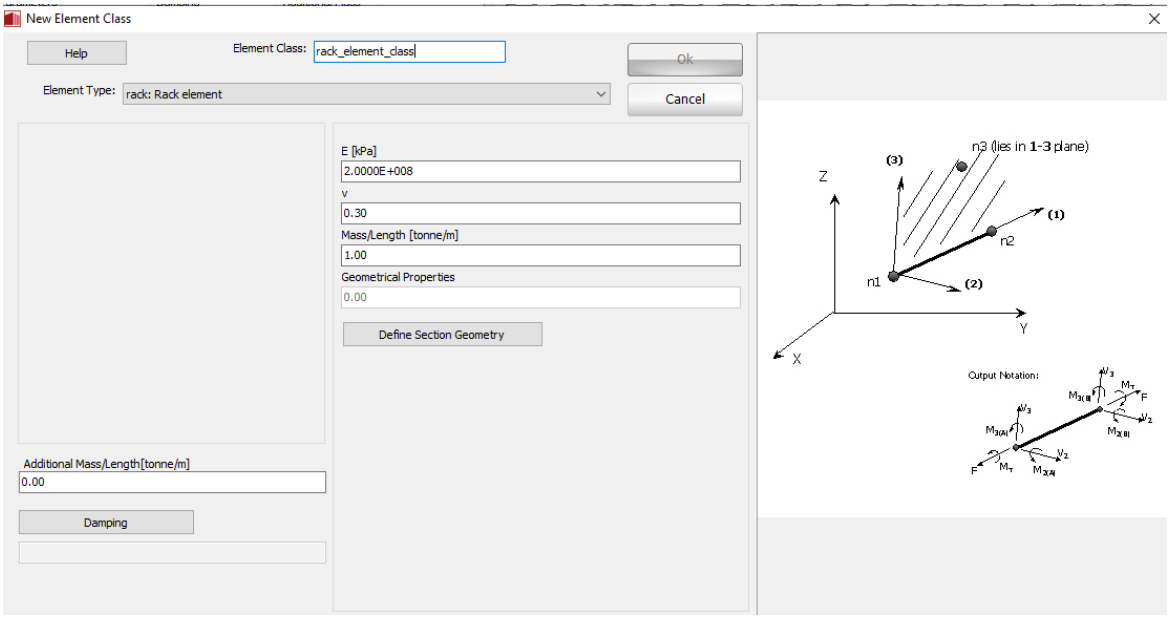
Local Axes and Output Notation for shell4n elements

Rack element type - rack

This element is a 3D beam element with thin-walled, open, cross-sections. The element is characterized by seven degrees of freedom per node, so as to correctly estimate both the displacements and the internal stresses, including warping displacements and bi-moment stresses, and to correctly predict the flexural-torsional and lateral-torsional buckling, derived by the coupling between flexure and torsion. Furthermore, the model accounts for the eccentricity of the shear centre from section centroid, and it considers all the Wagner coefficients, which makes it suitable for use with non-symmetric cross-sections.

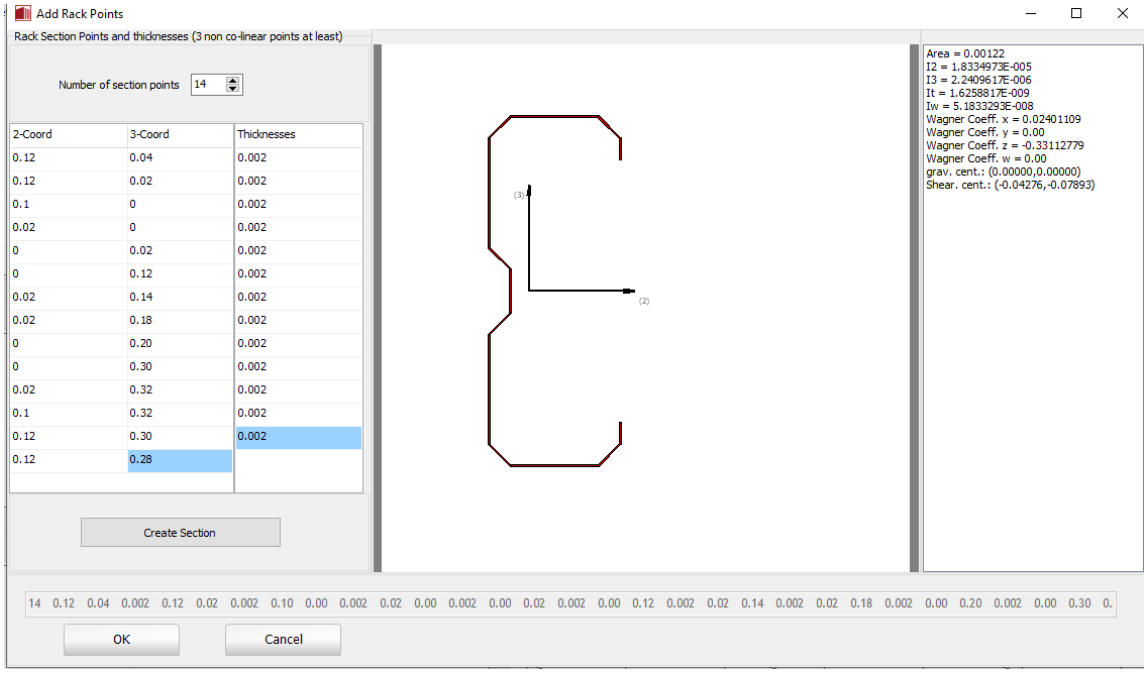
As a result, the formulation is ideal for the modelling of steel storage pallet racks, as well as scaffolding structures, which are generally composed by uprights which have mono-symmetric lipped channel cross-sections.

The rack element can be fully defined, if the (elastic) material properties (modulus of elasticity and Poisson ratio) and the section configuration are provided. The former are given on the main dialog box of the rack element class (see figure below).



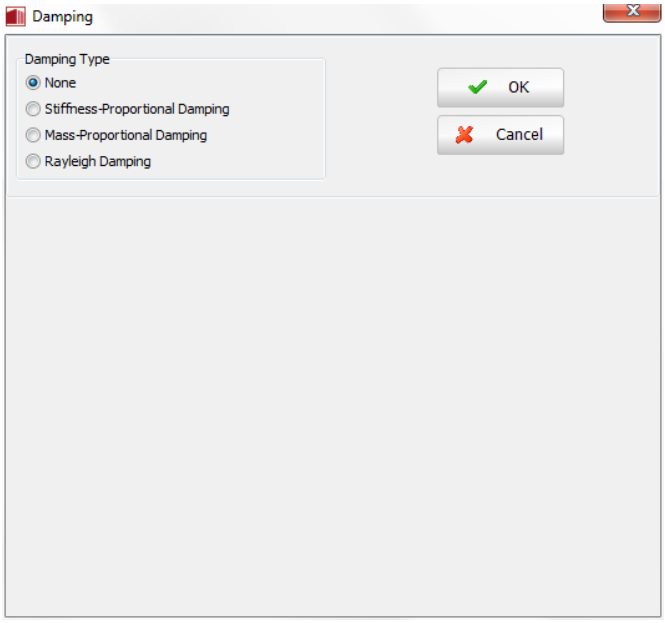
Definition of a new rack element

The section geometry can be defined in a dialog box. Any thin-walled open section configuration can be modelled, and different thicknesses may be assigned at the different parts of the section. After the user defines the coordinates of the corner points of the section, and clicks on the Create Section button, the section is shown on the screen and the elastic section properties, the Wagner coefficients and the position of the shear centre are automatically calculated.



Definition of the section configuration for the rack element

In this element's dialog box it is also possible to define an element-specific damping, as opposed to the global damping defined in *General* > *Project Settings* > *Damping*. To do so, users need simply to press the *Damping* button and then select the type of damping that better suits the element in question (users should refer to the Damping menu for a discussion on the different types of damping available and hints on which might the better options).

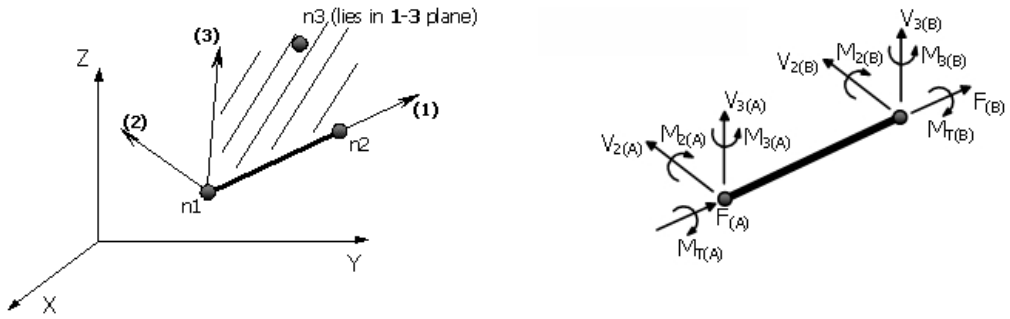


Definition of an element-specific damping

IMPORTANT: Damping defined at element level takes precedence over global damping, that is, the "globally-computed" damping matrix coefficients that are associated to the degrees-of-freedom of a given element will be replaced by coefficients that will have been calculated through the multiplication of the mass matrix of the element by a mass-proportional parameter, or through the multiplication of the element stiffness matrix by a stiffness-proportional parameter, or through the calculation of an element damping Rayleigh matrix.

NOTE: If Rayleigh damping is defined at element level, using varied coefficients from one element to the other, or with respect to those employed in the global damping settings, then non-classical Rayleigh damping is being modelled, classing Rayleigh damping requires uniform damping definition.

Local axes and output notation are defined in the figure below. Refer to the discussion on global and local frame axes in Appendix A for a detailed description of the determination of the orientation of the local element axis system.



Local Axes and Output Notation for rack elements

Rack Element with hinges - rackH

This element is a 3D beam element with thin-walled, open, cross-sections similarly to the rack element but with lumped plasticity at the element ends. The element is characterized by seven degrees of

freedom per node, so as to correctly estimate both the displacements and the internal stresses, including warping displacements and bi-moment stresses, and to correctly predict the flexural-torsional and lateral-torsional buckling, derived by the coupling between flexure and torsion. Furthermore, the model allows to model inelastic behaviour through nonlinear springs at the two ends of the element. As a result, during the element definition, a response curve describing the moment-deformation relationship for each of the two rotational degrees of freedom of each element edge is required.

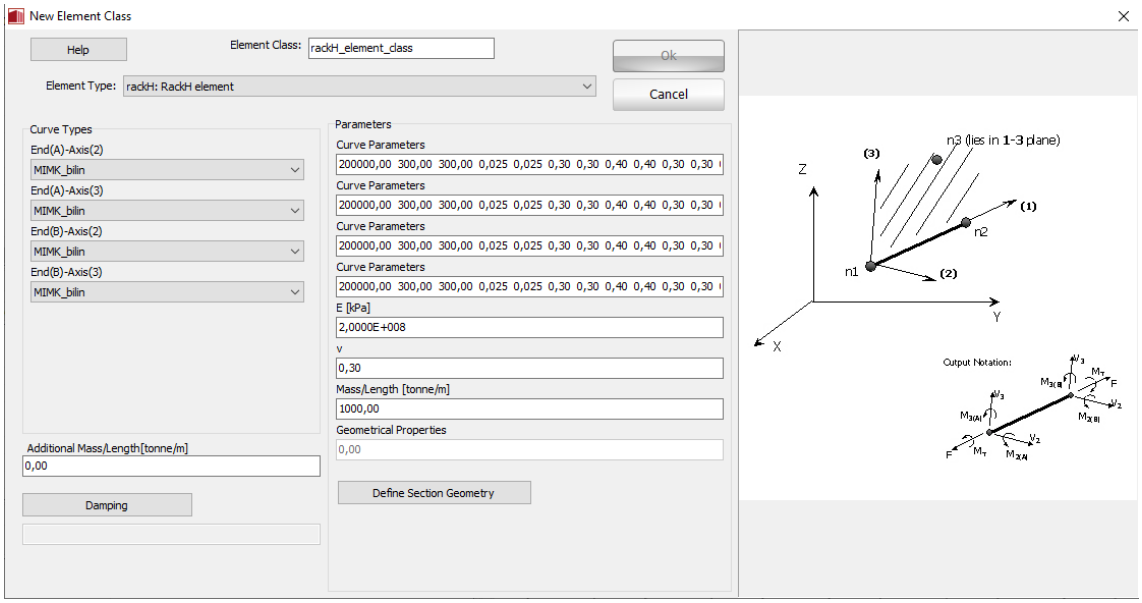
Currently, twenty-nine response curves are available, selectable from within the Element Class dialog box, whenever a rackH element type is selected.

- Linear symmetric curve - lin_sym
- Linear asymmetric curve - lin_asm
- Bilinear symmetric curve - bl_sym
- Bilinear asymmetric curve - bl_asm
- Bilinear kinematic hardening curve - bl_kin
- Trilinear symmetric curve - trl_sym
- Trilinear asymmetric curve - trl_asm
- Quadrilinear symmetric curve - quad_sym
- Quadrilinear asymmetric curve - quad_asm
- Pinched asymmetric curve - pinched_asm
- Modified Ibarra-Medina-Krawinkler Deterioration curve with Bilinear Hysteretic Response – MIMK_bilin
- Modified Ibarra-Medina-Krawinkler Deterioration Model with Peak-Oriented Hysteretic Response – MIMK_peak
- Modified Ibarra-Medina-Krawinkler Deterioration Model with Pinched Hysteretic Response – MIMK_Pinched
- Nonlinear elastic curve - Non_lin_Elast
- Plastic curve – plst
- Simplified bilinear Takeda curve –Takeda
- Asymmetric bilinear Takeda curve –Takeda_asm
- Ramberg Osgood curve - Ramberg_Osgood
- Modified Richard-Abbott curve - Richard_Abbott
- Soil-structure interaction curve - ssi_py
- Gap-hook curve - gap_hk
- Multi-linear curve – multi_lin
- Smooth curve – smooth
- Viscous Damper – vsc_dmp
- Bouc Wen curve - Bouc_Wen
- Elastic – Perfectly plastic Gap curve - gap_elpl
- Impact response curve - pound_hz
- Self Centering Brace response curve - scb
- Generic Hysteretic Curve - gen_hyst

For a comprehensive description of the available response curves associated to the rack element with hinges element refer to Appendix G.

Finally, as in the case of the simple rack element, the model accounts for the eccentricity of the shear centre from section centroid, and it considers all the Wagner coefficients, which makes it suitable for use with non-symmetric cross-sections.

The rackH element can be fully defined, if the rotational DOF response curves, the (elastic) material properties (modulus of elasticity and Poisson ratio) and the section configuration are provided. The former are given on the main dialog box of the rack element class (see figure below). Finally, the rackH element section geometry needs to be defined. The section geometry is defined similarly to the rack element.



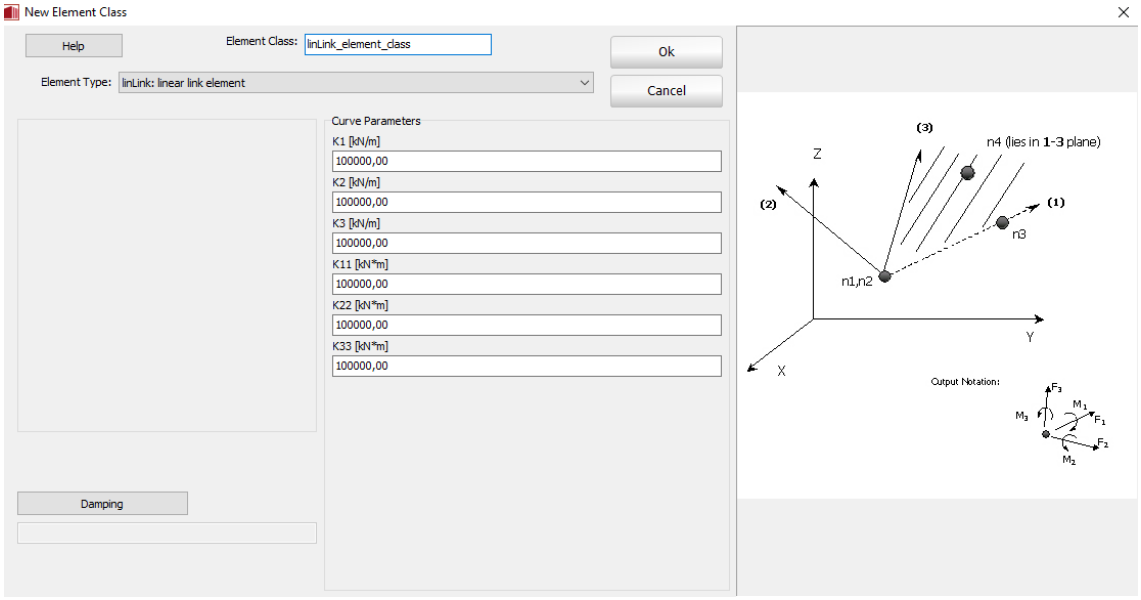
Definition of a new rackH element

It is also possible to define an element-specific damping, as opposed to the global damping defined in *General>Project Settings>Damping*, as described in the sections of the rack element. Finally, local axes and output notation are the same with the rack element.

LINK ELEMENT TYPES

Linear Link element type - linLink

These are the 3D link elements with uncoupled axial, shear and moment actions. All six local Degrees of Freedom of the element (F1, F2, F3, M1, M2, M3) follow a linear behaviour. As a result the user only needs to define the elastic stiffness K for each of the six Local Degrees of Freedom.



Definition of a new linLink element

The linLink elements connect two initially coincident structural nodes.

It is also possible to define an element-specific damping, as opposed to the global damping defined in *General*>*Project Settings*>*Damping*, as described in the sections of the NLLink element. Finally, local axes and output notation are the same with the NLLink element.

Nonlinear Link element type - NLlink

These are the 3D link elements with uncoupled axial, shear and moment actions that can be used to model, for instance, pinned or flexible beam-column connections, structural gapping/pounding, energy dissipating devices, bridge bearings, inclined supports, base isolation, foundation flexibility, and so on.

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).

Currently, twenty-nine response curves are available, selectable from within the Element Class dialog box, whenever a nonlinear link element type is selected.

- Linear symmetric curve - lin_sym
- Linear asymmetric curve - lin_asm
- Bilinear symmetric curve - bl_sym
- Bilinear asymmetric curve - bl_asm
- Bilinear kinematic hardening curve - bl_kin
- Trilinear symmetric curve - trl_sym
- Trilinear asymmetric curve - trl_asm
- Quadrilinear symmetric curve - quad_sym
- Quadrilinear asymmetric curve - quad_asm
- Pinched asymmetric curve - pinched_asm
- Modified Ibarra-Medina-Krawinkler Deterioration curve with Bilinear Hysteretic Response – MIMK_bilin
- Modified Ibarra-Medina-Krawinkler Deterioration Model with Peak-Oriented Hysteretic Response – MIMK_peak
- Modified Ibarra-Medina-Krawinkler Deterioration Model with Pinched Hysteretic Response – MIMK_Pinched
- Nonlinear elastic curve - Non_lin_Elast
- Plastic curve – plst
- Simplified bilinear Takeda curve – Takeda
- Asymmetric bilinear Takeda curve – Takeda_asm
- Ramberg Osgood curve - Ramberg_Osgood
- Modified Richard-Abbott curve - Richard_Abbott
- Soil-structure interaction curve - ssi_py
- Gap-hook curve - gap_hk
- Multi-linear curve – multi_lin
- Smooth curve – smooth
- Viscous Damper – vsc_dmp
- Bouc Wen curve - Bouc_Wen
- Elastic – Perfectly plastic Gap curve - gap_elpl
- Impact response curve - pound_hz
- Self Centering Brace response curve - scb
- Generic Hysteretic Curve - gen_hyst

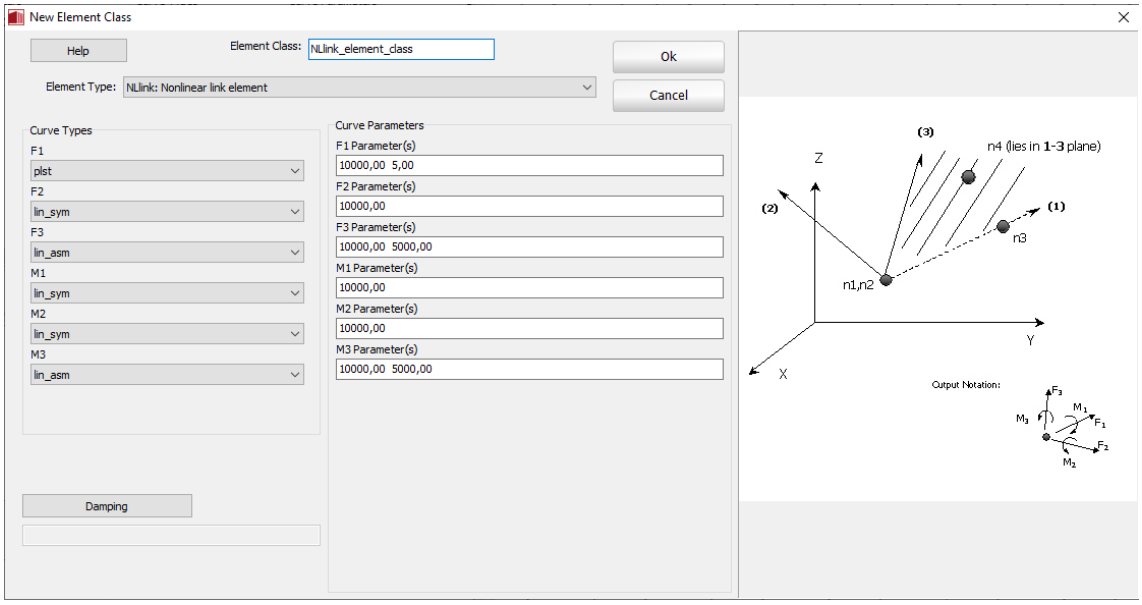
For a comprehensive description of the available response curves associated to the nonlinear link element refer to Appendix G.

In the Nonlinear Link element's dialog box it is also possible to define an element-specific damping, as opposed to the global damping defined in *General* > *Project Settings* > *Damping*. To do so, users need simply to press the *Damping* button and then select the type of damping that better suits the element in

question (users should refer to the Damping menu for a discussion on the different types of damping available and hints on which might the better options).

IMPORTANT: Damping defined at element level takes precedence over global damping, that is, the "globally-computed" damping matrix coefficients that are associated to the degrees-of-freedom of a given element will be replaced by coefficients that will have been calculated through the multiplication of the mass matrix of the element by a mass-proportional parameter, or through the multiplication of the element stiffness matrix by a stiffness-proportional parameter, or through the calculation of an element damping Rayleigh matrix.

The *element-specific damping* facility is typically used here to model radiation damping in soil-structure interaction springs (featuring varied force-displacement rules, such as ssi_py or any other response curve), thus avoiding the need for introducing parallel dashpot elements.



Definition of a new NLlink element

NOTE 1: Only the response curves that have been previously activated in the Constitutive Model tab window (*Tools > Project Settings > Constitutive Model*) can be selected from the drop-down menu and associated to a nonlinear link element

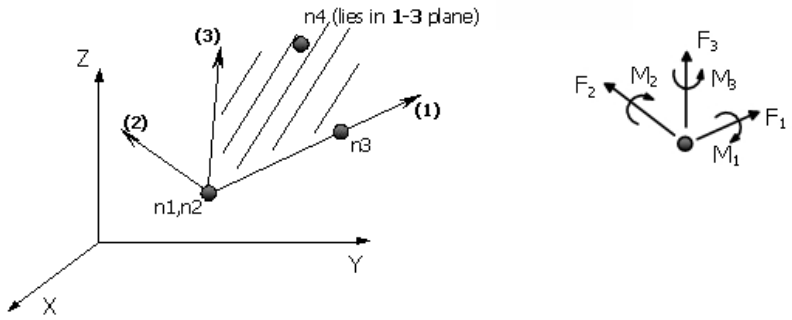
NOTE 2: When a nonlinear link element is introduced between two initially coincident nodes, a force-displacement relationship must compulsorily be defined for all six degrees-of-freedom, including those for which the response of the two nodes is identical. The latter are usually modelled by the adoption of linear response curves with very large stiffness values, so as to guarantee no relative displacement between the two nodes in that particular degree-of-freedom. The very large value to be adopted in such cases depends very much on the type of the analysis being carried out and on the order of magnitude of results being obtained. Too low a value will not reproduce infinitely stiff connection conditions, whilst a value that is too large may lead too numerical difficulties, especially when a force-based convergence criterion is adopted. Usually, and as a rule of thumb, users should consider a stiffness value that is 100 to 250 times larger than that of adjacent elements, noting however that only a sensitivity study will permit the determination of the optimum value.

NOTE 3: On some analyses, the adoption of K_0 = 0 to model pinned joint conditions may lead to difficulties in getting the analysis to converge. This usually can be easily solved by the adoption of a non-zero but still small value of stiffness (e.g. 0.001). Should the user wish to optimise the model (i.e. find the smallest possible stiffness value that will not give rise to accentuated numerical difficulties), then a sensitivity study ran on a case-by-case basis is highly recommended.

NOTE 4: If Rayleigh damping is defined at element level, using varied coefficients from one element to the other, or with respect to those employed in the global damping settings, then non-classical Rayleigh damping is being modelled, since classic Rayleigh damping requires uniform damping definition.

NOTE 5: Damping is here typically coupled with link elements for the introduction of Soil-Structure Interaction springs adequate for dynamic analysis (see also ssi_py response curve).

Local axes and output notation are defined in the figure below:



Local Axes and Output Notation

SSI Macro-element for Shallow Footings – ssilink1

This element is a nonlinear macro-element model for soil-structure interaction of shallow foundations based on the work of Correia and Paolucci (2019). This macro-element approach reduces the size of the problem significantly, since the footing and the soil are considered as a single macro-element characterized by six degrees-of freedom (6 DOFs), in the 3D case, whose formulation is based on the resultant forces and displacements. The geometry considered herein corresponds to a rectangular rigid footing, with coupling between all the macro-element DOFs and its definition as a single zero-length link element.Considering a planar loading, for simplicity of notation and visualisation, the footing will be subjected to a rocking moment and to both vertical and horizontal forces (My, N and Hx respectively), as depicted in Figure 1.

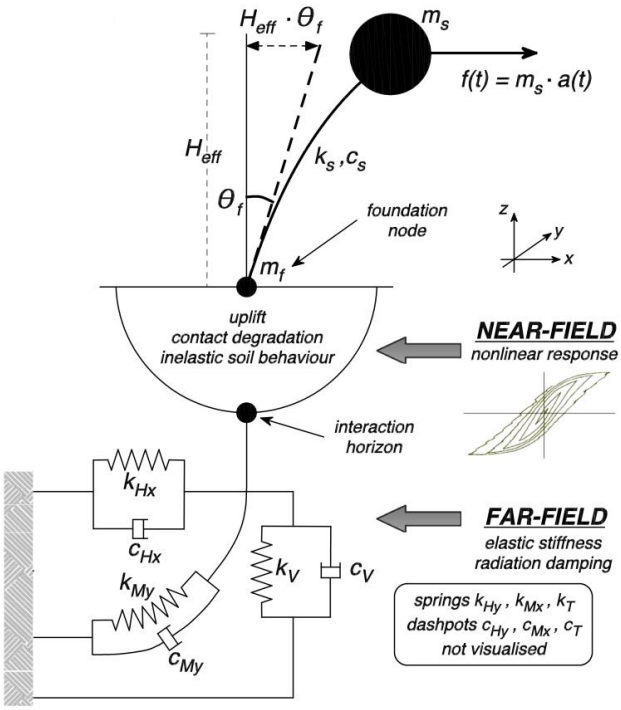


Figure 1. Nonlinear shallow foundation macro-element

An uplift model is adopted which is based on a nonlinear elastic-uplift response which also considers some degradation of the contact at the soil/footing interface due to irrecoverable changes in its geometry. A bounding surface plasticity model is also used which correctly takes into account the simultaneous elastic-uplift and plastic nonlinear responses. Finally, this macro-element formulation is fully applicable to three-dimensional loading cases. Figure 2 schematically represents the type of phenomena intended to be modelled with the macro-element.

