WHAT'S NEW IN CONSTEEL 5.0

CONSTEEL

Version 5.0

20.04.2010.

www.consteel.hu

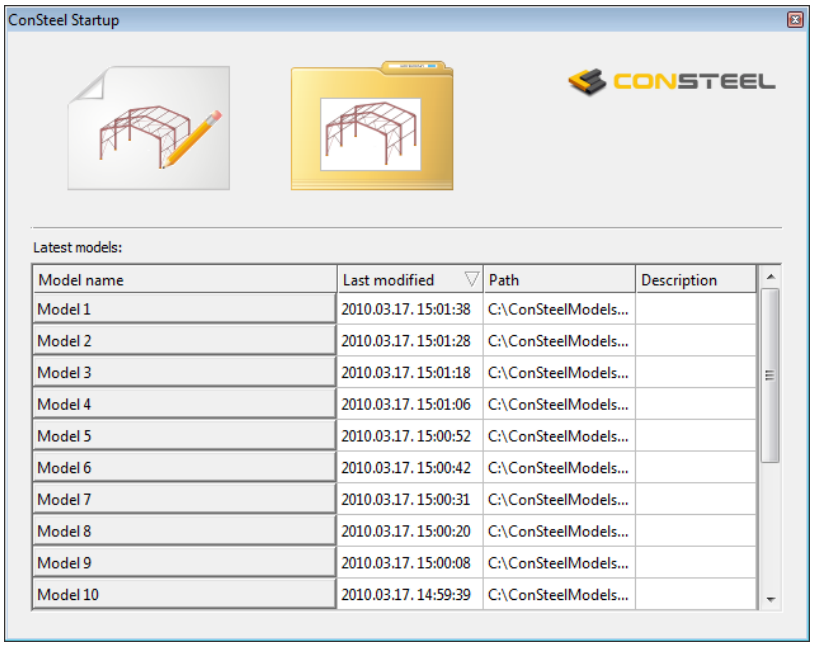
Content

1. PREFACE

ConSteel 5.0 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 5.0 has enhanced overall stability and faster analysis.
This documentation gives you a detailed overview about the new features of ConSteel 5.0

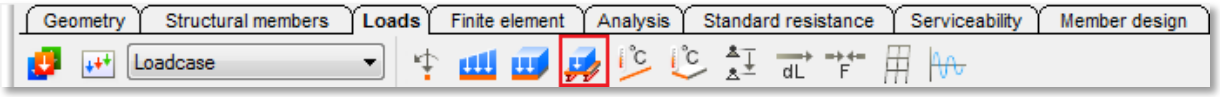
2. THE NEW STARTUP WINDOW

After starting ConSteel 5.0 there is a startup dialog window which allows creating and opening models easily. The latest models can be opened without browsing folders. They are sorted by the last modification date as a default setting but it can also be sorted by name or by model path.



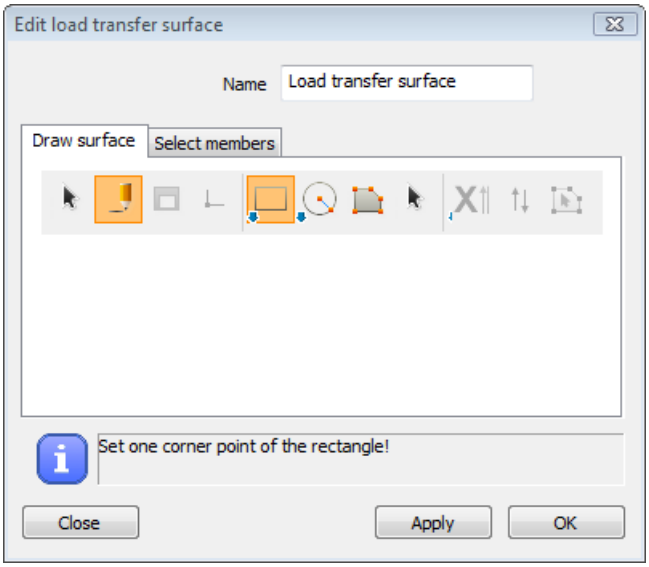
The two big icons are for creating a new model and for opening model from folder.

