NextFEM **Designer**

Struct module manual

Version 2.4

Summary

Chapter 1
Introduction

NextFEM Designer is an easy-to-use program for performing Finite Element analysis. The program has the Struct module, which imports and checks scaffolds and platforms created in the software PON CAD®, distributed by MEC CAD®.

Installation and first run

NextFEM Designer is designer for Windows 7 SP1 or above and it is available for 64 bit platform.

Program activation

To activate the program it is necessary to be connected to the Internet on the first run. Activation is automatic and without any additional cost. For the modules activation, see the relative paragraph.

Manuals and support

Along with NextFEM Designer is provided an 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.

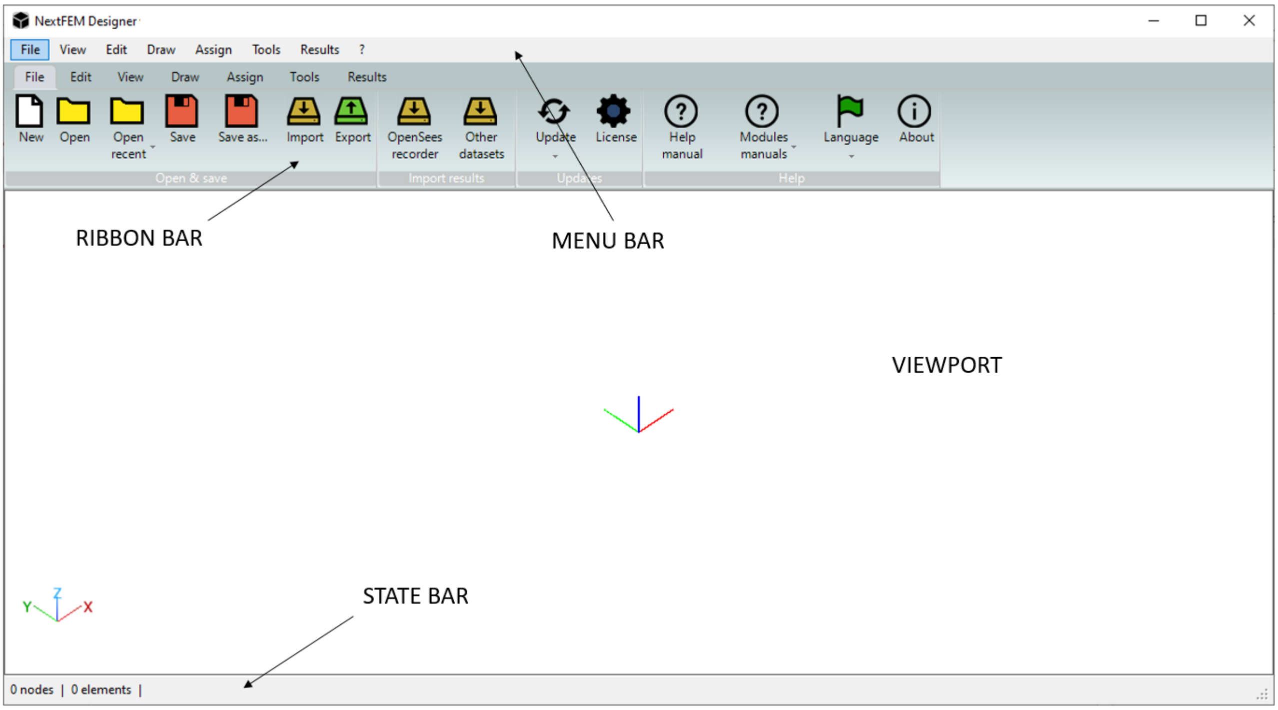
Design codes implemented in the Struct module

The following design codes has been followed to design the Struct module:

1. EN 1993-1-1: Eurocode 3 - Design of steel structures - Part 1-1: General rules and rules for buildings
2. EN 12811-1: Temporary works equipment - Part 1: Scaffolds - Performance requirements and general design
3. EN 1991-1-3 Actions on structures, Part 1-3: General actions – Snow loads
4. EN 1991-1-4 Actions on structures, Part 1-3: General actions – Wind actions

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.



License

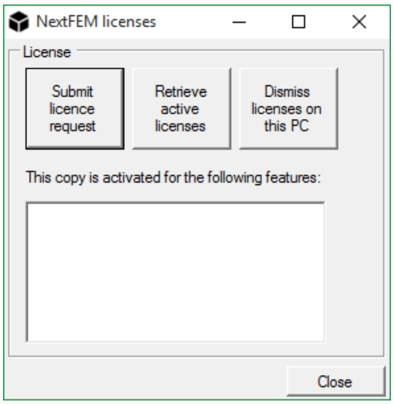
To activate the Struct module select the command *License…*, which handles user's license. Please be connected to the Internet before using it.

<u>License is valid only for the specific version bought and for the machine (PC) on which the user runs the activation procedure.</u>

The button *Submit license request* submits to the NextFEM servers the license request for the machine in which the program is installed. After the user clicked it, it is necessary to communicate via email at licensing@nextfem.it the username for which the license has been requested, and wait for a recall.

The command Retrieve active licenses allows to download the licensing data from NextFEM servers, and store them locally. The module is active if in the white lower box there are the letterings *PONCAD* and *GENDESIGN*.

The command Dismiss licenses on this PC deactivates the installed licenses and allows user to move them to another machine.



Chapter 2
Importing models and launching analysis

Importing the model

Model importing works with 2 files, placed in the same folder:

- *filename.SEZ* which contains the description of sections and materials;
- *filename.DXF* with the scaffold geometry.

  ⚠ WARNING: aside from the extension, the 2 files must have the same name(*filename*).

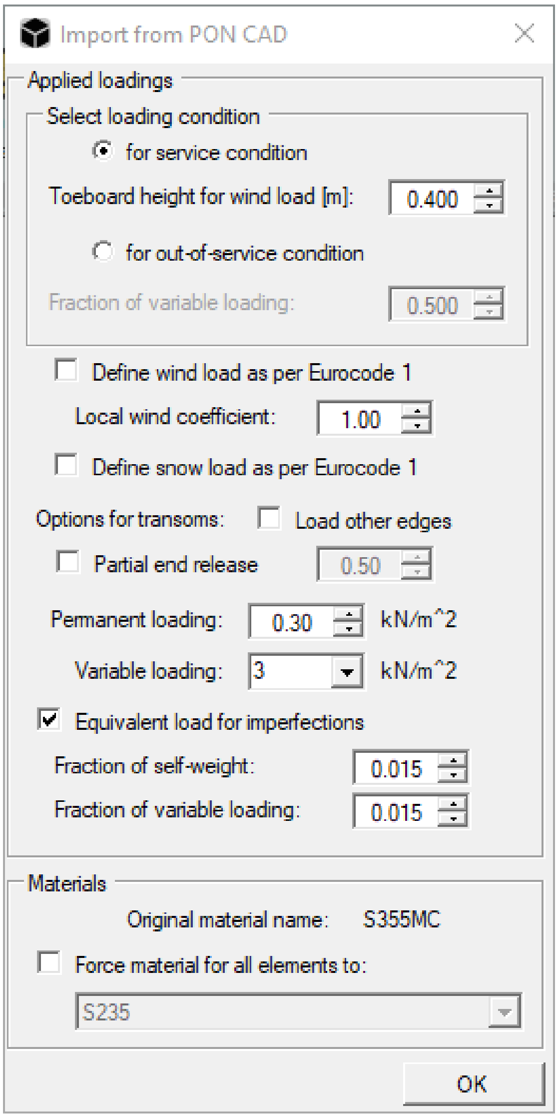
To import the model, select the command File/Import... and select the *.SEZ* file to import, or drag the *.SEZ* into the viewport.