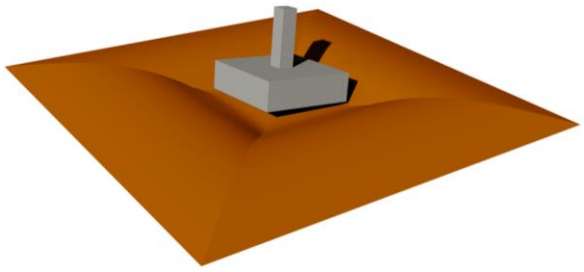


Figure 2. Schematic footing response in 3D case, accounting for uplift, inelasticity and contact degradation

The footing macro-element model represents the dynamic behaviour of isolated rigid footings, subjected to three-dimensional inertial loading, from the initial stages of loading up until reaching failure. The macro-element is based on the three major features of the response of footings, namely:

i) Initial elastic response,
ii) Uplift in rocking response,
iii) Failure loading conditions.

The bounding surface plasticity model is used to represent a continuous transition between the initial elastic response and the plastic flow at failure, for monotonic, cyclic and dynamic loading conditions. The uplift phenomenon is represented by a nonlinear elastic model which, however, takes into account and is influenced by the plastic deformation state in the underlying soil.

The bounding surface adopted in this macro-element depends on the type of soil and its drainage conditions during a seismic event. Therefore, different 3D failure surfaces are considered for drained and undrained conditions. The ultimate surface adopted to describe the drained behaviour corresponds to the “rugby-ball” shape, while for undrained loading the ultimate surface corresponds to the so-called "scallop" shape, which is represented in Figure 3 in terms of its intersection in the H-N and M-N planes of loading. The "rugby-ball" shape corresponds to have the ultimate surface represented by the continuous line in both planes of loading.

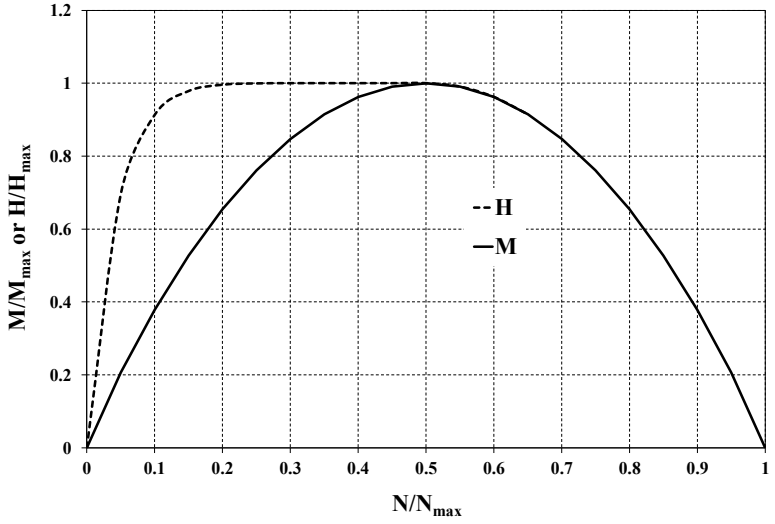


Figure 3. Scallop-shaped failure surface for undrained conditions

The macro-element model requires the definition of 25 input parameters, from which only 3 need to be calibrated. The model parameters along with their definition and suggested values are given in Table 1. They correspond to:

Two Geometric parameters

The footing dimensions (length, L, and width, B).

Twelve Elastic impedances parameters

The six foundation initial stiffness components, indicated as KN1, KH2, KH3, KM2, KM3, KM2, KTT for vertical, horizontal and rotational directions, respectively, can be evaluated by using formulas from literature (e.g. Gazetas, 1991), or calibrated based on test results. The same applies to the corresponding six equivalent dashpot coefficients for radiation damping representation.

Six Strength parameters

These characterise the failure surface and are defined as:

-the maximum centred vertical load capacity Nmax that corresponds to the ultimate static bearing capacity of the foundation and can be evaluated by standard superposition formulas (e.g. Brinch-Hansen,1970);

- the maximum base shear capacities Hmax2 and Hmax3 and maximum base moment capacities Mmax2, Mmax3, Tmax, which can be calibrated based either on material properties (e.g. soil friction angle) or on theoretical values.

Five Model specific parameters

Characterised as follows:

- the uplift initiation parameter, α, is only dependent on the assumed stress distribution of vertical stresses underneath the foundation and can be determined from simple static considerations, as shown in

	Symbol	Definition	Suggested Values	
Geometric / Elastic parameter	L, B	Footing dimensions	Geometry	
	KN	Footing initial vertical stiffness	Estimated based on literature(e.g. Gazetas, 1991) or calibrated on experimental results (when available)	
	KHH	Footing initial horizontal stiffness in x and y direction		
	KMM	Footing initial rotational stiffness in x and y direction		
Strength parameters	N_{max}	Centred vertical bearing capacity	Brinch-Hansen (1970) Vesic (1973) Eurocode8 Butterfield and Gottardi (1994)	
	H_{max}	Maximum base shear capacity along x and y direction		
	M_{max}	Maximum base moment capacity around x and y direction		
	BS	Bounding surface type	Rugby-ball shape, scallop shape or ellipsoid shape	
Model specific	α	Uplift initiation parameter	Statics	3 (2 to 10)

	Symbol	Definition	Suggested Values	
	n_{ur}	Exponent for loading history	Fixed	1
	d_θ	Soil/footing contact degradation	Calibratedon experimental results	0.1 (0.1 to 10)
	H_0^{pl}	Normalised reference plastic modulus		0.2 to 0.4
	χ_g	Plastic potential parameter		0.5 to 2

Table 1. Summary of macro-element parameters related to: (i) geometric and elastic parameters; (ii) strength parameters; (iii) model specific parameters

It is not affecting much the results, and is typically taken as 3, which corresponds to assuming a linear distribution stresses for the soil at the beginning of the analysis;

- the exponent for loading history in unloading/reloading, nUR, is usually equal to 1, being related to different plastic modulus values for unloading/reloading in comparison to the virgin loading;

- the soil/footing contact degradation parameter,dθ, takes into account the decrease of the contact area due to cumulative inelastic rocking in the damage model and can be evaluated based on experimental results;

- the normalised reference plastic modulus, H_0^{pl} , calibrated based on experimental results;

- and the plastic potential surface parameter,χg, also calibrated based on experimental results.

From the above, it turns out that, once the classical elastic and strength parameters for the soil-foundation system are known, a small number of 3 free-parameters remains to be calibrated in the validation process: H_0^{pl}, the normalised reference plastic modulus, χg, the plastic potential surface parameter, and dθ, the damage model parameter.

NOTE 1: Given that the SSI macro-element presents a nonlinear response from the outset of the analysis, it is very important to apply the initial loading in several steps in order to avoid lack of convergence or erroneous results. Typically a number of steps between 50 and 100 should be enough, although in more demanding cases of analyses might be needed.

NOTE 2: Care should be taken on the ground motion input when a dynamic time history analysis with the SSI macro-element is made. In fact, given that the two nodes of the macro-element should have the same motion while no inertial interaction is present, the soundest way of performing the analysis is not by imposing the ground motion acceleration history at the base node but by imposing the corresponding inertia forces on the structural masses above.

	Symbol	Definition	Suggested Values	
Geometric / Elastic parameter	L, B	Footing dimensions	Geometry	
	KN	Footing initial vertical stiffness	Estimated based on literature(e.g. Gazetas, 1991) or calibrated on experimental results (when available)	
	KHH	Footing initial horizontal stiffness in x and y direction		
	KMM	Footing initial rotational stiffness in x and y direction		
Strength parameters	N_{max}	Centred vertical bearing capacity	Brinch-Hansen (1970) Vesic (1973) Eurocode8 Butterfield and Gottardi (1994)	
	H_{max}	Maximum base shear capacity along x and y direction		
	M_{max}	Maximum base moment capacity around x and y direction		
	BS	Bounding surface type	Rugby-ball shape or scallop shape	
Model specific parameter	α	Uplift initiation parameter	Statics	3 (2 to 10)
	n_{ur}	Exponent for loading history	Fixed	1

	Symbol	Definition	Suggested Values	
	d_θ	Soil/footing contact degradation	Calibratedon experimental results	0.1 (0.1 to 10)
	H_0^{pl}	Normalised reference plastic modulus		0.2 to 0.4
	χ_g	Plastic potential parameter		0.5 to 2

Table 1. Summary of macro-element parameters related to: (i) geometric and elastic parameters; (ii) strength parameters; (iii) model specific parameters

Stress distribution below the footing				
α	2	3	4	$+\infty$

Table 2. Stress distribution below the footing and the corresponding value of uplift initiation parameter (α)

Some theoretical background on SSI analysis

SSI analysis can be carried out through the employment of a nonlinear solid finite element model (i.e. soil-block), or by means of a simpler and thus more practical substructure approach, which is the one that can be adopted in SeismoStruct.

In principle, when modelling SSI using the substructure method, one should first analyse the kinematic interaction with the full model of the soil and the structure, with the structural stiffness but no structural mass. In such procedure, the seismic input propagation in the soil is explicitly modelled, typically in the frequency domain (though not necessarily), and the end result is the foundation input motion (FIM), i.e. the motion of the foundation if it were massless. This initial step is, however, often avoided by assuming that the kinematic interaction may be neglected and thus using the free-field ground motion as the FIM (this free-field motion is also often assumed to result only from the vertical propagation of shear waves through horizontal soil layers).

A second stage in the modelling of SSI using the substructure method would then be the calculation of the foundation impedances (i.e. dynamic response properties of the foundation), typically represented by a set of springs, dashpots (and possibly fictitious masses to get the correct frequency-dependence of the impedances). This second step may be simplified by determining the impedances from existing expressions in the literature.

The final step is the analysis of the structure, with its stiffness and mass, supported on the foundation impedances and subjected to the FIM. This is what can be done in SeismoStruct, which features the added advantage of being able, through the employment of the SSI macro-element, of considering also the nonlinear response of the foundation system. In other words, an SSI analysis carried out using this macro-element corresponds to a hybrid approach in-between the inertial interaction analysis of the substructure approach, which is strictly valid only for linear response, and a nonlinear solid finite element modelling of SSI effects.

Within the context defined above, therefore, the following should be kept in mind by the user:

A. The substructure method is only theoretically correct if the response is linear, i.e. without sliding or uplifting of a footing, gapping of a pile, stiffness degradation, plastic behaviour, and irrecoverable displacements; in the presence of nonlinearities, therefore, this type of analysis inevitably involves some degree of approximation.
B. As already noted, the FIM is the input motion that the foundation would have only if it were massless (as well as the rest of the structure) and if it would behave linearly. Indeed, and for instance, if the foundation model of a footing simulates its sliding resistance, and if there is structural mass, the motion of the foundation will no longer be the FIM because of the inertial forces coming from above and from a possible sliding of the foundation. Moreover, even in the case of linear response and just a footing with its mass (no structure above), the motion of the foundation will not be exactly the FIM because of the inertia forces generated by the footing mass.
C. The seismic input for SSI analysis using the substructure approach (as done in SeismoStruct) can consist of one of the following:
    - acceleration time-history at the fixed base node of the macro-element (this should be the FIM, often assumed to be equivalent to the free-field motion, as discussed already), which will then propagate through the macro-element and excite the structural masses (including the foundation one);
    - inertia forces time-histories, computed as the product of structural masses (including the foundation one) by the FIM, applied to each of the masses of the structure.

    These two seismic input definition approaches are supposed to lead to identical analysis results in terms of nodal relative displacements (and hence material strains/stress and member internal forces). The first approach is easier to apply because only a base motion in the fixed nodes needs to be defined. However, it may give rise to numerical problems in special cases when the stiffnesses of the macro-element are very large. The second approach is more difficult to apply, because one has to apply a dynamic force time-history in all nodes with lumped masses and becomes cumbersome when distributed masses are used. But this method works in all cases.

SSI– Macro-element for Pile Foundations – ssilink2

This element extends the nonlinear macro-element approach to the analysis of laterally loaded flexible piles and soil-pile-structure interaction. It is based on the work of Correia and Pecker [2019b]. The lateral response of the entire soil-pile system to seismic actions is thus condensed at the pile-head, being represented by a zero-length element located at the base of the columns and subjected to the foundation input motion, as shown in Figure 4.

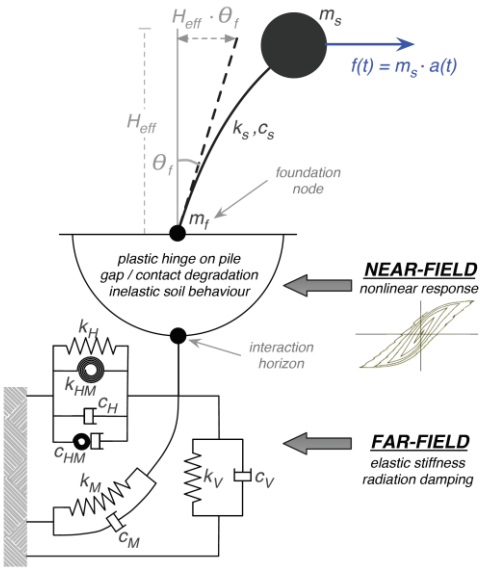


Figure 4.Nonlinear pile-head macro-element

The pile-head macro-element model represents the lateral behaviour of single vertical piles, subjected to a horizontal load and a moment, from the initial stages of loading up until reaching failure. The effects of vertical loading are not directly considered in this model except for its influence on the plastic moment of the pile cross-section. Otherwise, it is considered that the upper zone of the soil profile, until the depth at which the plastic hinge will form, only contributes to the lateral load resistance. The vertical load is assumed to be transferred to the surrounding soil below that depth, where there is no influence of gap opening.

A saturated soil deposit is considered and, upon seismic motion, it is assumed to be impervious. The soil is thus considered to have undrained behaviour since the aim of the macro-element is to simulate the pile response under seismic actions, or short-term cyclic loads, and the Tresca failure criterion is assumed to be valid. Figure5a represents the two simplified geotechnical scenarios considered, in terms of undrained shear strength (Su) distribution along the depth of the soil deposit: constant or linear. Figure5b illustrates the characteristic soil response for a laterally loaded long pile, namely: a soil passive wedge failure at shallow depths and flow-around failure at larger depths, with a possible gap formation at the back of the pile.

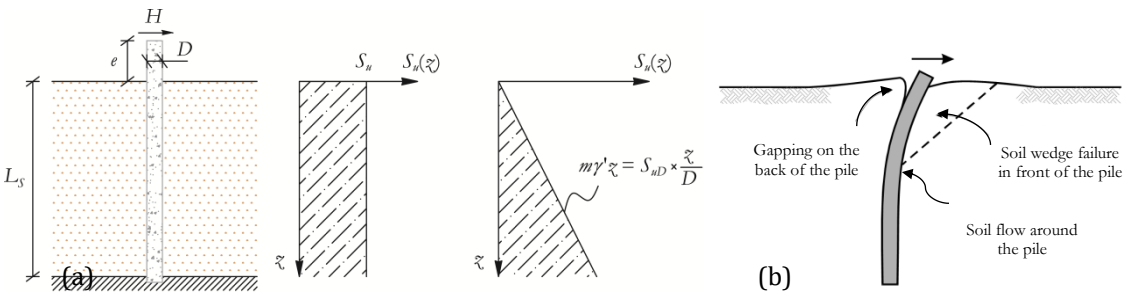


Figure5. Simplified (a) geotechnical scenarios and (b) soil response for pile-head lateral loading

The proposed macro-element is based on the three major features of the behaviour of laterally loaded piles, namely:

i) Initial elastic response,
ii) Gap opening and closure,
iii) Failure loading conditions.

The bounding surface plasticity model is used to represent a continuous transition between the initial elastic response and the plastic flow at failure, for monotonic as well as cyclic pile-head loading conditions. The gapping behaviour is represented by a nonlinear elastic model which, however, takes into account and is influenced by the plastic deformation state in the surrounding soil.

The bounding surface in the macro-element model corresponds to the failure surface for laterally loaded piles. Since there is no evidence showing that non-associative behaviour should be considered, associative plasticity is used and the bounding surface acts simultaneously as the plastic potential surface. No axial load effects are considered in this macro-element formulation and, consequently, the failure surface is defined in the loading space of the pile-head horizontal force and moment only. Furthermore, a planar loading is assumed.

A "rounded" approximate failure surface was proposed in Correia and Pecker [2019a], which is based on the so-called superellipse. Supposing a superellipse centred at the point (Hc, Mc), with a horizontal axis length $H_{u,e=0}$ and a vertical axis length My, which is also superimposed to a distortion of its shape, $\gamma < 0$, this approximate failure surface can be expressed as:

$$\left| \frac{\overline{H} - H_c}{H_{u,e=0}} - \gamma \frac{\overline{M} - M_c}{M_y} \right|^{n_H} + \left| \frac{\overline{M} - M_c}{M_y} \right|^{n_M} = 1 \qquad (1)$$

The positive exponents nH and nM control the curvature of the sides of the superellipse. Figure 6 represents such distorted superellipse configuration, centred at the origin (Hc = Mc = 0), with its parameters calibrated in order to fit the failure surface for the linear Su soil profile.

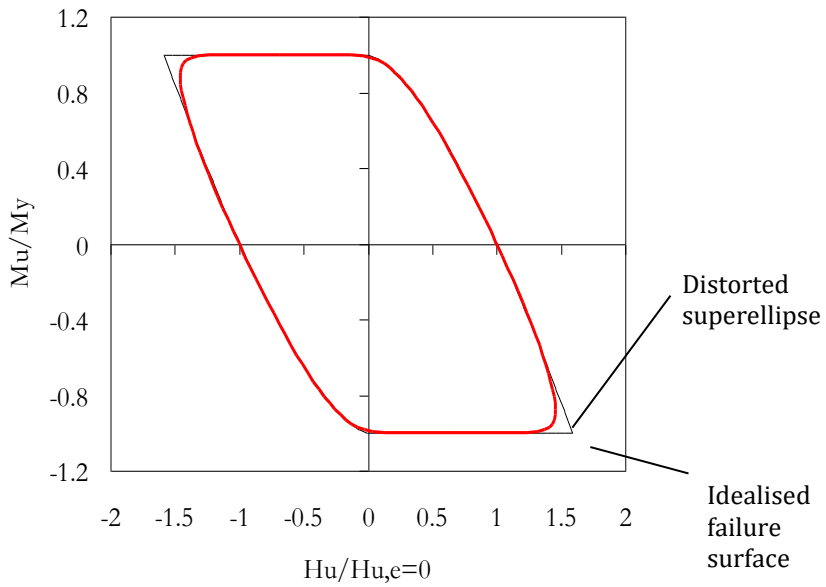


Figure 6. Distorted superellipse configuration for linear Su

The macro-element model requires the definition of 23 input parameters: D, K_{VV}, K_{HH}, K_{MM}, K_{HM}, K_{TT}, C_{VV}, C_{HH}, C_{MM}, C_{HM}, C_{TT}, $H_{u,\,e=0}$, M_y, n_H, n_M, γ, z_w, $\left(E_pI_p\right)_{eff}$, β, η, H_0^{pl}, n_{UR} and δ_{Lim}. Only the last 5 of these parameters must be calibrated, since all the remaining ones are computed directly through expressions developed in the literature.

The pile flexural stiffness, $\left(E_pI_p\right)_{eff}$, can be easily computed, while the pile yield moment, My, can be computed using any cross-section analysis tool (and considering the static vertical load on the pile). On the other hand, formulas for $H_{u,\,e=0}$ and z_w, are derived in Correia and Pecker [2019a].

Gazetas [1991] provides formulas for a direct computation of pile-head lateral and axial stiffness and damping coefficients. These are valid for soil profiles with constant, linear or parabolic increase of soil stiffness with depth, which are representative of OC clay, NC clay and sand, respectively. Figure 7 represents the soil stiffness evolution with depth in such idealised soil profiles. His expressions for pile-head static stiffnesses have been adopted, with slight modifications, in the current version of EC 8 – Part 5 [2003]. These are valid for flexible or long piles and are summarised in **Table 3.**. In those expressions, D is the pile diameter, ESD is the soil modulus of deformation at a depth equal to the pile diameter and Ep is the Young's modulus of the pile material. The pile-head stiffness matrix components follow the sign convention expressed in Figure 8.Gazetas [1991] also presents the corresponding pile-head damping coefficients, which are computed for each frequency f according to the expressions in **Figure 8**.

The dynamic components of the pile-head stiffnesses have been shown by Gazetas [1984] to be roughly equal to one, for the usual frequency range of interest for structural response. Hence, pile-head static stiffnesses may be approximately used as dynamic ones, for single flexible piles. Variation of the damping ratio components with frequency is linear, as predicted by the expressions in Table 4. This means that radiation damping behaviour may be approximated by physical dashpots with constant damping coefficient C.

The bounding surface parameters are fixed for each of the soil strength profiles and are shown in Table 5.The limit value δ_{Lim} is a parameter related to the numerical convergence and varies between 0.01 and 0.2, with a default value of 0.1.

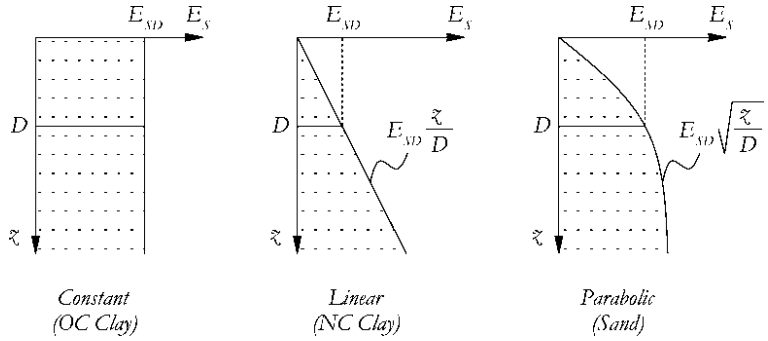


Figure 7. Idealised soil stiffness profiles

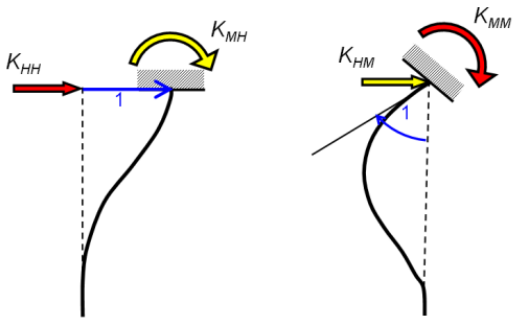


Figure 8. Sign convention for pile-head loading

Soil stiffness profile	$\frac{K_{HH}}{E_{SD}\,D}$	$\frac{K_{MM}}{E_{SD}\,D^3}$	$\frac{K_{HM}}{E_{SD}\,D^2}$
Constant $E_S = E_{SD}$	$1.08\left(\frac{E_p}{E_{SD}}\right)^{0.21}$	$0.16\left(\frac{E_p}{E_{SD}}\right)^{0.75}$	$-0.22\left(\frac{E_p}{E_{SD}}\right)^{0.50}$
Linear $E_S = E_{SD}\,z/D$	$0.60\left(\frac{E_p}{E_{SD}}\right)^{0.35}$	$0.14\left(\frac{E_p}{E_{SD}}\right)^{0.80}$	$-0.17\left(\frac{E_p}{E_{SD}}\right)^{0.60}$
Parabolic $E_S = E_{SD}\sqrt{z/D}$	$0.79\left(\frac{E_p}{E_{SD}}\right)^{0.28}$	$0.15\left(\frac{E_p}{E_{SD}}\right)^{0.77}$	$-0.24\left(\frac{E_p}{E_{SD}}\right)^{0.53}$

Table 3. Pile-head static stiffness coefficients for flexible piles (after EC8 – Part 5 [2003])

Finally, the remaining 4 calibration parameters, 2 of them are related to monotonic response –β and H_0^{pl}, and the 2 others are related to cyclic behaviour –η and nUR. Alternatively, 2 of the parameters are related to the gapping behaviour –β and η, and 2 others are related to the plasticity model – H_0^{pl} and nUR. The parameters β, H_0^{pl} and nUR are always positive, while η can also be equal to zero if no residual gap opening is considered. Their default values and ranges of variation are presented in Table 6.

NOTE: Care should be taken on the ground motion input when a dynamic time history analysis with the SSI macro-element is made. In fact, given that the two nodes of the macro-element should have the same motion while no inertial interaction is present, the soundest way of performing the analysis is not by imposing the ground motion acceleration history at the base node but by imposing the corresponding inertia forces on the structural masses above.

Soil stiffness profile	$\frac{\xi_{HH}\,V_{SD}}{f\,D}$	$\frac{\xi_{MM}\,V_{SD}}{f\,D}$	$\frac{\xi_{HM}\,V_{SD}}{f\,D}$

Constant $E_S = E_{SD}$	$1.10\left(\frac{E_p}{E_{SD}}\right)^{0.17}$	$0.35\left(\frac{E_p}{E_{SD}}\right)^{0.20}$	$0.85\left(\frac{E_p}{E_{SD}}\right)^{0.18}$
Linear $E_S = E_{SD}\, z/D$	1.80	0.40	1.00
Parabolic $E_S = E_{SD}\sqrt{z/D}$	$1.20\left(\frac{E_p}{E_{SD}}\right)^{0.08}$	$0.35\left(\frac{E_p}{E_{SD}}\right)^{0.10}$	$0.70\left(\frac{E_p}{E_{SD}}\right)^{0.05}$

Table 4. Pile-head radiation damping coefficients for flexible piles and fundamental shear frequencies of soil deposit (after Gazetas [1991])

S_u profile	n_H	n_M	γ
Constant S_u	8.435	2.000	-0.597
Linear S_u	7.040	2.000	-0.667

Table 5. Pre-determined failure surface parameters

β	η	H_0^{pl}	n_{UR}
1	1	0.4	1
0.1-10	0-100	0.1-10	0.5-2

Table 6. Default values and ranges of values for calibration parameters

Some theoretical background on SSI analysis

SSI analysis can be carried out through the employment of a nonlinear solid finite element model (i.e. soil-block), or by means of a simpler and thus more practical substructure approach, which is the one that can be adopted in SeismoStruct.

In principle, when modelling SSI using the substructure method, one should first analyse the kinematic interaction with the full model of the soil and the structure, with the structural stiffness but no structural mass. In such procedure, the seismic input propagation in the soil is explicitly modelled, typically in the frequency domain (though not necessarily), and the end result is the foundation input motion (FIM), i.e. the motion of the foundation if it were massless. This initial step is, however, often avoided by assuming that the kinematic interaction may be neglected and thus using the free-field ground motion as the FIM (this free-field motion is also often assumed to result only from the vertical propagation of shear waves through horizontal soil layers).

A second stage in the modelling of SSI using the substructure method would then be the calculation of the foundation impedances (i.e. dynamic response properties of the foundation), typically represented by a set of springs, dashpots (and possibly fictitious masses to get the correct frequency-dependence of the impedances). This second step may be simplified by determining the impedances from existing expressions in the literature.

The final step is the analysis of the structure, with its stiffness and mass, supported on the foundation impedances and subjected to the FIM. This is what can be done in SeismoStruct, which features the added advantage of being able, through the employment of the SSI macro-element, of considering also the nonlinear response of the foundation system. In other words, an SSI analysis carried out using this macro-element corresponds to a hybrid approach in-between the inertial interaction analysis of the

substructure approach, which is strictly valid only for linear response, and a nonlinear solid finite element modelling of SSI effects.

