Calculations in SCIA Engineer

Types of analysis, FE mesh, Soil-In, Seismicity, Plasticity, AutoDesign, Optimisation

Contacts

Belgium Headquarters SCIA nv Industrieweg 1007 B-3540 Herk-de-Stad Tel: +32 13 55 17 75 E-mail: info@scia.net Support Phone CAE (SCIA Engineer) Tel: +32 13 55 09 90 CAD (Allplan) Tel: +32 13 55 09 80 Support E-mail: support@scia.net	**France** SCIA France sarl Centre d'Affaires 16, place du Général de Gaulle FR-59800 Lille Tel.: +33 3.28.33.28.67 Fax: +33 3.28.33.28.69 E-mail: france@scia.net Agence commerciale 8, Place des vins de france FR-75012 Paris Tel.: +33 3.28.33.28.67 Fax: +33 3.28.33.28.69 E-mail: france@scia.net
Brazil SCIA do Brasil Software Ltda Rua Dr. Luiz Migliano, 1986 - sala 702 , CEP SP 05711-001 São Paulo Tel.: +55 11 4314-5880 E-mail: brasil@scia.net	**Germany** SCIA Software GmbH Technologie Zentrum Dortmund, Emil-Figge-Strasse 76-80 D-44227 Dortmund Tel.: +49 231/9742586 Fax: +49 231/9742587 E-mail: info@scia.de
Netherlands SCIA Nederland B.V. Wassenaarweg 40 NL-6843 NW ARNHEM Tel.:+31 26 320 12 30 Fax.: +31 26 320 12 39 E-mail: info@scia.net	**Switzerland** SCIA Swiss Office Dürenbergstrasse 24 CH-3212 Gurmels Tel.: +41 26 341 74 11 Fax: +41 26 341 74 13 E-mail: info@scia.ch
Czech Republic SCIA CZ s.r.o. Praha Evropská 2591/33d 160 00 Praha 6 Tel.: +420 226 205 600 Fax: +420 226 201 673 E-mail: info.praha@scia.cz SCIA CZ s.r.o. Brno Slavickova 827/1a 638 00 Brno Tel.: +420 530 501 570 Fax: +420 226 201 673	**Slovakia** SCIA SK s.r.o. Murgašova 1298/16 SK-010 01 Žilina Tel.: +421 415 003 070 Fax: +421 415 003 072 E-mail: info@scia.sk

E-mail: info.brno@scia.cz	
Austria SCIA Datenservice Ges.m.b.H. Dresdnerstrasse 68/2/6/9 A-1200 WIEN Tel.: +43 1 7433232-11 Fax: +43 1 7433232-20 E-mail: info@scia.at Support Tel.: +43 1 7433232-12 E-mail: support@scia.net	

Document created: 06 / 05 / 2018

SCIA Engineer 18.0

Introduction to calculation

Once the model of an analysed structure is created, the calculation of required type may be performed.

SCIA Engineer applies the deformation variant of finite element method. The employed beam finite element takes account of shear deformation.

Detailed information about the applied calculation methods may be found:

- in the following chapters and
- in a separate book Advanced calculations accessible via menu function Help > Contents > Advanced calculations.

Checking the data

Introduction to check of data

It is a good practice and sometimes even necessity to check the data of the model from time to time or at least before calculation. Especially for excessive models that have been modified by means of various manipulation functions, it may happen that the model contains some invalid or obsolete data. Such data should be removed from the project as they:

- occupy memory unnecessarily,
- could mislead some functions.

SCIA Engineer provides aN easy-to-use wizard that automatically searches the project and reveals improper or invalid data.

> Note : The check of data is important from one more point of view. By default the intersecting 1D members are not joined to each other. If they are supposed to act together, a linked node must be defined in their intersection. The Check of data function traces such places and suggests the user to make an automatic connection of affected 1D members. This operation may thus resolve possible future problems with numerically unstable solution.

Parameters of data check

The Check data function tries to reveal invalid data in the project.

Check of nodes

Search nodes	This option is ALWAYS ON. This check ensures that nodal data are correct. This option is a kind of protection against possible damage of saved data.
Search duplicate nodes	If ON, the program searches for nodes with identical co-ordinates If two nodes of identical position are found they are merged into a single node (i.e. one of them is removed). The value defined in Minimal distance between two points in the Mesh Setup dialogue is used for this check.
Ignore parameters	This option is effective only if parametric nodes have been defined in the project. If ON, only the co-ordinates (calculated from input parameters) are checked. If two nodes of the same co-ordinates are found, they are merged into one node. If OFF, the check procedure consists of two steps. First, the co-ordinates are checked. If any two nodes of the same co-ordinates are discovered, the defining parameters are check in the second step. If the two nodes are defined by means of the same parameters, they are considered duplicate and merged into one. If, however, the two nodes are defined using different parameters or different formulas, the nodes are let unchanged.

If the Check of nodes discovers any disorder or "mess" in nodal data, another dialogue is displayed.

Members with	This item shows the number of discovered undefined nodes. Such nodes MUST always be corrected

undefined nodes	and therefore the checkbox is ALWAYS ON.
Free nodes	If any free nodes are found in the project (i.e. nodes that do not belong to any member) the user may delete them. It is recommended to delete any free nodes unless the user has a specific reason for their existence in the project (e.g. free nodes may represent a temporary state during the definition of a complex model).
Duplicate nodes	Any duplicate nodes found in the project are reported here and it up to the user whether they will be deleted or not. It is recommended to delete duplicate nodes.

Check of 1D members

Check beams	The user may decide if 1D members in the project should be checked or not.
Search null beams	1D members of zero length are found. If such 1D members are discovered in the project, they are always deleted.
Search duplicate beams	This check goes through the model and traces double 1D members, i.e. 1D members of identical position, orientation and length. If such 1D members are discovered, the user may decide whether they should be preserved or whether only one of the identical 1D members should be kept in the project.

Note: Any two 1D members are considered identical if they have identical end nodes. If two different 1D members defined by means of four different end nodes "lie" one on another, they are not identical under the terms of this check. However, if standard check options are selected, the check procedure discovers duplicate nodes first, merges them, and consequently also the two 1D members become identical under the conditions of the check.

Check of structure

Contrary to original versions of SCIA Engineer, version 5 DOES NOT perfom the check of structure within this function. That means that any problems in connection of "touching" members are not solved by this function.

A separate function Connect members/nodes must be used for this task. The function can be found in tree menu Calculation, Mesh; on toolbar Geometry manipulation; or in menu Modify.

Check of additional data

Check additional data position	The program checks all additional data (e.g. loads, supports, etc.) and verifies the position of these data on members. For example, some loads might have got out of 1D member during manipulation functions. Such improper data are corrected.

Note: For the procedure read chapter Performing the check of data.

Performing the check of data

The procedure for the check of data

1. Start function Check of data:
    1. either using menu function Tree > Calculation, mesh > Check structure data,
    2. or using tree menu function Calculation, mesh > Check structure data.
2. The Check data wizard opens the setup dialogue on the screen.
3. Select the data types that should be searched and verified.
4. Start the check with button [Check].
5. The program scrutinises all the project data.
6. If no disproportion is revealed a message telling that no problems have been found is issued.
7. If something suspicious has been discovered, the wizard displays the statistics in the dialogue. Numbers of invalid entities for individual data types are stated.
8. Now, decide which data types should be corrected and which ones left unchanged (i.e. put a tick to the data type that should be corrected and remove the tick from those types that should be skipped during the correction phase).
9. Finish the Data check with button [Continue].
10. The invalid data are removed from the project.

Collision between entities

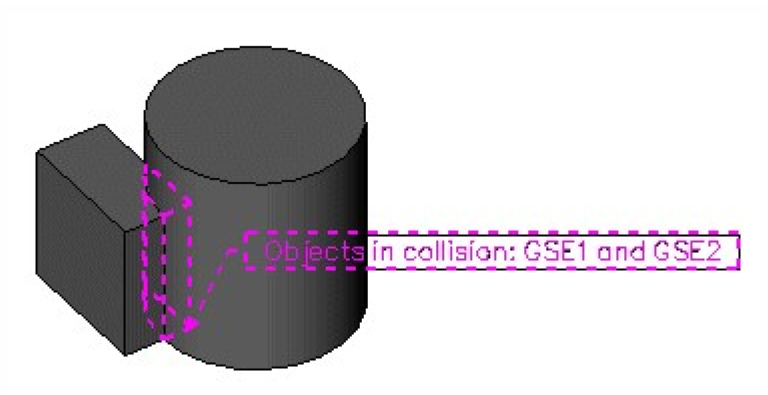
Sometimes you may need to find out if specific entities do or do not intersect each other. This can be verified through function Clash check of solids.

The function can process all types of entities: 1D members (beam, column, etc.), 2D members (plate, wall, etc.), general components (solid, open shell, etc.).

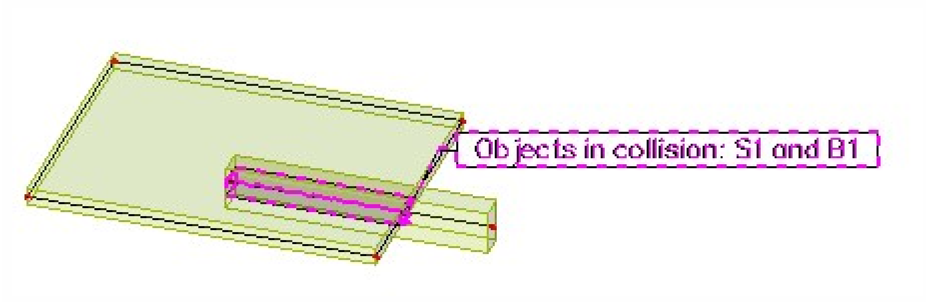
The function checks the selected entities and generates new entities (general components / solids) that correspond to the intersection of the selected entities. The original entities remain unaffected.

The following pictures demonstrate the use of the function.

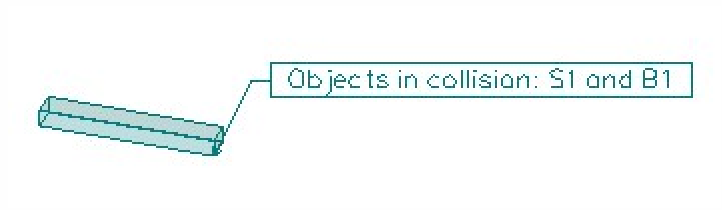
The first picture shows the result of the check on two solids (cylinder and prism).



The second picture shows the same for 1D member (beam) and 2D member (slab).



The last picture demonstrates the existence of the newly generated solid in the intersection of the checked entities. Here, the beam and slab from the previous picture were removed. What remains is a new entity (general solid) representing the intersection of the two above-mentioned entities.



The function can be used to check one or two groups of entities.

Check of one group of entities

If just one group of entities is selected, all the selected entities are checked if they collide with any other entity from the selection.

Check of two groups of entities

If two groups of entities are selected, the function checks whether any entity from the first group collides with any entity from the second group. If two entities in the same group collide, it is not reported.

Clash check parameters

Show tree of clashes
If ON, a list of clashing entities is displayed in a separate floating window. When a particular entity or clash is clicked in this tree, the corresponding entity or clash is highlighted in the graphical window.

Show collision description
If ON, each collision is displayed including a label.

Show transparent structure
If ON, the structure is displayed as transparent to allow for clearer view of the clashes.

Type of selection
Each group for the clash check can be defined as a (i) user selection (i.e. the user must manually select the required entities), (ii) layer (i.e. all the entities from the selected layer are in the group), (iii) named selection (i.e. the entities from the selected named selection are in the group), or (iv) element type (i.e. all the entities of the selected type, such as 1D members, are in the group).

The procedure to check the collision of entities

1) Open branch BIM toolbox and start function Evaluate - Clash check.

2) Select the entities for the first group to be checked.

3) Press [Esc] to complete the selection of the first group.

4) Select the entities for the second group to be checked. If only one-group check is requested, simply ignore this step

5) Press [Esc] to complete the selection of the second group.

6) The collisions are displayed on the screen. Moreover, they are selected.

7) If required, clear the selection, or do whatever necessary with the collisions.

> Note: This function also checks collisions between free reinforcement bars. For more information on free bars read the documentation for Concrete Code Check.

Generating the FE mesh

Parameters of FE mesh

The user may control the shape of the finite element mesh. The Mesh setup dialogue offers a whole range of parameters.

Mesh

Minimal distance between definition point and line	If the distance between a definition point (a structural node) and a definition line (2D member edge or 1D member centre line) is lower that the value specified here, a mesh node in the definition point and a mesh node on the definition line are automatically merged into a single mesh node.
Average size of 2D elements / curved members	The average size of edge for 2D elements. The size defined here may be altered through refinement of the mesh in specified points. Defines also the size of finite elements generated on curved members.
Average number of tiles of 1D element	If required, more than finite element may be generated on a single 1D member. The value here specifies how many finite elements should be created on a 1D member. The value is taken into account only if the original 1D member is longer than adjusted minimal length of beam element and shorter than adjusted maximal length of beam element. This option is useful mainly for stability, non-linear and dynamic calculations where more than one finite element is required per a structural member.
Definition of mesh element size for panels	Automatic: The mesh size is determined automatically according to the model Manual: This size is applied only to FEM method calculation of load panels and for the generation of loads for individual beams or edges. This mesh is not used for the main calculations of the whole model.
Average size of panel elements	(available for Manual definition in the item above) Defines the mesh size for load panels.
Elastic mesh	If ON mesh on 2D members is generated much variable than in previous versions. It also allows to use automatic mesh refinement functionality.
Use automatic mesh refinement	If ON mesh on 2D members can be refined automatically in places where is necessary for better results.

1D elements (1D members)

Minimal length of beam element	If a 1D member of a structure is shorter than the value here specified, then the 1D member is no longer divided into multiple finite elements even though the parameter above (Average number of tiles of 1D element) says so.
Maximal length of beam element	If a 1D member of a structure is longer than the value here specified, then the 1D member will be divided into multiple finite elements so that the condition of maximal length is satisfied.

Average size of cables, tendons, elements on sub-soil	It is necessary to generate more than one finite element on cables, tendons (prestressed concrete) and 1D members on subsoil. For more information about this issue see book Advanced calculations, chapter Analysis of a beam on elastic foundation versus mesh size. NOTE: This parameter also controls the size of finite elements for beams with a phased cross-section.
Generation of nodes in connections of beams	If this option is ON, a check for "touching" 1D members is performed. If an end node of one 1D member "touches" another 1D member in a point where there is no node, the two 1D members are connected by a FE node. If the option is OFF, such a situation remains unsolved and the 1D members are not connected to each other. The function has the same effect as performing function Check of data.
Generation of nodes under concentrated loads on beam elements	If this option is ON, finite elements nodes are generated in points where concentrated load is acting. This option is not normally required and it is off as default.
Generation of eccentric elements on members with variable height	If a beam is of variable height, the generator automatically generates eccentric finite elements along the haunch. Moreover, if this option is ON, the eccentricity of the elements may vary along the element, i.e. the start-node of the element may have different eccentricity than the end-node of the element. If this option is off, the eccentricity along individual finite elements is constant and the eccentricity changes in steps in nodes along the haunch.
Division on haunches and arbitrary members	Specifies the number of FE generated on a haunch.
Division for 2D - 1D upgrade	Specifies the number of section which are generated on beam after 2D-1D upgrade performance.
Mesh refinement following the beam type	Specifies the mode of refinement on 1D members. None The refinement is applied to 2D members only. Beams and columns The refinement applied to 1D members the type of which is adjusted to "beam (80)" or "column (100) All 1D members The refinement applied to all 1D members.

2D elements (slabs)

To generate predefined mesh	If ON, the generator first tries to generate in every slab a regular quadrilateral finite element mesh complying with the adjusted element-size parameters. Only if required, additional necessary nodes are added to the mesh. If OFF, the finite element mesh nodes are generated across the slab and the nodes are the elements are then created from the nodes.
Maximal out	This value determines whether a spatial quadrilateral whose nodes are not in one plane will be replaced

of plane angle of a quadrilateral	by triangular elements. This parameter is meaningful only for out-of-plane surfaces – shells. The assessed angle is measured between the plane made of three nodes of the quadrilateral and the remaining node of this quadrilateral.
Predefined mesh ratio	Defines the relative distance between the predefined mesh formed by regular quadrilateral elements and the nearest edge. The edge may consist of an internal edge, external edge or border of refined area. The final distance is calculated as a multiple of the defined ratio and adjusted average element size for 2D elements.

Average number of tiles of 1D element

For static linear calculation, value 1 is normally satisfactory. On the other hand, there are several configurations when finer division is required in order to obtain accurate results.

These are:

- beam laid on foundation requires a fine division – see chapter Analysis of a beam on foundation versus mesh size.
- dynamic calculation when a great number of eigenfrequencies is required – see chapter Natural vibration analysis versus mesh size
- buckling calculation
- non-linear calculations

Division on haunches and arbitrary beams

The number defined here determines the "precision" that is applied in modelling of the variable cross-section along a haunch. The higher the number, the more the model reflects the real shape. See chapter Analysis of a haunch versus mesh size.

In addition, the same rules as for a standard member must be followed here as well.

Generation of eccentric elements on members with variable height

The midline of the finite element model of a haunch-beam may be either straight (following the midline of the original "non-haunched" beam) or "curved" (following the real midline of the haunch beam corresponding to the centre of gravity of the cross-section).

See chapter Analysis of a haunch versus eccentric elements.

Average size of cables, tendons, elements on subsoil

Specifies the number of finite elements generated on a beam laid on foundation.

See chapter Analysis of a beam on foundation versus mesh size.

The procedure for the adjustment of mesh parameters

1. Call menu function Setup > Mesh.
2. Adjust the parameters (see above).
3. Confirm with [OK].

The finite element mesh may be previewed using function Mesh generation under tree menu Calculation.
07/10/2013

Previewing the FE mesh

For complex structures it may be useful to review the FE mesh before the results are scrutinised in detail.

It is possible to control the display style of the mesh through a set of view parameters.

Tab Structure > Group Mesh

Draw mesh	If ON, the mesh is displayed on the screen.
Free edges	Free edge is an edge of a 2D element that is not connected to any other element. It may be useful to see which parts of the structure are not connected to the rest of the model. If this option is ON, the free (unconnected) edges of 2D finite elements are highlighted using a thick line. This option is independent on the option above.
Display mode	The user may decide about the drawing style for the mesh (wired, rendered, transparent). Note: Rendered and transparent option may affect the adjustment of colours for symbols relating to the mesh (e.g. local axes).

Tab Structure > Group Local axes

Nodes	If ON, the program displays local axes of the nodes in the generated finite element mesh.
Mesh elements	If ON, the program displays local axes of the generated finite elements.

Tab Labels > Group Mesh

Display label	If ON, the selected labels are displayed together with the mesh. Note: If parameter Draw mesh from tab Structure - group Mesh is OFF, no labels are displayed.
Nodes	If ON, the numbers of nodes are displayed.
Elements 1D	If ON, the numbers of 1D finite elements are displayed.
Elements 2D	If ON, the numbers of 2D finite elements are displayed.

Tab Labels > Group Labels of local axes

Nodes	If ON, the labels (x, y, z) of node local axes are displayed.
Mesh	If ON, the labels (x, y, z) of finite element local axes are displayed.

The procedure for the preview of finite element mesh

1. Open View parameters setting dialogue.
2. Select Tab Structure or Labels.
3. In the required group adjust the required parameters.
4. Confirm the settings.
5. Check the mesh.
6. If required, switch the mesh off again.

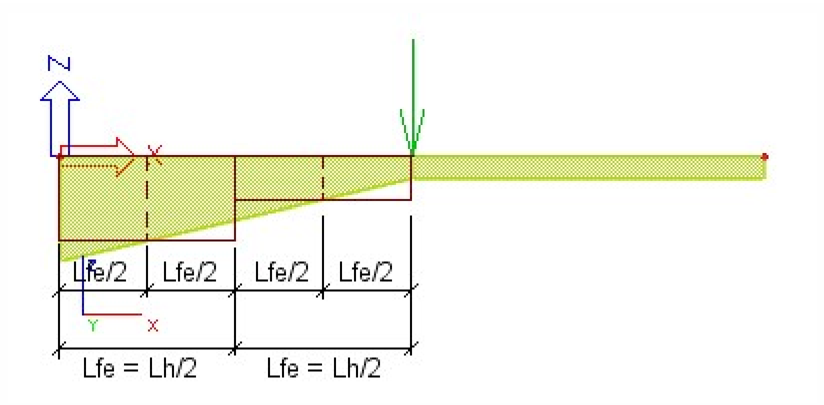
> Note: When the mesh generator has created mesh elements with an angle smaller than 5° an arrow is shown on the screen so the user can easily find such elements (elements with the angle smaller than 5° can sometimes lead to inaccurate results) Mesh refinement function is then recommended to generate a better mesh..

Analysis of a haunch versus mesh size

For haunch, the division of members into finite elements may play a significant role.

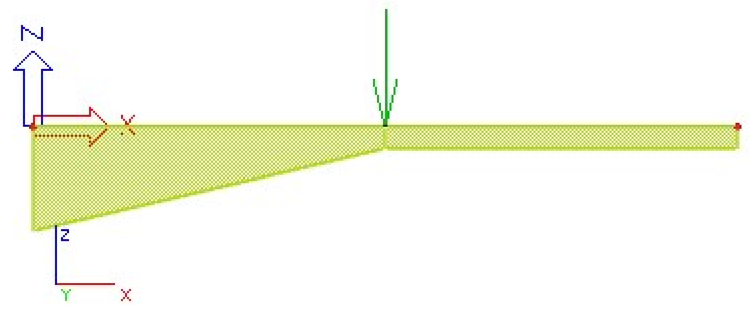
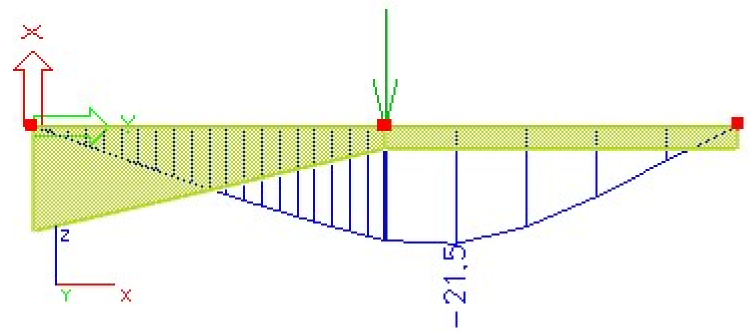
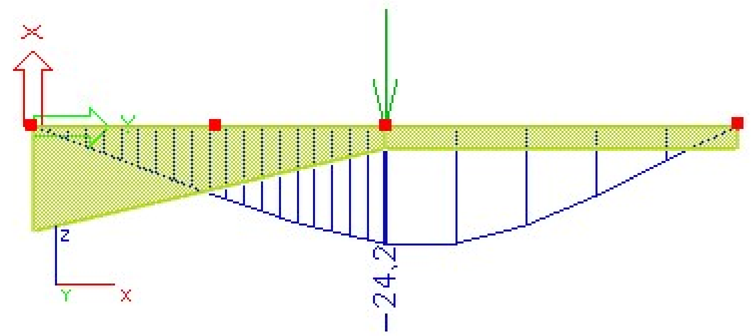
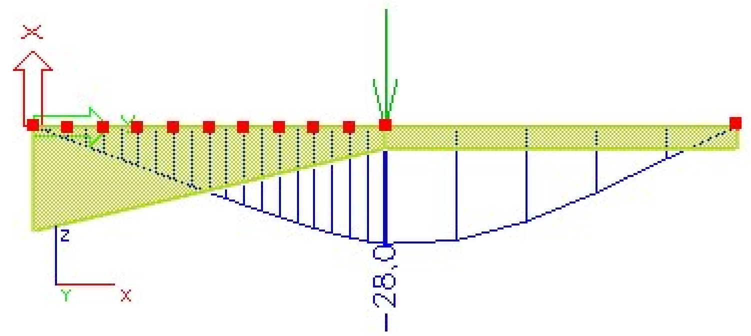
A haunch is an element of a variable cross-section. A 1D finite element used in SCIA Engineer, on the other hand, is an element of a constant cross-section. Therefore, the effect of the varying cross-section (most often of a varying depth of the cross-section) must be modelled by means of a finer finite element mesh.

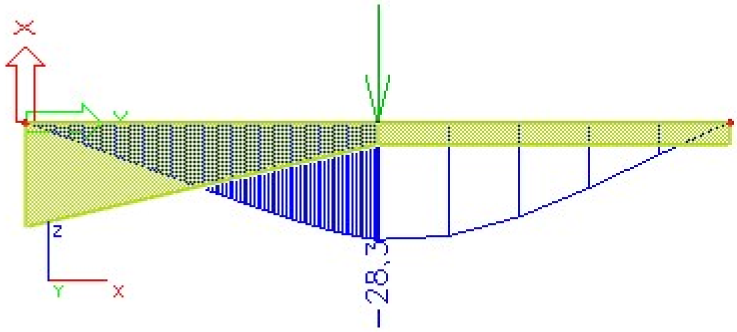
The practical application is shown in the figure below.



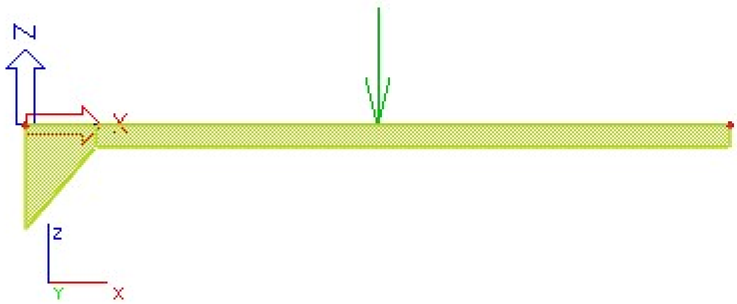
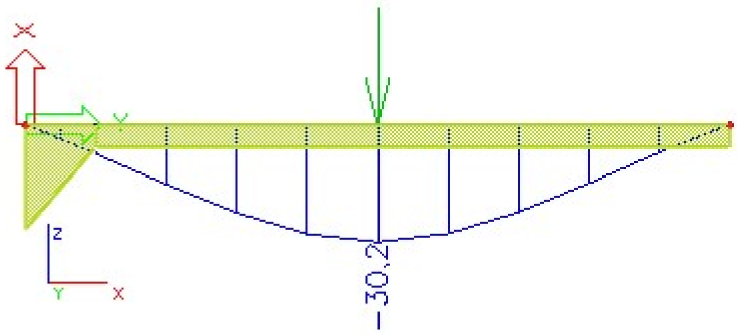
The example shows a beam with a haunch stretching over a half of total beam length. Let’s assume that the division is set to "2 finite elements per a haunch". When the finite element mesh is being generated, each haunch is cut into the specified number of segments, i.e. into two segments in our example. Then, the dimensions of the cross-section in the middle of each of the segments are calculated. These dimensions are used to create an ideal cross-section of the corresponding finite element.

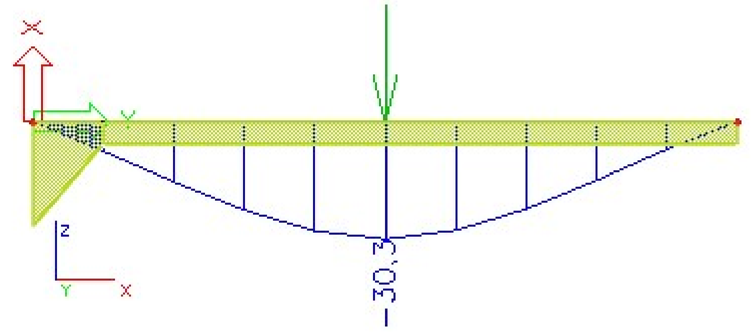
The approach presented above means, that the higher the number of finite elements per a haunch is, the more realistic model of the haunch is obtained. On the other hand, from a practical point of view, it is not necessary to generate "over-precise" haunches. The gain in the numerical precision is not in proportion to the number of finite elements per haunch. There is a big difference in the precision of results for very course division and for considerably fine division. But the difference between the considerably fine division and extremely fine division is almost negligible. Compare the deformation calculated for division equal to 1, 2, 10 and 50 finite elements per haunch.

analysed beam	
1 FE per haunch (deformation given in millimetres)	
2 FEs per haunch (deformation given in millimetres)	
10 FEs per haunch (deformation given in millimetres)	

50 FEs per haunch (deformation given in millimetres)	

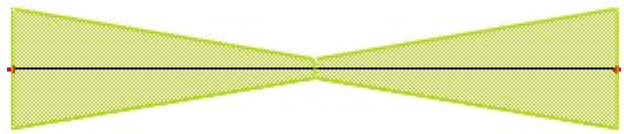
What's more, the above stated facts are not applicable to all cases. What also influences the "reasonable" division is the relative length of the haunch. If the haunch extends along a considerably smaller part of the beam, the required number of finite elements per haunch decreases. See another example.

analysed beam	
1 FE per haunch (deformation given in millimetres)	

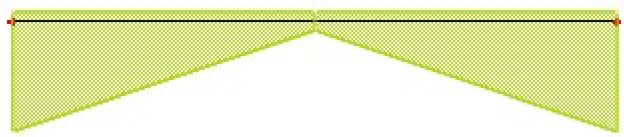
10 FEs per haunch (deformation given in millimetres)	

Analysis of a haunch with reference to eccentric elements

By default, a haunch is idealised by a set of finite elements that vary in cross-section from one element to another and whose middle axes lie in one line. This idealisation corresponds fully with a haunch whose midline is straight and whose both surfaces are inclined (see Fig.).

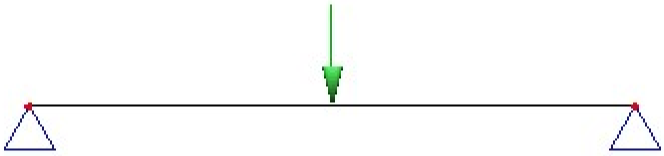


In practice, however, one more often comes across a haunch with an aligned top or bottom surface (see Fig.).

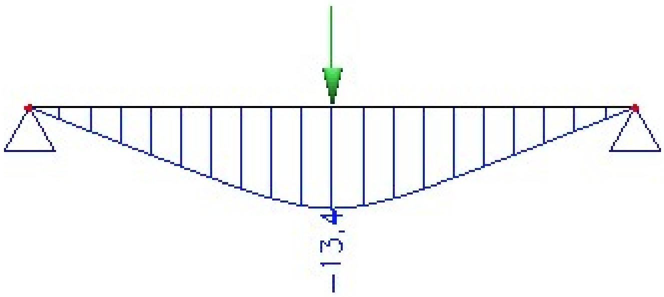


In this second case, the midline of the beam is not a straight line but it resembles an arch. This produces an arch effect whose practical outcome is shown in the following set of pictures.

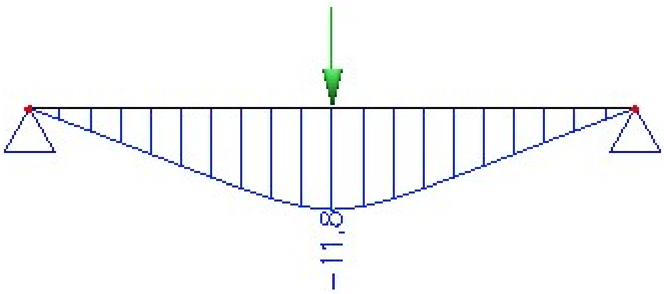
Let's assume a simply supported beam with both ends pinned subjected to a concentrated force load located in the middle of the span.



The first finite element model (option "Generate eccentric elements on haunches, arbitrary beams" is OFF) gives displacement in the middle of the span equal to 13.4 mm.

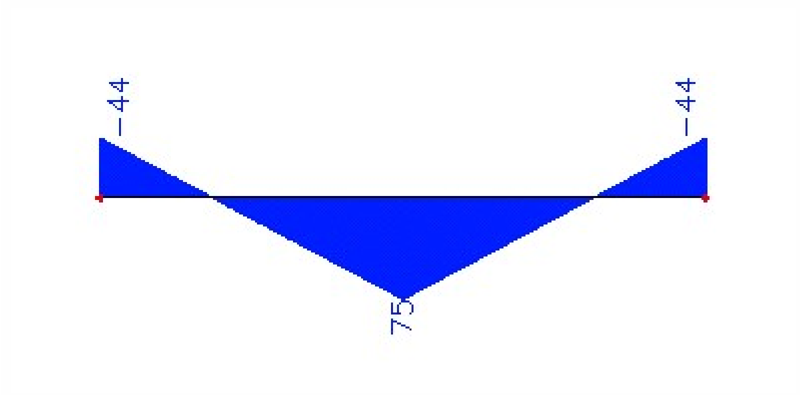


If however, the option "Generate eccentric elements on haunches, arbitrary beams" is set ON, the result displacement measured in the same place is only 11.8 mm.



The difference is about 12 percent, which is quite significant.

One must be aware of a "side effect" of the latter approach. Consider once again our model example of a beam pinned on both ends. None of the hinges provides for a horizontal movement. The midline of the beam is an "arch-like" curve and this means that under given loading conditions axial force appears in the beam. As there is an eccentricity introduced into the model, there will be a bending moment in both end-points of the beam. The moment in the support will be equal to the product of calculated axial force and introduced eccentricity.



Axial force diagram looks like:

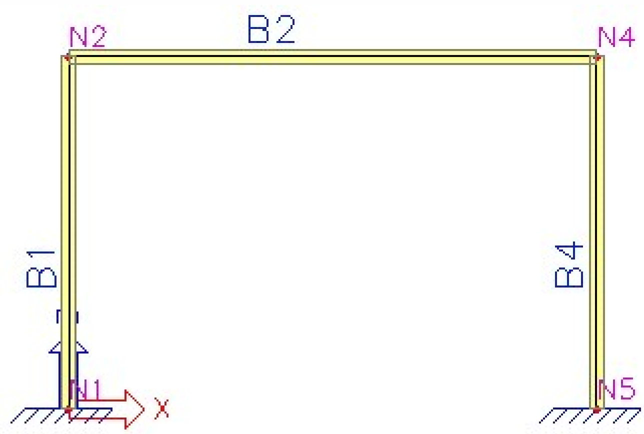
The depth of the haunch on its left-most side is equal to 1 metre. An easy calculation gives:

axial force * eccentricity equal to a half of the haunch depth = bending moment

(88) * (1/2 * 1.0) = 44

Natural vibration analysis versus mesh size

A natural vibration calculation is not a very complex problem. Nevertheless, one important point should be emphasised. In order to obtain greater number of eigenmodes (and natural frequencies as well) finer mesh of finite elements must be used. This must be fulfilled for both beam and plane elements. The effect of mesh density will be demonstrated on a simple planar frame.



The frame consists of three beam members. The default mesh division (i.e. the generation of a single finite element per a beam) would lead to a numerical model containing there are three finite elements and four nodes from which two nodes are supported. The structure has six degrees of freedom if solved as a 2D problem (Frame XZ project). The degrees of freedom are a vertical translation, a horizontal translation and a rotation in each of the corners. Therefore, six eigenmodes at most may be calculated for such a structure. If more eigenmodes than degrees of freedom are required a warning is issued and the calculation is terminated. Moreover, the calculation is abnormally terminated even if six eigenmodes are required since the algorithm works internally with increased number of eigenmodes. The fact that the higher eigenmodes are not calculated in our example has a significant advantage. If they were calculated they would show a significant numerical error caused by the coarse finite element mesh. Thus, the calculated results for the higher eigenmodes would be practically unusable. As a result, if higher natural frequencies should be calculated, finer mesh must be generated for the structure. The comparison of results for both coarse and fine finite element mesh is shown below. It can be clearly seen that while the first eigenmode is almost identical for both variants the other one differs.

coarse mesh

1 FE per beam)

Eigen frequencies

Eigen frequencies

Mass combination : CM1

N	f	omega	omega^2	T
-	[Hz]	[1/sec]	[1/sec^2]	[sec]
1	10,06	63,22	3996,97	0,10
2	33,81	212,43	45126,31	0,03

finer mesh

4 FEs per beam)

Eigen frequencies

Eigen frequencies

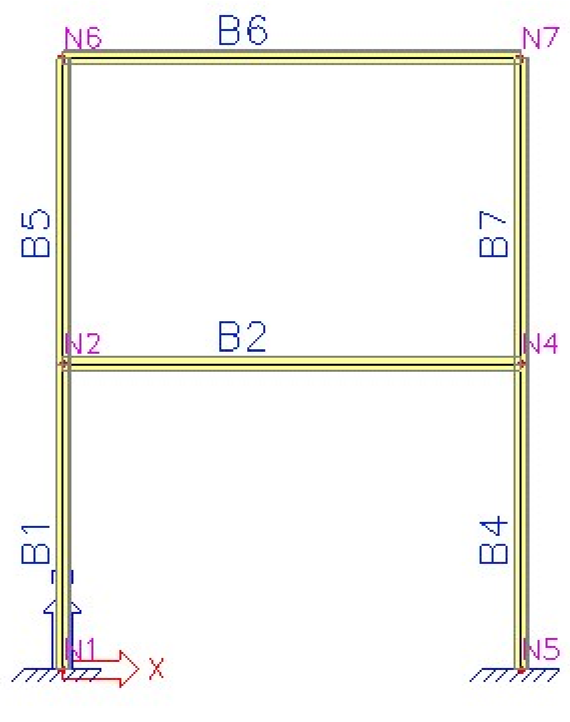
Mass combination : CM1

N	f	omega	omega^2	T
-	[Hz]	[1/sec]	[1/sec^2]	[sec]
1	10,05	63,15	3987,32	0,10
2	25,50	160,19	25661,14	0,04

Name

For instance, the deflection of the horizontal beam is completely different due to the absence of the node (mass degree of freedom) in the middle of the span. The natural frequency of the frame with lower number of degrees of freedom is higher just due to a lack of mass degrees of freedom, i.e. due to smaller portion of inertia energy in the total deformation energy of the system. This error grows rapidly with an increasing frequency.

A similar result may be examined for another example. Now, let's consider a two-storey frame:



The effect of the finite element size is now even more significant as one frequency has been skipped in the calculation performed for the single-element-per-beam division.

coarse mesh

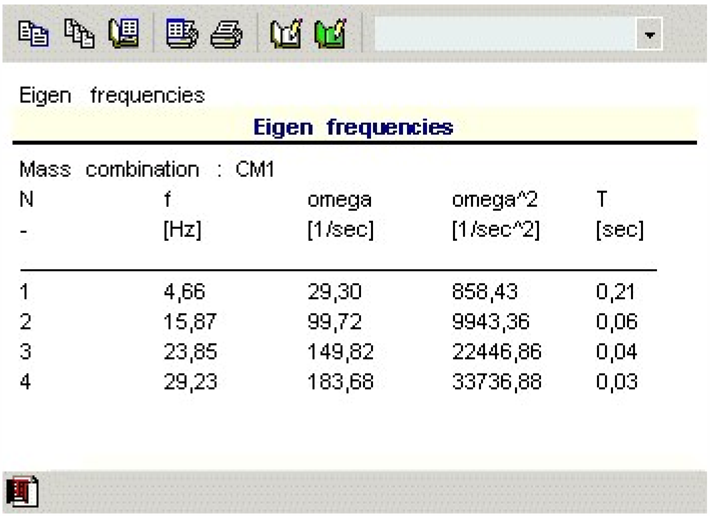
1 FE per beam)

Eigen frequencies

Eigen frequencies

Mass combination : CM1

N	f	omega	omega^2	T
-	[Hz]	[1/sec]	[1/sec^2]	[sec]
1	4,67	29,31	859,29	0,21
2	15,92	100,03	10005,28	0,06
3	29,65	186,28	34701,86	0,03
4	44,89	282,01	79531,66	0,02

finer mesh 4 FEs per beam)	

Eigen frequencies

Eigen frequencies

Mass combination : CM1

N	f	omega	omega^2	T
-	[Hz]	[1/sec]	[1/sec^2]	[sec]
1	4,66	29,30	858,43	0,21
2	15,87	99,72	9943,36	0,06
3	23,85	149,82	22446,86	0,04
4	29,23	183,68	33736,88	0,03

Analysis of a beam on elastic foundation versus mesh size

As stated earlier, finite element mesh where one 1D finite element corresponds to one beam member is satisfactory as far as the precision of results is concerned. However, and it was stated as well, there are exceptions to this rule.

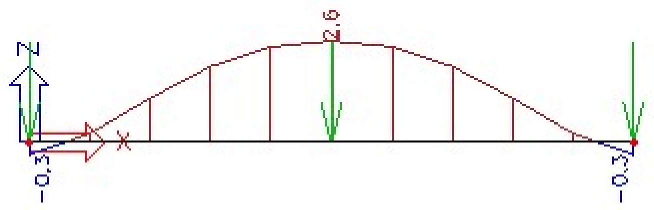
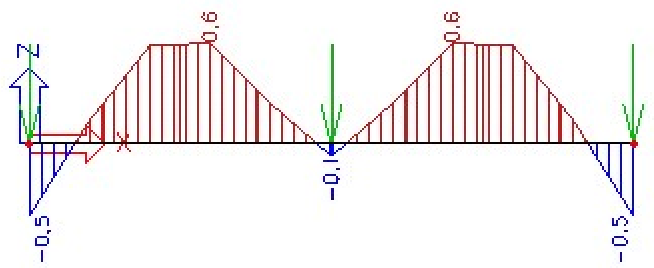
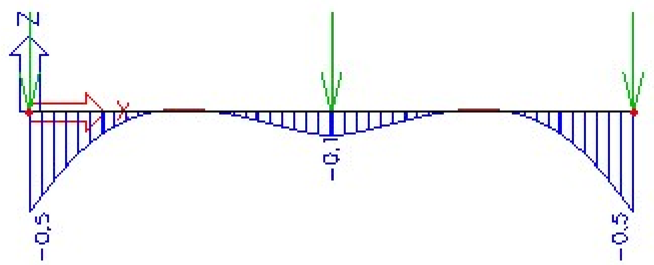
One of the exceptions is a beam laid on elastic foundation. The following table compares results for three different finite element divisions.

The table shown diagrams of vertical displacement, bending moment and shear force for three different meshes. The first one has got only one finite element on the beam. The other one has got two elements generated on the beam. The last one then shows results for a very fine mesh. It is clear that the distribution of both displacement and internal forces is considerably affected by the "coarseness" of the mesh. The reason is that for a beam supported by a foundation strip, the deflection curve (displacement diagram) is no longer a cubic parabola applied in the implemented finite element.

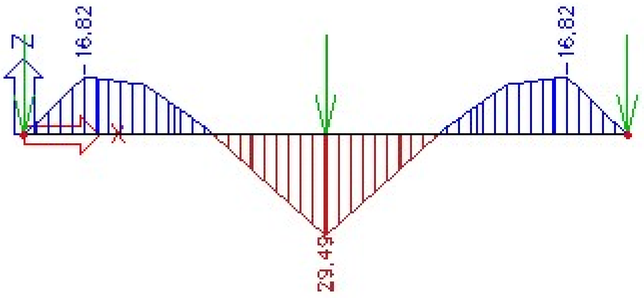
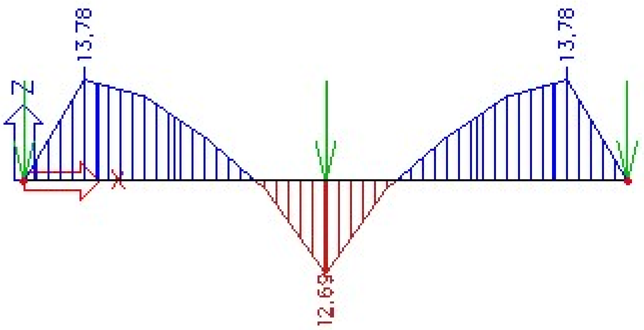
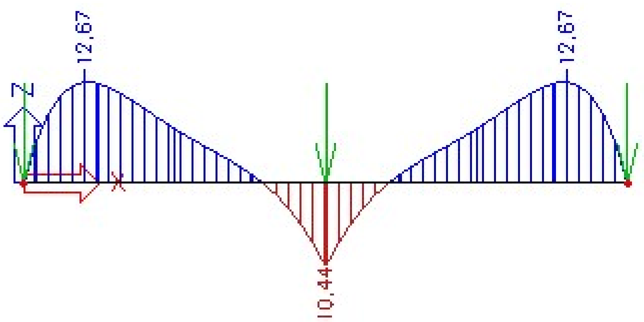
Therefore, it is important to remember that the finite element mesh fully sufficient for "standard" beams is completely unsuitable for analysis of members on foundation. The default settings reflect this phenomenon and are tailored for most of common structures. In some special cases however, additional, user-made, tuning of the mesh generation parameters may be necessary in order to obtain relevant and accurate results.

Vertical displacement

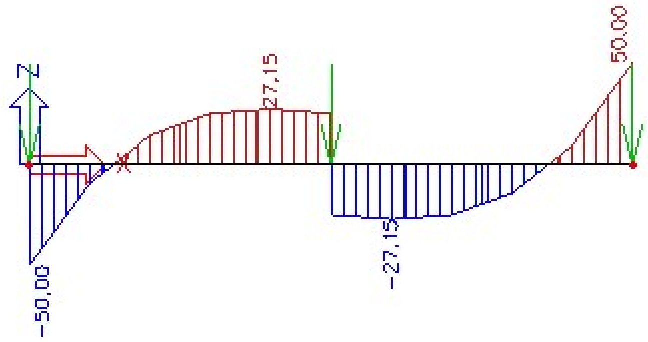
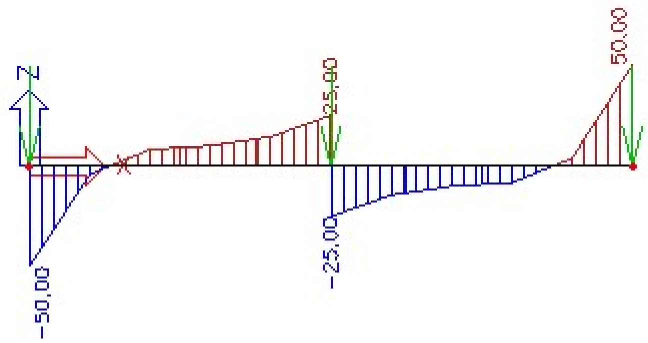
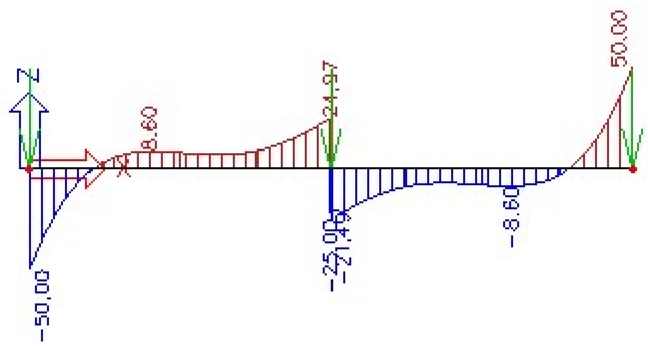
FE per beam	

1	2.6 Z X -0.3 -0.3
2	0.6 0.6 Z X -0.1 -0.5 -0.5
32	Z X -0.1 -0.5 -0.5

Bending moment

<table>
<tr><td>FE per beam</td><td></td></tr>
<tr><td>1</td><td>
</td></tr>
<tr><td>2</td><td>
</td></tr>
<tr><td>32</td><td>
</td></tr>
</table>

Shear force

FE per beam	
1	
2	
32	

Mesh refinement

Mesh refinement

The finer the finite element mesh is, the more accurate the obtained results are (i.e. the closer to the theoretically correct ones) and the more time consuming the solution is and the more disk space is needed both during the calculation and for storage of the results. The mesh size should be adjusted considering the load the structure is subject to and taking account of the requirements on the calculation.

The generation of the mesh is based on the adjusted size for 2D elements. The generator creates such elements whose edge size is as close to the adjusted value as possible. Also the division of slab / shell borders and internal edges is based on this. Any internal nodes of slabs / shells are taken into account as well.

The mesh must be made finer in certain areas. The mesh may be refines in a circular area around a specific point, in a band along a defined line or over the whole slab / shell.

If any two refinement areas overlap anywhere, the smaller element size is used. The refinement area does not have be fully inside the "master" slab /shell. Only a part of the refinement area may be located inside it.

Refinement around a node

The refinement area is circular with its centre in a specified point. The finite element size outside the circle is the standard FE size for 2D elements adjusted in FE mesh setup dialogue. The element size in the centre of the circle is the given refined value. The size of elements in between varies linearly from the two limits.

Name	Identifies the refinement.
Radius	Defines the radius of circular area where the mesh will be refined.
Ratio	Defines the ratio between the average element edge size in the centre of refinement area and the average preset element size.
dx, dy, dz	Defines possible shift of the centre of refinement area from the specified point. Thus the refinement area may be placed anywhere in the structure.
Type of point mesh refinement	Linear increment = elements at the end are larger, elements in the middle are smaller Equidistant - only 1D element = the elements are uniformly refined

The procedure for the adjustment of node refinement

1. Call function Node mesh refinement using tree menu function Calculation, mesh > Local mesh refinement > Node mesh refinement.
2. Adjust the parameters (see above).
3. Confirm with [OK].
4. Select nodes where the refinement should be used.
5. Close the function.

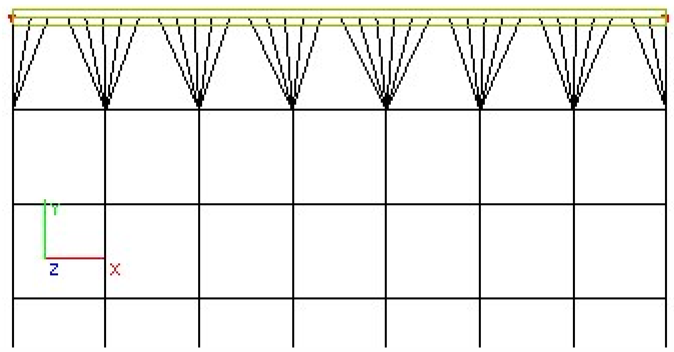
If you want to refine also mesh of 1D members , you must allow this in Mesh setup - see the Parameters_of_FE_mesh chapter.

Refinement along a line

The finite element size is reduced along the specified line.

Name	Identifies the refinement.
Size	Defines the size of refined elements.

Note : If this type of refinement is used without proper attention, it may result in really "strange" shapes of finite elements along the selected line. This may happen especially if the size along the line is too far from the standard element size that is used for other edges of the elements along the selected line (see the figure below).



The procedure for the adjustment of line refinement

1. Call function Line mesh refinement using tree menu function Calculation, mesh > Local mesh refinement > Line mesh refinement.
2. Adjust the parameters (see above).
3. Confirm with [OK].
4. Select the line along which the refinement should be used.
5. Close the function.

Refinement across an area

The finite element size is reduced over the specified area.

Name	Identifies the refinement.
Size	Defines the size of refined elements.

The procedure for the adjustment of line refinement

1. Call function Line mesh refinement using tree menu function Calculation, mesh > Local mesh refinement > Line mesh refinement.
2. Adjust the parameters (see above).
3. Confirm with [OK].
4. Select the regions over which the refinement should be used.
5. Close the function.

Calculation types

General calculation parameters

Any type of calculation can be controlled by means of a set of parameters.

Proper FEM analysis of cross-section parameters	If this option is ON, torsional constant and shear relaxation are calculated by means of finite element method for cross-sections defined as (i) general cross-section, (ii) geometric shapes or (iii) wooden sections.
Neglect shear force deformation	This setting has effects on beams. The option ON means ◦ The shear areas are equal to each other $A_x = A_y = A_z$ The option OFF means : ◦ Shear areas are computed exactly, using numerical integration. $A_y = \frac{A_x}{\beta_y}; A_z = \frac{A_x}{\beta_z}$ $\beta_y = \frac{A}{I_y^2} \int \frac{S_y^2}{t^2} dA > 1,0 - \text{effective shear coef.}$ NOTE: With this option ON, you get the same results for internal forces for SE and manual calculation.
Bending theory of plate/shell analysis	Defines if Kirchhoff or Mindlin banding theory is used.
Type of solver	Direct or iterative solution type may be selected.
Number of sections on average member	Defines the number of section for evaluation of results on a 1D member of "average length". Section is always created in both end points and under concentrated loads. The average length is determined from the real length. Shorter 1D members contain fewer sections while longer 1D members contain more sections.
Warning when maximal translation is greater than	If the maximal value of translation specified here is exceeded, the user is asked to confirm that s/he still want to review the results. This parameter is linear and nonlinear analysis .
Warning when maximal rotation is greater than	If the maximal value of rotation specified here is exceeded, the user is asked to confirm that s/he still want to review the results. This parameter is linear and nonlinear analysis .
Print time in Calculation Protocol	If ON, the calculation report contains start and end time for performed calculations. If OFF, this information is not stated.

Note: The adjustment of these parameters may affect the layout of the calculation dialogue that opens on the screen when a calculation is started.

Note: For Mindlin bending theory of shell is used a mixed interpolation of transverse displacements, section rotations and transverse shear strains. It is an extension of a pure displacement formulation. For curvatures there is used an interpolation as the pure displacement on based elements but for shear strain are used another one. The mixed elements are not locked and do not contain any spurious zero energy mode.

Number of result sections per member

The principle of finite element method is that the solution of the problem (in other words, the internal forces and deformations in the analysed structure) is given in finite number of points, i.e. in nodes of finite elements. These values may be further processed and result values for intermediate points of individual finite elements may be interpolated. In SCIA Engineer the user may decide how many intermediate points should be evaluated. This is made by means of solver option: **Number of sections on average member**.

When adjusting this option, one should remember that:

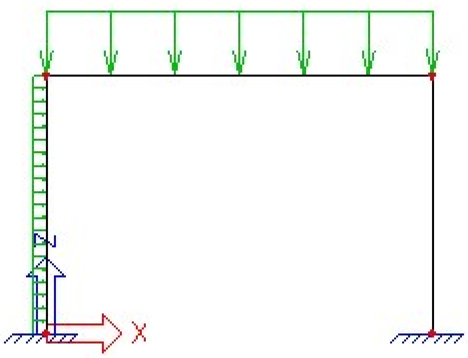
too few intermediate points:

- generates little data, saves the computer memory, increases the speed of the programme,
- may more or less distort the results.

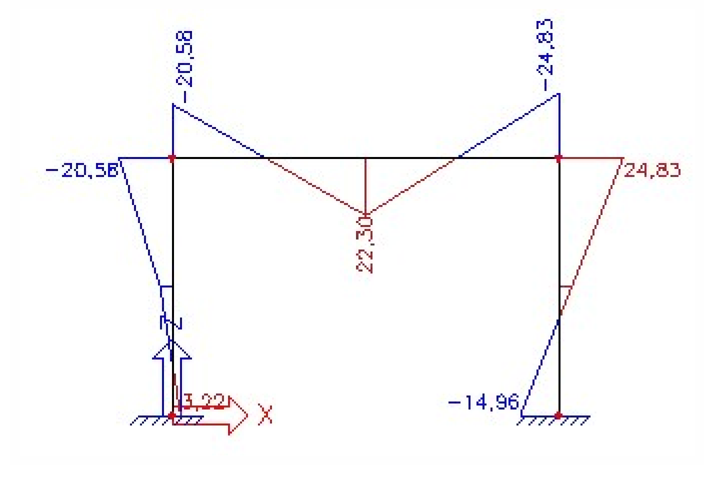
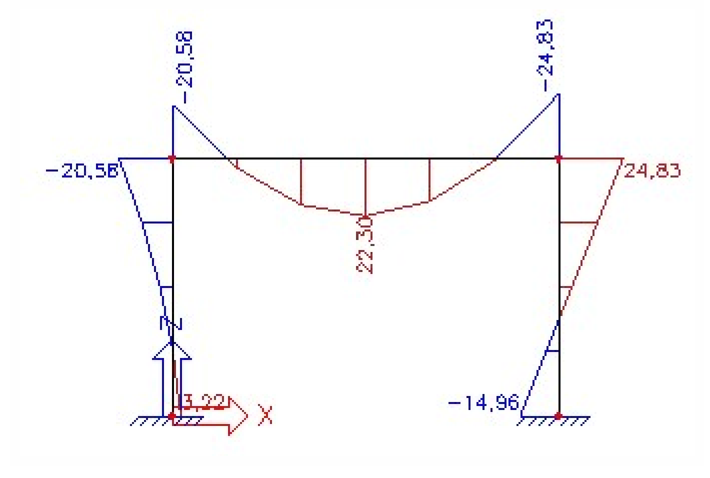
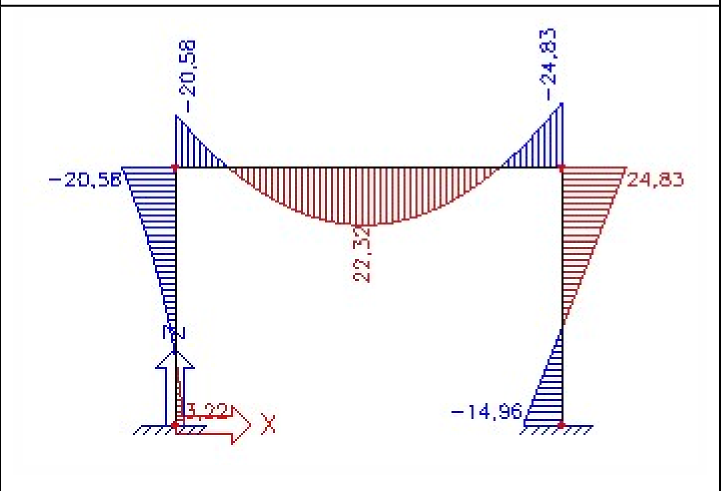
too many intermediate points:

- generates a huge amount of data and may lead to slower response of the programme,
- given more accurate distribution of result quantities.

Let's consider a simple frame subject to load as shown in the figure below.



The effect of number of result sections per member is shown on moment diagrams in the enclosed table. It can be seen that very course division may result in a completely distorted distribution of the result quantity. On the other hand, too fine division gives nicer picture indeed, but does not contribute to the real essence of the result.

2 sections per beam	
5 sections per beam	
50 sections per beam	

The programme remembers some of the previously made settings and may combine them with the current setting. It may happen that the currently adjusted number of result sections per member has no effect on the display of result diagrams. It will be most likely due to the fact that a different, finer division has been used before or for another calculation type. If this happens and if the reduction of the number of result sections is needed, all the results must be cleared from the computer memory and calculation repeated for the required value of the parameter. Use function **Tools > Cleaner** and option **General > All Results** to remove any possible remembered data from the memory.

Static linear calculation

When performing the static linear calculation, the user may specify the general calculation parameter to control the calculation method and process.

Static non-linear calculation

The main difference between a linear and non-linear calculation is that the non-linear calculation gives such results of deflections and internal forces for which equilibrium conditions are satisfied on a deformed structure. The user should think beforehand whether the load applied on the structure would lead to such a state of deformation that affects the resultant internal forces and deflections. If the user thinks so, they should use the non-linear calculation at least for one selected load and compare the results with those for linear calculation. Thus, the user can evaluate the effect on non-linearity on behaviour of the structure.

The non-linear calculation in SCIA Engineer is based on the following assumptions:

- equilibrium conditions are satisfied for deformed shape of a structure,
- effect of axial forces on flexural stiffness of beams is taken into account as well,
- non-linear supports are taken into consideration,
- material of the structure is considered as linearly elastic.

Setup parameters

If any kind of non-linearity should be taken into account in a project, it is necessary to select appropriate option (or options) in the **Project data setup** dialogue.

The options related to non-linearity are listed on tab **Functionality**.

In order to make the options accessible, the main option **Non-linearity** must be chosen. Once this is done, a list of options (or they may be called sub-options) is shown.

Initial deformation and curvature
This option enables the user to define an initial deformation of a structure.

2nd order - gGeometrical non-linearity
This option enables the user to perform geometrically non-linear calculation of analysed structure.

Physical non-linearity for reinforced concrete
This option enables the user to perform iterative calculation of interaction between concrete and reinforcement.

Beam local non-linearity
This option provides for the introduction of local non-linearities on individual beams.

Support non-linearity/soil spring
If this option is ON, it is possible take account of non-linearities in supports.

Friction support/soil spring
If this option is ON, it is possible take account of non-linearities in supports.

Membrane elements
This option enables the user to calculate 2D members with membrane effects.

Press only 2D members
This option enables the user to add nonlinear behavior "press only" on 2D members.

Sequential analysis

Solver parameters

In addition to general parameters controlling the calculation, the non-linear calculation enables the user to define additional options.

Geometrical nonlinearity	This option is available only for functionality "Geometrical nonlinearity" in project. This is a combobox with two possibilities, where user can define geomętical nonlinearity according "2nd order" or "3rd order". The "2nd order" is the exact solution of differential equations according Thimoshenko theory. This settings is suitable for most nonlinear behavior of buildings and it is default settings. Default iterative method of calculation is Picard method. The "3rd order" it is an iterative solution mostly for projects with membranes and cables, where there may be large deformations. Default iterative method of calculation is Newton-Raphson method.
Method of calculation	This item shows which iterative method will be use by solver for this nonlinear calculation. There are four types of method: Newton-Raphson, Modified Newton-Raphson, Picard and Picard and Newton-Raphson. User can edit this item only if geometrical nonlinearity is set as 3rd order.
Number of increments	This parameter is applied for both Newton-Raphson and Timoshenko method only. The values for individual methods are independent and remembered by the programme. Therefore, if you adjust 1 increment for Timoshenko method and four increments for Newton-Raphson method, this parameter will change every time you swap from one method to the other. Usually, one increment gives sufficient results. If deformation is large, the calculation issues a warning and the number of increments can be increased. The greater the value is, the longer it takes to complete the calculation.
Maximum iterations	Specifies the number of iterations for the non-linear calculation. This value is taken into account only for the Newton-Raphson method. For the Timoshenko method, the number of iterations is automatically set to 2. The termination of calculation is controlled by means of convergence accuracy or by means of the given maximal number of iterations. If the limit is reached, the calculation is stopped. If this happens, it is up to the user to evaluate the obtained results and decide whether (i) the maximum number of iteration must be increased or whether (ii) the results may be accepted. For example, if the solution oscillates, the increased number of iterations won't help.
Solver precision ratio	Coefficient that affects the tolerances used during non-linear analysis for convergence checks. Tolerance values are divided by that coefficient. 1 means that the nominal tolerance values are used. A coefficient value higher than 1 means that the tolerances will be smaller, hence the calculation will be more accurate. A coefficient value lower than 1 means that the tolerances will be larger, hence the convergence will be achieved more easily. In some case of heavy non-linearity (e.g. cable or membrane structures), it might be necessary to use less strict convergence criteria (e.g. ratio = 0.1) to allow for proper convergence of the analysis. Note that even with a ratio = 0.1, the convergence criteria remain very tight.
Solver robustness ratio	This parameter influences the convergence of nonlinear calculation of 1D members with nonlinear attributes. A high value of that parameter ensures a more stable but slower convergence of the calculation. It can help in case of sensitive nonlinear analysis where convergence is problematic.
Plastic hinge code	If this option is ON, the non-linear calculation takes account of plastic hinges. It is possible to select the required national standard that will be used to reduce limit moments. If no standard is selected, no reduction is performed.
Geometrical nonlinearity	If this option is ON, the second order effects are considered during the calculation. It is possible to select either Timoshenko or Newton-Raphson method.

	For both methods, the exact solution of 1D members is implemented. It takes account of normal forces and shear deformation for any kind of loading. Transformation of internal forces into the deformed 1D member axis is included. In addition, also available are: Modified Newton-Raphson method and Picard method. The Picard method is regarded as complementary method. It can be used when the Newton-Raphson method fails. The method is more robust but slower. The Picard method can be use alone (the direct iteration method, using the secant stiffness matrix) or in the combination with the Newton-Raphson method. In this case the calculation starts with Newton-Raphson and then switches to Piccard. Main differences: the Newton Raphons method uses tangent stiffnesses, but Picards method uses secant stiffnesses.
Allow compression in membrane members	If this checkbox is ON, the compression in membran elements is taken into account.

Limits of the calculation

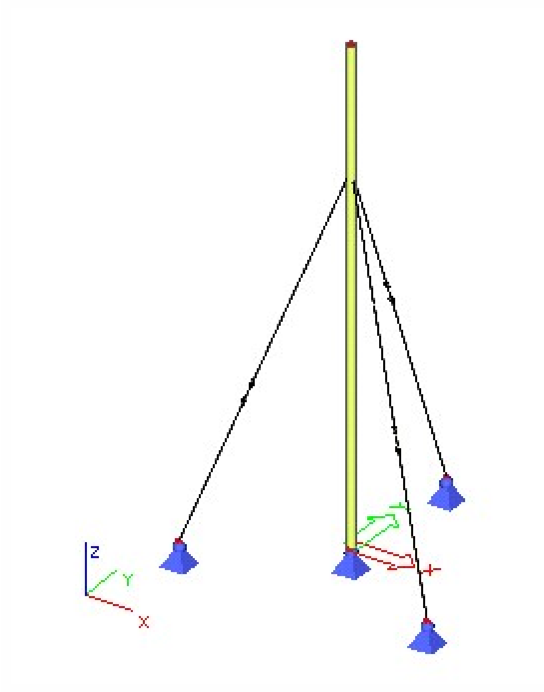
Total number of nodes and finite elements	unlimited
Total number of non-linear combinations	1000
Maximal number of iterations (in one increment)	999
Maximal number of increments	99

Note: Static non-linear calculation can ONLY be performed after the static calculation of the same project has been carried out successfully. In other words, non-linear calculation is a two-step procedure: (i) linear calculation must be completed, (ii) non-linear calculation can be started.

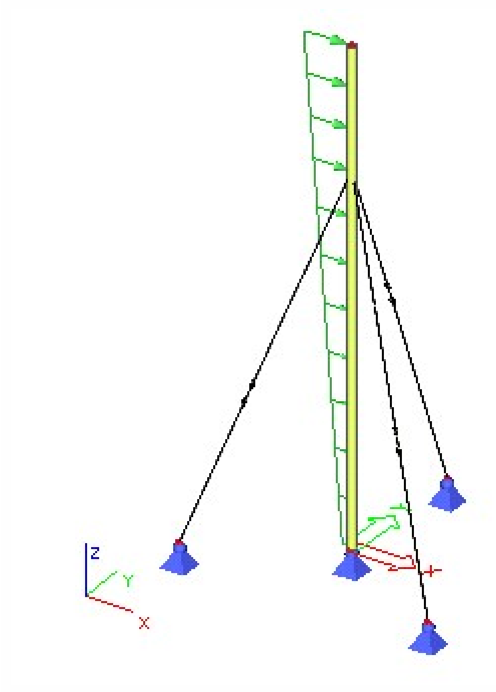
Sample analysis - guyed mast

Structure

A mast with three guy ropes. A steel tubular section has been used for a column shaft. Ropes are modelled as steel bars with a sectional area equal to the sectional area of the rope. The structure is subject to dead load and to the effect of wind.



The load scheme is in the picture below.



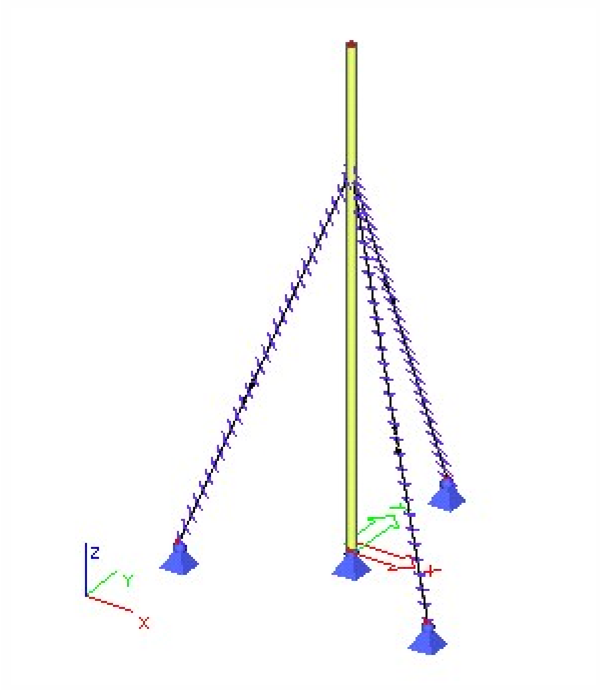
Aim of the analysis

The aim is to examine a deformation shape and internal forces for the given load conditions.

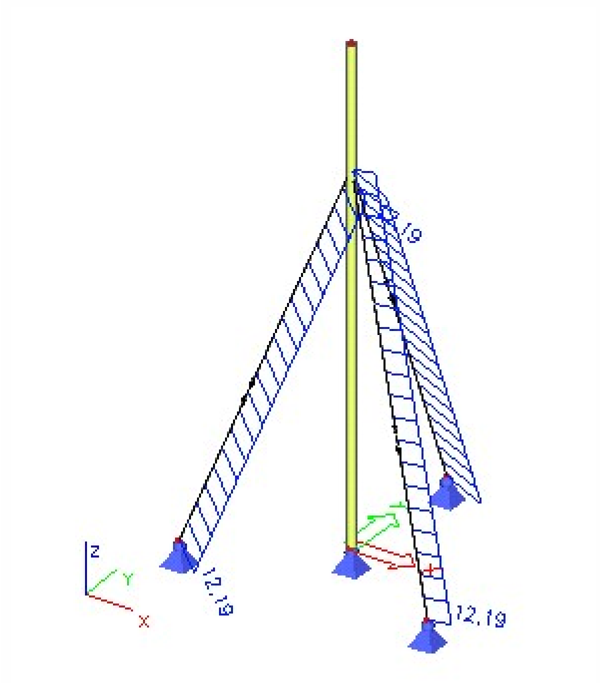
Analysis

It is quite obvious that horizontal stiffness of such a structure depends on stiffness of the ropes. In order to calculate with realistic rope stiffness, a deflection of ropes due to their self-weight must be included into the analysis. A slacked rope has significantly lower stiffness than a straight rope (e.g. a rope lying on a flat pad or a rope hanging vertically). Therefore, a full attention should be paid to proper modelling of the ropes with regard to their 'slackness' (self-weigh deflection).

The ‘slackness’ of ropes, of course, depends on their pre-stressing. The problem of guessing the proper pre-stressing may be an iterative process based mainly on engineer’s experience. The pre-stressing in our example has been introduced by means of temperature load – see the figure below.



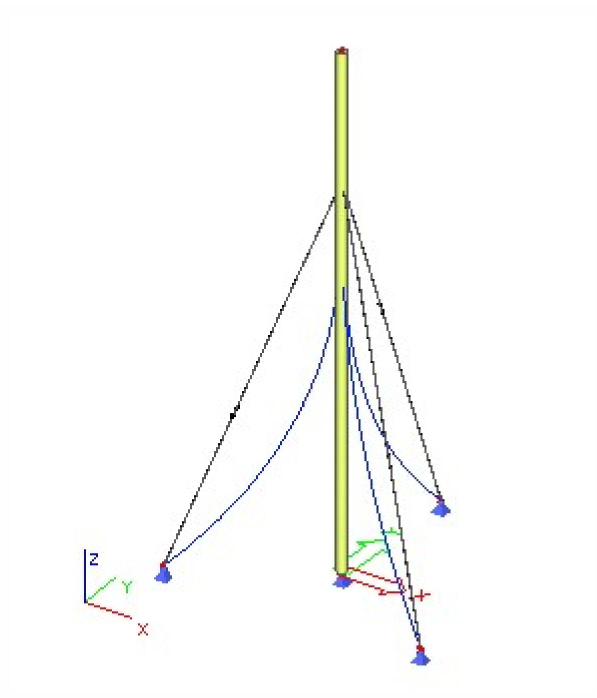
A fine finite element mesh has been used to obtain high quality results with respect to the deflection of ropes. The calculation of axial forces representing the pre-stressing due to thermal load has been carried out as linear calculation. The following picture shows the resultant axial forces.

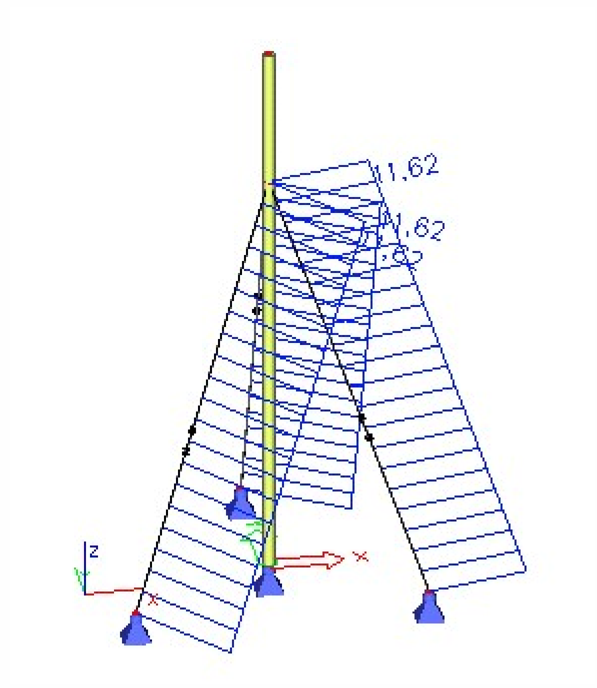


Let’s use these axial forces as the pre-stressing and use them for a successive calculation for the dead load. In other word, let’s start non-linear calculation with the pre-stressing taken into account.

> This calculation procedure corresponds with the following idea about a construction process. First, the structure is assembled in a state of weightlessness and is pre-stressed in this state. Then the structure is subject to the effect of gravity.

The deformation shape and axial force diagram is shown in the following figure. The axial forces are the sum of axial forces produced by pre-stressing and axial forces due to self-weight.

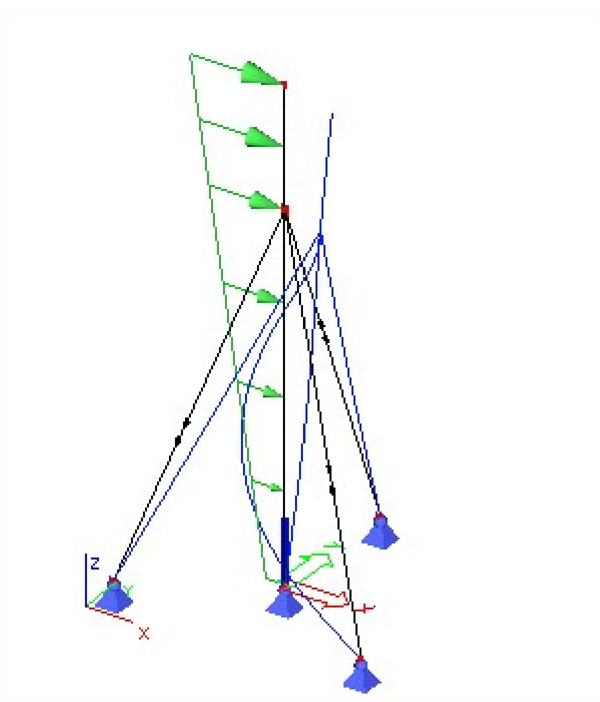




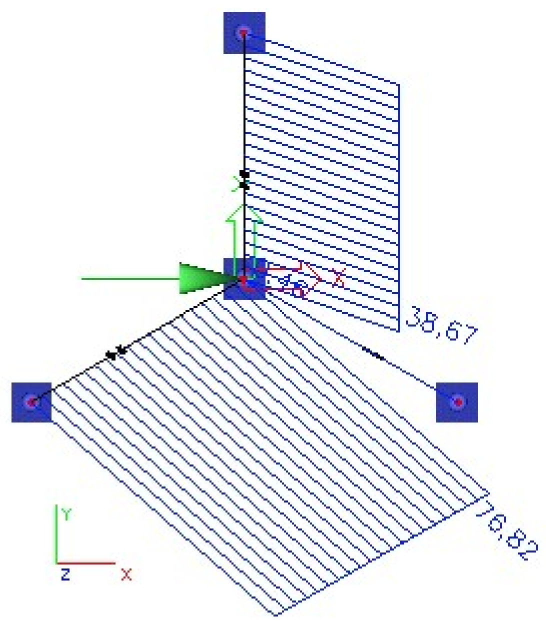
As a next step, the wind calculation has been performed. 'Tightening' of ropes on the windward side results in increased stiffness of the ropes on that side and simultaneous slacking of the ropes on the leeward side leads to dramatically decreased stiffness of the ropes there.

The superposition principle cannot be applied in the non-linear analysis. Therefore, the effect of the self-weight and the wind cannot be analysed separately and then combined in the postprocessor. A combination must be created in advance and the calculation must be carried out for this non-linear combination.

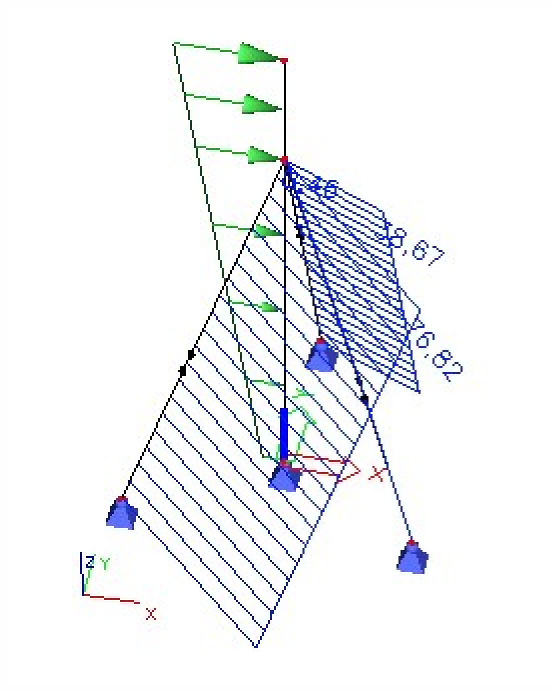
The resultant deformation shape is brought in the picture below.



And the next figure shows distribution of the axial force.



Below there is another view of axial force diagrams.



Initial stress options

A beam of the analysed structure may be subject to an initial stress. This pre-stress may be defined in several ways:

The approach can be adjusted in the Solver options setup dialogue.

Initial stress
If ON, some initial stress will be defined.

Initial stress as input
If ON, the initial stress is specified by a fixed user-input value.

If OFF, see below.

Stress from load case
The initial stress may be calculated automatically from the results of selected load case. The results of linear static calculation for the specified load cases are used to determine the initial stress in the beam.

Initial deformation and curvature

Initial deformation and curvature

If required, an initial deformation of analysed structure can be introduced. There are various approaches in SCIA Engineer to do so.

None
The structure is ideal without any imperfections.

Simple inclination
The imperfection is expressed in the form of a simple inclination. The inclination may be defined in millimetres per a metre of height of the structure. It means that only horizontal inclination in the global X and Y direction may be specified. The inclination is linearly proportion to the height of the building.

This option is applicable mainly for high-rise buildings. It has no or minimal influence on horizontal structures.

Inclination + curvature of beams

If this option is selected, the initial deformation may be defined the same way as above PLUS the curvature of beams may be specified as well. The curvature is the same for all the beams in the structure.

Inclination functions
The initial imperfection is defined by a function (or curve). The user inputs the curve by means of height-to-imperfection diagram.

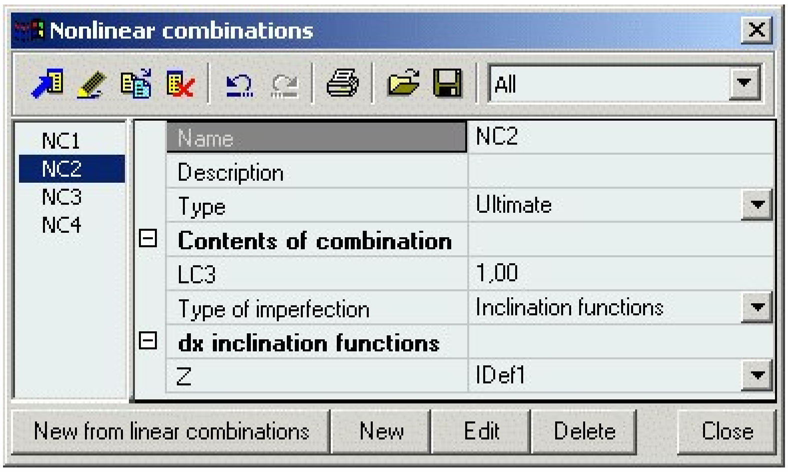
This option is applicable mainly for high-rise buildings. It has no or minimal influence on horizontal structures.

Functions + curvature of beams
The initial imperfection is expressed as a sum of inclination function and curvature. It is analogous to option **Inclination and curvature of beams**.

Deformation from load case
This option requires two-step calculation. First, a calculation for a required load case must be performed. The deformation due to this load case is then used as the initial imperfection for further calculation.

Buckling shape
This option requires two-step calculation. First, a stability calculation must be performed. The calculated buckling shape is then used as the initial imperfection for further calculation.

Regardless of the approach, the initial imperfection can be defined for a non-linear combination only. It is one of the parameters that the user may define in the **Non-linear combination manager**.



The calculated and displayed deformation is ALWAYS measured from the imperfect model, i.e. it does not represent the total deformation from the ideal shape, but the overall deformation from the imperfect shape.

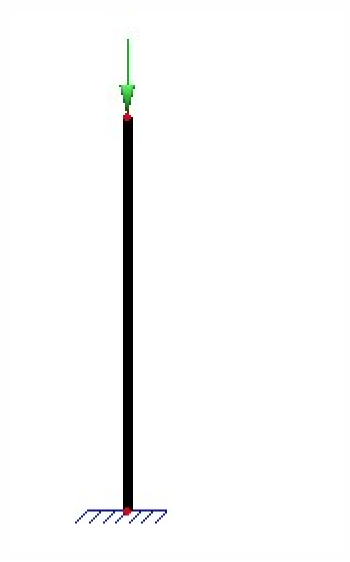
Calculations taking account of any type of imperfection are non-linear calculations and as that they are sensitive to the size of finite elements. Or to be precise, they are sensitive to the number of finite elements per a member. The user MUST remember that a division giving just one finite element per a beam is NOT sufficient and may give completely misleading results.

Simple inclination

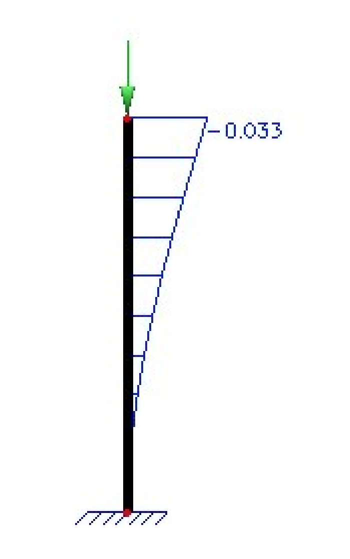
Simple inclination simply defines how much the structure is inclined to one side. The inclination is applicable to vertical structures only. It defines an inclination in horizontal direction per a unit of height. The inclination may be defined in the direction of global axes X and Y.

As an example, let's assume a vertical cantilever subject to vertical concentrated force.

If no inclination is defined, the horizontal displacement is zero.



If, however, a simple inclination is input, the result is affected by this imperfection and the column's horizontal displacement is non-zero.



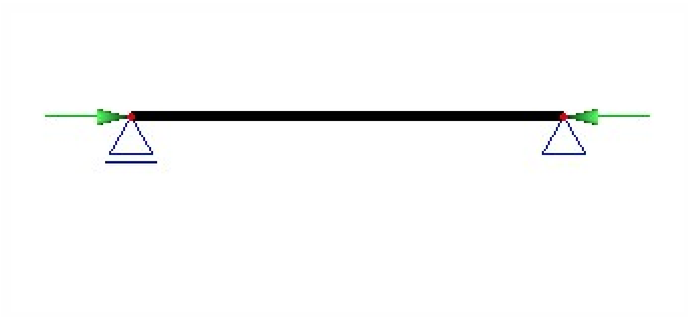
Inclination and curvature of beams

A simple inclination may be combined with an initial curvature of beams. It is also possible to define zero inclination and use only the curvature as a factor determining the initial imperfection.

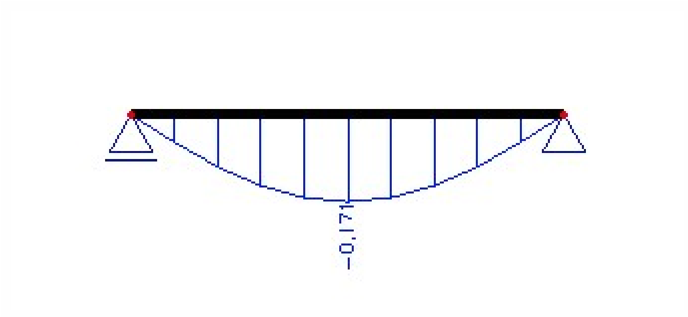
If specified, the inclination is considered the same way as if it is used as the only source of imperfection.

The given curvature is considered for all the beams in the structure. In other words, all the beams are subject to the same initial curvature. The programme automatically determines which direction of curvature is critical and uses that direction for calculation. The curvature is taken into account in all beams in the structure regardless of their spatial orientation. Unlike the simple inclination, it therefore affects even horizontal beams.

Let's assume a simply supported beam whose initial imperfection will be defined solely by means of the curvature. The beam is axially compressed by means of two concentrated forces.



If the curvature is adjusted to a non-zero value, the final vertical displacement of the beam is also non-zero – see the picture.



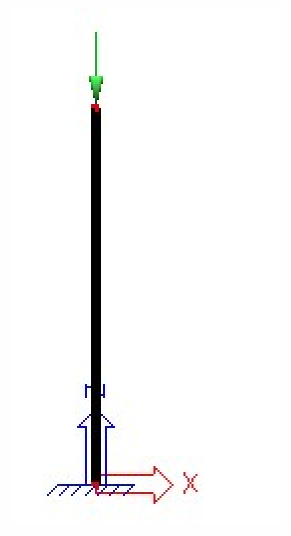
If, on the other hand, the curvature is not applied (i.e. is set to zero), the beam (which is ideally straight) remains straight even when subject to the pair of axially acting forces.

Inclination functions

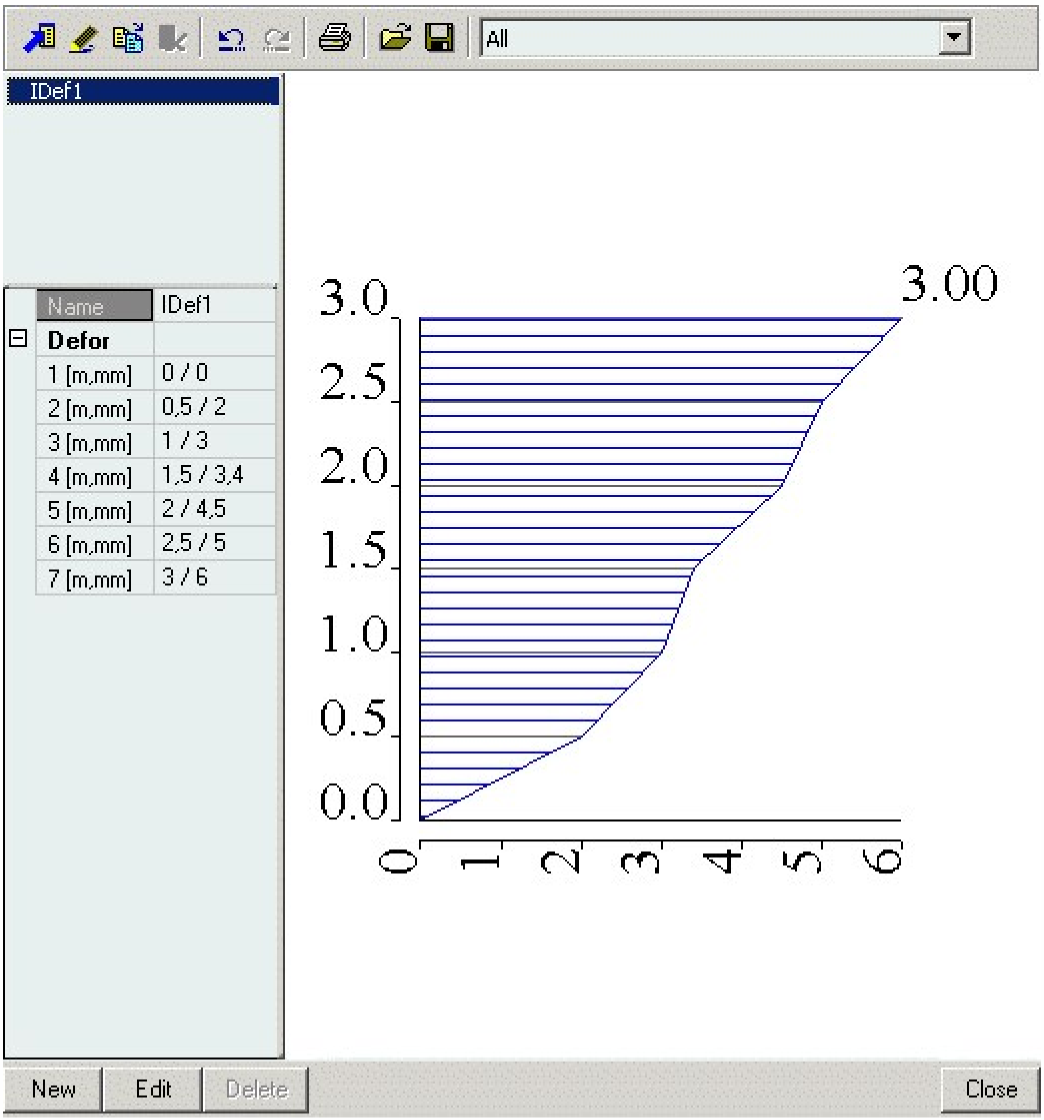
The inclination function may be considered similar to the simple inclination. It is applicable for vertical structures and it defines the horizontal inclination of the structure in the direction of global axes X and Y.

The inclination function is defined by means of an inclination-to-height curve. The curve is than assigned to the appropriate non-linear combination.

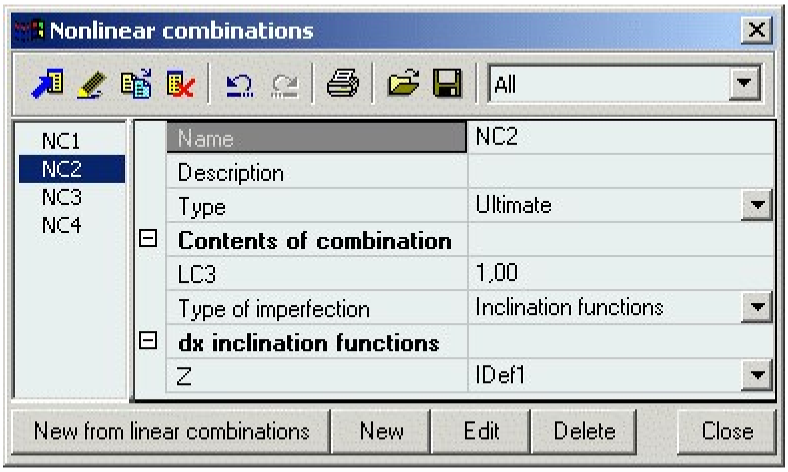
Let's assume a single vertical column fixed at its foot. The column is subject to a vertical force.



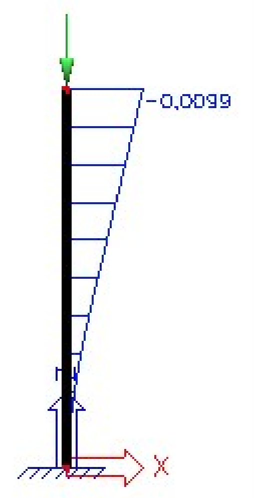
Let's define an inclination curve in the **Initial deformations manager**.



Let's create a non-linear combination, adjust type of imperfection to **Inclination** function and select the previously defined curve.

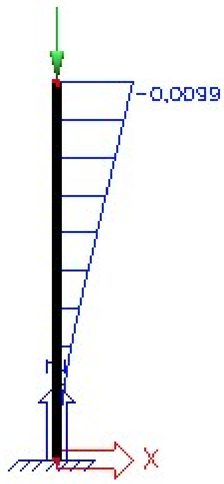
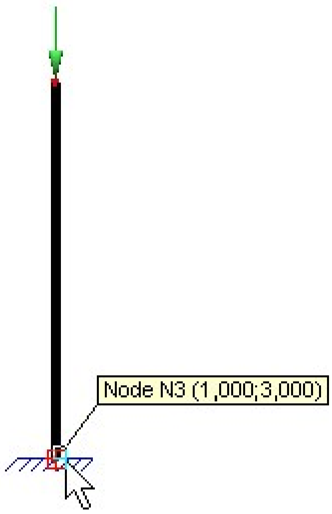


The calculated horizontal displacement of the column will be affected by the imperfection.



It must be emphasised here that the inclination function is defined in absolute co-ordinates, i.e. in co-ordinates of the global co-ordinate system. Therefore, in order to successfully introduce an inclination defined by an initial deformation curve, the user must ensure that the curve is defined at the level corresponding to global Z co-ordinates of the structure – see the table below.

The curve of Initial deformation is defined as above.	The curve is defined for the interval <0 – 3.0> (measured along the global Z axis).
Variant A The column foot is in the origin of the global co-ordinate system, i.e. its vertical co-ordinate is equal to 0.0 (zero).	The result is affected by the inclination curve.

	
Variant B The column foot is in the point whose global vertical co-ordinate is equal to 3.0.	The result is NOT affected by the inclination curve as the curve definition ends just at the foot of the column and therefore the column is not subject to any inclination. 

Inclination function and curvature of beams

The principle here is analogous to a simple inclination combined with a beam curvature. The only difference is that instead of a simple inclination expressed by a single number, the user can define a nonlinear inclination curve.

Deformation from load case

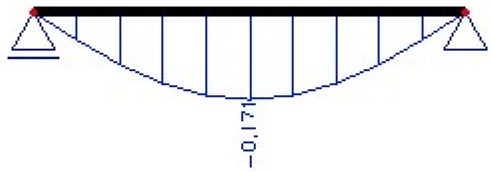
Quite often it may happen that some of the loads are acting on a structure before any other load is imposed. In addition, it is possible that this initial load leads to significant imperfections that may considerably affect the static behaviour of the structure.

A typical example is a rope structure, i.e. a structure whose parts are suspended on ropes or cables. Just the self-weight may cause a significant deflection of these ropes or cables. Also slender steel beams are a good example.

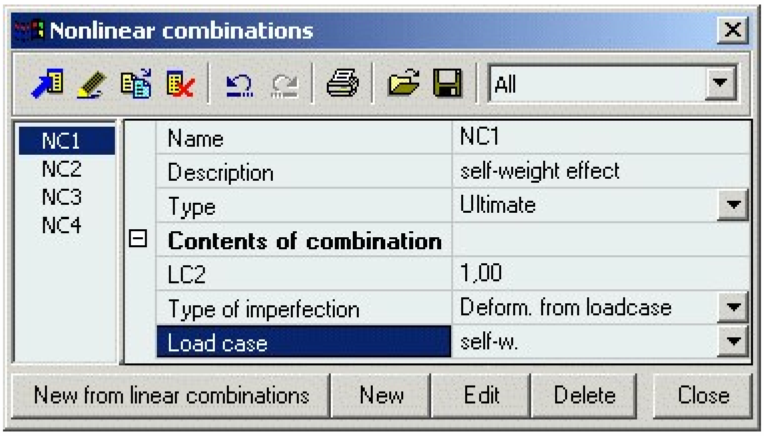
Example

Let's consider a simply supported beam subject to two horizontal forces that compress the beam.

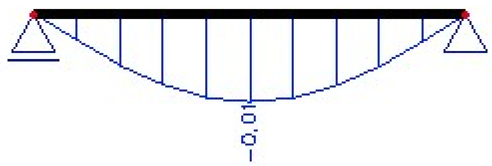
Let's assume that the beam is slender and that just the weight of the material would cause a significant deflection. The first calculation is therefore made for "self-weight load case" only. Its results are shown below.



The obtained result (for the self-weight) can be then selected in the **Non-linear combination manager** as the source of the initial imperfection.



The final non-linear calculation then gives results taking account of the pre-deformation.



If the initial imperfection had not been taken into account at all in our sample structure, the results would have been completely different. The beam, ideally straight and subject to axial load only, would show no deformation perpendicular to the longitudinal axis.

Usually (but not exclusively), a static linear calculation is sufficient enough for the determination of the initial imperfection due to a specific load case. Subsequent calculation whose aim is to take account of the initial imperfection, however, must already be a non-linear one.

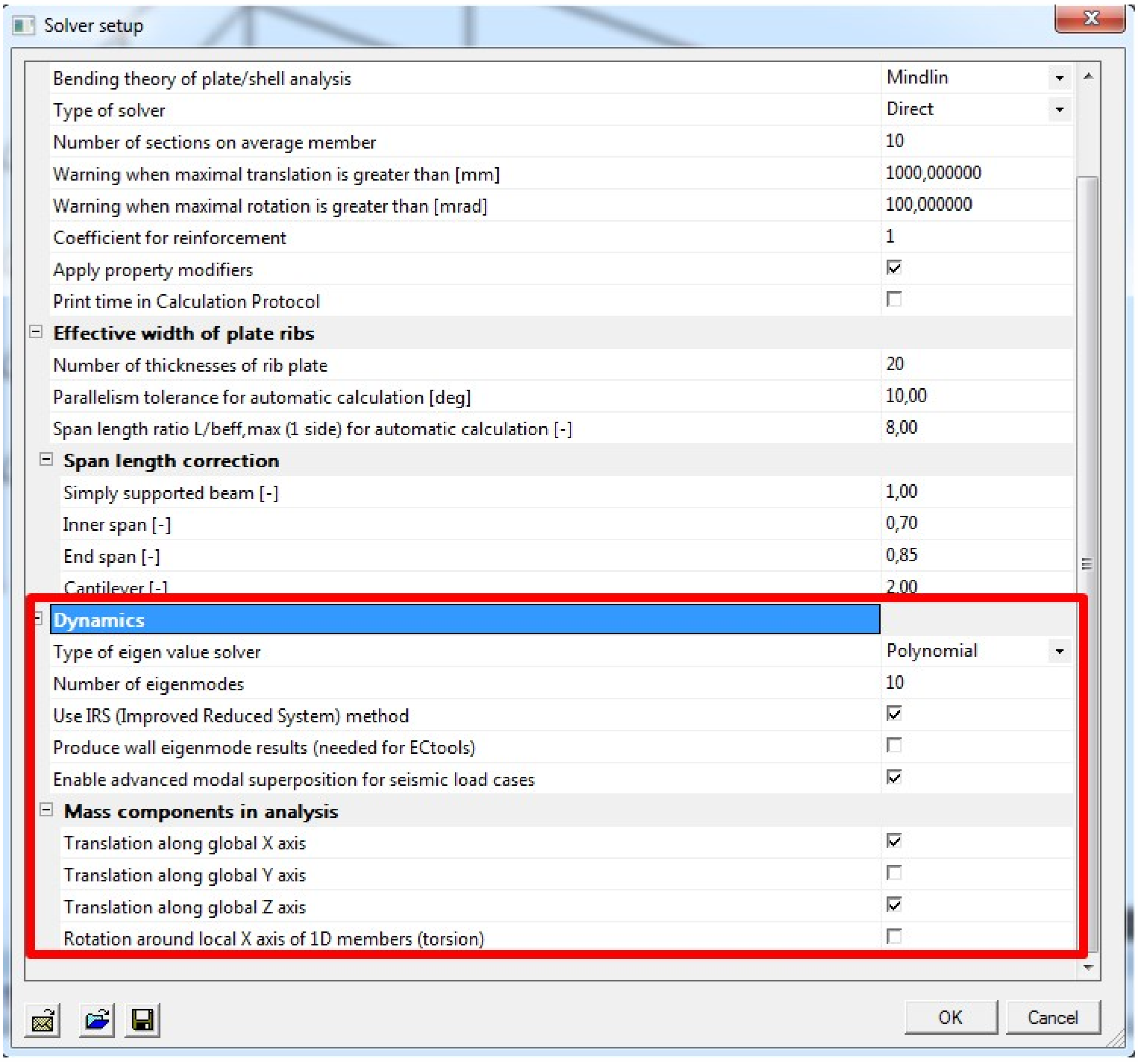
The calculated and displayed deformation is ALWAYS measured from the imperfect model, i.e. it does not represent the total deformation from the ideal shape, but the overall deformation from the imperfect shape.

Dynamic natural vibration calculation

The calculation of free vibration (determination of eigenfrequencies and eigenmodes) is carried out by means of subspace iteration. The result is the required frequencies and modes. All possible modes of vibration are affine. The amplitude can vary according to the amount of supplied energy. E.g. a guitar string may sound high or low but the frequency is the same for the basic tone as well as for higher harmonic tones. In practice, norm eigenmodes are used. The same approach is used worldwide today to adjust the scale of eigenmodes. It employs a weighted sum of squared Ä values as the norm. A mass constant M representing the weight is either the inertial mass in a node or the mass moment of inertia.

The specified masses are recalculated into nodes. By default, each member has only two end-nodes. That means that the mass is distributed into these nodes only. For structures consisting of a huge number of members, this approach leads to satisfactory results. But for structures made of a small number of members, it is necessary to increase the accuracy by means of appropriate refinement of finite element mesh.

In addition to general parameters controlling the calculation, the dynamic calculation enables the user to define additional options in solver setup:



Type of eigen value solver
The method for the calculation of eigenvalues can be selected here. Four methods are available:

The **subspace iteration** method was introduced by Krylov and improved by K.-J. Bathe.

The **Lanczos** method, which is generally faster than the subspace iteration, is described by K.-J. Bathe here.

The method denoted as „**iterative ICGC**“ is an internal development by A. Chirkov from Ukrainian Academy of Sciences and is destined for extremely large, especially 3D solid problems.

The method denoted as „**polynomial**“, is an internal development by Ivan Ševčík from FemConsulting and is based on finding roots of characteristic polynomial. It seems to be the most stable from all the suggested methods which, together with good performance, should indicate this method as the first choice. It is, however, slightly slower than Lanczos, especially on large models, which is why it is not selected by default.

Number of eigenmodes
Here the user specifies how many eigen frequencies should be calculated.

See chapter Natural vibration analysis versus mesh size.

See chapter Calculation model for dynamic analysis.

Use IRS (Improved Reduced System) method
See chapter Reduced Analysis Model

Produce wall eigenmode results (needed for ECTools)
Enable advanced modal superposition for seismic load cases

Method for time history analysis

The method for the time history analysis can be selected here. Two methods are available:

The **direct time integration** method is standard Newmark algorithm. Theoretical background about direct time integration here.

The **modal superposition** method, it is necessary in some cases to filter out local, high frequency modes and avoid irrelevant behaviour, especially for fast train dynamic analysis. Modal superposition is a powerful idea of obtaining solutions. It is applicable to both free vibration and forced vibration problems. The basic idea To use free vibrations mode shapes to uncouple equations of motion. The uncoupled equations are in terms of new variables called the modal coordinates. Solution for the modal coordinates can be obtained by solving each equation independently. A superposition of modal coordinates then gives solution of the original equations. Notices It is not necessary to use all mode shapes for most practical problems. Good approximate solutions can be obtained via superposition with only first few mode shapes.

Mass components in analysis

This settings in solver setup is for optional removal of mass components in dynamic analysis. There are four items with checkboxes, where user can choose, which parts of mass should be taken to dynamic calculation: (as default all masses are selected)

Translation along global X axis

Translation along global Y axis

Translation along global Z axis

Rotation around local X axis of 1D members (torsion)

Calculation for selected mass combinations

If general optionSingle nonlinear analysis is ON, the user may specify which mass combinations will be calculated. Otherwise, all non-calculated are always calculated.

Note: The dynamic calculation can be carried out for mass combinations only.

Dynamic forced harmonic vibration

The principles of how SCIA Engineer deals with a structure subject to a harmonic load are given in chapters:

- Loads > Load types > Dynamic loads > Harmonic load
- Loads > Load cases > Dynamic load cases > Defining the harmonic load case
- Results > Evaluating the results for harmonic load

And the core of dynamic calculations is laid down in:

- Loads > Load cases > Dynamic load cases > Dynamic load cases
- Loads > Load cases > Dynamic load cases > Defining a new dynamic load case

Harmonic band analysis

Context - Typical usage

In a nutshell, harmonic band analysis, aka Third Octave Analysis, is about assessing the sensitivity of a structure to an external source of vibration.

For example: a microsurgery room is located in a building close to a metro line. The metro line is the external vibration source, and the harmonic band analysis will:

- generate a series of harmonic load cases at a support (or close to it, not sure anymore, I think it was a spring support)

- filter them to get RMS values per band

- display the resulting spectrum in a particular point of the microsurgery room

- this spectrum is transfer function: for a given external vibration, typically at the foundation level, one can see the local impact in any point of the structure across a range of frequencies.

A transfer function is usually computed for a unit input at the foundation. It basically tells, for a unit action applied in a given point of the structure at a given frequency, how the response of the structure will be amplified or damped in another point of the structure. The amplification of the response usually depends on the frequency.

Harmonic Band Analysis is **not** meant for generating harmonic load cases that could be used in combination with other actions. It is solely designed to generate the transfer function, as described above. It does, however, carry out the calculation through a set of harmonic load case calculations on a range of frequencies.

Description

This calculation represents a new way of dealing with the calculations in harmonic analysis. Multiple analyses on a range of frequencies are carried out. The harmonic analysis is possible for a range of frequencies controlled by the user. In the standard harmonic analysis, the forces and the frequency are defined. In this type (Harmonic Band Analysis) of analysis, the frequency of the harmonic force varied over a range and the harmonic analysis is performed for multiple values in that range.

To fit the needs of this type of calculation, a new load case type named "Harmonic Band Analysis" has been introduced into SCIA Engineer. the properties of this load case are similar to the standard harmonic load case. But, instead of the frequency, there are 5 new parameters: A, n1, n2, C, N (explained below). The input of loads is the same as for the standard harmonic load cases.

SCIA Engineer generates a set of extra load cases:

1. one set of main F frequencies (their number is n=n2-n1+1) and
2. n sets of secondary frequencies (each of them with 2N items).

The secondary load cases are the standard SCIA Engineer harmonic load cases and have standard results.

The results of the main load cases are calculated by RMS (root mean square) method from the appropriate set of the secondary load cases.

SCIA Engineer generates the following result classes:

1. one with all main load cases and
2. n with the sets of the secondary load cases.

Output of results

Alphanumerical output

All the results of the main and secondary load cases are presented in the standard SCIA Engineer way in result tables using the generated results classes.

Graphical output

The results of the main frequencies or results of the bands around the main frequency can be presented also graphically in the form of a diagram. For that purpose a new tool has been integrated into SCIA Engineer.

Refresh after modifications of the structure and changes in other input values

When the user changes parameters n1, n2 or N, all the generated load cases and all the generated result classes are deleted and all the document items with band analysis load cases are not valid any more. If any other project data are changed, all generated items remain in the project and their content is updated after next calculation.

(Little) Theoretical background

The user defines constants A,n1,n2,C,N.

The default values are: A = 2, n1 = 6, n2 = 30, C = 3, N = 10.

From these data, a geometric series are generated using the following formula

$$F_i = A^{n/C}$$

where n varies from n1 to n2 with a step of 1.

The result is a series of so-called main frequencies F. The default set is: 4,00; 5,04; 6,35; 8,00; 10,08; etc. Around each of these values, an interval Fi- - Fi+ is defined:

$$F_{i-} = A^{(n-1/2)/C}$$
$$F_{i+} = A^{(n+1/2)/C}$$

The interval [Fi- - F] is now divided into N steps to generate the secondary frequencies "f".

For each value of "f" a harmonic analysis is carried out. The displacement or inner force in a specified node in a given direction is calculated, giving N result values. The same is done for the interval [F – Fi+]. From these 2N values, one value is calculated using RMS (root mean square) and assigned to the main frequency F.

Combination with other load cases

The results of this analysis can not be combined with other static and dynamic analyses.

Input of the load case for the Harmonic Band Analysis

The input of the load case for the Harmonic Band Analysis requires similar prerequisites as other dynamic load cases.

Procedure do define a new load case for the Harmonic Band Analysis

1. In the Project setup dialogue, on tab Functionality, select Dynamics and Harmonic band analysis.
2. In the Dynamics branch of the tree menu define at least one Mass group and at least one Combination of mass groups.
3. Then you may open the Load case manager and input a new load case for the Harmonic Band Analysis.
4. Select the following options and define the appropriate parameters:
    1. Action type = variable
    2. Load group = as required in the particular project
    3. Load type = dynamic
    4. Specification = Harmonic band analysis
    5. Parameters = as required in the particular project

   6. Master load case = none or as required in the particular project

   7. Mass combi = as required in the particular project

5. When ready, close the Load case manager.

> Note: Before the calculation is performed, the load case manager shows just this (these) input load case (cases). All the automatically generated load cases, generated according to the description provided above, are added to the Load case manager only after the calculation has been carried out.

Example

The list of load cases after performed Harmonic Band Analysis may look like

BA1-4 - 1.20
BA1-3 - 1.22
BA1-2 - 1.23
BA1-1 - 1.25
BA1-F1 - 1.26
BA1+1 - 1.28
BA1+2 - 1.29
BA1+3 - 1.31
BA1+4 - 1.32

This picture shows an extract of the list of load cases. It contains one main frequency (BA1-F1) and eight secondary frequencies (BA1-4, BA1-3, BA1-2, BA1-1, BA1+1, BA1+2, BA1+3, BA1+4).

Performing the Harmonic Band Analysis

In order to start the Harmonic Band Analysis, the linear static calculation must be run.

> Note: Similarly to other dynamic calculations, attention must be paid the size of the finite elements. This is true also in simple structures with a few 1D members only. The analysis may require a certain number of finite elements in order to calculate the total number of required bands.

Display of results of Harmonic Band Analysis

There is a special display mode for the results of the Harmonic Band Analysis. This mode is available in the following functions of service Results:

- Beams > Internal forces,
- 2D members > deformation of nodes,
- 2D members > Internal forces.

In this mode a new item (parameter) appears in the property window. This item is called Text output and can be set to two options: (i) Texts or (ii) Graph.

The Text option displays the results in a standard way, i.e. using the diagram in the graphical window and alphanumerical table in the Preview window.

The Graph option draws a special diagram in the Preview window. For this option one more item is added to the property window: Selection tool. This tool – accessible through the three-dot button – allows you to select the 1D members or slabs and nodes for which the diagram is to be displayed.

The later will be demonstrated on a few examples.

Example 1 - Setup for graphical result at main frequencies at a selected mesh node:

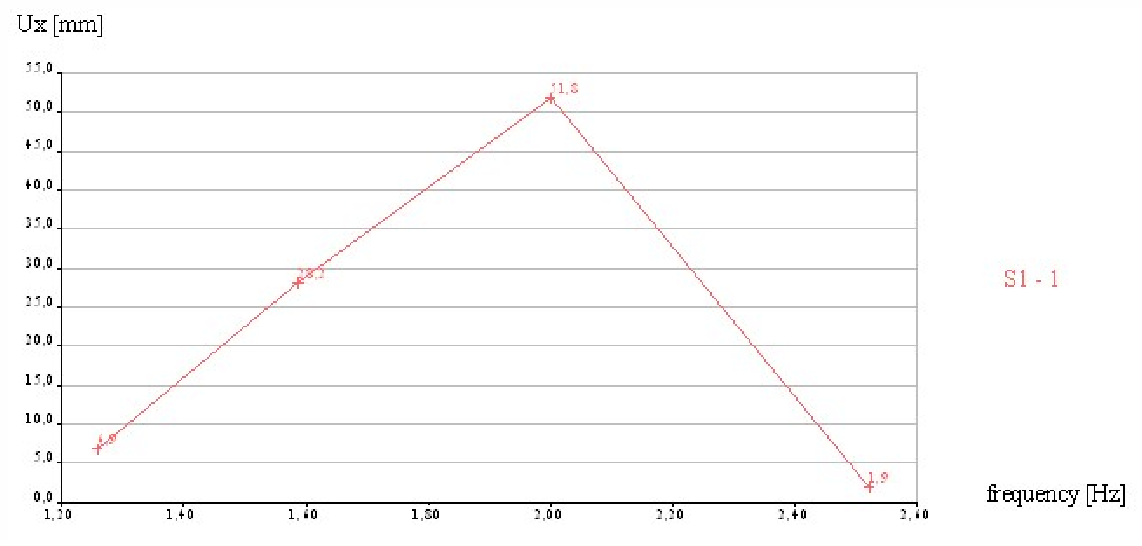
Function: Deformation of nodes

Type of load: Class

Class: Main

Text output: Graph

Selection tool: S1, node no. 1.



Example 2 - Setup for graphical result at a selected band for a selected mesh node:

Function: Deformation of nodes

Type of load: Class

Class: Sec3

Text output: Graph

Selection tool: S1, node no. 1.

Note that for a band, beside the deformation curve also the RMS is drawn.

Example 3 - Setup for envelope graphical result at main band frequencies, all nodes selected:

Function: Deformation of nodes

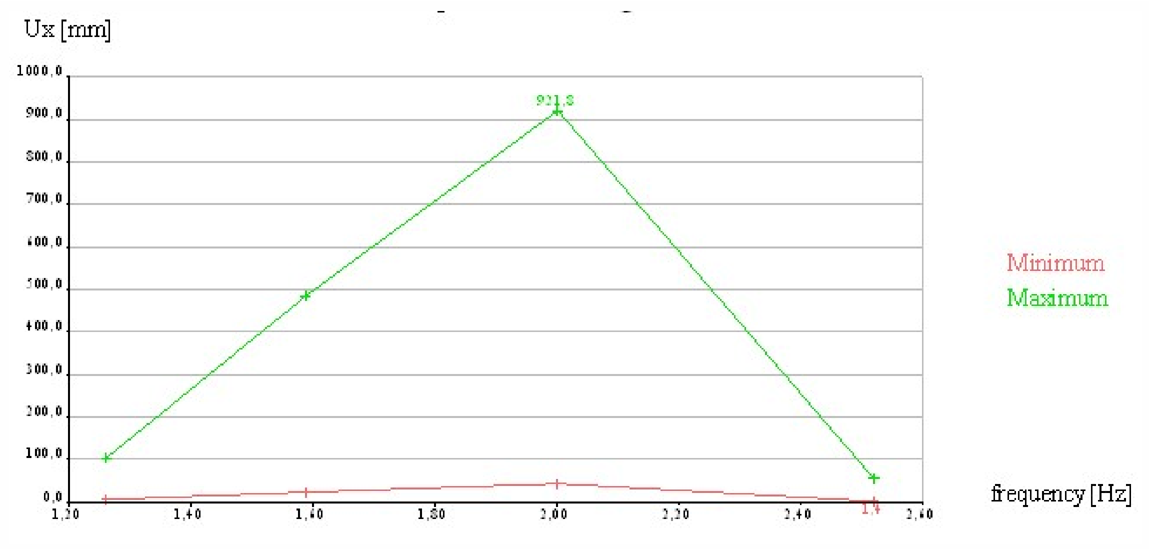
Type of load: Class

Class: Main

Text output: Graph

Selection tool: all members, (by default all nodes are selected)

Extreme: Global



Example 4 - Setup for graphical result at the main band frequencies for all nodes displayed in the same diagram:

Function: Deformation of nodes

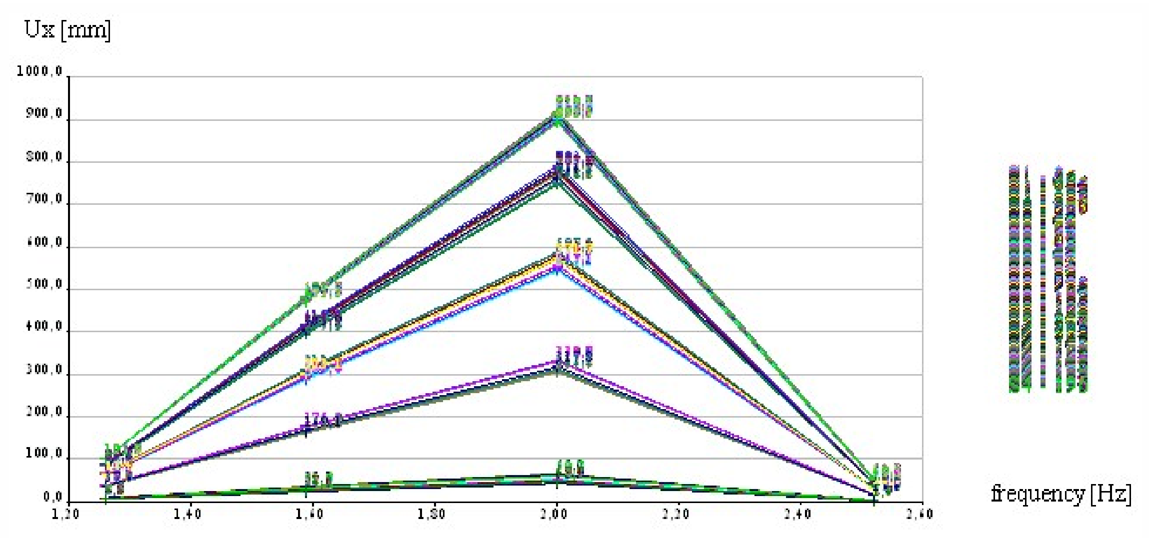
Type of load: Class

Class: Main

Text output: Graph

Selection tool: all members, (by default all nodes are selected)

Extreme: no



Calculation model for dynamic analysis

When a static analysis is being performed, there are usually no problems with the creation of a satisfactory model of the analysed structure. BUT, in dynamic analysis we have to think about the problem a bit more. Here's why.

Statics deals with the equilibrium of the structure

The imposed load and internal forces arising due to the elastic deformation of the structure must be balanced. In dynamics, the equilibrium is also required, but now with additional forces – inertial and damping.

Inertial forces

At school, they taught us about Newton Law – force is equal to mass multiplied by acceleration. This means that if the masses of a structure move with acceleration, inertial forces act on the structure. In order to analyse dynamic behaviour of the structure, we have to complete the calculation model and add data related to masses in the structure.

Damping forces

Nature has provided us with an invaluable principle. As soon as anything starts to move, it is stopped in a while. Against the motion the resistance of environment is acting – external and internal friction. The mechanical energy is transformed into another form, usually into the thermal one. In civil engineering practice, structures of higher damping capacity are more suitable – the higher damping capacity, the lower vibration. Steel structures have lower damping capacity than concrete or wooden ones. It is generally difficult to take damping forces into account in calculation. The user's point of view is that this task is rather simply s/he just defines a damping coefficient, e.g. logarithmic decrement. But nothing is as simple as it may seem to be. We've got just one coefficient, but its precision is rather arguable.

To conclude, dynamic calculations differ from static ones in one principle point. We do not seek only the equilibrium between imposed load and internal forces, but we also introduce inertial and damping forces. The outcome is that we have to complete the calculation model created for a static analysis with other data:

- masses in the structure (used for determination of inertial forces)
- damping data (for calculation of damping forces)

Method of decomposition into eigemodes

Dynamic calculations in SCIA Engineer are based on method of decomposition into eigemodes (called modal analysis). The basic task is therefore the solution of free vibration problem. The calculation finds eigenfrequencies and eigenmodes.

There exist bizarre conceptions among structural engineers of what the eigenfrequency and eigenmode actually is. It is important to realise that free vibration gives us only the conception of structure properties and allows us to predict the behaviour under time varying load conditions. In nature, each body prefers to remain in a standstill. If forced to move, it prefers the way requiring minimal energy consumption. These ways of motion are called eigenmodes. The eigenmodes do not represent the actual deformation of the structure. They only show deformation that is "natural" for the structure. This is why the magnitudes of calculated displacement and internal forces are dimensionless numbers. The numbers provided are orthonormed, i.e. they have a particular relation to the masses in the structure. The absolute value of the individual numbers is not important. What matter is their mutual proportion.

Let's assume that there is an engine mounted on a structure. The engine is equipped with a eccentrically connected parts revolving with frequency of 8 Hz. We determine that eigenfrequency of the structure is 7.7 Hz. This information means that it is natural for the structure to vibrate close to 8 Hz and that the applied load is too dangerous. The eigenmode corresponding to frequency 7.7 Hz can inform us only about the mode of the vibration. In other words, it can show us where the displacement due to vibration will be largest and where minimal. It suggests nothing about the real size of displacement or internal forces. These pieces of information can only be obtained from the calculation of the structure subjected to a particular load.

In order to consider the effect of inertial forces, we have to add masses to the calculation model. We can choose from concentrated masses in nodes or concentrated point and linear masses on beams.

In SCIA Engineer a concept of consistent mass matrix has been applied. This concept means that not only diagonal elements of mass matrix are considered in the calculation. Consequently, this solution is more precise than for matrix called lumped mass matrix. This advantage is significant e.g. when rigid arms are used in the model.

Self-weight of the structure due to cross-sections and material is not input. It is taken into account automatically. We only input masses attached to the structure, masses that will move with the structure when it moves. So be careful with suspended loads hanging on longer suspensions. Problems may arise with imposed load on floors, charges of tanks and other masses that may or may not be present. A safe side does not exist, which was already stated earlier. A very useful option in SCIA Engineer is the generation of masses from defined static load.

Masses on the structure may be sorted into several groups that can be combined in mass combinations in a way similar to standard load cases. For simple structures where all the masses are in a single group, this approach may be a small complication. On the other hand, it will be considerably advantageous for more complex structures. Masses in individual groups may be defined in nodes or on beams. The latter may be either concentrated or distributed.

The next step in the specification of mass distribution across the structure is the definition of mass groups. The combination then defines the masses in the dynamic calculation model. The combinations are input the same way as static load case combinations. We select appropriate mass groups and specify their coefficient. Why is this needed?

Let's consider an example of a structure supporting a large liquid tank. The amount of liquid in the tank will range between two limits: from an empty tank to a full tank. The dynamic behaviour of the structure will be different for the two limit states. If we need to perform a seismic analysis, we do not know when the earthquake happens. It is therefore suitable to evaluate both limit states, but also an intermediate one with the tank e.g. half full. It would mean three separate calculations. In SCIA Engineer however, we just need to define three mass combinations and run one calculation. We include all the masses corresponding to a full tank into one mass group. Then we define three combinations. The first one will contain the mentioned group with coefficient equal to 1.0. The second one will contain the mentioned group with coefficient equal to 0.5. And the third combination will not contain the group at all.

Other examples:

- Bridge – dynamic analysis of an "empty" bridge and bridge loaded with a vehicle or train (even though the mass of larger bridges is significantly bigger that the mass of vehicles),
- Building frame – response to vibration for various distribution of floor variable load.

The calculation of eigenmodes is usually carried out together with a static calculation. Each of the eigenmodes refers to a particular calculation model that is defined by means of a combination of mass groups and a corresponding model for static analysis.

The calculation of eigenmodes and eigenfrequencies is made on the finite element model of the structure. The structure is discretised, which means that instead of a general structure with an infinite number of degrees of freedom, a calculation model with a finite number of degrees of freedom is analysed. The number of degrees of freedom can be normally determined by a simple multiplication: number of nodes is multiplied by the number of possible displacements in the node (translation, rotation). It is important to know that the accuracy of the model is in proportion to the "precision of discretisation", i.e. to the number of elements of the finite element mesh. This refinement has almost no practical reason in static calculations. However, for dynamic and non-linear analyses, it significantly affects the accuracy of results (but also the calculation's time consumption).

The user must decide what number of eigenfrequencies should be calculated. The frequencies are always calculated from the lowest one. Generally, the number of frequencies that can be calculated for a discretised model is equal to the number of degrees of freedom. Due to the applied method, this is not always true in SCIA Engineer. Another problem is that the higher the frequency is, the less accuracy is achieved. Therefore, it is sensible to calculate only a few lowest frequencies. Usually, it is good enough to select the number frequencies as 1/20 to 1/10 of the number of degrees of freedom. On the other hand, it is not practical to select too high number of frequencies because it prolongs the calculation non-proportionally. Once again, no precise and strict limit can be established, but it can be said that 20 frequencies are usually sufficient even for excessive projects.

See also chapter Natural vibration analysis versus mesh size.

In eigenfrequency problem, the following equation system is solved:

$$\mathbf{M} \cdot \mathbf{r}_{\cdot\cdot} + \mathbf{K} \cdot \mathbf{r} = 0$$

where:

r is the vector of translations and rotations in nodes,

$\mathbf{r}_{\cdot\cdot}$ is the vector of corresponding accelerations,

K is the stiffness matrix assembled already for static calculation,

M is the mass matrix assembled during the dynamic calculation.

The solution itself is carried out by means of subspace iteration method.

Dynamic seismic calculation

The principles of how SCIA Engineer deals with a structure subject to a seismic load are given in chapters:

- Loads > Load types > Dynamic loads > Seismic load
- Loads > Load cases > Dynamic load cases > Defining the seismic load case

And the core of dynamic calculations is laid down in:

- Loads > Load cases > Dynamic load cases > Dynamic load cases
- Loads > Load cases > Dynamic load cases > Defining a new dynamic load case

Information about automatically generated combinations according code are here:

- Seismic combination according EN code
- Seismic combination according IBC code

Information about several seismic spectrums are here:

- Defining a seismic response spectrum

Information about various aspects of the seismic analysis of buildings is available here:

- Seismic Analysis of Buildings

Information about theoretical background, modelization & troubleshooting of dynamic analysis

- Dynamic analysis troubleshooting

Buckling analysis

Adjustment of general parameters may control the calculation.

Calculation for selected stability combinations

If general optionAdvanced solver option is ON, the user may specify which stability combinations will be calculated. Otherwise, all non-calculated are always calculated.

Assumptions of linear buckling calculation

The word 'linear' suggests that the following phenomena cannot be taken into account:

- non-linear elastic supports and foundation,
- beams acting in compression or in tension only.

Since the calculation is linear, both types of non-linearity listed above are considered in a way that corresponds to equilibrium equations assembled on a non-deformed structure. In other words, foundation stiffness is equal to stiffness for zero deflection and 'one-way-action' beams are taken as usual beams.

Let's start with the equilibrium equation

$$(\mathbf{K_E} + \mathbf{K_G})\ \mathbf{u} = \mathbf{R}$$

where $\mathbf{K_G}$ is a geometric stiffness matrix reflecting the effect of axial forces in beams and slabs.

The basic assumption is that the elements of the matrix $\mathbf{K_G}$ are linear functions of axial forces in beams. That means, matrix $\mathbf{K_G}$ corresponding to a K-th multiple of axial forces in the structure is the K-th multiple of original matrix $\mathbf{K_G}$. The aim of the buckling calculation is to work out such a multiple K for which the structure loses stability. Such a state happens when the equation below has a non-zero solution.

$$(\mathbf{K_E} + K\ .\ \mathbf{K_G})\ \mathbf{u} = \mathbf{0}$$

In other words, such a value K should be found for which the determinant of the total stiffness matrix (the term in the brackets) is equal to zero. For information about the methods for solving eigenvalue problems in SCIA Engineer, see the chapter about natural vibration analysis. Similarly to the dynamic calculation, the result is a series of eigenmodes and corresponding K-multiples. The first eigenmode is usually the most important and corresponds to the lowest K-multiple. A possible collapse of the structure usually happens for this first mode.

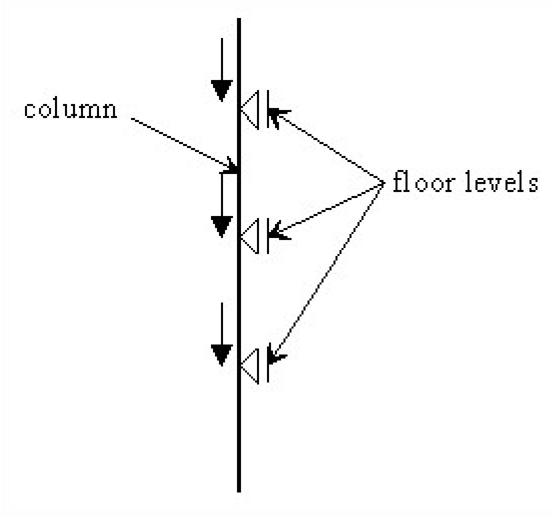
> Note: There is a difference in behaviour of beam and shell elements. For shell elements the axial force is not considered in one direction only. The shell element can be in compression in one direction and simultaneously in tension in the perpendicular direction. Consequently, the element tends to buckle in one direction but is being 'stiffened' in the other direction. This is the reason for significant post-critical bearing capacity of such structures.

The calculated buckling eigenmodes can help the structural engineer to get an idea about behaviour of a structure and about possible mechanisms of buckling failure. The resultant critical multiple suggests how far the structure is from possible buckling.

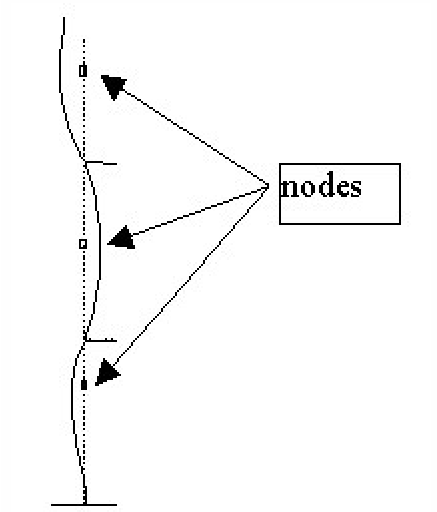
> Note: The buckling calculation can be carried out for stability combinations only.

Sample analysis - column

Similarly to dynamics, calculation of buckling depends on the finite element mesh. The effect of mesh density will be demonstrated on an analysis of a frame column that is supported in a horizontal direction by wind stiffeners at the floor levels (see figure).

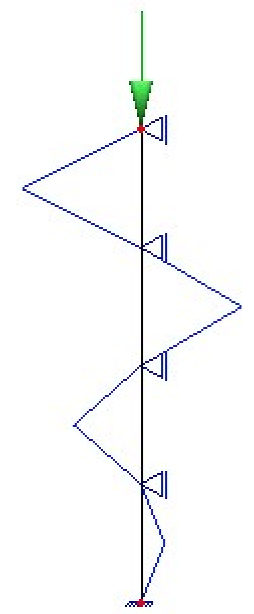
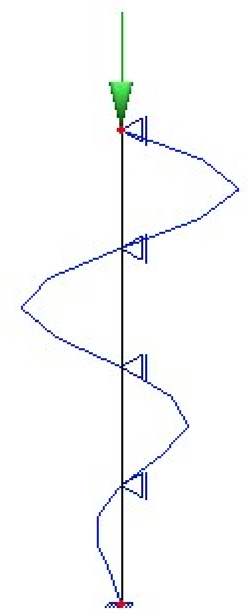


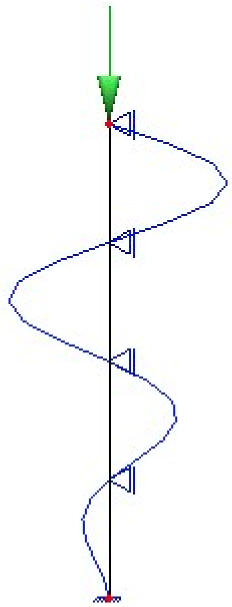
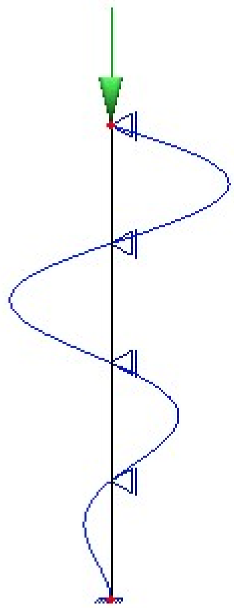
The supposed shape of buckling is in the following figure.



If the usual mesh size, i.e. one finite element per one beam, is used, the model has no degrees of freedom for horizontal translations in the middle of heights of individual floors, which prevents the buckling. Because, however, rotations in floor levels are not constrained even such a poor model would eventually buckle. But, the result would show a significant numerical error. The coarse division (one element per a floor) gives the critical multiple that is equal to 213, while a variant with nodes in the middle of the heights gives the multiple of 163. It is definitely clear that the difference is not negligible.

The following pictures demonstrate the above said facts. They also show that once an FE node is in the middle of the floor, further refinement of the mesh gives only small improvement in results.

4 finite elements per column, i.e. 1 FE per floor critical multiple = 213	
8 finite elements per column, i.e. 2 FEs per floor critical multiple = 163	

10 finite elements per column critical multiple = 161	
50 finite elements per column critical multiple = 160	

Nonlinear stability calculation

Non-linear stability is in its first phase calculated as normal nonlinear calculation using N-R method. The load is incremented in steps, but the incrementation does not stop at the load intensity defined by the non-linear combination and continues until the singularity is reached. Then the solution goes back to the last regular state and the critical load intensity is found from this state by means of eigenvalues, i.e. refined solution is sought in the interval between the last regular and singular state.The accuracy of the solution is therefore determined by the number of load increments. The load intensity defined in the non-linear combination has two meanings. First, when divided by the number of the increments, it specifies the size of load increment and, second, the calculated coefficient of the critical load is related just to it.

Non uniform damping in dynamic calculation

Non uniform damping

This type of calculation is a dynamic calculation that takes into account non-uniform damping on members and supports.

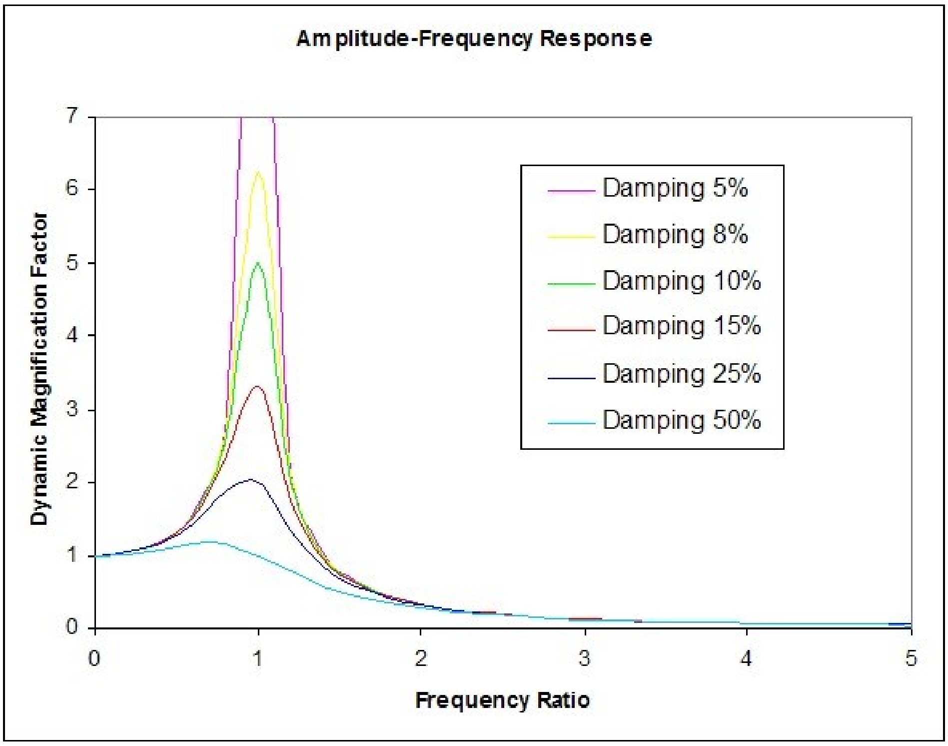
There is a possibility to input a damping value on each 1D and 2D member. It can be (i) relative damping, (ii) logarithmic decrement or (iii) Rayleigh damping. Moreover, a damper can be input in direction X, Y, Z of a nodal flexible support.

If a dynamic calculation (seismic + harmonic) is carried out and the load case has "Damping group" defined, then SCIA Engineer takes into account the non-uniform damping of the members and supports. The modal relative damping for each direction (i.e. the damping percentage for each mode and each direction) is calculated automatically for each load case.

All 1D and all 2D members must have the damping value assigned before the calculation starts or the default value is used. The input of damping in supports is possible only in the GCS directions.

Background information

The effect of damping is significant in the vicinity of resonance. The phenomenon resonance appears when the frequency of the source of vibration (=driving frequency) corresponds to the eigenfrequency of the system. In this case, large deformations are expected which can cause damage of the structure. Damping of the system is a solution to prevent this.



The most famous example of resonance was the collapse of the Tacoma Narrows Bridge in Washington state in 1940. Because of high windspeeds, this bridge began to vibrate torsionally firstly. Later, the vibrations entered a natural resonance frequency of the bridge which started to increase their amplitude.

Also the Erasmusbrug in Rotterdam became a danger due to resonance causing by the vibration of the cables. To prevent this in the future, hydraulic dampers were provided as a measure.

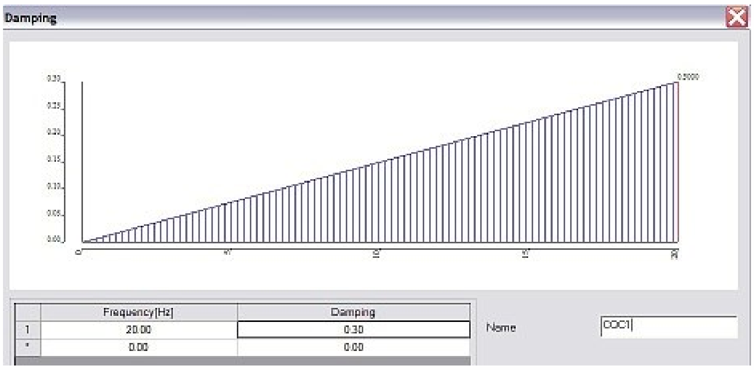
In SCIA Engineer, different damping methods are available.

First of all, the user is able to input uniform damping which influences the entire structure. For example, the damping value is taken into account in the harmonic analysis by means of the logarithmic decrement:

$$\Lambda = \frac{2\pi\xi}{\sqrt{1-\xi^2}}$$

with Xi being the damping ratio of the structure.

For the CQC-method in a seismic analysis, it's also possible to define a damping curve:



In the third case, a functionality non proportional damping is provided in SCIA Engineer.

Damping can have different causes. The component that is always present is structural damping. Structural damping is caused by hysteresis in the material: the transfer of small amounts of energy into warmth for each vibration cycle possibly increased by friction between internal parts.

Other causes can be the foundation soil of the building and aerodynamic damping due to the diversion of energy by the air. In many cases, damping is increased by adding artificial dampers to the structure.

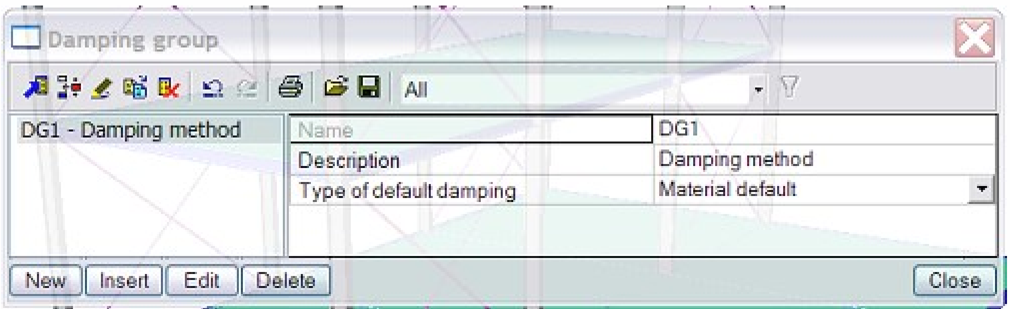
Non proportional damping allows the user to input manually dampers into the system and also to calculate the influence of the damping of the material. Structural systems composed of several structural elements with different properties can present high nonproportional damping.

Non proportional damping

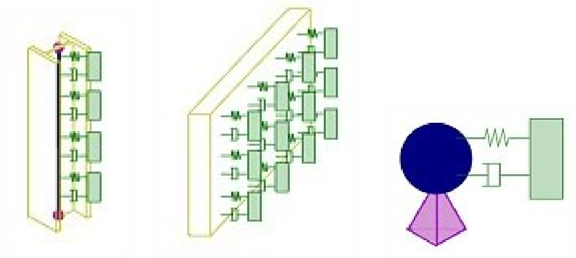
The module non proportional damping gives a solution to take into account the natural damping of the different kinds of materials in the structure. The logarithmic decrement of steel differs for example from that of concrete caused by another value of the damping ratio.

On top of this, the user is able to attribute manually dampers (by means of damping ratios) to the different elements of the system.

When no damping ratio is inputted on an element, a default value will be used. As default material damping or a global default for damping will be taken into account, dependent on the setting chosen by the user.



In SCIA Engineer, damping can be specified on 1D elements, 2D elements and supports.



The damping of each of these elements (or substructures) will be used to calculate a modal damping ratio for the whole structure for each Eigenmode. In the literature this is described as Composite Damping.

Composite damping is used in partly bolted, partly welded steel constructions, mixed steel-concrete structures, constructions on subsoil, ...

For structural systems that consist of substructures with different damping properties, the composite damping matrix C can be obtained by an appropriate superposition of damping matrices Ci for the individual substructures:

$$C = \sum_{i=1}^{N} C_i$$

With:

Ci= The damping matrix for the ith substructure in the global coordinate system.

N = The number of substructures being assembled.

Different ways of describing the damping can be assumed:

Rayleigh damping

In this method the damping matrix is formed by a linear combination of the mass and the stiffness matrices

$$C_i = \alpha_i \cdot M_i + \beta_i \cdot K_i$$

Stiffness-Weighted Damping

For structures that consist of major components with different damping characteristics, composite modal damping values can be calculated using the elastic energy of the structure:

$$\xi_j = \frac{\sum_{i=1}^{N} \xi_i \cdot E_i}{E}$$

Support damping

Additional to the damping of 1D and 2D elements, SCIA Engineer allows the input of a damper on a flexible nodal support. The modal damping ratio xi is calculated by the following formula:

$$\xi_j = Alpha \cdot \frac{\Phi_{s,j}^{\;T} \cdot \left[\sum_s C_s \right] \cdot \Phi_{s,j}}{4 \cdot \omega_j}$$

Damper setup

The damper setup provides for the input of global defaults.

Base value – logarithmic decrement	Default value of logarithmic decrement.
Alpha factor for supports	Factor for supports. Must be >0; default 1.
Maximal modal damping	Is used to limit the calculated damping. Default 30%.

Defining a new damping group

Procedure to define a new damping group

1. In the Project setup dialogue > tab Functionality options Dynamics and Non proportional damping must be selected.
2. Open service Dynamics.
3. Start function Damping group.
4. The Damping group manager is opened on the screen.
5. Click button [New].
6. A new damping group is added to the list of defined groups.
7. If necessary, change the name and/or other group parameters.

Damping group parameters

Name	Specifies the name of the group.
Description	Provides a short description of the group.
Type of default damping	Global default The default values are taken from the Damper setup. Material default The default values are taken from material properties.

Defining a new damper

A damper can be defined in a support, on a beam membe, on a slab.

Procedure to define a new damper

1. Open service Dynamics.
2. Start function Dampers.
3. If no damping has been defined so far, the Damping groups manager is opened on the screen. Define at least one damping group.
4. The Dampers branch is opened in the tree menu bar.
5. Select and start the function corresponding to the required type of damper:
   1. 1D damping,
   2. 2D damping,
   3. Node damping.

6. Fill in the parameters.
7. Select the appropriate 1D member/slab/support where the damper is to be installed.
8. End the function.

1D damping

Name	Specifies the name of the damper.
Type	Select the type of the damping parameter. Logarithmic decrement Relative damping Rayleigh damping
Value Alpha / Beta	Specifies the value of the parameter selected in the item above. Note: The Rayleigh damping requires the definition of two parameters. The remaining two types need just one value.

2D damping

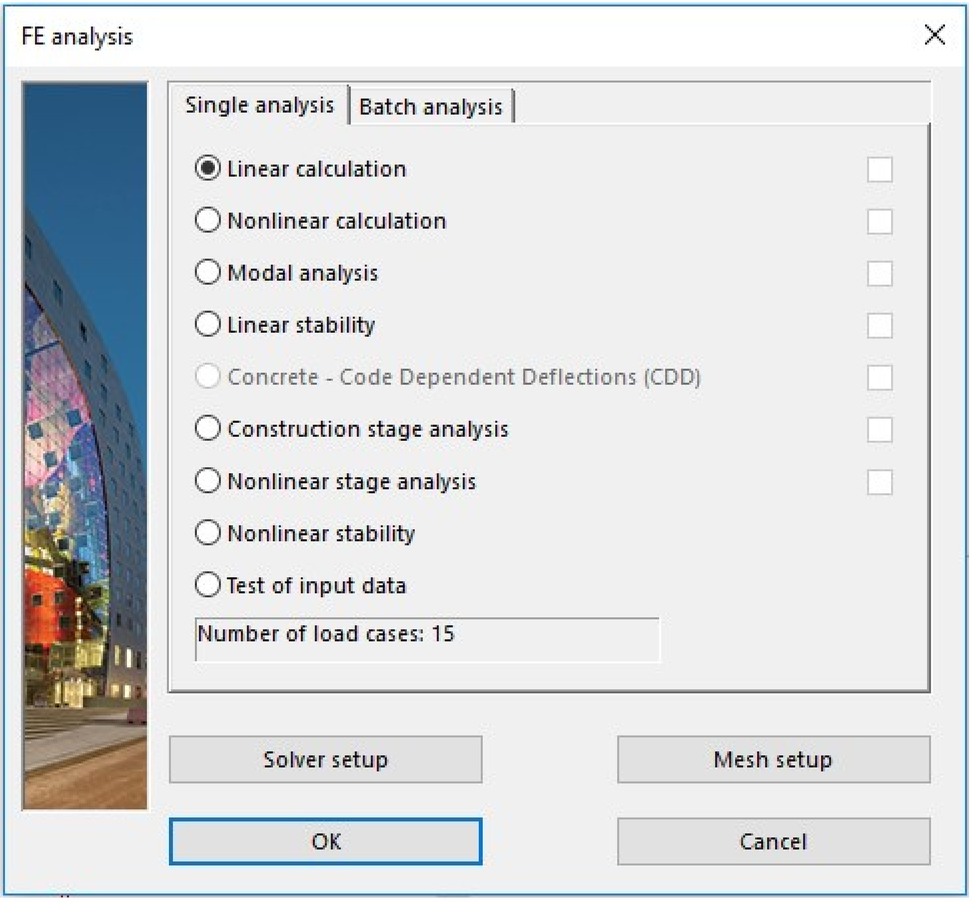
Name	Specifies the name of the damper.
Type	Select the type of the damping parameter. Logarithmic decrement Relative damping Rayleigh damping
Value Alpha / Beta	Specifies the value of the parameter selected in the item above. Note: The Rayleigh damping requires the definition of two parameters. The remaining two types need just one value.

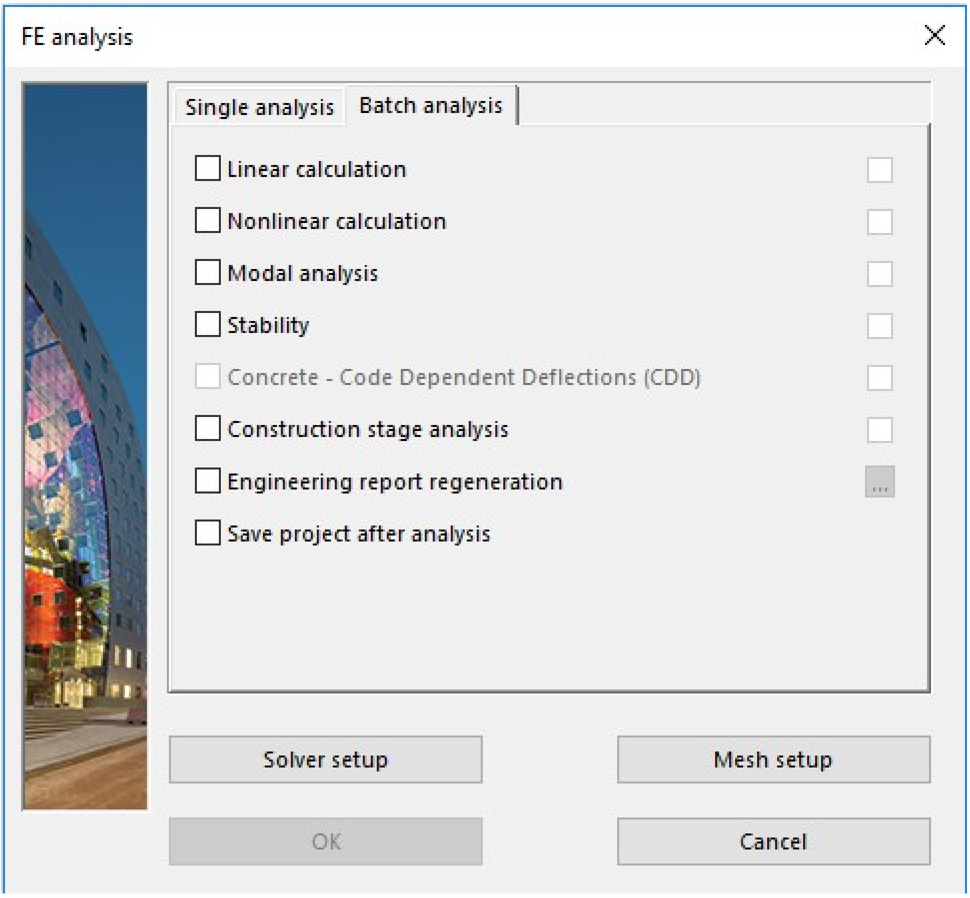
Node damping

Name	Specifies the name of the damper.
Damping X Damping Y Damping Z	Defines the damping in individual directions of the global coordinate system.

Performing the calculation

Calculation dialog for v16 and earlier





Single analysis / Batch analysis

In calculation dialog for version 16 and earlier there are two main view on list of possible analysis.

Single analysis where it is possible to choose one type of calculation and Batch analysis where it is possible to choose more than one type of calculation and these calculations will be perform behind a row.

Types of calculations

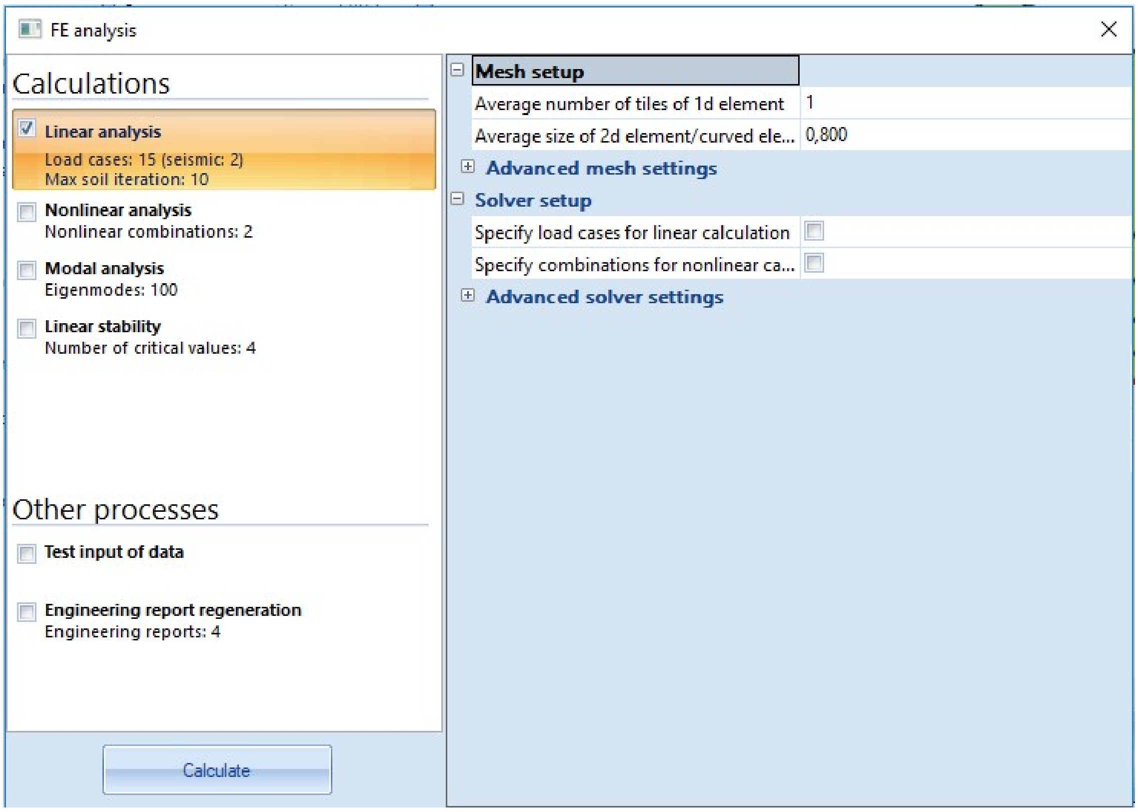
These items define all possible types of calculations, which can be performed on current model.

Linear calculation	Linear calculation Dynamic seismic calculation Harmonic band analysis Karman vibration Dynamic natural vibration calculation
Nonlinear calculation	Nonlinear calculation
Modal analysis	Calculation model for dynamic analysis
Linear stability	Stability combination
Concrete - Code Dependent Deflections (CDD)	Concrete - Code Dependent Deflections (CDD)
Construction stage analysis	Construction stage analysis Linear versus Non-linear construction stages
Nonlinear stage analysis	Nonlinear stage analysis
Nonlinear stability	Nonlinear stability
Test of input data	It is basically preparation of data from model for Linear calculation. It can be something like check if everything is OK before run of calculation. After performing it, new service "2D data viewer" is available in service "Calculation, mesh", where it is possible to check generated loads on members and other parameters of model.

Buttons

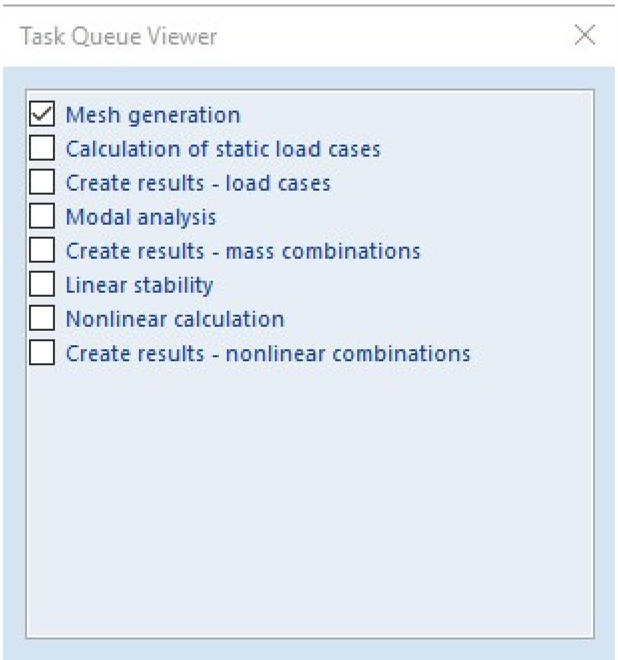
Solver setup	Settings for calculation - Solver setup
Mesh setup	Settings for FEM mesh on members - Mesh setup
OK	This button starts calculation.
Cancel	This button closes calculation dialog without calculation.

Calculation dialog for v17



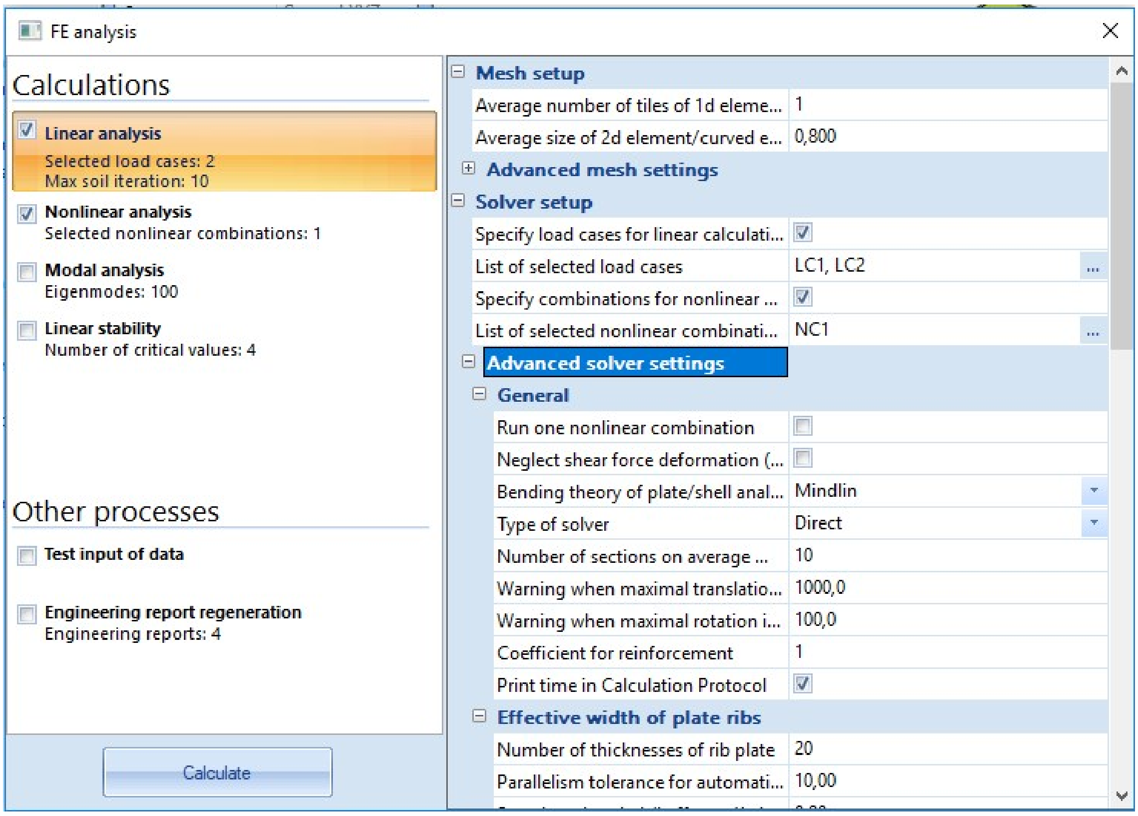
General description

Calculation dialog is designed as a batch analysis where it is possible to check several analysis to the calculation queue. If as an input for some types of analysis it necessary to perform another analysis before, this analysis will be automatically add to the Task queue viewer. This dialog is visible during calculation and it gives information about current order in batch analysis and current state of calculation.



Calculation dialog itself is divided into two main parts. In the left part there is a list of available calculations and other processes with some info about every calculations like number of load cases, number of seismic load cases, number of soilin iterations, number of eigenmodes or number of critical values for stability analysis.

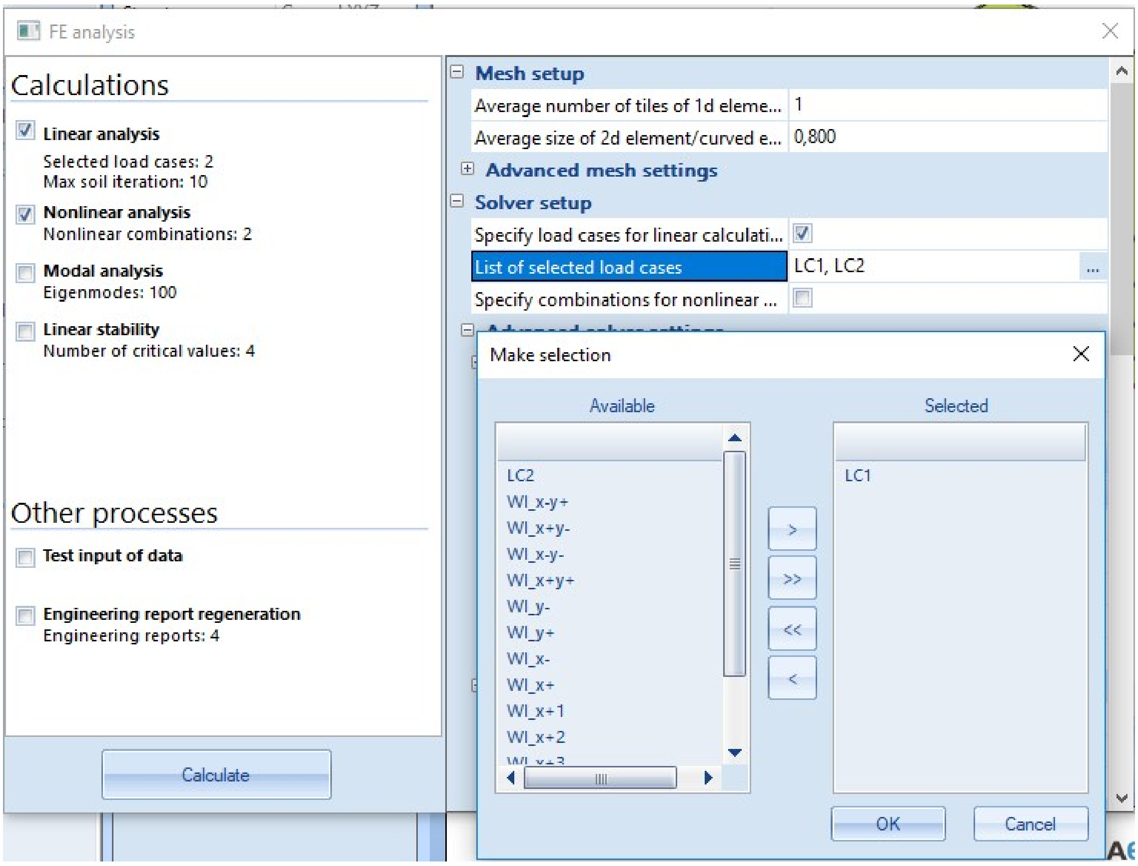
In the right part there is a direct view to mesh and solver setup. Items in both setup are divided into two groups...main group where are most important items and which are still visible in calculation dialog... and Advanced mesh settings and Advanced solver settings where are others items from setup which are packed. But it is possible to unpack them quickly and set some parameters directly in calculation dialog.

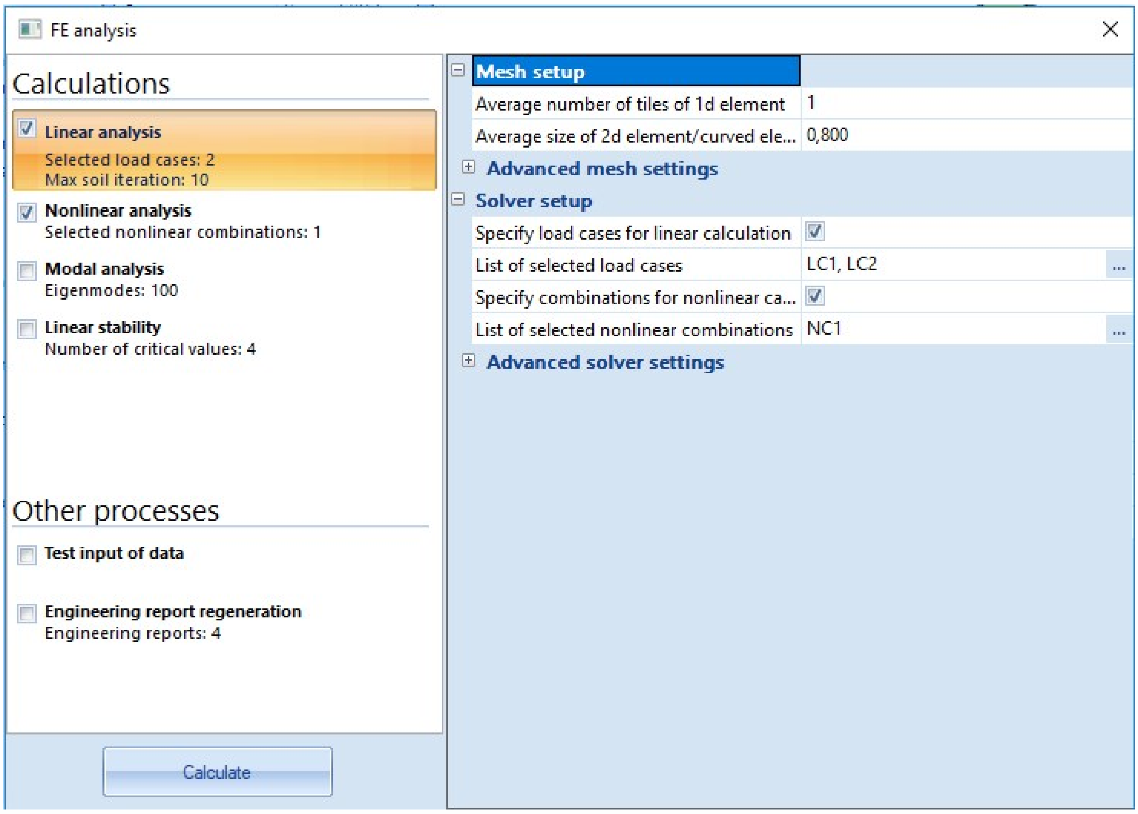


Selection of load cases or nonlinear combinations for calculation

For solver setup there are visible two items : "Specify load cases for linear calculation" and if there are some nonlinear combinations in model there is item "Specify combinations for nonlinear calculation". After check these items new dialog "Make selection" will appear with all possible load cases or nonlinear combinations and it is possible to make a selection for this calculation. After that selected items are displayed in solver setup and in info text before analysis there is new info for example : Selected load cases : 2 .