3. LOAD TRANSFER SURFACE

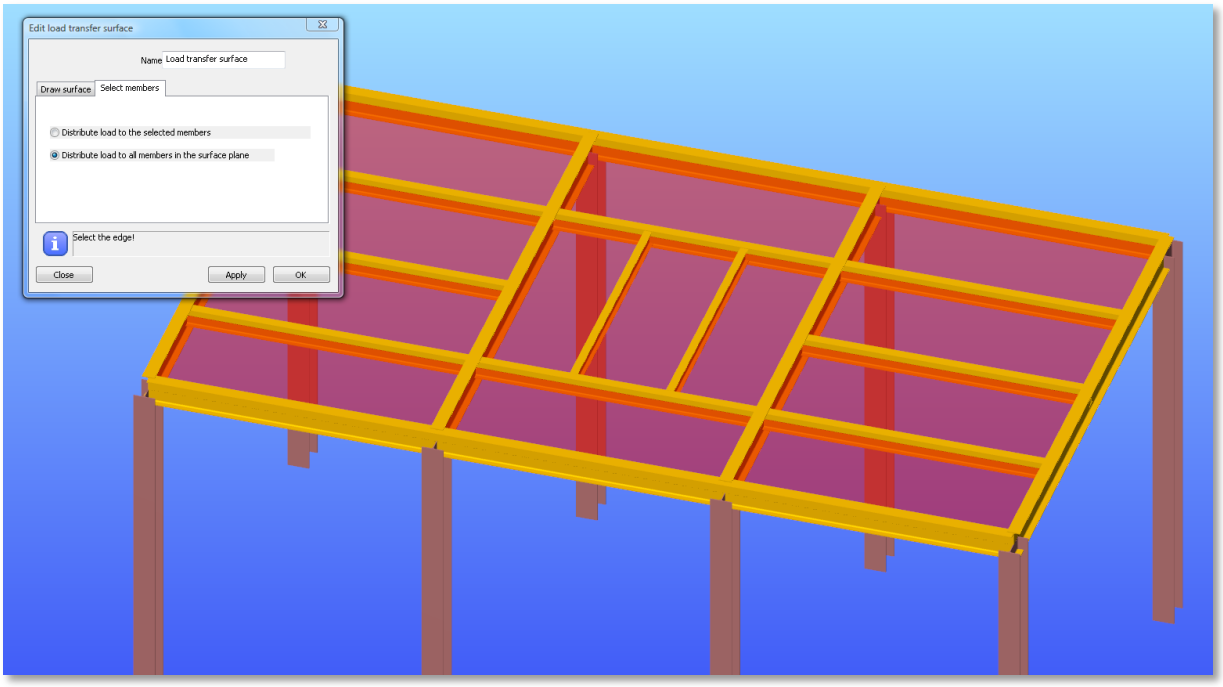


Load transfer surface is special surface which distributes surface loads to members as line loads. This is very useful in those situations when surface load needs to be distributed to members, like floor loads, snow and wind load, etc.

After clicking on the dedicated icon on the ***LOADS*** tab a dialog window appears.



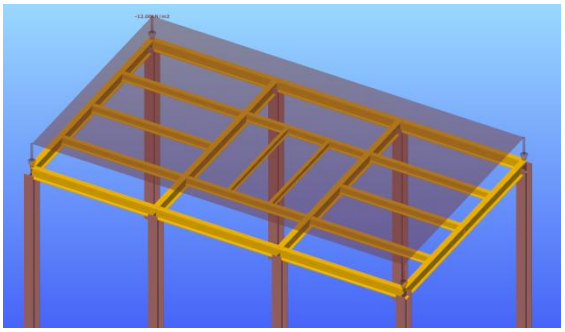
Just like in the two-dimensional figure drawing there are different possibilities to draw the surface: draw a rectangle, draw leaning rectangle, draw circle, draw polygon. After drawing the surface it is possible to select the members which the surface distribute the load. There are two options: distribute load to the selected members or distribute load to all members in the surface plane. If second option is clicked then the appropriate members will be highlighted. If none of the options is clicked, then the second one will be applied as a default.

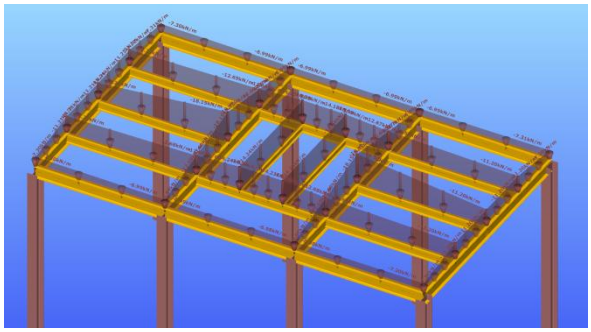


It is also possible to chose the second option and remove from the selection by using the Shift + Left mouse click on the member. After the corresponding members have been selected and OK is clicked then the surface is created.

Surface load can be placed by using the method described in chapter 6.4.3.

There are two visibility options: view the surface load or view the distributed load.





Changing the views is possible by clicking on the dedicated icon which can be found among the visibility of graphics symbols setting.

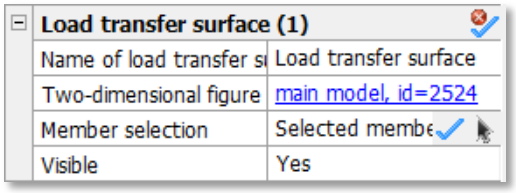
Converting surface loads to line loads is performed by a meeting the following requirements:

- line load is linear on member
- the value is the same on a node for all incoming members
- project equilibrium

The basic working method can be used as described below. First create the load transfer surface. Select the members which it distributes loads to or accept the default setting which is distribute loads to all planar members. Apply surface loads to the surface at every load case.

3.1.Modifying load transfer surface

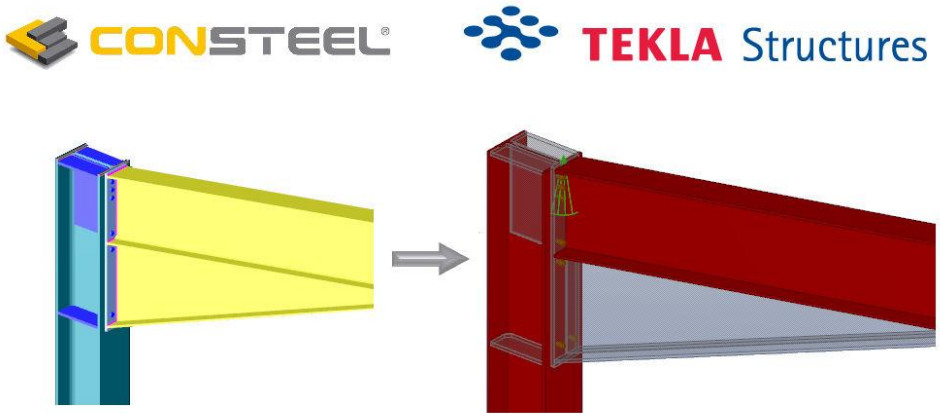
Load transfer surface can be modified by selecting it and changing the properties in the property bar. Member selection can be easily modified from the planar members to selected members. It is possible to highlight the previously selected members by clicking on the blue thick.



If selected members option is chosen and then the black arrow is clicked then previously selected members can be removed from the selection or new members can be added.

4. COMPLETE MODEL EXPORT INTO TEKLA STRUCTURES

The smooth and quick transition from the structural model to the detailing model can save significant time and cost, therefore the ConSteel has a high level interface module to the Tekla Structures. Besides the export of the global structural model (i.e. beams, columns, slabs, walls) all the joint models appear in ConSteel have the corresponding Tekla component, so the designed structural model with the placed joint models can be converted in one piece easily into detailing model without additional modeling efforts on the joints. This unique interface can save significant detailing time and therefore can appreciably reduce the project costs.

