This document gives a detailed summary of the new features and modifications of FEM-Design version 14. We hope you will enjoy using the program and its new tools and possibilities. We wish you success.

*Strusoft, the developers*

---

**Legend**

Pay attention / Note

Useful hint

Example

Clicking left mouse button

Clicking right mouse button

Clicking middle mouse button

## What's new in FEM-Design 14

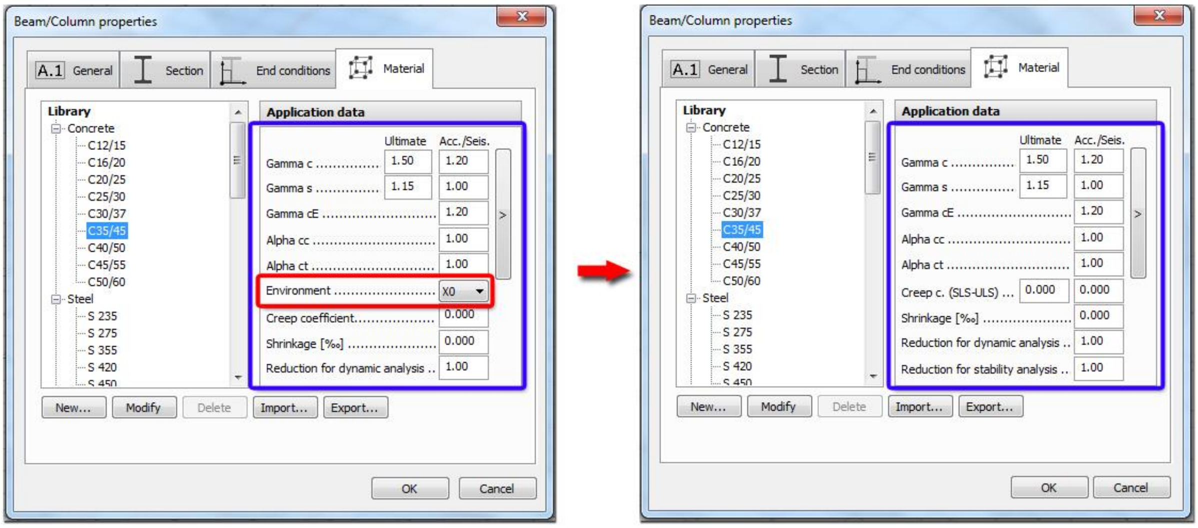
- Full 64-bit version (larger models can be calculated)
- List tables
  - for selected objects
  - using User-defined batches
  - to Excel by User-defined templates
- Single layer reinforcement for shells
- Fictitious shell
- Detailed result for line support group reaction and connection forces
- Local stability analysis
- Improved user interface
- Improved graphical performance

# Table of contents

# 1. STRUCTURE

## 1.1. Material properties

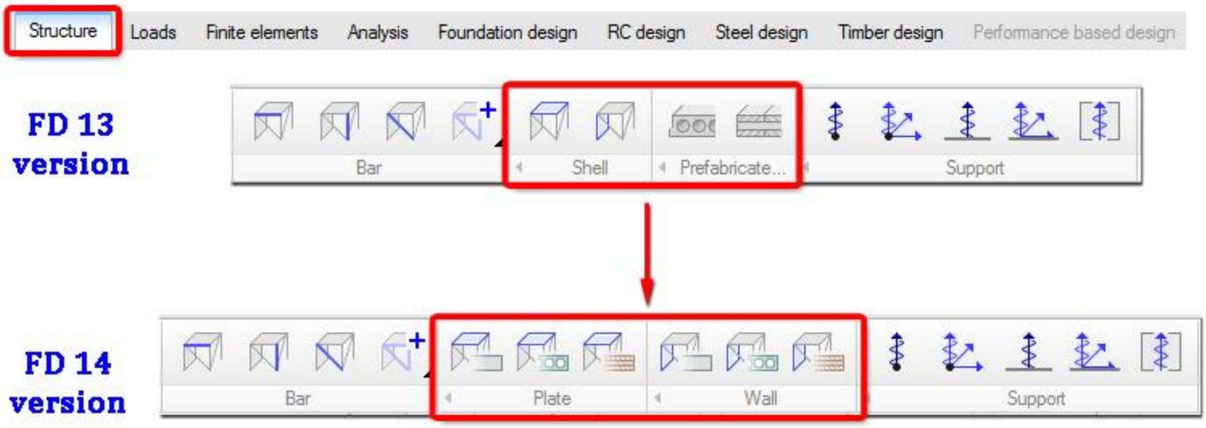
In FEM-Design 14 the Environmental class is not part of the Concrete/Application data dialog because it has no role any more in the calculations.




## 1.2. Shell objects

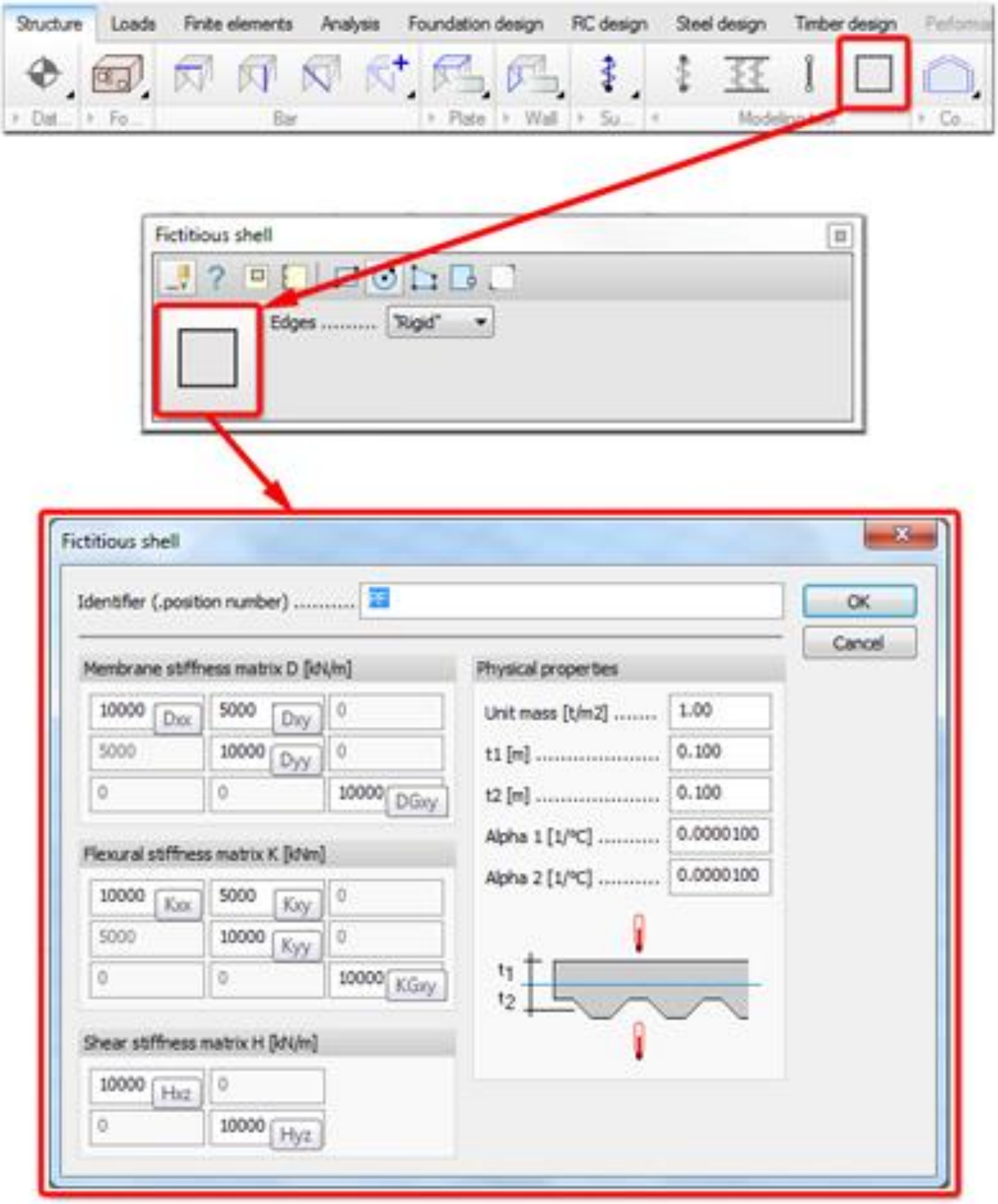
In *Structure* tab the Shell objects are separated both for in-situ (*Plate* and *Wall*) and prefabricated (*Profiled panel* and *Timber panel*) elements.

With this reorganization it is easier to choose the required element type.



## 1.3. Fictitious shell

A new object, the Fictitious shell is added to the Structure tab.



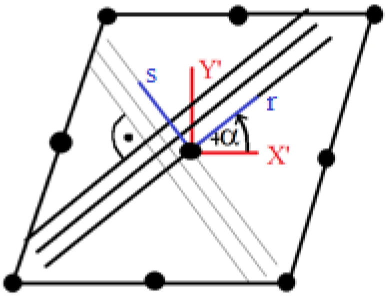

In the tool-window the type of the edges (Rigid or Hinged) can be chosen.

In the Default settings dialog the User can define the stiffness matrices (Membrane stiffness matrix, Flexural stiffness matrix and Shear stiffness matrix) and physical properties (Unit mass, distances of the planes where thermal load acts from the center plane, coefficient of thermal expansion).

Fictitious shell is ideal to model a section with ***composite material***, where the designer may calculate appropriate stiffness and fill the D, K and H stiffness values.

Also Fictitious shell is a powerful tool when calculating a section with ***complex geometry*** where the parameters are specified by the manufacturer of the structural element.

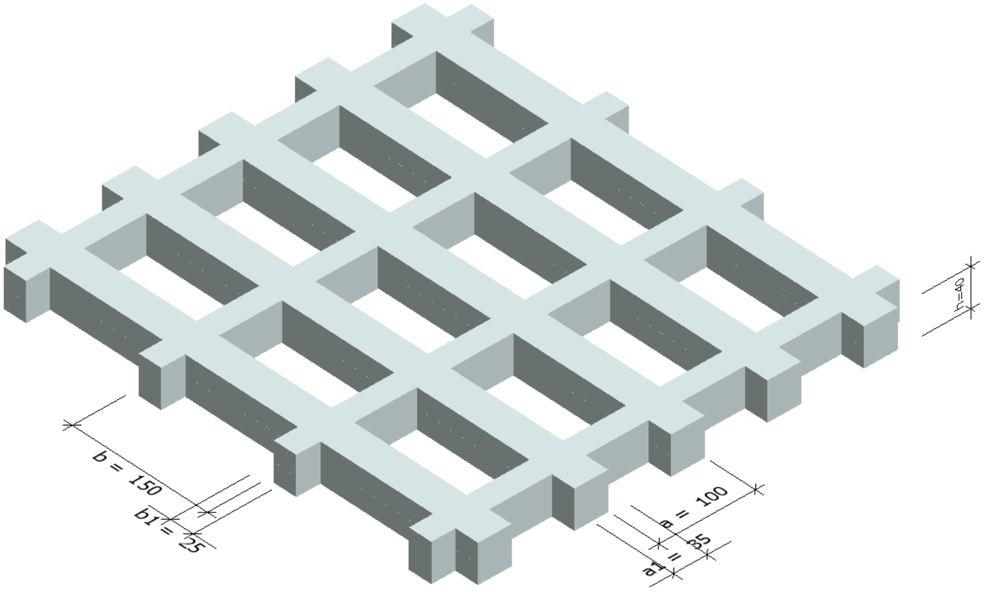
Stiffness of orthotropic shell element for fictitious shell can be calculated as follow:




**D** **K** **H**

$$\begin{bmatrix} \frac{E_r h}{1-\nu_{rs}\nu_{sr}} & \frac{\nu_{sr} E_r h}{1-\nu_{rs}\nu_{sr}} & 0 & 0 & 0 & 0 & 0 & 0 \\ \frac{\nu_{rs} E_s h}{1-\nu_{rs}\nu_{sr}} & \frac{E_s h}{1-\nu_{rs}\nu_{sr}} & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & G_{rs} h & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{E_r h^3}{12(1-\nu_{rs}\nu_{sr})} & \frac{\nu_{sr} E_r h^3}{12(1-\nu_{rs}\nu_{sr})} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{\nu_{rs} E_s h^3}{12(1-\nu_{rs}\nu_{sr})} & \frac{E_s h^3}{12(1-\nu_{rs}\nu_{sr})} & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{G_{rs} h^3}{12} & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & \frac{G_{rt} h}{1.2} & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & \frac{G_{st} h}{1.2} \end{bmatrix}$$

Numerical example below will show how to fill-up the stiffness matrix.




***Input Data***

E=30GPa

v=0.2

G=12.5GPa

h=40cm

a1=25cm

b1=35cm

a=1.5m

b=1.0m

***Membrane Stiffness***

| | | |
|---|---|---|
| Dxx= | E*h*a1/a= | 2000000 kN/m |
| Dxy= | Here lateral effect is negligible and assumed to be zero | 0 kN/m |
| Dyy= | E*h*b1/b= | 4200000 kN/m |
| DGxy= | G*h*(1-(a-a1)*(b-b1)/(a*b))= | 2291667 kN/m |

***Bending Stiffness***

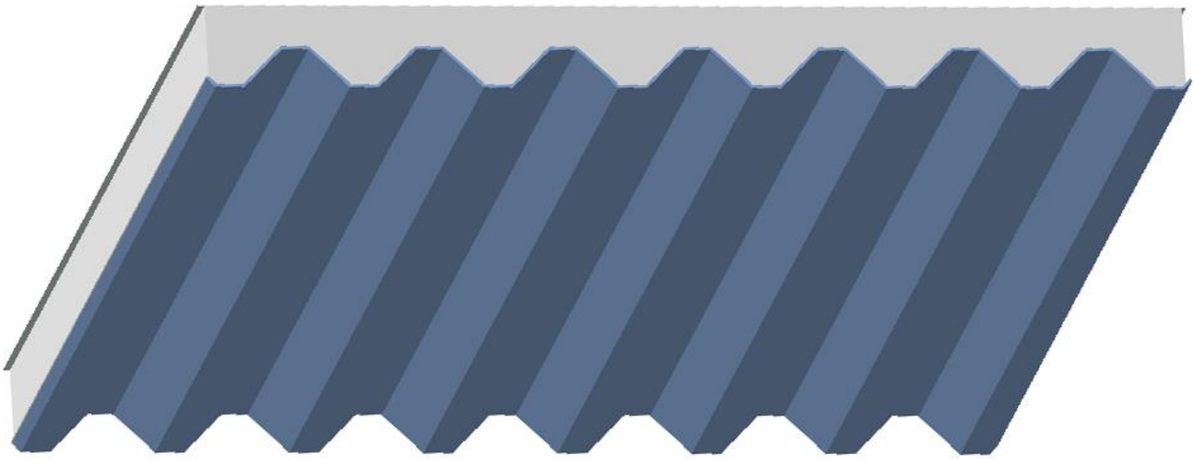
| | | |
|---|---|---|
| Kxx= | E*h*h*h/12*a1/a= | 26667 kNm^2/m |
| Kxy= | v*E*h*h*h/12*(a1*b1)/(a*b)= | 1867 kNm^2/m |
| Kyy= | E*h*h*h/12*b1/b= | 56000 kNm^2/m |
| KGxy= | E*h*h*h/12*(1-v*v*a1*b1/(a*b))/(a*b*(a/(a1*a1*a1)+b/(b1*b1*b1)))= | 892 kNm^2/m |

***Shear Stiffness***

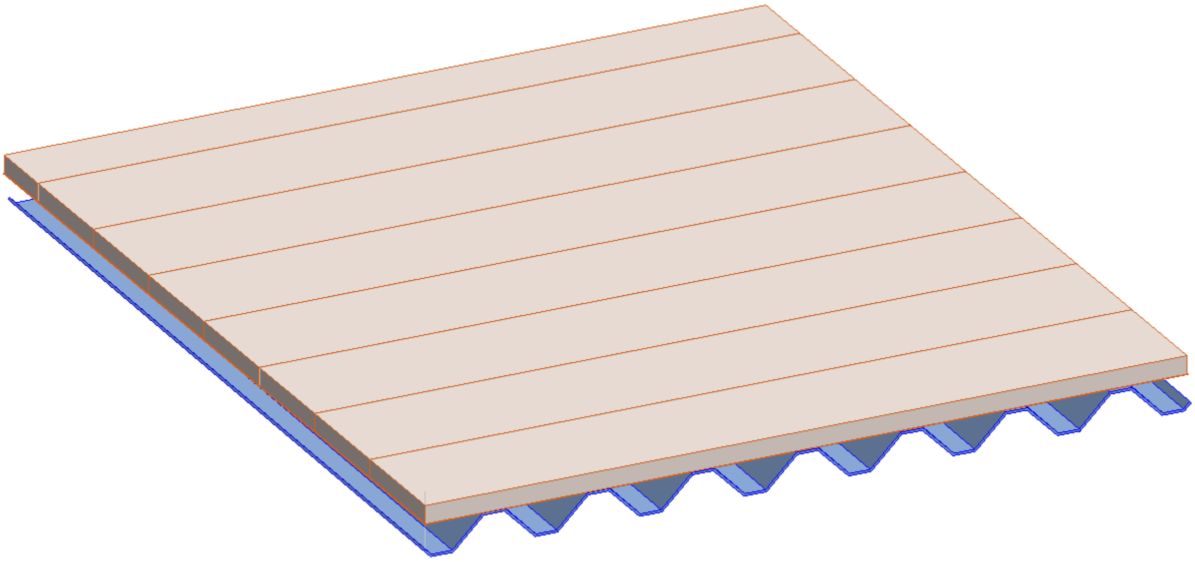
| | | |
|---|---|---|
| Hxz= | 5/6*G*h*a1/a= | 694444 kN/m |
| Hyz= | 5/6*G*h*b1/b= | 1458333 kN/m |

As it was mentioned above, if the D, K and H stiffness values are calculated and filled in, then it is possible to model the sections with complex geometry and composite materials with new fictitious shell object. Few section are show below which can be handled by the help of fictitious shell.

