AUTODESK UNIVERSITY 2014

Autodesk® Robot™ Structural Analysis: Steel and RC Design

Artur Kosakowski – Autodesk

SE54225

In this class we will cover the workflow for design of steel and RC structures starting with building the best possible model for a task followed by assigning correct design parameters and finding the optimal steel section sizes or reinforcement arrangement.

Learning Objectives

At the end of this class, you will be able to:

- Create a model of a structure that matches needs of the Steel and RC design modules of Autodesk Robot Structural Analysis Professional
- Define loads and load combinations to be correctly recognized in the design modules
- Apply principles of steel and concrete design modules and understand the influence of defined parameters on the design process
- Run concrete design of elements of a model

About the Speaker

Artur is a Structural Engineer. He graduated from Cracow's University of Technology. He has been working as a Support Specialist since 1996 at Robobat and since 2008 at Autodesk supporting Robot's users worldwide.

Introduction

Regardless of what kind of a structure you are about to design it is extremely important that you create its model in the manner that allows you to assign the correct design parameters to each of its elements. This document will not be a step by step guideline to follow but it will show you the ways for having models which are coherent with the design modules of RSA as well as indicate the rules that govern both steel and RC design in the program.

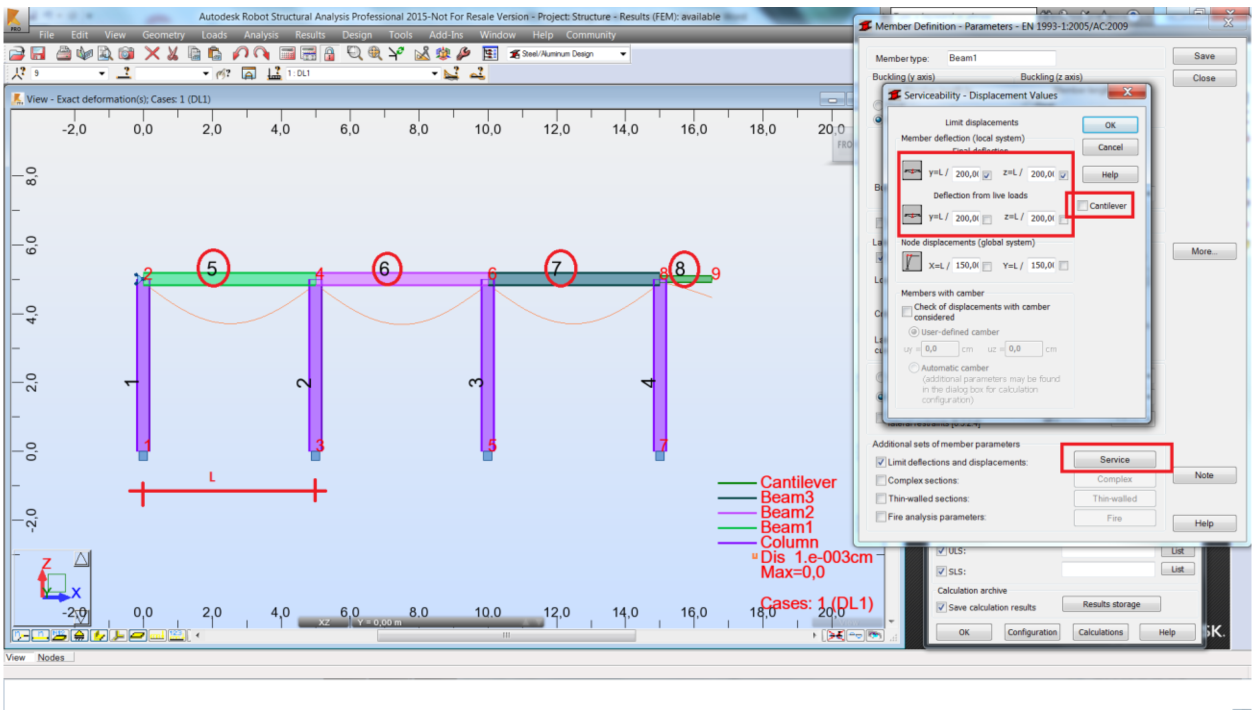
The topics discussed in this presentation are based both on my own experience as well as threads from Robot forum which I would like to encourage you to use as a platform to discuss them further or the place where you can both find help for your challenging issues and help others solve theirs.

1. Building a model

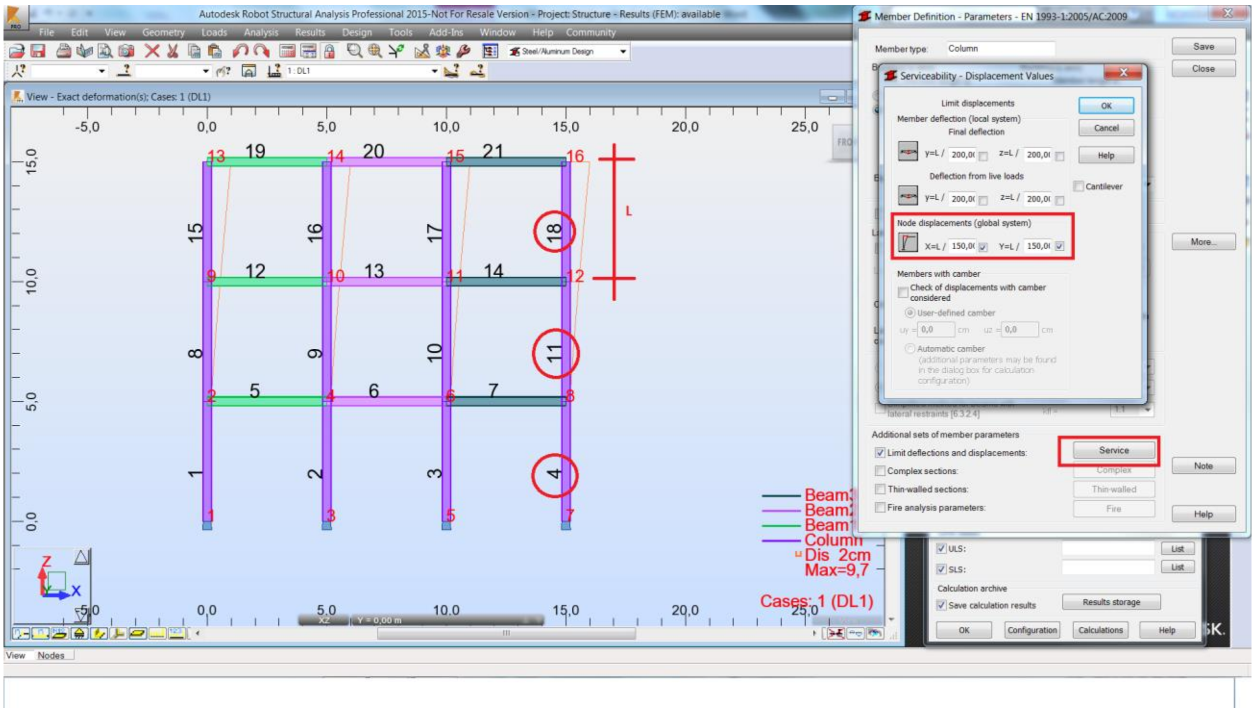
a) For Steel design

- Define bar elements in such a way it is easy to assign design parameters. Consider their role in a structure and design criteria

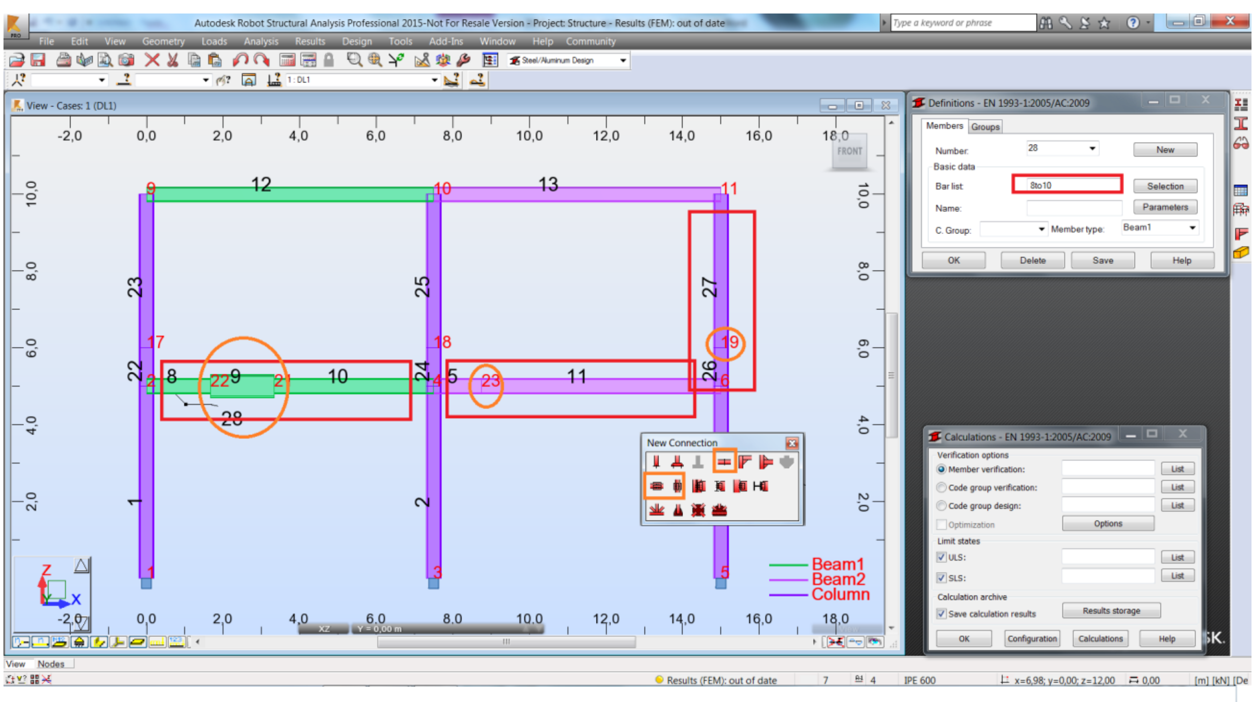
- Create multi span beams as single elements among supports (columns) rather than single elements and cantilevers as separate bars so that it is easy to define SLS (deflection) limits



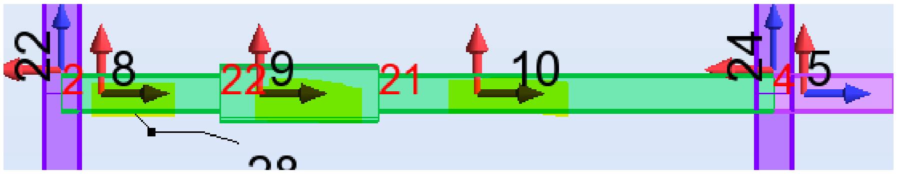
- Enter columns as single elements instead of being defined through the entire height of a structure which allows for both defining their displacement limits as well assigning them to the stories (seismic analysis)



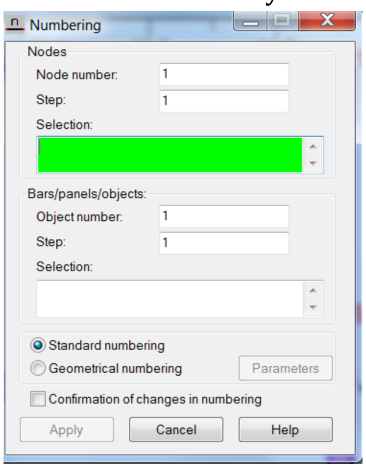
- Use "superbars" in case there is a need to split columns and beams between supports e.g. to define a connection or to change its section size (e.g. increase thickness of flanges of I section) on a part of its length



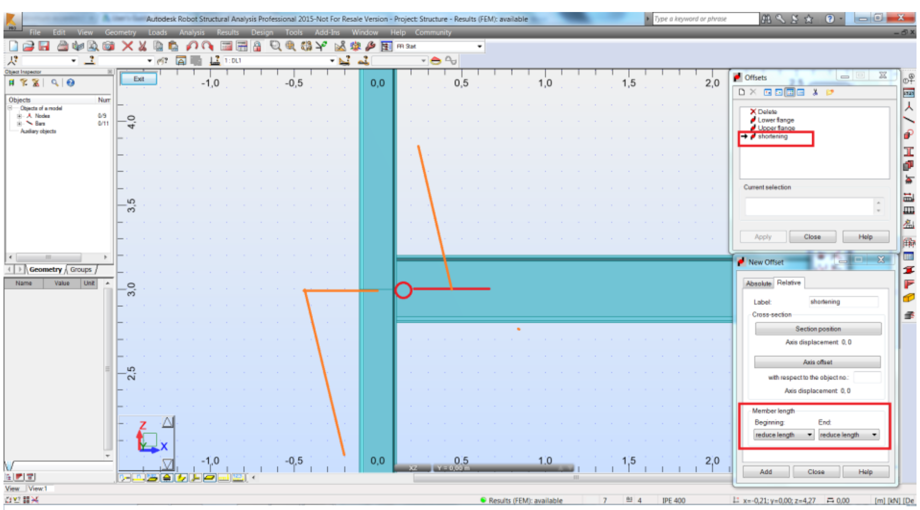
Mind that the local X axes of components of a "superbar" should point in the same direction.



Mind that the orientation of a "superbar" doesn't follow the direction of the local coordinate systems of its components. It is governed by the number of its origin and end node instead (lower node number is assumed as the start whereas the higher one as the end). To replace the origin with the end it is necessary to renumber the nodes which can be done using the numbering option.

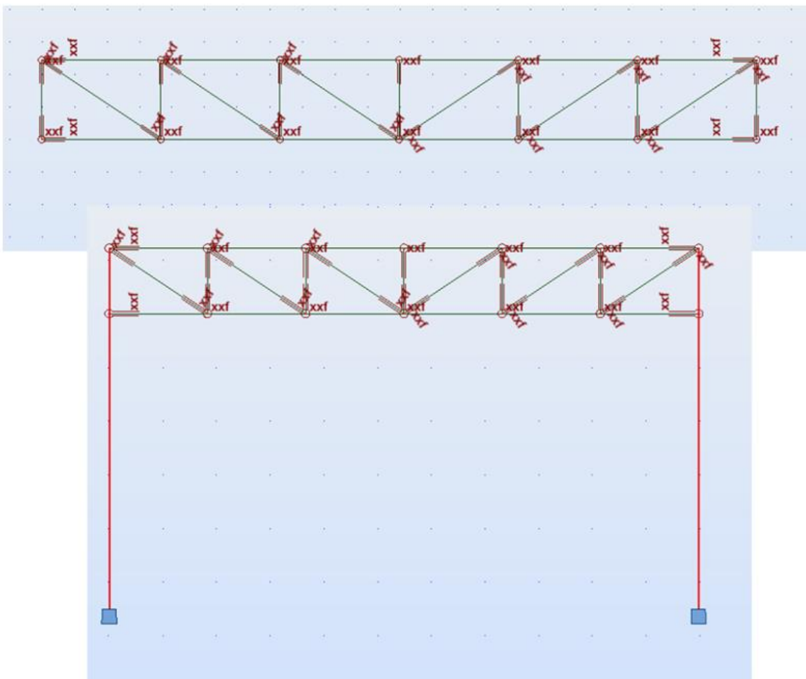


- Use offsets (bar shortening) to generate additional bending moments on columns which originate from pinned connections of beams to their flanges

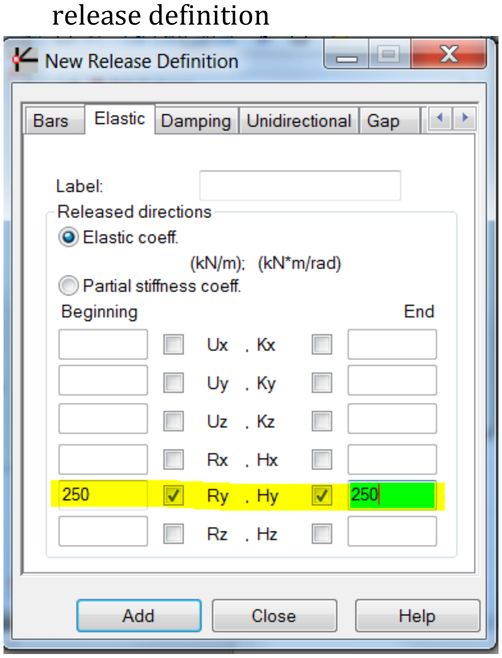


The relative definition of the offset for a member length automatically adjusts its value to a change in the size of a column.

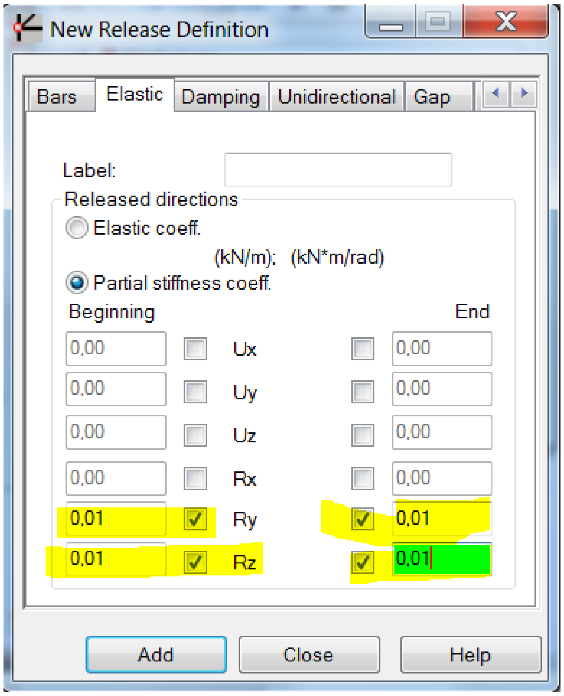
- Trusses from library structures – to avoid instabilities it is recommended to create continuous chords and use releases or truss bar definitions on posts and diagonals only. It should be remembered to delete side posts which for typical situations overlap with already defined columns



- To define partial releases which represent real stiffness of connections you can use elastic release definition



A similar approach (partial releases) may be used to obtain additional bending in pinned connections among elements of a model for having safety margin.



Additional references to the discussions on these topics on the Robot Forum:

http://forums.autodesk.com/t5/robot-structural-analysis/member-consisting-of-multiple-bars-for-design/td-p/3467536

http://forums.autodesk.com/t5/robot-structural-analysis/steel-design-divided-bar-2-identical-structures-one-pass-and-one/td-p/3535282

http://forums.autodesk.com/t5/robot-structural-analysis/dimensionnement-des-barres-acier/td-p/3750349

http://forums.autodesk.com/t5/robot-structural-analysis/lateral-buckling-parameters/td-p/3536254

http://forums.autodesk.com/t5/robot-structural-analysis/minimum-eccentricity-for-steel-simple-columns/td-p/3657566

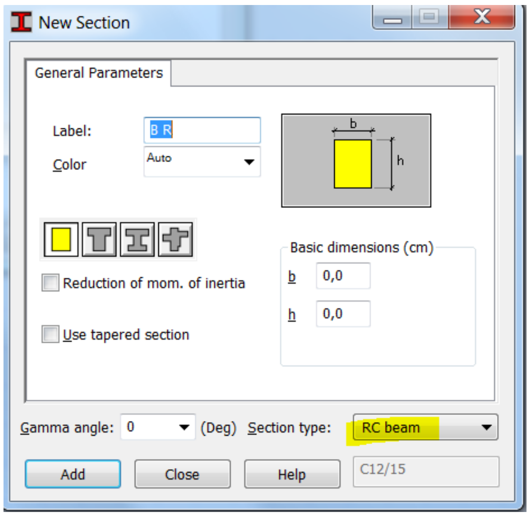
http://forums.autodesk.com/t5/robot-structural-analysis/beam-offsets/td-p/4344866

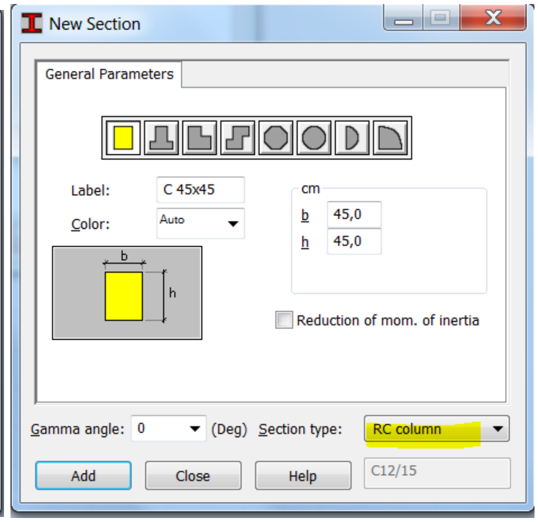
http://forums.autodesk.com/t5/robot-structural-analysis/truss-design/td-p/3596600

http://forums.autodesk.com/t5/robot-structural-analysis/steel-design/td-p/3573540

b) For RC design

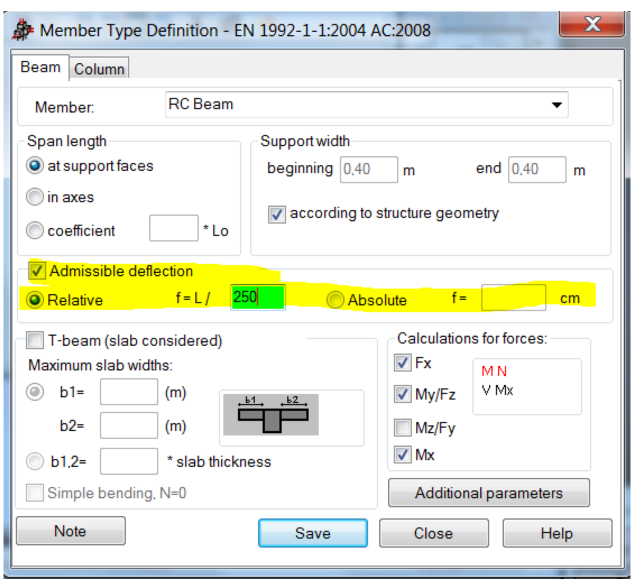
- Define bar elements with section types supported in design modules (neither use an RC Beam section for or an RC Column definition and vice versa nor use steel ones for either of them)



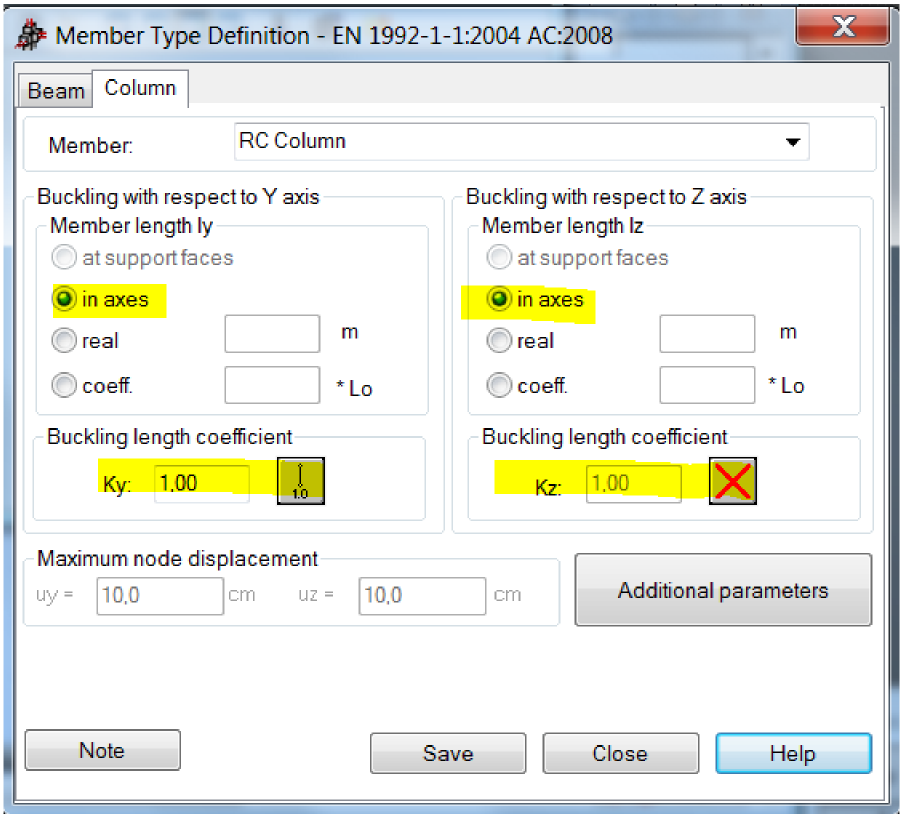


- Define bar elements and surfaces in such a way it is easy to assign design parameters. Consider their role in a structure and design criteria.

