WHAT'S NEW IN CONSTEEL 9



Version 9.0

18.03.2015

www.consteelsoftware.com

Content

1. World premiere functions 3
1.1 Frame corner wizard 3
1.2 Automatic selection of elastic critical factor for buckling check 6
2. Structural input 7
2.1 Easy creation of haunch 7
2.2 Multiple point support placement 7
2.3 Meteorological line load 8
2.4 Crane and moving load 8
2.5 Automatic meteorological load generation 9
3. Analysis 10
3.1 Plastic hinge analysis 10
3.2 Model portion consideration in second order analysis 10
3.3 Elastic analysis of influence graphs 11
3.4 Automatic amplitude calculation for imperfection 11
4. Standard design 12
4.1 New and more transparent design dialog 12
4.2 New functions on the Global checks tab 13
4.4 New Eurocode national annexes 13
5. Documentation 14
5.1 Fit page 14
5.2 Drag and drop function in documentation 14
5.3 Automatic standard design documentation 14
6. csJoint joint module 15
6.1 Joint model views 15
6.2 New joint placement 15

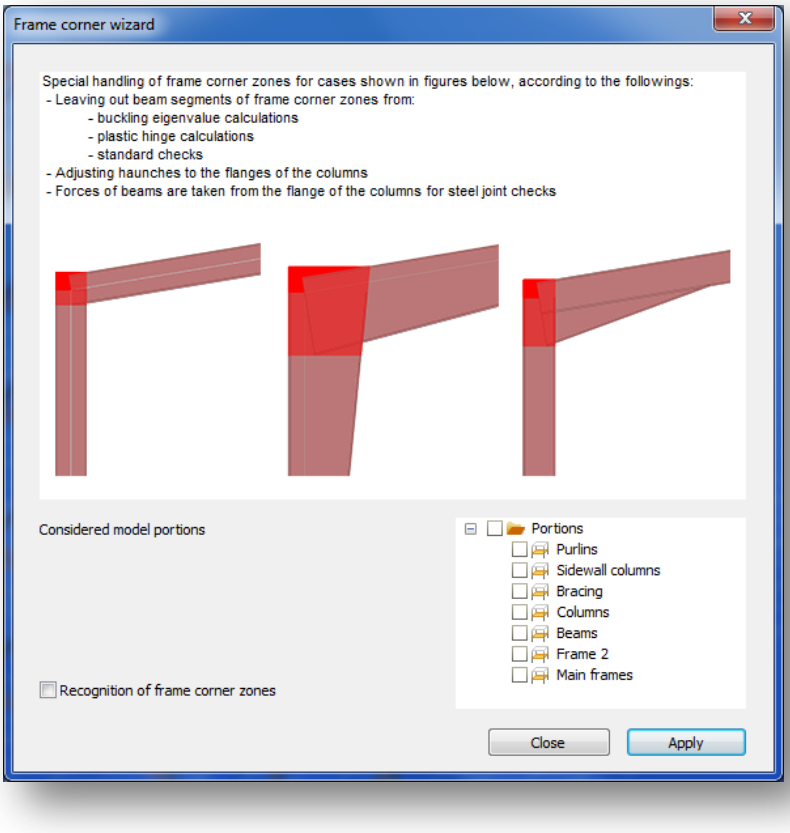
6.3 Prestressed bolts 15
6.4 Shear stub 16
6.5 New type of beam-to-column connection (coming soon)* 17
6.6 Pad foundation design (coming soon)* 17

1. WORLD PREMIERE FUNCTIONS

Two unique functions are implemented which have never been seen in other structural design software. Together these new functions can make the design process of steel structures even more realistic, safe and economic beside a significant increase in the working speed.

1.1 FRAME CORNER WIZARD

The ***FRAME CORNER WIZARD*** is an optional function for modelling and calculation of the corner regions of structural models more realistically considering the overlapping of connected members. The function is automatically identifies the corner zones and applies special treatment for the geometrical modelling, buckling analysis, plastic analysis and global checks. The main mechanical background is that these – generally stiffened – corner zones have usually significantly different behaviour than the connected beam members. Since these zones are very short the behaviour is influenced dominantly by the shear effect while the connected members have beam-column behaviour with dominant bending effect.



On the main dialogue two options are available:

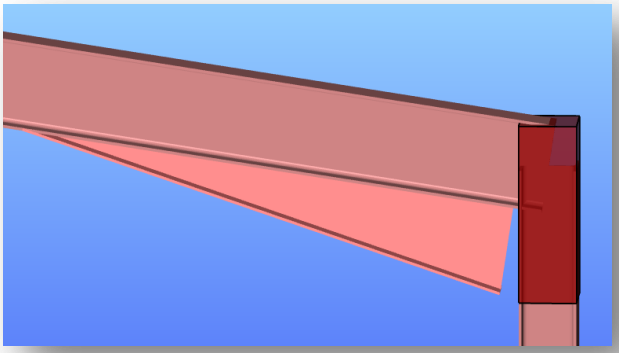
#1 – Turn on/off the automatic corner zone identification. The corner zones are defined around special connection nodes by the following rules:

- Applies to straight members with I or H cross-section
- Connection nodes are searched only between column (perfectly vertical) and beam members
- The local **y** axes of the column and beam members should be parallel (the webs of the connected members are in the same vertical plane)