The import module allows to:

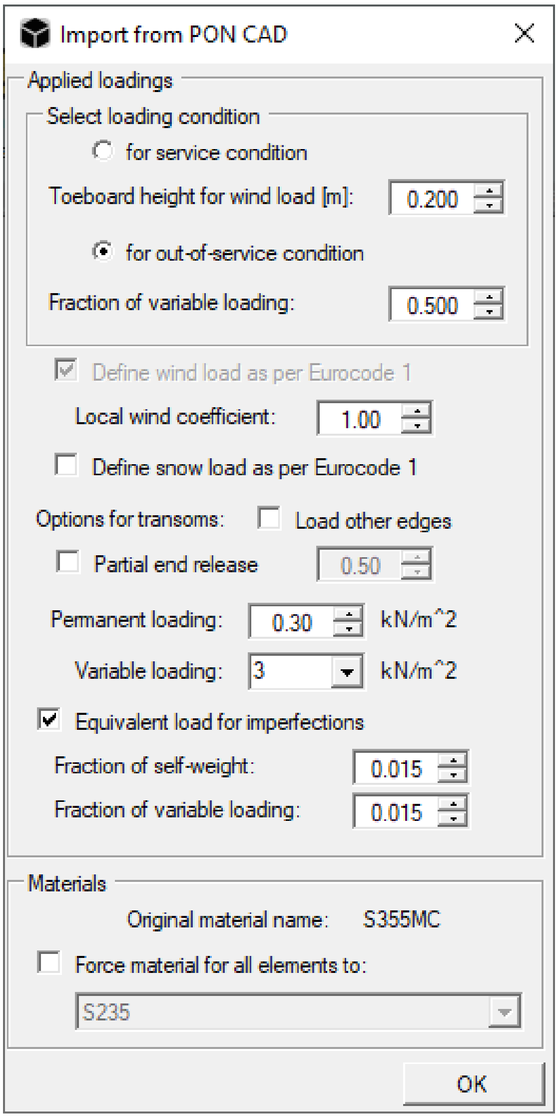
- import the geometry
- import the data for transversal sections
- import the material data
- define the loads to be used in the analysis
- automatically apply the loads to the structure
- compile automatically all the needed load combinations.

Import from PON CAD

Once opened the *.SEZ* file, the following mask shows up. Load and material settings can be defined here.



(a)



(b)

In the *Applied loading* box the following parameters can be set:

- *Select load combination* sets the model for the service or out-of-service load condition. By selecting:
  - *for service condition* the program applies the load as prescribed in EN 12811-1:2004, 6.2.9.2 (a), with a working wind load equal to 0.2kN/m^2 and for the height showed in *Toeboard height for wind load [m]*;
  - *for out-of-service condition* the program applies the load as prescribed in EN 12811-1:2004, 6.2.9.2 (b), with the wind load as defined by Eurocode 1 for the height showed in *Toeboard height for wind load [m]*. The reduction factor for this condition can be set through the value in *Fraction of variable loading*.
- *Define wind load as per Eurocode 1*: to define the wind load to be applied to the structure as per Eurocode 1 (only for European territory).
- *Loca wind coefficient*: to set a reduction factor for wind pressure as per Annex A of EN 12811-1.
- *Define snow load as per Eurocode 1*: to define the snow load as per Eurocode 1 (only for European territory)
- *[Options for transoms] Load other edges:* the program usually applies vertical loads along 2nd and 4th edges of 3DFACE representing the floors. Activating this option will force the beams under the 1st and 3rd edges to be loaded.
- *[Options for transoms] Partial end release:* it allows to force a partial end release to all transoms for moments around local y and z axes. The showed factor represents the percentage of the moment to be transmitted between standard and transom. The end releases are applied to both ledgers and transoms supporting each loading deck.

- *Permanent loading*: reports the dead load to be applied to the decks, identified by 3DFACE elements in the XY plane inside the DXF file. The default values is typical of a thin-walled steel deck.
- *Variable loading*: reports the variable load to be applied to the decks, identified by 3DFACE elements in the XY plane inside the DXF file. Its default values is set to 3.0kN/m^2, as in EN 12811-1:2004, Table 3, for loading class 4.
- *Equivalent load for imperfections*: as requested by EN 12811-1:2004, 10.2.2 (Imperfections), geometric imperfections are applied to the structure if this checkbox is active. They are expressed as the desired percentage of dead and variable loadings. The default values is set to 0.015, equal to the 1.5% of the load for each element. Such imperfections are applied to both X and Y directions, each one with the defined value.

⚠ WARNING: All wind loads, applied for *Height for working wind load [m]* are applied as concentrated loads at the first node of the beam closest to the deck area. The area for which the load is calculated is the half of the length of the deck times the set height.

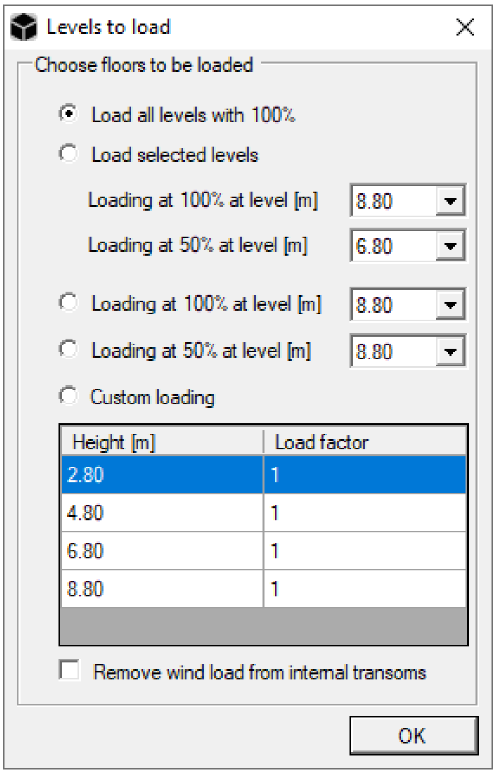
In the *Materials* box, with the *Force material for all elements to* check it is possible to force the application of the chosen material to all elements. By default, with this options unchecked, the program assigns the material specified in the *.SEZ* file.

By pressing OK, the program will ask for the definition of wind and snow loads, if they have been requested.

⚠ WARNING: The SEZ file could be modified for special needs. In this case, please pay attention to the progressive number, that must be unique. Moreover, the lines in which the first field don't start with "SEZ" must report the same name in the third field.

Set loading decks

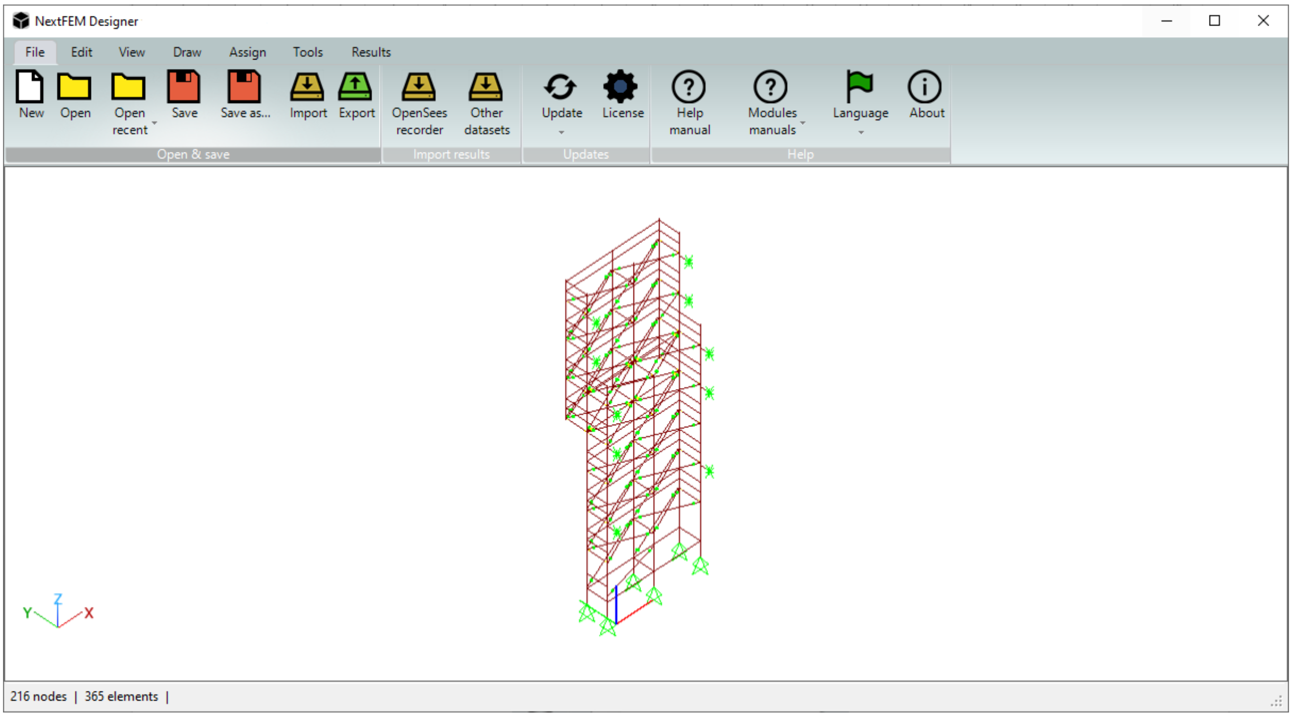
The following mask allows the definition of the loaded decks in the model.