- You can create multi span beams as single elements among supports (columns) or as a single bar between the outside ones when you want to design them in the RC Beam Design module as it automatically detects spans but define spans of beams as separate bars if you intend to calculate required area of reinforcement inside the RC Member Required reinforcement module to be able to correctly define deflection limits



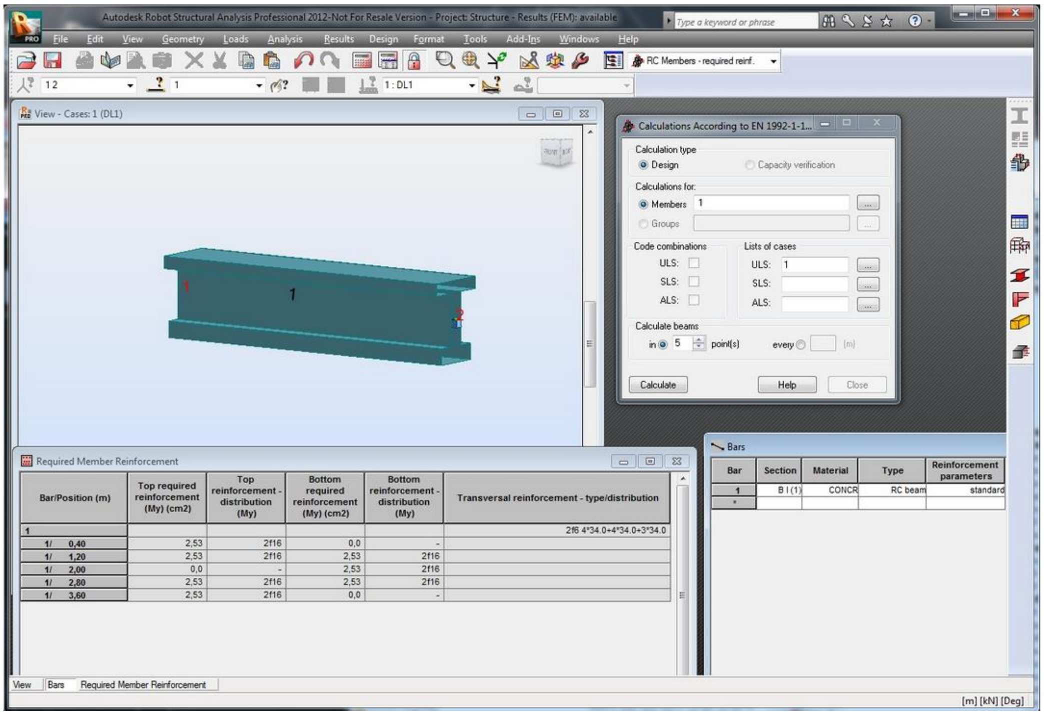
- Enter columns as single elements instead of being defined through the entire height of a structure which allows for both defining their buckling parameters as well assigning them to the stories (seismic analysis). In addition you may find the access to their results easier especially in the Forces table.



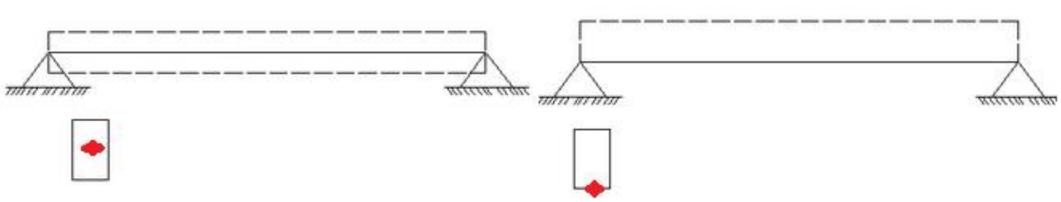
- Do not divide columns and beams between supports or stories (levels of beams) into smaller parts as such chains of elements are not supported correctly in the RC design module (with the exception of RC Beam Design one)

- To define RC Columns of different sizes at top of each other you can use offsets or (recommended) rigid links. If there is a slab "between" them you can define them in their real positions as they (top node of a bottom one and bottom node of a top one) will be connected by mesh elements of the slab.

- I shape RC Beams can only be designed in the RC Member Required reinforcement module.

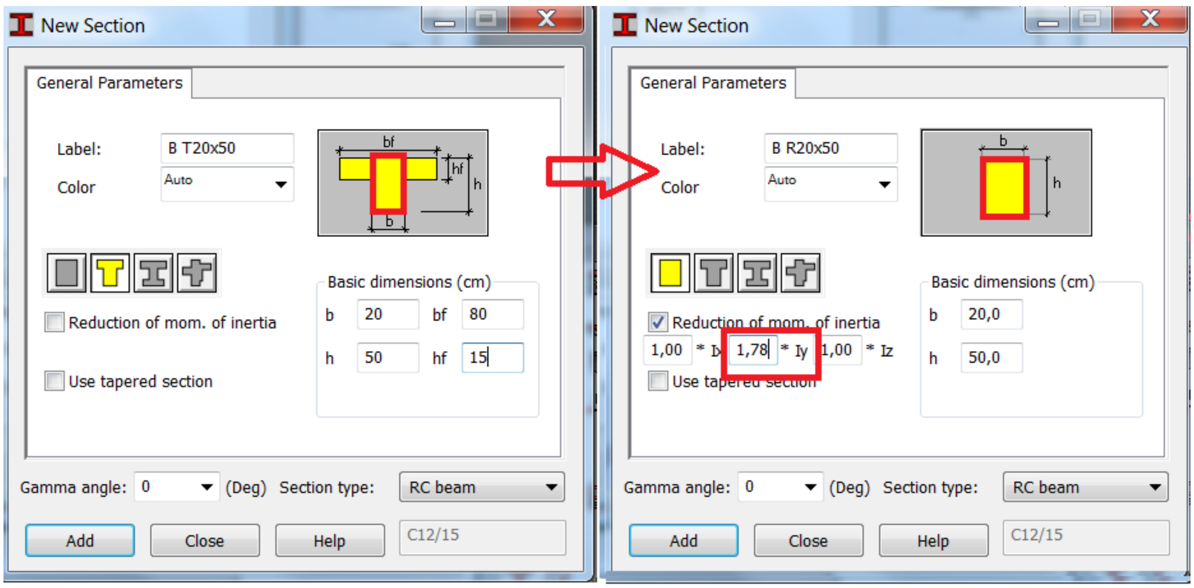


- It is not recommended to use vertical offsets for RC Beams you want to design reinforcement for as they influence values of internal forces causing large increase of the axial force (tension) and significant reduction of the bending moment

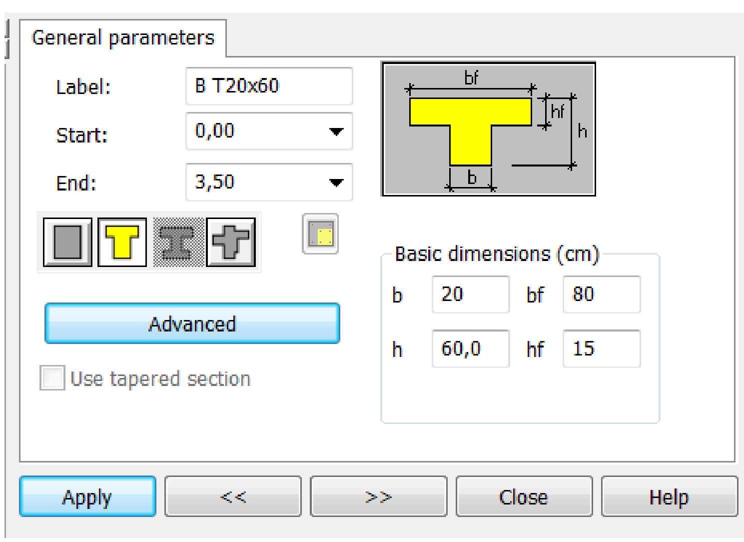


- To design a T shape beam under a slab (panel) without using the offsets (for the reasons explained above) you may follow these steps:

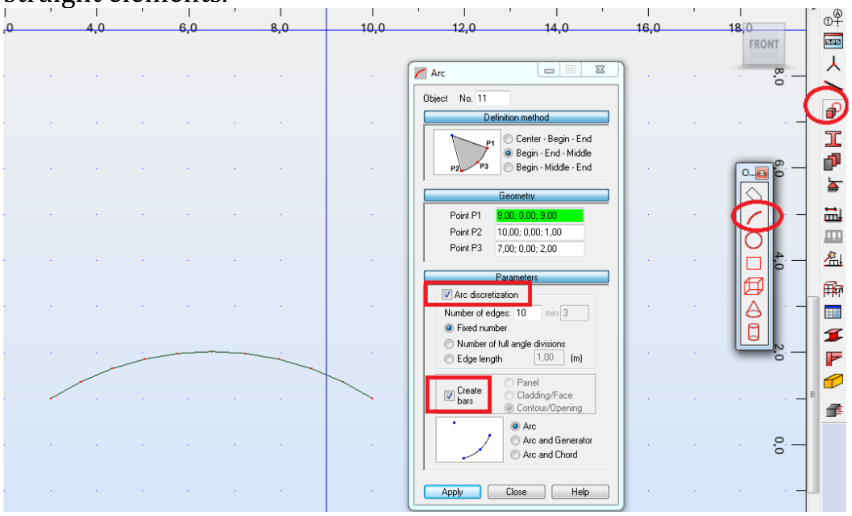
- define a beam with a rectangular cross section that represents T shape "web" with no offset
- increase its IY moment of inertia so that it is the same as the T shape it "replaces"



- replace the rectangular section with the "original' T one after exporting a beam from a model to the RC Beam Design module

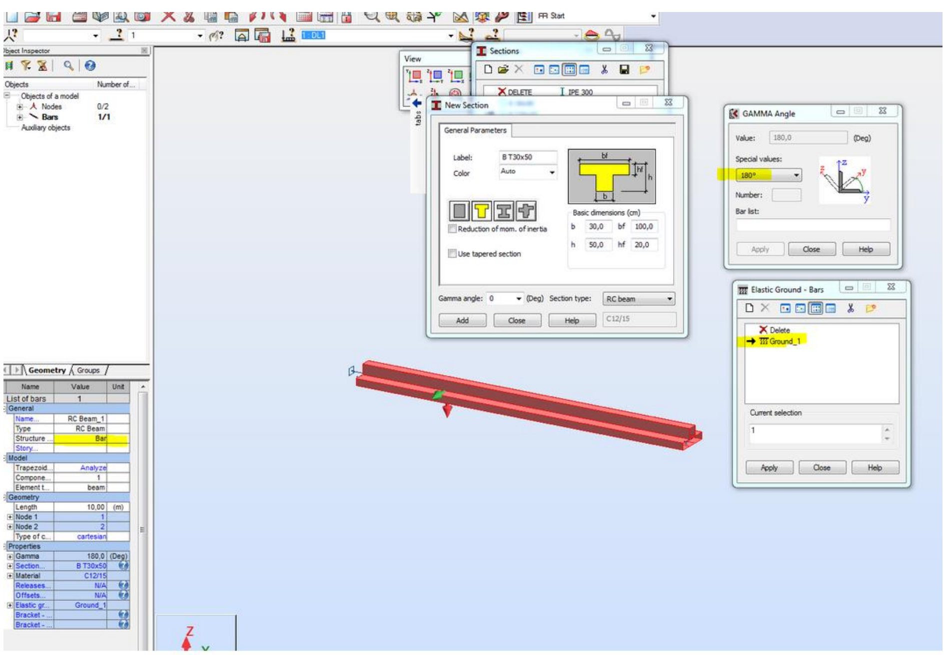


- In order to model curved RC beams you need to approximate them with number of smaller straight elements.

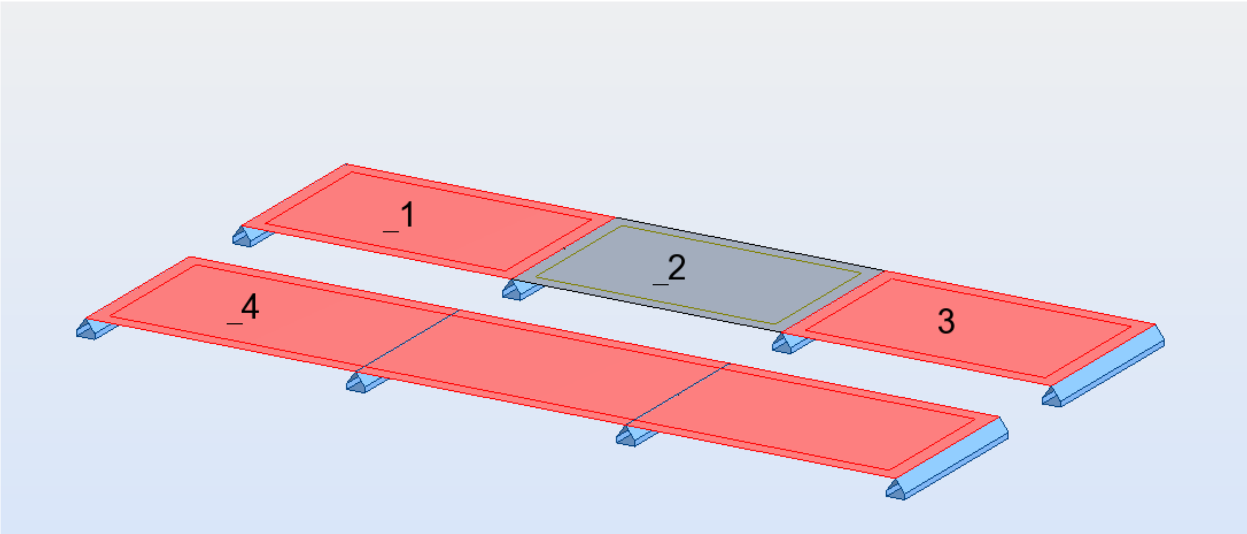


Then you can calculate required area of reinforcement in the RC Member Required reinforcement module.

- In order to correctly export a bar element from a model that represent a strap footing to the RC Continuous Footing (rather than to RC Beam) design module you should:
  - Define elastic soil as its attribute
  - Set it Structure object as Bar
  - In case of a T shape cross section assign it 180° (Gamma) rotation angle

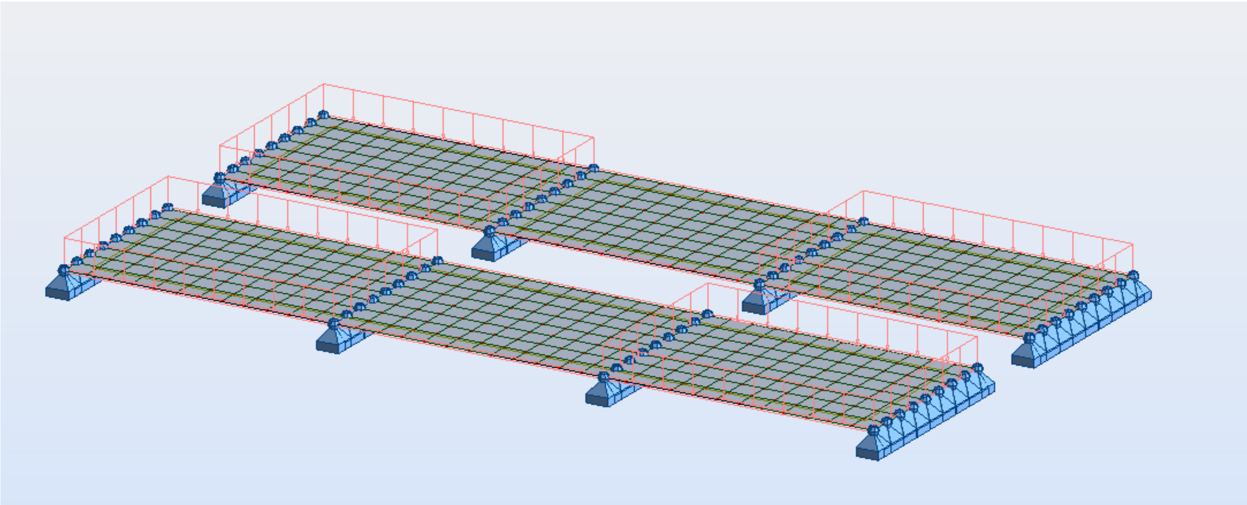


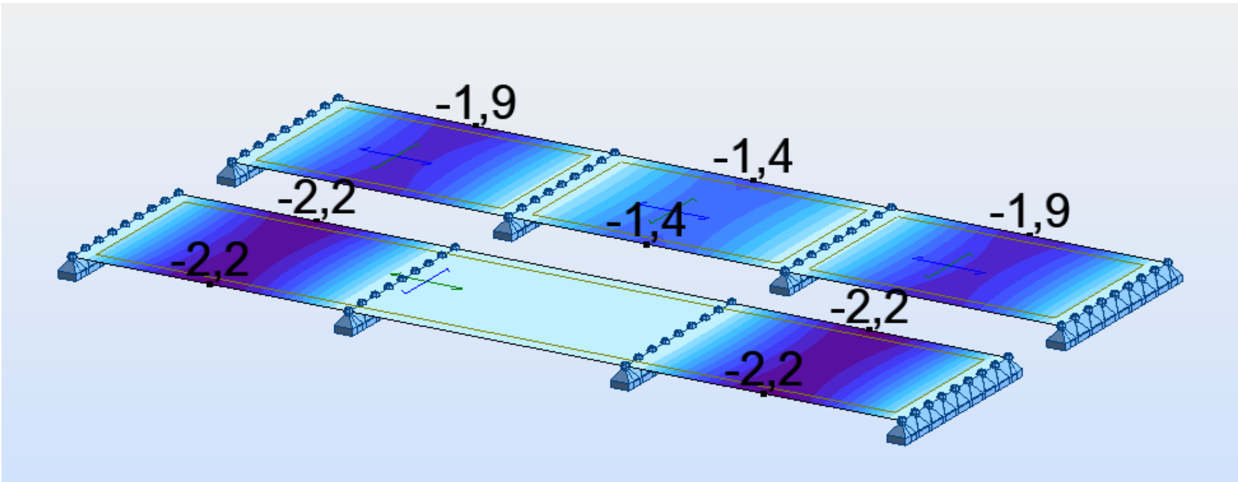
- To obtain correct values of "cracked" deflection of a multi span slab with use of the equivalent stiffness method you should create number of smaller panel among the support lines rather than a single one "covering" the entire shape of the floor.