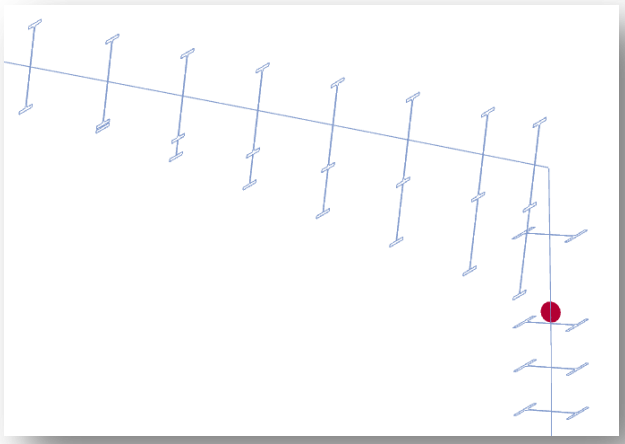
#2 – Set the suitable model portion(s) the ***FRAME CORNER WIZARD*** is applied for

After clicking on the ***APPLY*** button the program detects the overlapping zones of the members in the identified connection nodes including occurrent eccentricities, haunches or tapered members. The member segments inside the overlapping zones are separated and handled specifically:

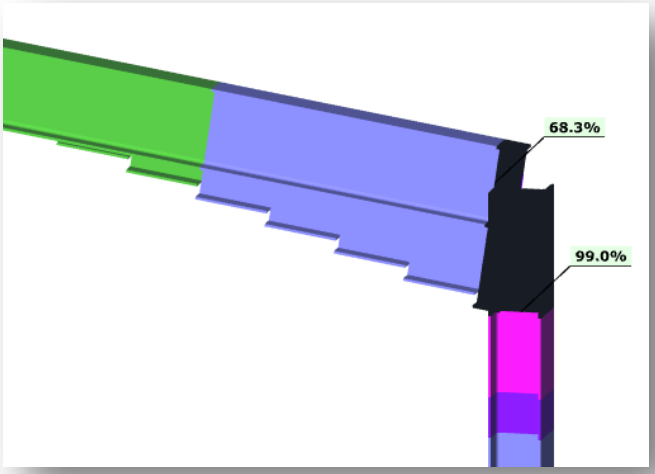
- *Geometry*: the haunch is started at the end of the truncated member (at the end of the overlapping member segment). In this way the real dimensions and position of the haunch is modelled.



- *Buckling analysis*: the finite elements belonging to the beam segments inside the overlapping nodes are left out from the buckling analysis (as it would be an unselected model portion). The reason is that in this zone member buckling cannot occur, and the effective buckling lengths of the members actually start outside of this zone.
- *Plastic analysis*: the plastic hinges cannot be formed inside the corner zone. Accordingly the plastic hinges are formed at the theoretically exact positions (for instance on the column below the haunch) and not in the connection node where the "virtual" bending moment is the highest.



- *Global checks*: the global checks (including section and stability checks) are performed only for the member parts outside of the corner zone. In this way the "virtual" peaks of the bending moments at the connection nodes inside the corner zones (which has no real mechanical meaning) are left out from the design checks and members are checked for the real, considerably bending moment (and the corresponding other internal force) values.



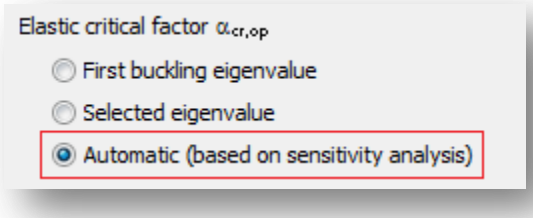
- *Connection design*: similarly to the earlier point the design bending moment and shear force values for the connection design are taken from the end of the truncated members where a beam-to-column connection is applied to the connection node where corner zone is defined.

1.2 AUTOMATIC SELECTION OF ELASTIC CRITICAL FACTOR FOR BUCKLING CHECK

In structural design it is always a basic problem to perform a realistic and safe stability design. The key questions are:

- Which are those critical members of the structural model where the stability design is important?
- Which buckling modes – among all the calculated modes – are the most relevant for these critical members?
- How to perform complete buckling design efficiently for these members?

In the 'Buckling sensitivity analysis' all the critical members are selected and assigned by a most relevant buckling mode in each load combination. In order to fully benefit of this information the elastic critical load factors (α_{cr}) corresponding to the relevant buckling modes of each member can be used in the global stability check based on the "General method" of EN 1993-1-1 6.4. in the calculation of the member slenderness value. Choosing the new 'Automatic (based on sensitivity analysis)' option on the design settings panel for the elastic critical load factor first the critical members are selected which may have stability problem the rest of the members are checked only for cross-section resistance.

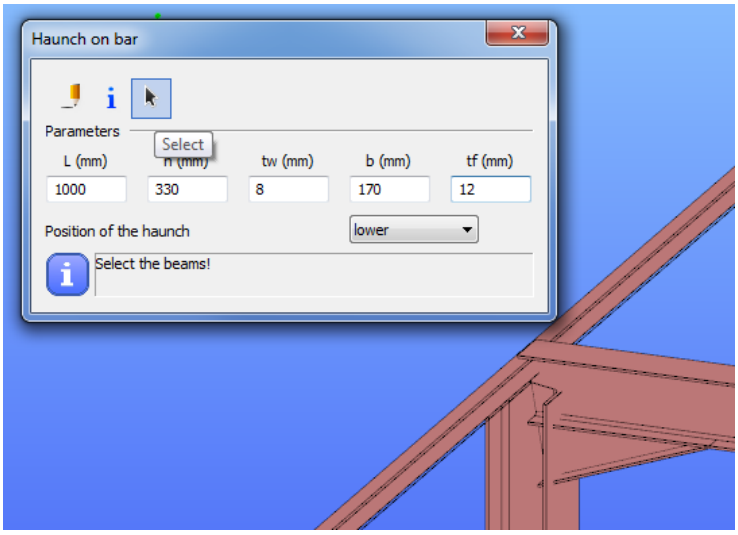


For the critical members the new function automatically calculates the most relevant slenderness value in each combination providing the most realistic base for stability design. Besides this automatic method selects highly safely the slenderness values based on the results of an advanced analysis of the structural model, this new function can extremely speed up the whole design process of large complicated models.

