

Version
February 2016

Program

RFEM

**Finite Element Analysis
Software**

Training Manual

All rights, including those of translations, are reserved.

No portion of this book may be reproduced – mechanically, electronically, or by any other means, including photocopying – without written permission of DLUBAL SOFTWARE GMBH.

© **Dlubal Software GmbH**

Am Zellweg 2 D-93464 Tiefenbach

Tel.: +49 (0) 9673 9203-0

Fax: +49 (0) 9673 9203-51

E-Mail: info@dlubal.com

Web: www.dlubal.com

Contents

	Contents	Page		Contents	Page
1.	About RFEM	6	2.4.2	LC 2: Imposed Loads	51
1.1	Finite Elements	6	2.4.3	LC 3: Wind in +X	53
1.2	Graphical User Interface	6	2.4.4	LC 4: Wind in +Y	57
1.3	Project Management	7	2.4.5	LC 5: Imperfection Towards +X	59
1.3.1	New Project	7	2.4.6	LC 6: Imperfection Towards +Y	60
1.3.2	New Model	8	2.5	Calculation	62
2.	Concrete Model	9	2.5.1	Plausibility Check	62
2.1	Sketch of System	9	2.5.2	FE Mesh	62
2.2	Model Data	12	2.5.2.1	Generation	62
2.2.1	General Settings	12	2.5.2.2	Mesh Refinements	63
2.2.2	Floor Slab	12	2.5.3	Calculation Parameters	64
2.2.2.1	Rectangular Surface	12	2.5.4	Calculation	65
2.2.2.2	Rows of Columns	14	2.6	Results	66
2.2.2.3	Walls	17	2.6.1	Results Output	66
2.2.2.4	Semicircular Surface	18	2.6.2	Deformations	67
2.2.3	Ceiling Slab	20	2.6.2.1	Surfaces	67
2.2.3.1	Surfaces	20	2.6.2.2	Members	68
2.2.3.2	Opening	24	2.6.3	Internal Forces	69
2.2.4	Walls	25	2.6.3.1	Selected Results of Surfaces	69
2.2.4.1	Lines	25	2.6.3.2	Specific Results of Members	72
2.2.4.2	Surfaces	28	2.6.4	Sections	75
2.2.5	Columns	31	2.6.5	Numerical Results	78
2.2.6	Downstand Beam	36	2.6.5.1	Work Window	78
2.2.7	Opening	38	2.6.5.2	Results Tables	81
2.2.8	Supports	40	2.7	Documentation	83
2.2.8.1	Line Supports	40	2.7.1	Printout Report	83
2.2.8.2	Surface Supports	41	2.7.2	Graphics	86
2.3	Load Cases and Combinations	43	2.7.2.1	Graphic Printout	86
2.3.1	Load Cases	43	2.7.2.2	Mass Print	88
2.3.2	Actions	46	2.7.3	Printout Report Header	90
2.3.3	Combination Expressions	47	2.7.4	Printout Report Template	90
2.3.4	Action Combinations	48	2.8	RF-CONCRETE Surfaces	91
2.3.5	Load Combinations	49	2.8.1	Input Data	91
2.3.6	Result Combinations	50	2.8.1.1	General Data	91
2.4	Loads	51	2.8.1.2	Reinforcement	92
2.4.1	LC 1: Self-Weight	51	2.8.2	Results	93

Contents

	Contents	Page		Contents	Page
2.8.2.1	Required Reinforcement	93	3.4	FE Mesh Settings	141
2.8.2.2	Serviceability Design	94	3.4.1	Node Refinement	141
2.8.2.3	Graphical Results	96	3.4.2	Line Refinement	143
2.8.2.4	Printout	97	3.5	Plausibility Check	145
2.9	RF-CONCRETE Members	99	3.6	Calculation	147
2.9.1	Downstand Beam	99	3.7	Results	150
2.9.1.1	General Data	99	3.8	RF-STEEL Surfaces	157
2.9.1.2	Ribs	100	3.8.1	Input Data	157
2.9.1.3	Supports	101	3.8.2	Results	159
2.9.1.4	Reinforcement	102	3.9	Printout Report	160
2.9.1.5	Required Reinforcement	103	4.	Stability Analysis	167
2.9.1.6	Provided Reinforcement	103	4.1	Model Data	167
2.9.1.7	Curtailment	105	4.1.1	Beam	167
2.9.1.8	Graphical Results	106	4.1.2	Openings	169
2.9.2	Columns	108	4.1.3	Supports	172
2.9.2.1	General Data	108	4.2	Load Data	173
2.9.2.2	Cross-Sections	110	4.2.1	LC1: Self-Weight	173
2.9.2.3	Reinforcement	110	4.2.2	LC2: Snow	173
2.9.2.4	Required Reinforcement	112	4.3	Load Combination	175
2.9.2.5	Provided Reinforcement	112	4.4	FE Mesh	176
2.9.2.6	Nonlinear Calculation	113	4.5	Calculation	177
2.9.2.7	Graphical Results	113	4.6	RF-STABILITY	178
2.9.2.8	Printout	116	4.6.1	Theory	178
3.	Steel Model	118	4.6.2	Input Data	178
3.1	Model	119	4.6.3	Results	180
3.1.1	Settings	119	4.7	RF-IMP	181
3.1.2	Base Plate	120	4.8	Second-Order Analysis	183
3.1.3	Column	125	4.9	RF-STEEL Surfaces	186
3.1.3.1	Manual Definition of Surfaces	125	4.9.1	Input Data	186
3.1.3.2	Generation of Surfaces from Member	129	4.9.2	Results	187
3.1.4	Rigid End Plate	134	5.	Parameterized Input	188
3.1.5	Supports	135	5.1	Model of Steel Sheets	188
3.2	Load Data	137	5.1.1	List of Parameters	189
3.2.1	LC 1: Self-Weight	137	5.1.2	Formula Editor	190
3.2.2	LC 2: Imposed Load	139	5.1.3	Parameterized Model	193
3.3	Superimposing the Loads	140	5.2	Parameterized Loads	194

Contents

	Contents	Page		Contents	Page
5.2.1	LC1: Self-Weight	194	7.	Special Topics	209
5.2.2	LC2/LC3: Variable Loads	196	7.1	Intersections	209
6.	Solids	197	7.1.1	Pipes	209
6.1	Model of Steel Hook	197	7.1.2	Intersections	211
6.1.1	Units and Grid Settings	197	7.1.3	Supports	213
6.1.2	Lines	198	7.1.4	LC1: Self-Weight	213
6.1.3	Surfaces	199	7.1.5	FE-Mesh Settings	213
6.1.4	Solids	201	7.1.6	Results	214
6.1.5	Supports	203	7.1.7	FE Mesh Refinement	215
6.2	Loads	204	7.2	Singularities	218
6.2.1	LC1: Self-Weight, Uniform Temperature	204	7.2.1	Theory	218
6.2.2	LC2: Temperature Difference	205	7.2.2	Example	219
6.3	FE Mesh	206			
6.4	Calculation and Results Evaluation	207			

1. About RFEM

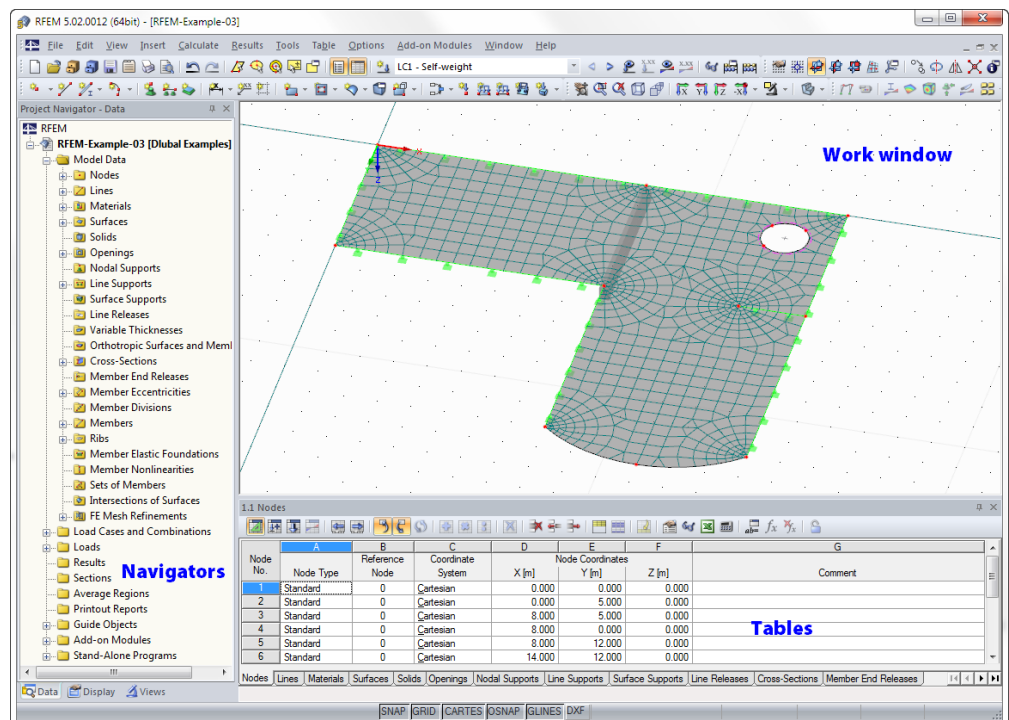
1.1 Finite Elements

There are three types of elements in RFEM. **1D member elements** with various nonlinearities like failure under tension or compression, yielding etc. can be used to model frameworks, trusses or cable structures. Plane **2D surface elements** are ideally elastic elements. Their thickness must be "small" so that they do not transfer any perpendicular stresses. By surface elements, plates, walls and shells can be modeled. **3D solid elements** are also ideally elastic elements. They should be used in a well-considered manner. Large parts of many models can often be represented by member and surface elements, with solid elements applied only at critical locations of the model. Contact problems are also represented by solid elements.

1.2 Graphical User Interface

There are three main modes in RFEM that can be used to define and control data:

- In the **work window** the model, graphics and dialog boxes are displayed. You can use the mouse for most tasks, for example to open a model, select and modify the structural elements or to display the results.
- In the **tables**, you will mainly use the keyboard for the numerical input.
- There are four **navigators** in the left part of the work window: *Data*, *Display*, *Views*, *Results*. The entries of the current model are listed hierarchically in those navigators.



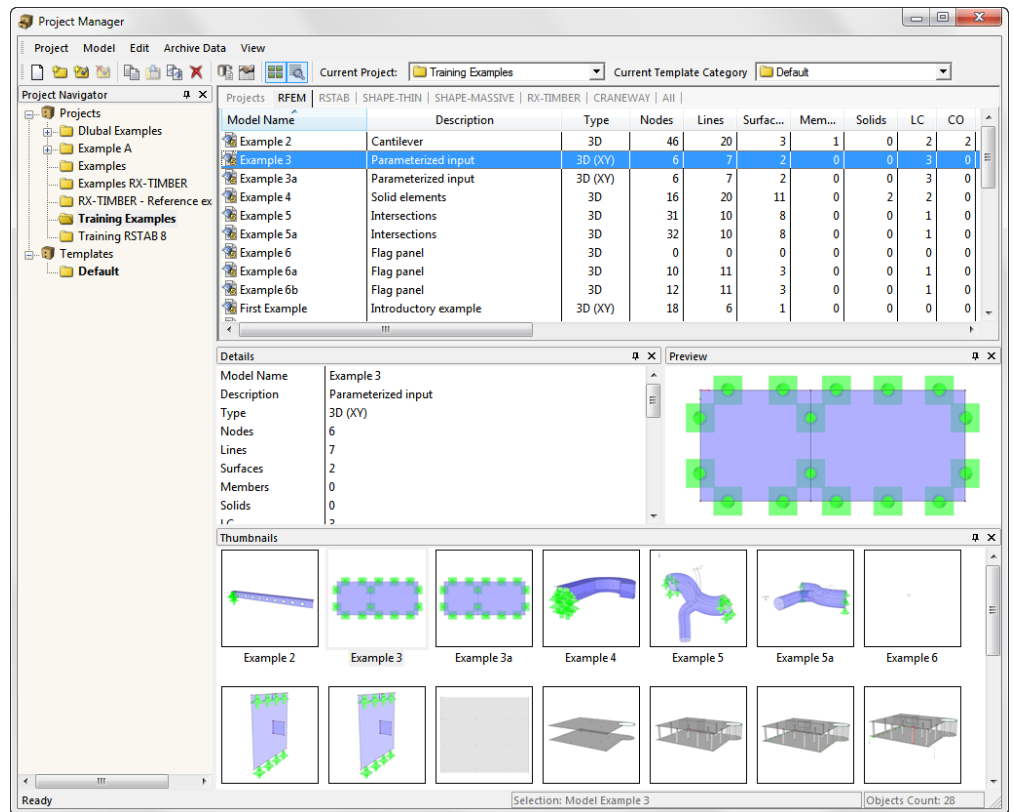
RFEM work window with navigator (left) and input tables (bottom)

There are many ways to use the mouse in the windows and the navigators, for example, to start a function or to edit elements.

1.3 Project Management

1.3.1 New Project

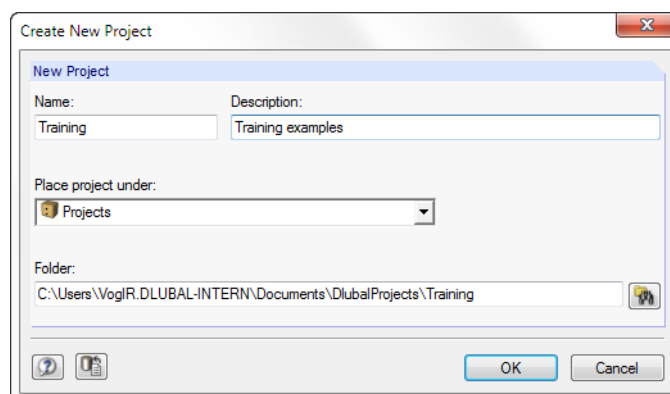
We open the Dlubal Project Manager by clicking the **[Project Manager]** button in the toolbar.



Project Manager



We click the **[New Project]** button in the toolbar of the Project Manager. The *Create New Project* dialog box opens. We enter the project name and description as follows.



Dialog box *Create New Project*



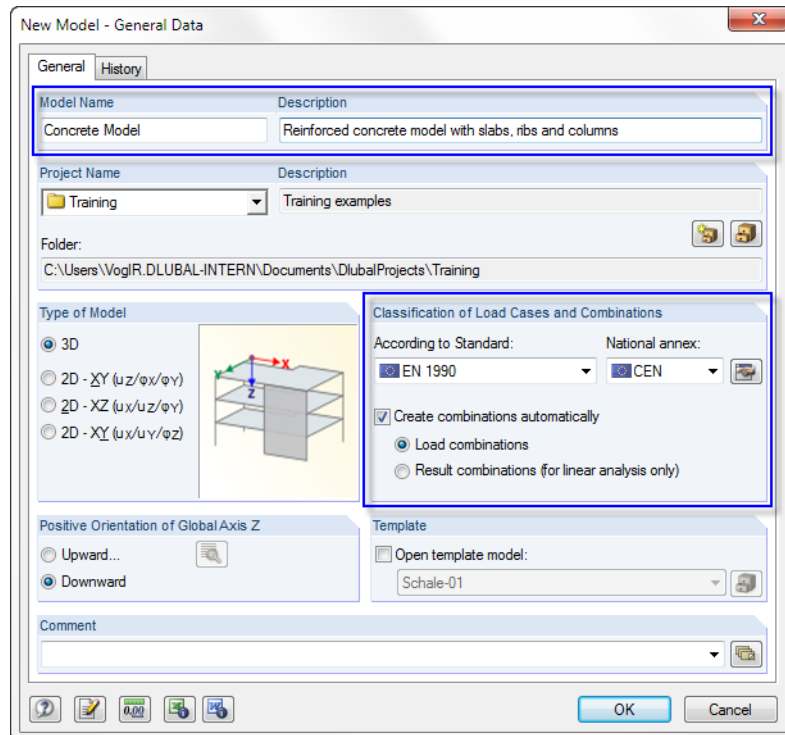
We select the folder for the project manually or by using the **[Browse]** button.

The new project folder *Training* is transferred to the Project Manager and set as default.

1.3.2 New Model

General Data

We close the Project Manager. Then we click [**New Model**] to open the dialog box **New Model - General Data** where we enter the *Name* and *Description* of the model.

Entering *Model Name* and *Description*

In the dialog section *Classification of Load Cases and Combinations*, we select the Standard **EN 1990** and the National annex **CEN**.

We activate the check box **Create combinations automatically** and make sure that the option **Load combinations** is set.

Units and Decimal Places

To check on the units and decimal places, we click the button [**Units and Decimal Places**].

A dialog box allows us to adjust the units and decimal places of the model, loading and results. The settings defined in the dialog box **Units and Decimal Places** can be changed any time in the design process.



2. Concrete Model

2.1 Sketch of System

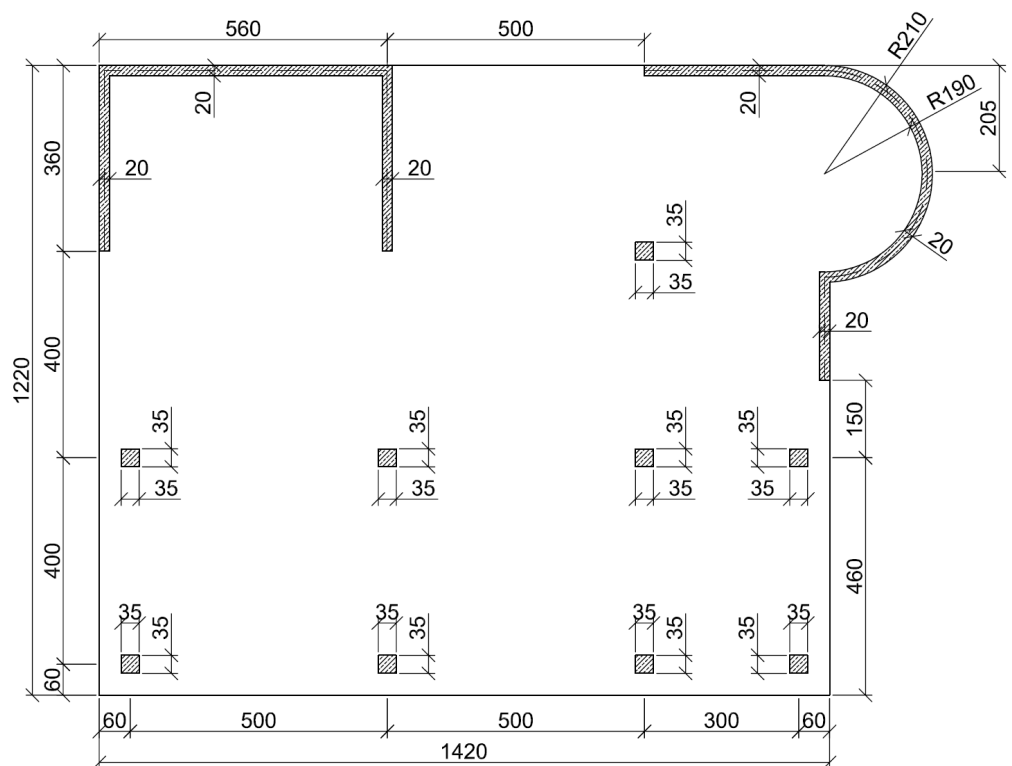
We will design one story of a reinforced concrete structure. This structure consists of a 20 cm thick floor slab and 30 cm thick ceiling slab. The ceiling slab is supported by nine columns with the dimensions 35 cm x 35 cm and two bracing wall systems.

We use concrete C30/37 and steel S 235 for all structural components.

The ceiling slab contains an opening for a spiral staircase with a diameter of 3.60 m.

The clear height between the two slabs is 3.00 m.

The dimensions of the floor slab are indicated in the following figure.

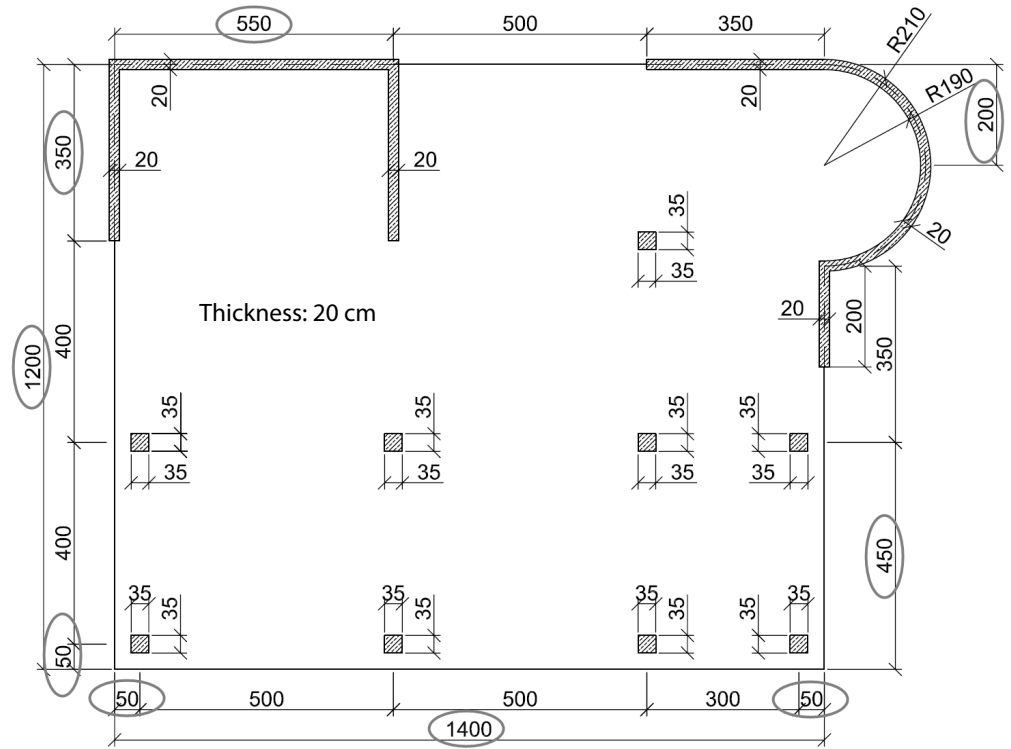


Sketch of system

The coordinates in RFEM refer to the axes of the walls and slabs. Therefore, we have to move the edges of the floor slab by 10 cm inwards so that they are placed on the center axes of the walls (which are 20 cm thick).

We use the following geometry for our model. The different values are circled.

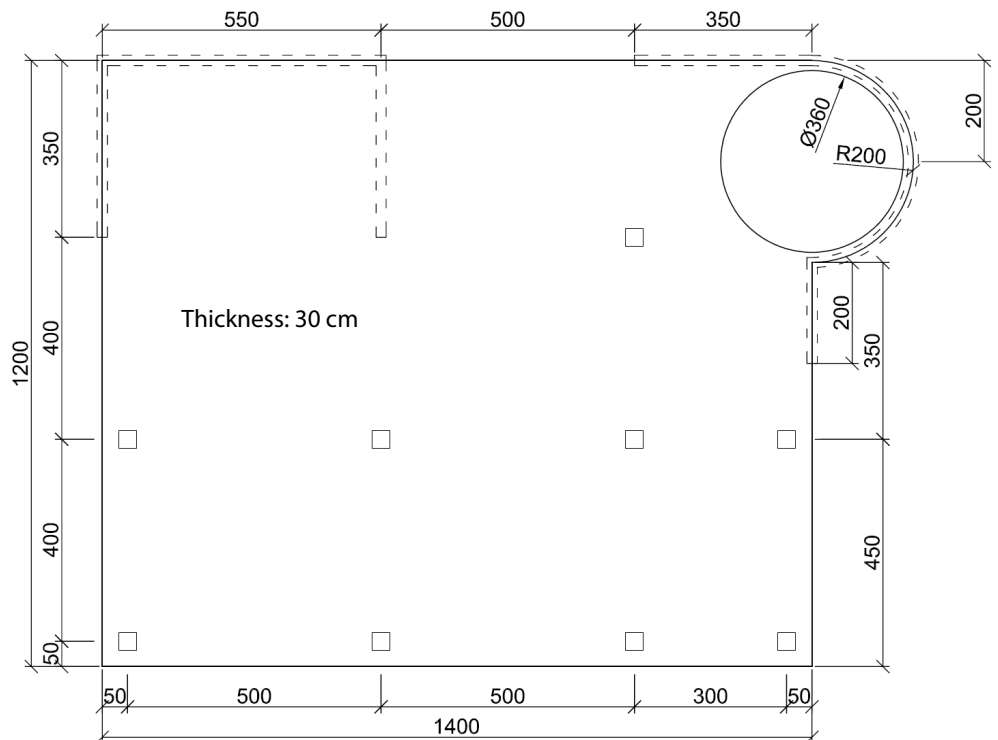
Floor slab



Floor slab

Ceiling slab

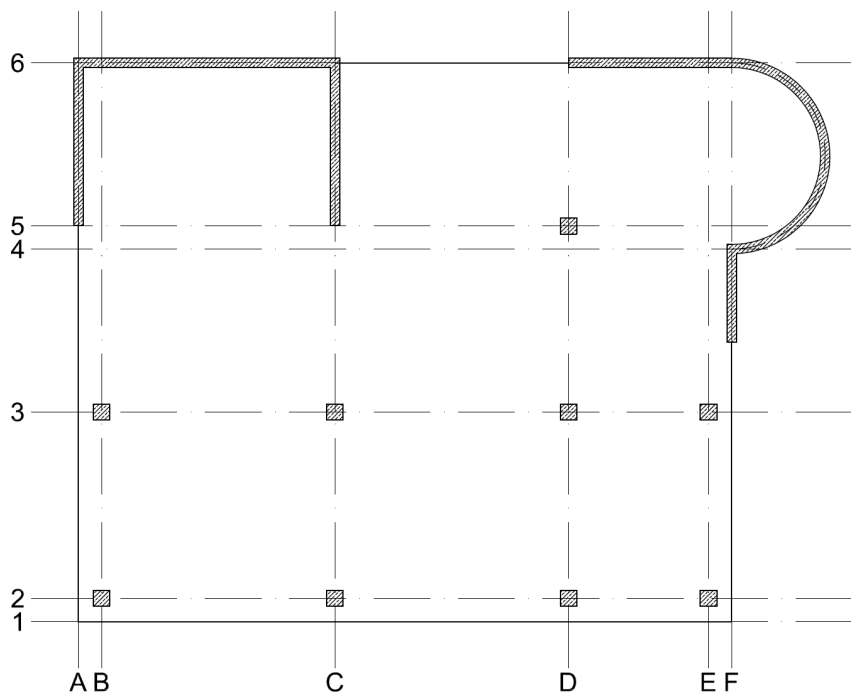
The geometry of the ceiling slab with the circular opening of the spiral staircase is as follows.



Ceiling slab

Axis system

We apply an axis system which is useful for sections or components such as columns.



Axis system

Supports

The floor slab represents a flat foundation with a foundation modulus of 100 MN/m³.

In axis A, the slab rests on the foundation of an adjacent building. Therefore, this line of the slab is fixed in the Z-direction.

Loads

Load case 1: Self-weight

Self-weight of slabs, walls and columns

Load case 2: Imposed loads

Floor slab: 5.0 kN/m²

Ceiling slab: 2.0 kN/m²

Load case 3: Wind in +X

Compression (axis A): $w_1 = 0.8 \cdot 0.5 \text{ kN/m}^2 = 0.40 \text{ kN/m}^2$

Suction (axis F): $w_2 = 0.5 \cdot 0.5 \text{ kN/m}^2 = 0.25 \text{ kN/m}^2$

Load case 4: Wind in +Y

Compression (axis 6): $w_1 = 0.8 \cdot 0.5 \text{ kN/m}^2 = 0.40 \text{ kN/m}^2$

Suction (axis 1): $w_2 = 0.5 \cdot 0.5 \text{ kN/m}^2 = 0.25 \text{ kN/m}^2$

Load case 5: Imperfections towards +X

According to EN 1992-1-1, clause 5.2 (5), imperfections are applied to all columns.

Inclination: $1/\varphi_0 = 200$

Precamber: $L/w_0 = 0$

Load case 6: Imperfections towards +Y

Inclination: $1/\varphi_0 = 200$

Precamber: $L/w_0 = 0$

2.2 Model Data

2.2.1 General Settings

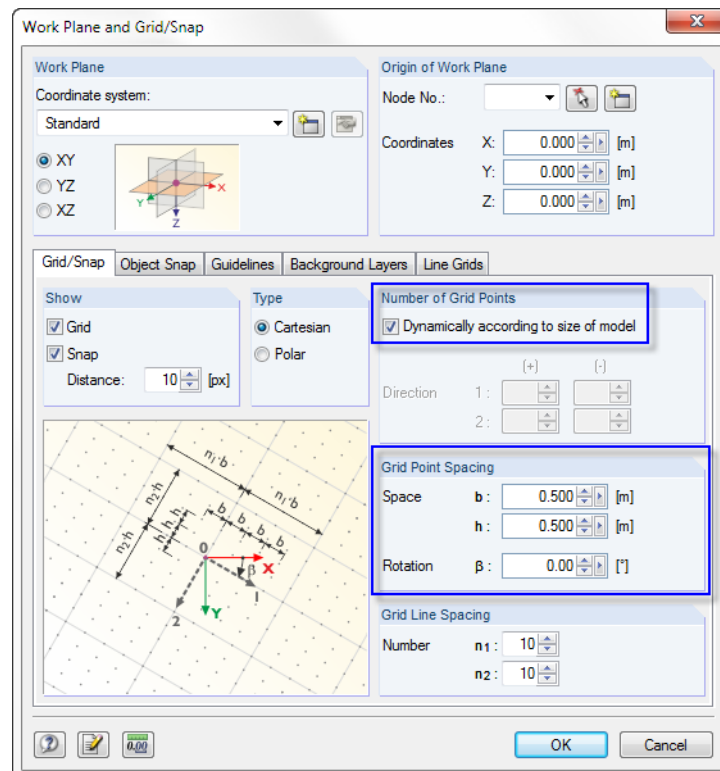
Set View



We click the [View in Z direction] button.

Set Work Plane / Customize Grid

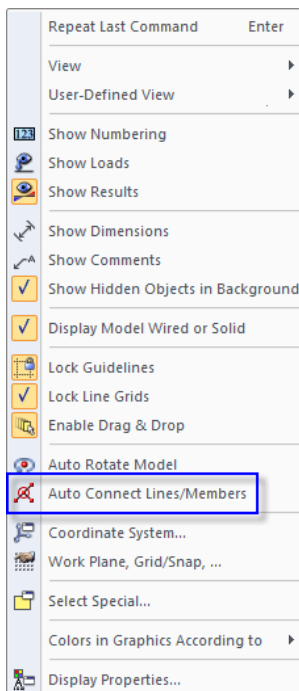
The default setting of the grid has a spacing of 1.00 m. We change those values to **0.50 m** in the dialog box **Work Plane and Grid/Snap** and activate the option **Dynamically according to size of model**.



Dialog box *Work Plane and Grid/Snap*

Check Default

We check in the general context menu whether the option **Auto Connect Lines/Members** is enabled. If so, we switch this function off.

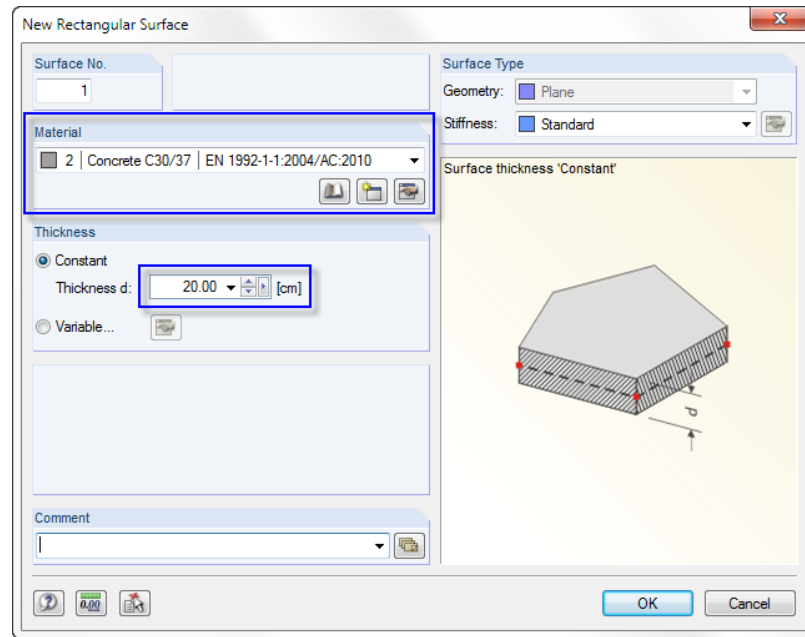


2.2.2 Floor Slab

2.2.2.1 Rectangular Surface

To create the first surface component of the floor slab, we click [New Rectangular Surface].

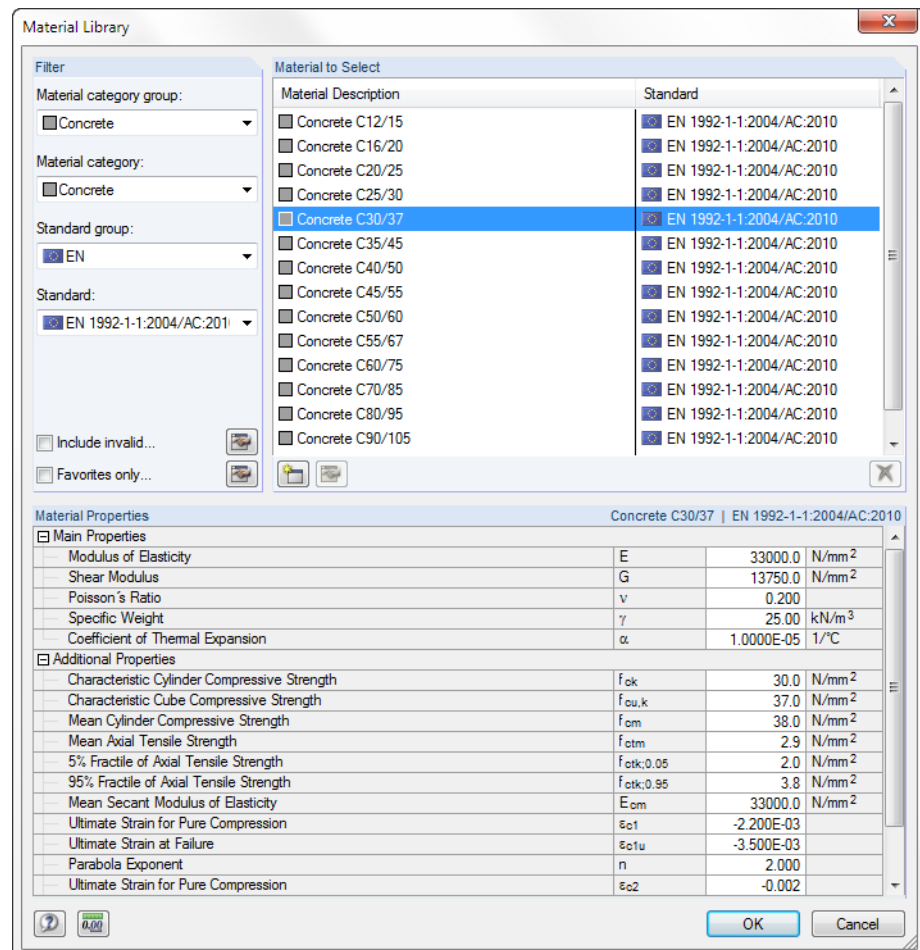
In the dialog box **New Rectangular Surface**, we specify the surface number, material, type and thickness of the surface.



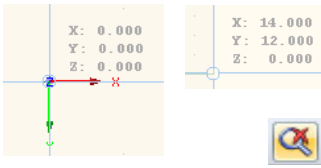
Dialog box *New Rectangular Surface*



We select the material **Concrete C30/37**. If necessary, we can click **[Library]** to change the material.

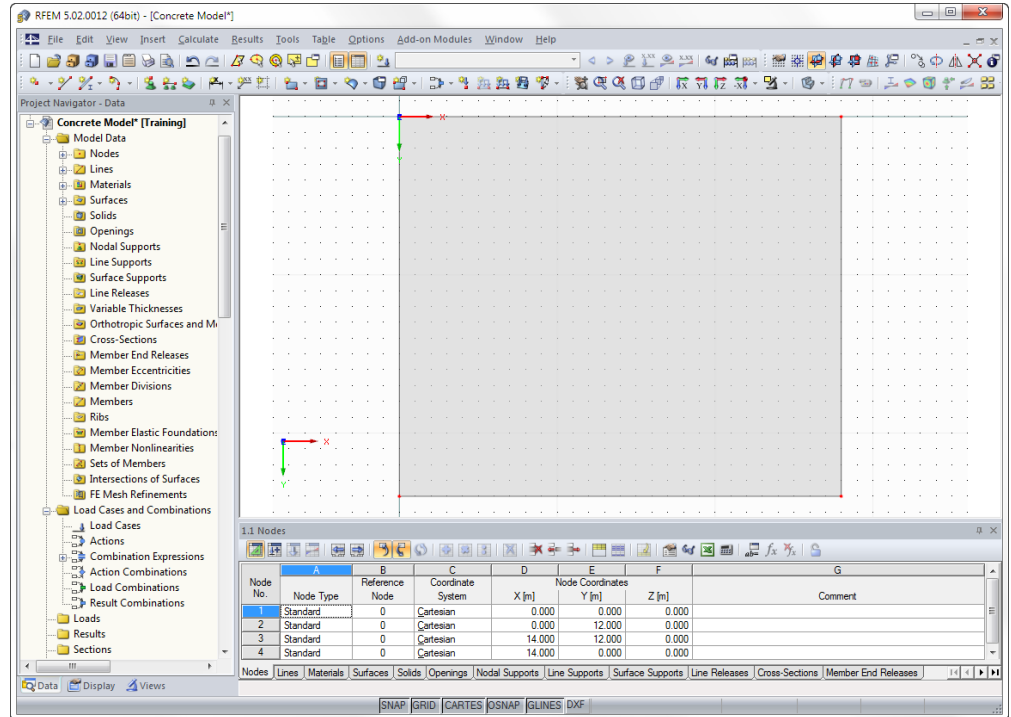


Material Library



The sketch on page 10 shows the dimensions 14.00 m x 12.00 m for the rectangular part of the floor slab. We set the first corner in the origin of the coordinate system **0.000/0.000/0.000** and then place the second corner at the grid point with the coordinates **X=14.00, Y=12.00**.

To show the entire slab scaled to the size of the work window, we click **[Show Whole Model]**.

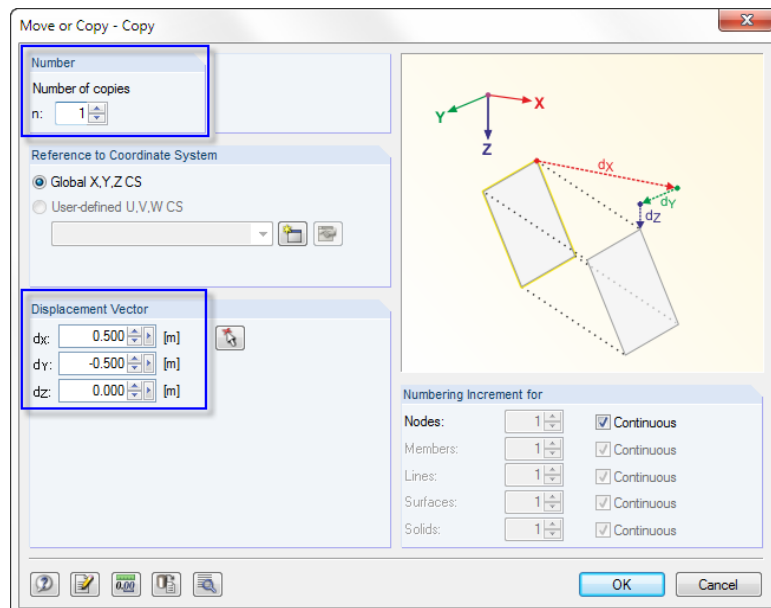
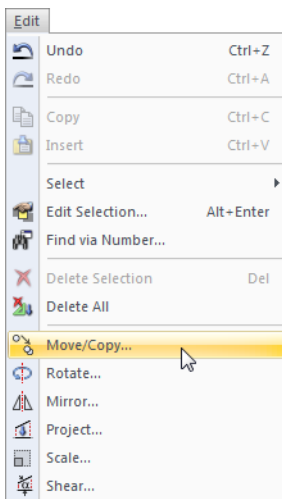


Rectangular part of floor slab

2.2.2.2 Rows of Columns

Copy node

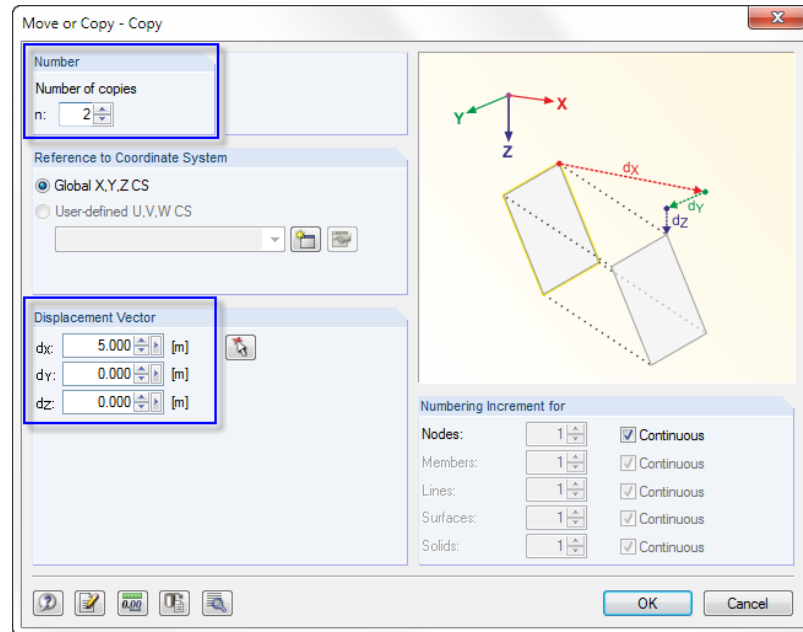
The distance of column B/2 from the corner is 0.50 m in X-direction and -0.50 m in Y-direction. We select node A/1 and use the **Move/Copy** function to copy it. We set the *Number of copies* to **1**, the *Displacement Vectors* d_x to **0.50** m and d_y to **-0.50** m.



Dialog box *Move or Copy*

Copy two nodes

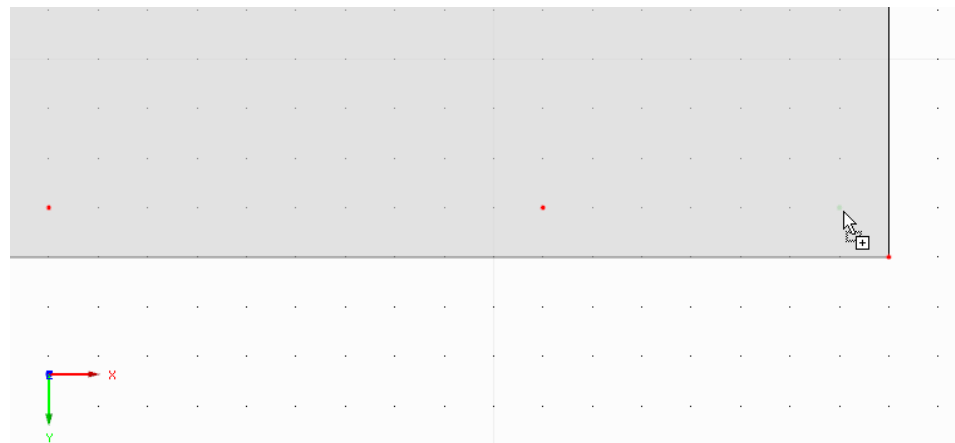
The spacing of the next two columns is 5.00 m. Since the new node is already selected, we just have to repeat the previous action. In the **Move or Copy** dialog box, we define **2** copies and set **dx** to **5.00 m**, **dy** to **0.00 m**.



Dialog box *Move or Copy*

Copy node graphically

Finally, we copy the node in the right bottom corner graphically. We select this node, press the [Ctrl] key and move the node to the adjacent grid point (distance 0.5 m).

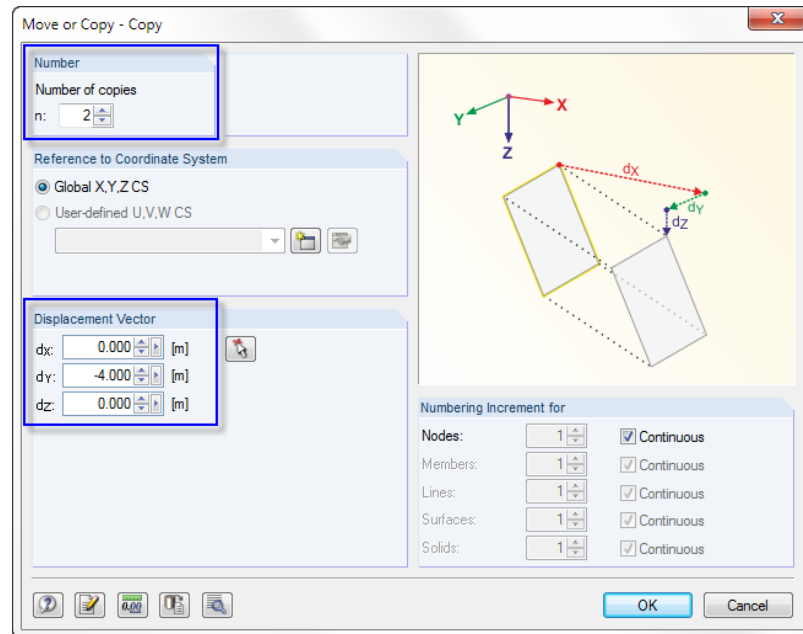


Copying node graphically

Copy column row

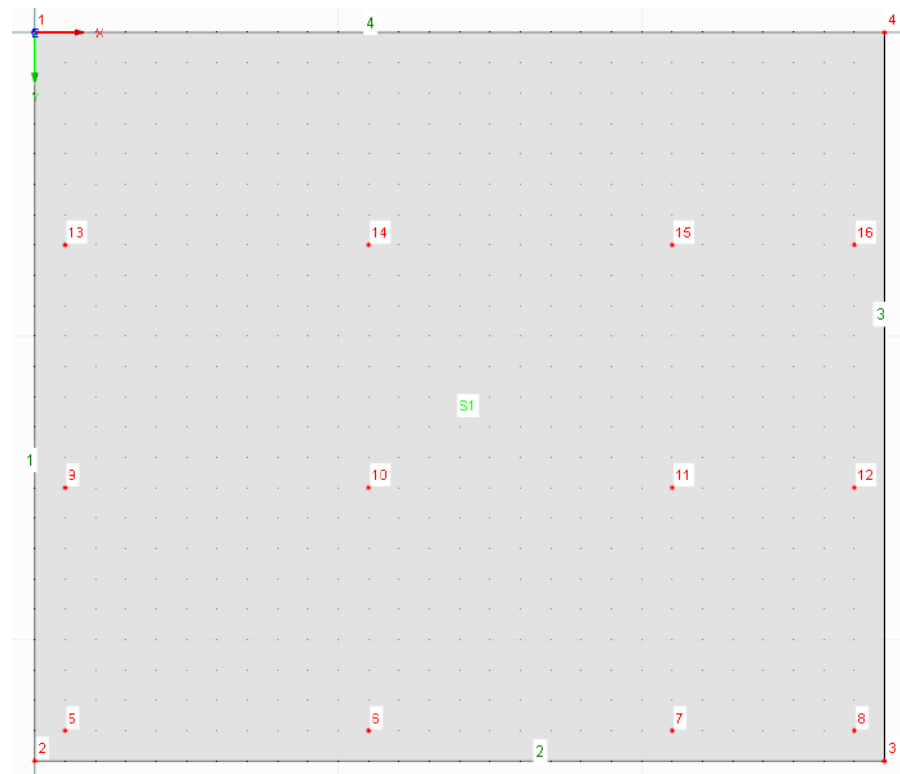
To copy all four column nodes, we have to select them. Either we click all points one after the other while holding the **[Shift]** key, or we draw a window across those four column points.

In the **Move or Copy** dialog box, we set the number copies to **2** and enter the *Displacement Vectors* $d_x = 0.00$ m, $d_y = -4.00$ m.



Dialog box *Move or Copy*

After that, the arrangement of column nodes is as follows.



Column nodes on floor slab

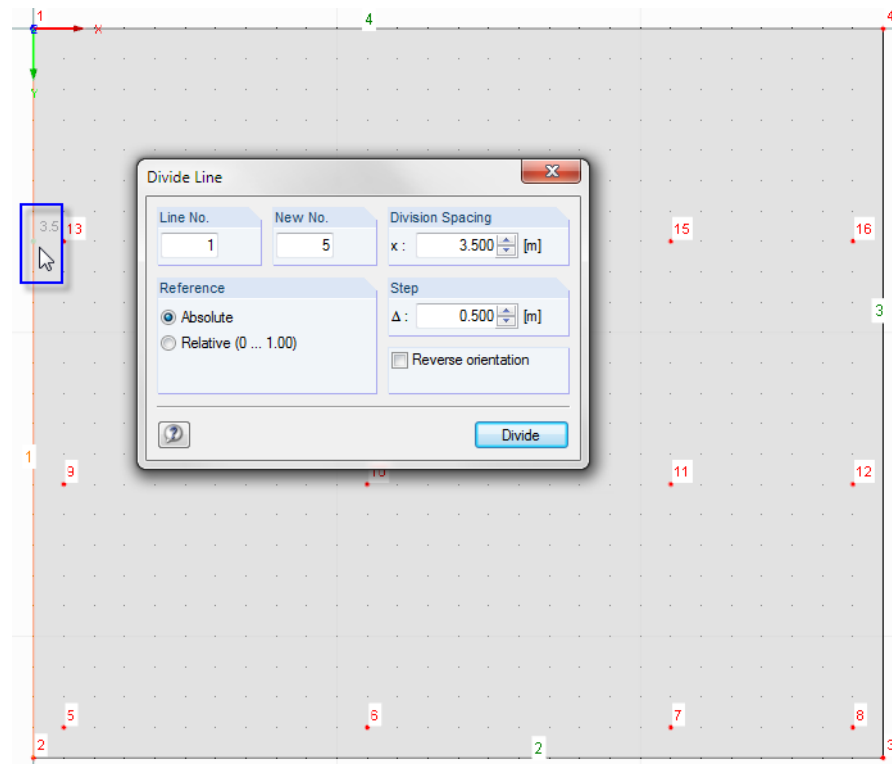
2.2.2.3 Walls

Divide line

To create the walls in axes A and C, we need the nodes of the points A/5 and C/6.

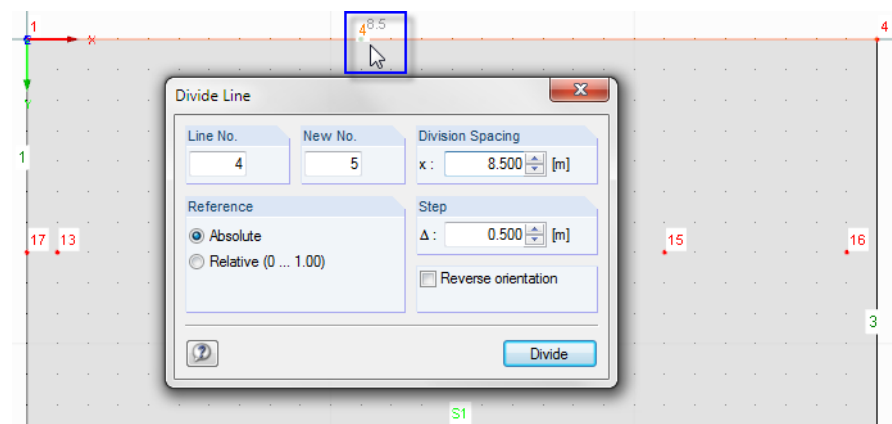
To create point A/5, we select line 1 by right-clicking it. On the context menu, we select **Divide Line** → **Graphically**.

When we move the mouse pointer along the line, the values of distances are shown. We divide the line close to point 13, that is, at the *Division Spacing* of **3.500 m** by clicking the line.



Divide line 1 graphically

We proceed with the division of line 4. At the location C/6, the value of the ordinate is **8.500 m** (this line runs from node 4 to node 1). We click the line at that location.



Divide line 4 graphically

Mouse functions: Move, Zoom, Rotate

Let us try out some of the mouse functions that are useful for working in the graphic window.

To activate the grab mode, we click the **[Move, Zoom, Rotate]** button in the toolbar. By clicking anywhere in the work window, the hand becomes a fist grabbing the according location. By moving the grabbing hand, we also move the graphic.

When we hold the **[Ctrl]** key in the grab mode, a curved arrow is shown in the hand. By holding the **[Ctrl]** key and the left mouse button at the same time, we can rotate the model.

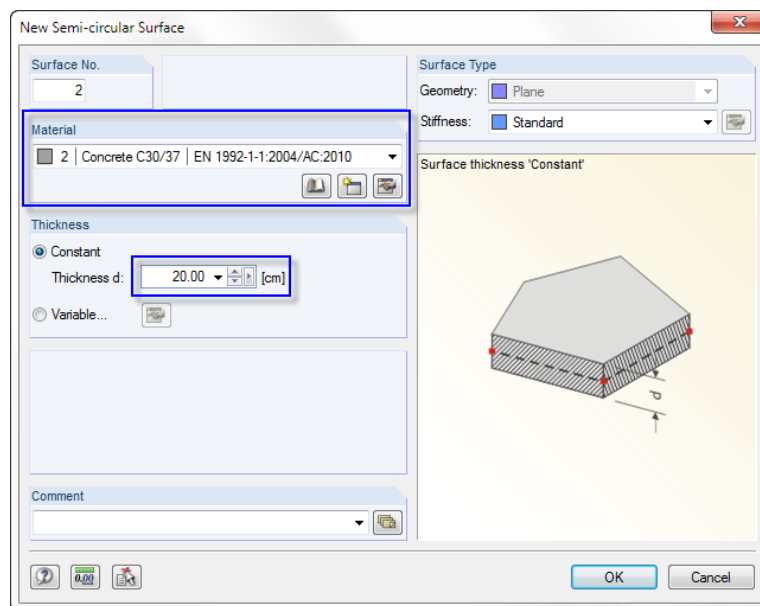
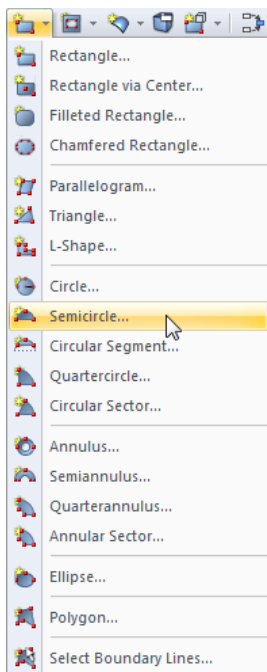
To activate the zoom mode, we press the **[Shift]** key and hold the left mouse button at the same time. A magnifying glass is shown now in the hand. By holding the left mouse button and moving the mouse pointer up, we zoom in and by moving it down we zoom out. The same effect is achieved by rotating the wheel button.

2.2.2.4 Semicircular Surface

We return to the previous view by clicking **[View in Z direction]**.

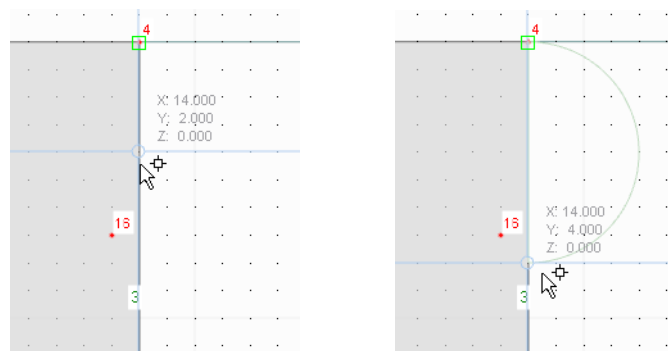
We create the semicircular surface via the list button as seen to left.

In the dialog box **New Semicircular Surface**, we set the material to **C30/37** and the slab thickness to **20 cm**.



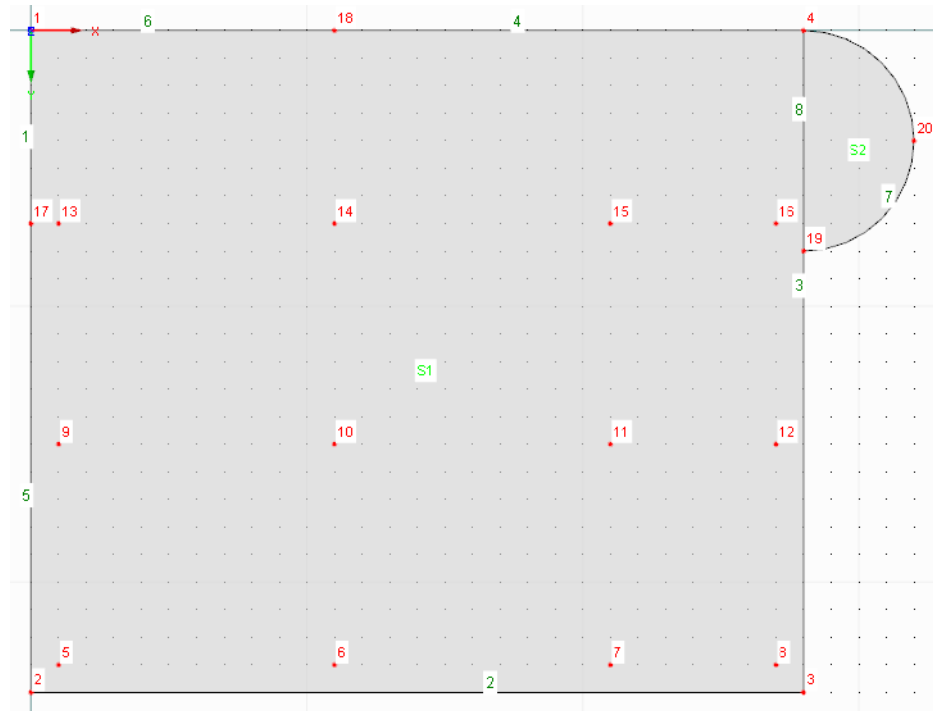
Dialog box *New Semicircular Surface*

We set the center of the semicircle at the grid point **X=14.000, Y=2.000**. Then we define the radius of the semicircle (2.00 m) by clicking the grid point **X=14.000, Y=4.000**.



Defining semicircle by center (left) and end point (right)

After that, surface 2 is created.



Floor slab consisting of two surfaces

2.2.3 Ceiling Slab

The geometry of the ceiling slab is almost identical to the geometry of the floor slab – with the difference that the thickness of the slab is 30 cm and that there is a circular opening.

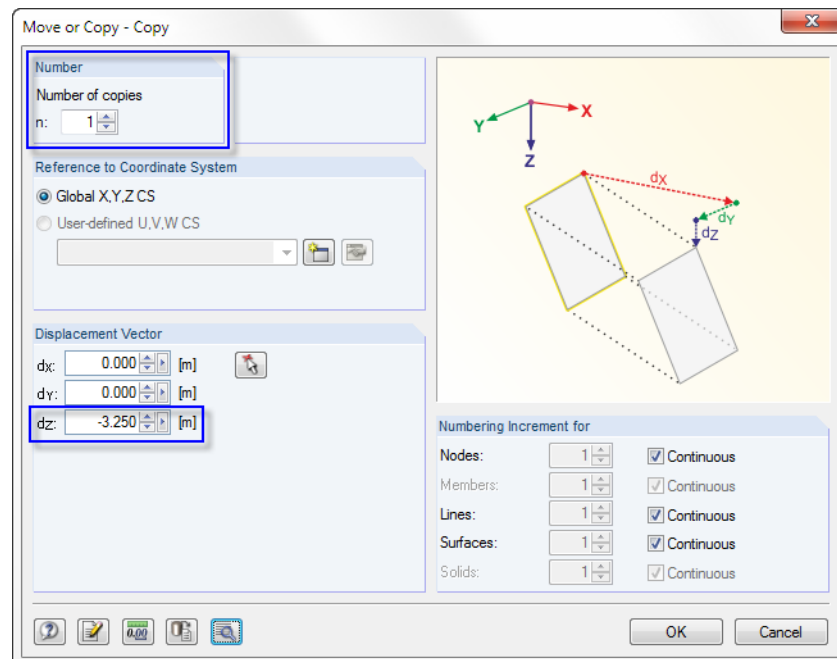
2.2.3.1 Surfaces

Copy surfaces

To obtain the clear height of 3.00 m between the two slabs, the center axes of the surfaces must have a spacing of 3.25 m ($3.00 + \frac{1}{2} \cdot 0.30 + \frac{1}{2} \cdot 0.20$). Thus, we create the ceiling by copying the floor slab by 3.25 m upward.

We select the floor slab by drawing a selection window across the entire model.

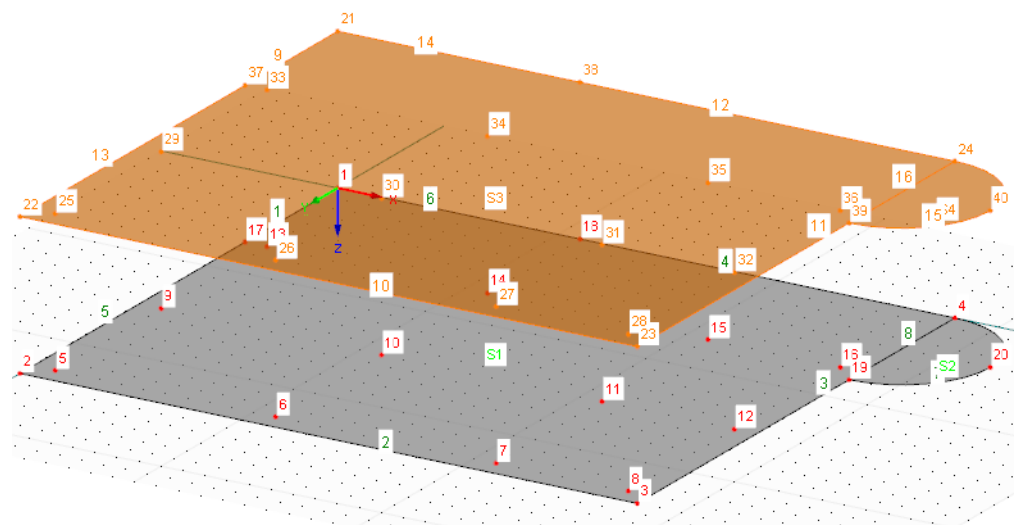
We then use the **[Move/Copy]** button to copy the slab by **3.25 m** in the negative Z-direction.



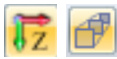
Dialog box *Move or Copy*



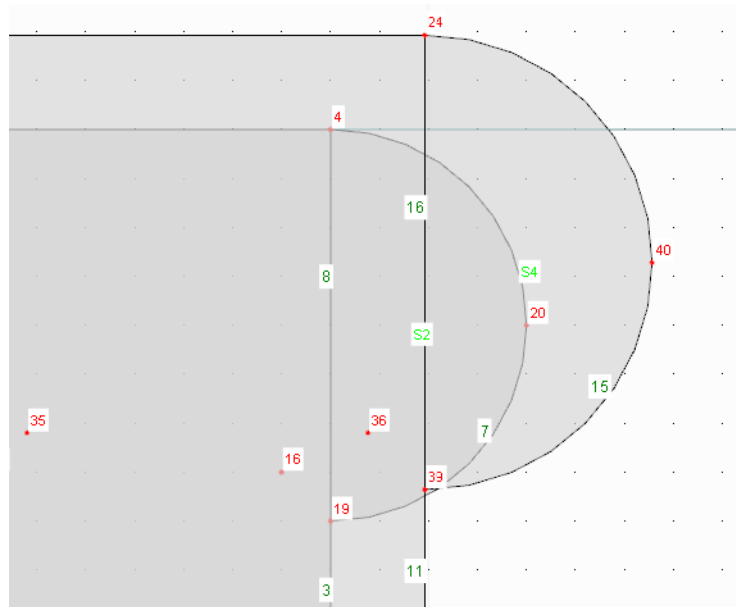
We set the **[Isometric View]** to check on the result.



Isometric view of the copied surfaces



To take a closer look at the semicircle slabs, we set the view by clicking **[View in Z direction]** and **[Perspective View]**. Then we zoom into the model.

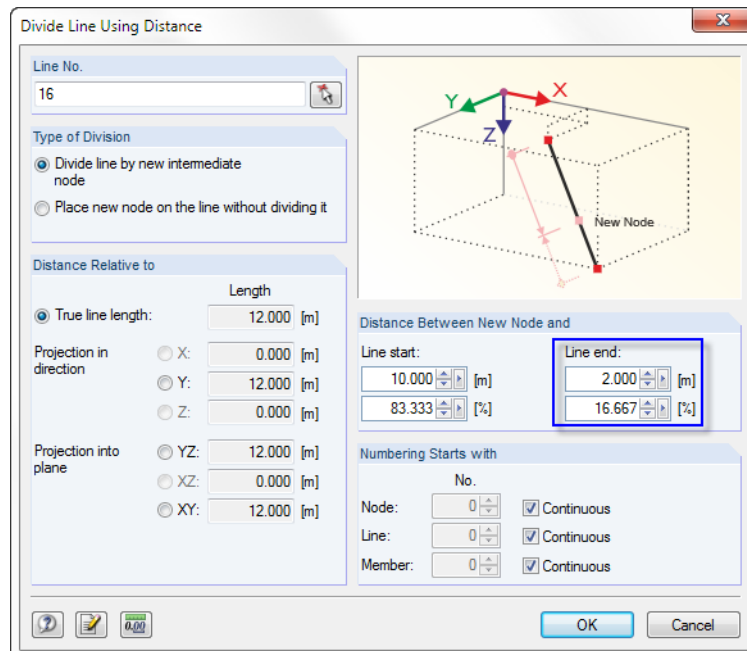
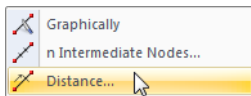


Perspective view of semicircle surfaces

Divide line

We divide the line of the semicircle diameter to obtain the center of the circular opening by right-clicking line 16. On the context menu, we select **Divide Line** → **Distance**.

In the dialog box **Divide Line Using Distance**, we define the distance of the division point at **2.00 m** from the end of the line.

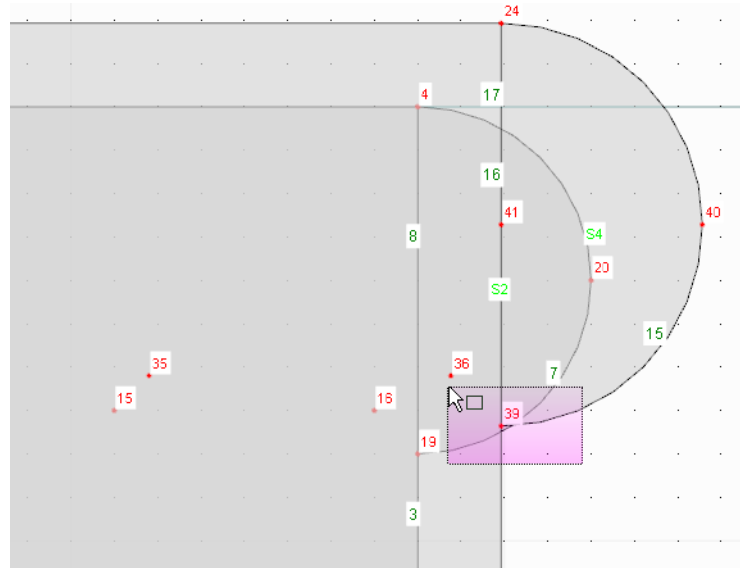


Dialog box *Divide Line Using Distance*

Connect lines

An opening can lie in only one surface. Thus, we have to remodel the boundary lines of the surface a bit before we can define the staircase opening in the ceiling slab.

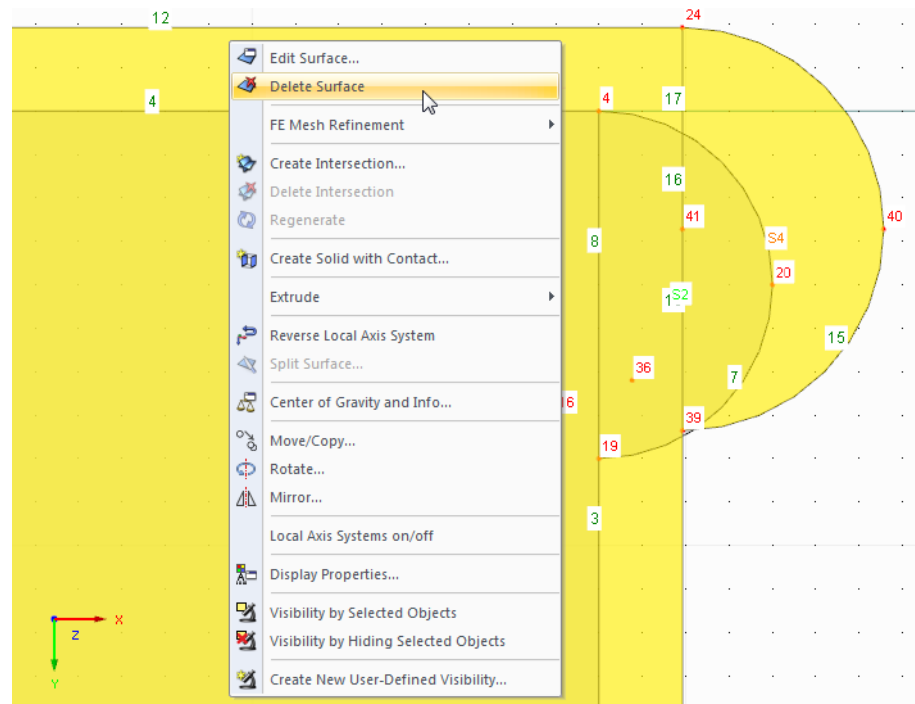
We use the **[Connect Lines or Member]** button to split up line 11. Then we draw a window around the area of node **39**.



Connecting lines

Delete surface

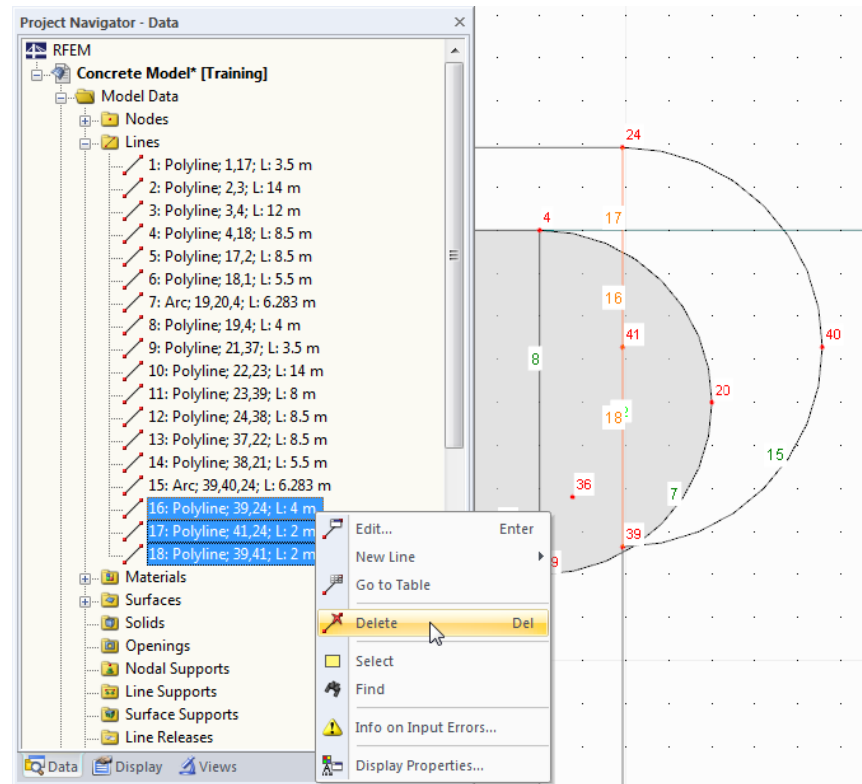
The ceiling slab consists of a rectangle and a semicircle surface. We select surfaces **3** and **4** and delete them via the context menu.



Deleting selected surfaces via context menu

Delete lines

We can remove the lines **16** and **17** and **18** because they are no longer necessary. This time, we use the navigator.

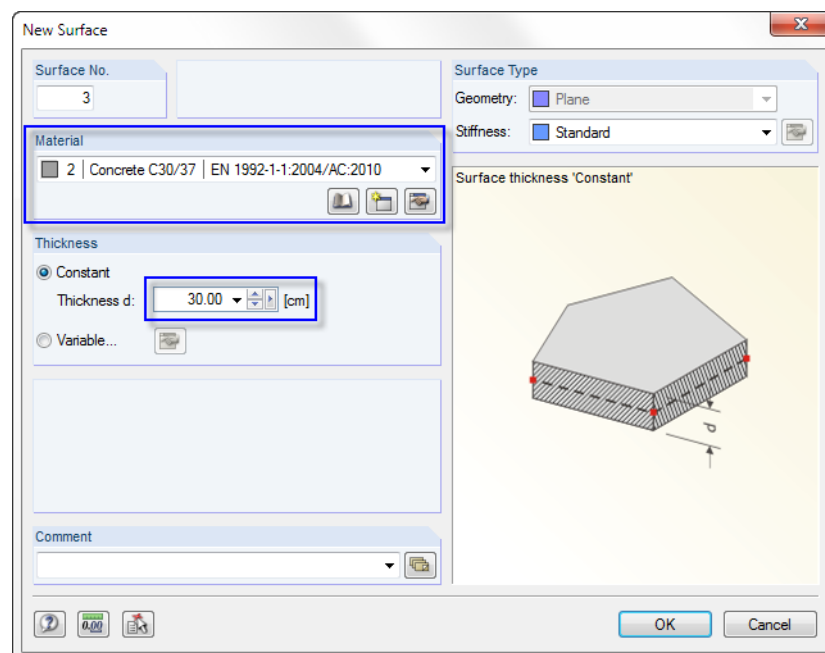


Deleting lines via *Data* navigator

Create surface

To restore the ceiling slab, we click the **[Select Boundary Lines]** list button.

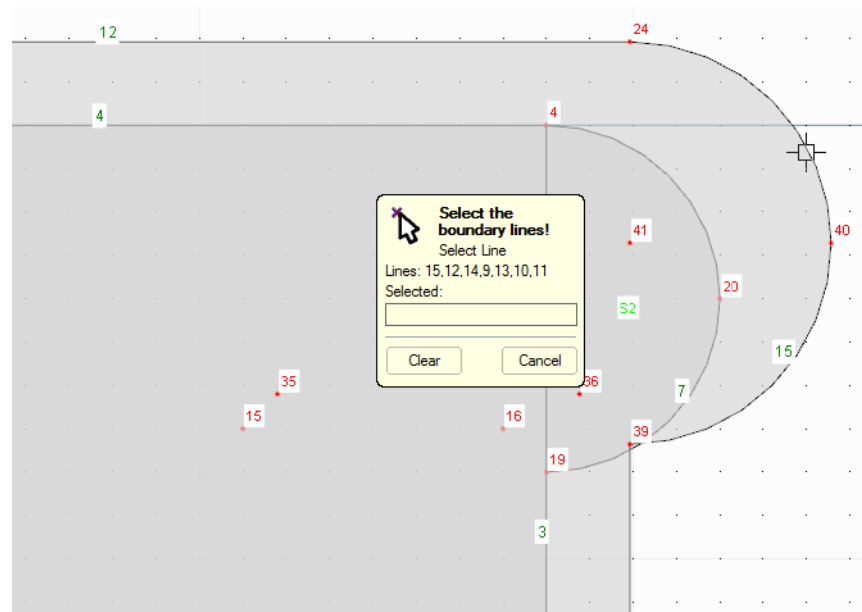
In the dialog box **New Surface**, we set **Concrete C30/37** and define the thickness of **30 cm**.



Dialog box *New Surface*



We now use the reticle to click a line of the upper plane and create the surface.



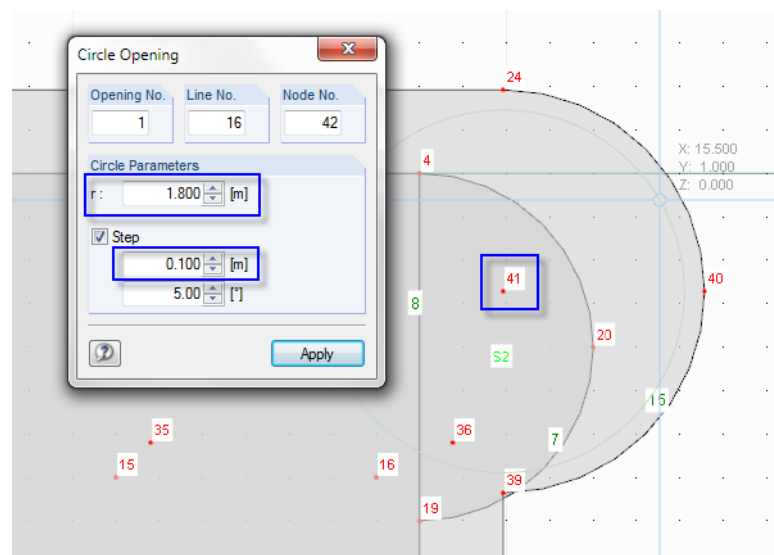
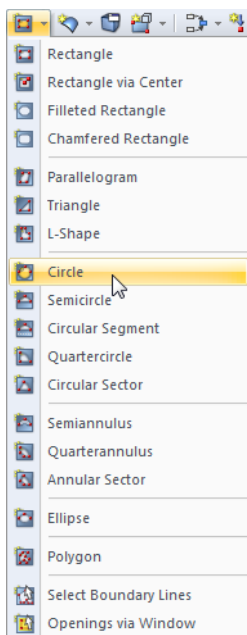
Creating surface by selecting boundary lines

2.2.3.2 Opening

We select the **[Circle]** option from list buttons for openings.

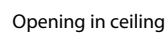
We click node **41** which represents the center of the circular opening.

In the **Circle Opening** dialog box, we activate the **Step** and change the value to **0.10 m**. It is easier then to set the radius r of **1.80 m** in graphics.



Dialog box *Circle Opening*

We click any point with this radius to create the opening.

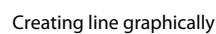


We now define the boundary lines of the walls.

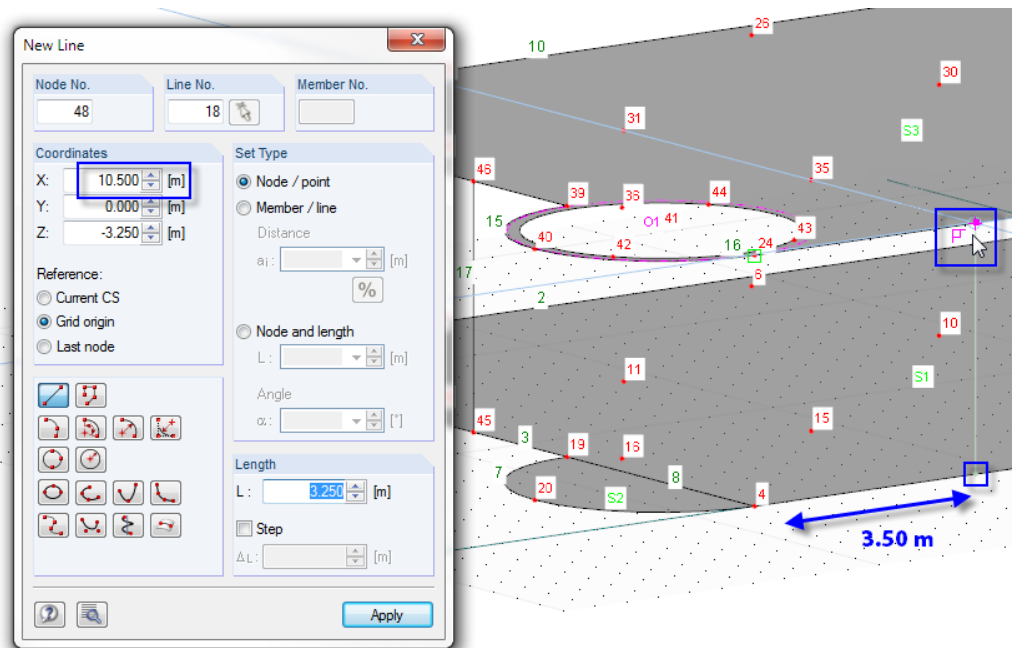
We now define the boundary lines of the walls.

We set the **[Isometric View]** and rotate the model so that the staircase area is in front.

We create a **[New Line]** by clicking the grid point at a distance of **2.00** m from the floor node 19 first. The second node is set perpendicularly on the ceiling line above. Note the symbol!



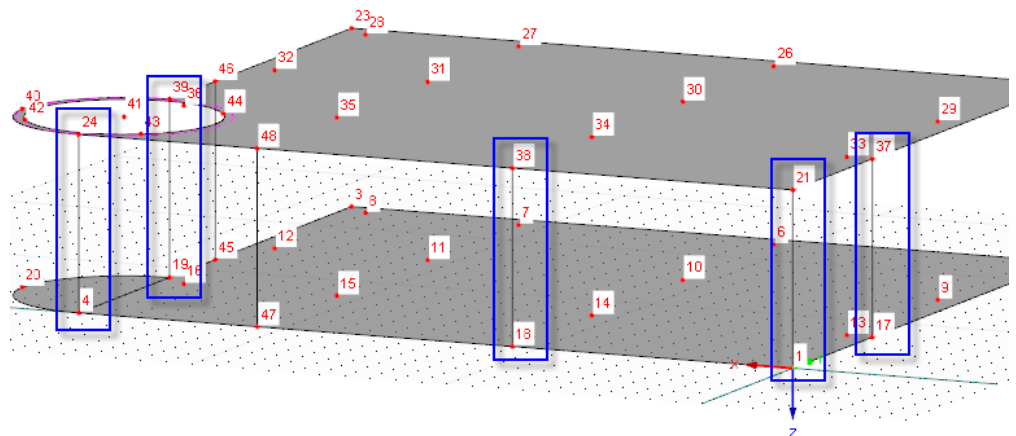
We rotate the model. Then we click the grid point at a distance of **3.50 m** from the floor node 4 (coordinate **X=10.50**). The second node is set perpendicularly on the ceiling line above.



Creating line graphically

Now we connect the pairs of nodes as follows:

- 19 - 39 (F/4)
- 4 - 24 (F/6)
- 18 - 38 (C/6)
- 1 - 21 (A/6)
- 17 - 37 (A/5)



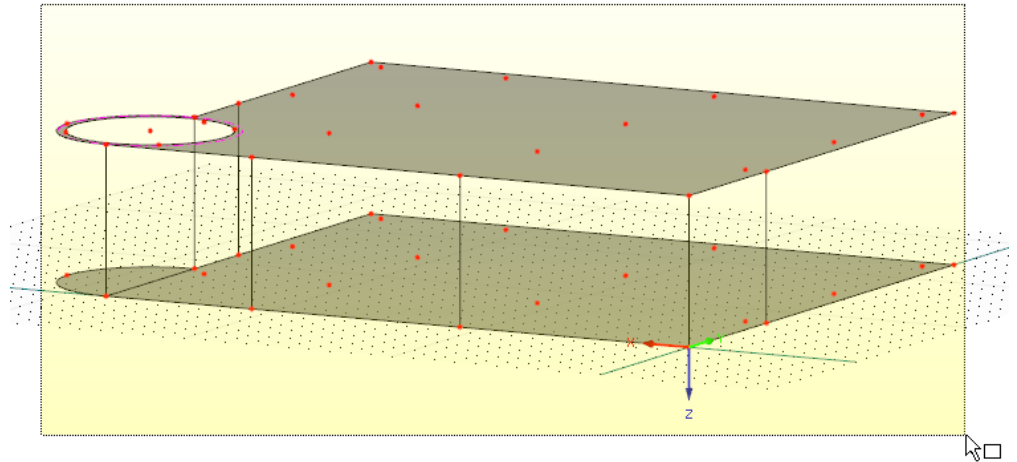
Connecting nodes by lines

Connect lines

Before we can create surfaces from the new lines, the adjacent lines have to be divided so that RFEM can detect all boundary lines. It is irrelevant for the calculation, however, whether there are polylines or single lines.

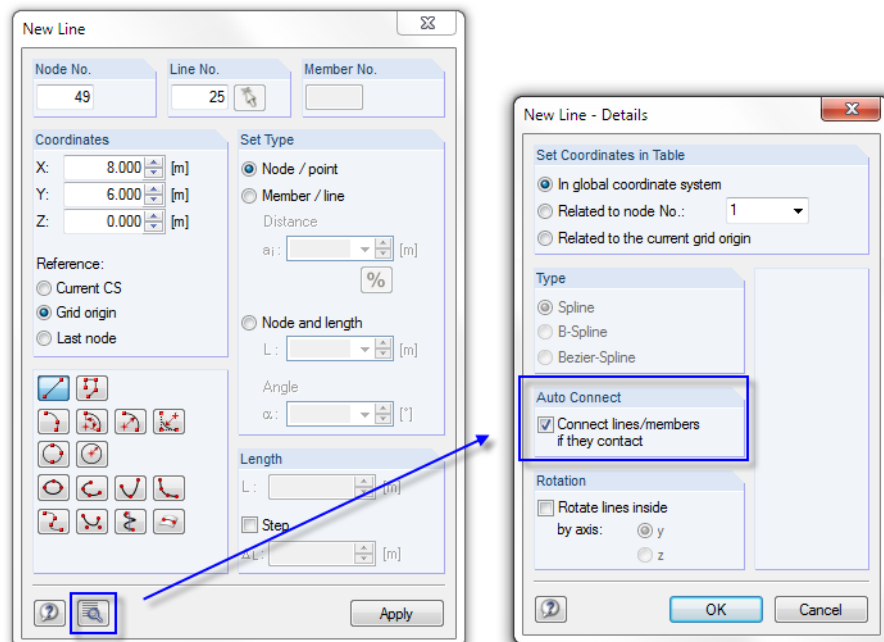


We use the **[Connect Lines or Member]** button again to split up all lines in the model. Then we simply draw a window around the entire model.



Connecting lines

Alternatively, we could have used the *Auto Connect* option when setting the lines.



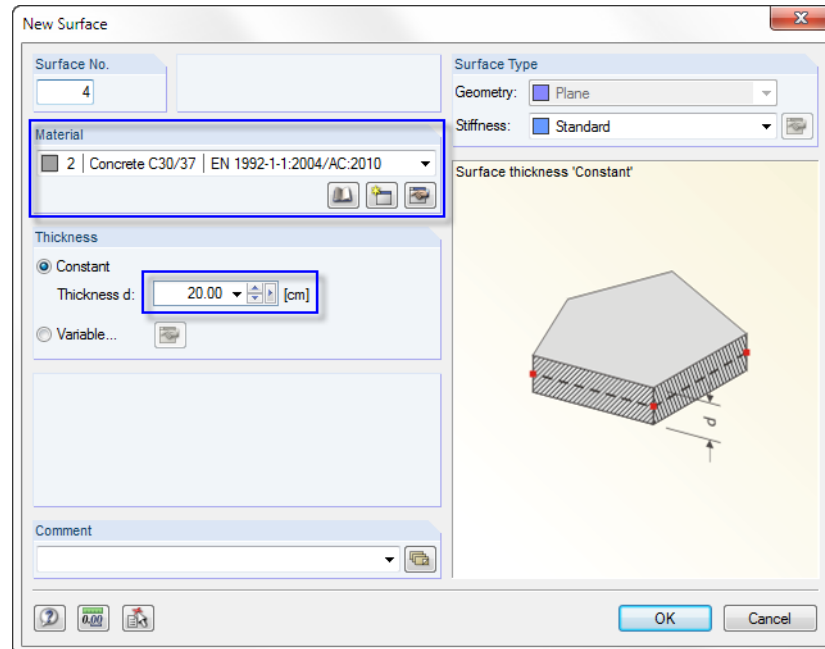
Dialog box *New Line - Details*

2.2.4.2 Surfaces

Walls in corner



Now we create the surfaces between the axes A and C. The walls can be defined by the lines that we just have set. We click the **[Select Boundary Lines]** button. In the dialog box, we select the material **C30/37** and set the surface thickness to **20 cm**.



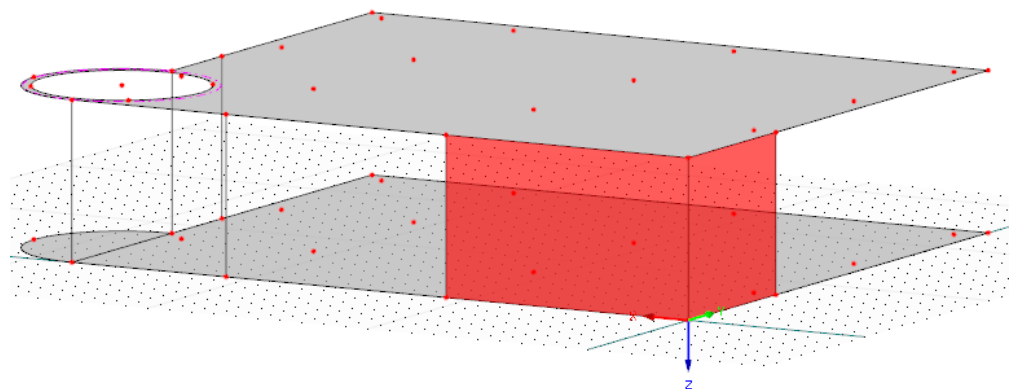
Dialog box *New Surface*



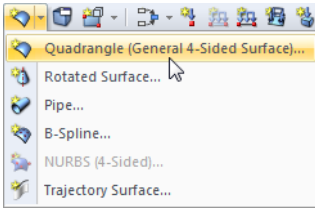
We click the boundary lines of the surface on the A-axis. Please take care to select the lines as a polygonal chain.

The surface function remains active as indicated by the reticle. We go on and create the wall in axis 6 by clicking the lines 21, 14, 22 and 6.

The two new surfaces are shown in the following figure.

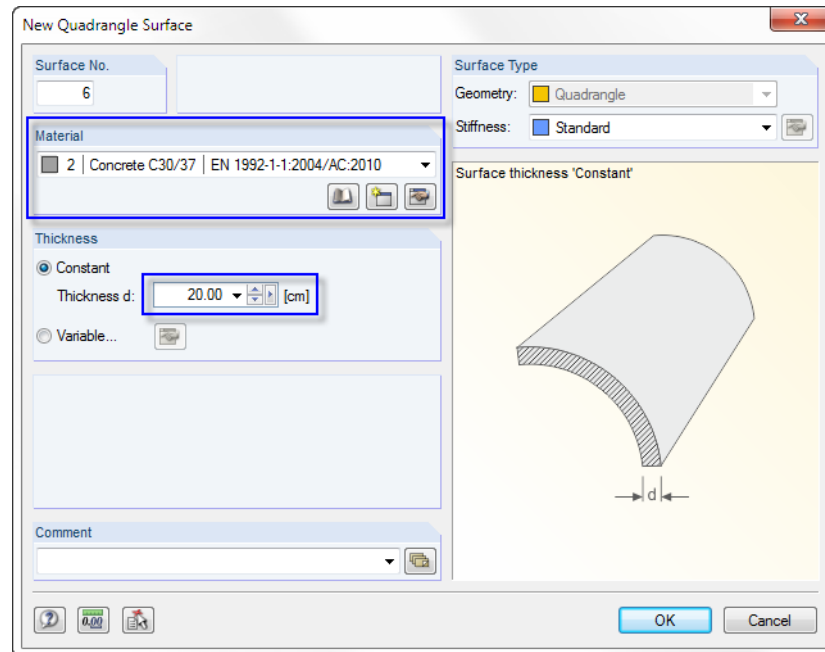


Walls in axes A and 6



Walls at staircase

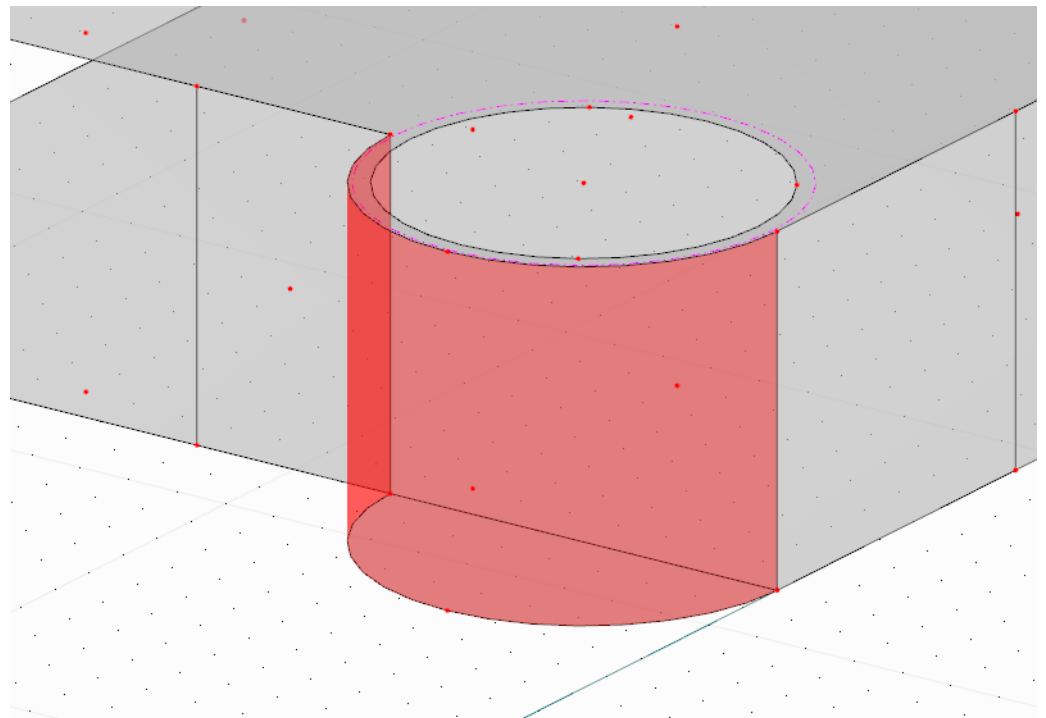
To create a curved surface, we click the **[New Quadrangle]** list button. In the **New Quadrangle Surface** dialog box, we specify the material **C30/37** and the thickness **20 cm**.



Dialog box *New Quadrangle Surface*



With the reticle, we click the boundary lines 20, 15, 19 and 7.

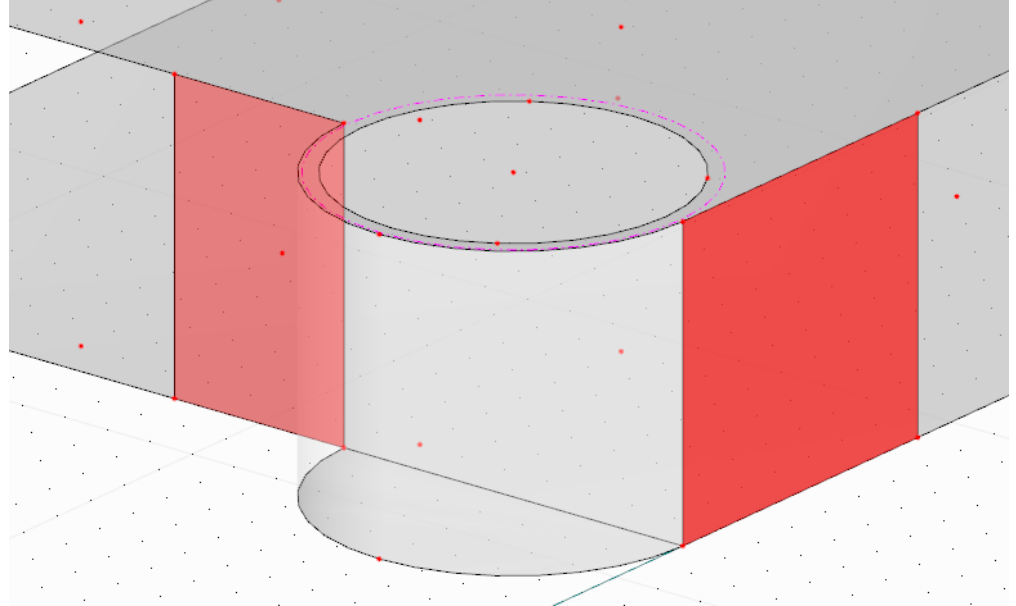


Staircase wall



Now we create the walls next to the curved surface via the **[Select Boundary Surface]** button. We apply the material **C30/37** and the thickness **20 cm** for both surfaces again.