There are two main reasons for such way of creating a model:

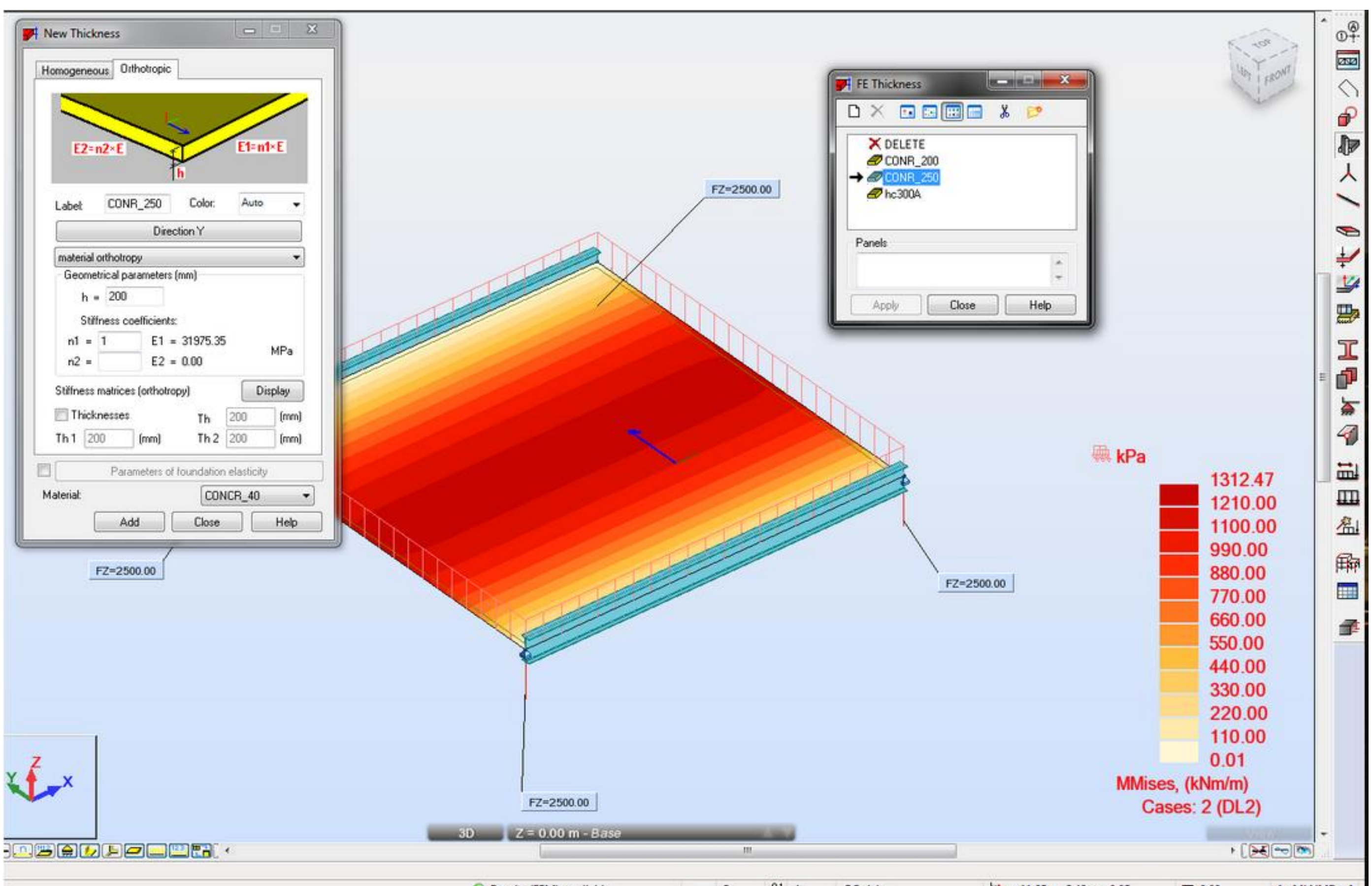
- Equivalent stiffness of RC plate (see: Help) is calculated for each of the "spans" (panels) separately which is important for different geometries and loads on each of panels
- Scaling of elastic displacements is done for load which causes maximal deformation for given panel rather than for the load which causes maximal displacements from the point found in the entire floor.





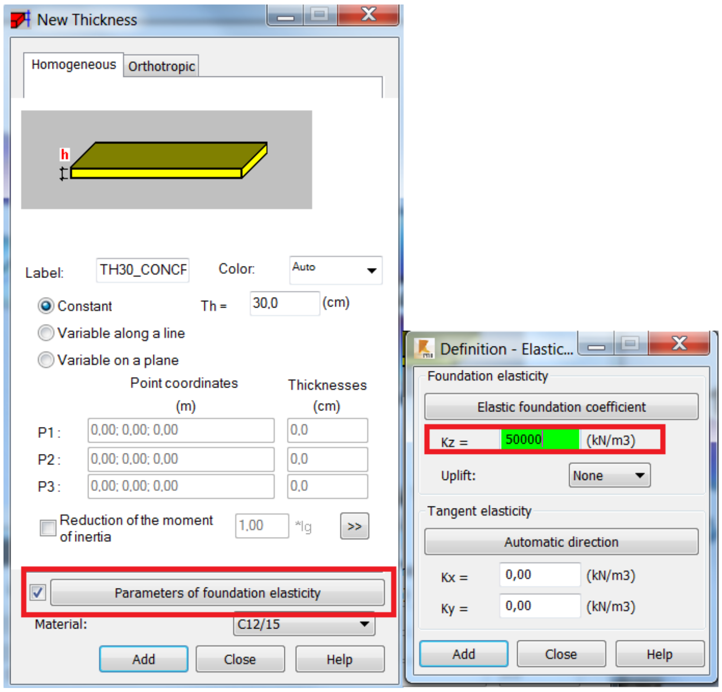
Otherwise the stiffness update method should be used instead.

- To obtain unidirectional behavior of a slab in a model you may use the orthotropic thickness definition instead of the homogenous one and reduce value of Young modulus in the direction the slab is not supposed to “work”

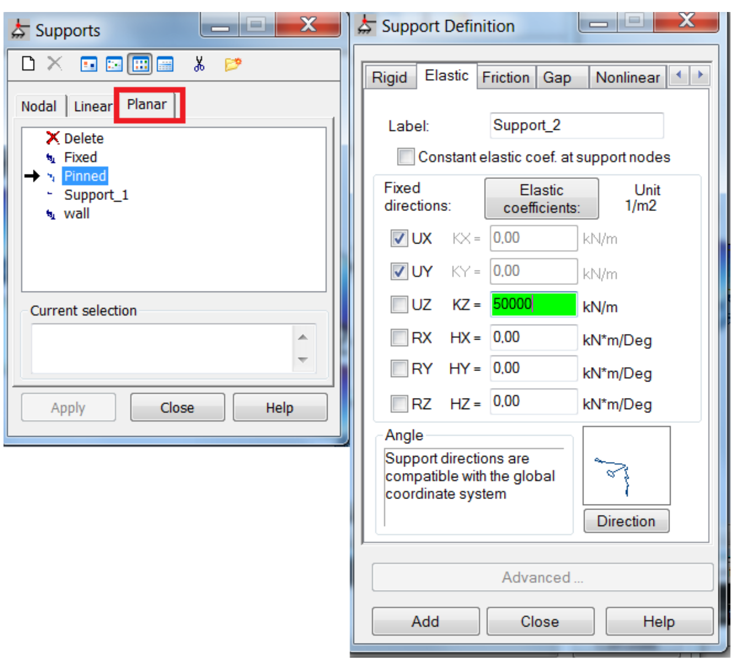


- To define raft foundation you should you should define elastic soil as either:

- parameter of its thickness (recommended for it allows for display of stress in soil as a map)



- or elastic planar (surface) elastic support. Mind not to mark the Constant coefficient check box to allow the stiffness of each support generated in panel's node to be automatically calculated based on tributary area of elements of a mesh.



Do not use both of them at the same time as the defined soil stiffness will add up.

Additional references to the discussions on these topics on the Robot Forum:

http://forums.autodesk.com/t5/robot-structural-analysis/buckling-length-coefficient-in-concrete-structure/td-p/3092910

http://forums.autodesk.com/t5/robot-structural-analysis/column-height-should-i-stop-at-slabs-or-go-all-the-way-down/td-p/3234202

http://forums.autodesk.com/t5/robot-structural-analysis/i-beam-section-choice-in-rc-elements-beam-cross-section-menu/td-p/3251898

http://forums.autodesk.com/t5/robot-structural-analysis/consequence-of-axis-position-offset-excentrement/td-p/3268158

http://forums.autodesk.com/t5/robot-structural-analysis/poutre-en-arc/td-p/3396489

http://forums.autodesk.com/t5/robot-structural-analysis/how-to-model-a-curved-beam-or-truss/td-p/3534426

http://forums.autodesk.com/t5/robot-structural-analysis/t-slab/td-p/3183798

http://forums.autodesk.com/t5/robot-structural-analysis/beam-offsets/m-p/3332921

http://forums.autodesk.com/t5/robot-structural-analysis/continuous-footing/td-p/5323957

http://forums.autodesk.com/t5/robot-structural-analysis/meshed-slab-supported-by-beams/td-p/3167096

http://forums.autodesk.com/t5/robot-structural-analysis/floors-span-direction/td-p/3349655

http://forums.autodesk.com/t5/robot-structural-analysis/sizing-slab/td-p/3280315

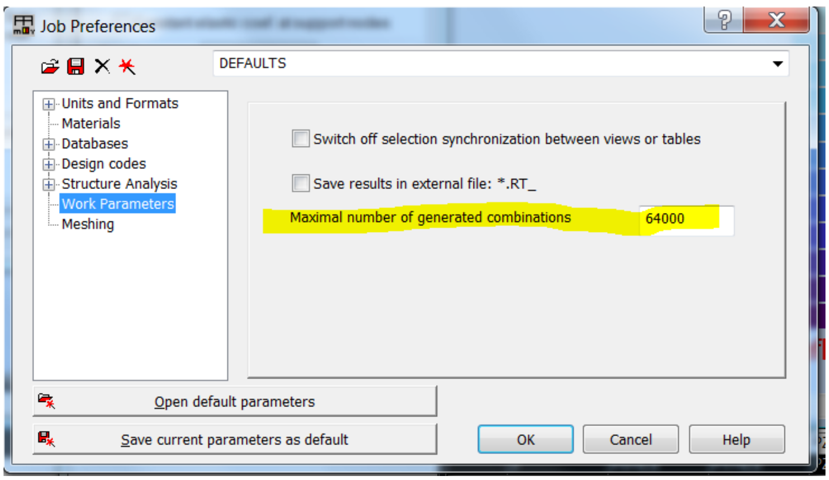
http://forums.autodesk.com/t5/robot-structural-analysis/mat-foundation-support/td-p/3899458

http://forums.autodesk.com/t5/robot-structural-analysis/aligment-of-two-column-elements-with-different-secction/td-p/4778935

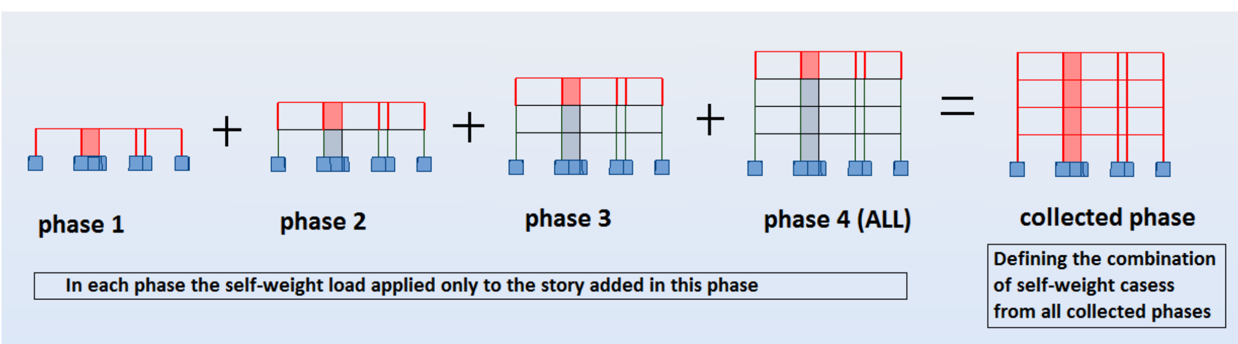
2. Defining loads

- Live load applied on a multi span beam or slab defined in the model of a structure acts as applied (there is no automatic pattern load distribution). In case this is needed it should be replace with number of live loads defined on each of spans separately. It is suggested to do so only for currently designed level (story) or do so for a substructure saved based on the selection of a part (e.g. floor plus walls and column above and below it) of the entire model

- In case of multi-story building limit number of combinations to a reasonable value. The limit of automatically generated combinations can be entered in Job Preferences



- In order to avoid the effect of unrealistic shortenings of RC Columns under self-weight of higher stories you can apply load in stages (phases)

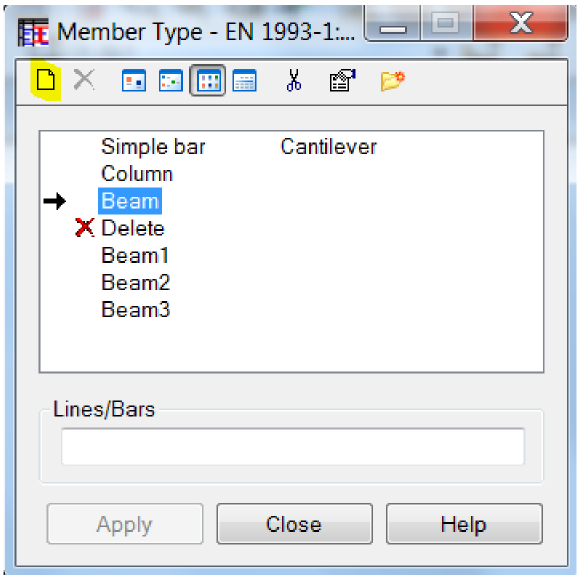


Additional references to the discussions on these topics on the Robot Forum:

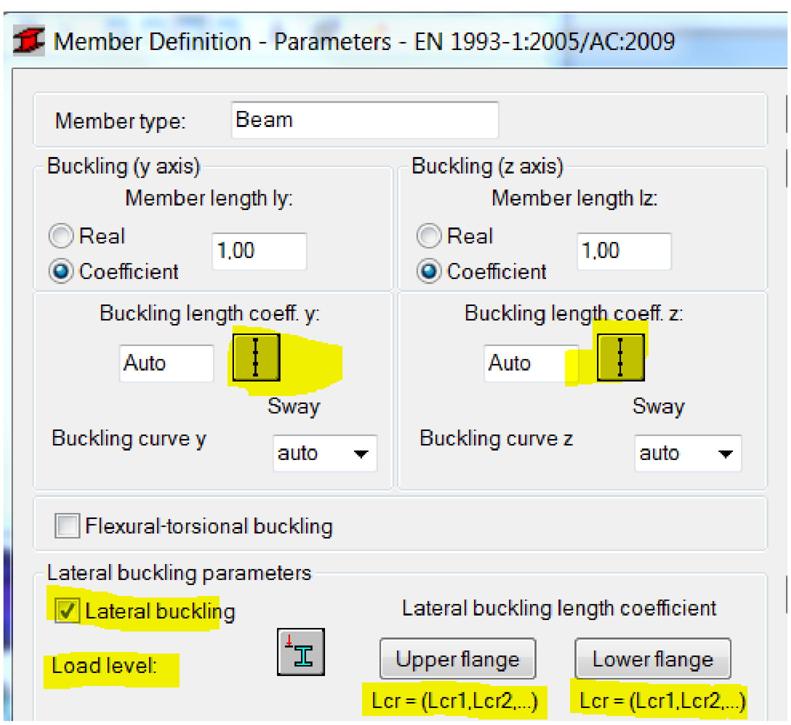
http://forums.autodesk.com/t5/robot-structural-analysis/load-sequencing-construction-stages/td-p/4568689

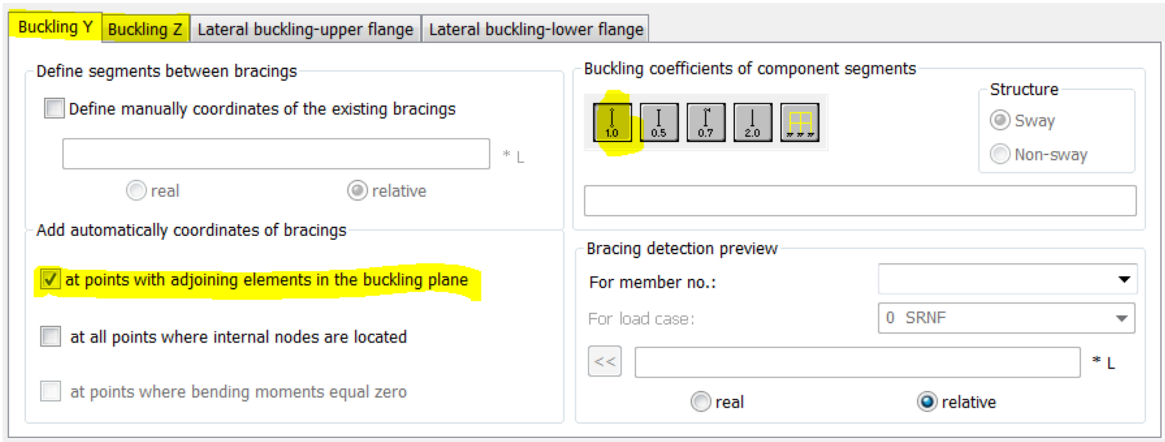
3. Working with the Steel Design module

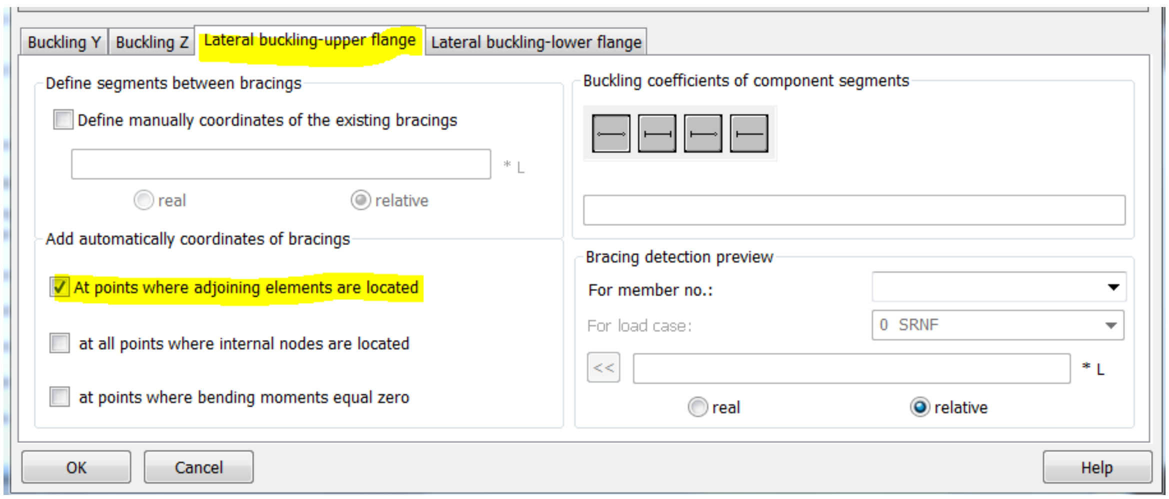
a) Setting design parameters (Member type definition)

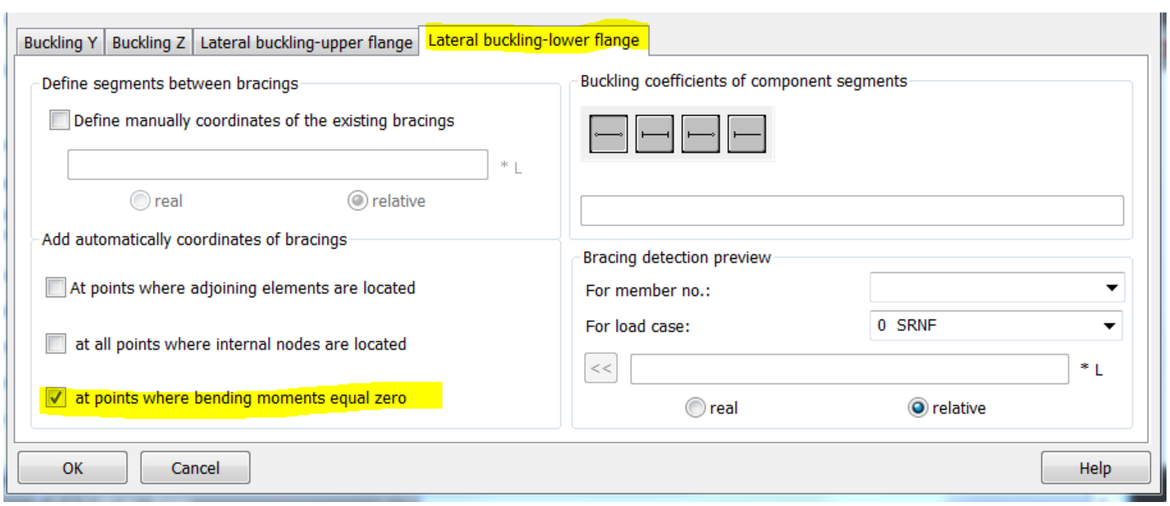


- **Beams** - the proposed setting for typical situations is to use automatic detection of bracings as shown below:



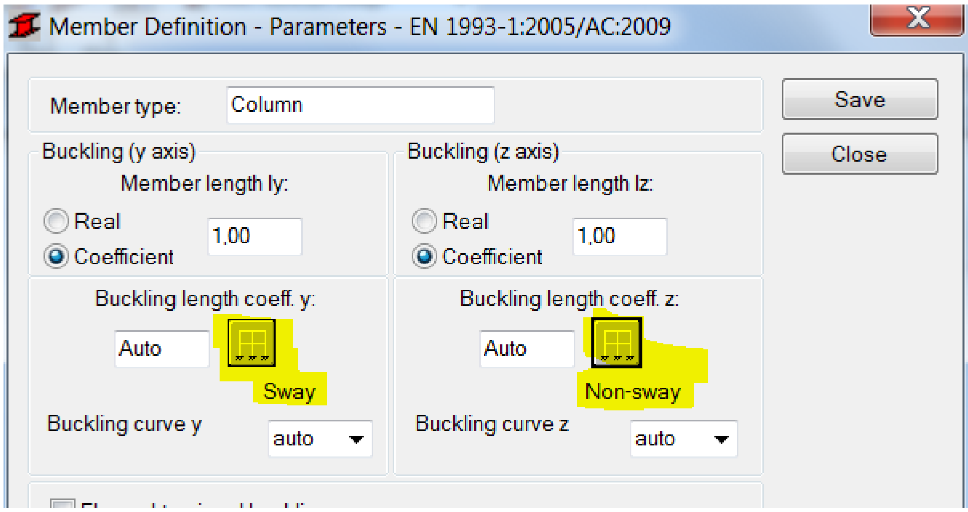


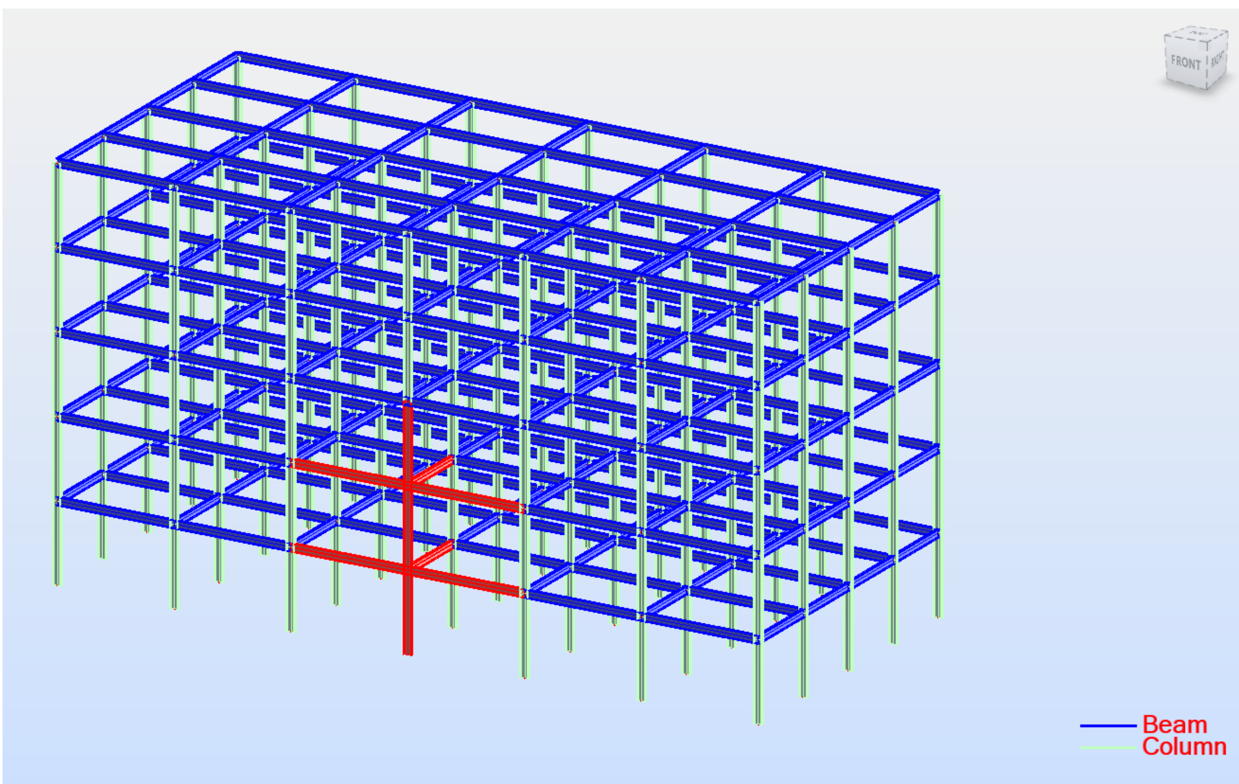




In addition for beams with large negative bending over supports you may decide to limit unrestrained lateral buckling length for the bottom flange to the parts where it is under compression

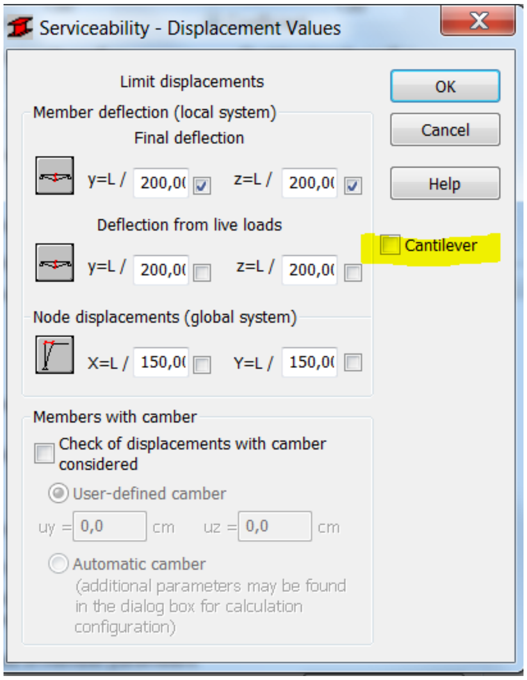
- **Columns** – the use of automatic buckling length should be limited to 'box' shaped structures with perpendicular beams defined among rows of columns at levels of horizontal floors



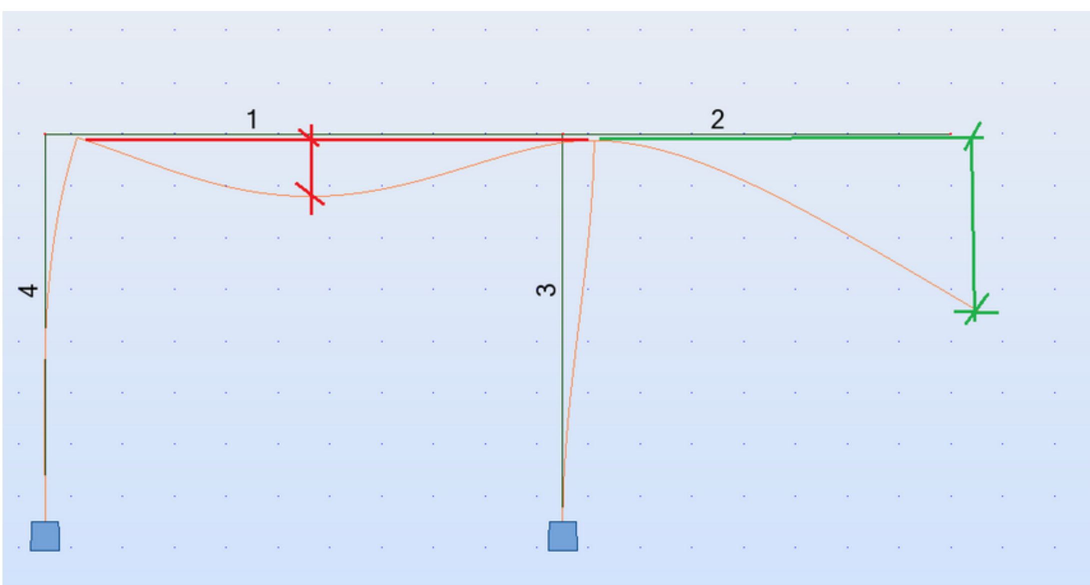


It is strongly recommended not to alter settings for member length and keep the default 1.00 coefficients in these fields.

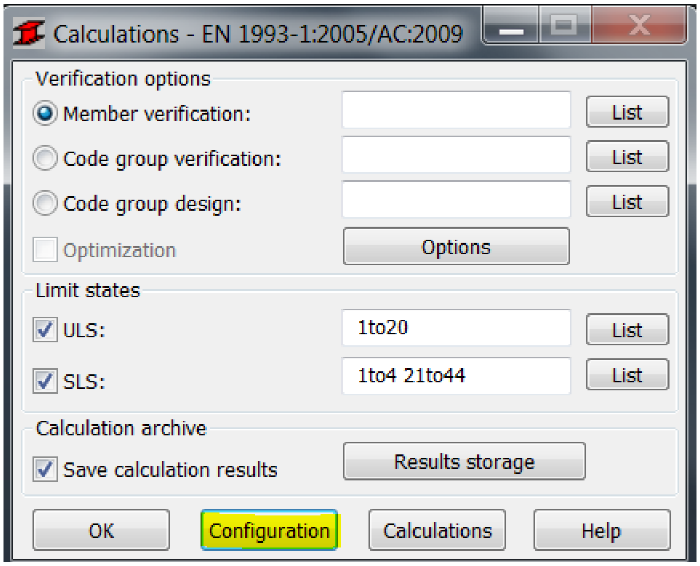
- A **beam vs. a cantilever** (use of Cantilever check box) for SLS verification



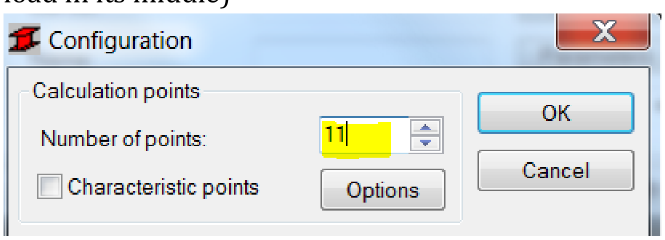
If the check box is not marked then the deflection is checked as the distance between the line that connects the beam ends in the positions they are after applying the load and the point along the beam that 'moves' the most whereas for the cantilever check box marked what is checked is the difference in displacement of the end nodes.



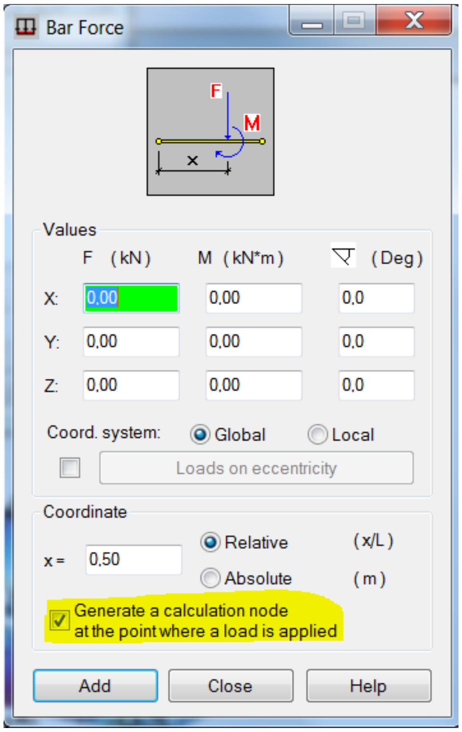
b) Setting Calculations parameters (Configuration dialog)



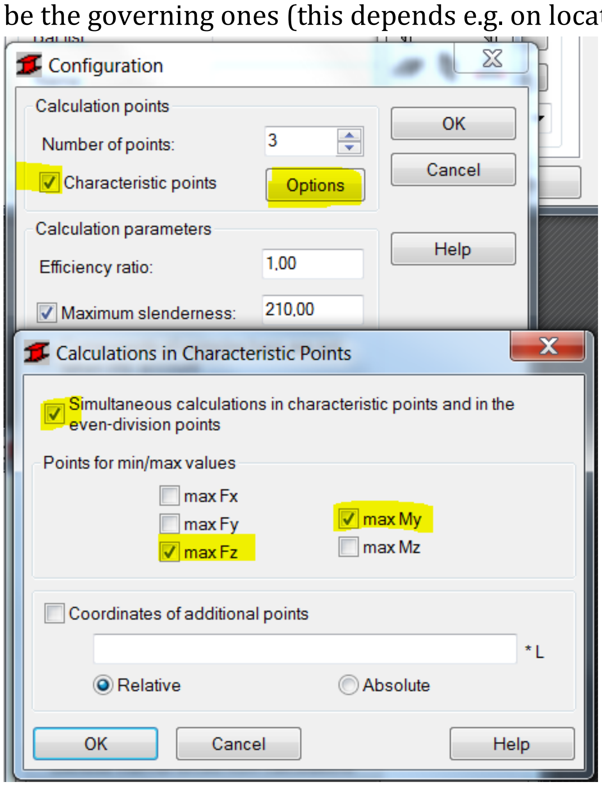
- Deciding on number of calculation points
  In general it is recommended to set relatively large number of verification points along a bar (usually the default 3 is good enough only for simply supported beams with uniform or point load in its middle)



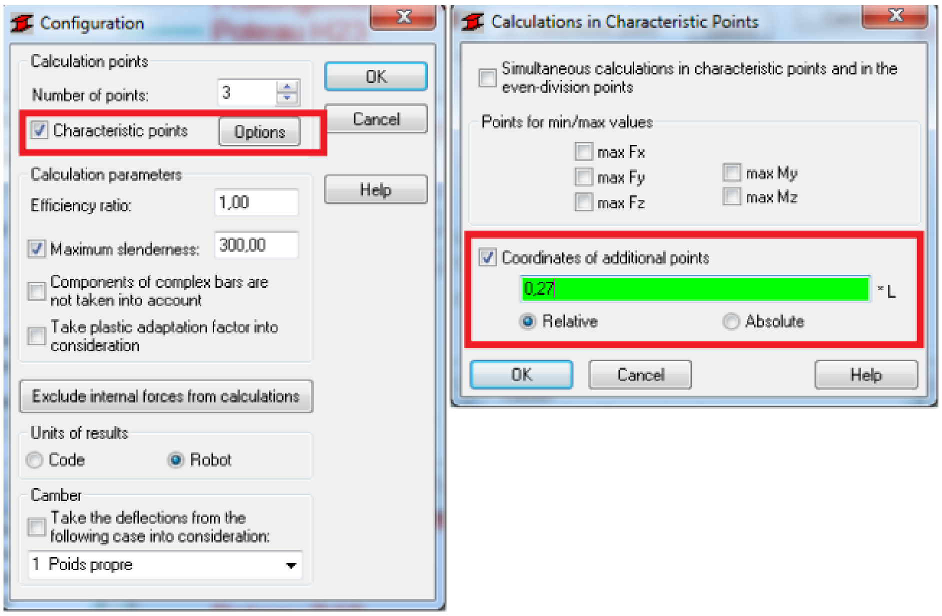
You may reduce their number when there are additional nodes along a bar as the points are set for calculation elements rather than the bar itself. This functionality is used to prevent "skipping" the point where a concentrated force is applied but in order to have it applied you need to define a point force with a calculation node:



It is recommended to have the Simultaneous calculations check box marked as max/min forces are assumed as the ones with largest absolute values (sign is ignored) and such points may not be the governing ones (this depends e.g. on locations of bracings as well)



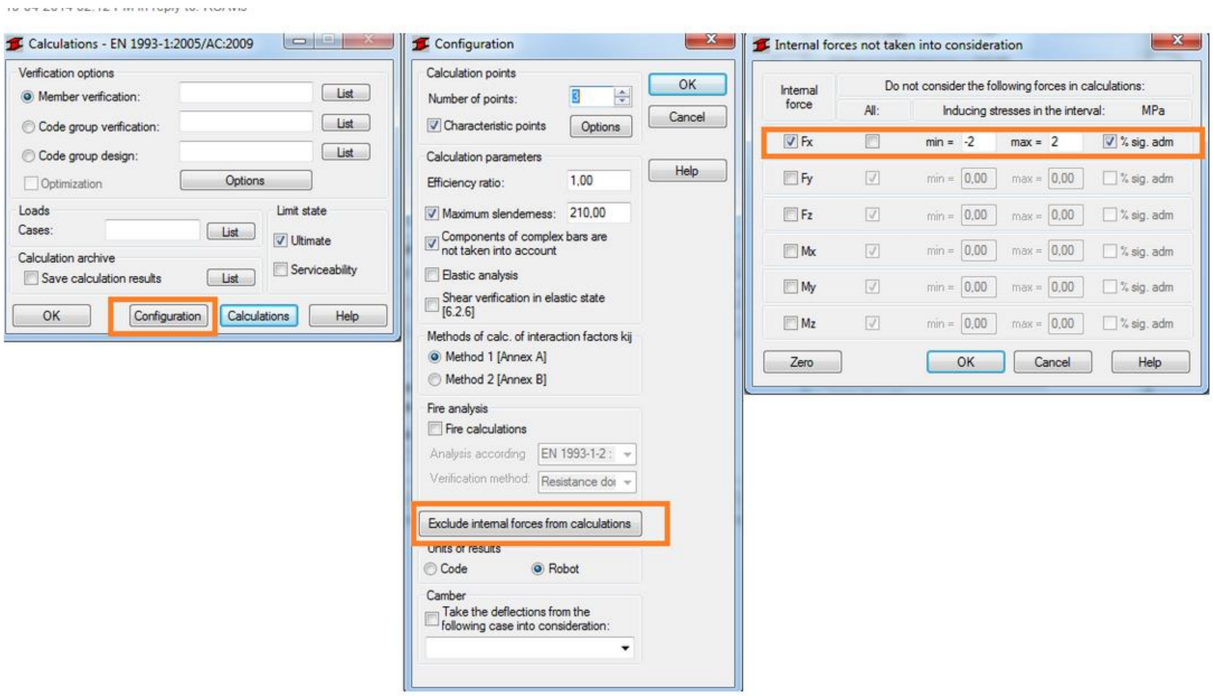
The results are displayed from the point where calculated ratio is the highest however you may run the check in an arbitrary selected location along a bar using the following settings:



- Excluding some internal forces from calculations

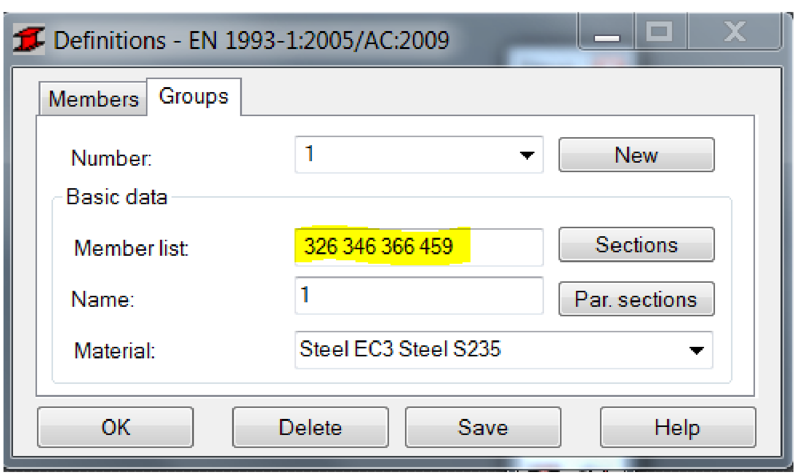
In some situation you may want to exclude small axial forces or bending moments to run design against simple bending or unidirectional bending with axial force instead of biaxial bending (with axial force)

This can be done by setting the limits below which the effect they cause are considered to be small enough to be neglected. It is recommended to use the relative stress limit as such definition is more "general" and can be applied to sections of different sizes. Mind to set both negative and positive values.

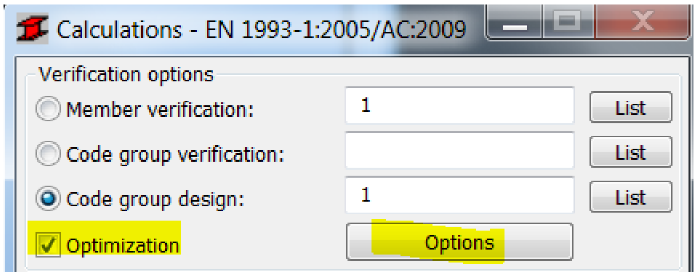


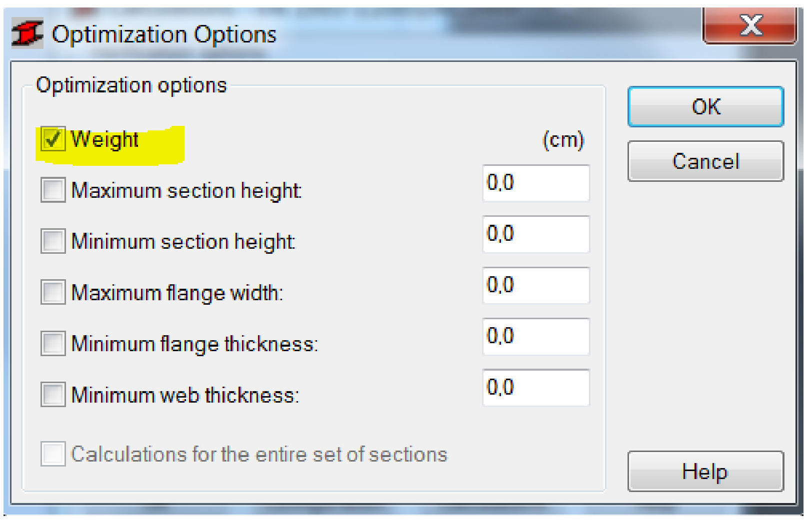
- Finding best matching section (Code group design)

In most situations you want to find a best (the same) section for selected group of elements of a model (e.g. internal columns on the 2nd floor or main girders of a roof) and the design criteria is weight (cost) rather than ratio closest to 1. In such case you need to create design groups based on list of elements from such group

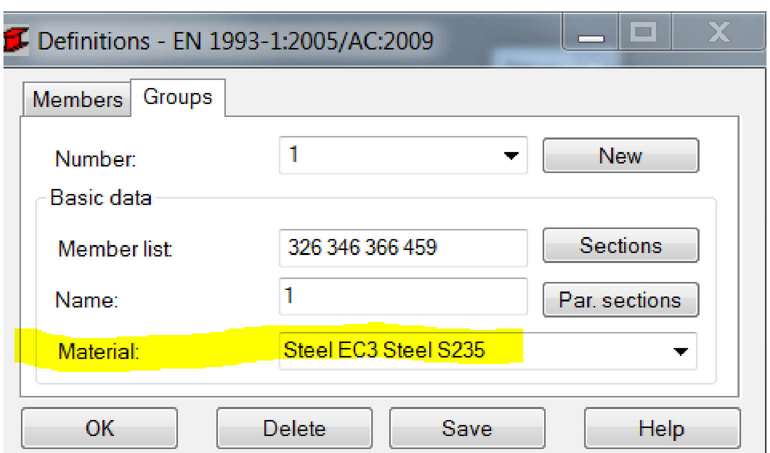


and select the optimization parameters



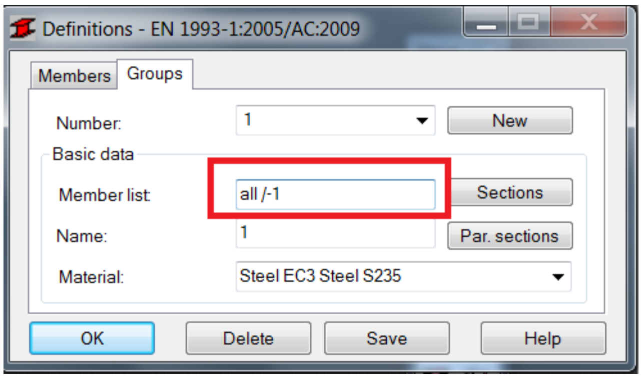


Mind that the code group design (contrary to the member verification) is based on the material assigned to the group rather than to elements that this group includes

