NextFEM

Designer
User's manual

Version 2.4

Index

1. Introduction

NextFEM Designer is a user-friendly Finite Element Analysis program, which can be used alone or to be a pre- or post-processor for several widely used FEM programs (i.e. OOFEM, SAP2000, Midas GEN, OpenSees, ABAQUS/CalculiX, Zeus-NL, and others).

The most important features are:

- general pre-processor capabilities: 3D and 2D views, customizable colours, accelerated rendering with *DirectX* technology;
- modelling with the most common structural elements, such as beams, shells and solid elements;
- importing input files from DXF drawings, OpenSees scripts, Midas GEN, SAP2000, ABAQUS, OOFEM, Zeus-NL;
- importing results from SAP2000, OOFEM and OpenSees;
- exporting models to SAP2000, ABAQUS, Midas GEN, OpenSees, OOFEM;
- post-processing capabilities with deformed shapes with contour display, beam diagrams, stress and strain contour.

1.1. Editions of NextFEM Designer

NextFEM Designer is available in the following editions:

- standard version, that can be downloaded from nextfem.it, having all the features described in this manual;
- *Lite* version, available only on Windows Store® for Windows 10® users. Such edition does not accept license codes and the following features are not available:
    - analysis on cloud;
    - manuals for dedicated modules.

  In this last edition, not all the features covered in this manual are present.

1.2. Installing NextFEM Designer

NextFEM designer is designed to work with Windows 7 SP1 or above and it is available for 64 bit Windows versions. The prerequisites are:

- .NET Framework 4.7.1;
- VC++ 2019 redistributable package.

To manually associate the NXF files to NextFEM Designer, open it once as administrator (right click/*Run as administrator* …)

1.3. License Activation

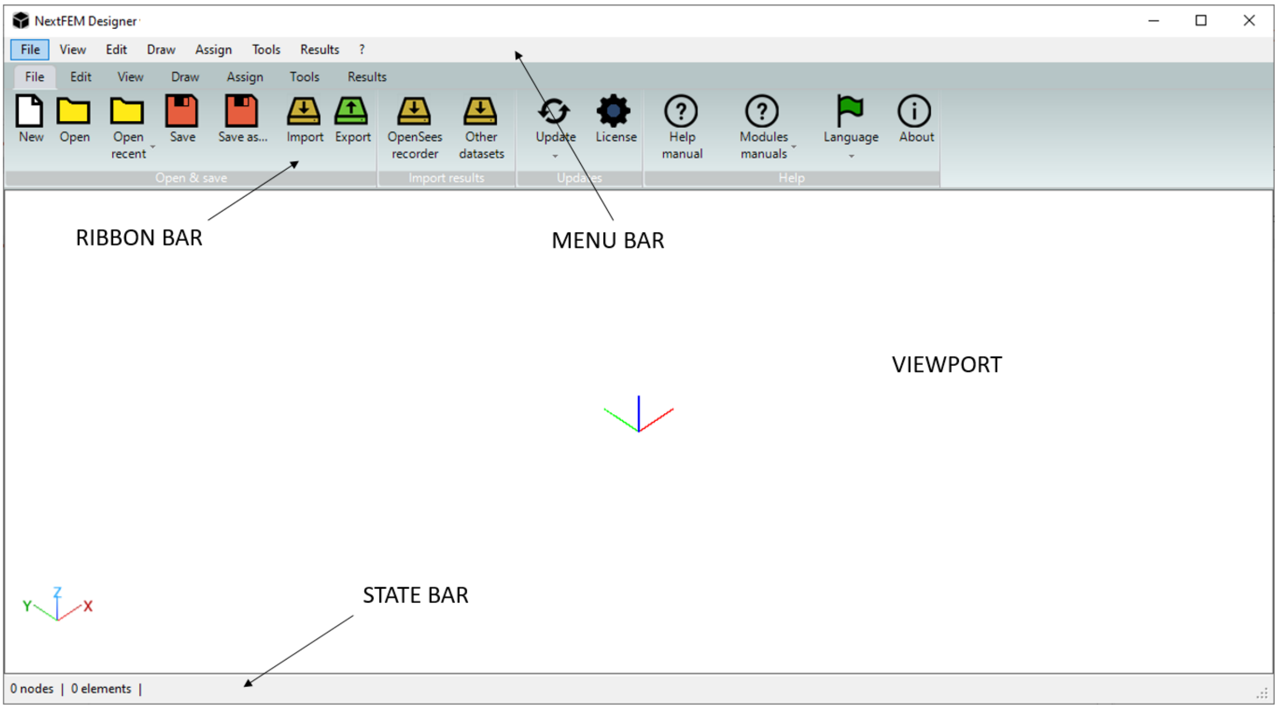
To activate the program you must be connected to the Internet at the first run. For the licensed installation, please refer to the command *?/License…*.

1.4. Manual and Support

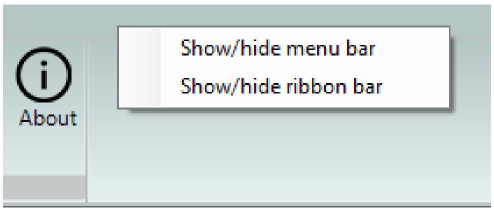
Along with NextFEM Designer is provided the user manual which describes the main controls and procedures to get started with the program. Moreover, online support is available. It is possible to ask questions or give suggestions in the dedicated part of the NextFEM forum (www.nextfem.it/it/nextfem-designer-support-forum/).

1.5. User interface

Display window looks like the figure below. It is possible to show the model (extruded or not), number of nodes or element, loads applied, global axes and other properties of the opened file. On the bottom left corner, number of nodes and elements of the model are shown.

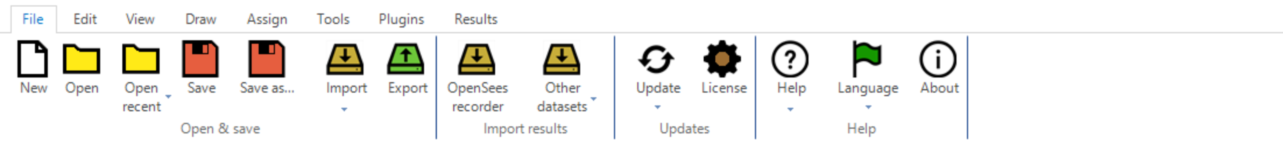


NextFEM Designer user interface presents a menu bar on the top which contains all the commands that can be used. Moreover, other toolbars, that can be anchored aside the viewport, which contain the major commands are available. A toolbar can be added in the main window by clicking the button 3 of the mouse (see User Controls chapter) and check it.

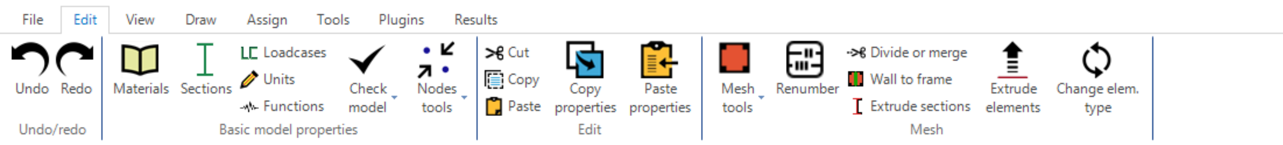


The following toolbars are available:

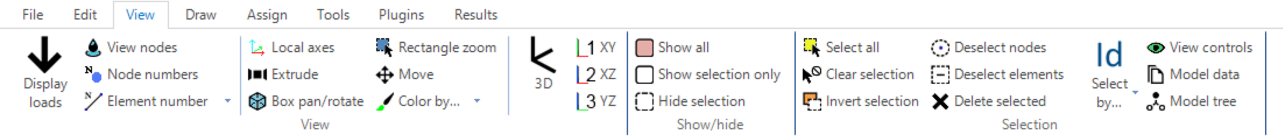
- *File*



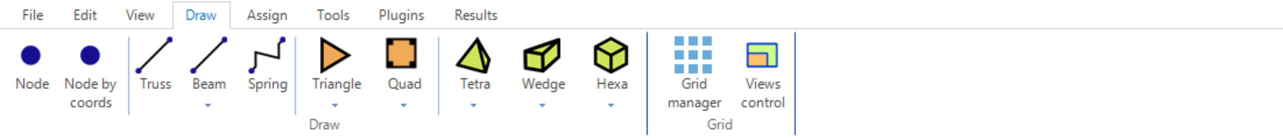
- *Edit*



- *View*

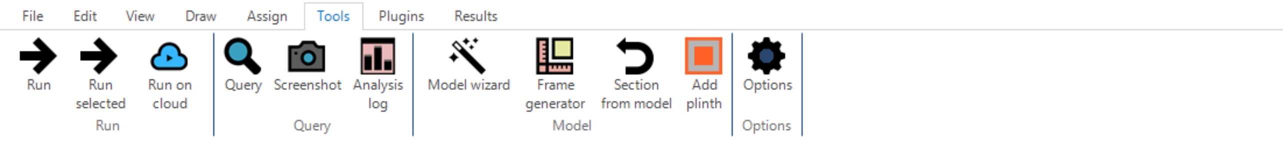


- *Draw*

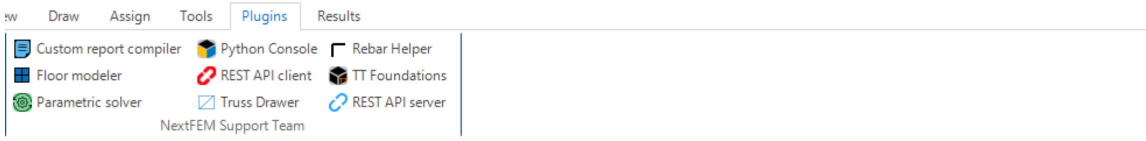


- *Assign*

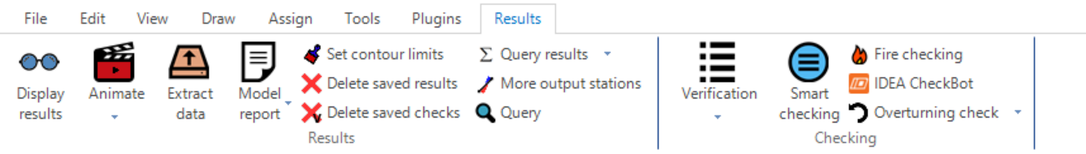
- *Tools (from v2.0)*



- *Plugins (from v2.0)*



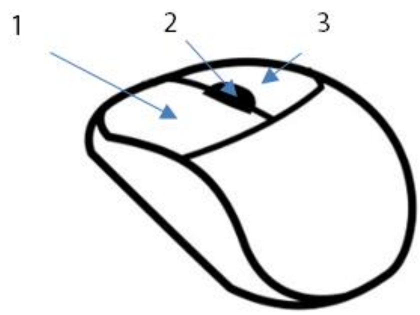
- *Results (from 2.4)*



2. Modelling with NextFEM Designer

2.1. User controls

Mouse usage



- ***1*** Button one permits to select objects in viewport by:
  - *Single object selection:* clicking on nodes or elements;
  - *Window selection:* by pressing and holding down the button, elements can be selected. Dragging the mouse <u>from left to right</u> only elements and nodes entirely contained in the window are selected. Dragging the mouse <u>from right to left</u> every element or nodes that intersect the window will be selected.
- ***2*** Button two (the mouse wheel) has a double function. Holding down it and moving the mouse, the model moves; scrolling it, zoom changes.
- ***3*** Holding down the button three, the model rotates.

Keyboard shortcuts

Keys	*Description*
Esc	clear current selection
Canc	delete selected nodes and elements
Alt+A	save screenshot
Alt+B	check linear element mesh
Alt+C	assign random colors to sections and paint the model
Alt+E	close the program
Alt+F4	close the program
Alt+G	open Groups mask
Alt+H	open Select by name / number mask
Alt+I	invert selection
Alt+J	show Select by property mask
Alt+M	show/hide menu bar

Alt+R	Run the model
Alt+Shift+R	Select cases to run and launch them
Alt+Shift+L	Show/hide local axes
Alt+Shift+K	modify the model without losing results
Alt+Shift+M	Import model and results from Midas programs via API
Alt+Shift+S	merge selected beam by selected nodes
Alt+Shift+U	merge meshed beams and imported results
Alt+S	copy screenshot
Alt+V	open views control
Alt+0	set view in 3D
Alt+1	set view in XY plane
Alt+2	set view in XZ plane
Alt+3	set view in YZ plane
Ctrl+0	Open Smart checking window
Ctrl+1	Steel checking as per Eurocode 3
Ctrl+2	Steel checking for scaffolds as per EN 12811-1
Ctrl+3	Aluminium alloy checking as per Eurocode 9
Ctrl+4	Aluminium alloy checking for scaffolds as per EN 12811-1
Ctrl+5	Timber checking as per Eurocode 5
Ctrl+6	Concrete checking as per Eurocode 2
Ctrl+7	Masonry checking as per Eurocode 6
Ctrl+A	select the entire model
Ctrl+C	Copy selected objects
Ctrl+D	deselect all nodes
Ctrl+F2	hide selection
Ctrl+E	export model
Ctrl+F	Fire checking
Ctrl+G	open Verifications window
Ctrl+I	import model
Ctrl+K	show loads
Ctrl+L	set contour limits
Ctrl+N	new model
Ctrl+O	open a model
Ctrl+Q	open Query dialog

Ctrl+R	show results
Ctrl+S	save the model
Ctrl+Shift+S	same model as
Ctrl+U	update program (including minor updates)
Ctrl+V	Paste selected objects
Ctrl+W	integrates the selected section cut (only when results are displayed)
Ctrl+X	Cut selected objects
Ctrl+Y	redo operations in the model
Ctrl+Z	undo operations in the model
Ctrl+"+"	auto-dimension rebar in concrete members
Shift+MouseR	open context menu
F1	open the user's manual
F2	show selection only
F3	show view options
F4	show the entire model
F5	show/hide nodes
F6	activate/disable extruded view
F7	show node numbers
F8	show element numbers
F9	show nodal masses
F10	show values in beam diagrams
F11	show values in displayed loads
F12	show constraints in the model

Command line options

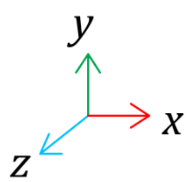
The following command line options are available:

- (-i *filename.ext*) allows to import a file specifying *filename*;
- (-c nodes) checks for free nodes in the loaded model;
- (-c elems) checks the counter-clockwise element connectivity in the loaded model;
- (-e *filename.ext*) allows to export the open model specifying *filename*, according to the specified extension *ext*.
  Available formats are:
    - ".s2K": SAP2000 text file;
    - ".mgt": Midas GEN text file;
    - ".dxf": DXF drawing file;
    - ".tcl": OpenSees script;
    - ".inp": ABAQUS/CalculiX input deck;
    - ".in": OOFEM input deck;
    - ".dat": ADAPTIC / Zeus NL input deck;
    - ".txt": Straus7/Strand7, versions higher than 2.2;
    - ".ifc" and ".ifcxml": IFC4 file format for BIM;
    - ".xml": SeismoStruct and WinStrand model.

- (-d lc data item dir) : extract data from the result file loaded and save selected output in CSV format; the following flags are currently supported:
    - *lc*: Loadcase name;
    - *data*: data type to be extracted: “react”, “disp” or “solls” for reactions, displacements or beam forces, respectively;
    - *item*: number of node/element or "all" for the sum for all nodes/elements. Optional, “all” is default;
    - *dir*: direction 1 2 3 4 5 6, according to global conventions (for nodes) or local ones (elements). Optional.
- (-r) : run the loaded/imported model
- (-s [*filename.ext*]) : save the loaded/imported model
- (-p) : print the model report in PDF format
- (-x) : closes the program after the batch operations as per any of the previous commands. The flag “-d” does not need this because it exits anyway.

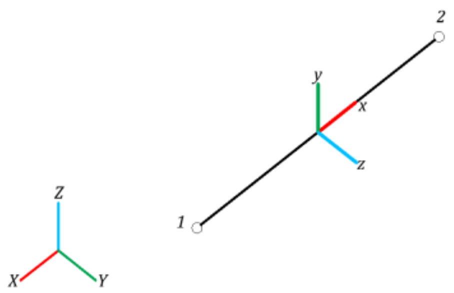
2.2. Local axis conventions

The following conventions are adopted for elements. Keep in mind that all the output forces/stresses/strains such as beam diagrams or area stresses are plotted with the following conventions along the local element axes.

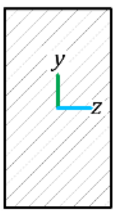


Beams

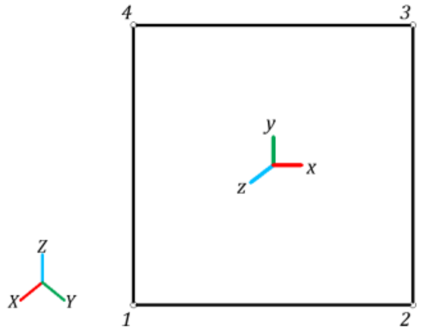
The second axis is always vertical and coincident with global Z.



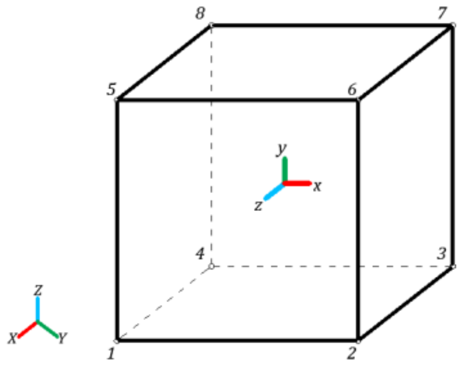
Sections



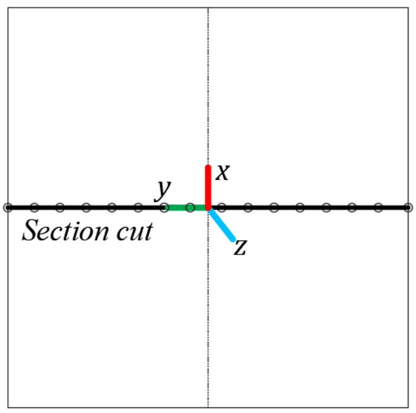
Planar elements



Solid elements



Section cut



2.3. File menu

New: Opens an empty model

Open: Opens an existing model

Save: Saves the model currently opened

Save as: Saves the model in *.nxf or *.xml formats.

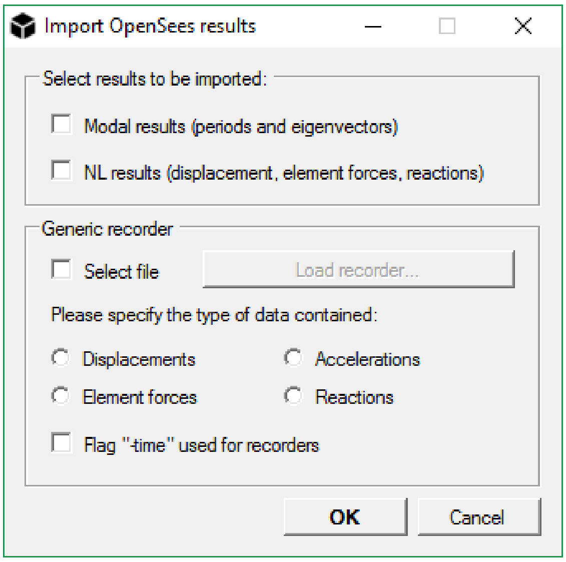
- *.nxf is a proprietary file format;
- *.xml saves the model in plain XML-formatted text.

Close: Closes the model currently opened.

Import: Imports model from other formats. See chapter 3 for supported formats.

Merge: Imports model from other formats merging it with the one in use.

Import OpenSees results: Imports OpenSees recorders



By the *box Select results to be imported* it is possible to import all the output files generated from the procedure007.tcl routines, that are included in the program. All the file with extension *.out* must be in the same folder of the already imported *TCL* model.

By the box *Generic recorder* it is possible to import a generic result by specifying its type. If the flag "-time" is used in the recorder, select the proper checkbox. The recorder file must be in the same folder of the already imported *TCL* model.

Import additional datasets: Imports additional datasets defined by user. This command is particularly suitable for importing data from external programs (i.e. OpenSees custom recorder). From the dropdown menu it is possible to import:

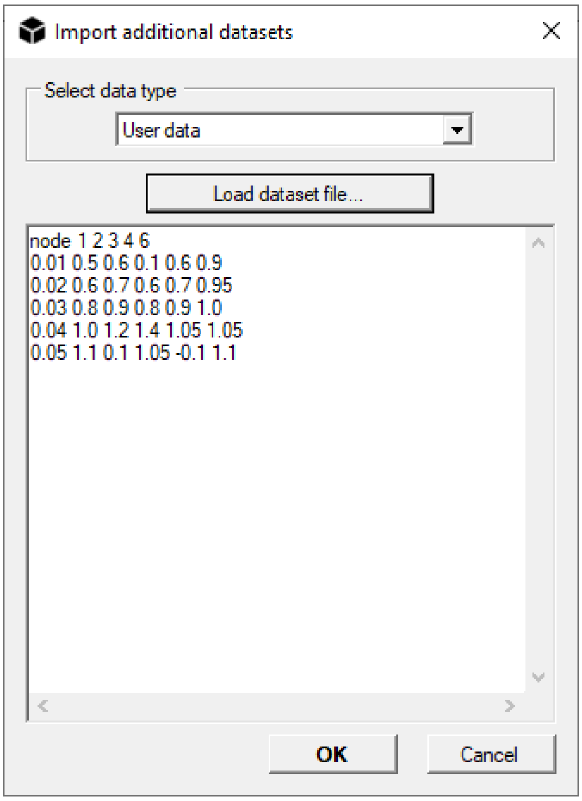
- *User data*: the format for such data is:
  node (node numbers series, separated by spaces, tabs or semicolon)
  time (series of data related to nodes, in the same order of the node numbers series in the first line)

  or

  element (element numbers series, separated by spaces, tabs or semicolon)
  time (series of data related to nodes, in the same order of the element numbers series in the first line)

  Such data will be imported in a new loadcase with the name of the source file, or with a temporary name if data are pasted in the textbox. All data can be saved in the NXF file and will be displayed in *Node data* or *Element data* results views.

- *Midas GEN results*: it allows the import of results tables from Midas GEN® (command Results/Result tables inside GEN). Tables can be pasted directly in the textbox. Supported tables are: Beam Forces, Wall Forces, Node Displacements.

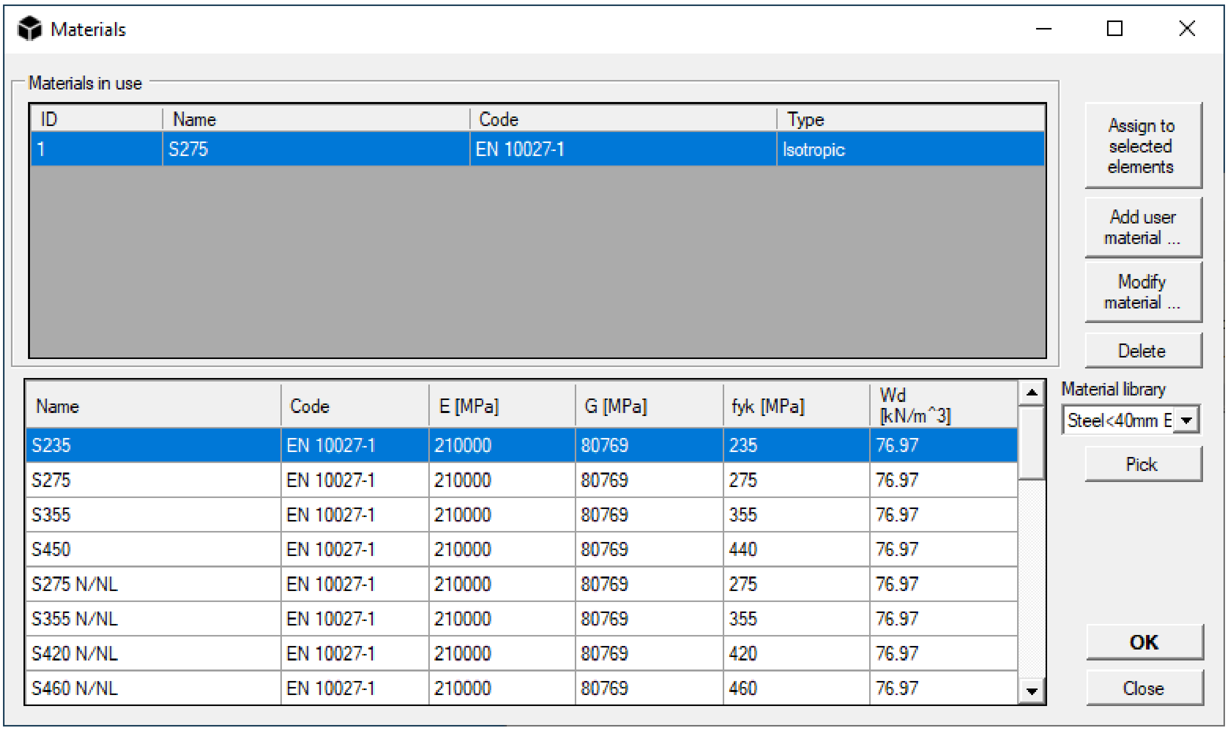


Export: Exports the model in many formats.

Exit: Closes the program.

2.4. Edit menu

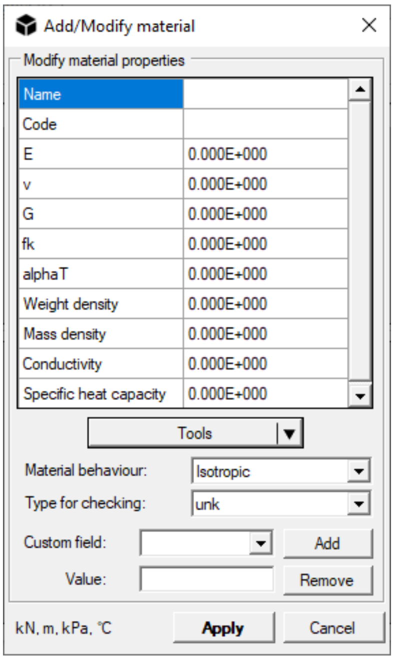
Material: To define materials properties:



- To add a new predefined material, click on the *Material library* combo box to show the available materials. Then select a material from the lower box and click on *Pick* to choose one of them;
- To add a new customized material, click the *Add user material* button

The following quantities are required:

- Name of the material
- (optional) reference code
- the Young's modulus *E*
- the Poisson's ratio *n*
- the shear modulus *G* (can be calculated automatically by clicking on *Shear modulus from E and G*)
- the characteristic strength *fk* (for elastic material behaviour)
- thermal expansion coefficient *alphaT*
- weight density *Wd*
- Mass density *Md* (can be calculated automatically by clicking on *Mass density from weight density*)



Click the *Assign to selected elements* to assign the selected material properties to the elements.

Click the button *Cancel* to exit this form.

Once the drop-down menu Tools has been selected, it is possible to choose between the following commands:

- With the command *Shear modulus from E and n* it is possible to get automatically the shear modulus G, computed through the following equation:

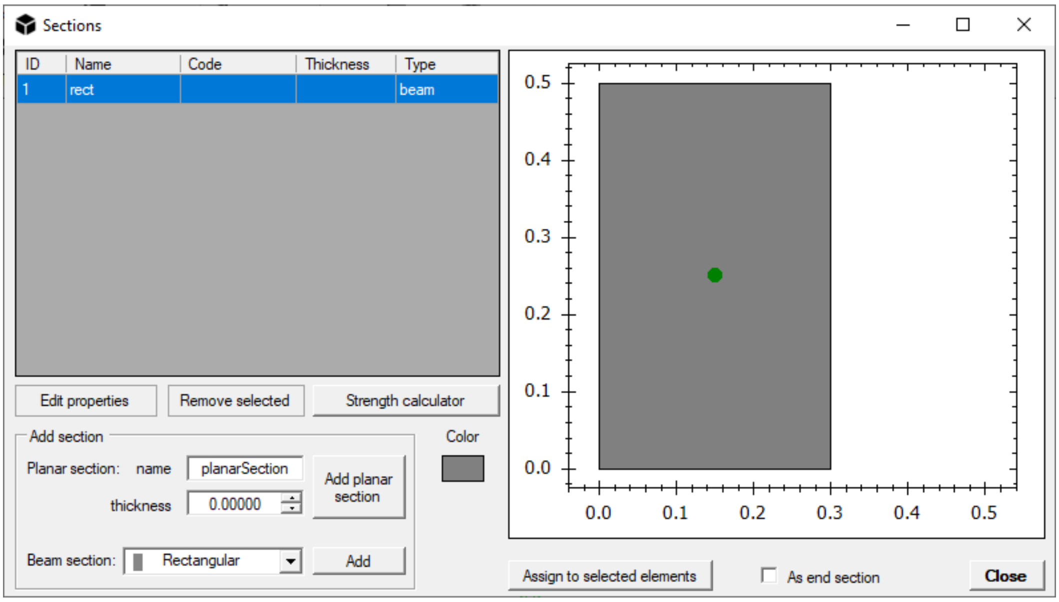
$$G = \frac{E}{2(1+\nu)}$$

- With the command *Mass density from weight density* it is possible to obtain automatically the mass density by dividing the weight density by the gravity acceleration, with is considered with consistent units.
- The button *Heat transfer settings* permits to specify the laws of variation of density, conductivity and specific heat on the base of temperature.
- With the *Set stiffness factor* command it is possible to set the stiffness factor to be considered for the current material.

To add a new value in table for material checking, use the button *Add* once the textboxes *Custom field* and *Value* have been compiled.

WARNING: it is strongly advised to insert units consistent with the initial choices (see ***Edit > Set units***).

Sections: to add Planar or Beam sections.



- To add a planar section, specify the name and the thickness in the *Planar section* boxes and then press *Add planar section*;
- To add a beam section, click the *Beam section* drop-down menu and choose the type among *Rectangular, C-shape, Circular, Pipe, Box, By point* options.

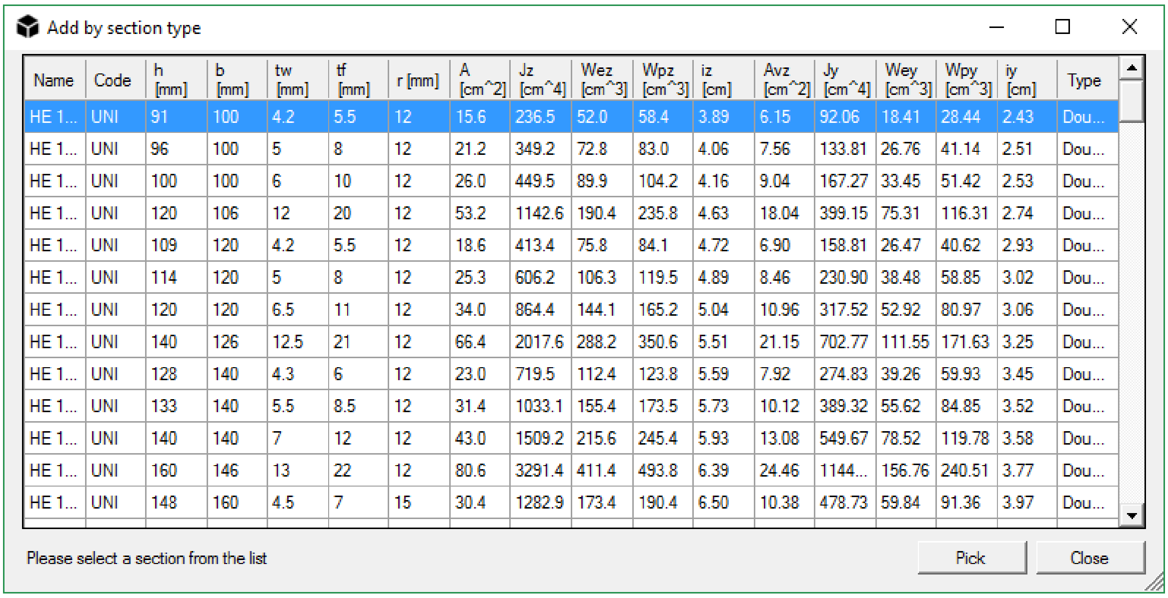
Click the *Assign to selected elements* button to assign the selected section to the selected elements.

The built-in beam sections available are:

Rectangular
Circular
C shape
T shape
I shape
Pipe
Box
L shape
By points
Double L
Double C

Further section shapes are available: Omega and cold-formed non-thin-walled C shapes, that can be set from inside the section properties.

Other custom libraries can be present in this list. For each of them, the following mask will be displayed to allow selection. Several libraries for steel sections are included in the program.

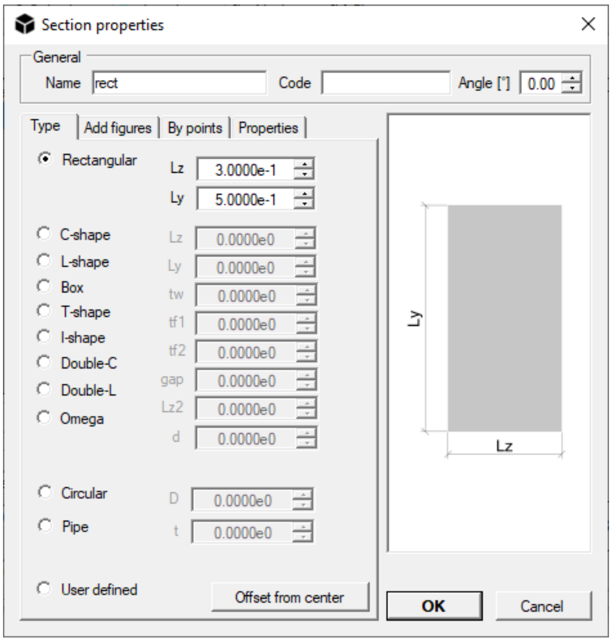


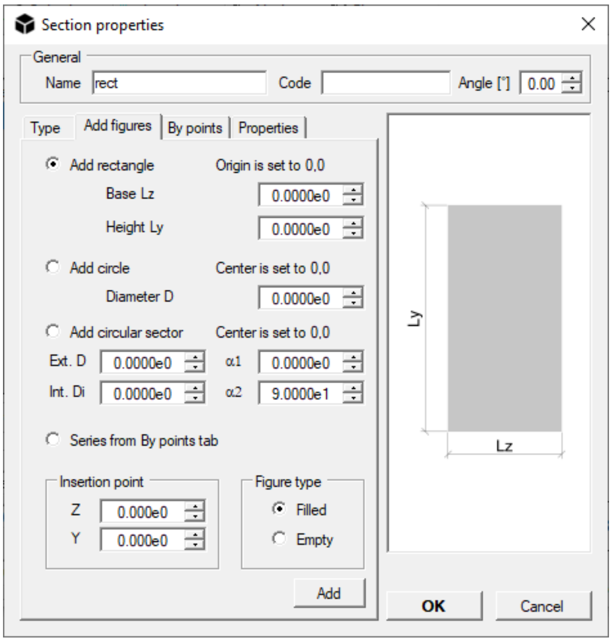
Add by section type

Name	Code	h [mm]	b [mm]	tw [mm]	tf [mm]	r [mm]	A [cm^2]	Jz [cm^4]	Wez [cm^3]	Wpz [cm^3]	iz [cm]	Avz [cm^2]	Jy [cm^4]	Wey [cm^3]	Wpy [cm^3]	iy [cm]	Type
HE 1...	UNI	91	100	4.2	5.5	12	15.6	236.5	52.0	58.4	3.89	6.15	92.06	18.41	28.44	2.43	Dou...
HE 1...	UNI	96	100	5	8	12	21.2	349.2	72.8	83.0	4.06	7.56	133.81	26.76	41.14	2.51	Dou...
HE 1...	UNI	100	100	6	10	12	26.0	449.5	89.9	104.2	4.16	9.04	167.27	33.45	51.42	2.53	Dou...
HE 1...	UNI	120	106	12	20	12	53.2	1142.6	190.4	235.8	4.63	18.04	399.15	75.31	116.31	2.74	Dou...
HE 1...	UNI	109	120	4.2	5.5	12	18.6	413.4	75.8	84.1	4.72	6.90	158.81	26.47	40.62	2.93	Dou...
HE 1...	UNI	114	120	5	8	12	25.3	606.2	106.3	119.5	4.89	8.46	230.90	38.48	58.85	3.02	Dou...
HE 1...	UNI	120	120	6.5	11	12	34.0	864.4	144.1	165.2	5.04	10.96	317.52	52.92	80.97	3.06	Dou...
HE 1...	UNI	140	126	12.5	21	12	66.4	2017.6	288.2	350.6	5.51	21.15	702.77	111.55	171.63	3.25	Dou...
HE 1...	UNI	128	140	4.3	6	12	23.0	719.5	112.4	123.8	5.59	7.92	274.83	39.26	59.93	3.45	Dou...
HE 1...	UNI	133	140	5.5	8.5	12	31.4	1033.1	155.4	173.5	5.73	10.12	389.32	55.62	84.85	3.52	Dou...
HE 1...	UNI	140	140	7	12	12	43.0	1509.2	215.6	245.4	5.93	13.08	549.67	78.52	119.78	3.58	Dou...
HE 1...	UNI	160	146	13	22	12	80.6	3291.4	411.4	493.8	6.39	24.46	1144...	156.76	240.51	3.77	Dou...
HE 1...	UNI	148	160	4.5	7	15	30.4	1282.9	173.4	190.4	6.50	10.38	478.73	59.84	91.36	3.97	Dou...

Please select a section from the list — Pick — Close

To see the properties (area and moments of inertia) of the inserted sections select a section and press the *Edit properties* button. This mask allows also to import a section from DXF file and define a section by points or by composing single figures.





⚠ WARNING: it is strongly advised to insert units consistent with the initial choices (see ***Edit > Set units***).

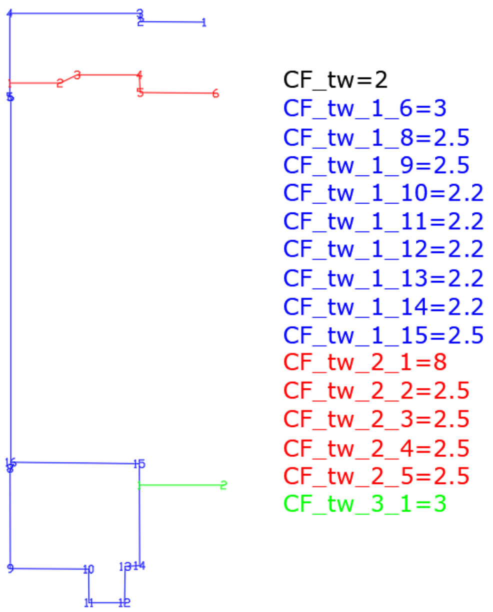
In the *Add figures* and *By Points* masks, it is possible to edit the component shapes of the section (for both voids and solids) or import a section from DXF with the *Import DXF* command. The contours of each figure must be polylines, defined clockwise for solids and anticlockwise for voids.

⚠ WARNING: the insertion of figures in sections does not take place in absolute co-ordinates, as the centre of gravity is recalculated for each change.

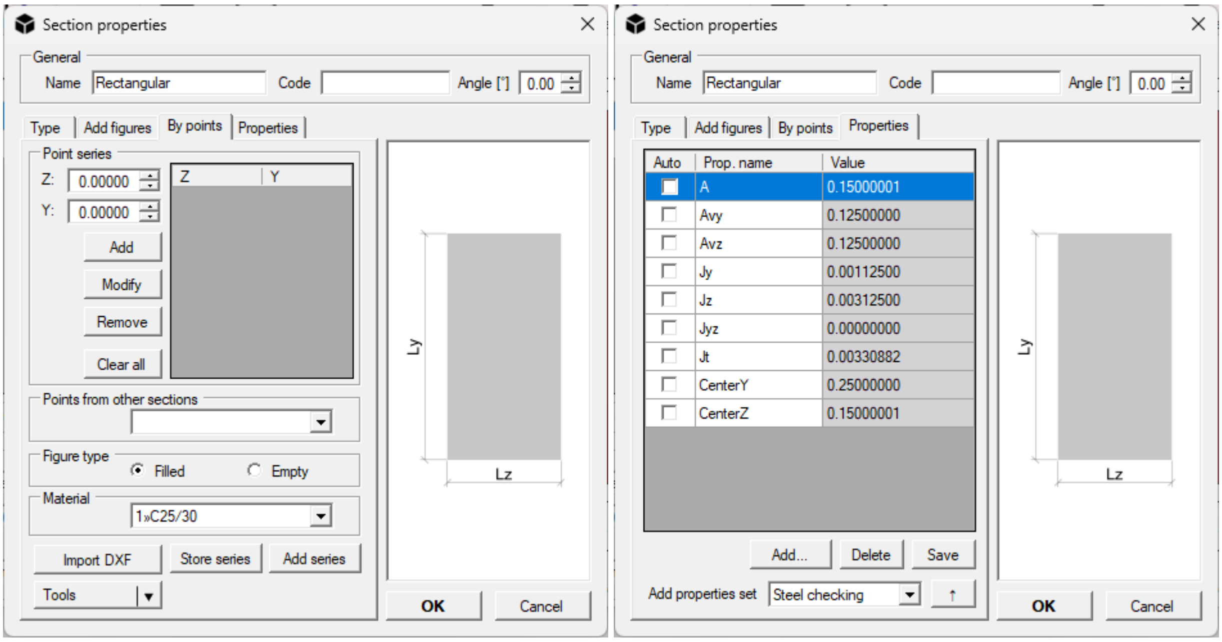
In the *Properties* mask it is possible to specify the design properties of the section to be used during checking. For sections without a defined type (e.g. point sections, customised), a class is not automatically assigned to the section during the verification for steel or aluminium material. For this, the user is asked to specify the class of the section via the Add Property drop-down menu, which presents the following options:

- *Steel verification* adds the following data for verification: SectClass (section class), alphaLT (coeff. flexural-torsional buckling), alphay (coeff. buckling around y-y), alphaz (coeff. buckling around z-z), Jw (warping constant)
- *Aluminium verification* adds the following data for verification: SectClass (section class), Jw (warping constant)
- *Composite column* adds the following data for analysis: Ec_factor (factor for the elastic modulus in the calculation of the homogenisation coeff., default at 0.5) and Reduction_factor (default at 0.9, inertia reduction factor)
- *Composite beam* adds the following data for analysis: Factor_Mpos (factor for positive moment inertia) and Factor_Mneg (factor for negative moment inertia)
- *Cold-formed section* adds the following data for the verification of thin sections, which is not performed without these flags: CF_tw (section thickness) and CF_rc (radius of curvature between segments)
- *Fibre section* adds the following data for the analysis in OpenSees: fibersX (number of fibres in the z-direction of the section) and fibersY (number of fibres in the y-direction of the section) to specify the fibre division for each direction
- *Skip verification* adds the flag skip = 1 to skip the verification of the section
- *From text* allows the loading of a set of user-defined flags, which are reported as variables in the verifications. The file must contain one variable per line in the format *nameVar=value*. This is particularly useful for defining different thicknesses in thin sections, for example for a model in mm and a section consisting of 3 figures (which in the case of a thin section are series of segments):

```
CF_tw=2
CF_tw_1_6=3
CF_tw_1_8=2.5
CF_tw_1_9=2.5
CF_tw_1_10=2.2
CF_tw_1_11=2.2
CF_tw_1_12=2.2
CF_tw_1_13=2.2
CF_tw_1_14=2.2
CF_tw_1_15=2.5
CF_tw_2_1=8
CF_tw_2_2=2.5
CF_tw_2_3=2.5
CF_tw_2_4=2.5
CF_tw_2_5=2.5
CF_tw_3_1=3
CF_rc=2
```



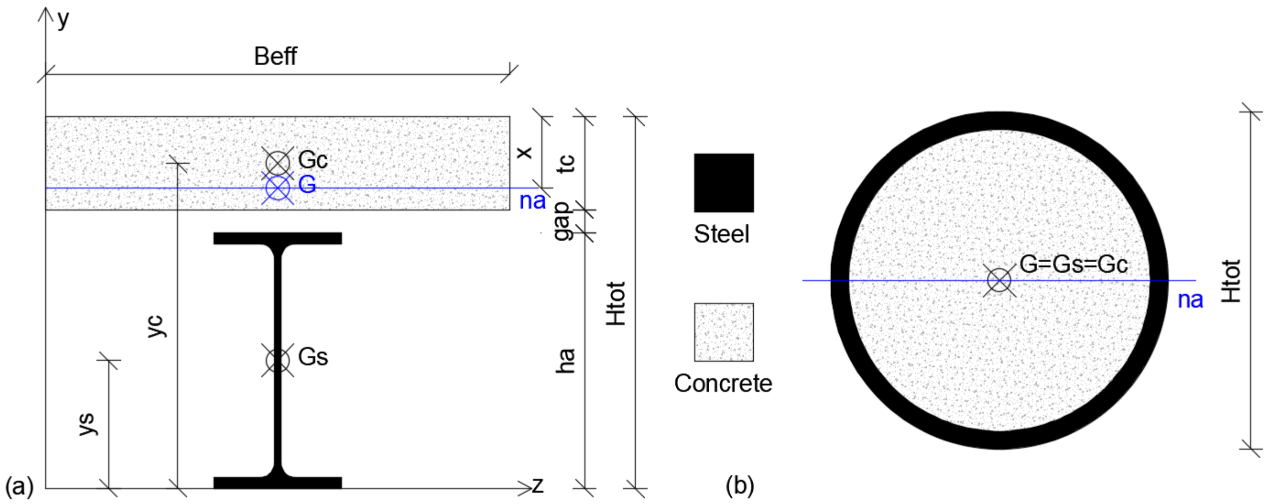
⚠ WARNING: by changing the sectional properties, the section will be converted in “Custom” type. Such operation is not reversible.



