

SEISMOSTRUCT



Nonlinear analysis
and assessment of structures



Copyright © 2023 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 technical information sheet may be reproduced or distributed in any form or by any means, without the prior explicit written authorization 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 technical information sheet 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.

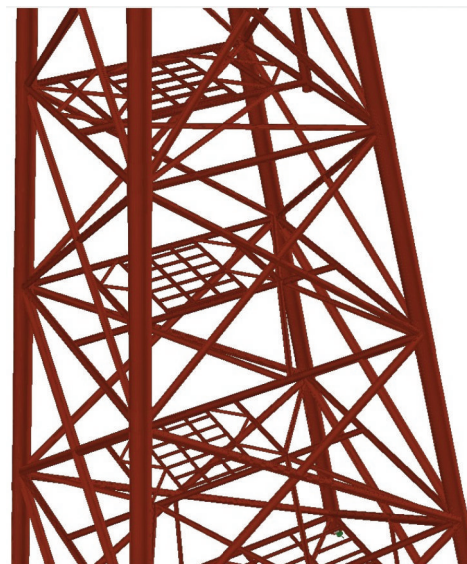
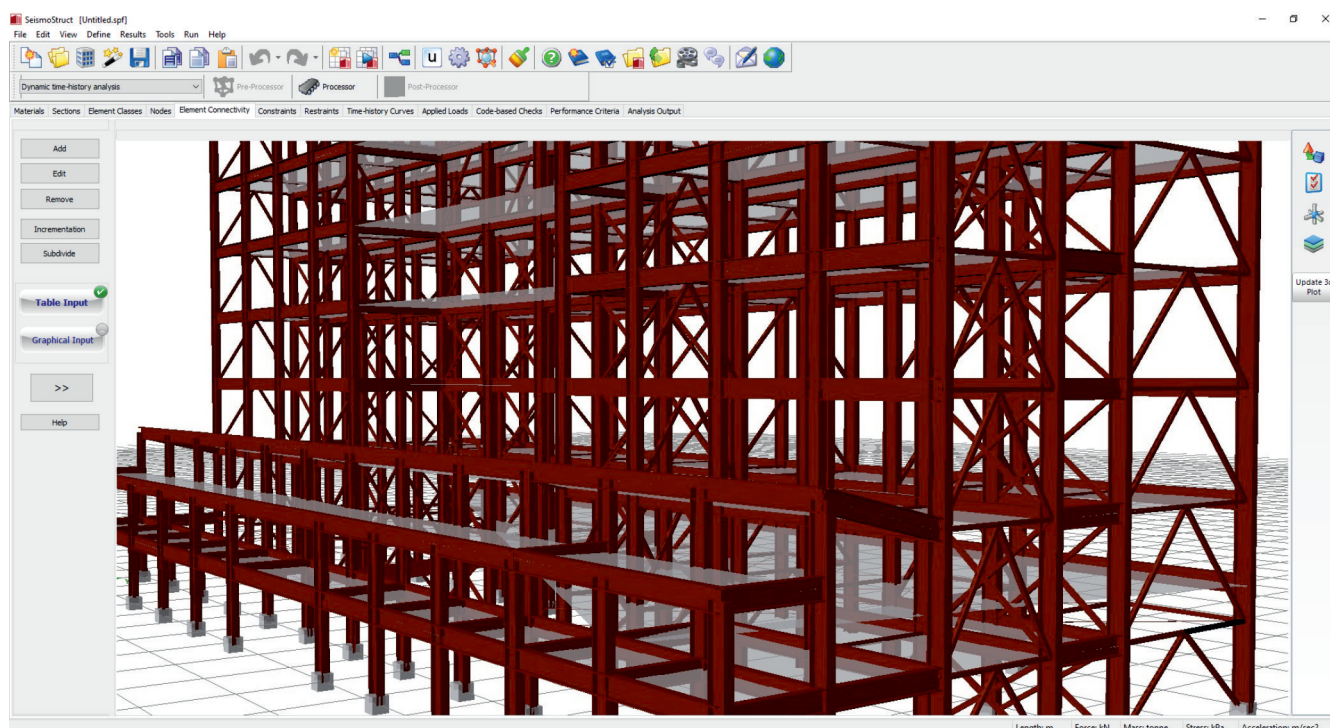
table of contents

Table of Contents	2
Summary	3
SeismoStruct Overview	5
System Requirements	7
Main SeismoStruct Features	8
Why use SeismoStruct	17
Technical Features	19
Geometric Nonlinearity	19
Material Inelasticity	19
Analysis Types	20
Material Models	21
Element Types	22
Company Overview	26
SeismoBuild	26
SeismoSignal	29
SeismoSignal 3D	29
SeismoSelect	30
SeismoMatch	31
SeismoArtif	31
SeismoSpect	32
FRP designer	32
SeismoRack	33
Awards	33
Documentation Available	34
References	35

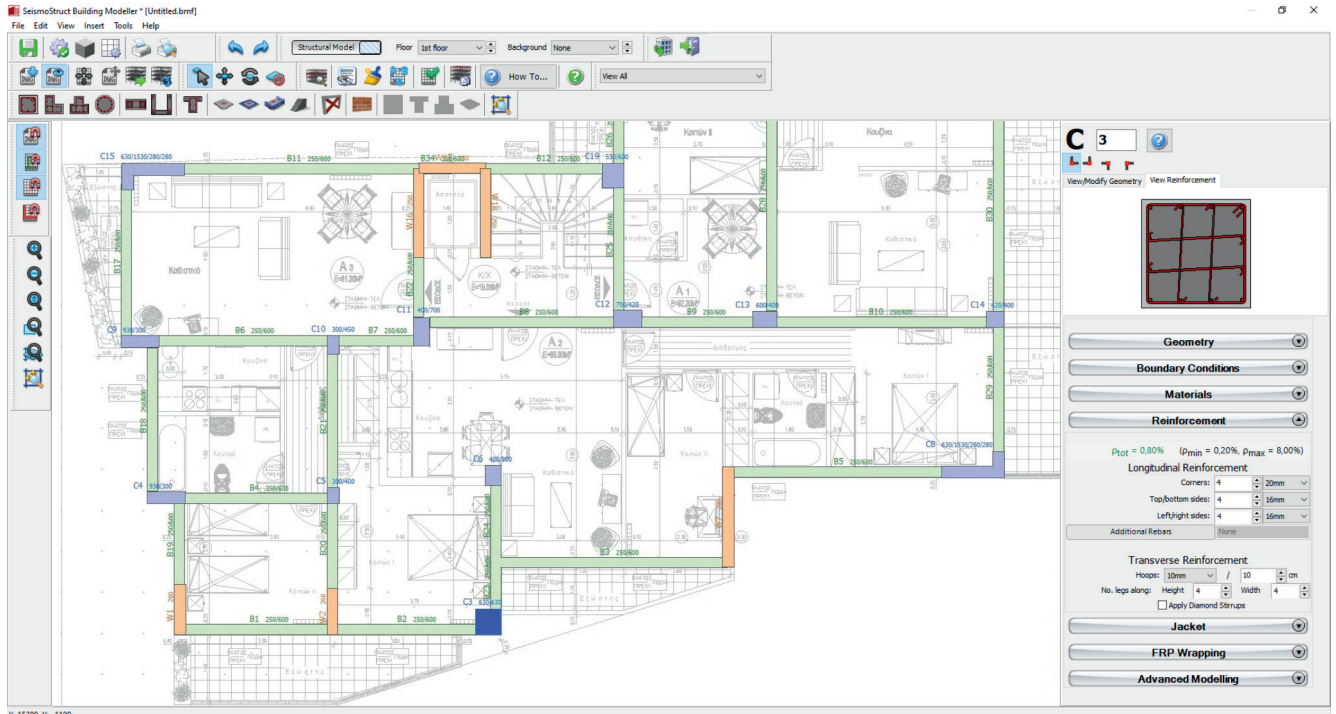
summary

SeismoStruct is an award-winning Finite Element package capable of predicting the large displacement behaviour of space frames under static or dynamic loading, taking into account both geometric nonlinearities and material inelasticity. Concrete, steel, masonry, FRP and superelastic shape-memory alloy material models are available, together with a large library of 3D elements that may be used with a wide variety of pre-defined steel, concrete and composite section configurations.

The nonlinear analysis solver of SeismoStruct, which features both distributed plasticity (fibre) and plastic hinge models and accounts for geometric nonlinearities and material inelasticity, has been extensively used and quality-checked by thousands of users for more than ten years. As described in the Verification Report of the program, its accuracy is very well demonstrated by its numerous successes in recent Blind Test Prediction Exercises.

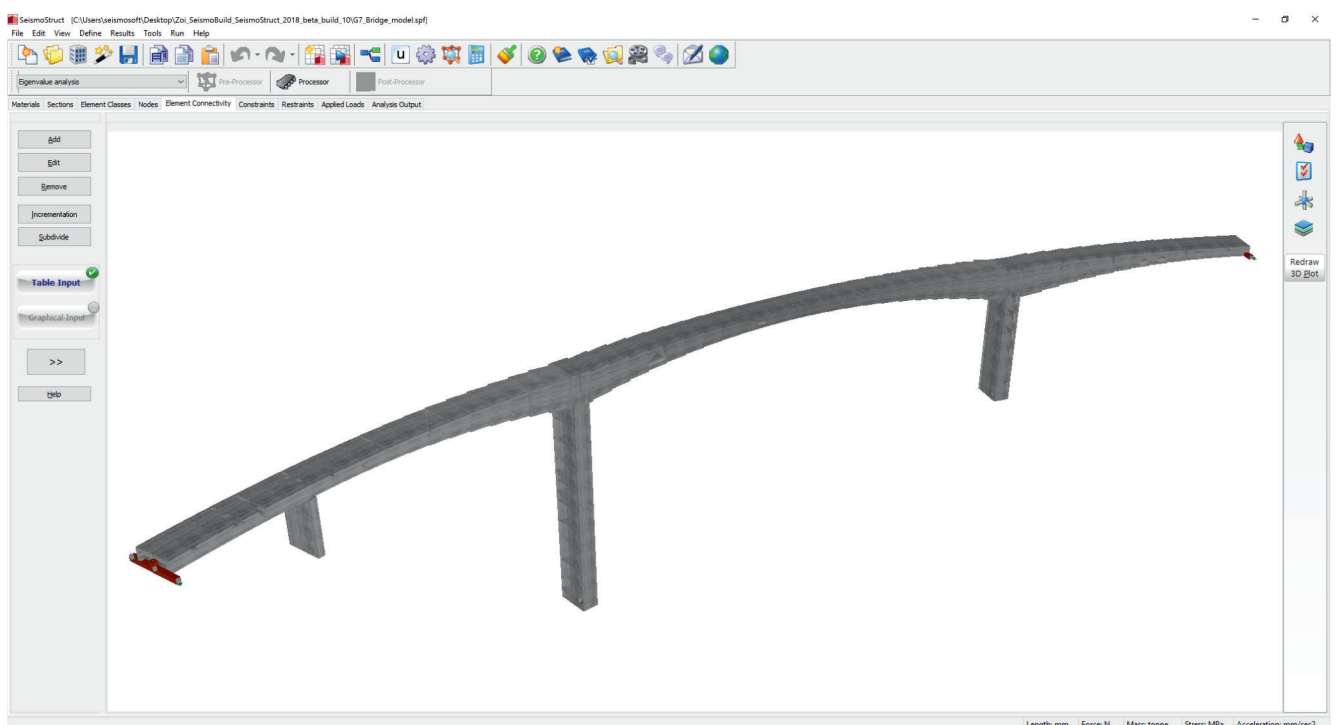


The rational and intuitive structure, as well as the simplicity of the package, which follows a very straightforward methodology with a series of input modules (tabs) from the materials definition to the performance criteria and the selected output, results in a very smooth learning curve even for engineers that do not have a deep knowledge of the Finite Elements method. Further, the Building Modeller and the Wizard facilities provide an extremely efficient, fast and intuitive way to create the structural model of regular and irregular reinforced concrete buildings, by introducing simply the geometry and the reinforcing details of the building's members.



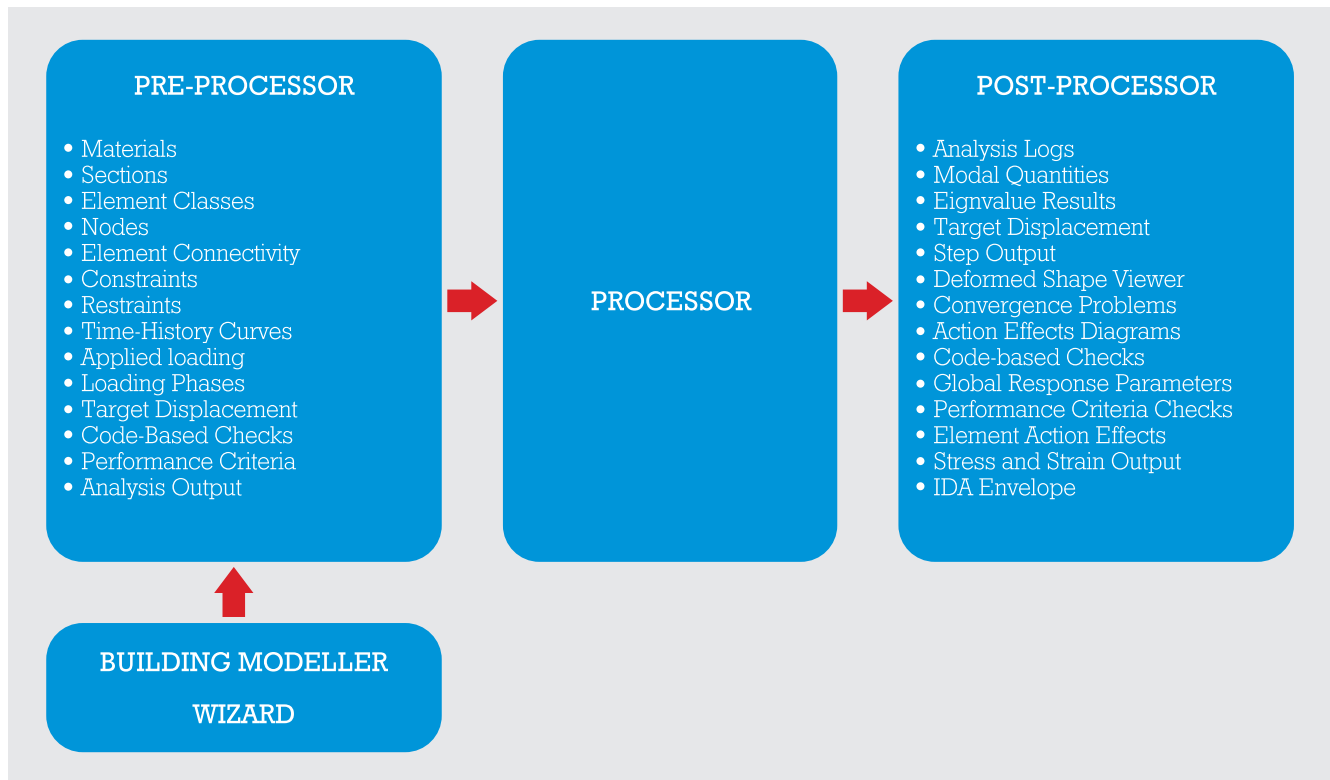
Seismosoft, which has been established as a leading enterprise in the field of earthquake engineering, is providing the engineering community with access to a powerful and state-of-the-art analytical tool that can be used by engineers that are not specialised in the Finite Elements Procedure.

The purpose of this document is to provide technical information about the SeismoStruct FE package. This includes information about SeismoStruct capabilities and its fundamental technical features.