We create the surface by clicking the four boundary lines **17, 19, 27, 24** for the left surface and **12, 18, 20, 4** for the right surface.

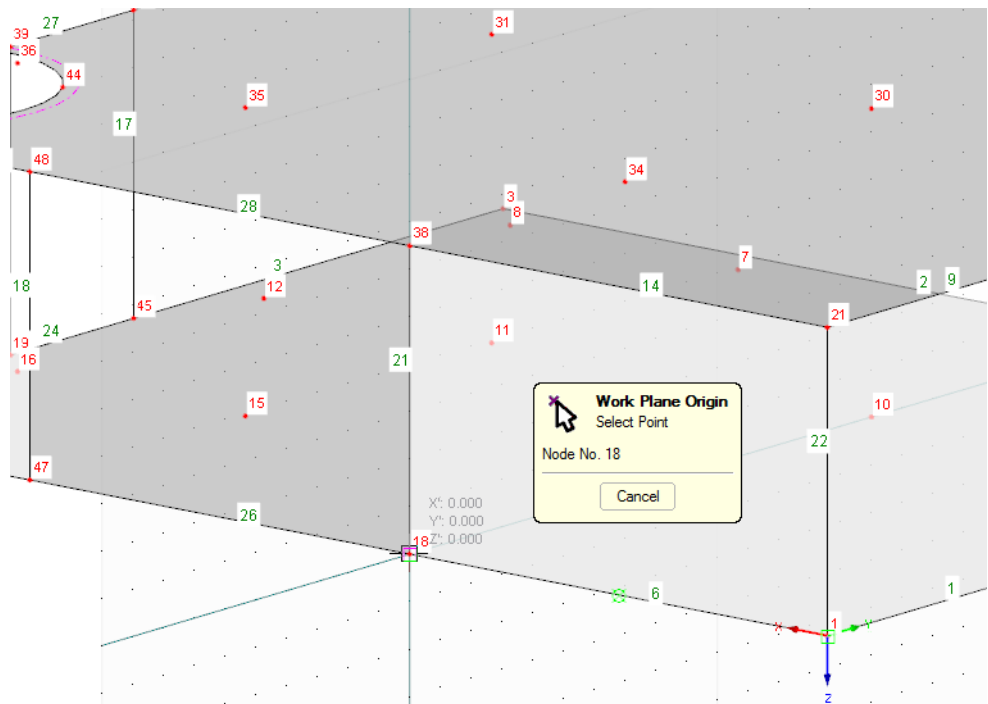


Staircase walls

Inner wall

To create the load-bearing inner wall in the axis C, we set the origin of our work plane into node **18**. For this, we use the button **[Set Origin of Grid/Work Plane]**.

Then we set the **[Work Plane YZ]**.

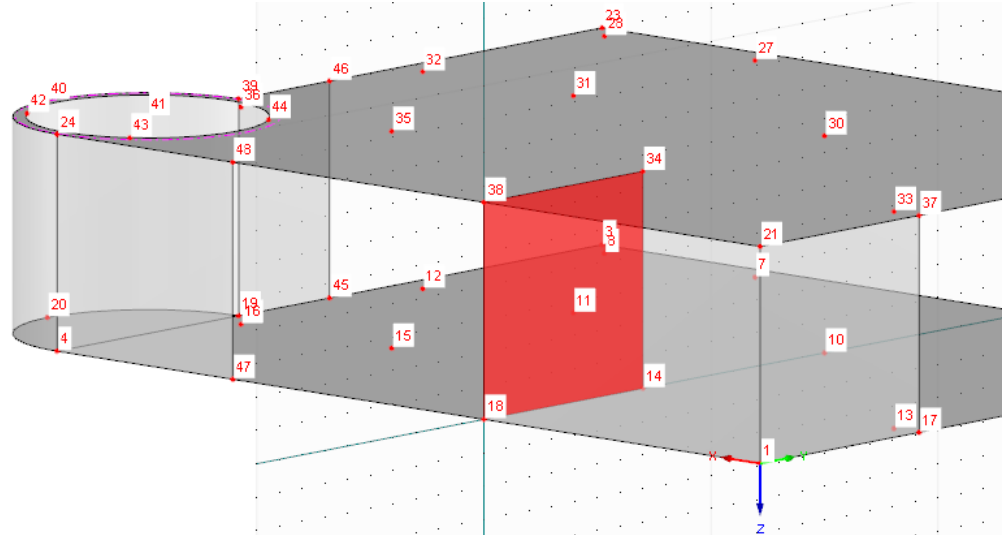


Setting new origin of work plane



We use the button **[New Rectangular Surface]** to define the inner wall. In the dialog box, we specify the material **C30/37** and the thickness **20 cm**.

We define the wall by setting the first corner in node **18** and the second corner in node **34**.



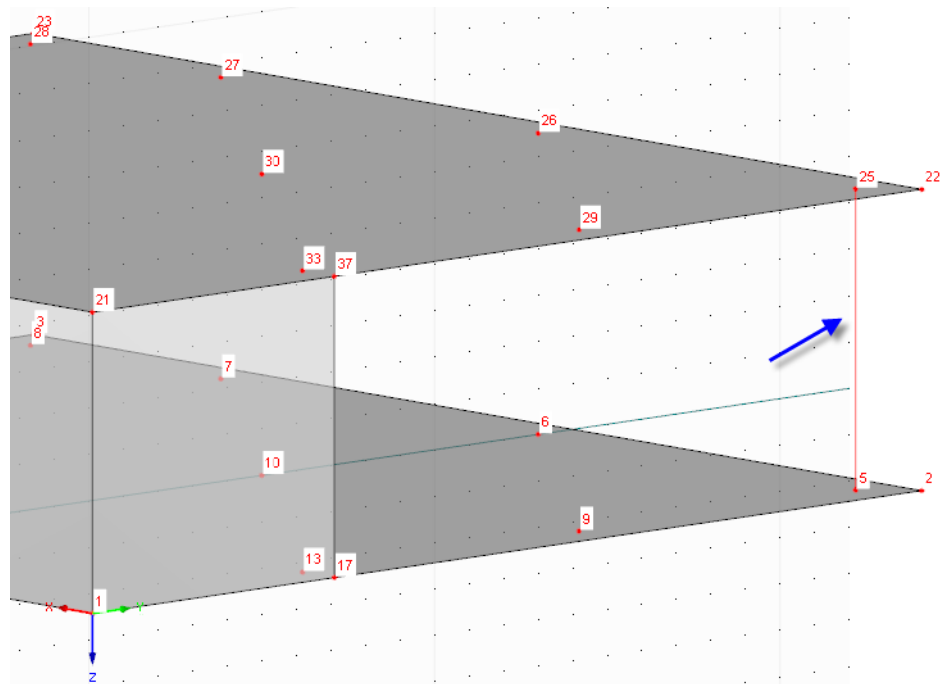
Inner wall

2.2.5 Columns

Create line

We will create one line, allocate member properties and then copy it.

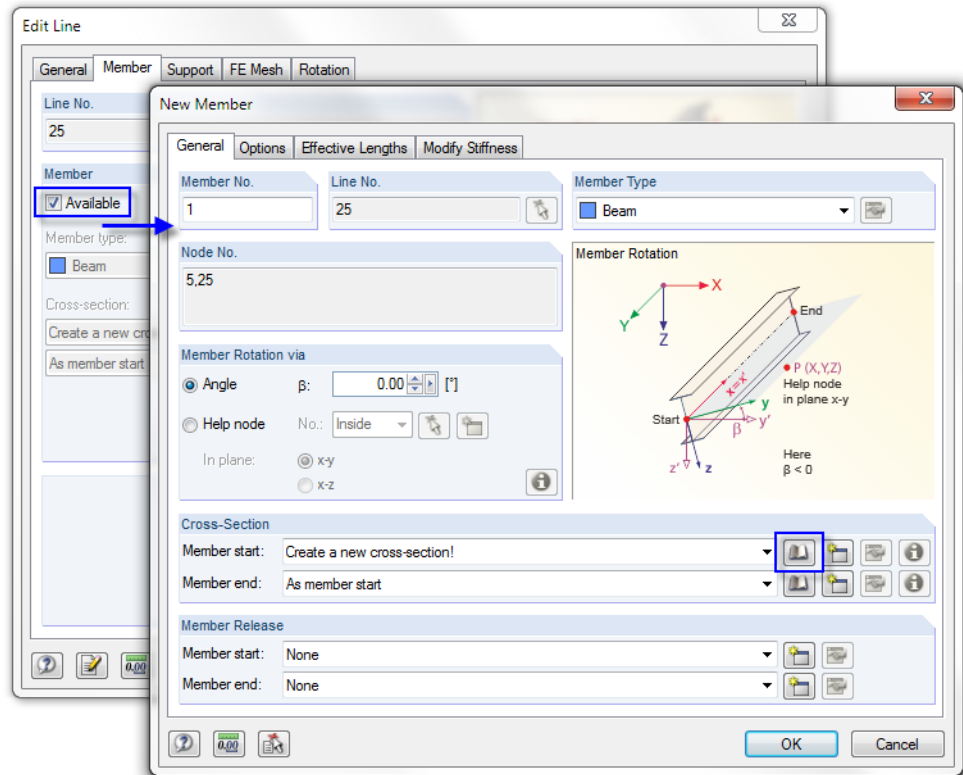
We click the **[New Line]** button. Then we connect nodes **5** and **25** by clicking those two points.



New line

Create member

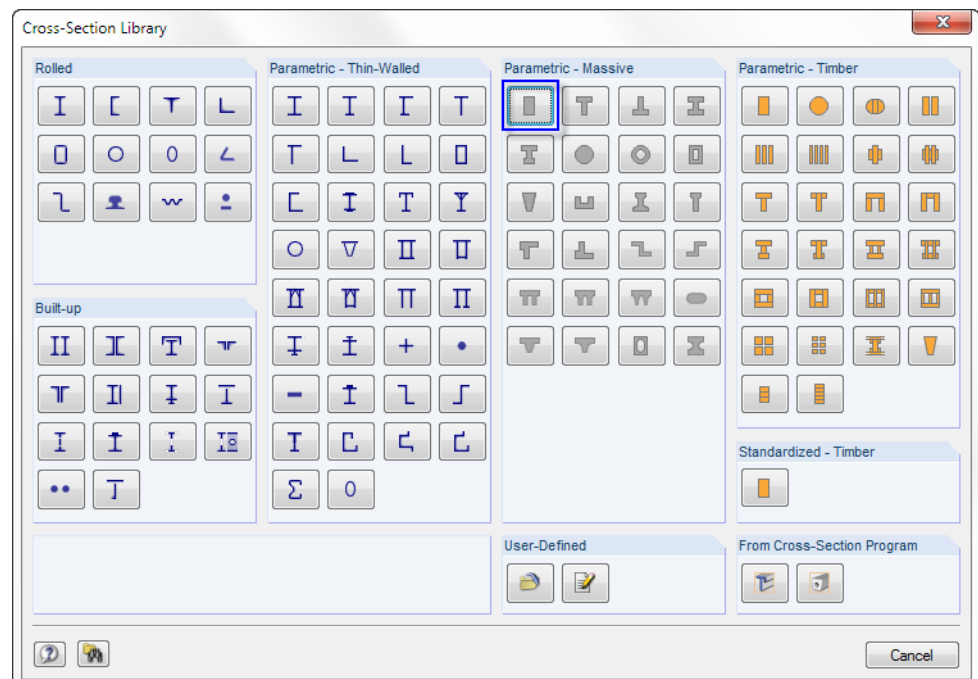
Now we double-click the new line 25 to open the **Edit Line** dialog box. In the **Member** tab, we select the **Available** check box. The **New Member** dialog box opens.



Dialog box *New Member*



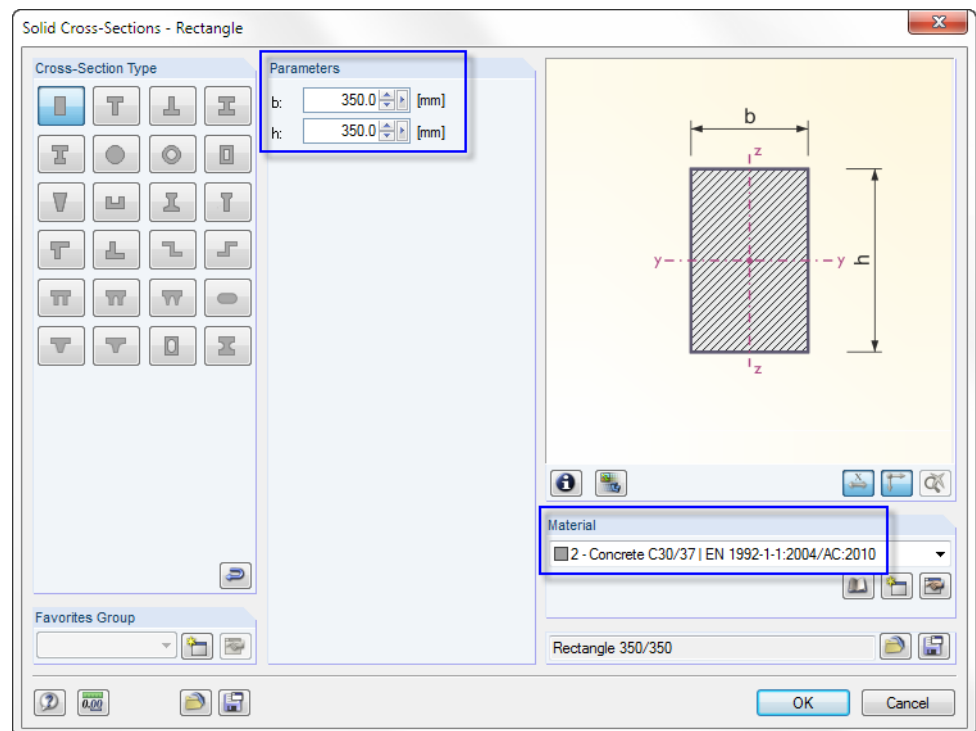
We use the **[Cross-Section Library]** button to open the **Cross-Section Library**.



Cross-Section Library



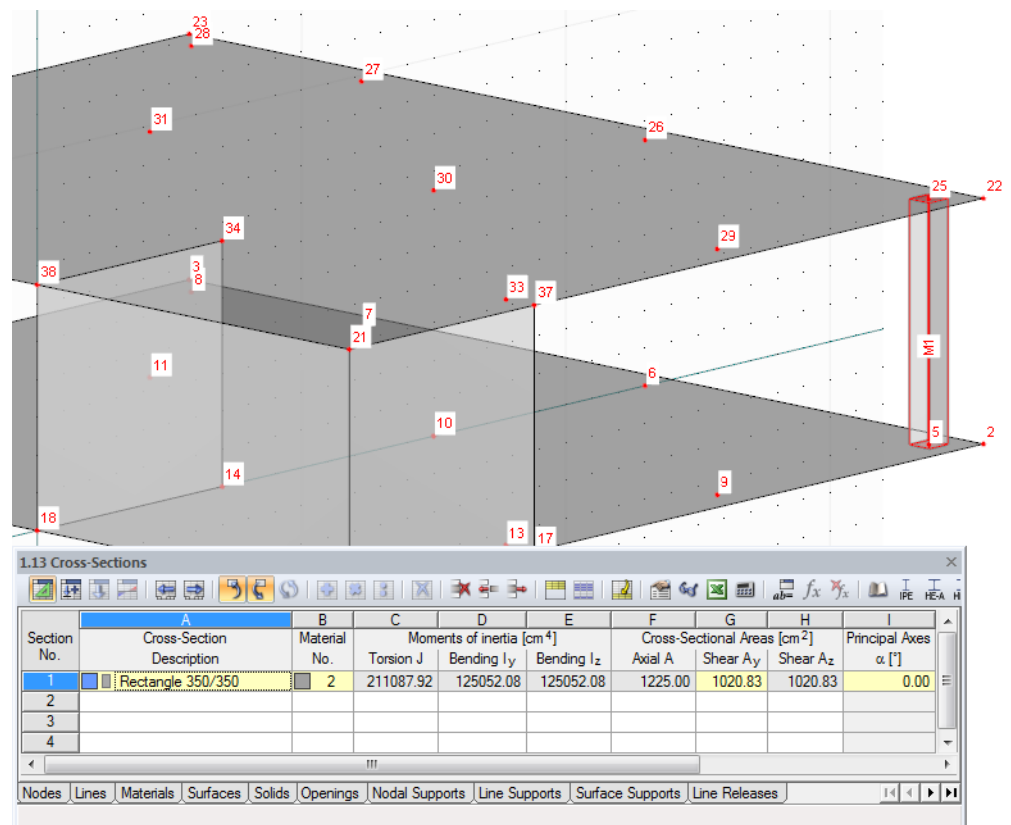
We click the **[Rectangle]** button to open the **Solid Cross-Sections - Rectangle** dialog box.



Dialog box *Solid Cross-Sections - Rectangle*

We enter **350** mm for each side (column 35/35) and set **Concrete C30/37**.

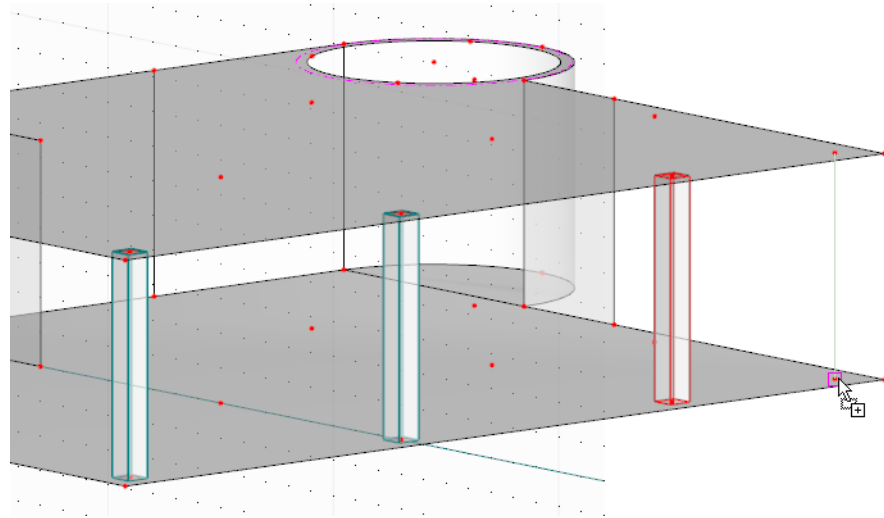
We click [OK] to confirm all dialog boxes. The cross-section is then displayed in the model.



Column cross-section

Copy members

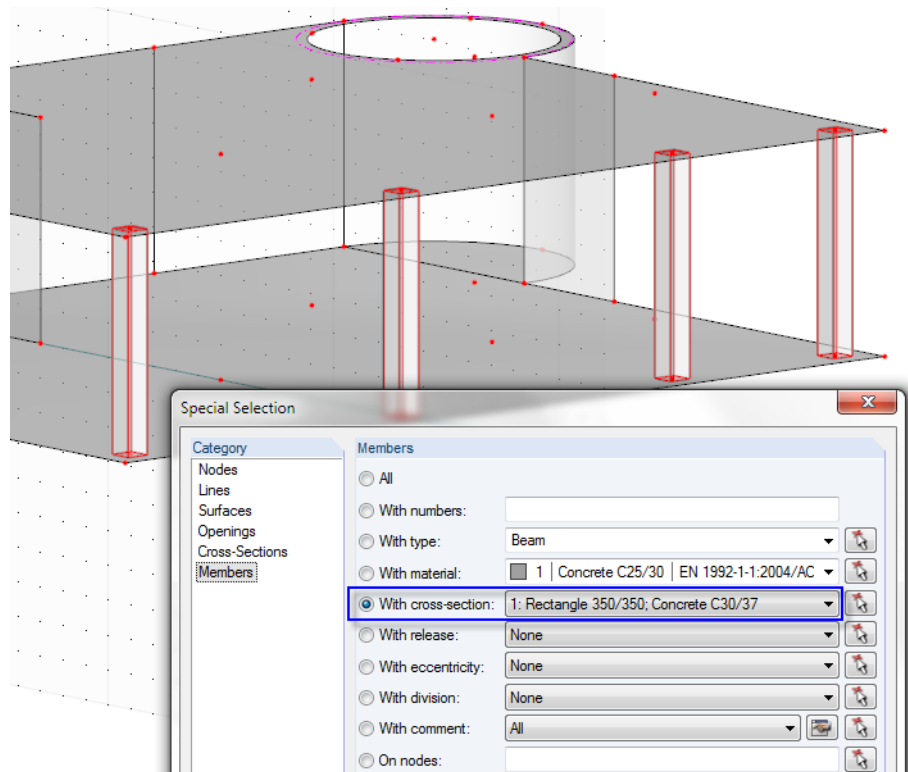
We now use the [Ctrl] key to copy the column graphically in axis 2.



Copying columns graphically

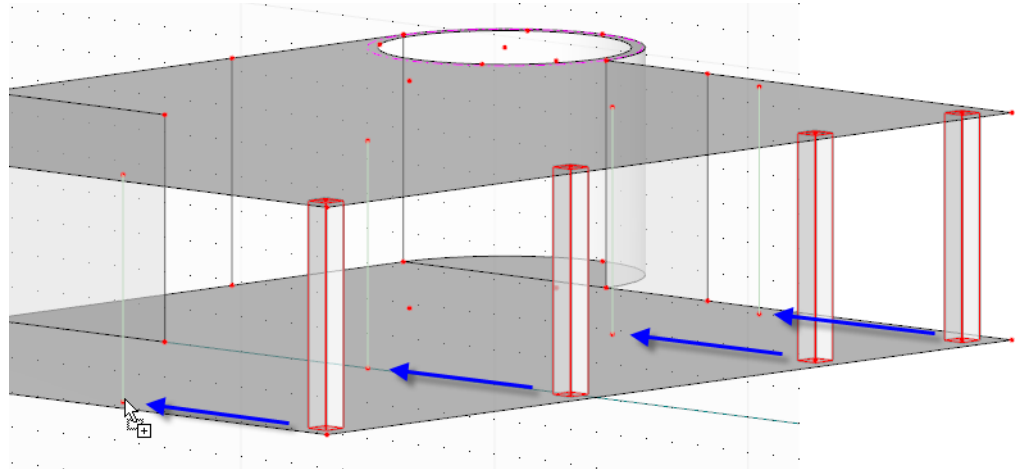


We select all four columns via the button **[Special Selection]**.



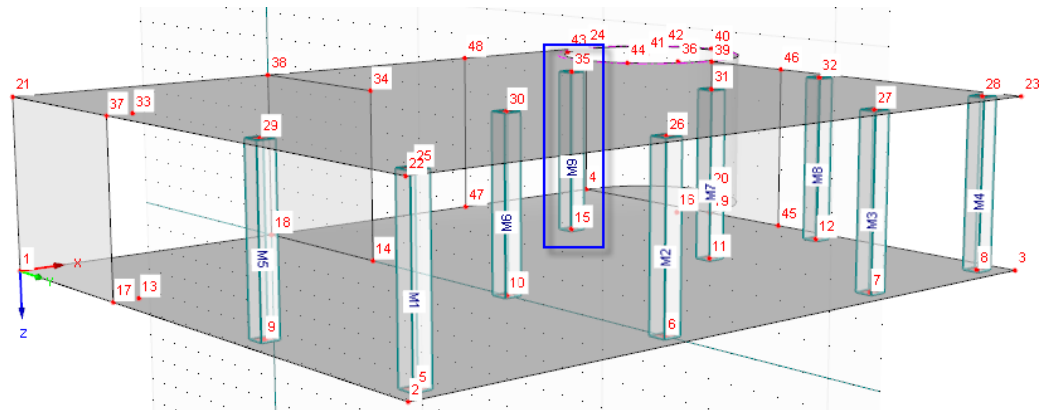
Dialog box *Special Selection*

We now copy the four columns at once to axis 3 by drag and drop, using the [Ctrl] key again.



Copying four columns

Finally, we copy one column to connect nodes 15 and 35 (axis D/5).

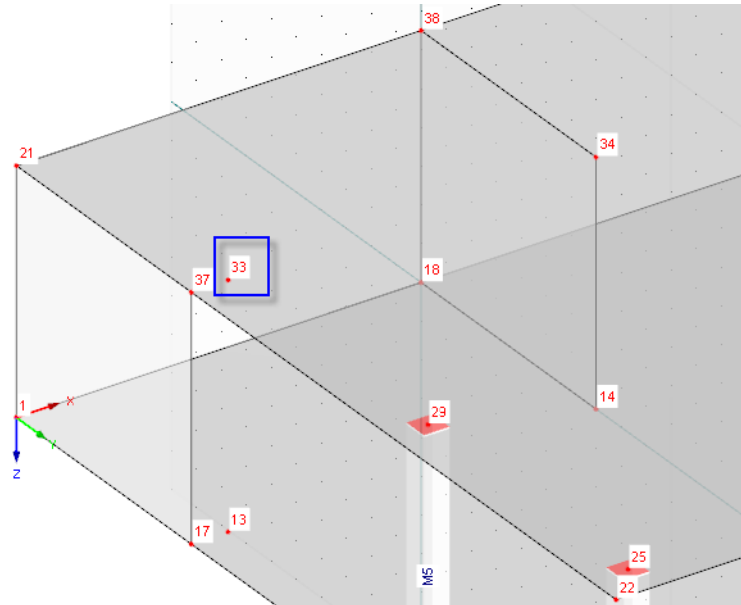


Model with columns

2.2.6 Downstand Beam

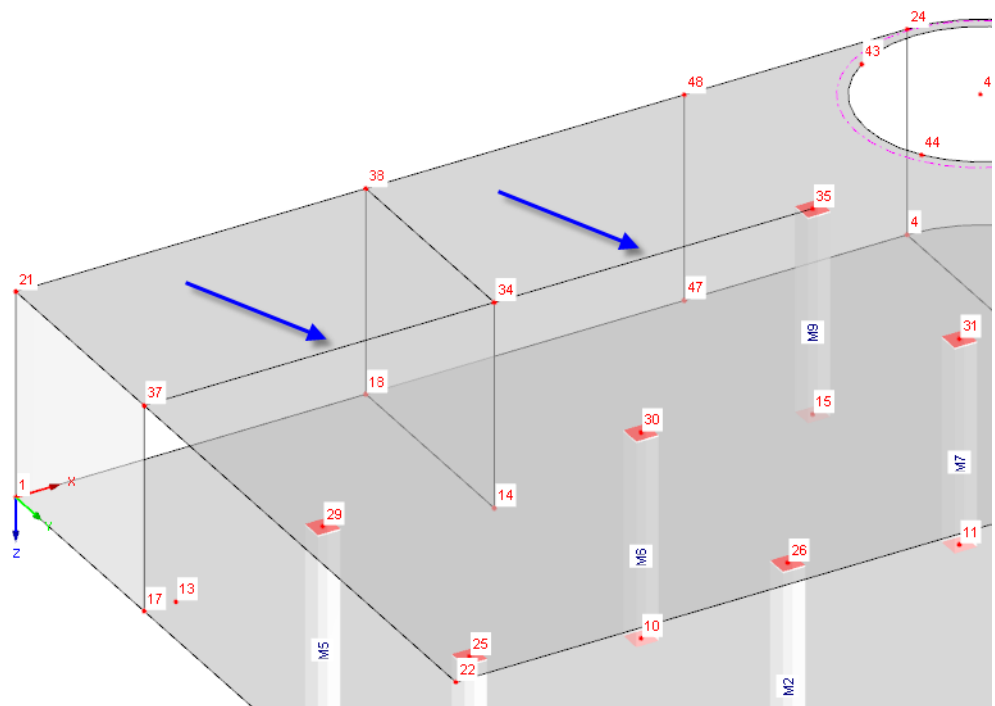
We create a downstand beam in axis 5. It runs from the wall in axis A to the column in axis D.

First we delete node 33 in the ceiling slab.



Deleting node 33

Now we draw two lines from node 37 to node 34 and from node 34 to node 35.



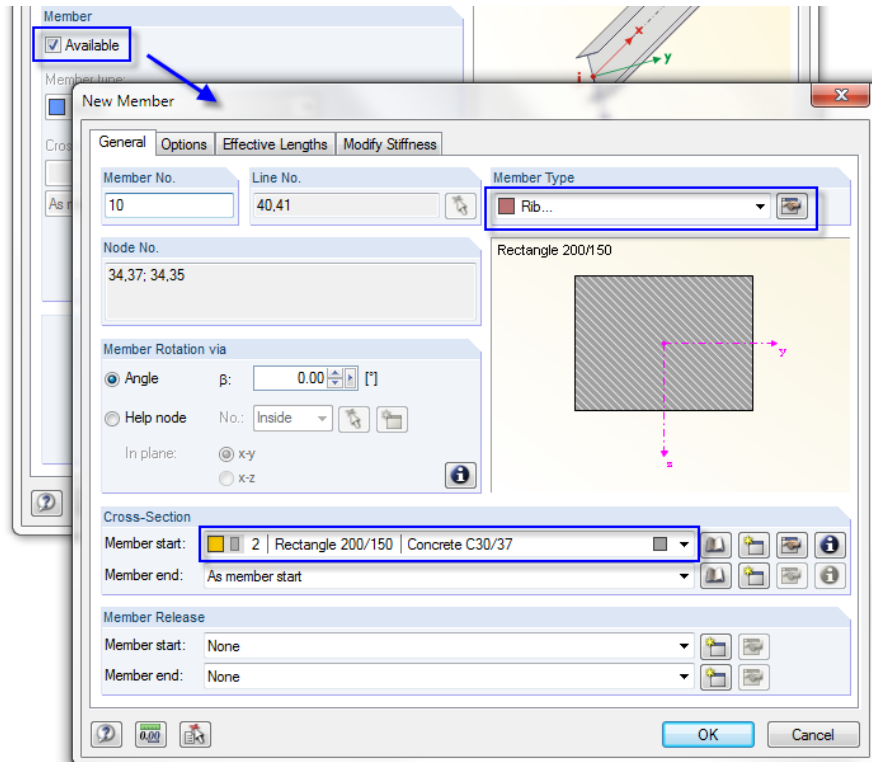
Setting lines

We select the two new lines and double-click them.

In the **Edit Line** dialog box, we check the **Available** option in the **Member** tab again.



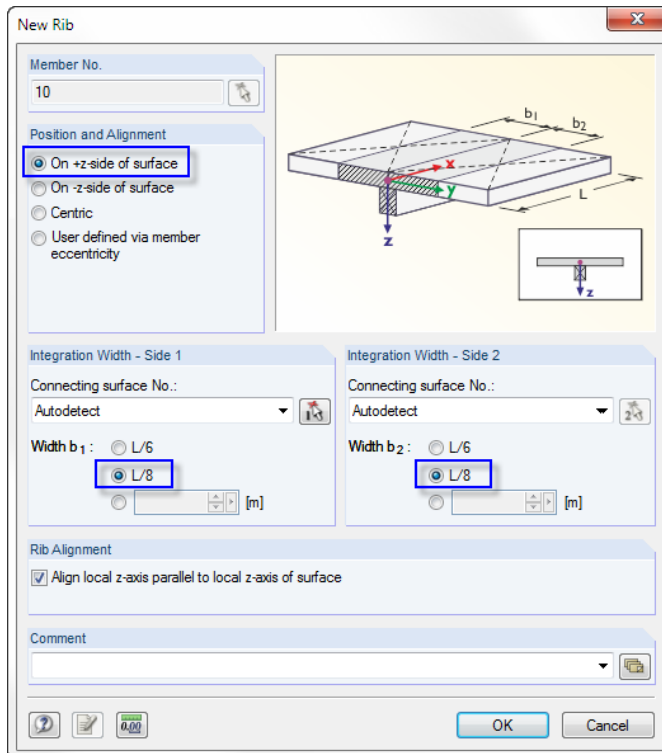
In the **New Member** dialog box, we select the **Member Type Rib**. We define a **[New]** rectangular cross-section with a width of **200 mm** and a height of **150 mm**.



Dialog box *New Member*

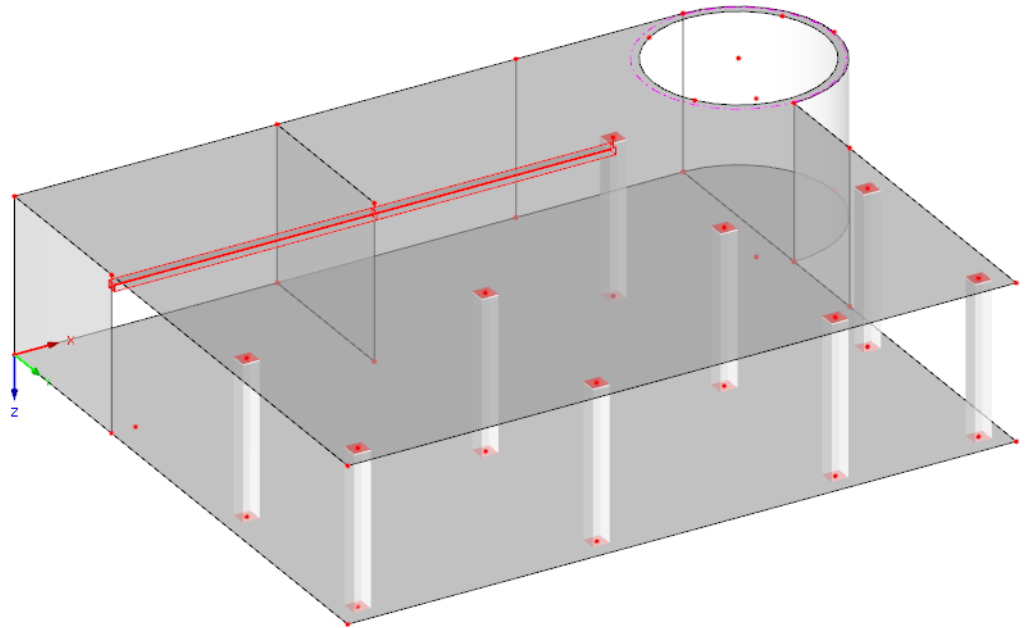


We click the **[Edit Member Type]** button to open the **New Rib** dialog box. There, we select the **On +z-side of surface** option. The integration width is to be **L/8** for sides **1** and **2**.



Dialog box *New Rib*

Having confirmed all dialog boxes with [OK], we can see the downstand beam in our model.

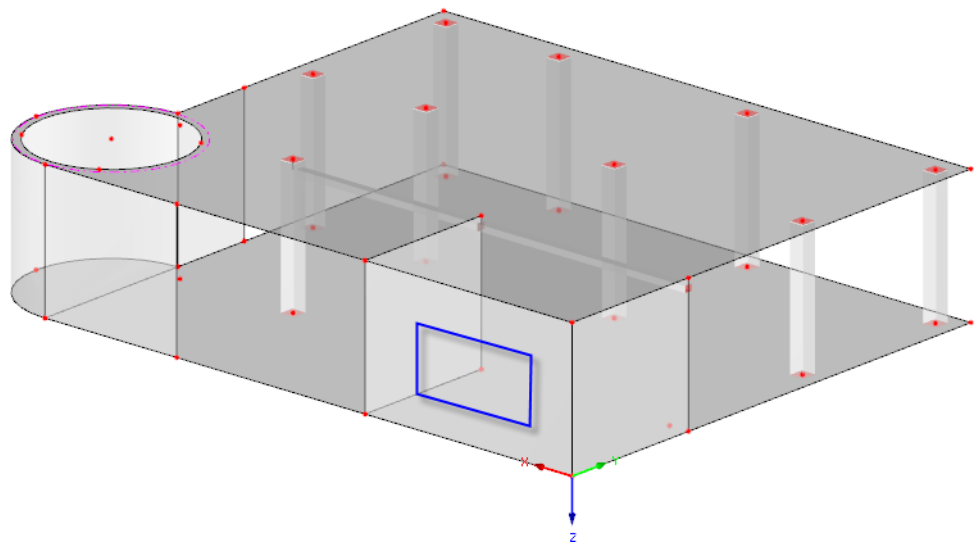


Rib in model

For ribs, the internal forces are determined according to the requirements of concrete design. The stiffness of a rib member results from the member height and the surface thickness due to the effective widths. The internal forces of a rib member include the member forces and the internal forces of the surface that are integrated from the widths b_1 and b_2 .

2.2.7 Opening

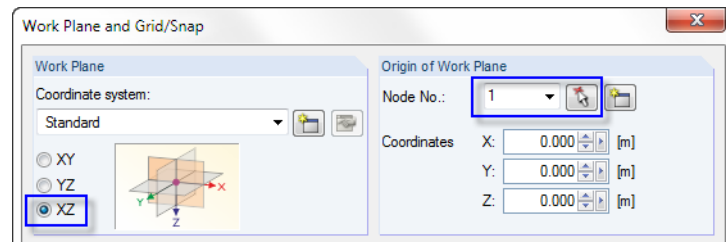
There is a window in the largest wall. It is 3.50 m wide, 1.50 m high and it is located 1.00 m above the surface axis.



Window opening



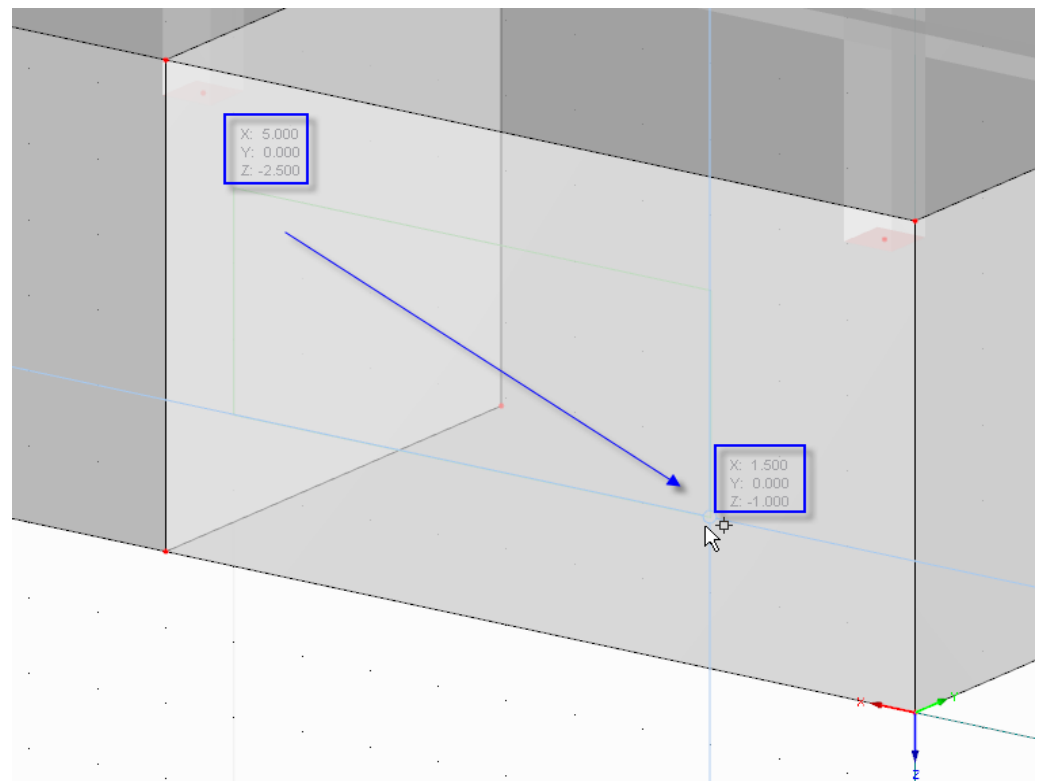
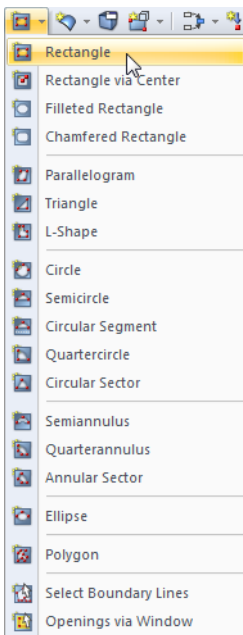
We change the work plane and the origin again because the surface lies in the **XZ** plane. Furthermore, we select node **1** as the new origin.



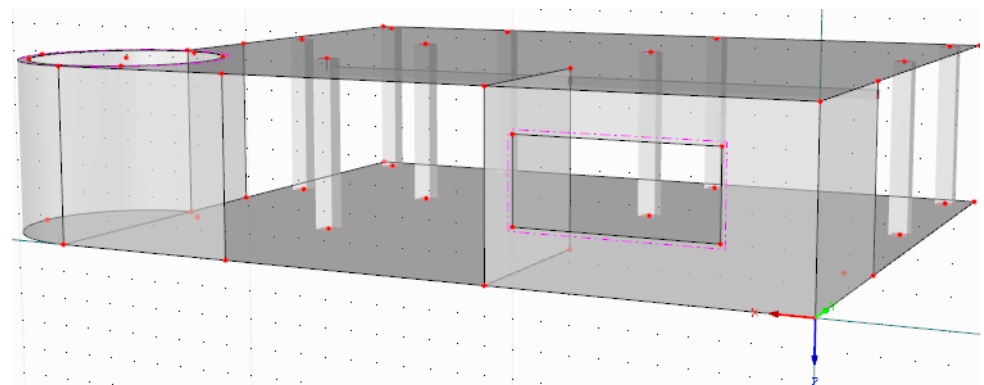
Dialog box *Work Plane and Grid/Snap*

We use the **[New Rectangular Opening]** list button to directly set the window.

The first corner point is **5.00/0.00/-2.50**, the second **1.5/0.0/-1.0**.



Setting opening



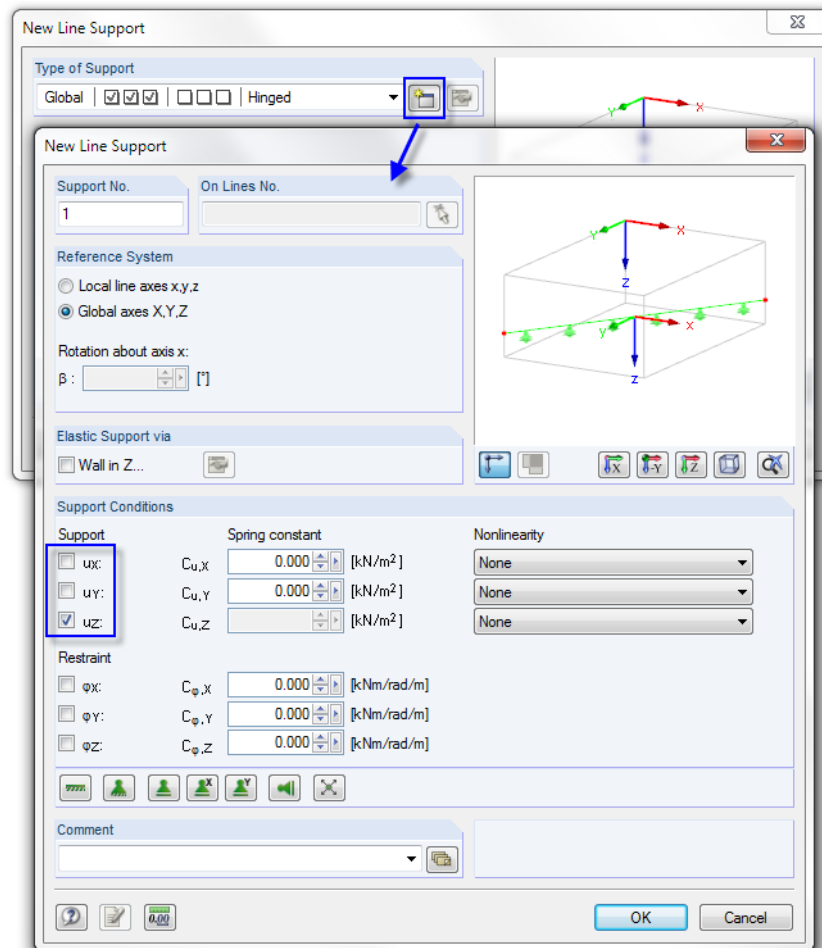
Window opening

2.2.8 Supports

2.2.8.1 Line Supports

Due to the foundation of an adjacent building, we assign support features to the line in axis A.

We click the **[New Line Support]** button to open the **New Line Support** dialog box. The default support is *Hinged* – all three axes are fixed, but not restrained. We have to define a **[New]** type of support because there is a support in the global Z-direction only.

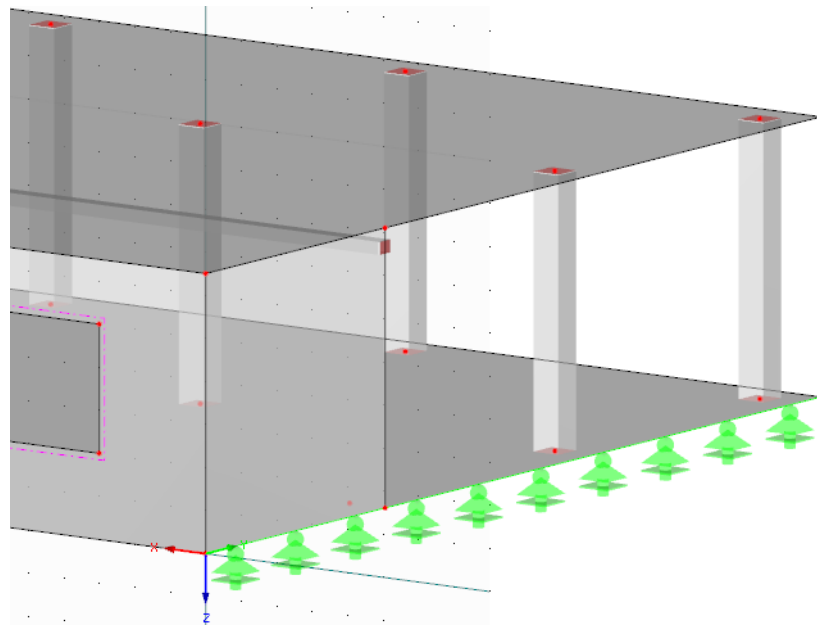


Dialog box *New Line Support*

We clear the check boxes for the displacements u_x and u_y . There is only a support for u_z .

Then we close all dialog boxes with [OK].

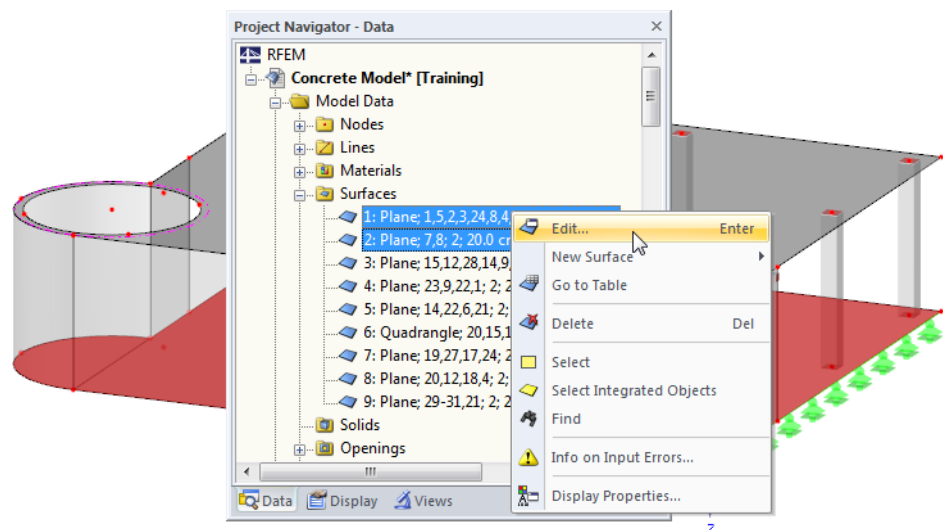
We click lines **1** and **5** to assign the supports. The support features are assigned continuously to the line.



Line supports

2.2.8.2 Surface Supports

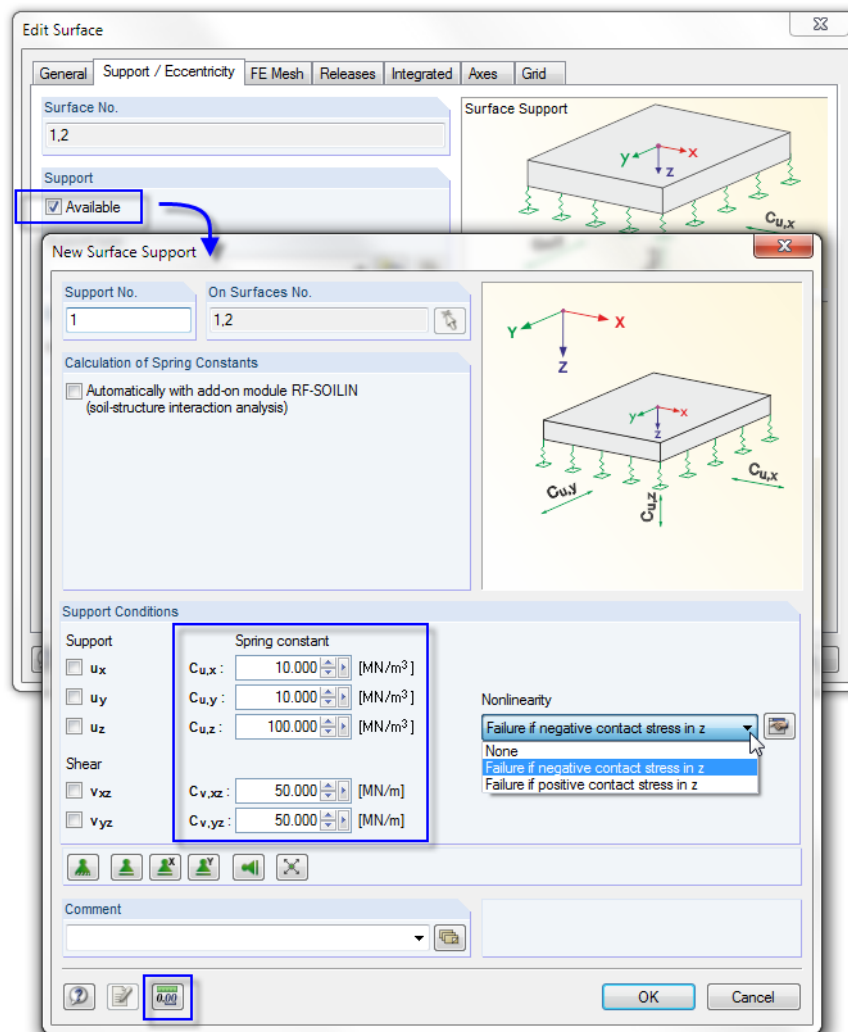
To enter the foundation for the floor slab, we select surfaces **1** and **2** in the navigator.



Context menu of floor slabs

To edit the surfaces, we use the **Edit** option of the context menu.

In the **Edit Surface** dialog box, we select the tab **Eccentricity / Support**. When we check the **Available** option there, the dialog box **New Surface Support** appears (see figure below).

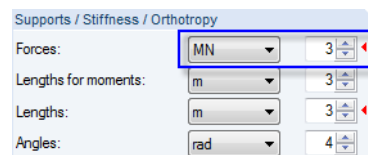
Dialog box *New Surface Support*

For our model, the foundation modulus 100 MN/m^3 for gravel is applicable.

We enter **100** MN/m^3 for the $C_{u,z}$ spring constant. The remaining spring constants are to be defined as seen in the figure above. The parameters $C_{v,xz}$ and $C_{v,yz}$ represent the shear capacities of the surface.



It is possible to adjust the units of spring constants via the **[Units and Decimal Places]** button.



Adjusting units

Furthermore, we select **Failure if negative contact stress in z** from the **Nonlinearity** list. This option corresponds to failure under tension, that is, if the slab is lifting, no more support is provided.

To model the subsidence basin appropriately, we would have to assign line and nodal supports at the edges and corners (see chapter 4.9 of the RFEM manual). However, we will neglect that.

When we have closed all dialog boxes with [OK], the input of the model is complete.

2.3 Load Cases and Combinations

We define six load cases, classify them and create the combinations. The loads will be allocated to each load case later.

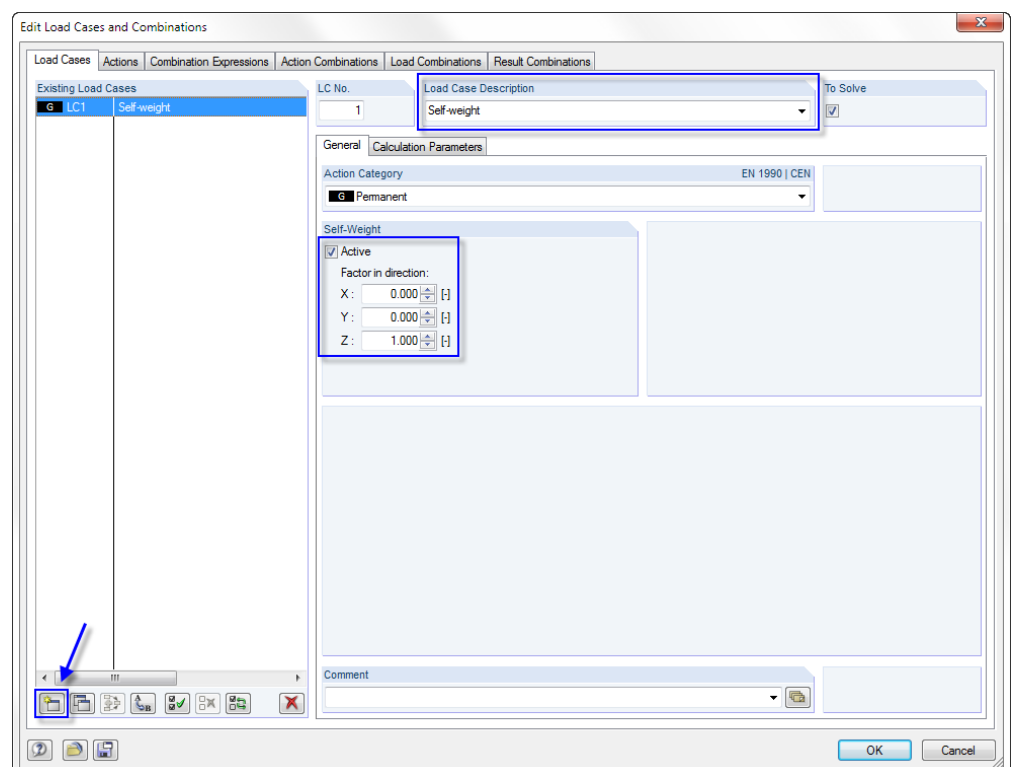
2.3.1 Load Cases

The entire loading is split into load cases, i.e. each load type that is to be treated separately is managed in one specific load case.

Load case 1: Self-weight

To generate this load case, we click the button **[New Load Case]**.

A new window opens. We choose the *Load Case Description* **Self-weight** from the list.



Dialog box *Edit Load Cases and Combinations*: LC1 Self-weight

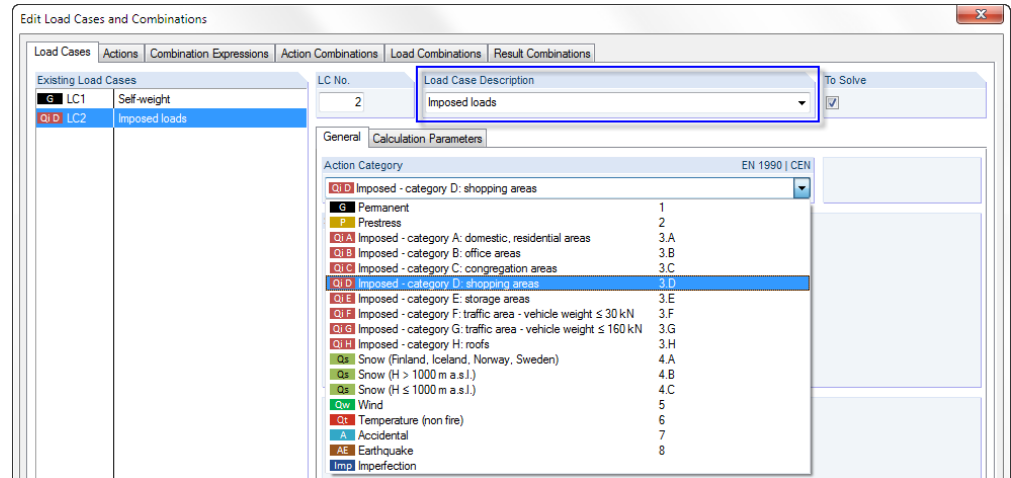
As *Action Category*, **Permanent** is set as default. Same applies to the factor **1.000** for the automatic self-weight which is to act in direction **Z**: The loads are generated from the properties of the material, surfaces and cross-sections.

We then use the **[Create New Load Case]** button to define the next load case.



Load case 2: Imposed loads

We set the *Description* **Imposed Loads** and the *Category* **Imposed - category D: shopping areas**. The self-weight is deactivated by default.



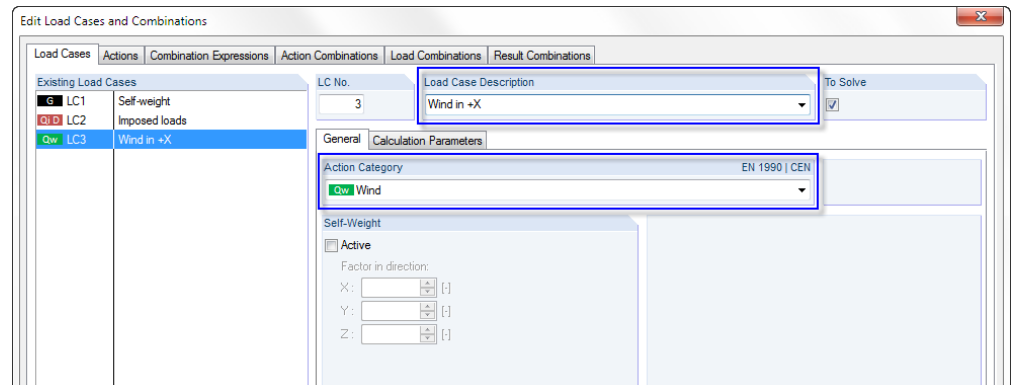
LC2: Imposed loads



We use the **[Create New Load Case]** button again.

Load case 3: Wind in +X

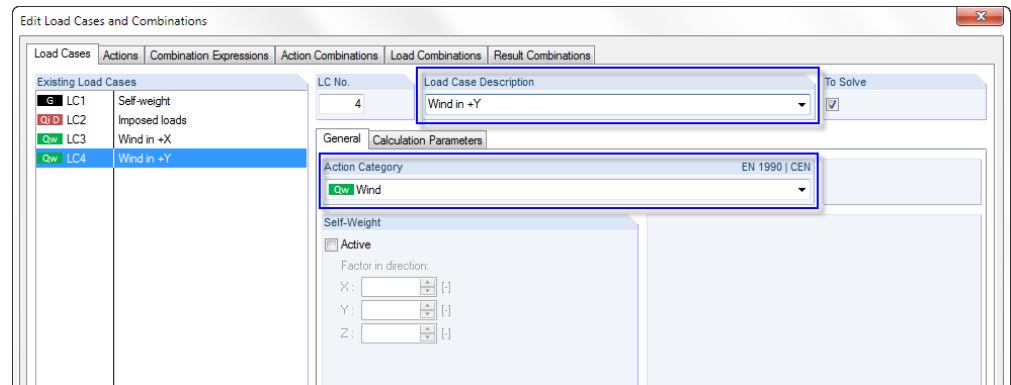
We select the *Description* **Wind in +X** from the list. The *Category* **Wind** is set automatically.



LC3: Wind in +X

Load case 4: Wind in +Y

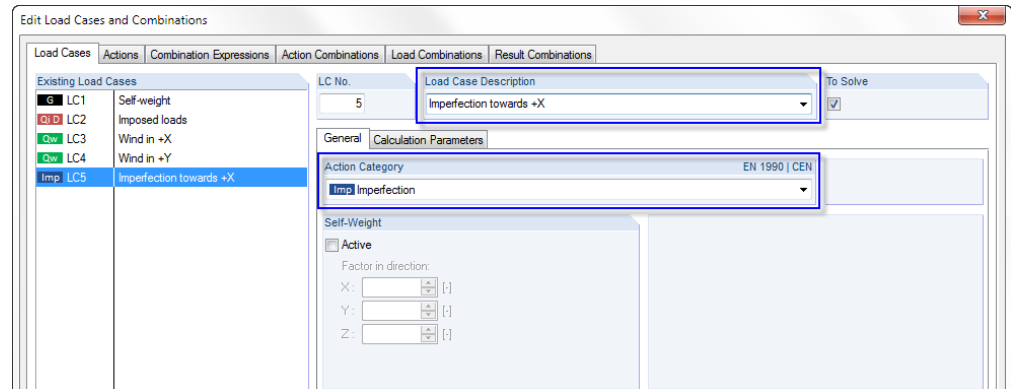
We select **Wind in +Y** from the list. The *Category* **Wind** is set automatically again.



LC4: Wind in +Y

Load case 5: Imperfections towards +X

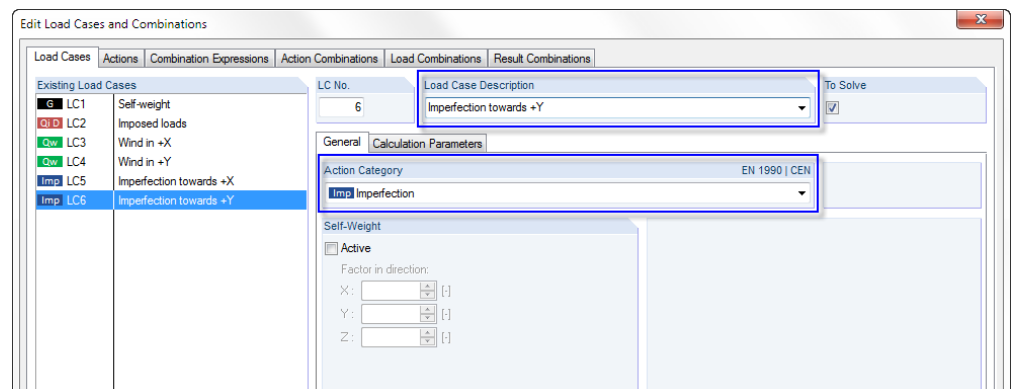
We select **Imperfections towards +X**. The **Category Imperfection** is set automatically.



LC5: Imperfection towards +X

Load case 6: Imperfections towards +Y

We select **Imperfections towards +Y**. The **Category Imperfection** is set automatically again.

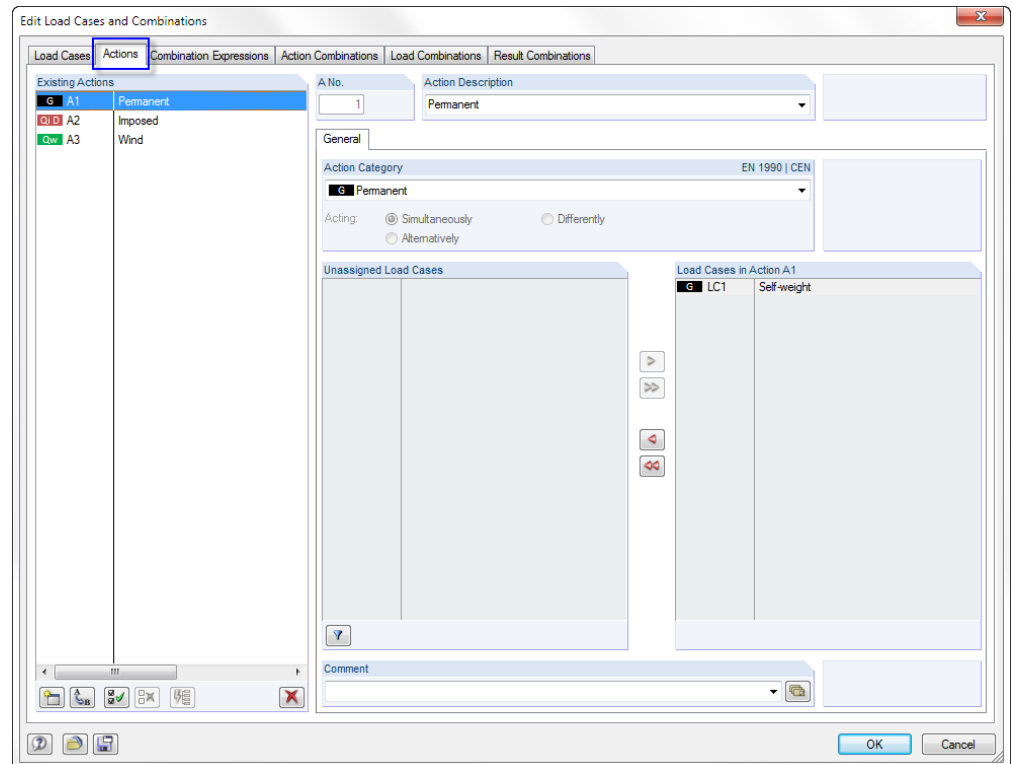


LC6: Imperfection towards +Y

2.3.2 Actions

We now assign the load cases to actions which can be combined according to EN 1990. Those actions represent independent influences that have different origins.

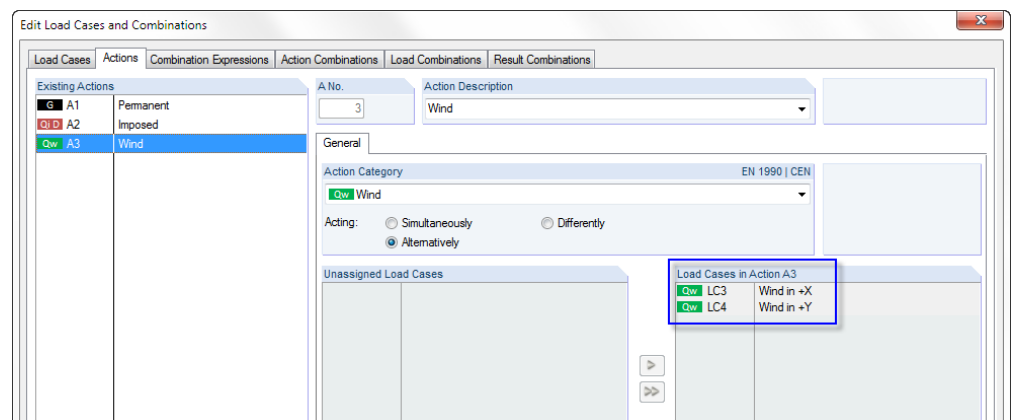
In the dialog box *Edit Load Cases and Combinations*, we click the next tab **Actions**.



Dialog box *Edit Load Cases and Combinations*, tab *Actions*

In the dialog section *Existing Actions*, three action categories are listed. We created them when defining the load cases. The *Imperfection* category is missing. This list includes only actions that have "real" loads.

The load cases have been automatically assigned to actions. Our two wind load cases are listed in the action A3 *Wind*.

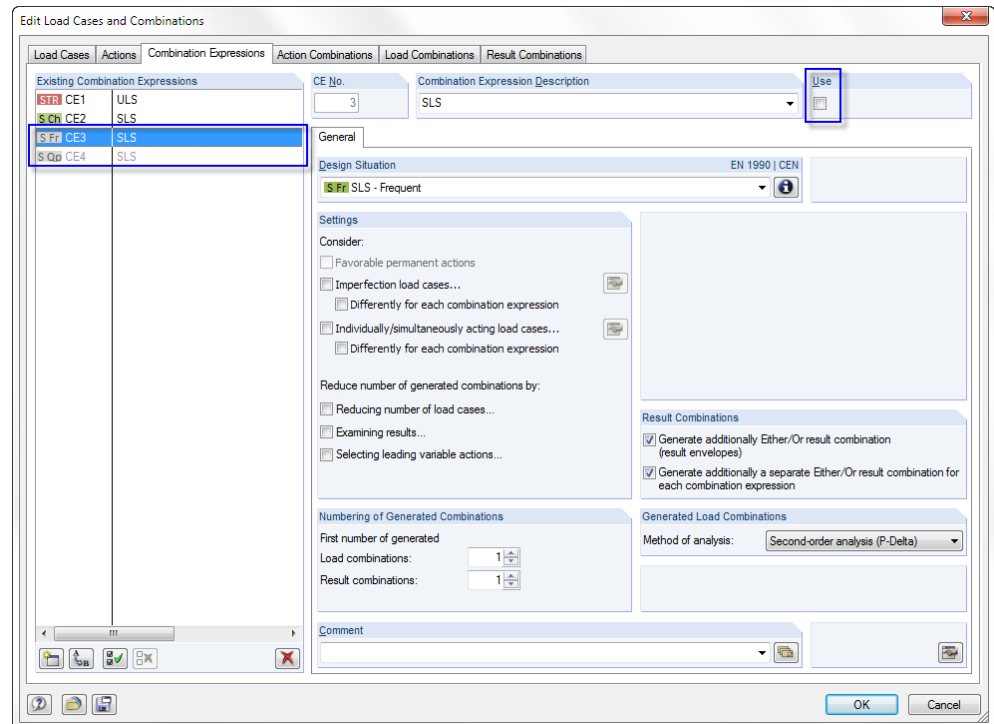


Load cases in action *Wind*

2.3.3 Combination Expressions

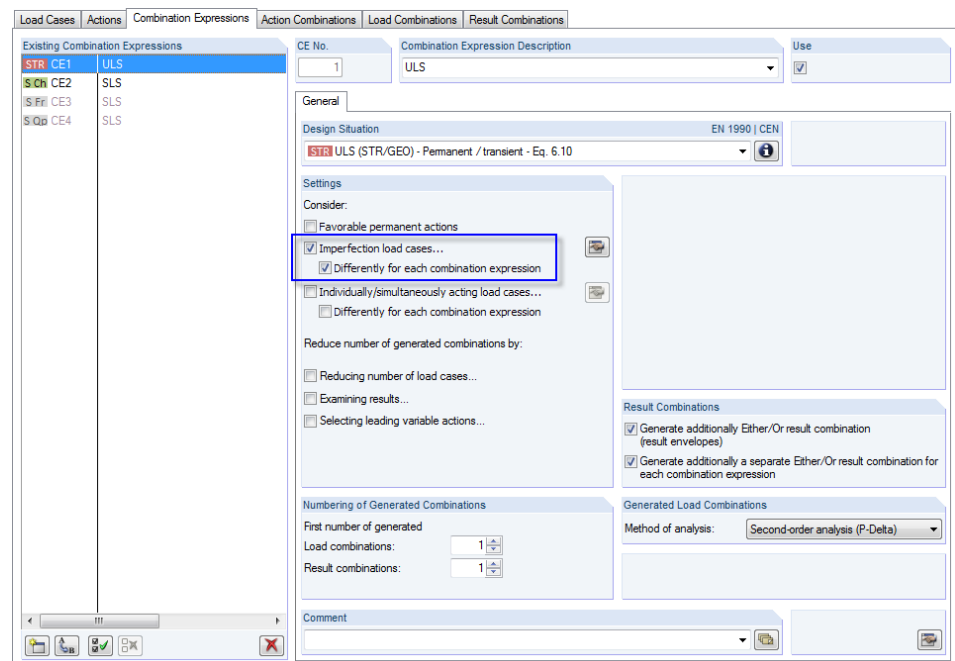
To combine the actions according EN 1990 for the ultimate and the serviceability limit states, we click the next tab **Combination Expressions**.

There are four combination expressions. We disable **CE3** (*SLS - Frequent*) and **CE4** (*SLS - Quasi-permanent*) so that we have only the *ULS (STR/GEO)* and *SLS - Characteristic* design situations.



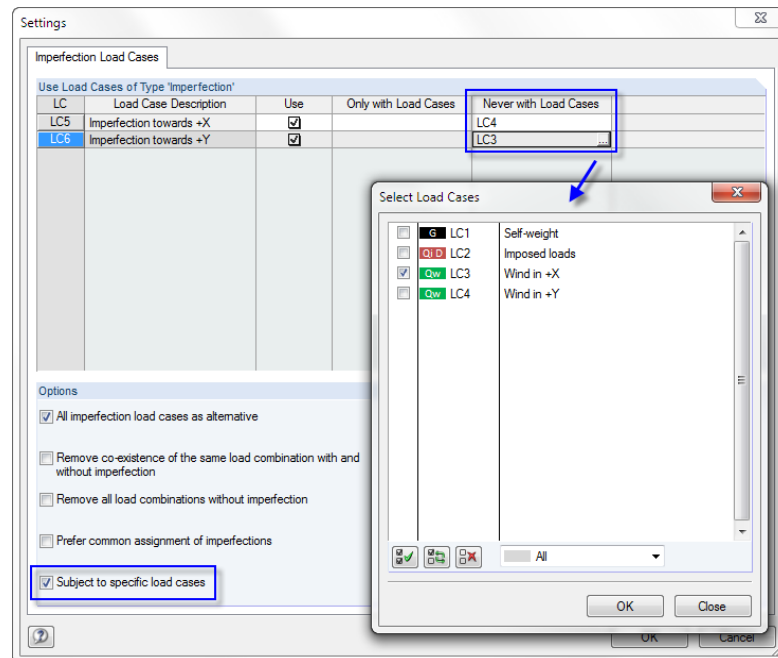
Disabling SLS design situation

We select **CE1**. To consider the imperfections, we check the **Imperfection load cases** option. As imperfections are not relevant for SLS, we select the **Differently for each combination expression** option as well (and will later deactivate the imperfection load cases for **CE2**).



Activating imperfections for *ULS* design situation

The *Settings* dialog box is opened. We activate the **Subject to specific load cases** option.

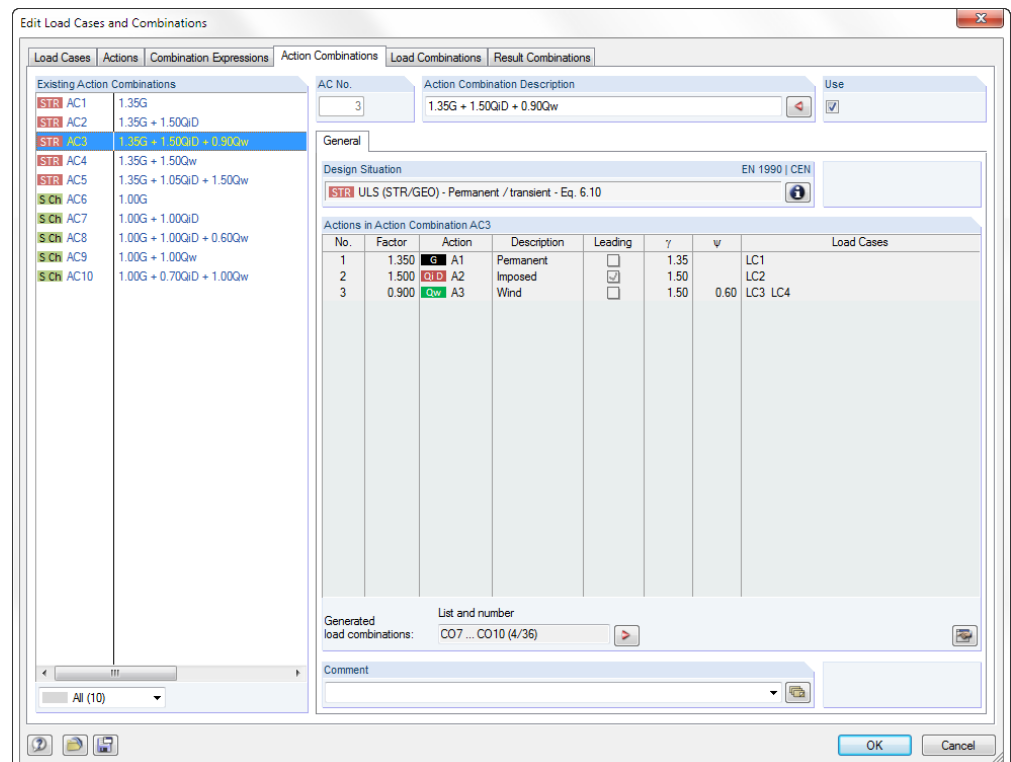


Dialog box *Settings* and *Select Load Cases*

To reduce the number of combinations, we exclude all combinations in which imperfections act in different directions than wind (see figure above). **LC5** is never to be combined with **LC4**, **LC6** never with **LC3**. We can use the [...] buttons to allocate the load cases.

2.3.4 Action Combinations

When we click the tab **Action Combinations**, RFEM combines the actions according to EN 1990.



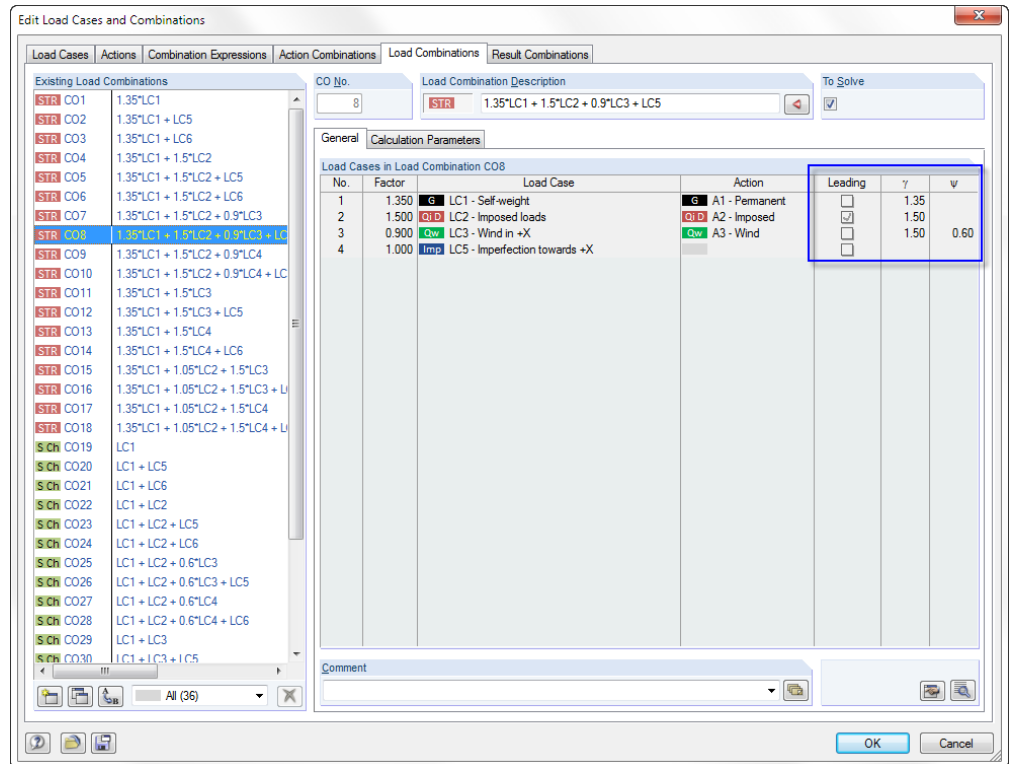
Dialog box *Edit Load Cases and Combinations*, tab *Action Combinations*

There are five combinations for the ULS and SLS design situations each.

We can select any combination and check on the combination parameters via the **[Info]** and **[Combination Coefficient]** buttons.

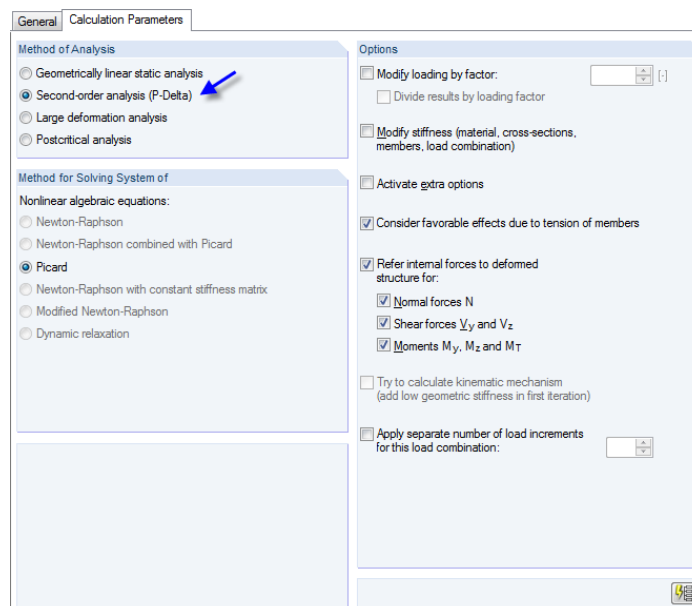
2.3.5 Load Combinations

Next we click the tab **Load Combinations**. RFEM calculates 36 load combinations altogether.



Dialog box *Edit Load Cases and Combinations*, tab *Load Combinations*

The loading of a load combination is treated as one load case which is calculated according to the second-order analysis (P-Delta) by default. We can check on the *Calculation Parameters*.



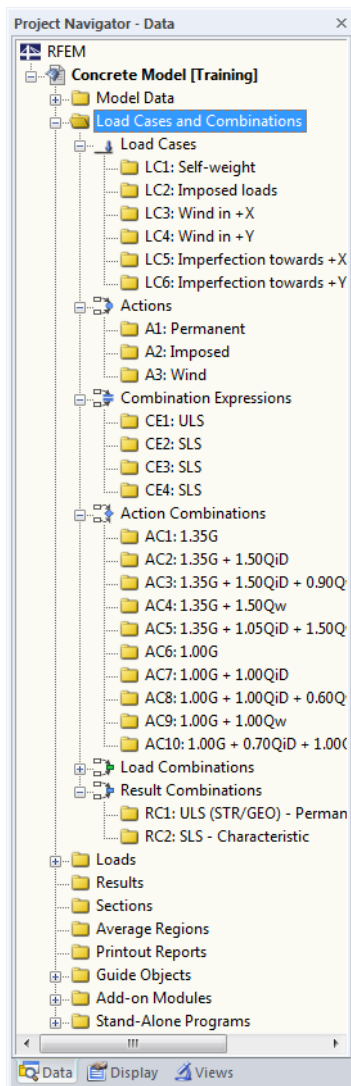
Tab *Calculation Parameters* of a load combination

Finally, we click the tab **Result Combinations** to automatically create an envelope of all results.



In a result combination, the maximum and minimum values of internal forces, deformations, support forces etc. are determined. The model is not recalculated. Instead, the results of the involved load cases or combinations are superimposed.

All load cases and combinations are listed in the *Data* navigator, too.



2.4 Loads

We will now define the loads of each load case.

2.4.1 LC 1: Self-Weight

When we created LC 1, the factor **1.000** in Z-direction was set by default (see chapter 2.3.1, page 43): The self-weight of all surfaces and members is applied automatically.

As there are no additional loads, we are done with the first load case.

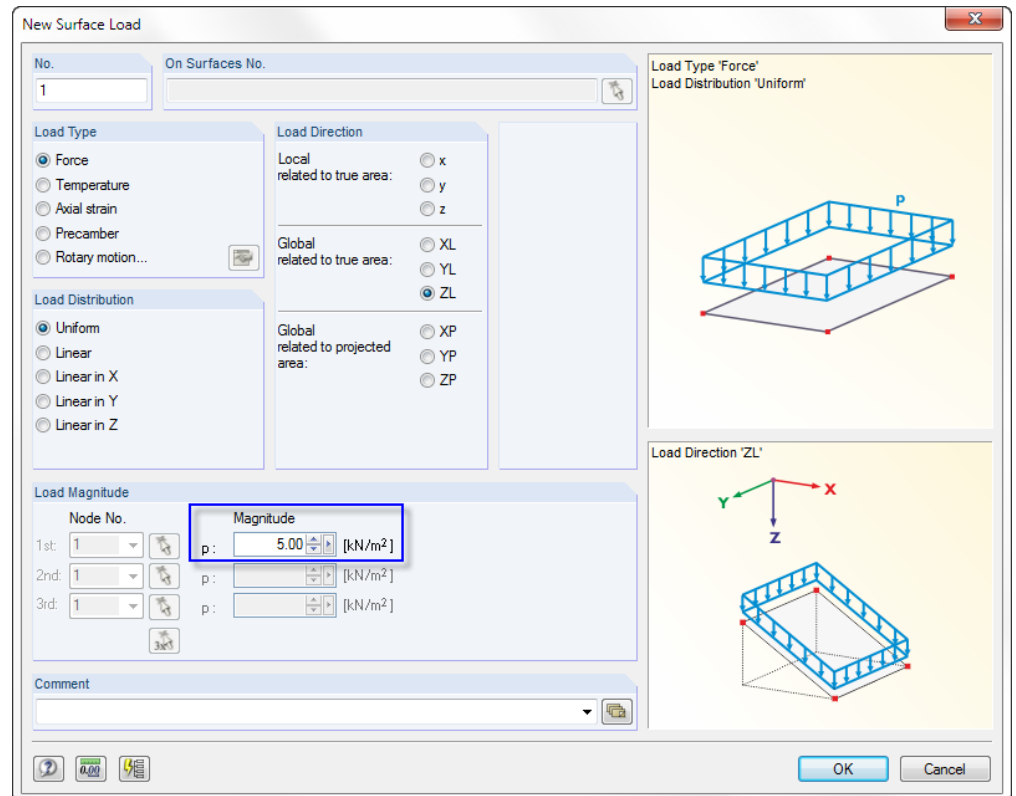
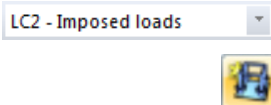
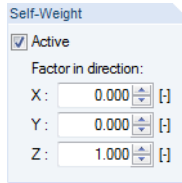
2.4.2 LC 2: Imposed Loads

We set LC2 in the list.

Then we click the **[New Surface Load]** button to define the loads of the floor and ceiling slabs.

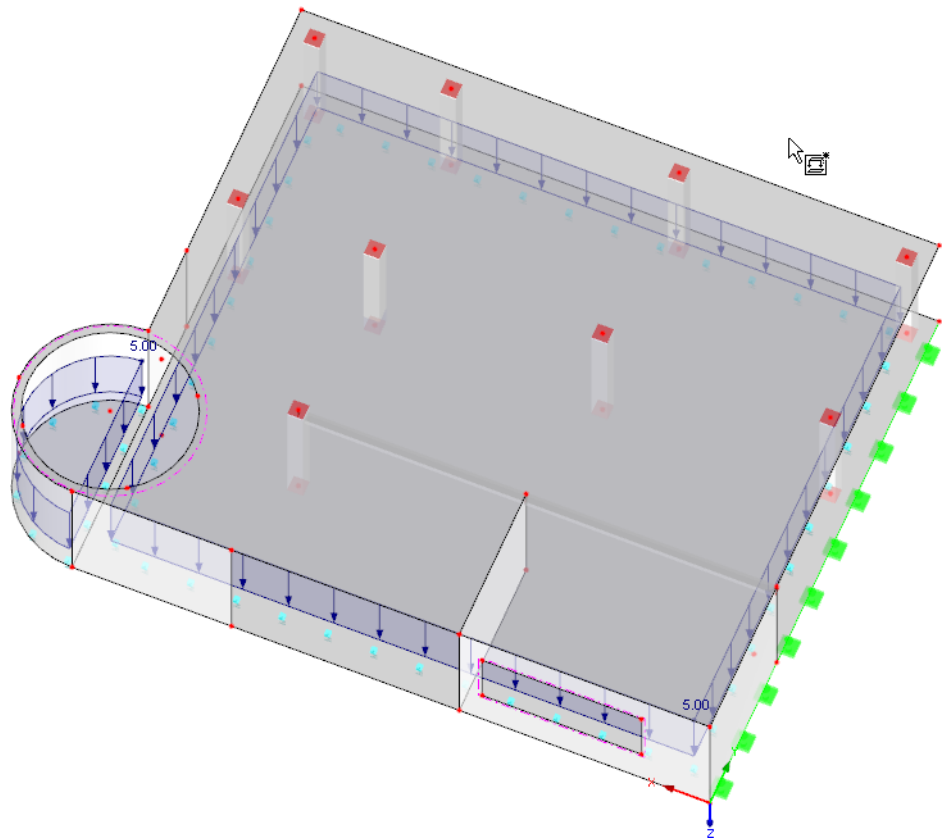
In the **New Surface Load** dialog box, we make sure that the **Load Type Force**, the **Load Direction ZL** and the **Load Distribution Uniform** are set.

We enter the **Load Magnitude 5.0** kN/m² (see page 11).



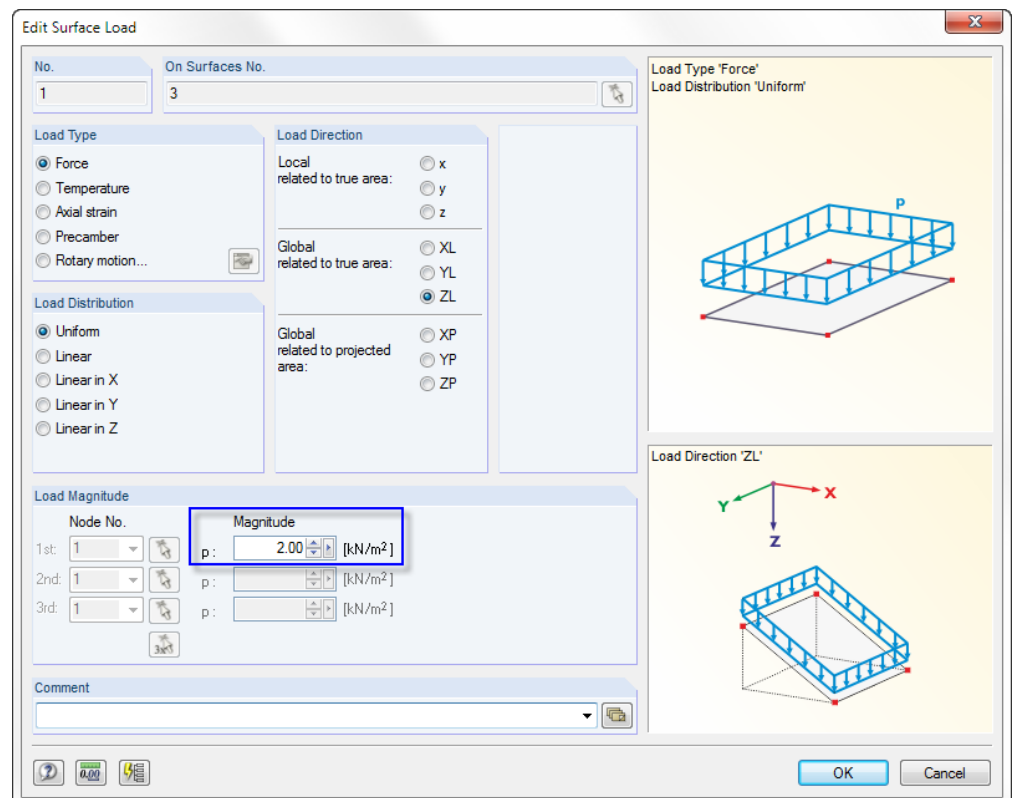
Dialog box *New Surface Load*

We set the load by clicking the two floor surfaces **1** and **2** (see figure below).



Setting surface loads

We continue and set the load on the ceiling surface **3**, too. After that, we double-click the ceiling load and correct the load value to **2.0 kN/m²** (see page 11).



Modifying surface load

2.4.3 LC 3: Wind in +X

LC3 - Wind in +X

We set LC3 in the list.

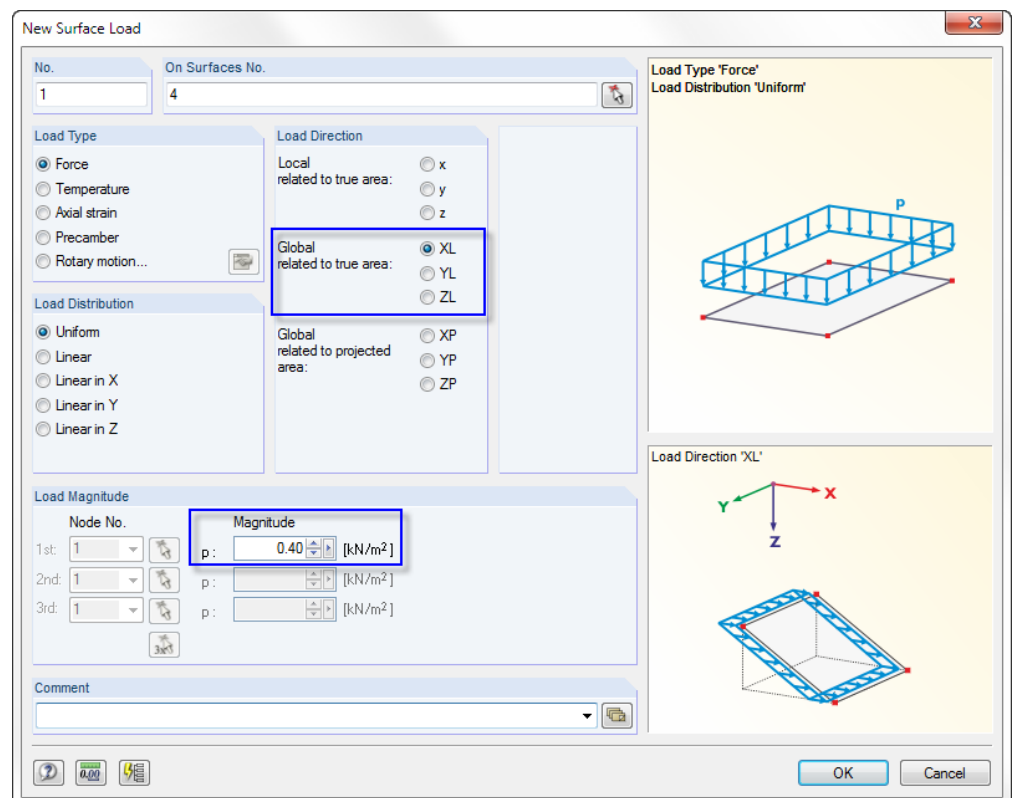
Compression side

For the wind compression in axis A, we apply one surface load and two line loads.

Surface load

We select the wall (surface **4**) and then click the **[New Surface Load]** button.

In the **New Surface Load** dialog box, we set the *Load Direction XL* and enter the *Load Magnitude 0.40 kN/m²* (see page 11).



