



NextFEM Designer  
Struct module manual

Version 2.5 and above

© NextFEM 2015-2026

# Summary

Chapter 1 Introduction .....	4
Installation and first run .....	4
Program activation .....	4
Manuals and support .....	4
Design codes implemented in the Struct module .....	4
User interface .....	4
License .....	5
Chapter 2 Importing models and launching analysis .....	6
Importing the model .....	6
Import from PON CAD .....	6
Set loading decks .....	8
Scaffold assistant .....	9
Check applied loads .....	10
Self-weight loads .....	11
Permanent loadings .....	11
Variable loads .....	12
Imperfection loadings .....	12
Wind loads .....	13
Snow loads .....	14
Automatic loads removal .....	15
Reload the scaffolding .....	16
Wind load definition .....	16
Snow load definition .....	17
Load combinations .....	17
Advanced load customization .....	18
Loads for particular conditions .....	18
Chapter 3 Analysis and results .....	19
Structural verifications .....	21
Checking of fittings .....	22
Joint as an element .....	24
Coupler as end-joint of a beam .....	28
Anchors checking .....	31
Base plate sliding check .....	32
Check for uplifting .....	33
Capsizing checks .....	33
Custom checking .....	36
Generic analysis report .....	36
Scaffolding analysis report .....	37



# 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.

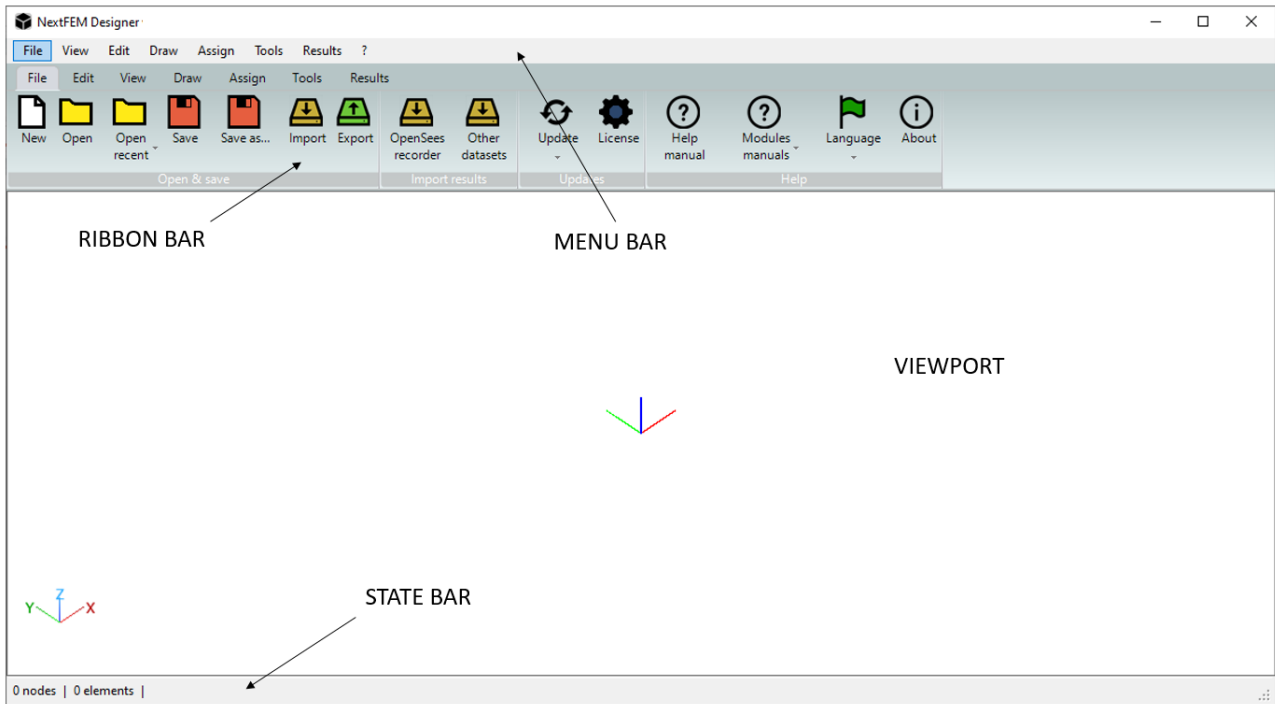
### Design codes implemented in the Struct module

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

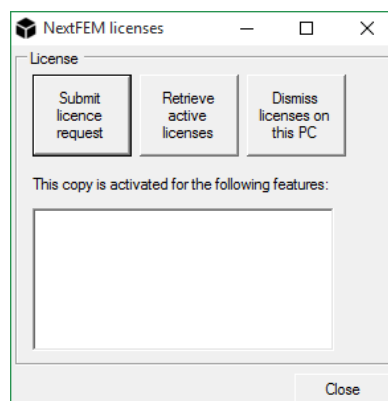
To activate the Struct module select the command *License...*, which handles user's license. Please be connected to the Internet before using it.

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

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](mailto: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*).

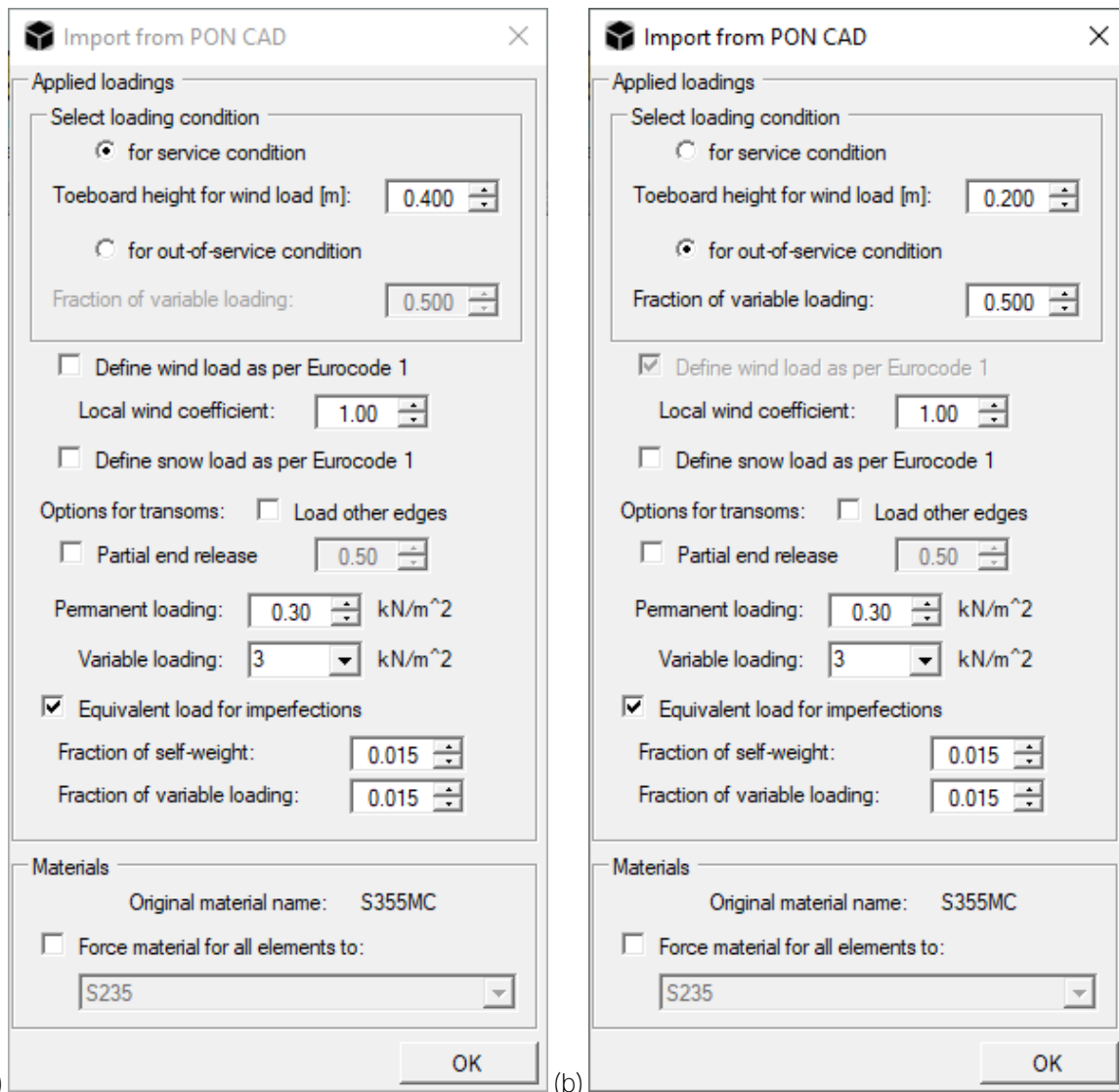
**To import the model, select the command File/Import... and select the .SEZ file to import, or drag the .SEZ into the viewport.**

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  $0.2 \text{ kN/m}^2$  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).
- *Local 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 2<sup>nd</sup> and 4<sup>th</sup> edges of 3DFACE representing the floors. Activating this option will force the beams under the 1<sup>st</sup> and 3<sup>rd</sup> 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<sup>2</sup>, 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.

The dialog box 'Levels to load' has the following options and data:

- Load all levels with 100%
- Load selected levels
- Loading at 100% at level [m]: 8.80
- Loading at 50% at level [m]: 6.80
- Loading at 100% at level [m]: 8.80
- Loading at 50% at level [m]: 8.80
- Custom loading

Height [m]	Load factor
2.80	1
4.80	1
6.80	1
8.80	1

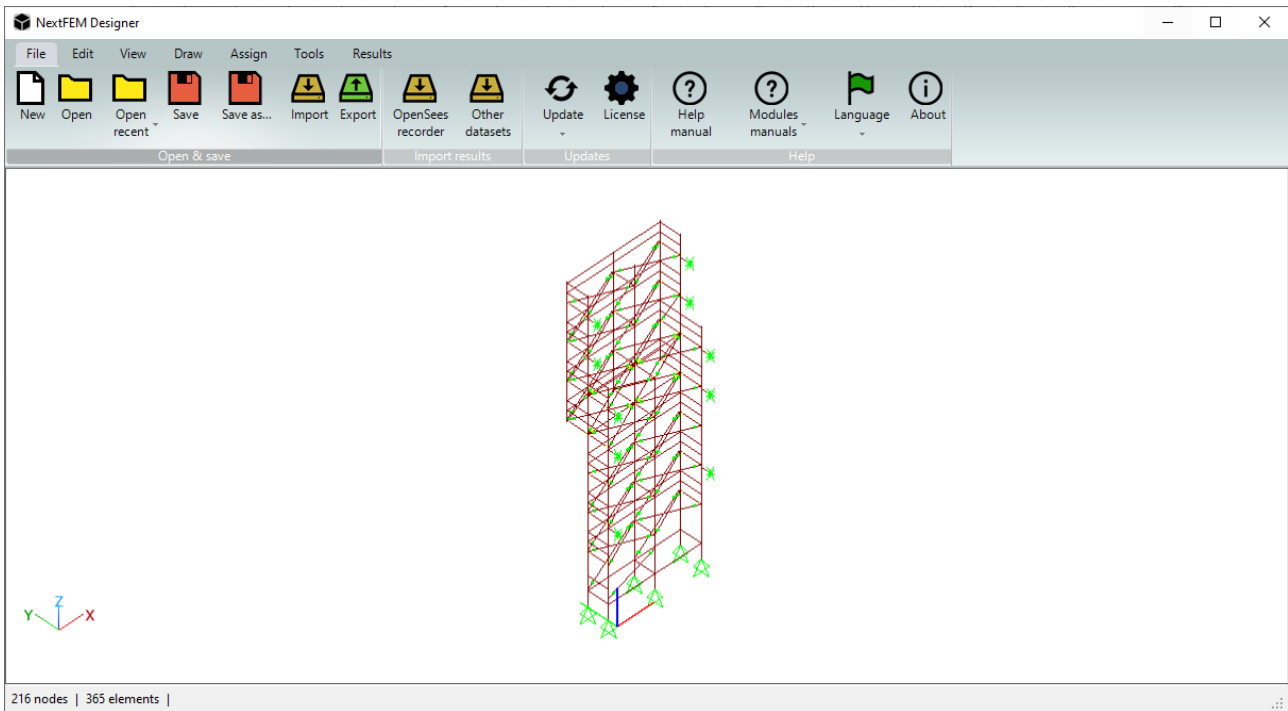
Remove wind load from internal transoms

OK

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.