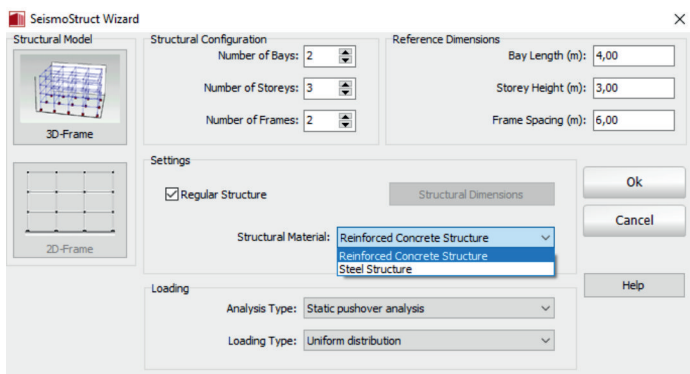


SeismoStruct overview

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. Further, the package includes two modules, the Building Modeller and Wizard facility, by which the user can create regular/irregular 2D or 3D models and run all types of analyses on the fly, with the whole process taking no more than a few minutes.



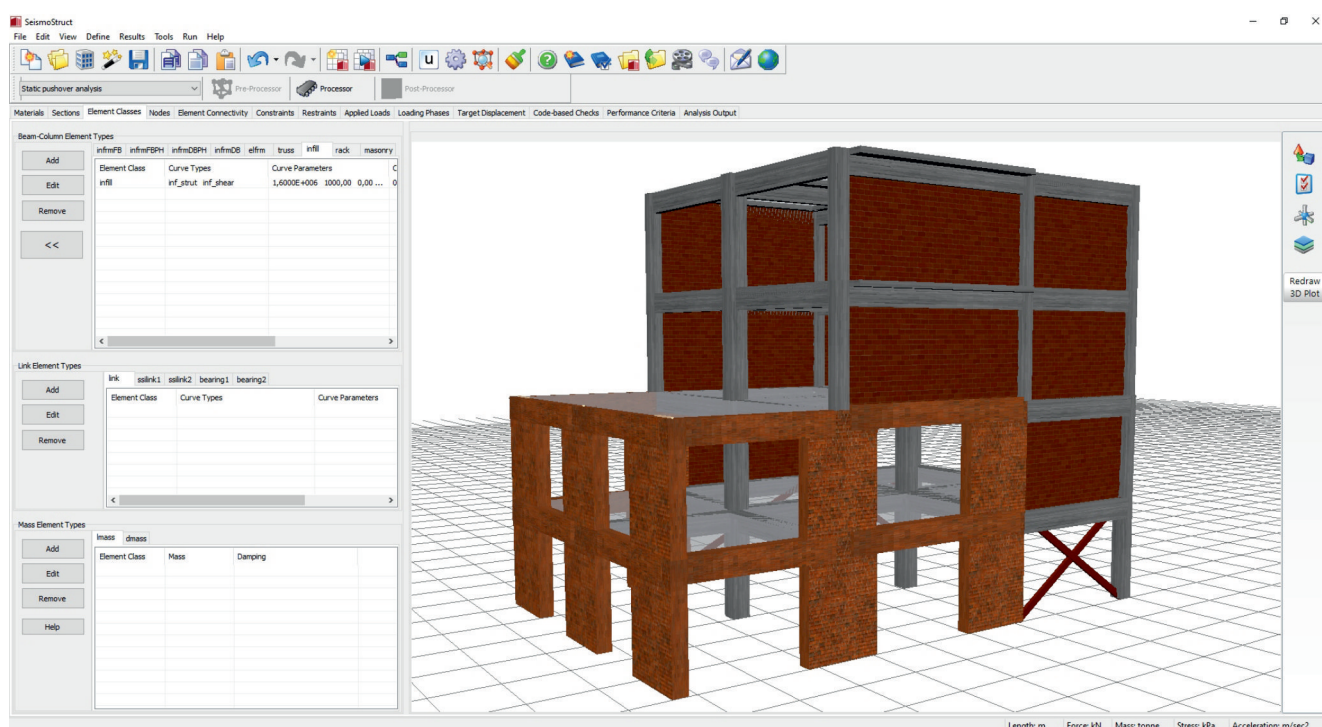
The Processor, moreover, features real-time plotting of the displacement curves of the structure, together with the possibility of pausing and re-starting the analysis to check the results. The Post-Processor offers advanced post-processing facilities, including the ability to custom-format all derived 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.

The software is fully integrated with the Windows environment. Input data created in spreadsheet programs, such as Microsoft Excel, may be pasted to the

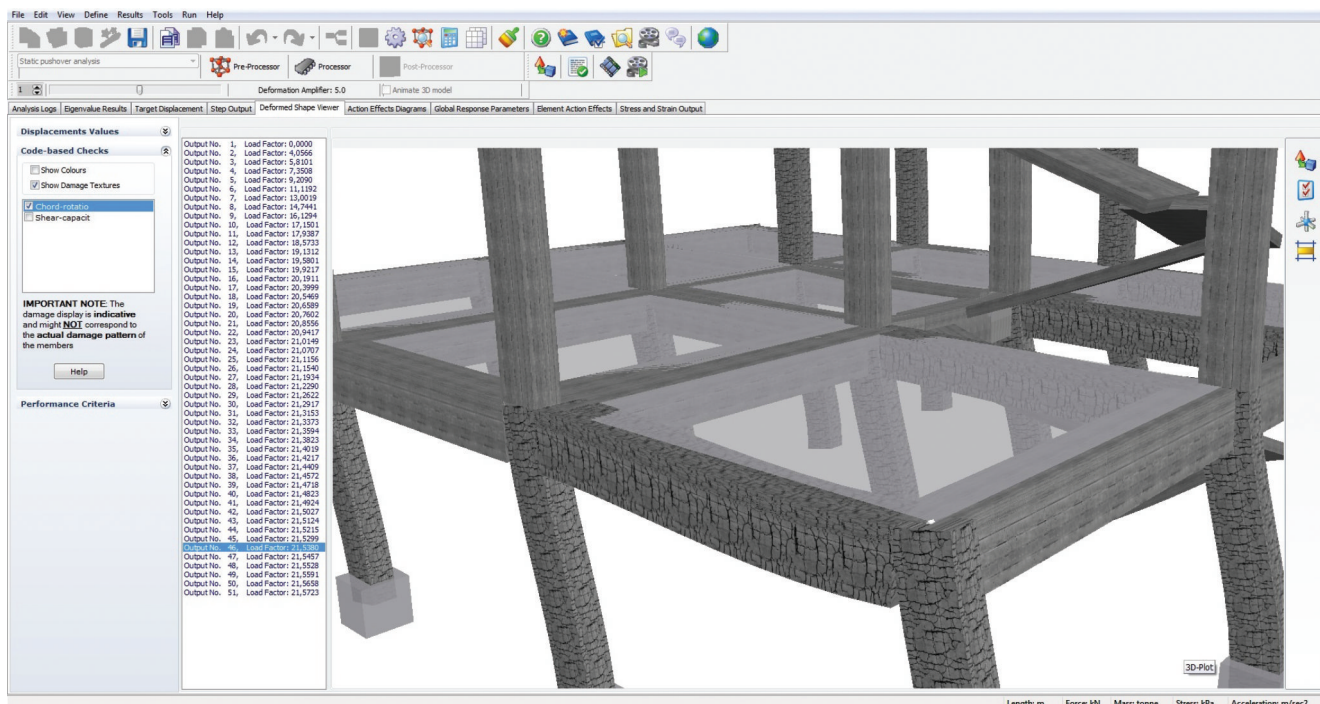
SeismoStruct input tables, for easier pre-processing. Conversely, all information visible within the graphical interface of SeismoStruct can be copied to external software applications (e.g. to word processing programs, such as Microsoft Word), including input and output data, high quality graphs, the models' deformed and undeformed shapes and much more.

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, response spectrum analysis, buckling, tsunami, eigenvalue analysis, and non-variable static loading.
- ✓ Twenty material models, such as nonlinear concrete models, a high-strength nonlinear concrete model, nonlinear steel models, a SMA nonlinear model, a general hysteretic model etc.



- ✓ A large library of 3D elements, such as nonlinear fibre beam-column elements, nonlinear plastic hinge elements, a nonlinear masonry element, a nonlinear rack element, a nonlinear truss element, a nonlinear infill panel element, nonlinear link elements, bearing elements, and soil-structure interaction elements, that may be used with a wide variety of pre-defined steel, concrete and composite section configurations.
- ✓ Twenty-nine hysteretic models, such as linear/bilinear/trilinear kinematic hardening response models, gap-hook models, soil-structure interaction model, Takeda model, Ramberg-Osgood model, Modified Ibarra-Medina-Krawinkler curves etc.
- ✓ Capacity checks for the frame and the masonry elements can be performed, according to Eurocode 8 and the Eurocodes framework, the American Code for Seismic Evaluation and Retrofit of Existing Buildings ASCE 41, the Italian National Seismic Code NTC-18, the Greek Seismic interventions Code KANEPE and the Turkish Seismic Evaluation Building Code TBDY, and for all the limit states of the specified Codes.
- ✓ Several Performance criteria for the frame, infill, masonry, link and truss elements are included 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 be in this manner readily obtained.
- ✓ 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.
- ✓ The program achieves numerical stability and accuracy at very high strain levels, enabling precise determination of the collapse load of structures.
- ✓ SeismoStruct's processor features real-time plotting of displacement curves and deformed shape of the structure, together with the ability of pausing and re-starting the analysis.
- ✓ Advanced post-processing facilities are available, including the ability to custom-format all derived plots and deformed shapes, thus increasing productivity of users.
- ✓ AVI movie files can be created to better illustrate the sequence of structural deformation.

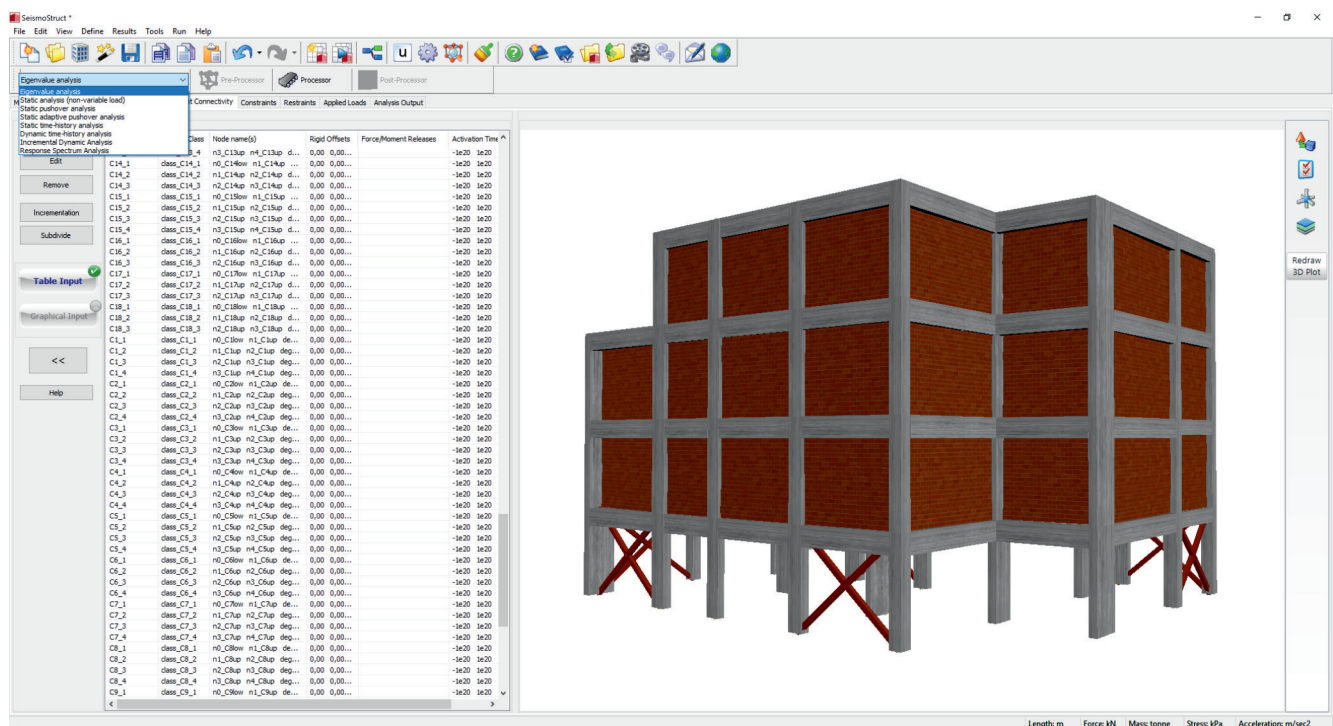
SYSTEM REQUIREMENTS

To use SeismoStruct, we suggest:

- ✓ A PC (or a "virtual machine") with one of the following operating systems: Windows 10, Windows 8 or Windows 7 (32-bit and 64-bit);
- ✓ 4 GB RAM;
- ✓ Screen resolution set to 1366x768 or higher (based on web browser statistics, as of 2022 about 96% of computers feature a screen resolution of 1366x768 or higher);
- ✓ An Internet connection (better if a broadband connection) for the registration of the software.

main SeismoStruct features

Below a list of the most important features of SeismoStruct is provided. SeismoStruct has been designed with both ease-of-use and flexibility in mind, and features a series of modules (pages) where the input can be defined in an intuitive way.



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 Analysis

UNITS

Units and reinforcing rebar types for both the metric and imperial system are supported.

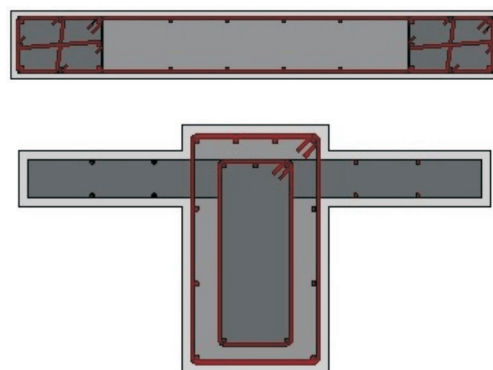
CODES AND STANDARDS

Five Standards worldwide are supported: Eurocode 8 and the Eurocodes framework, the American Code for Seismic Evaluation and Retrofit of Existing Buildings ASCE 41, the Italian National Seismic Code NTC-18, the Greek Seismic interventions Code KANEPE and the Turkish Seismic Evaluation Building Code TBDY.

MATERIAL MODELS

Currently, twenty material types are available in SeismoStruct, as depicted here after:

- ✓ Bilinear steel model - stl_bl
- ✓ Menegotto-Pinto steel model - stl_mp
- ✓ Dodd-Restrepo steel model - stl_dr
- ✓ Monti-Nuti steel model - stl_mn
- ✓ Buckling restrained steel braces model - stl_brb
- ✓ Bilinear steel model with isotropic strain hardening - stl_bl2
- ✓ Giuffre-Menegotto-Pinto Model with Isotropic Hardening - stl_gmp
- ✓ Ramberg-Osgood steel model - stl_ro
- ✓ 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 model - hyst_mat



SECTION TYPES

Currently, seventy-two (72) section types are available in SeismoStruct. These range from simple single-material solid sections to more complex reinforced concrete, steel, masonry and composite sections:

✓ **31 solid or steel sections:** rectangular and circular solid or hollow sections, symmetric I or T section, asymmetric general shape, and double angle or channel shaped section – dacss, several types of double angle, U-shaped or I sections, cruciform section

✓ **18 reinforced concrete sections:** rectangular section - rcrs, circular section - rccs, L-shaped column section - rclcs, T-shaped column section - rctcs, quadrilateral column section – rcqs, Z-shaped column section – rczcs, rectangular with rounded corners section – rccrccs, T-section - rcts, asymmetric rectangular section - rcars, rectangular wall section - rcrws, rectangular wall section without pseudo-columns – rcbws, U-shaped wall section - rcuws, L-shaped wall section - rclws, Z-shaped wall section – rczws, rectangular hollow section - rcrhs, circular hollow section - rcchs, box-girder section – rcbgs, rectangular hollow section with rounded corners - rccrchs

✓ **16 jacketed sections:** jacketed rectangular sections (1, 2, 3 and 4-sided), jacketed circular section, jacketed L-shaped column sections (3 and 4-sided), jacketed T-shaped column sections (3 and 4-sided), jacketed T-sections (1, 3 and 4-sided), jacketed asymmetric rectangular section – rcars (1 and 4-sided), jacketed rectangular section with rounded corners, jacketed Z-shaped column section, jacketed rectangular section with rounded corners

✓ **5 composite sections:** composite I-section - cpis, partially encased composite I section - peccs, fully encased composite I section - fecs, composite rectangular section - crs, composite circular section – ccs

✓ **2 masonry sections:** one masonry wall section and one masonry spandrel section

SeismoStruct allows also selecting predefined steel sections from a database of the most common steel sections (e.g. HEA, HEB, IPE, W, HSS etc.). More than 2000 steel section types have been included. Finally, user-defined sections can be introduced graphically for both reinforced concrete and steel members.