All decks can be loaded with the options *Load all levels with 100%*. Otherwise, as per EN 12811-1:2004, 6.2.9, it is possible to load only one deck with 100% of the load, and the upper or the lower one with the 50%. The option *Load selected levels* allows to choose the floors to which apply the loads (Z level in m). As further options, only one floor can be loaded at 100 or 50% or a custom load factor can be specified for each floor. Then press *OK* to continue.

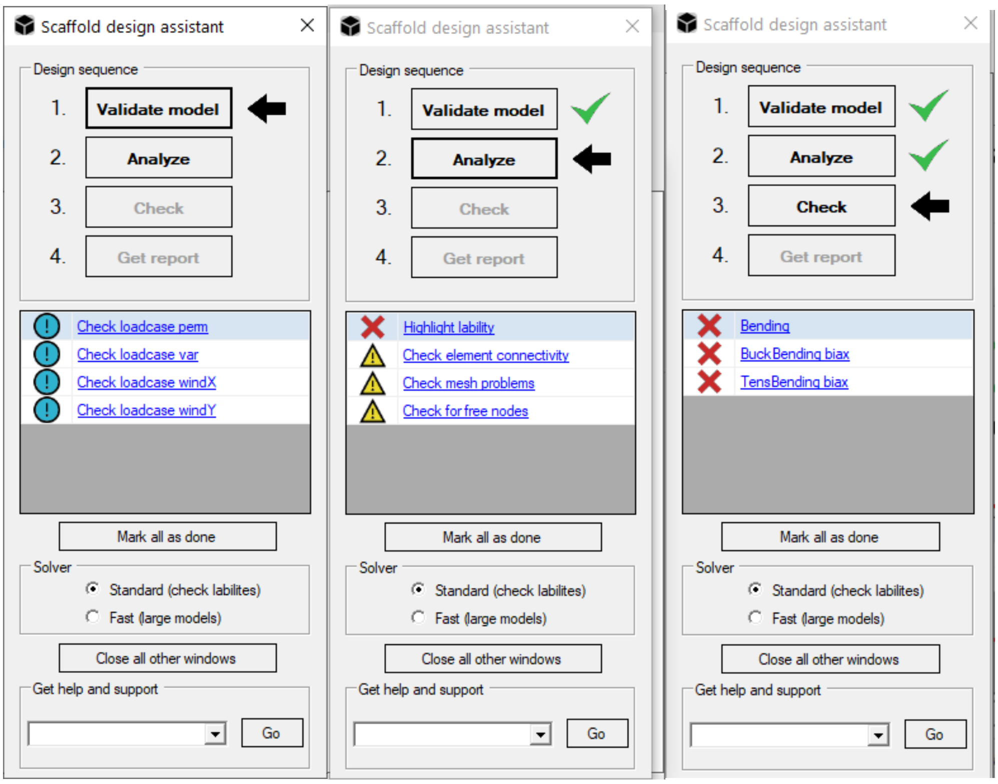
The option *Remove wind load from internal transoms* allows to avoid loading the internal transoms by the wind load, maintaining only the parietal ones. See the chapter describing the wind load in the following.

The resulting model is bounded with restraints in X and Y along the height, and supported at the base.



Finally, the imported file must be saved with the command *Save as…*

Scaffold assistant



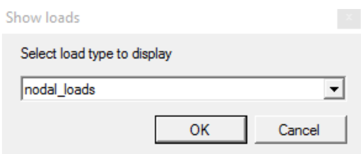
Since v.1.7, Scaffold design assistant is available to guide the scaffold design process.

- Firstly, press on "1. Validate model". By clicking on table rows, the program automatically shows the loading applied to the structure. Always check the load values, especially in case perm (permanent = decks weight) and var (variable loading). Example: 1.8m (influence length of 1 transom) x 3kN/mq (var.) = 5.4 kN/m
- Button "2. Analyze" performs the finite element analysis. Labilities, if present, can be highlighted.
- Button "3. Check" executes the code checking of beams and joints, for the first resisting load combination (ULS1).
- Command "4. Get report" produces the calculation report, including all the results listed and active in «Print model report» window.

The "Get help & support box" in Scaffold design assistant includes link to tutorial, manuals and for assistance purposes.

Check applied loads

After importing, it is very important to check the loads applied to the model. Select the command *View/Display loads…*, and the *beam_loads* from the dialog that appears.



To check also the nodal loads, repeat the procedure and select *nodal_loads* on the drop down menu.

WARNING: Starting from version 1.05, nodal loads are not applied automatically anymore. Beams loads are used instead.

Loads are applied for the following load cases, automatically generated by the program:

- **sw** for the self-weight
- **perm** for permanent loadings
- **var** for variable loads

- **imperf** for the imperfection loads (if they have been required)
- **wind_X** and **wind_Y** for the wind load
- **snow** for snow loads (if applied)

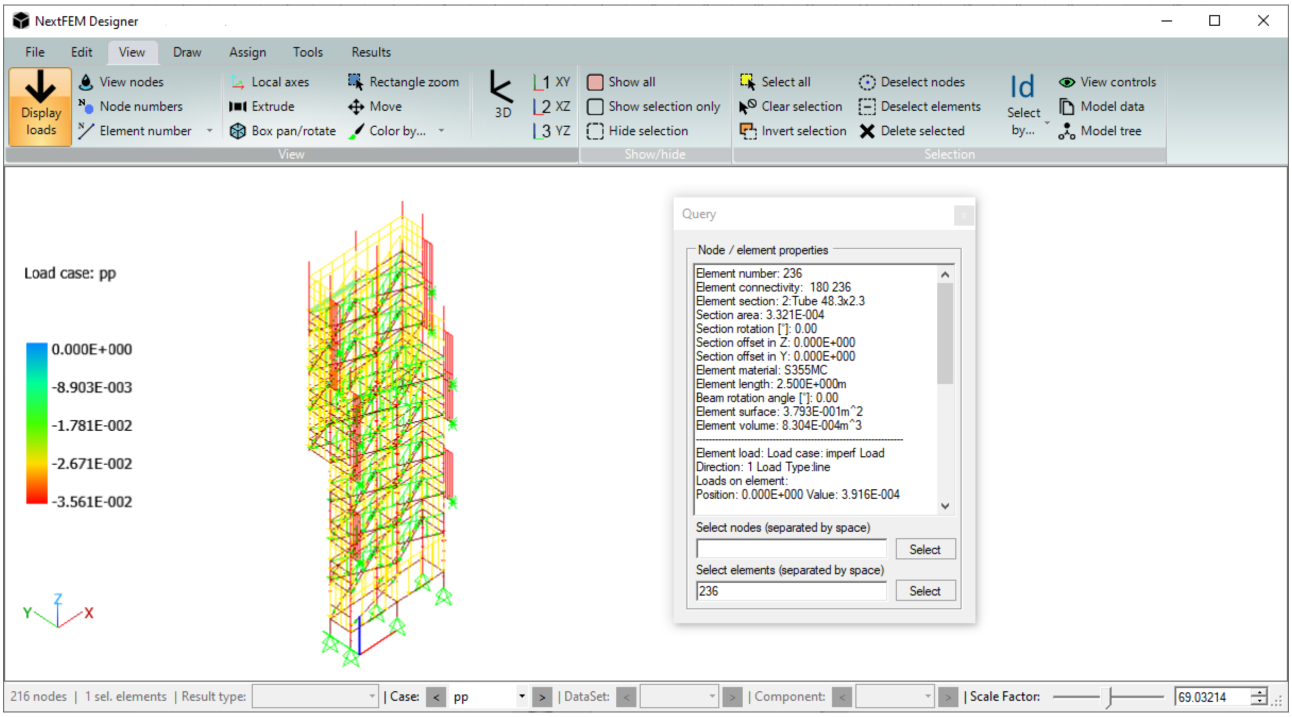
All these load cases are used to write the load combinations as per Eurocode 3.

⚠ WARNING: All the distributed loads are shown in **kN/m** and the point loads are in **kN**.

The loads for each element can be shown through the command Tools/Query, and by selecting the element to query.