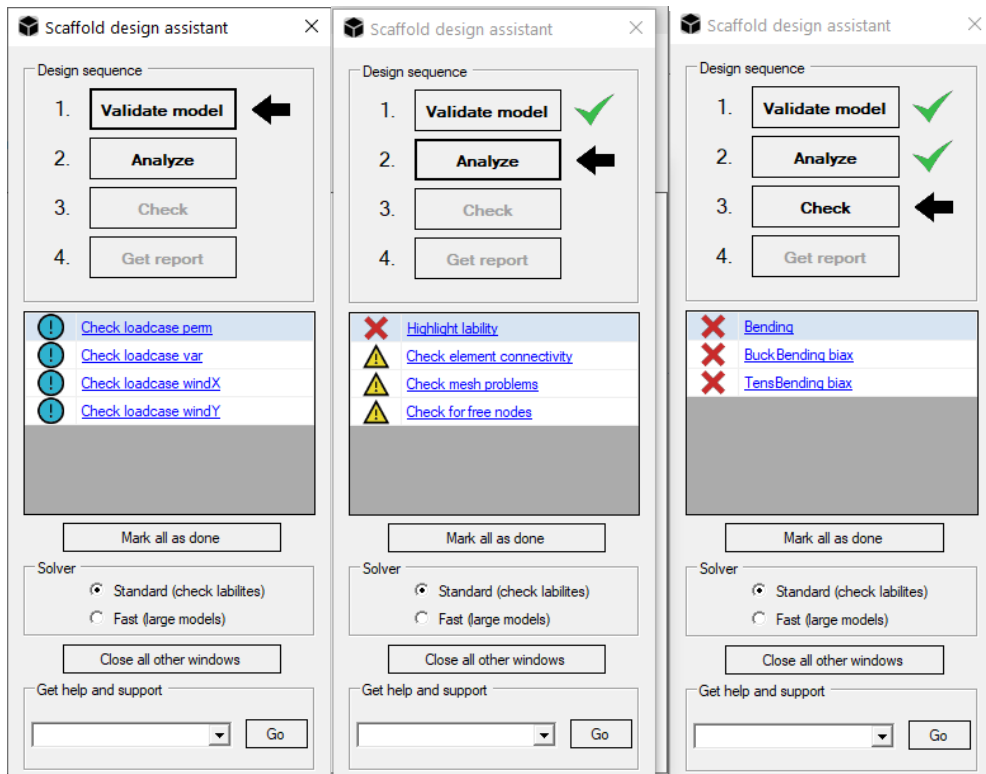
The resulting model is bounded with restraints in X and Y along the height, and supported at the base.



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

**⚠ NOTE ON HORIZONTAL BRACES FOR LOADING PLATFORMS:** The horizontal braces under the boards represent the stiffening effect in the horizontal plane of the boards (loading platform) and they are included in the model from PON CAD. Traditionally, in scaffolding calculations, they are modelled as connecting rods with the smallest section available for the scaffolding set (e.g., tubular 29.6x2.3mm). They may not necessarily be present in the actual scaffolding, so before considering the checks carried out on these elements to be significant or not, the user must verify that they are actually present.

*Scaffold assistant*



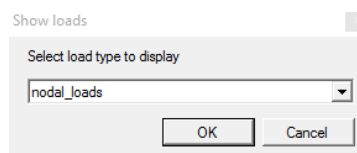
Since v.1.7, Scaffold design assistant is available to guide the scaffold design process.

- 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)

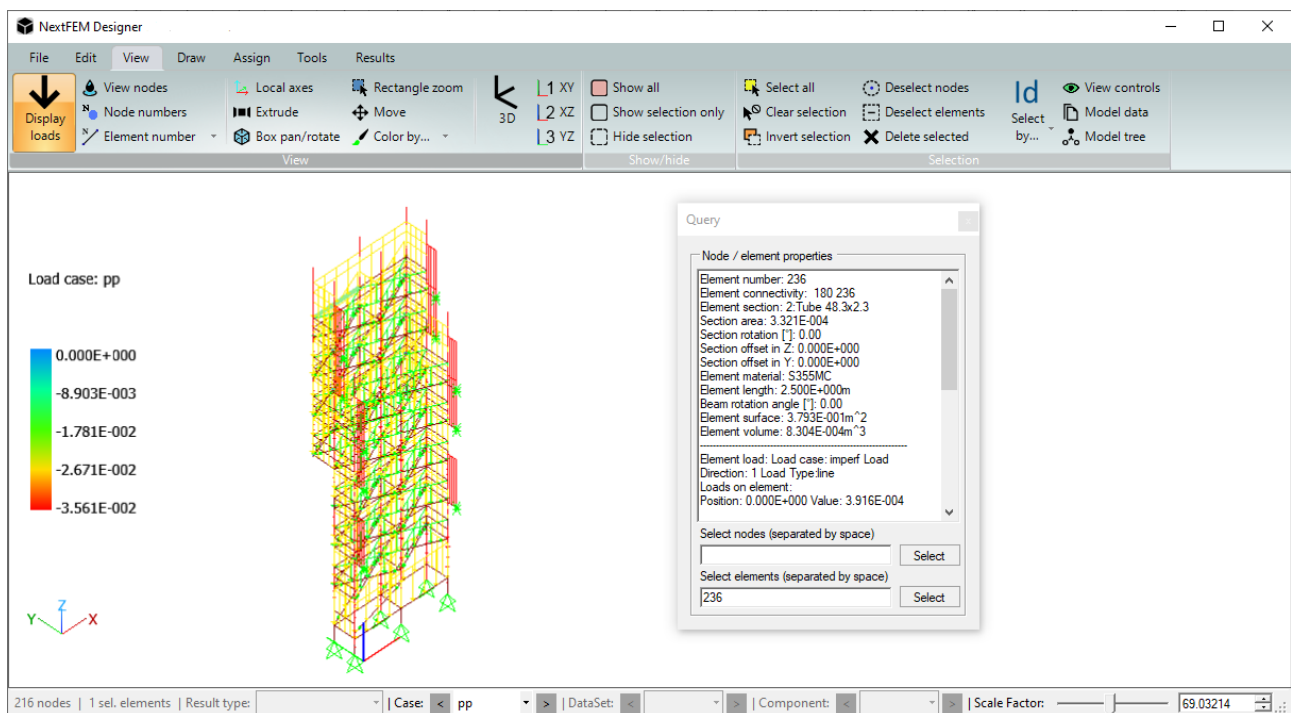
All these load cases are used to write the load combinations as per Eurocode 3.

 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.

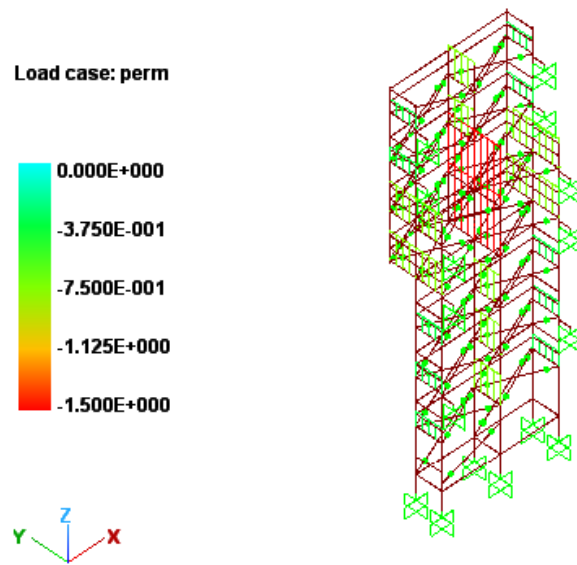
### Self-weight loads

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



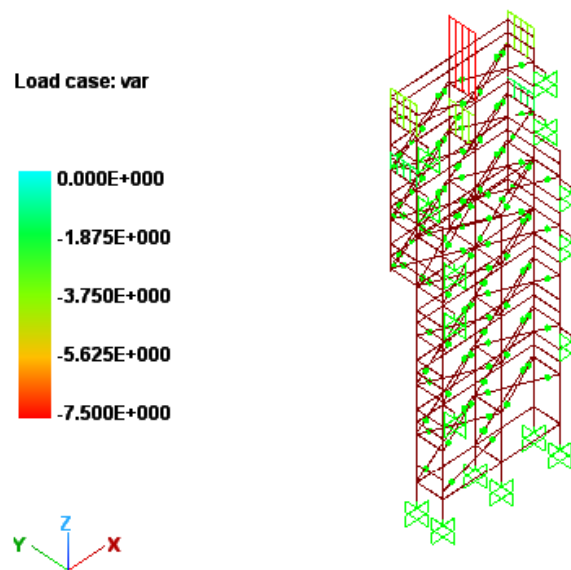
### Permanent loadings

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

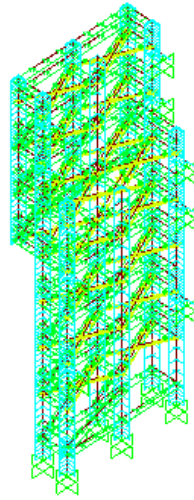
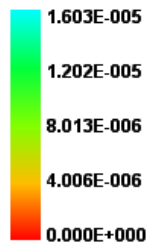
All the decks (or the desired ones) must have variable loads in var. All the loads are applied to the transoms supporting the decks.



### Imperfection loadings

All the elements must have loads (if required) in the load case imperf. The load case is automatically filled in with the load fraction (default 0.015) of the dead and variable loads; in this sense, the load case itself represents a combination.

Load case: imperf



### Wind loads

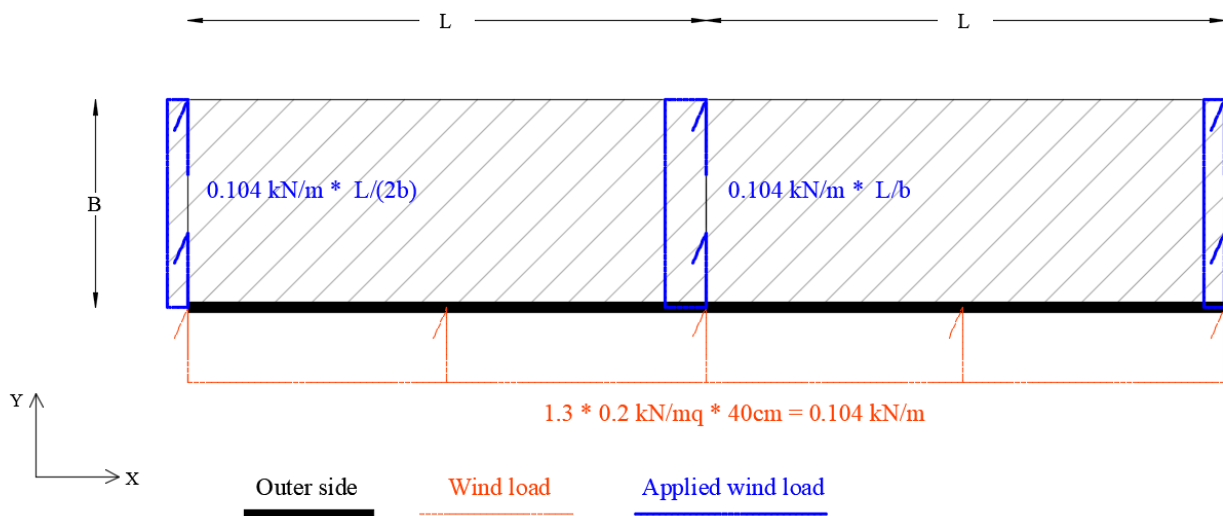
All the elements must have wind loads in the load cases wind\_X and wind\_Y assuming wind in global directions.

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

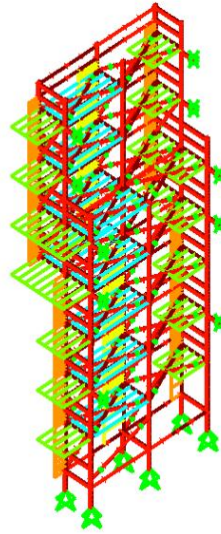
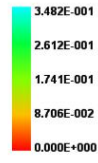
The lateral wind load (dir. X) is applied by default (and conservatively) to each transom, including the internal ones, assuming a toeboard on each side of every bay. To load only the transoms at both ends of the scaffold, select the option "Remove wind load from internal transoms".