Dialog box *New Surface Load*

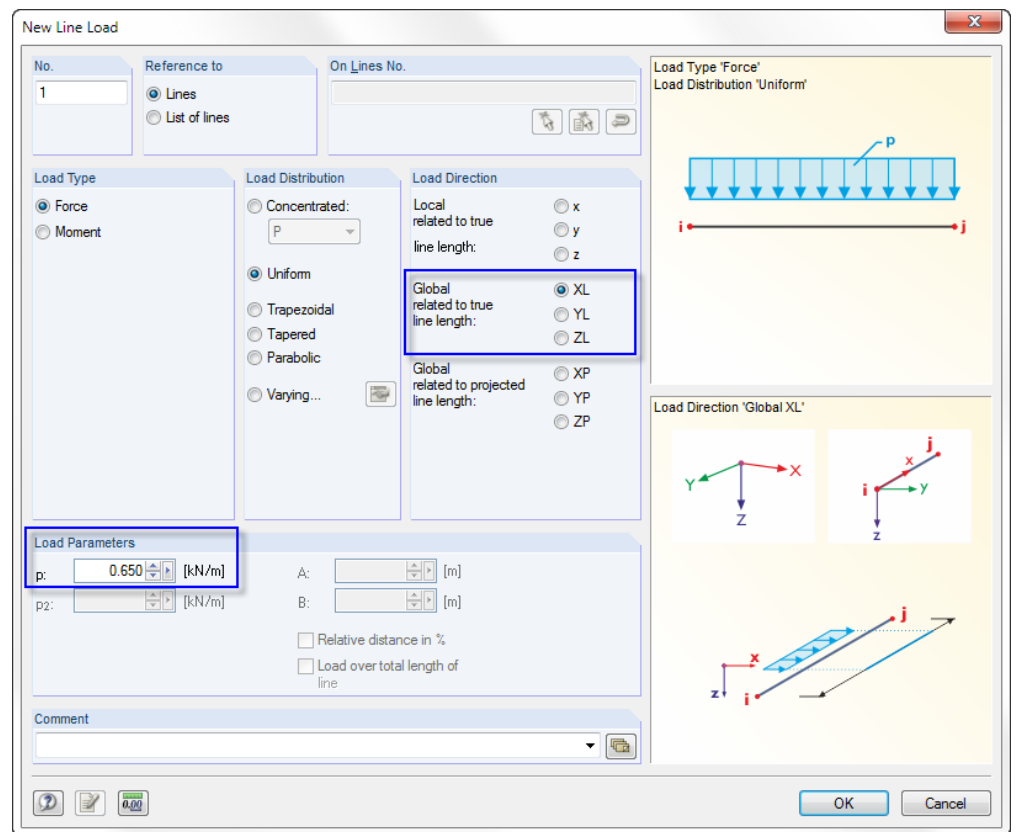
Line loads

For the remaining area of axis A, we have to allocate the surface load to the lines of the floor and ceiling slabs. The line load is $\frac{1}{2} \cdot 0.40 \text{ kN/m}^2 \cdot 3.25 \text{ m} = 0.65 \text{ kN/m}$ for each line.

We click the **[New Line Load]** button to open the **New Line Load** dialog box.

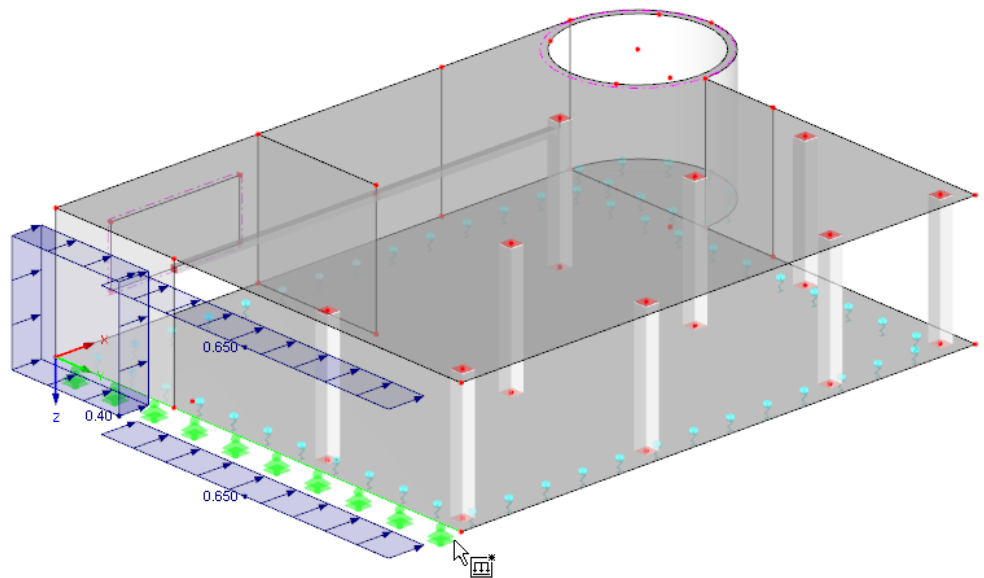
We make sure that the *Load Type Force* and the *Load Distribution Uniform* are set.

Then we set the *Load Direction XL* and enter the *Load Parameters 0.65 kN/m* (see figure below).



Dialog box *New Line Load*

We set the load by clicking lines **13** and **5**.



Setting line loads

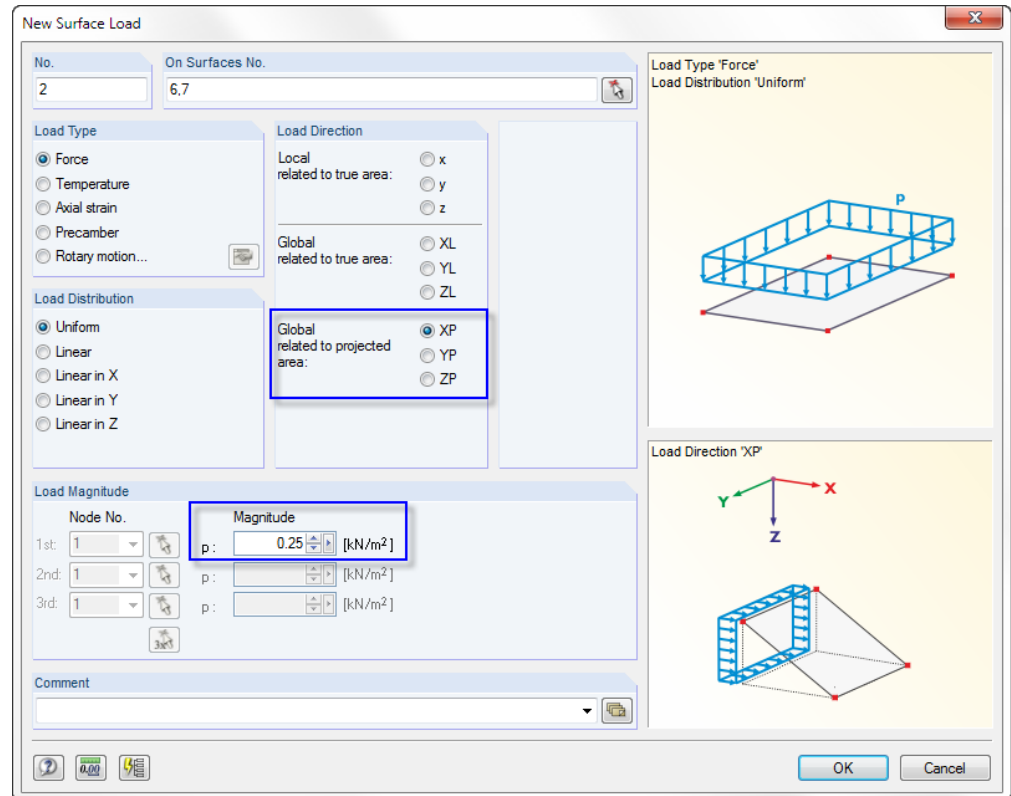
Suction side

For the wind suction in axis F, we apply two surface loads and two line loads.

Surface loads

We select surfaces **6** and **7** and then click the **[New Surface Load]** button again.

We set the *Load Direction* **XP** and enter the *Load Magnitude* **0.25 kN/m²** (see page 11).



Dialog box *New Surface Load*

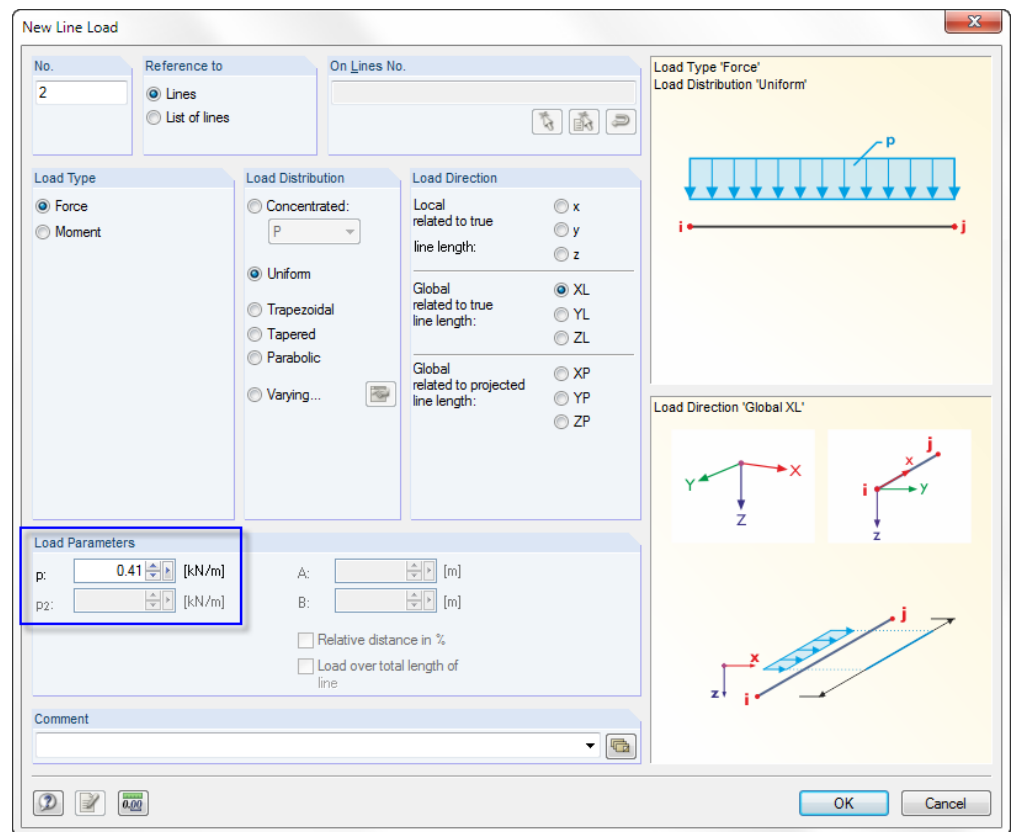
Line loads

For the remaining area of axis F, the converted line load is $\frac{1}{2} \cdot 0.25 \text{ kN/m}^2 \cdot 3.25 \text{ m} = 0.41 \text{ kN/m}$.

We click the **[New Line Load]** button.

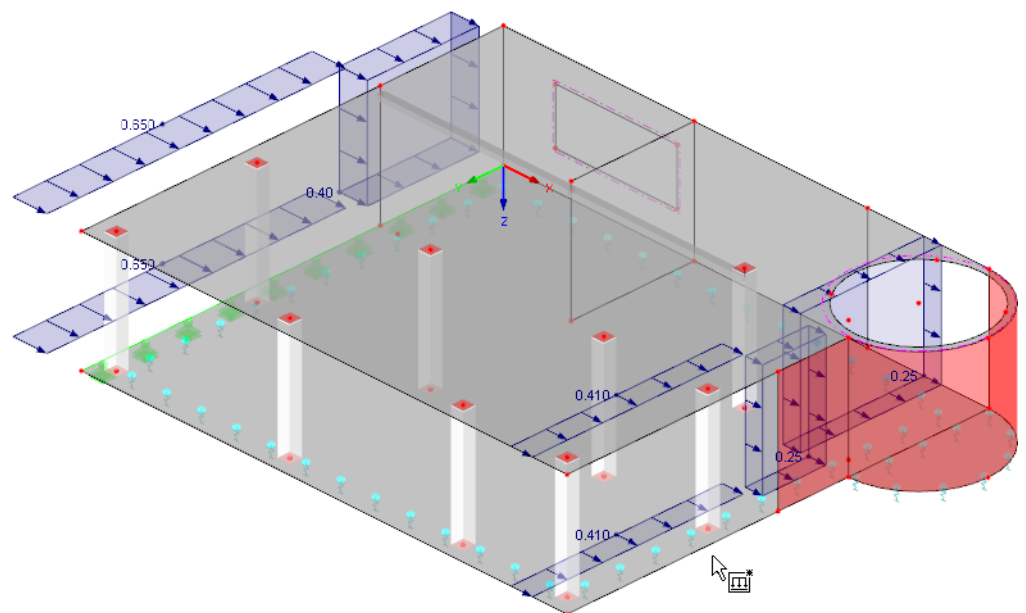
In the **New Line Load** dialog box, we enter the *Load Parameters* **0.41 kN/m** (see figure below). The *Load Direction* **XL** is preset.





Dialog box *New Line Load*

We set the load by clicking lines **3** and **11**.



Setting line loads

2.4.4 LC 4: Wind in +Y

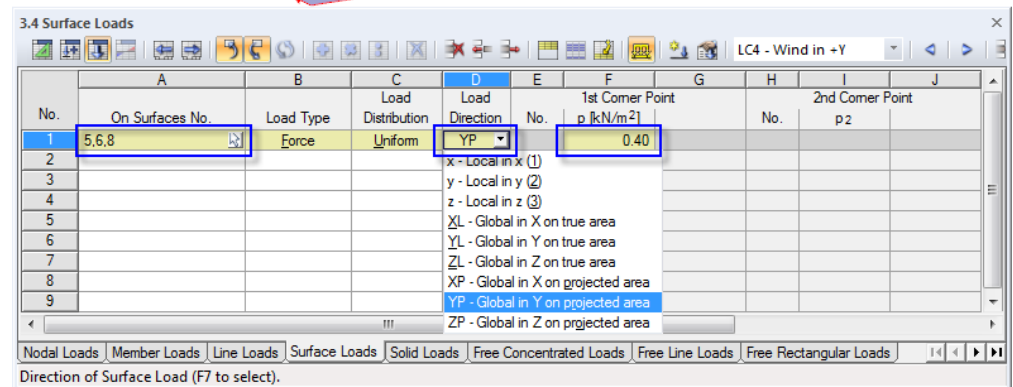
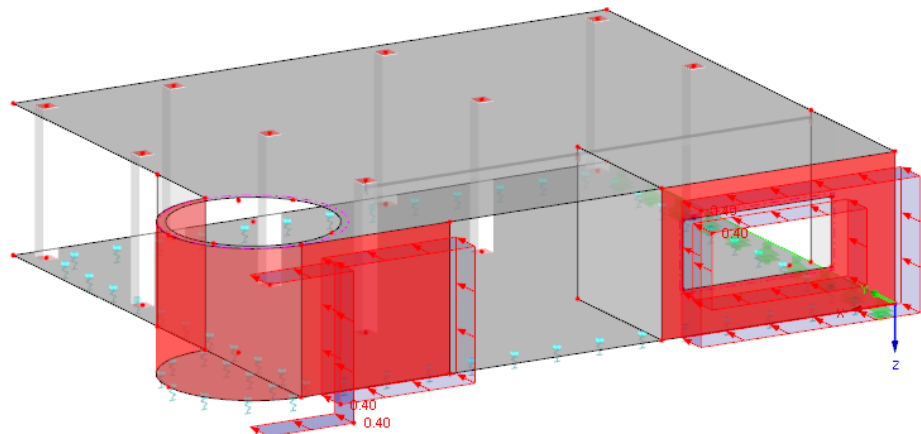
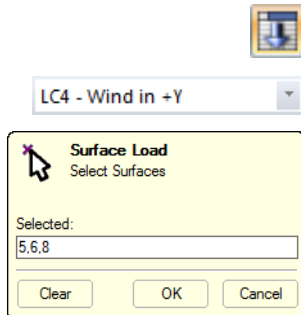
The wind loads of axis 6 and 1 can be defined analogously. Instead, we will use the tables now to enter the load data.

Surface loads

We go to table **3.4 Surface Loads**.

In the table toolbar, we set LC4 in the list.

By clicking in cell A1, the button [F7] becomes available. We use it to select surfaces **5, 6** and **8** in the work window.



Defining loads in table **3.4 Surface Loads**

Then we select the *Load Direction* **YP** from the list and enter the load p **0.40** kN/m².

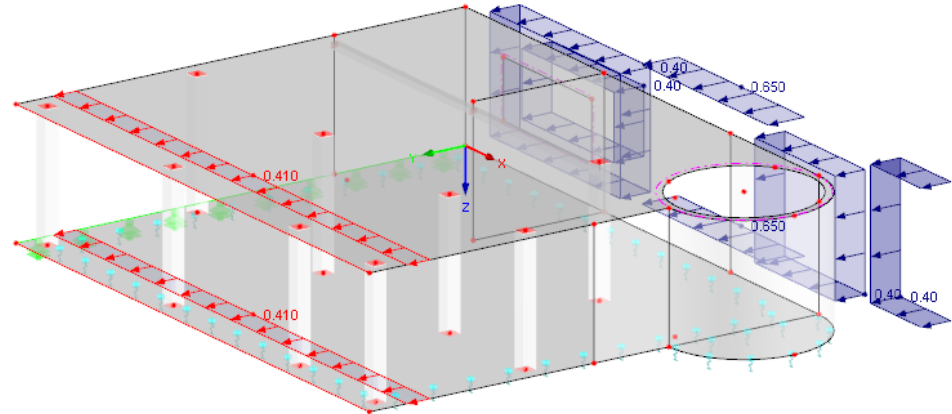
The load is immediately shown in the work window.

Line loads

We go to table **3.3 Line Loads**.

For the compression load, we again use the button [↖] to select lines **26** and **28** graphically. The *Load Direction* is **YL**, the load p **0.65** kN/m.

The line load on the suction side is to be set on lines **2** and **10**. The *Load Direction* is **YL** as well, but the load p is **0.41** kN/m.



3.3 Line Loads									
No.	A	B	C	D	E	F	G	H	I
	Reference to	On Lines No.	Load Type	Load Distribution	Load Direction	p [kN/m]	Line Load Parameters		
1	Lines	26,28	Force	Uniform	YL	0.650	p2	A	B
2	Lines	2,10	Force	Uniform	YL	0.410			
3									
4									
5									
6									

Defining loads in table 3.3 *Line Loads*

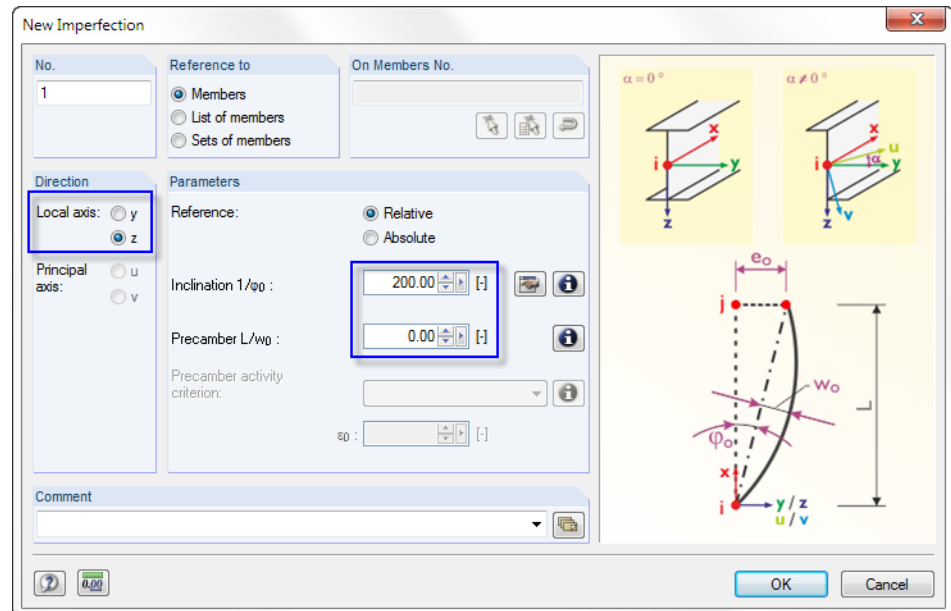
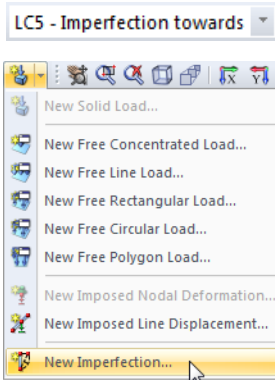
2.4.5 LC 5: Imperfection Towards +X

We set LC5 in the list.

Then we click the **[New Imperfection]** list button to open the dialog box **New Imperfection**.

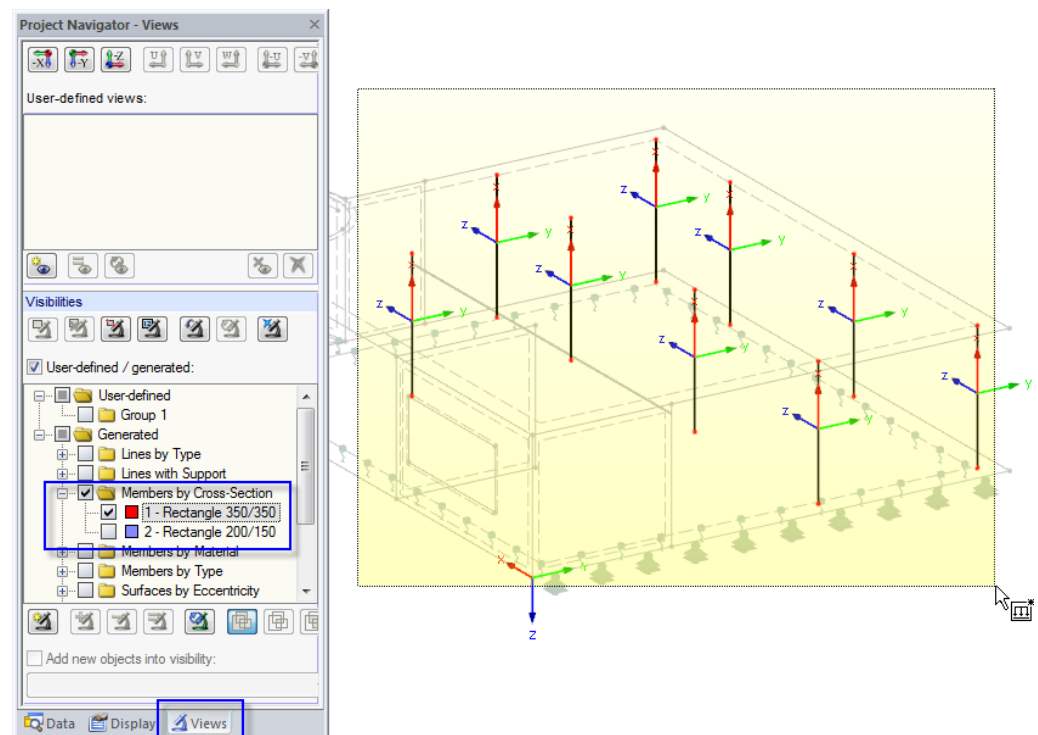
We make sure that the *Direction z* and the *Inclination $1/\varphi_0$ 200.00* are set.

According to EN 1992-1-1, we can reduce the *Precamber L/w_0* to **0.00** (see page 11).

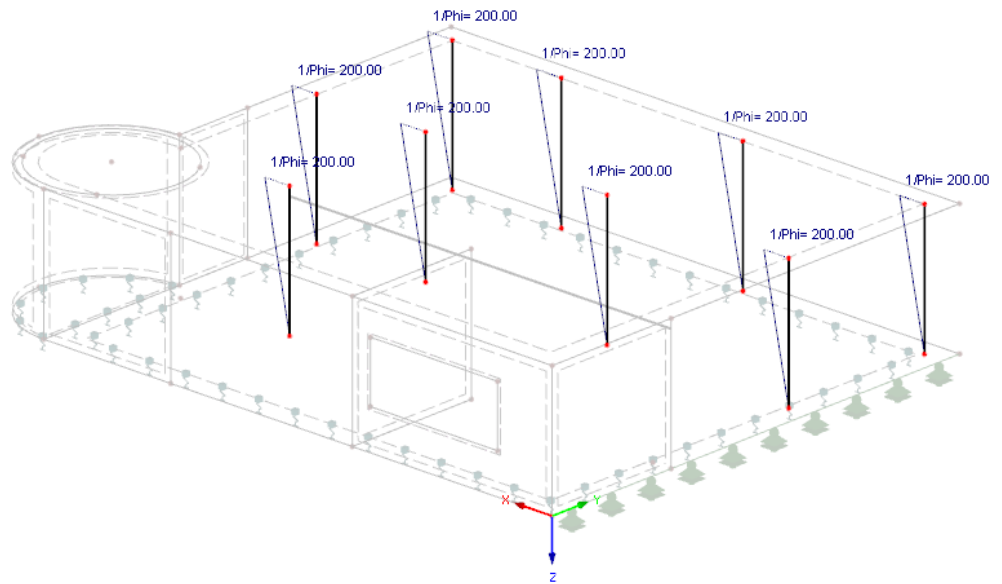


Dialog box *New Imperfection*

After [OK], we use the **Views Navigator** to display only the columns: Among the **Visibilities**, we check the option **Rectangle 350/350**. Then we draw a window around the entire model.



Activating columns in *Views navigator* and selecting all columns



Imperfections towards +X

2.4.6 LC 6: Imperfection Towards +Y

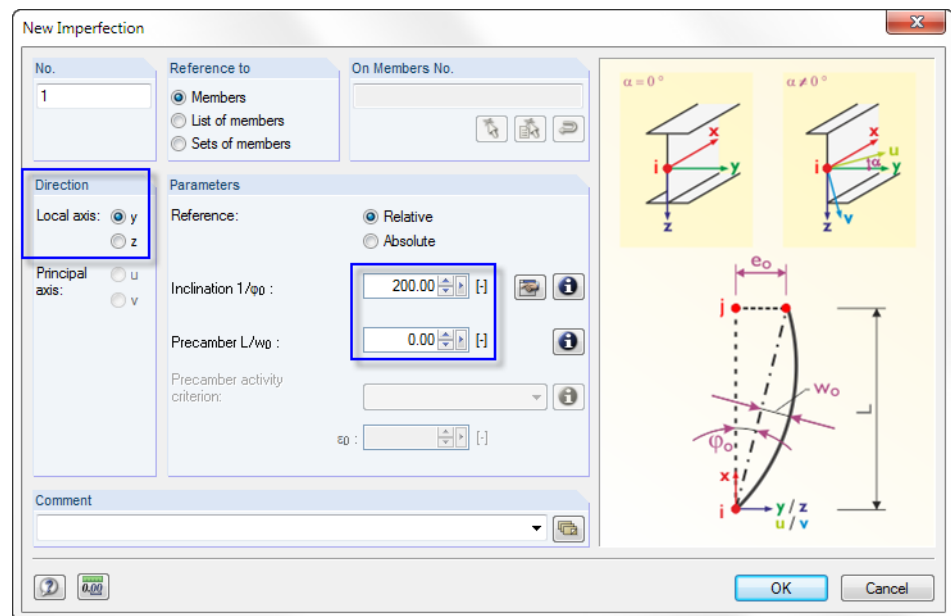
LC6 - Imperfection towards



Finally, we set LC6 in the list.

We click the **[New Imperfection]** list button again. In the dialog box **New Imperfection**, we make sure that the *Inclination* $1/\varphi_0$ **200.00** and the *Precamber* L/w_0 **0.00** are set.

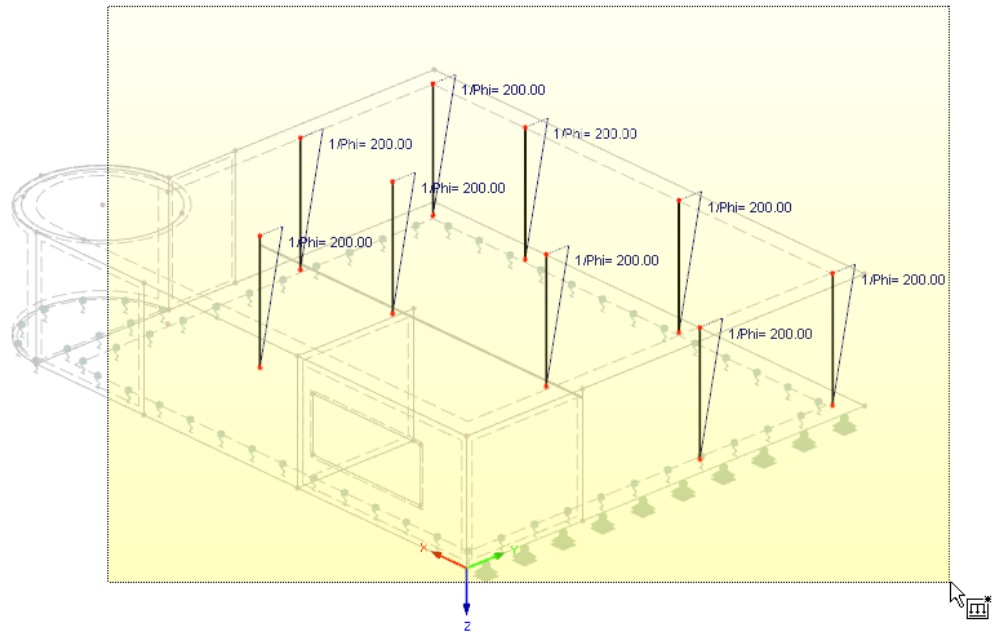
We change the *Direction* to the local axis **y**.



Dialog box *New Imperfection*

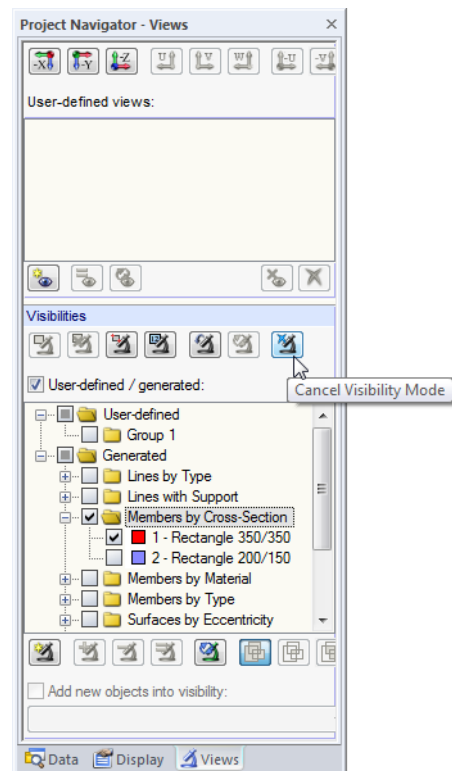


After [OK], we select all columns again by drawing a window around all columns.



Selecting all columns

To display the entire model again, we cancel the *Visibility Mode*.



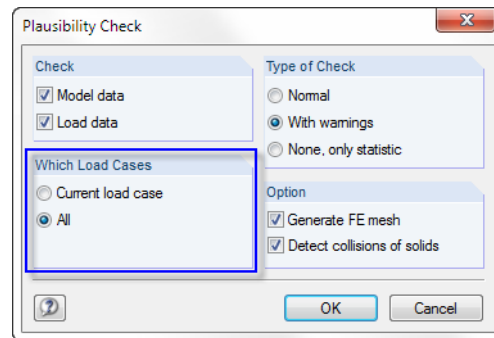
Cancelling *Visibility Mode* in Views navigator

2.5 Calculation

2.5.1 Plausibility Check



The plausibility check detects incomplete or missing entries of the input data. It will not reveal all input errors, especially instabilities resulting from release or support definitions.



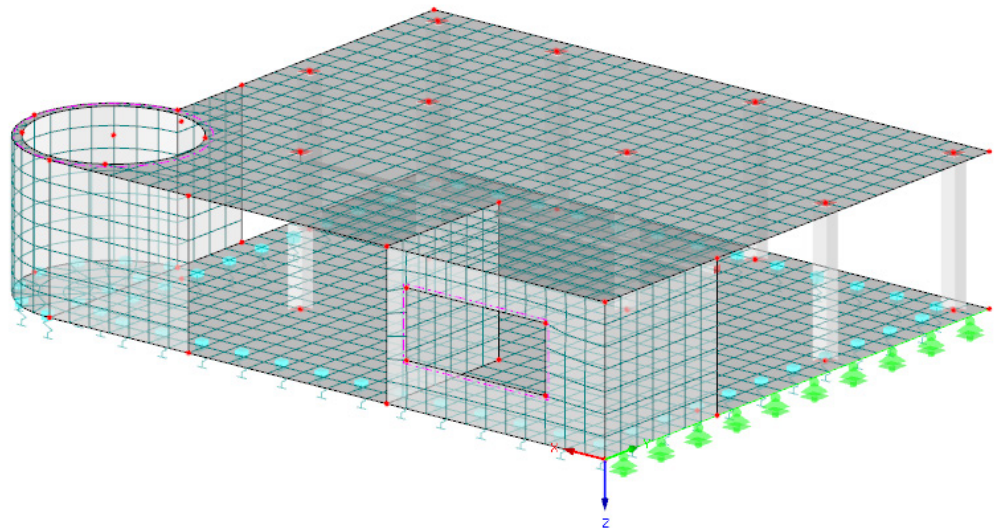
Dialog box *Plausibility Check*

We set **All** load cases for this check. If our input data is correct, RFEM detects no errors.

2.5.2 FE Mesh

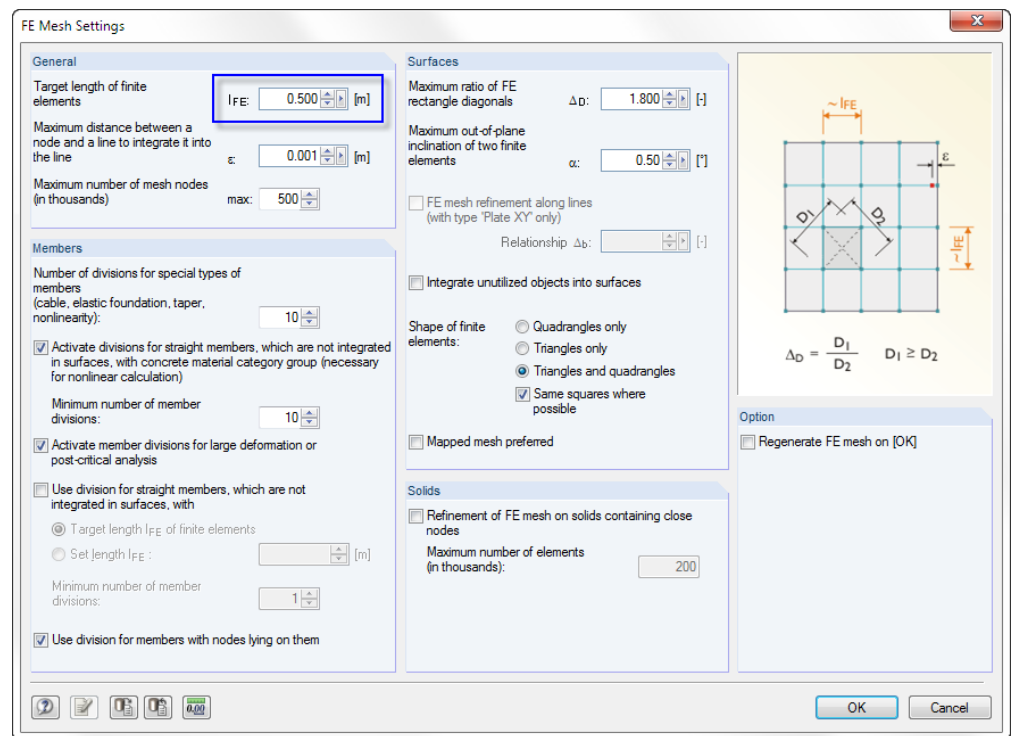
2.5.2.1 Generation

As the option **Generate FE mesh** was enabled in the dialog box *Plausibility Check* (see figure above), the FE mesh is generated automatically. RFEM applies a basic FE mesh width of 50 cm by default, which is convenient for our example.



Model with FE mesh

The settings of the mesh generator can be checked via menu **Calculate** → **FE Mesh Settings**.



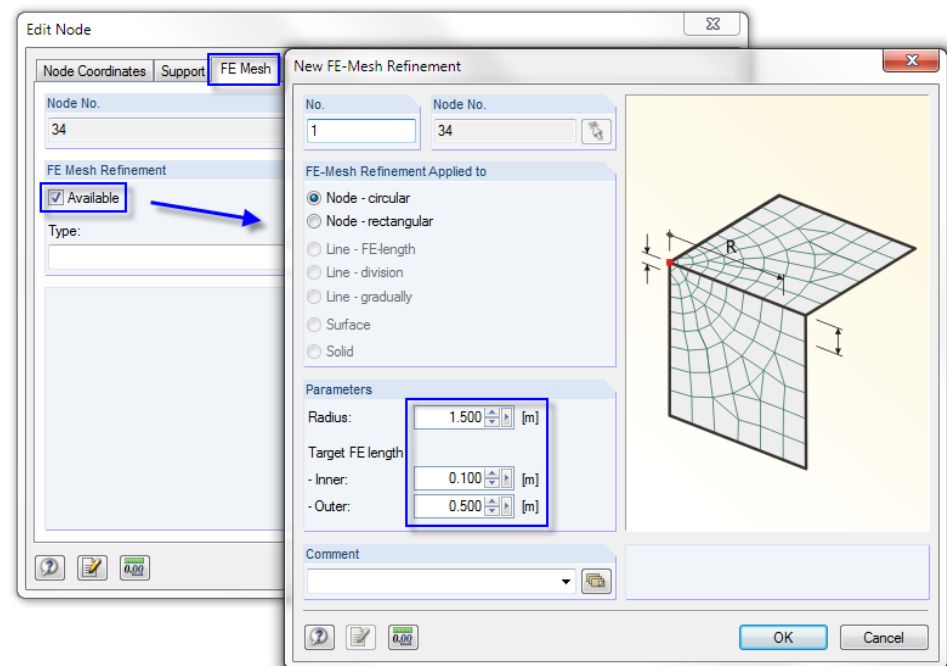
Dialog box *FE Mesh Settings*

The *target length* is globally applied to all lines, members, surfaces and solids.

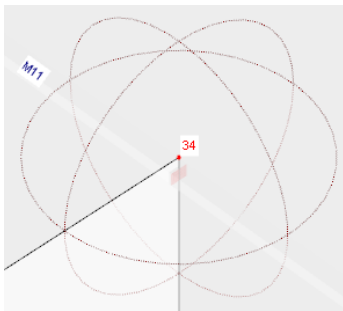
2.5.2.2 Mesh Refinements

Sometimes the FE mesh should be modified in specific parts of the model. Those are critical areas such as corners of openings or areas with single loads. We can use mesh refinements to define different lengths of the finite elements for particular areas or objects of the model.

To create a mesh refinement at the upper end of the inner wall (C/5), we double-click node **34**. In the *Edit Node* dialog box, we select the **FE Mesh** tab and then activate the **Available** option.



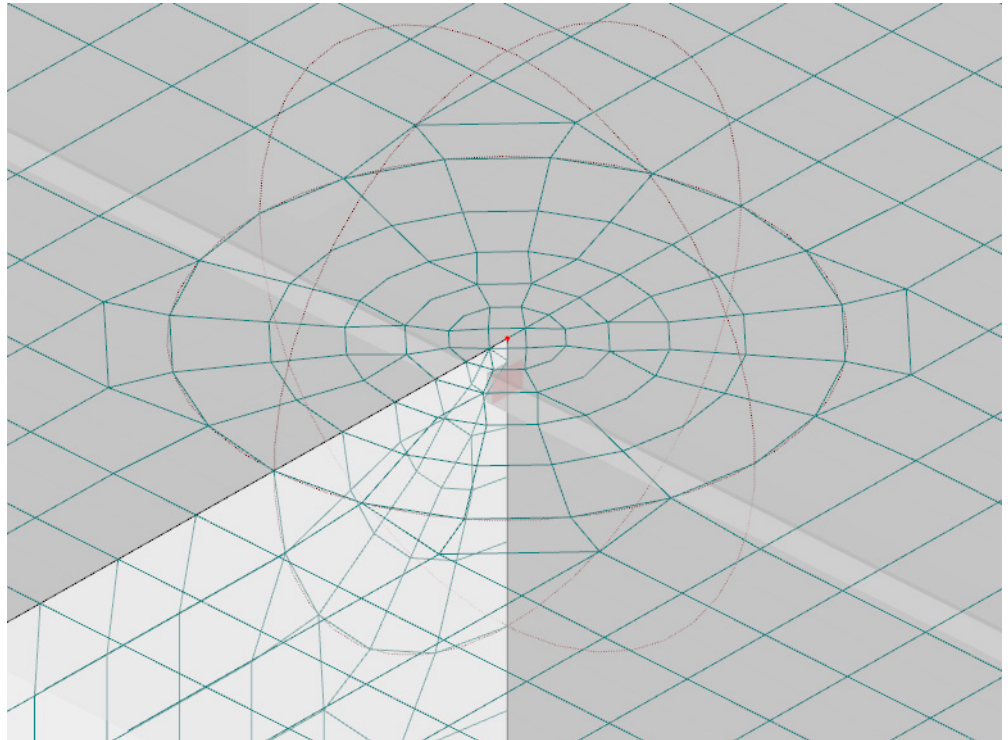
Dialog box *New FE-Mesh Refinement*



We define a mesh refinement within a radius of **1.50 m** of the node. The *Target FE length* at the center (*Inner*) is **0.10 m**, at the boundary line of the refinement (*Outer*) **0.50 m**.

After [OK], a sphere is shown around node 34 which symbolizes the refinement area. It affects all lines and surfaces within the sphere, thus including the ceiling and the wall.

To generate the mesh with the new settings, we select menu **Calculate** → **Generate FE Mesh**.



FE mesh with refinement

2.5.3 Calculation Parameters



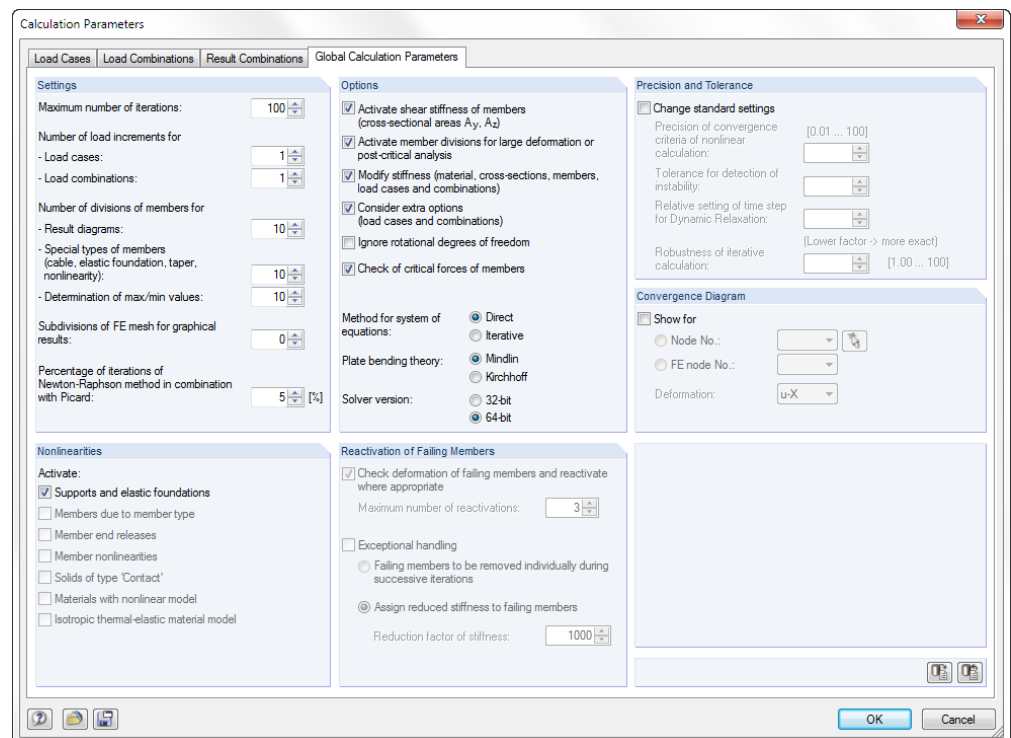
We now check the settings of the calculation by clicking the button [**Calculation Parameters**].

The **Calculation Parameters** dialog box opens. It consists of four tabs.

The *Global Calculation Parameters* tab controls all settings that apply to load cases, load and result combinations. The maximum number of iterations is preset to 100.

Due to the possible failure of the surface support, the option *Supports and elastic foundations* is activated. If there are problems in the calculation, we could deactivate the nonlinearities to find out if those are responsible.

We do not change the default settings in this tab.



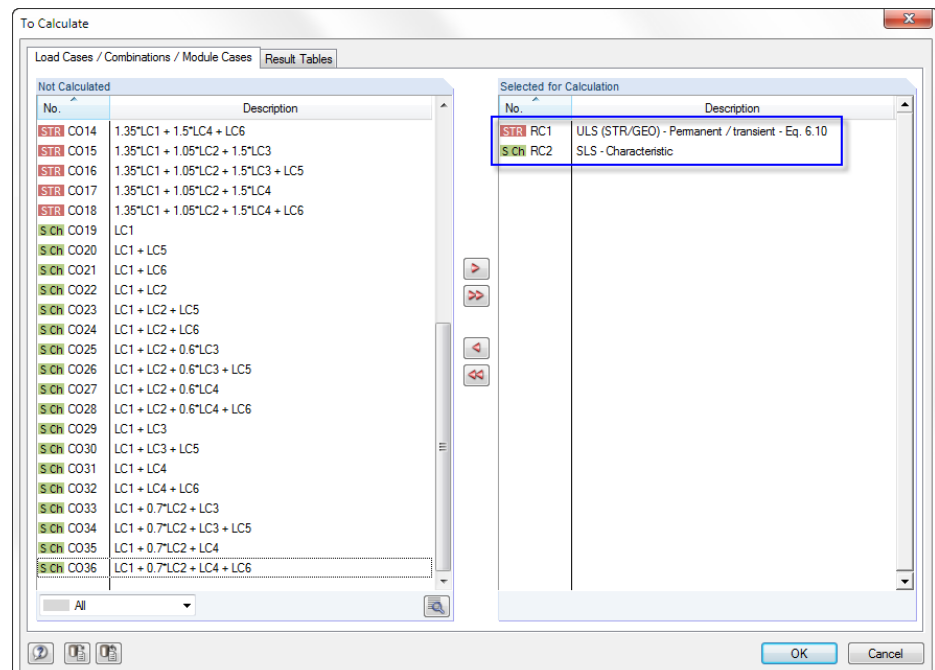
Dialog box *Calculation Parameters*, tab *Global Calculation Parameters*

We can check on the settings in the other tabs. In the **Load Cases** tab, the *Method of Analysis* is set to linear static analysis for all load cases. For **Load Combinations**, the default setting is the second-order analysis (P-delta). In the **Result Combinations** tab, the quadratic combination is disabled by default because it is only relevant for earthquake analyses.

2.5.4 Calculation

To select the load cases for the calculation, we apply menu **Calculate** → **To Calculate**.

We choose **RC1** and **RC2** by double-clicking them or via [►]. Then we start the calculation.

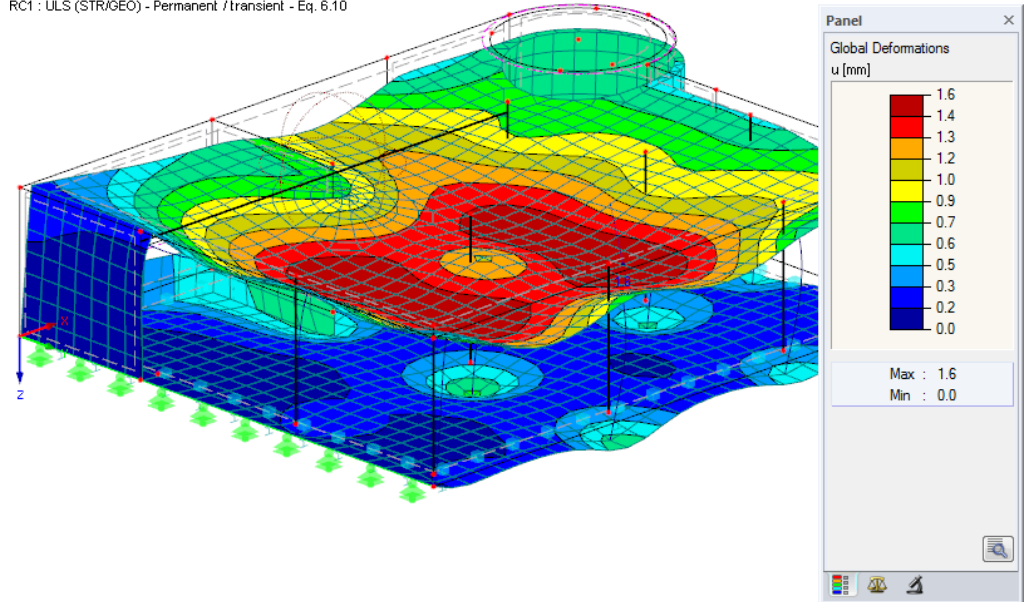


Dialog box *To Calculate*

2.6 Results

After the calculation, the deformations of the model are shown. We can adjust the display of the various types of results to evaluate the analysis.

Global Deformations u [mm]
RC1 : ULS (STR/GEO) - Permanent /transient - Eq. 6.10



Maximum deformations of RC1

2.6.1 Results Output

The results can be reviewed in the graphic window and in the tables. As with the loads, the results of each load case and combination are controlled by the list box in the toolbar.

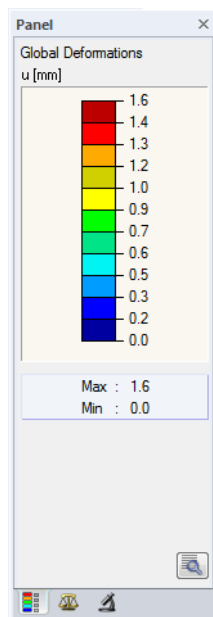
Work window

The **Results Navigator** controls the types of results that are displayed in the graphics.

In the **Panel**, the settings of the colors, scaling factors and filters can be adjusted.

Results tables

The numerical results of the nodes, supports, surfaces or members are listed in different tables.



4.2 Nodes - Deformations

Node No.		Displacements [mm]			Rotations [mrad]		
		uX	uY	uZ	φX	φY	φZ
1	Max	0.0	0.0	0.0	0.0	+0.1	0.0
	Min	0.0	0.0	0.0	0.0	-0.1	0.0
2	Max	0.0	0.0	0.0	0.0	+0.3	0.0
	Min	0.0	0.0	0.0	0.0	-0.4	0.0
3	Max	0.0	0.0	0.5	0.1	+0.1	0.0
	Min	0.0	0.0	0.4	0.1	+0.1	0.0
4	Max	0.1	0.1	0.6	-0.1	0.0	0.0
	Min	0.0	0.0	0.4	-0.2	+0.1	0.0
5	Max	0.0	0.0	0.2	0.1	+0.1	0.0
	Min	0.0	0.0	0.1	0.1	-0.1	0.0

Results - Summary | Nodes - Deformations | Lines - Support Forces | Members - Local Deformations | Members - Global Deformations

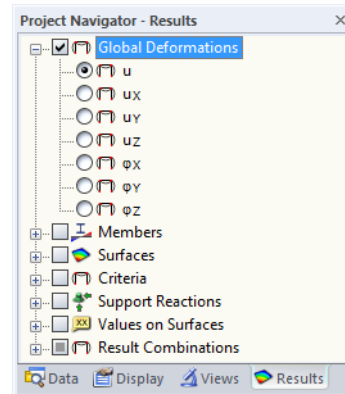
Table 4.2 Nodes - Deformations of RC1 (maximum/minimum results)

By default, the selection of objects is synchronized between tables and work window.

2.6.2 Deformations

2.6.2.1 Surfaces

The **Global Deformations** of all surfaces and members are set by default.

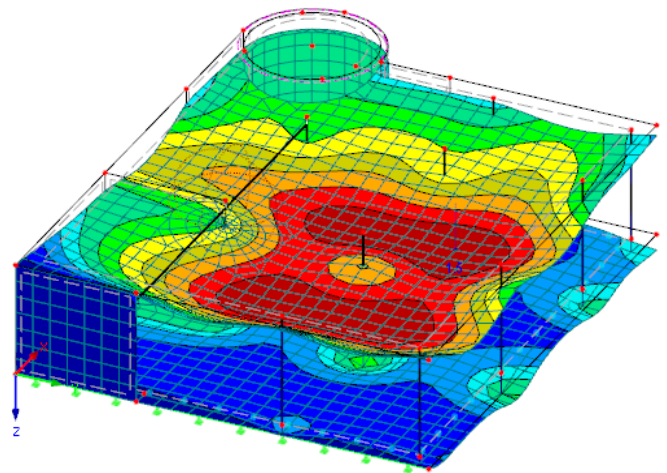
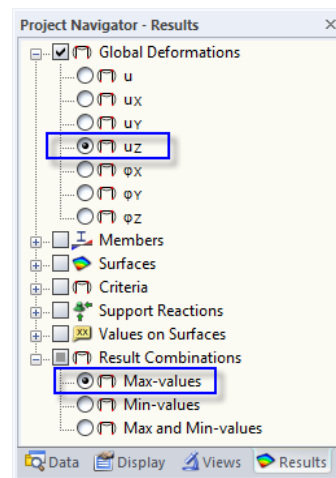


Setting *Global Deformations* in *Results* navigator

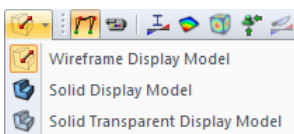
We can also display the components of the deformations and rotations with reference to the global axes X, Y and Z.

RC1 - ULS (STR/GEO) - Permanent /

We make sure the results of **RC1** are set. For a result combination, the extreme values can be displayed together or separately. We set the **Max-values** of the deformations **u_z**, for example.



Maximum deformations u_z of RC1

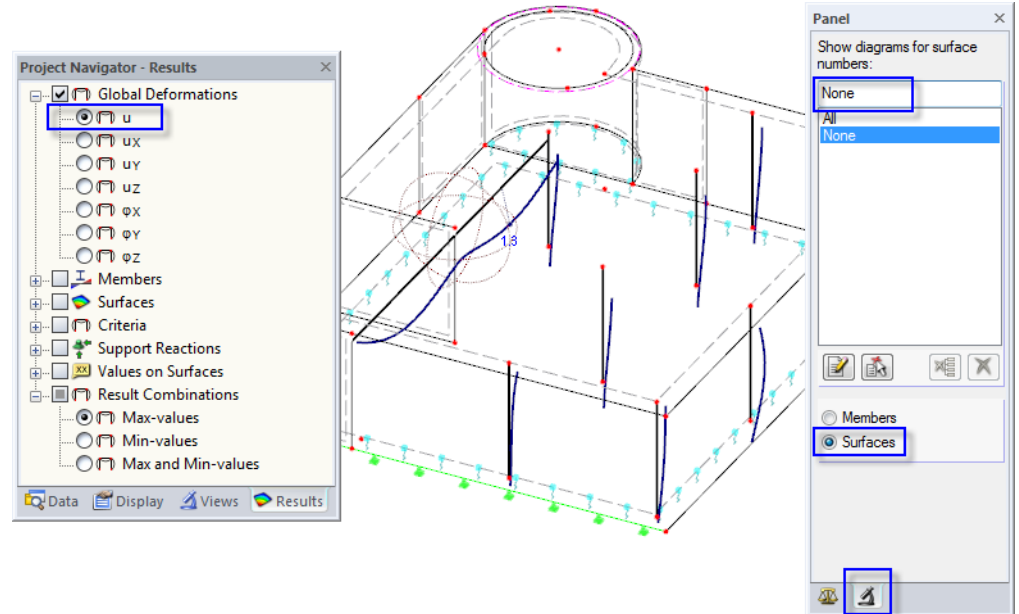


The *Wireframe* model renders an improved view of the results.

2.6.2.2 Members

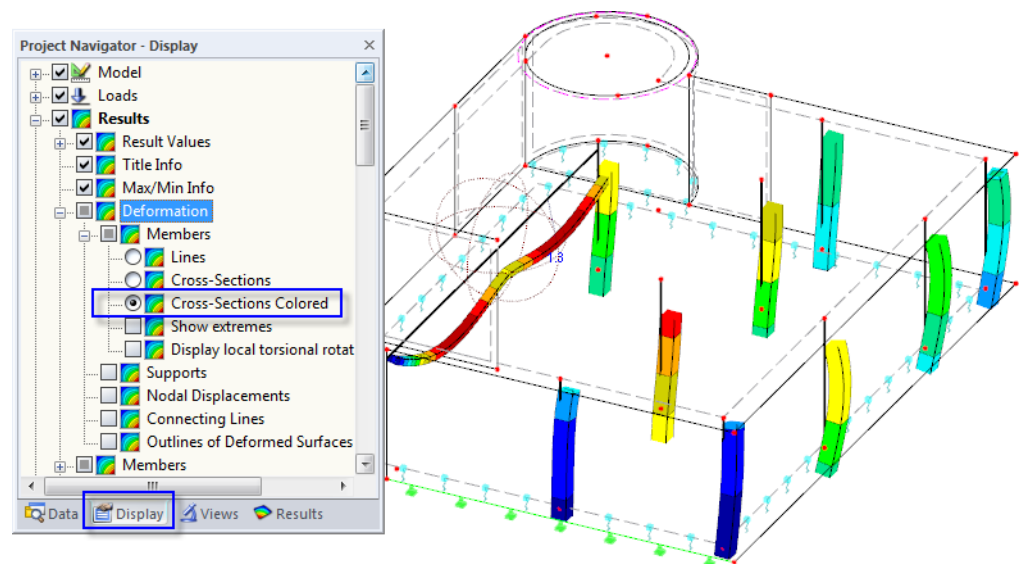
We display the *Global Deformations* **u** again.

In the *Filter* tab of the panel, we disable the surface results by setting **None**.



Maximum member deformations of RC1

The member results are shown as lines by default. We change those settings in the *Display navigator* by choosing **Cross-Sections Colored**.

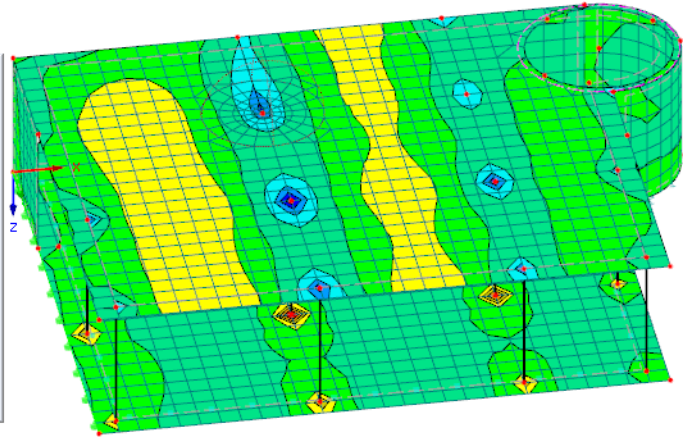
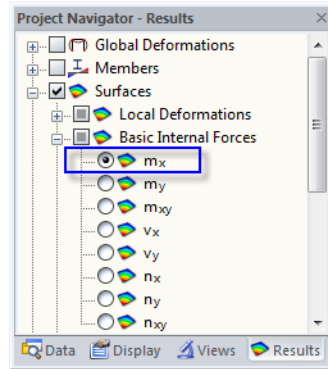
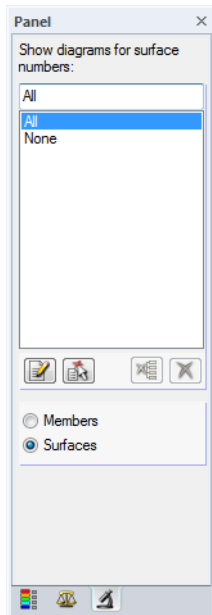


Colored display of cross-section deformations

2.6.3 Internal Forces

2.6.3.1 Selected Results of Surfaces

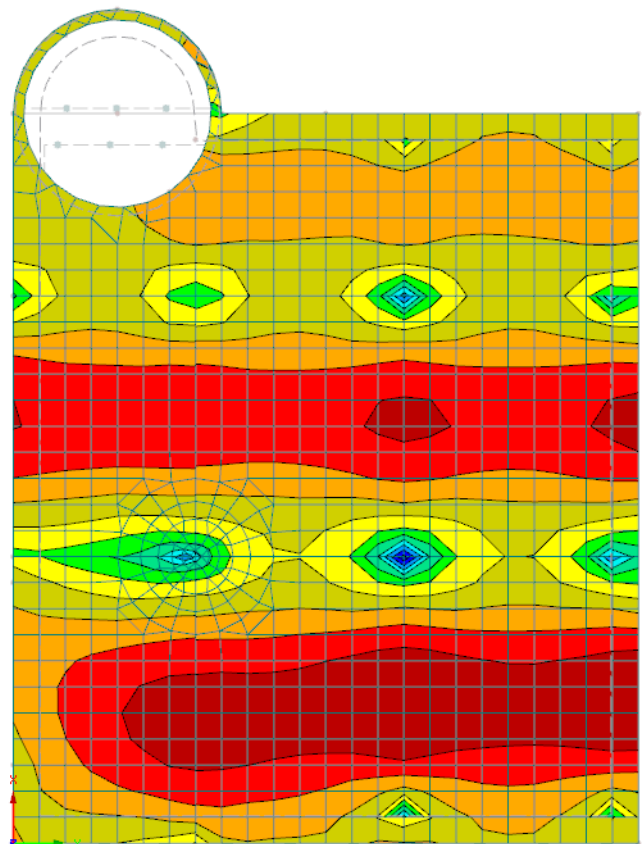
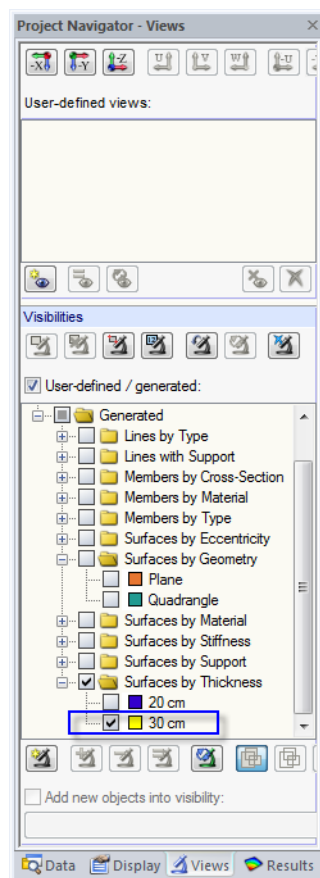
To display the bending moments of the surfaces, we set **All** surfaces again in the panel. In the **Results navigator**, we then activate the internal forces of the **Surfaces**, e.g. the moments m_x .



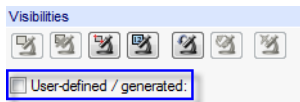
Basic internal forces m_x of RC1

Filter surfaces

To display only the moments of the ceiling slab, we can use the **Views navigator**. We activate the surfaces with **30 cm** thickness among the generated **Visibilities**.

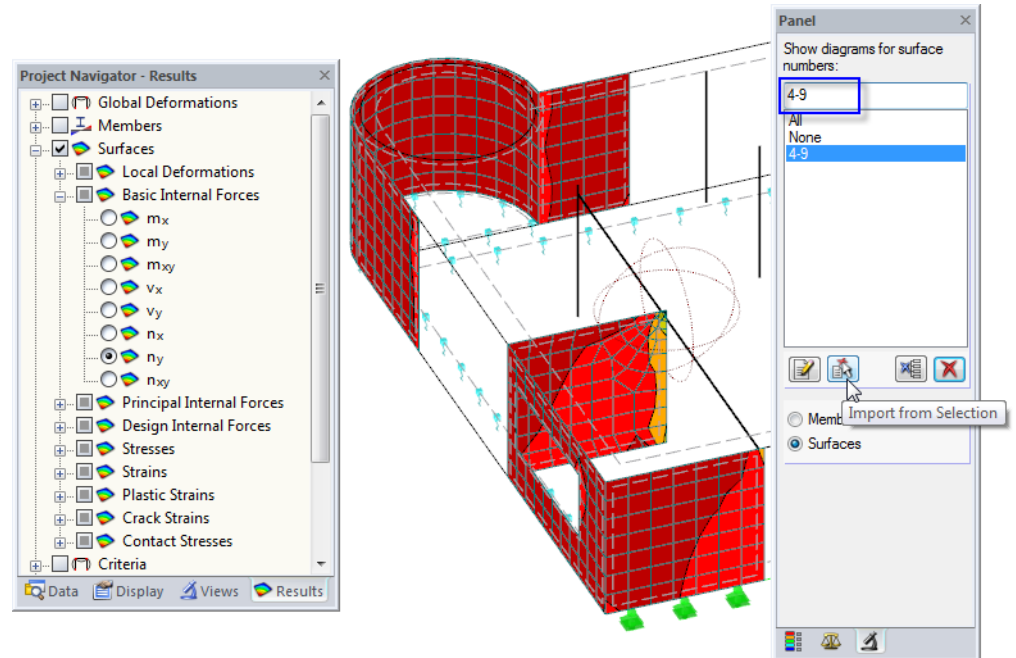


Activating visibility for moments of ceiling slab



We disable the visibility mode again. In the **Results navigator**, we set the internal forces n_y to display the axial forces. Then we draw a window that includes only the wall surfaces.

In the **Filter** tab of the panel, we then use the [^] button to set the wall surfaces in the list.

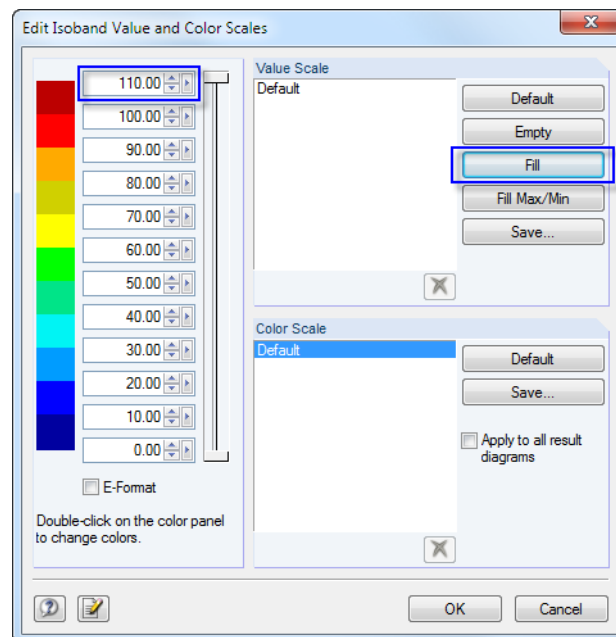


Basic internal forces n_y of walls (RC1)

Filter results

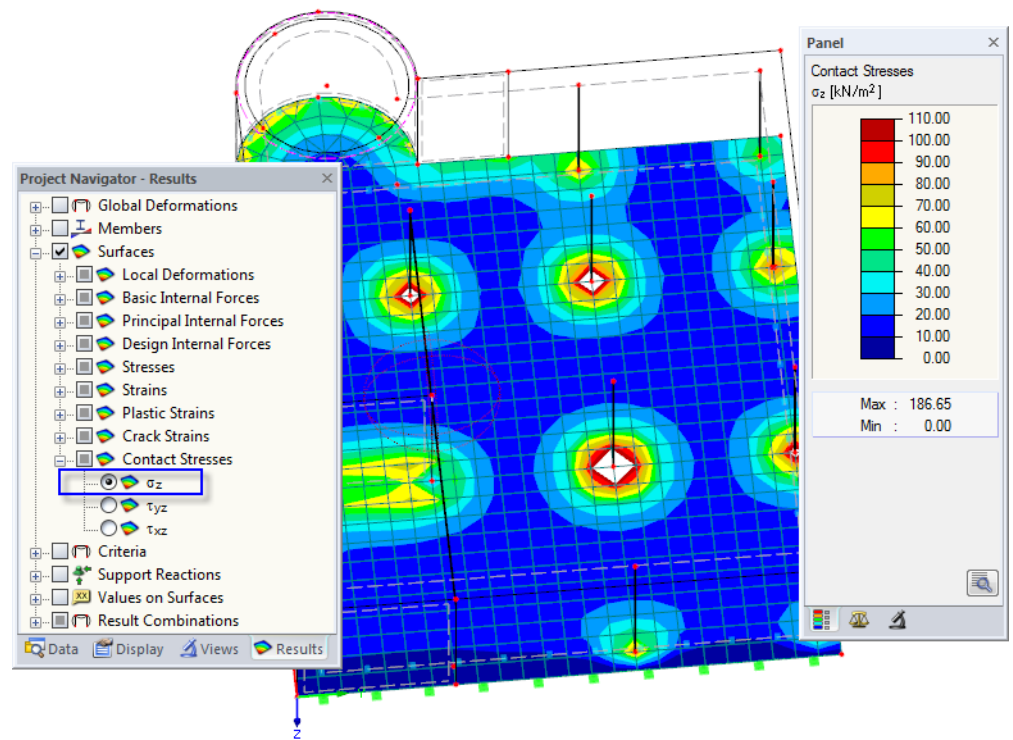
We reset **All** surfaces in the panel. In the **Results navigator**, we activate the contact stresses σ_x .

There are singularities at the column nodes that result in a slightly distorted view of the stresses. We can adjust the panel by double-clicking the color spectrum.



Dialog box *Edit Isoband Values and Color Scales*

We enter **110.00** as maximum stress and then [Fill] the intermediate values.

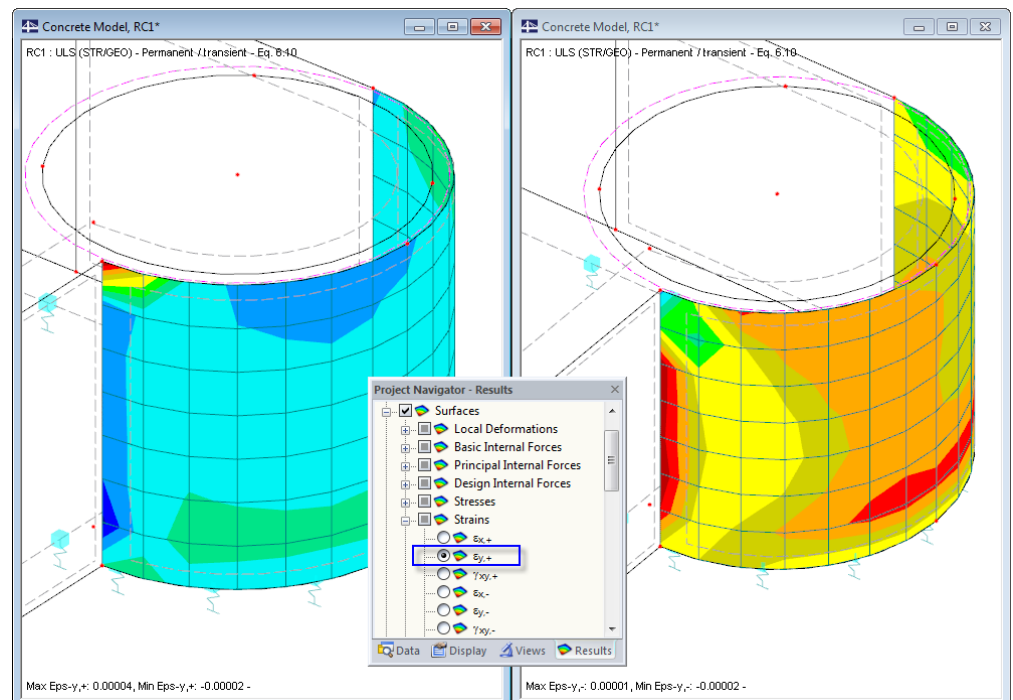


Contact stresses σ_z of floor slab (RC1)

Multi-window display

To use several windows with different views, we select **New Window** on the **Window** menu. Then we select **Tile Vertically** on the same menu.

In the following figure, the strains ϵ_y are displayed for both sides of the quadrangle surface.



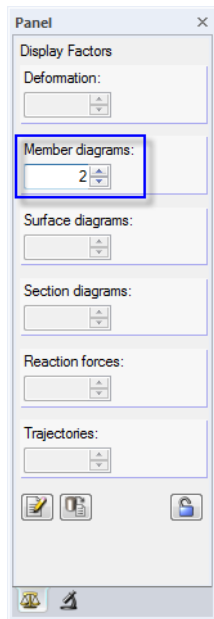
Strains $\epsilon_{y,+}$ and $\epsilon_{y,-}$ of quadrangle surface (RC1)

Then we **[Maximize]** one of the windows again.

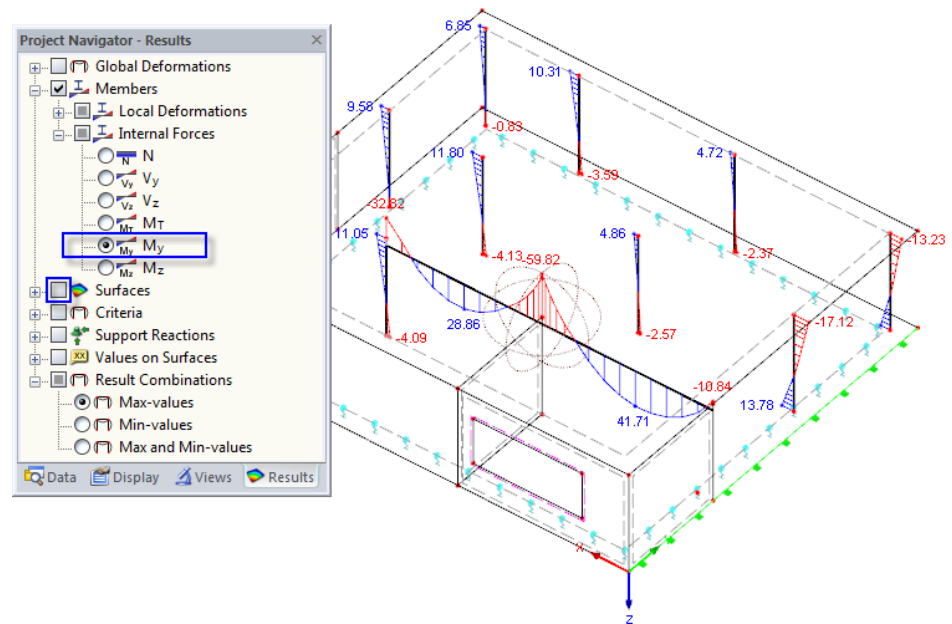


2.6.3.2 Specific Results of Members

To display the results of the members, we deactivate the surfaces in the **Results navigator**. Then we set the bending moments M_y .



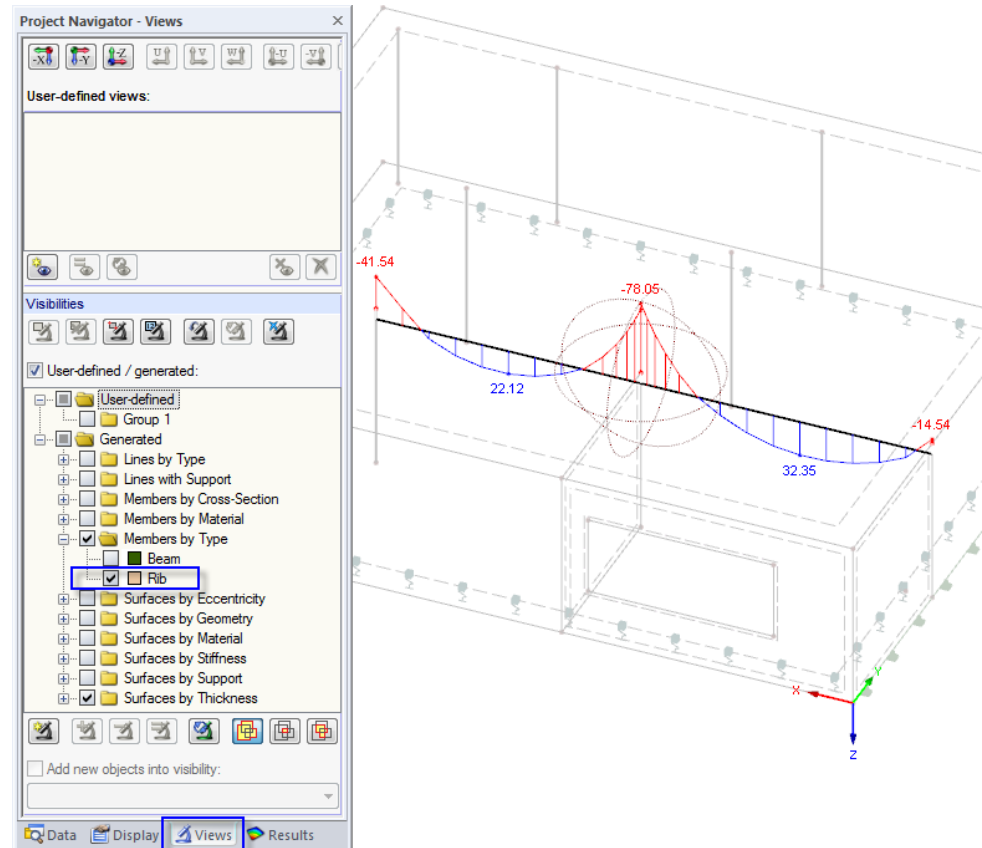
Panel tab **Factors**



Maximum moments M_y of columns and ribs (RC1)

In the **Factors** tab of the panel, we increase the display factor of the member diagrams.

We use the **Views navigator** to select the **Rib** member. When we set the **Min-values** in the **Results navigator**, the greatest negative moments of RC1 are displayed at the wall supports.

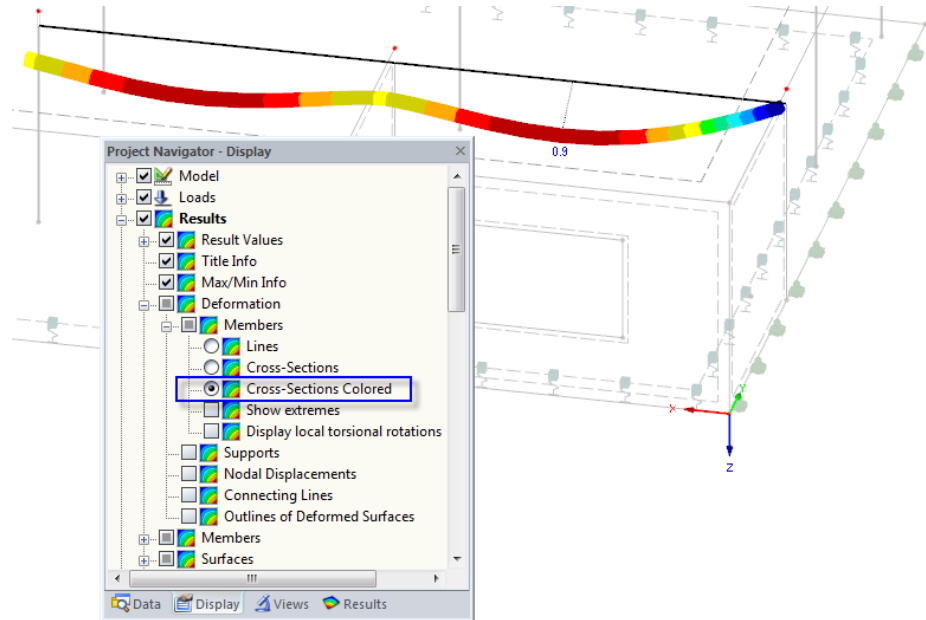


Minimum moments M_y of ribs (RC1)

Result display

We display the [Deformation] again.

In the **Display navigator**, we now set the option **Cross-Sections Colored**.

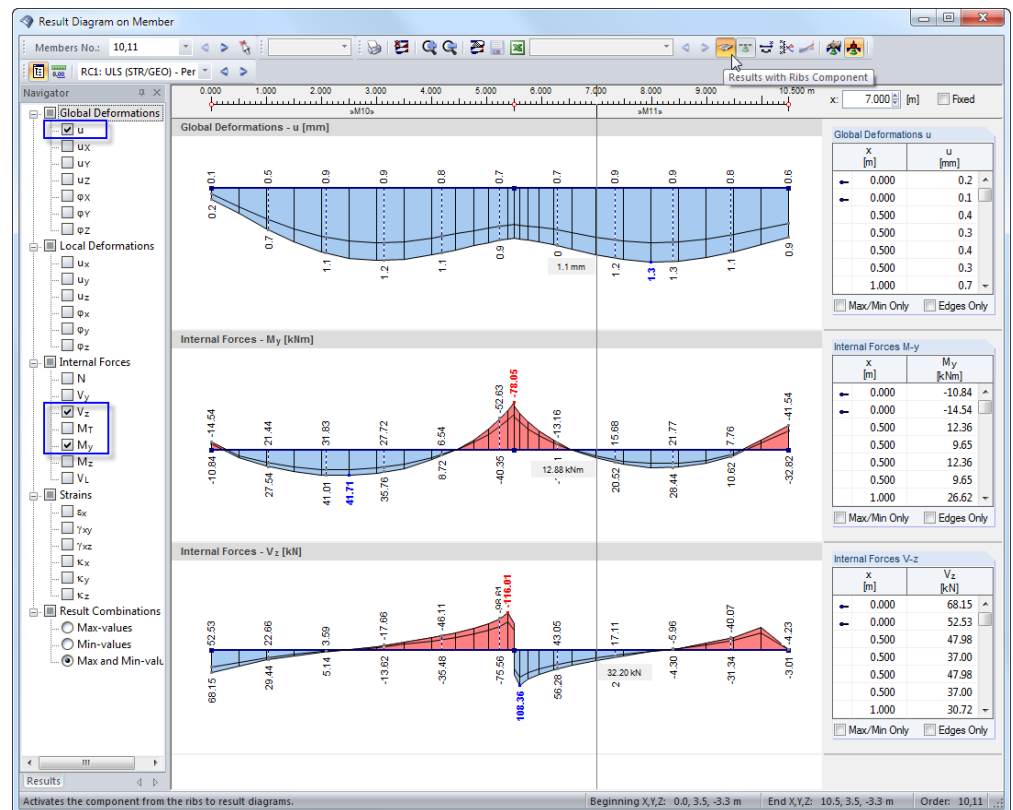


Minimum deformations of ribs (RC1)

Result diagrams

We select the two ribs. In the context menu, we click the option **Result Diagrams**.

In the dialog box **Result Diagram on Member**, we add **V_z** to display the shear forces of the ribs.



Dialog box **Result Diagram on Member**

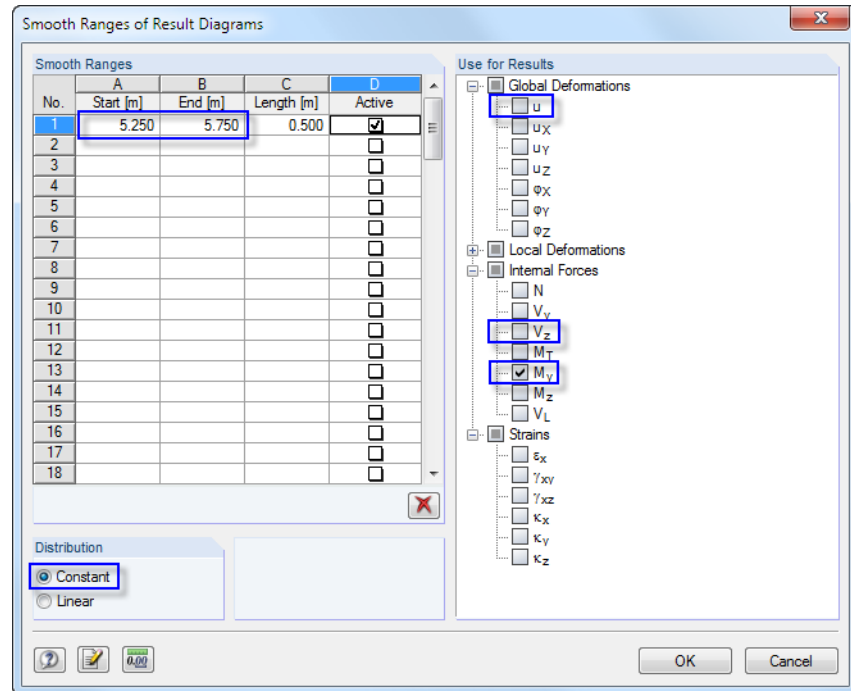
Smooth range



To view the moment redistribution effects at the wall support, we define a **[Smooth Range]**. Its length can be determined from the thicknesses of the wall and the slab as follows:

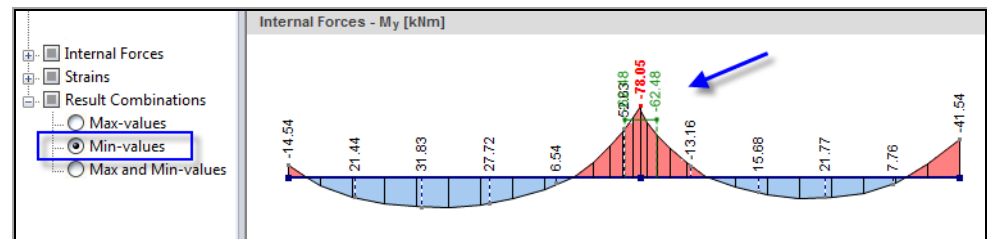
$$20 \text{ cm} + 2 \cdot 30 \text{ cm} / 2 = 50 \text{ cm}$$

In the dialog box *Smooth Ranges of Result Diagrams*, we enter **5.25 m** and **5.75 m**. Additionally, we apply the following settings.



Dialog box *Smooth Ranges of Result Diagrams*

We set the **Min-values** to display the redistributed negative moments of the smooth range.



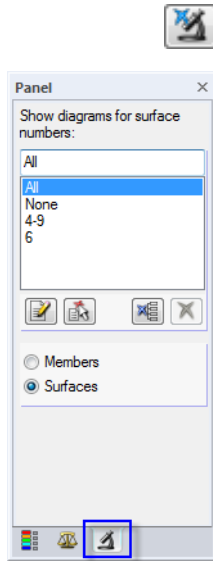
Result diagram with smooth range for moments M_y

2.6.4 Sections

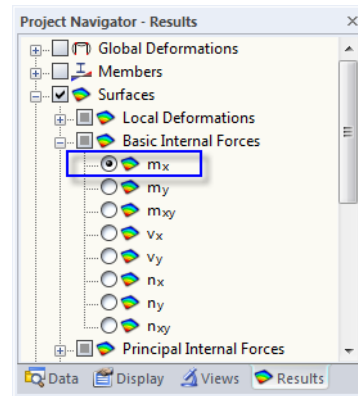
Before we create a section through the model, we [**Cancel**] the visibility mode.

We activate **All** surfaces again in the **Filter** tab of the panel.

In the **Results navigator**, we set the internal forces m_x of the surfaces.



Panel tab **Filter**

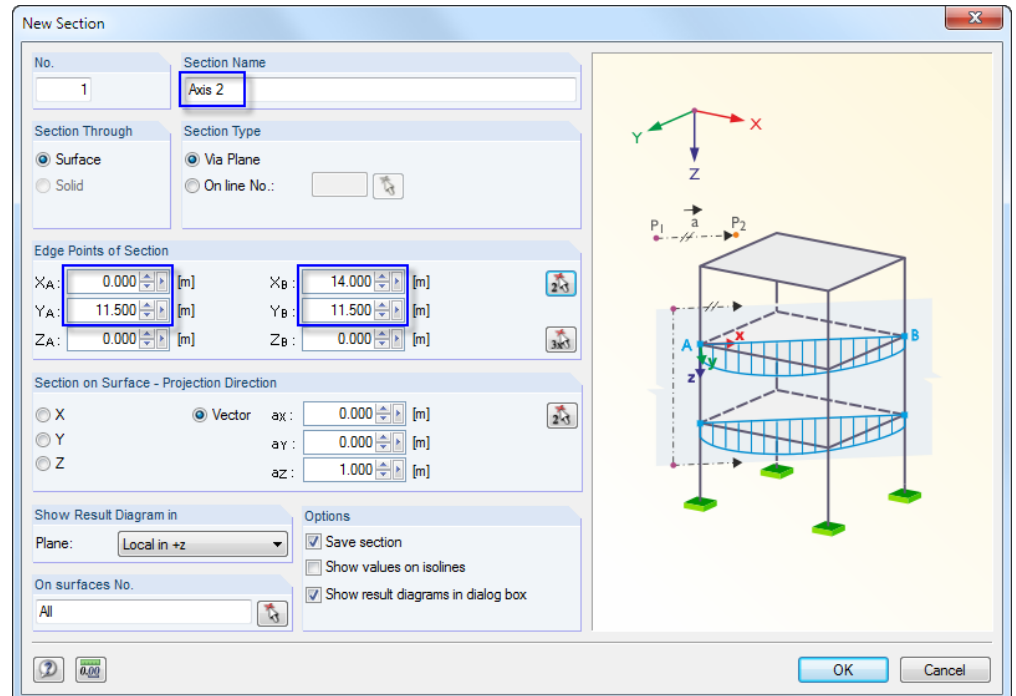
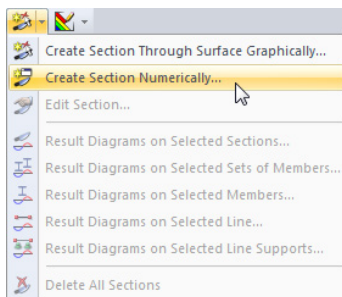


Setting basic internal forces m_x in **Results navigator**

Section in axis 2

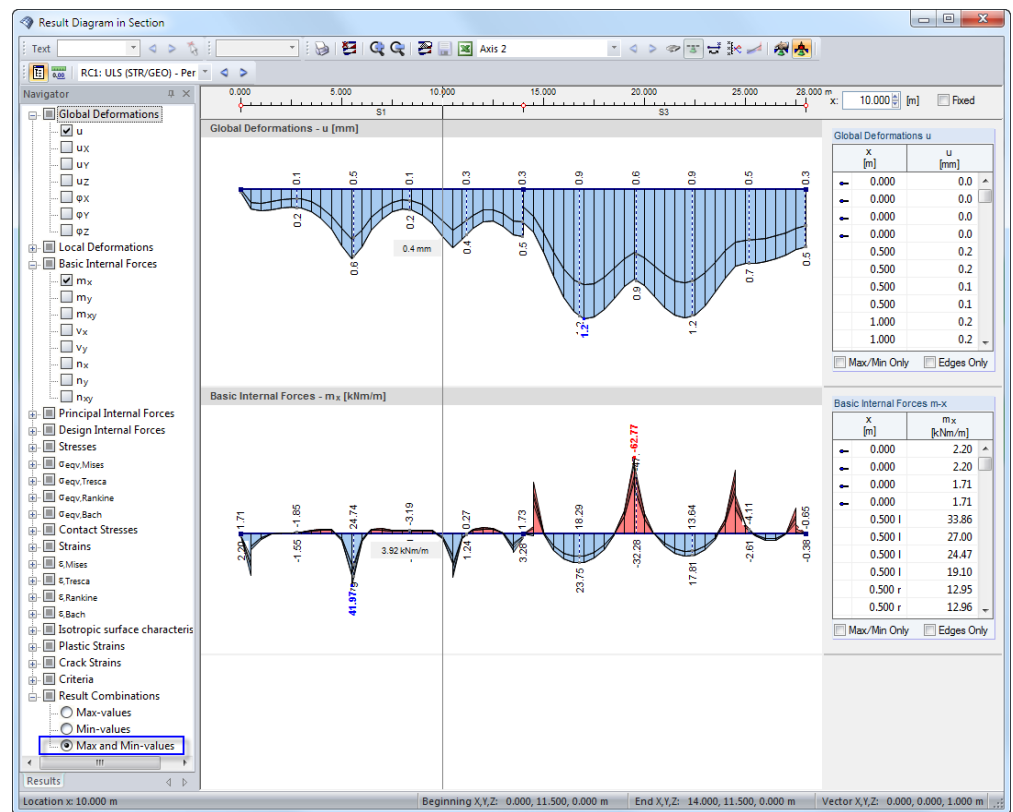
We use the [**Section**] list button to select the option **Create Section Numerically**.

In the **New Section** dialog box, we enter the name **Axis 2**. Then we define the edge points by entering the coordinates as seen in the following figure. We can also select the column nodes graphically via the button [$\square$] and modify the X-coordinates afterwards.



Dialog box **New Section**

The **Result Diagram in Section** window opens (see figure below). As we have chosen **All** surfaces in the **New Section** dialog box, the section plane cuts through the ceiling and the floor slabs, both diagrams are shown next to each other. This results in a total section length of 28 m.

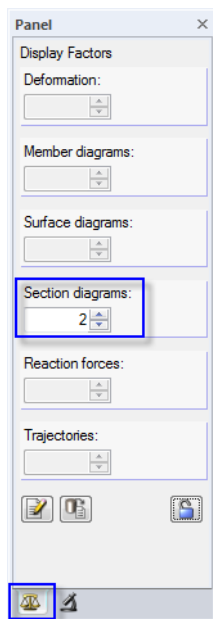


Dialog box *Result Diagram in Section*

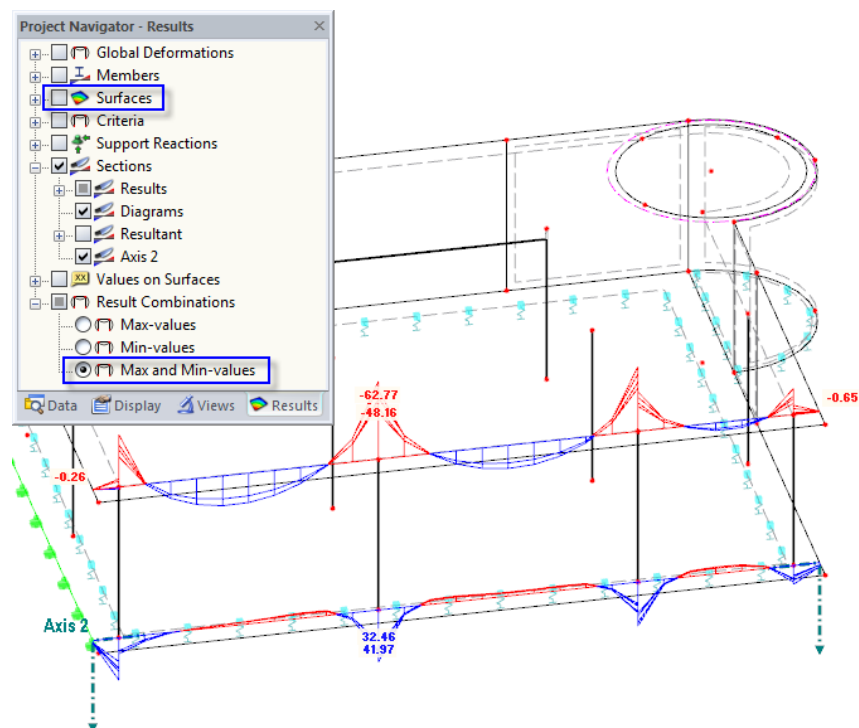
We activate the **Max and Min-values** of RC1.

When we close the window, the section diagrams are also shown in the surface results.

To improve the visibility, we turn off the surface results by clearing the **Surfaces** check box in the **Results navigator**. We can also increase the display factor of the diagrams in the panel.