Corrugated steel + Concrete floor



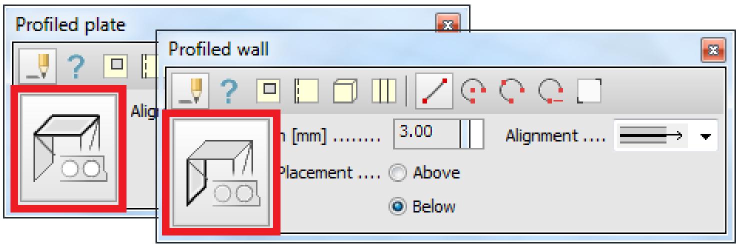
Corrugated steel + Timber floor

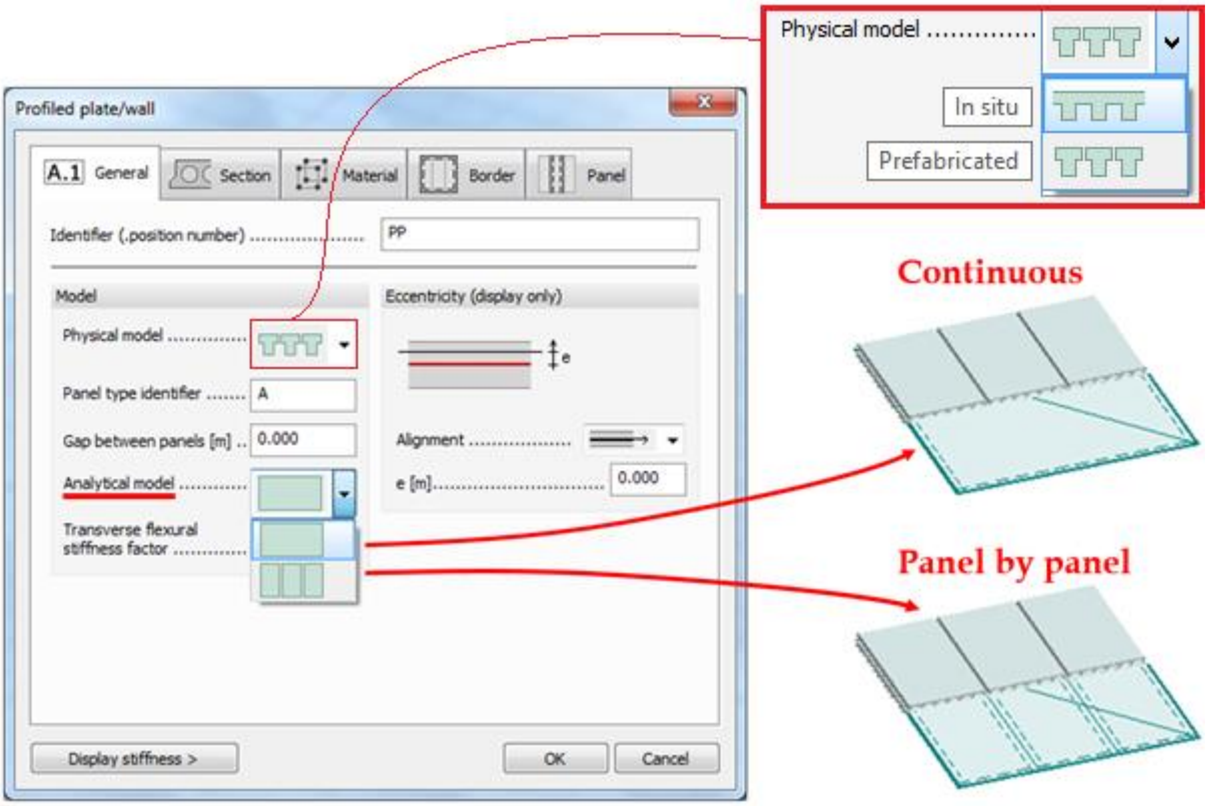


## 1.4. Profiled shell

### 1.4.1. Physical and Analytical system

In Profiled (also Timber) shell dialog there is opportunity to select the physical model (in-situ or prefabricated), then analytical system of shell, if applicable. The analytical system can be *Continuous* or *Panel by panel.* This option is available in profiled plate and profiled wall.




Choosing continuous analytical system can be useful when modelling the whole structure e.g. in the preliminary design phase. The memory usage of calculation with continuous analytical system is less than panel by panel system due to the reduced number of edge connections.

The use of panel by panel analytical system is reasonable in the phase of detailed design. In this case the results of the calculation are more accurate.

In case of continuous analytical system, effect of the connections between the panels can be considered by the transverse flexural stiffness factor.

### 1.4.2. Stiffness calculation

Profiled shells are calculated as Fictitious shells (the stiffness matrices are calculated automatically, according to the tables below).

$$D = \begin{bmatrix} \frac{E h_x}{1-\nu^2} & \nu\sqrt{\frac{E h_x}{1-\nu^2}\cdot\frac{E h_y}{1-\nu^2}} & \\ \nu\sqrt{\frac{E h_x}{1-\nu^2}\cdot\frac{E h_y}{1-\nu^2}} & \frac{E h_y}{1-\nu^2} & \\ & & G\frac{h_x+h_y}{2} \end{bmatrix}$$

$$K = \begin{bmatrix} \frac{E I_x}{1-\nu^2} & \nu\sqrt{\frac{E I_x}{1-\nu^2}\cdot q\frac{E I_y}{1-\nu^2}} & \\ \nu\sqrt{\frac{E I_x}{1-\nu^2}\cdot q\frac{E I_y}{1-\nu^2}} & q\frac{E I_y}{1-\nu^2} & \\ & & G\frac{\left(\frac{h_x+q h_y}{2}\right)^3}{12} \end{bmatrix}$$

$$H = \begin{bmatrix} \rho_x G h_x & \\ & \rho_y G h_y \end{bmatrix}$$

Section data:

$h_x$, $h_y$ : equivalent thickness in x and y direction

$I_x$, $I_y$: equivalent inertias in x and y direction

$\rho_x$, $\rho_y$: equivalent shear factor in x and y direction

x direction: strong axis of the panel cross-section

y direction: is perpendicular to the x direction in the plane of the panel

Material data:

E: Young modulus

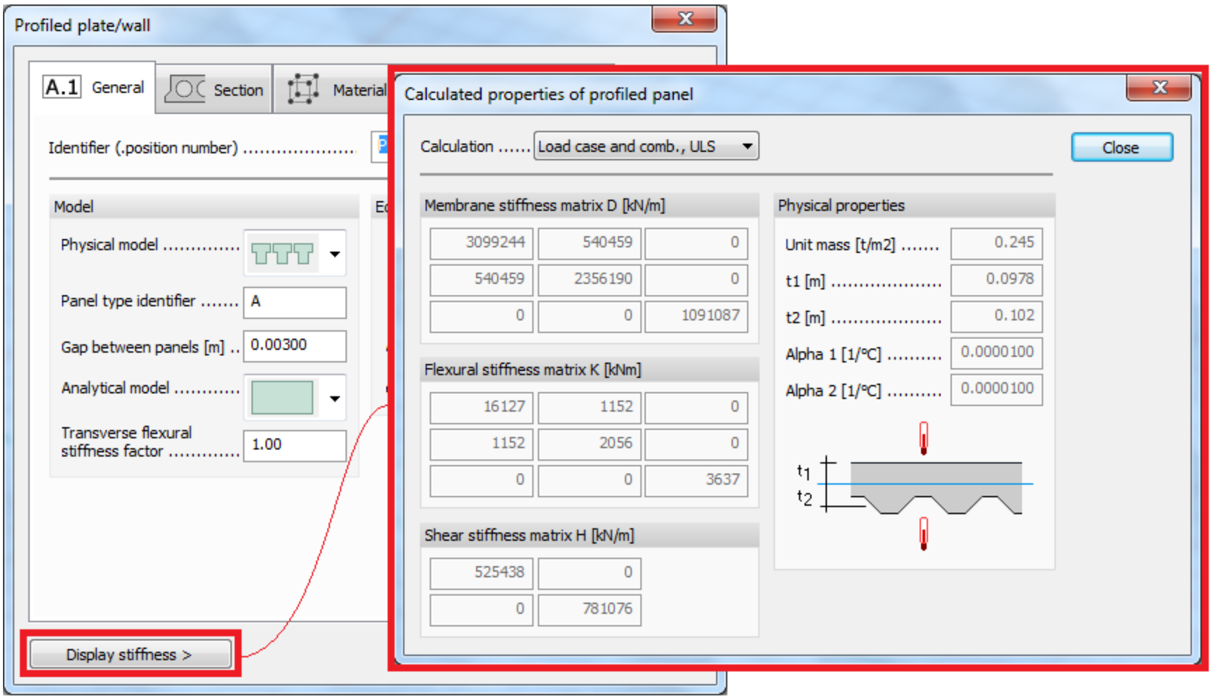
G: Shear modulus

ν: Poisson's ratio

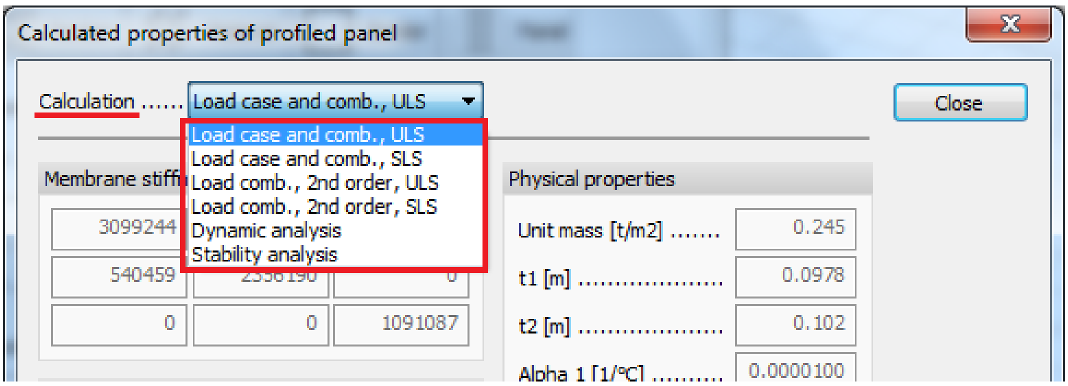
Panel data:

q: transverse flexural stiffness factor (applicable only in *Continuous* analytical system)

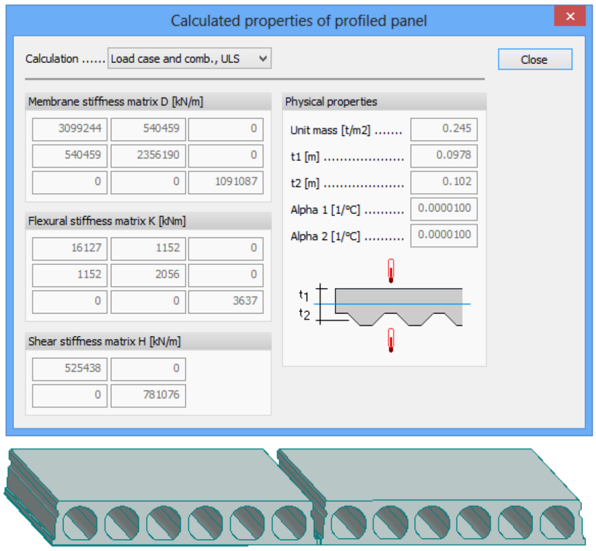
Modifying any above mentioned data will affect the stiffness of shell. The stiffness of shell is calculated automatically. The value of the stiffness (derived from properties of the shell) can be checked by clicking on the "Display stiffness" button.

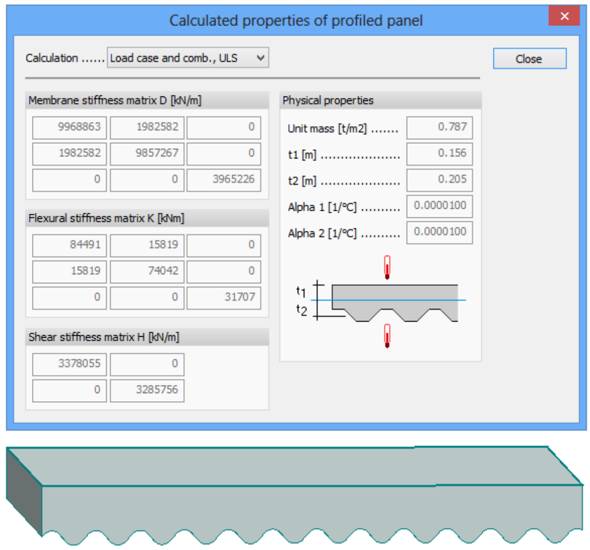


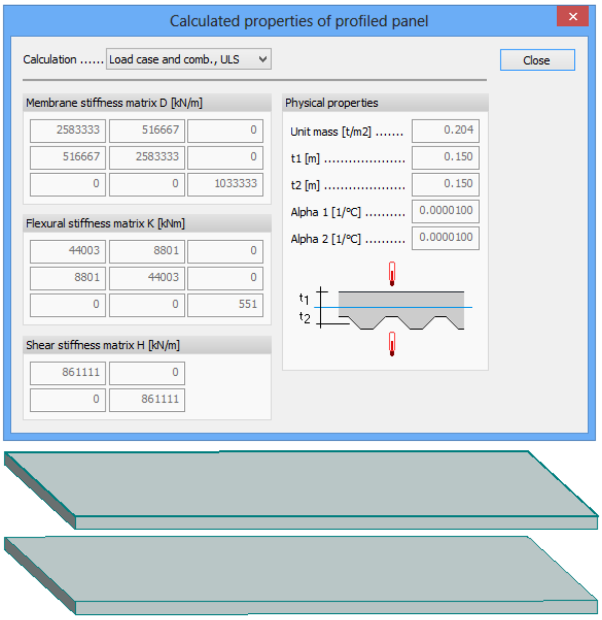

As it will be mentioned later (1.6), in version 14 there will be different types of Young moduli (except for steel material) in different types of calculations. This also affects the stiffness of shell in different types of calculations. These stiffness can easily be checked in "Display stiffness" dialog.

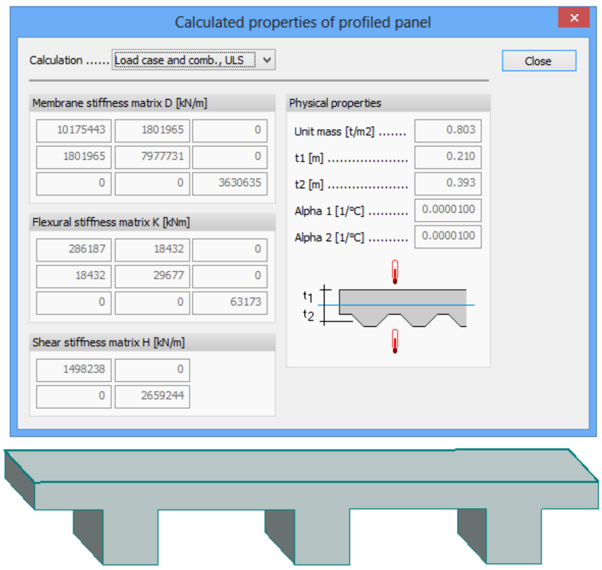


This powerful tool in version 14 gives possibility to automatically calculate stiffness of concrete shell having ***complex geometry***. Some typical examples are shown below (sections are made in section-editor).



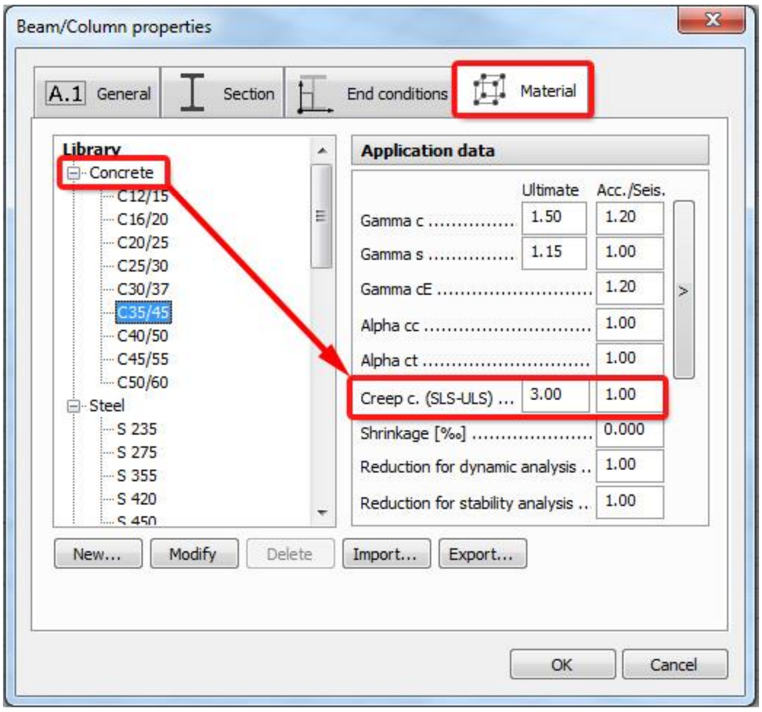




## 1.5. Creep coefficient

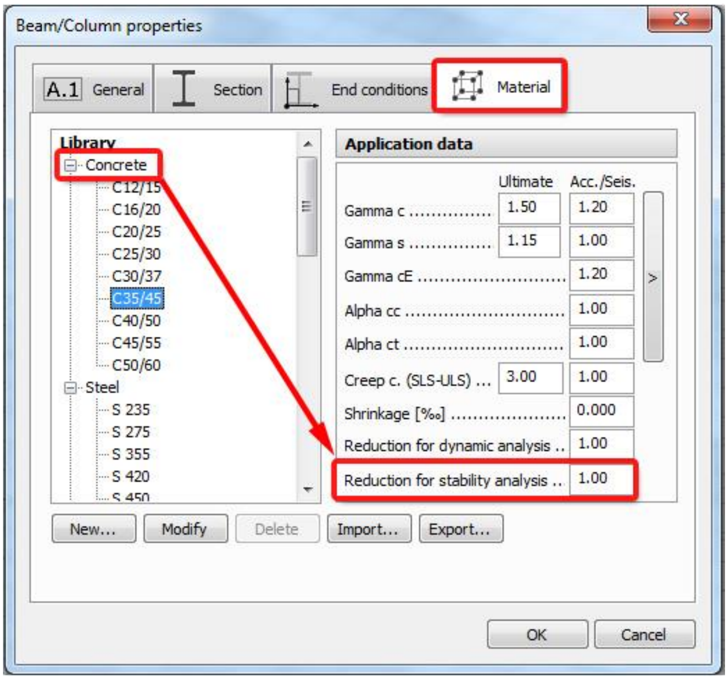
In Material settings dialog different creep coefficients can be specified for Serviceability Limit State (SLS) and Ultimate Limit State (ULS). It has some consequences in load case calculation and results:

- All load cases are calculated twice (first with the SLS Creep coefficient, than with the ULS Creep coefficient.
- The displayed displacements are the results of the SLS calculation.
- The displayed internal forces, reactions are the results of the ULS calculation.