After calculation in service results there are available results only for load cases and nonlinear combinations which were selected. Others have symbol * before name, which means no results.


FE analysis
Calculations
Linear analysis
Selected load cases: 2
Max soil iteration: 10
Nonlinear analysis
Nonlinear combinations: 2
Modal analysis
Eigenmodes: 100
Linear stability
Number of critical values: 4
Other processes
Test input of data
Engineering report regeneration
Engineering reports: 4
Calculate
Mesh setup
Average number of tiles of 1d eleme...
1
Average size of 2d element/curved e...
0,800
Advanced mesh settings
Solver setup
Specify load cases for linear calculati...
List of selected load cases
LC1, LC2
Specify combinations for nonlinear ...
Make selection
Available
Selected
LC2
WI_x-y+
WI_x+y-
WI_x-y-
WI_x+y+
WI_y-
WI_y+
WI_x-
WI_x+
WI_x+1
WI_x+2
LC1
>
>>
<<
<
OK
Cancel


FE analysis
Calculations
Linear analysis
Selected load cases: 2
Max soil iteration: 10
Nonlinear analysis
Selected nonlinear combinations: 1
Modal analysis
Eigenmodes: 100
Linear stability
Number of critical values: 4
Other processes
Test input of data
Engineering report regeneration
Engineering reports: 4
Calculate
Mesh setup
Average number of tiles of 1d element
1
Average size of 2d element/curved ele...
0,800
Advanced mesh settings
Solver setup
Specify load cases for linear calculation
List of selected load cases
LC1, LC2
Specify combinations for nonlinear ca...
List of selected nonlinear combinations
NC1
Advanced solver settings

Types of calculations

These items define all possible types of calculations, which can be performed on current model.

Linear calculation	Linear calculation Dynamic seismic calculation
Nonlinear calculation	Nonlinear calculation Limitations for version 17.0
Modal analysis	Calculation model for dynamic analysis
Linear stability	Stability combination

Types of other processes

These items define other processes, which can be performed on current model.

Test of input data	It is basically preparation of data from model for Linear calculation. It can be something like check if everything is OK before run of calculation. After performing it, new service "2D data viewer" is available in service "Calculation, mesh", where it is possible to check generated loads on members and other parameters of model.
Engineering report regeneration	Engineering report

Buttons

Calculate	This button starts calculation.

Note 1: If project is calculated and it has results, there is no change in geometry and other settings and user clicks on calculate - calculation is not performed and previous results are used.

Note 2: If project is calculated and it has results but in older version of SEN, there is no change in geometry and other settings and user clicks on calculate - results are deleted and project is recalculated.

Adjusting the calculation parameters

The procedure to adjust calculation parameters

1. Open the Solver setup dialogue
   1. either use menu function Setup > Solver,
   2. or use tree menu function Calculation > Solver setup.
2. The Setup dialogue is opened on the screen.
3. Adjust the parameters.
4. Confirm with [OK] button.

Note: The adjustment of these parameters may affect the layout of the calculation dialogue that opens on the screen when a calculation is started.

Performing the calculation

The procedure for performing of the calculation

1. Call function Calculation:
   1. either using menu function Tree > Calculation, Mesh > Calculation,
   2. or using tree menu function Calculation, Mesh > Calculation.
2. The Calculation settings dialogue opens on the screen (see below).
3. Adjust the parameters for calculation.
4. Confirm with [OK].
5. The calculation is started and solver report dialogue is opened on the screen (for small models the dialogue may just flash).
6. When the calculation has been finished, close the calculation report dialogue.

9. Proceed to evaluation of result.

Note: All the calculation parameters may be adjusted in the "Solver Setup"

Controlling and reviewing the calculation process

Once a calculation has been started a Solver report dialogue opens on the screen.

For small models the dialogue may just "flash" on the screen and disappear again.

On finishing the calculation, the program shows the dialogue with the result of the calculation.

If everything is OK, the Solver report dialogue can be closed and the user may proceed to the evaluation of results. If anything went wrong during the calculation, a message is displayed and it’s up to the user to resolve the situation.

Performing the repetitious calculations

Very often it may be necessary to repeat the calculation with the same calculation settings. It is possible to repeat a normal calculation. In addition, SCIA Engineer offers function Hidden calculation. This function starts the calculation without showing any information on the screen. Once the calculation is finished, all possible open windows with displayed results are automatically regenerated.

The Hidden calculation can be performed by means of:

- either menu function Tree > Calculation, Mesh > Hidden calculation,
- or tree menu function Calculation, Mesh > Hidden calculation,
- or button [Hidden calculation] () on toolbar Project.

Note: If just one type of calculation is available in the calculation dialogue, the hidden calculation simply runs on the background. If, however, two or more calculation types are accessible (depending on project and solver settings), the calculation dialogue is displayed and you must choose the required calculation type.

Repairing the instability of model

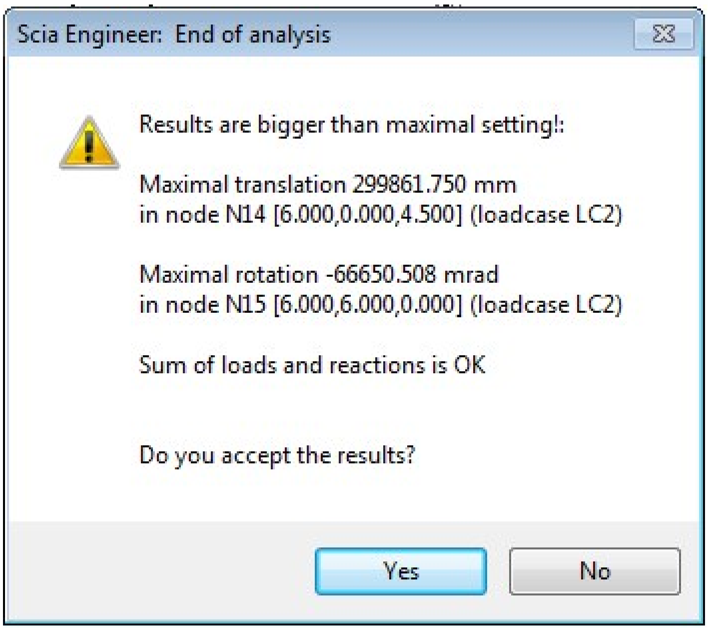
It may happen that the model is so defined that the numerical solution is impossible. Most often some kind of numerical instability may occur due to mistakes in the definition of boundary conditions.

Maximum displacement has been reached

The first case is that the numerical solution itself was correct, but the results seem to be distorted. This situation can be revealed by the check of maximum permissible vale of displacement and rotation. If the adjusted values are exceeded, a warning is given.

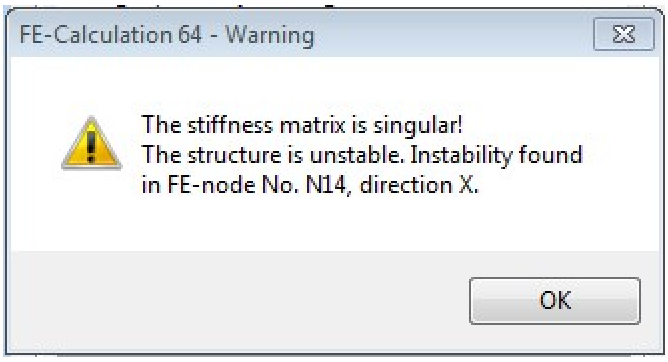
The results may be reviewed even if this situation happens. It is up to the user's experience to decide whether the structure is so soft and the large deformation is reasonable or whether a mistake was made in the definition of the model.

The point where the maximum displacement has been found is stated in the warning dialogue.

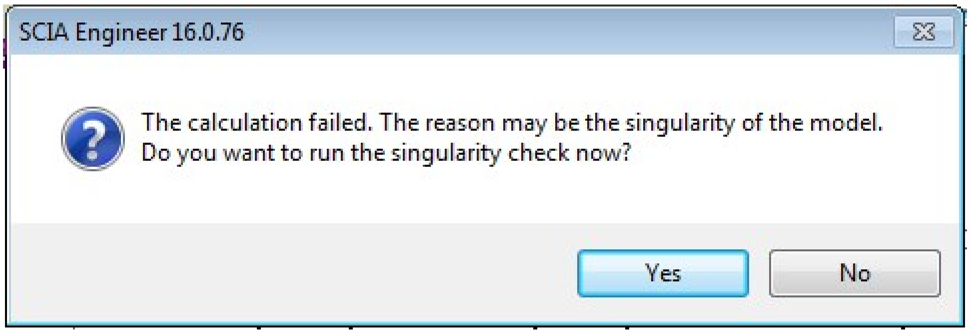


Singular stiffness matrix

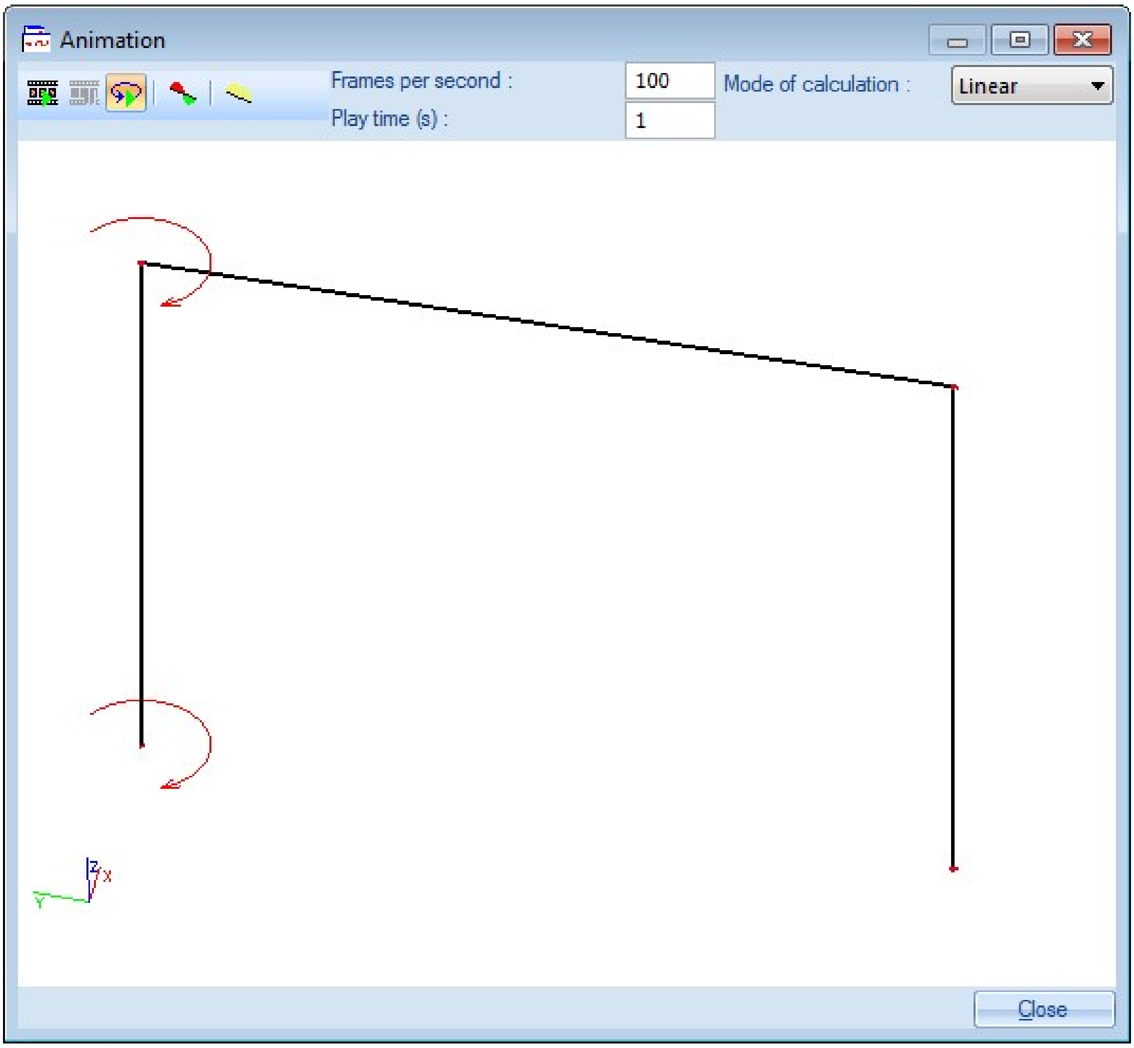
If the stiffness matrix is singular, the solution cannot be obtained at all. The user is informed about the problematic place in the model. The place is stated in the warning dialogue.



In case of linear analysis a possibility to run singularity checks is offered.

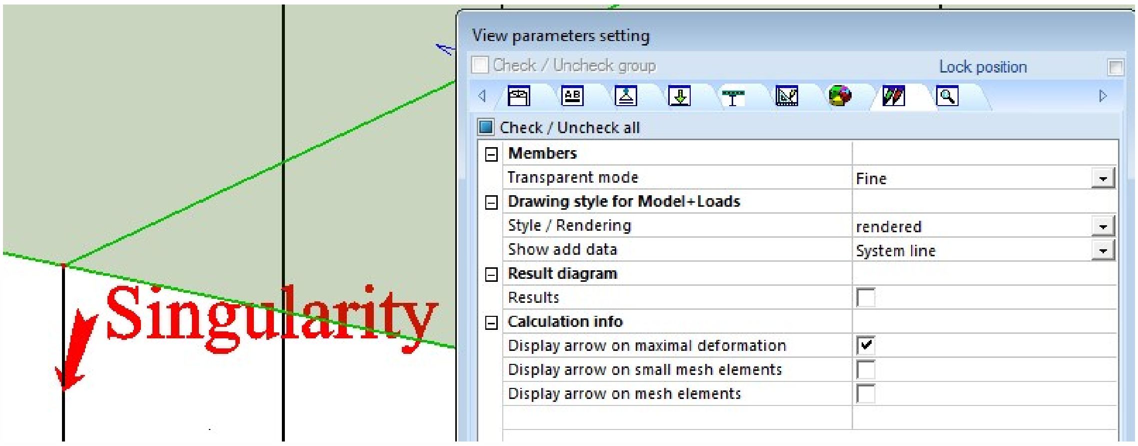


Then an animation window showing translations and rotations is displayed.



Note: The result of singularity check is limited. Not all input additional data are fully implemented in this check yet. Therefore in some cases the result of singularity check can differ from the real problem.

The place of instability can be also marked with an arrow which can be displayed by means of View parameter settings - tab Misc.- Display arrow on maximal deformation.



Solution methods

Direct solution

This is a standard Cholesky solution based on a decomposition of the matrix of the system. The advantage is that it can solve several right sides at the same time. This type of solution is effective especially for small and middle-size problems when disk swapping is not necessary. The limit depends on the size of the problem and on the size of available RAM memory.

It can be said that this solution is more convenient for most of problems.

Disadvantage of this solution may emerge with extremely large problems. The calculation time may rise significantly if RAM size is insufficient. What's more, if the available disk space is not large enough, the problem cannot be solved at all.

If the problem is excessive and of poor numerical condition, the rounding error may be so big that it exceeds the acceptable limit. This may result in imbalance between resultants of load and reactions. The difference between the total sums of loads and reactions should not be greater than about 0.5%. But even the value of 0.1% suggests that the results may be suspicious.

Generally, the direct solver should be used only for beam structures (without any 2D members) or planar structure composed of 2D members (i.e. a plate or a wall). In other cases the direct solver should be used as a default solution method. The application of iterative solution depends on the total number of nodes, band width and memory size of the particular computer. If the direct solution leads to an excessive disk swapping, the process is slowed down sifgnificantly and the iterative solution must be employed. This solver does not require so much memory – 150 000 nodes needs about 250 MB RAM. Other reason for the application of iterative solution may be poor determinetness of the equation system. These numerical problems can result in a discrepancy between the total load and sum of reactions. If this difference is greater than 5%, a warning is issued and the direct solver should be replaced by the iterative one.

Iterative solution

The Incomplete Cholesky conjugate gradient method is applied.

Its advantage is minimal demand on RAM and disk size. Therefore, the solution is convenient especially for extremely large problems that cannot be solved by means of direct solution or whose calculation time would be enormous for that kind of solution due to excessive disk operations.

Another advantage is that due to the ability of continuous improvement of accuracy the method is able to find technically accurate solution even for equation systems that would be numerically unstable in the direct solution.

The disadvantage is that the method can employ only one right side at a time and this increases the time demands for equation systems with several right sides.

Note: See the note in the Direct solution.

Timoshenko theory

The algorithm is based on the exact Timoshenko's solution of a 1D member. The axial force is assumed constant during the deformation. Therefore, the method is applicable for structures where the difference of axial force obtained by 1st order and 2nd order calculation is negligible (so called well defined structures). This is true mainly for frames, buildings, etc. for which the method is the most effective option.

The method is applicable for structures where rotation does not exceed 8°.

The method assumes small displacements, small rotations and small strains.

If 1D members of the structure are in no contact with subsoil and simultaneously they do not form ribs of shells, no fine division of 1D members into finite elements is required. If the axial force is lower than the critical force, this solution is robust. The method needs only two steps, which leads to a great efficiency of the method.

The first step serves only for solution of axial force. The second step uses the determined axial forces for Timoshenko´s exact solution. The original Timoshenko´s solution was generalised in SCIA Engineer and the shear deformations can be taken into account.

Newton-Raphson method

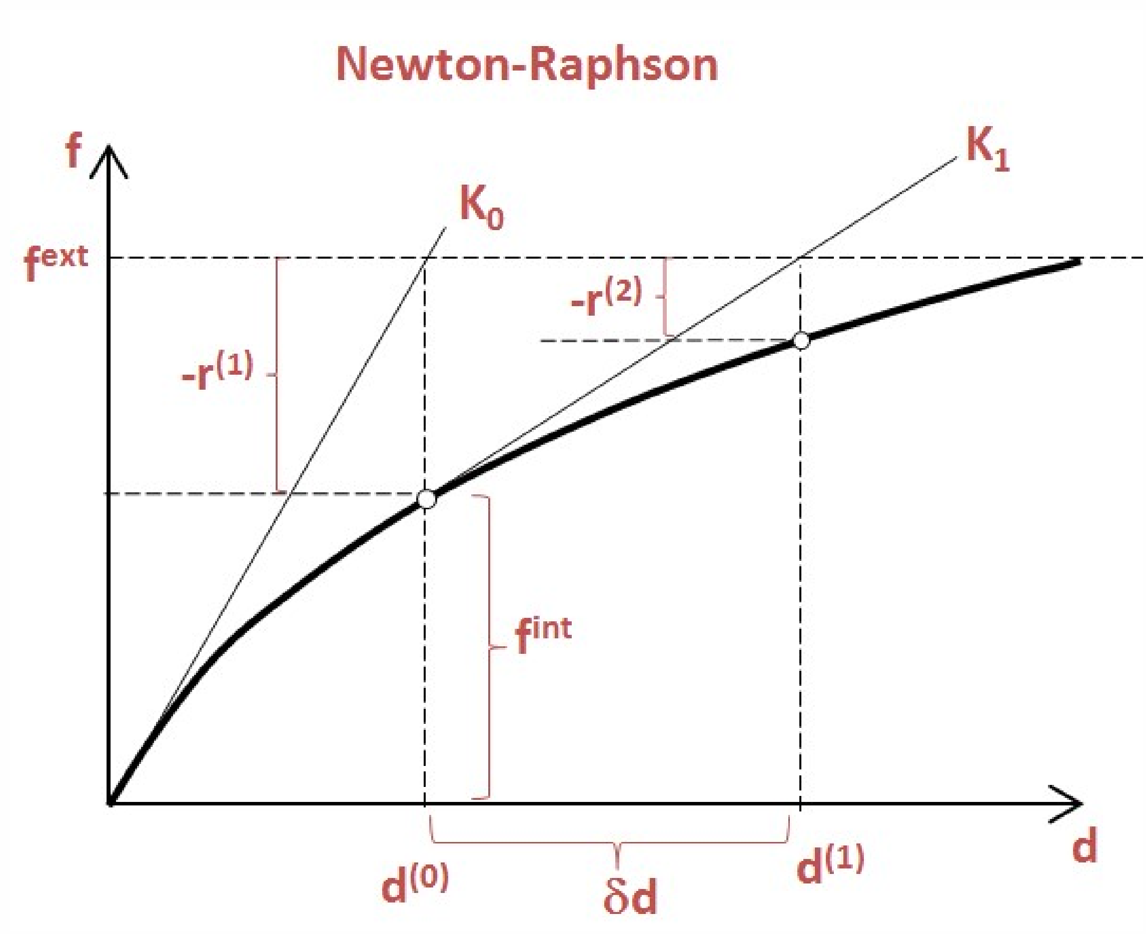
The algorithm is based on Newton-Raphson method for solution of non-linear problems. The method is robust for most of problems. It may, however, fail in the vicinity of inflection points of loading diagram. This may occur for example at compressed 1D members subject to small eccentricity or to small transverse load. Except for the mentioned example, the method can be applied for wide range of problems. It provides for solution of extremely large deformations.

The load acting on the structure can be divided into several steps. The default number of steps is eight. If this number is not sufficient, the program issues a warning.

The rotation achieved in one increment should not exceed 5°.

The accuracy of the method can be increased through refinement of the finite element mesh or by the increase in total number of increments. For example, the solution of a single beam divided to a single finite element will not give sufficient results.

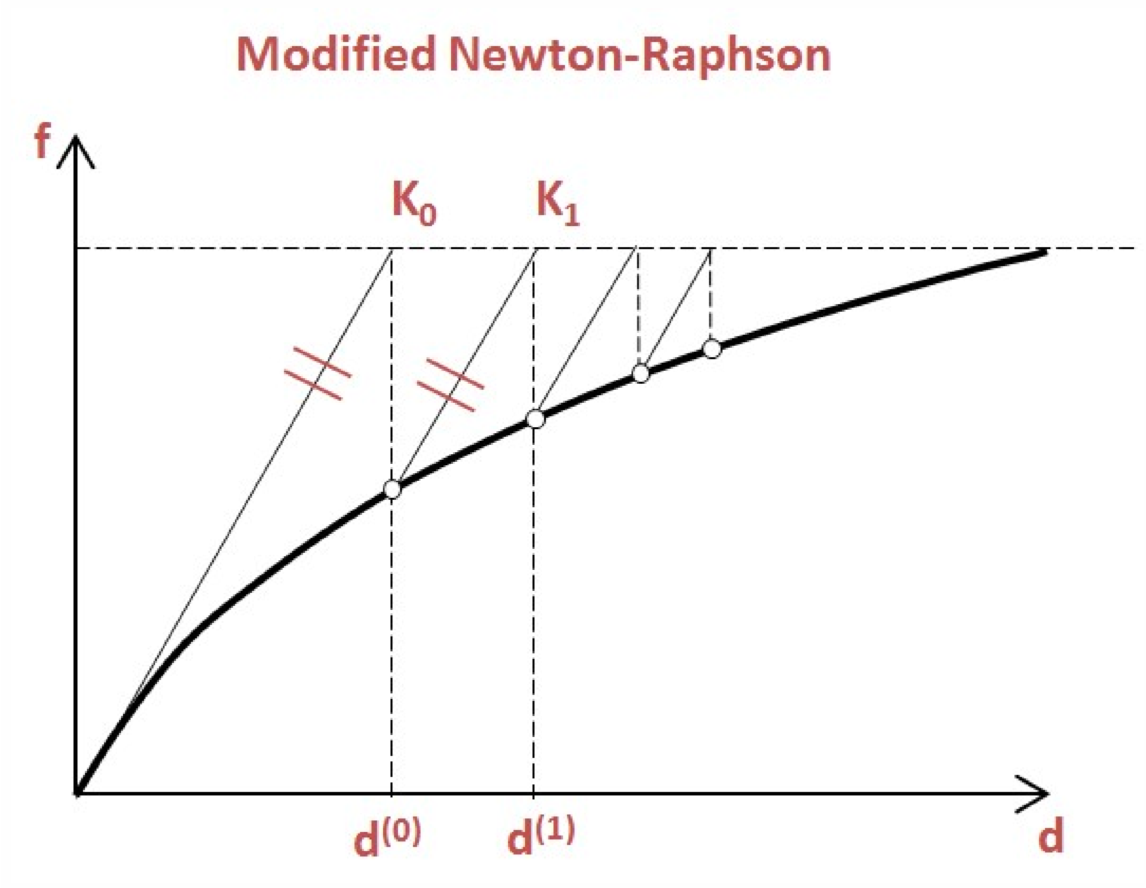
In some specific cases, high number of increments may solve even problems that tend to a singular solution which is typical for the analysis of post-critical states.



Note : This method requires that a 1D member is divided to at least four (4) finite elements. Usually, such division is adjusted automatically whenever Newton- Raphson method is selected for calculation.

Modified_Newton_Raphson_method

The algorithm is based on Newton-Raphson method for solution of non-linear problems. The method is better in some case of solution.

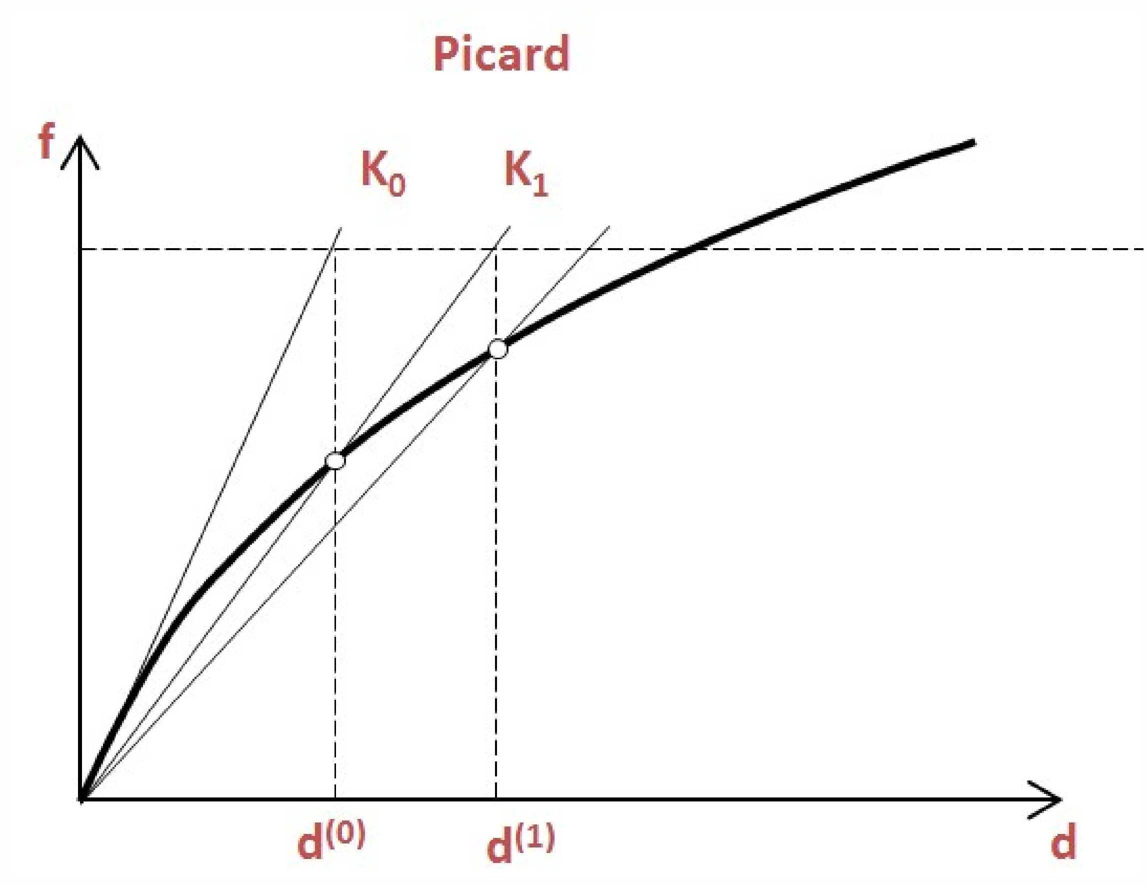


Note : This method requires that a 1D member is divided to at least four (4) finite elements. Usually, such division is adjusted automatically whenever Modified Newton- Raphson method is selected for calculation.

Picard_method

The Picard method is regarded as complementary method. It can be used when the Newton-Raphson method fails. The method is more robust but slower. The Picard method can be use alone (the direct iteration method, using the secant stiffness matrix) or in the combination with the Newton-Raphson method. In this case the calculation starts with Newton-Raphson and then switches to Piccard.

Main differences: the Newton Raphons method uses tangent stiffnesses, but Picards method uses secant stiffnesses.

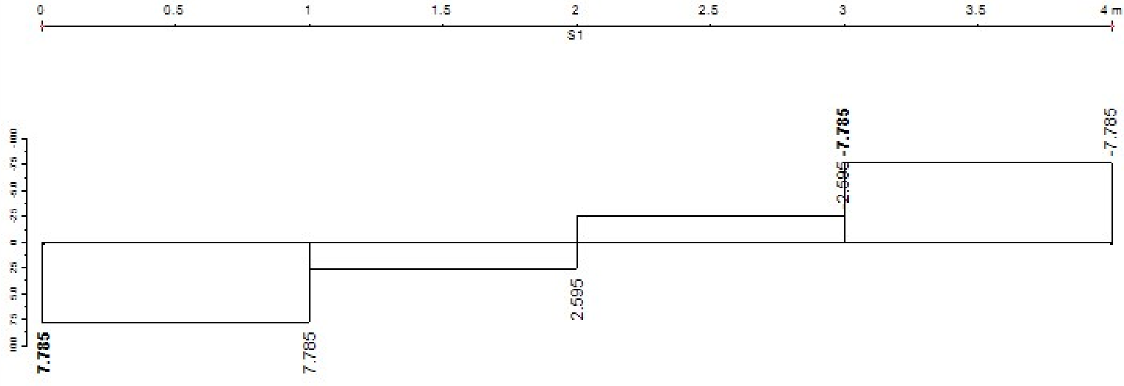


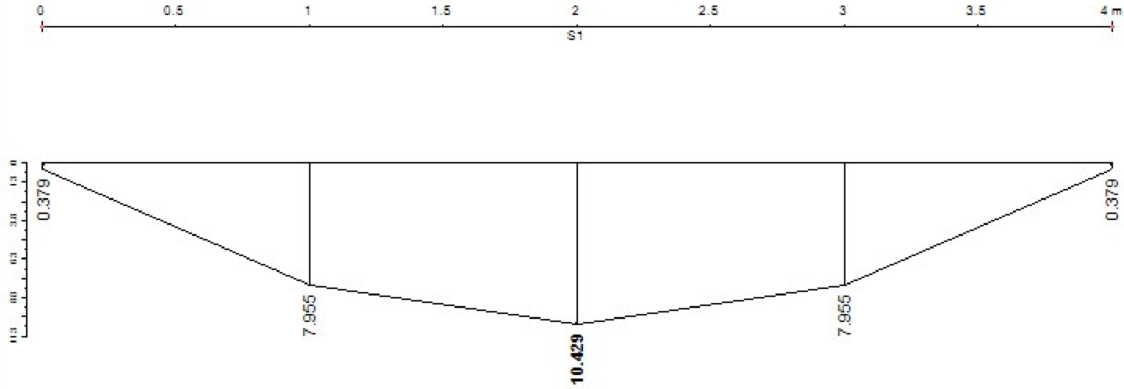
Note : This method requires that a 1D member is divided to at least four (4) finite elements. Usually, such division is adjusted automatically whenever Picard method is selected for calculation.

Smoothening of non-averaged values

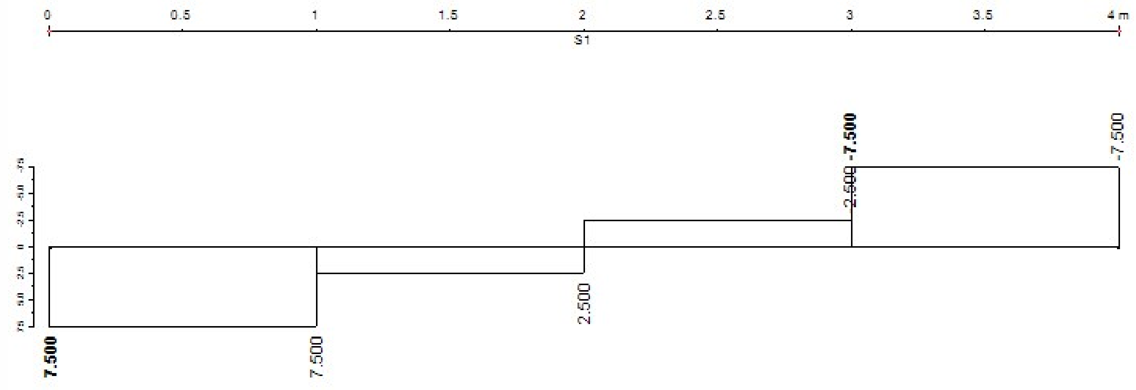
The solver internally calculates the internal forces in integration points and at the end the calculated values are extrapolated from the integration points to the nodes of the 2D element using the formulas for a hyperbolic paraboloid. These values are then declared the non-averaged nodal values. Smoothening of the non-averaged values is demonstrated on a calculation of a plate strip with the dimensions of 4x3 m. The plate strip is simply supported along the shorter edges, the Poisson coefficient μ = 0, and load p = 5 kNm-2. The strip is divided to 4 x 3 finite elements and is solved using the Kirchhoff's and Mindlin's theory. The picture shows the longitudinal section in the middle of the strip.

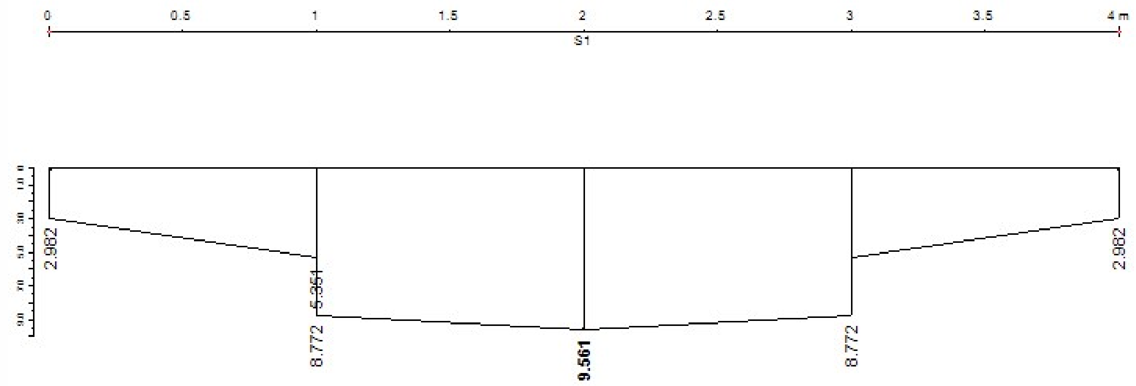
Distribution of vx and mx according to Kirchhoff, non-averaged values determined by the solver





Distribution of vx and mx according to Mindlin, non-averaged values determined by the solver



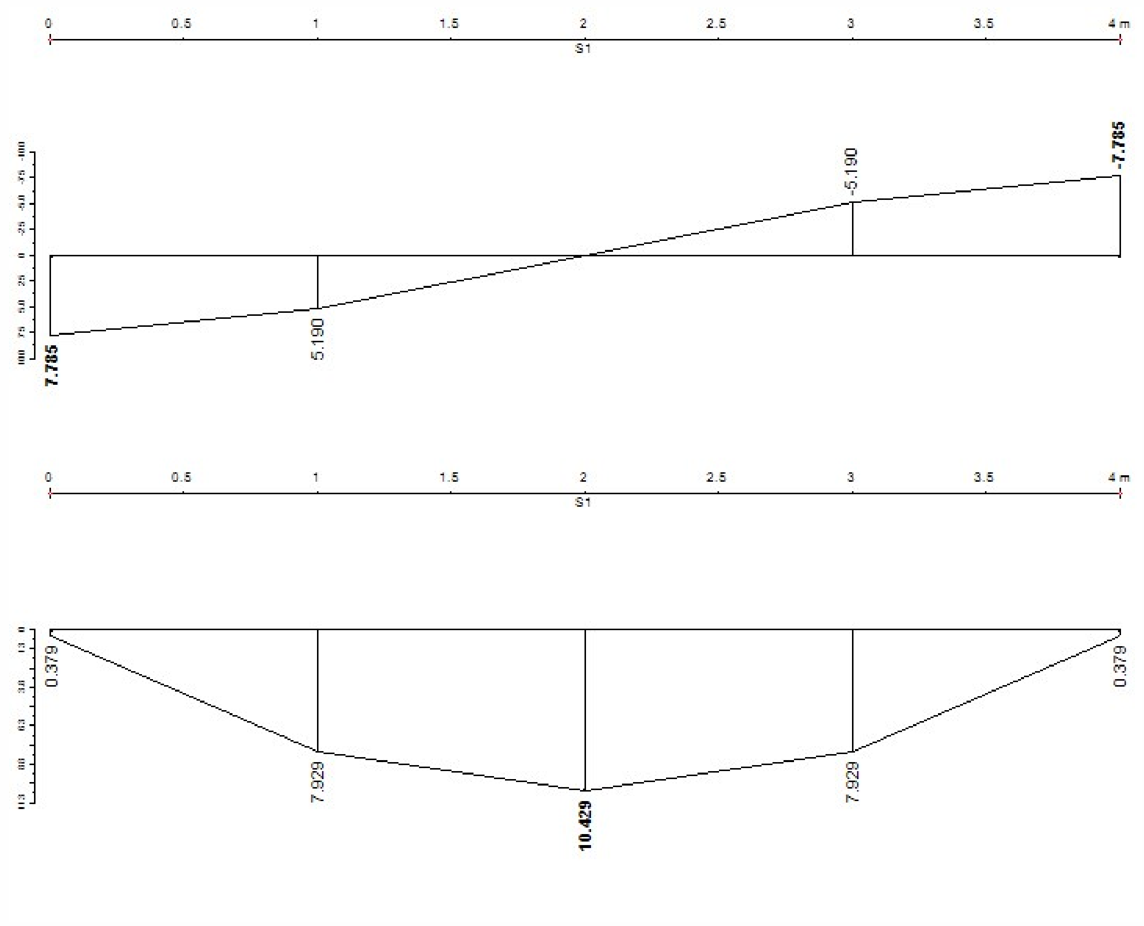


The following algorithms are used for the averaged values in the nodes:

a) Standard averaging

The nodal value is obtained as an arithmetic average from the values in the neighbouring selected 2D elements. When option "averaged values on 2D macros" is selected, the values from the neighbouring 2D elements are averaged only if the elements are located on one 2D macro, on the same side of possible internal edge and if the node is not defined by the user as an internal node. In that cases the values are not averaged and discontinuity in the distribution of internal forces occurs.

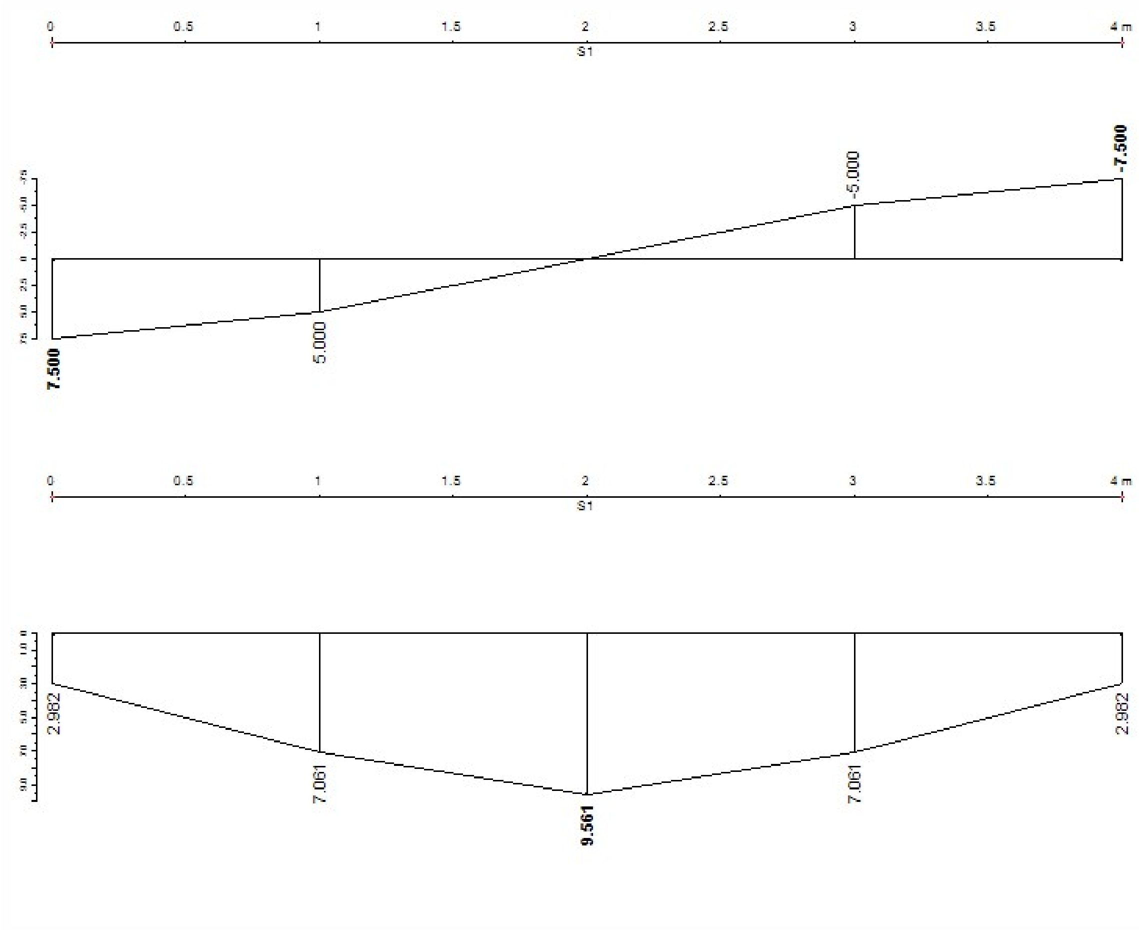
Distribution of vx and mx according to Kirchhoff, averaged values



E.g.

$$v_x(1) = \frac{7.785 + 2.595}{2} = 5.19$$
$$m_x(1) = \frac{7.954 + 7.904}{2} = 7.929$$

Distribution of vx and mx according to Mindlin, averaged values



E.g.

$$v_z(1) = \frac{7.5 + 2.5}{2} = 5$$

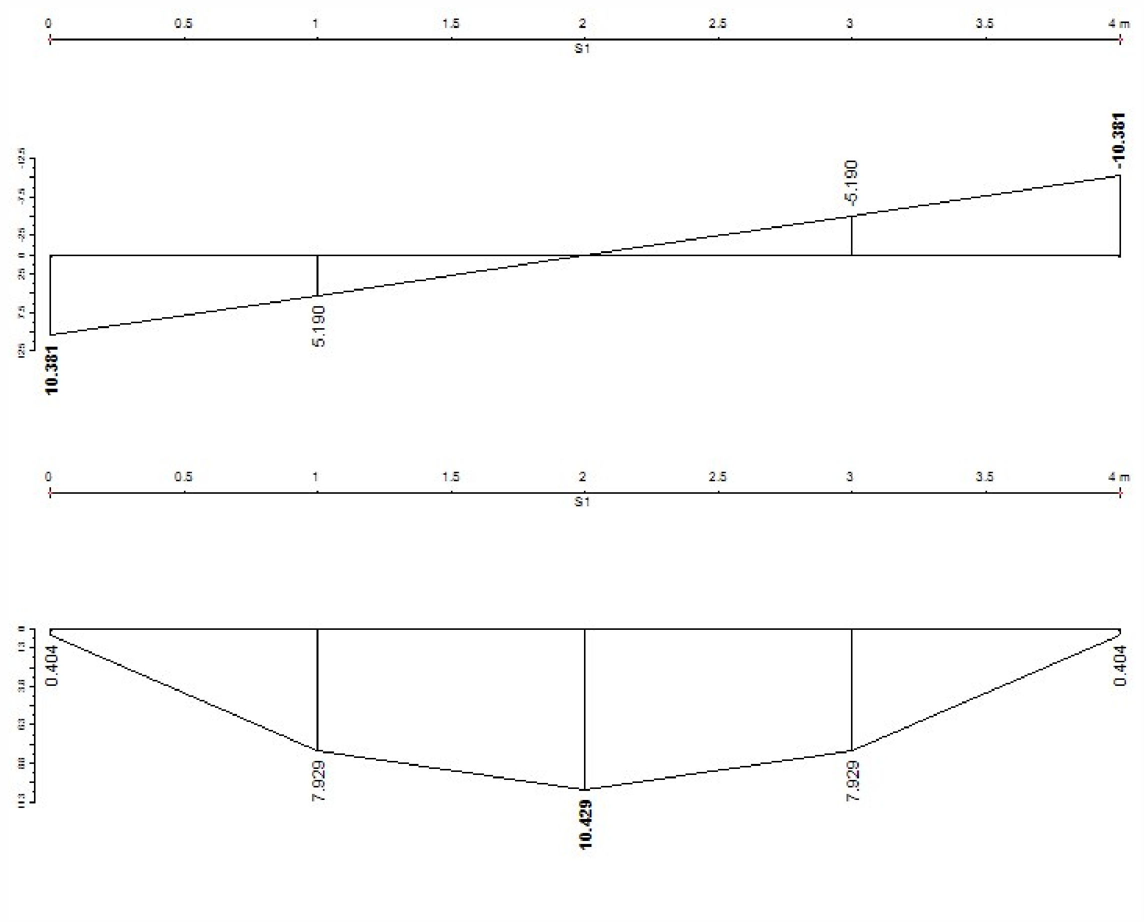
$$m_z(1) = \frac{8.772 + 5.351}{2} = 7.061$$

b) Smoothening along edges

It can be seen from the previous pictures that the standard averaging results in step-changes of the values along the edges of the structure. It results from the fact that there is nothing the value on the edge could be averaged with. Therefore, another approach is used along the edges.

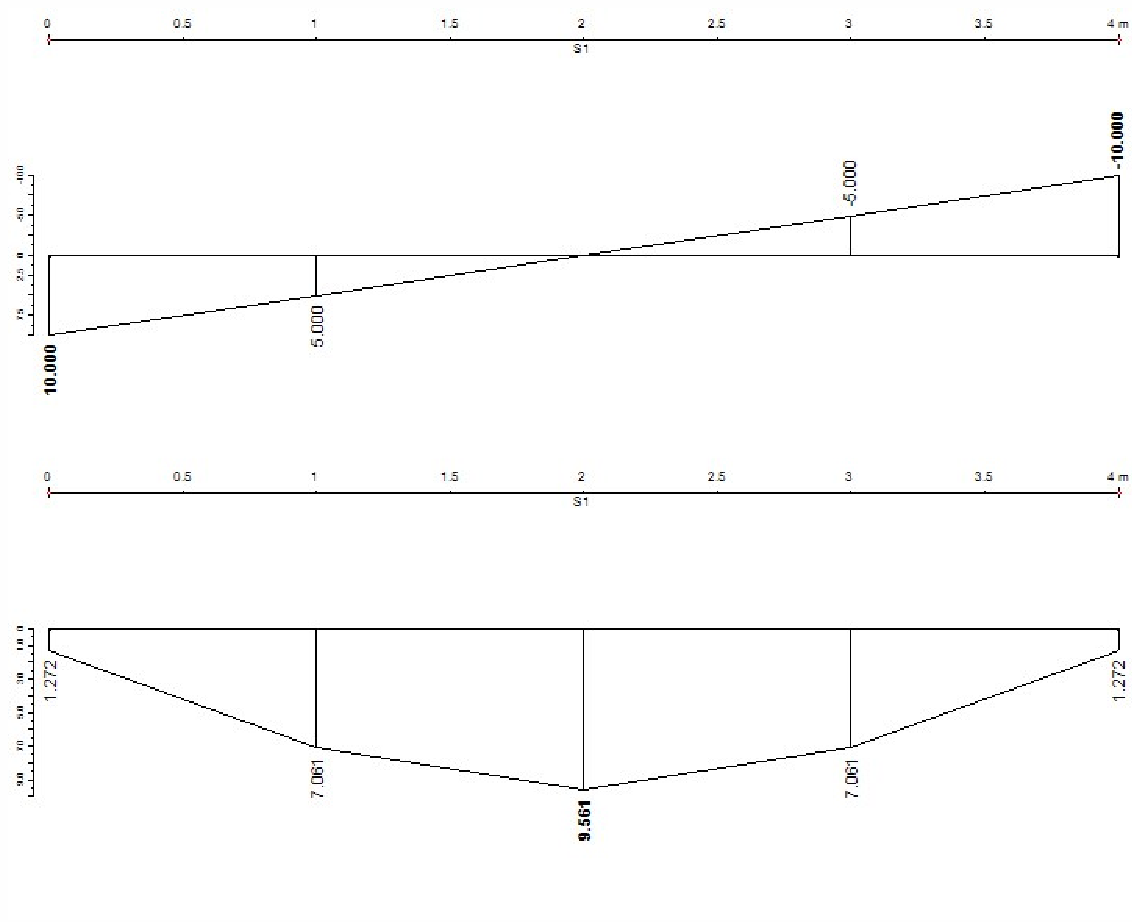
When option "averaged values on the whole structure" is selected, the nodes on a 2D macro edge that is not connected to another 2D macro or that is connected to a macro which is not selected are considered to be the edge of the structure. When option "averaged values on 2D macros" is selected then the nodes on all edges of 2D macros, nodes on internal edges and internal nodes defined by the user are considered to be the edge of the structure. In these situations, the averaging is done in two steps. In the first step, the averaged nodal are calculated in nodes that are not located on the edge of the structure. Then the values in the nodes of the elements on the edge of the structure are calculated using such an algorithm to preserve the original value in the middle of the element. In the second step, the averaged nodal are calculated in nodes that are located on the edge of the structure.

Distribution of vx and mx according to Kirchhoff, smoothening along edges



$$v_x(0) = 7.785 + (7.785 - 5.19) = 10.381$$
$$m_x(0.5) = \frac{(7.954 + 0.379)}{2} = 4.167$$
$$m_x(0) = 4.167 + (4.167 - 7.929) = 0.404$$

Distribution of vx and mx according to Mindlin, smoothening along edges



$$v_x(0) = 7.5 + (7.5 - 5) = 10$$

$$m_x(0.5) = \frac{(5.351 + 2.982)}{2} = 4.167$$

$$m_x(0) = 4.167 + (4.167 - 7.061) = 1.272$$

c) Shear forces

The Kirchhoff's theory calculates the shear forces as the third derivative of the deflection:

$$v_x = -\frac{Eh^3}{12(1-\mu^2)}\left(\frac{\partial^3 w}{\partial x^3} + \frac{\partial^3 w}{\partial x \partial y^2}\right)$$

$$v_y = -\frac{Eh^3}{12(1-\mu^2)}\left(\frac{\partial^3 w}{\partial y^3} + \frac{\partial^3 w}{\partial x^2 \partial y}\right)$$

When the third derivative is calculated a significant loss of accuracy occurs. Therefore the values of shear forces calculated by the solver using the Kirchhoff's theory are not used and are calculated by an improved approach using the derivatives of moments:

$$v_x = \frac{\partial m_x}{\partial x} + \frac{\partial m_{xy}}{\partial y}$$
$$v_y = \frac{\partial m_y}{\partial y} + \frac{\partial m_{xy}}{\partial x}$$

The formulas use moment values mx, my, mxy averaged using steps (a) and (b), which results in more accurate values of shear forces than determined by the solver.

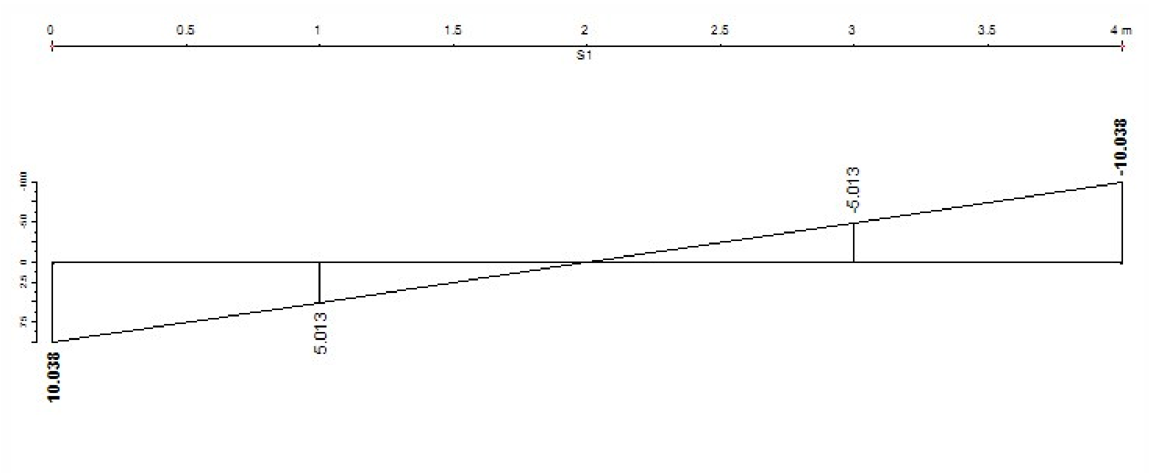
In the Mindlin's theory the shear forces are calculated as the first derivative of deflection.

$$v_x = \frac{5Gh}{6}\left(\frac{\partial w}{\partial x} + \Phi_y\right)$$
$$v_y = \frac{5Gh}{6}\left(\frac{\partial w}{\partial y} - \Phi_x\right),$$

these values are determined by the solver and are used directly.

The shear force are then averaged, according to the selected option, using steps (a) and (s).

Distribution of vx according to Kirchhoff, averaged values, smoothening along edges, values obtained as the derivative of moments



$$v_x(0)^+ = v_x(1)^- = 7.929 - 0.404 = 7.525$$
$$v_x(1)^+ = v_x(2)^- = 10.429 - 7.929 = 2.5$$
$$v_x(1) = \frac{7.525 + 2.5}{2} = 5.013$$
$$v_x(0) = 7.525 + 7.525 - 5.013 = 10.038$$

d) Moments

While the values of moments in the integration points correspond to the theory, their extrapolation to nodes using the hyperbolic paraboloid leads to some loss of accuracy. The reason is that the hyperbolic paraboloid does not replace the distribution of the moments over the element with a satisfactory precision. The calculation of the values of moments in nodes uses an improved algorithm that replaces the hyperbolic paraboloid extrapolation with an advanced shear-force-integral method. As the distribution of shear forces over the surface of the element is assumed to follow the shape of the hyperbolic paraboloid (second order surface), the integral of this surface represents a third order surface that approximates the moment distribution with a higher accuracy. This is based on the above-mentioned formulas for the calculation of shear forces using the derivative of moments. Moments are then written from these formulas:

$$m_x = \int\left(v_x - \frac{\partial m_{xy}}{\partial y}\right)dx + m_{x,0}$$

$$m_y = \int\left(v_y - \frac{\partial m_{xy}}{\partial x}\right)dy + m_{y,0}$$

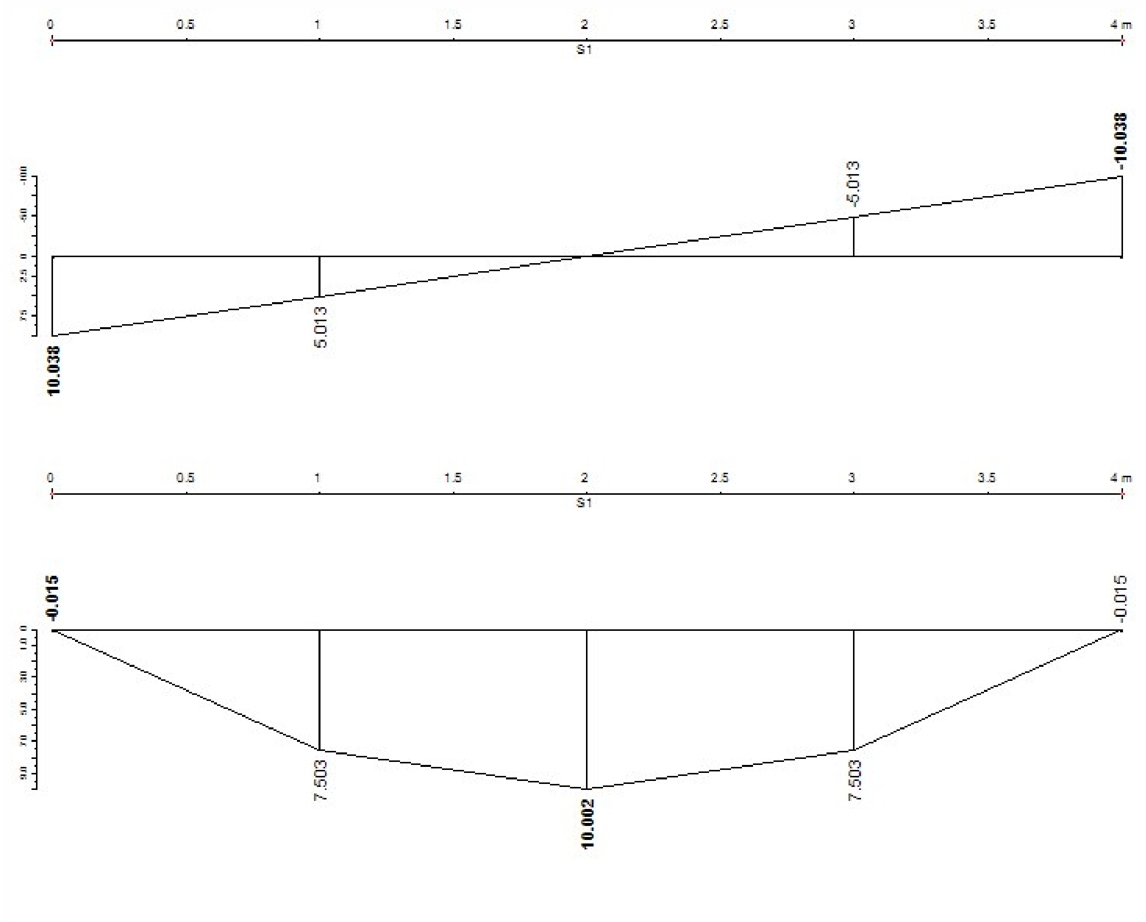
Integration constants mx,0, my,0 are calculated from the condition of the value in the middle of the element.

$$m_{x,C} = \frac{\int_S m_x dS}{S}$$

$$m_{y,C} = \frac{\int_S m_y dS}{S}$$

where S is the area of the element. The values of internal forces are calculated using steps (a), (b) and (c), the integrals are evaluated using numerical integration. The obtained values of moments in nodes are then, using the selected averaging option, averaged according to step (a).

Distribution of vx and mx according to Kirchhoff, smoothening along edges, moments obtained by integration of shear forces



E.g.

First element

$$v_x(x) = 10.038 - 5.025x$$

$$m_x(x) = \int v_x dx + m_{x,0} = 10.038x - \frac{5.025}{2}x^2 + m_{x,0}$$

$$m_{x,c} = m_x(0.5) = 4.167 = \int_0^1 10.038x - \frac{5.025}{2}x^2 + m_{x,0} =$$

$$\left[\frac{10.038}{2}x^2 - \frac{5.025}{6}x^3 + m_{x,0}x\right]_0^1 = 4.182 + m_{x,0}$$

$$m_{x,0} = m_x(0) = 4.167 - 4.182 = -0.015$$

$$m_x(1)^- = 10.038 - \frac{5.025}{2} - 0.015 = 7.511$$

Second element

$$v_x(x) = 5.013 - 5.013x$$

$$m_x(x) = \int v_x dx + m_{x,0} = 5.013x - \frac{5.013}{2}x^2 + m_{x,0}$$

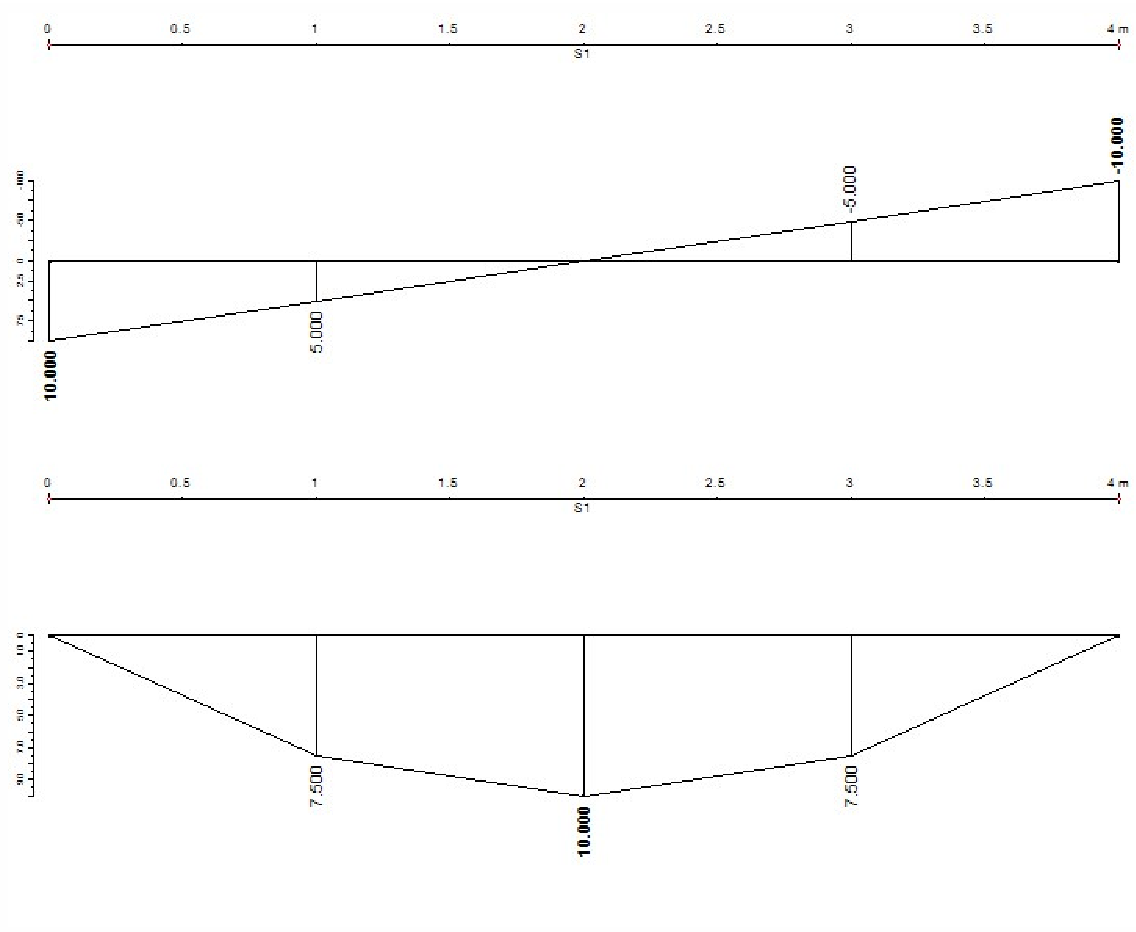
$$m_{x,C} = m_x(1.5) = 9.167 = \int_0^1 5.013x - \frac{5.013}{2}x^2 + m_{x,0} =$$

$$\left[\frac{5.013}{2}x^2 - \frac{5.013}{6}x^3 + m_{x,0}x\right]_0^1 = 1.671 + m_{x,0}$$

$$m_{x,0} = m_x(1)^+ = 9.167 - 1.671 = 7.496$$

$$m_x(1) = \frac{m_x(1)^- + m_x(1)^+}{2} = \frac{7.511 + 7.496}{2} = 7.503$$

Distribution of vx and mx according to Mindlin, smoothening along edges, moments obtained by integration of shear forces



E.g.

First element

$$v_x(x) = 10 - 5x$$

$$m_x(x) = \int v_x dx + m_{x,0} = 10x - \frac{5}{2}x^2 + m_{x,0}$$

$$m_{x,C} = m_x(0.5) = 4.167 = \int_0^1 10x - \frac{5}{2}x^2 + m_{x,0} =$$

$$\left[\frac{10}{2}x^2 - \frac{5}{6}x^3 + m_{x,0}x\right]_0^1 = 4.167 + m_{x,0}$$

$$m_{x,0} = m_x(0) = 4.167 - 4.167 = 0$$

$$m_x(1)^- = 10 - \frac{5}{2} + 0 = 7.5$$

Second element

$$v_x(x) = 5 - 5x$$

$$m_x(x) = \int v_x dx + m_{x,0} = 5x - \frac{5}{2}x^2 + m_{x,0}$$

$$m_{x,C} = m_x(1.5) = 9.167 = \int_0^1 5x - \frac{5}{2}x^2 + m_{x,0} =$$

$$\left[\frac{5}{2}x^2 - \frac{5}{6}x^3 + m_{x,0}x\right]_0^1 = 1.667 + m_{x,0}$$

$$m_{x,0} = m_x(1)^+ = 9.167 - 1.667 = 7.5$$

$$m_x(1) = \frac{m_x(1)^- + m_x(1)^+}{2} = \frac{7.5 + 7.5}{2} = 7.5$$

Initial deformations

Introduction to initial deformations

The initial deformation may be used in non-linear calculation to define the shape of the structure at the beginning of the analysis. Thus a state of initial imperfection in shape can be easily modelled.

Initial-deformation manager

The initial deformation curves can be defined and edited in the Initial-deformation manager. This manager is one the SCIA Engineer numerous database managers. Its operation and layout are analogous to other database managers.

In the Initial-deformation manager the user may:

- define a new initial deformation curve,
- edit an existing initial deformation curve,
- copy an existing initial deformation curve,
- delete an existing initial deformation curve,
- save the existing initial deformation curve to an external file.

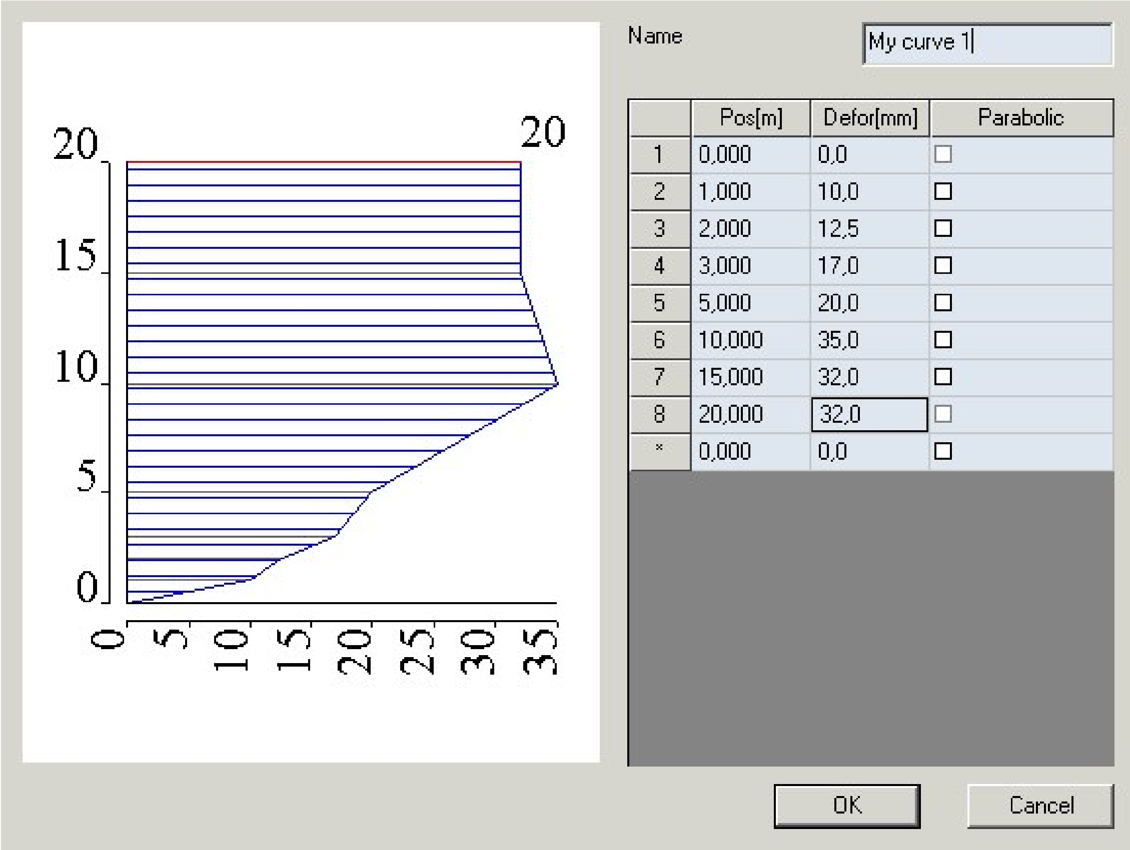
The Initial-deformation manager can be opened in two ways:

- using tree menu function Libraries > Initial deformations,
- using menu function Libraries > Initial deformations.

Initial deformation curve

The initial deformation is defined in the editing dialogue by means of a position-deformation curve.

The curve may be defined in a simple operated dialogue.



The user just has to type pairs of corresponding values for position and deformation. Next to the table the curve is displayed with the position on the vertical axis and deformation on the horizontal axis.

The curve can be then later assigned to required direction in the definition of a non-linear combination.

Defining a new initial deformation curve

The procedure for the definition of a new initial deformation curve

1. Open the Initial-deformation manager.
2. Click button [New] to insert a new curve.
3. A new curve is added to the list of defined curves.
4. Select the new curve.
5. Press button [Edit] to open the editing dialogue.
6. In the editing dialogue, type pairs of corresponding values for the position-deformation curve.
7. Confirm with [OK].
8. Repeat steps 2 to 7 as many times as required.
9. Close the manager.

Applying the initial deformation

The initial deformation curve may be used in a non-linear combination to define the initial imperfect shape of the structure.

The procedure for the application of the initial deformation curve

1. Create a new non-linear combination or edit the existing one.
2. Item Type of imperfection set to an option requiring the input of an initial deformation curve (i.e. either Functions + curvature on beams or Inclination functions).
3. In the appropriate items choose the required initial deformation curve (each direction can use different initial deformation curve).
4. Finish the definition or editing of the non-linear combination.
5. Use the combination for calculation.

Soil-In

Introduction

The analysis of foundation structures is challenged by the problem of modelling of the part of the foundation that is in contact with subsoil. The best solution is to use 2D model of the subsoil that properly represents the deformation properties of the whole under-foundation massif by means of surface model. The properties of such model are expressed by what is called interaction parameters marked C. These parameters are assigned directly to structure elements that are in contact with the subsoil and they influence the stiffness matrix.

To simplify the matter, we may imagine that C is the characteristics of elastic, more precisely pseudo-elastic, links, or surface spring constants that change according to the actual state of the analysed system. We may also use the professional slang that calls it "support on C parameters", which is the generalisation of standard Winkler idea of the supporting in the form of thick liquid g = C1 (MNm-3) or in the form of infinitely dense system of vertical springs. The generalisation is very important and deals mainly with the consideration of significant shear distribution in the subsoil that is neglected by Winkler model. The parameters of the interaction between the foundation and the subsoil depends on the distribution and loading level, or the contact stress between the structure surface and the surrounding subsoil, on the geometry of the footing surface and on mechanical properties of the soil.

Calculation module Soil-in takes account of all the mentioned dependencies.

As the C parameters influence the contact stress and vice versa – the distribution of the contact stress have impact on the settlement of the footing surface and thus the C parameters, it is necessary to use an iterative solution.

The influence of subsoil in the vicinity of the structure

The modelling of the interaction between a structure and subsoil requires that the influence of the subsoil outside of the structure be taken into account. This outside-subsoil supports the edges of the foundation slab due to shear stiffness. In the past, special procedures were recommended to model this phenomenon. The current versions of SCIA Engineer employ a sophisticated solution whose principle is described in the following paragraph.

The program automatically adds to the edge of the analysed foundation slab springs that approximately substitute the effect of what is termed support elements (1 to 2 metre wide strip located along the edges of the foundation slab, the thickness of this strip is almost zero). The solution obtained through this approach takes into account the effect of the subsoil outside (in the vicinity) of the analysed foundation slab.

In comparison with a solution without such springs, the results obtained with the springs gives smaller deformation of the foundation-slab edges which means larger bending moments in the foundation slab.

The springs oriented in the global z-direction are assigned to all edge nodes except the situation when a node already has another spring assigned or if a rotation of a node is specified. In that case, the program assumes that the user has already defined a special type of support and that it is not wanted to alter that special configuration automatically on the background.

These exceptions can be used to deliberately suppress the implementation of edge-springs along certain lines. The user can define very small line springs along required lines (edges) and thus eliminate the effect of the surrounding subsoil (e.g. if a sheet pile wall is installed).

Soil-in output

The results from the soil-in iteration are the C parameters C_{1z}, C_{2x} and C_{2y}.

Parameters C_{1x} and C_{1y} are always defined by the user.

C1z - resistance of environment against wP (mm) [C1z in MN/m3]

C2x - resistance of environment against wP/xP (mm/m) [C2x in MN/m]

C2y - resistance of environment against wP/yP (mm/m) [C2y in MN/m]

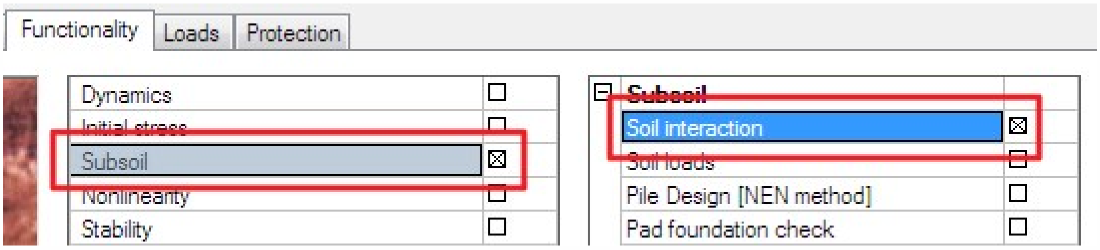
C1x - resistance of environment against uP (mm) [C1x in MN/m3]

C1y - resistance of environment against vP (mm) [C1y in MN/m3]

> Usually, C2x is considered equal to C2y and C1x equal to C1y, because the calculation is done by so called isotropic variant of the calculation of C2 parameter.

The soil-in calculation is available when the specific functionality is active.

Check Subsoil on the left part and the Soil interaction on the right part of the functionality tab:



> The Soil interaction is available only for Plate XY and General XYZ type of project.

C parameters (explanation from theoretical background of FEM solver)

C_1 - Parameters of the interaction of the foundation with the surface 2D model of the subsoil in physical relation containing components of displacement u, v, w.

Winkler formula for vertical components:

$$\sigma_z = r\,[\mathrm{kPa}] \quad = C_{1z}^{S}\,[\mathrm{MNm}^{-3}]\cdot w\,[\mathrm{mm}]$$

Winkler formula for horizontal shear components:

$$\tau_{zx} = s_x\,[\mathrm{kPa}] \quad = C_{1x}^{S}\,[\mathrm{MNm}^{-3}]\cdot u\,[\mathrm{mm}]$$

$$\tau_{zy} = s_y\,[\mathrm{kPa}] \quad = C_{1y}^{S}\,[\mathrm{MNm}^{-3}]\cdot v\,[\mathrm{mm}]$$

C_2- Parameters of interaction of the foundation with the surface 2D model of the subsoil in physical relations containing the first derivative of settlement.

Paternal formula for shear forces:

$$t_x\,[\mathrm{kNm}^{-1}] = C_{2x}^{S}\,[\mathrm{MNm}^{-1}]\cdot \partial w/\partial x\,[\mathrm{mm/m}]$$

$$t_y\,[\mathrm{kNm}^{-1}] = C_{2y}^{S}\,[\mathrm{MNm}^{-1}]\cdot \partial w/\partial y\,[\mathrm{mm/m}]$$

C_{1z} - foundation compression modulus of the Winkler type, expressing resistance to the vertical displacement of the subsoil surface.

C_{2x}, C_{2y},C_{2xy} - foundation shear modulus expressing resistance to the shear components in the x and y direction of the sub-soil surface, generally different in positive and negative shears g_{xy}, g_{yz}(dilatancy and contractancy effects).

Pasternak G_p modulus is our C_2.

Pasternak differential equation:

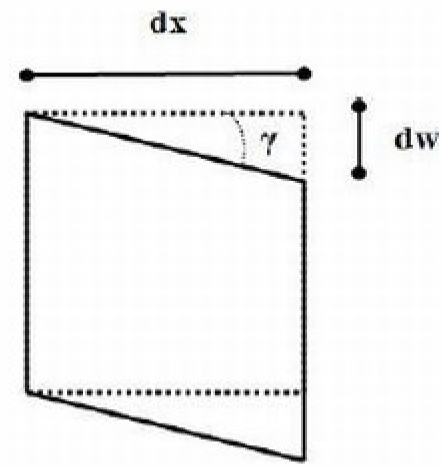
$$p = wk - G_p \frac{d^2w}{dx^2}$$

p - pressure

k - modulus of sub-grade reaction

G_p - shear modulus of the shear layer and it is related to rotation in the differential equation.

Rotation of the surface = bevel dw/dx (see the picture)



w - displacement

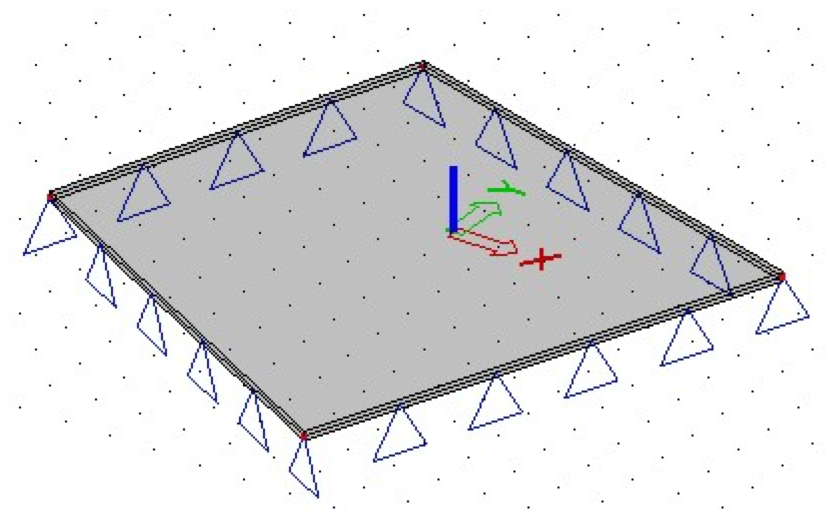
g - shear strain

Support on surface

The interaction between the structure and subsoil is calculated if the structure is put on a support of "Soilin" type.

The procedure to define a new Soilin support

1. Create the structure to be supported.
2. Open service Structure.
3. Start function Support > Surface (el. foundation).
4. Adjust the parameters (see chapter Surface support on slab).
5. Confirm with [OK].
6. Select the slab (groundslab) or slabs that should be supported with this type of support

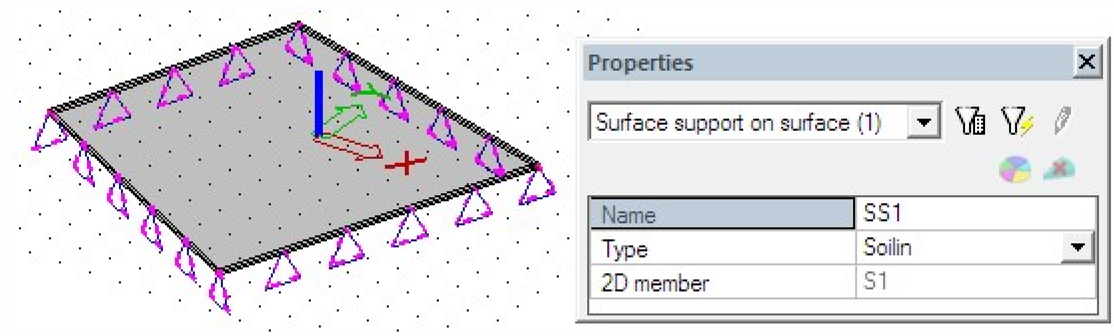


If the ground-slab is not horizontal, one should be aware of the following:

The correct calculation of C parameters assumes that the structure that is in contact with subsoil is more or less horizontal. Technically speaking, the inclination of the footing surface up to 5 to 8 degrees can be allowed. Program is capable of dealing with footing surface in several z-levels, but the results are acceptable only if the z-levels are within certain limits – see the following literature (in Czech):

- Kolář V.: Matematické modelování geomechanických úloh. Skriptum pro postgraduální studium FAST VUT Brno, 1990, 60 str.
- Buček J., Kolář V., Obruča J: Manuál k programu SOILIN, FEM consulting Brno, 1993
- Buček J., Kolář V.: Iterační výpočet NE-XX - SOILIN, FEM consulting Brno, 1995
- Kolář V.: Statické výpočty základových konstrukcí. Knižnice Aktualit České matice technické Praha, ed. plán 1994.
- Kolář V.: Teoretický manuál FEM-Z k programům DEFOR a NE-XX, seminář FEM consulting s.r.o., 5. - 6.10.1993 v Brně.

The surface support properties



Name: Specifies the name of the support.

Type: Defines the type of support – see below.

Subsoil: If necessary for the selected type, this item specifies the subsoil parameters.

Type

Individual:
A particular subsoil type is assigned to the slab.
The subsoil is defined by means of C parameters. These user-defined C parameters are used for the calculation (of e.g contact stress in the footing surface). It is possible to set for parameter C1z a nonlinear function.

Soil-in:
For such a support, the interaction of the structure with the foundation subsoil is carried out by means of SOIL-IN module.
Parameters C1z, C2x, C2y are calculated by SOIL-IN module.

Both:
Both of the above mentioned types are combined on the same slab.
The user defines which C parameters will be user-defined and which ones will be calculated by SOIL-IN module. Parameters can be defined in subsoil properties. Those C parameters that are input in the subsoil-property dialogue as zero, will be calculated by the SOIL-IN module. Nonzero parameters will be taken as they are input.

The additional springs are automatically added on the edges if soilin calculation doesn't recognize additional plates around the support. See chapter Advanced tips.

Subsoil in the 3D model

The subsoil in the 3D window is defined as a soil surface and soil borehole. The geologic profile is defined for each soil borehole. The position and the composition of the geologic profiles provide information about subsoil.

Soil borehole

The borehole is available in the project only when the functionality Soil interaction is checked.

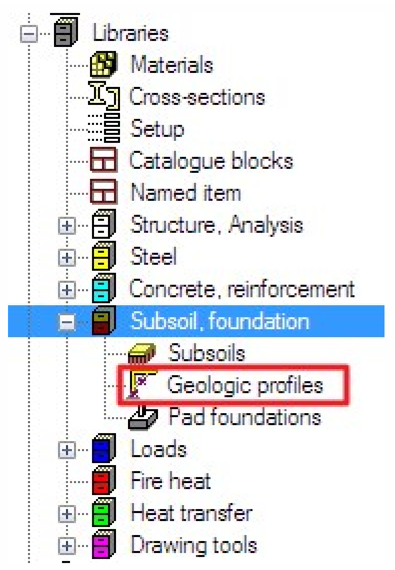
The special property in the inserting dialogue converts standard borehole to the Sand-gravel pile. See more in the separate chapter.

Soil types according to ČSN EN 73 1001

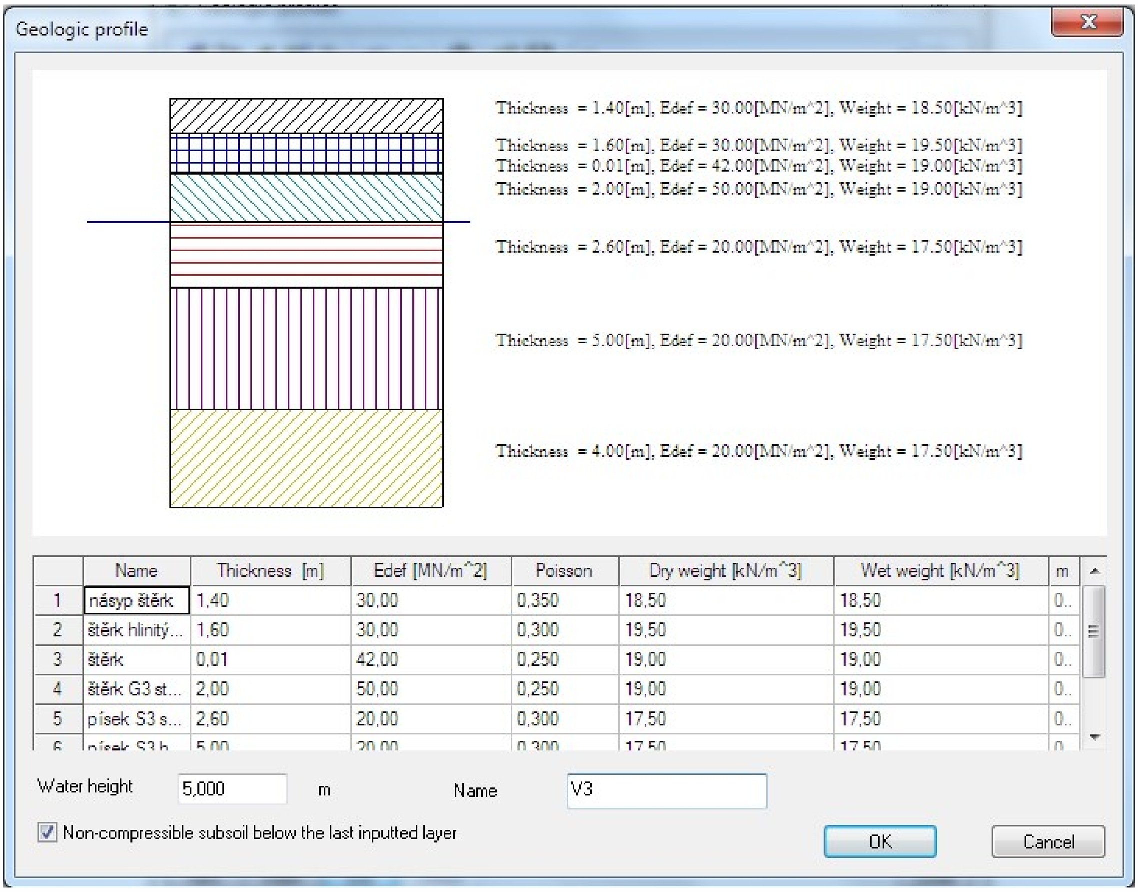
Fine-grained soils		**Sands**		**Gravel**	
F1(MG)	loam, fine-grained	**S1(SW)**	sand, well-grained	**G1(GW)**	gravel well-grained
F2 (CG)	clay, gravelly	**S2(SP)**	sand, poorly-grained	**G2(GP)**	gravel poorly-grained
F3 (MS)	loam, sandy	**S3(S-F)**	sand, with fine-grained soil	**G3(G-F)**	gravel with fine-grained soil
F4 (CS)	clay, sandy	**S4(SM)**	sand, with loam	**G4(GM)**	gravel, with loam
F5 (ML)	loam, small plasticity	**S5(SC)**	sand, with clay	**G5(GS)**	gravel, with clay
F5 (MI)	loam, middle plasticity				
F6 (CL)	clay, small plasticity				
F6 (CI)	clay, middle plasticity				
F7 (MH)	loam, high plasticity				
F7 (MV)	loam, very high plasticity				
F7 (CH)	loam, extremly high plasticity				
F8 (CV)	clay, very high plasticity				
F8 (CE)	clay, extremly high plasticity				

Geologic profile

All profiles are saved to the Geologic profiles library. The geologic profiles can be imported or exported by the DB4 format.



The borehole profile is defined as a simple grid with the preview. Each row represents one layer of soil with the same properties.



Each layer is defined by the soil parameters:

Name:
Specify the name of the layer

Thickness (m):
thickness of the layer

E_{def}:

The module of deformation E_{def} is defined as deformation characteristic of the soil. It is a ratio of the normal stress increment to the increment a linear transformation. For geotechnical categories 1 and 2 the indicative value from e.g. ČSN 73 1001 can be used, for category 3 a survey should be carried out to provide for the value. The value E_m from Eurocode 7 can be used instead of E_{def}.

E_{def} according to ČSN 73 1001:

Class of the subsoil	E_{def} (MPa)
F6-F8 (soft, medium consistency)	1,5-4
F6-F8 (stiff consistency)	6-8
F6-F8 (hard consistency)	10-15
F3-F5 (soft, medium consistency)	3-5
F3-F5 (stiff consistency)	8-10
F3-F4 (hard consistency)	based on survey
F5 (hard consistency)	10-20
F1, F2 (soft, medium consistency)	5-15
F1, F2 (stiff consistency)	12-25
F1, F2 (hard consistency)	based on survey
S4, S5	5-12
S3	12-19
S2	15-35
S1	30-60
G5	40-60
G4	60-80
G3	80-90
G2	100-190
G1	250-390
R6	10-75
R5	20-250
R4	40-750
R3	70-2500
R2	130-7500
R1	250-25000

The E_{def} for R is derived from the number of discontinuous parts in the soil.

Poisson:
Poisson's ratio, coefficient of transverse deformation, an indicative value or experimentally found value can be used, predefined range is 0 – 0.5

Poisson according to ČSN 73 1001:

Class of the subsoil	Poisson v
F8 (soft, medium, stiff consistency)	0,42
F8 (hard consistency)	based on survey
F5-F7 (soft, medium, stiff consistency)	0,40
F5-F7 (hard consistency)	based on survey

Class of the subsoil	Poisson v
F1-F4 (soft, medium, stiff consistency)	0,35
F1-F4 (hard consistency)	based on survey
S5	0,35
S4, S3	0,30
S1, S2	0,28
G4, G5	0,30
G3	0,25
G1, G2	0,20
R6	0,40-0,25
R4, R5	0,30-0,20
R3	0,25-0,15
R1, R2	0,20-0,10

Dry weight:
weight for the dry soil, normally within the range from 18 to 23 kN/m^3, range is 0 – 10000000000 kN/m^3

Wet weight:
weight for the wet (saturated) soil, this value is mostly about 2-3 kN/m^3 higher than the dry weight, range of the values is 10 – 10000000 kN/m^3

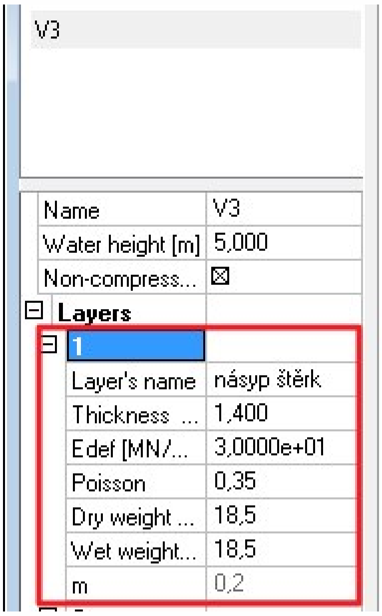
m coefficient:
structural strength coefficient, according to the Eurocode7 is 0,2 (ČSN 73 1001 defines a table). The m coefficient may be modified for the whole Eurocode.

Coefficient m according to ČSN 73 1001:

Class of the subsoil	m
F1-F8 with E_{def}<4MPa, not over consolidated and soft or solid consistency R1, R2 and R4, R5 not affected by erosion	0,1
F1-F8 which don't belong to the first group S1, S2, G1, G2 under the water level R3	0,2
S1, S2, G1, G2 above the water level S3-S5 G3-G5 R4, R5 which don't belong to the first group	0,3
R6	0,4
Loess, loess loam	0,5

Geologic profile must be defined up to such a depth where the bearing pressure is still active, otherwise the program does not have sufficient information.

The defined parameters are displayed in the library as properties.



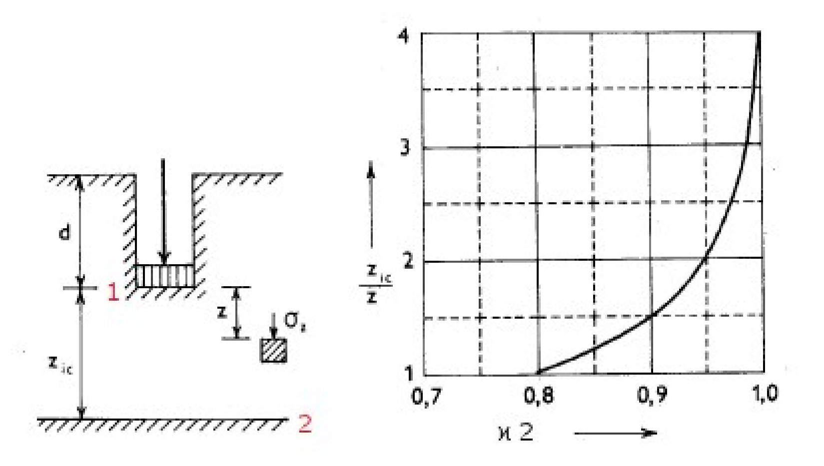
The height of the underground water is defined by the value in the properties. It is a positive value but it represents the depth.

Non-compressible subsoil below the last inputted layer

The checkbox "Non-compressible …" can be used if the soil below the last layer is non-compressible. The system applies coefficient of depth reduction $\varkappa_2$ in this case (calculation of $\varkappa_2$ can be found in ČSN 73 1001, art. 80). This option is recommended when the non-compressible layer is placed in a small depth under the borehole.

Calculation of $\varkappa_2$ according to ČSN 73 1001:

$\varkappa_2$=1-exp((z_{ic}/z) ln0,25 + ln0,8)



1 – foundation base

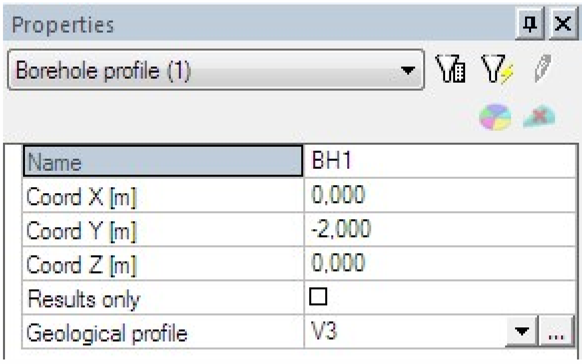
2 – non-compressible layer

z_{ic} – is the depth under the foundation base to the non-compressible layer

z – is the depth from the foundation base to the level where the contact stress σ_z should be calculated

The contact stress σ_z will be calculated by the reduced depth $z_{r2}=\varkappa_2$*z where z is the depth under the foundation base.

Properties of the borehole profile

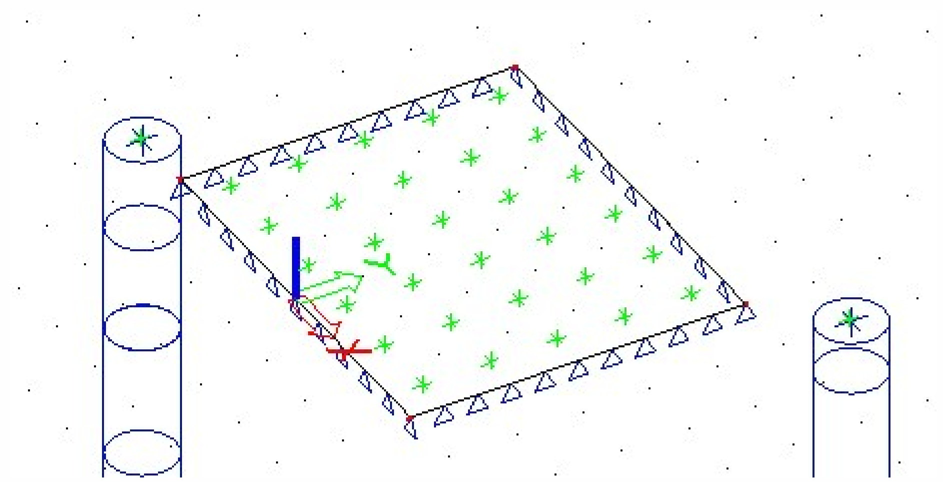


The borehole is defined by the geologic profile and the inserting point in the 3D window. The properties contain only name, its coordinates, the borehole profile and the checkbox Results only.

Settlement input data

Settlement is calculated for each mesh element (in its center of gravity) and for each borehole inserting point. The checkbox Results only exclude a borehole inserting point from the input data. It means that the point is used for the calculation of settlement but the geologic profile is not taken into account for the layers approximation.

The nodes for the settlement calculation (green vertexes):

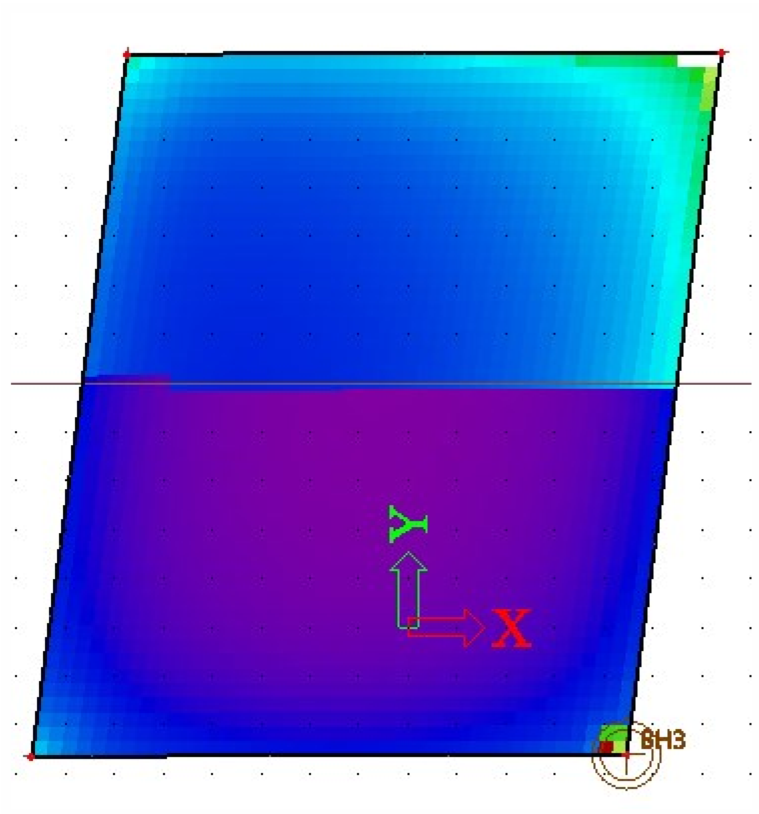


Geological areas

The main geological surface = area is calculated and displayed automatically. It is possible to create more areas in the main one.

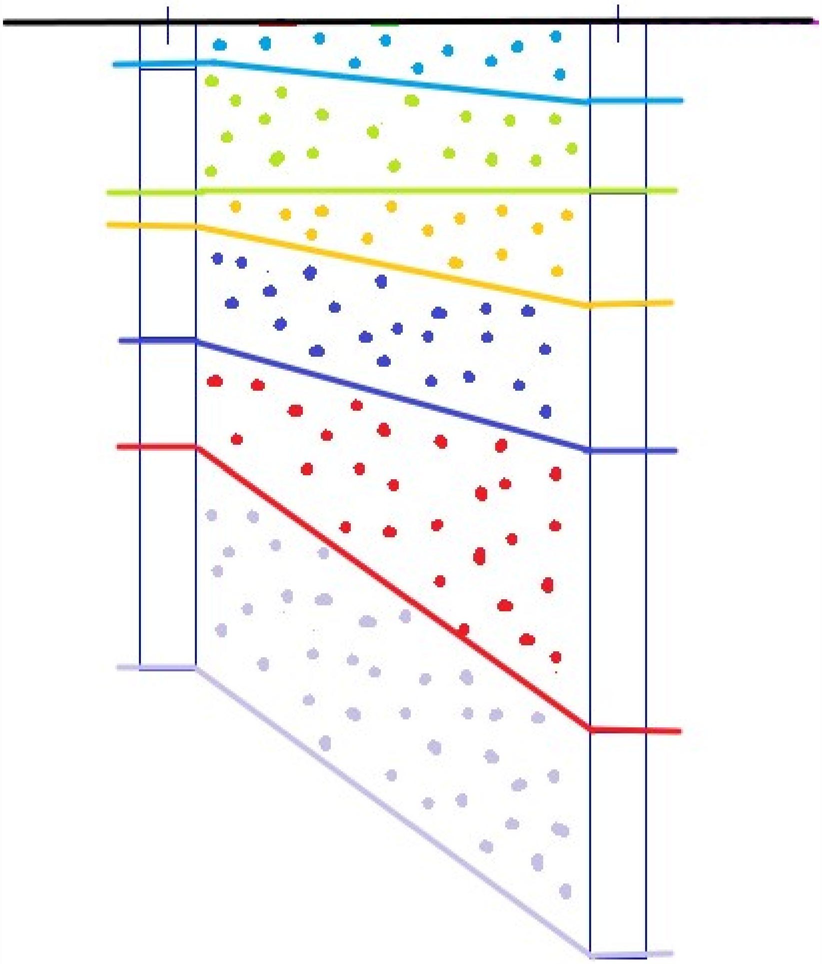
Layers approximation is calculated inside one geological area, independently on neighbouring areas. There is a geological fault on the border between two geological areas.

The Sand-gravel pile is a geological profile which is placed in one geological area.



Layers approximation

When more borehole profiles are used in the project then it must fulfil one important condition – the same number of layers. This is required because of the soil-in approximation.

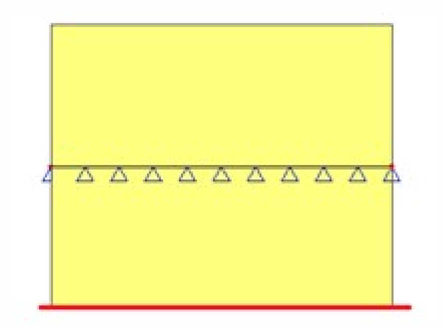


If there is some layer missing in one borehole, then it can be substituted by layer with minimum thickness – e.g. 1mm.o the soil-in has appropriate number of layers for approximation.

Foundation base

The level of the foundation base is considered on the bottom surface of the plate. The eccentricities are also taken into account.

Even the extrem example as this one place the foundation base to the bottom surface.



The red line indicates the foundation base.

Soil surface

Soil surface is a tool for initial approximation of subsoil surface and layers between boreholes.

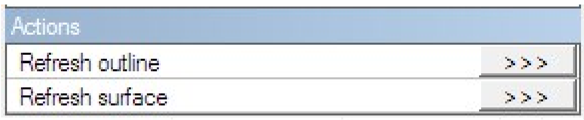
Surface is calculated automatically according to inserted structure and inserted boreholes.

If it is deleted then it is automatically regenerated before the calculation starts.

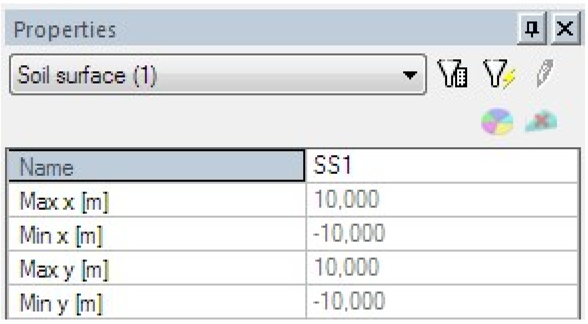
Surface has its borders at least 10m outside the structure.

The surface is editable by 2 action buttons:

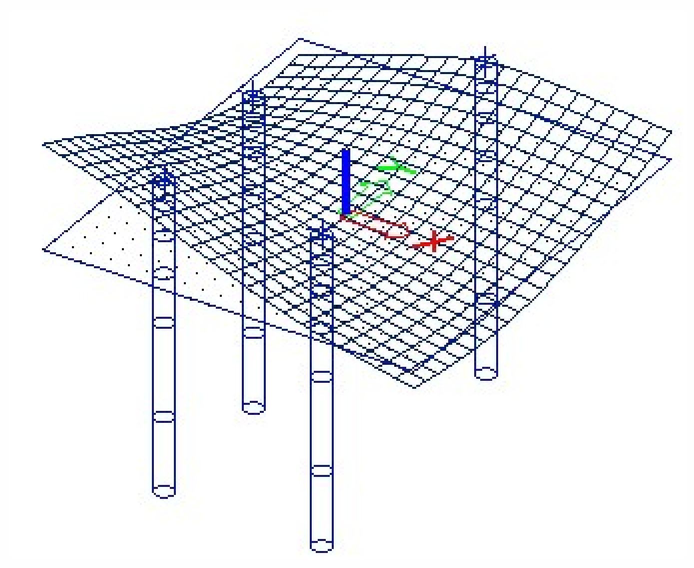
- Refresh outline: it recalculates the border
- Refresh surface: it recalculate the mesh of the surface



The properties of the surface are simple. Just a name and sizes:

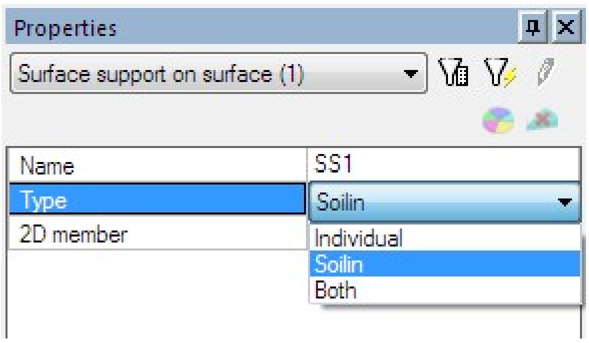


It is possible to display deformed subsoil surface. It is created by several boreholes with different Z coordinates. The mesh is used only for displaying of terrain, it is not used for the calculation.



Surface support

The surface support is basic structure object for soil-in. The support type is defined by combobox with 3 items.



Individual:
the C parameters are defined by user in the Subsoil manually (all of them). They are used for calculation. (e.g. contact stresses of the foundation surface)

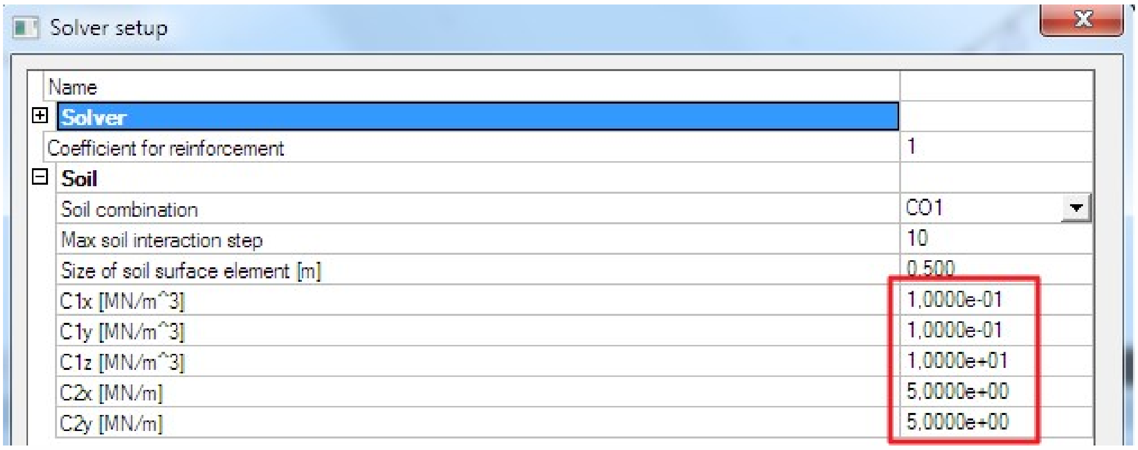
Soilin:
system calculates C parameters (C_{1z}, C_{2x}, C_{2y}) – this type is required for the complete soil-in calculation, C_{1x} and C_{1y} are taken from the solver setup

Both:
system calculates C_{1z}, C_{2x} and C_{2y} if they are set to zero in the Subsoil; the rest is taken from the Subsoil. This item is used rarely only for a very special cases.

Soil-in type

The only type which doesn't use the data from the Subsoil library. All initial values of C parameters are defined by the Solver setup. C_{1x} and C_{1y} are taken from this setup as the results and the rest is calculated by the Soil-in.

The initial values could influence a bit the calculation convergence but their major importance is for setting of non-compressible stiffnesses. These values are 100 times higher than the initial values. That's why a reduction of initial values (e.g. 10 times) can help in a convergence problems (higher depth, small loading, etc.)



Individual type

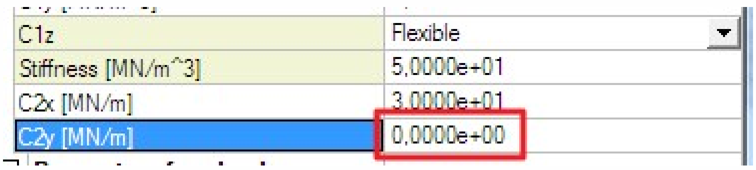
C_{1z}, C_{2x}, C_{2y} parameters are taken from the Subsoil library. It is predefined by the user. The calculation of Soil-in won't start in this case.

Both type

Soil-in calculates C_{1z}, C_{2x} and C_{2y} only when they are set to the zero value by the user.

The parameters with any other value are taken from the library.

Example with type Both:



In this case the C_{2y} parameter is calculated by soil-in. This item could be used only in a case when the soil-in would calculate any extreme values of C_2 parameters. It is a very sporadic case.

The type Both is not too common and it was introduced mainly for two reasons:

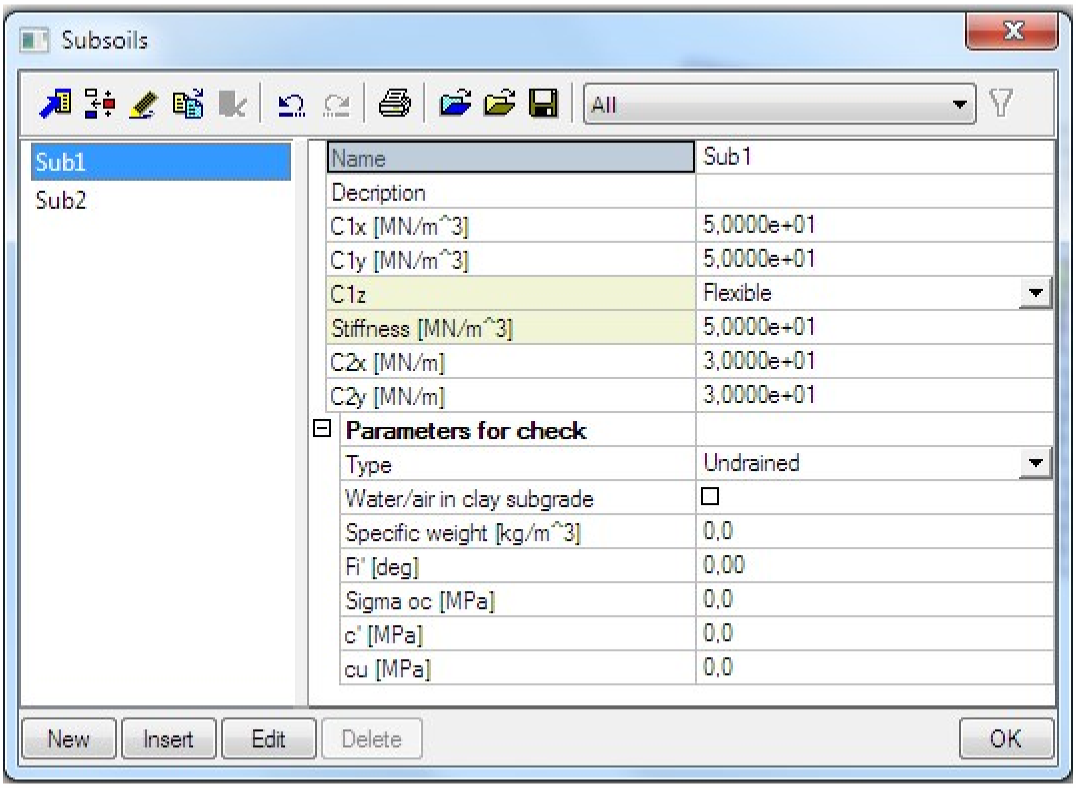
First, I use type Soil-in but I want to have different friction in different parts of the structure. Therefore, the solver setup dialogue is not enough for me, because is just one value can be adjusted there for the friction. Therefore, I can use type Both and thus I am able to define several subsoils with non-zero constants C_{1x} and C_{1y} with all other parameters adjusted to zero. When the Soil-in module runs, the non-zero constants C_{1x} and C_{1y} are of higher priority than those determined by the solver and are applied. Other "zero" values indicate that the values determined by the solver are applied.

Second, sometimes it may be necessary to "suppress" higher values of shear (C_{2x}, C_{2y}) calculated by Soil-in module. This may happen e.g. when a new plate is modelled on an old one and the old plate is defined as the first layer of the subsoil. It is a correct and proper solution, but as E modules of soil and concrete are dramatically different, the Soil-in module calculates high C_2 parameters. Consequently, the stiffness of the foundation slab in the model is bigger than if the two slabs were "joined" together and input as a homogenous monolith. Therefore, C_2 parameters may be reduced artificially. This can be achieved in type Both. I define the subsoil with zero C_{1z} (it will be determined by the Soil-in module) and other non-zero parameters (C_2 and friction). Thus the Soil-in module will provide only for C_{1z} parameter.

Subsoil library

The subsoil contains parameters which can be defined by the user or calculated by soil-in.

Parameters C_{1x} and C_{1y} are always defined by the user.

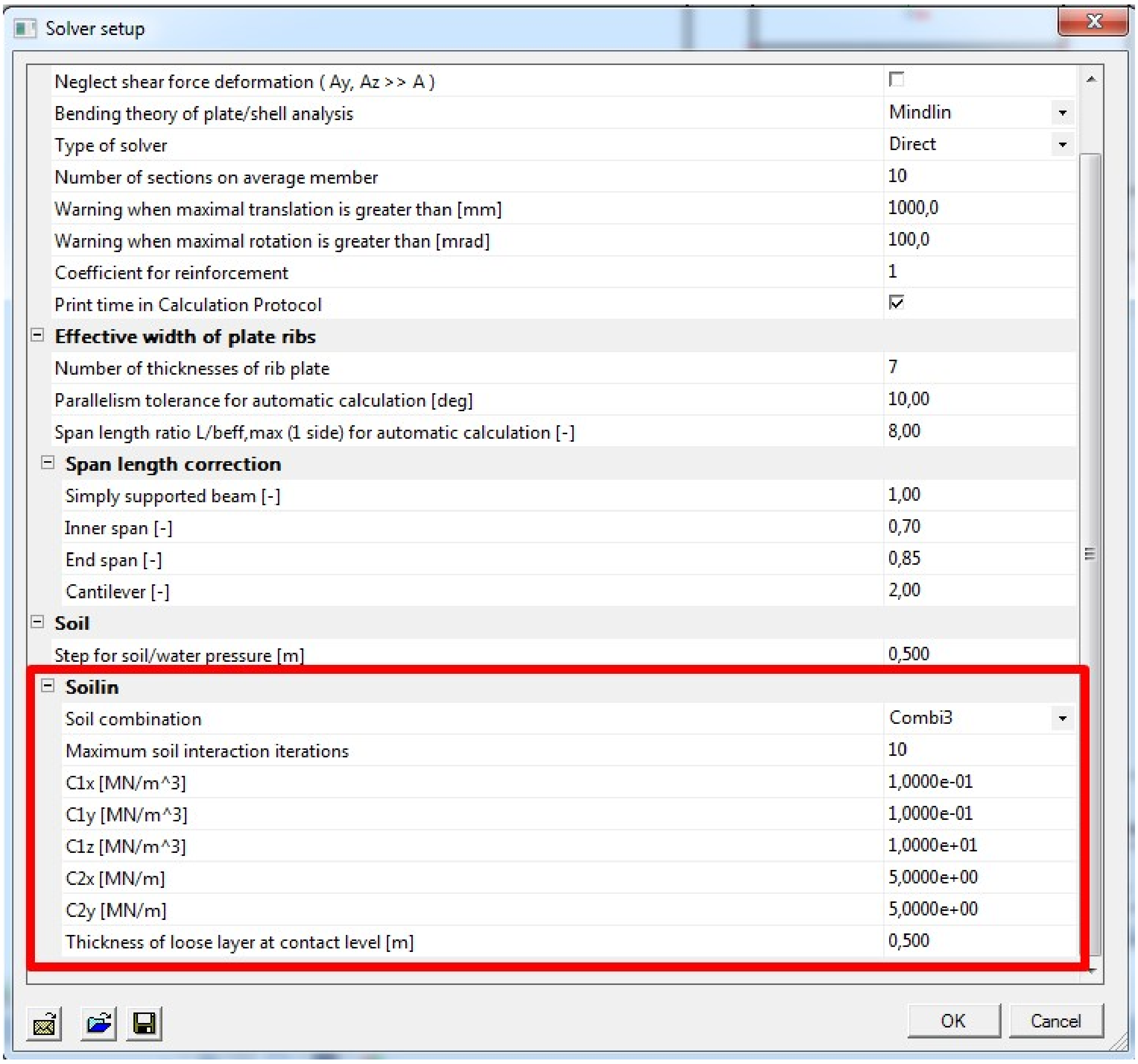


Required parameters for Soil-in calculation

What all must be defined:

- Project with at least one borehole with predefined geologic profile
- Structure with surface support type Soilin or Both
- Load
- Combination type Linear (ULS or SLS)

Soil-in settings in the Solver setup



Soil combination:
linear combination which is used for the soil-in calculation.
Even though it is not an exact solution, for practical reasons the C parameters are not calculated separately for each load case or each load case combination. The user must specify one particular reference combination that is used to calculate the C parameters. The calculated C parameters are then applied in all remaining defined load cases and combinations.

Maximum soil interaction iterationsstep:
number of iteration cycles (when the program stops iterations if there are still no proper C parameter calculated, in case that results diverge), the max. limit is 99 steps

Size of soil surface element:
define size of element of surface mesh. It is used for displaying of terrain.

C_{1x}:
the parameter defined by the user

C_{1y}:
the parameter defined by the user

C_{1z}:
initial value for soil-in (if the support type is Soilin)

C_{2x}:
initial value for soil-in (if the support type is Soilin)

C_{2y}:

initial value for soil-in (if the support type is Soilin)

Thickness of loose layer at contact level [m]:
setting in solver settings for automatic definition of loose layer at contact stress level for better handling of foundation at great depth and faster convergence of soilin iterations. This layer is automatically created and managed by soilin to handle the fact, that a thin layer of soil is always somewhat damaged and loose, just at the contact level. This item is set 0 as default. User can choose this layer 0 - 10m.

The source of not calculated parameters depends on the support type. It is described in the previous chapter.

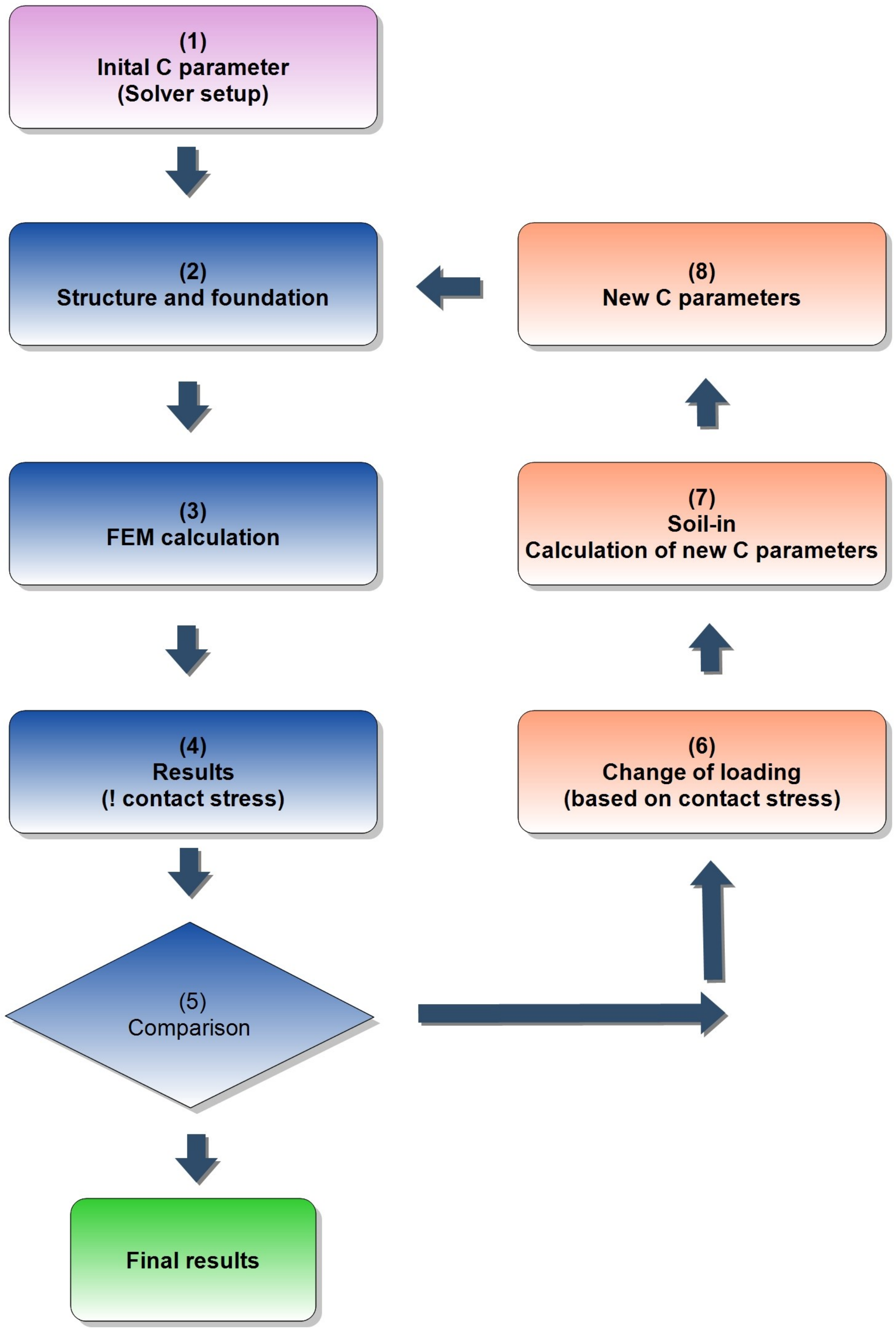
Soil-in calculation

Soil-in iterative cycle

The values from the top structure and the foundation are calculated by FEM. The values are used as the source data for the soil-in.

The iterative process is finished when the contact stress σ_z and displacement u_z does not change significantly in the two subsequent iterations. The special quadratic norms are evaluated in the each iteration cycle to find out if this condition is fulfilled.

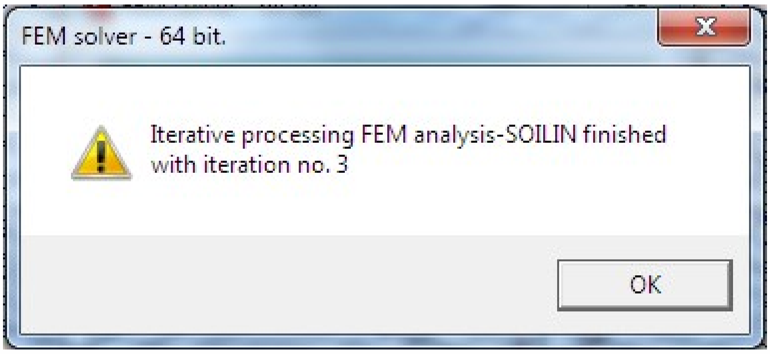
Diagram of the iterative cycle:



1. The values are taken from the solver setup, predefined by the user.
2. Data from the structure and its foundation.
3. FEM calculation – important results for soil-in contact stress σ_z and displacement u_z.
4. The results of i iteration.

5. Comparison of the in contact stress σ_z and u_z – it is based on the quadratic norms, when it does not change significantly, then the calculation is done and SCIA Engineer displays results.
6. 1st step of soil-in – the contact stress is recalculated to the new loading.
7. 2nd step of soil-in – the C parameters are recalculated, new loading is taken from the previous step.
8. 3rd step of soil-in – final C parameters from soil-in - the new input data.
9. New C parameters are used for the next FEM calculation.

There is a message when the last iteration is done.



Quadratic norm to compare the results from the last and the previous iteration

The calculation of the settlement of the subsoil and subsequent determination of the C parameters is performed in a standard way using an iterative process. The result of this process is the state in which the contact stress or displacement u_z in two subsequent iterations does not change significantly. For that reason, the following quadratic norms are evaluated in every j-th iteration:

$$\varepsilon_\sigma = \frac{\sum_{i=1}^{n}\left(\sigma_{z,i,j} - \sigma_{z,i,j-1}\right)^2 A_i}{\sum_{i=1}^{n}\left|\sigma_{z,i,j} \cdot \sigma_{z,i,j-1}\right| A_i}$$

$$\varepsilon_u = \frac{\sum_{i=1}^{n}\left(u_{z,i,j} - u_{z,i,j-1}\right)^2 A_i}{\sum_{i=1}^{n}\left|u_{z,i,j} \cdot u_{z,i,j-1}\right| A_i}$$

Where:

n number of nodes

$\sigma_{z,i}$ contact stress in node i

A_i area corresponding to node i

$u_{z,i}$ global displacement of node i in the z-direction

The iterative calculation is stopped if ε_σ<0,001 or ε_u<0,001

Theory about the derivation process

In this text we limit ourselves to a brief derivation for the purpose of the explanation that will follow:

1. The formula for the potential energy of internal forces of the 3D model has the following form:

$$\Pi^i_{3D} = \frac{1}{2}\int_V \underline{\boldsymbol{\sigma}}^T \underline{\boldsymbol{\varepsilon}}\, dV = \frac{1}{2}\int_V \underline{\boldsymbol{\varepsilon}}^T \underline{\mathbf{D}\boldsymbol{\varepsilon}}\, dV$$

2. Neglecting the effect of horizontal components of deformation, we get the following vectors:

$$\boldsymbol{\sigma} = \left[\sigma_z, \tau_{zx}, \tau_{yz}\right]^T = \mathbf{D}\boldsymbol{\varepsilon}$$

$$\boldsymbol{\varepsilon} = \left[\varepsilon_z, \gamma_{zx}, \gamma_{yz}\right]^T = \left[\frac{\partial w}{\partial z}, \frac{\partial w}{\partial x}, \frac{\partial w}{\partial y}\right]^T$$

3. This means the corresponding simplification of the matrix of physical constants D.

$$\mathbf{D} = \begin{bmatrix} E_z & 0 & 0 \\ 0 & G & 0 \\ 0 & 0 & G \end{bmatrix}$$

4. In order to be able to reduce the problem from 3D to 2D, it is necessary to integrate formula 1) over the z-axis. For this reason, a certain "damping function" f_z is introduced and it is defined by the ratio of the settlement in the given depth to the settlement of the surface $w_0(x,y)$.

$$f(z) = \frac{w(x,y,z)}{w_0(x,y)}$$

5. Modifying formulas from step 2) we get:

$$\boldsymbol{\varepsilon} = \left[w_0(x,y)\frac{\partial f(z)}{\partial z}, \frac{\partial w_0(x,y)}{\partial x} f(z), \frac{\partial w_0(x,y)}{\partial y} f(z)\right]^T$$

6. Substituting formula from step 5) into the formula for the potential energy of body V=ΩH, where Ω is the extent of the 2D model and H is the depth of the deformed zone of the 3D model, we obtain the following formula:

$$\Pi_{2D}^{i} = \Pi_{3D}^{i} = \frac{1}{2}\int_V \left[\sigma_z\varepsilon_z + \tau_{zx}\gamma_{zx} + \tau_{yz}\gamma_{yz}\right] dV =$$

$$= \frac{1}{2}\int_V \left[\varepsilon_z^2 E_z + (\gamma_{zx}^2 + \gamma_{yz}^2) G\right] dV =$$

$$= \frac{1}{2}\int_\Omega \left[w_0^2 \int_0^H E_z \left(\frac{\partial f}{\partial z}\right)^2 dz + \left(\frac{\partial w_0}{\partial x}\right)^2 \int_0^H f^2 G dz + \left(\frac{\partial w_0}{\partial y}\right)^2 \int_0^H f^2 G dz \right] d\Omega$$

7. Integrating over z , we get the formula for the potential energy of internal forces of the 2D model with two parameters C_1^S and C_2^S:

$$\Pi_{2D}^{i} = \frac{1}{2}\iint_\Omega \left[C_{1z}^S w_0^2(x,y) + C_{2x}^S \left(\frac{\partial w_0(x,y)}{\partial x}\right)^2 + C_{2y}^S \left(\frac{\partial w_0(x,y)}{\partial y}\right)^2 \right] d\Omega$$

8. Comparing formulas from step 6) and 7), we can define the relation between the parameters of the general (3D) and surface (2D) model:

$$C_{1z}^S = \int_0^H E_z \left(\frac{\partial f(z)}{\partial z}\right)^2 dz \qquad C_{2x}^S = C_{2y}^S = \int_0^H G f^2(z) dz$$

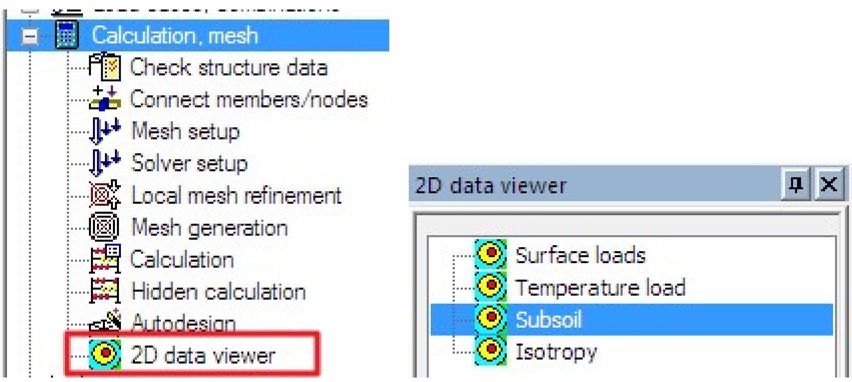
Conclusion:

It is also possible to eliminate the automatic calculation of some C parameters and define them manually. This can be achieved by special adjustment of the subsoil parameters and set the type to Both (!).

The results of soil-in

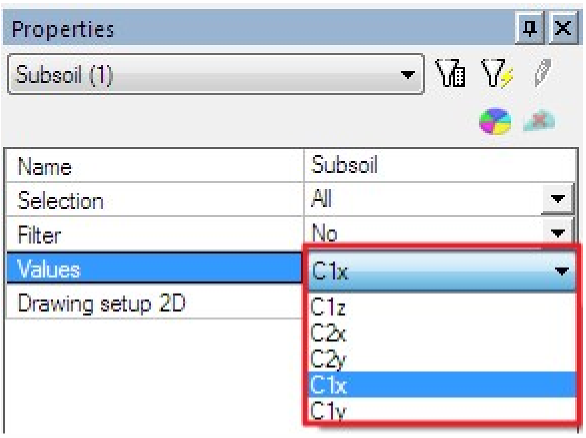
2D data viewer

The soil-in results are available in two different services. In the “Calculation, mesh” service is 2D data viewer. There are results for Subsoil.

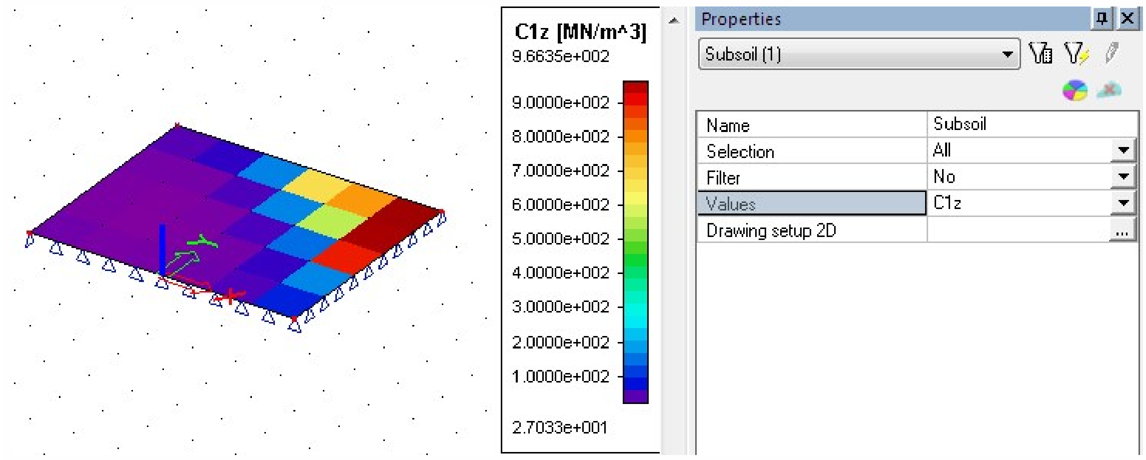


The C parameters are calculated for the mesh on the 2D member. It is displayed by the colour planes.

The results can be displayed for each of C parameters.



The example of calculated C1z:

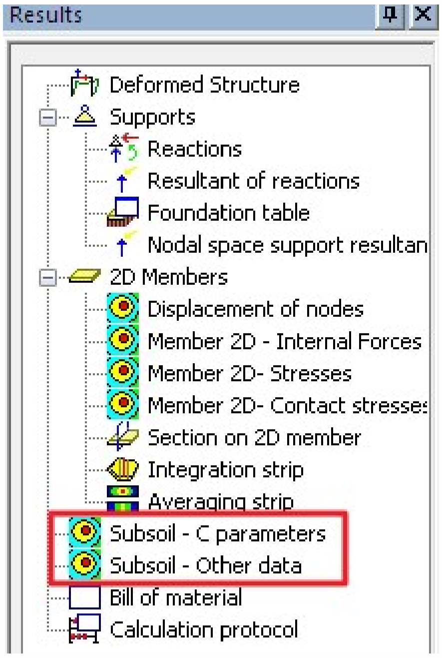


The preview with C parameters in the table can be also displayed in the 2D data viewer.

Results

The service results contain two result previews:

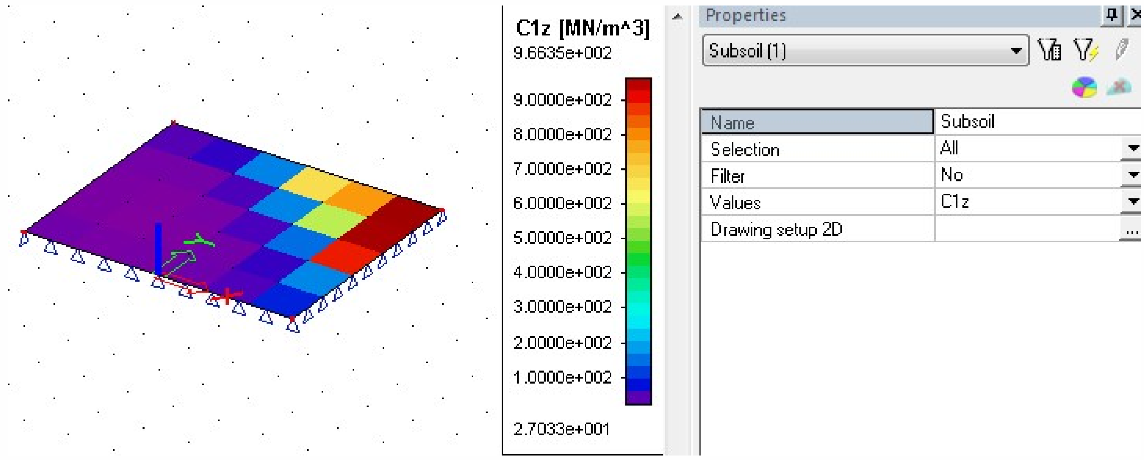
- Subsoil – C parameters
- Subsoil – Other data – this displays settlement (table and diagram for each node)

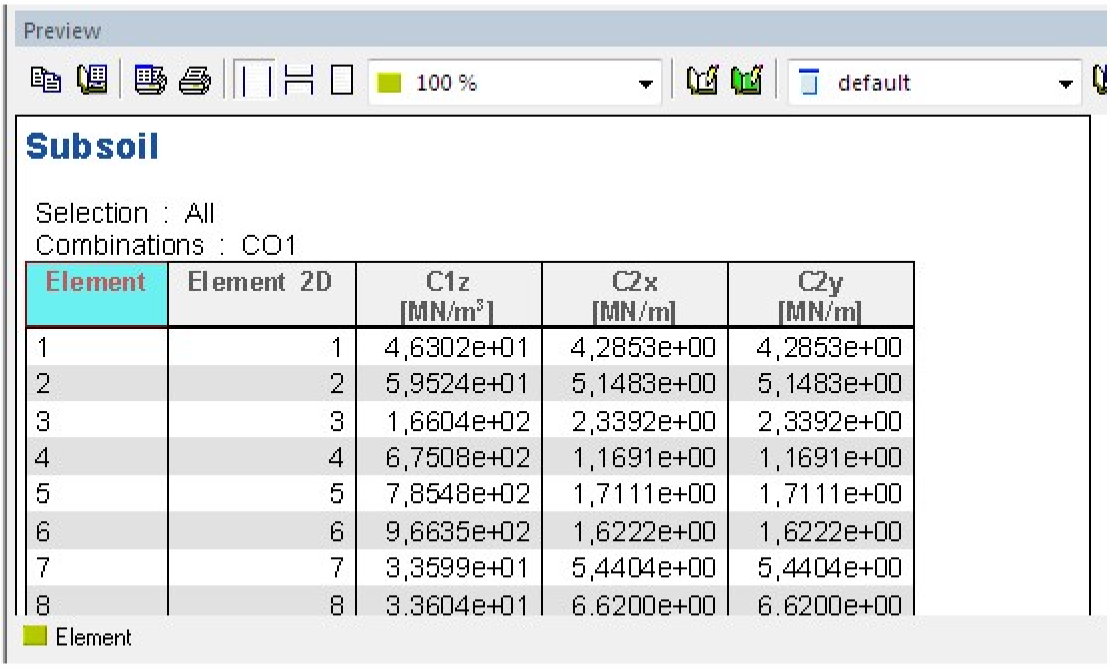


C parameter results

When the Soilin type of the support is used then the preview Subsoil – C parameters displays the same results as 2D data viewer.

When the Both type of the support is used then the preview Subsoil – C parameters displays results of the soilin calculation and the 2D data viewer display data from the Subsoil library.

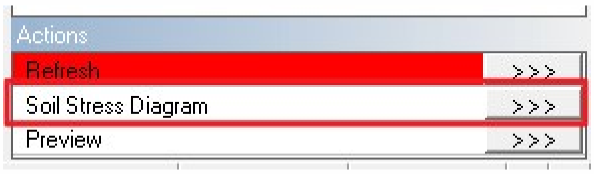


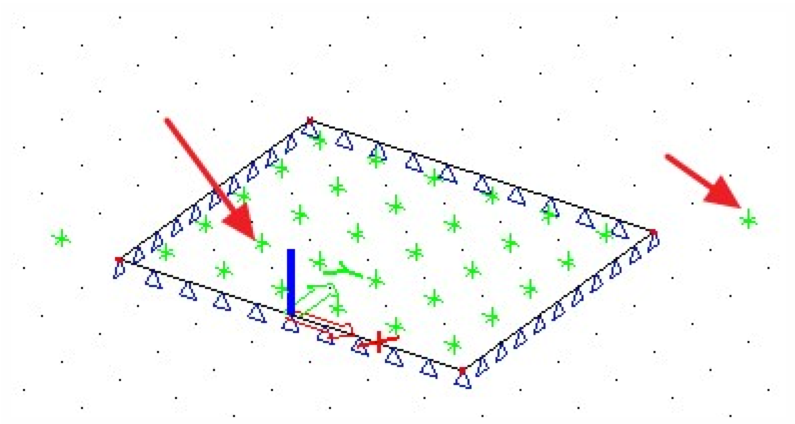


Element	Element 2D	C1z [MN/m³]	C2x [MN/m]	C2y [MN/m]
1	1	4,6302e+01	4,2853e+00	4,2853e+00
2	2	5,9524e+01	5,1483e+00	5,1483e+00
3	3	1,6604e+02	2,3392e+00	2,3392e+00
4	4	6,7508e+02	1,1691e+00	1,1691e+00
5	5	7,8548e+02	1,7111e+00	1,7111e+00
6	6	9,6635e+02	1,6222e+00	1,6222e+00
7	7	3,3599e+01	5,4404e+00	5,4404e+00
8	8	3,3604e+01	6,6200e+00	6,6200e+00

Soil stress diagram

The “Subsoil – Other data” allow to display Soil structure strength diagram for calculated points. The points are displayed by the action button “Soil stress diagram”.



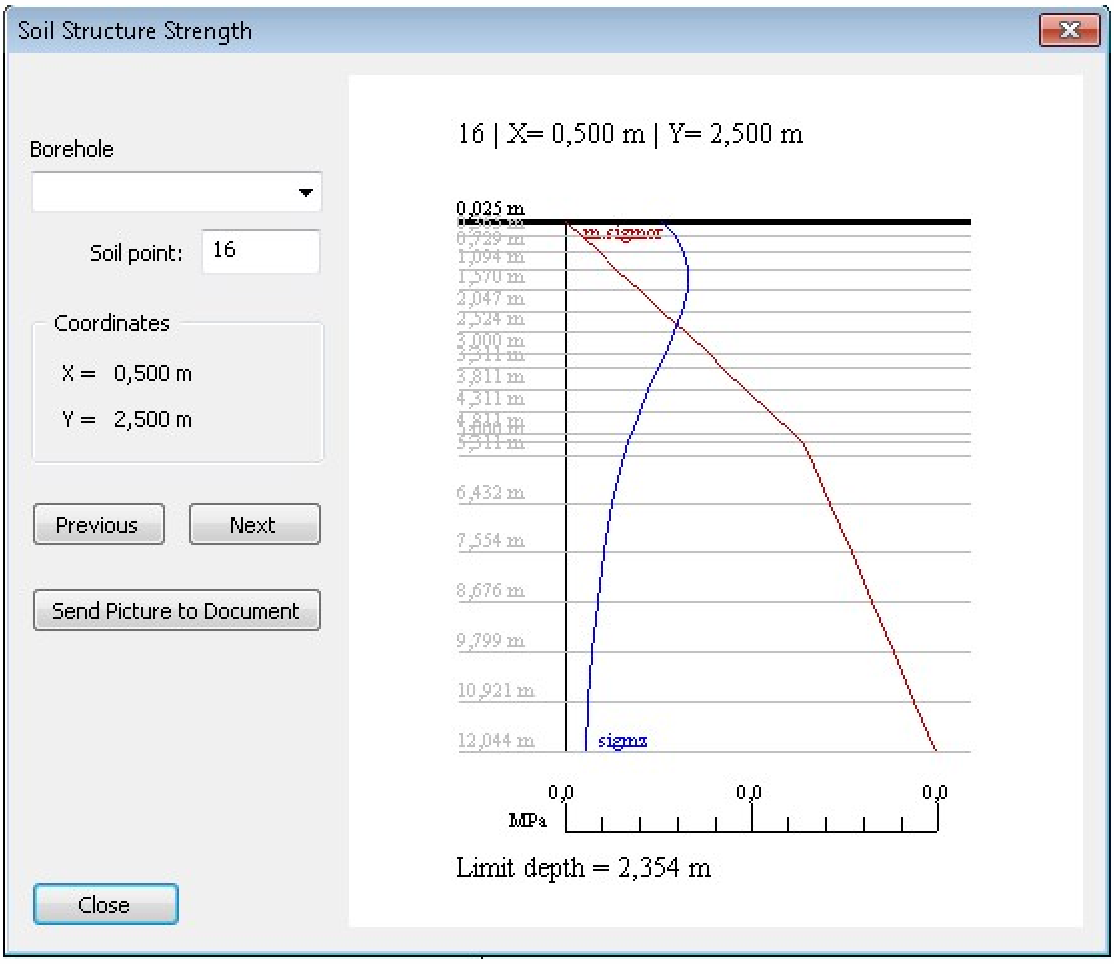


Green vertexes on the plate are centres of elements from 2D mesh. Two green vertexes outside the plate are inserting points from boreholes.

Points are displayed as a green vertex. The vertical axial components of stress and the structure strength (consequently the depth of the deformed subsoil zone) can be displayed for all points from the 2D mesh and for the inserting points of the bore-holes. User just selects the point and the diagram is displayed.

If the borehole is defined as “Results only”, then the point is available for displaying the diagram.

Example of the dialogue Soil Structure Strength:



Previous:
Display the Soil Structure Strength for the previous node

Next:
Display the Soil Structure Strength for the next node

Borehole:
Display the Soil Structure Strength for the selected borehole inserting point

Soil point:
Node number

m*Sigma,or:
The original soil stress

Sigma,z:
The overstress

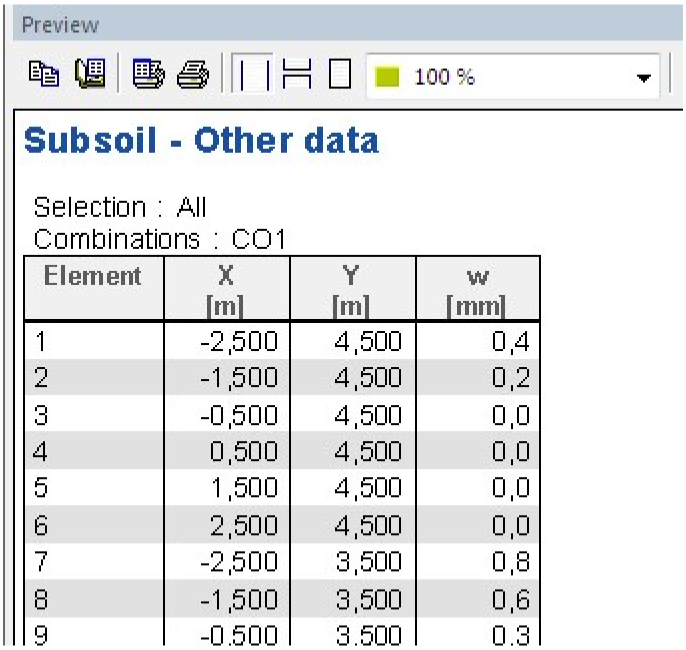
See more about Soil Structure Strength diagram here.

There are two lines - m*Sigma,or and Sigma,z. According to theory, settlement will occur if Sigma,z > m * Sigma,or.

Settlement table

The table is displayed in the Subsoil – other data results. The preview table contain values w for each node.

The settlement w is different from displacement u_z of the foundation plate because w is calculated without stiffness of structure and from the penultimate iteration. Therefore it is useful to watch values w only outside the foundation.



Preview

100 %

Subsoil - Other data

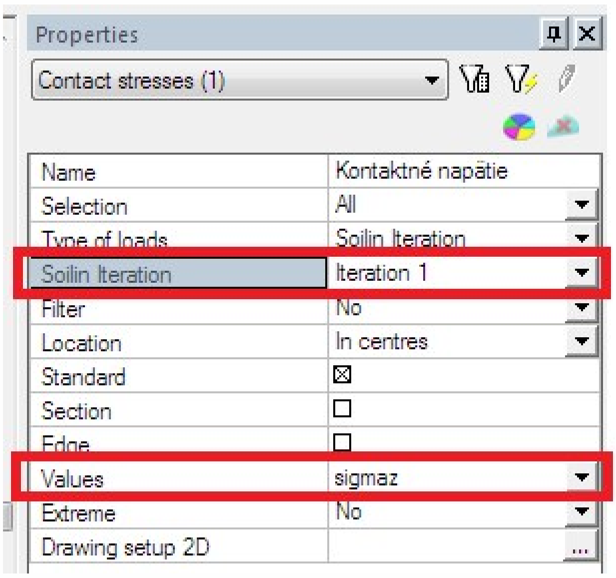
Selection : All
Combinations : CO1

Element	X [m]	Y [m]	w [mm]
1	-2,500	4,500	0,4
2	-1,500	4,500	0,2
3	-0,500	4,500	0,0
4	0,500	4,500	0,0
5	1,500	4,500	0,0
6	2,500	4,500	0,0
7	-2,500	3,500	0,8
8	-1,500	3,500	0,6
9	-0,500	3,500	0,3

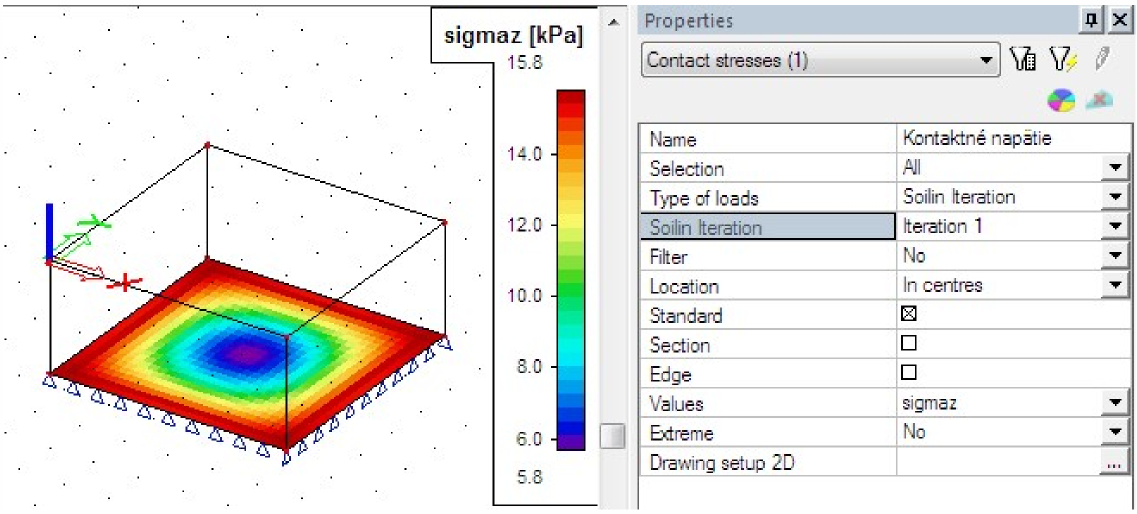
Results for each iteration cycle

When the soil-in won't finish its iteration process in a standard way, the calculation ends after the predefined number of cycles (the solver setup). User can display the contact stresses on the plate for each cycle separately so he is able to find the problem.

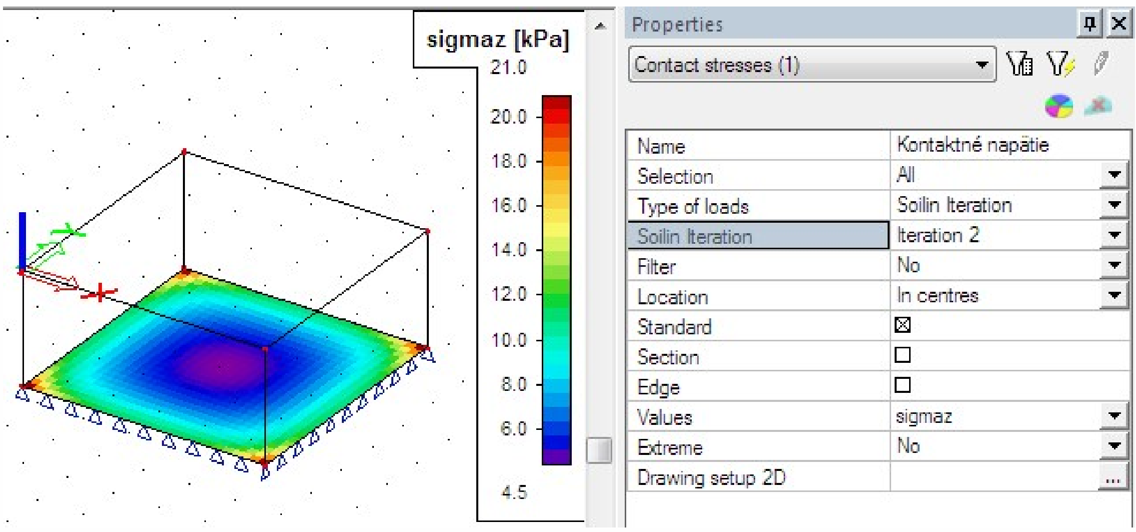
The calculated contact stresses for each iteration cycle can be found in the results.



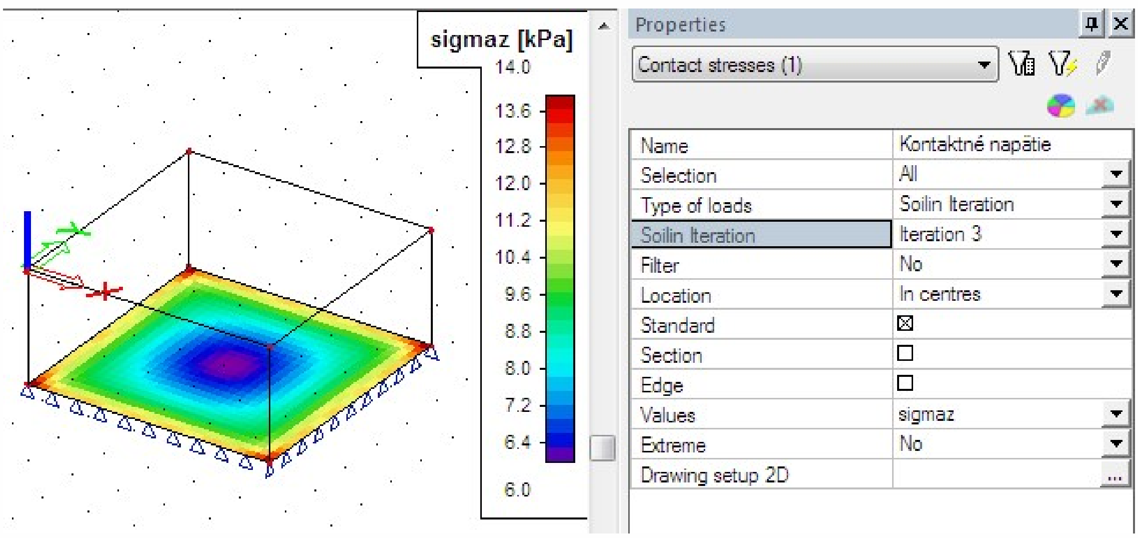
The first iteration cycle



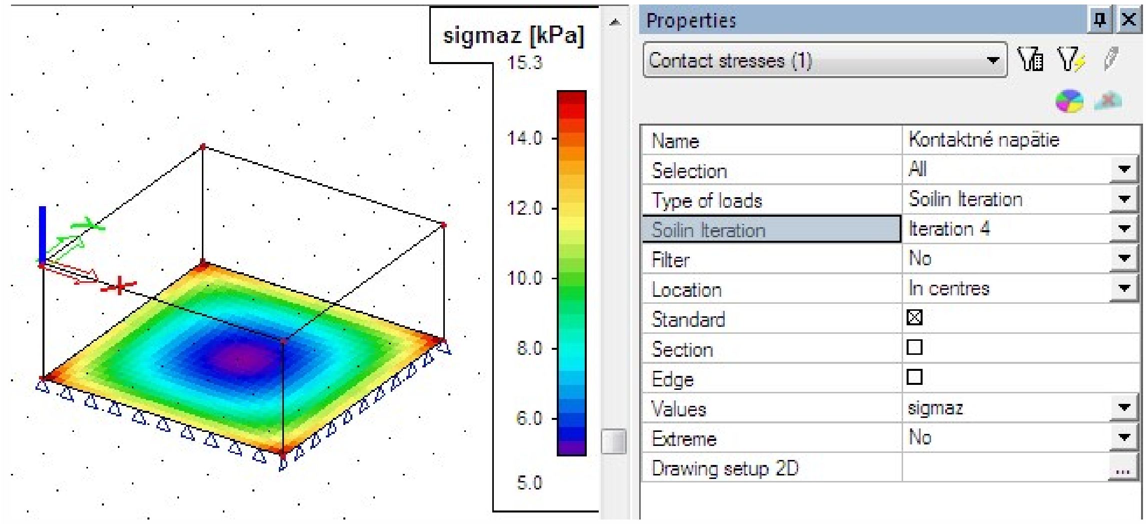
The second iteration cycle



The third iteration cycle



The fourth iteration cycle

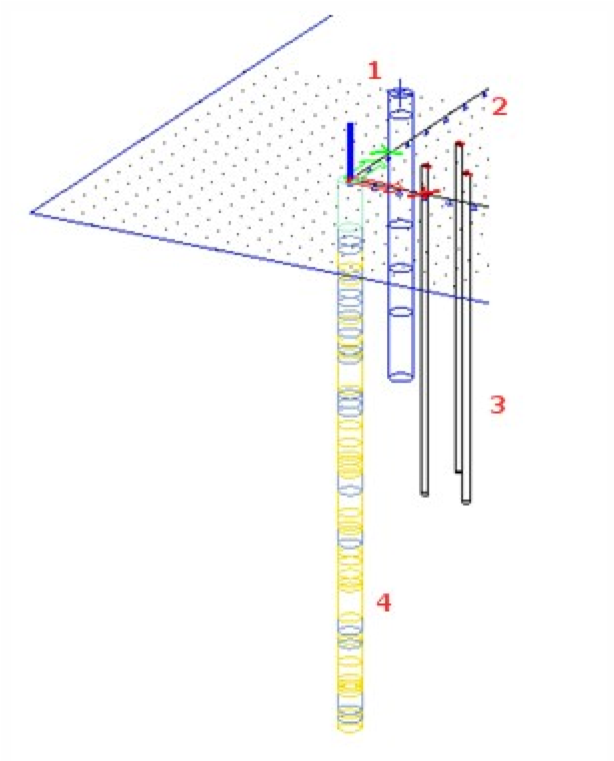


The additional springs are automatically added on the edges if soilin calculation doesn't recognize additional plates around the support. See chapter Advanced tips.

Soil-in and Pile design

Soil-in is a tool for calculation stiffness of the subsoil half-space. The pile is a type of support. Soil-in and piles can be used in one project and system will calculate it together.

Soil-in and Piles are using two different types of boreholes. Piles are based on the CPT profiles; soil-in boreholes are user-defined by layers. Both types of boreholes must be inserted in the project if the user wants to calculate soil-in and pile design.



1. Borehole profile for soil-in
2. Surface support for soil-in
3. Piles for Pile design
4. CPT profile for Pile design

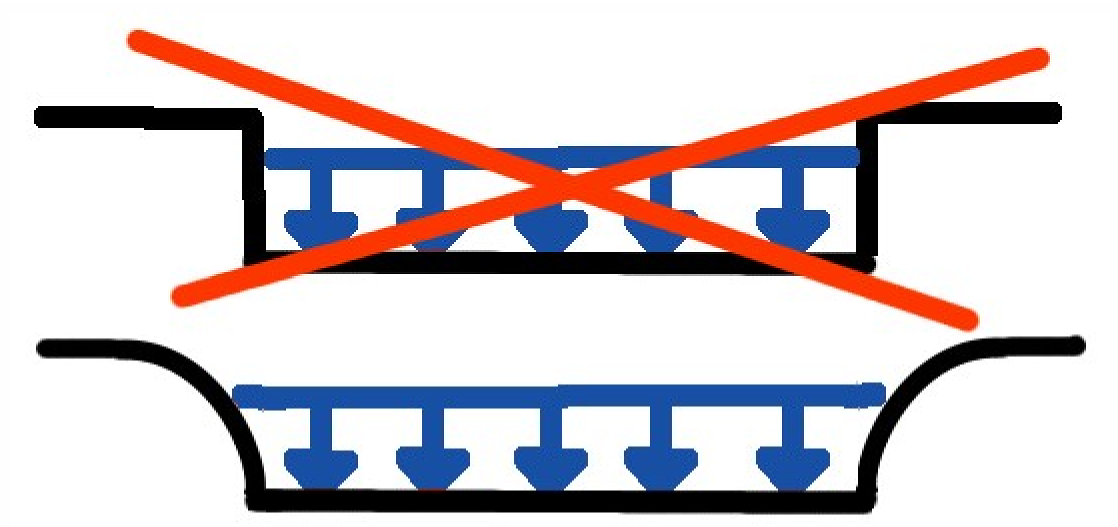
Advanced tips

Foundation at great depth

For better convergence of soilin iterations for foundation at great depth user can set at solver setup in group Soilin - Thickness of loose layer at contact level [m]. The recommended value is 0,500 m. More info in Required parameters for Soil-in calculation.

The effect of the subsoil outside the structure

The nearest subsoil around the loaded structure is also affected by its settlement. The better realistic picture how it works in the reality is displayed below.

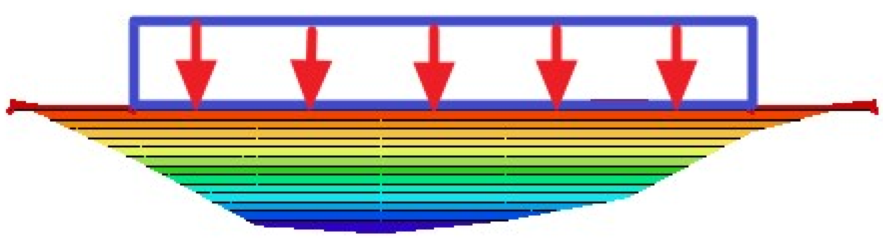


Calculation of the nearest surrounding of the structure is a specific use case. It is recommended to add one more plate to the structure for this purpose – additional subsoil element.

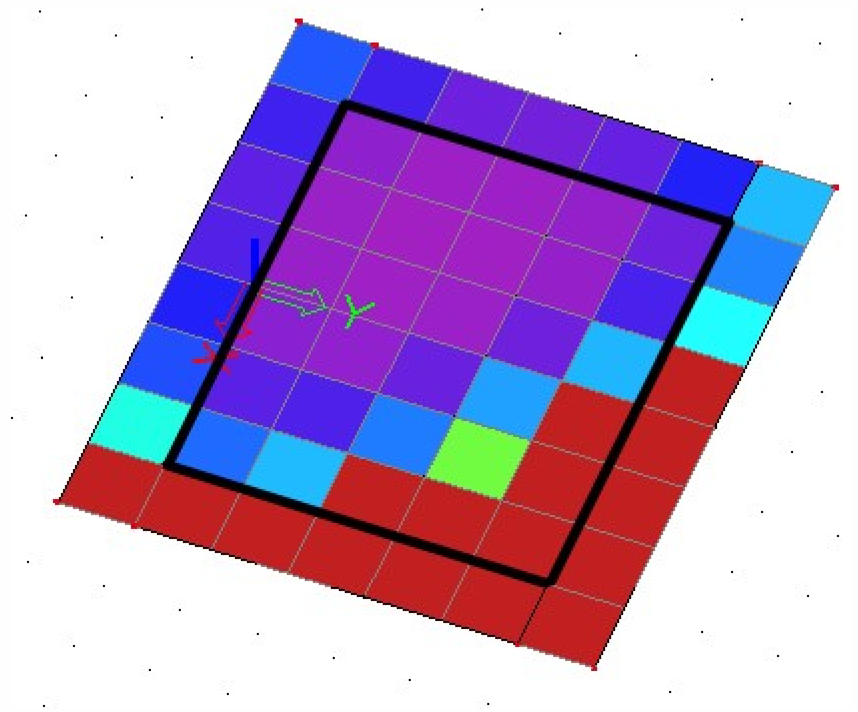
The new plate should be inserted with the minimum thickness (e.g. 0,01mm) and placed next to the foundation.

The C parameters for this affected subsoil around the structure are calculated this way also.

The deformed subsoil calculated by the SCIA Engineer:



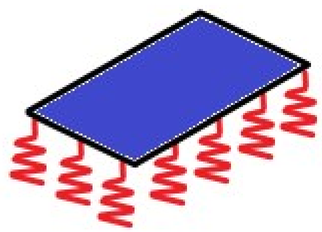
Calculated C parameters:



The structure is marked by the black rectangle and around this is one more plate - surrounding plate – with thickness 0,001mm.

Automatic calculation of the edge supports

When user don't use any subsoil elements then the program will eliminate the neglect of the subsoil on edges by an automatic inserting of vertical supports on the foundation edges.



The calculation of those supports is based on already known C parameters. The program try to support the plate in the same way as it should be supported by the subsoil itself. This leads to approximate model where the sum of reaction is contact stress with reactions in those nodes.

This solution can be sometimes undesirable – e.g. if there is a second foundation near by the calculated one or there is some other support under or near the foundation edge.

