

WHAT'S NEW IN CONSTEEL 6.0



CONSTEEL

Version 6.0
01.09.2011.

Content

1. Preface.....	2
2. Wind load generator	2
2.1 Wind load cases.....	2
2.2 Wind surfaces	3
2.3 Wind Generator.....	6
2.4. Line wind load	6
3 Design of Composite and Reinforced concrete columns	8
3.1 Section creation.....	8
3.2 Section design	10
4 Fire Design	12
4.1 Fire effect and protection	12
4.2 Analysis.....	13
4.3 Design	14
5. National Annex Manager	14
6. Tapered member.....	15
7. User configuration file for csJoint	16

1. PREFACE

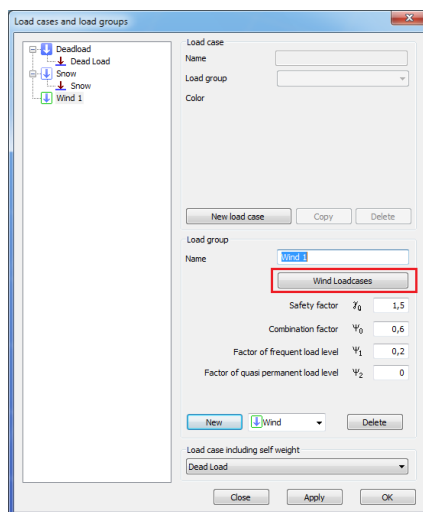
ConSteel 6.0 release includes extensive new software developments for more effective structural steel design and for more user friendly handling. In addition to the numerous new features, ConSteel 6.0 has enhanced overall stability and faster analysis. This documentation gives you a detailed overview about the main new features of ConSteel 6.0

2. WIND LOAD GENERATOR

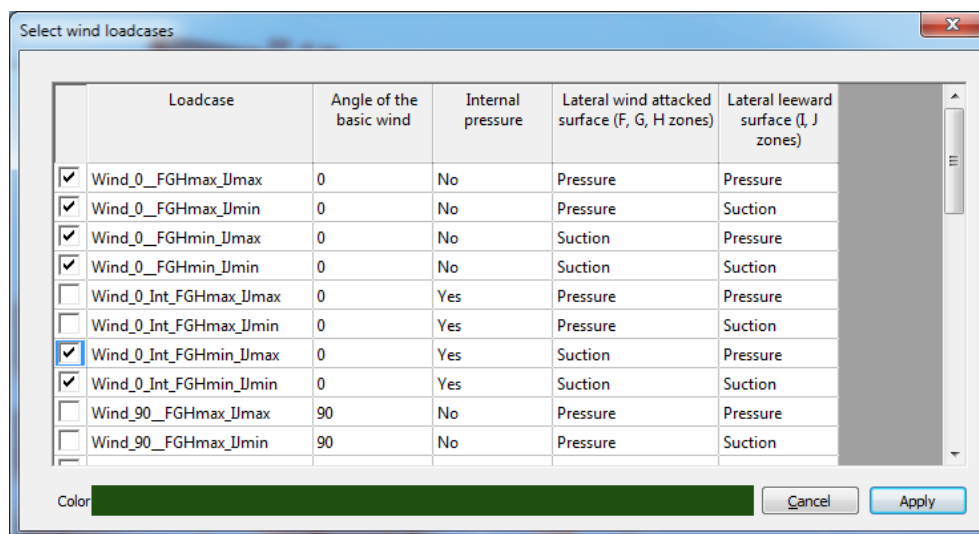
Wind load generator can generate wind load according to EuroCode 1 for the most common types of structures.

2.1 WIND LOAD CASES

Wind load cases must be defined as a first step of the wind load generation. If a wind loadgroup is created a dedicated button will be active to create the wind load cases.