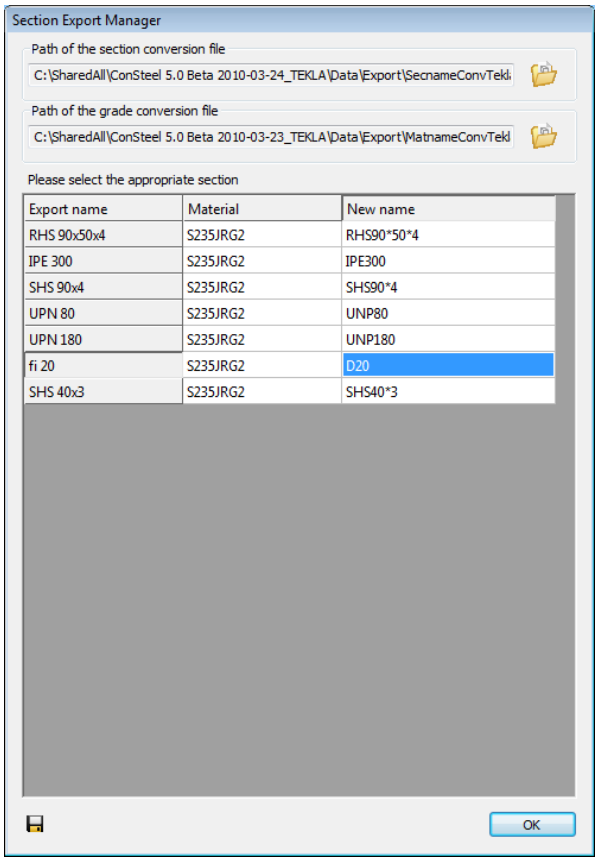


ConSteel uses Tekla Open API (Application Programming Interface), also known as the .NET API, provides an interface for third party applications to interact with model in Tekla Structures. Therefore both applications need to have installed in order to export model from ConSteel to Tekla Structures.

ConSteel export to Tekla Structures can be used from version 14 onward.

To export a model from ConSteel it is important to run both program at the same time and to have a model opened in Tekla Structures. Then click on the ***FILE*** menu and select ***EXPORT*** then select Tekla Structures. A dialog window opens asking for selecting model parts to be exported. Bear in mind that only those joints will be exported which are placed in the model and selected before exporting. The next step is to select a reference point and click on the export button.

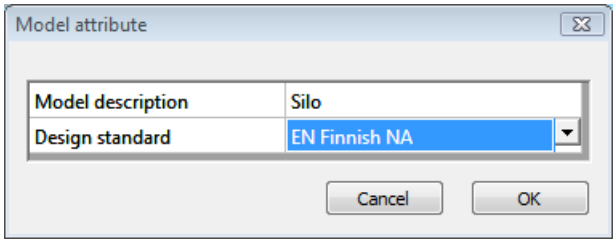
After selecting the reference point in Tekla the Section export manager window appears. It is possible to change the conversion file for grades and for sections, or edit the section name and grade manually.



If manual changes have been made then the list can be saved and can be used as a conversion file for future exports. After clicking OK the model will be exported.

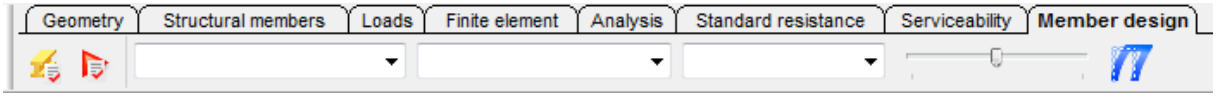
5. Handling of National Annexes

ConSteel 5.0 can handle different National Annexes for Eurocode. At the current stage Dutch, Finnish and Hungarian NA-s are included in the program.



6. MEMBER DESIGN

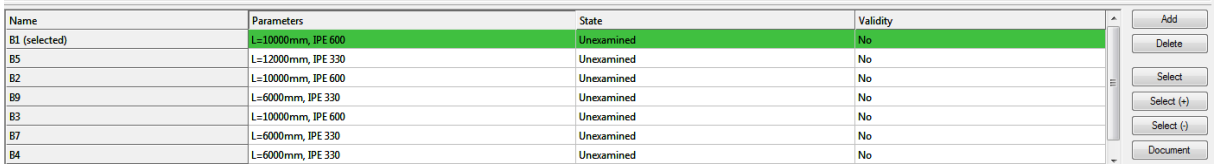
6.1 Basics



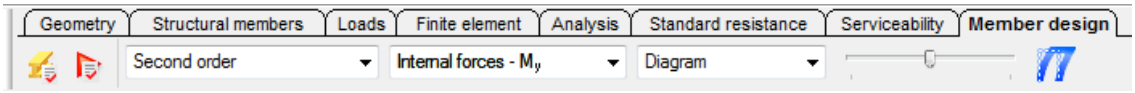
All member design related functions can be found on the ***MEMBER DESIGN*** tab.

6.2 Individual Member Design

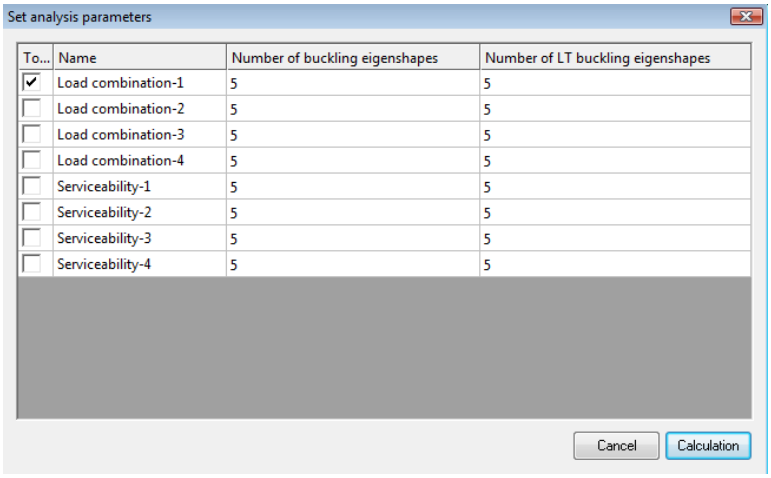
To run member design, first the members have to be selected and then added to the list. It is can be performed by clicking on the Add button. After that one member has to be chosen and Select button is clicked.