ELEMENT TYPES

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

- ✓ Three inelastic distributed plasticity frame elements (fibre modelling) - infrmFBPH, infrmFB and infrmDB
- ✓ One inelastic lumped plasticity element - infrmDBPH
- ✓ Elastic frame element - elfrm
- ✓ Elastic frame element with inelastic hinges - elfrmH
- ✓ Inelastic masonry element - masonry
- ✓ Inelastic infill panel element - infill
- ✓ Inelastic truss element - truss
- ✓ Rack element with geometric nonlinearities - rack
- ✓ Rack element with geometric nonlinearities and inelastic hinges - rackH
- ✓ Link element - link
- ✓ Elastomeric bearing element - bearing1
- ✓ Friction pendulum bearing element - bearing2
- ✓ Two soil-structure interaction elements - ssilink1 and ssilink2
- ✓ Mass elements - lmass & dmass
- ✓ Damping element - dashpt

By making use of these element types, the user is able to create an unlimited number of different element classes that are not only able to accurately represent intact/repared 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.

RESPONSE CURVES

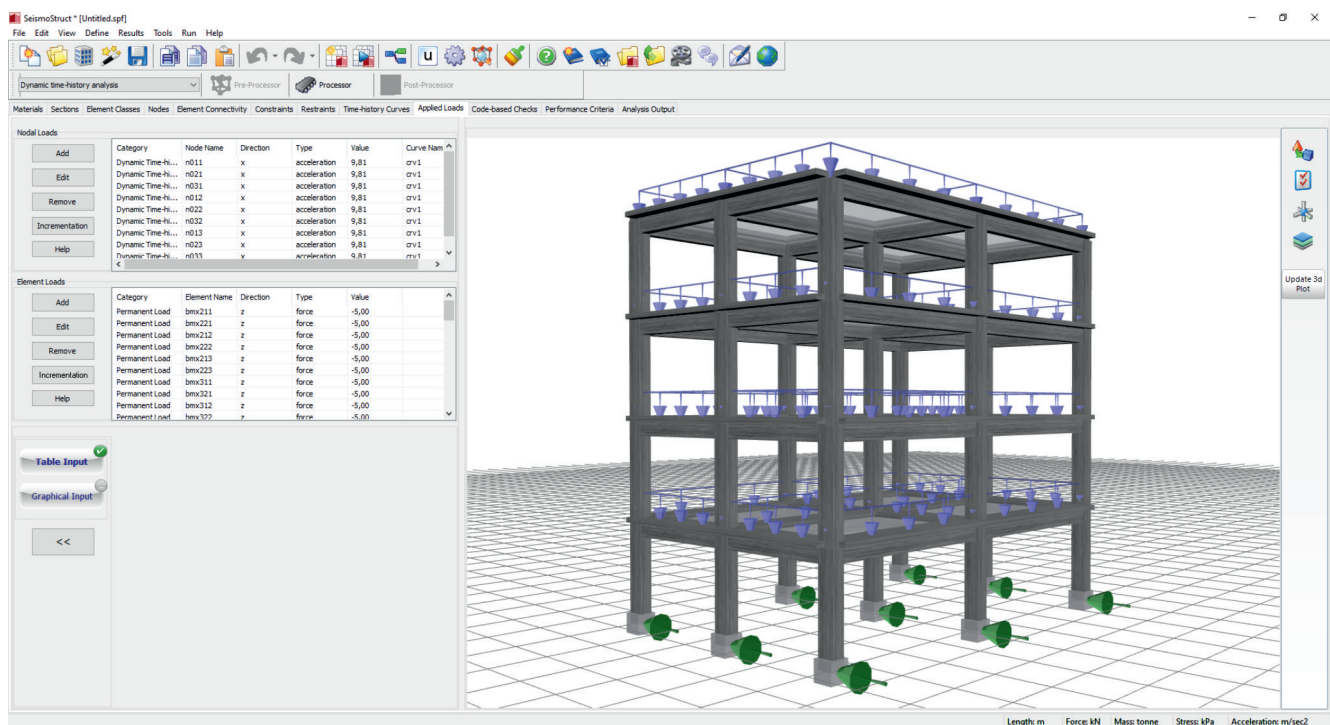
Currently, twenty nine response curves are available to be used with the nonlinear link element.

- ✓ 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 curve - scb
- ✓ Generic Hysteretic curve - gen_hyst

STRUCTURAL GEOMETRY

where the input for the definition of the structural configuration is defined.

- ✓ **Nodes:** The X, Y & Z global coordinates of the nodes are defined. A Graphical Input facility that features snapping, and an Incrementation facility are also available.
- ✓ **Element Connectivity:** The different elements of the structure are defined herein, where, apart from the type and the corresponding nodes, the rigid offsets at the member ends, the force/moment releases and the activation/deactivation time or loading factor are specified. The orientation of the linear elements is determined by means of a rotation angle or an orientation (non-structural) node. Graphical Input, Incrementation and Subdivision facilities are also available.
- ✓ **Constraints:** The different constraining conditions of the structure are defined by means of three different nodal constraint types available, Rigid Links, Rigid Diaphragms and Equal DOF constraints. As in the case of elements, Graphical Input and Incrementation are available.
- ✓ **Restraints:** In a list of all the structural nodes, restraints against deformation in any of the six degrees-of-freedom may be selected.



APPLIED LOADING

Four types of loading are available:

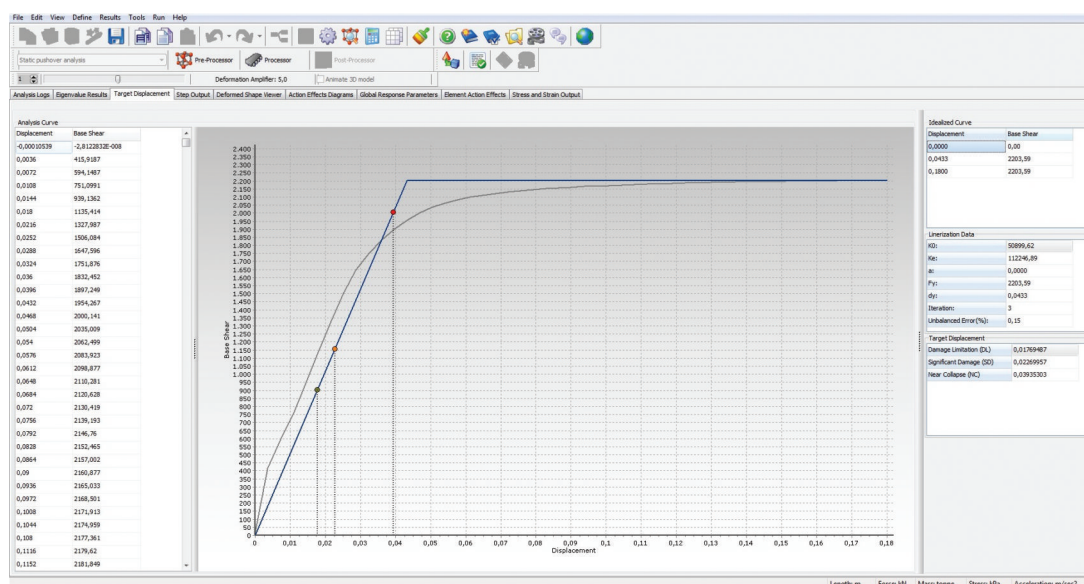
- ✓ **Permanent Load:** These are nodal or distributed loads along the element's length applied at the beginning of the analysis
- ✓ **Incremental Loads:** These represent pseudo-static loads (forces or displacements) that are incrementally varied. The magnitude of a load at any step is given by the product of its nominal value, and the current load factor, which is updated in automatic or user-defined fashion. Incremental loads are exclusively employed in pushover type of analyses. The loading factor is controlled in three ways: (i) load control, whereby the load factor is directly controlled, (ii) response control, whereby the loading factor is indirectly calculated, so that the response of a particular node assumes pre-defined values, and (iii) automatic response control, which is similar to the response control scheme, with the difference that the program automatically chooses which nodal degree-of-freedom to control during the analysis
- ✓ **Static time-history loads:** 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)

✓ **Dynamic time-history loads:** 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

TARGET DISPLACEMENT

The Target Displacement of the structure is automatically calculated in pushover analysis, according to the selected Code (Eurocodes, American Code for Seismic Evaluation and Retrofit of Existing Buildings ASCE 41, Italian National Seismic Code NTC-18, Greek Seismic Interventions Code KANEPE and Turkish Seismic Evaluation Building Code TBDY).



CODE-BASED CHECKS

SeismoStruct allows for the implementation of the code-based checks for the reinforced concrete members (chord rotation, shear and bending moment capacity) and for the masonry members (shear and bending moment capacity, drift and compressive force), according to the selected Standard (Eurocode 8, ASCE 41, NTC-18, KANEPE or TBDY) for all the limit states of the relevant Code. Parameters such as the specification of members as primary or secondary, the lap splices, the type of steel (ductile, ribbed etc.) or the availability of earthquake resistance may be easily defined.

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 during the analysis of a structure is automatically monitored by the program. Several performance criteria are available for frame, link, masonry, infill and truss elements, including material strains, section curvatures, chord rotation capacities or user-specified limits, shear capacities or user-specified limits, drifts, deformations, forces and bending moments.

CURVATURES, STRAIN & STRESS MONITORING

Users can specify the elements, for which output of curvatures and stress/strain peak values (maxima and minima) will be provided. It is also possible to follow the variation during the analysis of the stress and strain of a particular material, located at a given point of a specific integration section of an inelastic frame element. The definition of these stress points is done in a completely graphical manner.

DAMPING

Apart from the hysteretic damping, which usually is already implicitly included within the nonlinear formulation of the

inelastic frame and link elements, the possibility of defining viscous damping, either globally to all the structural members or locally to specific members, is provided. Four types of damping are available depending on the element type: stiffness-proportional, mass-proportional, Rayleigh damping and radiation damping.

CONVERGENCE CRITERIA AND ITERATIVE STRATEGY

Four different schemes are available in SeismoStruct for checking the convergence of a solution at the end of every iteration: (i) displacement/rotation based, (ii) force/moment based, (iii) displacement/rotation AND force/moment based, and (iv) displacement/rotation OR force/moment based. The user is asked to specify the convergence tolerance values, as well as certain iterative strategy parameters, such as the maximum number of iterations, the load subdivision factors, the number of initial stiffness updates etc.

INTEGRATION SCHEME

In dynamic time-history and IDA analysis, users may choose between the Hilber-Hughes-Taylor and the Newmark integration algorithms.

GRAVITY AND MASS

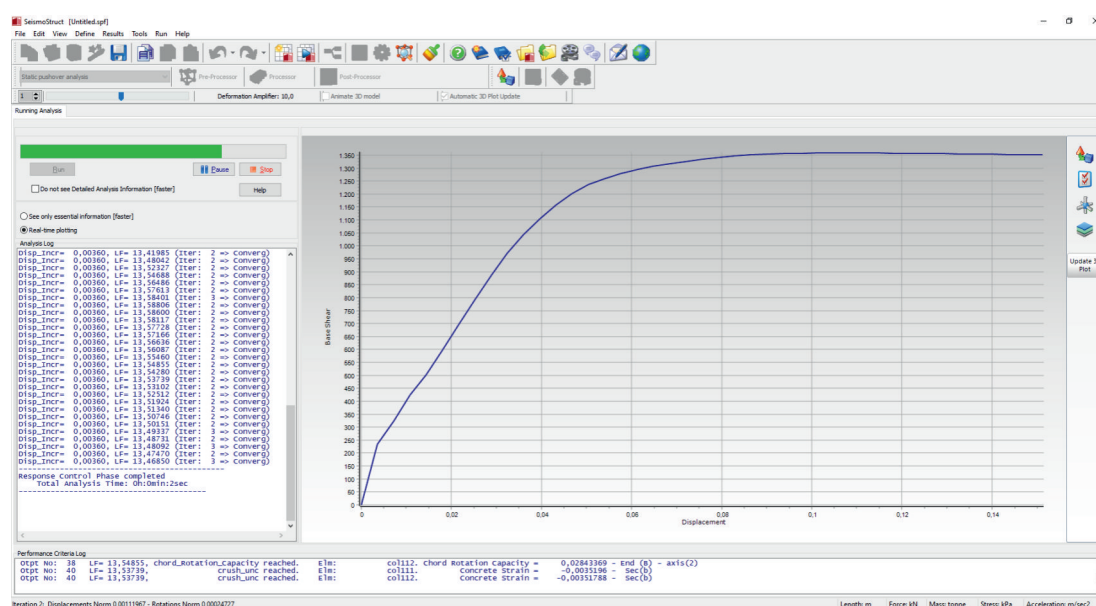
By defining the materials specific weights, the distributed self-mass of the structure can be considered. Additional distributed mass may also be assigned to frame elements, and lumped and distributed mass-only elements can also be defined. Furthermore, it is possible for users to define if and how such 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.

SOLUTION ALGORITHMS

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. 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 or the maximum number of iterations, specified by the user, has been reached.

For the solution of the linear equations 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 Mackay et al. [1991] and featuring a compact row storage scheme using elimination trees proposed by Liu [1986];
- ✓ A Parallel Sparse/Profile Solver for sparse systems, which is the parallel version of the Mackay et al. algorithm. The method was introduced by Law and Mackay [1992].



AUTOMATIC ADJUSTMENT OF LOAD INCREMENT OR TIME-STEP

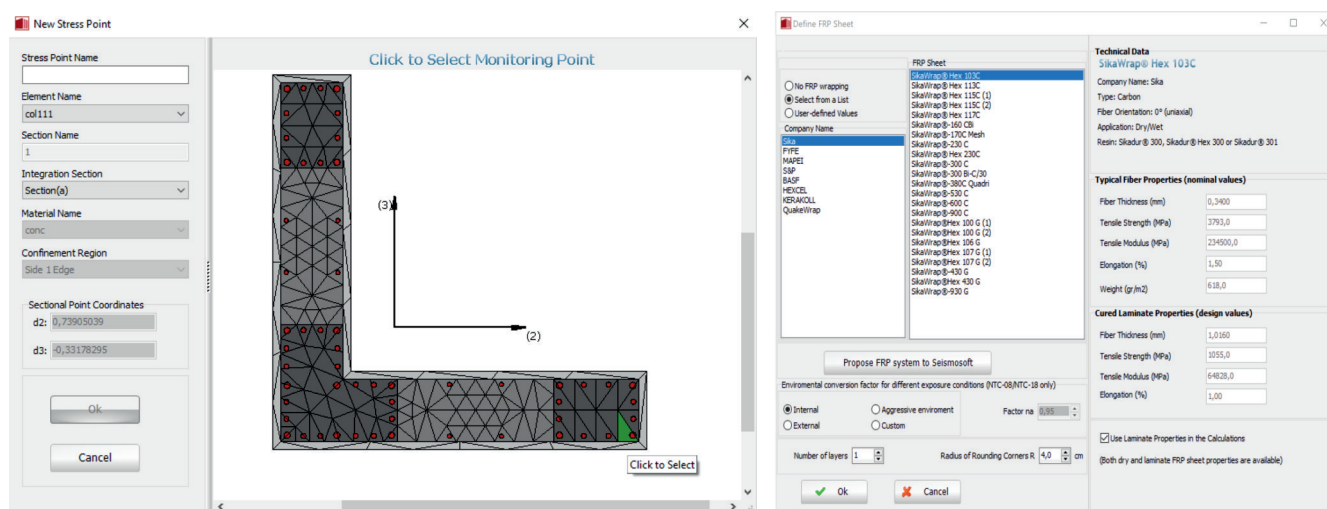
SeismoStruct possesses the ability to smartly subdivide the loading (or time-step) increment, whenever convergence problems arise, with the level of subdivision depending on the convergence difficulties encountered. When the convergence difficulties are overcome, the program automatically increases the loading increment back to its original value. In this way, the program achieves numerical stability and accuracy at very high strain levels, enabling precise determination of the collapse load of structures.