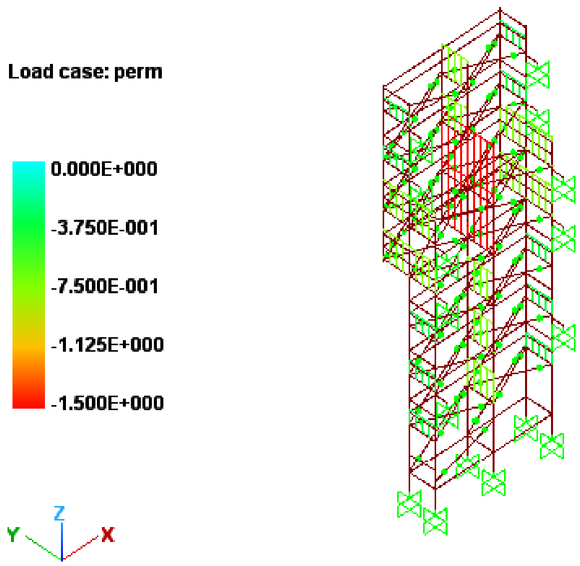
Self-weight loads

All the elements must have their self-weight in the load case **sw**.



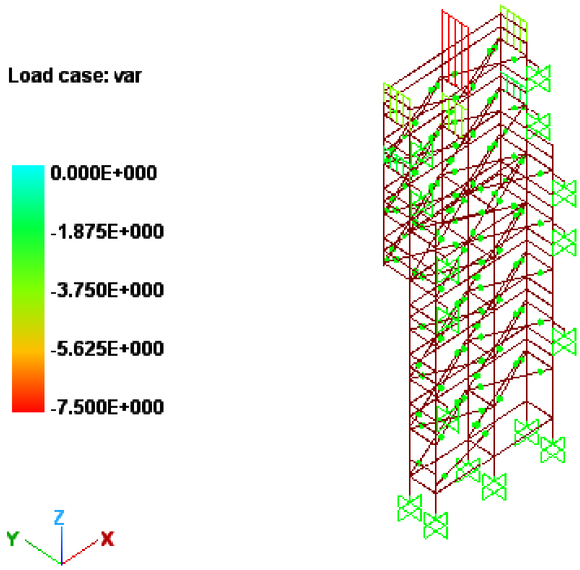
Permanent loadings

All the decks must have a load defined in load case **perm**. All the loads are applied to the transoms supporting the decks.



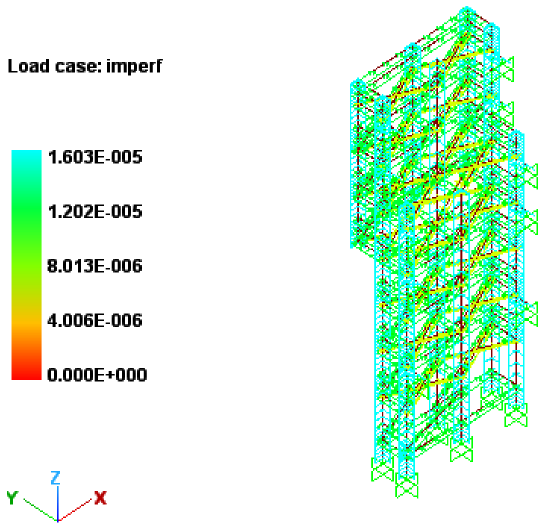
Variable loads

All the decks (or the desired ones) must have variable loads in **var**. All the loads are applied to the transoms supporting the decks.



Imperfection loadings

All the elements must have loads (if required) in the load case **imperf**.



Wind loads

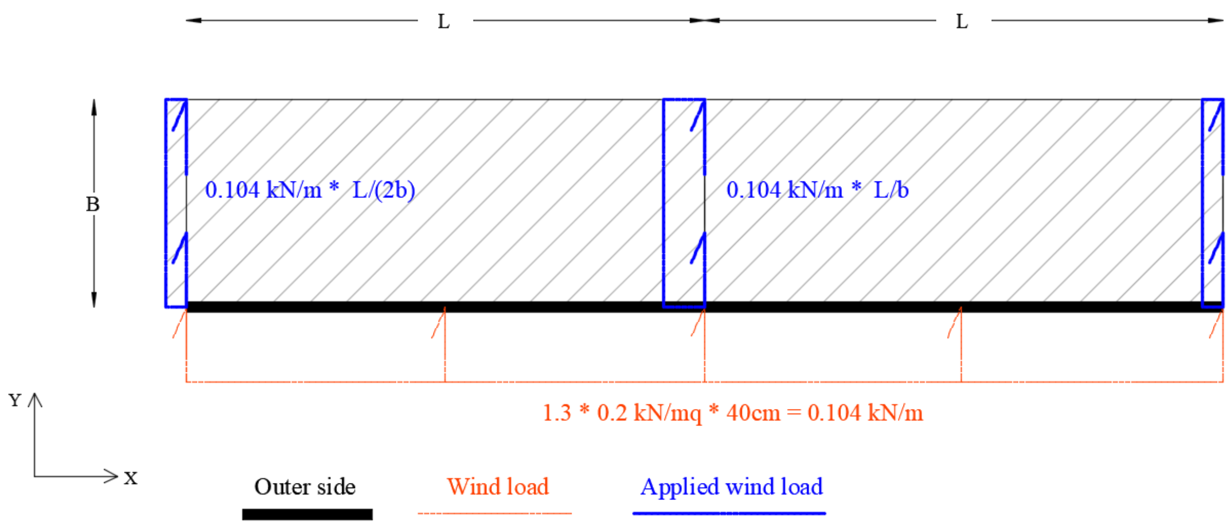
All the elements must have wind loads in the load cases **wind_X** and **wind_Y** assuming wind in global directions.

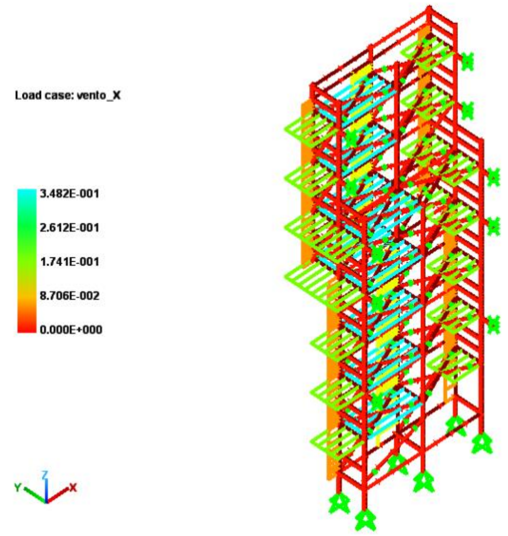
The loading normal to the façade (dir. Y) is applied to the transoms supporting the loading decks, as depicted in the picture below.

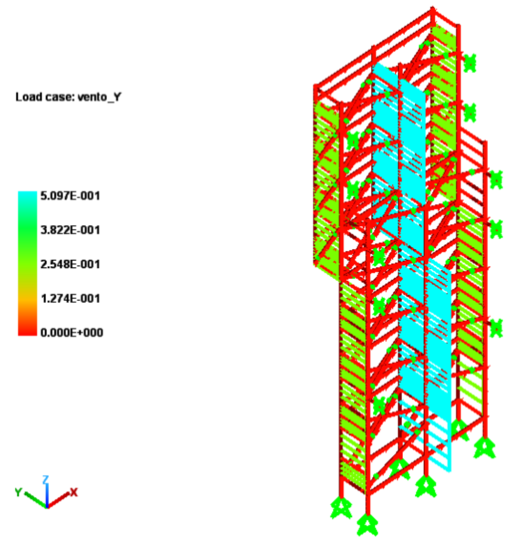
The lateral wind load (dir. X) is applied by default (and conservatively) to each transom, including the internal ones, assuming a toeboard on each side of every bay. To load only the transoms at both ends of the scaffold, select the option "Remove wind load from internal transoms".

⚠ WARNING: wind loading is applied for the equivalent toeboard height as distributed local load of a transoms. In this way, by using X and Y as wind directions, the load is not projected on such global directions. The program selects automatically the load case wind_X or wind_Y allowing a maximum misalignment of 45°.

⚠ WARNING: wind loading is eventually applied for sheeting or netting. Conservatively, the netting coefficients for orthogonal (1.3) and parallel (0.3) wind are assumed.