2. STRUCTURAL INPUT

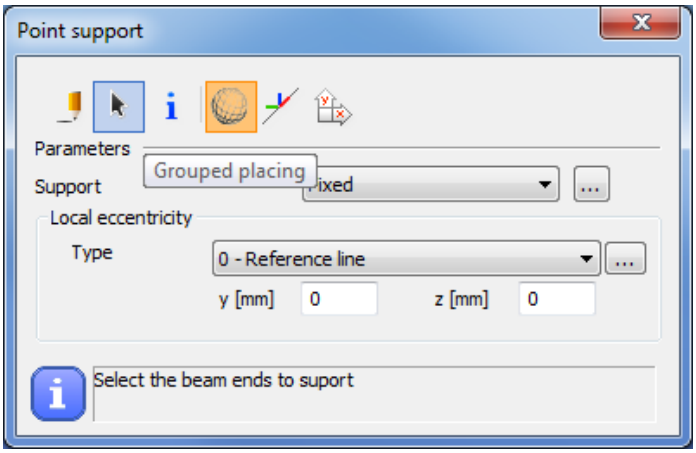
2.1 EASY CREATION OF HAUNCH

With the new black arrow button, the parameters (plate thickness, width) of the selected member can be used to create a new haunch.



2.2 MULTIPLE POINT SUPPORT PLACEMENT

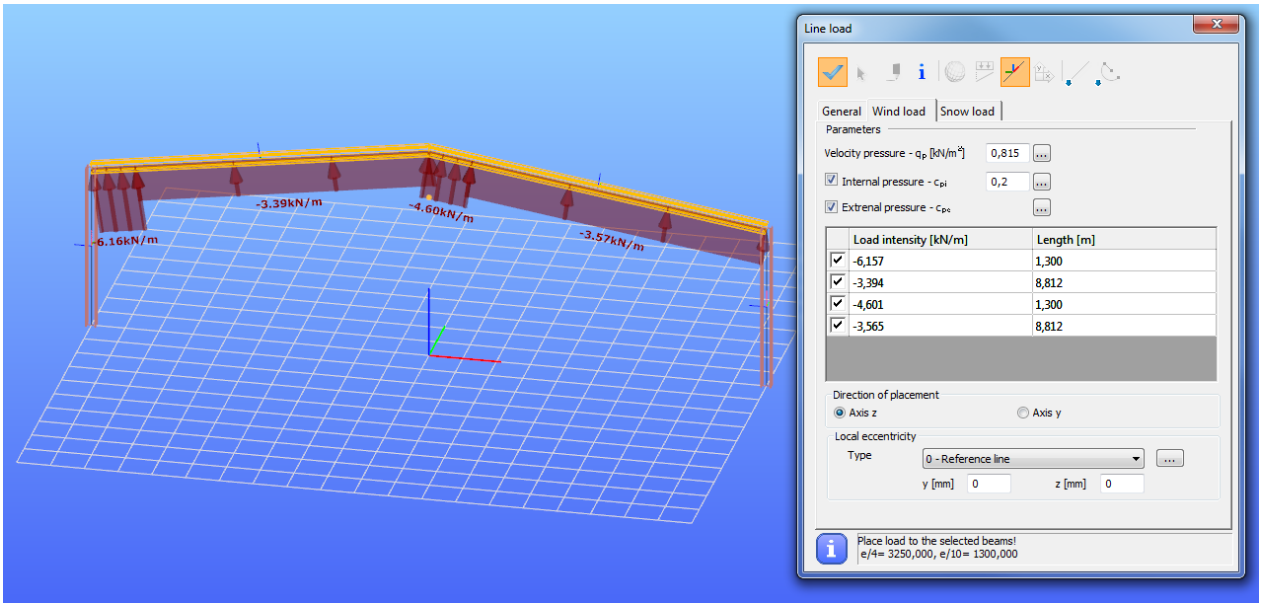
With the new black arrow button, multiple point support can be placed with window selection.



2.3 METEOROLOGICAL LINE LOAD

Existing Wind line load was further developed and redesigned and the line load function was completed with Snow line load.