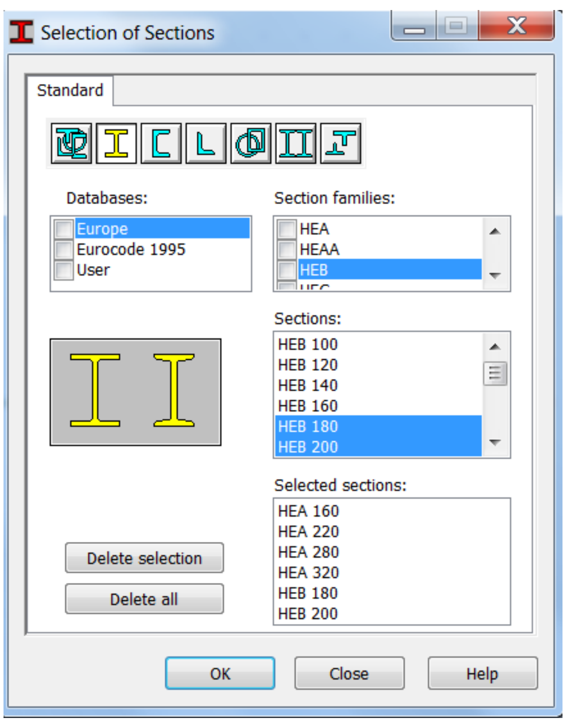


- Finding best matching section for each beam or column of a model separately

As such operation can only be performed on groups it is necessary to create them for each of element of a model separately (one element = one group). This can be done automatically by typing all/-1 in the list of bars and pressing the ENTER key

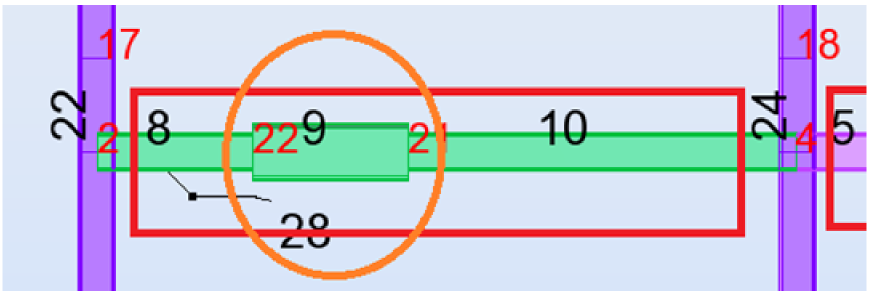


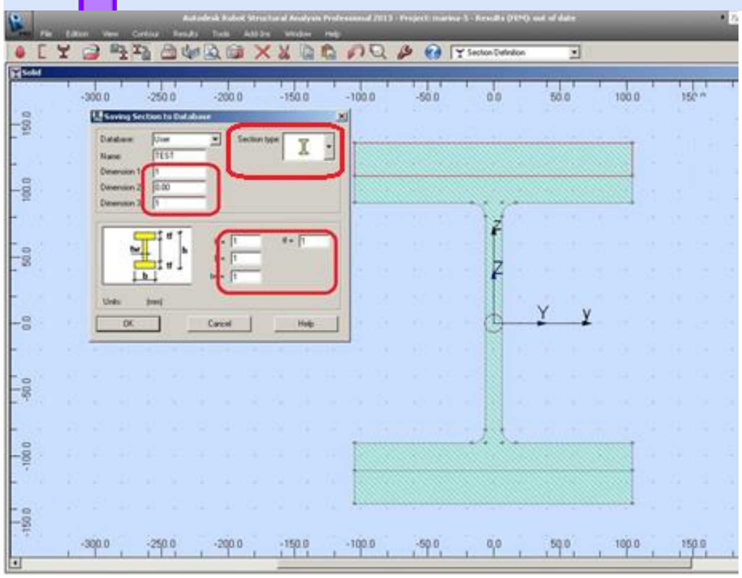
- Limiting search to only these profiles which are available (on stock)



Marking the check box moves all profiles from the marked database or family to the selected (available for design) section list.
Selecting a database allows for picking up families or single profiles
If no selection is done the design assumes that all profiles from the family of a section assigned to a beam can be used.

- Design of user created section is based on selected section type (verification path and formulas to be used) as well as entered section geometrical properties (section class determination) e.g. for I section with additional welded plates as discusses at the beginning of the presentation

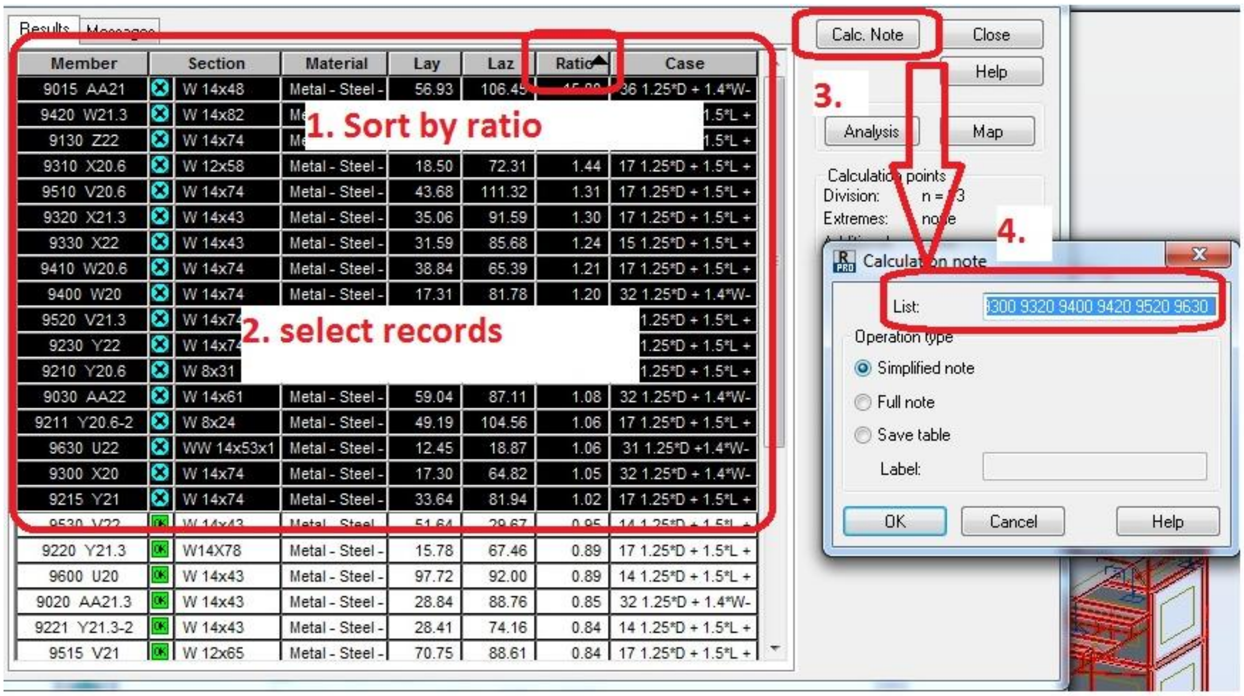


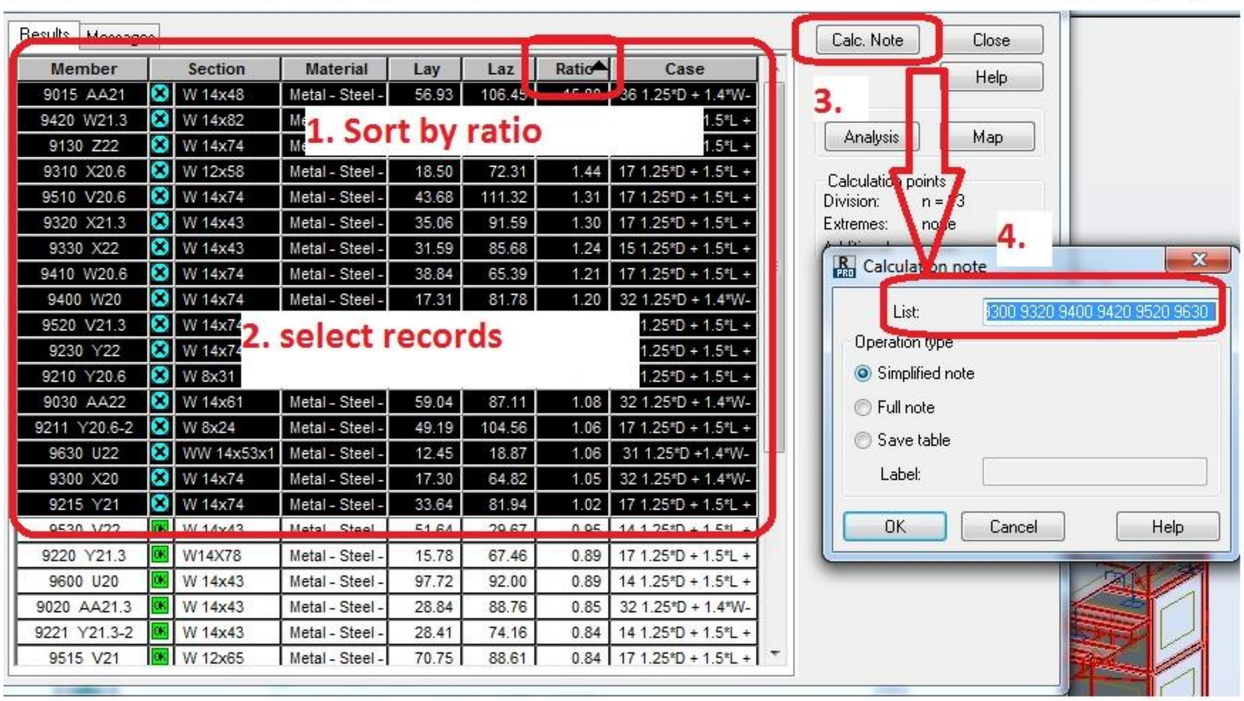


When section type is left as “?” the verification is done as for a solid rectangular profile

- Managing the results of code checking

To check which elements failed and to what degree you can follow the steps illustrated on the pictures below:





Additional references to the discussions on these topics on the Robot Forum:

http://forums.autodesk.com/t5/robot-structural-analysis/problem-with-custom-sections/td-p/3379315

http://forums.autodesk.com/t5/robot-structural-analysis/bar-deflection-code-check-cantilever-check-box-usage/td-p/3300335

http://forums.autodesk.com/t5/robot-structural-analysis/calculation-points-in-steel-design/td-p/3345899

http://forums.autodesk.com/t5/robot-structural-analysis/steel-design-results/td-p/3446635

http://forums.autodesk.com/t5/robot-structural-analysis/steel-design/m-p/3340849

http://forums.autodesk.com/t5/robot-structural-analysis/ltb-amp-steel-bracket/td-p/3385765

http://forums.autodesk.com/t5/robot-structural-analysis/need-help-in-design-a-steel-building/td-p/3443633

http://forums.autodesk.com/t5/robot-structural-analysis/autodesk-robot-structural-analysis-steel-design-elements-which/m-p/3501700

http://forums.autodesk.com/t5/robot-structural-analysis/steel-beam-design-issue/td-p/3622270

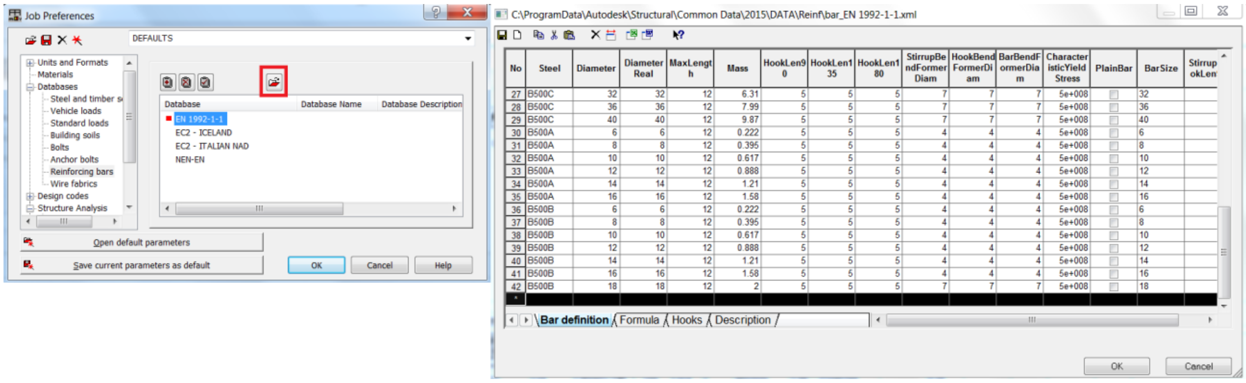
http://forums.autodesk.com/t5/robot-structural-analysis/code-group-verification-vs-member-verification/td-p/3667094

http://forums.autodesk.com/t5/robot-structural-analysis/steel-beams-verification-without-axial-force/td-p/4948536

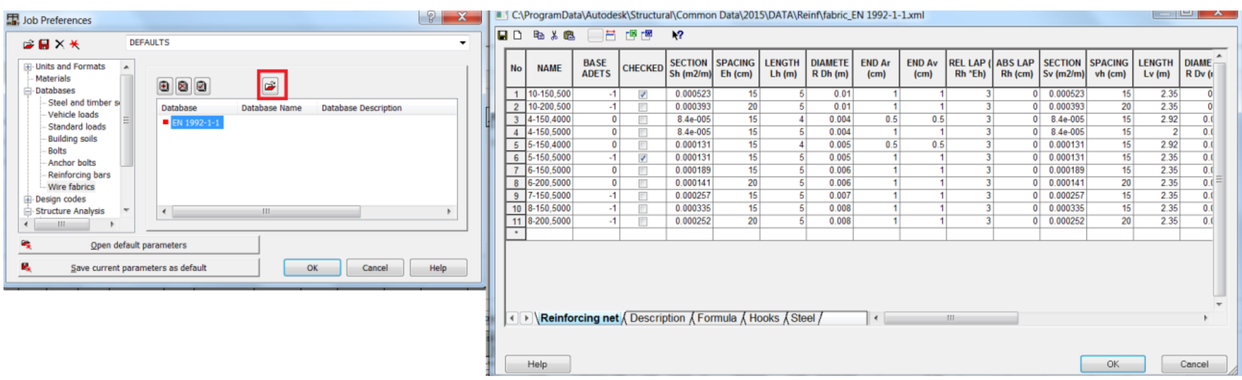
http://forums.autodesk.com/t5/robot-structural-analysis/metal-beam/td-p/4962944

4. Working with the RC Design modules

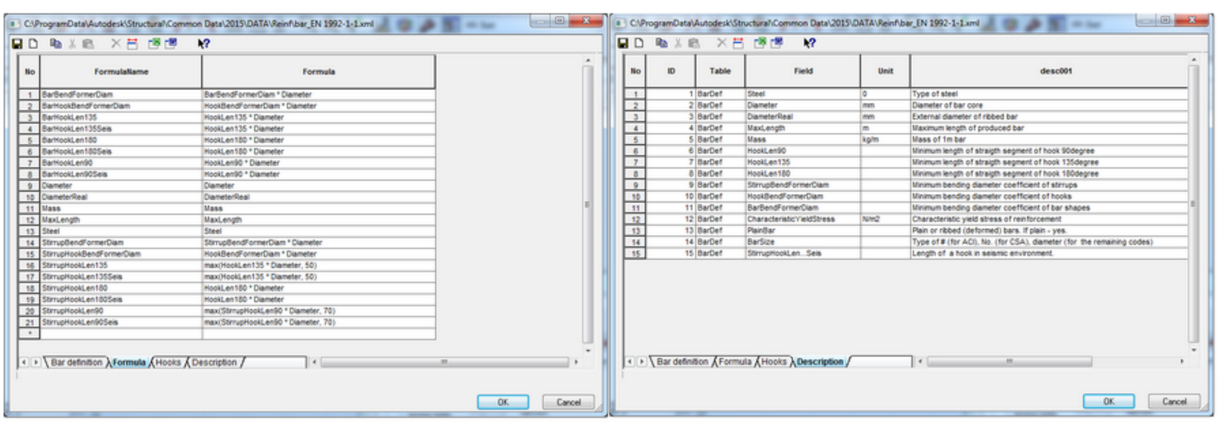
- To add new reinforcement bar grade or diameter you select and edit reinforcement bar database



In the same way it is possible to add or edit the list of wire fabrics (meshes)



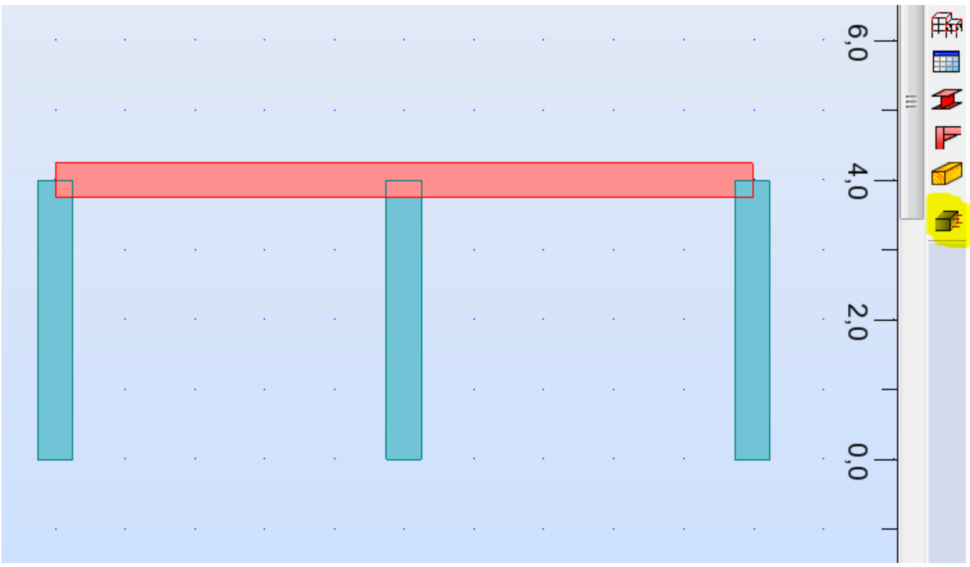
The description tabs provide information about each of the columns.



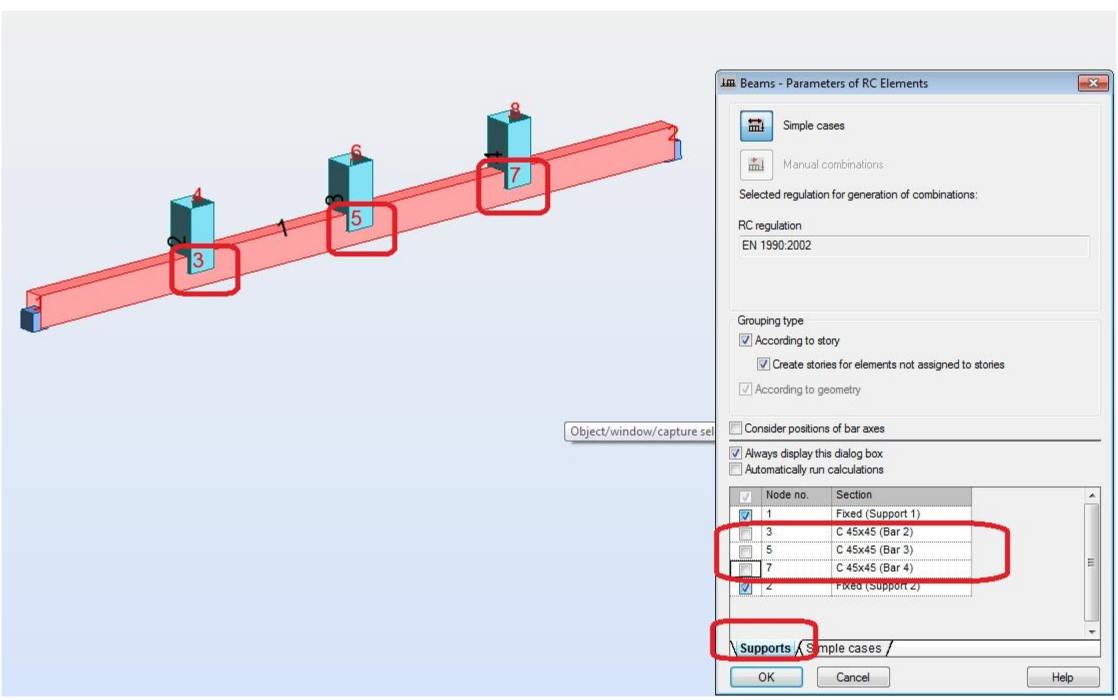
a) RC Beam Design module

- Import form a model

To import a beam from a model you need to select bars which define it and press the indicated button

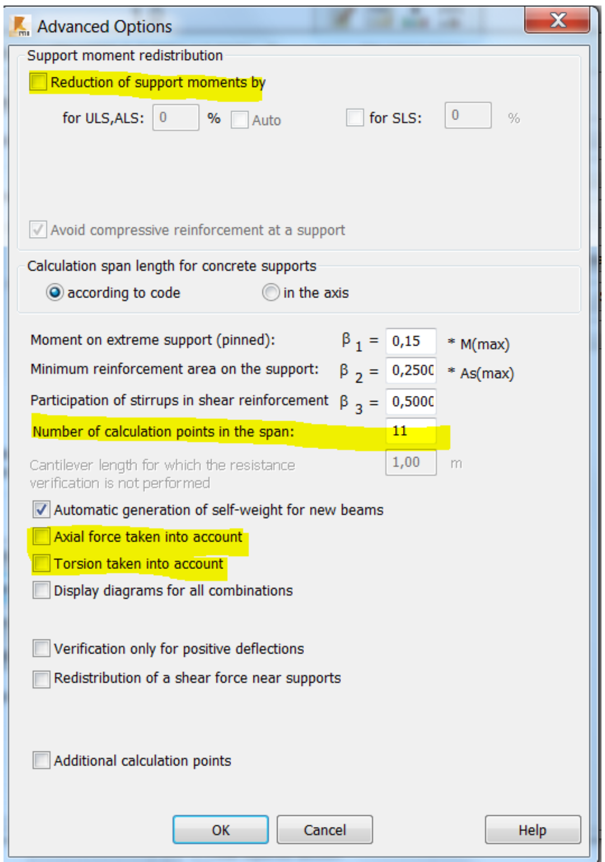


In addition to decision about loads you want to import mind to check the supports tab to see if the automatic selection (based on supporting element type and its size) has been done correctly