Panel tab *Factors*

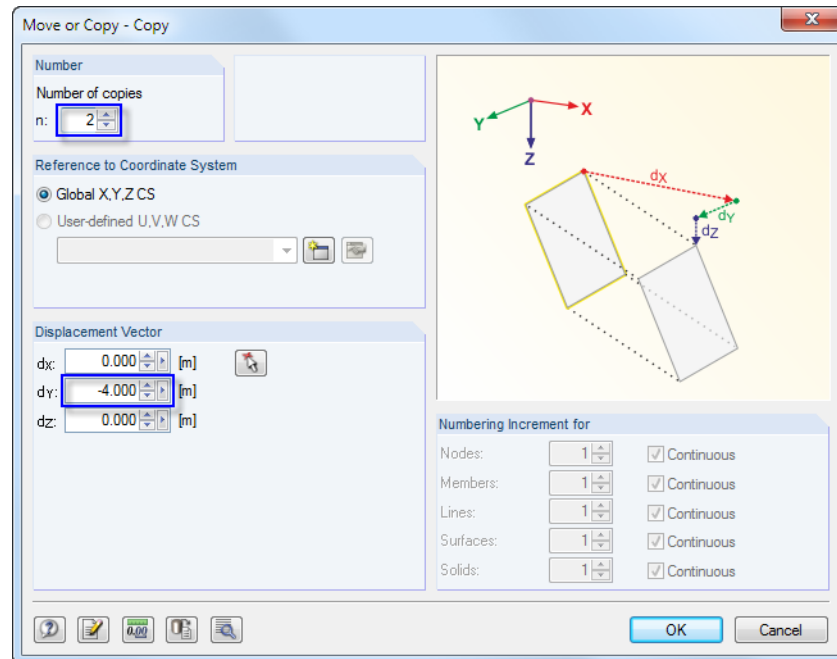
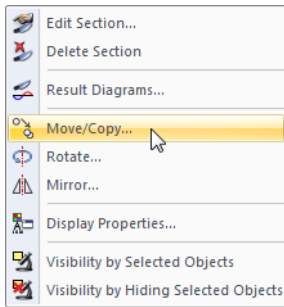


Section *Axis 2* displaying moments m_x

Sections in axes 3 and 5

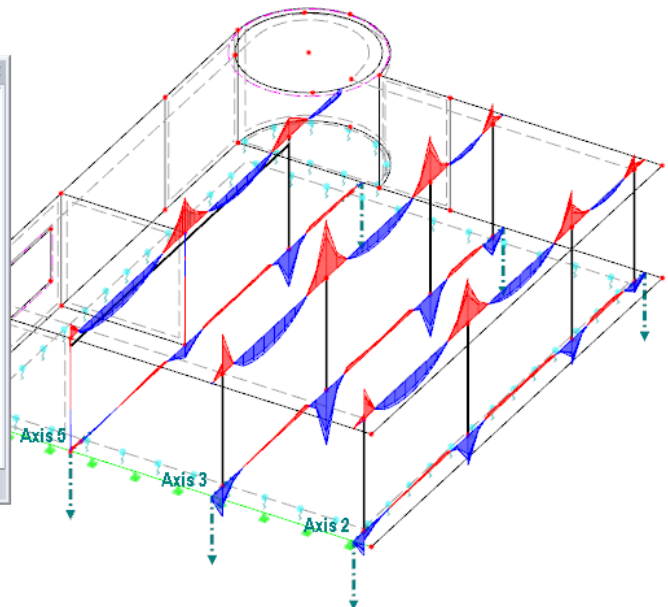
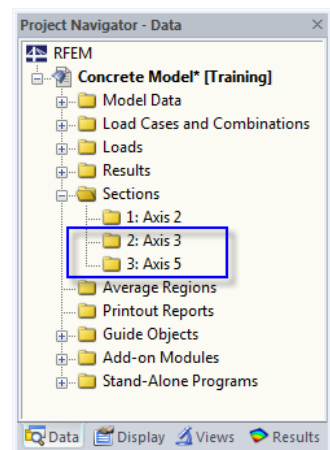
We now copy the section via its context menu.

In the dialog box *Move or Copy - Copy*, we define **2** copies and the displacement vector of **-4.0 m**.



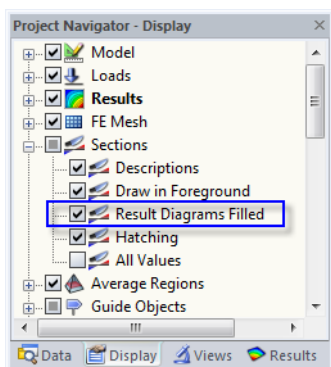
Dialog box *Move or Copy - Copy*

After [OK], we modify the names of the copied sections in the **Data navigator**.



Renamed sections with filled diagram display

We can activate the option **Result Diagrams Filled** in the **Display navigator** for a better view.



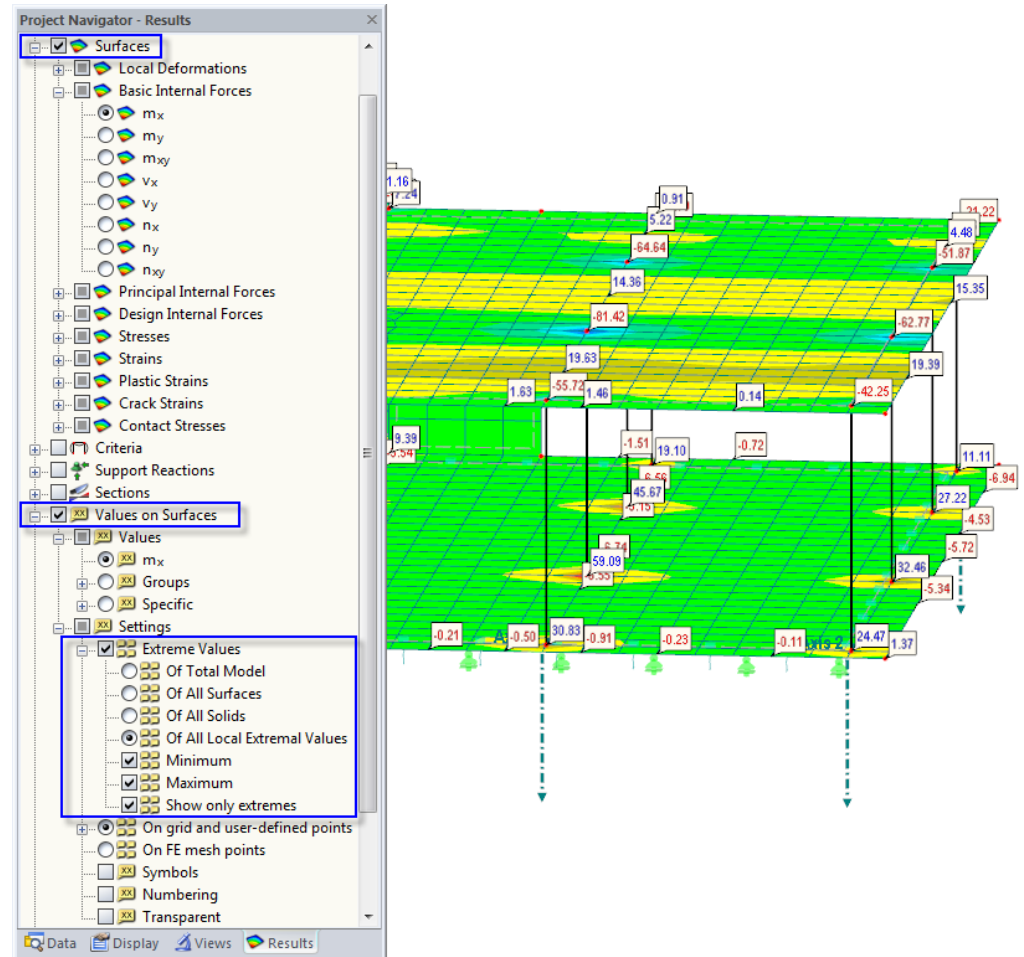
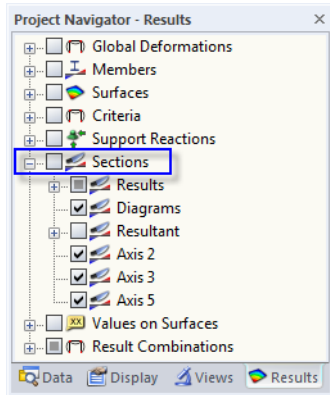
2.6.5 Numerical Results

2.6.5.1 Work Window

We turn off the **Sections** in the **Results** navigator. Then we display the surface results m_x .

Extreme values

When we activate the option **Values on Surfaces**, the **Extreme Values** are set by default.



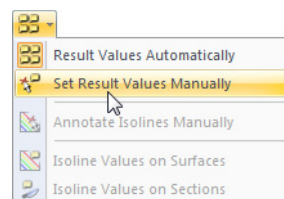
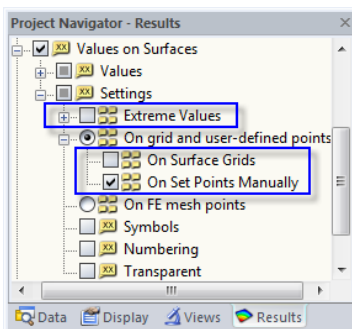
Extreme values of surface results m_x (RC1)

User-defined values

To add values manually in the ceiling slab, we set the **[Work Plane XY]** and make sure that the **[Origin]** lies in this plane.

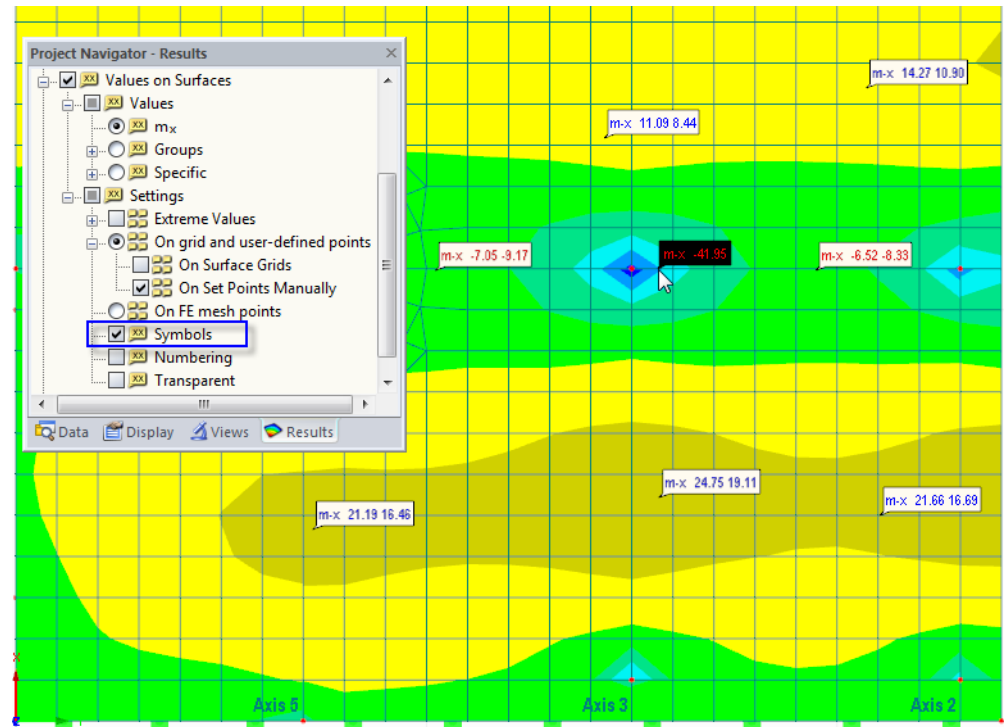
We disable the **Extreme Values**. Then we enable the option **On Set Points Manually** only.

Next we choose **Set Result Values Manually** in the list button **[Surface Result Values]**.



List button [Surface Result Values]

We can now set values by clicking specific points in the slab. For a result combination, both maximum and minimum values are shown.

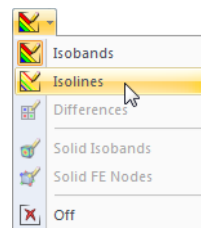


Setting user-defined values in ceiling slab (RC1)

With the **Symbols** option, the descriptions of the internal forces are displayed, too.

Isolines

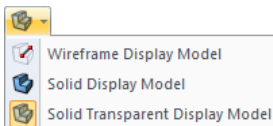
Next we select the **Isolines** in the list button [Display Options].

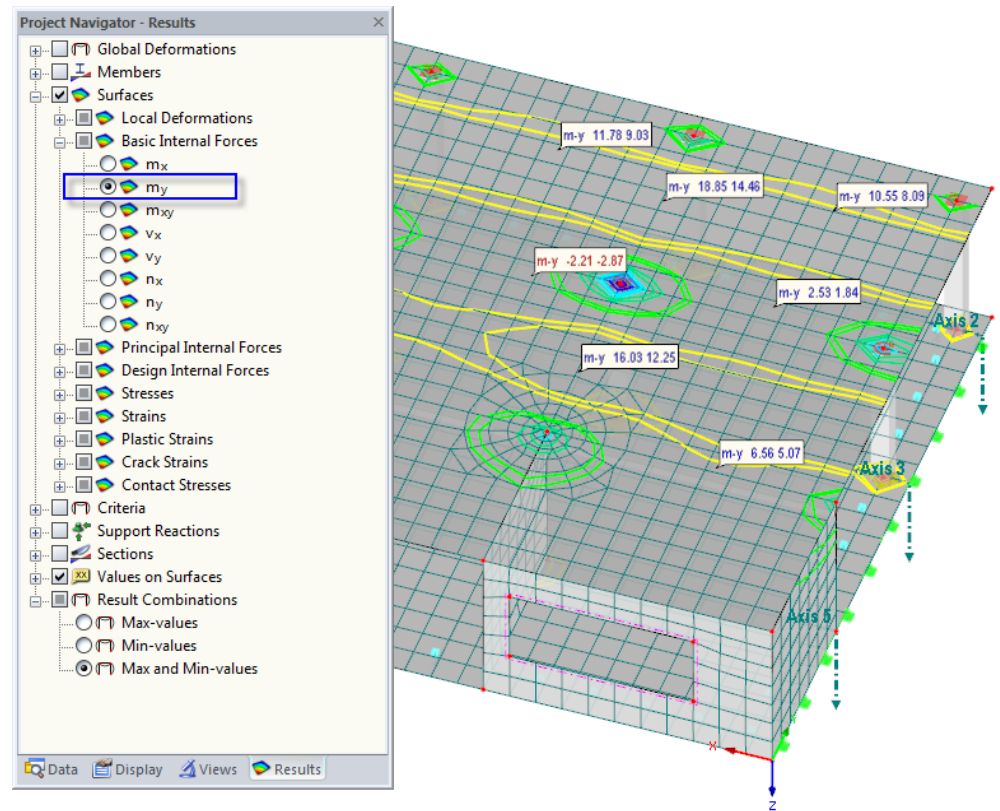


List button [Display Options]

In the **Results navigator**, we set the surface results **m_y**.

For a better view of the isolines, we choose the **[Solid Transparent Display Model]** (see figure on next page).

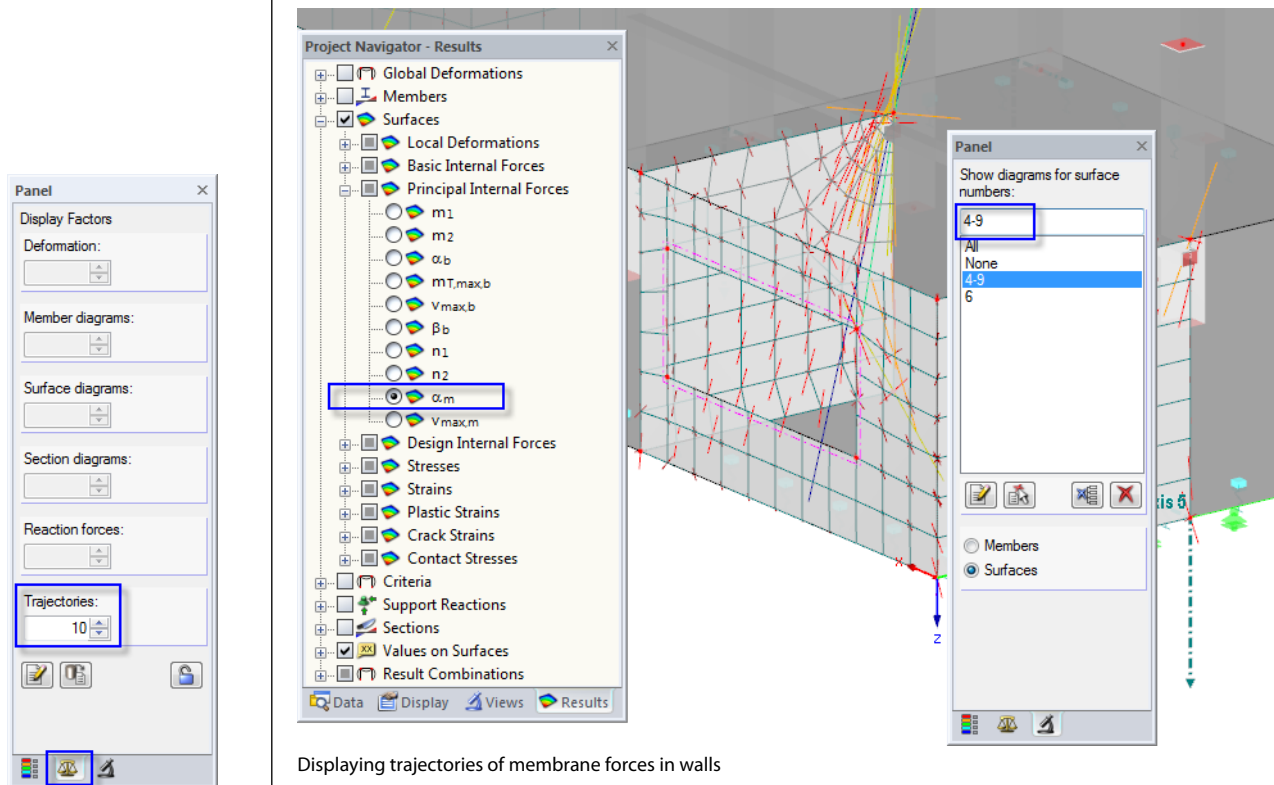




Isolines of surface results m_y (RC1)

Trajectories

We can display the trajectories of the membrane forces of the walls by setting the angle α_m in the **Results navigator** and by selecting the wall surfaces **4-9** in the **Filter** tab of the panel.



Panel tab *Factors*

Displaying trajectories of membrane forces in walls

In the **Factors** tab of the panel, we can increase the display factor of the trajectories.

2.6.5.2 Results Tables

The numerical results are also listed in the tables. We set the membrane forces n_x of **CO18**. In the **Filter** tab of the panel, we select surface **6**.

To display the results in all grid points, we activate the option **On Surface Grids** in the **Results navigator**. Then we set table **4.15 Surfaces - Basic Internal Forces**.

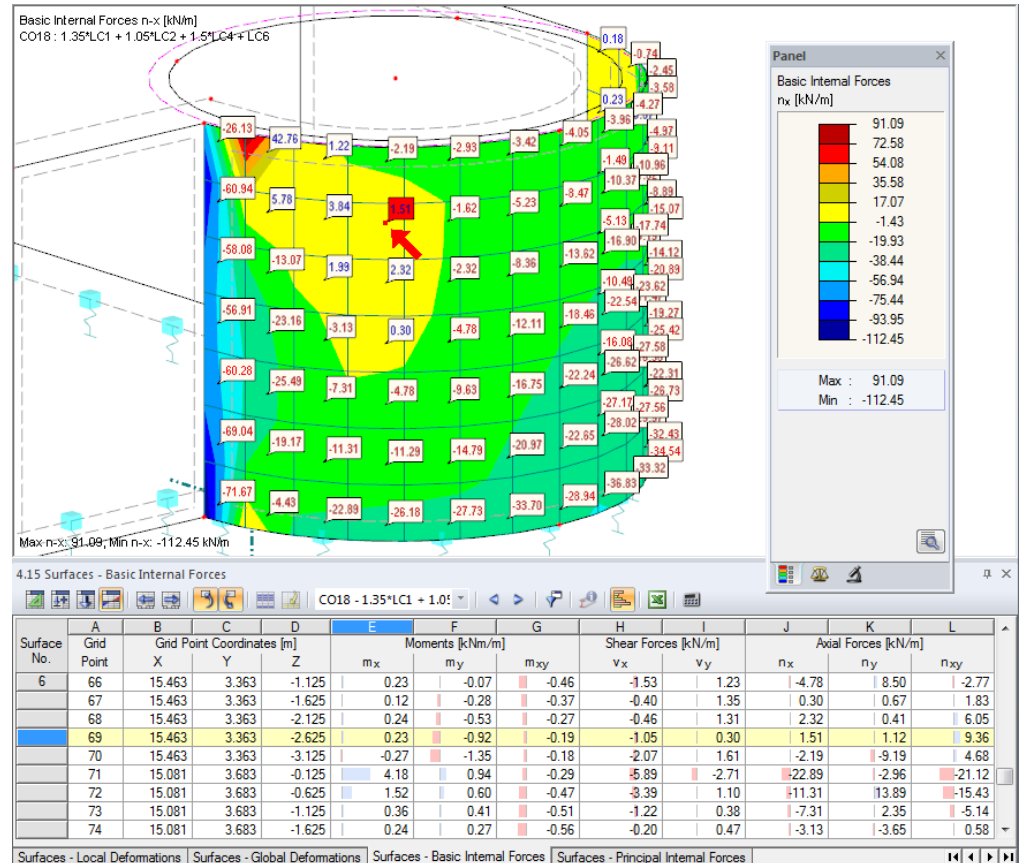
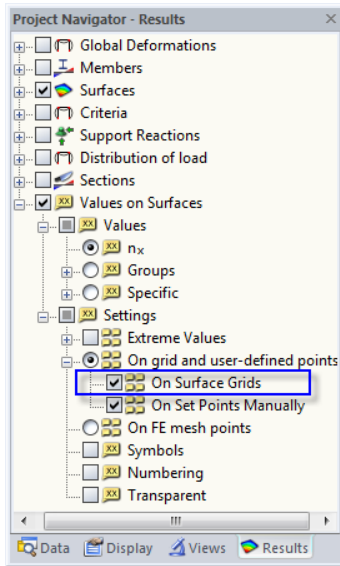


Table 4.15 Surfaces - Basic Internal Forces and graphics with current grid point highlighted (CO18)

The selection of results is synchronized between tables and graphics.

Table filter

We set table **4.6 Members - Internal Forces** to view the results of the columns and the ribs.

4.6 Members - Internal Forces



CO18 - 1.35*LC1 + 1.0:

Member No.	A Node No.	B Location x [m]	C N	D Forces [kN] V _y	E V _z	F M _T	G Moments [kNm] M _y	H M _z	I Cross-Section
10	37	0.000	8.63	0.72	42.09	4.38	-9.66	-2.13	2 - Rectangle
	34	5.500	-44.56	28.09	-75.70	-5.58	-51.58	16.76	
	Max N	2.000	36.75	5.49	3.61	1.11	23.91	-15.45	
	Min N	5.385	-49.82	15.67	-78.42	-7.73	-41.73	18.55	
	Max V _y	5.500	-44.56	28.09	-75.70	-5.58	-51.58	16.76	
	Min V _y	4.441	-5.38	-11.05	-27.93	-1.33	-0.24	-0.06	
	Max V _z	0.000	8.63	0.72	42.09	4.38	-9.66	-2.13	
	Min V _z	5.385	-49.82	15.67	-78.42	-7.73	-41.73	18.55	
	Max M _T	0.000	8.63	0.72	42.09	4.38	-9.66	-2.13	
	Min M _T	5.385	-49.82	15.67	-78.42	-7.73	-41.73	18.55	
	Max M _y	2.500	36.22	1.82	-1.56	0.55	24.41	-15.74	
	Min M _y	5.500	-44.56	28.09	-75.70	-5.58	-51.58	16.76	
	Max M _z	5.385	-49.82	15.67	-78.42	-7.73	-41.73	18.55	
	Min M _z	2.500	36.22	1.82	-1.56	0.55	24.41	-15.74	

Members - Local Deformations

Members - Global Deformations

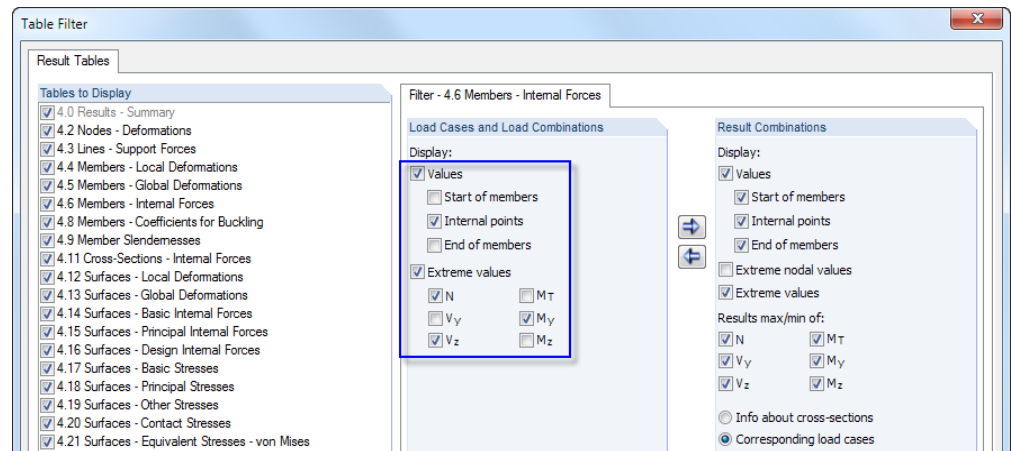
Members - Internal Forces

Members - Coefficients for Buckling

Table 4.6 Members - Internal Forces (CO18)

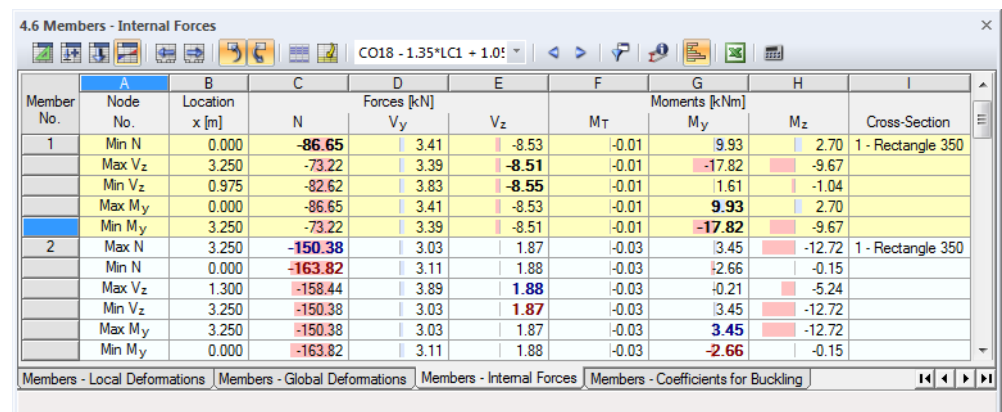


To reduce the results and display the extreme values of selected internal forces only, we apply the **[Filter]** button. In the *Table Filter* dialog box, we disable the values at the **Start** and **End of members**, as well as the extreme values of V_y , M_T and M_z .



Dialog box *Table Filter*

The table is now shorter.



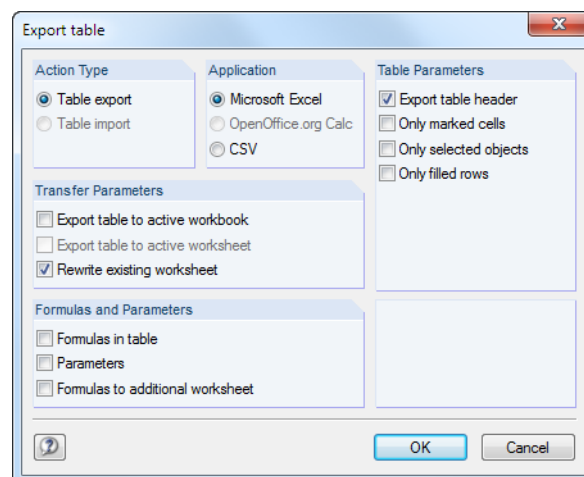
Member No.	Node No.	Location x [m]	N	V _y	V _z	M _T	M _y	M _z	Cross-Section
1	Min N	0.000	-86.65	3.41	-8.53	-0.01	9.93	2.70	1 - Rectangle 350
	Max V _z	3.250	-73.22	3.39	-8.51	-0.01	-17.82	-9.67	
	Min V _z	0.975	-82.62	3.83	-8.55	-0.01	1.61	-1.04	
	Max M _y	0.000	-86.65	3.41	-8.53	-0.01	9.93	2.70	
2	Min M _y	3.250	-73.22	3.39	-8.51	-0.01	-17.82	-9.67	
	Max N	3.250	-150.38	3.03	1.87	-0.03	3.45	-12.72	1 - Rectangle 350
	Min N	0.000	-163.82	3.11	1.88	-0.03	2.66	-0.15	
	Max V _z	1.300	-158.44	3.89	1.88	-0.03	-0.21	-5.24	
	Min V _z	3.250	-150.38	3.03	1.87	-0.03	3.45	-12.72	
	Max M _y	3.250	-150.38	3.03	1.87	-0.03	3.45	-12.72	
	Min M _y	0.000	-163.82	3.11	1.88	-0.03	-2.66	-0.15	

Table 4.6 Members - Internal Forces

Table export



The **[Excel]** button enables us to export the current table data to Excel or OpenOffice Calc.



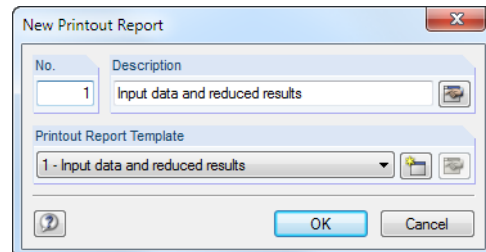
Dialog box *Export table*

2.7 Documentation

2.7.1 Printout Report

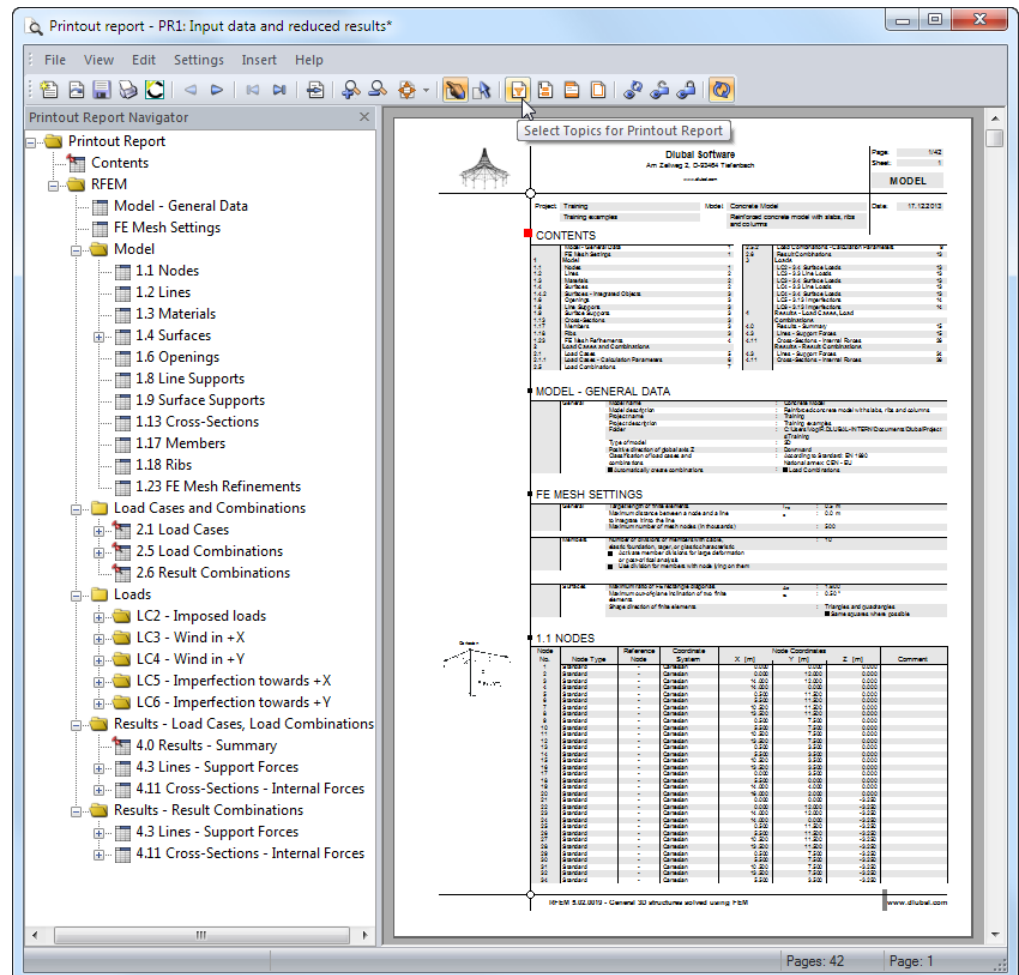


Before printing the data, we create a preview of the documentation – the so-called printout report. We use the **[Printout Report]** button to open the dialog box *New Printout Report*.



Dialog box *New Printout Report*

We apply the **Input data and reduced results** template which includes all input data and the most important results.

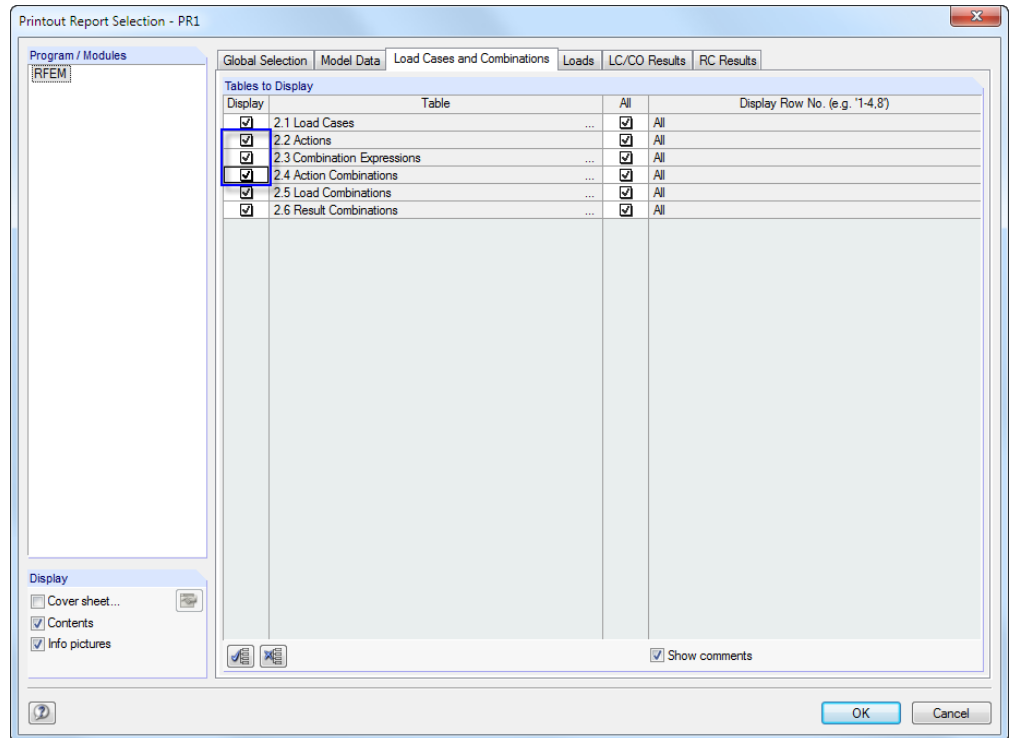


Printout report



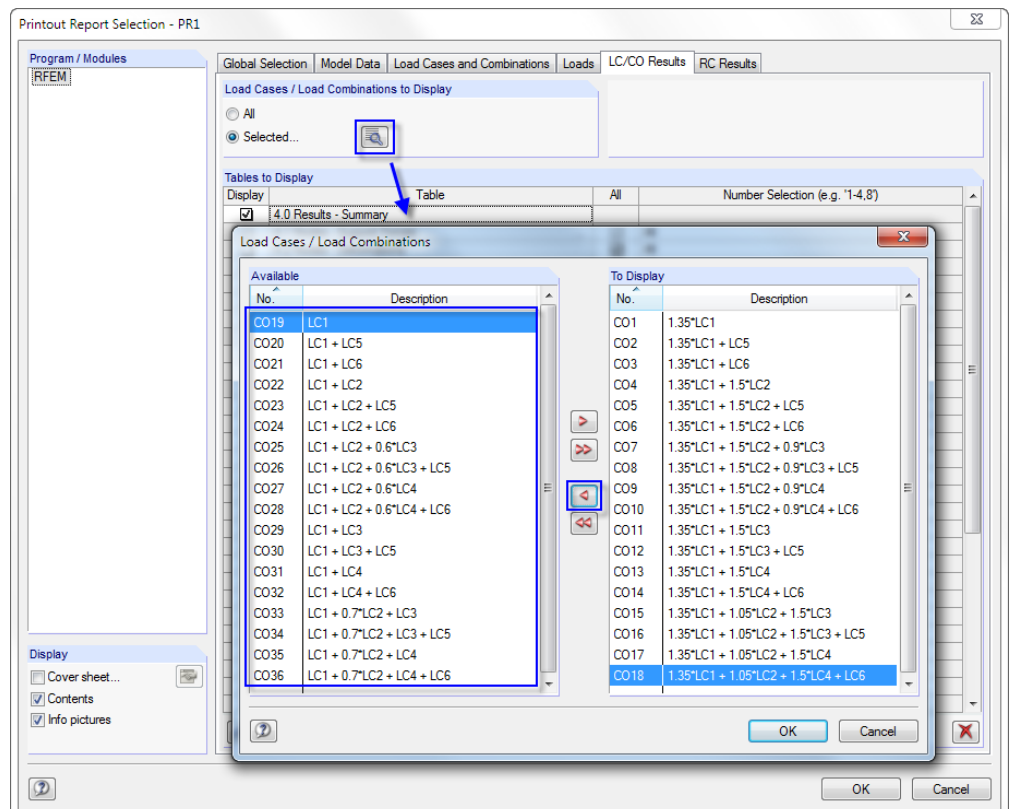
We can modify the contents via the **[Select]** button. The *Printout Report Selection* dialog box opens (see figure below).

In the tab **Load Cases and Combinations**, we activate the tables **2.2 Actions**, **2.3 Combination Expressions** and **2.4 Action Combinations**.



Dialog box *Printout Report Selection*, tab *Load Cases and Combinations*

In the tab **LC/CO Results**, we apply the button **[Select]** to remove all SLS load combinations. We select **CO19 through CO36** in the right column by using [Shift]. Then we press [◀].



Dialog box *Printout Report Selection*, tab *LC/CO Results* with deselected SLS load combinations

Global Selection | Model Data | Load Cases and Combinations | Loads | LC/CO Results | RC Results

Result Combinations to Display

☒ All
☐ Selected...

Tables to Display

Display	Table	All	Number Selection (e.g. '1-4,8')
☒	4.1 Nodes - Support Forces	☒ All	
☐	4.2 Nodes - Deformations	☒ All	
☒	4.3 Lines - Support Forces	☒ All	
☒	4.4 Members - Local Deformations	☐ Members: 10,11	
☐	4.5 Members - Global Deformations	☒ All	
☒	4.6 Members - Internal Forces	☒ All	
☐	4.7 Members - Contact Forces	☒ All	
☐	4.8 Members - Coefficients for Buckling	☒ All	
☐	4.9 Member Slendernesses	☒ All	
☐	4.10 Sets of Members - Internal Forces	☒ All	
☒	4.11 Cross-Sections - Internal Forces	☒ All	
☒	4.12 Surfaces - Local Deformations	☐ 3	
☐	4.13 Surfaces - Global Deformations	☒ All	
☒	4.14 Surfaces - Basic Internal Forces	☒ All	
☐	4.15 Surfaces - Principal Internal Forces	☒ All	
☐	4.16 Surfaces - Design Internal Forces	☒ All	
☐	4.17 Surfaces - Basic Stresses	☒ All	
☐	4.18 Surfaces - Principal Stresses	☒ All	
☐	4.19 Surfaces - Other Stresses	☒ All	
☐	4.20 Surfaces - Contact Stresses	☒ All	
☐	4.21 Surfaces - Equivalent Stresses von Mises	☒ All	
☐	4.22 Surfaces - Equivalent Stresses Tresca	☒ All	
☐	4.23 Surfaces - Equivalent Stresses Rankine	☒ All	
☐	4.24 Surfaces - Equivalent Stresses Bach	☒ All	
☐	4.25 Surfaces - Basic Strains	☒ All	
☐	4.26 Surfaces - Principal Strains	☒ All	
☐	4.27 Surfaces - Maximum Strains	☒ All	
☐	4.28 Surfaces - Strains von Mises	☒ All	
☐	4.29 Surfaces - Strains Tresca	☒ All	

☒ Show corresponding load cases

After [OK], the printout report is created with the new settings.

Printout report - PR1: Input data and reduced results

File View Edit Settings Insert Help

Printout Report Navigator

Printout Report

Contents

RFEM

Model - General Data

FE Mesh Settings

Model

Load Cases and Combinations

Loads

Results - Load Cases, Load Combinations

Results - Result Combinations

4.3 Lines - Support Forces

4.4 Members - Local Deformations

RC1: ULS (STR/GEO) - Permanent

RC2: SLS - Characteristic

4.6 Members - Internal Forces

4.11 Cross-Sections - Internal Forces

4.12 Surfaces - Local Deformations

4.14 Surfaces - Basic Internal Forces

4.4 MEMBERS - LOCAL DEFORMATIONS

Member No.	RC	Node No.	Location x [m]		Displacements [mm]		
					U _x	U _y	U _z
10	RC1	37	0.000	max	0.2	0.1	0.1
				min	0.1	0.0	0.1
	34	5.500	max	0.3	0.1	0.8	
			min	0.2	0.0	0.6	
RC2	37	0.000	max	0.1	0.0	0.1	
			min	0.1	0.0	0.1	
	34	5.500	max	0.2	0.0	0.6	
			min	0.1	0.0	0.4	
11	RC1	34	0.000	max	0.3	0.1	0.8
				min	0.2	0.0	0.6
	35	5.000	max	0.3	0.1	0.8	
			min	0.2	0.0	0.6	
RC2	34	0.000	max	0.2	0.0	0.6	
			min	0.1	0.0	0.4	
	35	5.000	max	0.2	0.1	0.6	
			min	0.1	0.0	0.4	

4.6 MEMBERS - INTERNAL FORCES

Member No.	RC	Node No.	Location x [m]	Forces [kN]			
				N	V _y	V _z	M _T

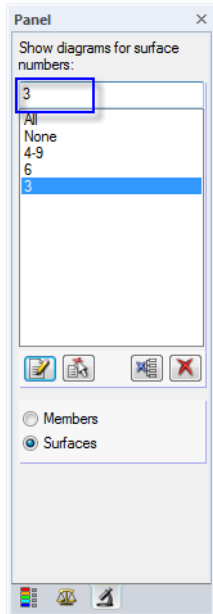
Pages: 159 Page: 37

We now close the report and save it.

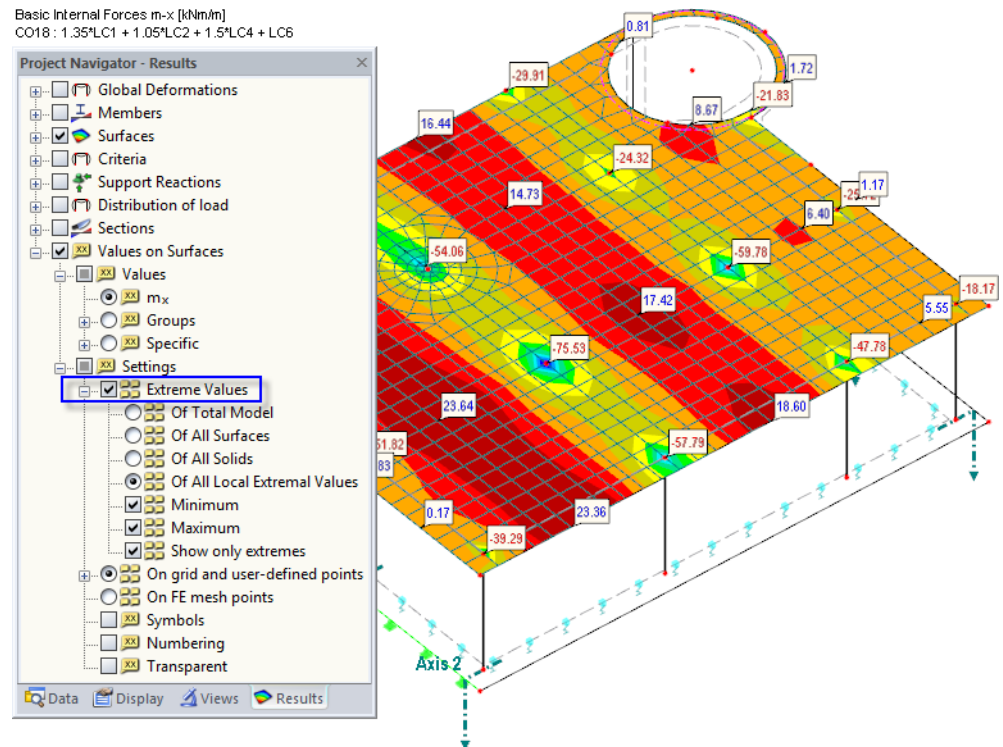
2.7.2 Graphics

2.7.2.1 Graphic Printout

In the RFEM work window, we display the moments m_x of CO18 for the ceiling surface 3. Additionally, we activate the **Extreme Values**.



Panel tab Filter

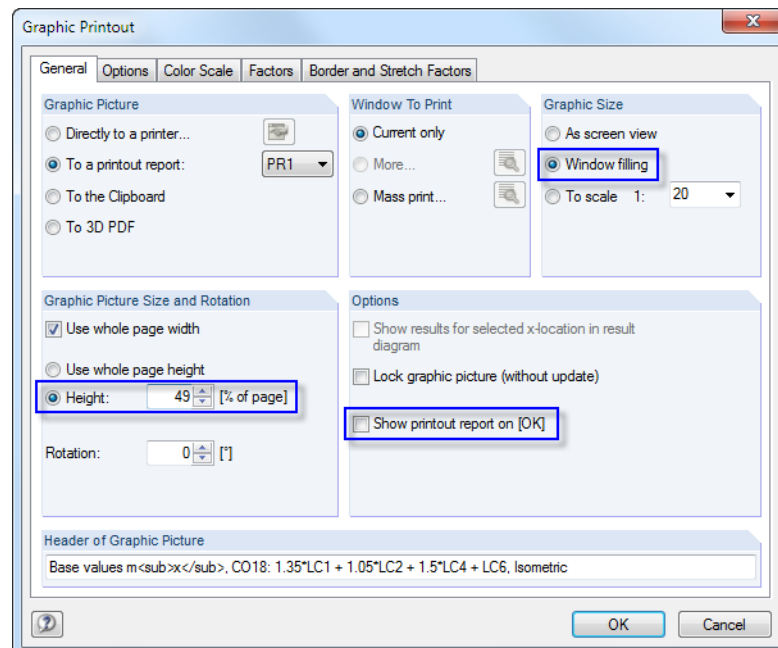


Moments m_x of ceiling slab (CO 18)



To print this image, we click the **[Print]** button.

In the **Graphic Printout** dialog box, we choose the following settings.

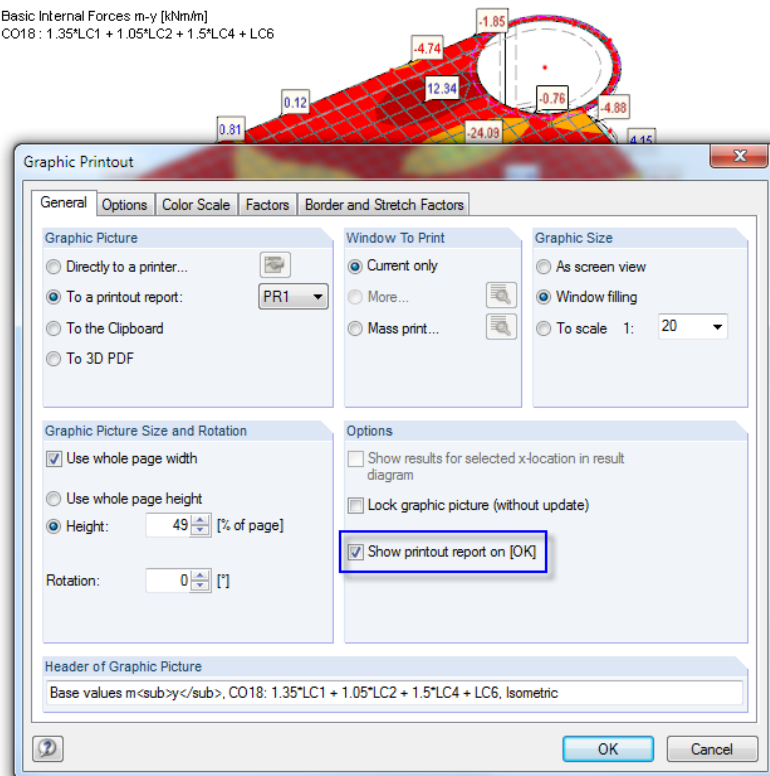


Dialog box *Graphic Printout*



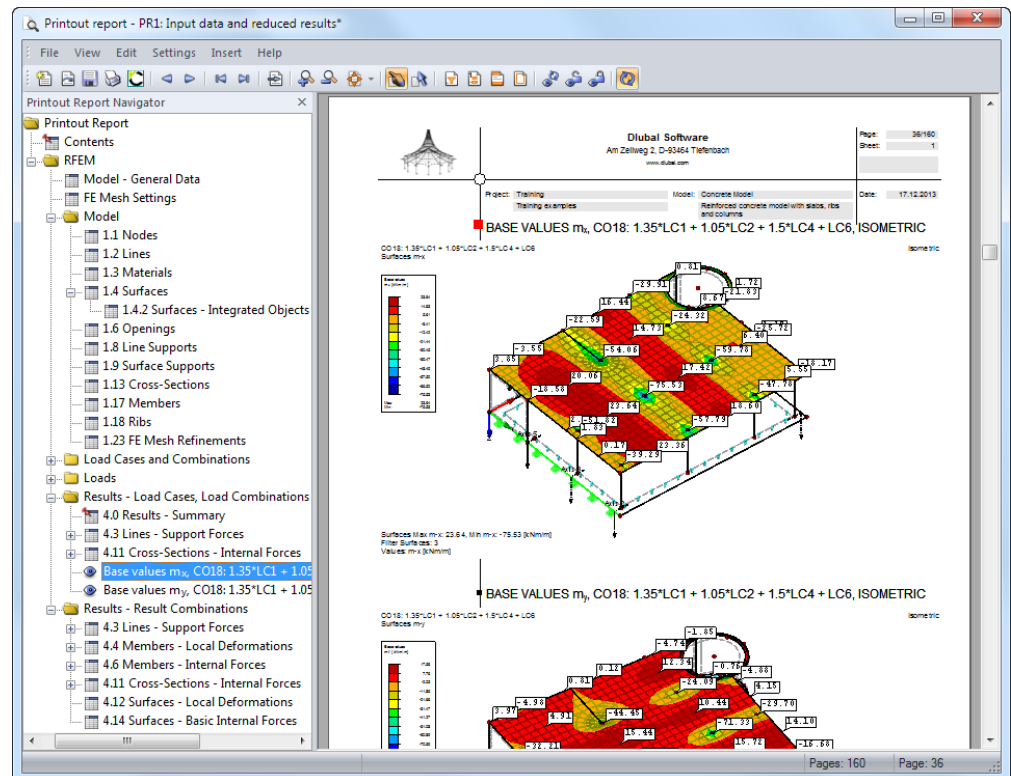
After [OK], we set the moment m_y of the ceiling surface. Then we open the **Graphic Printout** dialog box again. We activate the option **Show printout report on [OK]**.

Basic Internal Forces m-y [kNm/m]
CO18: 1.35*LC1 + 1.05*LC2 + 1.5*LC4 + LC6



Dialog box *Graphic Printout*

After [OK], the printout report is created.

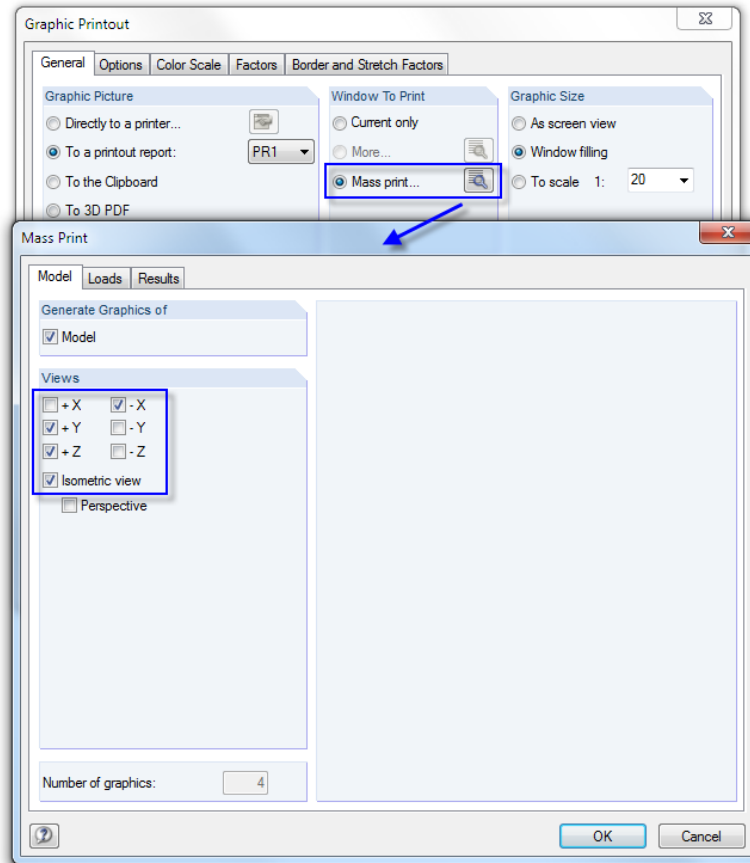
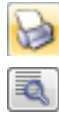


Graphics in printout report

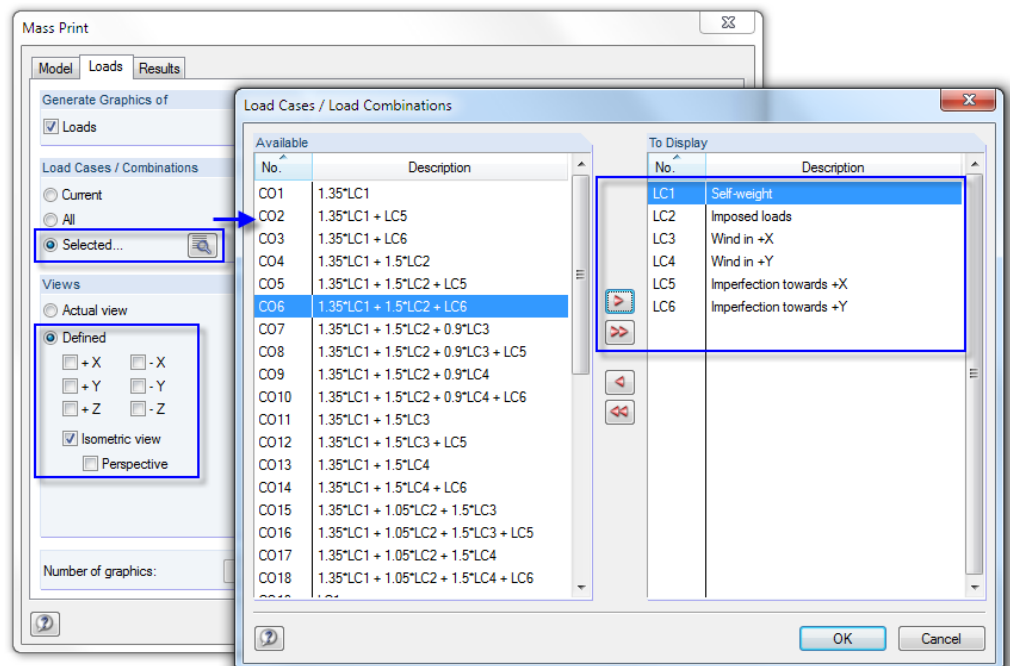
2.7.2.2 Mass Print

To include graphics that are created automatically by RFEM, we use the **[Print]** button again.

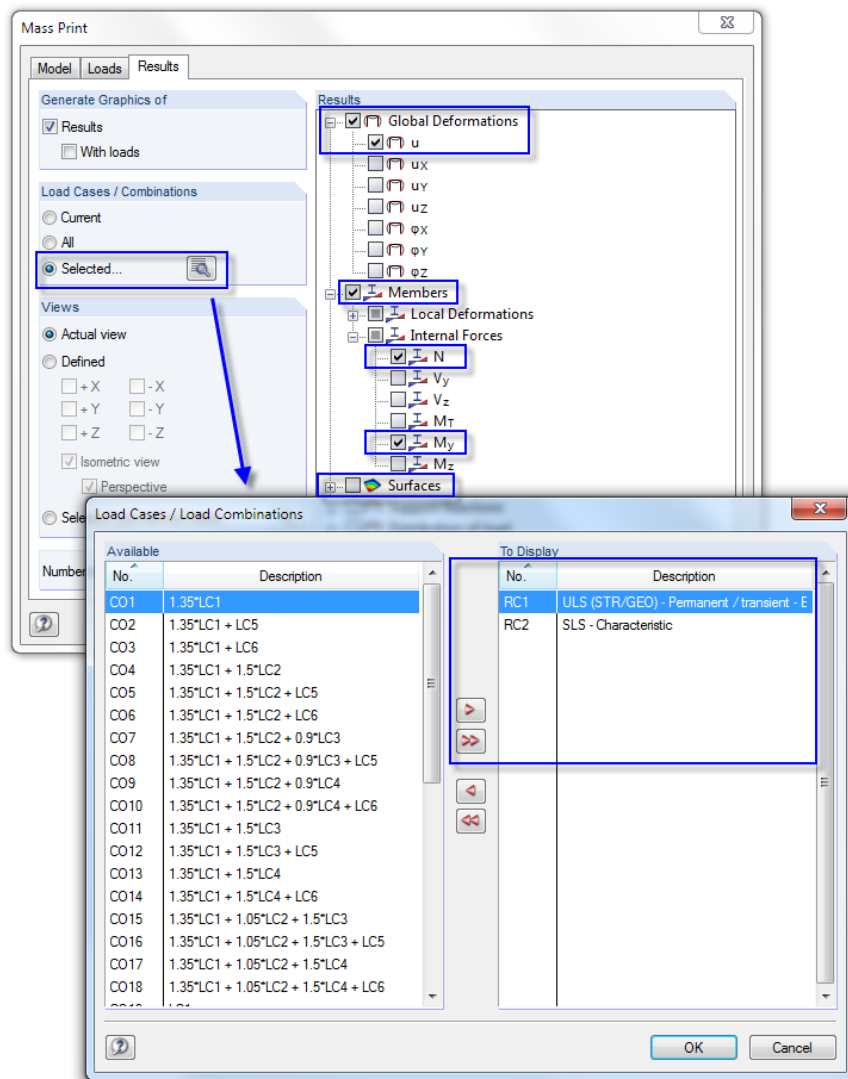
We select the **Mass print** option, click the **[Edit]** button and apply the following settings.



Dialog box *Mass Print*, tab *Model*

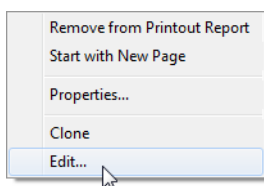


Dialog box *Mass Print*, tab *Loads*

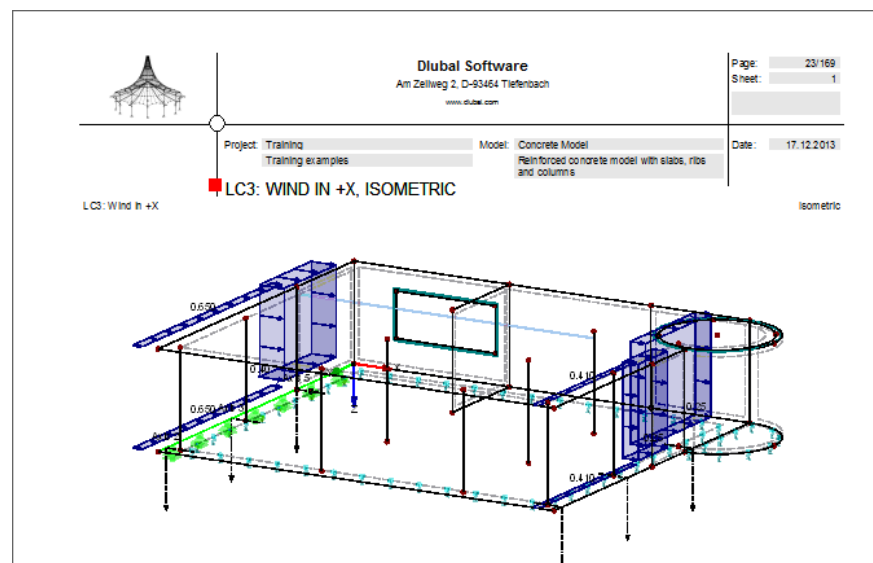


Dialog box *Mass Print*, tab *Results*

After [OK], the printout report is created again. If required, we can **Edit** the graphics.



Context menu of graphics



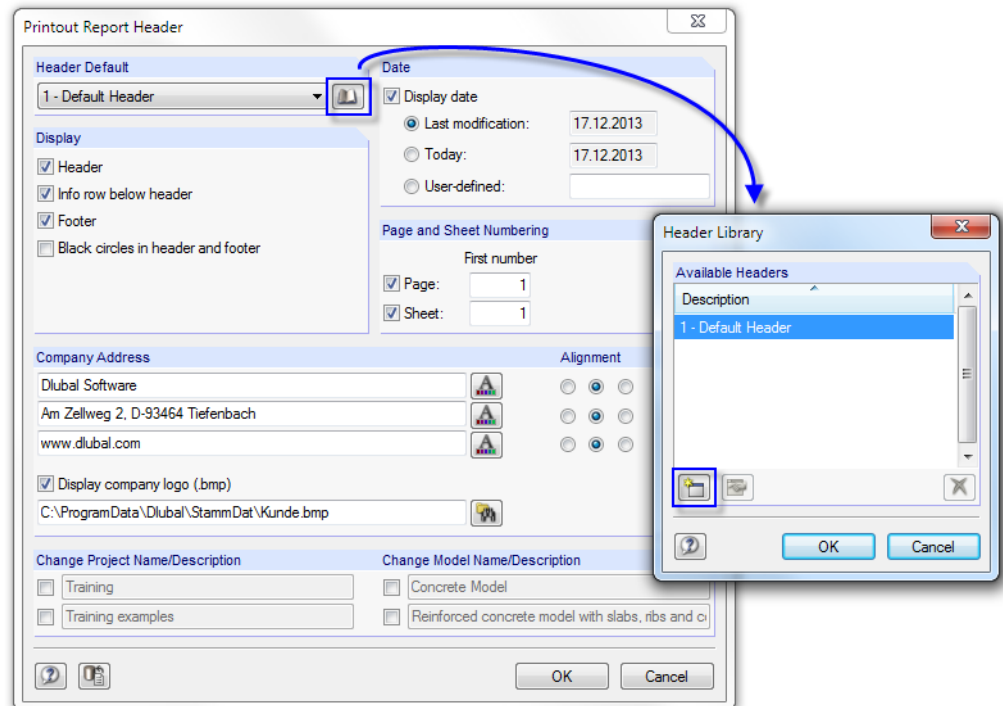
Mass print image in printout report

2.7.3 Printout Report Header



We can change the header of the printout report by clicking the **[Header]** button.

The **Printout Report Header** dialog box provides many options to define the company address, logo, date, page numbering, etc.



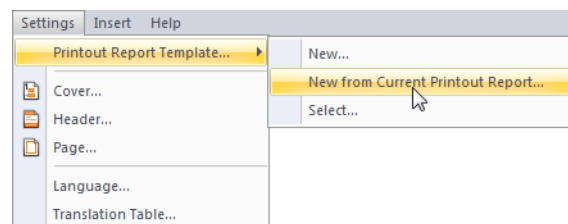
Dialog boxes *Printout Report Header* and *Header Library*



In the *Header Library*, it is possible to define a **[New]** header which can then be set as *Header Default*.

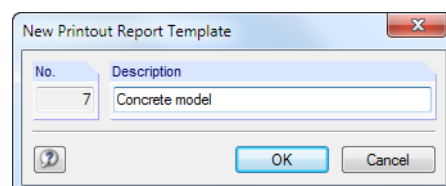
2.7.4 Printout Report Template

We create a new template from our selection so that we apply it in similar models.



Menu *Settings* → *Printout Report Template*

In the dialog box **New Printout Report Template**, we enter the *Description* of our template.



Dialog box *New Printout Report Template*

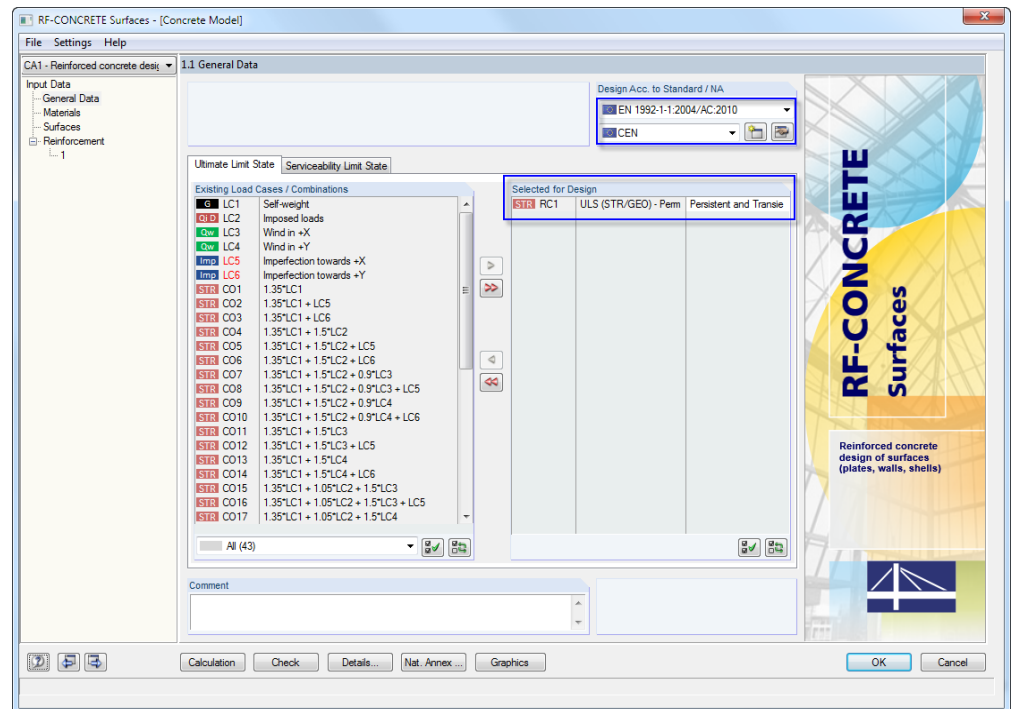
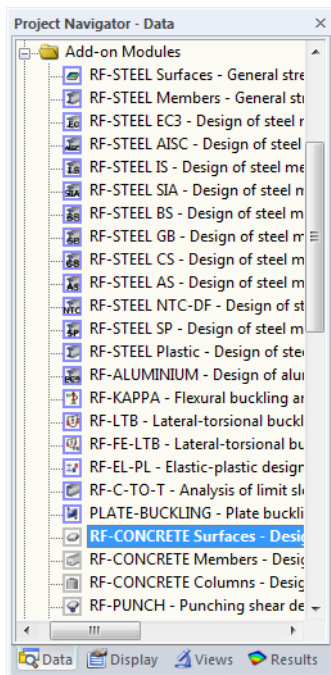
After [OK], we close the printout report.

2.8 RF-CONCRETE Surfaces

We double-click **RF-CONCRETE Surfaces** in the **Data navigator** to open the add-on module.

2.8.1 Input Data

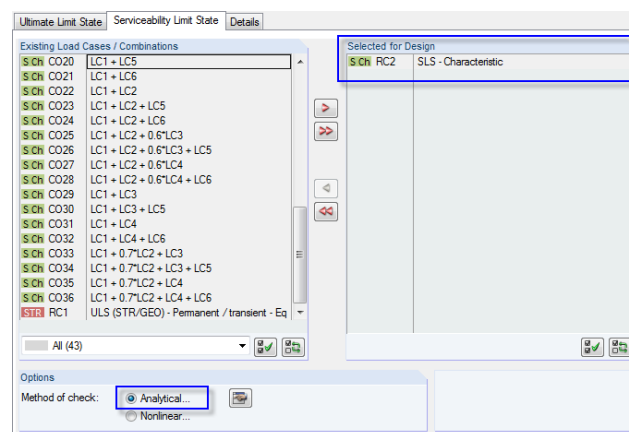
2.8.1.1 General Data



Window RF-CONCRETE Surfaces, 1.1 General Data, tab Ultimate Limit State

In window 1.1, we select **RC1** for the **Ultimate Limit State** design according to **EN 1992-1-1** and the **CEN** recommendations.

In the **Serviceability Limit State** tab, we set **RC2** and the **Analytical** design method.



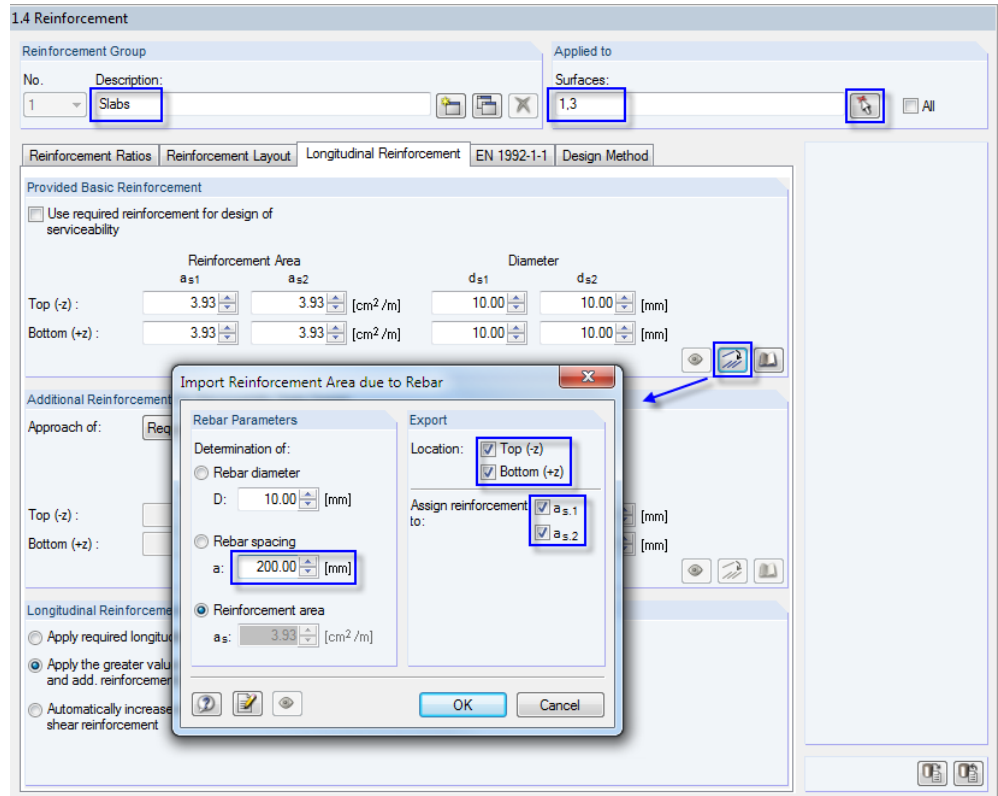
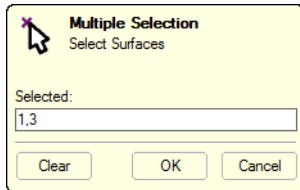
Window 1.1 General Data, tab Serviceability Limit State

We check the settings in windows **1.2 Materials** and **1.3 Surfaces** but do not change them.

2.8.1.2 Reinforcement

The **1.4 Reinforcement** window contains five tabs that control the reinforcement parameters. We define two reinforcement groups with different settings for the slabs and walls. For the **Slabs**, we select surfaces **1,3** graphically via [^].

In the **Longitudinal Reinforcement** tab, we use the **[Rebars]** button to define a basic reinforcement as seen in the figure below.

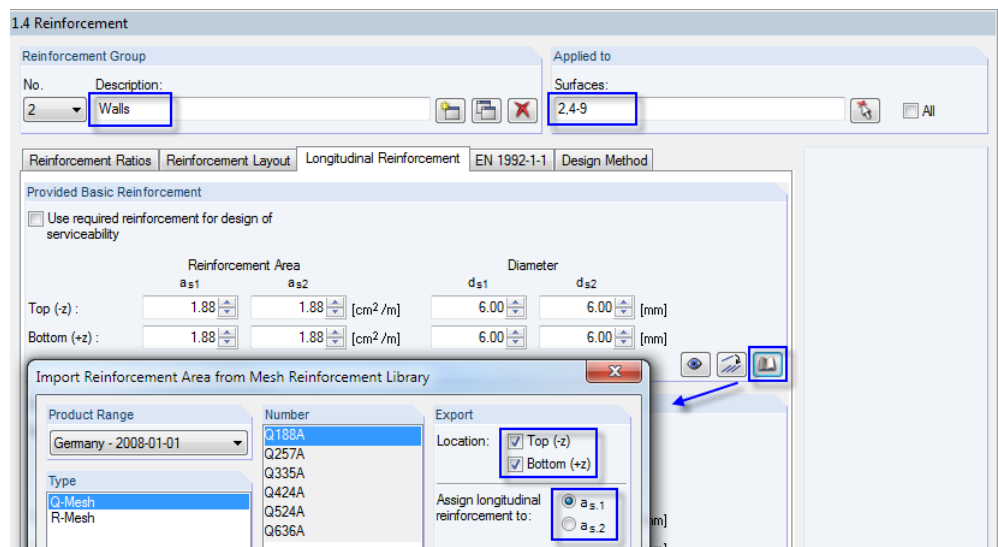


Window 1.4 Reinforcement, tab Longitudinal Reinforcement



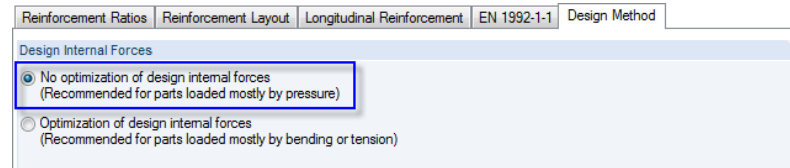
We create a **[New]** reinforcement group for the **Walls**. The numbers of the wall surfaces are entered automatically.

In the **Longitudinal Reinforcement** tab, we use the **[Library]** to define the basic reinforcement.



Window 1.4 Reinforcement, tab Longitudinal Reinforcement

In the **Design Method** tab, we select the option **No optimization of design internal forces**.



Window 1.4 Reinforcement, tab Design Method

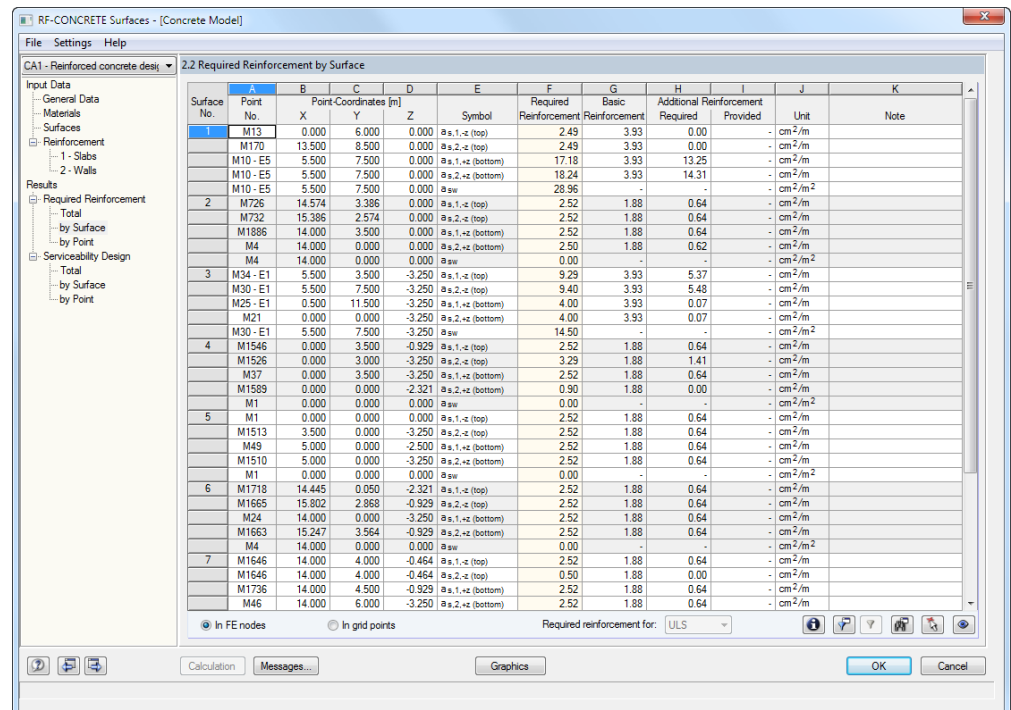
Calculation

We can check on the settings in the other tabs. Then we start the [Calculation].

2.8.2 Results

The results of the ULS and SLS designs are listed separately.

2.8.2.1 Required Reinforcement



Surface No.	Point No.	Point-Coordinates [m]	X	Y	Z	Symbol	Required Reinforcement	Basic Reinforcement	Additional Reinforcement	Unit
1	M13	0.000	6.000	0.000	0.000	$\sigma_{s,1,-z}$ (top)	2.49	3.93	0.00	- cm²/m
	M170	13.500	8.500	0.000	0.000	$\sigma_{s,2,-z}$ (top)	2.49	3.93	0.00	- cm²/m
	M10 - E5	5.500	7.500	0.000	0.000	$\sigma_{s,1,-z}$ (bottom)	17.18	3.93	13.25	- cm²/m
	M10 - E5	5.500	7.500	0.000	0.000	$\sigma_{s,2,-z}$ (bottom)	18.24	3.93	14.31	- cm²/m
	M10 - E5	5.500	7.500	0.000	0.000	σ_{sw}	28.96	-	-	- cm²/m²
2	M726	14.574	3.386	0.000	0.000	$\sigma_{s,1,-z}$ (top)	2.52	1.88	0.64	- cm²/m
	M732	15.386	2.574	0.000	0.000	$\sigma_{s,2,-z}$ (top)	2.52	1.88	0.64	- cm²/m
	M1886	14.000	3.500	0.000	0.000	$\sigma_{s,1,-z}$ (bottom)	2.52	1.88	0.64	- cm²/m
	M4	14.000	0.000	0.000	0.000	$\sigma_{s,2,-z}$ (bottom)	2.50	1.88	0.62	- cm²/m
	M4	14.000	0.000	0.000	0.000	σ_{sw}	0.00	-	-	- cm²/m²
3	M34 - E1	5.500	3.500	-3.250	0.000	$\sigma_{s,1,-z}$ (top)	9.29	3.93	5.37	- cm²/m
	M30 - E1	5.500	7.500	-3.250	0.000	$\sigma_{s,2,-z}$ (top)	9.40	3.93	5.48	- cm²/m
	M25 - E1	0.500	11.500	-3.250	0.000	$\sigma_{s,1,-z}$ (bottom)	4.00	3.93	0.07	- cm²/m
	M21	0.000	0.000	-3.250	0.000	$\sigma_{s,2,-z}$ (bottom)	4.00	3.93	0.07	- cm²/m
	M30 - E1	5.500	7.500	-3.250	0.000	σ_{sw}	14.50	-	-	- cm²/m²
4	M1546	0.000	3.500	-0.929	0.000	$\sigma_{s,1,-z}$ (top)	2.52	1.88	0.64	- cm²/m
	M1526	0.000	3.000	-3.250	0.000	$\sigma_{s,2,-z}$ (top)	3.29	1.88	1.41	- cm²/m
	M37	0.000	3.500	-3.250	0.000	$\sigma_{s,1,-z}$ (bottom)	2.52	1.88	0.64	- cm²/m
	M1589	0.000	0.000	-2.321	0.000	$\sigma_{s,2,-z}$ (bottom)	0.90	1.88	0.00	- cm²/m
	M1	0.000	0.000	0.000	0.000	σ_{sw}	0.00	-	-	- cm²/m²
5	M1	0.000	0.000	0.000	0.000	$\sigma_{s,1,-z}$ (top)	2.52	1.88	0.64	- cm²/m
	M1513	3.500	0.000	-3.250	0.000	$\sigma_{s,2,-z}$ (top)	2.52	1.88	0.64	- cm²/m
	M49	5.000	0.000	-2.500	0.000	$\sigma_{s,1,-z}$ (bottom)	2.52	1.88	0.64	- cm²/m
	M1510	5.000	0.000	-3.250	0.000	$\sigma_{s,2,-z}$ (bottom)	2.52	1.88	0.64	- cm²/m
	M1	0.000	0.000	0.000	0.000	σ_{sw}	0.00	-	-	- cm²/m²
6	M1718	14.445	0.050	-2.321	0.000	$\sigma_{s,1,-z}$ (top)	2.52	1.88	0.64	- cm²/m
	M1655	15.802	2.868	-0.929	0.000	$\sigma_{s,2,-z}$ (top)	2.52	1.88	0.64	- cm²/m
	M24	14.000	0.000	-3.250	0.000	$\sigma_{s,1,-z}$ (bottom)	2.52	1.88	0.64	- cm²/m
	M1663	15.247	3.564	-0.929	0.000	$\sigma_{s,2,-z}$ (bottom)	2.52	1.88	0.64	- cm²/m
	M4	14.000	0.000	0.000	0.000	σ_{sw}	0.00	-	-	- cm²/m²
7	M1646	14.000	4.000	-0.464	0.000	$\sigma_{s,1,-z}$ (top)	2.52	1.88	0.64	- cm²/m
	M1646	14.000	4.000	-0.464	0.000	$\sigma_{s,2,-z}$ (top)	0.50	1.88	0.00	- cm²/m
	M1736	14.000	4.500	-0.929	0.000	$\sigma_{s,1,-z}$ (bottom)	2.52	1.88	0.64	- cm²/m
	M46	14.000	6.000	-3.250	0.000	$\sigma_{s,2,-z}$ (bottom)	2.52	1.88	0.64	- cm²/m

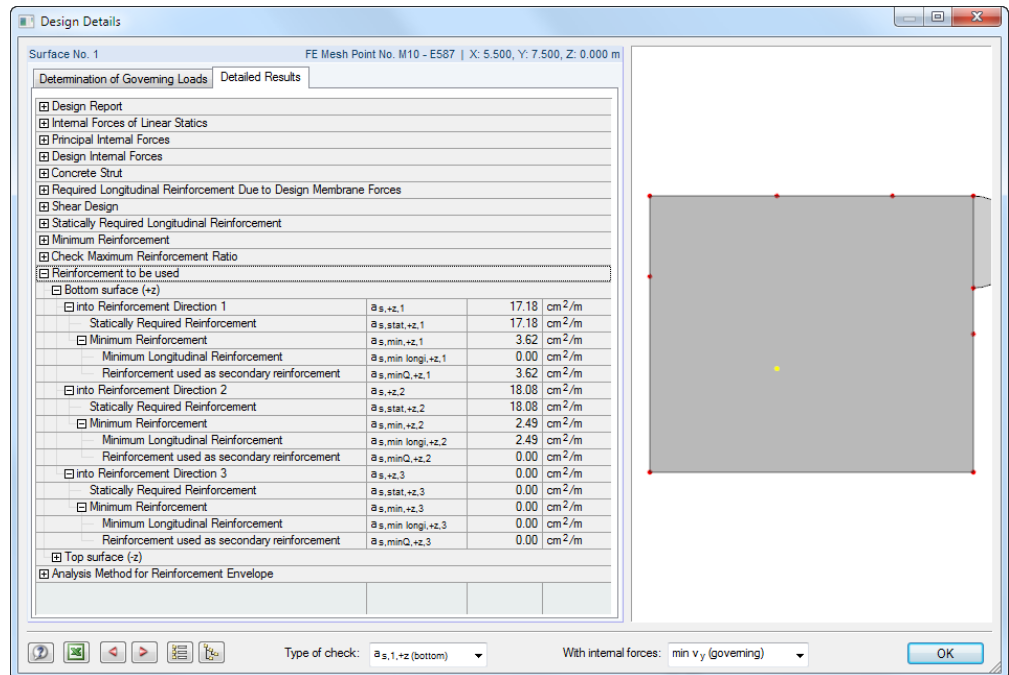
Window 2.2 Required Reinforcement by Surface

The ULS design results are listed for the **top** and **bottom** surface sides in each of the directions **1** and **2**. Additionally, the shear reinforcement σ_{sw} is displayed.

The tables can be sorted by the results *In FE Nodes* and *In grid points*.



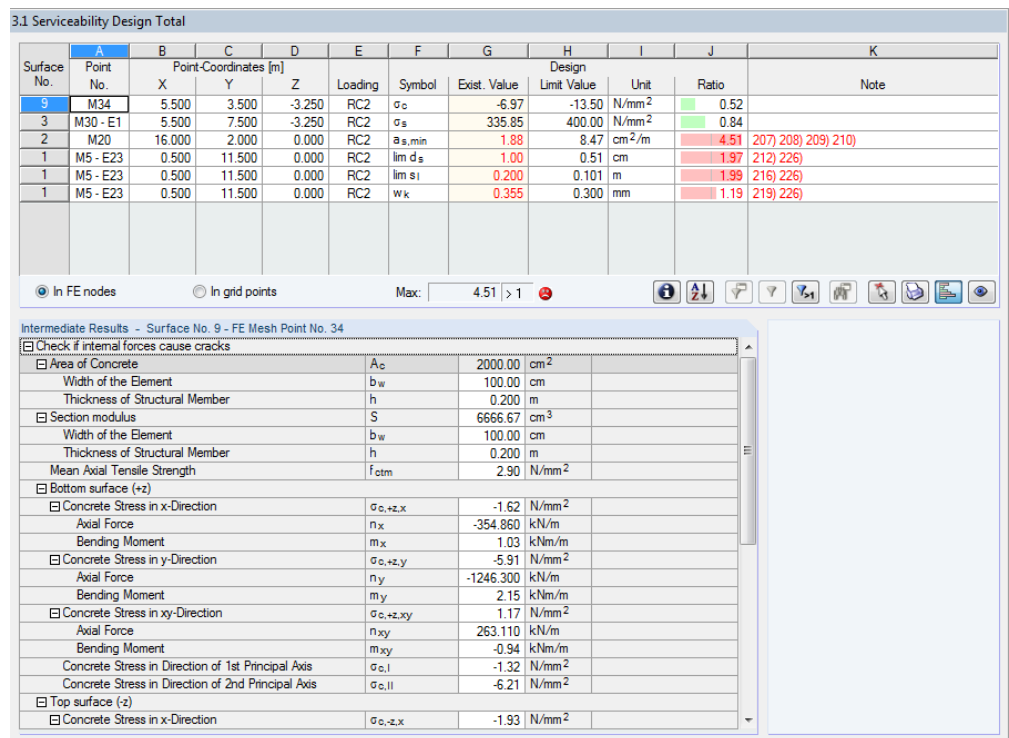
We can check on the design details of every location by clicking the **[Info]** button.



Dialog box *Design Details*

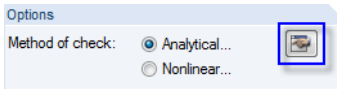
2.8.2.2 Serviceability Design

The SLS design includes the verification of stresses, limiting bar diameters and bar spacings, minimum reinforcement and crack widths.

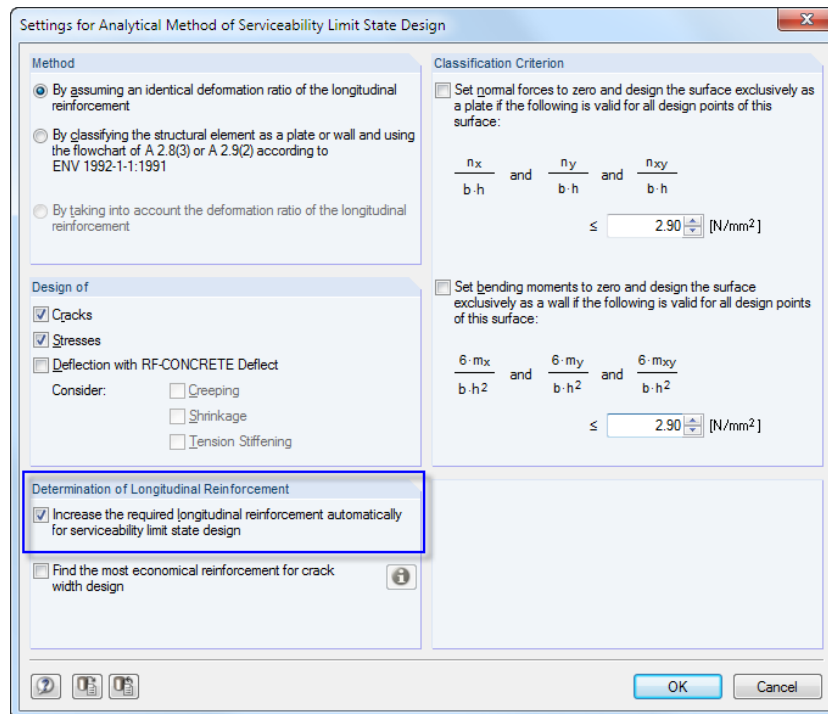


Window 3.1 *Serviceability Design Total*

Not every single aspect must be fulfilled for the SLS design. Nevertheless, the minimum reinforcement $a_{s,min}$ is a problem.

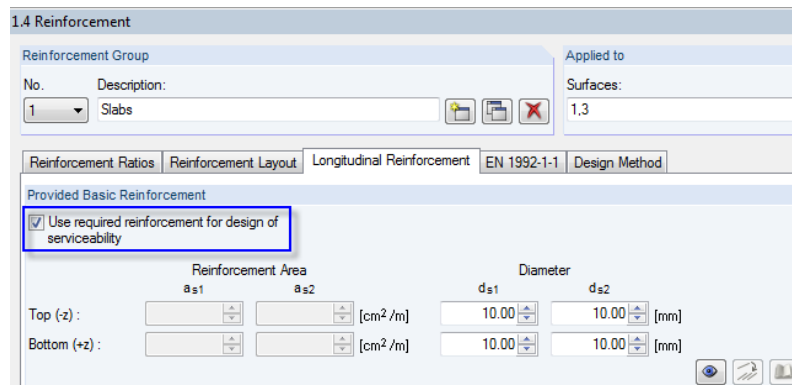


We return to window **1.1 General Data**, tab **Serviceability Limit State** and click the [Edit] button. In the dialog box, we then tick the option **Increase the required longitudinal reinforcement automatically for serviceability design**.



Dialog box *Settings for Analytical Method of SLS Design*

In window **1.4 Reinforcement**, we tick the option **Use required reinforcement for design of serviceability** – for both reinforcement groups **Slabs** and **Walls**.



Window *1.4 Reinforcement*, tab *Longitudinal Reinforcement*

Calculation

After the **[Calculation]**, the SLS design is fulfilled.

3.1 Serviceability Design Total

Surface No.	A Point No.	B X	C Y	D Z	E Loading	F Symbol	G Exist. Value	H Design Limit Value	I Unit	J Ratio	K Note
9	M34	5.500	3.500	-3.250	RC2	σ_c	-6.97	-13.50	N/mm ²	0.52	
3	M34 - E1	5.500	3.500	-3.250	RC2	σ_s	328.74	400.00	N/mm ²	0.83	226)
1	M1	0.000	0.000	0.000	RC2	$a_{s,min}$	7.60	7.60	cm ² /m	1.00	
3	M31 - E1	10.500	7.500	-3.250	RC2	$lim\ d_s$	1.00	1.00	cm	1.00	
3	M31 - E1	10.500	7.500	-3.250	RC2	$lim\ s_l$	0.108	0.108	m	1.00	
3	M31 - E1	10.500	7.500	-3.250	RC2	w_k	0.299	0.300	mm	1.00	

☒ In FE nodes
 ☐ In grid points
 Max: 1.00 ≤ 1

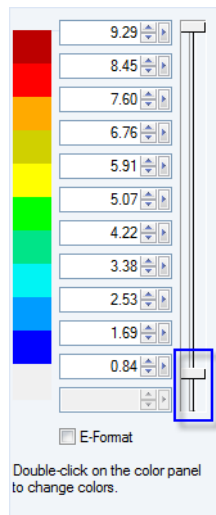
Window *3.1 Serviceability Design Total*

2.8.2.3 Graphical Results

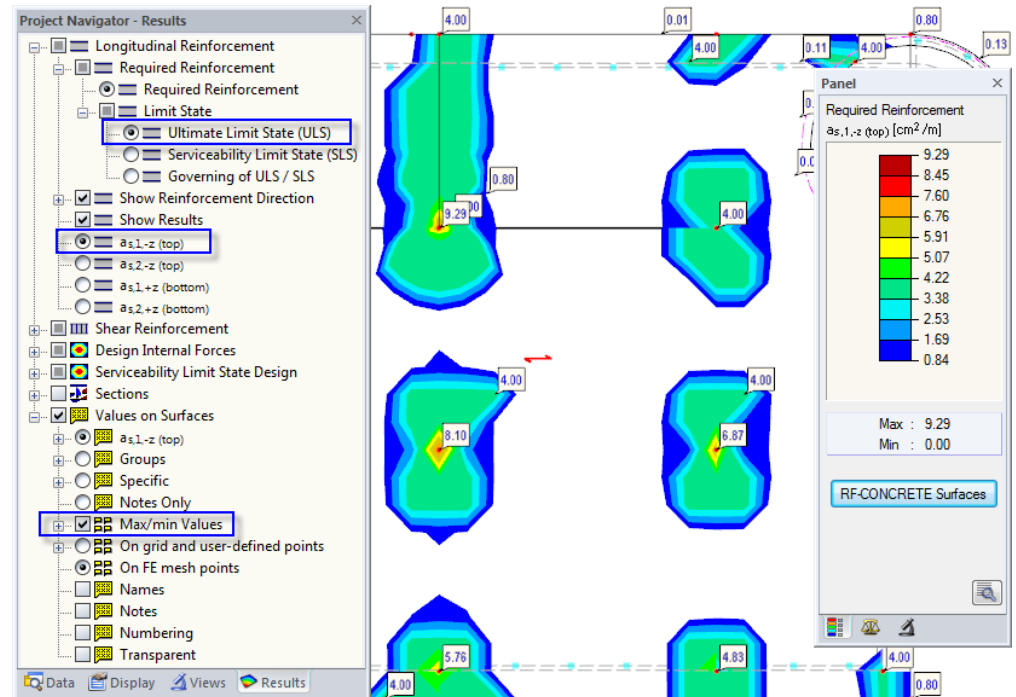
We use the **[Graphics]** button to evaluate the design results in the work window of RFEM.

In the **Result navigator**, we set the required reinforcement $a_{s,1,-z}$ (top) of the **ULS** design. To display the results of the ceiling slab only, we set surface **3** in the **Filter** tab of the panel.

Additionally, we disable the regions with very low reinforcement by removing the dark blue color: We double-click the panel colors and then move the slider one step up in the dialog box.

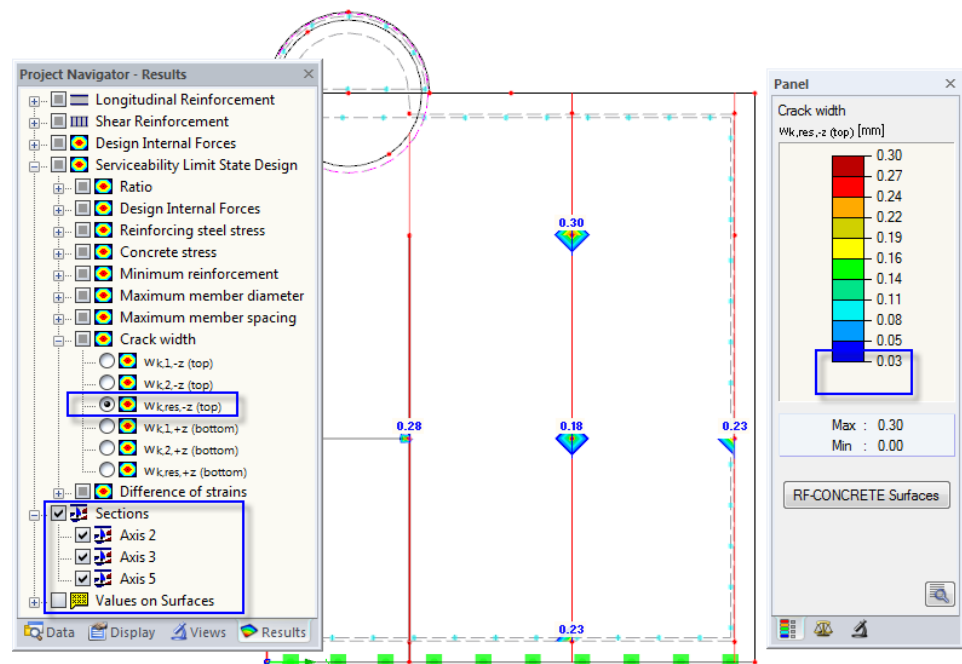


Reducing panel colors



Top reinforcement in direction 1 of ceiling slab with edited color scale

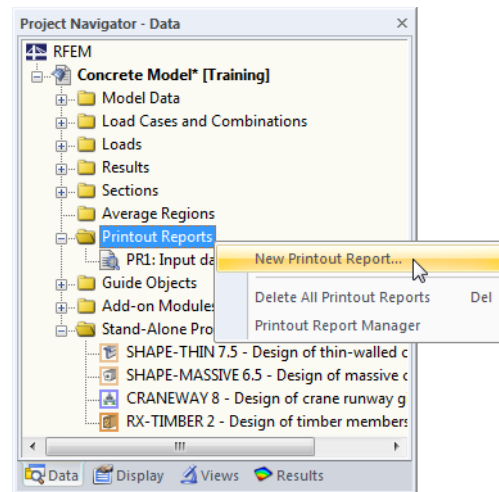
As for the SLS designs results, we select the resulting crack widths $w_{k,res,-z}$ (top) of the ceiling slab. We activate the display of the **Sections**, too. By removing the blue color, we get a better view.