Within the context defined above, therefore, the following should be kept in mind by the user:

D. The substructure method is only theoretically correct if the response is linear, i.e. without sliding or uplifting of a footing, gapping of a pile, stiffness degradation, plastic behaviour, and irrecoverable displacements; in the presence of nonlinearities, therefore, this type of analysis inevitably involves some degree of approximation.
E. As already noted, the FIM is the input motion that the foundation would have only if it were massless (as well as the rest of the structure) and if it would behave linearly. Indeed, and for instance, if the foundation model of a footing simulates its sliding resistance, and if there is structural mass, the motion of the foundation will no longer be the FIM because of the inertial forces coming from above and from a possible sliding of the foundation. Moreover, even in the case of linear response and just a footing with its mass (no structure above), the motion of the foundation will not be exactly the FIM because of the inertia forces generated by the footing mass.
F. The seismic input for SSI analysis using the substructure approach (as done in SeismoStruct) can consist of one of the following:
    - acceleration time-history at the fixed base node of the macro-element (this should be the FIM, often assumed to be equivalent to the free-field motion, as discussed already), which will then propagate through the macro-element and excite the structural masses (including the foundation one);
    - inertia forces time-histories, computed as the product of structural masses (including the foundation one) by the FIM, applied to each of the masses of the structure.

    These two seismic input definition approaches are supposed to lead to identical analysis results in terms of nodal relative displacements (and hence material strains/stress and member internal forces). The first approach is easier to apply because only a base motion in the fixed nodes needs to be defined. However, it may give rise to numerical problems in special cases when the stiffnesses of the macro-element are very large. The second approach is more difficult to apply, because one has to apply a dynamic force time-history in all nodes with lumped masses and becomes cumbersome when distributed masses are used. But this method works in all cases.

Elastomeric Isolator Element (Bouc Wen) - isolator 1

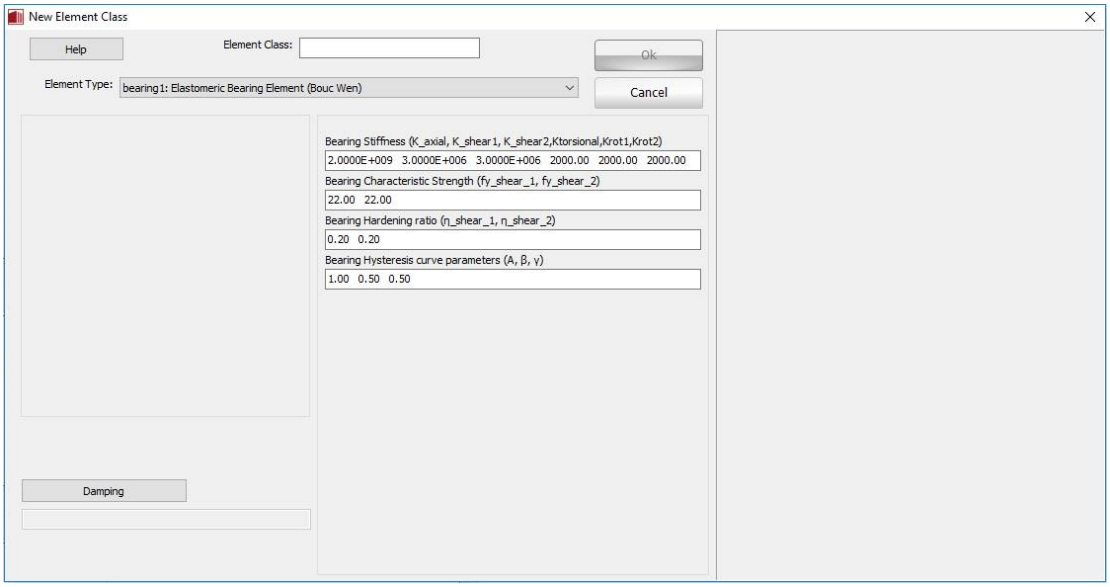
Isolator 1 Elements are 3D elements with zero length used to model the behaviour of elastomeric isolators used in Seismic Isolation Applications. Isolator 1 Elements have coupled plasticity properties for the two shear directions (axes 2 and 3 in the local coordinate system of the isolator 1 element) 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]. In the shear directions the force-deformation relationships follow the equations below:

$$f_2 = \eta_shear_2 \;\; K_Shear_2 \, u_2 \; + (1 - \; \eta_shear_2) \, Y_2 \, z_2$$

$$f_3 = \eta_shear_3 \;\; K_Shear_3 \, u_3 \; + (1 - \; \eta_shear_3) \, Y_3 \, z_3$$

where η_shear_2 and η_shear_3 are the ratios of the Post Yield Stiffness to the Elastic (Pre-Yielding) Stiffness of the isolator in each shear direction (Isolator Hardening Ratios), K_2 and K_3 are the elastic Stiffnesses of the isolator in each direction, Y_2 and Y_3 the yielding deformations in each shear direction while z_2 and z_3 are internal hysteretic variables.

Thirteen parameters are needed in order to describe the isolator 1 element behaviour:



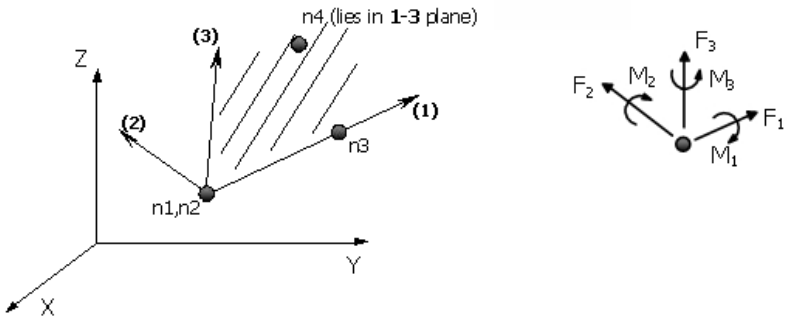
Element Properties	Typical values	Default values
Elastic Stiffness in the axial (local axis 1) direction - K_axial	2E+05 - 150E+05 [kN/m]	2E+06 [kNm]
Elastic Stiffness in the shear (local axis 2 and 3) directions - K_shear_1, K_Shear_2	3000 – 40000 [kN/m]	3000 [kNm]
Elastic Stiffnesses in the torsional and rotational degrees of freedom - Ktorsional, Krot1, Krot2	-	20000 [kNm]
Isolator Characteristic Strength for each shear direction – fy_shear_1, fy_shear_2	20-600 [kN]	22 [kN]
Isolator Hardening ratio for eachshear direction – η_shear_1, η_shear_2	0.1-0.8	0.2
Variable controlling the hysteresis Amplitude – A	1	1
First hysteretic shape variable – β	0.1-0.9	0.5
First hysteretic shape variable – γ	0.1-0.9	0.5

NOTE: In general the rule A/(β+γ) = 1 should be followed.

In this element's dialog box it is also possible to define an element-specific damping, as opposed to the global damping described in here. To do so, users need simply to press the Damping button and then select the type of damping that better suits the element in question (users should refer to the Damping menu for a discussion on the different types of damping available and hints on which might the better options). Users are reminded also that damping defined at element level takes precedence over global damping, that is, the "globally-computed" damping matrix coefficients that are associated to the degrees-of-freedom of a given element will be replaced by coefficients that will have been calculated through the

multiplication of the mass matrix of the element by a mass-proportional parameter, or through the multiplication of the element stiffness matrix by a stiffness-proportional parameter, or through the calculation of an element damping Rayleigh matrix.

Local axes and output notation are defined in the figure below:



Local Axes and Output Notation

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:

$$\mu = ffast_1 - (ffast_1 - fslow_1)exp(-rate_1 \left| v \right|)$$

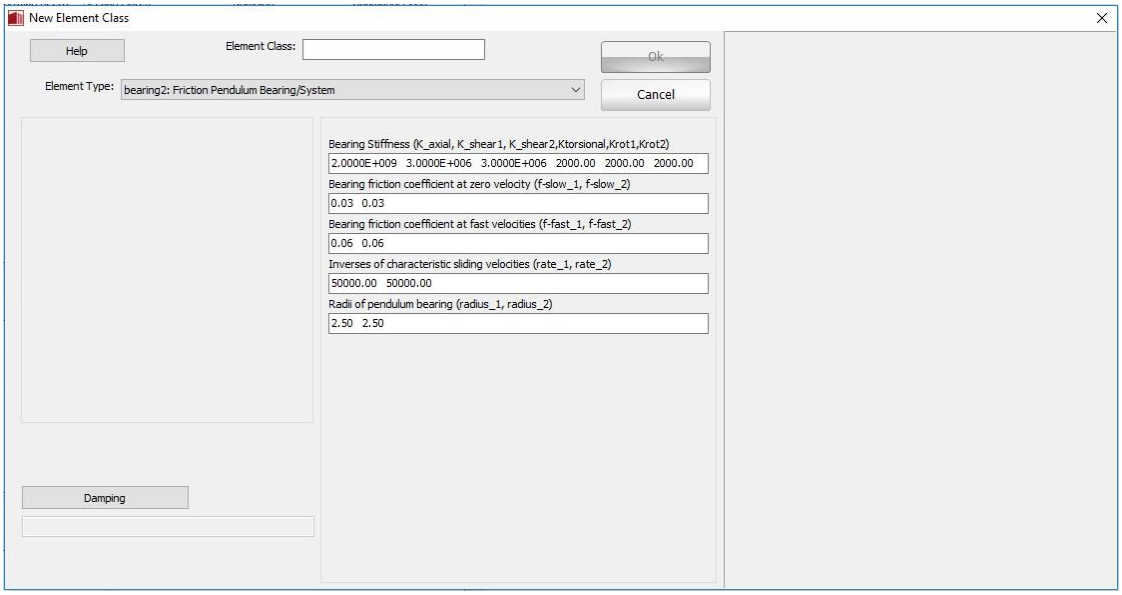
where ffast_1 and fslow_1 are the isolator friction coefficients at fast and slow velocities respectively, v is the isolator velocity and rate_1 is the rate controlling the transition from low to high velocities.

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.

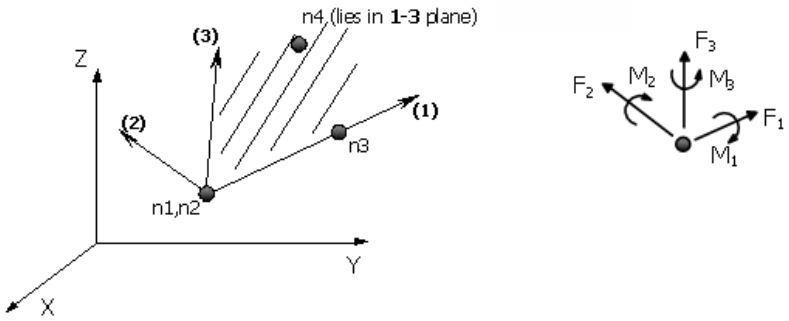
Fourteen parameters are needed in order to describe the isolator 2 element behaviour:



Element Properties	Typical values	Default values
Elastic Stiffness in the axial (local axis 1) direction - K_axial	1E+06 - 30E+06 [kN/m]	2E+06 [kNm]
Elastic Stiffness in the shear (local axis 2 and 3) directions - K_shear_1, K_Shear_2	500 – 20000 [kN/m]	3000 [kNm]
Elastic Stiffnesses in the torsional and rotational degrees of freedom - Ktorsional, Krot1, Krot2	-	20000 [kNm]
Isolator Friction Coefficient at slow velocities –fslow_1, fslow_2	0.02-0.05	0.03
Isolator Friction Coefficient at fast velocities –ffast_1, ffast_2	0.04-0.1	0.06
Inverse of characteristic sliding velocities – rate_1, rate_2	50-100 [s/m]	50 [s/m]
Curvature radii of the friction pendulum – radius_1, radius_2	2-7 [m]	2,50 [m]

In this element's dialog box it is also possible to define an element-specific damping, as opposed to the global damping described in here. To do so, users need simply to press the Damping button and then select the type of damping that better suits the element in question (users should refer to the Damping menu for a discussion on the different types of damping available and hints on which might the better options). Users are reminded also that damping defined at element level takes precedence over global damping, that is, the "globally-computed" damping matrix coefficients that are associated to the degrees-of-freedom of a given element will be replaced by coefficients that will have been calculated through the multiplication of the mass matrix of the element by a mass-proportional parameter, or through the multiplication of the element stiffness matrix by a stiffness-proportional parameter, or through the calculation of an element damping Rayleigh matrix.

Local axes and output notation are defined in the figure below:



Local Axes and Output Notation

MASS AND DAMPING ELEMENT TYPES

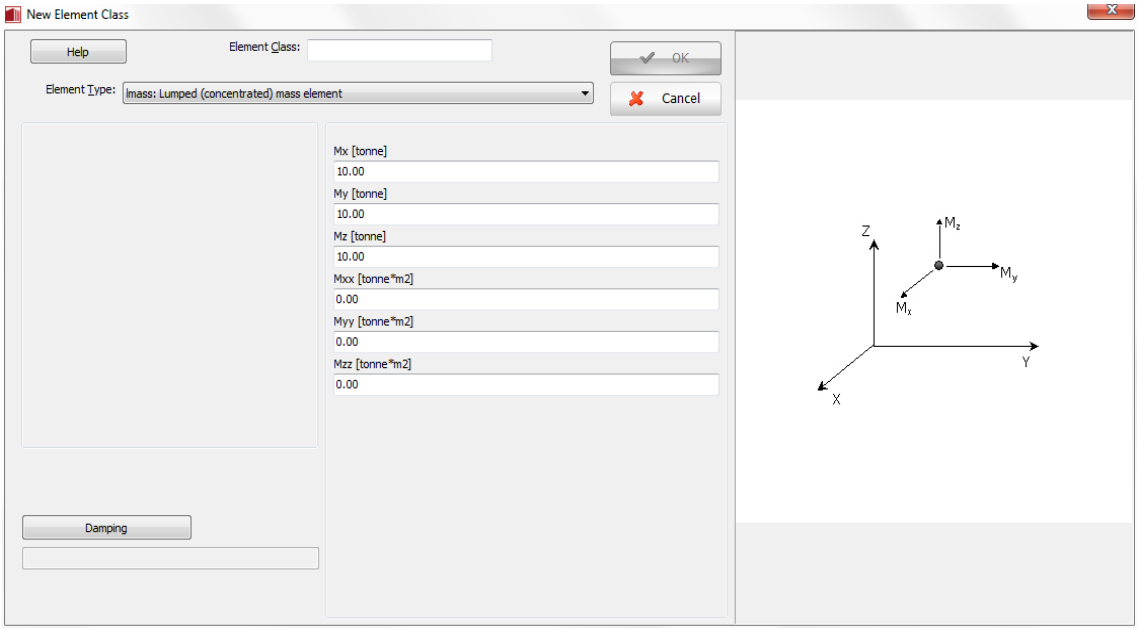
Mass elements - lmass & dmass

As indicated in the **Materials** module, users have the possibility of defining the materials specific weights, with which the distributed self-mass of the structure can then be calculated. More, in the **Sections** module, additional distributed mass may also be defined, which will serve to define any mass not associated to the self-weight of the structure (e.g. slab, finishings, infills, variable loading, etc).

Here, *lumped* (lmass) and *distributed* (dmass) mass-only elements can also be defined and then added to the structure in the Element Connectivity module, so that users may model mass distributions that cannot be obtained using the aforementioned **Materials/Sections** facilities; e.g. water tank with concentrated mass on top.

NOTE: Analyses of large models featuring distributed mass/loading are inevitably longer than those where lumped masses, and corresponding point loads, are employed to model, in a more simplified fashion, the mass/weight of the structure. If users are not interested in obtaining information on the local stress state of structural elements (e.g. beam moment distribution), but are rather focused only on estimating the overall response of the structure (e.g. roof displacement and base shear), then the employment of a faster lumped mass/force modelling approach may prove to be a better option, with respect to its distributed counterpart.

The *lumped mass* element (lmass) is a single-node mass element, characterised by three translational and three rotational inertia values. The latter are defined by means of the mass moment of inertia (not to be confused with the second moment of area, commonly named also as moment of inertia), and may be computed using formulae available in the literature [e.g. Pilkey, 1994; Gere and Timoshenko, 1997]. The inertia mass values are to be defined with respect to the global reference system (X, Y and Z), and lead to a diagonal 6x6 element mass matrix.



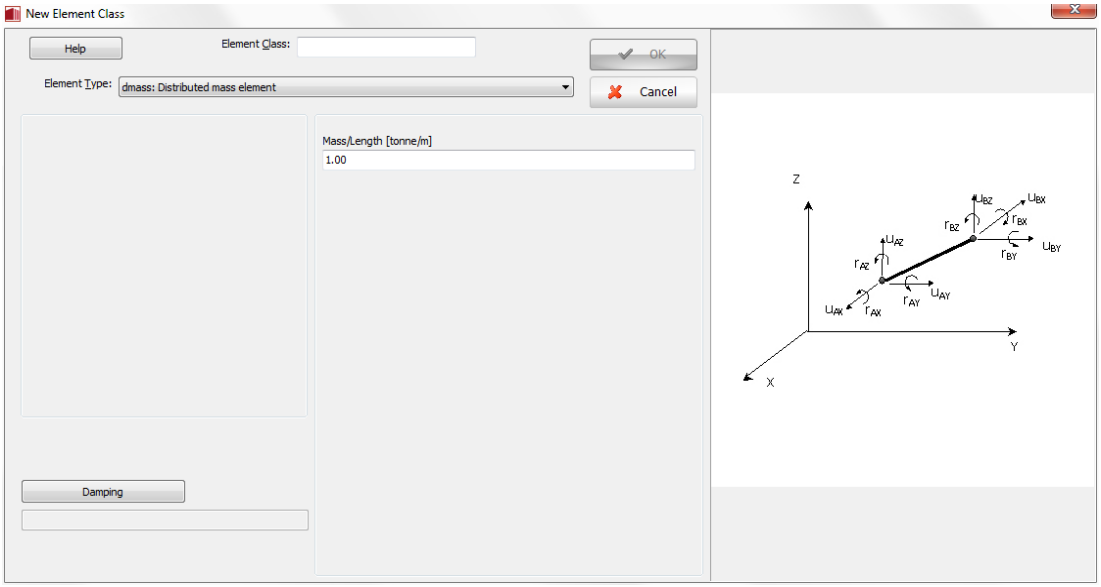
Definition of a new lmass element

The *distributed mass* (dmass) is a two-node mass element. The user needs only to specify the unitary mass (mass/length) value, from which the program computes internally the total element mass M, and subsequently derives the respective *diagonal mass matrix* with reference to the global translational degrees-of-freedom of the member.

NOTE 1: When the structure is subjected to very large deformations (e.g. buckling), the employment of two or more dmass elements per member is recommended, for accurate modelling.

NOTE 2: If the *loads are derived from masses (in the gravity direction based on the g value or in any translational direction, according to user-defined coefficients)*, then the program will automatically compute and effectively apply distributed permanent loads.

NOTE 3: Distributed loads obtained from dmass elements are not considered in stress-recovery operations (because they are separate elements from the beams/columns), hence moment values throughout an element's length are bound to be wrong. Users interested in obtaining correct moments throughout an element's length, should define distributed mass/load using the 'material volumetric weight' in the Materials module and/or 'section added mass' in the Sections module.

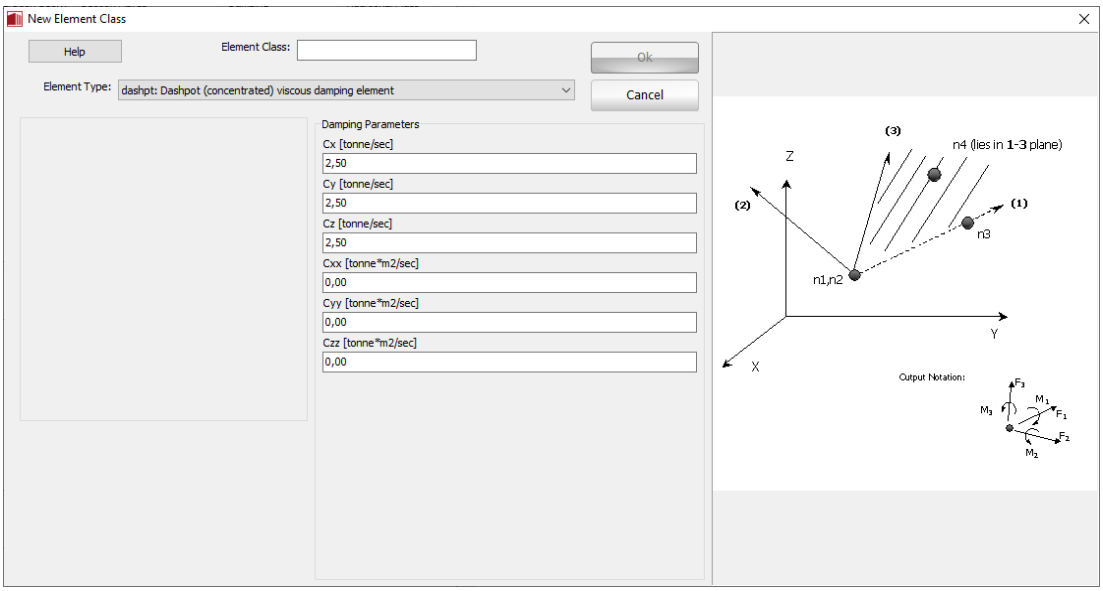


Definition of a new dmass element

Damping element - dashpt

IMPORTANT: In SeismoStruct, dampers are normally modelled by means of link elements with adequate response curves that may be able to characterise the non-velocity-dependent (at least within the typical range of earthquake velocities) force-displacement relationship of a given damper. However, in those cases where velocity dependence is important, this dashpt element may be employed instead, noting that currently only a linear force-velocity relationship is featured.

This is a two-node damping element, which may be employed to represent a linear dashpot between any two given nodes, e.g. it can be used to represent a damper installed on a steel brace.. Damping coefficients may be defined on all six global degrees-of-freedom, though, commonly, dampers will work only in one or two directions. The dashpot accounts for the relative motion between the two element nodes, in order to calculate the dashpot forces.



Definition of a new dashpt element

NOTE: This dashpt element may also be employed whenever the need arises for the introduction of a Maxwell model (i.e. series coupling of damping and stiffness), by placing in series a link and a dashpt element. For a Kelvin-Voigt model (i.e. parallel coupling of damping and stiffness), one may again make use of a link element, this time placed in parallel with a dashpt, though in these cases it may result easier to simply assign directly to the link element a given viscous damping value.

Nonlinear Dashpot Viscous Damping element - NLdashpt

IMPORTANT: In SeismoStruct, dampers are normally modelled by means of link elements with adequate response curves that may be able to characterise the non-velocity-dependent (at least within the typical range of earthquake velocities) force-displacement relationship of a given damper. However, in those cases where velocity dependence is important, and in particular a nonlinear relationship between force and velocity this NLdashpt element may be employed instead.

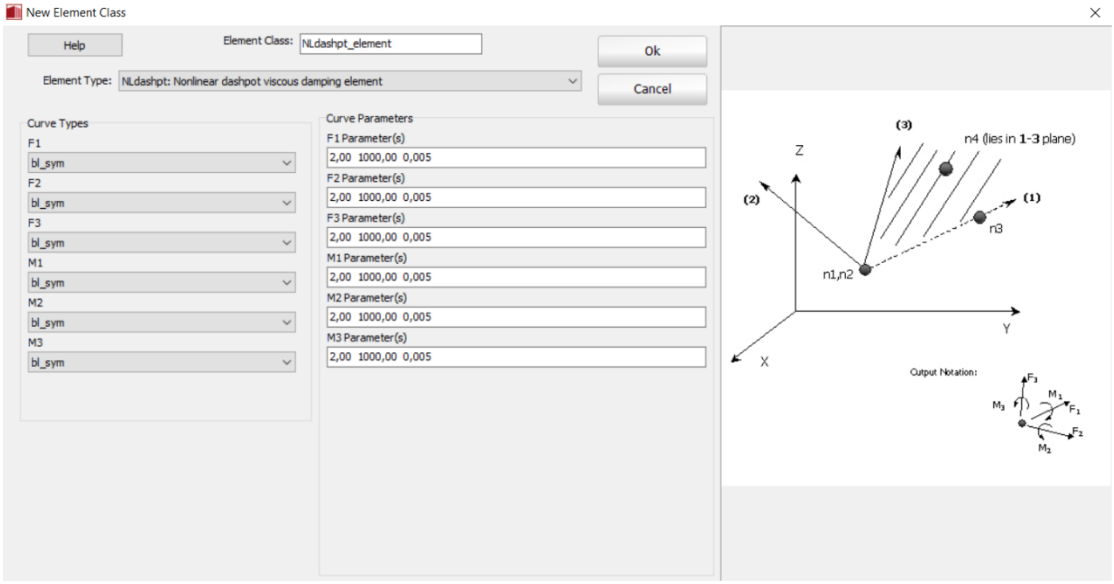
This is a two-node damping element, providng damping in all six global degrees of freedom. Damping in each degree of freedom is modelled through a Force-Velocity relationship defined by one of the available response curves. For the definition of the element the user needs to define a response curve for the modelling of the damping action in each direction.

Currently, twenty-nine response curves are available, selectable from within the Element Class dialog box, whenever a rackh element type is selected.

- Linear symmetric curve - lin_sym
- Linear asymmetric curve - lin_asm
- Bilinear symmetric curve - bl_sym
- Bilinear asymmetric curve - bl_asm
- Bilinear kinematic hardening curve - bl_kin
- Trilinear symmetric curve - trl_sym
- Trilinear asymmetric curve - trl_asm
- Quadrilinear symmetric curve - quad_sym
- Quadrilinear asymmetric curve - quad_asm
- Pinched asymmetric curve - pinched_asm
- Modified Ibarra-Medina-Krawinkler Deterioration curve with Bilinear Hysteretic Response – MIMK_bilin
- Modified Ibarra-Medina-Krawinkler Deterioration Model with Peak-Oriented Hysteretic Response – MIMK_peak
- Modified Ibarra-Medina-Krawinkler Deterioration Model with Pinched Hysteretic Response – MIMK_Pinched
- Nonlinear elastic curve - Non_lin_Elast
- Plastic curve – plst
- Simplified bilinear Takeda curve –Takeda
- Asymmetric bilinear Takeda curve –Takeda_asm
- Ramberg Osgood curve - Ramberg_Osgood
- Modified Richard-Abbott curve - Richard_Abbott
- Soil-structure interaction curve - ssi_py
- Gap-hook curve - gap_hk
- Multi-linear curve – multi_lin
- Smooth curve – smooth
- Viscous Damper – vsc_dmp
- Bouc Wen curve - Bouc_Wen
- Elastic – Perfectly plastic Gap curve - gap_elpl
- Impact response curve - pound_hz

- Self Centering Brace response curve - scb
- Generic Hysteretic Curve - gen_hyst

For a comprehensive description of the available response curves associated to the nonlinear dashpot viscous damping element element refer to Appendix G.



Definition of a new NLdashpt element

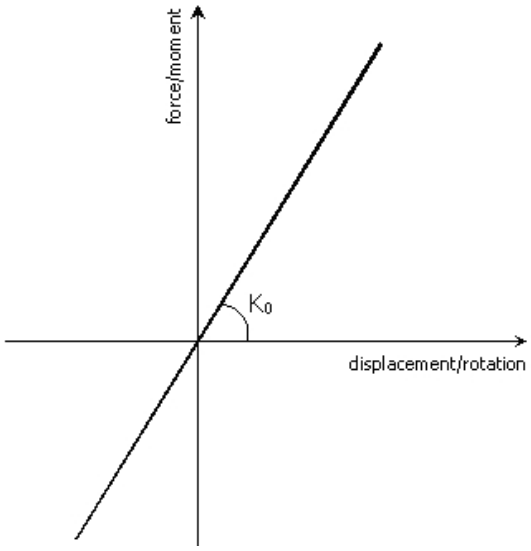
Appendix G - Response Curves

In this appendix the available response curves are described in details.

Symmetric linear curve - lin_sym

IMPORTANT: In previous releases of SeismoStruct, link elements featuring lin_sym response curve were typically employed to model pinned joints (zero stiffness) and/or constraints. However, users may now use the *Equal DOF* facility (see Constraints) to achieve the same objective; e.g. a pin/hinge may be modelled by introducing an 'Equal DOF' constrain defined for translation degrees-of-freedom only.

This is a curve frequently employed to model idealised linear behaviour, soil/foundation flexibility, laminated-rubber bearings (if their usually low viscous damping is ignored), and so on.