Composite sections

The support for composite sections includes:

- Beams with slab like in figure (a) and
- Columns with coincident centres of gravity (b).



A composite section is automatically detected when:

- Two or more filled figures, one of them associated to a steel material and one to a concrete material;
- Properties of the composite section are defined inside the mask *Section properties / Properties* in order to distinguish a beam or a column section.

⚠ WARNING: The effective width of the section *Beff* must be evaluate by the user as per current code of practice.

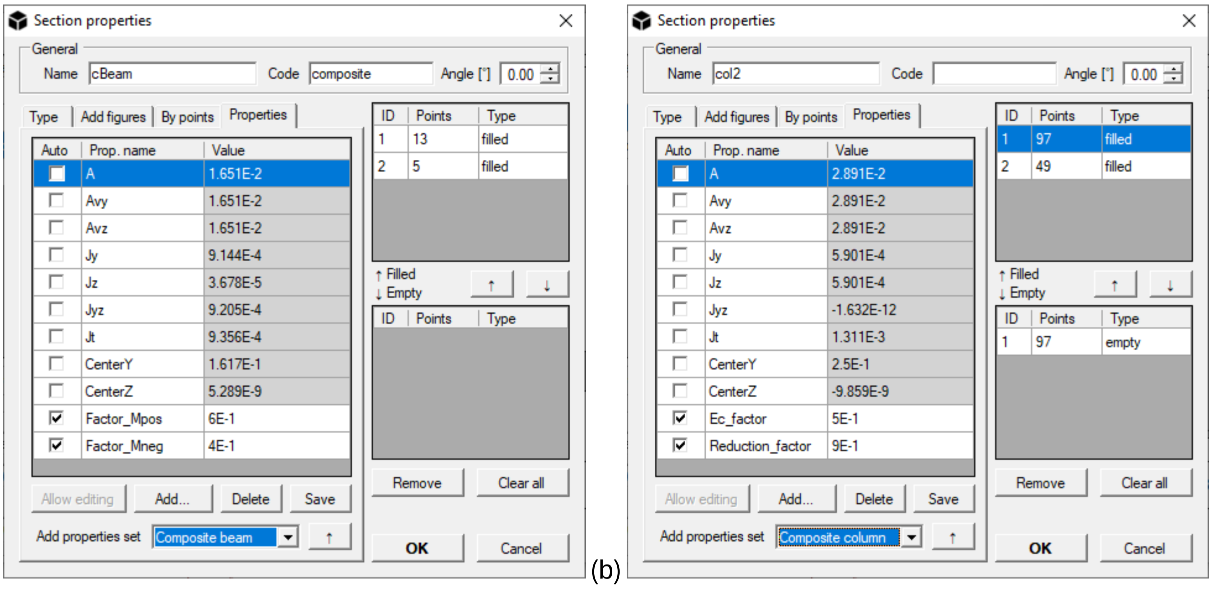
The stiffness of beam elements having such sections is evaluated as follows. With subscript *s*, steel properties are described, while the ones related to concrete have the subscript *c*.

For beams, the moment of inertia for positive bending around the elastic neutral axis *na* is calculated as:

$$I_{z1} = I_{zs} + A_{zs} \cdot (H_{tot} - y_s - x)^2 + \frac{I_{zc}}{n} + \frac{A_{zc}}{n}(x - H_{tot} + y_c)^2$$

$$with \quad n = \frac{E_s}{E_c} \quad and \quad x = H_{tot} - \frac{n \cdot A_{zs} \cdot y_s + A_{zc} \cdot y_c}{n \cdot A_{zs} + A_{zc}}$$

The moment of inertia of the section for negative bending around the plastic neutral axis is equal to the inertia given by the steel section, as the concrete part is considered as cracked.
As a whole, bending stiffness is estimated as per EC4 and as a function of the factors *Mpos* and *Mneg* which can be changed in the “Properties” input mask (figure a).



(a) (b)

Hence, the resulting bending stiffness is:

$$I_z = f_{Mpos} \cdot I_{z1} + f_{Mneg} \cdot I_{zs}$$

The remaining properties for the beam section are all calculated by dividing the concrete contribution by *n*.

For composite columns, once the factors Ec and R have been specified in the “Properties” mask (figure b), the inertial properties of the section are calculated as follows:

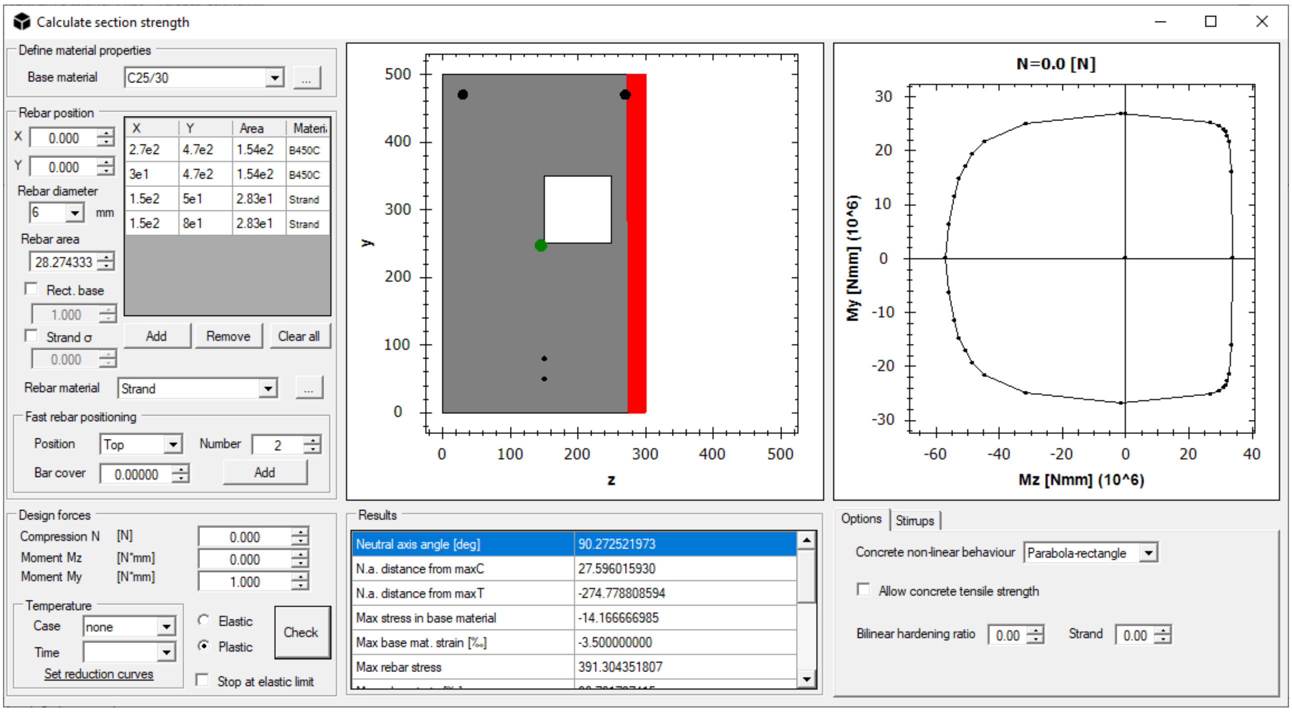
$$I_z = R \cdot \left(I_{zs} + \frac{I_{zc}}{n} \right); \; I_y = R \cdot \left(I_{ys} + \frac{I_{yc}}{n} \right); I_{yz} = R \cdot \left(I_{yzs} + \frac{I_{yzc}}{n} \right);$$

$$with \quad n = \frac{E_s}{f_{Ec} E_c}$$

The remaining properties are all calculated by dividing the concrete contribution by *n*.

Strength calculator

By the *Strength calculator* button it is possible to define the rebar in a RC section and to evaluate its flexural strength.



In *Define material properties* the base material is specified.

The box *Rebar position* can be used to specify the rebar coordinates (eventually of rectangular shape by checking the option *Rect. rebar* and by specifying the base dimension of the plate in *Rect. base*). The rebar coordinates are input on the base of the section local coordinate system, by specifying the diameter and the associated material (that can be set or added by the command "…"). The available materials should be customized by modifying or adding *.nfm files in *data/design* subfolder in the software installation directory.

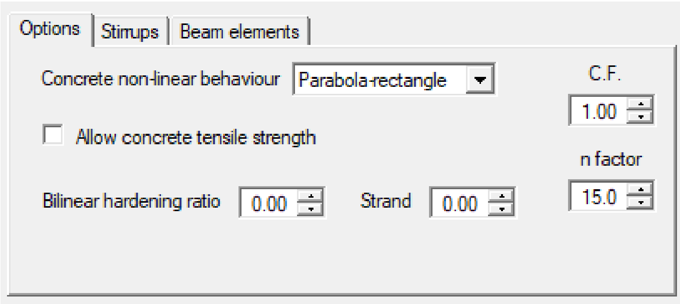
In order to insert strands for prestressed RC sections, it is possible to input the initial strand stress by activating the *Strand σ* option and by inputting the stress in consistent dimensions.

To obtain the resistant values of the section, specify the *Design forces* in the required units. By selecting the *Elastic* option, the response of the section is computed with elastic materials. By selecting *Plastic*, elastic-perfectly plastic laws are used for material behaviour. In the *Results* box, the results of the performed computation are reported, linked to the overlying graphs.

The window allows also resistant domains calculations reduced by thermal action. The software will compute the strengths parameters of the section at the selected instant of time and thermal case, on the base of strengths reductions laws for mechanical and resistant features of base material and rebar, if present.

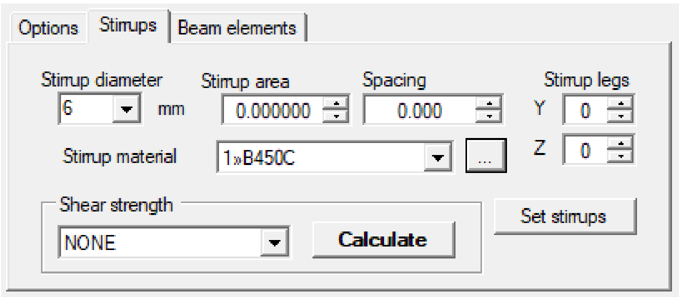
The available options in this mask are:

- *Concrete non-linear behaviour* allows to choose between parabola-rectangle and bilinear material behaviour for the concrete material. If a simple rectangular or circular section are present, the *Confined concrete* option is available in this menu, allowing to consider the confined part of the section inside stirrups. In this case, data in *Stirrups* tab are required. Spirals are not supported.

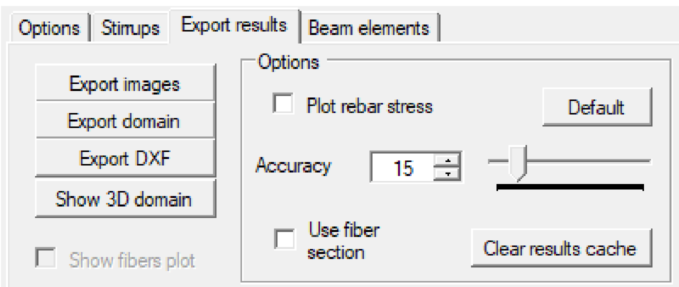


- *Allow concrete tensile strength* can be enabled to activate the contribution of tensile strength of the base material;
- *Bilinear hardening ratio* allows to specify the ratio between the plastic branch of rebar and their elastic modulus. In case of Steel or Aluminium base material, it specifies the hardening ratio to be used.
- *Steel class section* sets the class of the section in case of Steel or Aluminium base material, i.e. if the section can be calculated in plastic field or not.
- *Confidence Factor (C.F.)* sets the confidence factor for strength of base material and rebar.
- *n factor* specifies the value of the homogenization factor (modular ratio between steel and concrete). The default is 15.

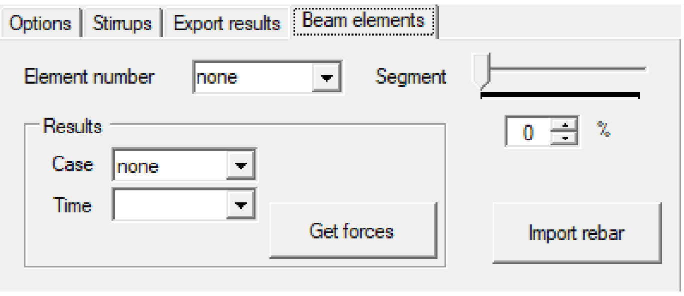
Stirrups tab helps to define the stirrups for the current section and to obtain a resisting strength for both direction of the section.



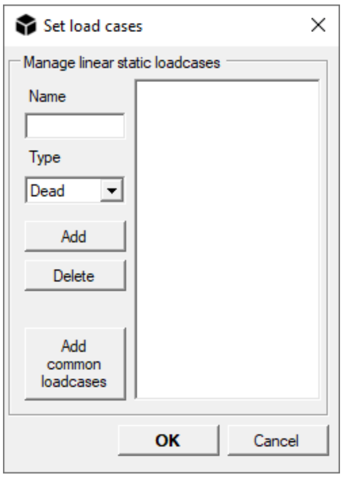
Export Results contains options to:
- plot the tension in the reinforcement bars,
- to perform the calculation with a triangular fiber section
- to change the accuracy of the calculation (recommended value 15, to be increased in the case of very thin parts)
- delete the calculation cache, which is useful when changing reinforcement materials or bars to prevent the program from returning the same results as before when these materials were changed
- export images of the analysed section or points of the resisting domain
- export the section to DXF with the reinforcement
- display the domain in 3D.



Beam elements allows the import of reinforcements from individual elements, temporarily overwriting those of the section. The *Get forces* button allows the stresses for the desired abscissa taken from the calculated model.



Load cases: Inserts the linear static load cases by writing the *Name* box and then clicking on the *Add* button. Different load can be inserted without closing the *Set load cases* window.

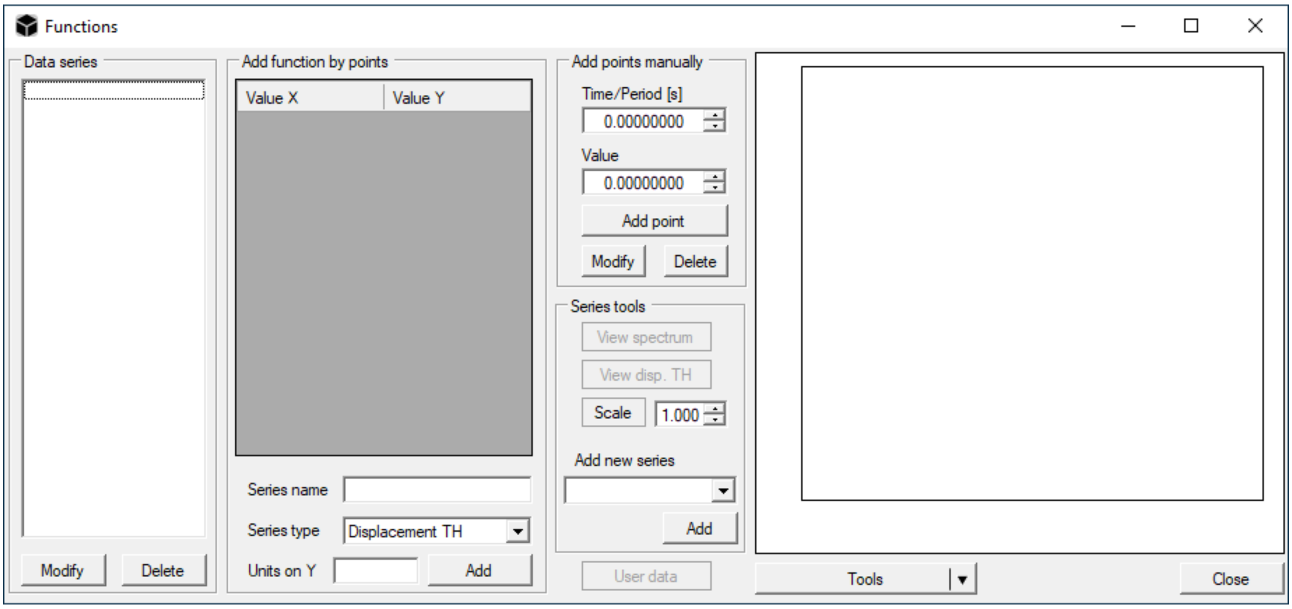


⚠ WARNING: only linear static load cases can be inputted from here.

From the *Type* drop-down menu it is possible to specify the type of base load case. This setting assumes importance in the automatic generation of load combinations, for which reference is made to the appropriate description of the command.

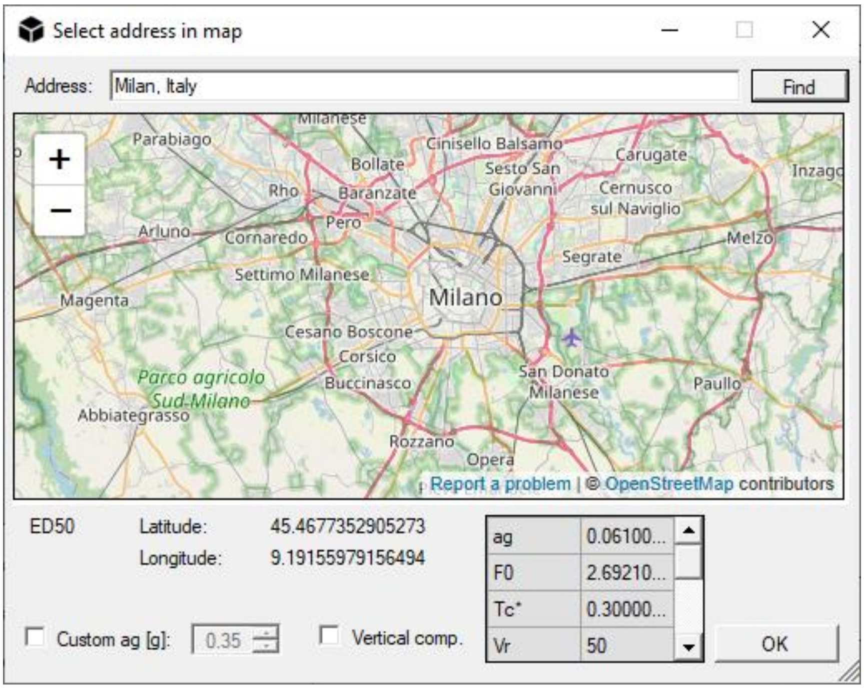
The *Add common loadcases* button automatically adds the most used load cases (self weight, permanent, variable, etc.).

Functions: Adds Time History (TH) or Spectrum functions by points or file.



From the dropdown menu in *Series tools* you can set several types of TH functions, such as:

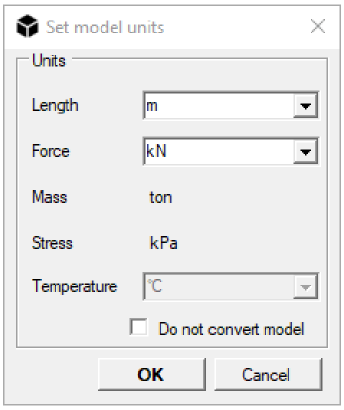
- *Eurocode 8 Spectrum*: returns the design acceleration spectrum as defined in Eurocode 8 – EN 1998-1-1
- *NTC2008 Spectrum*: returns the design acceleration spectrum as defined in NTC2008 (Italian code of practice D.M. 14-01-2008). The program uses an automatic geo-referencing algorithm that allows to search for postal addresses finding the desired site.



- *NTC2008 Wind Load*: gives the wind action as per NTC2008 (Italian code of practice D.M. 14-01-2008) as a function of building height
- *NTC2008 Snow Load*: gives the snow load as per NTC2008 (Italian code of practice D.M. 14-01-2008) as a function of height of the building site
- *Linear TH*: returns a linear loading ramp
- *Sinusoidal TH*: returns a sinusoidal function on the base of the properties chosen by the user
- *Calculated Spectrum*: this option is enabled only if user has requested the acceleration spectrum of the displayed acceleration TH through the button View spectrum

- *From File*: allows the user to load a 2-columns text file containing a data series
- Constant load: defines a constant loading plateau
- *EC8 Seismic Forces*: allows to define a set of static lateral forces for each rigid diaphragm defined as per EC8/NTC2008, to be used in an equivalent static analysis;
- *EC1 Wind Load*: wind load as per Eurocode 1
- *EC1 Snow Load*: snow load as per Eurocode 1
- *EC1 Standard Fire Curve*: fire curve as per Eurocode 1. The resulting series has time in seconds.
- *EC1 External Fire Curve*: external fire curve as per Eurocode 1. The resulting series has time in seconds.
- *EC1 Hydrocarbon Fire Curve*: hydrocarbon fire curve as per Eurocode 1. The resulting series has time in seconds.

Set units: Custom user units can be defined by clicking on the opposite drop-down menu and choosing between:



- Length:
  - Metres [m] (default option);
  - Centimetres [cm];
  - Millimetres [mm];
  - Kilometres [km];
  - Inches [in];
  - Feet [ft]
- Force:
  - Newton [N];
  - DecaNewton [daN];
  - KiloNewton [kN] (default option);
  - KiloPounds-force [kipf]
- Temperature:
  - Celsius degrees [°C] (default option);
  - Kelvin degrees [°K].

The following units are computed automatically in a consistent way:

- Mass:
  - Megagram [Mg];
  - Kilogram [kg] (default option);
  - Gram [g];
  - Ton [t];
  - Ounce [oz];
  - Pound [lb].
- Stress:
  - MegaPascal [MPa];

- KiloPascal [kPa];
- Pascal [Pa] (default option).

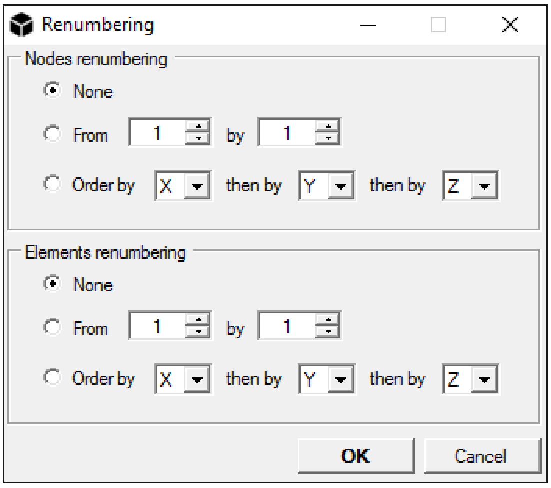
Check element connectivity: Checks counter-clockwise connectivity for elements with more than two nodes.

Check element overlap: Checks if there are some overlaid elements.

Check for free nodes: Check whether every node is connected to an element.

Check element properties: Checks if section and material have been assigned to each element.

Renumbering: Renumber the selected nodes and/or elements by the chosen criteria.

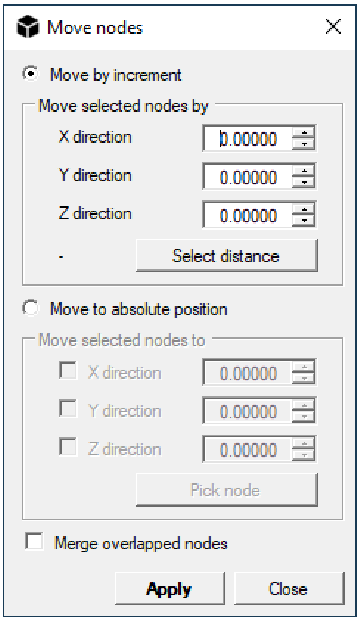


Check line mesh: Check all the line elements in the model to find out is there are disconnected beams (or trusses). This command corrects the mesh by dividing the lines that present apparent intersections (i.e. a node belonging to one element that lies on the joining line of the considered beam or truss).

Nodes tools

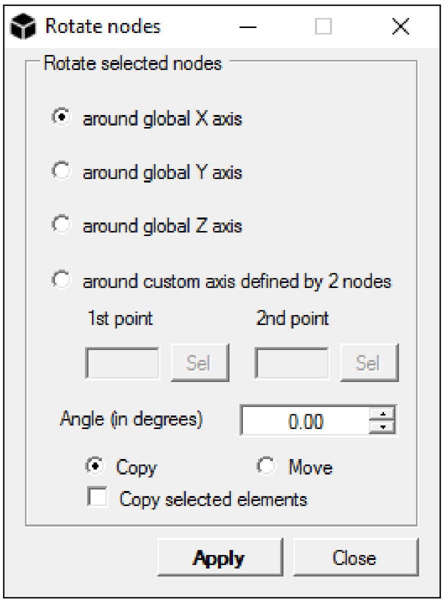
Merge overlapped nodes: Merges the nodes with the same coordinates in the whole model.

Move nodes: By selecting *Move by increment*, it moves the selected nodes of the specified quantity. By selecting *Move to absolute position*, it moves the selected nodes in the chosen directions to the specified position.

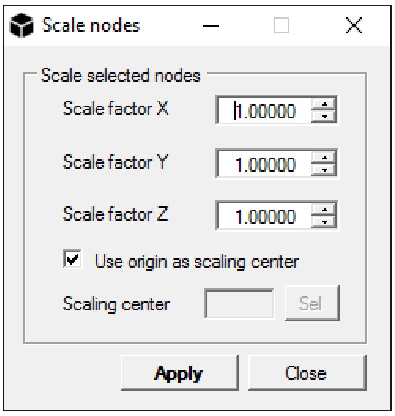


Rotate nodes: Rotate the selected nodes with respect of a specified axis.

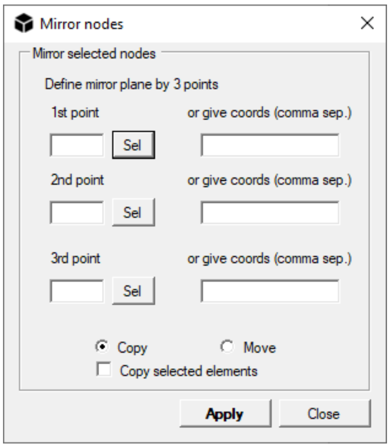
To define a custom axis select two nodes in the model.



Scale nodes: Scale the selected nodes (and elements) with respect to a specified point in the 3D space.

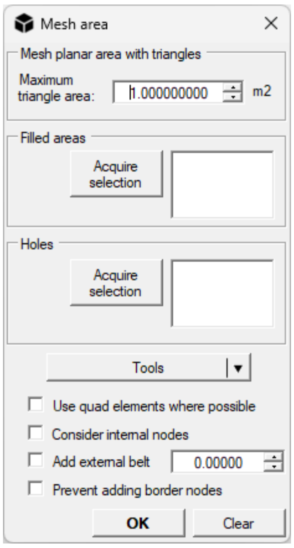


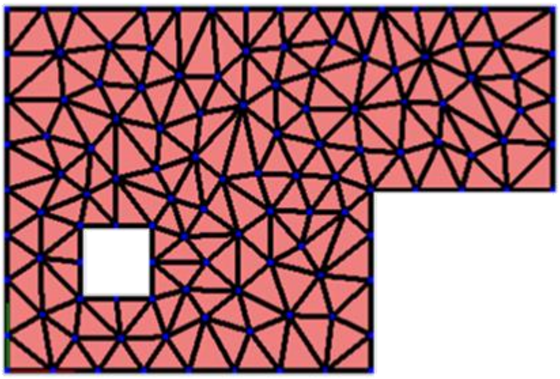
Mirror nodes: Mirror the selected nodes (and elements) with respect to a specified plane in the 3D space defined by 3 points.



Mesh tools

Mesh area: Automatically meshes an area into triangular elements. Choose the maximum area of the triangles to be generated, select the nodes of the perimeter of the area to mesh and then click on the *Acquire selection* button under the *Filled areas* section. To insert a hole in the mesh, select the nodes of the hole and then press the *Acquire selection* button under the *Holes* section. To confirm press *OK*.





Options:

- "Use quad elements where possible" combines triangles generated in pairs where possible into quad elements
- When the option "Consider internal nodes" is selected, only the groups of nodes forming filled polygons are considered, even if non-convex.
- The "Add external belt" option allows to mesh convex surfaces, extending them outwards for the desired length.
- The "Prevent adding border nodes" option allows the creation of a mesh without adding nodes on its edge (e.g., flexible plane).

Tools:

- The "Add internal nodes" button allows the selected nodes to be associated with the selected solid figure, forcing the nodes to be included in mesh (e.g. the base of columns in a slab).

– The "Follow border beams" button allows a mesh to be created using a concave or convex contour created by the selected beams.
– “Draw planar border” allows to draw on screen the border of the figure to be meshed.

Add wall: The command allows to insert and mesh walls of arbitrary dimensions, made of Quad elements.

To insert a wall insert the origin coordinates on the *Origin* box and specify its dimension on the *Length* and *Height* box.

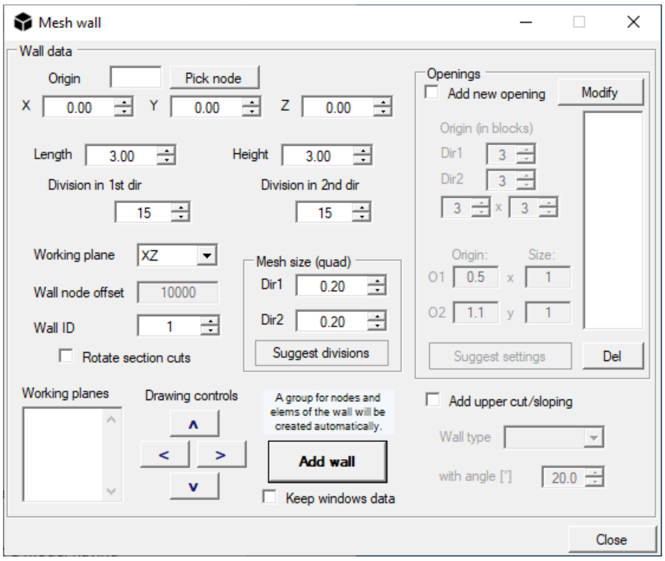
Enabling the *Add upper cut/sloping* option it is possible to insert sloping walls by specifying the *Wall type* and the angle of inclination (*with angle [°]* box). Clicking on the *Wall type* drop down menu the following options are available:

- *x*;
- *y*;
- *vertical* that is in the working plane direction.

x and *y* can be used only if the *working plane* is set as *XY*, otherwise they have no effect.

Through the *Drawing controls* pan it is possible to move in the working plane to insert quickly adjacent panels.

Enabling the *Add openings* option, holes can be added in the wall.



Automesh wall: The command allows to automatically mesh all the quad elements in the model, using the size specified in the *Options* under *Size for IfcWall mesh*.

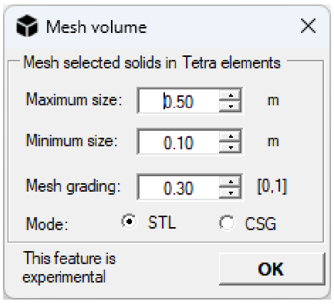
Mesh tapered beams: The command allows to automatically mesh all line elements of a model having variable cross section, i.e., different initial and final sections.

Expand macroelements: The command allows to automatically mesh all macro elements in a model, expanding them as specified by the submodels in the *macro* folder.

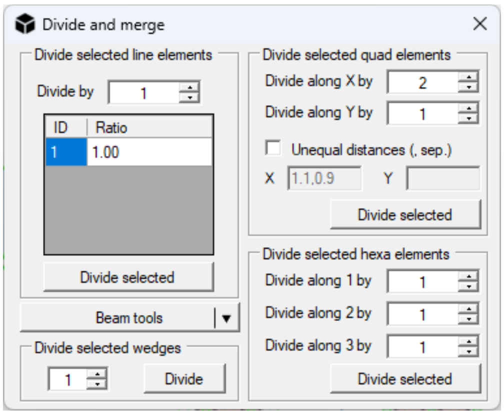
Add belt: The command allows to mesh both concave and convex belt, for example, on an existing slab.

Mesh volume: this command allows to mesh solids into tetrahedral elements. In *Maximum size* box the maximum characteristic size of the mesh element must be inserted; *Minimum size* sets the minimum characteristic size and *Mesh grading* sets the percentage variation of characteristic dimension amongst elements in target mesh. Press *OK* to mesh selected elements.

Select STL if the volume is delimited by planar facets; select CSG is the volume is described by solid elements.



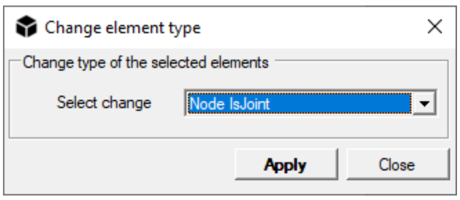
Divide and merge: this command allows to divide in equal parts the selected *line* (2 nodes) and *quad* (4 nodes) elements. Moreover, it is possible to merge the selected *line* elements having the same first local axis. Divide commands are available also for Quad and Hexa elements.



Wall to frame: this command allows to obtain a frame from a shell wall model. This is particularly useful to convert into beams all the Wall elements imported from a Midas GEN® model or to apply plastic hinges to the beams obtained from walls. The local y axis of Quad element becomes the x axis of the beam.

Extrude section: this command extrudes the selected beam elements accordingly to their transversal section into shell or solid models. This is particularly useful to convert a part of a frame into a shell or solid model to allow detailing analyses, such as local buckling.

Change element type: This command allows changing the type of the selected elements. Pick a transformation from the menu and then press *OK*.



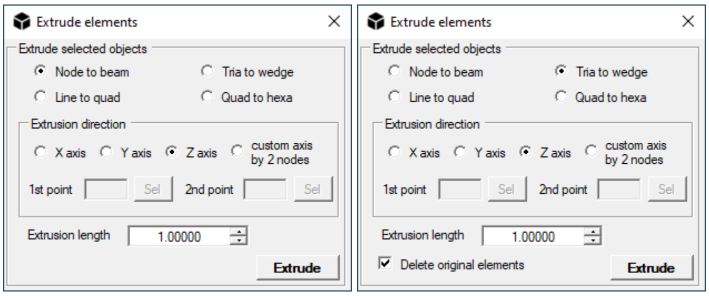
The possible conversions are:

- Truss->Beam: from truss to beam
- Beam->Truss: from beam to truss
- Beam->Beam3: from beam to 3 node beam (middle node)
- Beam3->Beam: from 3 node beam to beam
- Spring->Beam: from spring to 2 node beam
- Beam->Spring: from 2 node beam to spring
- Quad->Tria: from quad to tria

- Quad->Quad8: from quad to quad8
- (planar)->PlaneStress: from planar element to plane stress
- (planar)->Shell: from planar element to shell
- (planar)->PlaneStrain: from planar element to plane strain
- Tetra->Tetra10: from tetra to 10 nodes tetra
- Hexa->Hexa20: from hexa to 20 nodes hexa.

⚠ WARNING: PlaneStress and PlaneStrain elements can be used only in XY plane.

Extrude elements: this command allows to extrude elements from selected nodes, tria or quad elements. In all cases, the *Extrusion direction* and the *Extrusion length* must be provided by the user.



If *Node to beam* is checked, the selected nodes will be extruded to beam elements. Remember to assign a material and a section to the obtained beams.

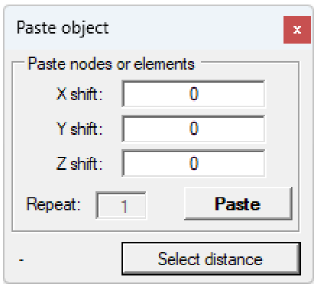
If *Tria to wedge* is selected, the selected tria elements will be extruded to wedge elements. By checking *Delete original elements*, the initial tria elements will be deleted after extrusion. The material of newly created elements will be assigned equal to material of original tria elements.

If *Quad to hexa* is selected, the selected quad elements will be extruded to hexa elements. By checking *Delete original elements*, the initial quad elements will be deleted after extrusion. The material of newly created elements will be assigned equal to material of original quad elements.

Cut: Cuts the selected elements (CTRL+X).

Copy: Copies the selected elements (CTRL+C).

Paste: To paste the copied/cut elements insert the shift coordinates in the opposite boxes and then click in *Paste* button (CTRL+V).



Undo: Undo the last action.

Redo: Redo the previous action.

2.5. View menu

Select all: Select all nodes and elements in the model.

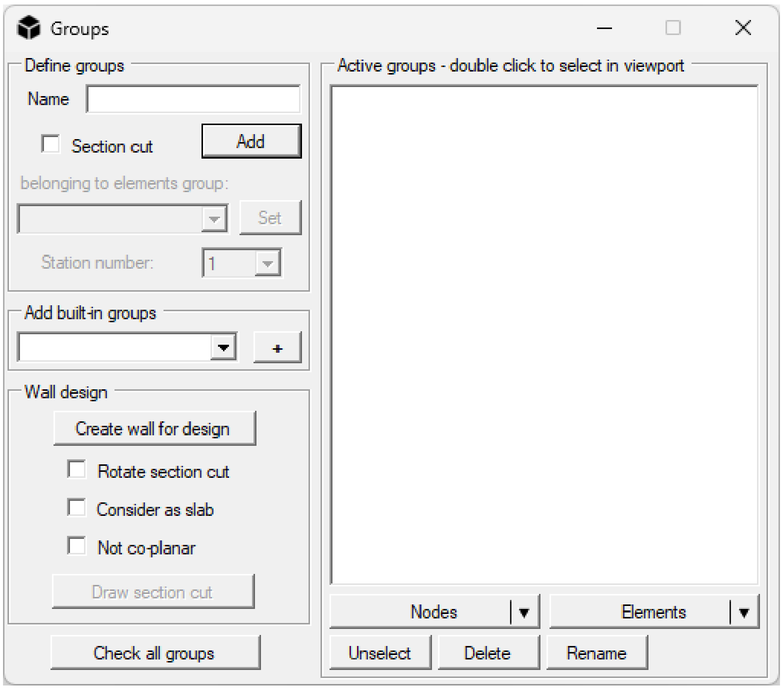
Invert selection: Select the complementary elements and nodes to those that have been already selected.

Clear selection: Deselect all elements and nodes.

Delete selected: Delete the elements that have already been selected.

Gr *Select group:* This command allows to define group and to assign element and/or nodes to it. It permits also to select nodes and elements previously assigned to a group.

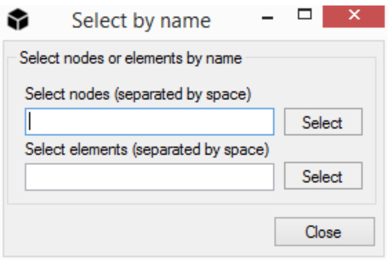
To define a group, write its name in the *Name* box and press the *Add* button. To assign elements and nodes to each group select the group in the *Active groups* pan, select the nodes and the element to assign to the group and then click the *Add selected nodes* or *Add selected elements* button.



To select nodes or elements previously assigned to a group activate the desired group in the *Active groups* panel and then press the *Select nodes in group/Select elements in group* button.

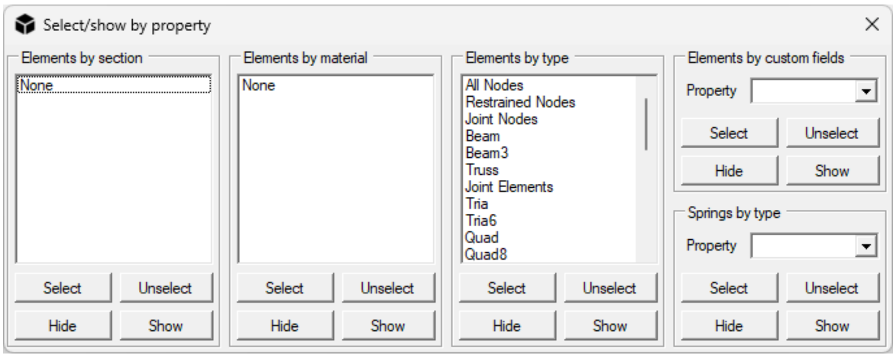
To create a group of planar element that can hold reinforcements, use the commands in *Wall design* box. Select the planar elements and press *Create wall for design*. Automatically, 3 horizontal section cuts will be created (both ends and in the middle) to check the group by forces integration. To have vertical section cuts (e.g. floor spandrels), select *Rotate section cut*. To check slabs and avoid generation of section cuts, activate *Consider as slab*. The *Not co-planar* option allows to set groups of not aligned area elements.

Select by name: This command allows to select nodes and/or elements by name (ID). The names must be written separated by a space.



Pr *Select by property:* By this option, elements can be selected by section, by material or by their type.

- To select all the elements with the same section activate it in the *Elements by section* pan and the click the *Select* button. Press *Unselect* to deselect, *Hide* to hide them and *Show* to add them to the view.
- To select all the element with the same material activate it in the *Elements by material* panel and then press the *Select* button.
- To select all the elements of the same type such as Node, Line, Triangle, Quad, and so on, select the type(s) in the *Elements by section* pan and the click the *Select* button.



Show all: Shows all the elements and nodes in the model

Show selection only: Limits the view only to selected elements

Hide selection: Hides selected elements

Box pan/rotate: Rotates the model avoiding the graphic regeneration of each element. To be used in old PCs.

Move: Move the model (pan command).