Resulting crack widths of ceiling slab

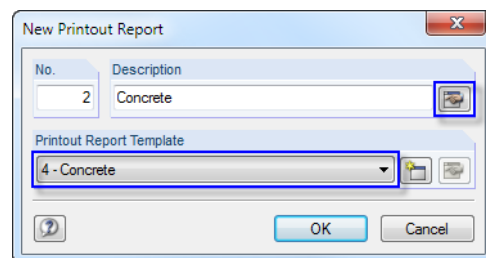
2.8.2.4 Printout

We create a new printout report which includes only the design results.



Creating new printout report in *Data* navigator

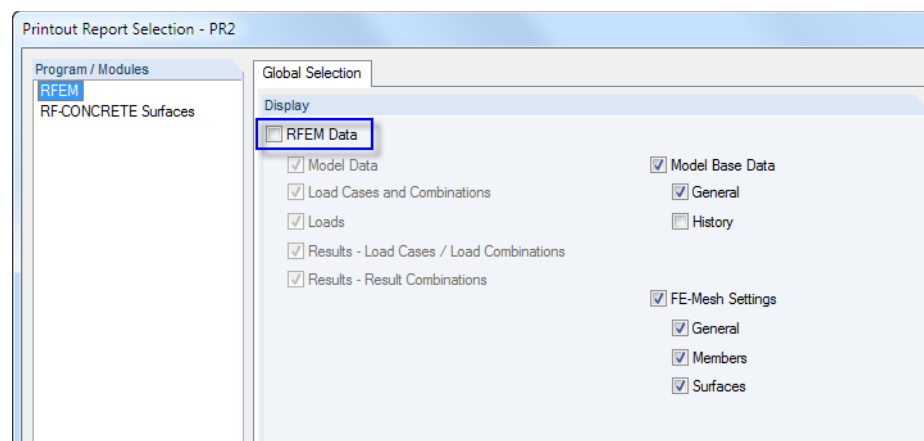
In the **New Printout Report** dialog box, we select template **4 - Concrete**.



Dialog box *New Printout Report*



We **[Edit]** the template settings. In the **Printout Report Selection** dialog box, we disable all **RFEM Data**.



Dialog box *Printout Report Selection - PR2*

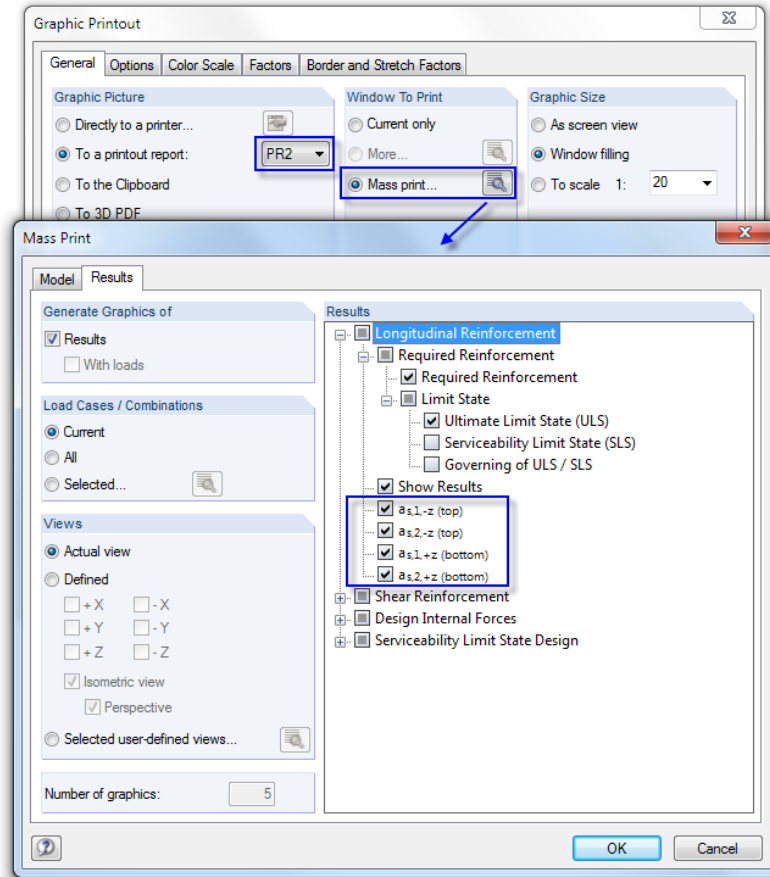
After [OK] in all dialog boxes, the second printout report is created.

We can check on the data before we close the printout report again.

In the RFEM work window, we activate the **Values on Surfaces** again.

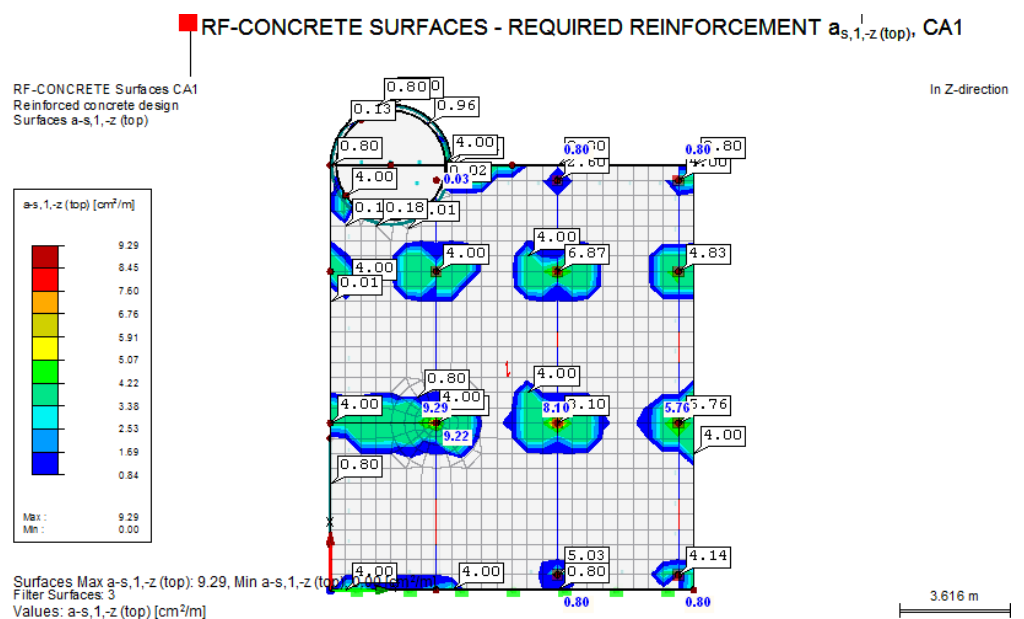


We open the **[Graphic Printout]** dialog box again. There we select printout report **PR2** and the **Mass Print** option. We tick all four reinforcement options **a_s** as seen in the figure below.



Dialog boxes *Graphic Printout* and *Mass Print*

After [OK], the pictures of the reinforcement are included in the printout report.



Mass print image in printout report

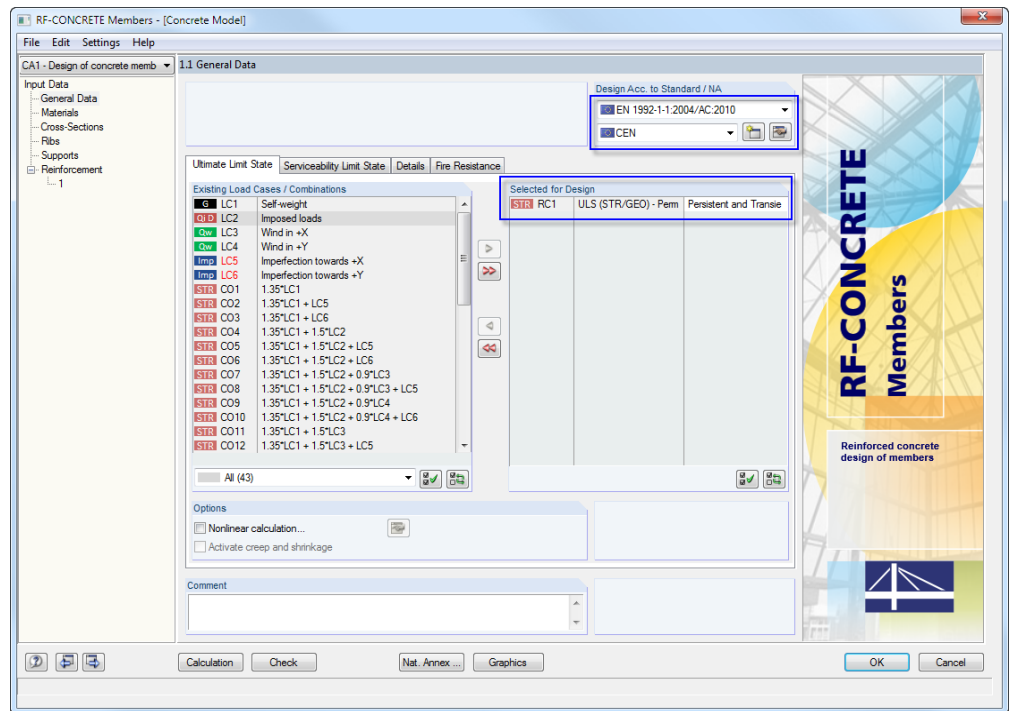
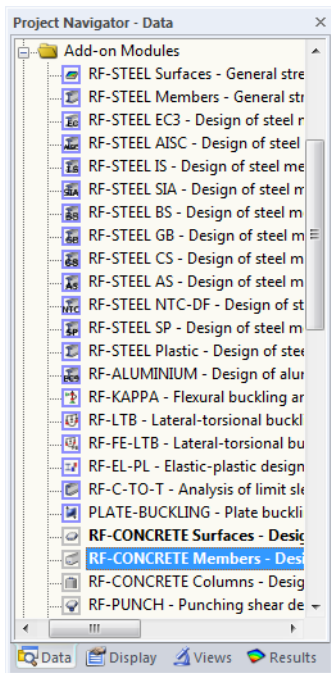
2.9 RF-CONCRETE Members

We double-click **RF-CONCRETE Members** in the **Data navigator** to open the add-on module.

2.9.1 Downstand Beam

2.9.1.1 General Data

We design the downstand beam and the columns in separate design cases.



Window RF-CONCRETE Members, 1.1 General Data, tab Ultimate Limit State

In window 1.1, we select **RC1** for the **Ultimate Limit State** design according to **EN 1992-1-1** and the **CEN** recommendations.

We neglect the SLS design for the concrete members.

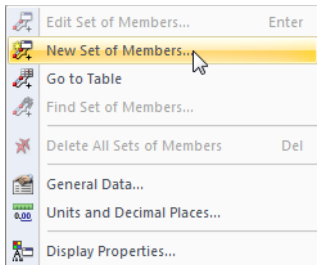
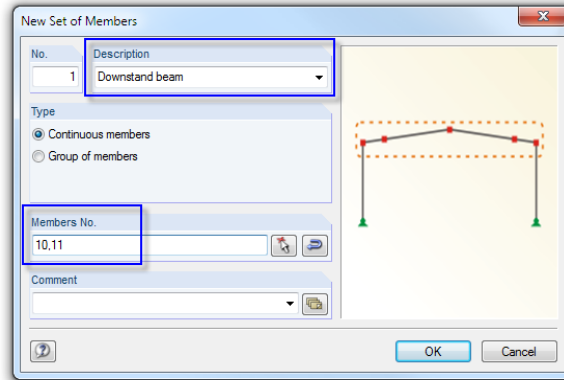
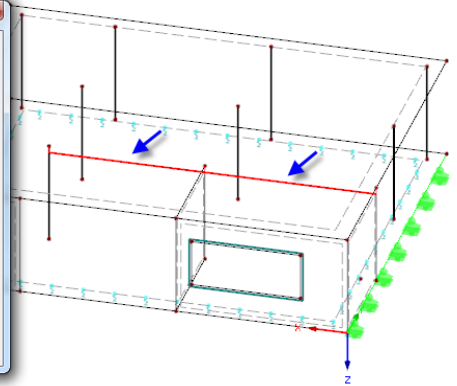
We check the settings in windows **1.2 Materials** and **1.3 Cross-Sections** but do not change them.

2.9.1.2 Ribs

Create Set of Members

To design the downstand beam as a unit, we have to create a new set of members. We close the module window by clicking [OK].

In the RFEM work window, we select members **10** and **11**. We then use the context menu of the members or of the *Data* navigator to call up the dialog box *New Set of Members*.

Context menu in *Data* navigatorDialog box *New Set of Members*

If the list of the member numbers is **11,10**, we change the orientation via the **[Reverse]** button.

We enter the *Description* **Downstand beam**.

After [OK], we open RF-CONCRETE Members again and set window **1.4 Ribs**.

Modify effective widths

1.4 Ribs

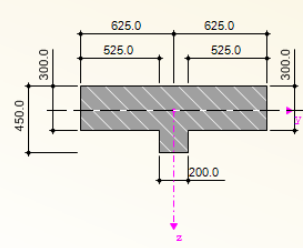
Member No.	Cross-Section No.		Surface 1	Effective Widths		Notes
	Start	End		b _{eff,1} [m]	b _{eff,2} [m]	
10	2	2	3	0.625	3	0.625
11	2	2	3	0.625	3	0.625

Rectangle 200/150

Options

☐ Show rib effective widths for determination of internal forces

FBU 450/525/525/300/300/200


Window *RF-CONCRETE Members*, 1.4 Ribs

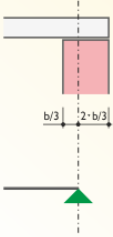
To obtain a consistent geometry for the design, we reduce the effective widths to **0.625 m** at both sides for member **10**.

2.9.1.3 Supports

For horizontal beams, the real widths of the supports are useful to apply reduced moments and forces for the design.

1.5 Supports

Support No.	A Node No.	B Support Width b (mm)	C Direct Support	D Monolithic Connection	E End Support	F M Ratio $\delta [-]$	G Comment
1	34	200.0	☒	☒	☐	1.00	
2	35	350.0	☒	☒	☒	1.00	
3	37	350.0	☒	☐	☒	1.00	
4							
5							
6							
7							
8							
9							



- ☐ Consideration of limited moment redistribution of the supporting moments according to 5.5
- ☐ Reduction of the moments or dimensioning for the moments at the face of a monolithic support according to 5.3.2.2
- ☒ Reduction of the shear forces in the support face and distance d acc. to 6.2.1 (8)
- ☒ Reduction of the shear forces with concentrated load acc. to 6.2.2 (6) and 6.2.3 (8)

Window RF-CONCRETE Members, 1.5 Supports

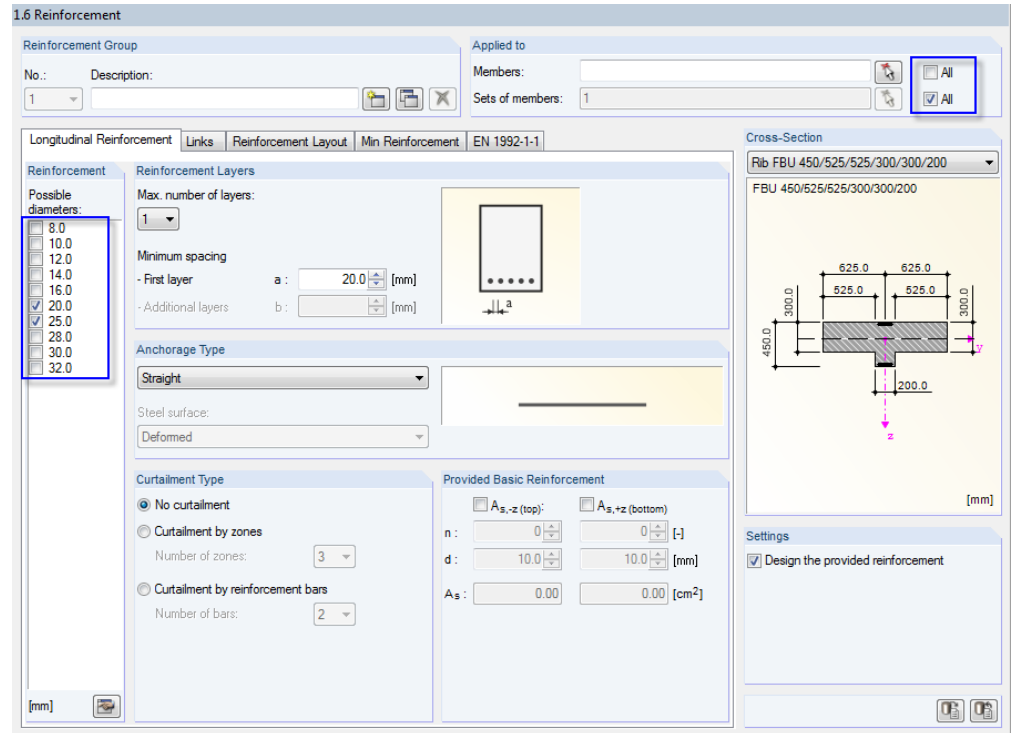


We define the support widths as shown in the figure above. The nodes can be selected graphically via the [...] button.

2.9.1.4 Reinforcement

The **1.6 Reinforcement** window contains five tabs that control the reinforcement parameters.

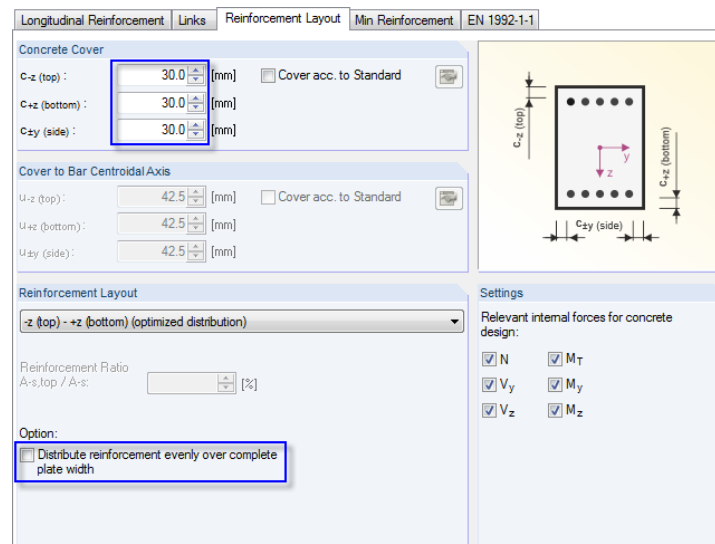
As we design only the downstand beam, we clear the selection of *All Members*.



Window 1.6 Reinforcement, tab Longitudinal Reinforcement

In the *Longitudinal Reinforcement* tab, we select the diameters **20** and **25** mm for the design.

The *Reinforcement Layout* tab controls the arrangement of the rebars.



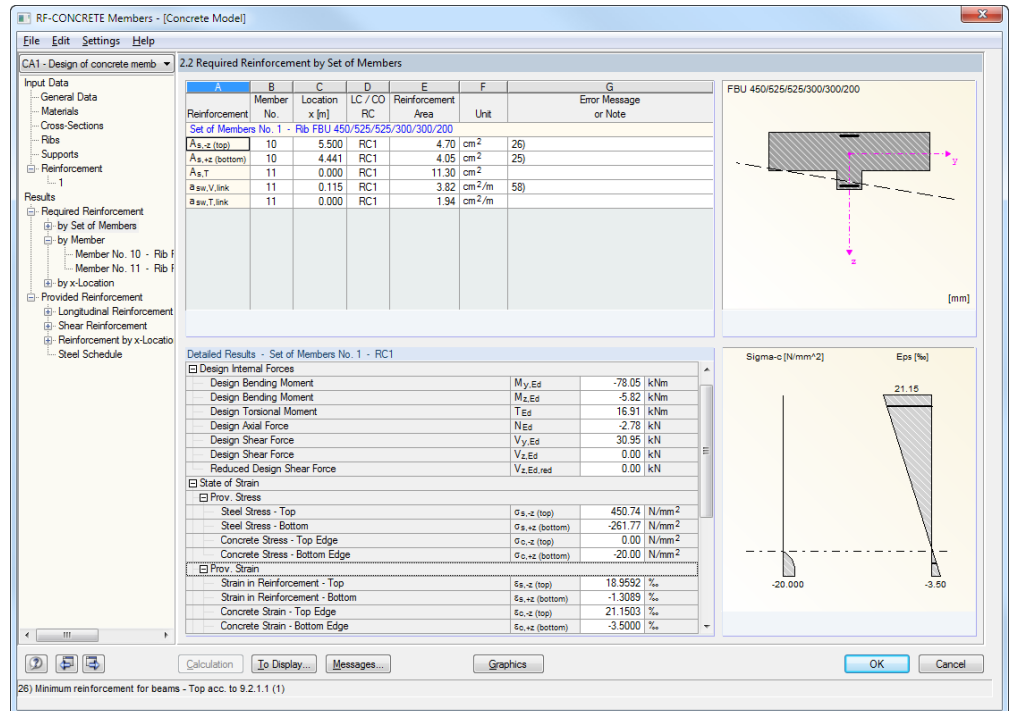
Window 1.6 Reinforcement, tab Reinforcement Layout

We make sure that the *Cover* is **30** mm for all sides. Then we disable the option *Distribute reinforcement evenly over complete plate width*.

We can check on the settings in the other tabs. Then we start the **[Calculation]**.

Calculation

2.9.1.5 Required Reinforcement



Window 2.2 Required Reinforcement by Set of Members

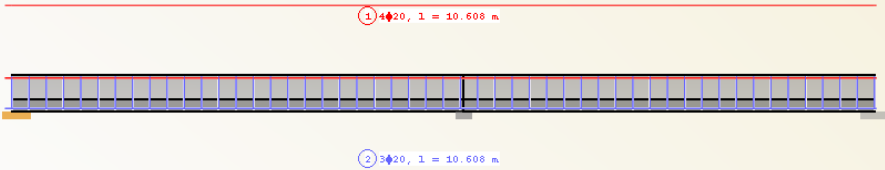
The longitudinal reinforcements **A_s** are listed for the **top** and **bottom** sides of the members. Additionally, the torsional and shear reinforcements **a_{sw}** are displayed.

2.9.1.6 Provided Reinforcement

RF-CONCRETE Members creates a reinforcement proposal which is based on the required reinforcement and the reinforcement parameters. The rebars are shown in window **3.1 Provided Longitudinal Reinforcement**.

3.1 Provided Longitudinal Reinforcement

A	B	C	D	E	F	G	H	I	J
Item No.	Reinforcement Position	No. of Bars	d _s [mm]	Length [m]	x-Location [m] from	x-Location [m] to	Anchorage	Weight [kg]	Notes
Set of Members No. 1 - Rib FBU 450/525/525/300/300/200									
1	-z (top)	4	20.0	10.608	-0.083	10.525	☒	104.65	
2	+z (bottom)	3	20.0	10.608	-0.083	10.525	☒	78.49	



Window 3.1 Provided Longitudinal Reinforcement

The parameters of each rebar group can be modified via the [Edit] button.

Dialog box *Edit Longitudinal Reinforcement* - Set of Members No. 1, Item No. 1

Location
 x from: 0.00 [m]
 to: 10.50 [m]
 Length: 10.500 [m]

Reinforcement Layout
 -z (top)

Reinforcement Diameter
 D: 20.0 [mm]

Bar Positioning
 Number of bars: 4
 Bar coordinates and hook rotation:

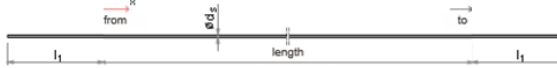
No.	Coordinates	Hook Rotation
	y [mm] z [mm]	β [°]
1	20.0 -126.7	180.0
2	60.0 -126.7	180.0
3	-60.0 -126.7	180.0
4	-20.0 -126.7	180.0

☒ Place rebars evenly

Anchorage

Anchorage Type	Bond	Anchorage and Additional Lengths	Total	Bending Diameter d _{br}
		l _{bd} l ₁ l ₂		
Start: Straight	poor	0.20 0.08	0.08 [m]	
End: Straight	poor	0.20 0.03	0.03 [m]	

Straight



Total length: 10.608 [m]
 Total weight: 104.6 [kg]

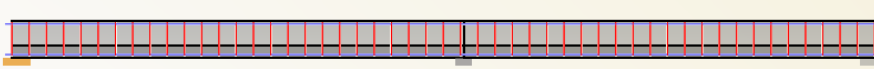
OK Cancel

Dialog box *Edit Longitudinal Reinforcement*

The links are displayed in window **3.2 Provided Shear Reinforcement**.

3.2 Provided Shear Reinforcement

A	B	C	D	E	F	G	H	I	J	K
Item No.	No. of Links	d _s [mm]	Length [m]	x-Location [m] from	to	Spacing s _{ll} [m]	Link Dimensions [mm]	Number of Legs	Weight [kg]	Notes
Set of Members No. 1 - Rib FBU 450/525/525/300/300/200										
1	51	10.0	10.500	0.000	10.500	0.210	410.0/160.0/113.1	2	42.96	

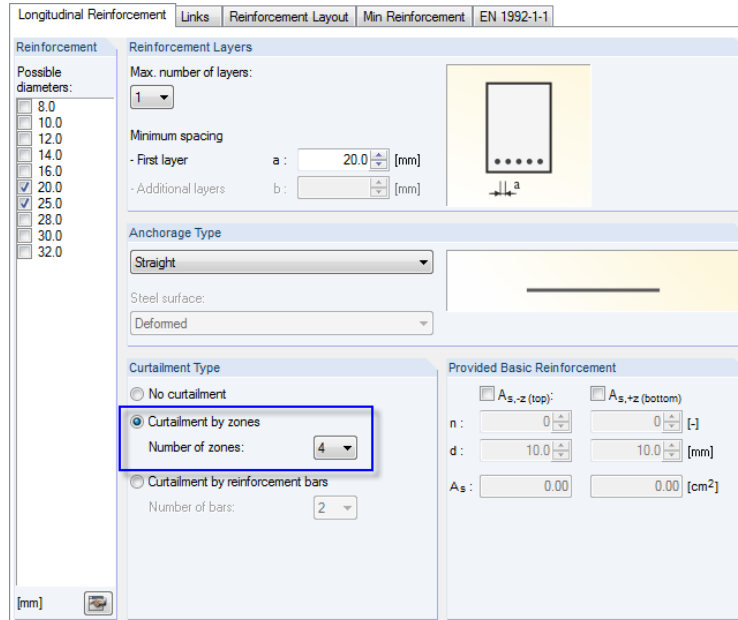


51 links 10-0.210 m

Window *3.2 Provided Shear Reinforcement*

2.9.1.7 Curtailment

We return to window **1.6 Reinforcement**, tab **Longitudinal Reinforcement**. In the dialog section **Curtailment Type**, we tick the option **Curtailment by zones** and set **4 zones**.



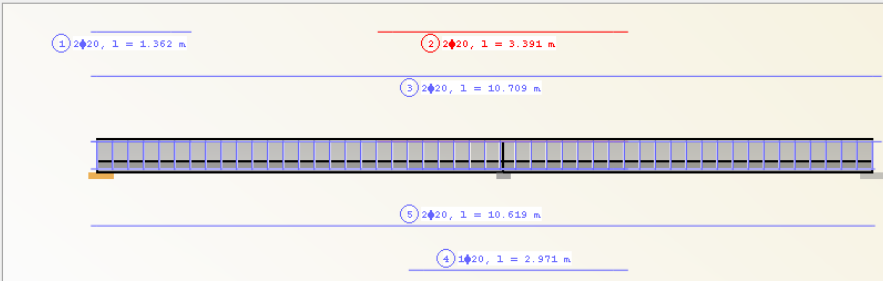
Window 1.6 Reinforcement, tab Longitudinal Reinforcement

Calculation

After the **[Calculation]**, the rebar layout is adapted to the required reinforcement.

3.1 Provided Longitudinal Reinforcement

A	B	C	D	E	F	G	H	I	J
Item No.	Reinforcement Position	No. of Bars	d _s [mm]	Length [m]	x-Location [m] from	x-Location [m] to	Anchorage	Weight [kg]	Notes
Set of Members No. 1 - Rib FB4 450/525/525/300/200									
1	-z (top)	2	20.0	1.362	-0.083	1.279	☑	6.72	
2	-z (top)	2	20.0	3.391	3.800	7.191	☑	16.73	
3	-z (top)	2	20.0	10.709	-0.083	10.626	☑	52.82	
4	+z (bottom)	1	20.0	2.971	4.220	7.191	☑	7.33	
5	+z (bottom)	2	20.0	10.619	-0.083	10.536	☑	52.38	



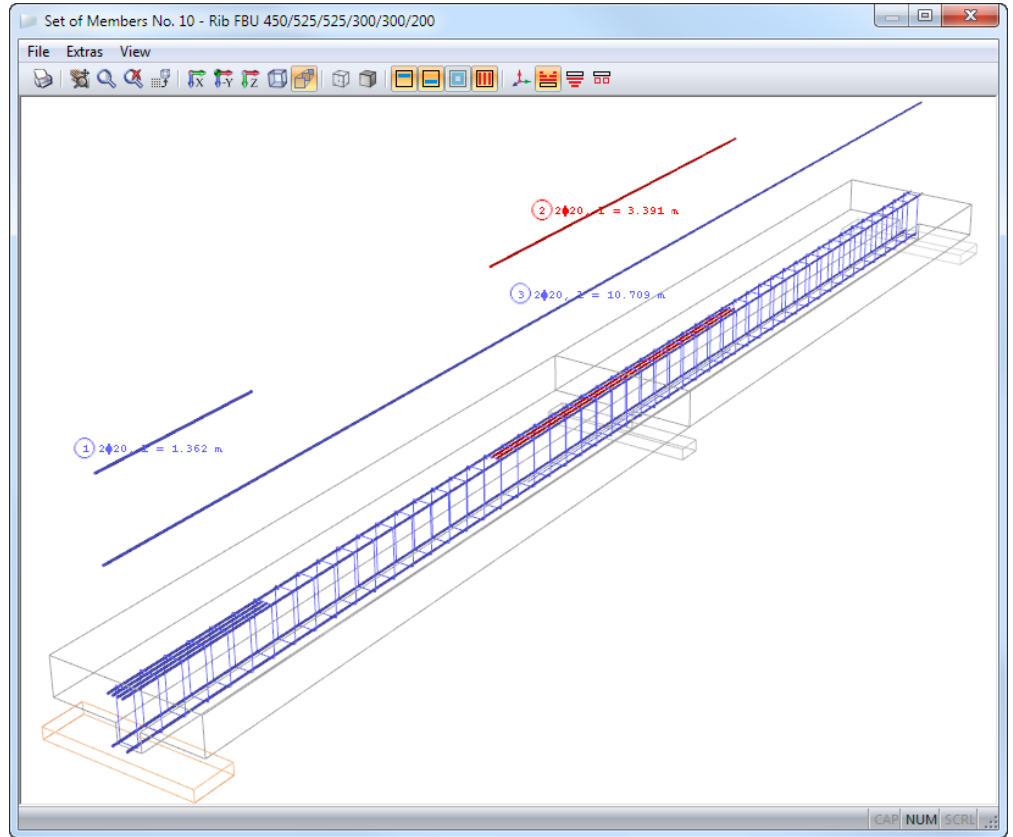
Window 3.1 Provided Longitudinal Reinforcement

2.9.1.8 Graphical Results

3D view

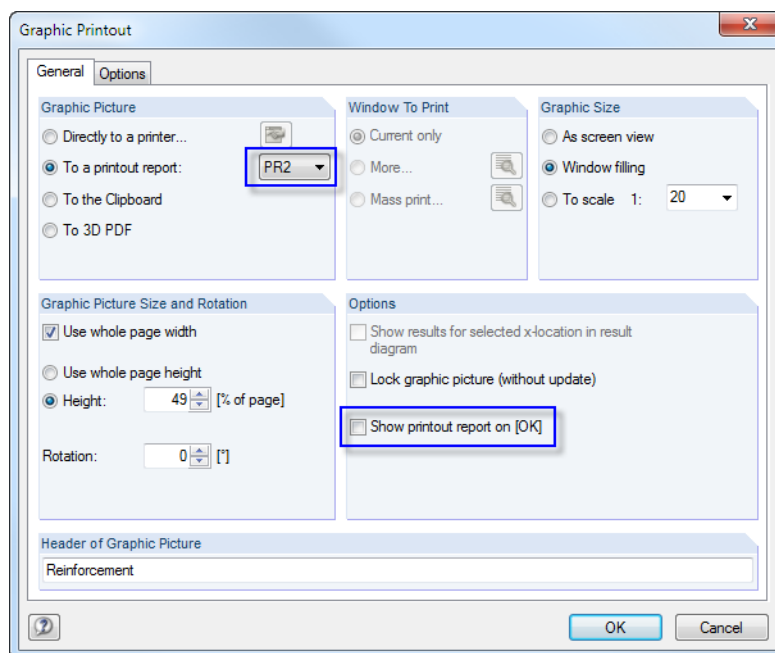
3D-Rendering

By clicking the **[3D-Rendering]** button we can display the provided reinforcement in 3D.



3D view of longitudinal reinforcement

We print this view to the printout report **PR2**, but without opening the printout report on [OK].

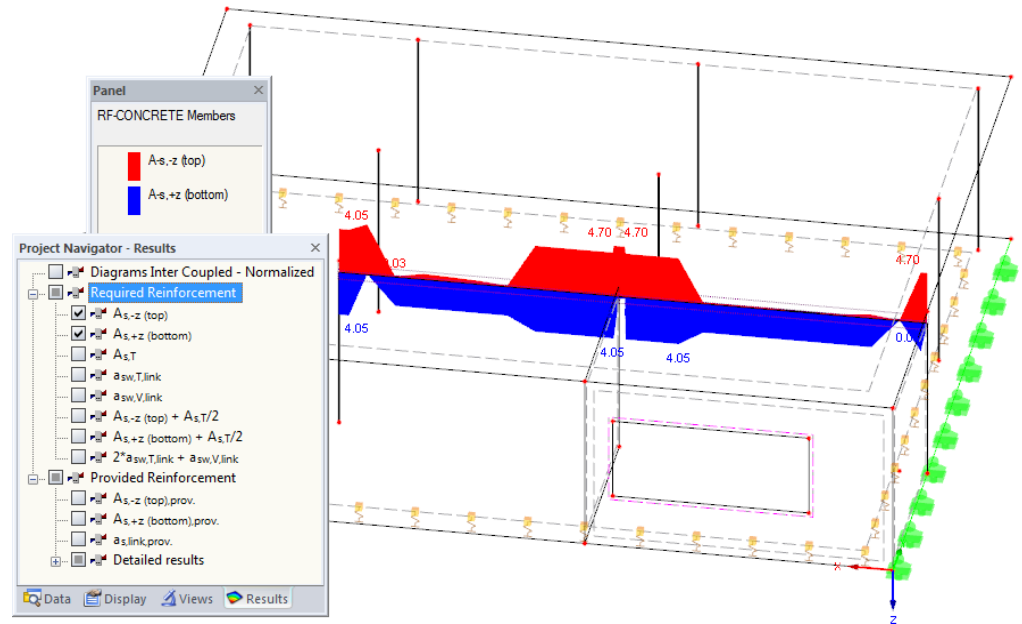
Dialog box *Graphic Printout*

Graphics

RFEM work window

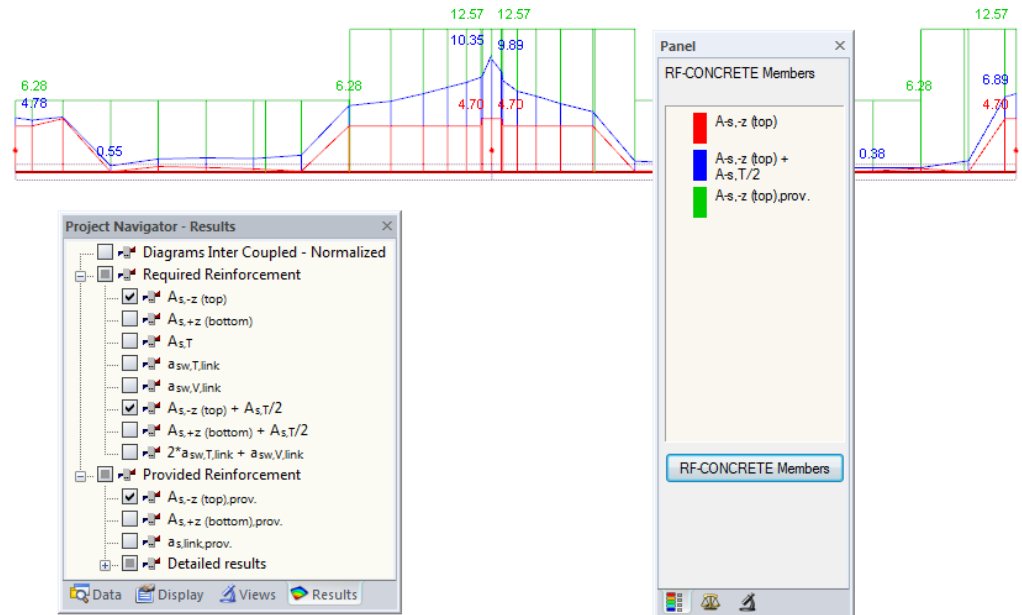
We use the **[Graphics]** button to evaluate the design results in the work window of RFEM.

In the **Result navigator**, we set the required reinforcement $A_{s,z}(\text{top})$ and $A_{s,z}(\text{bottom})$.



Required top and bottom reinforcement

We can also display the shear reinforcement or contrast the required reinforcement $A_{s,z}(\text{top})$ and $A_{s,z}(\text{top}) + A_{s,T}/2$ with the provided reinforcement $A_{s,z}(\text{top}).\text{prov.}$



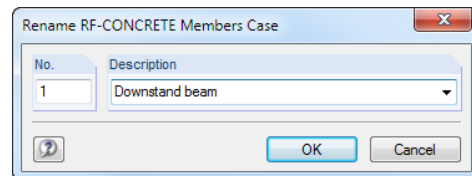
Required reinforcement and provided reinforcement

2.9.2 Columns

Eurocode 2 provides three methods for the design of columns: The simplified *method based on nominal curvature* (clause 5.8.8), the *general method* (clause 5.8.6) based on a nonlinear analysis including second order effects, and the *method based on nominal stiffness* (clause 5.8.7). For our example, we apply the general method, i.e. we do not have to use RF-CONCRETE Columns.

Rename case

On the *File* menu, we click *Rename Case*.

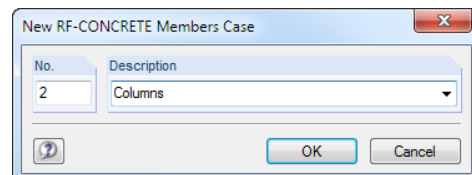


Dialog box *Rename RF-CONCRETE Members Case*

We enter **Downstand beam** as new description of the first design case.

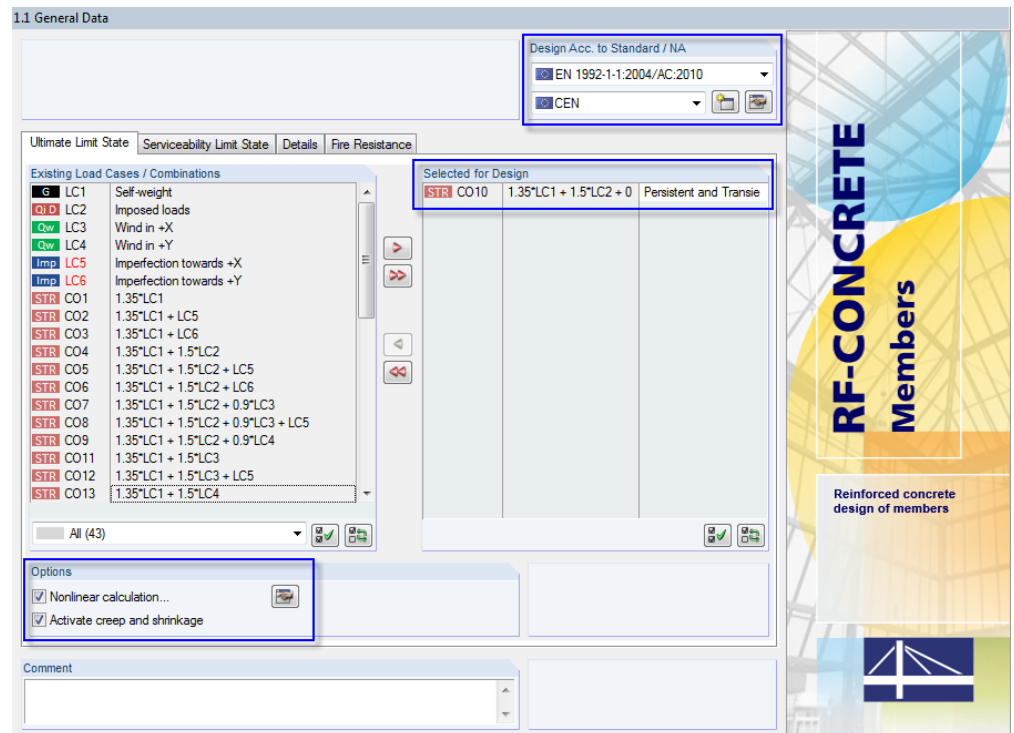
Create new case

On the *File* menu, we click *New Case*. We then enter **Columns** to describe the second case.



Dialog box *New RF-CONCRETE Members Case*

2.9.2.1 General Data

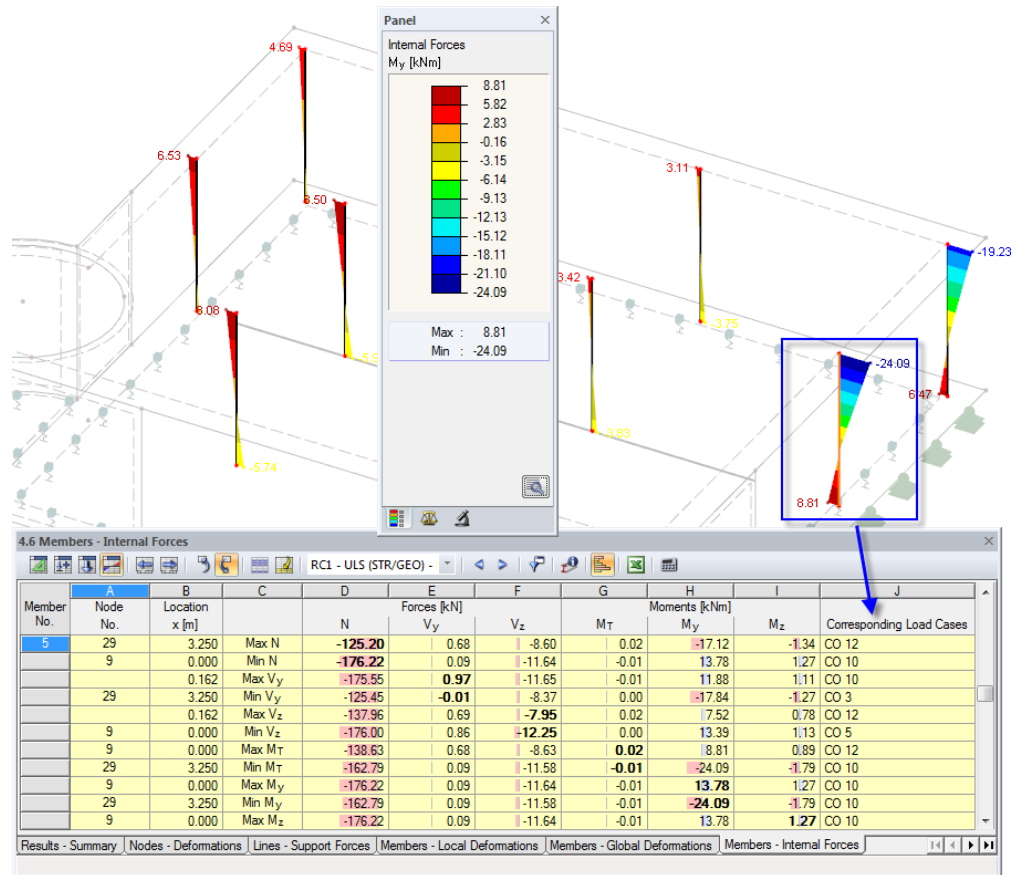


Window *RF-CONCRETE Members*, 1.1 General Data, tab *Ultimate Limit State*



In window 1.1, we select **CO10** for the **Ultimate Limit State** design according to **EN 1992-1-1** and the **CEN** recommendations. We have to specify a load combination now because a non-linear design is not possible for result combinations. CO10 is governing for the columns (see figure below).

We check the options **Nonlinear calculation** and **Activate creep and shrinkage**.

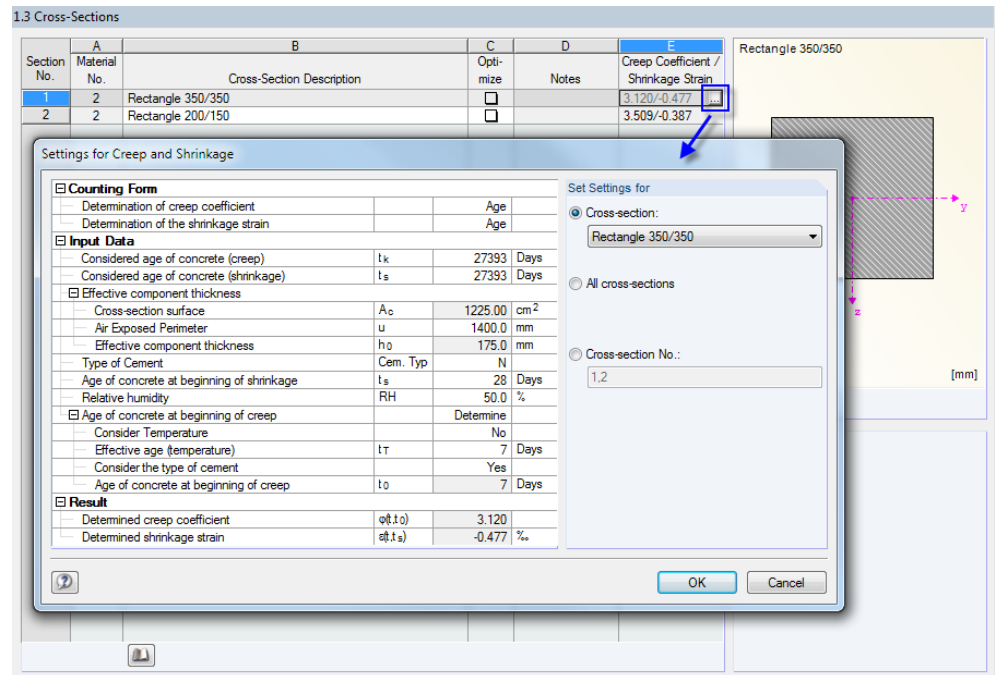


Internal forces and governing load combinations of RC1

Again, there is no SLS design.

2.9.2.2 Cross-Sections

In window **1.3 Cross-Sections**, we can check on the settings for creep and shrinkage by clicking the [...] button. We do not change them, however.

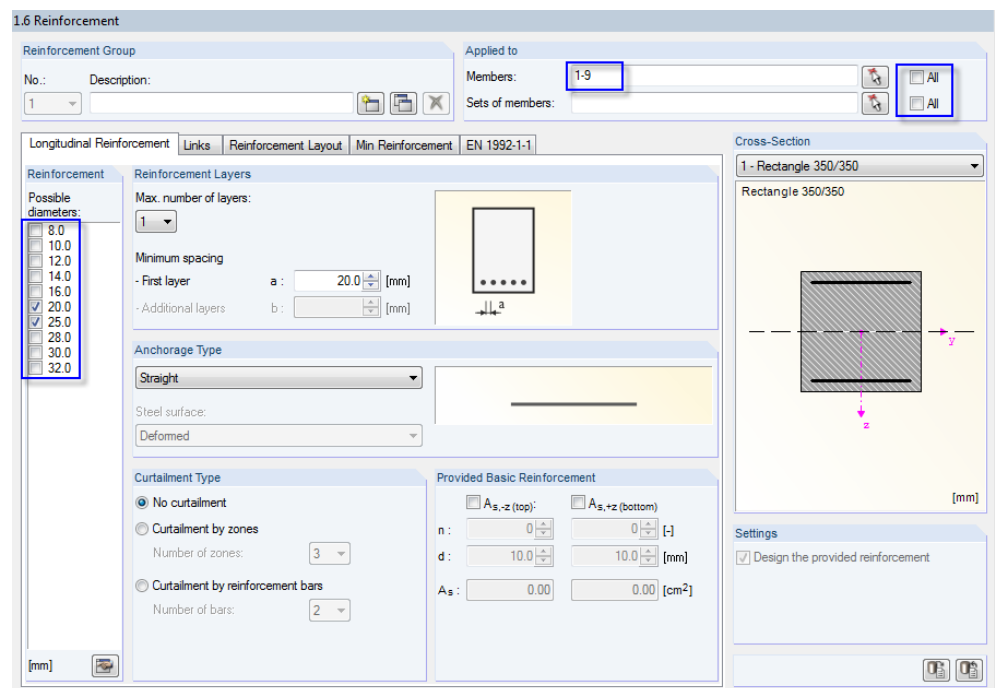


Window 1.3 Cross-Sections, dialog box Settings for Creep and Shrinkage

Windows **1.4 Ribs** and **1.5 Supports** are not relevant for the column design.

2.9.2.3 Reinforcement

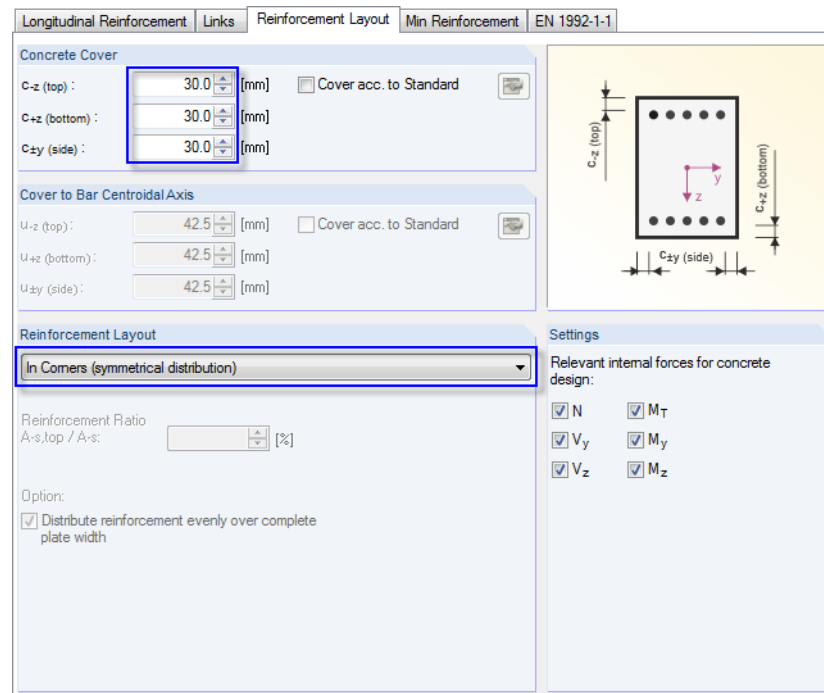
As we design only the columns, we enter the member numbers **1-9**.



Window 1.6 Reinforcement, tab Longitudinal Reinforcement

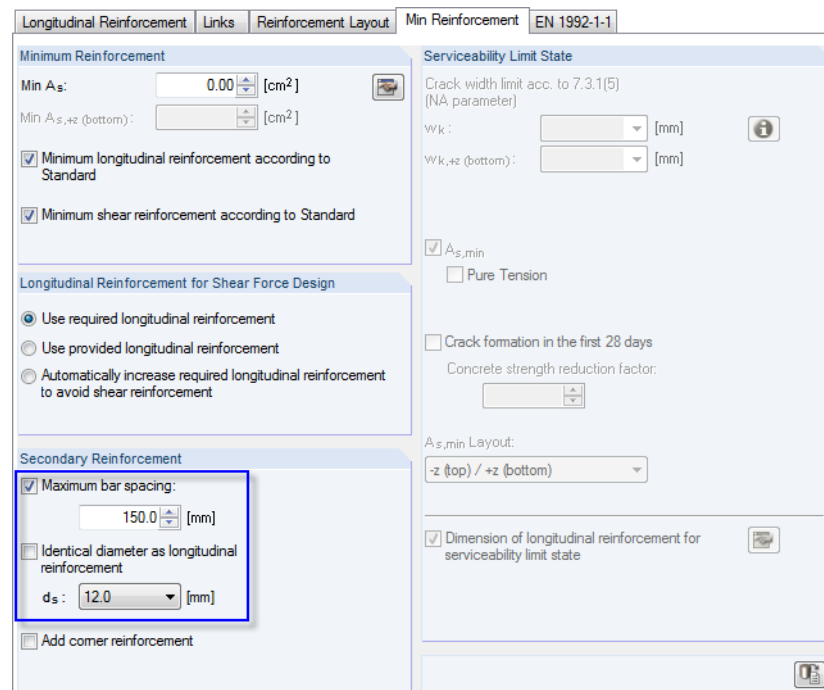
In the *Longitudinal Reinforcement* tab, we select the diameters **20** and **25** mm for the design.

The *Reinforcement Layout* tab controls the arrangement of the rebars. The *Cover* is **30 mm** for all sides again. We set the layout option **In corners (symmetrical distribution)**.



Window 1.6 Reinforcement, tab Reinforcement Layout

In the *Min Reinforcement* tab, we define a maximum bar spacing of **150 mm**. The diameter of the secondary reinforcement is **12 mm**.



Window 1.6 Reinforcement, tab Min Reinforcement

Calculation

Then we start the **[Calculation]**. The nonlinear design requires several load steps and iterations.

2.9.2.4 Required Reinforcement

For all columns, the required reinforcement $A_{s,total}$ can be covered by the minimum reinforcement for compression members.

2.3 Required Reinforcement by Member

A	B	C	D	E	F	G
Reinforcement	Member No.	Location x [m]	LC / CO RC	Reinforcement Area	Unit	Error Message or Note
Member No. 5 - Rectangle 350/350						
$A_{s,total}$	5	0.000	CO10	2.45	cm ²	22) 25)
$A_{s,T}$	5	3.250	CO10	0.01	cm ²	
$A_{s,w,V,link}$	5	0.000	CO10	0.00	cm ² /m	58) 933)
$A_{s,w,T,link}$	5	3.250	CO10	0.00	cm ² /m	
Member No. 6 - Rectangle 350/350						
$A_{s,total}$	6	0.000	CO10	2.45	cm ²	22) 25)
$A_{s,T}$	6	0.000	CO10	0.01	cm ²	
$A_{s,w,V,link}$	6	0.000	CO10	0.00	cm ² /m	58) 933)
$A_{s,w,T,link}$	6	0.000	CO10	0.00	cm ² /m	
Member No. 7 - Rectangle 350/350						
$A_{s,total}$	7	0.000	CO10	2.45	cm ²	22) 25)
$A_{s,T}$	7	0.000	CO10	0.01	cm ²	

Detailed Results - Member No. 5 - Rectangle 350/350 - CO10

Reinforcement

Design Internal Forces

Design Bending Moment	$M_{y,Ed}$	13.78	kNm
Design Bending Moment	$M_{z,Ed}$	1.27	kNm
Design Torsional Moment	T_{Ed}	-0.01	kNm
Design Axial Force	N_{Ed}	-176.22	kN
Design Shear Force	$V_{y,Ed}$	0.09	kN
Design Shear Force	$V_{z,Ed}$	-11.64	kN
Reduced Design Shear Force	$V_{z,Ed,red}$	-11.64	kN

State of Strain

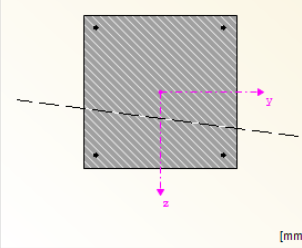
Prov. Stress

Steel Stress - Top	$\sigma_{s,-z} (top)$	-435.46	N/mm ²
Steel Stress - Bottom	$\sigma_{s,+z} (bottom)$	-	
Concrete Stress - Top Edge	$\sigma_{c,-z} (top)$	-20.00	N/mm ²
Concrete Stress - Bottom Edge	$\sigma_{c,+z} (bottom)$	0.00	N/mm ²

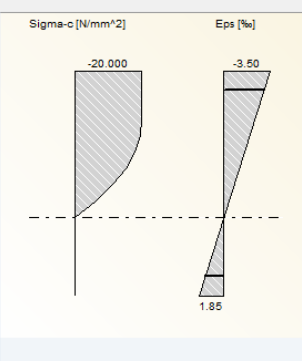
Prov. Strain

Strain in Reinforcement - Top	$\epsilon_{s,-z} (top)$	-2.8888	‰
Strain in Reinforcement - Bottom	$\epsilon_{s,+z} (bottom)$	-	
Concrete Strain - Top Edge	$\epsilon_{c,-z} (top)$	-3.5000	‰

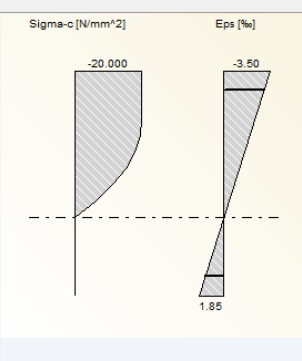
Rectangle 350/350



Sigma-c [N/mm²]



Eps [‰]



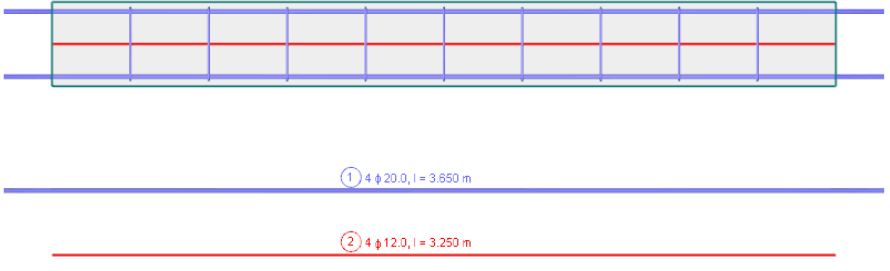
Window 2.3 Required Reinforcement by Member

2.9.2.5 Provided Reinforcement

The reinforcement proposal includes the reinforcement in the corners and the secondary reinforcement at the cross-section sides.

3.1 Provided Longitudinal Reinforcement

A	B	C	D	E	F	G	H	I	J	K
Item No.	Reinforcement Position	No. of Bars	d_s [mm]	A_s [cm ²]	Length [m]	x-Location [m] from	x-Location [m] to	Anchorage	Weight [kg]	Notes
Member No. 1 - Rectangle 350/350										
1	Corners	4	20.0	12.57	3.650	-0.200	3.450	☒	36.01	
2	Secondary	4	12.0	4.52	3.250	0.000	3.250	☐	11.54	
Member No. 2 - Rectangle 350/350										
1	Corners	4	20.0	12.57	3.650	-0.200	3.450	☒	36.01	
2	Secondary	4	12.0	4.52	3.250	0.000	3.250	☐	11.54	
Member No. 3 - Rectangle 350/350										
1	Corners	4	20.0	12.57	3.650	-0.200	3.450	☒	36.01	
2	Secondary	4	12.0	4.52	3.250	0.000	3.250	☐	11.54	
Member No. 4 - Rectangle 350/350										
1	Corners	4	20.0	12.57	3.650	-0.200	3.450	☒	36.01	
2	Secondary	4	12.0	4.52	3.250	0.000	3.250	☐	11.54	
Member No. 5 - Rectangle 350/350										
1	Corners	4	20.0	12.57	3.650	-0.200	3.450	☒	36.01	
2	Secondary	4	12.0	4.52	3.250	0.000	3.250	☐	11.54	
Member No. 6 - Rectangle 350/350										



① 4 ϕ 20.0, l = 3.650 m

② 4 ϕ 12.0, l = 3.250 m

Window 3.1 Provided Longitudinal Reinforcement

2.9.2.6 Nonlinear Calculation

The results of the nonlinear analysis are listed in windows **6.1 Ultimate Limit State for Nonlinear Calculation** and **6.2 Design Details**.

6.1 Ultimate Limit State for Nonlinear Calculation					
A	B	C	D	E	F
Member No.	Location x [m]	LC No.	Safety factor γ [-]	Utilization $1/\gamma$ [-]	Notes
1	0.325	CO10	24.036	0.042	
	0.488	CO10	26.000	0.038	
	0.650	CO10	28.071	0.036	
	0.975	CO10	32.440	0.031	
	1.300	CO10	34.341	0.029	
	1.625	CO10	30.481	0.034	
	1.950	CO10	26.054	0.038	
	2.275	CO10	21.896	0.046	
	2.600	CO10	18.311	0.055	
	2.763	CO10	16.793	0.060	
	2.925	CO10	15.454	0.065	
	3.088	CO10	14.284	0.070	
	3.250	CO10	13.262	0.075	
Detailed Results - Member No. 1, x: 3.250, CO10					
Ultimate Internal Forces R _{Ed}					
Ultimate Moment About Axis y		M _{yRd}	-167.71	kNm	
Ultimate Moment About Axis z		M _{zRd}	-84.58	kNm	
Ultimate Normal Force		N _{Rd}	-1295.96	kN	
Stress and Strain Analysis					
Ultimate Curvature		1/r _{Rd}	1.4181E-02	1/m	
Ultimate Curvature in y-Direction		1/r _{yRd}	-7.0074E-03	1/m	
Ultimate Curvature in z-Direction		1/r _{zRd}	-1.2329E-02	1/m	
Direction of the Principal Curvature		α_{Rd}	-150.4	°	
Design Details					
Steel Stress in Governing Fiber		σ_{sRd}	403.54	N/mm ²	
Steel Strain in Governing Fiber		ϵ_{sRd}	2.018	‰	
Concrete Stress in Governing Fiber		σ_{cRd}	-25.50	N/mm ²	
Concrete Strain in Governing Fiber		ϵ_{cRd}	-3.500	‰	
Assessed Concrete Residual Tensile Strength		σ_{ctRd}	0.00	N/mm ²	
Maximum Concrete Tensile Strain		ϵ_{ctRd}	3.268	‰	
Design Internal Forces R _{Ed} (γ _R = 1.30)					
Bending Moment About the y-Axis		M _{yEd}	-12.65	kNm	

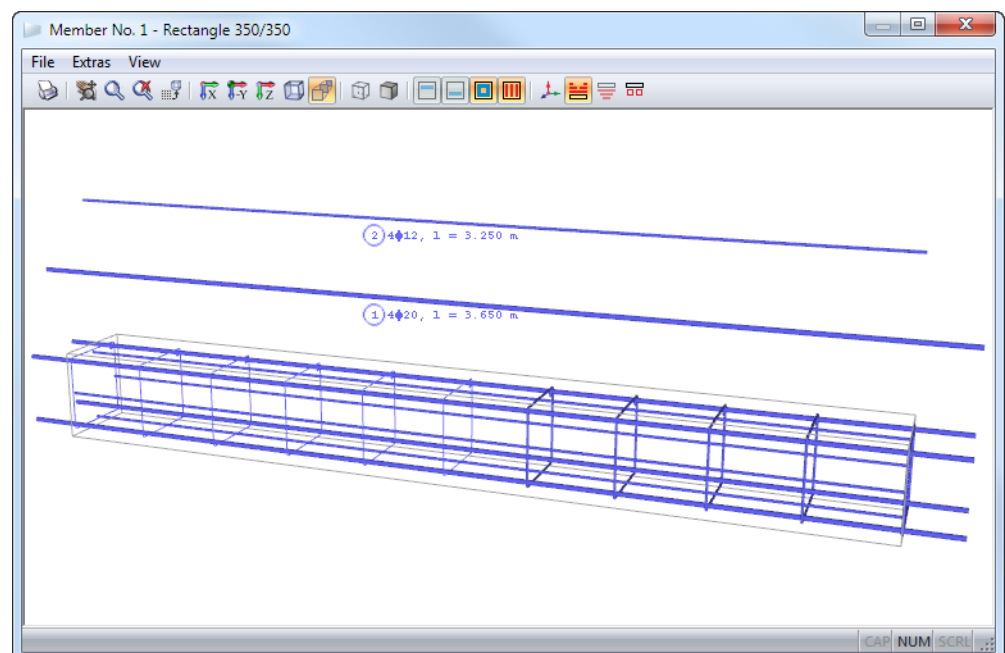
Window 6.1 Ultimate Limit State for Nonlinear Calculation

2.9.2.7 Graphical Results

3D view

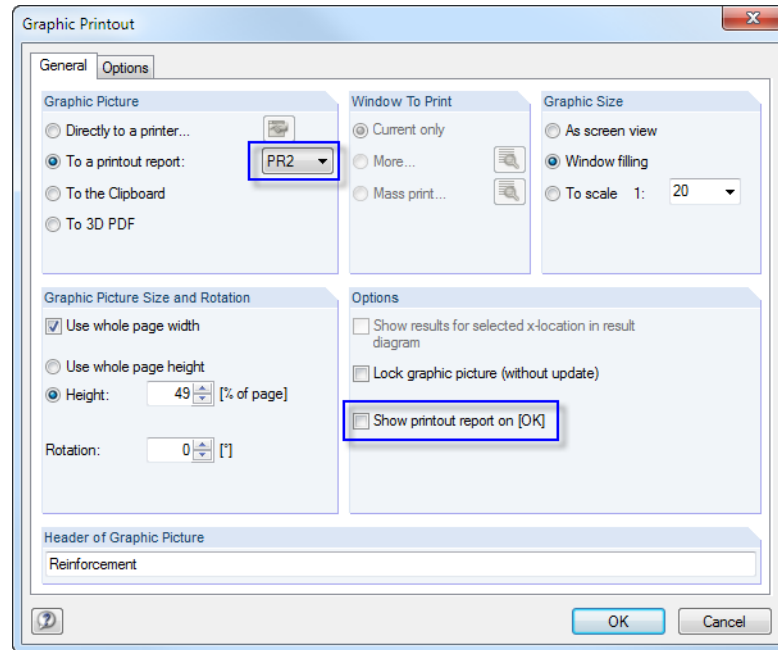
3D-Rendering

We return to window **3.1 Provided Longitudinal Reinforcement** and use the **[3D-Rendering]** button again to display the provided reinforcement in 3D.



3D view of reinforcement

We print this view to the printout report **PR2**, but without opening the printout report on [OK].

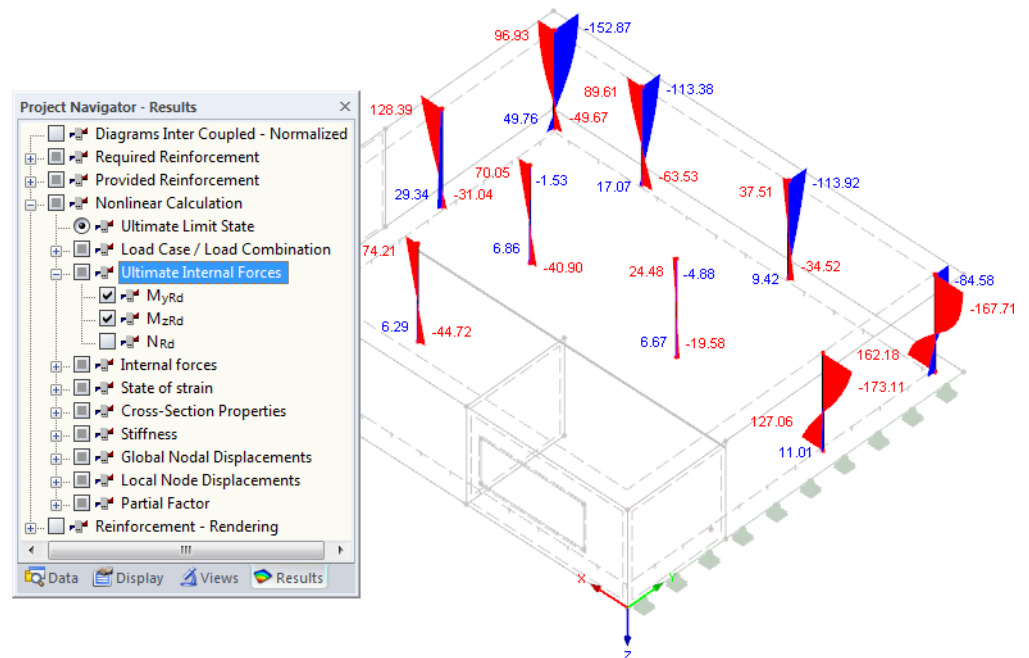


Dialog box *Graphic Printout*

RFEM work window

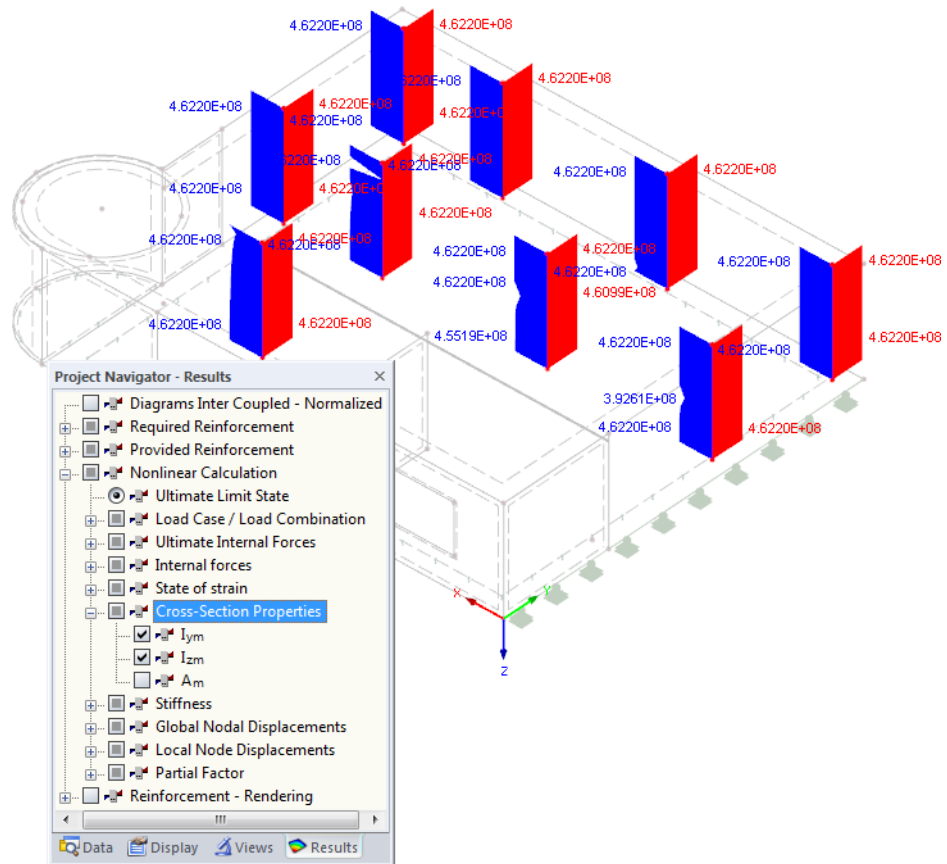
We use the **[Graphics]** button to evaluate the design results in the work window of RFEM.

In the **Result navigator**, we set the ultimate internal forces M_{yRd} and M_{zRd} of the nonlinear calculation.



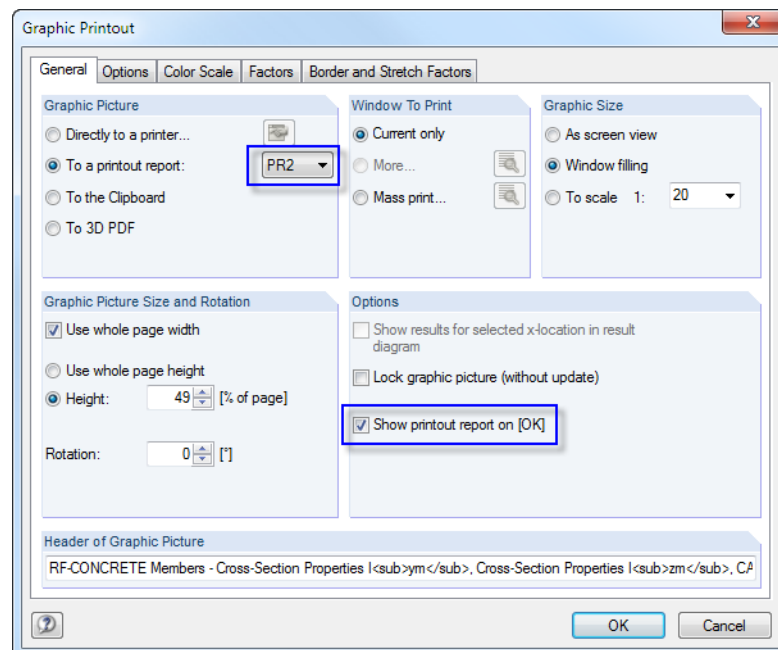
Ultimate moments of nonlinear calculation

We can also display the mean moments of inertia I_{ym} and I_{zm} of the columns that are used for the nonlinear design.



Mean moments of inertia of nonlinear calculation

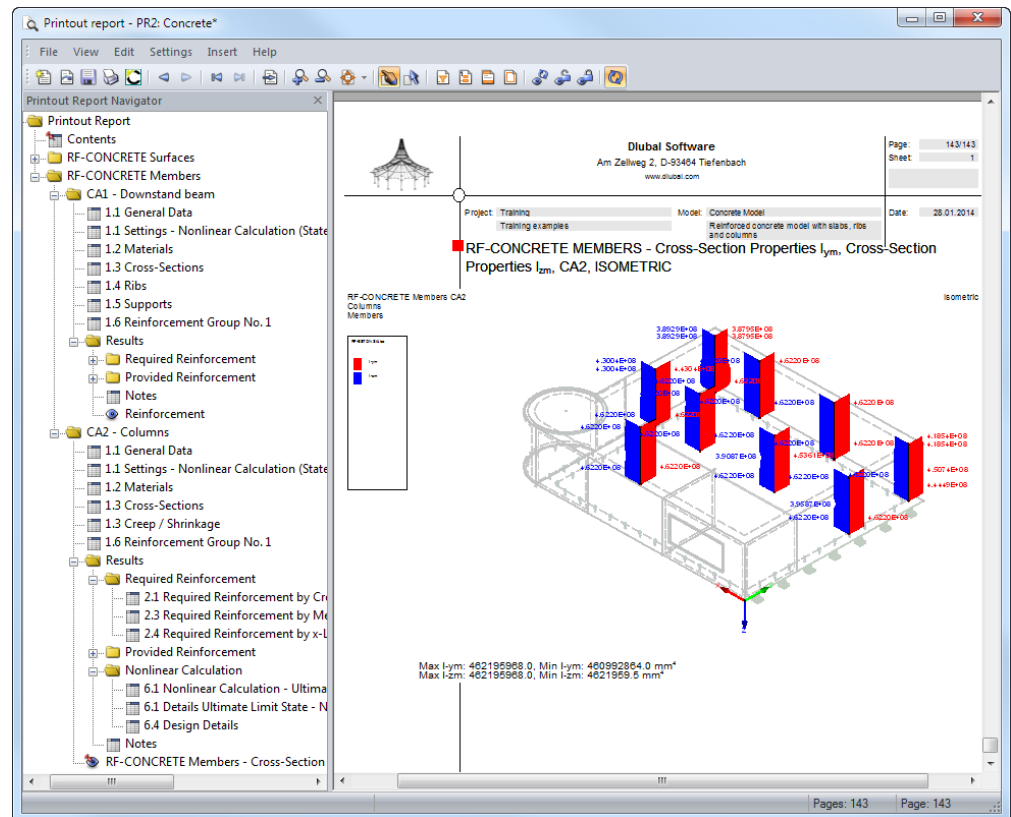
We print this view to the printout report **PR2**, but now showing the printout report on [OK].



Dialog box *Graphic Printout*

2.9.2.8 Printout

The printout report PR2 is opened. It contains all data of the concrete design, including the two 3D-rendering graphics of the *Reinforcement*.



Graphic in printout report PR2

We can check on the data of the two design cases.



Dlubal Software
 Am Zellweg 2, D-93464 Tiefenbach
 www.dlubal.com

Project: Training
 Model: Concrete Model
 Reinf. concrete model with slabs, ribs and columns

Page: 10/143
 Sheet: 1
 Date: 28.01.2014

3.1.1 PROVIDED LONGITUDINAL REINFORCEMENT - ANCHORAGE

Item No.	Anchorage Type	Bond	l _a [m]	l ₁ [m]	l ₂ [m]	Total [m]	d _{sr} [m]
Set of Members No. 1 - Continuous Members 1							
1 Start	Straight	poor	0.200	0.083	-	0.083	-
1 End	Straight	poor	0.279	0.279	-	0.279	-
2 Start	Straight	poor	0.200	0.200	-	0.200	-
2 End	Straight	poor	0.200	0.200	-	0.200	-
3 Start	Straight	poor	0.200	0.083	-	0.083	-
3 End	Straight	poor	0.301	0.126	-	0.126	-
4 Start	Straight	good	0.200	0.200	-	0.200	-
4 End	Straight	good	0.200	0.200	-	0.200	-
5 Start	Straight	good	0.200	0.083	-	0.083	-
5 End	Straight	good	0.211	0.036	-	0.036	-

RF-CONCRETE MEMBERS - SET OF MEMBERS NO. 1 - CONTINUOUS MEMBERS 1

2#20, l = 1.362 m

② 2#20, l = 3.391 m

③ 2#20, l = 10.709 m



⑤ 2#20, l = 10.619 m

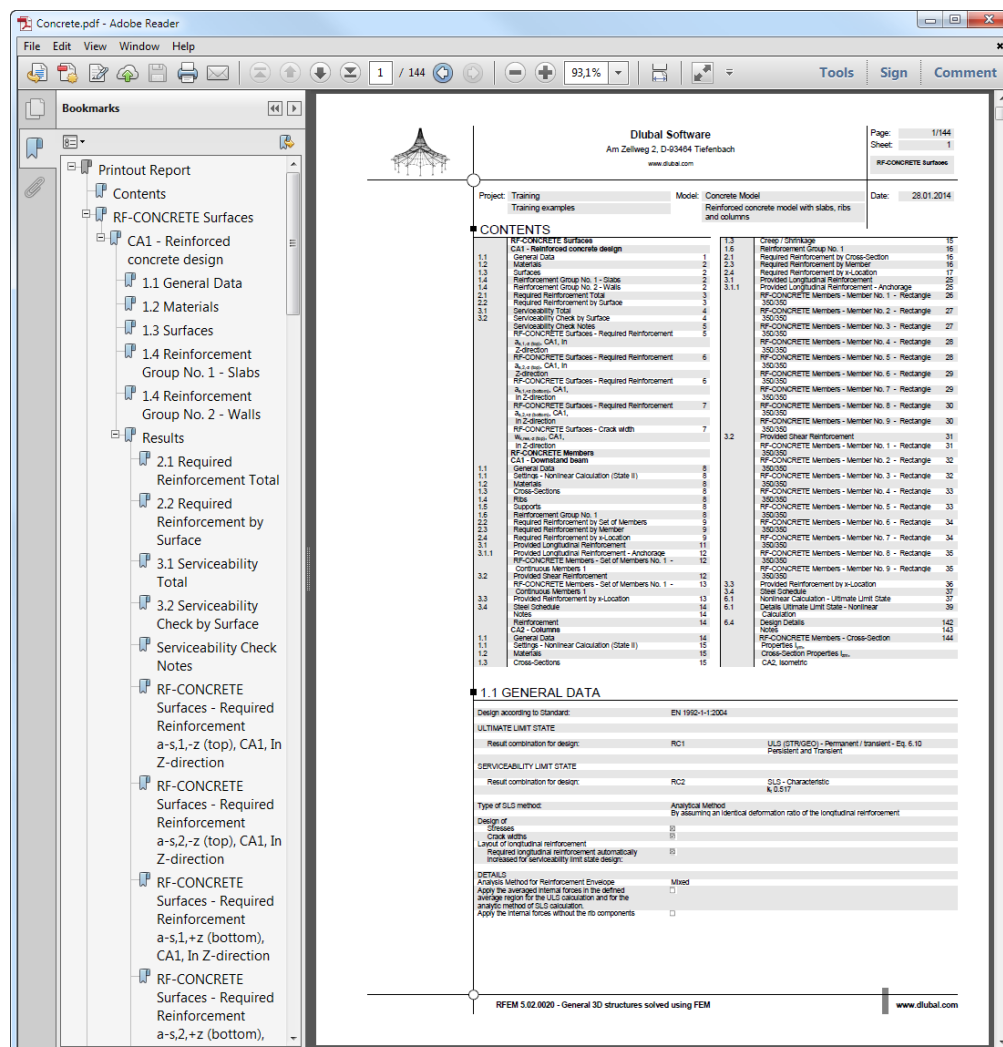
④ 1#20, l = 2.971 m

Provided reinforcement of downstand beam



Create PDF file

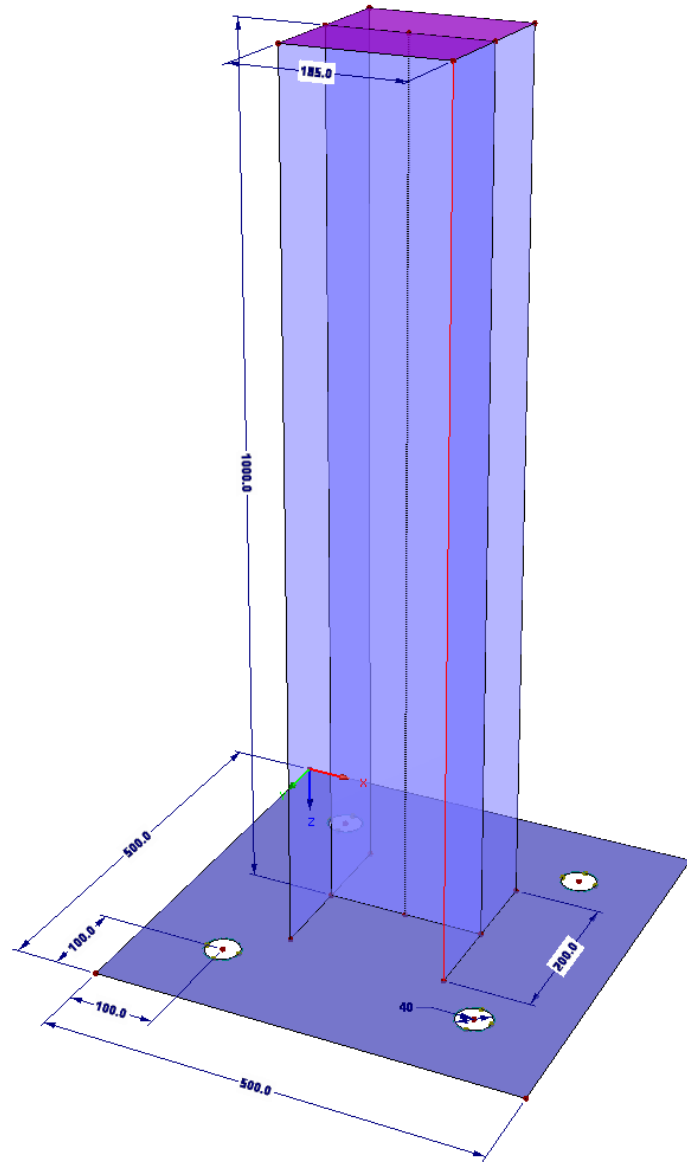
We now create a PDF file of the concrete data. The function **Export to PDF** is available on the **File** menu.



PDF of concrete data

3. Steel Model

We will analyze the steel column base shown in the following figure. At the end of this example, we will perform a design by means of the add-on module RF-STEEL Surfaces.



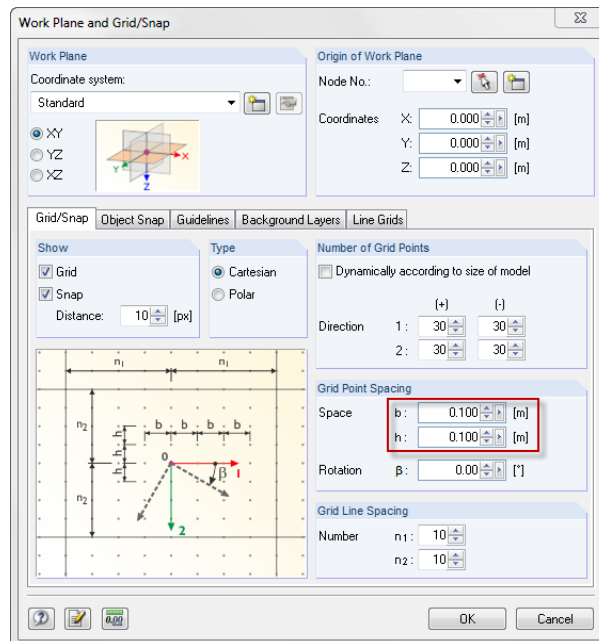
Column base with dimensions

3.1 Model

We create a new model file as described in chapter 1.3.2. This time, however, we disable the option *Create combinations automatically*.

3.1.1 Settings

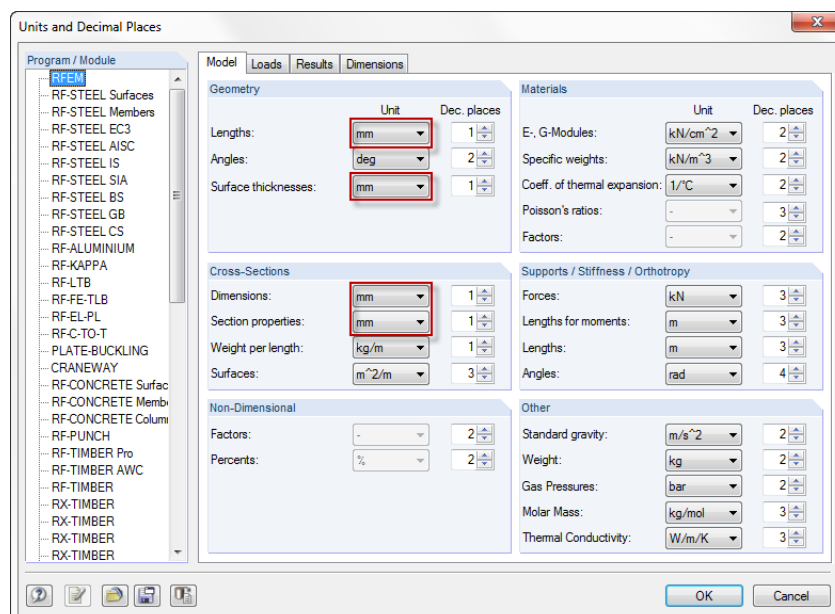
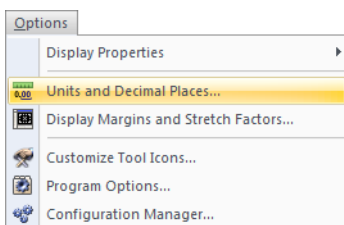
For the design of steel models, it is useful to refine the grid points to **0.1 m**. We open the **Work Plane and Grid/Snap** dialog box by clicking the toolbar button [Settings of Work Plane, Grid/Snap, Object Snap, Guide Lines].



Dialog box *Work Plane and Grid/Snap*

We set the units commonly used in the design of steel structures. To do this, we open the **Units and Decimal Places** dialog box by using the **Options** menu.

Here we set the unit **mm** for the lengths, surface thicknesses, dimensions and sections.



Dialog box *Units and Decimal Places*

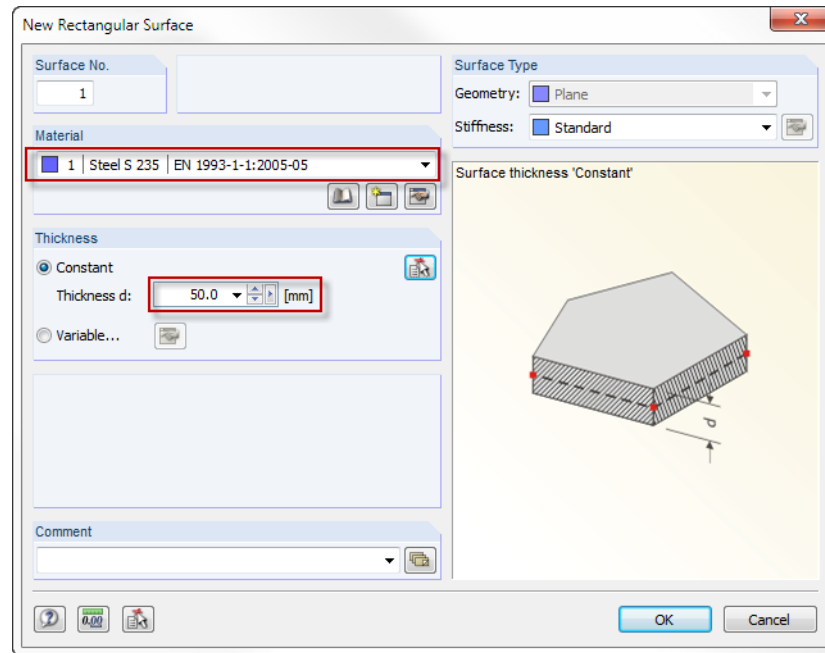
3.1.2 Base Plate

Rectangular surface

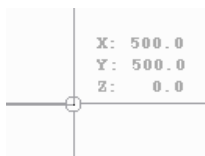


We open the dialog for the input of plate properties by clicking **[New Rectangular Surface]**.

In the **New Rectangular Surface** dialog box, we select **Steel S 235**. We keep the thickness type as **Constant**, set the value of the surface thickness to **50 mm** and then click **[OK]**.



Dialog box *New Rectangular Surface*

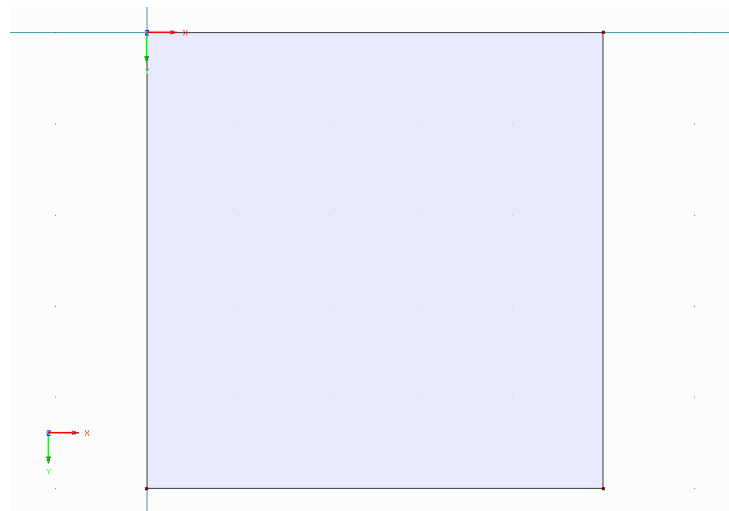


The mouse pointer becomes a reticle. For a better view, we click **[View in Z direction]**.

We move the reticle to the point **0.0; 0.0; 0.0** and click it, thus setting the first corner of the plate. We next place the reticle on the point **500.0; 500.0; 0.0** and click it.

Now we have set the rectangular surface and can quit the input function by right-clicking anywhere in the work window.

When we click **[Show Whole Model]** or **[F8]**, the rectangular surface should look like this.



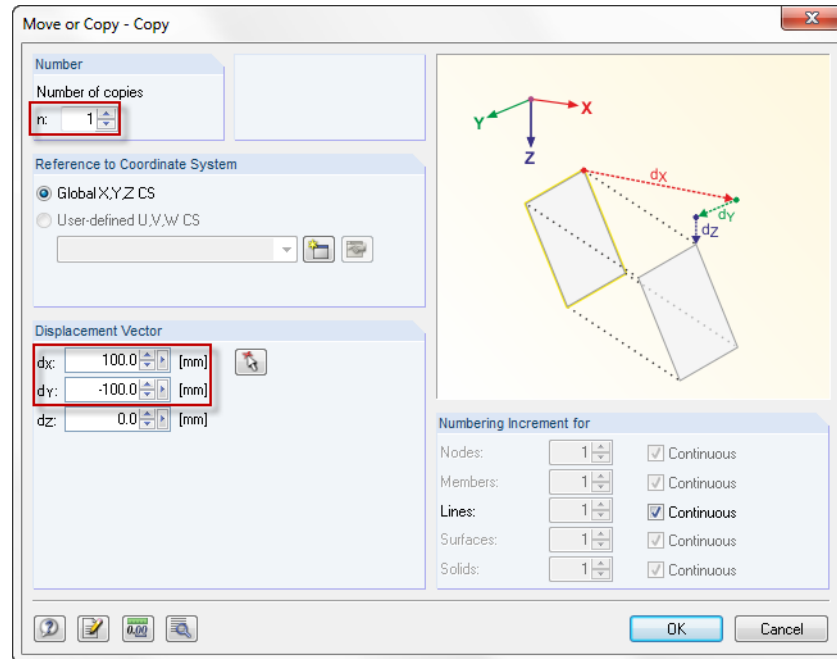
Base plate

Fastener holes

We copy the lower left node of the plate and then move it by **100 mm** in **X-direction** and by **-100 mm** in **Y-direction**. This node will be the center point of the fastener opening.

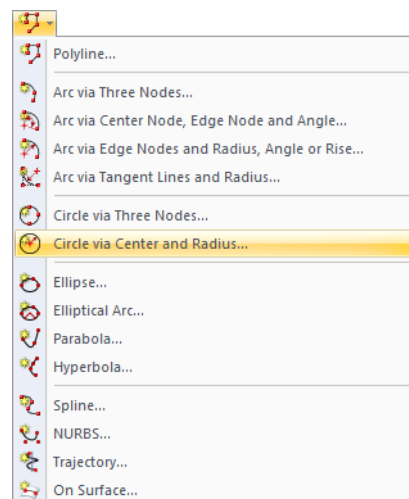


We select this node by clicking it. Then we click **[Move and/or Copy]** to open the **Move or Copy** dialog box. We set the **Number of copies** as **1**, because we want to copy the node. In the **Displacement Vector** section, we enter the value **100** for d_x , and **-100** for d_y , and then click [OK].