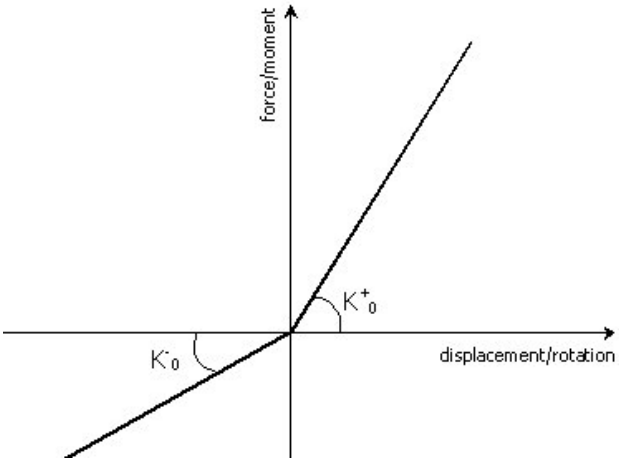


A single parameter needs to be defined in order to fully characterise this response curve:

Curve Properties	Typical values	Default values
Stiffness – **K_0**	-	10000 (-)

Asymmetric linear curve - lin_asm

This is a curve employed to model idealised linear asymmetric behaviour, soil/foundation flexibility, and so on.

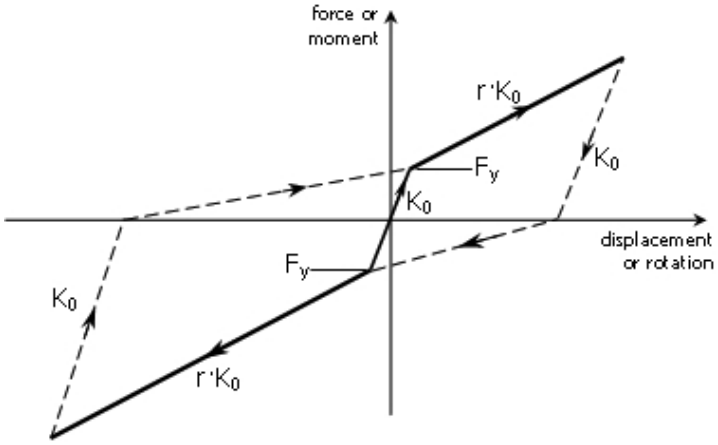


Two parameters need to be defined in order to fully characterise this response curve:

Curve Properties	Typical values	Default values
Stiffness in positive region – $K_{0(+)}$	-	10000 (-)
Stiffness in negative region – $K_{0(-)}$	-	5000 (-)

Symmetric bilinear curve - bl_sym

This is a curve frequently employed to model idealised symmetric elastic-plastic behaviour. An isotropic hardening rule is adopted.



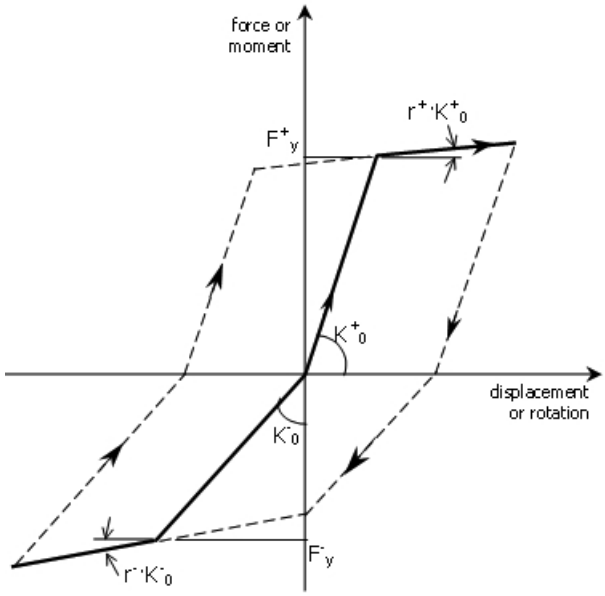
Three parameters need to be defined in order to fully characterise this response curve:

Curve Properties	Typical values	Default values
Initial stiffness – K_0	-	20000 (-)
Yield force – F_y	-	1000 (-)
Post-yield hardening ratio – **r**	-	0.005 (-)

NOTE: Evidently, in those (relatively common) cases where the post-yield stiffness is not very high and the maximum force does not thus reach a value that is twice its yield counterpart, this response curve will behave in the same manner as curve bl_kin.

Asymmetric bilinear curve - bl_asm

This is a curve frequently employed to model idealised asymmetric elastic-plastic behaviour. An isotropic hardening rule is adopted.



Six parameters need to be defined in order to fully characterise this response curve:

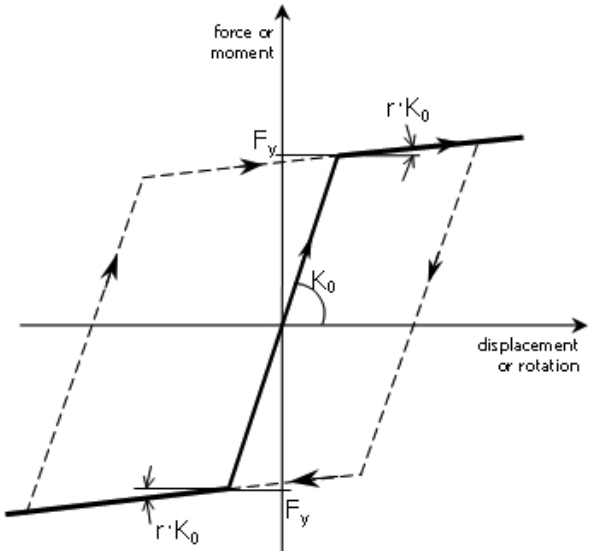
Curve Properties	Typical values	Default values
Initial stiffness in positive region – $K_{0(+)}$	-	20000 (-)
Yield force in positive region – $F_{y(+)}$	-	1000 (-)
Post-yield hardening ratio in positive region – $r_{(+)}$	-	0.005 (-)
Initial stiffness in negative region – $K_{0(-)}$	-	10000 (-)
Yield force in negative region – $F_{y(-)}$	-	-1500 (-)
Post-yield hardening ratio in negative region – $r_{(-)}$	-	0.01 (-)

NOTE 1: Stiffness values $K_0(+)$ and $K_0(-)$ must be positive.

NOTE 2: The image above reflects those (relatively common) cases where the post-yield stiffness is not very high and the maximum force does not thus reach a value that is twice its yield counterpart, making the curve behaviour resemble that of a kinematic-hardening curve such as bl_kin. This however will not be the case on all instances, and hence an isotropic-hardening type of response (such as that shown clearly in here) should be expected.

Bilinear kinematic curve - bl_ kin

This is a kinematic-hardening bilinear symmetrical curve frequently employed to model idealised elastic-plastic behaviour, semi-rigid connections, lead-rubber bearings, steel hysteretic dampers, and so on.

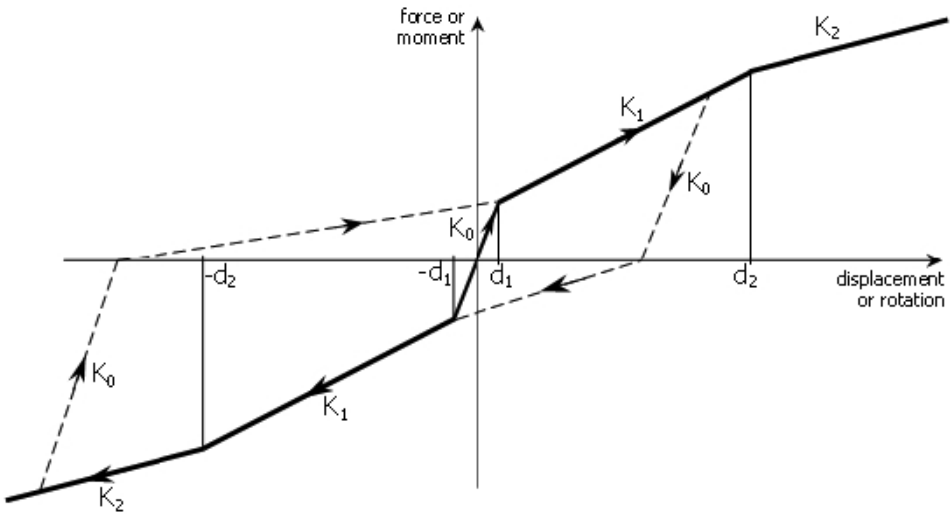


Three parameters need to be defined in order to fully characterise this response curve:

Curve Properties	Typical values	Default values
Initial stiffness – **K0**	-	20000 (-)
Yield force – **Fy**	-	1000 (-)
Post-yield hardening ratio – **r**	-	0.005 (-)

Trilinear symmetric curve - trl_sym

This is a curve frequently employed to model idealised trilinear behaviour. An isotropic hardening rule is adopted.



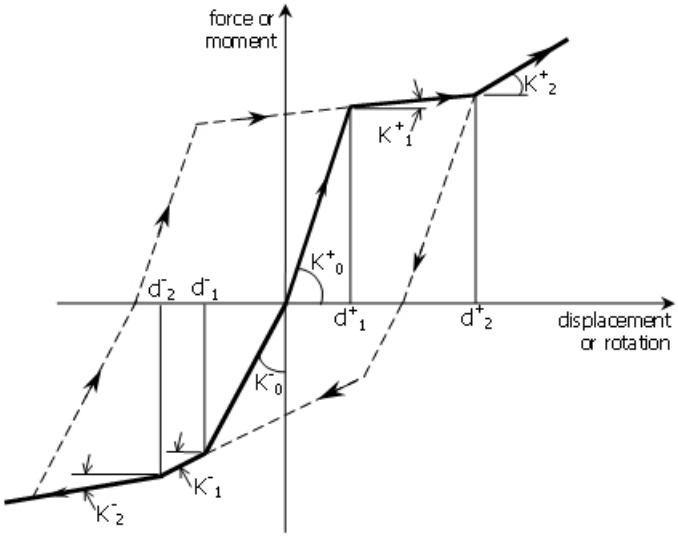
Five parameters need to be defined in order to fully characterise this response curve:

Curve Properties	Typical values	Default values
Initial stiffness – **K0**	-	1000 (-)
First branch displacement limit – **d1**	-	1 (-)
Second branch stiffness – **K1**	-	10 (-)
Second branch displacement limit – **d2**	-	5 (-)
Third branch stiffness – **K2**	-	100 (-)

NOTE: Stiffness values K_0, K_1 and K_2 must be positive. Further, K_1 and K_2 should always be smaller than K_0.

Trilinear asymmetric curve - trl_asm

This is a curve frequently employed to model idealised trilinear asymmetric behaviour. An isotropic hardening rule is adopted.



Ten parameters need to be defined in order to fully characterise this response curve:

Curve Properties	Typical values	Default values
Initial stiffness in positive region – $K_{0(+)}$	-	1000 (-)
First branch positive displacement limit – $d_{1(+)}$	-	1 (-)
Second branch positive stiffness – $K_{1(+)}$	-	50 (-)
Second branch positive displacement limit – $d_{2(+)}$	-	5 (-)
Third branch stiffness in positive region – $K_{2(+)}$	-	100 (-)
Initial stiffness in negative region – $K_{0(-)}$	-	10000 (-)
First branch negative displacement limit – $d_{1(-)}$	-	-5 (-)
Second branch negative stiffness – $K_{1(-)}$	-	35 (-)
Second branch negative displacement limit – $d_{2(-)}$	-	-15 (-)
Third branch stiffness in negative region – $K_{2(-)}$	-	100 (-)

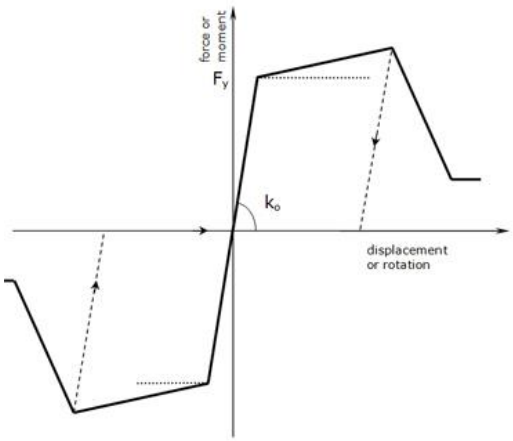
NOTE 1: Stiffness values $K_{0(+)}$, $K_{1(+)}$, $K_{2(+)}$ and $K_{0(-)}$, $K_{1(-)}$, $K_{2(-)}$ must be positive. Further, K_1 and K_2 should always be smaller than K_0 in both positive and negative displacement regions.

NOTE 2: Example. To model the pounding of two adjacent buildings separated by an expansion joint of 20 mm, the following trl_asm curve parameters could be adopted: $K_{0(+)}$=1e12, $d_{1(+)}$=0, $K_{1(+)}$=0, $d_{2(+)}$=1e10, $K_{2(+)}$=0, $K_{0(-)}$=1e12, $d_{1(-)}$=0, $K_{1(-)}$=0, $d_{2(-)}$=-20,$K_{2(-)}$=1e10. However, the employment of response curve gap_hk is recommended for these cases.

NOTE 3: Users may refer to the figure relating to the trl_sym curve, for further indications on the cyclic rules employed this response curve. Ultimately, users are always advised to run simple cyclic load analyses (e.g. using a single link element connected to the ground on one end, and then imposing cyclic displacements at its free node) in order to gain a full understanding of this hysteretic relationship, before its employment within more elaborate models.

Quadrilinear symmetric curve - quad_sym

This is a simple curve employed to model idealised quadrilinear behaviour. An isotropic hardening rule is adopted.

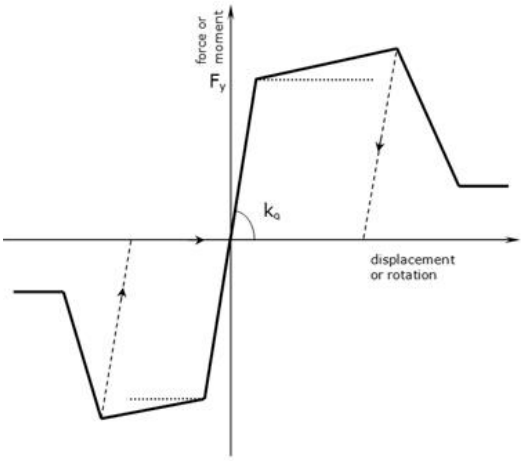


Five parameters need to be defined in order to fully characterise this response curve:

Curve Properties	Typical values	Default values
Yielding strength – F_y	-	500
Yielding displacement – d_y	-	0.005
Ultimate strength – F_u	-	550
Ultimate displacement capacity – d_u	-	0.02
Residual strength	-	200

Quadrilinear asymmetric curve - quad_asm

This is a simple curve employed to model idealised quadriilinear asymmetric behaviour. An isotropic hardening rule is adopted.

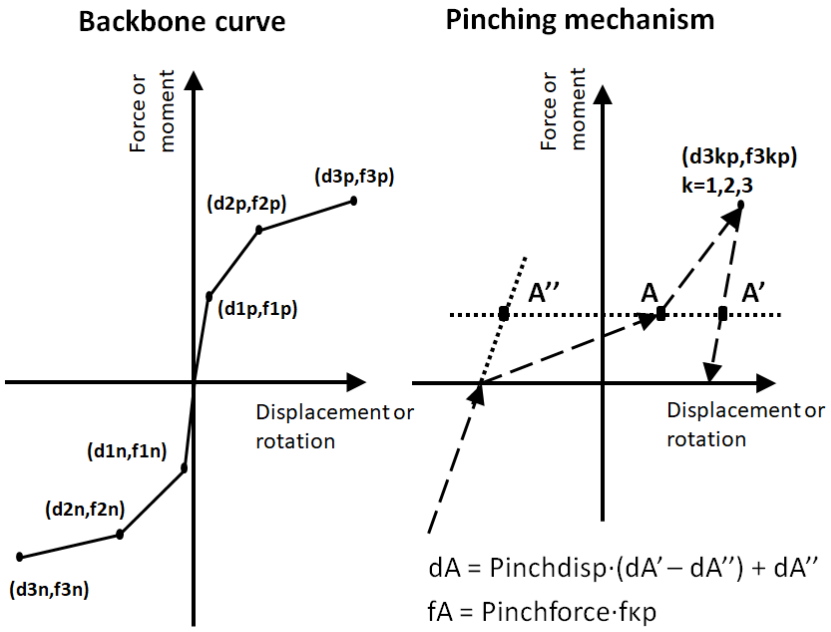


Ten parameters need to be defined in order to fully characterise this response curve:

Curve Properties	Typical values	Default values
Yielding strength for positive loading direction – $\mathbf{F_{y(+)}}$	-	500
Yielding displacement for positive loading direction – $\mathbf{d_{y(+)}}$	-	0.005
Ultimate strength for positive loading direction – $\mathbf{F_{u(+)}}$	-	550
Ultimate displacement capacity for positive loading direction – $\mathbf{d_{u(+)}}$	-	0.02
Residual strength for positive loading direction **(+)**	-	200
Yielding strength for negative loading direction – $\mathbf{F_{y(-)}}$	-	400 (-)
Yielding displacement for negative loading direction – $\mathbf{d_{y(-)}}$	-	0.005 (-)
Ultimate strength for negative loading direction – $\mathbf{F_{u(-)}}$	-	450 (-)
Ultimate displacement capacity for negative loading direction – $\mathbf{d_{u(-)}}$	-	0.02 (-)
Residual strength for negative loading direction **(-)**	-	200 (-)

Pinched asymmetric curve - pinched_asm

The Pinched_Asm curve is a uniaxial force-displacement model characterized by non-linear stiffness and strength deterioration rules and also by pinching of force and displacement. Deterioration because of damage due to ductility and energy may be added. The envelope curve of the model is defined by the user by supplying three force-displacement pairs for the positive branch and three force-displacement pairs for the negative branch of the envelope curve. Force and displacement must be given in ascending order for the positive branch and in descending order for the negative branch.



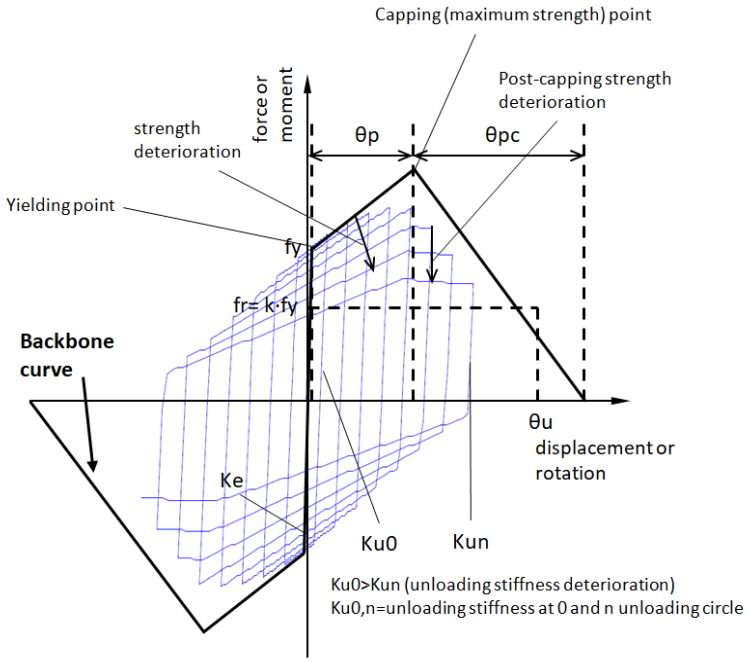
Seventeen parameters need to be defined in order to fully characterise this response curve:

Material Properties	Typical values	Default values
Displacement at first point of envelope in the positive direction– $\mathbf{d_{1p(+)}}$	-	0.001
Force at first point of envelope in the positive direction – $\mathbf{f_{1p(+)}}$	-	100.00
Displacement at second point of envelope in the positive direction– $\mathbf{d_{2p(+)}}$	-	0.115
Force at second point of envelope in the positive direction – $\mathbf{f_{2p(+)}}$	-	150
Displacement at third point of envelope in the positive direction– $\mathbf{d_{3p(+)}}$	-	0.2
Force at third point of envelope in the positive direction – $\mathbf{f_{3p(+)}}$	-	145.00
Displacement at first point of envelope in the negative direction– $\mathbf{d_{1n(-)}}$	-	-0.001
Force at first point of envelope in the negative direction – $\mathbf{f_{1n(-)}}$	-	-100.00
Displacement at second point of envelope in the negative direction– $\mathbf{d_{2n(-)}}$	-	-0.115
Force at second point of envelope in the negative direction – $\mathbf{f_{2n(-)}}$	-	-150.00
Displacement at third point of envelope in the negative direction– $\mathbf{d_{3n(-)}}$	-	-0.2
Force at third point of envelope in the negative direction – $\mathbf{f_{3n(-)}}$	-	-145.00
Pinching factor for displacement during reloading– **pinchdisp**	0-1.0(-)	0.8
Pinching factor for force during reloading– **pinchforce**	0-1.0(-)	0.2

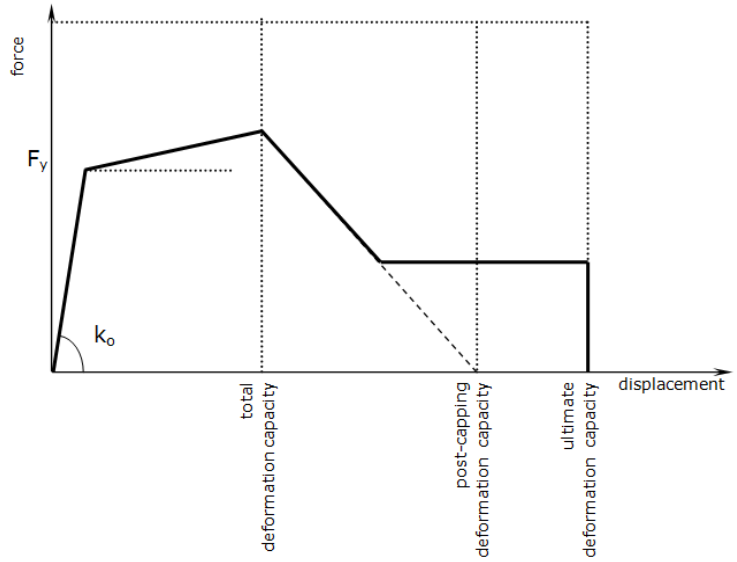
Material Properties	Typical values	Default values
Damage due to ductility– **dam1**	0-0.05(-)	0.0
Damage due to Energy– **dam2**	0-0.5(-)	0.0
Power used to determine the degraded unloading stiffness based on ductility – **mu**	0-0.5(-)	0.0

Modified Ibarra-Medina-Krawinkler Deterioration curve with Bilinear Hysteretic Response – MIMK_bilin

The Modified Ibarra-Medina Krawinkler (MIMK) Deterioration curve is based on the model initially proposed by Ibarra et al. [2005] and later modified by Lignos and Krawinkler [2011]. The MIMK is based on a backbone curve that represents the behavior for monotonic loading and establishes strength and deformation bounds, and a set of rules that define the basic characteristics of the hysteretic behavior between the bounds. Hysteresis is modeled by standard bilinear hysteretic rules with kinematic strain hardening. The model also includes three modes of cyclic deterioration namely: a. basic strength deterioration, b. post-capping strength deterioration and c. unloading/reloading stiffness deterioration. The model can be utilized to model the cyclic moment-rotation relationship in plastic hinge areas of beams and the model variables for the case of steel beams can be defined by empirical relationships deduced from experimental data, from more than 300 experiments, by Lignos and Krawinkler [2011]. The model can be applied to any force-deformation relationship, though it has initially been described in terms of moment and rotation quantities.



The capacity deformations are better described in the following graph:



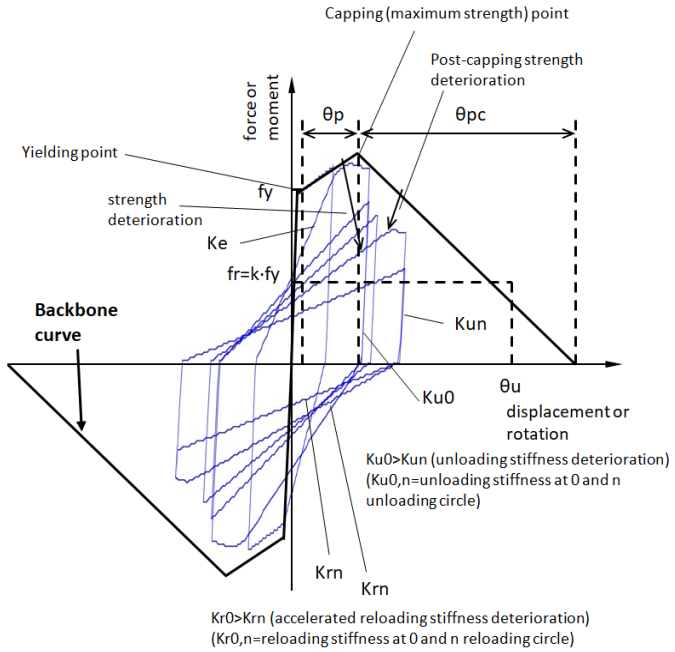
Twenty Two parameters need to be defined in order to fully characterise this response curve(an online tool for defining the model parameters in the case of steel beams can be found in http://dimitrios-lignos.research.mcgill.ca/databases/component/):

Curve Properties	Typical values (for steel beams)	Default values
Elastic Stiffness – **Ke**	-	200000
Effective yield strength for positive loading direction – $\mathbf{f_{y(+)}}$	-	300
Effective yield strength for negative loading direction – $\mathbf{f_{y(-)}}$	-	300
Plastic rotation capacity for positive loading direction – $\mathbf{\theta_{p(+)}}$	-	0.025
Plastic rotation capacity for negative loading direction – $\mathbf{\theta_{p(-)}}$	-	0.025
Post - capping rotation capacity for positive loading direction – $\mathbf{\theta_{pc(+)}}$	-	0.3
Post-capping rotation capacity for negative loading direction – $\mathbf{\theta_{pc(-)}}$	-	0.3
Ultimate rotation capacity for positive loading direction – $\mathbf{\theta_{u(+)}}$	-	0.4
Ultimate rotation capacity for negative loading direction – $\mathbf{\theta_{u(-)}}$	-	0.4
Residual Strength Ratio for positive loading direction – $\mathbf{k_{(+)}}$	0.0 - 0.5	0.3
Residual Strength Ratio for negative loading direction – $k_{(-)}$	0.0 - 0.5	0.3
Strain Hardening Ratio for positive loading direction – $\mathbf{a_{s(+)}}$	0.0 - 0.5	0.005
Strain Hardening Ratio for negative loading direction – $\mathbf{a_{s(-)}}$	0.0 - 0.5	0.005
Cyclic deterioration parameter for strength deterioration – Λ_s	0.0-1000.0 (Zero value leads to no deterioration)	1000

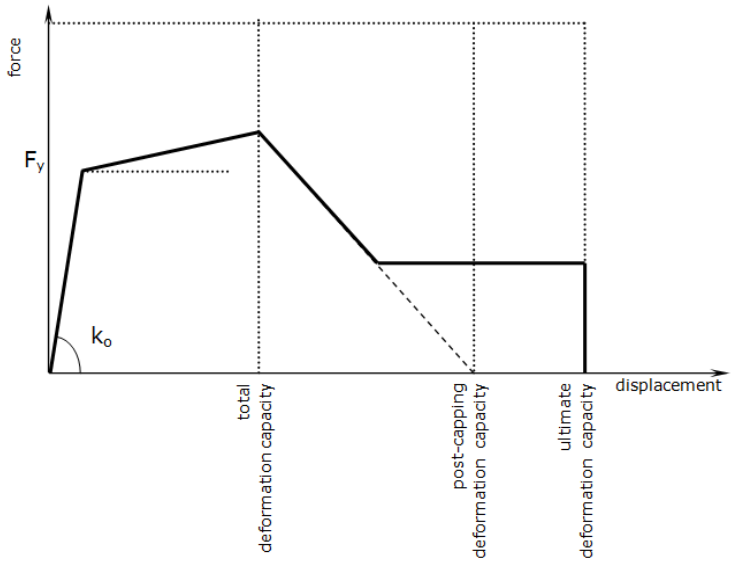
Curve Properties	Typical values (for steel beams)	Default values
Cyclic deterioration parameter for unloading stiffness deterioration – Λ_K	0.0-1000.0 (Zero value leads to no deterioration)	1000
Cyclic deterioration parameter for post-capping strength deterioration– Λ_c	0.0-1000.0 (Zero value leads to no deterioration)	1000
Strength deterioration rate – C_s	Usually equal to 1.0 (1.0-2.0)	1.0
unloading stiffness deterioration rate– C_K	Usually equal to 1.0 (1.0-2.0)	1.0
post-capping strength deterioration ratio– C_c	Usually equal to 1.0 (1.0-2.0)	1.0
Rate of cyclic deterioration in positive loading direction– $D_{(+)}$	$0 \leq D^+ \leq 1$	1.0
Rate of cyclic deterioration in negative loading direction– $D_{(-)}$	$0 \leq D^- \leq 1$	1.0
Factor for elastic stiffness amplification– N	Zero for no elastic stiffness amplification. If elastic Stiffness amplification is to be used a value of n around 1000 should be used for Concentrated Plastic Hinges according to Ribeiro et al. [2014]	0.0

Modified Ibarra-Medina-Krawinkler Deterioration Model with Peak-Oriented Hysteretic Response – MIMK_peak

The Modified Ibarra-Medina Krawinkler (MIMK) Deterioration curve with Peak-Oriented Hysteretic Response is based on the model initially proposed by Ibarra et al. [2005] and later modified by Lignos and Krawinkler [2009]. The model resembles to the Modified Ibarra-Medina-Krawinkler Deterioration Model with Bilinear Hysteretic Response since it is based on a backbone curve that represents the behavior for monotonic loading and establishes strength and deformation bounds, but uses a Peak-Oriented hysteretic Model to model hysteresis of the backbone curve. The Peak-Oriented hysteretic Model follows the rules proposed by Clough and Johnston [1966] and later modified by Mahin and Bertero [1976]. The model includes four modes of cyclic deterioration: a. basic strength deterioration, b. post-capping strength deterioration, c. unloading/reloading stiffness deterioration and d. accelerated reloading stiffness deterioration. Modified Ibarra-Medina Krawinkler Deterioration Model with Peak-Oriented Hysteretic Response is able to simulate the behaviour of reinforced concrete beams that primarily fail in a flexural mode and has been calibrated with more than 200 Reinforced Concrete beams by Lignos and Krawinkler [2012].