**⚠ 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°.

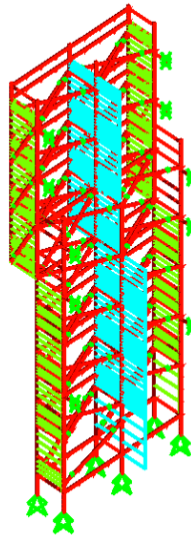
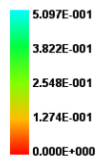
**⚠ WARNING:** wind loading is eventually applied for sheeting or netting. Conservatively, the netting coefficients for orthogonal (1.3) and parallel (0.3) wind are assumed.




Load case: vento\_X



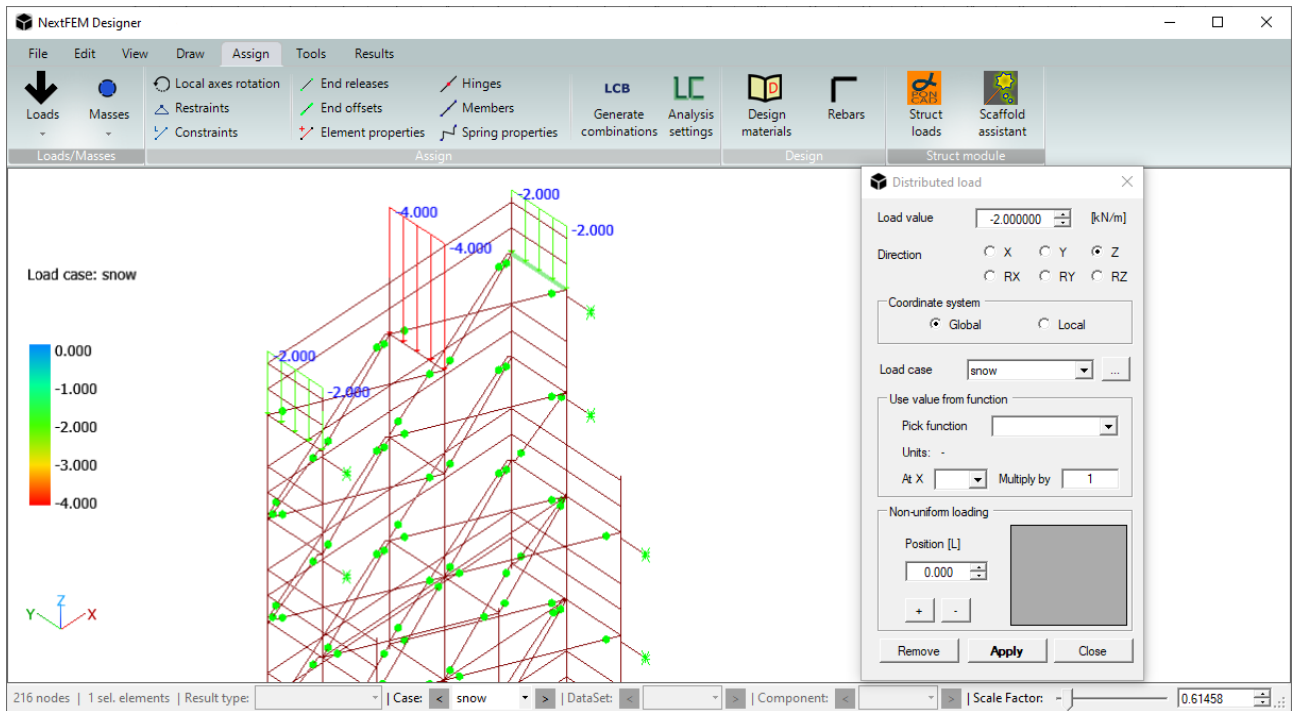
Load case: vento\_Y



### 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.

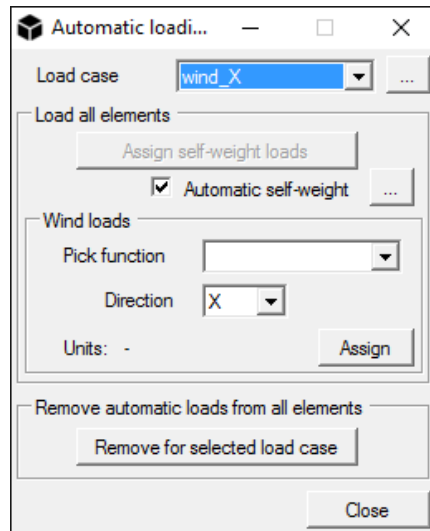
**⚠ WARNING:** Always specify the minus sign in *Distributed load value*.

Finally, press *Apply*.

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

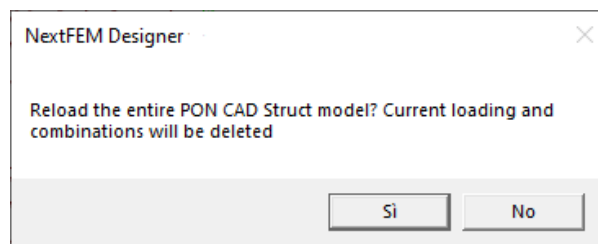
### *Automatic loads removal*

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.  $V_b$  [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  $C_o$* : the Orography factor according to Eurocode 1-1-4.
- *Pressure coeff.  $C_p$* : 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 for other structures.

## 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  $s_k$  [kN/m<sup>2</sup>]: the reference snow loads on the ground for the selected site;
  - Exposure coeff.  $C_e$ : the exposure coefficient as per Eurocode 1-1-3.
  - Thermal coeff.  $C_t$ : 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.

**Analysis settings**

**Analysis mode**

- Linear
- Non-Linear

**Analysis type**

- Static
- Dynamic

**Combination mode**

- Base
- SRSS
- Linear add
- Envelope

**TH load or spectrum**

Default (ramp)  
vento\_1

Factor: 1.00000

X: 0.000000  
Y: 0.000000  
Z: 0.000000

**Analysis sequence and combination**

Name: SLU1

Type: unk

Change order: ↑ ↓

Type for checking: ultimate

Number of steps: 1

Start time: 0.000000

Time increment: 1.000000

End time [s]: 1.000

Maximum iterations: 1

Tolerance: 0.0001000

Number of modes: 3

Name	Type	Steps/Modes	Combo
pp	LinearStatic	1:1.0000	base
pem	LinearStatic	1:1.0000	base
var	LinearStatic	1:1.0000	base
imperf	LinearStatic	1:1.0000	base
vento_X	LinearStatic	1:1.0000	base
vento_Y	LinearStatic	1:1.0000	base
SLU0	LinearStatic	1:1.0000	linearadd
SLU1	LinearStatic	1:1.0000	linearadd
SLU2	LinearStatic	1:1.0000	linearadd
SLU3	LinearStatic	1:1.0000	linearadd
SLU4	LinearStatic	1:1.0000	linearadd
SLU5	LinearStatic	1:1.0000	linearadd

Add Remove Modify Add envelope of selected cases

**Linear loadcases**

Name	Factor
vento_Y	1.5
pp	1.5
pem	1.5
var	1.04999995...
imperf	0.89999997...

Factor: 1.000

Add Modify Remove

Recalculate results in combos Export combos in CSV file Close

- ⚠ 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);
  - *SnowFunc*: 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<sup>2</sup>;
  - *VarLoad*: default variable loading value in kN/m<sup>2</sup>;
  - *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<sup>2</sup>);
  - *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;
  - *WindFactCs*: default value of 1.0, specifies the value of the local coefficient for wind.
- ⚠ 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.

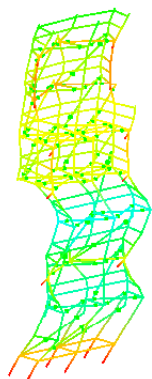
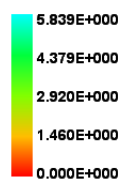
In the viewport, it is possible to get:

### Nodal displacements

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

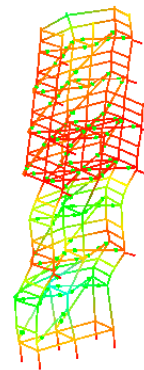
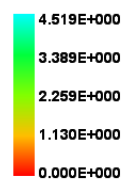
Caso di carico: vento\_X

Node Displacements  
Componente: xyz



Caso di carico: vento\_Y

Node Displacements  
Componente: xyz

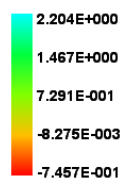


### Forces diagrams

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

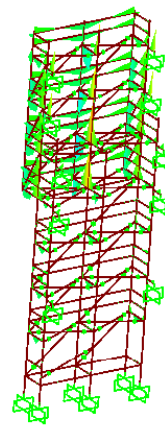
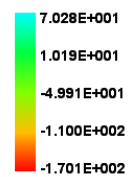
Caso di carico: vento\_Y