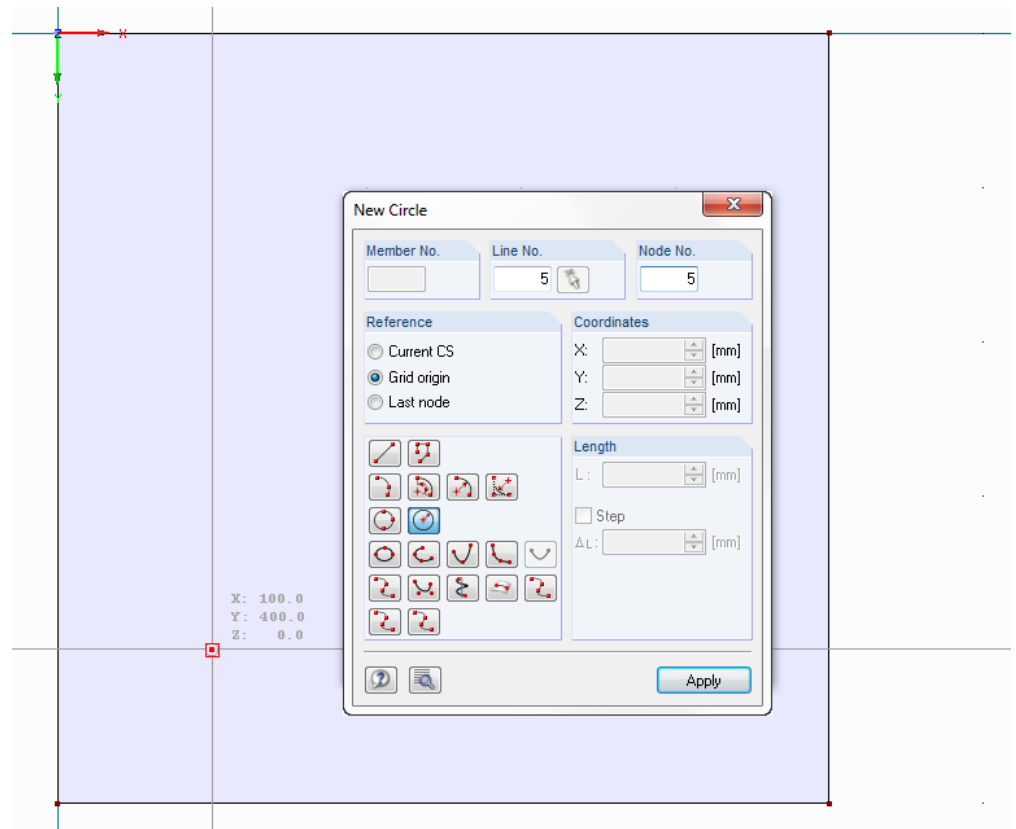
Dialog box *Move or Copy - Copy*

Now we open the **New Circle** dialog box. To do this, we select **New Circle via Center and Radius** on the menu of the **[Polyline]** button as shown on the following figure.

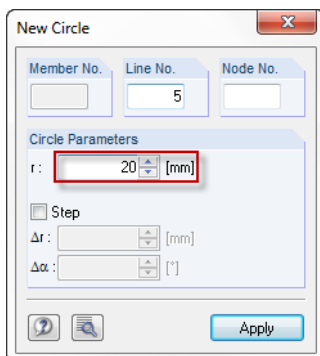


List button [Polyline]

The **New Circle** dialog box opens. The mouse becomes a reticle.



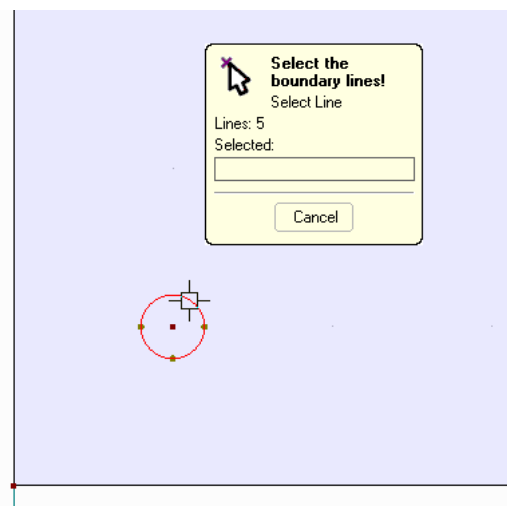
Dialog box *New Circle*



Now we select the copied node. The dialog box shown on the left opens. We specify the value **20 mm** for the radius and then confirm it by pressing **[Enter]**. Please make sure that you do not move the mouse pointer outside the dialog box. If the mouse pointer is in the work window, the value changes because then the graphical selection is active.

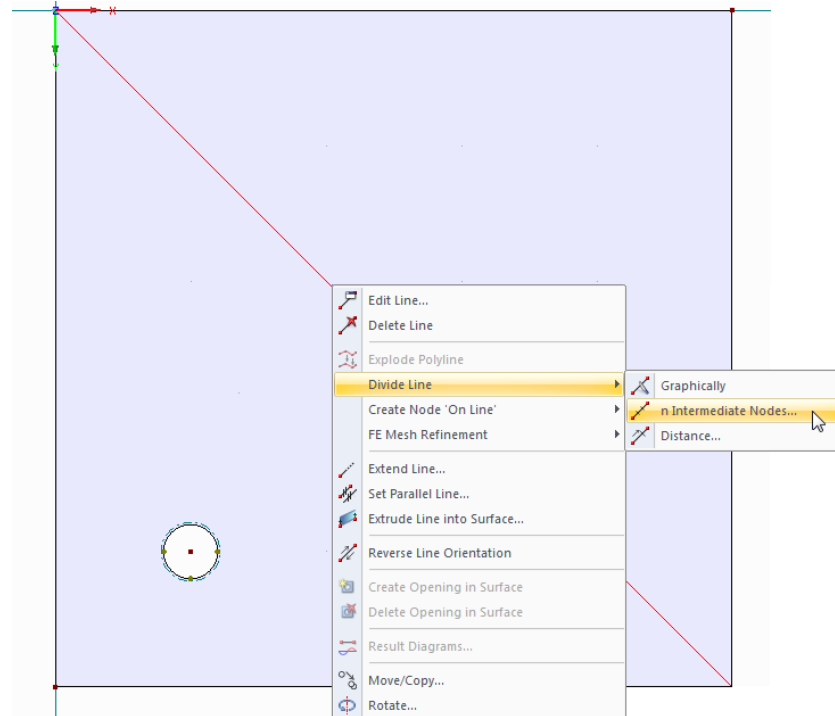
To close the **New Circle** function after creating the circle, we right-click in the work window.

To create an opening in this circle, we click **[New Opening via Selection of Boundary Lines]**, and then click the circle as shown in the figure below. Thus we create a circular opening in the plate. We quit the function by right-clicking anywhere in the work window.



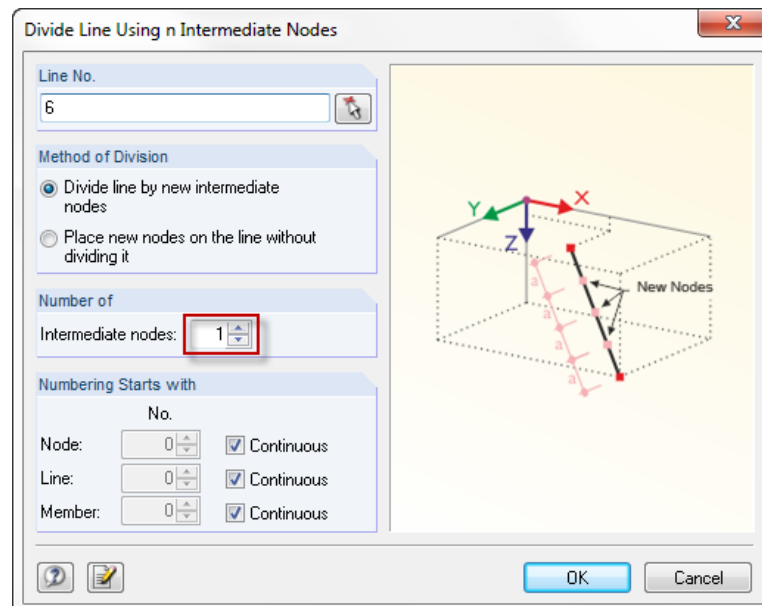
Creating circular opening

We want to create the other fastener openings by using the **Rotate** function. To do this, we first must determine the center point of the rectangular plate as the reference point for the rotation. To this end, we create a diagonal between two corner points of the plate. Then we divide it via the context menu of this line.



Line context menu

In the **Divide Line Using n Intermediate Nodes** dialog box, we set the number of intermediate nodes to **1**. Then we click [OK].

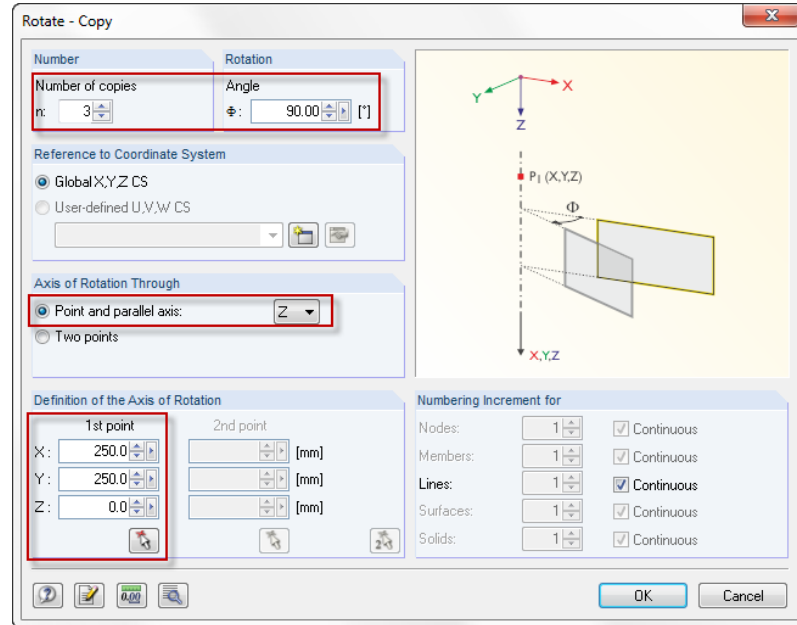


Dialog box *Divide Line Using n Intermediate Nodes*

We have now divided the diagonal into two equal lines. Now we delete the diagonal since we needed it only to create the center point for the rotation.

Next we want to select the entire opening with the circle. To facilitate the selection, we change the view to **[Wireframe Display Model]**. Now we draw a selection window across the opening, thus selecting it including all the lines and nodes.

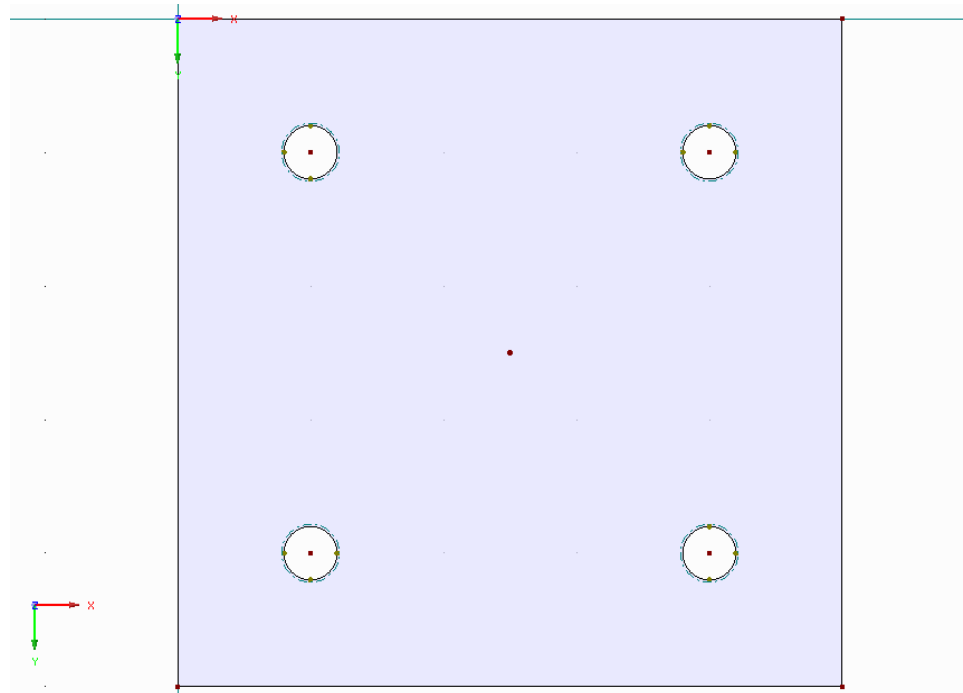
To rotate the selected opening, we click **[Rotate]**. The **Rotate** dialog box opens.



Dialog box *Rotate - Copy*

Since we need three additional openings, we set the number of copies to **3**.

The rotation is defined by a point and one principle axis. The point of rotation is the center point of the plate. We can enter its coordinates manually or by using the **[Select]** button. We rotate the openings about the **Z axis** by **90°**. When we confirm the dialog box, RFEM creates the additional openings.



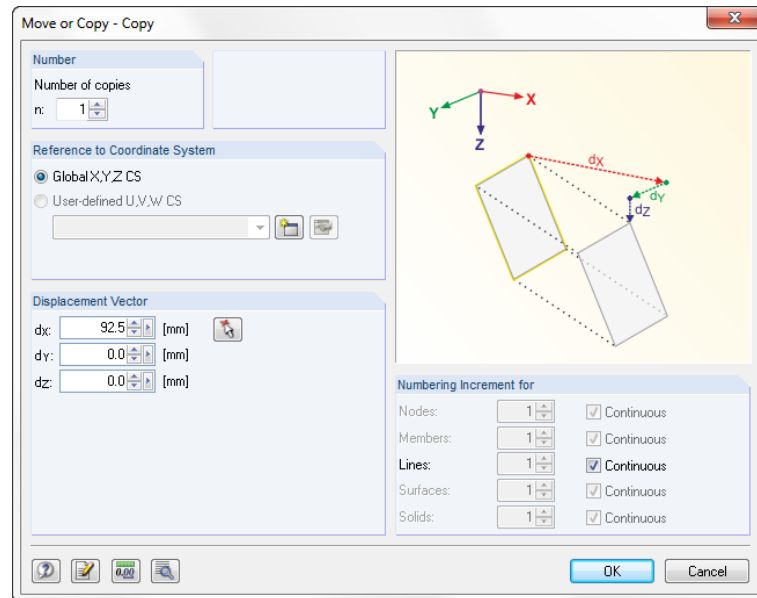
Fastener holes in base plate

3.1.3 Column

There is a "complicated" and a "simple" way to create the HEB 200 column at the base plate. First we want to look at the complicated way, as this allows us to compare it to the more efficient way.

3.1.3.1 Manual Definition of Surfaces

We copy the center point of the base plate by **92.5 mm** in the **X**-direction. To this end, we select the point and open the **Move or Copy** dialog box.



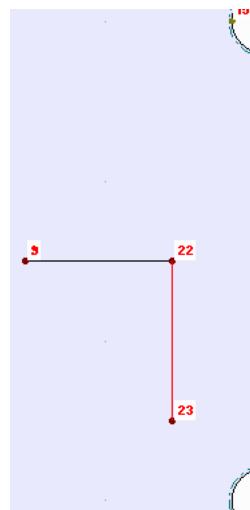
Dialog box *Move or Copy - Copy*



When copying the node, we want to create connection lines between the original node and the new one. To do this, we click [**Edit Expanded Settings**]. The **Detail Settings** dialog box appears where we check **Create new lines between the selected nodes and their copies**.

With these settings, the center point of the plate is copied by 92.5 mm in X-direction. The two nodes are connected by a line.

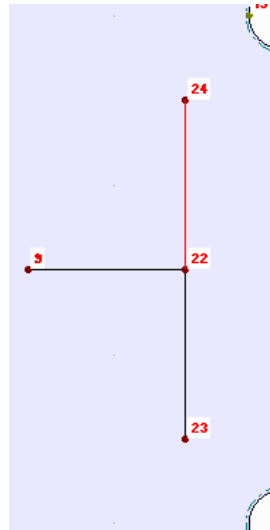
Now we copy the node by **100 mm** in the **Y**-direction in a similar way. After that, two new nodes and two connecting lines are created.



Copies of center node with connecting lines

We now copy the vertical line up by using the drag-and-drop function: We hold the **[Ctrl]** key and then grab the selected line. Please grab the line at its lower half (near node 23). Thus, we can activate the object snap and connect nodes 22 and 23 when they are near each other. Otherwise, we activate the other node (here node 22).

The lines should look like this now.



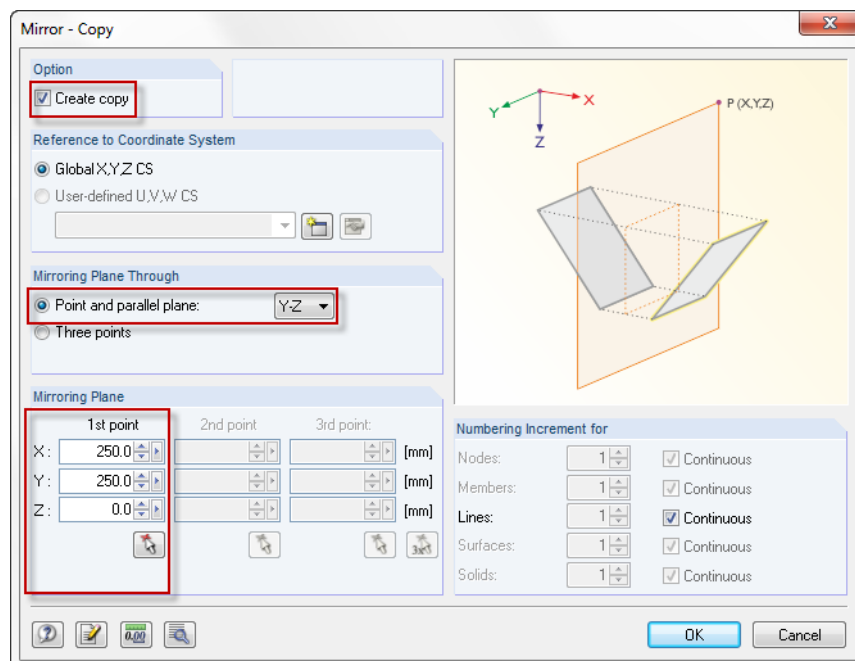
Copied line



Next we mirror all created lines about the center point of the plate (**node 9**). We click the **[Mirror]** button to open the **Mirror - Copy** dialog box.

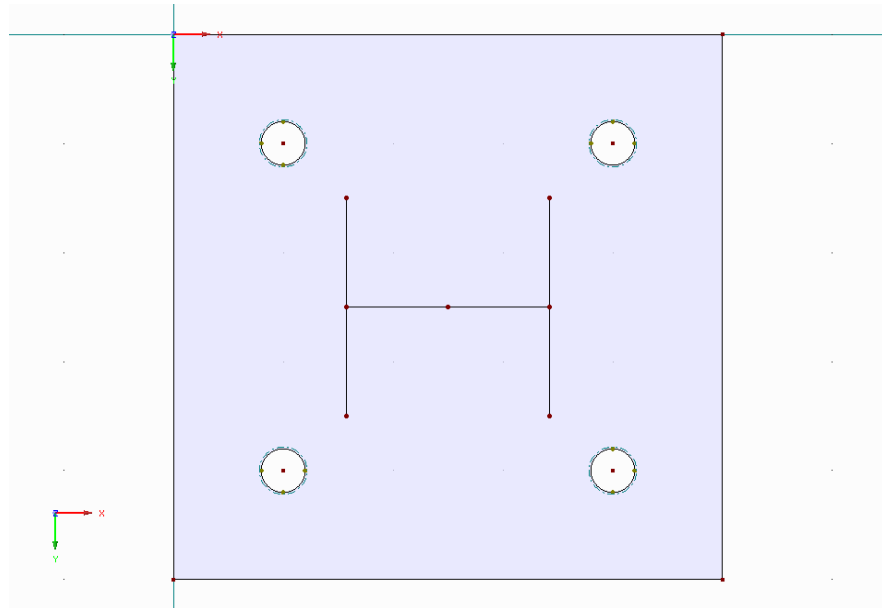


We want to create a copy and, therefore, select the **Create copy** check box. The mirror plane is the **Y-Z** plane. The mirror point is the center point of the plate. This time, it is important to clear the selection of the check box **Create new lines between the selected nodes and their copies** in the **Detail Settings** dialog box!



Dialog box **Mirror - Copy**

When we confirm the dialog boxes with [OK], the H-shape of the HEB 200 cross-section appears as shown in the following figure.

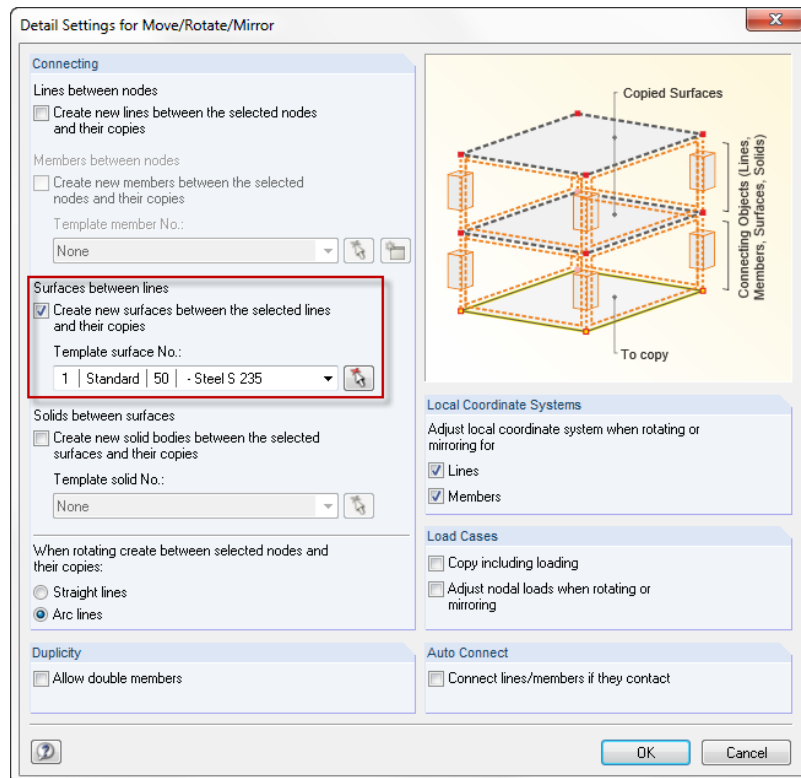


Copied lines in base plate

Finally, we copy the H-shaped cross-section lines by **-1000 mm** in **Z**-direction. To this end, we select the H-shaped lines and then open the **Move or Copy** dialog box again.

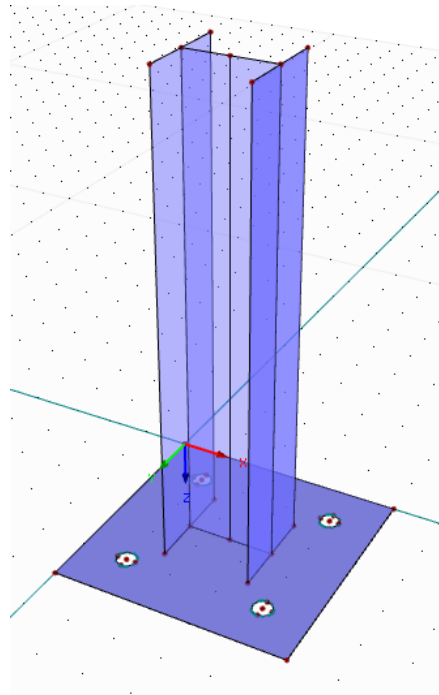


To create surfaces between the original lines and the new ones when copying, we open the **Detail Settings** dialog box. There we check the **Create new surfaces between the selected lines and their copies** option. As template surface, we select the surface of the base plate. We can edit the web and flange surfaces later.



Dialog box *Detail Settings for Move/Rotate/Mirror*

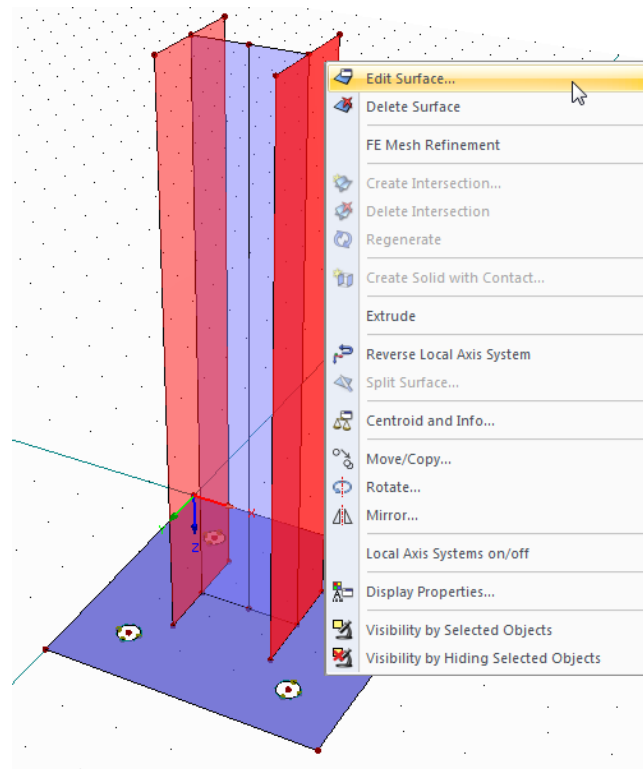
After confirming the data, the model looks like this.



Result of copied lines

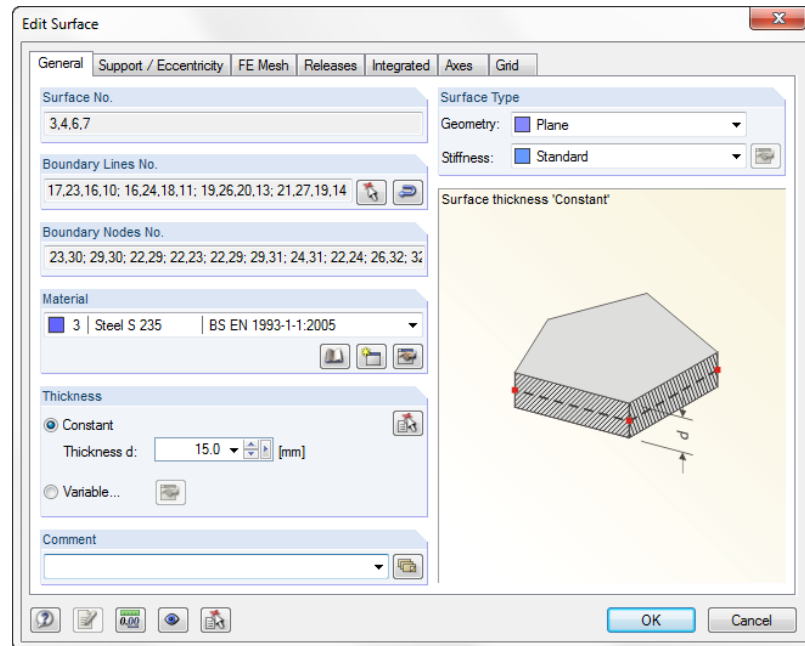
We now adjust the surface thickness of the web and the flanges. The web of a HEB 200 section has a thickness of 9 mm, the flanges are 15 mm thick.

First, we select the surfaces of the flanges. Then we right-click one of the selected surfaces. On the context menu, we select **Edit Surface**.



Surface context menu

The **Edit Surface** dialog box appears. Here we enter the value **15 mm** for the *Thickness d* and then confirm with [OK].



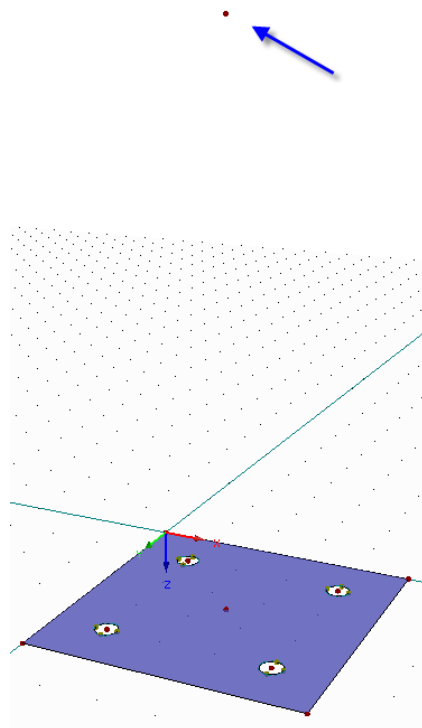
Dialog box *Edit Surface*

We change the thickness of the web in the same way by setting the surface thickness to **9 mm**.

3.1.3.2 Generation of Surfaces from Member

We can create the cross-section much faster. We return to the original base plate (without the lines) by using the **[Undo]** button.

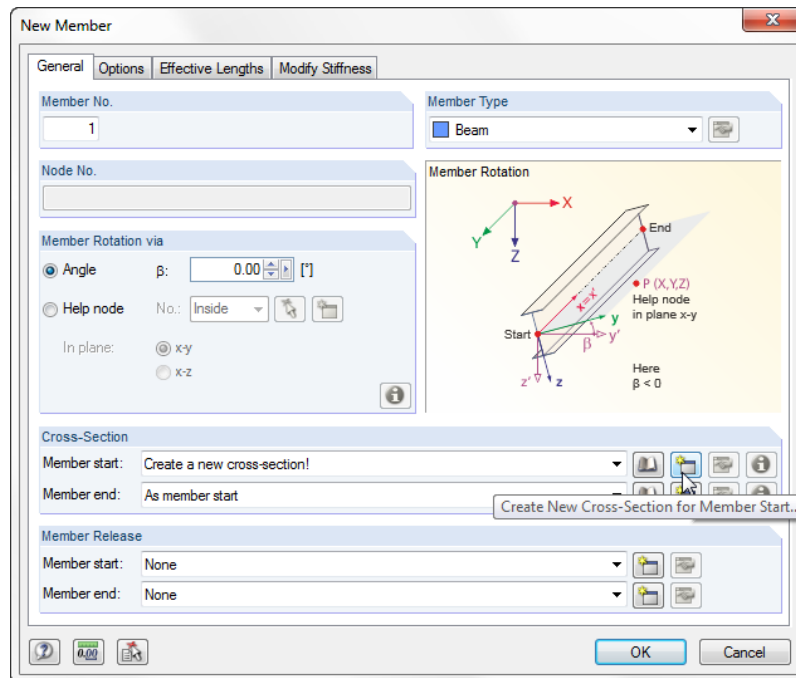
We select the center point and copy it by **-1000 mm** in **Z**-direction. The model looks as follows:



Copy of center node



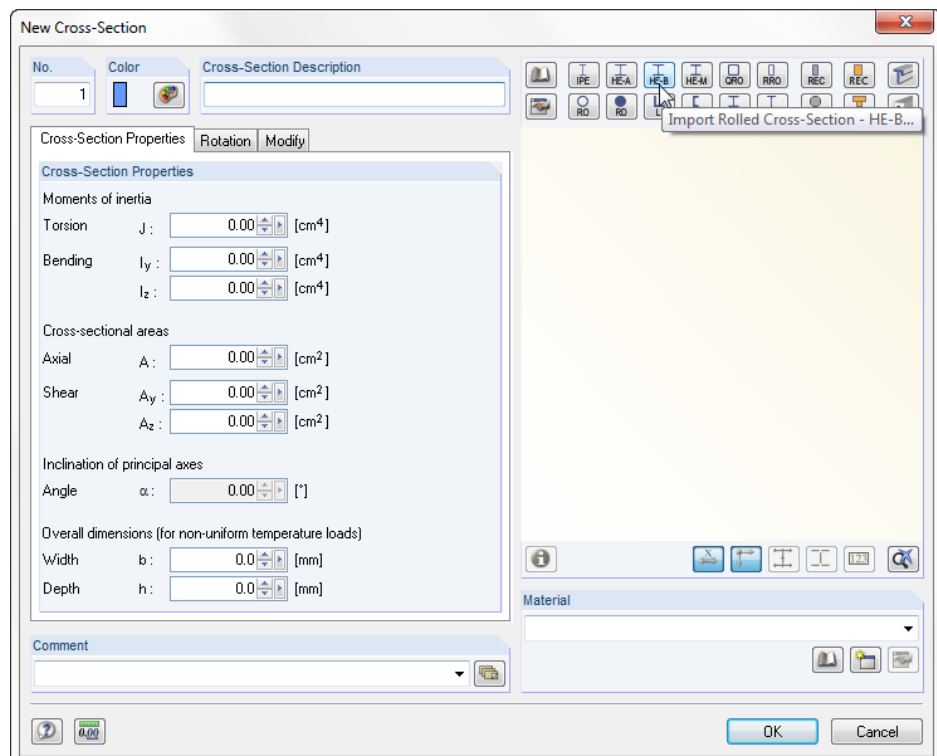
Now we define a member with the cross section HEB 200. We click **[New Member Single]** to open the **New Member** dialog box.



Dialog box *New Member*



We now click **[Create New Cross-Section for Member Start]**. The **New Cross-Section** dialog box opens.

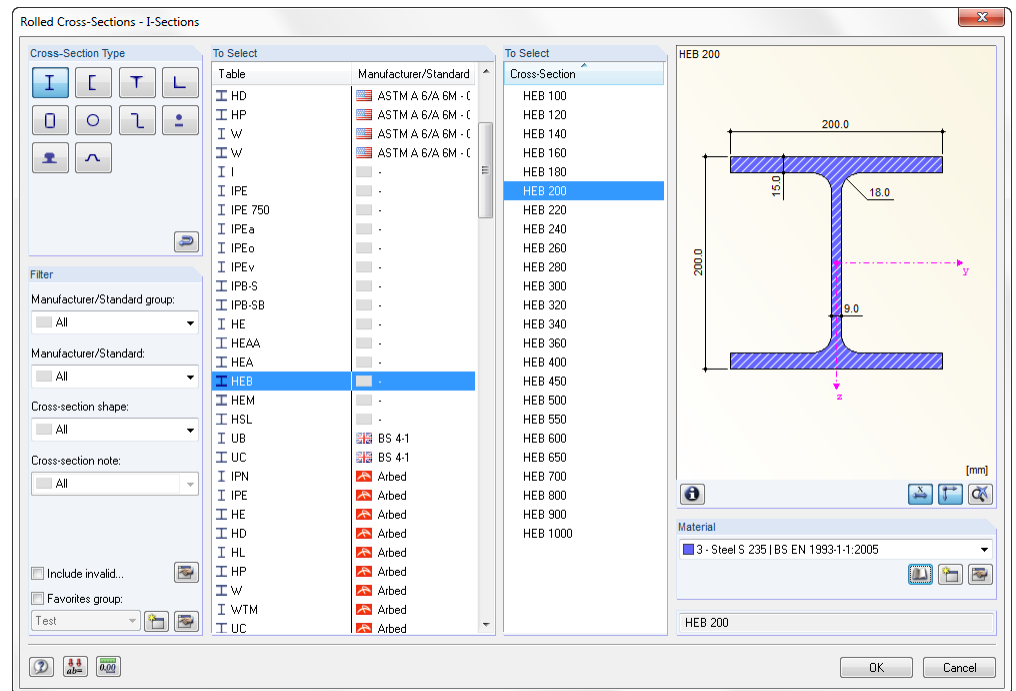


Dialog box *New Cross-Section*



Here we select Steel S 235. Then we click the **[Import Rolled Cross-Section - HE-B]** button.

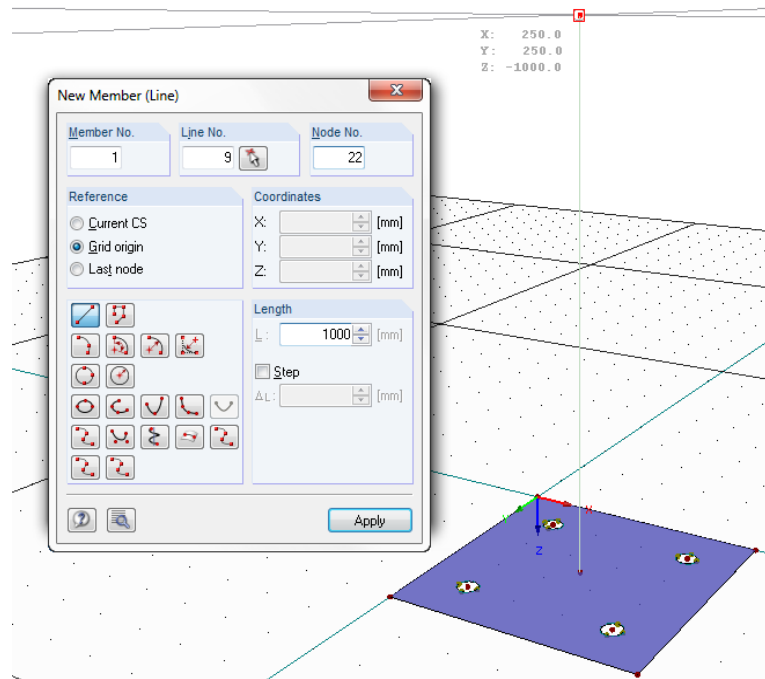
The following dialog box appears for the input of rolled cross-sections.



Cross-section library

In the left column, the HEB cross-section table is preset. We select the cross-section **HEB 200** in the right table and confirm all three dialog boxes by clicking [OK].

The **New Member** dialog box opens. The mouse pointer becomes a reticle. We now select the start node (center point of plate) and the end node (copied node) of the member by clicking them with the reticle.

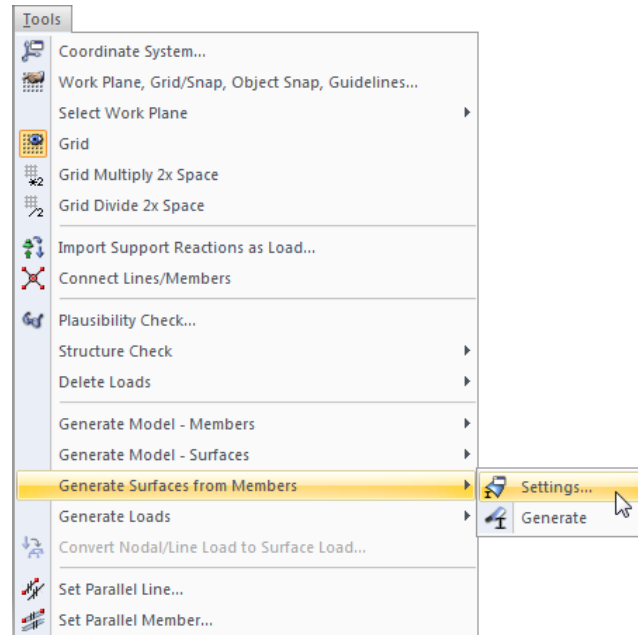


Defining member

In this way, we create a member between the two nodes with the cross-section HEB 200.

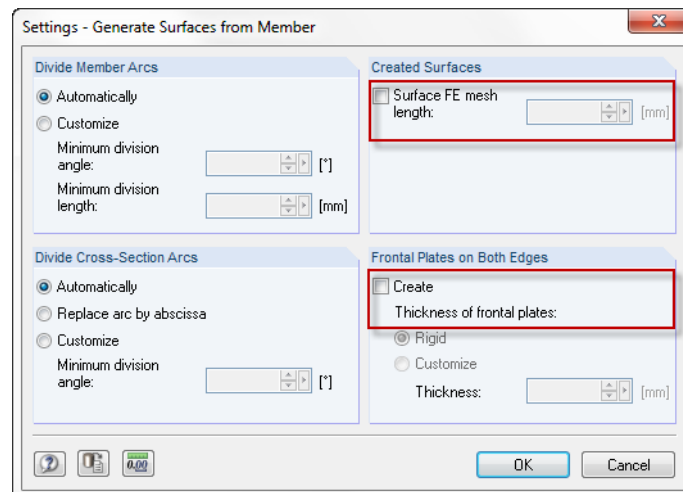
We now divide the member into individual surfaces. Before we do this, however, we open the **Settings** for this command by selecting

Tools → Generate Surfaces from Members → Settings.



Menu Tools

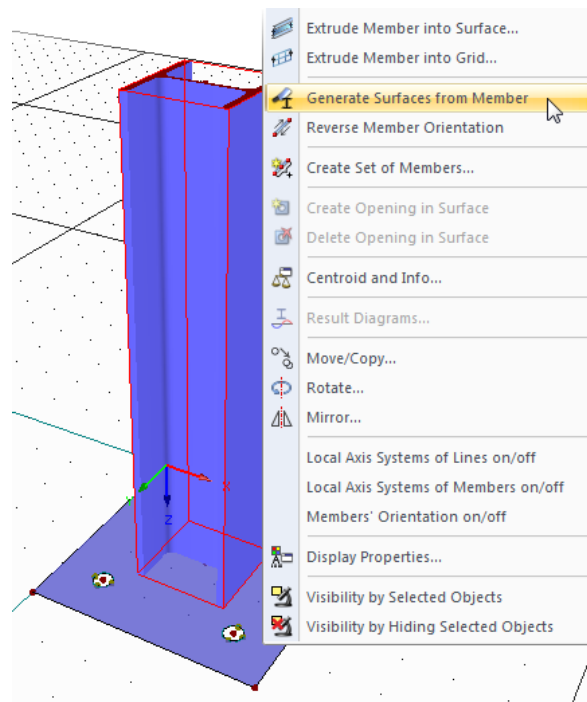
The **Settings - Generate Surfaces from Members** dialog box opens. This dialog box offers many setting options. Since we want to achieve the same results as in the "manual" specification of HEB 200 section, we deactivate the **Surface FE mesh length** check box in the **Created Surfaces** section, and the **Create** check box in the **Frontal Plates on Both Edges** section.



Dialog box Settings - Generate Surfaces from Member

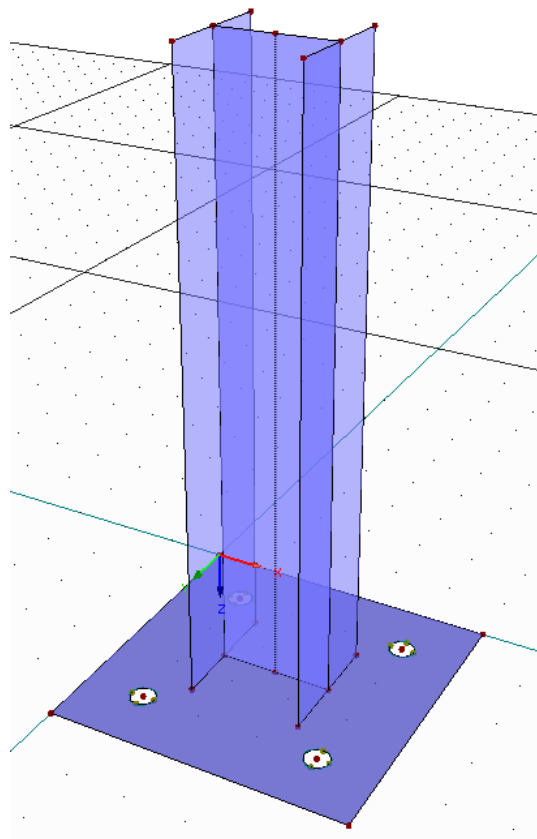
We confirm the input with [OK].

Now we right-click the member to open its context menu.



Member context menu

We select the **Generate Surfaces from Members** function. The member model is converted into a surface model. RFEM sets the surface thicknesses automatically.

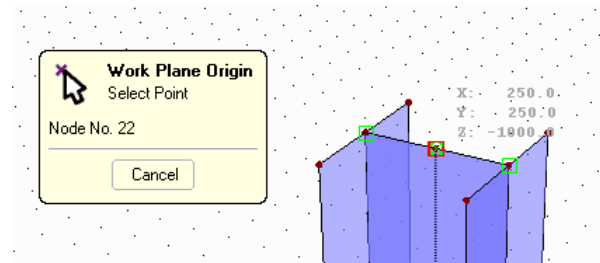


Surface model

3.1.4 Rigid End Plate



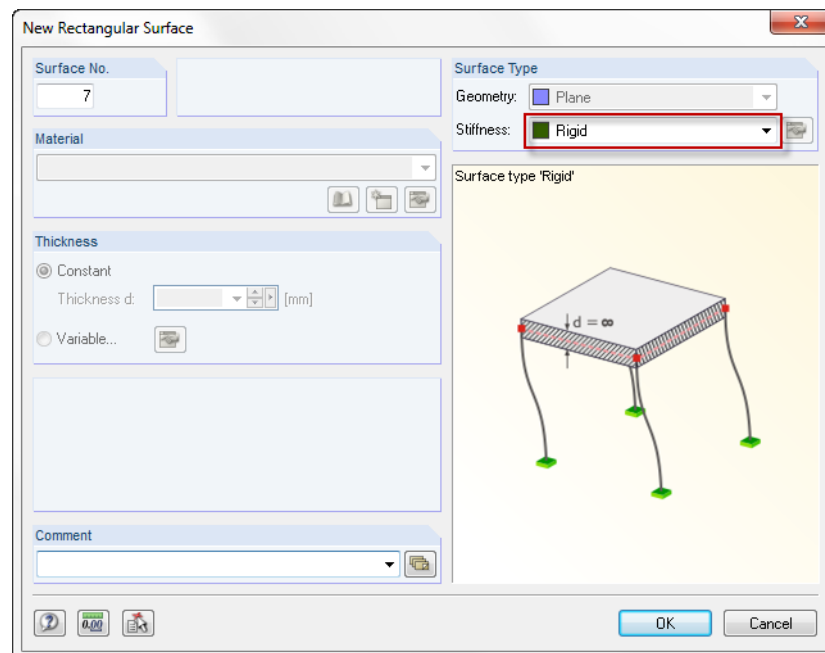
To apply the loads without singularity effects, we create a rigid surface on the upper end of the columns. We move the origin of the work plane by clicking **[Set Origin of Grid/Work Plane]**. Then we click one of the upper nodes of the surface where we want to set the new plane.



Defining new origin of work plane

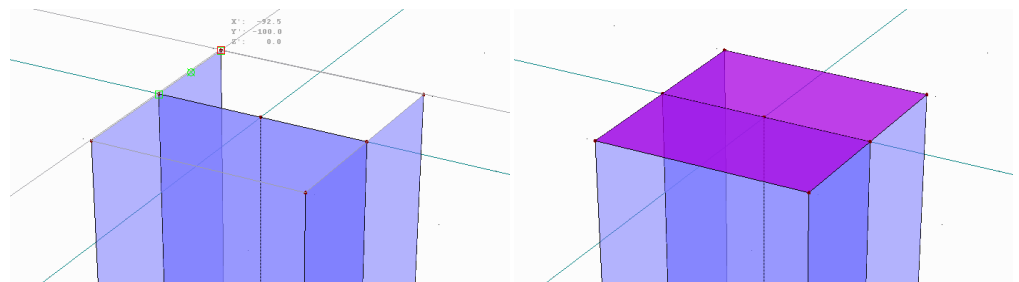


We open the **New Rectangular Surface** dialog box with the according button. In the dialog box, we select **Rigid** as stiffness type of the plate.



Dialog box *New Rectangular Surface*

After **[OK]**, we define the corners of the end plate by selecting two diagonally opposing corner points of the flanges. The boundary line of the web runs through the middle of the rigid plate. It is automatically integrated in the rigid surface (i.e. it is part of this surface).



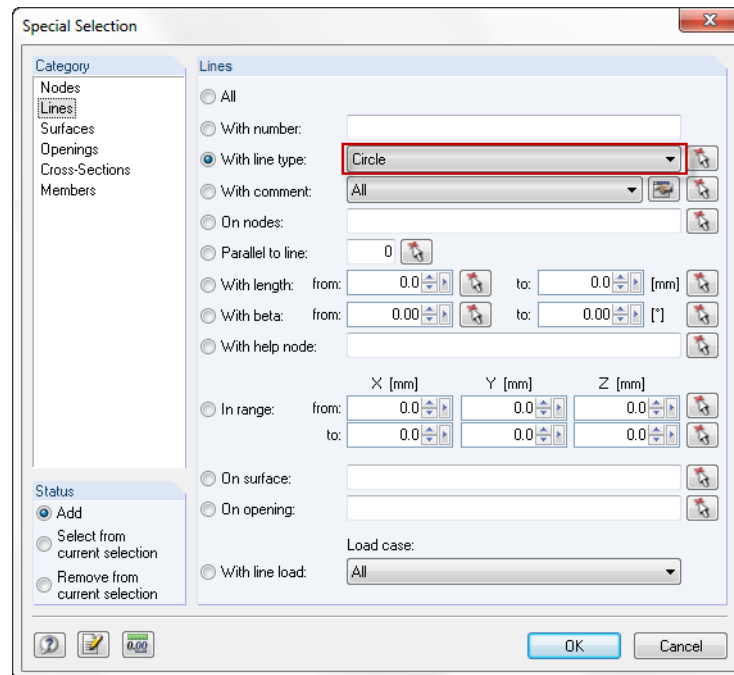
Creating rectangular surface of type *Rigid*

3.1.5 Supports

Line support



To support the fastener holes, we apply four line supports. We select all circles by using the **[Select Special]** button. The **Special Selection** dialog box opens where we set **Circle** from the **With line type** drop-down list as shown in the figure.

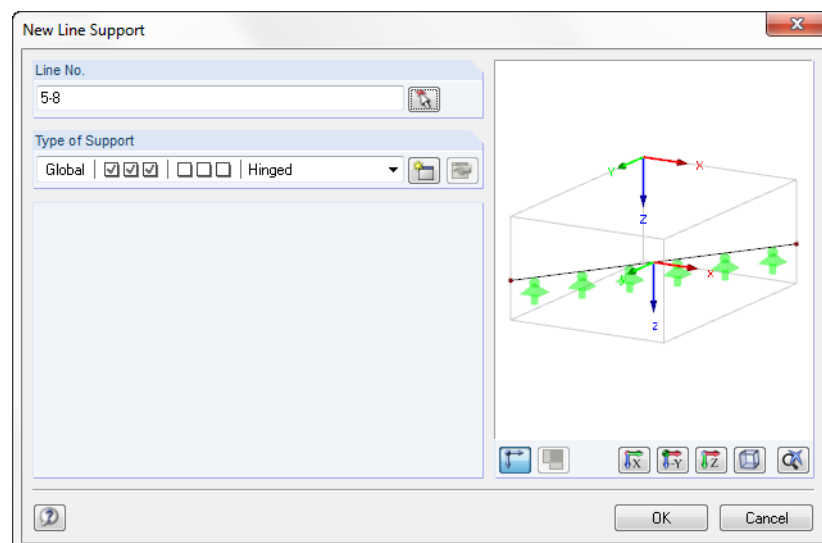


Dialog box *Special Selection*

After **[OK]**, all circles are selected (the boundary lines of the fastener openings).



Next we click **[New Line Support]** to open a dialog box in which we define the parameters of the support. From the **Type of Support** drop-down list, we select the **Hinged** type of support.

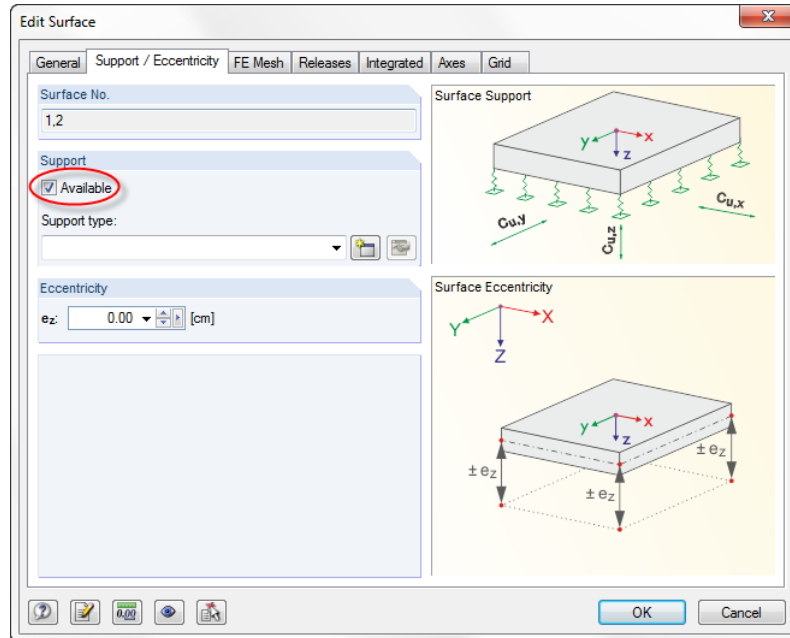


Dialog box *New Line Support*

After **[OK]**, the supports are assigned to the selected lines around the holes.

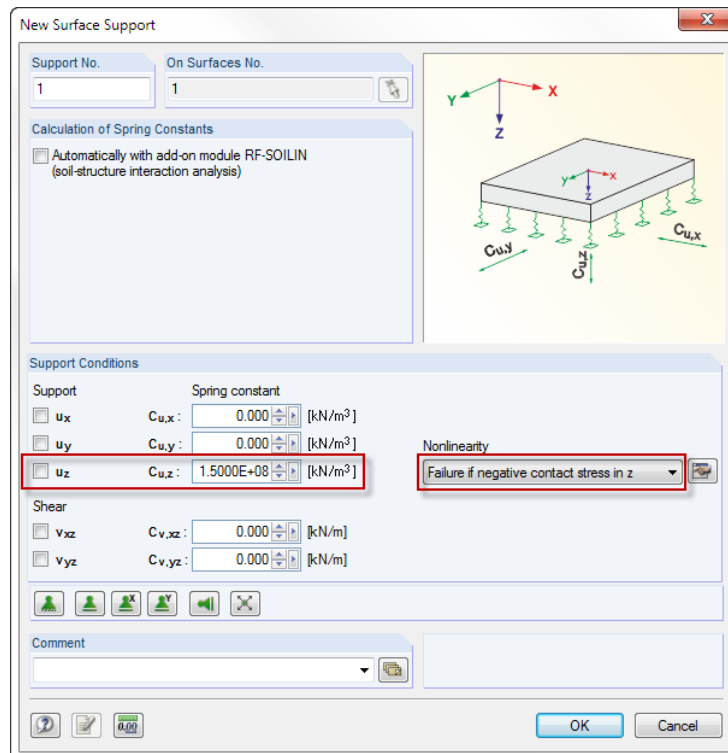
Surface support

To assign a surface support with failure under tension, we double-click the bottom plate. The **Edit Surface** dialog box opens. We go to the **Support / Eccentricity** tab and select the **Available** check box.



Dialog box *Edit Surface*

The **New Surface Support** dialog box is opened. We define the translational spring $C_{u,z}$ with the constant **150 000 000** [kN/m³]. Additionally, we consider the failure under tension. Tension forces are directed opposite to the z-axis of the surface so that the contact forces are negative. We select the **Nonlinearity** option **Failure if negative contact stress in z**.



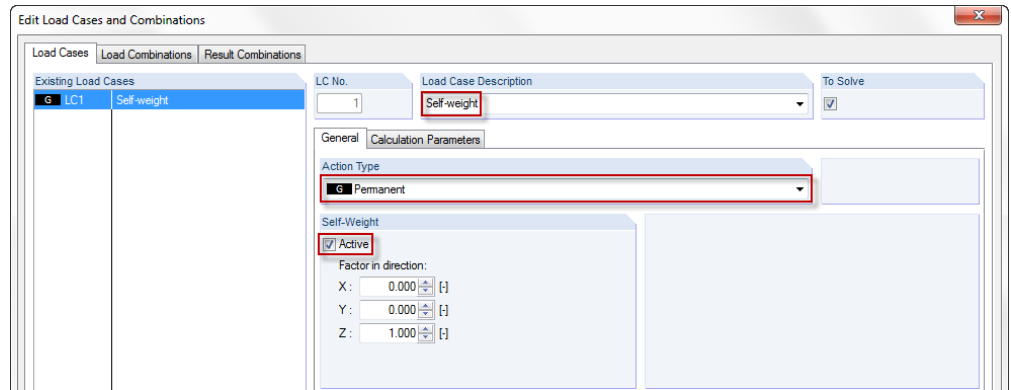
Dialog box *New Surface Support*

3.2 Load Data

3.2.1 LC 1: Self-Weight



To specify a load case, we click **[New Load Case]**. The **Edit Load Cases and Combinations** dialog opens showing the **Load Cases** tab.



Dialog box *Edit Load Cases and Combinations*

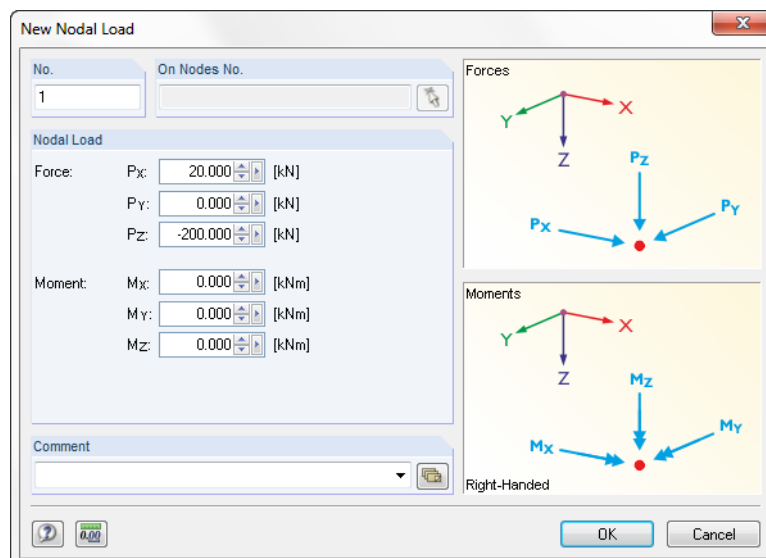
We assign the self-weight and all permanently acting loads to the first load case. As **Load Case Description**, we select **Self-weight** from the drop-down list. Furthermore, we set the **Action Type Permanent**.

The **Self-Weight** is activated by default. It is considered with the factor 1.0 in direction of the Z-axis. We confirm the input by **[OK]**. Then we enter the other permanently acting loads.

To specify a new nodal load, we click **[New Nodal Load]**. The following dialog box opens.



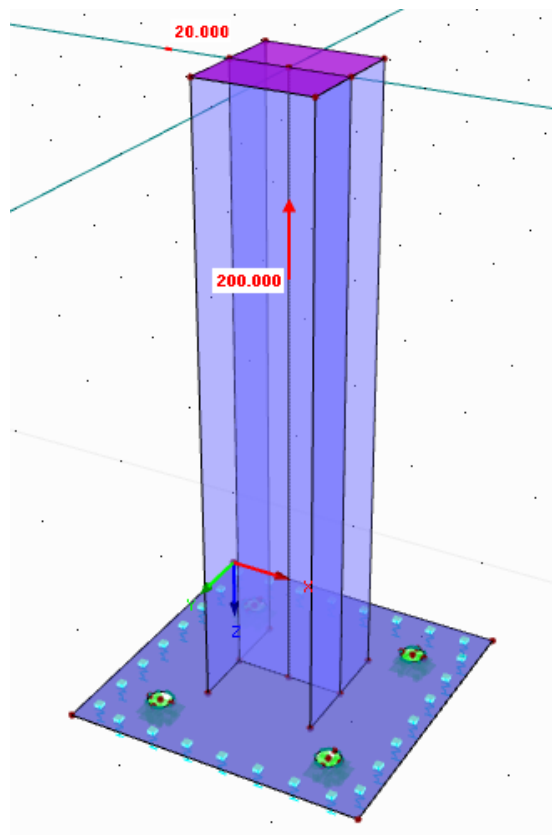
For our example, we want to apply a nodal load to the center point of the upper rigid plate. The nodal load has the components **20 kN** in the direction of the X-axis and **-200 kN** in the direction of the Z-axis.



Dialog box *New Nodal Load*



After **[OK]**, the symbol of the mouse pointer changes. We click the center point of the rigid end plate. Then the nodal load is displayed as shown in the following figure. To quit the **New Nodal Load** function, we right-click anywhere in the work window. Alternatively, you can press **[Esc]**.

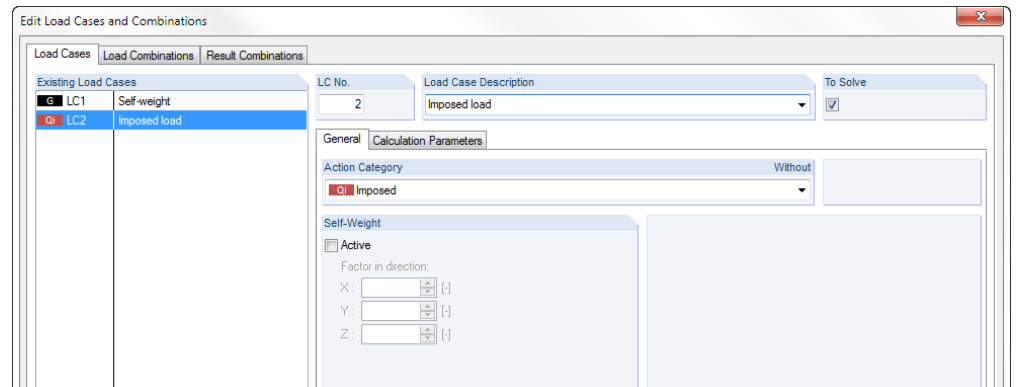


Model with permanent nodal load in the center point of the rigid end plate

3.2.2 LC 2: Imposed Load

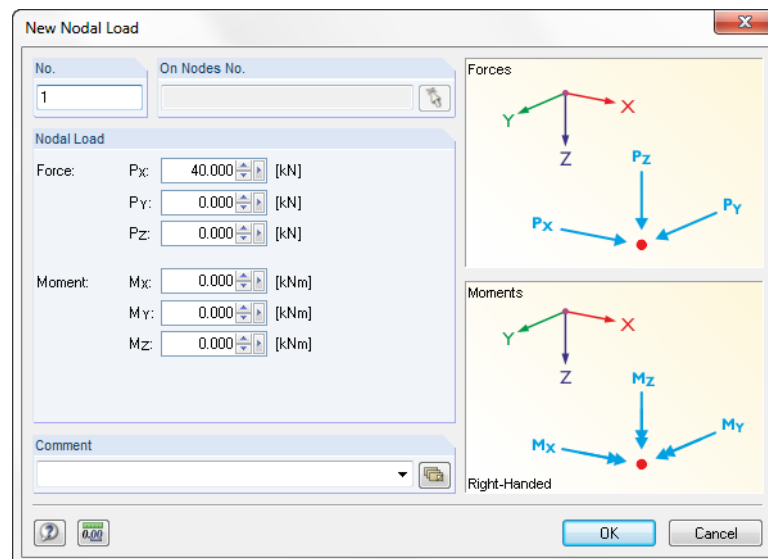
We define the second load case in the same way. First, we reopen the **Edit Load Cases and Combinations** dialog box on the **Load Cases** tab and the **General** subtab.

That load case contains a variable load. Therefore, we select **Imposed** as **Action Category**. The self-weight is not taken into account.



Dialog box *Edit Load Cases and Combinations*

Having confirmed the input, we open the **New Nodal Load** dialog box as we did for LC1. This time, we enter the nodal load in X-direction with the value **40 kN**.



Dialog box *New Nodal Load*



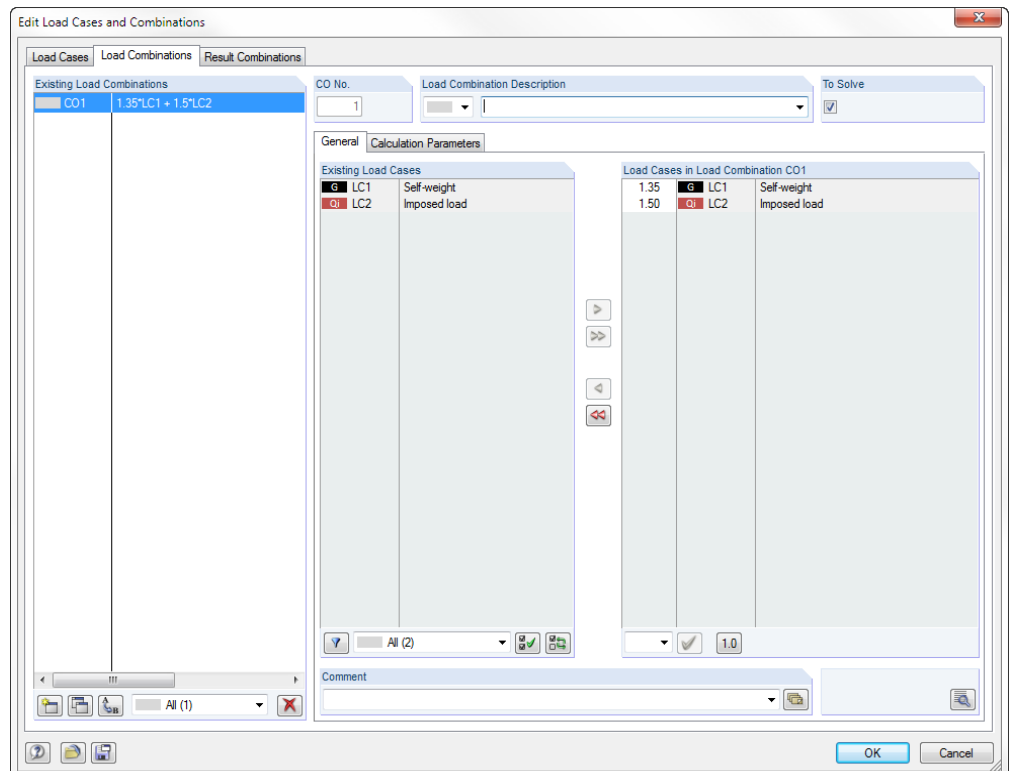
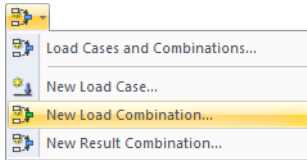
After closing this dialog box by clicking **[OK]**, we click the center point of the rigid frontal plate by using the nodal load symbol.

When the nodal load appears, we right-click anywhere in the work window to quit this function.

3.3 Superimposing the Loads

We superimpose the loads defined in chapter 3.2 in load combinations since they consider the influences of the second order analysis correctly. In load combinations, the loads are superimposed. In result combinations, however, the internal forces are superimposed, which is not allowed for nonlinear calculations.

To define load combinations, we select **New Load Combination** on the menu of the button **[Edit Load Cases and Combinations]**. The **Edit Load Cases and Combinations** dialog box opens showing the **Load Combinations**.



Dialog box *Edit Load Cases and Combinations*

The **Existing Load Cases** section lists all defined load cases. This table also shows the action type next to the respective LC. Depending on the action type and the selected standard, a partial safety factor is automatically chosen with which the load case is multiplied when it is added to the load combination. We can also define an individual partial safety factor.



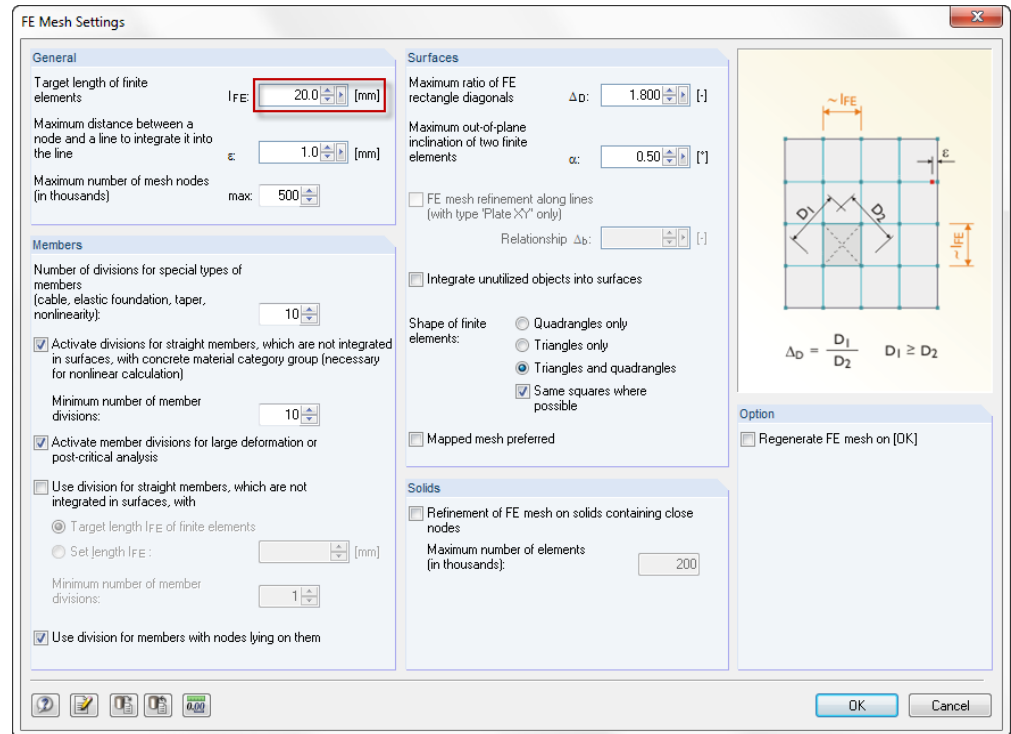
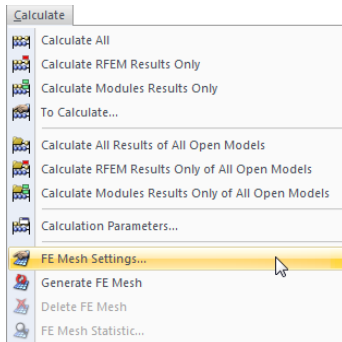
To add a load case to the load combination, we double-click a load case. You can also select a load case and then click **[Add Selected Load Cases]**.

Now we add the load case **LC1** with the partial safety factor **1.35** for permanent loads and **LC2** with the partial safety factor **1.50** for traffic loads to CO1. Then we click **[OK]**.

3.4 FE Mesh Settings

We select the **FE Mesh Settings** on the **Calculate** menu. The **FE Mesh Settings** dialog box opens.

We change the **Target length of finite elements** to **20 mm** and then confirm by clicking [OK].



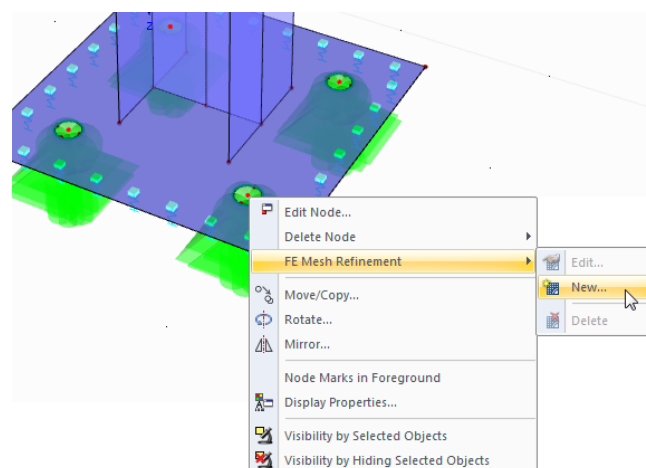
Dialog box *FE Mesh Settings*

To reduce computing time, the mesh should not be too refined. On the other hand, certain areas of the model should be represented as fine as adequate. To this end, we use the so-called **FE mesh refinement**.

3.4.1 Node Refinement

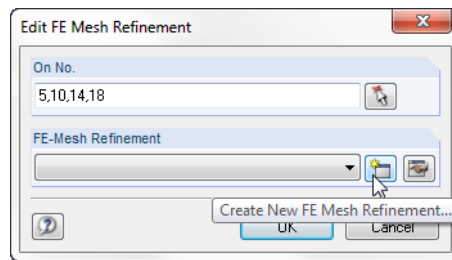
The areas around the fastener openings should be treated specially. They can best be refined by using node refinements.

We select the four center points of the openings by clicking them one by one while holding the [Ctrl] key. Then we right-click one of the selected nodes to open the node context menu.



Node context menu

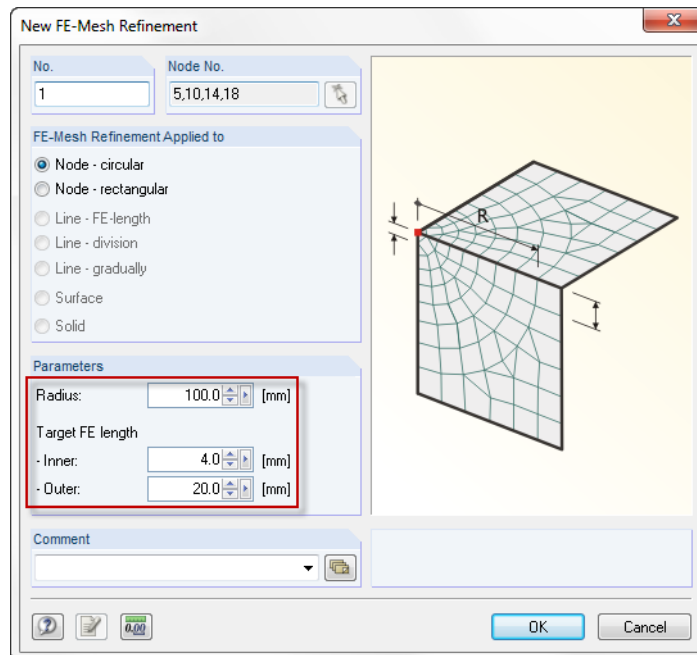
We select the option **FE Mesh Refinement** → **New**. The **Edit FE Mesh Refinement** dialog box opens.



Dialog box *Edit FE Mesh Refinement*

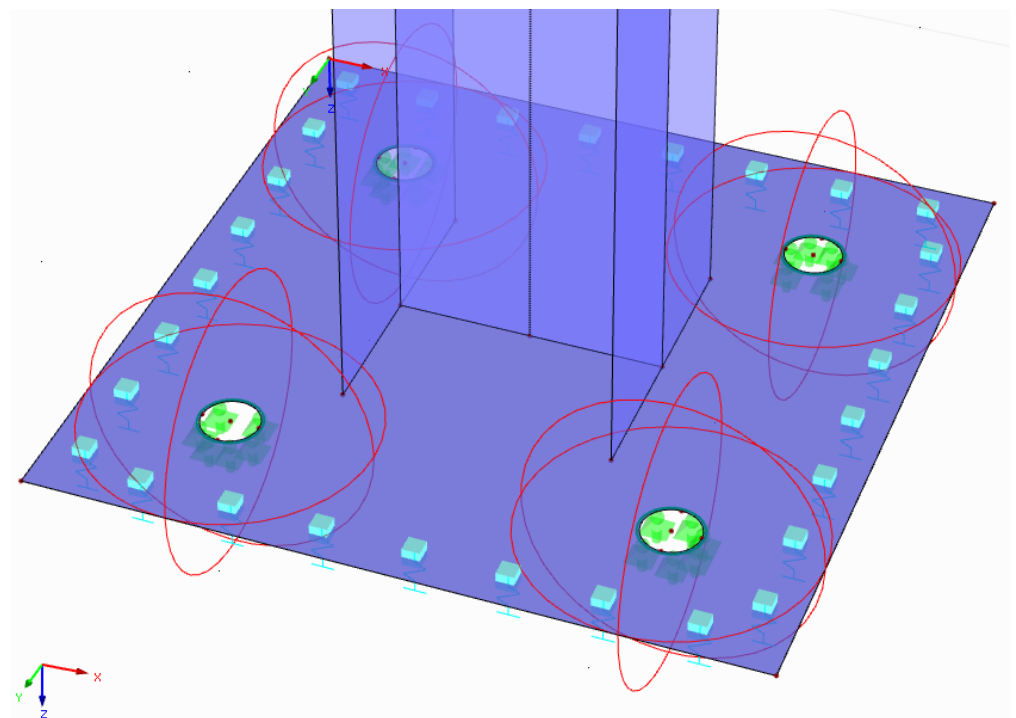
We click [**Create New FE Mesh Refinement**] to define the parameters of the refinement.

We keep the option **Node - circular** in the **FE Mesh Refinement Applied to** section. We enter **100 mm** for the **Radius** of the mesh refinement. Under **Target FE length**, the **Inner** input field refers to the FE length of the elements at the center of the mesh refinement. We enter **4.0 mm** for this value. This length increases until it reaches the **Outer** periphery of the radius. In this input field, you usually enter the length of the global FE mesh (here **20.0 mm**).



Dialog box *New FE Mesh Refinement*

When we confirm the dialog boxes by clicking [**OK**], the refinements areas of the FE mesh are shown around the selected nodes (see figure below).

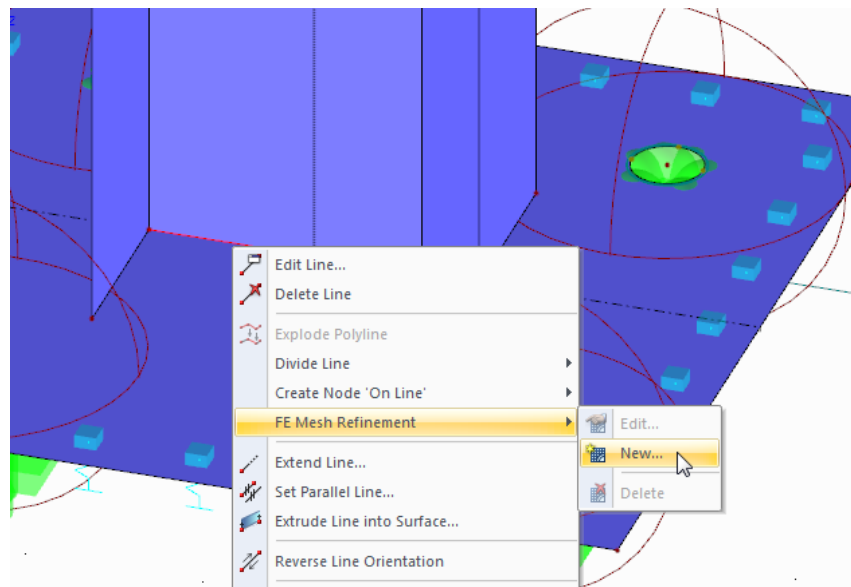


Node refinement areas for FE mesh

3.4.2 Line Refinement

We also refine the weld seam between the web and the base plate by a **line mesh refinement**.

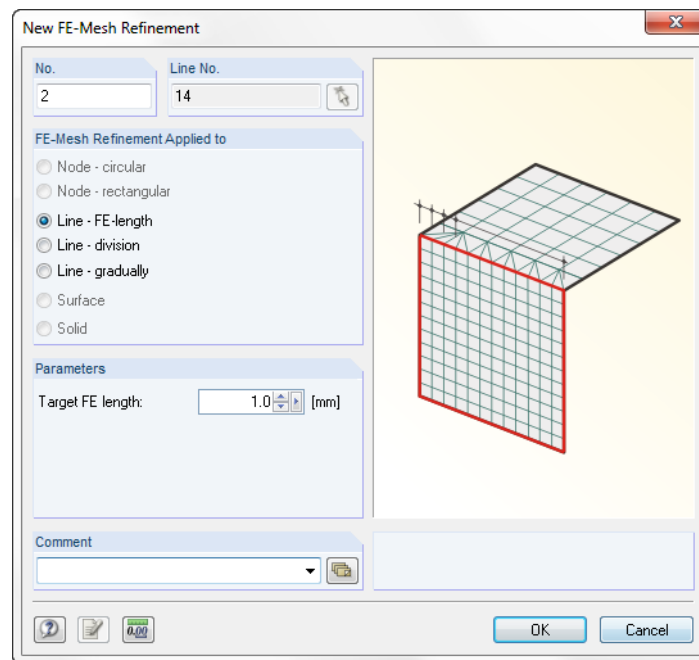
By right-clicking the line of the weld seam, we open the line context menu. Again, we select the option **FE Mesh Refinement** → **New**.



Line context menu

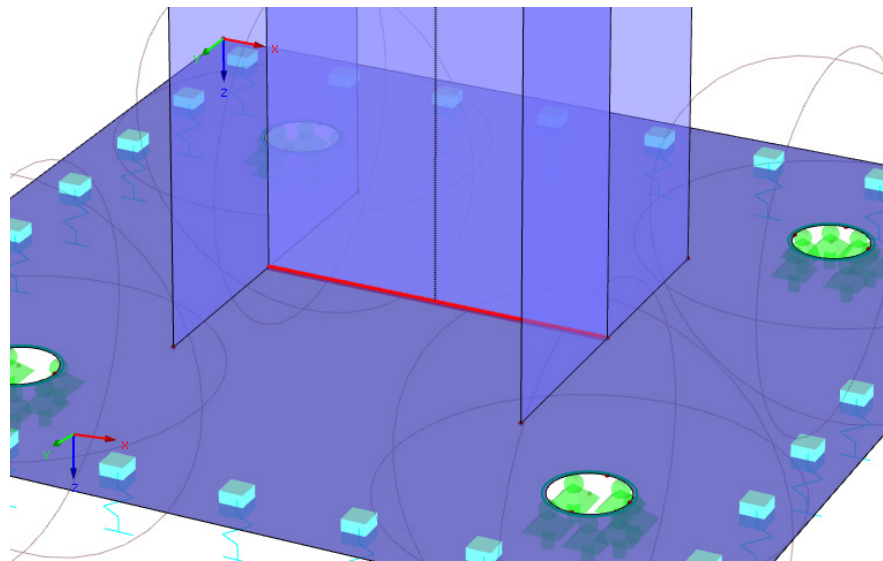
The definition of the line refinement is similar to the one of a node refinement. There are different options available in the **New FE-Mesh Refinement** dialog box, however.

We keep the **Line - FE-length** option under **FE-Mesh Refinement Applied to** dialog box. In the **Target FE length** input field, we enter the value **1.0 mm** (see figure below).



Dialog box *New FE Mesh Refinement*

After [OK], the line mesh refinement is displayed in the work window.



Line mesh refinement of the FE mesh

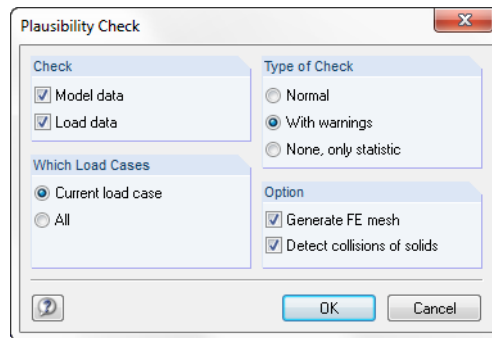
Mesh refinements of **Surfaces** and **Solids** can be generated in a similar way.

3.5 Plausibility Check

The input of the model data is complete now. Before we start the calculation, however, we should perform some checks.



The **Plausibility Check** is useful to check whether the model has been entered correctly or not. This check is automatically performed before the calculation. You can also start it manually by clicking **[Plausibility Check]**.

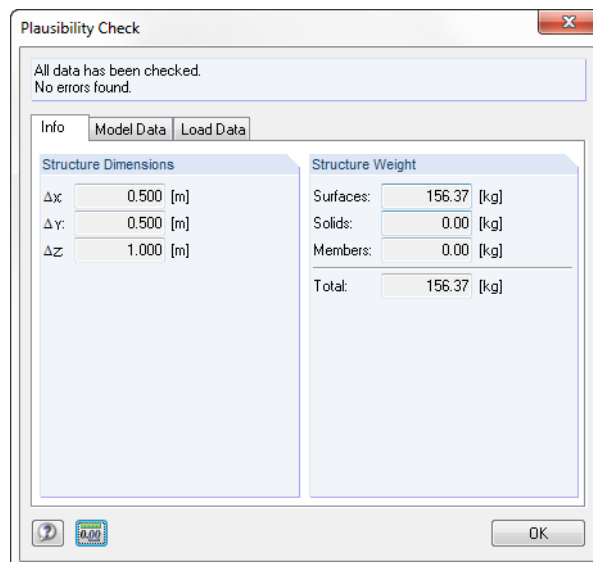


Dialog box *Plausibility Check*

Before the check starts, it is possible to specify some settings. In the **Check** section, we can choose if we want to check the model data and/or the load data.

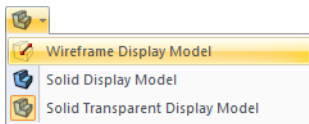
If we select the **Generate FE mesh** check box, the FE mesh is generated during the check.

We start the check by clicking **[OK]**. If the model contains no errors, the following dialog box appears after the check. After **[OK]**, we can view the model with the generated FE mesh.



Result of plausibility check

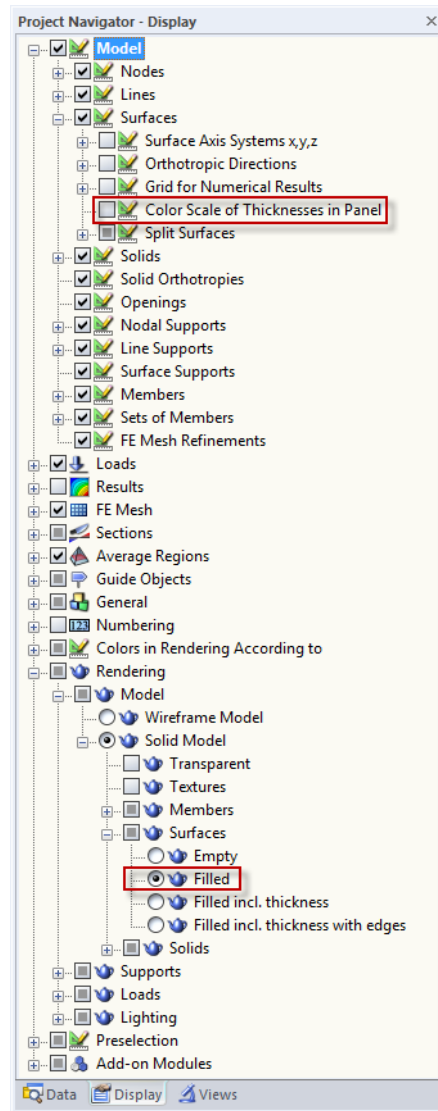
This function checks the model only with regard to modeling, that is, it checks whether or not there are input errors for certain objects. The plausibility check cannot evaluate whether the entered model corresponds to the model in your mind or on paper. This can only be done by the user. However, RFEM offers us various tools that facilitate this kind of checking.



For example, we can use the **[Display Model]** button to switch between the wireframe and the solid display model including the rendered member cross-sections. This allows us to detect parts of the model that are not correct.

There are different options to display surfaces. These options are available in the **Display** navigator. For example, we can display the surfaces including their thicknesses.

Additionally, there is the possibility to display the thicknesses with or without the actual thicknesses. The colors are displayed depending on the thicknesses in the control panel.



Display navigator

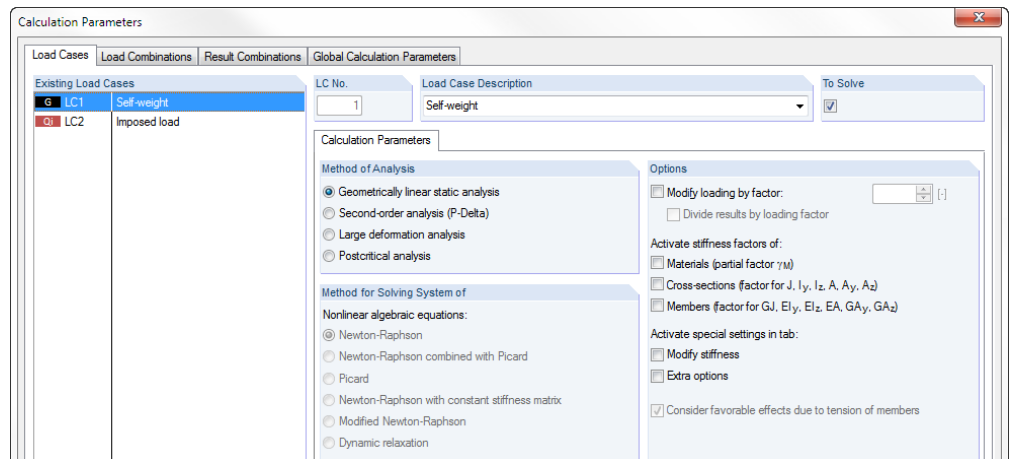
3.6 Calculation

Calculation parameters



The calculation can be controlled in the **Calculation Parameters** dialog box. To open this dialog box, we click [**Calculation Parameters**]. In this dialog box, we select the **Load Cases** tab.

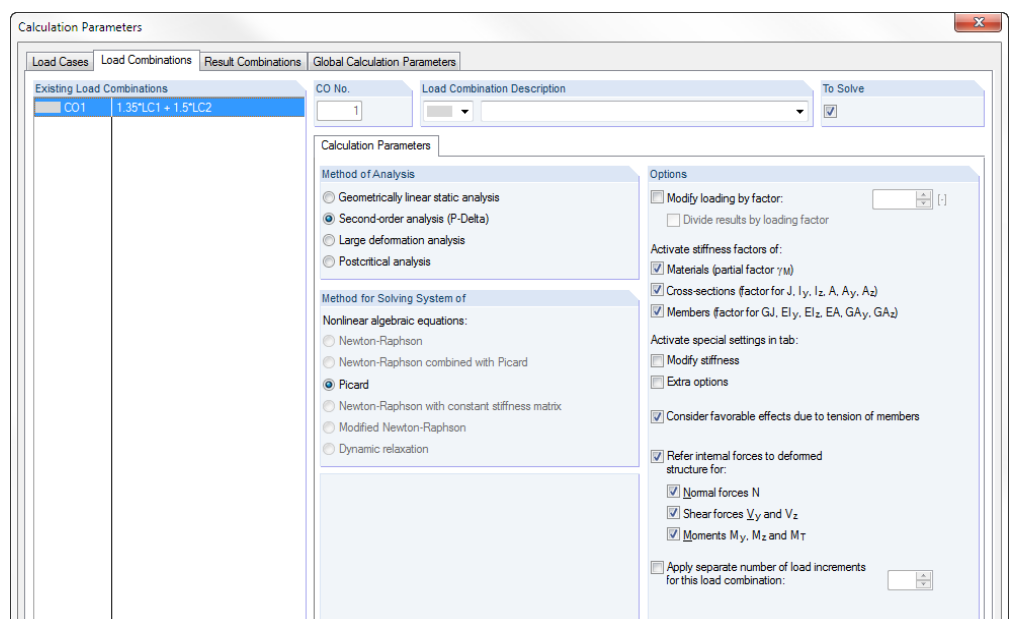
The section on the left lists all defined load cases. We can select one or several load cases and modify the options for the selection. For example, we can choose according to which method of analysis the calculation is to be performed. By default, load cases are calculated according to the **Geometrically linear static analysis**.



Dialog box *Calculation Parameters*, tab *Load Cases*

The tab **Load Combinations** has a similar structure. Instead of load cases, the section on the left lists all defined load combinations.

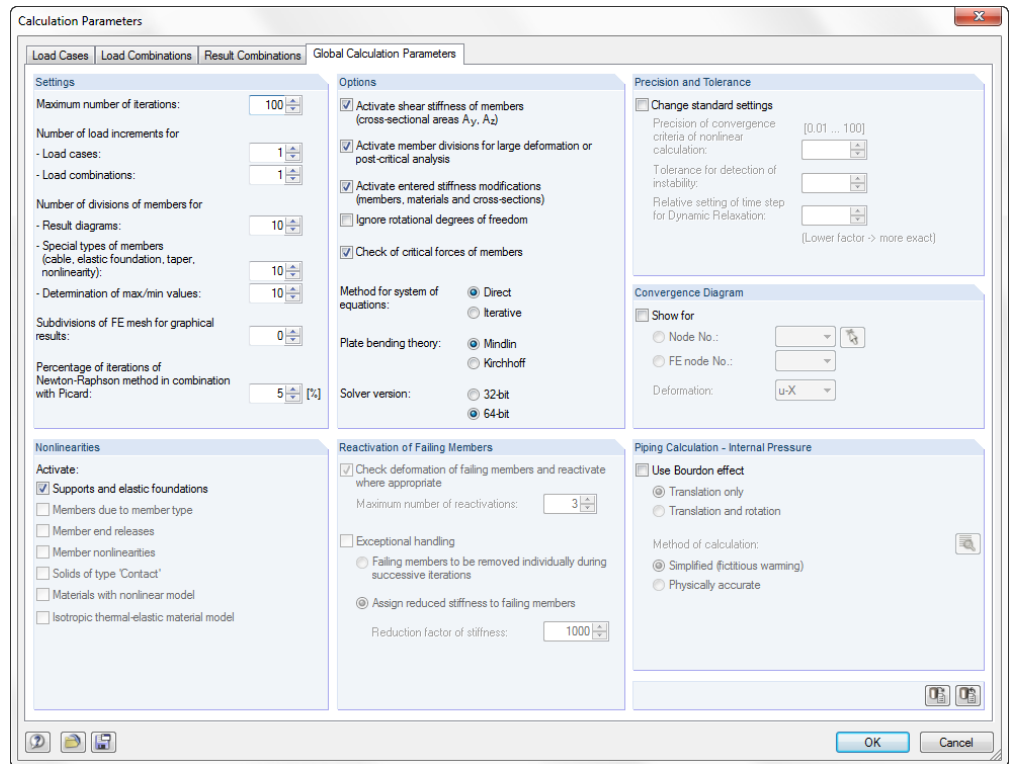
Load combinations are calculated according to the second-order analysis by default. However, we can select any of the four methods of analysis for load cases and load combinations. By selecting a nonlinear method of calculation, three further options become available (this also applies to the **Load Cases** tab). We can consider favorable effects due to tension of members. Moreover, we can freely define a loading factor and also divide the results by this factor.



Dialog box *Calculation Parameters*, tab *Load Combinations*

Since we have not defined any result combinations, we go to the tab **Global Calculation Parameters**. This tab offers many possibilities to influence the calculation. In the **Nonlinearities** section, it is possible to clear the consideration of any defined nonlinearities. This may be useful if the calculation is stopped or does not converge.

In the **Options** section, we can select the method for the system of equations, the plate bending theory or the solver version. Some settings should be changed only by advanced users: maximum number of iterations, the number of load increments, the precision of convergence criteria, etc. These settings influence the duration and the convergence behavior.



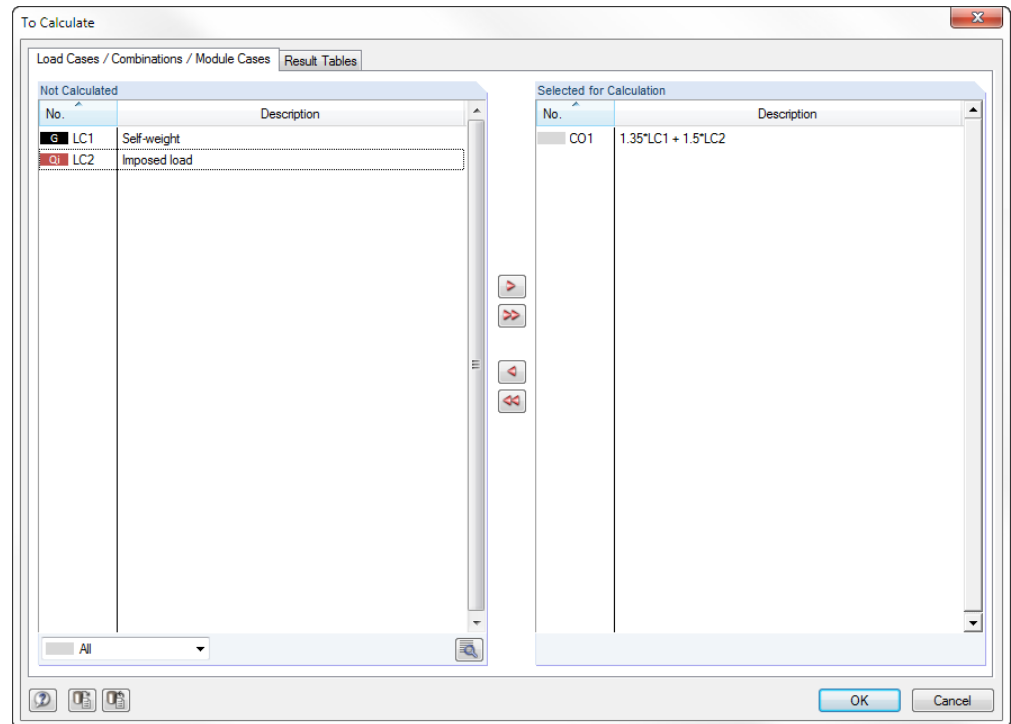
Dialog box *Calculation Parameters*, tab *Global Calculation Parameters*

We do not change the settings. Then we close the dialog box.

Calculation

To open the dialog box to select the load cases, load and result combinations that are to be calculated, we click menu **Calculate** → **To Calculate**.

The **To Calculate** dialog box appears. By double-clicking a load case, we transfer it to the **Selected for Calculation** list. Alternatively, you can also select an entry, and then click **[Add to Calculation]**. It is also possible to add several or all entries.