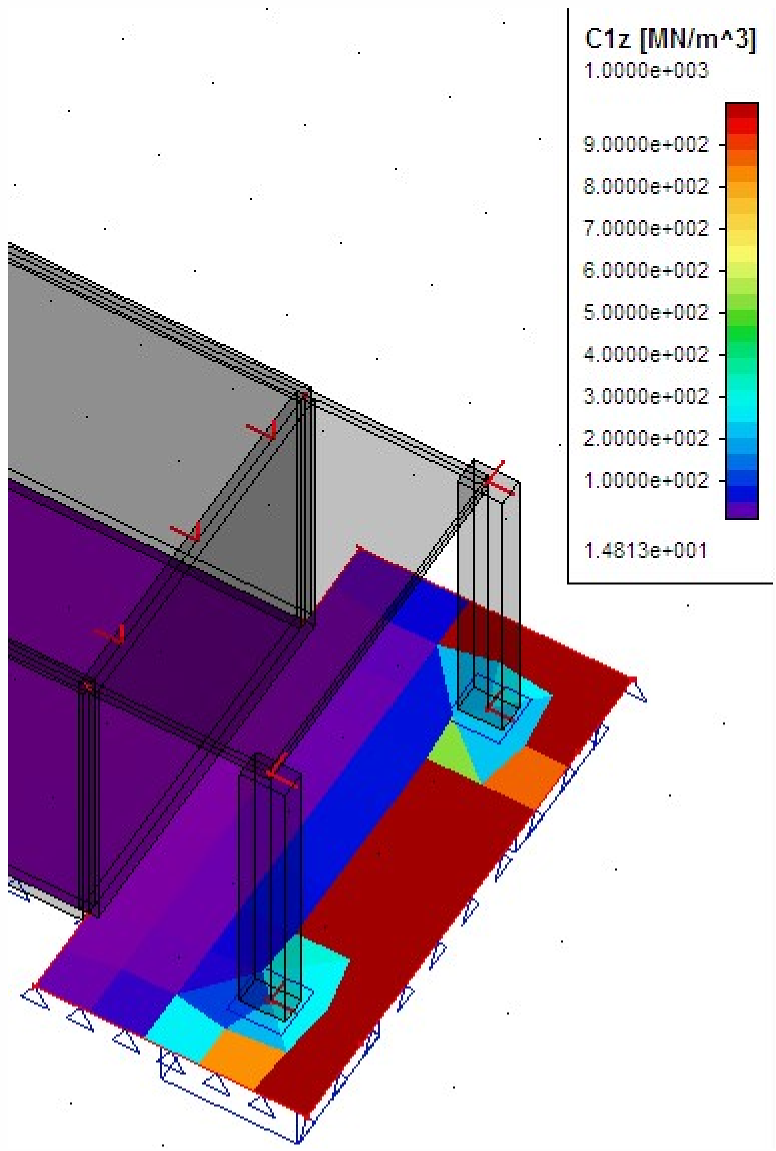
This automatic input can be avoided manually. User can insert a spring with a small stiffness on the plate edges and then the system won't use automatic input of vertical supports. This could be the additional subsoil elements.

Pad foundation and soil-in

The pad foundation is not connected with the soil-in calculation.

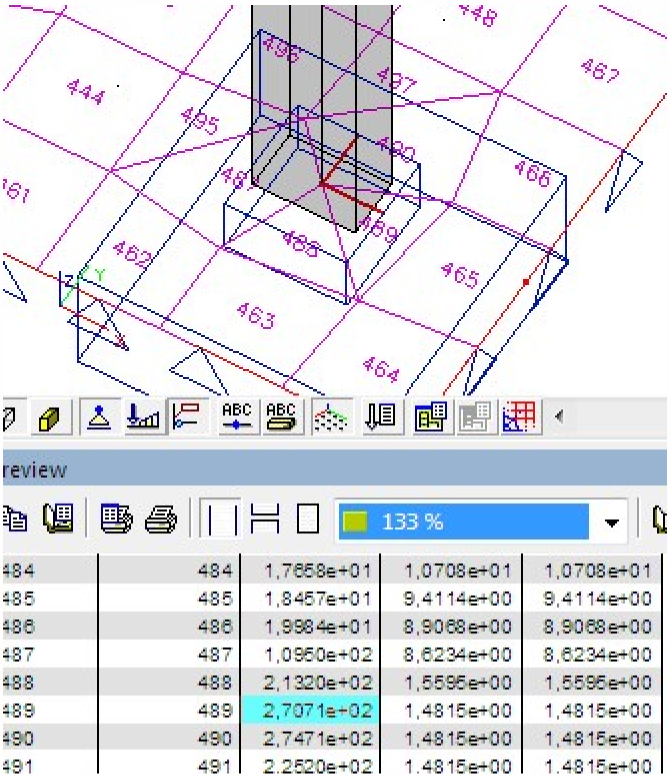
How to use soil-in for the pad foundation check:

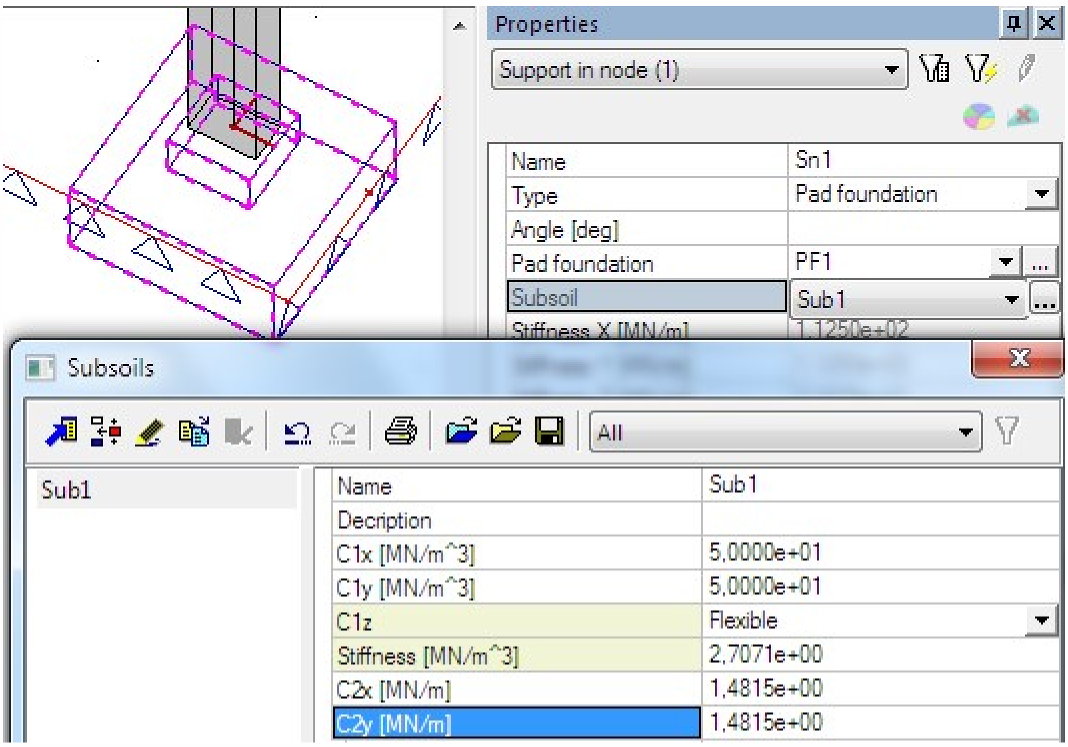
1. Create additional structure to calculate the C parameters in the nearest surrounding (it is described in the previous tip)



Calculated C parameters on the surrounding plate –> C parameters for the pad foundation

2. Calculated C parameters can be used in the Subsoil library. Put the values from the table to the Subsoil library.





3. Run the linear calculation again.
4. Check the pad foundation in a standard way.

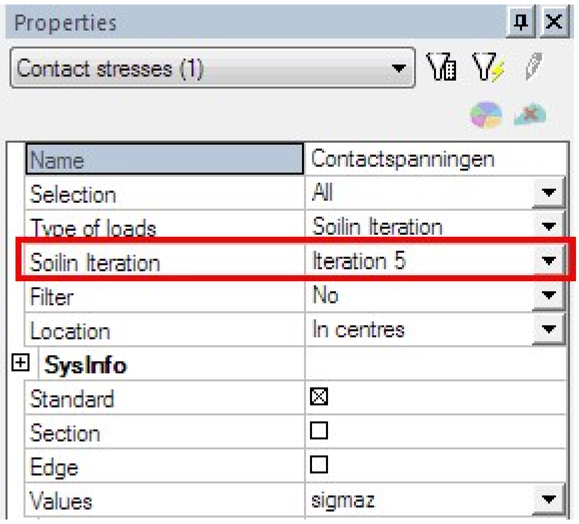
What if the model is correct but the iteration is not finished

Sometimes the model is correct but some circumstances may cause unfinished iterative process. The results in cycles don't lead to one set of C parameters but on the contrary, the results are more and more different.

This can be caused by some tensions in the foundation plate, specific foundation members and similar problems.

How to solve those problems:

1. It is necessary to check the model. It must be correct – the mesh elements are not triangular, the element´s Z axis is upward, the foundation plate must be under the soil surface and so on.
2. Check the iteration cycles in results – contact stresses, type of loads – soilin iteration.
   First few iteration cycles will be probably quite OK and after some time the results become messy.
   Find one cycle (between those correct ones) where the results seem to be close to the reality – e.g. 5thcycle. Use this value in the solver setup for number of iteration cycles.

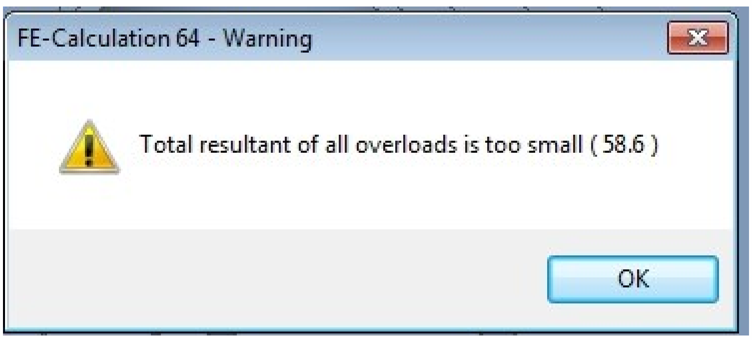


3. Start the linear calculation again, it will be finished after the 5th iteration cycle with results most closest to the reality. The correct cycle is between 2nd and 5th cycle in the most cases.

What if the load is wrongly inserted?

When the plate is not in compression, then soilin cannot be calculated properly.

There could be a message about wrong total resultant:



This may happen when loads are from the bottom to the top, or when there is some change in local LCS of the plate.

What if the symmetrical structure gives non-symmetrical results?

This may happen when additional subsoil elements are not added around the structure.

Also when the soilin didn't find the correct result and calculation is stopped too soon. (For example when solver setup defines only few soilin cycles.)

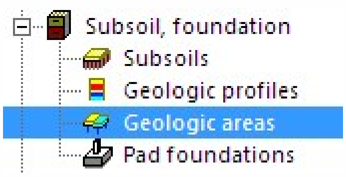
What if geological fault in the subsoil is needed?

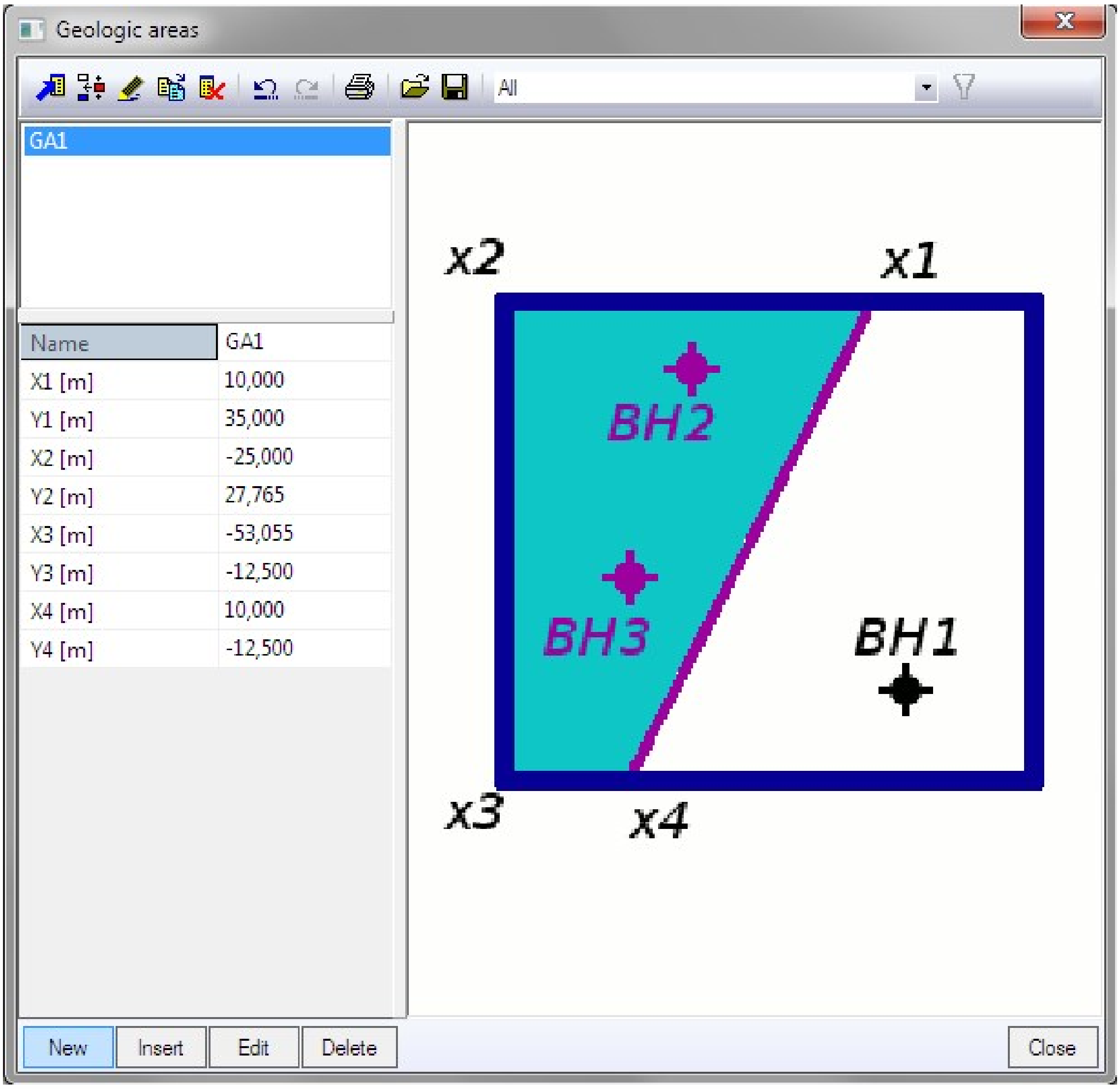
The main subsoil surface is created automatically when a structure and a borehole is inserted to the project. The size of the subsoil outline is calculated from the structure size, positions of boreholes and some offset around.

The geological fault is created when the second area is inserted to the main one.

1. Create structure and insert boreholes.
2. The main subsoil surface is created.

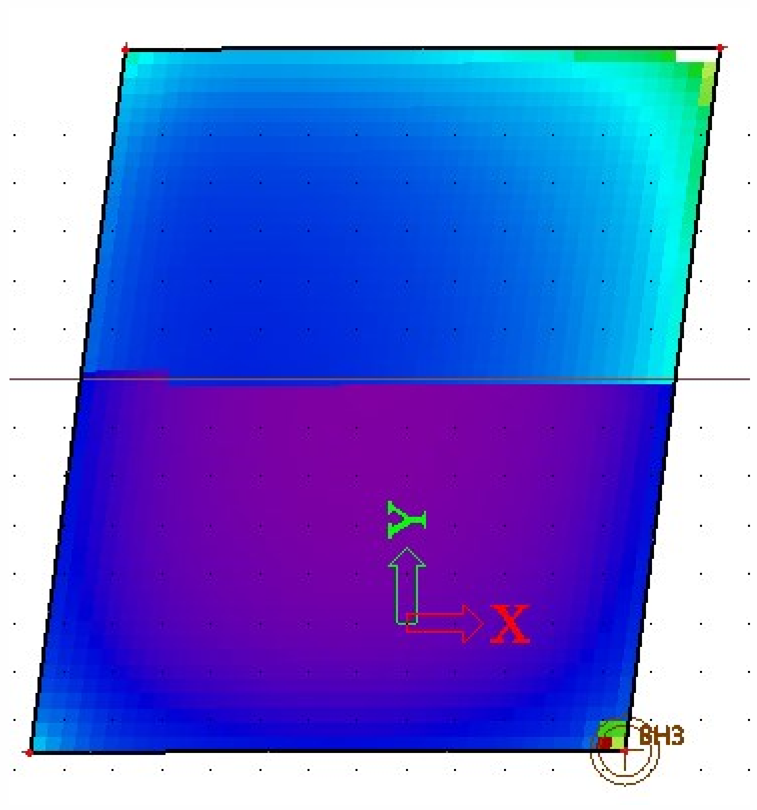
3. Go to Libraries / Subsoil, Foundation / Geological areas, create a new area by its coordinates.





4. Run standard Soilin calculation.

5. Check results where geological fault is visible as a border between two gradients.

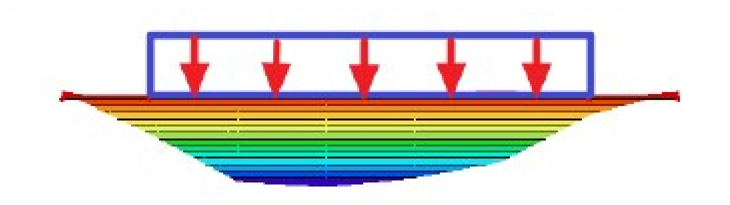


How to use additional plates

Soilin is a tool which calculates C parameters of the subsoil under the surface support. Using the additional plates around the support provides more realistic results.

About C parameters:

1. C parameters are parameters of interaction, so their value depends on the structure, load, stiffness and subsoil. Change in any of those parts causes different C parameters.
2. The whole plate is supported vertically by the soil stiffness – parameter C_1 (winkler) and also in the shear direction – parameter C_2 (pasternak).
3. The plate edges are more supported by the C_2 parameters because it is affected by neglecting.
4. The area around the support is affected by the shear stiffness of the soil and the degrease basin is created.

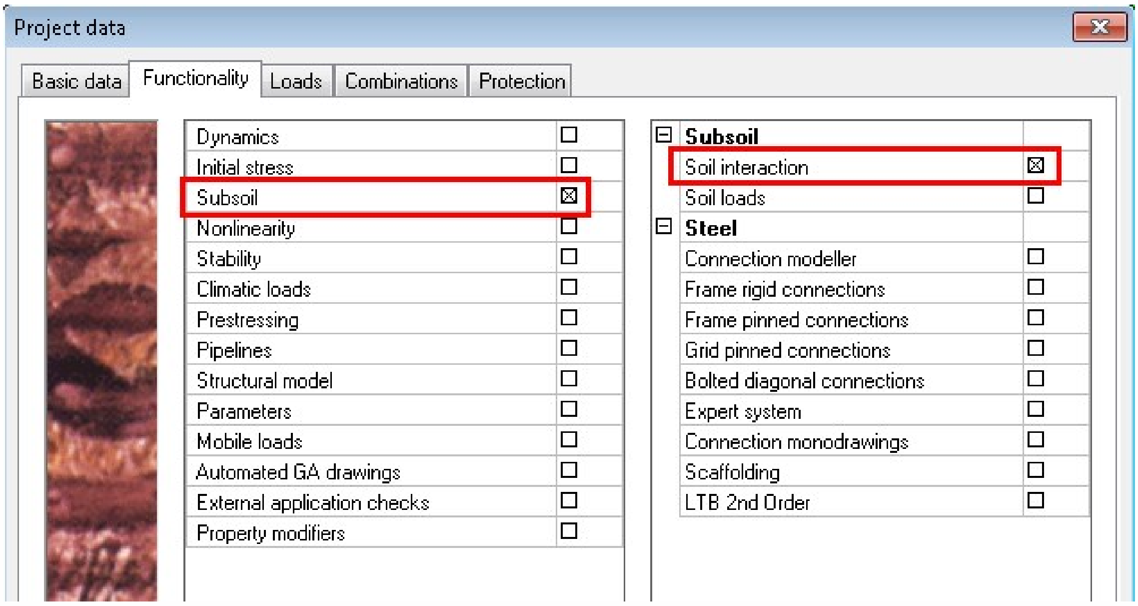


5. The degrease basin can be substituted by spring supports around the plate – this is done automatically in SCIA Engineer when user don't add plates around.
6. When user uses the plates around the support, the springs are not added and the C parameters are calculated for the whole area.

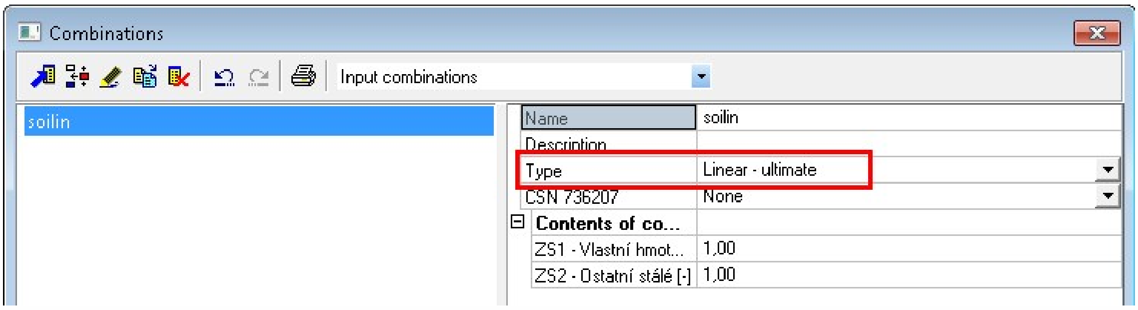
-> The next text describes how to create plates around the support - additional plates.

Settings for soilin calculation

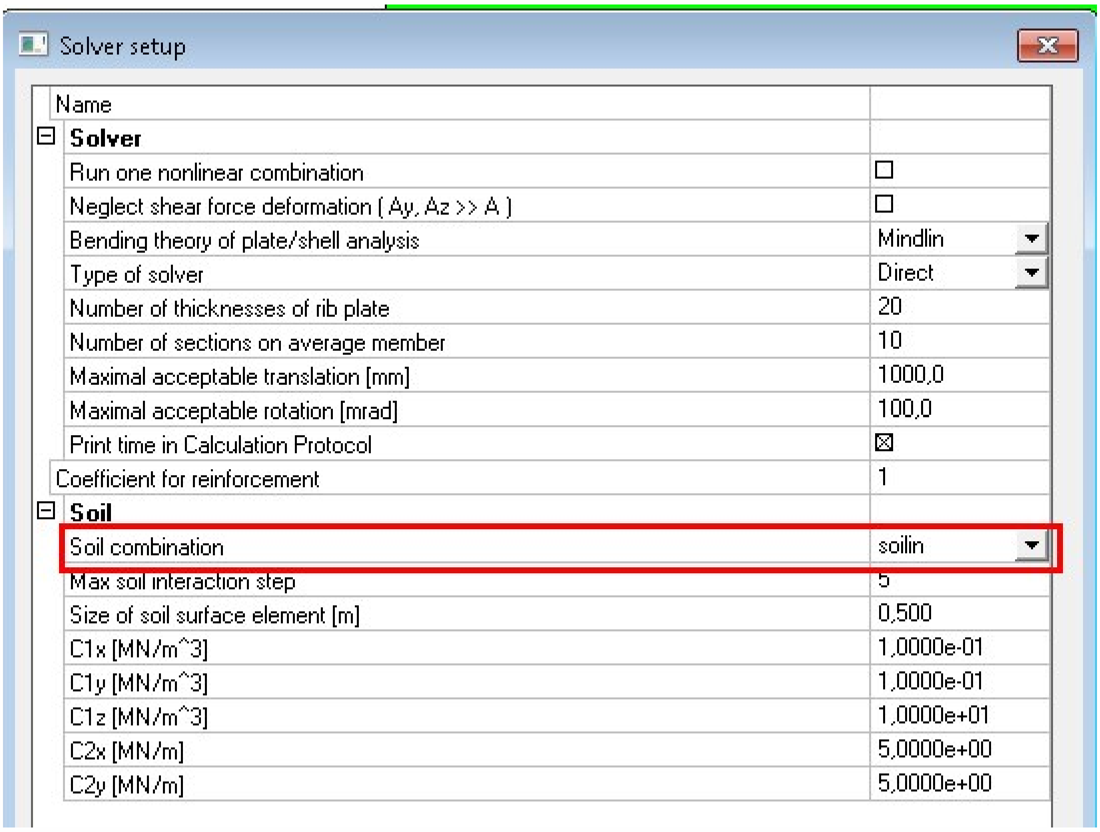
1. The functionality Subsoil and Soil iteration must be checked.



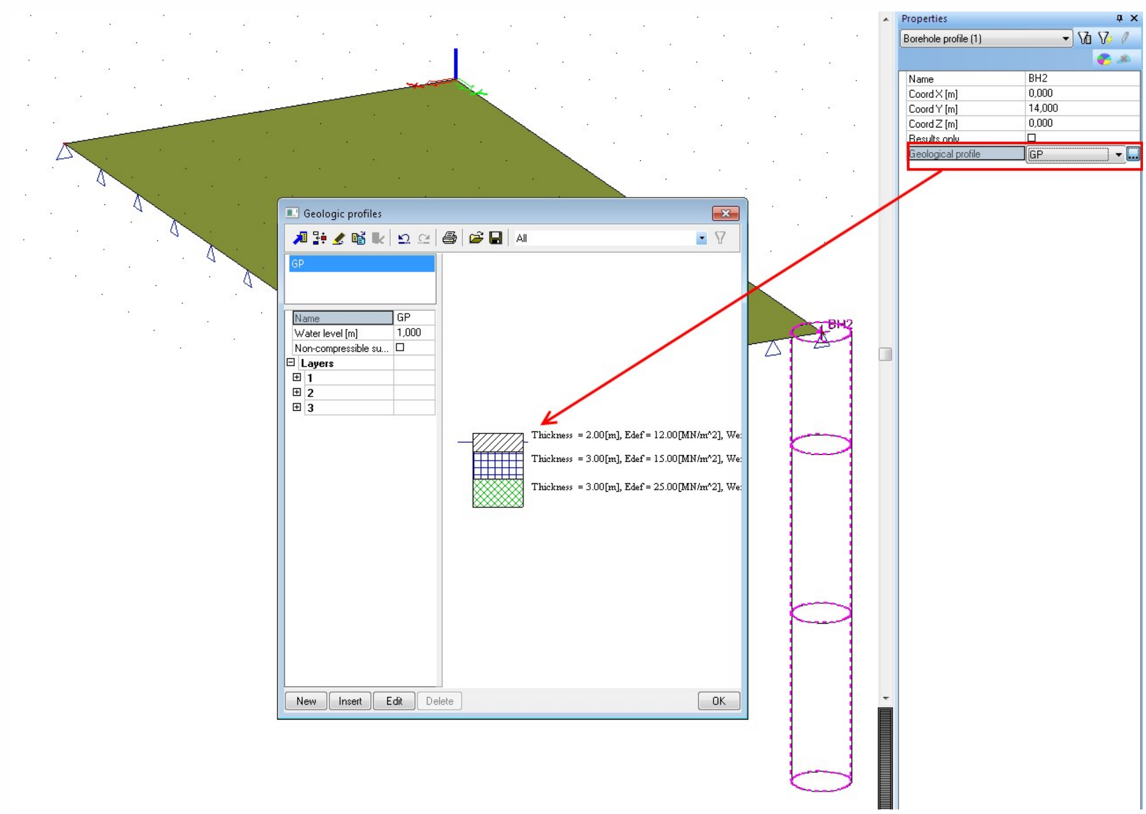
2. One combination must be linear - this combination is used for soilin calculation.



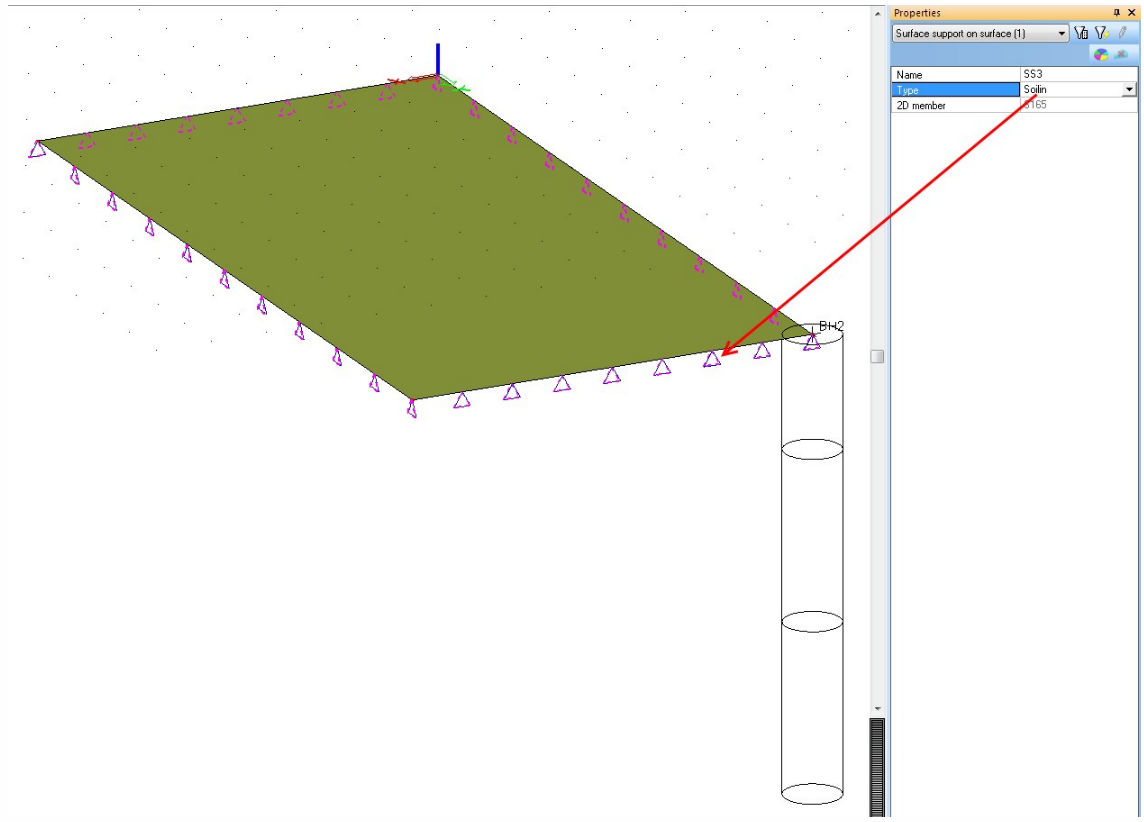
3. This linear combination must be selected in Solver setup to run soilin with it.



4. The project must contain borehole with geologic profile.



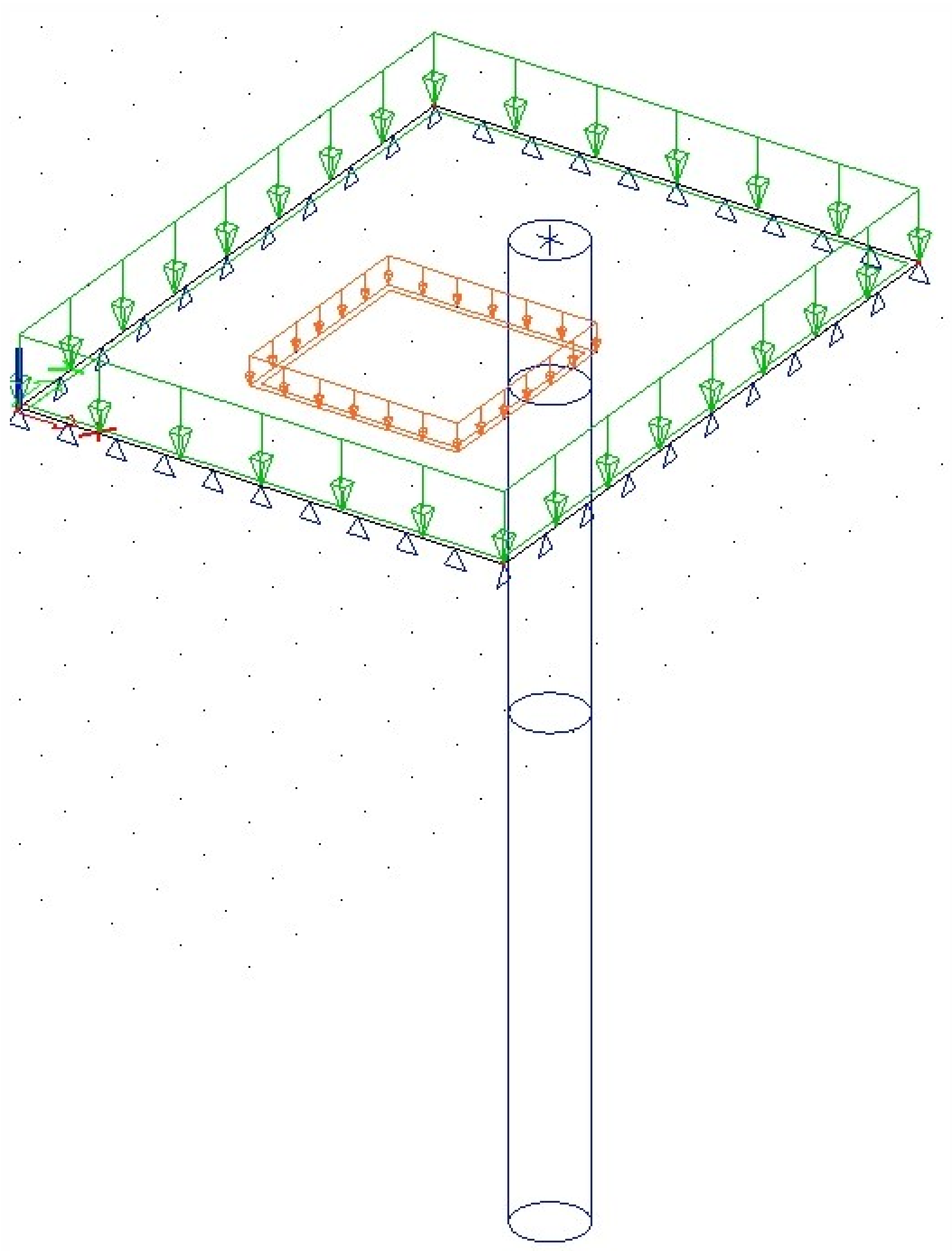
5. The project must contain surface support type soilin.



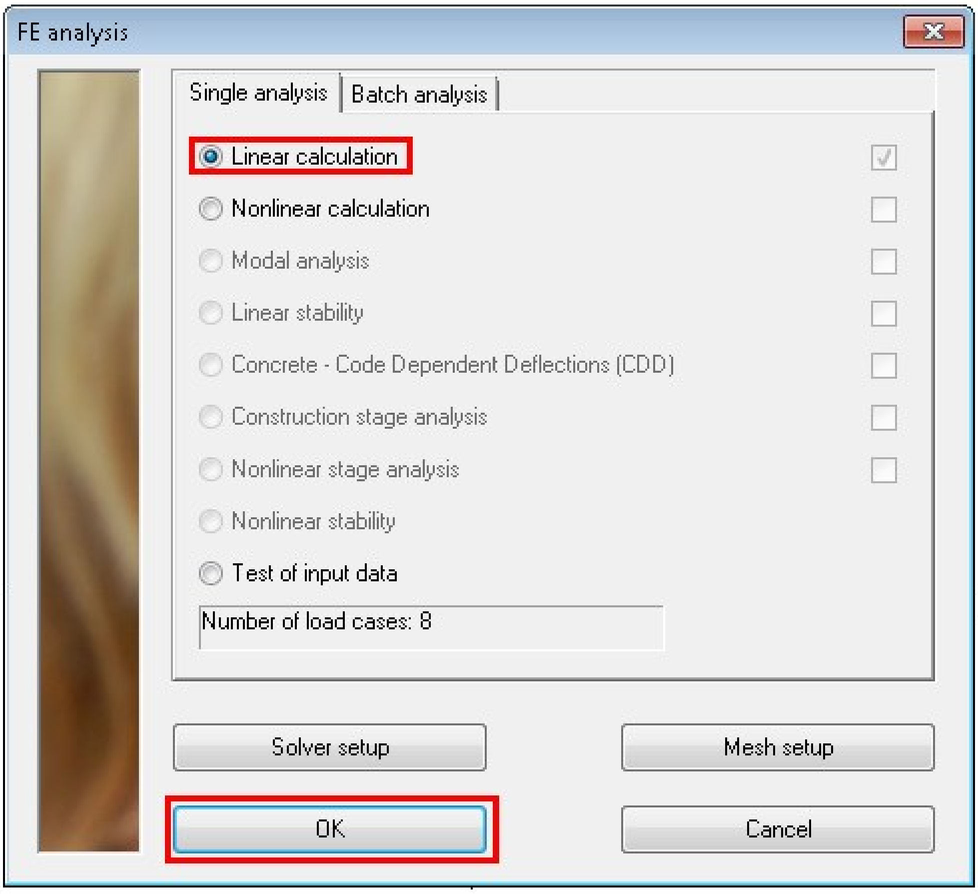
How to calculate the plate without soilin

1. Open the project "soilin_start.esa".

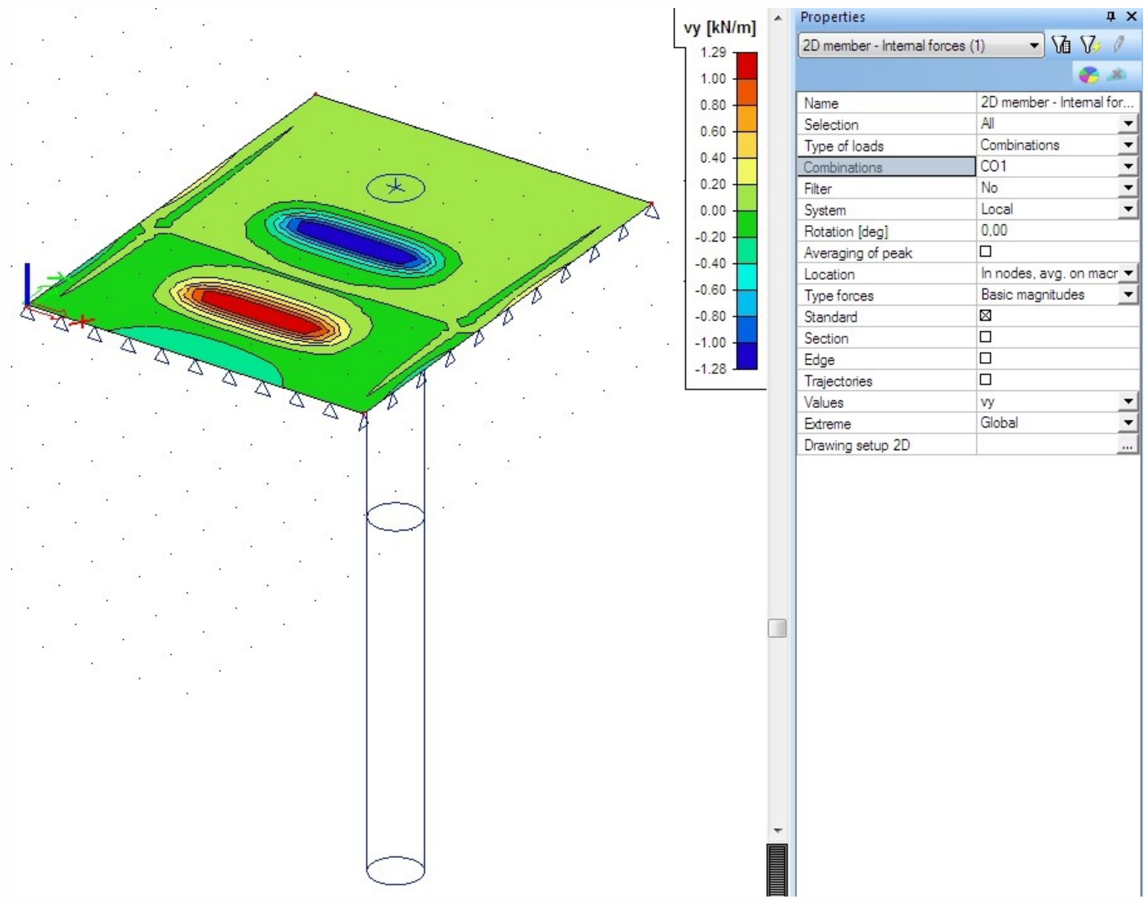
2. There is one plate with the surface support type Individual. This type of the support has a constant parameters C_1 and C_2.



3. Run the linear calculation with the default settings.

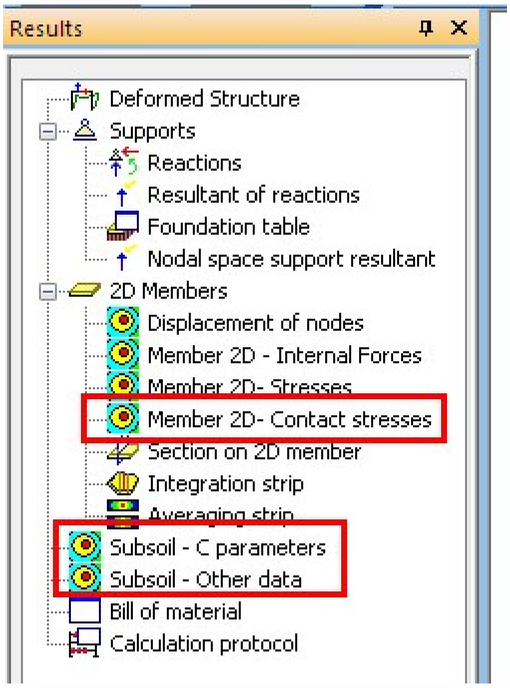


4. Go to the service Results. Display the results for internal forces. There are no results for C parameters.
5. Internal forces - for example vy:

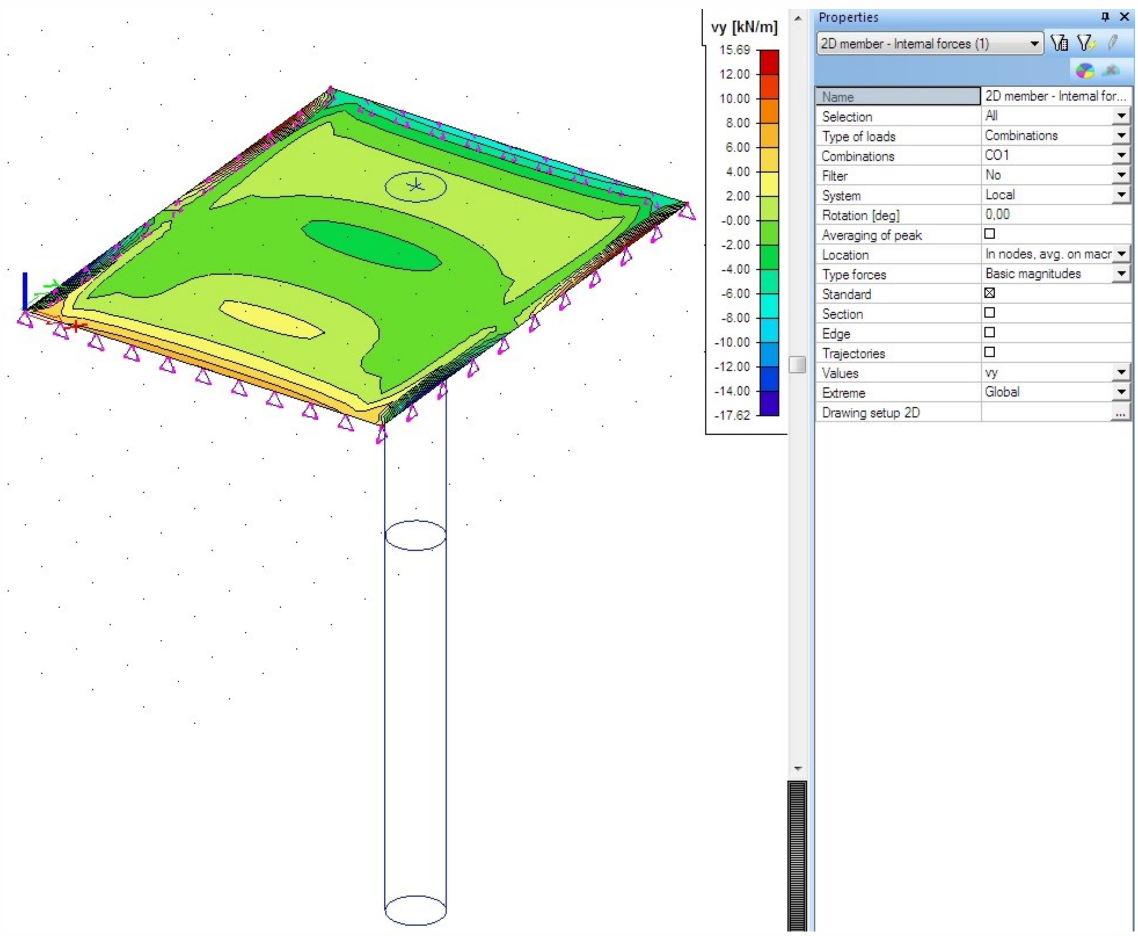


How to calculate the plate with soilin.

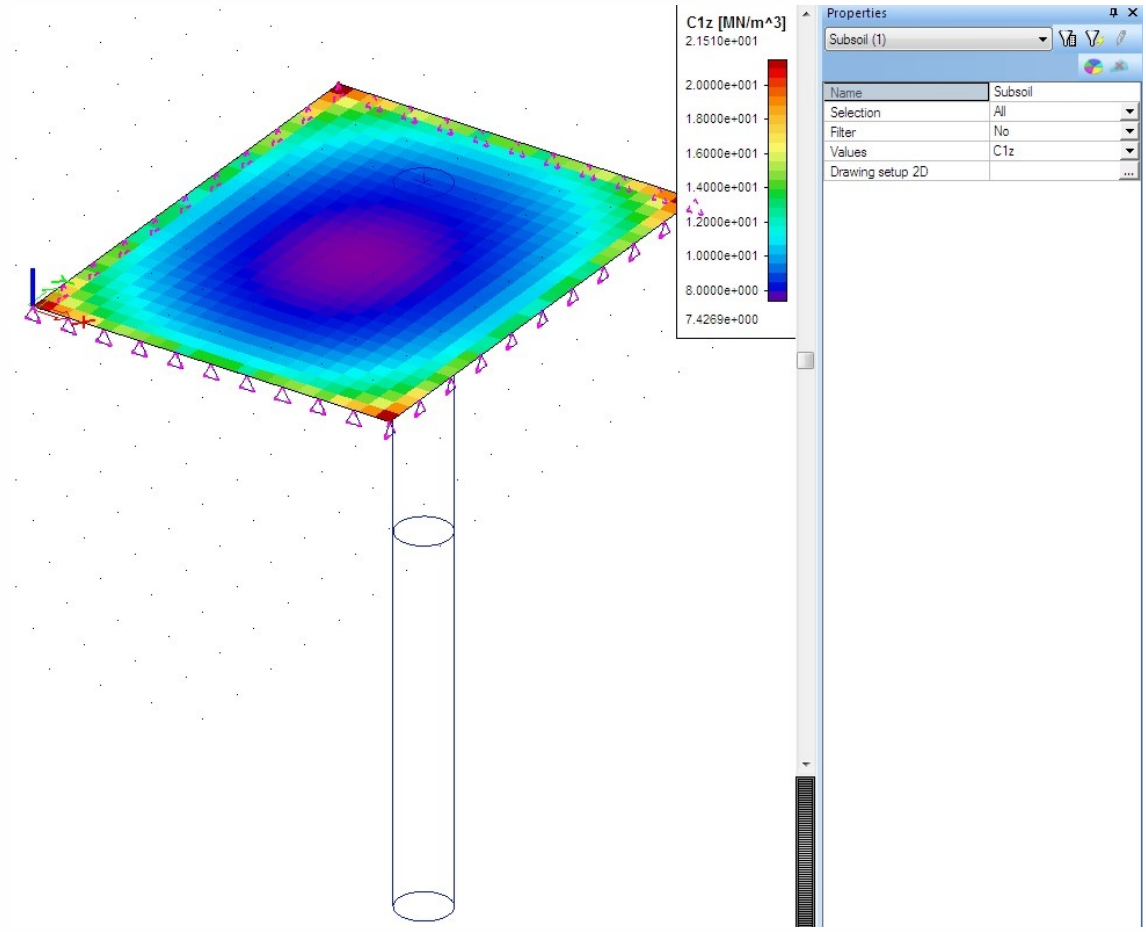
1. Change the support type to soilin.
2. Run the linear calculation again.
3. Go to the service Results. Display the results for internal forces and soilin for combination C01.



4. Internal forces - vy:



5. Subsoil - C parameters - parameter C1z:



6. Subsoil - Other data (see the preview with the table for the settlement):

Preview

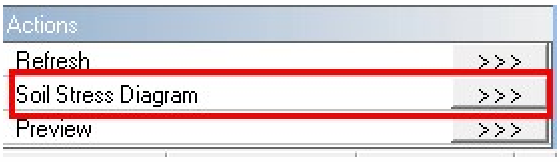
120 %

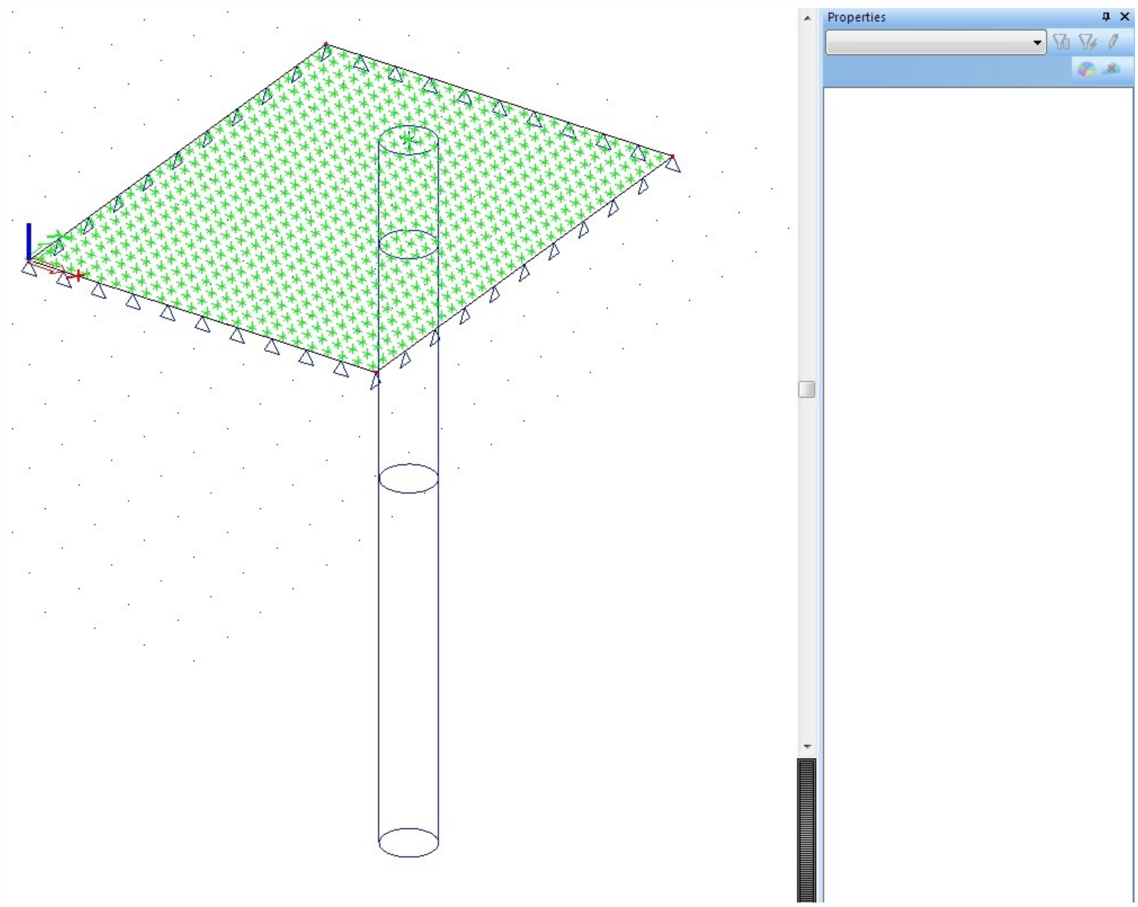
Subsoil - Other data

Selection : All
Combinations : CO1

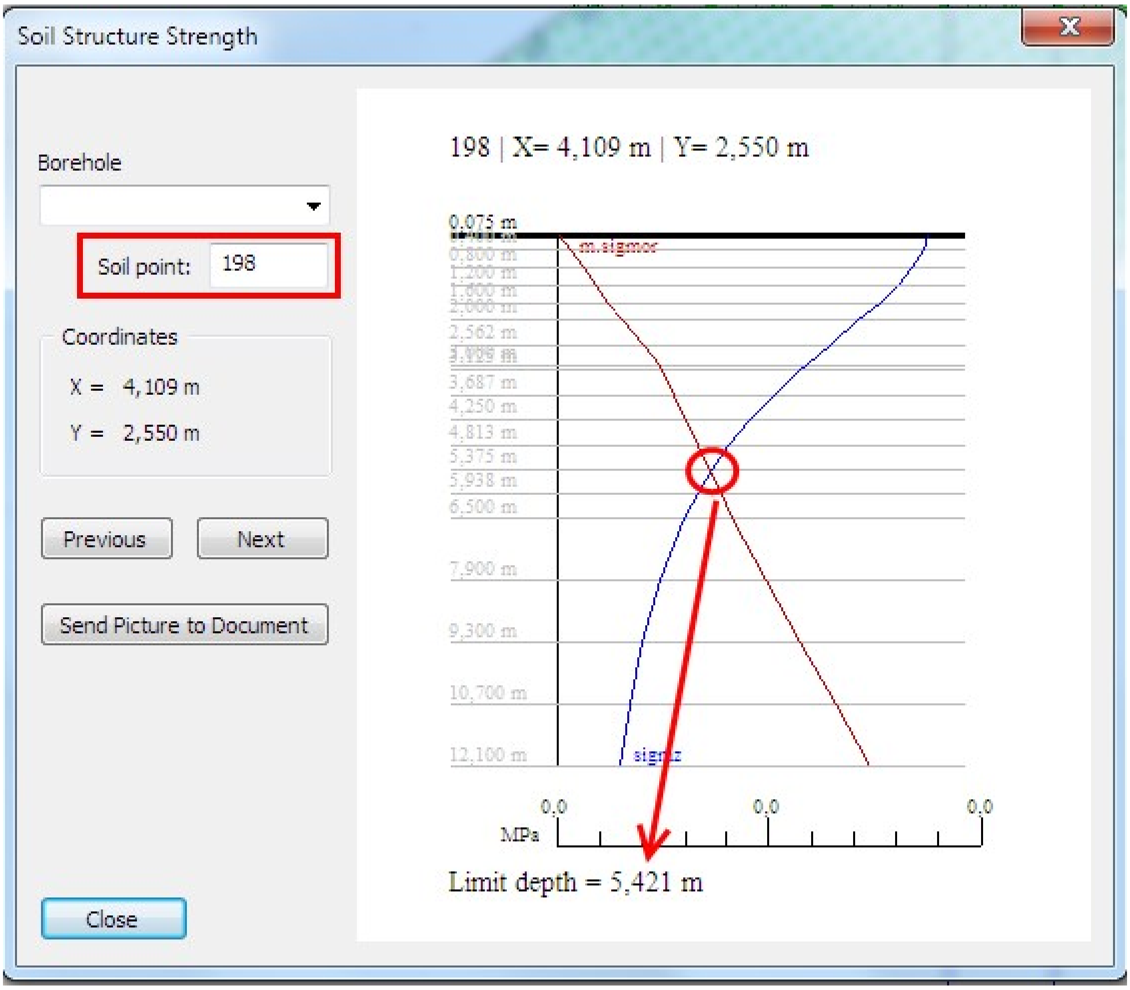
Element	X [m]	Y [m]	w [mm]
1	0,152	0,150	0,7
2	0,457	0,150	1,1
3	0,761	0,150	1,5
4	1,065	0,150	1,6
5	1,370	0,150	1,4
6	1,674	0,150	2,0
7	1,978	0,150	2,2
8	2,283	0,150	2,4
9	2,587	0,150	2,3
10	2,891	0,150	1,8
11	3,196	0,150	2,4
12	3,500	0,150	2,4
13	3,804	0,150	2,4
14	4,109	0,150	1,8
15	4,413	0,150	2,4
16	4,717	0,150	2,3
17	5,022	0,150	2,2
18	5,326	0,150	1,9

7. Subsoil - Other data - use the action button "Soil Stress Diagram" and select one green vertex:





8. A new dialogue appears - there is a stress diagram for the selected mesh element:

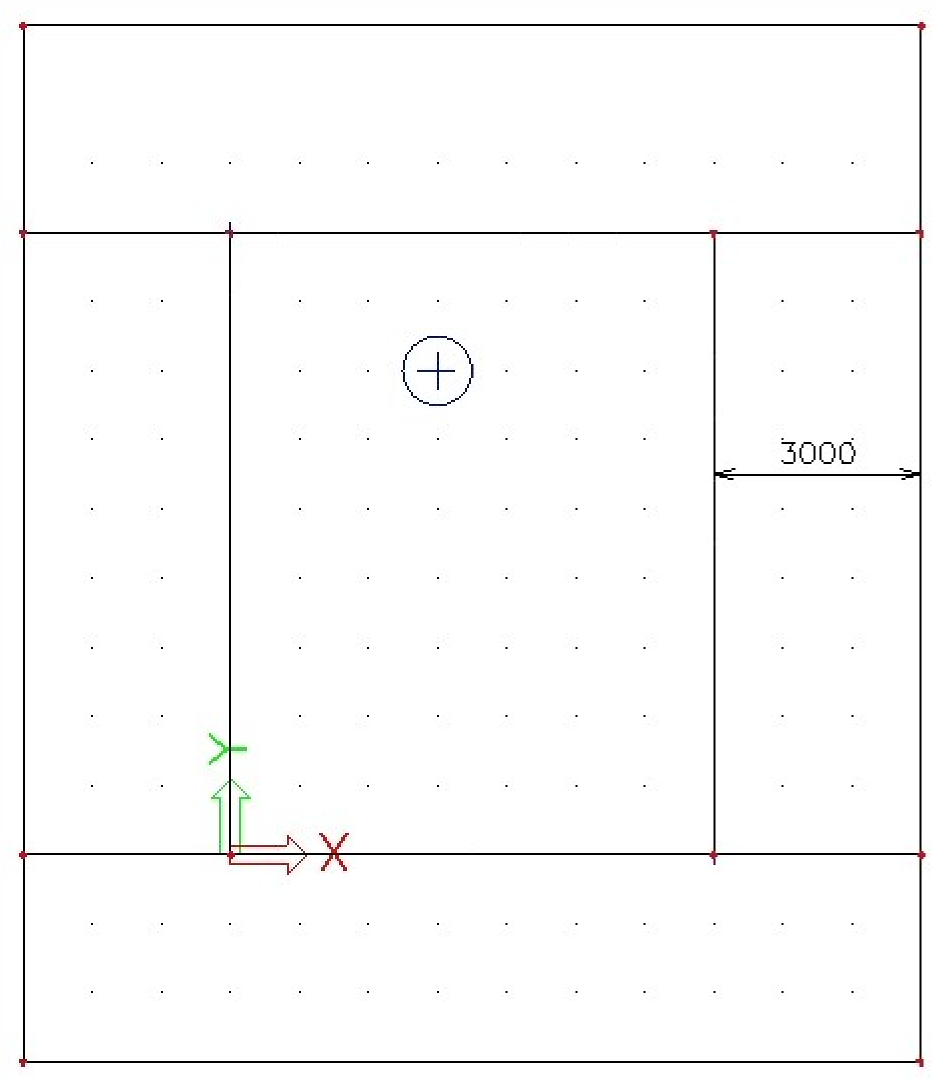


9. Close the dialogue.
10. Use ESC to finish the action.

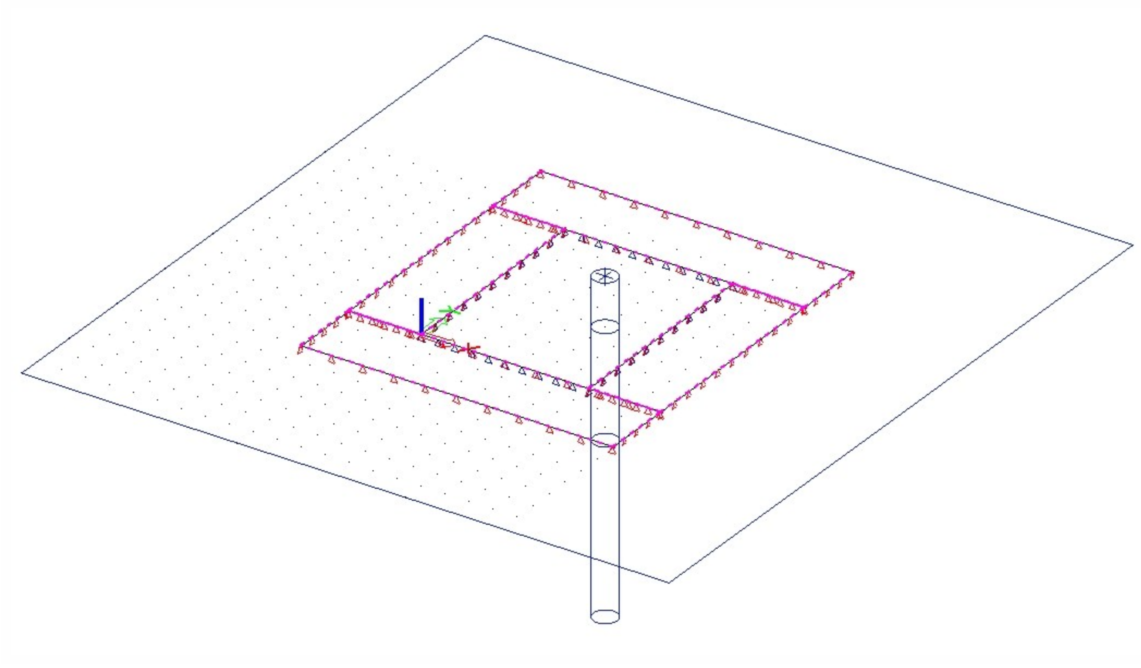
> The edges of the plate are supported by springs automatically.

How to create the additional plates around

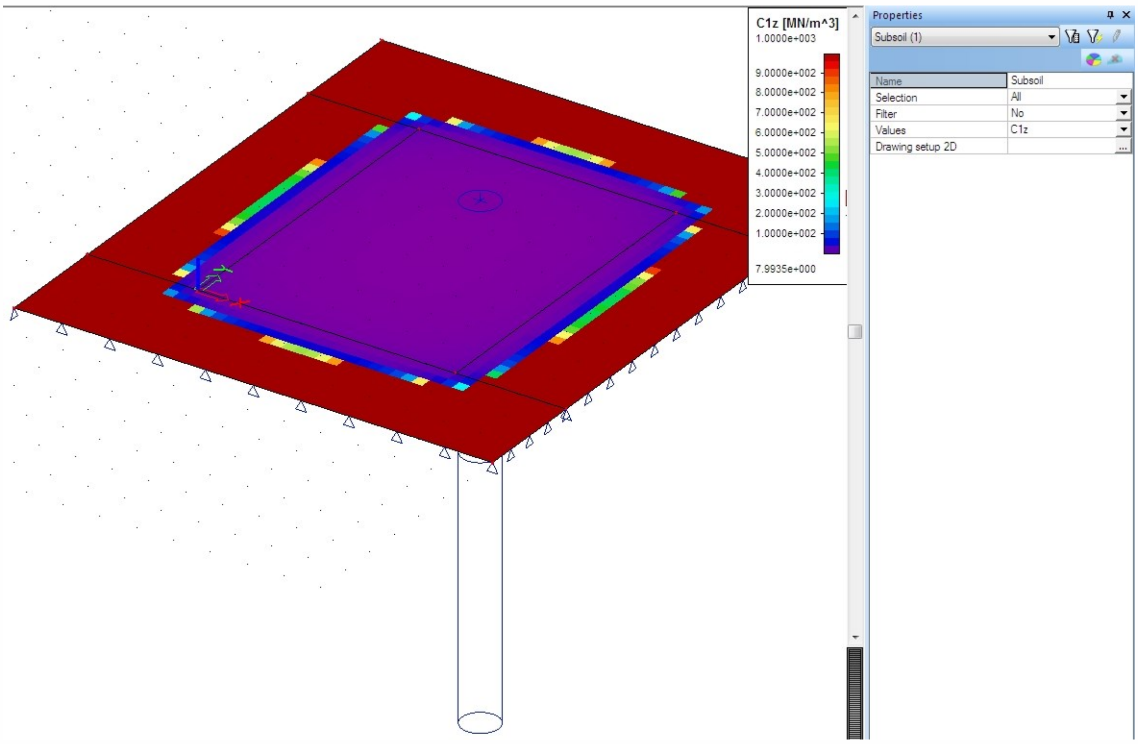
1. Use the same project.
2. Open the Structure service and start the command for inserting a new plate.
3. Set the thickness of the plate to 1mm.
4. Create 4 plates around the surface support according to the picture. The width from the original plate is 3m.



5. Add the surface support type soilin on those plates.



6. Run the linear calculation with the same settings again.
7. Go to the service Results. Display the results for soilin.
8. Subsoil - C parameters - parameter C1z:



9. Subsoil - Other data (see the preview with the table for settlement):

Preview

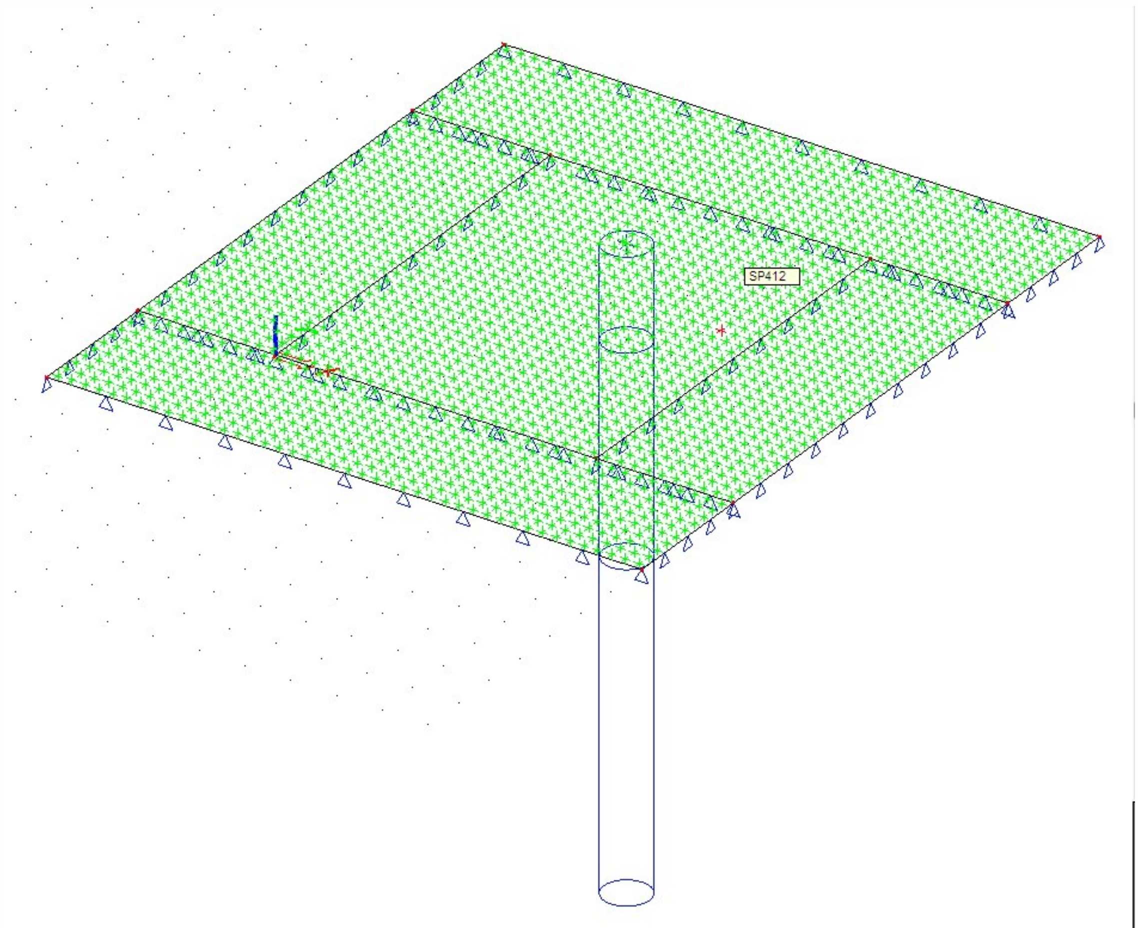
120 %

Subsoil - Other data

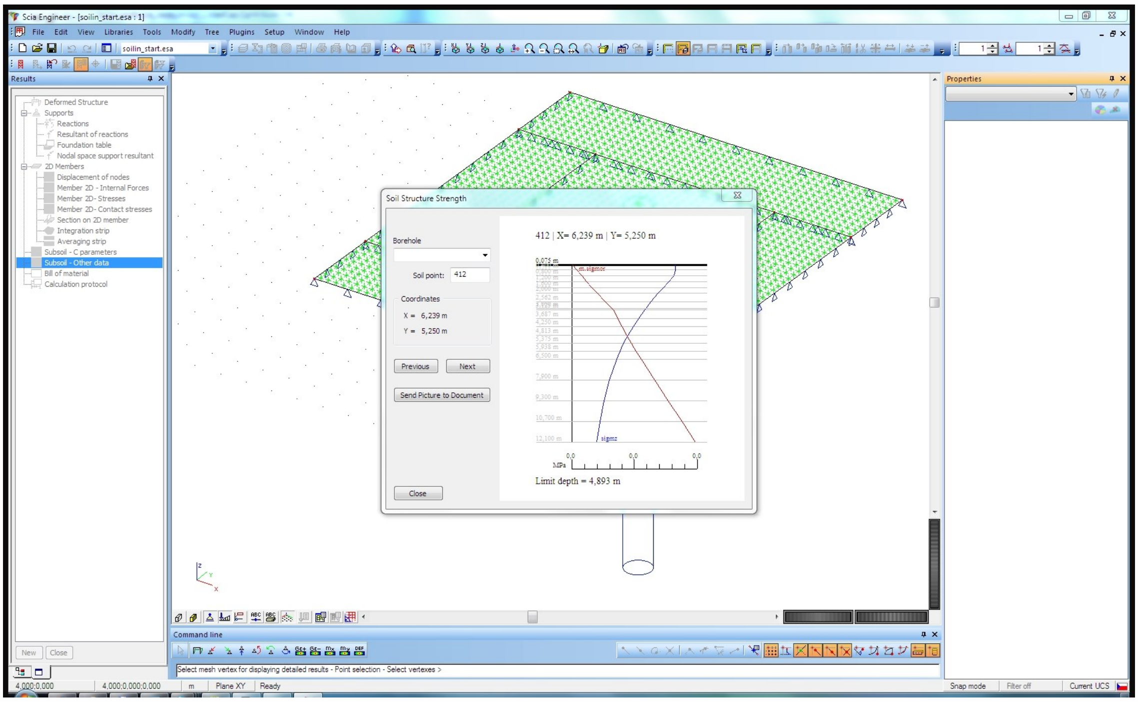
Selection : All
Combinations : CO1

Element	X [m]	Y [m]	w [mm]
1	0,152	0,150	0,6
2	0,457	0,150	0,8
3	0,761	0,150	1,1
4	1,065	0,150	1,3
5	1,370	0,150	1,4
6	1,674	0,150	1,6
7	1,978	0,150	1,7
8	2,283	0,150	1,8
9	2,587	0,150	1,8
10	2,891	0,150	1,9
11	3,196	0,150	1,9
12	3,500	0,150	1,9
13	3,804	0,150	1,9
14	4,109	0,150	1,9
15	4,413	0,150	1,9
16	4,717	0,150	1,8
17	5,022	0,150	1,7
18	5,326	0,150	1,6

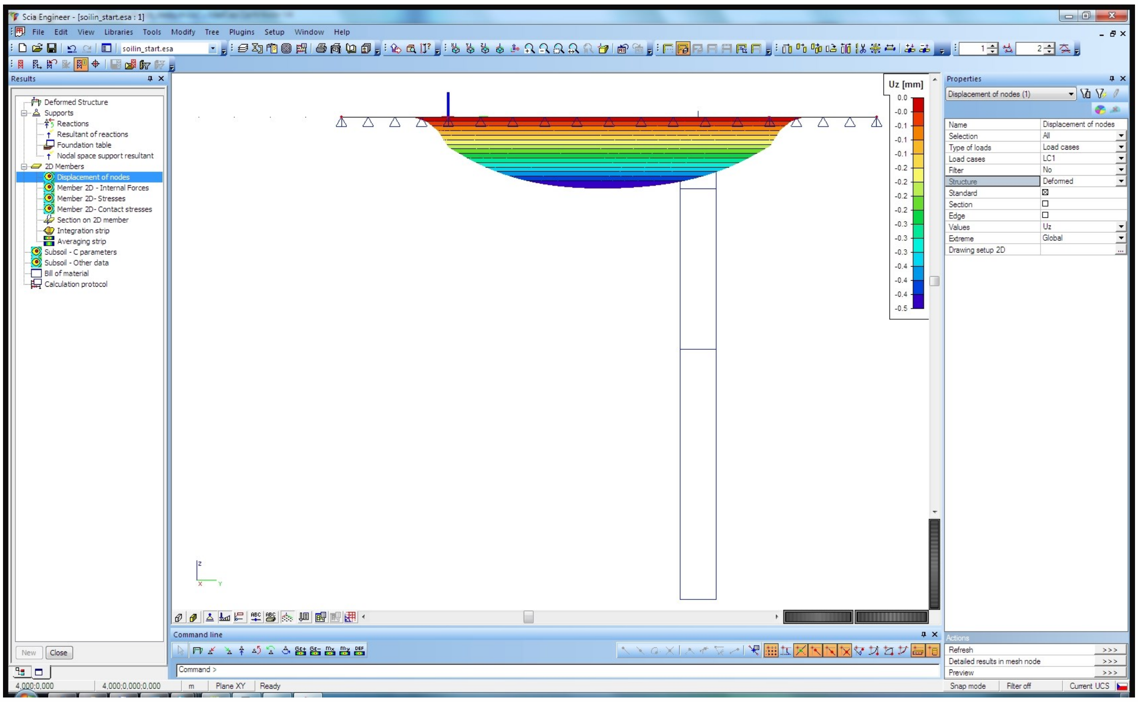
10. Subsoil - Other data - use the action button "Soil Stress Diagram" and select one green vertex:



11. Stress diagram for selected mesh element:



12. Close the dialogue.
13. Use ESC to finish the action.
14. The interesting results are deformations.
15. See the result "Displacement of nodes", value Uz on Deformed structure:



The deformed structure shows the degrease basin.

16. The result is in project "soilin_finished.esa".

Seismic Analysis of Buildings

Introduction

As a general 3D structural analysis software, SCIA Engineer is able to provide a detailed analysis of most civil engineering structures. However, the particular field of seismic design of buildings requires specific tools in order to improve the efficiency of that process. The tools presented in this document are primarily intended for an effective modelization and analysis buildings under seismic action.

The overall seismic behaviour of a building can very often be analyzed accurately by considering each floor as a mass linked to neighbouring floors by walls and columns. Therefore the concept of storeys is widely used in this context. Several aspects of data input, as well as analysis model, result output and design functionality will be directly linked to the storeys of a building.

The IRS (Improved Reduced System) analysis will be used for the computation of eigenmodes of the structure. That technique allows reducing drastically the number of degrees of freedom of a large finite element mesh. The computation time is reduced accordingly. Moreover, most of the local vibration modes, which are irrelevant for the overall seismic response of the structure, will not appear in the results – unlike in standard FE analysis.

SCIA Engineer also supports seismic analysis of buildings using the simplified method of Equivalent Lateral Forces (ELF). Although it is a purely static analysis of the structure, some input of data related to dynamic analysis is required, such as masses and combinations of mass groups, which is used when generating the ELF loading of the structure.

Storey results give a synthetic view of results in each storey. Displacements, accelerations, internal forces and resultant forces may be displayed per storey, either as bulk values for each storey, or as detailed results for each wall and column. That feature may also be used for fast and comprehensive load descending output.

Most seismic design standards require that torsional effects due to accidental mass eccentricity are taken into account. Based on storey data, the effects of accidental eccentricity may be included in the model according to various methods.

This chapter gives specific information about the seismic analysis of buildings. Read also the chapter dedicated to general seismic analysis in SCIA Engineer.

Important information and help about troubleshooting of dynamic and seismic analysis is provided in Dynamic analysis troubleshooting.

Reduced Analysis Model

Introduction: the modified IRS method

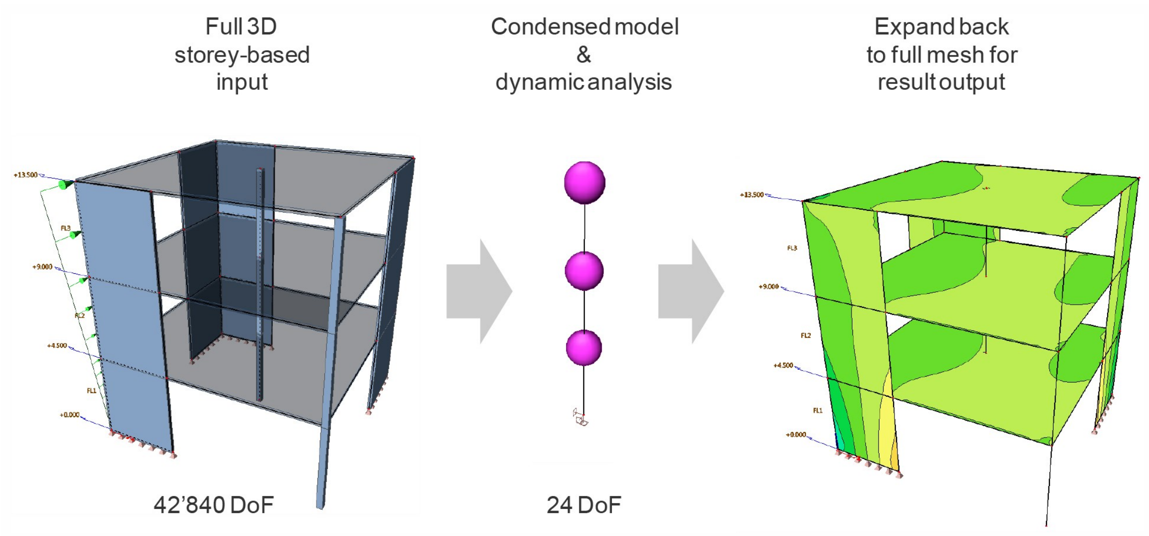
The actual tendency in FE structural analysis is using full 3D modelization of the considered structure. SCIA Engineer obeys that rule as structures are usually modelized in 3D using beam and shell elements, including buildings.

Once a detailed 3D modelization is ready for statical analysis of a structure, it is natural to use it also for dynamic analysis and, more specifically, for seismic design. A typical issue of full 3D modelization is, that seismic design regards mostly the overall behaviour of the structure whence the full mesh of the structure provides a lot of information about local behaviours. More specifically for modal analysis, the full mesh lets appear local vibration modes, which are irrelevant for the overall seismic response of the structure. It appears hence logical to use a different, reduced mesh for the dynamic analysis.

Well known matrix condensation techniques (Guyan Reduction, also known as static condensation [1]) allow obtaining very efficiently a reduced system, but those methods are not well suited for dynamic analysis. An Improved Reduced System (IRS) method has been developed [2] which takes into account not only the stiffness matrix of the system, but also the mass matrix during the condensation process. That method proved to give excellent results for dynamic analysis, with both modal analysis and direct time integration methods.

The algorithm implemented in SCIA Engineer used the IRS method and consists of 3 steps:

1. The IRS method is used for condensing the mesh of the analysis model.
2. The modal analysis is performed using the reduced mesh, which has typically 1'000 times less degrees of freedom than the original full mesh. This makes the calculation of eigenvalues massively faster on large structures and also avoids unwanted local modes. The latter is particularly interesting for seismic analysis.
3. The results of the reduced system are expanded to the original full mesh, allowing for output of detailed results in the entire structure.



Although the reduced system (center picture) seems to look a lot like a classical 1-node-per-storey model, with equivalent stiffness of the supporting members, it is in reality a much more advanced. The main difference lies in the fact, that the mass matrix that is used is a full matrix (non diagonal), which allow to take into account the real eccentricity of the masses of each storey. Moreover, as the modal results are re-extrapolated to the original 3D mesh after the modal analysis, it provides a much higher level of detail.

IRS analysis vs diaphragms

Modelizing deck slabs as diaphragms in a building for seismic analysis is a common technique. It has several purposes:

1. elimination of irrelevant, local bending vibration modes in the slabs
2. elimination of unwanted frame effects from the structural behaviour (by removing the bending stiffness of slabs)
3. reduction of computation time
4. easy handling of mass eccentricities for each deck

Items 1, 3 and 4 are addressed directly by the IRS analysis:

- local modes in all structural elements are implicitly removed, due to the elimination of unwanted degrees of freedom. Of course, adding more reduction nodes would allow for more detailed analysis of local modes, but it is particularly interesting for seismic analysis to keep in the reduced model only the nodes that are strictly necessary to reproduce the typical seismic behaviour of a building. Ultimately, it is up to the user to choose the reduction points in such a way that the wanted eigenmodes are obtained.

- the computation time is reduced, due to the drastic reduction of the number of degrees of freedom; actually, the reduction is even more important than with diaphragms, as supporting members are also condensed. The reduction of computation time affects only the computation of the eigenmodes itself. The pre- and post-processing times remain practically unchanged, which include overall data preparation, mesh generation and modal superposition of local results. These operations are, obviously, still performed on the full mesh of the model and can be quite time consuming.
- the IRS analysis uses a full mass matrix, which allows for exact implementation of mass eccentricity in each node of the reduced system

Item 2 – removal of frame effects – is not addressed by the IRS analysis in itself, as it does not modify the mechanical behaviour of the structure. However, as unwanted local bending modes are implicitly removed from the reduced system, so-called flexible diaphragms may be easily simulated by significantly reducing the bending stiffness of deck slabs. Not only does that allow to obtains classical diaphragm behaviour by means of a very low bending stiffness, but also intermediate behaviour where the bending stiffness is less drastically reduced and frame effects are therefore reduced, but not completely removed.

> Using diaphragms and IRS analysis together is actually possible in theory - although not supported yet in SCIA Engineer.

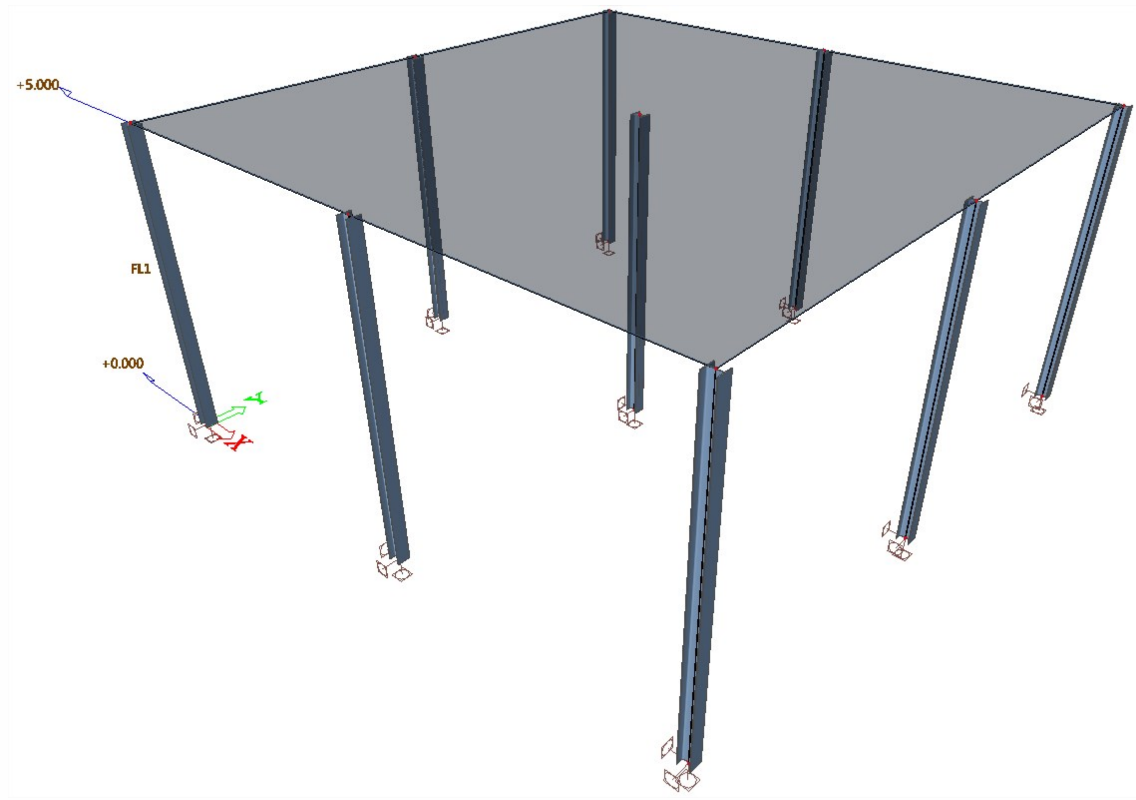
> Using diaphragms and IRS analysis together is actually possible. Diaphragm constraints may be used to eliminate completely frame effects due to the bending stiffness of the slabs, and still benefit from the other advantages of the reduced system. Diaphragms are currently supported only for composite decks in SCIA Engineer.

Benchmarks

Example 1: simple platform

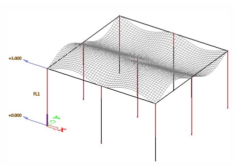
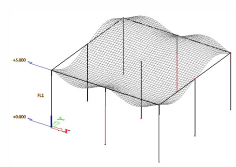
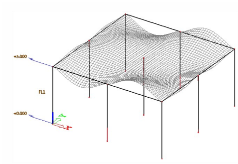
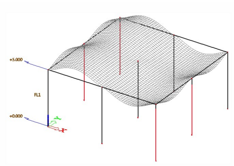
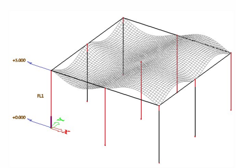
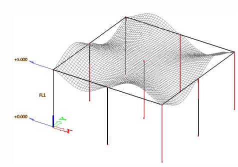
The structure is a simple platform, with 9 steel columns and a thin plate on the top. This academic example only intends to illustrate some aspects of the method.

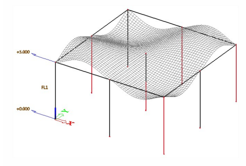
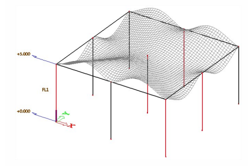
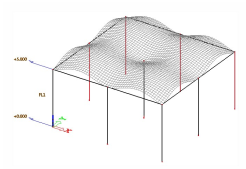
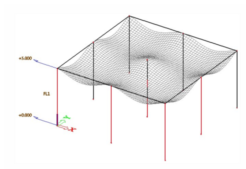
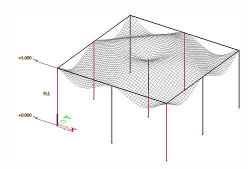
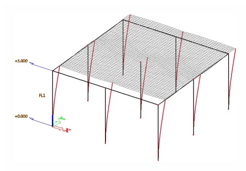
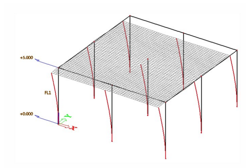
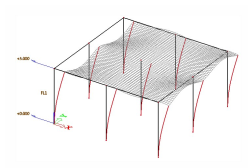
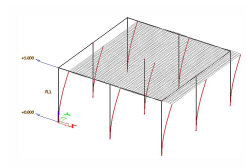
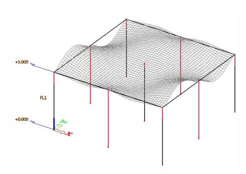
The reduced model consists of 2 R-nodes, located at the top and at the bottom of the middle column.



The table below shows the natural frequencies and eigenshapes for both full mesh and IRS analysis. The 11 first modes are displayed for the full mesh analysis, along with the corresponding modes from the IRS analysis.

As expected, there are gaps in the IRS list, as the DoF of the reduced model cannot represent all the bending modes of the plate. Those modes are anyway irrelevant for the seismic behaviour of the structure. Apart from the change of sign for most eigenmodes, it is clear that the correspondence is excellent for both frequency and mode shape for eigenmodes that are significant for seismic behaviour.

Full Mesh analysis	IRS analysis	Full Mesh analysis	IRS analysis
Mode 1 – 1.15 Hz 	Mode 1 – 1.17 Hz 	Mode 7 – 2.10 Hz 	N/A
Mode 2 – 1.15 Hz 	Mode 2 – 1.17 Hz 	Mode 8 – 2.15 Hz 	N/A
Mode 3 – 1.21 Hz	N/A	Mode 9 – 2.27 Hz	N/A

Full Mesh analysis	IRS analysis	Full Mesh analysis	IRS analysis
			
Mode 4 – 1.23 Hz 	Mode 3 – 1.23 Hz 	Mode 10 – 2.30 Hz 	N/A
Mode 5 – 1.42 Hz 	Mode 4 – 1.42 Hz 	Mode 11 – 2.32 Hz 	Mode 5 – 2.32 Hz 
Mode 6 – 2.10 Hz 	N/A		

The next table shows the modal masses for both models. Significant modes for seismic behaviour are spread up to mode 11 for full mesh analysis. For IRS analysis, they are all grouped are the beginning of the list. Note that the first two modes are pure bending modes and would definitely not occur in a building.

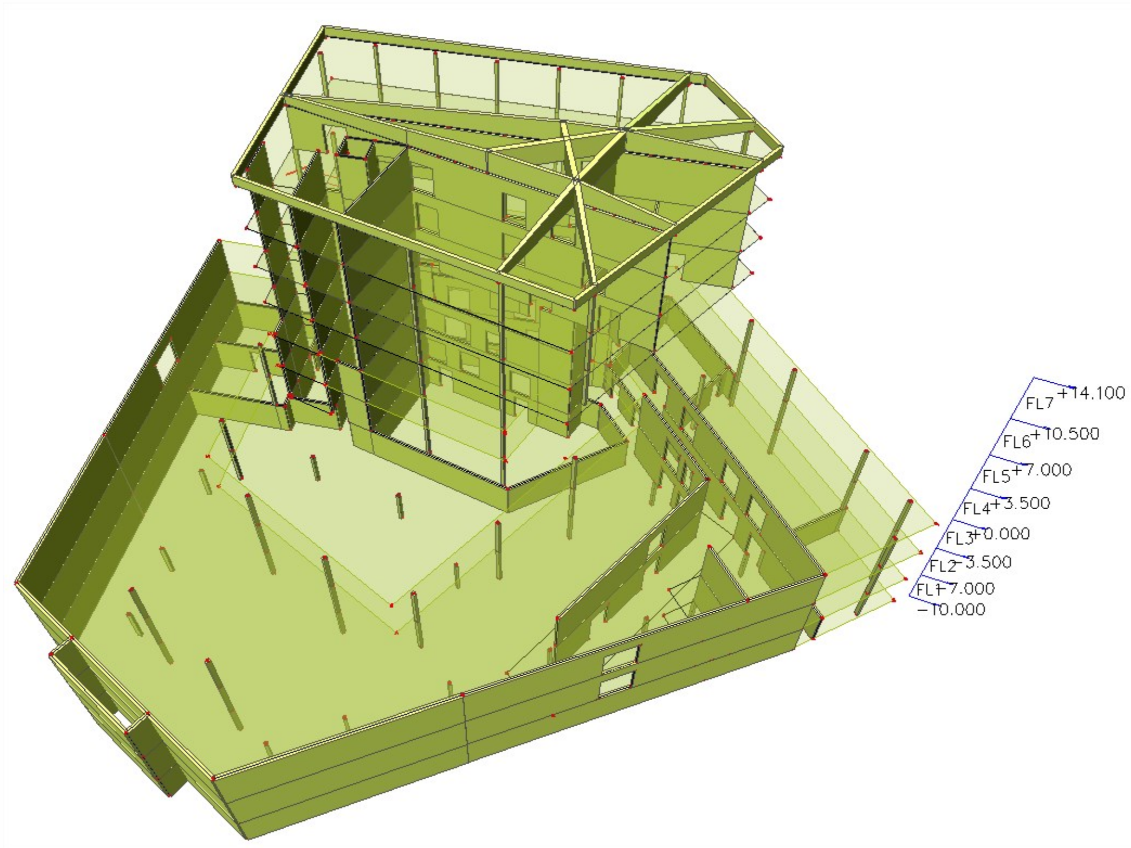
Another important point is, that the obtained total modal mass is higher with the IRS analysis. The 12 listed modes represent all the eigenmodes of the reduced model, as it has 12 DoF. That is the reason why the IRS analysis obtains 100% of the mass. For the full mesh analysis, it is normal that the obtained modal mass is lower, as there are many more modes that are not listed here.

Full Mesh analysis						IRS analysis				
Mode	Freq.	Wxi /	Wyi /	Wzi /		Mode	Freq.	Wxi /	Wyi /	Wzi /
	[Hz]	Wxtot	Wytot	Wztot			[Hz]	Wxtot	Wytot	Wztot
1	1.15	0	0.0001	0		1	1.17	0	0.0003	0
2	1.15	0	0	0		2	1.17	0	0	0
3	1.21	0	0	0		3	1.23	0	0	0.8934
4	1.23	0	0	0.887		4	1.42	0	0.9987	0
5	1.42	0	0.9981	0		5	2.32	0.999	0	0
6	2.10	0	0.0001	0		6	2.71	0	0	0
7	2.10	0.0004	0	0		7	130.72	0	0.001	0
8	2.15	0	0	0		8	205.43	0.001	0	0

Full Mesh analysis						IRS analysis				
9	2.27	0	0	0		9	248.14	0	0	0
10	2.30	0	0	0.0001		10	413.53	0	0	0.1066
11	2.32	0.9644	0	0		11	711.66	0	0	0
12	2.47	0	0.0011	0		12	737.23	0	0	0
		0.9648	0.9994	0.8871				1.00	1.00	1.00

Example 2: 7-storey building

This example is a real, 7-storey building. No particular simplification has been done to create the model for the seismic analysis from the static modelization.



Building of the ACPC, Fribourg, Switzerland

Courtesy of GIBES Engineering Group, Lausanne, Switzerland

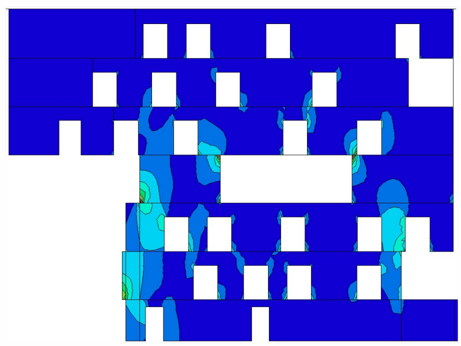
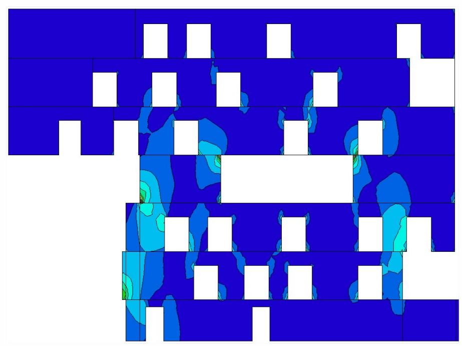
	Full Mesh analysis	IRS analysis
Degrees of freedom	152’988	48
Requested modes	320	48
Total time: data generation, modal analysis, seismic analysis	31’27”	5’17”
Obtained total modal mass (X / Y / Z)	90% / 94% / 78%	95% / 96% / 98%
Count of most significant modes to obtain 90% modal mass	187 / 103 / >320	14 / 17 / 8

> Note: 320 is the number of eigenmodes that is required in this case to obtain 90% of cumulated modal mass in both X and Y directions. For the vertical direction, over 400 modes would be necessary (not calculated).

	Full Mesh analysis		IRS analysis	
1st most significant mode in X direction	3.12 Hz (3)	32.2%	3.12 Hz (3)	33.0%
2nd most significant mode in X direction	7.38 Hz (18)	8.4%	14.56 Hz (30)	11.5%
1st most significant mode in Y direction	1.71 Hz (1)	20.7%	1.71 Hz (1)	20.7%
2nd most significant mode in Y direction	2.44 Hz (2)	13.2%	2.44 Hz (2)	13.2%
1st most significant mode in Z direction	5.16 Hz (7)	9.6%	115.57 Hz (46)	36.4%
2nd most significant mode in Z direction	5.14 Hz (6)	7.0%	5.60 Hz (7)	19.1%

About the Z direction: eigenmode 46 @ 115 Hz is not represented in the full mesh analysis because the last computed mode (320) reaches only 29 Hz. That particular mode is anyway irrelevant for the seismic analysis as it is completely out of the frequency range of an earthquake. The modes nr 7 in both models, although their frequency slightly differs, show practically the same deformed shape.

The final internal forces after modal superposition for the seismic analysis show very close results. In the main wall of the structure (picture below), the peak values of the vertical membrane force ny are only 2% apart. The entire result distribution is close to identical.

Full Mesh analysis	IRS analysis
	
ny_{max} = 2'362 kN/m	ny_{max} = 2'416 kN/m

Using the IRS model in SCIA Engineer

Unlike the classical modal analysis, which typically uses a lumped mass matrix (only diagonal terms are non-zero), the reduced system uses a full mass matrix , with non-zero values out of the diagonal. This means that mass eccentricities can be taken into account easily by the reduced system. The very small size of the reduced system allows using the full mass matrix.

Therefore the reduction points – or so-called R-nodes – that will constitute the reduced model do not need to be located in a particular position, such as the mass center of each storey. As the structure may have to be calculated several times with various distributions of the masses, the mass center of each storey is likely to be slightly different depending on the selected mass combination. Thanks to the use of a full mass matrix, the same R-nodes may be used in all cases.

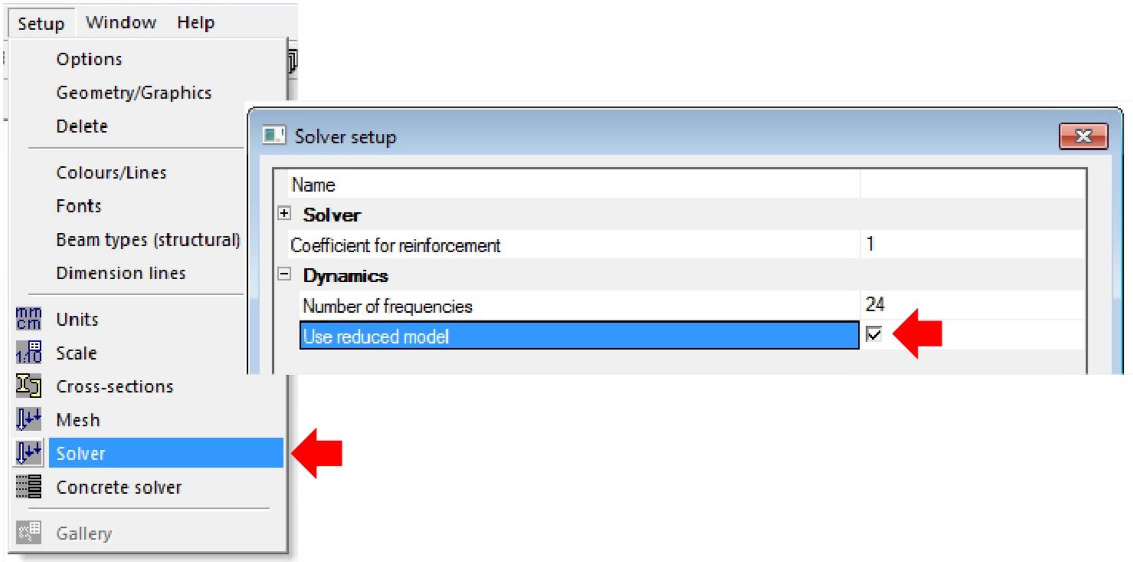
Another benefit of the IRS analysis is, that the exact position of the mass center of each storey does not need to be know beforehand. It is automatically calculated during the condensation process and can be obtained as a result of the analysis.

During the analysis, the reduced model is computed automatically from the full mesh. Each node of the full mesh is mapped to the closest R-node of the reduced model.

It is up to the user to define how many R-nodes will be used for the analysis and, therefore, how many degrees of freedom the reduced model will have. For seismic analysis of buildings, the typical reduced model consists of 1 R-node per storey, i.e. per deck slab.

Enabling the reduced model

First of all, the reduced model analysis must be enabled in the project. For this, tick the option “use reduced model” in the solver settings.

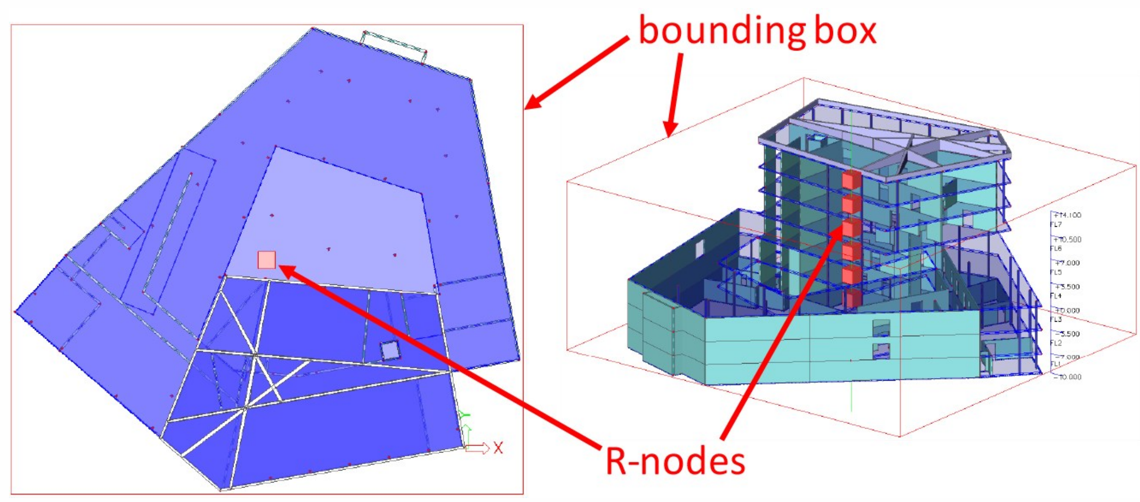


Definition of the R-nodes

In SCIA Engineer 2013, the reduced model is defined directly from the storey data.

Storeys must be defined in the modelization. The program will create the reduced analysis model by creating one R-node per storey. This means that the reduced analysis model will be valid for buildings that have one deck slab per storey. Each deck slab may be made of several 2D members.

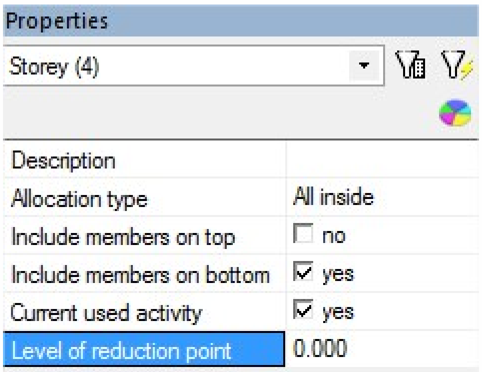
The program will place a R-node at each storey, in the middle of the bounding box of the structure:

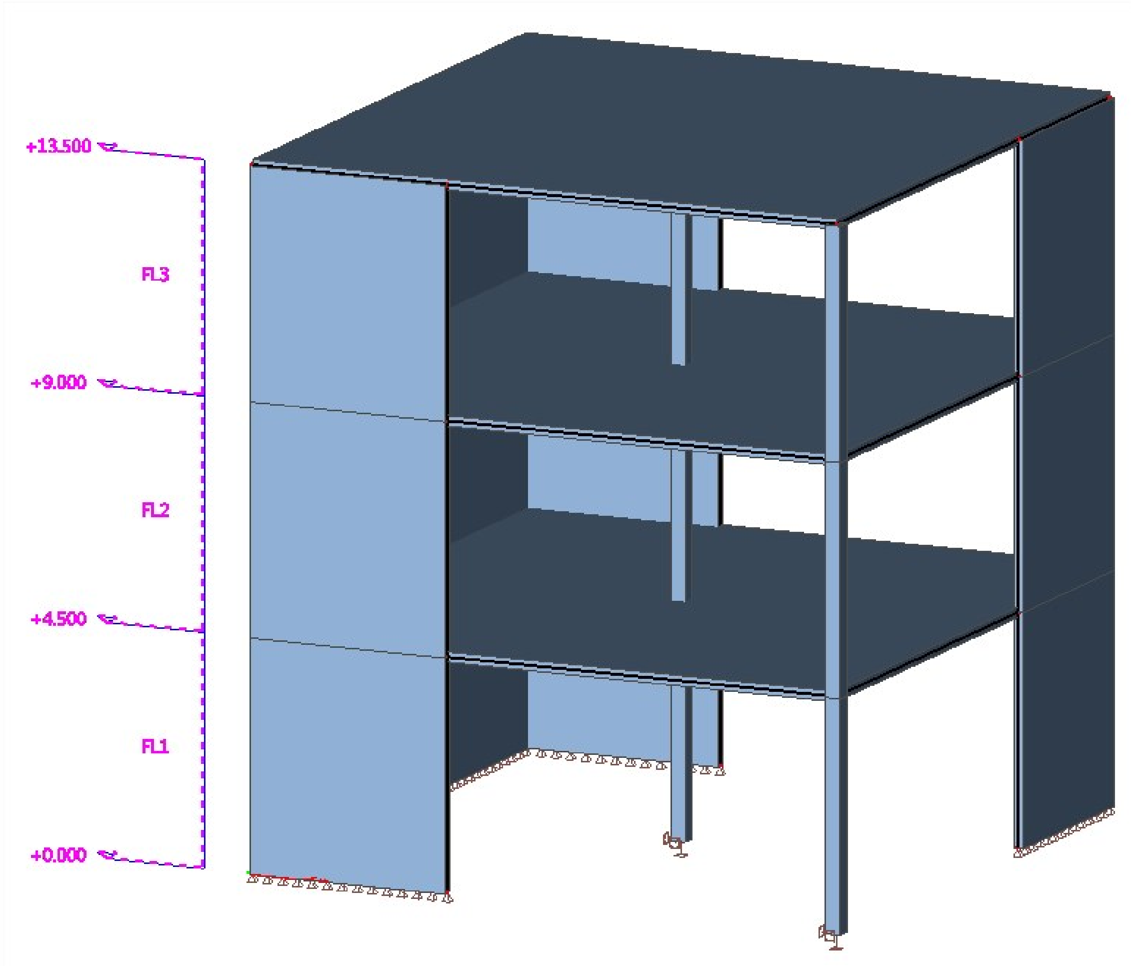


Note: this is a principle drawing. R-nodes are not displayed in SCIA Engineer.

Further development is planned, allowing more advanced layout of R-nodes, i.e. more than one per storey. The current version of SCIA Engineer allows for one R-node per storey.

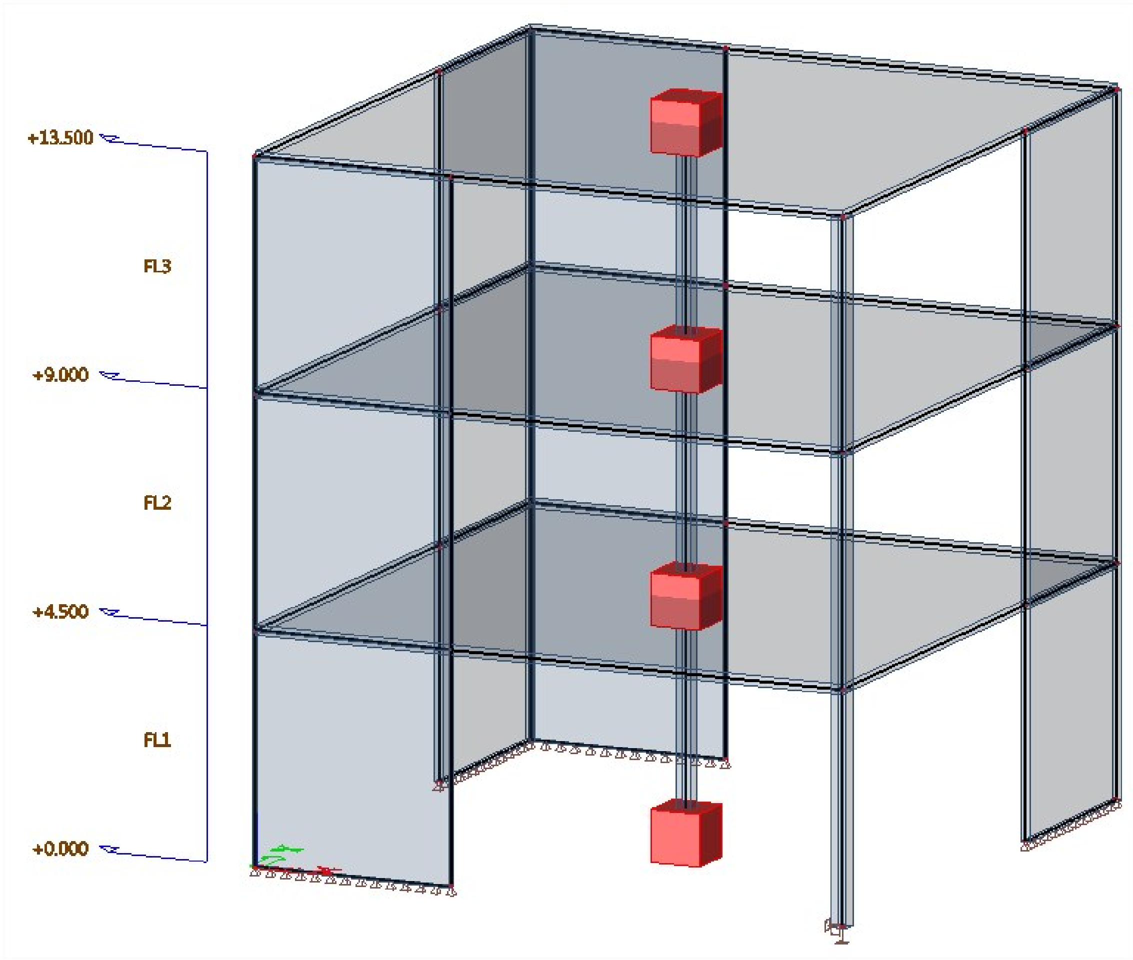
Optionally, R-nodes may be placed at any level in each storey. The storey property “level of reduction point” allows to select the exact height of the reduction point for each storey separately. 0 corresponds to the bottom of the storey, 1 to the top of the storey.



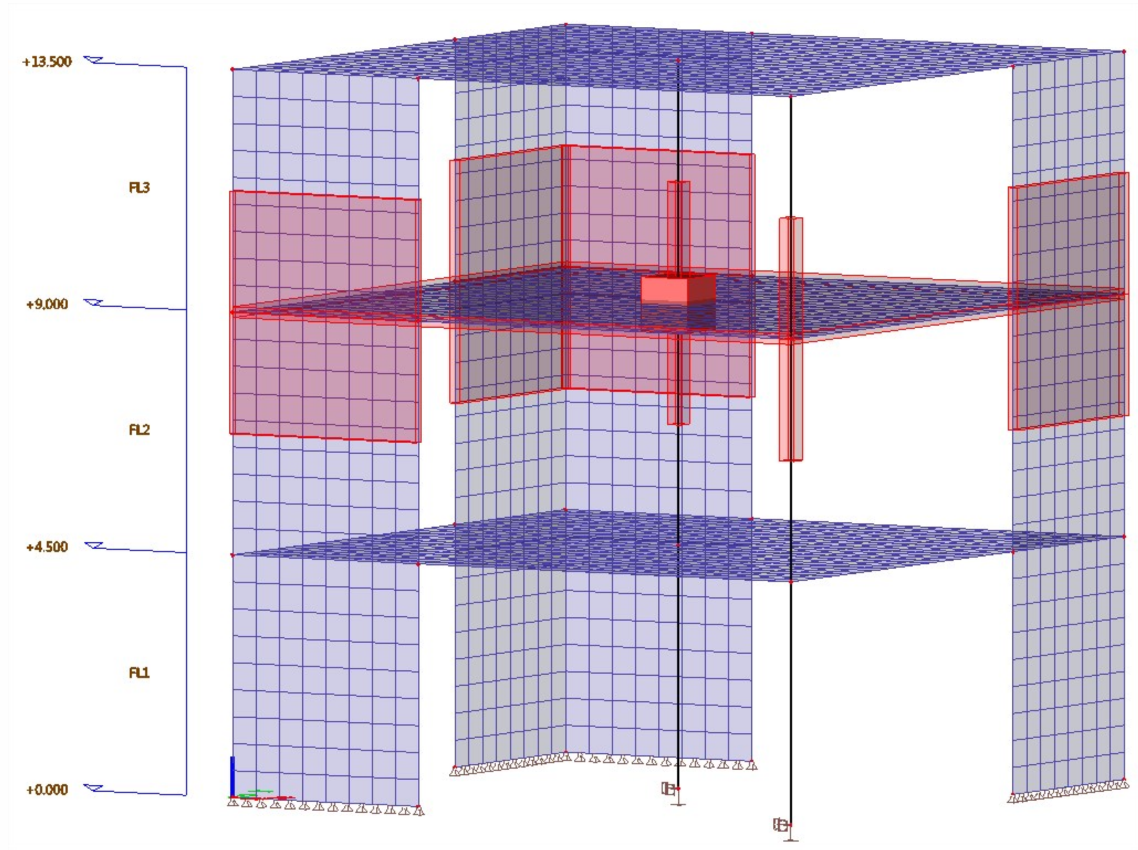


With the default settings (see above), the deck slab of each storey is located at the bottom of the storey, and so is the corresponding R-node. It is recommended to use those default settings.

As described before, R-nodes are placed in each storey, at the specified level, in the middle of the structure (all R-nodes are located on the same vertical axis).



During the analysis, the reduced model is automatically generated from the full mesh of the structure. Each node of the full mesh is mapped to the closest R-node. In a typical building configuration, this means that each R-node will receive the stiffness, loads and masses from the corresponding deck slab, from the top half of the supporting members below the slab and from the bottom half of the supporting members above the slab.



The concept of storeys in SCIA Engineer implies, that each structural member is allocated to at least one storey. That information is relevant for detailed storey results. However, **it does not affect the reduced system**. The nodes of the full mesh are mapped to the R-nodes of the reduced system purely based on their location (coordinates).

Running the analysis & Results output

Once the data has been prepared, the analysis is run exactly in the same way as for a standard analysis.

In SCIA Engineer, the IRS analysis will be used only for the calculation of the eigenmodes of the structure. On the basis of those eigenmodes, the actual calculation of seismic load cases will be performed on the original, full mesh.

The use of the IRS analysis appears in the calculation protocol:

Calculation protocol

Solution of Free Vibration

Number of 2D elements	28781
Number of 1D elements	778
Number of mesh nodes	25498
Number of equations	152988
Combination of mass groups	MC 1 exc0
Number of frequencies	48
Method	Lanczos
Bending theory	Mindlin
Type of analysis model	Standard using improved reduced system (IRS)
Start of calculation	12.04.2013 17:07
End of calculation	12.04.2013 17:07

There are fundamentally two types of results available after an IRS analysis:

- The results of the reduced model are automatically expanded to the original mesh and are accessible through standard output. This will not be detailed here.
- Some dedicated results, coming directly from the reduced model, are available in the result service Summary Storey Results. This typically gives information about the masses, displacements and accelerations at the mass center each storey in the reduced model.

See information about typical solver error when using IRS with a model that is not properly configured: non-associated R-node.

IRS: Too many eigen values requested / Non-associated R-node detected

Issue

When running IRS dynamic analysis, the calculation fails with the error message:

IRS modal reduction: Detect of nonassociated r-node

or

IRS modal reduction: More then computed eigen values are demand

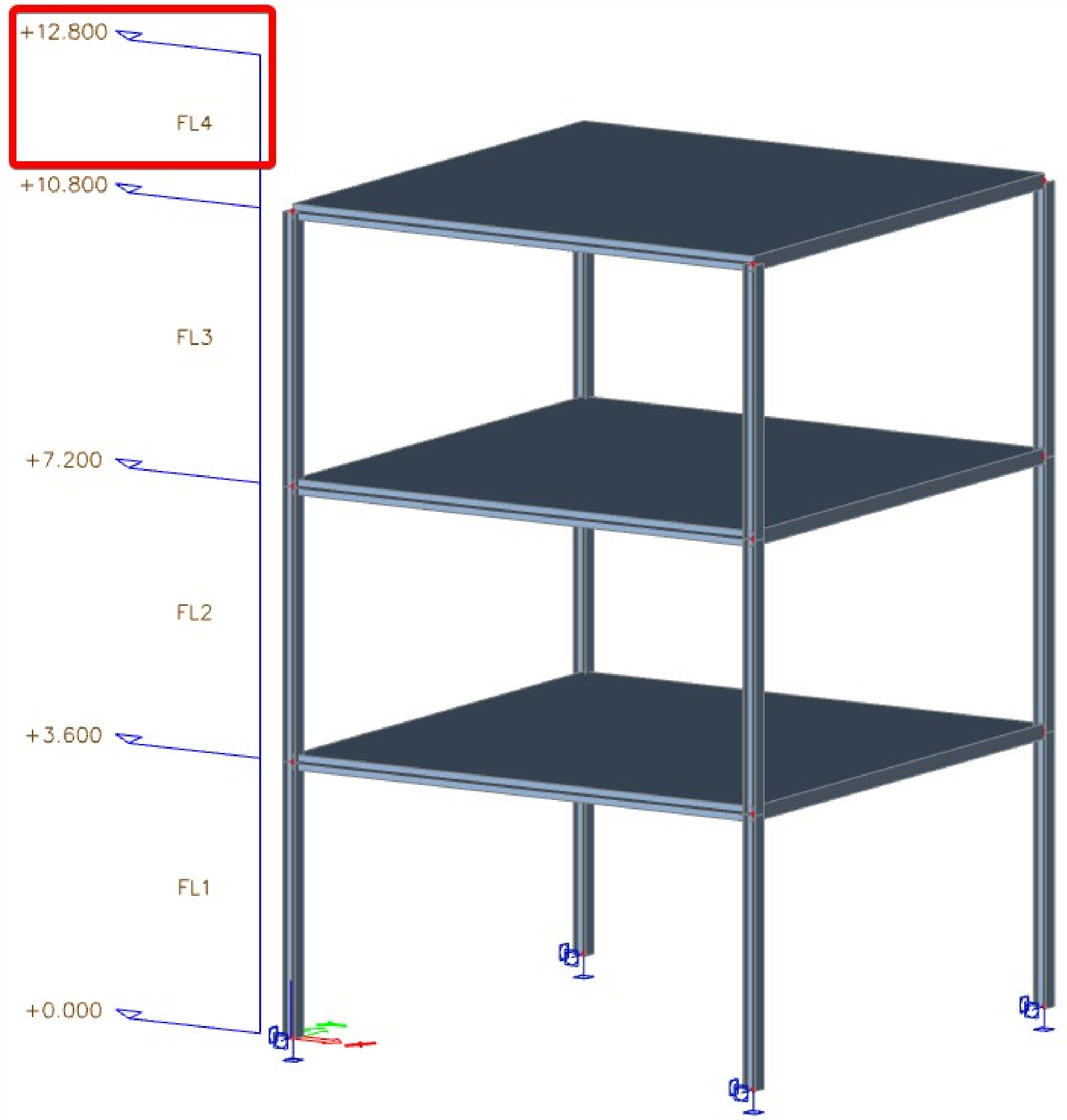
Meaning

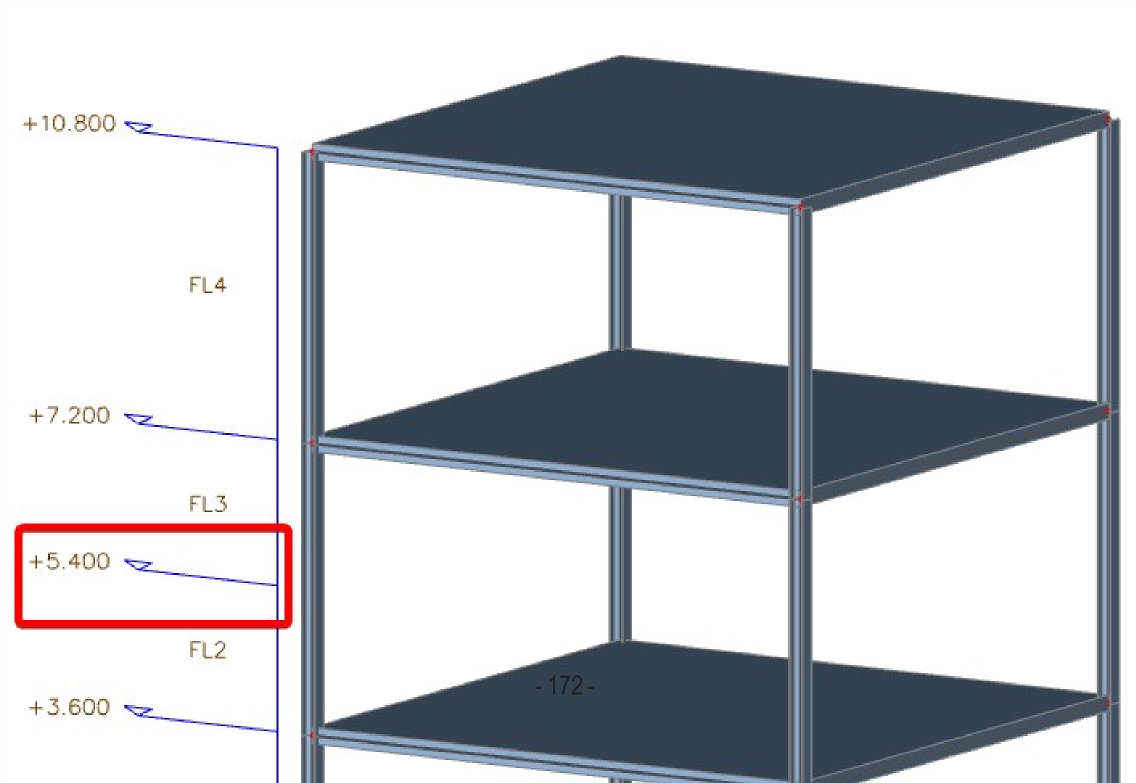
During the generation of the IRS analysis model, the solver detected one or more degree of freedom of some reduction point (R-node) without any mass associated to it. It means that all the nodes of the original finite element mesh that are linked to that R-node have a zero mass value for the component corresponding to that degree of freedom.

Solution

This can happen in several cases. The solution depends on the root cause.

Bad definition of storeys





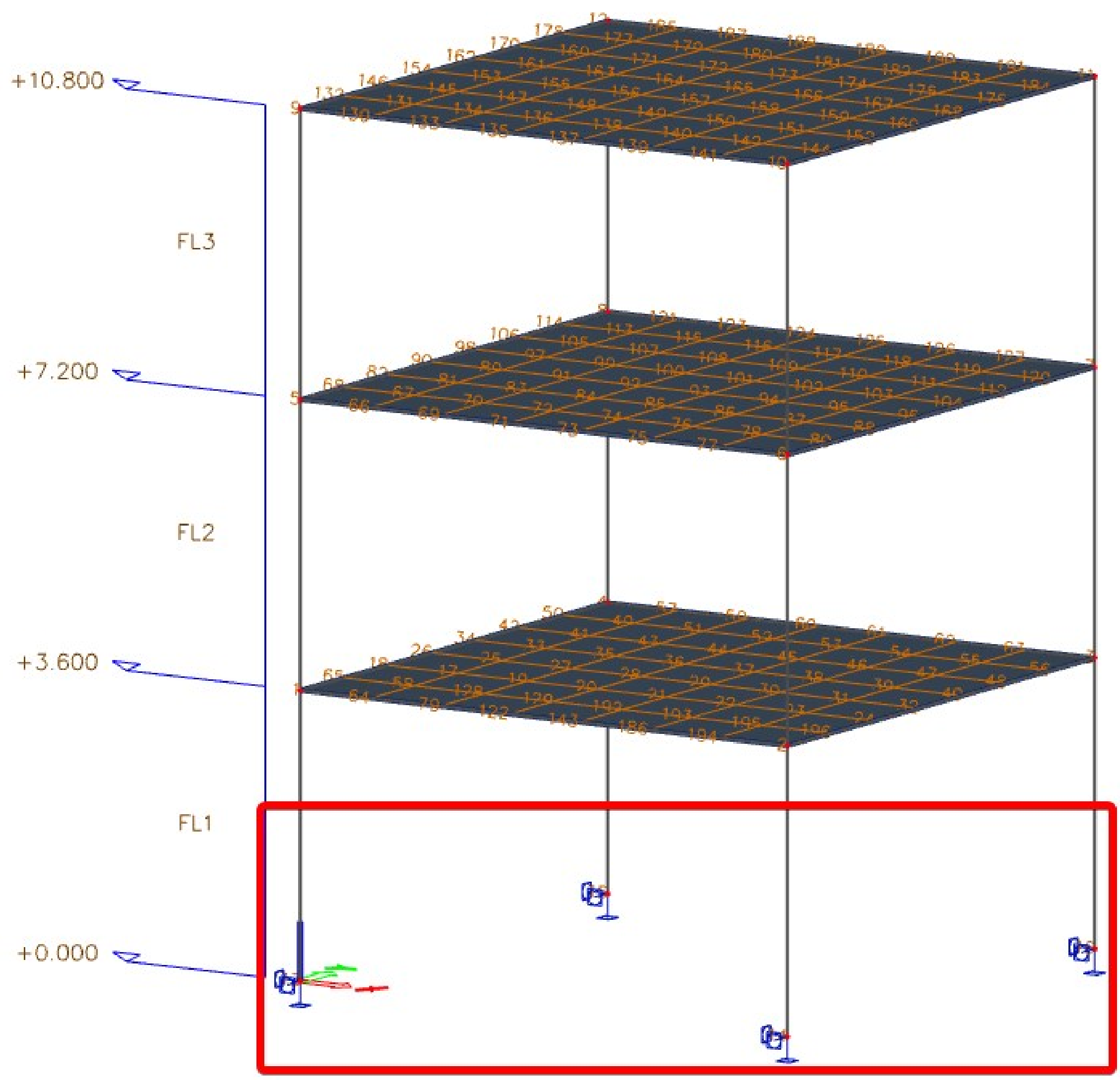
The storeys are defined in such a way, that there is no available mesh node at the level of some storey.

Each node of the finite element mesh is assigned to the closest R-node, i.e. to the closest storey level. In both examples shown above, the highlighted level has no mesh node assigned to it, because for any mesh node, there is another level that is closer.

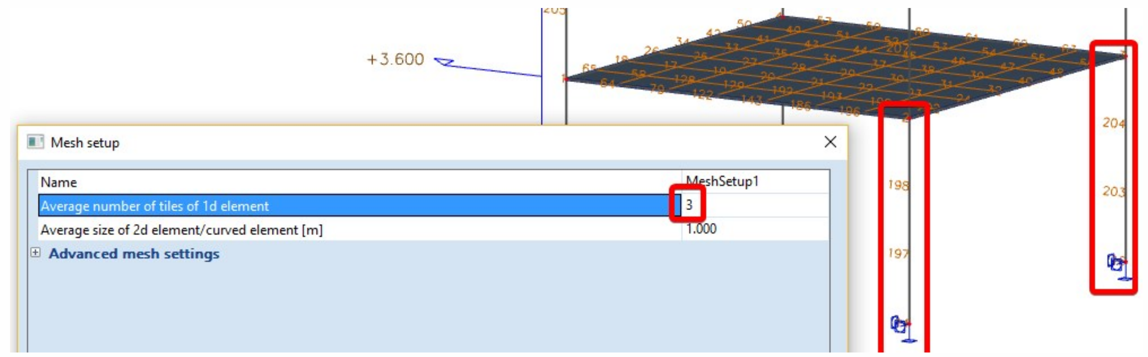
Solution: remove useless storeys

No moveable node is associated to a R-node

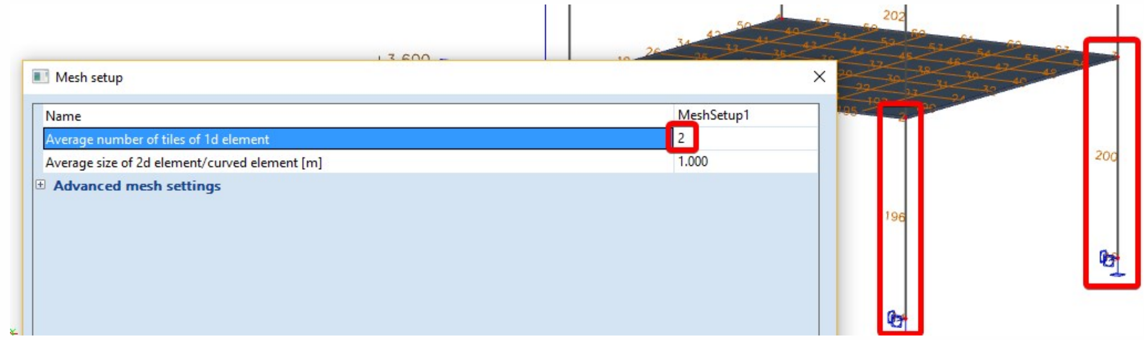
This typically happens at foundation level, where all nodes are fully blocked by a support. If the finite element mesh is not refined, all the nodes that are associated to the bottom R-node are fully blocked and are removed from the analysis model, leading again to a R_node without any associated mesh node,



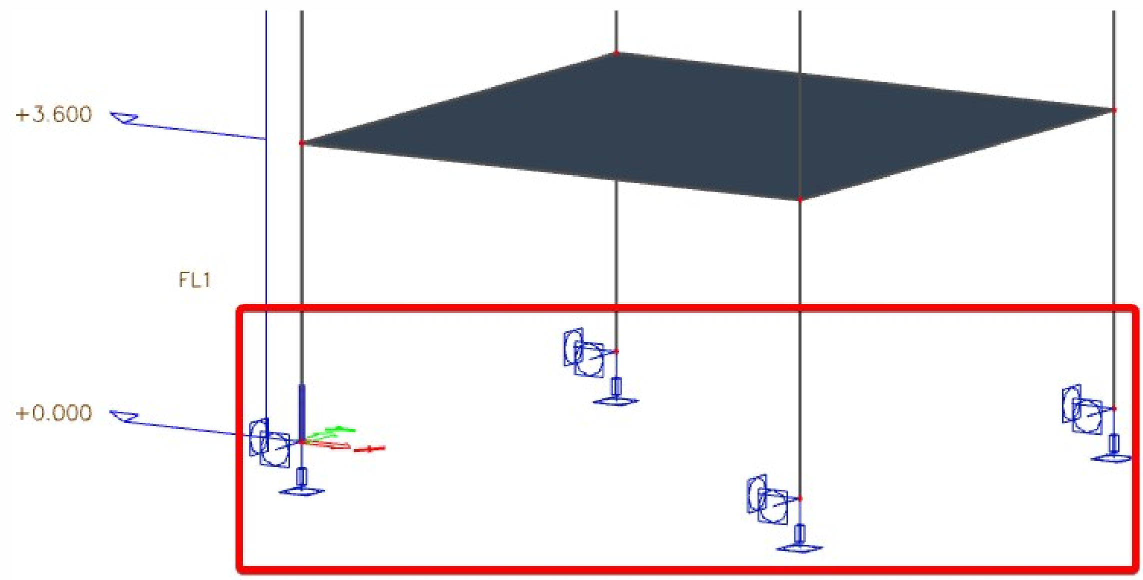
Solution: refine the mesh, if possible only on the members of the bottom storey.



Dividing the columns in two parts, thus inserting just one mesh node at mid-height, might not be sufficient, as the mid-node might be assigned either to the storey above or below it, depending on numerical sensitivity. To be safe, use a subdivision of at least 3.

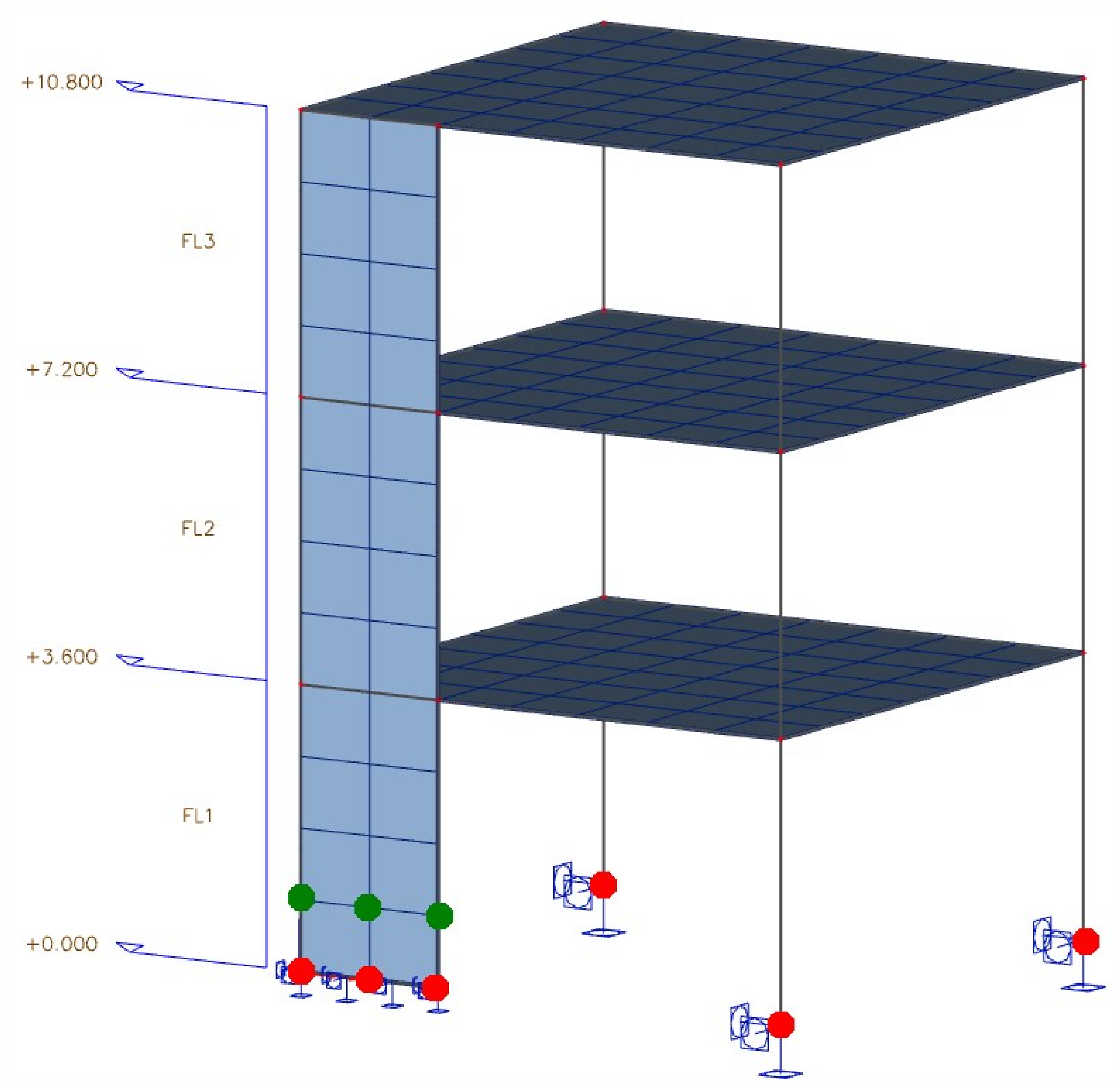


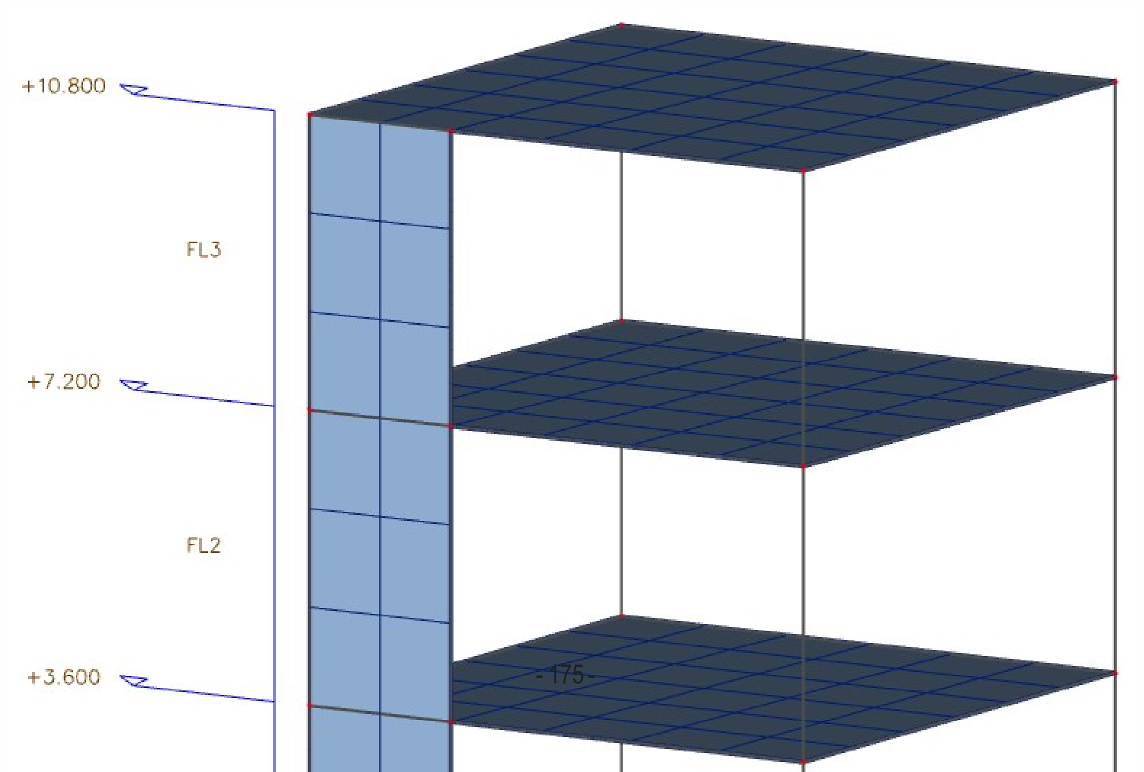
Having partially blocked supports could also lead to a similar situation. For instance, if all supports have a flexible vertical component, but all horizontal degrees of freedom blocked, the horizontal mass components will remain non-associated. The solution is also the same.



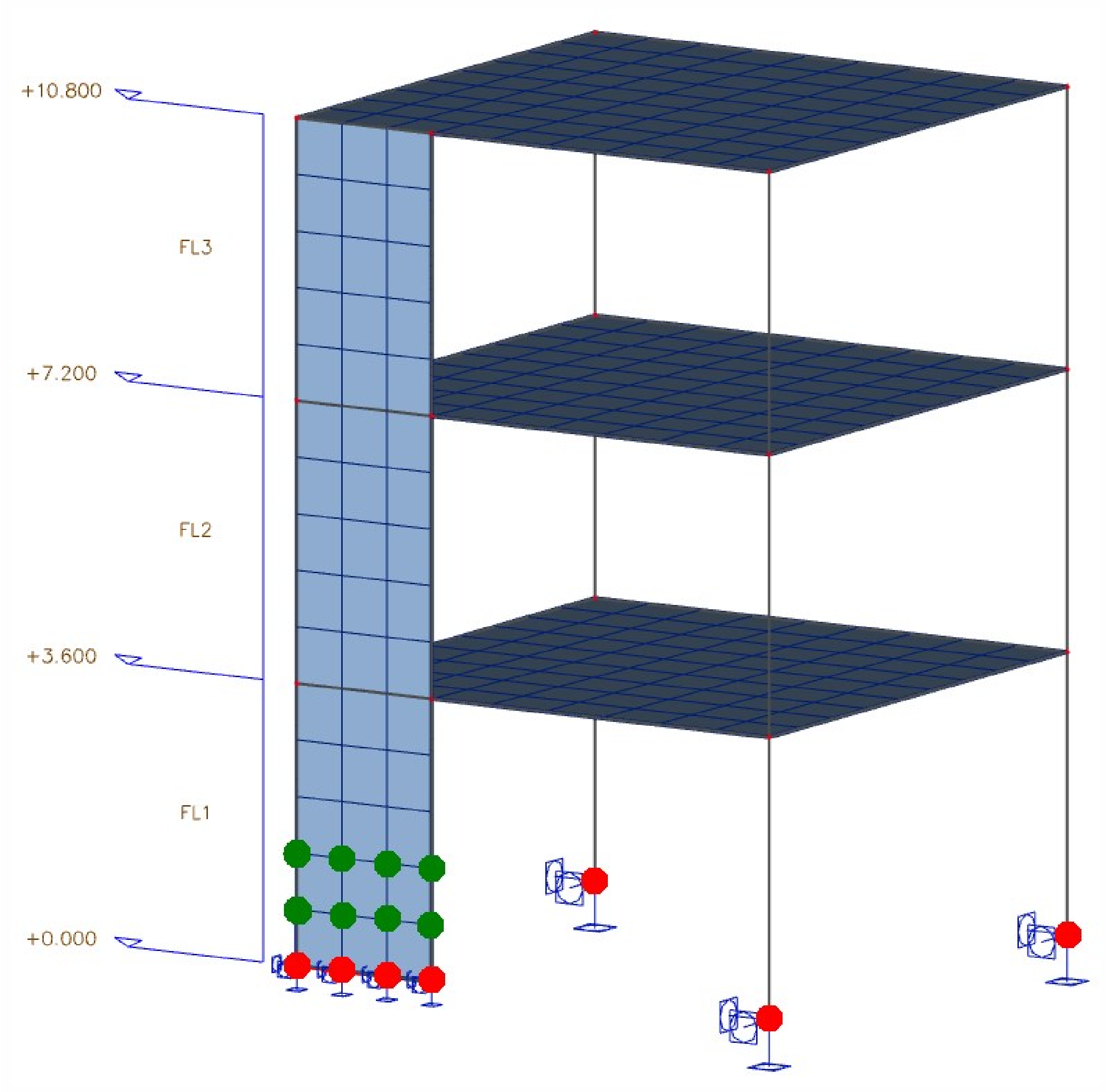
Insufficient rotational inertia of associated nodes on 2D members

In some cases, usually quite simple models, it is possible that some rotational component of the reduced mass matrix is zero. This can happen, for instance, when all movable mesh nodes of the bottom level are on a single line (see green nodes in pictures below).



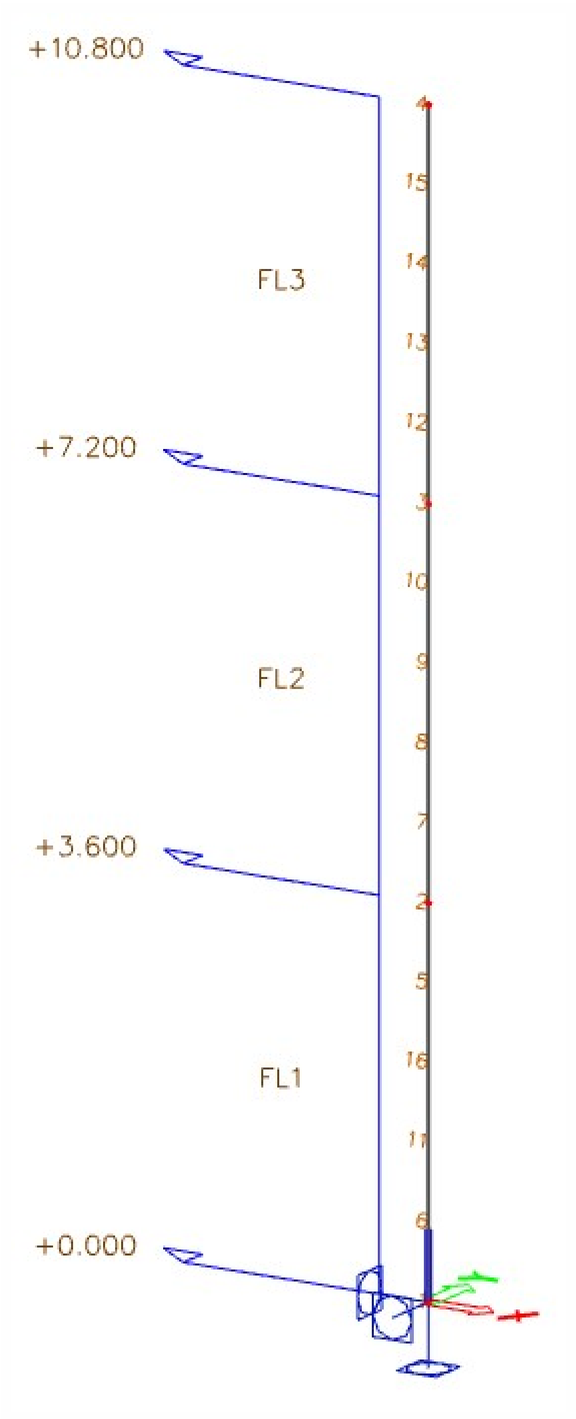


Solution: most of the time, refining the mesh of 2D members at the bottom level will solve the issue, by including a 2nd row of mesh nodes, thus ensuring a rotation inertia of the group of nodes.



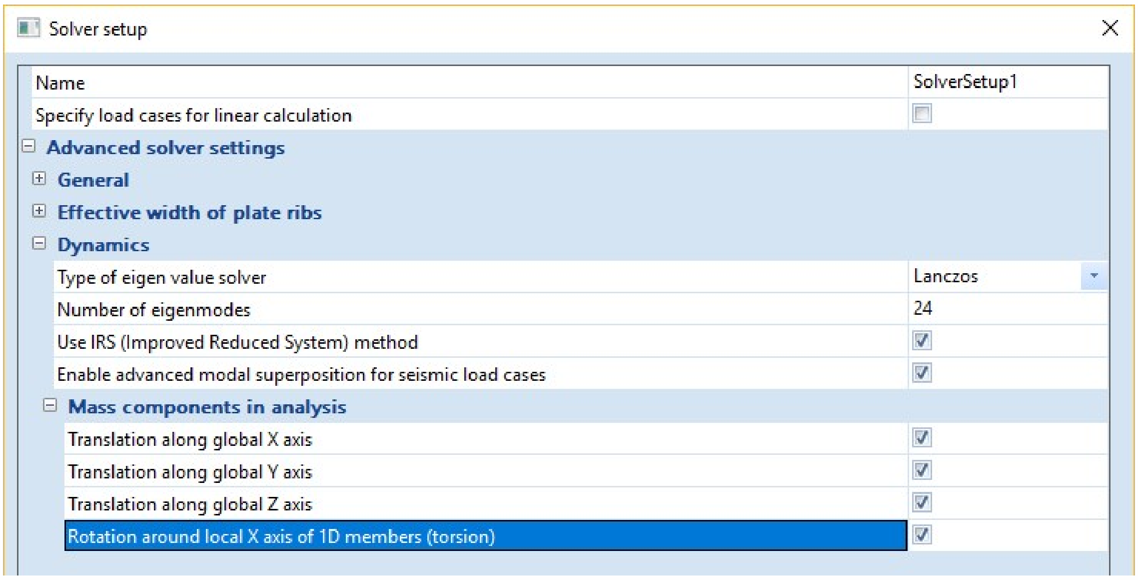
Insufficient rotational inertia of associated nodes on 1D members

By default, SCIA Engineer ignores the rotational inertia of 1D members around their axis. The reason for that is, that that mass component is most of the time useless in 3D systems, as the rotational inertia components of the system are mostly due to the lever arm between the nodal masses of the structure. There are, however, structures where that torsional behaviour is a significant of the dynamic behaviour.



In the simple example above, although the mesh of 1D members is sufficiently refined, the IRS analysis fails, because of the total absence of rotational inertia of all column members.

Solution: enable the *Rotation around local X axis of 1D members* option in the *solver setup*.



Numerical singularity and numerical divergence

When some mass component is zero in the resolution for eigen values, the analysis normally fails because of a division by zero during the process. This is typically what happens when the solver returns a message about a non-associated R-node.

In some cases, however, the solver attempts to provide a meaningful solution by replacing zero components with very small values. In the case of dynamic modal analysis, this allows to obtain reliable results for load frequencies. On the other hand, it usually generates very high frequency, irrelevant modes. Those can simply be ignored.

Relative modal masses

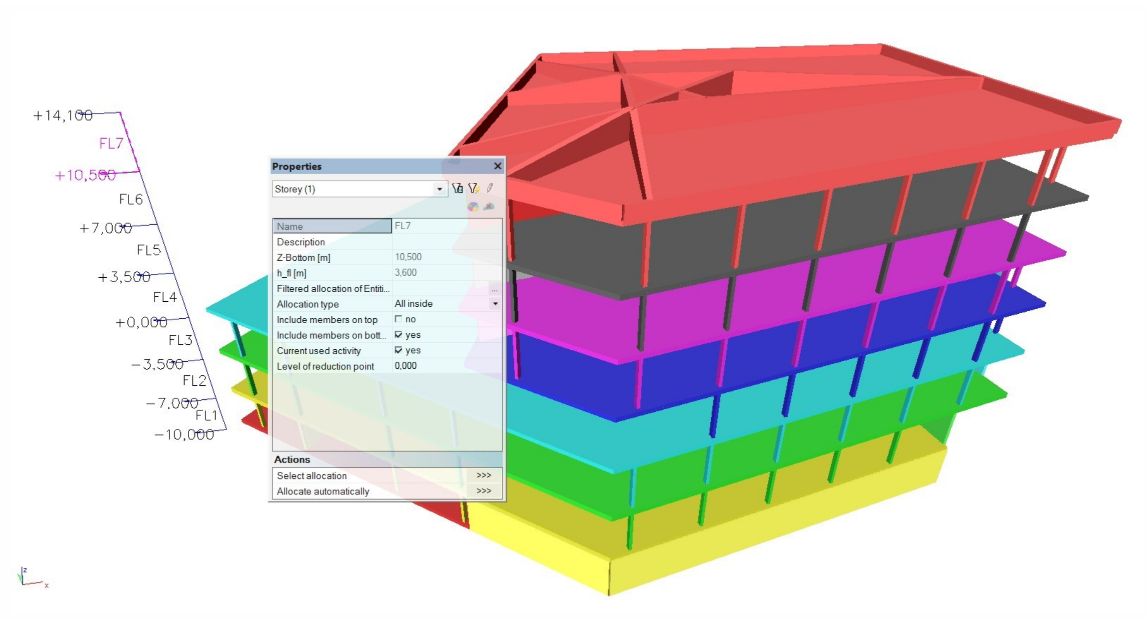
Mode	mega [rad/:	Period [s]	Freq. [Hz]	W_{xi}/W_{xtot}	W_{yi}/W_{ytot}	W_{zi}/W_{ztot}	W_{xi_R}/W_{xtot_l}	W_{yi_R}/W_{ytot_l}	W_{zi_R}/W_{ztot_l}
1	17.4205	0.36	2.77	0	0.643055	0	0.350941	0	0
2	28.8274	0.22	4.59	0.64435	0	0	0	0.349772	0
3	107.713	0.06	17.14	0	0.198931	0	0.262555	0	0
4	175.38	0.04	27.91	0.201642	0	0	0	0.270038	0
5	297.452	0.02	47.34	0	0.0683869	0	0.142922	0	0
6	472.251	0.01	75.16	0.0699763	0	0	0	0.148818	0
7	591.285	0.01	94.11	0	0.0369122	0	0.0923715	0	0
8	911.993	0.01	145.15	0.0389357	0	0	0	0.0996571	0
9	951.588	0.01	151.45	0	0.024944	0	0.0680887	0	0
10	1127.56	0.01	179.46	0	0	0.850805	0	0	0
11	1395.39	0.00	222.08	0.0235828	0	0	0	0.0658947	0
12	1989.17	0.00	316.59	0	0.0275541	0	0.0824139	0	0
13	2516.51	0.00	400.51	0.0215125	0	0	0	0.065816	0
14	3412.31	0.00	543.09	0	0	0.0883127	0	0	0
15	6029.74	0.00	959.66	0	0	0.0531712	0	0	0
16	1e+10	0.00	591549430.92	0	0	0	0	0	0
17	1e+10	0.00	591549430.92	0	0	0	0	0	0
18	1e+10	0.00	591549430.92	0	0	0	0	0	0
				0.999999	0.999783	0.992289	0.999293	0.999996	0

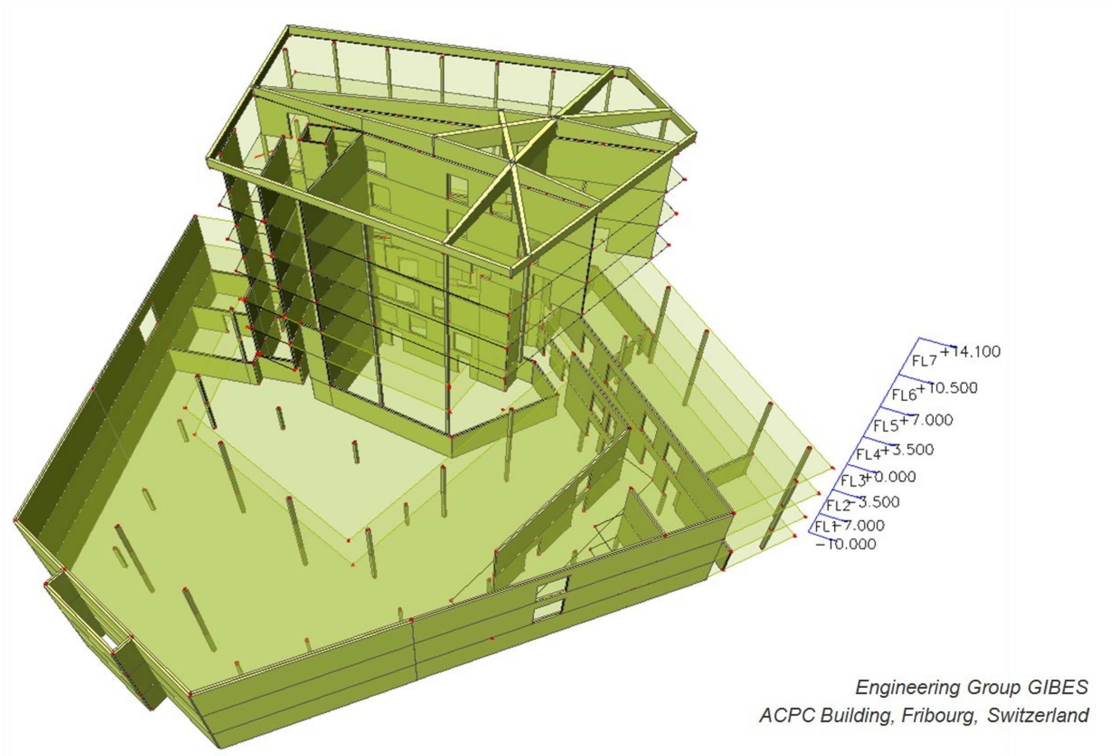
In such a case, preferably reduce the requested number of modes to avoid irrelevant modes.

Storey Results

For a detailed description of storey results and their usage, see the following pages:

- Summary Storey Results
- Detailed Storey Results





Engineering Group GIBES
ACPC Building, Fribourg, Switzerland

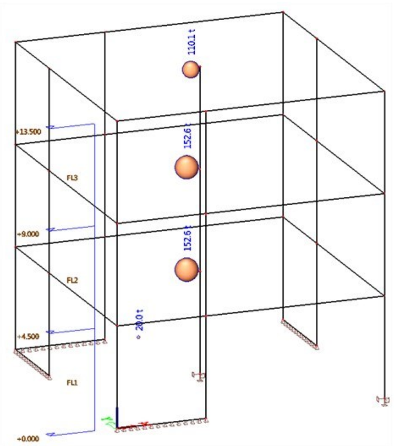
Summary storey results

The service "Summary Storey Results" provides results directly produced by the IRS analysis (see Reduced analysis model). At this time, this service is dedicated to result output for the seismic analysis of buildings. It provides single results per storey, such as mass, position of mass center, displacement, acceleration…

> Summary storey results are available only when the reduced modal analysis is enabled and storey are defined.

Types of results:

For mass combinations	Storey data: mass & mass center of each storey Displacements of storey mass center per mode Accelerations of storey mass center per mode
For seismic load cases	Displacements of storey mass center Accelerations of storey mass center Inter-storey drift at storey mass centers



Summary storey result

Storey Displacements:
Eigen solution, Extreme: No, System: Principal
Selection: All
Mass combinations : CM1/1 - 2.07

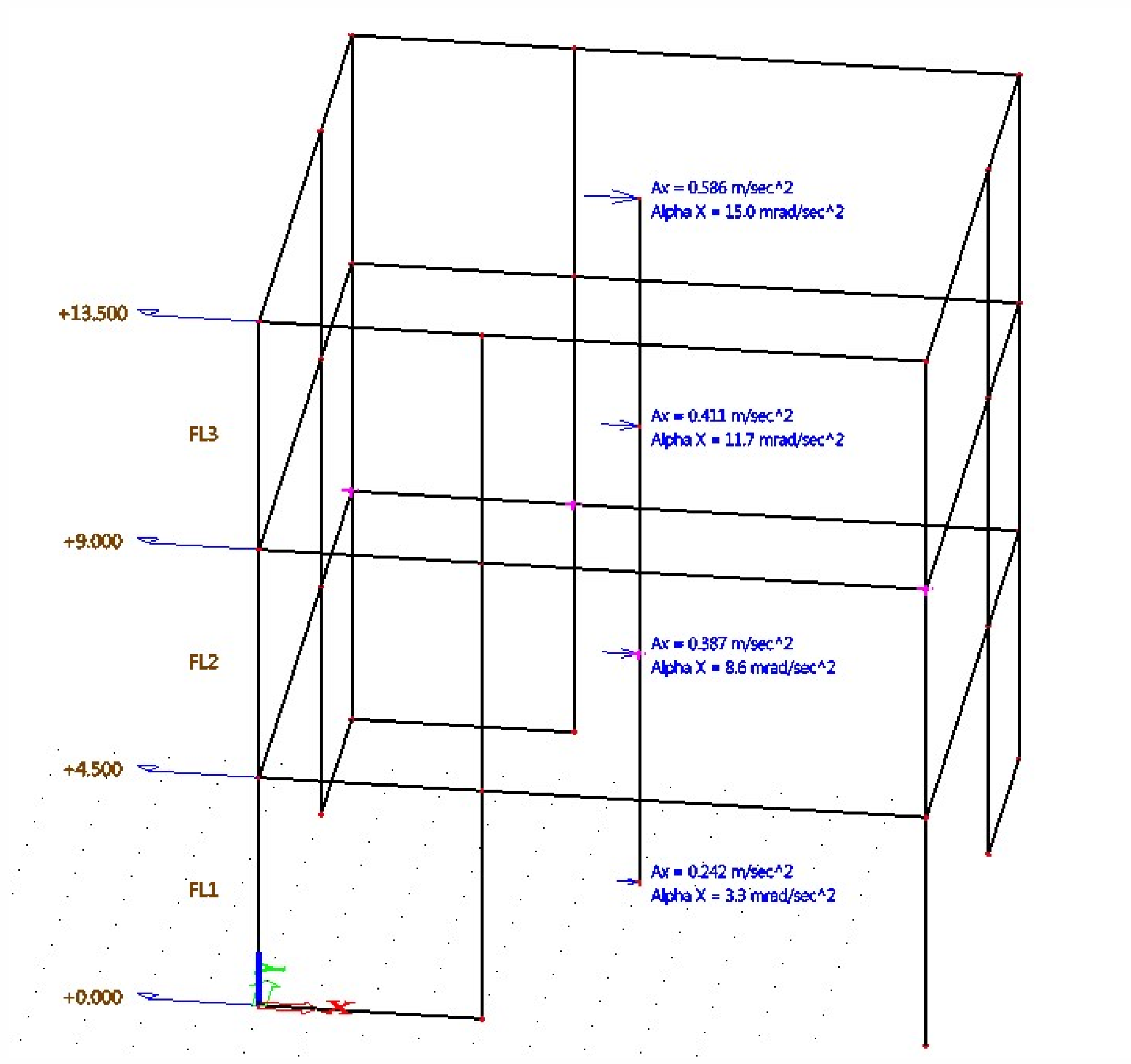
Name	Ux [mm]	Uy [mm]	Uz [mm]	Phix [mrad]	Phiy [mrad]	Phiz [mrad]
FL1	-6.1e-02	6.1e-02	0.0e+00	-2.0e-03	-2.0e-03	0.0e+00
FL2	-3.3e-01	3.3e-01	7.7e-02	-8.0e-03	-8.0e-03	0.0e+00
FL3	-9.9e-01	9.9e-01	1.1e-01	-1.1e-02	-1.1e-02	0.0e+00
FL4	-1.7e+00	1.7e+00	1.3e-01	-1.0e-02	-1.0e-02	0.0e+00

Summary storey result

Storey Accelerations:
Eigen solution, Extreme: No, System: Principal
Selection: All
Mass combinations : CM1/1 - 2.07

Name	Ax [m/sec^2]	Ay [m/sec^2]	Az [m/sec^2]	Alpha X [mrad/sec^2]	Alpha Y [mrad/sec^2]	Alpha Z [mrad/sec^2]
FL1	-0.010	0.010	0.000	-3.38e-01	-3.38e-01	0.00e+00
FL2	-0.056	0.056	0.013	-1.35e+00	-1.35e+00	0.00e+00
FL3	-0.167	0.167	0.019	-1.86e+00	-1.86e+00	0.00e+00
FL4	-0.291	0.291	0.022	-1.69e+00	-1.69e+00	0.00e+00

Picture: Summary storey results



Picture: Accelerations

Detailed Storey Results

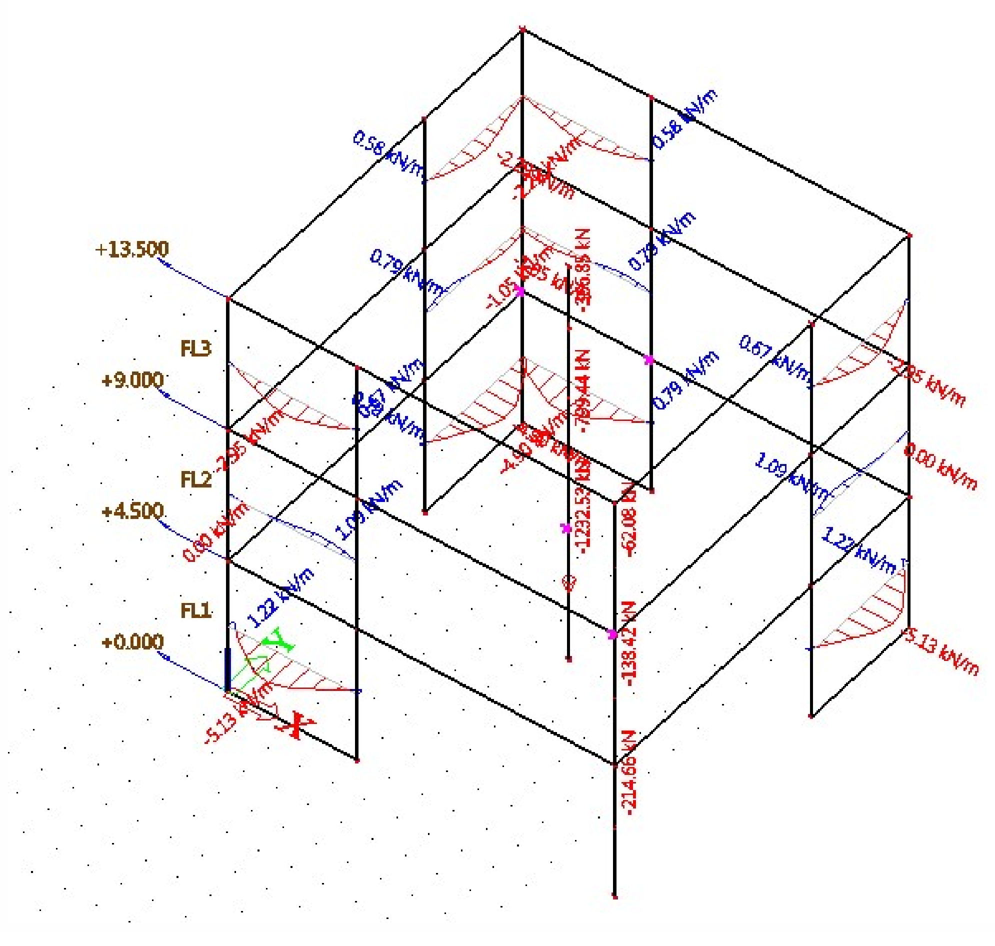
The service "Detailed Storey Results" provides results from the full mesh analysis. At this time, it may be used for results from any linear analysis, with or without dynamic analysis, with or without IRS analysis. It provides results in all supporting members, with easy selection of members per storey. Walls and columns may be represented on the same drawing. Typical provided results are: internal forces, resultants per wall or per storey…

> Tip: before using detailed storey results, make sure that all supporting members of the building are properly allocated to storeys. That information is essential for proper handling of storey results.