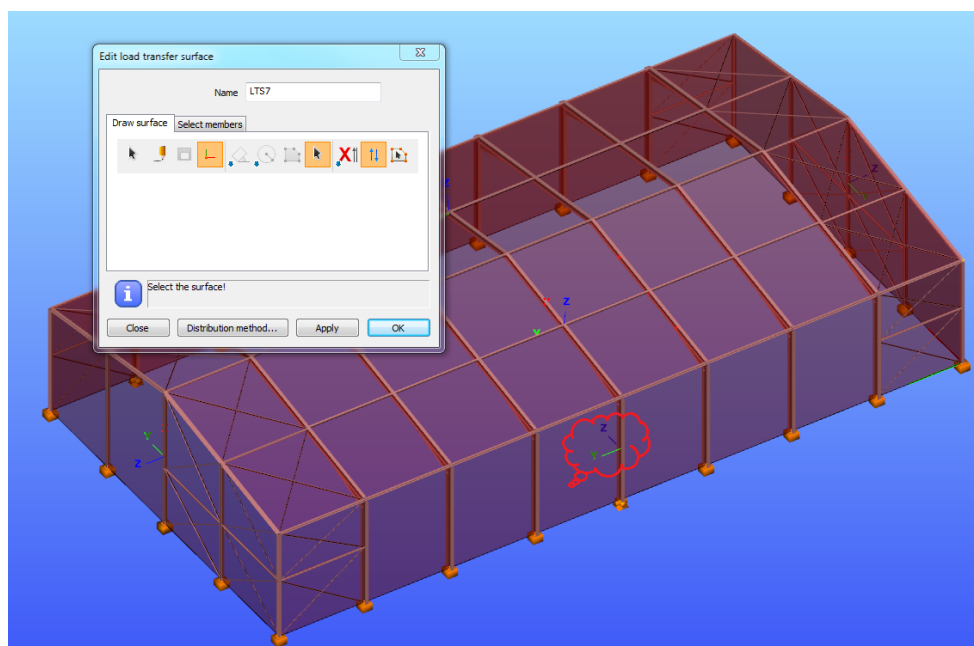
The dialog window contains all the possible EC wind load cases. Multiple selections are possible by pressing the Ctrl and Shift buttons.



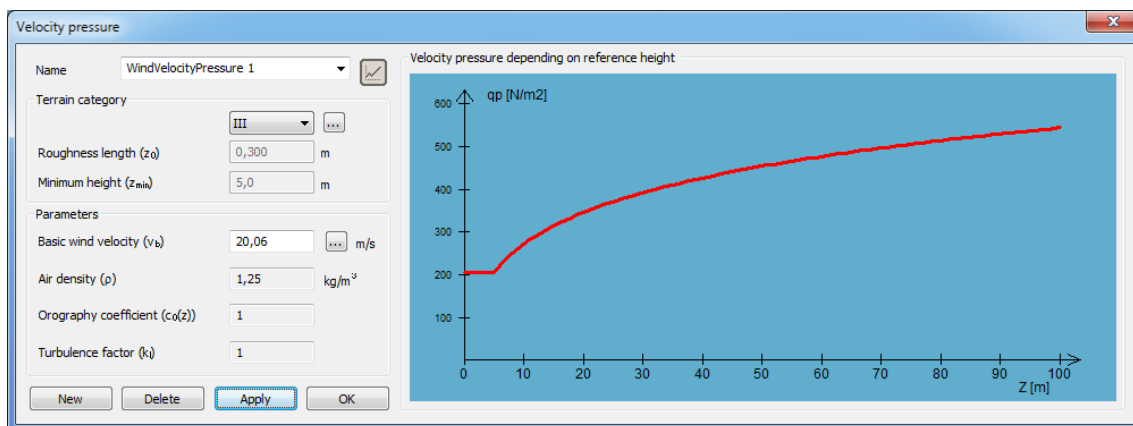
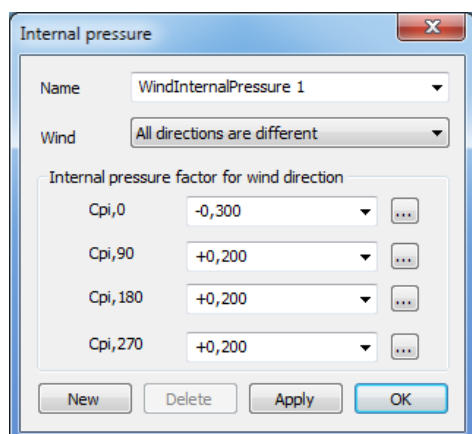
2.2 WIND SURFACES

Any load transfer surface can be selected as a wind surface. It is advised to cover all wind attacked sides of the model with load transfer surfaces.

It is important to check the local axes directions of the load transfer surfaces. All Z axes must point out of the building. If modification is necessary then it is recommended to use the *invert z axis* tool on the *Edit load transfer surface* dialog (see image below).

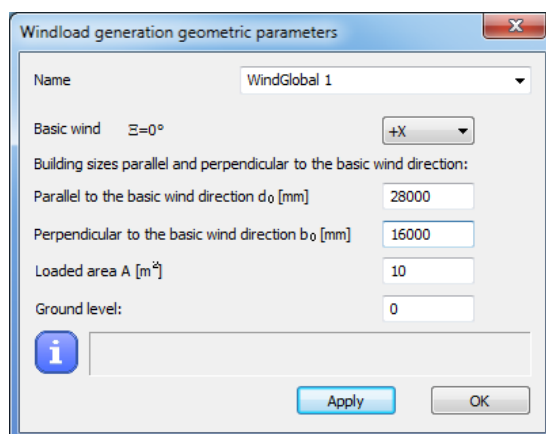


Before selecting the wind surfaces, velocity pressure, internal pressure and geometric parameters must be set.