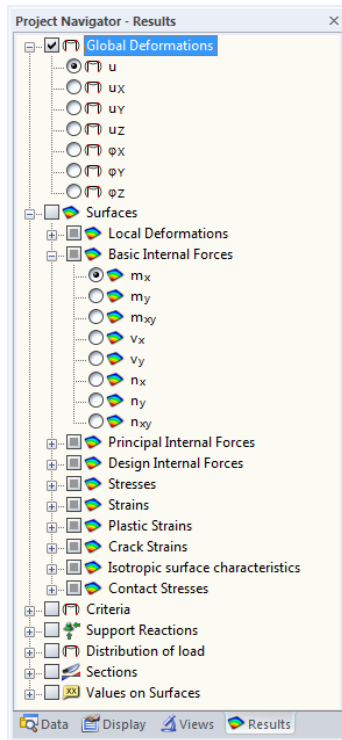


Dialog box *To Calculate*

We select load combination **CO1**. To start the calculation, we click **[OK]**.

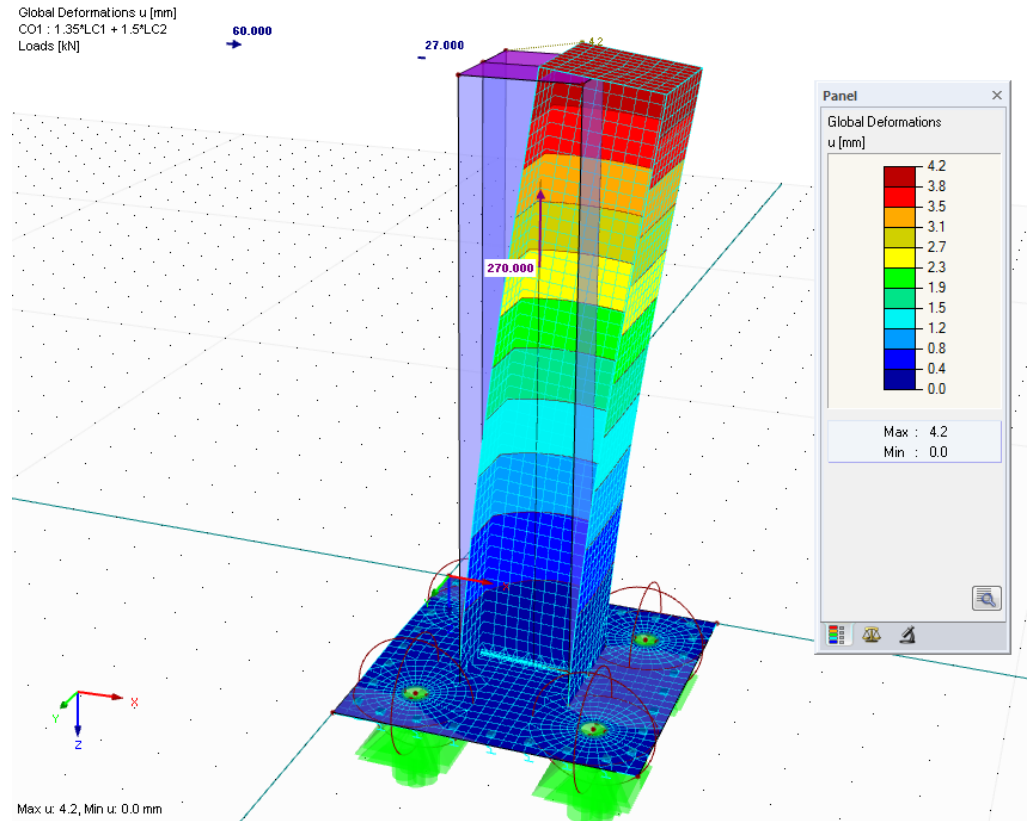
3.7 Results

After the calculation, the graphic of the deformations is shown in the work window.



Results navigator

CO1 - 1.35*LC1 + 1.5*LC2



Deformations of load combination CO1

Results Display

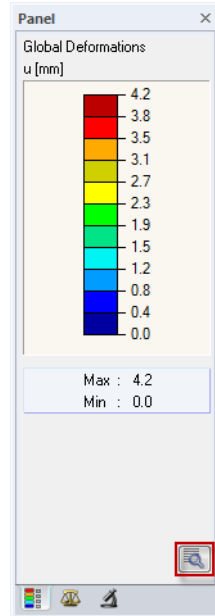
If the **Results** option was not deactivated in the **Display** navigator, the new tab **Results** appears in the navigator. Here you can control the display of all results. The results are sorted by element type, that is, depending on the elements in your model, the results can be displayed in members, surfaces or solids.

In the **drop-down** list in the toolbar, we can select the loading whose results are to be shown.

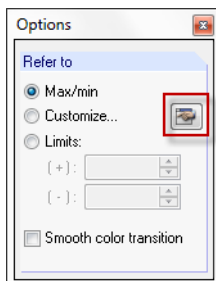
Panel



If the **[Panel]** button is switched on, the **Control Panel** appears. It contains three tabs. The first tab shows a results scale. The scale refers to the minimum and maximum values of the results. However, it is possible to edit the spectrum of the scale by using the **[Options]** button.

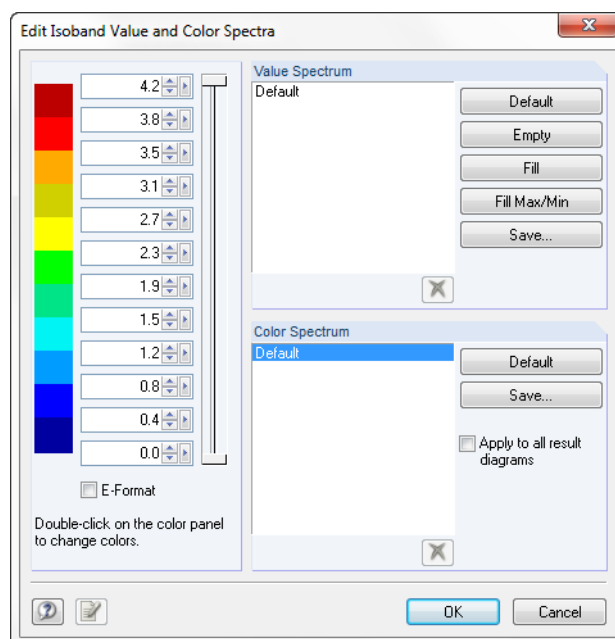


Panel with [Options] button



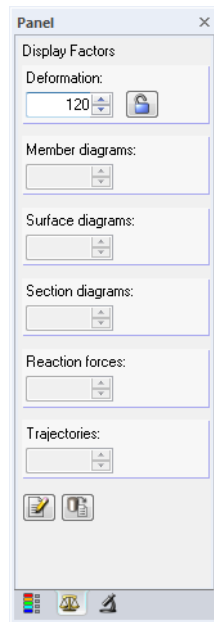
We click **[Options]** to open the **Options** dialog box offering various settings. For example, it is possible to display a **Smooth color transition**. The **Max/min** option sets the value spectrum according to the minimum and maximum values of the results. By selecting **Limits**, we can manually enter an upper and a lower limit value.

By selecting **Customize**, we can edit the value and color spectra. We open the **Edit Isoband Value and Color Spectra** dialog box by clicking the **[Edit]** button. This dialog box allows us to modify the settings of the spectra. The modified values and colors can be saved and imported in other models.



Dialog box *Edit Isoband Value and Color Spectra*

The second tab of the panel offers us the option to change some factors in order to increase the display of the results. Depending on the displayed results, we can edit different factors.

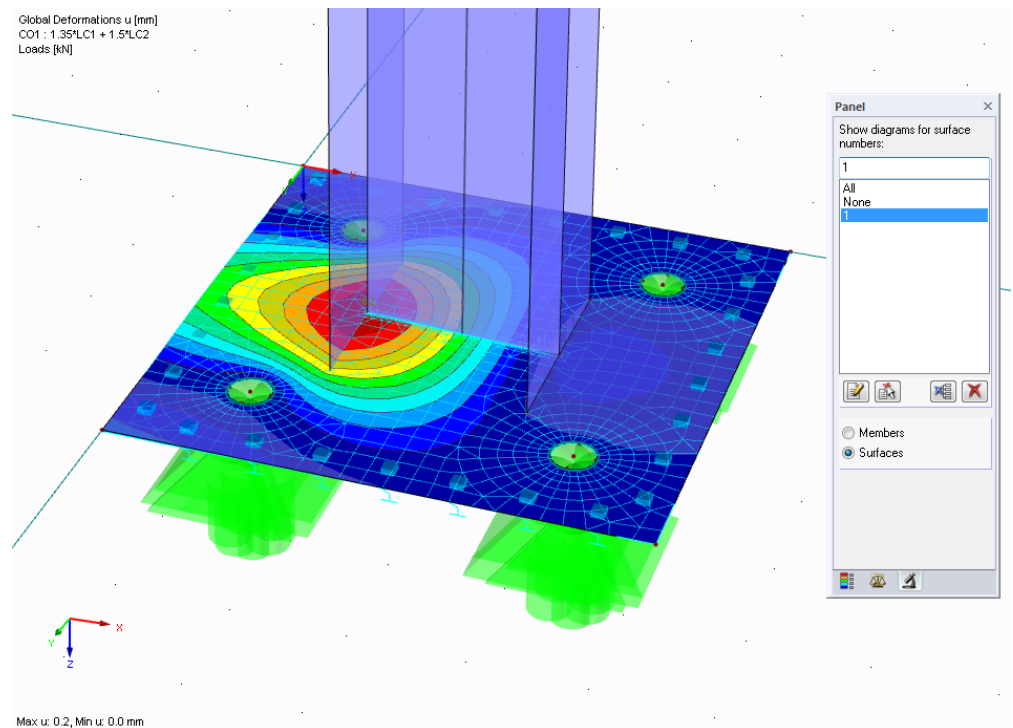


Panel tab *Factors*

The third tab of the panel offers us the possibility to display the results of individual parts of the model. For a better visualization, we can select specific members or surfaces whose results we want to see.

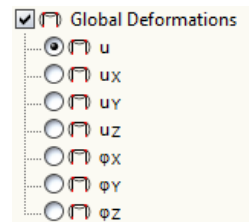


In the following figure, we have selected the base plate for the results display: It is surface **1** in the list. We can either type the number in the input field or select it graphically and then click **[Import from Selection]**.



Displaying deformations

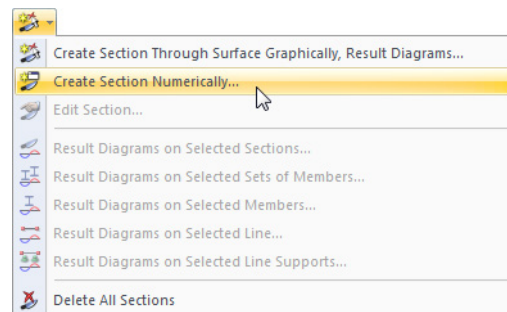
To display the deformations, we select the **Global Deformations** check box in the **Results** navigator. In the submenu, we can choose the type and direction of the deformation.



Setting deformations in *Results* navigator

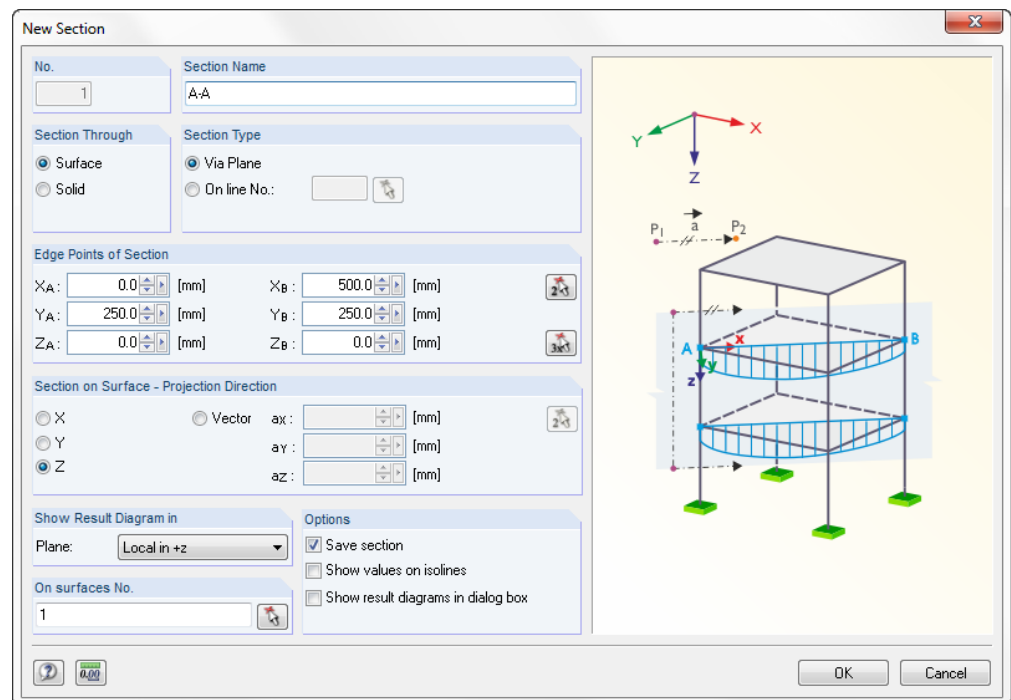
Displaying results in sections

To display the results along a section, we select **Create Section Numerically** on the menu of the **[Define Section and Result Diagrams]** button.



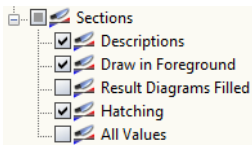
List button for sections and result diagrams

In the **New Section** dialog box, we select two center points of the base plate sides as *Edge Points*.

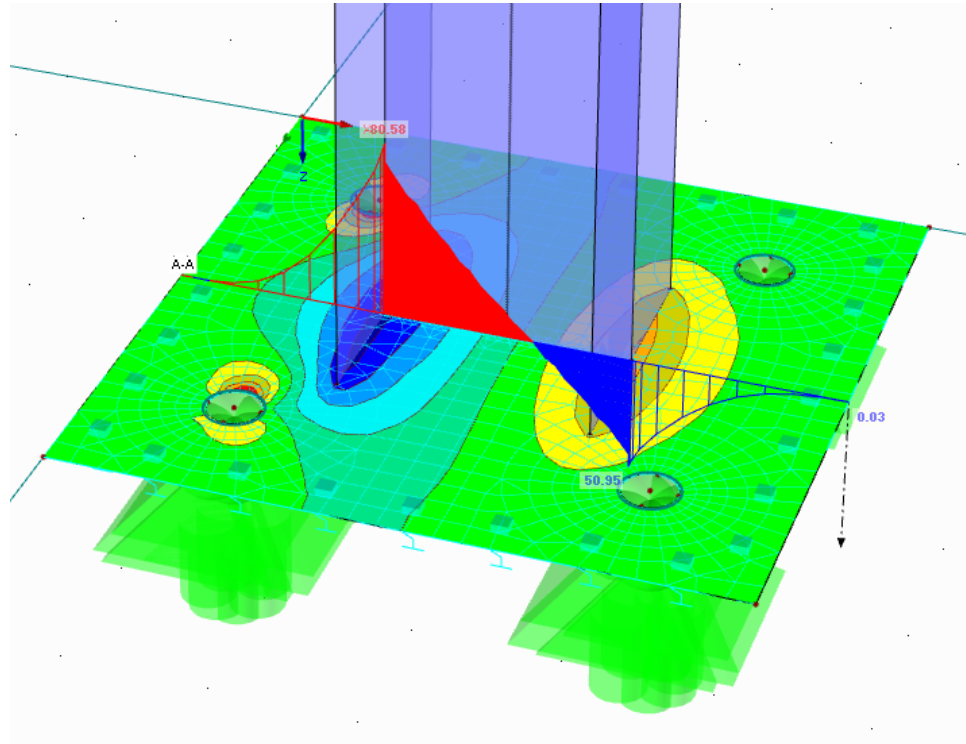


Dialog box *New Section*

We confirm the input by clicking **[OK]**.



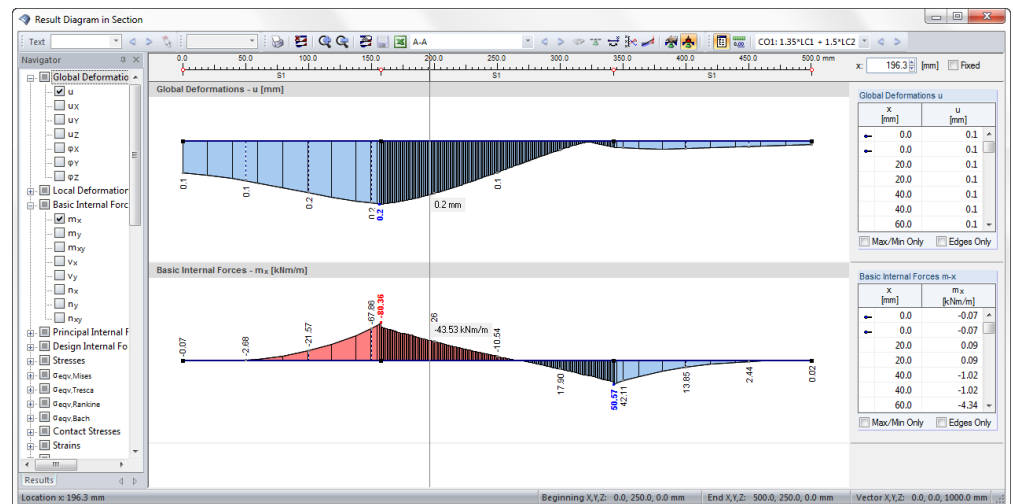
If the view of the sections is set in the **Display** navigator, the sections are shown in the model. The **Display** navigator contains some more options to control the results display of a section.



Section in base plate

There is also the possibility to display the results in a diagram. We right-click the section. On the context menu, we select **Result Diagrams**.

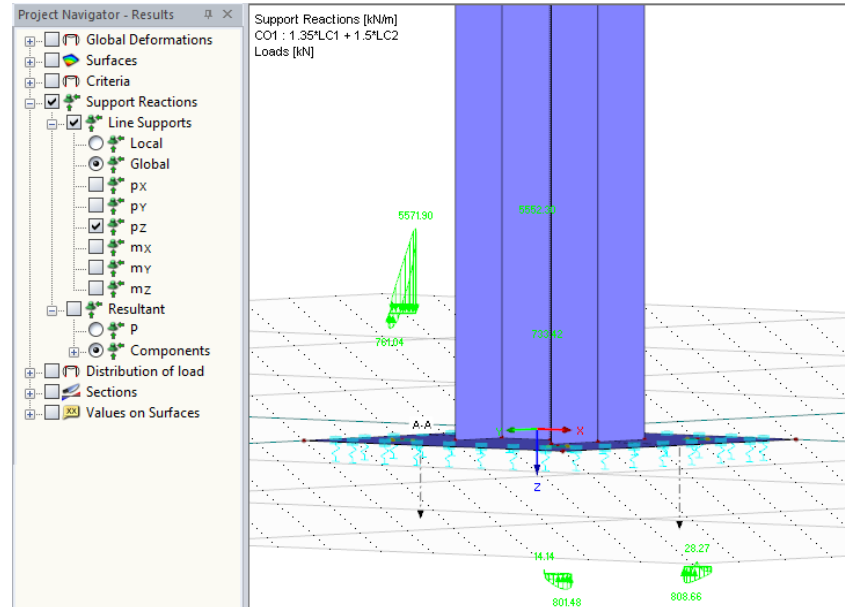
The **Result Diagram in Section** dialog box opens. Here we can review several types of results at the same time. On the right side, the selected results are listed numerically.



Window **Result Diagram in Section**

Displaying support reaction diagrams

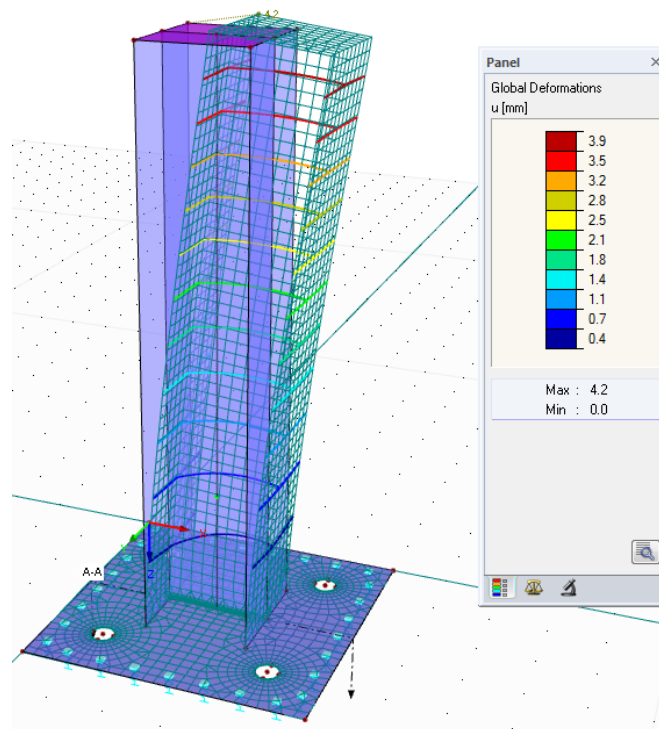
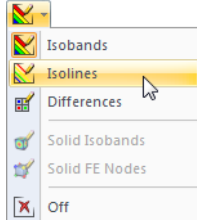
We tick the **Support Reactions** check box in the **Results** navigator. In the **Line Supports** sub-menu, we can then select specific components of the support reactions. We can either show the support reactions in relation to the **Global** or the **Local** coordinate system of the supports.



Line support forces p in global direction Z

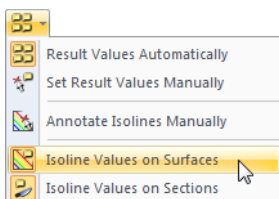
Displaying results on isolines

Sometimes it is useful to display the results as isolines. We select **Isolines** on the menu of the **[Isoband]** button.



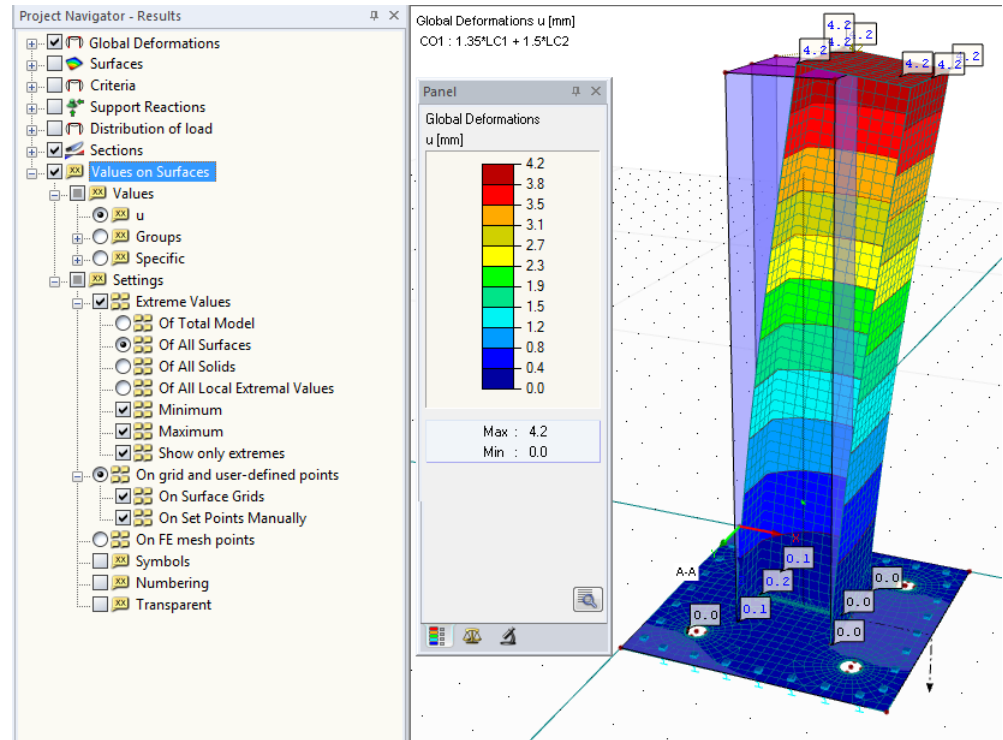
Isolines of deformations

To switch the result values of the isolines on and off, we can use the **Isoline Values on Surfaces** option on the menu of the **[Surface Result Values]** button.



Numerical results output

We can include numerical values in the graphical results by activating the **Values on Surfaces** check box. Below this check box, there are various options for the display of those values, e.g. on automatically detectable extreme locations as well as in manually definable points.



Deformations showing extreme values

3.8 RF-STEEL Surfaces

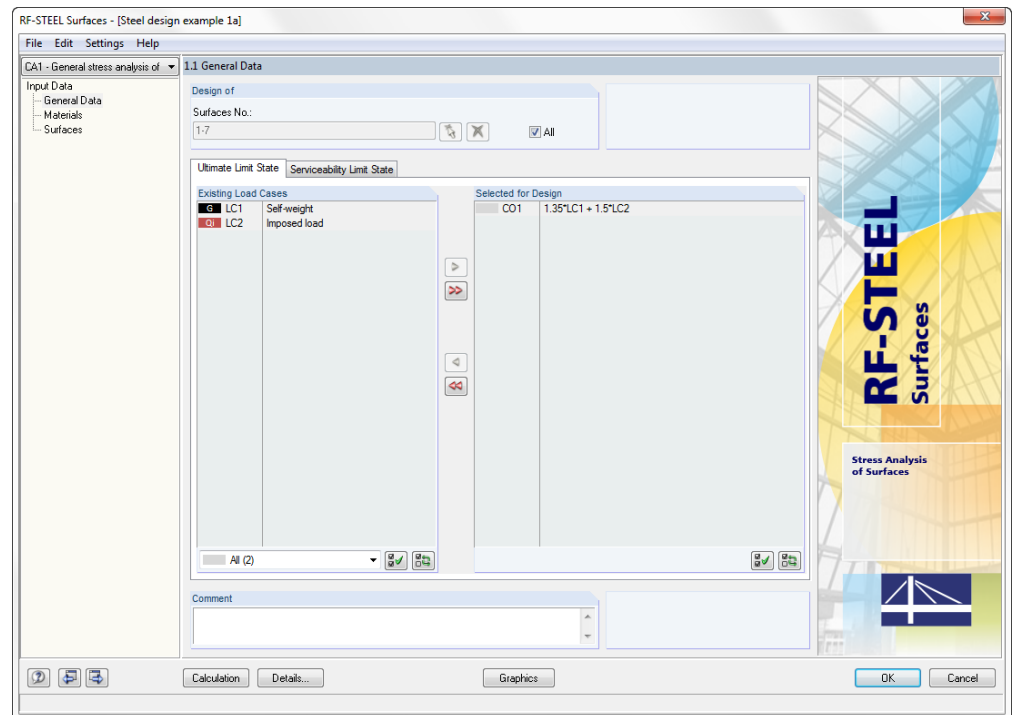
The add-on module RF-STEEL Surfaces analyzes the stresses of surface elements according to the elastic-elastic method. It includes a library with the limit stresses of common steel grades, e.g. according to EN 10025 (for the calculation according to EC 3) or DIN 18800. It is also possible to add user-defined materials.

We start this add-on module by selecting the **Add-on Modules** folder in the **Data** navigator where we double-click **RF-STEEL Surfaces**.

3.8.1 Input Data

In the module window 1.1, we set the surfaces that are to be designed. To select all surfaces of the model, we use the **All** check box.

We also have to select at least one loading for the stress analysis. For our example, we select the load combination **CO1**.

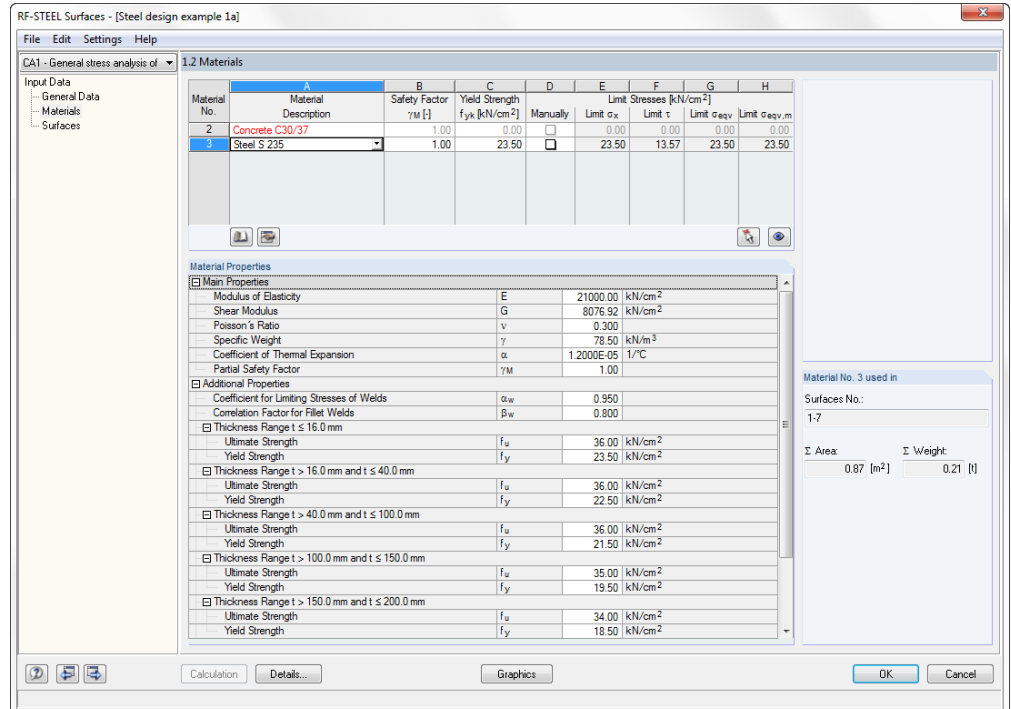


RF-STEEL Surfaces window 1.1 General Data



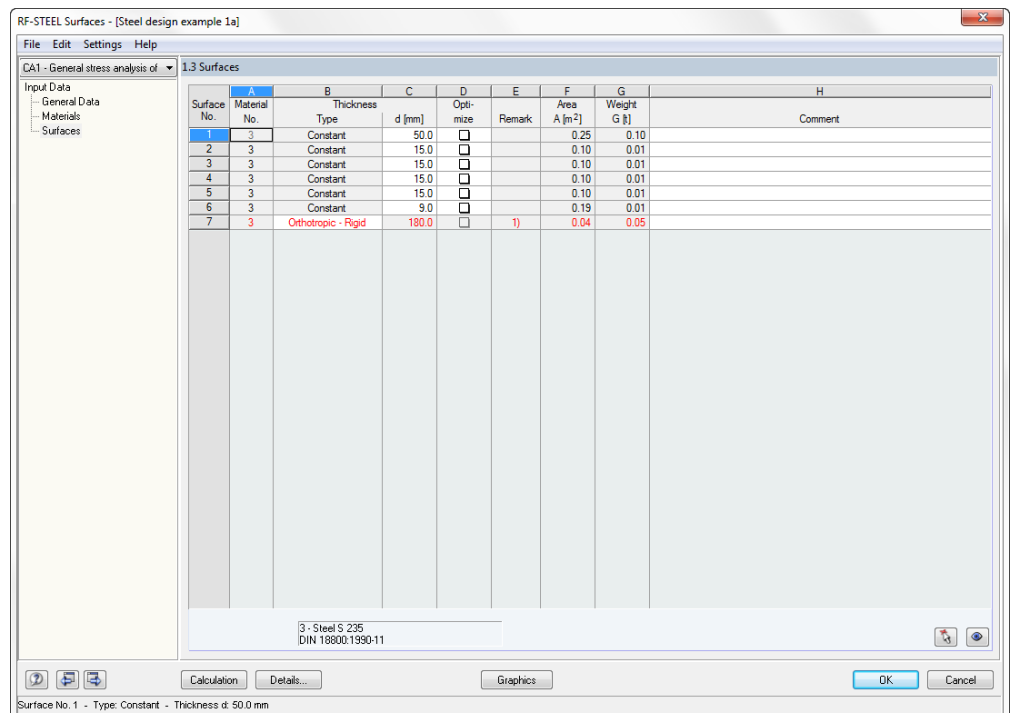
On the left side of the module window, we see a navigator showing all available windows. We can directly open a module window there by clicking the corresponding entry. It is also possible to use the **[Previous/Next Window]** buttons.

In window **1.2 Materials**, all materials used in RFEM are listed. The limit stresses depend on the thickness of the structural components and are adjusted automatically. If we activate the **Manually** check box in column D, we can edit the limit stresses for the stress analysis. Invalid materials are shown in red. If we click a material, all material parameters are shown below.



Window 1.2 Materials

Window **1.3 Surfaces** lists all surfaces with their thicknesses. The thicknesses can be modified, which influences the stresses. The changes are not transferred to RFEM automatically. Therefore, any modifications should be exported to RFEM for a new calculation of the results.



Window 1.3 Surfaces

3.8.2 Results

Calculation

After the **[Calculation]**, the window **2.1 Stresses by Load Case** is shown. It presents the maximum design ratios of the stress types that have been set in the **Details** dialog box.

RF-STEEL Surfaces - [Steel design example]

File Edit Settings Help

CA1 - General stress analysis of ▾

Input Data

- General Data
- Materials
- Surfaces

Results

- Stresses by Load Case
- Stresses by Material
- Stresses by Surface
- Stresses by Line
- Parts List

2.1 Stresses by Load Case

Load- ing	A Surface No.	B FE Mesh Point No.	C Point X	D Coordinates [mm] Y	E Z	F Symbol	G Stress [kN/cm ²] Existing	H Limit	I Stress Ratio [-]	J
CO1										
	1	380	112.9	115.3	0.0	$\sigma_{1,max}$	13.86	12.41	1.12	
	5	34	157.5	350.0	0.0	$\sigma_{1,+}$	64.28	23.50	2.74	
	1	589	157.5	294.1	0.0	$\sigma_{2,+}$	-18.24	21.50	0.85	
	5	34	157.5	350.0	0.0	$\sigma_{1,-}$	47.63	23.50	2.03	
	2	29	342.5	150.0	0.0	$\sigma_{2,-}$	-27.67	23.50	1.18	
	5	34	157.5	350.0	0.0	$\sigma_{1,m}$	55.95	23.50	2.38	
	3	31	342.5	350.0	0.0	$\sigma_{2,m}$	-23.68	23.50	1.01	
	5	34	157.5	350.0	0.0	$\sigma_{eqv,max}$	58.63	23.50	2.49	
	5	34	157.5	350.0	0.0	$\sigma_{eqv,m}$	50.57	23.50	2.15	

☒ FE mesh points

☐ Grid points

Max: 2.74 > 1

Calculation

Details...

Graphics

OK

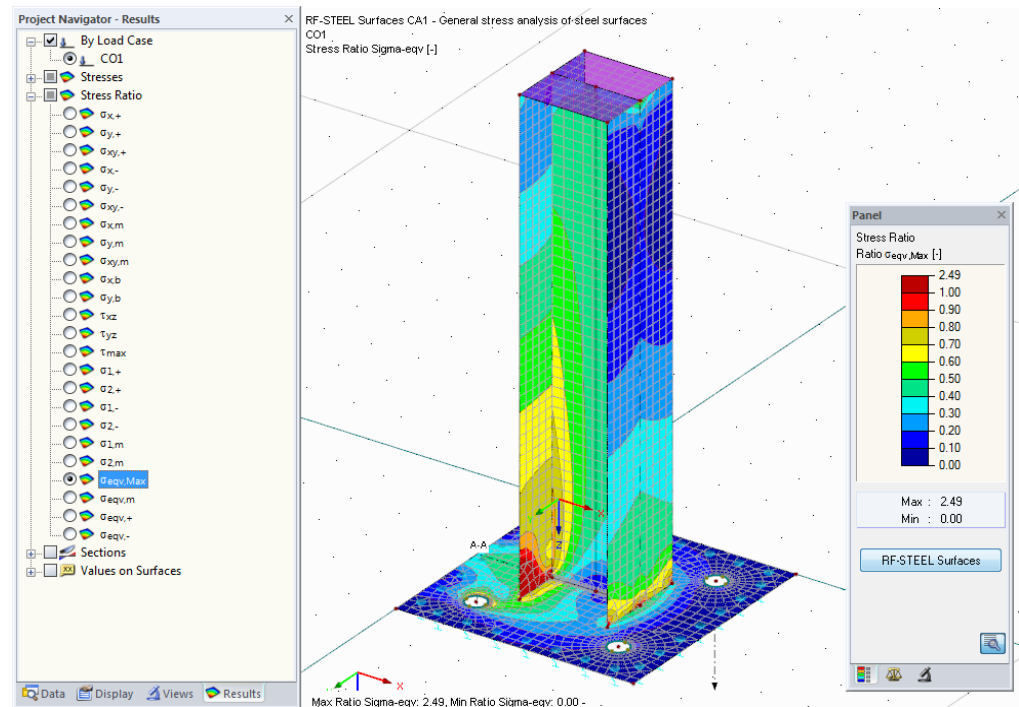
Cancel

Window 2.1 Stresses by Load Case

The windows 2.2 to 2.4 show the maximum stresses and ratios by material, surface or line.

Graphics

We click **[Graphics]** to return to the work window where we can see the results graphically.



Graphical results in RFEM work window

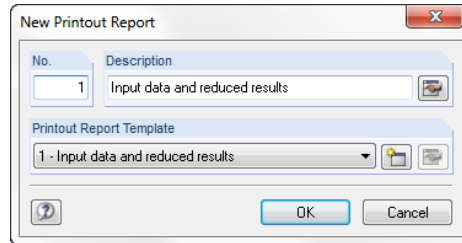
In the **Results** navigator, we can set the type of stress and the corresponding ratios.

3.9 Printout Report



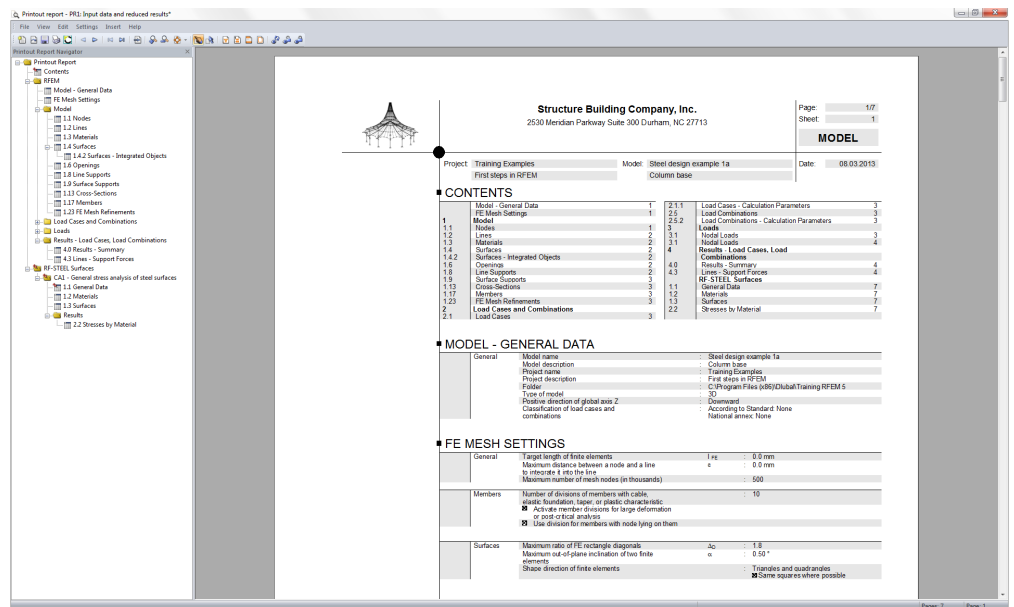
We click [**Current Printout Report**] in the toolbar to open the printout report where we can prepare the results that are to be printed.

The **New Printout Report** dialog box opens.

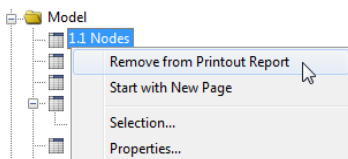


Dialog box *New Printout Report*

We confirm this dialog box with [**OK**], thus opening the printout report.



Printout report



On the left side of the printout report, we see the navigator showing all selected chapters. The main window shows the preview of the printout. To go to a particular chapter of the report, we click the corresponding entry in the navigator. We can open the context menu of a chapter by right-clicking that entry. In the context menu, we can remove a chapter or start it on a new page. By using the **drag-and-drop** function, we can move a chapter to a different location.

When we open the printout report, the [**Grab Mode**] is preset. The mouse pointer becomes a hand when it is placed within the printout report, allowing us to grab it and go to the desired part of the report.

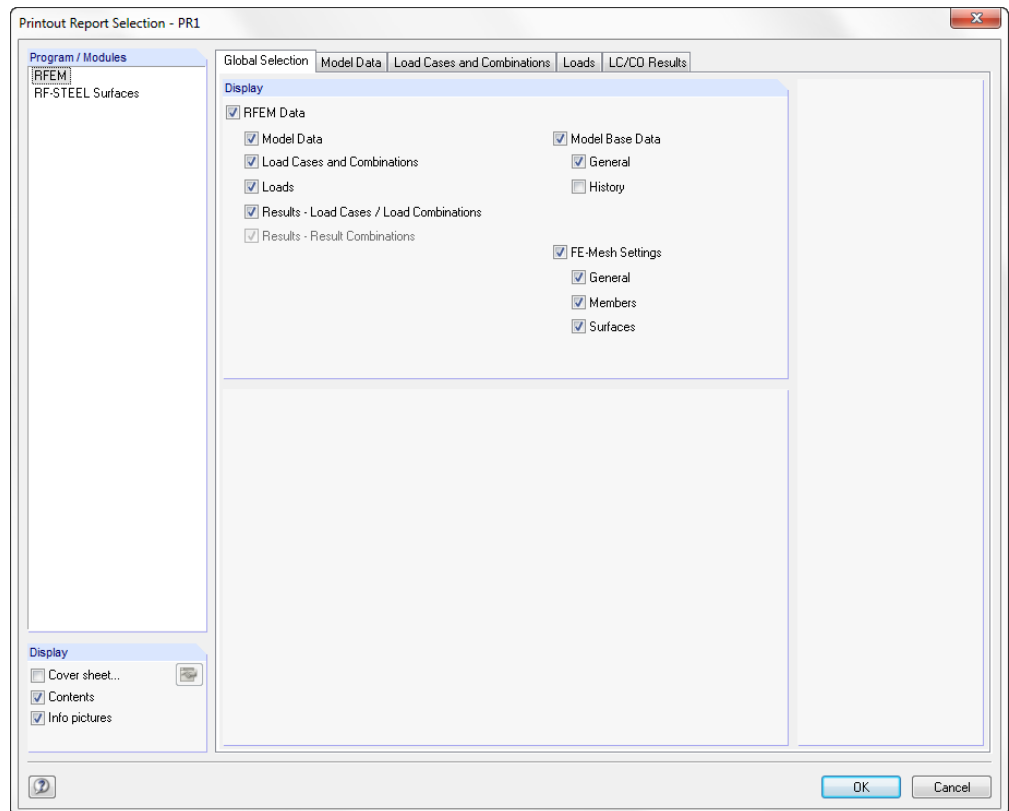
To select and edit chapters, we can click [**Select Mode**]. This mode allows us to select some chapter or part in the work window and then to delete or edit it (after double-clicking it).

To zoom the display of the dialog box, we use the [**Zoom in**] or the [**Zoom out**] button.

Since FE analyses lead to very big data, the printout report also becomes quite large. Therefore, it is recommended to select the data before printing. We use the [**Select Topics for Printout Report**] button for that.



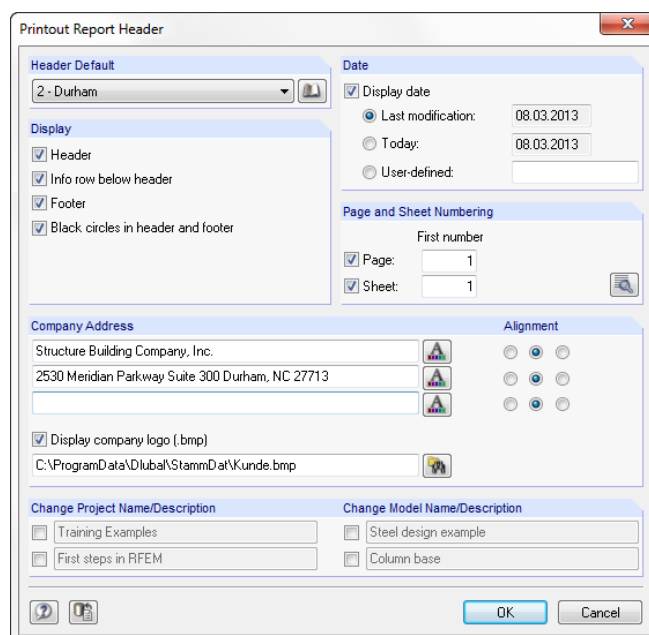
The **Printout Report Selection** dialog box opens.



Dialog box *Printout Report Selection*

The **Program / Modules** section on the left shows all modules containing data. Each module has a **Global Selection** tab. All other tabs are controlled by the settings in this tab. If the **Model Data** check box in the **Global Selection** tab is activated, we can select in the **Model Data** tab which structural data we want to include in the printout report.

To change the header, we click [**Header Setup**]. The **Printout Report Header** dialog box opens.

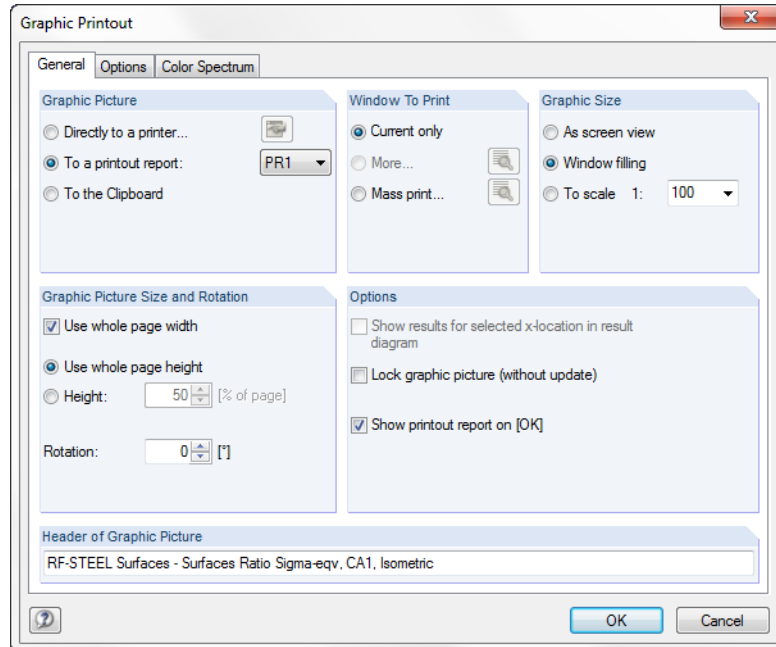


Dialog box *Printout Report Header*



Graphics can be printed directly from the RFEM work window into the printout report. To do this, we close the printout report via the [X] button and saving the data. Then we click **[Print Graphic]**. The **Graphic Printout** dialog box opens.

The **General** tab of this dialog box controls whether the picture is sent to the printer, to the printout report or to the clipboard.



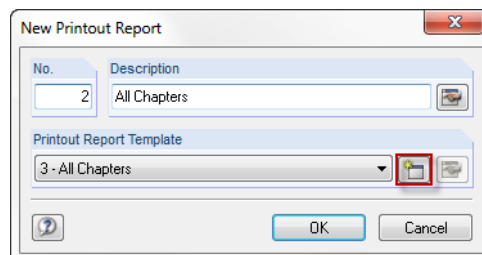
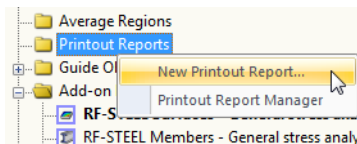
Dialog box *Graphic Printout*, tab *General*

User-defined template

In RFEM, we can define printout report templates for different purposes. Those templates can then be used to quickly create a user-defined printout report.

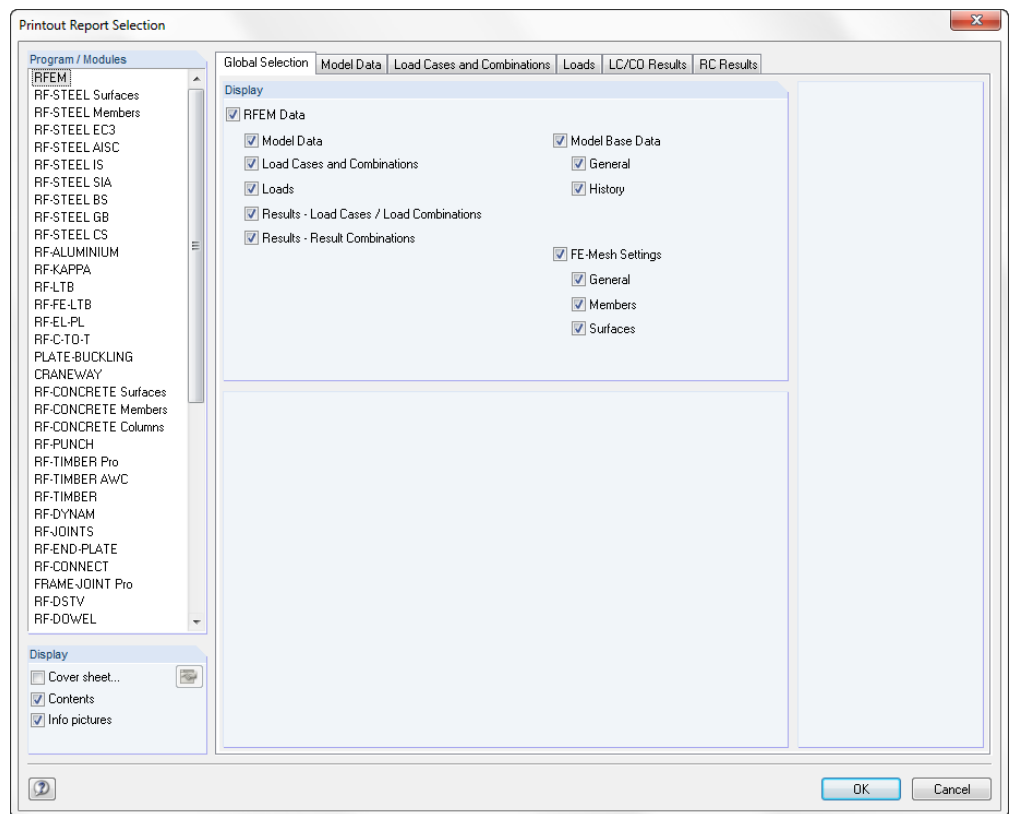
We right-click the **Printout Reports** entry in the **Data** navigator. On the context menu, we select **New Printout Report**.

The **New Printout Report** dialog box appears. We click the **[Edit Selection and Create New Printout Report Template]** button.

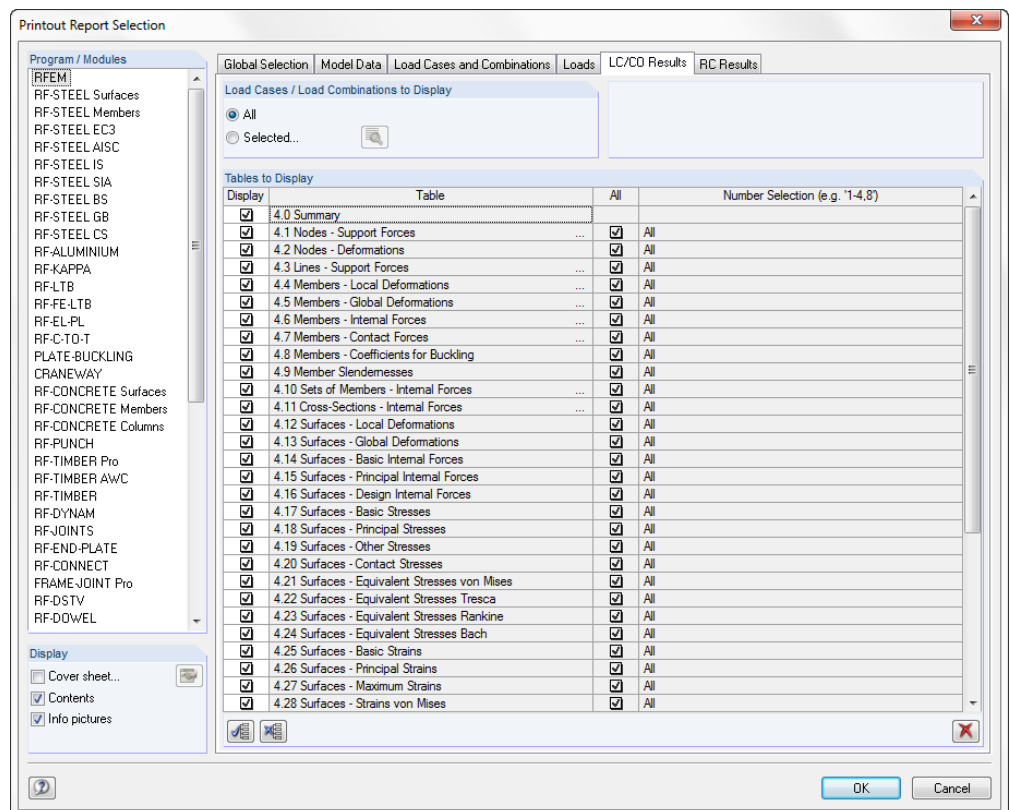


Dialog box *New Printout Report*

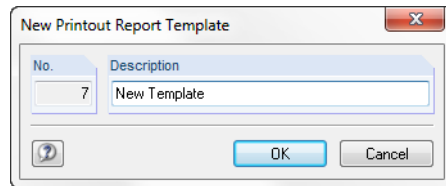
The **Printout Report Selection** dialog box opens (see figure below). It is similar to the familiar selection dialog box. On its left side, however, this dialog box shows all the installed modules rather than just the ones containing data. This allows us to create a generally applicable report template that is independent of the used add-on modules.

Dialog box *Printout Report Selection*, tab *Global Selection*

In the **Global Selection** tab, we keep the selection of all data. Then we go to the **LC/CO Results** tab. Here we can select all results that we want to be included in the template.

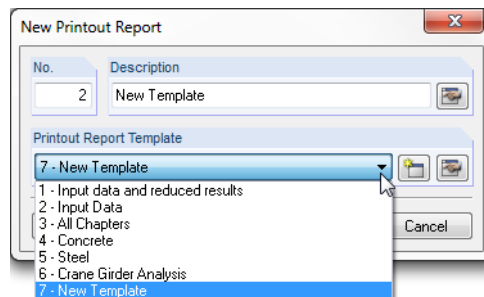
Dialog box *Printout Report Selection*, tab *LC/CO Results*

After [OK], RFEM asks us to enter a description for the printout report template.



Dialog box *New Printout Report Template*

Having entered a description and confirmed it with [OK], we can see that the new template is included in the list of templates. We select this template and then clicking [OK].

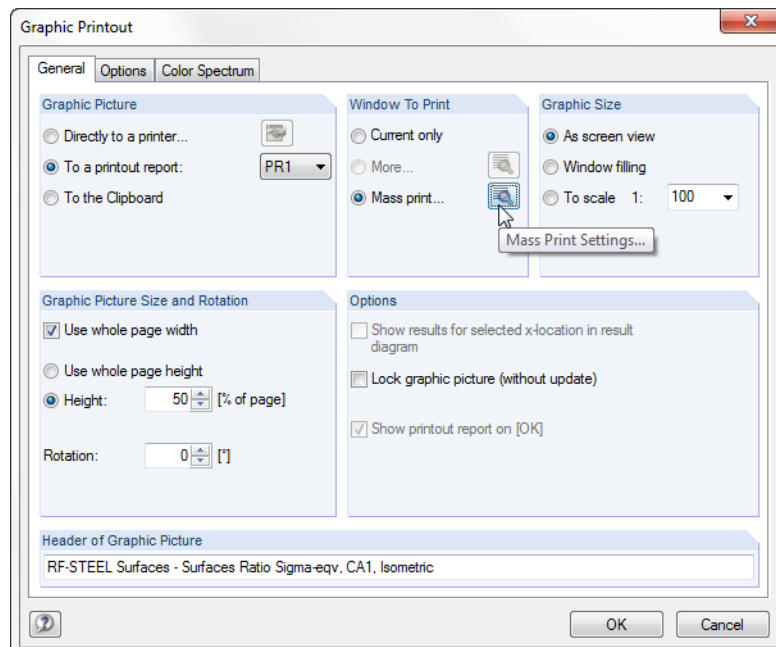


Selecting printout report template

Mass Print

To print, for example, the deformations of all load cases and load combinations, we click [Print Graphic].

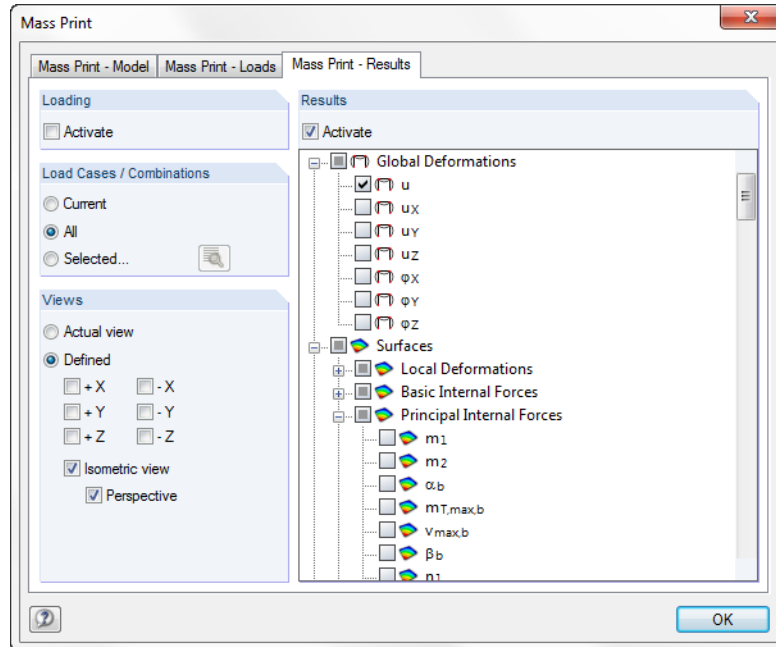
The **Graphic Printout** dialog box opens. In the **Window to Print** section, we select the **Mass Print** option. Then we click [Mass Print Settings].



Dialog box *Graphic Printout*

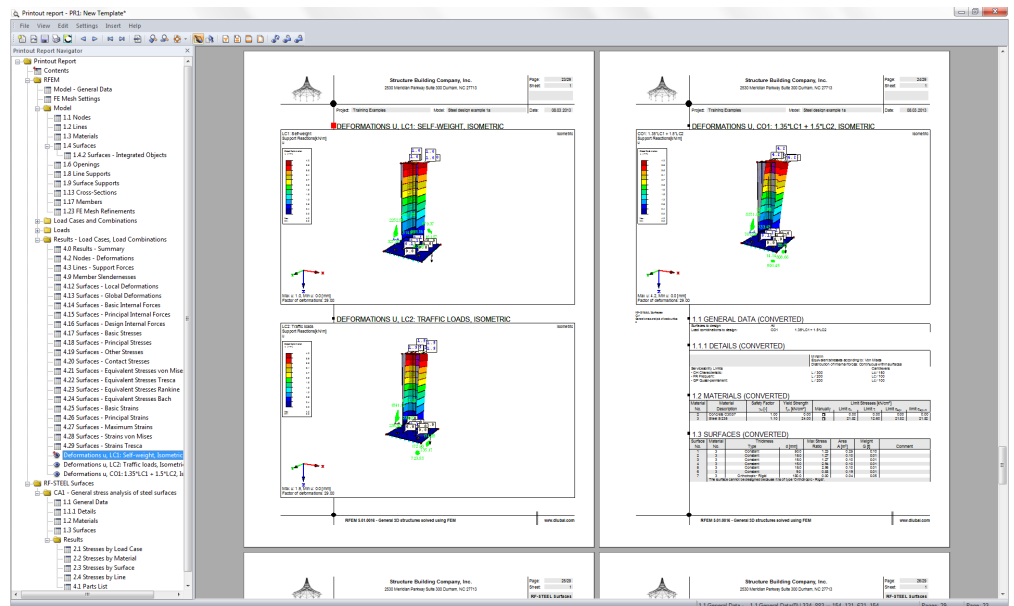
The **Mass Print** dialog box appears. It has three tabs where we can select the desired graphics.

We go to the **Mass Print - Results** tab and select the **u** check box under **Global Deformations**. Additionally we select **All** in the **Load Cases / Combinations** section. In the **Views** section, we can then define the view of the graphics.

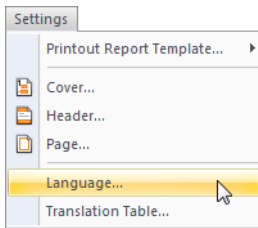


Dialog box *Mass Print*

After [OK], RFEM prints the graphics into the printout report.



Printout report with automatic graphics



Languages

The data can be printed in different languages. To change a language for the print, we select **Language** on the **Settings** menu of the printout report. The **Languages** dialog box opens.

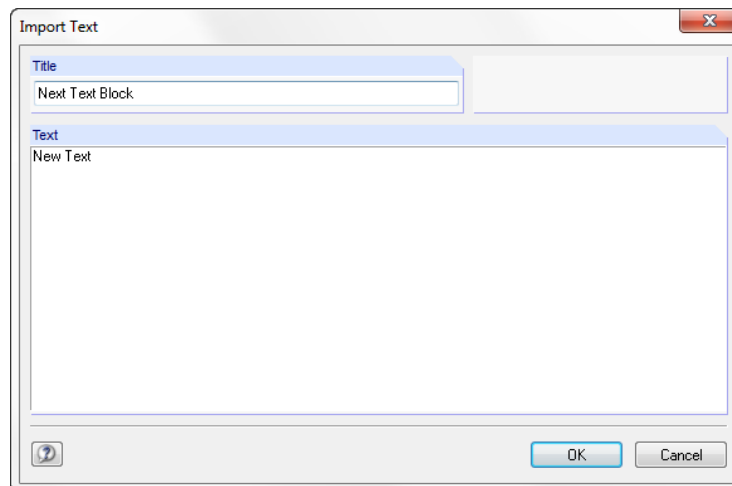
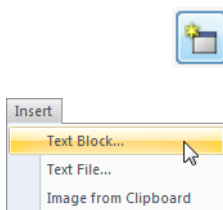


Dialog box *Languages*

To create additional languages, we can use the **[Create New Language]** button.

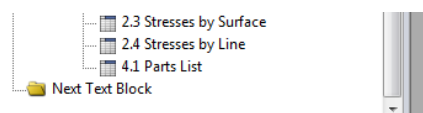
Inserting user-defined text

It is also possible to insert user-defined texts. We select **Text Block** on the **Insert** menu. The **Import Text** dialog box opens. In the **Title** input field, we can type a user-defined header. In the **Text** field, we can enter any text.



Dialog box *Import Text*

After **[OK]**, the text is included in the printout report.

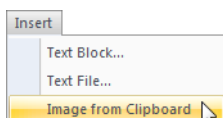


NEXT TEXT BLOCK
New Text

User-defined text in printout report

Inserting image from Clipboard

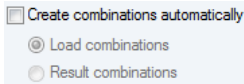
To insert any pictures in the printout report, we first copy the graphic to the clipboard. Then we can select **Image from Clipboard** on the **Insert** menu, thus inserting the graphic.



4. Stability Analysis

This example presents a second-order stability analysis.

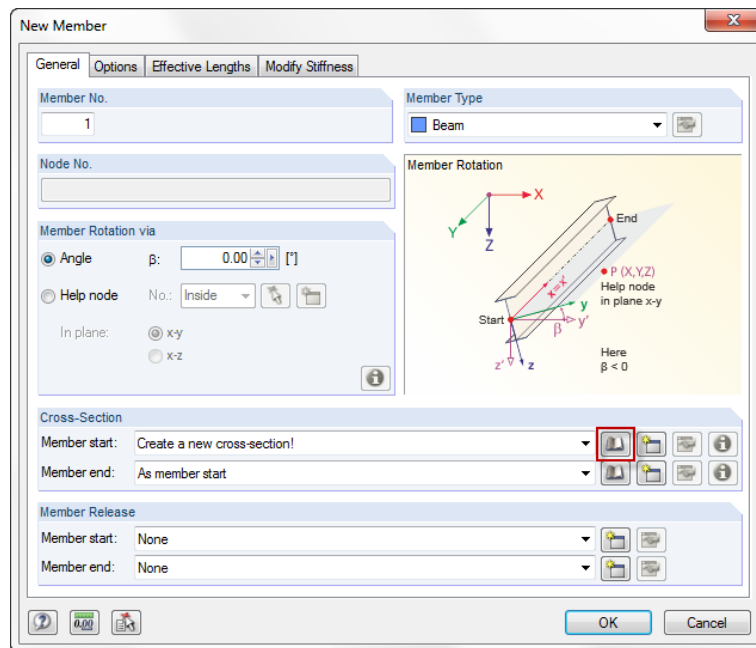
We create a **[New Model]**. In the **General Data** dialog box, we type **Example 2** as **Model Name** and **Cantilever** as **Description**. We disable the option *Create combinations automatically*.



4.1 Model Data

4.1.1 Beam

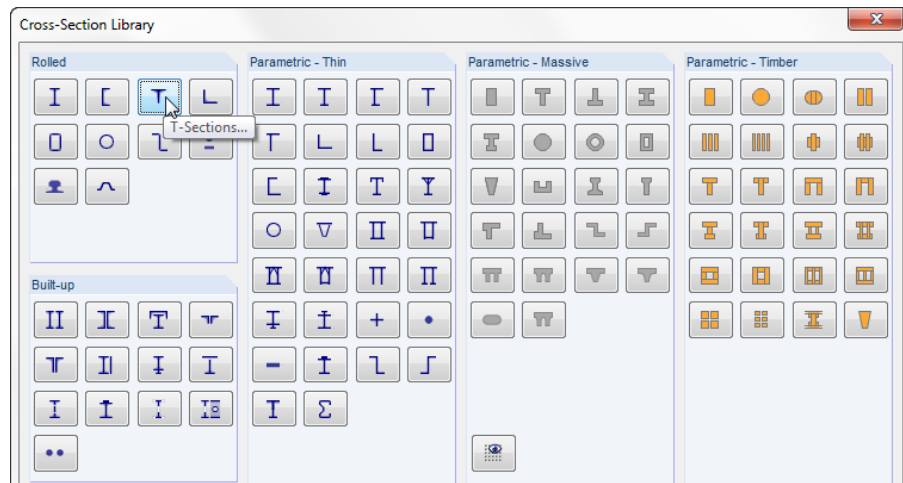
The model consists of a 1/2 IPE 500 Cross-Section with a length of 3.0 m. We open the **New Member** dialog box by clicking **[New Member Single]**.



Dialog box *New Member*

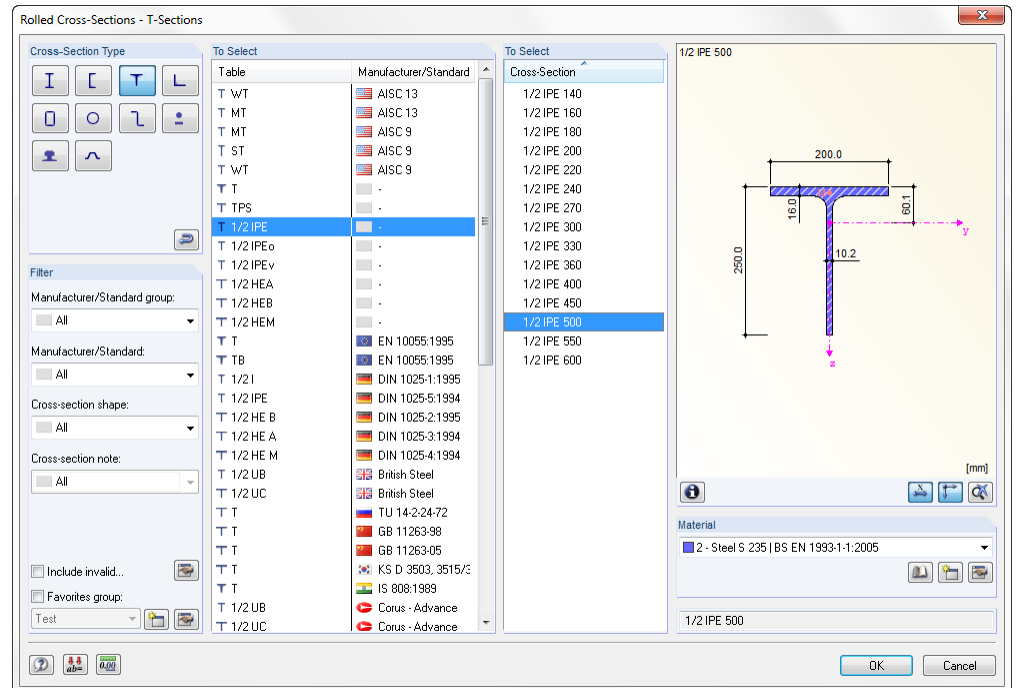
In the **Member start** row of the Cross-Section dialog box section, we click **[Import from Cross-Section Library]**. The **Cross-Section Library** dialog box opens.

We click **[T-Sections]** in the **Rolled** dialog box section.



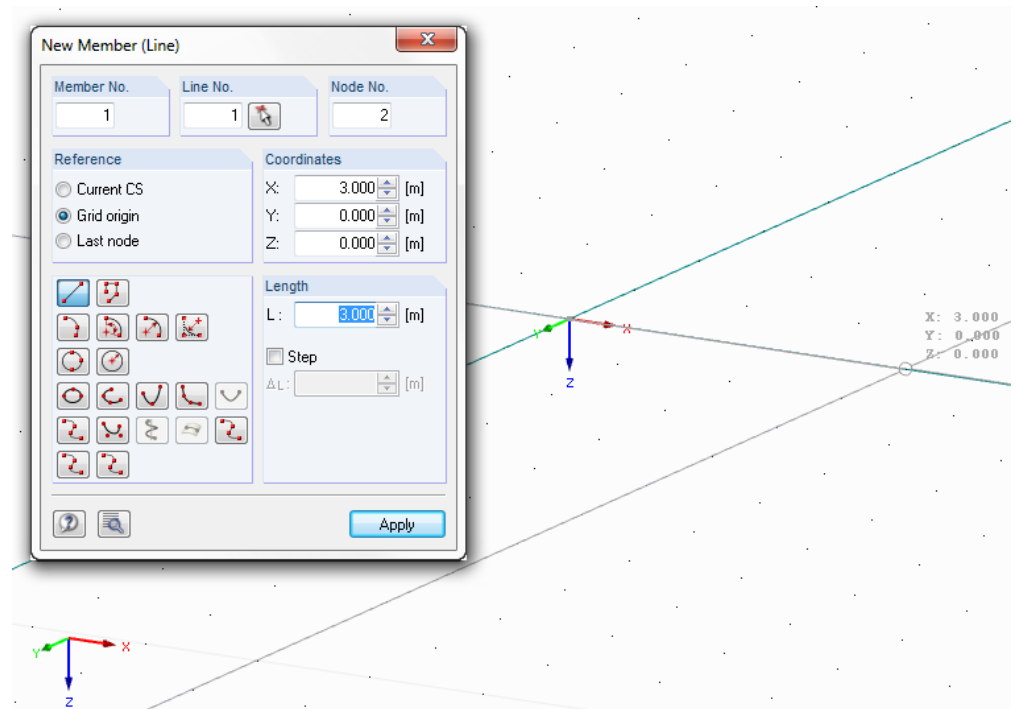
Cross-Section Library

The **Rolled Cross-Sections - T-Sections** dialog box opens. The **To Select** dialog box section on the left shows a list with cross-section tables. We select the cross-section table **1/2 IPE**. Then we can select the **1/2 IPE 500** cross-section in the list on the right.



Selecting 1/2 IPE 500 cross-section

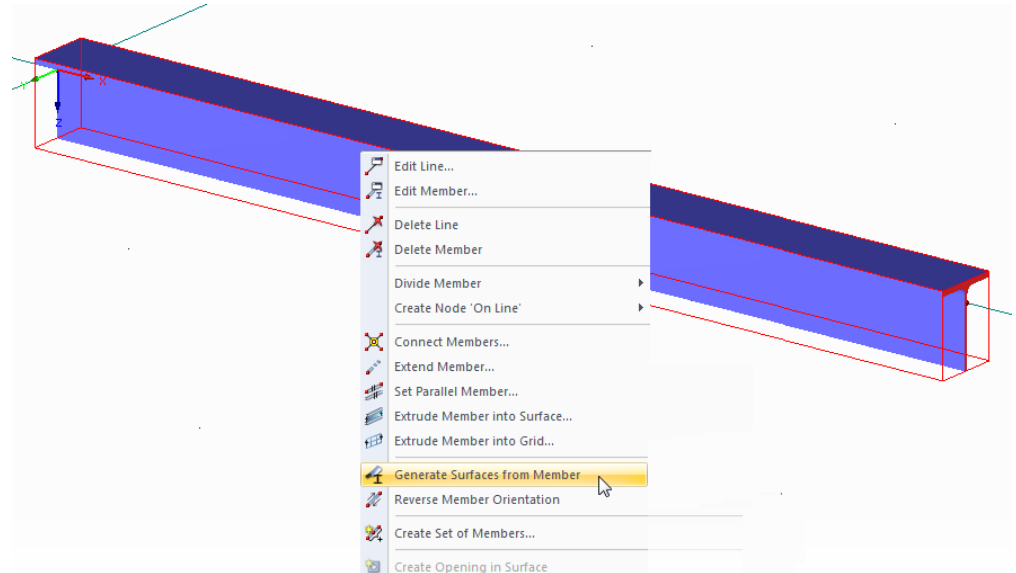
We confirm the dialog boxes with **[OK]** and return to the RFEM work window. The mouse pointer becomes a reticle. We use this reticle to select the start node and end node of the member. The member has the length of **3.0 m**. We select the origin **[0;0;0]** as the member start and the grid point **[3;0;0]** as the member end.



Creating member

We finish the input by right-clicking anywhere in the work window.

We want to create openings in the web of the beam. To do this, we have to divide the member into surfaces. We open the context menu for editing the member by right-clicking it. Then we select **Generate Surfaces from Member**.

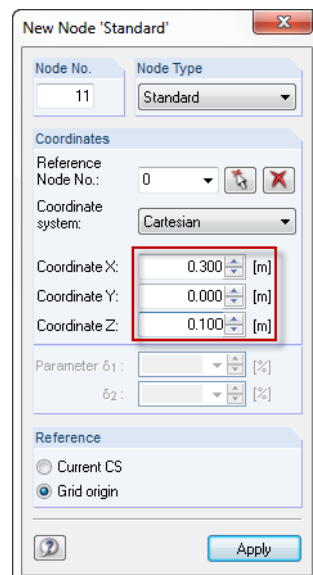


Member context menu

After that we set the work plane in the web by clicking **[Work Plane XZ]**.

4.1.2 Openings

We next create the center point of the first circular opening with the coordinates **[0.3; 0.0; 0.1]** by using the **[New Node]** function.



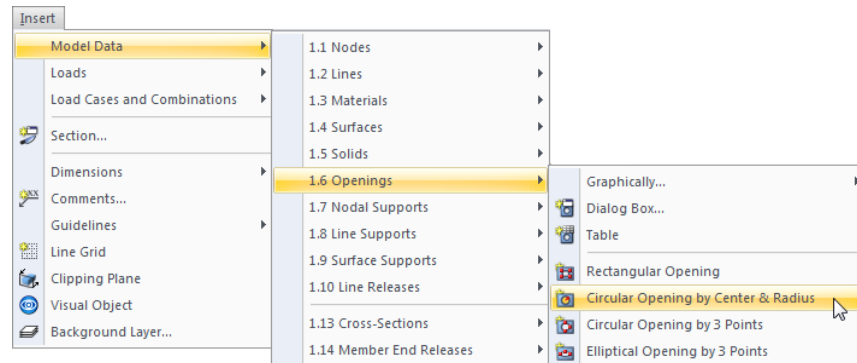
Dialog box New Node 'Standard'



We click **[Apply]** and then close the dialog box.

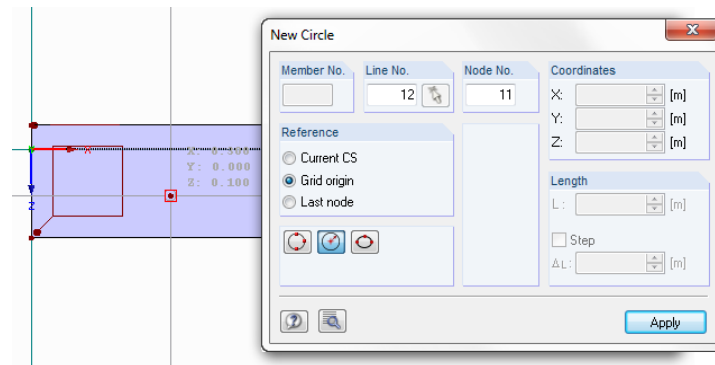
Next we create the opening by selecting

Insert → Model Data → Openings → Circular Opening by Center & Radius.



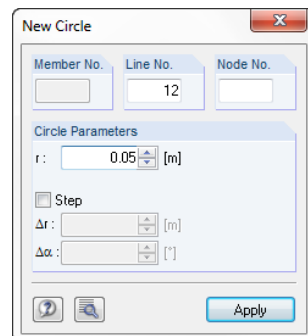
Menu Insert

The **New Circle** dialog box appears. We use the reticle of the mouse pointer to select the node that we have just created as the center point of the circle.



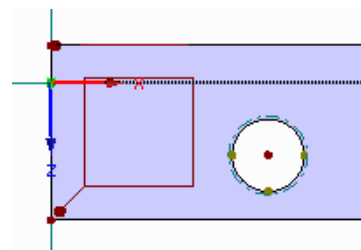
Selecting center point of circle

After the selection of the node, a new window opens. We enter the circle radius **0.05m**.



Dialog box New Circle

We **[Apply]** the circle parameters to create the opening.

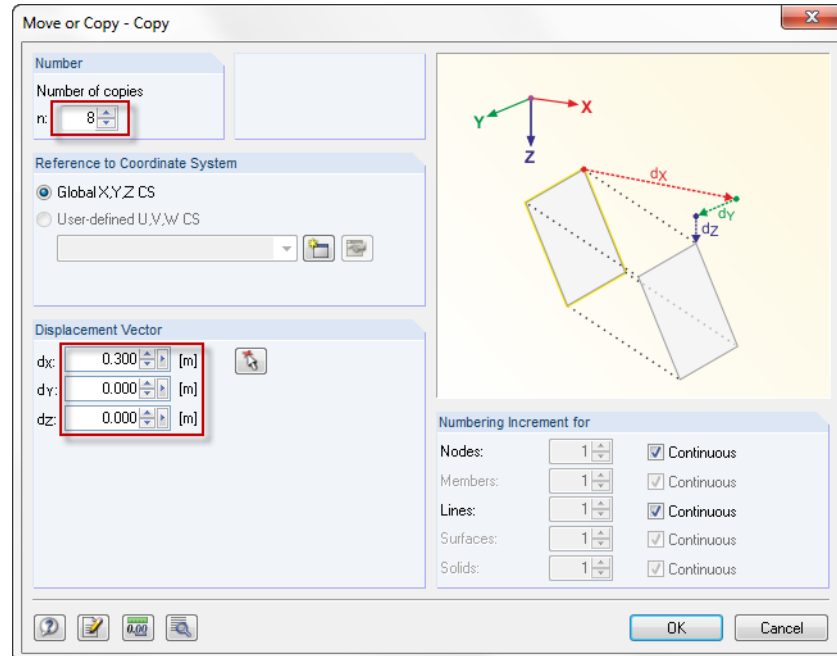


Circular opening

We create the other openings quickly by copying the opening eight times. Before we do this, we have to select the opening with all nodes and lines. If we start the selection within the surface, it is useful to press the **[Alt]** key so that we do not select the surface, too.

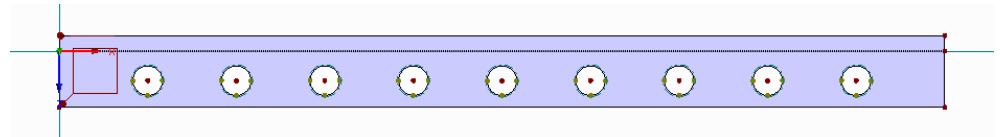


We click **[Move and/or Copy]** to open the **Move or Copy** dialog box. We enter **8** as **Number of copies**. In the **Displacement Vector** section, we enter the distance **0.3 m** for d_x .



Dialog box **Move or Copy - Copy**

After **[OK]**, eight new openings are created.

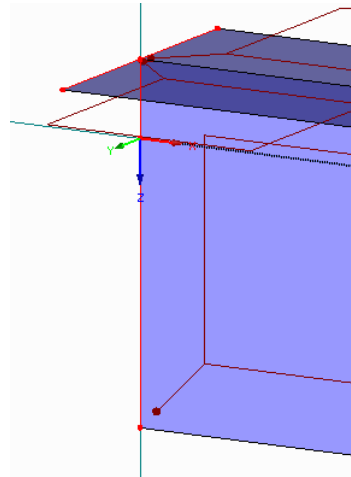


Openings

4.1.3 Supports

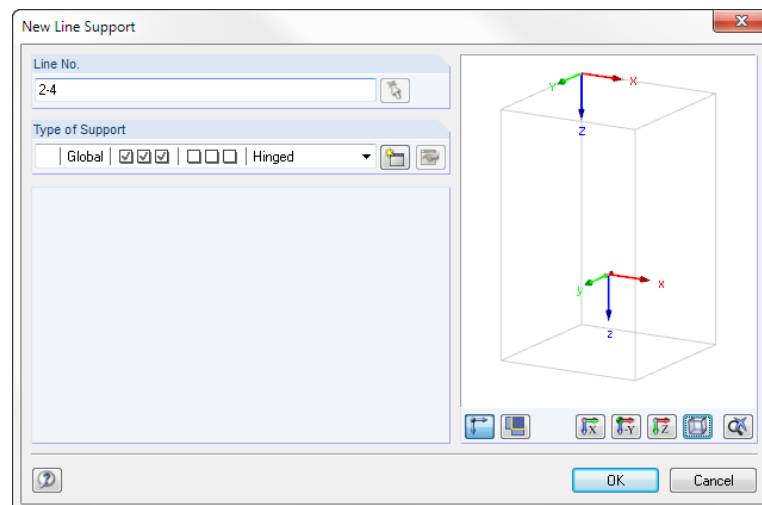
To finish the input of the model data, we define the support conditions. The beam is to be supported on its left side (plane of the origin).

We select all lines that lie in the plane of the origin. Then we click **[New Line Support]**.



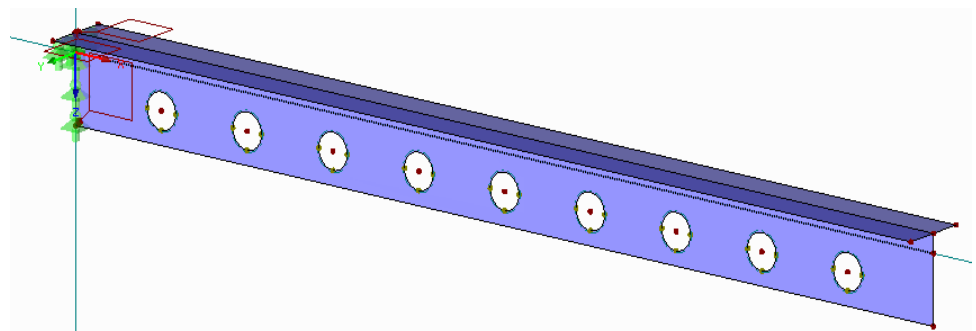
Selecting lines of support

The **New Line Support** dialog box opens. The selected lines are preset. We select **Hinged** as support type.



Dialog box *New Line Support*

After **[OK]**, the restraint is displayed as hinged line support of the boundary lines to the left.



Model with openings and support

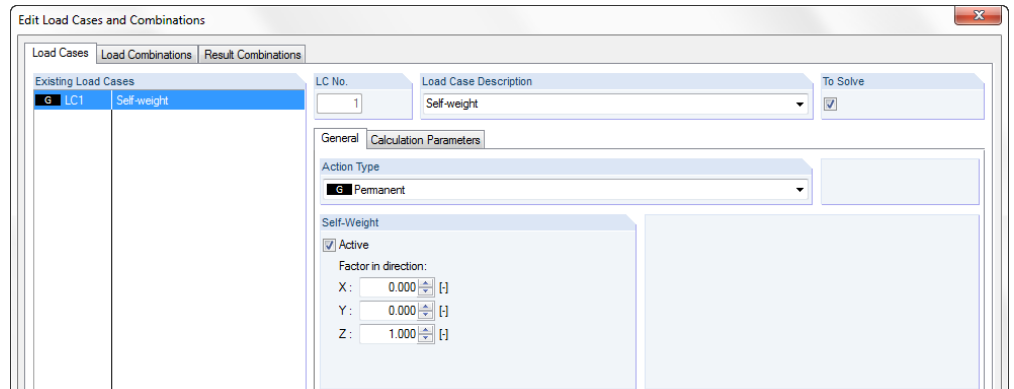
4.2 Load Data

We create two load cases and one load combination of both load cases.

4.2.1 LC1: Self-Weight



The first load case contains only the self-weight of the beam. We click **[New Load Case]**. The **Edit Load Cases and Combinations** dialog box opens showing the **Load Cases** tab.



Dialog box *Edit Load Cases and Combinations*

The self-weight as the action type **Permanent** is considered with the factor 1.0 in **Z**-direction. In the **Load Case Description** section, we enter **Self-weight**.

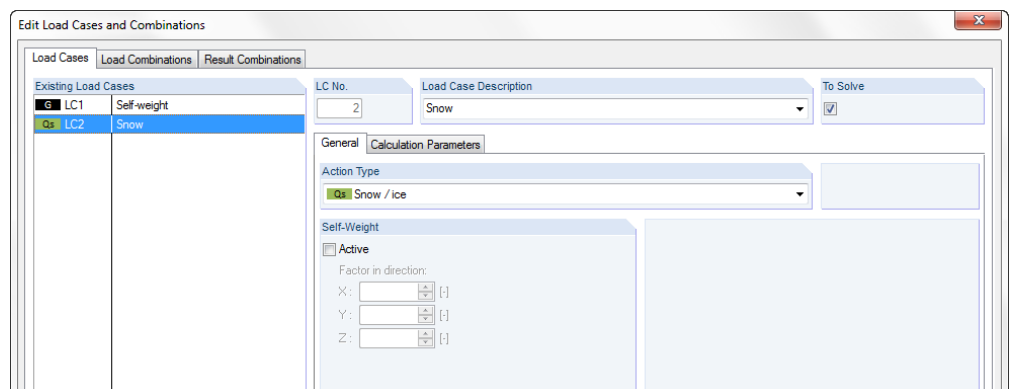
By clicking **[OK]**, the first load case is created.

4.2.2 LC2: Snow



In the second load case, we define a line load that represents a snow load. Again, we click **[New Load Case]**. In the **Edit Load Cases and Combinations** dialog box, the self-weight is deactivated now.

When we select **Snow** as **Load Case Description** from the drop-down list, the **Action Type** is automatically set to **Snow / ice**.

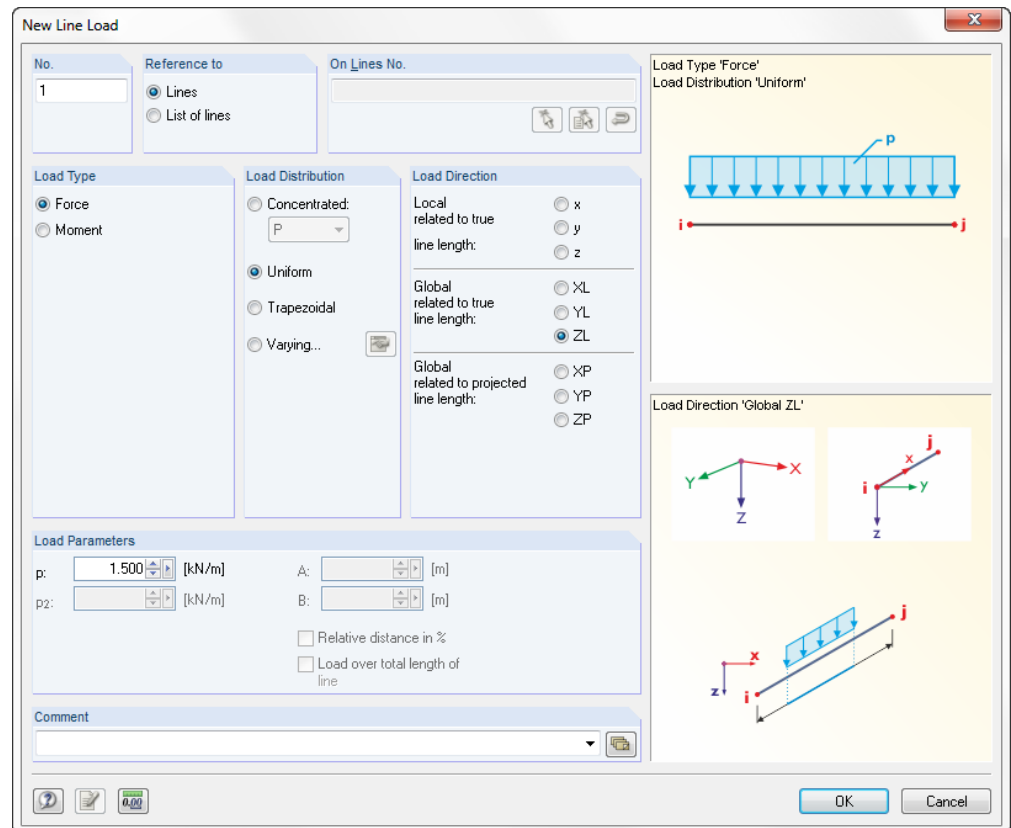


Dialog box *Edit Load Cases and Combinations*

We confirm the input by clicking **[OK]**.



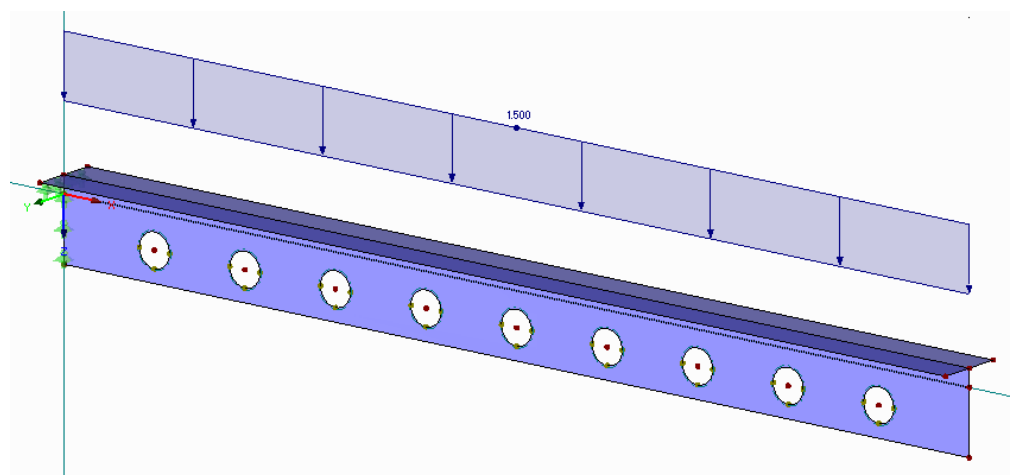
Now we can define the line load. We click **[New Line Load]**. The **New Line Load** dialog box opens. We select a **Uniform** load distribution in the direction of the global Z-axis (**ZL**). We enter the value **1.5 kN/m** for **p** in the **Load Parameters** section.



Dialog box *New Line Load*

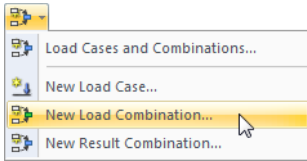
We click **[OK]** to confirm the input.

Now we click the line to which the load is to be applied: It is the center line of the flange.



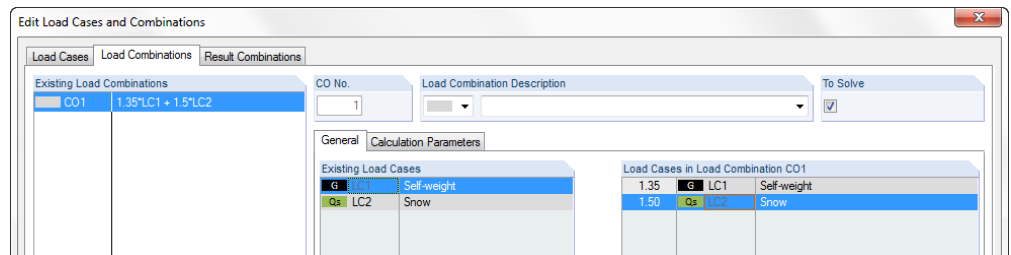
Snow load

4.3 Load Combination



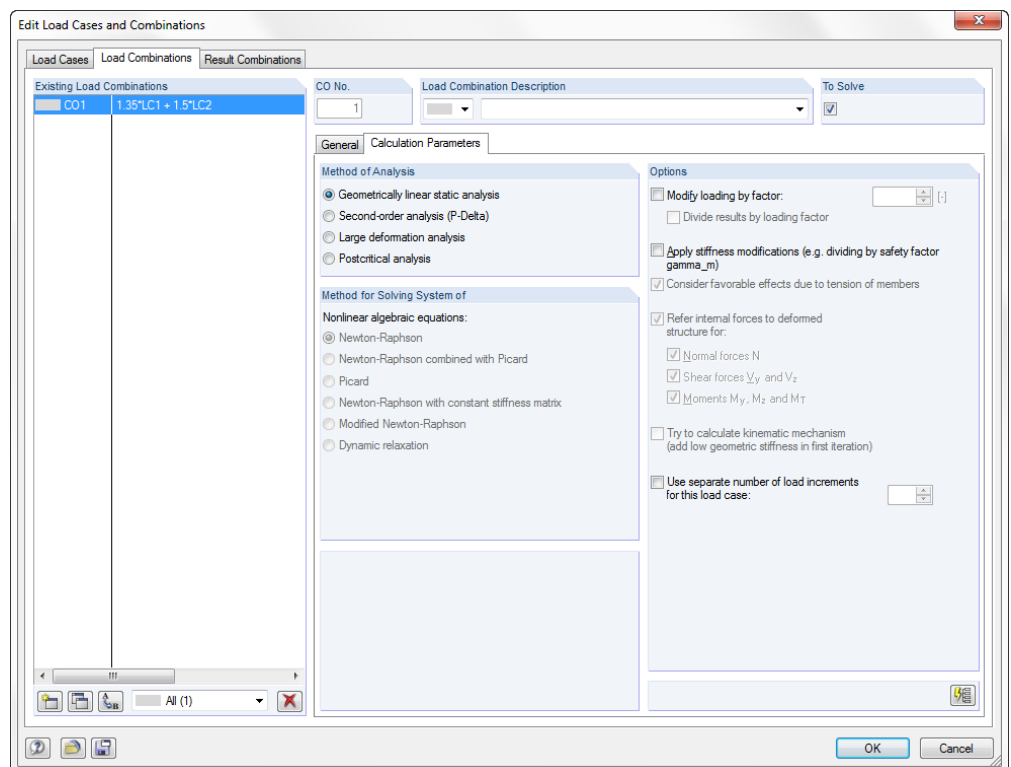
Now we superimpose the two load cases in a load combination. We open the **Edit Load Cases and Combinations** dialog box as shown in the figure to the left.

The **Load Combinations** tab contains the two subtabs **General** and **Calculation Parameters**. In the **General** tab, we select the load cases that are to be combined. The partial safety factors are assigned automatically.