The capacity deformations are better described in the following graph:



Twenty four parameters need to be defined in order to fully characterise this response curve:

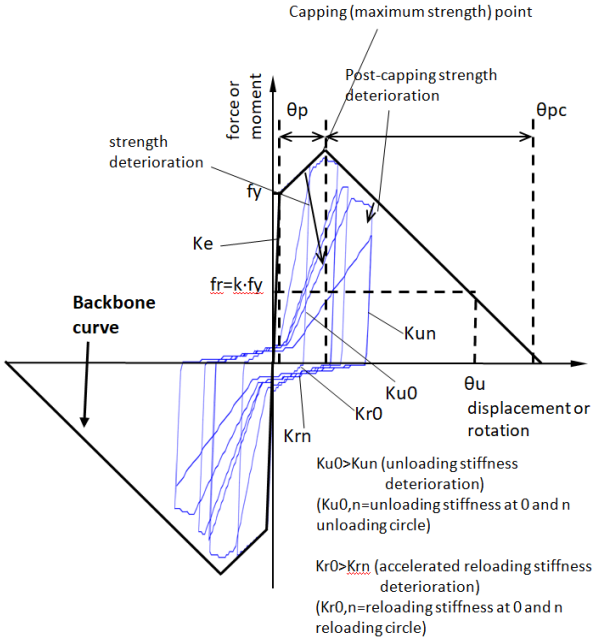
Curve Properties	Typical values (for steel beams)	Default values
Elastic Stiffness – **Ke**	-	200000
Effective yield strength for positive loading direction – $\mathbf{f_{y(+)}}$	-	300
Effective yield strength for negative loading direction – $\mathbf{f_{y(-)}}$	-	300
Plastic rotation capacity for positive loading direction – $\boldsymbol{\theta_{p(+)}}$	-	0.025

Curve Properties	Typical values (for steel beams)	Default values
Plastic rotation capacity for negative loading direction – $\theta_{p(-)}$	-	0.025
Post - capping rotation capacity for positive loading direction – $\theta_{pc(+)}$	-	0.3
Post-capping rotation capacity for negative loading direction – $\theta_{pc(-)}$	-	0.3
Ultimate rotation capacity for positive loading direction – $\theta_{u(+)}$	-	0.4
Ultimate rotation capacity for negative loading direction – $\theta_{u(-)}$	-	0.4
Residual Strength Ratio for positive loading direction – $k_{(+)}$	0.0 - 0.5	0.3
Residual Strength Ratio for negative loading direction – $k_{(-)}$	0.0 - 0.5	0.3
Strain Hardening Ratio for positive loading direction – $a_{s(+)}$	0.0 - 0.5	0.005
Strain Hardening Ratio for negative loading direction – $a_{s(-)}$	0.0 - 0.5	0.005
Cyclic deterioration parameter for strength deterioration – Λ_s	0.0-1000.0 (Zero value leads to no deterioration)	1000
Cyclic deterioration parameter for post-capping strength deterioration– Λ_c	0.0-1000.0 (Zero value leads to no deterioration)	1000
Cyclic deterioration parameter for accelerated reloading deterioration– Λ_a	0.0-1000.0 (Zero value leads to no deterioration)	1000
Cyclic deterioration parameter for unloading stiffness deterioration – Λ_K	0.0-1000.0 (Zero value leads to no deterioration)	1000
Strength deterioration rate – C_s	Usually equal to 1.0 (1.0-2.0)	1.0
Post-capping strength deterioration ratio– C_c	Usually equal to 1.0 (1.0-2.0)	1.0
Accelerated reloading deterioration ratio– C_a	Usually equal to 1.0 (1.0-2.0)	1.0
Unloading stiffness deterioration rate– C_K	Usually equal to 1.0 (1.0-2.0)	1.0
Rate of cyclic deterioration in positive loading direction– $D_{(+)}$	$0 \leq D^+ \leq 1$	1.0
Rate of cyclic deterioration in negative loading direction– $D_{(-)}$	$0 \leq D^- \leq 1$	1.0
Factor for elastic stiffness amplification– N	Zero for no elastic stiffness amplification. If elastic Stiffness amplification is to be used a value of n around 1000 should be used for Concentrated Plastic Hinges according to Ribeiro et al. [2014]	0.0

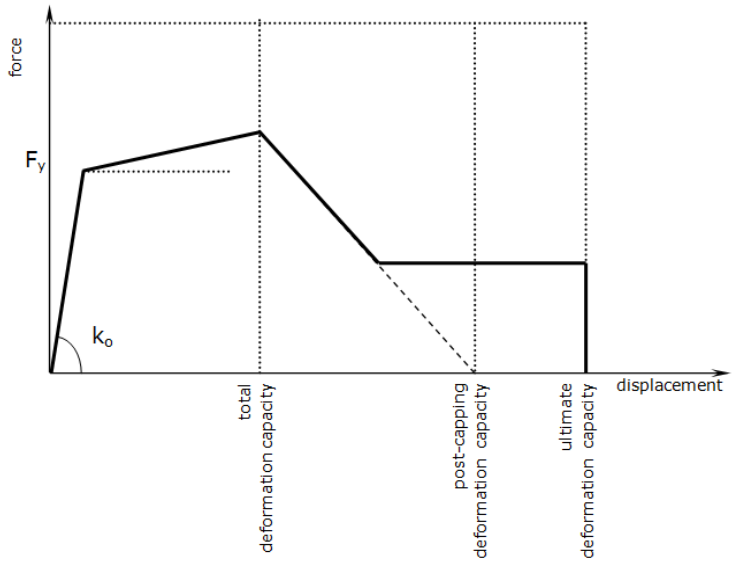
Modified Ibarra-Medina-Krawinkler Deterioration Model with Pinched Hysteretic Response – MIMK_Pinched

The Modified Ibarra-Medina Krawinkler (MIMK) Deterioration curve with Pinched Hysteretic Response is based on the model that was primarily proposed by Ibarra et al. [2005] and was later modified by Lignos and Krawinkler [2009]. The model resembles to the Modified Ibarra-Medina-Krawinkler

Deterioration Models with Bilinear and Peak-Oriented Hysteretic Response since it is based on a backbone curve that represents the behavior for monotonic loading and establishes strength and deformation bounds, but uses a Pinched hysteretic Model to model hysteresis of the backbone curve. The Pinched hysteretic is similar to the peak-oriented one, except that reloading consists of two parts. Initially the reloading path is directed towards a 'break point', which is a function of the maximum permanent deformation and the maximum load experienced in the direction of loading (Ibarra et al.[2005]. The model includes four modes of cyclic deterioration: a. basic strength deterioration, b. post-capping strength deterioration, c. unloading/reloading stiffness deterioration and d. accelerated reloading stiffness deterioration. Modified Ibarra-Medina Krawinkler Deterioration Model with Pinched Hysteretic Response is able to simulate the behaviour of reinforced concrete beams that primarily fail in a shear mode. This model is also able to simulate the hysteretic behaviour of shear connections, beam-to-column gusset plate connections and wooden components.



The capacity deformations are better described in the following graph:



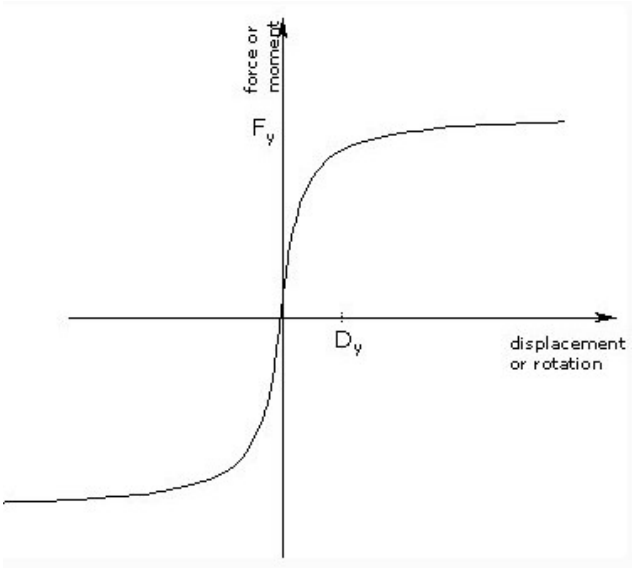
Twenty seven parameters need to be defined in order to fully characterise this response curve:

Curve Properties	Typical values (for steel beams)	Default values
Elastic Stiffness – **Ke**	-	200000
Effective yield strength for positive loading direction – $\mathbf{f_{y(+)}}$	-	300
Effective yield strength for negative loading direction – $\mathbf{f_{y(-)}}$	-	300
Plastic rotation capacity for positive loading direction – $\mathbf{\theta_{p(+)}}$	-	0.025
Plastic rotation capacity for negative loading direction – $\mathbf{\theta_{p(-)}}$	-	0.025
Post - capping rotation capacity for positive loading direction – $\mathbf{\theta_{pc(+)}}$	-	0.3
Post-capping rotation capacity for negative loading direction – $\mathbf{\theta_{pc(-)}}$	-	0.3
Ultimate rotation capacity for positive loading direction – $\mathbf{\theta_{u(+)}}$	-	0.4
Ultimate rotation capacity for negative loading direction – $\mathbf{\theta_{u(-)}}$	-	0.4
Residual Strength Ratio for positive loading direction – $\mathbf{k_{(+)}}$	0.0 - 0.5	0.3
Residual Strength Ratio for negative loading direction – $k_{(-)}$	0.0 - 0.5	0.3
Strain Hardening Ratio for positive loading direction – $\mathbf{a_{s(+)}}$	0.0 - 0.5	0.005
Strain Hardening Ratio for negative loading direction – $\mathbf{a_{s(-)}}$	0.0 - 0.5	0.005
Ratio between force at start of reloading to the force corresponding to the maximum experienced deformation for positive loading direction –$F_{pr(+)}$	0.0-1.0	0.2
Ratio between force at start of reloading to the force corresponding to the maximum experienced deformation for negative loading direction –$F_{pr(-)}$	0.0-1.0	0.2
Ratio of Reloading Stiffness– K_r	0.0-1.0	0.2
Cyclic deterioration parameter for strength deterioration – Λ_s	0.0-1000.0 (Zero value leads to no deterioration)	1000
Cyclic deterioration parameter for post-capping strength deterioration– Λ_c	0.0-1000.0 (Zero value leads to no deterioration)	1000
Cyclic deterioration parameter for accelerated reloading deterioration– Λ_a	0.0-1000.0 (Zero value leads to no deterioration)	1000
Cyclic deterioration parameter for unloading stiffness deterioration – Λ_K	0.0-1000.0 (Zero value leads to no deterioration)	1000
Strength deterioration rate – C_s	Usually equal to 1.0 (1.0-2.0)	1.0
Post-capping strength deterioration ratio– C_c	Usually equal to 1.0 (1.0-2.0)	1.0
Accelerated reloading deterioration ratio– C_a	Usually equal to 1.0 (1.0-2.0)	1.0
Unloading stiffness deterioration rate– C_K	Usually equal to 1.0 (1.0-2.0)	1.0
Rate of cyclic deterioration in positive loading direction– $D_{(+)}$	$0 \le D^+ \le 1$	1.0

Curve Properties	Typical values (for steel beams)	Default values
Rate of cyclic deterioration in negative loading direction– $D_{(-)}$	0≤D-≤1	1.0
Factor for elastic stiffness amplification– N	Zero for no elastic stiffness amplification. If elastic Stiffness amplification is to be used a value of n around 1000 should be used for Concentrated Plastic Hinges according to Ribeiro et al. [2014]	0.0

Nonlinear elastic curve – Non_lin_Elast

This hysteresis loop is a simplified version of the Ramberg-Osgood model, whereby no hysteretic dissipation is allowed (the same curve is employed for loading and unloading). It has been proposed and initially programmed by Otani [1981] for modelling of prestressed concrete elements.

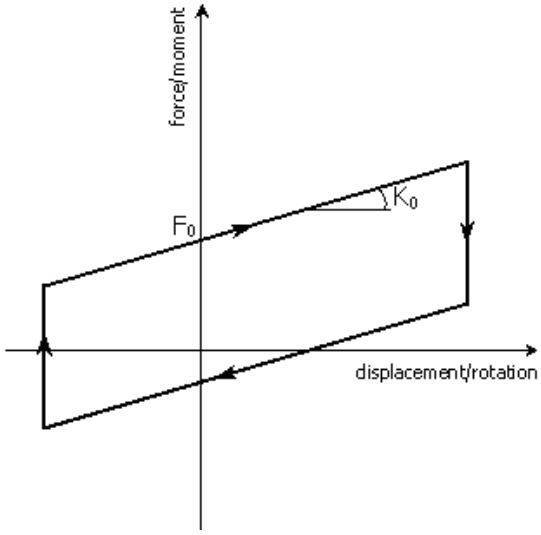


Four parameters need to be defined in order to fully characterise this response curve:

Curve Properties	Typical values	Default values
Yield strength – $\mathbf{F_y}$	-	500 (-)
Yield displacement – $\mathbf{D_y}$	-	0.0023 (-)
Ramberg-Osgood parameter – γ	-	5.5 (-)
Convergence limit for the Newton-Raphson procedure – β_1	-	0.001 (-)

Plastic curve - plst

This is a curve frequently employed to model idealised rigid-plastic behaviour, sliding bearings, FPS (friction pendulum system) isolating devices, hydraulic or lead-extrusion dampers, and so on. A kinematic hardening rule is adopted.



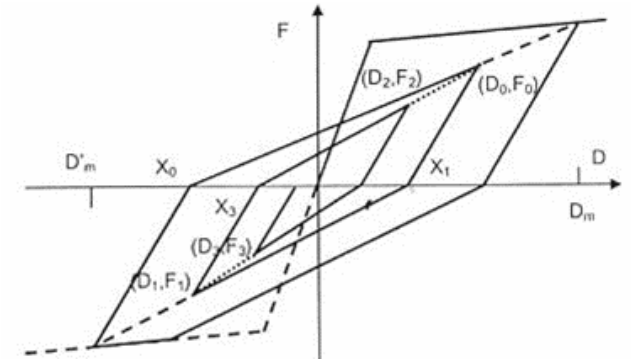
Two parameters need to be defined in order to fully characterise this response curve:

Curve Properties	Typical values	Default values
Initial force – **F_0**	-	10000 (-)
Post-yield stiffness – **K_0**	-	5 (-)

NOTE: Unloading and reloading stiffness is taken as infinite, which means that, if a sufficiently small analysis time-step is used, then the unloading/reloading branches of this response curve result practically vertical. With large time-steps, on the other hand, a finite unloading/reloading stiffness is obtained through the ratio $2F_0/\Delta t$.

Simplified bilinear Takeda curve - Takeda

This is the modified Takeda hysteresis loop described in Otani [1974], featuring however the unloading rules proposed by Emori and Schonobrich [1978]. Essentially, the model consists of a bilinear simplification of the original trilinear model proposed by Takeda et al. [1970], the inner cyclic rules of which were diverse from those proposed by Clough and Johnston [1966] in their original bilinear hysteresis model. This response curve has been initially programmed by Otani [1981].



Five parameters need to be defined in order to fully characterise its behaviour:

Curve Properties	Typical values	Default values
Yield strength – **F_y**	-	500 (-)
Initial stiffness – **K_y**	-	200000 (-)

Curve Properties	Typical values	Default values
Post-yielding to initial stiffness ratio – α	-	0.1 (-)
Outer loop stiffness degradation factor – β_0	-	0.4 (-)
Inner loop stiffness degradation factor – β_1	-	0.9 (-)

NOTE 1: The unloading stiffness from the post yielding curve in outer hysteresis loop is defined by:

$$K_{rout} = K_y \left(\frac{D_y}{D_m}\right)^{\beta_0}$$

where:
K_y is the initial stiffness;
D_y is the yielding displacement
D_m is the previous maximum displacement
β_0 is the outer loop stiffness degradation factor (K_{rout}), i.e. for unloading from the primary curve

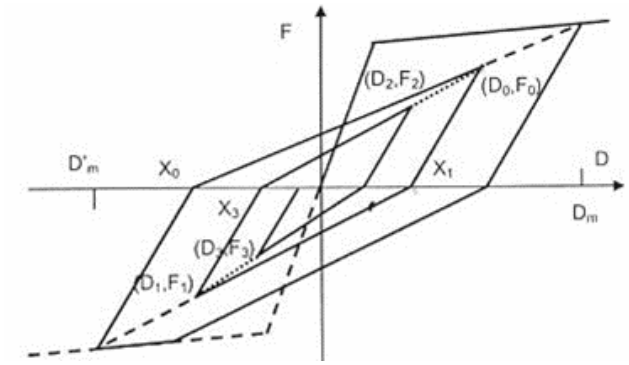
NOTE 2: The unloading stiffness in inner hysteresis loop is defined by:

$$K_{rin} = K_y \left(\frac{D_y}{D_m}\right)^{\beta_0} * \beta_1$$

where:
β_1 is the inner loop stiffness degradation factor ($K_{rin} = \beta_1 * K_{rout}$), i.e. for unloading and reloading in the inner cycles

Asymmetric bilinear Takeda curve - Takeda_asm

This is a model similar to the modified Takeda hysteresis loop described above, whereby different yield values may be specified for tension Fy(+) and compression Fy(-), so that to enable the modelling of asymmetric behaviour.



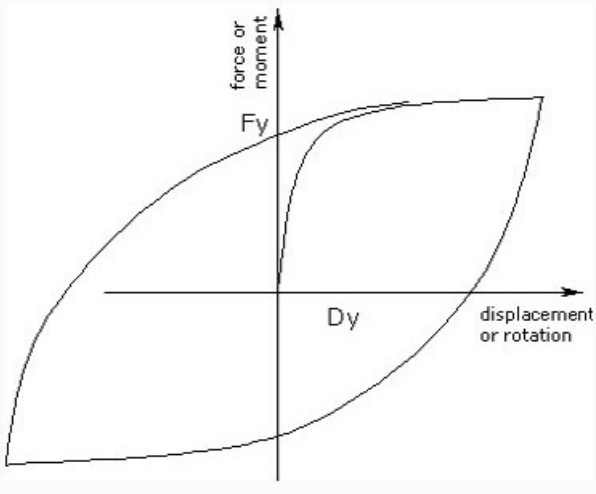
Six parameters need to be defined in order to fully characterise its behaviour:

Curve Properties	Typical values	Default values
Yield strength in positive region – $F_{y(+)}$	-	500 (-)
Yield strength in negative region – $F_{y(-)}$	-	400 (-)
Initial stiffness – K_y	-	200000 (-)
Post-yielding to initial stiffness ratio – α	-	0.1 (-)
Outer loop stiffness degradation factor – β_0	-	0.4 (-)

Curve Properties	Typical values	Default values
Inner loop stiffness degradation factor – β_1	-	0.9 (-)

Ramberg-Osgood curve - Ramberg_Osgood

This is the Ramberg-Osgood hysteresis loop [Ramberg and Osgood, 1943], as described in the work of Kaldjian [1967]. It has been initially programmed by Otani [1981].



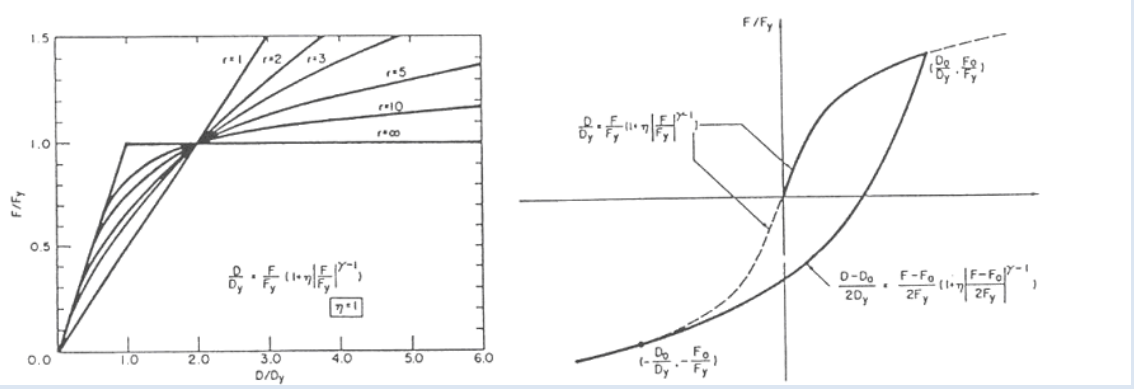
Four parameters need to be defined in order to fully characterise its behaviour:

Curve Properties	Typical values	Default values
Yield strength – F_y	-	500 (-)
Yield displacement – D_y	-	0.0025 (-)
Ramberg-Osgood parameter – γ	-	1.5 (-)
Convergence limit for the Newton-Raphson procedure – β_1	-	0.001 (-)

NOTE 1: The loading curve defined by:

$$\frac{D}{D_y} = \frac{F}{F_y}\left(1 + \left|\frac{F}{F_y}\right|^{\gamma-1}\right)$$

NOTE 2: The curve passes at (F_y, ($1+D_y$)) for any value of γ, which controls the shape of the primary curve. As shown below, the loading curve may vary from a linear elastic line for γ = 1.0, to an elasto-plastic bilinear segment for γ = infinity.



NOTE 3: The unloading curve from the maximum point (D_0, F_0) follows the equation:

$$\frac{D - D_0}{2D_y} = \frac{F - F_0}{2F_y}\left(1 + \left|\frac{F - F_0}{2F_y}\right|^{\gamma-1}\right)$$

NOTE 4: The force is computed by an iterative procedure using the Newton–Raphson method.

NOTE 5: As pointed out by Otani [1981] this hysteretic model dissipates energy even if the ductility factor is less than one. The dissipated energy is sensitive to γ, increasing with the increasing of this parameter.

Modified Richard-Abbott curve - Richard_Abbott

This is a modified Richard-Abbott hysteresis loop, programmed and implemented by Nogueiro et al. [2005a] based on the proposals of De Martino et al. [1984] and Della Corte et al. [2000], who in turn had built upon the original work of Richard and Abbott [1975]. The model is very flexible, being capable of modelling all sorts of steel and composite connections (e.g. welded-flange bolted-web connection, extended end-plate connection, flush end-plate connection, angle connection, etc.), for as long as the model parameters are calibrated accordingly, as demonstrated by Della Corte et al. [2000], Simoes et al. [2001] and Nogueiro et al. [2005a], amongst others.

Thirty parameters need to be defined in order to fully characterise this response curve.

For the ascending (positive) branches the corresponding input parameters are:

Curve Properties	Typical values	Default values
Initial stiffness for the upper bound curve – **K_a**	15000 - 50000 (kNm/rad)	12000 (kNm/rad)
Strength for the upper bound curve – **M_a**	75 - 250 (kNm)	45 (kNm)
Post-elastic stiffness for the upper bound curve – **K_{pa}**	0.02K_a - 0.05K_a	200 (kNm/rad)
Shape parameter for the upper bound curve – **N_a**	4 (-)	4 (-)

Curve Properties	Typical values	Default values
Initial stiffness for the lower bound curve – **K_{ap}**	K_a	12000 (kNm/rad)
Strength for the lower bound curve – **M_{ap}**	$0.45M_a$ - $0.65M_a$	5 (kNm)
Post-elastic stiffness for the lower bound curve – **K_{pap}**	K_{pa}	200 (kNm/rad)
Shape parameter for the lower bound curve – **N_{ap}**	4 (-)	4 (-)
Empirical parameter related to the pinching – **t_{1a}**	5 - 20 (-)	30 (-)
Empirical parameter related to the pinching – **t_{2a}**	0.15 - 0.5 (-)	0.03 (-)
Empirical parameter related to the pinching – **C_a**	1 (-)	1 (-)
Empirical coefficient related to the stiffness damage rate – **i_{Ka}**	3 - 25 (-)	0 (-)
Empirical coefficient related to the strength damage rate – **i_{Ma}**	0.01 - 0.1 (-)	0.03 (-)
Empirical coefficient defining the level of isotropic hardening – **H_a**	0.01 - 0.04 (-)	0.02 (-)
Maximum value of deformation reached in the loading history – **E_{maxa}**	0 - 0.2 (rad)	0.5 (rad)

NOTE: If a symmetric behaviour is sought, the second set of 15 parameters is identical to the first half.