The 'Internal pressure' dialog box shows the following settings:

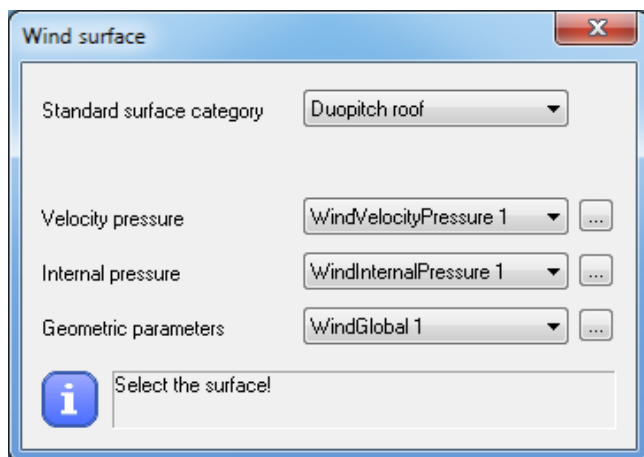
- Name: WindInternalPressure 1
- Wind: All directions are different
- Internal pressure factor for wind direction:
 - $C_{pi,0}$: -0,300
 - $C_{pi,90}$: +0,200
 - $C_{pi,180}$: +0,200
 - $C_{pi,270}$: +0,200



The 'Windload generation geometric parameters' dialog box shows the following settings:

- Name: WindGlobal 1
- Basic wind: $\Xi=0^\circ$
- Building sizes parallel and perpendicular to the basic wind direction:
 - Parallel to the basic wind direction d_0 [mm]: 28000
 - Perpendicular to the basic wind direction b_0 [mm]: 16000
- Loaded area A [m²]: 10
- Ground level: 0

The next step is to assign standard categories to the load transfer surfaces. There are 7 categories: wall, flat roof (sharp eaves), flat roof (with parapets), flat roof (curved eaves), flat roof (mansard eaves), monopitch roof, duopitch roof. If duopitch roof is chosen two surfaces must be selected directly one after the other.



The 'Wind surface' dialog box shows the following settings:

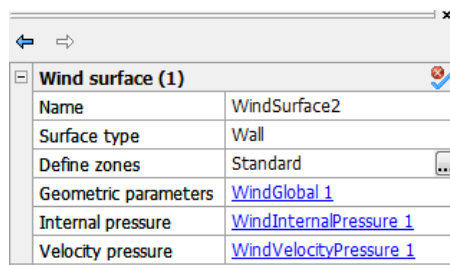
- Standard surface category: Duopitch roof
- Velocity pressure: WindVelocityPressure 1
- Internal pressure: WindInternalPressure 1
- Geometric parameters: WindGlobal 1

At the bottom, there is an information icon and the text 'Select the surface!'.

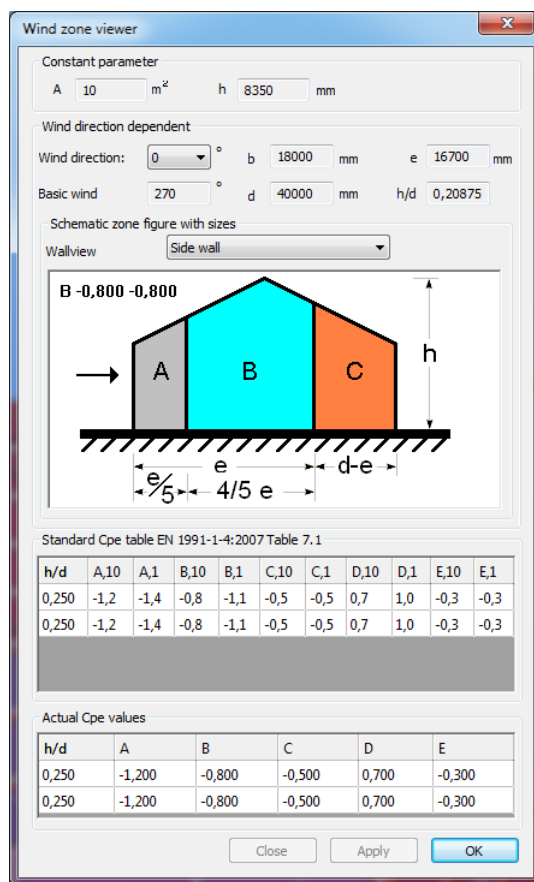


Wind surface has a symbol of an air sleeve.