Rectangle Zoom: zooms the area selected by a selection rectangle

Views: Changes the view options. Choose between:

3D: Shows the entire model in a 3D view;

1 *XY:* Shows the model from the front in the XY plane;

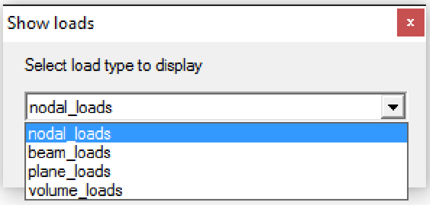
2 *XZ:* Shows the model from the front in the XZ plane;

3 *YZ:* Shows the model from the top in the YZ plane;

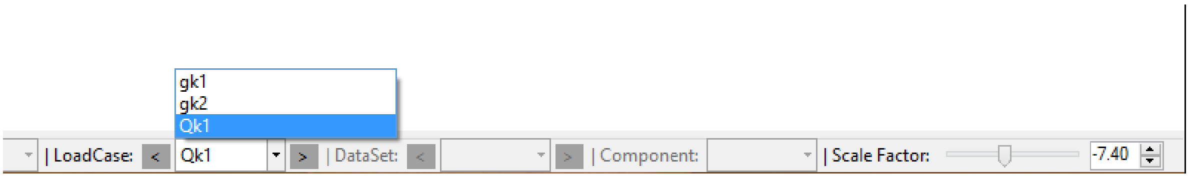
View nodes: Enable/Disable viewing of nodes in the model

Extrude: Enable/Disable viewing of extrude model

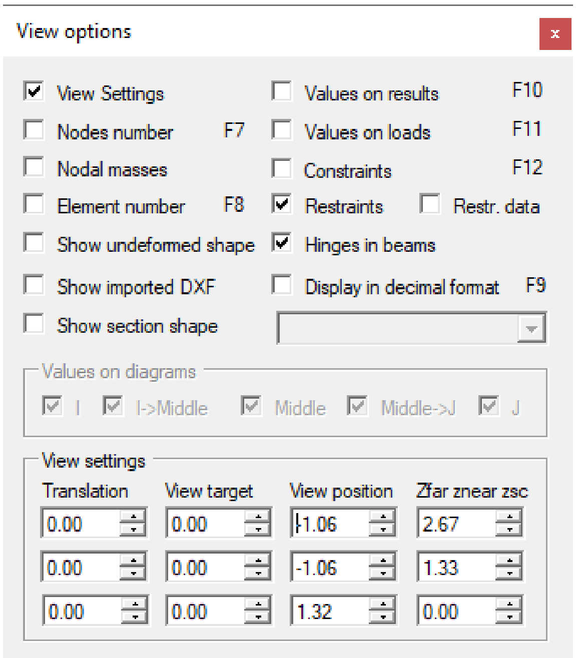
🠋 *Display loads:* shows loads applied to the structure depending on their type. To show a load type, click on the drop-down menu and choose between *point*, *line*, *area* or *volume*.



Then select the wanted load case form the bottom toolbar.



View controls: Through this panel it is possible to change the parameters of the view.

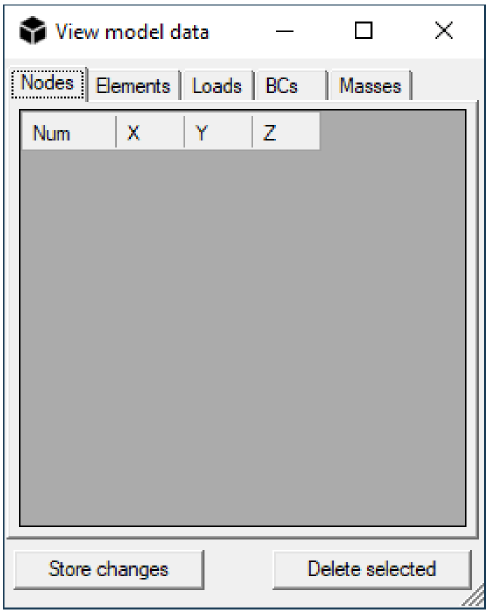


- *Partition opening:* experimental function that enables loading ADAPTIC partitions;
- *View Settings* shows the point of view and other graphics options of the model;
- *Nodes number* shows the number of the nodes in the model;
- *Nodes masses* displays the positions of nodal masses in the model;
- *Element number* shows the number of the elements in the model;
- *Values on diagrams* shows values in the beams force diagrams;
- *Values on loads* displays the values of applied loadings;
- *Constraints* displays the applied constraints in the model;
- *Restraints* displays the applied restraints in the model (active by default);
- *Hinges in beams* displays the hinges applied to the beam elements (active by default);

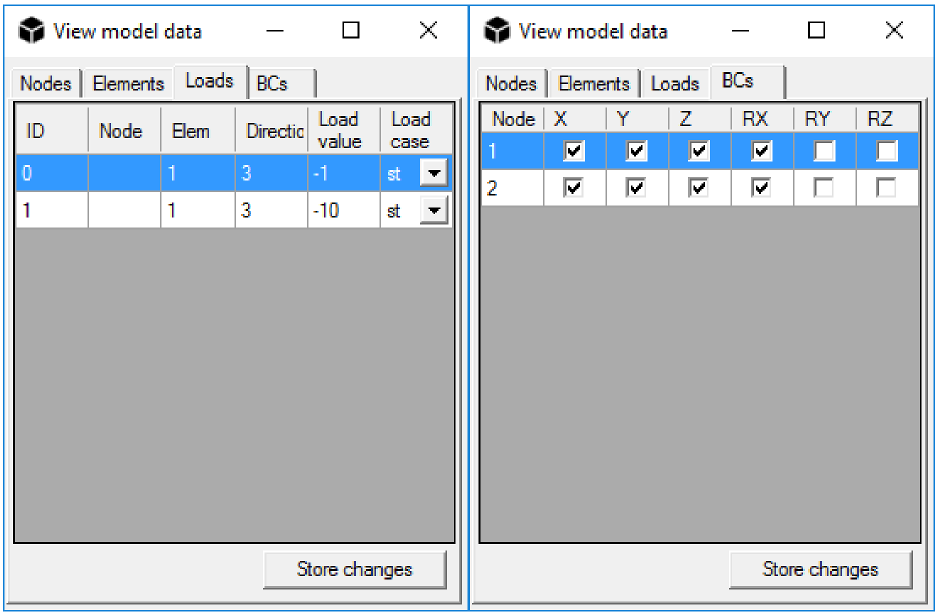
- *Display in decimal format* shows all number in viewport (both for loads and results) in the desired format (e.g. “0.000”).

In the box called *Values on diagrams* it is possible to specify which values have to be reported in the viewport when *Values on results* is activated. This allows to enable/disable values for each one of the 5 output stations in a beam diagram.

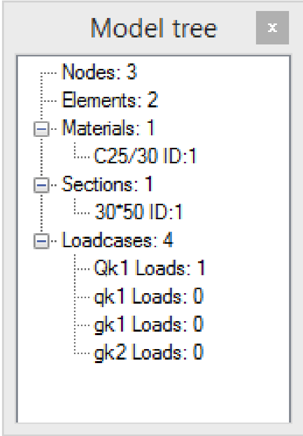
Model data: This tool allows to see the *Nodes*, *Elements* or *Loads* in the model as long as theirs coordinates.



From this mask it is also possible to modify loads and restraints simply by changing the values in the showed table and then clicking on *Store changes*.



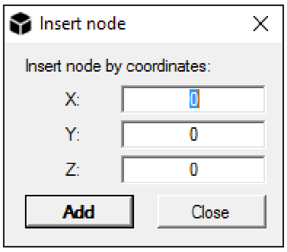
Model tree: Shows the main properties of the model (materials, sections and loads currently applied).



2.6. Draw menu

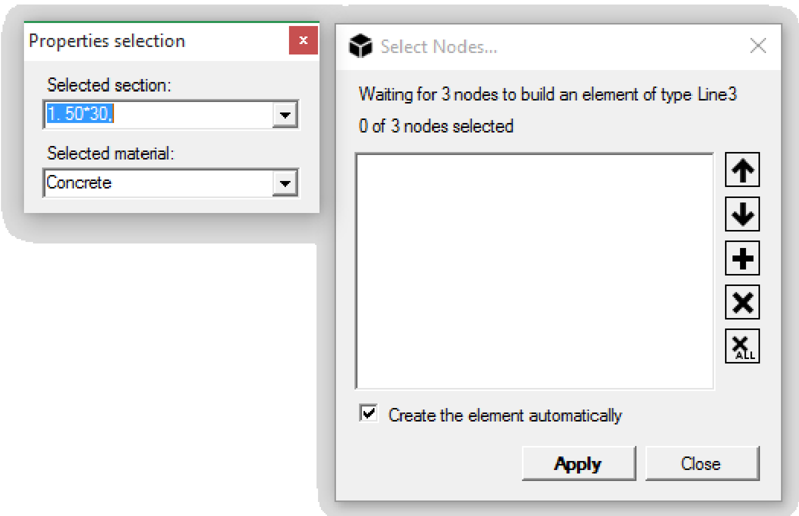
Node: Inserts nodes in the model, after a working plane is selected.

Node by coordinates: Inserts nodes by coordinates. To insert a nodes by coordinates, insert them on the textboxes and then click on the *Add* button. It is possible to insert nodes until the *Close* button is pressed.



Truss: Inserts truss elements between two nodes. This element can react only in tension or compression.

Beam: Inserts beam elements between two nodes. To insert a beam element select a section and a material from , if it has already been defined, from *Selected section* drop-down menu in the *Section* selection window and then click on the nodes to be connected. Once finished click on the *Close* button in the *Selected nodes* window.



Beam3: Inserts a 3 nodes beam elements. The three nodes must be aligned.

Triangle: Creates a 3 nodes triangular plane element.

Triangle6: Creates a 6 nodes triangular plane element.

⚠ WARNING: Triangle6 elements are transformed in a group of 3 nodes elements with the default solver.

Quad: Creates a four nodes quadrilateral plane element.

Quad8: Creates an eight nodes quadrilateral plane element.

⚠ WARNING: Triangle6 elements are transformed in a group of 3 nodes elements with the default solver.

Tetra: Creates a 4 nodes tetrahedron.

Tetra10: Creates a 10 nodes second-order tetrahedron.

Wedge: Creates a 6 nodes wedge.

Wedge15: Creates a 15 nodes wedge.

Hexa: Create an 8 nodes hexahedral element.

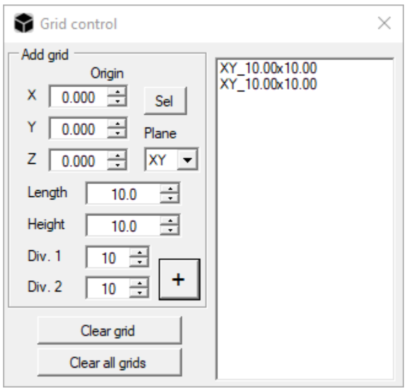
Hexa16: Create a 16 nodes hexahedral element.

Hexa20: Create a 20 nodes hexahedral element.

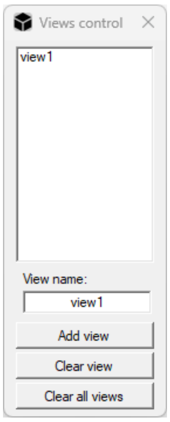
Spring: Creates a 2 nodes spring.

⚠ WARNING: To create a spring element, the spring properties must be specified in *Assign/Spring properties* and then assigned to the element.

Grid manager: it manages all grids in the viewport, to support drawing nodes. In grids, at each line intersection, snap is available.



Views control: it creates and handles model views.



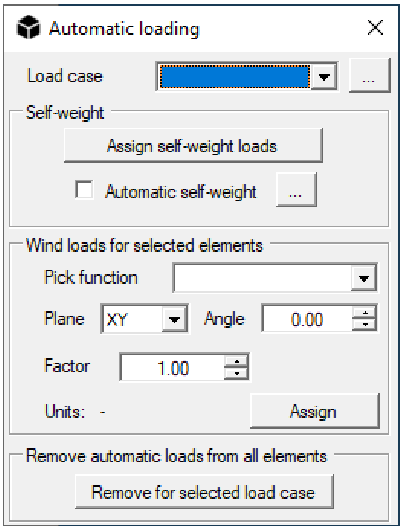
2.7. Assign menu

Loads

Automatic loading: Allows to assign to all the elements in the model some automatic-generated loads, such as self-weight and wind loads.

The command *Assign self-weight loads* produces the distributed load only for the elements having a material and a section assigned to.

The command *Assign* in the *Wind loads* box produces the assignation of wind load to all the Line elements in the model, in the chosen direction. The loading is calculated from a defined user function, and applied to the lateral surface of each element, obtained from the largest edge of the transversal section.

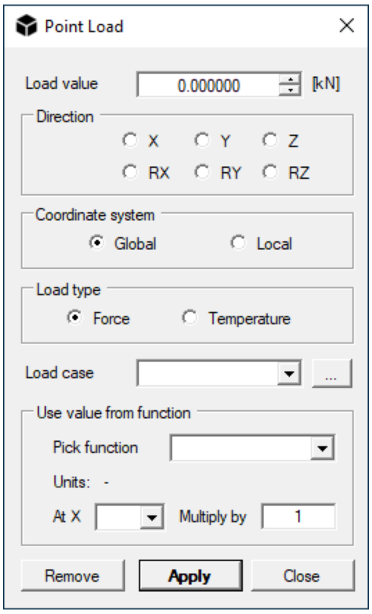


Point load: Applies a concentrated loads to selected nodes. To insert a concentrate load:

- Select one or more nodes;
- Insert the loads value in the *Load value* box;
- Select the direction clicking in a radio-button on the *Direction* box;
- Select the load case on the *Load case* drop-down list;
- Click on the *Apply* button to set the load.

To use a value from a defined function, pick the desired function from the *Pick function* menu and then specify the value from which read the function the in the *At X* menu. Optionally, a multiplier for the resulting value can be specified.

Finally, the option *Temperature* allows to assign a nodal temperature to be used in *Heat-transfer* load cases.

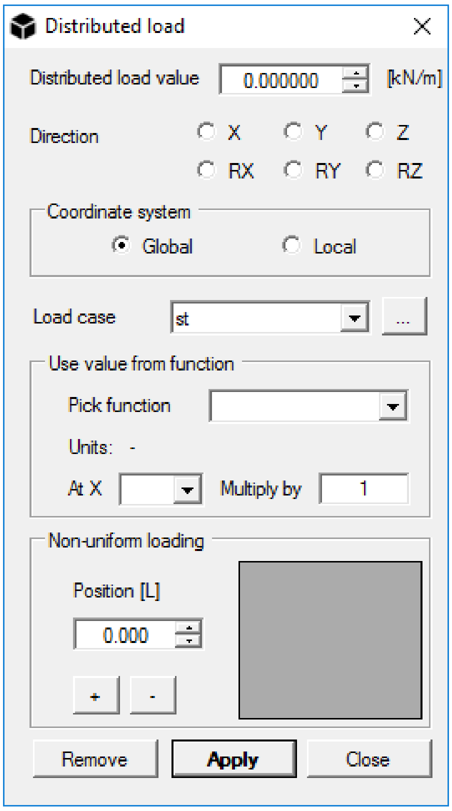


Distributed load: To insert a distributed load on a line element:

- Insert the value of the load in *Uniform load value* box specifying the correct sign in global coordinate system;
- Select the *Direction* of the load;
- Select the proper *Load case*;
- Select the element in the model and click on the *Apply* button.

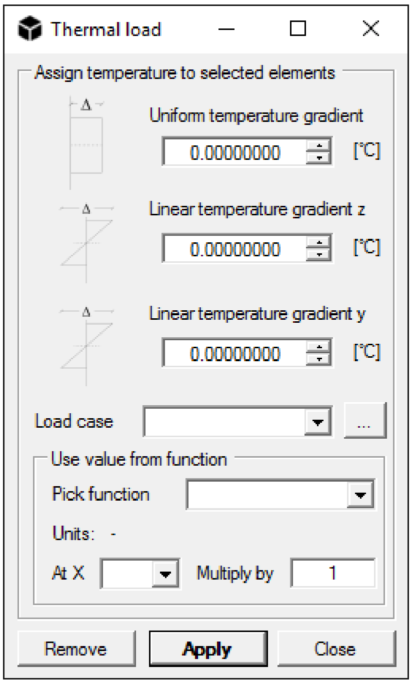
To remove a distributed in the selected load case, select the element in the model and click on the *Remove* button.

By the *Non-uniform loading* box it is possible to insert linear distributed load by points, specifying the proper distance for the input values.

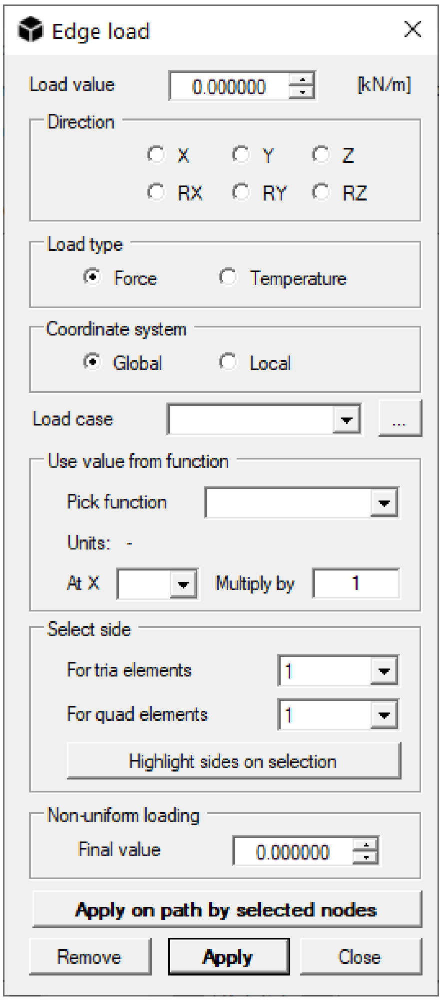


- ⚠ WARNING: The units need to be consistent with the initial choices [Force/Length].
- ⚠ WARNING: Since version 1.3, non-uniform loading is supported even in the default solver. An indefinite number of multi-linear distributed loads can be applied on the same beam.
- ⚠ WARNING: *Rigid offsets* applied on beams having non-uniform loads must be placed before and/or after the beginning or the end of the loads, otherwise they will not be considered.
- ⚠ WARNING: *Line3* elements, for compatibility reasons, do not support non-uniform loading. *Line* elements can be used without losing accuracy.

Temperature: Applies a thermal load, which causes a structural deformation, to the selected beams and planar elements. Beam element support the uniform gradient and the linear temperature gradient, in both y and z directions of the transversal cross section. Planar elements support only uniform gradient. Finally, solid elements do not support thermal loading.



Edge load: Applies an uniform load on an edge of an element. It can be used to assign forces per unit of length (*Force* option) or temperatures (*Temperature* option). This last option is intended to be used of heat transfer analysis only. After you selected the planar element, from the *Select side* box it is possible to highlight the chosen edges.



Non-uniform loading box allows to set a final value for load. The button *Apply on path by selected nodes* allows to apply edge loads following selected nodes.

Floor load: Applies a floor load on a triangular surface defined by Line elements or quadrangular by selecting 3 or 4 nodes. Choose the floor type within:

- Centroid-trilateral plane: to load a triangular plane distributing the load all over its sides;
- Centroid-quadrilateral plane: to load a quadrilateral plane distributing the load all over its sides;
- One-way quadrilateral plane: to load a quadrilateral plane on 2 or more sides taking into account the load direction (i.e. floor frames);
- Two-way quadrilateral plane: to load quadrilateral plane with 2-way distribution (tributary area method).

In the *Set floor load* mask target load cases and relatives values can be specified.

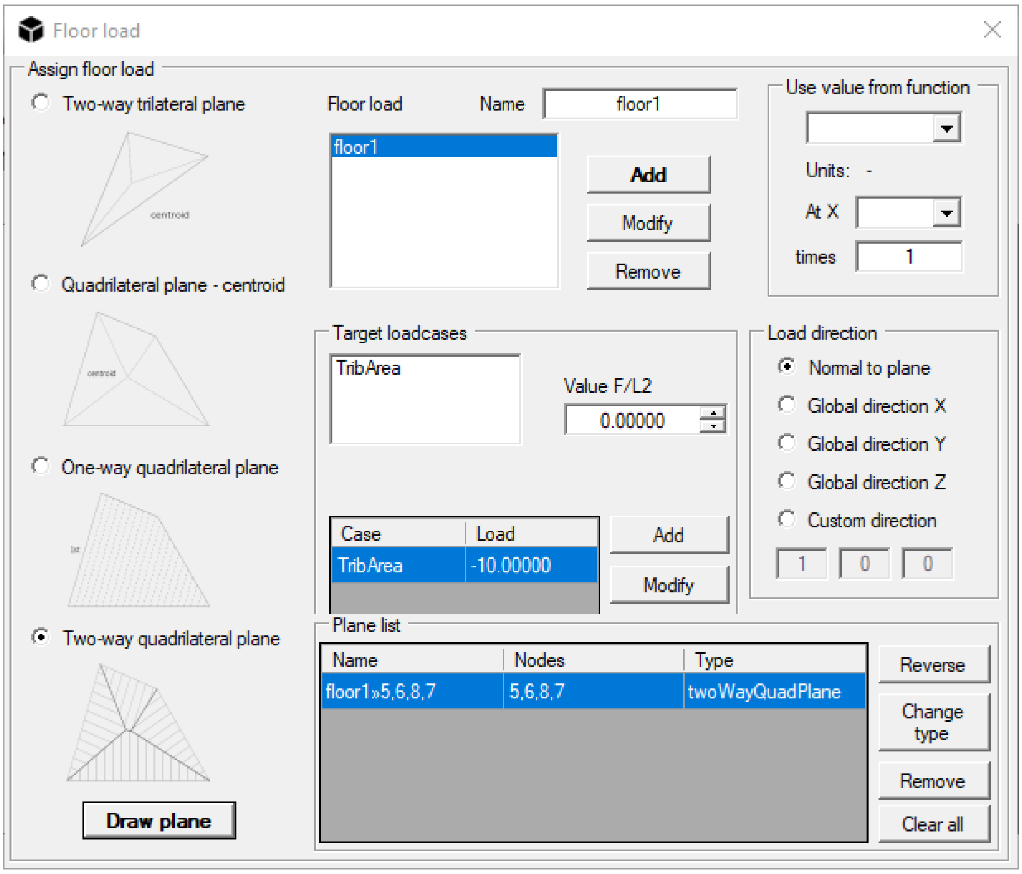
In the *Load direction* box it is possibile to specify a direction for applied loads. The default setting *Normal to plane* applies the loading normal to the plane by considering its clockwise or counterclockwise orientation. This setting will be ignored if another direction is specified.

To draw the plane once selected the desired plane load, select *Draw plane*.

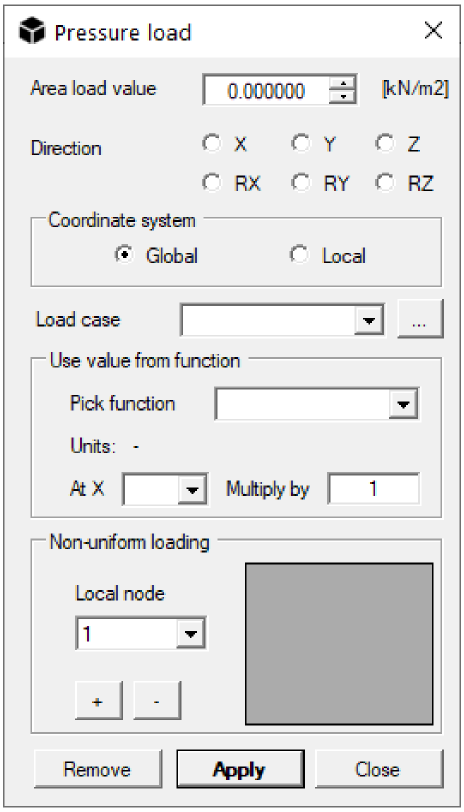
⚠ WARNING: input loads in this section must have F/L^2 dimension.

⚠ WARNING: edge plane elements can only be of Line type.

⚠ WARNING: the plane drawing direction defines the load sign (positive or negative) in the global coordinate system - draw planes counter clockwise to keep the value defined in the *Set load* table.

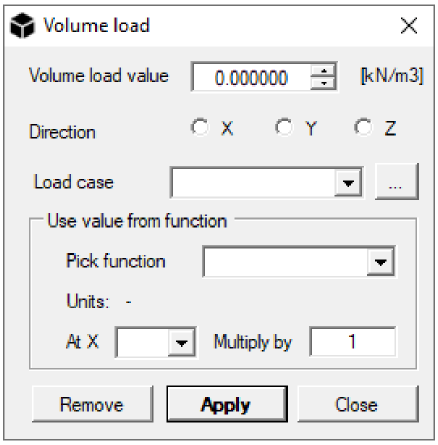


Pressure load: Applies a pressure load on planar elements.

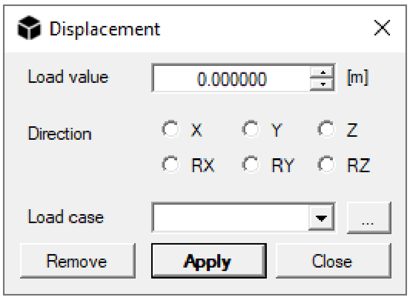


The *Non-uniform loading* box allows to apply a bilinear load on a face of planar element.

Volume load: It applies a load per unit of volume on solid elements.



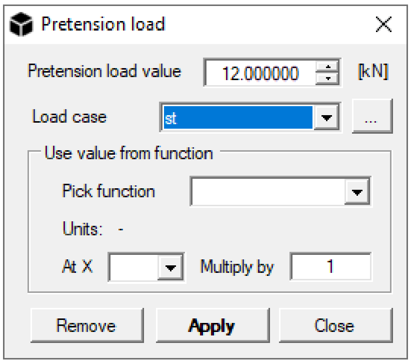
Point displacement: Applies displacements to selected nodes.



Initial temperature: Set the initial temperature in the model.

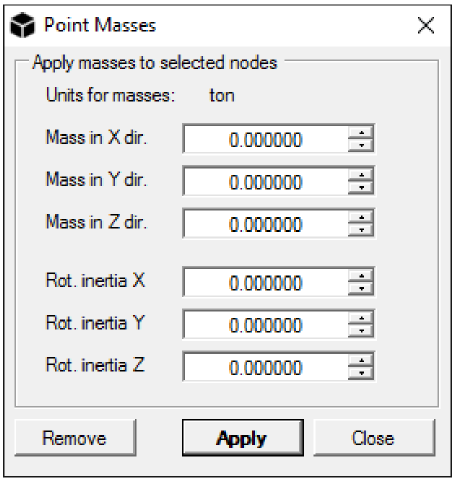
⚠ WARNING: This command affects only thermal analyses. It does not affect the loads induced by thermic distortion.

Pretension load: Applies pretension loads to linear (truss and beams) elements. A positive value signifies a tensioned element (i.e. tendons, cables). Only positive values are permitted (i.e. element in tension).



⚠ WARNING: The load is displayed as a uniform thermal distortion on the element. The value of distortion will have the opposite sign of the assigned pretension.

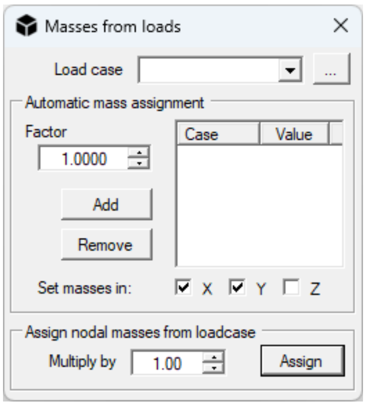
Point Mass: Applies masses to selected nodes.



⚠ WARNING: The units need to be consistent with the initial choices.

⚠ WARNING: You may want to add the nodal mass in all the three translational directions.

Masses from loads: Allows to transform, automatically or once at a time, the loads contained in a load case into masses. Checks in 'Set masses in' activate or deactivate the mass in the corresponding direction.



The "Assign nodal masses from loadcase" box is used for assigning masses in the directions specified above without changing them as loads change.

Material: Assigns material to the selected elements.

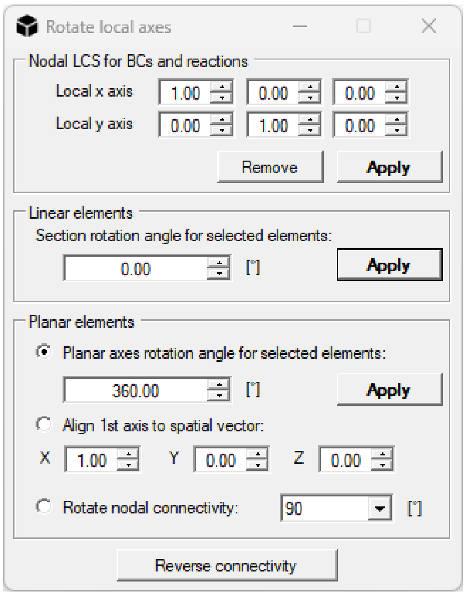
Section: Assigns section to the selected elements.

Local axes rotation: In the *Beams* box, the command rotates the selected beam sections by the counter-clockwise angle specified.

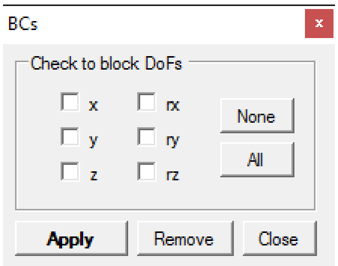
In the *Planar Elements* box, it rotates the planar local axes by changing the element connectivity.

The *Reverse connectivity* command is used to reverse the Z axes of plane elements.

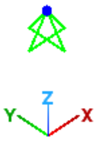
⚠ NOTICE: With non-regular meshes (e.g., triangular), all local axes should be rectified in order to read the results in terms of stresses and strains consistently



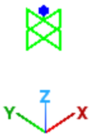
Restraints: Assigns restraint to the degrees of freedom selected by checking the appropriate check box.



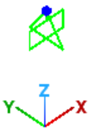
In viewport, the following drawing conventions applies:

- 

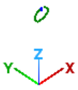
x, y and z checked (simple support)

- 

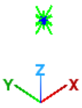
all checked (fixed end)

- 

x, y, z and ry, rz blocked (fixed in XZ plane, simply supported in YZ plane)

- 

y and z fixed, slider in plane xz

- 

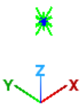
x, y, z and rx blocked

- 

only ry blocked

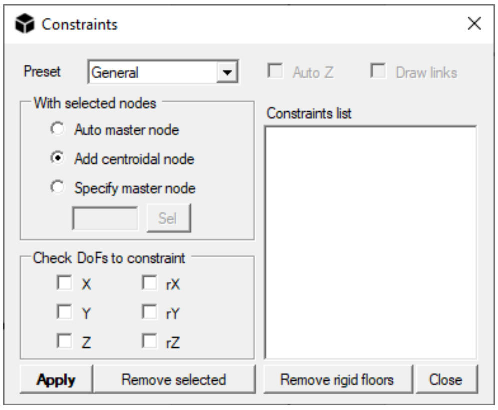
- 

rx and ry blocked

- 

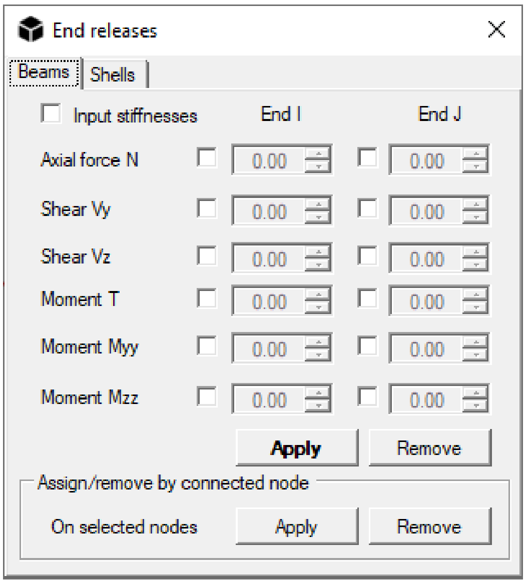
all other restraint types.

Constraints: Assign a constraint (i.e. rigid diaphragm or master-slave link) to selected nodes. By the *Rigid diaphragm* preset, a rigid floor can be applied to the selected nodes.

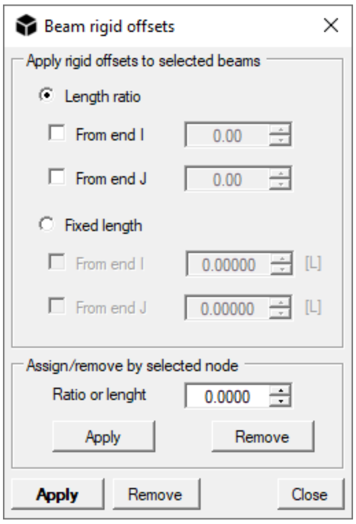


WARNING: The master node must be bounded properly to avoid singularities in the model. This procedure always proposes the proper boundary conditions automatically.

End releases: Assigns end releases to beam elements. By specifying a value between 0 (fully released) and 1 (fully fixed) in the textbox next to the check, a partial end release to be used in a linear elastic analysis can be obtained. The reduction factor *r* is applied to reduce the stiffness of the beam by the ratio *r/(1-r)*.
The option "*Input stiffnesses*" allows to directly set stiffness values instead of force ratios.

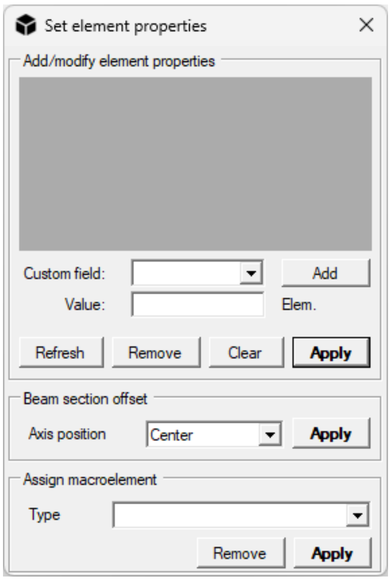


Rigid offset: Assigns one or more rigid offsets to beam elements.



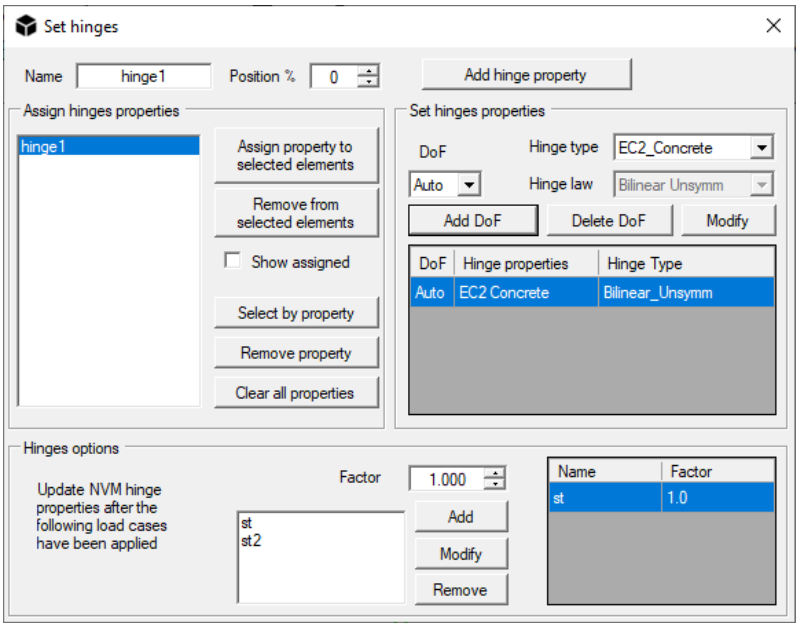
⚠ WARNING: Loads on a beam with rigid offsets are not modified not changed or adapted to the rigid segments. As a result, distributed loads on beams can decrease in their overall intensity due to the lesser length of application.

Element properties: Assigns custom properties to beam elements. For the property names and reserved values see chapter 4.
In addition, from this mask it's possible to assign a beam section offset at element level, from the box *Beam section offset*.



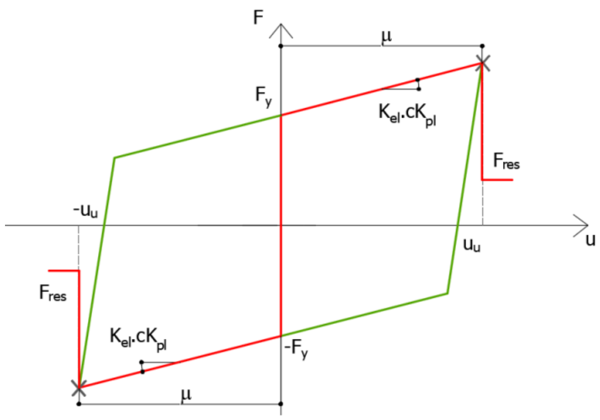
Hinges: Assigns plastic hinges to beam elements. After the name of the hinge has been inserted, press *Add*. At this point, each DoF of the hinge can be set by inserting the properties listed below. Press *Modify* to save the hinge properties.

The *Hinges options* box allows to specify under which loading conditions all hinges have to be recalculated. This is particularly useful for hinges with NVM interaction.



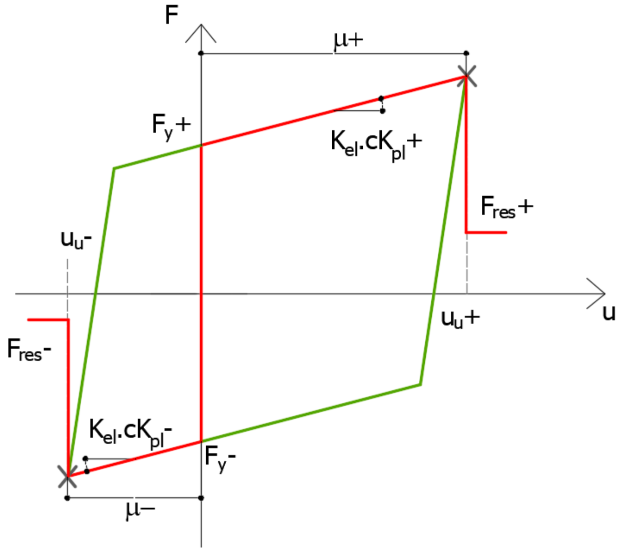
Data requested for a *Bilinear Symmetric* plastic hinge are:

- *Fy*: strength at elastic limit;
- *cKpl*: coefficient to set the slope of the plastic branch Kpl=cKpl * Kel;
- μ: hinge ductility;
- *Fres/Fy*: residual strength in percentage of Fy.



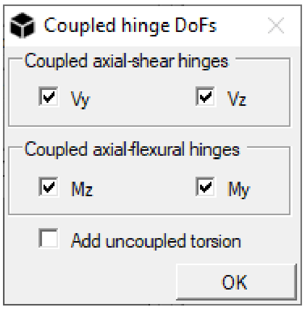
Data requested for a *Bilinear Unsymmetrical* plastic hinge are:

- *Fy+*: strength at elastic limit;
- *cKpl+*: coefficient to set the slope of the plastic branch Kpl=cKpl * Kel;
- μ*+*: hinge ductility;
- *Fres/Fy+*: residual strength in percentage of Fy+
- *All the above quantities followed by a "-" sign:* to define the negative part of the hinge law.

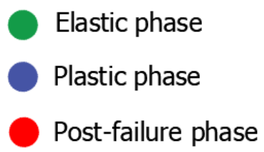


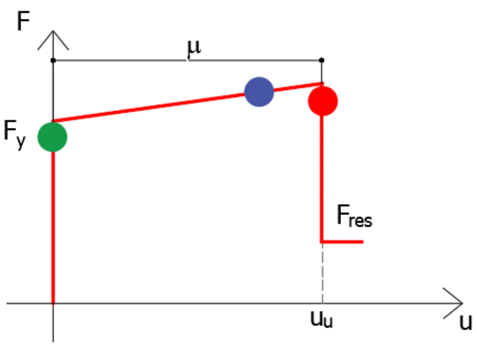
The hinges inserted with *Custom properties* are defined by setting all the above data.

By selecting *NVM* from the *DoF* dropdown menu it is possible to assign an axial/shear/flexure interacting hinge. The calculation of all its properties is made automatically on the assignation on the base of checking type selected from the dropdown menu *Hinge type*. *NVM* hinges always have an unsymmetrical bilinear law.

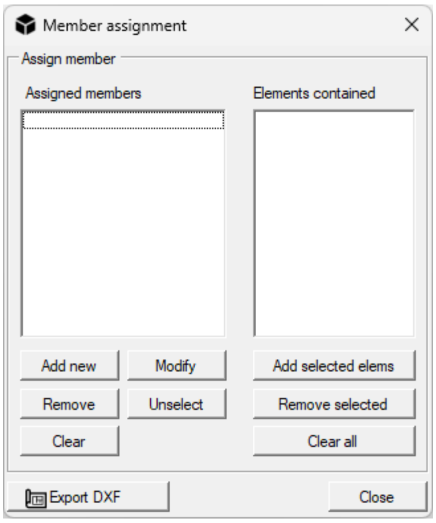


Hinge status can be seen together with beam diagrams results, with the following convention.





Member: Assigns one or more elements to a member. This affects the verification of a beam element.



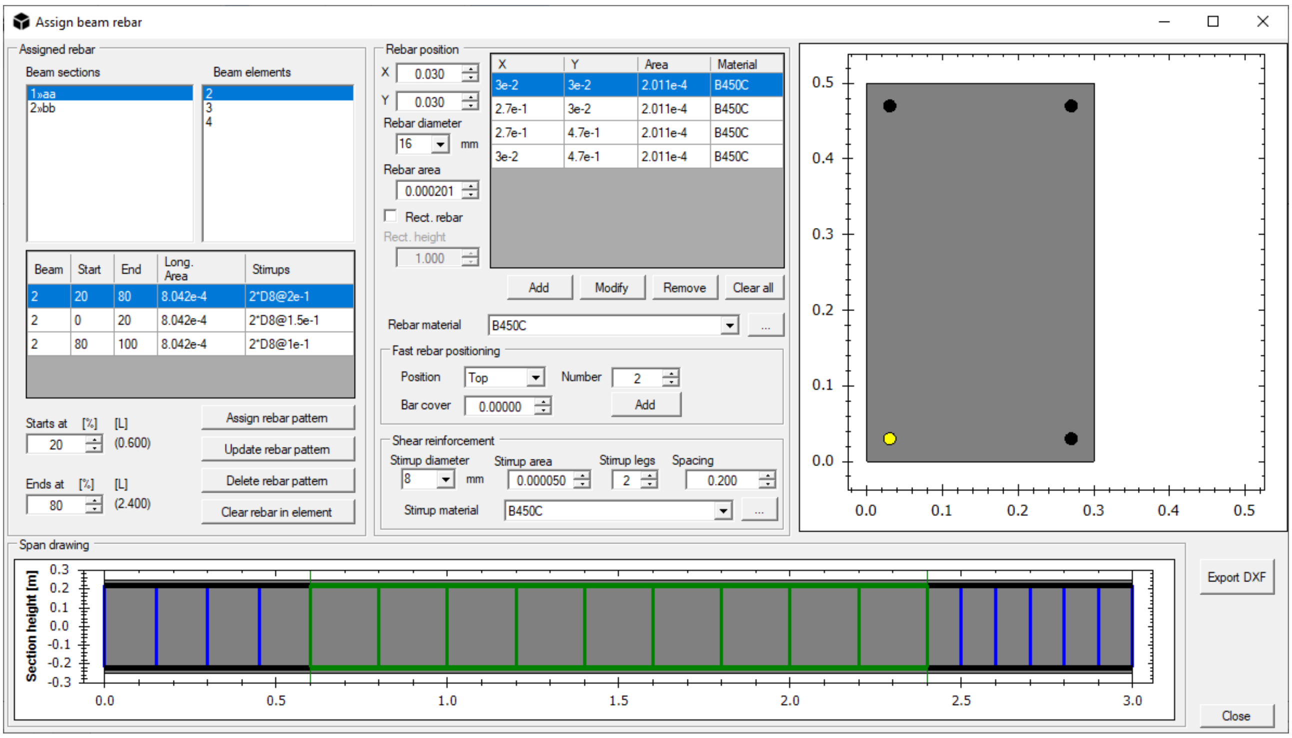
Rebars: Assigns reinforced bars to beam elements. By selecting the desired section from the *Beam sections* box, the potential rebar defined in *Assign/Sections/Strength* is shown and it can be applied to the beams having that section.