## 1.6. Reduction for stability analysis

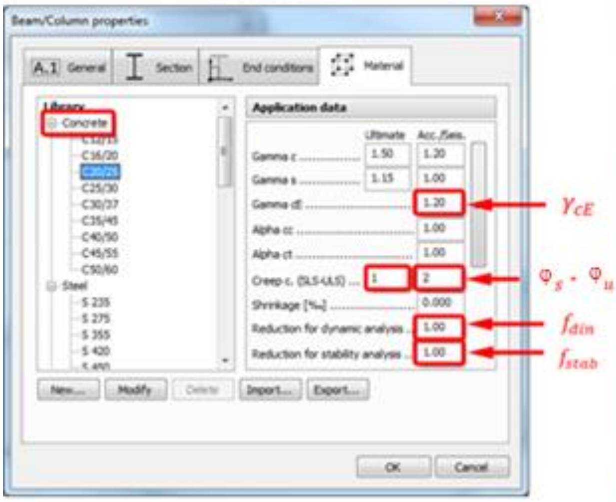
In Material settings dialog the stiffness for stability analysis can be decreased. Taking a reduction factor into account is needed in those calculations where it is specified by the standards (e.g. at the cracked section analysis).



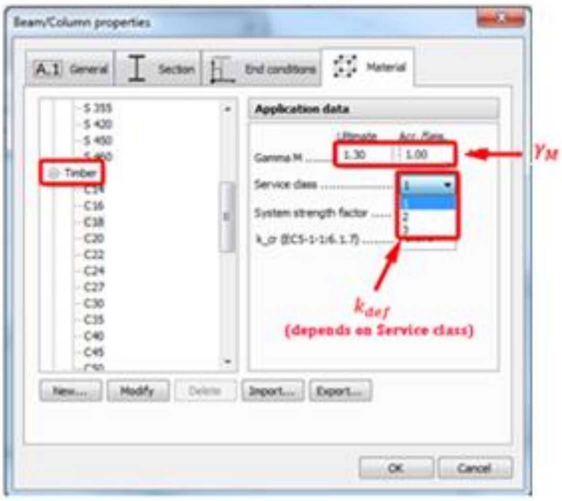
The following table and figures summarize the calculation of Young moduli used in different analysis types for concrete, steel and timber materials.

| Analysis type | | Concrete | Steel | Timber |
|---|---|---|---|---|
| Load cases,<br>Load group,<br>1st order load combination | ULS | $\frac{E_{cm}}{(1+\varphi_u)}$ | $E_k$ | $\frac{E_{o,mean}}{(1+k_{def})}$ |
| | SLS | $\frac{E_{cm}}{(1+\varphi_s)}$ | | |
| 2nd order load combination,<br>Imperfection calculation | ULS | $\frac{E_{cm}}{(1+\varphi_u)} \cdot \frac{1}{\gamma_{cE}}$ | | $\frac{E_{o,mean}}{\gamma_M}$ |
| | SLS | $\frac{E_{cm}}{(1+\varphi_s)} \cdot \frac{1}{\gamma_{cE}}$ | | |
| Stability analysis | | $f_{stab} \cdot E_{cm}$ | | $\frac{E_{o,mean}}{(1+k_{def})}$ |
| Eigenfrequency calculation<br>Seismic analysis | | $f_{din} \cdot E_{cm}$ | | $E_{o,mean}$ |

**Concrete**




**Timber**

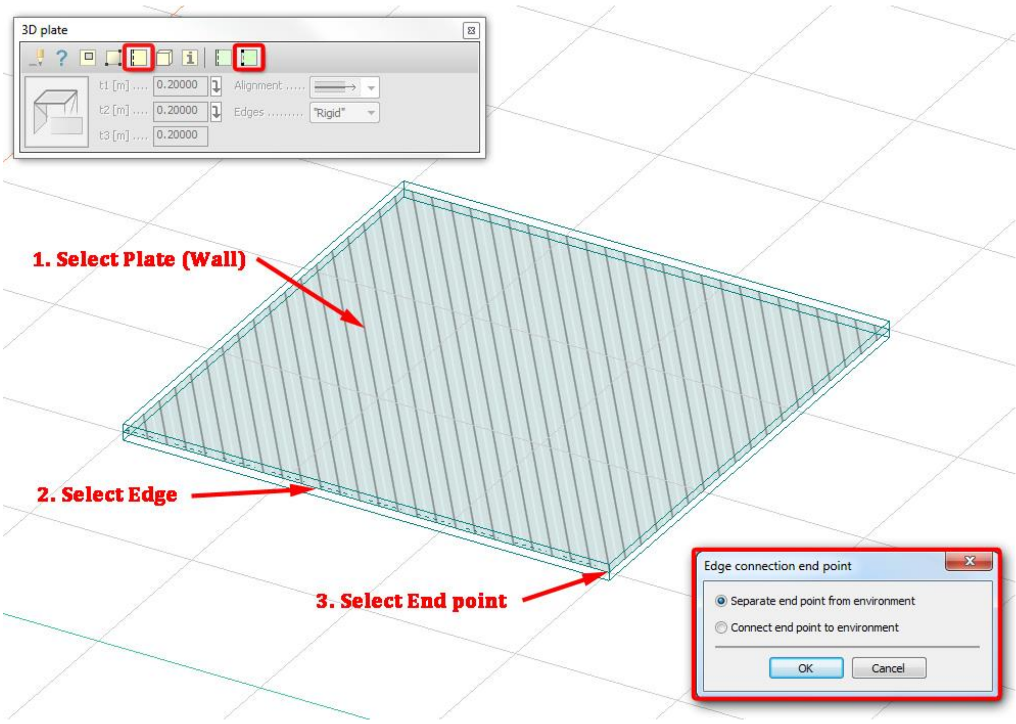



## 1.7. Edge connection / End point behaviour

In FD 14 for the end points of the edge connections two options are available:

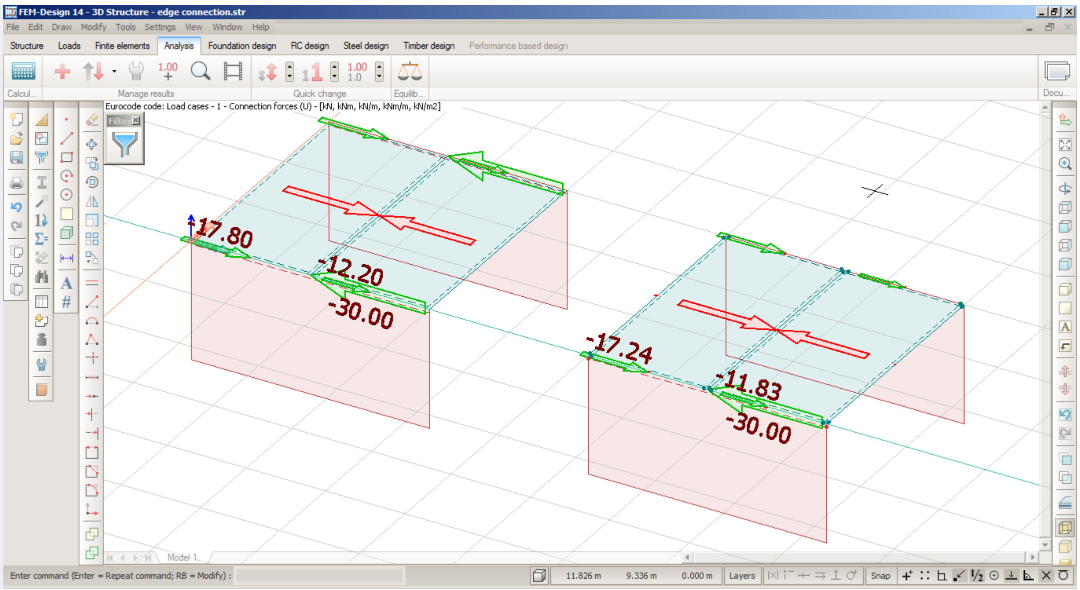
- Separate end points from environment,
- Connect end point to environment.

In all Plate and Wall Toolwindows this setting is available by choosing 'Edge connection' then clicking  on 'End point behaviour'  button. The end point can be chosen in three steps by selecting the Plate or Wall, then the edge and finally the end point.




Using this function can solve problems like the one on the picture below:

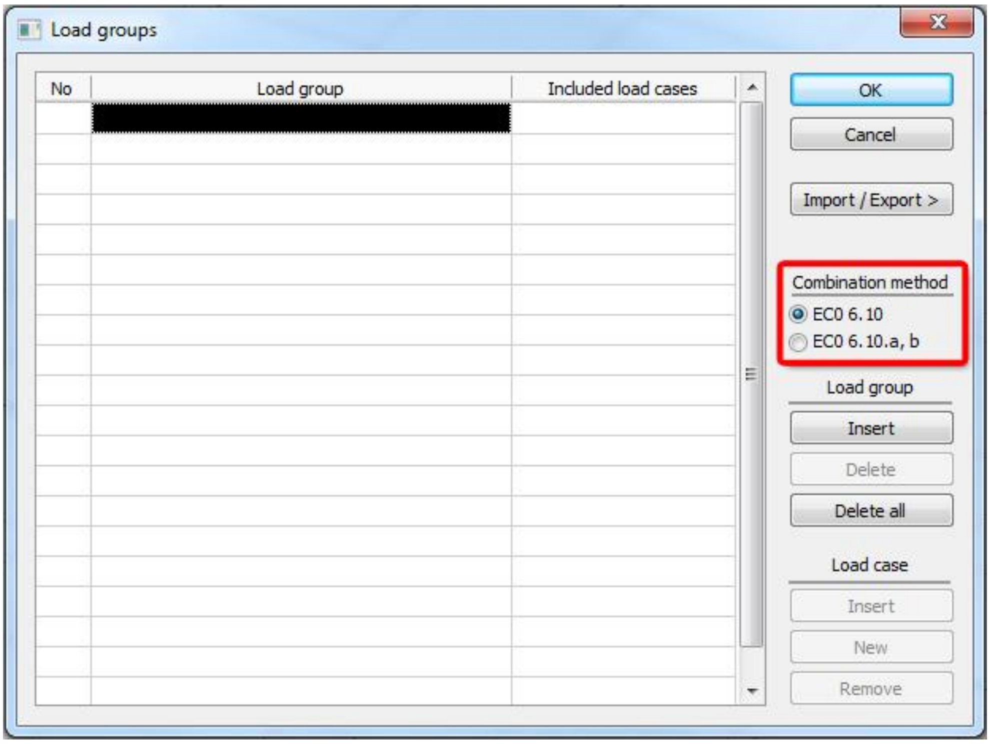
On the right side structure all edge connection end points are connected to the environment which cases difference between the shear force on the wall and the sum of shear forces on the plate panels, which should be in balance according to common sense and this is exactly the case on the left side structure where edge connection end points are separated from the environment.



# 2. LOADS

## 2.1. Load groups

In *Load groups* dialog the User has the opportunity to choose one of the combination methods offered by Eurocode 0. Two methods of determining the combination of actions are allowed for the STR Ultimate Limit States.




The first approach is to use expression 6.10.

$$\sum_{j\geq 1}\gamma_{G,j}G_{k,j}\text{ " + "}\gamma_{p}P\text{" + "}\gamma_{Q,1}Q_{k,1}\text{" + "}\sum_{i\geq 1}\gamma_{Q,i}\psi_{0,i}Q_{k,i} \qquad (6.10.)$$

The second approach is to use the more onerous of expressions 6.10.a and 6.10.b.

$$\sum_{j\geq 1}\gamma_{G,j}G_{k,j}\text{ " + "}\gamma_{p}P\text{" + "}\gamma_{Q,1}\psi_{0,1}Q_{k,1}\text{" + "}\sum_{i\geq 1}\gamma_{Q,i}\psi_{0,i}Q_{k,i} \qquad (6.10.a)$$

$$\sum_{j\geq 1}\xi_{j}\gamma_{G,j}G_{k,j}\text{ " + "}\gamma_{p}P\text{" + "}\gamma_{Q,1}Q_{k,1}\text{" + "}\sum_{i\geq 1}\gamma_{Q,i}\psi_{0,i}Q_{k,i} \qquad (6.10.b)$$

The subtle attraction of this pair of expression derives from two important changes from 6.10.:

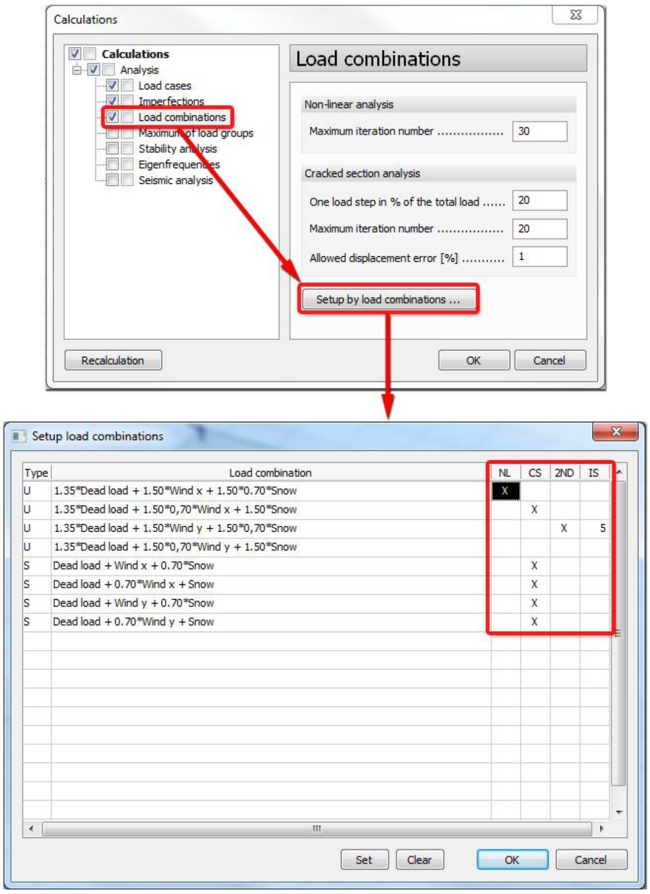
- The application of the $\psi_0$ factor to the leading variable action in expression 6.10.a (not applied in 6.10.)
- The introduction of a reduction factor $\xi$ applied to the permanent actions in expression 6.10.b (not applied in 6.10.)

The whole expressions appear on the screen as the cursor is moved on the name of the combination method.

# 3. ANALYSIS

## 3.1. Setup calculation by load combinations

The calculation of the load combinations can be run with different options. They can be set in Calculations dialog by selecting the Load combination items and clicking on *Setup by load combinations*.




The User has the opportunity to choose

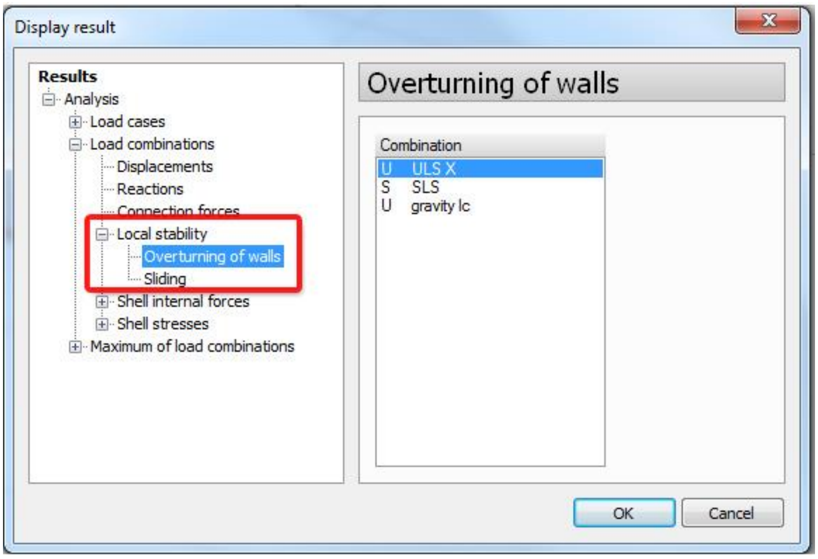
- Non-linear calculation (NL),
- Cracked-section analysis (CS),
- Second order analysis (2ND),
- Imperfection calculation (IS, the selected shape will be taken into account in Second order analysis)

for each Load combinations.

For example, in practice it can be useful to set 2nd order analysis only for the ULS and Cracked-section analysis only for the SLS combinations.