EIGEN-SOLVER ALGORITHMS

Users may choose between two different eigensolvers, the Lanczos algorithm presented by Hughes [1987] or the Jacobi algorithm with Ritz transformation.

STRESS RECOVERY

Stress-recovery algorithms are employed in the frame element formulations, so that to retrieve the correct internal forces of an element subjected to distributed loading even if its nodes do not displace.



BUILDING MODELLER

The Building Modelling facility constitutes an extremely efficient, fast and intuitive way to create the structural model of a reinforced concrete building. Its main characteristics are the following:

- ✓ Ability to import CAD drawings as background to facilitate the structural input and the definition of the members' geometry and location. Snap, grid and ortho facilities are available. Other floors plan view may also be employed as background.
- ✓ The structural input of one floor may easily be copied to any other floor. Further, a renumbering facility is available.
- ✓ The cross section of any column or beam can be defined by employing one of the predefined built-in cross-sections from SeismoStruct's internal library. Both existing and strengthened (jacketed, 1, 2, 3 or 4-sided) cross-sections are available in various shapes (rectangular, L-shaped, T-shaped, circular, shear walls, normal or inverted T-beams). Detailed guidelines are provided for the correct modelling of more complicated wall configurations (e.g. U-shaped or Z-shaped walls).
- ✓ The modelling of the typical reinforcing bars, both longitudinal and transverse, can be quickly defined at specific locations (corners or sides of the section) by using reinforcing patterns, whereby the number of bars, and their sizes is defined. Additional rebars may be graphically defined with a single click.
- ✓ A large number of material types may be defined for both existing and new members. Predefined material sets with material characteristics, according to the regulations (e.g. C20/25 and S500) are also available. In strengthened (jacketed) sections, both new and existing materials may be employed.

✓ The user can easily model strengthening with Fibre-Reinforced Polymer (FRP) wraps by specifying just the very basic characteristics of the FRP sheets (thickness, strength, stiffness and maximum elongation). Moreover, a large library of existing FRP materials available in the market has been included and may be easily used by specifying just the name of the product.

✓ The strengthening with steel braces can be easily introduced. Currently the following configurations are supported: (i) X-Brace with connected diagonals, (ii) X-Brace with disconnected diagonals, (iii) diagonal brace, (iv) inverted diagonal brace and (v) V-Brace. The cross section of the brace members is selected from a large library of pre-defined steel sections.

✓ Infill panels can also be defined. The strength and stiffness of the panel are automatically calculated by the main structural parameters that affect them (geometry, brick and mortar dimensions, material strengths and openings). When the infills do not cover the entire floor height, the short concrete columns that are formed above them are automatically identified and modelled by the program.

✓ Slabs may be introduced with a single mouse click, after which the program graphically determines the slab edges at the surrounding beams. Live loading can be defined by selecting the category of the loaded area, as specified in the chosen Standard.

The boundary conditions and the slab reinforcement may be easily adjusted by the user. Finally, slab edges and slab cantilevers may be defined by specifying the free edge of the slab as a line and by then adding the slab.

✓ The stairs can be defined by specifying the centreline and some basic geometric parameters, such as the stairs width and the riser height.

✓ Inclined beams may be efficiently modelled by specifying the height of the two ends. The height of the supporting columns is then automatically adapted. 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.

✓ In a similar fashion, inclined slabs are modelled by specifying the slab level at three points that are graphically selected by the user. The neighbouring beams' levels and columns' heights are automatically adjusted, and the columns are subdivided in shorter elements by the program, if needed.

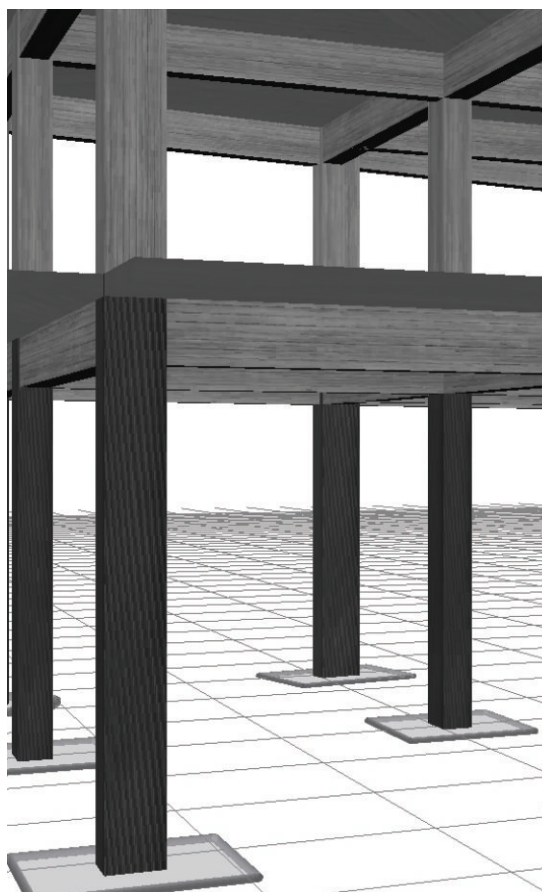
✓ Parameters such as the specification of members as primary or secondary, the lap splices, the cover thickness, the type of steel (ductile, ribbed etc) or the availability of earthquake resistance may be easily specified, as well.

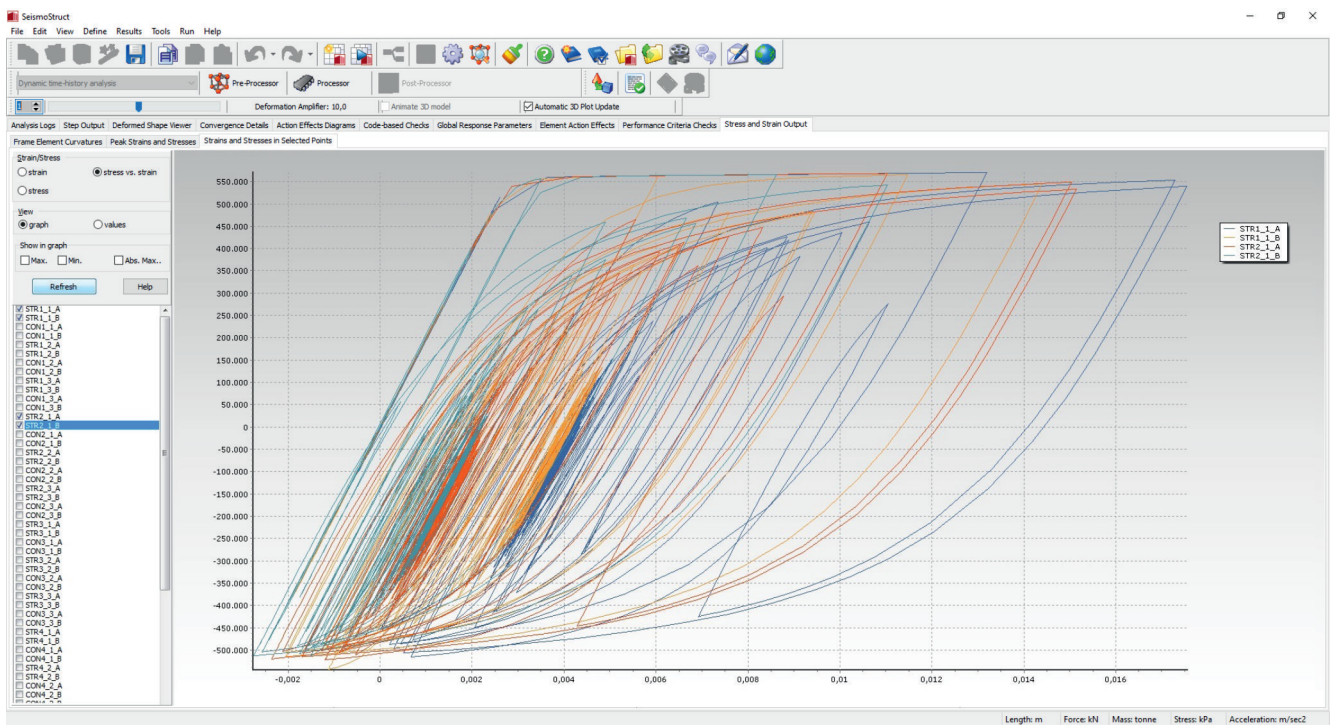
WIZARD

The wizard is a facility that enables the creation of frame/building models on-the-fly. By providing some basic geometric parameters and the type of analysis, the program automatically creates a structural model that is ready to run.

PROCESSOR, REAL-TIME PLOTTING

After completing the pre-processing phase, the user may run the analysis in the Processor area of SeismoStruct. As the analysis is running, a progress bar provides the user with a percentage indication of how far has the former advanced to. Further, the real-time plotting of a selected deformation parameter is allowed (e.g. displacement of a specific node). The analysis can also be paused, enabling users to check the results obtained up to that point, which may be useful to decide the worthiness of progressing with a lengthy analysis.

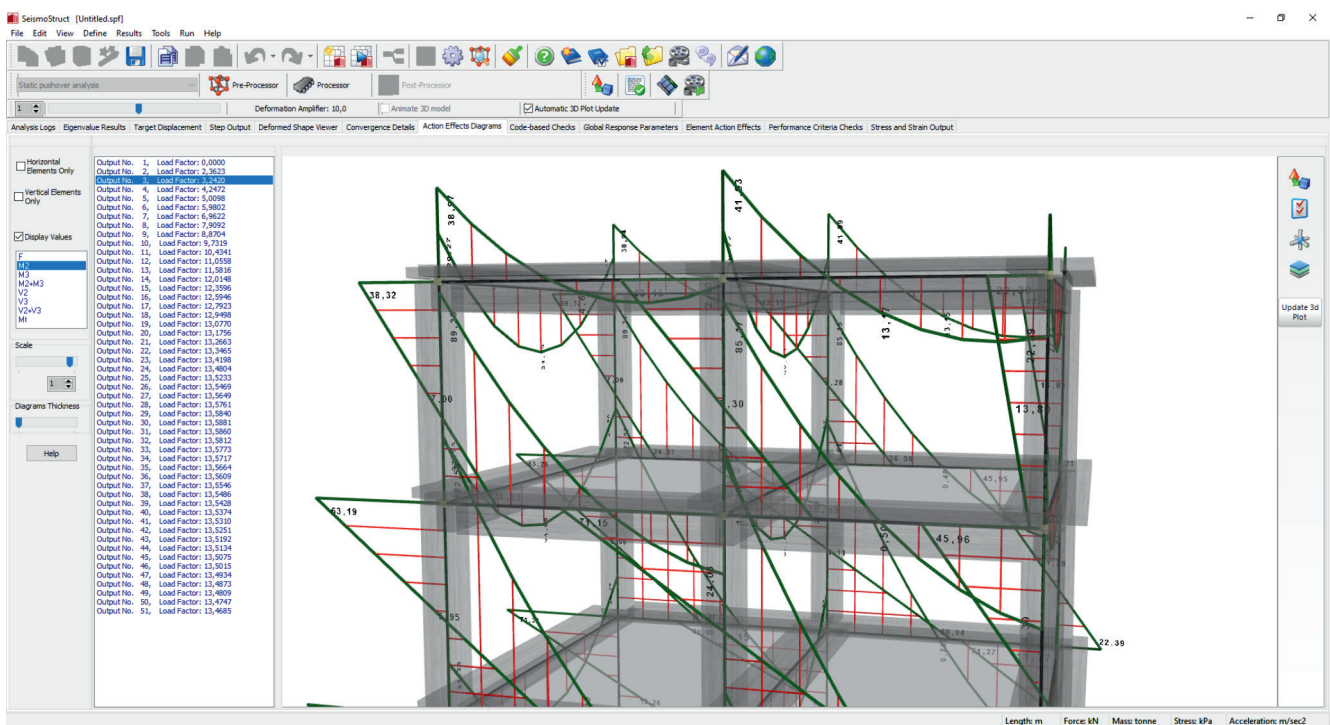




POST-PROCESSOR

The post-processor area features a series of modules, where results from the different analyses may be obtained in table or graphical format. Deformations, support forces, members internal forces and curvatures, stresses and strains can be viewed. Further, the deformation shape of the structural model at the different steps of the analysis is displayed, as well as the 3D plot of the building with the action effects diagrams (axial and shear forces, bending and torsional moments).

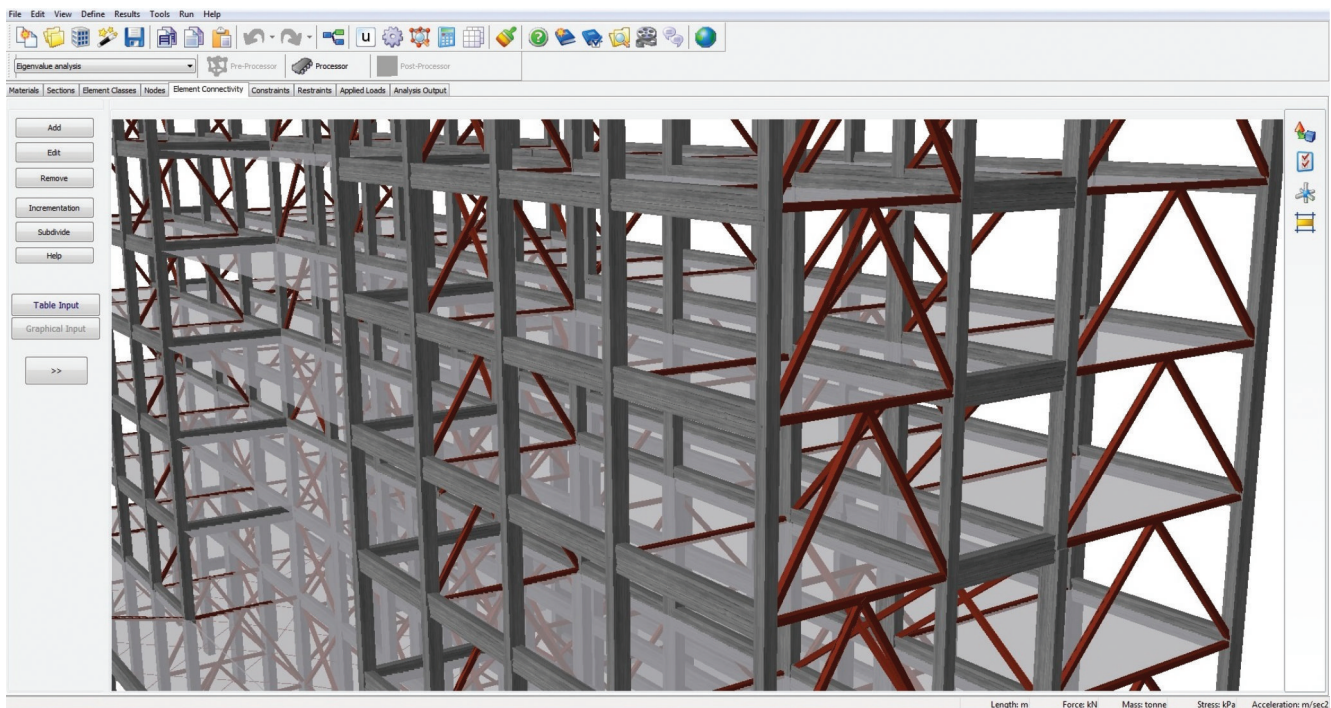
All the results can be copied into any other Windows application (e.g. tabled results can be copied into a spreadsheet applications like Microsoft Excel, whilst the results plots and high quality graphs of the models' deformed and undeformed shapes can be copied into a word-processing application, like Microsoft Word). It is also possible to create AVI movie files to better illustrate the sequence of structural deformations.