Type of results

Internal forces in supporting members	Selection by storey Extreme by member (also for walls !) Walls & columns on the same drawing

	Simple choice of section level Display of average & total value for walls Available for static & seismic results Also suitable e.g. for load descending
Resultant forces in supporting members	Resultant for each wall on the same drawing Clear display of more components + all the key points mentioned previously
Resultant forces in supporting members per storey	Resultant of all supporting members at once (walls + columns combined) + all the key points mentioned previously



Picture: Internal forces

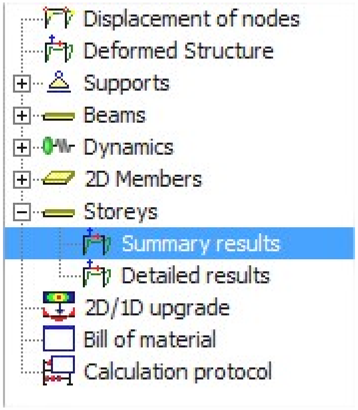
Summary Storey Results

This service provides results directly produced by the IRS analysis (see Reduced analysis model). At this time, this service is dedicated to result output for the seismic analysis of buildings. It provides single results per storey, such as mass, position of mass center, displacement, acceleration…

Pre-requisites for using Summary Storey Results:

- **storeys must be defined**
- **the reduced analysis model must be enabled in the solver settings**

The service may be found in the Results service. It is available only after a successful dynamic analysis using the reduced model.



Output Settings

The output settings are presented here according to the various types of results that can be obtained from this service. There are 4 types of results:

- Storey Data: information of the reduced analysis model, such as the mass and the position of the mass center of each storey
- Displacements: displacements at the mass center of each storey
- Accelerations: accelerations at the mass center of each storey
- Inter-storey drift: relative displacements at the mass center of each storey, relative to the storey immediately below

Result type Storey Data

For mass combinations, it displays for each storey the total mass and the coordinates of the mass center.

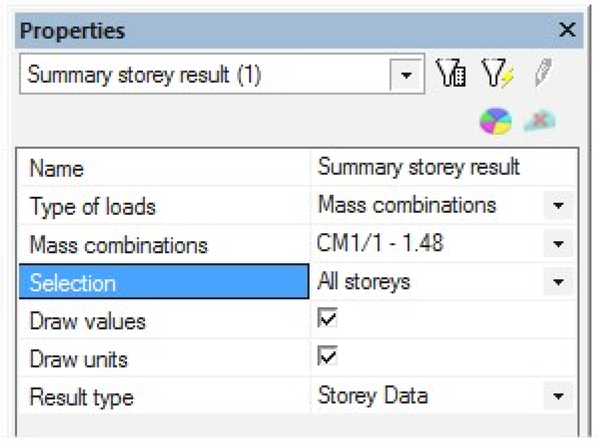
For ELF seismic load cases, it displays summary information about computed storey forces.

Type of loads
selection of the type of load

Mass combinations
selection of the combination of mass groups or eigenmode for a given combination of mass groups.

Selection
type of selection; the possible choices are

- All storeys
- Named selection
- Single storey



Named selection
named selection that contains a set of storeys; only if Selection = Named selection

Storey
dropdown menu for selection of a single one; only if Selection = Single storey

Draw values
tick to draw values on the drawing

Draw units
tick to draw values with their units on the drawing

Result type
types of results; possible choices are

- Displacements
- Inter-storey drift
- Accelerations
- Storey Data

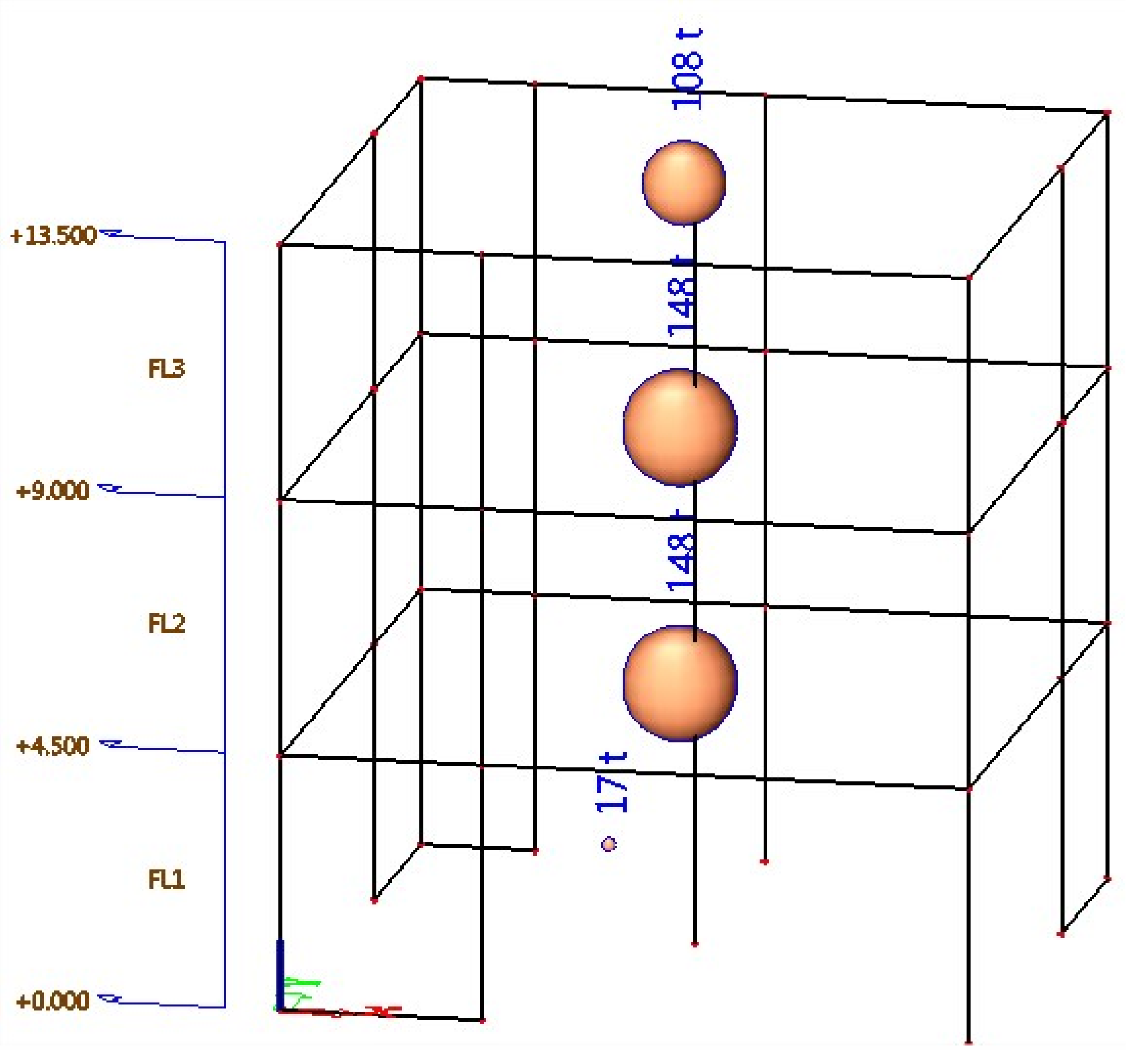
Values
main component to be displayed; *only when an ELF seismic load case is selected*

Fx, Fy, Fz : components of the seismic storey forces

Ftot : resultant seismic force per storey

M : mass of each storey

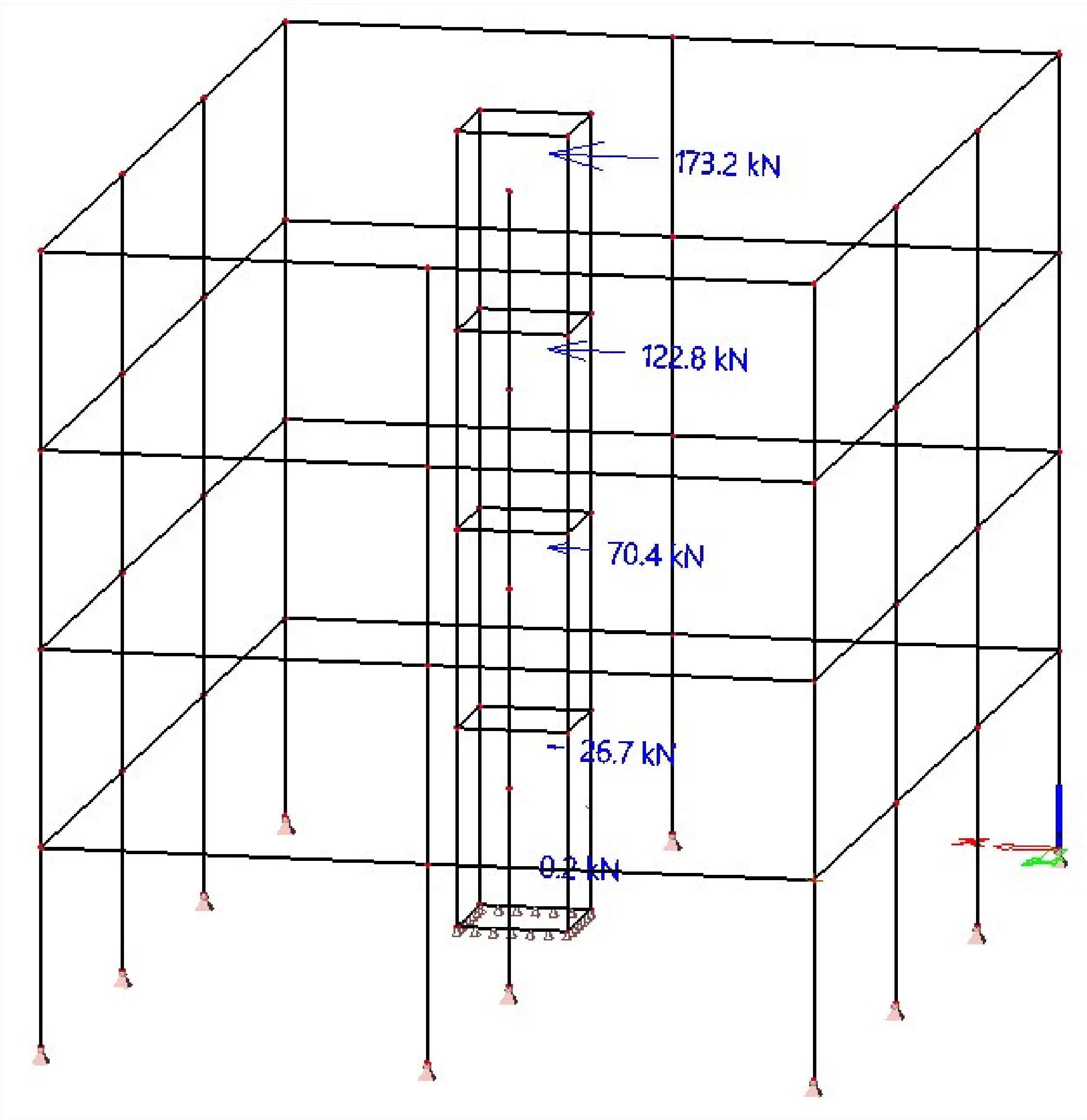
Additional values
allow to show more than one result component simultaneously on the drawing.



Summary storey result

Storey Storey Data:
Eigen solution, Extreme: No, System: Principal
Selection: All
Mass combinations : CM1/1 - 1.48

Name	M [t]	XG [m]	YG [m]	ZG [m]
FL1	17	4.157	7.704	1.250
FL2	148	5.663	6.350	4.500
FL3	148	5.663	6.350	9.000
FL4	108	5.769	6.240	13.343



Summary storey result

Storey data:

Linear calculation, Extreme: No, System: Principal

Selection: All

Load cases : EQX1

Equivalent Lateral Forces (ELF) settings

ELF method	Polynomial distribution of accelerations (ASCE 7-10 12.8.3)
Seismic force from	Selected eigenmode
Fundamental period [s]	1.28
Distribution factor k	1.39
Mode shape	1

Equivalent Lateral Forces (ELF) per storey

Name	M [t]	Zg [m]	Fx [kN]
FL1	7.1	0.993277	**0.2**
FL2	165.9	3.603689	**26.7**
FL3	166.7	7.204320	**70.4**
FL4	165.5	10.800000	**122.8**

Content of the output table for mass combinations

In the case of storey data output for a mass combination, the following columns are displayed in the preview and table output, for each storey:

M: mass of each storey, taken as the max value of Mxx, Myy and Mzz

XG, YG, ZG: GCS-coordinates of the mass center of each storey

Mxx, Myy, Mzz: mass components of each storey, in X, Y, Z direction of the GCS

Ixx, Iyy, Izz: mass inertia components of each storey, around X, Y, Z axes of the GCS

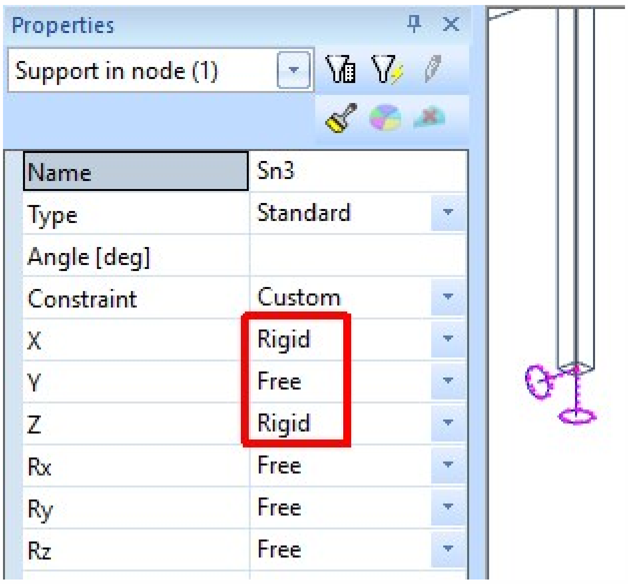
$$I_{xx} = \int \left((y - y_G)^2 + (z - z_G)^2 \right) dM$$

$$I_{yy} = \int \left((x - x_G)^2 + (z - z_G)^2 \right) dM$$

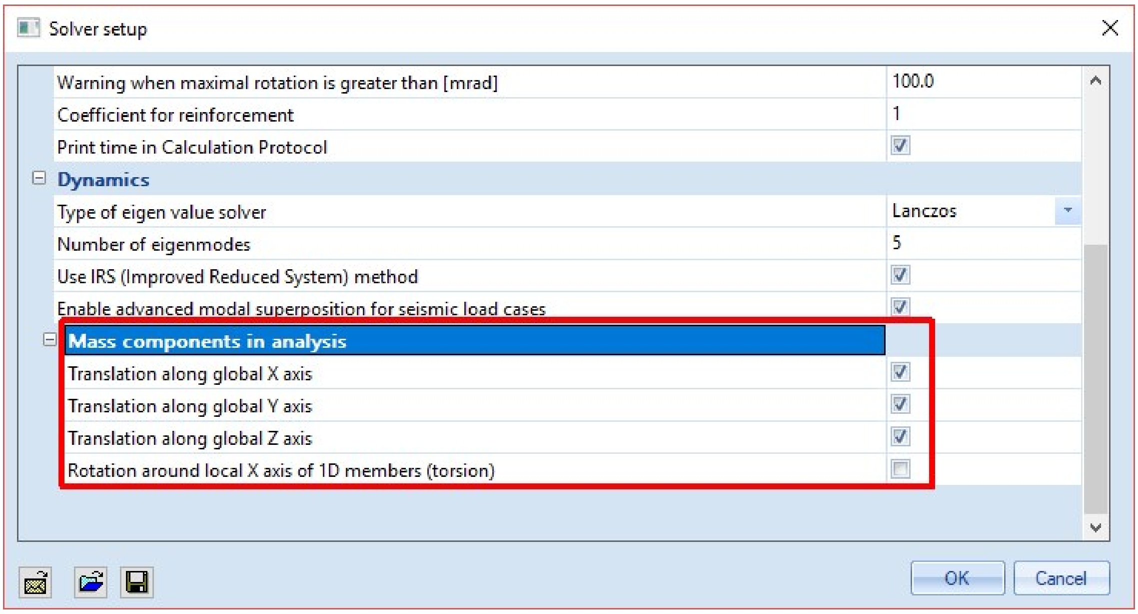
$$I_{zz} = \int \left((x - x_G)^2 + (y - y_G)^2 \right) dM$$

In most cases, M = Mxx = Myy = Mzz. However, they may differ in the following cases:

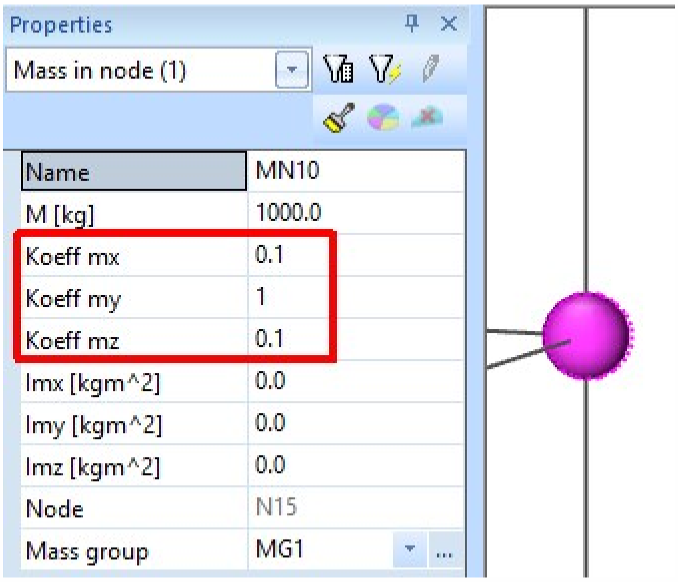
- non-isotropic boundary conditions: rigid supports with different properties in X, Y and Z direction and the supported structural entity has non-zero mass



- use of mass direction components (see Dynamics section in the solver setup)



- use of directional mass coefficients in mass definition



Result types Displacements & Accelerations

Displacements & Accelerations are available for eigenmodes and seismic load cases. The values of displacement & acceleration components are given at the mass center of each storey.

Results for mass combinations are raw, normalized results from modal analysis, without effect of response spectrum.

Results for seismic load cases are values after modal superposition.

Type of loads
selection of the type of load

- Mass combinations
- Load cases

Mass combination

list of available eigenmodes for each computed combination of mass groups; only if Type of loads = Mass combinations

Load cases
list of available seismic load cases; only if Type of loads = Load cases

Selection
type of selection; the possible choices are

- All storeys
- Named selection
- Single storey

Named selection
named selection that contains a set of storeys; only if Selection = Named selection

Storey
dropdown menu for selection of a single one; only if Selection = Single storey

Extreme
filter extreme results; possible choices are

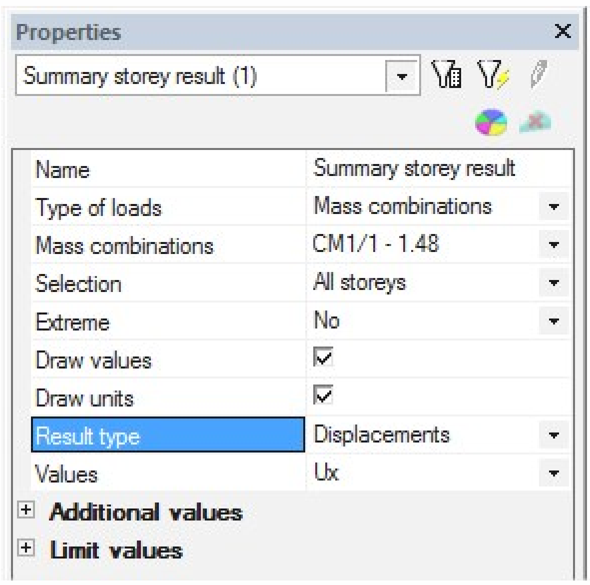
- No
- Section (relevant only for seismic load cases)
- Member (relevant only for seismic load cases)
- Global

Draw values
tick to draw values on the drawing

Draw units
tick to draw values with their units on the drawing

Result type
types of results; possible choices are

- Displacements
- Inter-storey drift
- Accelerations
- Storey Data



Values

main component to be displayed;

for Result type = Displacements

u_x, u_y, u_z : displacement at mass center according to GCS axes

φ_x, φ_y, φ_z : rotation at mass center around GCS axes

for Result type = Inter-storey drift

Δux, Δuy, Δuz : relative displacement at mass center according to GCS axes, relative to the storey immediately below

Δφ_x, Δφ_y, Δφ_z : relative rotation at mass center around GCS axes, relative to the storey immediately below

for Result type = Accelerations

Ax, Ay, Az : acceleration at mass center according to GCS axes

AlphaX, AlphaY, AlphaZ : rotational acceleration at mass center around GCS axes

Summary storey result

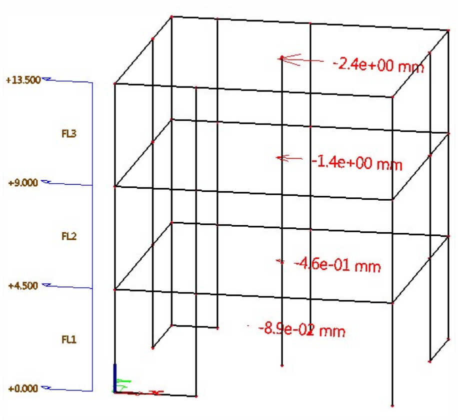
Storey Displacements:
Eigen solution, Extreme: No, System: Principal
Selection: All
Mass combinations : CM1/1 - 1.48

Name	Ux [mm]	Uy [mm]	Uz [mm]	Phix [mrad]	Phiy [mrad]	Phiz [mrad]
FL1	-8.9e-02	1.6e-02	-4.5e-04	-9.2e-04	-1.9e-03	1.9e-03
FL2	-4.6e-01	1.1e-01	4.3e-02	-7.6e-03	-6.5e-03	5.3e-03
FL3	-1.4e+00	3.5e-01	6.3e-02	-1.1e-02	-9.5e-03	1.3e-02
FL4	-2.4e+00	6.2e-01	7.1e-02	-1.2e-02	-8.2e-03	1.9e-02

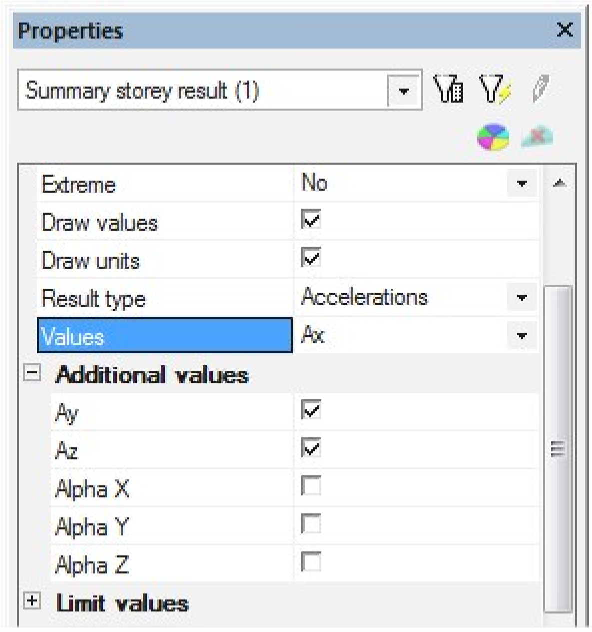
Additional values

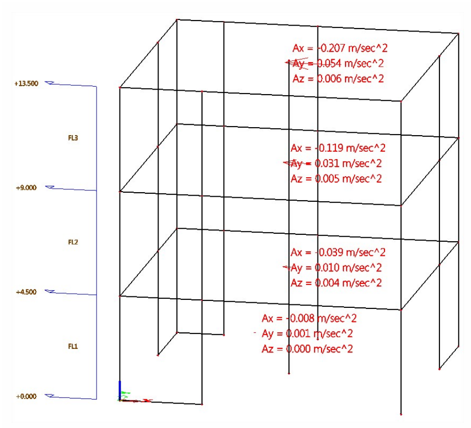
allow to show more than one result component simultaneously on the drawing.

When only the main component is selected (no additional value ticked), the selected component is drawn in the corresponding direction.



When one or more additional value is ticked, all selected components are listed in the plane of the screen for better readability.





Limit values

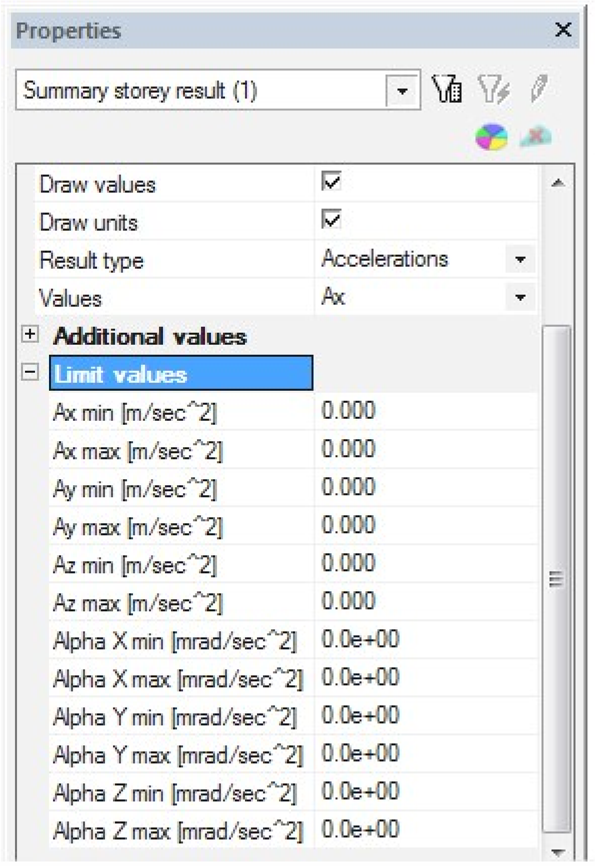
for each result component, definition of min and max value for colour coding on the drawing. Colours may be configured in Settings > Colours/Lines.

Default colour coding is as follows:

Values lower than V_{min} are display in red

Values between V_{min} and V_{max} are displayed in gray

Values greater than V_{max} are displayed in blue



Detailed Storey Results

This service provides results from the full mesh analysis. It may be used for results from any linear analysis, with or without dynamic analysis, with or without IRS analysis. It provides results in all supporting members, with easy selection of members per storey. Walls and columns may be represented on the same drawing.

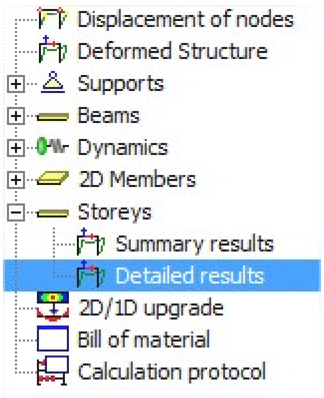
Typical provided results are: internal forces, resultant forces per wall or per storey…

Pre-requisites for using Summary Storey Results:

- **storeys must be defined**
- **supporting members must be properly allocated to storeys**

> Before using detailed storey results, make sure that all supporting members of the building are properly allocated to storeys. That information is essential for proper handling of storey results.

The service may be found in the Results service. It is available only after a successful analysis, if storeys are defined.



Output Settings

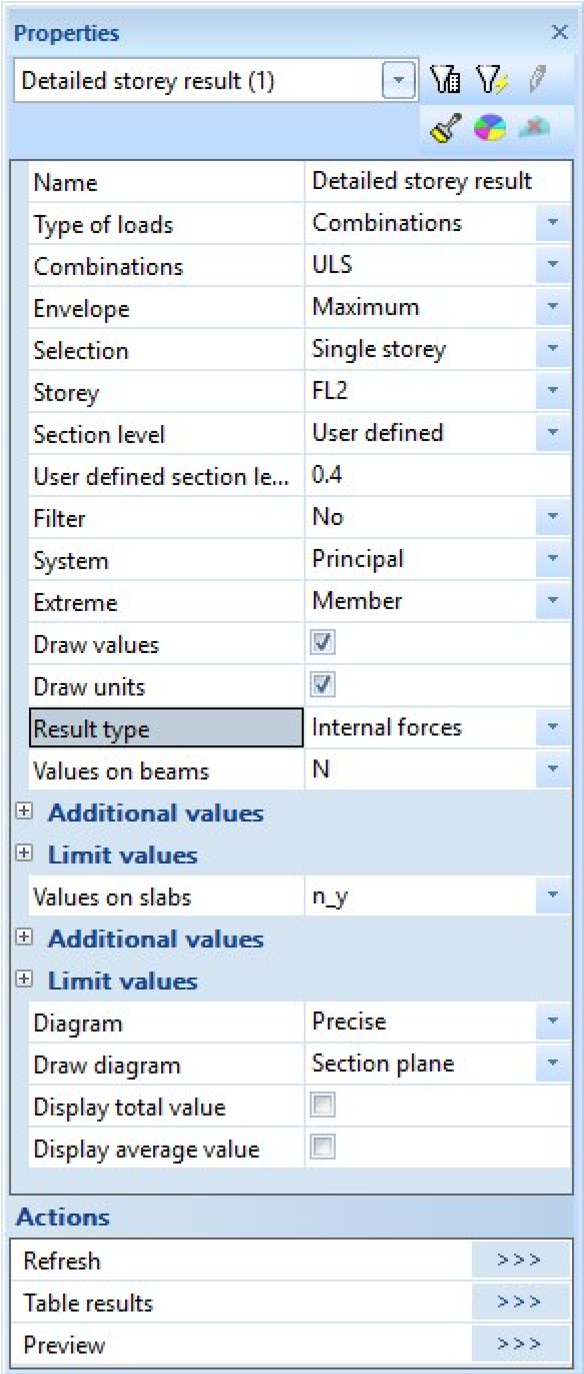
Mainly 2 types of results are available in this service:

- Internal forces in supporting members
- Resulting forces

For resulting forces, the Member grouping may be selected:

- per member: compute the resulting forces for each supporting member separately
- per storey: compute the resulting forces for each entire storey at once, combining 1D and 2D members

Result type Internal forces



Type of loads
selection of the type of load

- Load cases
- Combinations
- Class

Load cases
list of available load cases; only if Type of loads = Load cases

Combinations
list available load case combinations; only if Type of loads = Combinations

Class
list of available result classes; only if Type of loads = Class

Envelope
Maximum / Minimum; select the maximum or minimum value of an envelope; only if Type of loads = Combinations or Class

Selection
type of selection; the possible choices are

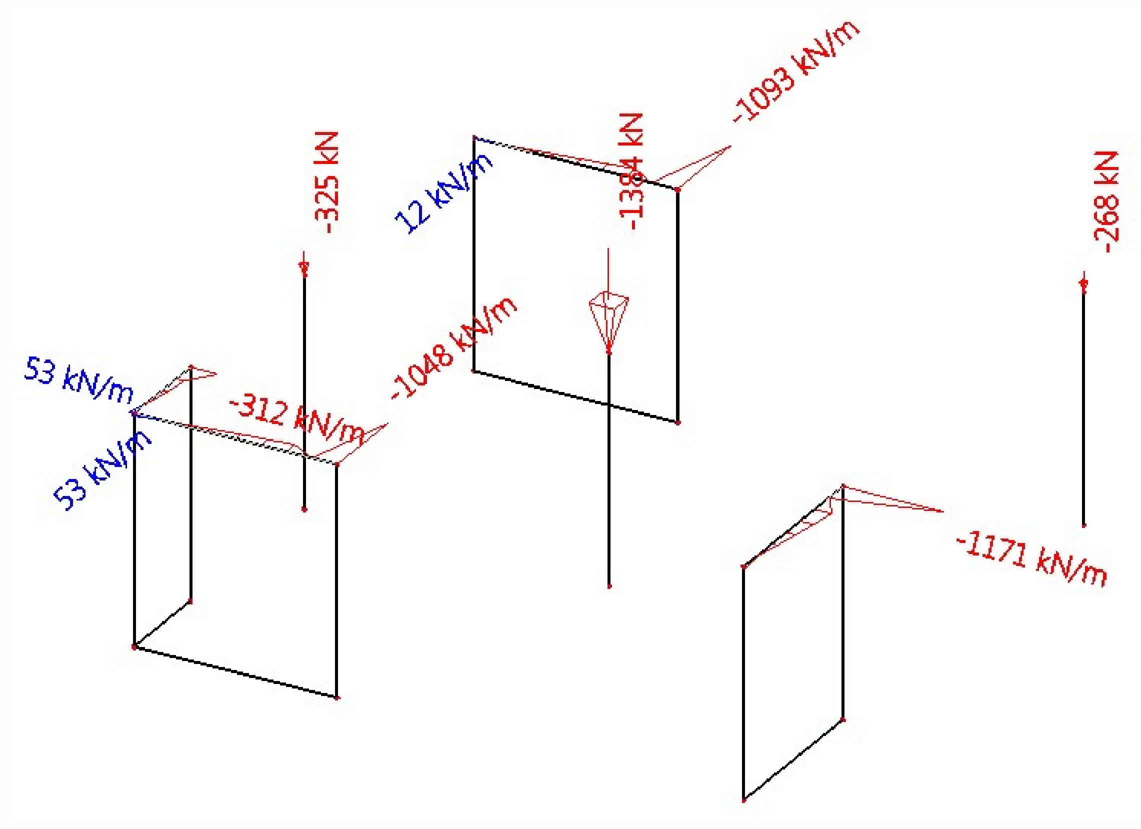
- All storeys
- Named selection
- Single storey

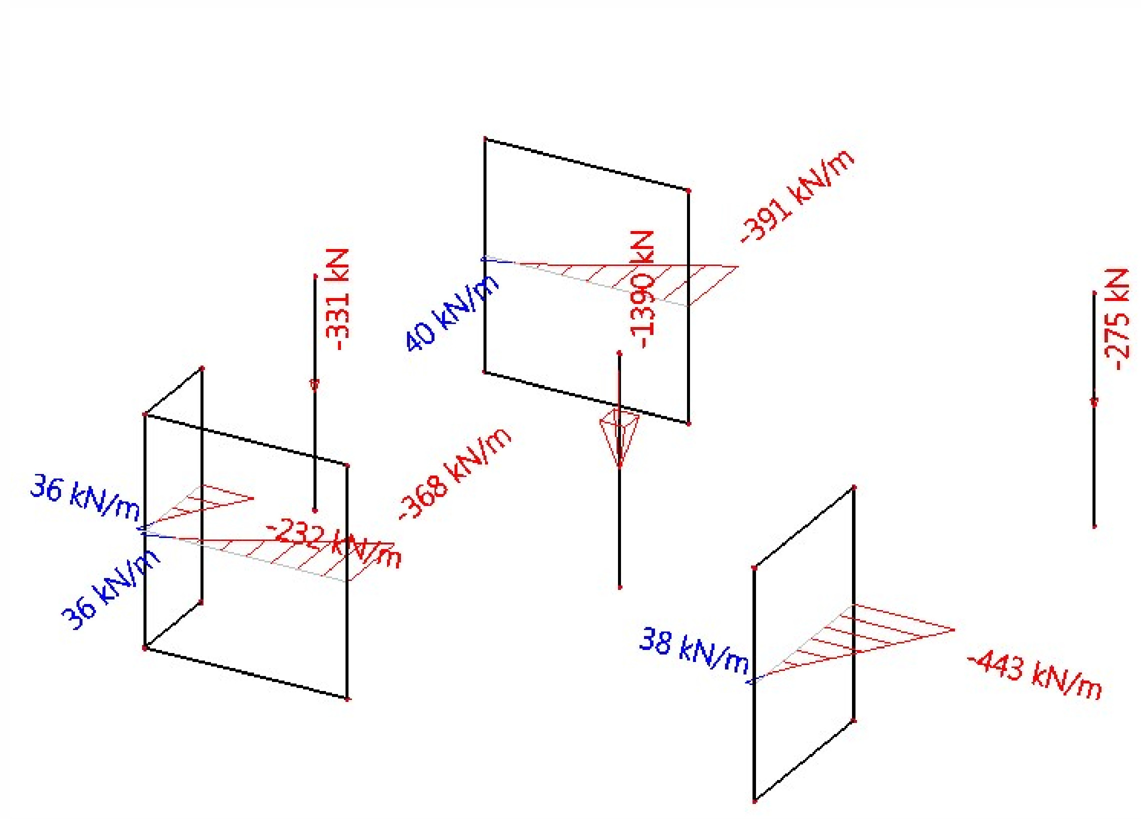
Named selection
named selection that contains a set of storeys; only if Selection = Named selection

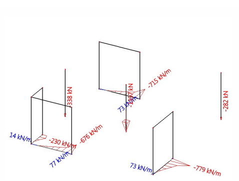
Storey
dropdown menu for selection of a single storey; only if Selection = Single storey

Section level
level where the section must be made across the supporting members in each storey; possible choices:

- Top (section at the top of each storey)
- Middle (section at mid-height of the each storey)
- Bottom (section at the bottom of each storey)
- User defined


-325 kN
12 kN/m
-1384 kN
-1093 kN/m
-268 kN
53 kN/m
-312 kN/m
-1048 kN/m
53 kN/m
-1171 kN/m


-391 kN/m
-331 kN
40 kN/m
-1390 kN
-275 kN
-368 kN/m
36 kN/m
-232 kN/m
36 kN/m
38 kN/m
-443 kN/m



User defined section level
level of section in each storey; 0 = bottom of the storey, 1 = top of the storey

Filter
filtering of supporting members for the output; possible choices are

- No
- Wildcard
- Material
- Thickness / CSS
- Layer

Wildcard
wildcard for name filtering of members; only if Filter = Wildcard

Material
list of available materials for material filtering of members; only if Filter = Material

Thickness
value of thickness for thickness filtering of members; only if Filter = Thickness/CSS; 0 = all thickness values

CSS
list of available cross-sections for filtering of members; only if Filter = Thickness/CSS

Layer
list of available layers for filtering of members; only if Filter = Layer

System
selection of coordinate system for output of internal forces in 1D members; possible choices are

- principal (principal axes of the cross-section)
- LCS (LCS of the 1D member)

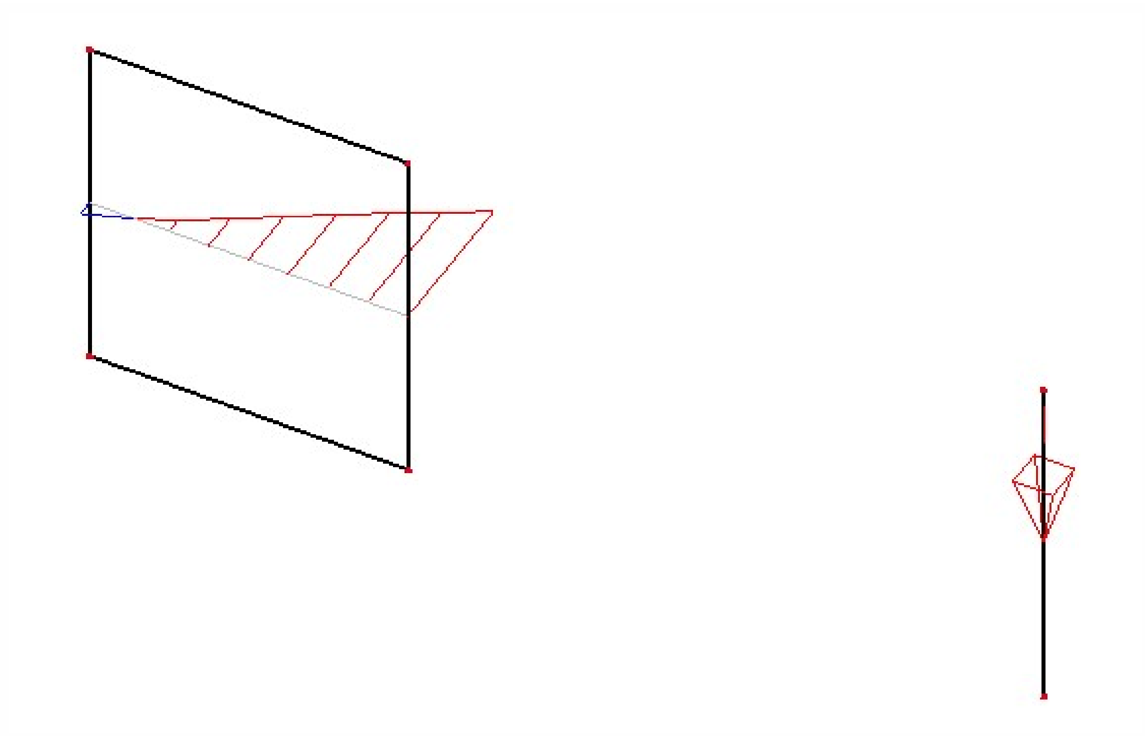
for 2D members, the LCS is always used

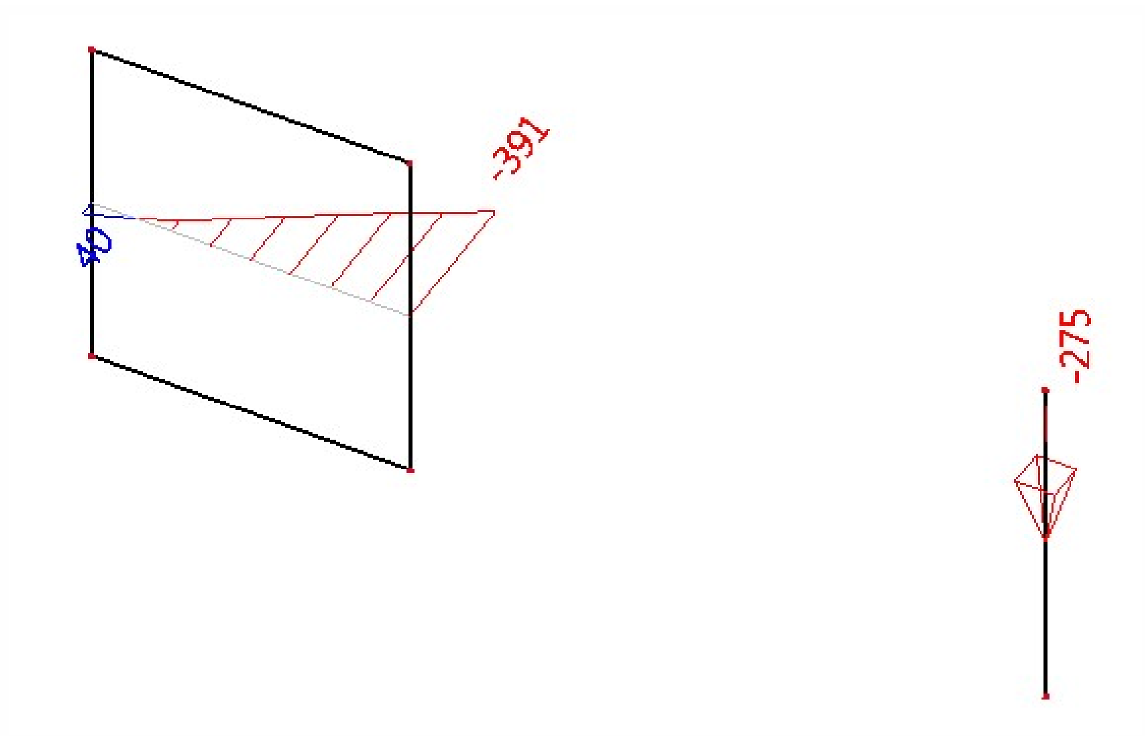
Extreme
filter extreme results; possible choices are

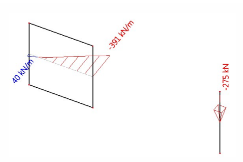
- No
- Section
- Member
- Global

Draw values
tick to draw values on the drawing

Draw units
tick to draw values with their units on the drawing







Result type
selection of result type; possible choices are

- ***internal forces***
- resulting forces

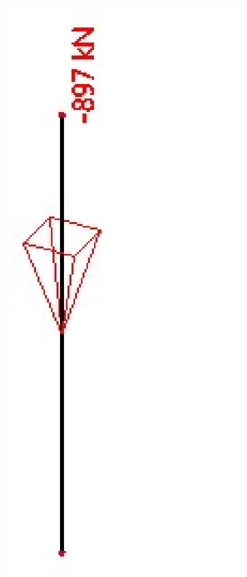
Values on beams
main component to be displayed on 1D members (columns); possible choices are

- N = axial force
- Vy = shear force according to Y-axis of selected system
- Vz = shear force according to Z-axis of selected system
- Mx = torsional moment
- My = bending moment around the Y-axis of selected system
- Mz = bending moment around the Z-axis of selected system

Additional values
allow to show more than one result component simultaneously on the drawing for 1D members; possible choices: see Values on beams

When only the main component is selected (no additional value ticked), the selected component is drawn in the corresponding direction.

When one or more additional value is ticked, all selected components are listed in the plane of the screen for better readability.



Limit values
for each result component on 1D members, definition of min and max value for colour coding on the drawing. Colours may be configured in Settings > Colours/Lines.

Default colour coding is as follows:

Values lower than V_{min} are display in red

Values between V_{min} and V_{max} are displayed in gray

Values greater than V_{max} are displayed in blue

Values on slabs
main component to be displayed on 2D members (walls); possible choices are

- nx = membrane axial force in X-direction of member LCS
- ny = membrane axial force in Y-direction of member LCS
- nxy = membrane shear force in member LCS
- mx = bending moment around Y-axis of member LCS
- my = bending moment around X-axis of member LCS

- mxy = torsional moment according to member LCS
- vx = shear force according to X-axis of member LCS
- vy = shear force according to Y-axis of member LCS

Detailed storey result

Linear calculation, Extreme: Global, System: Principal
Selection: FL2
Load cases : SW
Columns:

Name	Storey	x [m]	y [m]	z [m]	N [kN]	V_y [kN]	V_z [kN]	M_x [kNm]	M_y [kNm]	M_z [kNm]
B38	FL2	12.000	0.000	6.300	**-123**	0	0	**0**	0	0
B41	FL2	6.000	12.000	6.300	-147	**0**	**0**	**0**	**0**	**0**
B23	FL2	6.000	6.000	6.300	**-594**	0	0	0	0	**0**

Walls:

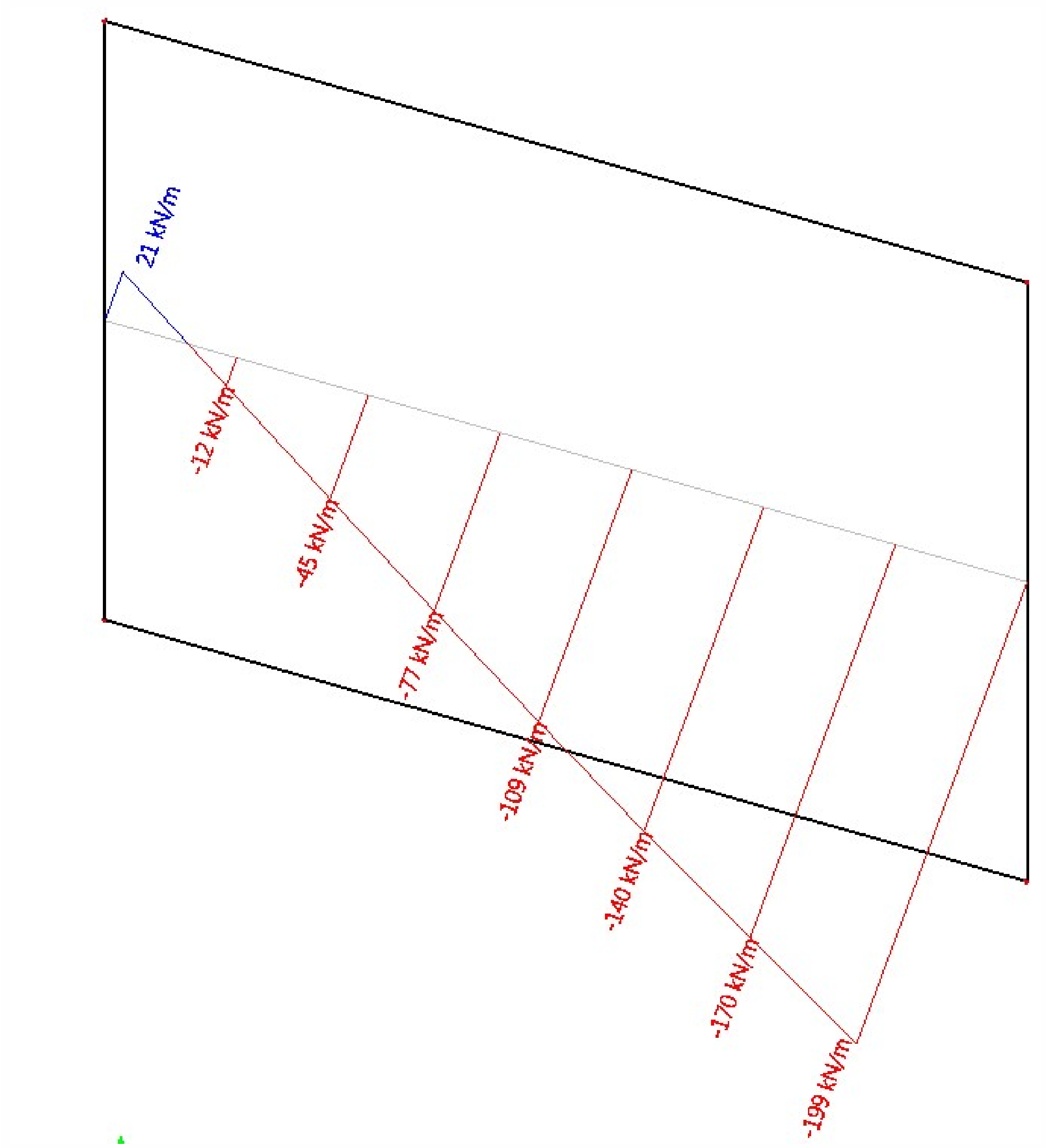
Name	Storey	x [m]	y [m]	z [m]	n_x [kN/m]	n_y [kN/m]	n_{xy} [kN/m]	m_x [kNm/m]	m_y [kNm/m]	m_{xy} [kNm/m]	v_x [kN/m]	v_y [kN/m]
S14	FL2	2.000	0.000	6.300	**3**	-115	-1	0	0	0	0	0
S14	FL2	3.500	0.000	6.300	**-1**	**-198**	0	0	0	0	0	0
S5	FL2	12.000	12.000	6.300	0	**23**	0	0	0	0	0	0
S14	FL2	1.000	0.000	6.300	2	-47	**-5**	0	0	0	0	0
S8	FL2	0.000	8.500	6.300	1	-147	**7**	0	0	0	0	0
S11	FL2	0.000	12.000	6.300	0	18	1	**-1**	**-1**	**0**	0	**1**
S5	FL2	12.000	11.500	6.300	0	-4	-3	0	0	**0**	0	0
S8	FL2	0.000	8.000	6.300	-1	-165	0	0	0	0	**0**	0
S8	FL2	0.000	12.000	6.300	0	18	-2	**1**	**1**	0	**1**	1
S11	FL2	0.500	12.000	6.300	0	-16	1	0	-1	0	0	**0**

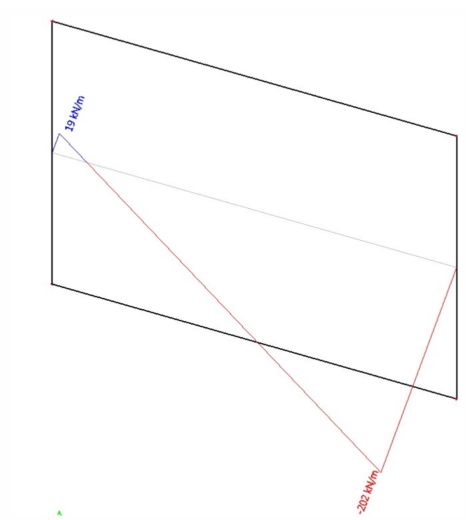
Additional values
allow to show more than one result component simultaneously on the drawing for 2D members; possible choices: see Values on slabs

Limit values
for each result component on 2D members, definition of min and max value for colour coding on the drawing; see detailed description above

Diagram
style of diagram for internal forces in 2D members; possible choices are

- Precise: raw computed results, without alteration
- Trapezoidal: trapezoidal regression of diagram, for each 2D member separately
- None: the diagram is hidden

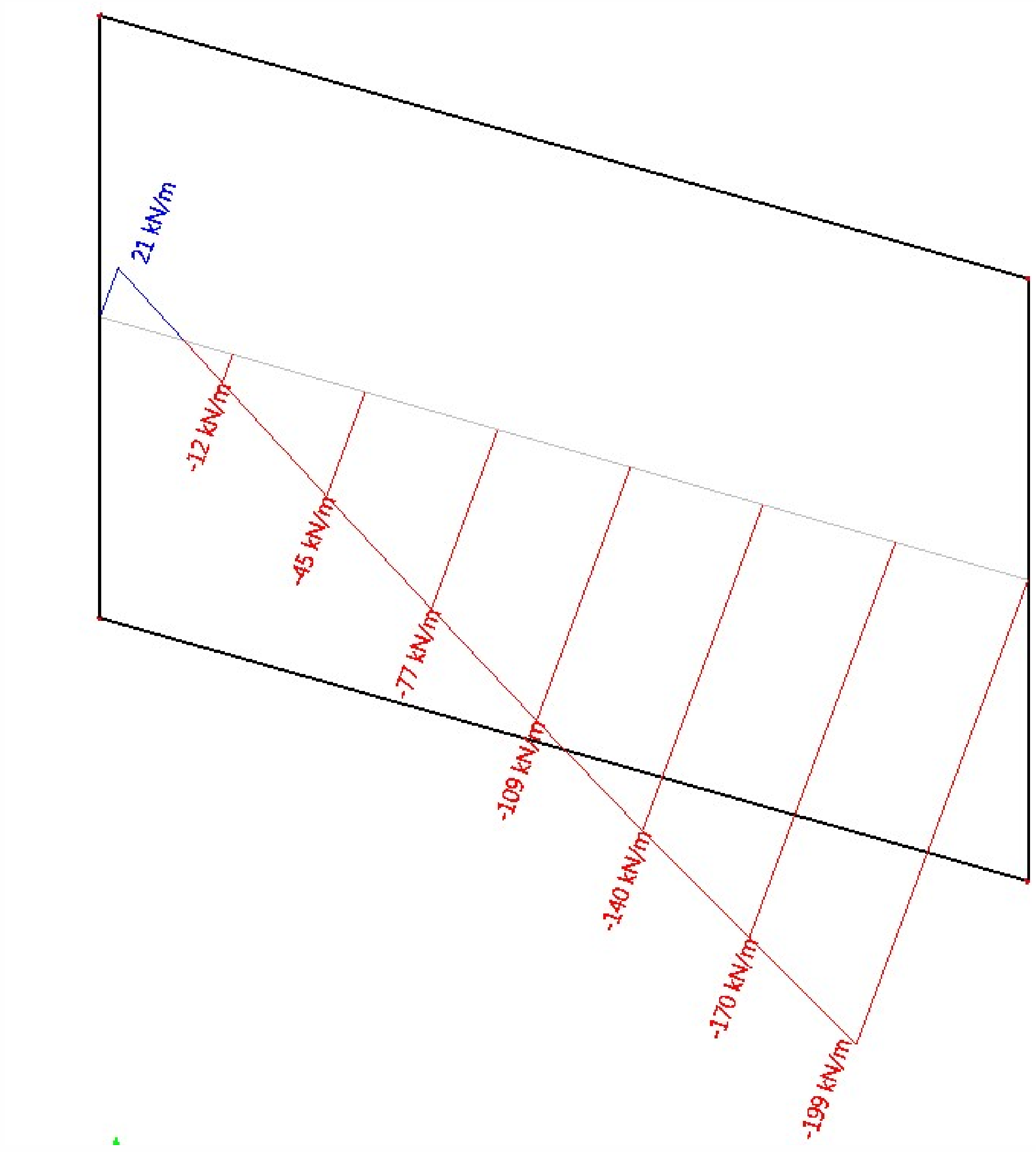


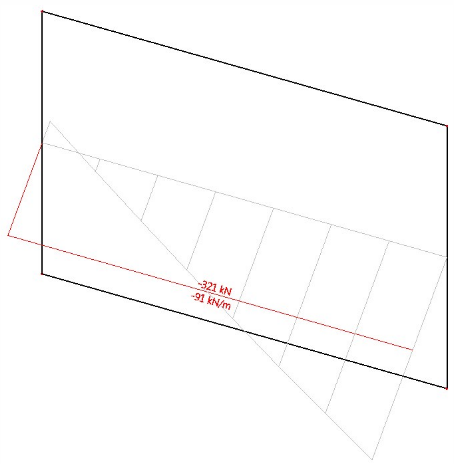


Display total value

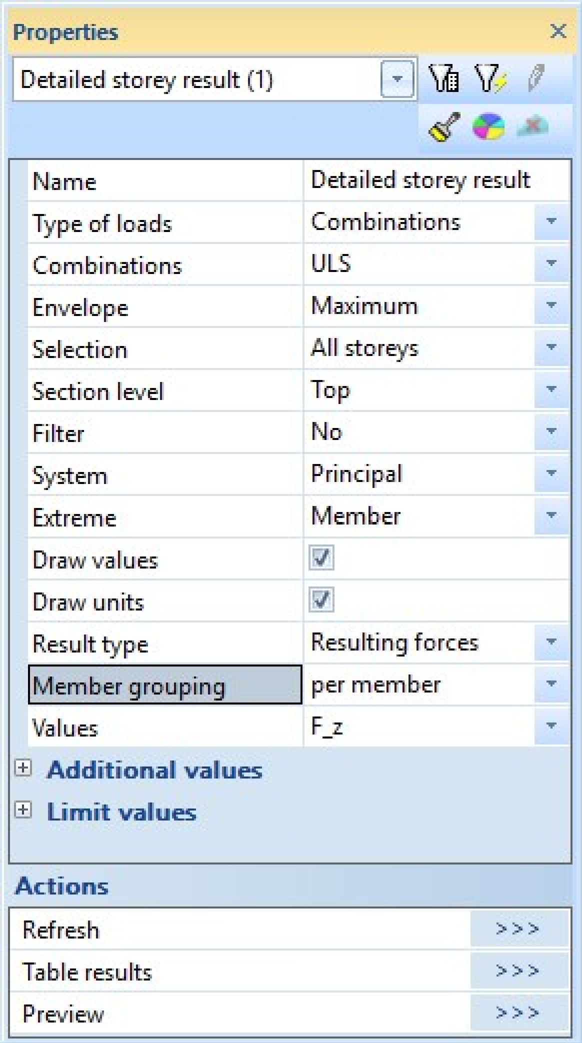
when ticked, the integral value of the displayed diagram is written, for each 2D member separately

Display average value
when ticked, the average value of the displayed diagram is written and the corresponding uniform diagram is drawn, for each 2D member separately





Result type Resulting forces – Member grouping per member



Only settings that behave differently from previous paragraph are listed here. For information about settings that are not listed here, please refer to previous paragraph “Internal forces”.

Result type
selection of result type; possible choices are

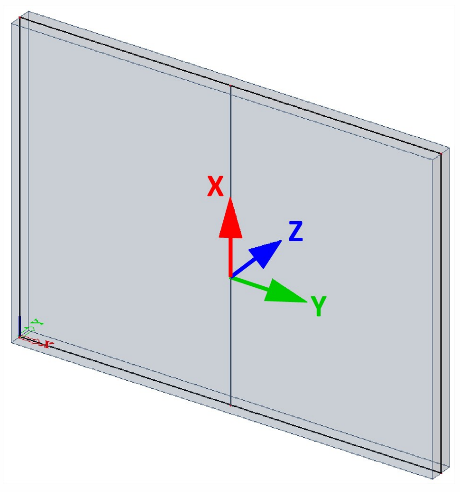
- internal forces
- ***resulting forces***

Member grouping
selection of location type for resulting forces; possible choices are

- ***per member***: the resulting forces are computed for each supporting member separately
- per storey: the resulting forces are computed for each storey, considering all the supporting members at once; 1D (columns) and 2D members (walls) are taken into account together

Resulting forces in 1D members (columns) are identical to internal forces in 1D members.

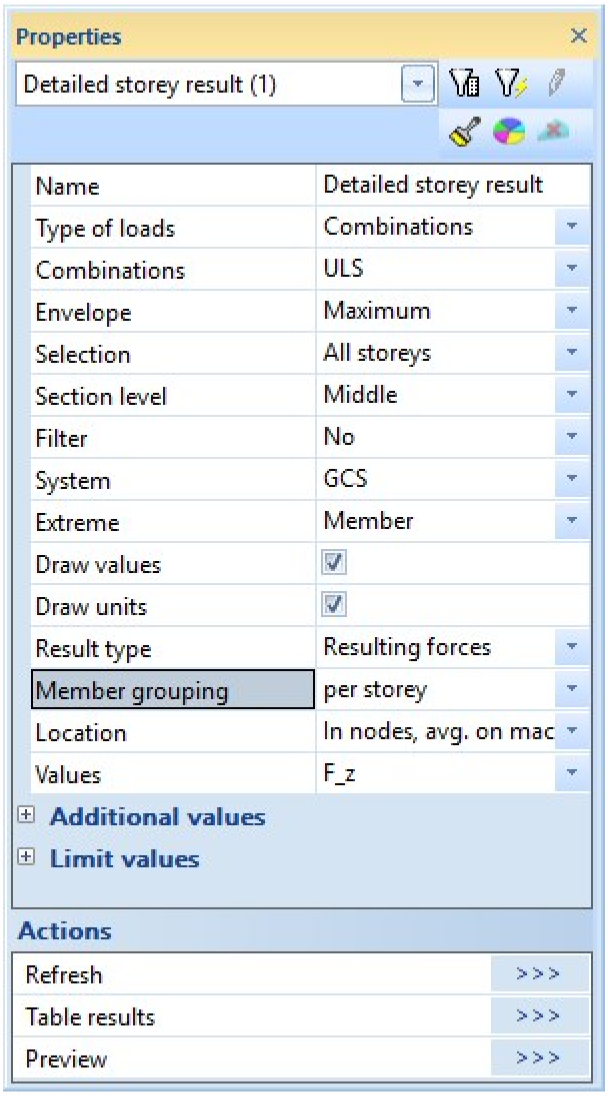
Resulting forces in 2D members (walls) compute the resultant at the center of each wall, according to a dedicated local coordinate system, regardless of the System output setting. The coordinate system that is used is the same as the LCS of a vertical rib placed in the middle of the wall. It is also the same coordinate system that is used for integration strips.



- The local X-axis is vertical, upwards
- The local Z-axis is identical to the Z-LCS of the wall
- Y = Z ^ X

In this way, resulting forces in walls can be easily displayed together, consistently with internal forces in columns on a single drawing.

Result type Resulting forces – Member grouping per storey



Only settings that behave differently from paragraph “Internal forces” are listed here. For information about settings that are not listed here, please refer to paragraph “Internal forces”.

Result type
selection of result type; possible choices are

- internal forces
- ***resulting forces***

Member grouping
selection of location type for resulting forces; possible choices are

- per member: the resulting forces are computed for each supporting member separately
- ***per storey***: the resulting forces are computed for each storey, considering all the supporting members at once; 1D (columns) and 2D members (walls) are taken into account together

Location
defines how values are obtained from the 2D finite elements, using various interpolation methods (for more information, see Location)

System
selection of coordinate system for output of resulting forces by storey; possible choices are

- GCS
- UCS

UCS

selection of a UCS from the UCS library, to be used as reference system for output of resulting forces by storey

Values

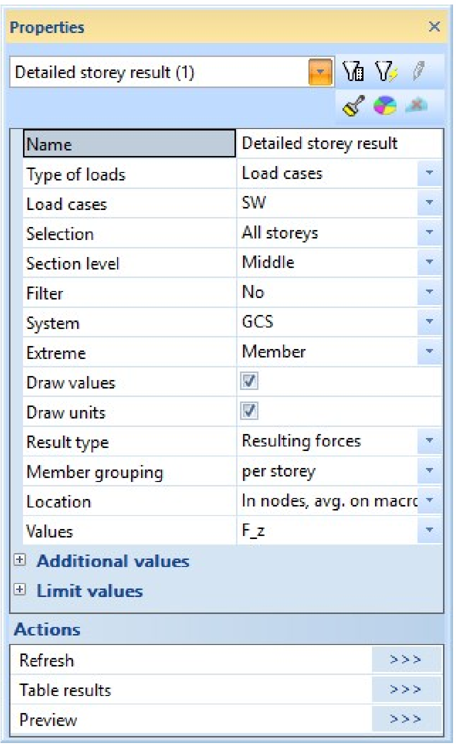
main component of resulting force; possible choices are

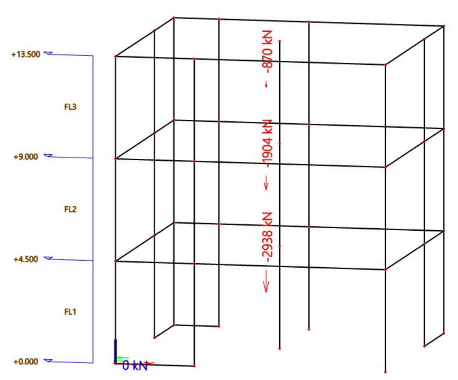
Fx, Fy, Fz = resulting force according to X,Y,Z axis of selected coordinate system

Mx,My,Mz = resulting moment around X,Y,Z axis of selected coordinate system

Some examples

Output of total vertical forces in all storeys (load descending)





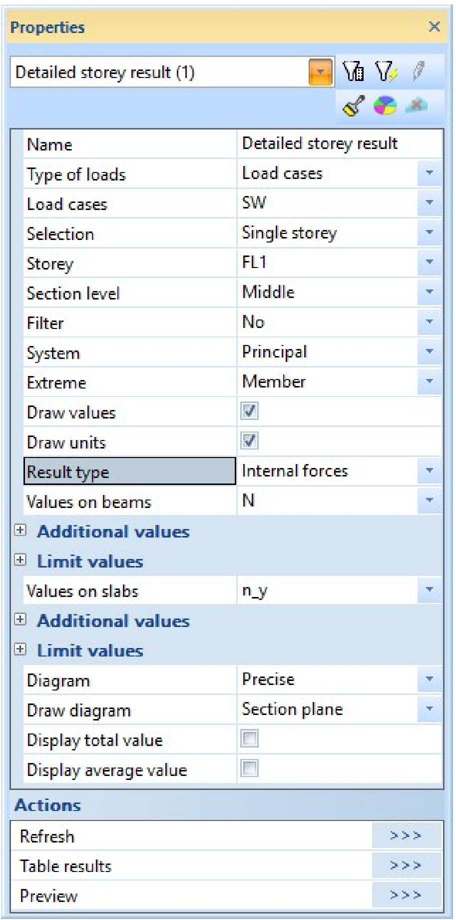
Detailed storey result

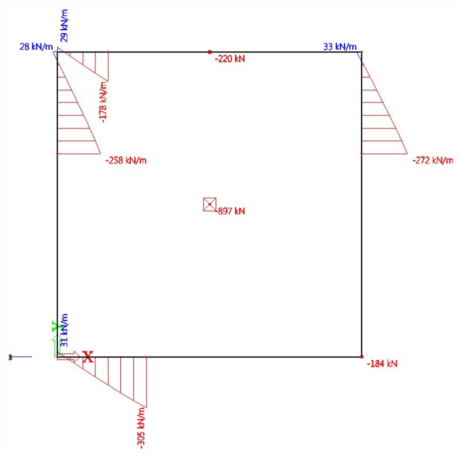
Linear calculation, Extreme: Member, System: GCS
Selection: All
Load cases : SW

Resulting forces per storey

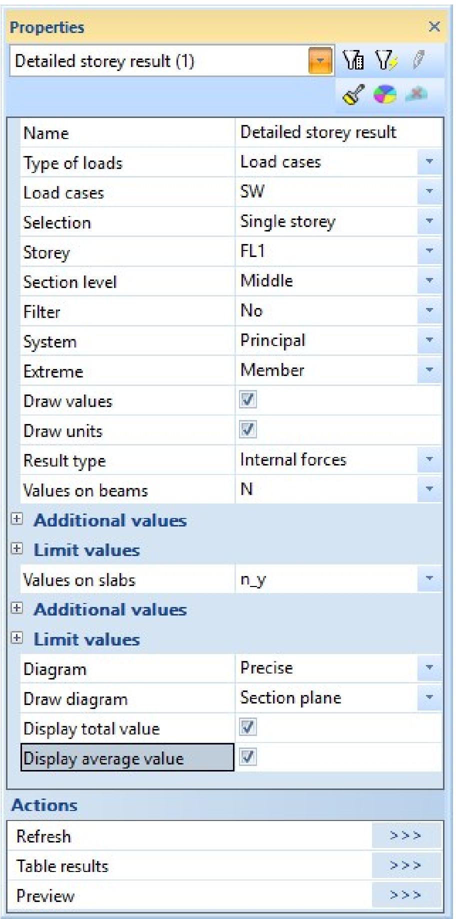
Name	Storey	x [m]	y [m]	z [m]	F_x [kN]	F_y [kN]	F_z [kN]	M_x [kNm]	M_y [kNm]	M_z [kNm]
FL1		5.118	7.278	2.250	0	1	-2938	2485	1368	4
FL2		5.118	7.278	6.750	0	0	-1904	1671	945	0
FL3		5.118	7.278	11.250	0	0	-870	858	523	-1
FL4		0.000	0.000	0.000	0	0	0	0	0	0

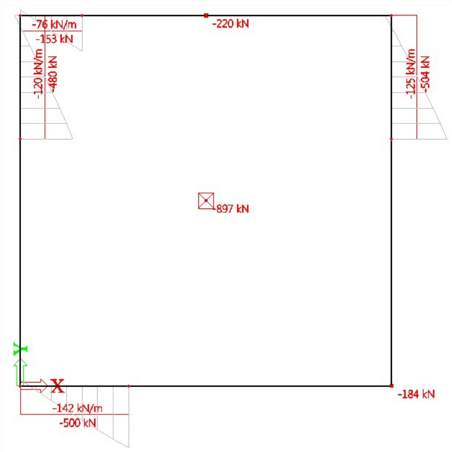
Internal forces in all supporting members of a storey



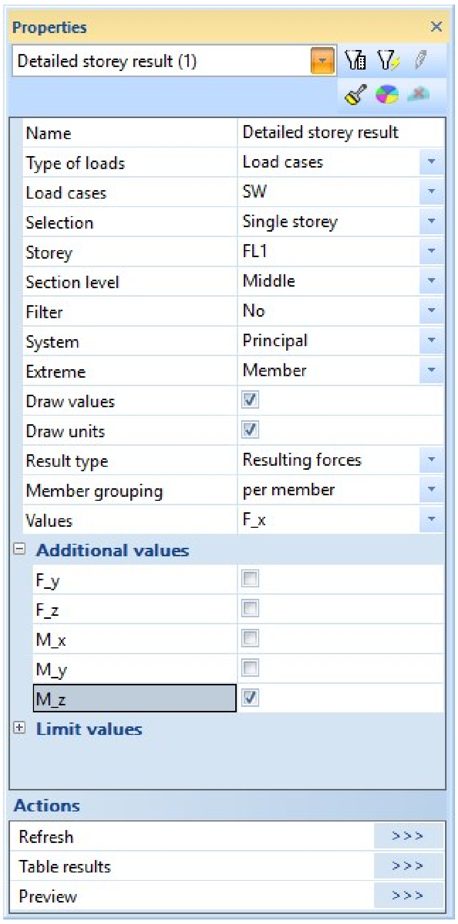


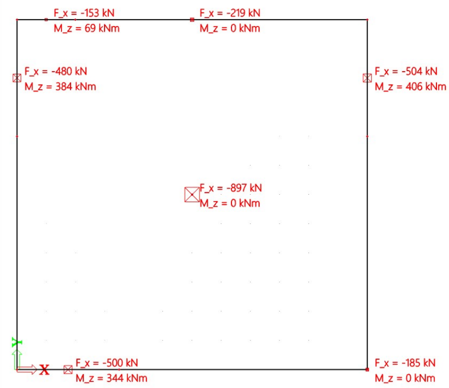
Average forces in all supporting members of a storey





Resulting forces in all supporting members of a storey





Detailed storey result

Linear calculation, Extreme: Member, System: Principal
Selection: FL1
Load cases : SW
ResultingForces:

Name	Storey	x [m]	y [m]	z [m]	N [kN]	Vy [kN]	Vz [kN]	Mx [kNm]	My [kNm]	Mz [kNm]
S4	FL1	12.000	10.000	2.250	**-504**	**-2**	**0**	**0.01**	**0.00**	**408.43**
S7	FL1	0.000	10.000	2.250	**-480**	**0**	**0**	**0.29**	**3.41**	**385.44**
S10	FL1	1.000	12.000	2.250	**-153**	**1**	**0**	**-0.02**	**-2.23**	**60.11**
S15	FL1	1.750	0.000	2.250	**-500**	**-1**	**0**	**0.02**	**0.00**	**329.18**
B22	FL1	6.000	6.000	2.250	**-897**	**-2**	**0**	**0.01**	**-1.06**	**-2.39**
B37	FL1	12.000	0.000	2.250	**-184**	**0**	**0**	**0.01**	**0.00**	**0.00**
B40	FL1	6.000	12.000	2.250	**-220**	**0**	**0**	**0.01**	**0.00**	**0.00**

Modal Superposition

Introduction – theoretical background

This section gives a brief overview of the seismic analysis features of SCIA Engineer. For more details, read also the chapter dedicated to seismic loading.

Modal superposition

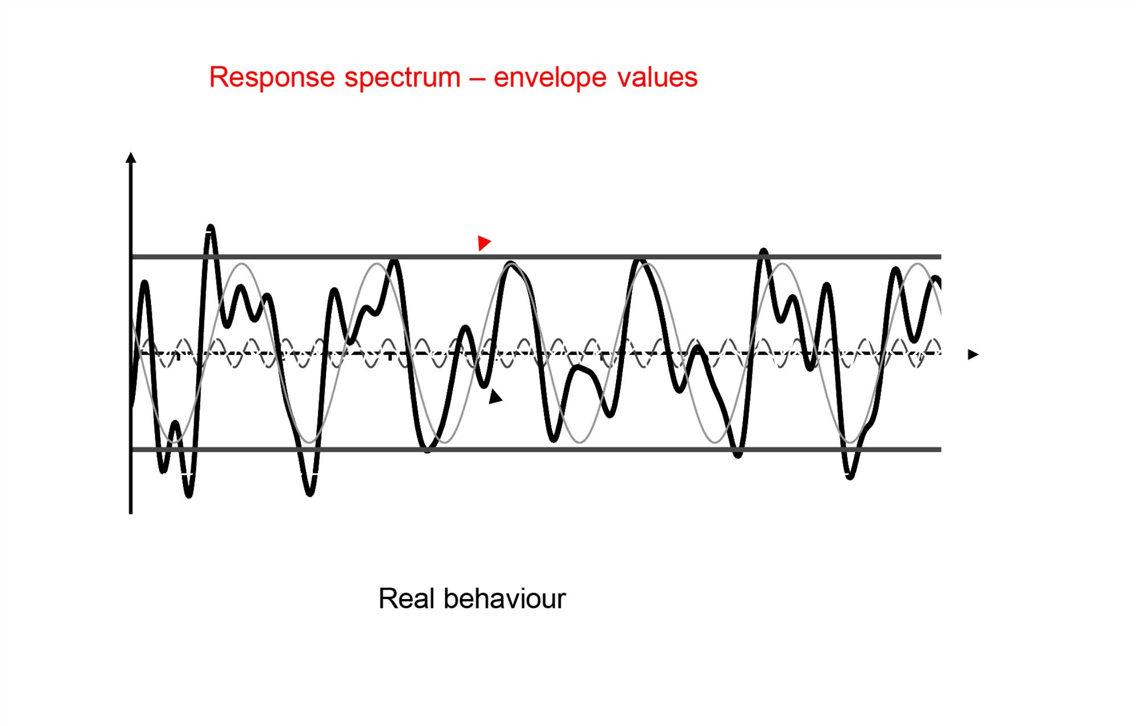
The response spectrum method is one of the most widely used methods for the seismic analysis of structures. It has many advantages on other methods:

- unlike equivalent lateral forces (ELF) method, it takes several vibration modes into account and allows analyzing structure without restriction on the geometry
- unlike explicit time history dynamic analysis, it takes into account a whole range of earthquakes, thus covering an envelope of possible earthquakes for a given site; a time history analysis takes only one earthquake into account at a time (one input accelerogram)
- the computation cost of the response spectrum method is much lower than for time history analysis, especially when non-linearities need accounting for

The response spectrum method uses a modal superposition of the relevant eigenmodes of the structure. The method allows to calculate the magnitude of each mode, but not their phase shift. The values of phase shift actually depend on the real accelerogram that will be applied to the structure. As a response spectrum represents the envelope of a family of accelerograms, it is not possible to define unique values of phase shift for each mode: these variables are random.

This is where the concept of modal superposition comes in: various statistical techniques allow to determine envelope values which cover the real behaviour with a reasonable probability of occurrence.

The most widely used techniques are the SRSS (Square Root of Sum of Squares) and CQC (Complete Quadratic Combination).

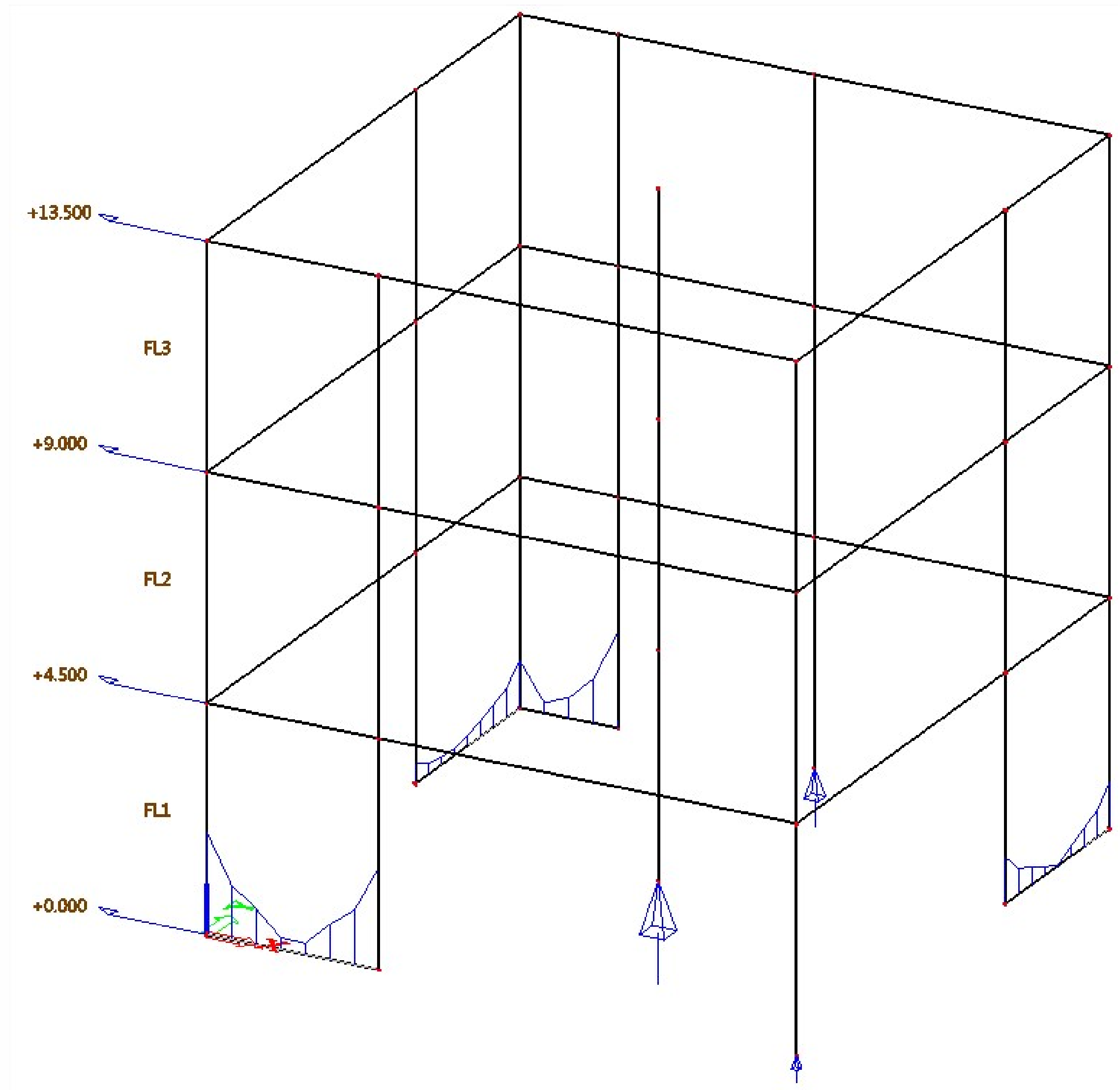


These methods have the advantage of providing very easily design values of all results (displacements, internal forces…) without knowing the real phase shift values, but only part of the information is available:

- likely min and max values of any result can be determined
- the actual sign of a result cannot be defined
- the concomitance of separate results cannot be defined

The last point above is actually not entirely accurate, as probabilistic correlations can be established between results, but those techniques are currently not commonly used in practice and are beyond the scope of this document.

The loss of concomitance and sign of results is an issue typically when computing resulting forces in shear walls: it is not possible to compute a resultant from internal forces after modal superposition, as typically all raw results are positive.

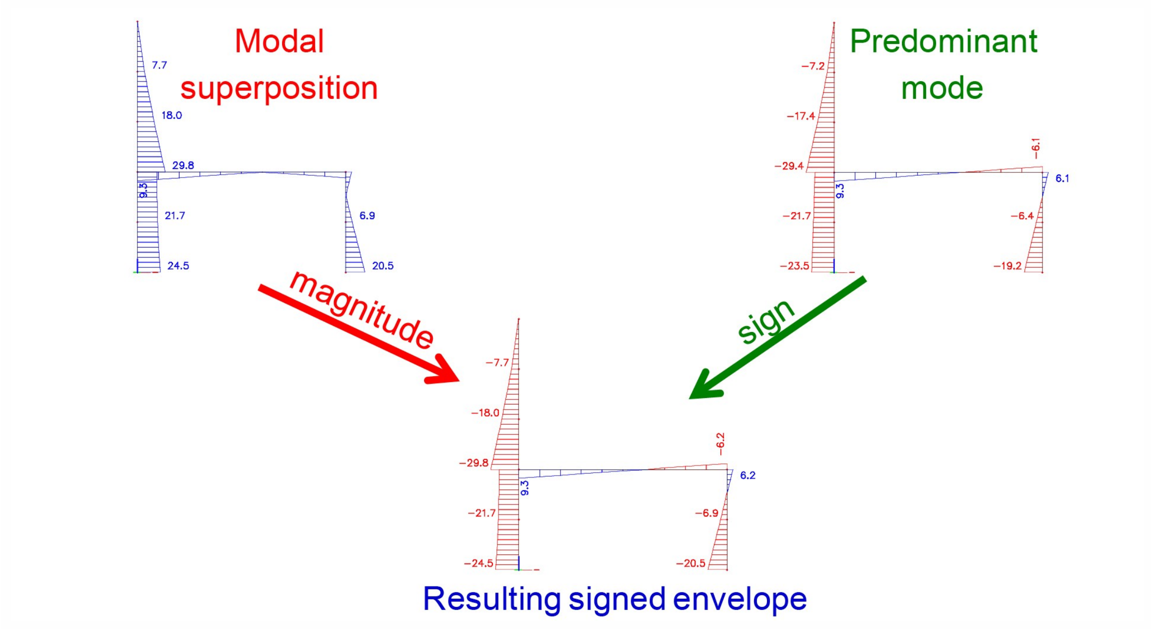


Computing resultant forces in one of those shear walls would typically give near-zero moments and extremely over-estimated axial forces.

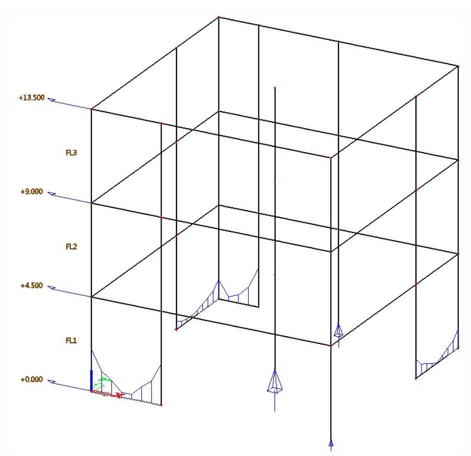
Predominant mode & signed results

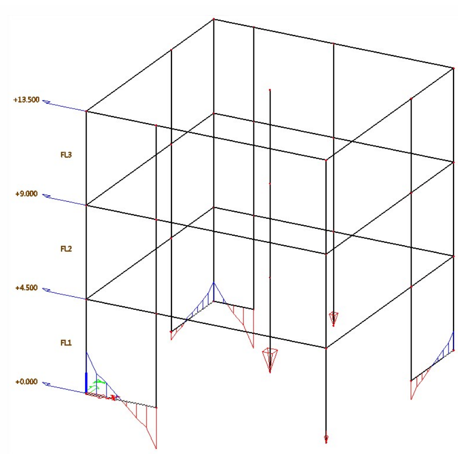
To obtain usable values of resulting forces, a possibility is the so-called "signed results" method.

It consists of applying some signature scheme to raw results of the modal superposition. A classical approach uses the sign of the most significant eigenmode.



Applying this to shear walls, it is possible to “sign” the internal forces, making them suitable for computation of resulting forces:





Modal superposition of resultants

The method of signed results might be seen as a workaround to obtain usable resulting forces. This method will be referred to as “calculation of resultants with modal pre-superposition”. It is convenient from a point of view of computation, because modal superposition needs to be done only once on all local results and the resultant can be computed directly from a unique set of results.

However, the rigorous method for computation of resultants in the context of the response spectrum method can be summarized as follows:

- compute the local internal forces for each eigenmode
- compute the resultant force for each eigenmode separately
- apply the modal superposition to the obtained modal resultant values

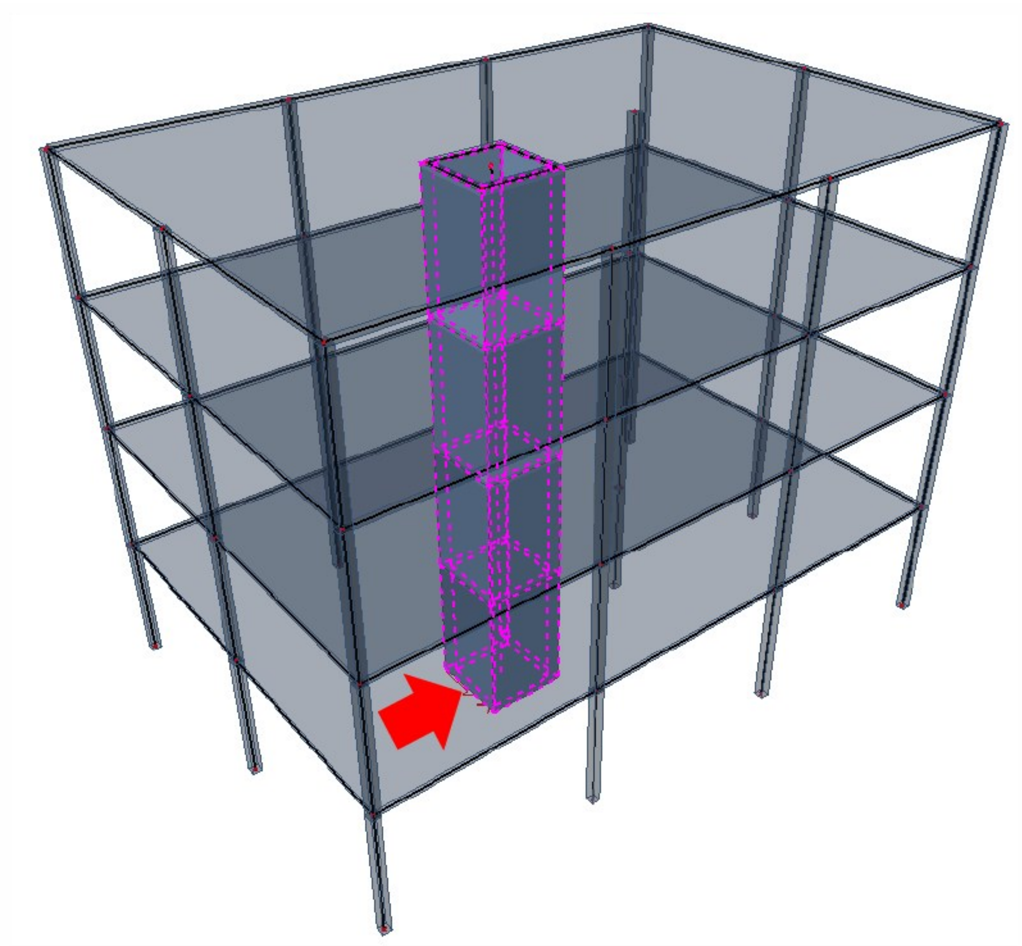
This method will be referred to as “calculation of resultants with modal **post-superposition**”.

When proceeding so, no result signature is necessary to obtain correct values of resulting forces. Moreover there are cases where the method described in the previous paragraph gives overestimated results of most result components and can therefore only be seen as an approximation. The method described here is clearly more robust and accurate.

As shown in the example below, structures with predominant torsional behaviour are especially sensitive to that phenomenon.

Important discrepancies can be seen on most resultant components when using the calculation of resulting forces from superposed signed results (up to 4 times the reference result).

On the other hand, all values obtained by modal superposition of resulting forces computed in each mode separately are very close to the reference model (max 6% difference).



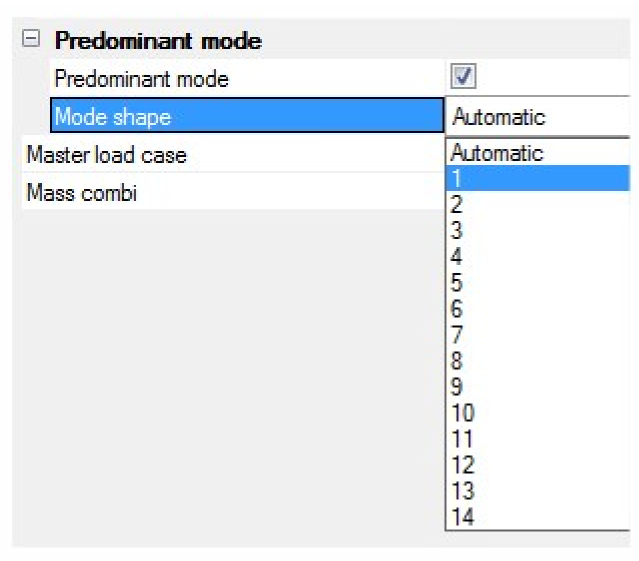
Forces at the bottom of the core	Fx	Fy	Fz	Mx	My	Mz
Resultant from results signed after superposition	518	855	15	2509	2732	1691
Modal superposition after resultant calculation	249	198	26	1900	2394	1614
Reference model (1D member)	264	209	25	1911	2429	1640

Modal superposition in SCIA Engineer

SCIA Engineer offers all methods described above. All related settings are located in the properties of the seismic load case.

Signed results

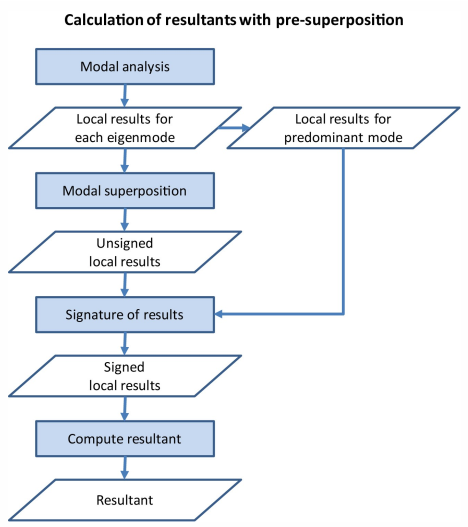
The signature of results can be enabled using the setting “Predominant mode”. The eigenmode to be used as reference for the signature can be selected automatically or manually.

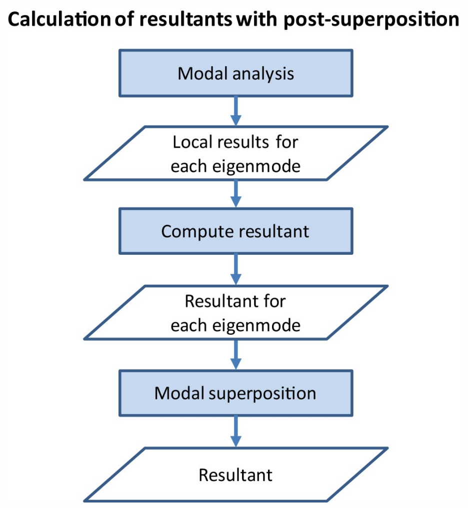


Note that the automatic selection of the mode shape will select the mode which has the highest modal mass, regardless of the actual direction of the earthquake. This is most of the time correct for 2D models (2D frames) but **often wrong for 3D models! For 3D models, it is highly recommended to select the mode shape manually.**

Modal superposition of resultants

Resultants can be calculated in two different ways: by pre-superposition or post-superposition (see above). The flowcharts below summarize the process for both methods. Note that although the post-superposition method might look much simpler than the pre-superposition method, actually all the part related to local results must be done anyway in order to provide local results for seismic load cases. It is not represented on the flowchart because it is irrelevant in this context.





In the current version of SCIA Engineer, the following result services use the **pre-superposition method** and their accuracy is therefore **affected by the selection of an appropriate predominant mode for results signature**:

- resultant of reactions
- integration strips

The following result services use the **post-superposition method**:

- resultant of forces in a section on 2D members
- detailed storey results, resultant per member
- detailed storey results, resultant per storey
- seismic resultant in calculation protocol

Accidental Eccentricity

Introduction

Most of the seismic standards require that structures are checked for torsion due to mass eccentricity including an additional eccentricity – so-called accidental eccentricity. This is required to cover inaccuracies between the real structure and the modelization, as well as the fact that masses that are linked to service loads may vary during the life of the structure.

Two types of eccentricity must be distinguished for the analysis: the structural eccentricity and the accidental eccentricity.

The structural eccentricity is the offset between the center of mass and the center of stiffness of the structure. It is part of the structure. In a simplified seismic analysis via 2D models, where typically the X and Y directions are analyzed separately, the impact of the structural eccentricity is taken into account by manually distributing the torsional effects on the structure. An additional safety factor is usually applied to the structural eccentricity to cover inaccuracies due to that simplified method.

When using a 3D modelization of the structure, the structural eccentricity is automatically taken into account due to the fact that the X and Y and linked and analyzed together, allowing torsional effects to appear directly in the analysis without having to add them manually afterwards.

The accidental eccentricity accounts for inaccuracies in the distribution of masses in the structure. Design standards usually take it into account as an additional mass eccentricity that is defined as a fraction of the size of the structure.

In the Eurocode 8, the accidental eccentricity for a given floor is defined as 5% of the width of the floor perpendicularly to the direction of the acting seismic action.

In simplified modelizations where the structural eccentricity appears explicitly, it is very simple to add the accidental eccentricity in the calculation. In general 3D modelizations, the structural eccentricity does not appear as such and it is therefore more complex to take its effects into account in such a case.

In SCIA Engineer, using the IRS condensed model allows introducing accidental eccentricity easily, since the condensed model uses only one R-node per storey. The accidental eccentricity may be taken into account either as real mass eccentricity or as additional torsional actions (simplified method according to the design codes).

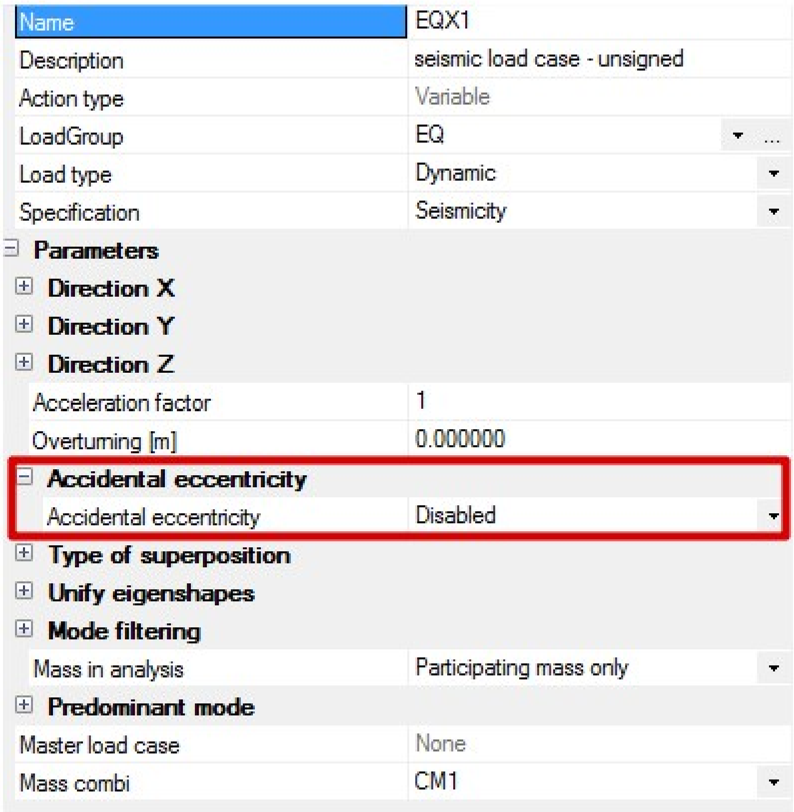
However, the method using real mass eccentricity in the modal analysis has not been implemented yet in SCIA Engineer.

> For now, only the simplified method using additional torsional moment is available.

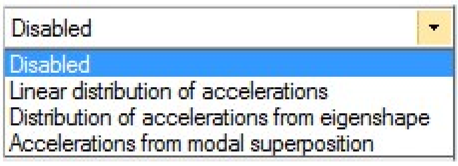
Definition of the accidental eccentricity

First of all, please note, that accidental eccentricity may be used only together with the reduced model analysis. See how to enable it in Enabling the reduced model.

The accidental eccentricity settings are defined in the properties of the considered seismic load case, in the sub-group ."Accidental eccentricity". Accidental eccentricity is disabled by default.



The method for the calculation of the accidental eccentricity can be selected in the combobox "Accidental eccentricity":



Depending on the selected method, various settings must be defined.

Eccentricity
value of the accidental eccentricity, defined as a fraction of the width of the considered storey in the direction perpendicular to the seismic action;

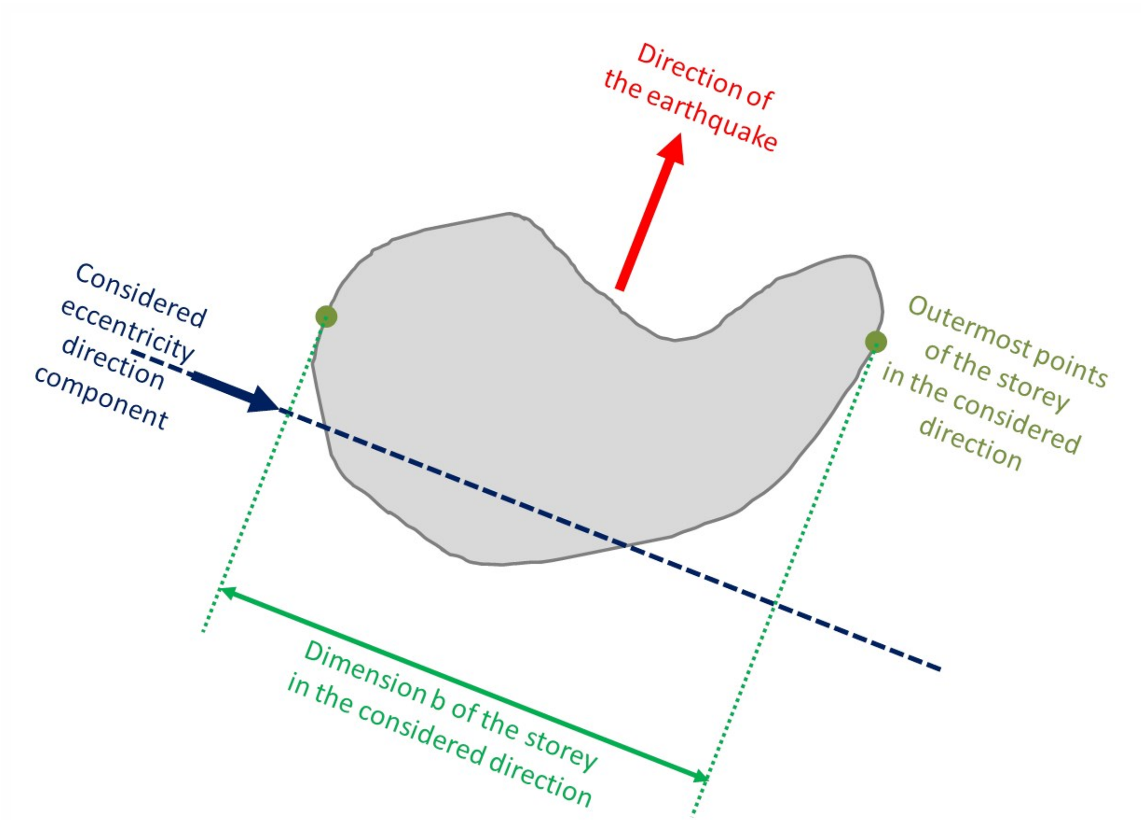
most seismic design standards specify a value of 0.05 for this ratio (EN1998-1 § 4.3.2(1)P and formula (4.3)).

Mode shape
in case the distribution of accelerations is calculated according to a mode shape, the user must specify which mode should be used for that purpose

These settings are explained more in detail below.

Value of the accidental eccentricity

Regardless of the selected method, the value of the accidental eccentricity must be specified. That value is defined as a fraction of the width of the considered storey in the direction perpendicular to the seismic action (EN1998-1 § 4.3.2(1)P).



The actual eccentricity is then calculated as follows:

$$e_{A,i} = e_{Ar,i} \cdot b_i$$

where $e_{Ar,i}$ is the user defined value of the relative eccentricity, e.g. 0.05, and b_i is the width of the considered storey. The value $e_{A,i}$ is computed separately for each storey.

Calculation of the effects of eccentricity

The accidental eccentricity is taken into account as follows:

- a dynamic analysis of the structure is performed without accidental eccentricity, using the response spectrum method
- the effects of accidental eccentricity are added by applying static torsional moments to the structure about the vertical axis of each storey. This method is described in detail in the Eurocode 8 (EN1998-1 § 4.3.3.3.3)

The general principle for the calculation of torsional moments is as follows:

$$F_j = F_{base} \cdot \frac{\alpha_j \cdot m_j}{\sum_k \alpha_k \cdot m_k}$$

$$M_{z,j} = F_j \cdot e_{A,j}$$

where

F_j is the horizontal force acting on storey j

F_{base} is the total horizontal force acting on the structure (aka base shear) in the considered earthquake direction obtained from the response spectrum analysis of the structure

m_j is the mass of storey j

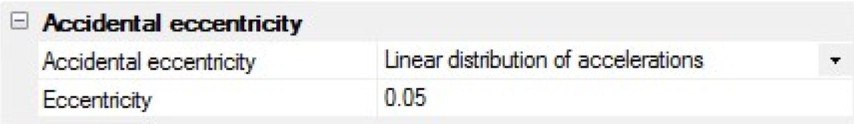
α_j is the distribution key of the accelerations; this depends on the selected method; at this time, 3 methods are offered to define the distribution of accelerations (see below)

$e_{A,j}$ is the accidental eccentricity of storey j, as defined in the previous paragraph

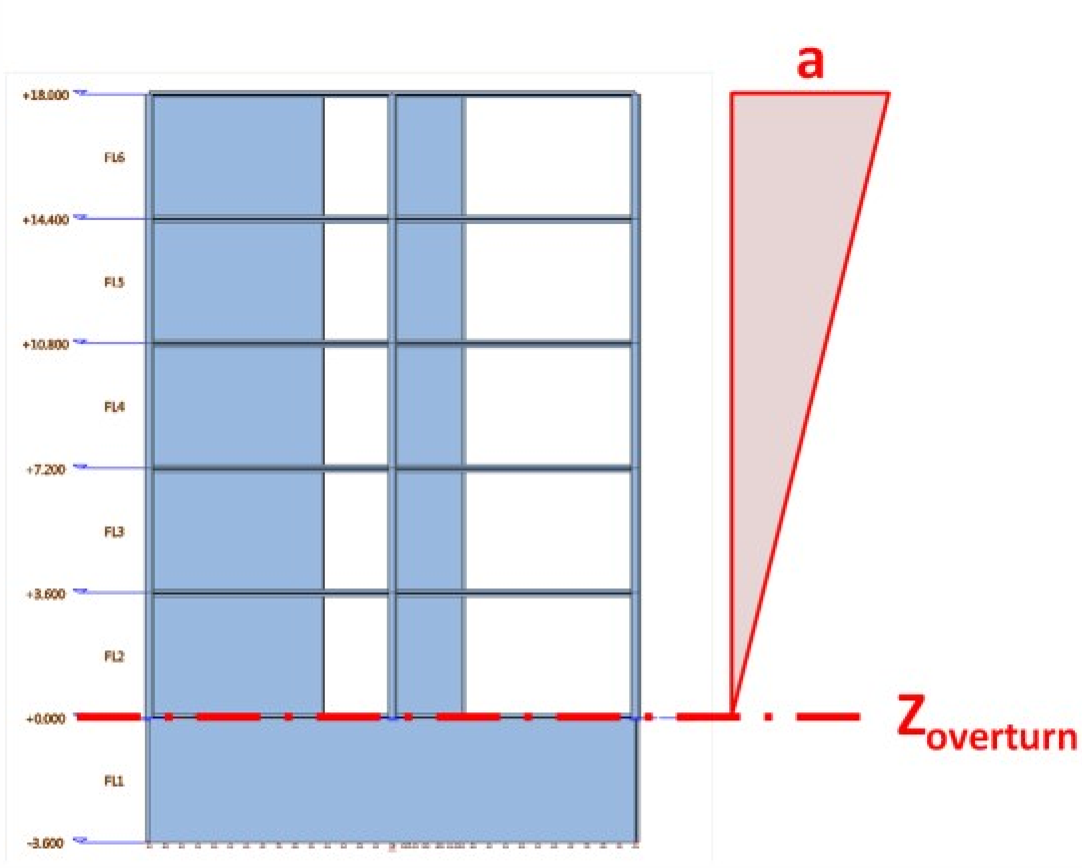
$M_{z,j}$ is the applied torsional moment about the Z-axis for the storey j

About accidental eccentricity and ELF seismic analysis: when the ELF method is selected for the seismic analysis, the storey forces are directly obtained from the ELF calculation. In that case, the method for the distribution of accelerations for accidental eccentricity can only be the same as the method selected for the ELF calculation.

Linear distribution of accelerations



In this case, the distribution of acceleration is assumed to be linear, proportional to the height. The reference level is the overturning level defined in the seismic load case properties.



$$\alpha_j = \max(z_j - z_{overturn}; 0)$$

where z_j is the level of the mass center of storey j. $z_{overturn}$ is defined by the user in the properties of the seismic load case.

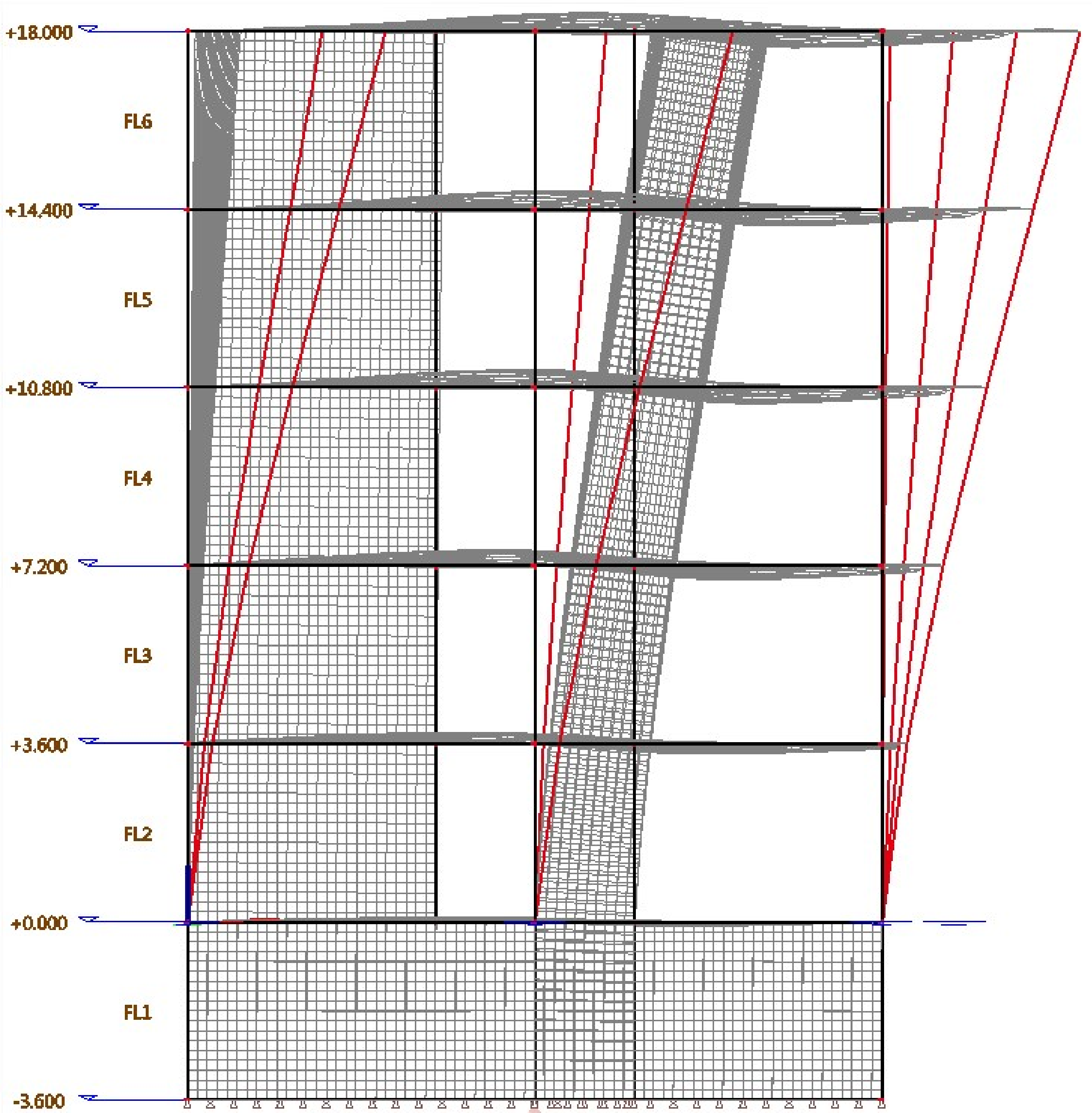
This method corresponds to the simplified approach defined in EN1998-1 § 4.3.3.2.3(3) and formula (4.11).

Distribution of accelerations from eigenshape

Accidental eccentricity	
Accidental eccentricity	Distribution of accelerations from eigenshape
Eccentricity	0.05
Mode shape	1

In this case, the distribution of accelerations is assumed to be proportional to the displacements of the structure in the relevant mode shape. The user must specify the reference mode (fundamental mode).

When the mode shape selection is set as "automatic", the program selects the mode that has the highest modal mass in the direction of the seismic action.



$$\alpha_j = U_{G,j}$$

where $U_{G,j}$ is the modal displacement of the mass center of storey j in the direction of the seismic action, obtained from modal analysis of the reduced model.

This is the preferred approach in Eurocode 8, defined in EN1998-1 § 4.3.3.2.3(2) and formula (4.10).

Accelerations from modal superposition

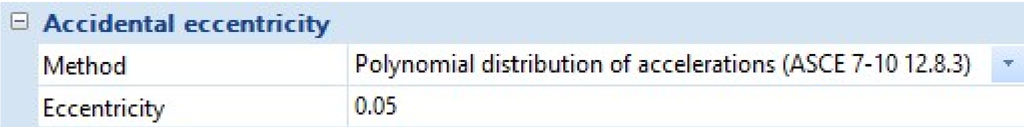
In this case, no distribution key is used. The accelerations are obtained directly from the seismic load case after modal superposition. The acting forces on storeys are obtained as follows:

$$F_j = a_{Gj} \cdot m_j$$

where a_{Gj} is the acceleration at the mass center of storey j in the direction of the seismic action obtained from the modal superposition in the reduced model.

This approach is not described in the Eurocode 8. It is more conservative than the other approaches, as it uses an envelope of accelerations instead of a distribution of the resultant base shear. However, it has the advantage of covering cases where higher order modes cannot be neglected for accidental eccentricity.

Polynomial distribution of accelerations (ASCE 7-10 12.8.3)

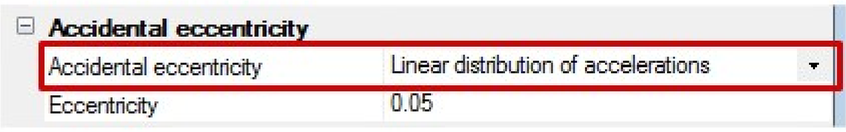


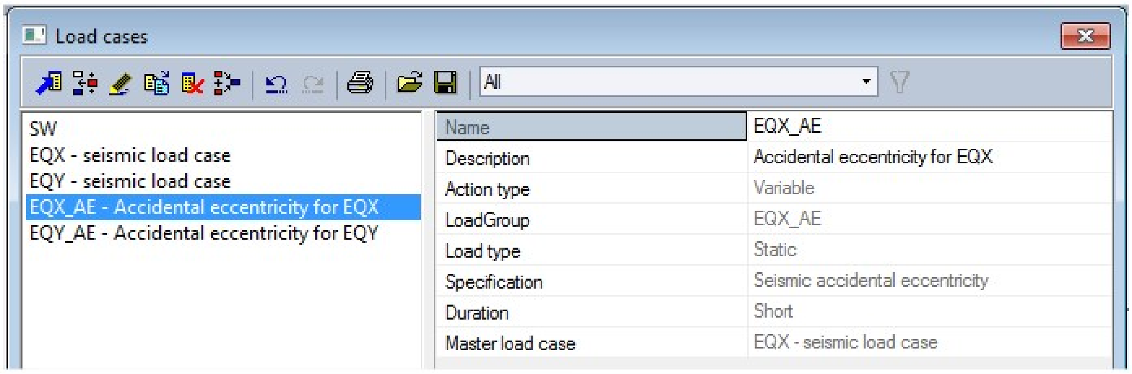
This method is available only when using the ELF method for the seismic analysis (static method). It is similar to the linear distribution method, but uses a different formula to calculate the vertical distribution of the horizontal acceleration. See the chapter about the ELF method for more details.

Analysis & results of accidental eccentricity

Accidental eccentricity load cases

When enabling accidental eccentricity in a seismic load case, the program will automatically create an accidental eccentricity "slave" load case.





Accidental eccentricity load cases are read-only and cannot be deleted. To remove an AE load case, disable accidental eccentricity in the corresponding seismic load case.

All properties of AE load cases are read-only, except their name and description which may be edited by the user. The default values of the properties are:

Name
name of the source seismic load case with added suffix "_AE"

Description
"Accidental eccentricity for XXX" where XXX is the name of the master seismic load case

Action type
Variable

Load group
identical to Name; see next paragraph for details

Load type
static; the torsional effect is indeed computed as a set of static loads (moments) applied to the structure

Specification
Seismic accidental eccentricity

Duration
Short

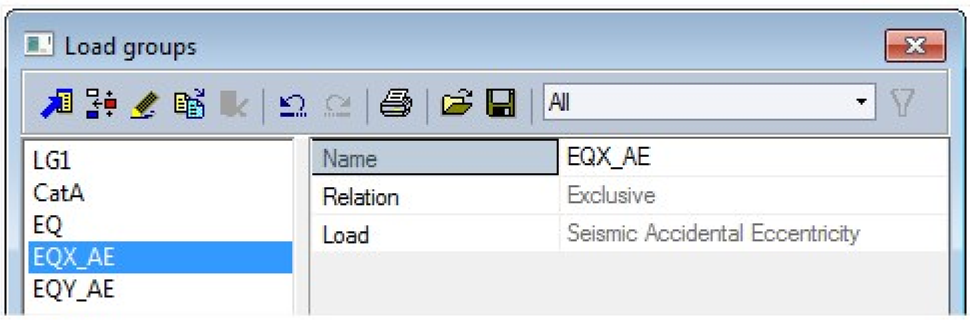
Master load case
the master seismic load case; thanks to this, the accidental eccentricity will be applied in envelopes only when the corresponding seismic action is also present in the envelope

The content of an AE load case cannot be viewed nor edited. No loads can be added to it. The applied moments due to the accidental eccentricity are automatically computed during the analysis.

The generated AE load cases may be inserted in seismic envelope combinations in order to account for AE effects. Their results may also be viewed separately for validation.

Load groups

For each seismic load case with enabled accidental effect, the program will create automatically a read-only load group with the following properties:



Name
same as the corresponding AE load case

Relation
same as the relation of the load group of the source seismic load case

Load
Seismic accidental eccentricity

This load group is automatically assigned to the corresponding AE load case.

Combinations

When using an AE load case in an envelope combination together with its source seismic load case, it will automatically be combined with the seismic action, assigning + and – sign to it. The AE load case will not be taken into account without its source seismic load case.

The AE load cases must be added manually to combinations.

A typical use case:

		Possible combinations					
LC	Description	1	2	3	4	5	6
SW	Self weight (static)	1	1	1	1	1	1
EQX	Seismic load case (dynamic)	1	-1	1	-1	1	-1
EQX_AE	Accidental eccentricity load case for EQX (static)			1	1	-1	-1

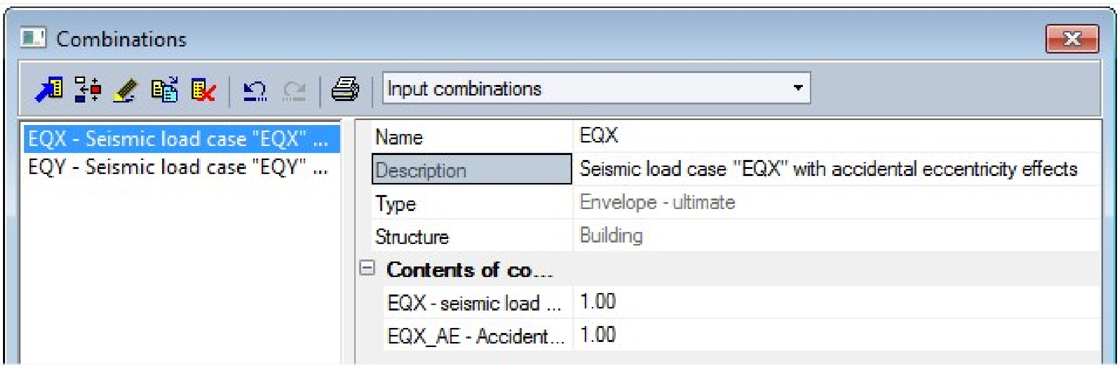
Reminder: this table shows the principle of combination, but it is not applied strictly as such. When using a seismic envelope combination in checks, the program also uses **for unsigned seismic load cases** other cases where the sign of internal forces components are switched independently to account for the non-concomitance of extremes after modal superposition in the response spectrum method.

Reminder: when dealing with seismic load cases, for the reason above, use only “envelope” or “code” combinations. Do not use “linear” combinations, nor explode envelope or code combinations to linear.

Generated combinations

As mentioned previously, AE load cases must be added manually to load case combinations.

However, for convenience, the program generates automatically an envelope combination for each seismic load case, containing the source seismic load case and its AE load case and named after the source seismic load case. This allows to check easily the results for the full seismic action, including the effects of accidental eccentricity.



Equivalent Lateral Forces (ELF)

Introduction

Seismic ELF analysis is the most well known method for the seismic analysis of structures. Although it is quite conservative, its simplicity makes it a very popular method for seismic design.

The ELF method is a static analysis method. However, using it in SCIA Engineer requires the input of some data related to dynamic analysis: masses and at least one combination of mass groups must be defined, as the calculation of the seismic equivalent lateral forces is based on the distribution of masses in the structure. The calculation of storey forces is based on the definition of storeys as well as on the reduced system, which must therefore be defined in order to allow using the ELF analysis.

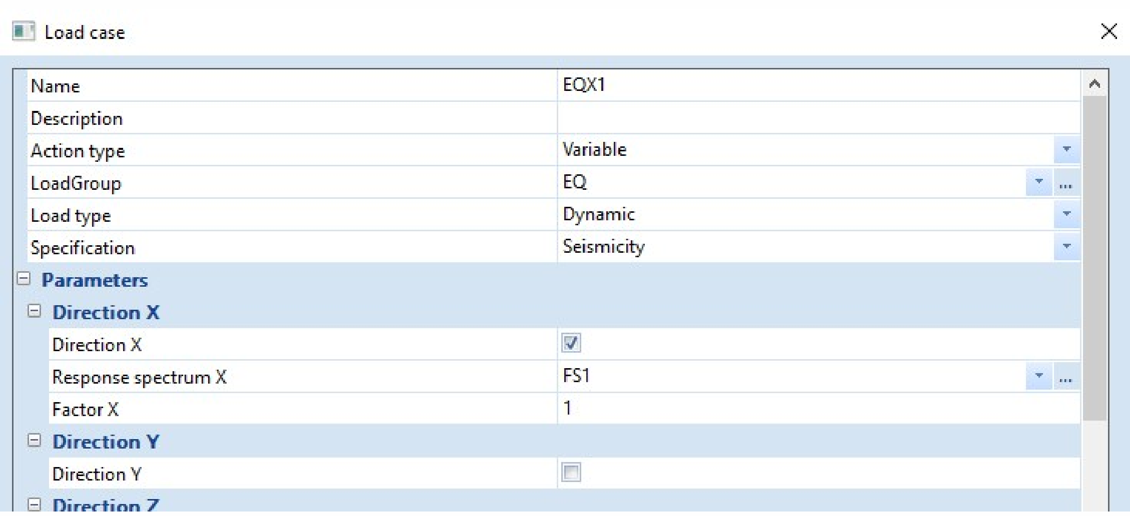
Defining an ELF seismic load case

Pre-requisites - before creating an ELF seismic load case

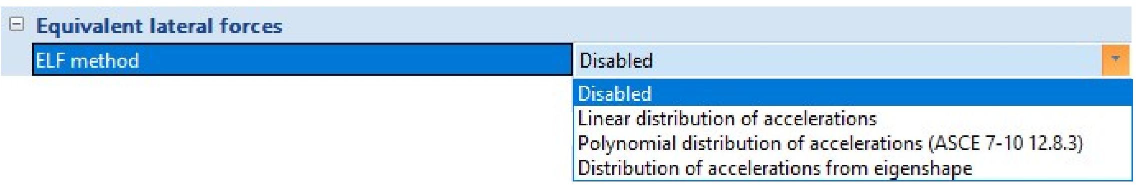
- enable dynamics and seismic analysis in the project settings
- define masses and a combination of mass groups (same as for any dynamic load case)
- define storeys
- enable the reduced model

Then create an ELF seismic load case

- create a new load case
- select *Action type = Variable*
- select *Load type = Dynamic*
- select *Specification = Seismicity*



Then, in order to set the seismic load case to ELF, go to the settings group *Equivalent lateral forces* and select the desired ELF method (disabled by default)



A this point, most of the settings of a standard seismic load case will disappear from the dialogue, as many of them are relevant only for a dynamic response spectrum analysis. For an ELF load case, the following remain:

Direction X (resp. Y, Z)
Tickbox that enables the seismic action in the X (resp. Y, Z) direction

Response spectrum X (resp. Y, Z)
Selection of the seismic response spectrum for direction X (resp. Y, Z)

Factor X (resp. Y, Z)

Multiplying factor for the seismic action in direction X (resp. Y, Z)

Acceleration factor
Multiplying factor for the entire seismic action

Overturning reference level
Reference level for the calculation of overturning moments. Also used as reference level when using a linear or polynomial distribution of accelerations

Accidental eccentricity
Accidental eccentricity settings act the same as for dynamic seismic analysis

Mass combi
Mass combination selected for the calculation of the seismic action

Specific settings for an ELF seismic load case

Equivalent lateral forces

ELF method
Defines how the distribution of accelerations will be calculated in the building. The various methods are detailed in the next section. The available choices are:

> **Disabled**
> ELF calculation of the seismic load case is disabled, i.e. the load case will use dynamic, multi-modal response spectrum analysis
>
> **Linear distribution of accelerations**
> The acceleration is increasing linearly with the global Z coordinate, starting with zero at the overturning reference level (see above)
>
> **Polynomial distribution of accelerations (ASCE 7-10 12.8.3)**
> The acceleration is increasing according to a polynomial function, according to ASCE 7-10 standard, starting with zero at the overturning reference level (see above)
>
> **Distribution of accelerations from eigenshape**
> The acceleration is distributed proportionally to the mode shape of an eigenmode selected by the user

Seismic force from
Defines how the total seismic force (base shear) is calculated. The available options depend on the selected ELF method.

> **Max acceleration of spectrum**
> The maximum acceleration of the selected response spectrum is used
>
> **Input fundamental period**
> The acceleration corresponding to a user input value of fundamental period is used
>
> **Selected eigenmode**
> The acceleration corresponding to the period of the user selected eigenmode is used; please note, that this implies a modal analysis of the structure
>
> **Approximate fundamental period (ASCE 7-10 12.8-7)**
> The acceleration corresponding to approximate fundamental period according ASCE 7-10 eq. 12.8-7. This is available only when *ELF method = Polynomial distribution of accelerations (ASCE 7-10 12.8.3)*

Fundamental period
User input fundamental period to be used for the calculation of the seismic force when *Seismic force from = Input fundamental period*

Mode shape
Eigenmode to be used for the distribution of the accelerations and/or the calculation of the seismic force

ASCE additional parameters

Structure type
This item is for values of approximate period parameters, see ASCE 7-10 table 12.8-2

> **Steel moment frame**

Concrete moment frame
Steel eccentrically braced frames
Steel buckling-restrained braced frames
All other structural systems

Structural height from model
When this option is ticked, the structural height is obtained from the model as the maximal z-coordinate of top storey in project minus the *Overturning reference level* (Reference level for the calculation of overturning moments). When it is not ticked, the structural height must be defined manually.

h_n (structural height)
Structural height of building, used for the calculation of the approximate fundamental period according to ASCE 7-10 eq. 12.8-7.

C_t
Coefficient for ASCE 7-10 eq. 12.8-7

x
Exponent for ASCE 7-10 eq. 12.8-7

T_a [s]
Approximate fundamental period, see ASCE 7-10 section 12.8.2.1 and equation 12.8-7

Limit fundamental period
When this option is ticked, the fundamental period used for the calculation of the seismic force is limited by the upper boundary value defined in ASCE 7-10 section 12.8.2 .

C_u
Coefficient for upper limit on calculated period, see ASCE 7-10 table 12.8-1

Maximum fundamental period [s]
Maximum fundamental period according to ASCE 7-10 section 12.8.2, defined as T_a multiplied by C_u.

Calculation of the Equivalent Lateral Forces

Equivalent Lateral Forces are applied as one concentrated force at the mass center of each storey of the building.

First, the total seismic force (base shear) is calculated. It is then distributed to the storeys according to the selected method (vertical distribution of accelerations). The same procedure is applied for each direction (X,Y,Z).

Calculation of the seismic force

The total seismic force F_{base} is calculated as follows

$$F_{base} = M_{tot} \cdot a_{ref} \cdot c_{dir} \cdot c_{acc}$$

where

M_{tot} is the total mass of the structure, obtained from the selected mass combination

a_{ref} is the reference acceleration, obtained from the selected seismic response spectrum

c_{dir} is the *direction factor* defined in the seismic load case settings

c_{acc} is the *acceleration factor* defined in the seismic load case settings

The value of a_{ref} is extracted from the response spectrum, according to the *Seismic force from* setting

Max acceleration of spectrum: the maximum value of acceleration defined in the selected spectrum

Input fundamental period: the value of acceleration corresponding to the period defined in the *Fundamental period* setting

Selected eigenmode: the value of acceleration corresponding to the period of the eigenmode selected in the *Mode shape* setting

Approximate fundamental period (ASCE 7-10 12.8-7) : the value of acceleration corresponding to the calculated approximate fundamental period

In the 2nd and 3rd cases above, the used fundamental period may be reduced by the calculated upper limit, if *Limit fundamental period* is ticked

Additionally, when a seismic response spectrum according to ASCE 7-10 is used (spectrum generator for IBC standard), the reference acceleration a_{ref} cannot be less than the specified minimum acceleration according to section 12.8.1.1:

$$a_{ref} \geq C_{s,\min} \cdot g$$

where

$$C_{s,\min} = \max(\, 0.044 \cdot S_{DS} \cdot I_e \quad ; \quad 0.01)$$

additionally, if $S_1 \geq 0.6$ the following applies:

$$C_{s,\min} \geq \frac{0.5 \cdot S_1}{R/I_e}$$

The parameters S_{DS}, S_1, I_e and R are defined in the settings of the IBC response spectrum generator.

g is the acceleration of gravity, defined in the project settings.

Distribution of the seismic force to the storeys

The storey force for the *j-th* storey j is calculated as follows

$$F_j = F_{base} \cdot \frac{\alpha_j \cdot m_j}{\sum_k \alpha_k \cdot m_k}$$

where

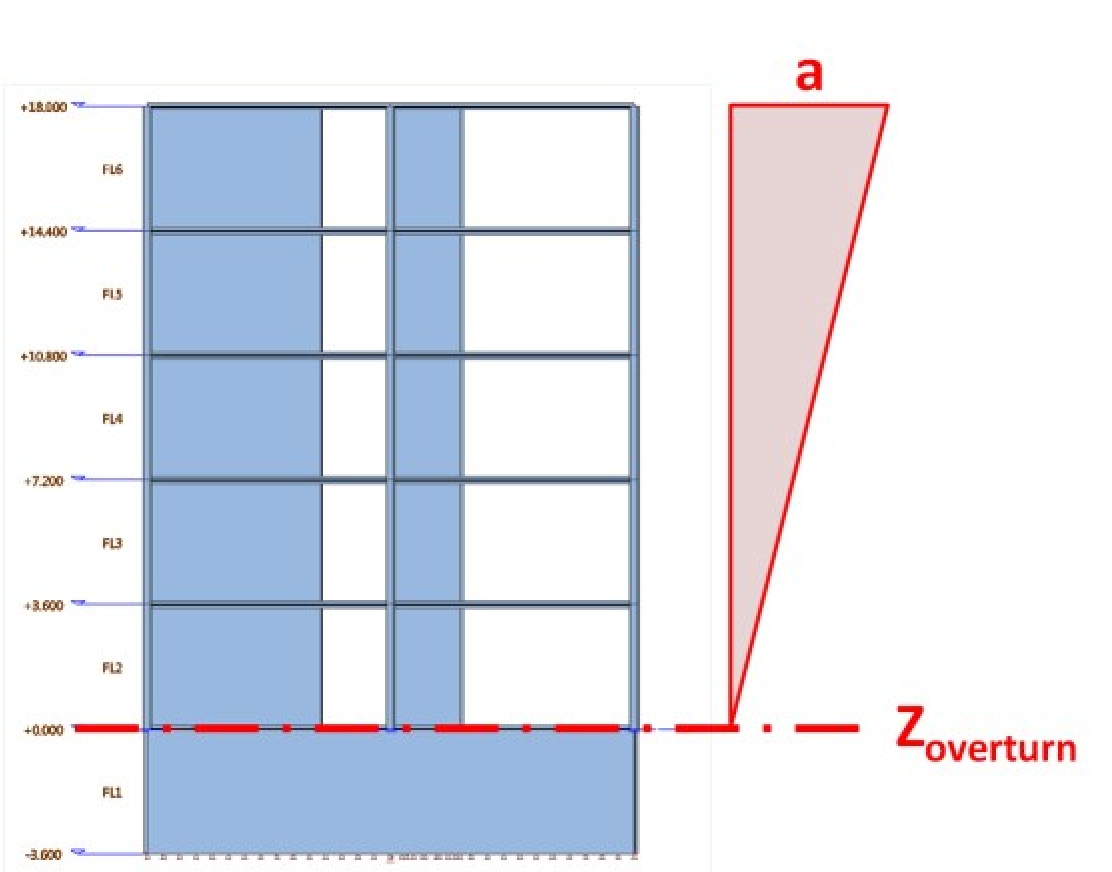
F_j is the horizontal force acting on storey j

F_{base} is the total horizontal force acting on the structure (see above)

m_j is the mass of storey j

α_j is the distribution key of the accelerations, according to the selected *ELF method* setting

Linear distribution of accelerations



$$\alpha_j = \max(z_j - z_{overturn}; 0)$$

where

z_j is the level of the mass center of storey *j*

$z_{overturn}$ is the *Overturning reference level*, defined by the user in the properties of the seismic load case.

This method corresponds to the simplified approach defined in EN1998-1 § 4.3.3.2.3(3) and formula (4.11).

Polynomial distribution of accelerations

$$\alpha_j = (\max(z_j - z_{overturn}; 0))^k$$

$$k = 1 \leq \left(1 + \frac{T - 0.5}{2}\right) \leq 2$$

where

z_j is the level of the mass center of storey *j*

$z_{overturn}$ is defined by the user in the properties of the seismic load case

T is the reference fundamental period, depending on the selected *Seismic force from* setting:

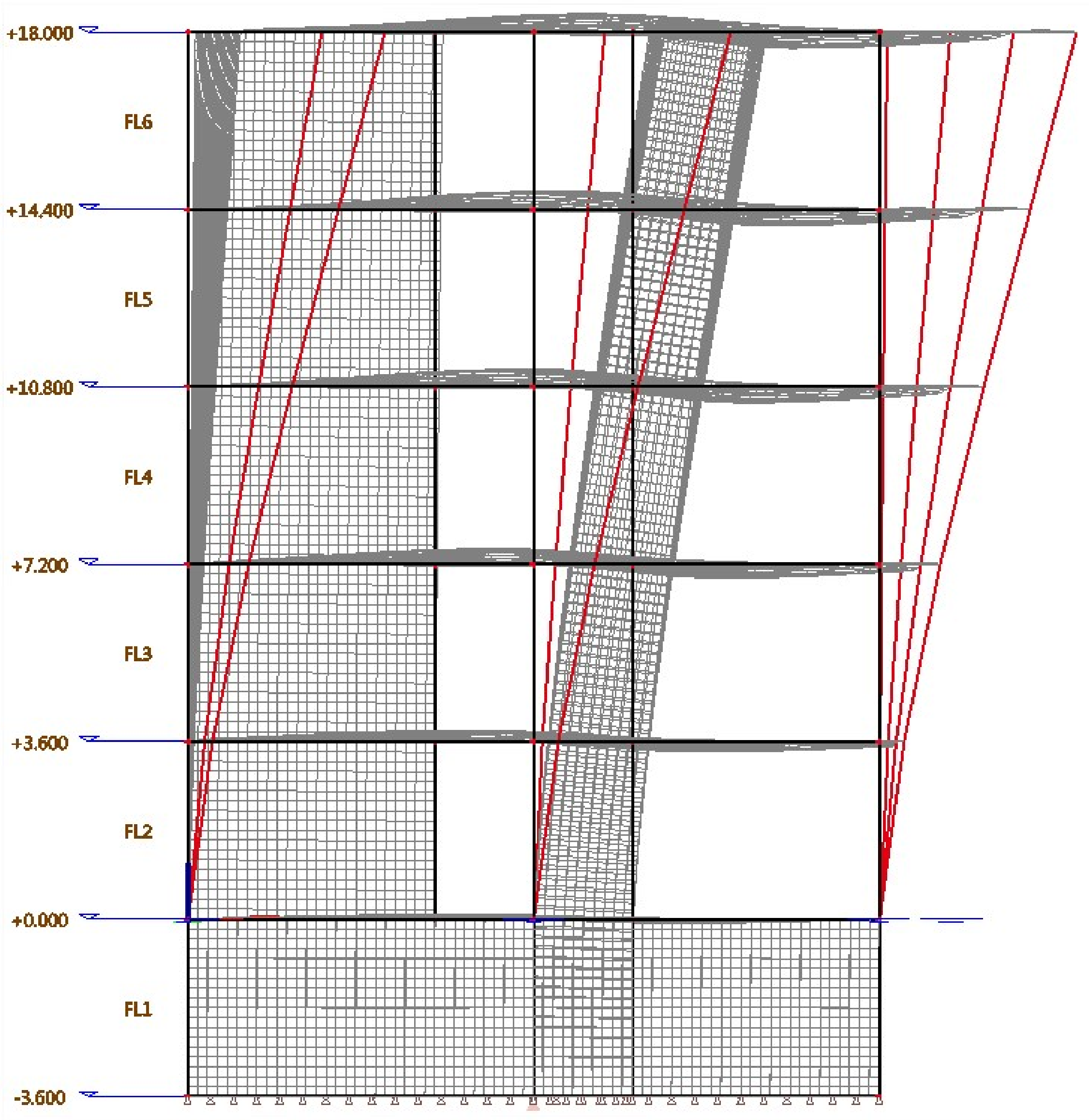
Max acceleration of spectrum: *T* is unknown, the conservative value *k*=2 is used

Input fundamental period: *T* is the period defined in the *Fundamental period* setting

Selected eigenmode: T is the period of the eigenmode selected in the *Mode shape* setting

This method corresponds to the approach defined in ASCE 7-10 section 12.8.3

Distribution of accelerations from eigenshape



$$\alpha_j = U_{G,j}$$

where

> $U_{G,j}$ is the modal displacement of the mass center of storey j in the direction of the seismic action, obtained from modal analysis of the reduced model.

This is the preferred approach in Eurocode 8, defined in EN1998-1 § 4.3.3.2.3(2) and formula (4.10).

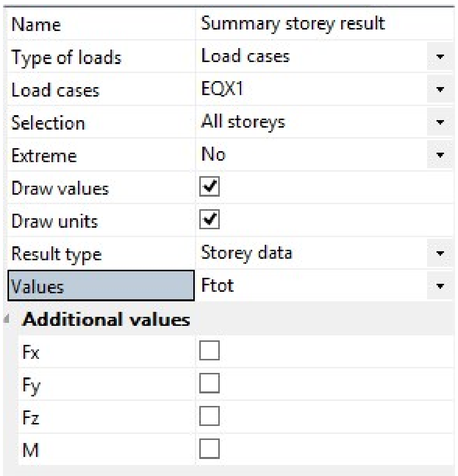
Application of the storey forces to the model

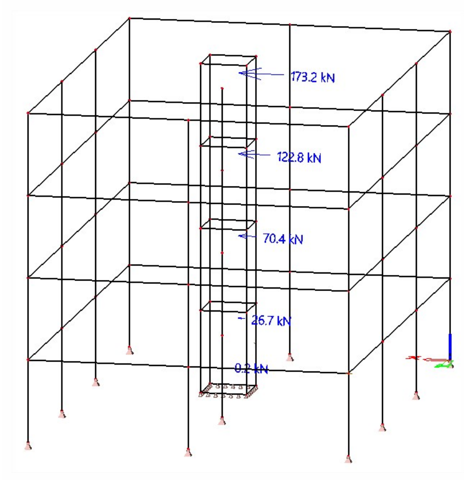
The calculated storey forces are applied to the structure using the reduced system. The transformation matrices of the IRS method allow to "smear" the concentrated storey forces in such a way that the resultant of each storey force is applied at the mass center of the corresponding storey. The loads are, however, applied in a distributed way to the entire storey, hence avoiding any numerical singularity, as would be the case if point loads would be applied in a conventional way.

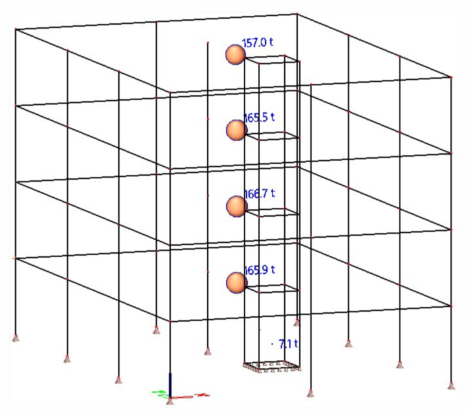
Results

As an ELF load case is fundamentally a static load case, all standard result output can be used in SCIA Engineer, without restriction. Also, because it is a static load case, none of the issues related to the loss of sign due to the modal superposition apply here.

Additionally, the Summary Storey Results service allows to display the storey forces applied to the structure.







Summary storey result

Storey data:

Linear calculation, Extreme: No, System: Principal

Selection: All

Load cases : EQX1

Equivalent Lateral Forces (ELF) settings

ELF method	Polynomial distribution of accelerations (ASCE 7-10 12.8.3)
Seismic force from	Selected eigenmode
Fundamental period [s]	1.28
Distribution factor k	1.39
Mode shape	1

Equivalent Lateral Forces (ELF) per storey

Name	M [t]	Zg [m]	Fx [kN]
FL1	7.1	0.993277	**0.2**
FL2	165.9	3.603689	**26.7**
FL3	166.7	7.204320	**70.4**
FL4	165.5	10.800000	**122.8**
FL5	157.0	14.354149	**173.2**
Total	**662.2**		**393.4**

References

[1]
Guyan, R.J., Reduction of Stiffness and Mass Matrices, AIAA Journal, Vol. 3, No. 2, February, 1965

[2]
O'Callahan, j., A Procedure for an Improved Reduced System (IRS) Model, Proceedings of the 7th International Modal Analysis Conference, Las Vegas, Nevada, February, 1989

[3]
EC-EN 1998-1, Eurocode 8 - Design of structures for earthquake resistance - Part 1: General rules, seismic actions and rules for buildings

[4]
Kabeláč, J., Rossier, S., Reduction for High-Rise Buildings Seismic Analysis, 20th International Conference Engineering Mechanics, Svratka, Czech Republic, May, 2014

General Plastic Analysis

A general plastic analysis can be carried out in SCIA Engineer for any 2D members (plates, walls, shells).

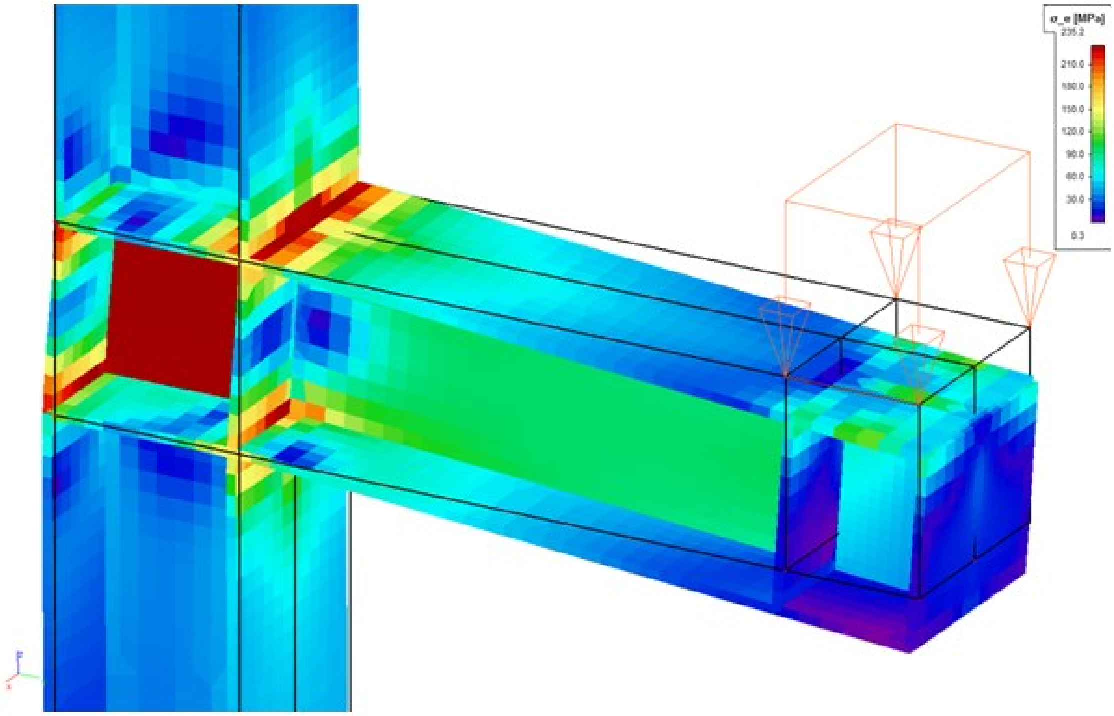
There are four types of general plasticity in Scia Engineer : Tresca, von Mises, Drucker-Prager and Mohr-Coulomb.

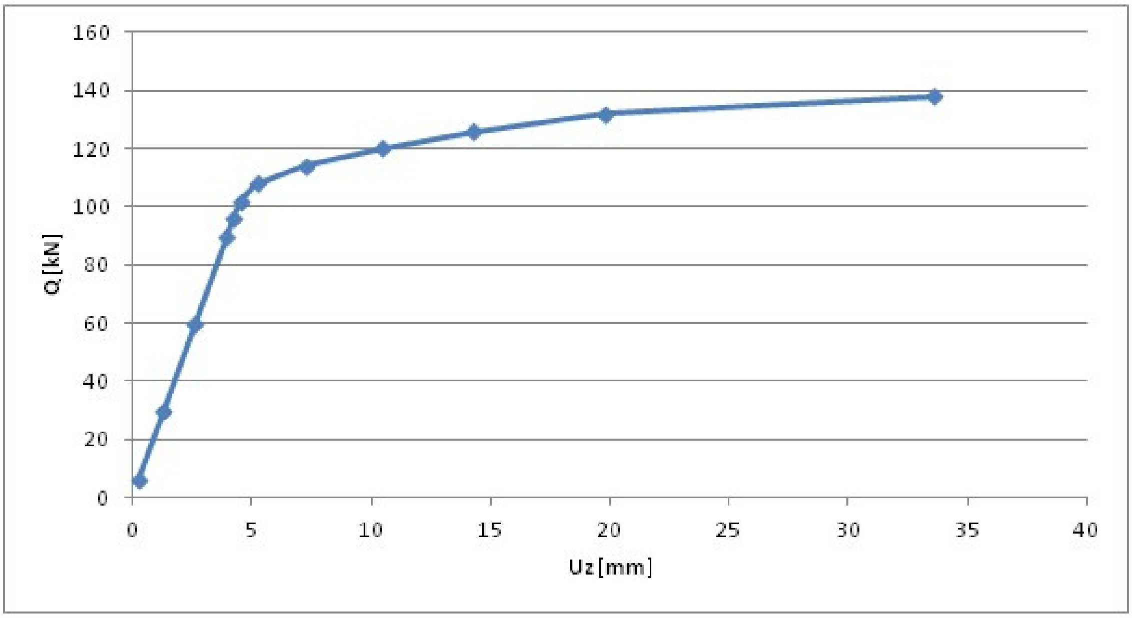
Tresca and The von Mises yield conditions are is currently available, which is suitable for ductile materials in general, such as metals (steel, aluminium...). It is a symmetric behaviour, acting in the same way in tension and compression, with or without hardening in the plastic branch. Drucker-Prager and Mohr-Coulomb yield conditions are suitable for materials with different strength in tension and compression (concrete, soil). More types of plastic behaviours will be added in further versions.

The plastic behaviour of materials may be combined with other types of non-linearity in SCIA Engineer. For instance, plasticity, press only supports and large displacement analysis can be used together. Tension only 1D members with a plastic limit forces may be used to model the behaviour of bolts in a connection.

The typical first application of general plasticity is the detailed analysis of non-standard steel construction connections, where simplified methods do not apply. It may however be applied to any structure that can be modelled using 2D members.

Plasticity is not supported yet for 1D members. Any beam or truss member that is present in the model will be considered as elastic.





Theoretical background

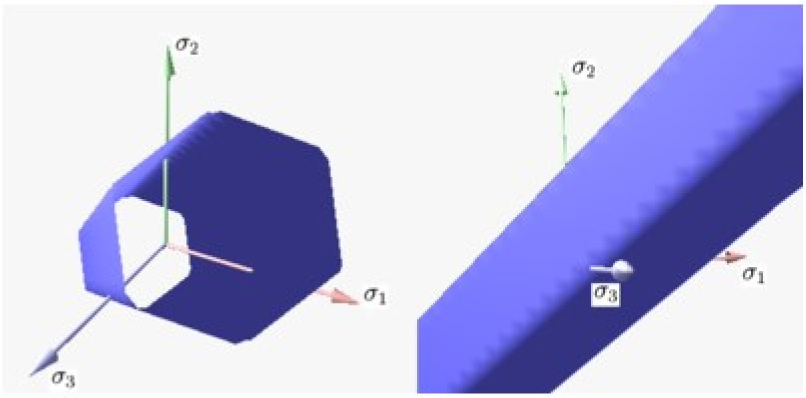
Tresca yield criterion

The **Tresca yield criterion** is taken to be the work of Henri Tresca. It is also known as the maximum shear stress theory (MSST) and the Tresca–Guest (TG) criterion. In terms of the principal stresses the Tresca criterion is expressed as:

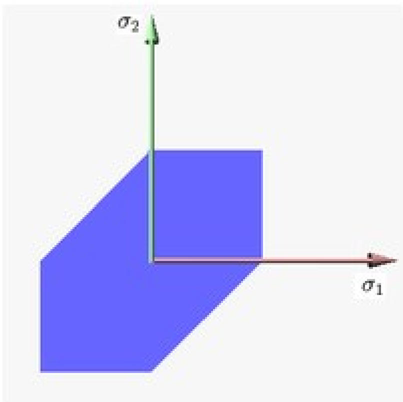
$$\frac{1}{2}\max(|\sigma_1 - \sigma_2|, |\sigma_2 - \sigma_3|, |\sigma_3 - \sigma_1|) = S_{sy} = \frac{1}{2}S_y$$

Where S_{sy} is the yield strength in shear, and S_y is the tensile yield strength.

On the first picture there is the Tresca–Guest yield surface in the three-dimensional space of principal stresses. It is a prism of six sides and having infinite length. This means that the material remains elastic when all three principal stresses are roughly equivalent (a hydrostatic pressure), no matter how much it is compressed or stretched. However, when one of the principal stresses becomes smaller (or larger) than the others the material is subject to shearing. In such situations, if the shear stress reaches the yield limit then the material enters the plastic domain.



On the second picture there is the Tresca–Guest yield surface in two-dimensional stress space, it is a cross section of the prism along the plane.



source: Wikipedia http://en.wikipedia.org/wiki/Yield_surface#Tresca_yield_surface

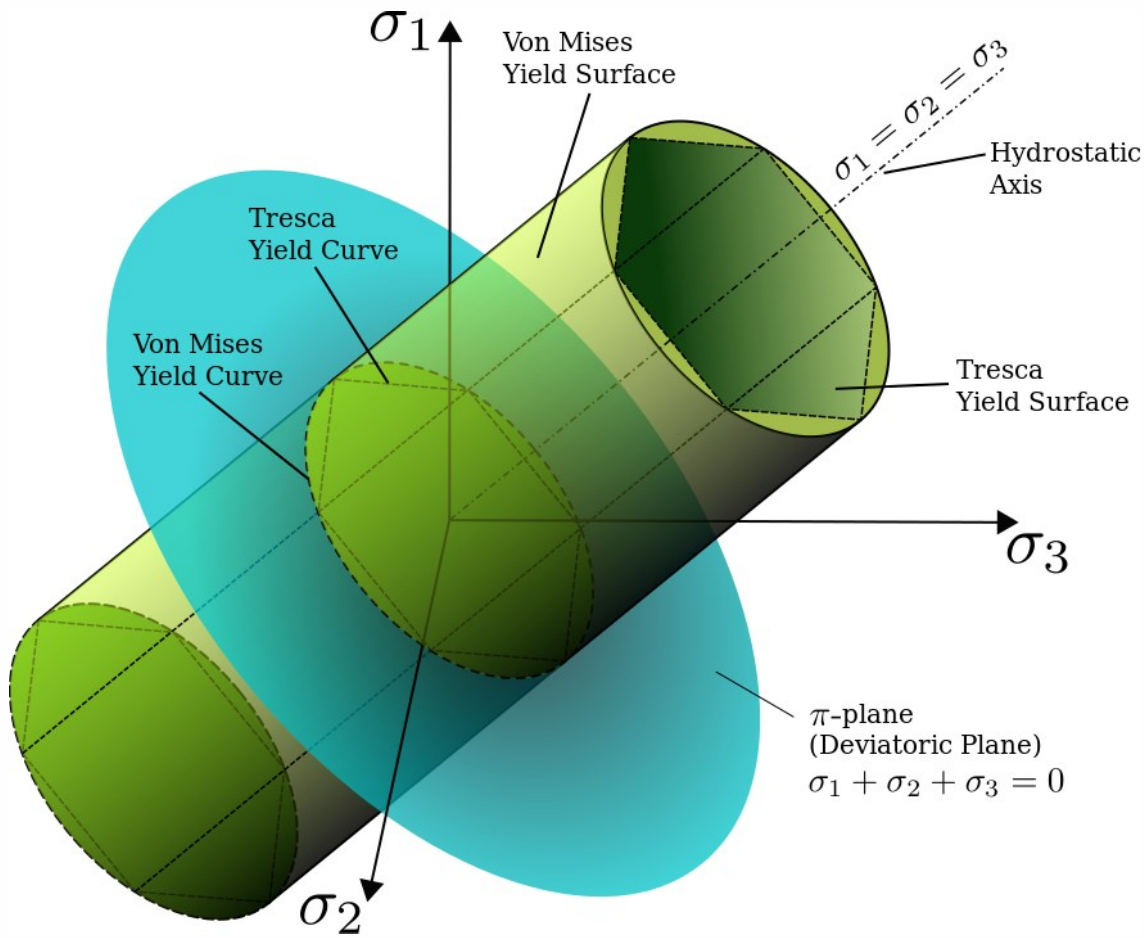
Von Mises yield criterion

The **von Mises yield criterion** suggests that the yielding of materials begins when the second deviatoric stress invariant J_2 reaches a critical value. For this reason, it is sometimes called the J2-plasticity or J2 flow theory. It is part of a plasticity theory that applies best to ductile materials, such as metals. Prior to yield, material response is assumed to be elastic.

In materials science and engineering the von Mises yield criterion can be also formulated in terms of the von Mises stress or equivalent tensile stress, σ_E, a scalar stress value that can be computed from the Cauchy stress tensor. In this case, a material is said to start yielding when its von Mises stress reaches a critical value known as the yield strength, σ_y. The von Mises stress is used to predict yielding of materials under any loading condition from results of simple uniaxial tensile tests. The von Mises stress satisfies the property that two stress states with equal distortion energy have equal von Mises stress.

Because the von Mises yield criterion is independent of the first stress invariant, I_1, it is applicable for the analysis of plastic deformation for ductile materials such as metals, as the onset of yield for these materials does not depend on the hydrostatic component of the stress tensor.

Although formulated by Maxwell in 1865, it is generally attributed to Richard Edler von Mises (1913). Tytus Maksymilian Huber (1904), in a paper in Polish, anticipated to some extent this criterion. This criterion is also referred to as the Maxwell–Huber–Hencky–von Mises theory.



The formulation of the von Mises comparison stress in a general 3D stress-state is given by:

$$\sigma_E = \sqrt{\frac{1}{2} \cdot \left[(\sigma_{11} - \sigma_{22})^2 + (\sigma_{22} - \sigma_{33})^2 + (\sigma_{33} - \sigma_{11})^2 + 6 \cdot (\sigma_{12}^2 + \sigma_{23}^2 + \sigma_{31}^2) \right]}$$

source: Wikipedia http://en.wikipedia.org/wiki/Von_Mises_yield_criterion

Drucker-Prager yield criterion

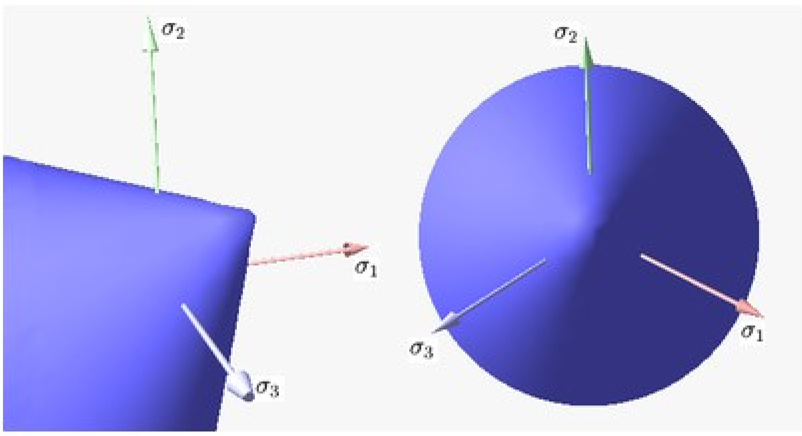
The **Drucker-Prager yield criterion** is similar to the von Mises yield criterion, with provisions for handling materials with differing tensile and compressive yield strengths. This criterion is most often used for concrete where both normal and shear stresses can determine failure. The Drucker–Prager yield criterion may be expressed as

$$\left(\frac{m-1}{2}\right)(\sigma_1+\sigma_2+\sigma_3)+\left(\frac{m+1}{2}\right)\sqrt{\frac{(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2}{2}}=S_{yc}$$

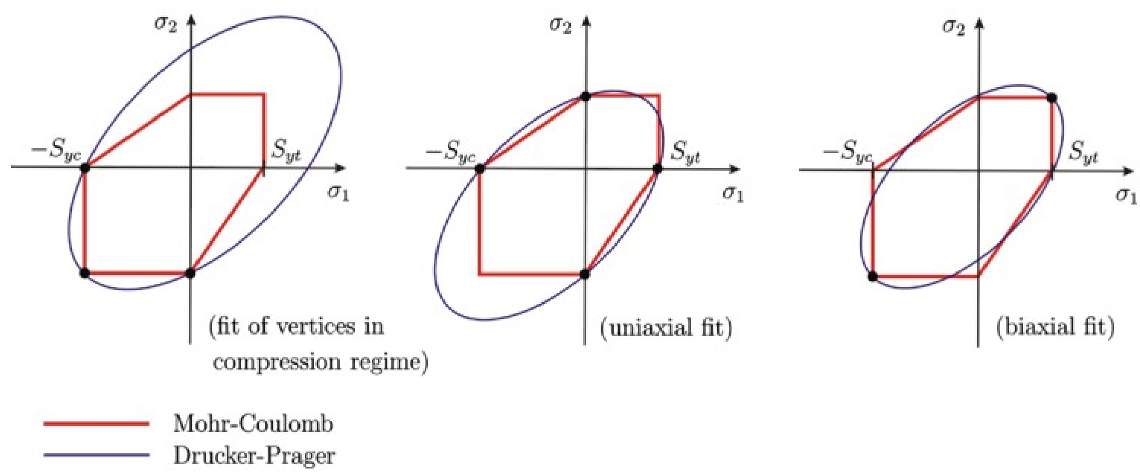
where

$m=\frac{S_{yc}}{S_{yt}}$ where S_{yc} and S_{yt} are the uniaxial yield stresses in compression and tension respectively. The formula reduces to the von Mises equation if $S_{yc}=S_{yt}$.

On the first picture there is Drucker–Prager yield surface in the three-dimensional space of principal stresses. It is a regular cone.



On the second picture there is Drucker–Prager yield surface in two-dimensional space. The elliptical elastic domain is a cross section of the cone on the plane of σ_1 ;σ_2.



It can be chosen to intersect the Mohr–Coulomb yield surface in different number of vertices. One choice is to intersect the Mohr–Coulomb yield surface at three vertices on either side of the $\sigma_1=-\sigma_2$ line, but usually selected by convention to be those in the compression regime. Another choice is to intersect the Mohr–Coulomb yield surface at four vertices on both axes (uniaxial fit) or at two vertices on the diagonal $\sigma_1 = \sigma_2$ (biaxial fit).The Drucker-Prager yield criterion is also commonly expressed in terms of the material cohesion and friction angle.

source: Wikipedia http://en.wikipedia.org/wiki/Yield_ surface#Drucker.E2.80.93Prager_ yield_surface

Mohr-Coulomb yield criterion

The **Mohr-Coulomb yield** is similar to the Tresca criterion, with additional provisions for materials with different tensile and compressive yield strengths. This model is often used to model concrete, soil or granular materials. The Mohr–Coulomb yield criterion may be expressed as:

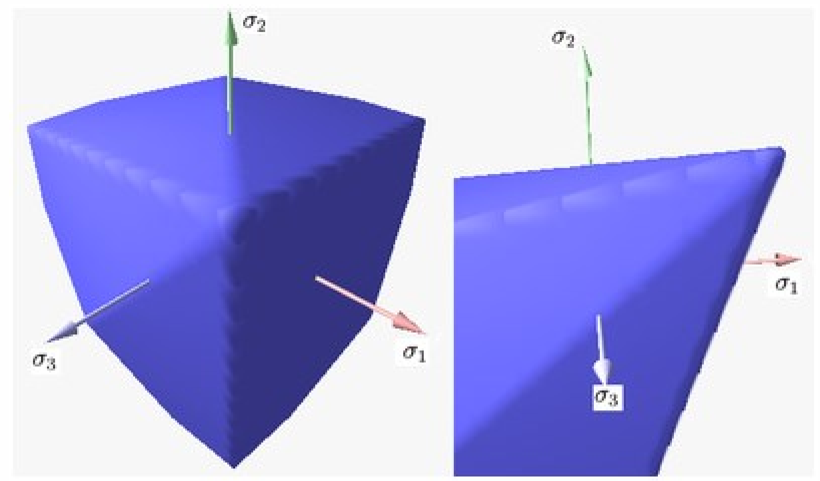
$$\frac{m+1}{2}\max(|\sigma_1-\sigma_2|+K(\sigma_1+\sigma_2),|\sigma_1-\sigma_3|+K(\sigma_1+\sigma_3),|\sigma_2-\sigma_3|+K\left(\sigma_2+\sigma_3\right))=S_{yc}$$

Where

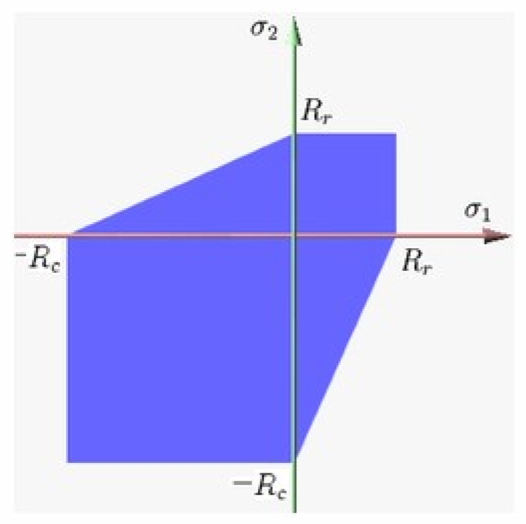
$$m=\frac{S_{yc}}{S_{yt}} S_{sy}\,;\ K=\frac{m-1}{m+1}$$

and the parameters S_{yc} and S_{yt} are the yield (failure) stresses of the material in uniaxial compression and tension, respectively. The formula reduces to the Tresca criterion if $S_{yc}=S_{yt}$.

On the first picture there is Mohr–Coulomb yield surface in the three-dimensional space of principal stresses. It is a conical prism and determines the inclination angle of conical surface.