Frame forces  
Componente: N



Caso di carico: perm

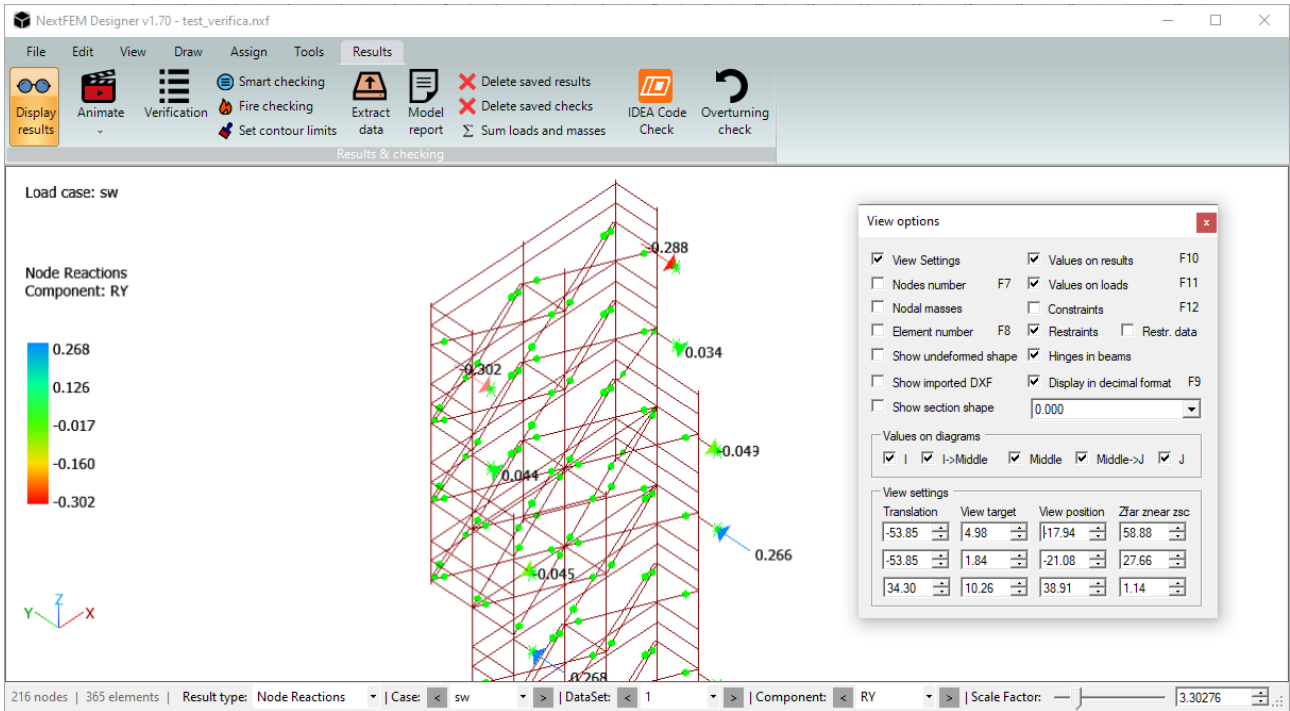
Frame forces  
Componente: Mzz



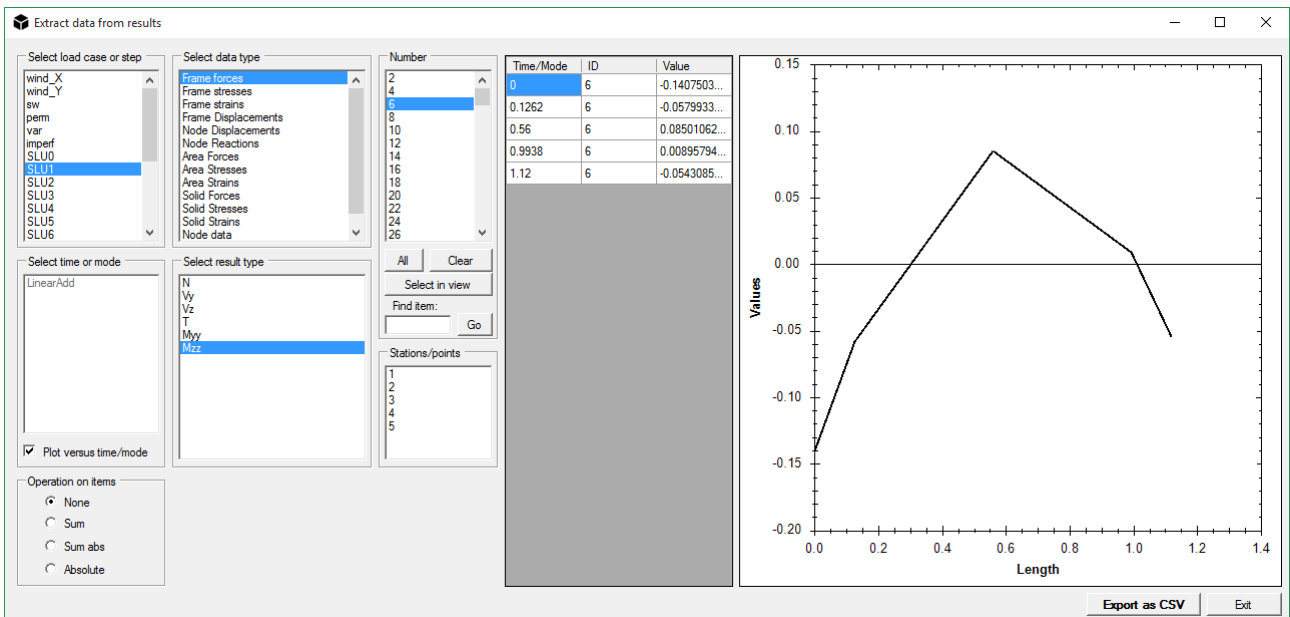
To obtain the values for the diagrams, use the option *Values in diagrams* available in *View controls...* (as in the following figure) by pressing F3.

## Reactions

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...*

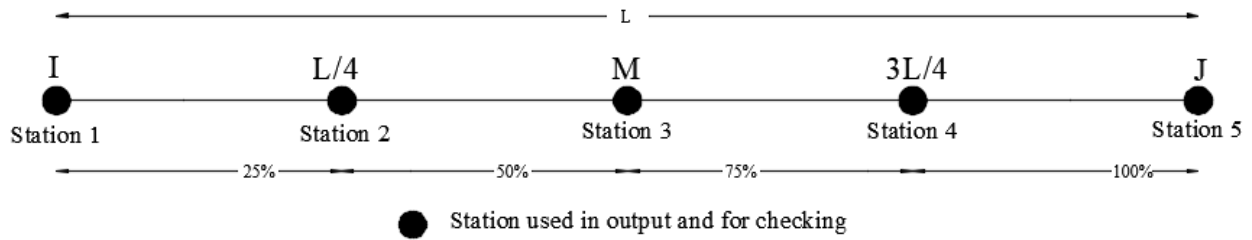
To perform the Ultimate Limit State (ULS) checking on steel members, follow these steps:

- *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:
  - o *on the following quantities*: *Element results* for the beam forces;
  - o *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;
  - o *for time/mode*: *ALL*
  - o *Built-in checking*: to perform the checking of a scaffold, select *Steel/Scaffolds*;
  - o *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.
  - o 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.

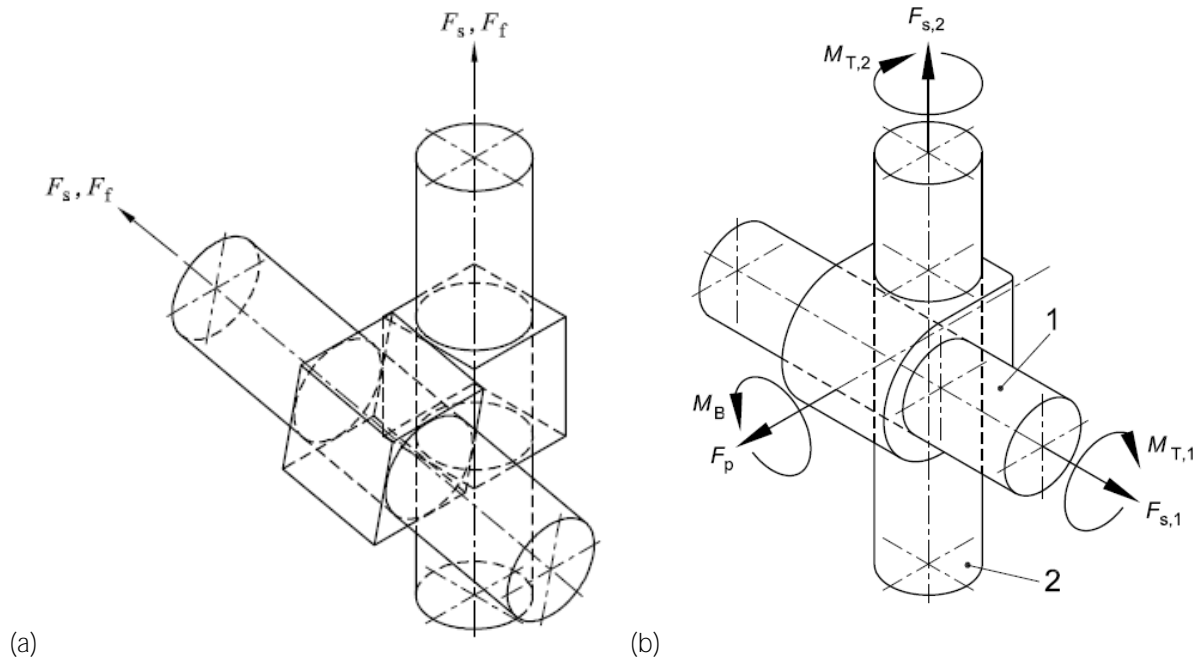
To use only 3 sections (I, J and Middle of the span), please select the option *Use 3 stations* (I, M e J).



*Checking of fittings*

The verification of fittings in a scaffolding is a very onerous operation due to the number of connections. The stresses involved in the verification must also refer to the local reference system of the retained rod. Without claiming to be exhaustive, the main types of joints in a scaffolding structure are:

- a. Swivel joint, which connects two rods with any angle and with a hinge-type constraint;
- b. Right-angle joint, which rigidly connects two orthogonal rods ;
- c. Parallel joint, which connects two rods keeping them aligned;
- d. Sleeve coupler, which connects coaxially aligned rods.



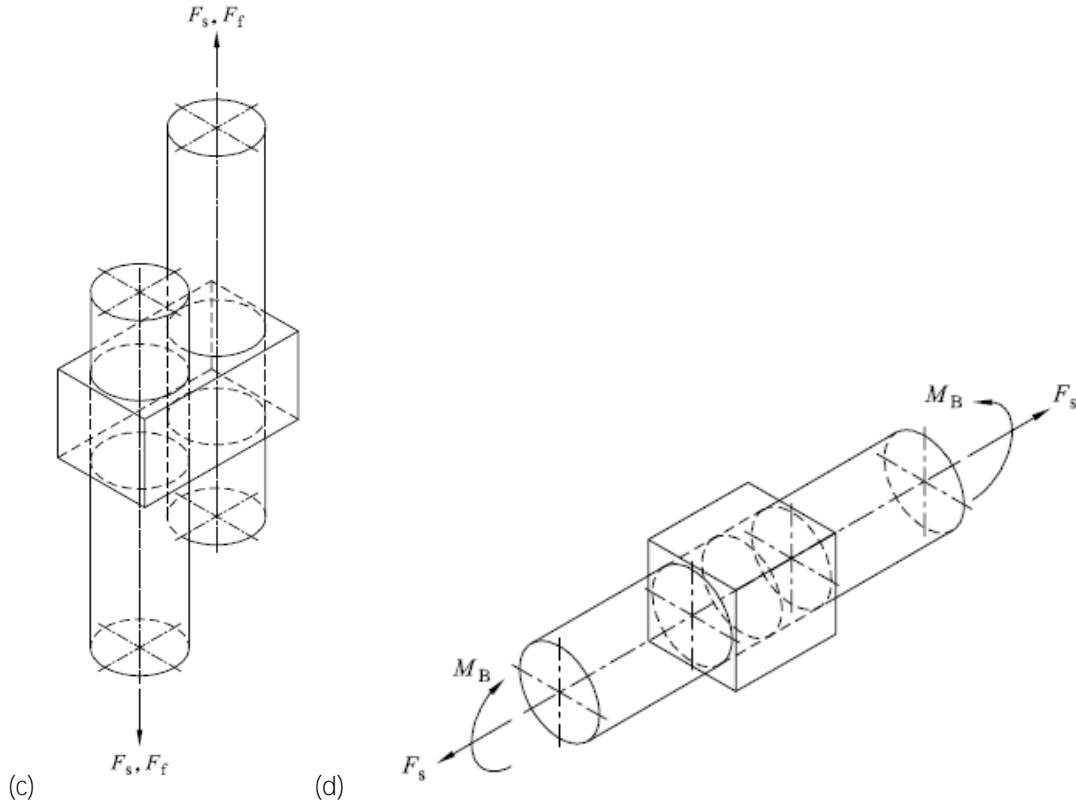


Diagram and nomenclature of the forces acting on the swivel (a), right-angle (b), parallel (c) and sleeve (d) coupler – from EN 74 -1 and EN 12811-1

The UNI EN 74-1 standard describes the requirements and test procedures for the joints, also listing their classes. We will consider class B (highest resistances).

The verification is described in §10.3.3.5 of EN 12811-1 "Temporary works equipment - Part 1: Scaffolds - Performance requirements and general design", with particular reference to right-angle and sleeve couplers. The report reported for orthogonal joints can be easily extended to all other types. According to §10.3.3.5, formula (10), the relationship between demand and capacity of the joint is:

$$\rho_{Sj} = \frac{F_{s1} + F_{s2}}{2F_{sd}} + \frac{F_p}{F_{pd}} + \frac{M_B}{2M_{Bd}}$$

With  $F_{sd} = \frac{F_{sk}}{\gamma_M}$  joint resistance of the joint

$F_{pd} = \frac{F_{pk}}{\gamma_M}$  resistance to joint separation

$M_{Bd} = \frac{M_{Bk}}{\gamma_M}$  resistance to twisting / bending of the joint , with  $\gamma_M = 1.1$ .

This formula represents a conventional domain, which explicitly refers to 2-way flow, the separation of the joint and the resistance to torsion / cruciform moment of the joint. In particular, for the sliding reference is made to the average of the stresses  $F_{s1}$  and  $F_{s2}$  specific to each connected rod. This relationship applies only to orthogonal joints, since we assume the average of  $F_{s1}$  and  $F_{s2}$ . From a more general point of view, the same domain can be written as follows:

$$\rho_{Sj} = \frac{F_{s,res}}{F_{sd}} + \frac{F_p}{F_{pd}} + \frac{M_{B,res}}{M_{Bd}}$$

where  $F_{s,res}$  is the resultant of the sliding forces and  $M_{B,res}$  the resulting conventional moment. This relationship, which is of a general nature (think for example of the one proposed by the same EN standard regarding sleeve joints) conservatively includes all 3 mechanisms of collapse of the joint, i.e. sliding, separation and bending failure, and will be applied to checks of all joints.