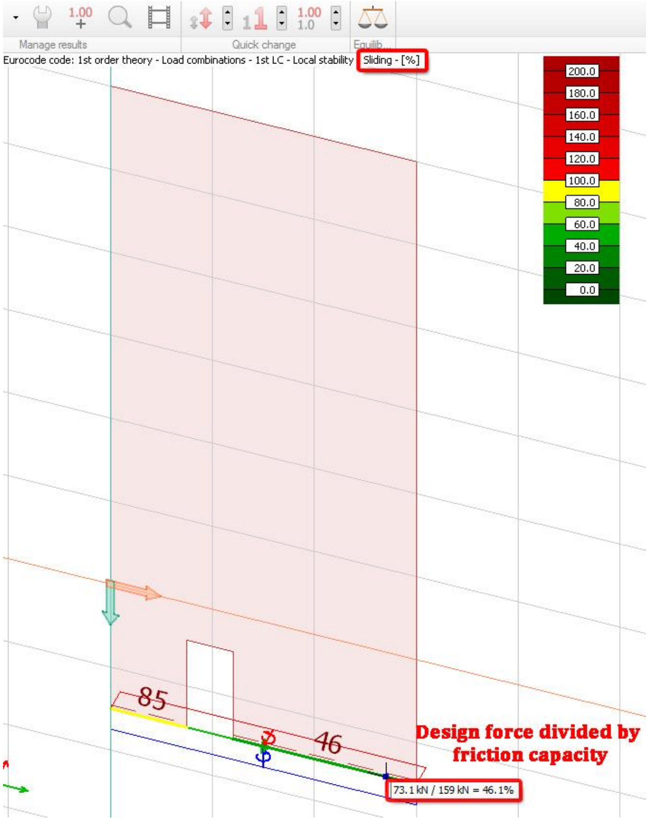
## 3.2. Local stability results

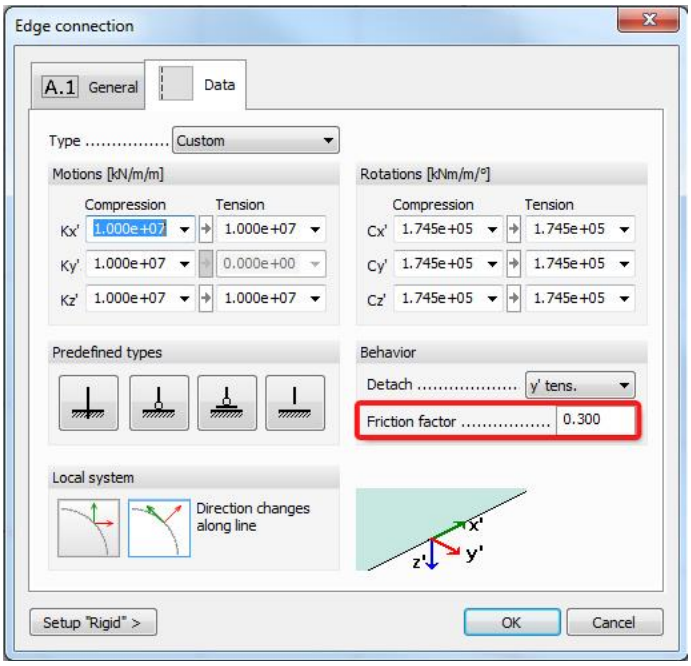
In FD 14 local stability results can be displayed. After calculating the load combinations the Local stability results (*Overturning of walls* and *Sliding*) are available in Display results dialog.




- Overturning of walls: only those walls can be calculated which have at least one horizontal edge in the bottom and edge connection is defined for it. The result is expressed as a percentage:
  - 0% belongs to the case when the vertical force acts at the centre of bottom edges,
  - 100% belongs to the case when the vertical force acts at one of the corners,
  - 1000% belongs to the case when the resultant is outside the wall edge.

- Sliding of edge connections: the result is the ratio of the design force and the friction capacity. The friction factor can be set in the *edge connection* dialog.

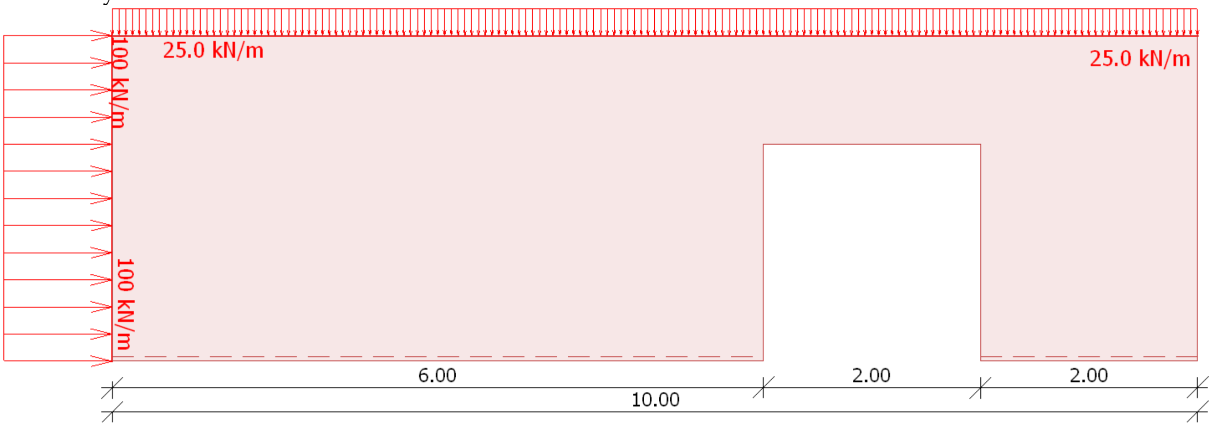


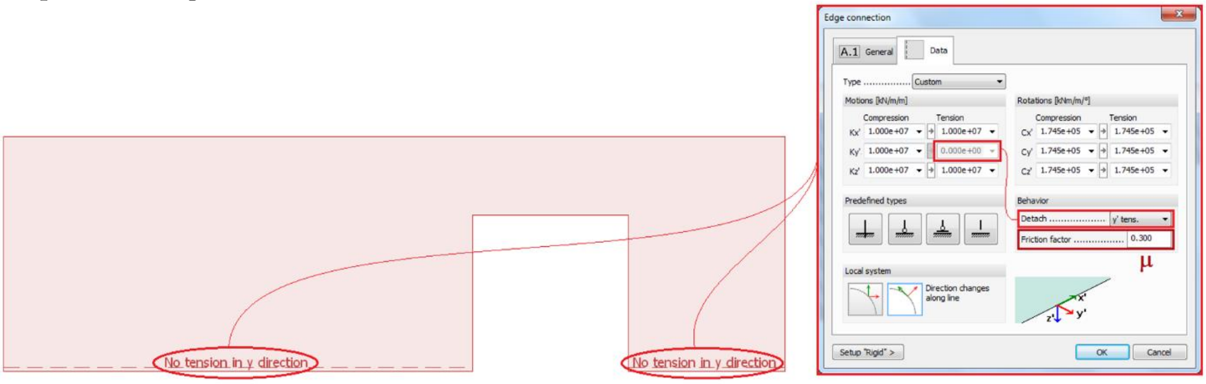

⚠ Overturning of walls results are informative. Without accurate modelling it may lead to incorrect results!

Numerical example below will illustrate the *Local Stability*.

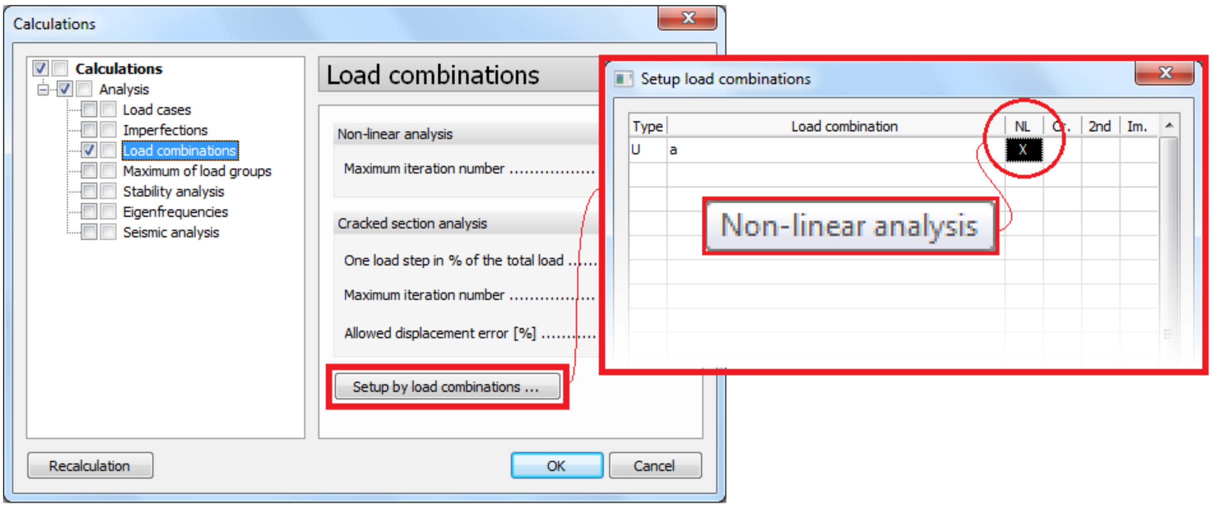
Geometry and Loads



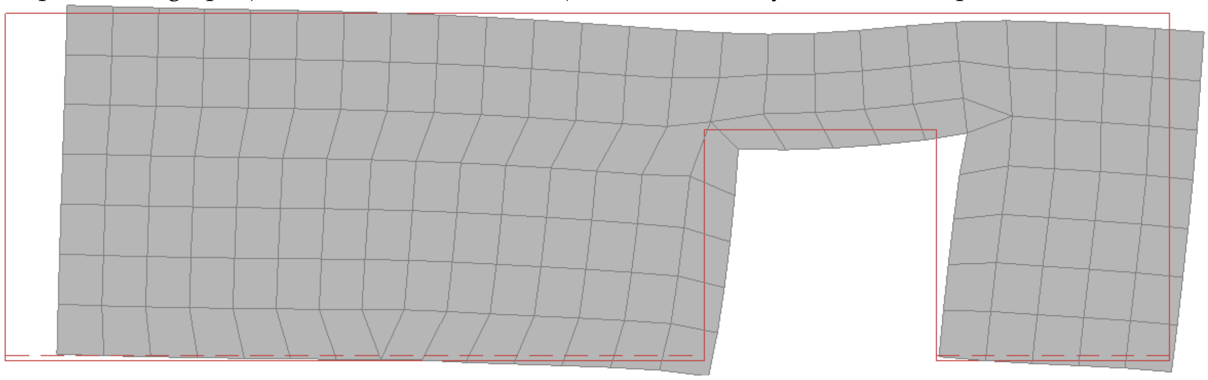

Properties of edge connections




Non-linear calculation (which allows uplift) is recommended to get correct result for local stability.

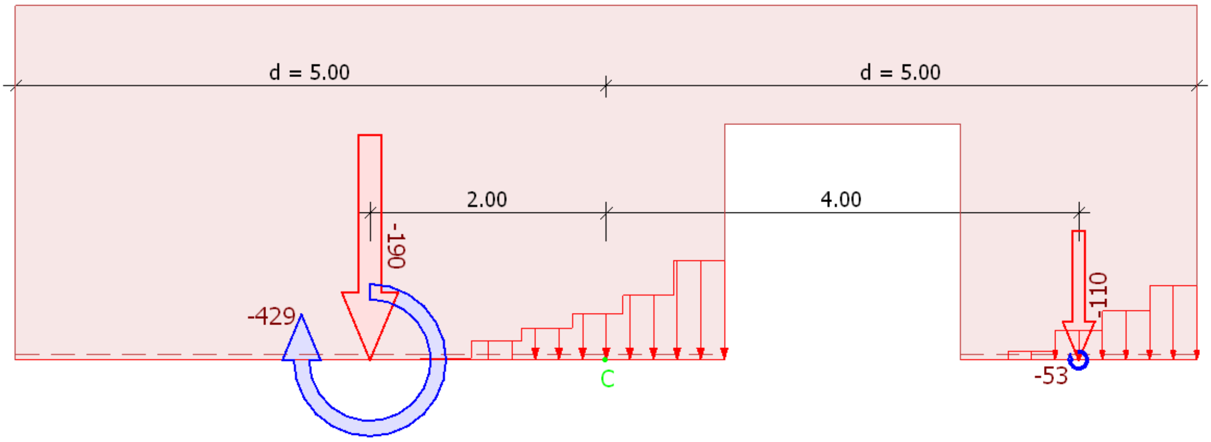


Displacement graph (as well as connection force) is the easiest way to check the uplift.



**Overturning of wall**

With the help of resultants of edge connections, wall's overturning can be examined as below.

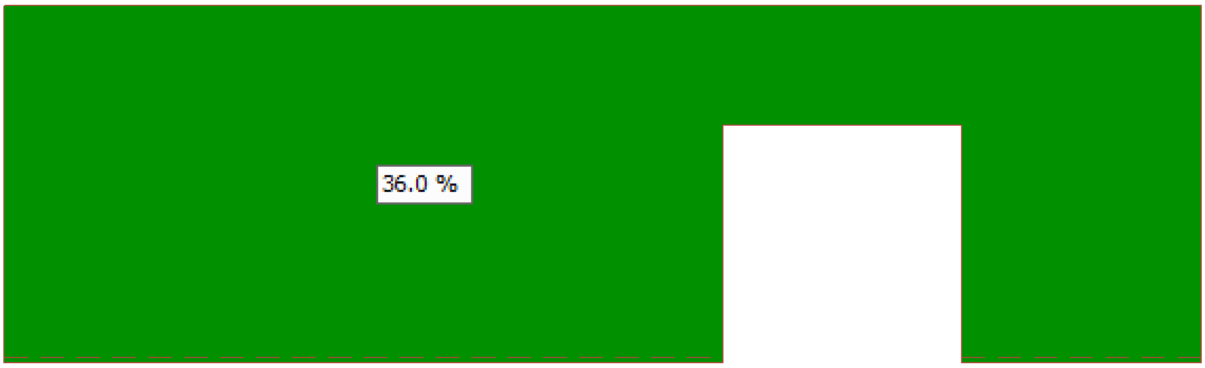



$$F = \sum_{i=1}^{n} F_i = -190 - 110 = -300\ kN$$

$$M^{(C)} = \sum_{i=1}^{n} M_i + F_i * v_i = -429 + 190 * 2 - 53 - 110 * 4 = -542\ kNm$$

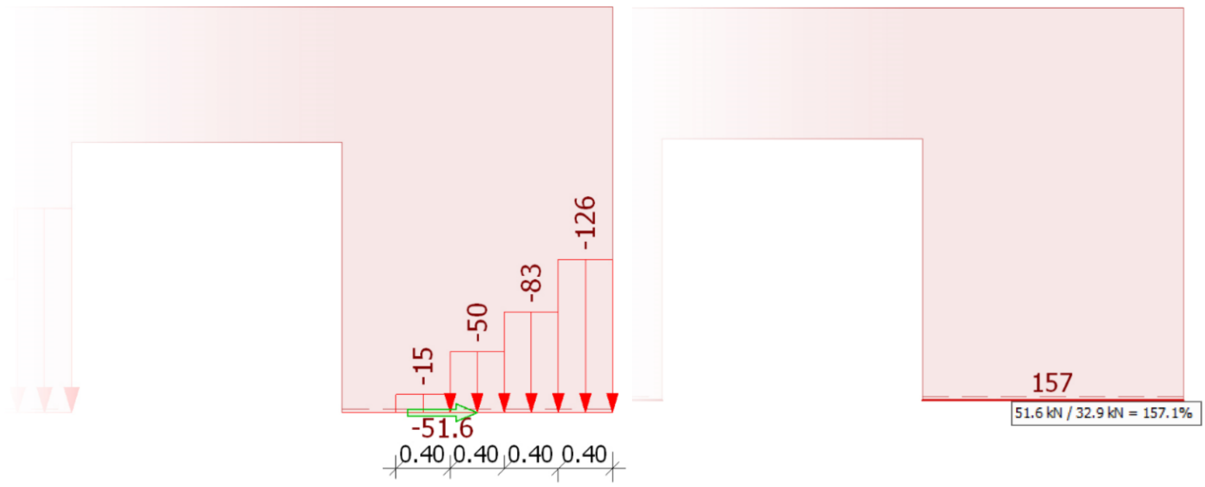
$$e = \frac{M^{(C)}}{F} = \frac{-542}{-300} = 1.81\ m$$

$$Utilization = \frac{e}{d} * 100\ \% = \frac{1.81}{5.00} * 100 = 36\%$$



**Sliding of edge connections**

Edge connection's sliding is calculated in each edge connection *separately* by comparing the x' component of the connection force as design force, and the limit force calculated by the y' components of the connection forces and the friction coefficient of the edge connection.