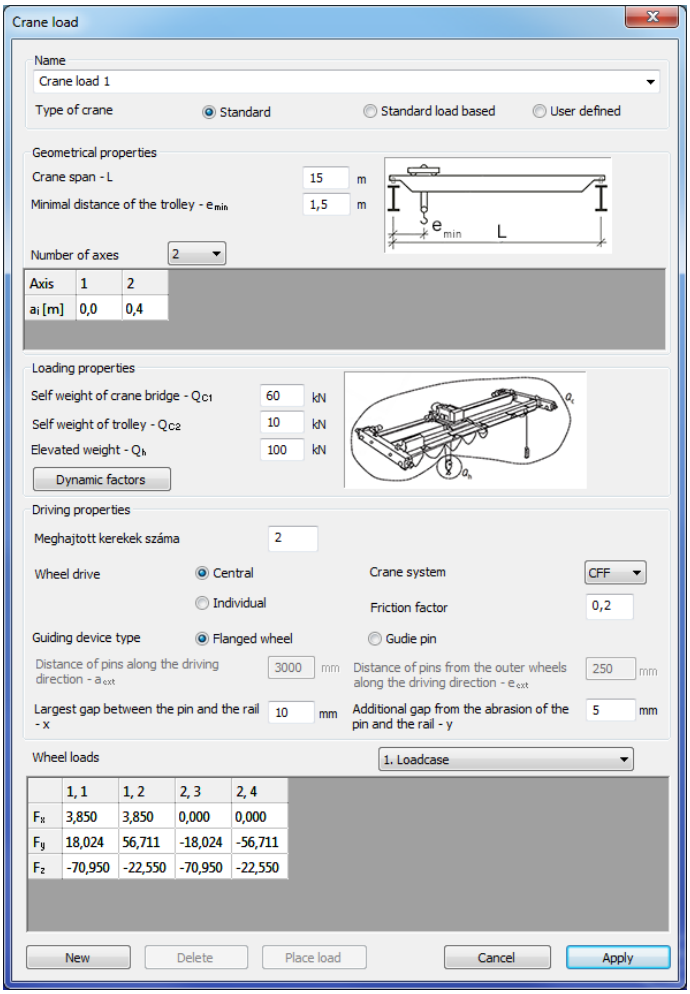
With these new functions, the meteorological loads can be easily and quickly calculated and placed as a line load.



2.4 CRANE AND MOVING LOAD

Eurocode or user defined crane () and moving load () can be placed manually on the model or by loading a previously analysed influence graph.

New function automatically calculates the proper wheel loads and creates the necessary crane load groups. In case of influence graph loading, the dominant crane load group and the positions of the trolley are automatically identified.

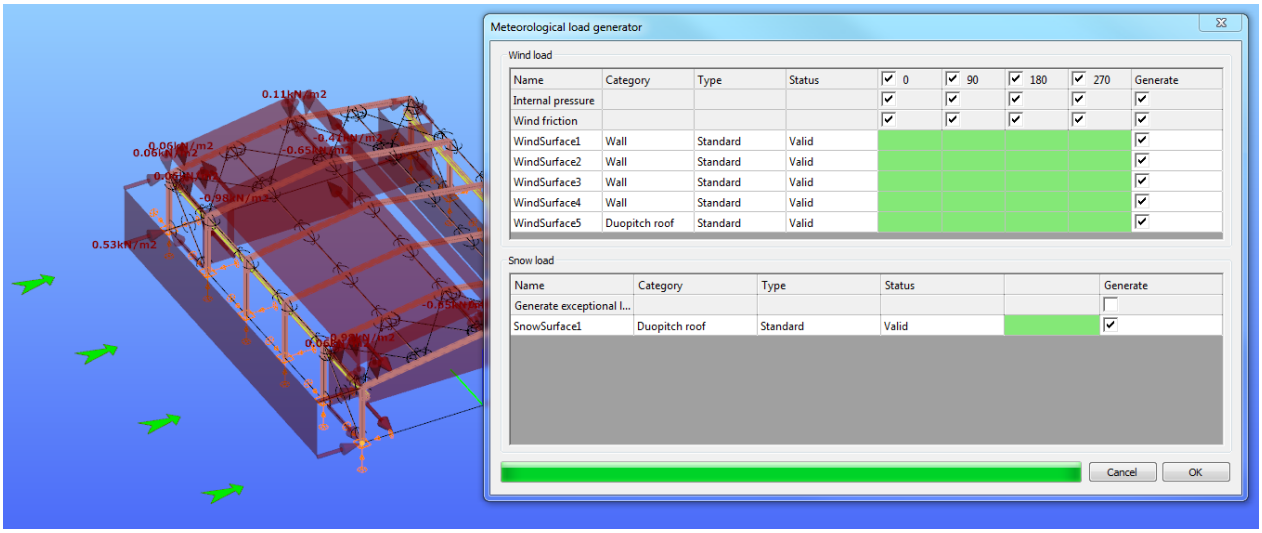


2.5 AUTOMATIC METEOROLOGICAL LOAD GENERATION

ConSteel 9 contains a full Eurocode based automatic meteorological load generation (), including the snow load and wind friction beside the existing wind load generation.

Thanks for the new functionalities the meteorological load generation automatically creates the necessary load cases depend on the shape of the structure, generate all loadings even the wind friction in the selected wind directions.

And from now not just one internal pressure can set for each of wind directions, but alternative pressure can be set also.

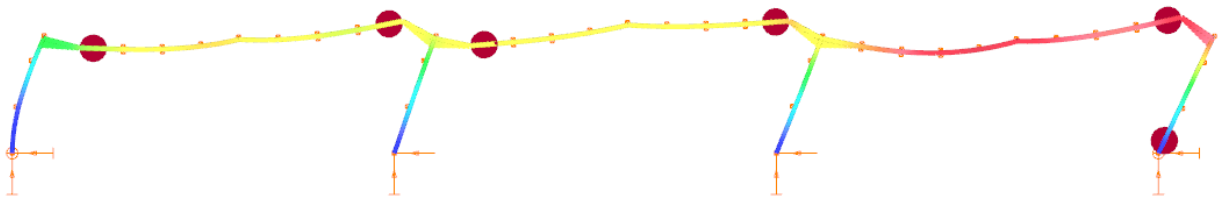


3. ANALYSIS

3.1 PLASTIC HINGE ANALYSIS

As a new analysis type, plastic hinge analysis is available in ConSteel 9.

Plastic analysis can be performed not just for the first and second order analysis, but for the buckling analysis also.



3.2 MODEL PORTION CONSIDERATION IN SECOND ORDER ANALYSIS

Thanks for the new feature, second order analysis can be performed not just for the whole model, but for a partial model also. With this selection unstable parts of the model can be taken out which would block the second order analysis.