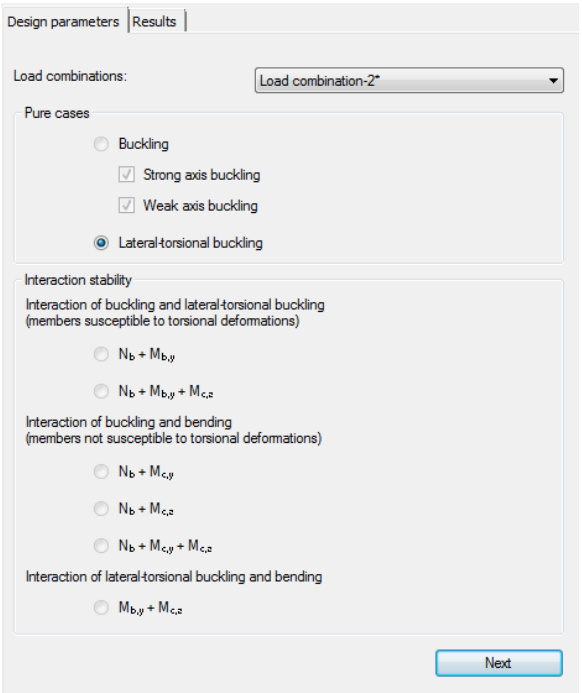
If a member is selected, the corresponding row highlighted with green and analysis results are automatically loaded. Member design can be run if there is analysis result (first and second order) and cross check for the member. Same like in the analysis the type of the result view can be set by three drop down menus.



Special analysis can be run by clicking on the second icon. Only one load combination can be selected at the same time. Buckling and lateral torsional buckling analysis cannot be performedfor all members. If the normal force is negligible then no buckling analysis can be run.

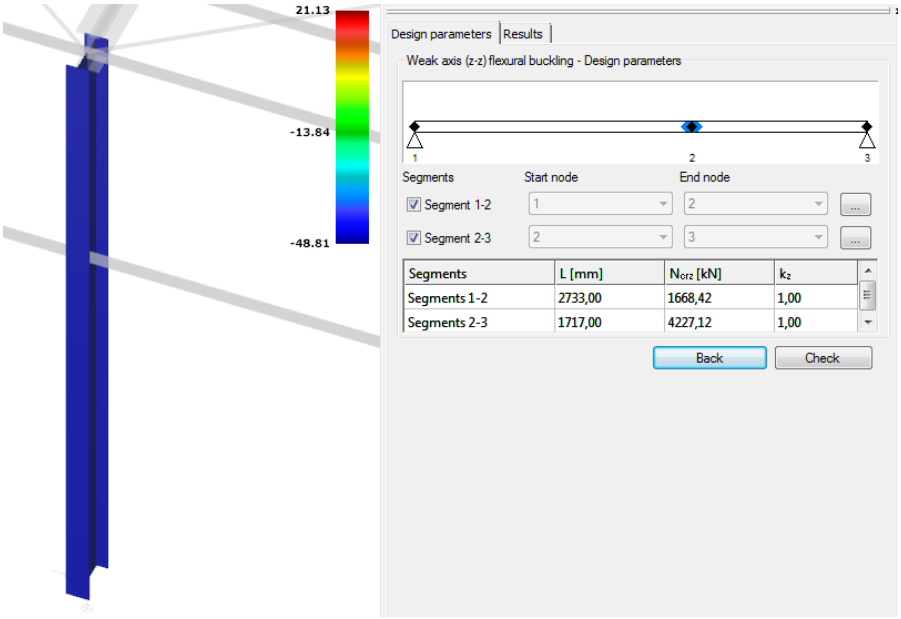


The next step is to choose the load combination and design type from the options: pure cases (strong axis buckling, weak axis buckling, lateral-torsional buckling) and interaction stability (interaction of buckling and lateral torsional buckling, interaction of buckling and bending, interaction of lateral-torsional buckling and bending). The dominant load combination is automatically selected and marked with a * symbol.

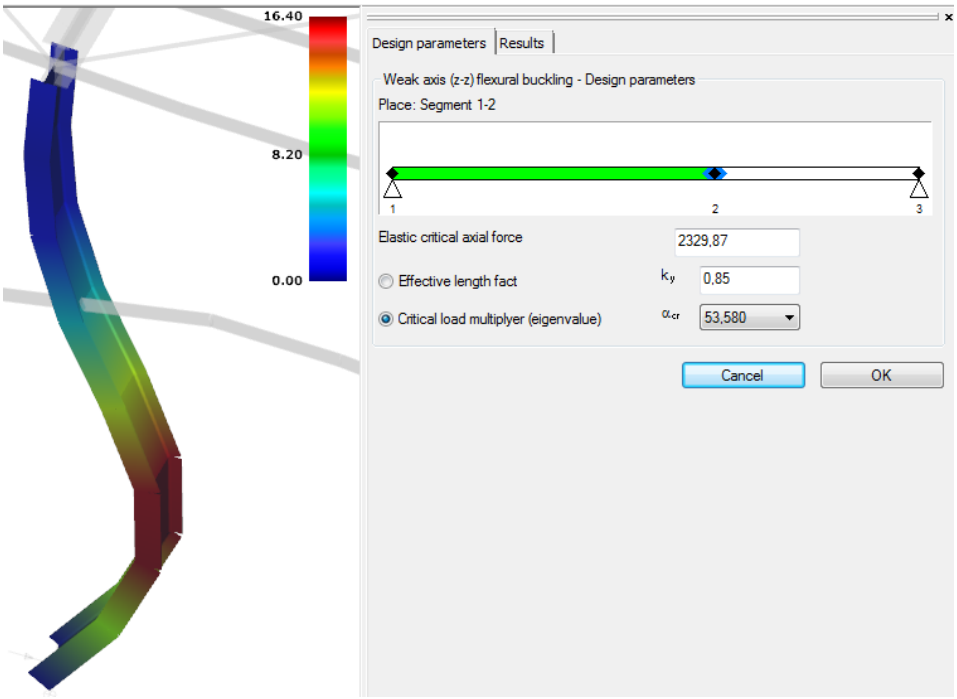


Depending on the member not all checks can be performed. If the normal force is negligible then strong or weak axis buckling design cannot be run.