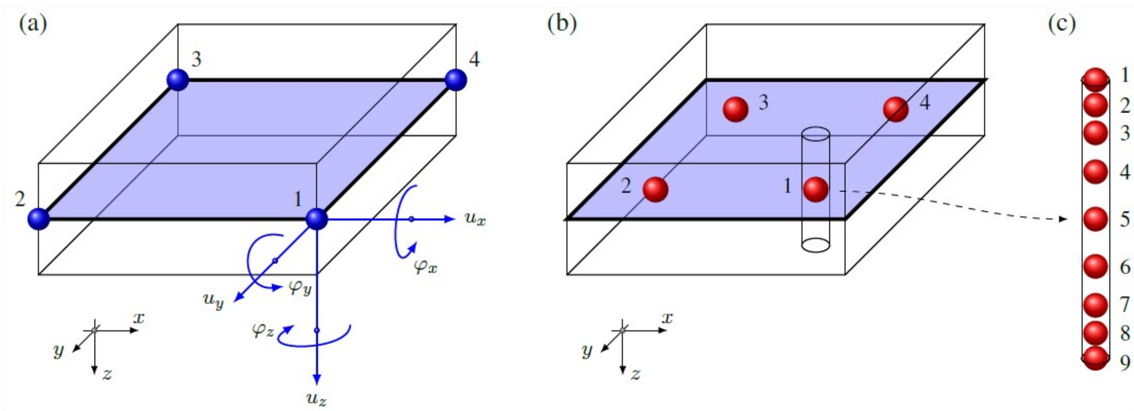
On the second picture there is Mohr–Coulomb yield surface in two-dimensional stress space. It is a cross section of this conical prism on the plane of σ_1, σ_2.



source: Wikipedia http://en.wikipedia.org/wiki/Yield_ surface#Mohr.E2.80.93Coulomb_ yield_surface

Finite element model

Drilling rotations at each node is used for in-plane loading. This means that element has six degrees of freedom at each node and is therefore compatible with other types of elements (beam/solid elements). Within the element area the Gauss 2x2 quadrature points are used. Each of these Gauss quadrature points is realized by nine Gauss-Lobatto quadrature points throughout the thickness, so the four-node element has 2x2x9=36 quadrature points in total. Due to these Gauss-Lobatto points the element can handle bending loading with high accuracy. In all of these points the nonlinear model is computed independently using the plane stress formulation. Linear transversal shear stiffness is assumed.



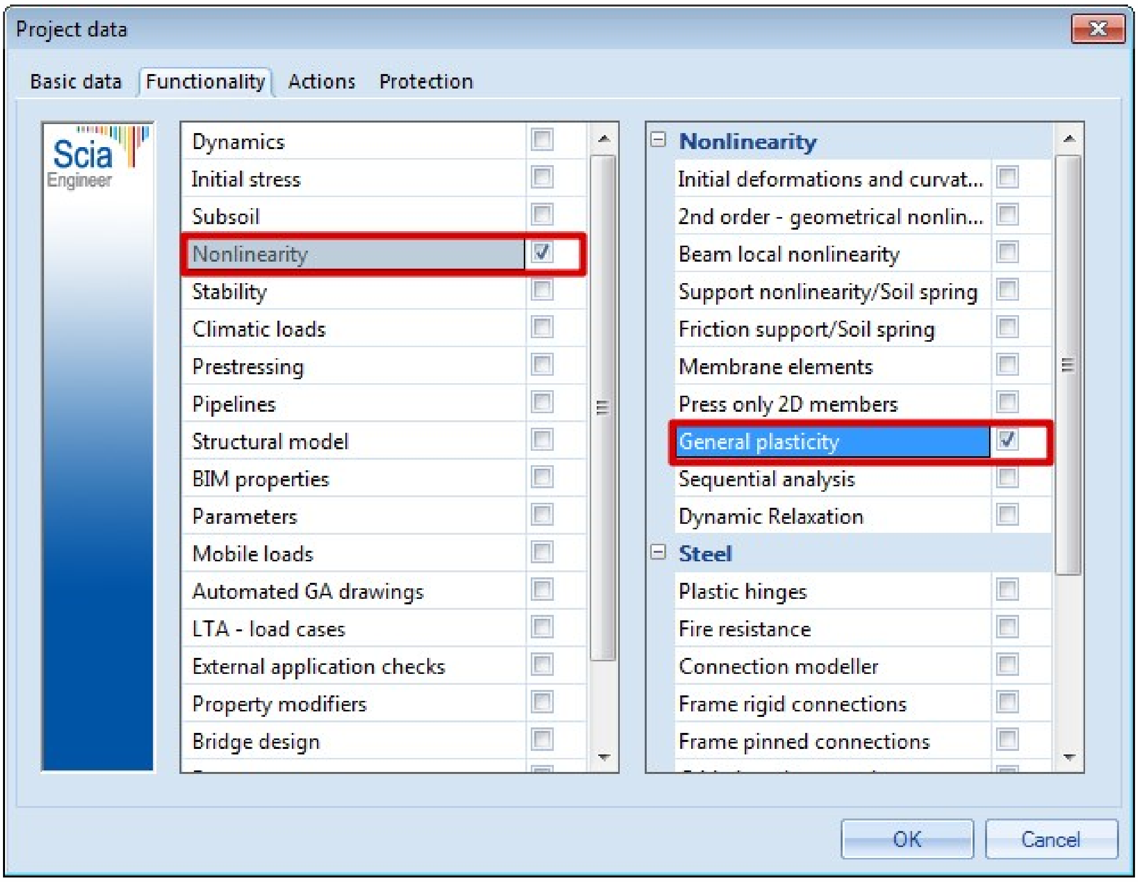
About Gauss-Lobatto quadrature: https://en.wikipedia.org/?title=Gaussian_quadrature

Using general plasticity in SCIA Engineer

General plasticity is a specific type of non-linearity in SCIA Engineer. After defining the suitable data in the project a non-linear analysis must be carried out to calculate the plastic behaviour of the structure. Please refer to the general information about non-linear analysis in SCIA Engineer.

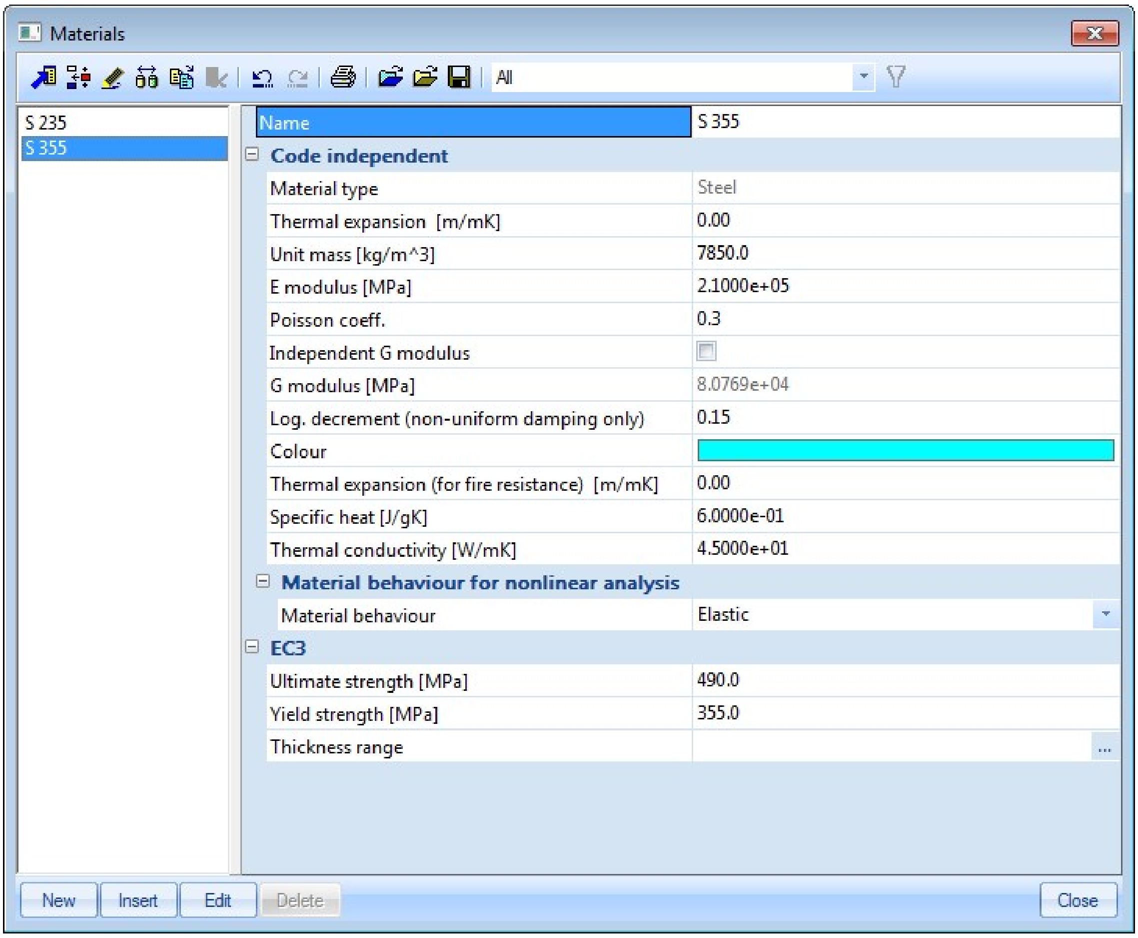
Project settings

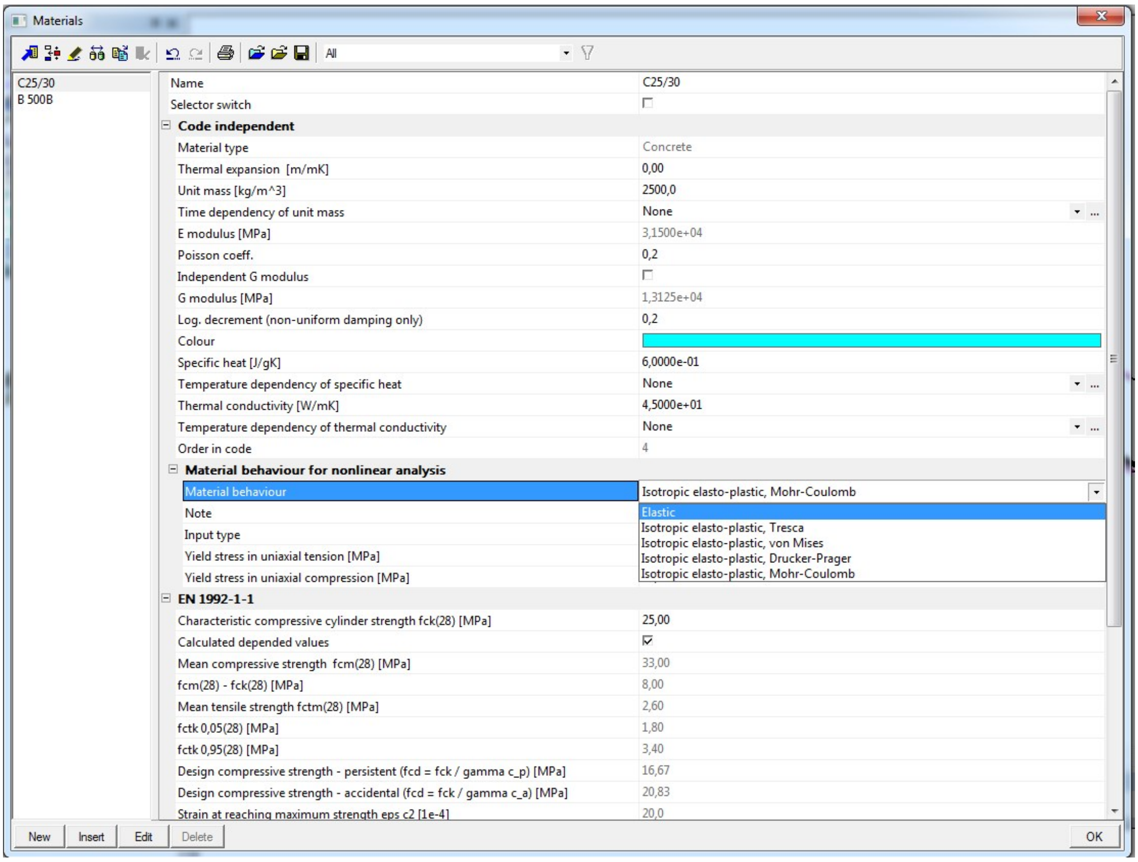
General plasticity is a sub-functionality of non-linear analysis. In the project settings, in the *Functionality* tab, enable *Non-linearity* and *General plasticity*.



Nonlinear properties of materials

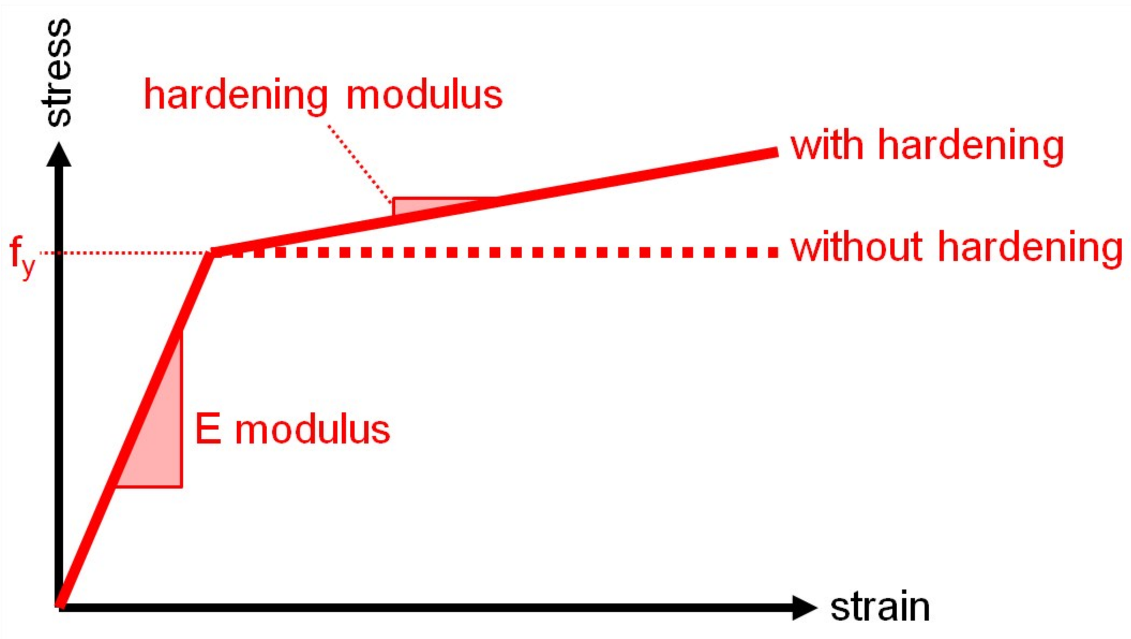
The non-linearity of materials is defined directly in the material library. See the property group *Material behaviour for non-linear analysis*. By default, all materials in the library are set as *elastic*. This means, that the selected material will behave elastically during a non-linear analysis. The plastic properties of materials are generic, code independent in SCIA Engineer and are therefore available for any material, regardless of the selected design code.


Materials
All
S 235
S 355
Name
S 355
Code independent
Material type
Steel
Thermal expansion [m/mK]
0.00
Unit mass [kg/m^3]
7850.0
E modulus [MPa]
2.1000e+05
Poisson coeff.
0.3
Independent G modulus
G modulus [MPa]
8.0769e+04
Log. decrement (non-uniform damping only)
0.15
Colour
Thermal expansion (for fire resistance) [m/mK]
0.00
Specific heat [J/gK]
6.0000e-01
Thermal conductivity [W/mK]
4.5000e+01
Material behaviour for nonlinear analysis
Material behaviour
Elastic
EC3
Ultimate strength [MPa]
490.0
Yield strength [MPa]
355.0
Thickness range
New
Insert
Edit
Delete
Close

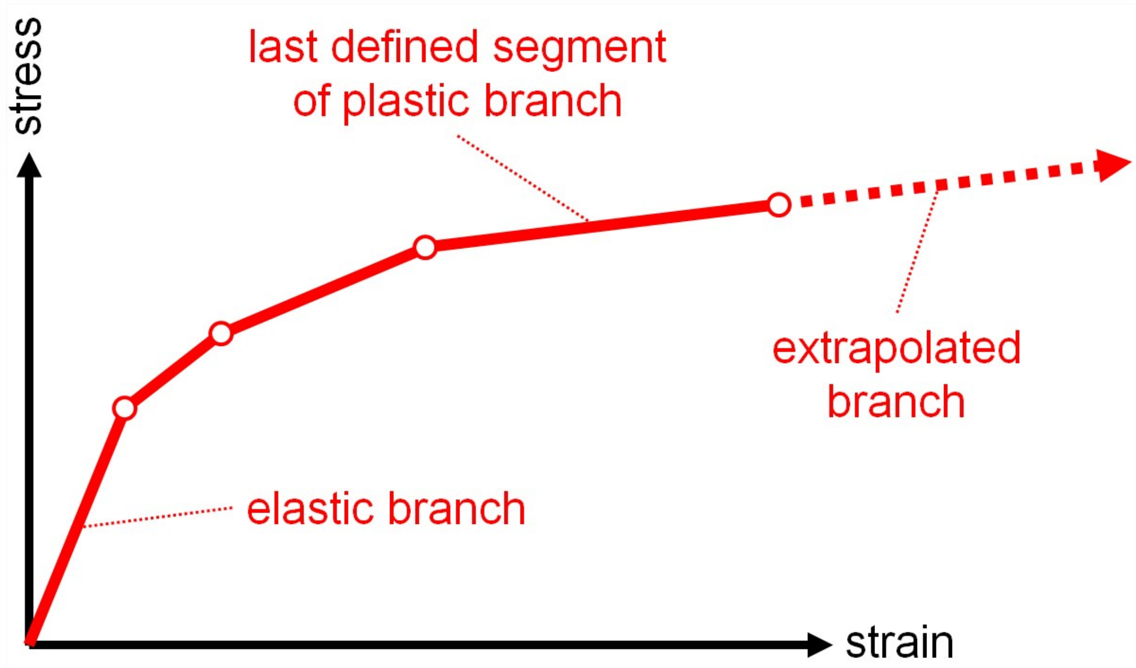

Materials
All
C25/30
B 500B
Name
C25/30
Selector switch
Code independent
Material type
Concrete
Thermal expansion [m/mK]
0,00
Unit mass [kg/m^3]
2500,0
Time dependency of unit mass
None
E modulus [MPa]
3,1500e+04
Poisson coeff.
0,2
Independent G modulus
G modulus [MPa]
1,3125e+04
Log. decrement (non-uniform damping only)
0,2
Colour
Specific heat [J/gK]
6,0000e-01
Temperature dependency of specific heat
None
Thermal conductivity [W/mK]
4,5000e+01
Temperature dependency of thermal conductivity
None
Order in code
4
Material behaviour for nonlinear analysis
Material behaviour
Isotropic elasto-plastic, Mohr-Coulomb
Note
Input type
Yield stress in uniaxial tension [MPa]
Yield stress in uniaxial compression [MPa]
Elastic
Isotropic elasto-plastic, Tresca
Isotropic elasto-plastic, von Mises
Isotropic elasto-plastic, Drucker-Prager
Isotropic elasto-plastic, Mohr-Coulomb
EN 1992-1-1
Characteristic compressive cylinder strength fck(28) [MPa]
25,00
Calculated depended values
Mean compressive strength fcm(28) [MPa]
33,00
fcm(28) - fck(28) [MPa]
8,00
Mean tensile strength fctm(28) [MPa]
2,60
fctk 0,05(28) [MPa]
1,80
fctk 0,95(28) [MPa]
3,40
Design compressive strength - persistent (fcd = fck / gamma c_p) [MPa]
16,67
Design compressive strength - accidental (fcd = fck / gamma c_a) [MPa]
20,83
Strain at reaching maximum strength eps c2 [1e-4]
20,0
New
Insert
Edit
Delete
OK

Plasticity can be enabled by selecting a type of plastic behaviour. Currently, the only available type is *isotropic elasto-plastic von Mises*. It corresponds to a bilinear stress-strain relationship, identical in tension and compression. The plastic branch may have a slope (hardening modulus) or not.

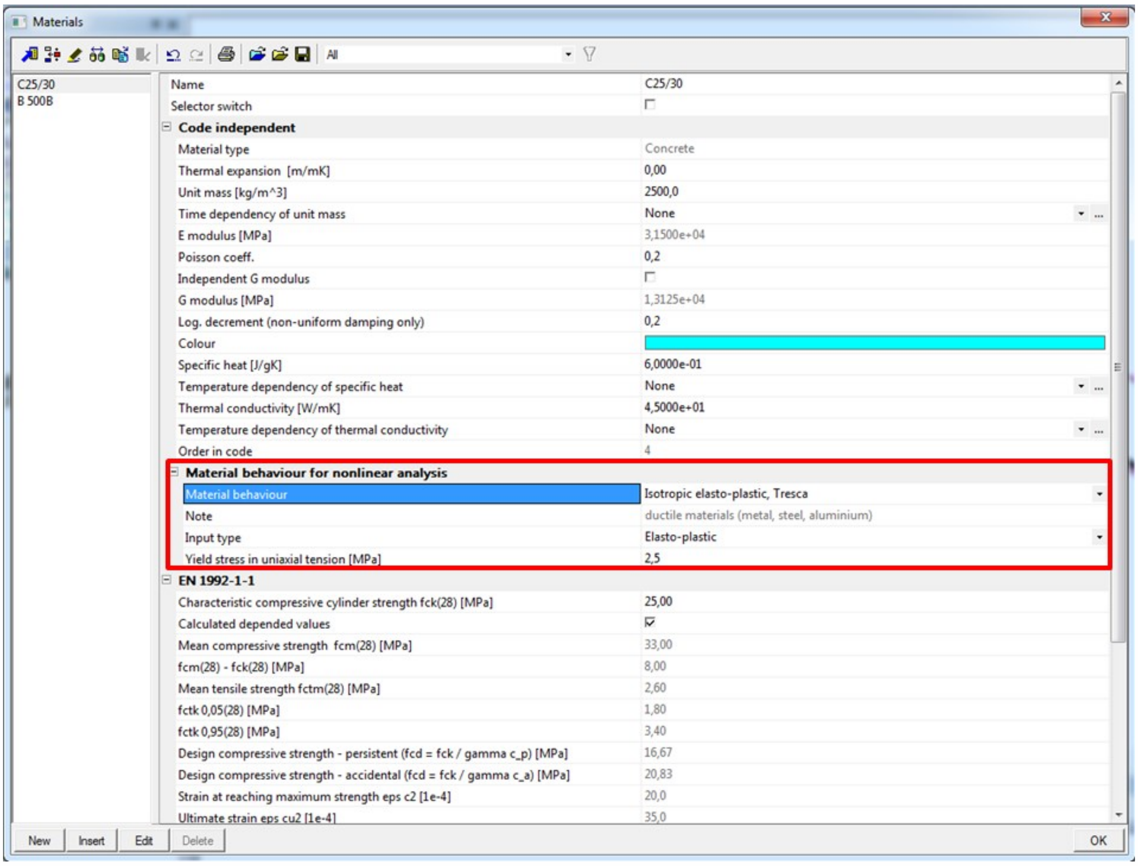
The stress-strain relationship is automatically generated from 3 parameters: Young's modulus (elastic part), yield stress for uniaxial tension and, optionally, hardening modulus (slope of the plastic branch).

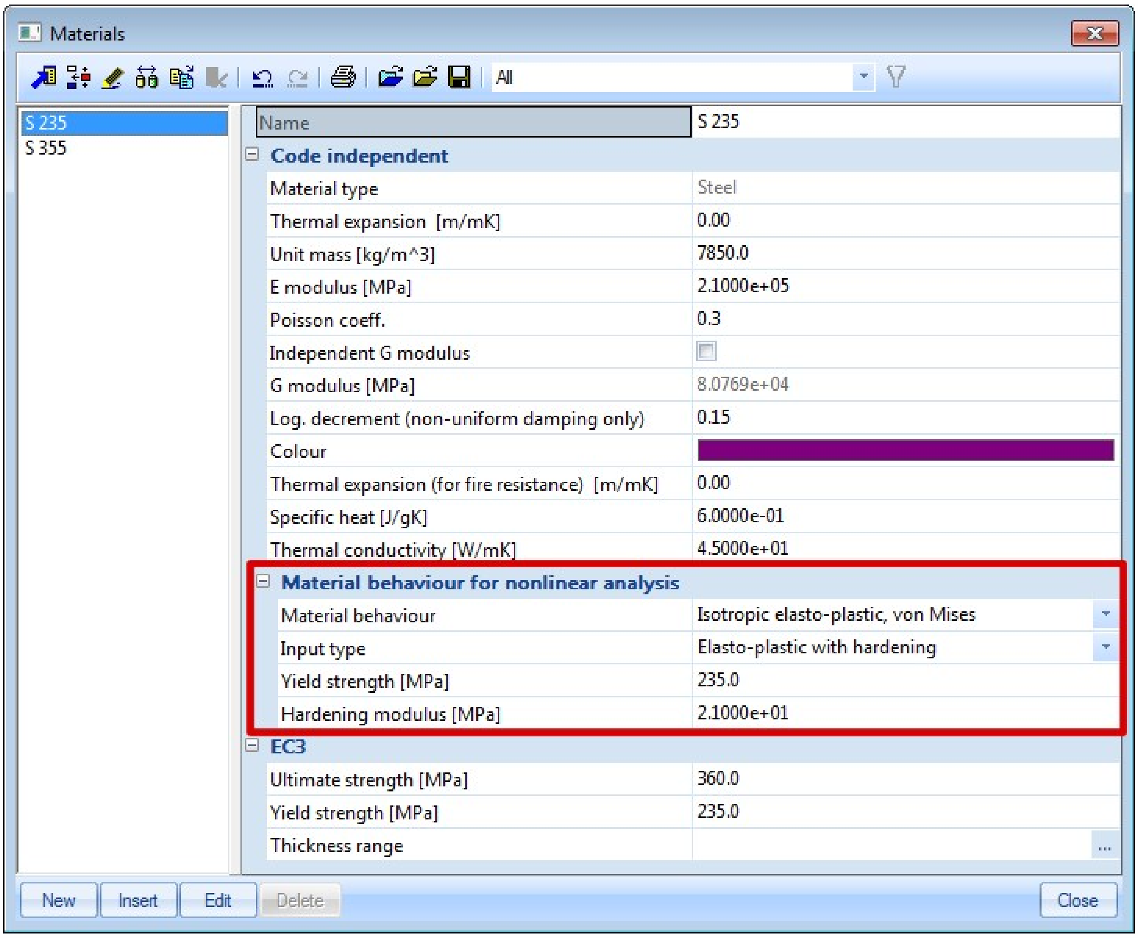


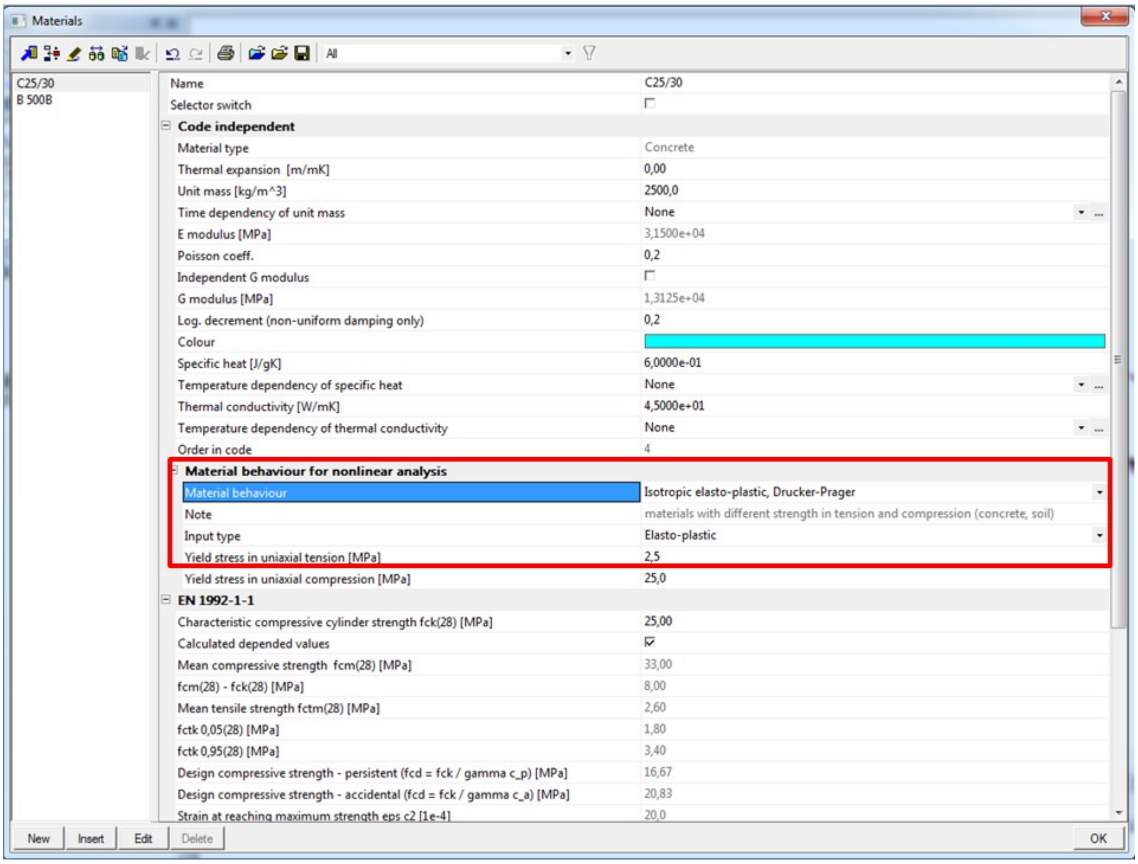
Only the tension part of the diagram is defined, as it is related to a plastification condition in general 3D stress state in principal stress directions. Some plastification models allow for a different yield stress in compression, which is defined separately. There is no limit (ultimate) strain value for the analysis. When the actual strain value in the structure exceeds the defined diagram, the diagram is extrapolated, tangent to the last defined segment of the stress-strain relationship. The reason for that is, that the analysis would then fail and it would be impossible for the user to find where the problem is located in the structure. It is therefore preferable, that the analysis continues and that the user checks the obtain strain values after the analysis.

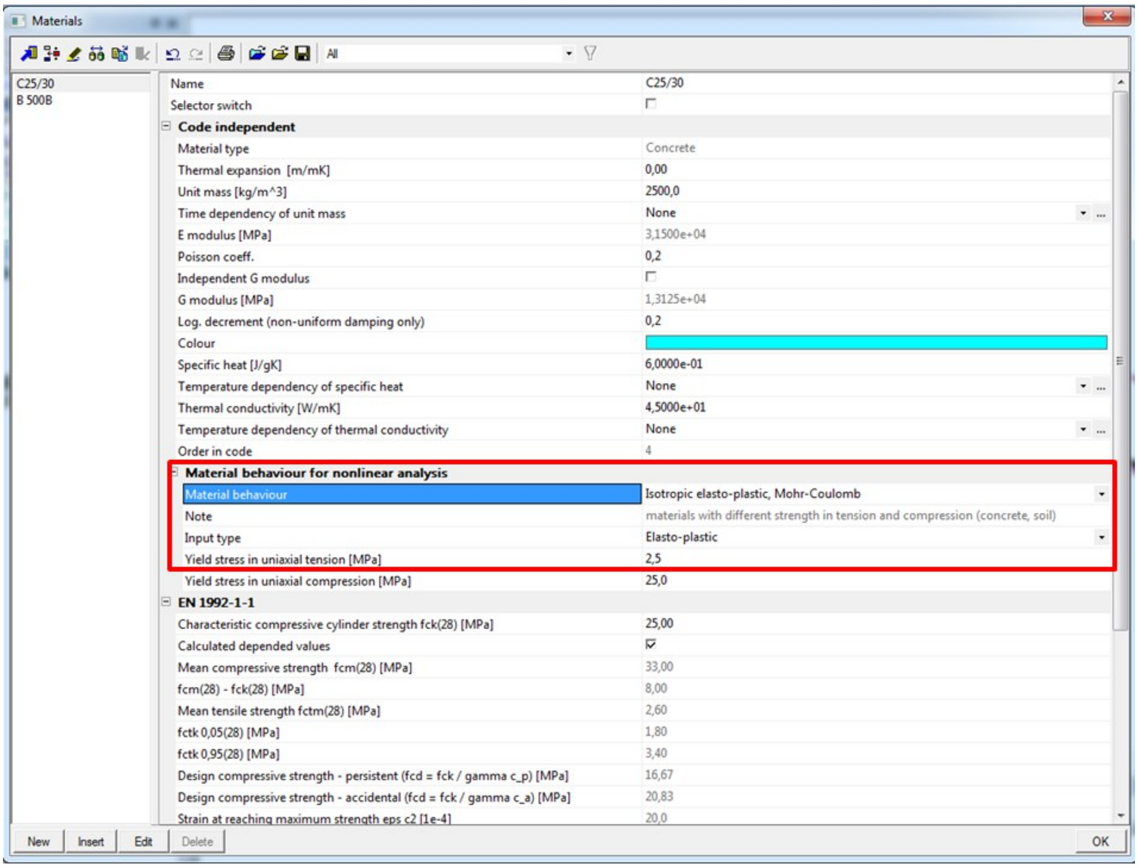


The following properties define the nonlinear behaviour of the material in the material library.




Materials
All
S 235
S 355
Name
S 235
Code independent
Material type
Steel
Thermal expansion [m/mK]
0.00
Unit mass [kg/m^3]
7850.0
E modulus [MPa]
2.1000e+05
Poisson coeff.
0.3
Independent G modulus
G modulus [MPa]
8.0769e+04
Log. decrement (non-uniform damping only)
0.15
Colour
Thermal expansion (for fire resistance) [m/mK]
0.00
Specific heat [J/gK]
6.0000e-01
Thermal conductivity [W/mK]
4.5000e+01
Material behaviour for nonlinear analysis
Material behaviour
Isotropic elasto-plastic, von Mises
Input type
Elasto-plastic with hardening
Yield strength [MPa]
235.0
Hardening modulus [MPa]
2.1000e+01
EC3
Ultimate strength [MPa]
360.0
Yield strength [MPa]
235.0
Thickness range
New
Insert
Edit
Delete
Close


Materials
All
C25/30
B 500B
Name
C25/30
Selector switch
Code independent
Material type
Concrete
Thermal expansion [m/mK]
0,00
Unit mass [kg/m^3]
2500,0
Time dependency of unit mass
None
E modulus [MPa]
3,1500e+04
Poisson coeff.
0,2
Independent G modulus
G modulus [MPa]
1,3125e+04
Log. decrement (non-uniform damping only)
0,2
Colour
Specific heat [J/gK]
6,0000e-01
Temperature dependency of specific heat
None
Thermal conductivity [W/mK]
4,5000e+01
Temperature dependency of thermal conductivity
None
Order in code
4
Material behaviour for nonlinear analysis
Material behaviour
Isotropic elasto-plastic, Drucker-Prager
Note
materials with different strength in tension and compression (concrete, soil)
Input type
Elasto-plastic
Yield stress in uniaxial tension [MPa]
2,5
Yield stress in uniaxial compression [MPa]
25,0
EN 1992-1-1
Characteristic compressive cylinder strength fck(28) [MPa]
25,00
Calculated depended values
Mean compressive strength fcm(28) [MPa]
33,00
fcm(28) - fck(28) [MPa]
8,00
Mean tensile strength fctm(28) [MPa]
2,60
fctk 0,05(28) [MPa]
1,80
fctk 0,95(28) [MPa]
3,40
Design compressive strength - persistent (fcd = fck / gamma c_p) [MPa]
16,67
Design compressive strength - accidental (fcd = fck / gamma c_a) [MPa]
20,83
Strain at reaching maximum strength eps c2 [1e-4]
20,0
New
Insert
Edit
Delete
OK

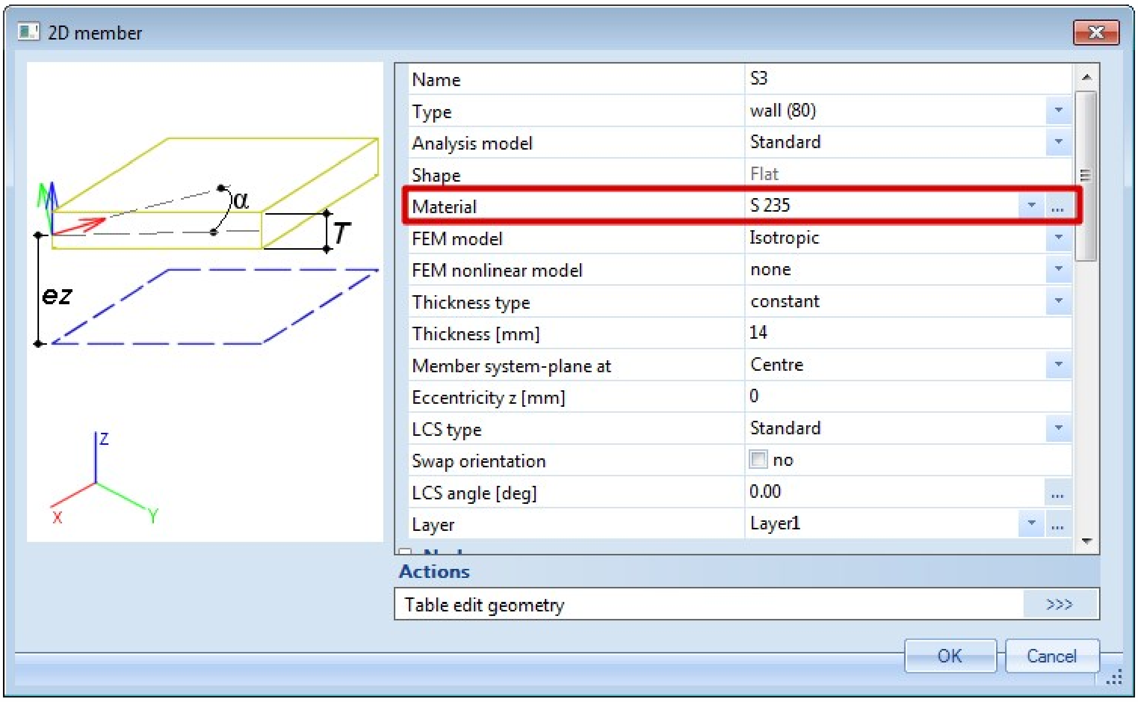


E modulus	Young's modulus of the material; it defines the slope of the elastic part of the stress-strain relationship
Material behaviour	selects the type of behaviour of the material in case of nonlinear analysis. The available types are currently: • *elastic* • *isotropic elasto-plastic, Tresca* • *isotropic elasto-plastic, von Mises* • *isotropic elasto-plastic, Drucker-Prager* • *isotropic elasto-plastic, Mohr-Coulomb*
Input type	type of definition of the plastic branch of the stress-strain relationship. The available types are: • *elasto-plastic*; in the plastic domain, the stress remains constant when the strain increases • *elasto-plastic with hardening*; in the plastic domain, the stress increases with the strain
Yield strength stress in uniaxial tension [MPa]	elastic limit for plastification due to tension shear (see von Mises theory)
Yield stress in uniaxial compression [MPa]	elastic limit for plastification due to compression
Hardening modulus	slope of the plastic branch of the stress-strain relationship

Important: default values have been defined for the yield strength and the hardening modulus, as much as possible according to the corresponding design code of each material proposed in the system libraries. However, **those values should always be reviewed** before use.

Assigning plastic behaviour to a 2D member

To enable plastic behaviour on a 2D member, just assign to it a material whose plastic behaviour has been enabled. There is no other specific setting.



Although various types of non-linearity may be combined in the same project, it is not possible to cumulate several types of non-linearity on the same 2D member. The property *FEM non-linear model*, when combined with a plastic material, will behave as follows:

- Plastic material and *2D press-only* behaviour: the press-only behaviour will be ignored and the 2D member will behave as plastic
- Plastic material and *membrane* behaviour: the plastic behaviour will be ignored and the 2D member will behave as an elastic membrane

Additionally, a warning message will be displayed when starting the analysis, giving the same information about functionality conflicts.

Using a non-linear material in the properties of the cross-section of a 1D member will not affect the behaviour of that member. General plasticity is currently not supported for 1D members and the behaviour of the material will **remain elastic** for such member.

Plastic hinges

Introduction to plastic hinges

If a normal linear calculation is performed and limit stress is achieved in any part of the structure, the dimension of critical elements must be increased. If however, plastic hinges are taken into account, the achievement of limit stress causes that plastic hinges are inserted into appropriate joints and the calculation can continue with another iteration step. The stress is redistributed to other parts of the structure and better utilisation of overall load bearing capacity of the structure is obtained.

On the other hand, there is a risk in this approach. If a hinge is added to the structure, its statical indeterminateness is reduced. If other hinges are added, it may happen that the structure becomes a mechanism. This would lead to a collapse of the structure and the calculation is stopped.

Plastic hinges can be thus used to calculate the plastic reliability margin of the structure. The applied load can be increased little by little (e.g. by increasing the load case coefficients in load case combination) until the structure collapses. This approach can be used to determine the maximum load multiple that the structure can sustain.

Plastic hinges are considered only at ends of individual 1D members. No selection of 1D members is made for the calculation with plastic hinges. If this type of calculation is selected, all 1D members in the structure are tested .

The calculation is similar to the calculation of beams with gaps. All 1D members in the structure are tested and if the limit stress is reached, the plastic hinge is inserted. If however the stress lowers in the next iteration step, the plastic hinge may be removed.

The Solver setup offers an item where a particular national code can be selected for correction of the limit moments. If option No Code is selected , the modification of the limit moments is performed as for option EC (Eurocode).

Approaches described in EC3, DIN 18800 and NEN codes are implemented in SCIA Engineer.

Plastic hinges to EC3

Axis	Axial load	V =<0.5 V_	V>0.5 V_
yy	NSd=<0.25 NRd	Mpl,y,Rd	Mpl,y,Rd (1-r)
yy	Nsd >0.25 NRd	Mpl,y,Rd1.11 (1-n)	Mpl,y,Rd1.11 (1-n-r)
zz	NSd=<0.25 NRd	Mpl,z,Rd	Mpl,z,Rd(1-r)
zz	NSd>0.25 NRd	Mpl,z,Rd1.56 . . (1-n)(n+0.6)	Mpl,z,Rd1.56 . . (1-n-r)(0.6+n/(1-r))

where:

r	(2 VSd / VRd –1)2
a	NSd / NRd
Nsd	axial force
VSd	shear force
Mpl,y,Rd	full plastic moment around yy axis
Mpl,z,Rd	full plastic moment around zz axis

VRd	plastic shear force
NRd	plastic axial force

Plastic hinges to DIN 18800

Axis	Axial load	V =<0.33 V_	V>0.33 V_
yy	N =<0.10 Npl,d	Mpl,y,d	Mpl,y,d (1.136-0.42r)
yy	N >0.10 Npl,d	Mpl,y,d 1.111 (1-n)	Mpl,y,d (1.25-1.113 n - - 0.4125 r)

Axis	Axial load	V =<0.25 V_	V>0.25 V_
zz	N =<0.30 Npl,d	Mpl,z,d	Mpl,z,d (1-0.82r2) / 0.95
zz	N >0.30 Npl,d	Mpl,z,d (1-n2) / 0.91	Mpl,z,d (1-0.95 n2 - - 0.75 r2)/0.87

where:

r	V / Vpl,d
a	N / Npl,d
A	axial force
V	shear force
Mpl,y,d	full plastic moment around yy axis
Mpl,z,d	full plastic moment around zz axis
Vpl,d	plastic shear force
Npl,d	plastic axial force

Plastic hinges to NEN

For IPE sections

Axis	Condition	
yy	n / 0.18 + r <= 1	Mpl,y,d
yy	a <=0.18	Mpl,y,d
yy	a >0.18	Mpl,y,d 1.22 (1-n)
yy	r<=0.3	Mpl,y,d
yy	r>0.3	Mpl,y,d (1.1-0.3 n)
zz	n <=0.36	Mpl,z,d
zz	n >0.36	Mpl,z,d (1-((n-0.36) / 0.64)2)
zz	r<=0.3	Mpl,z,d
zz	r>0.3	Mpl,z,d (1.1-0.3 n)

For other sections

Axis	Condition	
yy	n / 0.10 + r <= 1	Mpl,y,d
yy	n <=0.10	Mpl,y,d
yy	n >0.10	Mpl,y,d 1.11 (1-n)
yy	r<=0.3	Mpl,y,d
yy	r>0.3	Mpl,y,d (1.1-0.3 n)
zz	n <=0.20	Mpl,z,d
zz	n >0.20	Mpl,z,d (1-((n-0.20) / 0.80)2)
zz	r<=0.3	Mpl,z,d
zz	r>0.3	Mpl,z,d (1.1-0.3 n)

where:

r	V / Vpl,d
a	N / Npl,d
A	axial force
V	shear force
Mpl,y,d	full plastic moment around yy axis
Mpl,z,d	full plastic moment around zz axis
Vpl,d	plastic shear force
Npl,d	plastic axial force

Calculating with plastic hinges

To perform the calculation with plastic hinges taken into account, it is necessary to:

- select Nonlinearity in Project Setup dialogue,
- select required Plastic hinge code in Solver setup dialogue,
- define non-linear load case combination / combinations,
- have linear calculation of the structure successfully completed,
- start nonlinear calculation and obtain successful solution.

AutoDesign - Global optimization

Introduction

In many cases, the design of a building involves a typical calculation with only some variation in certain predefined parameters, such as the buildings dimensions, loading, boundary conditions, etc. Often, the principle of the analysis in all projects is the same. If all these principles can easily be defined, parameterized and stored, the design of the building can be made much faster than in the traditional way of engineering. In SCIA Engineer it is easily possible to parameterize structures and save them in a library for later reuse in other projects. Parameters, such as height of a section, length of a beam/column, span, cross-section data (including built-up sections), etc. can be easily defined without the need for programming or scripting. The designer can choose the required structural element from the library and easily edit its boundary conditions, loads (wind, snow, etc.) according to a chosen code and the code specific combinations are applied automatically to enable the design of the structure. Once the structure is defined, SCIA Engineer Autodesign capabilities will automatically run the analysis and find the optimal definition of the structure according to the specific design rules that are chosen by the user.

Steel and concrete members can be designed individually or within a common set of elements to satisfy the criteria of the appropriate code. The Autodesign capabilities in SCIA Engineer offer a lot of flexibility. They offer the users different levels of control. They considerably reduce the time needed to select appropriate sections. For example, the user can select the maximum check value and type of cross-section, including I-sections, angles and welded sections. Then SCIA Engineer will determine the optimal profile that satisfies the code check. Automatic profile optimization can be applied to all standard and parametric sections. For parametric sections, the user chooses which parameter - whether height, flange thickness or other – should be adapted. The program displays the check values graphically in the 3D view of the structure with colours giving a clear overview of (over-)dimensioned and (un)satisfactory parts of the construction.

Principles of Autodesign

Once a structure has been designed and calculated, it is the time to perform checking and usually a kind of optimisation of the original design. SCIA Engineer contains a powerful tool for this task. The optimisation of applied profiles may be done automatically or semi-automatically. The process of Autodesign results in what may be called an economical and good solution.

Autodesign in general represents a complex task. A full, complete and really "optimal" optimisation would usually lead to a long and often recursive process. Therefore, SCIA Engineer implements a kind of compromise.

One Autodesign step takes account of a single cross-section only

It is possible to optimise one cross-section at a time. The user selects the cross-section from a list of all cross-sections used in the structure.

One Autodesign step considers only "selected" members

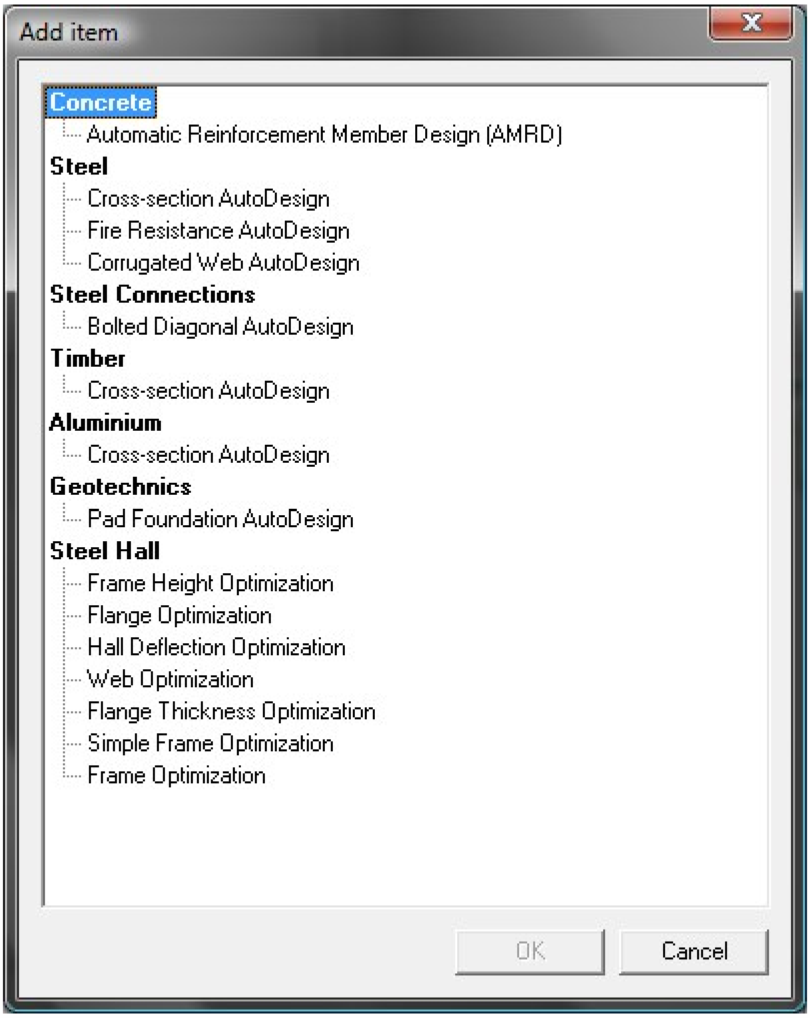
It is possible to limit the Autodesign process to only a set of selected members. The user may make a selection to specify which beams of the given cross-section should be considered for the Autodesign calculations.

One Autodesign step affects the whole structure

Once the optimised cross-section is found, it is applied to ALL members in the structure that are of the specified cross-section. It is of no importance whether the Autodesign calculation was limited to a selected number of beams or not. The final effect of the Autodesign is that the original cross-section is simply replaced with the new, i.e. optimised, cross-section.

Autodesign types

Within SCIA Engineer, there are different possibilities that guide you through the optimization process. The basic option is the use of parameters. Again, almost each entity in SCIA Engineer can be covered by a parameter. Once a project is defined and analyzed, the user can always save the project as a template for later reuse. More general and very useful tool is the Autodesign tool. This allows the user to optimize different parts of the structure. With the optimization, one has the option to define relations between dimensions and specify iteration steps. Last but not least, the defined optimization groups can be combined in the Overall Autodesign in order to optimize multiple types of members or entire structure.



SCIA Engineer enables you to perform an optimisation of the whole structure or of its selected part. The optimisation can be run for steel, concrete, aluminium and timber structures or for steel or timber parts of multi-material projects. Most of the items there concern the default Autodesign (cross-section steel, timber, aluminium, fire resistance, corrugated web check) which is the standard Autodesign of cross-sections that can be found in the appropriate steel, aluminium and other services.

There are several advantages in the overall Autodesign function over the individual Autodesign in services:

- The ability to Autodesign more than one member at a time.
- The ability to run more than one type of Autodesign at a time (steel, timber, concrete, aluminum…etc).
- The ability to use an iterative optimization

There are several different Autodesign procedures mentioned in the following table:

Material	Autodesign item
Concrete	Automatic member reinforcement design (AMRD)
Steel	Cross-section AutoDesign
Steel	Fire resistance AutoDesign

Material	Autodesign item
Steel	Corrugated web AutoDesign
Steel	Lapped purlin/girt Autodesign (only IBC code)
Steel connection	Bolted diagonal Autodesign
Timber	Cross-section Autodesign
Aluminium	Cross-section Autodesign
Geotechnics	Pad foundation Autodesign
Steel hall	In-block Autodesign item
Steel hall	Frame - Autodesign manager
Steel hall	Frame – CSS height Autodesign
Steel hall	Frame - deflection Autodesign
Steel hall	Frame - flange Autodesign
Steel hall	Frame – web Autodesign
Steel hall	Frame - flange thickness Autodesign

It is also possible to perform several of the above mentioned optimisation types and then compare the results. And it is up to you to select the cross-section types and bolted diagonal connections that are relevant to your work. It is also your responsibility to think in advance and define and assign to 1D members as many cross-section types as necessary for a proper design and optimisation of the project.

Note: In order to perform the Autodesign, calculation must be already performed.

Note: Autodesign for material concrete use Automatic member reinforcement design (AMRD) from Concrete Advanced service.

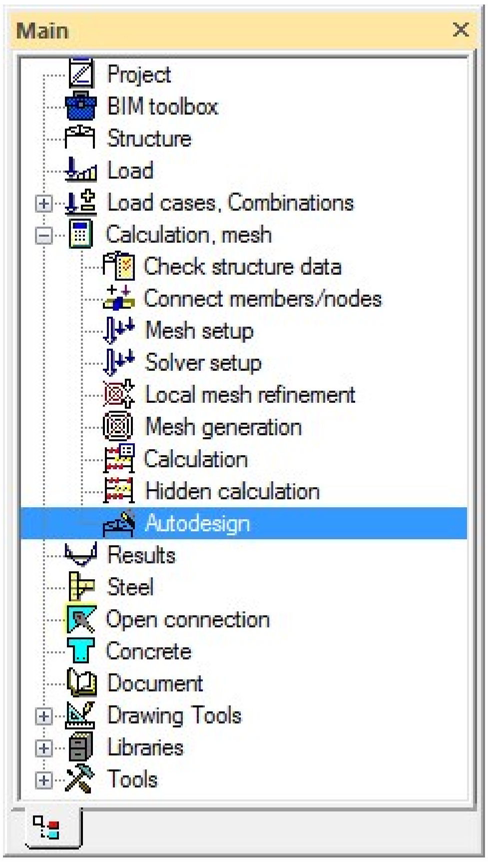
Autodesign manager

As stated in the introduction you may perform several different optimisations. You may run the Autodesign and compare the results for different parts of the structure, for different optimisation types (e.g. standard and fire resistance code check). Therefore, all the defined optimisations are stored in the Autodesign manager. Thus you do not have to define all the Autodesign criteria and parameters again and again. The Autodesign manager is a standard SCIA Engineer database manager with usual features and functions.

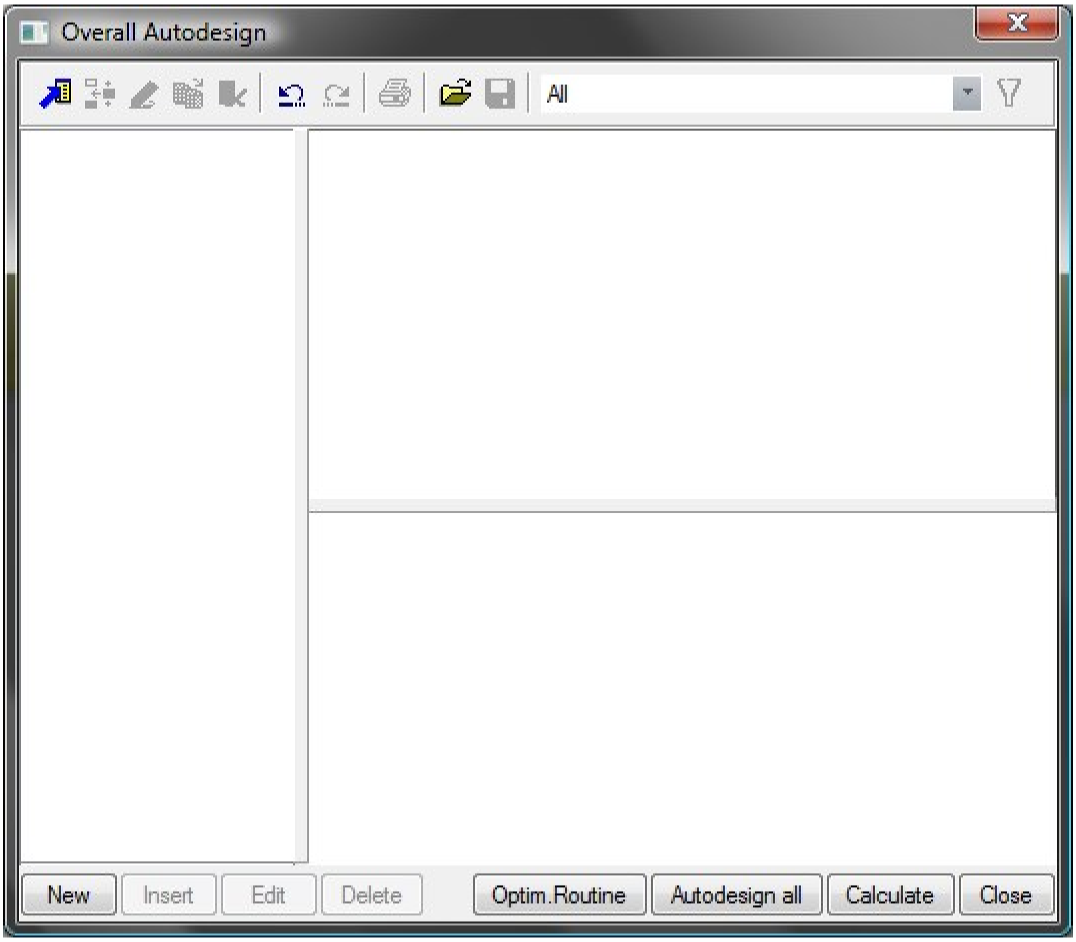
Procedure to open the Autodesign manager

1. Open service Calculation, Mesh.

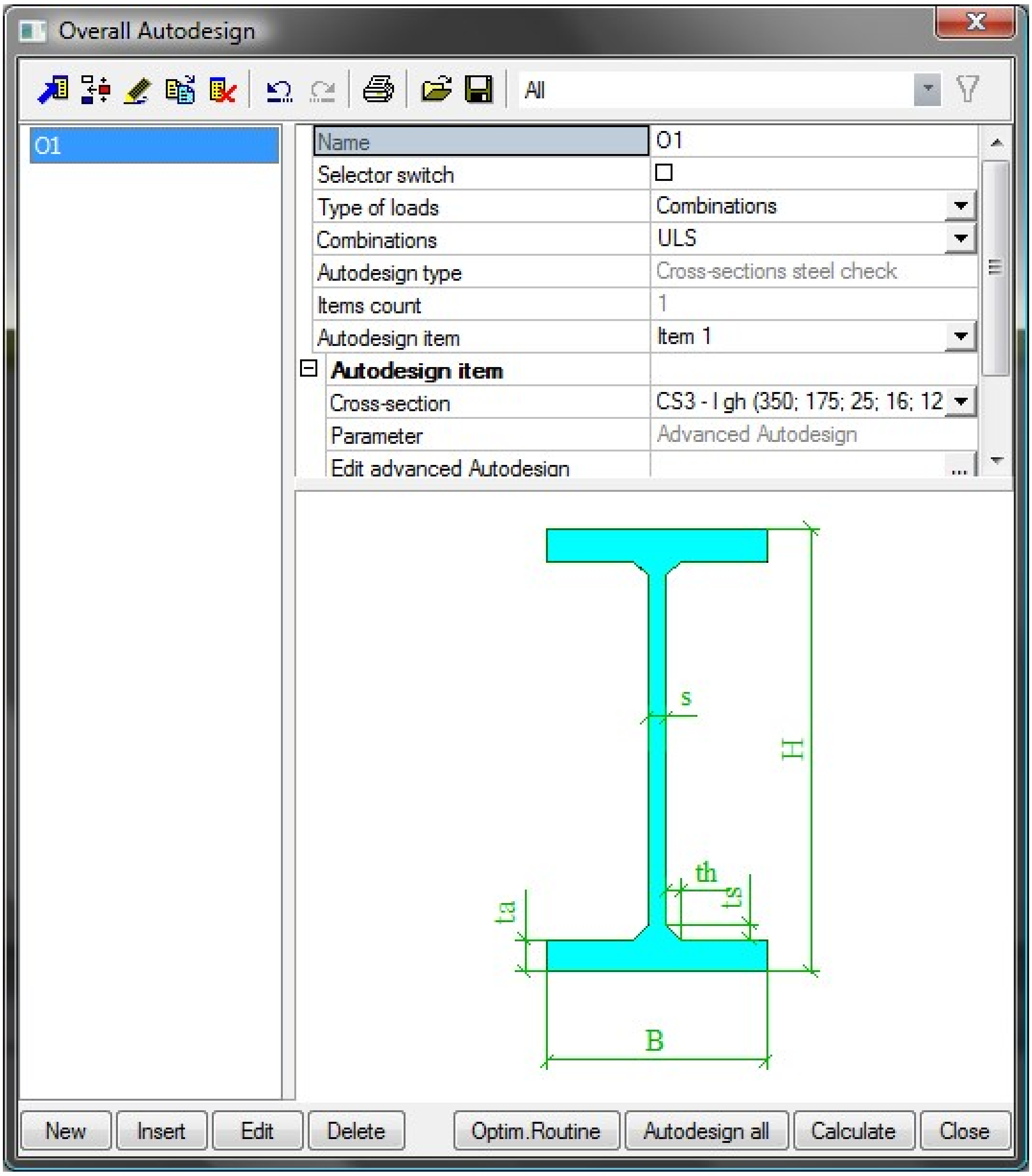
2. Start (double-click) function Autodesign.



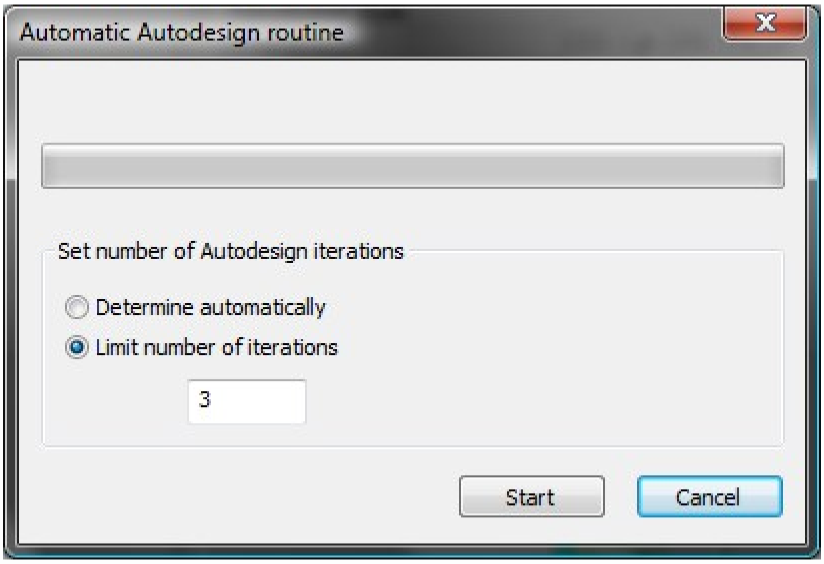
3. Autodesign manager is opened. Initially it is an empty library with standard library functions (read from file, save to file and others).



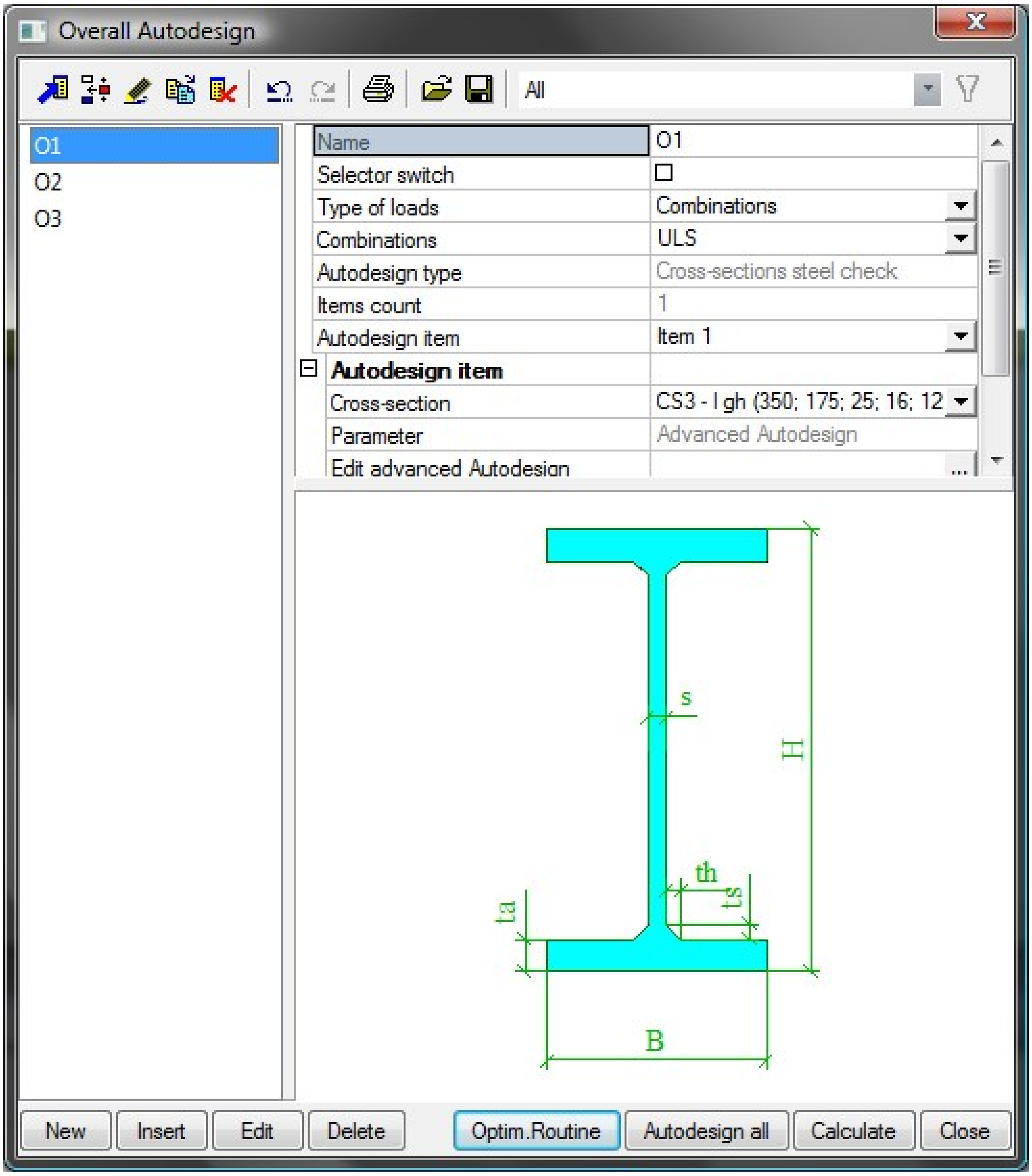
4. After defining the procedure (see next chapter) the following dialogue is displayed.



5. The user is able to optimize the selected Autodesign function using iterative Autodesign clicking on Optim.Routine. There are two possibilities how to set the number of Autodesign iterations – Determine automatically or input Limit number of iterations. We can set a number of iterations for the optimization, or we can let SCIA Engineer iterate until an optimum solution is reached.



6. There is also possibility to run all Autodesign functions in one step using Autodesign all.

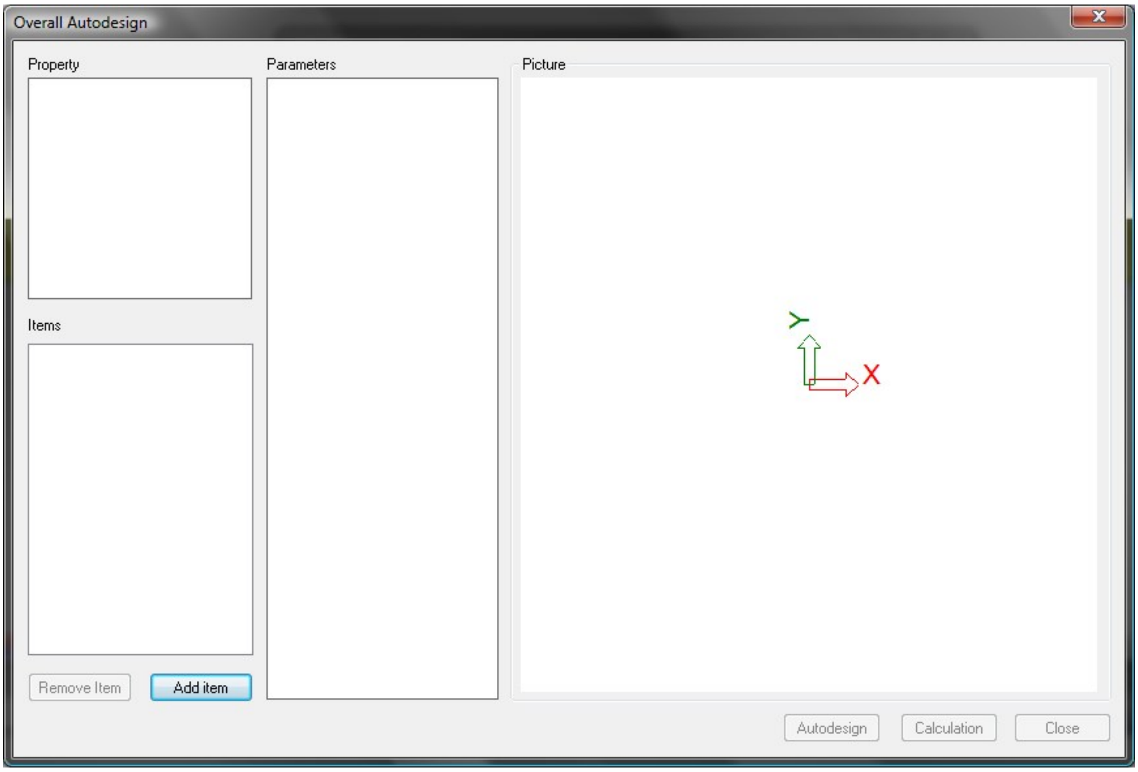


Defining a new optimisation

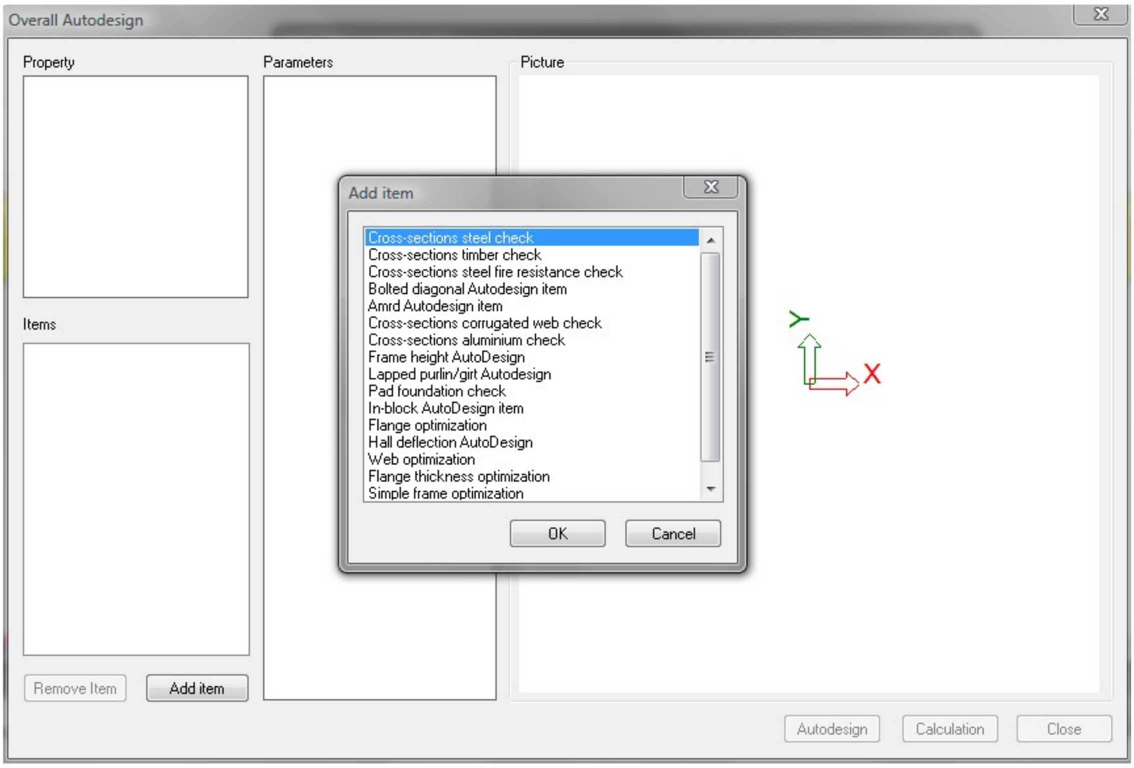
Procedure to define and run a new optimisation

1. Start the Autodesign manager.

2. Click button [New] to open the Overall Autodesign dialogue.

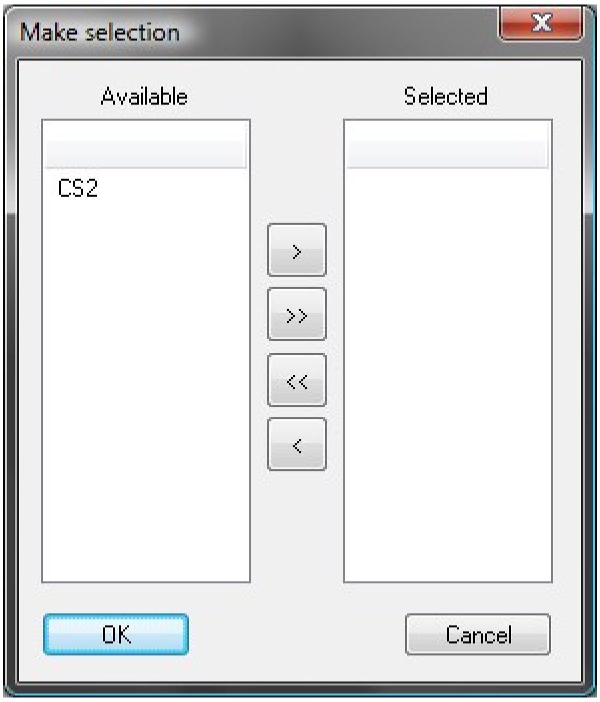


3. Using button Add item it is possible to add a new item of the optimized structure of structural part.

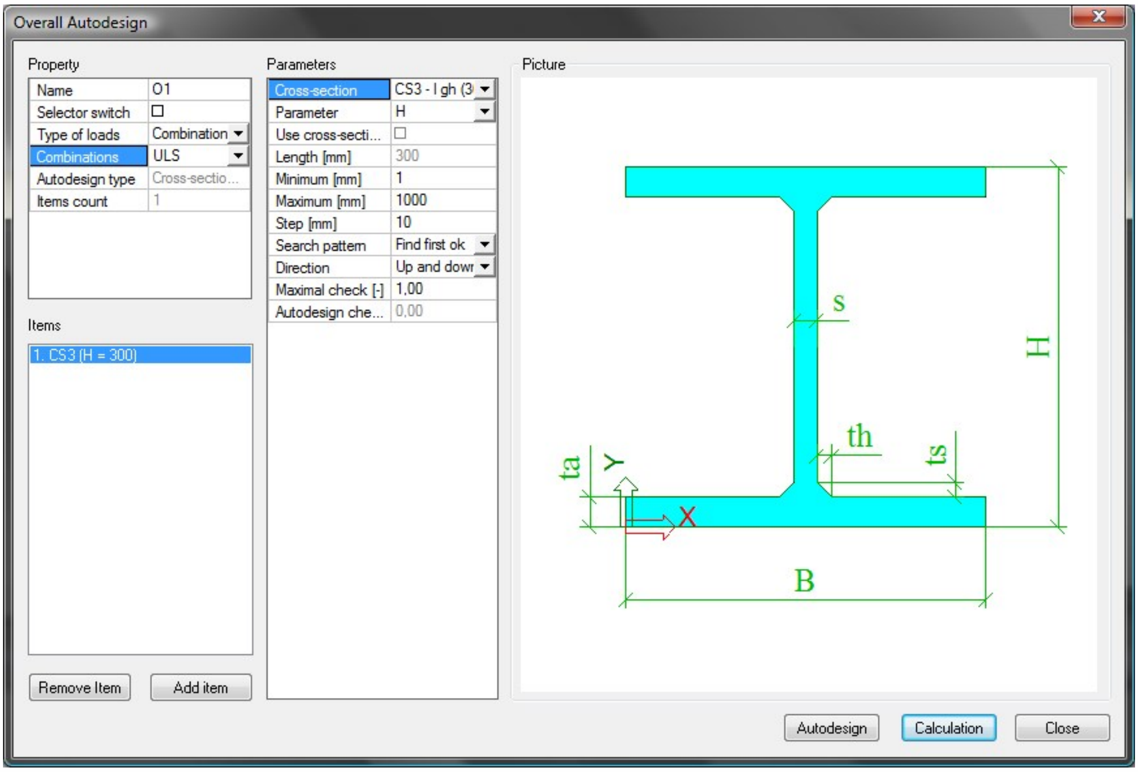


Note – the basic functionality of Autodesign is explained on the first item Cross-sections steel check.

4. The user is asked to select one from the used cross-sections that will be optimized.



5. Define the Autodesign property, parameters and criteria.



6. Click button [Autodesign] to run the calculation and see its result.

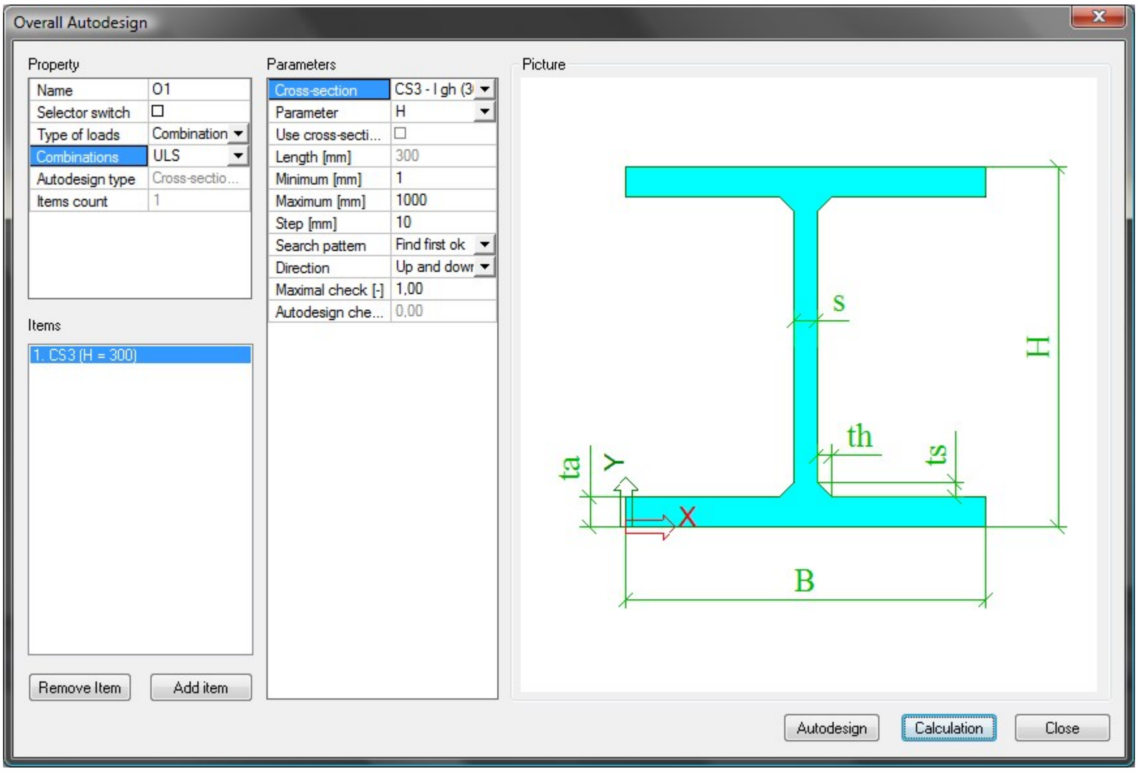
7. If required, click button [Calculation] to re-calculate the model in order to reflect the results of the optimisation.

8. Depending on what you exactly need and want, you may repeat steps 5 to 7 as many times as required.

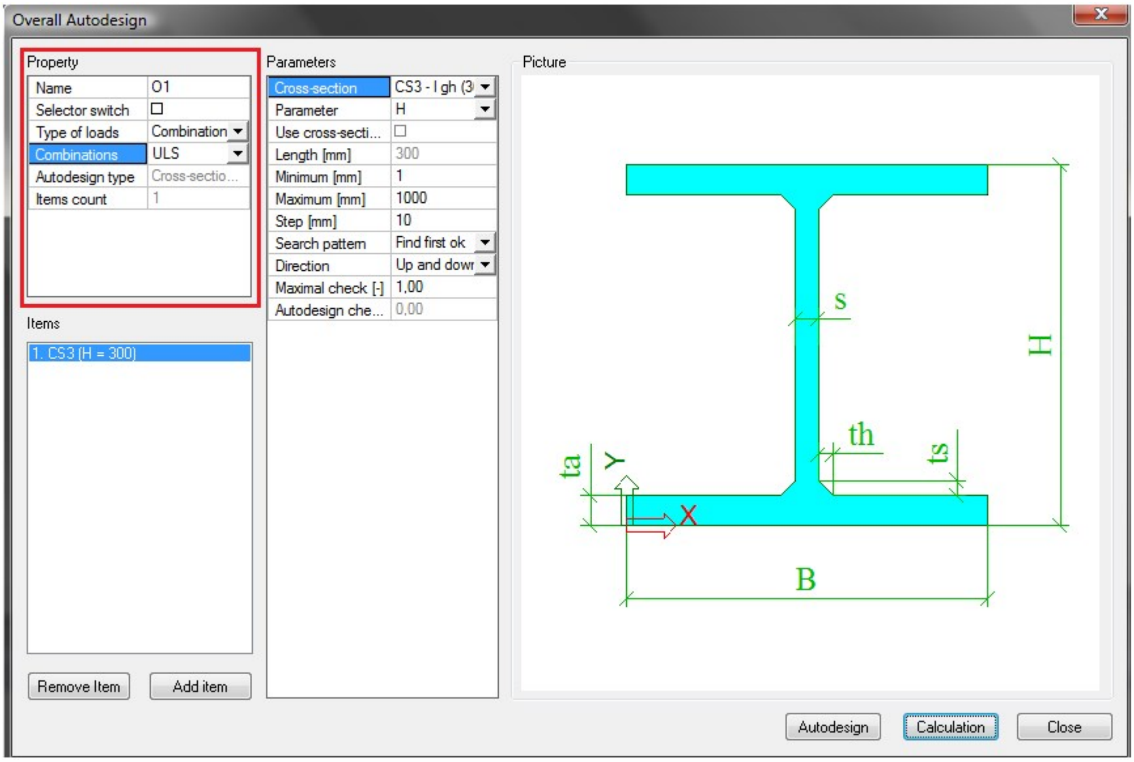
Note: Please note, that pure repetition of Autodesign and Calculation in turns may lead to a "never-ending" cycle. The Autodesign may find cross-section "A" as optimal. When you perform the calculation, the internal forces are redistributed to reflect the Autodesign results. When you run Autodesign now, it may find cross-section "B" as optimal. And another re-calculation once more redistributes the internal forces. And it may happen that the subsequent Autodesign finds the cross-section "A" as optimal once again. And so on, and so on, and so on.

Autodesign parameters and criteria

This chapter describes all buttons, settings and functionality in details for a typical thin-walled geometrical section.



Property



Name
Defines the name of the optimisation (criteria).

Selector switch
Enables you to parameterize Autodesign item using a library type of parameter

Type of loads
Autodesign may be performed for load cases, load case combinations, result classes, etc.

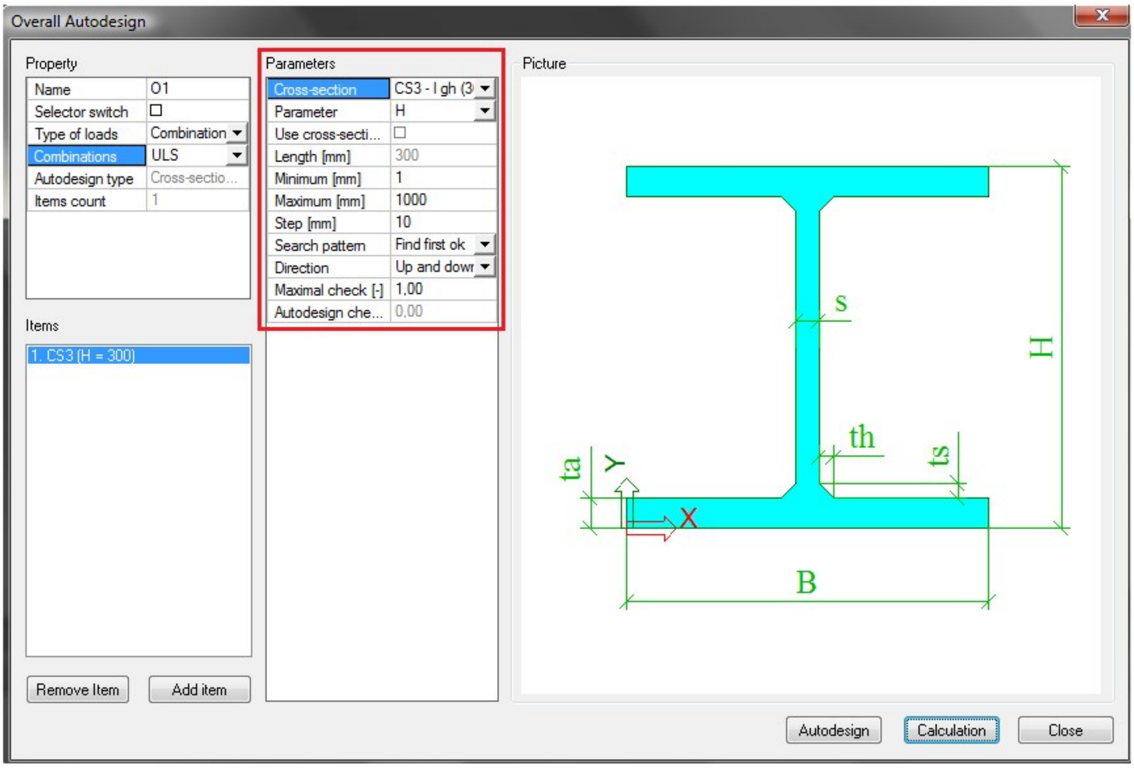
Load
Specifies the particular load case, combination, etc. for which the selected cross-section type will be optimised.

Autodesign type (informative)
Tells the type of the optimisation. Cross-section steel check in this case

Item count (informative)
Shows the number of defined Autodesign items.

Parameters

The displayed parameters depend on type of used cross-section.



Autodesign parameters for rolled and cold-formed cross-sections

The user may control the process of Autodesign by means of a set of parameters.

Check parameter

Maximal check
This parameter tells the program what is the maximal allowable value for a satisfactory check.

Maximum unity check
This item shows the found maximal check result for the optimised cross-section.

Shape parameters for Autodesign

Sort by height
The sequence of cross-sections is based on the height.

Sort by A
(sectional area) The sequence of cross-sections is based on the sectional area.

Sort by Iy
(moment of inertia) The sequence of cross-sections is based on the moment of inertia.

Buttons for manual Autodesign

Set value
This button enables the user to set manually the required value of selected dimension (see above).

Next down
This button finds one-step smaller cross-section according to the defined shape parameters (see above).

Next up
This button finds one-step larger cross-section according to the defined shape parameters (see above).

Autodesign parameters for welded and solid cross-sections

The user may control the process of Autodesign by means of a set of parameters.

Cross-section

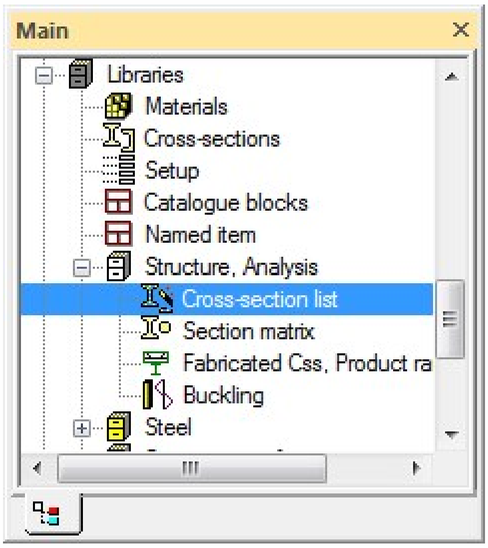
Defines the cross-section type to be optimised.

Parameter

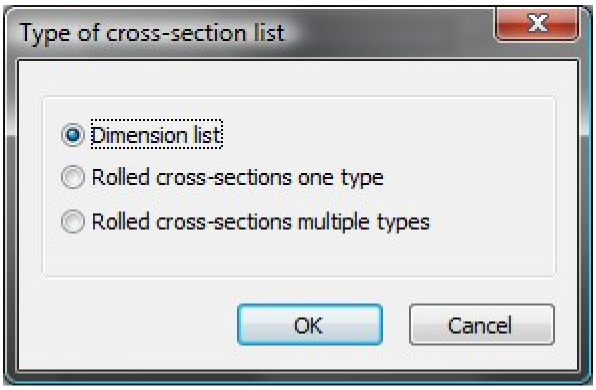
Selects the dimension (e.g. section depth, width, etc.) that will be optimised. All dimensions of optimized item are offered for selection. The optimized item is displayed in the part Items according to the selected parameters (see above CS3 (H=300)). There is also possibility to select Advanced Autodesign (for more information see chapter Edit Advanced Autodesign).

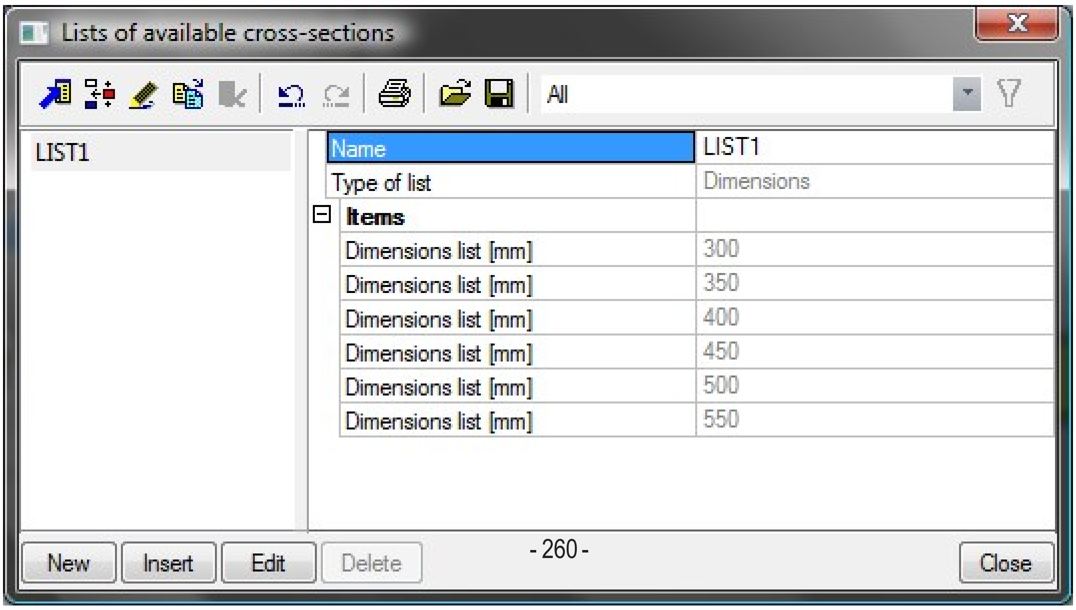
Use cross-section list

Enables you to use predefined values of one dimension according to the list defined in the Cross-section list library. This library is stored in Libraries > Structure, analysis > Cross-section list.



There is a possible to define three types of cross-section list (see the following figure). Cross-section list type Dimension list can be used for Autodesign only.





Length (informative)
Shows the current size of the selected dimension.

Minimum
Defines the minimal applicable size for the optimised parameter.

Maximum
Defines the maximal applicable size for the optimised parameter.

Step
Defines the step for the Autodesign.

Search pattern
Combobox (Find first OK / Find from all) - option enabling to find first solution which fits the requirements or the best from all the solutions.

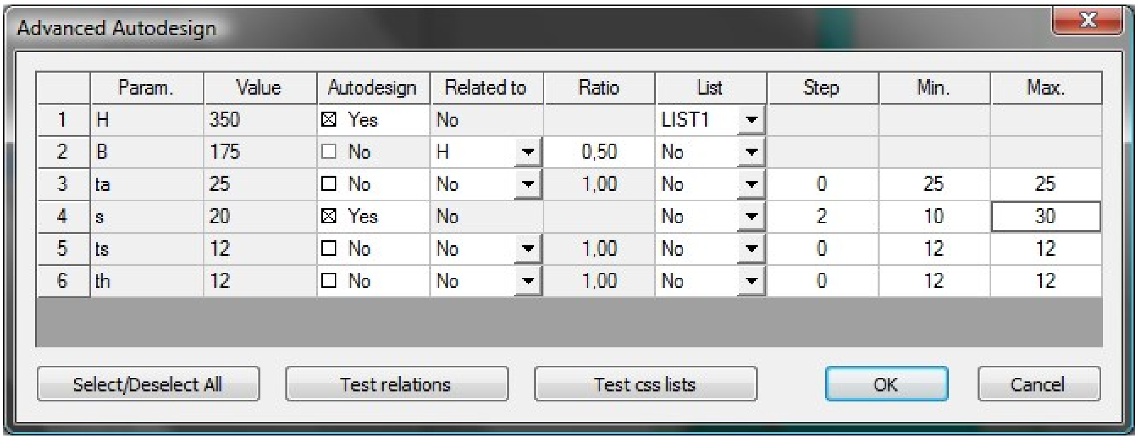
Direction
Combobox (Up and down / Up) – option that specifies the direction of searching for the optimized solution.

Maximal check
Defines the maximal acceptable value of the unity check of the optimised cross-section.

Autodesign check (informative)
Shows the unity check for the optimised connection.

Edit Advanced Autodesign

This option is seen only if Advanced Autodesign is selected as the parameter. Advanced Autodesign enables to optimize several or all parameters of a cross-section in one step. It is possible to use dependencies between parameters and use cross-section list as well.



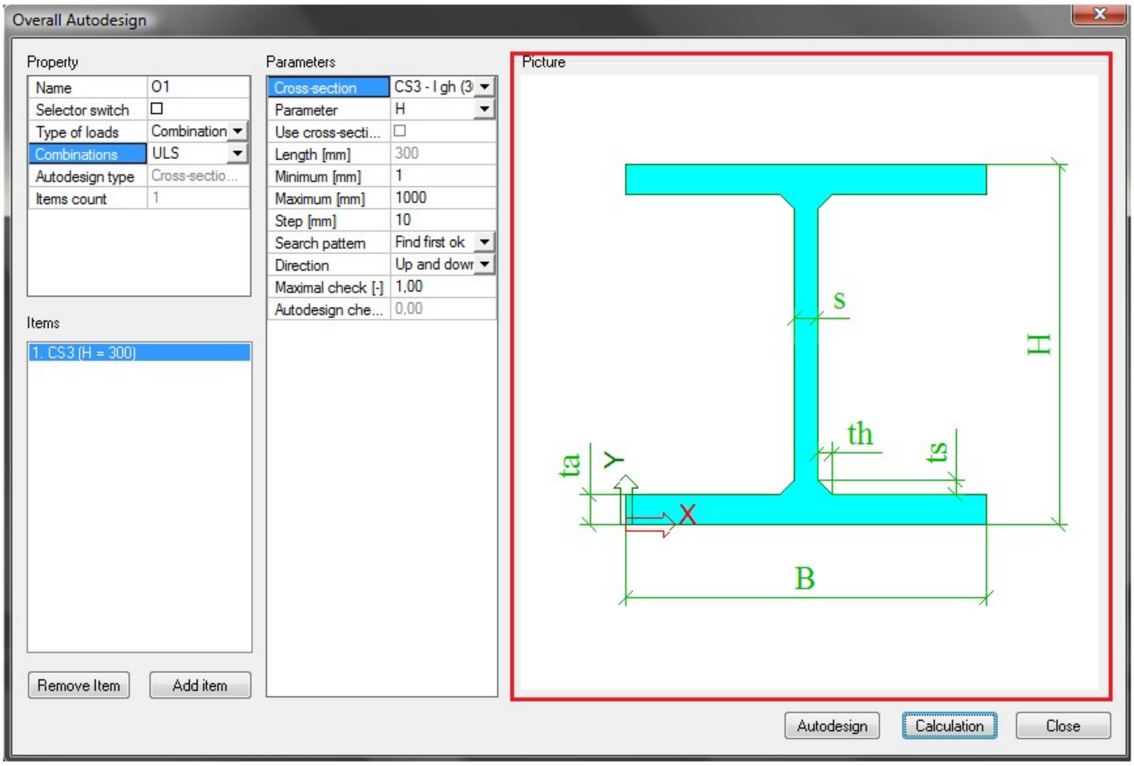
All items are described in the following table.

Item	Description
Parameter	Parameter which needs to be optimized
Value	Value of the optimized parameter
Autodesign	Checkbox if the parameter should be optimized or not. Inactive when Related to is assigned to some parameters
Related to	Relation between parameters. Selected (dependent) parameter can be optimized depending on the optimization of other parameter. Inactive when Autodesign is YES.
Ratio	Ratio gives the relation between the optimized parameter and dependent parameter, see above (value B calculated as 0.5 * optimized value H). Inactive when Related to is No.
List	Link to Cross-section list library. The selected parameter can be optimized according to the requirements stored in the Cross-section list library (dimension list)

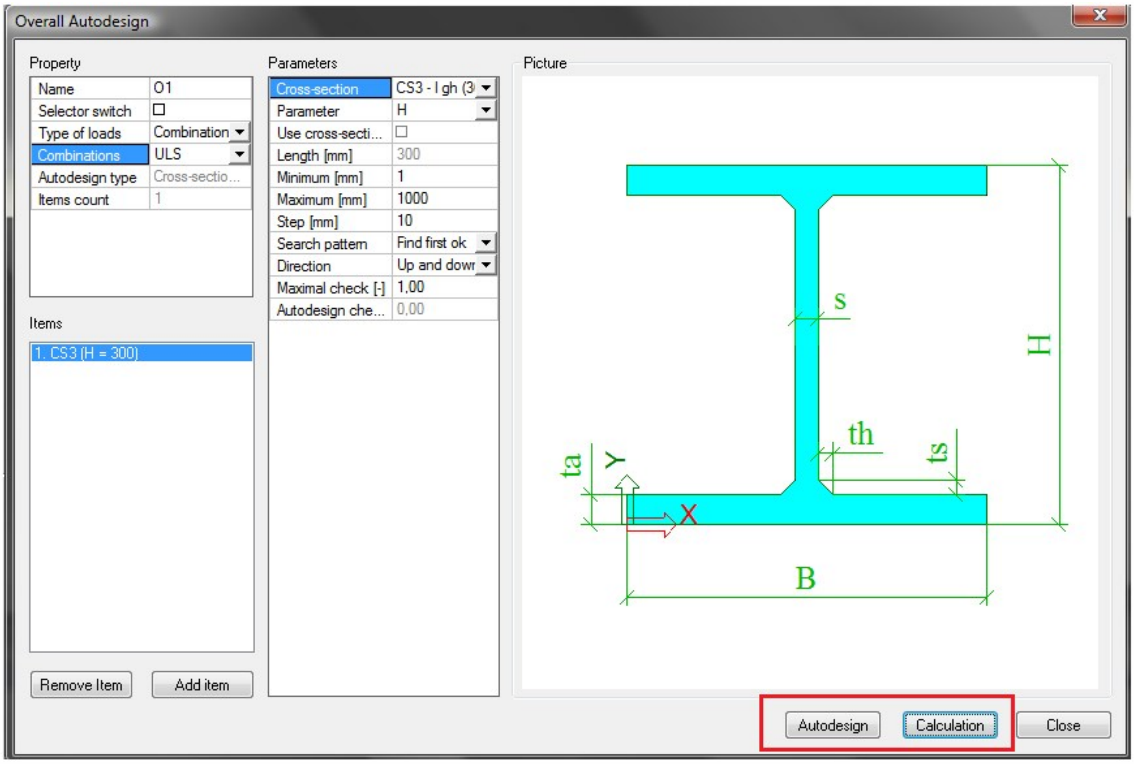
Item	Description
Step	Defines the step for Autodesign
Min.	Defines the minimal applicable size for the optimised parameter
Max	Defines the maximal applicable size for the optimised parameter

Picture

The picture shows the shape of the optimised item (cross-section, pad foundation or the symbol of the bolted diagonal connection etc).



Control buttons



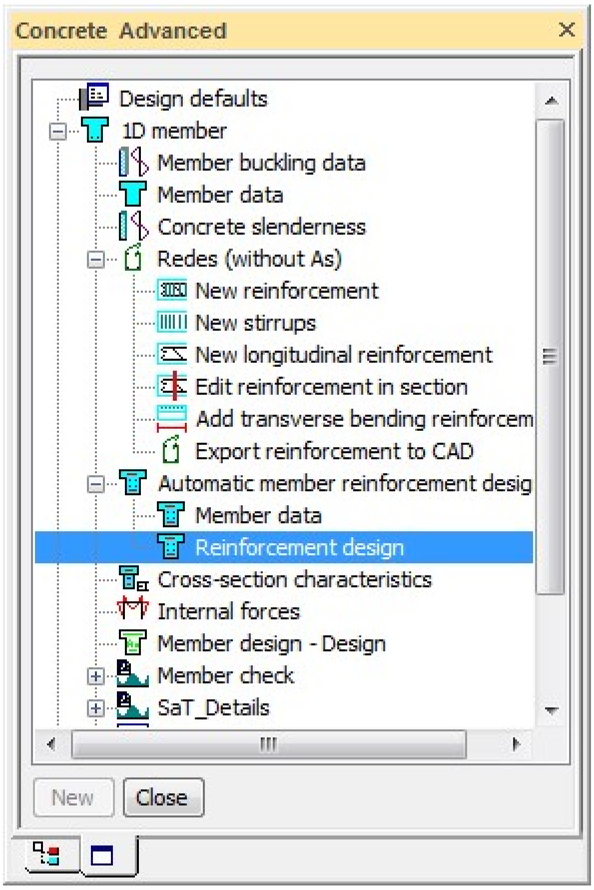
Autodesign
Performs the optimisation for the defined Autodesign items.

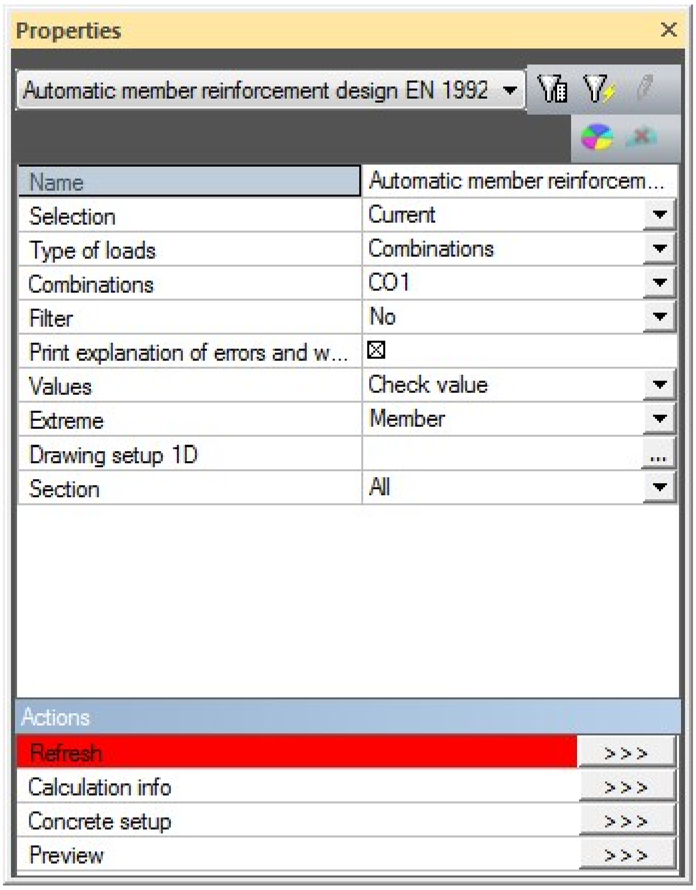
Calculation
Carries out the calculation for the optimised model.

Concrete – Automatic member reinforcement design (AMRD)

Autodesign of concrete section is the same part as the Reinforcement design performed in the service Concrete Advanced > Automatic member reinforcement design > Reinforcement design

Autodesign in Concrete Advanced service





When the user clicks on action button Refresh then the same procedure of Autodesign is performed. For concrete cross-section the procedure is called Automatic Member Reinforcement Design (AMRD). The non-prestressed reinforcement is designed in selected beam. Both longitudinal reinforcement and stirrups are designed.

Theoretical background for AMRD

The non-prestressed reinforcement in beams can be defined manually by the user or it can be calculated automatically by the program.

The latter designs the reinforcement on the basis of parameters defined in:

- reinforcement template,
- setup dialogue of service Concrete Advanced,
- member data related to the automatic design,
- practical reinforcement defined manually.

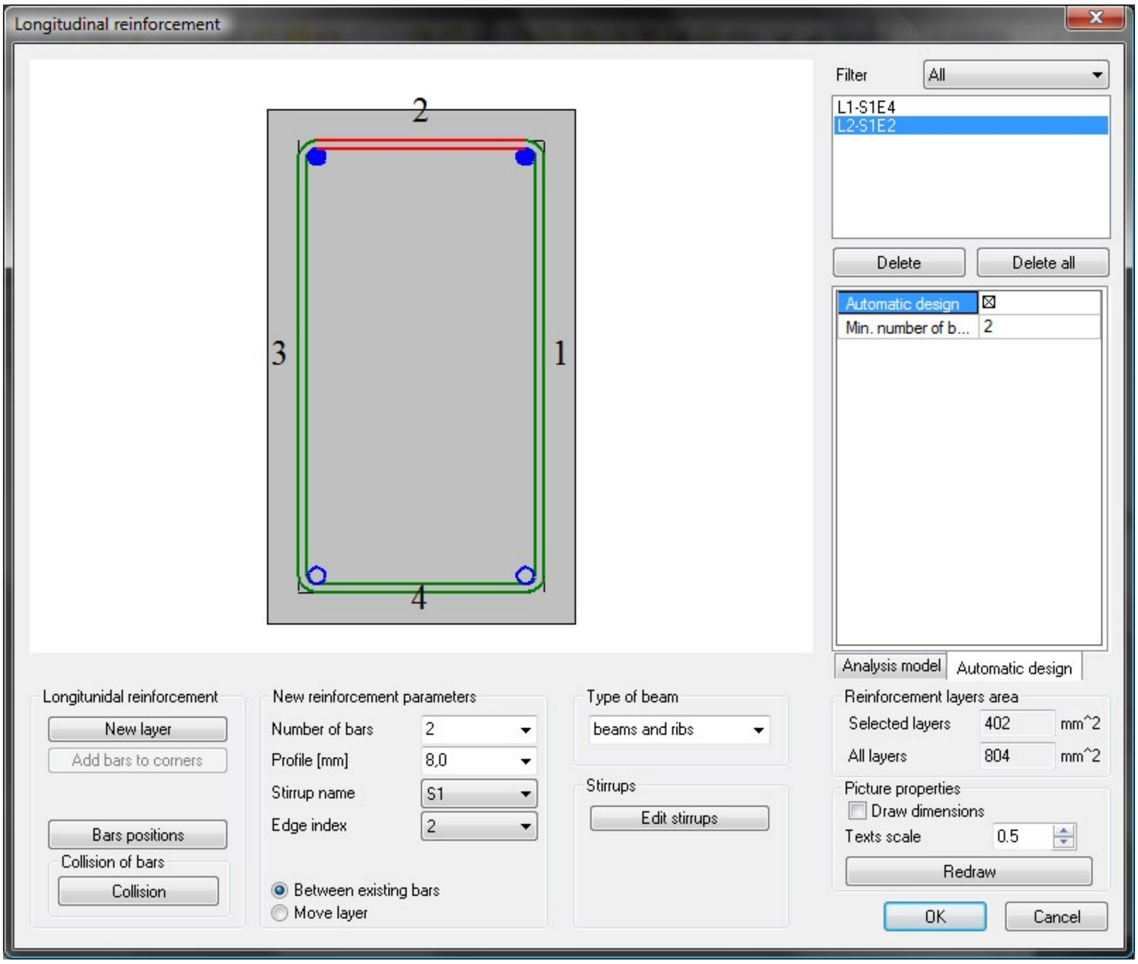
The automatic design takes into account the combination of bending moments and axial force and shear forces. It does not include torsion and deflections. It works within the ultimate limit state. The automatic design can be used for loads cases, ULS (not SLS) combinations and classes with ULS or ULS+SLS combinations. Concerning the parameters mentioned above, the practical reinforcement is of the highest priority. That means, if some reinforcement has been defined, the automatic design uses in the first step the diameter of this practical reinforcement. There is no output to the document. The results of the automatic design can be reviewed only on the screen in the graphical window and/or in the Preview window. Of course, the bill of reinforcement can be inserted into the document in order to show the reinforcement that has been automatically designed. The automatic design uses only the layers of the reinforcement marked in the reinforcement template. The automatic design is not capable of adding a new layer in the situation when the required reinforcement cannot be put into just one layer. Therefore, it may happen that the automatic design can fail.

The basic procedure is the following:

- First, the parameters that may affect the automatic design should be defined. It is necessary to define reinforcement template for longitudinal reinforcement, which layers can be optimise during the automatic reinforcement procedure,
- Specify the default parameters in
  - Setup dialogue of service Concrete Advanced affecting and controlling the procedure for the automatic design,
  - member data related to the automatic design, if required. These data overwrites the default values by member data that are specific for a particular beam.
- During Autodesign the standard design of reinforcement for combination N+My+Mz is performed (maximal bending moments along whole member are considered for design of reinforcement). The maximal amount of upper and lower reinforcement is designed.
- The designed reinforcement template is used for checks (Interaction diagram, detailing provisions). The calculation is evaluated based on the maximal utilisation (check value) in concrete setup. When the calculated check value is still less than the defined utilisation, then bars are deleted to achieve the optimal utilisation.

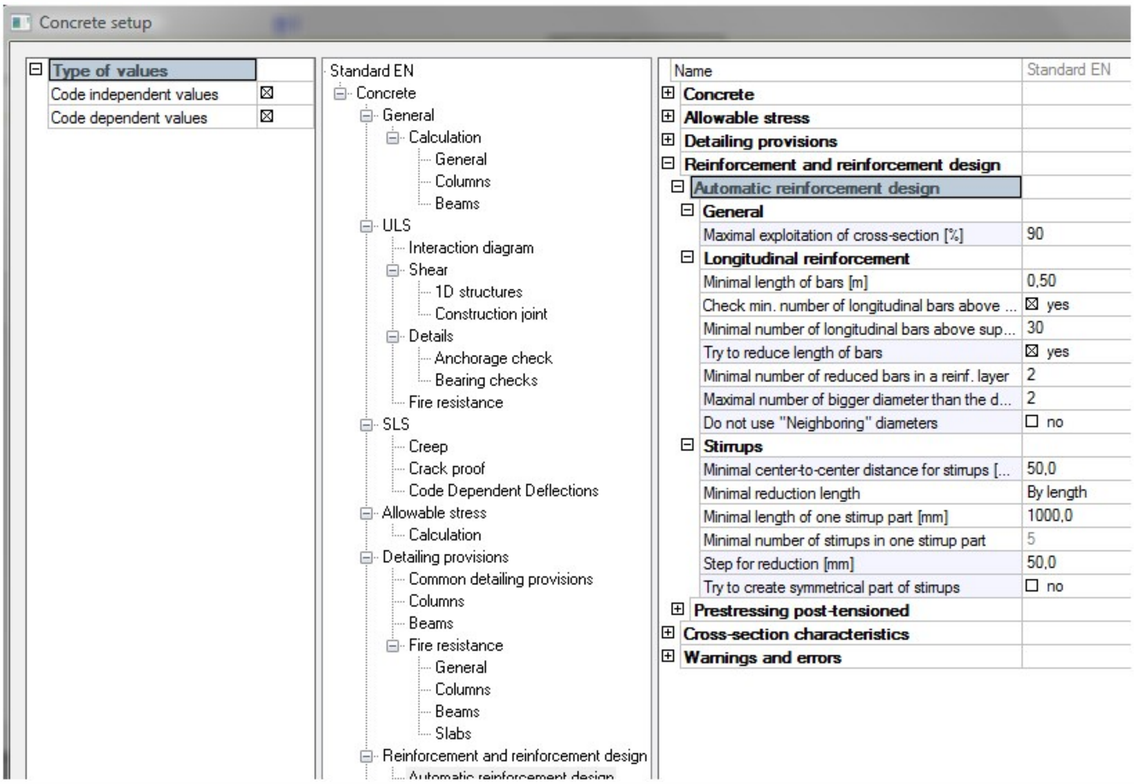
Template preparation

The template used for the Autodesign is prepared in a standard way. One difference is that the check box Automatic member design is switched ON.



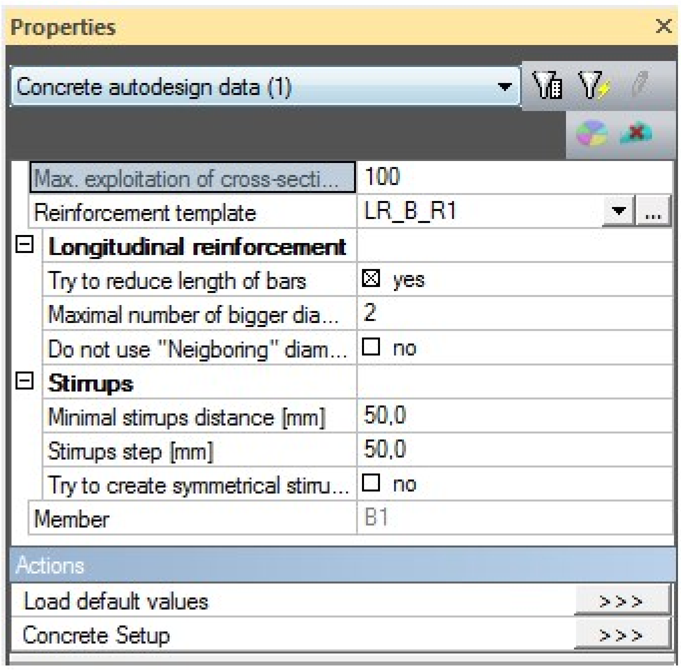
Concrete setup

The default settings used for Autodesign are stored in Concrete setup.



Member data for the automatic reinforcement design

The member data used for AMRD looks like this:



General

Max. exploit-ation of cross- Specifies the maximal utilisation of the cross-section in the automatically reinforced beam. The value may be between 1 and 100%.

section	
Reinforcement template	Shows the used reinforcement template. Note: This item appears in the dialogue ONLY when the already defined member data are edited. If the member data are being assigned to a new member, this item is not accessible.

Longitudinal reinforcement

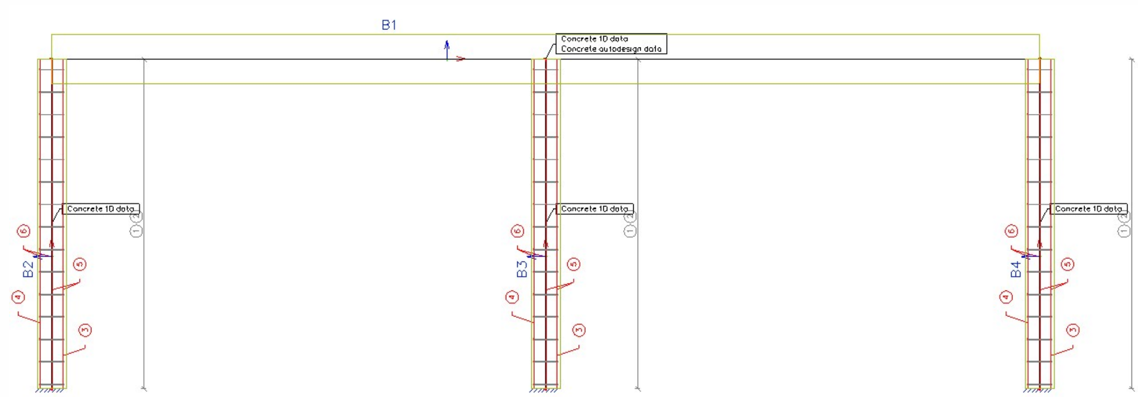
Try to reduce length of bars	If OFF, the program uses only bars that extent over the whole length of the beam. If ON, some bars may be shortened if the unity check is satisfied without them.
Maximal number of bigger diameters than the default	Defines how many different (bigger) diameters of the reinforcement can be used for the optimisation. Let us assume that the default diameter specified in the Design default tab is 10 mm. If this parameter is set to 2, the program can use diameters 10, 12 (i.e. +1 item in the manufacturing programme) and 14 (i.e. +2 item in the manufacturing programme) for the design.
Do not use "Neighbouring" bars	Some standards recommend that "neighbouring" profiles from the manufacturing programme should not be used in one beam (in order to avoid unintentional interchange of the profiles). Let us assume that the default diameter specified in the Design default tab is 10 mm. Further assume that Maximal number of bigger diameters than the default is set to 2. If this option is ON, the following bars can be inserted into the beam: (i) either 10 mm, (ii) or 12 mm, (iii) or 14 mm, (iv) or 10 mm and 14 mm can be combined together. 10 mm and 12 mm are not permitted to be combined in one beam

Stirrups

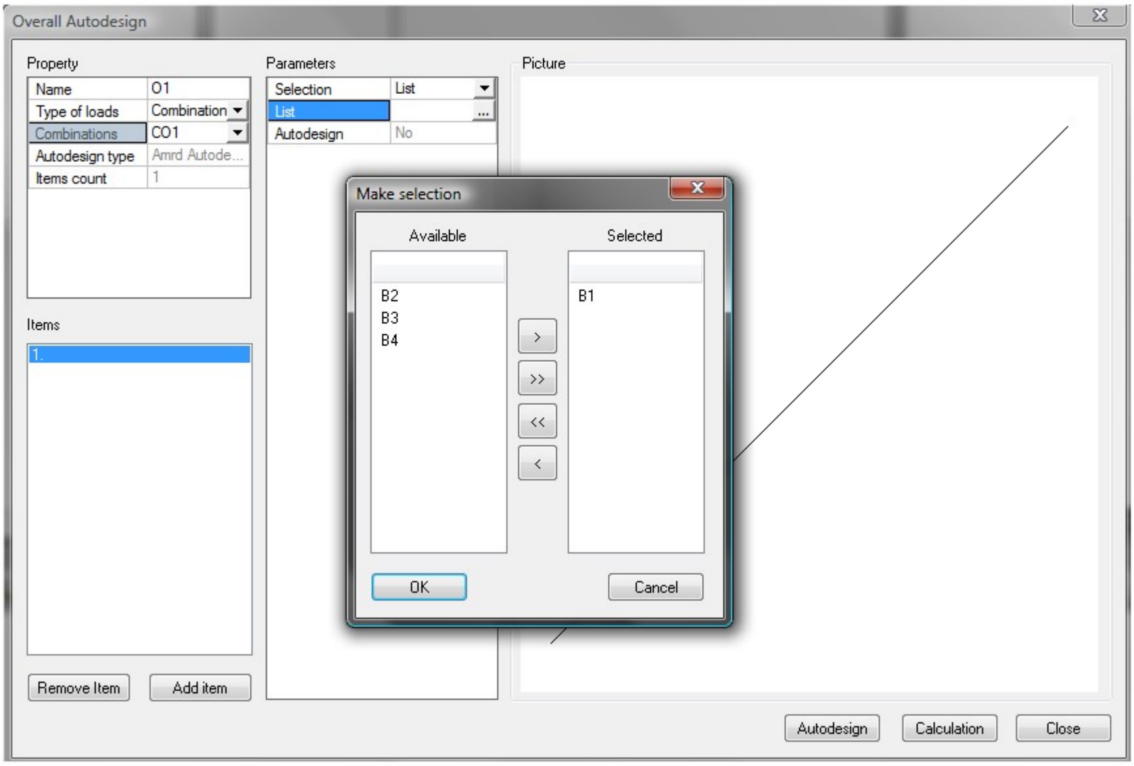
Minimal stirrups distance	Specifies the minimal distance between stirrups measured from the centre of a bar to the centre of an adjacent bar.
Stirrups step	Defines the step for the reduction of the distance between two adjacent stirrups. This ensures that the distance between stirrups is always a "rounded" number – e.g. 200 mm, then 250 mm, then 300 mm, etc. (and not e.g. 200, 246 mm, 298 mm, etc.).
Try to create symmetrical stirrups parts	This parameter may enforce that the stirrup parts are symmetrical along the length of the beam.
Member (informative only)	Shows the name of the beam where the member data are assigned to

Illustrative example

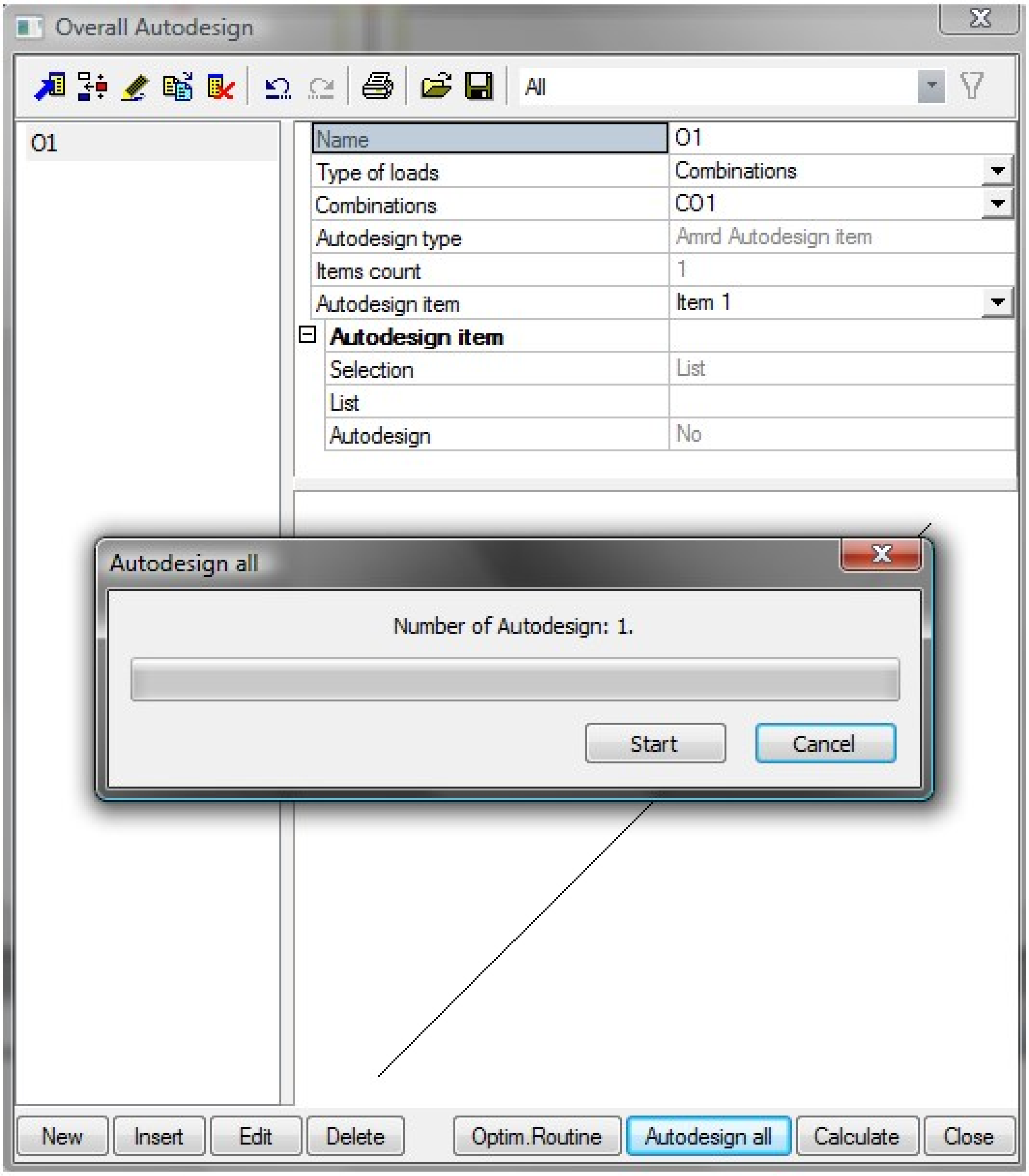
Let us consider a very simple example of concrete frame. The structure is subjected to several loads (self weight, permanent, variable, wind etc.). The aim of this example is to find the optimal reinforcement pattern in the continuous horizontal member of the frame.



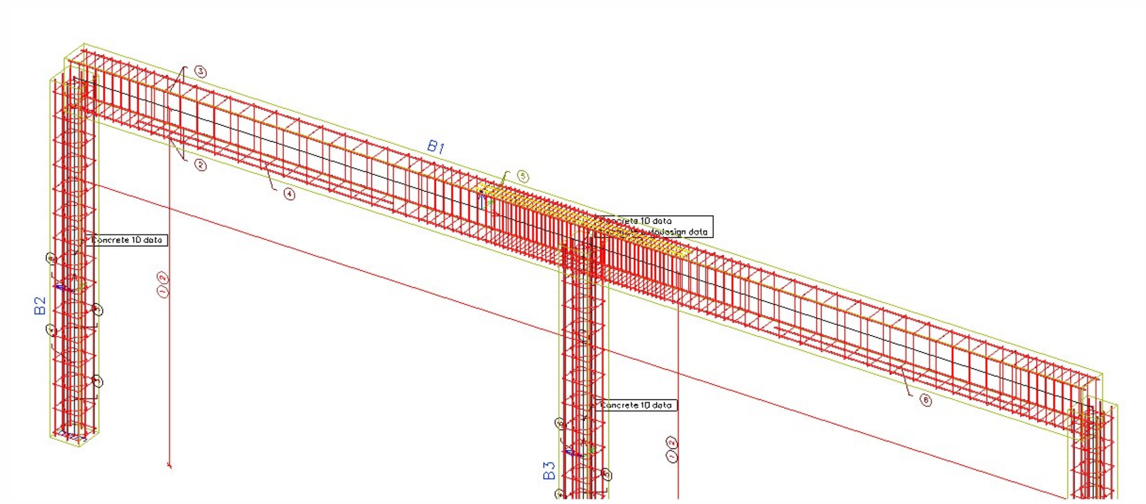
One Autodesign function with type AMRD Autodesign is defined and reinforcement pattern for beam B1 is optimized. No additional settings related to parameters are defined for this case. Possibly AMRD data can be defined on the beam only.



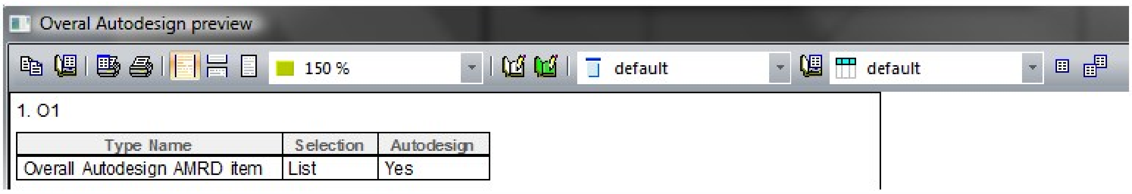
Autodesign starts after pressing button Autodesign all.



The obtained results of designed reinforcement for beam B1 are the following. The reinforcement is automatically designed and input on beam B1. You can see higher density of stirrups near the supports and additional longitudinal bars in the span and above the middle support.



The document output is very simple.


Overal Autodesign preview

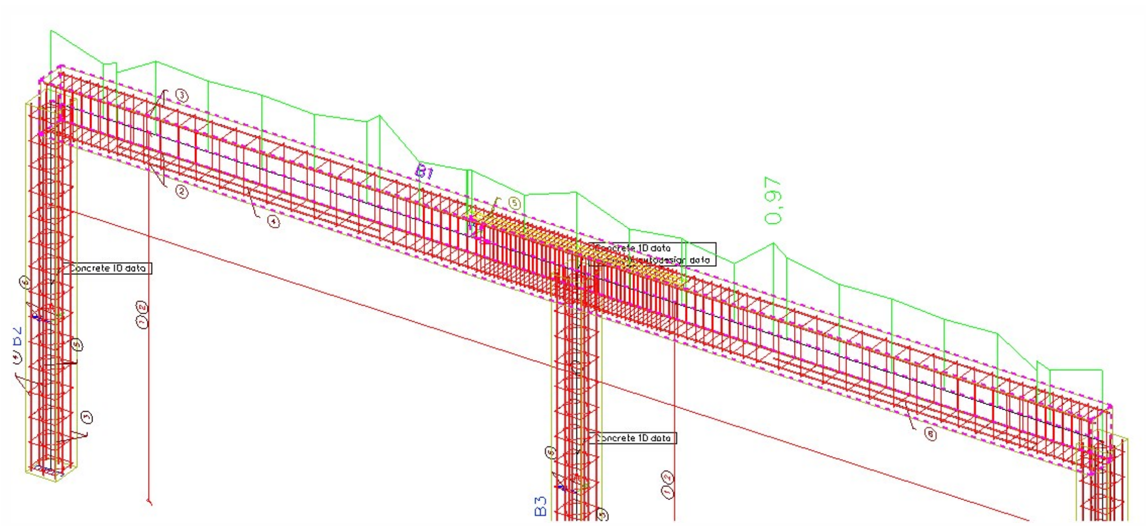
150 % default default

1. O1

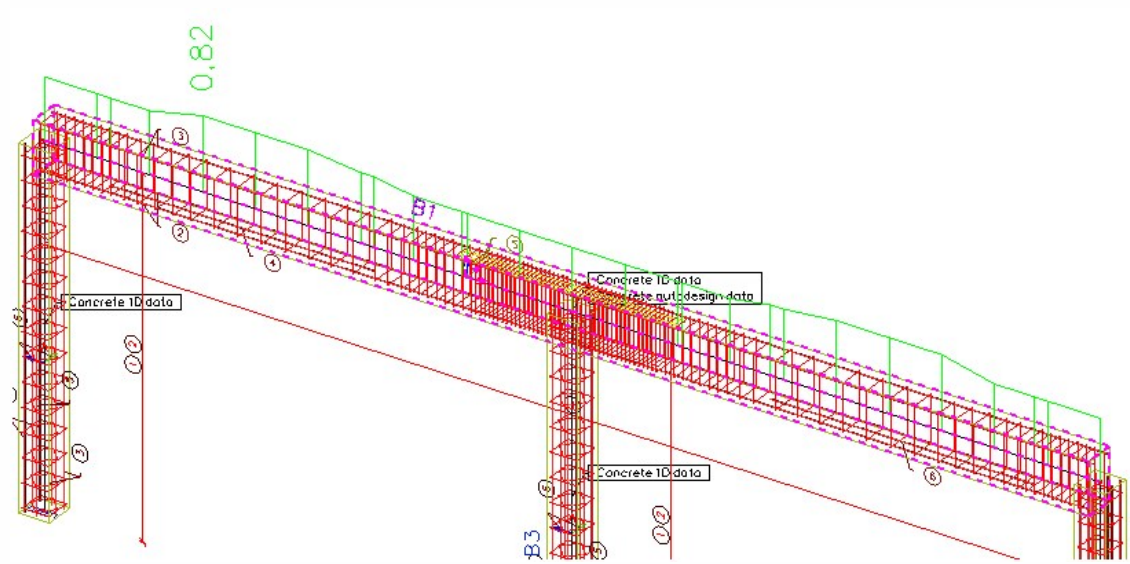
Type Name	Selection	Autodesign
Overall Autodesign AMRD item	List	Yes

The results can be verified in the standard concrete checks:

- Check capacity (max check value 0,97)



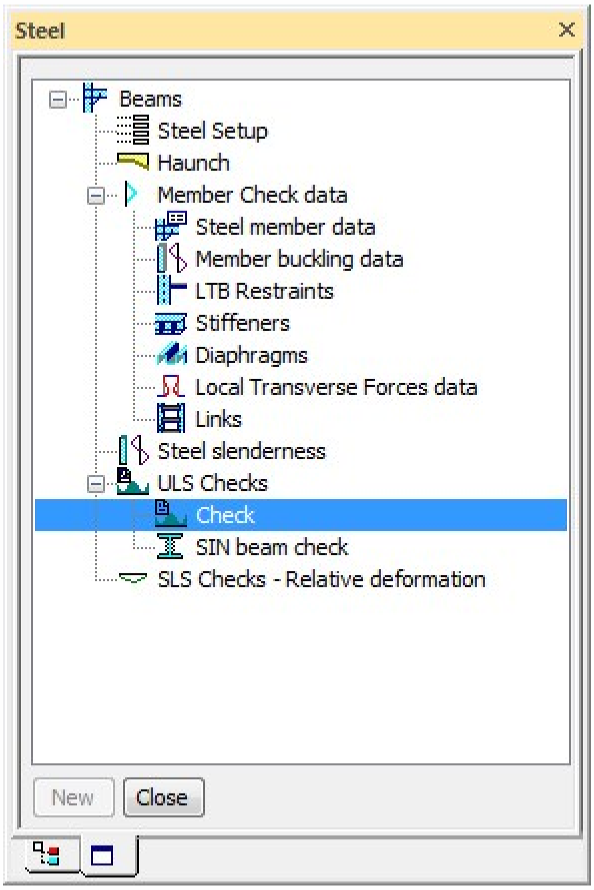
- Detailing provision (max check value 0,82)

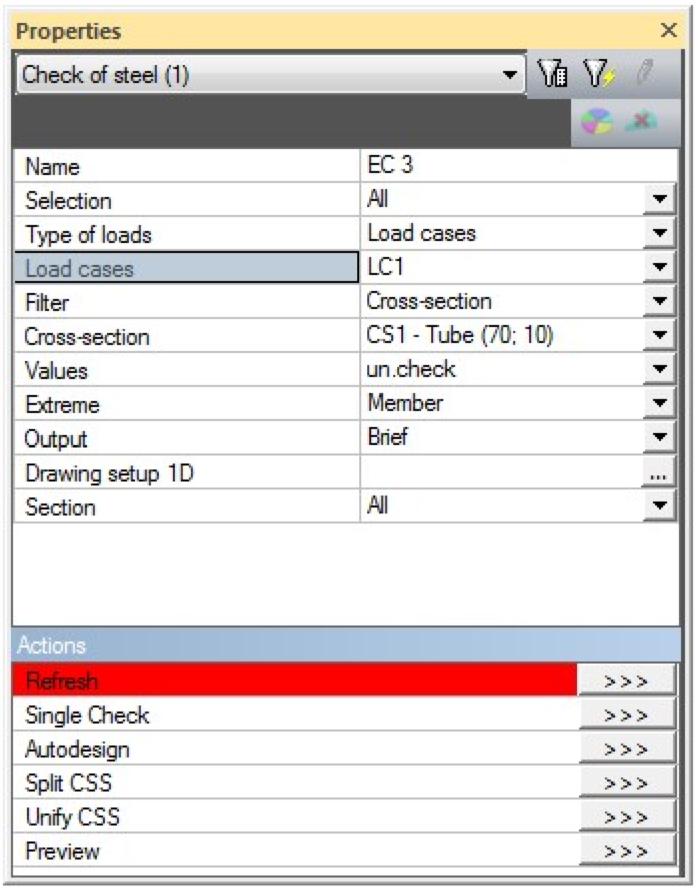


Steel – Cross-section AutoDesign

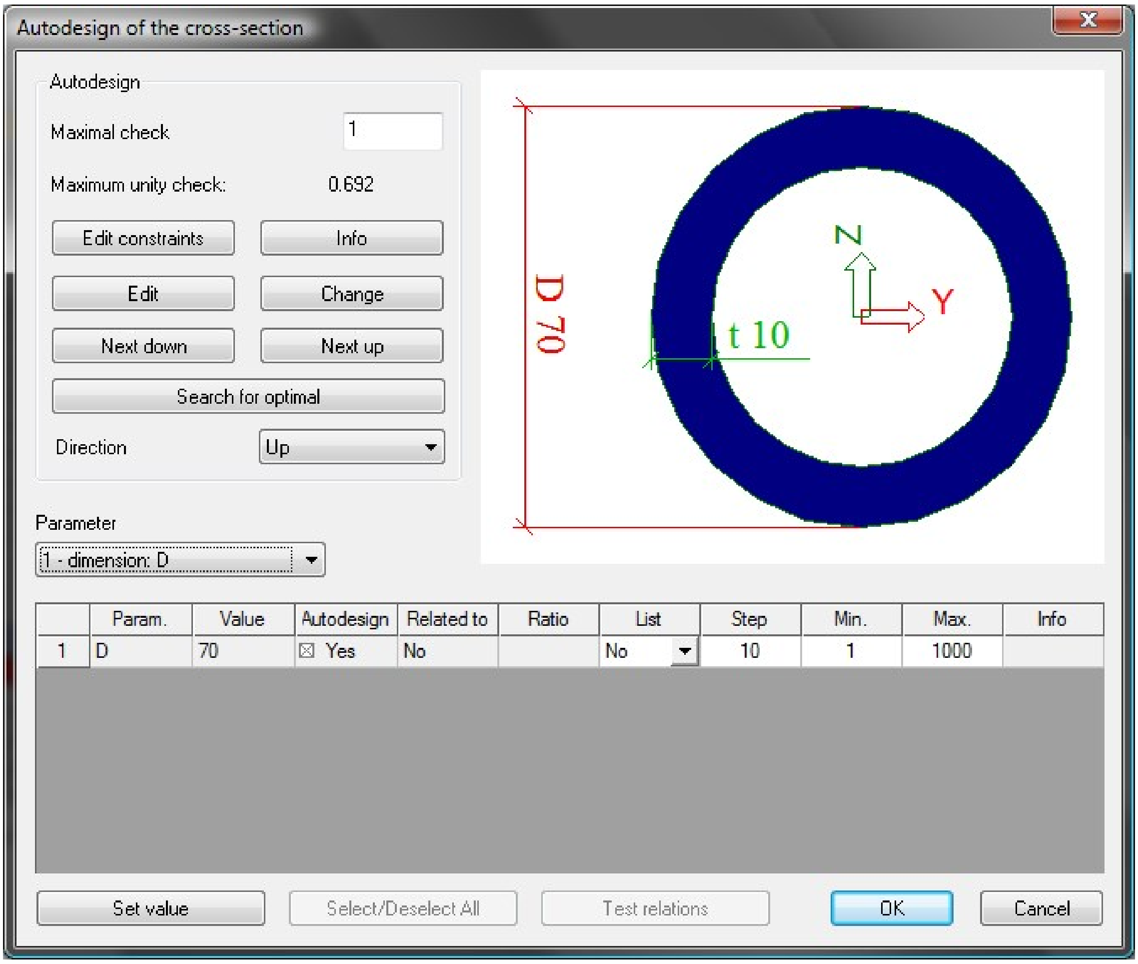
Autodesign of cross-section steel check is the same part as the Autodesign located in the standard Steel >ULS Check> Check.

Autodesign in Steel service





When user clicks on action button Autodesign then the following dialogue is displayed. The dialogue is a bit different than the one used in the general Autodesign dialogue but the functionality is the same.

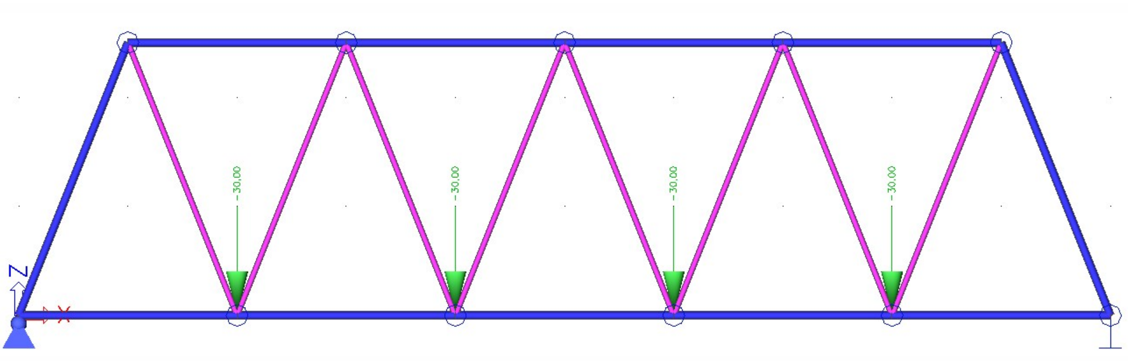


We will focus on Autodesign running from Autodesign service in this case. Generally, results based on the same settings in Autodesign and the individual service have to be the same.

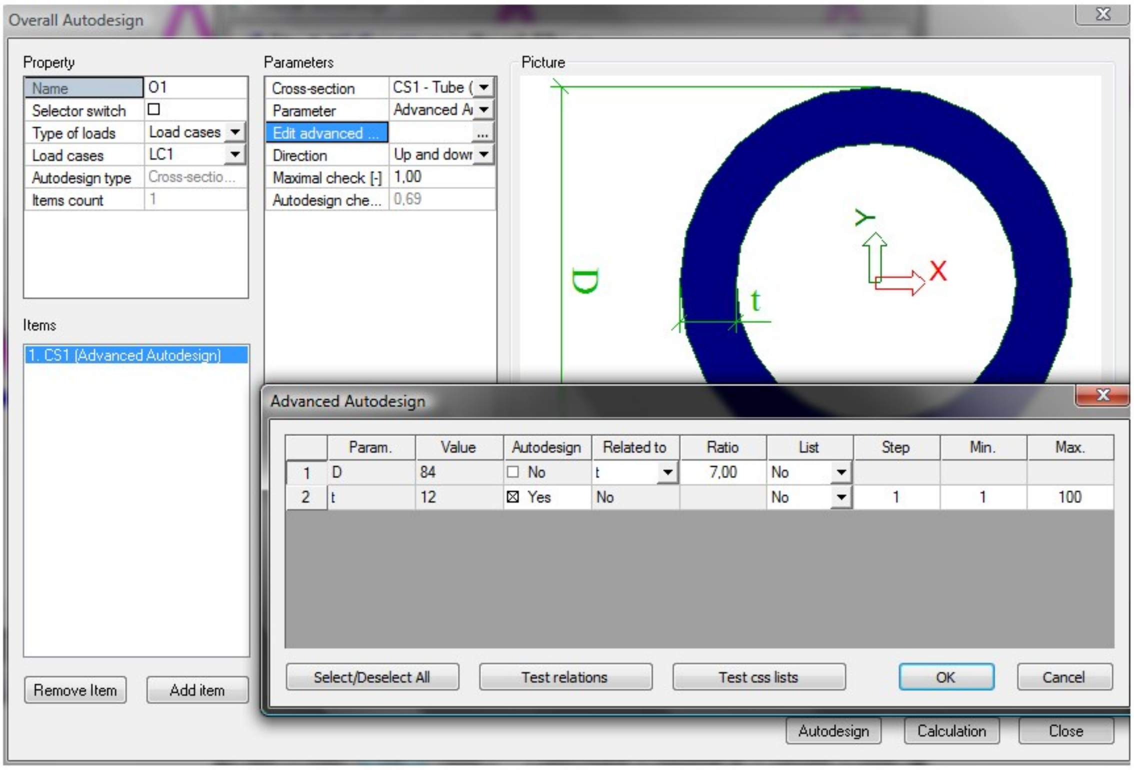
Illustrative example

Let us consider a very simple example of a steel truss girder for Autodesign. The structure is shown in the following figure. The structure is subject to four point loads acting on the bottom chord. The aim of this example is to find the optimal dimensions of two tubular cross-sections. The initial dimensions of the tubular cross-sections are:

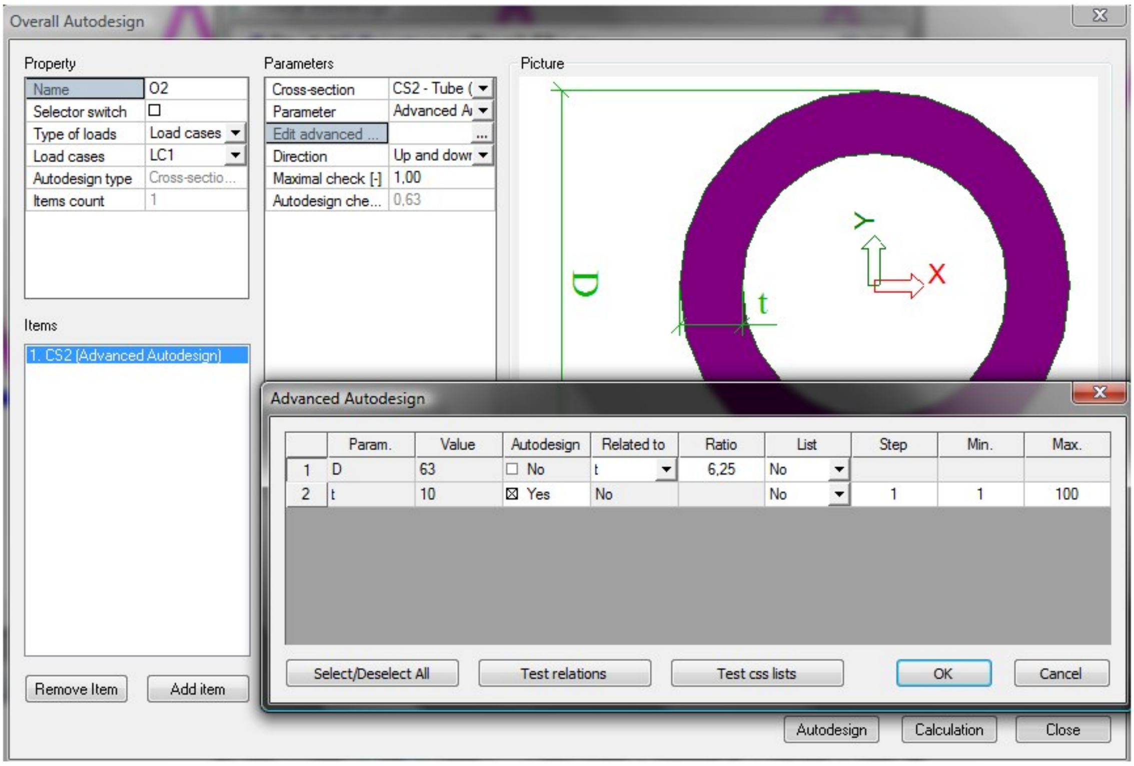
Cross-section	t [mm]	D [mm]
CS1 - tube	12	84
CS2 - tube	10	62.5



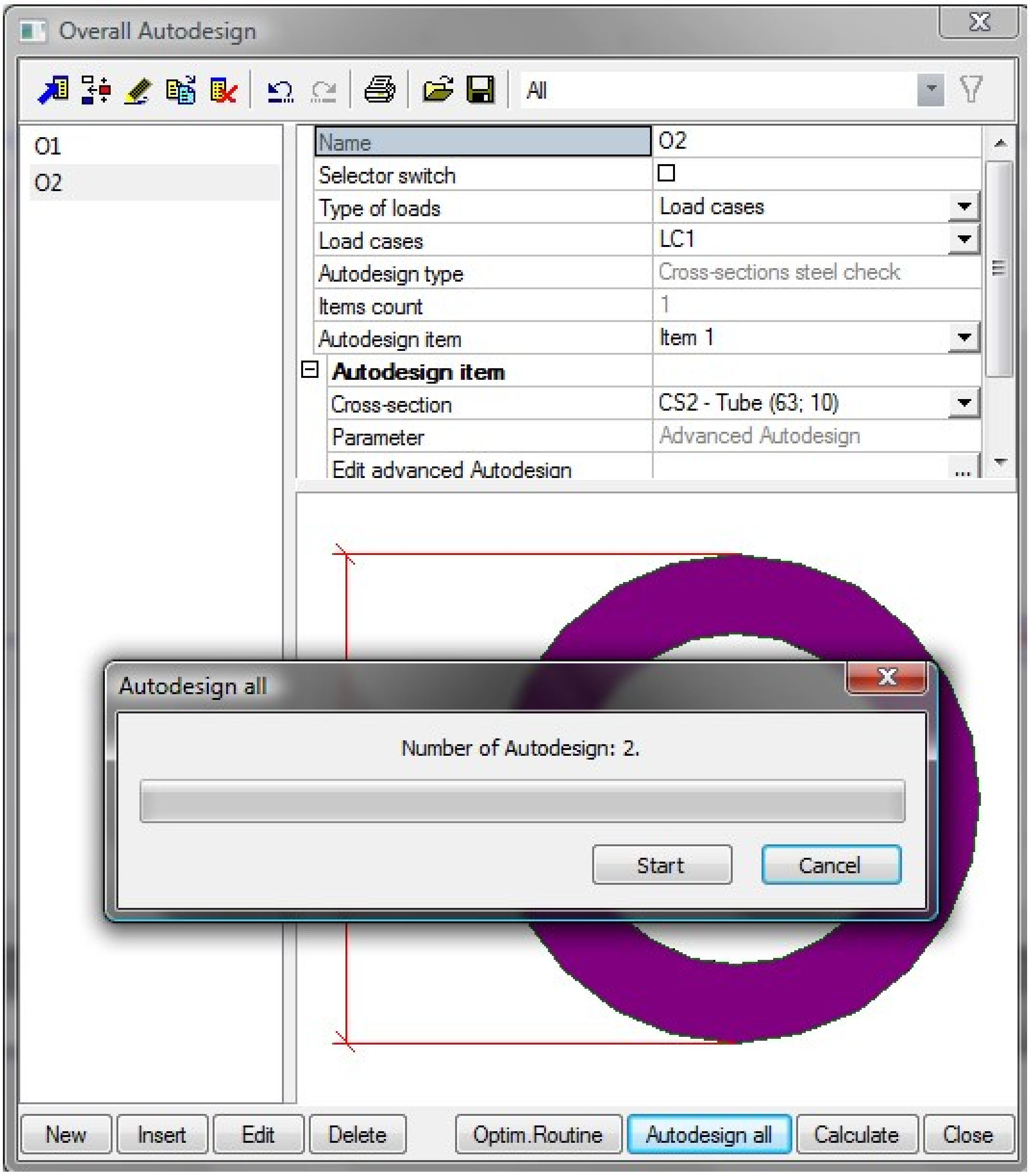
The Autodesign function is defined for each of the cross-sections. You can see the settings for CS1 – Tube,



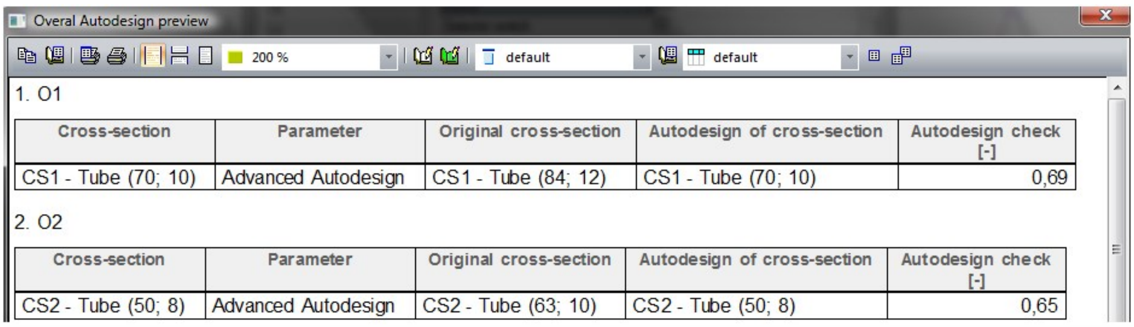
and for the CS2 – Tube. The advanced Autodesign is used for both cross-sections. Thickness of the tube (t) is optimized (Autodesign is YES) and the diameter (D) depends on the thickness (Related to t through the defined ratio).



Autodesign of both cross-sections can be run in one step using Autodesign all..



The results are automatically printed in preview.



Overal Autodesign preview

1. O1

Cross-section	Parameter	Original cross-section	Autodesign of cross-section	Autodesign check [-]
CS1 - Tube (70; 10)	Advanced Autodesign	CS1 - Tube (84; 12)	CS1 - Tube (70; 10)	0,69

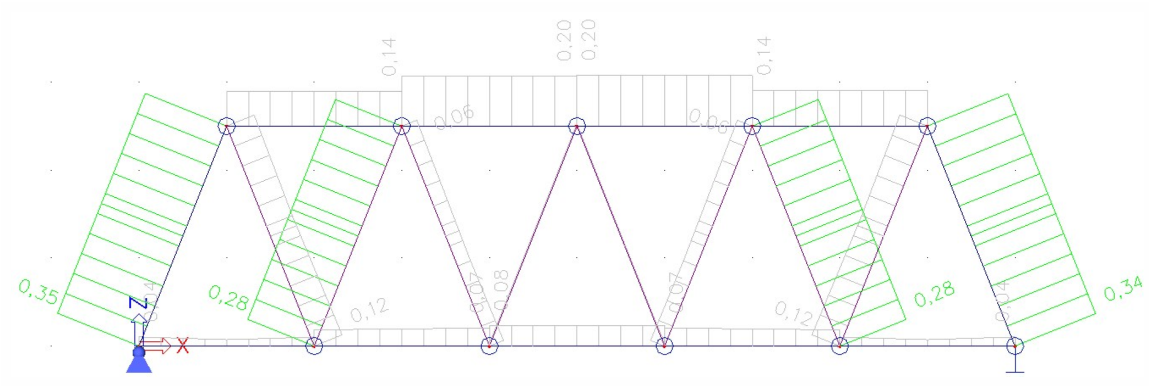
2. O2

Cross-section	Parameter	Original cross-section	Autodesign of cross-section	Autodesign check [-]
CS2 - Tube (50; 8)	Advanced Autodesign	CS2 - Tube (63; 10)	CS2 - Tube (50; 8)	0,65

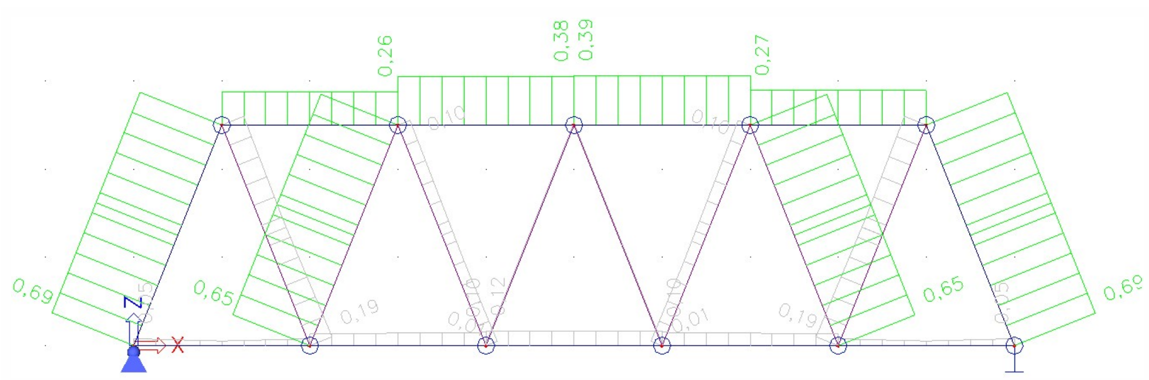
The comparison of the dimensions is in the following table.

Cross-section	Initial		Optimized	
	t [mm]	D [mm]	t [mm]	D [mm]
CS1 - tube	12	84	10	70
CS2 - tube	10	62.5	8	50

Evaluation of the steel unity check along the beam is compared in the following figures.



Evaluation of steel check for initial cross-section dimensions

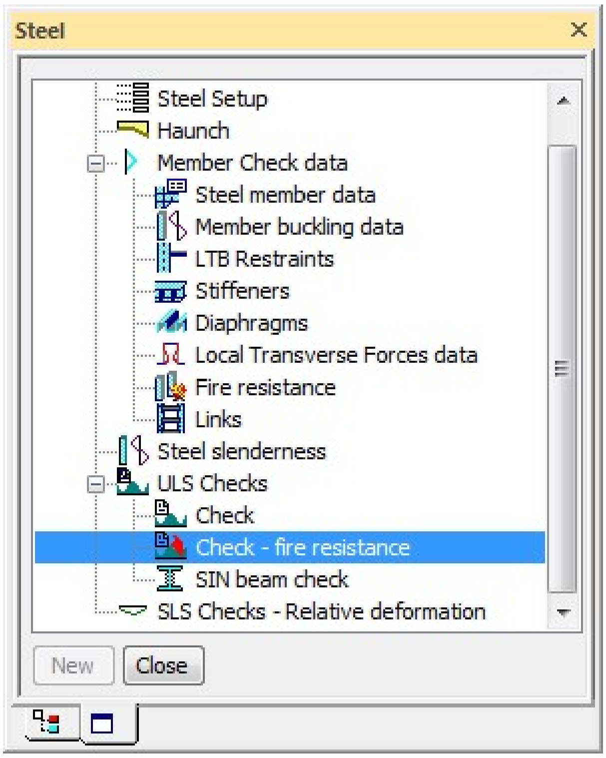


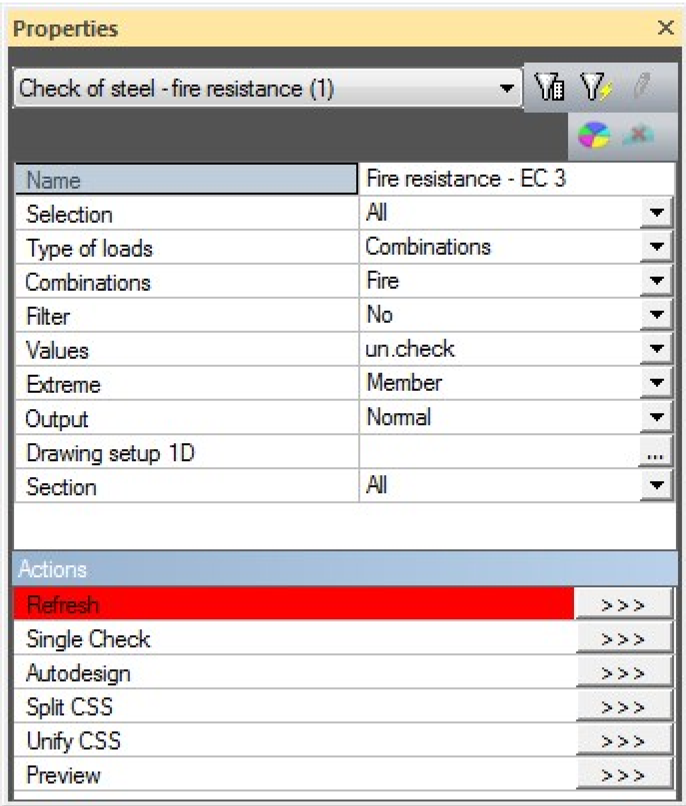
Evaluation of steel check for optimized cross-section dimensions

Steel - Fire resistance AutoDesign

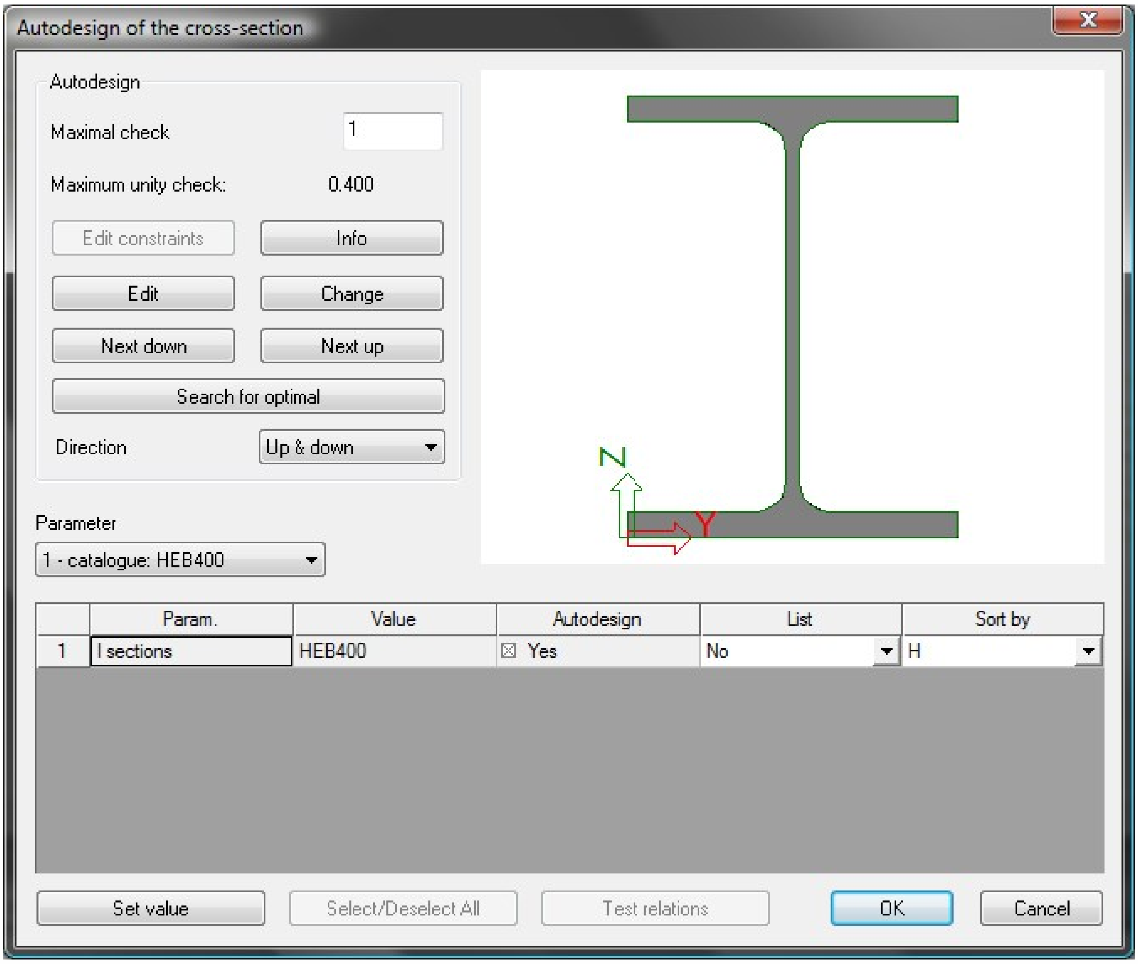
Autodesign of steel fire resistance check is the same part as the Autodesign located in the standard Steel > ULS Check > Check - Fire resistance.

Autodesign in Steel service



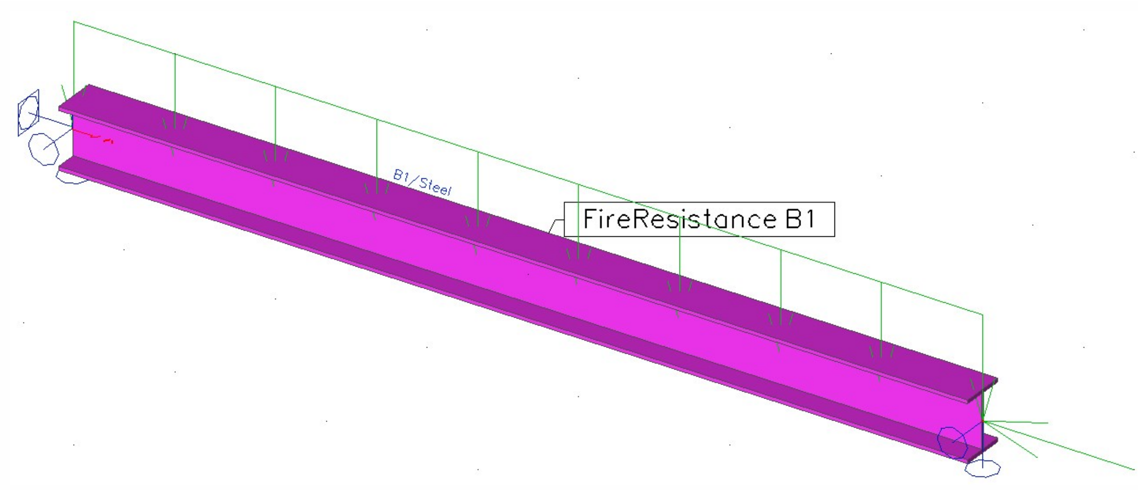


When the user clicks on action button Autodesign then the following dialogue is opened. The dialogue is a bit different from the one used in the general Autodesign, but the functionality is the same.