The table in the box named *Assigned rebar* shows the longitudinal and shear reinforcements assigned to each beam, or to a portion of a beam by specifying the *Start at* and *Ends at* values.

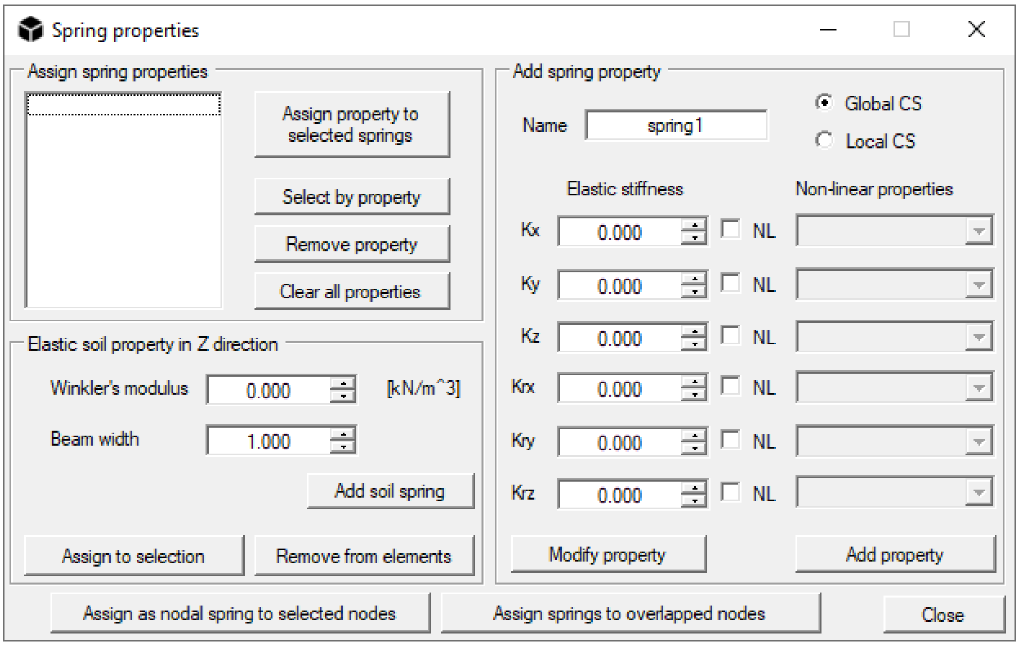
The *Rebar position* box allows to define and change longitudinal and shear reinforcements. Stirrups are defined in the *Shear reinforcement* sub-box. The *Fast rebar positioning* box allows to easily insert simple rebar schemes for the most common sections.

Span drawing shows the longitudinal section of the beam, highlighting the selected part.

⚠ WARNING: units of measure used in this mask are the same defined for the model, except for the diameters of the available bars in the dropdown menus, with are shown in *mm*.



Spring properties: Add, modify or assign one set of properties (elastic stiffnesses) for linear springs. It is also possible to specify and assign subsoil distributed springs on beam elements (as elastic foundations) by using the box *Elastic soil properties in Z direction* by using the button *Add soil spring* after having set the *Winkler's modulus* and the *Beam width*.



The *Assign as nodal spring to selected nodes* button allows to assign to a node the selected spring, connected to another fully fixed node that is automatically restrained.

The *Assign springs to overlapped nodes* command allows to easily assign zero-length springs to the selected overlapped nodes.

It is possible to assign non-linear properties for a translational (x, y, z) or rotational (rx, ry, rz) DoFs, either in global or local coordinates as specified in the upper right corner of the property box. By clicking on the *NL* checkbox, the non-linear properties dropdown will list all the hysteretic laws available. Select one item from the dropdown to set properties.

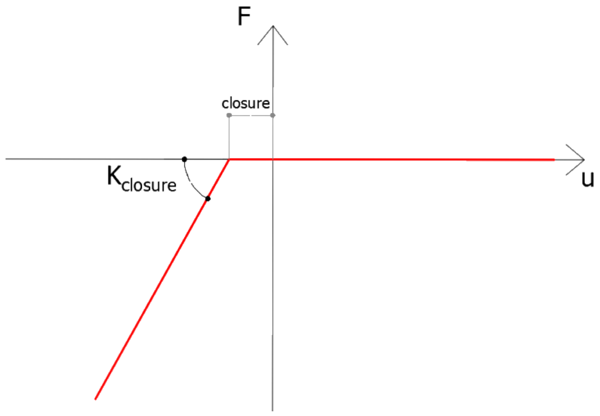
WARNING: in order to define the non-linear properties, it is necessary to add a spring property first. Once the non-linear properties have been defined, click on Modify property to store them.

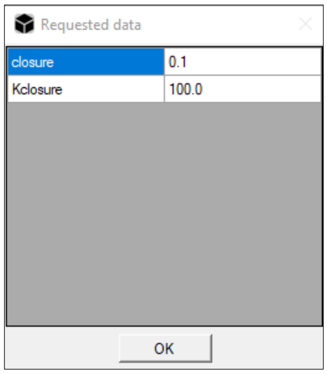
The non-linear models available in NextFEM Designer are:

- Gap: gap spring is particularly useful for representing contact (i.e. gap with null opening). Such model works only in compression with the stiffness defined by the user.

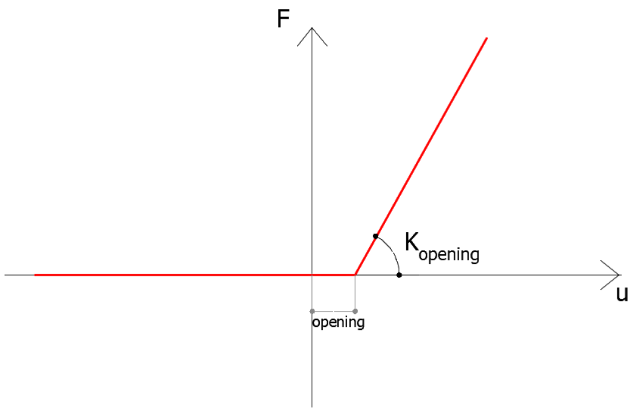
  Data requested for Gap are:

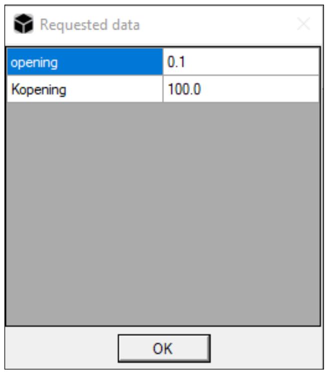
  - *closure*: maximum closure of the gap;
  - *Kclosure*: slope of the compression branch.



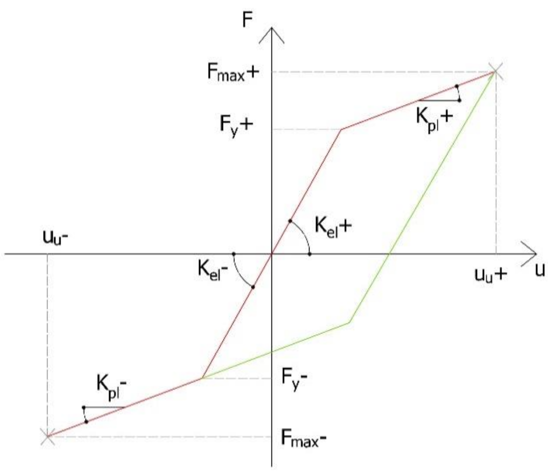


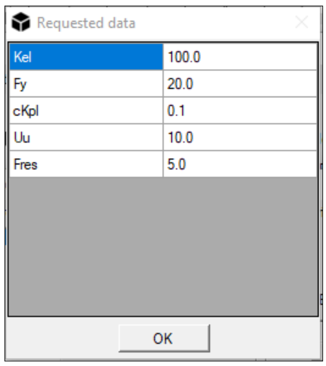
- Hook: this law reacts only in tension. Data requested for Hook are:
    - *opening*: displacement at which tension stiffness is not null;
    - *Kopening*: slope of the tension branch.





- Bilinear Plastic: the elastic behaviour is maintained until the yielding force *Fy* is reached, then the plastic field begins. Once the ultimate displacement *Uu* is achieved, the spring breaks, maintaining a residual force *Fres*. This law has a symmetrical behaviour.

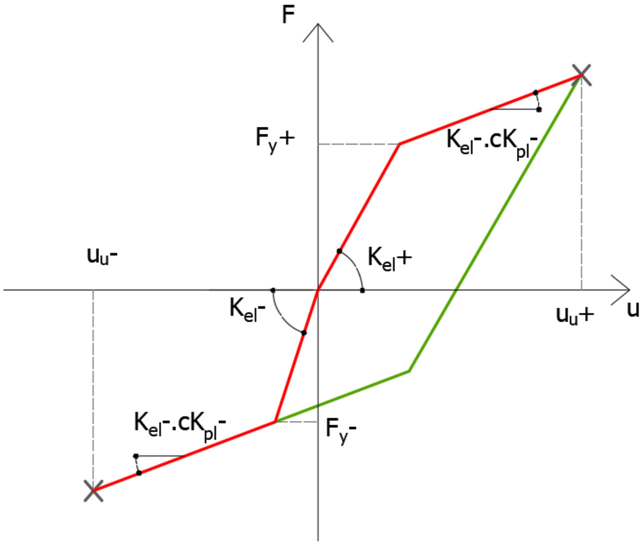




Data requested for a *Bilinear Plastic* behaviour are:

- *Kel*: elastic stiffness assigned to the spring;
- *Fy*: strength at elastic limit;
- *cKpl*: coefficient to set the slope of the plastic branch Kpl=cKpl * Kel;
- *Uu*: ultimate displacement;
- *Fres*: residual strength after collapse.

- Bilinear Unsymmetrical: this behaviour is suitable for modelling non-symmetric behaviour in structural elements. This law has an unsymmetrical behaviour.



Requested data

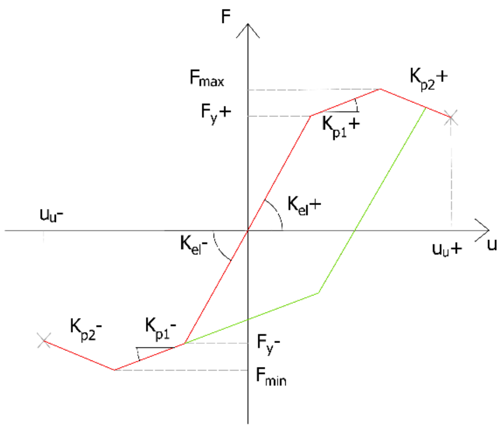
Kel+	100.0
Fy+	20.0
cKpl+	0.1
Uu+	10.0
Fres+	5.0
Kel-	100.0
Fy-	-20.0
cKpl-	0.1
Uu-	-10.0
Fres-	-5.0

OK

Data requested for a *Bilinear Unsymmetrical* type behaviour are:

- *Kel+*: elastic stiffness assigned to the spring;
- *Fy+*: strength at elastic limit;
- *cKpl+*: slope of the positive plastic branch;
- *Uu+*: ultimate displacement allowed;
- *Fres+*: residual force for
- *All the above quantities followed by "-"* are the same properties for the compression side.

- Trilinear Plastic: after reaching the plastic field, the skeleton curve is characterized by an hardening branch bringing to *Fmax*, followed by a hardening/softening one that leads to failure at *Uu*. This law may have or not a symmetrical behaviour.



Requested data

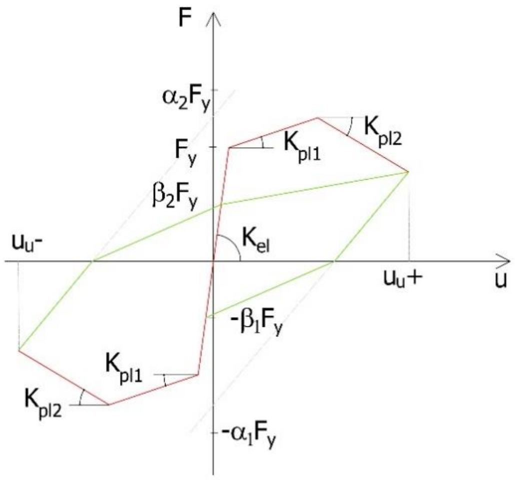
Kel+	100.0
Fy+	20.0
Kp1+	5.0
Fmax	25.0
Kp2+	-5.0
Kel-	100.0
Fy-	-20.0
Kp1-	5.0
Fmin	-25.0
Kp2-	-5.0
Uu+	40.0
Uu-	-40.0

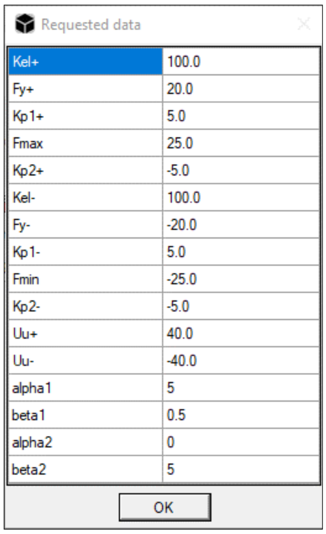
OK

Data requested for a *Trilinear Plastic* behaviour are:

- *Kel+*: elastic stiffness assigned to the spring;
- *Fy+*: strength at elastic limit;
- *Kp1+*: slope of the first plastic branch;
- *Fmax*: maximum strength reachable;
- *Kp2+*: slope of the second plastic branch;
- *Uu+*: ultimate displacement in positive direction;
- *All the above quantities (except Fmax) followed by "-"* are the ones for the compression side;
- *Fmin*: maximum strength reachable in compression.

- <u>Pivot rule</u>: such model follows the rules defined in *Dowell RK, Frieder S, Wilson LE. Pivot hysteresis model for reinforced concrete members. Struct J 1998;95(5):607–17*. The law can be set as not symmetrical. The definitions of the pivot points can be made by setting the α and β parameters.
  Referring to the *Trilinear Plastic* rule, the additional parameters are:
  - *alpha1*: sets the pivot point for unloading from a tensile force to zero;
  - *beta1*: sets the pivot point for reverse loading from zero to a negative force. Must be set between 0 and 1;
  - *alpha2*: sets the pivot point for unloading from a compressive force to zero;
  - *beta2*: sets the pivot point for reverse loading from zero to a positive force. Must be set between 0 and 1.



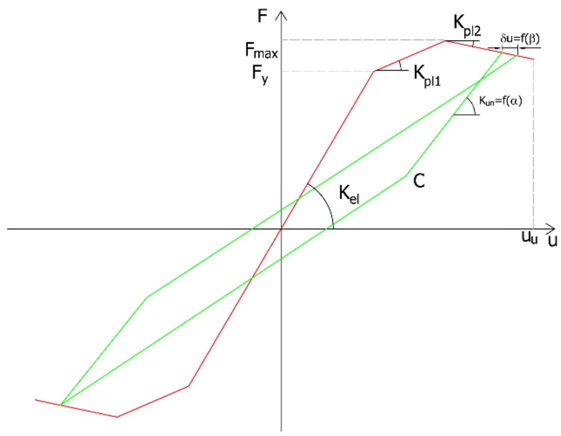


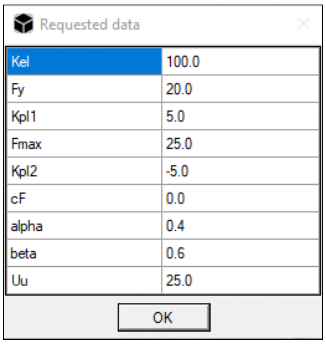
Requested data

Kel+	100.0
Fy+	20.0
Kp1+	5.0
Fmax	25.0
Kp2+	-5.0
Kel-	100.0
Fy-	-20.0
Kp1-	5.0
Fmin	-25.0
Kp2-	-5.0
Uu+	40.0
Uu-	-40.0
alpha1	5
beta1	0.5
alpha2	0
beta2	5

OK

- <u>Tomazevic-Lutman model</u>: represents a trilinear behaviour as defined in *Tomazevic M, Lutman M. Seismic behavior of masonry walls – modeling of hysteretic rules. J Struct Eng 1996:1048–54*. It is influenced by parameters alpha and beta, which determine the stiffness and the strength degradations, respectively. This model is suitable for shear DoFs in masonry walls. This law has a symmetrical behaviour.



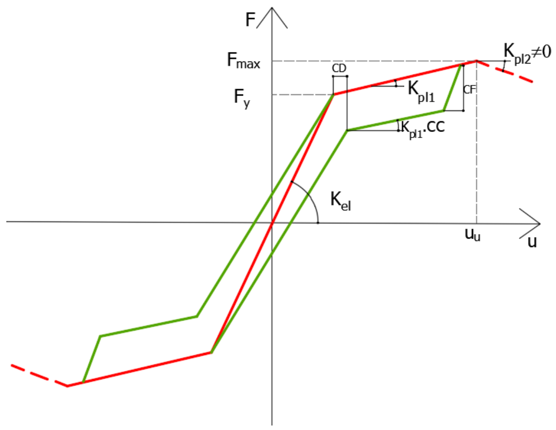


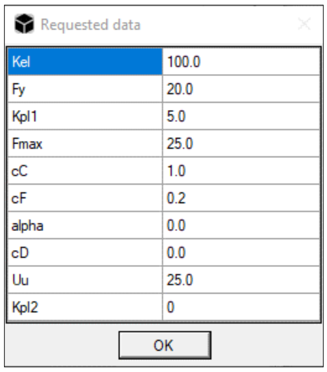
Kel	100.0
Fy	20.0
Kpl1	5.0
Fmax	25.0
Kpl2	-5.0
cF	0.0
alpha	0.4
beta	0.6
Uu	25.0

Data requested for *Tomazevic-Lutman* model are:

- *Kel*: elastic stiffness assigned to the spring;
- *Fy*: strength at elastic (cracking) limit;
- *Kpl1*: slope of the first plastic branch;
- *Fmax*: maximum strength reachable;
- *Kpl2*: slope of the second hardening/softening plastic branch;
- *cF*: coefficient to set unloading force ratio from the backbone;
- *alpha*: coefficient for linear stiffness degradation at unloading (Kult=alpha*Kel);
- *beta*: coefficient for strength degradation on the base of energy dissipated in a cycle (default 0.06);
- *Uu*: ultimate displacement.

- Ring-shape: this model represent a poor dissipative cyclic behaviour and can be used, for example, for flexural DoFs in slender masonry panels. The backbone can be bilinear if *Kpl2* is set to zero.
  This law has a symmetrical behaviour.





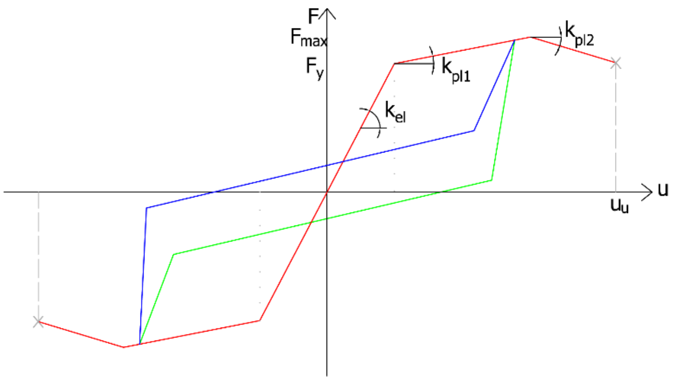
Kel	100.0
Fy	20.0
Kpl1	5.0
Fmax	25.0
cC	1.0
cF	0.2
alpha	0.0
cD	0.0
Uu	25.0
Kpl2	0

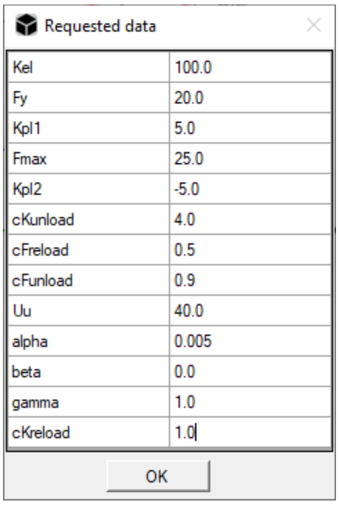
Data requested for the *Ring-shape* model are:

- *Kel*: elastic stiffness assigned to the spring;
- *Fy*: strength at elastic (cracking) limit;
- *Kpl1*: slope of the first plastic branch;
- *Fmax*: maximum strength reachable;
- *cC:* coefficient to set the slope of the unloading path;

- *cF*: coefficient to set unloading force ratio from the backbone;
- *alpha*: coefficient for linear stiffness degradation at unloading (Kult=alpha*Kel);
- *cD*: way point for the unloading path - distance from elastic limit in backbone curve;
- *Uu*: ultimate displacement;
- *Kpl2*: slope of the seconnd plastic branch, can be set to 0 to remove it.

- Slip type: trilinear behaviour exploitable for shear connection in timber structures (nails, screws, angle brackets, ect.). Such law is based on the work *Rinaldin G., Fragiacomo M. A Component Model for Cyclic Behaviour of Wooden Structures. Materials and joints in Timber structures: recent developments of technology, RILEM Bookseries, Vol. 9, pp. 519-530, 2014, DOI: 10.1007/978-94-007-7811-5_48, ISBN: 978-94-007-7811-5*. This law has a symmetrical behaviour.



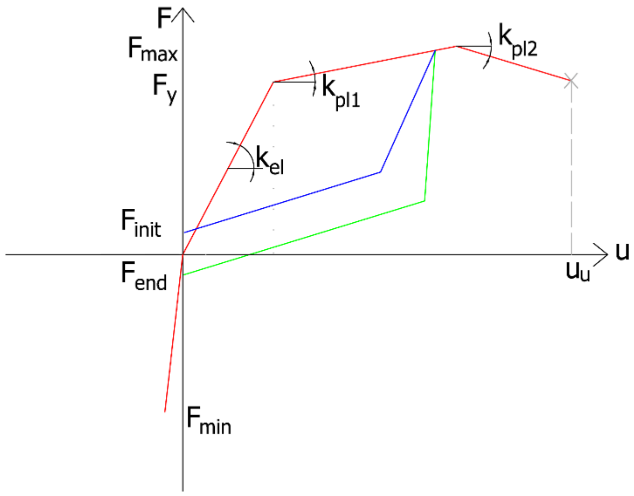


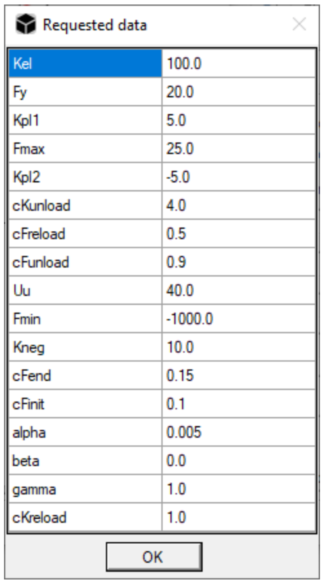
Data requested for the *Slip-type* model are:

- *Kel*: elastic stiffness assigned to the spring;
- *Fy*: strength at elastic limit;
- *Kpl1*: slope of the positive plastic branch, must be ≠0;
- *Fmax*: maximum strength reachable;
- *Kpl2*: slope of the negative plastic branch, must be ≠0;
- *cKunload*: it sets the unloading stiffness of the first branch of unloading (green) and of reloading (blue) path by multiplying the elastic stiffness by this factor;
- *cFreload*: it sets the lower limit of last branch of reloading (blue) and unloading (green) path, in percentage of the force value in the backbone. Such parameter must be set between 0 and 1;
- *cFunload*: it sets the lower limit of first branch of unloading (green) and reloading (blue) path, in percentage of the force value in the backbone. Such parameter must be set between 0 and 1;
- *Uu*: ultimate displacement;
- *alpha*: exponential strength degradation parameter based on dissipated energy;
- *beta*: exponential strength degradation parameter based on maximum displacement reached;
- *gamma*: linear strength degradation parameter;
- *cKreload*: sets the unloading stiffness of last branch of the reloading (blue) and unloading (green) path by multiplying the elastic stiffness by this factor.

- Unsymmetrical Slip-type: this behaviour is suitable for modelling hold-down connections in timber structures. Such law is based on the work *Rinaldin G., Fragiacomo M. A Component Model for Cyclic Behaviour*

of Wooden Structures. Materials and joints in Timber structures: recent developments of technology, RILEM Bookseries, Vol. 9, pp. 519-530, 2014, DOI: 10.1007/978-94-007-7811-5_48, ISBN: 978-94-007-7811-5.
This law has an unsymmetrical behaviour.




Requested data

Kel	100.0
Fy	20.0
Kpl1	5.0
Fmax	25.0
Kpl2	-5.0
cKunload	4.0
cFreload	0.5
cFunload	0.9
Uu	40.0
Fmin	-1000.0
Kneg	10.0
cFend	0.15
cFinit	0.1
alpha	0.005
beta	0.0
gamma	1.0
cKreload	1.0

OK

Data requested for an *Unsymmetrical Slip-type* behaviour are:

- *Kel*: elastic stiffness assigned to the spring;
- *Fy*: strength at elastic limit;
- *Kpl1*: slope of the positive plastic branch, must be ≠0;
- *Fmax*: maximum strength reachable;
- *Kpl2*: slope of the negative plastic branch, must be ≠0;
- *cKunload*: it sets the unloading stiffness of the first branch of unloading (green) and of reloading (blue) path by multiplying the elastic stiffness by this factor;
- *cFreload*: it sets the lower limit of last branch of reloading (blue) and unloading (green) path, in percentage of the force value in the backbone. Such parameter must be set between 0 and 1;
- *cFunload*: it sets the lower limit of first branch of unloading (green) and reloading (blue) path, in percentage of the force value in the backbone. Such parameter must be set between 0 and 1;
- *Uu*: ultimate displacement allowed;
- *Fmin*: maximum strength reachable in compression;
- *Kneg*: stiffness coefficient for compression branch, multiplies the elastic stiffness;
- *cFend*: coefficient for setting the force value at the end of the last branch of the unloading path (green), multiplies the yielding force;
- *cFinit*: coefficient for setting the force value at the beginning of the first branch of the reloading path (blue), multiplies the yielding force;
- *alpha*: exponential strength degradation parameter based on dissipated energy;
- *beta*: exponential strength degradation parameter based on maximum displacement reached;
- *gamma*: linear strength degradation parameter;
- *cKreload*: sets the unloading stiffness of last branch of the reloading (blue) and unloading (green) path by multiplying the elastic stiffness by this factor.

- Dashpot: represents a non-linear damper element accordingly to Kelvin's model. This law is suitable for dynamic analysis, and gives a response defined as:

$$F = C \cdot \dot{u}^{\alpha} + K \cdot u + F_0$$

where the stiffness is

$$K = \frac{1}{\frac{1}{K_r} + \frac{1}{K_0}}$$

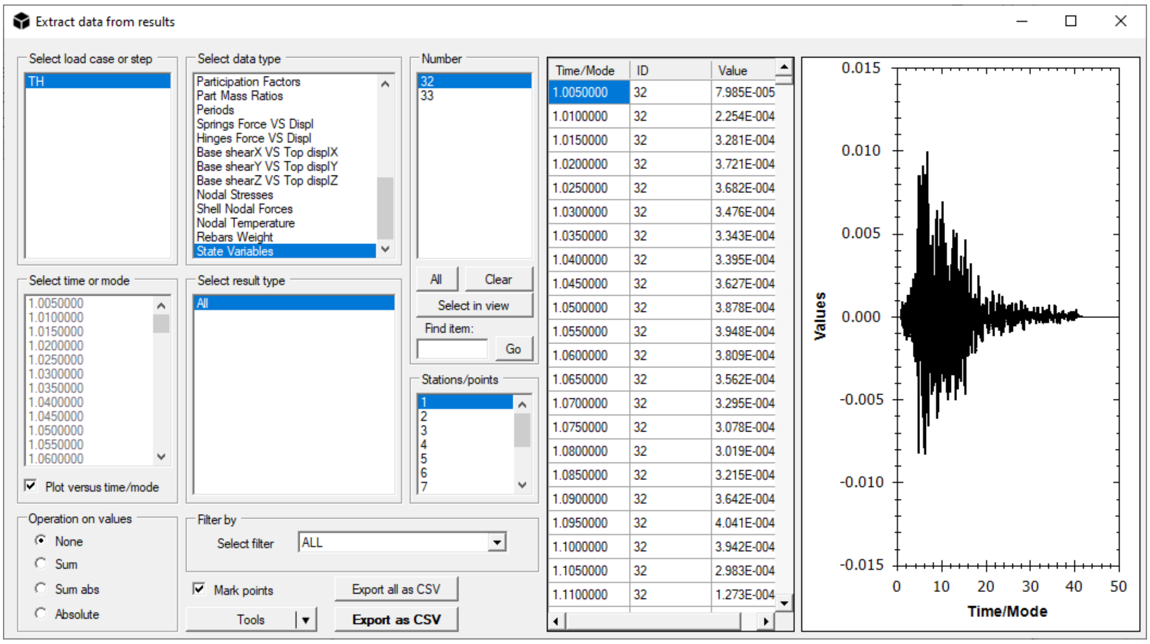
The needed parameters are:

- *Kr*: restoring stiffness of the dashpot;
- *C*: damping coefficient;
- α: damping exponent;
- *K0*: initial stiffness of the dashpot;
- *F0*: preloading force. Below this value, damper is not active.

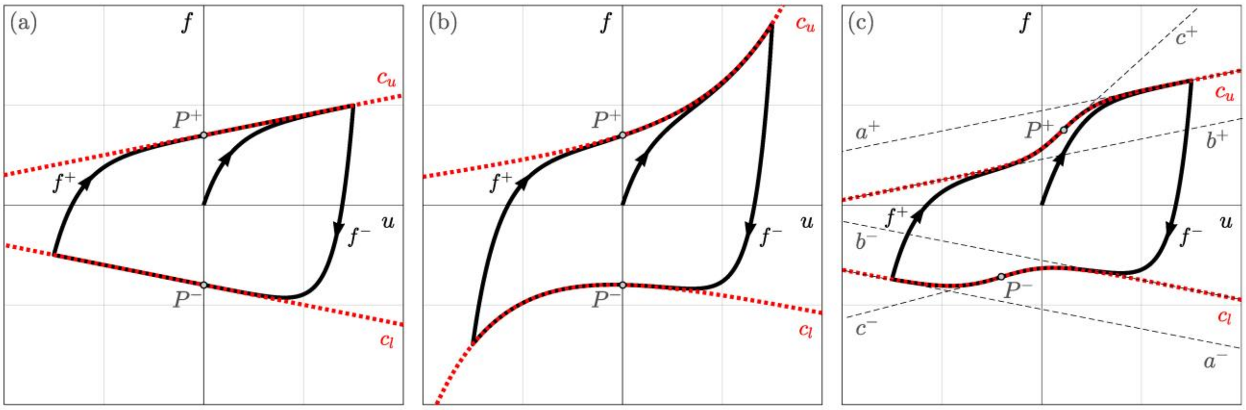
- AMD: represents a linear damper (except for *Fmax*, *Vmax*, and *Dmax* limits) as per Maxwell model (damper and spring in series). If *Fmax* is non-zero, the damper limits the supplied force by behaving elastically-perfectly plastic. *Vmax* and *Dmax* can be used to represent an Active Mass Damper with limits in velocity and displacement. Required parameters are:
  - K: damper stiffness
  - G: damping coefficient
  - α: damping exponent;
  - Fmax: maximum force for the damper
  - Dmax: maximum displacement of moving mass
  - Vmax: maximum velocity of moving mass

State variables are available for this element, which can be consulted in *Extract data* mask, under 'Stations/Points'. They contain, in order:

- s1: speed at i-th step
- s2: force at pitch
- s3: energy dissipated by the damper
- s4: acceleration rel. mass
- s5: mass-related velocity
- s6: displacement rel. mass.



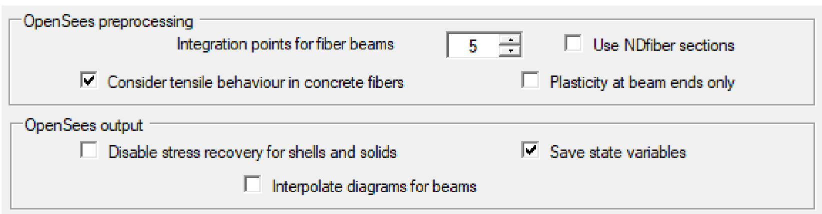
- VRM: simulates a broad class of complex hysteresis loops as described in *Vaiana N, Rosati L (2023) Classification and unified phenomenological modeling of complex uniaxial rate-independent hysteretic responses. Mech Syst Sig Process 182: 109539* and in *Vaiana N, Rosati L (2023) Analytical and differential reformulations of the Vaiana-Rosati model for complex rate-independent mechanical hysteresis phenomena. Mech Syst Sig Process 199: 110448.*



The data required by the Vaiana Rosati model are:

- *kb+:* angular coefficient of the upper boundary line c_u (Fig. a)
- *f0+:* ordinate of the point P^+ (Fig. a)
- *alpha+:* parameter that adjusts the curvature of the load curve f^+ (Fig. a)
- *beta1+:* parameter that transforms c_u into a curve without an inflection point by adjusting its curvature (Fig. b)
- *beta2+:* parameter that transforms c_u into a curve without an inflection point by adjusting its curvature (Fig. b)
- *gamma1+:* parameter that transforms c_u into a curve with inflection point P^+ by adjusting the distance between a^+ and b^+ (Fig. c)
- *gamma2+*: parameter that transforms c_u into a curve with inflection point P^+ by adjusting the slope of c^+ (Fig. c)
- *gamma3+:* abscissa of the inflection point P^+ (Fig. c)
- *umax:* positive ultimate displacement
- All previous parameters followed by "-": the same properties for the unloading phase
- *umin:* negative ultimate displacement.

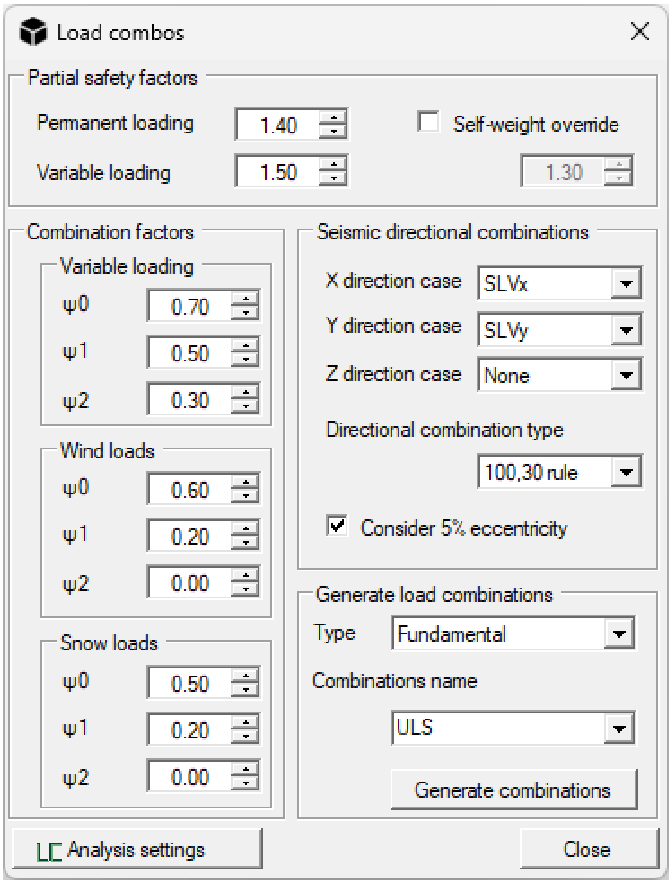
WARNING: for the use in OpenSees of non-linear springs, see www.nextfem.it/it/opensees/ to install the OpenSees solver. Also set the desired options in the boxes in the *Options* mask, tab *Misc.*



LCB *Generate combinations*

This command allows to the safety factors and the partial coefficients for the automatic generation of load combinations as per Eurocode.

The *Self-weight override* option permits to apply a different coefficient to self-weight loadcase, as requested by some codes (e.g. Italian regulations).

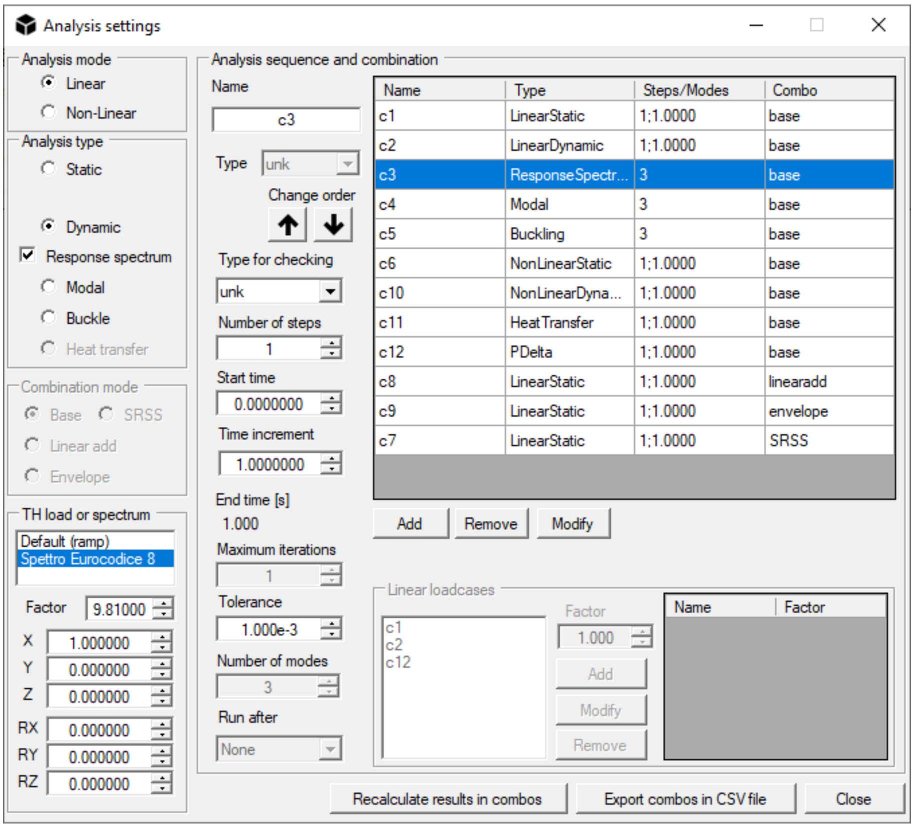


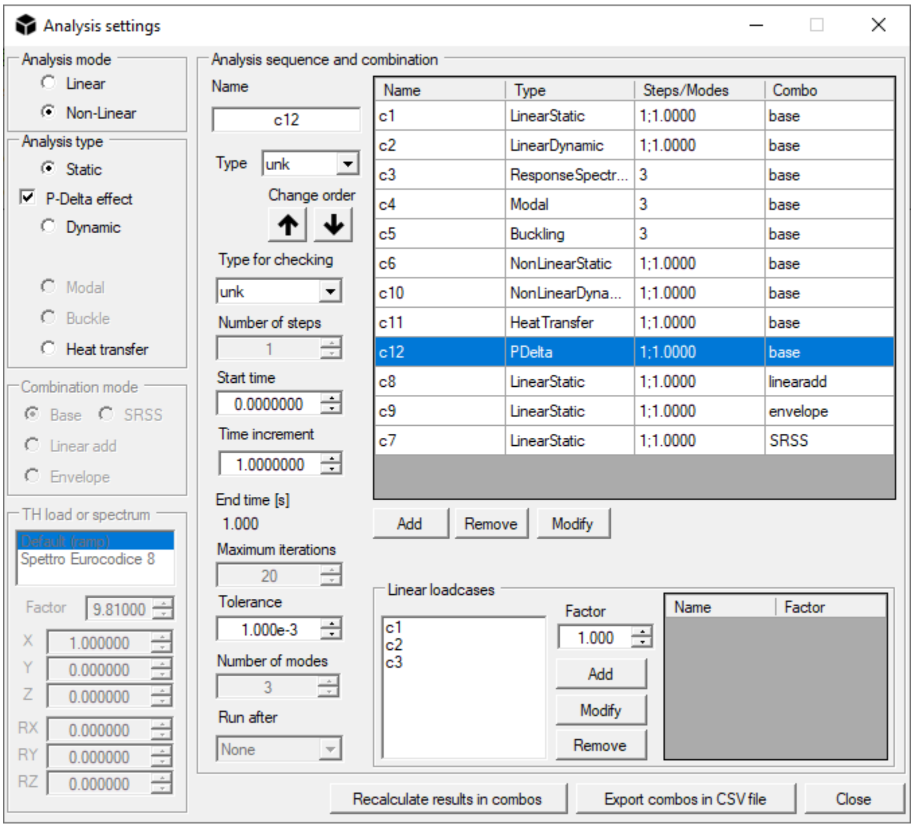
It is possible to generate fundamental (ULS resistance), service (SLS) or seismic combinations, including the direction combination of 2 or 3 seismic cases specified in the *Seismic Directional Combinations* box. It is possible to prefix the name of the combination, which is particularly useful when writing seismic combinations referring to different spectra (e.g. LLS and DLS).

The option *Consider 5% eccentricity* enables the assignment of equivalent torques to the plane forces in the dedicated cases named *exc5x and *exc5y, which are automatically generated and combined. For an analysis with a response spectrum in the X and Y directions and with active eccentricities, 32 combinations are thus generated instead of the 8 without the option.

The calculated combinations can be edited from the *Analysis settings* command.

LC *Analysis settings:* Defines the type and the sequence of the analyses. The combinations can be defined adding linear load cases to a defined case. To add an analysis, select an *Analysis mode*, an *Analysis type* and, if needed, a *Combination type*. Then click on *Add*.





The program allows the following types of linear and non-linear analysis:

- Static
- Dynamic

- Response spectrum analysis (user must check *Dynamic* first, and the option *Response spectrum*. Hence, a design spectrum can be selected from the *TH load or spectrum box*)
- Modal, to find structural periods and eigen-vectors of a model;
- Buckle, to find the buckling modes of a model;
- Heat transfer, to analyze models in XY plane and to get thermal maps (for example from a section);
- P-Delta, for static analysis including second-order geometrical effects. Such kind of analysis are carried-out iteratively, but in only one loading step, and results can be used in combinations. To include second order geometrical effects in multi-steps non-linear analyses, flag the option named "2nd order".

In the *Combination mode* box, user can choose the type of combination to be performed:

- *Base* for all the linear static load cases
- *Sum* for the load combinations in which the cases specified in the *Linear loadcases* box have to be summed;
- *Envelope* for the load combinations in which the cases (or combinations) specified in *the Linear loadcases* box have to be enveloped;
- *SRSS* to perform a Square Root of the Sum of Squares of the cases (or combinations) specified in the *Linear loadcases* box.

From the *Type* drop-down menu in the *Analysis sequence and combination* box, the type of each loadcase or combination can be chosen, for using in checking. If this has not been specified, checking could not be run for the specific combination.

With the command *Export combos in CSV file*, a CSV file containing all the combinations and their combination coefficient can be obtained in a tabular format.

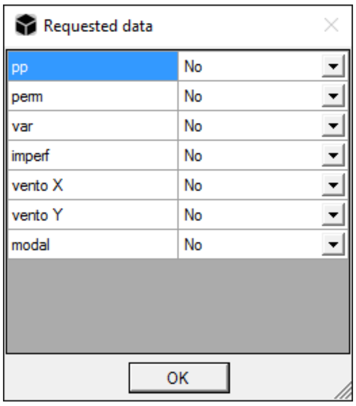
For dynamic analyses, damping can be specified in terms of Rayleigh's coefficients, while for the Response Spectrum analysis damping ratio of the spectrum employed is required.