If **strong or weak axis buckling option** is selected then first the design parameters has to be set. The program automatically identifies the supports which could have effect on the buckling check.

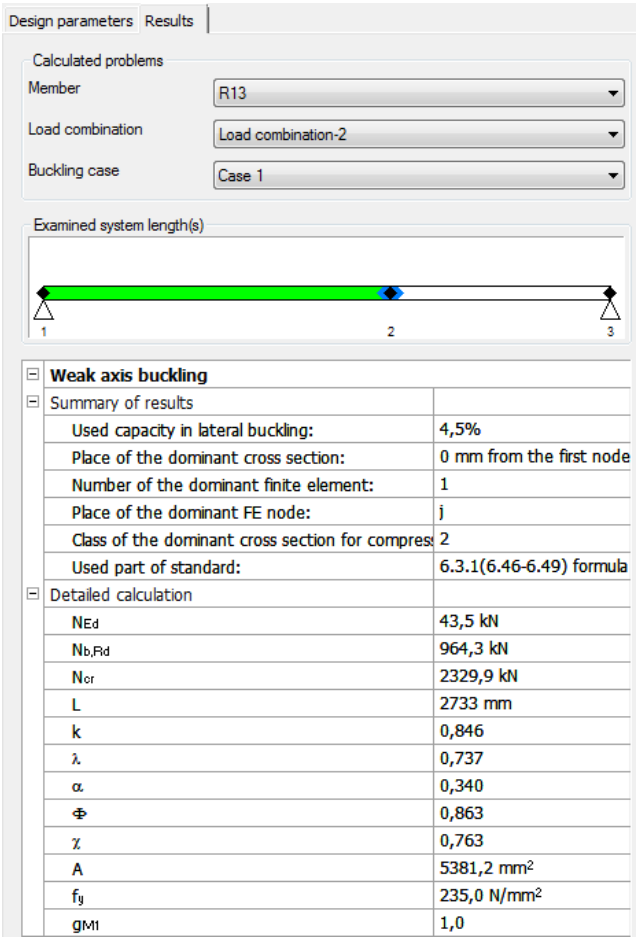


Among the supports the member is divided to segments. Segments can also be turned off in order not to calculate them during the analysis. The design parameters can be individually set for each segment by clicking on the three dots icon.



The actual segment is highlighted with green. There are two options to set the design parameters: set the effective length fact manually by typing in the appropriate value or by choosing the right critical load multiplier. The second option is possible if the special analysis has already been run. In both cases the elastic critical axial force is automatically calculated.

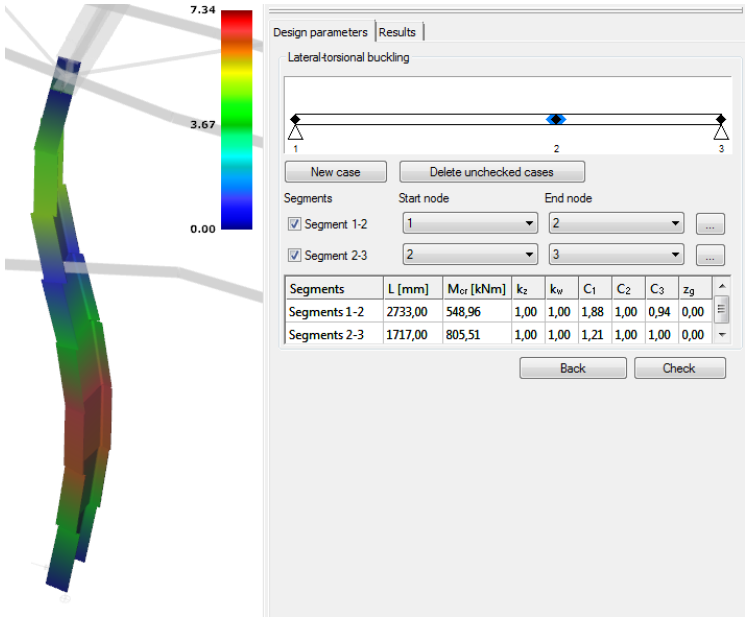
If all the design parameters has been properly set then design check can be performed by clicking on the Check button. Results will be displayed.