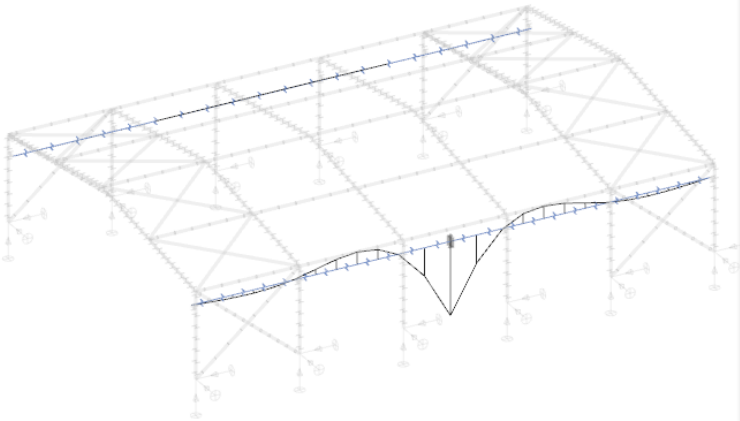
3.3 ELASTIC ANALYSIS OF INFLUENCE GRAPHS

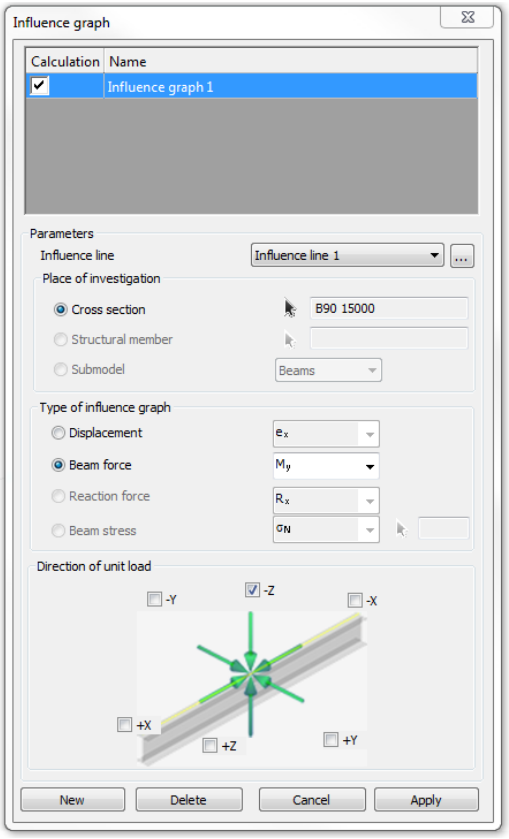
Elastic analysis of influence graphs can be performed for a selected influence line.

Three types of influence graph are available in ConSteel 9:

- Displacement
- Internal force
- Reaction force

Influence graph can be automatically loaded by moving loads.





3.4 AUTOMATIC AMPLITUDE CALCULATION FOR IMPERFECTION

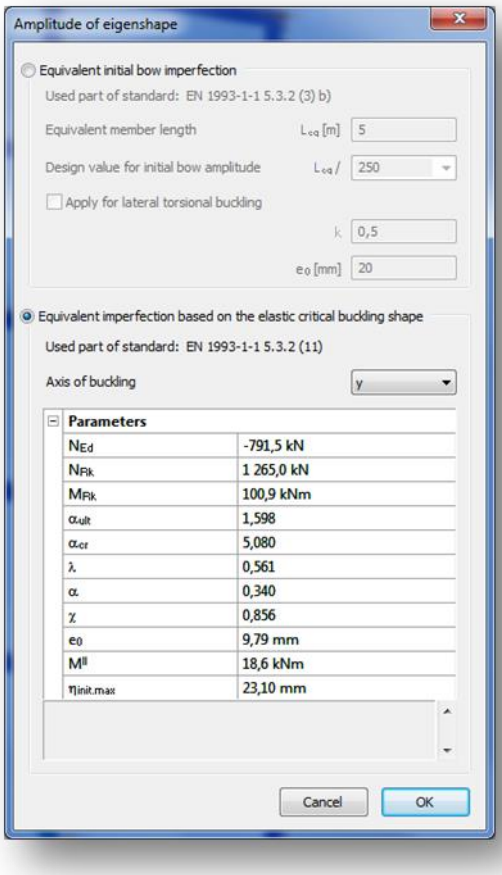
Thanks for the new functionality there are two options to calculate automatically the amplitude for imperfection:

- Equivalent initial bow imperfection

This option is based on the table EN 1993-1-1 5.3.2 (3)b where initial bow amplitude ratios are defined depending on the corresponding buckling curve of the selected member.