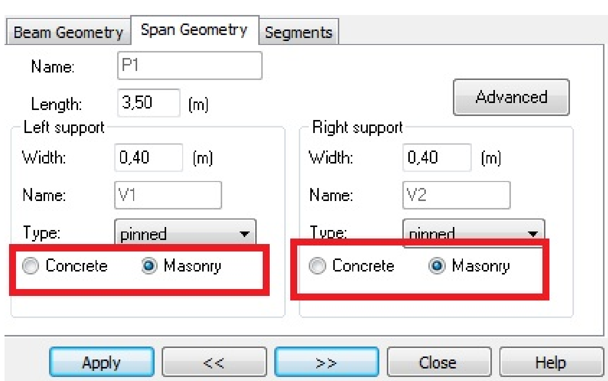


- Calculation options

  By default a beam is designed against the simple bending with values checked in 11 points. These settings can be changed in the Advanced (calculation) options dialog



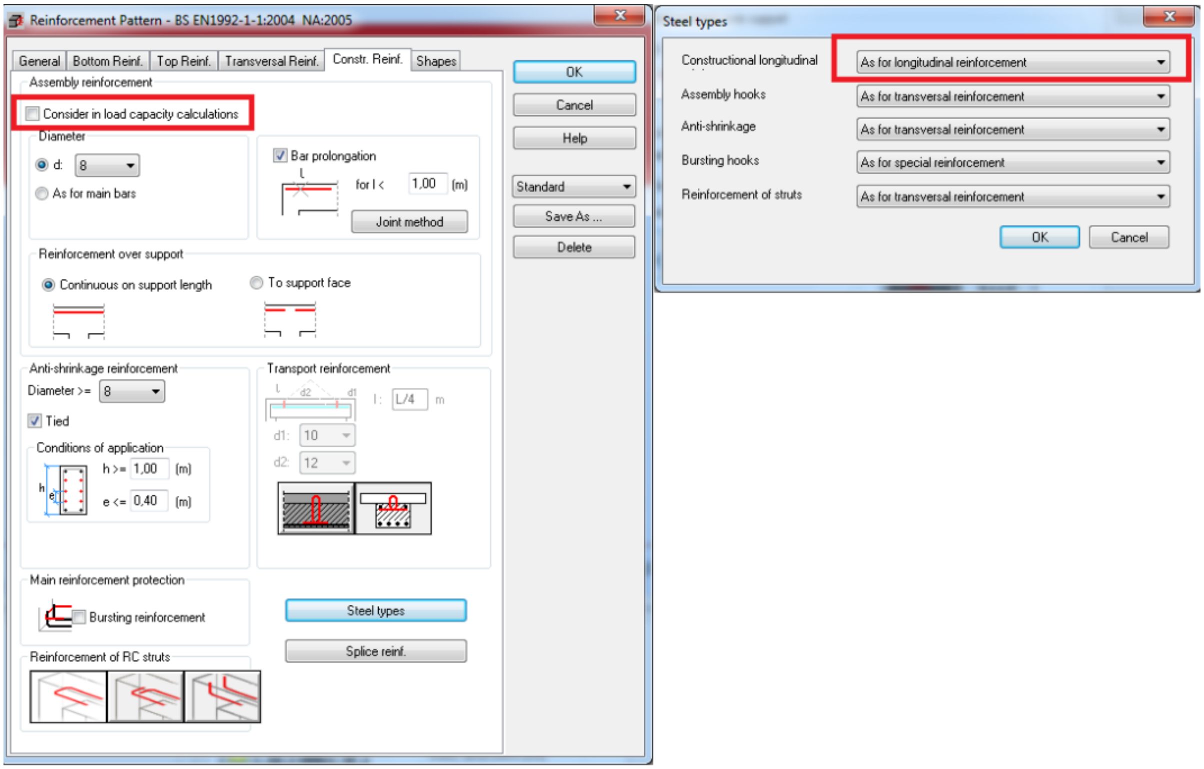
In addition it is possible to decide if you want to calculate reinforcement for bending moment and shear force from support's face or its middle by selecting a support type type



- Reinforcement pattern (distribution)

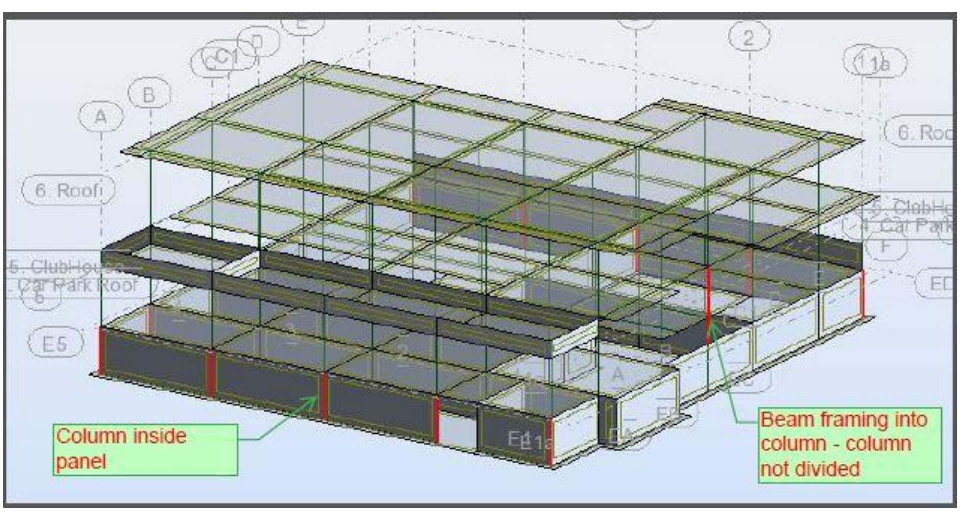
The decision of support type influences the range of generation of stirrups. In case you want to design a beam with concrete supports for bending moments from its middle but you don't want to have any stirrups generated over supports you may need to change the default settings

To include constructional reinforcement in beam's capacity you need to set its steel grade as being the same as the one selected for the longitudinal (main) one.



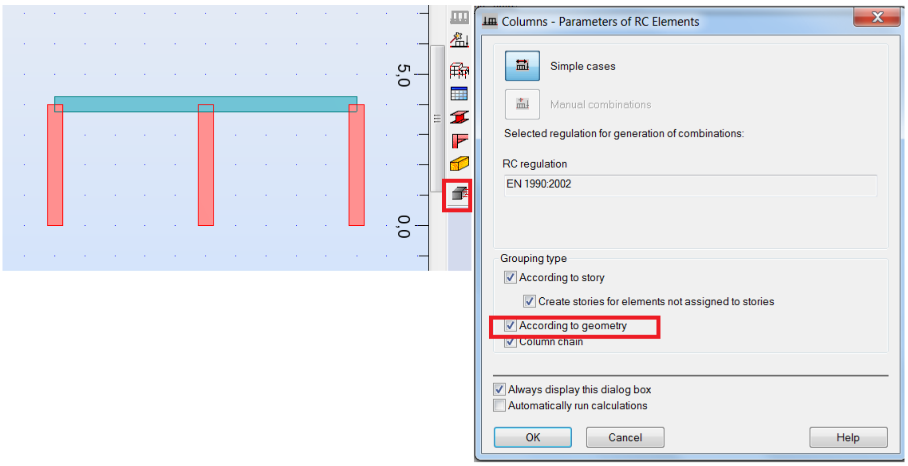
b) RC Column Design module

RC Columns in a model should be defined as a single bar elements (between levels of beams or floors) rather than a chain of smaller ones or a single element running through all the stories. In case of a column defined "inside" a "meshed" wall you should ignore the warning about nodes defined along its length.



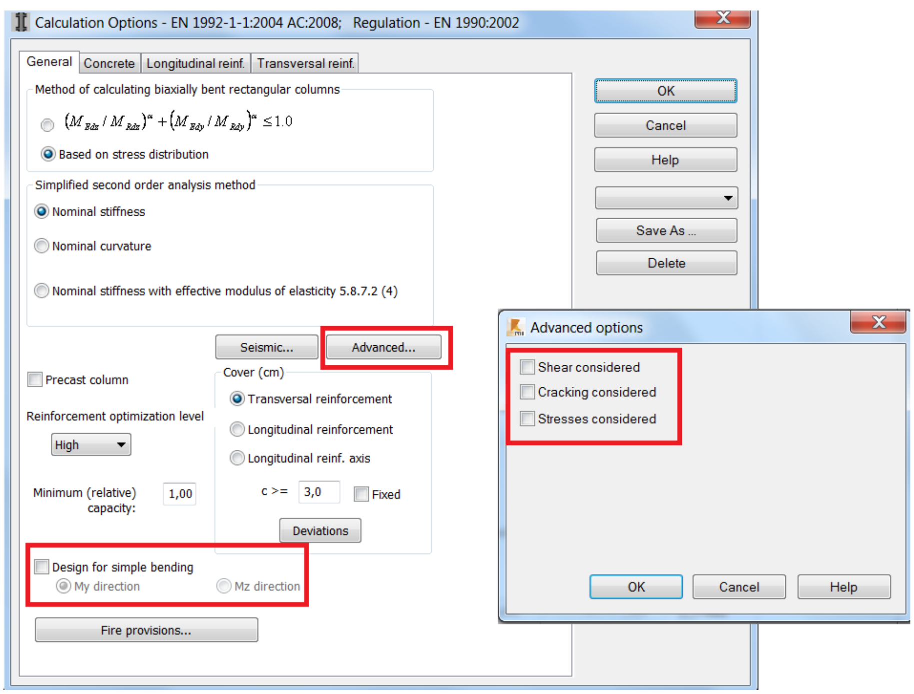
- Import form a model (grouping)

RC columns are exported from a model in the same way as beams but it is possible (similarly to the group design in the Steel Design module) calculate identical reinforcement for group of RC columns of the same geometry

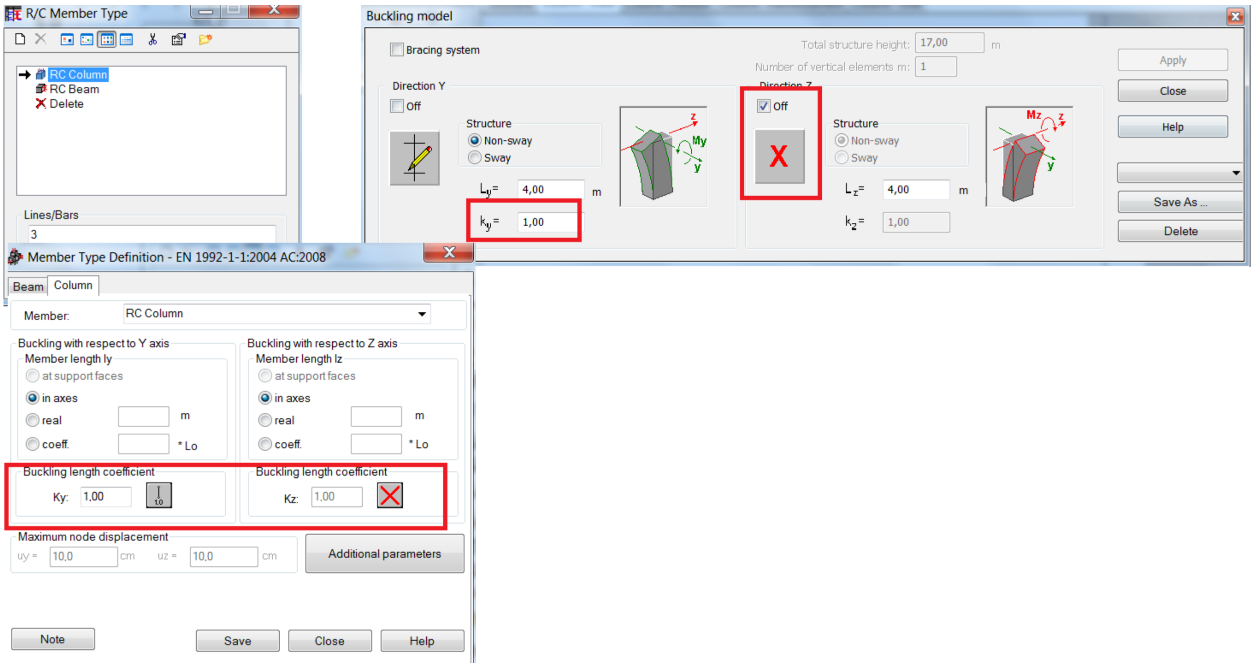


- Calculation options

By default columns are calculated for axial force and bi-directional bending but it is possible to include shear and SLS design or limit bending to one direction only (when the other direction bending moment is small enough to be neglected or a column is a 'part' of a wall)



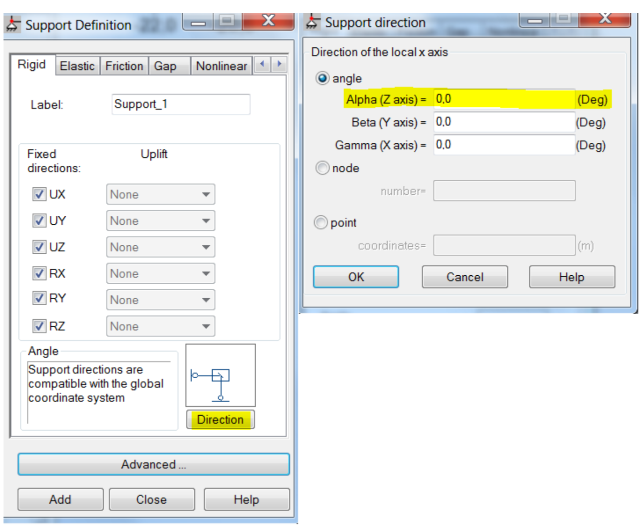
Mind that buckling parameters are taken from the Member type label assigned to a column in a model.



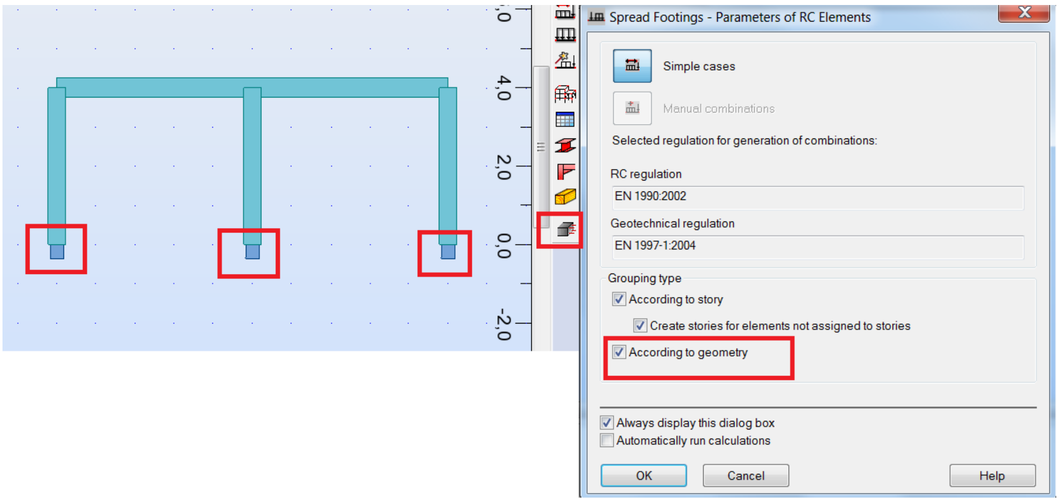
c) RC Spread Footing Design module

- Import form a model (grouping)

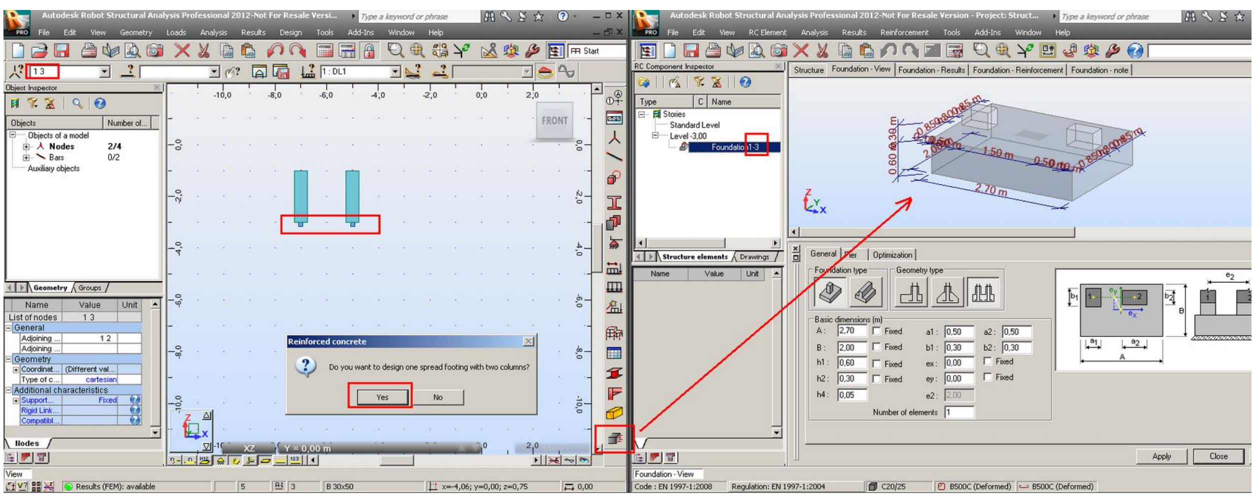
Design of a spread footing is based on reactions imported from selected support nodes. It is important to match the directions of support with the direction of the local coordinate system of a column as the design of the spread footing is based on obtained values of reactions.



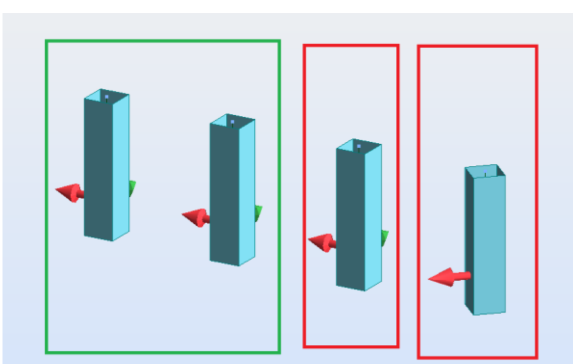
It is possible to group nodes in order to have identical foundations in these locations.