For the descending (negative) branches the corresponding input parameters are:

Curve Properties	Typical values	Default values
Initial stiffness for the upper bound curve – **K_d**	15000 - 50000 (kNm/rad)	12000 (kNm/rad)
Strength for the upper bound curve – **M_d**	75 - 250 (kNm)	45 (kNm)
Post-elastic stiffness for the upper bound curve – **K_{pd}**	$0.02K_d$ - $0.05K_d$	200 (kNm/rad)
Shape parameter for the upper bound curve – **N_d**	4 (-)	4 (-)
Initial stiffness for the lower bound curve – **K_{dp}**	K_d	12000 (kNm/rad)
Strength for the lower bound curve – **M_{dp}**	$0.45M_d$ - $0.65M_d$	5 (kNm)
Post-elastic stiffness for the lower bound curve – **K_{pdp}**	K_{pd}	200 (kNm/rad)
Shape parameter for the lower bound curve – **N_{dp}**	4 (-)	4 (-)
Empirical parameter related to the pinching – **t_{1d}**	5 - 20 (-)	30 (-)
Empirical parameter related to the pinching – **t_{2d}**	0.15 - 0.5 (-)	0.03 (-)
Empirical parameter related to the pinching – **C_d**	1 (-)	1 (-)

Curve Properties	Typical values	Default values
Empirical coefficient related to the stiffness damage rate – i_{Kd}	3 - 25 (-)	0 (-)
Empirical coefficient related to the strength damage rate – i_{Md}	0.01 - 0.1 (-)	0.03 (-)
Empirical coefficient defining the level of isotropic hardening – H_d	0.01 - 0.04 (-)	0.02 (-)
Maximum value of deformation reached in the loading history – E_{maxd}	0 - 0.2 (rad)	0.5 (rad)

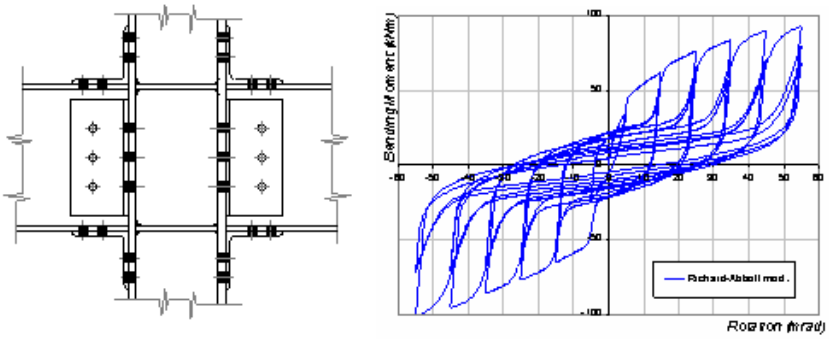
Below, example applications extracted from the work of Nogueiro et al. [2005a] are given, in order to illustrate the modelling capacities of this response curve:

NOTE: In the Steel Connection below some parameters assume non-typical values.

K_a=25000
M_a=85
K_{pa}=750
n_a=0.6
K_{ap}=25000
M_{ap}=45
K_{pap}=250
n_{ap}=0.6
t_{1a}=50
t_{2a}=0.5
C_a=1
i_{Ka}=0.000001
i_{Ma}=0.03
H_a=0.0000001
E_{maxa}=0.1

K_d=25000
M_d=85
K_{pd}=750
n_d=0.6
K_{dp}=25000
M_{dp}=45
K_{pdp}=250
n_{dp}=0.6
t_{1d}=50
t_{2d}=0.5
C_d=1
i_{Kd}=0.000001
i_{Md}=0.03
H_d=0.0000001
E_{maxd}=0.1

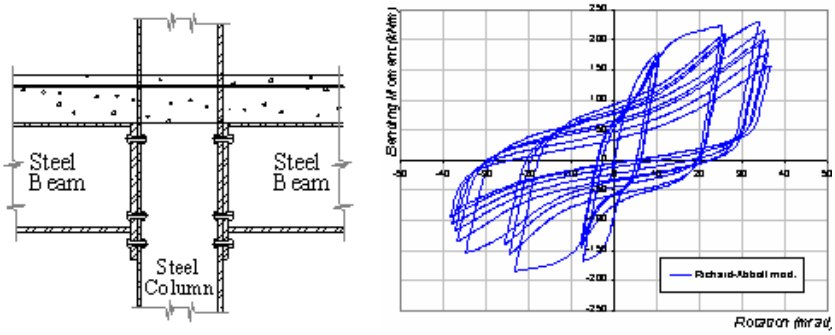
Steel Connection



K_a=36950
M_a=200
K_{pa}=1025
n_a=2.0
K_{ap}=36950
M_{ap}=120
K_{pap}=1025
n_{ap}=1.0
t_{1a}=10
t_{2a}=0.5
C_a=1
i_{Ka}=20
i_{Ma}=0.05
H_a=0.04
E_{maxa}=0.1

K_d=40830
M_d=170
K_{pd}=1020
n_d=2.0
K_{dp}=40830
M_{dp}=75
K_{pdp}=1020
n_{dp}=1.0
t_{1d}=15
t_{2d}=0.5
C_d=1
i_{Kd}=20
i_{Md}=0.08
H_d=0.0000001
E_{maxd}=0.1

Steel-Concrete Connection

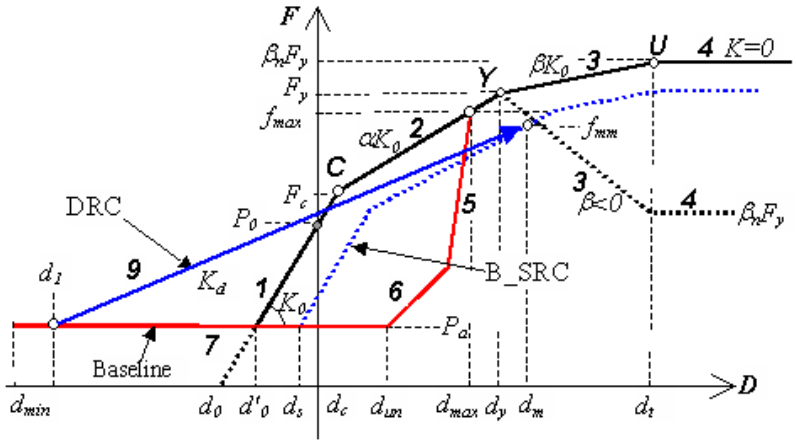


Soil-structure interaction curve - ssi_py

IMPORTANT: This versatile hysteretic model is still being tested and further developed. For instance, currently this curve caters for the normal force-displacement direction only (i.e. it does not account for the tangential force-slip response). In addition, or perhaps in tandem, the DOFs are not fully coupled (a limitation that is also a consequence of the currently uncoupled nature of the link elements in SeismoStruct). It is envisaged that both of these issues will be addressed in future releases of SeismoStruct.

This is a nonlinear dynamic soil-structure interaction (SSI) model, developed and implemented by Allotey and El Naggar [2005a; 2005b], adequate for analysing footings, retaining walls and piles under different loading regimes (the nomenclature chosen for this curve puts in evidence the fact that this model can be used to carry out lateral pile analyses, where p-y curves are commonly employed). It accounts for gap formation with the option of considering soil cave-in, it features cyclic hardening/degradation under variable-amplitude loading, and it can model responses that are bounded or unbounded within their initial backbone curves.

Cyclic degradation/hardening due to pore pressure and volumetric changes is accounted for through the use of elliptical damage functions implemented within the framework of a modified rainflow counting algorithm [Anthes, 1997]; the equivalent number of cycles approach [e.g. Seed et al. 1975; Annaki and Lee, 1977] is also used. The effect of soil cave-in is, on the other hand, modelled using an empirically developed hyperbolic function.



Evidently, this hysteretic model, on its own, is not sufficient to model a given foundation system. Instead, a series of springs (i.e. link elements) featuring an appropriately calibrated ssi_py curve must be used, normally in association with a beam-column element, in order to model whatever foundation system the user needs to represent. In other words, this response curve is to be employed within the realms of beam-on-a-nonlinear Winkler foundation (BNWF) model, whereby a number of spring elements are used under the foundation and the response curves have to be given for each. For a footing, the parameters are the same for all springs, whilst for a pile or retaining wall, since the overburden increases with depth, the parameters change with depth.

In addition, viscous damping may be assigned to the link element whenever the user wishes to somehow account for radiation damping effects (this will be similar to the introduction of a dashpot element parallel to the soil spring). Users may refer to the literature [e.g. Wolf, 1994; Allotey and El Naggar, 2005b] for indications on how to compute appropriate values of damping, as a function of the vibration characteristics of the soil-structure system. Commonly, if the vibration period of the soil-structure system is below that of the site, then the effects of radiation damping may be considered as negligible.

Nineteen parameters need to be defined in order to fully characterise this response curve:

Curve Properties	Typical values	Default values
Initial stiffness – $\mathbf{K_0}$	-	10000 (kNm/rad)
Soil strength ratio at first turning point – $\mathbf{F_c}$	0 - 1 (-)	0.5 (-)
Yielding soil strength – $\mathbf{F_y}$	-	100 (kNm)
Initial force ratio at zero displacement – $\mathbf{P_0}$	0 - 0.9	0 (-)
Minimum force ratio at baseline – $\mathbf{P_a}$ – OR Side-shear force factor ratio – $\mathbf{f_s}$	$0 \le P_a \le P_0$; $P_a \le \beta_n F_y$; $P_a \le F_c$ $0 \le f_s \le 0.9$	0 (-)
Stiffness ratio after first turning point – $\boldsymbol{\alpha}$	0.001 - 1 (-)	0.5 (-)

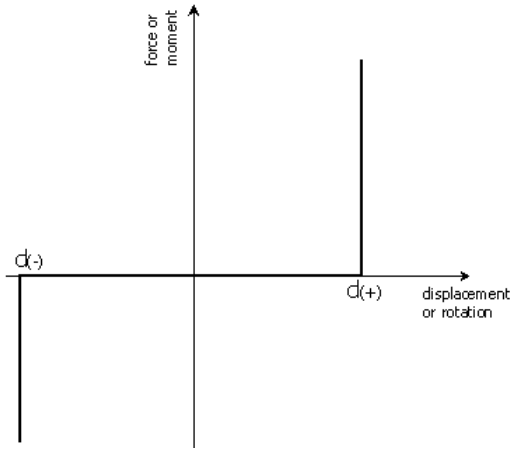
Curve Properties	Typical values	Default values
Unloading stiffness factor – α_N	-	1 (-)
Yielding stiffness ratio – β	-	0 (-)
Ultimate soil strength – β_N	-	1 (-)
Flag settings combination indicator – **Flg**	See Help System	31 (-)
DRC starting stiffness ratio – e_{p1}		1 (-)
Gap force parameter – p_1		1 (-)
Soil cave-in parameter – p_2		0 (-)
Stiffness degradation/hardening parameter – p_k		1 (-)
Stiffness degradation/hardening parameter – e_k		1 (-)
Strength degradation/hardening parameter – p_s		1 (-)
Strength degradation/hardening parameter – e_s		1 (-)
Slope of the S-N curve – k_s		0.1 (-)
Soil stress corresponding to point S1 in S-N curve – f_0		200 (-)

NOTE 1: Future releases of SeismoStruct are also likely to introduce a significantly more user-friendly way of calibrating/adjusting the parameters of this response curve, using drop-down menus and/or radio buttons to select the different modelling options.

NOTE 2: In recent years, an alternative approach to the modelling of foundation systems, consisting in the employment of a fully-coupled V-H-M (vertical-horizontal-rotation) macro-model has been proposed [e.g. Cremer at al, 2002]. It uses just one element to model the whole footing response and is based on a plasticity-type yield surface formulation. Although this is certainly a promising approach to SSI modelling, it is felt that, given the current state of development and practice, the more traditional BNWF procedure, currently implemented in SeismoStruct through the employment of the powerful ssi_py response curve, provides users with all the facilities required for an adequate modelling of the static, and above all dynamic, interaction between soils, foundations and structures.

Gap/Hook curve - gap_hk

This is a curve employed to model structural gapping/pounding, expansion joints, deck restrainers, and so on.



Four parameters need to be defined:

Curve Properties	Typical values	Default values
Locking/Engaging displacement in positive region – **d_+**	-	5 (-)
Gap/Hook stiffness in positive region – **K_+**	-	1.00E+012 (-)
Locking/Engaging displacement in negative region – **d_-**	-	-5 (-)
Gap/Hook stiffness in negative region – **K_-**	-	1.00E+012 (-)

NOTE: Stiffness values K_+ and K_- must be positive.

Multi-linear curve – multi_lin

This is the polygonal hysteresis loop, as described in the work of Sivaselvan and Reinhorn [1999]. The model can simulate the deteriorating behaviour of strength, stiffness, and bond slip. Sixteen parameters need to be defined in order to fully characterise this response curve. There are two groups of parameters: common parameters (the same as for the smooth curve), related to the backbone curve, and then specific parameters for the hysteretic rules.

Sixteen parameters need to be defined:

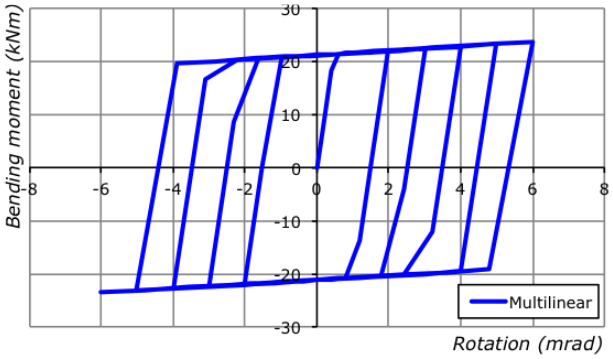
Curve Properties	Typical values	Default values
Initial flexural rigidity – **EI**	-	45400 (-)
Cracking moment (positive) – **PCP**	-	10 (-)
Yield moment (positive) – **PYP**	-	22 (-)
Yield curvature (positive) – **UYP**	-	0.002 (-)
Ultimate curvature (positive) – **UUP**	-	0.006 (-)
Post-Yield flexural stiffness (positive) as ratio of elastic **EI**	-	0.0088 (-)
Cracking moment (negative) – **PCN**	-	-10 (-)
Yield moment (negative) – **PYN**	-	-22 (-)
Yield curvature (negative) – **UYN**	-	-0.002 (-)
Ultimate curvature (negative) – **UUN**	-	-0.006 (-)
Post-Yield flexural stiffness (negative) as ratio of elastic **EI**	-	0.0088 (-)
Stiffness degrading parameter – **HC**	-	200 (-)

Curve Properties	Typical values	Default values
Ductility-based strength decay parameter – **HBD**	-	0.001 (-)
Hysteretic energy-based strength decay parameter – **HBE**	-	0.001 (-)
Slip parameter – **HS**	-	1 (-)
Model parameter. 0 for trilinear model, 1 for bilinear model, 2 for Vertex-oriented model	-	0 (-)

Below, example applications are given, in order to illustrate the modelling capacities of this response curve (it is noted that the 'bordered' parameters have been changed with respect to the default values):

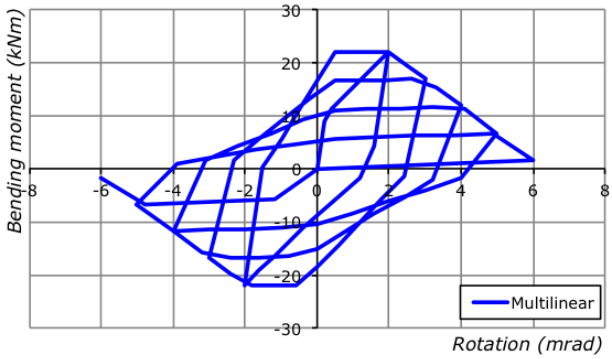
EI=45400
PCP=10
PYP=22
UYP=0.002
UUP=1
EI3P=0.0088
PCN=-10
PYN=-22
UYN=-0.002
UUN=-1
EI3N=0.0088

HC=200
HBD=0.001
HBE=0.001
HS=1
IBILINEAR=1



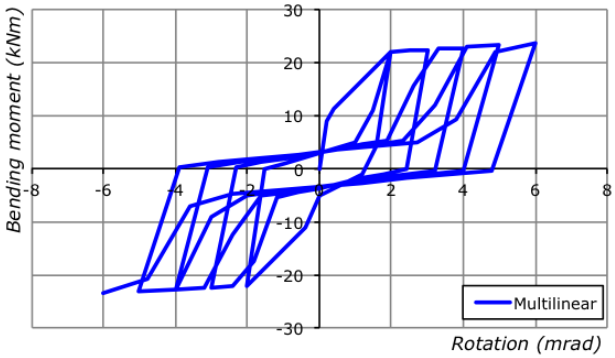
EI=45400
PCP=10
PYP=22
UYP=0.002
UUP=0.006
EI3P=0.0088
PCN=-10
PYN=-22
UYN=-0.002
UUN=-0.006
EI3N=0.0088

HC=200
HBD=1
HBE=0.001
HS=1
IBILINEAR=0



EI=45400
PCP=10
PYP=22
UYP=0.002
UUP=1
EI3P=0.0088
PCN=-10
PYN=-22
UYN=-0.002
UUN=-1
EI3N=0.0088

HC=200
HBD=0.001
HBE=0.001
HS=0.25
IBILINEAR=0



Smooth curve – smooth

This is the smooth hysteresis loop, as described in Sivaselvan and Reinhorn [1999] and Sivaselvan and Reinhorn [2001]. The model is a variation of that originally proposed by Bouc [1967] and modified by several others (Wen [1976], Baber and Noori [1985], Casciati [1989] and Reinhorn et al. [1995]). It has been formulated with rules for stiffness and strength degradation, and pinching. Twenty-two

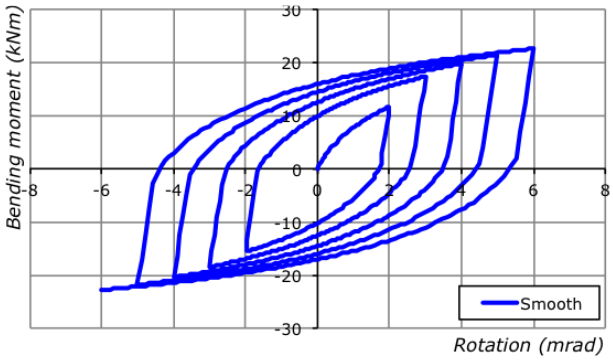
parameters need to be defined in order to fully characterise this response curve. There are two groups of parameters: common parameters (the same as for the multilinear curve), related to the backbone curve, and then specific parameters for the hysteretic rules.

Twenty-two parameters need to be defined:

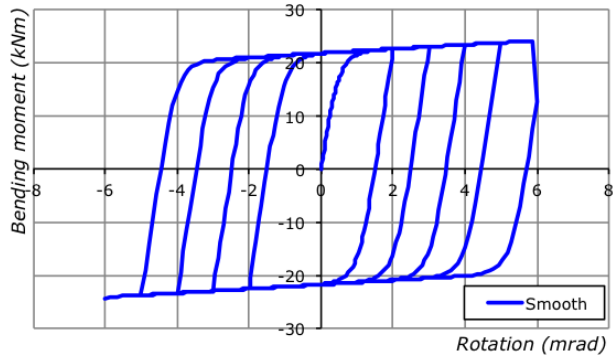
Curve Properties	Typical values	Default values
Initial flexural rigidity – **EI**	-	45400 (-)
Cracking moment (positive) – **PCP**	-	10 (-)
Yield moment (positive) – **PYP**	-	22 (-)
Yield curvature (positive) – **UYP**	-	0.002 (-)
Ultimate curvature (positive) – **UUP**	-	0.006 (-)
Post-Yield flexural stiffness (positive) as % of elastic **EI3P**	-	0.0088 (-)
Cracking moment (negative) – **PCN**	-	-10 (-)
Yield moment (negative) – **PYN**	-	-22 (-)
Yield curvature (negative) – **UYN**	-	-0.002 (-)
Ultimate curvature (negative) – **UUN**	-	-0.006 (-)
Post-Yield flexural stiffness (negative) as % of elastic **EI3N**	-	0.0088 (-)
Stiffness degrading parameter – **HC**	-	200 (-)
Ductility-based strength decay parameter – **HBD**	-	0.001 (-)
Hysteretic energy-based strength decay parameter – **HBE**	-	0.001 (-)
Smoothness parameter for elastic-yield transition – **NTRANS**	-	10 (-)
Parameter for shape of unloading – **ETA**		0.5 (-)
Slip length parameter – **HSR**		0 (-)
Slip sharpness parameter – **HSS**		100 (-)
Parameter for mean moment level of slip – **HSM**		0 (-)
Exponent of gap closing spring – **NGAP**	-	10 (-)
Gap closing curvature parameter – **PHIGAP**		1000 (-)
Gap closing stiffness coefficient – **STIFFGAP**	-	1 (-)

Below, example applications are given, in order to illustrate the modelling capacities of this response curve (it is noted that the 'bordered' parameters have been changed with respect to the default values):

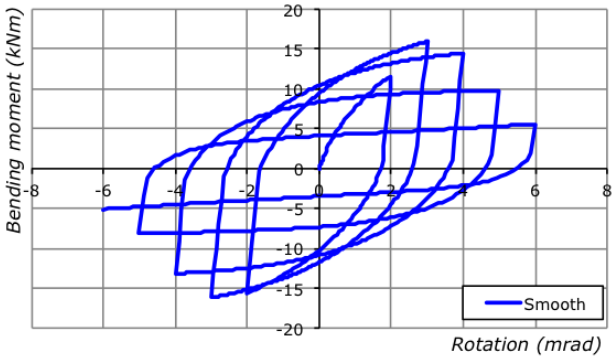
EI=45400	**HC**=200
PCP=10	**HBD**=0.001
PYP=22	**HBE**=0.001
UYP=0.002	**NTRANS**=0.1
UUP=0.006	**ETA**=0.5
EI3P=0.0088	**HSR**=0
PCN=-10	**HSS**=100
PYN=-22	**HSM**=0
UYN=-0.002	**NGAP**=10
UUN=-0.006	**PHIGAP**=1000
EI3N=0.0088	**STIFFGAP**=1



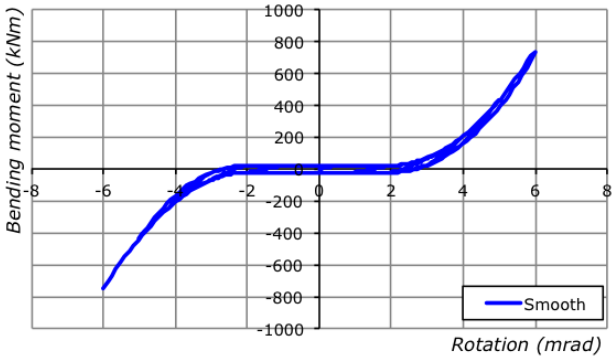
EI=45400	**HC**=200
PCP=10	**HBD**=0.001
PYP=22	**HBE**=0.001
UYP=0.002	**NTRANS**=2
UUP=0.006	**ETA**=0.5
EI3P=0.0088	**HSR**=0
PCN=-10	**HSS**=100
PYN=-22	**HSM**=0
UYN=-0.002	**NGAP**=10
UUN=-0.006	**PHIGAP**=1000
EI3N=0.0088	**STIFFGAP**=1



EI=45400	**HC**=200
PCP=10	**HBD**=0.001
PYP=22	**HBE**=0.5
UYP=0.002	**NTRANS**=0.1
UUP=0.006	**ETA**=0.5
EI3P=0.0088	**HSR**=0
PCN=-10	**HSS**=100
PYN=-22	**HSM**=0
UYN=-0.002	**NGAP**=10
UUN=-0.006	**PHIGAP**=1000
EI3N=0.0088	**STIFFGAP**=1

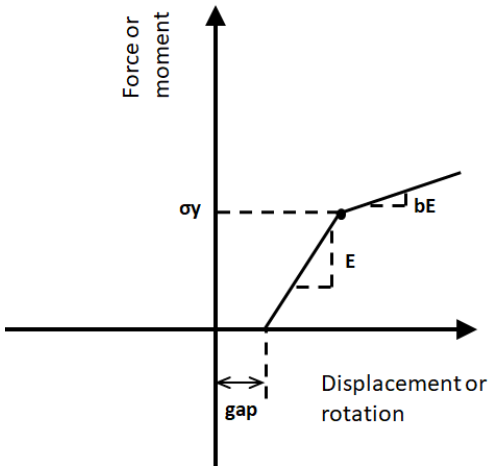


EI=45400	**HC**=200
PCP=10	**HBD**=0.001
PYP=22	**HBE**=0.001
UYP=0.002	**NTRANS**=0.5
UUP=0.006	**ETA**=0.5
EI3P=0.0088	**HSR**=0
PCN=-10	**HSS**=100
PYN=-22	**HSM**=0
UYN=-0.002	**NGAP**=2
UUN=-0.006	**PHIGAP**=0.002
EI3N=0.0088	**STIFFGAP**=2



Elastic – Perfectly plastic Gap curve - Gap_elpl

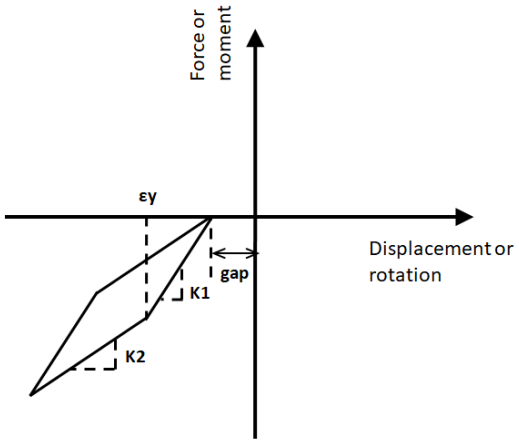
The Gap_elpl curve is characterized by zero stress until the deformation surpases the value of the initial gap and a perfectly elastic behaviour afterwards until the yielding point. After the yielding point the curve enters the plastic region. Kinematic hardening is also available. Four parameters must be defined in order to fully describe the mechanical characteristics of the curve:



Material Properties	Typical values	Default values
Modulus of Elasticity – **E**	-	100 (-)
Yield Stress – **σy**	-	200 (-)
Gap length	-	0.4 (-)
Post yielding to initial stiffness ratio	-	0.1(-)

Impact response curve – pound_hz

The Impact response curve is described by Muthukumar and DesRoches [2006] and is based on the Herz Law. The model can be used for modeling impact and pounding events between oscillating structures. The model is characterised by compressive stress after the closing of the initial gap between the two members included in the pounding event. Four parameters have to be defined for the model:

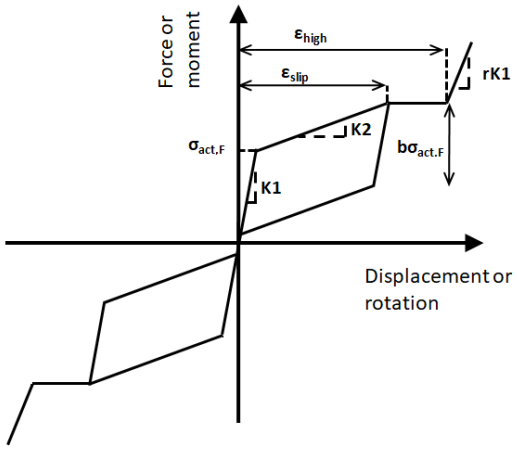


Material Properties	Typical values	Default values
Initial Stiffness – **K1**	-	30(-)
Post Yield Stiffness – **K2**	-	2.6 (-)
Yield displacement– **εy**	-	0.02 (-)
Gap length– **gap**	-	0.005 (-)

Self Centering Brace response curve- SCB

The Self Centering Brace (SCB) response curve is a uniaxial response curve modeling the behaviour of Self Centering Energy dissipating Braces. The model is described by Christopoulos et al. (2008). The SCB model resembles ordinary Flag-shaped models giving also the opportunity to incude two additional

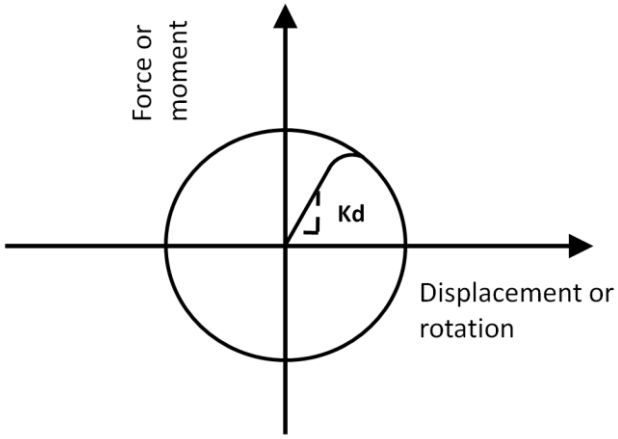
characteristics consisting of a non-recoverable slip of an external friction fuse available by the brace structure, and a steep increase of the stiffness of the brace at high deformations. In total seven parameters have to be defined for the model:



Material Properties	Typical values	Default values
Initial Stiffness – **K1**	100000 – 2000000 (kN/m)	200000 (kN/m)
Post-Yield Stiffness – **K2**	10000- 50000 (kN/m)	12000 (kN/m)
Activation Force– $\sigma_{act,F}$	200 – 2000 (kN)	200 (kN)
Ratio of Forward to Reverse activation force – **b**	-	0.5
Slip displacement – $\varepsilon_{,slip}$	0-0.01(m)	0 (m)
Displacement at the start of stiffness increase – $\varepsilon_{,high}$	-	0.3 (m)
Ratio of Increased Stiffness to Initial Stiffness – **r**	1.5-2	1.5

Viscous Damper – vsc_dmp

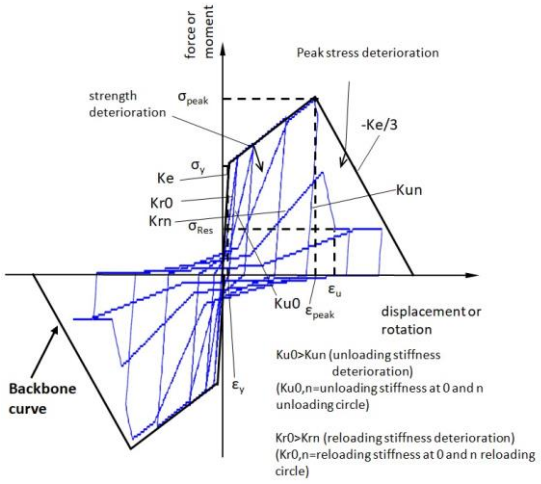
The Viscous Damper (vsc_dmp) response curve is a uniaxial response curve modeling the behaviour of a NonLinear Viscous Damper. The Viscous Damper Model consists of a linear spring and a dashpot connected in series and is based on the fractional-derivative Maxwel model described by Makris and Constantinou [1991]. In total four parameters need to be defined for the model:



Material Properties	Typical values	Default values
Initial Stiffness – K_d	1000 – 400000 (kN/m)	200000 (kN/m)
Damping Constant – C_d	20-5000 (kN/(m/s)$^{\alpha}$)	1200 (kN/(m/s)$^{\alpha}$)
Velocity Exponent – α	0 - 1	0.4
Pin Tolerance – **s**	-	0 (m)

Generic Hysteretic Curve– Gen_Hyst

Gen_Hyst is a generic uniaxial response curve characterised by Pinching effect (controlled by a pinching factor with values from 0 for no pinching to 1 for high pinching) and by four deterioration modes including strength deterioration, peak stress deterioration, reloading and unloading stiffness deterioration. All deterioration modes are controlled by the deterioration factor (with values from 0 for no deterioration to 1 for high deterioration). An initial backbone curve consisting of the yielding point, the peak stress point and the reidual strength is initialy defined and deteriorates after each unloading incident.