- Equivalent imperfection based on the elastic critical buckling shape

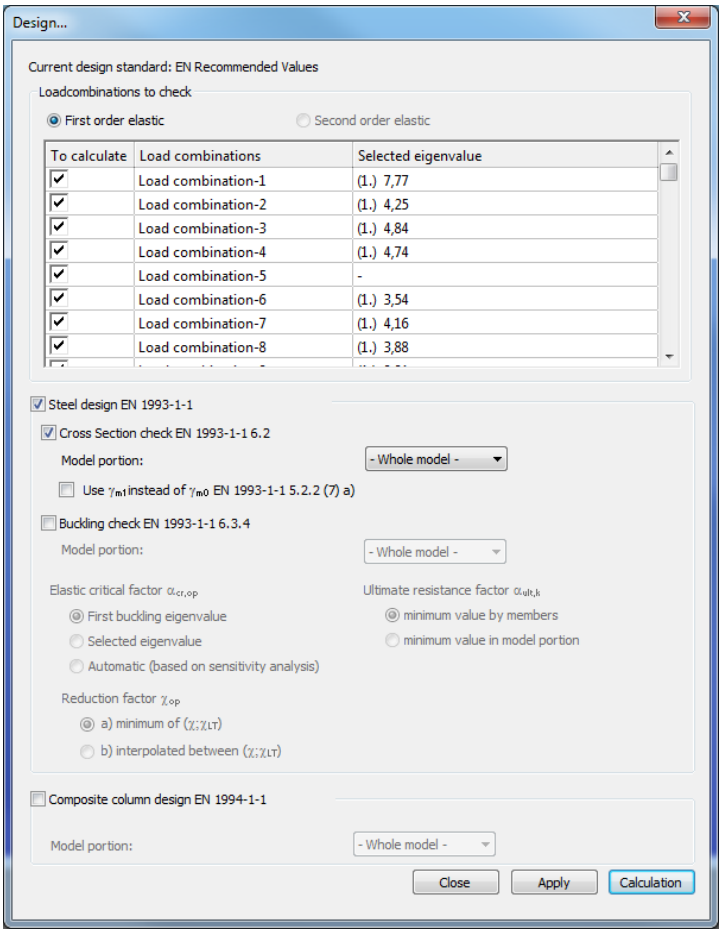
This option is based on the rules of EN 1993-1-1 5.3.2 (11). This method calculates fully automatically the amplitude value based on the selected axis of buckling.



4. STANDARD DESIGN

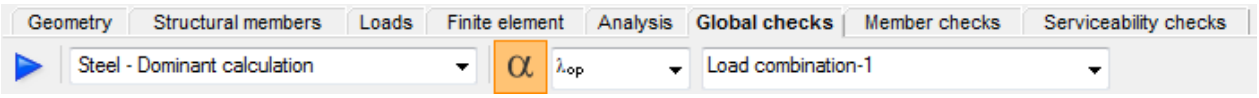
4.1 NEW AND MORE TRANSPARENT DESIGN DIALOG

On the new dialog the different design options are much more transparent and manageable. Settings of the different design checks are ordered in separated groups.



4.2 NEW FUNCTIONS ON THE GLOBAL CHECKS TAB

With the new functionality () the used α_{cr}, α_{ult} and λ_{op} parameters can be checked by graphical visualisation and table for each members.



4.4 NEW EUROCODE NATIONAL ANNEXES

Two more national annexes can be used to standard design:

- Greece national annex
- Bulgarian national annex

5. DOCUMENTATION

5.1 FIT PAGE

In the documentation module, as a new function the fitting of the documentation can be selected:

- Fit page width 
- Fit page height 

5.2 DRAG AND DROP FUNCTION IN DOCUMENTATION

Content of the documentation can be moved with drag and drop function in the documentation tree.

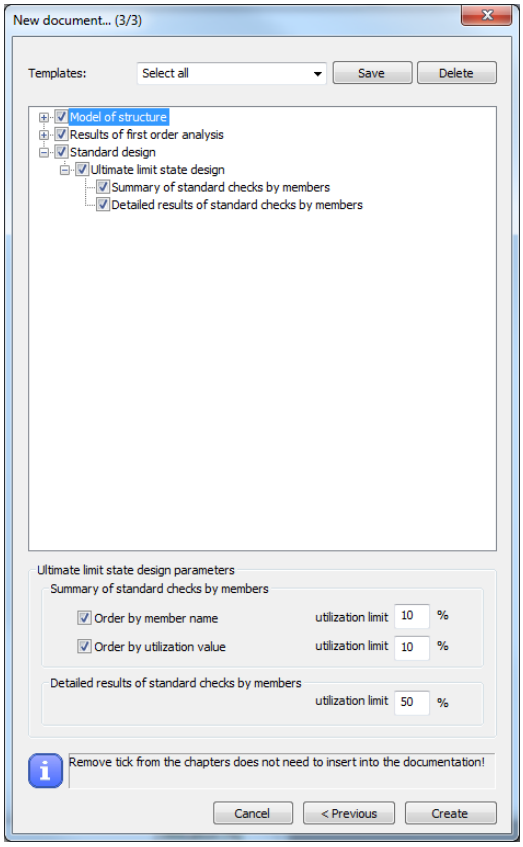
5.3 AUTOMATIC STANDARD DESIGN DOCUMENTATION

Thanks for the new function the results of the standard design can be added to the documentation automatically.

Two types of documentation can be added:

- Summary by members in table format
- Detailed design by members

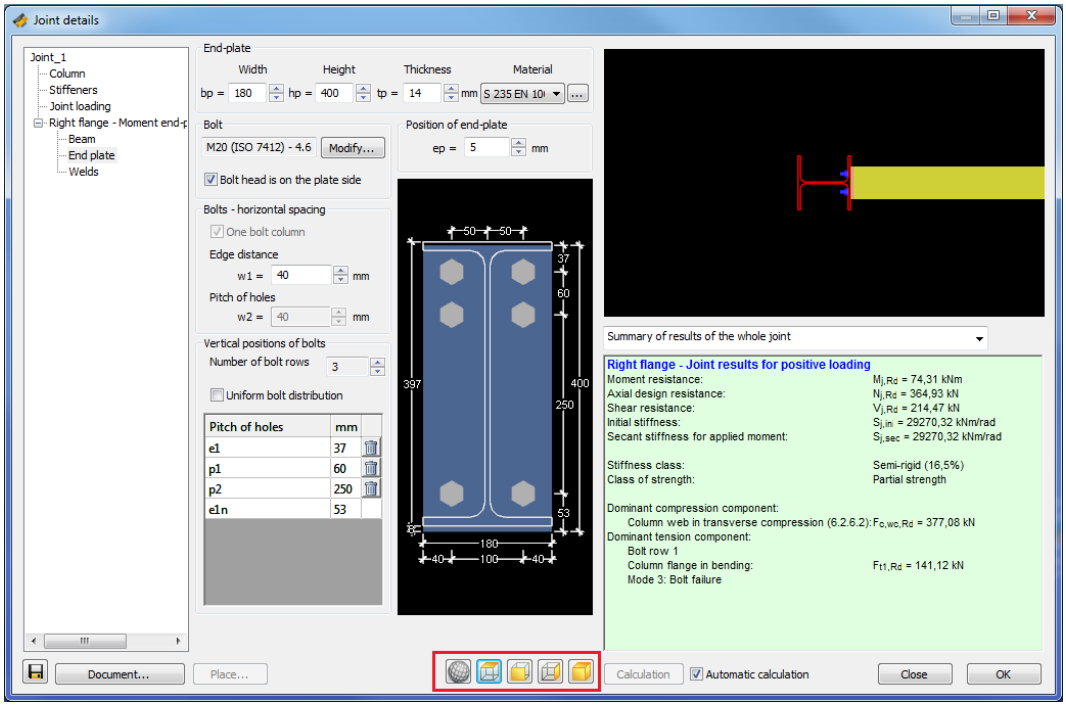
Members can be ordered by member name or utilization and a minimum utilization limit can be set.



6. CSJOINT JOINT MODULE

6.1 JOINT MODEL VIEWS

Thanks for the new features the 3D model of the joint can be viewed easily from the top, front, side or perspective view.



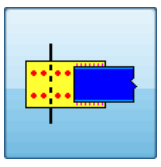
6.2 NEW JOINT PLACEMENT

With the new ***PLACE…*** button, joint can be placed to the 3D model directly from the Joint details dialog, during the joint creation. No longer need to create the joint first and close the Joint details dialog and place the joint to the model with the Place joint function.

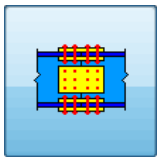
6.3 PRESTRESSED BOLTS

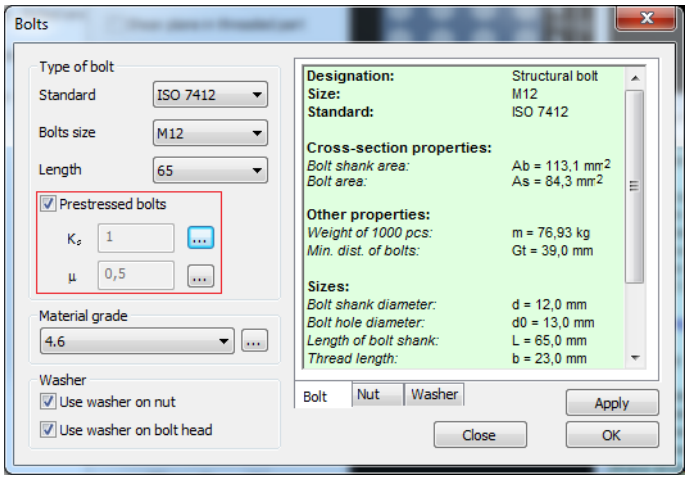
Prestressed bolts can be used in the following connection types:

- Bracing splice plate connections



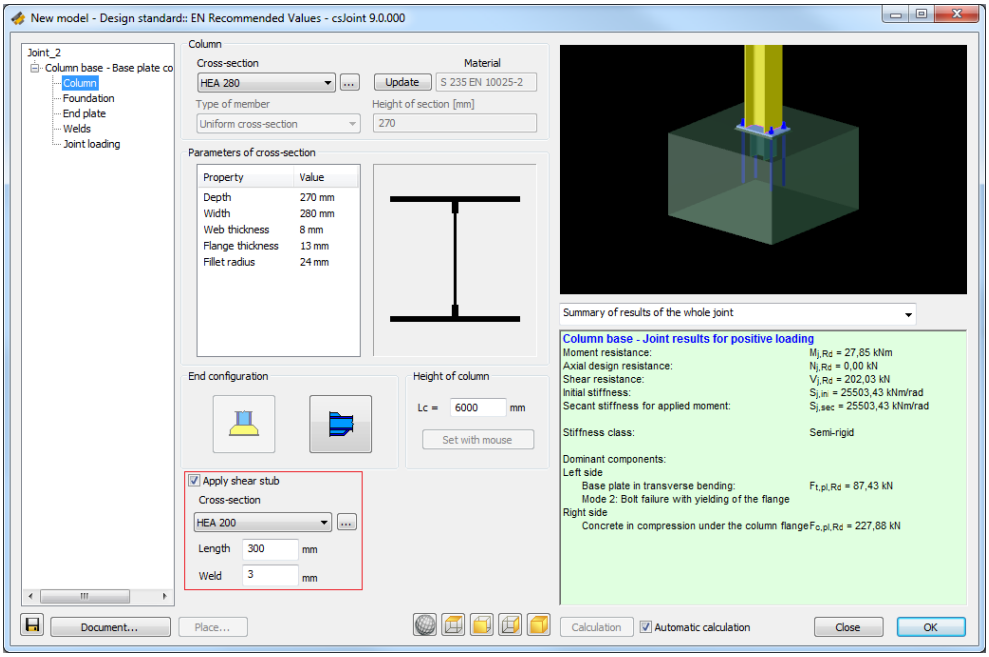
- Beam splice plate connections





6.4 SHEAR STUB

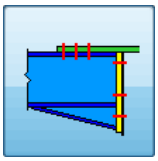
As a new option of the column base joint, shear stub can be applied to the base plate connection to increase the shear capacity of the connection.