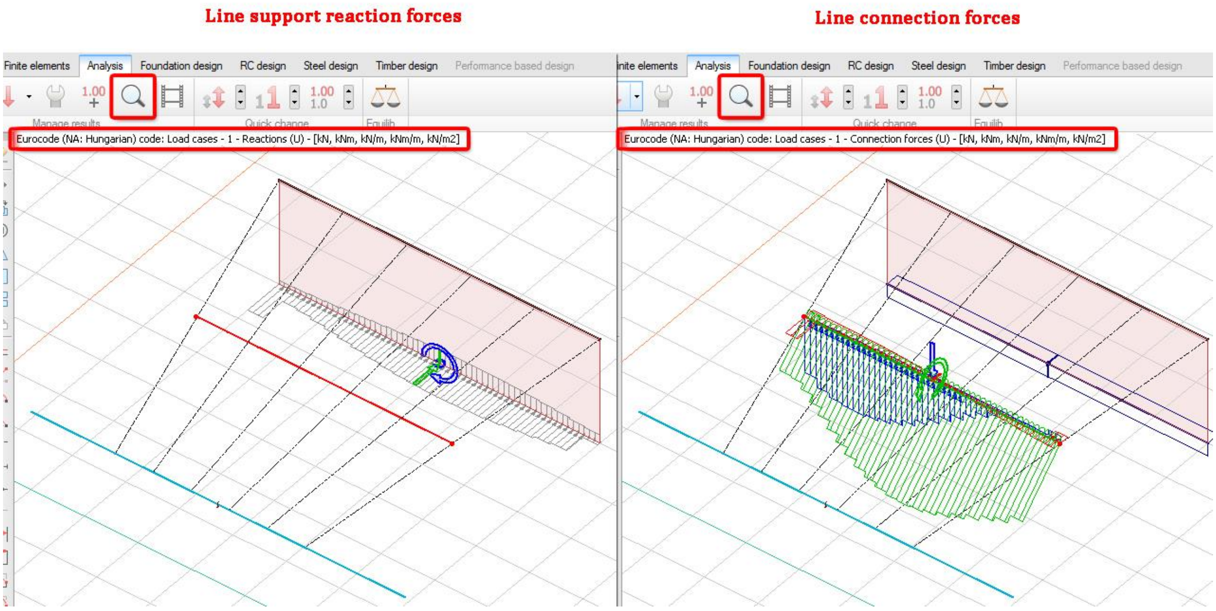



$$F_{lim} = \mu * \sum_{i=1}^{n} f_i * dx_i = 0.3 * (15 * 0.4 + 50 * 0.4 + 83 * 0.4 + 126 * 0.4) = 32.9\ kN$$

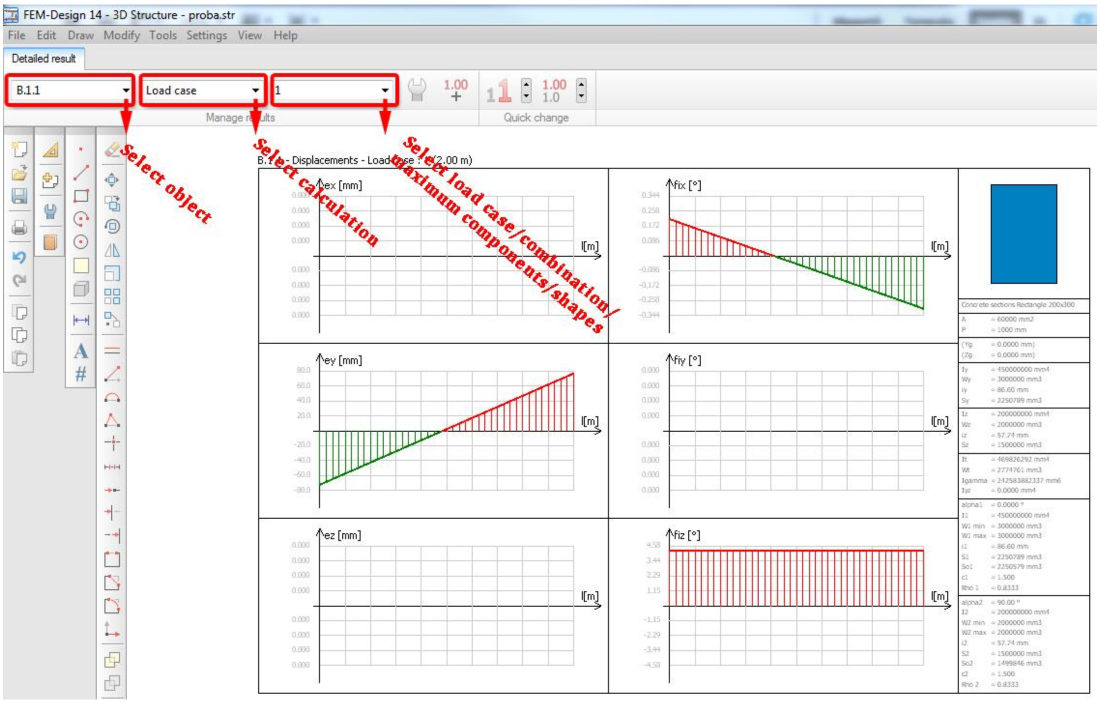
$$Utilization = \frac{F}{F_{lim}} * 100\ \% = \frac{51.6}{32.9} * 100 = 157.1\%$$

## 3.3. Detailed result

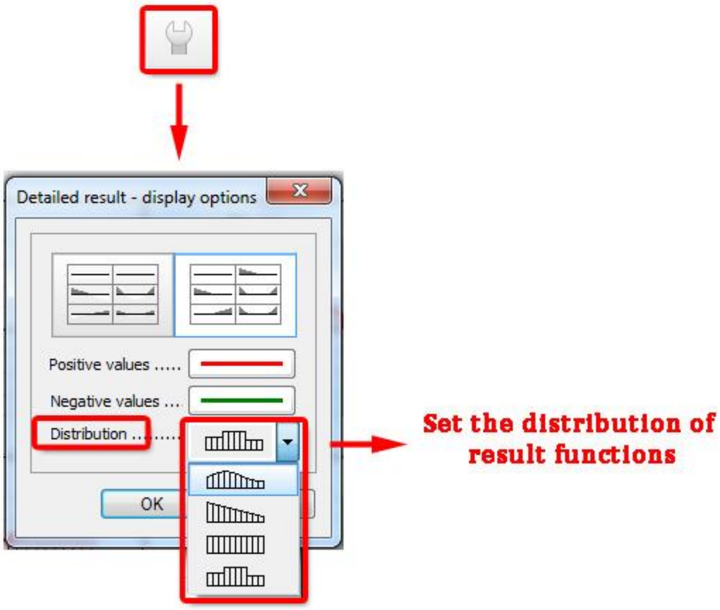
Detailed analytical results are available for two-dimensional analytical objects (e.g. bars, line support groups and connections) after clicking on  button and by selecting the object.




In *Detailed result* window the User can manage the results by selecting the object, the calculation and other – calculation dependent – options.



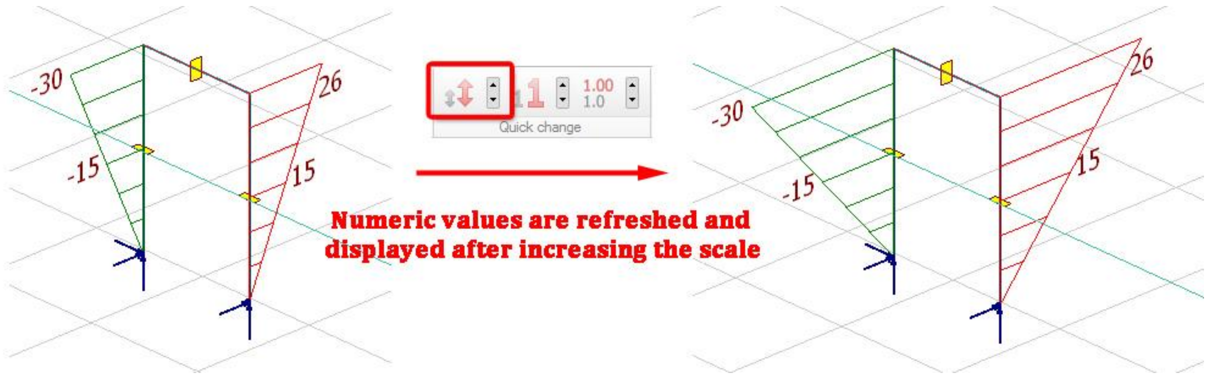

The dialog of the display options is renewed, as the distribution of the selected result functions can be chosen (calculated, linear, constant or constant by element) for line support groups and connections.




## 3.4. Refreshing numeric values

The numeric values are refreshed automatically when the display options of a result are modified.

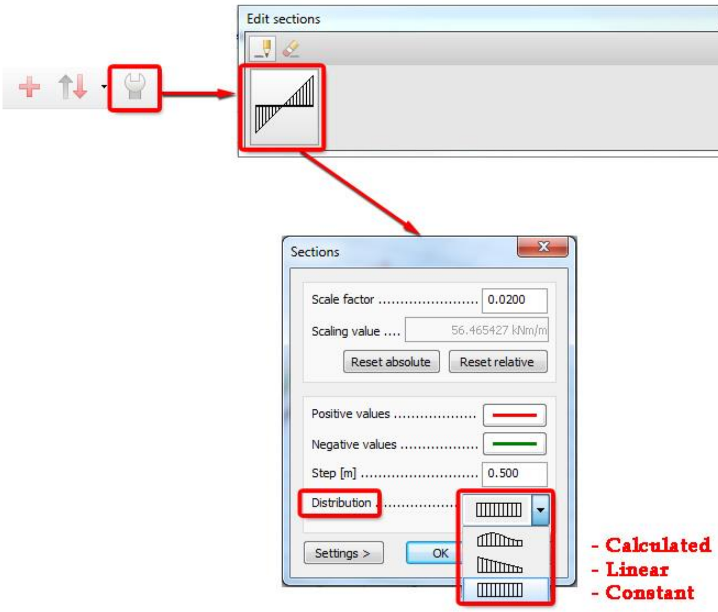
 The following figure describes this feature on an example of a simple frame structure.




## 3.5. Section result distribution for shells

The distribution of the displayed section results can be chosen by the user. Besides the calculated distribution it can be linear or constant.

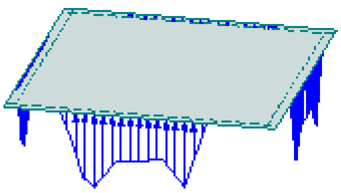
This function can be useful at checking the average in-plane shear force between profiled panels.



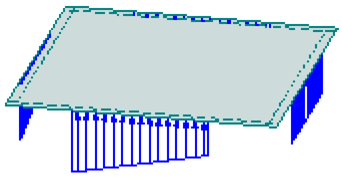

## 3.6. Displaying support reactions and connection forces according to uplift

In FD 14 displaying of line support reactions and connection forces has been improved in case of linear and constant distributions by considering which part of the support/connection is "uplifted".

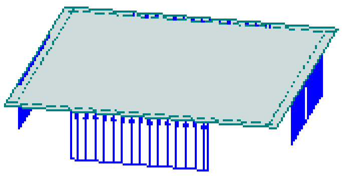
Calculated distribution



Linear distribution



Constant distribution



# 4. DESIGN

## 4.1. Manual design

In FEM-Design 14 the manual design can be done without calculated analysis results in the following categories:

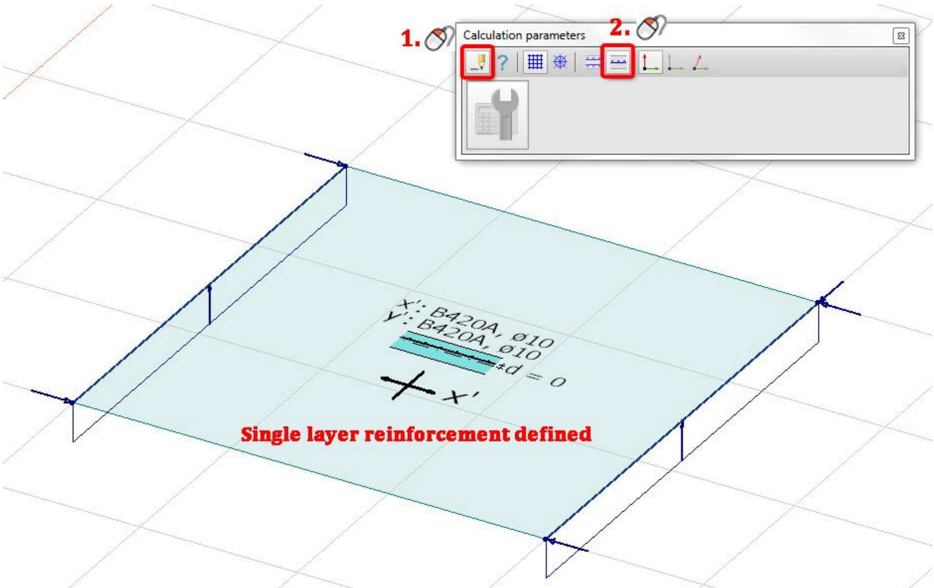
- RC design
- Steel design
- Timber design

This function is useful especially at the analysis of an existing building.

# 5. RC DESIGN

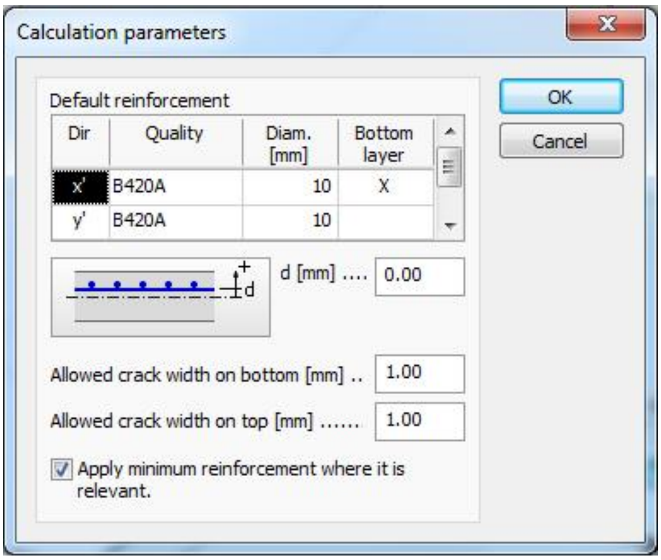
## 5.1. Single layer reinforcement

In RC design the User has the possibility to place single layer surface reinforcement. A Plate or Wall can be specified as single layer reinforced by defining “Single layer reinforcement” Calculation parameter for it.

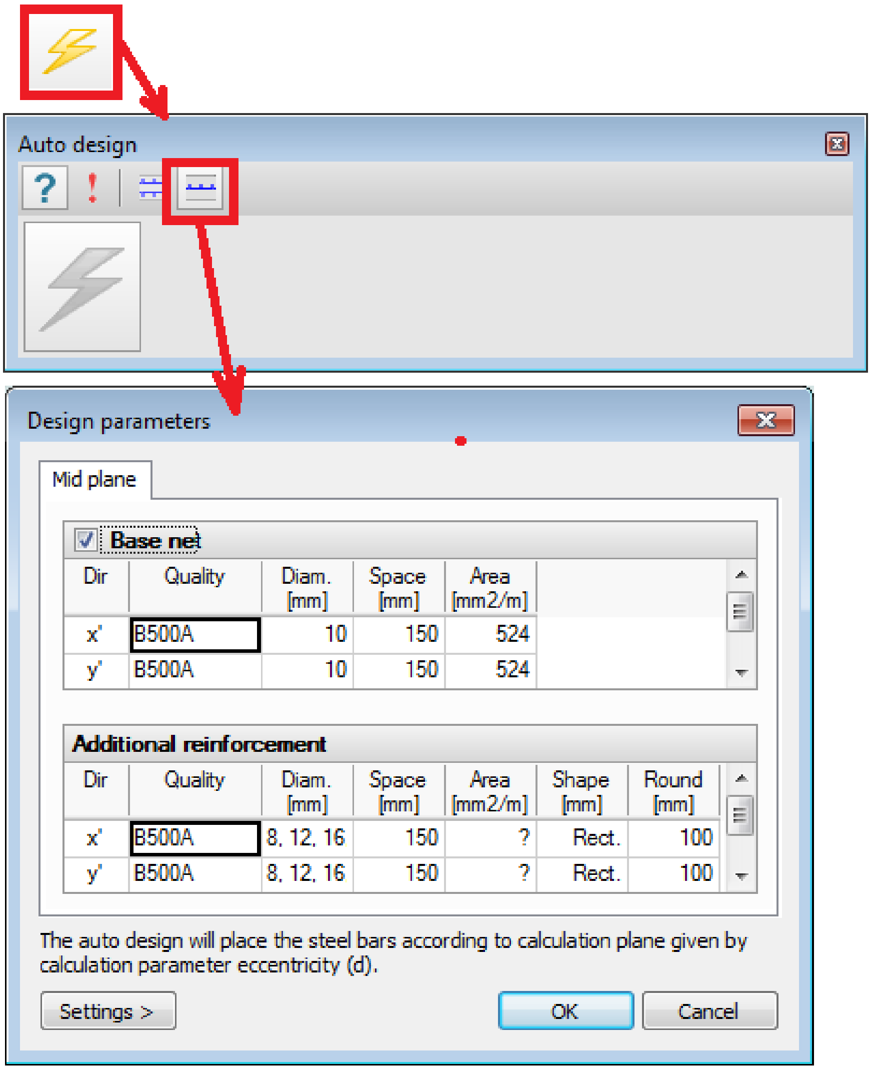



In the Calculation parameters dialog after clicking on the single layer reinforcement and the default settings  button the calculation parameters can be set in the dialog. The User can define the followings:

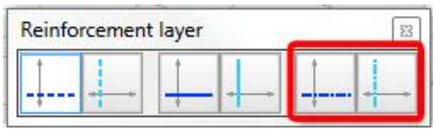
- the quality and diameter of the reinforcement for both directions,
- the direction of the bottom layer,
- the distance of the reinforcement from the centreline,
- the allowed crack width on the bottom and on the top of the structure.



If a shell has “Single layer reinforcement” Calculation parameter, its Design parameter can be modified only if “Single layer reinforcement” option is selected in Auto design/Parameters.




In case of Manual design, single layer reinforcement can be placed only on „Mid, x’/r” and „Mid, y’/t” layers.

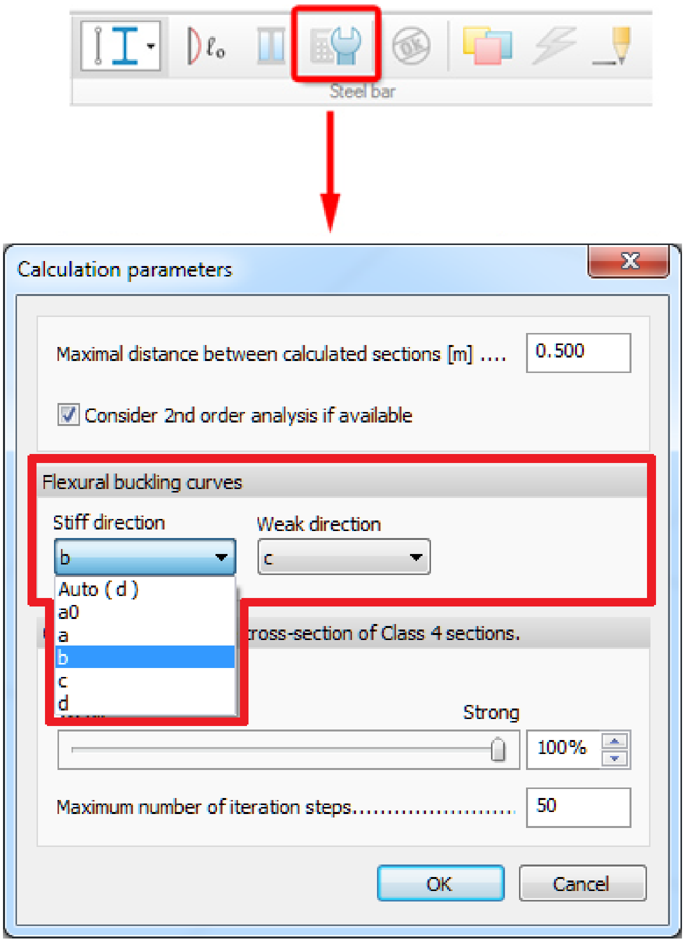



⚠ Single and double layer reinforcements cannot be used in the same Plate or Wall element.