Design resistances must be found by the manufacturer based on the tests performed. As an indication, the EN standard contains the following values:

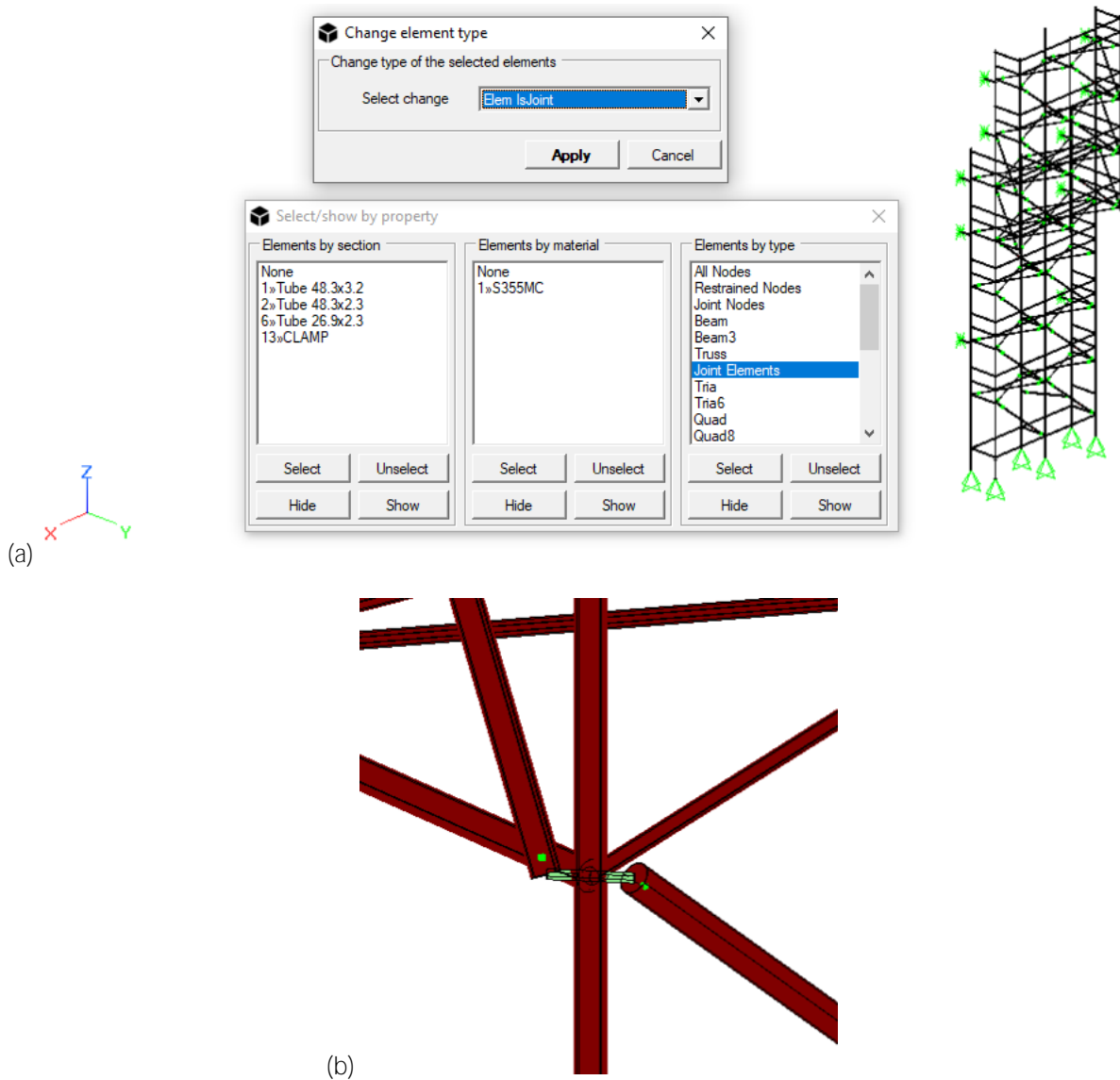
Table C.1 EN 12811-1

Type of joint		Resistance [ kN or kNm ]	
		Class A	ClassB
Orthogonal	fSK	10	15
	Mbk		0.8
	Fpk	20	30
Sleeve type	fSK	6	9
	Mbk		1.4
Swivel	fSK	10	15
Parallel	fSK	10	15

The modelling of the joint is another matter of fundamental importance: the single joint can be modelled as a "short" element (for example as long as the offset between the axes of the connected pipes), or it can be considered as "implicit" in the auction, i.e. incorporated at its ends. The NextFEM Designer Struct module supports both modes, and automatically imports all modelled joints from [PON CAD](#), both as an element and implicit.

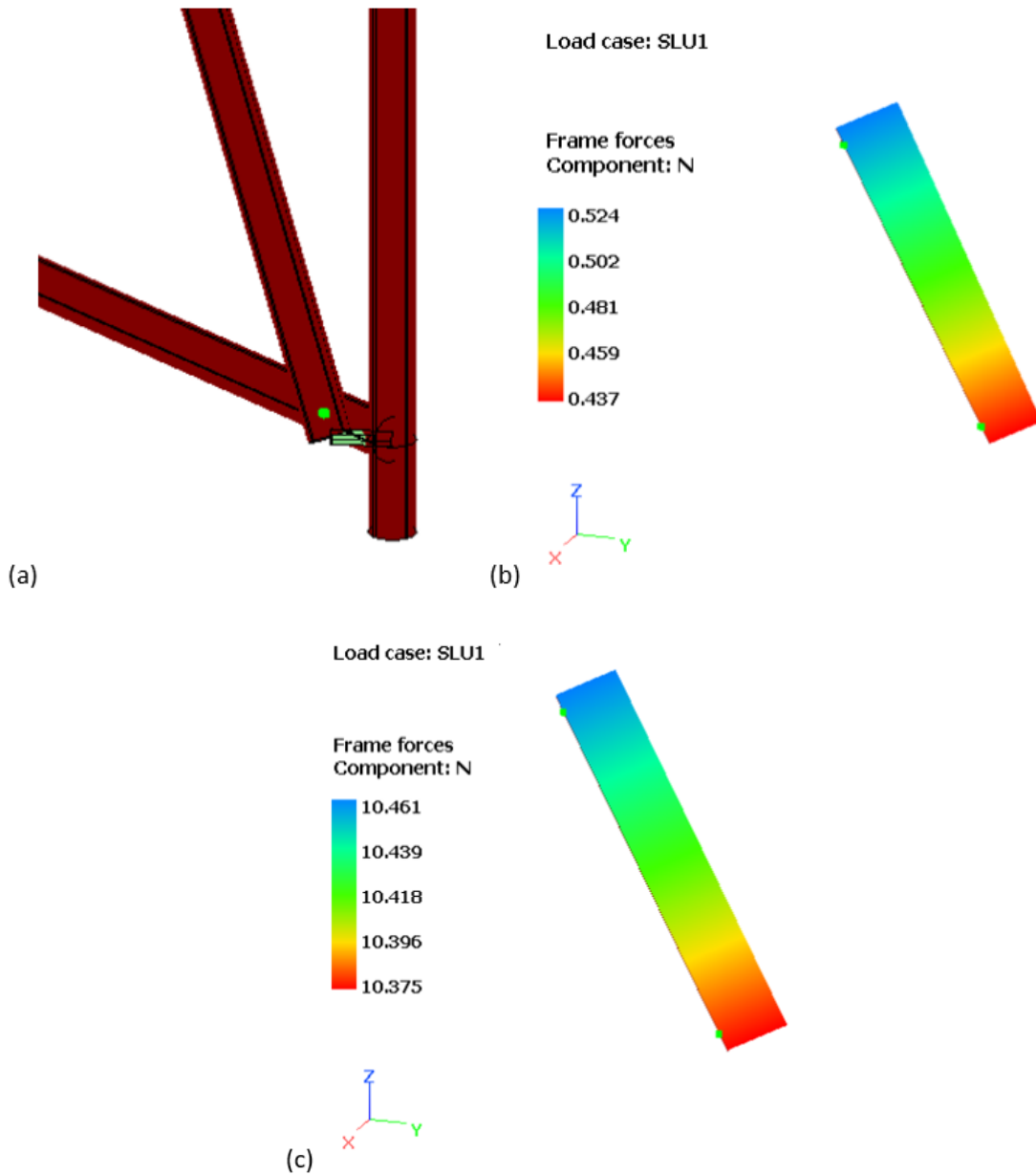
#### *Joint as an element*

NextFEM Designer, with the Struct module, allows you to treat a single beam element as a "joint element". Setting this flag is performed automatically in case of import of PON CAD scaffolding, and can be enabled or disabled for each item from the *Edit / Change element type / item element-> IsJoint*.



*Selection and modification masks for the joint elements (a) and detail of the model in the diagonal-upright node*

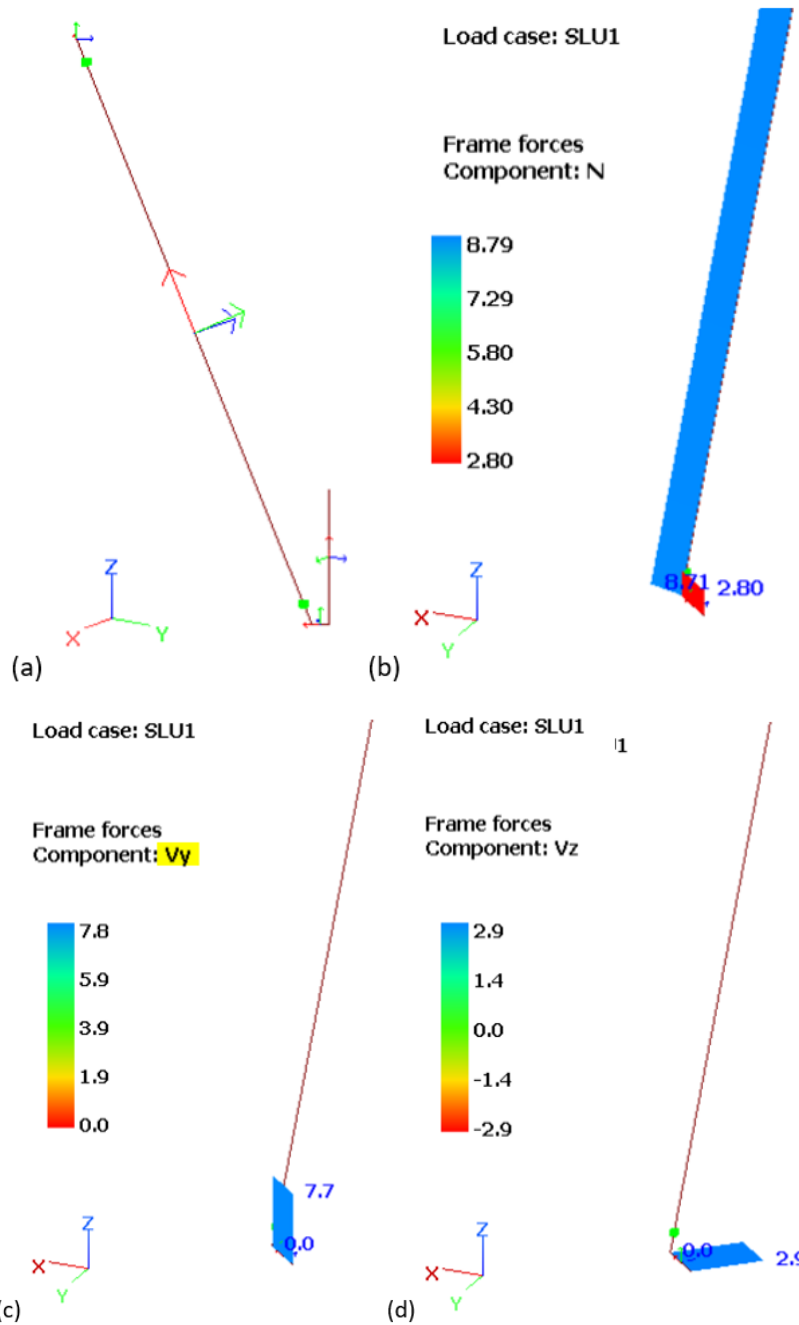
The joint is modelled as a single "squat" beam element, i.e. with a cross section of dimensions comparable to its length. In the following figure, the diagonal on the façade is connected to the upright through an element orthogonal to the development of the diagonal, which will then react to shear. The slenderness of this element is particularly important: the thicker it is, the more it will absorb shear stress, and consequently the axial load passing through the diagonal will increase, being the structure hyperstatic. A good approach is to use a cross-section with an area more or less equal to that of the connecting bolt, also taking into consideration the free span of this pin.



*Modelling detail imported from PON CAD of element ( clamp ) connecting the riser to the diagonal of the facade*

As a demonstration of this, the normal effort of the diagonal of figure (a) of the same scaffolding, connected with joint elements having a circular section full of diameter  $D = 5\text{mm}$  (b) or  $D = 50\text{mm}$  (c), as shown in the previous figure, with the same length of the joint element (once the diameter of the connected pipe, about 48mm). Notice how the axial effort that passes through the beam changes about 20 times.

Apart from modelling, please note in the following figure that the axial force in the diagonal is the result of the 2 shear forces that pass through the joint element that connects it to the upright. For the verification of the joint element, the forces referred to its local axes  $x$ ,  $y$  and  $z$  are considered, oriented as depicted in the following figure (a).