Finally, for applying a displacement history (as defined by imposing displacement) in non-linear analyses, set *Factor* to 0.

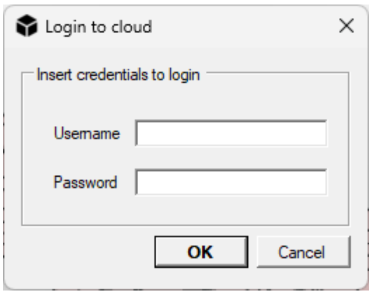
2.8. Tools menu

Run: This option allows to run all the analyses.

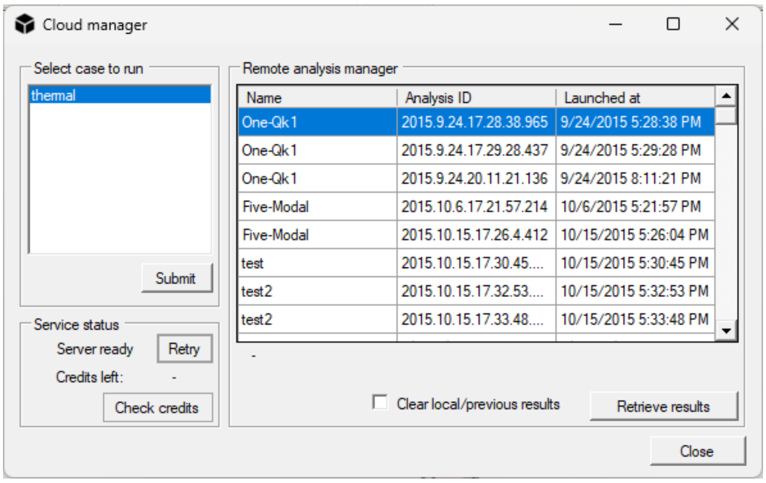
Run selected: This option allows to run the selected analyses. To exit without running anything, set all to *No* and press *OK*.



Login to cloud: This option allows to run analyses on the NextFEM cloud by connecting to the NextFEM server using the user's credential. To log into the cloud insert your *Username* and your *Password* obtained from the registration on the NextFEM Internet site.

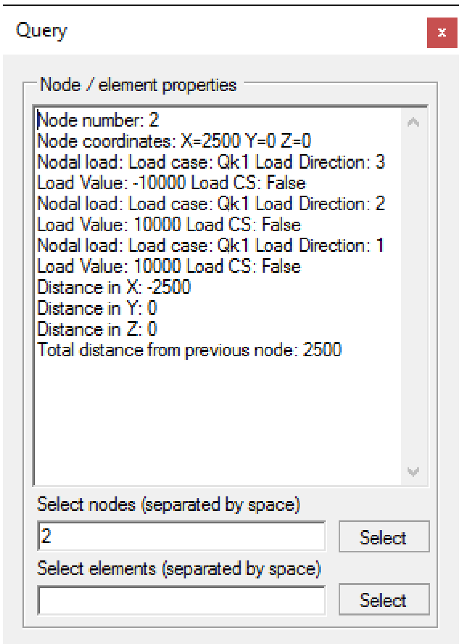


Once connected to the cloud it is possible to launch analyses by selecting them from the *Select case to run* list and then click on the *Submit* button.



Once the analysis has been submitted, it will be completed in the cloud. When the analysis is completed it is possible to view the result at any time clicking on the *Retrieve results* button.

Query: Provides information about the selected element/node. By picking 2 nodes, it is possible to measure the distance between them.

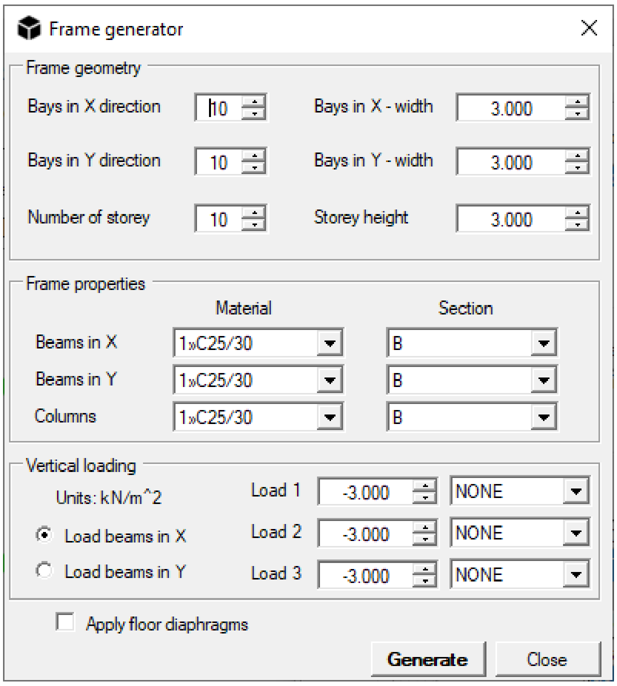


Screenshot: Copies to the clipboard the current viewport

Model wizard: it allows the guided setting of a new model through the definition of materials, sections, load cases and masses.

Frame generator: This command generates automatically a regular spatial frame. The input data are:

- The number of bays along X and Y, the number of story and their span;
- The material and the transversal section to be assigned to the beams and columns
- OPTIONAL: the loads to be assigned to beams by choosing the roof framework direction. Target load cases must already be defined to proceed;
- OPTIONAL: you can apply automatic floor diaphragms.

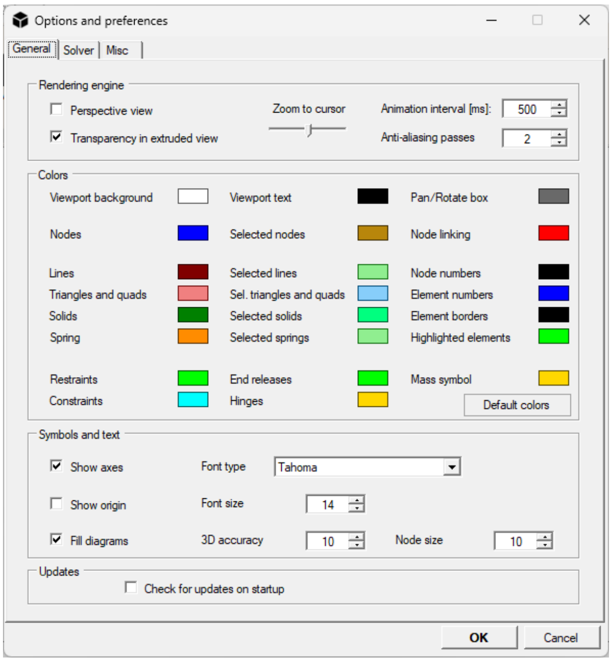


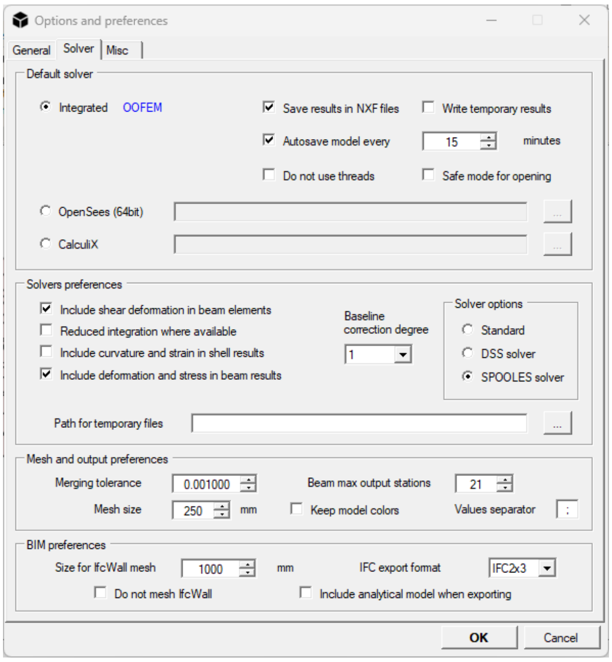
Section from model: It allows a section to be created from a model in which the thermal analysis has been carried out in the XY plane. The command then asks for certain parameters for choosing the functions for fire testing (fire curve, material degradation curves, etc.).

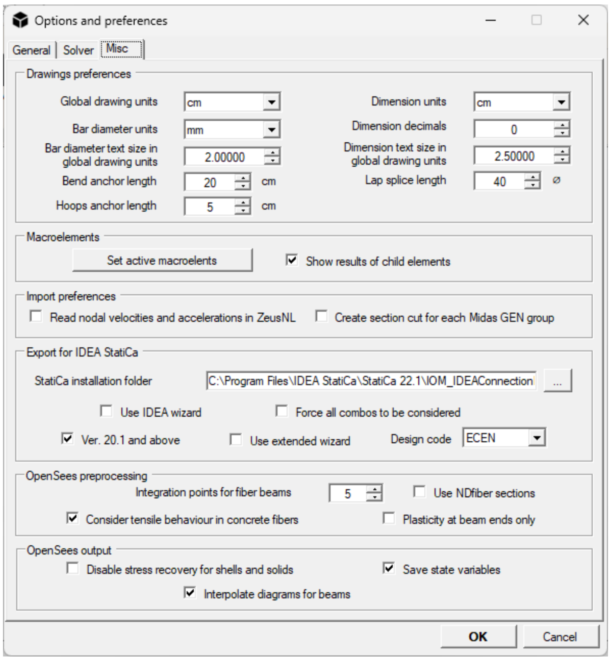
Add plinth: Adds a shell-modelled plinth on elastic soil to the selected node. The restraints of the node are overwritten with a constraint compatible with the elastic soil (X, Y and RZ). The command asks as input the plan dimensions of the plinth, the plinth thickness and material, and the elastic soil type which must already be defined.

Options: It allows to change the visualization options as long as the solver options.

To change from the default solver (OOFEM) to OpenSees the user must enter the *Path to OpenSees.exe* box.; to use the solver CalculiX the user must enter the *Path to ccx.exe.*







WARNING: OpenSees and CalculiX solvers are not fully supported, and not distributed by NextFEM.

The option *Save results in NXF files* allows to save the results of a model in the NXF file, only if the calculations are performed with OOFEM.

The option *Autosave model every n minutes* allows to automatically save the model every n minutes as specified by the user.
The option *Include curvature and strain in shell results* allows to get curvatures and strains in results for shell elements. Enabling this option may slow down the analysis and increase the model weight.
The option *Solver* allows the user to choose the default solver:
- *Standard*: it's the standard solver which allows to detect labilities in the model;
- *DSS solver*: Direct Sparse Solver, optimized for fast solving of large problems. Labilities will not be detected;
- *SPOOLES solver*: SParse Object Oriented Linear Equations Solver, optimized for fast solving of large

problems. Labilities will not be detected.

Path for temporary files shows the temporary folder used by the solver.

OOFEM Preferences

The option *Include shear deformation in beam elements* activates the Timoshenko contributions for beams in the default solver.

The option *Reduced integration for 4-node shells* activates the reduced integration for 4-node *quad* elements in the default solver.

ADAPTIC Preferences

The option *Read nodal velocities and accelerations from NUM files* enables the reading of velocities and accelerations in results import from ADAPTIC NUM file.

BIM Preferences

The option *Size for IfcWall mesh* allows to specify the mesh size to be used during import of IfcWall elements from IFC or IFCxml files.

General preferences

It allows to set the tolerance for merging nodes and to set the separator for CSV export.

2.9. Plugins

This menu section lists all the plugins loaded at startup. Plugins are in English language only, and they are completely independent from Designer. For further information: www.nextfem.it/dev/

Some plugins are included by default in NextFEM Designer, such as:

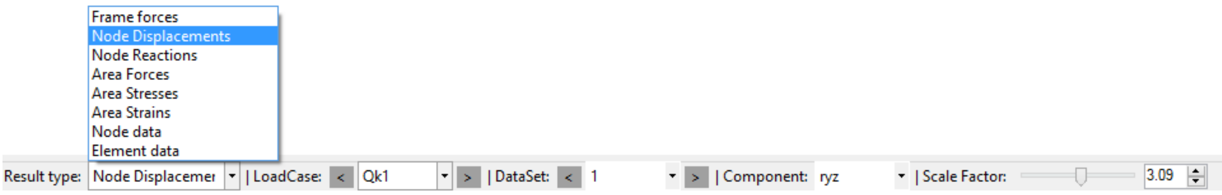
Floor modeller, to transform a loading plane to a shell plane loaded on surface;

Custom report compiler, to edit a DocX document by including data and images from model;

Parametric solver to carry-out parametric analyses in an easy way.

2.10. Results and Cheking menu

Display results: It shows the result of the analysis by load. To show the appropriate *Result type, LoadCase; DataSet* and *Component*, select them from the respective drop-down lists.



Available results can vary on the base of the performed analysis, and can be:

- *Frame forces* to show beam diagrams
- *Frame stresses* to show a contour of beam generalized stresses
- *Frame strains* to show a contour of beam deformations
- *Frame Displacements* to show a contour of beam deflections
- *Frame subsoil reactions* to show a contour of the soil reactions for beams with subsoil springs
- *Node Displacements* to show the deformed shape and a contour of nodal displacements
- *Node Velocities* to show the deformed shape and a contour of nodal velocities
- *Node Accelerations* to show the deformed shape and a contour of nodal accelerations
- *Node Reactions* to show the nodal reactions in restrained nodes as arrows
- *Area Forces* to show forces per unit of length in planar elements
- *Area Stresses* to show stresses in planar elements

- *Area Strains* to show deformations in planar elements
- *Solid Forces* to show solid elements forces
- *Solid Stresses* to show solid elements stresses
- *Solid Strains* to show solid elements deformations
- *Node data* to show node-related data such as checking ratios or custom values
- *Element data* to show element-related data such as checking ratios or custom values
- *Area temperature* to show the thermal map of planar elements
- *Solid temperature* to show the thermal map of solid elements (not yet available).

Animation: Shows animation of deformed shapes through time or exports a video on the selected result.

Extract data: Extracts results data in tabular format.

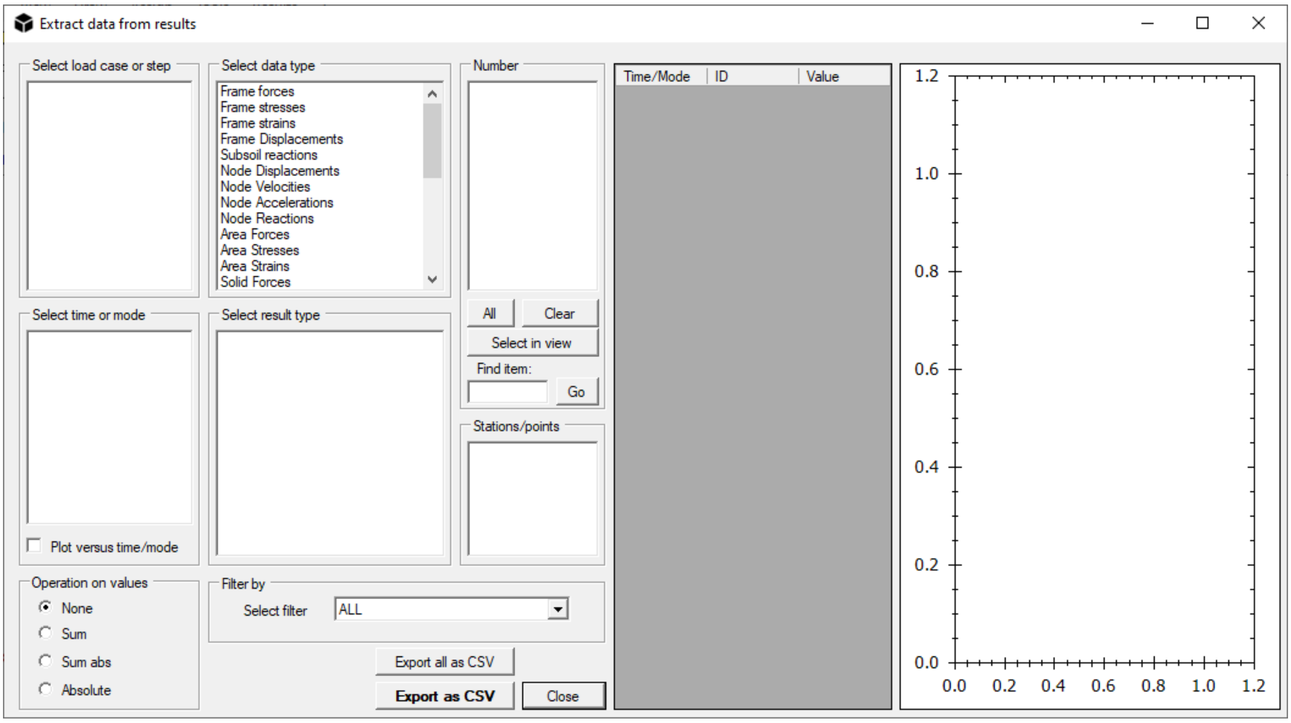
To extract the result follow the following procedure:

- Select a load/step by clicking on it in the *Select load case or step* list;
- Select a time or mode by clicking on it in the *Select time or mode* list;
- Select a data type, so that a result type from the *Select data type* list;
- Select the number of the node or element in the *Number* list.
- Optionally, choose the number of stations/points in the *Stations/points* window.

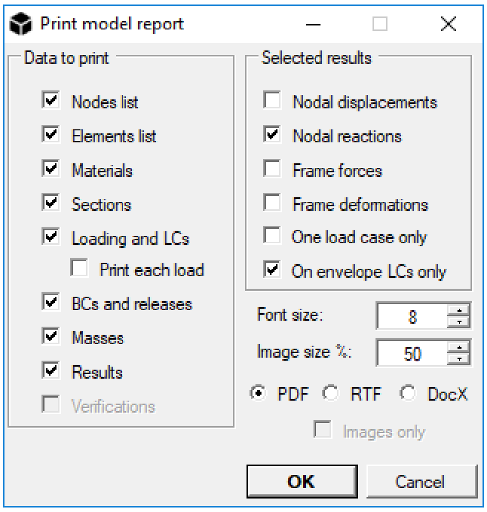
To export the shown data in a tabular format, press *Export as CSV* button. The command *Export all as CSV* allows to export all the data of the selected type (in the *Select data type* box) for all elements or nodes.

The *Operation on values* box allows to apply simple orations (sum, sum of absolute values, absolute values) on the requested (multiple) data.

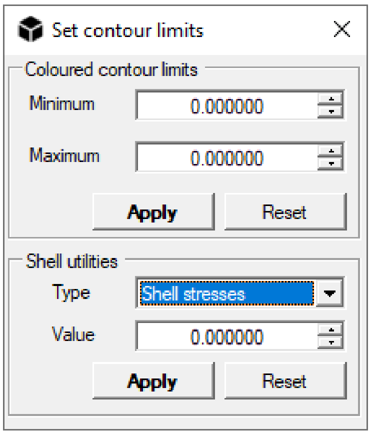
The *Filter by* box allows to display only data associated to selected materials, sections, etc.



Model report: Produces a model report in PDF, RTF or DocX formats. Tick the proper items to export them.

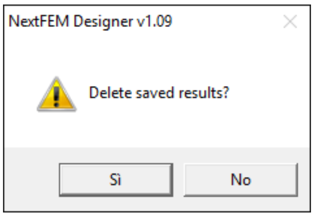


Set contour limits: It allows to set the limits in the current displayed contour plot. This mask contains also commands to plot the isocurve of a certain values for some selected shell results (stresses, forces, temperature).



⚠ WARNING: The contour limits affect the colours associated to the extreme values of contour. Any value exceeding the limits will be associated with the colour of the nearest extreme limit (red or blue).

Delete saved results: Allows the deletion of the saved results in the NXF file.



Delete saved checks: Allows the deletion of the saved checks in the NXF file.

Σ *Sum of loads and masses*: produces a report on the sum of reactions in Z direction for each base loadcase, reporting also the summation of the masses present in the model.

⚠ WARNING: for wind loadcases, the returned value is the highest sum amongst the X, Y and Z directions, calculated on reactions. For all the other cases, applied loads are summed.

Σ *Bill of materials*: produces a report on the sum of volume and weight for each material in the model, ordered also by sections.

More output stations: allows continuous diagrams to be displayed by increasing the number of stations employed.

⚠ WARNING: Additional stations are not saved in the file. The command resets when saving the model or opening.

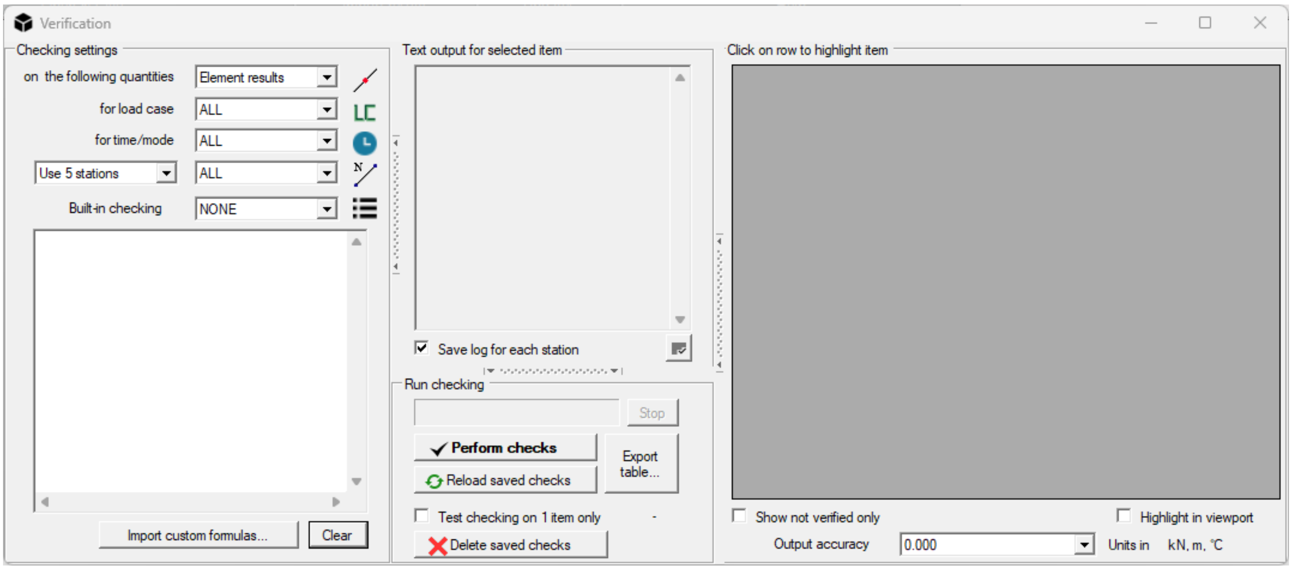
Verifications: It allows to handle and checks some defined quantities on nodes/elements.

To check nodes/elements, the following procedure has to be done:

- choose a quantity to check from the *quantities* drop-down list;
- Choose a load case and a time/mode (or select ALL for all times/modes);
- Import the formulas from a customized **.txt* file (see chapter 4);
- Click on the *Perform checks* button.

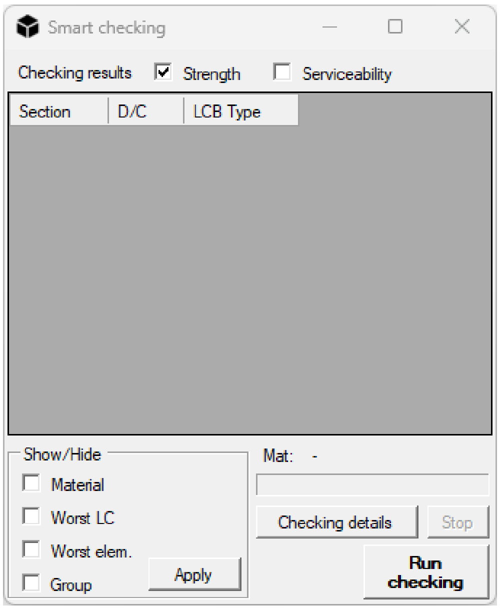
The results are shown on the output area in tabular and textual format, if required with the *"Save log for each station"* check. They can be exported in text format by clicking on the *Export table…* button.

Finally, the areas 'Verification Settings', 'Textual Results on Selected Object' and 'Launch Verification' can be hidden by pressing the grip button on the respective border.



Please refer to the dedicated chapter for a full reference of the *Verification* tool.

Smart checking: allows to automatically select the proper verification on the base of the element material. In addition, displays checking results filtered by transversal section.

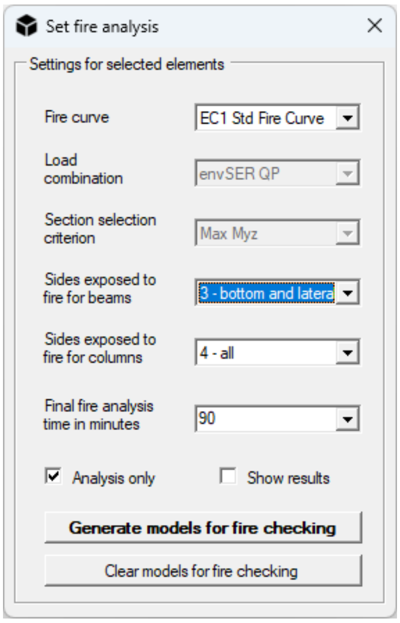


WARNING: the selected codes for checking for each material are the same associated to the keys combinations CTRL+1 – 7.

Fire checking: the command starts the first phase of the fire checks for the frame structures on the selected beam elements.

In the window that appears you can select:
- *Analysis and design curves*: select Eurocode 2 for structures in reinforced concrete, Eurocode 3 for steel, Eurocode 9 for aluminium;
- *Load combination*: select the exceptional load combination to use for quick verification;
- *Section selection criterion*: the section used for verification must meet the selected criterion (maximum moment My, maximum moment Mz, maximum in both directions Myz)
- *Sides exposed to fire for beams*: the sides considered as exposed for the thermal analysis for the horizontal or sub-horizontal elements;
- *Sides exposed to fire for columns*: the sides considered as exposed for the thermal analysis for vertical elements;
- *Final fire analysis time in minutes*: select the analysis time for which the resistant check of the section will be conducted.

The *Run all analyses* option allows you to launch the files created in the same folder as the current model, saving the results for each. The *Show Results* option displays the 500°C isotherm for each thermal analysis in a separate window.



The command returns a table containing the demand / capacity ratios of the analysed sections in the *Check-NMM* (pressure and bending) and *Check-V* (shear) columns. The images relating to the resistant verification are saved in the same folder as the model (representation of the section and neutral axis, reduced resistant domain due to the effect of fire).

Overturning check: executes the overturning check on the whole model, assuming it as a rigid body. This command calculates the overturning and stabilizing contributions for each load present in the model (except for thermal loads and distortions) and produces their summation.
The check is conducted with respect to rotation axes in XY plane, passing through the nodes bounded in Z direction. The program can select automatically the 4 peripheral nodes or the user can specify such nodes. It is possible to specify a rotation angle, defined in XY plane, for the rotation axis (default 0° = X global axis).
The checking is carried out automatically for the specified axis and its normal axis, producing a table with each load case, containing base cases and combinations of type *linear add*, showing the overturning moment. If this is negative, the configuration is stable. Otherwise, the structure is not.

> WARNING: the overturning check is performed with respect to the only loads applied as forces, concentrated of uniformly distributed. Applied moments and reactions are always neglected.
>
> WARNING: linear distributed loads are not supported. The loading resultant is always assumed in the middle of the loading span.
>
> WARNING: the automatic algorithm for the seeking of nodes looks for all the nodes restrained in Z direction and assumes their maximum obstruction. Always check the position of the nodes restrained in Z direction; if you're in doubt please proceed by manually selecting the reference nodes.

Seismic capsizing of masonry walls: performs verification of the tilting kinematics of user-selected *Wall* or *MasonryWall* elements. The tilt direction is provided by the Z axis of the elements - to change it use the *Assign / Rotate Local Axes / Reverse Connectivity* command. Refer to the Wall element tutorial for more information. The command requires a license for the *MasonryCheck* module. They are calculated:

Participation factor

$$e_p = \frac{W \cdot d_1 + P \cdot d_2}{(W + P) \cdot \left(W \cdot {d_1}^2 + P \cdot {d_2}^2\right)}$$

Mobilization acceleration

$$a = \frac{a_g \cdot g \cdot S}{q}$$

Collapse multiplier

$$\alpha_0 = \frac{M_s}{M_r}$$

Mechanism-activation acceleration

$$a_c = \frac{\alpha_0 \cdot g}{e_p \cdot FC}$$

With W own weight and d1 distance from hinge, P permanent and d2 distance from hinge, FC Confidence Factor, Ms stabilizing moment and Mr overturning moment.

IDEA Check Manager: this command executes IDEA StatiCa® Code Check Manager on selected nodes and/or elements.

WARNING: in order to work properly, the option "StatiCa installation folder" under Options must be correctly set.

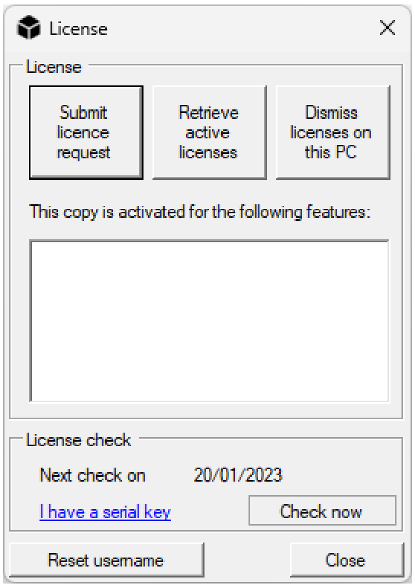
2.11. Help menu (?)

Help: Opens the manual

Check for updates: Checks if the program is updated. The sub-command *Check for minor updates…* allows to select further updates with minor changes and corrections.

Language: You can choose the interface to be in English or Italian. You must restart the program for the changes to take effect.

License: Manage license requests.



About: Credits and licenses.

3. Import/export features

3.1. Import

OpenSees

Pre-processing features:

- Nodes
- UniaxialMaterial (partial reading)
    - elastic
    - elasticPP
    - elasticPPgap
    - ENT
    - Concrete01
    - Concrete02
    - Concrete03
    - Steel01
    - Steel02
- NDMaterial
    - ElasticIsotropic
- Section (partial reading)
    - Fiber
    - PlateFiber
    - LayeredShell
- Elements
    - ShellMITC4
    - quad
    - bbarQuad
    - enhancedQuad
    - quadUP (both)
    - ShellDKGQ
    - ShellNLDKGQ
    - ShellNL (treated as 8-node shell)
    - ElasticBeamColumn
    - ElasticTimoshenkoBeam
    - ForceBeamColumn
    - DispBeamColumn
    - beamWithHinges
    - truss and trussf
    - corotTruss
    - rotspring2dir
    - zeroLength
    - zeroLengthND
    - zeroLengthSection
    - stdBrick
    - bbarBrick
    - Brick8N
    - brickUP

    - Brick20N
    - 20_8_BrickUP
- Fix, fixX, fixY, fixZ Commands
- GeomTransf Commands
- Uniform Beam Loads
- Nodal Loads

Post-processing features:

- Displacements recorder
- Reactions recorder
- Modal Eigenvectors recorder
- Custom spring data

⚠ WARNING: Import from OpenSees could be incomplete. Please always use recorder as complete as possible in DoFs and objects (ex. "recorder Node -file disp.out -time -nodeRange 0 124 -dof 1 2 3 4 5 6 disp").

Midas GEN®

Pre-processing features:

- Nodes
- Elements
    - Beam
    - Truss
    - Tension-only truss
    - Compression-only truss
    - Plate
    - Wall
    - Wall-opening
    - Elastic link
    - NL-link
- Frame Sections
    - Rectangular
    - Circular
    - SRC (EPC only)
    - Tapered
    - H, C, L, 2C, cold-formed C, T, reversed T, box, 2L
- Section color (see Options)
- Thickness color
- Plate and Wall sections (thickness)
- Materials
- Member assignments
- Story
- Constraints
- Groups
- Local axes
- Surface springs
- Frame releases and offsets
- Static load cases
- Load combinations
- Distributed beam loads
- Point loads
- Beam rebar
- Column rebar

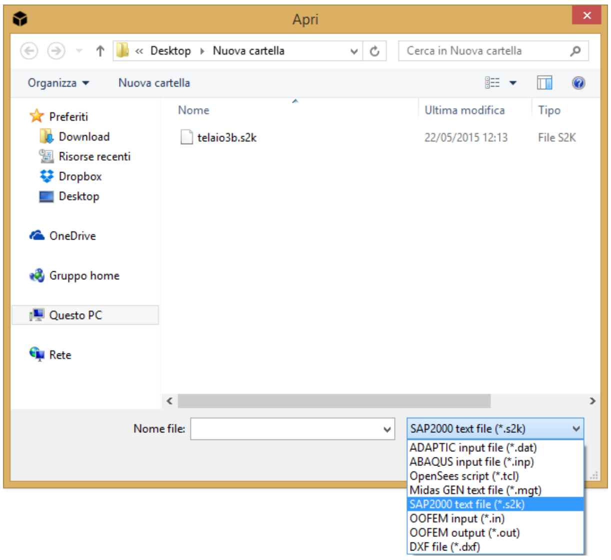
- Wall rebar
- Meshed-slabs rebar (basic rebar only)

Post-processing features:

- Beam forces from table
- Wall forces from table
- Truss forces from table
- Elastic and general link forces from table
- Nodal displacements from table
- RS nodal forces.

SAP2000®

To import a model from SAP2000®, click on the *Import* option in the *File* menu and choose the *SAP2000 text file (*.s2k)* option from the *file type* drop-down list.



The supported pre-processing features are:

- Joint coordinates
- Joint local axes assignments (typical and advanced)
- Joint restraint assignments
- Frame section properties
- RC columns and beams frame properties
- Non-prismatic section properties
- Section designer properties: box/tube and plate, shape solid rectangle, shape polygon and single reinforcing bars
- Cable section definitions
- Frame connectivity
- Link connectivity
- Cable connectivity
- Area connectivity
- Solid connectivity
- General frame section properties
- Area section properties
    - Rectangular
    - circular
- Frame release assignments: total and partial
- Frame section assignments

- Frame local axes assignments
- Area section assignments
- Area local axes assignments
- Basic mechanical material properties
- Material properties for concrete data
- Material properties for rebar data
- Groups definition and assignment
- Joint constraints
- Joint loads
- Frame distributed loads
- Frame temperature loads
- Cable loads
- Load case definitions
- Load patterns definitions
- Modal case
- Functions: response spectrum and time-history from file
- Added mass assignment
- Combinations

Post-processing:

- Modal periods and frequencies
- Joint displacements
- Joint reactions
- Frame element forces
- Shell element forces (averaged on nodes)
- Shell element stresses
- Section cuts forces (analysis and design types)

OOFEM

Pre-processing features:

- Analysis type
- Node
- Rigidarmnode
- Beam3D
- Truss3D
- quad1mindlinshell3d
- mitc4shell
- tr_shell01
- tr_shell02
- LSpace
- LTRSpace
- QSpace

Post-processing features:

- Node displacements
- Node reactions
- Element forces and moments
- Element strains
- Element stresses

ADAPTIC and Zeus-NL

Pre-processing features:

- Materials
- Groups
- Structural Nodal Coordinates (with or without repetitions)
- Non-structural Nodes (with or without repetitions)
- Element connectivity (with or without repetitions)
  - CBP3
  - CBP2
  - LNK3
  - JEL3
  - IN16 (partitioning supported)
  - BK20 (partitioning supported)

Post-processing features for NUM files:

- Nodal displacements
- Nodal velocities (disable by default, check Options)
- Nodal accelerations (disable by default, check Options)
- Reactions
- Frame diagrams for cbp2, cbp3, lnk3 (in global coordinates), jel3.

ABAQUS® and CalculiX

Pre-processing Import features:

- Nodes
- Element
  - B22
  - B23
  - B31
  - B33
  - S3
  - CPS3
  - S4
  - S4R
  - CPS4
  - C3D4
  - C3D10
  - C3D8
  - C3D20

Post-processing features:

- Not available yet

Dxf drawing

Pre-processing features:

- Points
- Line elements

⚠ WARNING: The correctness of the results is not guaranteed. Check carefully your model after import.

Straus7®

Pre-processing features:

- Units
- Nodes
- Beam
- Beam3

- Tri3
- Quad4
- Hexa8
- Beam rotation angles
- Restraints
- Rigid links
- Master-slave links
- Masses
- Beam sections (rectangular, round, I, angle)
- Shell sections
- Nodal forces
- Global distributed forces on beams
- Local distributed forces on beams

BIM files – IFC and IFCxml

Import of BIM models is supported for IFC and IFCxml formats. During import, IfcWall elements are automatically meshed with quad elements in a structured mesh. The mesh dimension can be set in the program options. Therefore, tolerances on position of windows and doors do not exceed the half of mesh size.

The following elements can be read:

- IfcBeam (Swept Solid body, Axis 2d and Mapped body representations)
- IfcColumn (Swept Solid body, Axis 2d and Mapped body representations)
- IfcWall (Swept solid representations)
- IfcSlab (Swept solid representations)
- IfcFooting (Swept solid representations)
- IfcReinforcingBar
- IfcStructural classes.

SAF models (Structural Analysis Format)

Import of SAF models happens through .XLSX files. Supported features are:

- StructuralMaterial
- StructuralCrossSection
- StructuralPointConnection
- StructuralCurveMember
- StructuralSurfaceMember
- StructuralPointSupport
- RelConnectsStructuralMember
- RelConnectsRigidLink
- StructuralLoadCase
- StructuralLoadCombination
- StructuralPointAction
- StructuralPointMoment
- StructuralCurveAction
- StructuralCurveMoment
- ResultInternalForce1D

3.2. Export

ABAQUS®

- Nodes
- Elements
  - B33
  - CPE2
  - CPS3
  - S3
  - CPE4
  - CPS4

OpenSees

- Nodes
- Elements
  - elasticBeamColumn
  - ShellMITC4
  - twoNodeLink (for springs)
  - stdBrick
  - Brick20N
- Sections
  - PlateFiber
- Isotropic Materials
  - nDMaterial ElasticIsotropic
- Restraints
- Nodal loads

Midas GEN®

- Units
- Nodes
- Elements
  - Beam
  - Triangular plates
  - Quadrilateral plates
- Groups
- Frame releases
- Isotropic materials
- Sections and general sections
- Wall and plate sections
- Restraints
- Elastic link
- Surface springs
- Members
- Constraints
- Nodal masses
- Loads cases
- Concentrated loads
- Distributed loads on beams
- Temperature loads
- Pressure loads
- Load combinations
- Subsoil springs
- Column rebar

- Beam rebar
- Wall rebar

SAP2000®

- Joints
- Frame connectivity
- Area connectivity
- Solid connectivity
- Basic material properties
- Frame sections
- Frame local axes
- Frame releases
- Area section (⚠ WARNING: thick shells with drilling DOFs)
- Section assignments
- Load cases (⚠ WARNING: Self weight set to zero)
- Restraints
- Constraints, rigid diaphragms
- Linear links (⚠ WARNING: not completely supported)
- Concentrated masses
- Concentrated loads
- Distributed loads on beams
- Temperature loads
- Groups
- Section cuts
- Functions: response spectrum and time-history
- Load patterns
- Load cases
- Modal loadcase
- Buckling loadcase
- Response spectrum analysis
- Dynamic analysis
- Load combinations

OOFEM

- Analysis
    - Linear static
- Nodes
- Elements
    - Beam3D
    - Truss3D
    - Tr_shell01
    - Quad1mindlinshell3d
    - Lumped mass
    - LTRSpace
    - LSpace
    - QSPace
- Sections
- Isotropic materials
- Boundary conditions
- Sets

Dxf

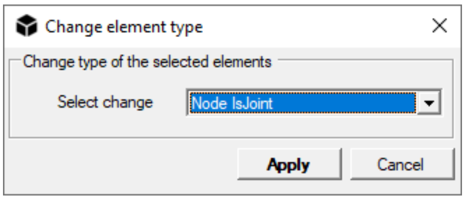
- Points
- Line elements
- Plain elements (⚠ WARNING: triangular and quadrilateral only)

BIM files – IFC e IFCxml

- IfcBeam (Swept Solid body, Axis 2d and Mapped body representations)
- IfcColumn (Swept Solid body, Axis 2d and Mapped body representations)
- IfcWall only for walls defined as group of shell (see *Edit/Mesh wall* command) (Swept solid representations)
- IfcSlab (Swept solid representations)
- IfcFooting (Swept solid representations)
- IfcReinforcingBar for longitudinal rebar and stirrups.

IDEA StatiCa

For steel structures, the entire model is exported. To design the joint in IDEA StatiCa Connection, the corresponding node must have the "Node isJoint" attribute, to be assigned by the command *Edit/Change element type*.



⚠ WARNING: the correct export of geometry and results is not guaranteed.

SAF models (Structural Analysis Format)

Export of SAF models happens through .XLSX files. Supported features are:

- StructuralMaterial
- StructuralCrossSection
- StructuralPointConnection
- StructuralCurveMember
- StructuralSurfaceMember
- StructuralPointSupport
- RelConnectsStructuralMember
- RelConnectsRigidLink
- StructuralLoadCase
- StructuralLoadCombination
- StructuralPointAction
- StructuralPointMoment
- StructuralCurveAction
- StructuralCurveMoment
- ResultInternalForce1D

4. Customization

4.1. Expand material library

The material library can be expanded by writing a CSV file (i.e. a text file using the semicolon separator), as in the following examples.

- *Steel*

```
Name;Code;E [MPa];G [MPa];n_;fyk [MPa];ftk [MPa];Wd [kN/m^3];Md [ton/m^3];CheckType
S235;UNI EN 10025-2;210000;80769;0.3;235;360;78.6;0.801;1
S275;UNI EN 10025-2;210000;80769;0.3;275;430;78.6;0.801;1
```

- *Concrete*

```
Name;Code;E [MPa];G [MPa];n_;fck [MPa];Wd [kN/m^3];Md [ton/m^3];CheckType
C8/10;NTC2008;25331;10555;0.2;8;25;254.841998
C12/15;NTC2008;27085;11285;0.2;12;25;254.841998
```

The column *CheckType* specifies the verifications associated with the material. Insert:

- 1 for *Steel*
- 2 for *Aluminium/Alloy*
- 3 for *Concrete*
- 4 for *Timber*
- 0 for other materials.

For other types of materials, all the columns are mandatory, except for "Md" – mass density, which will be calculated by the program in consistent units from Wd – weight density.

Natively supported fields are:

- E: Young modulus
- n_: Poisson ratio
- G: shear modulus
- fk: characteristic strength
- Wd: weight density
- a_T: thermal coefficient for linear deformation
- K: heat conductivity
- Cp: specific heat capacity

The property name can be followed by square brackets containing the unit of measure (eg. "E [MPa]").

The file must be placed in the "data" folder and its extension must be **.nfm*.

4.2. Expand section Library

The section library can be expanded by writing a CSV file (i.e. a text file using the semicolon separator) as follows:

```
Name;Code;h [mm];b [mm];tw [mm];tf [mm];r [mm];A [cm^2];Jy [cm4];Wey [cm^3];Wpy [cm^3];iy [cm];Avz [cm^2];Jz [cm^4];Wez [cm^3];Wpz [cm^3];iz [cm]
IPE A 80;UNI;78;46;3.3;4.2;5;6.375401837;64.37774091;16.50711305;18.97743881;3.177708713;3.070001837;6.852668856;2.979421242;4.692462888;1.036754887
...
```

The first four columns are mandatory.