Wind surface can be modified by selecting and setting the parameters

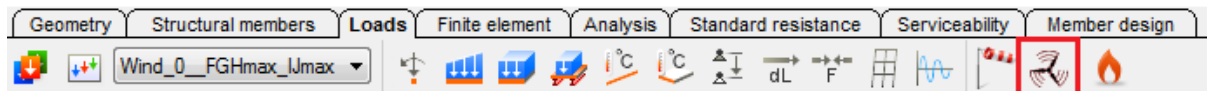


If the three dots icon is clicked the Wind zone viewer window is opened and the standard zones and Cpe values can be checked.

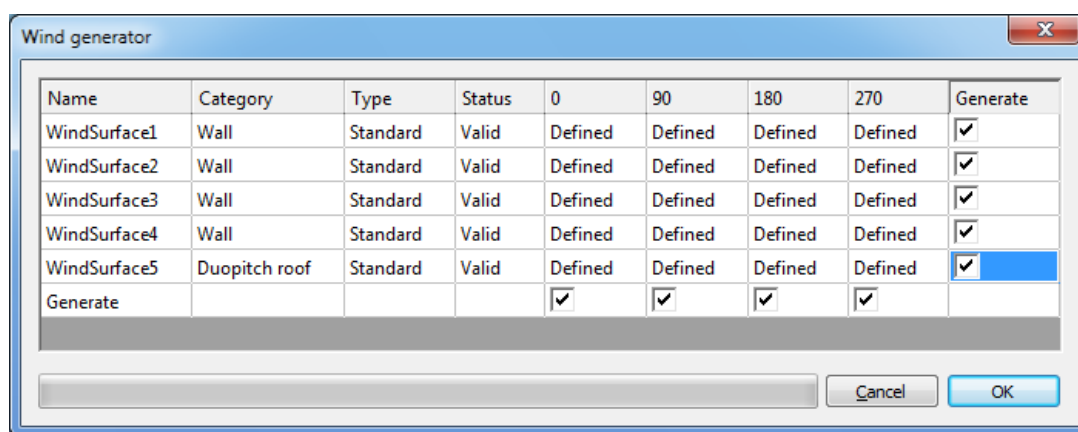


2.3 WIND GENERATOR

Wind generator function can be found on the **LOADS TAB**

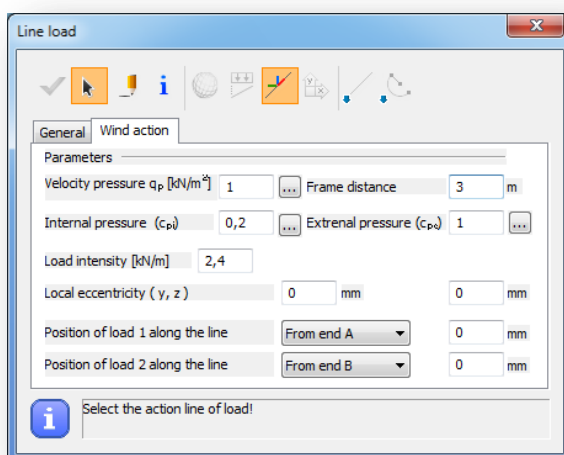


The function can be used to automatically generate wind load based on the wind surface settings (internal pressure, velocity pressure, geometric parameters). Each direction and wind surface can be turned on and off.

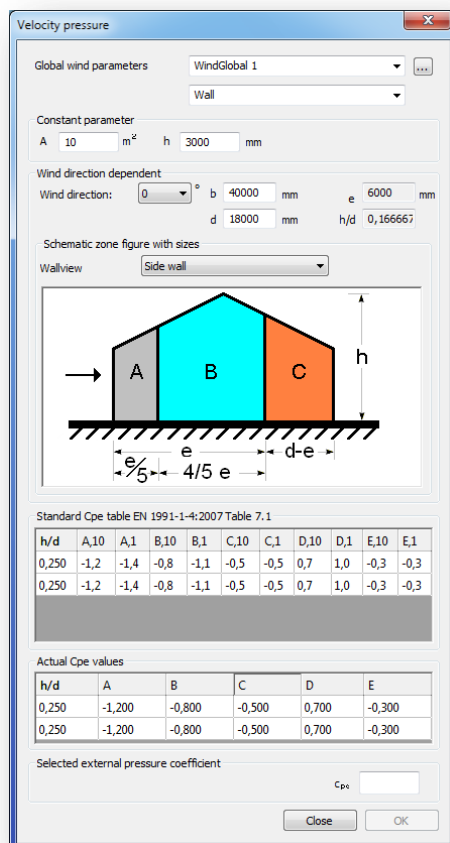


2.4. LINE WIND LOAD

Line wind load is a great tool to apply wind load to members easily. It saves time because only the basic data must be added and ConSteel automatically calculates the appropriate wind load. The three dots icons provide help determining the correct values of the parameters. It is important to know that load intensity is calculated using the parameters added on the window and these parameters can be modified later but every line wind load is independent from the others. Therefore modifying the parameter of a line wind load will not have effect on the other line loads. If one or more parameter(s) need to be modified for all line wind loads then all must be selected and the appropriate parameter(s) must be modified.