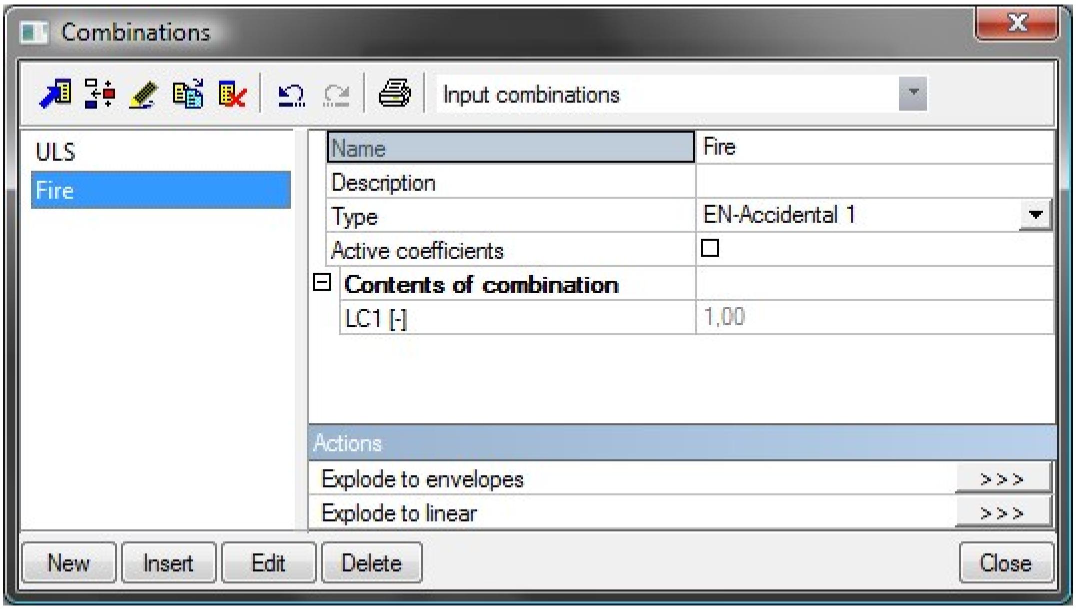


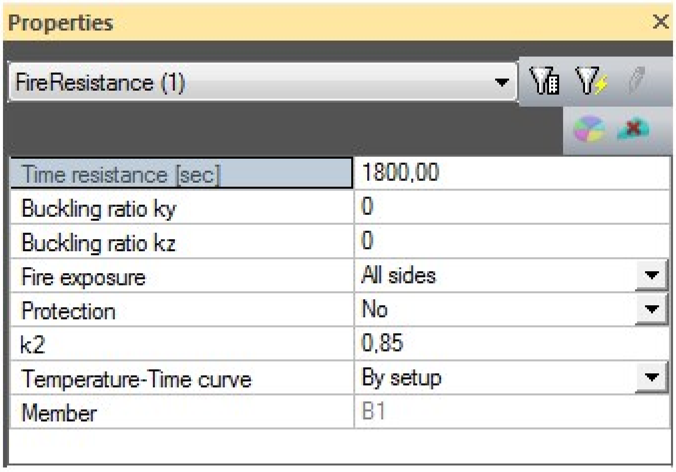
Illustrative example

Let us consider a very simple example of a steel beam with HEB400 cross-section for Autodesign. The structure is subject to the uniform load and to the axial force at one end. The aim of this example is to find the optimal height of the cross-section.

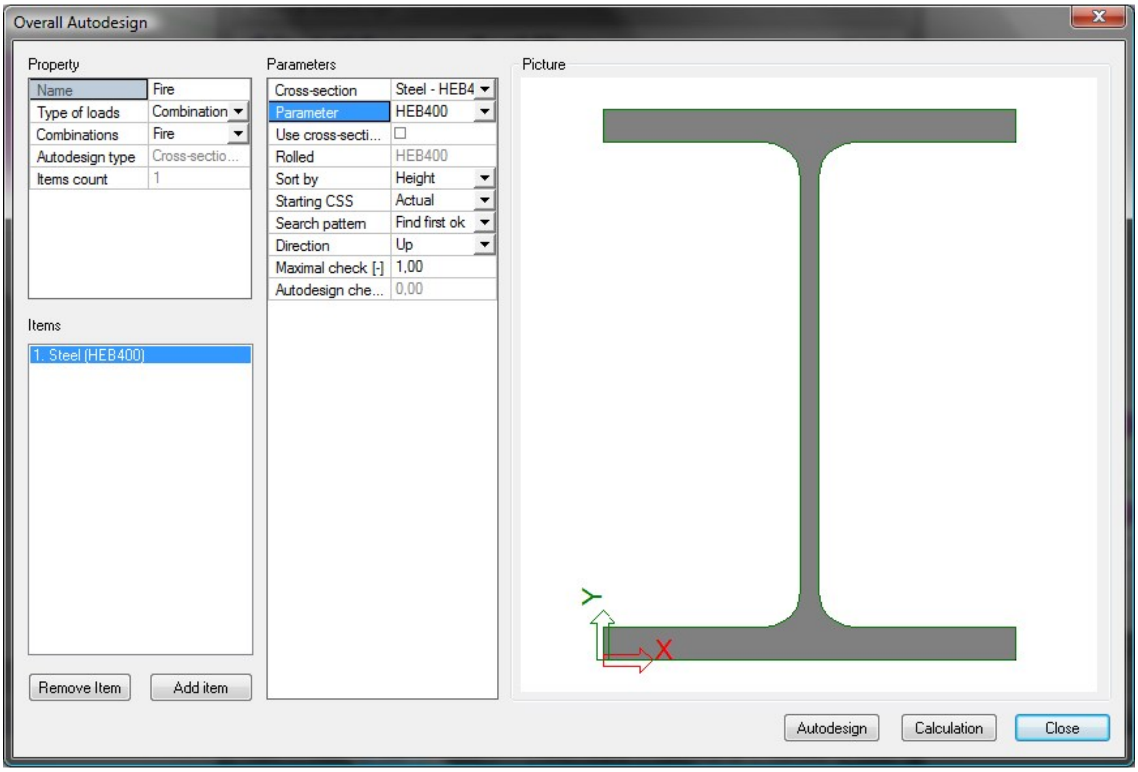


The accidental combination has to be defined. The fire resistance data are used as defined on the following figure.

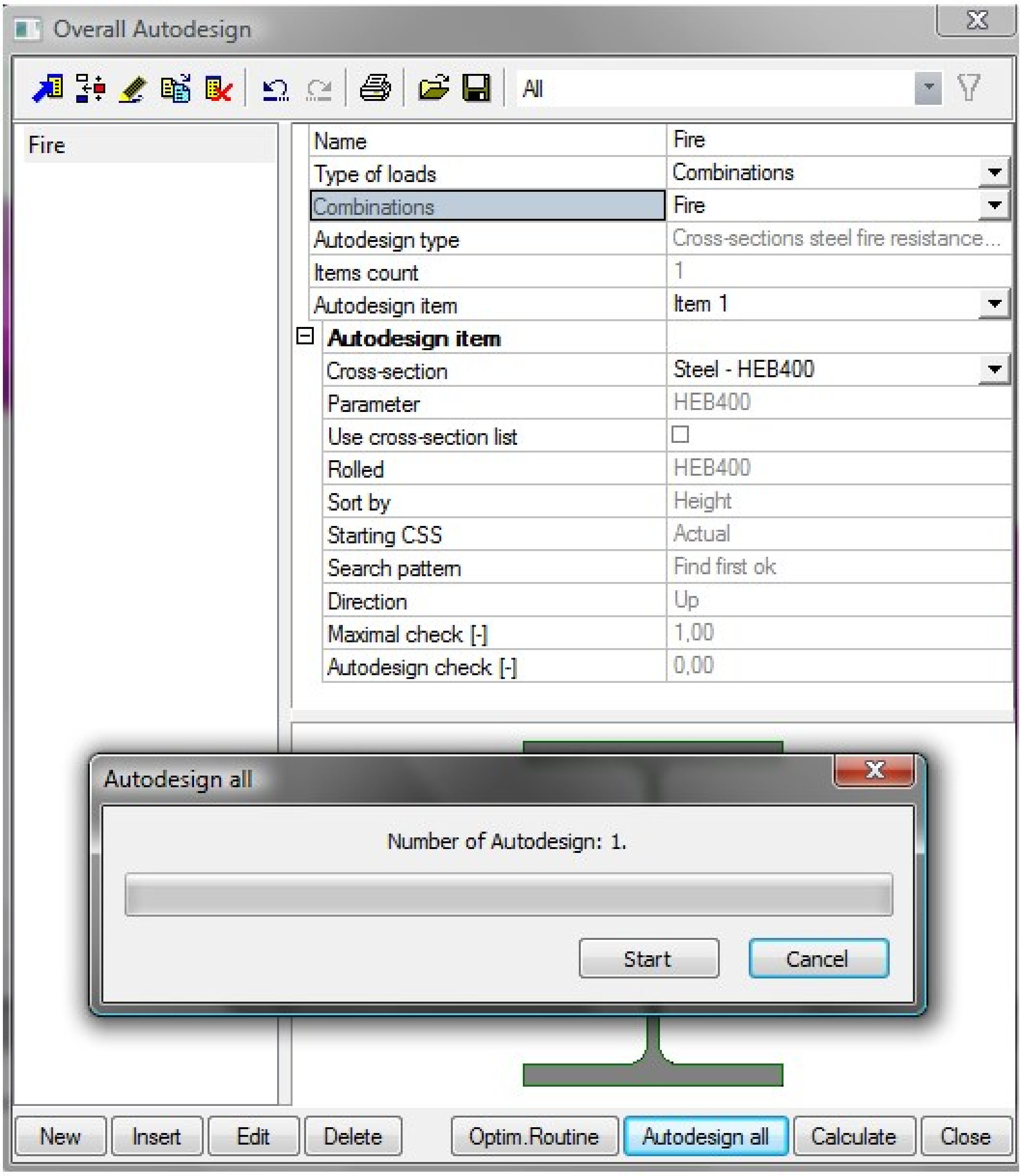




The Autodesign function is defined for HEB400 cross-section. The cross-section is rolled therefore the properties of Autodesign for rolled cross-section are used. You can see the settings for CS1 – Tube,



Autodesign of both cross-sections can be run in one step using Autodesign all.



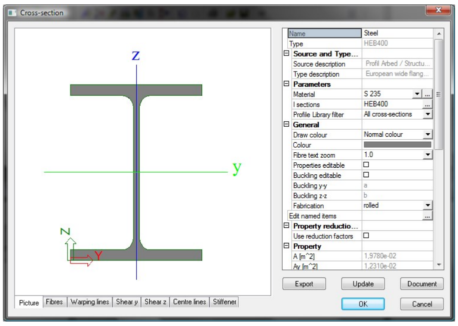
The results are automatically printed in preview.

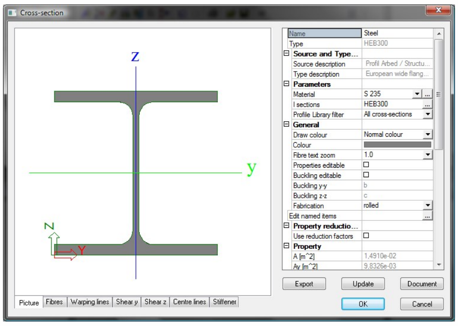
1. Fire

Cross-section	Parameter	Sort by	Original cross-section	Autodesign of cross-section	Autodesign check [-]
Steel - HEB300	HEB300	Height	Steel - HEB300	Steel - HEB300	0,86

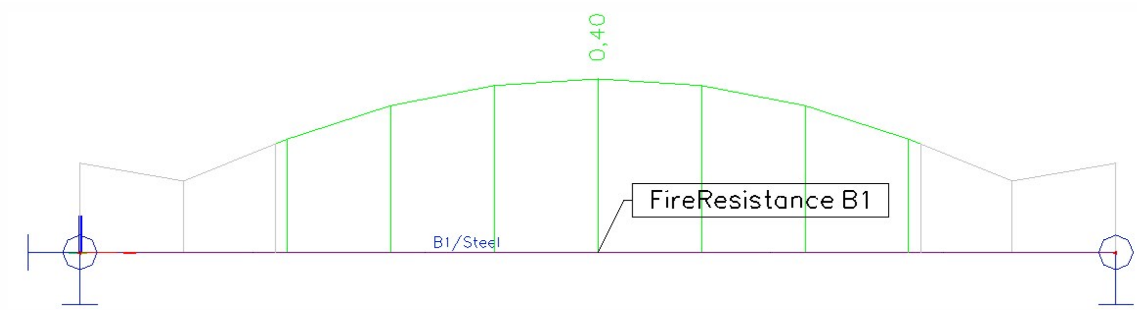
The comparison of initial and optimized values of cross-section is clear from the following table. Initially HEB400 was used and the optimal cross-section is HEB300.

Initial Optimized





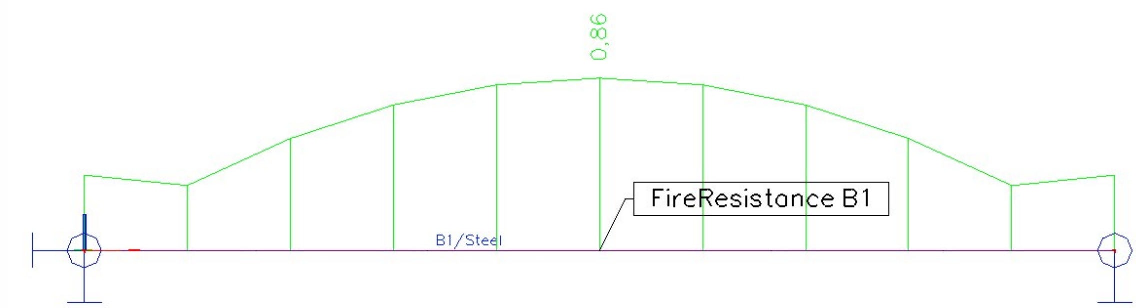
Evaluation of the rolled cross-section unity check along the beam is compared in the following figures. The first figure together with table is for the initial cross-section.



Check of steel - fire resistance

Type Name	Case	Member	css	mat	dx [m]	un.check [-]	sec.check [-]	stab.check [-]
Check of steel - fire resistance	Fire/1	B1	Steel - HEB400	S 235	3,000	0,40	0,33	0,40

The second figure together with table is for the optimized cross-section.



Check of steel - fire resistance

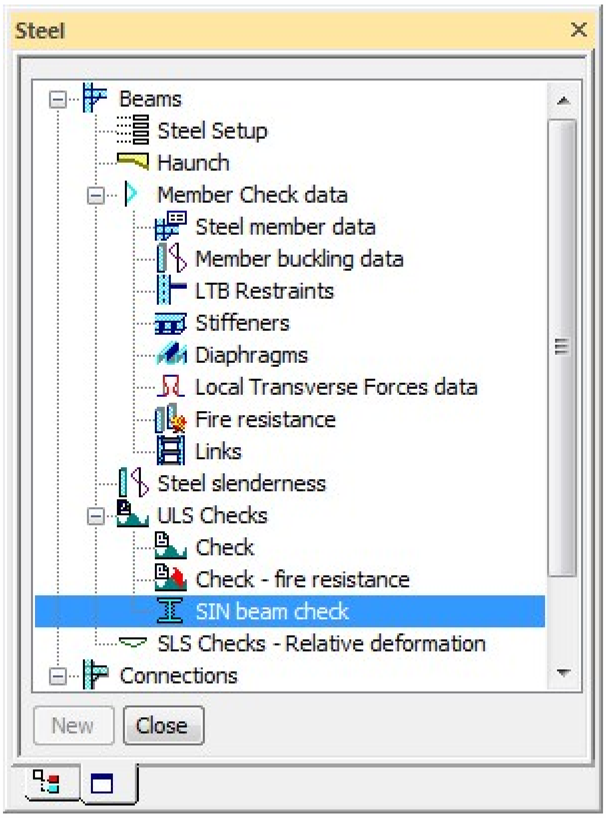
Type Name	Case	Member	css	mat	dx [m]	un.check [-]	sec.check [-]	stab.check [-]
Check of steel - fire resistance	Fire/1	B1	Steel - HEB300	S 235	3,000	0,86	0,70	0,86

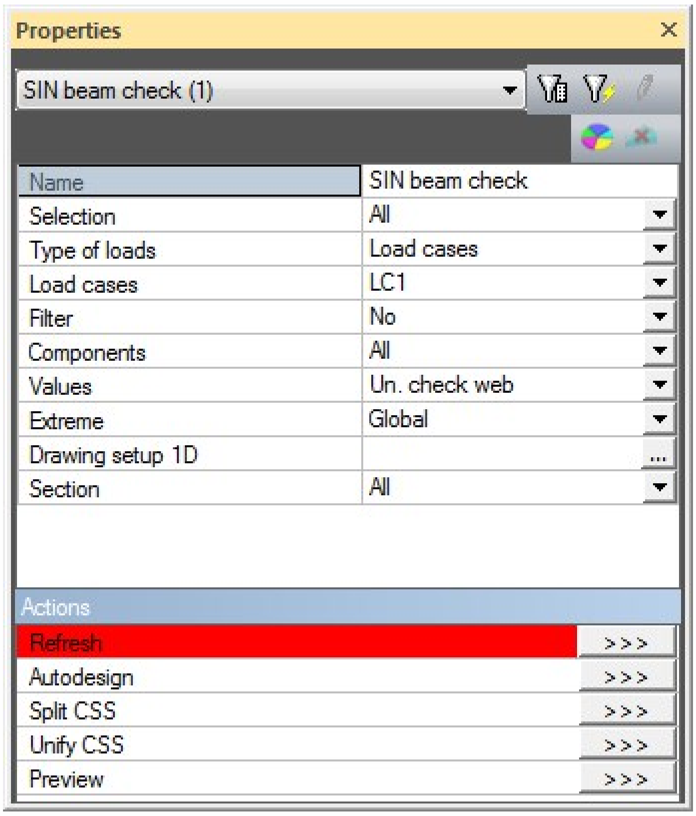
Evaluation of steel check for optimized cross-section dimensions

Steel - Corrugated web AutoDesign

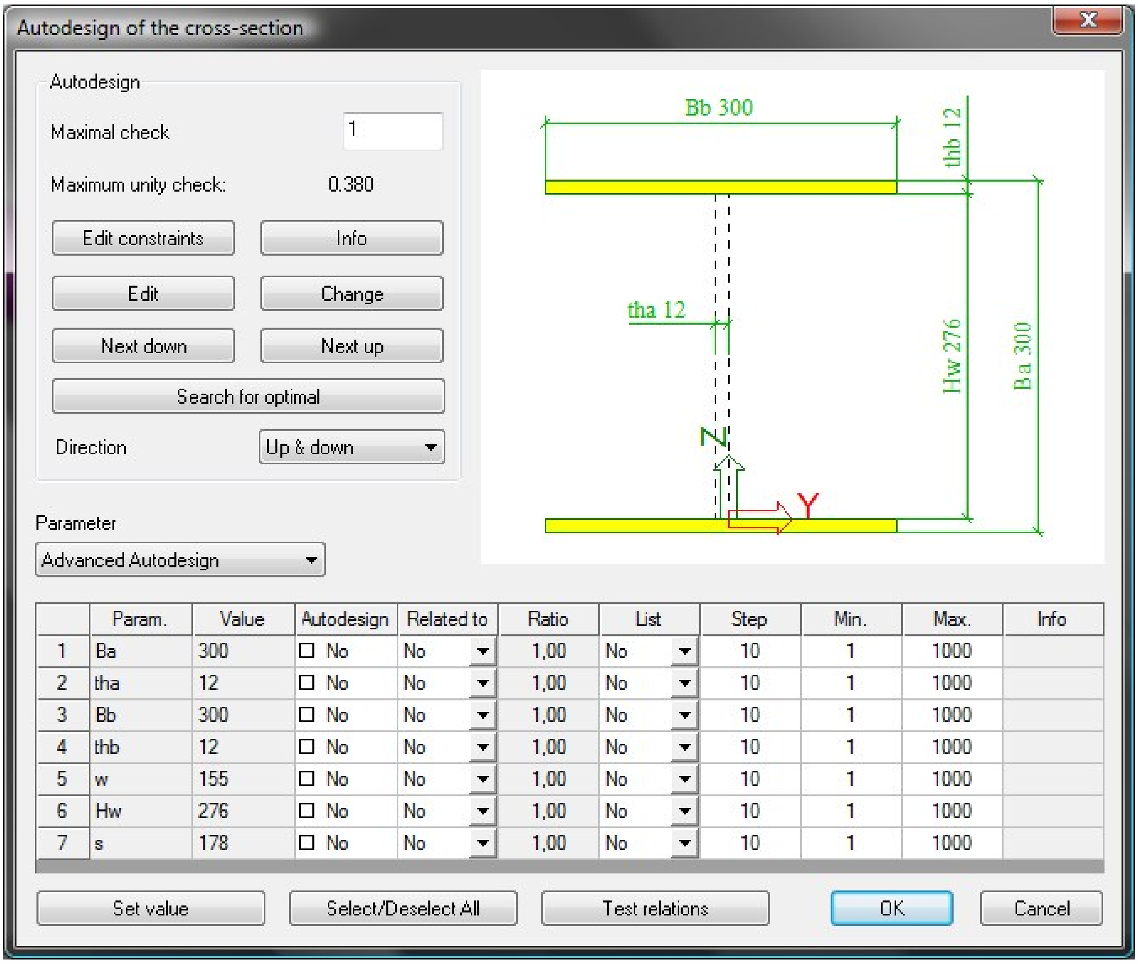
Autodesign of corrugated cross-section check is the same part as the Autodesign located in the standard Steel > ULS Check > SIN beam check.

Autodesign in SIN beam check service





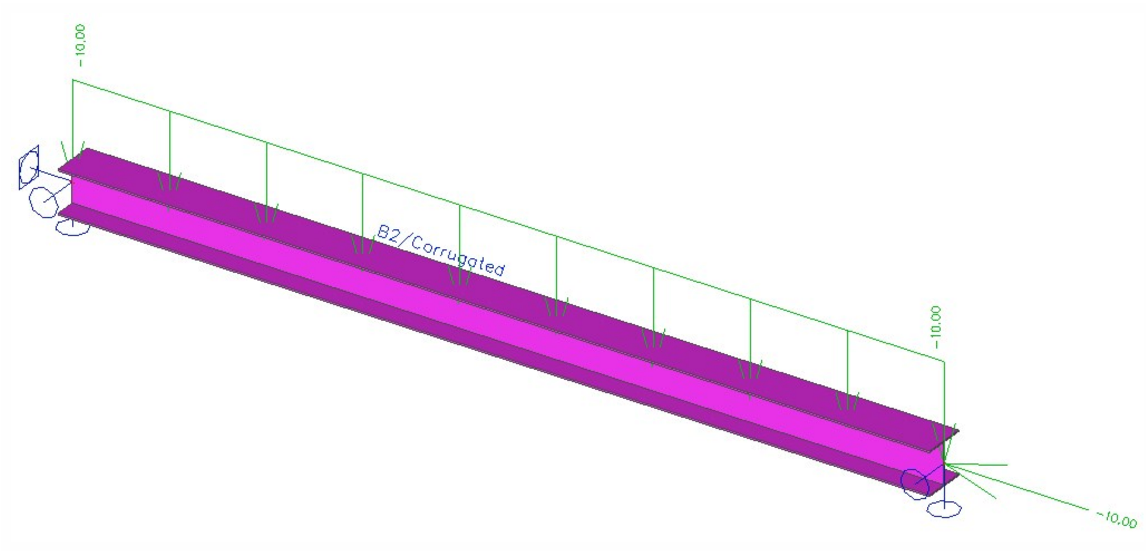
When the user clicks on action button Autodesign then the following dialogue appears. The dialogue is a bit different from the one used in the general Autodesign dialogue but the functionality is the same.



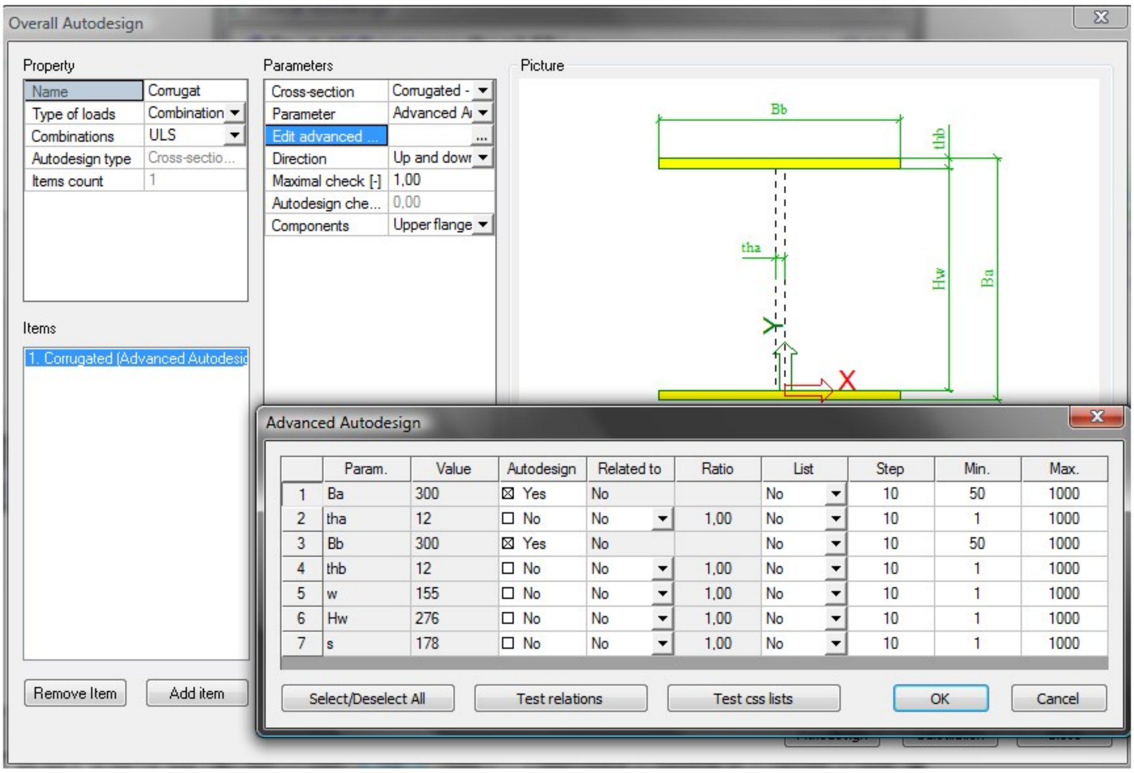
We will focus on Autodesign running from the Autodesign service in this case. Generally, results based on the settings in Autodesign and in the individual service have to be the same.

Illustrative example

Let us consider a very simple example of a steel beam with corrugated cross-section for Autodesign. The structure is subject to the uniform load and to the axial force at one end. The aim of this example is to find the optimal dimensions of the corrugated SIN cross-section.

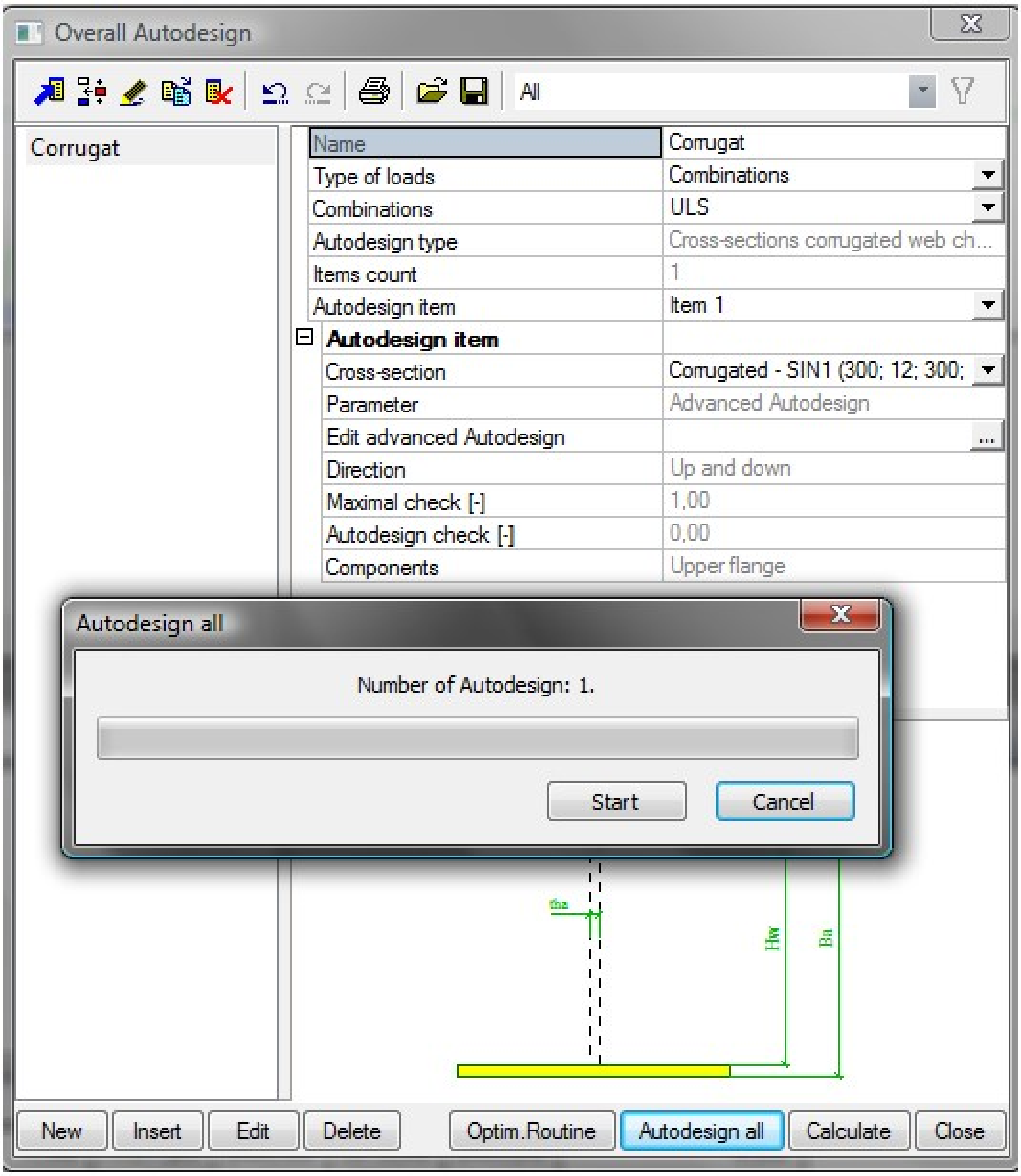


The Autodesign function is defined for this cross-section. You can see the settings on the following figure.

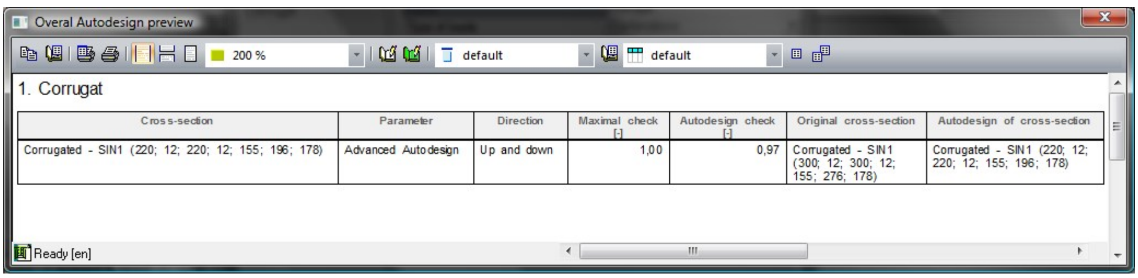


In this case the Advanced Autodesign is selected. The settings are made according to the figure above. The properties are similar to the standard steel code check.

Autodesign can be run in one step using Autodesign all.

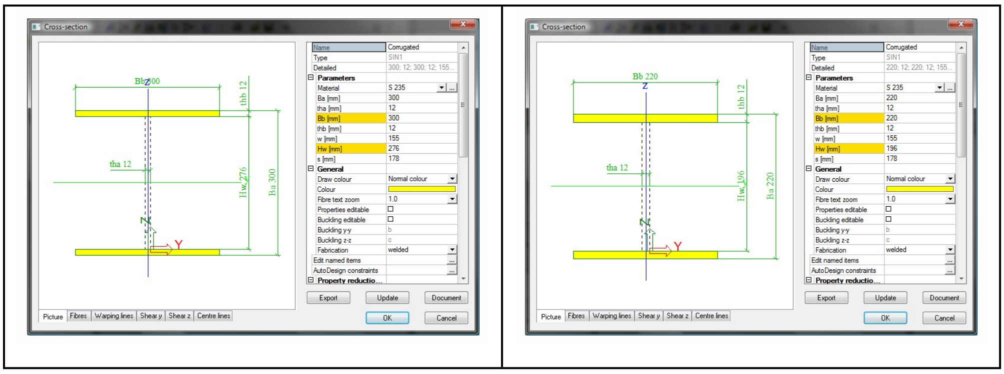


The results are automatically printed in preview.

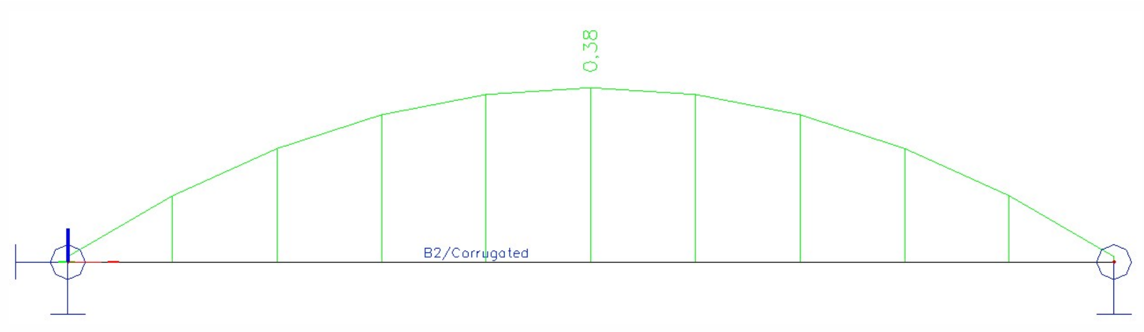


The comparison of the initial and optimized values of the corrugated cross-section is clear from the following table.

Initial	Optimized



Evaluation of the SIN beam unity check along the beam is compared in the following figures. The first figure together with the table is for the initial cross-section.



SIN beam check

Linear calculation, Extreme : Global
Selection : All
Combinations : ULS
SIN beam check
SIN beam checks - outer flange

Member name	Dx [m]	Load case	N,Ed [kN]	My,Ed [kNm]	Vz,Ed [kN]	c top [m]	kc top [-]	Ned - [kN]	Ny [kN]	Ngl [kN]	Nl [kN]	Max unity check [-]
B2	0,000	ULS/1	-13,50	0,00	40,50	6,000	1,00	-6,75	846,00	573,38	846,00	0,01
B2	3,000	ULS/1	-13,50	60,75	0,00	6,000	1,00	-217,69	846,00	573,38	846,00	0,38

SIN beam checks - inner flange

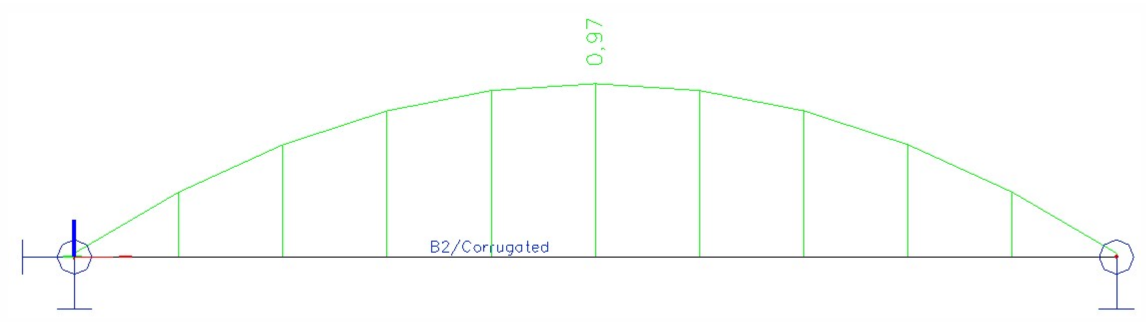
Member name	Dx [m]	Load case	N,Ed [kN]	My,Ed [kNm]	Vz,Ed [kN]	c bot [m]	kc bot [-]	Ned + [kN]	Ned - [kN]	Ny [kN]	Ngl [kN]	Nl [kN]	Max unity check [-]
B2	0,000	ULS/1	-13,50	0,00	40,50	6,000	1,00		-6,75	846,00	573,38	846,00	0,01
B2	3,000	ULS/1	-13,50	60,75	0,00	6,000	1,00	204,19		846,00			0,24

SIN beam checks - web

Member name	Dx [m]	Load case	h [m]	Vz,Ed [kN]	V_web [kN]	Vrd [kN]	Max unity check [-]
B2	0,000	ULS/1	0,288	40,50	40,50	449,36	0,09
B2	3,000	ULS/1	0,288	0,00	0,00	449,36	0,00

SIN beam checks - Summary
Unity Check - OK - satisfies. (0.380)

The second figure together with the table is for the optimized cross-section.



SIN beam check

Linear calculation, Extreme : Global
Selection : All
Combinations : ULS
SIN beam check

SIN beam checks - outer flange

Member name	Dx [m]	Load case	N,Ed [kN]	My,Ed [kNm]	Vz,Ed [kN]	c top [m]	kc top [-]	Ned - [kN]	Ny [kN]	Ngl [kN]	Nl [kN]	Max unity check [-]
B2	0,000	ULS/1	-13,50	0,00	40,50	6,000	1,00	-6,75	620,40	308,35	620,40	0,02
B2	3,000	ULS/1	-13,50	60,75	0,00	6,000	1,00	-298,82	620,40	308,35	620,40	0,97

SIN beam checks - inner flange

Member name	Dx [m]	Load case	N,Ed [kN]	My,Ed [kNm]	Vz,Ed [kN]	c bot [m]	kc bot [-]	Ned + [kN]	Ned - [kN]	Ny [kN]	Ngl [kN]	Nl [kN]	Max unity check [-]
B2	0,000	ULS/1	-13,50	0,00	40,50	6,000	1,00		-6,75	620,40	308,35	620,40	0,02
B2	3,000	ULS/1	-13,50	60,75	0,00	6,000	1,00	285,32		620,40			0,46

SIN beam checks - web

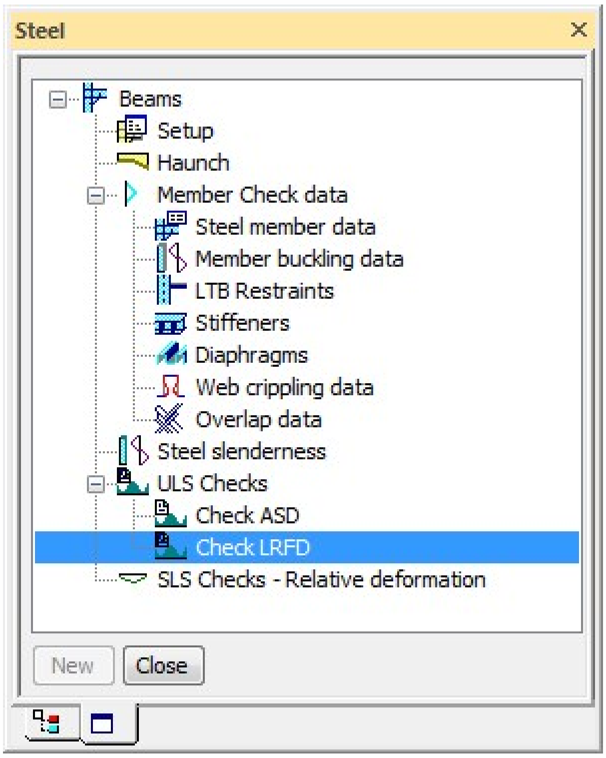
Member name	Dx [m]	Load case	h [m]	Vz,Ed [kN]	V_web [kN]	Vrd [kN]	Max unity check [-]
B2	0,000	ULS/1	0,208	40,50	40,50	319,11	0,13
B2	3,000	ULS/1	0,208	0,00	0,00	319,11	0,00

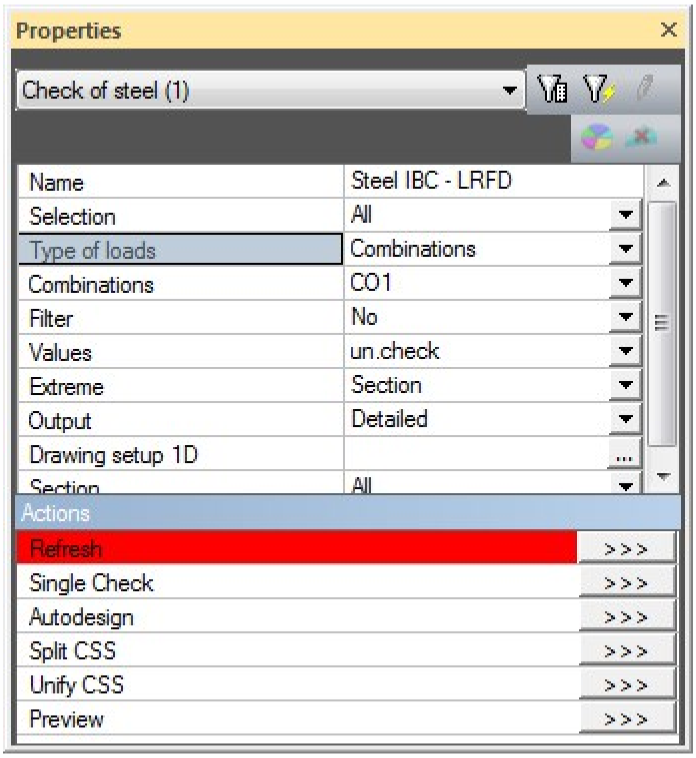
SIN beam checks - Summary
Unity Check - OK - satisfies. (0.969)

Steel - Lapped purlin/girt AutoDesign (only IBC code)

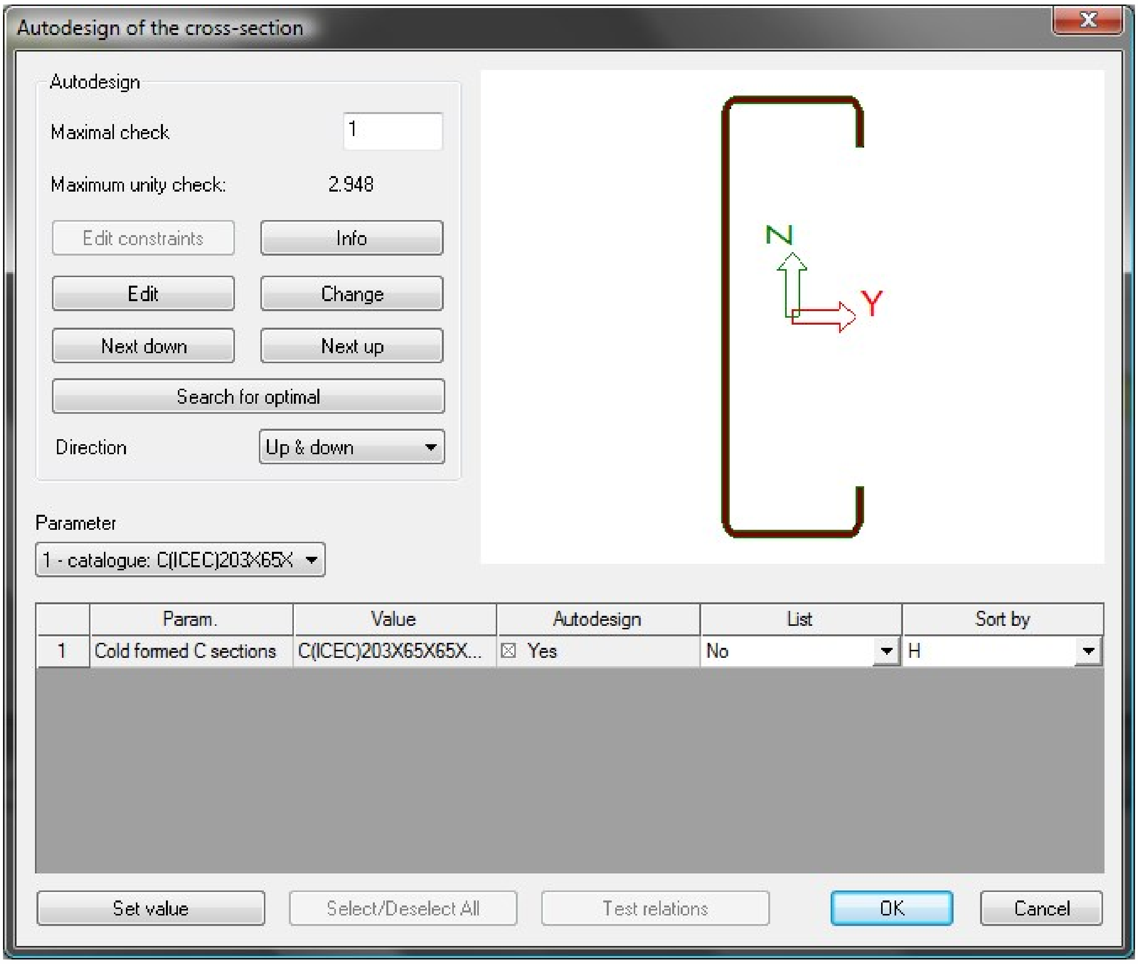
Generally, there is special Autodesign available only for NAS 2007 check for purlins (ASD or LRFD code). The algorithm changes either the cross-section or the length of the overlap. The Autodesign of lapped purlins is the same part as the Autodesign located in the standard Steel > ULS Check > Check LRFD resistance.

Autodesign in Check LRFD service



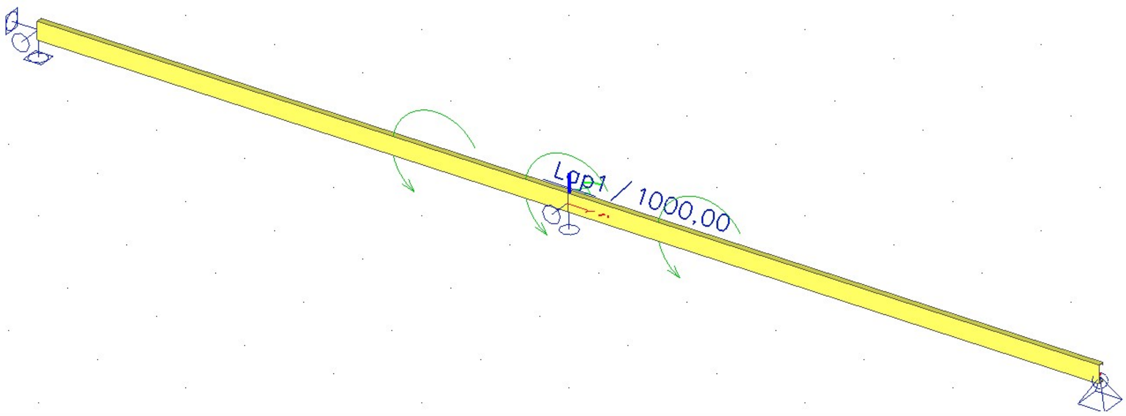


When the user clicks on action button Autodesign then the following dialogue is shown. The dialogue is a bit different from the one used in the general Autodesign dialogue but the functionality is the same.

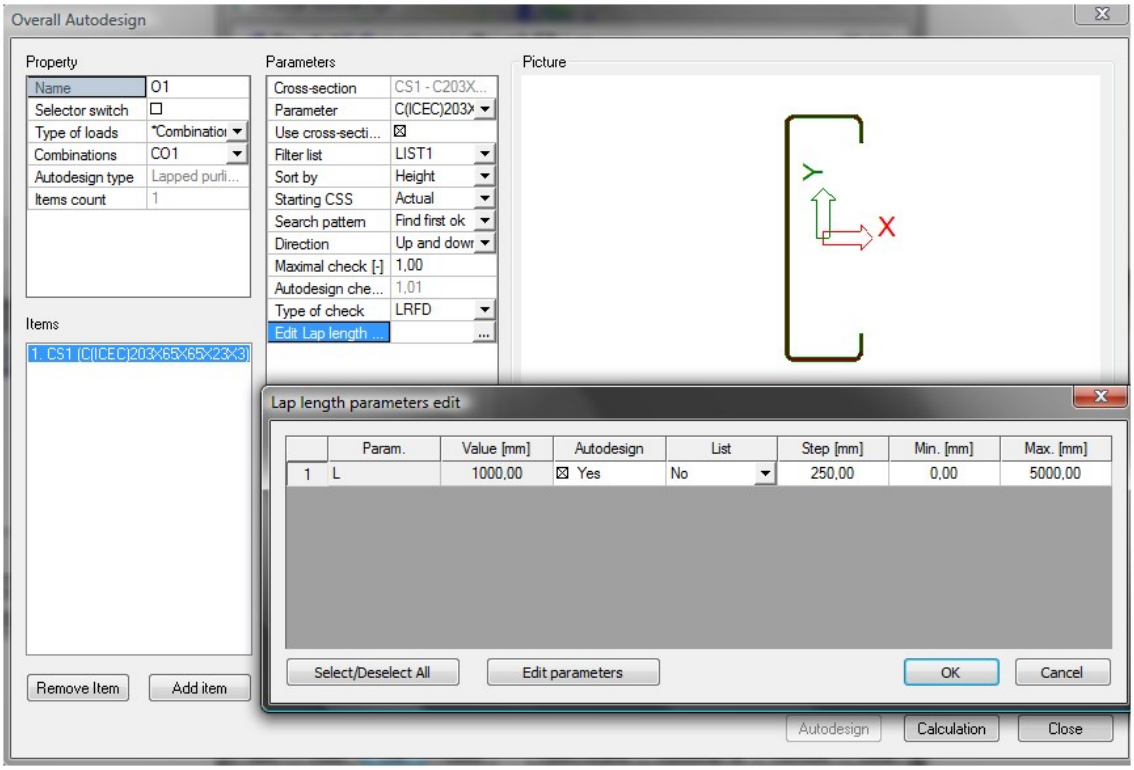


Illustrative example

Let us consider a very simple example of a continuous steel purlin made of a cold formed C section for Autodesign. The purlin overlap should be designed for bending moment 20kNm. The aim of this example is to find the optimal height of the cross-section or length of the overlap.



The Autodesign function is defined for the cold-formed C section. The cross-section is cold formed therefore the properties of Autodesign for rolled and cold formed cross-section are used. You can see the settings in the following figure.

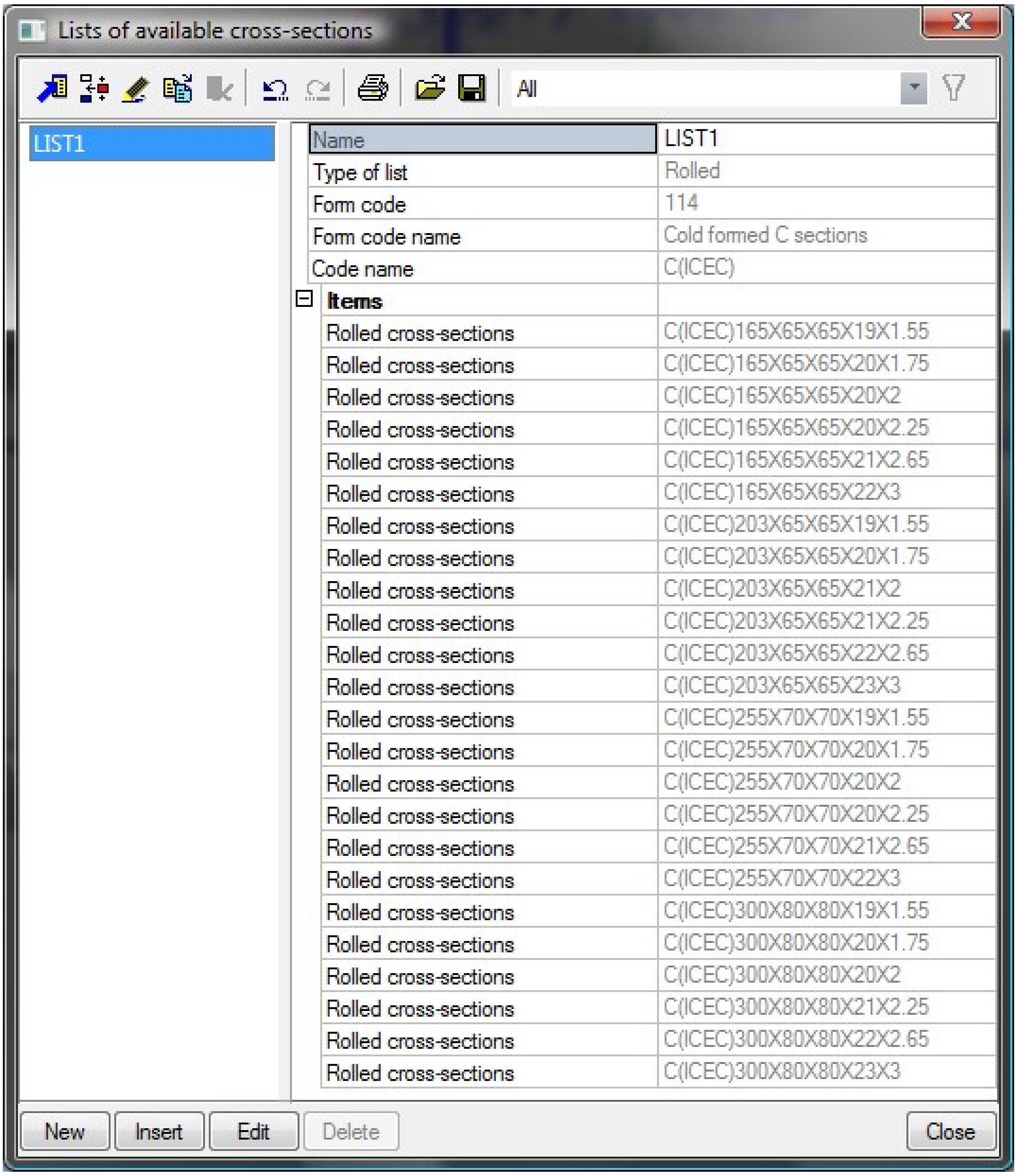


The additional properties are:

Type of check
it is possible to select between ASD code and LRFD code

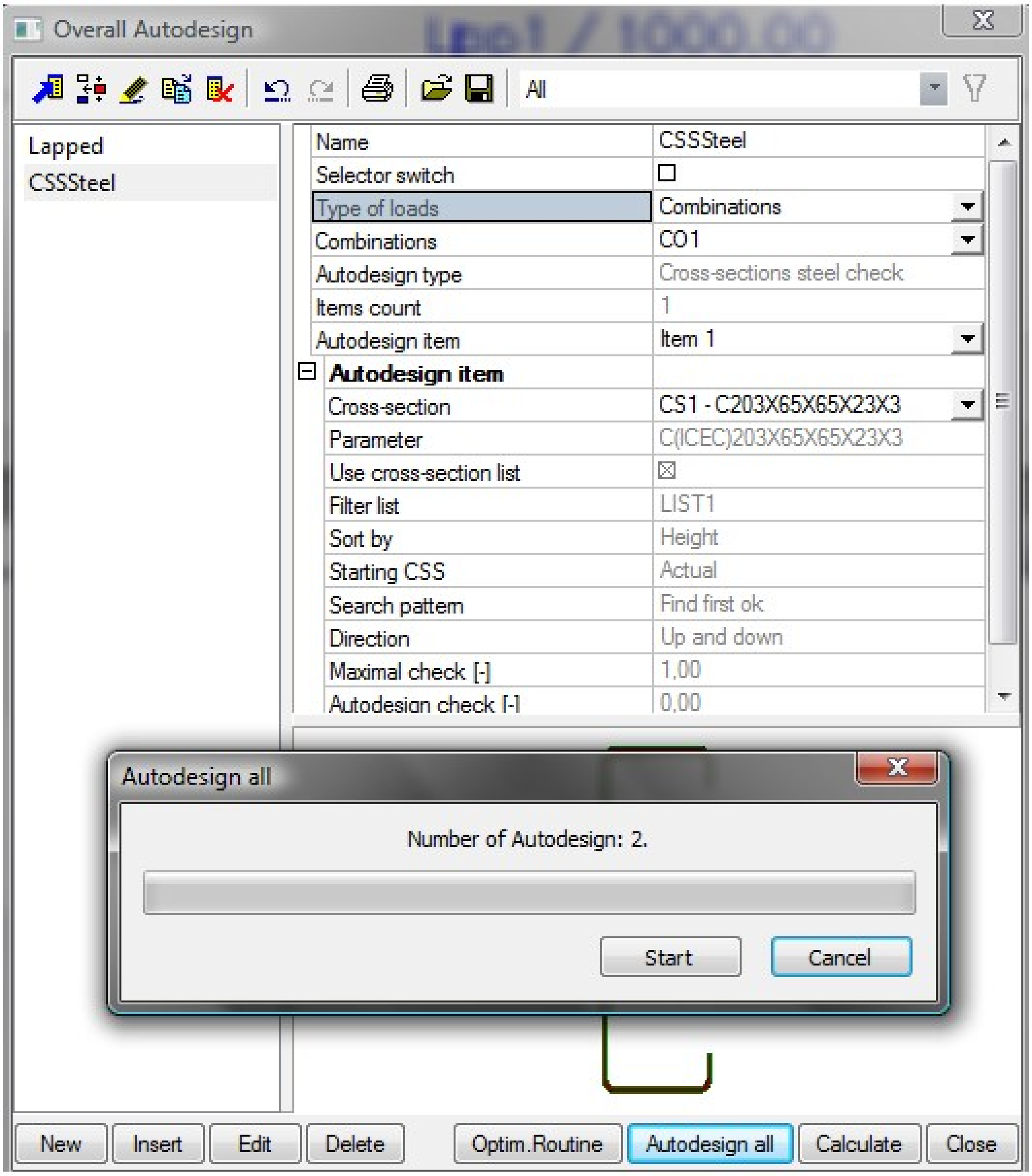
Edit Lap length
Autodesign settings for the overlap

The following cross-sections are inserted to the Cross-section list.



In fact the lapped purlin Autodesign decides between the cross-section change and length change. The cross-section optimization of the whole member is already included. Note that in the example, a fixed list was used to avoid a change in cross-section so specifically the change in length could be tested.

Autodesign of both cross-sections can be run in one step using Autodesign all.



The results are automatically printed in preview.

1. Lapped

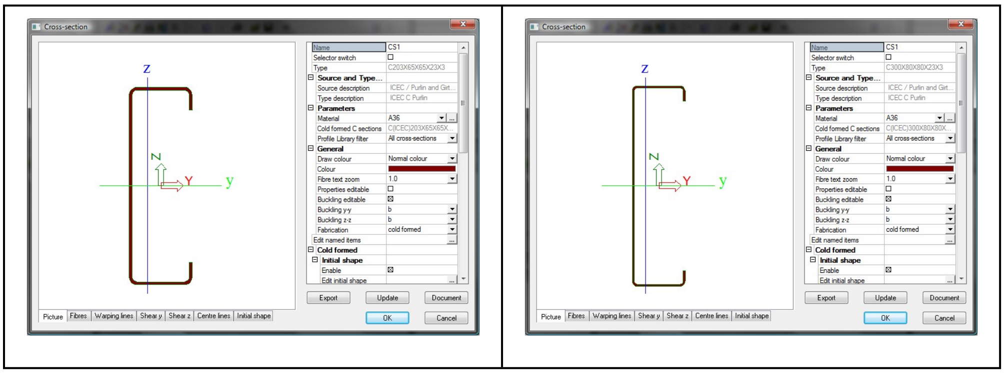
Cross-section	Parameter	Sort by	Filter list	Autodesign check [-]	Type of check
CS1 - C300X80X80X23X3	C(ICEC)300X80X80X23X3	Height	LIST1	0,91	LRFD

2. CSSSteel

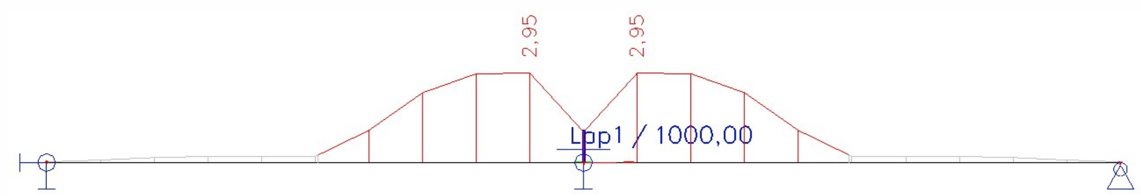
Cross-section	Parameter	Sort by	Filter list	Original cross-section	Autodesign of cross-section	Autodesign check [-]
CS1 - C300X80X80X23X3	C(ICEC)300X80X80X23X3	Height	LIST1	CS1 - C255X70X70X21X2.65	CS1 - C300X80X80X23X3	0,96

The comparison of the initial and optimized values of the cross-section is clear from the following table. The length of the overlap increased from 1000 to 2500mm.

Initial	Optimized



Evaluation of the rolled cross-section unity check along the beam is compared in the following figures. The first figure together with the table is for the initial cross-section.

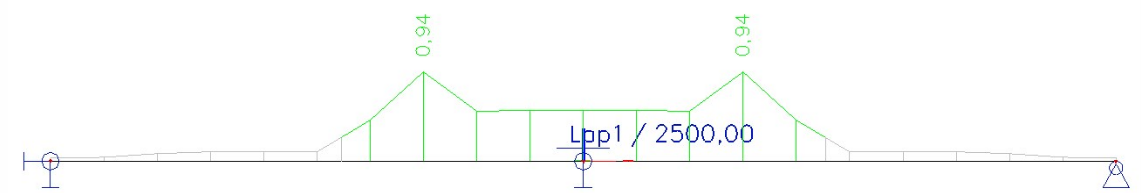


Check of steel

Linear calculation, Extreme : Member
Selection : All
Combinations : CO1

Case	Member	css	mat	dx [m]	un.check [-]
CO1/1	B1	CS1 - C(ICEC)203X65X65X23X3	A36	5,400	**2,95**
CO1/1	B2	CS1 - C(ICEC)203X65X65X23X3	A36	0,600	**2,95**

The second figure together with the table is for the optimized cross-section.



Check of steel

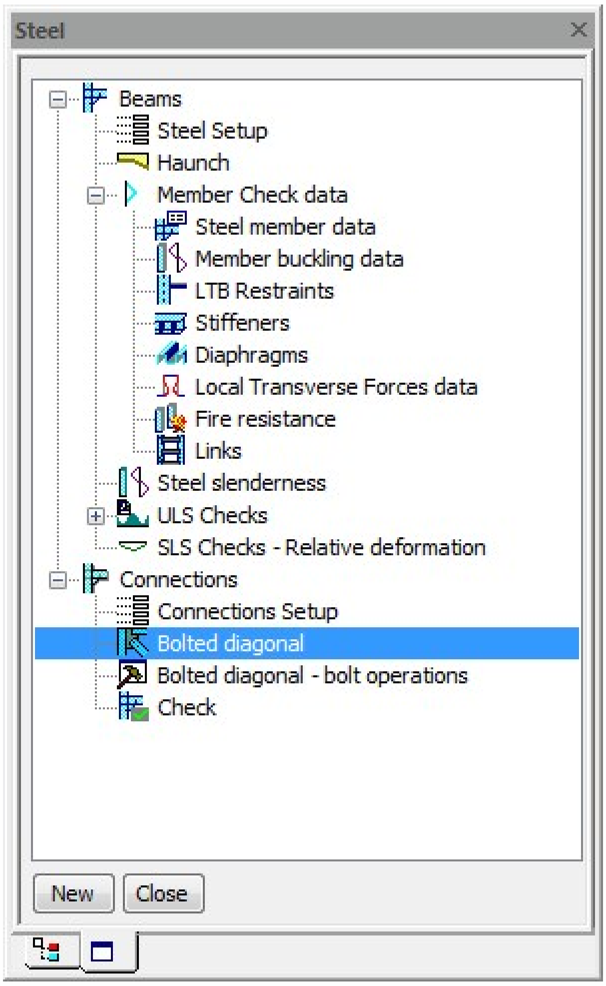
Linear calculation, Extreme : Member
Selection : All
Combinations : CO1

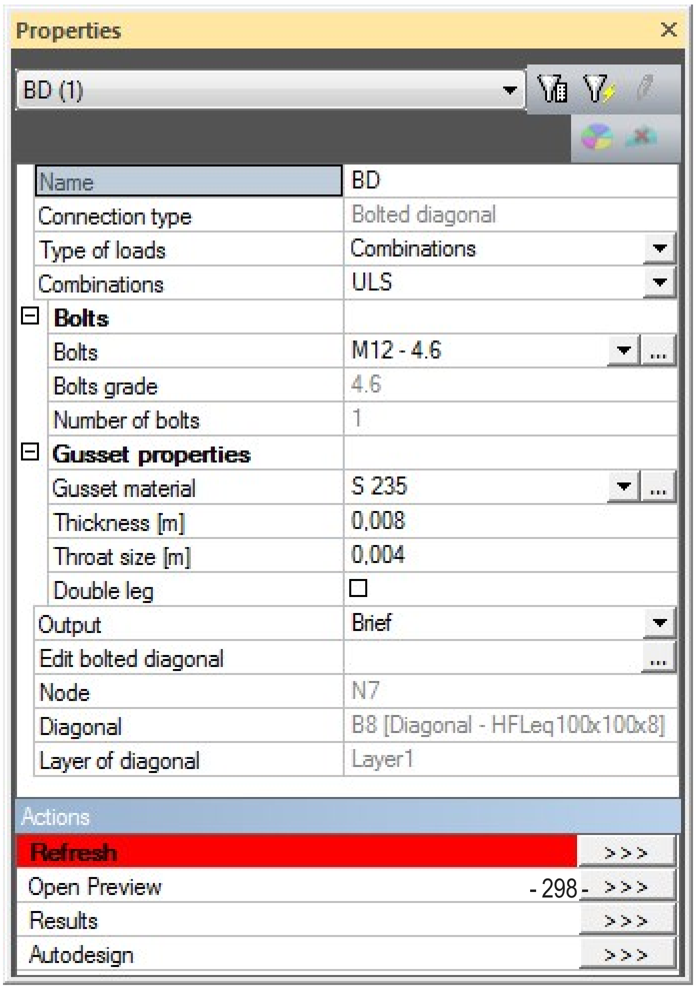
Case	Member	css	mat	dx [m]	un.check [-]
CO1/1	B1	CS1 - C(ICEC)300X80X80X23X3	A36	4,200	0,94
CO1/1	B2	CS1 - C(ICEC)300X80X80X23X3	A36	1,800	0,94

Steel connection - Bolted diagonal AutoDesign

Autodesign of bolted diagonal is the same part as the Autodesign located in the standard Steel > Connections > Bolted diagonal.

Autodesign in Bolted diagonal service



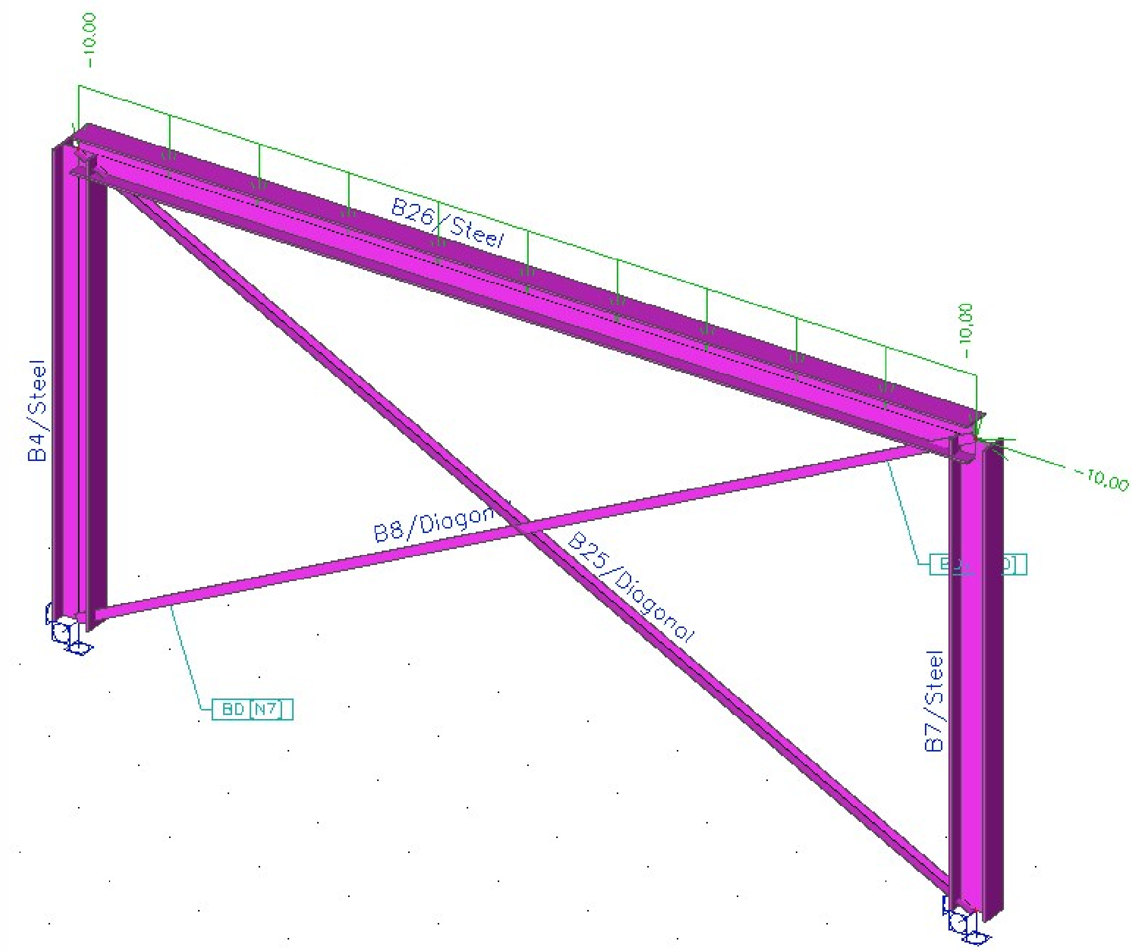


When the user clicks on action button Autodesign then the following dialogue appears. The dialogue is a bit different than the one used in the general Autodesign dialogue but the functionality is the same.

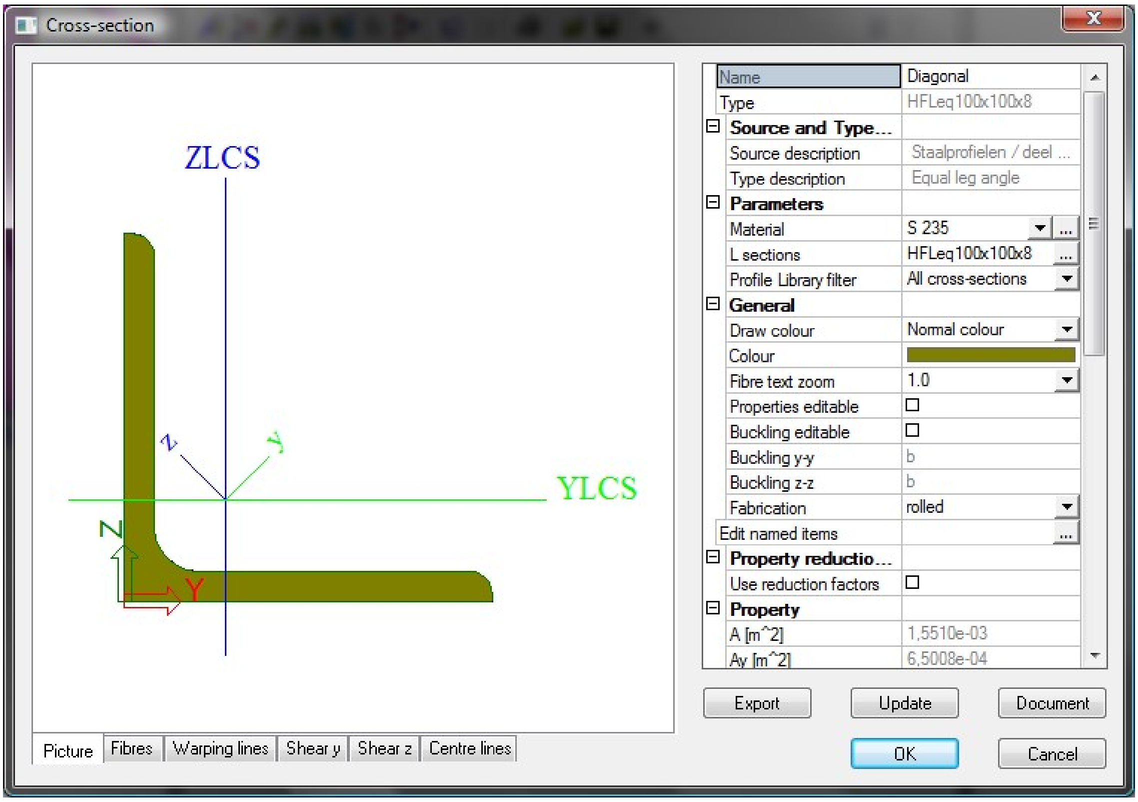
> We will focus on Autodesign running from Autodesign service in this case. Generally, results based on the same settings in Autodesign and in the individual service have to be the same.

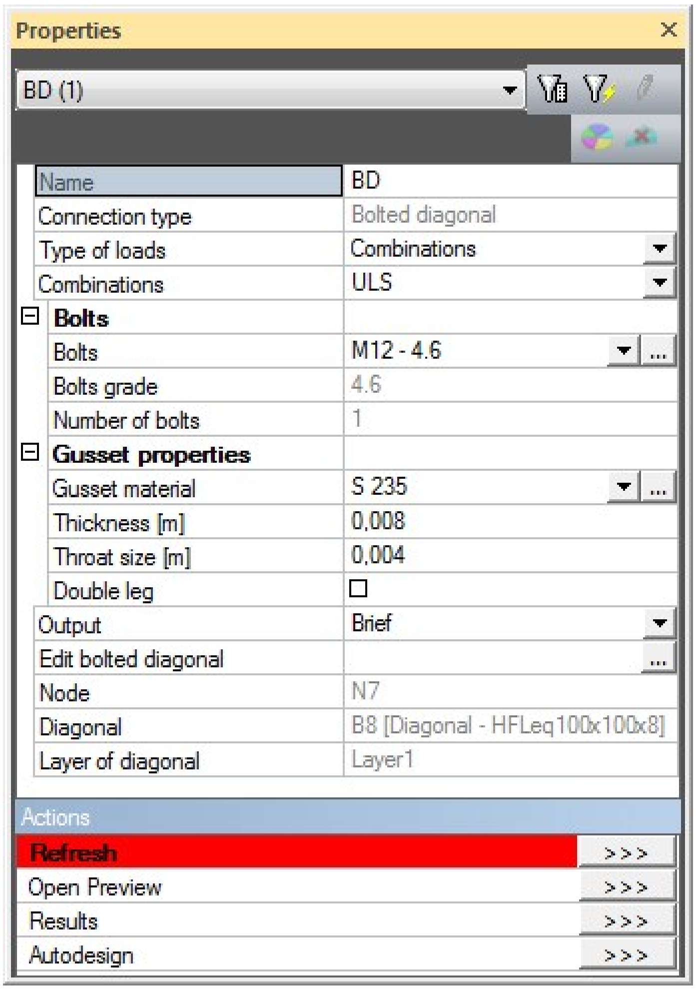
Illustrative example

Let us consider a very simple example of a steel frame with bolted diagonals for Autodesign. The structure is subject to the uniform load and to the axial force at the end. The aim of this example is to find the optimal dimensions of the bolted diagonal connection.

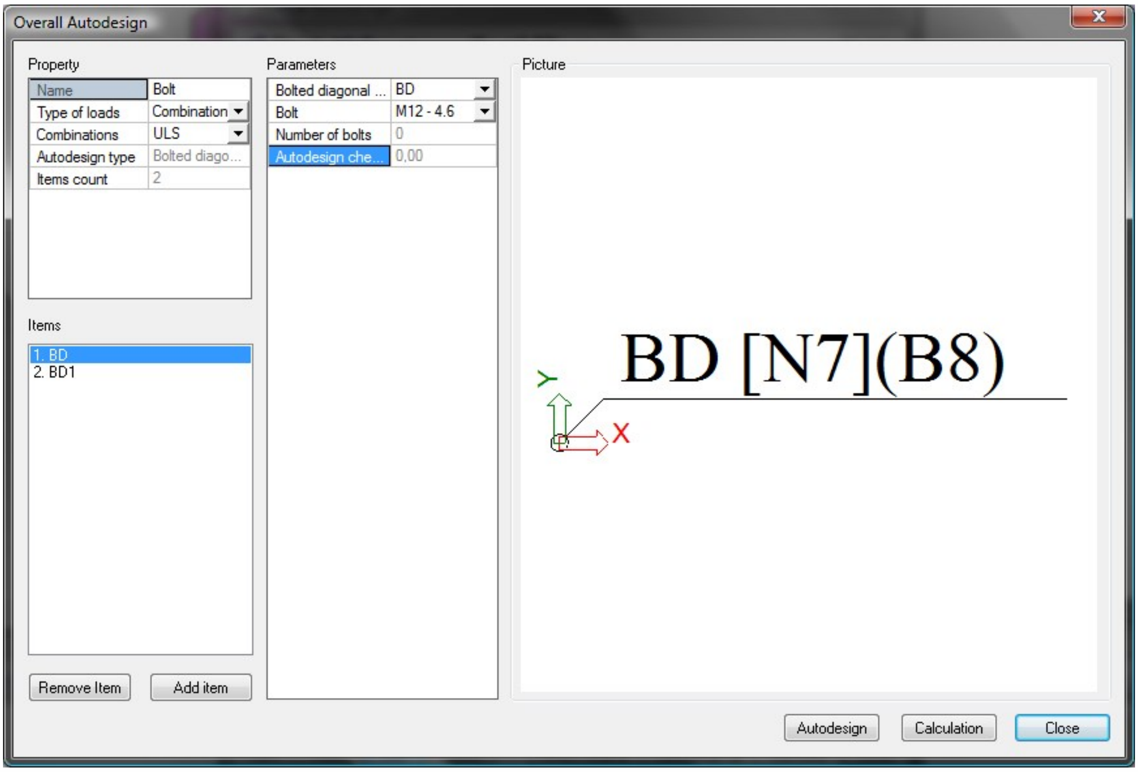


Bolt diagonal has the following cross-section. The parameters of bolted connection are shown in the following figure.


Cross-section
ZLCS
YLCS
Name
Diagonal
Type
HFLeq100x100x8
Source and Type...
Source description
Staalprofielen / deel ...
Type description
Equal leg angle
Parameters
Material
S 235
L sections
HFLeq100x100x8
Profile Library filter
All cross-sections
General
Draw colour
Normal colour
Colour
Fibre text zoom
1.0
Properties editable
Buckling editable
Buckling y-y
b
Buckling z-z
b
Fabrication
rolled
Edit named items
Property reductio...
Use reduction factors
Property
A [m^2]
1,5510e-03
Ay [m^2]
6,5008e-04
Export
Update
Document
OK
Cancel
Picture
Fibres
Warping lines
Shear y
Shear z
Centre lines


Properties
BD (1)
Name
BD
Connection type
Bolted diagonal
Type of loads
Combinations
Combinations
ULS
Bolts
Bolts
M12 - 4.6
Bolts grade
4.6
Number of bolts
1
Gusset properties
Gusset material
S 235
Thickness [m]
0,008
Throat size [m]
0,004
Double leg
Output
Brief
Edit bolted diagonal
Node
N7
Diagonal
B8 [Diagonal - HFLeq100x100x8]
Layer of diagonal
Layer1
Actions
Refresh
Open Preview
Results
Autodesign

The Autodesign function is defined for this cross-section. You can see the settings on the following figure.



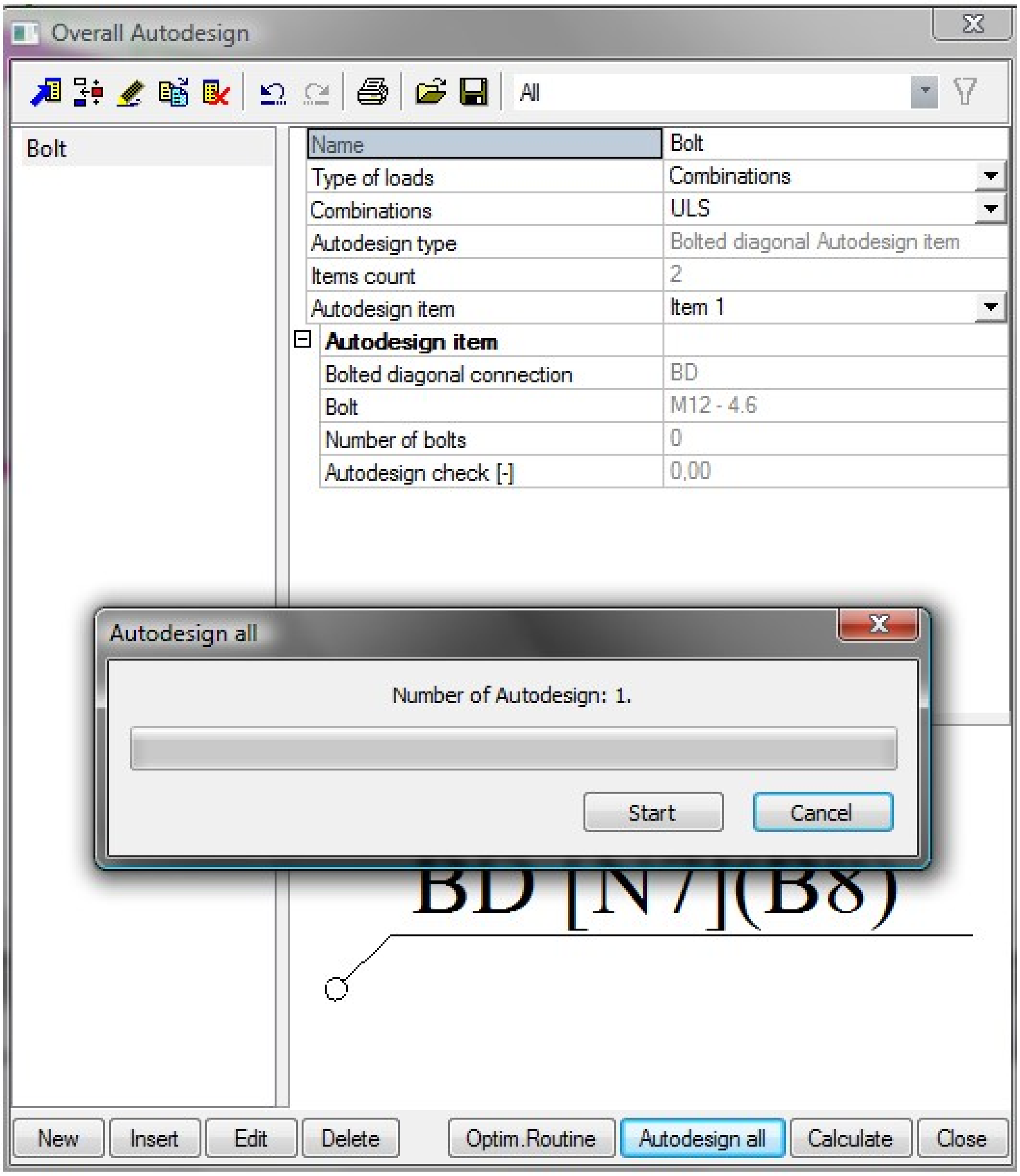
The settings are shown in the figure above.

Bolted diagonal
Specifies the bolted diagonal to be optimised.

Bolt
Specifies the bolt used.

Optimised check (informative)
Shows the unity check for the optimised connection.

Autodesign can be run in one step using Autodesign all.



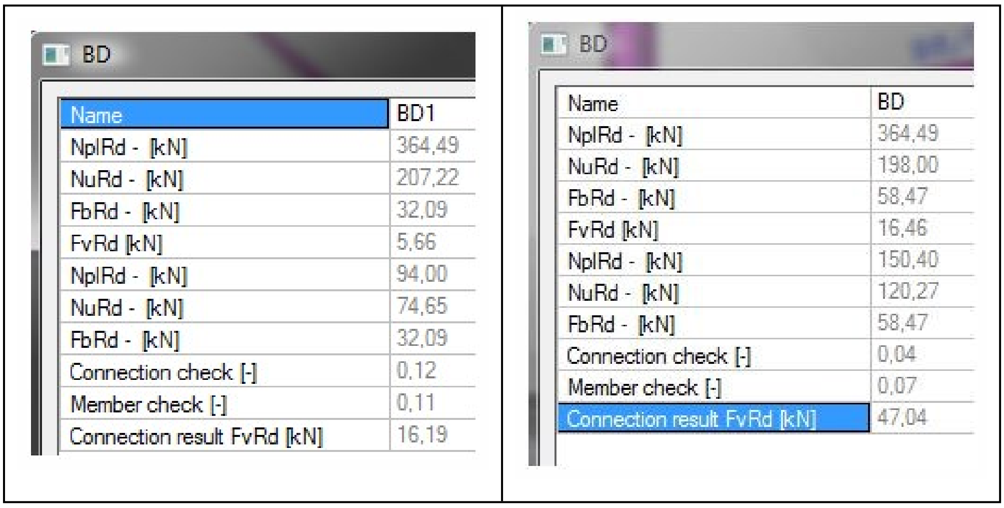
The results are automatically printed in preview. One bolt for each connection is enough. Instead of originally designed four bolts.

1. Bolt

Type Name	Bolted diagonal connection	Bolt	Number of bolts	Autodesign check [-]
Overall Autodesign bolted diagonal item	BD	M12 - 4.6	1	0,49
Overall Autodesign bolted diagonal item	BD1	M12 - 4.6	1	0,49

The comparison of initial and optimized values of bolted connections is clear from the following table.

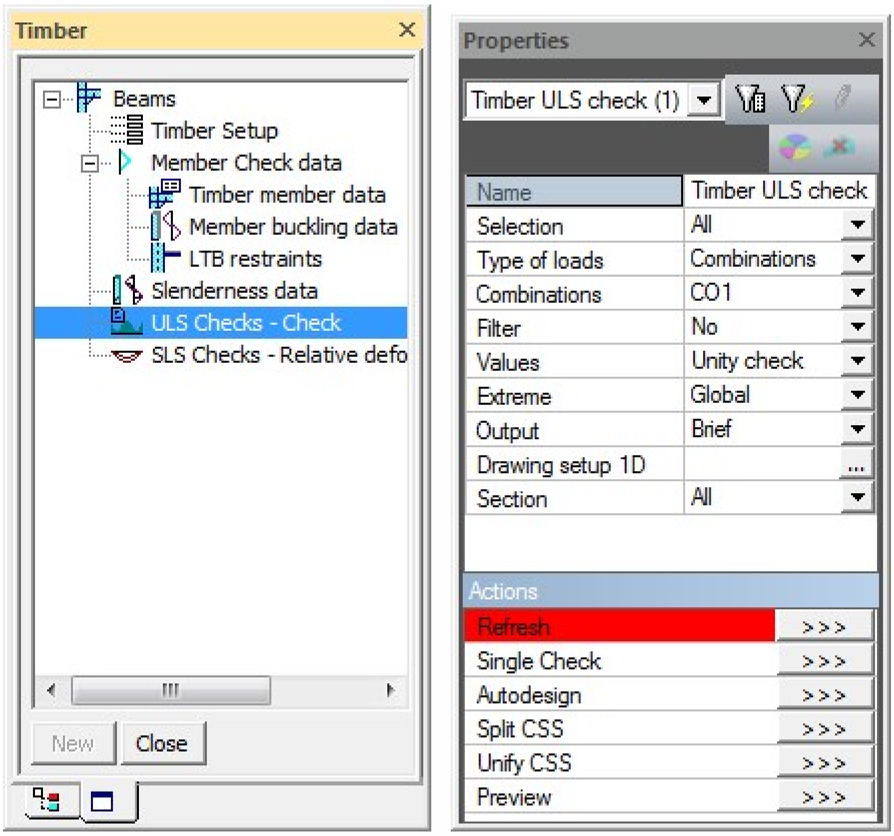
Initial	Optimized



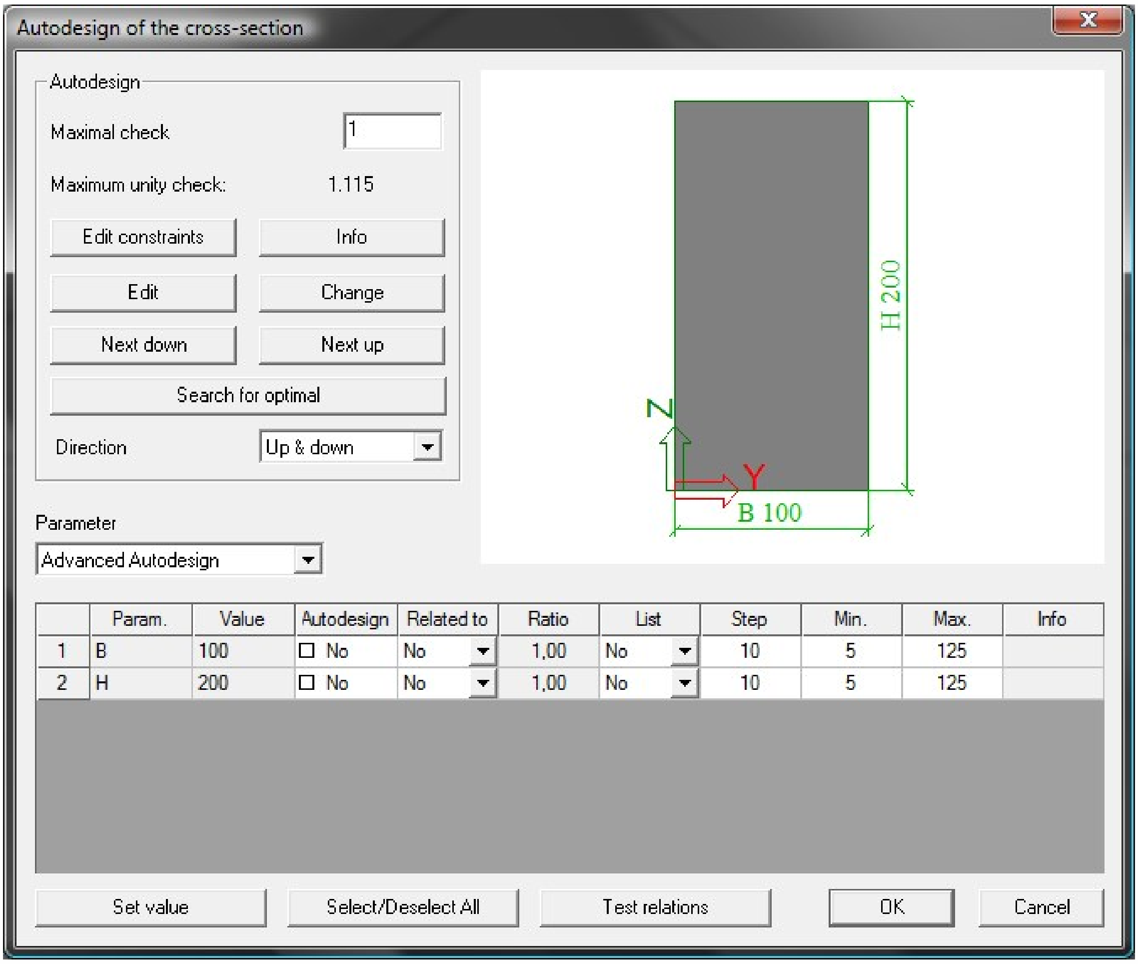
Timber – Cross-section AutoDesign

Autodesign of cross-section timber check is the same part as the Autodesign placed in the standard Timber > ULS Checks - Check.

Autodesign in Timber service



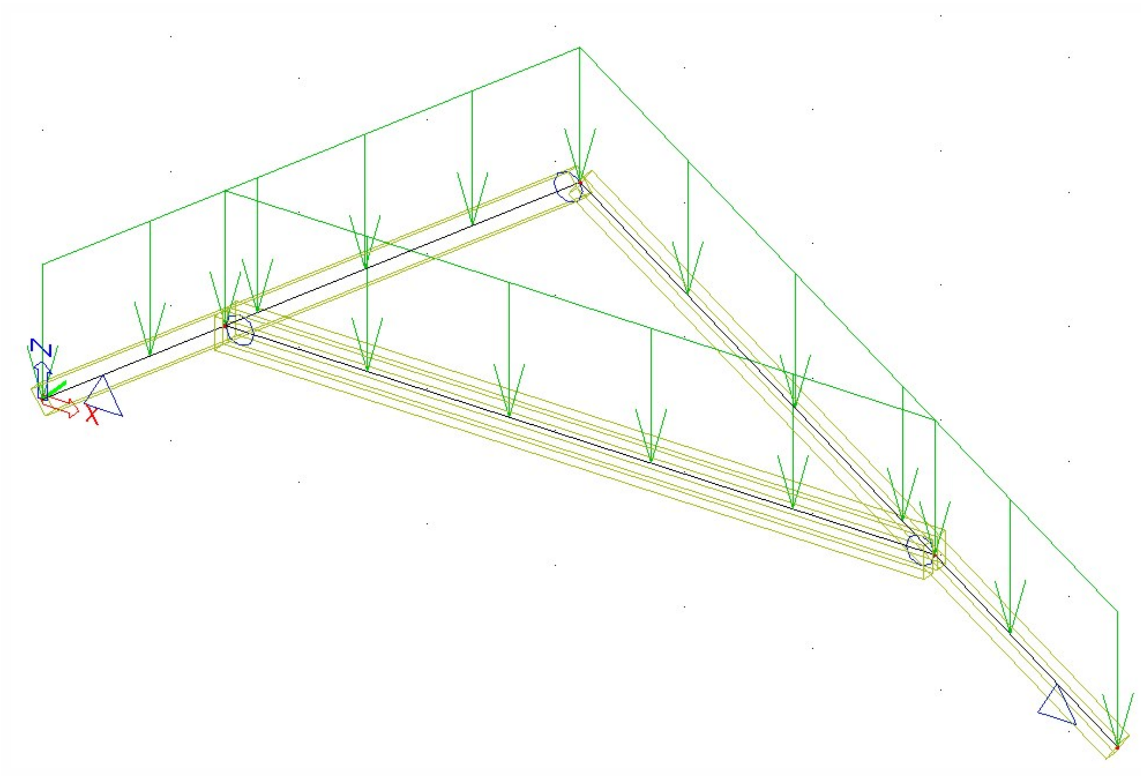
When the user clicks on action button Autodesign the following dialogue is displayed. The dialogue is a bit different from the one used in the general Autodesign dialogue but the functionality is the same.



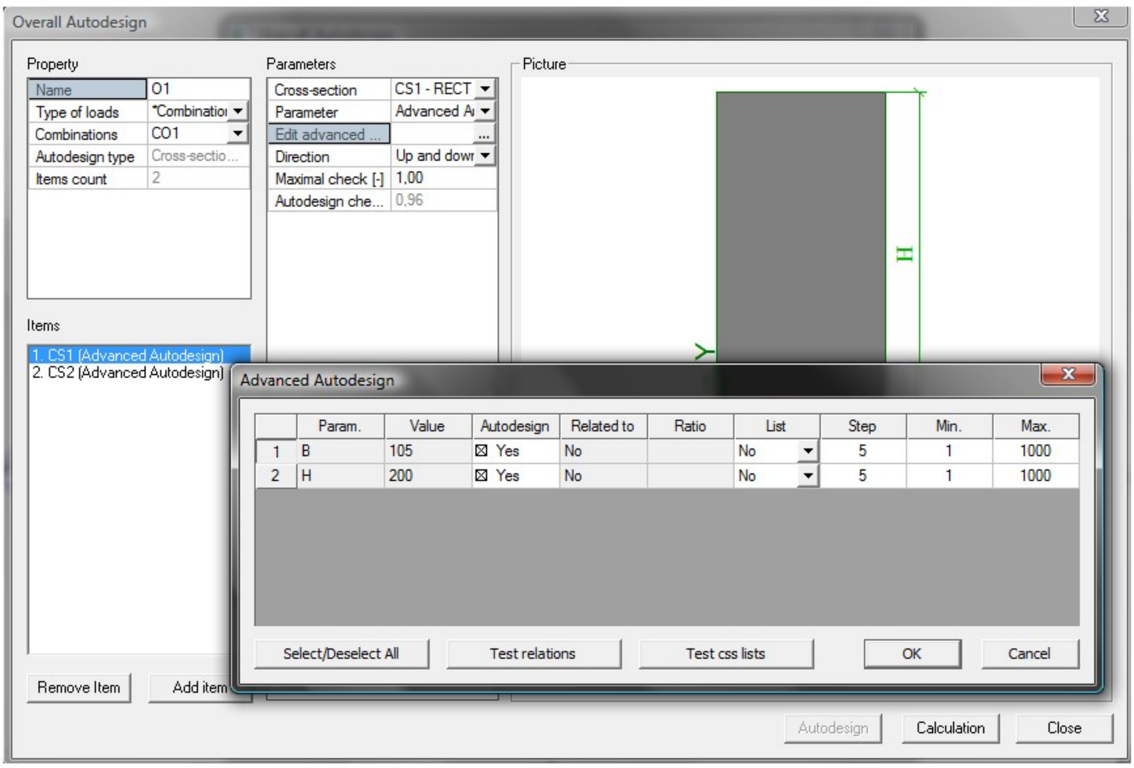
We will focus on Autodesign running from Autodesign service in this case. Generally results based on the same settings in Autodesign and the individual service have to be the same.

Illustrative example

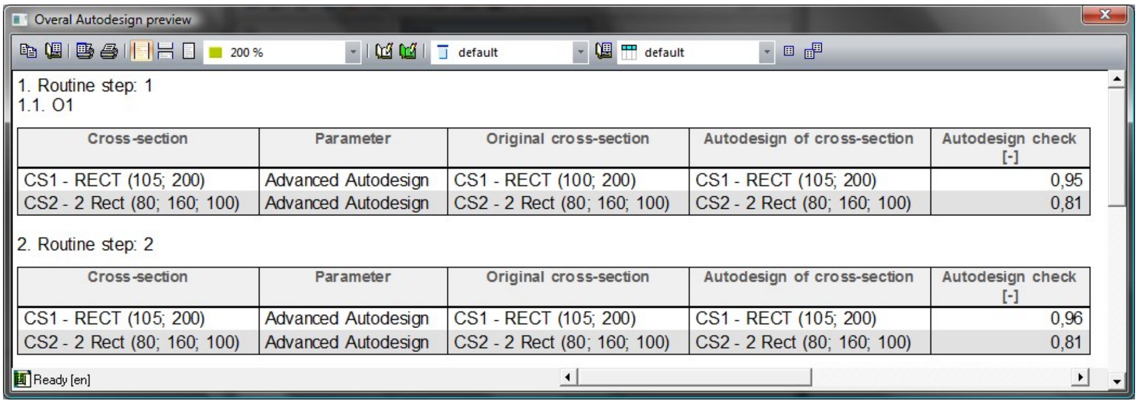
Let us consider a very simple example of timber roof structure with rectangular cross-sections for Autodesign. The structure is subject to self-weight, uniform permanent load, wind and snow load. The aim of this example is to find the optimal dimensions of cross-sections.



The Autodesign function is defined for each cross-section. You can see the settings for CS1.The settings are completely the same as for the steel check. The advanced Autodesign is used for both cross-sections.



Autodesign of both cross-sections can be run in one step using Autodesign all.


Overal Autodesign preview

1. Routine step: 1
1.1. O1

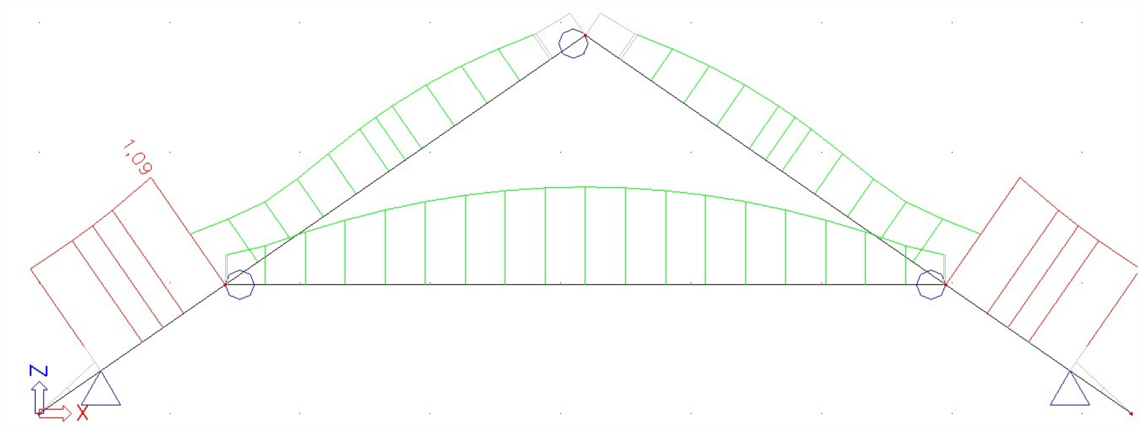
Cross-section	Parameter	Original cross-section	Autodesign of cross-section	Autodesign check [-]
CS1 - RECT (105; 200)	Advanced Autodesign	CS1 - RECT (100; 200)	CS1 - RECT (105; 200)	0,95
CS2 - 2 Rect (80; 160; 100)	Advanced Autodesign	CS2 - 2 Rect (80; 160; 100)	CS2 - 2 Rect (80; 160; 100)	0,81

2. Routine step: 2

Cross-section	Parameter	Original cross-section	Autodesign of cross-section	Autodesign check [-]
CS1 - RECT (105; 200)	Advanced Autodesign	CS1 - RECT (105; 200)	CS1 - RECT (105; 200)	0,96
CS2 - 2 Rect (80; 160; 100)	Advanced Autodesign	CS2 - 2 Rect (80; 160; 100)	CS2 - 2 Rect (80; 160; 100)	0,81

Ready [en]

Evaluation of the timber unity check along the beam is compared in the following figures. The first figure and table are for the initial structure.

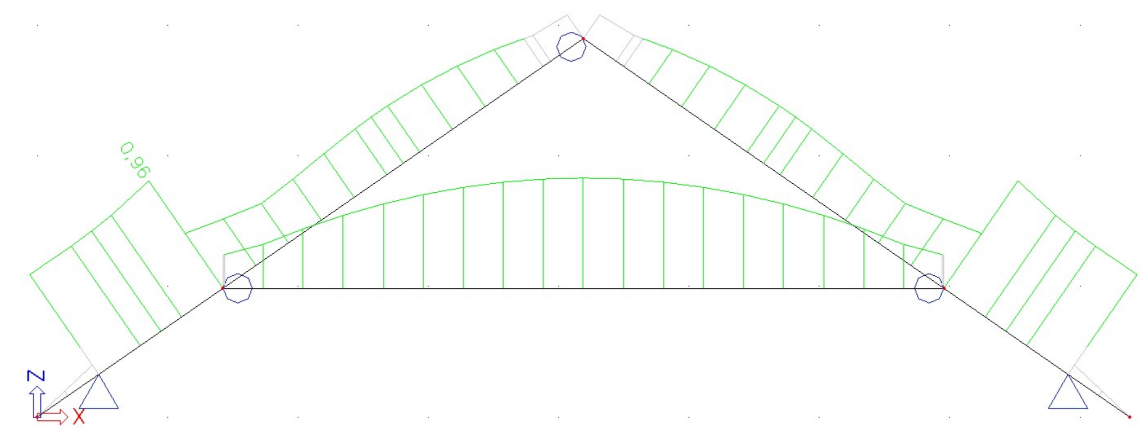


Timber ULS check

Linear calculation, Extreme : Global
Selection : All
Combinations : CO1
Cross-section : CS1 - RECT (100; 200)

Beam	Cross-section	Material	dx [m]	Load case	Unity check [-]	Section check [-]	Stability check [-]
B1	CS1 - RECT	C22	1,734	CO1/1	**1,09**	0,22	1,09

The second ones are for the optimised structure.



Timber ULS check

Linear calculation, Extreme : Global
Selection : All
Combinations : CO1

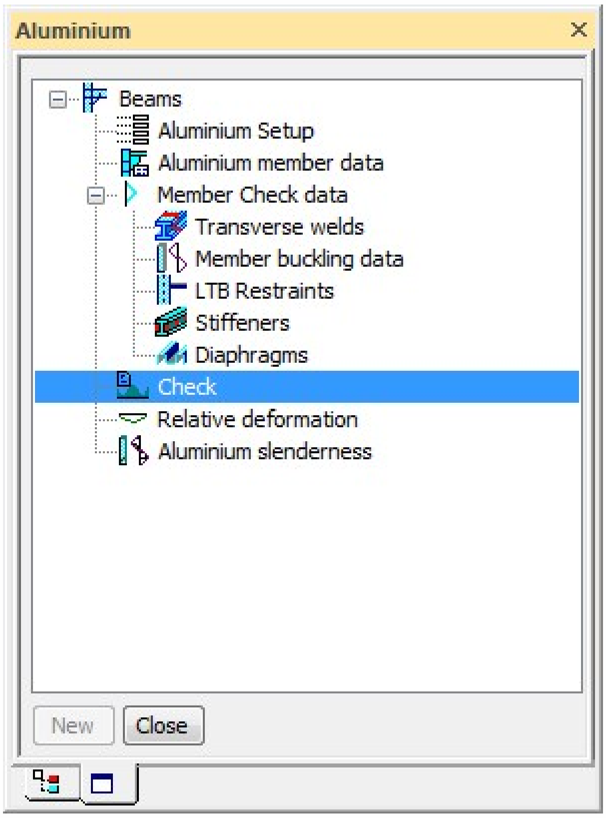
Beam	Cross-section	Material	dx [m]	Load case	Unity check [-]	Section check [-]	Stability check [-]
B1	CS1 - RECT	C22	1,734	CO1/1	**0,96**	0,21	0,96

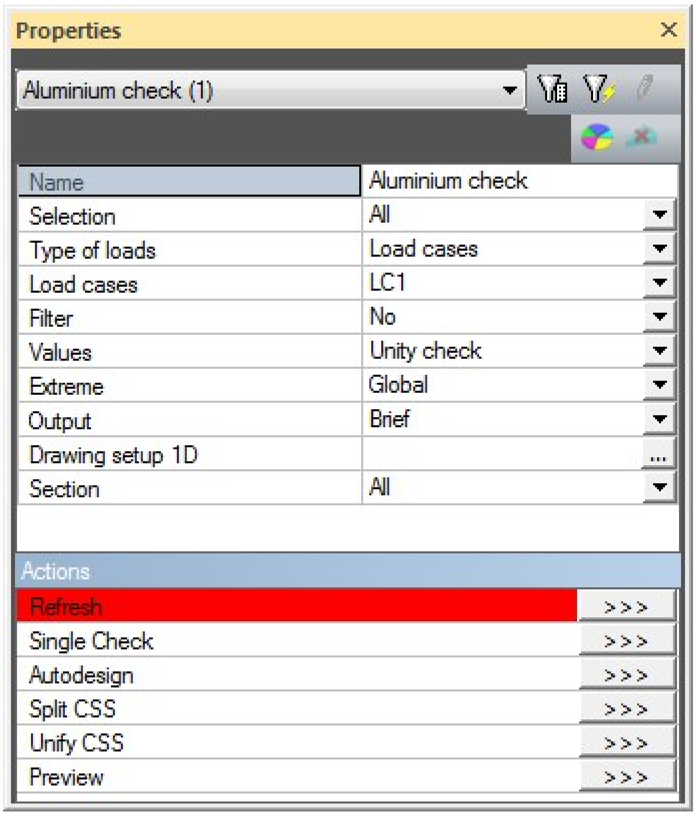
The width of rafters were optimised from 100 to 105mm.

Aluminium – Cross-section AutoDesign

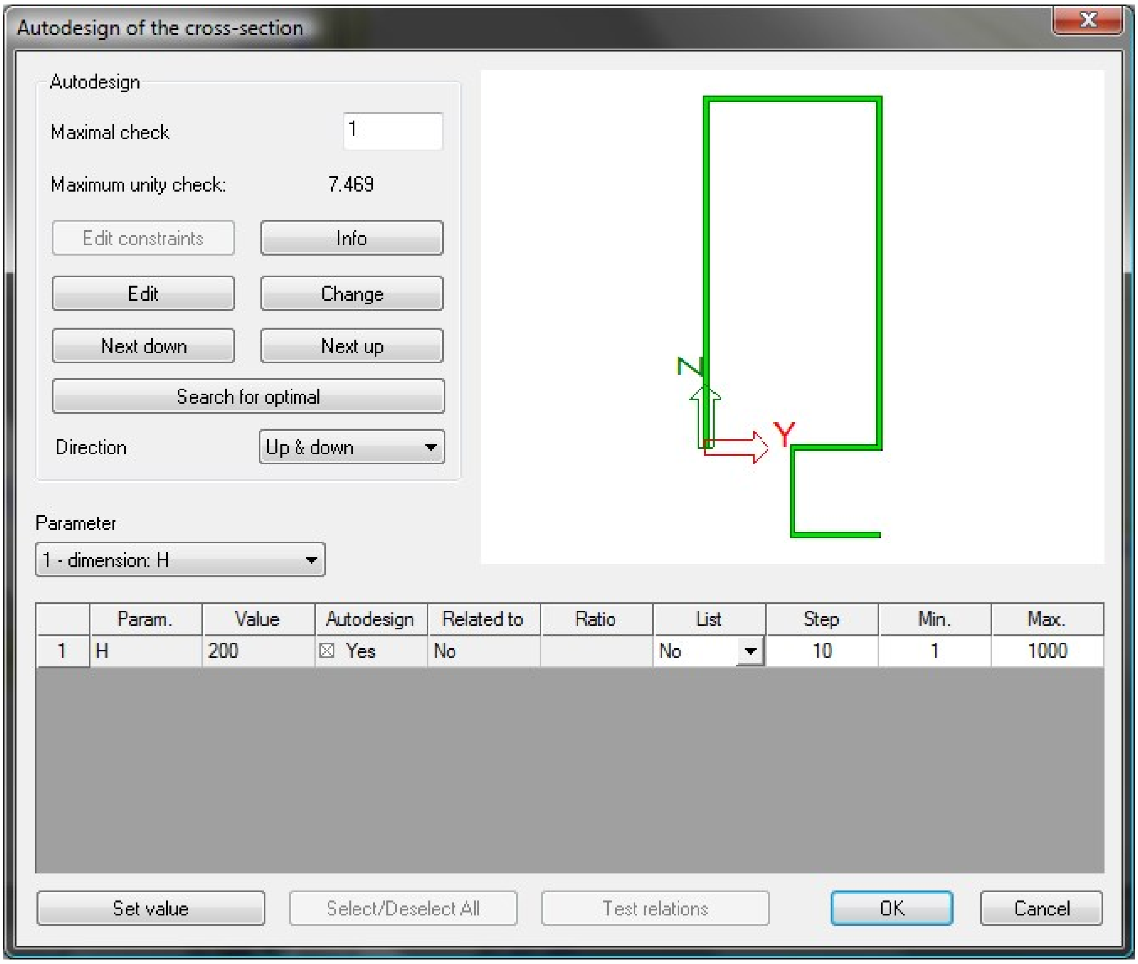
Autodesign of cross-section steel check is the same part as the Autodesign locates in the standard Aluminium > Check.

Autodesign in Aluminium service





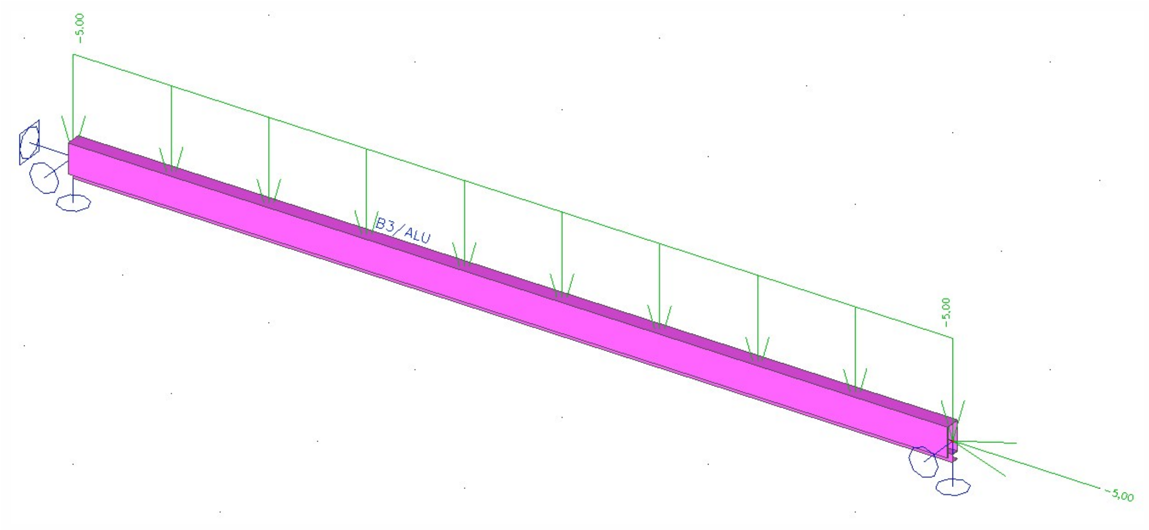
When the user clicks on action button Autodesign then the following dialogue appears. The dialogue is a bit different from the one used in the general Autodesign but the functionality is the same.



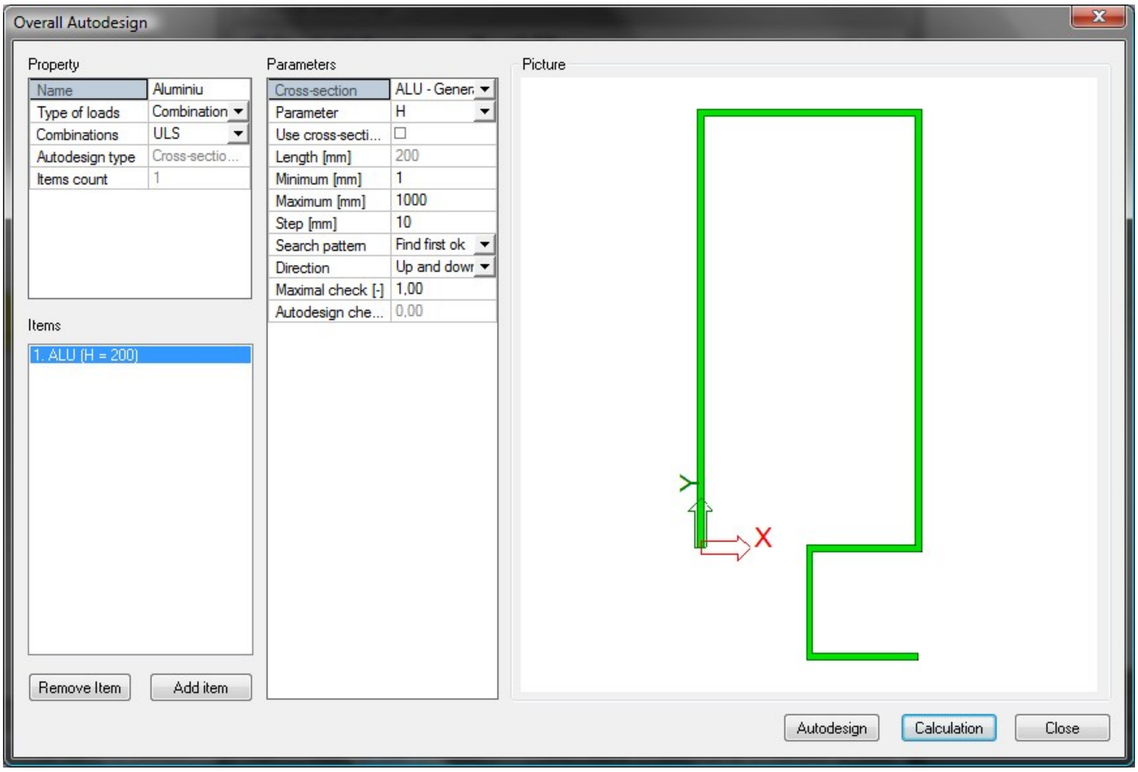
We will focus on Autodesign running from Autodesign service in this case. Generally results based on the same settings in Autodesign and the individual service have to be the same.

Illustrative example

Let us consider a very simple example of an aluminium beam for Autodesign. The structure is subject to the uniform load and to the axial force at the end. The aim of this example is to find the optimal height of aluminium cross-section defined as a general cross-section.

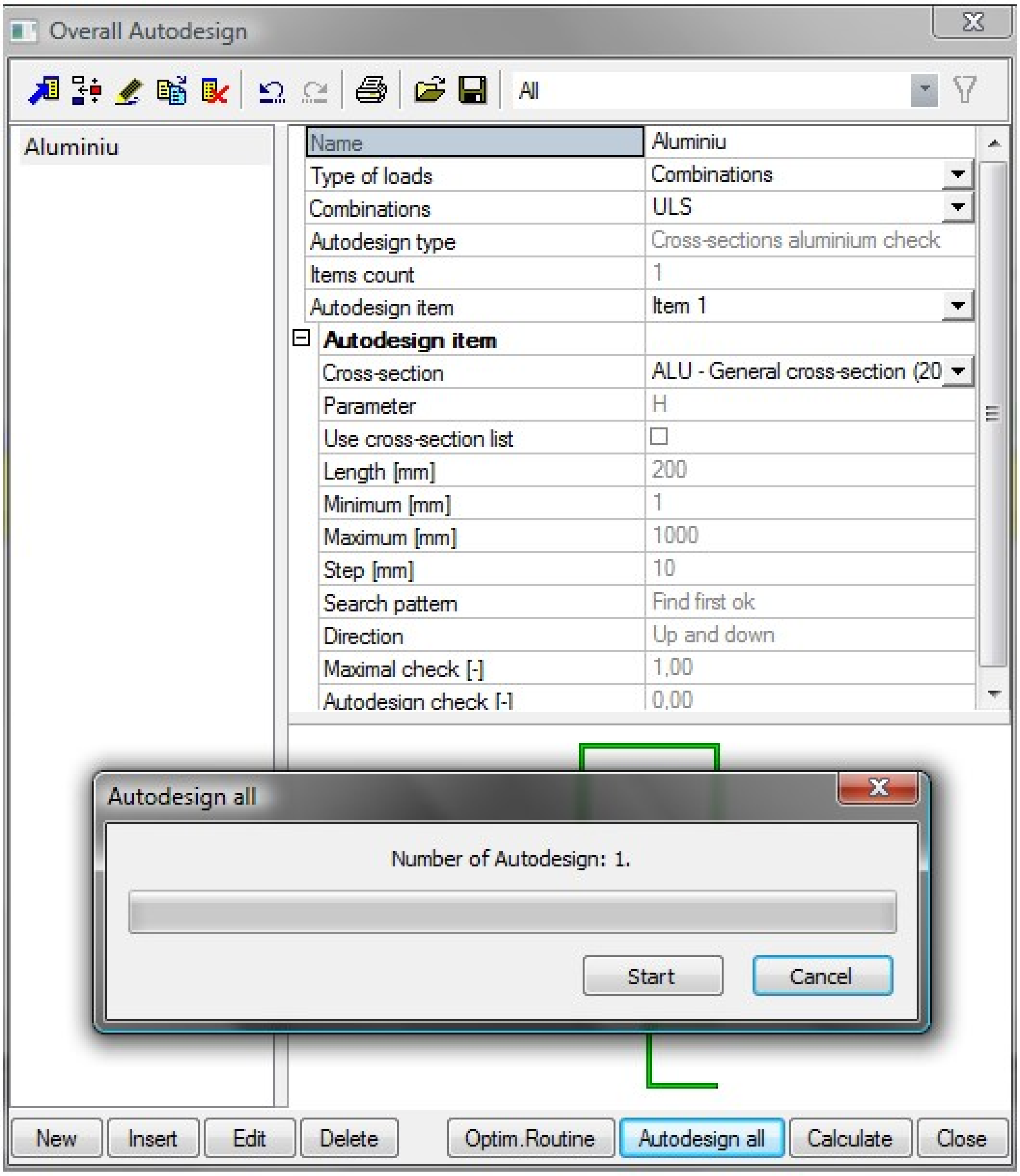


The Autodesign function is defined for this cross-section. You can see the settings in the following figure,

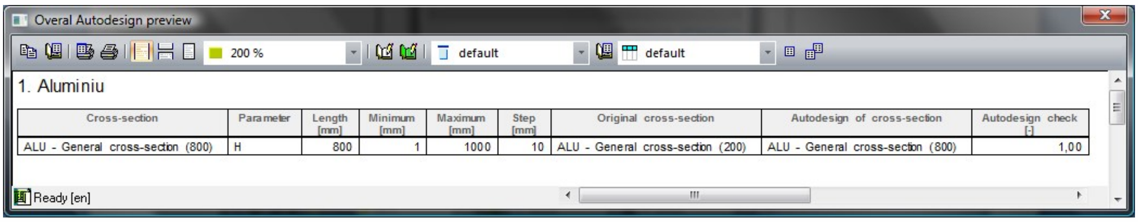


In this case only one parameter (height of cross-section) is available. Therefore advance Autodesign is not used. Other settings are the same as for the standard steel Autodesign.

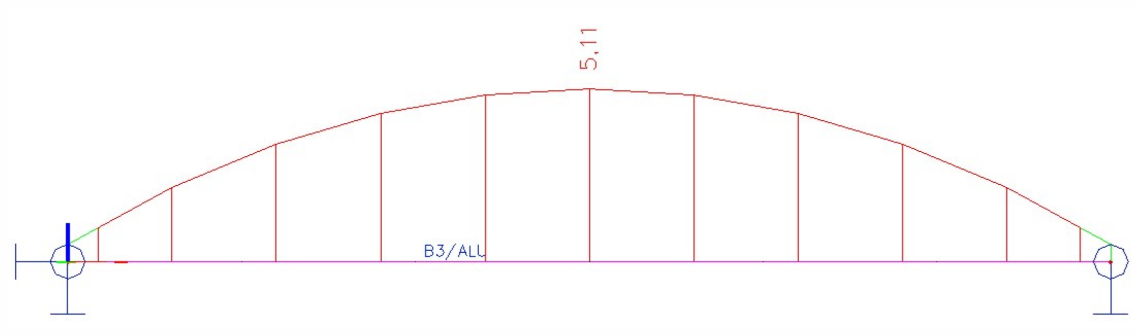
Autodesign can be run in one step using Autodesign all.



The results are automatically printed in preview.



Evaluation of the aluminium unity check along the beam is compared in the following figures. The initial height of the cross-section was 200mm. The optimized value is 800mm. Evaluation of the steel check for the initial cross-section dimensions.

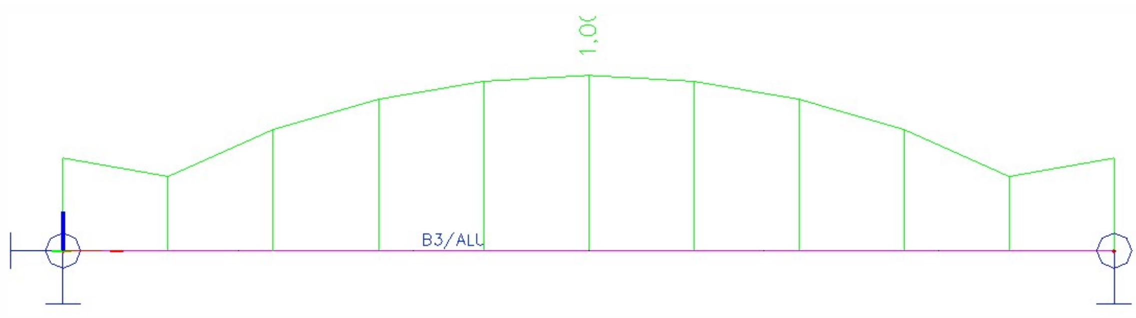


Aluminium check

Linear calculation, Extreme : Global
Selection : All
Combinations : ULS
Cross-section : ALU - General cross-section (200)

Beam	Case	Css	Material	dx [m]	Unity check [-]	Section check [-]	Stability Check [-]
B3	ULS/1	ALU - General cross-section	EN-AW 6005A (EP/O,ER/B) T6 (0-5)	3,000	**5,11**	1,92	5,11

Evaluation of the steel check for the optimized cross-section dimensions.



Aluminium check

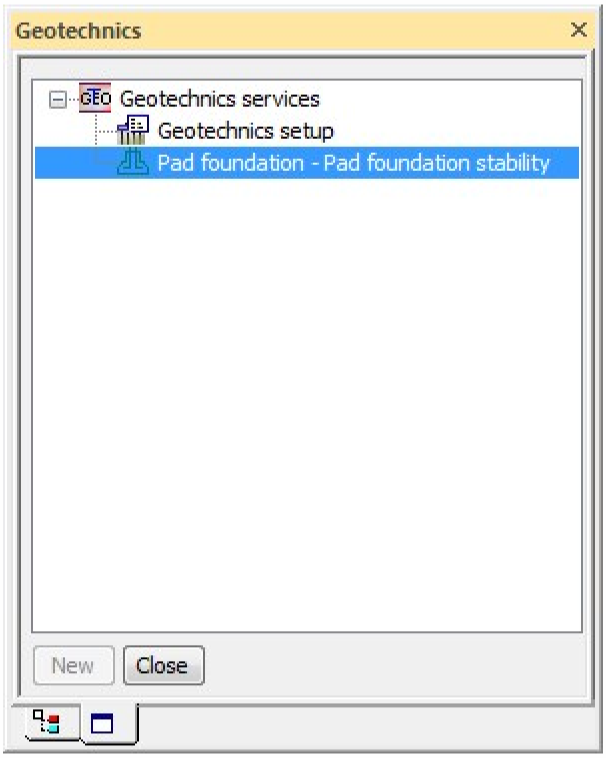
Cross-sections were changed during Autodesign. The structure has to be recalculated !
Linear calculation, Extreme : Global
Selection : All
Combinations : ULS
Cross-section : ALU - General cross-section (800)

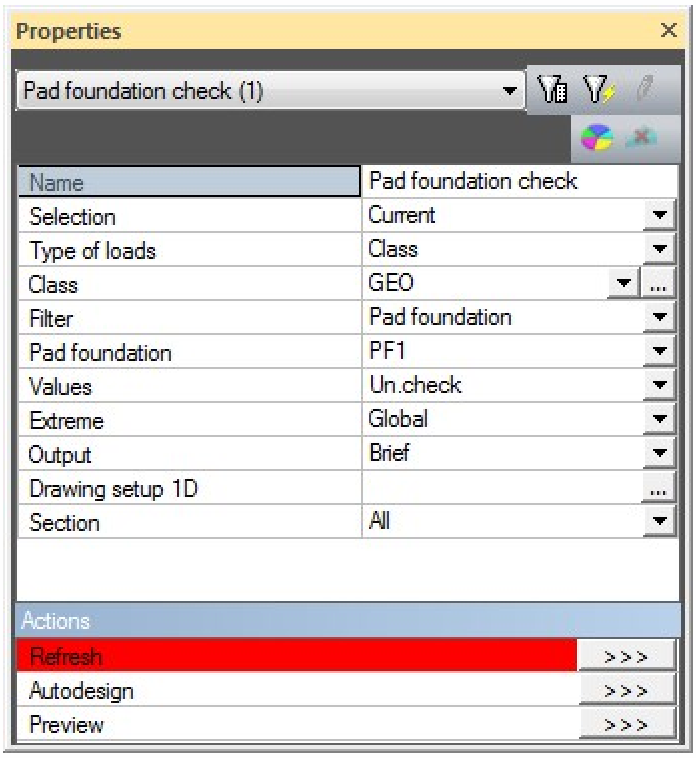
Beam	Case	Css	Material	dx [m]	Unity check [-]	Section check [-]	Stability Check [-]
B3	ULS/1	ALU - General cross-section	EN-AW 6005A (EP/O,ER/B) T6 (0-5)	3,000	**1,00**	0,55	1,00

Geotechnics – Pad foundation AutoDesign

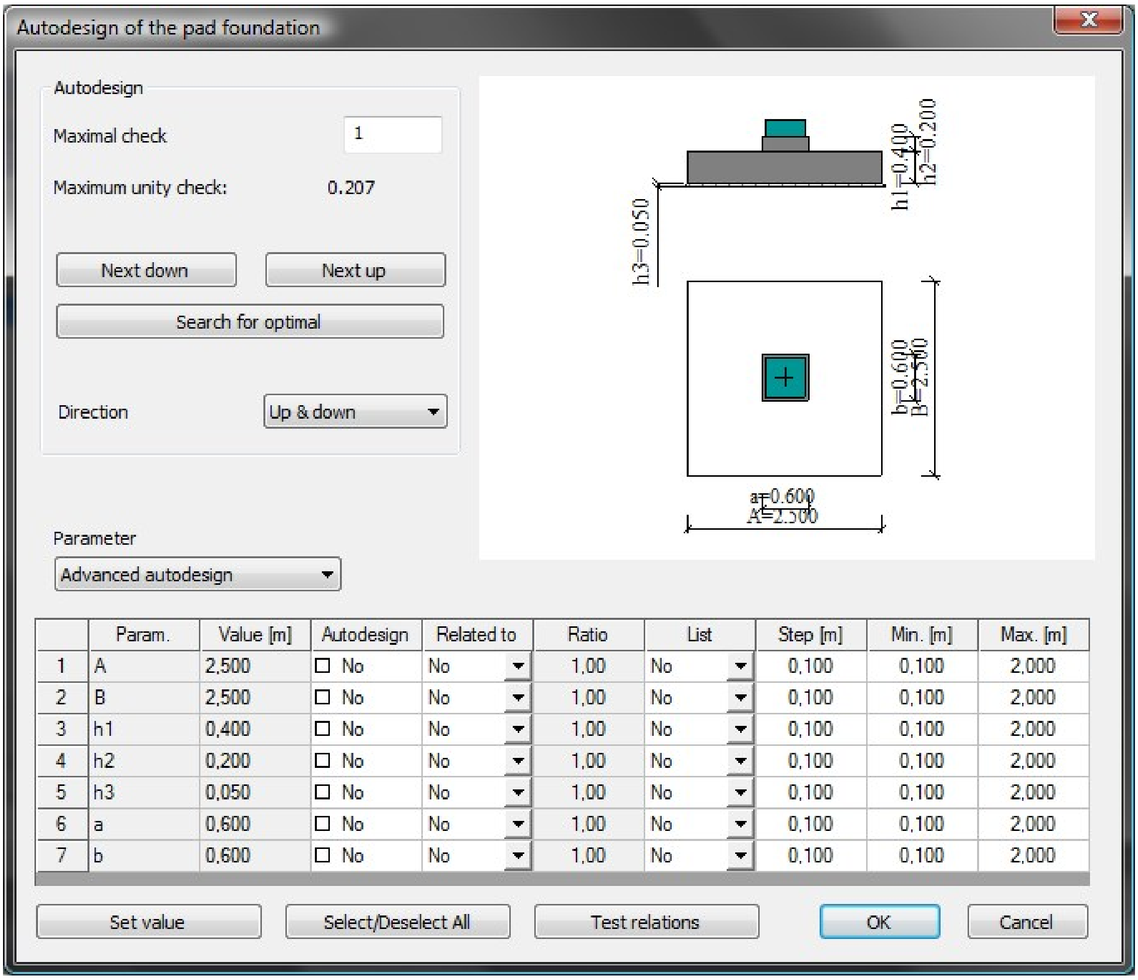
Autodesign of a concrete pad foundation is the same part as the Autodesign located in the standard Geotechnics > Geotechnics services > Pad foundation – stability check.

Autodesign in Geotechnics service



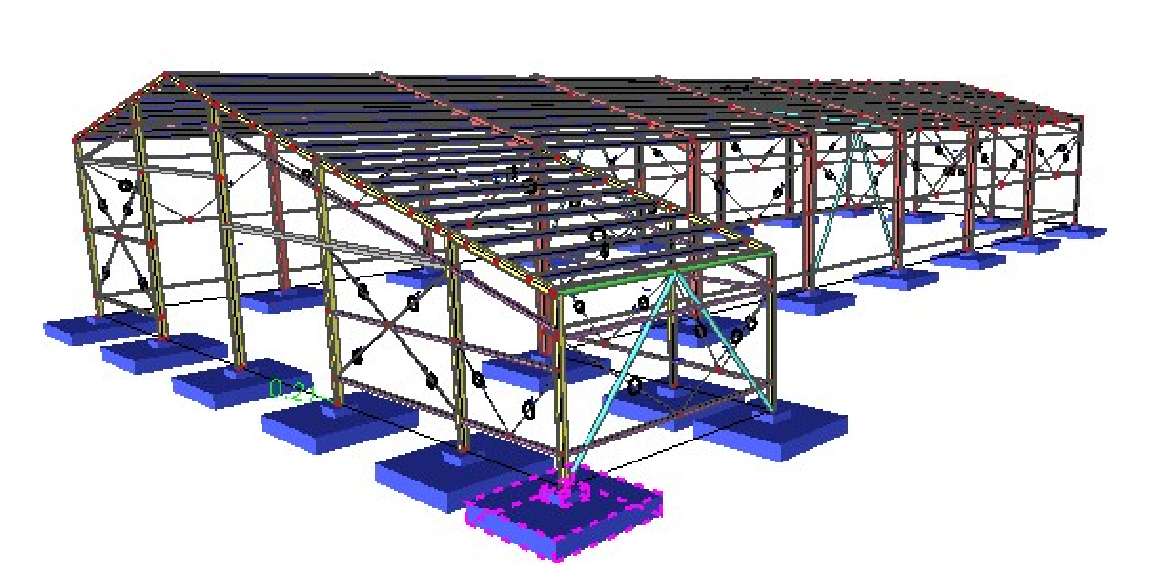


When the user clicks on action button Autodesign then the following dialogue is shown. The dialogue is a bit different than the one used in the general Autodesign but the functionality is the same.

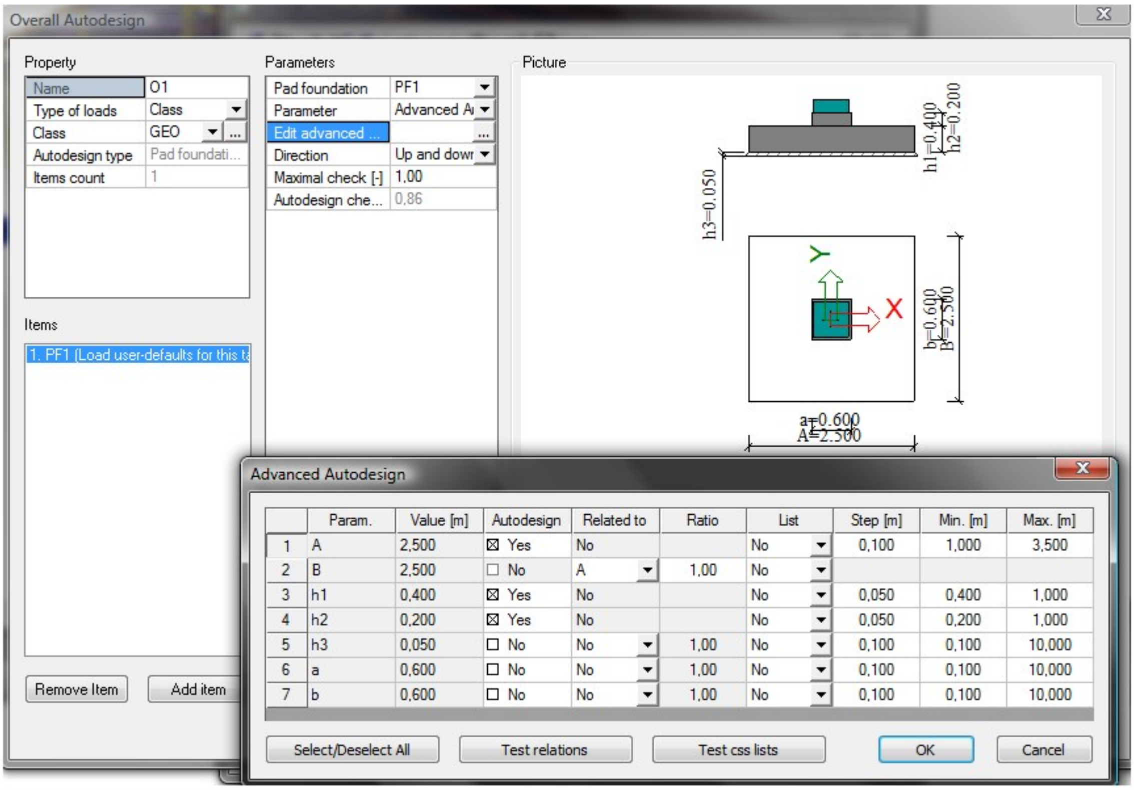


Illustrative example

Let us consider a very simple example of a steel hall laid on concrete pad foundations. The aim of this example is to find the optimal dimensions of one pad foundation. The structure is shown in the following figure.



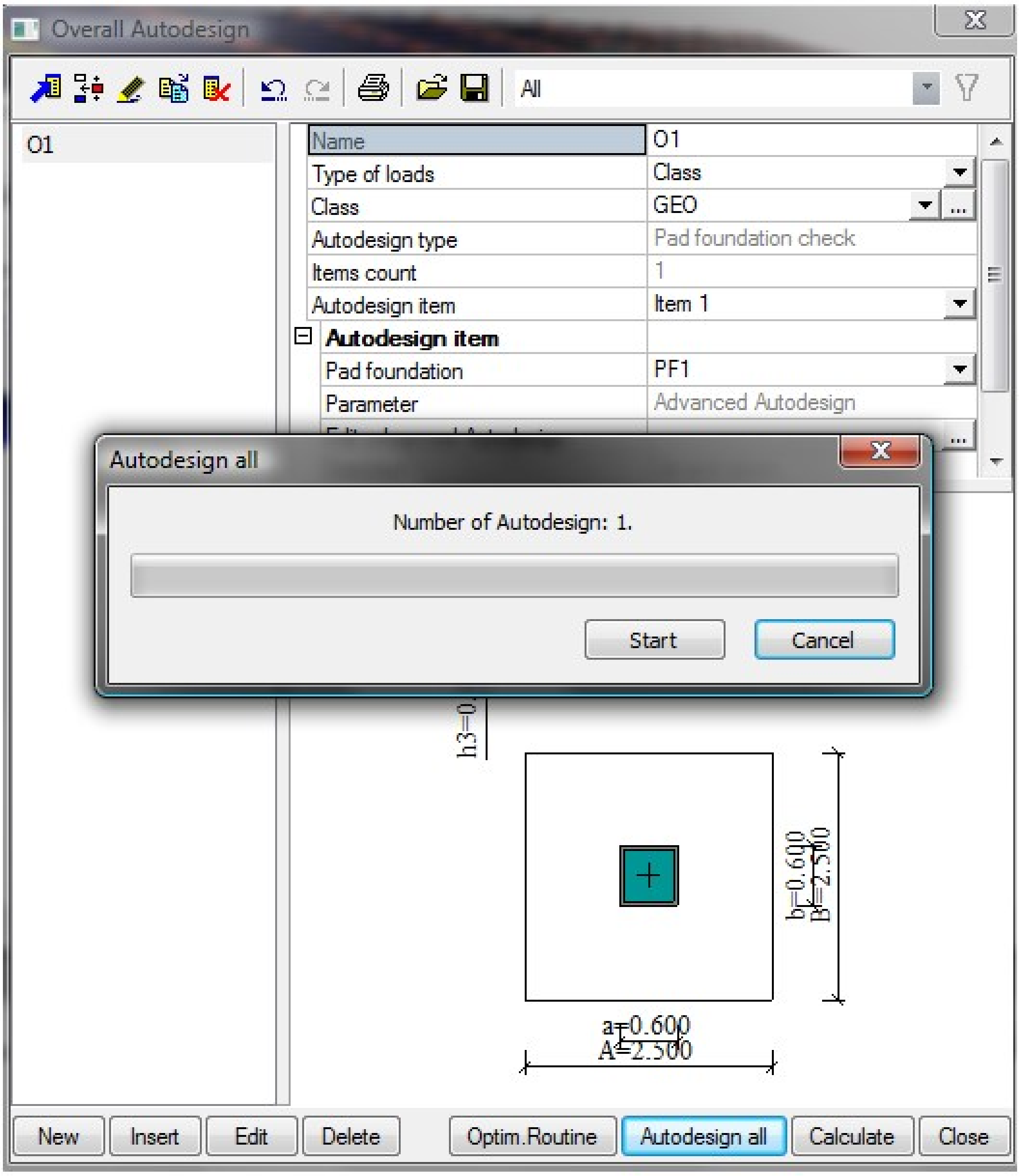
The Autodesign function is defined for one selected pad foundation. You can see the settings.



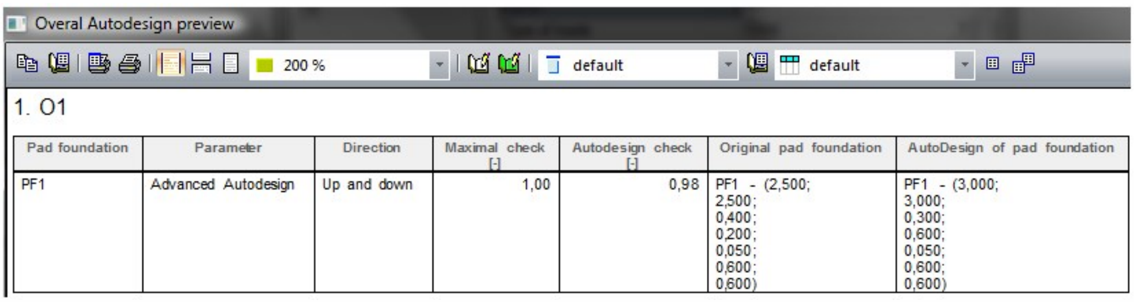
	Param.	Value [m]	Autodesign	Related to	Ratio	List	Step [m]	Min. [m]	Max. [m]
1	A	2,500	☒ Yes	No		No	0,100	1,000	3,500
2	B	2,500	☐ No	A	1,00	No			
3	h1	0,400	☒ Yes	No		No	0,050	0,400	1,000
4	h2	0,200	☒ Yes	No		No	0,050	0,200	1,000
5	h3	0,050	☐ No	No	1,00	No	0,100	0,100	10,000
6	a	0,600	☐ No	No	1,00	No	0,100	0,100	10,000
7	b	0,600	☐ No	No	1,00	No	0,100	0,100	10,000

The advanced Autodesign is used. Dimensions A and B are considered to be the same.

Autodesign of both cross-sections can be run in one step using Autodesign all..



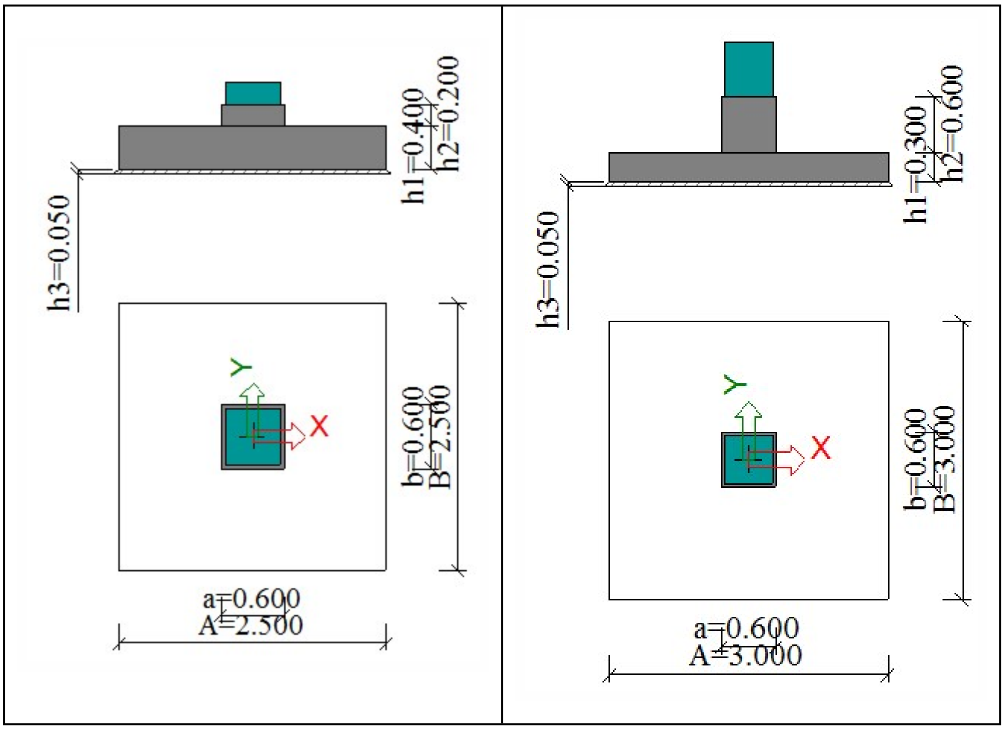
The results are automatically printed in preview.



1. O1

Pad foundation	Parameter	Direction	Maximal check [-]	Autodesign check [-]	Original pad foundation	AutoDesign of pad foundation
PF1	Advanced Autodesign	Up and down	1,00	0,98	PF1 - (2,500; 2,500; 0,400; 0,200; 0,050; 0,600; 0,600)	PF1 - (3,000; 3,000; 0,300; 0,600; 0,050; 0,600; 0,600)

Initial	Optimized

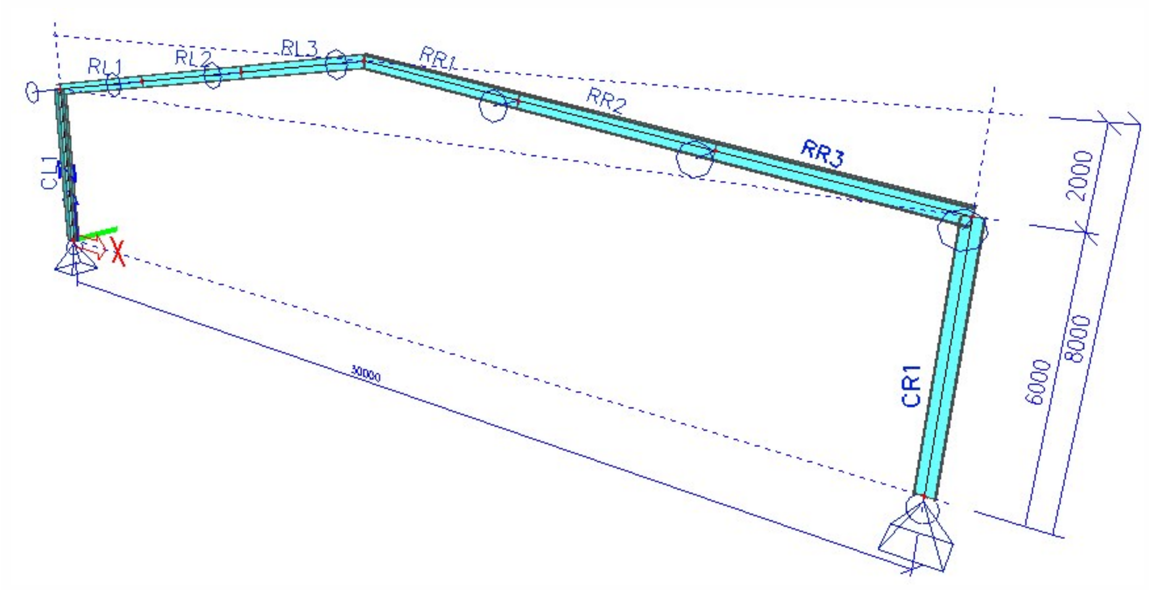


Steel hall - Frame Autodesign

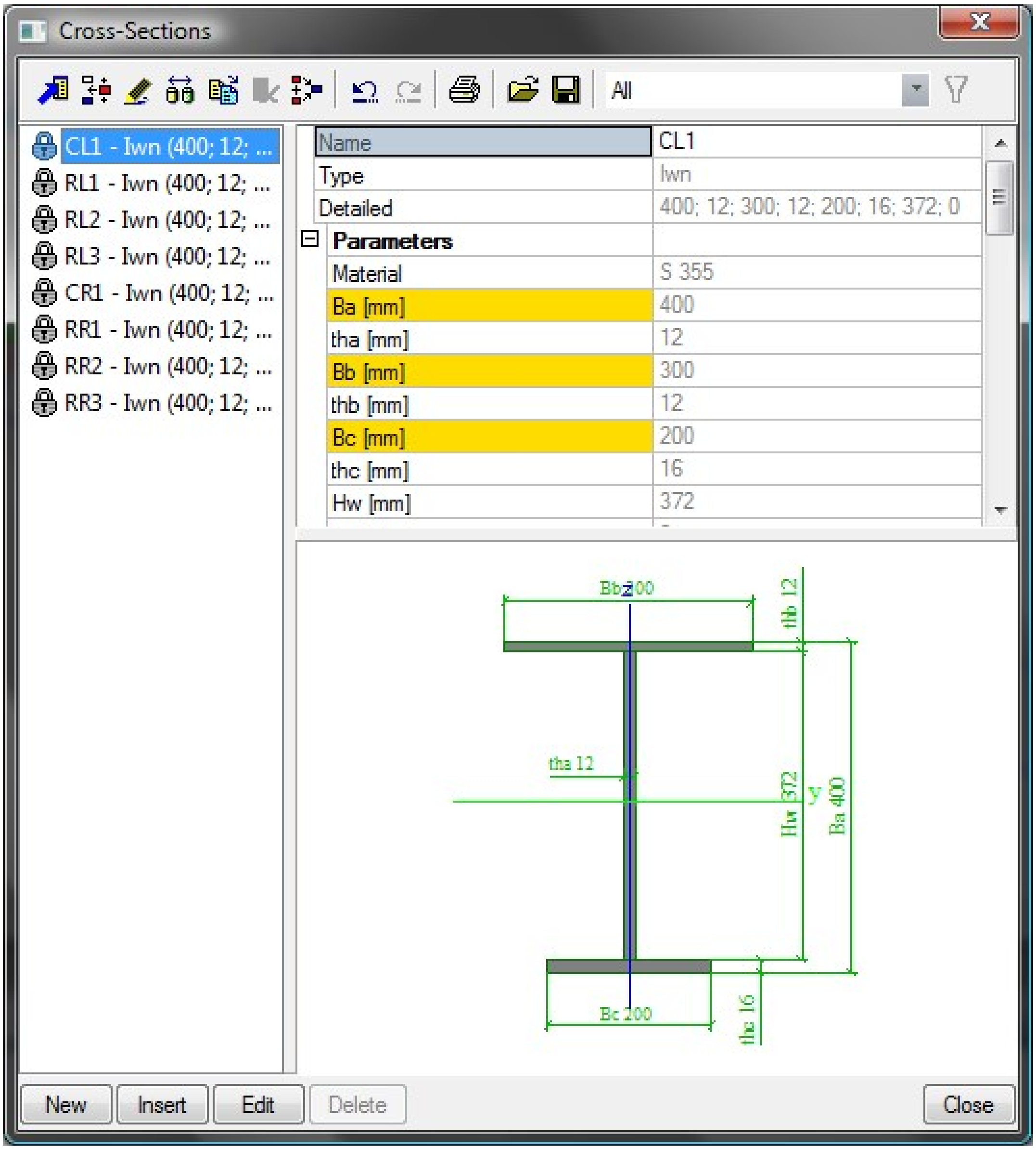
Most of Autodesign items are related to the optimal design of steel frames or halls. The following items are related to the frame design:

- Frame - Autodesign manager
- Frame – CSS height Autodesign
- Frame - deflection Autodesign
- Frame - flange Autodesign
- Frame – web Autodesign
- Frame - flange thickness Autodesign

All items mentioned above are demonstrated on a simple steel frame (S355) made of Iwn cross-sections. The frame is 30m long with 6.0m high columns. The top point is 2.0 above the head of the columns. The structure is subject to self-weight, permanent load (-3kN/m), variable load (-5kN/m) and automatically generated climatic load (wind, snow) according to EN.

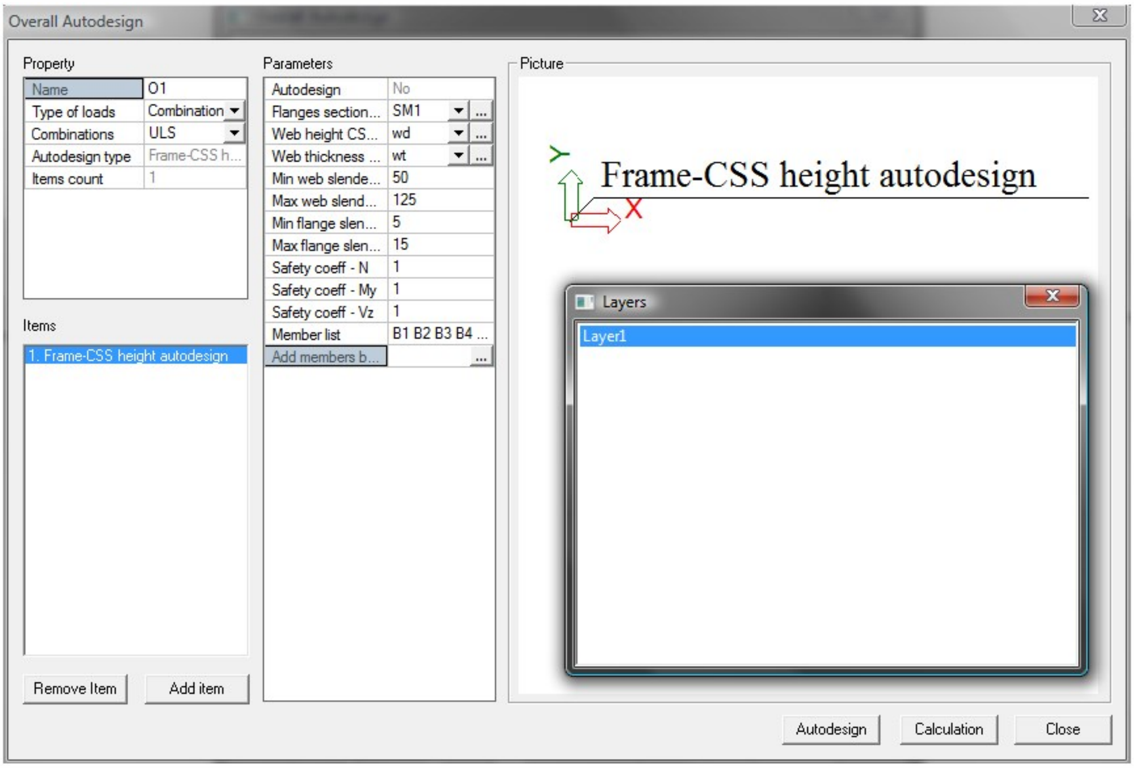


Eight different cross-sections are used (two for columns and six for rafters). All cross-sections are welded.



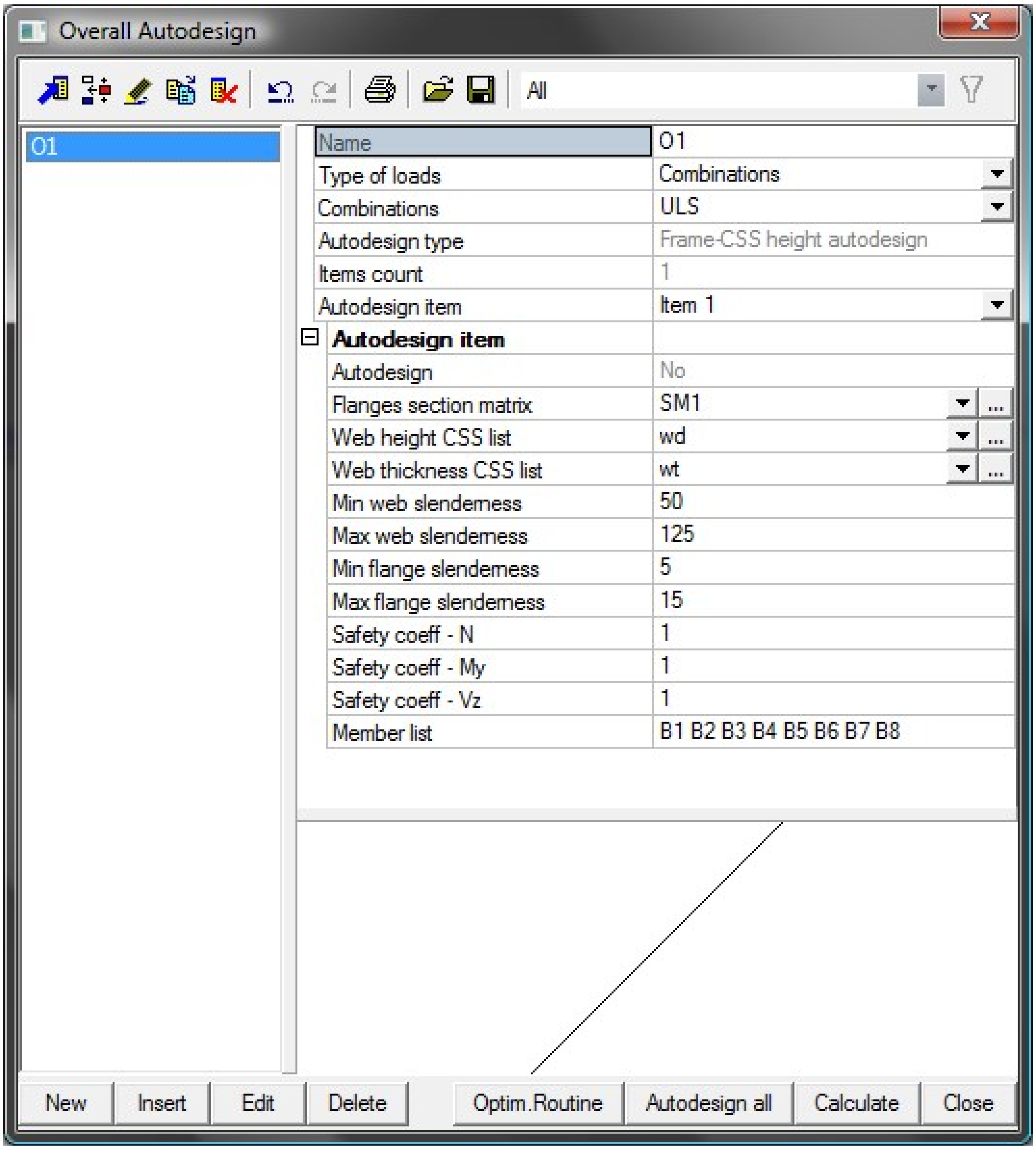
Frame – CSS height Autodesign

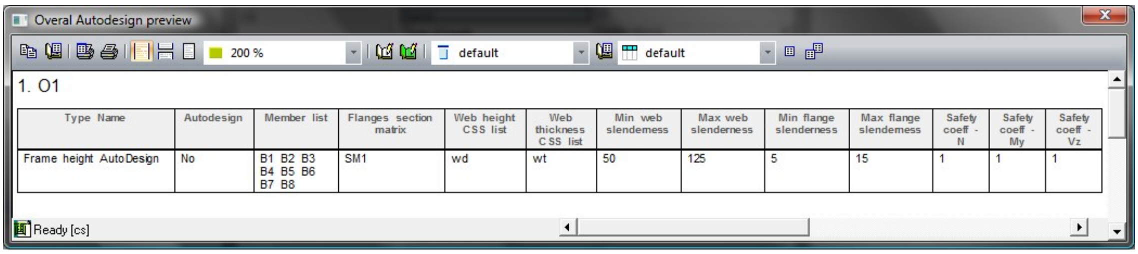
CSS height Autodesign is the first of steel frame AutoDesigns. To simplify the design procedure, the height design optimization is based on a symmetrical cross section. The height design is based on the interaction formula for combined bending and compression.



The property of this Autodesign is the following:

Flanges section matrix	link to library of section matrices
Web height CSS	link to library of cross-section lists
Web thickness	link to library of cross-section lists
Min web slenderness	minimal slenderness of the web working as constraint
Max web slenderness	maximal slenderness of the web working as constraint
Min flange web slenderness	minimal slenderness of the flange working as constraint
Max flange web slenderness	maximal slenderness of the flange working as constraint
Safety coeff – N	XXX
Safety coeff - My	XXX
Safety coeff – Mz	XXX
Member list	list of Autodesigned members
Add members by layer	link to layer database for selection of a layer





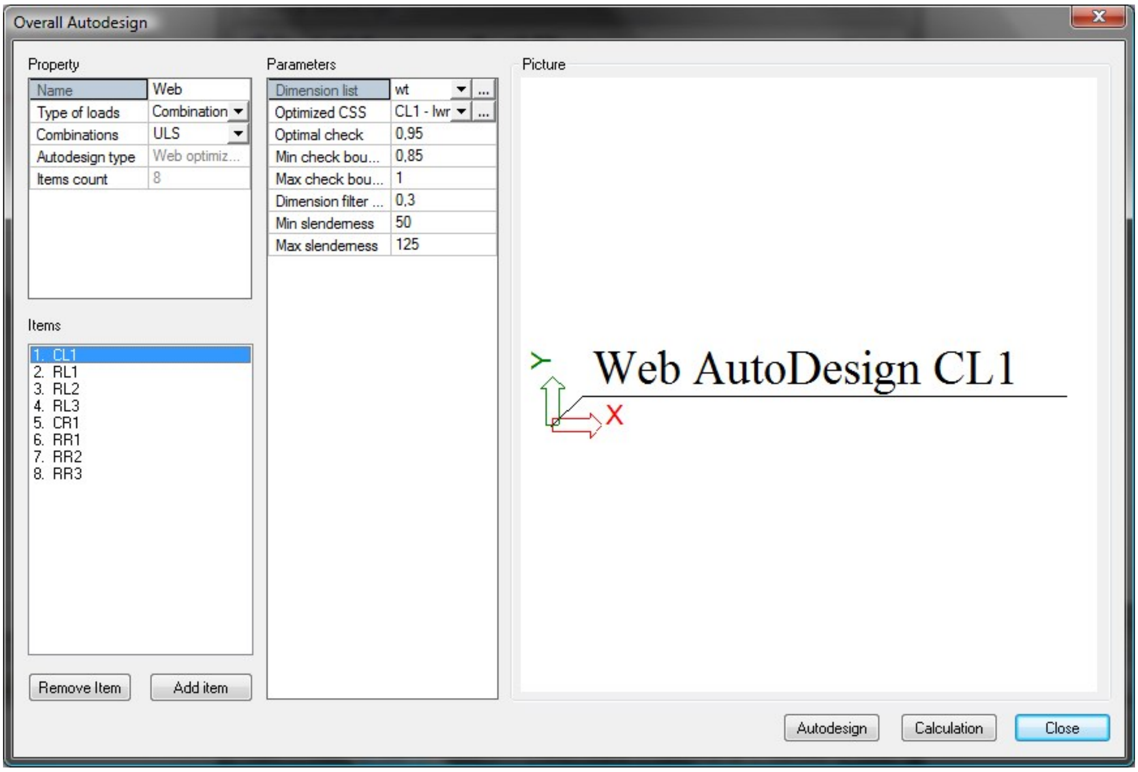
The height of the cross-section is Autodesigned.

Frame – web Autodesign

Web optimization is the second step of the steel frame design. The goal of this item is the web thickness. The Autodesign function of the web optimization type is defined for each cross-section.

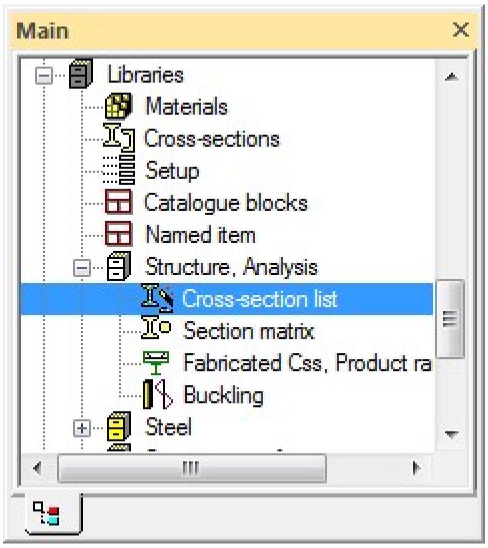
The program is automatically looking for the smallest web thickness from the defined Thickness list with satisfying steel check. It works only with the CSS shear check and web thickness. It means that the values of steel shear check are verified in each section (default value is 10 on a member) for the load class/combination which is set by the user. The program checks the web slenderness to make sure that the Autodesign does not set a web thickness which is outside the defined range. The starting web thickness of this Autodesign is the value after the CSS height design. It checks the value of the shear check for this thickness.