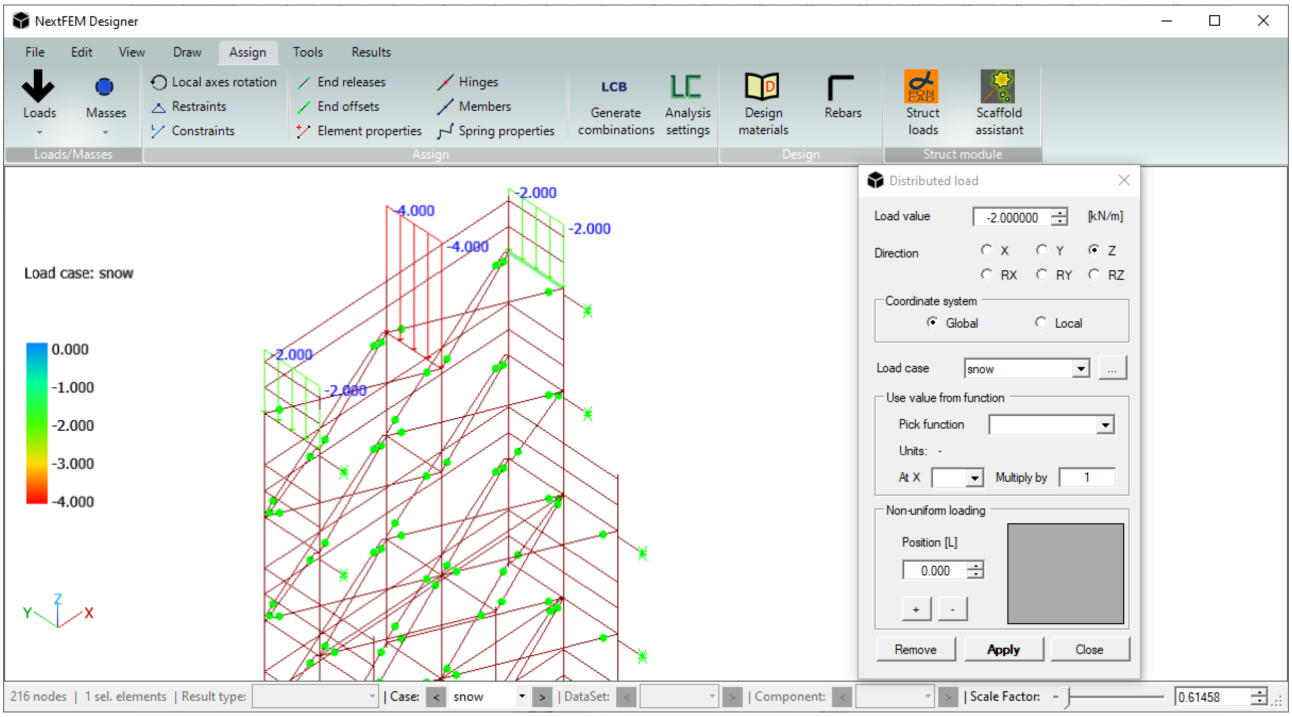




Snow loads

⚠ WARNING: snow loads are automatically applied by the program ONLY for the highest planar floor or in the case of sloped roof.

To apply the snow loads, select the elements to load and use the command *Loads/Distributed load…* .



To apply snow loads, select the Z direction and the load function *snow_2* in the *Distributed load* mask.

In the box *Use value from function*, if the snow load definition has been requested during import, the lettering *snow_2* is available. Choose for *X* (the height of the snow load) the proper value equal to the altitude, in *m*.

In the Multiply by textbox, the user has to specify the length of influence in order to obtain a linear load [F/L]. The uniform load value to apply is automatically updated.

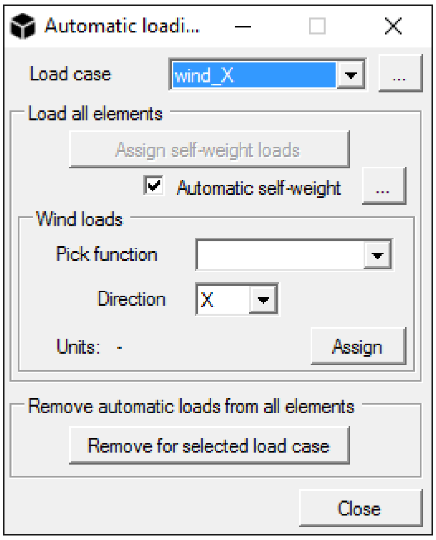
WARNING: Always specify the minus sign in *Distributed load value*.

Finally, press *Apply*.

The selected elements must have a snow load as defined by the used in the load case **snow**.

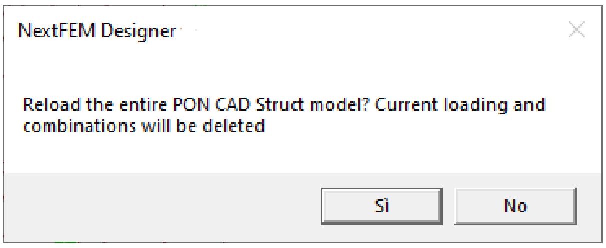
Automatic loads removal

To remove a load set assigned automatically during the import stage, you can use the command *Remove for selected load case* in *Assign/Loads/Automatic loading…* .



Reload the scaffolding

To reload a scaffold, the load masks displayed in the import procedure can be recalled with the command *Assign/Loads/PON CAD Struct loads…* . This is particularly useful to reload a model used for service condition with the out-of-service loads.



⚠ WARNING: This procedure is not guaranteed in case of severe modifications applied in the imported model before this command. For example, if some transoms have been removed and then redrawn, the automatic procedure will not load them. Please check carefully the loading sets you obtain.

Wind load definition

Wind load as per Eurocode 1 is defined on the base of the site of the structure and on the selected properties.

In the window *Requested data*, select or input the proper values through the textboxes and the dropdown menus available.

Requested data are:

- *Wind vel. Vb [m/s]*: the reference wind velocity on the building site.
- *Terrain category*: the type of the terrain according to Eurocode 1-1-4.
- *Orography factor Co*: the Orography factor according to Eurocode 1-1-4.
- *Pressure coeff. Cp*: the drag coefficient used, see note below.

⚠ WARNING: The pressure (drag) coefficient C_p must be left equal to 1.3 (default) to satisfy the code EN 12811-1:2004, 6.2.7 (Wind loads). The local effect coefficient c_s is taken always as 1.0 by the program.

Snow load definition

Snow load as per Eurocode 1 is defined on the base of the site of the structure and on the selected properties.

In the window *Requested data*, select the proper values through the textboxes and the dropdown menus available.

The requested data are:

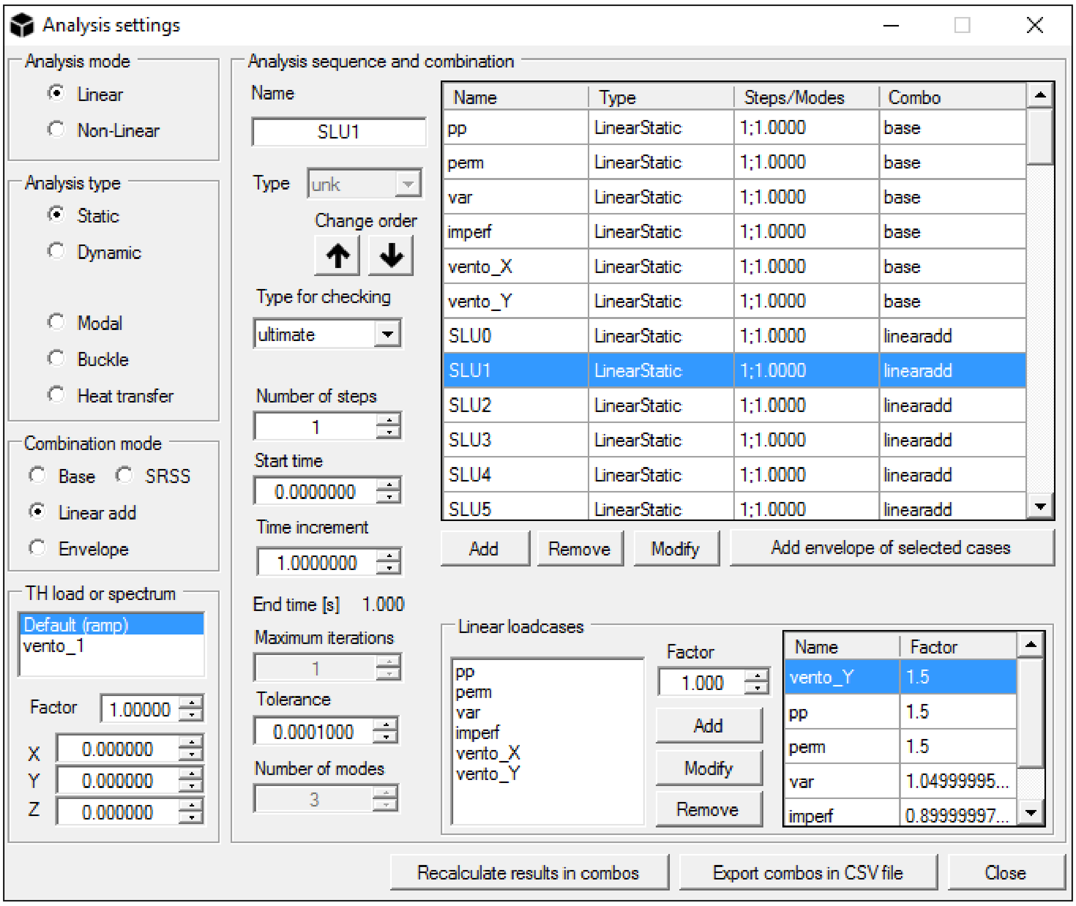
- Snow load sk [kN/m^2]: the reference snow loads on the ground for the selected site;
- Exposure coeff. Ce: the exposure coefficient as per Eurocode 1-1-3.
- Thermal coeff. Ct: the thermal coefficient as per Eurocode 1-1-3.
- Pitch angle of roof [°]: angle, in degrees, for the pitch of the roof (for normal scaffolds is usually 0°).