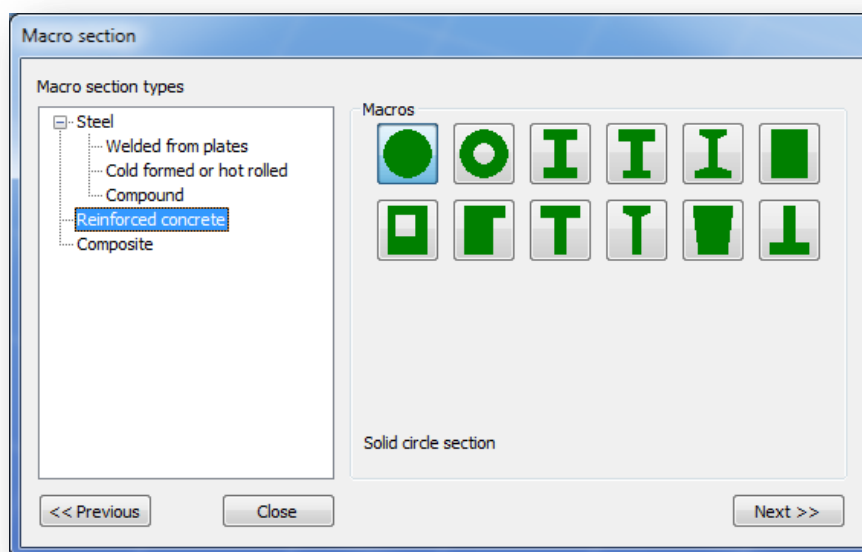
By clicking on the *Velocity Pressure* the window below will appear. It is possible to view and select the C_{pe} values after choosing the type of the wind surface (wall, flat roof, duopitch roof, etc).



3 DESIGN OF COMPOSITE AND REINFORCED CONCRETE COLUMNS

3.1 SECTION CREATION

Reinforced concrete and composite sections can be found among the macro sections.



The creation of these sections is very similar to the creation of the steel macro sections. If a reinforced concrete section is created on the bottom of window there are three possibilities for the applied modulus of elasticity in structural analysis: the secant, the design value and the effective modulus of elasticity. In the last case the creep ratio must be set. The nominal stiffness calculation according to EN 1992-1-1 5.21 formula can be turned on or off.