Natively supported fields are:

- h: height
- b: base
- d: diameter
- t: thickness (for pipe sections)
- tw: web thickness
- tf: flange thickness
- A: area of the section
- Jz: moment of inertia with respect to horizontal axis
- Jy: moment of inertia with respect to vertical axis
- Wez: elastic strength modulus with respect to horizontal axis
- Wey: elastic strength modulus with respect to vertical axis
- Wpz: plastic strength modulus with respect to horizontal axis
- Wpy: plastic strength modulus with respect to vertical axis
- Avz: shear area with respect to horizontal axis (local z)
- Avy: shear area with respect to horizontal axis (local y)
- gap: horizontal distance for double sections (eg. double L or double C).

The property name can be followed by square brackets containing the unit of measure (eg. "h [cm]").

The *Type* columns specifies the shape of the section. Insert:

- 0 for unknown or generic
- 1 for rectangular
- 2 for circular
- 3 for C shape
- 4 for T shape
- 5 for double-T o I
- 6 for L shape
- 7 for box
- 8 for pipe
- 9 for double L
- 10 for double C.

The file must be placed in the "data" folder and its extension must be **.nfs*.

4.3. Library of sections defined by points

Sections defined by points can be inserted in a **.nfs* file like in the previous case, but with the following fields:

```
Name;Code;PointsXf1 [mm];PointsYf1 [mm];PointsXe2 [mm];PointsYe2 [mm];PointsXf3 [mm];PointsYf3 [mm];Class;OffsetX [mm];OffsetY
[mm];alphaLT;alphay;alphaz;Jw [mm^4]
TestSect;custom;0,20,20,0;0,0,20,20;4,8,8,4;4,4,8,8;0;0;3;0;0;0.76;0.76;0.76;0
…
```

First 2 columns are mandatory, and value of Code column must be set to "custom". Series of points can be whichever needed, by following this codification for columns names:

PointsX*pN* [*units*];PointsY*pN* [*units*];

in which *p* is equal to "f" for filled figures and "e" for holes; *N* is the progressive number of series, the same for filled and empty shapes. All the columns after section offsets are inserted in section custom values (e.g. *alphaLT;alphay;alphaz;Jw*).

4.4. Custom verifications

It is possible to customize the list of verifications with the notation described in the following. The text file containing checking can be placed in the "*verification*" folder in the program installation directory and must have the "*.nvv*" extension (see for example *trusses.nvv*).

Such file can be edited with:

- Notepad++, with the setting for the syntax highlight that can be found in the "*verification*" folder (file *NextFEMVerifications.xml*);

- Visual Studio Code with Custom Coloring plugin for syntax highlight. Setting for this plugin are in the file *NextFEMverification_vsCode.json* contained in the folder "*verification*" as well.

The checking engine supports the code execution in blocks. Each block is delimited by the identifiers as in the following example and must the named with a floating point number (0.6).

$$0.6

 # this is a comment

execif(SecType==1,1.0)

$!

To call each block, the following keywords are available:

- *exec(0.6)* : executes the code block named 0.6
- *execif(condition, 0.6)*: executes the code block named 0.6 if *condition* is true;
- *execwhile(condition, 0*.6): executes the code block named 0.6 while condition is true. The variable *exitdo* equal to 1 within the code block forces the exit. Limited to 10000 cycles.

Formulas can be written using the following operators/functions:

Addition: **+**
Subtraction: **-**
Multiplication: *****
Division: **/**
Modulo: **%**
Exponentiation: **^**
Less than: **<**
Less than or equal: **<=** or ≤
More than: **>**
More than or equal: **>=** or ≥
Equal: **==**
Not Equal: **!=** or ≠
Sine: **sin**
Cosine: **cos**
Arcsine: **asin**
Arccosine: **acos**
Tangent: **tan**
Cotangent: **cot**
Arctangent: **atan**
Arc cotangent: **acot**
Natural logarithm: **loge**
Common logarithm: **log10**

Logarithm: **logn**
Square root: **sqrt**
Conditional key: **if(var<var2,1,0)**
Boolean operator: **and(cond1,cond2)**
Boolean operator: **or(cond1,cond2)**
Boolean operator: **not(cond)**

The hardcoded variables are:

- **Model units handling**
    - **unitconv**: converts between units. Usage: *unitconv(oldUnits,newUnits,Value)*.
      Example: Eps=sqrt(235/*unitconv(model_S,MPa,fk)*)
      converts fk from *stress units in the model* to *MPa*
    - **rcsect**: calculates resisting moments of a RC section, storing them in Mry and Mrz. Usage: *rcsect(N,Myy,Mzz)*
    - **skipItem**: if =1, skips the subsequent checking. To be used only in the time-dependent load cases (for example, linear dynamic analysis)
    - **model_L**: placeholder for the length unit in the model
    - **model_F**: placeholder for the force unit in the model
    - **model_FL**: placeholder for the force per length unit in the model
    - **model_T**: placeholder for the temperature unit in the model
    - **model_M**: placeholder for the mass unit in the model
    - **model_S**: placeholder for the stress unit in the model
    - **isVarDefined(*var*)**: checks if a variable is defined (1) or not (0)
    - **Halt**: stops the execution;
    - **addUtable:** adds an user table at runtime, and returns the ID of the table. Usage: addUtable(numColumns, columnsHeaders, data by row including row header). Example: tid=addUtable(3,1,2,3,100,4,5,6,101,4,5,6,102,4,5,6,103,4,5,6)
      adds the following table:

	1	**2**	**3**
100	4	5	6
101	4	5	6
102	4	5	6
103	4	5	6

    - **UtableAt**: returns the value at i, j of a table. Usage: UtableAt(table ID,i,j)
    - **UtableInterpValues:** gives the bilinear interpolation for the table. Usage: results=UtableInterpValues(table ID, value on row headers, value on column headers). Example: res1= UtableInterpValues(tid,101.3,2.5) gives 5.5 as a result.
    - **addRebar**: adds longitudinal rebar to the current element.
      The function returns 0 if it fails or design material cannot be found.
      Usage: addRebar(zCoord, yCoord, rebarArea, design material ID, startAt [0,1), endAt (0,1]).
      Example: addRebarL(40,40,201.0,2,0,1)
      adds a rebar of area 201.0 at position 40, 40 from the bottom left corner of the section, for the whole element.
    - **addStirrups**: adds stirrups to the current element.
      The function returns 0 if it fails or design material cannot be found.
      Usage: addStirrups(legs in Y, legs in Z, single bar area, spacing, design material ID, startAt [0,1), endAt (0,1]).

Example: addStirrups(2, 2, 50.0, 200, 2, 0,1)
adds 2-by-2 legs stirrups, each one of area 50.0, at 200 of spacing, for the whole element.

- **clearRebar:** clear all rebar, including stirrups, in the current element.
- **getBarDiam**: gets the minimum diameter of a longitudinal bar required to satisfy the given area. The output is in mm. Example: getBarDiam(2.01) # for a model in cm, returns 16.
- **getStirrupDiam**: gets the minimum diameter of a stirrup bar required to satisfy the given area. The output is in mm. Example: getStirrupDiam(0.5) # for a model in cm, returns 8.
- **TranslateMomentZZ**: gives maximum and minimum moments around z local axis for the given position, translated by a quantity *delta*. Syntax: TranslateMomentZZ(position, delta). The argument *position* is contained in the built-in dataset for elements in the variable *pos*. This function returns or overwrite the variables *Mzzmax* and *Mzzmin*.
- **TranslateMomentYY**: gives maximum and minimum moments around y local axis for the given position, translated by a quantity *delta*. Syntax: TranslateMomentYY(position, delta). The argument *position* is contained in the built-in dataset for elements in the variable *pos*. This function returns or overwrite the variables *Myymax* and *Myymin.*
- **FireSectionStrength**: loads a thermal map of a section and use it to estimate element strength. It is available in the FireSafe module. The filename containing thermal results for the station must be saved inside the "fireSect" key in custom properties of the element. Syntax: FireSectionStrength(station,N,Myy,Mzz). The argument *station* is the checking station, typically from 1 to 5.
- **round**: round a floating point number to the nearest integer. Ex. round(12.6) returns 13.0.
- **ceil**: round up a floating point number. Ex. ceil(12.3) returns 13.0.
- **floor**: round down a floating point number. Ex. floor(12.6) returns 12.0.

- **Built-in general dataset:**
  - **ServType:** indicates if the current serviceability combination is characteristic (rare) (1), frequent (2) or quasi-permanent (2). The value is 0 if the combination is not of serviceability type.
  - **Seismic**: indicates if the current combination is seismic (1) or not (0). The value 1 is always associated to ServType=0.
  - **time:** current time.

- **Built-in dataset for elements:**
  - **A**: Area
  - **Jz**: Moment of inertia around z-axis
  - **Jy**: Moment of inertia around y-axis
  - **Jmin**: Minimum moment of inertia
  - **Jt**: Torsional Inertia
  - **D**: Diameter of circular cross sections
  - **Di**: Inner diameter of pipe cross sections
  - **te**: Thickness of pipe cross sections
  - **b**: Base for any other cross sections
  - **h**: Height for any other cross sections
  - **tw**: web thickness
  - **tf1**: thickness of bottom flange
  - **tf2**: thickness of upper flange
  - **t**: thickness for planar sections
  - **N**: Axial force of the current section of a beam
  - **Vy**: Shear force along y direction of the current section of a beam

- **Vz**: Shear force along z direction of the current section of a beam
- **Mt**: Twisting moment of the current section of a beam
- **Myy**: Moment around y local axis of the current section of a beam
- **Mzz**: Moment around z local axis of the current section of a beam
- **rMyIJ**: ratio between Myy at end I and at end J of the beam
- **rMzIJ**: ratio between Mzz at end I and at end J of the beam
- **MmaxY**: maximum moment around y axis for the whole element or member
- **MmaxZ**: maximum moment around z axis for the whole element or member
- **MminY**: minimum moment around y axis for the whole element or member
- **MminZ**: minimum moment around z axis for the whole element or member
- **Em**: material Young modulus
- **Gm**: material shear modulus
- **Nlm:** material Poisson's ratio
- **fk**: material characteristic strength
- **WelZ**: section modulus for z axis
- **WelY**: section modulus for y axis
- **WplZ**: plastic section modulus for z axis
- **WplY**: plastic section modulus for y axis
- **iz**: radius of inertia for z axis
- **iy**: radius of inertia for y axis
- **imin**: minimum radius of inertia
- **SecType**: 1=beam, 2=planar, 0=unknown
- **SecBeamType**: 0=unknown, 1=rectangular, 2=circular, 3=Cshape, 4=Tshape, 5=DoubleTshape, 6=Lspahe, 7=box, 8=pipe
- **MatType**: type of material. Steel 1, Aluminium 2, Concrete 3, Timber 4, Masonry 5, Fragile in tension 6, unknown 0.
- **dx**: axial relative displacement along beam axis
- **dy**: transversal deflection in local direction y
- **dz**: transversal deflection in local direction z
- **L0yy**: buckling length for bending around y local axis
- **L0zz**: buckling length for bending around z local axis
- **L0**: maximum buckling length between L0yy and L0zz
- **pos**: of the current section of a beam, in model units
- **LvyI**: contraflexure length from end I, ratio Myy/Vz in station 1
- **LvyJ**: contraflexure length from end J, ratio Myy/Vz in station 5
- **LvzI**: contraflexure length from end I, ratio Mzz/Vy in station 1
- **LvzJ**: contraflexure length from end J, ratio Mzz/Vy in station 5
- **fxx**: force in x local direction (element centre)
- **fyy**: force in y local direction (element centre)
- **fxy**: shear force in xy direction (element centre)
- **fzz**: force in z local direction (element centre)
- **fxz**: shear force in xz direction (element centre)
- **fyz**: shear force in yz direction (element centre)
- **mmxx**: moment around x local direction (element centre)
- **mmyy**: moment around y local direction (element centre)
- **mmxy**: moment around xy direction (element centre)
- **mmzz**: moment around z local direction (element centre)
- **mmxz**: moment around xz direction (element centre)
- **mmyz**: moment around yz direction (element centre)

- **time**: current time step
- **temp**: average element temperature in the current step
- **isWall**: if wall groups is defined for the current planar element returns 1, 0 otherwise.
- **Column**: if the current beam element is vertical returns 1, 0 otherwise. It is not defined for other types of elements.

- **Built-in dataset for nodal results:**
  - **dx**: nodal displacement in X direction
  - **dy**: nodal displacement in Y direction
  - **dz**: nodal displacement in Z direction
  - **rx**: nodal rotation around X axis
  - **ry**: nodal rotation around Y axis
  - **rz**: nodal rotation around Z axis
  - **vx**: nodal velocity in X direction
  - **vy**: nodal velocity in Y direction
  - **vz**: nodal velocity in Z direction
  - **vrx**: nodal velocity around X axis
  - **vry**: nodal velocity around Y axis
  - **vrz**: nodal velocity around Z axis
  - **ax**: nodal acceleration in X direction
  - **ay**: nodal acceleration in Y direction
  - **az**: nodal acceleration in Z direction
  - **arx**: nodal acceleration around X axis
  - **ary**: nodal acceleration around Y axis
  - **arz**: nodal acceleration around Z axis
  - **Rex**: nodal reaction in X direction
  - **Rey**: nodal reaction in Y direction
  - **Rez**: nodal reaction in Z direction
  - **Rerx**: nodal reaction around X axis
  - **Rery**: nodal reaction around Y axis
  - **Rerz**: nodal reaction around Z axis
  - **sxx**: nodal stress in X direction
  - **syy**: nodal stress in Y direction
  - **sxy**: nodal shear stress in XY direction
  - **szz**: nodal stress in Z direction
  - **sxz**: nodal shear stress in XZ direction
  - **syz**: nodal shear stress in YZ direction
  - **fxx**: nodal force in X direction
  - **fyy**: nodal force in Y direction
  - **fxy**: nodal shear force in XY direction
  - **fzz**: nodal force in Z direction
  - **fxz**: nodal shear force in XZ direction
  - **fyz**: nodal shear force in YZ direction
  - **mxx**: nodal moment around X direction
  - **myy**: nodal moment around Y direction
  - **mxy**: nodal moment around XY direction
  - **mzz**: nodal moment around Z direction
  - **mxz**: nodal moment around XZ direction
  - **myz**: nodal moment around YZ direction

- **time**: current time step
- **temp**: nodal temperature in the current step.

5. Getting started and validation

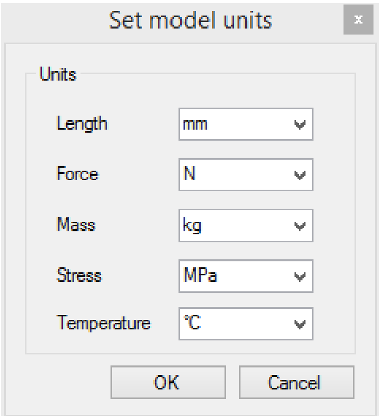
5.1. Tutorial One

This tutorial will show how to model a 5 metres long fixed-ended beam, loaded with concentrated loads of 10kN in directions x, y and z in the middle of its span. The results from NextFEM Designer (Frame forces and displacement) are compared with hand calculations.

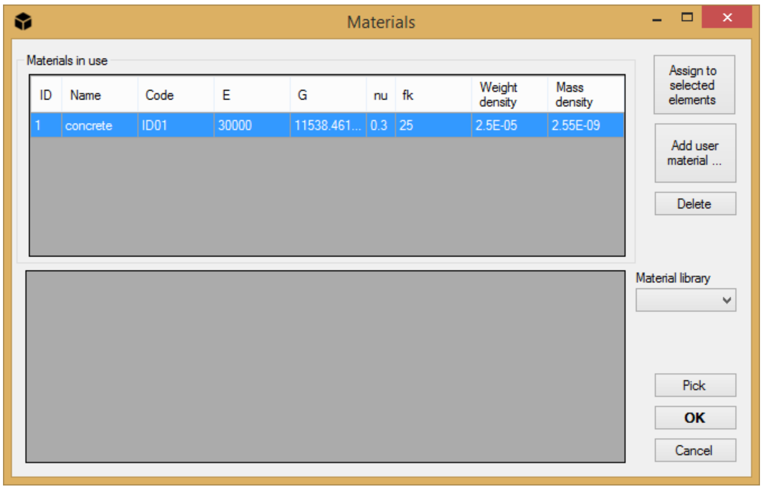
⚠ **<u>WARNING: Both flexural and shear deformations are considered. To enable this option, click on *Tools>Option>Solver* and check the *Include shear deformations in beam elements* tick under the *OOFEM preferences* box</u>**

The following sequence of operations are needed to create the model:

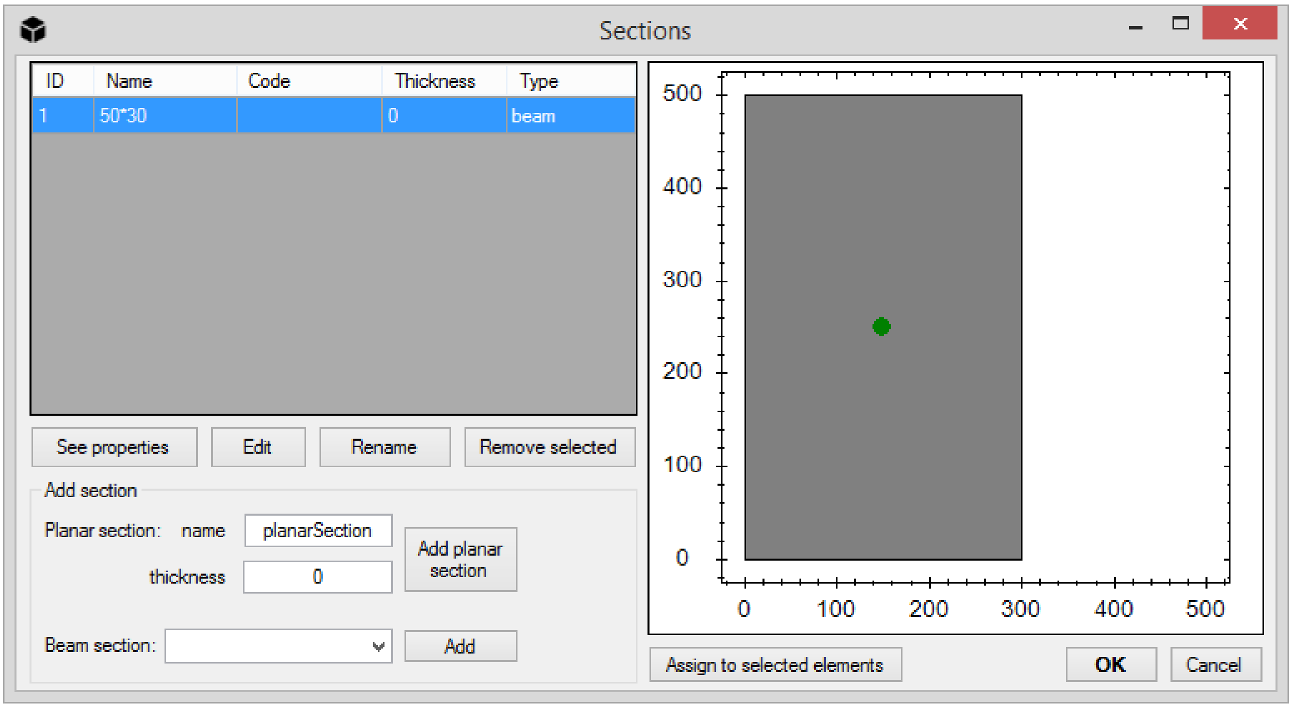
1. Set the Units: *N* for force and *mm* for length.



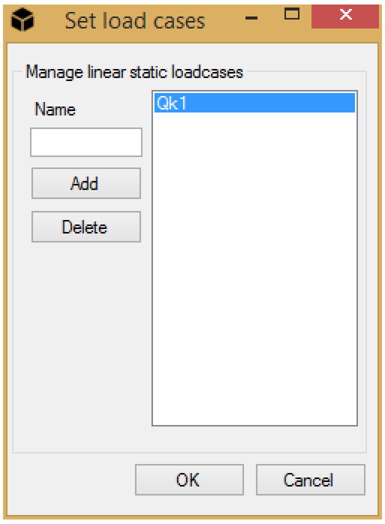
2. Define the Material properties:
   - Name: Concrete;
   - E=30000 N/mm^2;
   - Nu=0.3
   - Fk=25 N/mm
   - Weight=2.5e-5 N/mm^3;
   - Mass=2.55e-9 N/mm^2/g



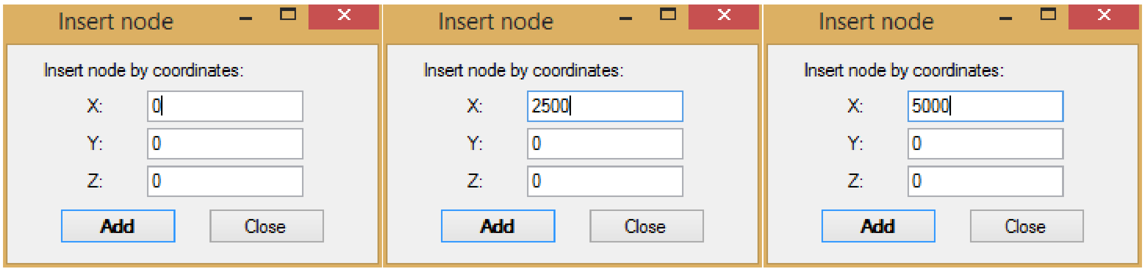
3. Define the Section properties:
   - b=300 mm (z direction);
   - h=500mm (y direction);



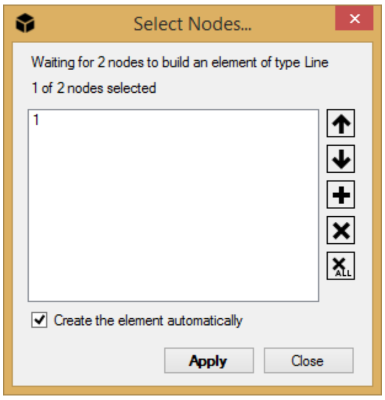
4. Define the Loads cases: Only one load case called *Qk1* is considered



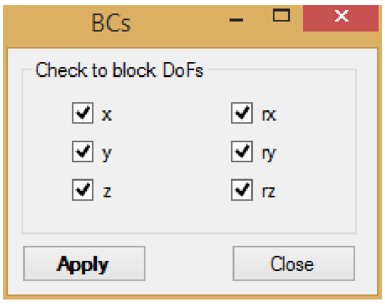
5. Insert the Geometric properties using *Node by Coordinates*:
   - L=5000 mm;
   - Distance from fixed end to loads=2500 mm;



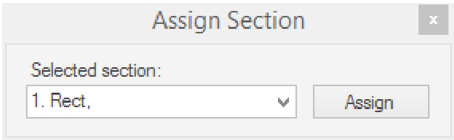
6. Insert the beams using the *Beam* command.



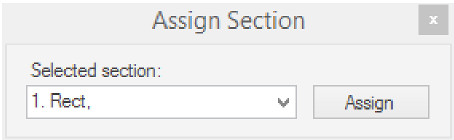
7. Assign the boundary conditions using the *Restraints* command: fix all DoFs for nodes 1 and 3.



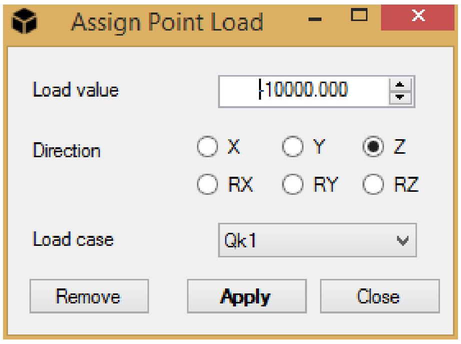
8. Assign the material using the *Assign>Material* command at the beams by selecting them and then click on *Assign to selected elements*



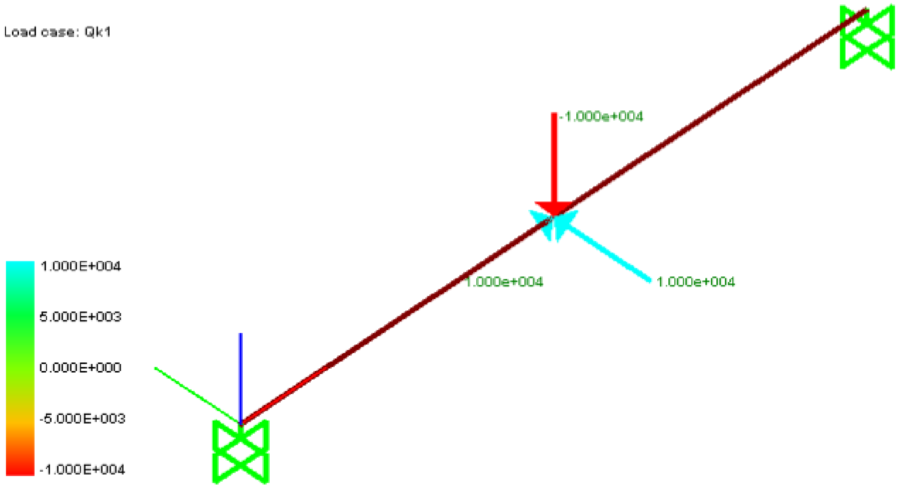
9. Assign the section using the *Assign>Section* command at the beams by selecting them and then click on *Assign*



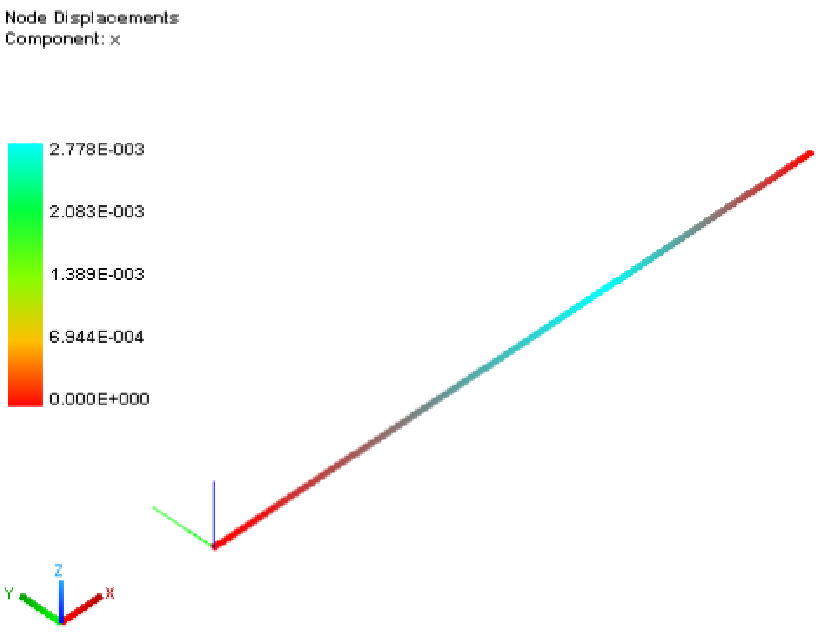
10. Assign the point load to the node number 2.
    - Px=10000 N;
    - Py=10000 N;
    - Pz=-10000 N.



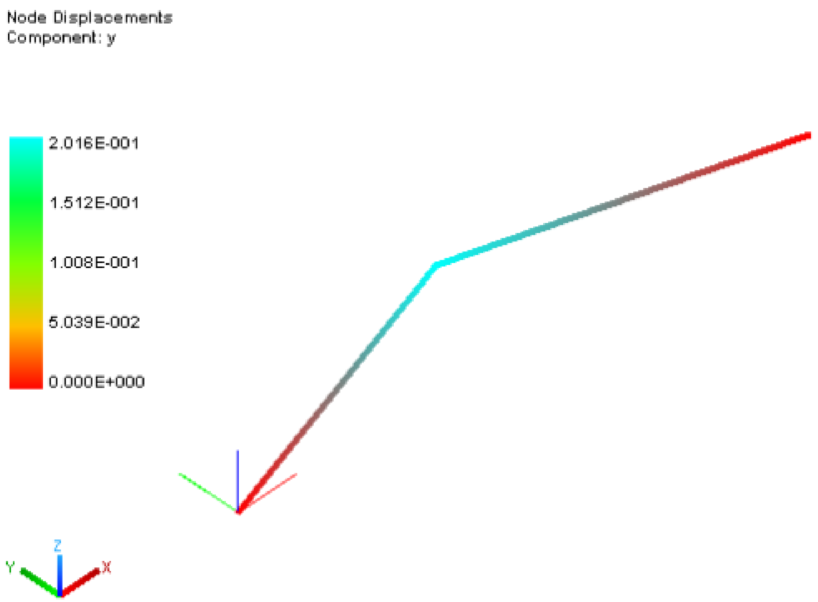
11. Run the analysis.



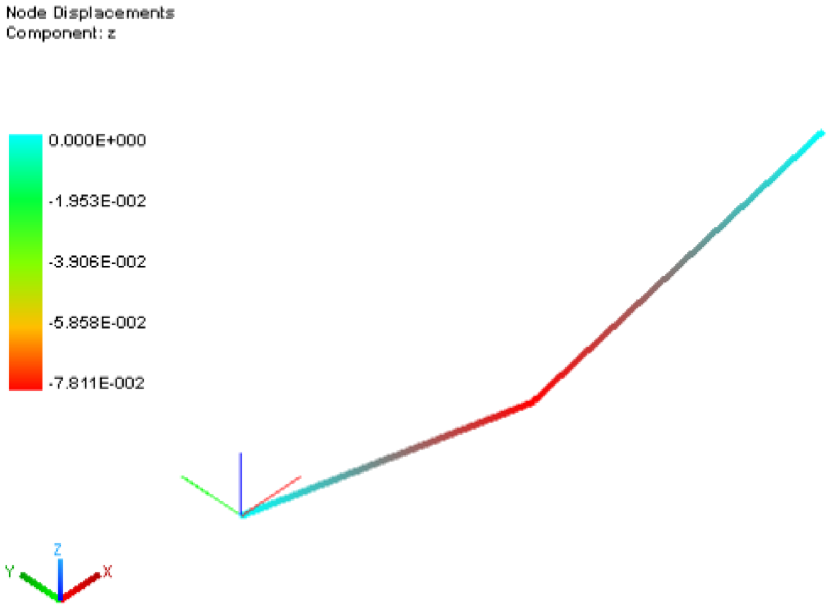
- NextFEM Designer's Results:
    - Displacement in x direction: Node 2=0.002778mm



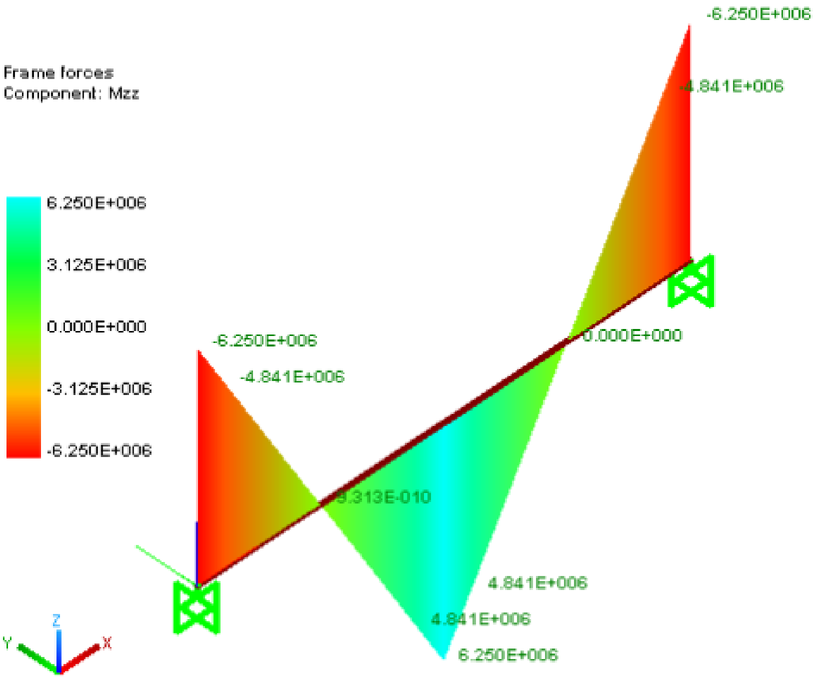
- Displacement in y direction: Node 2=0.2016mm



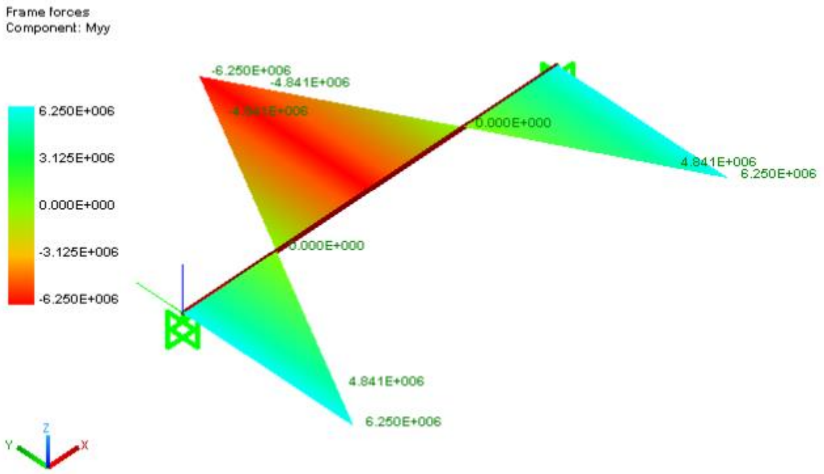
- Displacement in z direction: Node 2=-0.00781 mm



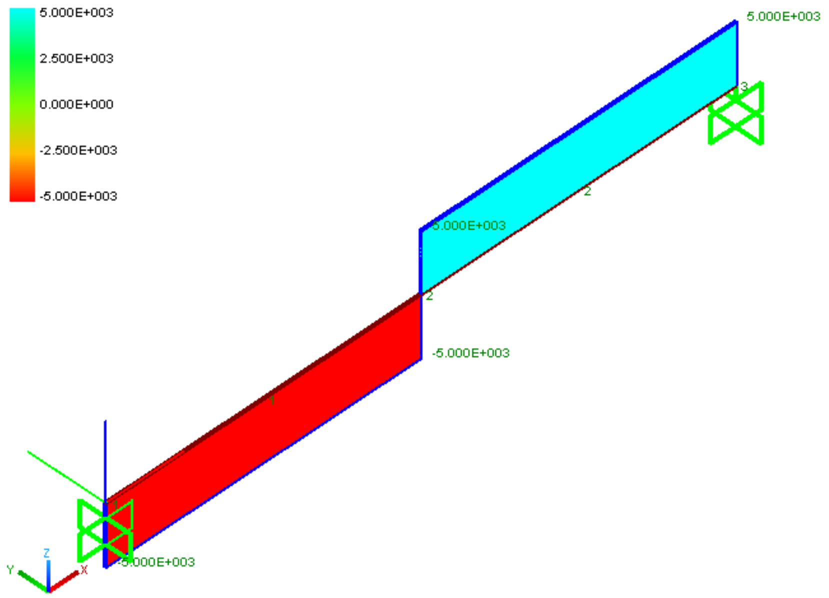
- Moment Diagram: Values from ***Results>Extract Data***
  - Mzz



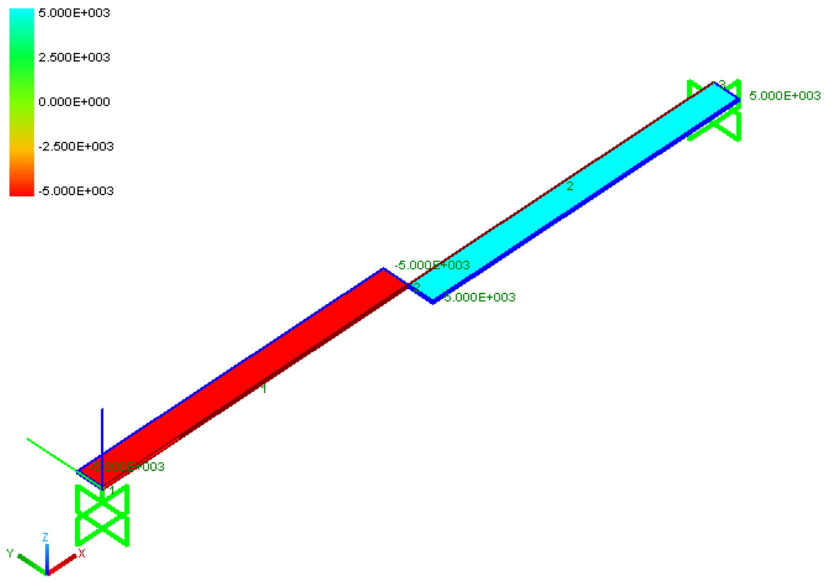
  - Myy



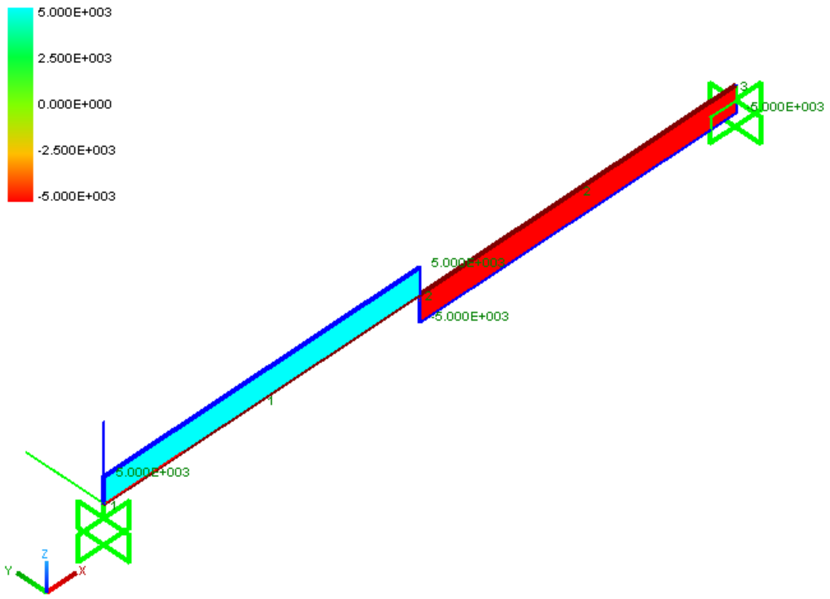
- Shear Diagram:
  - Vy



  - Vz



○ Normal forces Diagram:



○ Internal displacements in y direction: values from ***Results>Extract data***

Position [mm]	Displacement [mm]
0	0
281.8	0.00778
1250	0.1008
2218	0.1938
2500	0.2016
2500	0.2016
2781.8	0.1938

3750	0.1008
4718	0.00778
5000	0

- Internal displacements in z direction: values from ***Results>Extract data***

Position [mm]	Displacement [mm]
0	0
281.8	-0.003424
1250	-0.03906
2218	-0.07469
2500	-0.07811
2500	-0.07811
2781.8	-0.07469
3750	-0.03906
4718	-0.003424
5000	0

- Hand Calculations:
  - Section properties

    $A = b \cdot h = 150000mm^2$

    $J_y = \frac{bh^3}{12} = 3125e6mm^4$

    $J_z = \frac{hb^3}{12} = 1125e6mm^4$
  - Moment diagram:
    - Mzz

$M_{\max} = \frac{P_z l}{8} = 6250000Nmm$; $M_{\min} = -\frac{P_z l}{8} = -6250000Nmm$

    - Myy

$M_{\max} = \frac{P_y l}{8} = 6250000Nmm$; $M_{\min} = -\frac{P_y l}{8} = -6250000Nmm$

  - Shear Diagram:
    - Vy

$V_{\max} = \frac{P_z}{2} = 5000N$; $V_{\min} = -\frac{P_z}{2} = -5000N$

    - Vz

$V_{\max} = \frac{P_y}{2} = 5000N$; $V_{\min} = -\frac{P_y}{2} = -5000N$

  - Axial force Diagram:

$N_{\max} = \frac{P_x}{2} = 5000N$; $N_{\min} = -\frac{P_x}{2} = -5000N$

  - Displacement in x direction: Node2

$$u_{2,x} = \frac{N_{\max}(l/2)}{EA} = 0.00278mm$$

- Displacement in y direction: Node 2

$$u_{2,y} = \frac{1}{192}\frac{P_y l^3}{EJ_z} + \chi\frac{P_y l}{4GA} = 0.201568mm$$

- Displacement in z direction: Node 2

$$u_{2,z} = \frac{1}{192}\frac{P_z l^3}{EJ_y} + \chi\frac{P_z l}{4GA} = -0.07811mm$$

- Displacement in y direction: internal point at the coordinate x

$$u_{x,y} = \frac{1}{24}\frac{P_y x^2\left(\frac{3}{2}l - 2x\right)}{EJ_z} + \chi\frac{P_y x}{2GA} \text{ for } 0 \le x \le L/2$$

$$u_{x,y} = \frac{1}{24}\frac{P_y\left(L-x\right)^2\left(2x - \frac{L}{2}\right)}{EJ_z} + \chi\frac{P_y\left(L-x\right)}{2GA} \text{ for } L/2 \le x \le L$$

Position [mm]	Displacement [mm]
0	0
281.8	0.00778
1250	0.1008
2218	0.1938
2500	0.2016
2500	0.2016
2781.8	0.1938
3750	0.1008
4718	0.00779
5000	0

- Displacement in z direction: internal point at the coordinate x

$$u_{x,z} = \frac{1}{24}\frac{P_z x^2\left(\frac{3}{2}l - 2x\right)}{EJ_y} + \chi\frac{P_z x}{2GA} \text{ for } 0 \le x \le L/2$$

$$u_{x,z} = \frac{1}{24}\frac{P_z\left(L-x\right)^2\left(2x - \frac{L}{2}\right)}{EJ_y} + \chi\frac{P_z\left(L-x\right)}{2GA} \text{ for } L/2 \le x \le L$$

Position [mm]	Displacement [mm]
0	0
281.8	-0.003425

1250	-0.03906
2218	-0.07468
2500	-0.07811
2500	-0.07811
2781.8	-0.07469
3750	-0.03906
4718	-0.003429
5000	0

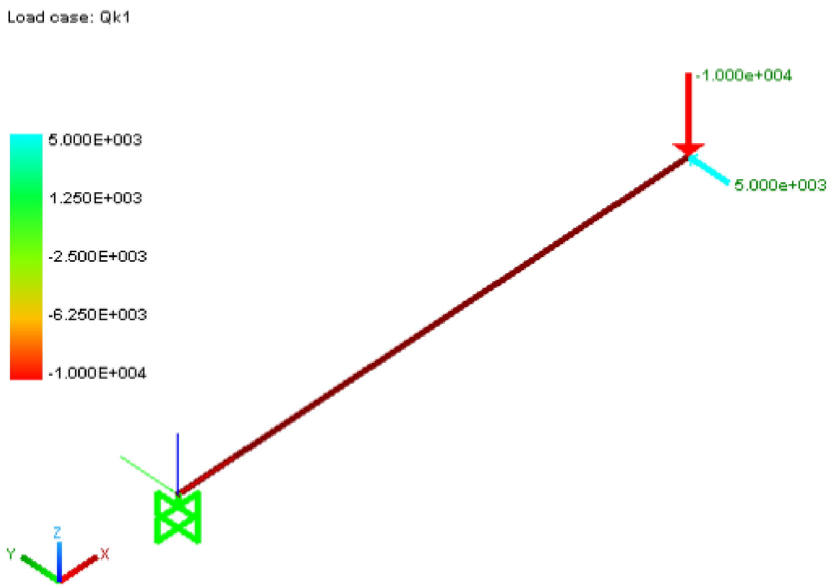
5.2. Tutorial Two

The second tutorial consists in a cantilever beam loaded with points load in directions y and z. The output results of NextFEM Designer (Frame forces and displacement) are compared with hand calculations.