why use SeismoStruct

1. It has a very smooth learning curve

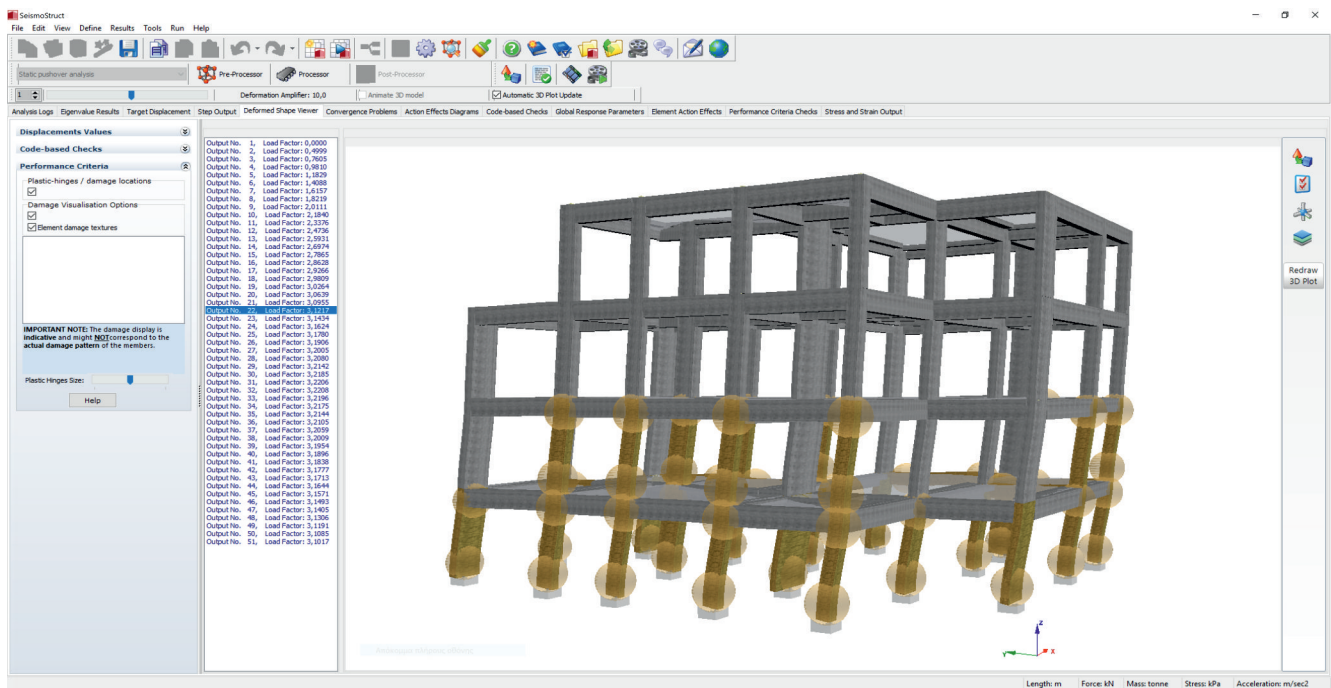
Featuring an extremely intuitive graphical environment with a series of input modules (tabs) and a very straightforward methodology from the materials definition to the performance criteria and the selected output, SeismoStruct is extremely easy to learn. Neither manuals, nor tutorial examples are required. Any engineer with just a limited knowledge of the Finite Elements procedures can open the program and start using it. Further, the Building Modeller and the Wizard facilities constitute two modules, with which a user can create on-the-fly reinforced concrete building models, by introducing just the geometric and reinforcement parameters of the structure.



2. It offers enhanced accuracy and reliability

SeismoStruct has been extensively used and verified by thousands of users for more than ten years. It employs the fibre approach to represent the cross-section behaviour, where each fibre is associated with a uniaxial stress-strain relationship. This approach is generally more accurate, especially with non-ordinary sections, which is why it gradually gains significance in the modelling of framed members. Moreover, the distributed plasticity force-based elements with the integration sections located along the length of the member are most suitable for large shear walls, where inelasticity occurs along their entire height and not just at their ends, something that is explicitly assumed by the programs that employ the plastic hinge approach.

The accuracy of the solver in the nonlinear analysis of framed structures is very well demonstrated by the numerous successes of SeismoStruct in recent Blind Test Prediction Exercises, such as the Concrete Column Blind Prediction Contest 2010 (UCSD, San Diego, USA), which was sponsored by the Network for Earthquake Engineering Simulation (NEES) and the Pacific Earthquake Engineering Research Center (PEER, Berkeley University), or the Blind Test Challenge of the 15th World Conference on Earthquake Engineering (LNEC, Lisbon, Portugal), amongst others.

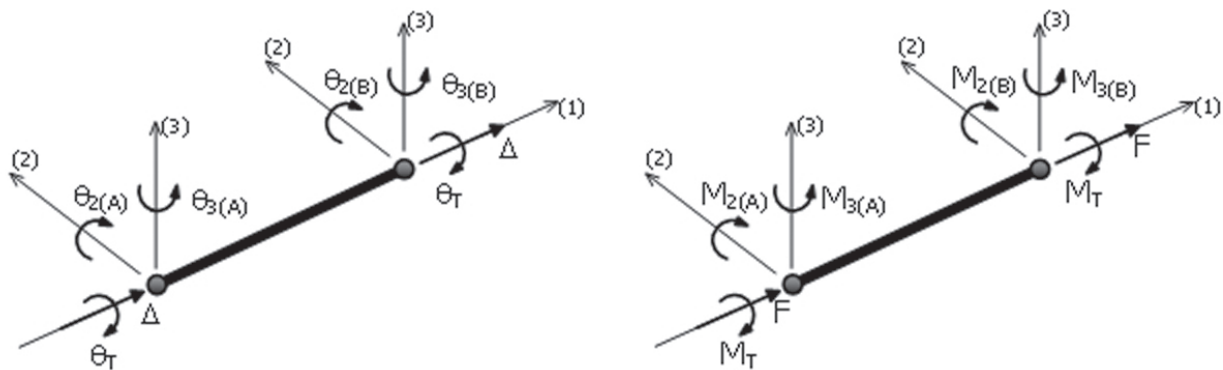


GEOMETRIC NONLINEARITY

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 loosing 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:

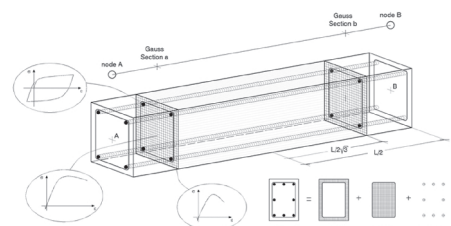


MATERIAL INELASTICITY

Distributed inelasticity elements are becoming widely employed in earthquake engineering applications, either for research or professional engineering purposes. 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 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.

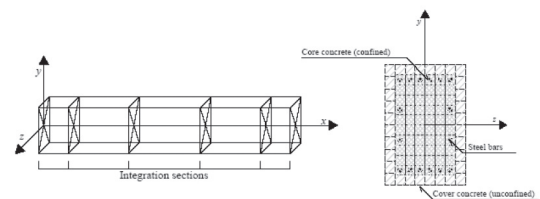
Distributed inelasticity frame elements can be implemented with two different finite elements (FE) formulations: the classical displacement-based (DB) ones [e.g. Hellebrand and Scordelis 1981; Mari and Scordelis 1984], and the more recent force-based (FB) formulations [e.g. Spacone et al. 1996; Neuenhofer 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. Alemndar 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. 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. The use of Gauss-Lobatto integration sections 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.

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.

ANALYSIS TYPES

Currently, ten analysis types are available in SeismoStruct:

✓ **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.

✓ **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.

✓ **Static Pushover Analysis** This is the conventional (non-adaptive) pushover analysis that is frequently utilised to estimate the horizontal capacity of structures. The applied incremental load P is kept proportional to the pattern of nominal loads (P^0) initially defined by the user: $P = \lambda(P^0)$. 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.

✓ **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. Apart from most common force distributions, adaptive pushover in SeismoStruct 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.

✓ **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 $P_i = \lambda_i(t)P_i^0$, i.e. a function of the time-dependent load factor $\lambda_i(t)$ and the nominal load P_i^0 . This type of analysis is typically used to model static testing of structures under various force or displacement patterns.

✓ **Dynamic Time-history Analysis** Dynamic analysis is commonly used to predict the nonlinear inelastic response of a structure subjected to earthquake loading. 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, progressive collapse etc.), in which case force pulse functions of any given shape (rectangular, triangular, parabolic, and so on), can be employed to describe the transient loading at the appropriate nodes, or the removal of specific members.

✓ **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.

✓ **Buckling Analysis** In slender steel structures, the limit point that recognises the transition from a stable to an unstable structure 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 an incremental one. Besides these two assumptions, the element geometric stiffness matrices are linear functions of their end forces.

✓ **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. 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 user-defined.

MATERIAL MODELS

SeismoStruct has a range of material models to be used, in order to cover all modelling needs. The available models are listed below:

Bilinear steel model - stl_bl: is a uniaxial bilinear stress-strain model with kinematic strain hardening, whereby the elastic range remains constant throughout the various loading stages, and the kinematic hardening rule for the yield surface is assumed as a linear function of the increment of plastic strain. This simple model is also characterised by its computational efficiency and can be used in the modelling of both steel structures, where mild steel is usually employed, as well as reinforced concrete models, where worked steel is commonly utilised.

Menegotto-Pinto steel model - stl_mp: 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.

Dodd-Restrepo steel model - stl_dr: 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.

Monti-Nuti steel model - stl_mn: 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).

Buckling restrained steel brace model - stl_brbr: is a uniaxial steel material model describing the behaviour of steel in Buckling Restrained Braces. The model has been presented by Zona et al. [2012].

Bilinear steel model with isotropic strain hardening - stl_bl2: 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.

Ramberg-Osgood steel model - stl_ro: is the Ramberg-Osgood stress-strain model [Ramberg and Osgood, 1943], as described in the work of Kaldjian [1967] and Otani [1981].

Giuffre-Menegotto-Pinto Model with Isotropic Hardening – `stl_gmp`: 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.

Trilinear concrete model - `con_tl`: is a simplified uniaxial trilinear concrete model that assumes no resistance to tension and features a residual strength plateau.

Mander et al. nonlinear concrete model - `con_ma`: 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.

Chang-Mander nonlinear concrete model - `con_cm`: 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.

Kappos and Konstantinidis nonlinear concrete model - `con_hs`: 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. It is noted that the need for a special-purpose high-strength concrete model arises 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.

Engineered Cementitious Composites material - `con_ecc`: is a uniaxial generic material modelling the behaviour of ductile fiber-reinforced cement-based composites as described by Han et al. [2003].

Kent - Scott - Park concrete model - `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.

Trilinear masonry model- `mas_tl`: is a simplified uniaxial trilinear material model that assumes no resistance to tension and features a residual strength plateau.

Parabolic masonry model- `mas_par`: is a uniaxial trilinear material model for masonry that is based on the hysteretic rules of the `con_ma` typical constant confinement concrete model.

Superelastic shape-memory alloys model - `se_sma`: 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.

Trilinear FRP model - `frp_tl`: is a simplified uniaxial trilinear FRP model that assumes no resistance in compression.

Elastic material model - `el_mat`: is a simplified uniaxial elastic material model with symmetric behaviour in tension and compression.

Generic Hysteretic Model - `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 residual strength is initially defined and deteriorates after each unloading incident.

ELEMENT TYPES

Nineteen element types are available in SeismoStruct to cover your modelling needs:

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. 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. Hence, its use allows for very high accuracy in the analytical results.

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 advantage of such formulation is a reduced analysis time, since fibre integration is carried out for the two member end sections only.

Inelastic displacement-based plastic hinge frame element type - `infrmDBPH`

This is a displacement-based plastic-hinge 3D beam-column element with concentrated plasticity at the two element's ends. It is a typical one-component Giberson model [Giberson, 1967], which consists of one elastic girder and four nonlinear rotational springs attached at the two element's ends in both the 2nd and the 3rd local axis.

All nonlinear deformations of the element are lumped in these rotational springs, whereas the rest of the member remains elastic. The moment-rotation curves in the two local axes at each end are independent. This is obviously a simplification with respect to the force-based plastic hinge element, where inelastic deformations spread over a finite region at the ends of the girder and the behaviour in the two local axes is correlated. However, this lack of accurate modelling is compensated by increased stability and significantly shorter analysis times.

The `infrmDBPH` formulation is capable of modelling the geometric nonlinearities, whereas material inelasticity at the plastic hinges is modelled with an asymmetric Takeda type of curve that features different yield values in tension and compression. The monotonic curve is described by a trilinear skeleton curve, which accounts for cracking of concrete and yielding of reinforcing steel. It is noted that the hysteretic curve parameters are automatically calculated by the program, hence users need only specify the member's section.

Inelastic displacement-based frame element type – `infrmDB`

This is the displacement-based 3D beam-column element type capable of modelling members of space frames with geometric and material nonlinearities. As described in the Material inelasticity paragraph, the sectional stress-strain state is obtained through the integration of the nonlinear uniaxial material response of the individual fibres in which the section has been subdivided, fully accounting for the spread of inelasticity along the member length and across the section depth.

The displacement-based formulation follows a standard FE approach [e.g. Hellesland and Scordelis 1981; Mari and Scordelis 1984], where the element deformations are interpolated from an approximate displacement field, before the PVD is used to form the element equilibrium relationship. The DB formulation features two integration sections per element, and the Gauss quadrature is employed for higher accuracy.

In order to approximate nonlinear element response, constant axial deformation and linear curvature distribution are enforced along the element length, which is exact only for prismatic linear elastic elements. Consequently, `infrmDB` should be employed with members of small length, for which reason `infrmDB` elements are used in `SeismoStruct` only to model short columns and beams.

Elastic frame element - `elfrm`

For the 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.), the employment of an elastic linear frame element might be desirable, for which reason element type `elfrm` has been developed and implemented in `SeismoStruct`.

Elastic frame element with hinges - `elfrmH`

This is the linear elastic element that features plastic hinges at its two ends in both local axes. The internal elastic sub-element and the two external links are connected in series. The hinges are simulated through the inelastic response curves that are available in `SeismoStruct`.

Inelastic masonry frame element – `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.

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, in order 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.

Rack element with hinges- rackH

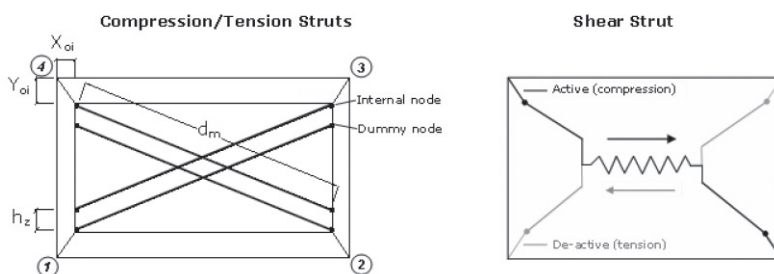
This is the standard rack element that features plastic hinges at its two ends in both local axes. The internal elastic sub-element and the two external links are connected in series, and the hinges are simulated through the inelastic response curves that are available in SeismoStruct.

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.).

Inelastic infill panel element type

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.



Link element type - link

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.

Currently, twenty-nine response curves are available to be used with the nonlinear link element.

- ✓ 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 curve - scb
- ✓ Generic Hysteretic curve - gen_hyst

Shallow footings macro-element - [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.

Pile Foundations macro-element - [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.