Rectangular section

Name

rc Trectangular

Concrete section

bc: 250 mm hc: 500 mm

Material: C20

Reinforcement

Material: B500A

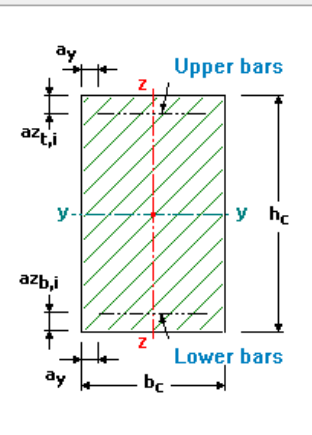
Cover - ay: 20 mm

Upper bars - number of rows: 1

Mem...	Size of membe...	Number of ...	Cover - az t (m...
1. row	16	4	20

Number of lower bar rows: 1

Mem...	Size of membe...	Number of ...	Cover - az b (m...
1. row	16	4	20



Modulus of elasticity applied in structural analysis

☒ Secant modulus of elasticity
 $E_{cm} = 22(f_{cm}/10)^{0.3} = 29962,0$

☐ Design value of elastic modulus
 $E_{cd} = E_{cm}/\gamma_c = 19974,7$

☐ Effective elastic modulus
 $E_{cd,eff} = E_{cd}/(1+\phi_{ef}) = 6658,2$

creep ratio ϕ_{ef}

2

☐ Nominal stiffness calculation according to EN 1992-1-1 5.21 formula

factor for the effects of shrinkage, creep, cracking K_c

0,5

Factor for contribution of reinforcement K_s

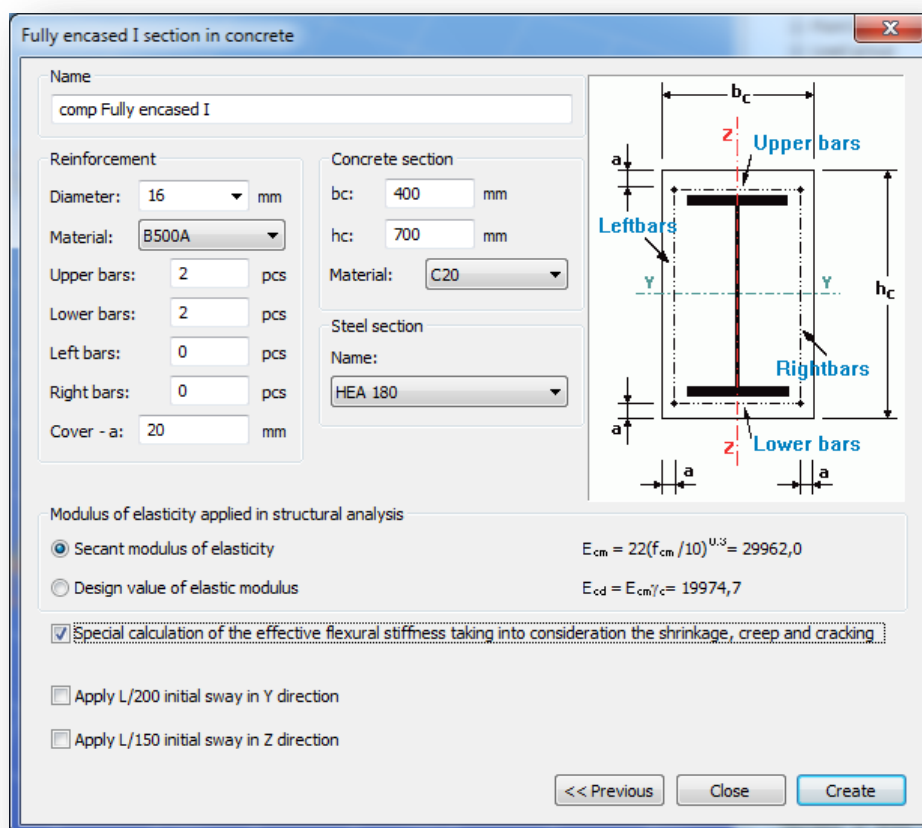
1

<< Previous

Close

Create

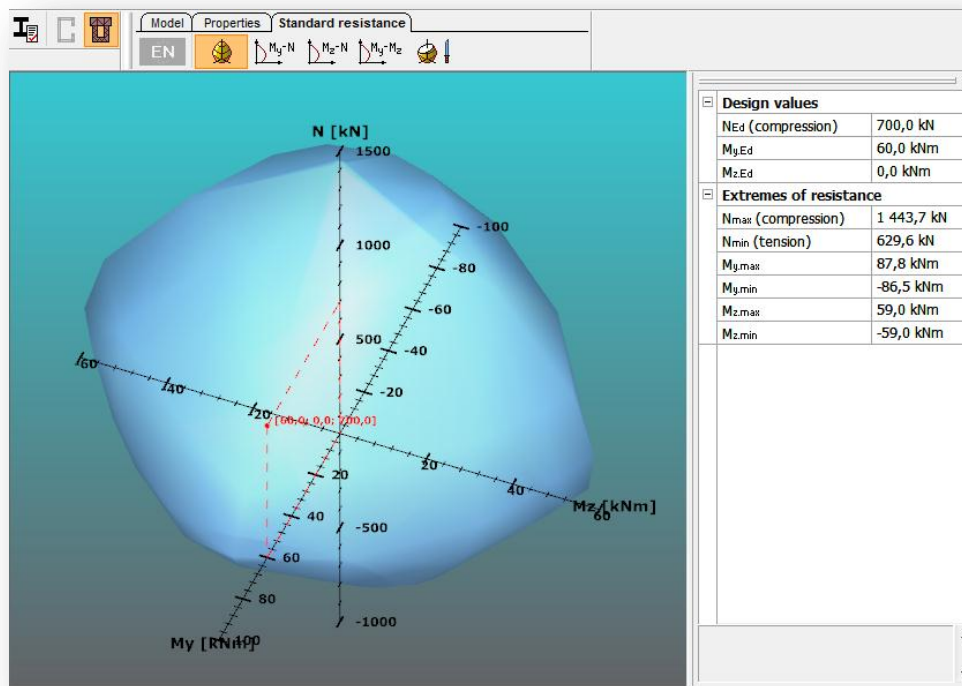
When composite section is selected, then it is possible to apply initial sway in Y and Z direction. This case no stability check must be performed according to the Eurocode standard.



3.2 SECTION DESIGN

If the section of a member is reinforced concrete or composite then the element type automatically set to beam-column 12 DOF finite element.

In the section module the resistance surface of the section can be viewed. It is also possible to view the resistance line along three different axes and the arbitrary section.