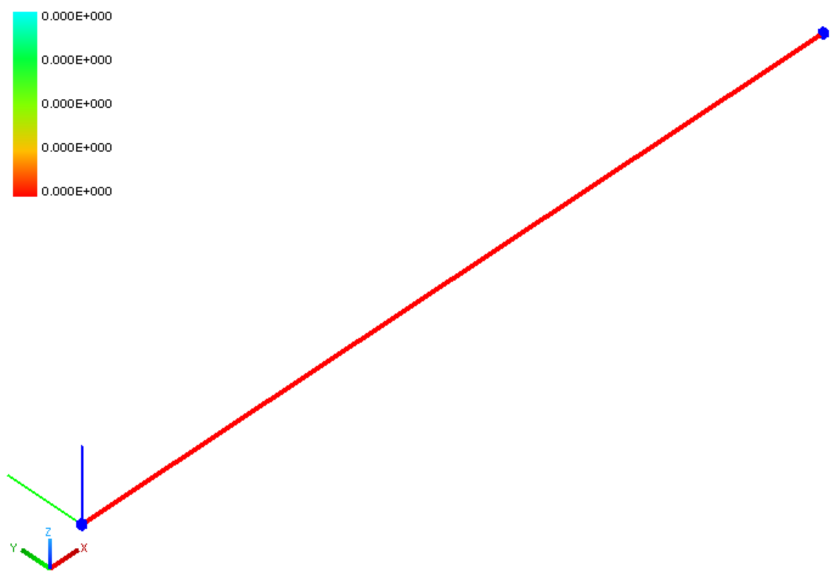
Case a

⚠ **<u>Only flexural deformations are considered.</u>**

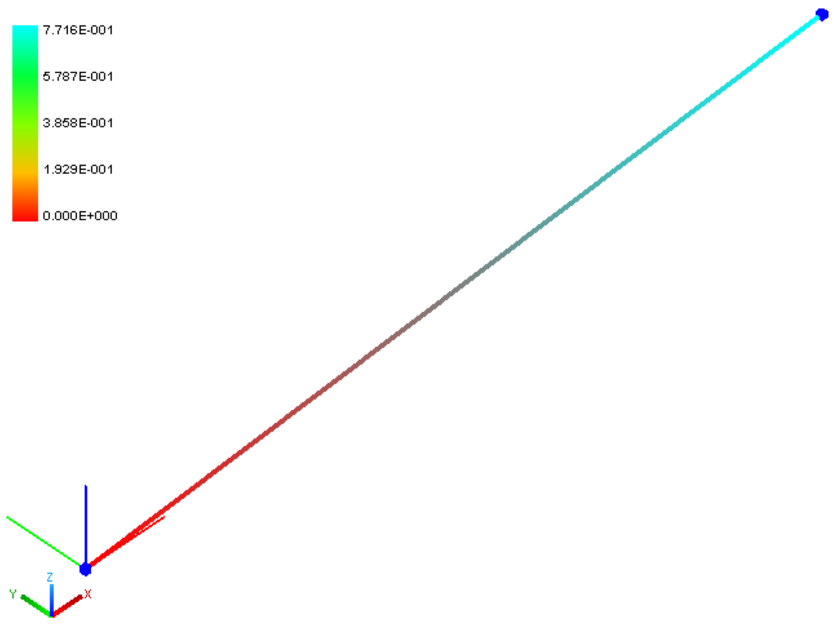
- Units: N for forces and mm for lengths.
- Material Properties:
  - Name: Concrete;
  - E=30000 N/mm^2;
  - Nu=0.3
  - Fk=25 N/mm
  - Weight=2.5^10-5 N/mm^3;
  - Mass=2.55^10-9 N/mm^2/g
- Section properties:
  - B=300 mm (z direction);
  - H=500mm (y direction);
- Geometric properties:
  - L=2500 mm;
- Loads:
  - Py=5000 N;
  - Pz=-10000 N.



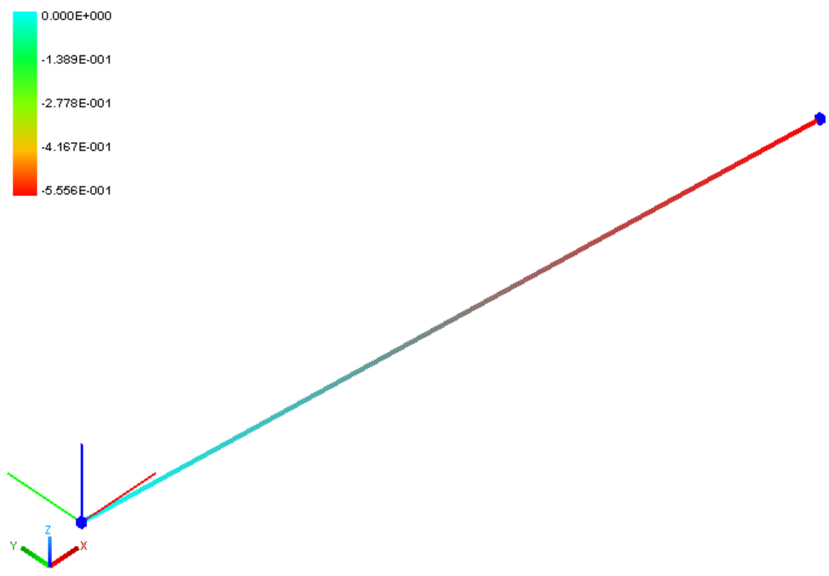
- NextFEM Designer's results:
  - Displacement in x direction: Node 2=0.00mm



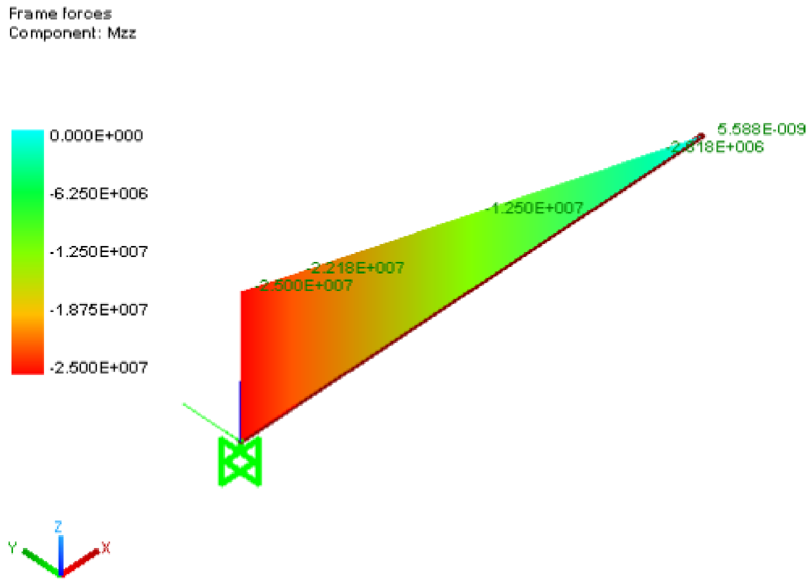
  - Displacement in y direction: Node 2=0.7716mm



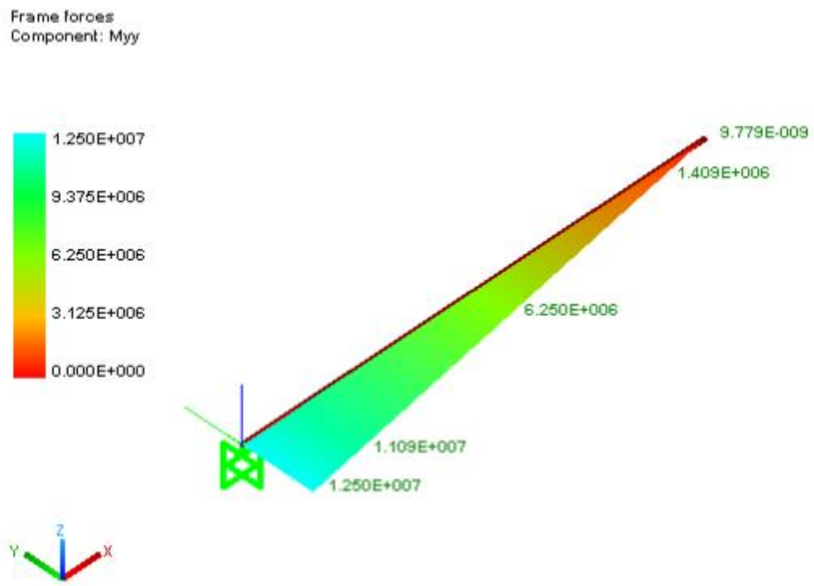
- Displacement in z direction: Node 2=-0.5556mm



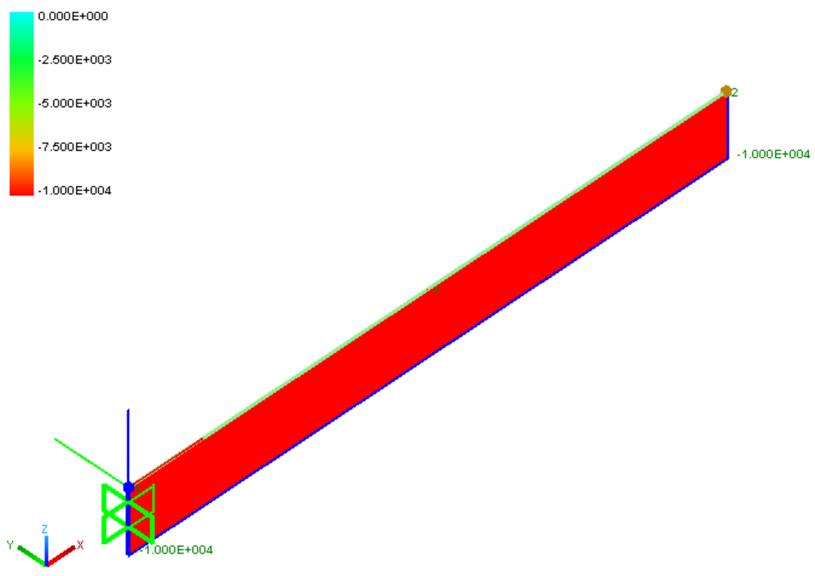
- Moment Diagram: Values from ***Results>Extract Data***
  - Mzz



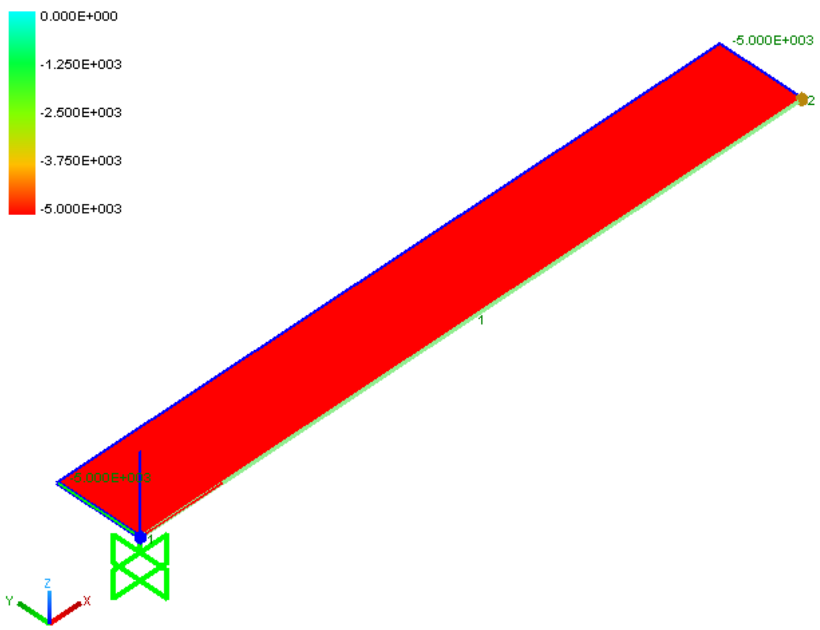
- Myy



- Shear Diagram:
  - Vy



- Vz



- Internal displacement in y direction: values from ***Results>Extract data***

Position [mm]	Displacement [mm]
0	0
281.8	0.01415
1250	0.2411
2218	0.6417
2500	0.7716

- Internal displacement in z direction: values from ***Results>Extract data***

Position [mm]	Displacement [mm]

0	0
281.8	-0.01019
1250	-0.1736
2218	-0.4620
2500	-0.5556

- Hand Calculations:
  - Moment diagram:
    - Mzz

$M_{\max} = P_z l = 25000000 Nmm$;

    - Myy

$M_{\max} = P_y l = 12500000 Nmm$

  - Shear Diagram:
    - Vy

$V_{\max} = P_z = 10000 N$;

    - Vz

$V_{\max} = P_y = 5000 N$;

  - Axial force Diagram:

$N_{\max} = 0 N$;

  - Displacement in x direction: Node 2

$u_{2,x} = 0$

  - Displacement in y direction: Node 2

$$u_{2,y} = \frac{1}{3}\frac{P_y l^3}{EJ_z} = 0.77160 mm$$

  - Displacement in z direction: Node 2

$$u_{2,z} = \frac{1}{3}\frac{P_z l^3}{EJ_y} = -0.0.5556 mm$$

  - Displacement in y direction: point at coordinate x

$$u_{x,y} = \frac{1}{6}\frac{P_y x^2 (3l - x)}{EJ_z}$$

Coordinate x [mm]	Displacement [mm]
0	0
281.8	0.01415
1250	0.2411
2218	0.6416
2500	0.7716

  - Displacement in z direction: point at coordinate x

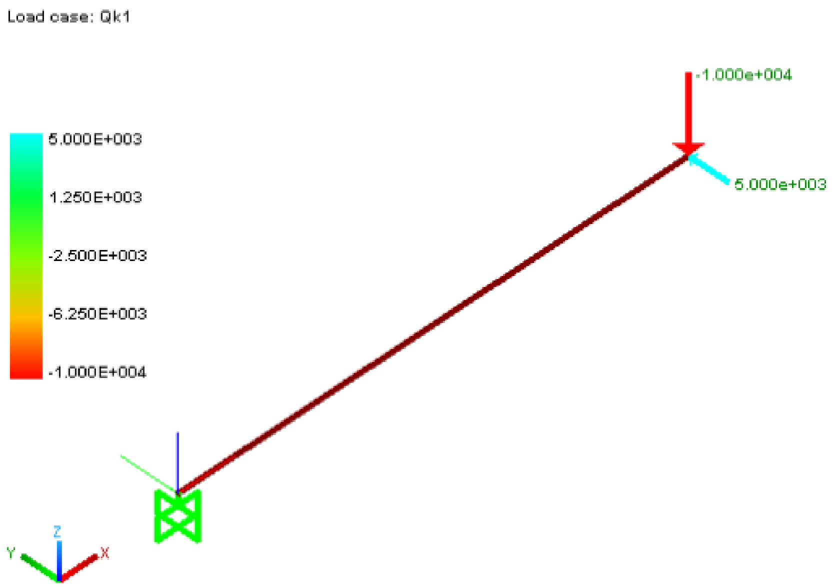
$$u_{x,z} = \frac{1}{6}\frac{P_z x^2 (3l - x)}{EJ_y}$$

Coordinate x [mm]	Displacement [mm]
0	0
281.8	-0.01019
1250	-0.1736
2218	-0.4620
2500	-0.5556

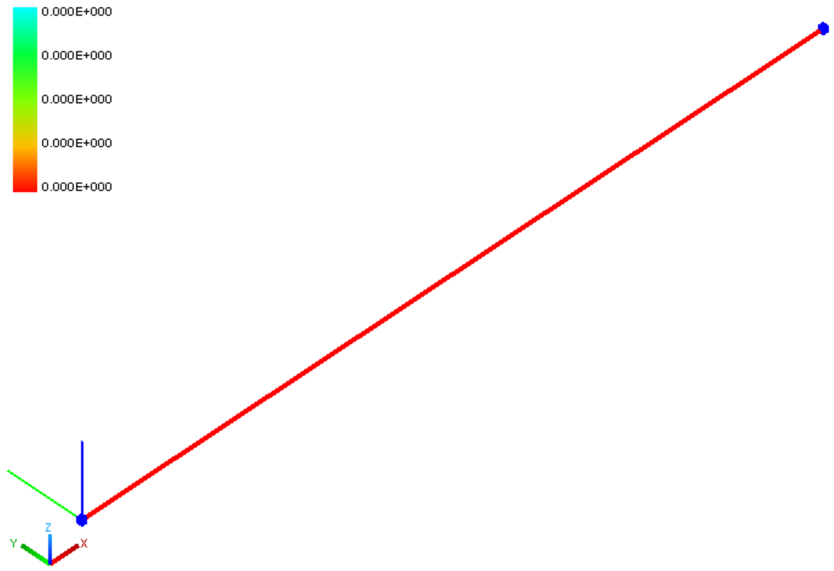
Case b

⚠ **<u>Both flexural and shear deformations are considered. To enable this option, click on *Tools>Option>Solver* and check the *Include shear deformations in beam elements* tick under the *OOFEM preferences* box</u>**

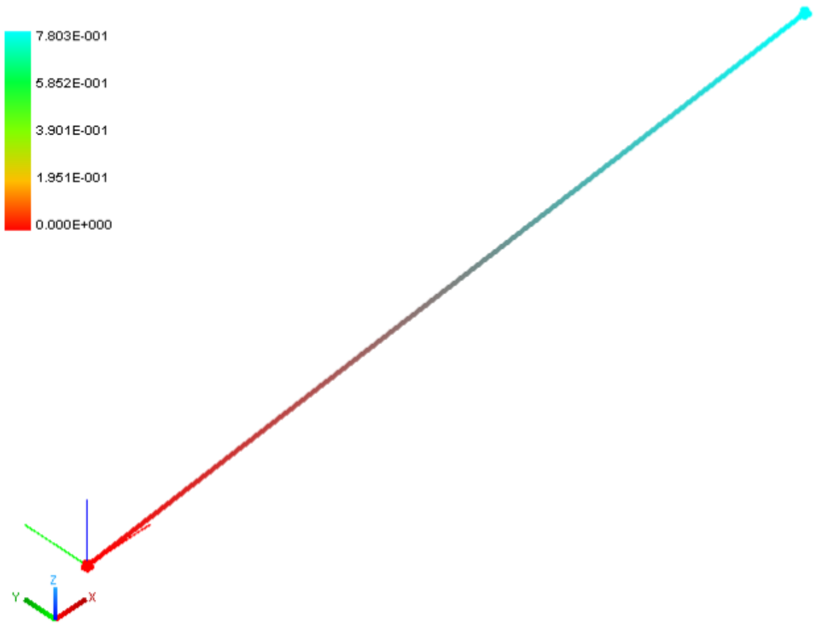
- Units: N for forces and mm for lengths.
- Material Properties:
  - Name: Concrete;
  - E=30000 N/mm^2;
  - Nu=0.3
  - Fk=25 N/mm
  - Weight=2.5^10-5 N/mm^3;
  - Mass=2.55^10-9 N/mm^2/g
- Section properties:
  - B=300 mm (z direction);
  - H=500mm (y direction);
- Geometric properties:
  - L=2500 mm;
- Loads:
  - Py=5000 N;
  - Pz=-10000 N.



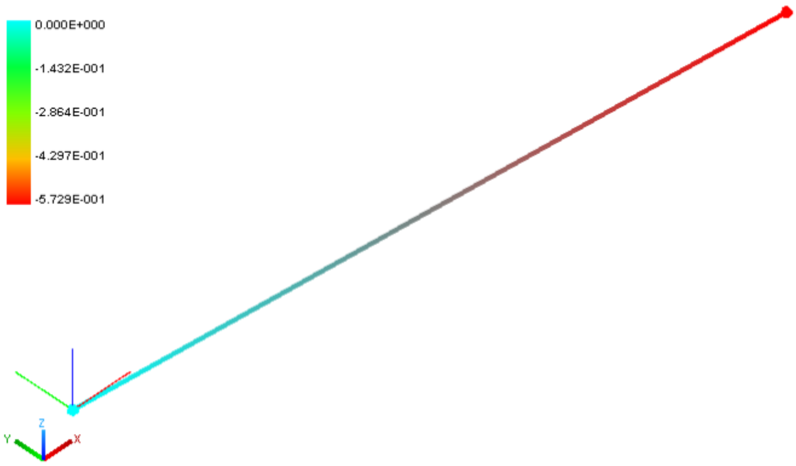
- NextFEM Designer's results:
    - Displacement in x direction: Node 2=0.00mm



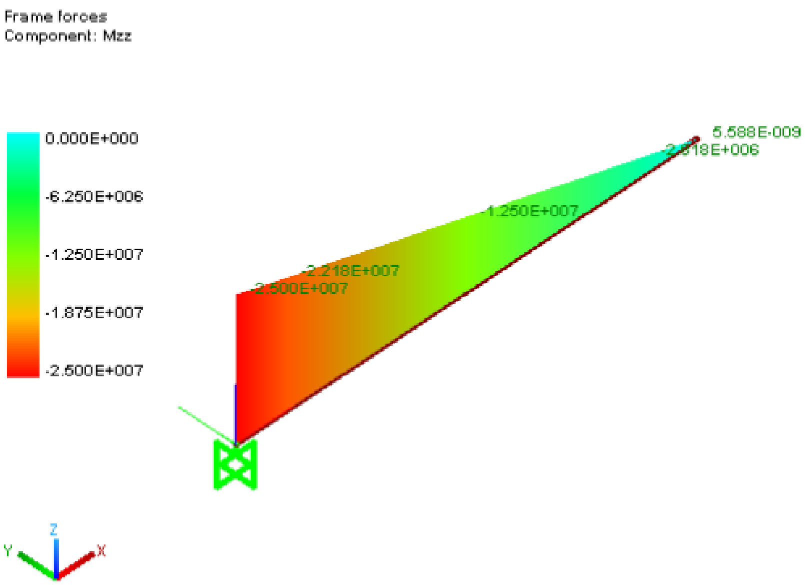
    - Displacement in y direction: Node 2=0.7803mm



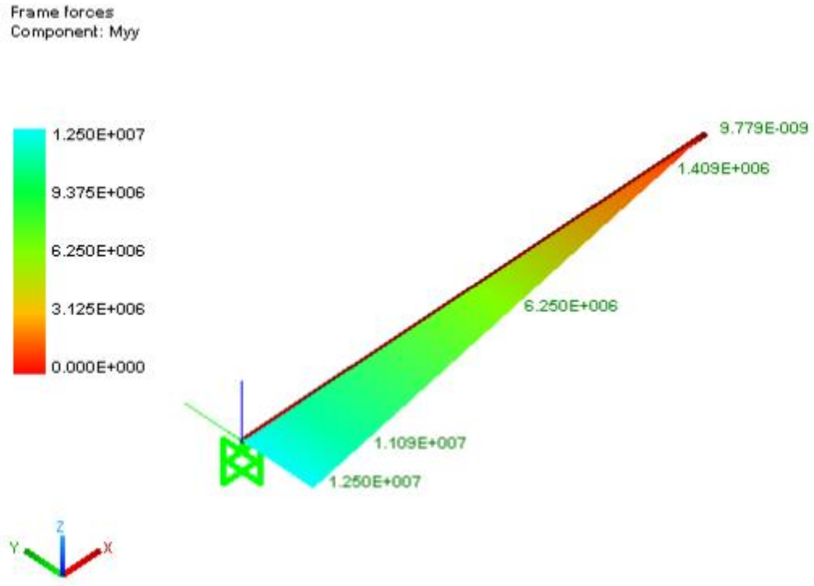
    - Displacement in z direction: Node 2=-0.5729mm



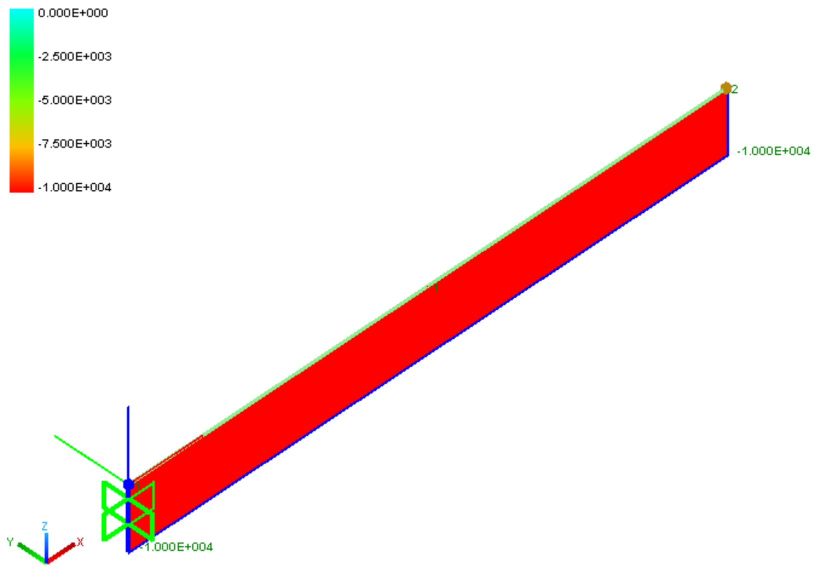
- Moment Diagram: Values from ***Results>Extract Data***
  - Mzz



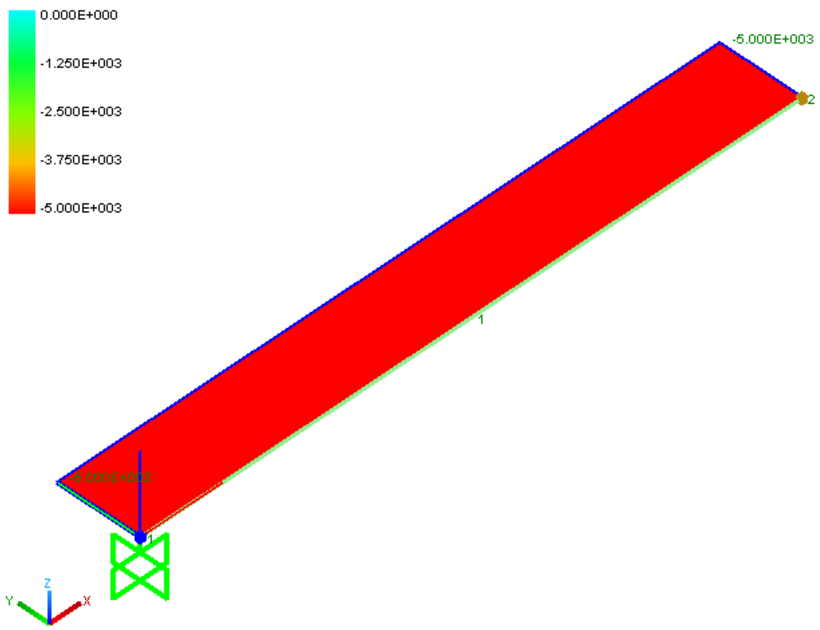
  - Myy



- Shear Diagram:
  - Vy



  - Vz



- Internal displacement in y direction: values from ***Results>Extract data***

Position [mm]	Displacement [mm]
0	0
281.8	0.01513
1250	0.2455
2218	0.6494
2500	0.7803

- Internal displacement in z direction: values from ***Results>Extract data***

Position [mm]	Displacement [mm]
0	0
281.8	-0.01214
1250	-0.1823
2218	-0.4774
2500	-0.5729

- Hand Calculations:
  - Moment diagram:
    - Mzz

$M_{\max} = P_z l = 25000000 Nmm$;

    - Myy

$M_{\max} = P_y l = 12500000 Nmm$

  - Shear Diagram:

- Vy

$V_{\max} = P_z = 10000N$;

- Vz

$V_{\max} = P_y = 5000N$;

- Axial force Diagram:

$N_{\max} = 0N$;

- Displacement in x direction: Node 2

$u_{2,x} = 0mm$

- Displacement in y direction: Node 2

$$u_{2,y} = \frac{1}{3}\frac{P_y l^3}{EJ_z} + \chi\frac{P_y l}{GA} = 0.7803mm$$

- Displacement in y direction: point at coordinate x

$$u_{x,y} = \frac{1}{6}\frac{P_y x^2 \left(3l - x\right)}{EJ_z} + \chi\frac{P_y x}{GA}$$

Coordinate x [mm]	Displacement [mm]
0	0
281.8	0.01513
1250	0.2455
2218	0.6493
2500	0.7803

- Displacement in z direction: Node 2

$$u_{2,z} = \frac{1}{3}\frac{P_z l^3}{EJ_y} + \chi\frac{P_z l}{GA} = -0.5729mm$$

- displacement in z direction: point at coordinate x

$$u_{x,z} = \frac{1}{6}\frac{P_z x^2 \left(3l - x\right)}{EJ_y} + \chi\frac{P_z x}{GA}$$

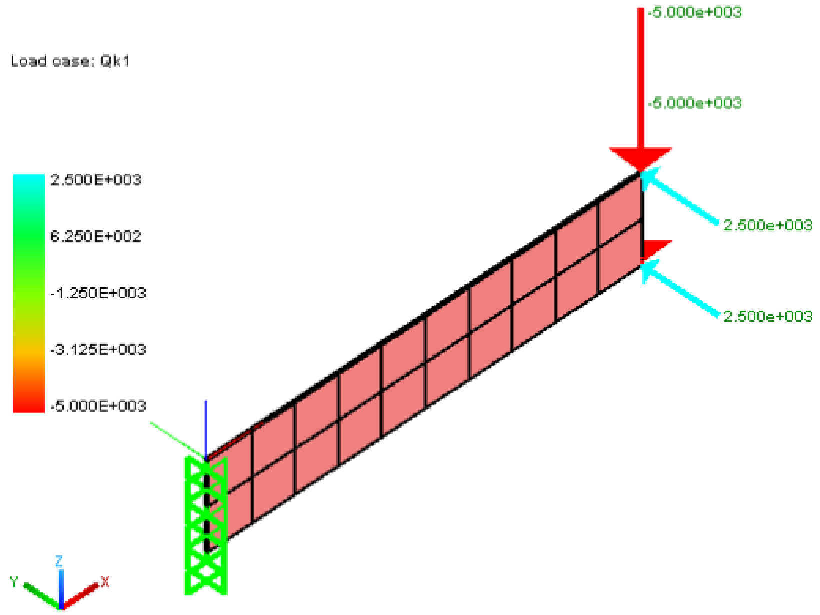
Coordinate x [mm]	Displacement [mm]
0	0
281.8	-0.01214
1250	-0.1823
2218	-0.4773
2500	-0.5729

5.3. Tutorial Three

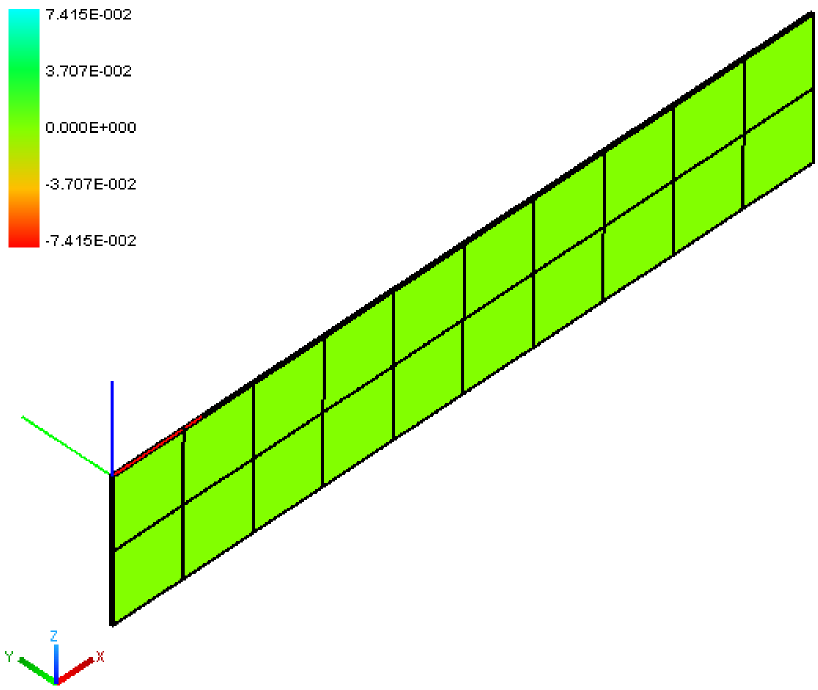
The third tutorial consists in a cantilever beam loaded with points load in direction y with modelled by shell elements (Mindlin-Reissner theory). The output results of NextFEM Designer (Frame forces and displacement) are compared with hand calculations.

⚠ **Only flexural deformations are considered.**

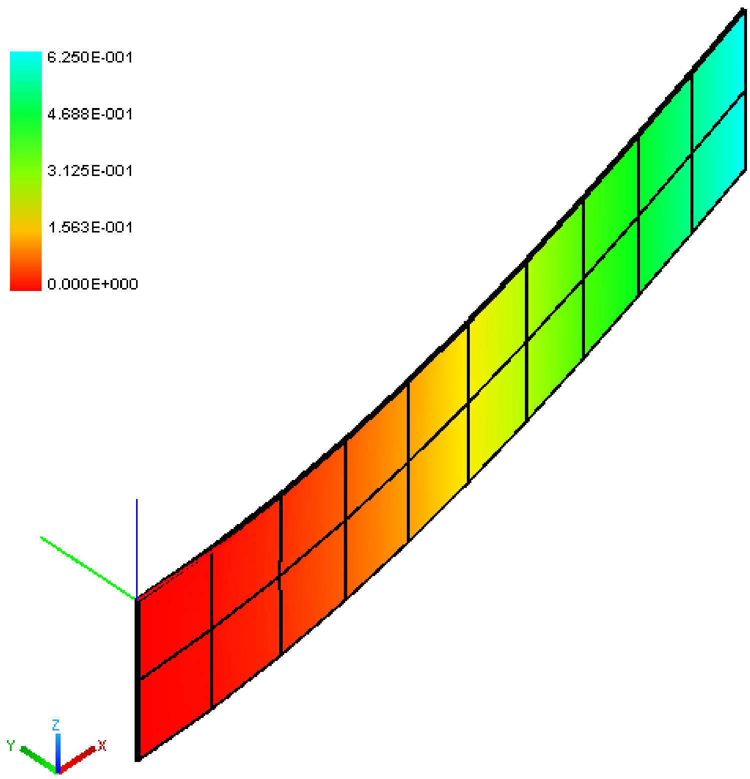
- Units: N for forces and mm for lengths.
- Material Properties:
  - Name: Concrete;
  - E=30000 N/mm^2;
  - Nu=0.3
  - Fk=25 N/mm
  - Weight=2.5^10-5 N/mm^3;
  - Mass=2.55^10-9 N/mm^2/g
- Section properties:
  - B=300 mm (y direction); Planar section;
- Geometric properties:
  - L=5000 mm;
- Loads:
  - Py=5000 N;
  - Pz=10000 N;
- Mesh size: 250x250 mm



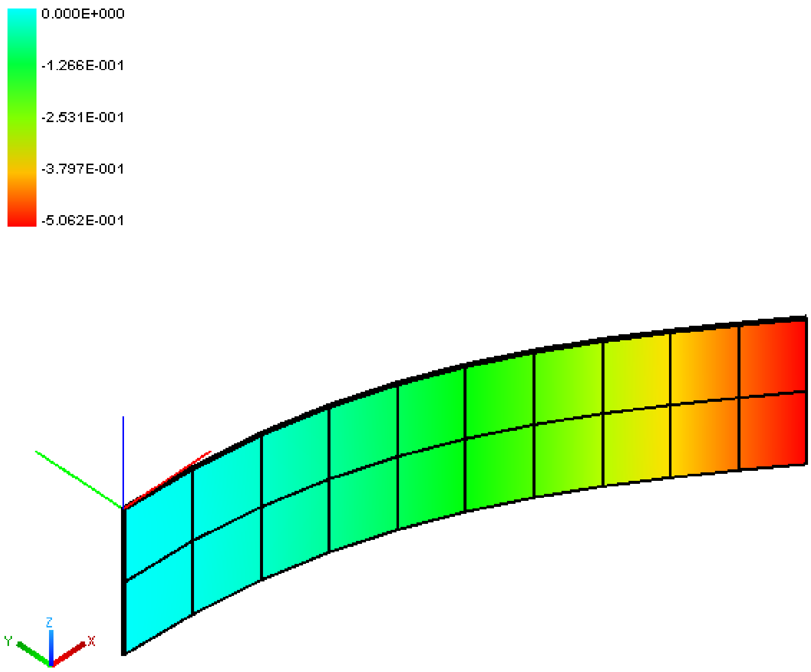
- NextFEM Designer's Results:
  - Displacement in x direction: Node 2=0.00mm



- Displacement in y direction: Node 2=0.6250 mm



- Displacement in z direction: Node 2=-0.5062mm

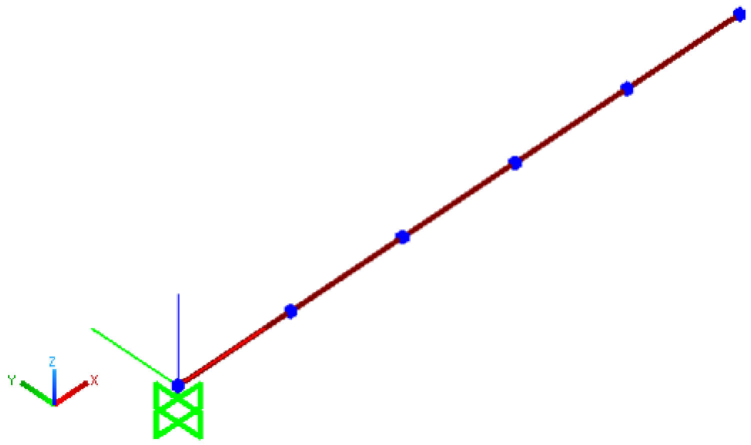


- Comparison with hand Calculations (see tutorial two):
  - Displacement in y direction
    - Hand Calculation: 0.77160mm
    - NextFEM designer: 0.6250mm
    - Percent difference 19%
  - Displacement in z direction:
    - Hand Calculation: -0.5556mm
    - NextFEM designer: -0.5062mm
    - Percent difference: 9%

Note that the difference is due to the choice of the mesh size.

5.4. Tutorial Four

The fourth tutorial consists in a cantilever beam loaded with distributed loads in directions x, y and z. The output of NextFEM Designer (Frame forces and displacement) is compared with hand calculations.

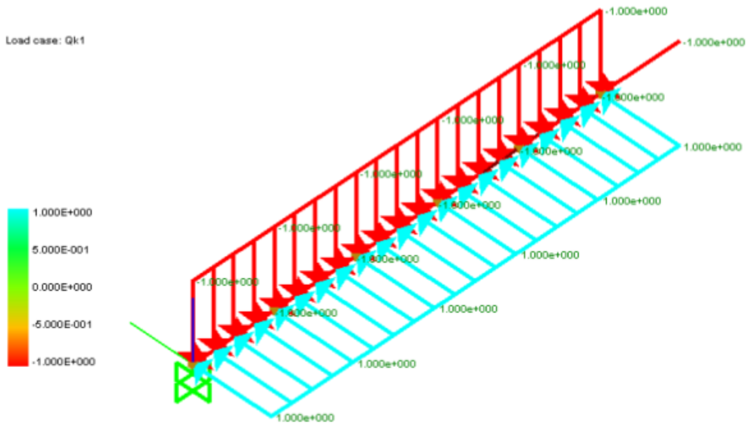


⚠ **<u>Only flexural deformations are considered.</u>**

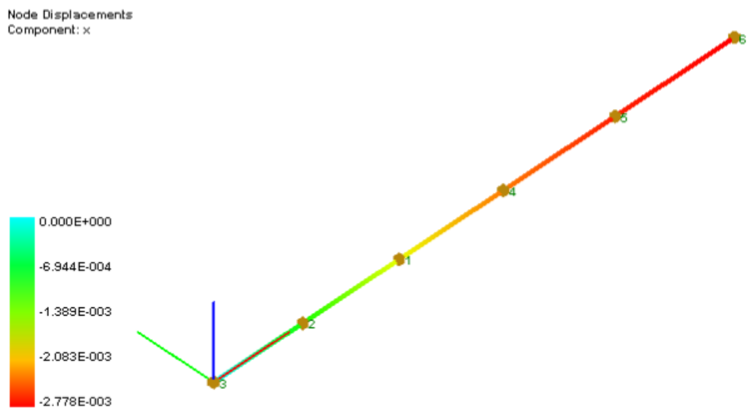
- Units: N for forces and mm for lengths.
- Material Properties:

- Name: Concrete;
- E=30000 N/mm^2;
- Nu=0.3
- Fk=25 N/mm
- Weight=2.5^10-5 N/mm^3;
- Mass=2.55^10-9 N/mm^2/g

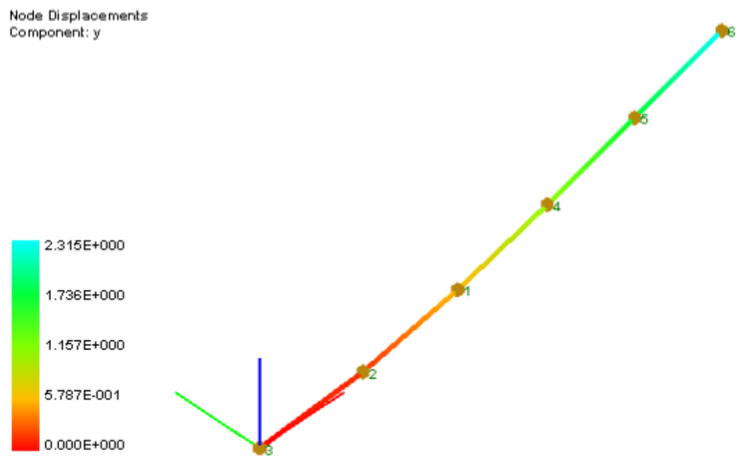
- Section properties:
  - B=300 mm (y direction);
  - H=500mm (z direction);
- Geometric properties:
  - L=5000 mm;
- Loads properties:
  - qy=1 N/mm;
  - qz=-1 N/mm;
  - qx=-1 N/mm.