Elastomeric Bearing 1 Element (Bouc Wen) - [bearing 1](#)

This is a 3D element with zero length used to model the behaviour of elastomeric bearings employed in Seismic Isolation applications. Bearing 1 elements have coupled plasticity properties for the two shear directions (axes 2 and 3 in the local coordinate system), while they are characterised by linear elastic behaviour for the remaining four deformation types. The behaviour in the shear directions is based on the hysteretic behaviour proposed by Wen [1976] and Park et al. [1986].

Friction Pendulum Bearing/System – [bearing 2](#)

This is a 3D element with zero length used to model the behaviour of single friction pendulum systems employed in Seismic Isolation applications. Bearing 2 elements have coupled plasticity properties for the two shear directions (axes 2 and 3 in the local coordinate system), 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 bearing sliding surface.

Mass elements - [lmass & dmass](#)

Lumped (lmass) and distributed (dmass) mass-only elements can be defined and added to the structure, so that users may model mass distributions that cannot be obtained using the Materials/Sections facilities; e.g. water tank with concentrated mass on top.

Damping element - [dashpt](#)

This is a single-node damping element, which may be employed to represent a linear dashpot fixed to the ground. Damping coefficients may be defined on all six global degrees-of-freedom, though, commonly, dampers will work only in one or two directions.

company overview

Seismosoft was founded in 2002 with the aim of providing the earthquake engineering community with access to powerful and state-of-the-art analytical tools, and is now recognised as a leading enterprise in this field. It has always focused not only on technical advancement but also on ease-of-use, so as to facilitate effective adoption and employment by both researchers and practitioners. As a result, its software applications are requested at a rate of 1000 downloads per month, from more than 110 countries, and are employed in hundreds of international academic/research institutions, practicing companies and governmental bodies.

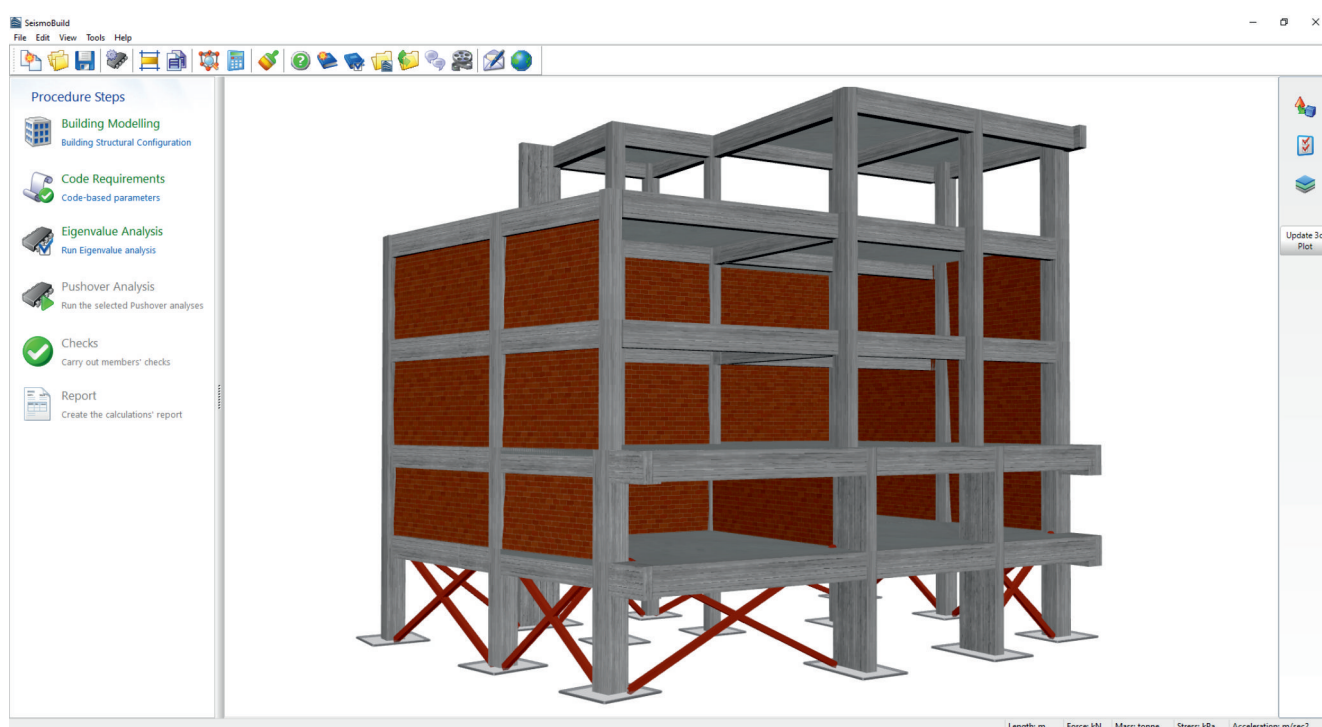
Ultimately, by rendering readily available to the full spectrum of earthquake engineering stakeholders tools and methods that feature not only technical excellence but also efficiency and user-friendliness, Seismosoft hopes to somehow contribute, even if modestly, to the continuous search for higher mitigation of the risks that earthquakes pose to humankind.

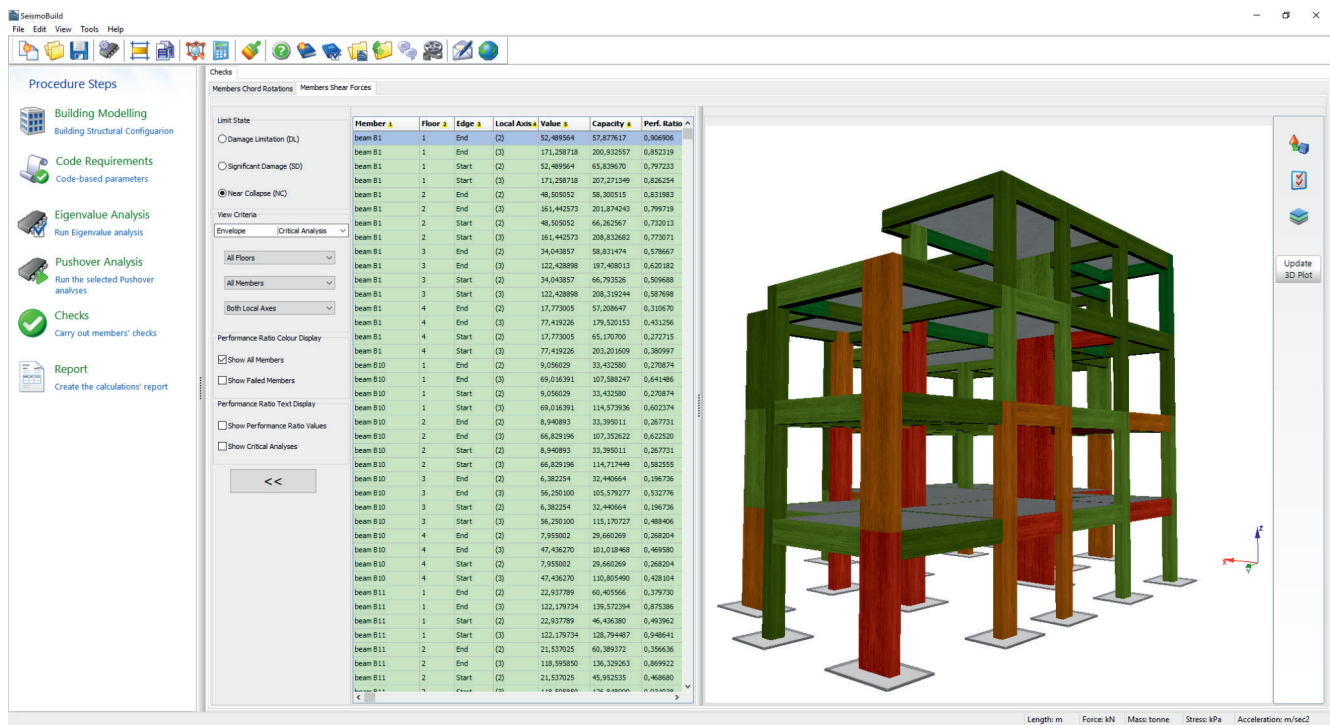
Seismosoft goal in having a place in earthquake engineering field has led in the development of ten programs, including SeismoStruct. A brief reference of the other nine programs and their capabilities is exhibited below:

SEISMOBUILD

SeismoBuild is an innovative Finite Element package wholly and exclusively dedicated to seismic assessment and strengthening of reinforced concrete buildings. The program is capable of fully carrying out the Code defined assessment methodologies from the structural modelling through to the required analyses and the corresponding member checks. Currently five Standards are supported (Eurocode 8 and the Eurocodes framework, the American Code for Seismic Evaluation and Retrofit of Existing Buildings ASCE 41, the Italian National Seismic Code NTC-18, the Greek Seismic interventions Code KANEPE and the Turkish Seismic Evaluation Building Code TBDY). Both metric and imperial units, as well as European and US reinforcing bars types are supported.

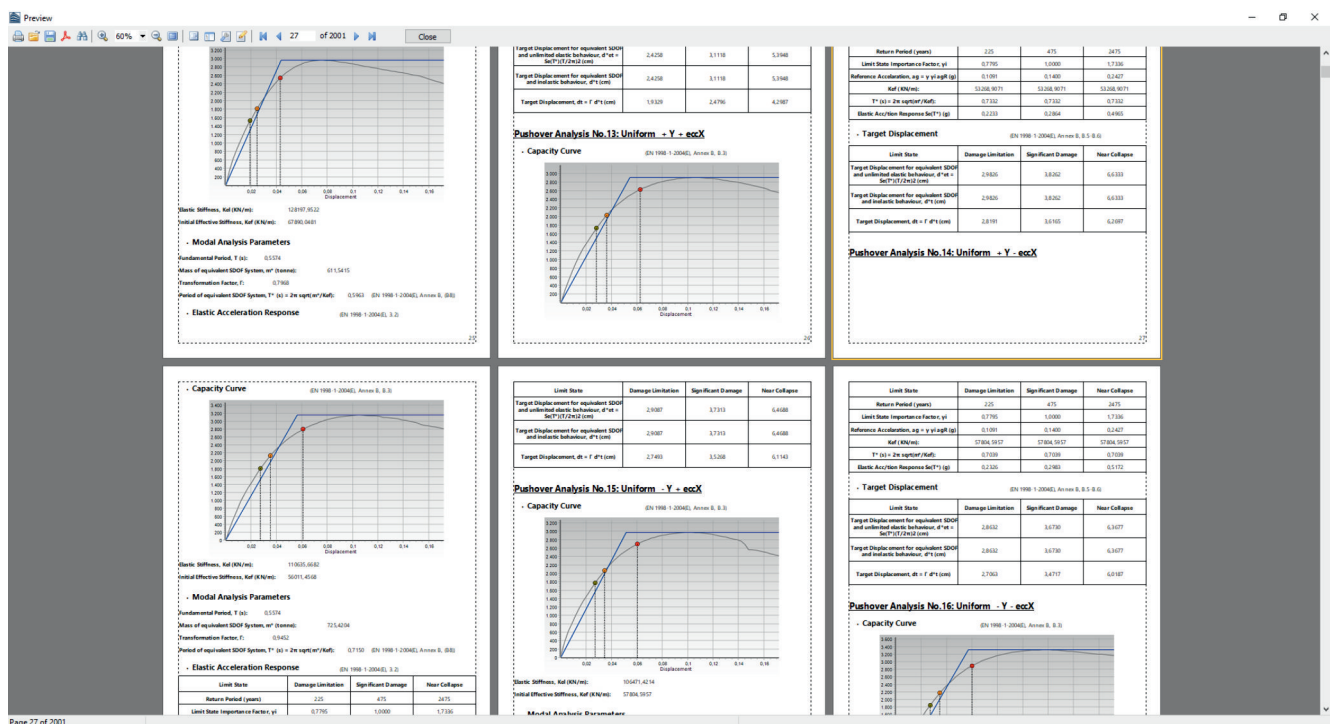
The rational and intuitive structure of the package, which are mainly attributed to the fact that it is the only software worldwide that is totally committed to seismic assessment, result in a very smooth learning curve even for engineers that are not familiar with the Finite Elements method. Further, the user-friendly, CAD-based, graphical user interface

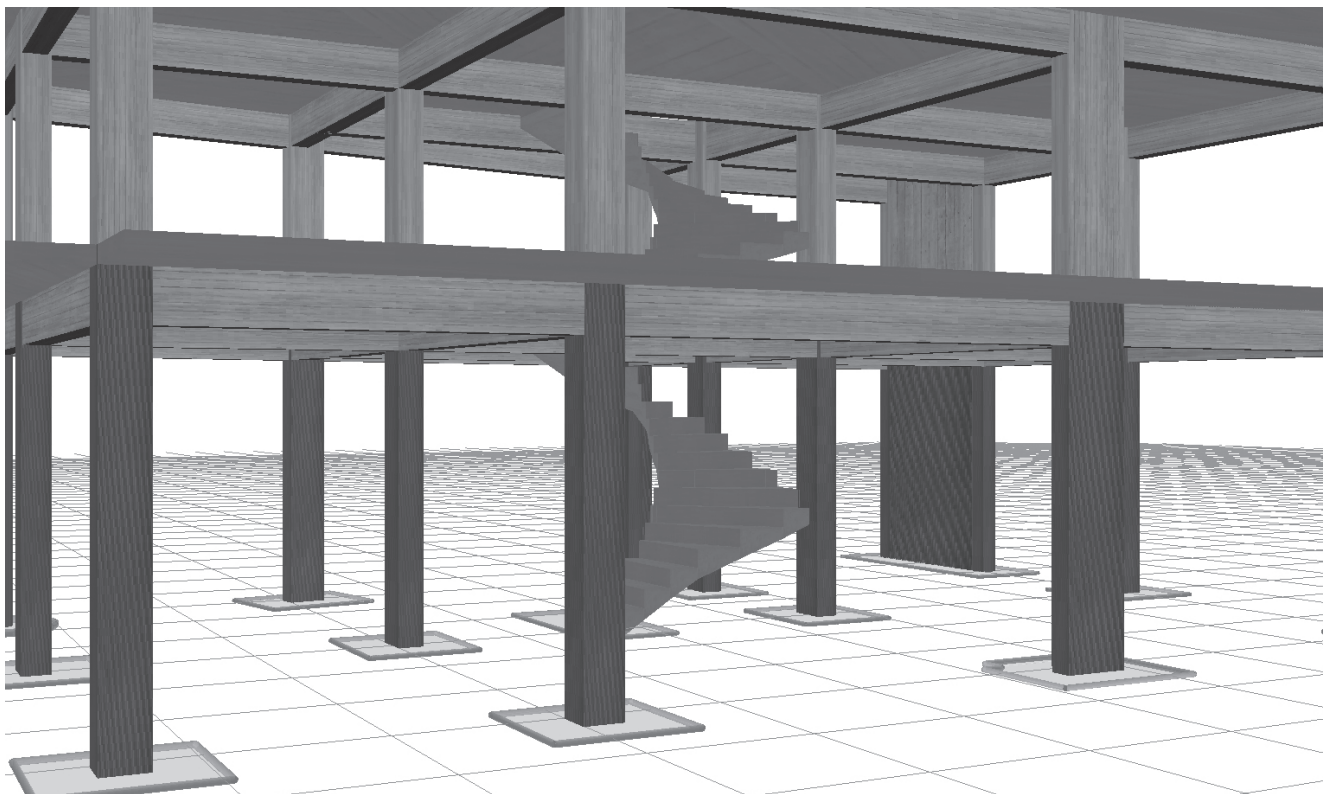




may increase the productivity significantly, to the point that the assessment of a multi-storey RC building can be completed, within a few minutes, including the creation of the report and the CAD drawings to be submitted to the client.