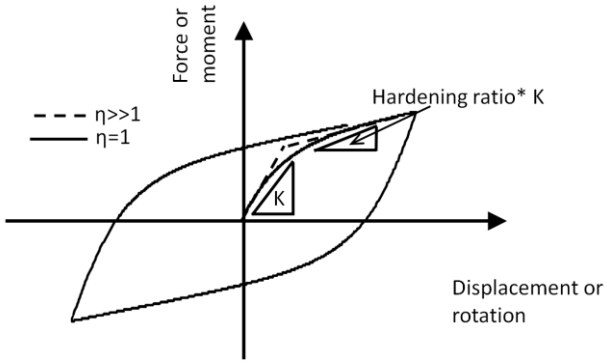


Twelve parameters are needed in order to describe the mechanical characteristics of the material. Pairs of stress strain defining the backbone curve must be given in ascending order

Material Properties	Typical values	Default values
Elastic Stiffness – $\mathbf{K_e}$	-	12000 (-)
Yield Stress in positive direction – $\boldsymbol{\sigma_{y,pos}}$	-	300 (-)
Yield Stress in negative direction – $\boldsymbol{\sigma_{y,neg}}$	-	300 (-)
Peak Strain in positive direction – $\boldsymbol{\varepsilon_{peak,pos}}$	-	0.3 (-)
Peak Stress in positive direction – $\boldsymbol{\sigma_{peak,pos}}$	-	500 (-)
Peak Strain in negative direction – $\boldsymbol{\varepsilon_{peak,neg}}$	-	0.3 (-)
Peak Stress in negative direction – $\boldsymbol{\sigma_{peak,neg}}$	-	500 (-)
Residual Strength in positive direction – $\boldsymbol{\sigma_{Res,pos}}$	-	120 (-)
Residual Strength in negative direction – $\boldsymbol{\sigma_{Res,pos}}$	-	120 (-)
Pinching factor – **FPinch**	0-1	0.5 (-)
Deterioration factor – **FDet**	0-1	0 (-)

Bouc Wen Curve – Bouc_Wen

The Bouc_Wen curve follows the hysteresis model proposed by Bouc [1967] and modified by Wen [1976]. In total five marameters need to be defined for the model, including the Initial curve stiffness and the hardening ratio, parameter η controlling the smoothness of the curve, parameter A which controlos the hysteresis amplitude and parameters β and γ wich control the shape of the hysteresis curve.



Material Properties	Typical values	Default values
Hardening Ratio	-	0.2(-)
Initial Stiffness (K)	-	210000 (-)
η – parameter controlling transisiton from elastic to plastic behaviour	-	1.00
γ – parameter affecting the shape of the hysteresis loop	-	0.50
β - parameter affecting the shape of the hysteresis loop	-	0.50
A – parameter controlling the hysteresis amplitude	-	1.00

Appendix H – Codes

Appendix H1 - EUROCODES

In this appendix the parameters used for the structures assessment according to the Eurocodes (EC8-Part1 and Part3) are presented.

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.

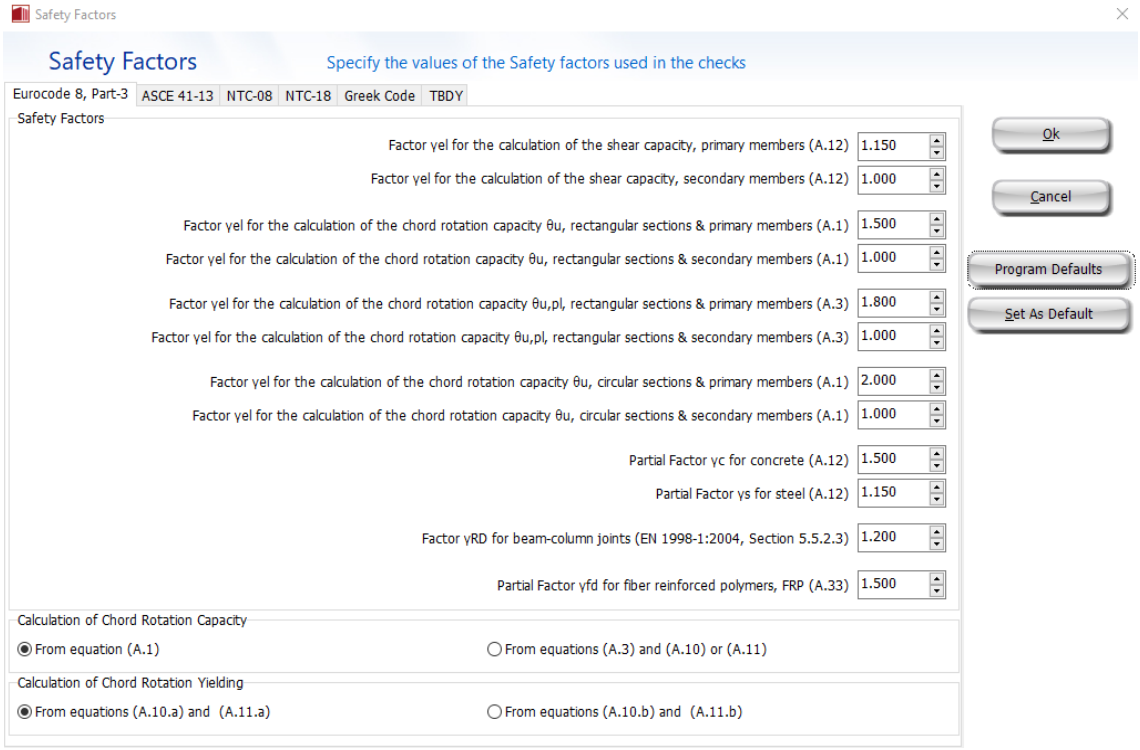
Knowledge Level	Geometry	Details	Materials	Analysis	CF
KL1	From original outline construction drawings with sample **visual** survey **or** from **full** survey	Simulated design in accordance with relevant practice **and** from **limited** in-situ inspection	Default values in accordance with standards 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).

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):

$$\theta_{um} = \frac{1}{\gamma_{el}} \cdot 0{,}016 \cdot (0{,}3^{\nu}) \left[\frac{\max(0{,}01;\omega')}{\max(0{,}01;\omega)} f_c\right]^{0{,}225} \cdot \left(\min(9, \frac{L_V}{h})\right)^{0{,}35} 25^{\left(\alpha\rho_{sx}\frac{f_{yw}}{f_c}\right)} (1{,}25^{100\rho_d})$$

(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:

$$\theta_{um}^{pl} = \theta_{um} - \theta_y$$
$$= \frac{1}{\gamma_{el}} \cdot 0{,}0145 \cdot (0{,}25^{\nu}) \left[\frac{\max(0{,}01;\omega')}{\max(0{,}01;\omega)}\right]^{0{,}3} \cdot {f_c}^{0{,}2} \cdot \left(\min\left(9, \frac{L_V}{h}\right)\right)^{0{,}35} 25^{\left(\alpha\rho_{sx}\frac{f_{yw}}{f_c}\right)} (1{,}275^{100\rho_d})$$

(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 rectangular 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 rectangular 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 non-linearities 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 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 SeismoStruct the following equations 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:

$M_y^* = M_y$ (A.18) EC8: Part 3

The chord rotation at yield:

$\theta_y^* = 1.05\theta_y$ (A.19a) EC8: Part 3

The ultimate chord rotation:

$\theta_u^* = \theta_u$ (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

The shear capacity of beams, columns and walls 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.

$$V_R = \frac{1}{\gamma_{el}}\left[\frac{h-x}{2L_V}\min(N; 0{,}55A_c f_c) + \left(1 - 0{,}05\min\left(5; \mu_\Delta^{pl}\right)\right)\cdot\left[0{,}16\max(0{,}5; 100\rho_{tot})\left(1 - 0{,}16\min\left(5; \frac{L_V}{h}\right)\right)\sqrt{f_c}A_c + V_w\right]\right]$$ (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 rest of the 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:

$$V_{R,max} = \frac{0{,}85\left(1-0{,}06\min\left(5;\mu_\Delta^{pl}\right)\right)}{\gamma_{el}}\left(1 + 1{,}8\min\left(0{,}15; \frac{N}{A_c f_c}\right)\right)\left(1 + 0{,}25\max(1{,}75; 100\rho_{tot})\right)\cdot\left(1 - 0{,}2\min\left(2; \frac{L_V}{h}\right)\right)\sqrt{f_c}b_w z$$ (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.

Masonry Elements

All the masonry element checks (shear capacity, bending moment and drift) should be carried out for all the elements of every floor, according to Annex C 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).

Shear Force Capacity

The shear force capacity of unreinforced masonry walls controlled by flexure under an axial load N is calculated according to the following expression:

$$V_f = \frac{DN}{2H_0}(1 - 1{,}15v_d) \qquad \text{(C.1) EC8: Part 3}$$

The shear force capacity of unreinforced masonry walls controlled by shear under an axial load N is calculated according to the following expression:

$$V_f = f_{vd} D' t \qquad \text{(C.2) EC8: Part 3}$$

Bending Moment Capacity

The Bending moment capacity of unreinforced masonry wall is calculated based in the following expression:

$$V_f = {}^{2M}/_{L}$$

Where V_f is the shear force capacity calculated according to equations (C.1) of EC8: Part 3.

Drift

The capacity of unreinforced masonry walls may be expressed in terms of drift, which is calculated from the following equations:

For primary seismic walls controlled by flexure:

$$0{,}008 \cdot {}^{H_0}/_{D} \qquad \text{C.4.2.1(2) EC8: Part 3}$$

For secondary seismic walls controlled by flexure:

$$0{,}012 \cdot {}^{H_0}/_{D} \qquad \text{C.4.2.1(2) EC8: Part 3}$$

The drift for masonry walls and spandrels with reinforcement controlled by shear is taken equal to 0,004 for primary seismic walls and 0,006 for secondary ones, according to C.4.3.1(2) of EC8: Part 3.

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 $\Phi_n=1$, 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:

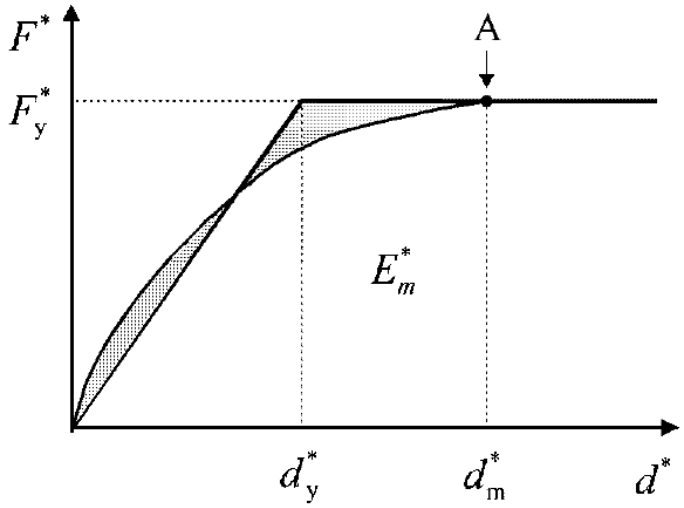


Figure B.1 of EN 1998-1 : Determination of the idealized elasto-perfectly plastic force-displacement relationship

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 H2 - 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.

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 nonexistent 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 nonexistent 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.

Safety Factors

In ASCE 41-23 the safety factors are directly incorporated in the member's strengths and deformation limits, and should not be defined.

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 10.4.2.3 of ASCE 41-23.

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.

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:

  $$\theta_{yE} = \left(\frac{M_{CyGE}}{(E_{cE}I)_{eff}}\right) l_p \qquad (7.4.1.1.1)$$

  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 369.1-2223

- 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 7.4.1.1.3of 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:

$$V_n = A_{cv}\left(\alpha_c\lambda\sqrt{f'_c} + \rho_t f_y\right)$$ (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 318-14.

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 detailed calculation of section 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:

$$V_f = \frac{A_{fv} f_{fe} (\sin\alpha + \cos\alpha) d_{fv}}{s_f} \qquad \text{(11.4a) ACI 440}$$

where

$$A_{fv} = 2 n t_f w_f \qquad \text{(11.4b) ACI 440}$$

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.

Masonry Elements

All the masonry element checks (shear capacity, bending moment capacity, compression capacity and drift) should be carried out for all the elements of every floor, according to Section 11.3.2 of ASCE 41-23 and Sections 9.3.4.1 and 9.3.5 of TMS 402, 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).

Shear Force Capacity

Unreinforced Masonry Walls

The in-plane shear force capacity of unreinforced masonry walls is calculated as the lesser value of rocking strength, bed-joint sliding strength, toe-crushing strength and diagonal tension strength, which are calculated according to the following expressions:

Rocking strength:

$$Q_{CE} = V_r = 0.9(\alpha P_D + 0.5P_W)L/h_{eff} \quad \text{(11-9) ASCE 41-23}$$

Bed-Joint sliding strength:

$$Q_{CE} = V_{bjs1} = v_{me}A_n \quad \text{(11-10) ASCE 41-23}$$

Diagonal Tension strength:

$$Q_{CL} = V_{dt} = f'_{dt}A_n\beta\sqrt{1 + \frac{f_\alpha}{f'_{dt}}} \quad \text{(11-13) ASCE 41-23}$$

Unreinforced Wall Spandrels

The in-plane shear force capacity of unreinforced masonry spandrels is calculated as the lesser value of the flexural strength and the shear strength.

The in-plane flexural strength of rectangular unreinforced wall spandrels with and without timber, concrete, or steel lintels is determined as the minimum value of the following equations:

$$V_{f1} = \left(f_t + p_{sp}\right)\frac{h_{sp}^2 b_{sp}}{3l_{sp}} \quad \text{(11-15) ASCE 41-23}$$

and

$$V_{fl,r} = \frac{p_{sp}h_{sp}^2 b_{sp}}{l_{sp}}\left(1 - \frac{p_{sp}}{0.85f_{hm}}\right) \quad \text{(11-17) ASCE 41-23}$$

The shear strength of unreinforced masonry spandrel is computed as the lesser value calculated from the following expressions:

$$V_{s1} = \frac{2}{3}\left(c_{bj} + \mu_f p_{sp}\right)h_{sp}b_{sp} \quad \text{(11-18) ASCE 41-23}$$

and

$$V_{s2} = f'_{dt}\beta_{sp}h_{sp}b_{sp}\sqrt{1 + \frac{p_{sp}}{f'_{dt}}} \quad \text{(11-19) ASCE 41-23}$$

Where equation (11-18) is the peak shear strength associated with the formation of cracks through head and bed joints over almost the entire height of the spandrel and equation (11-19) is used when fracture of the masonry units will occur.

Masonry Walls/Spandrels with reinforcement

The shear force capacity of masonry walls and spandrels with reinforcement is computed using the following equations:

$$V_n = (V_{nm} + V_{ns})\gamma_g \quad \text{(Equation 9-21) TMS 402}$$

Where V_n shall not exceed the following:

- If $M_u/(V_u d_v) \leq 0.25$

$$V_n \leq \left(6A_{nv}\sqrt{f'_m}\right)\gamma_g \quad \text{(Equation 9-22) TMS 402}$$

- If $M_u/(V_u d_v) \geq 1.0$

$$V_n \leq \left(4A_{nv}\sqrt{f'_m}\right)\gamma_g \quad \text{(Equation 9-23) TMS 402}$$

Users are advised to refer to the Code for the definition of the parameters and further details on the expressions.

Compressive Capacity

The nominal axial compressive strength shall not exceed Equation 9-19 or Equations 9-20, as appropriate:

- For members having an h/r ratio not greater than 99

$$P_n = 0.80\left[0.80f'_m(A_n - A_{st}) + f_y A_{st}\right]\left[1 - \left(\frac{h}{140r}\right)^2\right] \quad \text{(Equation 9-19) TMS 402}$$

- For members having an h/r ratio greater than 99

$$P_n = 0.80\left[0.80f'_m(A_n - A_{st}) + f_y A_{st}\right]\left(\frac{70r}{h}\right)^2 \quad \text{(Equation 9-20) TMS 402}$$

Users are advised to refer to the Code for the definition of the parameters and further details on the expressions.

Bending Moment Capacity

The Bending moment capacity of unreinforced masonry walls and spandrels is based on the lower-bound toe crushing strength calculated in accordance with the following expression:

$$Q_{CL} = V_{tc} = (\alpha P_D + 0.5P_W)\left(\frac{L}{h_{eff}}\right)\left(1 - \frac{f_\alpha}{0.7f'_m}\right) \quad \text{(11-12) ASCE 41-23}$$

The Bending moment capacity of masonry walls and spandrels with reinforcement is calculated based in the following expression of section 9.3.5.2 of TMS 402:

$$M_n = \left(P_u/\varphi + A_s f_y\right)\left(\frac{t_{sp} - \alpha}{2}\right) + A_s f_y\left(d - \frac{t_{sp}}{2}\right)$$

where

$$\alpha = \frac{A_s f_y + P_u/\varphi}{0.80f'_m b}$$

The above equations are valid for both centered and noncentered flexural reinforcement. In the case of centered flexural reinforcement, $d=t_{sp}/2$, the bending moment capacity is obtained as:

$$M_n = \left(P_u/\varphi + A_s f_y\right)\left(d - \frac{\alpha}{2}\right)$$

Users are advised to refer to the Code for the definition of the parameters and further details on the expressions.

Drift

The capacity of unreinforced masonry walls and spandrels may be expressed in terms of drift, which is calculated from the Table 11-4 of ASCE 41-23.

Table 11-4. Nonlinear Procedures: Simplified Force–Deflection Relations for URM In-Plane Walls, Wall Piers[a], and Spandrels with $v_{rL} \geq 30$ lb/in.^{2}

Limiting Behavior Mode	Modeling Parameters: Residual Strength Ratio	*d* (%)	*e* (%)	*f* (%)	Acceptance Criteria Performance Level	IO (%)	LS (%)	CP (%)
Wall and Wall Pier Rocking[b]	$V_{tc,r}/V_r$	$100\Delta_{tc,r}/h_{eff}$	$100\Delta_{tc,r}/h_{eff}$	$100(\Delta_{tc,r}+\Delta_y)/h_{eff}$	Simplified	0.1	$0.4h_{eff}/L$ but not greater than 1.50%[c]	$0.6h_{eff}/L$ but not greater than 2.25%[d]
					Comprehensive[d]	0.1	$0.6h_{eff}/L$ but not greater than 2.25%	$100\Delta_{tc,r}/h_{eff}$ but not greater than 2.5%
Wall and Wall Pier Bed-Joint Sliding	V_{bjs2}/V_{bjs1}[e]	0.4	1.0	$1.0+100\Delta_y/h'$		0.1	0.75	1.0
Spandrels with Prismatic Lintels	$\text{Min}(V_{flr}, V_{s,r})/\text{Min}(V_{fl}, V_s)$	0.3	3.0	3.1		0.1	2.25	3.0
Spandrels with Shallow Arch Lintels	$\text{Min}(V_{flr}, V_{s,r})/\text{Min}(V_{fl}, V_s)$	0.3	0.75	0.85		0.1	0.56	0.75

[a] Interpolation shall be used between table values.
[b] All rocking walls and wall piers shall comprise a minimum thickness of 6 in. and, for solid brick masonry, a minimum of two wythes. Multi-wythe solid brick walls and wall piers shall be connected with bonded solid headers. $V_{tc,r}$ is the seismic shear force associated with the onset of toe crushing after rocking initiates. The axial compressive stress on the toe caused by gravity loads, f_a, shall be based on the strain of the rocking pier and an equivalent compression zone of the effective net section of the rocking pier that is in bearing immediately before the onset of crushing, consistent with Section 9.3.2(g) of TMS 402, or some other analytical approach based on engineering mechanics and the stress–strain response of the materials that comprise the pier and its interface with supporting components. $\Delta_{tc,r}$ is the lateral displacement associated with the onset of toe crushing $V_{tc,r}$.
[c] In no case shall the LS acceptance criteria exceed 0.75 times the CP acceptance criteria.
[d] Comprehensive acceptance criteria may be used if an analysis based on moment-curvature is used to explicitly calculate $\Delta_{tc,r}$.
[e] V_{bjs1} and V_{bjs2} shall be calculated in accordance with Section 11.3.2.2.2.
[f] Point *f* on the force–deformation curve where vertical-load-carrying capacity is diminished shall be based on the drift associated with no greater than one-half the width of the masonry units or units at the spring line of masonry arches, assuming that bed-joint sliding occurs entirely within one bed joint in a wall or pier.

Table 11-4 of ASCE 41-23

The drift for masonry walls and spandrels with reinforcement is taken from Table 11-7 of ASCE 41-23.

Table 11-7. Modeling Parameters and Acceptance Criteria for Nonlinear Procedures—Reinforced Masonry In-Plane Walls

f_{ae}/f_{me}	h_{eff}/L	$\rho_g f_{ye}/f_{me}$	Residual Strength Ratio *c*	*d* (%)	*e* (%)	Acceptable Drift Ratio (%)[a] Performance Level: IO (%)	LS (%)	CP (%)
Wall Components Controlled by Flexure								
0.00	≥2.0	0.01	0.5	2.6	5.3	1.0	3.9	5.3
		0.05	0.6	1.1	2.2	0.4	1.6	2.2
		0.20	0.7	0.5	1.0	0.2	0.7	1.0
	1.0	0.01	0.5	2.1	4.1	0.8	3.1	4.1
		0.05	0.6	0.8	1.6	0.3	1.2	1.6
		0.20	0.7	0.3	0.6	0.1	0.5	0.6
	≤0.5	0.01	0.5	1.6	3.3	0.6	2.5	3.3
		0.05	0.6	0.6	1.3	0.2	0.9	1.3
		0.20	0.7	0.2	0.4	0.1	0.3	0.4
0.038	≥2.0	0.01	0.4	1.0	2.0	0.4	1.5	2.0
		0.05	0.5	0.7	1.4	0.3	1.0	1.4
		0.20	0.6	0.4	0.9	0.2	0.7	0.9
	1.0	0.01	0.4	0.8	1.5	0.3	1.1	1.5
		0.05	0.5	0.5	1.0	0.2	0.7	1.0
		0.20	0.6	0.3	0.6	0.1	0.4	0.6
	≤0.5	0.01	0.4	0.6	1.2	0.2	0.9	1.2
		0.05	0.5	0.4	0.7	0.1	0.5	0.7
		0.20	0.6	0.2	0.4	0.1	0.3	0.4
0.075	≥2.0	0.01	0.3	0.6	1.2	0.2	0.9	1.2
		0.05	0.4	0.5	1.0	0.2	0.8	1.0
		0.20	0.5	0.4	0.8	0.1	0.6	0.8
	1.0	0.01	0.3	0.4	0.9	0.2	0.7	0.9
		0.05	0.4	0.4	0.7	0.1	0.5	0.7
		0.20	0.5	0.2	0.5	0.1	0.4	0.5
	≤0.5	0.01	0.3	0.3	0.7	0.1	0.5	0.7
		0.05	0.4	0.3	0.5	0.1	0.4	0.5
		0.20	0.5	0.2	0.3	0.1	0.2	0.3
Wall Components Controlled by Shear								
All cases[b]	All cases[b]	All cases[b]	0.4	0.75	2.0	0.4	0.75	1.5

[a] Interpolation shall be used between table values.
[b] For wall components governed by shear, the axial load on the member must be less than or equal to $0.15A_g f'_m$; otherwise, the component shall be treated as force controlled.

Table 11-7 of ASCE 41-23

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 muli 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

$$C_1 = 1 + \frac{\mu_{strength}^{-1}}{\alpha T_e^2} \text{ for } 0.2s \le T < 1s\,, \qquad \text{(7-30) ASCE 41-23}$$

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;

$$C_2 = 1 + \frac{1}{800}\left(\frac{\mu_{strength}-1}{T_e}\right)^2 \quad \text{(7-31) ASCE 41-23}$$

Where the strength ratio $\mu_{strength}$ is calculated according to the following equation:

$$\mu_{strength} = \frac{S_\alpha}{V_y/W} C_m \quad \text{(7-32) ASCE 41-23}$$

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 non-linear 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).

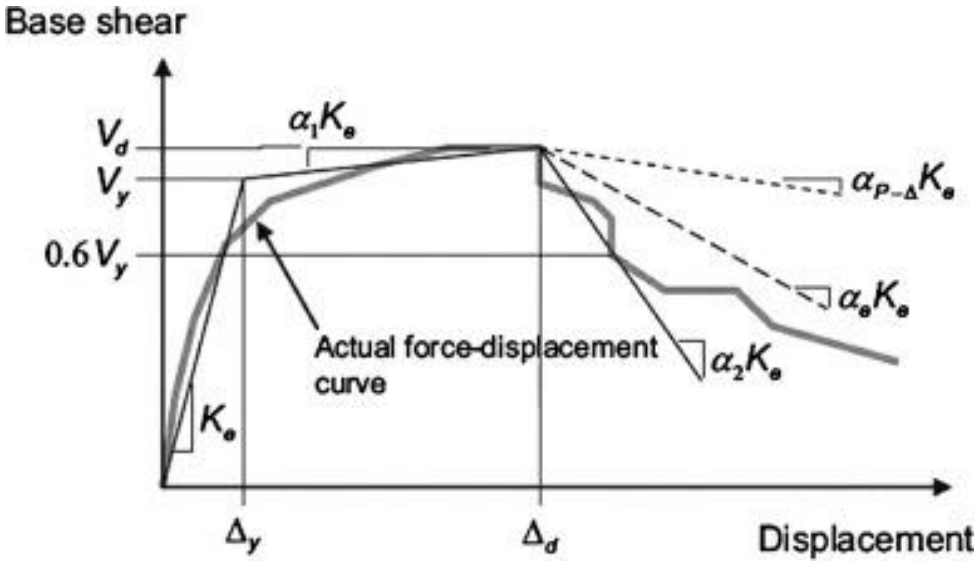


Figure 7-3 of ASCE 41-23 Idealized Force -_Displacement Curve

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 H3 - NTC-18

In this appendix the parameters used for the structures assessment according to the Italian National Seismic Code – NTC-18 are presented.