When two supported nodes are selected it is possible to design a common foundation for both columns located in these nodes



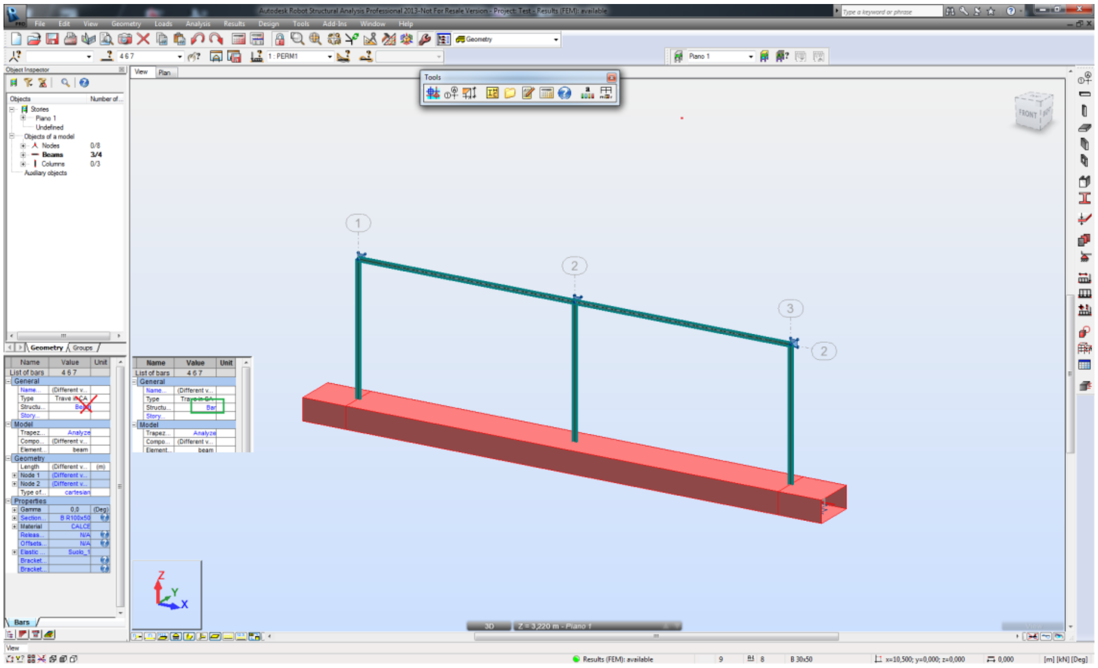
It is important to match the directions of local coordinate systems of the columns



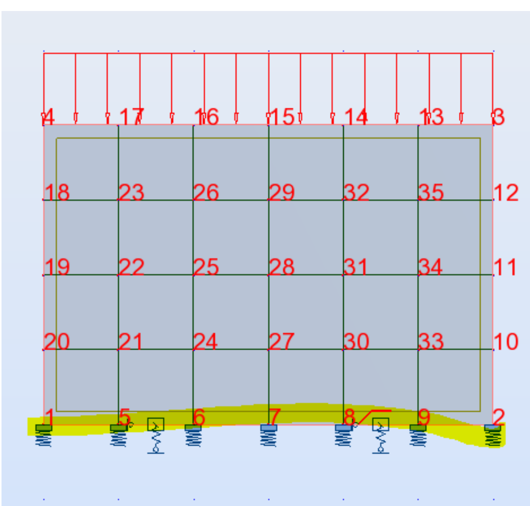
d) Design of RC Continuous Footing

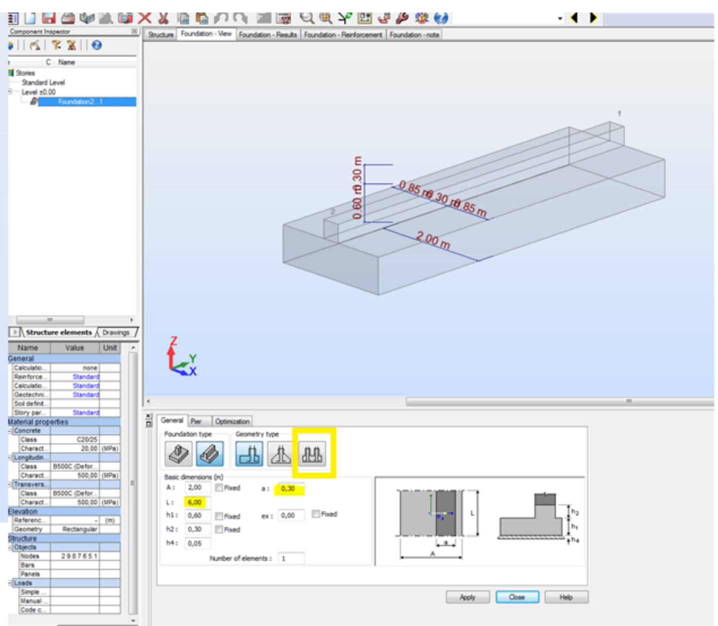
- For a strap foundation under isolated columns use the RC Continuous Footing module

Mind to set the Structure object type as a Bar rather than a Beam



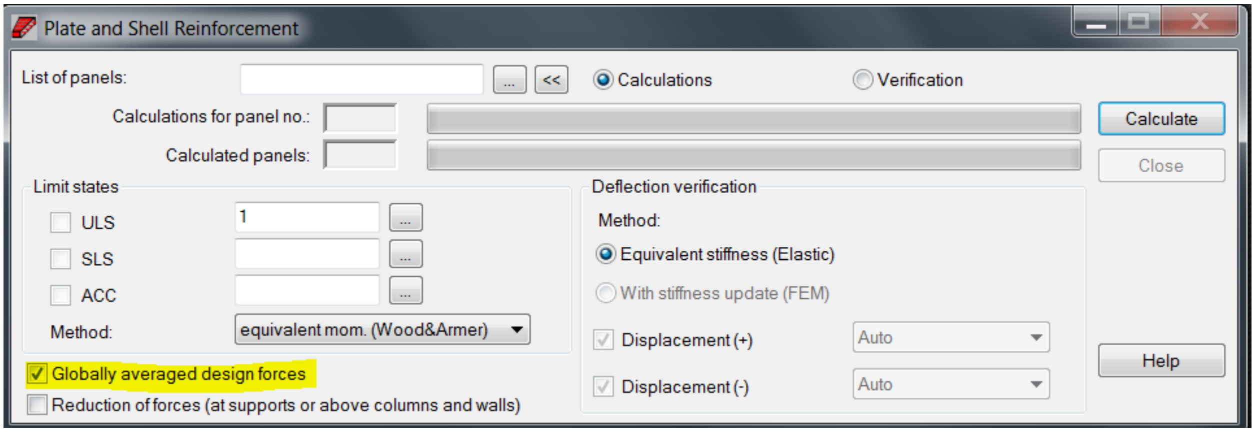
- For a strap foundation under a wall with defined linear support at the bottom edge use the RC Spread footing module in the continuous footing mode



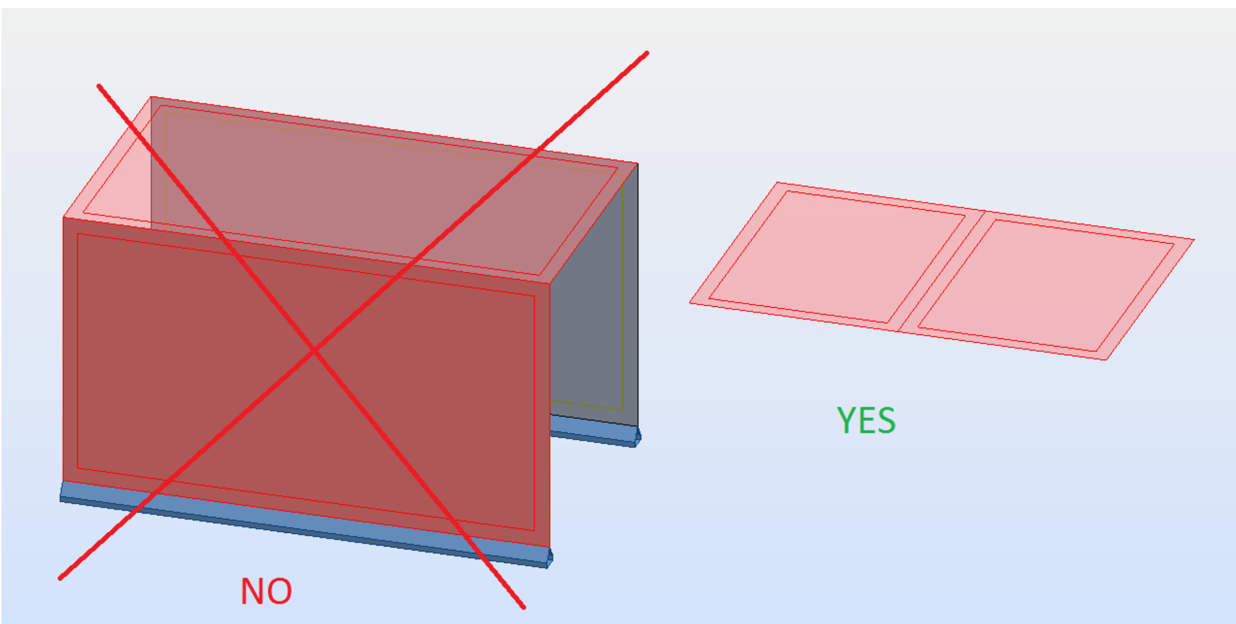


e) RC Slab Required Reinforcement module

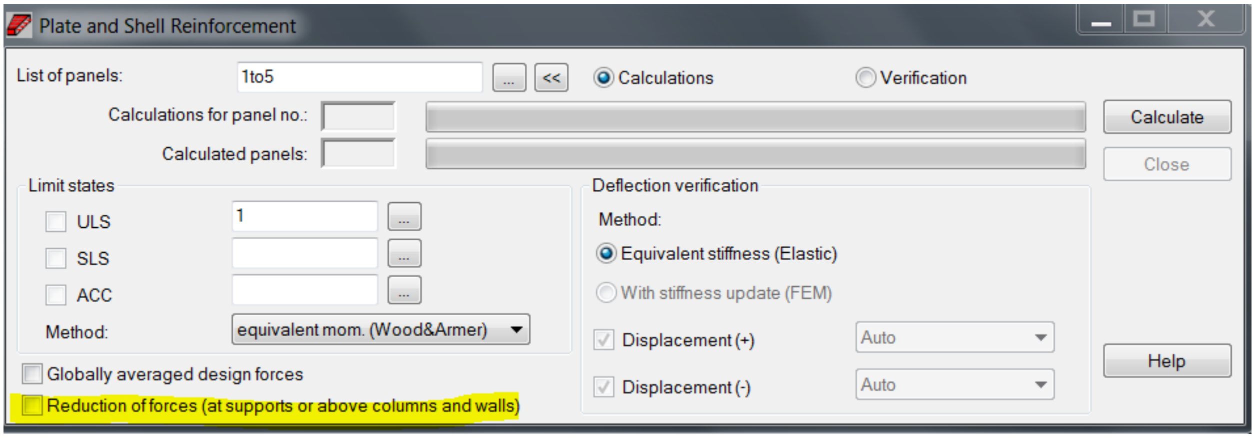
- Global averaging of forces



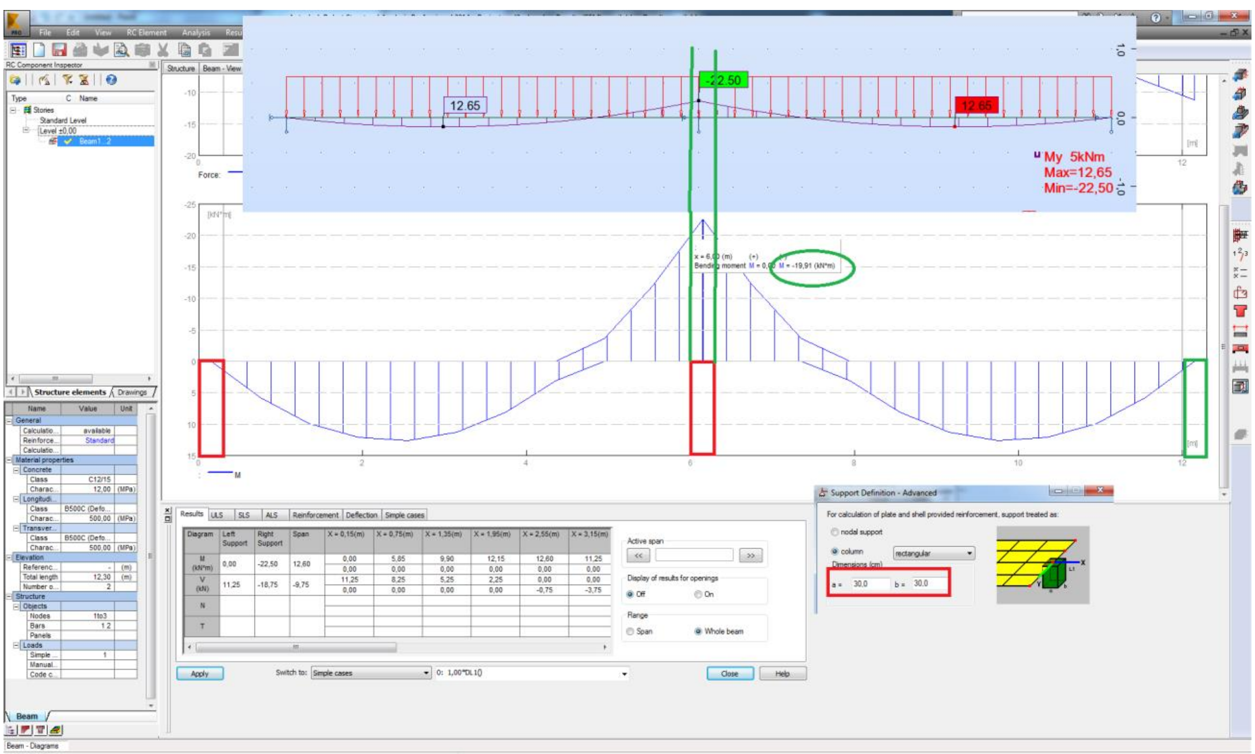
The option should only be used for a situation when a slab is modeled as number of smaller panels and there are no other panels defined in another planes. Otherwise it should be switched off.

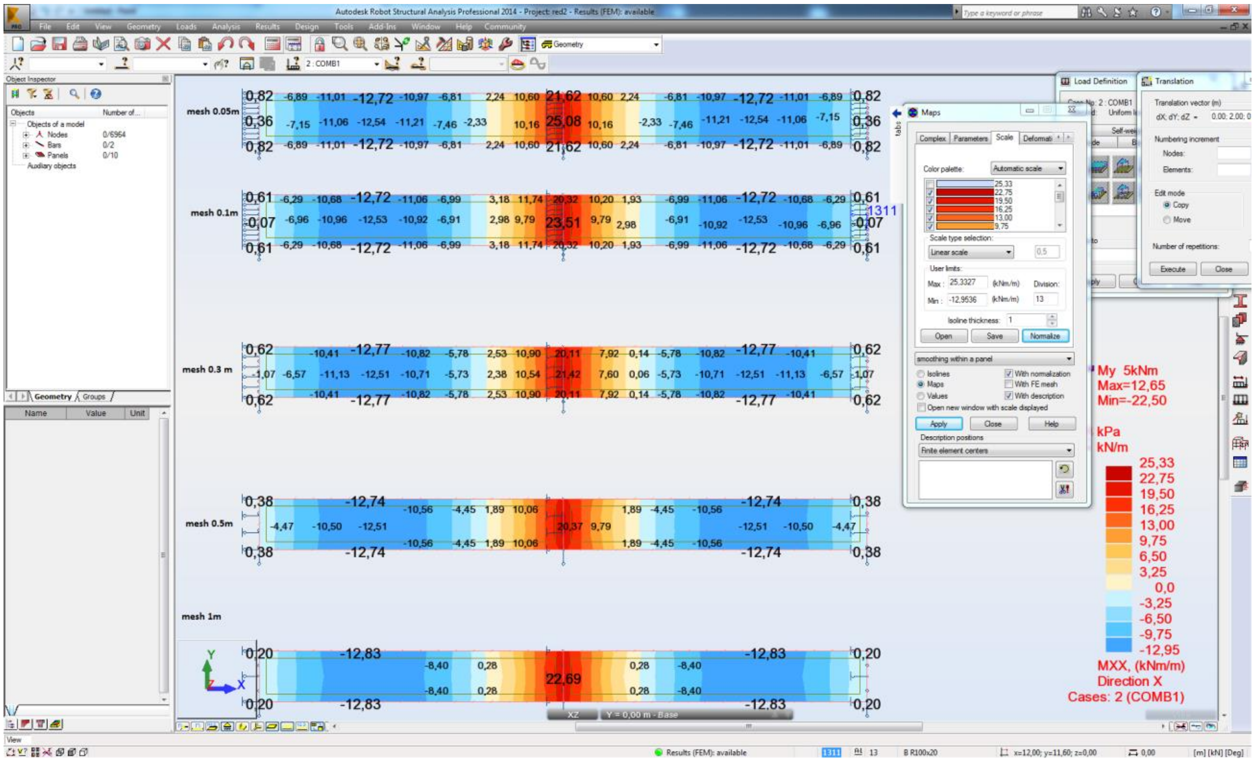


- Reduction of peek of bending moments over point supports

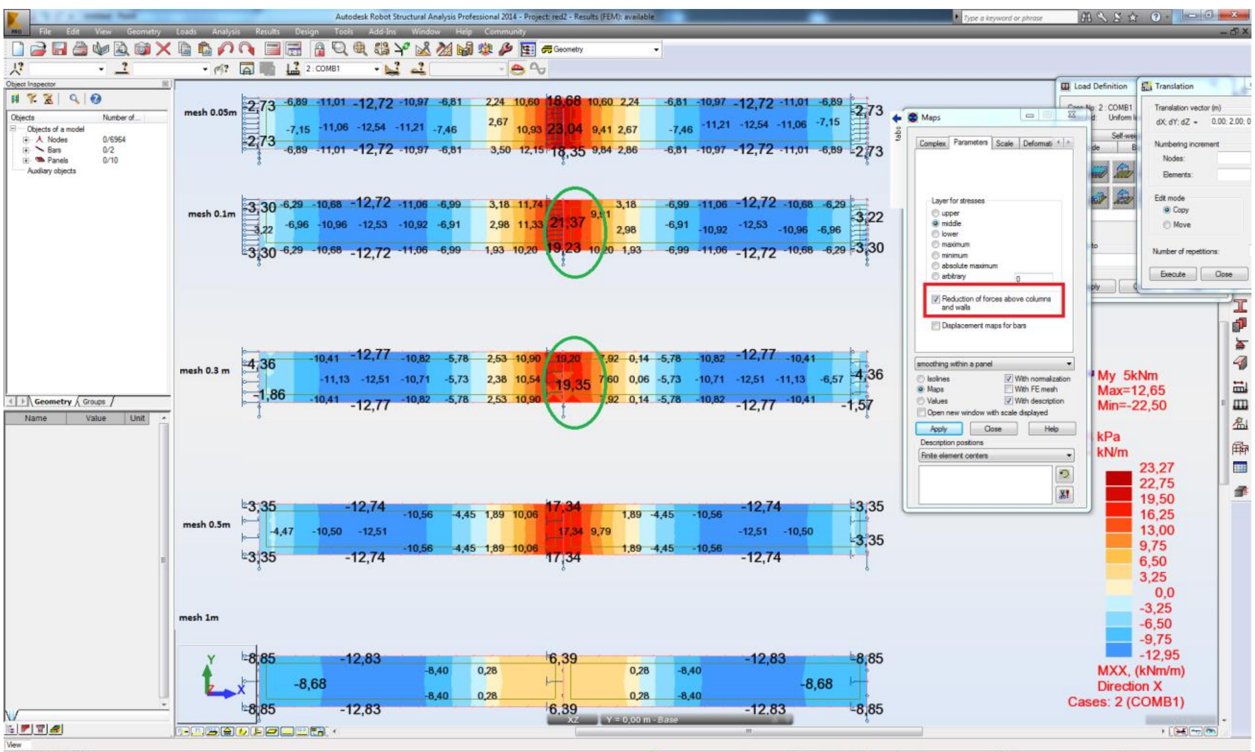


As the values of bending moments in these locations are larger than existing in reality you can reduce them based on the actual size of a support (column). The range of the reduction depends on the mesh size therefore you need to match the dimensions of elements of the mesh and panel's supports e.g. size of elements being approximately from a half to the size of a real support.





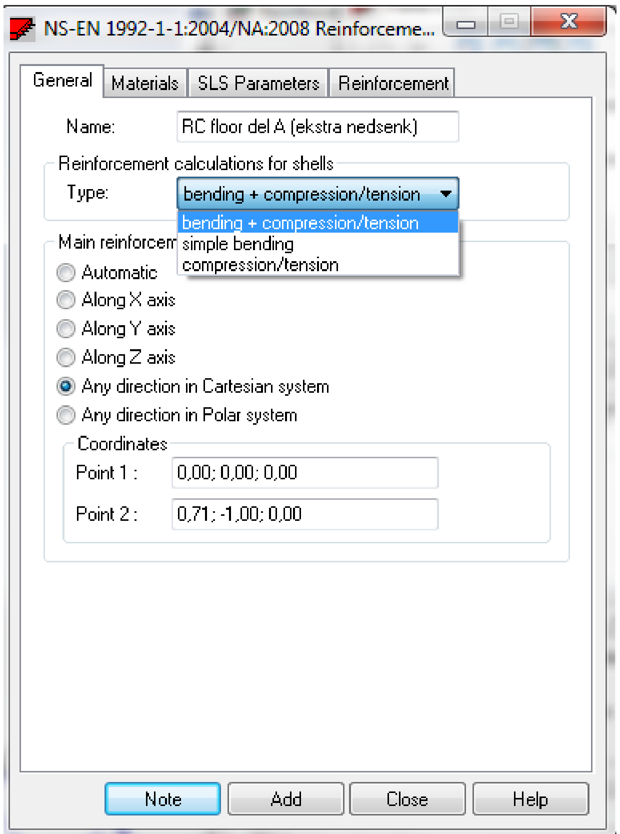
Autodesk Robot Structural Analysis Professional 2014 - Project: red2 - Results (FEM): available
mesh 0.05m
mesh 0.1m
mesh 0.3 m
mesh 0.5m
mesh 1m
Maps
My 5kNm
Max=12,65
Min=-22,50
kPa
kN/m
25,33
22,75
19,50
16,25
13,00
9,75
6,50
3,25
0,0
-3,25
-6,50
-9,75
-12,95
MXX, (kNm/m)
Direction X
Cases: 2 (COMB1)



Autodesk Robot Structural Analysis Professional 2014 - Project: red2 - Results (FEM): available
mesh 0.05m
mesh 0.1m
mesh 0.3 m
mesh 0.5m
mesh 1m
Maps
Layer for stresses
Reduction of forces above columns and walls
Displacement maps for bars
My 5kNm
Max=12,65
Min=-22,50
kPa
kN/m
23,27
22,75
19,50
16,25
13,00
9,75
6,50
3,25
0,0
-3,25
-6,50
-9,75
-12,95
MXX, (kNm/m)
Direction X
Cases: 2 (COMB1)

- Selection of forces included in the reinforcement design

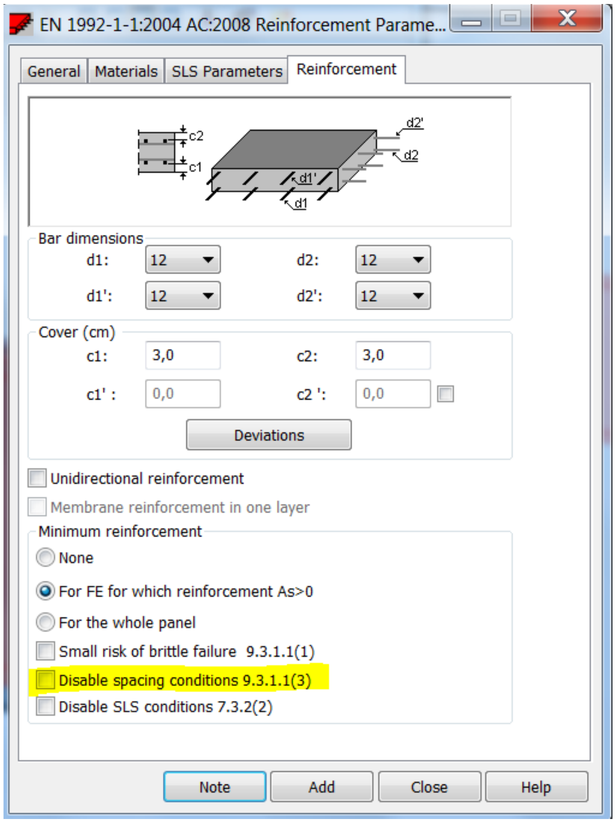
You can exclude small axial forces from reinforcement calculations selecting the simple bending mode as well as exclude bending moments using the compression/tension one.