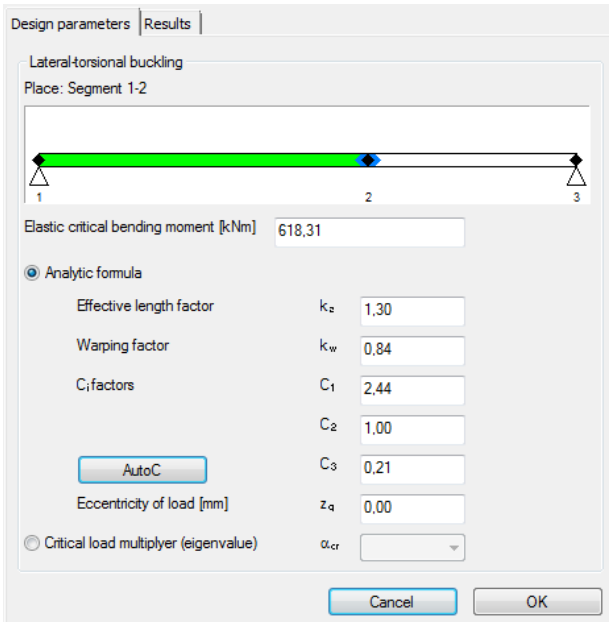
Weak axis buckling	
Summary of results	
Used capacity in lateral buckling:	4,5%
Place of the dominant cross section:	0 mm from the first node
Number of the dominant finite element:	1
Place of the dominant FE node:	j
Class of the dominant cross section for compres	2
Used part of standard:	6.3.1(6.46-6.49) formula
Detailed calculation	
N_{Ed}	43,5 kN
$N_{b,Rd}$	964,3 kN
N_{cr}	2329,9 kN
L	2733 mm
k	0,846
λ	0,737
α	0,340
Φ	0,863
χ	0,763
A	5381,2 mm^2
f_y	235,0 N/mm^2
γ_{M1}	1,0

There are three drop down menus in order to view different results. The first on can be used to change member, the second one is to change load combinations and the third one is to change the buckling case. The corresponding case is highlighted with green in the image.
A short explanation can be read in the bottom when any of the rows is selected.

Lateral torsional buckling design is more or less the same as the buckling design in ConSteel usage, there are only small differences. New cases can be defined and unnecessary cases can be deleted. For each segment start and end node can be set.



By clicking on the three dots icon the design parameters can be altered. The elastic critical bending moment can be calculated from the analytic formula or from the critical load multiplier. When the first method is used the factors can be typed in or a special automatic C factor calculator can be used by clicking on the AutoC button. The second option is to choose the corresponding critical load multiplier from the list.



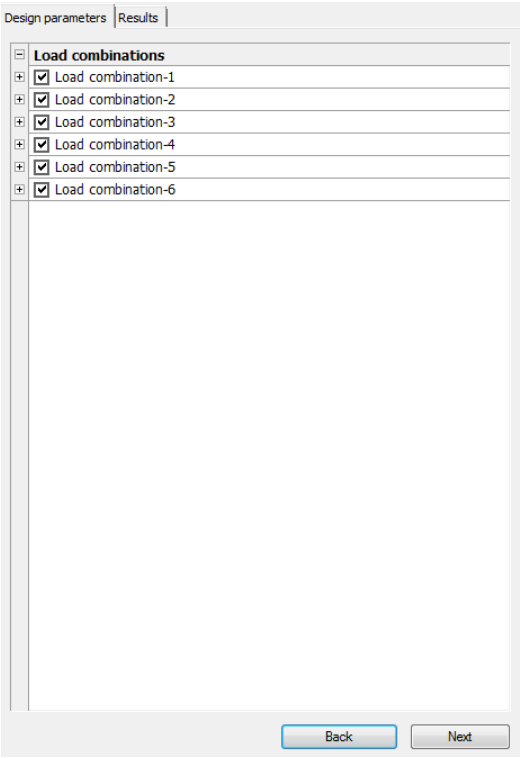
Interaction stability design goes through the steps of the pure cases.

After the check is made the state of the member in the list at the bottom changes to 'Examined' from 'Unexamined'.

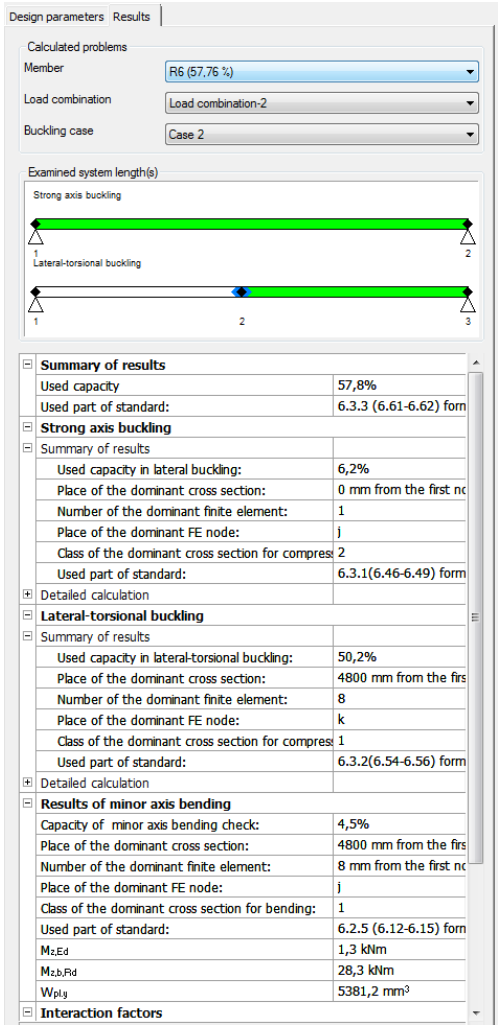
6.3 Member Group Design

By using the Select (+) button it is possible to select more members for the member design. Select (-) can be used for remove members from the selection. Member group design works nearly the same as individual member design there are just minor differences.
No special analysis can be run in member group design therefore the critical load factor cannot be selected.

If the members are selected then the next step is to select the design type and click Check button similar to the individual member design.



For each load combination member design can be turned off. It is important to know that not all design can be run for all members in load combinations. It depends on the analysis result.

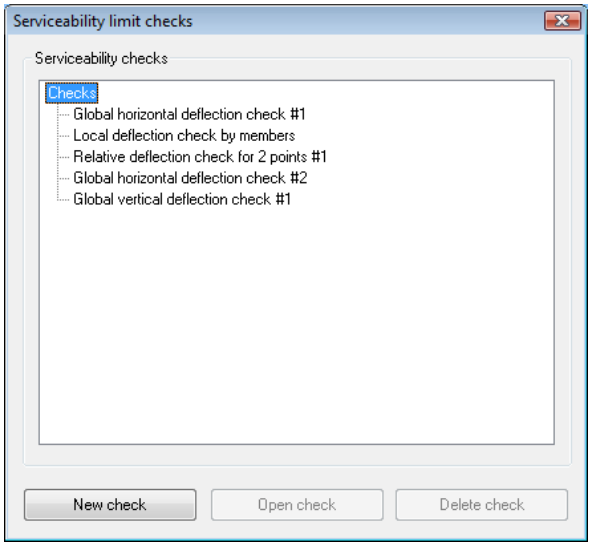


In the results tab the dominant member is automatically shown. The number next to the member number shows the capacity.