⚠ WARNING: snow loads are automatically applied by the program ONLY for the highest planar floor or in the case of sloped roof.

⚠ WARNING: snow loads are applied to sloped roofs with the load shape factor taken 0.8, as for planar surfaces. Moreover, the reduction for higher slope angles >30° and <60° is not applied. To modify the load, change the load function instead, by inputting the desired slope in *Pitch angle of roof*.

Load combinations

Load combinations are generated automatically by the program and are visible under *Assign/Analysis settings…* . Please see the User's manual for the explanation of this mask.



⚠ WARNING: The applied loading sets refer to the code EN 12811-1:2004, 6.2.9.2 (Load combinations) for façade scaffolds. This procedure may not be valid for other types of scaffolds.

⚠ WARNING: Dynamic loads due to moving weighs are not actually supported by the program. They can be added manually in the model.

Advanced load customization

Loads defined in a *Struct* model can be customized in order to modify the import stage through the settings in the file *general.pcd*, which is located in the installation directory, in the *data* subfolder.

```
#Lang;LoadCode;WindFunc;SnowFunc;VarRatioOutOfServ;PermLoad[units];
#VarLoad[units];Imperfection;FractSWImperf;FractVarImperf;ComboName;UnitsOut;WindStd[units];WindStdHei[unitsOut];
#NameSW;NameG2;NameVar;NameSnow;NameWind;WindCnormal;WindCortho;SerComboName;AdmStr;AdmLCs;
en;Eurocode 1;12;13;0.5;0.3;3.0;True;0.015;0.015;ULS;m;0.2;0.4;sw;perm;var;snow;wind;1.3;0.3;SLS;0;Condition;
it;NTC2018;2;3;0.5;0.3;3.0;True;0.015;0.015;SLU;m;0.2;0.4;pp;perm;var;neve;vento;1.3;0.3;SLE;0;Condizione;
```

Such file permits to set the following parameters:

- *Lang*: language code related to the parameters set;
- *LoadCode*: name of the reference code;
- *WindFunc*: code for wind loading function (2 for NTC2018, 4 for NTC2008, 12 for Eurocodice 1);
- *SnowFinc*: code for snow loading function (3 for NTC2018, 5 for NTC 2008, 13 for Eurocodice 1);
- *VarRatioOutOfServ*: quota of the variable load to be applied to the deck in the less conservative position for the out-of-service condition;
- *PermLoad*: default permanent loading value in kN/m^2;
- *VarLoad*: default variable loading value in kN/m^2;
- *Imperfection*: True/False to activate/deactivate by default the option for imperfection loading;
- *FractSWImperf*: self-weight quota to be applied as imperfection load;
- *FractVarImperf*: variable loading quota to be applied as imperfection load;
- *ComboName*: prefix for Ultimate combinations;
- *UnitsOut*: target length unit for the model (m);
- *WindStd*: default wind load for service condition (0.2 kN/m^2);
- *WindStdHei*: equivalent toeboard height for wind loads (0.4 m);
- *NameSW*: name of the self-weight loadcase;
- *NameG2*: name of the loadcase for permanent loading;
- *NameVar*: name of the loadcase for variable loading;
- *NameSnow*: name of the loadcase for snow loading;
- *NameWind*: name of the loadcase for wind loading;
- *WindCnormal*: load coefficient for normal wind (1.3 as per EN 12811-1);
- *WindCortho*: load coefficient for parallel wind (0.3 as per EN 12811-1);
- *SerComboName*: prefix for Serviceability combinations;
- *AdmTA*: 0/1 to disable/enable the combinations generation for Allowable Tension design;
- *AdmLCs*: prefix for the combinations for Allowable Tension design;
- *PreMesh*: if 1, it executes a preliminary meshing of the model, if 0 is disabled (default). It can be useful for tube & fittings scaffolds having undivided transoms.

⚠ WARNING: user is strongly advised to NOT modify such values. Changing these values must be in any case made only in accordance with NextFEM, as to avoid program malfunctioning.

Loads for particular conditions

Stairways modelled in PON CAD are transformed in load decks by PON CAD itself before exporting to Struct, hence the associated loading can be incorrect. Please always check the loads applied to stairways, and eventually integrate them.

Concentrated loads can be added directly in PON CAD. Such weights, always defined in kN, are imported in Struct in the “perm” loadcase, holding permanent loads.

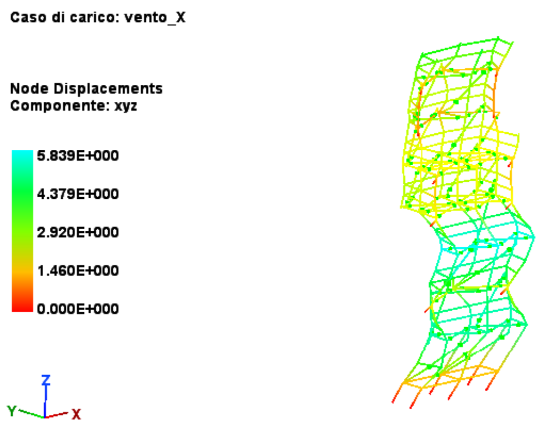
Chapter 3
Analysis and results

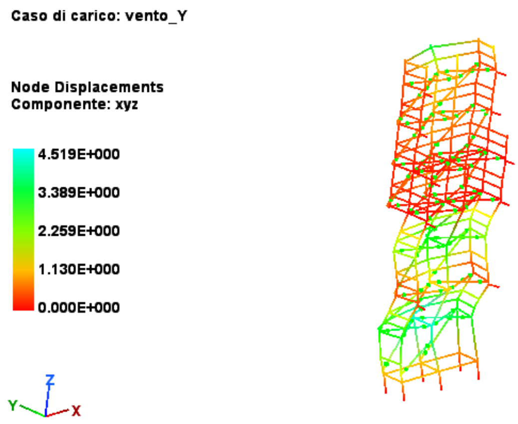
Once loads and restraints are checked, the analysis can be performed on the model through the command *Run*. Each load case will be analysed separately and, at the end, the command *Display results* will be activated automatically.

In the viewport, it is possible to get:

<u>Nodal displacements</u>

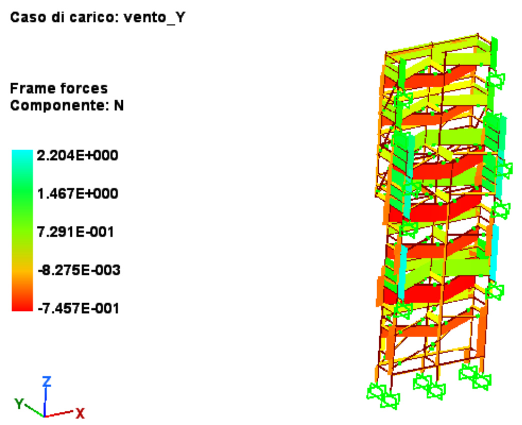
Select *Node displacements* from the dropdown menu *Results* in the bottom bar.