# 6. STEEL DESIGN

## 6.1. Flexural buckling curve defined by the user

User can specify the flexural buckling curves (EC3-1-1: 6.3.1.2) for each steel bar in Calculation parameters dialog or they can let the program to calculate it as in the previous versions by selecting "Auto" option.



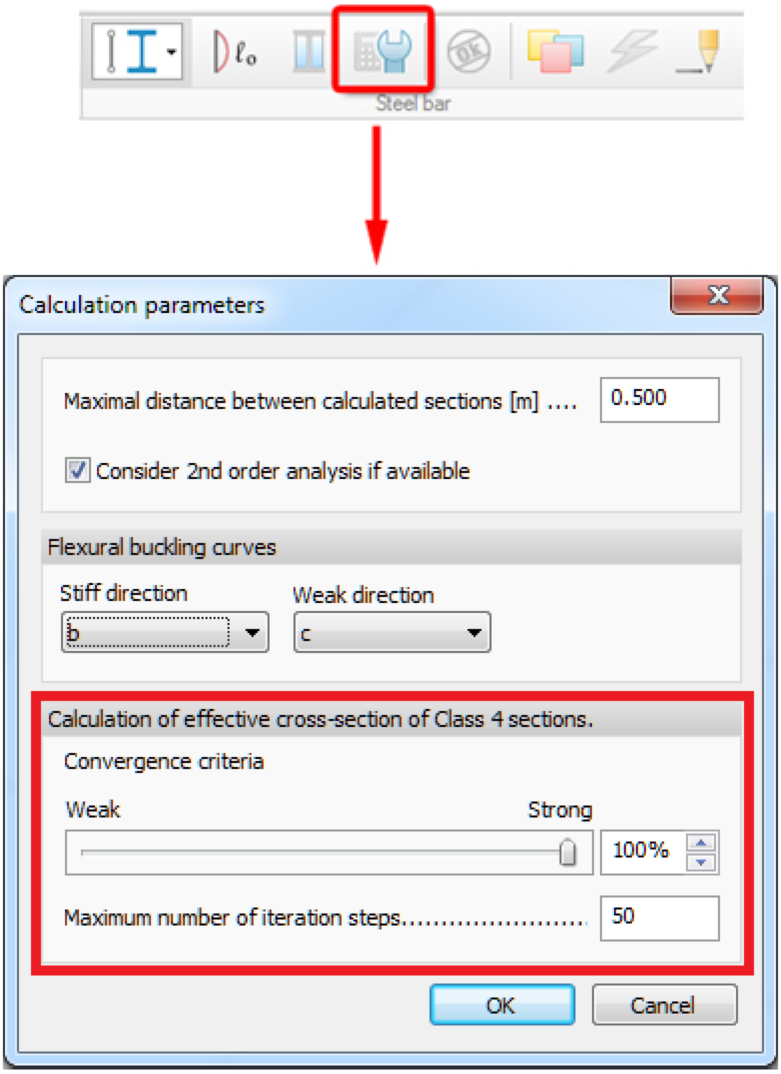

⚠ For steel bars with varying section the "Auto" option cannot display the automatically calculated curve, since it is determined during the design calculation.

⚠ When the section of a steel bar is modified, buckling curve options of the calculation parameter is reset to "Auto"

⚠ When buckling curve is calculated automatically, applied section is considered, if it exist.

## 6.2. Convergence criteria for Class 4 steel sections

Convergence criteria and the maximum number of iteration steps can be set for effective cross-section calculation of Class 4 steel bar section in Calculation parameters dialog.

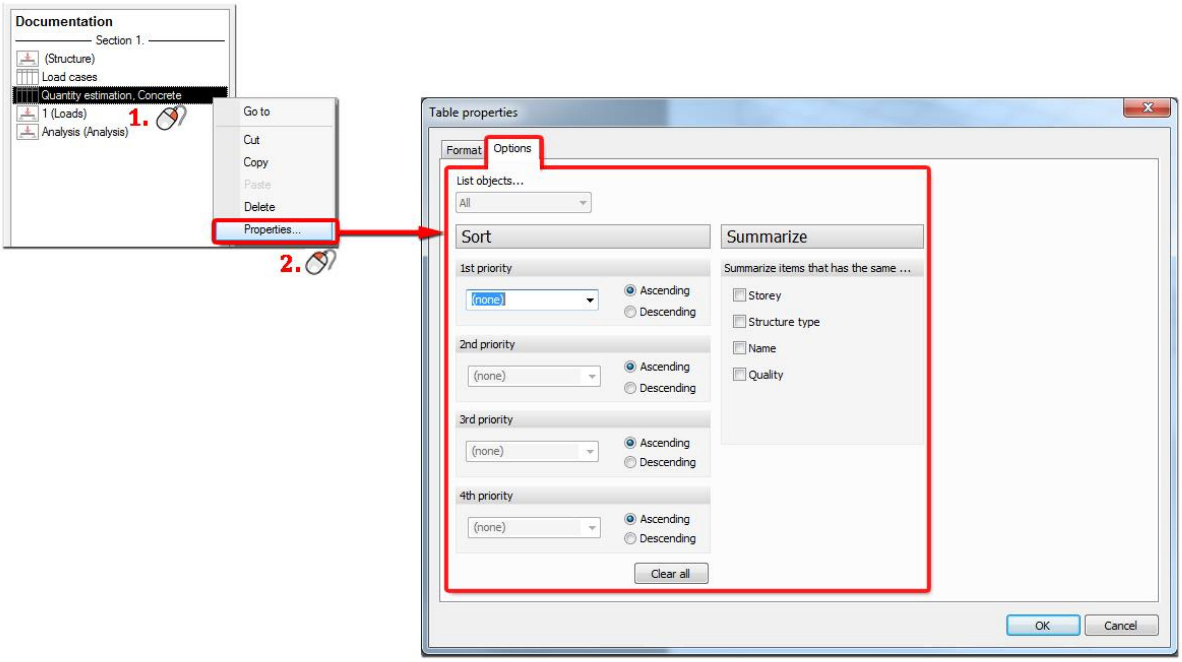



In some cases the iteration for effective cross-section fails because of the too strong convergence criteria. In this case reducing its factor or increasing the number of the iteration steps may solve the problem.

# 7. DOCUMENTATION

## 7.1. Options tab in table properties

Options tab (if available) is added to the Table properties dialog in documentation.



# 8. SYSTEM

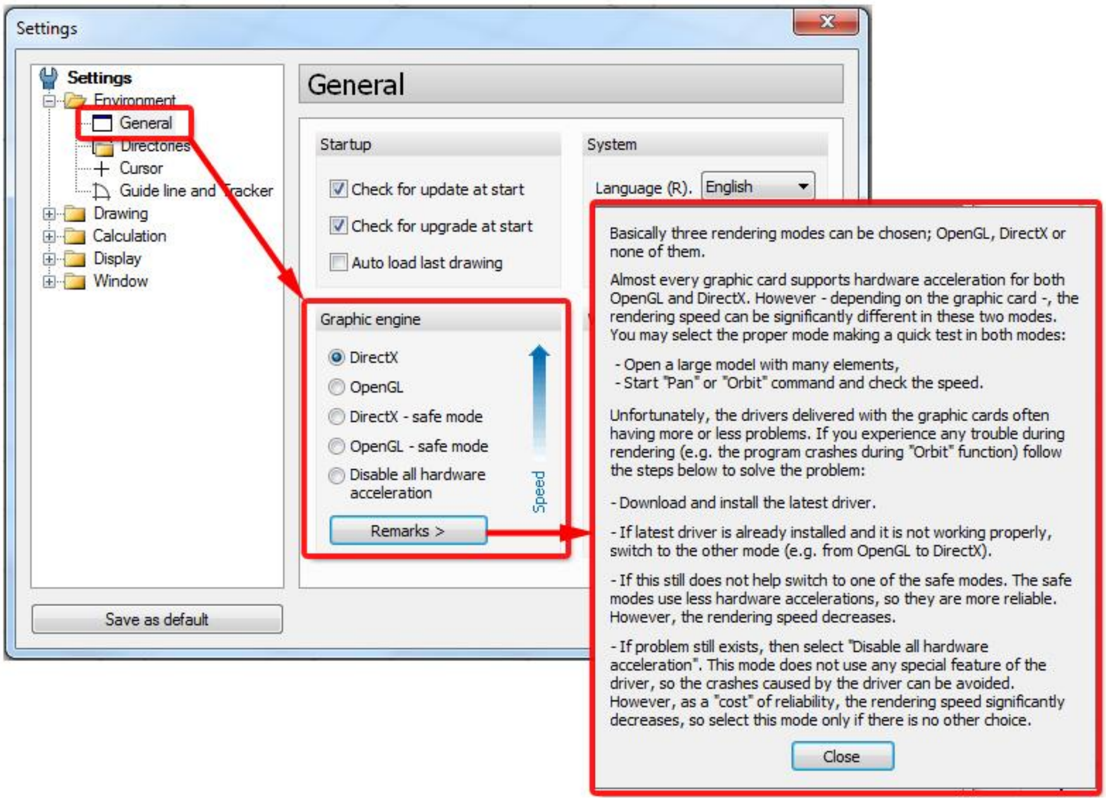
## 8.1. Full 64 bit version

As FD 14 is a full 64-bit software, all memory of the computer can be utilized. In previous 32-bit versions the memory usage was limited to 2 Gigabytes, which could cause out of memory error message when displaying results at large models. The analysis core was already 64-bit in earlier versions.

⚠ **32-bit operating systems are not supported anymore!**

## 8.2. Increased graphical performance

The graphical performance is increased and more development is expected in future versions. In FD 14 the DirectX graphic engine is faster than OpenGL on most of the computers. We recommend the User to check which graphic engine performs better on his computer.

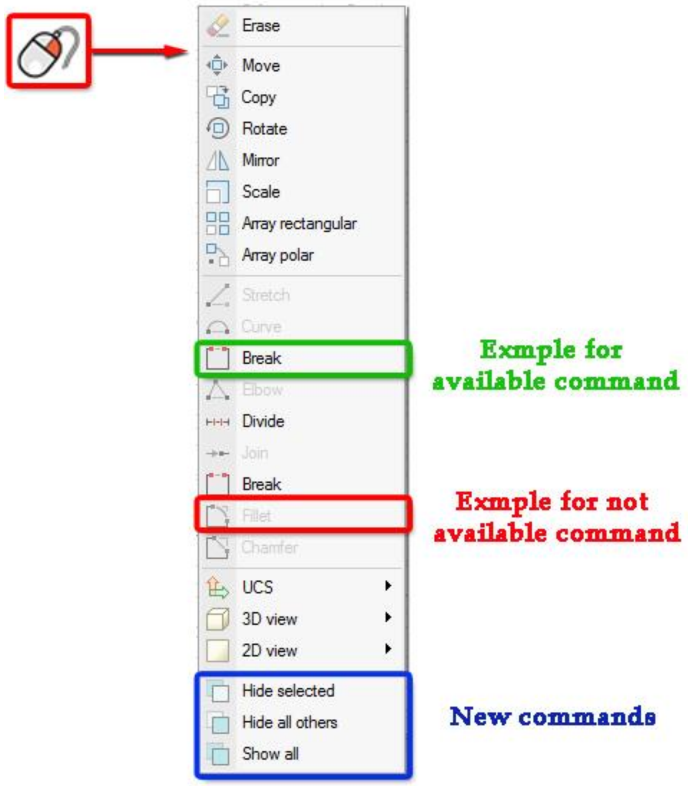



For more information about the graphic engine read the Remarks.

# 9. CAD/USER INTERFACE

## 9.1. Quick menu

By clicking right mouse button, a renewed quick menu appears. After clicking on an arbitrary element all the commands of quick menu are visible but those that are not available for the chosen element(s) are disabled.




3 new commands are added to the quick menu:

- **Hide selected**

  For this command FD hides the selected elements

- **Hide all others**

  For this command FD hides all the elements except the selected ones.

- **Show all**

  For this command FD shows all the hidden elements.

## 9.2. Quick selection

Holding 

pressed and by using  objects can be added to or subtract from the current selection. By pressing 

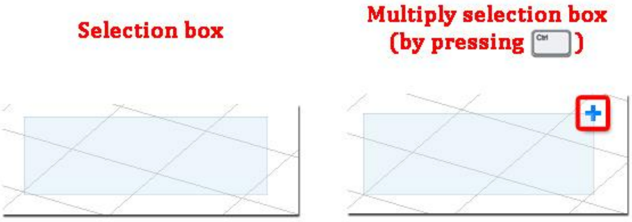
+A all the visible objects on the screen get selected.

## 9.3. Selection box, multiple selection

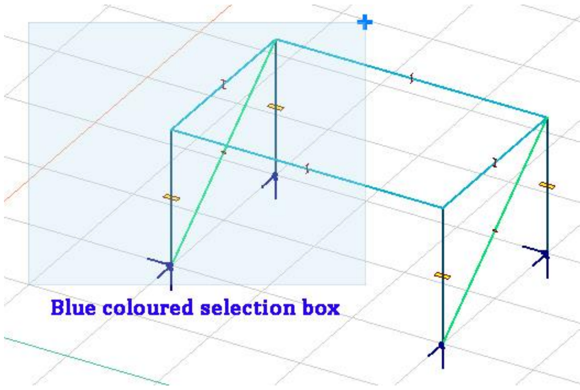
The selection box changed in FD 14:

- '+' sign is drawn if multiply selection is active (by pressing 

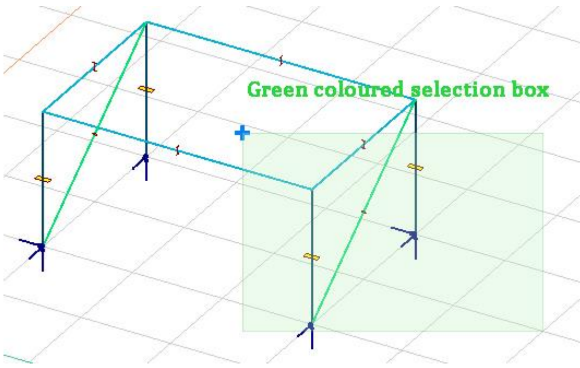
)



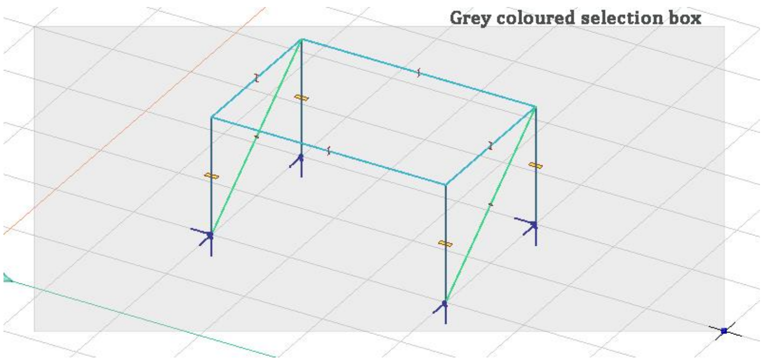

- The selection box has different colours depending on the way of the selection:
    a. By 'left-to-right' selection (the included objects get selected) the colour of the selection box is blue.




    b. By 'right-to-left' selection (all intersected objects get selected) the colour of the selection box is green.




    c. By area selection (e.g. to print the selected area) the colour of the selection box is grey.



Multiple selection can be ended by Ctrl + (it has the same effect like pressing Enter).

- For pressing Ctrl + in the application window (without any active tool palette) the quick menu appears.
- When a tool palette (e.g. Beam tool palette or Point load tool palette) is active, for pressing Ctrl + the properties of the items belonging to the active window appear.

## 9.5. Using Esc in tools

Using Esc in tools has changed in FD 14:

- Pressing 1st Esc breaks the current command (e.g. placing a structural element).
- For pressing 2nd Esc the active tool palette closes.

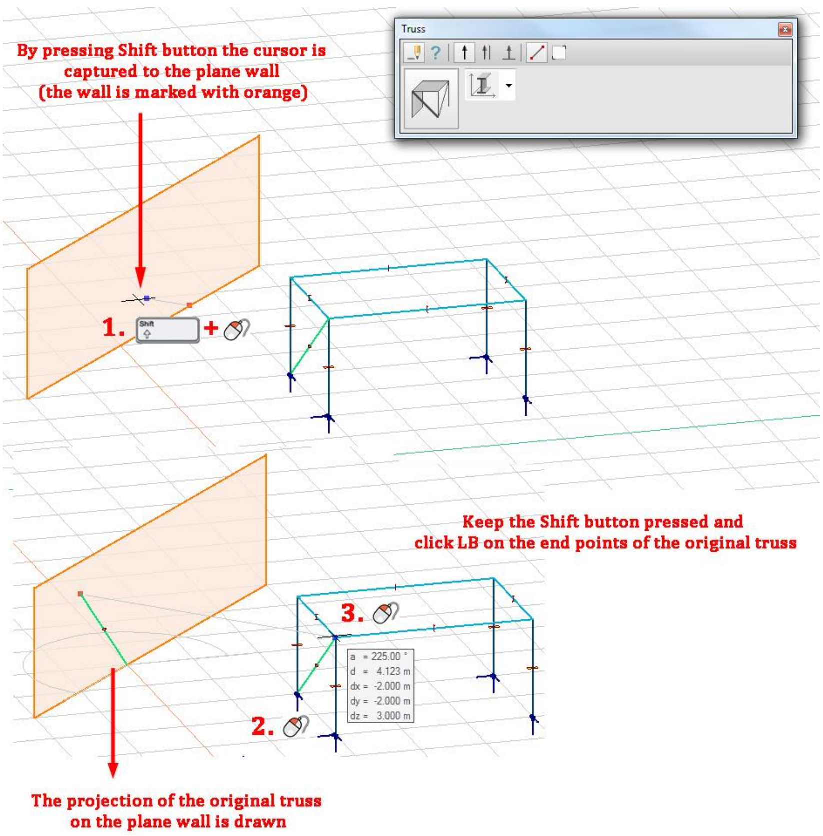
## 9.6. Using Undo/Redo

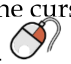
Undo/Redo command does not close the current action window only resets it.