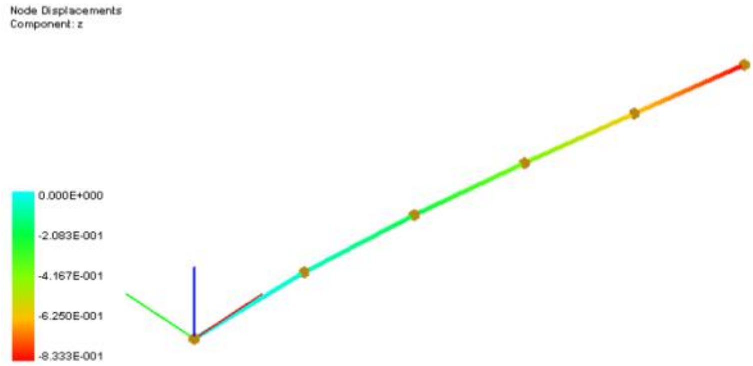
- NextFEM Designer’s Results:
  - Displacement in x direction: Node 6=-0.00278mm



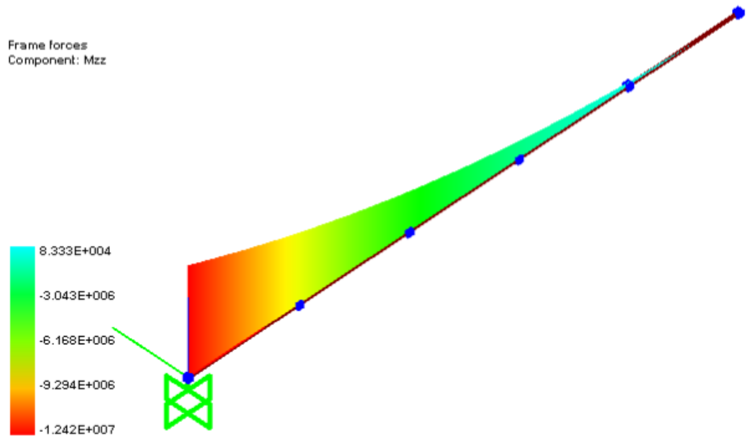
  - Displacement in y direction: Node 6=2.315mm



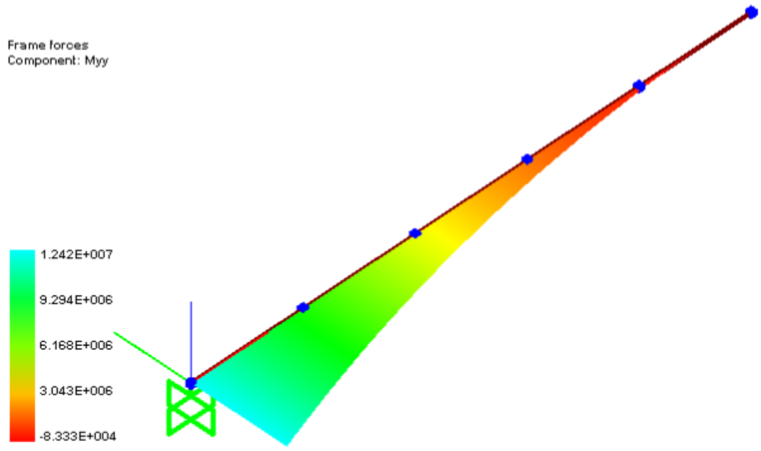
- Displacement in z direction: Node 6=-0.8333mm



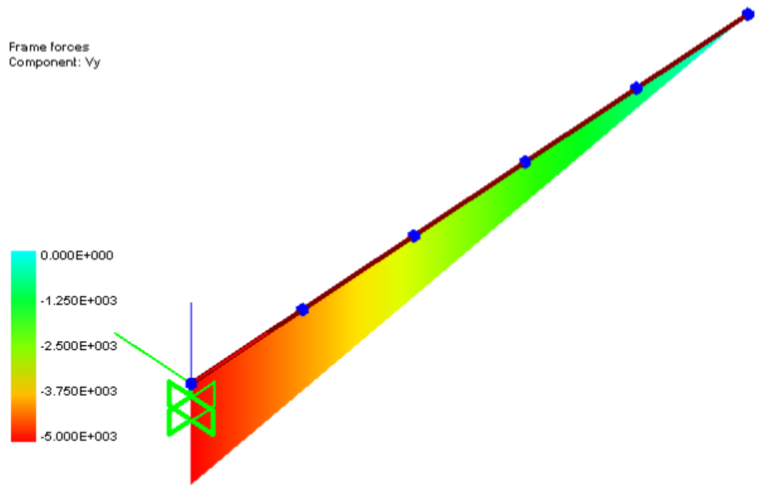
- Moment Diagram: Values from ***Results>Extract Data***
  - Mzz max: node 1: 125000000000 Nmm



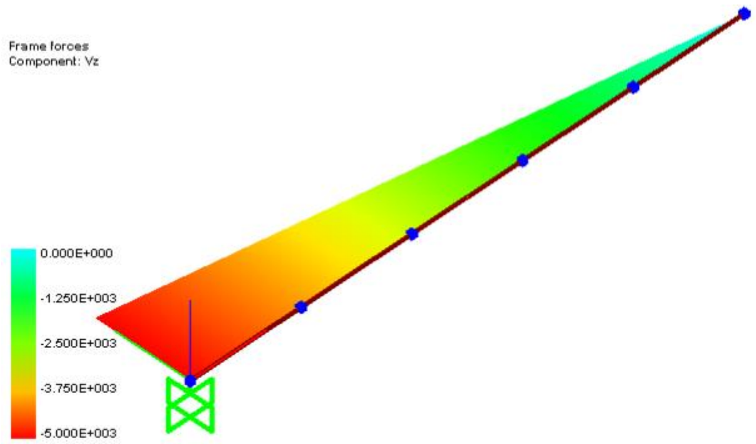
  - Myy max: node 1: 125000000000 Nmm



- Shear Diagram:
  - Vy



  - Vz



- Hand Calculations:
  - Moment diagram:
    - Mzz

$$M_{\max} = \frac{q_z l^2}{2} = 125000000000 Nmm;$$

  - Myy

$$M_{\max} = \frac{q_y l^2}{2} = 125000000000 Nmm;$$

- Shear Diagram:
  - Vy

$$V_{\max} = q_z l = 5000N;$$

  - Vz

$$V_{\max} = q_y l = 5000N;$$

- Axial force diagram:

$$N_{\max} = q_x l = 5000N;$$

- Max Displacement in x direction: Node 6

$$u_{6,x} = \frac{q_x l^2}{2EA} = 0.00278mm$$

- Max Displacement in y direction: Node 6

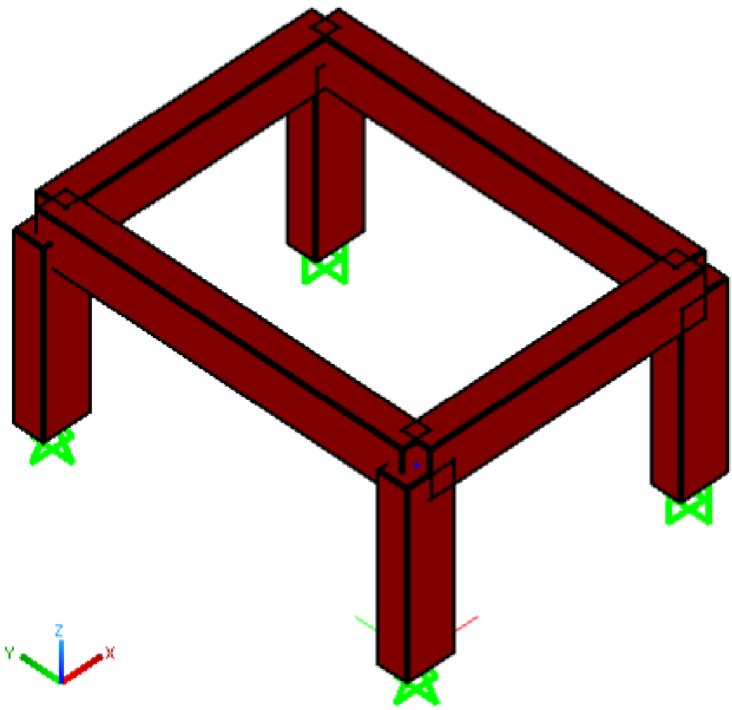
$$u_{6y} = \frac{q_y l^4}{8EJ_y} = 2.315mm$$

- Max Displacement in z direction: Node 6

$$u_{6,z} = \frac{q_z l^4}{8EJ_z} = -0.8333mm$$

5.5. Tutorial Five

The fifth tutorial consists in a modal analysis of a 3D wooden frame-building . The output of NextFEM Designer (modes of vibration) is compared with the output of SAP2000®.



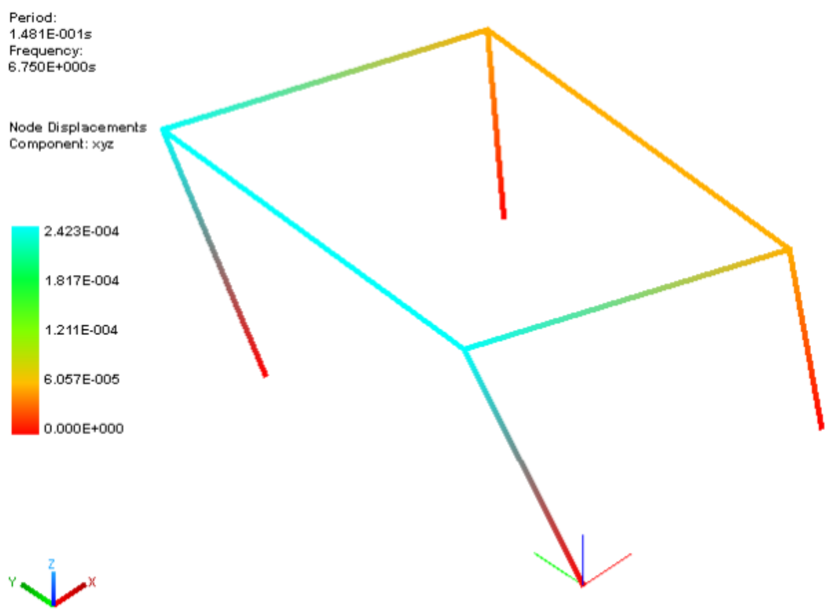
⚠ **Only flexural deformations are considered.**

- Units: kN for forces and m for lengths.
- Material Properties:

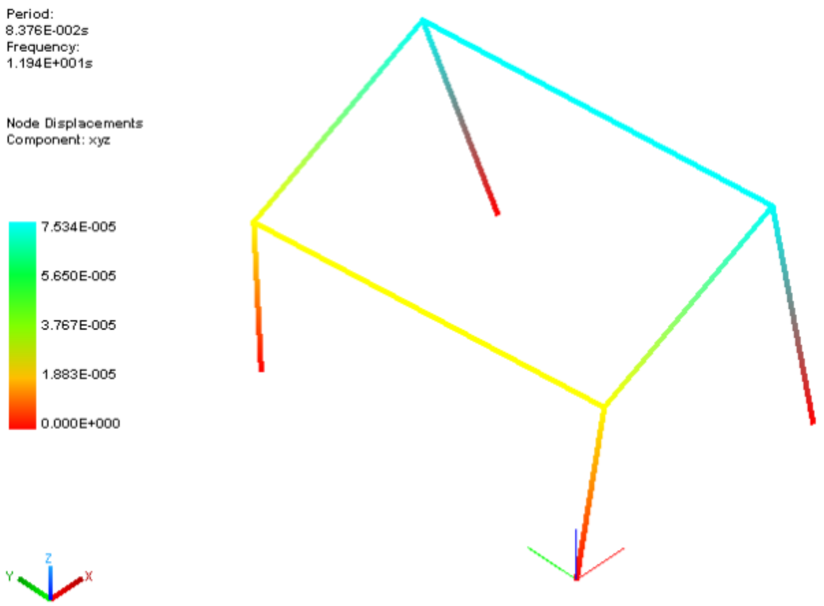
- Name: GL24H;
- E=9.40^10+6 kN/m^2;
- Nu=0.3
- Weight=3.8 kN/m^3;
- Mass=0 kN/m^3/g

- Section properties:
  - B=300 mm (y direction);
  - H=500mm (z direction);
- Geometric properties:
  - Lx=3 m;
  - Ly=4m;
  - Lz=2m;
- Mass properties: at every nodes of the 1st storey
  - my=2.5 kN/g;
  - mz=-2.5 kN/g;
  - mx=-2.5 kN/g

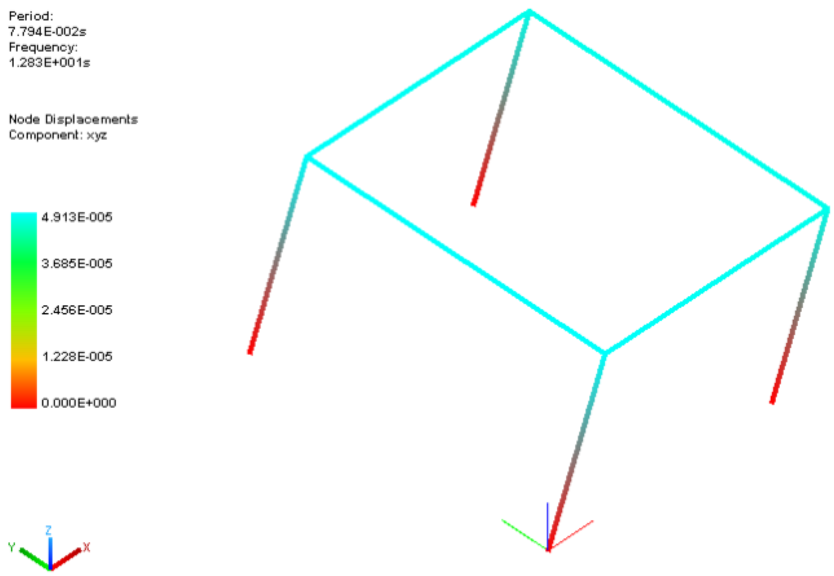
- NextFEM Designer's Results:
  - First mode:



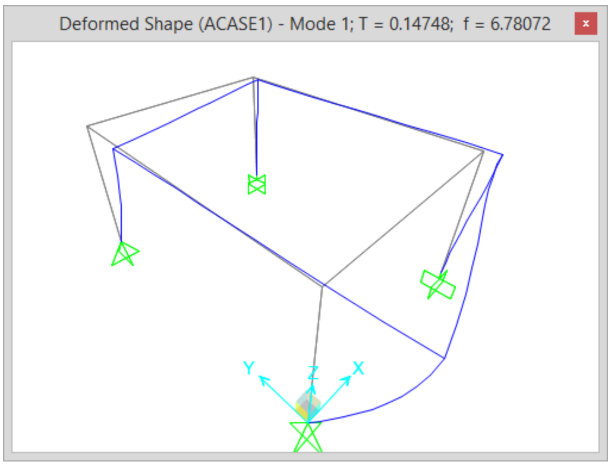
  - Second mode:



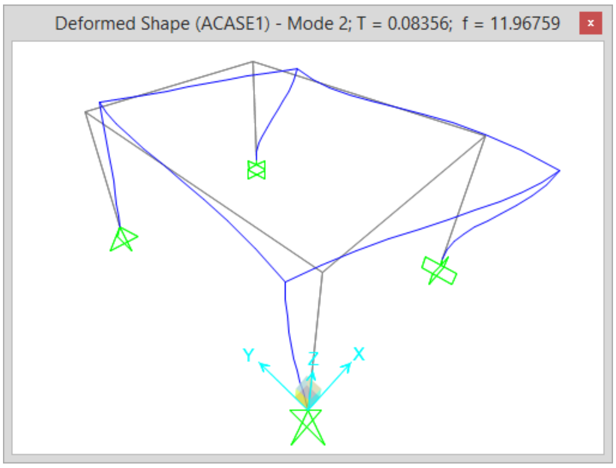
- Third mode:



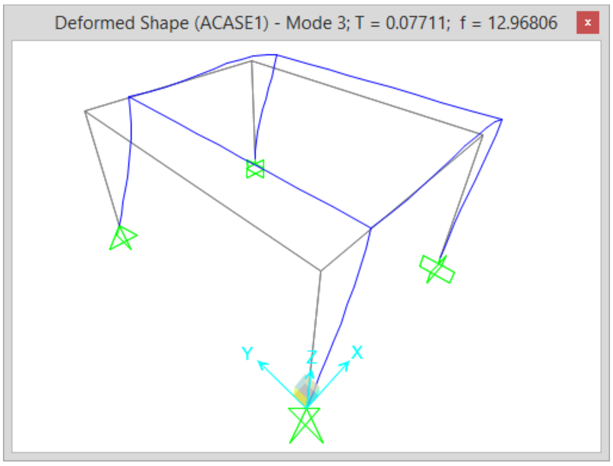
- SAP2000® results:
  - First mode:



- Second mode:



- Third mode:

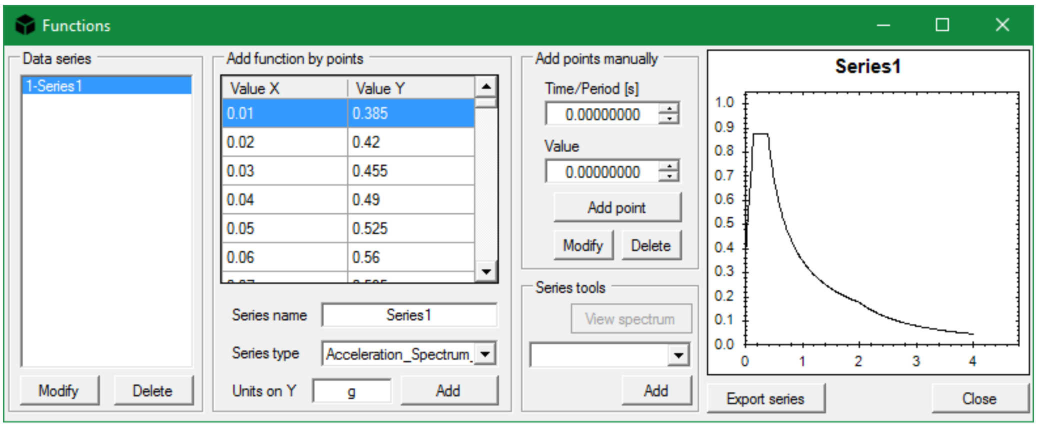


The test model, calculated with two different programs, exhibits comparable results.

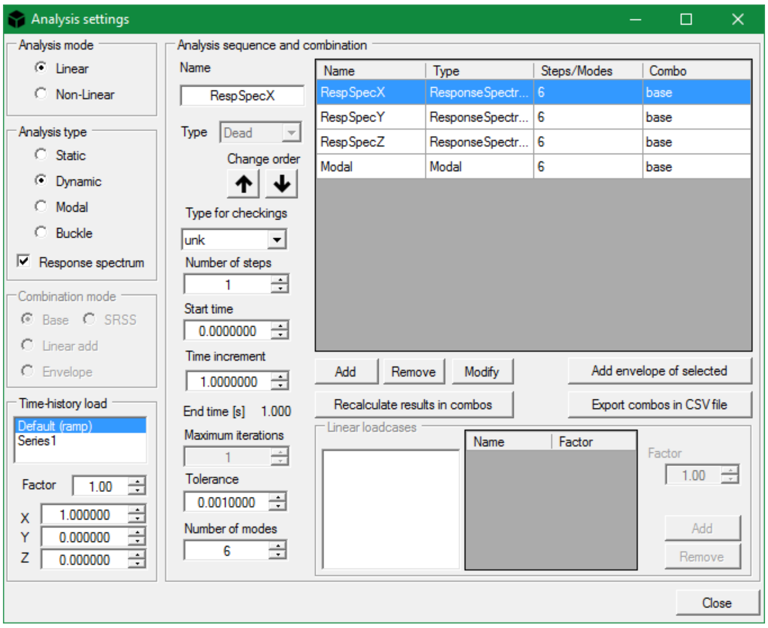
5.6. Tutorial Six

The sixth tutorial consists in a response spectrum analysis of a 3D wooden frame-building, as showed in tutorial five.

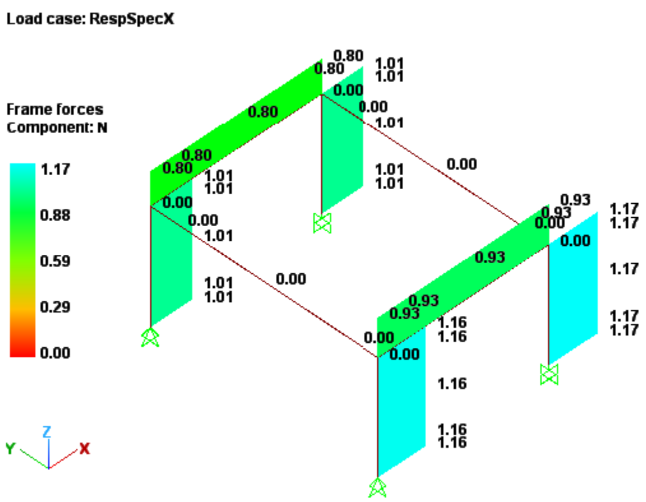
The acceleration spectrum employed is generated from inside the *Functions* form, and it is consistent with the Eurocode 8 Type 1 spectrum, on soil A and with PGA 0.35g.



The Response Spectrum linear dynamic analysis is performed separately in each spatial direction.

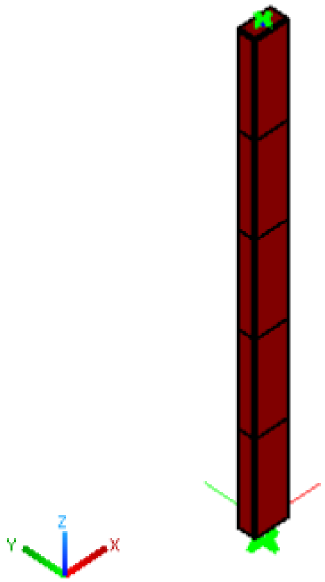


⚠ WARNING: all the results given by a *ResponseSpectrum* analysis must be interpreted without sign.



5.7. Tutorial Seven

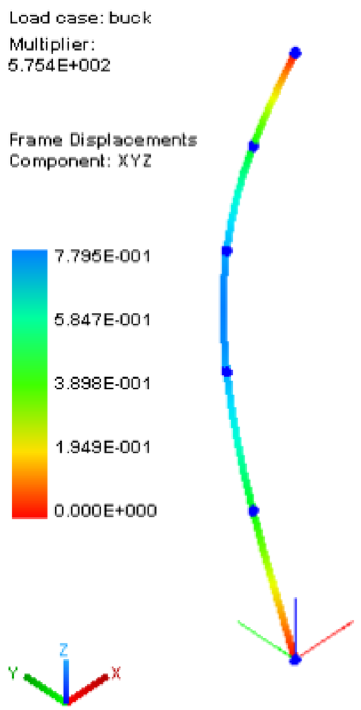
The sixth tutorial consists in a buckling analysis of a simply supported concrete column. The output of NextFEM Designer (i.e. the eigenvalues representing the load multipliers) is compared with the results computed by the Eulerian instability theory. The column is meshed into 5 elements of equal length.



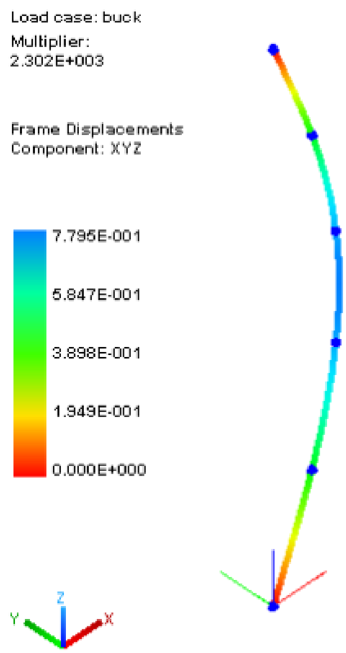
Only flexural deformations are considered.

- Units: kN for forces and m for lengths.
- Material Properties:
    - Name: C25/30;
    - E=3.15e+6 kN/m^2;
    - ν=0.2
    - Weight =25 kN/m^3;
    - Mass =2.5 kN/m^3/g
- Section properties:
    - B=100 mm (z);
    - H=200 mm (y);
- Geometric properties:
    - Ltot=3.0 m;
- Loads:
    - Qz=-1 kN;

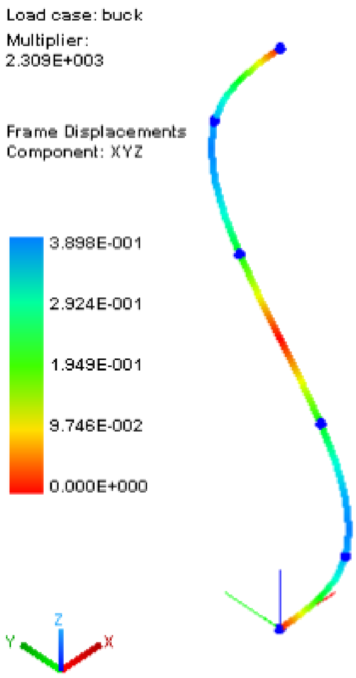
- NextFEM Designer's Results:
    - First mode:



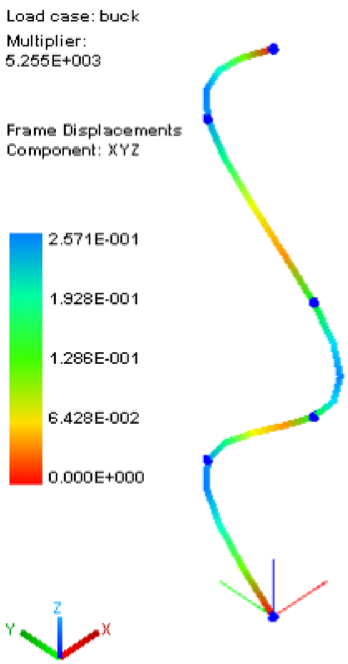
- Second mode:



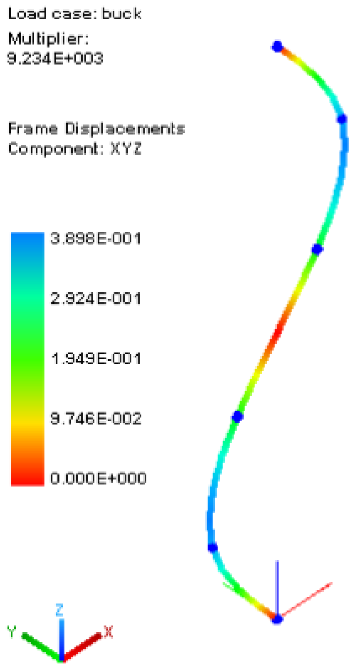
- Third mode:



- Fourth mode:



- Fifth mode:



- Manual calculation:
  - The critical load is computed as come $P_{cr} = \pi^2 \dfrac{EJ}{(l/n)^2}$ with $n = 1, 2, 3, \ldots$, and the inertia J of the direction of inflection.
  - Section inertia:

$$J_{yy} = \frac{1}{12} hb^3 = 16.7 \cdot 10^6 mm^4$$

$$J_{zz} = \frac{1}{12} bh^3 = 66.7 \cdot 10^6 mm^4$$

  - Theoretical results:

Bending around yy	Bending around zz
$\pi^2 \dfrac{EJ_{yy}}{l^2} = 575kN, \lambda = 575$ $\pi^2 \dfrac{EJ_{yy}}{(l/2)^2} = 2301kN, \lambda = 2301$ $\pi^2 \dfrac{EJ_{yy}}{(l/3)^2} = 5178kN, \lambda = 5178$	$\pi^2 \dfrac{EJ_{zz}}{l^2} = 2301kN, \lambda = 2301$ $\pi^2 \dfrac{EJ_{zz}}{(l/2)^2} = 9206kN, \lambda = 9206$

NextFEM Designer's results are in agreement with the theoretical results.

6. License Terms

END USER LICENSE AGREEMENT

By using the software provided by NextFEM SRLS, user explicitly agreed to the following terms and conditions. The contents of NextFEM website and of the supplied software, including the following license terms, could be changes and/or updated at any time. By using the software supplied by NextFEM SRLS or our website, the privacy policy of NextFEM SRLS, published on https://www.nextfem.it/it/privacy-policy/, is accepted.

Art. 1 – User license –

1. On the basis of the following terms and conditions, NextFEM SRLS, in the person of its in office pro tempore legal representative (specified as "Company" or "Licensor" in the following), grants to the Customer ("User" or "Licensee" in the following) the license to use the software (or "Program" or "Programs" in the following) provided by NextFEM SRLS, for PC and for Windows® operating system, including manuals and documentation. The license hereby granted is not exclusive and not transferable in any case.

2. The present agreement does not transfer to the Licensee the source code of the supplied software, neither the logic and/or design documentation.

Art. 2 – Duration – Agreement termination –

1. The present agreement is valid for one year, starting from the license issue date. At the end of the year, this agreement will be considered automatically rescinded, unless a renewal request by the Customer is received by NextFEM SRLS via email at licensing@nextfem.it at least 30 days before the license expiration, or unless the Customer buys the renewal. Different conditions may apply for Educational licenses.

2. NextFEM SRLS has the right to terminate earlier the present agreement due to gross negligence or wilful misconduct of the Customer and/or for violations of the present agreement. Termination will be communicated via email to the address supplied by the Customer during registration onto the Company's website. In any case, the Licensor has the right to the compensation for damage.

Art. 3 - License delivery -

1. The free software provided by NextFEM SRLS which is freely available can be independently downloaded by the Customer from the site nextfem.it.

2. The paid software provided by NextFEM SRLS which is subjected to a fee can be independently downloaded by the Customer from the site nextfem.it. License request are fulfilled by the Licensor in the indicative and not binding term of 7 days after the reception of the payment. Different conditions may apply for Educational licenses.

3. In any case, NextFEM SRLS is not responsible of any damage directly or indirectly connected to delays not dependent on its will during the release of the license.

Art. 4 – Installation -

1. NextFEM software is auto-installing. Once the installation is performed, the software is considered as accepted by the Customer.

2. Any other service (e.g. installation, verification, assistance requested by the Customer to let his employees to use NextFEM software) will be performed by NextFEM SRLS after a Customer's request and in any case after a quotation made by NextFEM Srls and its acceptance by the Customer.

3. The Customer is solely responsible that his technological equipment (hardware and software) meets the minimum and essential requirements to install and use the software, as indicated in the users' manual included in the program.

Art. 5 – Programs usage –

1. The Customer must use NextFEM software for lawful and legal purposes.

2. The Customer undertakes to not remove or alter any trademarks, serial number or other information related to right reservation, which are included in software produced by NextFEM SRLS, even after the termination of the present agreement.

3. According to the Italian law art. 64 ter Law 22nd of April 1941, n. 633 as amended and supplemented and art. 5 par. 2 Directive 2009/24/EC of the European Parliament and of the Council of 23 April 2009 as amended and supplemented, it is allowed to the Licensee Customer, having the right to use a copy of the software produced by NextFEM SRLS, to make a back-up copy of the software and of the included documentation, so far as it is necessary for that use.

4. According to the Italian law art. 64 bis, lett. a) and b) ,64 ter Law 22nd of April 1941, n. 633 as amended and supplemented and art. 4 par. 1 lett. a) and b) Directive 2009/24/EC of the European Parliament and of the Council of 23 April 2009 as amended and supplemented, it is explicitly forbidden to the Customer the permanent or temporary reproduction of the software produced by NextFEM SRLS and of the included documentation, by any means and in any form, in part or in whole, without the authorisation by the right-holder. In so far as loading, displaying, running, transmission or storage of the aforementioned software necessitate such reproduction, even such acts shall be subject to authorisation by the right-holder. At the same conditions and with the same restrictions, it is equally forbidden to the Customer the translation, adaptation, arrangement and any other alteration of the software produced by NextFEM SRLS and of the included documentation and the reproduction of the results thereof, without prejudice to the rights of the person who alters the program.

5. The acts of the aforementioned paragraphs 2 and 3, even when they are necessary for the use of the aforementioned software by the lawful Licensee Customer in accordance with its intended purpose, including for error correction, are subjected to the authorisation by the right-holder.

6. According to the Italian law art. 64 ter, subparagraph 3 Law 22nd of April 1941, n. 633 as amended and supplemented and art. 5 par. 3 Directive 2009/24/EC of the European Parliament and of the Council of 23 April 2009 as amended and supplemented, it is allowed to the Licensee Customer to observe, study or test the functioning of the software produced by NextFEM SRLS and of which he holds the License, in order to determine the ideas and principles which underlie any element of the program if he does so while performing any of the acts of loading, displaying, running, transmitting or storing the program which he is entitled to do.

7. According to the Italian law art. 64 quater Law 22nd of April 1941, n. 633 as amended and supplemented and art. 6 Directive 2009/24/EC of the European Parliament and of the Council of 23 April 2009 as amended and supplemented, the previous authorisation of NextFEM SRLS shall not be required where reproduction of the code and translation of its form within the meaning of art. 64bis, lett. a) and b) Law 22nd of April 1941, n. 633 as amended and supplemented and of points (a) and (b) of Article 4 par.1 are done to modify the form of the code and are indispensable to obtain the information necessary to achieve the interoperability of an independently created computer program with other programs, provided that the following conditions are met:
(a) those acts are performed by the Licensee or by another person having a right to use a copy of a program, or on their behalf by a person authorised to do so;
(b) the information necessary to achieve interoperability has not previously been readily available to the persons referred to in point (a);
(c) those acts are confined to the parts of the original program which are necessary in order to achieve interoperability.

8. The provisions of preceding paragraph 7 shall not permit the information obtained through its application:
(a) to be used for goals other than to achieve the interoperability of the independently created computer program;
(b) to be given to others, except when necessary for the interoperability of the independently created computer program;
(c) to be used for the development, production or marketing of a computer program substantially similar in its expression, or for any other act which infringes copyright.

9. In accordance with the provisions of the Berne Convention for the protection of Literary and Artistic Works, enacted in Italy with the Italian law 20th of June 1978, n. 399 as amended and supplemented, the provisions of article 64 quater Law 22nd of April 1941, n. 633 as amended and supplemented and art. 6 of Directive 2009/24/EC of the European Parliament and of the Council of 23rd April 2009 as amended and supplemented may not be interpreted in such a way as to allow its application to be used in a manner which unreasonably prejudices the right-holder's legitimate interests or conflicts with a normal exploitation of the computer program.

Art. 6 – Property – Transfer prohibition –

1. The software provided by NextFEM SRLS and the included documentation remain exclusive property of NextFEM SRLS. The Customer is explicitly forbidden to distribute products of NextFEM SRLS or copies to anyone or to sell them or assign them in license to third parts or to lease them, or in any case to allow others to use the programs, either in exchange for payment or not. In these cases, NextFEM SRLS can revoke the user's license of the free program or of any of the paid modules at any time.

2. This clause will remain in force even after the rescission or the termination to any title of this contract.

Art. 7 – Right holders – Secret – Modifications –

1. The software provided by NextFEM SRLS, the included documentation, the program code, its layout, the structures and the program files organization, the program name, the Company logo and any other representation form within the software are subjected to copyright; this one, and any rights coming from it or in any way connected to the copyright are property of NextFEM SRLS. Other trademarks belong to the respective owners.

2. The Customer is required to keep secret the content of the software provided by NextFEM SRLS and the included documentation, and to protect NextFEM SRLS and his suppliers' rights; in particular, the Customer is required either to make no modifications to

the software provided by NextFEM SRLS or to incorporate it entirely or in part in other software without preventive written authorisation by NextFEM SRLS, without prejudice to current mandatory legislation on the matter. In these cases, NextFEM SRLS can revoke the user's license of the free program or of any of the paid module at any time.

3. This clause will remain in force even after the termination or the expiration in whatever manner of this agreement.

Art. 8 – Fee – Solve et repete –

1. NextFEM SRLS provides the software "as is" and is not obliged to provide maintenance, support, updates, improvements or changes. Different conditions may apply to paid software and for Education licenses.

2. To ensure the continuity of the license and of paid modules, the Customer must pay the relative fee at least 15 days before the current license expires; if the deadline is not respected, NextFEM SRLS can't guarantee such continuity, and the licensed modules may be blocked. For no reason the payment of the annual fee regarding the software and/or its single component module and/or required services can be delayed or suspended; eventual exceptions or Customer's disputes will be managed and solved separately.

3. NextFEM SRLS provides software updates for 12 (twelve) months starting from the delivery date, limited to the functionalities of paid module/s. During this period, encountered malfunctions in paid modules will be fixed to ensure the correct functionality. This guarantee does not apply to functionalities not included in the paid modules.

Art. 9 – Warranty and liability –

1. For both the basic software version and the paid modules, NextFEM SRLS provides the software "as is" and it is not obliged to provide maintenance, support, updates, improvements or changes. During the validity of this agreement, eventual software updates or patches may be released.

2. NextFEM SRLS is committed, only for functionalities of the paid modules, and for 12 months from the purchase, to keep the software able to perform the tasks described in the user manual. During this period, encountered malfunctions of the paid modules will be fixed to ensure the correct functionality. This guarantee does not apply to functionalities not included in paid modules, nor to Educational licenses, as stated in art. 14 of this agreement.

3. The warranty is conditioned to the correct original functioning of the Client's machine, hardware and system software and the existence of the minimum requirements prescribed for the correct software installation, as well to the circumstances in which the Customer installs the updates and patches that can be made available by the Licensor via an independent download made from the nextfem.it website, and also to the correct use of the system and software by the Customer.

4. The Customer is the sole responsible for the choice of the software produced by NextFEM SRLS and its compliance to his own needs and purpose of use, for any input fed to the software and any output coming from the program or from its parts, and must verify results, reports and the checks conducted with it.

5. The software provided by NextFEM SRLS are a representation of the current state of development, so NextFEM SRLS cannot grant that they will always work correctly in every applications and in any situation.

6. Customer is responsible of installation, launch, and usage of the software produced by NextFEM SRLS and of the application of the related updates and patches, their transfer to the computer, the settings, and everything not explicitly stated in this Contract that burden on NextFEM SRLS.

7. This warranty is not valid whenever a software fault is due to accident, improper and/or non-conforming and/or wrongful usage. Any change to the software made directly by the Customer will result in the withdrawal of this warranty.

8. NextFEM SRLS does not take any responsibility and is not liable for any direct and/or indirect, special, collateral, incidental and/or consequential damage, including lost profits, incurred by the Customer or third parts caused through the use or lack of use of the software and by any means related and/or consequential to eventual software quality, adequacy, use and usability flaws, which are therefore to be exclusively borne by the Customer, except as what is compulsorily required by law.

9. NextFEM SRLS does not take any responsibility and is not liable for any direct and/or indirect, special, collateral, incidental and/or consequential damage, including lost profits, incurred by the Customer or third parts caused through the use or lack of use of the software and by any means related and/or consequential to eventual software quality, adequacy, use and usability, caused by suppliers or by parts of program developed by third parts. To these parts of programs developed by third parts are applied their own licensing conditions, which can be found inside NextFEM SRLS software from *? / Information …*.

Art. 10 – Software restitution - Software deletion –

1. Within one month from the termination of this agreement for any reason, the Customer must delete and eliminate any copy of the software he owns, even if they are backup copies. The Customer must confirm this by sending to NextFEM SRLS an e-mail within the same deadline.

2. As a consequence of the termination of this agreement as in the previous subparagraph, the license of use is revoked and cannot be use anymore by the Customer for any purpose.

Art. 11 – Support and/or consultation -

1. Upon Customer's request, NextFEM SRLS is willing to give, by a preventive stipulation of specific separate contracts, the necessary support and/or consultation to maintain or launch or update or personalize or implement the software provided by NextFEM SRLS, also for a potential training of Customer's staff who is appointed for its use.

2. The possible existence of other contractual relationships between NextFEM SRLS and the Customer does not affect other connections between them, that will remain separated and independent.

3. The free licenses of use granted by NextFEM SRLS (i.e. basic program, Educational license, etc.) are not covered by any kind of support.

4. The paid licenses of use granted by NextFEM SRLS to the Customer can be issued with a first-installation assistance via email, until 7 days after the payment. Further paid support can be supplied for one year starting from the license acquisition. Support period can be bought or renewed only together with the program license or renewal, respectively. Assistance is supplied only via email and concerns the sole software use or program functioning. NextFEM SRLS does not supply support on the engineering choices made or to be made for designing any structures. Any advice given by support cannot substitute the engineering judgement of the Customer, who is the sole responsible of the structure designed, analysed and checked with the program, including the obtained results.

Art. 12 – Communication -

1. Any communication from one part to another of this agreement must be sent as a registered letter with signed return receipt or as a hand-delivered registered letter addresses to "NextFEM SRLS, Piazza del Foro Romano 12, 31046 Oderzo (TV)" or as a certified e-mail to nextfem@pec.nextfem.it.

Art. 13 – Litigation – Applied law -

1. This contract is subjected to the Italian law.

2. Any litigation in any case connected to this agreement shall be exclusively of the competence of the Court of Treviso.

Art. 14 – Educational license –

1. Educational licenses are distributed for a predetermined number of PCs prior oral or written agreement with NextFEM SRLS. NextFEM SRLS solely decides the number of distributed licenses prior consultation with the Customer.

2. Educational licenses can be given freely to a private or public Customer, when it is a training institution, a research and development company or a school, at the incontestable discretion of NextFEM SRLS. In such case, the given Educational licenses do not grant to the Customer the right to use them after the planned time period conceded to the Licensee, and they can be revoked at any time by NextFEM SRLS without any justification or notice to the Customer.

3. When given freely, Educational license does not allow for any refund of the cost of the software, for any reason.

4. When given freely, Educational license does not allow for any kind of support supplied by NextFEM SRLS, neither for malfunctioning of the program. Hence, the warranties described in art. 9, paragraph 2 of this agreement are excluded.

Art. 15 – Changes of terms in this agreement –

1. The Licensor has the right to modify the conditions of the present License user's agreement for the software of NextFEM SRLS by email to be sent to the Licensee to the address given during registration on nextfem.it website. The Licensee has the right of withdrawal by sending a registered letter with signed return receipt or as a hand-delivered registered letter addresses to "NextFEM SRLS, Piazza del Foro Romano 12, 31046 Oderzo (TV)" or as a certified e-mail to nextfem@pec.nextfem.it compulsorily within 14 days from the receipt of the communication related to the changes of the agreement terms.

Art. 16 – Final provisions -

1. Whenever one of the clauses contained in this Contract will be declared invalid or without effects, entirely or in part, this will not invalidate the other clauses, except when the Licensor considers in bona fide the clause as essential, and consequently shall ask for the resolution of the contract.

2. For anything not expressively provided in this contract, the Italian Civil Code rules shall be applied and the Legislative Decree of 29th of December 1992, n. 518 as amended and supplemented, regarding the implementation of the Council Directive n. 91/250 CEE of 14th of May 1991 on the legal protection of computer programs which modifies and integrates Law 22nd of April 1941, n.

633, and this last law as amended and supplemented, and the Directive 2009/24/EC of the European Parliament and the Council of 23rd of April 2009 as amended and supplemented.

Oderzo (TV), (date of acceptance of this contractual conditions)

NextFEM SRLS Licensee

Pursuant to and in accordance with art. 1341 and 1342 c.c., the Customer specifically approves, for having them read, understood and known, the articles: 2 (Duration – Agreement termination) paragraph 2, 3 (License delivery) paragraph 3, 4 (Installation) paragraph 3, 5 (Programs usage), 6 (Property – Transfer prohibition), 7 (Right holders – Secret – Modifications) paragraph 2, 8 (Fee – Solve et repete) paragraph 2, 9 (Warranty and liability) paragraphs 3,4,5,6,7,8,9, 10 (Software restitution - Software deletion) paragraph 4, 11 (Support and/or consultation), 13 (Litigation – Applied law), 14 (Educational license) paragraphs 2 and 3, 15 (Changes of terms in this agreement), 16 (Final provisions).

Oderzo (TV), (date of acceptance of this contractual conditions)

The Customer

NextFEM Designer uses:

OOFEM v.2.6
ZedGraph Library v.5.1
Clipper2 Library v.1.0.6
netDxf Library v.3.0.1
Poly2Tri Library
RichTextBoxLinks Library
SlimDX Library
Sonic Library
UnitConversionLib Library
DotNetZip Library v1.13.7
Triangle.NET Library Beta 4
Netgen Library v.6.0
with OpenCascade Libraries
PDFSharp v.1.32
RTFWriter Library v.1.0
xBIM Essentials v.5.1
xBIM Geometry v.5.1
xBIM GLTF v.5.1
XbimWebUI for xBIM Toolkit
Desktop Bridge Helpers v.1.1.0
DocX Library v.1.0.0.22
Splicer Library v.1.0
DirectShowLib v.2.1
OpenStreetMap site
IDEA StatiCa IOM
Newtonsoft Json.NET v12.0
AutoUpdater.NET v1.1
SdxSpriteTextRenderer v2.0
RibbonWinforms v5.0.1.0
glTF-CSharp-Loader v1.1.3-alpha
SpreadsheetLight v3.5
WPF-Math v0.11
Collapsible control
OpenTK library
WeifenLuo.WinFormsUI.Docking library

All the licenses, including the general one for the program and for paid modules, can be found in the dialog *?/About …* .