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

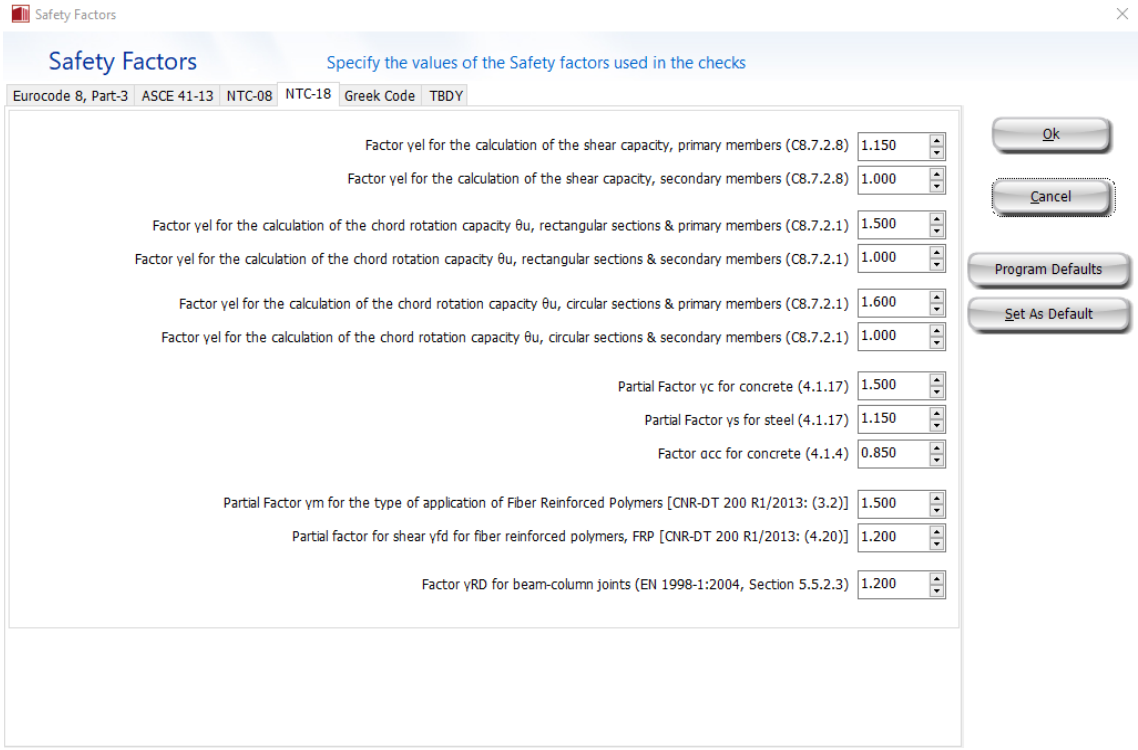
In the following table of section C8A.1.B.3 of the commentary of NTC-18 a summary and recommendations for the confidence factors and the analysis methods are provided for each knowledge level.

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 insitu	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 insitu	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.

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:

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

(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

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 C8.7.2.3.4 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 non-linearities 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 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 SeismoStruct the following equations 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.

$$\theta_u = \left(\theta_y + (\varphi_u - \varphi_y)L_{pl}\left(1 - 0.5\,L_{pl}/L_s\right) + \alpha_{sl}\Delta\theta_{u,slip}\right)/\gamma_{el}$$

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:

$M_y^* = 0.9M_y$ (8.7.4.2) commentary of NTC-18

The chord rotation at yield:

$\theta_y^* = 0.9\theta_y$ (8.7.4.3) commentary of NTC-18

The ultimate chord rotation:

$\theta_u^* = \theta_u$ (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 and its plastic part for the members of rectangular sections with corners rounded is calculated through the expressions 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:

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

(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.

$$V_{Rd} = \min(V_{Rsd}, V_{Rcd}) \qquad (4.1.29)\ \text{NTC-18}$$

where V_{Rsd} is the shear strength that corresponds to the contribution of the shear reinforcement and is calculated according to the equation below:

$$V_{Rsd} = 0{,}9 \cdot d \cdot \frac{A_{sw}}{s} \cdot f_{yd} \cdot (\text{ctg}\alpha + \text{ctg}\theta) \cdot \sin\alpha \qquad (4.1.27)\ \text{NTC-18}$$

and V_{Rcd} is the shear strength that corresponds to the confined concrete core and is calculated according to the following equation:

$$V_{Rcd} = 0{,}9 \cdot d \cdot b_w \cdot \alpha_c \cdot v \cdot f_{cd} \cdot (\text{ctg}\alpha + \text{ctg}\theta)/(1 + \text{ctg}^2\theta) \qquad (4.1.28)\ \text{NTC-18}$$

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.

$$V_R = \frac{1}{\gamma_{el}}\left[\frac{h-x}{2L_V}\min(N; 0{,}55A_c f_c) + \left(1 - 0{,}05\min\left(5; \mu_{\Delta,pl}\right)\right) \cdot \left[0{,}16\max(0{,}5; 100\rho_{tot})\left(1 - 0{,}16\min\left(5; \frac{L_V}{h}\right)\right)\sqrt{f_c}A_c + V_w\right]\right] \qquad (8.7.2.8)\ \text{commentary of NTC-18}$$

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:

$$V_R^* = 0.9V_R \qquad (8.7.4.1)\ \text{commentary of NTC-18}$$

FRP wrapping

The cyclic resistance V_R, may be calculated from the section 4.1 of NTC-18 adding to V_w the contribution of the FRP jacket to shear resistance. The contribution of the fully wrapped FRP jacket to V_w is computed according to 4.19 equation of CNR-DT 200 R1/2013 in the following form:

$$V_{Rd,f} = \frac{1}{\gamma_{Rd}} \cdot 0.9 \cdot d \cdot f_{fed} \cdot 2 \cdot t_f \cdot (\cot\theta + \cot\beta) \cdot \sin\beta$$

Masonry Elements

All the masonry element checks (shear capacity, bending moment and drift) should be carried out for all the elements of every floor, according to sections 7.8.2.2 and 7.8.3.2 of NTC-18 and section C8.7.1.3.1.1 of Commentary of NTC-18.

Shear Force Capacity

The shear force capacity of unreinforced masonry walls controlled by flexure under an axial load N is calculated according to the following expression:

Unreinforced Masonry Walls

Mohr-Coulomb equation, as described in section 7.8.2.2.2 of NTC-18 and section 6.2 of EC6, for masonry with blocks

$$V_t = l' t f_{vd} \qquad (7.8.3)\ \text{NTC-18}$$

Turnšek-Čačovič equation, as described in section C8.7.1.3.1.1 of commentary of NTC-18, for irregular masonry (irregular stones) and for existing regular masonry (bricks, regular stones):

$$V_t = l \cdot t \frac{1.5\tau_{0d}}{b} \sqrt{1 + \frac{\sigma_0}{1.5\tau_{0d}}} = l \cdot t \frac{f_{td}}{b} \sqrt{1 + \frac{\sigma_0}{f_{td}}}$$

Mann-Müller equation, as described in section C8.7.1.3.1.1 of commentary of NTC-18)for existing regular masonry (bricks, regular stones):

$$V_t = \frac{lt}{b}\left(\tilde{f}_{v0d} + \tilde{\mu}\sigma_0\right) = \frac{lt}{b}\left(\frac{f_{v0d}}{1+\mu\varphi} + \frac{\mu}{1+\mu\varphi}\sigma_0\right) \leq V_{t,lim} \qquad (8.7.1.24)\ \text{commentary of NTC-18}$$

In some cases, it is necessary to take the lower value between the different expressions above. For example, in the case of Unreinforced Masonry Wall with bricks the Commentary of the NTC-18 requires to take the lower value between the Turnšek-Čačovič and Mann-Müller criterion.

Reinforced Masonry Walls

The shear capacity of reinforced masonry walls is calulcated, according to section 7.8.3.2.2 of NTC-18, as the sum of the contribution of masonry ($V_{t,M}$) and of the reinforcement ($V_{t,S}$), as shown in the following expression:

$$V_t = V_{t,M} + V_{t,S} \qquad (7.8.7)\ \text{NTC-18}$$

where:

$$V_{t,M} = d \cdot t \cdot f_{vd} \qquad (7.8.8)\ \text{NTC-18}$$

and

$$V_{t,S} = \left(0{,}6 \cdot d \cdot A_{sw} \cdot f_{yd}\right)/s \qquad (7.8.9)\ \text{NTC-18}$$

Further, the shear capacity should not exceed the following value:

$$V_{t,c} = 0{,}3 \cdot f_d \cdot t \cdot d \qquad (7.8.10)\ \text{NTC-18}$$

Users are advised to refer to the Code for the definition of the parameters and further details on the expressions.

Bending Moment Capacity

The Bending moment capacity of unreinforced masonry walls calculated according to the following expression:

$$M_u = \left(l^2 \cdot t \cdot \frac{\sigma_0}{2}\right)\left(1 - \frac{\sigma_0}{0{,}85 f_d}\right) \qquad (7.8.2)\ \text{NTC-18}$$

The Bending moment capacity of unreinforced masonry spandrels calculated according to the following expression:

$$M_u = H_p \cdot \frac{h}{2} \cdot \left[1 - \frac{H_p}{(0{,}85 \cdot f_{bd} \cdot h \cdot t)}\right] \qquad (7.8.5)\ \text{NTC-18}$$

The Bending moment capacity of masonry walls and spandrels with reinforcement is calculated based in the following expression of section 9.3.5.2 of TMS 402:

$$M_n = \left(P_u/\varphi + A_s f_y\right)\left(\frac{t_{sp} - \alpha}{2}\right) + A_s f_y \left(d - \frac{t_{sp}}{2}\right)$$

where

$$\alpha = \frac{A_s f_y + P_u/\varphi}{0.80 f'_m b}$$

The above equations are valid for both centered and noncentered flexural reinforcement. In the case of centered flexural reinforcement, $d=t_{sp}/2$, the bending moment capacity is obtained as:

$$M_n = \left(P_u/\varphi + A_s f_y\right)\left(d - \frac{\alpha}{2}\right)$$

Users are advised to refer to the Code for the definition of the parameters and further details on the expressions.

Drift

The capacity of masonry walls may be expressed in terms of drift. The drift for masonry walls controlled by flexure is taken equal to 0,01 for walls without reinforcement and 0,016 for walls with reinforcement, according to sections 7.8.2.2.1 and 7.8.3.2.1 of NTC-18. Further, the drift for masonry walls controlled by shear is taken equal to 0,005 for walls without reinforcement and 0,008 for for walls with reinforcement, according to section 7.8.2.2.2 and 7.8.3.2.2 of NTC-18.

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 $\Phi_n=1$, 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:

$$\Gamma = \frac{\varphi^T M \tau}{\varphi^T M \varphi} \qquad \text{(C7.3.5) commentary of NTC-18}$$

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:

$$F^* = F_b/\Gamma$$

$$d^* = d_c/\Gamma \qquad \text{(C7.3.4) commentary of NTC-18}$$

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:

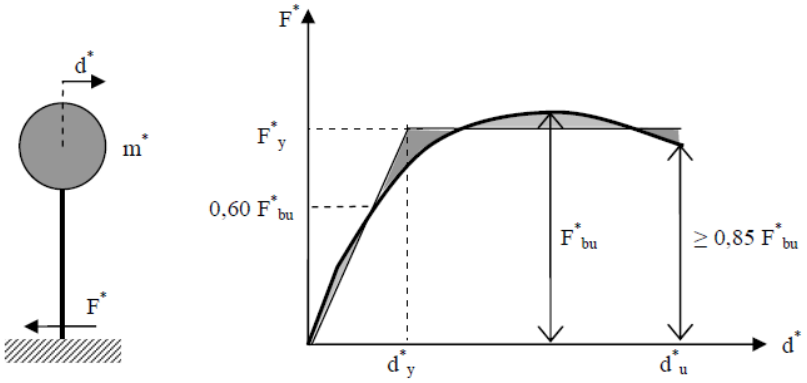


Figure C7.3.1 of commentary of NTC-18: Determination of the idealized elasto-perfectly plastic force-displacement relationship of NTC-18

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:

$$d^*_{max} = d^*_{e,max} = S_{De}(T^*) \qquad \text{(C7.3.7) commentary of NTC-18}$$

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 $q^* \leq 1$ then $d^*_{max} = d^*_{e,max}$.

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 H4 - KANEPE

In this appendix the parameters used for the structures assessment according to the Greek Seismic Interventions Code KANEPE are presented.

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), but 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

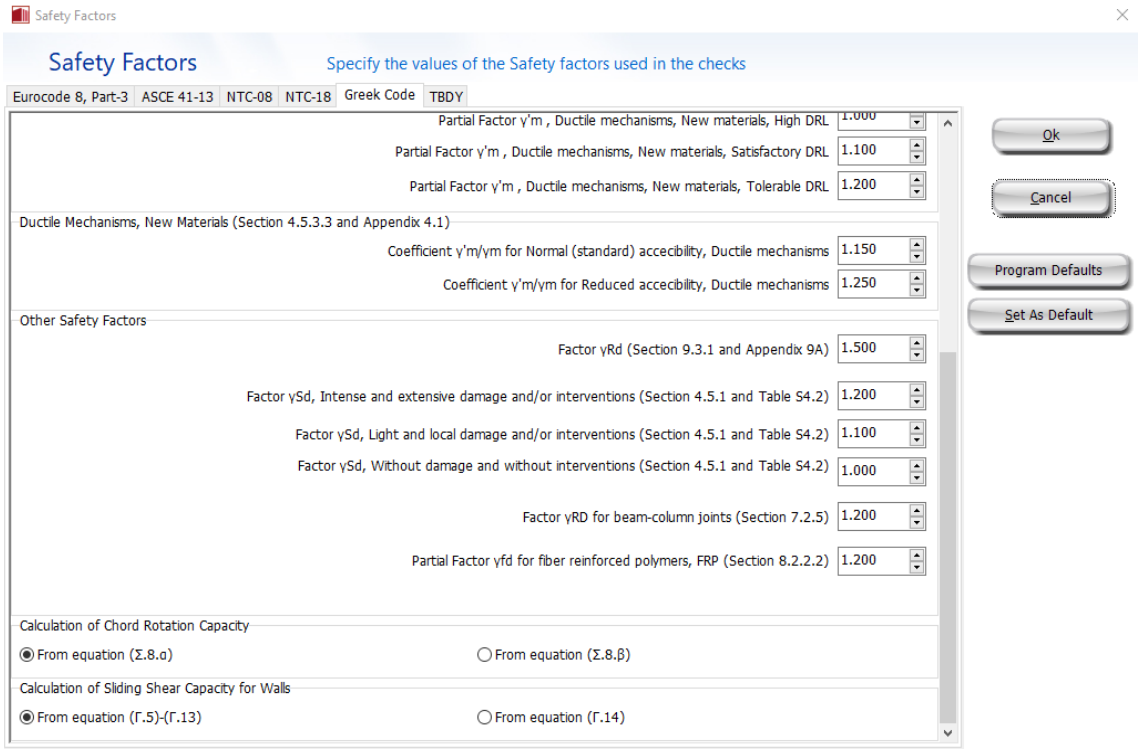
The high reliability level corresponds to a state of knowledge where the overall structural geometry and member sizes are known from a comprehensive survey or from the complete set of outline construction drawings used for both the original construction and subsequent modifications, as well as a sufficient 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.

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:

$$\theta_y = (1/r)_y \frac{L_s + \alpha_V z}{3} + 0{,}0014\left(1 + 1{,}5\frac{h}{L_s}\right) + \frac{(1/r)_y d_b f_y}{8\sqrt{f_c}} \qquad \text{(S.2a) KANEPE}$$

- For walls:

$$\theta_y = (1/r)_y \frac{L_s + \alpha_V z}{3} + 0{,}0013 + \frac{(1/r)_y d_b f_y}{8\sqrt{f_c}} \qquad \text{(S.2b) KANEPE}$$

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:

$$\theta_{um} = 0{,}016 \cdot (0{,}3^{\nu}) \left[\frac{\max(0{,}01;\omega')}{\max(0{,}01;\omega-\omega')} f_c\right]^{0{,}225} \cdot (\alpha_s)^{0{,}35} 25^{\left(\alpha\rho_s \frac{f_{yw}}{f_c}\right)} (1{,}25^{100\rho_d}) \qquad \text{(S.11a) KANEPE}$$

- 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:

$$\theta_{um} = 0{,}01 \cdot (0{,}3^{\nu}) \left[\frac{\max(0{,}01;\omega')}{\max(0{,}01;\omega-\omega')} f_c\right]^{0{,}225} \cdot (\alpha_s)^{0{,}35} 25^{\left(\alpha\rho_{sx} \frac{f_{yw}}{f_c}\right)} (1{,}25^{100\rho_d}) \qquad \text{(S.11a) KANEPE}$$

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:

$$\theta_{um}^{pl} = \theta_u - \theta_y = 0{,}0145 \cdot (0{,}25^{\nu}) \left[\frac{\max(0{,}01;\omega')}{\max(0{,}01;\omega-\omega')}\right]^{0{,}3} (f_c)^{0{,}2} (\alpha_s)^{0{,}35} 25^{\left(\alpha\rho_s \frac{f_{yw}}{f_c}\right)} (1{,}275^{100\rho_d})$$

(S.11b) KANEPE

- For 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.

$$\varphi_y = (1/r)_y = \frac{f_y}{E_s(1-\xi_y)d} \quad \text{(A.1) KANEPE}$$

If the section yields due to the deformation non-linearities 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.

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). Finally, 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.

In the case of circular column sections, the equations above cannot be employed for the calculation of the elements' chord rotation capacity. In SeismoStruct the following equations 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.

$$\theta_u = \left(\theta_y + (\varphi_u - \varphi_y)L_{pl}\left(1 - 0.5\,L_{pl}/L_s\right) + \alpha_{sl}\Delta\theta_{u,slip}\right)/\gamma_{el}$$

Where γ_{el} is equal to 2.0 for 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:

$$\Delta\theta_{u,slip} = 10 d_{bl}\left(\varphi_u + \varphi_y\right)/2$$

$$L_{pl} = 0.6D\left[1 + \frac{1}{6}\min\left(9; \frac{L_s}{D}\right)\right]$$

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 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.

$$V_R = \frac{h-x}{2L_s}\min(N; 0{,}55A_c f_c) + \left(1 - 0{,}05\min\left(5; \mu_\theta^{pl}\right)\right)\left[0{,}16\max(0{,}5; 100\rho_{tot})\,(1 - 0{,}16\min(5; \alpha_s))\sqrt{f_c}A_c + V_w\right]$$

(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:

$$V_{R,max} = 0{,}85\left(1 - 0{,}06\min\left(5; \mu_\theta^{pl}\right)\right)\left(1 + 1{,}8\min\left(0{,}15; \frac{N}{A_c f_c}\right)\right)\left(1 + 0{,}25\max(1{,}75; 100\rho_{tot})\right)\cdot\left(1 - 0{,}2\min(2; \alpha_s)\right)\sqrt{f_c}b_w z$$

(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:

$$V_{R,SLS} = V_i + V_f + V_d \quad \text{(C.6) KANEPE}$$

with

$$V_i = \sum A_{si} f_{yi} \cos\varphi \quad \text{(C.7) KANEPE}$$

$$V_f = \min\left(\mu\left[\left(\sum A_{sv} f_{yv} + N\right)\xi + M_y/z\right]; 0.3 f_c A_{compr}\right) \quad \text{(C.8) KANEPE}$$

where ξ is calculated according to equations C.10-C13 of Appendix 7C of KANEPE, and

$$V_d = 1.6 \sum A_{sv}\sqrt{f_c f_{yv}} \le \sum A_{sv} f_{yv}/\sqrt{3} \quad \text{(C.9) KANEPE}$$

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:

$$V_{R,SLS} = \left(1 - 0{,}025\mu_\theta^{pl}\right)\min\begin{pmatrix} 0{,}5\sum\left[A_s f_y (0{,}6\sin\varphi + \cos\varphi)\right] + 0{,}6N + 1{,}1\sum\left[A_s\sqrt{f_c f_y}\sin\varphi\right]; \\ 0{,}2\min\left(0{,}55; 0{,}55\left(\frac{30}{f_c}\right)^{\frac{1}{3}}\right) f_c A_c \end{pmatrix} \quad \text{(C.14) KANEPE}$$

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 $\alpha_s \ge 1.0$. For walls with low shear ratio $\alpha_s \le 1.2$, the shear capacity should be calculated from the following equation of Appendix 7C of KANEPE:

$$V_{R,squat} = V_{si} + V_c \quad \text{(C.15) KANEPE}$$

with

$$V_s = \min\begin{Bmatrix} \rho_h b_w \min((d - x)/\tan\theta_{cr}, L_s) f_{yh} \\ \left(\rho_v b_w \min(L_s \tan\theta_{cr}, d - x) f_{yv} + A_s f_y\right)/\tan\theta_{cr} \end{Bmatrix} \quad \text{(C.16) KANEPE}$$

and

$$V_c = (1 + 150\rho)(1 - 0.725\alpha_s)\left(\frac{2}{3} A_c f_{ct}\sqrt{1 + \frac{N}{A_c f_{ct}}}\right) \quad \text{(C.17) KANEPE}$$

Users are advised to refer to the relevant publication for the definition of the parameters and further details on the expressions.

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:

$$V_R^* = 0.9 V_R$$

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.

Masonry Elements

All the masonry element checks are carried out according to Eurocodes.

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:

$$\delta_t = C_0 \cdot C_1 \cdot C_2 \cdot C_3 \cdot (T_e^2/4\pi^2)S_{e(T)} \qquad \text{(5.6) KANEPE}$$

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 non-linear 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).

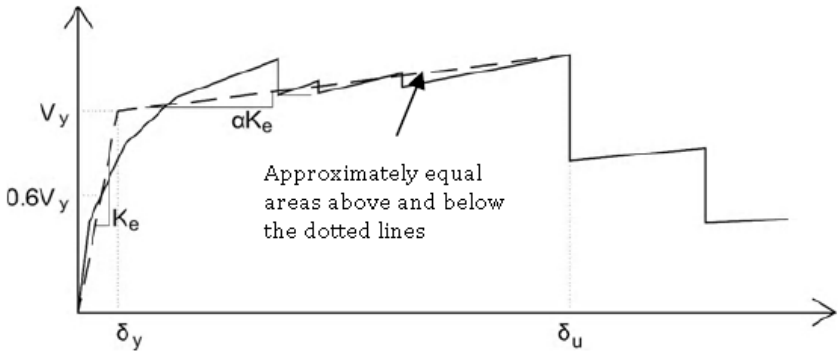


Figure 5.2 of KANEPE Idealisation of a (indicative) capacity curve with a bilinear curve

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) non-linear 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:

$$T_e = T\sqrt{\frac{K_0}{K_e}} \qquad \text{(5.5) KANEPE}$$

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 H5 - TBDY

In this appendix the parameters used for the structures assessment according to the Turkish Earthquake Building Regulations-TBDY are presented.

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), but 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 nonexistent 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.

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.

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:

$$\theta_y = \frac{\varphi_y L_s}{3} + 0{,}0015\eta\left(1 + 1{,}5\frac{h}{L_s}\right) + \frac{\varphi_y d_b f_{ye}}{8\sqrt{f_{ce}}} \qquad \text{(5.3) TBDY}$$

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.

$$\theta_p^{(HK)} = 0 \qquad \text{(5.8b) TBDY}$$

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:

$$\theta_p^{(CG)} = 0{,}75\theta_p^{(GO)} \qquad \text{(5.7b) TBDY}$$

The value $\theta_p{}^{(GO)}$ is calculated from the following equation:

$$\theta_p^{(GO)} = \frac{2}{3}\left[(\varphi_u - \varphi_y)L_p\left(1 - 0{,}5\frac{L_p}{L_s}\right) + 4{,}5\varphi_u d_b\right] \qquad \text{(5.6) TBDY}$$

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.

$$\varphi_y = (1/r)_y = \frac{f_y}{E_s(1-\xi_y)d} \qquad \text{(A.1) KANEPE}$$

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:

$$\varphi_y = (1/r)_y = \frac{\varepsilon_c}{\xi_y d} \approx \frac{1.8 f_c}{E_c \xi_y d} \quad \text{(A.2) KANEPE}$$

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 SeismoStruct 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.

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.

$$V_r = V_c + V_w + V_f \leq V_{max} \quad \text{(15B.1) TBDY}$$

where V_c is calculated according to equations 8.1 and 8.4 of TS500:

$$V_{cr} = 0.65 f_{ctd} b_w d \left(1 + \gamma \frac{N_d}{A_c}\right) \quad \text{(8.1) TS500}$$

$$V_c = 0.8 V_{cr} \quad \text{(8.4) TS500}$$

The contribution of the transverse reinforcement to the shear capacity is calculated according to equation 8.5 of TS00:

$$V_w = \frac{A_{sw}}{s} f_{ywd} d \qquad (8.5)\ TS500$$

The contribution of the FRP jacket to the shear resistance is computed through the following expression:

$$V_f = \frac{2\, n_f\, t_f\, w_f\, E_f\, \varepsilon_f\, d}{s_f} \qquad (15B.2)\ TBDY$$

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:

$$V_{max} \leq 0.22\, f_{cd}\, b_w d \qquad (8.7)\ TS500$$

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.

Masonry Elements

All the masonry element checks are carried out according to ASCE 41-23.

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.

$$d_{1,max}^{(X)} = S_{di}(T_1) \qquad (5B.12)\ TBDY$$

where

$$S_{di}(T_1) = C_R S_{de}(T_1) \qquad (5B.13)\ TBDY$$

$S_{de}(T_1)$ 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:

$$C_R = \frac{\mu(R_y, T_1)}{R_y} \qquad (5B.14)\ TBDY$$

where R_y is the yield reduction factor calculated from the following equation:

$$R_y = \frac{f_e}{f_y} = \frac{S_{ae}(T_1)}{\alpha_{y1}} \qquad (5B.15)\ TBDY$$

The equation (5B.14) for the modification factor C_R takes the following form by using the equations (5B.16) of TBDY:

$C_R = 1$ for $T_1 > T_B$ (5B.17a) TBDY

$C_R = \frac{1+(R_y-1)\frac{T_B}{T_1}}{R_y} \geq 1$ for $T_1 \leq T_B$ (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.

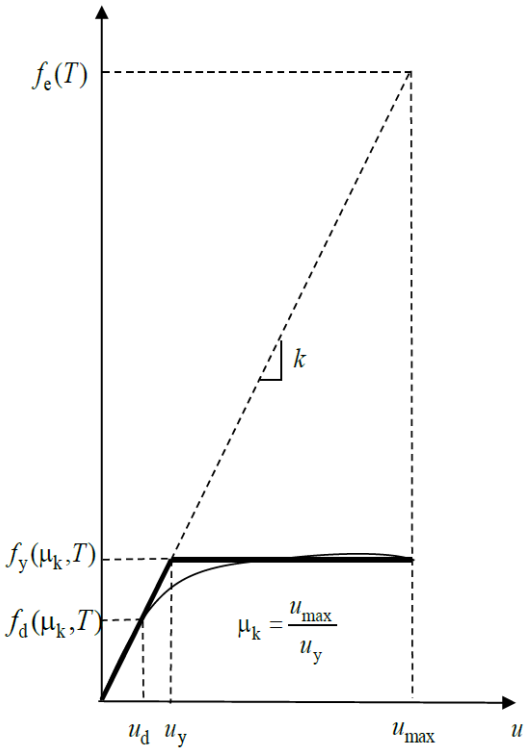


Figure 4A.1 of TBDY Determination of the idealized elasto-perfectly plastic force-displacement relationship