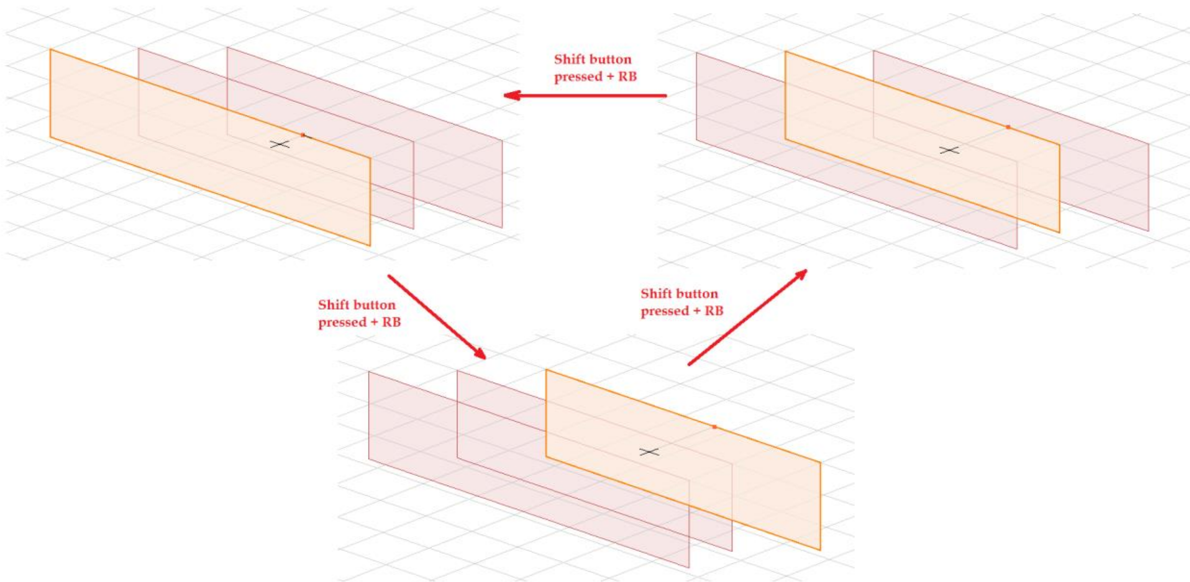
## 9.7. Editing point

Cursor can be captured by Shift not only to guide lines but to arbitrary lines or surfaces.

With the help of this feature the projection of a truss can be easily drawn on an arbitrary plane wall.

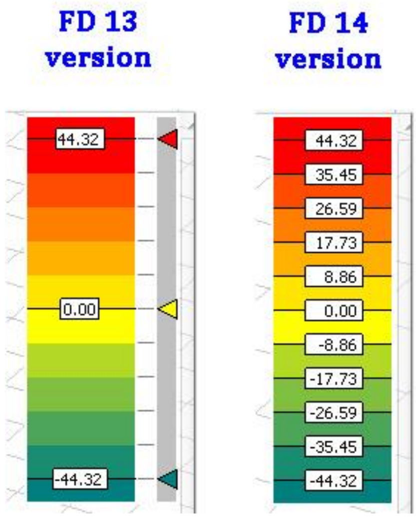


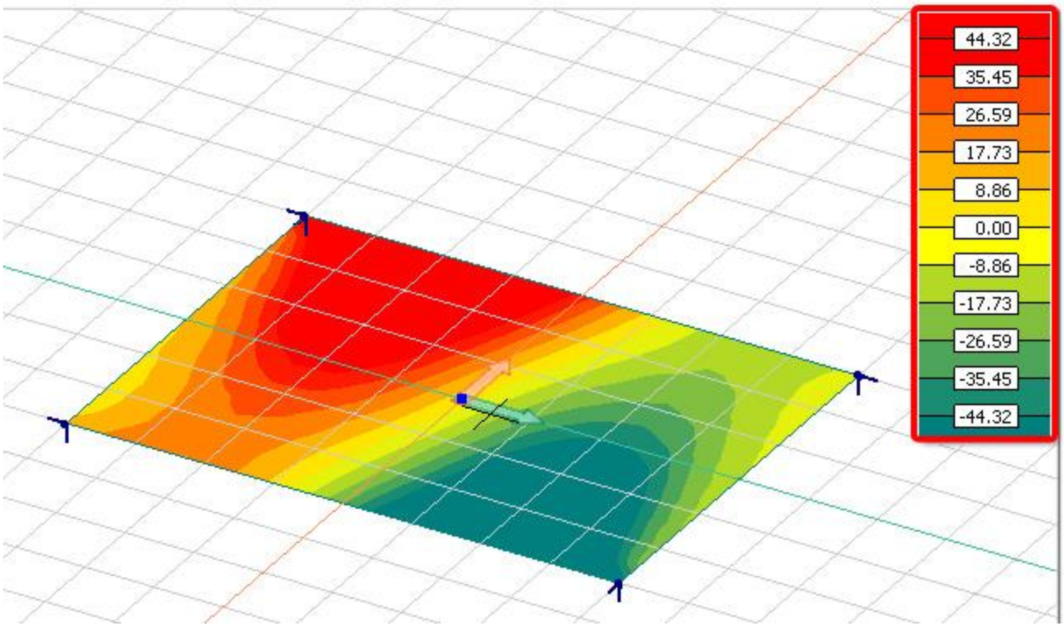
If the ray that is perpendicular to the plane of the screen and crosses the cursor intersects more than one plane, User can switch between the planes by pressing  + 



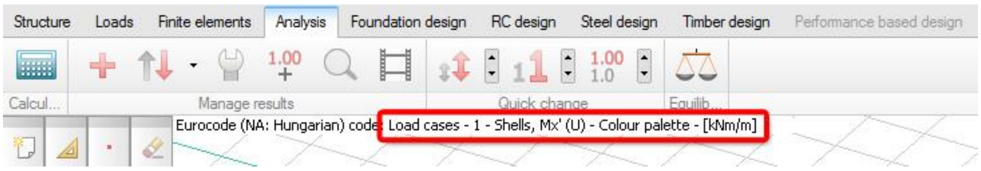

## 9.8. Display result

In FD 14 the labels of the colour palette changed. While in the previous version only the zero and the two extreme values were displayed, in FD 14 to all colour tones belongs a label.



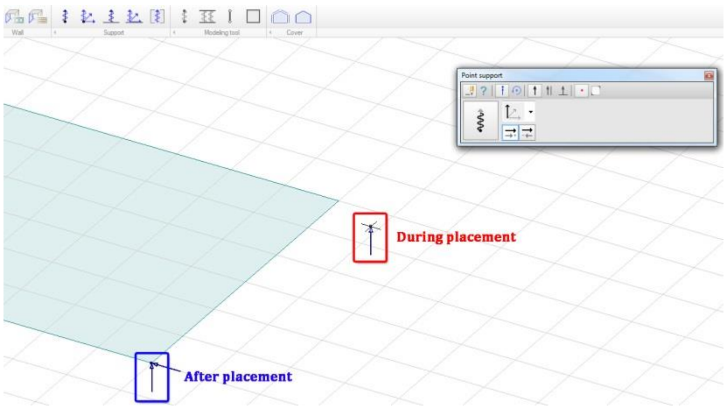



⚠ The dimension of the currently displayed result is located under result's title.



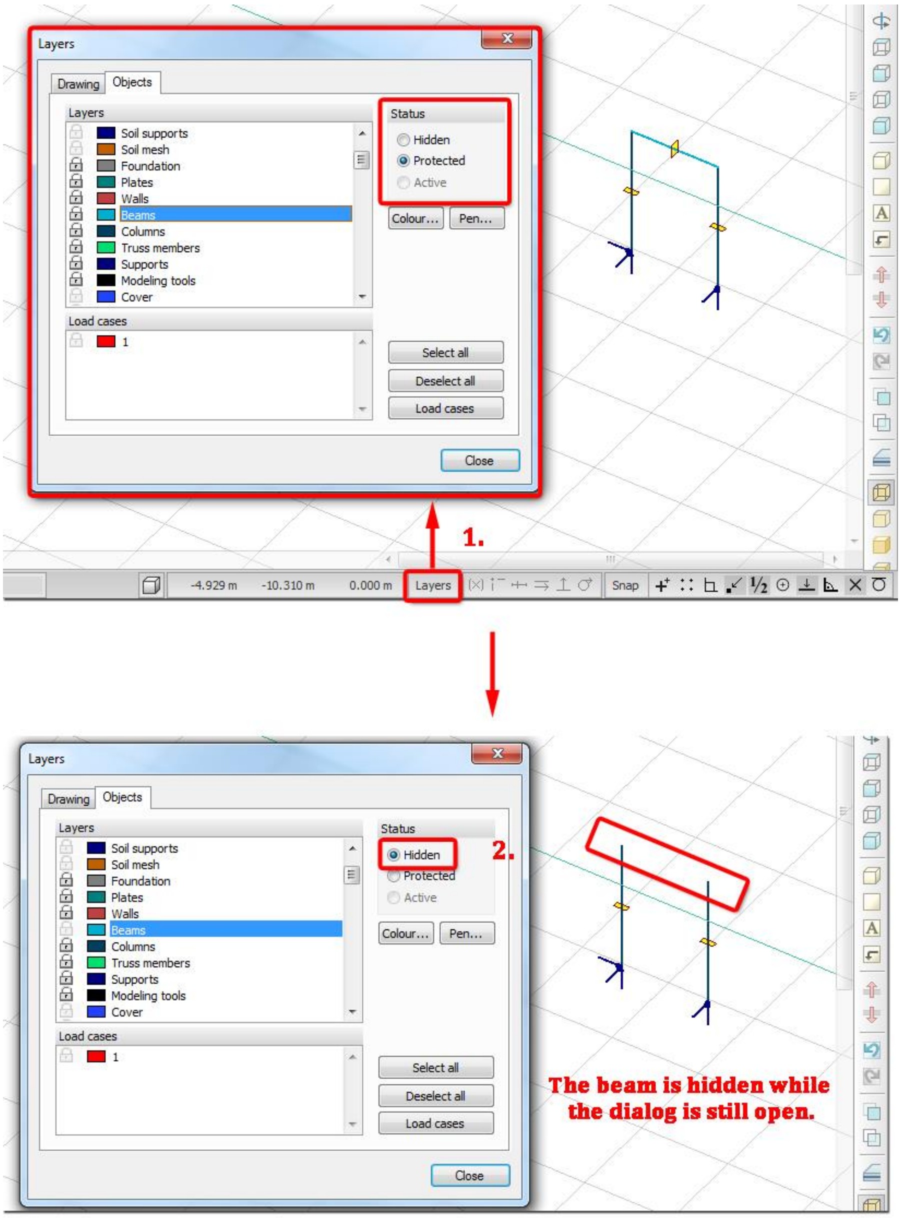

## 9.9. Cursors in "Draw" menu

Cursors in "Draw" menu and in the first two tabs (Structure and Loads) are drawn real time.




## 9.10. Layer settings

In Layers dialog changing the layer status is real-time, which means that the selected active (hidden) drawing or object layers become hidden (active) without closing the dialog.




# 10. TOOLS

## 10.1. List

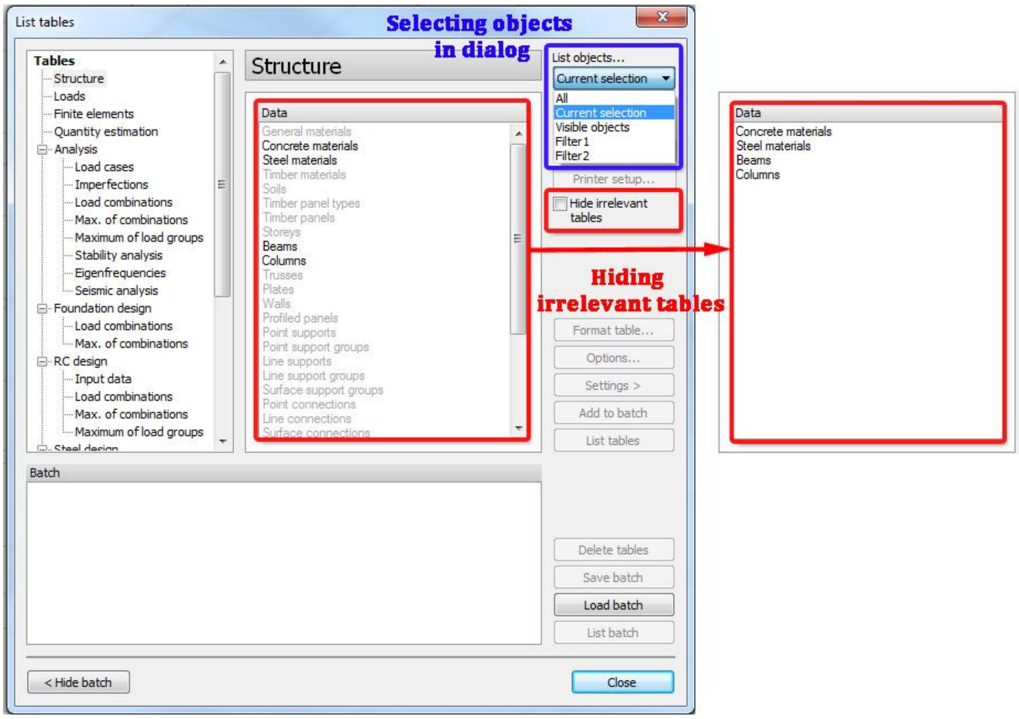
In FD 14 List dialog has improved in several aspects.

After launching the List  command, the User can select the **required objects** – using Filter and/or common graphical selection methods - before the List tables dialog appears. The whole database can be listed by pressing 

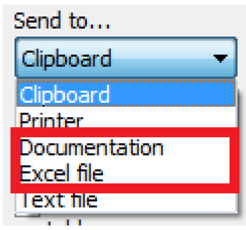
button. Selection can be carried out in the List tables dialog as well using the drop-down menu in the top right corner. The User can choose the followings:

- All (all objects of the model),
- Current selection (objects selected before List tables appeared; this option is not available when all objects were selected (by pressing Enter) after launching the List command),
- Visible objects (objects visible in the Application window),
- User defined filter (objects belonging to pre-defined filters).

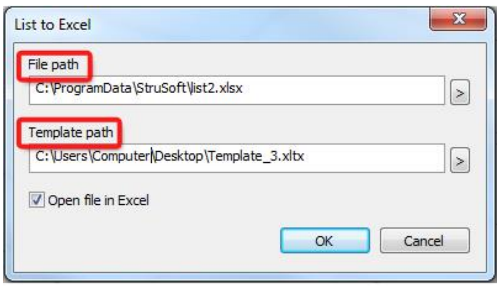
Listed data depend on the selected objects. By ticking 'Hide irrelevant tables' check box those tables which are irrelevant by the current selection disappear (otherwise they are shown in grey).




In 'Send to…' drop-down menu the destination of the listed tables can be set. Beside Clipboard, Printer and Text file, two new destinations are available:



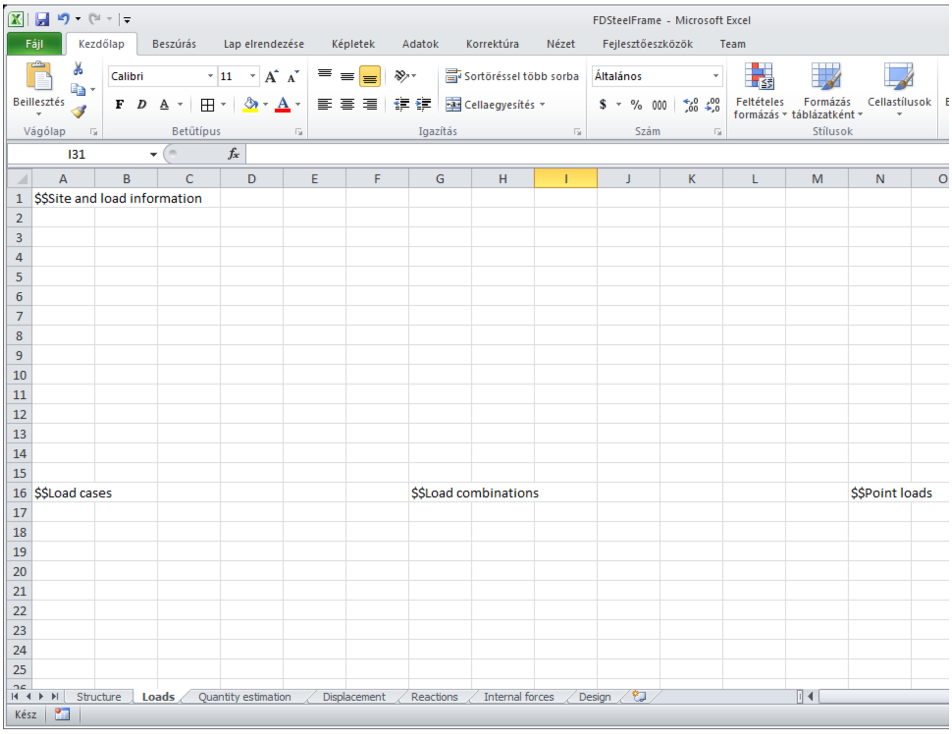

- **Documentation** (listed tables appear in the documentation),
- **Excel file** (listed tables can be exported to an Excel file)



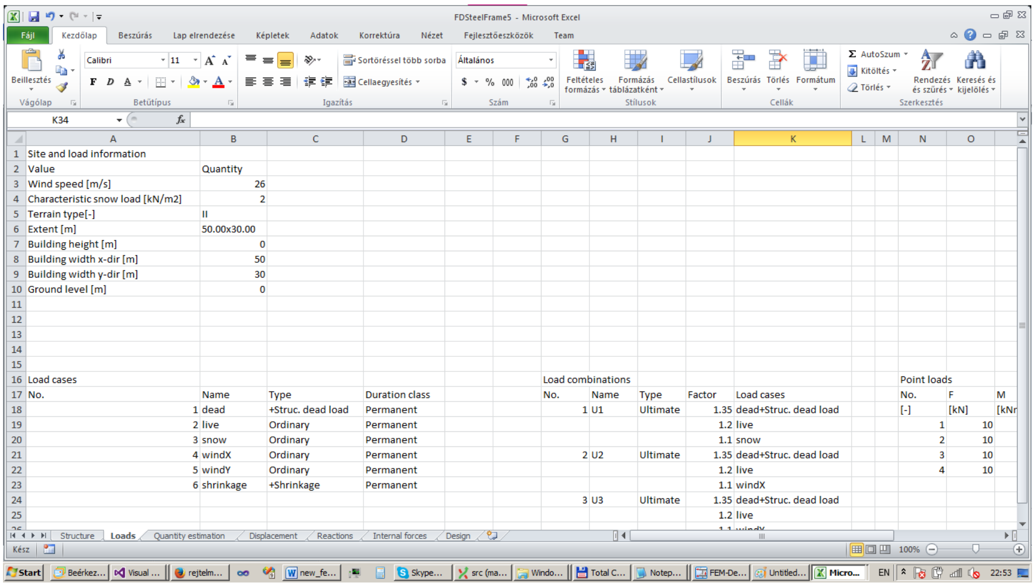
In List to Excel dialog besides the File path also a Template path can be set if the User has a pre-defined Excel-template (with .xltx, .xlt, or .xltm extension). The created Excel file can be opened immediately by ticking 'Open file in Excel' checkbox.