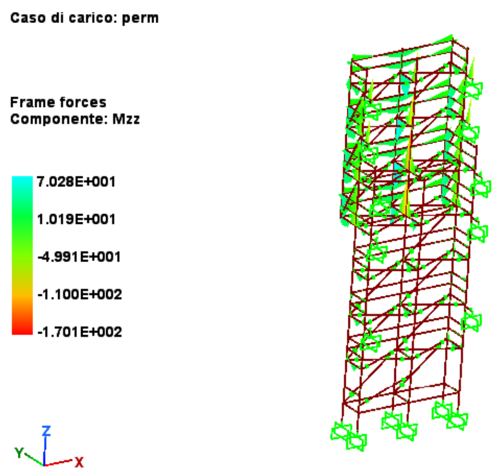




<u>Forces diagrams</u>

Select *Frame forces* from the dropdown menu *Results* in the bottom bar.

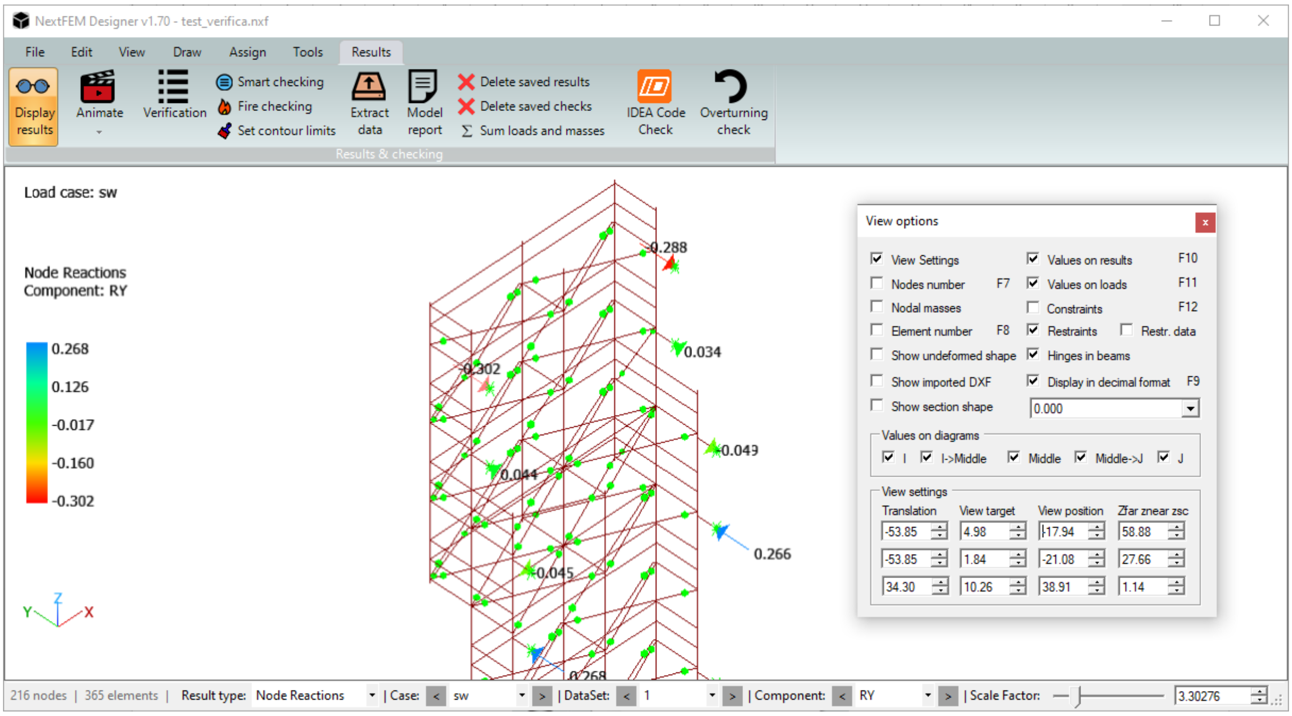




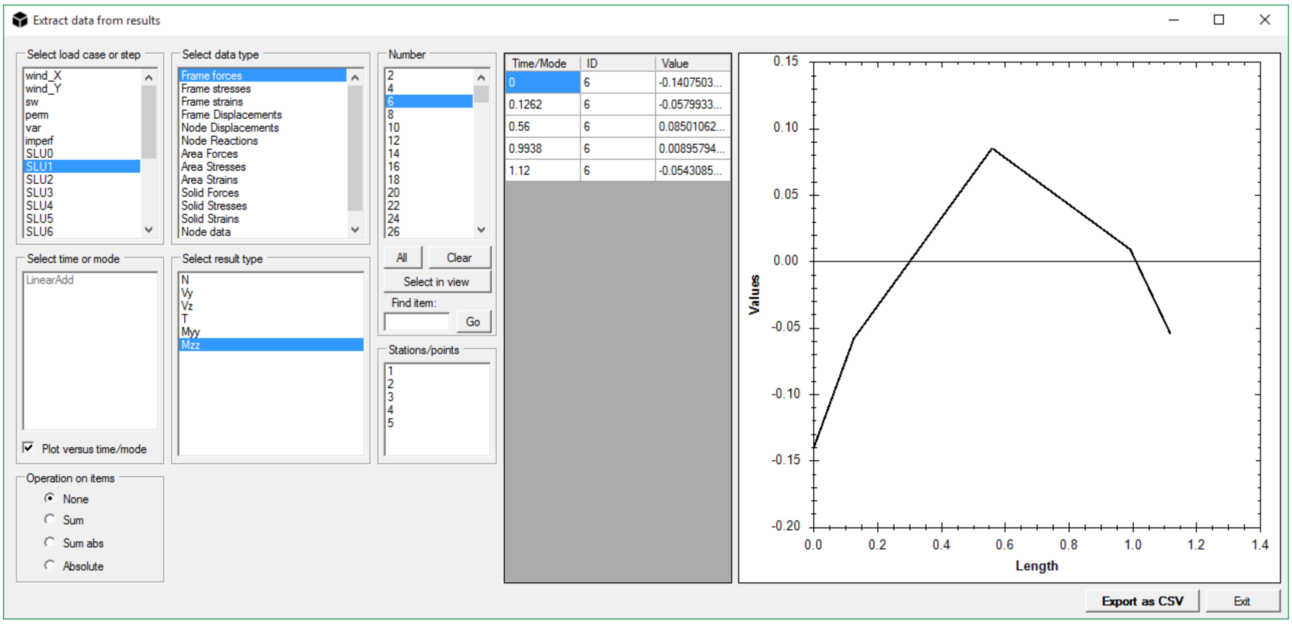
To obtain the values for the diagrams, use the option *Values in diagrams* available in *View controls…* (as in the following figure) by pressing F3.

<u>Reactions</u>

Select *Node reactions* from the dropdown menu *Results* in the bottom bar.



Results in tabular format are available from the command *Extract data…* . Please see the User's manual for a proper explanation and to know how to export data in Excel® format.

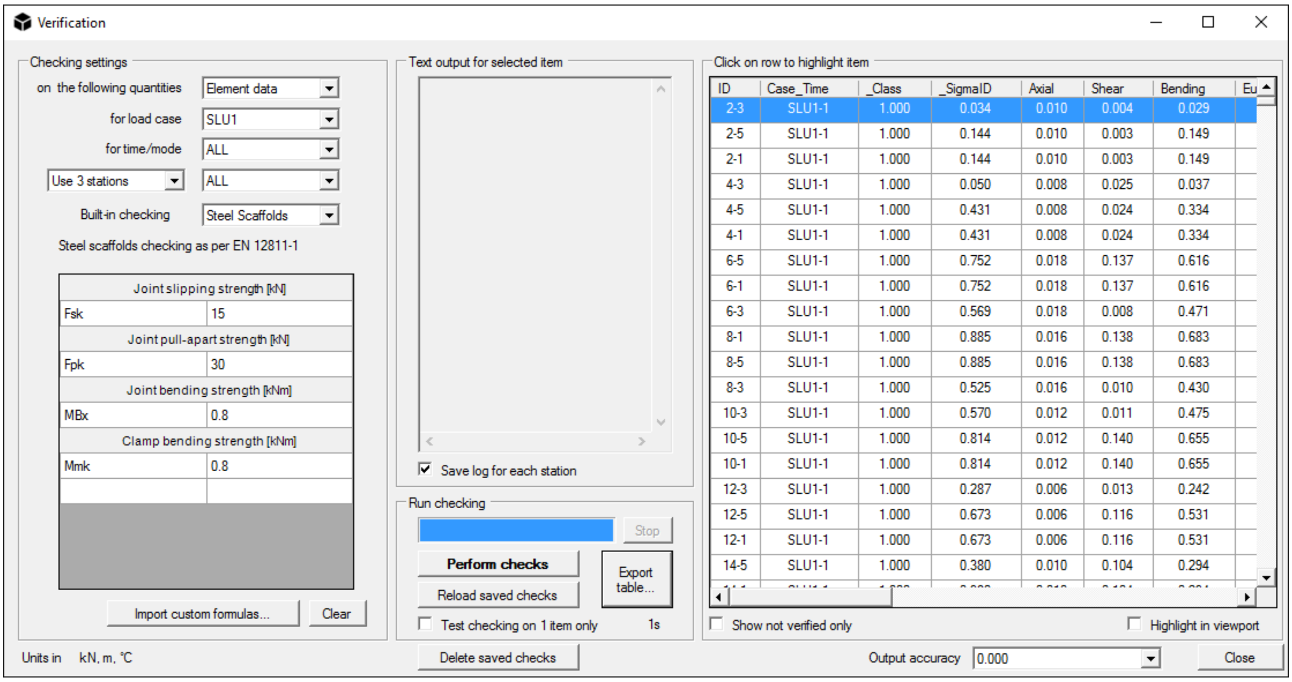


Structural verifications

Once obtained the results, the structural verifications can be performed through the command *Tools/Verifications…*