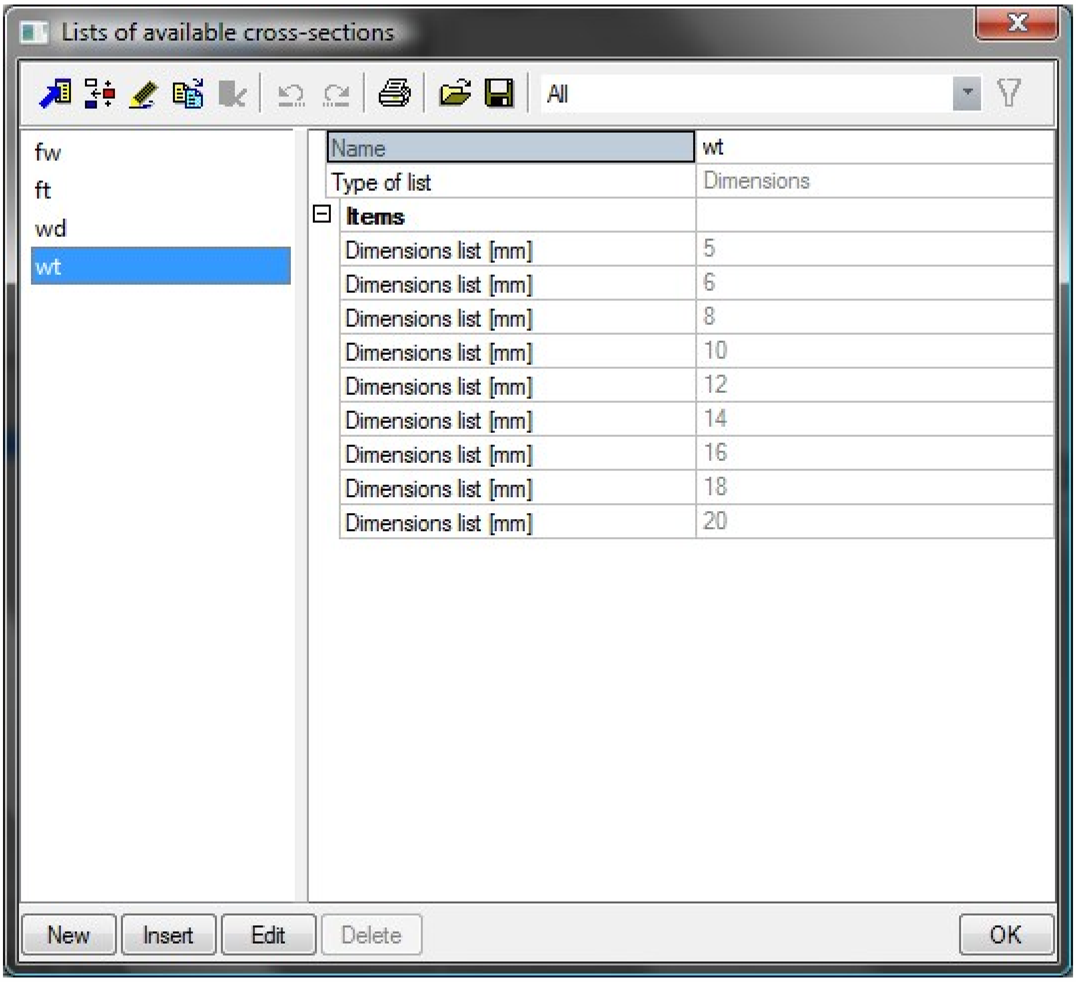
- If the value of the check is bigger than 1 then it finds in the list the first thickness which has the check lower than 1.
- If the value of the check is smaller than 1 then it tries all thicknesses from the list which are smaller than the current one and returns the thickness which gives the check closest to 1.



The optimal (0.95), minimal (0.85) and maximal (1.0) check values are the same as for the flange optimization. Minimal (50) and maximal (125) slenderness of web parts are different. This web optimization has different properties than the flange optimization:

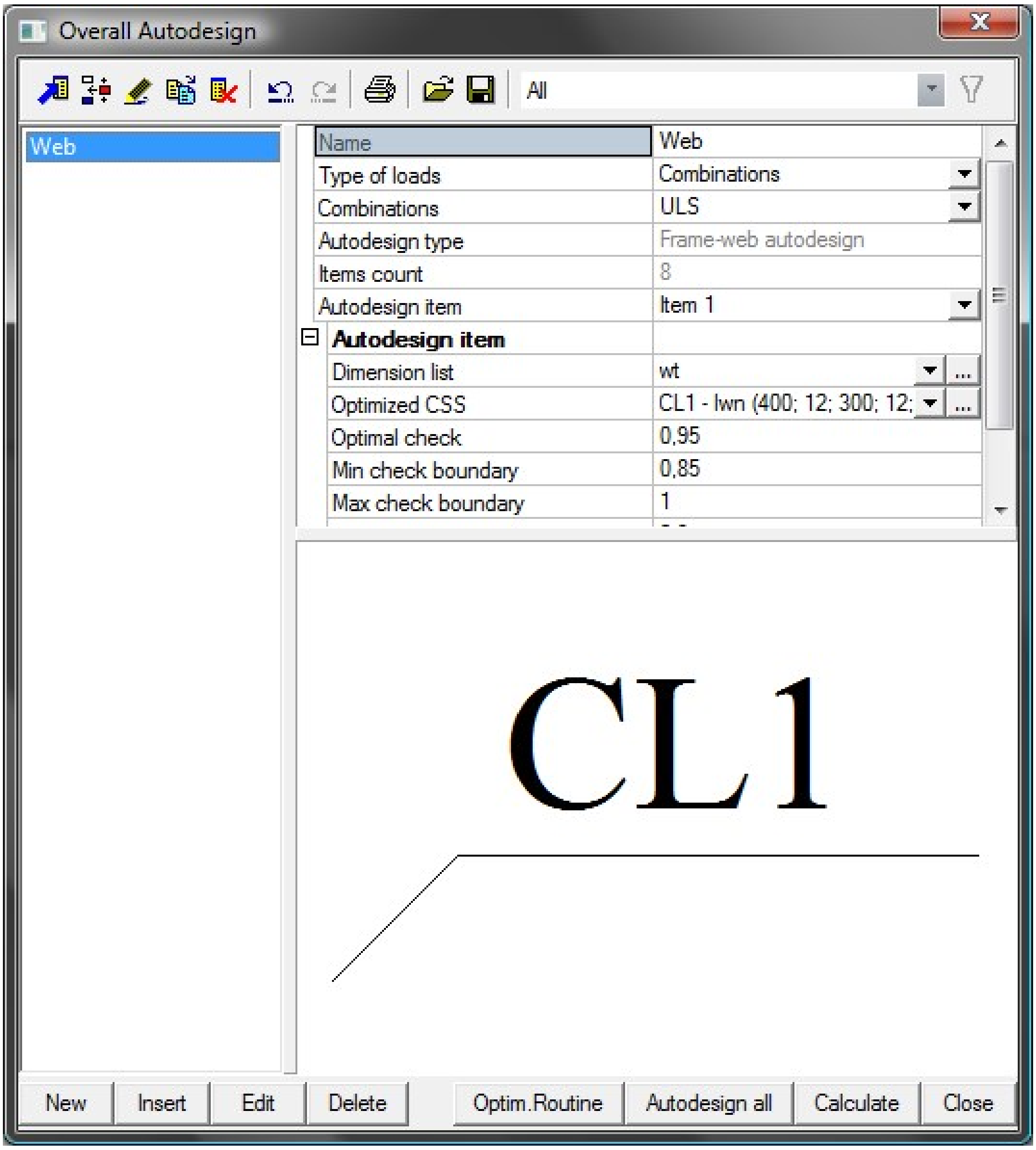
Dimension list
Enables the user to use predefined values of one dimension according to the list defined in the Cross-section list library. This library is stored in Libraries > Structure, analysis > Cross-section list.



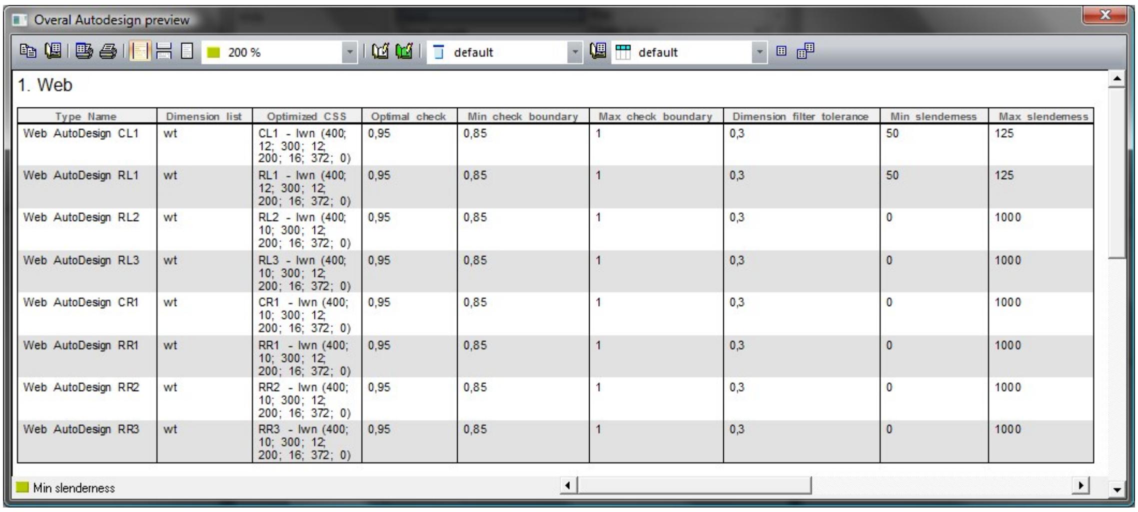


It is possible to define three types of cross-section lists (see the following figure). Cross-section list of type Dimension list can be used for Autodesign only. The dimension list wt is used for optimization.

When all values are set the dialogue looks like this

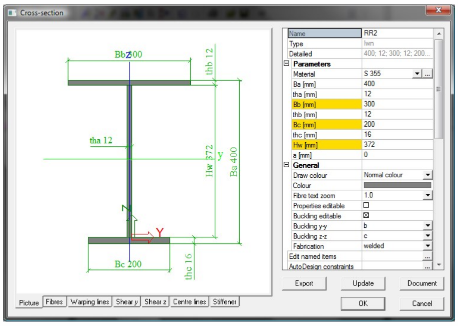
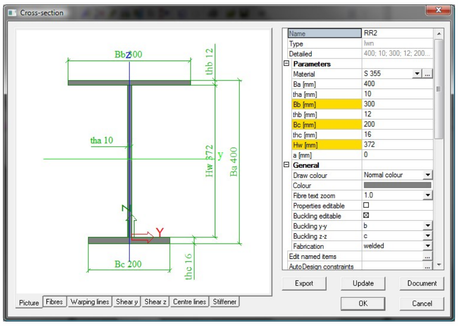


The results preview is printed



Type Name	Dimension list	Optimized CSS	Optimal check	Min check boundary	Max check boundary	Dimension filter tolerance	Min slenderness	Max slenderness
Web AutoDesign CL1	wt	CL1 - Iwn (400; 12; 300; 12; 200; 16; 372; 0)	0,95	0,85	1	0,3	50	125
Web AutoDesign RL1	wt	RL1 - Iwn (400; 12; 300; 12; 200; 16; 372; 0)	0,95	0,85	1	0,3	50	125
Web AutoDesign RL2	wt	RL2 - Iwn (400; 10; 300; 12; 200; 16; 372; 0)	0,95	0,85	1	0,3	0	1000
Web AutoDesign RL3	wt	RL3 - Iwn (400; 10; 300; 12; 200; 16; 372; 0)	0,95	0,85	1	0,3	0	1000
Web AutoDesign CR1	wt	CR1 - Iwn (400; 10; 300; 12; 200; 16; 372; 0)	0,95	0,85	1	0,3	0	1000
Web AutoDesign RR1	wt	RR1 - Iwn (400; 10; 300; 12; 200; 16; 372; 0)	0,95	0,85	1	0,3	0	1000
Web AutoDesign RR2	wt	RR2 - Iwn (400; 10; 300; 12; 200; 16; 372; 0)	0,95	0,85	1	0,3	0	1000
Web AutoDesign RR3	wt	RR3 - Iwn (400; 10; 300; 12; 200; 16; 372; 0)	0,95	0,85	1	0,3	0	1000

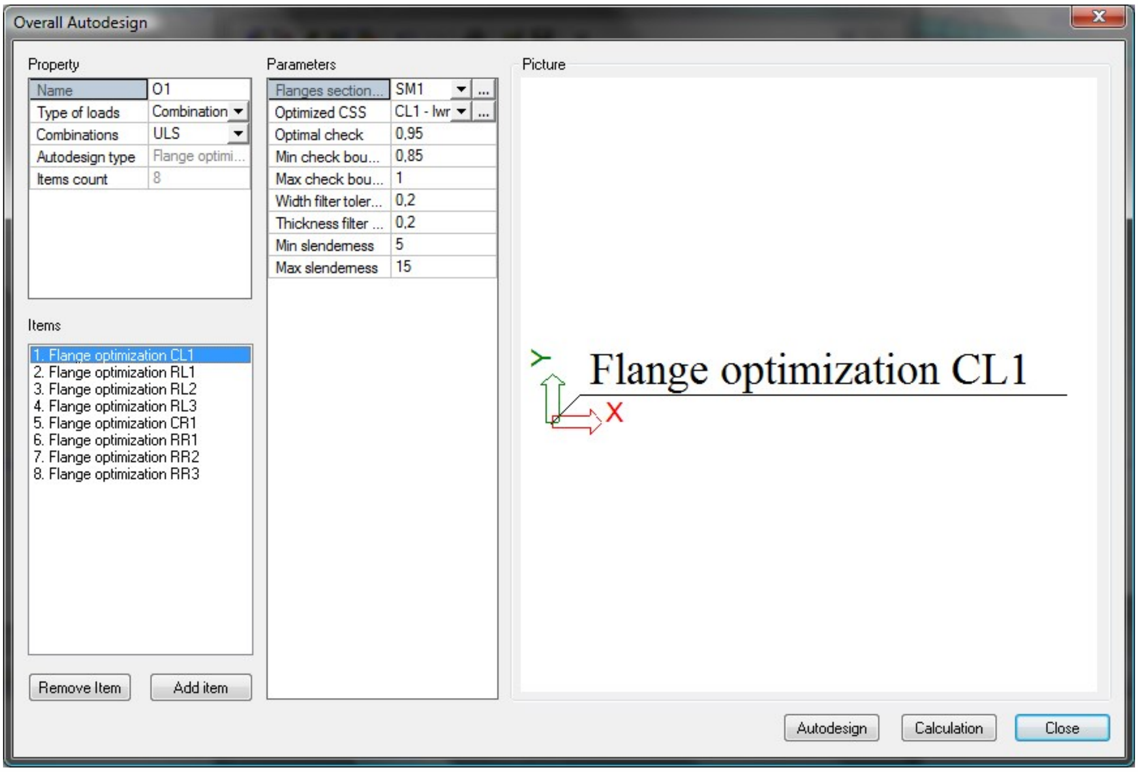
The comparison of the initial and optimized values of the cross-section is clear from the following table. Web thickness decreased from 12 to 10mm.

Initial	Optimized
	

Frame - flange Autodesign

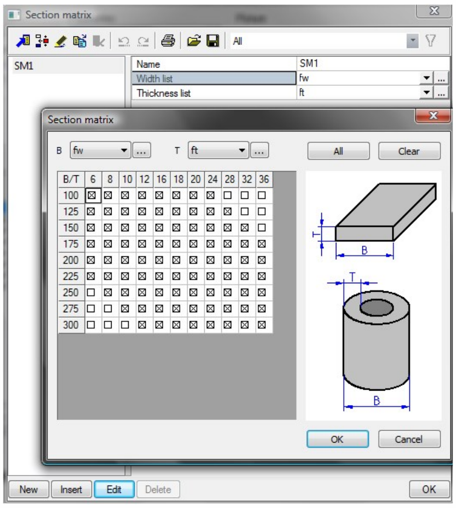
The goal of this item is to design the optimal dimension of flange parts of steel cross-sections. The Autodesign function of the Flange optimization type is defined for each cross-section.

In this step the CSS is designed as symmetrical which implies that the top and bottom flanges have the same geometry. The program is automatically looking for the lightest geometry of the flange from the defined Flange section matrix. It takes all defined combinations of flange widths and thicknesses and sorts them by the flange area. Then the program looks for the first item from this list with CSS check smaller than 1.



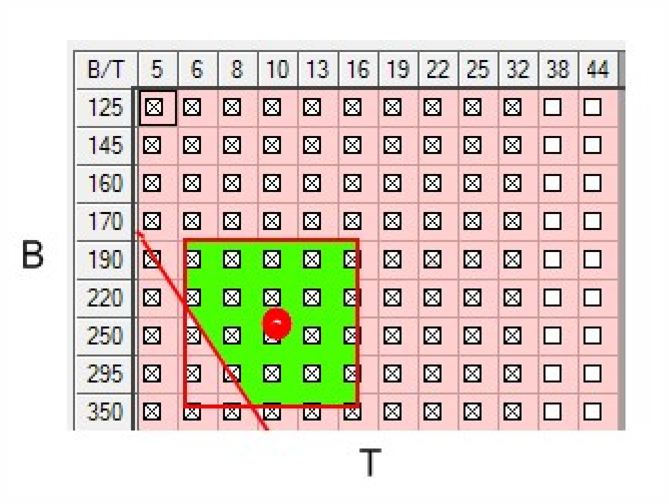
This Flange optimization has different properties:

Flanges section — here is the link to Section matrix library. Section matrix library is defined in Libraries > Structure, Analysis> Section matrix. Generally, the section matrix is the matrix of the defined sets of flange width and thickness. It can be defined by the user and it can represent the manufacturing possibilities of the steel provider. The section matrix used for this example is displayed in the following figure.



Optimized CSS	Link to the cross-section which should be optimized from library. For each optimized cross-section a different Autodesign item is created. (8 items of flange optimization type).
The following items work as constraints.	
Optimal check	check values which should be achieved for the structure with optimal dimensions
Min. boundary check	minimal check value which is accepted for the optimal configuration of the structure
Max. boundary check	maximal check value which is accepted for the optimal configuration of the structure
Width filter tolerance	Used tolerance for width parameters
Thickness filter tolerance	Used tolerance for thickness parameter
Min slenderness	minimal slenderness of the flange of the cross-section
Max slenderness	maximal slenderness of the flange of the cross-section

The tolerance is defined relatively, from the actual definition only those inside the range are checked. Graphically it can be interpreted as shown in the following picture.

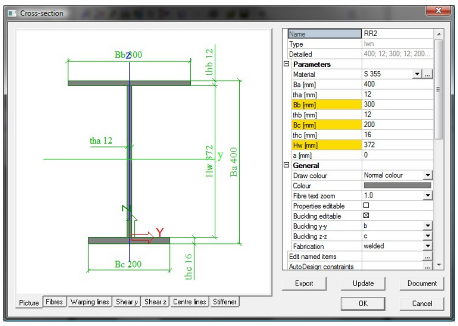
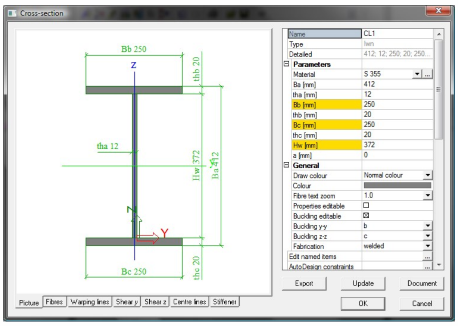


- The red dot in the picture shows the flange geometry as a result of the height design. The flange geometry of the matrix which most closely matches this result is used. Using the tolerance parameters from the design setup, a range of widths and thicknesses is defined as illustrated with the red rectangle.
- In addition to the previous tolerance, the flange slenderness limit set in the relative limitations is accounted for. This is illustrated in the picture by the inclined line.
- The final set of flange geometries taking into account both the tolerance and the slenderness limit is shown in the picture in green.

When all values are set for each optimized cross-section then the overall Autodesign can run.

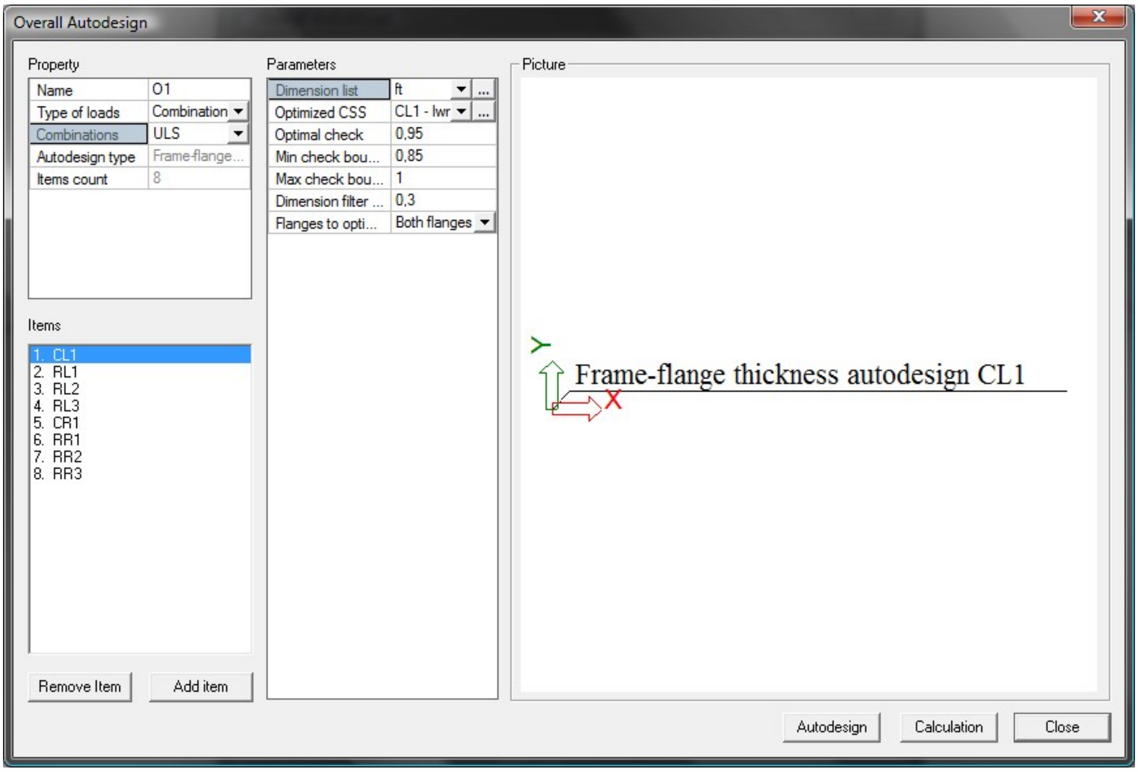
> It is not recommended to use the flange optimization only. In that case the I cross-section is transformed to flange cross-section which is a non-sense.

The comparison of the initial and optimized values of the cross-section is clear from the following table. The initial flange dimensions for upper (12x300mm) and for lower (16x200mm) flange changed to 20x250mm for upper and lower respectively.

Initial	Optimized
	

Frame - flange thickness Autodesign

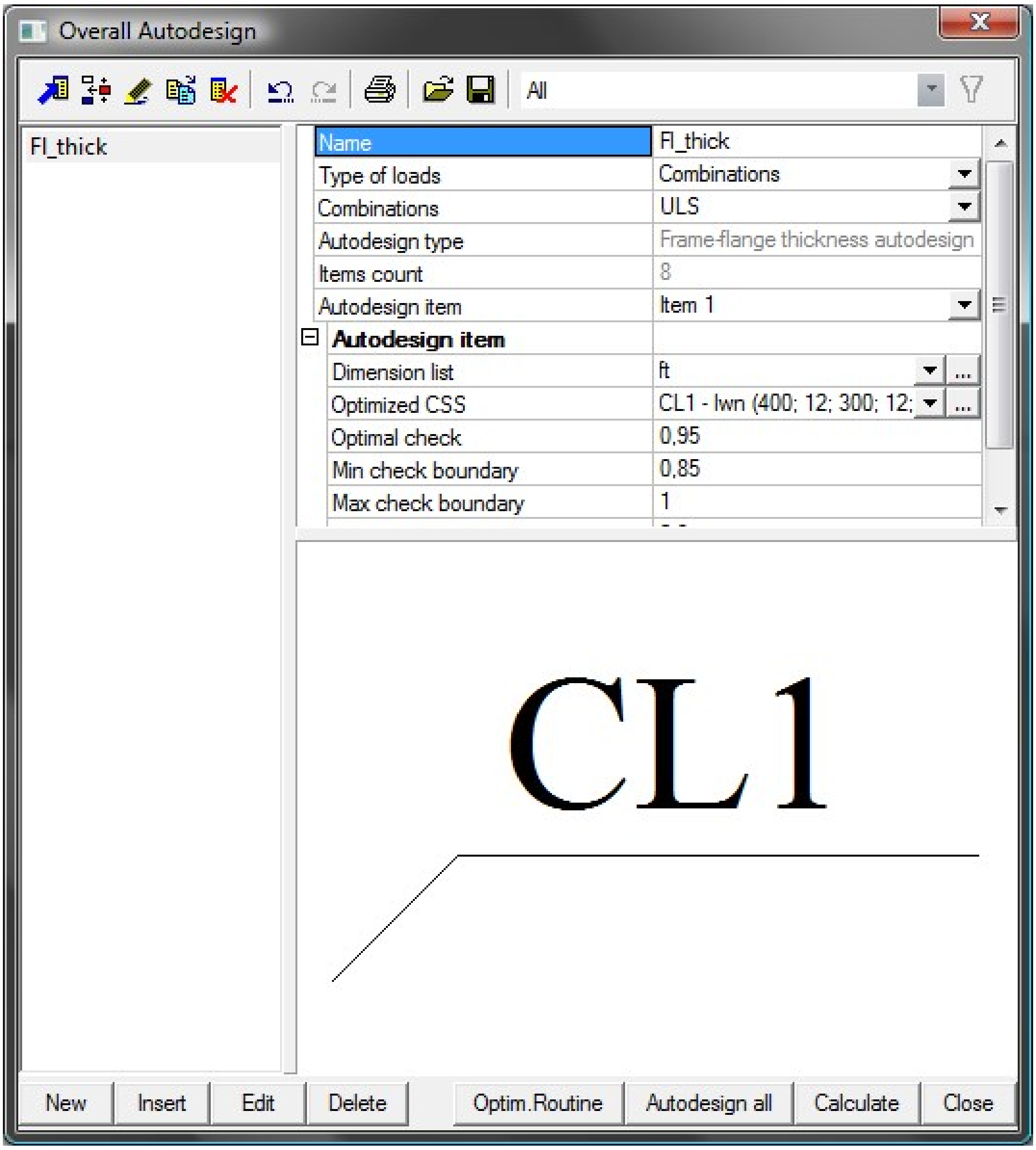
Flange thickness optimization is the next step in the Autodesign of steel frame cross-sections. The goal of this item is to design the optimal dimension of the flange. The Autodesign function of the Flange thickness optimization type is defined for each cross-section.



The optimal (0.95), minimal (0.85) and maximal (1.0) check values are the same as for the flange optimization. The dimension filter tolerance is set to 0.3. There is only one property different for the web and flange Autodesign:

Flange to optimize
Selection of Inner / Outer / Both flanges.

When all values are set the dialogue looks like this



It is not recommended to use the flange optimization only. In that case the I cross-section is transformed to flange cross-section which is a non sense.

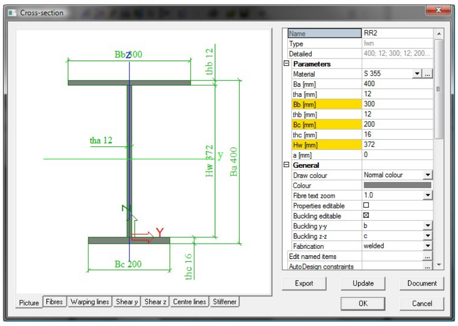
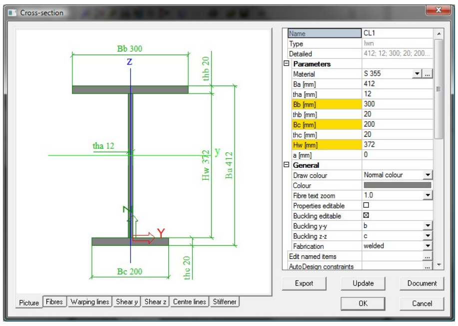
Overal Autodesign preview

1. Fl_thick

Type Name	Dimension list	Optimized CSS	Optimal check	Min check boundary	Max check boundary	Dimension filter tolerance	Flanges to optimize
Frame-flange thickness autodesign CL1	ft	CL1 - Iwn (412; 12; 300; 20; 200; 20; 372; 0)	0,95	0,85	1	0,3	Both flanges
Frame-flange thickness autodesign RL1	ft	RL1 - Iwn (412; 12; 300; 20; 200; 20; 372; 0)	0,95	0,85	1	0,3	Both flanges
Frame-flange thickness autodesign RL2	ft	RL2 - Iwn (412; 12; 300; 20; 200; 20; 372; 0)	0,95	0,85	1	0,3	Both flanges
Frame-flange thickness autodesign RL3	ft	RL3 - Iwn (412; 12; 300; 20; 200; 20; 372; 0)	0,95	0,85	1	0,3	Both flanges
Frame-flange thickness autodesign CR1	ft	CR1 - Iwn (412; 12; 300; 20; 200; 20; 372; 0)	0,95	0,85	1	0,3	Both flanges
Frame-flange thickness autodesign RR1	ft	RR1 - Iwn (412; 12; 300; 20; 200; 20; 372; 0)	0,95	0,85	1	0,3	Both flanges
Frame-flange thickness autodesign RR2	ft	RR2 - Iwn (412; 12; 300; 20; 200; 20; 372; 0)	0,95	0,85	1	0,3	Both flanges
Frame-flange thickness autodesign RR3	ft	RR3 - Iwn (412; 12; 300; 20; 200; 20; 372; 0)	0,95	0,85	1	0,3	Both flanges

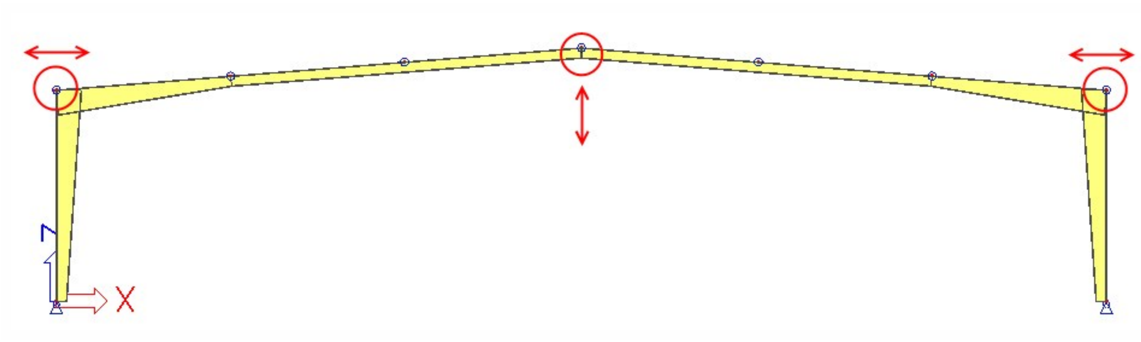
Max check boundary

The comparison of the initial and optimized values of the cross-section is clear from the following table. The initial flange dimensions for upper (12x300mm) and lower (16x200mm) flange changed to 20x300mm and 20x200mm for upper and lower respectively.

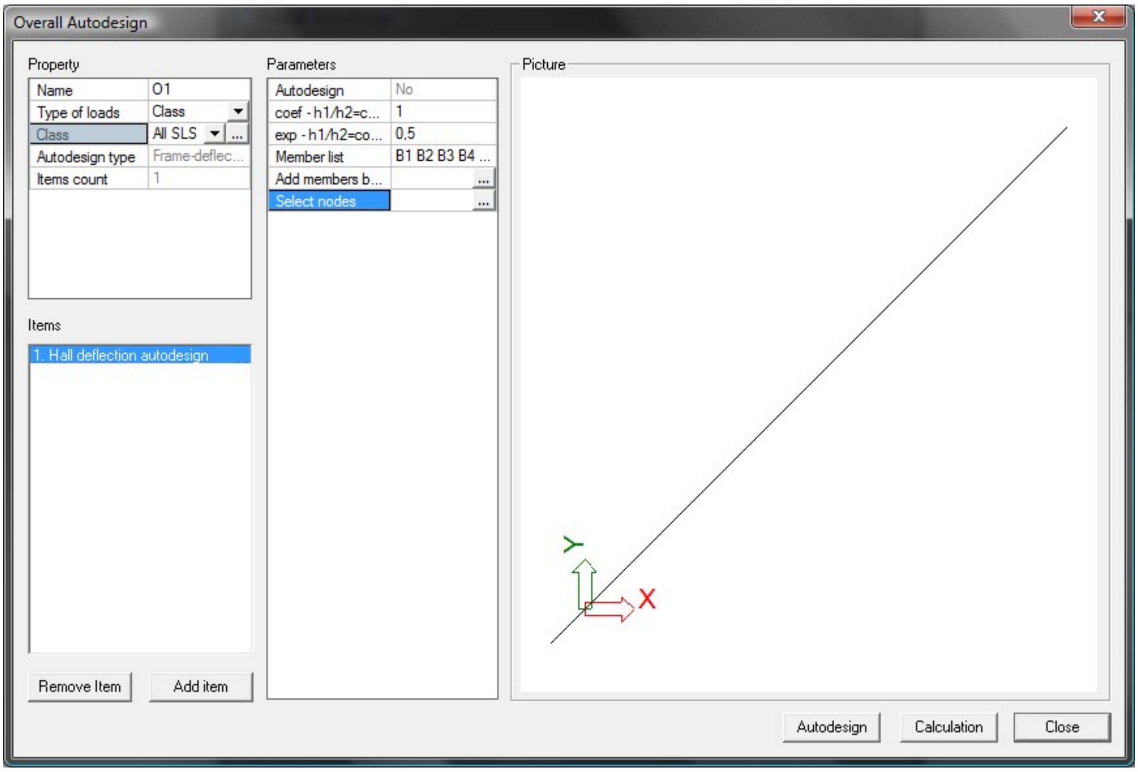
Initial	Optimized
	

Frame - deflection Autodesign

First, the program calculates the values of the deflections in all defined nodes. Then it compares these values with the limits of deflections. If the calculated values of deflections are smaller than the limits of deflections then the design of frames is OK and the SLS design stops. Otherwise the program starts a new Autodesign loops. The program checks both horizontal and vertical deflections in all nodes. The following picture shows an example of typical critical nodes of a simple frame.



As specified the program checks the deflections at all nodes since the critical nodes differ per frame type.



There are different properties for this kind of Autodesign:

coef – h1/h2=coef*(check^exp-1)+1
the value used for the optimal design of CS based on the limit deflection (see formula)

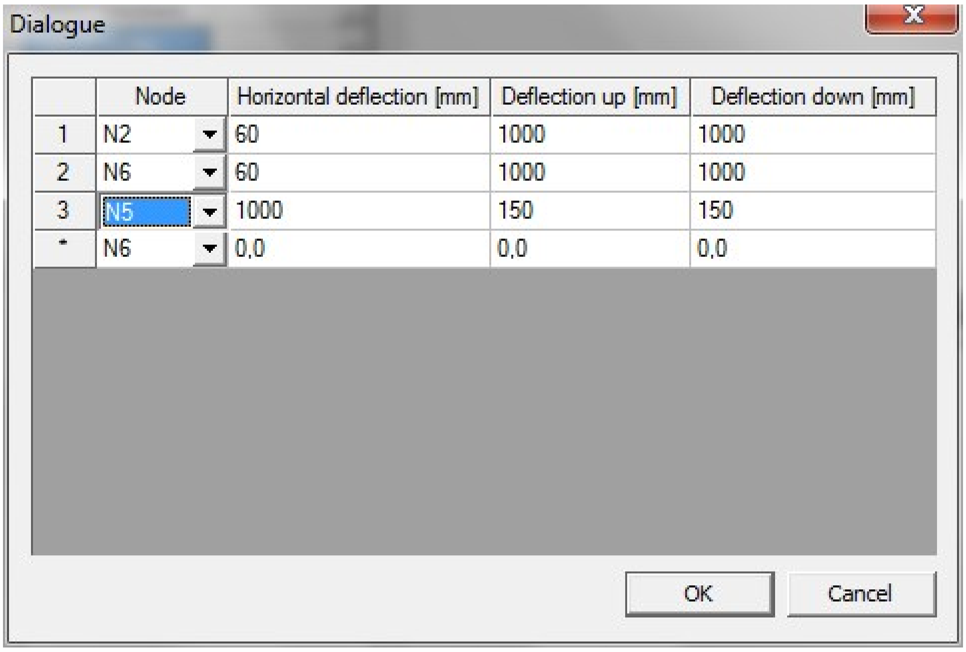
exp – h1/h2=coef*(check^exp-1)+1
the value used for the optimal design of CS based on the limit deflection (see formula)

> It is necessary to use a class as a type of load, otherwise the frame-deflection Autodesign will not work correctly.

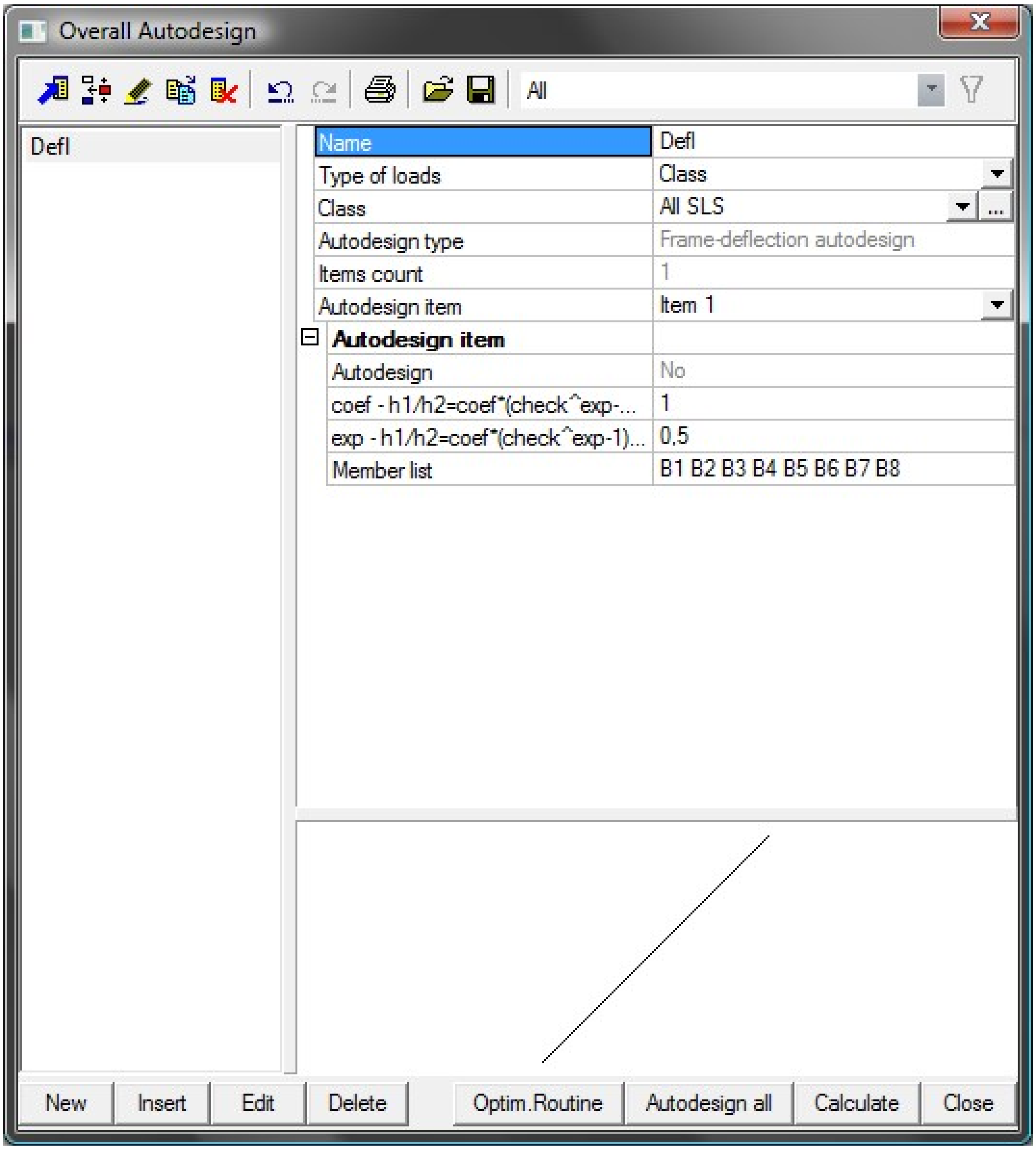
1. Select node N2 (left eave node) from the first list and set values
   Horizontal deflection = 60mm
   Deflection up = 1000mm (criterion will not be used)
   Deflection down = 1000mm (criterion will not be used)
2. Select node N6 (right eave node) from the last list and set values
   Horizontal deflection = 60mm

Deflection up = 1000mm (criterion will not be used)
Deflection down = 1000mm (criterion will not be used)

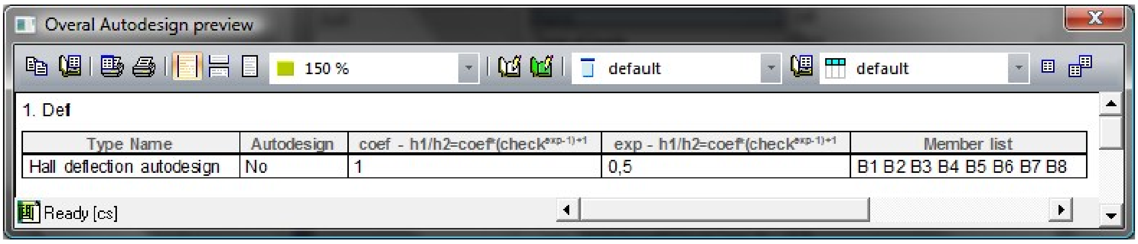
3. Select node N5 (top node) from the last list and set values
   Horizontal deflection = 1000mm (criterion will not be used)
   Deflection up = 150mm
   Deflection down = 150mm



When all values are set the dialogue looks like:

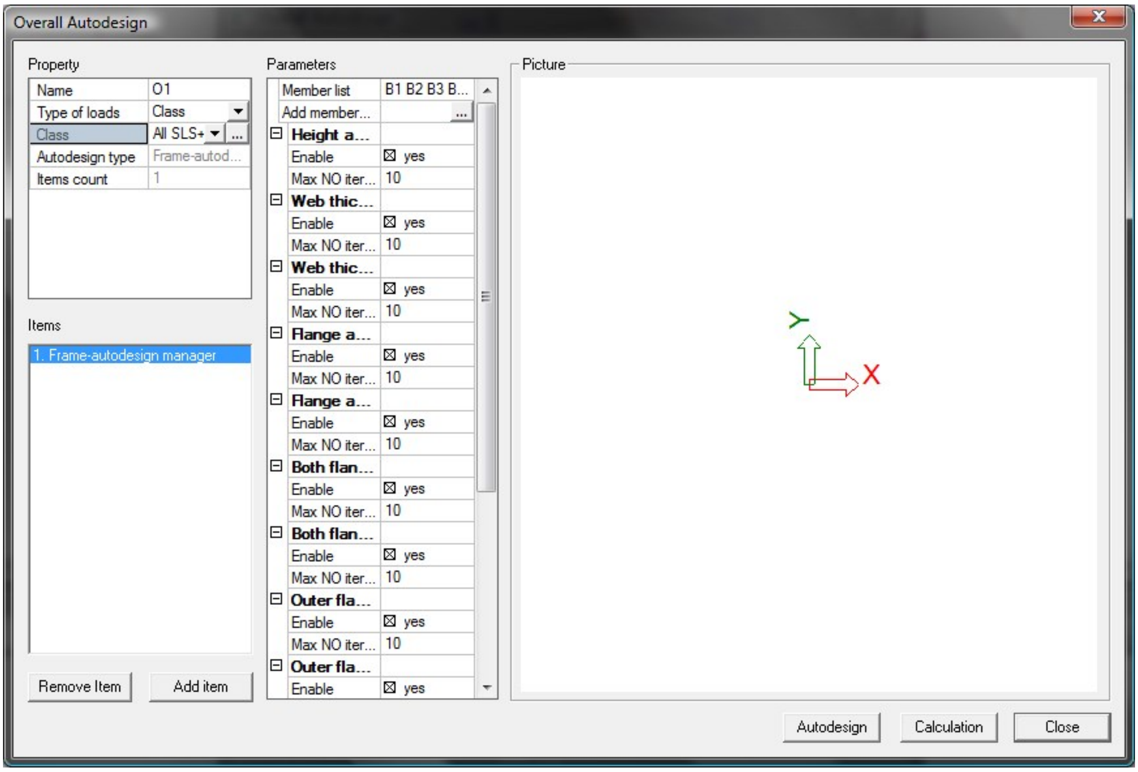


The results preview is printed.



Frame – Autodesign manager

All above-mentioned Autodesign items are used for the optimal design (using special type of Autodesign function (Frame - Autodesign manager). This type includes flange, web, flange thickness, CSS height, and deflection Autodesign.



The property of this Autodesign is the same for particular checks.

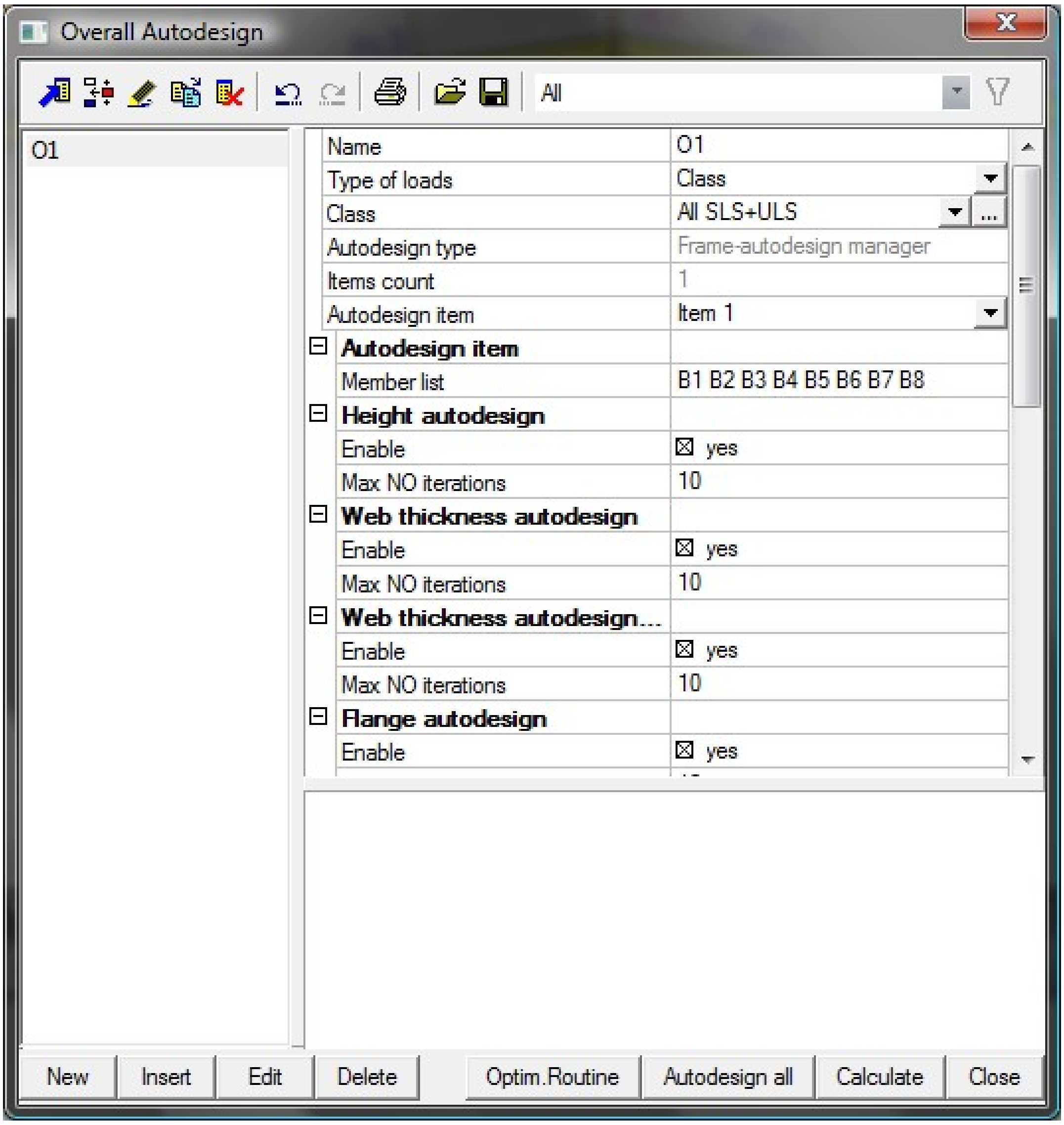
Member list
selected members (read only)

Name of the check
name of the used check for Autodesign (height, web thickness, web thickness in deflection, flange, flange in deflection, bottom flange, and bottom flange in deflection, outer flange, outer flange in deflection, inner flange, and inner flange in deflection, internal column, and deflection design)

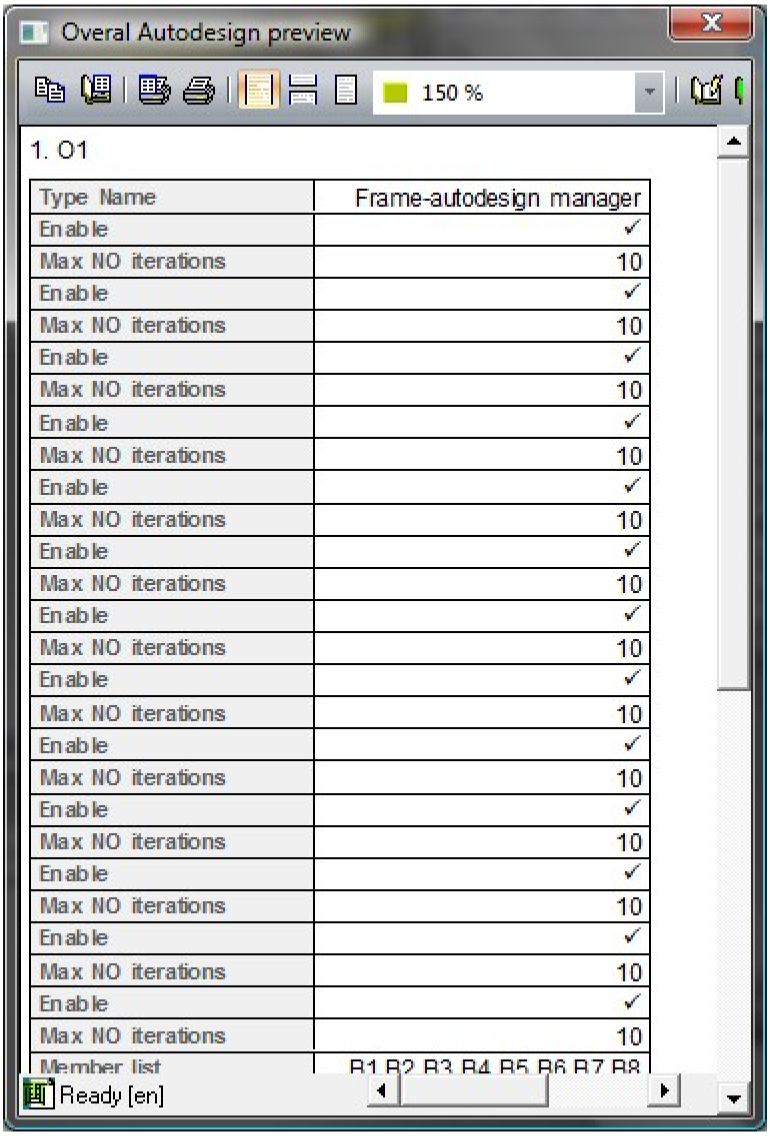
Enable
option, if this check should be considered on Autodesign or not

Max. NO iteration
maximal number of iterations used for Autodesign

When all values are set the dialogue looks like



The results preview is printed


Overal Autodesign preview
150 %
1. O1
Type Name
Frame-autodesign manager
Enable
Max NO iterations
10
Enable
Max NO iterations
10
Enable
Max NO iterations
10
Enable
Max NO iterations
10
Enable
Max NO iterations
10
Enable
Max NO iterations
10
Enable
Max NO iterations
10
Enable
Max NO iterations
10
Enable
Max NO iterations
10
Enable
Max NO iterations
10
Enable
Max NO iterations
10
Enable
Max NO iterations
10
Enable
Max NO iterations
10
Ready [en]

SCIA Optimizer

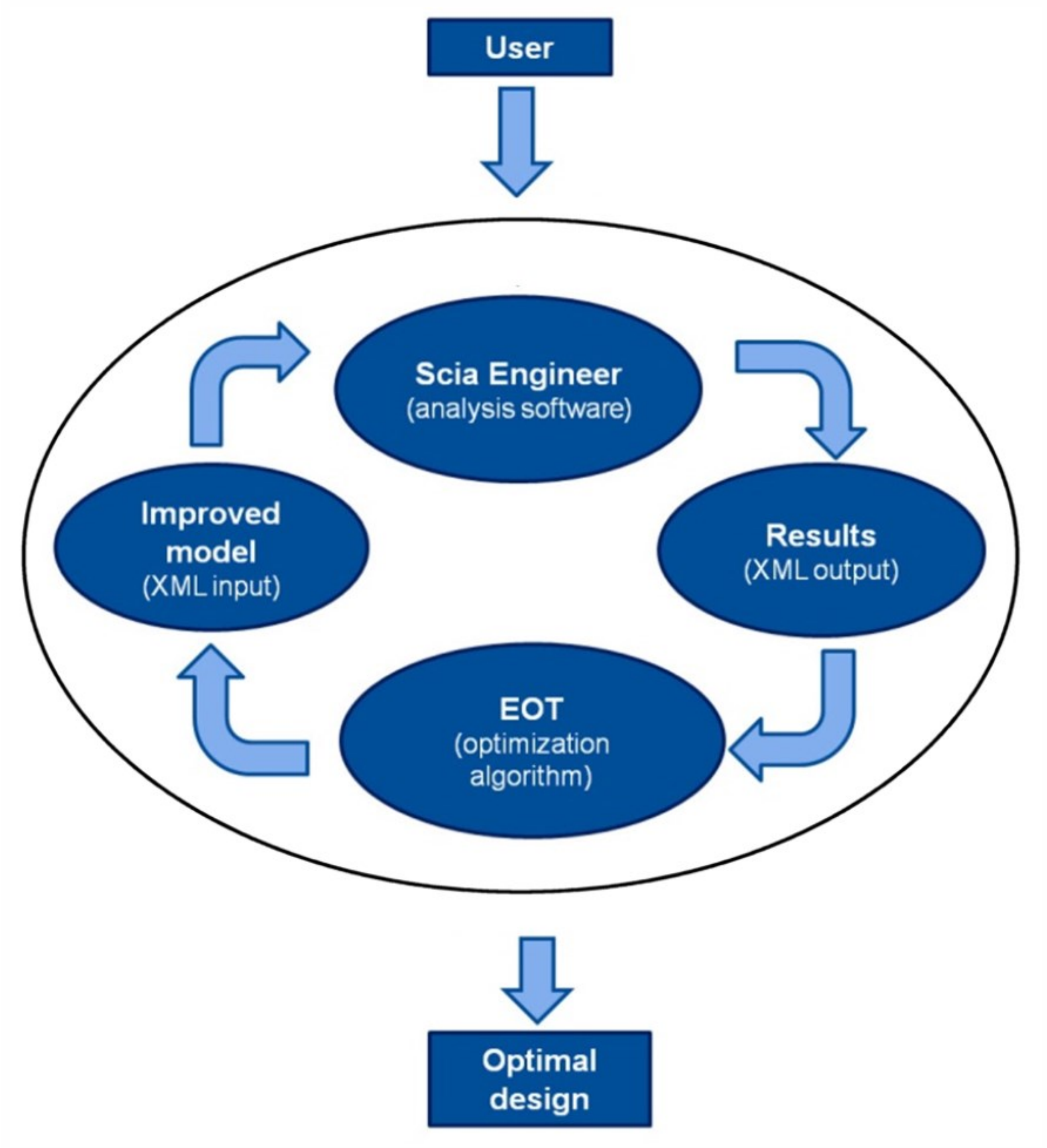
Introduction

The aim of this document is to introduce a brand new tool for optimization of civil engineering structures developed by SCIA and to demonstrate how it can be helpful and effective to everybody who deals with structural design.

About SCIA Engineer Optimizer

SCIA Engineer Optimizer is an example of a new generation of software for the design of structures. It is software which calculates internal forces, checks the compliance to the code, and on top of that, this software is able to "find" the final optimal structural design.

It represents a combination of the widespread structural analysis software - SCIA Engineer - and a separate optimization engine (EOT – Engineering Optimization Tool). The two programs have been integrated together and offer a versatile and complete optimization solution for all types of civil engineering structures.



Motivation

SCIA Engineer Optimizer is very general and flexible because the demands which need to be considered during a really optimal structural design are also rather complex and general. However, thanks to the power of the current computational technologies, all requirements for cost reduction, material savings and environmental protection can be now easily met. Despite the complexity of the general optimization, the optimization process itself is not complicated and this manual will lead you how to proceed.

Once all the required input data are entered, i.e. the model of the analysed structure is defined, the search for the optimal solution runs fully automatically and no interaction from the user is required. For real-life problems several optimal solutions can be found. In such situations, it is up to the user to make the final decision.

Depending on our requirements we may seek different targets. The goal of the optimization depends on us: is it the total weight, costs, eigen frequency or something else. And with respect to the goal, there are many possibilities of what can be optimized, what means, which properties or parameters of the structure are to be changed to reach the optimum: geometry, cross-sections, reinforcement, prestressing, structural arrangement and others.

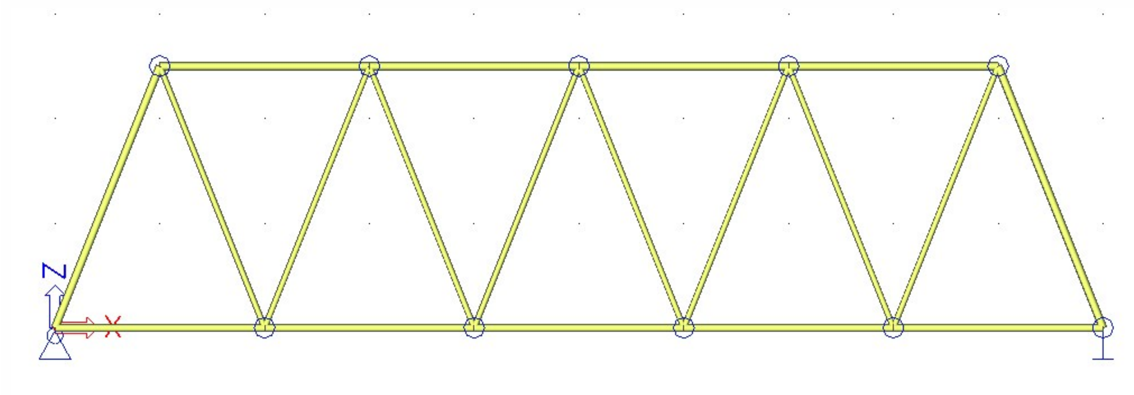
As the constraints we use various values obtained from SCIA Engineer analysis: unity check, capacity, eigen frequency, deflection etc.

Worked example

This example shows the optimization of the geometrical shape with the aim to reach the minimum mass of a steel truss girder with respect to fulfilling the capacity according to for 1st limit state code check implemented in SCIA Engineer (EC3 in this case).

The structure is a simply supported truss girder with the span of 10 metres, see the next picture. The goal of optimization is find optimal height of the truss (which will most probably vary along the girder length, see results at the end of this tutorial).

All members have tubular cross-sections but the thickness and diameter are variable. However both values have to be within certain limitations.

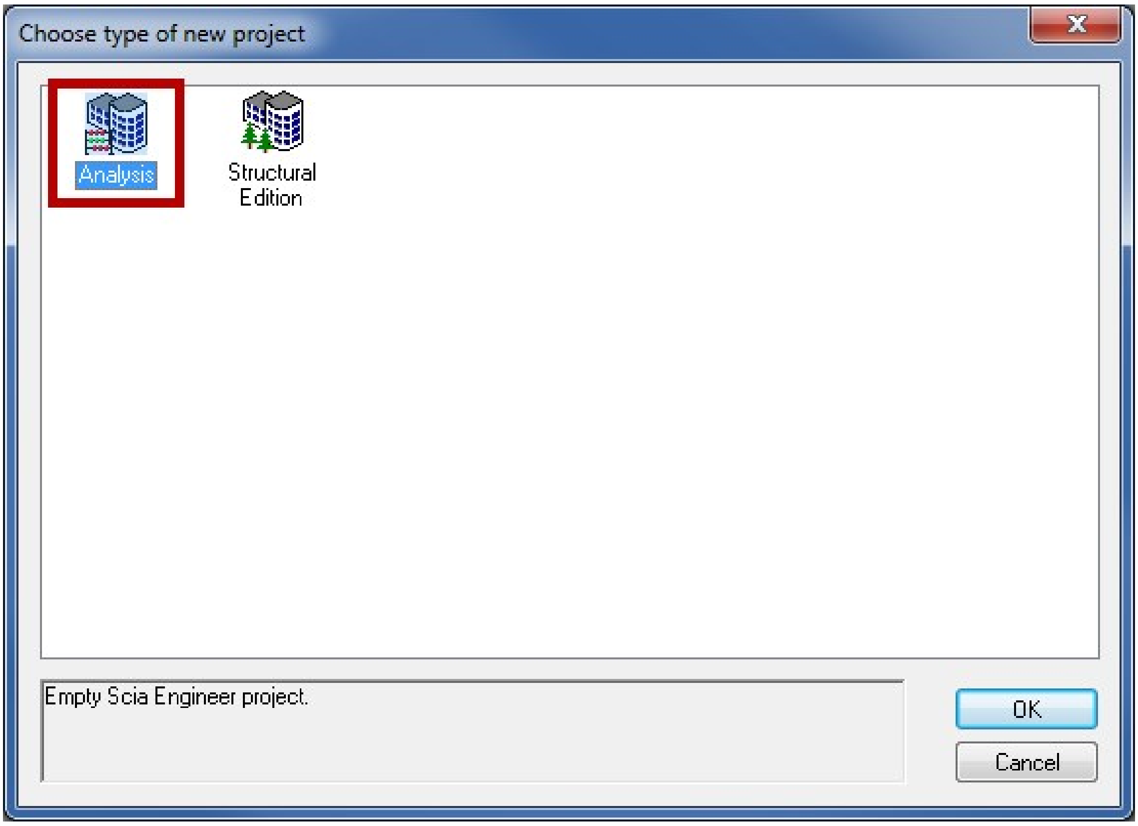


Structure modelling

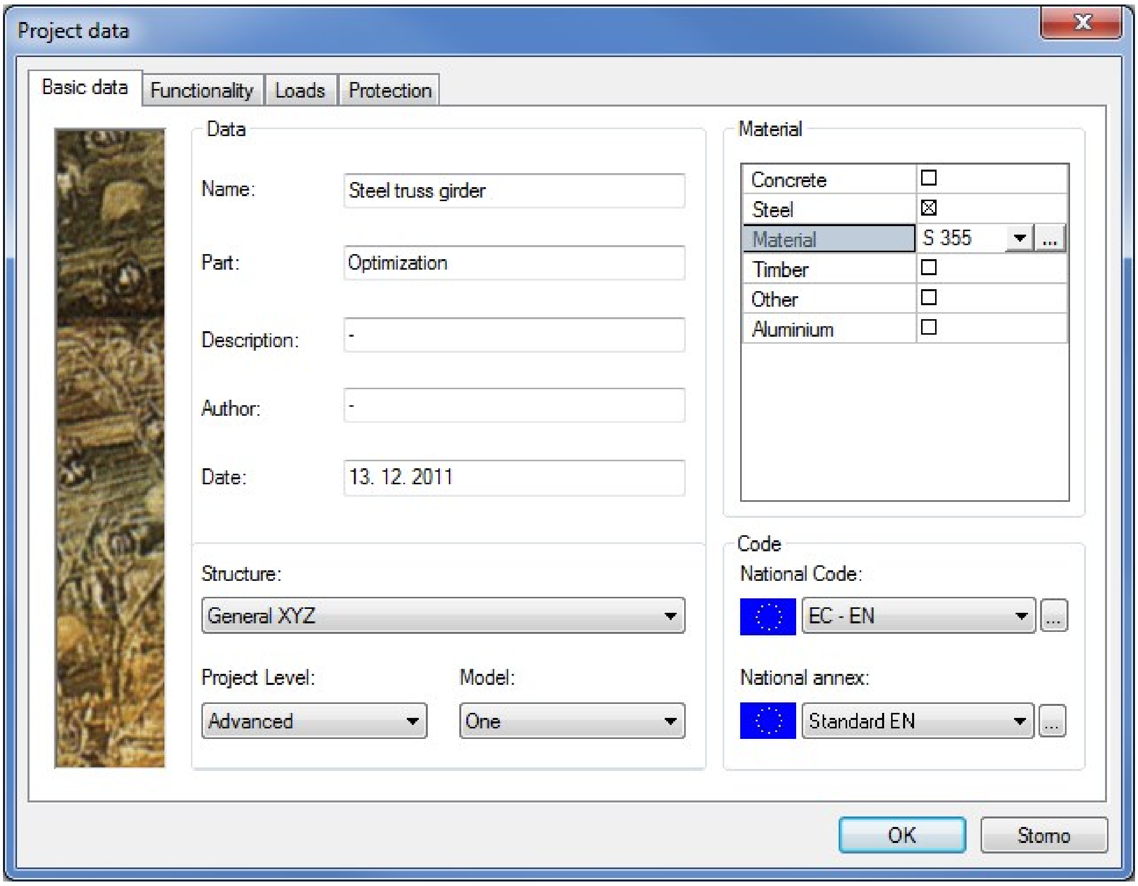
Starting the project in SCIA Engineer

First of all the standard SCIA Engineer project must be prepared. Therefore, create a model of a structure with all the necessary aspects of the future optimization taken into account.

Run SCIA Engineer and create a new project of the Analysis type.



The basic Project data can be filled in according to the picture below:



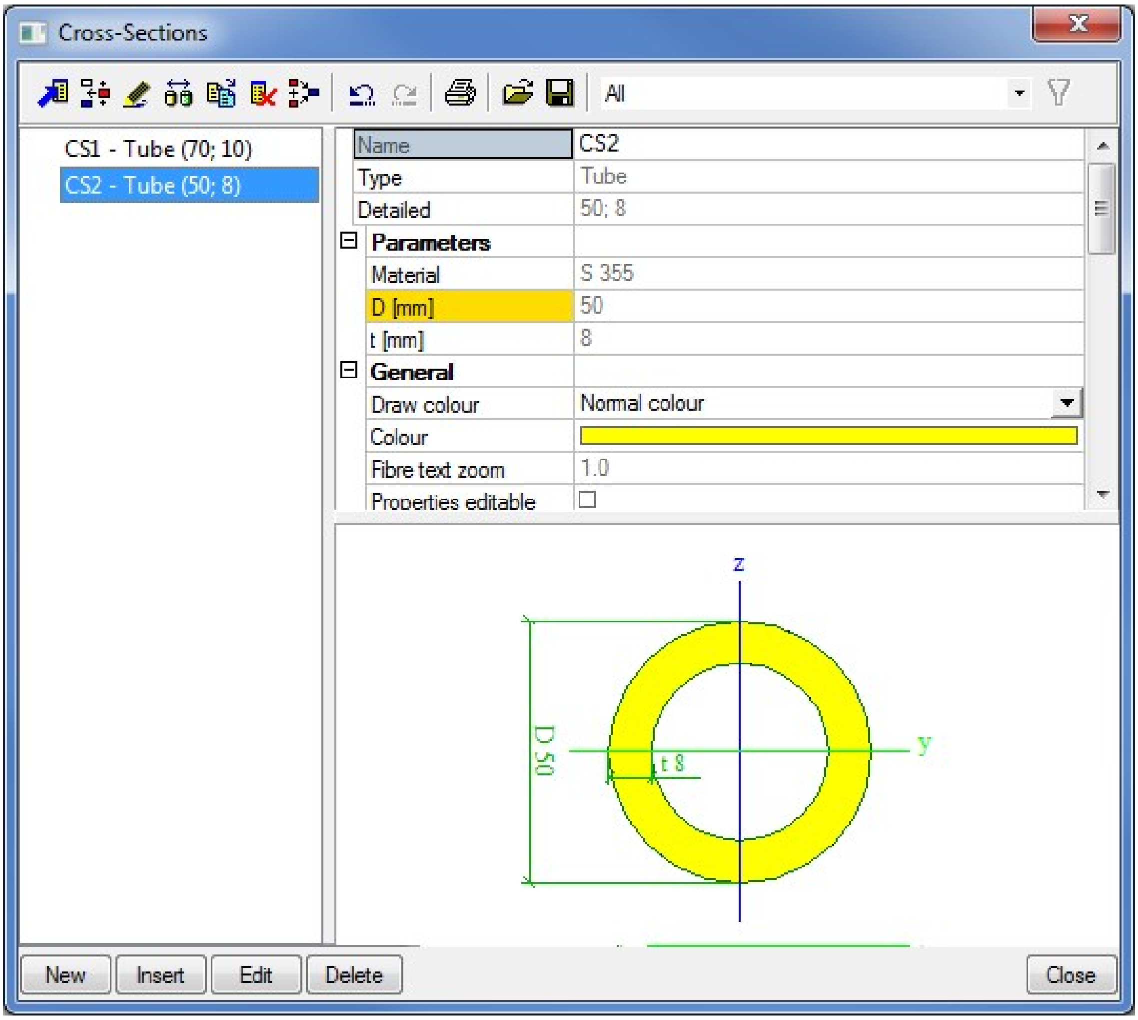
Although the structure is planar, the structure type is set to General XYZ to keep it more general. The project level is set to Advanced, which is a generally recommended setting. The code of this particular project is set to EC-EN with the standard EN annex.

Press OK to confirm the settings and to open a blank project.

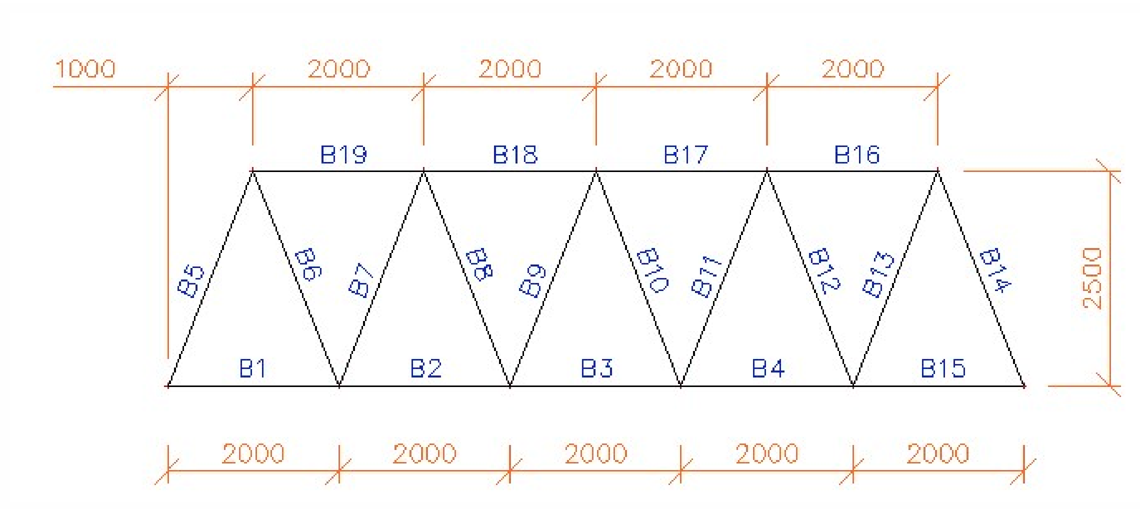
Model the truss girder with 1D members. First, you will be asked to select cross-sections for the current project. Add two tubular cross-sections with the dimensions:

CS1:D = 70 mm; t = 10 mm

CS2:D = 50 mm, t = 8 mm

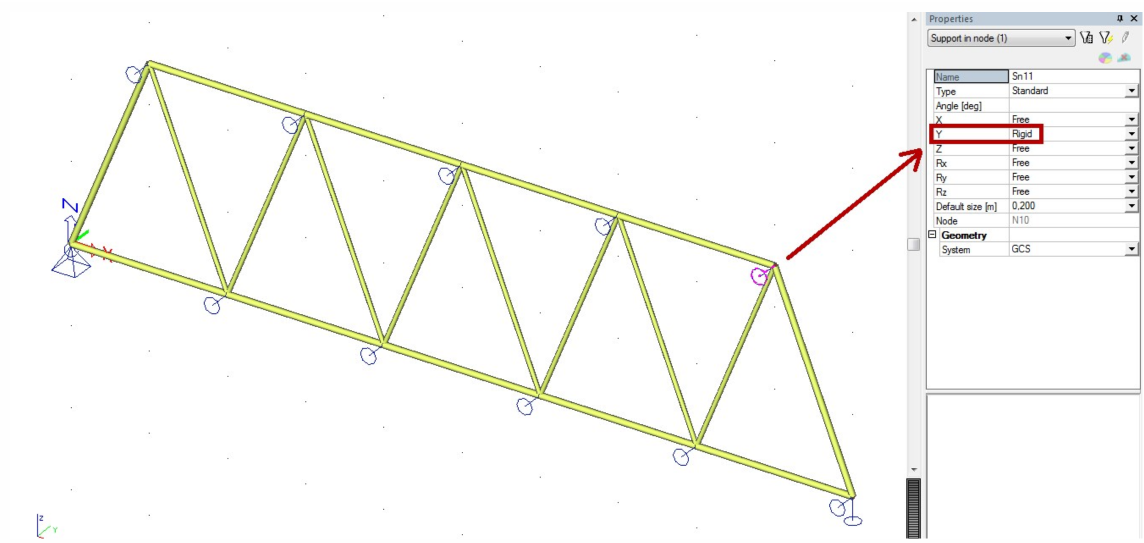


Create the structure according to the following scheme and table:

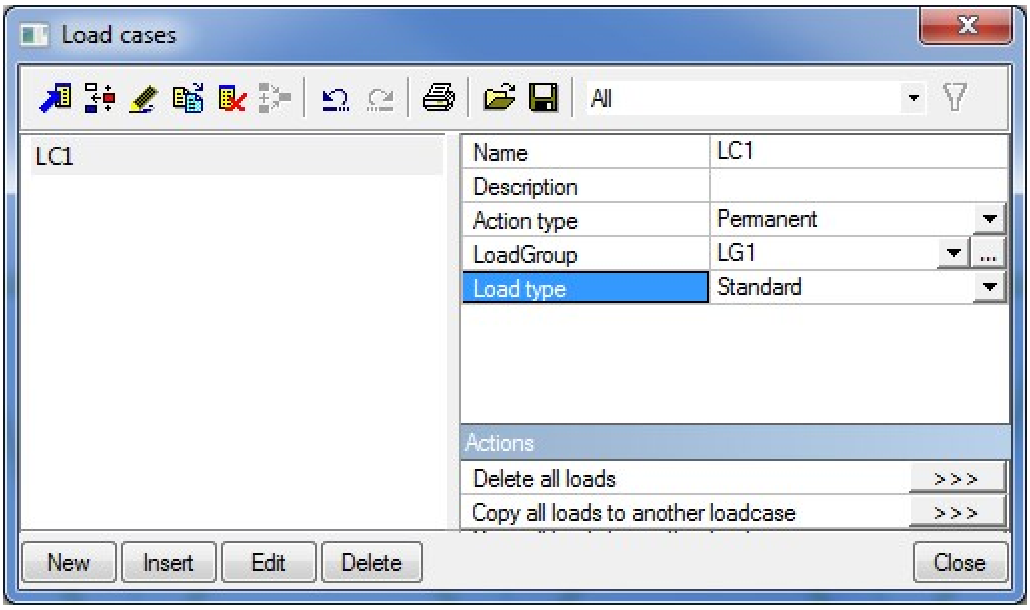


Name	CrossSection	Material	Length [m]	Shape	Beg. node	End node	Type	FEM type
B1	CS1 - Tube (70; 10)	S 355	2,000	Line	N1	N2	general (0)	standard
B2	CS1 - Tube (70; 10)	S 355	2,000	Line	N2	N3	general (0)	standard
B3	CS1 - Tube (70; 10)	S 355	2,000	Line	N3	N4	general (0)	standard
B4	CS1 - Tube (70; 10)	S 355	2,000	Line	N4	N5	general (0)	standard
B5	CS1 - Tube (70; 10)	S 355	2,693	Line	N1	N6	general (0)	standard
B6	CS2 - Tube (50; 8)	S 355	2,693	Line	N6	N2	general (0)	standard
B7	CS2 - Tube (50; 8)	S 355	2,693	Line	N2	N7	general (0)	standard
B8	CS2 - Tube (50; 8)	S 355	2,693	Line	N7	N3	general (0)	standard
B9	CS2 - Tube (50; 8)	S 355	2,693	Line	N3	N8	general (0)	standard
B10	CS2 - Tube (50; 8)	S 355	2,693	Line	N8	N4	general (0)	standard
B11	CS2 - Tube (50; 8)	S 355	2,693	Line	N4	N9	general (0)	standard
B12	CS2 - Tube (50; 8)	S 355	2,693	Line	N9	N5	general (0)	standard
B13	CS2 - Tube (50; 8)	S 355	2,693	Line	N5	N10	general (0)	standard
B14	CS1 - Tube (70; 10)	S 355	2,693	Line	N10	N11	general (0)	standard
B15	CS1 - Tube (70; 10)	S 355	2,000	Line	N11	N5	general (0)	standard
B16	CS1 - Tube (70; 10)	S 355	2,000	Line	N10	N9	general (0)	standard
B17	CS1 - Tube (70; 10)	S 355	2,000	Line	N9	N8	general (0)	standard
B18	CS1 - Tube (70; 10)	S 355	2,000	Line	N8	N7	general (0)	standard
B19	CS1 - Tube (70; 10)	S 355	2,000	Line	N7	N6	general (0)	standard

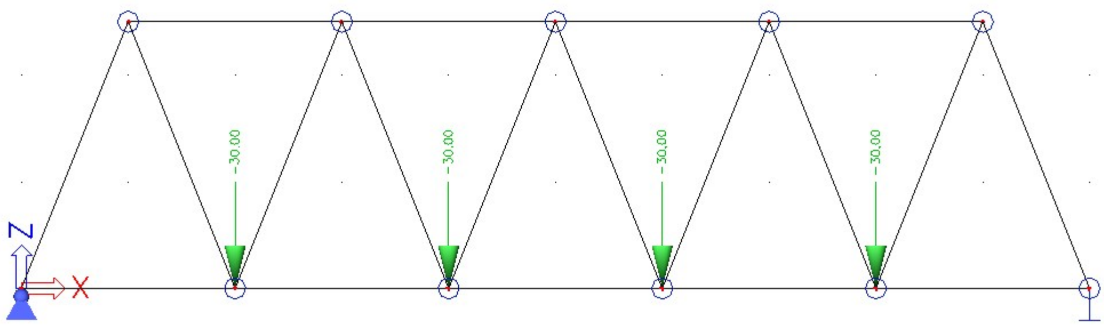
The truss as a whole structure is simply supported. To restrain the lateral displacements, supports in all nodes must be defined to prevent from the deformation in the Y direction.



When the structure is modelled, loads have to be specified as well. The truss girder is subjected to a simple load case with vertical point forces in the bottom nodes. Create a new load case with the action type Permanent and load type Standard.

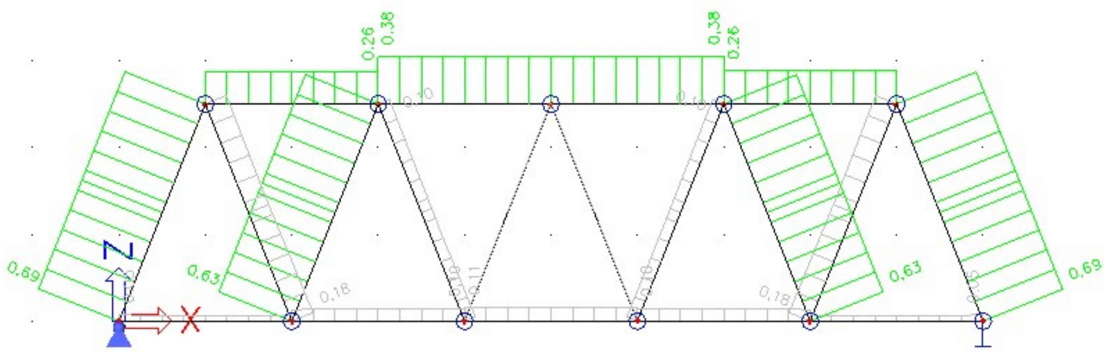


This load case is represented by 4 point forces with the magnitude of -30 kN in the Z axis direction (the load is going downwards).

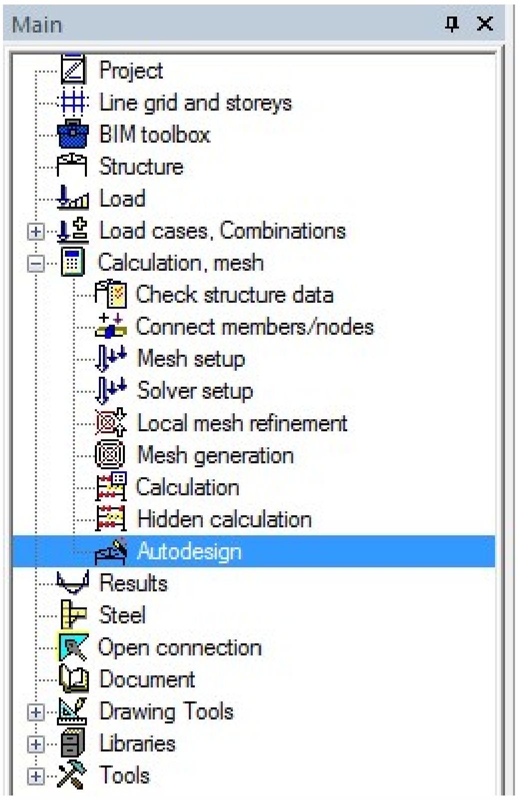


Calculation and Autodesign

The structure is calculated using standard functions of SCIA Engineer first. To see utilization of profiles, a unity check can be performed in Steel service, see next picture.



Maximal unity check value 0,69 shows, profiles are too thick. In SCIA Engineer we can use very efficient tool to design cross-sections, resulting in a good utilization of individual members under a certain load. The function is called Autodesign and it can be found under Calculation, mesh group in the Main tree.

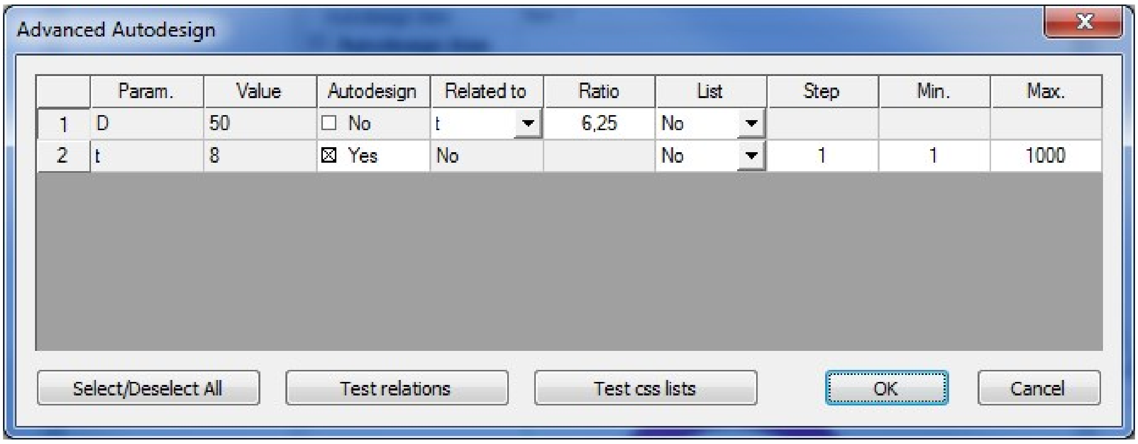


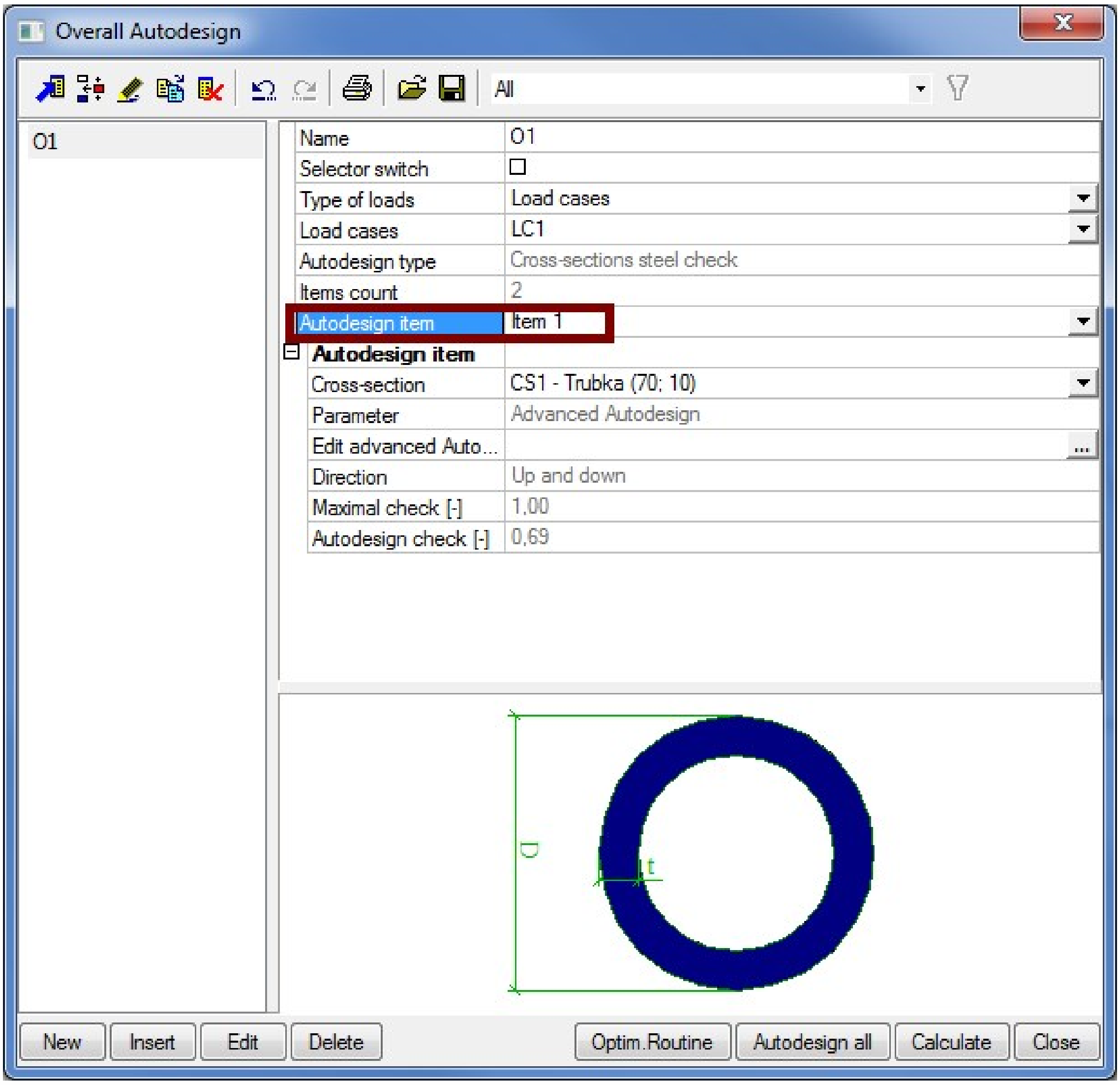
Autodesign can be used for various purposes. The Cross-section steel check will be used in this case. This Autodesign function finds an optimal cross-section with respect to the unity check for all members with this cross-section. However, as the change of a profile in a statically indeterminate structure affects the internal forces, the project has to be recalculated. For new internal foces, after recalculation, we can run Autodesign again. The user can do those two steps several times to reach proper cross-section design.

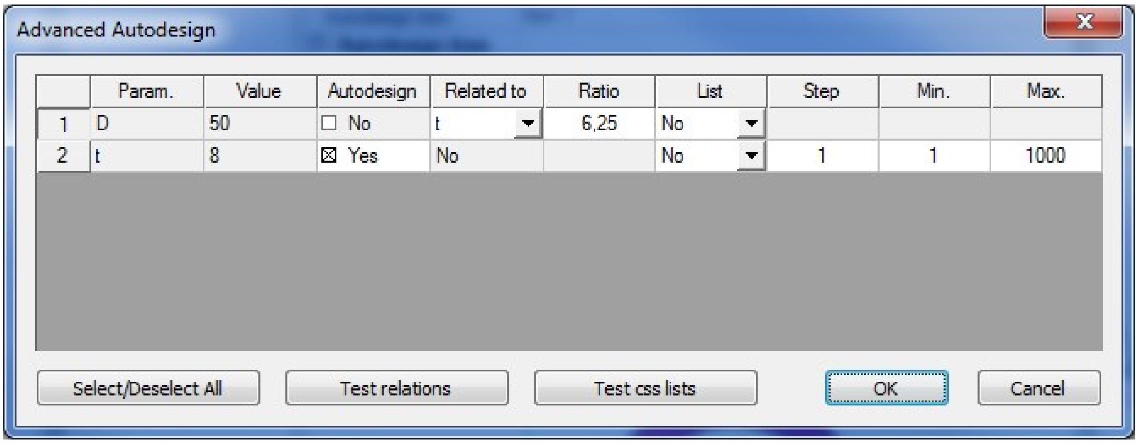
Autodesign is also integrated in optimization loop s of EOT. It means, during iterative search of optimal structure geometry, the program will make design of proper cross-sections. It means, in each iterative step, both geometry and cross-sections will be improved at the same time.

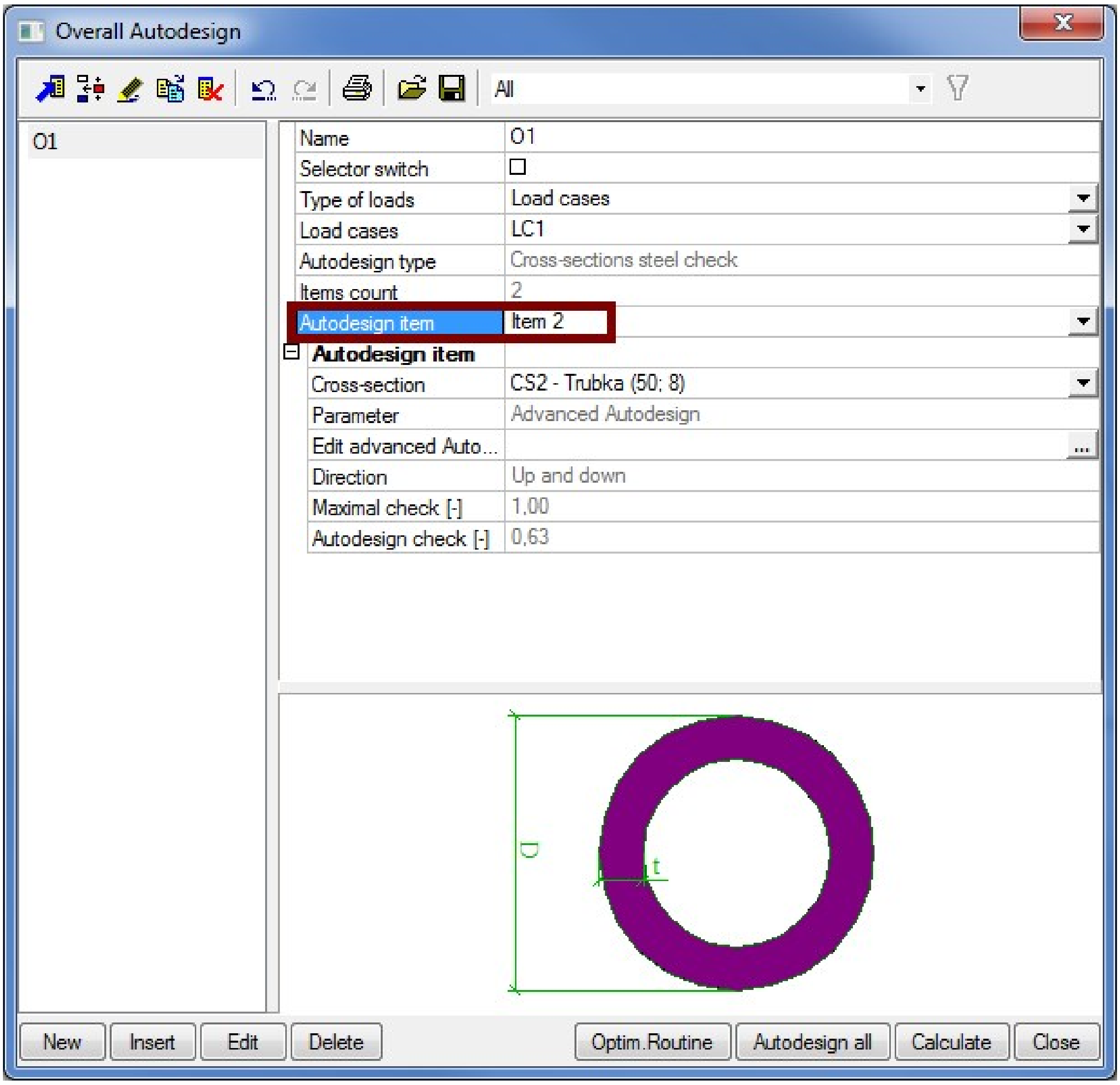
Prepare a new entry (called O1) in the Overall Autodesign library. Select both cross-sections (CS1 and CS2) in the selection dialogue. It means that the items count will be 2.

Each cross-section has got two dimensions in their properties – thickness t and diameter D. Both dimensions can be changed upwards and downwards in order to search for the optimum. To optimise both variables in one run, we will keep a fixed ratio between D and t. Advanced Autodesign is used for this.


Advanced Autodesign
Param. Value Autodesign Related to Ratio List Step Min. Max.
1 D 50 No t 6,25 No
2 t 8 Yes No No 1 1 1000
Select/Deselect All
Test relations
Test css lists
OK
Cancel


Overall Autodesign
All
O1
Name O1
Selector switch
Type of loads Load cases
Load cases LC1
Autodesign type Cross-sections steel check
Items count 2
Autodesign item Item 1
Autodesign item
Cross-section CS1 - Trubka (70; 10)
Parameter Advanced Autodesign
Edit advanced Auto...
Direction Up and down
Maximal check [-] 1,00
Autodesign check [-] 0,69
D
t
New
Insert
Edit
Delete
Optim.Routine
Autodesign all
Calculate
Close





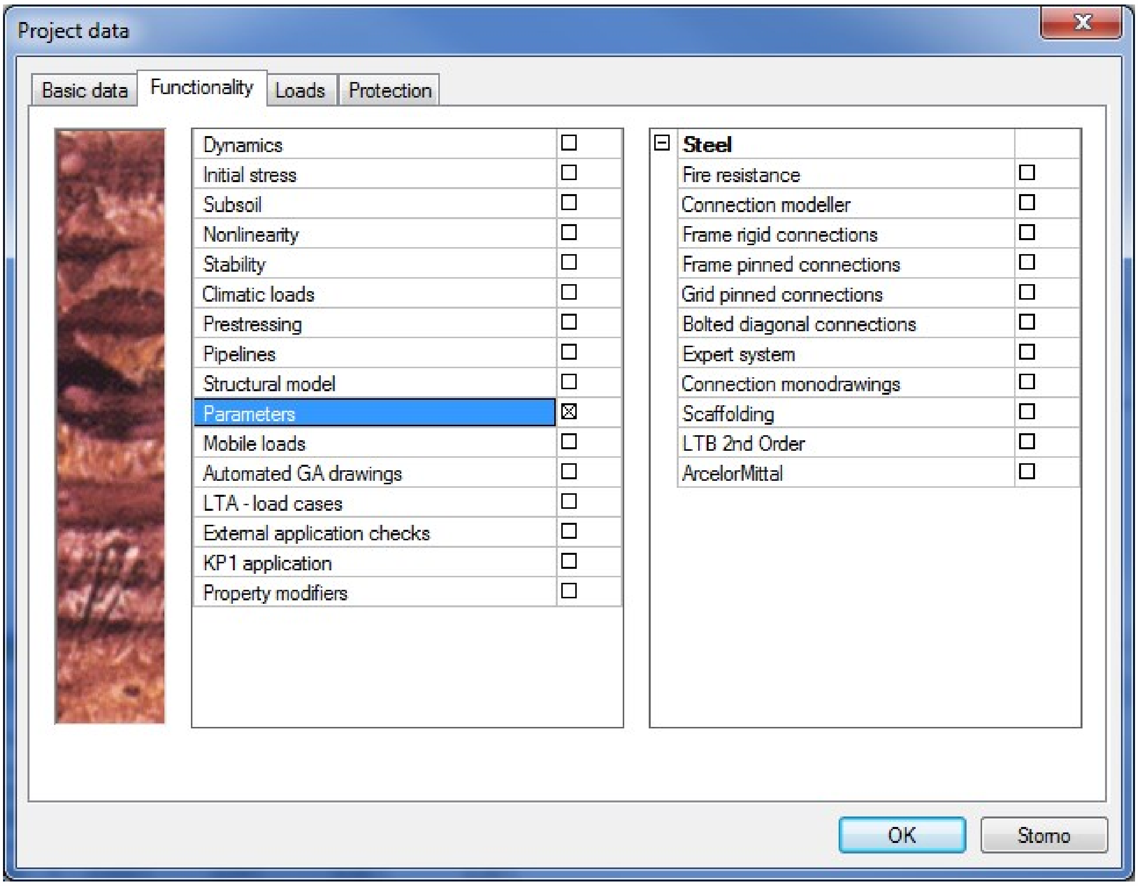
It is not needed to do the Autodesign at this moment. EOT application will use this stored setting afterwards.

Parameters

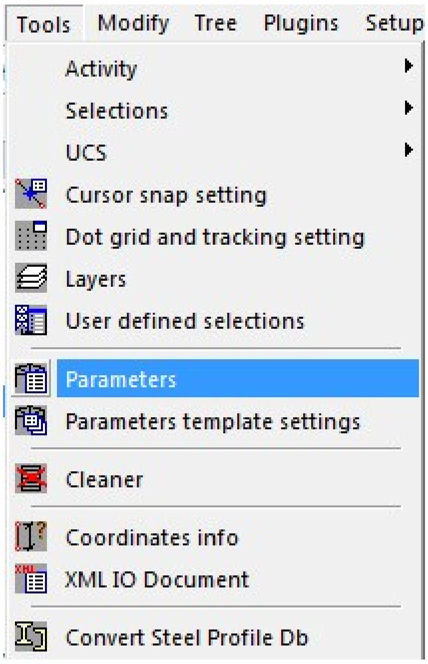
The optimization is based on parameters. SCIA Engineer allows the user to prepare lot of different kinds of parameters, which can be assigned to various entities and/or properties.

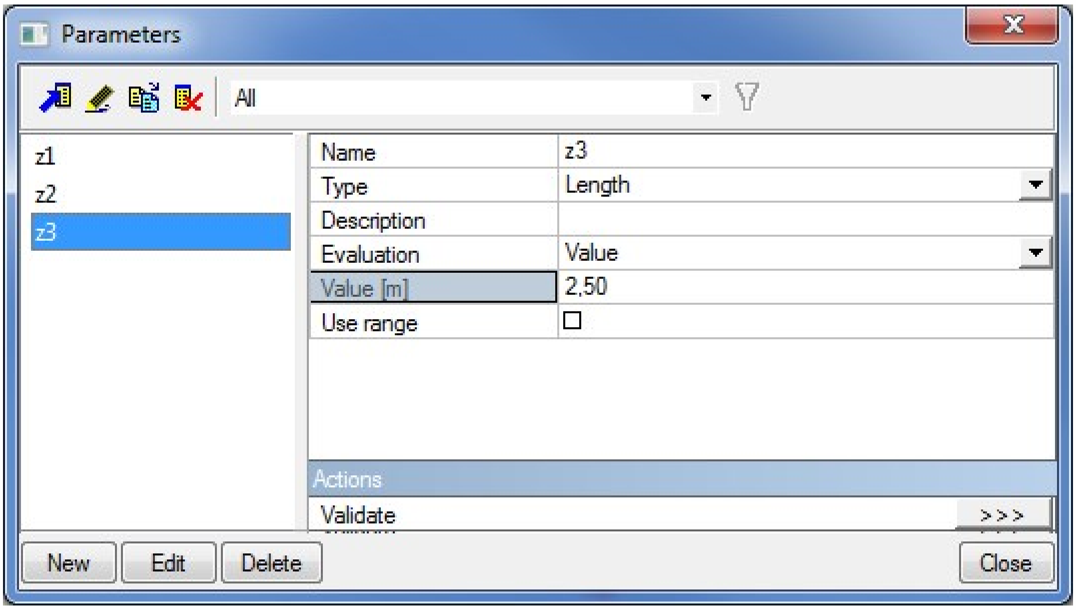
In this particular example we want to adapt the shape of the upper chord to get the minimum mass of the structure. This means that some nodes will change their positions (z-coordinates). Therefore, we will to make a set of parameters and we will assign them to properties of nodes.

To make parameters available, we have to switch them on in functionality setting:



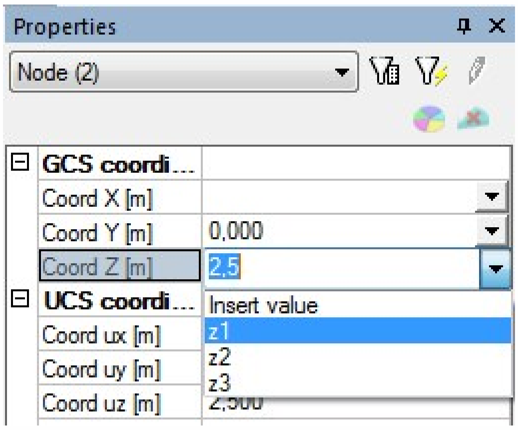
Then, let us open parameters library under Tools > Parameters. New items can be added with the button New in the top left or the bottom left corner (both these buttons do the same).



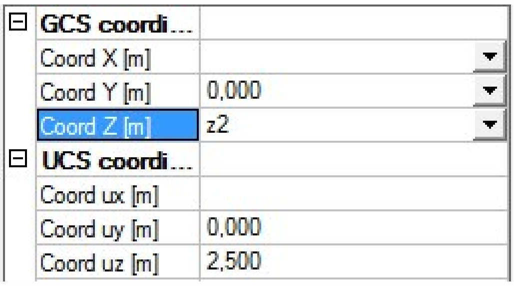


Prepare three parameters which will specify the Z coordinate of nodes of the upper chord. As we need a symmetric structure three parameters will be sufficient. The type of parameters is set to Length, Evaluation to Value and the default value is 2.5 m at the moment.

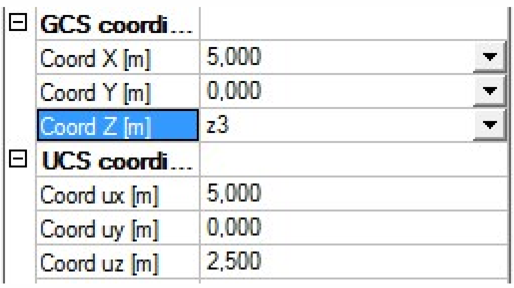
Assigning parameter to a node is simple. Just select a node and then, assign the parameter to the value of this property of selected node.



Select the outside nodes N6 and N10 and change their Coord Z to z1.



Select the inner nodes N7 and N9 and change their Coord Z to z2.



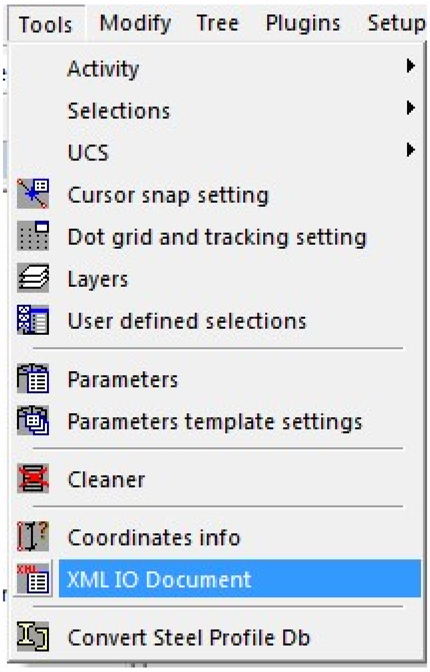
Select the the middle node N8 and change its Coord Z to z3.

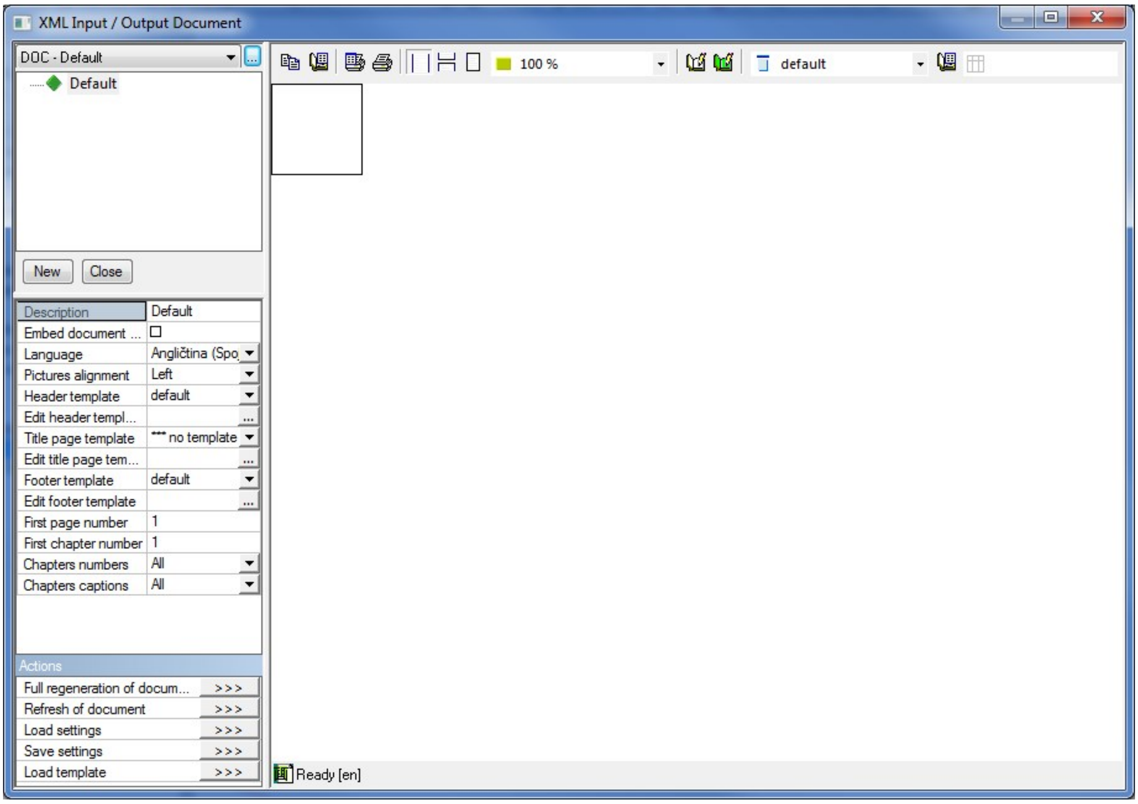
When the value of particular parameter is changed, the structure reacts immediately. However, let us assign this job to EOT. It shall calculate the correct value for each of the parameters.

XML documents

SCIA Engineer Optimizer is an external application and the optimization process is running “outside” SCIA Engineer. We need to transfer the necessary information between these two applications. Format XML (Extensible Markup Language) is used for this task.

SCIA Engineer has a tool for creating XML documents in a similar way as the basic SCIA Engineer output document. Go to Tools > XML IO Document and a new window appears.



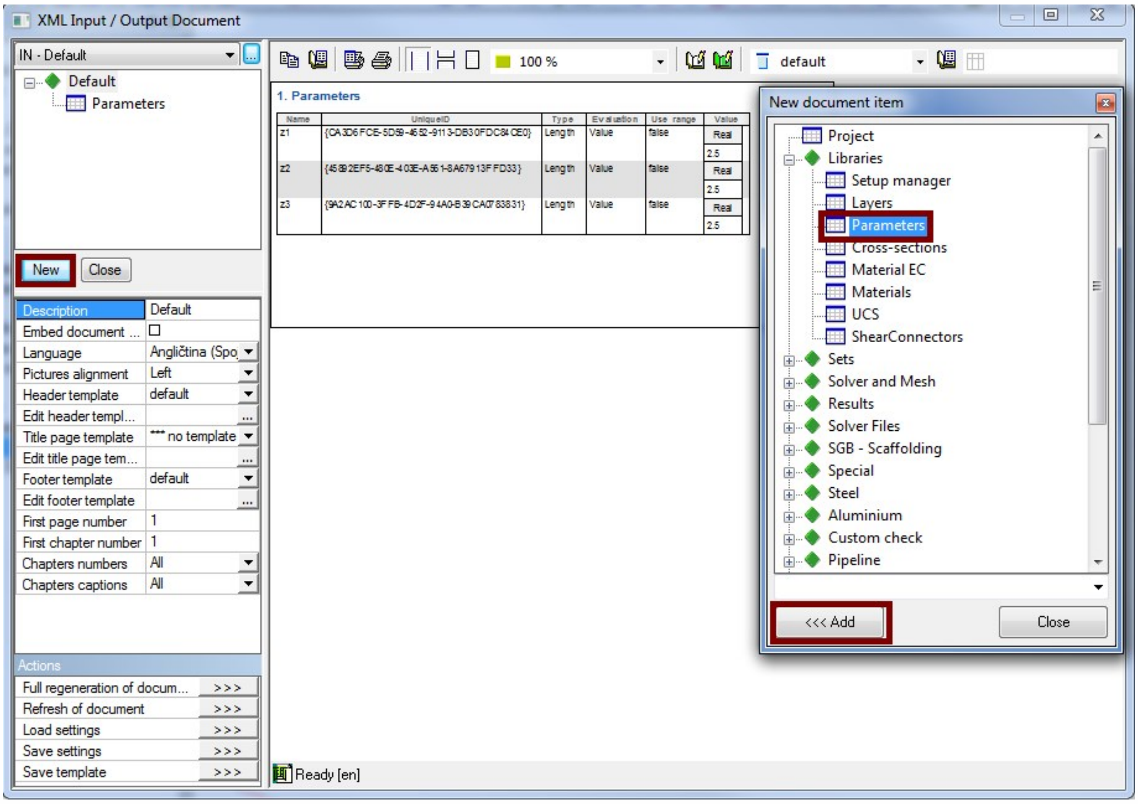


If you open the XML document for the first time, an option to open the Document template in *.TDX format appears. You can skip this if you don't have any XML template. Hit Cancel button.

Two documents have to be created for each project - one with input parameters and second one with output parameters. These two documents differ in their content which is described below.

Input XML

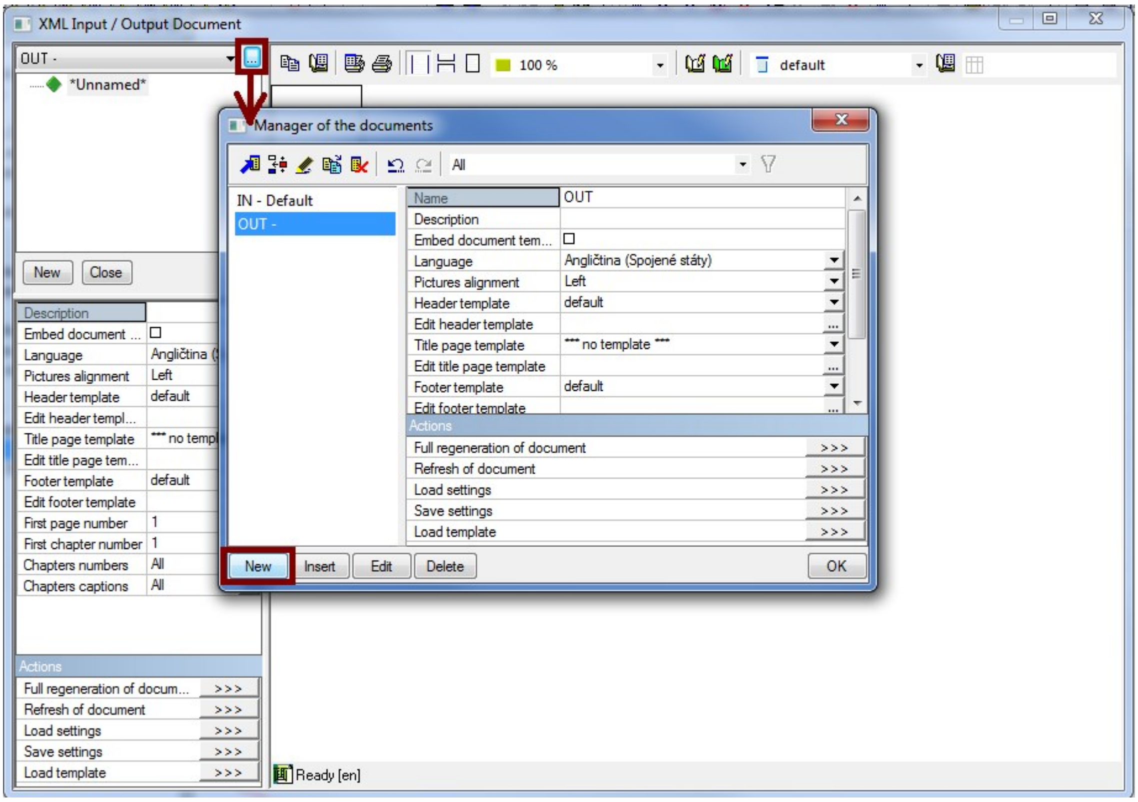
The input document has to include the table of parameters only. Click on the New button on the top left corner and select Parameters table from the Libraries group.



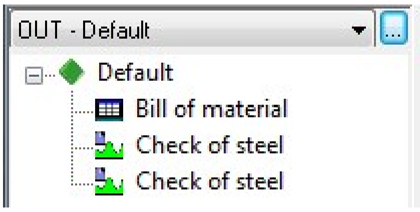
Output XML

The output document consists of more tables, namely the Bill of material (because mass optimization is the basic task) and Checks of steel (because we want to design the structure as well).

Click the button with three dots at the top and insert a new document.



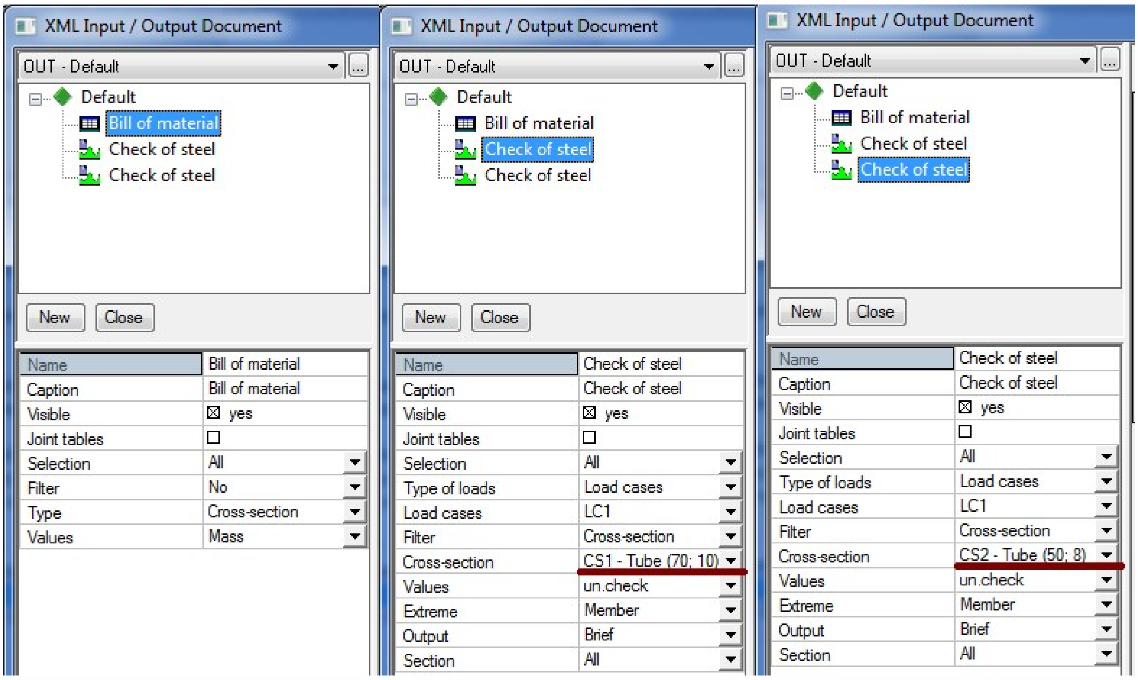
Click on the New button in the top left corner and add the desired tables:



Bill of material from the Results group

Check of steel from the Steel group (this has to be input twice – for each cross-section)

Below you can see the settings for each table included in the document. Note that the check of steel is filtered by cross-section and the first and the second item differ in this setting only.



Refresh the document to see if all the data are correctly loaded and close the window by the red cross in the top right corner.

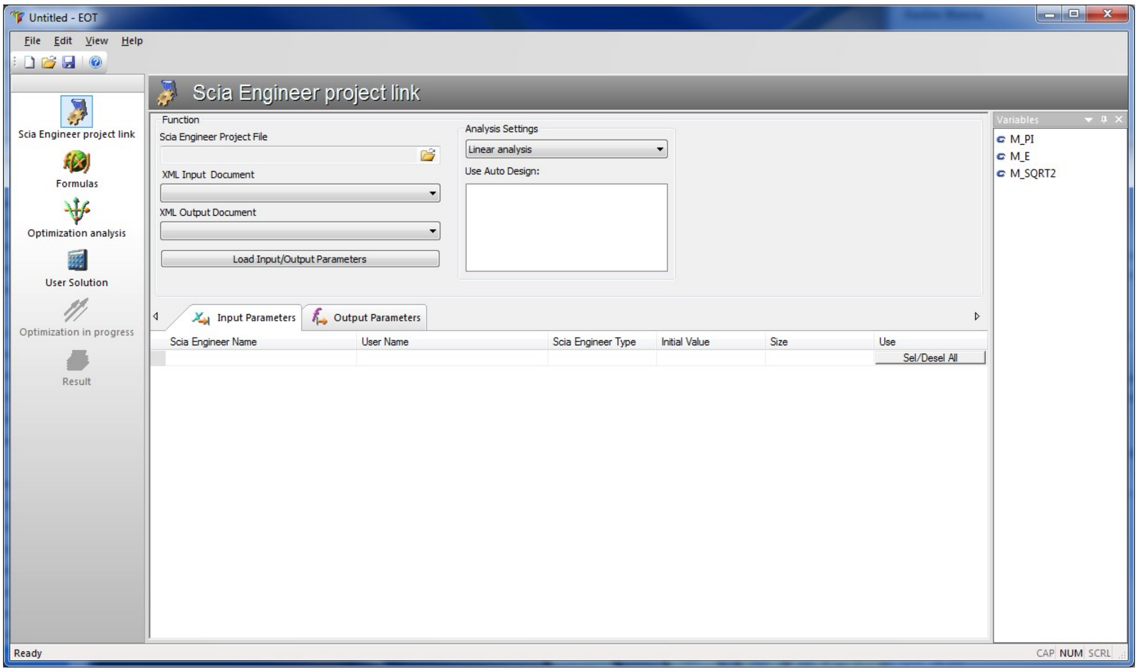
> It is wise to type a name of both documents so that they could be better recognized later.
> "IN" and "OUT" is the simplest possibility and it works well.

Optimizing tool

SCIA Engineer Optimizer is a part of standard setup version. It means that it is installed on your computer together with SCIA Engineer. You will find it in the particular folder under Start > All programs > SCIA Engineer, or you may have an icon on your desktop. However, Optimizer can be launched also from the program files folder, e.g.:

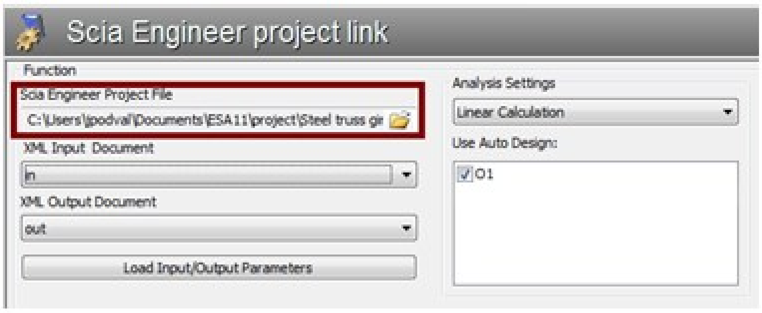
c:\Program Files\SCIA\Engineer2011.0\Eot.exe.

The user interface looks similar to SCIA Engineer. There is the main tree on the left with 6 functions that manage everything from model loading to results checking. These 6 steps will be described in the following chapters of this tutorial. There is also a special window with variables on the right and a few icons on top. These icons can be used to create a new EOT project, open an existing project or save the current project.



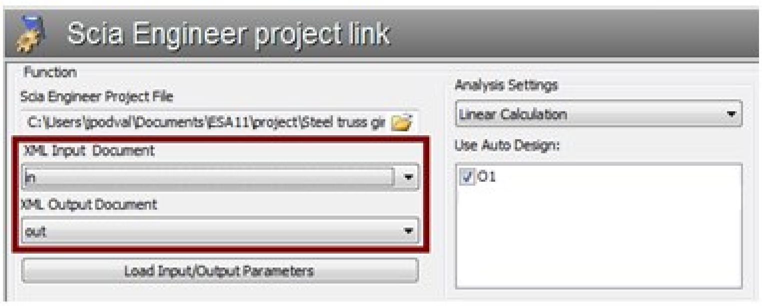
SCIA Engineer project link

In the first step you have to link EOT with a project file made in SCIA Engineer. At the top there is a field for the path to the *.esa file. Use the yellow "open folder" button to open an explorer and search for the project.

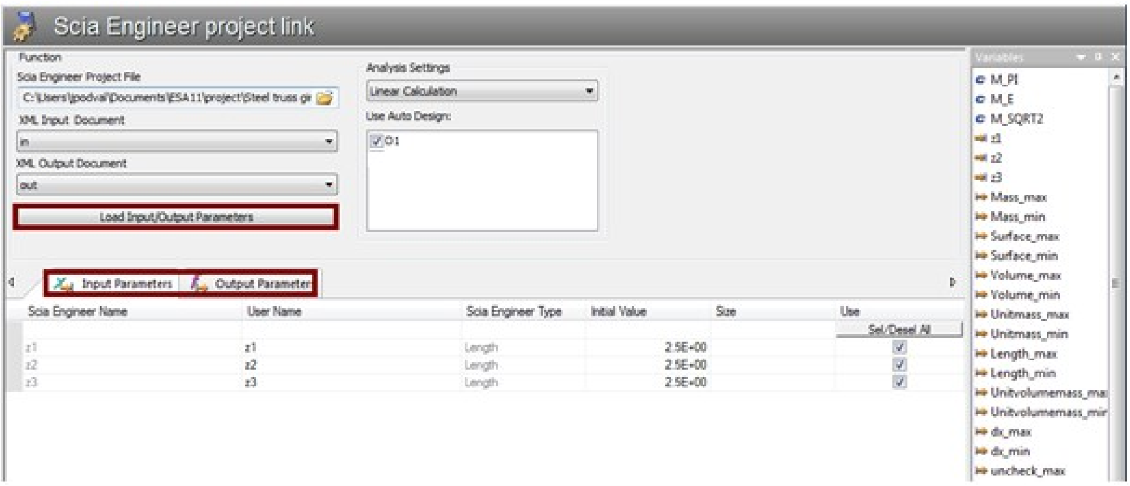


If necessary wait a few second to initiate the SCIA Engineer project file.

Once the project is loaded, the input and output documents are ready to be selected from the list of available XML documents. Usually, only the output document has to be switched as it is the second one from the offer.

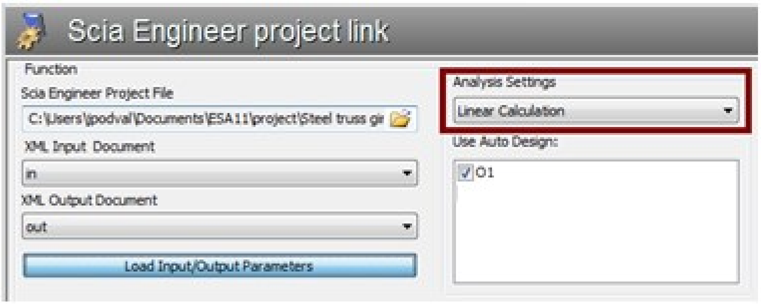


Select the right input and output document and click the button below called Load Input/Output Parameters. Wait a few seconds again until all parameters are displayed in the table (and in the right panel as well).

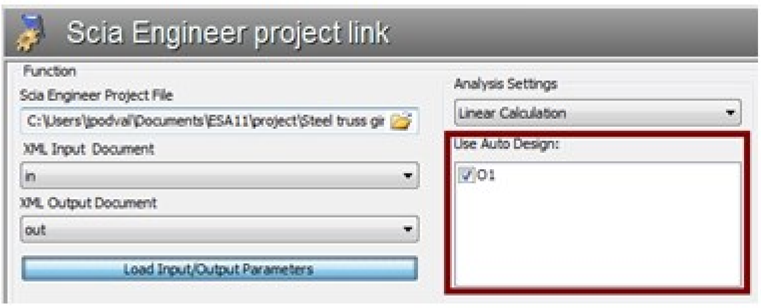


At this point new files are automatically generated in the folder where the project is saved. The files are related to the input and output XML documents and two files are created for both of them*_in.xml*_out.xml*_in.xml.def*_out.xml.def

The middle part of the screen shows also the available types of Analysis Settings. Leave the default Linear Calculation option selected in the combo-box.

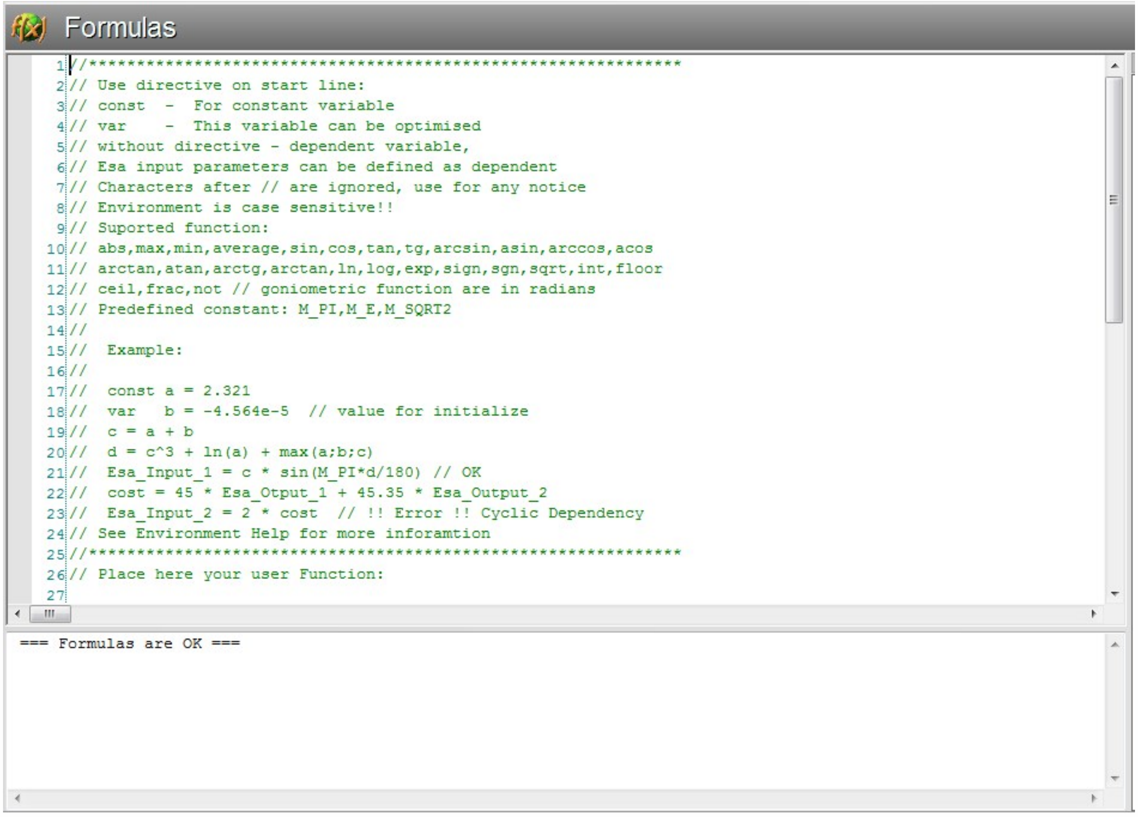


If any autodesign item has been created in the used SCIA Engineer project it will be listed in the white box here. Tick the items you want to be included in the optimization procedure. We want to design all cross-sections in SCIA Engineer, thus tick the O1 Autodesign.



Formulas

The SCIA Engineer Optimizer is a scientific tool where you don't have to fully rely on what have been programmed but where you can add your own formulas. At the beginning of the text editor for formulas there is a paragraph with hints and instructions about how this environment can be used. There is also a list of supported functions.

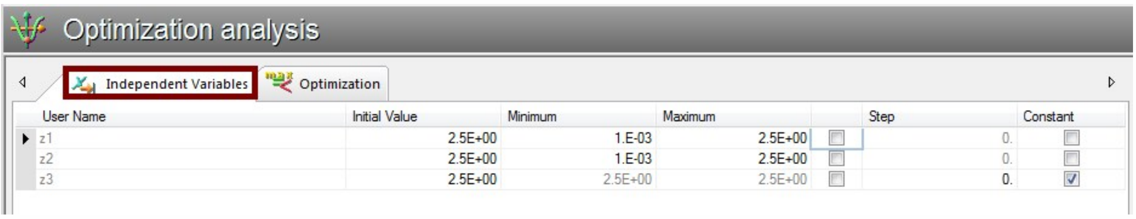


However, the whole step is optional. We won't be dealing with it in this tutorial.

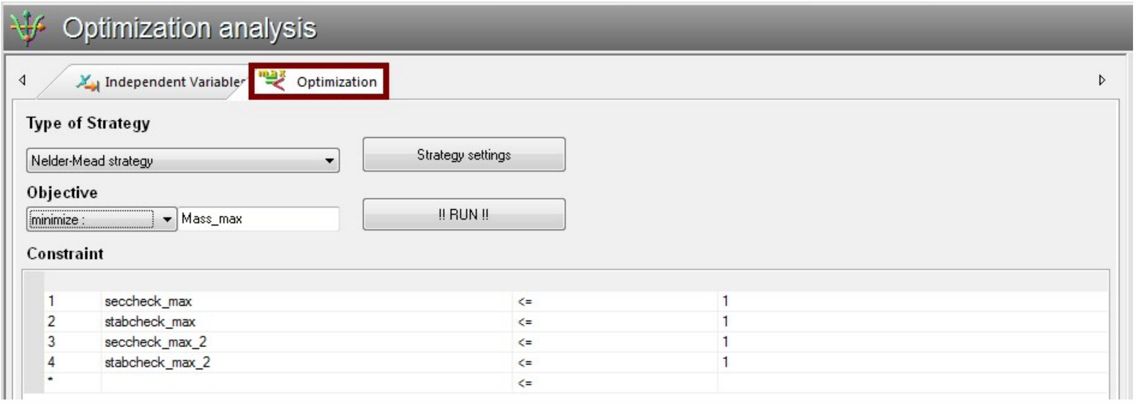
Optimization analysis

In the third step the optimization analysis is set. Variables, constraints and objective are selected and the type of strategy is chosen.

All independent variables ticked in the input parameters are displayed in the first tab. Please, pay attention to their minimum and maximum values to keep the design inside reasonable boundaries. Any variable can be set as constant; basically it is the same as omitting it. We decided to remain the top level of the girder, thus z3 is constant.



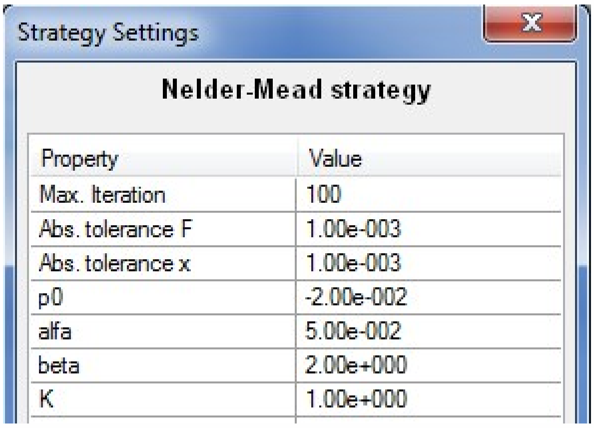
The Optimization tab requires more settings - all related to the analysis itself:



Type of strategy

There are 5 strategies implemented in EOT. Each strategy has its own specifications and is suitable for different examples. Sometimes it is necessary to test more strategies, but the detailed discussion about the selection of the strategies is out of scope of this tutorial. For our example select the heuristic method called Nelder –Mead strategy.

Strategy settings



This button enables advanced users to adjust the properties of the selected strategy. Leave the defaults.

Objective

The intention of the whole optimization is specified here. There are two extremes that can be determined – maximum or minimum. Type a variable name from the list of dependent variables (see the right stripe of the window) into the white field.

Constraint

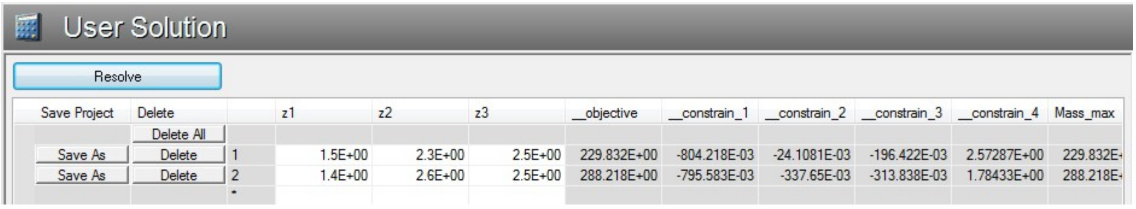
There may be some restrictions for the optimization. In our case we want to design the structure with respect to code regulations. The unity check of all members has to be lower than 1.0. There are two cross-sections and for both of them the section check and the stability check is considered. There are four constraints in total: the name is taken from the list of variables and the condition set is smaller or equal to 1.

The control in the bottom window tells us if all criteria are all right.

When ready, click button !! RUN !!

User Solution

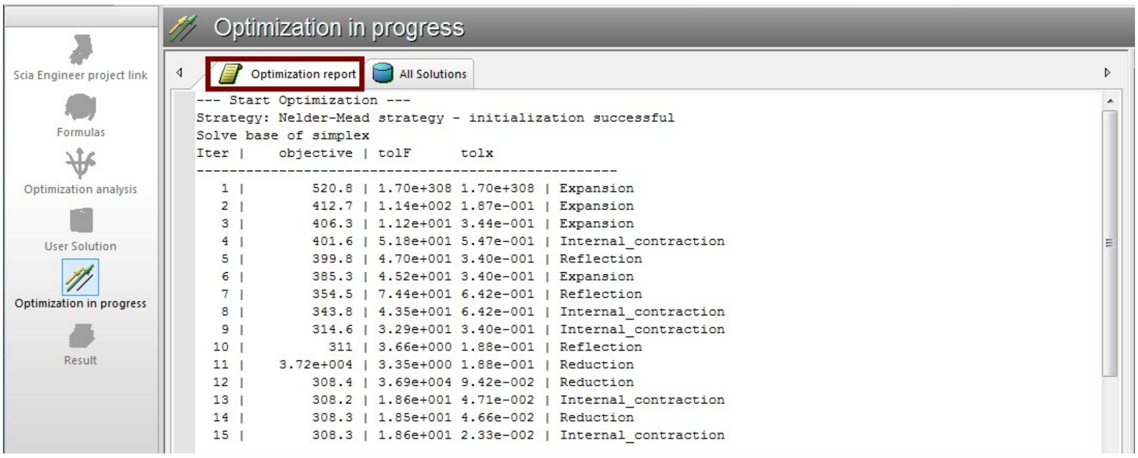
The user can use his own independent variables and calculate the objective and other dependant variables anytime before or after the optimization. Fill in the white fields in the table and click the Resolve button to run the calculation, if desired.



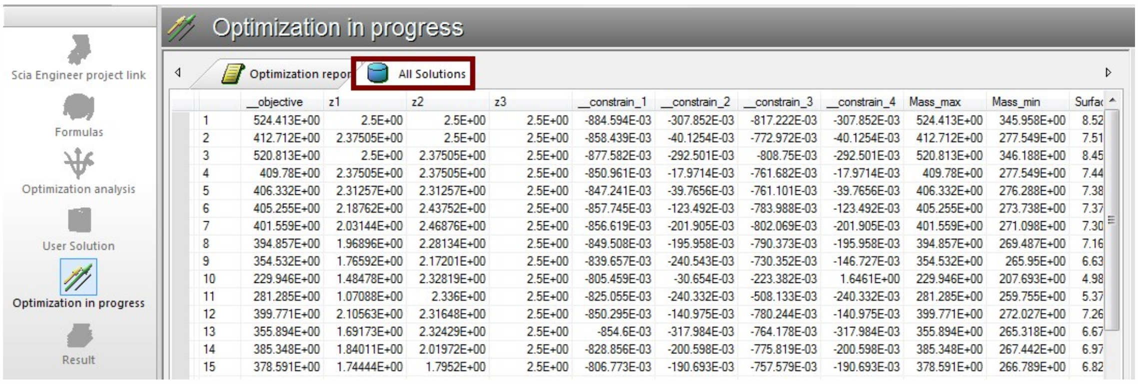
A typical advantage of this tool is when the results is for example 0,98 and you want to apply rounded values. In this case type 1,0 and resolve the structure with user solution.

Optimization in progress

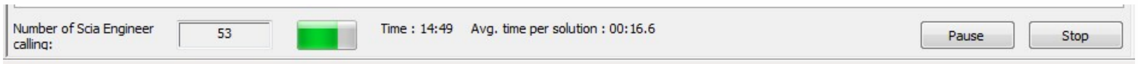
During the optimization only the current step in the left menu is available. The iterations can be monitored in the main window under the Optimization report tab.



There is also a second tab with All solutions the number of which increases during the calculation.



The bottom status bar shows the progress, total time, average time per one solution and there are also two important buttons for pausing or stopping the optimization.

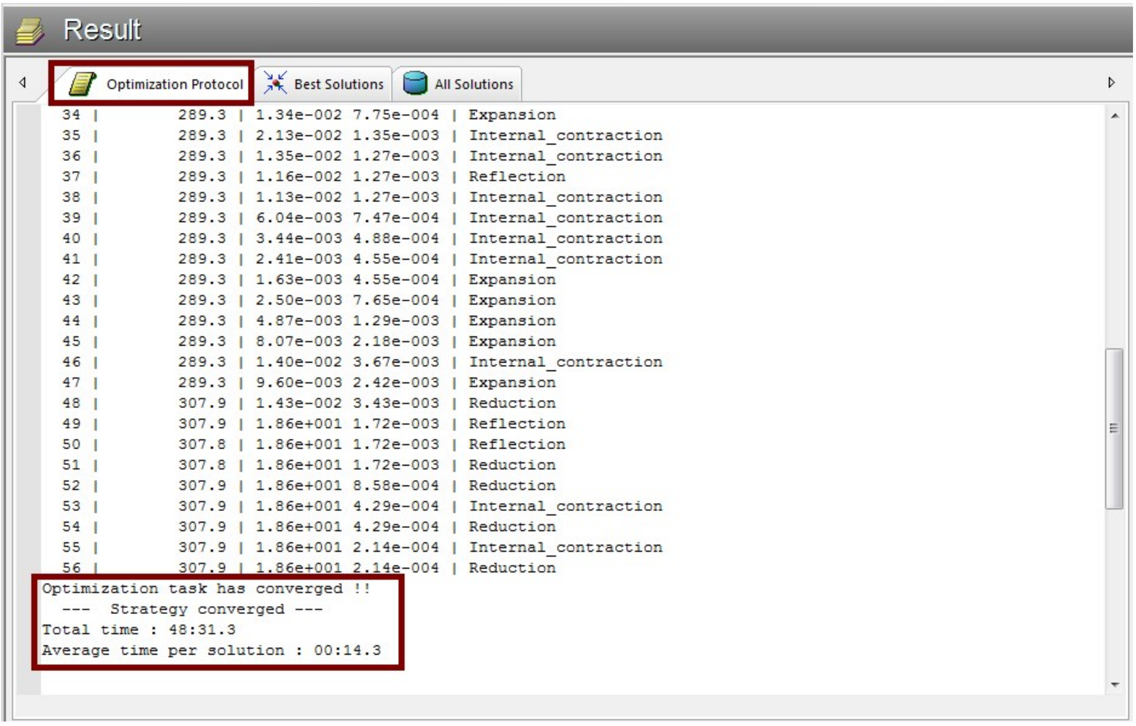


Result

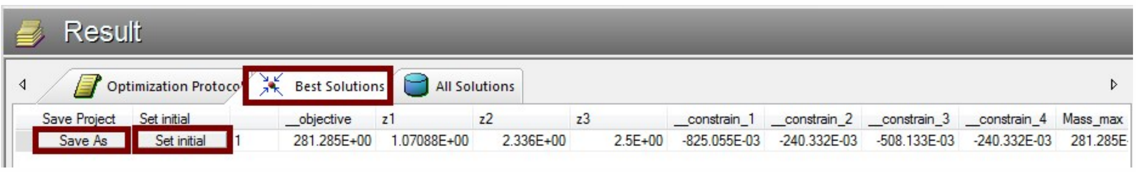
If the optimization is successful, no signal appears at the end.

There are three tabs in results which offer different outputs.

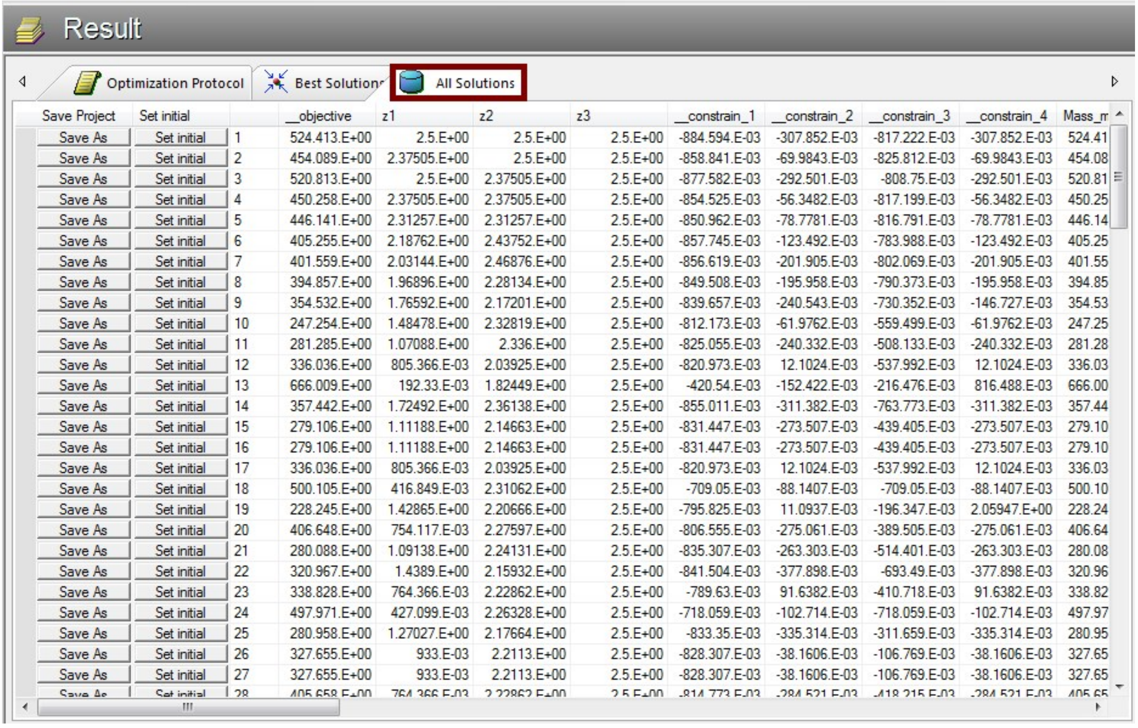
The Optimization Protocol shows information about iterations and you can study the progress of the optimization. At the end there is also a note about the completion and total calculation time.



Best Solutions tab offers one or more solutions that are considered to be the best. You can save such a solution as a SCIA Engineer project (Save As button) or set the calculated values as initial ones and run a new optimization (Set Initial button).

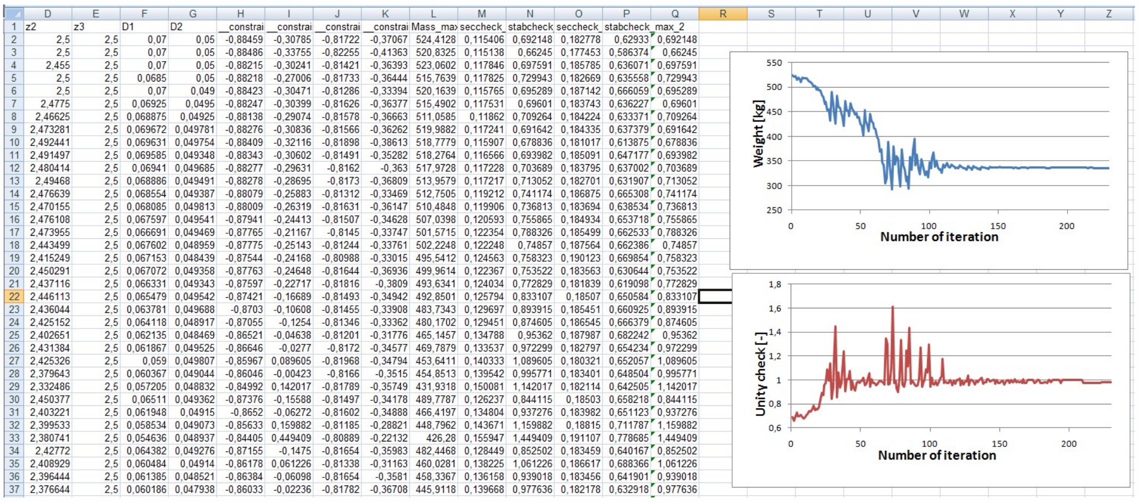


There are also All Solutions that have been found during the optimization. One can compare them or search for a solution which is not necessarily the best one but more suitable for the user. Any item can be also saved as a project or set as initial for the next optimization attempt.



Save Project	Set initial		_objective	z1	z2	z3	_constrain_1	_constrain_2	_constrain_3	_constrain_4	Mass_m
Save As	Set initial	1	524.413.E+00	2.5.E+00	2.5.E+00	2.5.E+00	-884.594.E-03	-307.852.E-03	-817.222.E-03	-307.852.E-03	524.41
Save As	Set initial	2	454.089.E+00	2.37505.E+00	2.5.E+00	2.5.E+00	-858.841.E-03	-69.9843.E-03	-825.812.E-03	-69.9843.E-03	454.08
Save As	Set initial	3	520.813.E+00	2.5.E+00	2.37505.E+00	2.5.E+00	-877.582.E-03	-292.501.E-03	-808.75.E-03	-292.501.E-03	520.81
Save As	Set initial	4	450.258.E+00	2.37505.E+00	2.37505.E+00	2.5.E+00	-854.525.E-03	-56.3482.E-03	-817.199.E-03	-56.3482.E-03	450.25
Save As	Set initial	5	446.141.E+00	2.31257.E+00	2.31257.E+00	2.5.E+00	-850.962.E-03	-78.7781.E-03	-816.791.E-03	-78.7781.E-03	446.14
Save As	Set initial	6	405.255.E+00	2.18762.E+00	2.43752.E+00	2.5.E+00	-857.745.E-03	-123.492.E-03	-783.988.E-03	-123.492.E-03	405.25
Save As	Set initial	7	401.559.E+00	2.03144.E+00	2.46876.E+00	2.5.E+00	-856.619.E-03	-201.905.E-03	-802.069.E-03	-201.905.E-03	401.55
Save As	Set initial	8	394.857.E+00	1.96896.E+00	2.28134.E+00	2.5.E+00	-849.508.E-03	-195.958.E-03	-790.373.E-03	-195.958.E-03	394.85
Save As	Set initial	9	354.532.E+00	1.76592.E+00	2.17201.E+00	2.5.E+00	-839.657.E-03	-240.543.E-03	-730.352.E-03	-146.727.E-03	354.53
Save As	Set initial	10	247.254.E+00	1.48478.E+00	2.32819.E+00	2.5.E+00	-812.173.E-03	-61.9762.E-03	-559.499.E-03	-61.9762.E-03	247.25
Save As	Set initial	11	281.285.E+00	1.07088.E+00	2.336.E+00	2.5.E+00	-825.055.E-03	-240.332.E-03	-508.133.E-03	-240.332.E-03	281.28
Save As	Set initial	12	336.036.E+00	805.366.E-03	2.03925.E+00	2.5.E+00	-820.973.E-03	12.1024.E-03	-537.992.E-03	12.1024.E-03	336.03
Save As	Set initial	13	666.009.E+00	192.33.E-03	1.82449.E+00	2.5.E+00	-420.54.E-03	-152.422.E-03	-216.476.E-03	816.488.E-03	666.00
Save As	Set initial	14	357.442.E+00	1.72492.E+00	2.36138.E+00	2.5.E+00	-855.011.E-03	-311.382.E-03	-763.773.E-03	-311.382.E-03	357.44
Save As	Set initial	15	279.106.E+00	1.11188.E+00	2.14663.E+00	2.5.E+00	-831.447.E-03	-273.507.E-03	-439.405.E-03	-273.507.E-03	279.10
Save As	Set initial	16	279.106.E+00	1.11188.E+00	2.14663.E+00	2.5.E+00	-831.447.E-03	-273.507.E-03	-439.405.E-03	-273.507.E-03	279.10
Save As	Set initial	17	336.036.E+00	805.366.E-03	2.03925.E+00	2.5.E+00	-820.973.E-03	12.1024.E-03	-537.992.E-03	12.1024.E-03	336.03
Save As	Set initial	18	500.105.E+00	416.849.E-03	2.31062.E+00	2.5.E+00	-709.05.E-03	-88.1407.E-03	-709.05.E-03	-88.1407.E-03	500.10
Save As	Set initial	19	228.245.E+00	1.42865.E+00	2.20666.E+00	2.5.E+00	-795.825.E-03	11.0937.E-03	-196.347.E-03	2.05947.E+00	228.24
Save As	Set initial	20	406.648.E+00	754.117.E-03	2.27597.E+00	2.5.E+00	-806.555.E-03	-275.061.E-03	-389.505.E-03	-275.061.E-03	406.64
Save As	Set initial	21	280.088.E+00	1.09138.E+00	2.24131.E+00	2.5.E+00	-835.307.E-03	-263.303.E-03	-514.401.E-03	-263.303.E-03	280.08
Save As	Set initial	22	320.967.E+00	1.4389.E+00	2.15932.E+00	2.5.E+00	-841.504.E-03	-377.898.E-03	-693.49.E-03	-377.898.E-03	320.96
Save As	Set initial	23	338.828.E+00	764.366.E-03	2.22862.E+00	2.5.E+00	-789.63.E-03	91.6382.E-03	-410.718.E-03	91.6382.E-03	338.82
Save As	Set initial	24	497.971.E+00	427.099.E-03	2.26328.E+00	2.5.E+00	-718.059.E-03	-102.714.E-03	-718.059.E-03	-102.714.E-03	497.97
Save As	Set initial	25	280.958.E+00	1.27027.E+00	2.17664.E+00	2.5.E+00	-833.35.E-03	-335.314.E-03	-311.659.E-03	-335.314.E-03	280.95
Save As	Set initial	26	327.655.E+00	933.E-03	2.2113.E+00	2.5.E+00	-828.307.E-03	-38.1606.E-03	-106.769.E-03	-38.1606.E-03	327.65
Save As	Set initial	27	327.655.E+00	933.E-03	2.2113.E+00	2.5.E+00	-828.307.E-03	-38.1606.E-03	-106.769.E-03	-38.1606.E-03	327.65

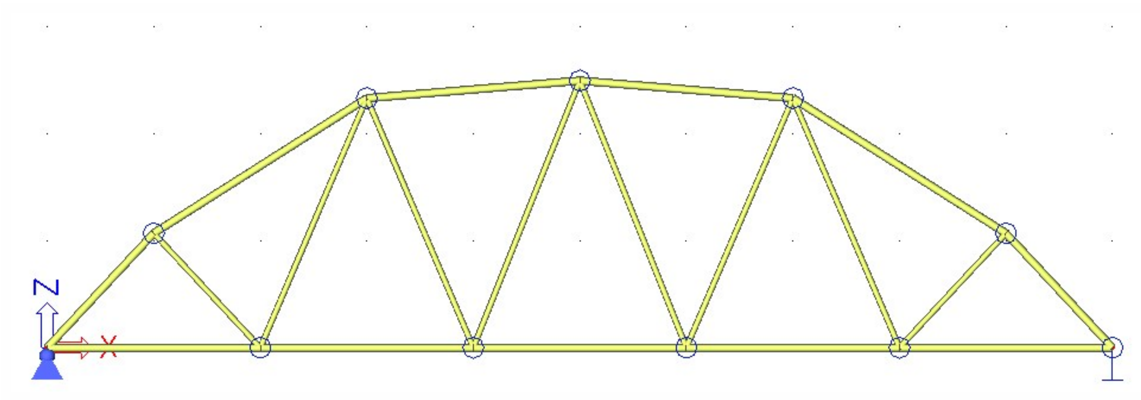
Three buttons in the bottom right corner enables the user to Save protocol (the same as you see on the screen, in *.txt format), Save tables as CVS or Export (results) to Excel. The last option opens Microsoft Excel with a new file containing three tabs – exactly the same as in Results of the EOT application. The advantage of such an export is that you can insert illustrative graphs to demonstrate the optimization process.



This result shows quite remarkable material savings, as the original mas 524 kg was finally reduced to 281,2 kg, only by means of optimization of geometry of the structure!

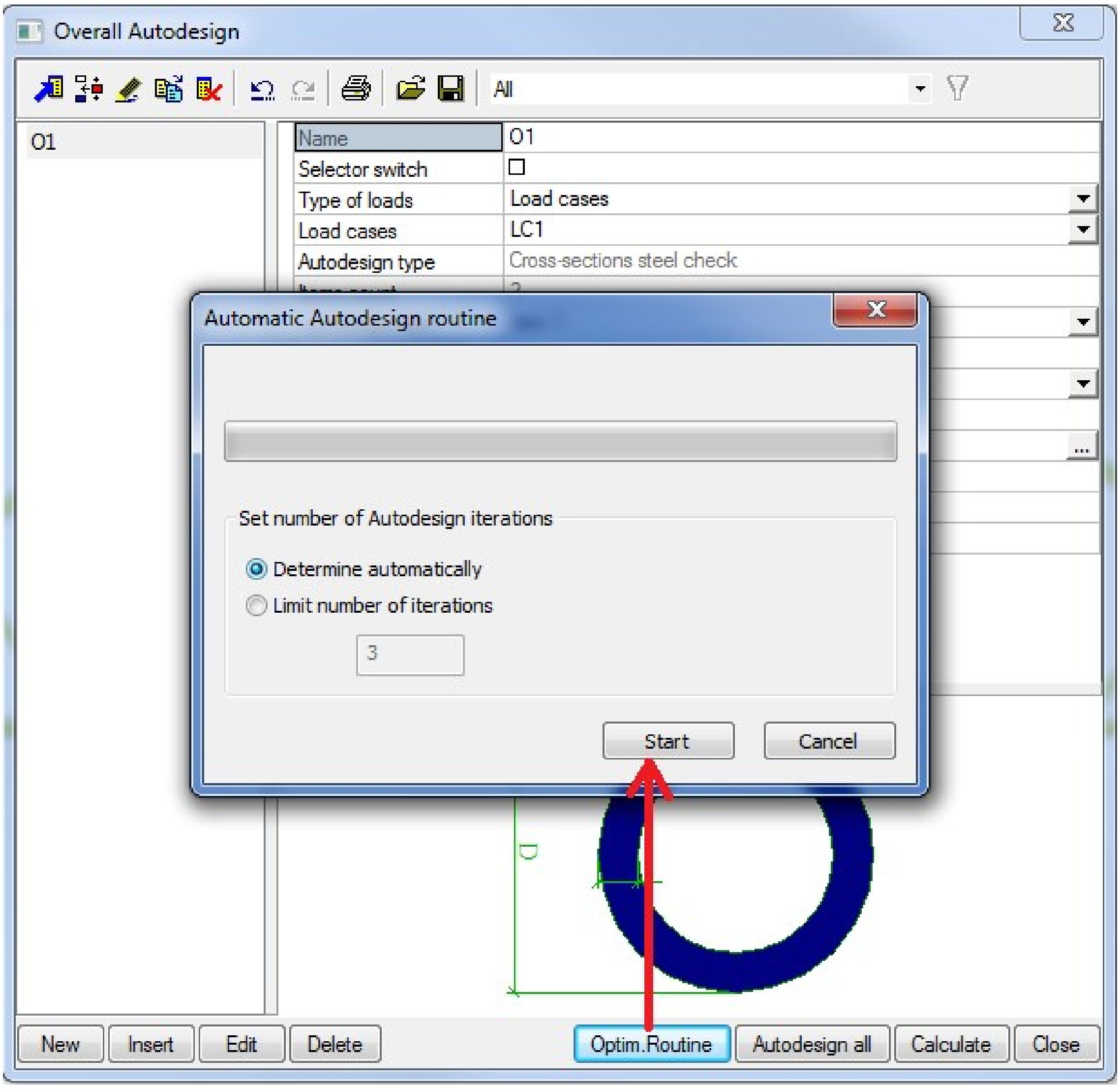
Conclusion

If you save the selected result using button “Save As” (see picture above) and open the saved .esa file, you will get the corresponding structure (see also attached file Steel truss girder_best.esa).

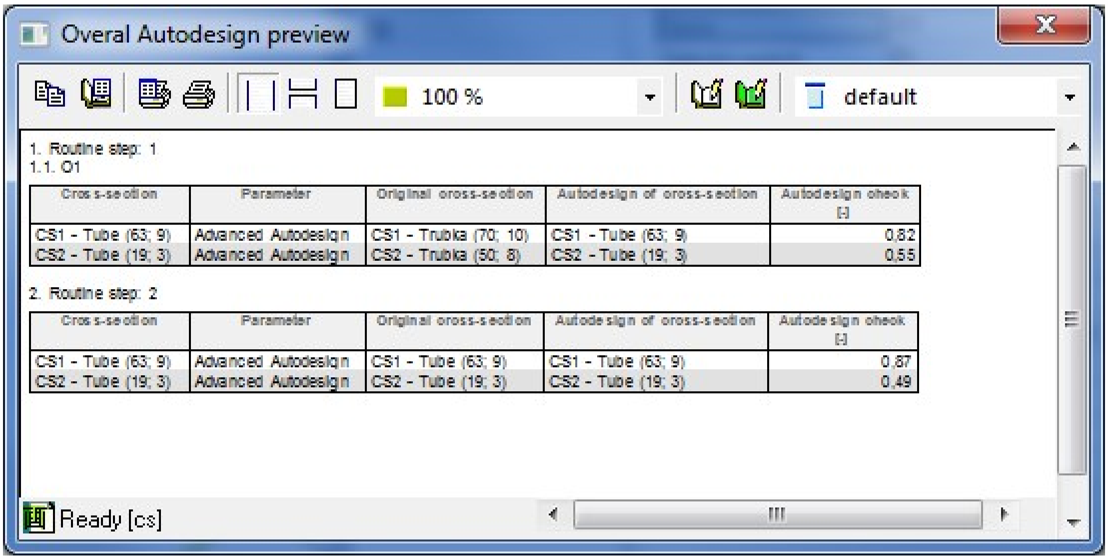


The structure saved this way contains the status corresponding to the values of parameters. To get also correct cross-sections, it is necessary to run Autodesign function manually!

Run linear analysis and Autodesign afterwards in SCIA Engineer. Use the Optimization Routine for iterative design (due to reasons described in chapter 2.2). Let the software determine automatically how many iterations are necessary and click the button Start.



For more information, you can see the final cross-section dimensions with optimal utilization in the preview window which appears at the end of Autodesign.



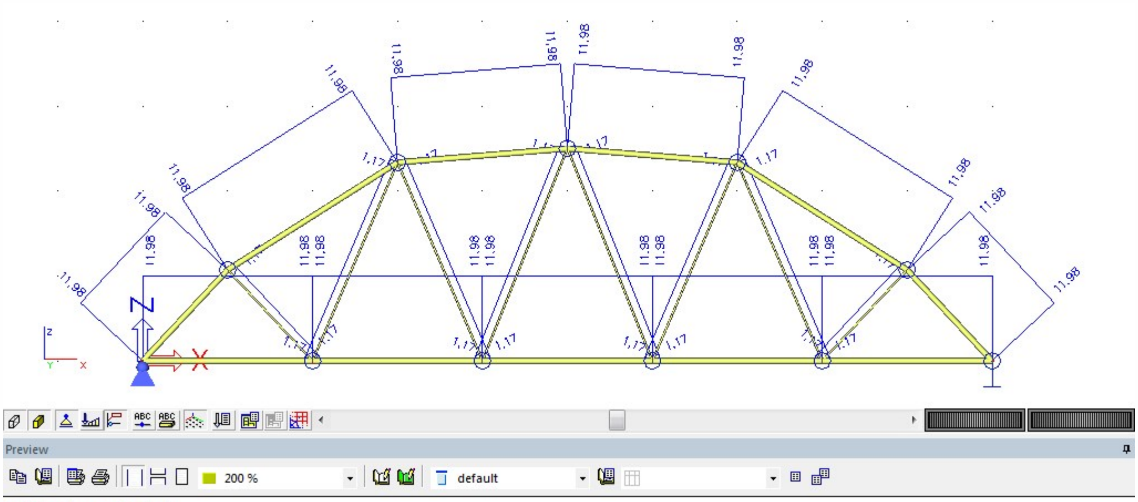
1. Routine step: 1
1.1. O1

Cross-section	Parameter	Original cross-section	Autodesign of cross-section	Autodesign check [-]
CS1 - Tube (63; 9)	Advanced Autodesign	CS1 - Trubka (70; 10)	CS1 - Tube (63; 9)	0,82
CS2 - Tube (19; 3)	Advanced Autodesign	CS2 - Trubka (50; 8)	CS2 - Tube (19; 3)	0,55

2. Routine step: 2

Cross-section	Parameter	Original cross-section	Autodesign of cross-section	Autodesign check [-]
CS1 - Tube (63; 9)	Advanced Autodesign	CS1 - Tube (63; 9)	CS1 - Tube (63; 9)	0,87
CS2 - Tube (19; 3)	Advanced Autodesign	CS2 - Tube (19; 3)	CS2 - Tube (19; 3)	0,49

Using function Bill of material you can see detailed output of mass related to each cross-section.



Bill of material

Name	Mass [kg]	Surface [m²]	Volume [m³]
Total results :	281,28	5,379	3,5832e-02

CSS	Material	Unit mass [kg/m]	Length [m]	Mass [kg]	Surface [m²]	Unit volume mass [kg/m³]	Volume [m³]
CS1 - Tube (63; 9)	S 355	11,98	21,677	259,76	4,290	7850,00	3,3090e-02
CS2 - Tube (19; 3)	S 355	1,17	18,480	21,53	1,088	7850,00	2,7426e-03

Steel frame design

Getting started

Starting a project

Before you can start a project, you need to start the program first.

Starting the program

1. Double-click on the SCIA Engineer shortcut in the Windows Desktop.

or

2. If the shortcut is not installed, click [Start] and choose Programs > SCIA Engineer > SCIA Engineer. If the program does not find any protection, you will obtain a dialogue indicating that no protection was found. A second dialogue will list the restrictions of the demo version. Click [OK] in both windows.