The structural model is introduced with the so-called Building Modeller, which constitutes a special CAD-based module that facilitates significantly the input. With the help of the program's predefined built-in sections, columns and beams of different shapes (rectangular, circular, L-shaped and T-shaped columns, beams and walls, as well as their 1, 2, 3 or 4-sided jacketed counterparts), any RC building configuration can be easily modelled. The reinforcement, the other modelling parameters (e.g. new or existing member, material strengths, lap splices, members' detailing, cover thickness) and the Code-defined settings (e.g. primary/secondary member, with/without detailing for earthquake resistance) may be easily introduced with a couple of mouse clicks. Slabs and stairs can be effectively introduced, whereas inclined slabs and beams are also supported. Furthermore, infill panels and the



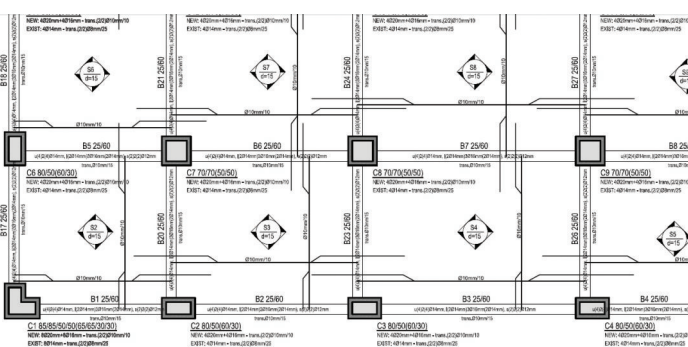


strengthening with steel braces of different configurations (X-braces, V-braces or diagonal braces) may also be defined. Finally, a large built-in library of FRP wrapping sheets is available for the strengthening of reinforced concrete members. It is important to note that the input provided by users is only related to either the structural details (geometry, materials, and reinforcement), or clearly defined Code-based parameters, such as the limit states, the seismic action, or the knowledge level. The entire structural model (including the material models parameters, the nodes location and the FE mesh, the elements geometry and offsets, the mass distribution and the loading patterns) is automatically created by the program without any intervention by the user.

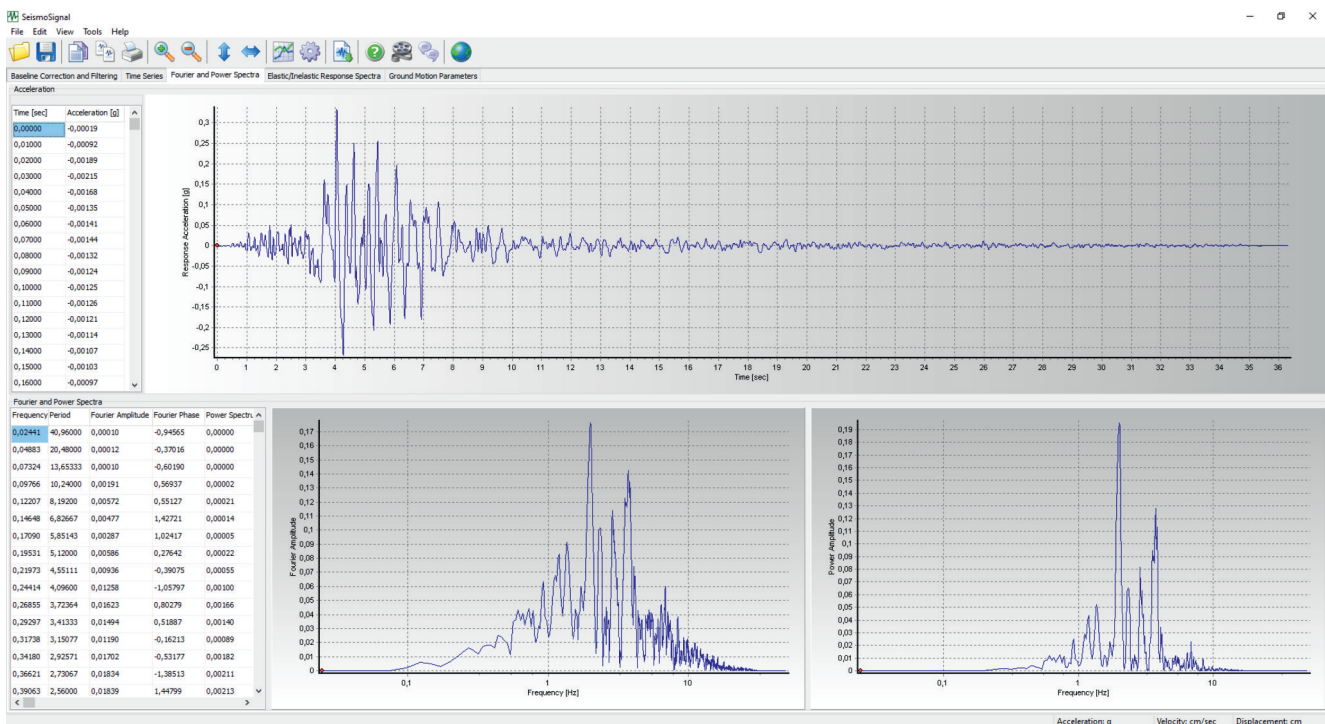
The Code requirements (limit states, knowledge levels, seismic input, types of checks to be performed, safety factors, etc.) according to the selected Standard, as well as the analysis parameters, can be easily introduced from a series of specially designed modules.

All the analysis types proposed by the Standards are supported: (i) linear static procedure LSP, (ii) linear dynamic procedure LDP, (iii) nonlinear static procedure NSP, (iv) nonlinear dynamic procedure NDP. The nonlinear analysis solver features both geometric nonlinearities and material inelasticity, and is based on the advanced solution algorithms of SeismoStruct. Carrying out the analyses, including the calculation of the target displacement in pushover analysis (NSP) and the creation and matching of the input accelerograms in dynamic analysis (NDP) is straightforward.

Similarly, the extraction of the results and the creation of the envelopes of the demand-to-capacity ratios for all the analyses is done automatically by the program. The results for all the selected checks can be viewed both in tabular format and graphically, where the members, for which the demand has exceeded the capacity, are displayed in colour. The checks available in SeismoBuild pertain to the members' chord rotations, bending moment and shear capacities, the inter-storey drift levels, as well as the checks for the beam-column joints.



Finally, the technical report of the analysis can be created on-the-fly in PDF, RTF or HTML file formats, whereas a variety of CAD drawing files of the building structural model (plan views, cross sections and reinforcement tables) may be quickly created and exported.



SEISMOSIGNAL

SeismoSignal constitutes an easy and efficient way to process strong-motion data, featuring a user-friendly visual interface and being capable of deriving a number of strong-motion parameters often required by engineer seismologists and earthquake engineers. SeismoSignal calculates the following quantities:

- ✓ Elastic and constant-ductility inelastic response spectra
- ✓ Fourier and Power spectra
- ✓ Arias (Ia) and Characteristic (Ic) Intensities
- ✓ Cumulative Absolute Velocity (CAV) and Specific Energy Density (SED)
- ✓ Root-Mean-Square (RMS) of acceleration, velocity and displacement
- ✓ Sustained Maximum Acceleration (SMA) and Velocity (SMV)
- ✓ Effective Design Acceleration (EDA)
- ✓ Acceleration (ASI) and Velocity (VSI) Spectrum Intensity
- ✓ Predominant (Tp) and mean (Tm) periods
- ✓ Husid and Energy Flux plots
- ✓ Bracketed, Uniform, Significant and Effective durations

SeismoSignal is able to read accelerograms defined in both single and multiple-values per line formats (the two most popular formats used by strong-motion databases), and can apply baseline correction and filtering of the unwanted frequency content, prior to time-integration of the signal to obtain velocity and displacement time-histories. Polynomials of up to the 3rd degree can be employed for baseline correction, whereas three different digital filter types are available (Butterworth, Chebyshev and Bessel), all of which are capable of carrying out highpass, lowpass, bandpass and bandstop filtering.

Finally, and due to its full integration with the Windows environment, SeismoSignal allows for numerical and graphical results to be copied to any Windows application (e.g. MS Excel, MS Word, etc.), noting that the plots characteristics can be fully customised from within the program itself.

SEISMOSIGNAL 3D

SeismoSignal3D is an extension of SeismoSignal that allows for the simultaneous processing of acceleration components in 2 or 3 dimensions, and the derivation of the orientation-independent spectra for the ground motion (RotD100, RotD50, RotD00, RotDXX), considering the two horizontal components of the records. The main features of the software are as follows:

- ✓ Automatic loading of files that are downloaded from the NGA West, the NGA East and the ESM strong motion databases and automatic import of the acceleration time-histories for all three components.

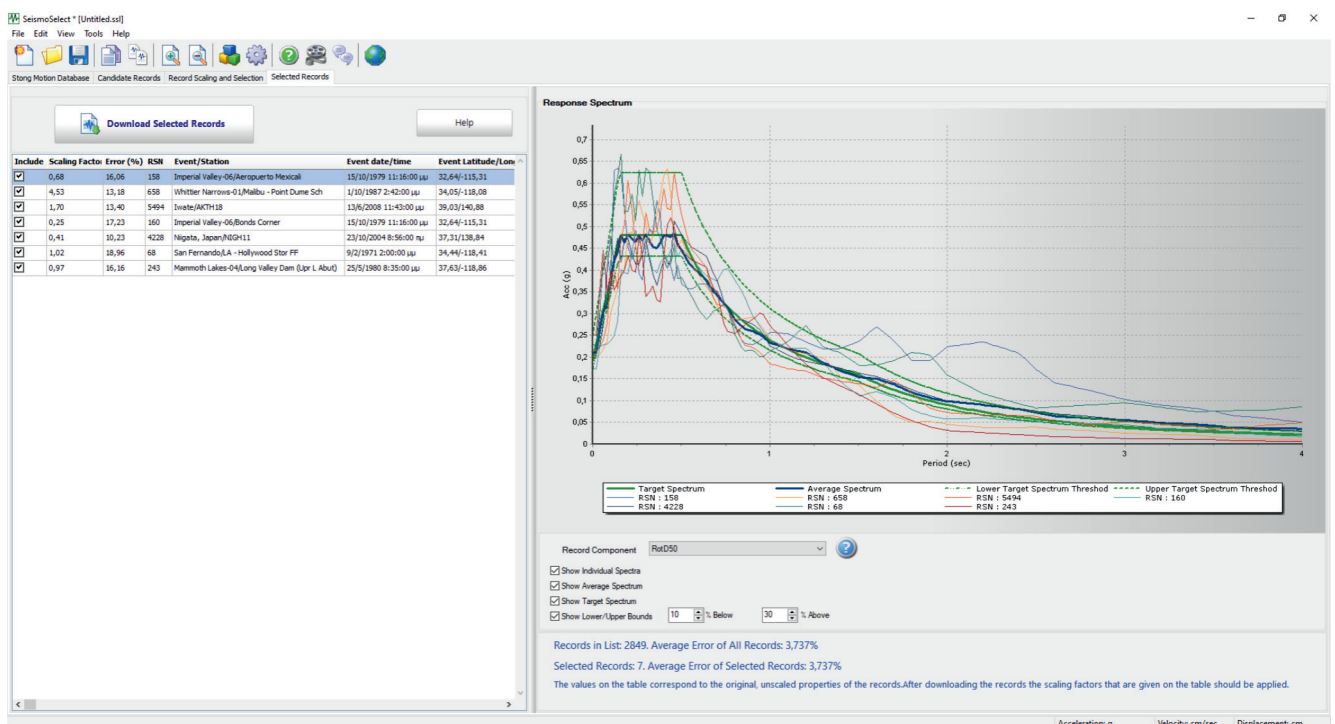
- ✓ Baseline correction and filtering of the time-histories
- ✓ Derivation of the displacement & velocity time-histories for each individual component
- ✓ Derivation of the orientation-independent spectra for the ground motion (RotD100, RotD50, RotD00, RotDXX), considering the two horizontal components of the record.
- ✓ Derivation of the Orbit Plots, considering the two horizontal components of the record.
- ✓ Calculation of several ground motion parameters, such as the PGA, PGV, PGD, the Arias Intensity and the Cumulative Absolute Velocity.

Data and results presented in tables or charts and easily exported to other Windows application (e.g. MS Word and MS Excel).

SEISMOSELECT

SeismoSelect is an easy and efficient way to search, select, scale and download ground motion data from three of the mostly used strong motion databases that are available on-line. Different criteria may be employed for the selection, and based on these parameters the software carries out searches for sets of compatible records and provides ways to easily download the selected records. The main characteristics of the program are described below:

- ✓ Support for (i) the NGA West 2 Ground Motion Database, (ii) the NGA East Ground Motion Database and (iii) the Engineering Strong Motion Database ESM.
- ✓ More than 30 code-based spectra can be automatically loaded as the target spectrum
- ✓ Different criteria may be employed as the parameters of interest, with which to carry out the searches. These include target spectral ordinates, different ground motion parameters (e.g. PGA, PGV, Arias or Housner Intensity), information regarding the event (e.g. magnitude, faulting style location, date) or the recording site (e.g. $V_{s,30}$, epicentral distance).
- ✓ Many options are available for the selection of the best set of records. These include the selection and sorting of all the records according to their errors from the given criteria or a more sophisticated greedy algorithm, in order to get a set of records for which the average spectrum has the best fit.
- ✓ The records may be scaled with direct scaling in order to achieve a better fit to the target spectrum.
- ✓ Several options are provided for shortlisting and optimizing the final selection.
- ✓ The final list of records may be downloaded, either automatically (if this is permitted by the strong-motion database), or manually with the record IDs that are provided by SeismoSelect.



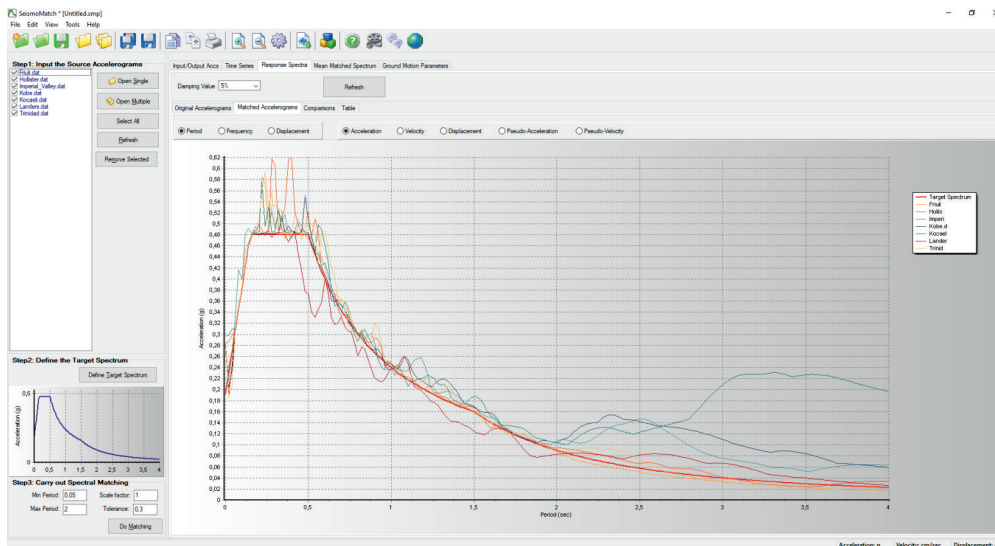
SEISMOMATCH

SeismoMatch is an application capable of adjusting earthquake accelerograms to match a specific target response spectrum, using the wavelets algorithm proposed by Abrahamson [1992] and Hancock et al. [2006]. Some of its features are:

- ✓ Simultaneous matching of a number of accelerograms, and then creation of a mean matched spectrum whose maximum misfit respects a pre-defined tolerance.
- ✓ Ability to combine some of the matched accelerograms, in order to obtain a combined mean spectrum that fulfils the user's requirements regarding maximum and mean misfit.
- ✓ Possibility of using this software, in combination with records selection tools and records appropriateness verification algorithms, to define adequate suites of records for nonlinear dynamic analysis of new or existing structures.
- ✓ Capability of reading single accelerograms defined in both single- or multiple-values per line formats (the two most popular formats used by strong-motion databases) or of reading a number of accelerograms at the same time (if they are defined in the single-value per line format).
- ✓ Creation of the target spectrum by following Eurocode 8 rules, by computing the spectrum of a specific accelerogram or by simply loading a user-defined spectrum.