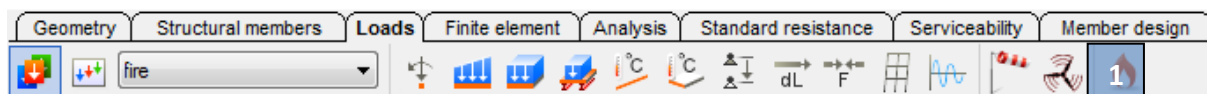
Section design according to EC 4 can be performed for composite sections

Summary	
	Adequate!
R _{max}	81,1 %
Dominant case	Interaction check
Dominant formula	Compression + Bending about the major axis
Load combination	1. Lc.
Pure resistances	
Compression	
Bending about the major axis	
Utilization	89,3 %
Applied part of standard	EN 1994-1-1 6.7.3.2 6.19 figure 6.7.3.5
M _{y,Ed}	200,0 kNm
I _{eff,Iy}	2 561 406 630,0 mm ²
M _{pl,y,Rd}	224,1 kNm
Interaction check	
Compression + Bending	
Utilization	81,1 %
Applied part of standard	EN 1994-1-1 6.7.3.6 (6.45) formula
N _{Ed}	1 000,0 kN
M _{y,Ed}	200,0 kNm
M _{pl,y,Rd}	224,1 kNm
M _{pl,y,N,Rd}	274,1 kNm
μ _{dy}	1,22
α _{M,y}	0,90
M _{pl,y,Rd} Design resistance	

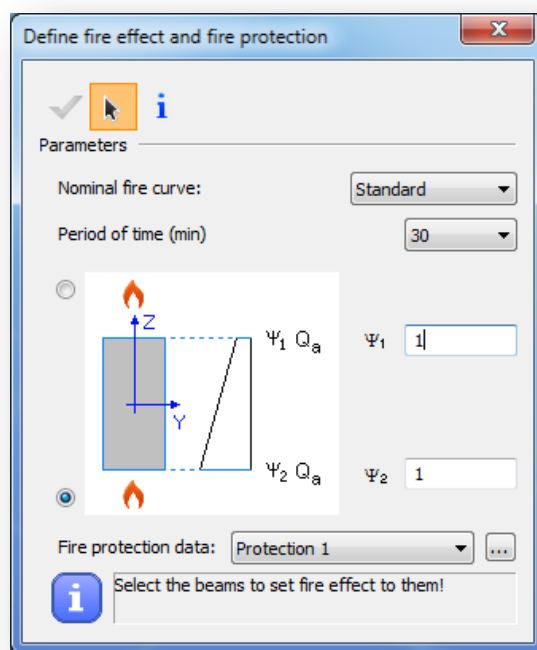
4 FIRE DESIGN

4.1 FIRE EFFECT AND PROTECTION

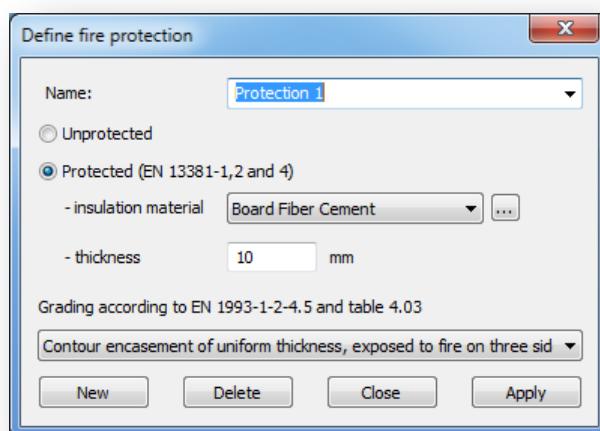
Before define fire effect an accidental load group and at least on accidental load case must be created. The fire effect and protection can be defined by clicking the appropriate icon (#1) on the **LOADS** tab.



The first option is to set the nominal fire curve. The second one is to set the fire resistance time. The fire effect can be set to different intensity in the top and the bottom of the section.



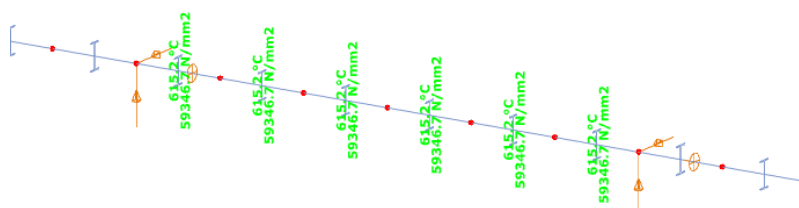
Different fire protection can be created by clicking on the three dot icon next the Fire protection data. In this window it is possible to select between Unprotected and Protected. If Protected is chosen then the insulation material and the thickness must be specified. The third parameter to set is the grading according to EN 1993-1-2-4.5.