7. SERVICEABILITY

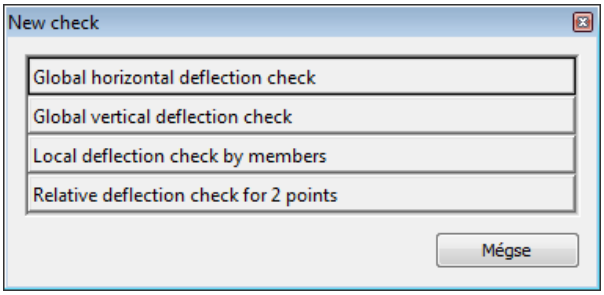
7.1 Basics

All serviceability related functions can be found on the ***SERVICEABILITY*** tab. By clicking on the blue arrow icon a new dialog appears. The previously defined serviceability checks can be found and it is also possible to open or delete the selected checks.



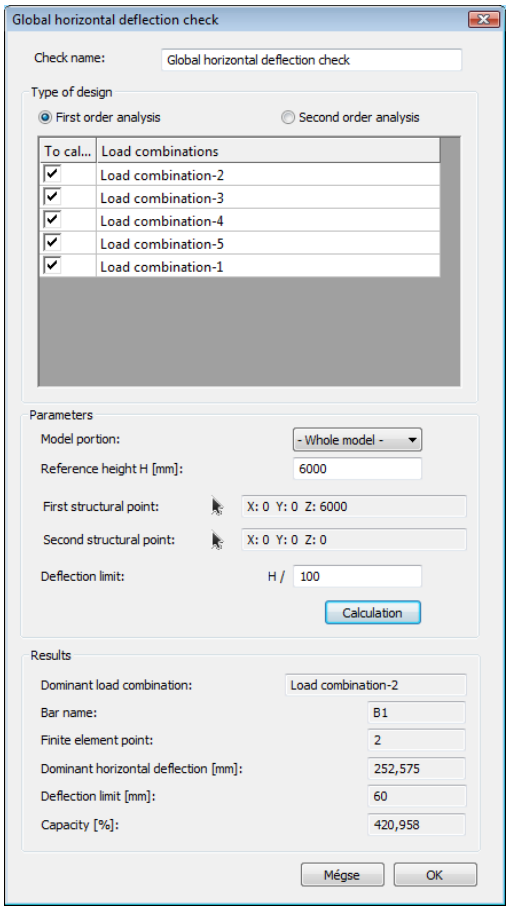
In order to create a new serviceability check ***NEW CHECK*** button is clicked. The following checks can be performed:

- Global horizontal deflection check
- Global vertical deflection check
- Local deflection check by members
- Relative deflection check for 2 points



7.2 Global horizontal deflection check

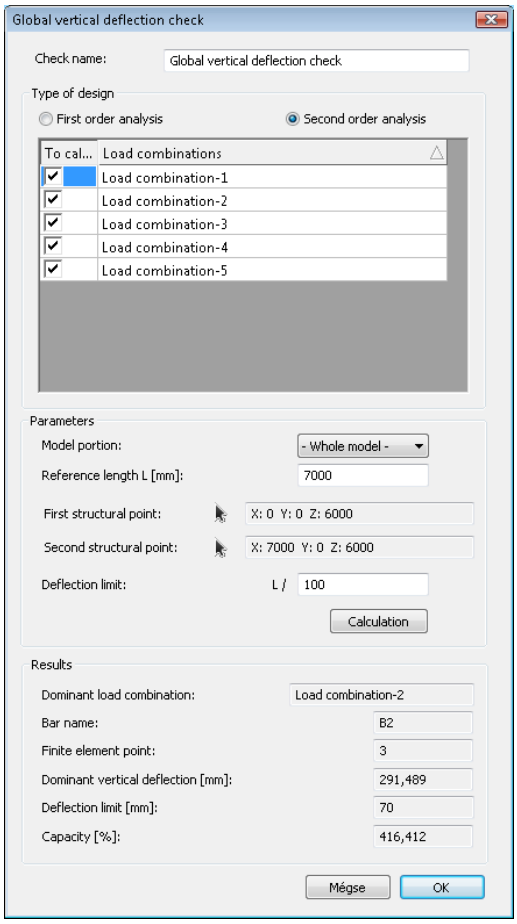
Global horizontal deflection check can be used to check the biggest horizontal deflection in the model or in the model portion. First or Second order analysis can be selected just like the serviceability load combinations that need to be taken into account.



Reference height can be typed in or two points can be selected and program calculates the height distance between them. Check is done after clicking the ***CALCULATION*** button. The dominant bar is automatically selected in the model to make the interpretation easier.