Local axes of joint and diagonal façade elements (a), axial stresses for the first load combinations SLU (b), Vy shear (c) and Vz shear (d)

The resultant of the sliding for the joint element is evaluated during the verification phase by the Struct module as:

$$F_{s,res} = \sqrt{V_y^2 + V_z^2}$$

while the separation force is equal to  $N$  if it is traction, otherwise 0.

Conservatively, the cruciform moment it is calculated as the maximum of the results at the ends

$$M_{B,res} = \max\left(\sqrt{M_{yI}^2 + M_{zI}^2 + M_{TI}^2}, \sqrt{M_{yJ}^2 + M_{zJ}^2 + M_{TJ}^2}\right)$$

to represent both the twist of clamps and the cruciform moment for orthogonal joints. This formulation particularly in favour of safety is used since, in the case of sleeve joints, the resistance considered must be associated with the moment of equilibrium and therefore the factor 2

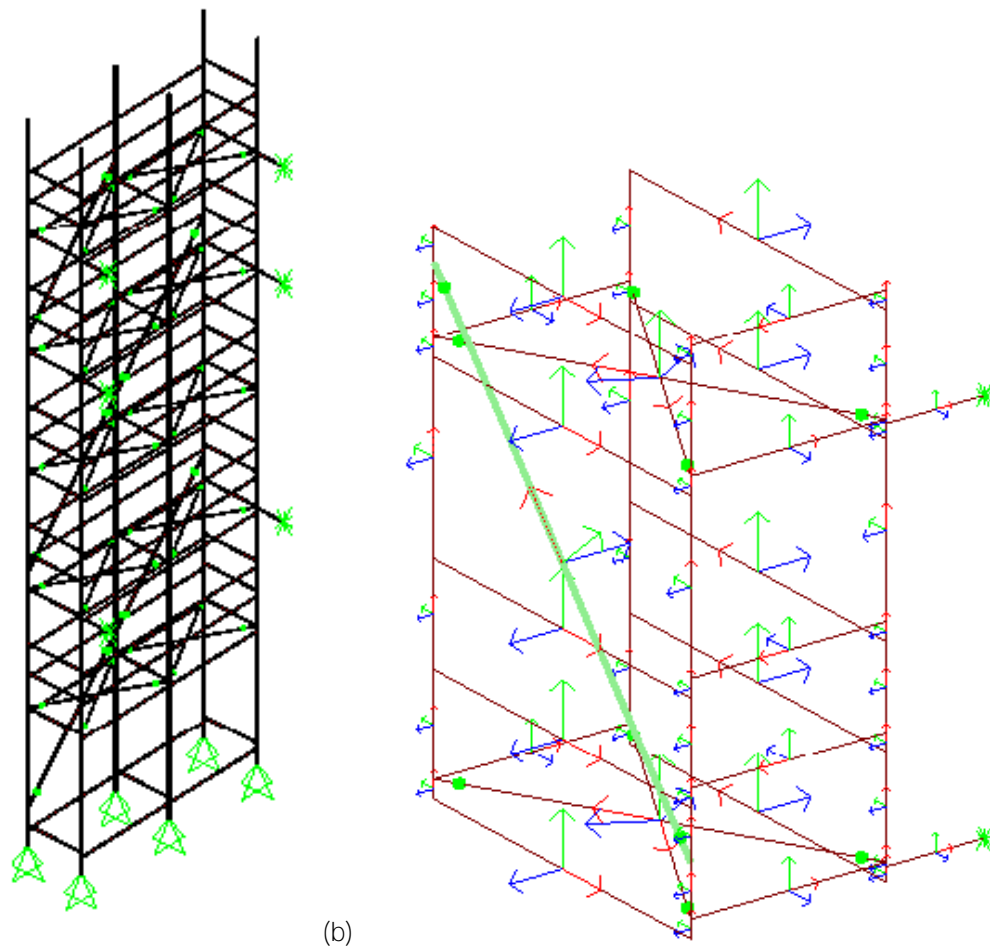
is not found in the denominator of the verification relationship. Likewise for the sliding, since for the sleeve joints the ratio between the results of the pulls and once the sliding resistance is particularly conservative.

The general formula presented above is then used to evaluate the ratio of demand / capacity of the joint.

In prefabricated multidirectional scaffolding, the verification is not always significant: in general, the resistance of the façade / diagonal connection is over-resistant with respect to the critical Eulerian load of the diagonal itself. The Struct module allows a complete personalization of the verification values to allow the designer complete control of these connections.

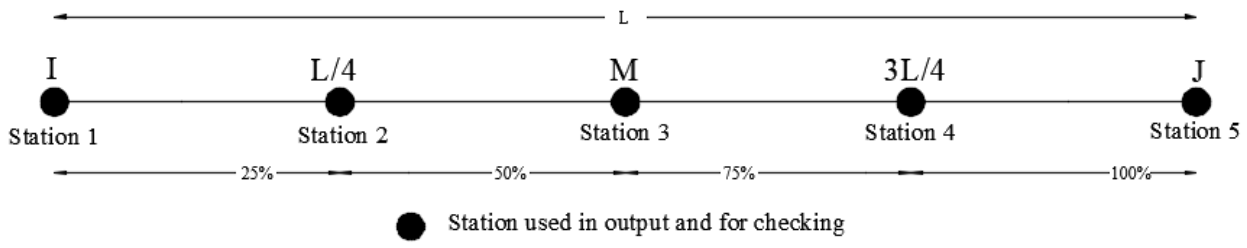
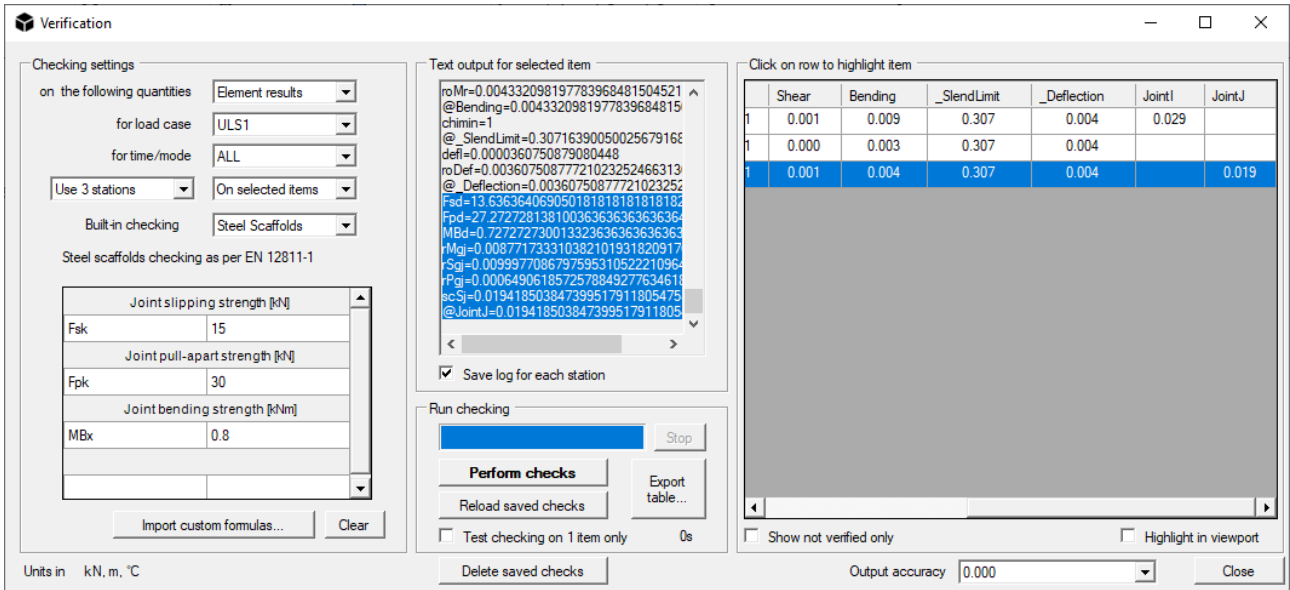
*Coupler as end-joint of a beam*

The most common case of joint modelling (and also the least expensive in terms of input time of the structural model) is certainly to consider the joints as "incorporated" in the ends of each rod. Consider, for example, a scaffolding made entirely with tubes & fittings.

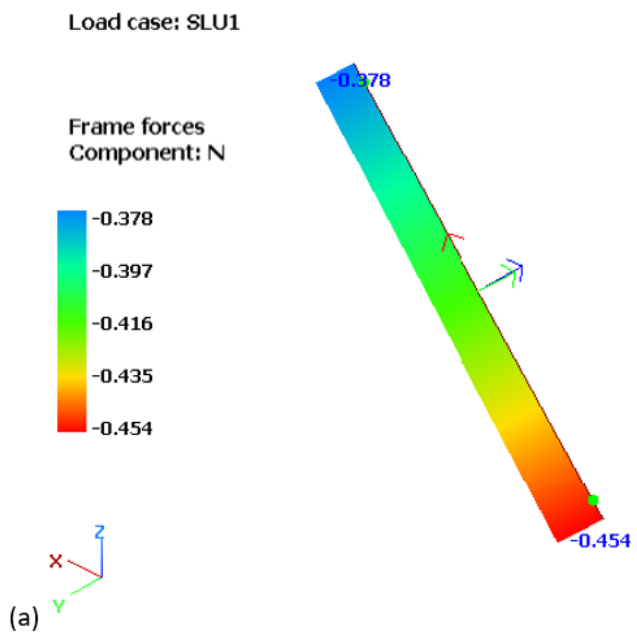


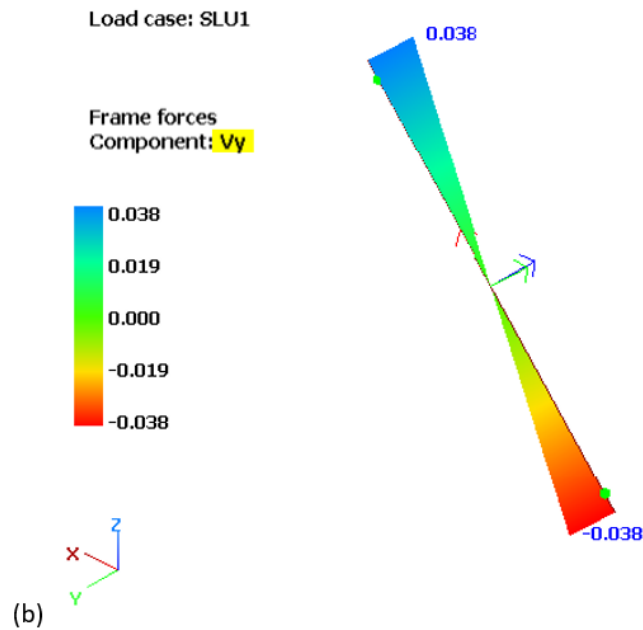
*Axonometric view of the tubes & fittings scaffolding (a) and local axes of a portion, in which the diagonal is highlighted in green (b)*

The Struct module, having received the indication of the pipe-joint rods from PON CAD (in this case all, excluding the risers), assigns the "joint-node" flag to the end nodes of the same rods. In this way, it will be possible to identify them in the verification of the elements: in station 1 (*extreme I*) the joint will be checked at the first end, in station 5 (*end J*) the one at the opposite end.



This automatism simplifies and shortens the operations that the designer should lead otherwise for the verification of each joint. The verification is carried out as for the joint elements, taking care to obtain the actions acting differently, always according to the local axes of the connected element.





Axial stress (a) and Vy shear due to self-weight (b) for the SLU1 load combination for the diagonal under examination

The resultant of the sliding for each joint node is evaluated during the verification phase by the Struct module as:

$$F_{s,res} = \sqrt{V_y^2 + N^2}$$

while the separation force is equal to  $V_z$ , with any sign (the crushing of the joint is also conservatively counted).