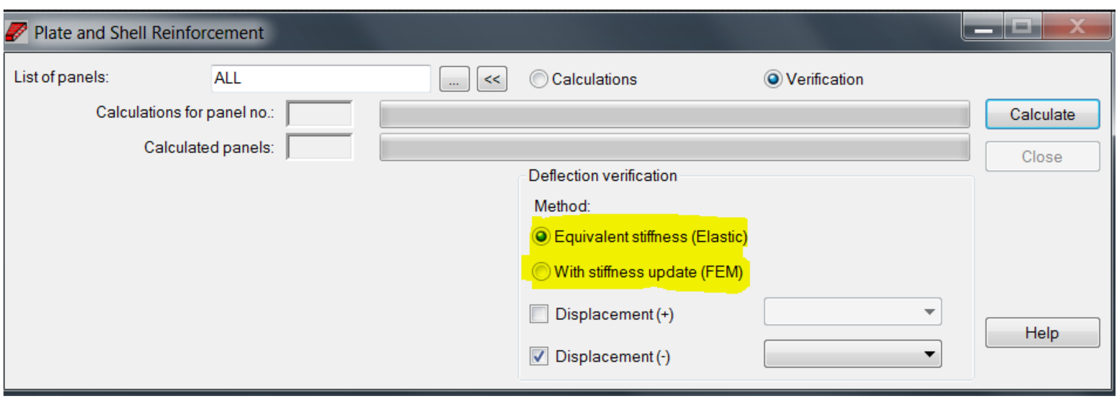
Mind that for some codes there is large difference between minimal area of reinforcement for elements under pure tension (such situation may happen for the bending + compression/tension) and under bending.

- Influence of maximal allowed bar spacing on calculated required area of reinforcement

At the stage of calculations of required area of reinforcement you may not know what bar diameter will be finally used therefore you may assume it as larger one for safety reasons (arm of internal forces; cracks). In such case for lightly loaded slabs you may get large values due to maximal allowed bar spacing. This effect can be disabled by checking the check box marked below:



- Calculations of "cracked" slab deflection

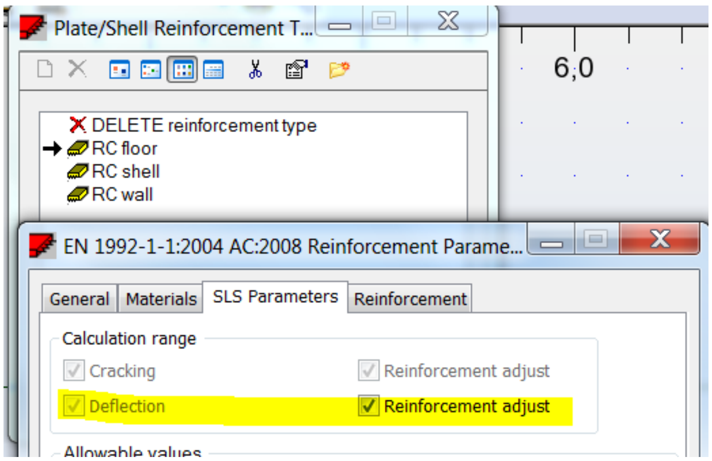


The equivalent stiffness method is the approximated approach based on scaling the deflection obtained from the static analysis of a model by elastic stiffness of a panel to its "averaged" cracked stiffness factor. Due to assumption of this method it should not be used to raft foundation on elastic soil and you should check the values of displacements obtained from static analysis instead.

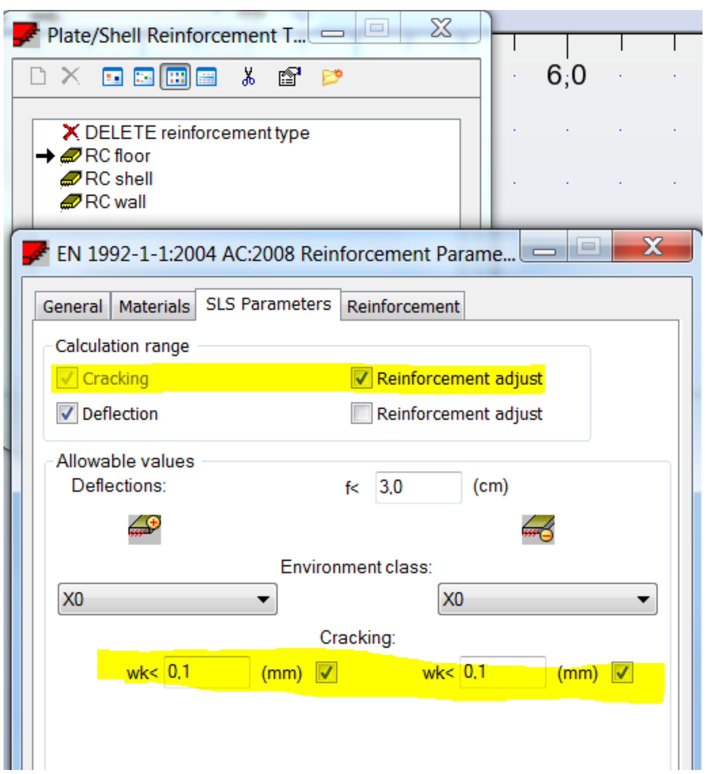
The stiffness update method is based on calculations of entire model with each element of a panel having its stiffness updated (reduced) according to calculated area of reinforcement and crack width. This verification can only be done for a single load case or combination at a time.

- Correction of an excessive deflection by additional reinforcement

If you mark Reinforcement adjust for deflection then the additional reinforcement is added in the locations where the calculated cracked stiffness of a slab is much smaller than in other locations. In some situations this may result in having very large area of reinforcement in isolated places across the slab.



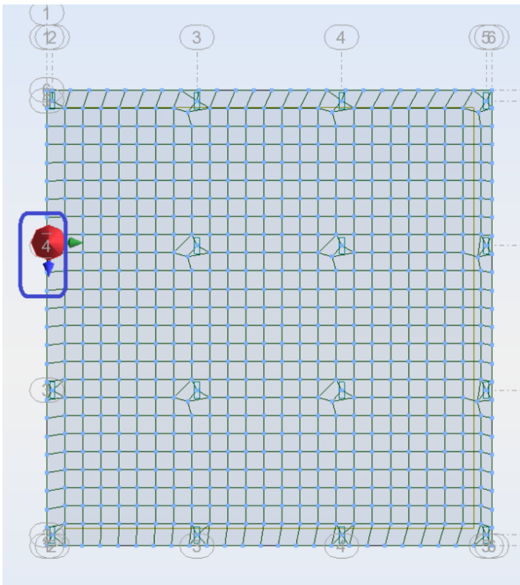
Alternative approach is to reduce the allowable crack width which will result in having much more uniform distribution of “additional” reinforcement needed to keep cracks below the new limits which will also reduce the deflection of a slab.

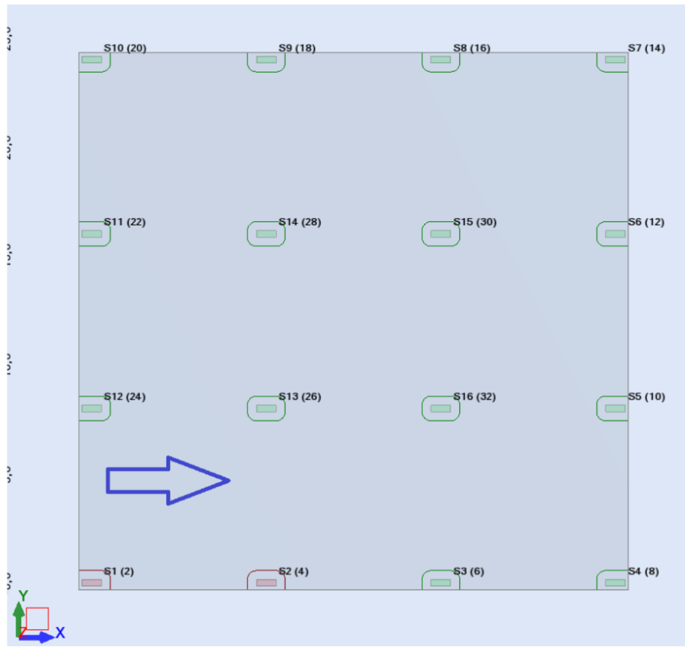


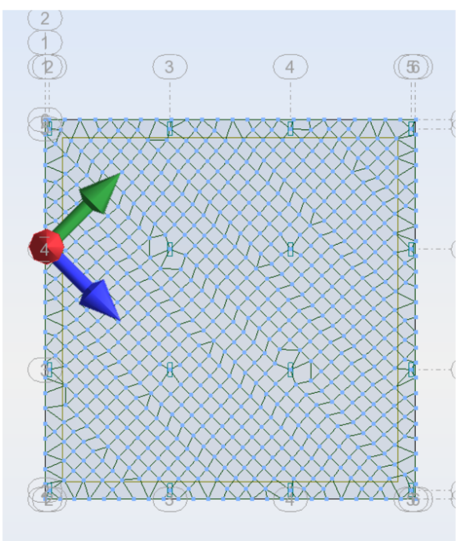
f) RC Slab Provided Reinforcement module

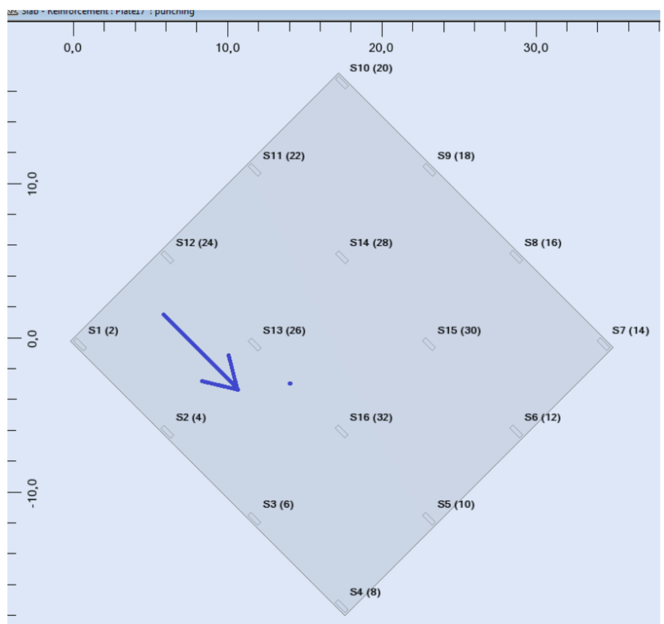
- Orientation of a slab imported from a model

The direction is governed by the local X axis of a panel (as defined in a model)



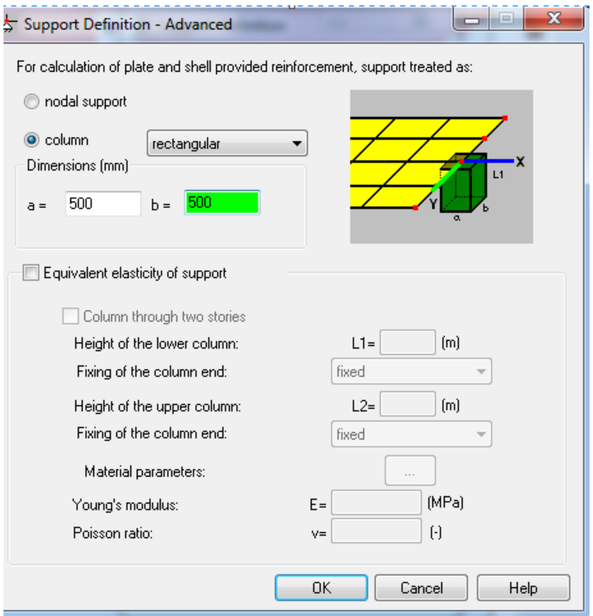






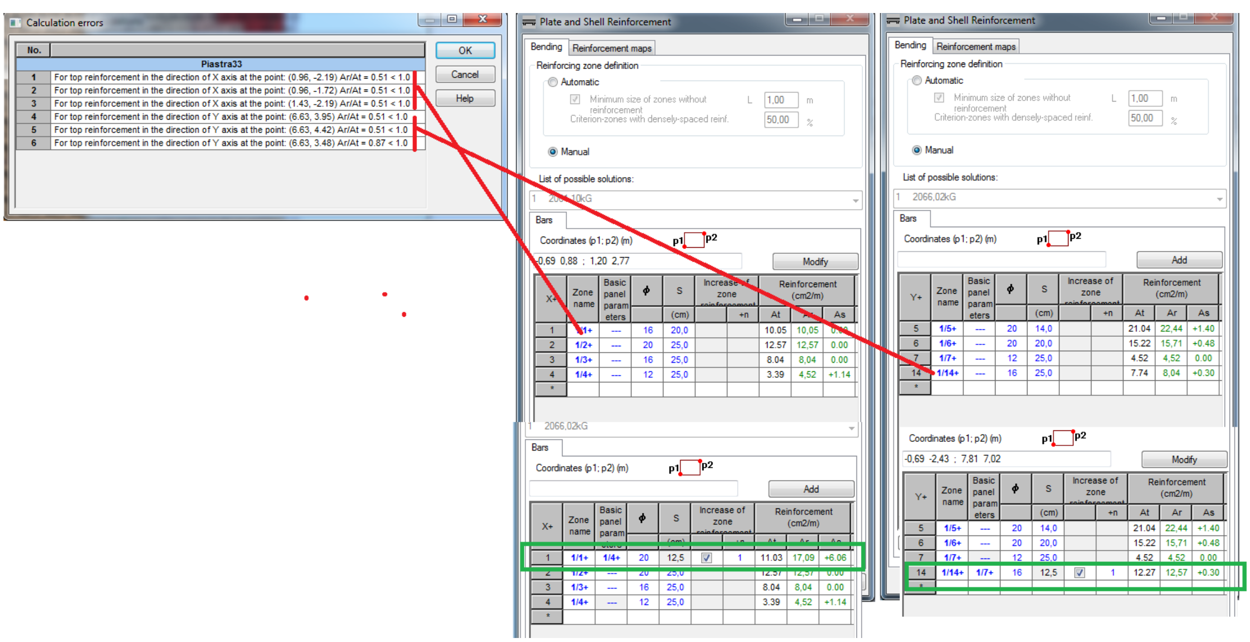
- Punching verification

The check is based on real support (column) size therefore it is required to provide this information when nodal supports are defined. This can be done in their advanced parameters.



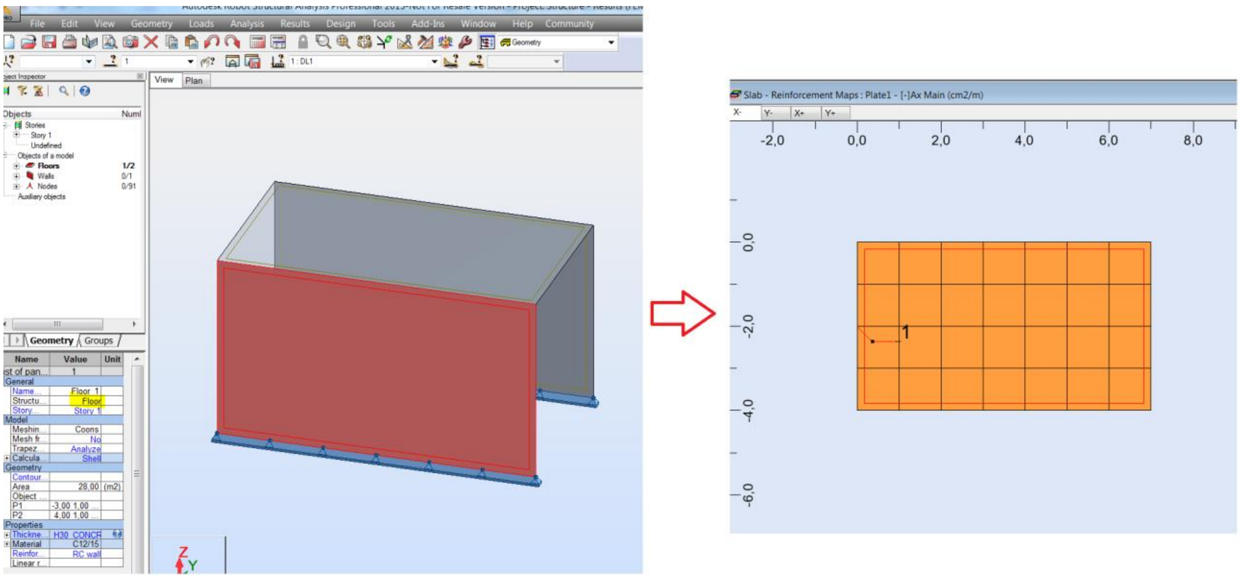
- Manual definition of reinforcement zones

As program deletes reinforcement bars which are generated in the same place it is recommended to indicate a basic panel while creating overlapping zones.

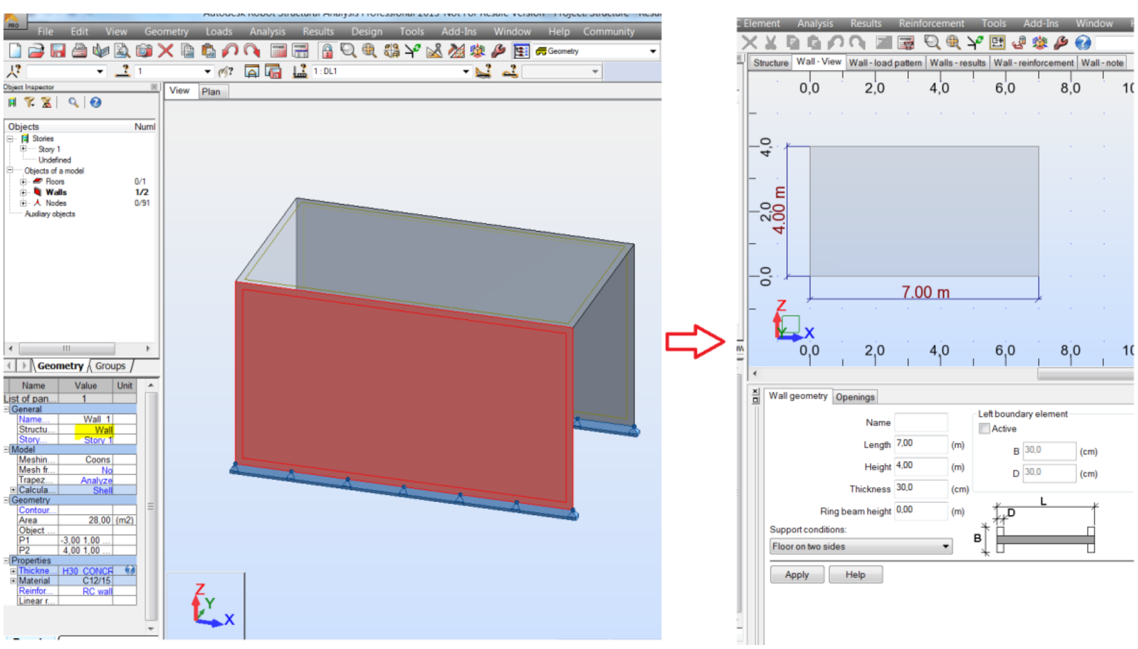


g) Design of RC Walls

The design of provided reinforcement in a vertical panel (wall) can be done either in the same way as for slab (RC Slab Required Reinforcement calculations in either compression/tension or bending + compression tension mode followed by the RC Slab Provided Reinforcement) when its Structure Object Type is set as Panel or Floor



or (for selected design codes) in the RC Wall Design module when a panel Structure Object type is set as Wall. In this case out of the plane bending is neglected.



Additional references to the discussions on these topics on the Robot Forum:

http://forums.autodesk.com/t5/robot-structural-analysis/provided-real-reinforcement-bar-diameter-list-how-to-add-new/td-p/3263604

http://forums.autodesk.com/t5/robot-structural-analysis/rc-beams-design-of-structure-beam-at-the-column-face/td-p/3229182

http://forums.autodesk.com/t5/robot-structural-analysis/beam-design/td-p/3486810

http://forums.autodesk.com/t5/robot-structural-analysis/rc-beams-primary-beam-secondary-beam-connection/td-p/3712360

http://forums.autodesk.com/t5/robot-structural-analysis/dimensionnement-poutre/td-p/4303006

http://forums.autodesk.com/t5/robot-structural-analysis/afp-and-support-reinforcement/td-p/3330609

http://forums.autodesk.com/t5/robot-structural-analysis/construction-reinforcement-in-beams/td-p/3370355

http://forums.autodesk.com/t5/robot-structural-analysis/robot-2013-crack-width-in-column-according-to-eurocode/td-p/3400611

http://forums.autodesk.com/t5/robot-structural-analysis/rc-column-design-iternal-calculation-nodes/td-p/3518270

http://forums.autodesk.com/t5/robot-structural-analysis/combined-footings/td-p/3187330

http://forums.autodesk.com/t5/robot-structural-analysis/combined-footing/td-p/4889394

http://forums.autodesk.com/t5/robot-structural-analysis/footing/td-p/3417443

http://forums.autodesk.com/t5/robot-structural-analysis/robot-continuous-foundation-design/td-p/5030176

http://forums.autodesk.com/t5/robot-structural-analysis/elastic-and-nonelastic-slab-deflection-verifications/td-p/3217594

http://forums.autodesk.com/t5/robot-structural-analysis/how-dose-reinforcement-be-increased-to-reduce-deflections/td-p/3269435

http://forums.autodesk.com/t5/robot-structural-analysis/minimum-reinforcement-on-slabs/td-p/3729178

http://forums.autodesk.com/t5/robot-structural-analysis/global-smoothing-and-reduce-forces-above-supports-methodology/td-p/3796129

http://forums.autodesk.com/t5/robot-structural-analysis/slab-punching-check/td-p/3792458

http://forums.autodesk.com/t5/robot-structural-analysis/rc-slab-punching-shear-check/td-p/4609147

http://forums.autodesk.com/t5/robot-structural-analysis/design-of-core-wall-by-using-robot/m-p/3596518

http://forums.autodesk.com/t5/robot-structural-analysis/design-of-core-wall-by-using-robot/m-p/3596518