7.3 Global vertical deflection check

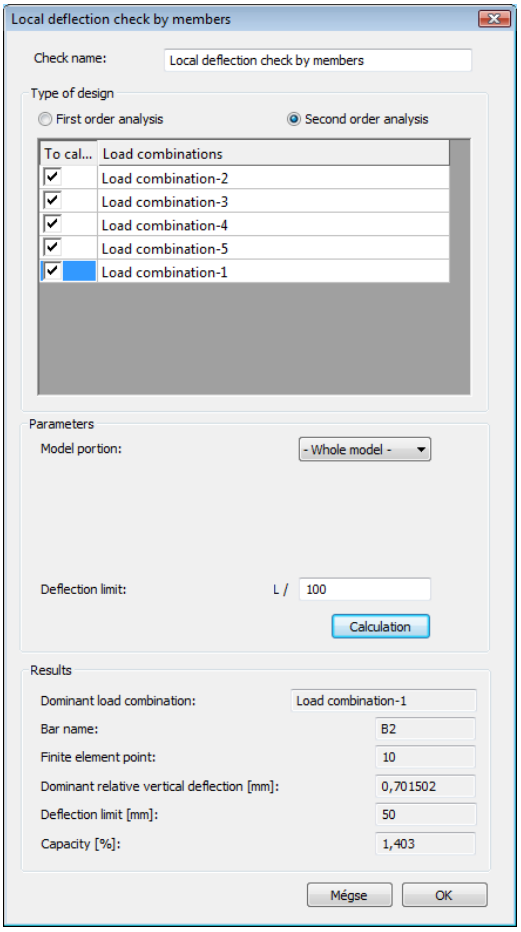
Global vertical deflection check works similar to the global horizontal deflection check. There is just minor difference which is the following: reference length has to be set instead of reference height. This can be set manually or by clicking the two end points of the bar.



7.4 Local deflection check by members

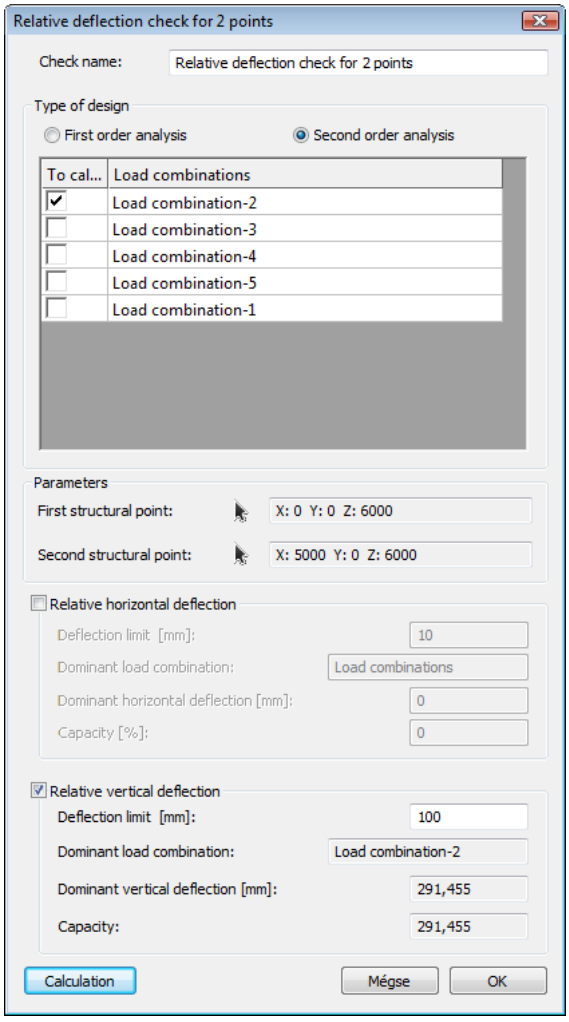
Local deflection check by members goes through all the beams in the model, and checks the local deflection, and then selects the maximum. This check is suitable only for horizontal beams i.e. members in YX plane.

Local deflection check by members is not adequate for cantilever beams just for beams which are supported by two other beams or supports in both ends.



7.5 Relative deflection check for 2 points

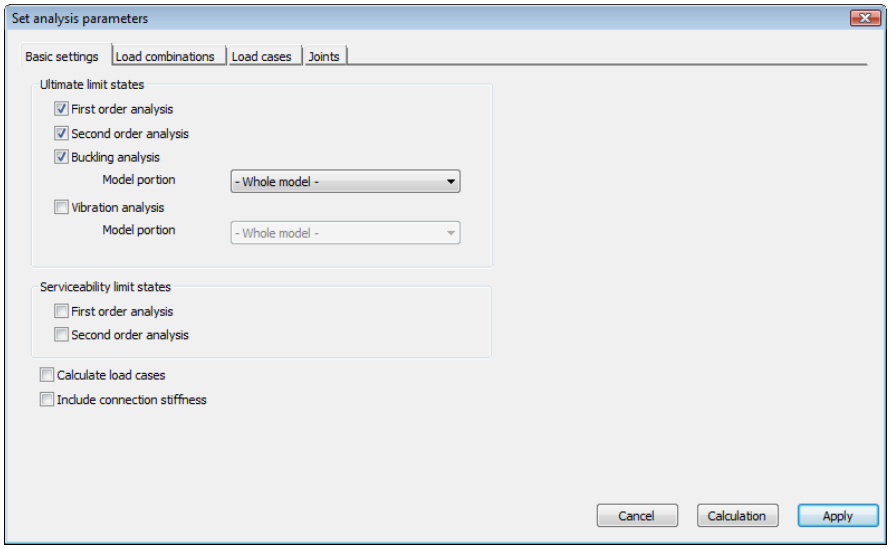
Relative deflection check for 2 points can be used for two clicked points. Relative horizontal or vertical check can be selected and performed.



8. NEW ANALYSIS AND DESIGN DIALOG WINDOWS

The analysis and design dialog windows have been redesigned in order to be more user-friendly and to provide some special settings

The required analysis types can be set on the ***ANALYSIS PARAMETERS*** panel. All types can be run for all the finite elements. Basically the analysis types can be defined for the existing load combinations.



By using the first tab it is possible to set analysis parameters for all load combinations at the same time. If buckling analysis is clicked 10 eigenvalues are calculated. The buckling analysis can be performed for the whole model or for a model portion. For each load combination unique settings can be set on the second tab. Load combinations can be turned off or type of analysis can be set.

The new design dialog window gives the possibility to calculate cross section check and buckling check also for model portions.

There are three additional new settings on the panel:

- elastic critical factor (first or selected)
- ultimate resistance factor (minimum by members, minimum by model portion)
- Reduction factor (minimum or interpolated)