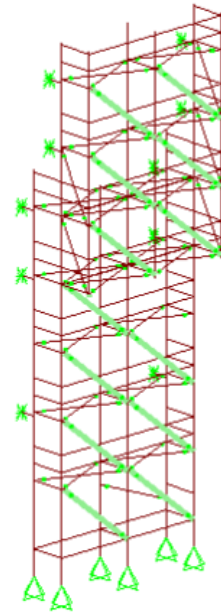
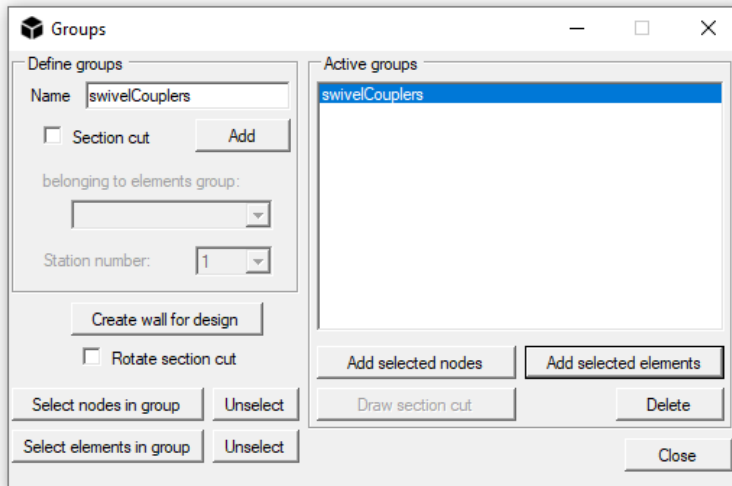
Conservatively, the cruciform moment is calculated as  $M_{B,res} = \sqrt{M_y^2 + M_z^2 + M_T^2}$  to represent both the bending of clamps and the cruciform moment for orthogonal joints. The general formula presented above is then used to evaluate the ratio of demand / capacity of the joint.

Finally, the Struct module allows to exclude from the verification a mechanism (for example the flexural one for orthogonal Class A fittings) by setting the relative resistance to zero (eg  $MB_x = 0$ ).

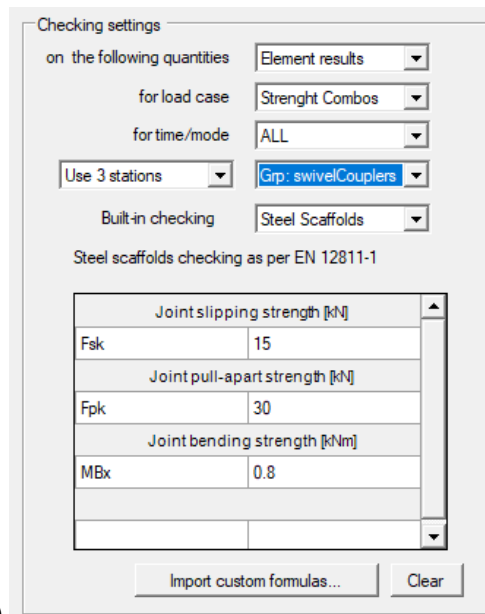
Joint slipping strength [kN]	
Fsk	15
Joint pull-apart strength [kN]	
Fpk	30
Joint bending strength [kNm]	
MBx	0

As we have seen, *Struct* deals with the verification of fittings in the same way, being already undifferentiated in the CAD environment. This saves the user the classification in groups, which is however possible manually with the aim, for example, of verifying with orthodontic parameters the orthogonal, revolving, parallel joints, etc.

NextFEM Designer makes it possible to create groups, so that they can be recalled during verification, with the command *View / Select by groups ...*



(a)



(b)

In the case of joint nodes, it will be sufficient to select the elements involved, while the joint-elements already represent the desired members. It will be enough to select the desired group during the verification phase, as in figure (b).

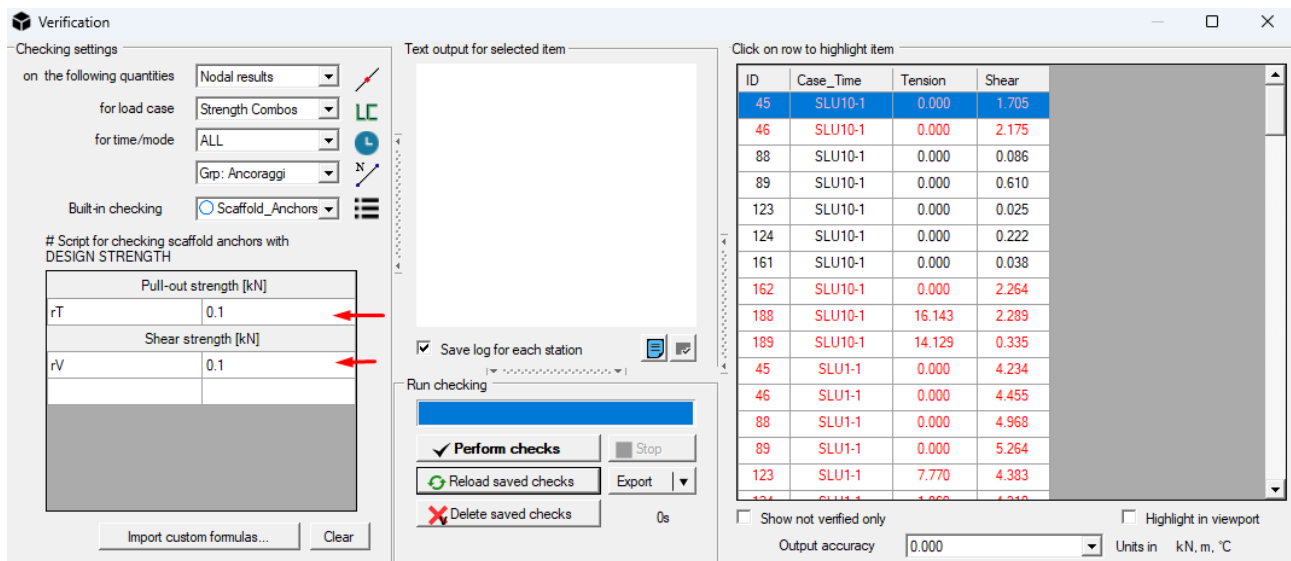
#### *Anchors checking*

From version 2.7 onwards, the anchors of the scaffolding to the structure are verified with the "Scaffold Anchors" verification set, in which the values of  $rT$  (tensile strength) and  $rV$  (shear strength) in kN calculated on the reference material (reinforced concrete, masonry, etc.) of the structure to which the scaffolding is anchored must be specified. The values are design values, i.e. they already contain the appropriate material safety factors, and can be obtained from a calculation performed according to EN 1992-4 (Part 4: Connections) or from the anchor manufacturer's tables.

The verification must be set up using:

- Nodal results (NOTE: the verification does NOT work on elements, although the programme can still perform it)

- combinations for strength check (Strength Combo)
- select the group of anchor nodes ("Grp: Anchors"). If the group is not present in the window, the scaffolding must be re-imported from PON CAD into Struct using NextFEM Designer 2.7 or later.



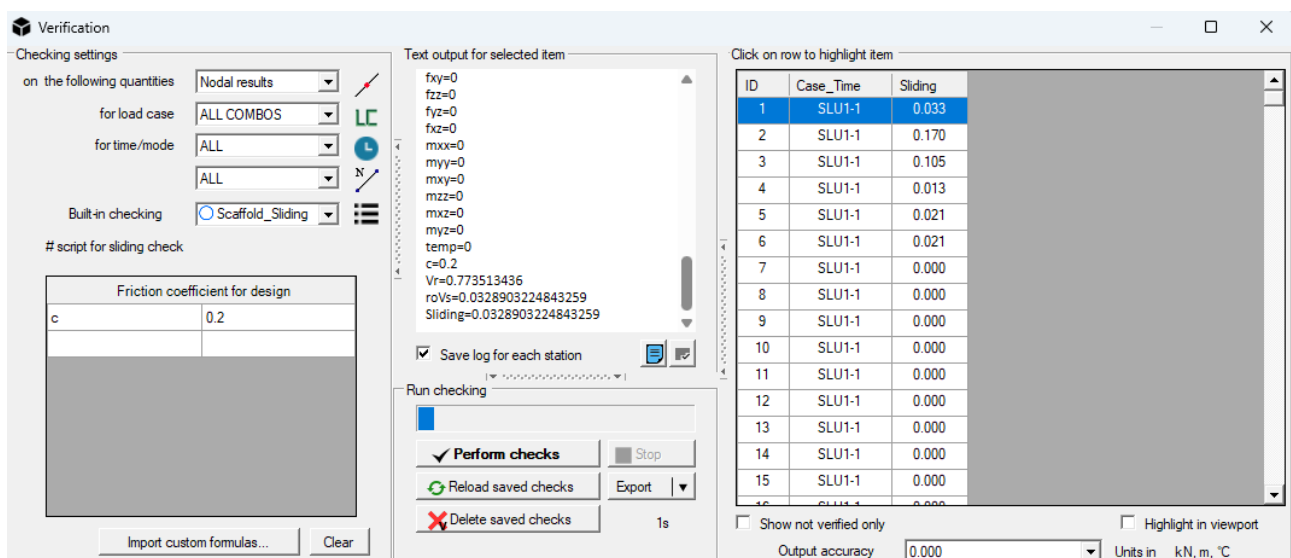
### Base plate sliding check

The total sliding check is carried out by checking that the scaffolding does not slide on its horizontal support plane, that means there are no slips at the interface between the base plates and the ground.

The adjective "total" reminds us that it's a global verification. Unless there are force compensations between the base plates, the check can also be carried out locally, making sure that each individual plate is always compressed with an effort that will generate a friction effect at the interface with the ground that is higher than the transmitted shear force.

In terms of modelling results, it will be sufficient to compare the base shear with the product between a friction coefficient (chosen appropriately by the user) and the axial stress.

The friction coefficient depends on the roughness of the surfaces in contact, i.e. base plates and ground: an appropriate and conservative value in case of normal use is 0.2. This coefficient is dimensionless, since it represents the ratio between tangential and normal stresses (and, by extension multiplying by the same area, between axial force and shear).



### *Check for uplifting*

The uplifting check is considered satisfied when none of the base plates works in tension, i.e. it occurs through the linear model that no base plate is uplifted, under every combination. This is a check that can be carried out locally: the base plates are represented in the model by supports, and therefore it is sufficient to check for each of the minimum envelopes of the load combinations (at Ultimate and Serviceability Limit States) that the vertical reactions of the basic constraints are always positive (directed upward).

### *Capsizing checks*

The overturning checks, mandatory for all types of construction, are always onerous since they must be carried out manually. In fact, the infinite element method is of no help, since it has been designed to provide the deformations of the calculated structure, and cannot consider the construction as a rigid body, except through dedicated approaches.

Overturning is a kinematic analysis, aimed at ascertaining the static balance of the artefact, which consists in verifying that the overturning moments are less than the stabilizing ones. The analysis is usually carried out by hypothesizing a point (structures in 2D) or an axis (in 3D) of rotation which acts as a pole for the calculation of the moments of each individual load acting on the structure, including its own weight.

This type of analysis is therefore used to test the overall stability of the product under the design actions in ultimate conditions (SLU), and is regulated by the application of combination coefficients that can vary from norm to norm. In Eurocodes, the coefficients for the equilibrium combinations as rigid body EQU vary according to the type of load:

- For G1 permanents (own weight): 0.9 if favourable, 1.1 if unfavourable
- For non-structural G2 permanents (carried): 0.8 if favourable, 1.5 if unfavourable
- For Q variables: 0 if favourable, 1.5 if unfavourable.