- Listing without a template file: creates an Excel file in which all the tables are located on separate spreadsheets.
- Listing with a template file: the User can create a template in which the locations of the required tables in the spreadsheet are marked with '$$' followed by the exact title of the table. Using a template allows the User to gain exactly the required data.

  Template file:



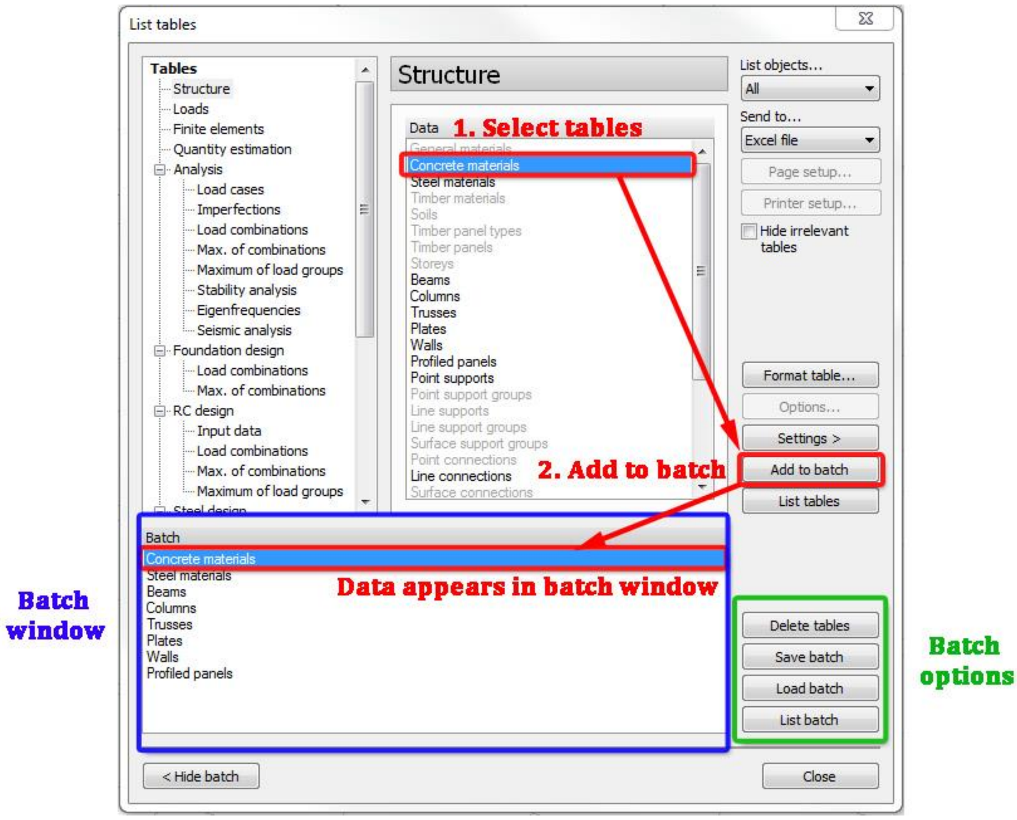

  Template-based exported file:




After listing to Excel without any template and examining the number of columns belonging to a table, it is easier to create a template.

2 batch files + 2 templates are delivered with the program (FDSteelFrame, FDRCStruct).

To avoid the time-consuming procedure of selecting the tables for listing one by one **Batch** is implemented for the tables which can be saved and reloaded, even in different models. This way all the required tables can be listed by a few click. Tables can be added to the **Batch** by selecting the tables in the "Data" window and clicking on *Add to batch* button.




The following options can be done with the batch:

- Delete tables (selected data will be deleted),
- Save batch (the batch can be saved to a batch script file with .bsc extension),
- Load batch (the saved batches can be loaded and used in other models; those tables which are not relevant in other models are displayed in grey),
- List batch (all tables in batch window will be listed in accordance with the settings of 'List objects…' and 'Send to…' drop-down menus.

The batch window and options can be hidden by clicking on <*Hide batch* button.

In batch window more data can be selected by holding pressed or with the help of Ctrl and Shift buttons.

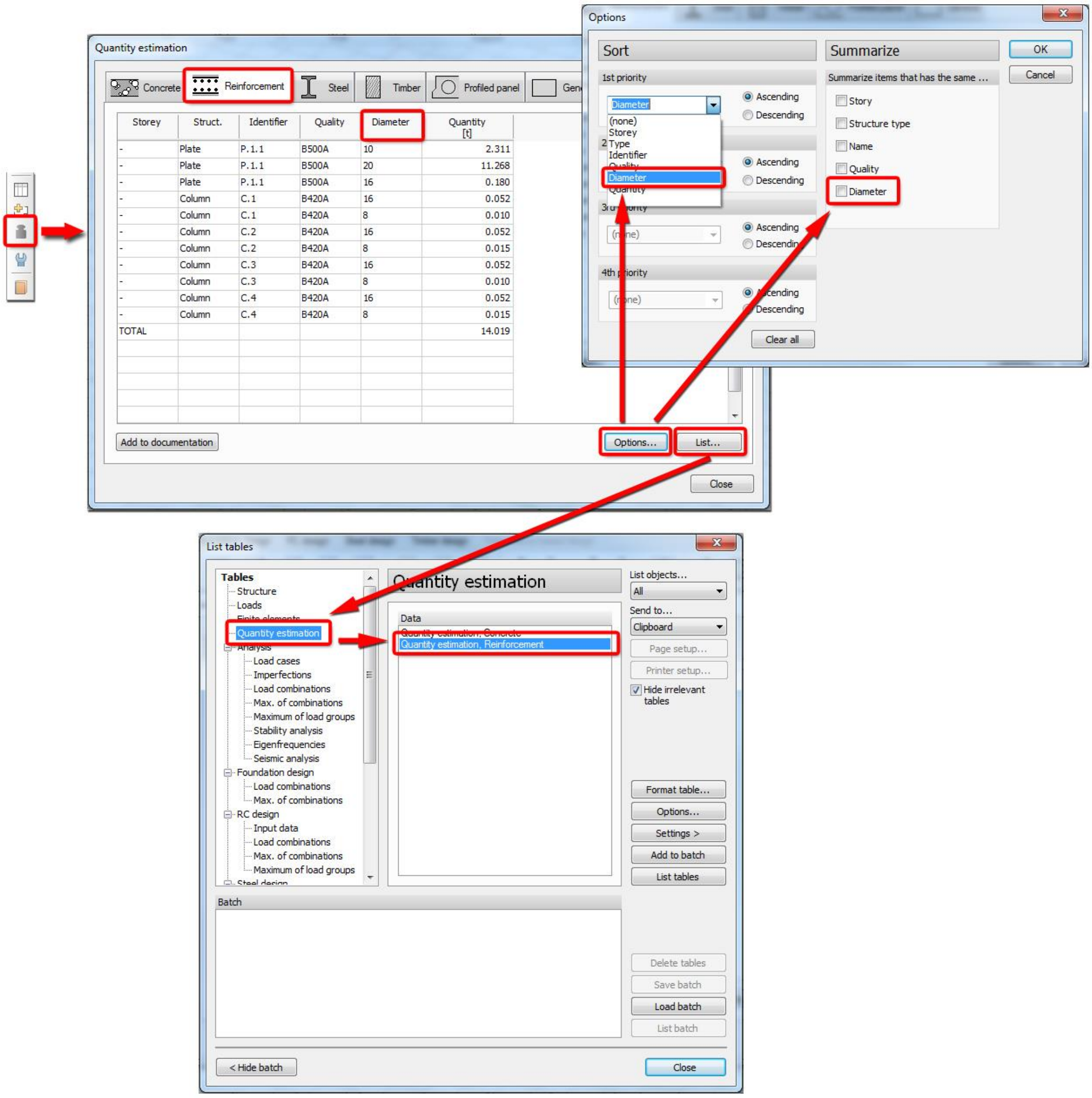
In a listed batch the order of the tables is the same as in the batch window. The order of the tables in batch window can be rearranged with Alt pressed and using .

A saved batch file with load cases (or load combinations, stability analysis etc.) can be used in other models when the names of the load cases (or load combinations, stability analysis etc.) are the same due to that the identification is based on the name. If all load cases (or load combinations, stability analysis etc.) are selected in the batch window and the batch file is loaded in another model, all the load cases (or load combinations, stability analysis etc.) of the other model will be taken into account.

## 10.2. Quantity estimation

In *Quantity estimation*  dialog diameter is added to reinforcement quantity estimation. From now in Options menu the items can be sorted or summarized by diameter as well as by the already existing parameters (Storey, Structure type, Name, Quality, Quantity).

Diameter is also added to the Table of reinforcement quantity estimation which is available after clicking on *List…* menu.



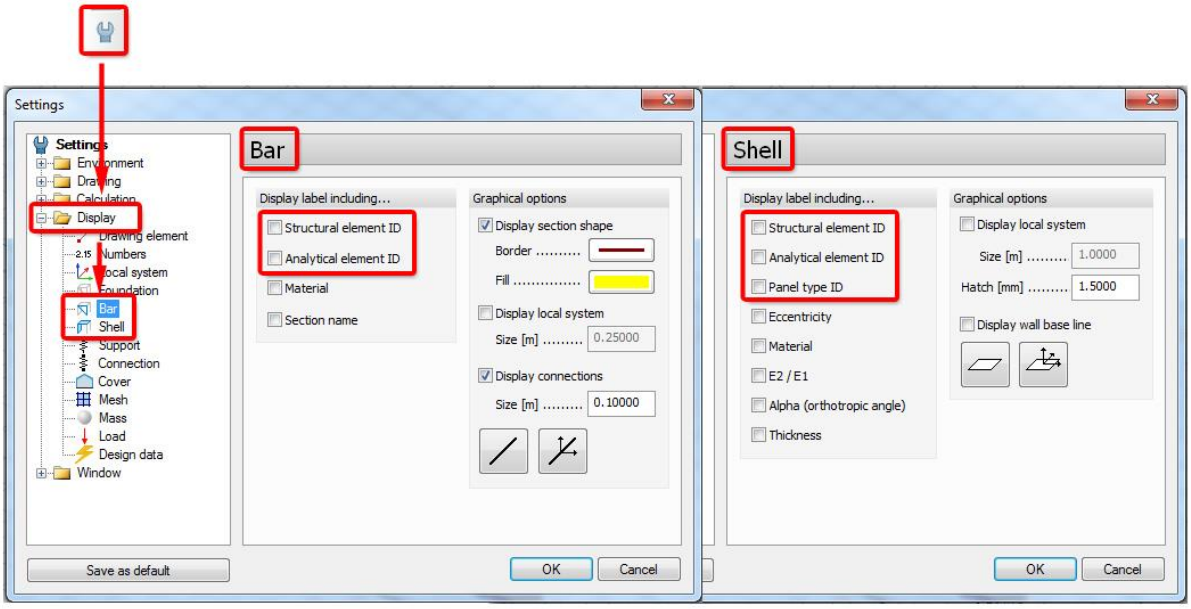

Reinforcement quantities are displayed after RC design only.

The table can be easily added to the documentation with the *Add to documentation* button.

# 11. SETTINGS

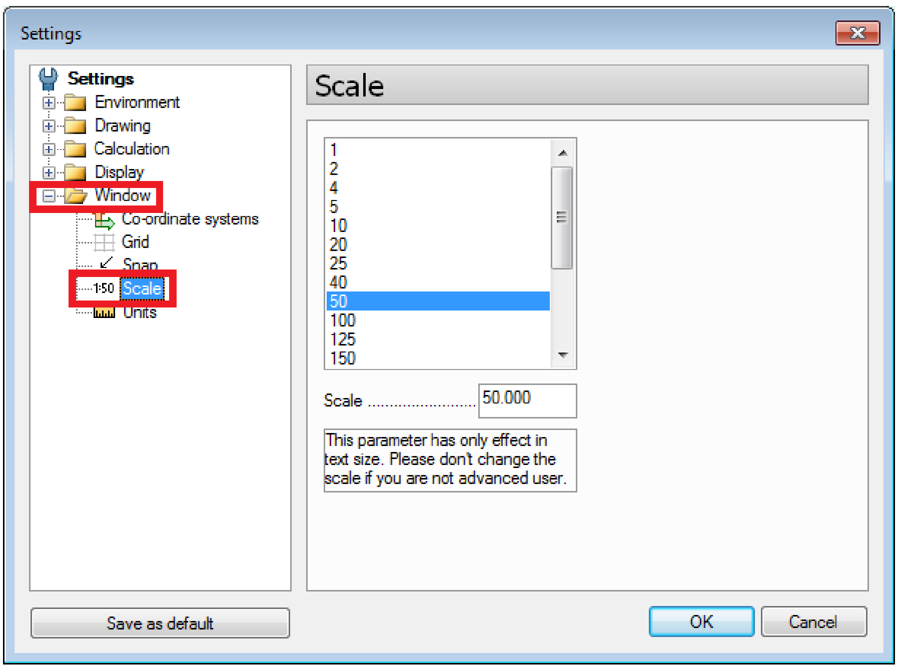
## 11.1. Displaying element ID

Displaying structural and analytical element ID's is separated in order to avoid duplicate labels on the screen. The required ID can be set by *Bar* and *Shell* elements in Display settings dialog.

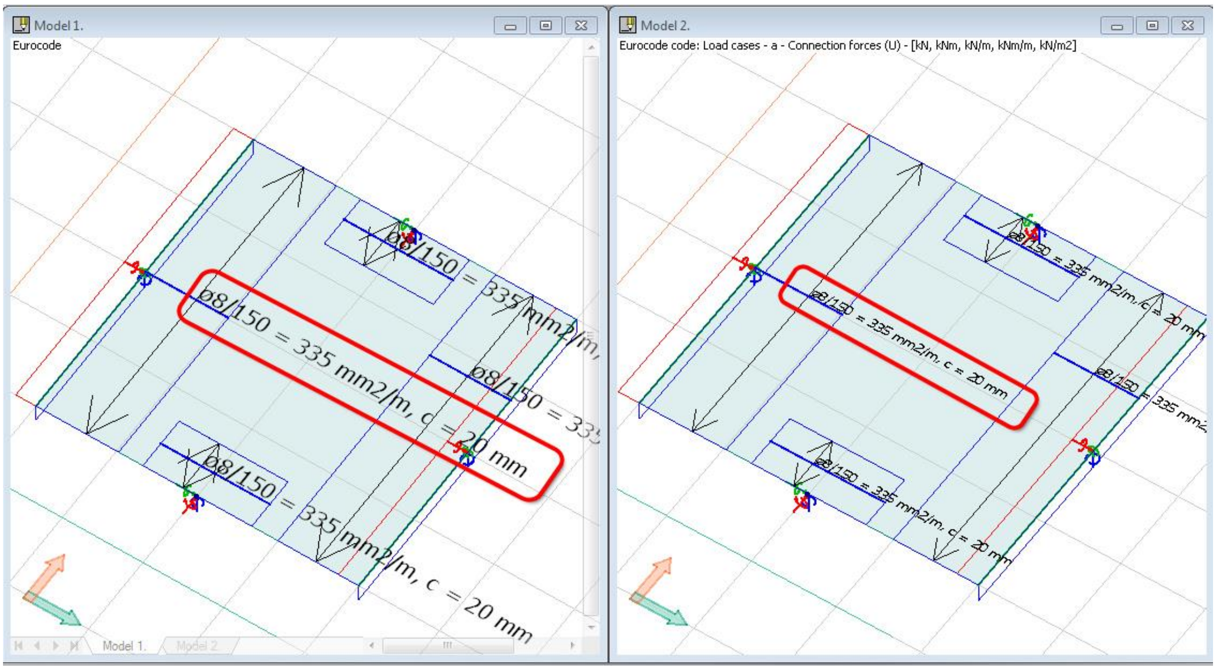



## 11.2. Scale options

In different windows different scales can be defined by the User, which can be specially useful in documentation module.




E.g. by displaying the results of a reinforced concrete plate once it can be necessary to add a figure to the documentation in which the parameters of the reinforcement are easy to read. In other case the parameters can be decreased to display another result.


Model 1.
Eurocode
ø8/150 = 335 mm2/m, c = 20 mm
Model 1.
Model 2.
Model 2.
Eurocode code: Load cases - a - Connection forces (U) - [kN, kNm, kN/m, kNm/m, kN/m2]
ø8/150 = 335 mm2/m, c = 20 mm