The following strong-motion parameters are then computed for the matched accelerograms:

- ✓ Overdamped and constant-ductility inelastic response spectra
- ✓ Root-Mean-Square (RMS) of acceleration, velocity and displacement
- ✓ Arias (Ia) and Characteristic (Ic) Intensities
- ✓ Cumulative Absolute Velocity (CAV) and Specific Energy Density (SED)
- ✓ Acceleration (ASI) and Velocity (VSI) Spectrum Intensity
- ✓ Housner Intensity
- ✓ Sustained Maximum Acceleration (SMA) and Velocity (SMV)
- ✓ Effective Design Acceleration (EDA)
- ✓ Predominant Period (Tp)
- ✓ Significant duration



SEISMOARTIF

SeismoArtif is an application capable of generating artificial earthquake accelerograms matched to a specific target response spectrum using different calculation methods and varied assumptions. It is noted that the use of real accelerograms and spectrum matching techniques (see SeismoMatch), together with records selection tools, tends to be recommended for the derivation of suits of records for use in nonlinear dynamic analysis of structures. However, in those cases where access to real accelerograms is, for whatever reason, challenging or inappropriate, then a tool such as SeismoArtif will be of pertinence and usefulness.

Four calculation methods for the simulation of artificial ground motions are supported in SeismoArtif:

- ✓ Synthetic Accelerogram Generation & Adjustment
- ✓ Artificial Accelerogram Generation
- ✓ Artificial Accelerogram Generation & Adjustment
- ✓ Real Accelerogram Adjustment

The target spectrum can be created by following Eurocode 8 rules, by computing the spectrum of a specific accelerogram or by simply loading a user-defined spectrum. With both the target spectrum and the envelope type defined, an artificial accelerogram is generated and a comparison between its response spectrum and the target spectrum is given including the mean error in percentage, coefficient of variation, and PGA of the accelerogram.

SEISMOSPECT

SeismoSpect is a platform that allows to create a personal library of ground motion records and save them all in a single file making it easy to handle and share large numbers of records. The program can read accelerograms defined in both single or multiple values per line formats (the two most popular formats used by strong-motion databases). Before the integration for the derivation of the velocity and displacement time-histories, users are able to apply several digital filter types to the record, and to perform baseline-correction using up to third order polynomials.

Apart from the individual spectra for each record, the mean spectrum of the collection of accelerograms and the mean plus/minus one or two standard deviations spectra are also calculated. These spectra are then compared to a reference spectrum. Users can thus combine different records in order to individuate those whose average spectrum will meet a certain reference spectrum.

A module is also introduced to compute for all the loaded records a number of strong-motion parameters often required by engineer seismologists and earthquake engineers, such as:

- ✓ Maximum acceleration, velocity and displacement
- ✓ Effective design acceleration (EDA)
- ✓ Root-Mean-Square (RMS) of acceleration, velocity and displacement
- ✓ Arias (Ia) and Characteristic (Ic) Intensities
- ✓ Cumulative Absolute Velocity (CAV) and Specific Energy Density (SED)
- ✓ Acceleration (ASI) and Velocity (VSI) Spectrum Intensity
- ✓ Sustained Maximum Acceleration (SMA) and Velocity (SMV)
- ✓ Predominant Period (Tp)
- ✓ Significant duration

Finally, the program is fully integrated with the Windows environment and data and plots can be easily copied to other applications.

FRP DESIGNER

FRP Designer is a simple and efficient application for the analysis and design of the strengthening of reinforced concrete members with fibre-reinforced polymers (FRPs).

The input of the structural data for the existing member and for the FRP strengthening is done easily with a completely visual interface. A large library of the most common FRP fabrics and laminates available in the market has been included in the program, whereas user-defined FRP materials can also be defined by specifying their very basic characteristics (e.g. thickness, width, strength, stiffness and maximum elongation). The program is capable of automatically carrying out all the calculations for the determination of the member capacity with and without the retrofit interventions. Currently interventions in shear and bending have been included. The supported Standards are: (i) the US guidelines of ACI PRC-440.2-17, (ii) Eurocode 8, Part-3, (iii) the Italian guidelines CNR-DT 200 R1/2013 and (iv) the fib Bulletin 90. The results and the checks can be exported in a technical report in PDF, RTF or HTML file format.

SEISMORACK

SeismoRack is Finite Element package wholly and exclusively dedicated to analysis and assessment of structures with members with thin-walled, open, cross-sections. Consequently, the program is ideal for the modelling of steel storage pallet racks, as well as scaffolding structures, which are generally composed by uprights with mono-symmetric lipped channel cross-sections.

Any thin-walled open section configuration can be modelled, and different thicknesses may be assigned at the different parts of the section. The analysis can be performed with either 6 or 7 degrees of freedom per node. In the latter case, the displacements and the internal stresses, including warping displacements and bi-moment stresses, can be correctly estimated and the flexural-torsional and lateral-torsional buckling, derived by the coupling between flexure and torsion, may be accurately predicted. The modelling also 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. Finally, the flexibility of the supports at the edges of the rack elements may be modelled with linear or nonlinear springs.

The rational structure of the program, which is mainly attributed to the fact that it is the only software worldwide that is totally committed to racks and scaffoldings, results in a very smooth learning curve even for engineers that are not familiar with the Finite Elements method. Furthermore, the user-friendly graphical user interface increases the productivity significantly, to the point that the assessment of a large rack structure can be completed within a few minutes including the creation of the technical report.

Currently the European Standards for steel static storage systems and adjustable pallet racking systems are supported (EN 15512). The Code requirements (limit states, seismic input, types of checks to be performed, loading combination coefficients, member imperfections etc.), as well as the analysis parameters, can be easily introduced from a series of specially designed modules. The program is capable of running the following analyses specified by the Standard: linear static analysis, eigenvalue analysis, buckling analysis and nonlinear pushover analysis.

The extraction of the results and the creation of the envelopes of the demand-to-capacity ratios for all the analyses is done automatically by the program. The results for all the selected checks can be viewed both in tabular format and graphically, where the members, for which the demand has exceeded the capacity, are displayed in colour. The checks available in SeismoRack are the following: axial tension, axial compression, biaxial bending, shear, combined tension and bending, combined compression and bending, combined shear axial force and bending, lateral torsional buckling of beams, flexural buckling, flexural-torsional buckling, bending and axial compression without lateral-torsional buckling, bending and axial compression with lateral-torsional buckling, deflection of beams, vertical deflection and twist in beams.

The technical report of the analysis with all the checks can be created and exported on-the-fly in PDF, RTF or HTML file formats.

AWARDS

As mentioned above, the nonlinear analysis solver of SeismoStruct has been extensively used and verified by thousands of users for more than ten years.

The accuracy of the solver in nonlinear analysis of framed structures is well demonstrated by the successes in recent Blind Test Prediction Exercises, such as the Concrete Column Blind Prediction Contest 2010 (UCSD, San Diego, USA), the Blind Test Challenge of the 15th World Conference on Earthquake Engineering (LNEC, Lisbon, Portugal) and the Blind prediction Contest organized in 2011 by the Earthquake resistance and Disaster Prevention Branch of the Architectural Society of China, amongst others.

documentation available

Users are provided full documentation for SeismoStruct in different formats. The sources of information currently available are:

- ✓ The User Manual provided with the program installation
- ✓ The application's Help System provided with the program installation
- ✓ The Verification Report provided with the program installation
- ✓ The SeismoStruct models for all the examples presented in the Verification Report are provided with the program installation
- ✓ Video tutorials available in Seismosoft YouTube channel
- ✓ Seismosoft's Forum available at www.seismosoft.com/forum/
- ✓ Seismosoft's Support page available at www.seismosoft.com/support/. Amongst other resources, the page provides dozens of samples model for download.
- ✓ Seismosoft's Assessment and Strengthening Blog available at www.seismosoft.com/blog
- ✓ Seismosoft's pages in the social media (Facebook, Twitter and LinkedIn page)

references

- Abrahamson N.A. [1992] "Non-stationary spectral matching," *Seismological Research Letters*, Vol. 63, No. 1, p. 30.
- ACI 318-19 [2019] American Concrete Institute: Building Code Requirements for Structural Concrete (ACI 318-19) and Commentary.
- ACI 440.2R-08 [2008] American Concrete Institute: Guide for the Design and Construction of Externally Bonded FRP Systems for Strengthening Concrete Structures.
- 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., Pinho R. [2004b] "Development and Verification of a Displacement-based Adaptive Pushover Procedure," *Journal of Earthquake Engineering*, Vol. 8, No. 5, pp. 643-661.
- ASCE 41-17 [2017] American Society of Civil Engineers: Seismic Evaluation and Retrofit of Existing Buildings.
- Auricchio F., Sacco E. [1997] "A superelastic shape-memory-alloy beam," *Journal of Intelligent Materials and Structures*, Vol. 8, pp. 489-501.
- 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.
- 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.
- Chopra A.K. [1995] *Dynamics of Structures: Theory and Applications to Earthquake Engineering*, Prentice-Hall.
- 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.
- 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 *Geotechnique*.
- 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.
- Crisafulli F.J. [1997] *Seismic Behaviour of Reinforced Concrete Structures with Masonry Infills*, PhD Thesis, University of Canterbury, New Zealand.
- 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.
- EN 1998-1 (English) [2004]: 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.
- 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.
- 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. Charnpis, N.D. Lagaros and Y. Tsompanakis, A.A. Balkema Publishers – Taylor & Francis, The Netherlands.
- 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.
- 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.
- Hamburger R.O., Foutch D.A., Cornell C.A. [2000] "Performance basis of guidelines for evaluation, upgrade and design of moment-resisting steel frames," *Proceedings of the Twelfth World Conference on Earthquake Engineering*, Auckland, New Zealand, paper No. 2543.
- Han T.-S., Feenstra P.H., Billington S.L. [2003] "Simulation of Highly Ductile Fiber-Reinforced Cement-Based Composite Components Under Cyclic Loading," Vol. 100, No. 6, pp. 749-757.
- Hancock J., Watson-Lamprey J., Abrahamson N.A., Bommer J.J., Markatis A., McCoy E., Mendis R. [2006] "An improved method of matching response spectra of recorded earthquake ground motion using wavelets," *Journal of Earthquake Engineering*, Vol. 10 pp. 67-89.
- Hellesland J., Scordelis A. [1981] "Analysis of RC bridge columns under imposed deformations," *IABSE Colloquium*, Delft, pp. 545-559.
- Hughes T.J.R. [1987] *The Finite Element Method, Linear Static and Dynamic Finite Element Analysis*, Prentice-Hall.
- Hellesland J., Scordelis A. [1981] "Analysis of RC bridge columns under imposed deformations," *IABSE Colloquium*, Delft, pp. 545-559.
- 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.

KANEPE [2022]: Code of Structural Interventions, 3rd revision.

Kaldjian M.J. [1967] "Moment-curvature of beams as Ramberg-Osgood functions," Journal of Structural Division, ASCE, Vol. 93, No. ST5, pp. 53-65.

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.

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.

Liu, J. W. H. [1986]. "A compact row storage scheme for Cholesky factors using elimination trees." ACM Trans. Math. Software 12, 127-148 (1986).

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.

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.

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., 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.

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.

Mpampatsikos V., Nascimbene R., Petrin L. [2008] "A critical review of the R.C. frame existing building assessment procedure according to Eurocode 8 and Italian Seismic Code," Journal of Earthquake Engineering, Vol. 12, Issue SP1, pp. 52-58.

Nagashima T., Sugano S., Kimura H., Ichikawa A. [1992] "Monotonic axial compression tests on ultra high strength concrete tied columns," Proceedings of the Tenth World Conference on Earthquake Engineering, Madrid, Spain, pp. 2983-2988.

Neuenhofer A., Filippou F.C. [1997] "Evaluation of nonlinear frame finite-element models," Journal of Structural Engineering, Vol. 123, No. 7, pp. 958-966.

NTC 2018, D.M. Infrastrutture Trasporti 17 gennaio 2018 (G.U. 20 febbraio 2018 n. 8 - Suppl. Ord.). "Norme tecniche per le Costruzioni" (NTC18).

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.

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.

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.

Ramberg W., Osgood W.R. [1943] Description of Stress-Strain Curves by Three Parameters, National Advisory Committee on Aeronautics, Technical Note 902.

Rosenblueth, E. [1951] A basis for a Seismic Design, PhD Thesis, University of Illinois, Urbana, USA.

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.

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.

Spacone E., Ciampi V., Filippou F.C. [1996] "Mixed formulation of nonlinear beam finite element," Computers & Structures, Vol. 58, No. 1, pp. 71-83.

TBDY [2018]: Turkish Seismic Building Regulations.

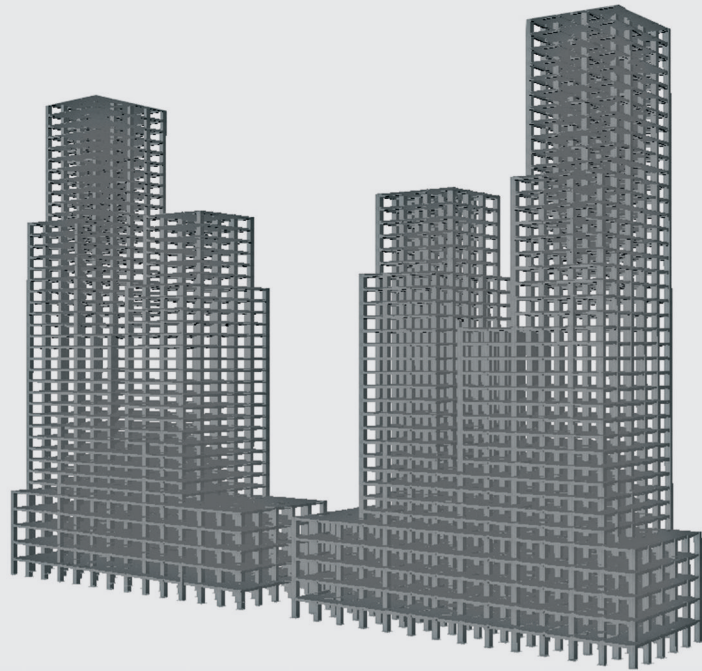
Vamvatsikos D., Cornell C.A. [2002] "Incremental dynamic analysis," Earthquake Engineering and Structural Dynamics, Vol. 31, No. 3, pp. 491-514.

Wen Y.K. [1976] "Method for Random Vibration of Hysteretic Systems," Journal of the Engineering Mechanics Division, ASCE, Vol. 102, Issue 2, Pg. 249-263.

Yassin M.H.M. [1994] Nonlinear analysis of prestressed concrete structures under monotonic and cyclic loads, PhD Thesis, University of California, Berkeley, USA.

Zona A. and Dall'Asta A. [2012] "Elastoplastic model for Steel buckling-restrained braces", Journal of Constructional Steel Research, Vol. 68, pp. 118-125.





Seismosoft in brief

Founded in 2002, Seismosoft provides the earthquake engineering community with access to powerful and state-of-the-art analytical tools, such as SeismoBuild, SeismoStruct, SeismoSignal, SeismoSelect, SeismoMatch, SeismoArtif, SeismoSpect and FRP Designer.

With more than 1000 software license requests per month, and users in thousands of international academic/research institutions and practicing companies in more than 110 countries worldwide, Seismosoft is now recognised as a leading enterprise in this field.

Seismosoft provides the full spectrum of earthquake engineering stakeholders, tools and methods that feature not only technical excellence but also efficiency and user-friendliness.

Ultimately, we hope to somehow contribute, even if modestly, to the continuous search for higher mitigation of the risks that earthquakes pose to humankind.



SEISMOSOFT
EARTHQUAKE ENGINEERING SOFTWARE SOLUTIONS

How to Contact

Up to date Seismosoft contact details are always available on our website at

www.seismosoft.com



<https://seismosoft.com/blog/>