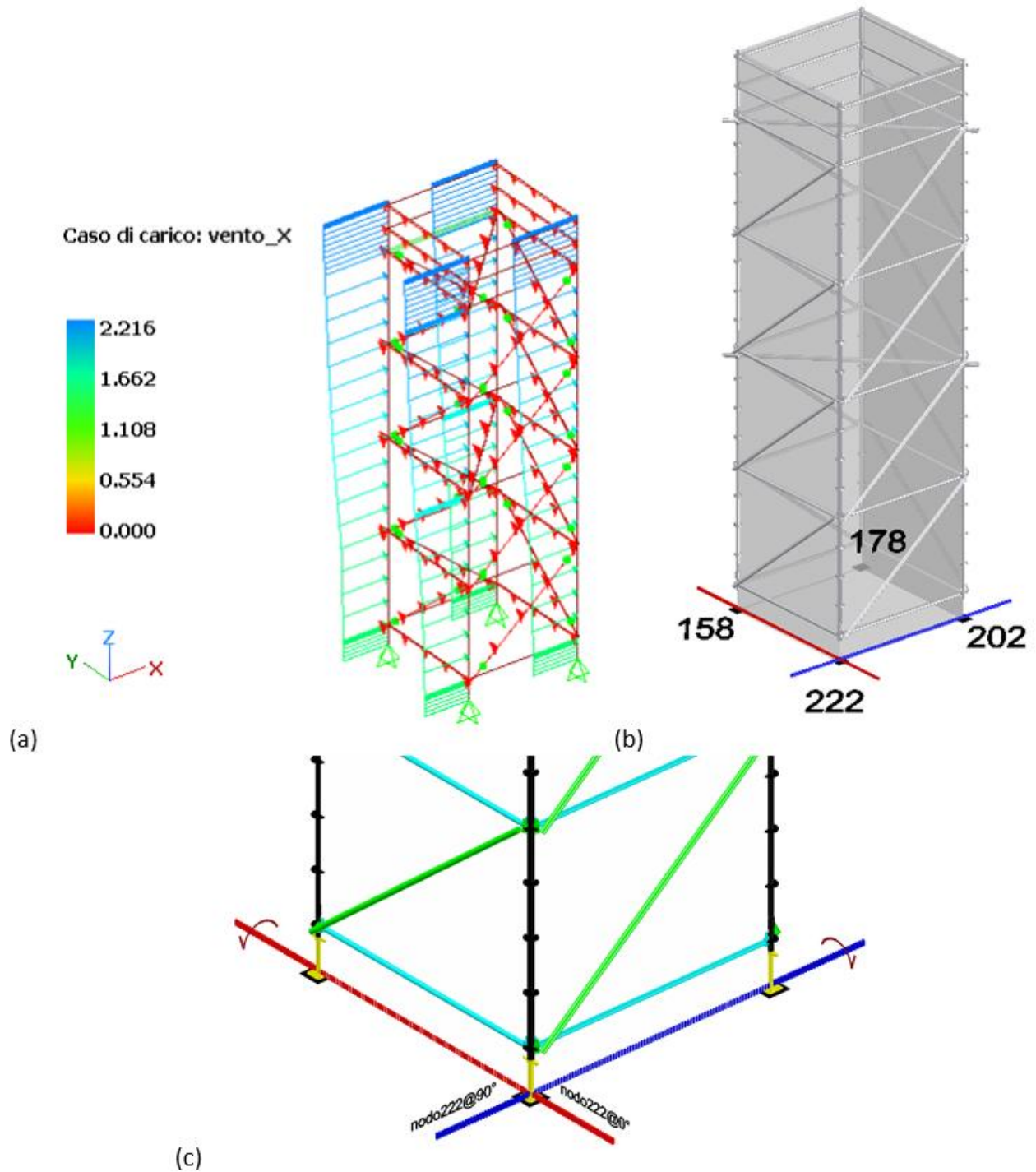
For scaffolding and stages for example, the EN 12811-1 standard explicitly refers to EN 12812 *Support structures for permanent works - Performance requirements and general design*, which reports the following coefficients:

- Permanent loads Q1 (own weight): 0.9 if favourable, 1.35 if unfavourable
- Persistent variable loads Q2 (e.g. supported construction): 0.9 if favourable, 1.35 if unfavourable
- All other loads: 0 if favourable, 1.5 if unfavourable.

Note that the safety margin against tipping is always 1.5, even in the case of only self-weight and weight bearing ( $1.35 / 0.9 = 1.5$ ).

We now come to the calculation example. Let's consider an audio tower built with scaffolding sets as a case study. This is the typical temporary construction which requires an appropriate overturning check, for example against the wind. The audio tower has dimensions 2.5 x 2.5 x 9.5 m

The calculation, if carried out manually, must consider the worst situation, that is, less conservative, between the 2 axes, orthogonal to each other, passing through each node tied to the base. In other words, we consider all the possible tilting axes of the structure, 1 per side.



Wind load in X for an audio tower (a), numbering of the base nodes (b) and calculation axes for overturning with respect to a node (c)

Now let's launch the command *Results / Advanced data / Verify overturning* . The following mask will appear:

The user is asked for the angle in degrees on which to set the axes. This angle measures the deviation of the sides of the building with respect to the global X and Y axes. case is null.

The safety factors listed above for EN 12811-1 are also preset.

The command returns the stabilizing (- sign) and overturning (+ sign) moments added, in the unit of measurement of the model, already inclusive of the specified coefficients.

Therefore, a negative moment means that the overturning verification is satisfied, a negative one that the structure overturns.

The verification is carried out for all the axes passing through each base node, at 0 ° (parallel to global axis X) and at + 90 °. Unverified (positive) values are highlighted in red.

As you can see in the cell highlighted in the following table, the system does not *tip over* in the *SLUb* combination calculated manually previously.

Caso di carico	222@0°	222@90°	178@0°	178@90°	158@0°	158@90°	202@0°	202@90°	Inviluppo
pp	-4.622	-4.622	-4.622	-4.622	-4.622	-4.622	-4.622	-4.622	0.000
pem	-35.999	-18.000	0.000	-18.000	0.000	-18.000	-35.999	-18.000	0.000
var	35.386	0.028	0.000	0.028	0.000	0.028	35.442	0.028	35.442
vento_X	0.000	0.000	0.000	570.561	0.000	0.000	0.000	568.897	570.561
vento_Y	0.000	0.000	570.561	0.000	568.897	0.000	0.000	0.000	570.561
imperf	0.000	0.000	0.486	0.486	0.484	0.000	0.000	0.484	0.486
SLU1	-5.235	-22.593	-4.136	548.453	-4.137	-22.593	-5.178	546.788	548.453
SLU2	-5.235	-22.593	566.425	-22.108	564.760	-22.593	-5.178	-22.109	566.425
SLU3	-5.235	-22.593	-4.136	548.453	-4.137	-22.593	-5.178	546.788	548.453
SLU4	-5.235	-22.593	566.425	-22.108	564.760	-22.593	-5.178	-22.109	566.425
SLU5	-5.235	-22.593	-4.136	548.453	-4.137	-22.593	-5.178	546.788	548.453
SLU6	-5.235	-22.593	566.425	-22.108	564.760	-22.593	-5.178	-22.109	566.425
SLU7	-4.403	546.304	-3.304	-22.108	-3.306	547.968	-4.347	-22.109	547.968
SLU8	563.663	-21.761	-4.136	-21.276	-4.137	-21.761	565.382	-21.277	565.382
SLU9	-4.403	546.304	-3.304	-22.108	-3.306	547.968	-4.347	-22.109	547.968
SLU10	563.663	-21.761	-4.136	-21.276	-4.137	-21.761	565.382	-21.277	565.382
SLU11	-4.403	546.304	-3.304	-22.108	-3.306	547.968	-4.347	-22.109	547.968
SLU12	563.663	-21.761	-4.136	-21.276	-4.137	-21.761	565.382	-21.277	565.382
SLUb	-0.613	-17.972	0.000	-17.972	0.000	-17.972	-0.557	-17.972	0.000

[kN\*m] Chiudi

The algorithm implemented in NextFEM Designer for overturning does not require the finite element calculation, treating the whole model as a rigid body and without considering constraints and their possible reactions.

Finally, double-clicking on a row will generate a detailed report of the calculation in tabular format (Excel).

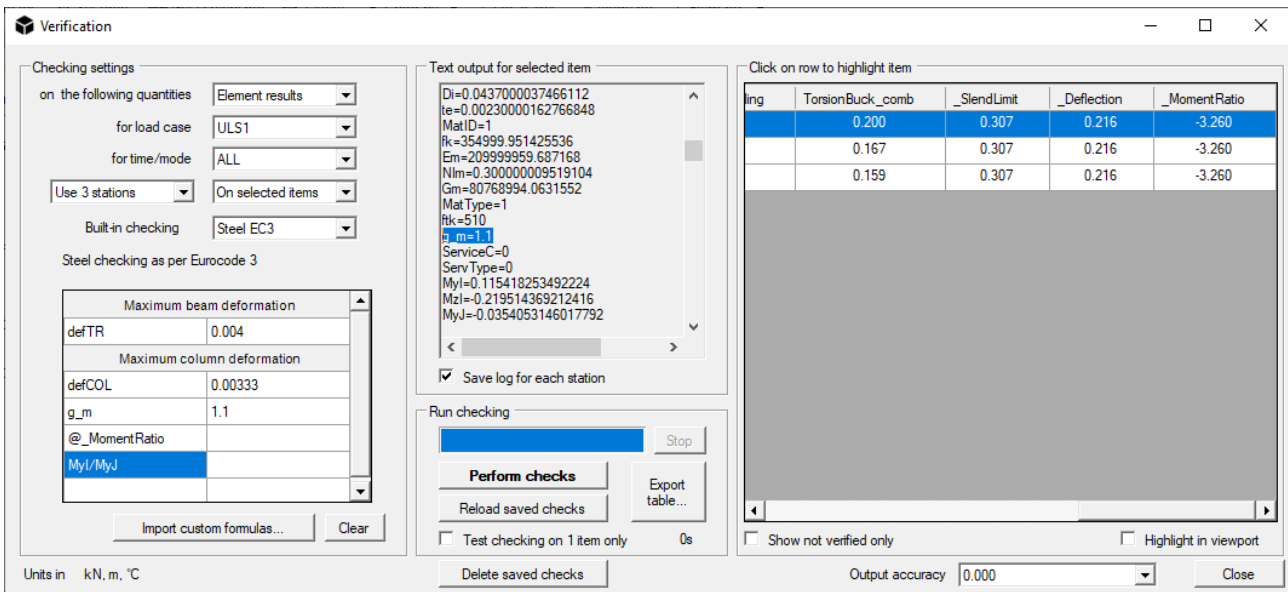
	A	B	C	D	E	F	G	H	I	J
1	###	Format: Stabilizing factor * Stabilizing capsizing moment + Capsizing factor * Capsizing moment ###								
2	###	Row header: loadcase __ reference node __ angle [deg] from X axis ###								
3	###	Result per row: positive values are capsizing moments - negative values are holding moments ###								
4	SLU7 0°	combinatio	34.44002							
5	SLU7 90°	combinatio	-111.374							
6	SLU7 0°	combinatio	34.44002							
7	SLU7 90°	combinatio	-58.5754							
8	SLU7 0°	combinatio	-26.1491							
9	SLU7 90°	combinatio	-58.5754							
10	SLU7 0°	combinatio	-26.1491							
11	SLU7 90°	combinatio	-111.374							
12										

### Custom checking

All the checks performed can be customized at will with 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 be 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.

### Generic analysis report

The report described in this chapter is replaced by the one generated from the template provided by PON CAD, generated using the relevant button in the *Scaffold Assistant*.

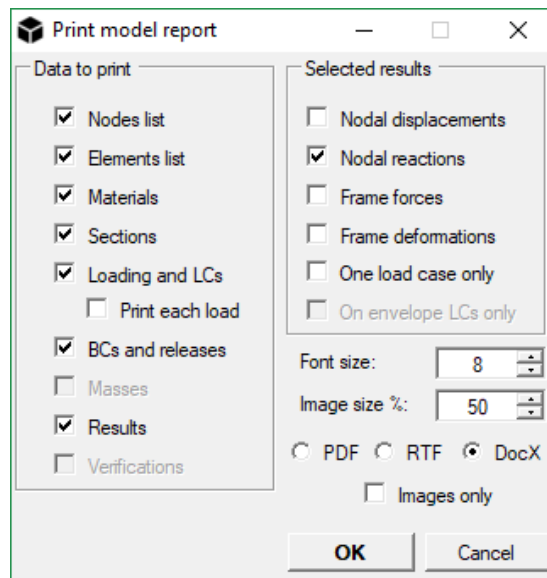
The report of the performed analysis and verifications can be obtained with the command *Tools/Print report...*

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...*).

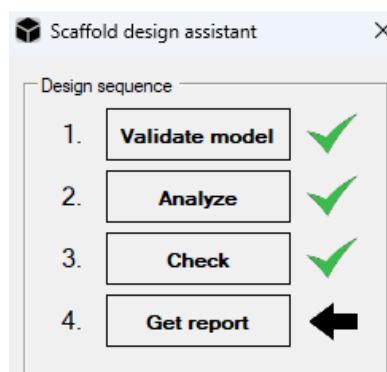
Export can take several minutes.



- ⚠ 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. ...*)

Scaffolding analysis report

The report is generated from the template provided by PON CAD, generated using the relevant button in the *Scaffold Assistant*.



Up to version 2.6, you may be asked to press "Write report" in the "Custom report compiler" window, which should look like this. If "Use provided template" is not active, load the Word template provided by PON CAD into the same folder as the template using the "Load .docx template" button.