To perform the Ultimate Limit State (ULS) checking on steel members, follow these steps:

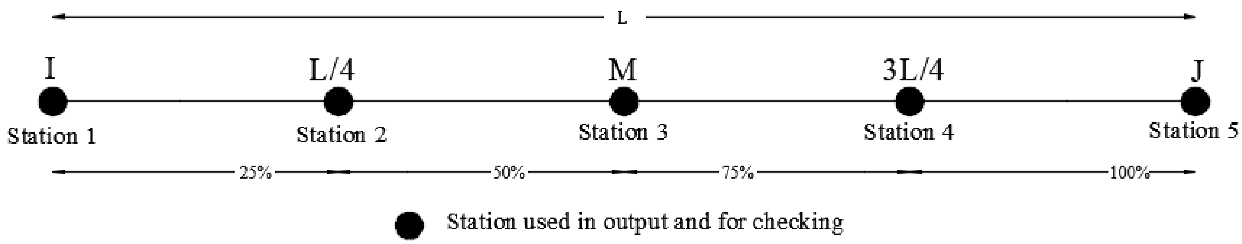
- *OPTIONAL*: select the elements to check and turn on the option On selected items only. If this options is not active, all the elements in the model will be checked;
- Select, in the box *Checks to be performed*, the quantities to be checked:
  - *on the following quantities: Element results* for the beam forces;
  - *for load case*: select *ALL* to analyse all the load cases including combinations, *ALL COMBOS* to check only combinations; Strength Combos to check the Ultimate Limit State combinations; Serviceability Combos to check the Serviceability Limit State combinations, or, alternatively, a single load case or combination;
  - *for time/mode*: *ALL*
  - *Built-in checking*: to perform the checking of a scaffold, select *<u>SteelScaffolds</u>*;
  - *OPTIONAL*: to overwrite any on the parameters used in verifications, i.e. the partial safety factors, add the desired variables on the table on the left.
  - in the table on left side, some parameters are required to perform the joint checking, please specify the proper ones changing the default.
- To start the checks, press *Perform checks*. Results, all in terms of force/strength, are reported in the columns of the table shown on the right. By selecting one row, the corresponding element will be selected in the viewport.



For each element, 5 sections are used for the checking. They are spread along the beam as follows:

[1] end I;
[2] 25% of the beam length from end I;
[3] middle point;
[4] 75% of the beam length from end I;
[5] end J.

To use only 3 sections (I, J and Middle of the span), please select the option *Use 3 stations* (I, M e J).

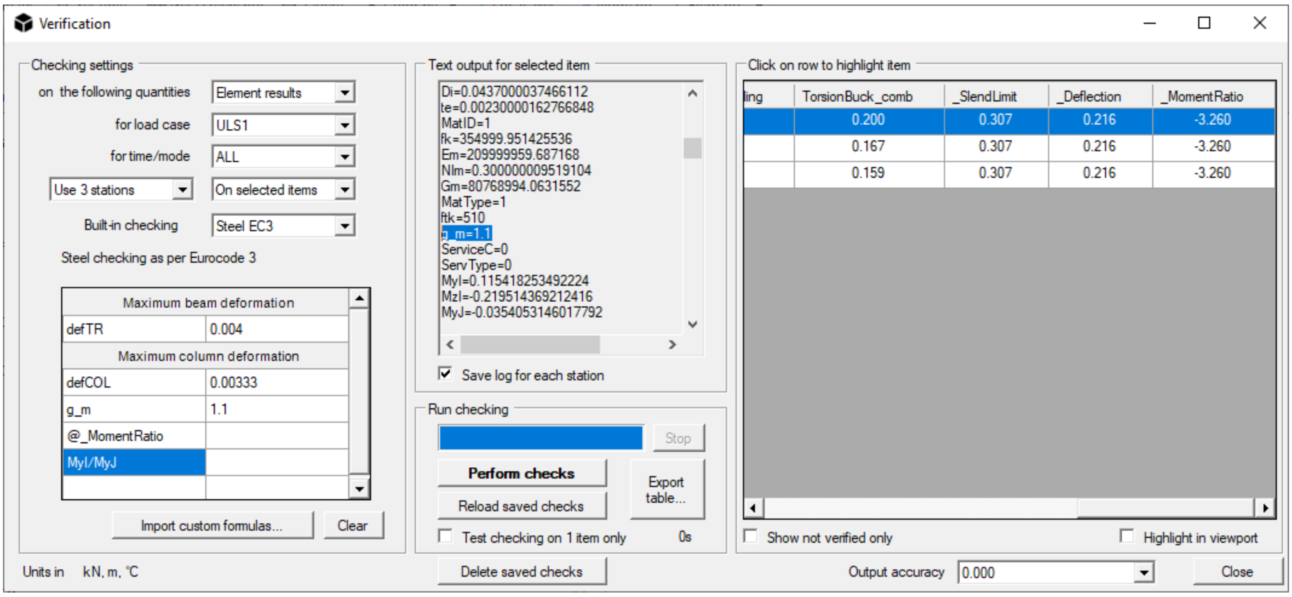


Custom checking

All the checks performed can be customized at will with a the desired relationships appended to the existing table displayed on the left. Please refer to the users' manual for further information on the proper syntax to be used.

For example, to override the default Gamma_M0 safety coefficient, given by a material property, add the following row:

g_m	1.1



To obtain also the visual representation of the custom check, the value must the numeric. For example:

@_MomentRatio	
MyI/MyJ	

will show the column named "_MomentRatio" and the relative visualization in *Element Data*. If the column name begins with the underscore character "_", then such quantity will not be taken into account in evaluating if the section does or does not satisfy the whole check.

Analysis report

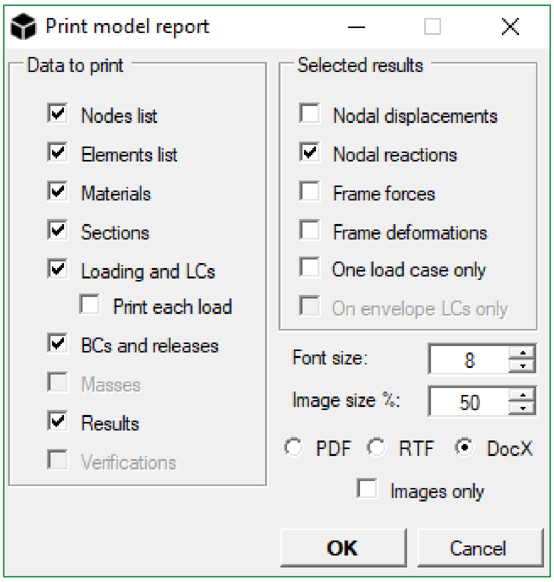
The report of the performed analysis and verifications can be obtained with the command *Tools/Print report* … .

Select the proper items to write in the report and press OK.

The report can be saved in PDF or RTF format where specified in the dialog that will appear. RTF or DocX format is advised, since it allows further modifications and contains several screenshots, automatically generated by the program.

The first lines of the report contain the reaction results for each load case (*Total load for case…*).

Export can take several minutes.



- ⚠ WARNING: user is strongly advised to limit the results report to the sole envelope combinations, as to avoid excessive length of the report itself.
- ⚠ WARNING: The results included that are reported in text format follow the pattern written at the beginning of each paragraph (i.e. *Reactions: X Y Z RX RY RZ* means that the reactions are reported, one node at a line, in the format *node, reaction along X, reaction along Y,* etc. …)