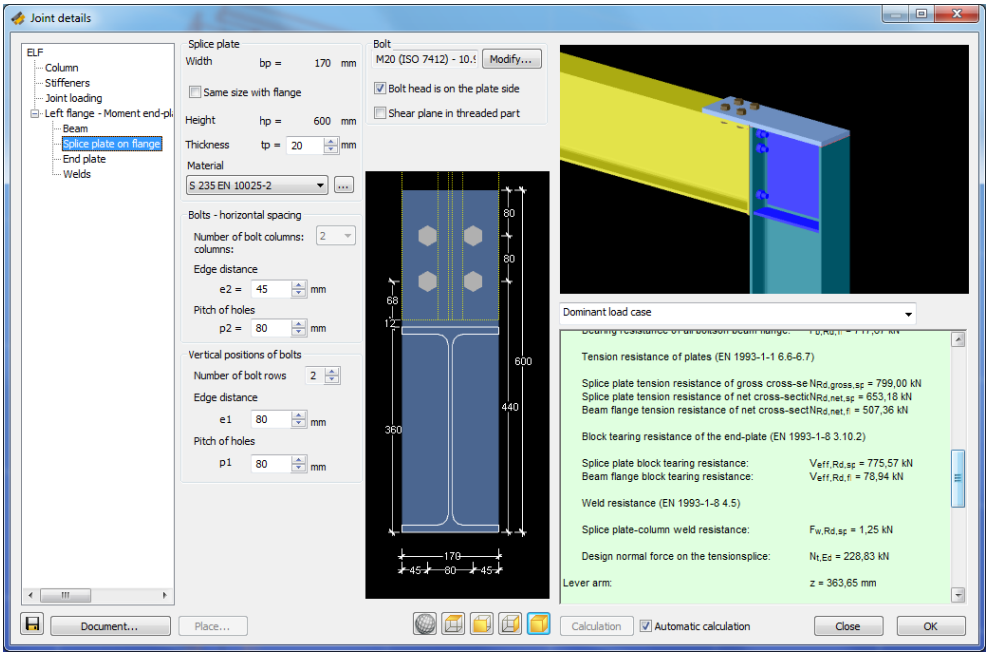


6.5 NEW TYPE OF BEAM-TO-COLUMN CONNECTION (COMING SOON)*

New type of beam-to-column moment end-plate connection can be created and checked:

- Moment end-plate connection with bolted tension splice

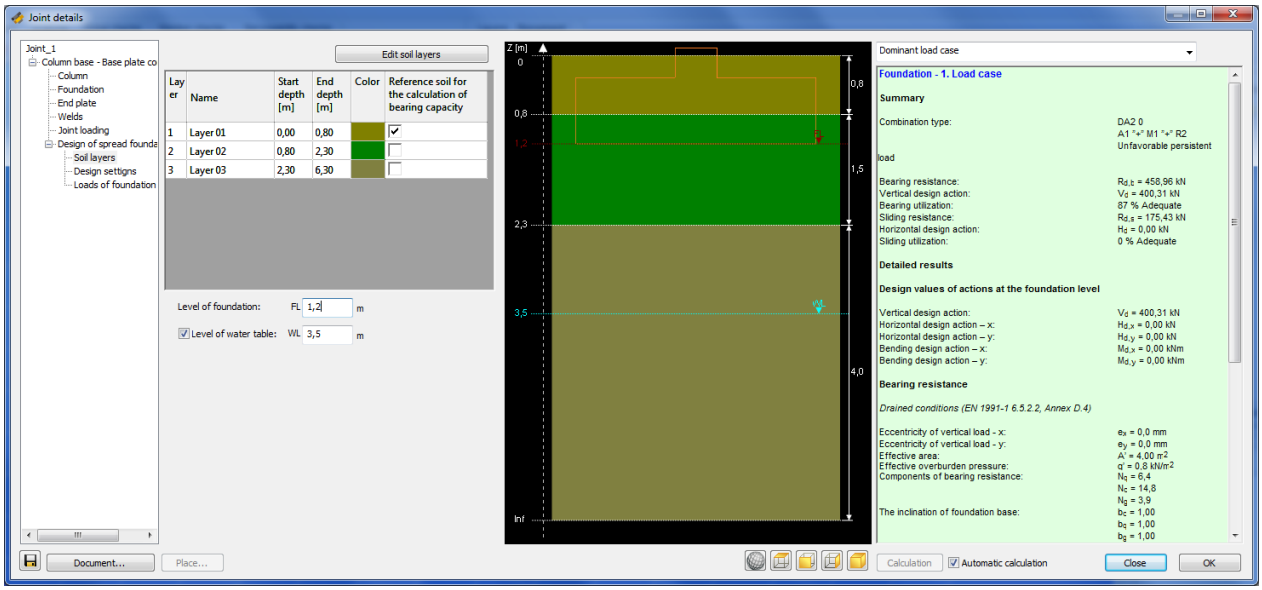




6.6 PAD FOUNDATION DESIGN (COMING SOON)*

As a new function, besides the check of the base plate connection, pad foundation design can be carried out. Several numbers of soil layers and the level of the ground water can be set.

Foundation loads can be automatically loaded from the global model if the column base joint is placed and the necessary analysis are calculated. The bearing and sliding resistances and the settlements are checked according to the selected Design Approach (DA).



** marked functions will be released at the end of March.*