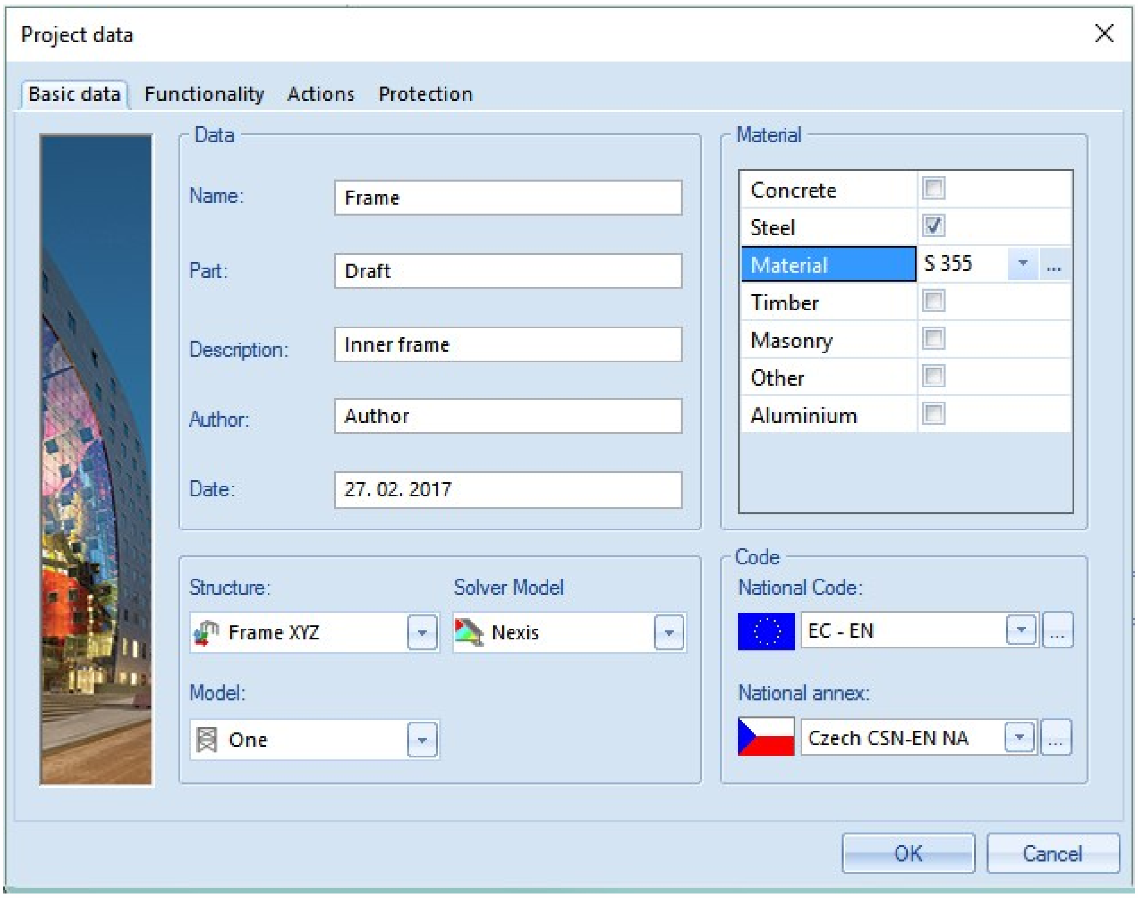
For this tutorial, you must start a new project.

Starting a new project

For this tutorial we have to start a new project. If the dialog Open appears, click [Cancel].

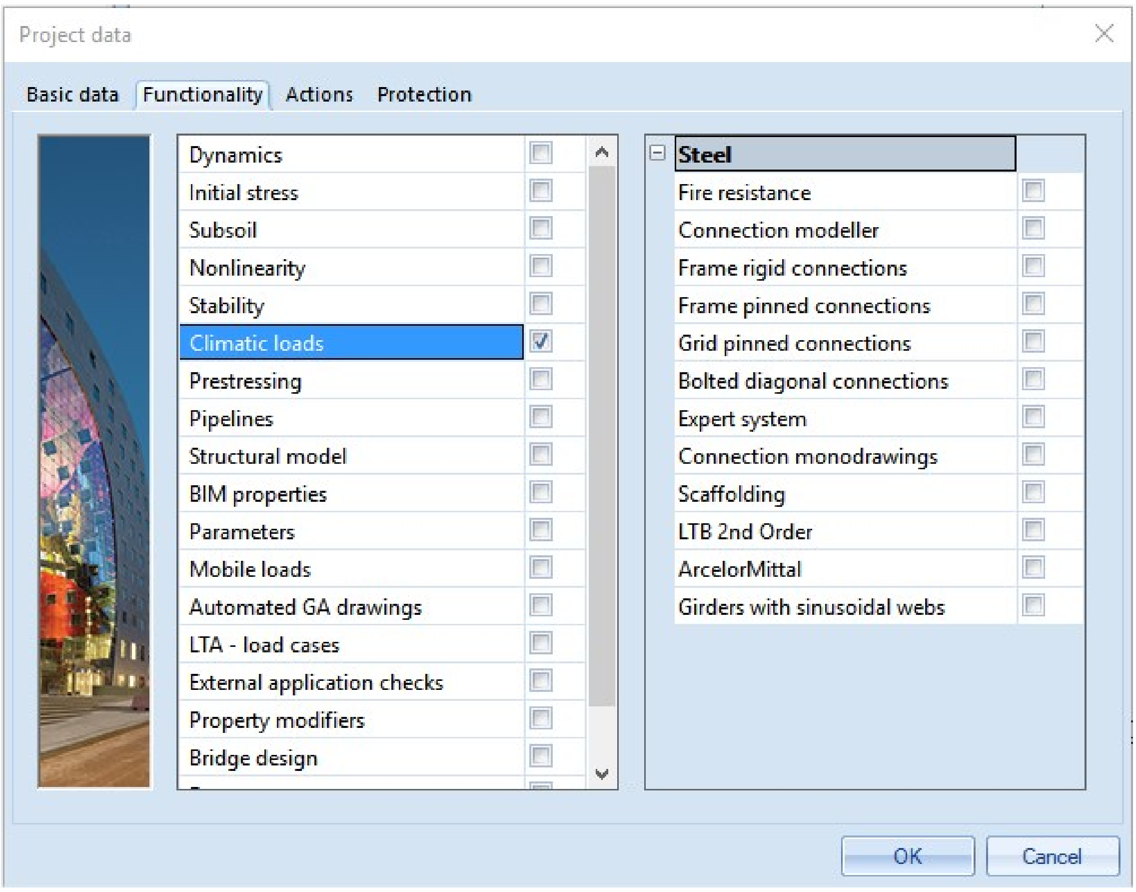
Click on the New icon  in the toolbar.

Now, the Project data dialogue is opened. Here, you can enter general data about the project.

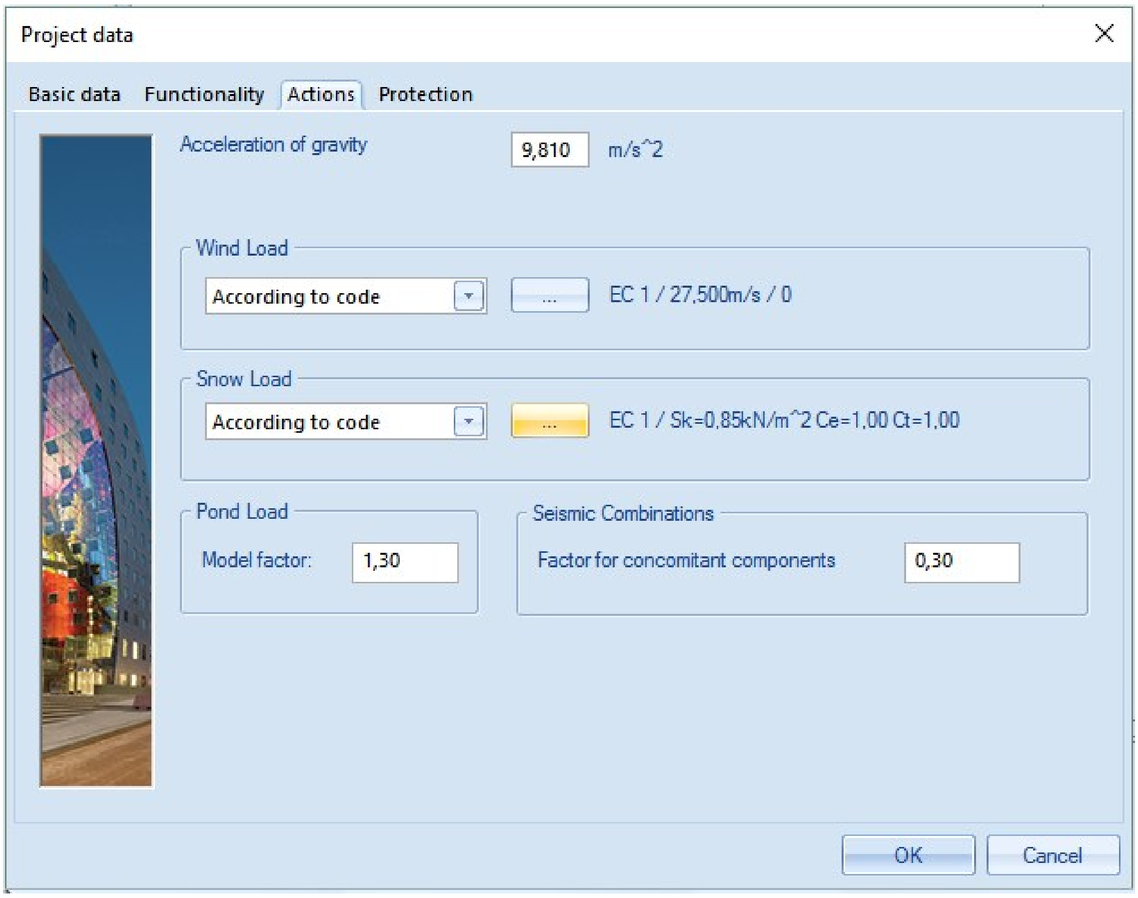


1. In the Data group, enter your preferred data. These data can be mentioned on the output, e.g. in the document and on the drawings.
2. Select Frame XYZ in the Structure field.
3. Choose Model: One.
4. Click on the rectangular button  below National Code to choose the default code for the project. This code will determine the available materials, combination rules and code checks. For the project of this Tutorial, choose EC-EN.
5. Select a National annex. For this example choose Czech CSN – EN NA.
6. In the Material group, select Steel.
   Below the item Steel, a new item Material will appear - choose S355 from the menu.

On the Functionality tab choose the options Climatic loads.



On the Load tab set the value of check boxes Wind Load and Snow Load to the option According to code.



If you complete settings, confirm your choice by click on the button [OK].

On the Basic data tab, you can set a project level. If you choose “default”, the program will only show the most frequently used basic functions. If you choose “advanced”, all basic functions will be shown.

On the Functionality tab, you choose the options you need. The non-selected functionalities will be filtered from the menus, thus simplifying the program.

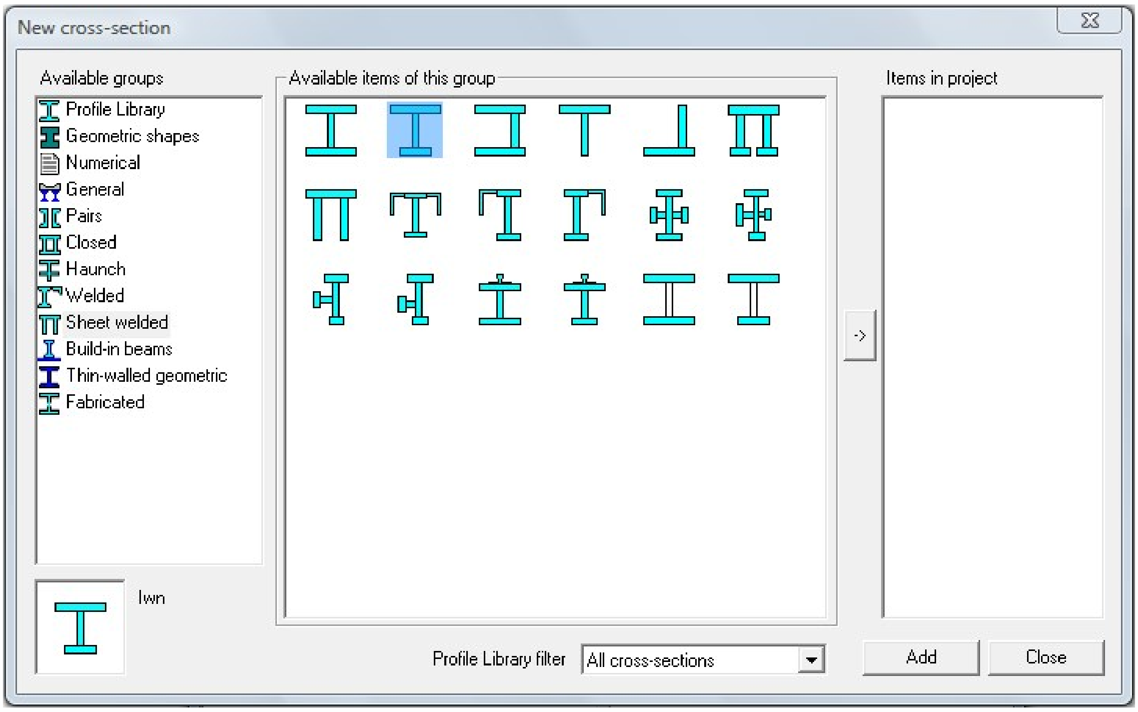
Structure definition

If you start a new project, the geometry of the structure must be entered.

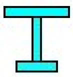
Cross-sections, lists and matrices definition

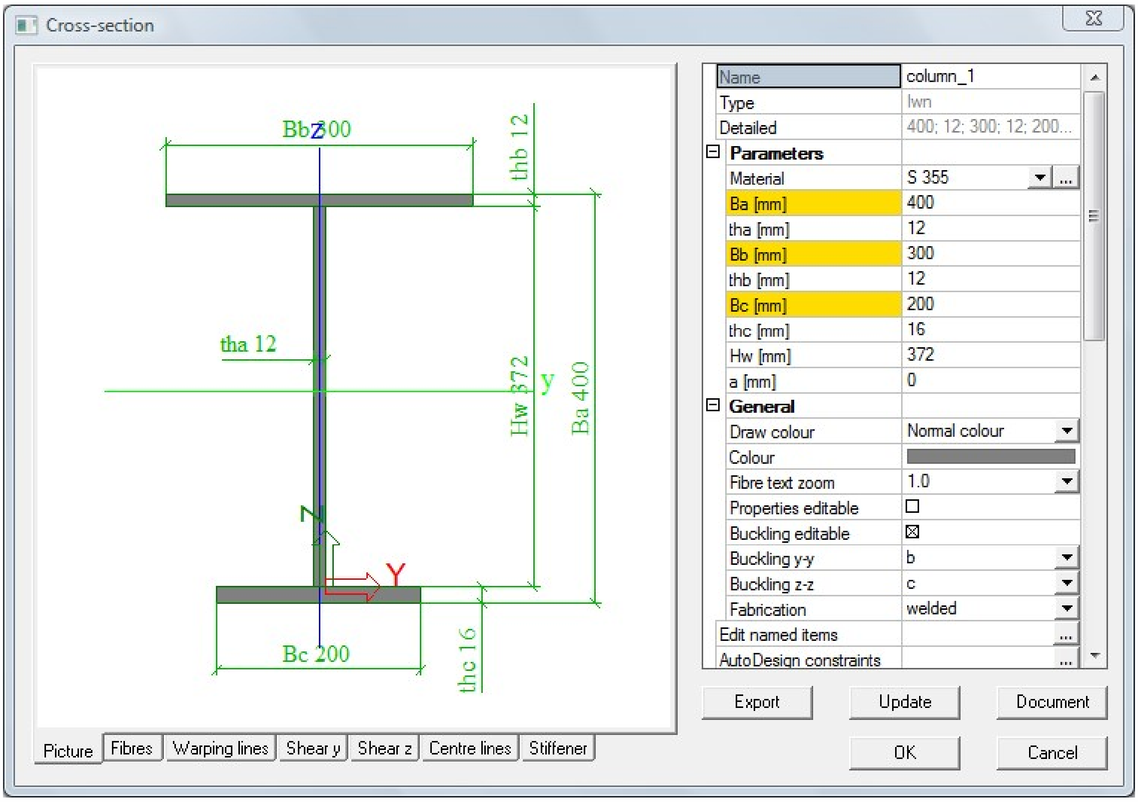
Cross-sections

1. Click on the Cross-Sections  icon in the toolbar.
   The Cross-Sections manager is opened. If no profiles have been entered in the project, New cross-section window will be automatically opened.



2. Click Sheet welded in the group Available groups.

3. In the Available items of this group, choose Iwn profile  from the list.

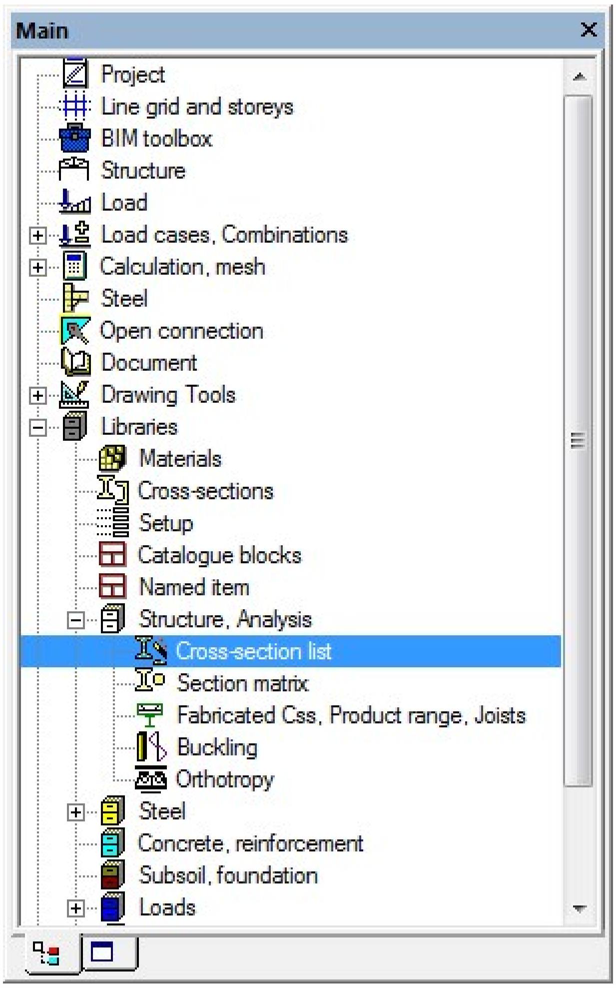


4. Set the name to CL1.
5. Confirm by clicking [OK].
6. Add in the same way cross-sections RL1, RL2, RL3, CR1, RR1, RR2 and RR3.
7. Click [Close] to close the New cross section dialogue.
8. Click [Close] to close the Cross-Sections manager and to return to the project.

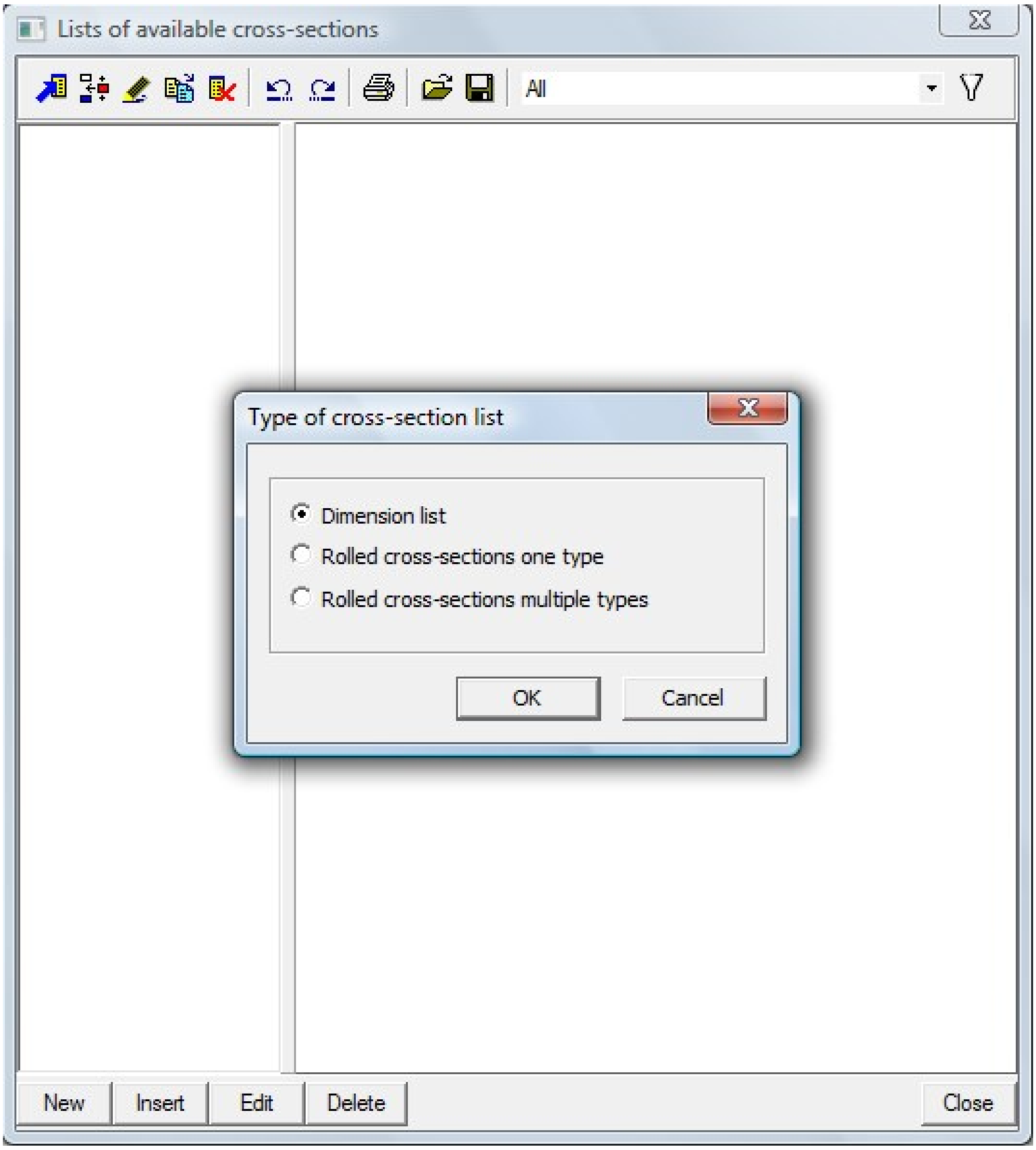
> It is necessary to define a unique CSS for each member else frame autodesign will not work correctly.

Cross-section lists

1. To define CSS lists, use the option Main window→Libraries→Structure, Analysis→ Cross-section list



2. Type of cross-section list dialog appears.



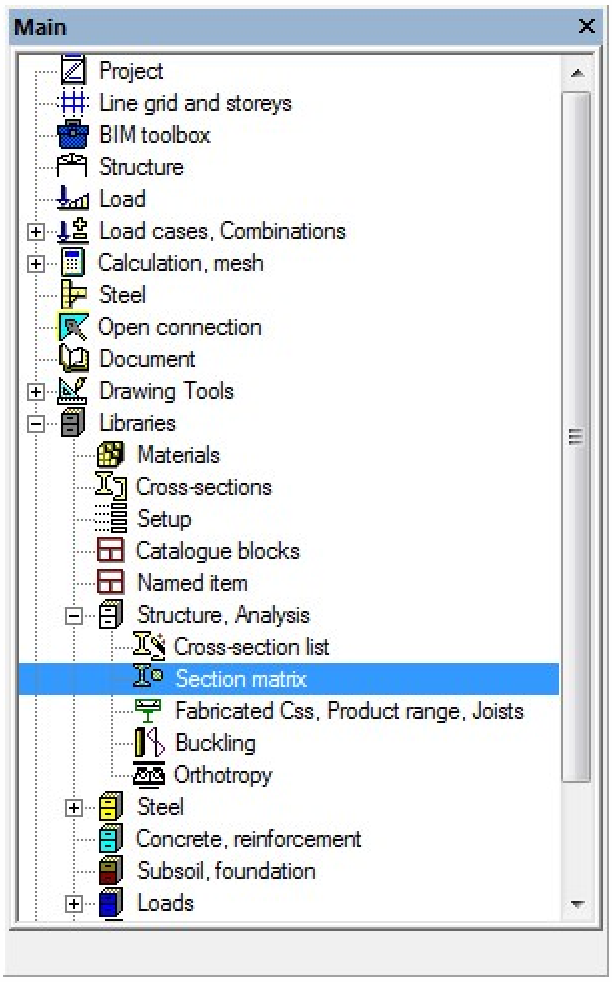
3. Use option Dimension list.
4. Dimensions dialog appears.
5. Define values 100,125,150,175,200,225,250,300
6. Confirm your input with [OK].
7. A new list LIST1 appears.
8. Change the name of the list to fw.
9. Click 

 or  to create a next list
10. Repeat points 3-8 to define lists:
    ft (6,8,10,12,16,18,20,24,28,32,36)
    wd (250,300,350,400,450,500,600,700,800,900,1000,1100,1200,1300,1400,1500)
    wt (5,6,8,10,12,14,16,18,20)

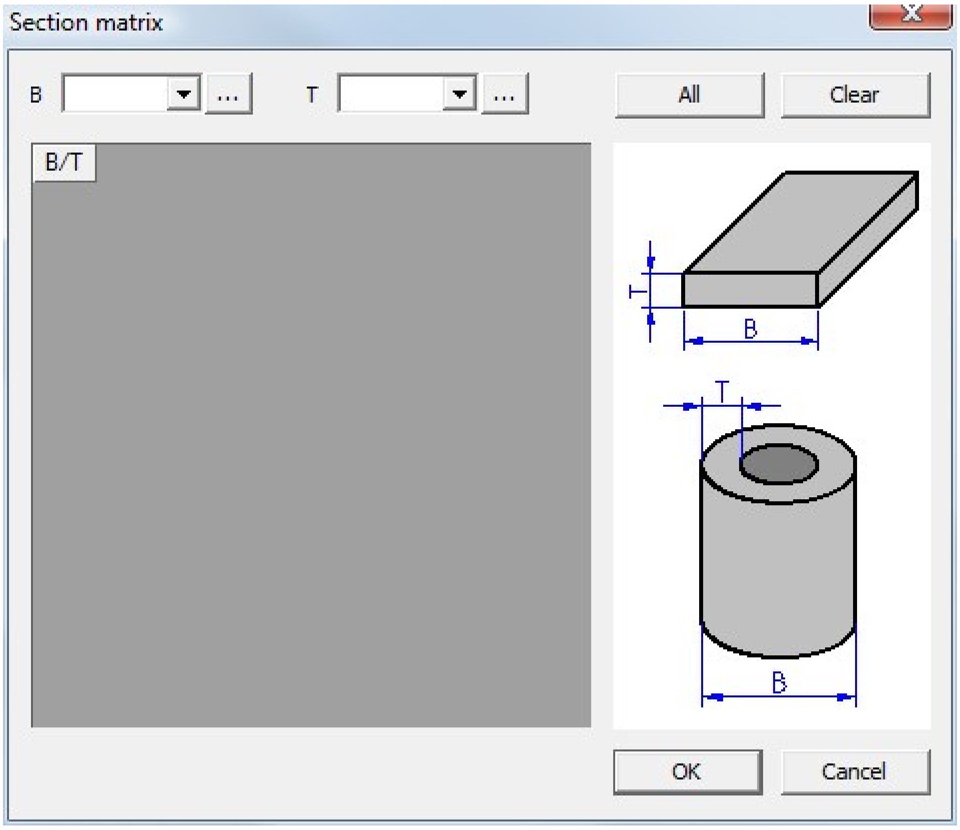
11. Click [Close] to close the List of available cross-sections dialogue.

Section Matrix

1. To define section matrix, use the option Main window→Libraries→Structure, Analysis→ Section matrix

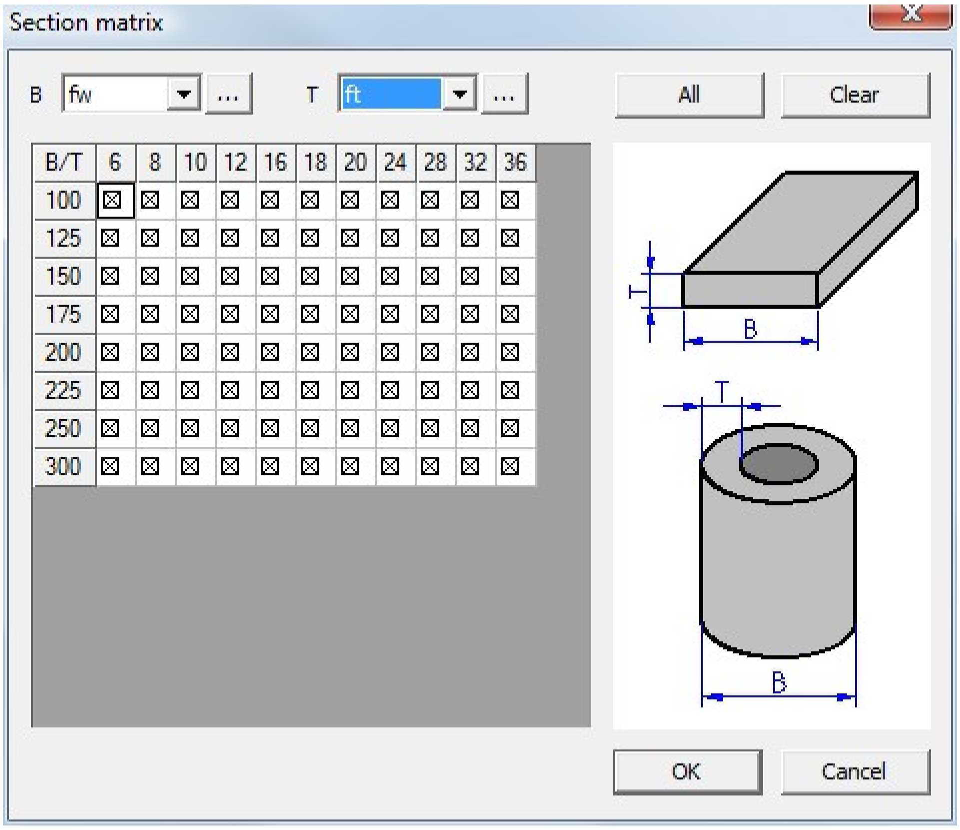


2. Section matrix dialog appears.

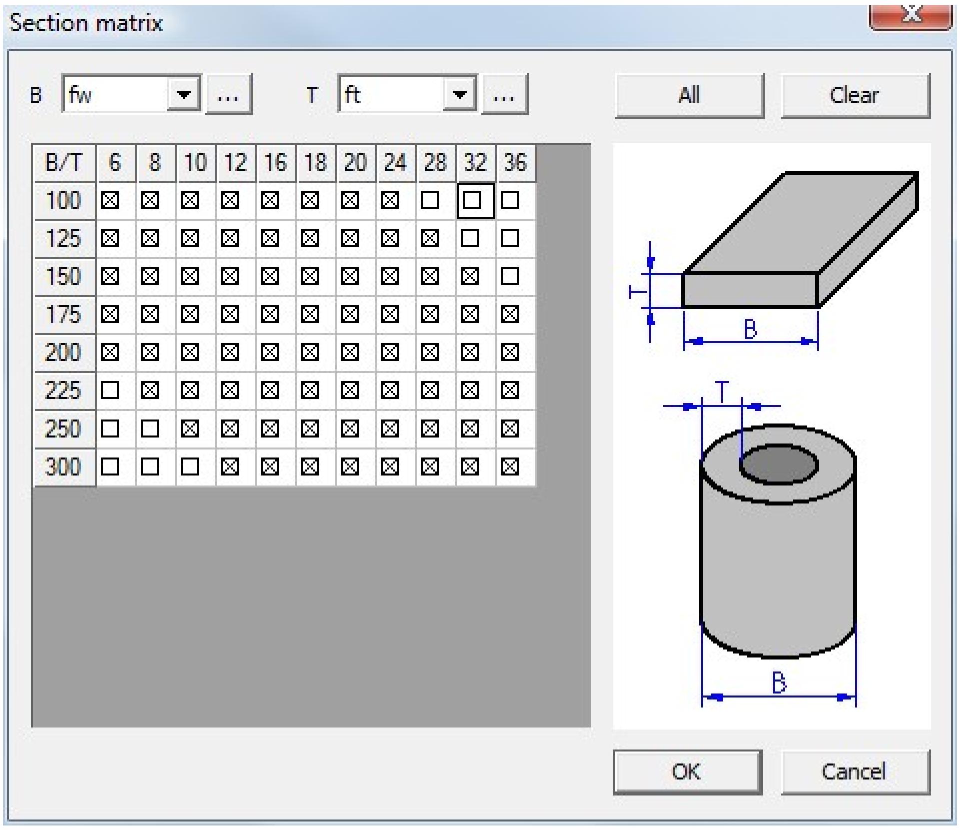


3. Select list fw in B combo–box.
4. Select list ft in T combo–box.

5. Matrix from defined list appears.



6. Unselect some options according to following drawing

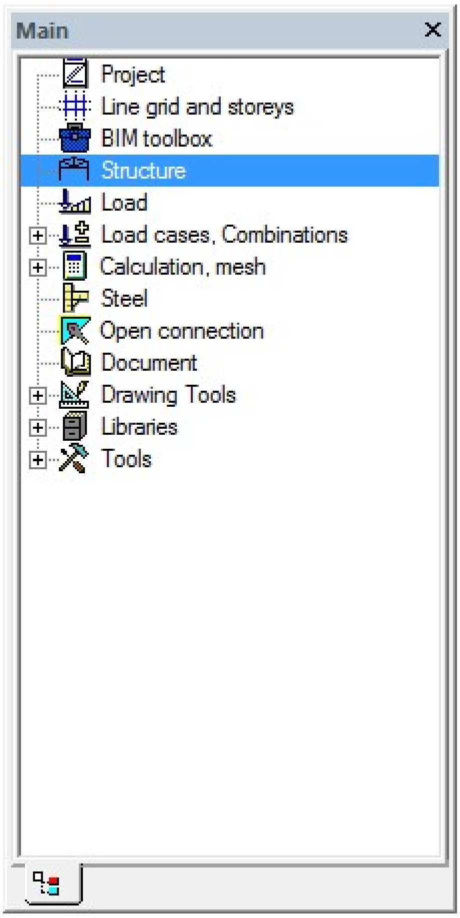


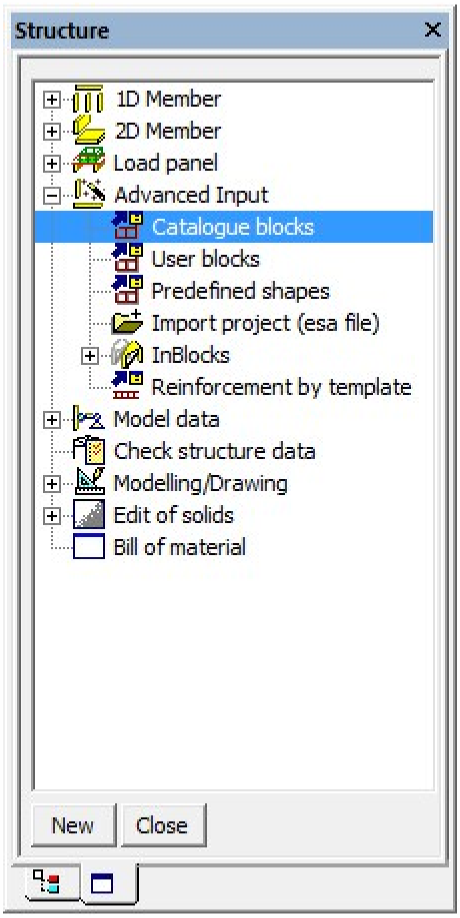
7. Confirm your input with [OK].
8. Change the name of the matrix to FM.
9. Click [Close] to close the Section matrix dialogue.

Geometry

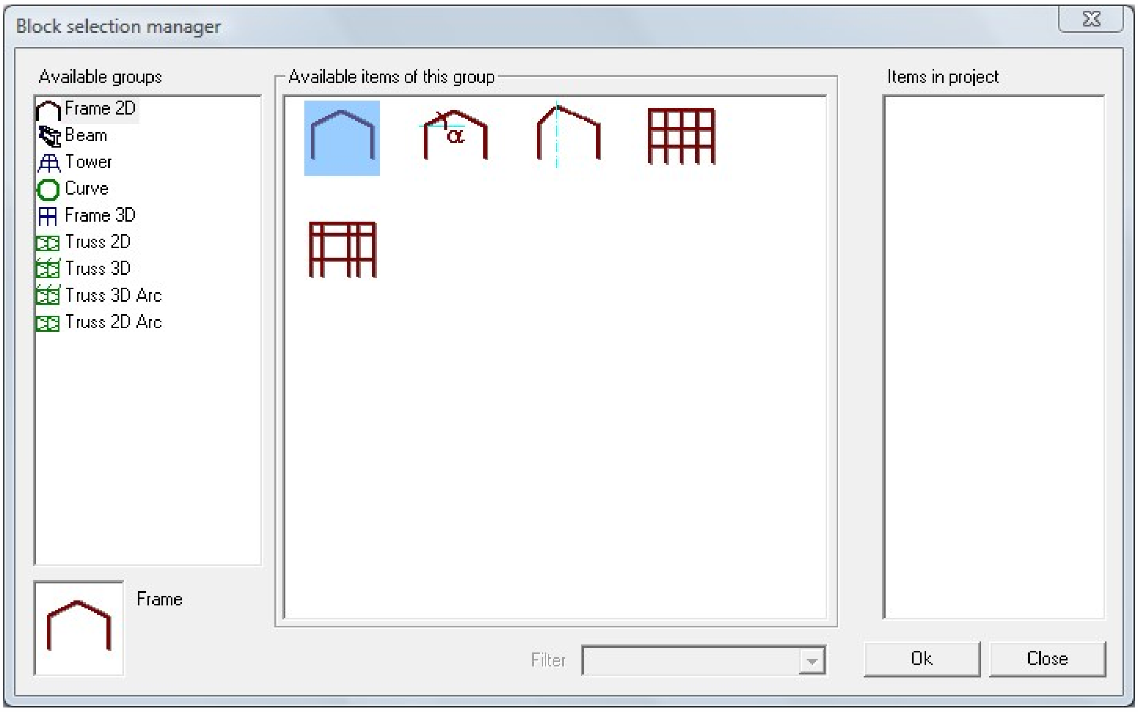
Structure input

1. Double-click on Structure in the Main window.

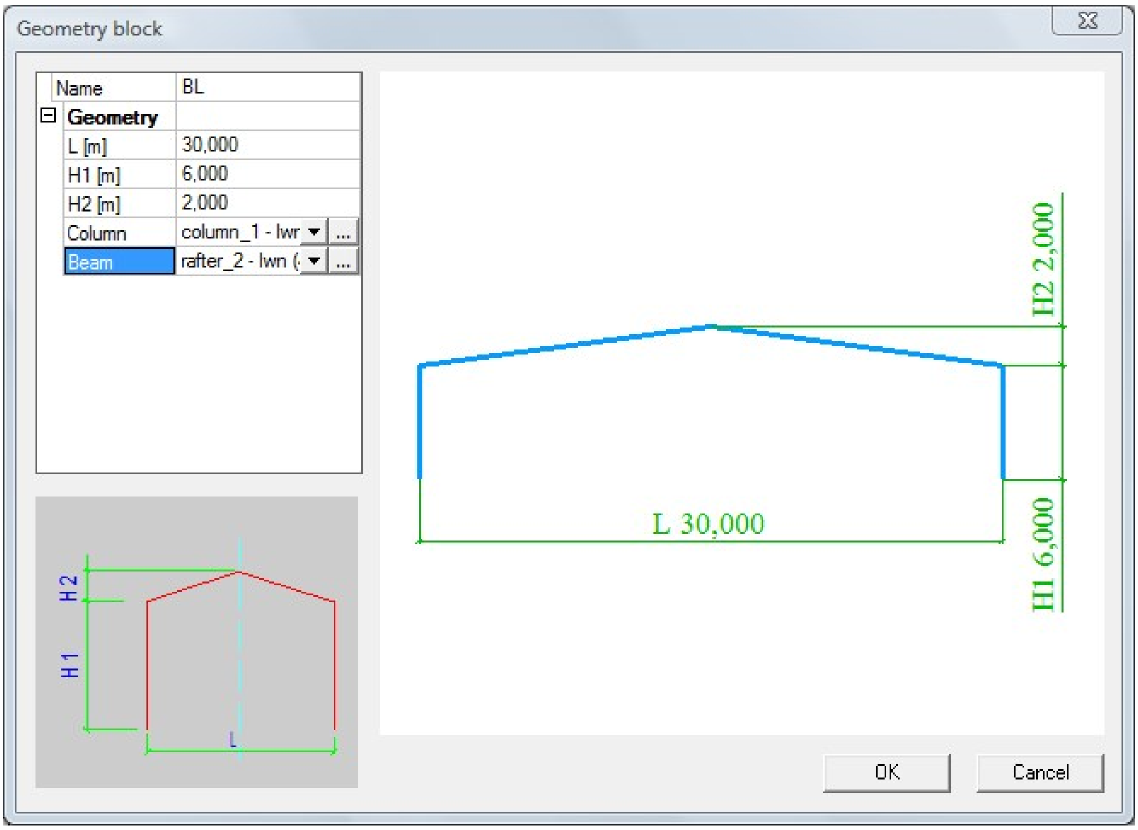




2. To enter a new frame, use the option Advanced Input and Catalogue Blocks in the Structure menu. The Catalogue block manager is opened.

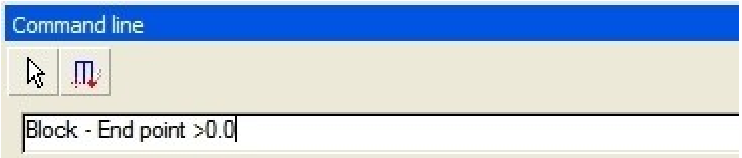


3. In the Available Groups choose a Frame 2D
4. In the Available items of this group choose the first option (frame).
5. Confirm your choice with [OK]. The Geometry block window appears.



6. Enter the frame dimensions: L = 30m, H1 = 6m and H2 = 2m

7. In the pull-down menus, choose CL1 for the RL1 for the Beam.
8. Confirm your input with [OK]. The Catalogue block manager appears.
9. Click [OK] to return to the project.
10. Left column origin is used as an input point of the frame. Type the coordinates 0 0 0 in the Command line and press <Enter> to confirm your input.

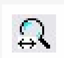


11. End the input with the <Esc> key.

The properties of selected elements are shown and can be modified in the Properties window.

If no section has been defined in the project, the New cross-section window will automatically appear, as soon as you will try to enter a structural element (column, beam…).

You can end your input by pressing either the <Esc> key either the right mouse button.

With Zoom All icon  in the toolbar, you can visualize the entire structure.

For coordinates separation use <space> key¨ ¨.

Structure modification

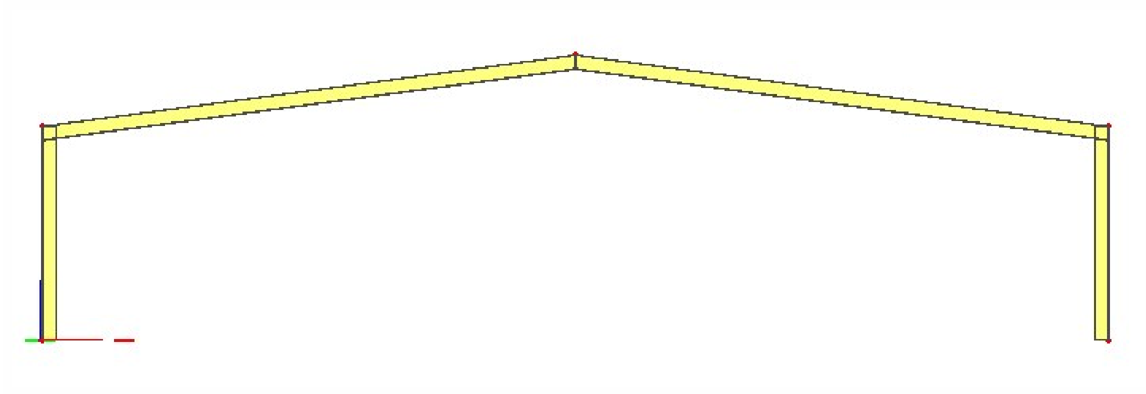
System line alignment

1. Select all members
2. Set Member system-line at to top in Property window.
3. Finish selection with <Esc> key.

LCS Rotation

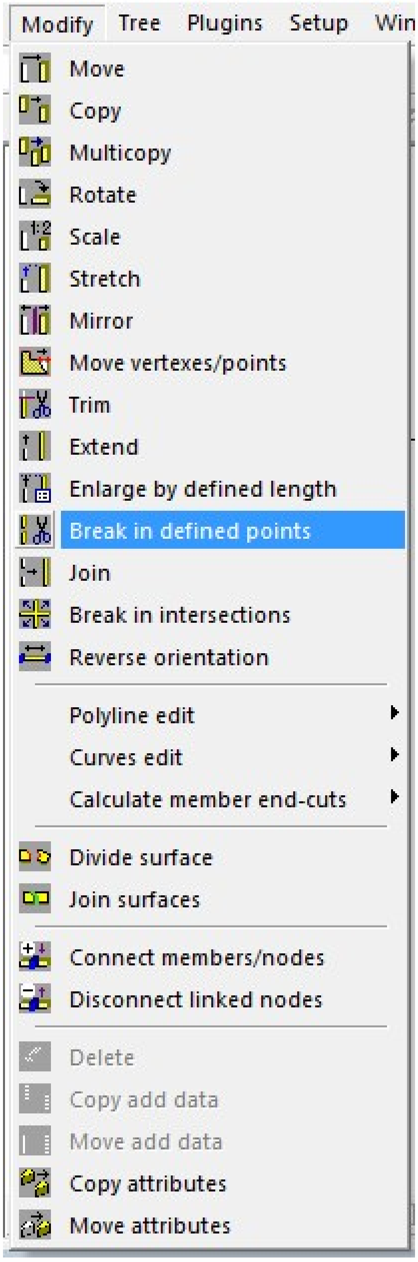
1. Select right column
2. Set value of LCS rotation to 180 [deg] in Property window.
3. Finish selection with <Esc> key.
4. Switch on  Show/hide surfaces button.
5. Switch on  Render geometry button.

6. Check generated structure

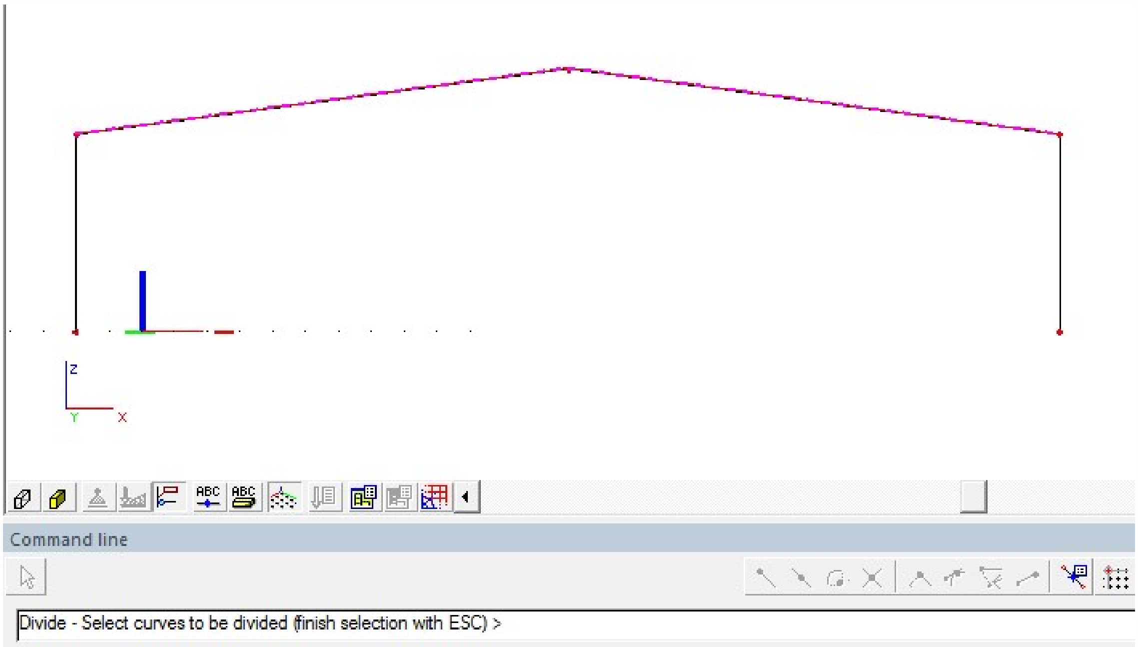


Rafters splitting

1. To split rafter beam into more parts use an option Break in defined points from Modify tab in the Main menu.

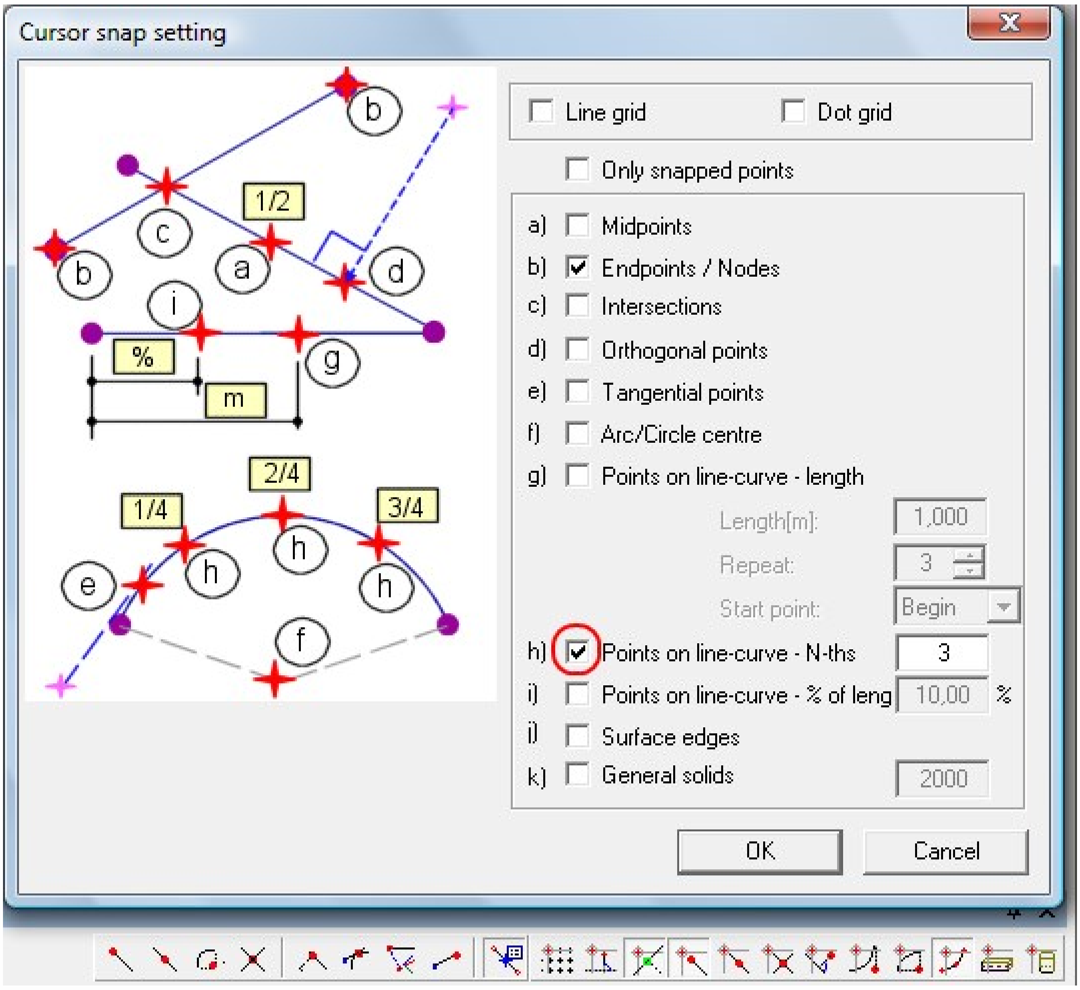


2. Select both rafter beams and finish selection with <Esc> key.

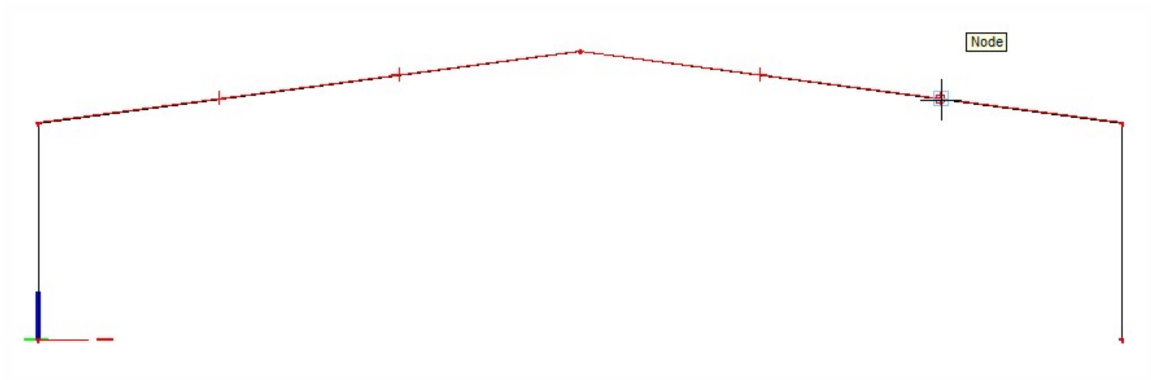


3. Click on the Snap mode button. The Cursor snap setting manager is opened.

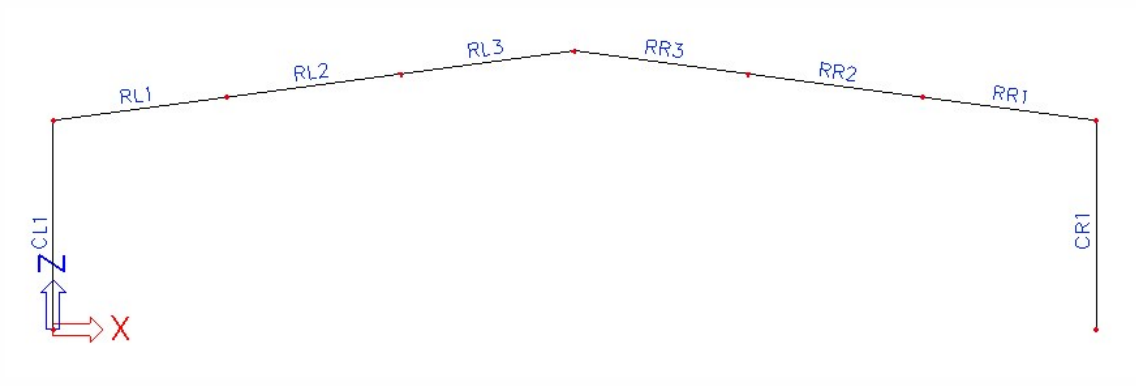
4. Activate option Points on line-curve and set it's value to 3.



5. Select all rafter points on line.



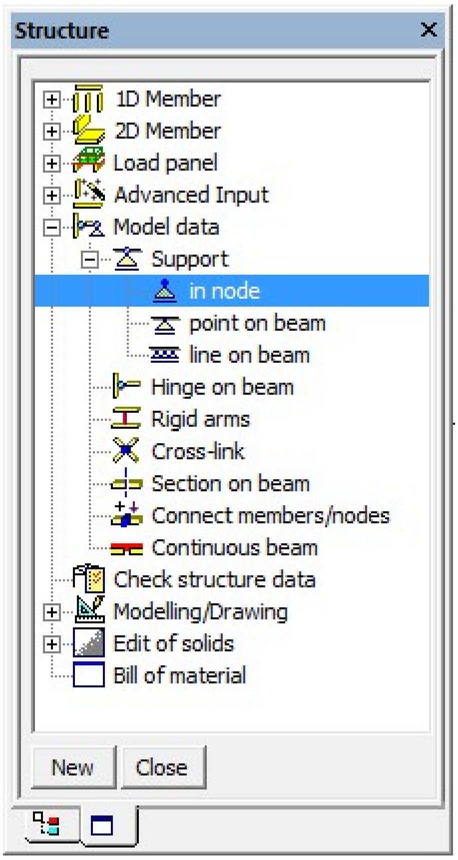
6. Finish selection with <Esc> key.
7. Select left middle rafter beams and change CSS to RL2 in Property window.
8. Finish selection with <Esc> key.
9. Repeat points 7 and 8 to set cross-sections according to the following drawing :



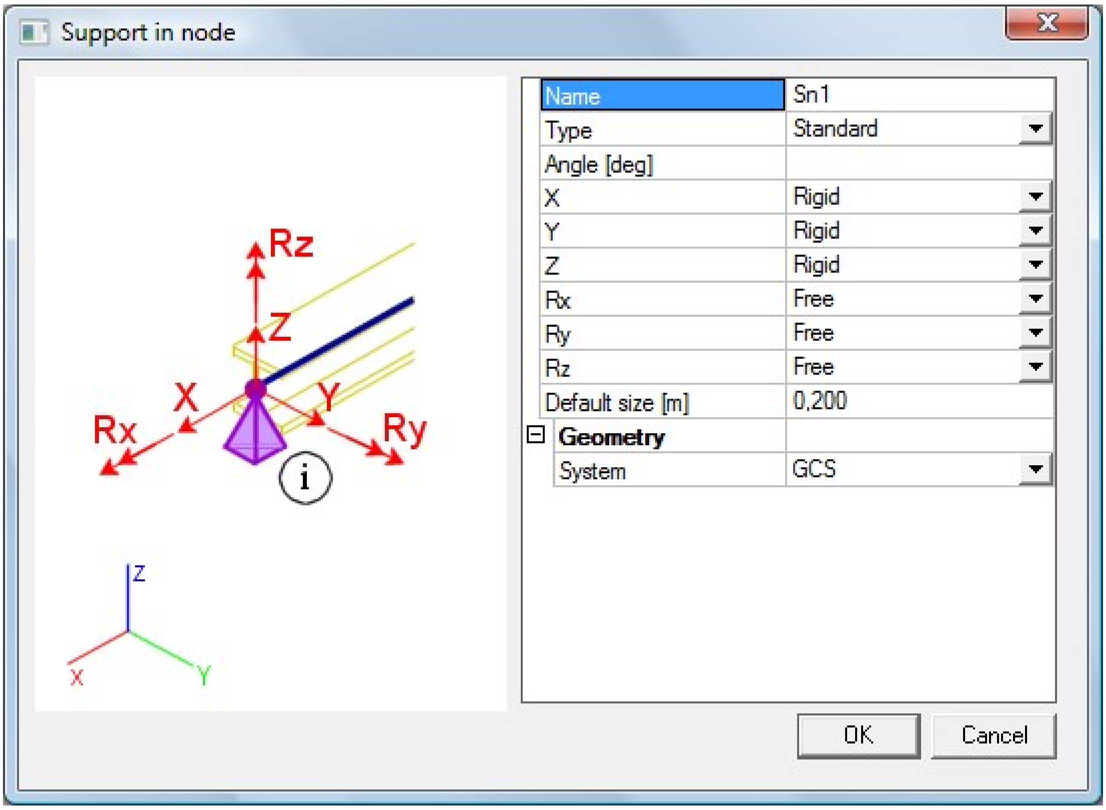
Supports

The geometry input can be completed with supports definition.

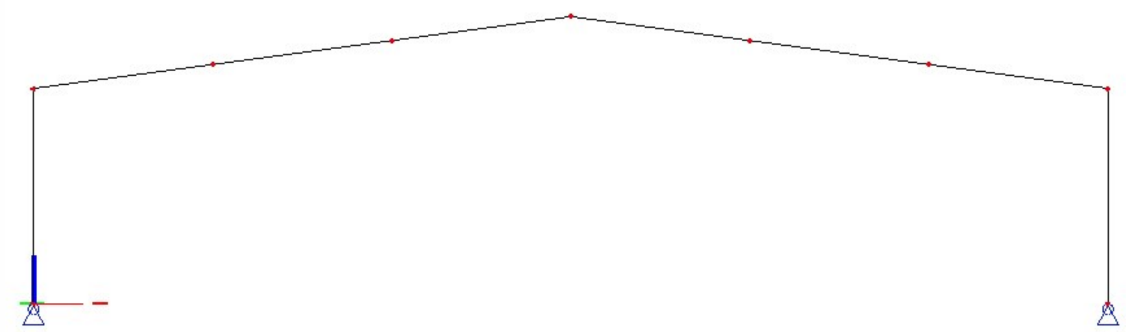
1. To enter supports, use the option Main window→Structure→Model data→Support→in node



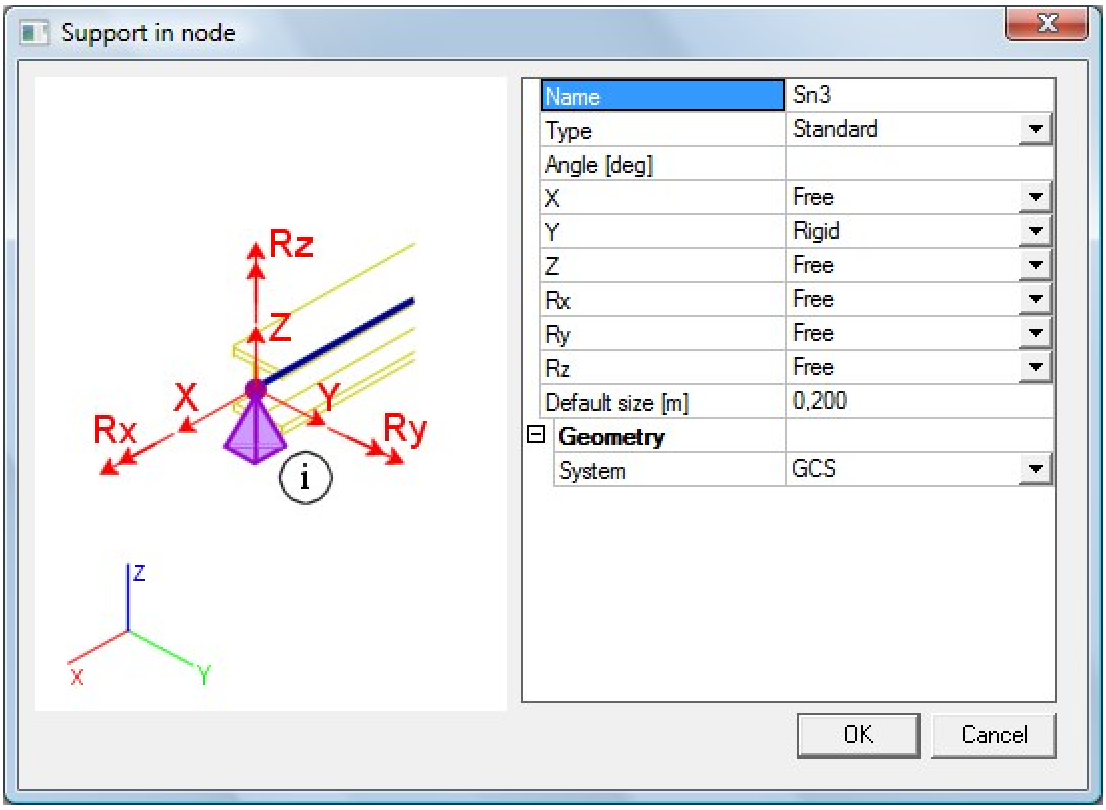
2. Define all support translations Rigid and rotations Free.

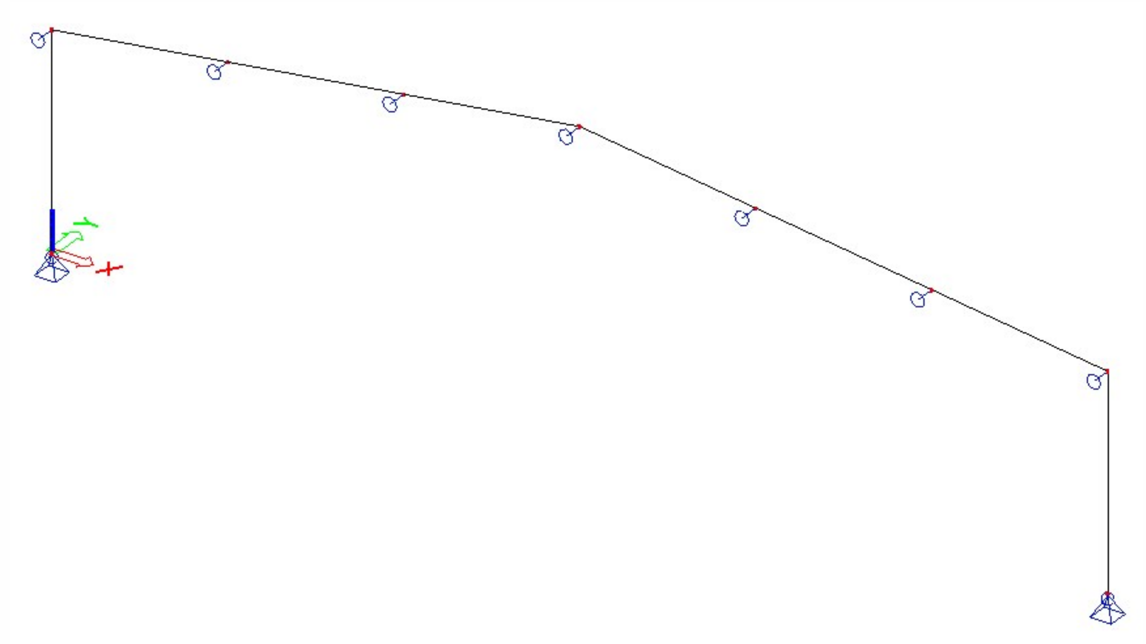


3. Confirm your input with [OK].
4. Select both column bases and press <Esc> to finish the selection.



5. Use the previous workflow and define supports in remaining nodes with the following settings:

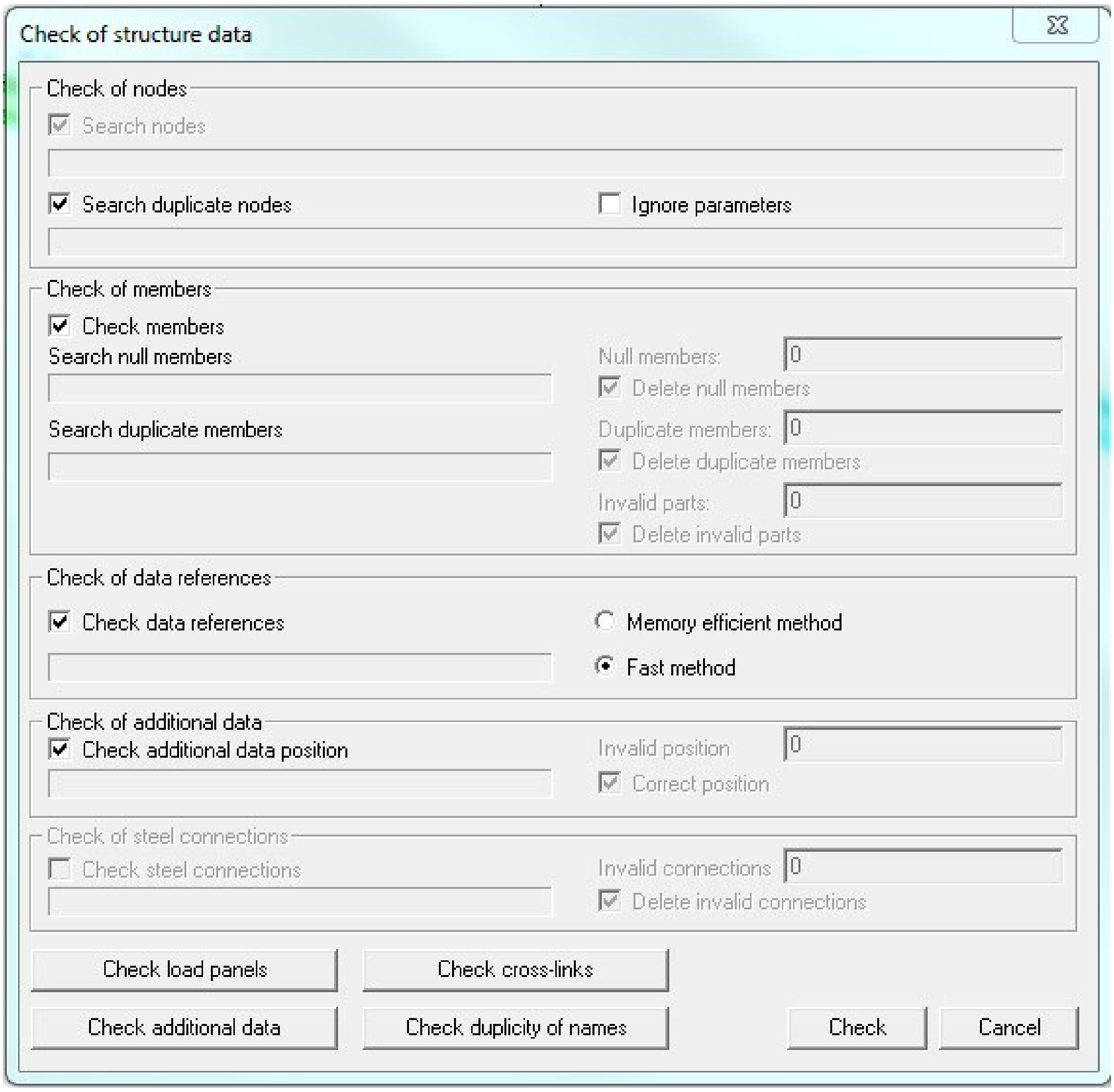




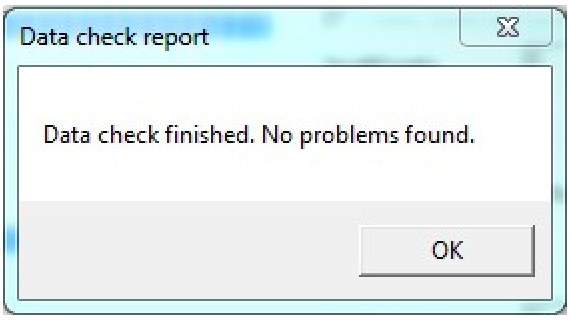
Check Structure data

After input of the geometry, the input can be checked for errors by means of the option Check Structure data. With this tool, the geometry is checked for duplicate nodes, zero bars, duplicate bars, etc.

1. Double-click on the Check Structure data option in the service Structure or click on the icon in the toolbar. The Structure data check window appears, list the different available checks.



2. Click [Check] to perform the checks.
   The Data Check Report window appears, indicating that no problems were found.



3. Close the check by clicking [OK].

Loads and combinations

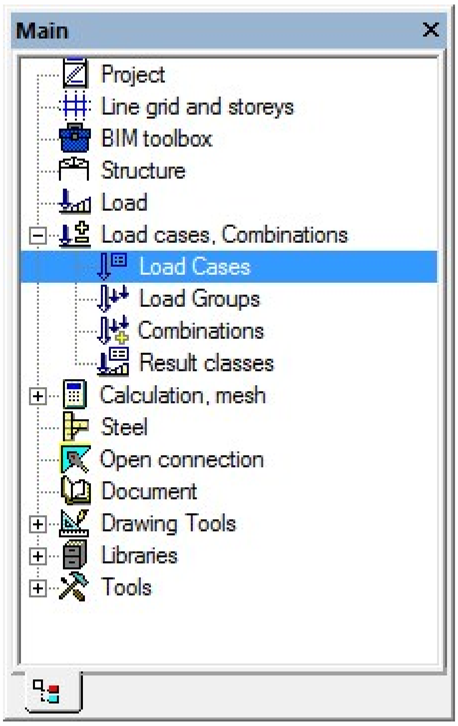
Load Cases and Load Groups

In this project, six load cases are entered:

- LC1 – Self weight; action type Permanent; load group LG1; load type Self weight; direction –Z
- LC2 – Dead load; action type Permanent; load group LG1; load type Standard
- LC3 – Live Load; action type Variable; load group LG2; load type Static; specification Standard; duration Short; Master load case None
- WND-L - Wind from left; action type Variable; load group Wind; load type Static; specification Standard; duration Short; Master load case None
- WND-L - Wind from left; action type Variable; load group Wind; load type Static; specification Standard; duration Short; Master load case None
- SN - Snow loads; action type Variable; load group Snow; load type Static; specification Standard; duration Short; Master load case None

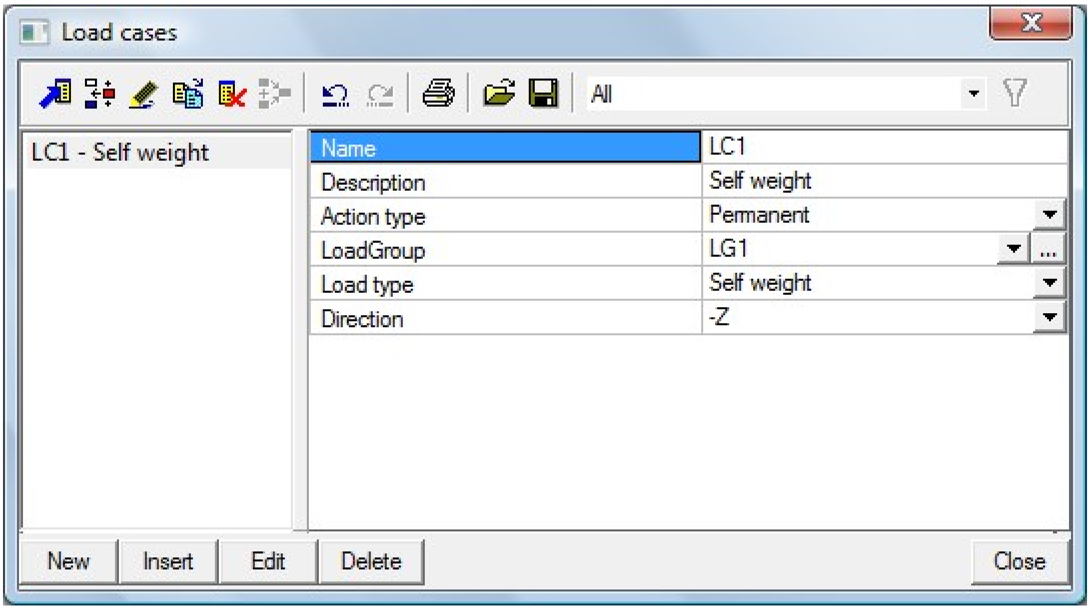
Self weight

1. Double-click on the Load Cases in the service Load cases, Combinations in the Main window.



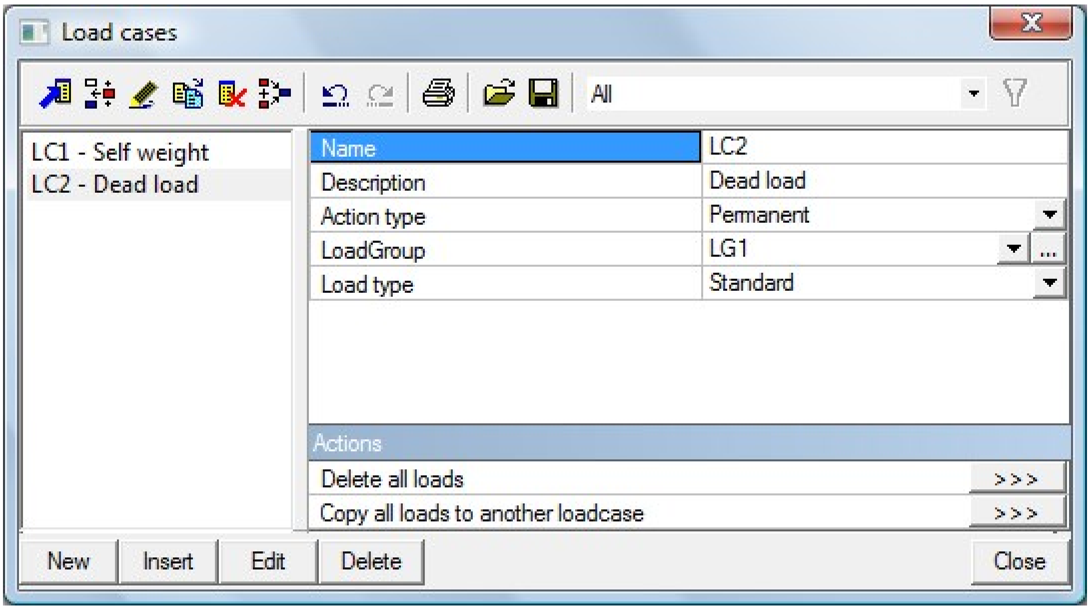
2. By default, the load case LC1 is created. This load is a permanent load of the Self weight load type. The self weight of the structure is automatically calculated by means of this type. You can describe the content of this load case. For this project,

enter the description "Self weight".



Dead load

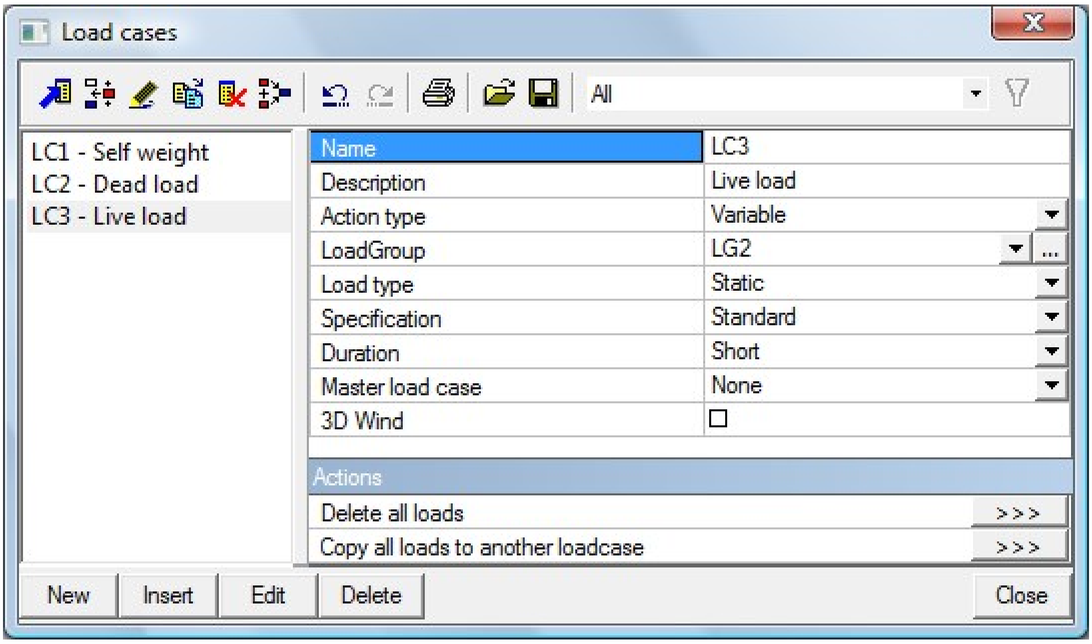
1. Click  or  to create next load case (LC2). Enter the description "Dead load".



Live load

1. Click 

or  to create next load case (LC3). Enter the description “Live load”.
2. As this is a variable load, change the Action type to Variable. The Load Group LG2 is automatically created.



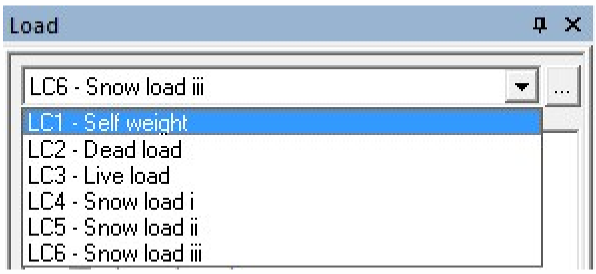
3. Confirm your input by click on the button [Close].

Load

After input of the Load cases, go to the service Load from Main window.

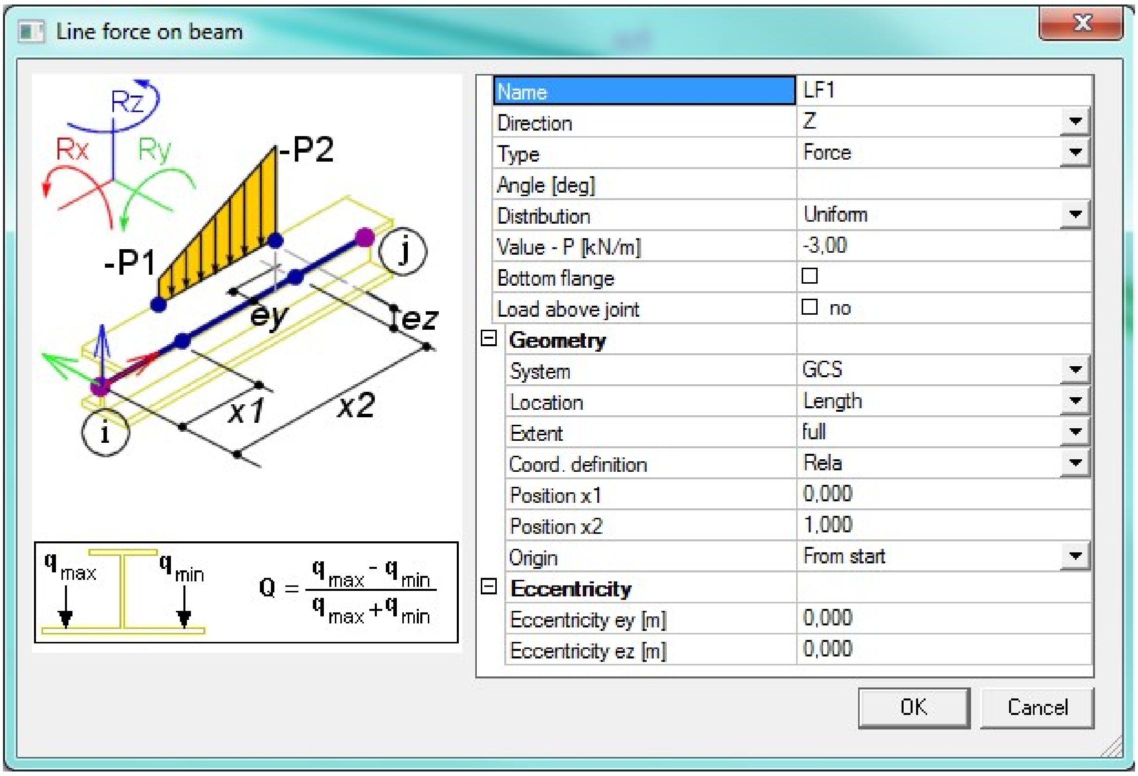
Switching between load cases

You can switch between load cases with the mouse pointer in the list box.



Dead load

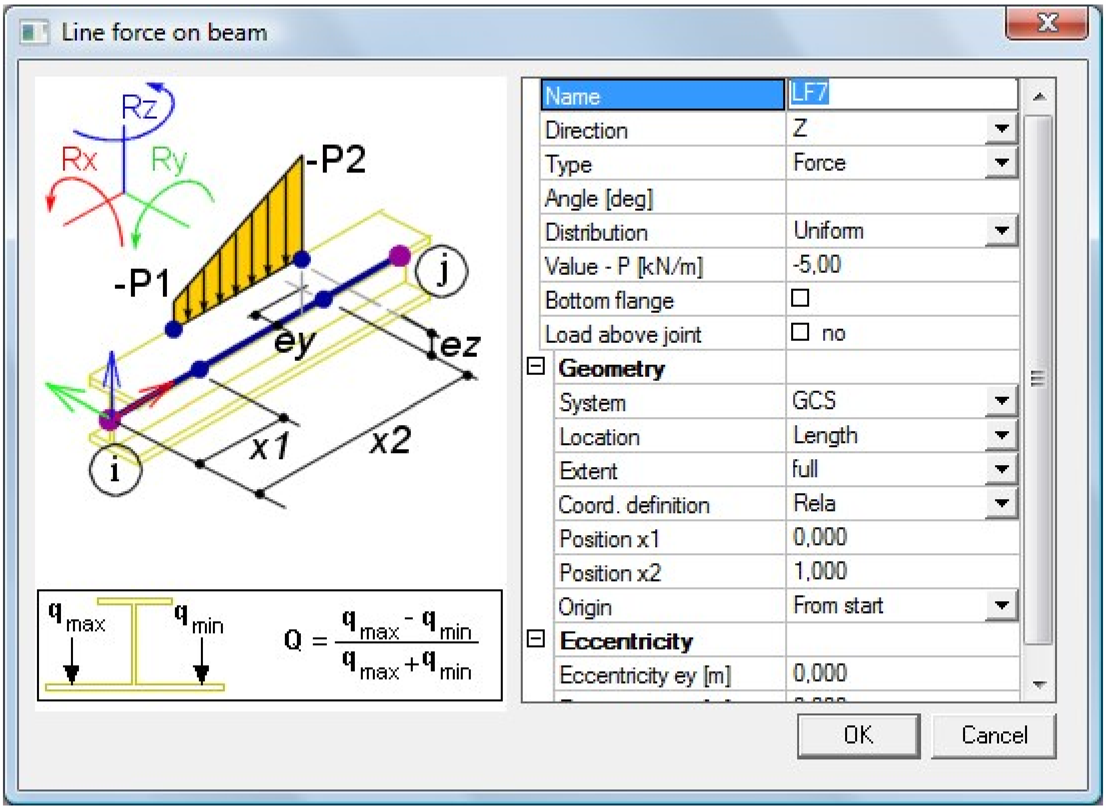
1. According to the previous paragraph select the load case LC2 – Dead load.
2. Click Line force - on beam in the Load Menu. The following dialog appears.



3. Set the Value to -3,0 kN/m and System to GCS
4. Confirm your input with [OK].
5. Select all rafter beams to enter defined load
6. Press <Escape> to finish the input.
7. Press <Escape> once more to finish the selection.

Live load

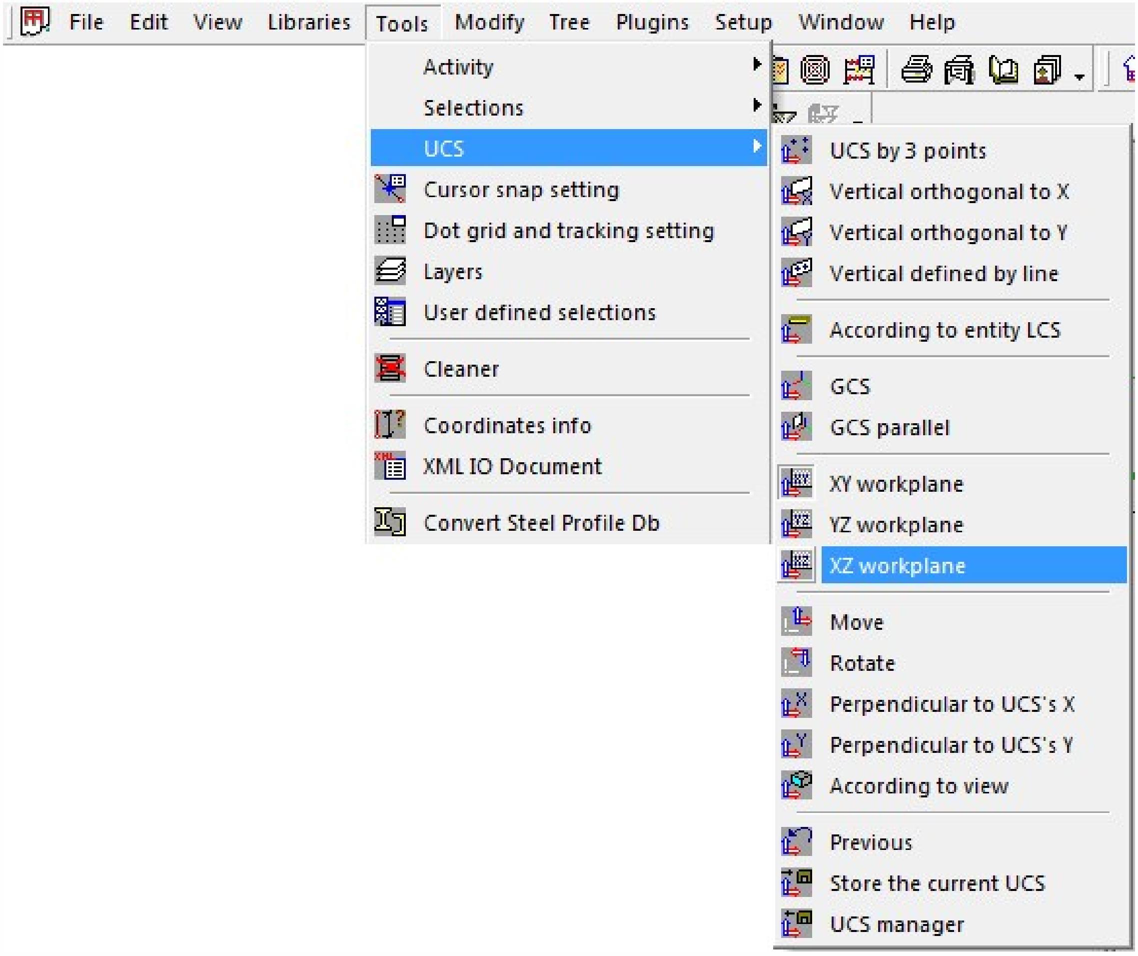
1. Select the load case LC3 – Live load.
2. Click Line force - on beam in the Load Menu. The following dialog appears.



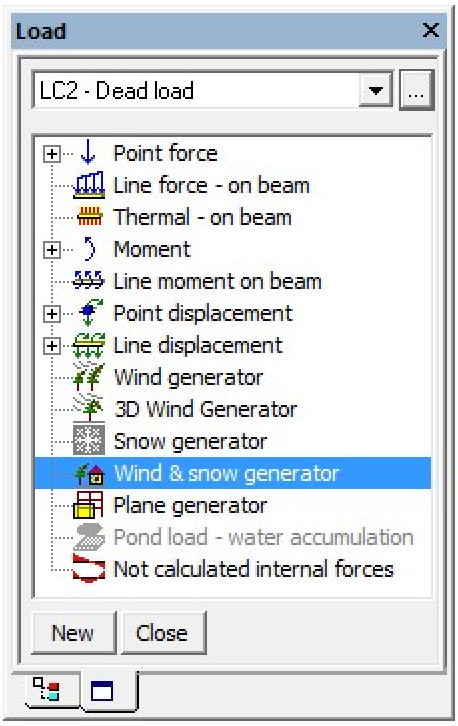
3. Set the Value to -5,0 kN/m and System to GCS
4. Confirm your input with [OK].
5. Select all rafter beams to enter defined load
6. Press <Escape> to finish the input.
7. Press <Escape> once more to finish the selection.

Wind

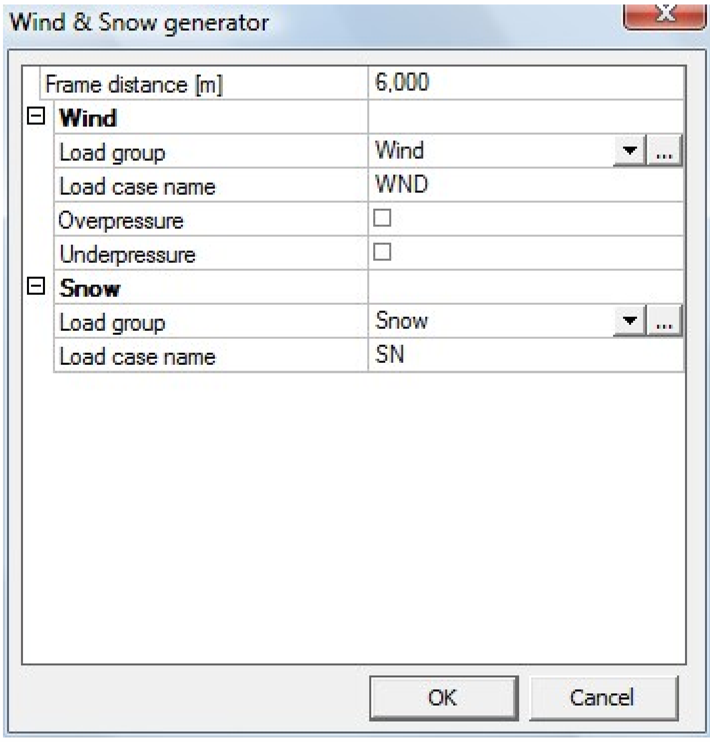
1. Set workplane as XZ by an option Main menu→Tools→UCS→XZ workplane



2. Click Wind & Snow generator in the Load Menu.

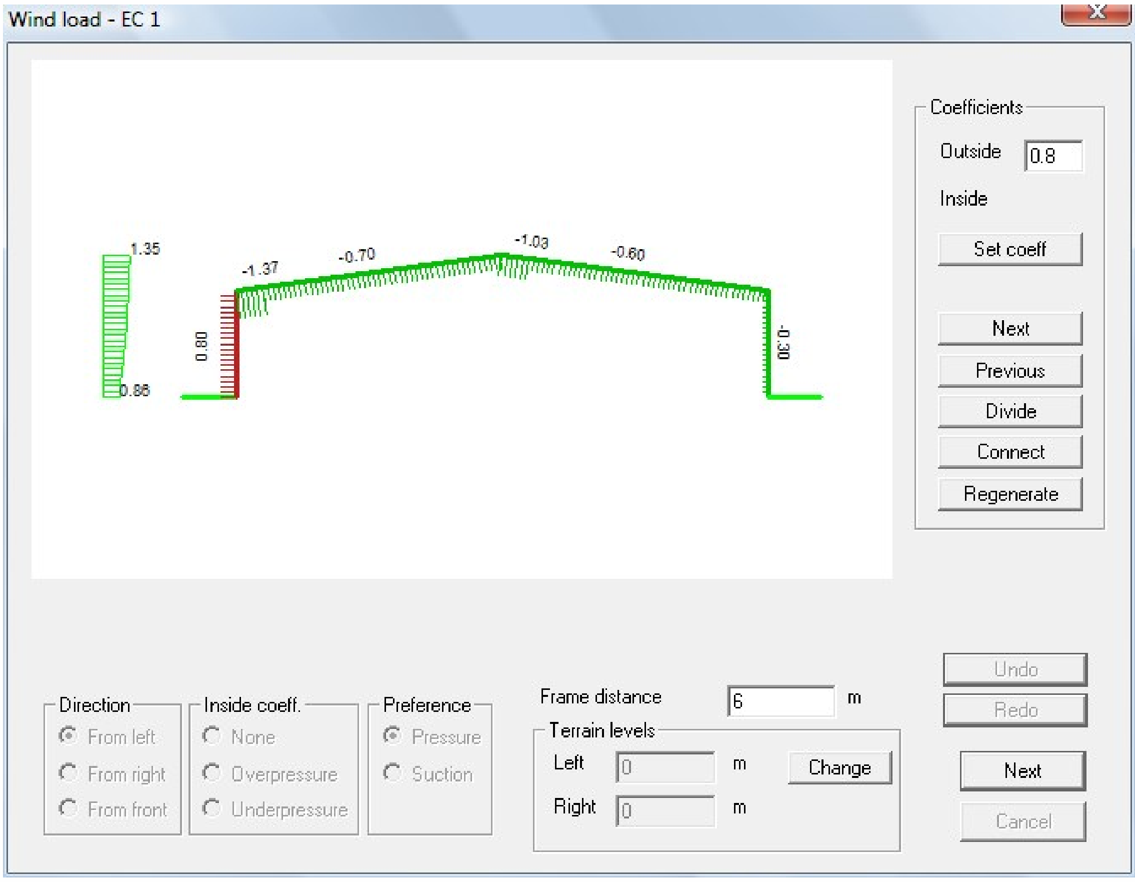


3. Following dialogue appears.

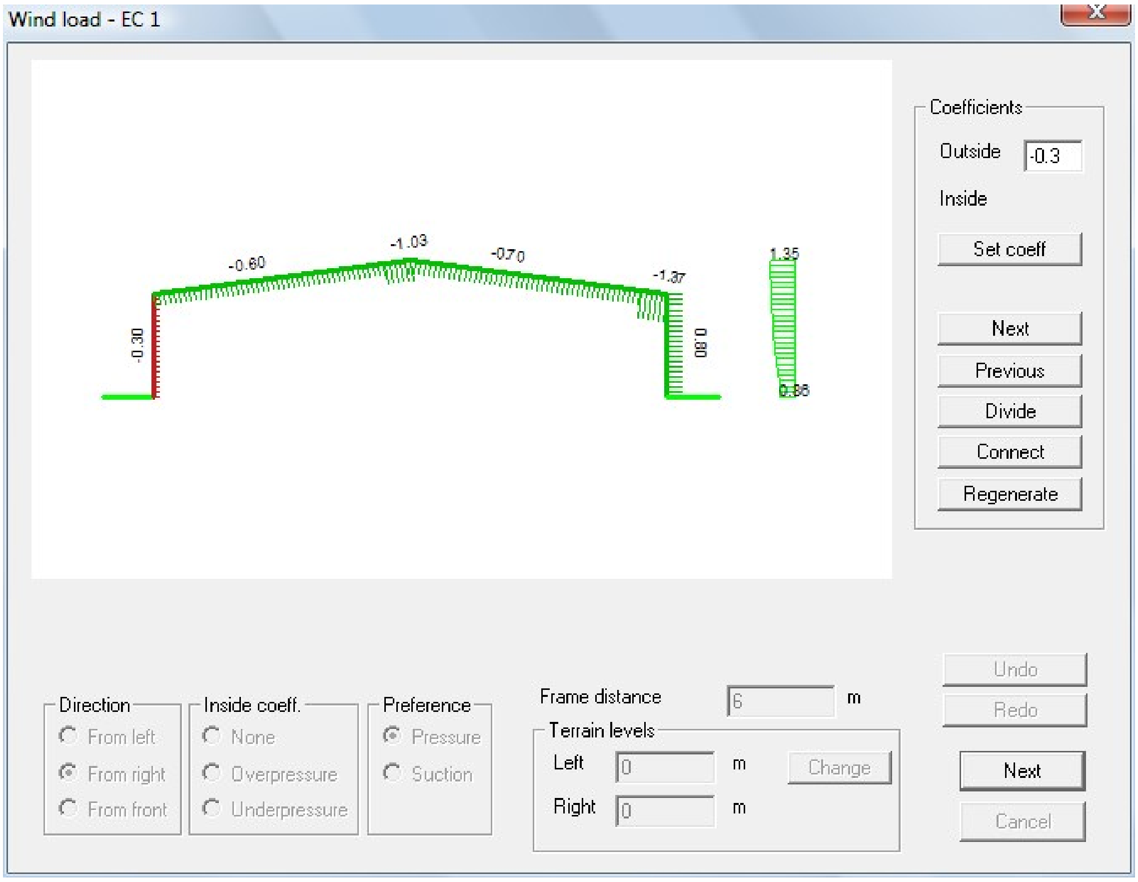


4. Confirm default settings with [OK].

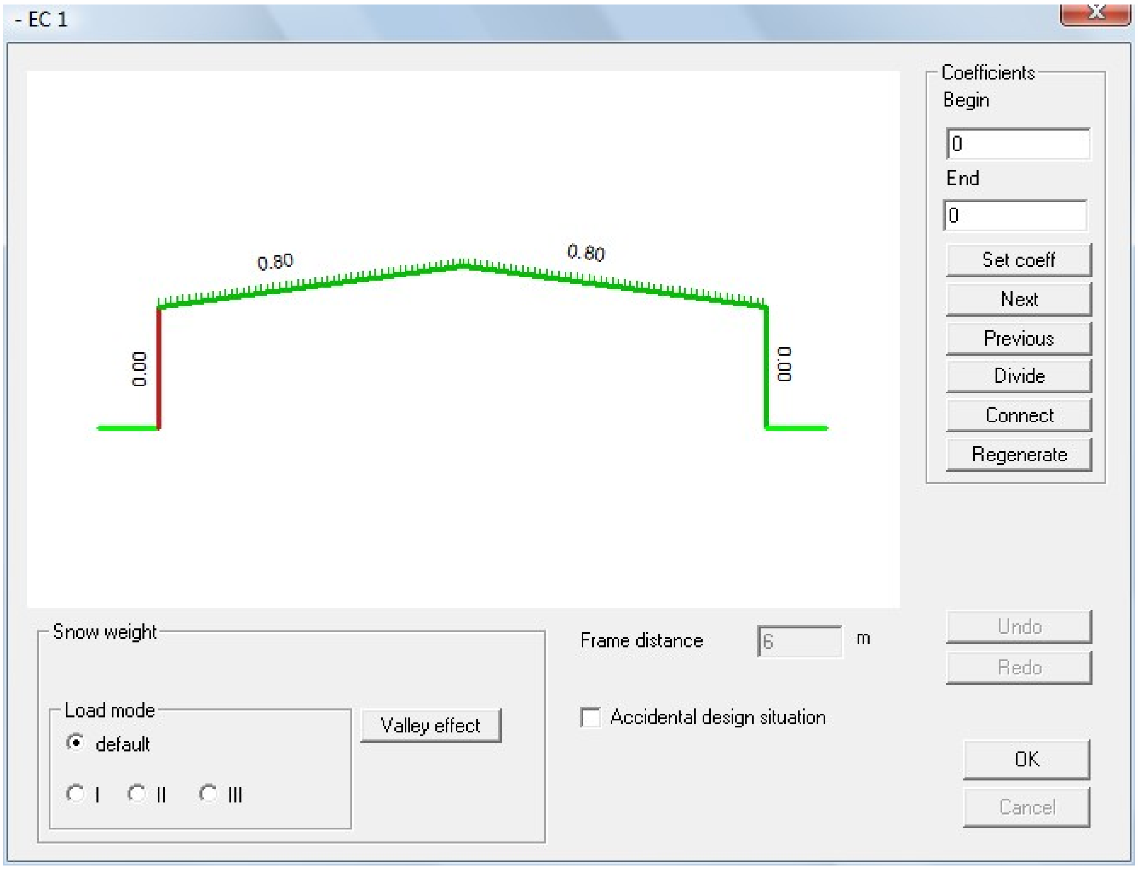
5. Left wind load generator manager appears.



6. Accept default settings and press [Next].
7. Left wind load generator manager appears.

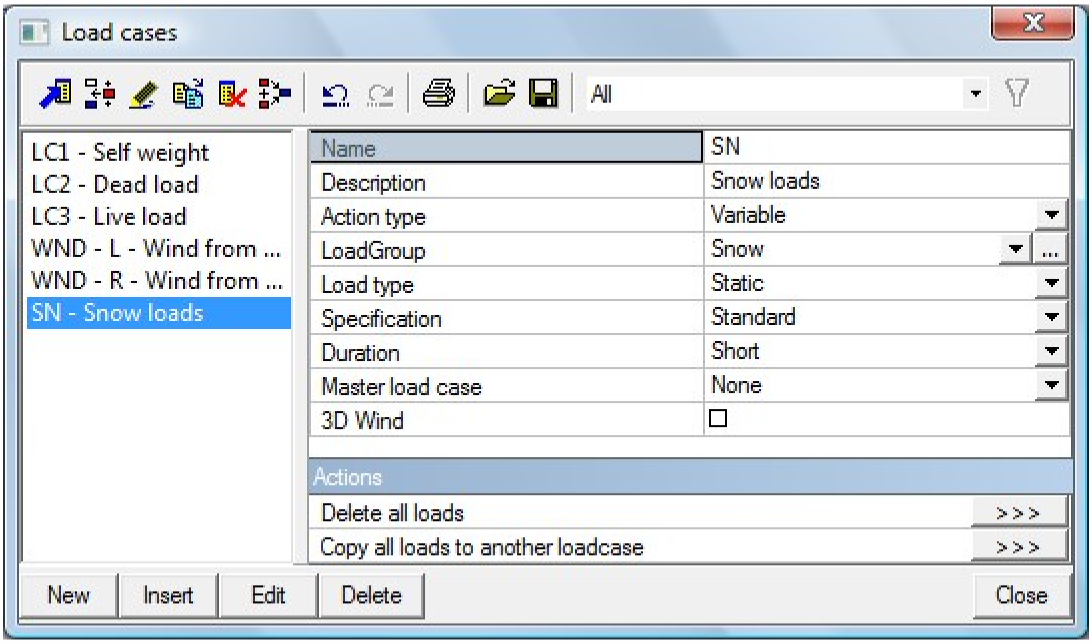


8. Accept default settings and press [Next].
9. Snow load generator manager appears.



10. Accept default settings and press [OK].

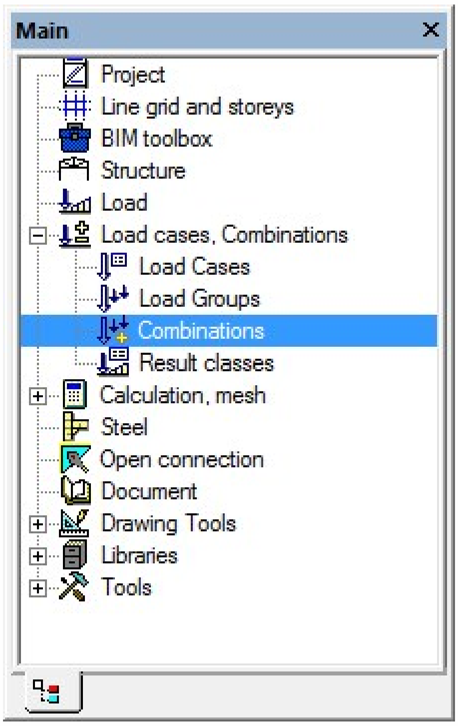
11. New loadcases:
WND-L-Wind from left
WND-L-Wind from left
SN-Snow loads
are generated



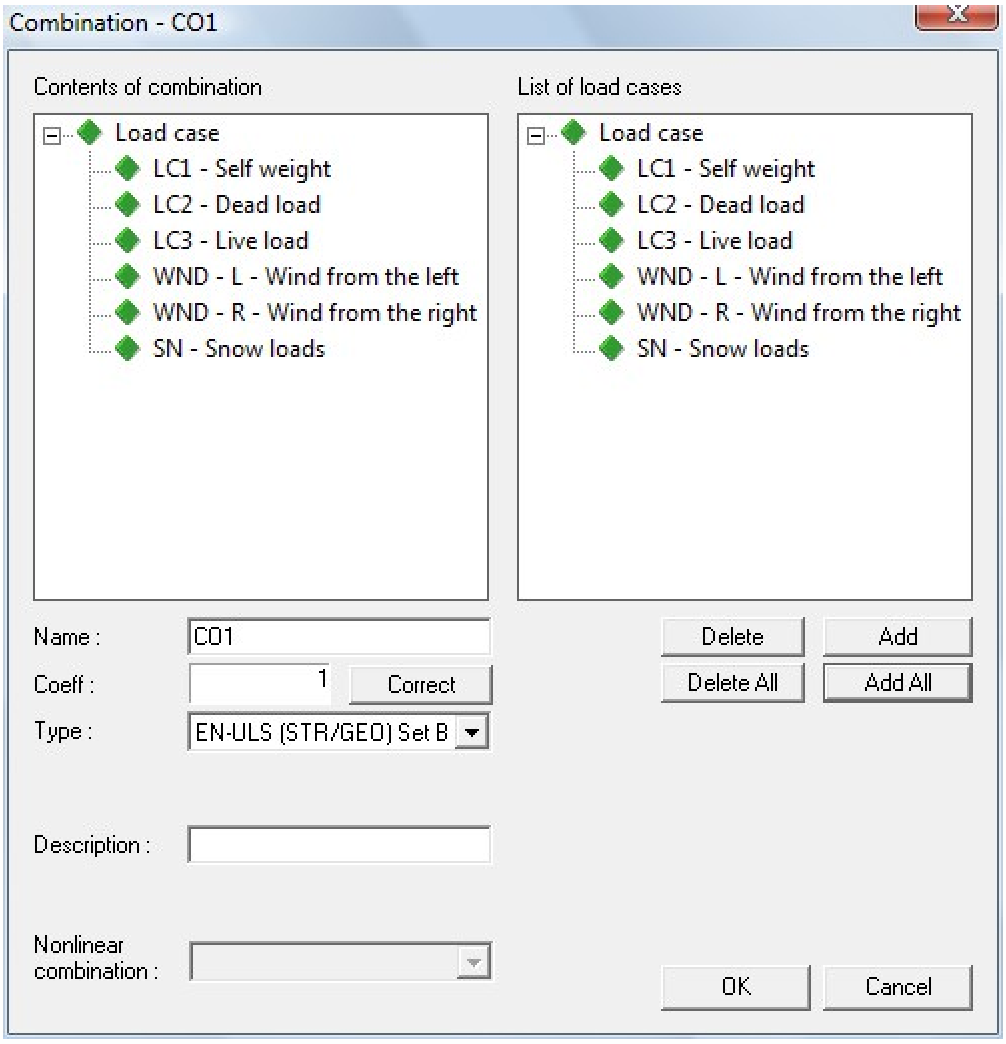
Combinations

After input of the load cases, the latter can be grouped in combinations. In this project, two linear combinations are created, one for the Ultimate Limit State and one for the Ultimate Serviceability State.

1. Double-click on Combinations below Load cases, Combinations in the Main window.

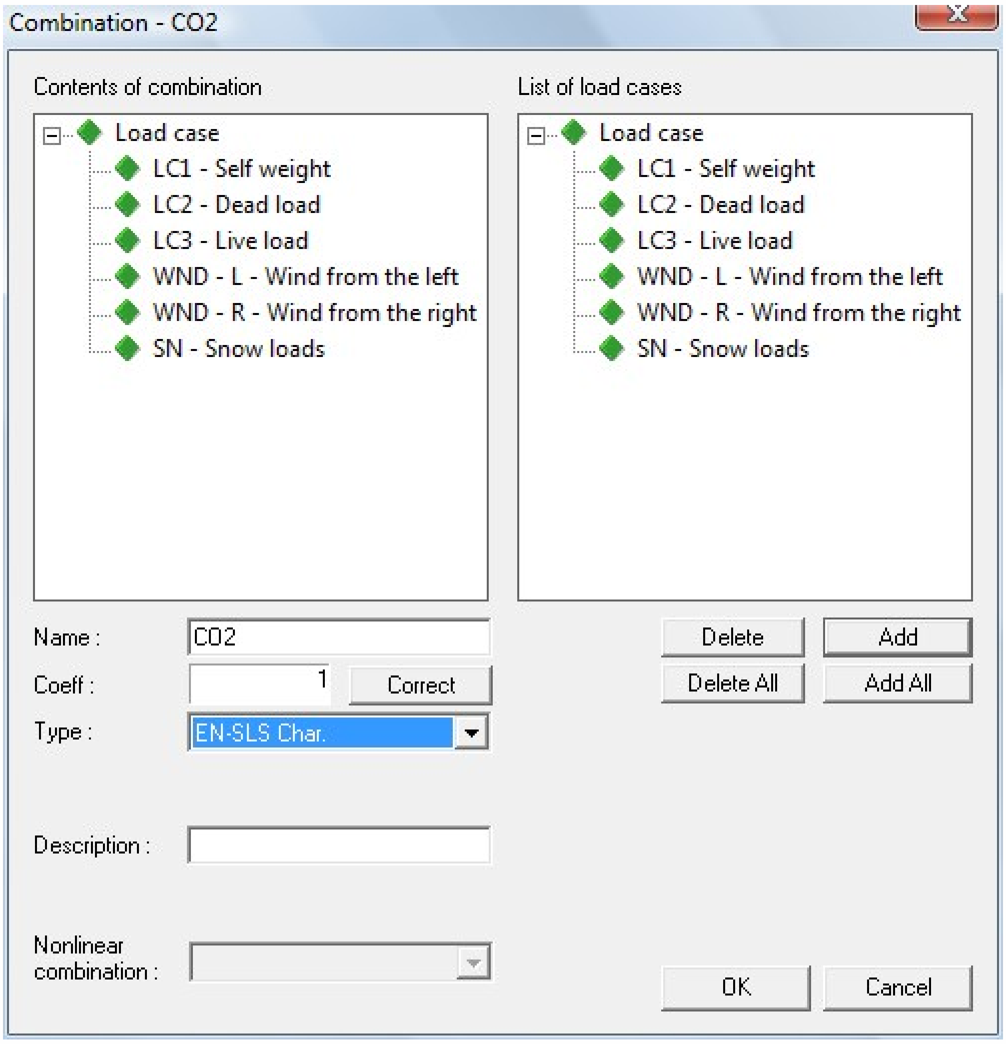


2. Since no combination has been entered yet, the window to create a new combination will automatically appear.

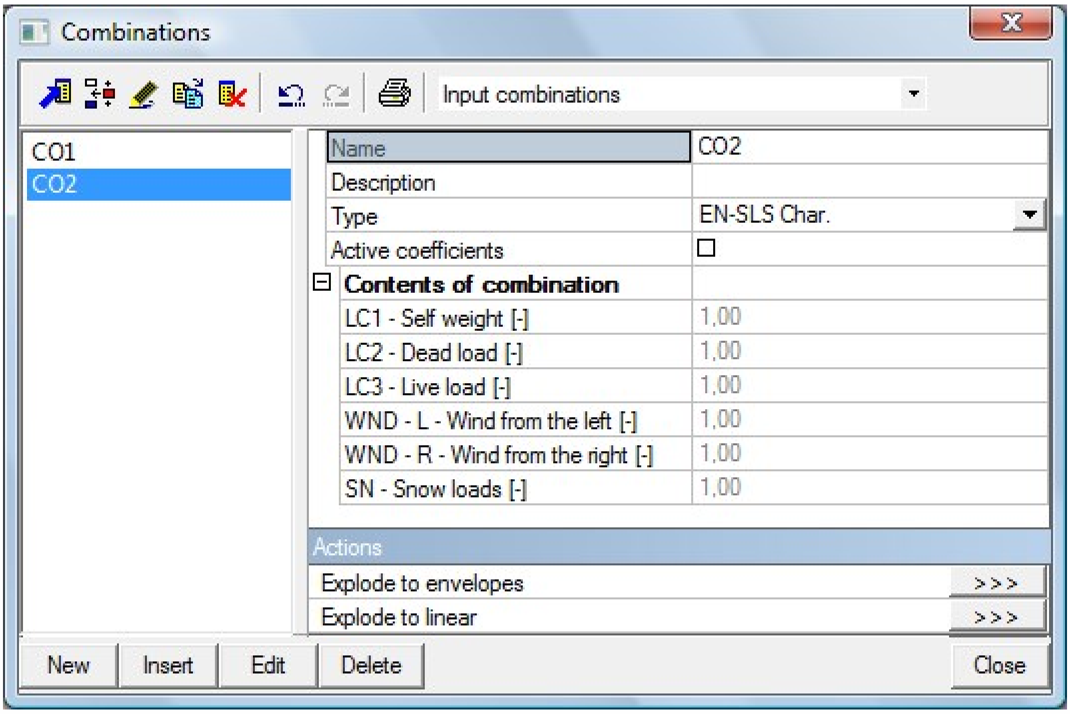


3. Change the type of the combination EN-ULS (STR/GEO) Set B. With this combination type, SCIA Engineer will automatically generate combinations in accordance with the composition rules of the Eurocode.
4. By means of the button [Add all], all load cases can be added to the combination.
5. Confirm your input with [OK]. The Combination Manager is opened.
6. Click 

 or  to create a second combination.
7. Change the Type of the combination to EC - SLS char.

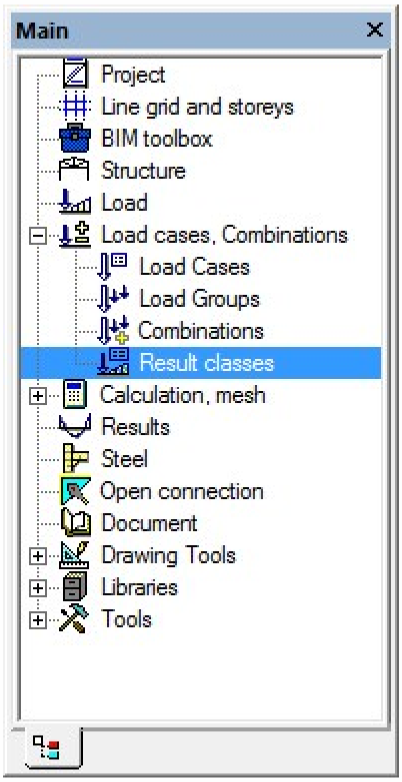


8. Confirm your input with [OK].
9. Click [Close] to close the Combination manager.



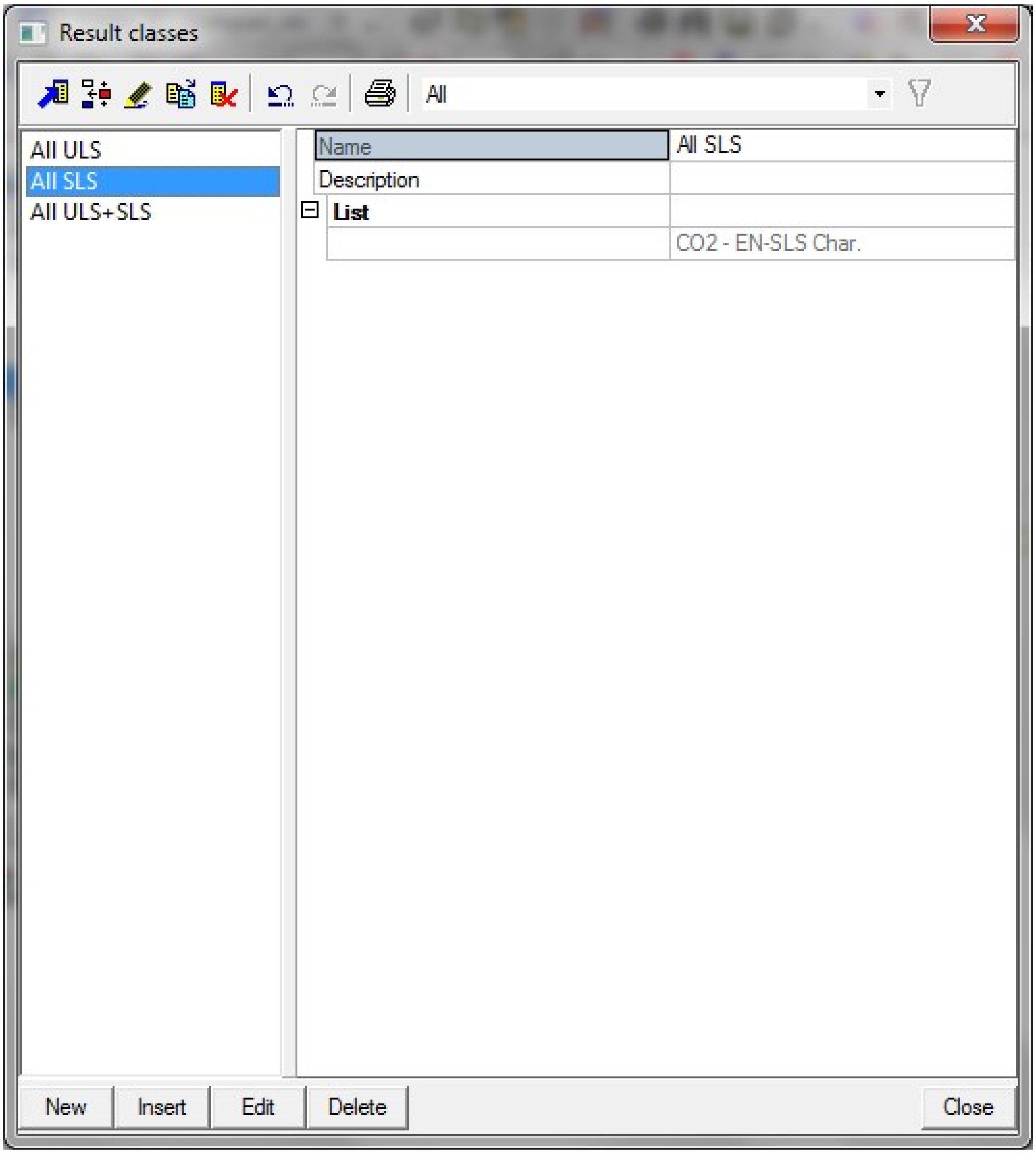
Classes

1. Double-click on Result classes below Loade cases, Combinations in the Main window.

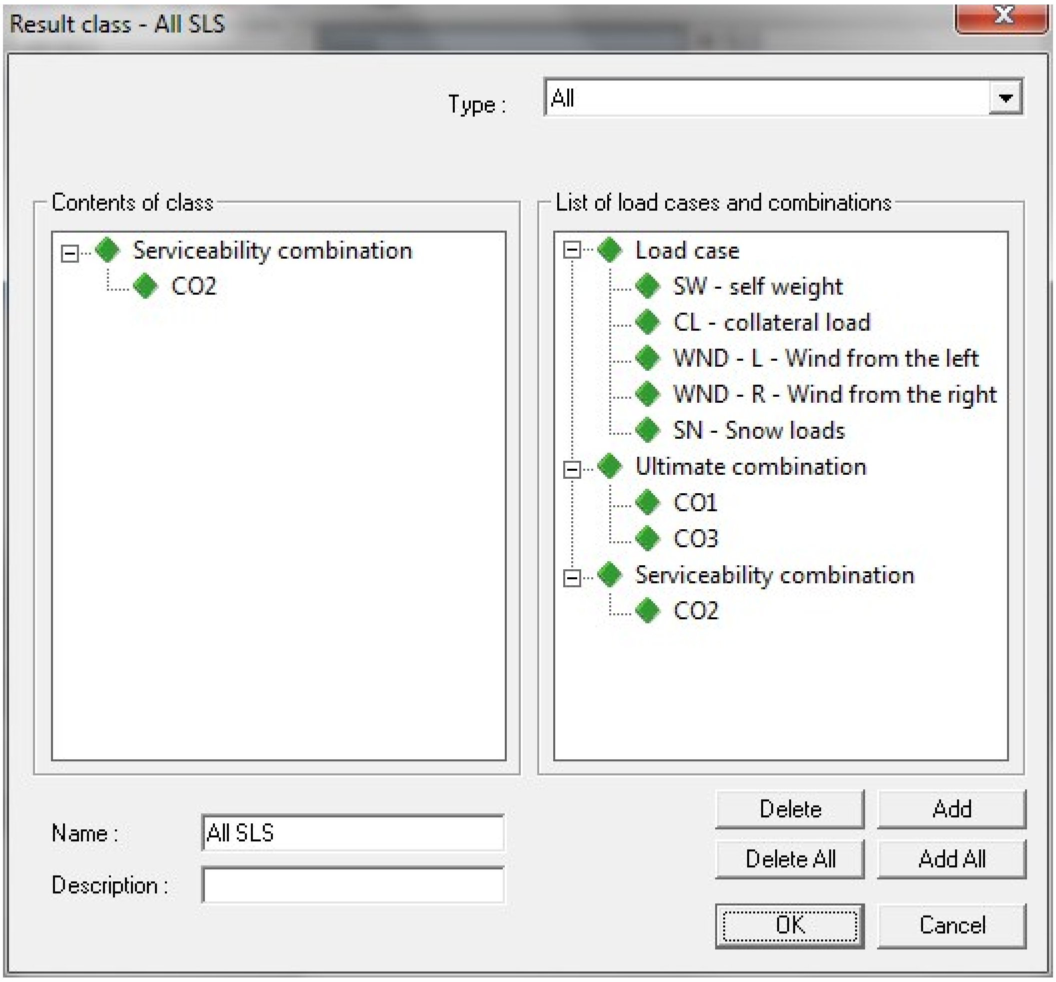


2. Select class All SLS and click 

.



3. All SLS class dialog appears

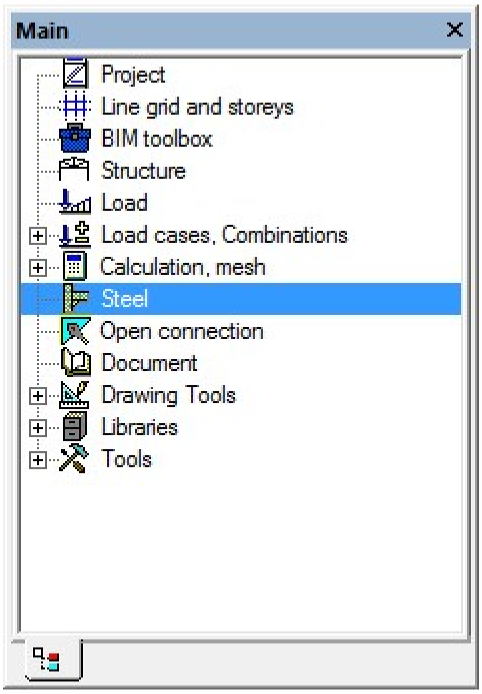


4. Check Contents of class. Combination CO2 is automatically added.
5. Confirm your input with [OK].

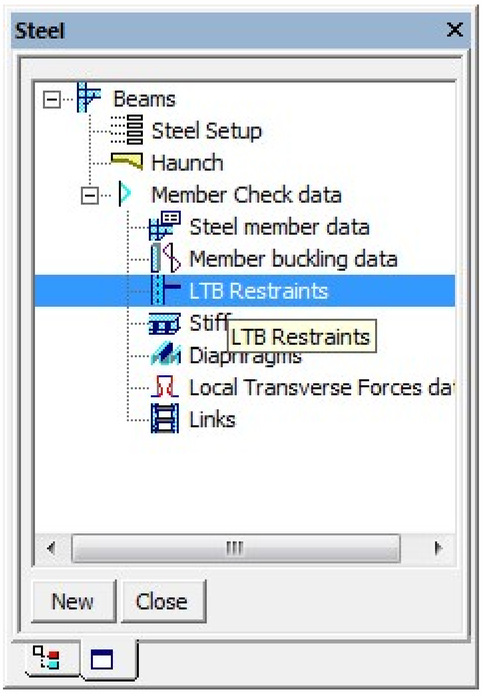
Steel

LTB restrains

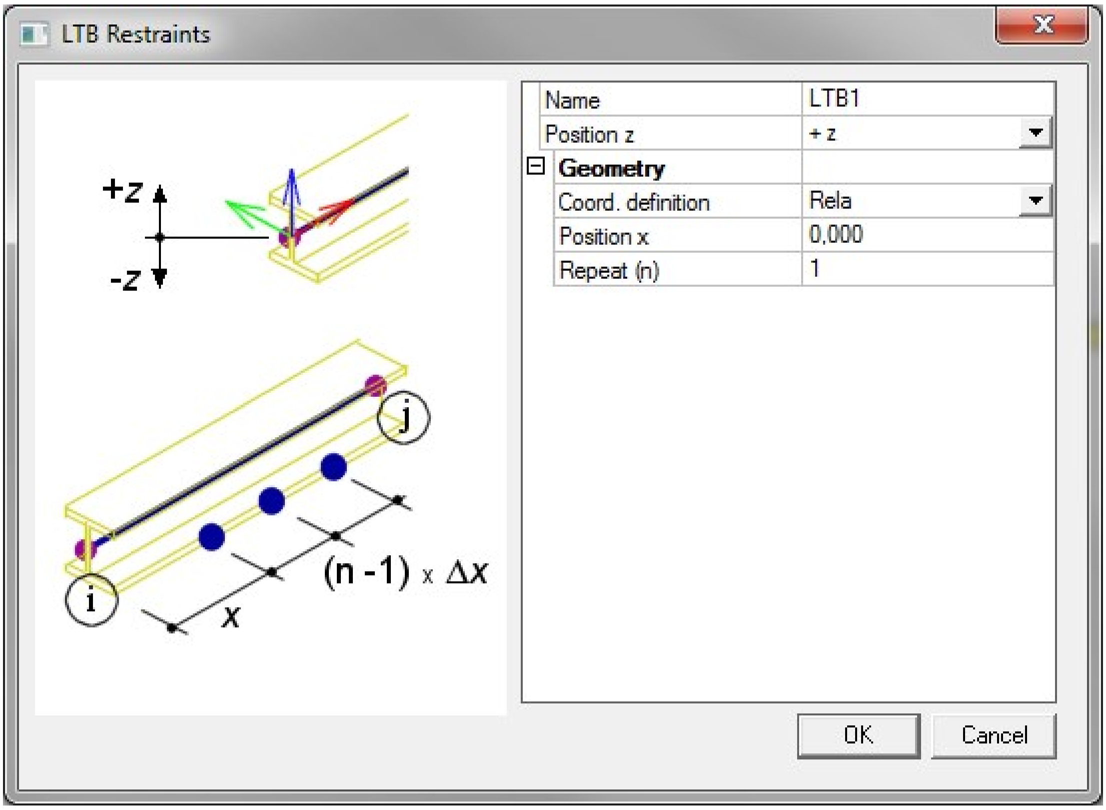
1. Double-click on the service Steel in the Main window.



2. Double-click on the LTB Restraints in the service Steel



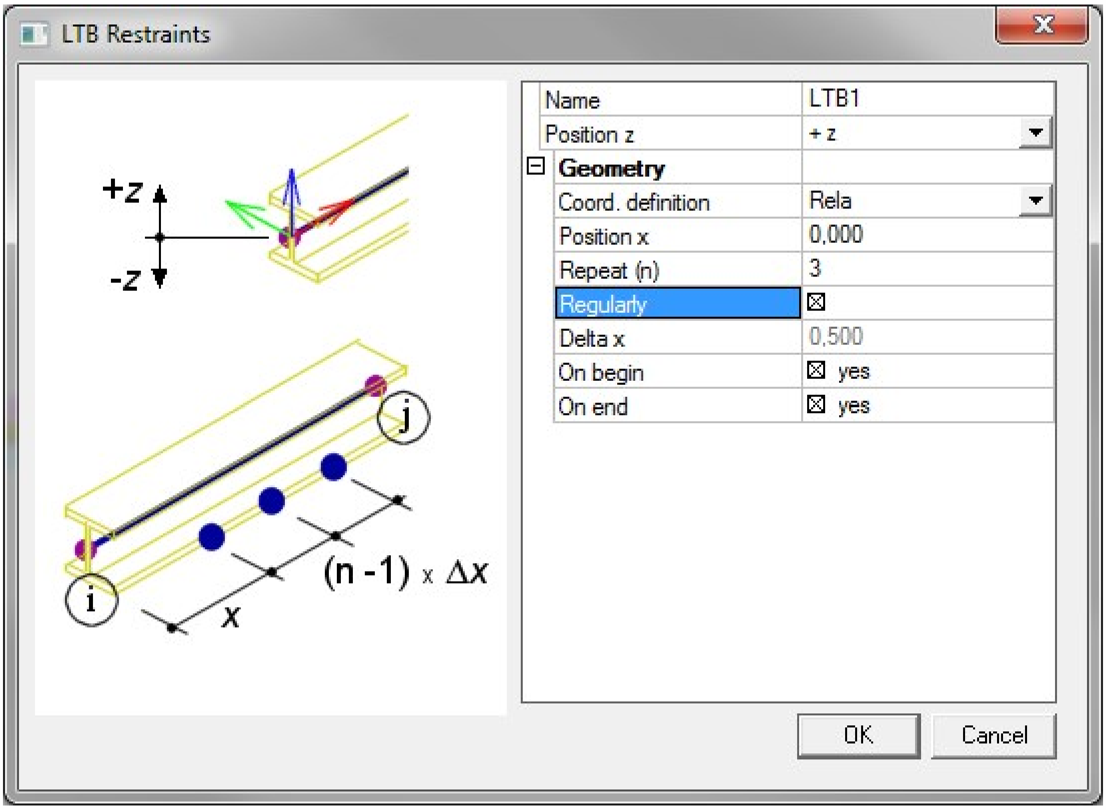
3. LTB restraints dialog appears



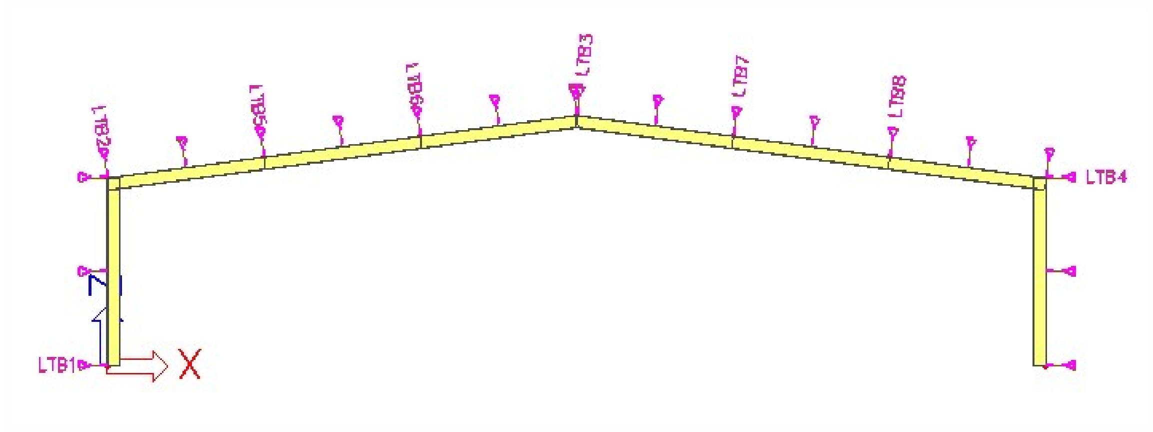
4. Set the value of Repeat (n) to 3.

5. Active Regularly check box.

6. Confirm your input with [OK].



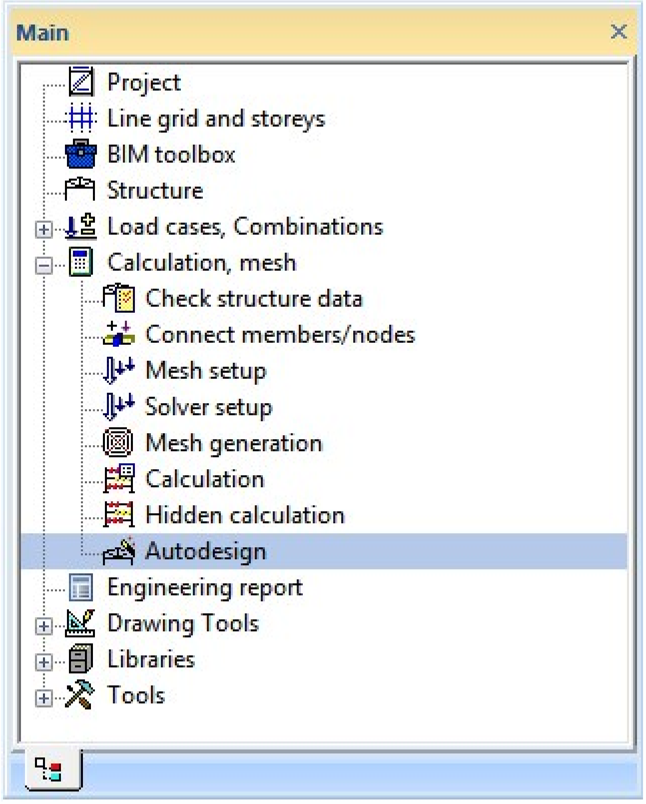
7. Select all members to enter LTB restraints.
8. Press <Escape> to finish the input.



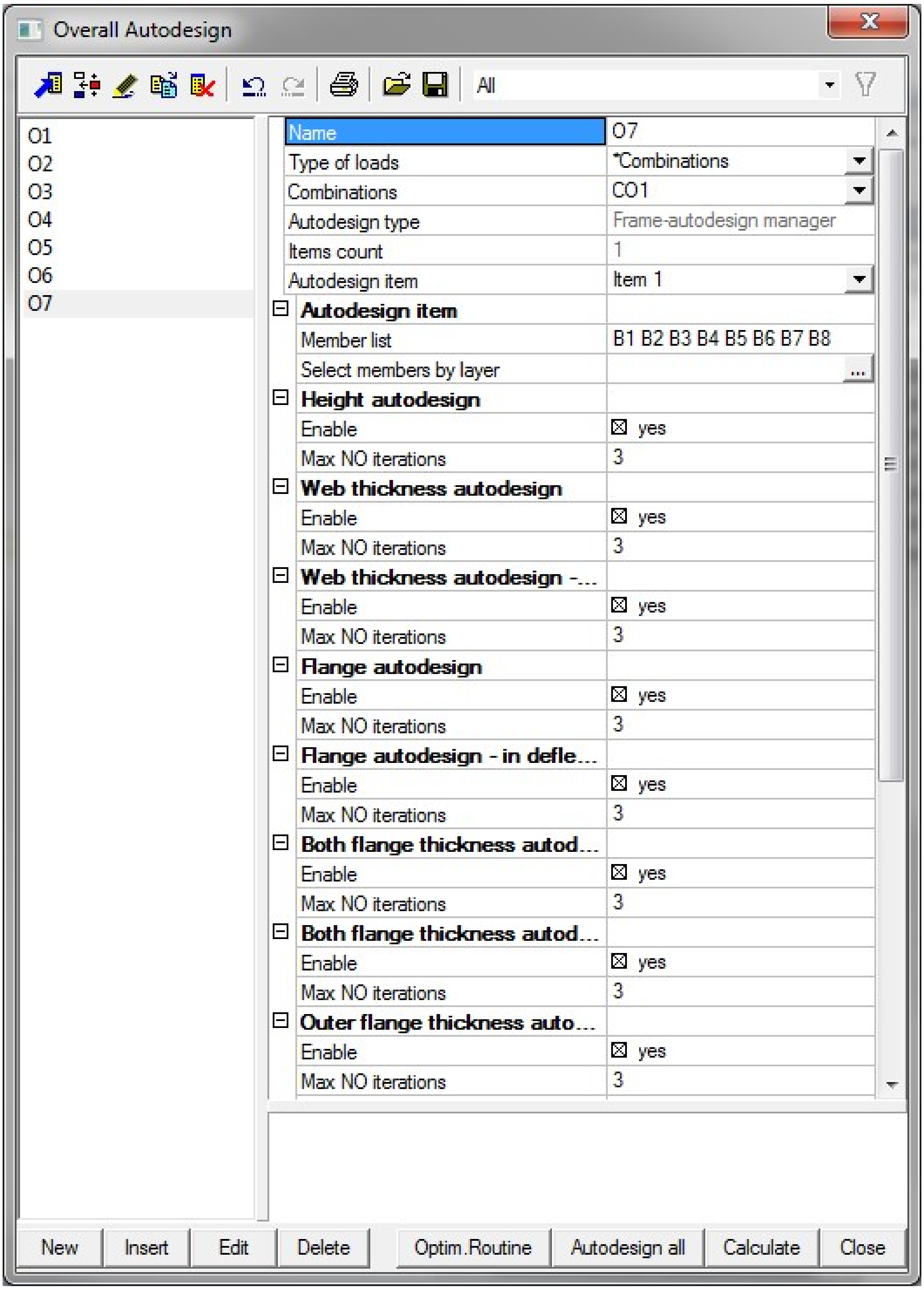
9. Press <Escape> once more to finish the selection.

Optimization

1. To start frame optimization process use the option Main window→ Calculation, mesh→Autodesign

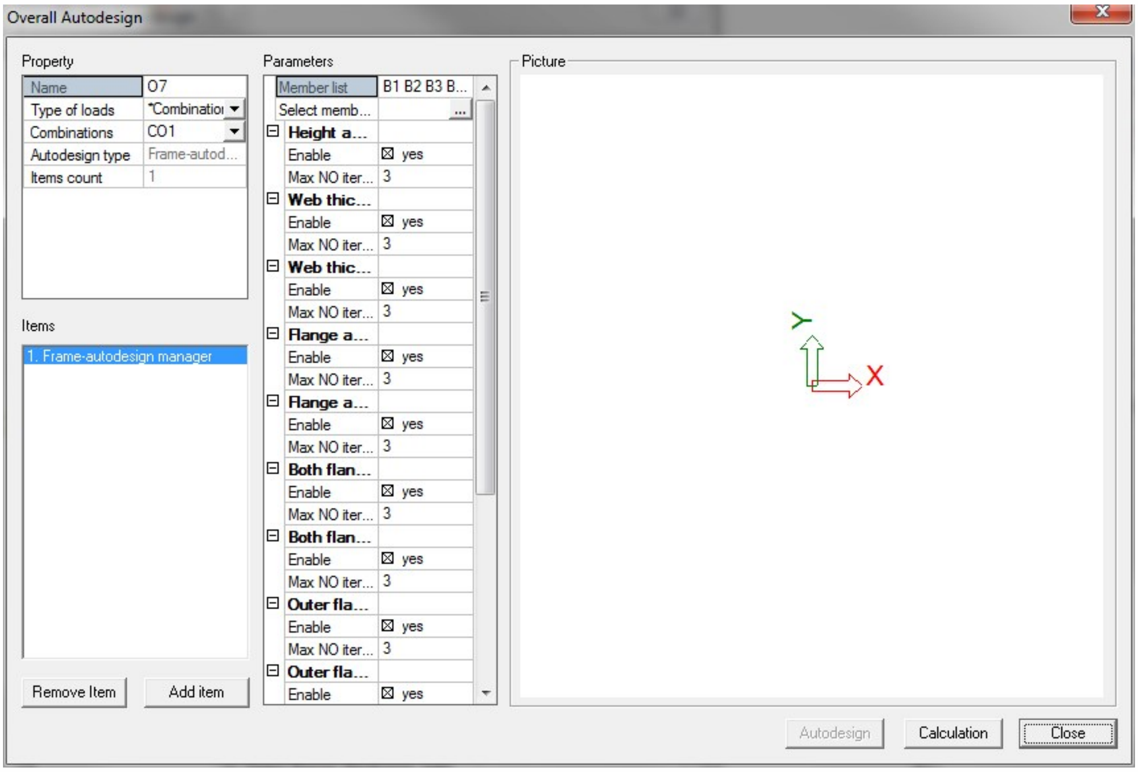

Main
Project
Line grid and storeys
BIM toolbox
Structure
Load cases, Combinations
Calculation, mesh
Check structure data
Connect members/nodes
Mesh setup
Solver setup
Mesh generation
Calculation
Hidden calculation
Autodesign
Engineering report
Drawing Tools
Libraries
Tools

2. Overall Autodesign dialog appears.

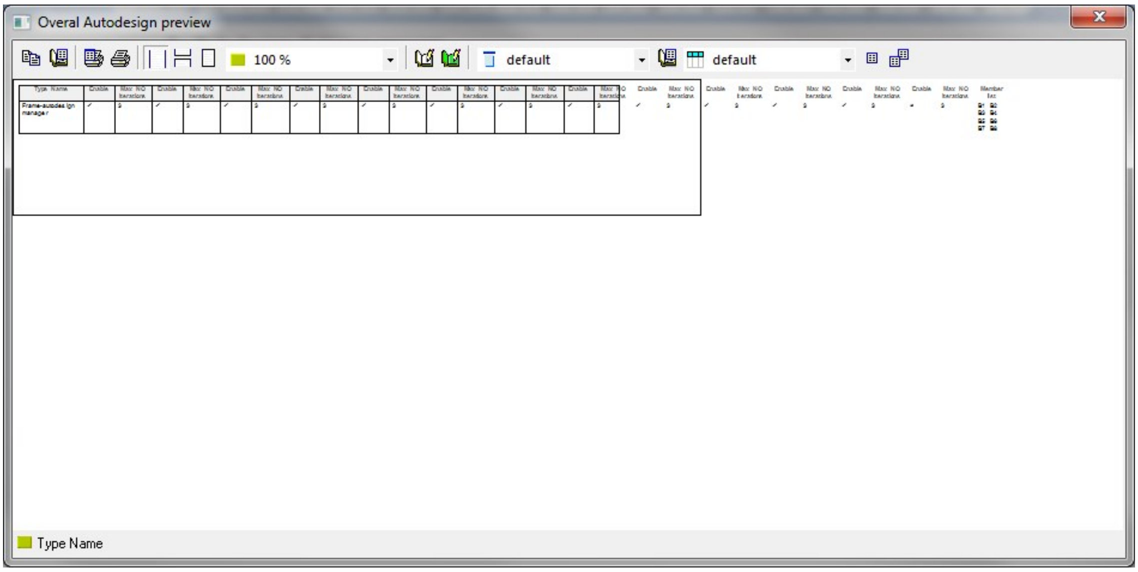


3. Select autodesignO7 (Frame-autodesign manager and click [Edit] button.

4. Autodesign dialogue appears



5. Click [Calculation].
6. Structure is calculated.
7. Click [Autdesign].
8. Autodesign preview appears

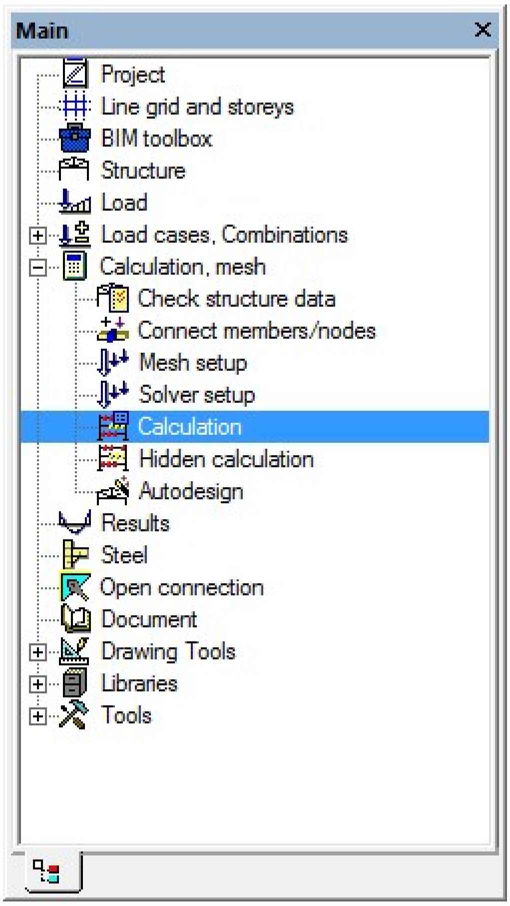


9. Close autodesign preview window
10. Click [Close] to close Autodesign dialogue.
11. Click [Close] to close Overall Autodesign dialogue.

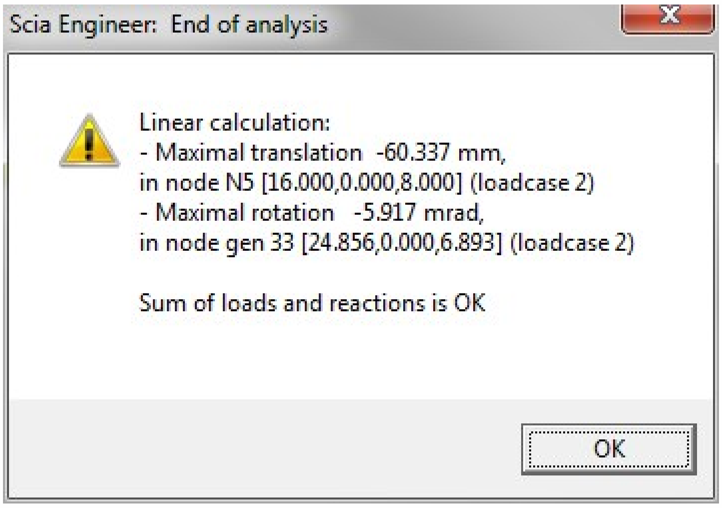
Check of designed structure

Calculation

1. Double-click on the Calculation in the service Calculation, mesh in the Main window.



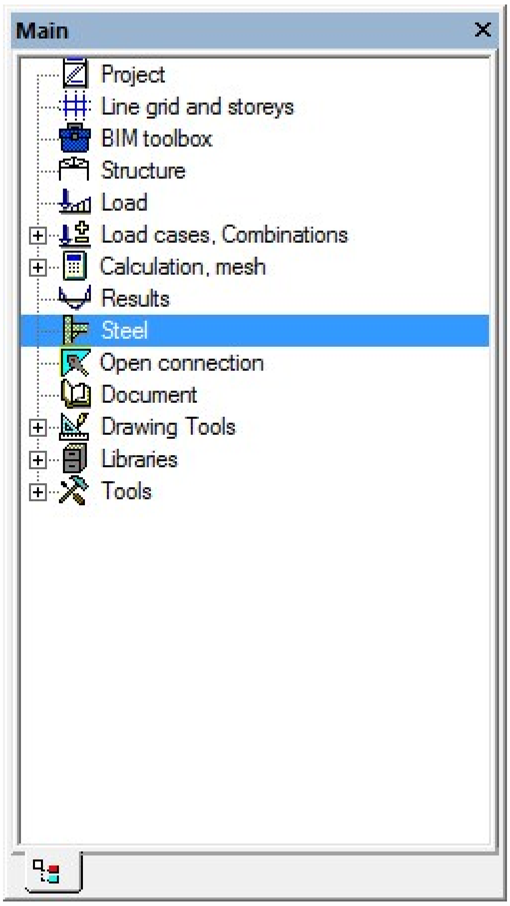
2. FE analysis dialogue appears
3. Click on [OK].
4. Analysis report appears.



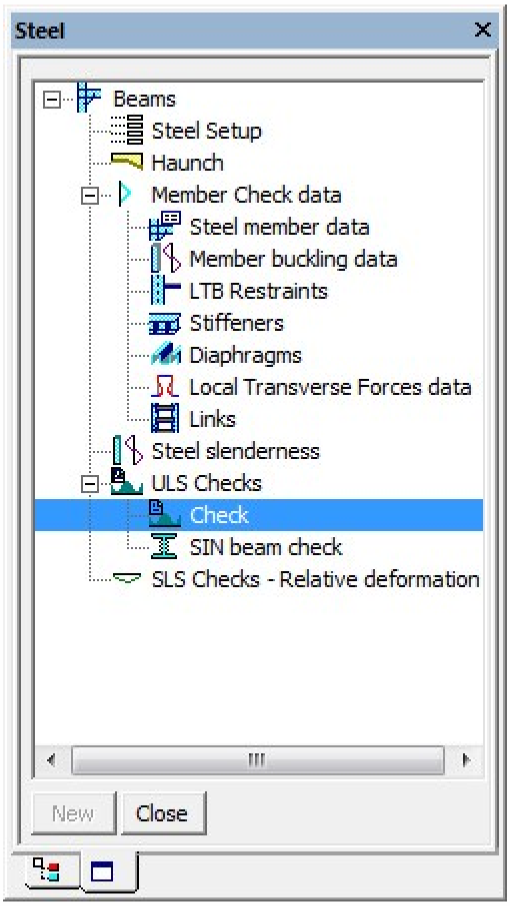
5. Click [OK] to close.

Steel checks

1. Double-click on the Steel service in the Main window.

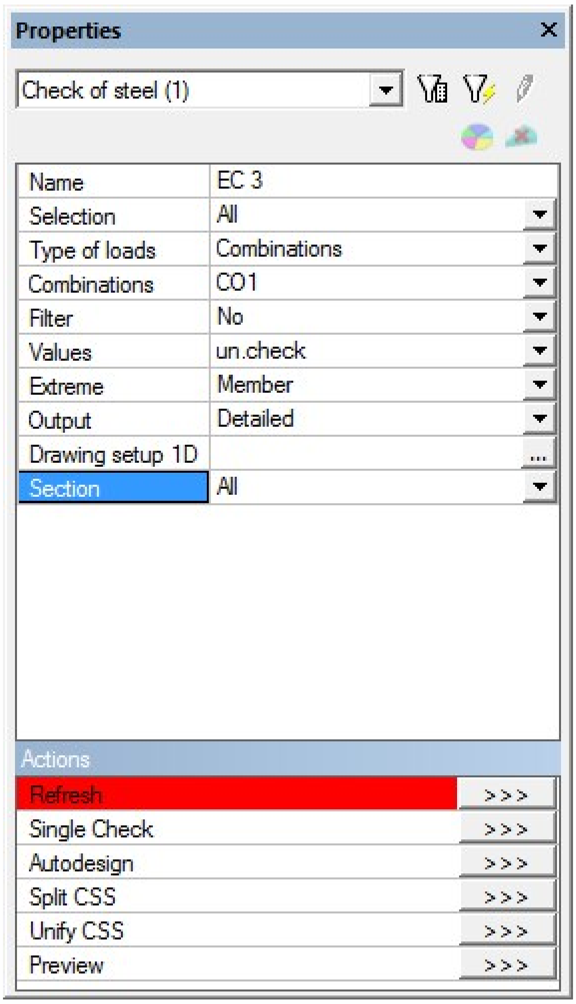


2. Click on Check from the subgroup ULS Checks of the group Beams in the Steel service.

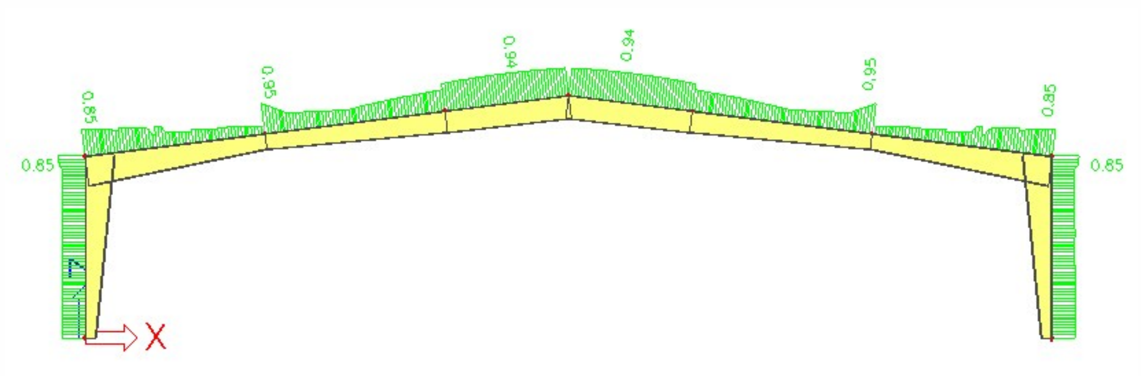


3. Define following settings in the Properties window.

Selection = all
Type of loads = Combinations
Filter = No
Values = un. check
Extreme = Member
Output = Detailed
Section = All

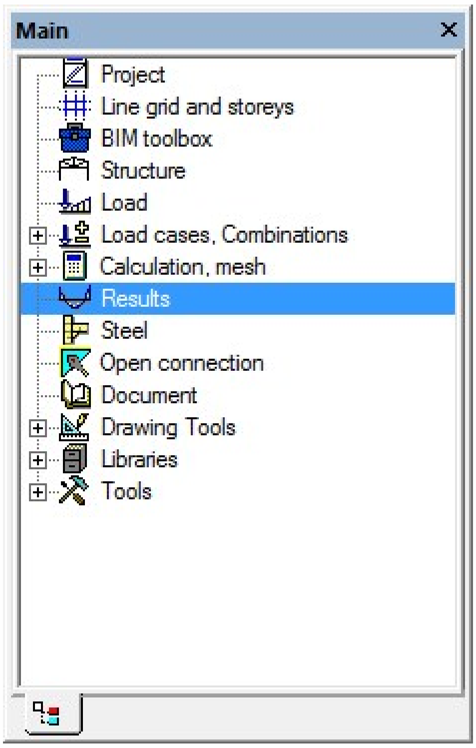


4. Click Refresh in the Properties window.
5. Check displayed steel checks values.

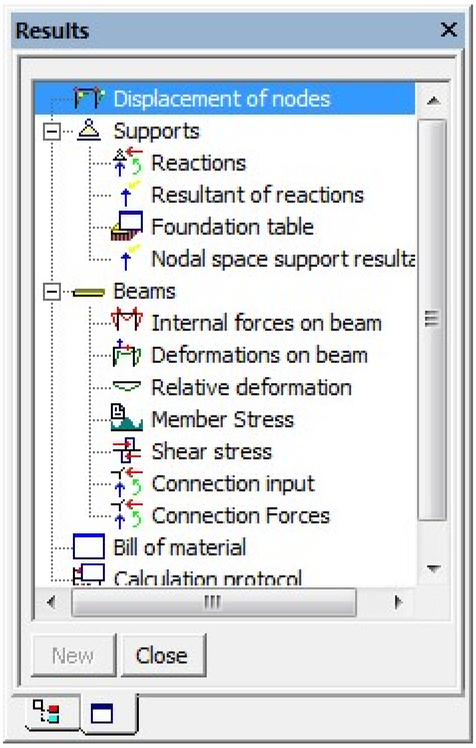


Displacements of nodes

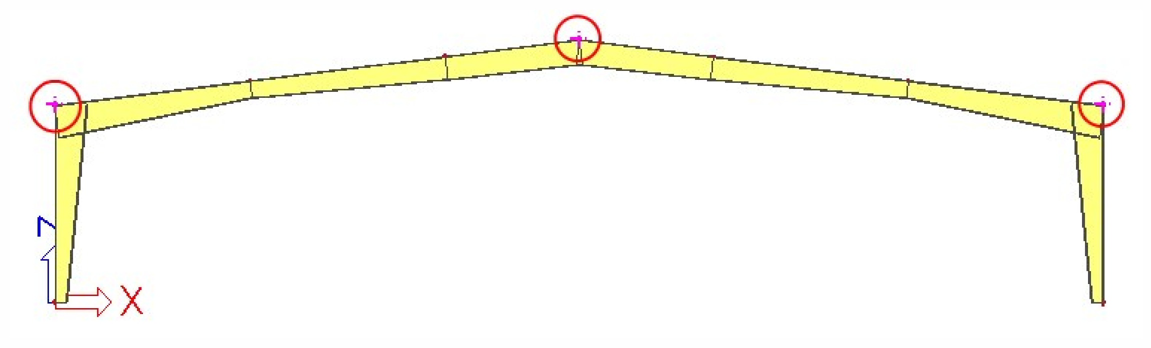
1. Double-click on the Results service in the Main window.



2. Click on Displacement of nodes

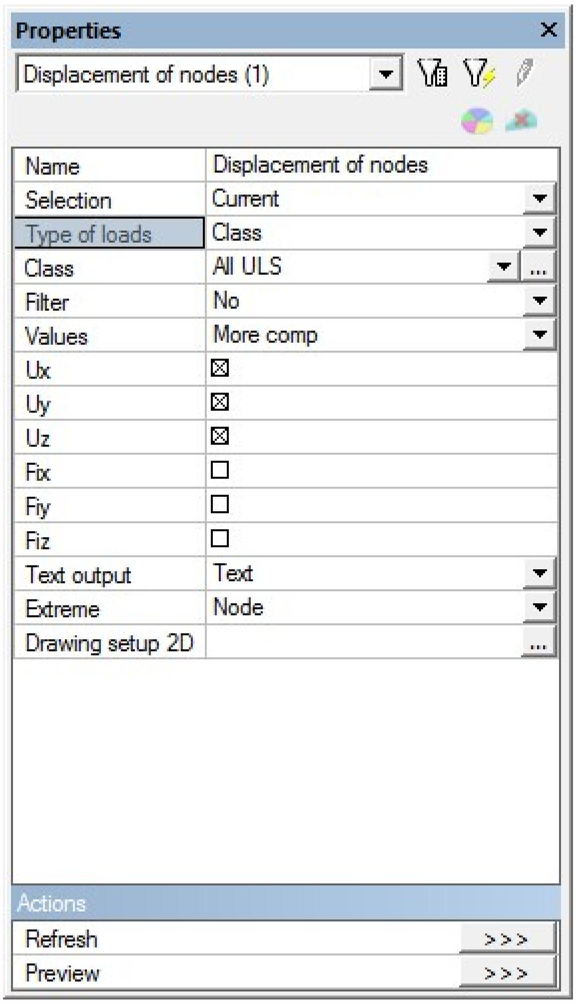


3. Select eave nodes and top node.



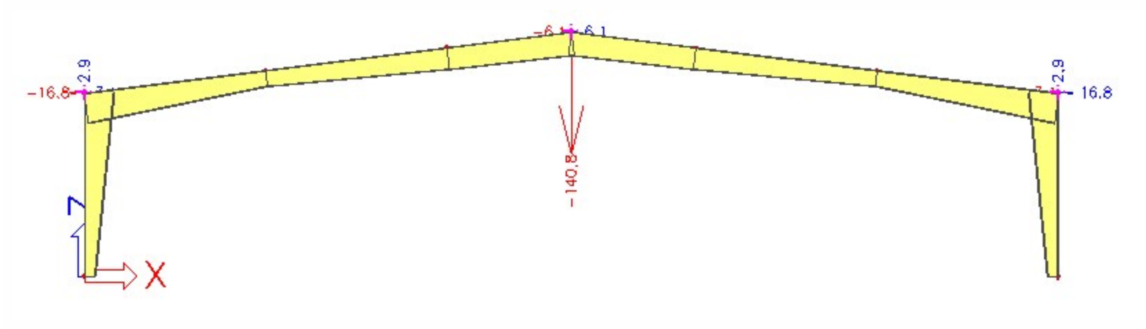
4. Define following settings in the Properties window.

Selection = Current
Type of loads = Class
Class = All ULS



5. Click Refresh in the Properties window.

6. Check displayed displacements of nodes.



Global optimisation

Introduction

SCIA Engineer enables you to perform an optimisation of the whole structure or of its selected part. The optimisation can be run for steel and timber structures or for steel or timber parts of multi-material projects.

It is possible to optimise the value of:

- standard steel code check,
- fire resistance steel check,
- timber code check,
- bolted diagonal check.

It is also possible to perform several of the above mentioned optimisation types and then compare the results.

It is always the cross-section size or the bolt size that is optimised. In general, you must select which cross-section types or bolted diagonal connections used in your model are to be optimised. And it is up to you to select the cross-section types and bolted diagonal connections that are relevant to your work. It is also your responsibility to think in advance and define and assign to 1D members as many cross-section types as necessary for a proper design and optimisation of the project.

> Note : In order to perform the AutoDesign, calculation must be already performed.

AutoDesign manager

As stated in the introduction you may perform several different optimisations. You may run the AutoDesign and compare the results for different parts of the structure, for different optimisation types (e.g. standard and fire resistance code check). Therefore, all the defined optimisations are stored in the AutoDesignmanager. Thus you do not have to define all the AutoDesign criteria and parameters again and again.

TheAutoDesign manager is a standard SCIA Engineer database manager with usual features and functions.

Procedure to open the AutoDesign manager

1. Open service Calculation, Mesh.
2. Start (double-click) function AutoDesign.

Defining a new optimisation

Procedure to define and run a new optimisation

1. Start the AutoDesign manager.
2. Click button [New] to open the OverallAutoDesign dialogue.
3. Define the AutoDesign parameters and criteria.
4. Click button [AutoDesign] to run the calculation and see its result.

5. If required, click button [Calculation] to re-calculate the model in order to reflect the results of the optimisation.
6. Depending on what you exactly need and want, you may repeat steps 3 to 5 as many times as required.

Note: Please note, that a mechanical repetition of AutoDesign and Calculation in turns may lead to a "never-ending" cycle. The AutoDesign may find cross-section "A" as optimal. When you perform the calculation, the internal forces are redistributed to reflect the AutoDesign results. When you run AutoDesign now, it may find cross-section "B" as optimal. And another re-calculation once more redistributes the internal forces. And it may happen that the subsequent AutoDesign finds the cross-section "A" as optimal once again. And so on, and so on, and so on.

AutoDesign parameters and criteria

Items

The item defines the type of the optimisation and the cross-section type that should be optimised. The type of optimisation (e.g. standard and fire resistance code check) must be defined for the first item only. All the other items in one AutoDesign definition are of the same type. One AutoDesign item represents one cross-section type or one bolted diagonal connection that will be optimised.

[Add item]	Adds a new optimisation item into the list.
[Remove item]	Removes the existing optimisation item from the list.

Property

Name	Defines the name of the optimisation (criteria).
Type of loads	The AutoDesign may be performed for load cases, load case combinations, result classes, etc.
Load	Specifies the particular load case, combination, etc. for which the selected cross-section type will be optimised.
AutoDesign type	(informative) Tells the type of the optimisation.
Item count	(informative) Shows the number of defined AutoDesign items.

Parameters

Cross-section AutoDesign

Cross-section	Defines the cross-section type to be optimised.
Parameter	Selects the dimension (e.g. section depth, width, etc.) that will be optimised.
Length	(informative) Shows the current size of the selected dimension.
Minimum	Defines the minimal applicable size for the optimised parameter.
Maximum	Defines the maximal applicable size for the optimised parameter.
Step	Defines the step for the AutoDesign.
Maximal check	Defines the maximal acceptable value of unity check of the optimised cross-section.
Optimised check	(informative) Shows the unity check for the optimised connection.

Bolted diagonal AutoDesign

Bolted diagonal	Specifies the bolted diagonal to be optimised.
Bolt	Specifies the bolt used.
Optimised check	(informative) Shows the unity check for the optimised connection.

Picture

The picture shows the shape of the optimised cross-section or the symbol of the bolted diagonal connection.

Control buttons

AutoDesign	Performs the optimisation for the defined AutoDesign items.
Calculation	Carries out the calculation for the optimised model.