If the fire protection is set then the members must be selected. It is possible to apply fire effect to more members at the same time. This case the members should be selected before clicking the fire icon.

4.2 ANALYSIS

During the finite element generation ConSteel calculates the steel temperature and the reduced modulus of elasticity. These values can be viewed in the **FINITE ELEMENT** tab by selecting the appropriate load case.



The analysis for the accidental (fire) load combinations are performed in global level. Therefore the calculated deformations and internal forces include the effect of the expansion and decreased elasticity modulus of those members which are subjected to fire.

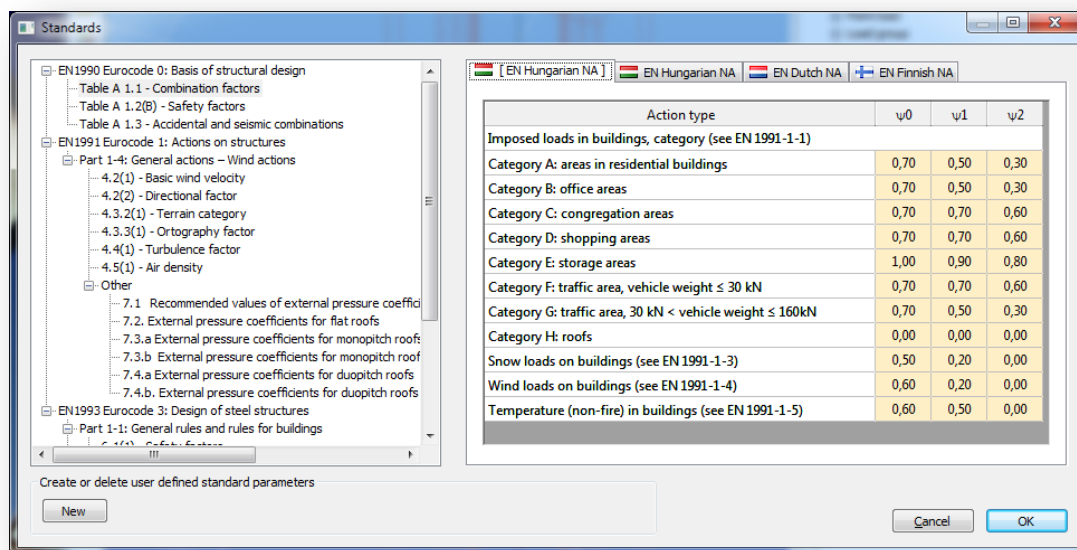
4.3 DESIGN

ConSteel performs cross section resistance checks for all members which are subjected to fire effect by using the adequate formulas from the EuroCode standard. More detailed information about the evaluated formulas can be found in the Manual of ConSteel 6. A typical cross section check in ConSteel looks like below.

Compression	
Utilization	5,4 %
Section class	1
Applied part of standard	EN 1993-1-2: 4.2.3.2
N_{Ed}	-30,0 kN
$N_{fi,Ed}$	558,8 kN
$k_{y,\theta}$	0,772
θ_A	502,6 °C
f_y	235,0 N/mm ²
γ_{M0}	1,0
$\gamma_{M,fi}$	1,0

5. NATIONAL ANNEX MANAGER

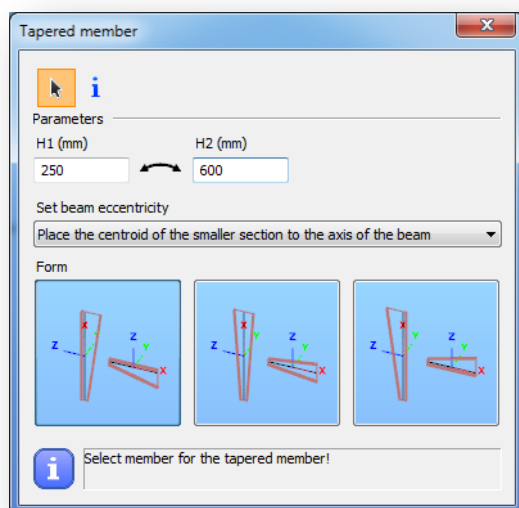
The **STANDARDS** menu provides a great opportunity to view existing and to create new standard parameters in an easy way.



The first tab shows the applied NA which is stored in the model. If a user defined National Annex is created it will be saved to the Documents\ConSteel folder as Standard.xml.

6. Tapered member

Tapered member creation is easier in ConSteel 6.0 thanks to the redesigned dialog window. The three different form types can be simply selected by clicking on the images. The arrow helps to change the start and end height effortlessly.



7. User configuration file for csJoint

By using the new user configuration file for csJoint the basic settings for the joints (i.e. default bolt size, plate thickness, weld size, etc) can be set and exported.