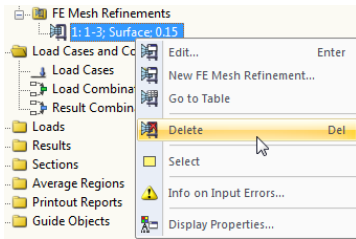
Dialog box *Edit Load Cases and Combinations*

In the **Calculation Parameters** tab, we select the method of analysis, and, if necessary, other calculation settings. In our example, we want to calculate according to the second-order analysis with consideration of an imperfection. This imperfection is to be derived from the deformations of the superimposed load cases. To determine the imperfection, it is necessary to select the **Geometrically linear static analysis** in the **Method of Analysis** section since we want to calculate the internal forces without additional moments due to P-delta. Those linear internal forces can then be used for the analysis in the RF-STABILITY module.



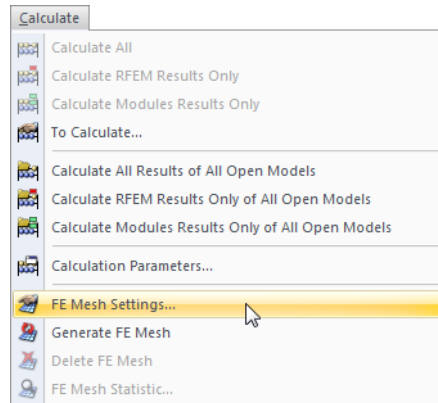
Dialog box *Edit Load Cases and Combinations*

4.4 FE Mesh



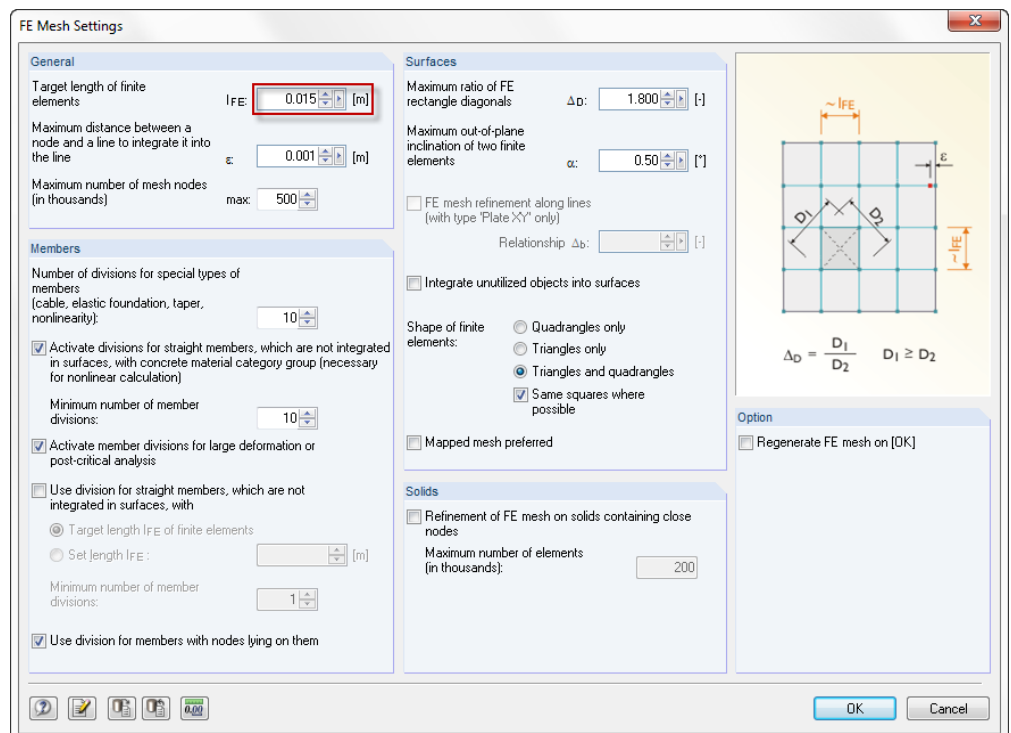
When the surfaces were created from the beam member, mesh refinements were defined for all surfaces. We delete those mesh refinements via the context menu of the **Data** navigator (see figure to the left). Alternatively, we can use the **[Del]** key.

We select **FE Mesh Settings** on the **Calculate** menu in order to modify the general size of the finite elements.



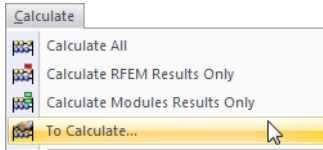
Menu Calculate

In the **FE Mesh Settings** dialog box, we enter the value **0.015 m** for the **Target length of finite elements**.

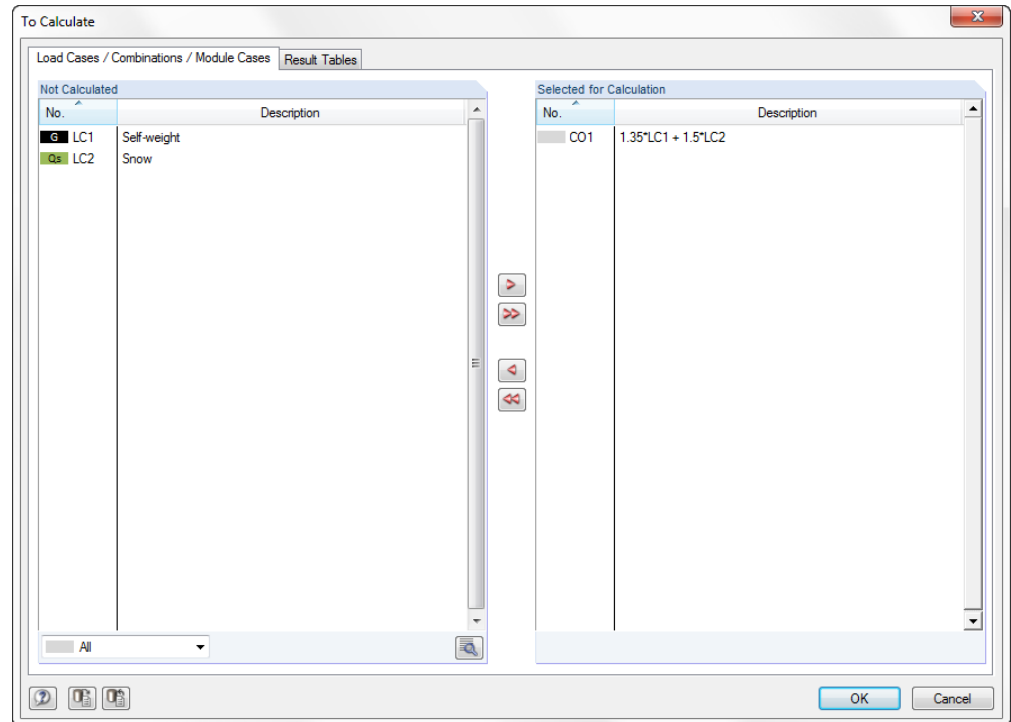


Dialog box FE Mesh Settings

4.5 Calculation



On the same **Calculate** menu, we select **To Calculate**. We set load combination **CO 1** for the calculation. Then we start the calculation by clicking **[OK]**.



Dialog box *To Calculate*

The FE mesh is generated automatically.

After the analysis, the deformations of the girder are displayed in the work window.

4.6 RF-STABILITY

With the add-on module **RF-STABILITY**, we can perform a stability analysis of any structure. To do this, we determine the **critical load factors** η_{ki} and the according **buckling shapes**.

4.6.1 Theory

By means of the critical load factors, the program calculates the **magnification factor** α according to this formula:

$$\alpha = \frac{1}{1 - \frac{1}{\eta_{cr}}}$$

With the **magnification factor** α , we can determine the moment M^{II} according to the second-order analysis from the moment M^{I} according to the linear static analysis under consideration of the equivalent loading for the deformation as follows:

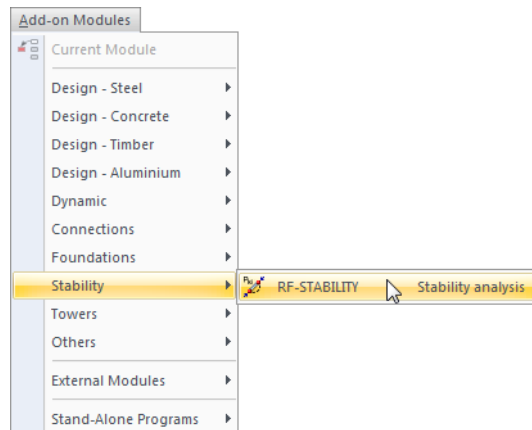
$$M^{\text{II}} = \alpha \cdot M^{\text{I}}$$

The **critical load factor** η_{cr} is a measure for the stability risk of the system.

The buckling shape provides information on the position of the structural component prone to instability risks.

4.6.2 Input Data

To open the add-on module RF-STABILITY, we point to **Stability** on the **Add-on Modules** menu, and then click **RF-STABILITY - Stability analysis**.

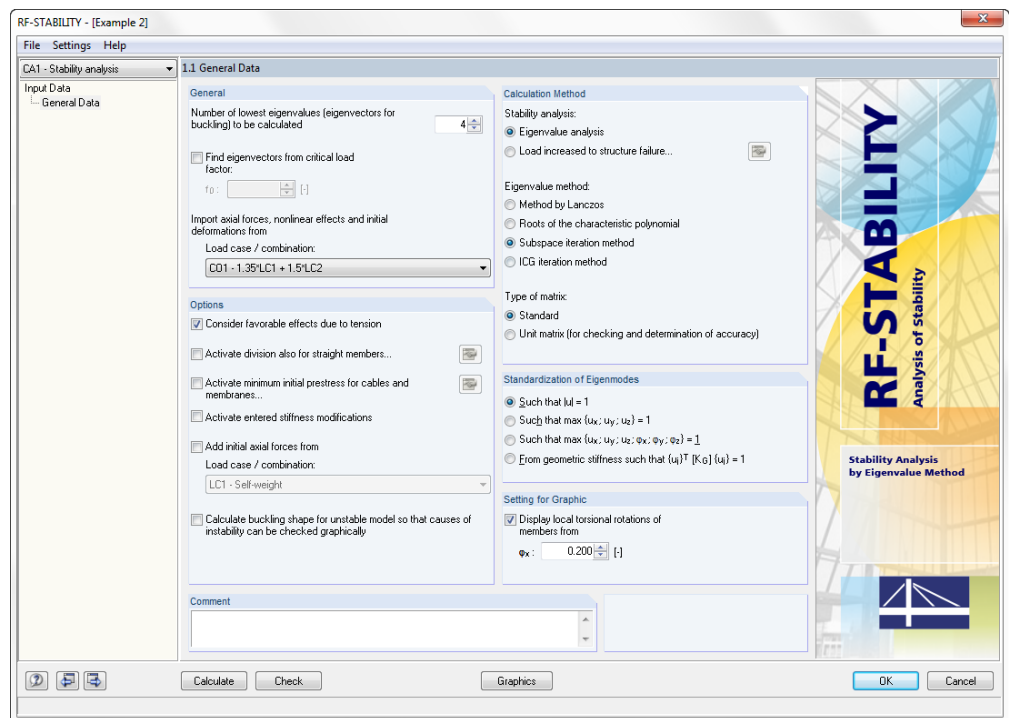


Menu *Add-on Modules*

RF-STABILITY shows only one window: **1.1 General Data** (see figure below). In the **General** section of this window, we can specify the number of lowest eigenvalues (eigenvectors for buckling) that are to be calculated. For our example, we specify **4** eigenvalues.

Under the option **Import axial forces, nonlinear effects and initial deformations**, we have to specify a load case or combination whose axial forces are to be applied for the analysis. We select the load combination **CO1**. In addition to this, we can *Consider the favorable effects due to tension*. However, this effect only refers to member elements, not surface elements.

According to EN 1993-1-1, no stiffness modifications (material etc.) need to be activated.



RF-STABILITY module window 1.1 General Data

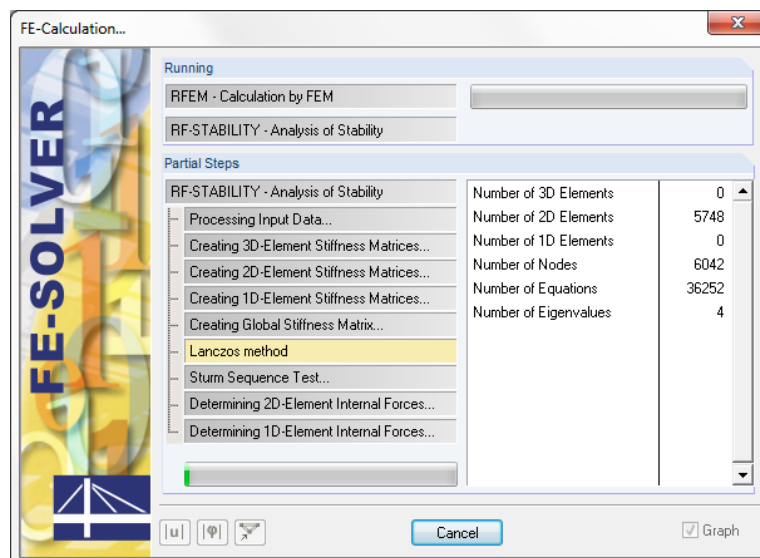
In the **Calculation Method** section, there are four **Eigenvalue Methods** available of which we will mention only two:

- The **Subspace iteration method** is rather fast. However, it needs a lot of RAM because all eigenvalues are calculated in one step. This method is suitable for small and medium sized models for which you want to calculate a large number of eigenvalues.
- The **ICG iteration method** is significantly slower. All eigenvalues are calculated one after the other. The RAM requirements are, therefore, small. However, the computing time increases depending on the number of calculated eigenvalues. This method should be applied in case of extensive models with only few eigenvalues.

We set the **Subspace iteration method**.

Calculate

When we click **[Calculate]**, the solver window appears.

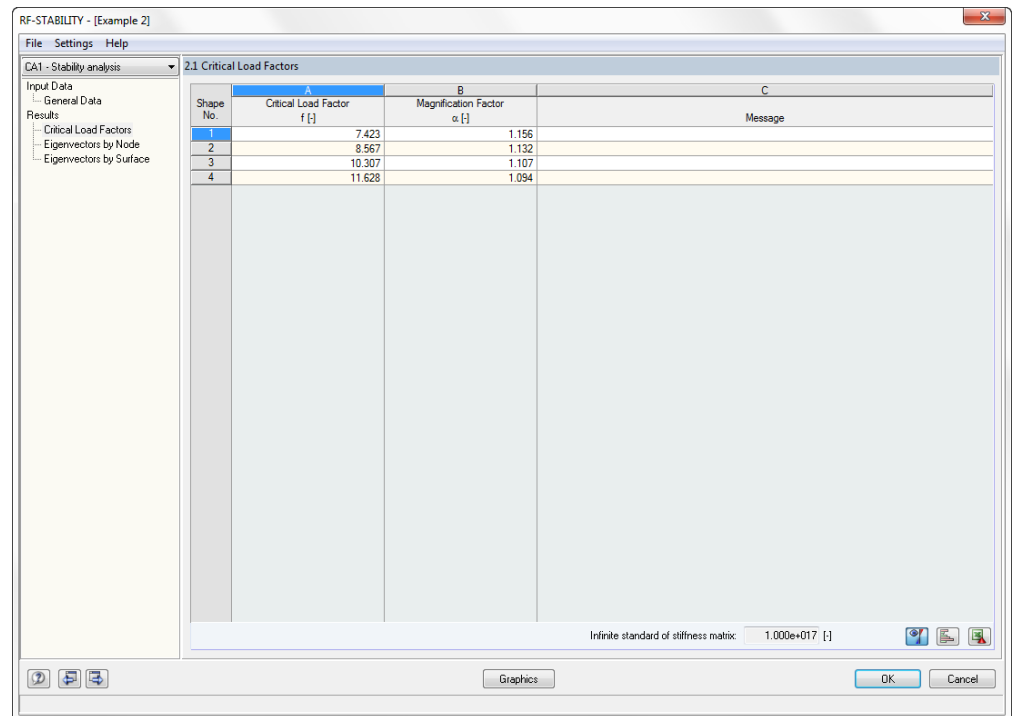


Dialog box FE-Calculation

4.6.3 Results

When the calculation is successfully completed, the module shows several results windows. The number of the windows depends on the types of elements used in the model.

Window **2.1 Critical Load Factor** lists all critical load factors as well as all magnification factors. If a critical load factor is within an interval between 1 and 10, the model is still stable; however, the second-order analysis is required.



Shape No.	A Critical Load Factor $f [-]$	B Magnification Factor $\alpha [-]$	C Message
1	7.423	1.156	
2	8.567	1.132	
3	10.307	1.107	
4	11.628	1.094	

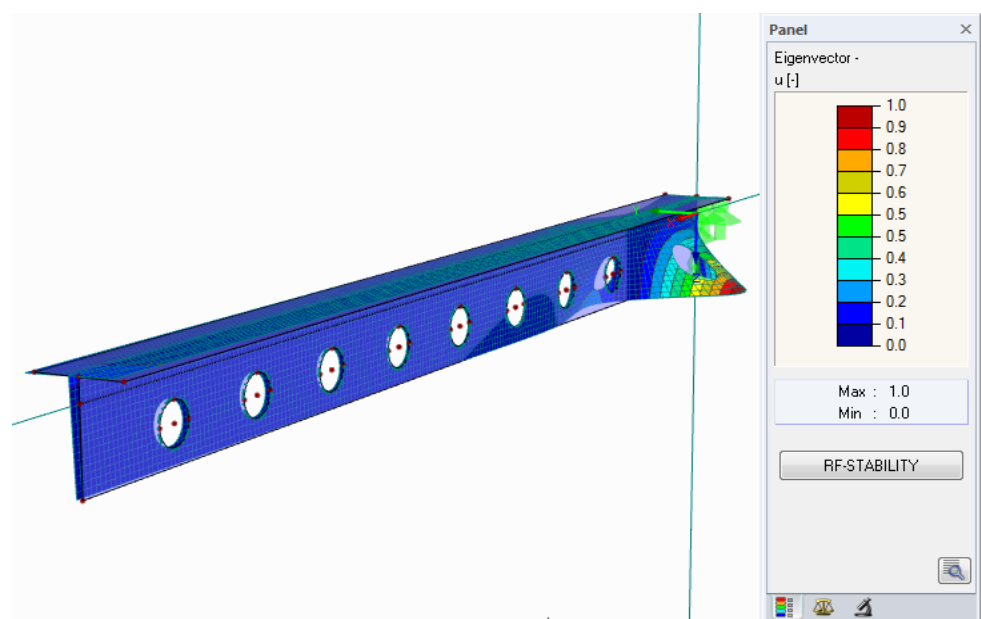
Infinite standard of stiffness matrix: 1.000e+017 [-]

Module window 2.1 Critical Load Factors

The other windows show the normalized eigenvectors (buckling shapes) in tables. If members are available, the buckling lengths are displayed, too.

Graphics

To go to the work window, we click **[Graphics]**.



First buckling shape

4.7 RF-IMP

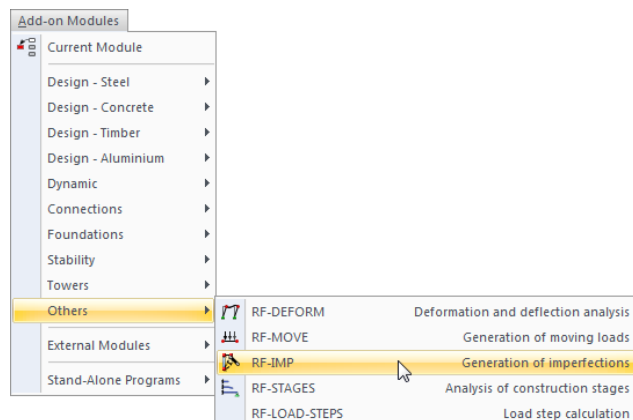
The add-on module RF-IMP allows us to generate imperfections for the calculation according to second-order analysis as required by many standards. There are two main ways to do this:

- Using equivalent loads (see EN 1993-1-1:2010-12, 5.2.2(5)).
- We can shift nodes of the model according to the requirements of the standard. The calculation is then carried out for this equivalent system.

The first method can only be applied to member elements, not to surfaces or solids. The second method can be used for all types of elements. RF-IMP offers both possibilities.

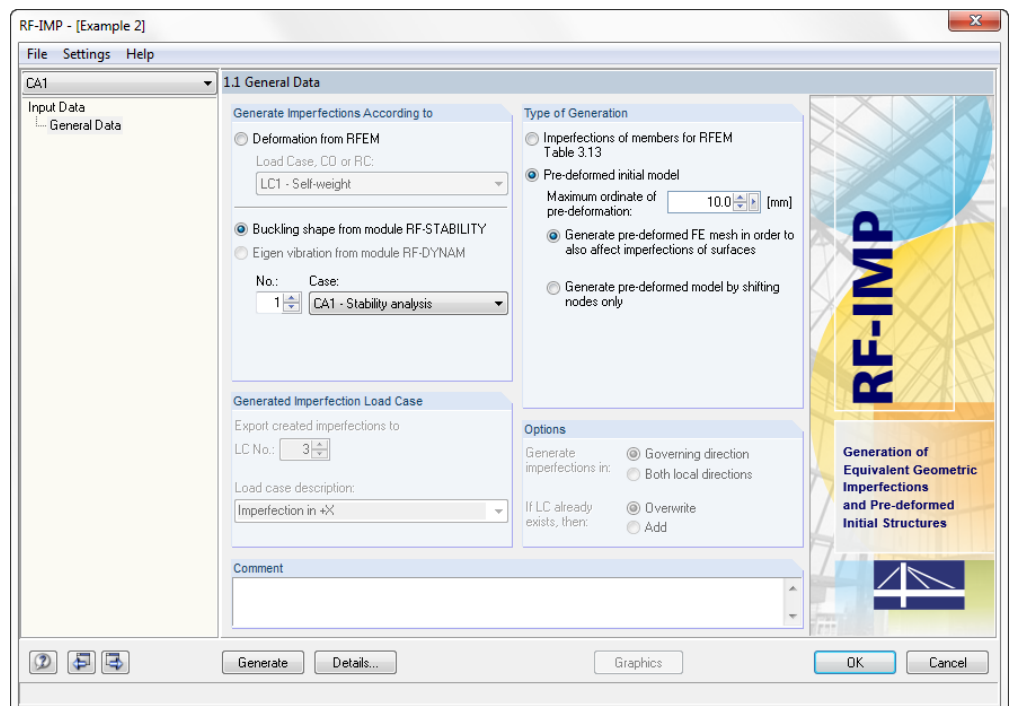
To start RF-IMP, we click

Add-on Modules → Others → RF-IMP.



Menu Add-on Modules

In the RF-IMP module window **1.1 General Data**, we can choose whether the imperfections are to be based on the deformations of a load case or combination, a buckling shape of RF-STABILITY or an eigenvector of RF-DYNAM. We select the buckling shape **1** from the option **Buckling shape from module RF-STABILITY**.

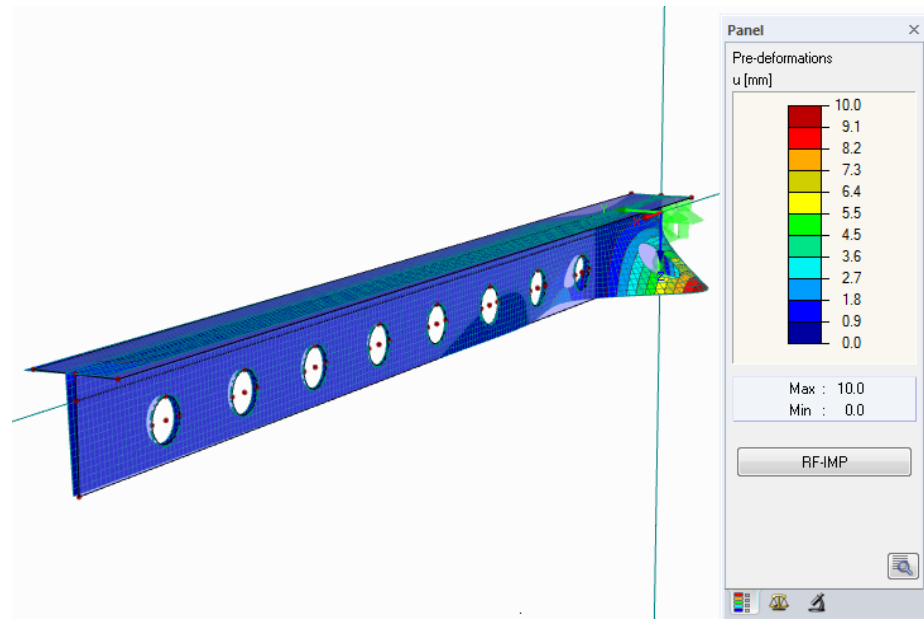


RF-IMP module window 1.1 General Data

Generate

Graphics

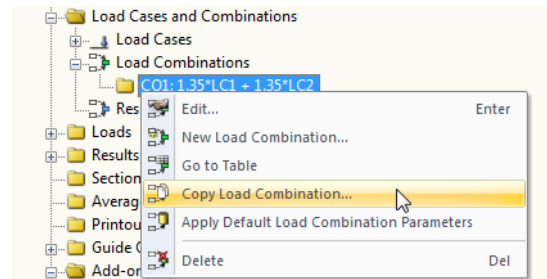
By clicking **[Graphics]**, we can go to the work window to see the pre-deformations.



The imperfections correspond to the first buckling shape of the RF-STABILITY module. The two figures differ only with regard to the units of their scale: The RF-STABILITY buckling shape is normalized to 1, and is, therefore, without unit. In contrast, the imperfections in RF-IMP have a maximum deformation amplitude of 10 mm.

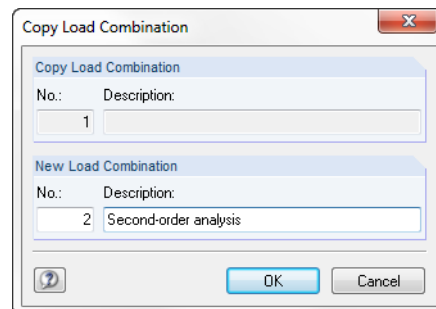
4.8 Second-Order Analysis

The imperfection shape is now available. We can use it for a new analysis according to second-order theory. The easiest way to do that is to copy load combination CO1 by right-clicking it in the *Data* navigator.



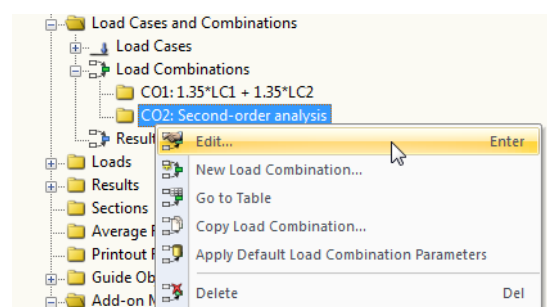
Load Combinations context menu

The **Copy Load Combination** dialog box opens. We can enter the **No.** of the new load combination and its **Description**.



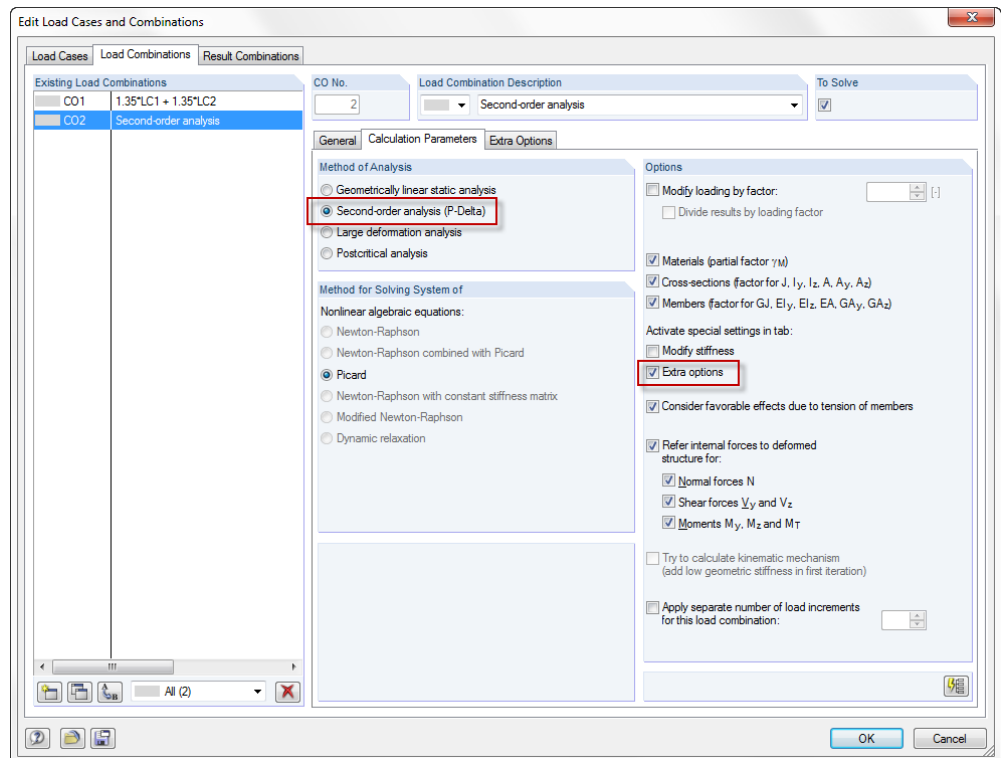
Dialog box Copy Load Combination

After **[OK]**, the navigator shows the new load combination CO2. We must change the method of analysis for this load combination, however. Again, we use the context menu of the navigator to adjust the settings of load combination CO2. On the context menu, we select **Edit**.



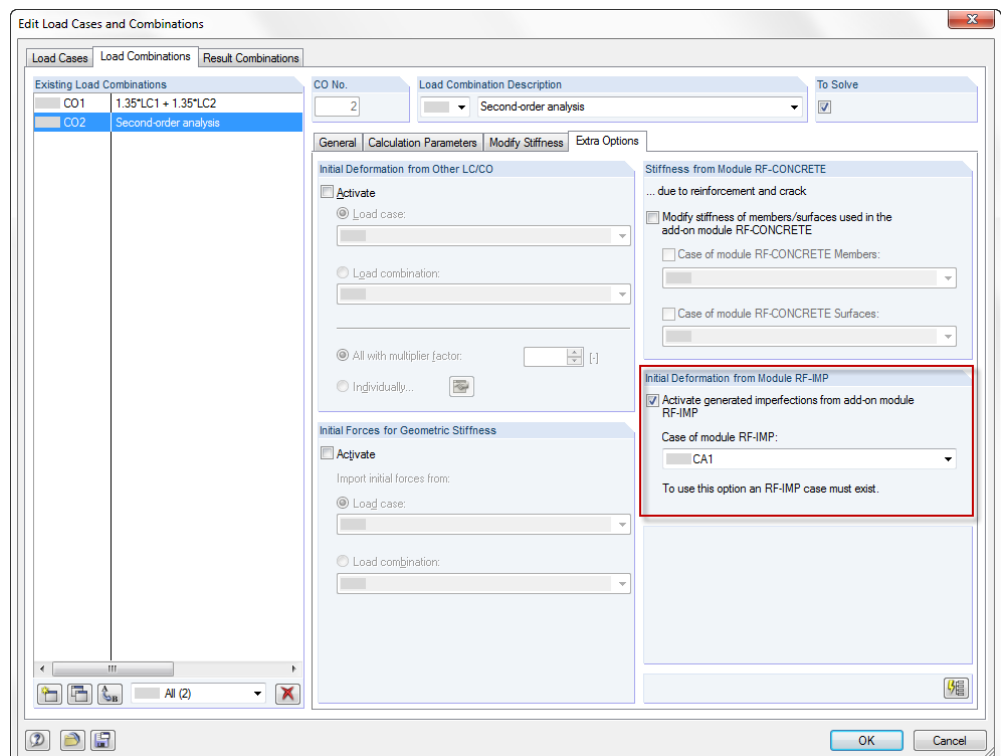
Load Combinations context menu

We select load combination **CO2** and its **Calculation Parameters** subtab. In the **Method of Analysis** section, we select **Second-order analysis**. Additionally, we activate the **Extra Options**.



Dialog box *Edit Load Cases and Combinations*, subtab *Calculation Parameters*

Now we go to the **Extra Options** tab. In the section **Initial Deformation from Module RF-IMP**, we select the imperfection that we have generated in RF-IMP. A second-order analysis using an imperfection that is based on the first buckling shape results in unfavorable internal forces.

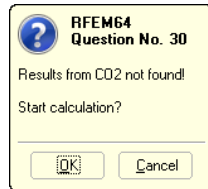


Dialog box *Edit Load Cases and Combinations*, subtab *Extra Options*



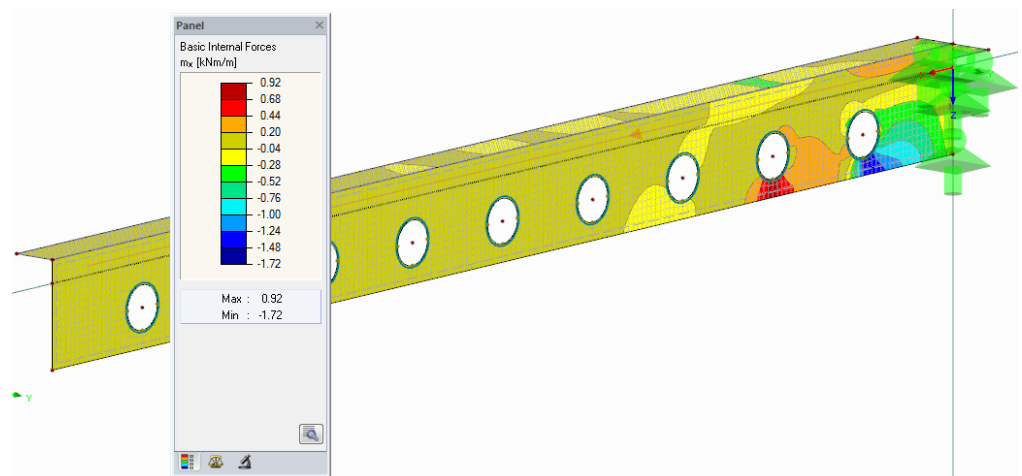
We confirm the input by clicking **[OK]**. Now we can start the calculation of load combination CO2 by clicking **[Show Results]**.

We confirm the following message with **[OK]**.



RFEM question

After the calculation, we can display the moments m_x , for example.



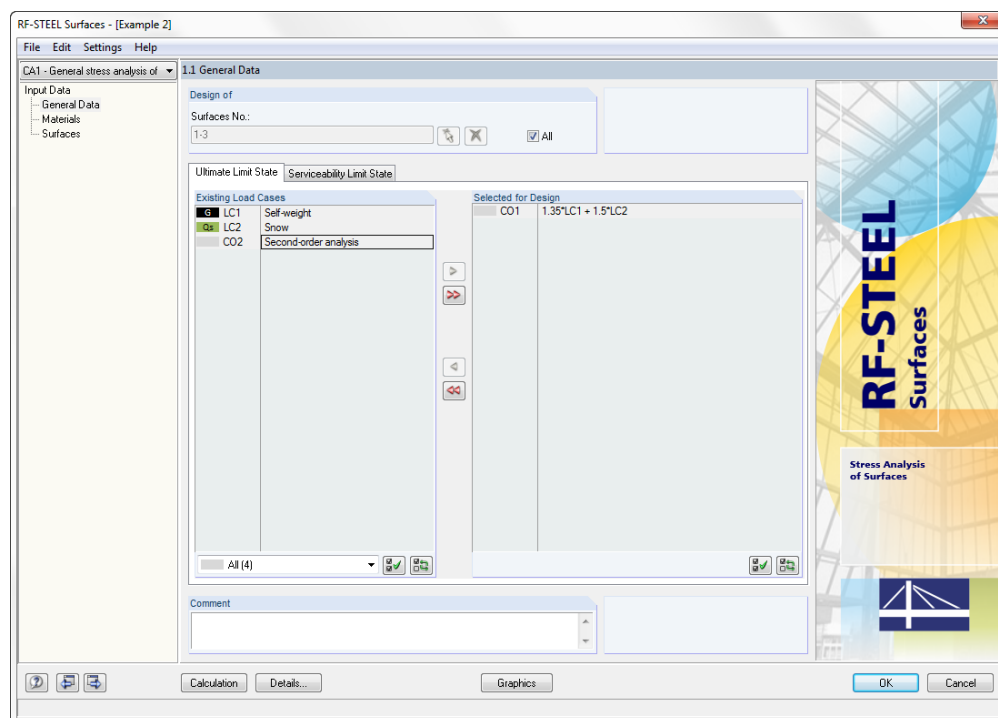
Moments m_x of CO2 with consideration of imperfections

4.9 RF-STEEL Surfaces

Finally we perform a stress analysis with the **RF-STEEL Surfaces** add-on module. We open this module by clicking it in the **Data** navigator.

4.9.1 Input Data

In the module window **1.1 General Data**, we select **CO1** for the design. To compare the stresses due to the linear static and the second-order analyses, we will design CO2 afterwards.



RF-STEEL Surfaces module window 1.1 General Data

Window **1.2 Materials** lists the materials defined in the model. Materials that are non-designable in RF-STEEL Surfaces like concrete are highlighted in red.

The third window lists all surfaces selected for design and shows their thicknesses.

1.3 Surfaces

Surface No.	A Material No.	B Thickness Type	C d [mm]	D Optimize	E Remark	F Area A [m ²]	G Weight G [t]	H Comment
1	2	Constant	16.0	☐		0.30	0.04	
2	2	Constant	16.0	☐		0.30	0.04	
3	2	Constant	10.2	☐		0.66	0.05	

Window 1.3 Surfaces

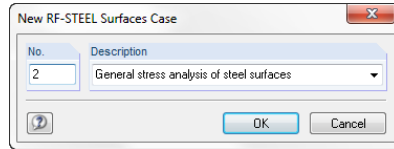
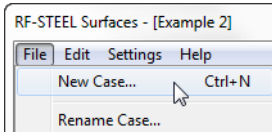
We start the **[Calculation]** of the stresses.

Calculation

4.9.2 Results

After the calculation, we create a new design case for CO2 by selecting menu

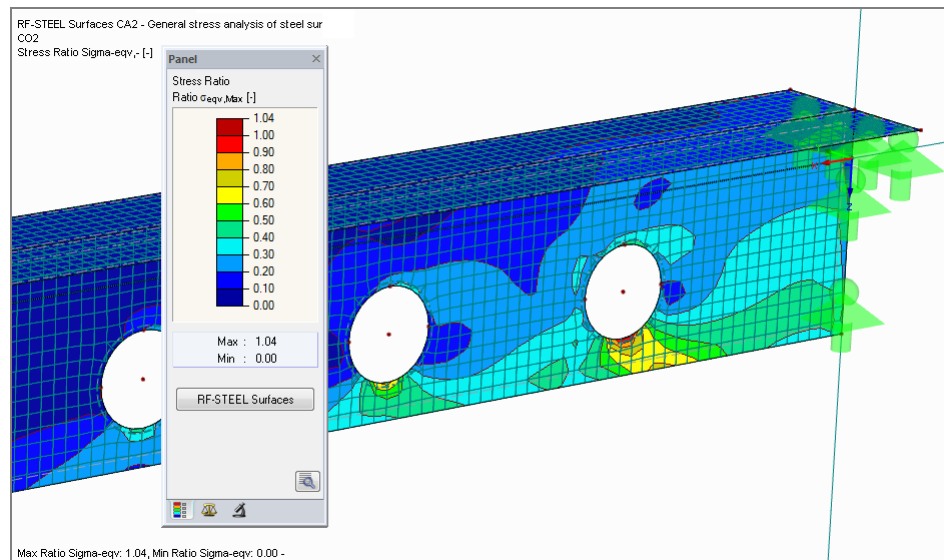
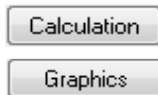
File → New Case.



Dialog box *New RF-STEEL Surfaces Case*

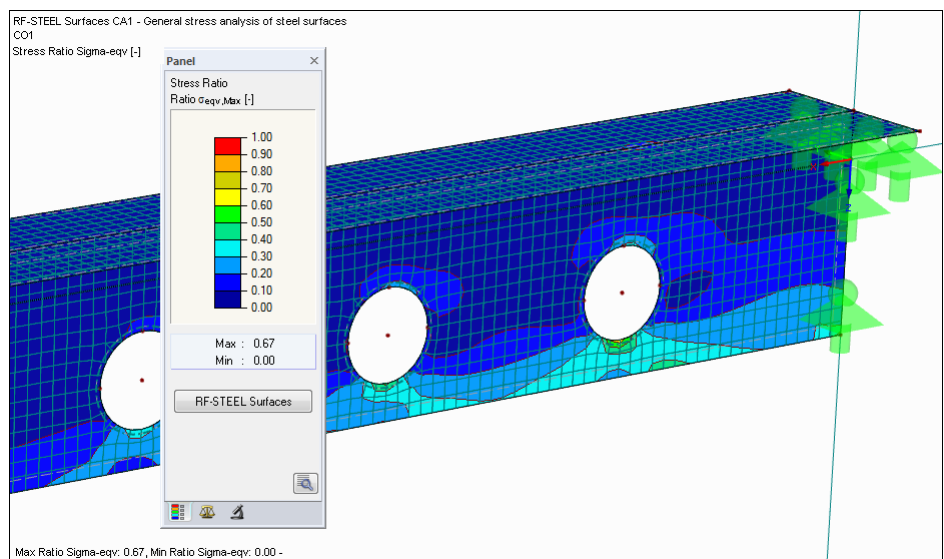
In window **1.1 General Data** of the new case, we select **CO2**. Then we start the **[Calculation]**.

To view the results of the stress design graphically, we click **[Graphics]**. The following figure shows the ratio of σ_{eqv} of **CO2** according to second-order analysis. We can see that the limit stress is exceeded at the location with the maximum plate-buckling effect.



Stresses due to nonlinear calculation with consideration of imperfections

By contrast, the figure below shows the ratios σ_{eqv} from **CO1** (linear analysis). There is an increase in stress below the first opening, too. The stresses are far below the limit, however.



Stresses due to linear static analysis

5. Parameterized Input

If the model or the loading depend on variables, the input can be parameterized. It is particularly suited for similar models whose values of the parameters vary.

The parametric input is not suited for branching models such as multi-span beams with a variable number of spans. It is, however, possible to model a parameterized multi-span beam with variable span lengths.

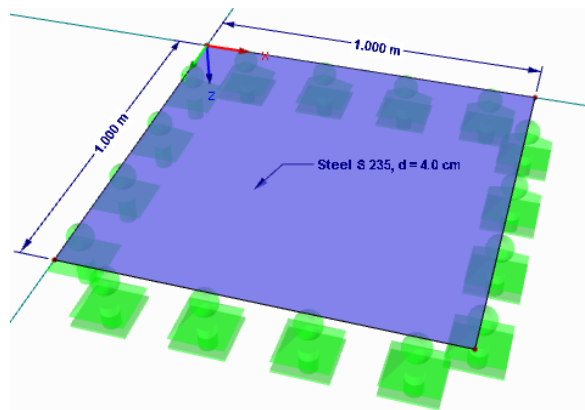
In this example, we will create a parameter list and use those parameters to define the model and the loading.

5.1 Model of Steel Sheets

We model two-span sheets for which we define the span lengths, the width and the surface thickness as variable parameters.

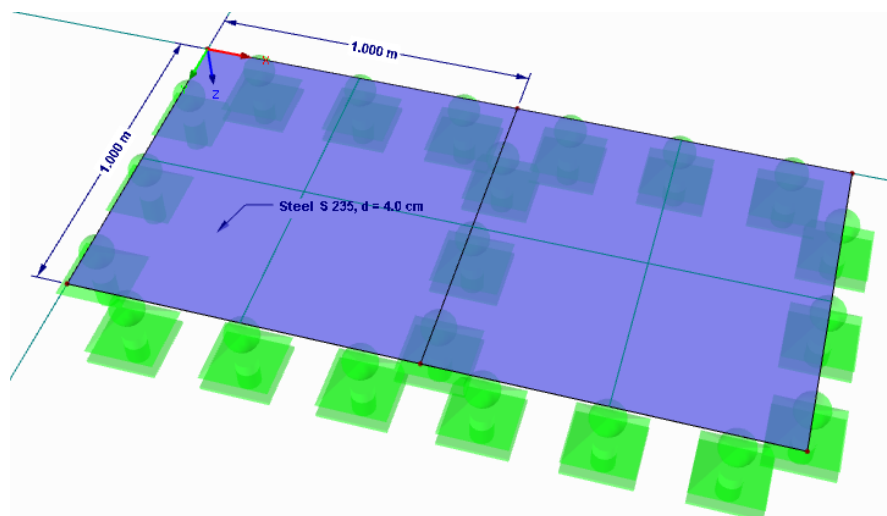
First we create a surface with the dimensions **1x1 m** that has a thickness of **4.0 cm** and is made of **Steel S 235** as shown in the figure below.

The surface is supported by hinged line supports at its four boundary lines. One boundary line is to be supported as horizontally braced, the remaining three lines as horizontally unbraced.



Surface 1x1 m

We use the drag-and-drop function while pressing the **[Ctrl]** key to create the second surface directly next to the first one and to support it as horizontally unbraced.



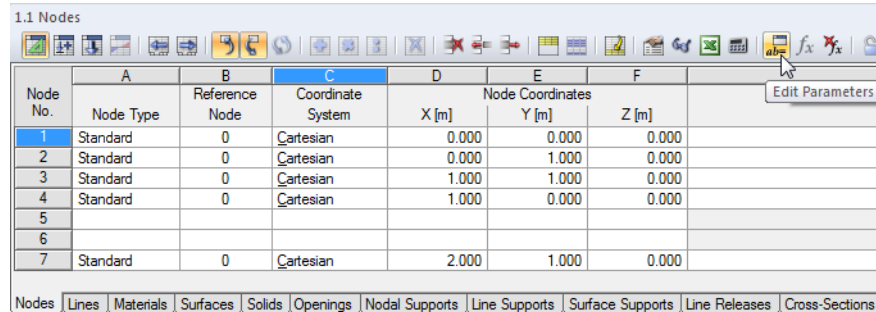
Model and copy

5.1.1 List of Parameters



Now we create a list of the required parameters. We open the **Edit Parameters** dialog box by clicking the **[Edit Parameters]** button in the table.

1.1 Nodes



Node No.	A Node Type	B Reference Node	C Coordinate System	D X [m]	E Node Coordinates Y [m]	F Z [m]	
1	Standard	0	Cartesian	0.000	0.000	0.000	
2	Standard	0	Cartesian	0.000	1.000	0.000	
3	Standard	0	Cartesian	1.000	1.000	0.000	
4	Standard	0	Cartesian	1.000	0.000	0.000	
5							
6							
7	Standard	0	Cartesian	2.000	1.000	0.000	

Buttons: Nodes | Lines | Materials | Surfaces | Solids | Openings | Nodal Supports | Line Supports | Surface Supports | Line Releases | Cross-Sections

Table button [Edit Parameters]

In the dialog box **Edit Parameters**, we define the parameters **Length1**, **Length2**, **Width** and **Thickness** as seen in the following figure. Then we click **[OK]** to close the list of parameters.

Edit Parameters

List of Parameters

	A Parameter	B Unit Type	C Value	D Unit	E Formula	F Range of Values	G Comment
1	Length1	Length	1.000	m			
2	Length2	Length	1.000	m			
3	Width	Length	1.000	m			
4	Thickness	Length	0.030	m			
5							

Comment

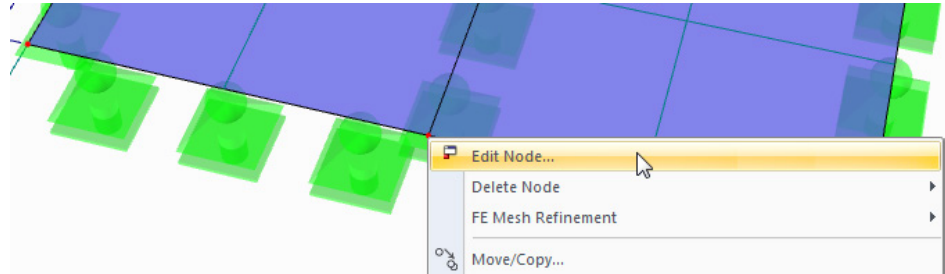
Name of parameter (e.g. "Width_w", not possible: "e" and "l")

Buttons: OK Cancel

Dialog box **Edit Parameters**

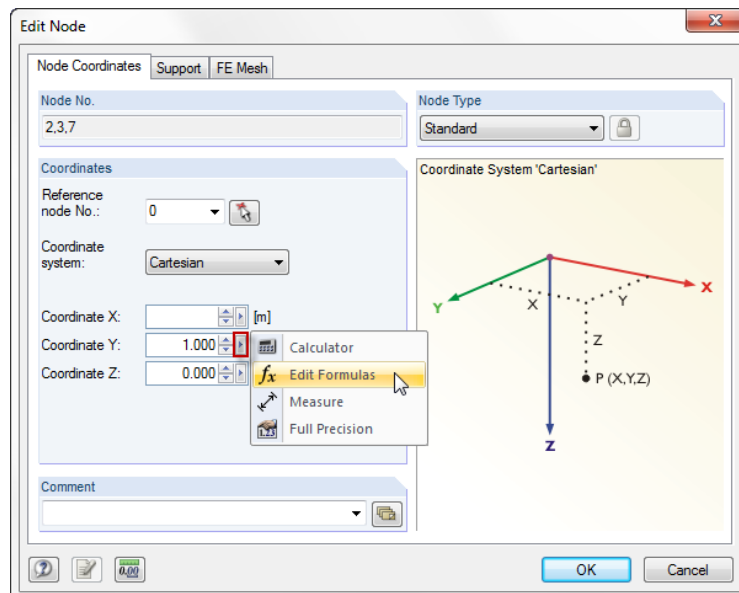
5.1.2 Formula Editor

Now we can parameterize the width of the two surfaces: We select the three nodes that do not lie on the X-axis. Then we right-click on one of those nodes to open the context menu.



Node context menu – selected nodes for *Width*

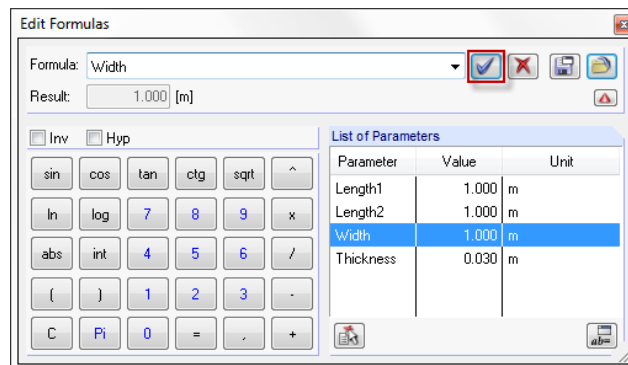
We click in the selected field next to the Y-coordinate, and then select the **Edit Formulas** option.



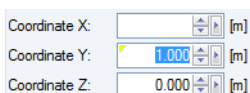
Dialog box *Edit Node*



In the formula editor (dialog box *Edit Formulas*), we double-click the **Width** parameter. This parameter is transferred to the **Formula** field. We close the editor by clicking the **[Apply]** button.

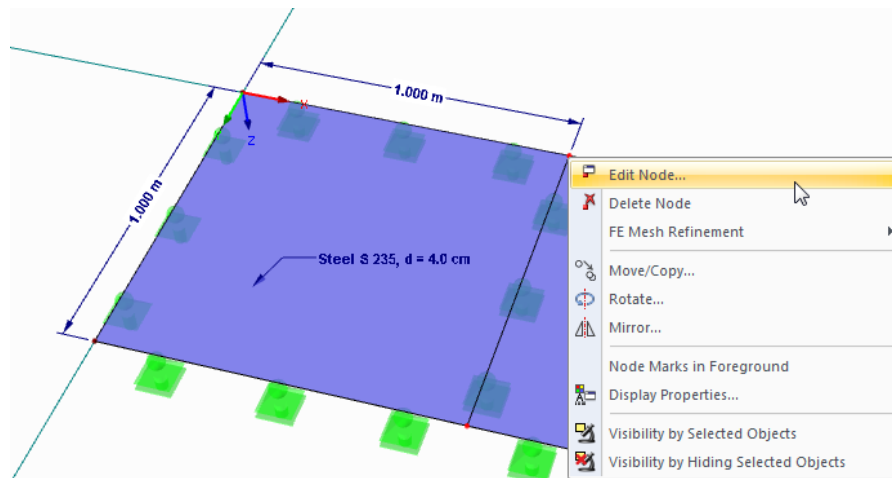


Dialog box *Edit Formulas* (formula editor)



In the **Edit Node** dialog box, we now see a yellow flag in the field of the parameterized coordinates. This flag is shown in all fields containing a parameterized quantity. We can return to the formula editor by clicking this flag.

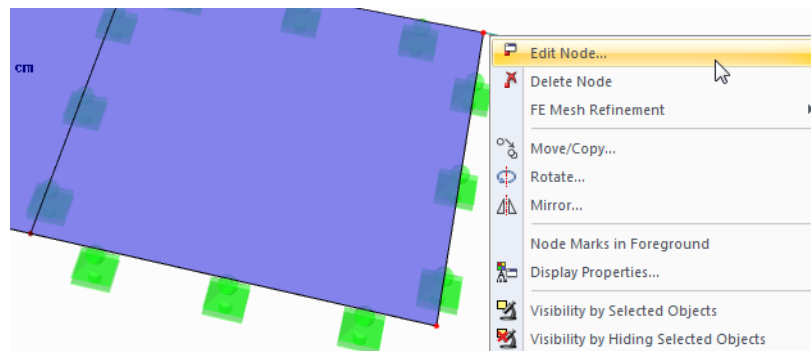
Next we parameterize the span width of the first surface. We select the two nodes in the middle of the plate as shown in the figure. Then we edit them via the context menu again.



Node context menu – selected nodes for *Length1*

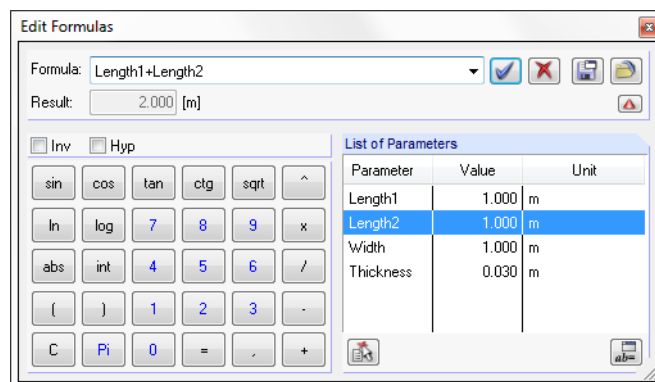
We open the formula editor by clicking the function button next to the **X**-coordinate now. The **Edit Formulas** dialog box opens again where we select **Length1**. We **[Apply]** this formula, too.

The two nodes on the right side are the last pair of nodes that are to be parameterized.



Node context menu – selected nodes for *Length1+Length2*

We open the formula editor next to the **X**-coordinate of the two nodes. In the **Formula** field, we first enter **Length1**. We click **[Plus]** and then enter **Length2**.



Dialog box *Edit Formulas* (formula editor)

After that, we click **[OK]**.

Finally, we parameterize the surface thickness.



To open the formula editor, we use an alternative way: We select table **1.4 Surfaces** and then click **Thickness d** (column F) of surface No. **1**. To start the formula editor, we click the button **[Edit Formula]**.

1.4 Surfaces



Surface No.	Surface Type		Boundary Lines No.	Material No.	Thickness		Nodes No.
	Geometry	Stiffness			Type	d [cm]	
1	 Plane	 Standard	1-4	 2	Constant	4.0	0.0
2	 Plane	 Standard	3,6-8	 2	Constant	4.0	0.0
3							
4							
5							
6							
7							

Nodes

Lines

Materials

Surfaces

Solids

Openings

Nodal Supports

Line Supports

Surface Supports

Line Releases

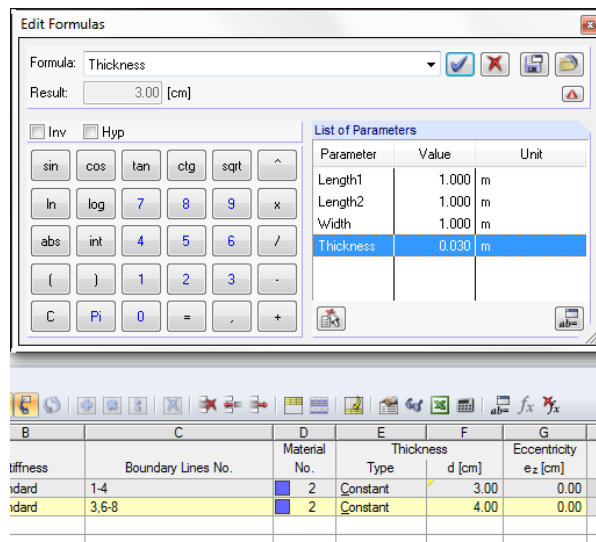
Cross-Sections

Member En...

Table 1.4 Surfaces



The **Edit Formulas** dialog box opens. We transfer the parameter **Thickness** to the **Formula** field and then click **[Apply Formula]**. This time, the formula editor is not closed because it was started from the table. The advantage of this is that in the table, you can go directly to the row of surface No. **2** below and also parameterize its thickness by the **Thickness** parameter.



Formula: Thickness

Result: 3.00 [cm]

☐ Inv ☐ Hyp

sin cos tan ctg sqrt ^

ln log 7 8 9 x

abs int 4 5 6 /

{ } 1 2 3 .

C Pi 0 = +

Parameter	Value	Unit
Length1	1.000	m
Length2	1.000	m
Width	1.000	m
Thickness	0.030	m

B	C	D	E	F	G
Stiffness	Boundary Lines No.	Material No.	Thickness Type	Thickness d [cm]	Eccentricity e _z [cm]
Standard	1-4	2	Constant	3.00	0.00
Standard	3,6-8	2	Constant	4.00	0.00

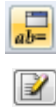
Dialog box *Edit Formulas*



Having done this, we can close the formula editor by clicking **[X]**.

5.1.3 Parameterized Model

Now we open the **Edit Parameters** dialog box by using the **[Edit Parameters]** button. When we modify the parameters in the list and **[Apply]** the changes, we can see the effects in the work window.



Dialog box **Edit Parameters**

	A	B	C	D	E	F	G
	Parameter	Unit Type	Value	Unit	Formula	Range of Values	Comment
1	Length1	Length	1.500	m			
2	Length2	Length	2.100	m			
3	Width	Length	1.500	m			
4	Thickness	Length	0.050	m			
5							

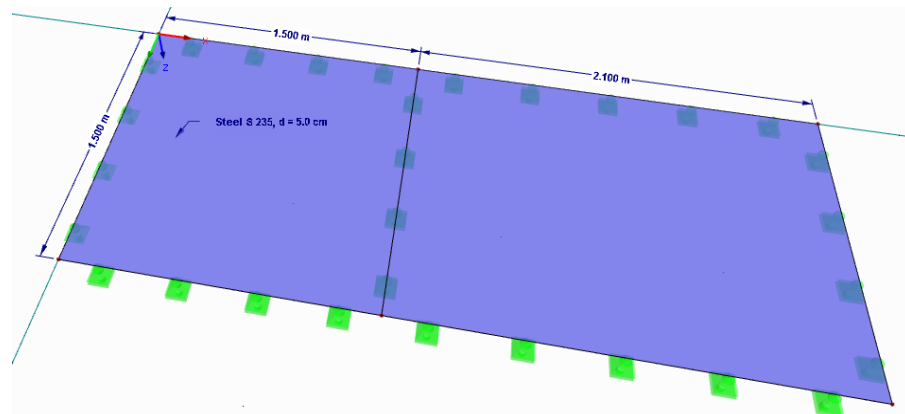
Comment

Name of parameter (e.g. 'Width_w', not possible: 'e' and 'l')

OK Cancel

Dialog box *Edit Parameters*

With the set of parameters as shown above, the model looks as follows.



Parameterized model

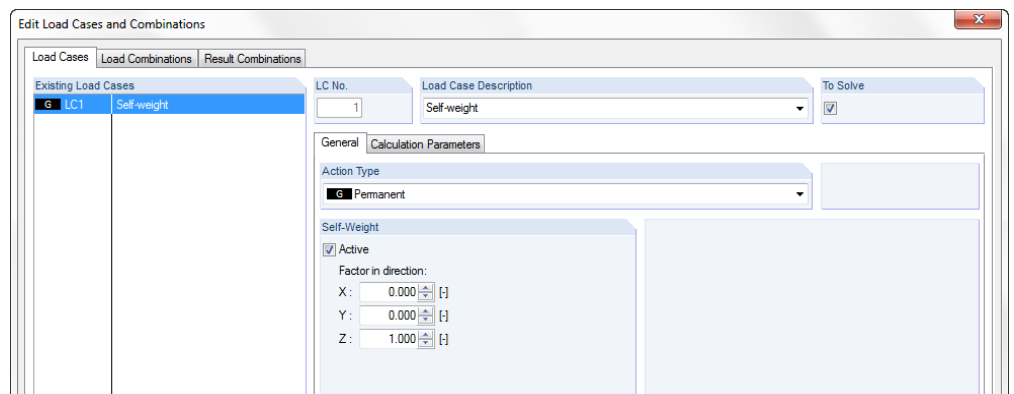
5.2 Parameterized Loads

We can parameterize load data just like model data. As described in the previous chapter, we add the following parameters to the list:

- **g**, unit type **force per area**, value **1 kN/m²**
- **p**, unit type **fore per area**, value **1 kN/m²**

5.2.1 LC1: Self-Weight

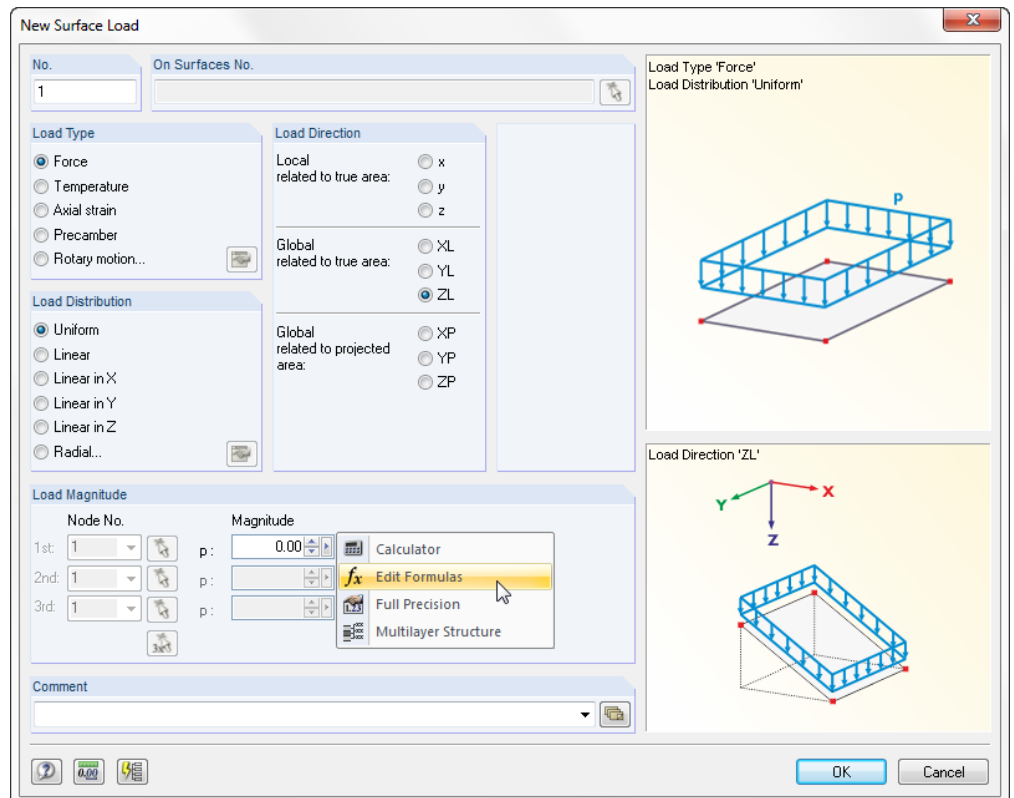
Then we create a load case **LC1 Self-weight** by clicking **[New Load Case]**. A dialog box opens where we define the self-weight as shown in the following figure. Then we click **[OK]**.



Dialog box *Edit Load Cases and Combinations*



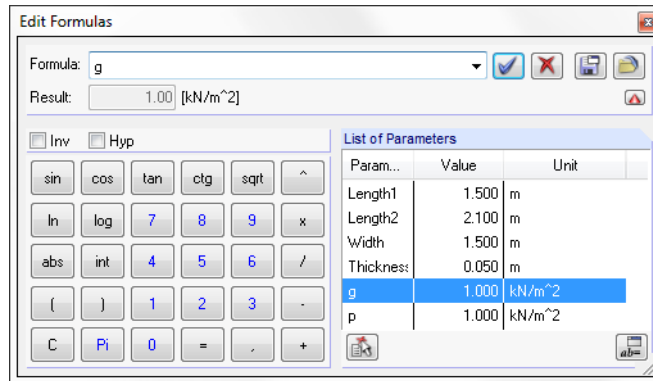
Next we click **[New Surface Load]** to open the **New Surface Load** dialog box. We specify the settings as shown in the figure below. In the **Loading Magnitude** dialog box section, we click the arrow showing to the right, thus opening a menu. Here we select the **Edit Formulas** option.



Dialog box *New Surface Load*

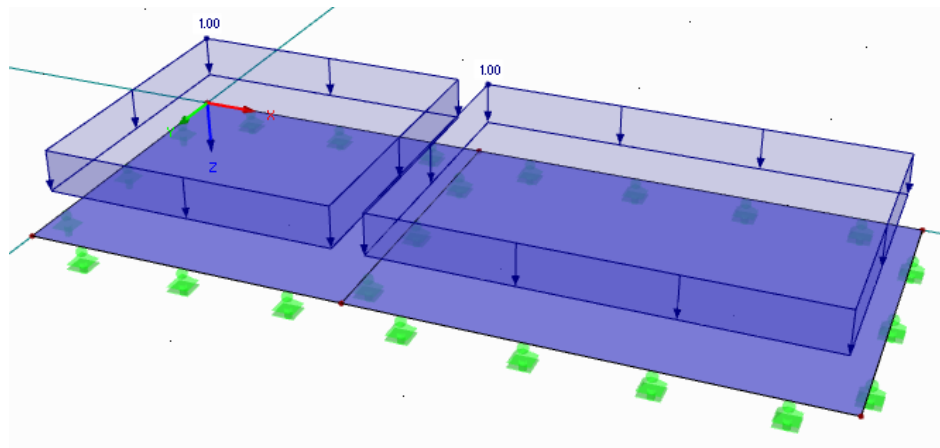


The formula editor opens where we enter the parameter **g**. Then we click **[Apply Formula]**.



Dialog box *Edit Formulas*

We confirm the **New Surface Load** dialog box. Then we select the two surfaces to which we apply the surface load as follows.



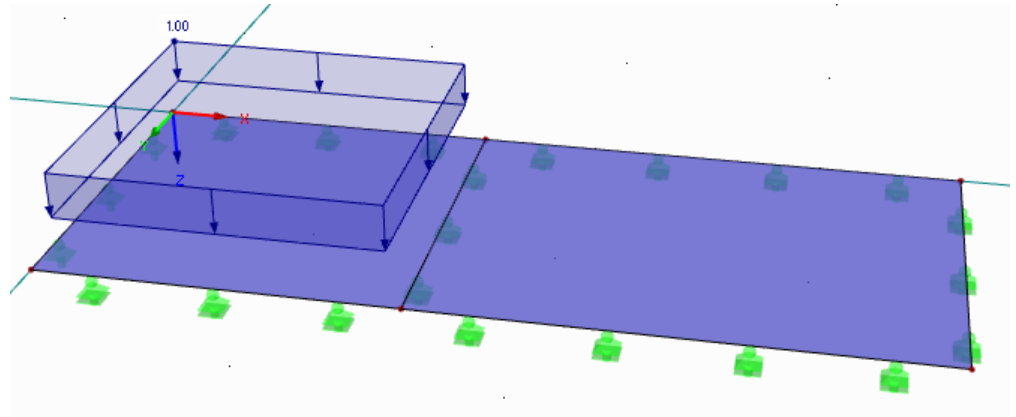
LC1: Surface loads

5.2.2 LC2/LC3: Variable Loads

In the same way, we define two more load cases: **LC2 - Imposed load 1** and **LC3 - Imposed load 2**.

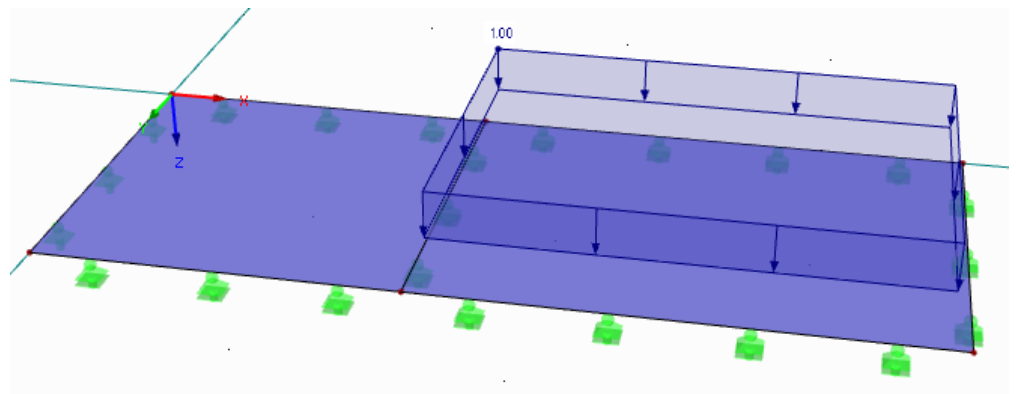
In both load cases, we select the parameter p instead of g .

We apply **LC2** to surface 1.



LC2: Imposed load 1

LC3 contains the surface load of surface 2.



LC3: imposed load 2

After that, we can modify the parameters g and p to change the surface loads – in the same way as it is described in chapter 5.1.3 *Parameterized Model* for the model data.

6. Solids

In this example, we will explore special features related to solid elements.

Since the calculation of solids is rather time-consuming, solid elements should be applied only where necessary. In general, at least three 3D elements are to be created between the closer sides of a solid so that we get plausible results. Solids are also required when we want analyze the stresses perpendicular to the plane of a surface.

The 3D elements of RFEM can have the shapes of a cuboid or a tetrahedron. Each element node has three degrees of freedom – three translational and three rotational degrees of freedom. This makes sure that they are compatible with the surface and member elements.

Solid elements are defined by boundary surfaces of the type **Null**.

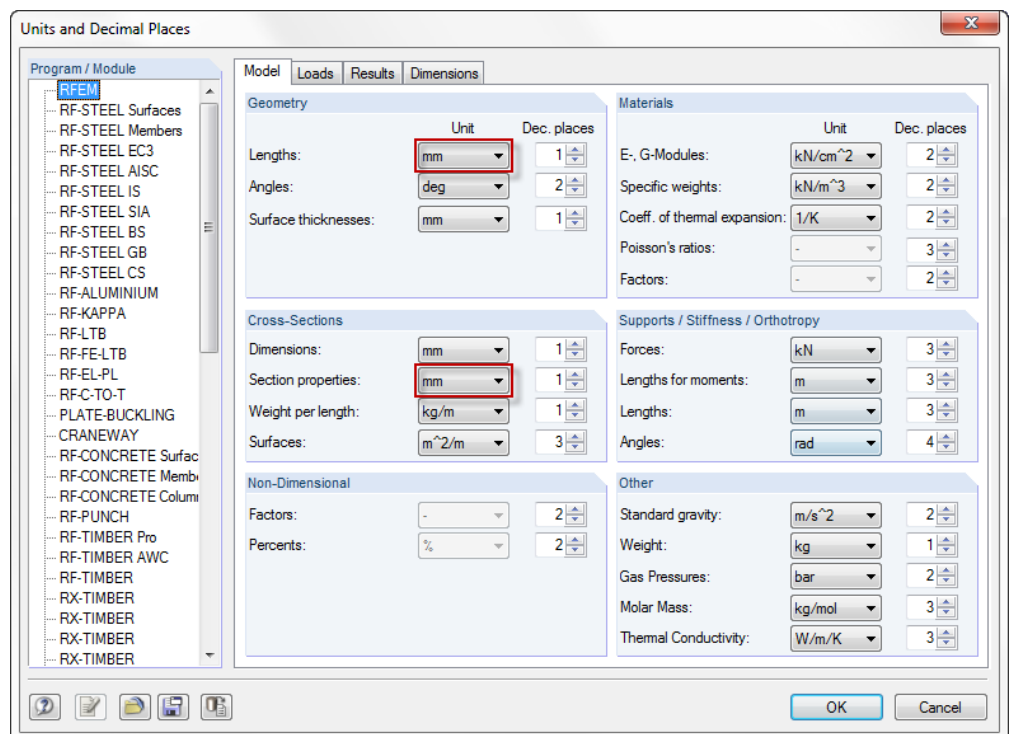
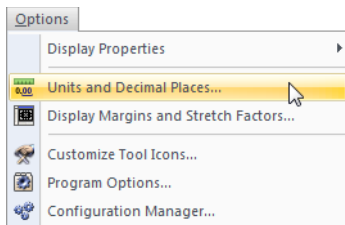
6.1 Model of Steel Hook

6.1.1 Units and Grid Settings

First, we adjust the units and the grid according to the dimensions of the model. To open the dialog box for the adjustments of units and decimal places, we click menu

Options → **Units and Decimal Places**.

Here we change the units of the lengths and the cross-section properties to **[mm]**.

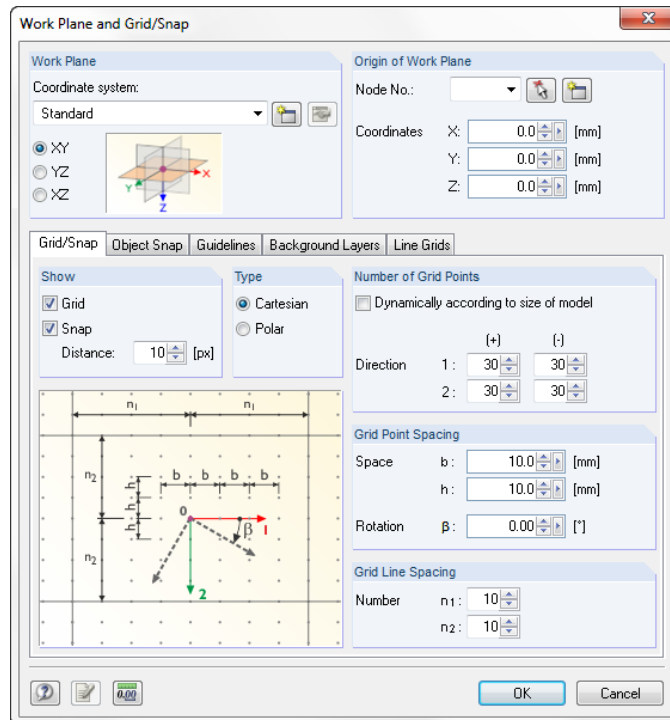


Dialog box *Units and Decimal Places*



We now open the dialog box **Work Plane and Grid/Snap** by clicking the button [Settings of Work Plane, Grid/Snap, Object Snap, Guide Lines].

In the dialog box, we change the space of the grid points **b** and **h** to **10 mm**. Then we click [OK].

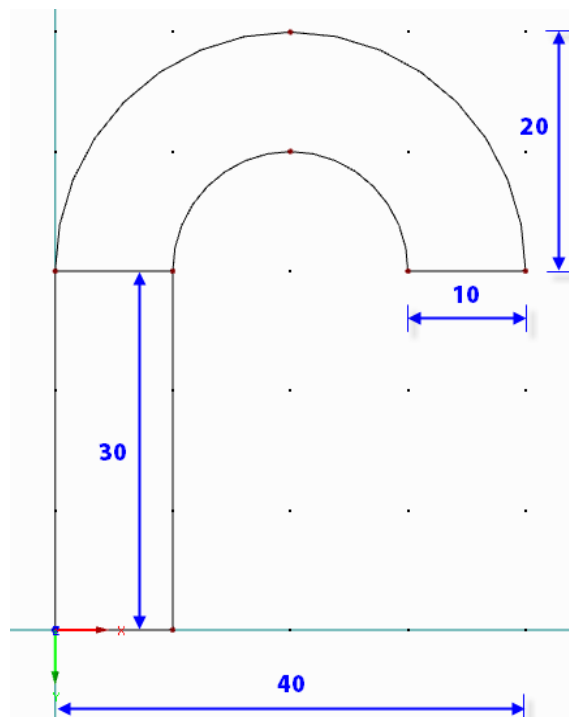


Dialog box *Work Plane and Grid/Snap*

6.1.2 Lines



We draw the lines as shown on the following figure. For the arc lines, we use the [New Arc via Three Nodes] button.

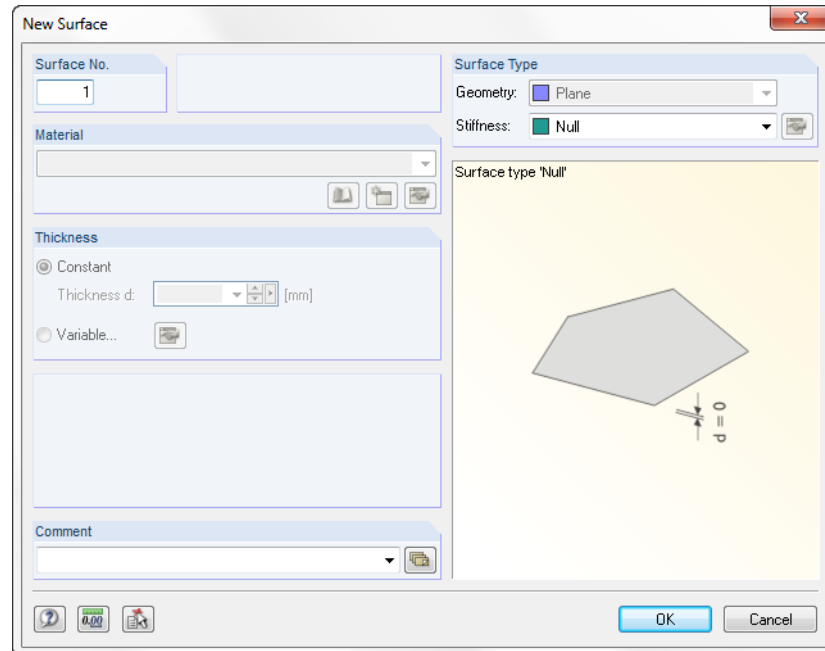


Lines and distances

6.1.3 Surfaces

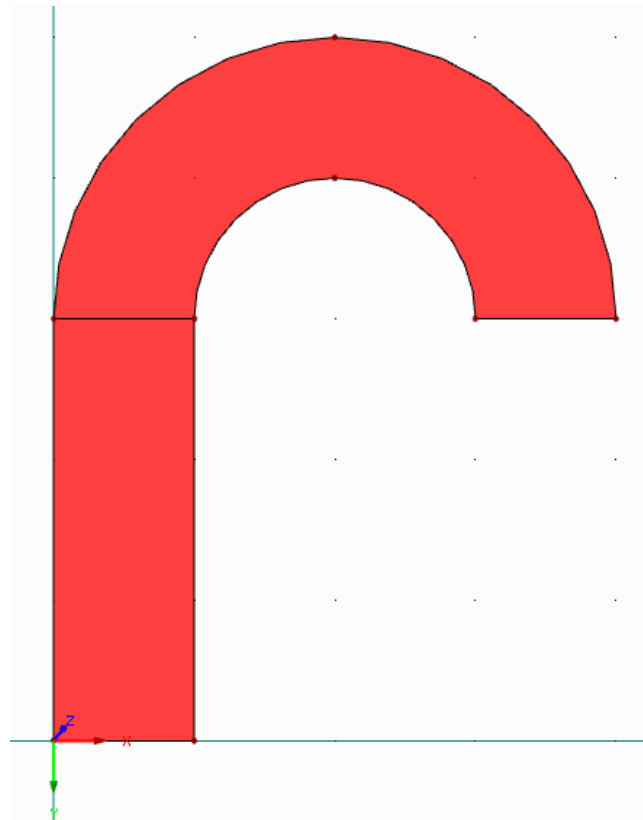


Next, we insert two **Null** surfaces within the boundary lines by using the **[Select Boundary Lines]** button.



Dialog box *New Surface*

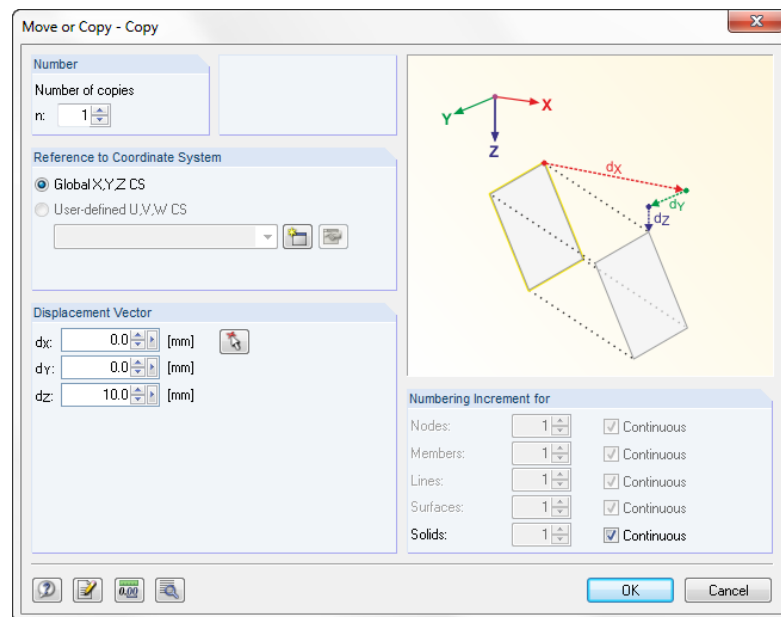
The result looks as follows:



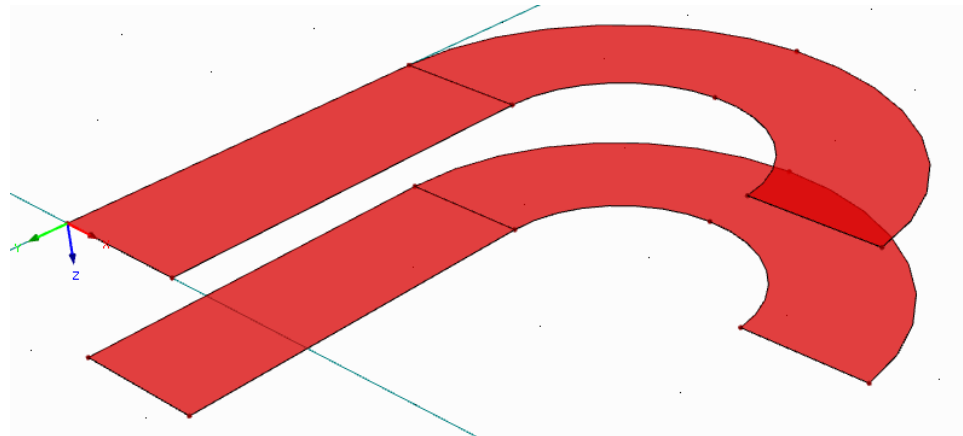
Two surfaces of type *Null*



We select the two surfaces with their lines and nodes. Then we click the **[Move and/or Copy]** button. In the dialog box, we set the number of copies to **1** and enter the value **$d_z = 10$ mm** in the **Displacement Vector** section.

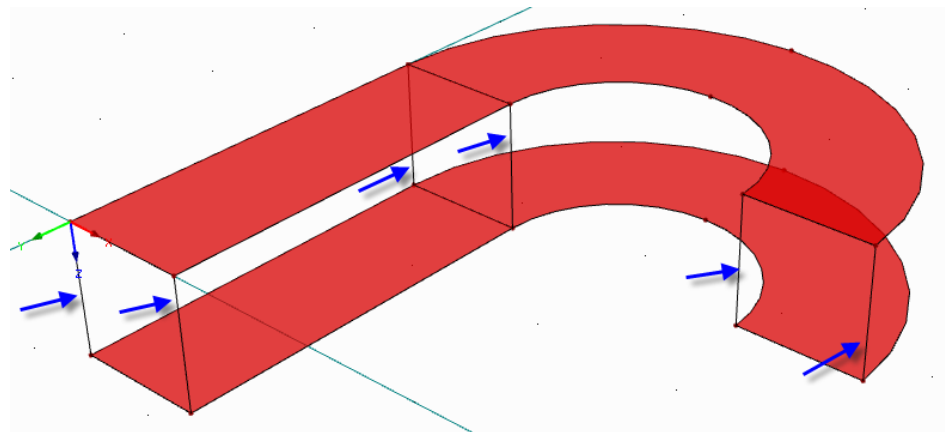


Dialog box *Move or Copy - Copy*



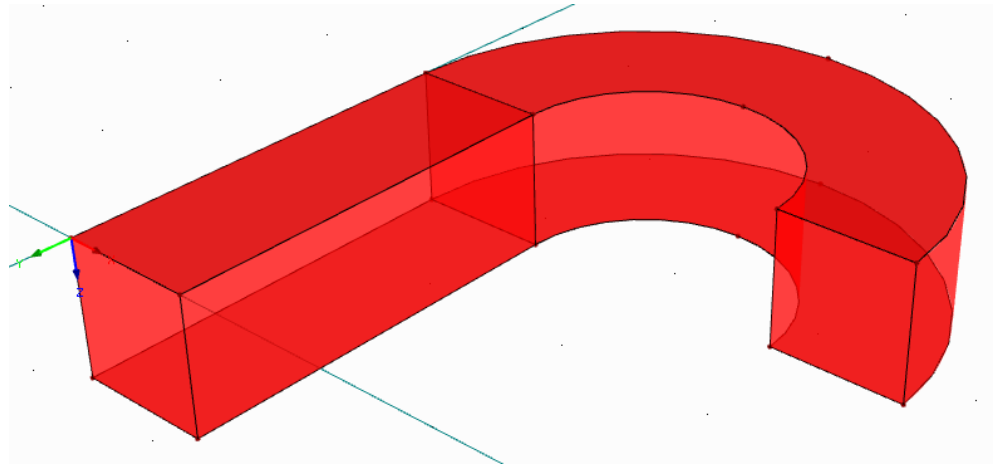
Copied surfaces

Now we connect the corner nodes of the surfaces by lines as follows.



Connecting lines

We insert new **Null** surfaces between the lines. Please note that the curved surfaces cannot be modeled by plane surfaces. We have to use surfaces of the type **Quadrangle** instead.



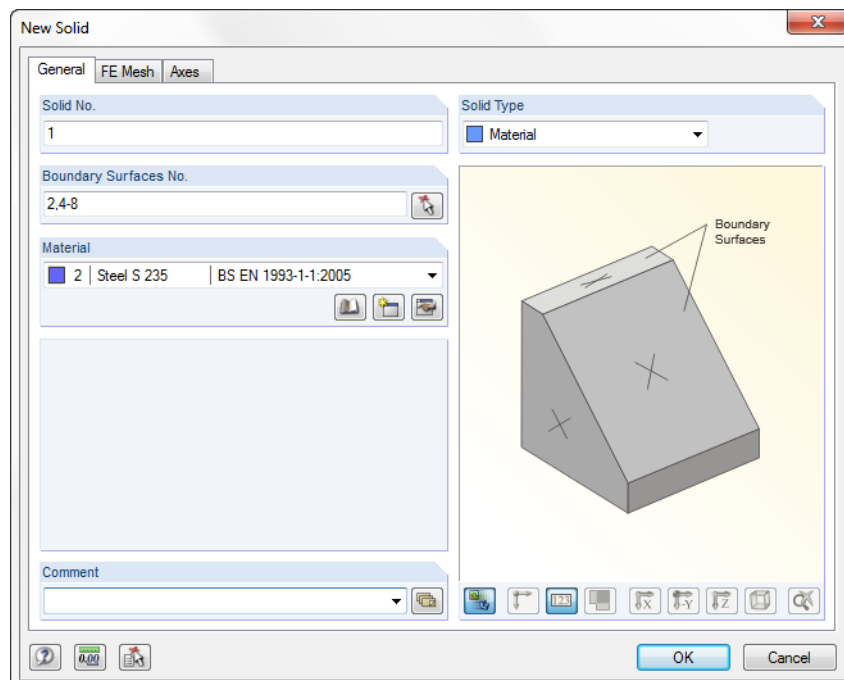
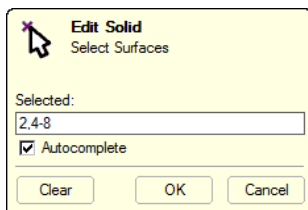
Surface model

6.1.4 Solids

Now we can create solids from the boundary surfaces. We will model the hook by two solids: The first solid consists of the cuboid, the second solid of the curved part. We click **[New Solid]**.

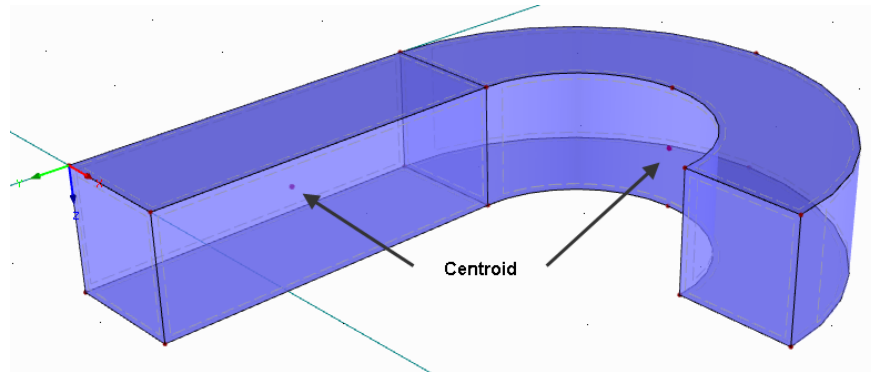
The **New Solid** dialog box opens. In the **Boundary Surface No.** section, we click **[Select Boundary Surface of the Solid]** and select the six surfaces of the first solid.

The material is **Steel S 235**. Then we close the window by clicking **[OK]**.



Dialog box *New Solid*

In the same way, we define the curved solid No. 2. The result is shown in the following figure.

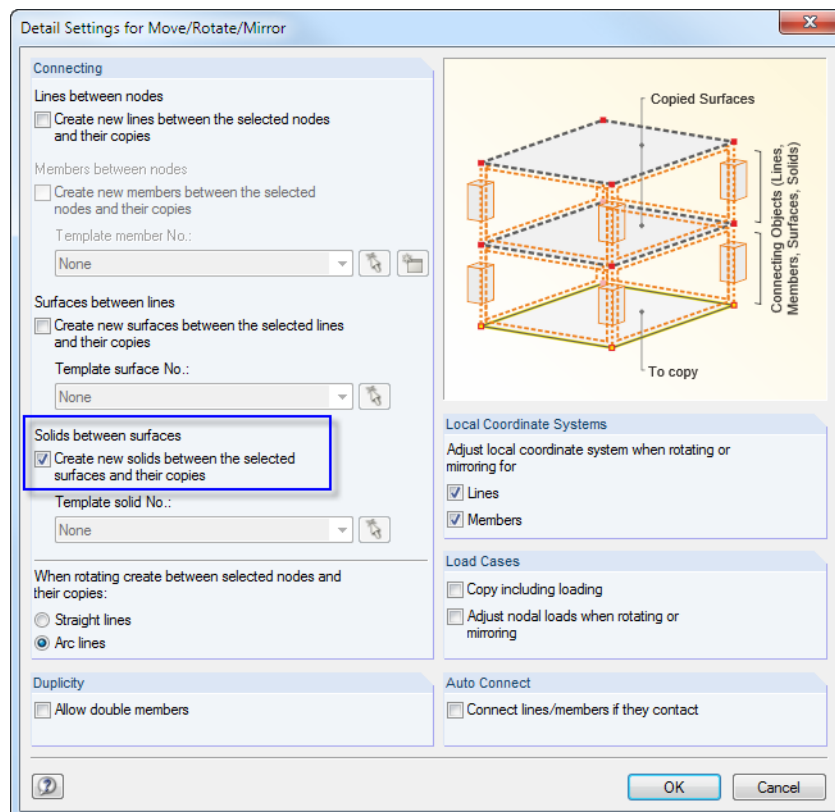


Solid model of steel hook

Solids are symbolized by purple points. Those represent their centroids.

Note:

Similar to example 2 *Steel Model*, there is a "complicated" and a "simple" way to create the solid (see chapter 3.1.3). We have chosen the difficult one. It would have been much faster to use the option **Create new solids between the selected surfaces and their copies** which is available in the [Detail Settings] of the **Move or Copy** dialog box.

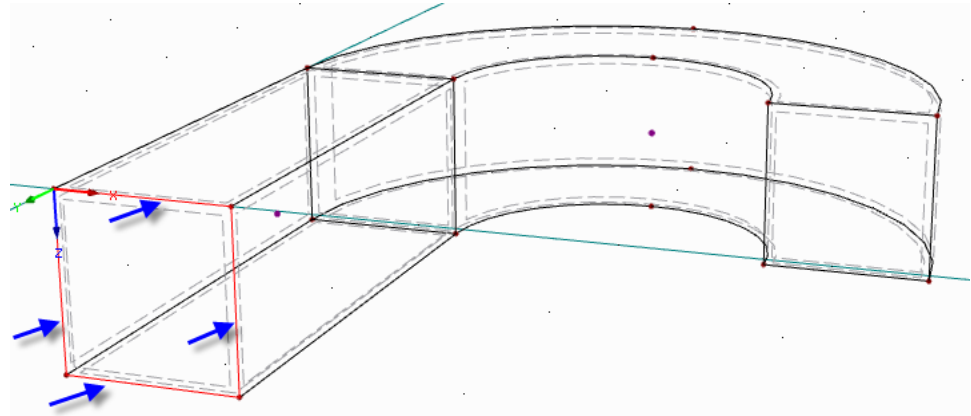


Dialog box *Detail Settings for Move/Rotate/Mirror*

6.1.5 Supports

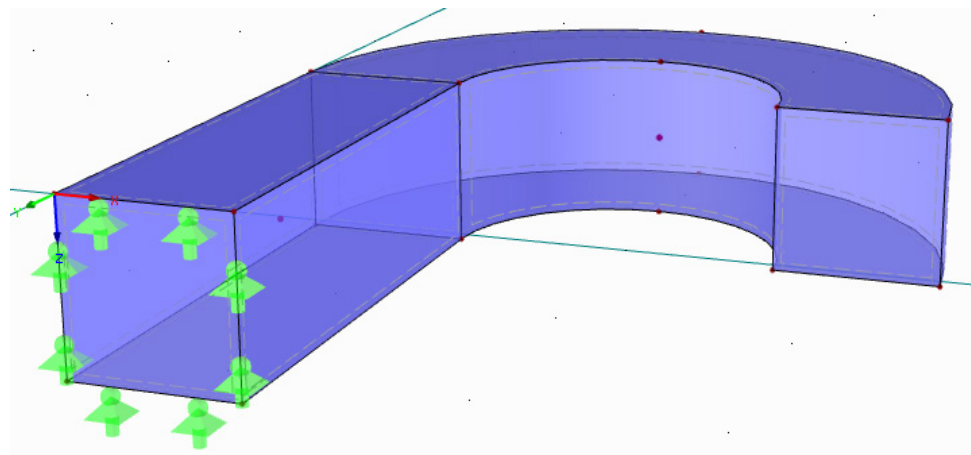


To allocate the bearings of the model, we select the four boundary lines as shown in the figure. Then we click the **[New Line Support]** button.



Selecting lines for supports

In the **Line Support** dialog box, we select the **Hinged** support type. After **[OK]**, the supported model looks as follows.



Solid model with line supports

6.2 Loads

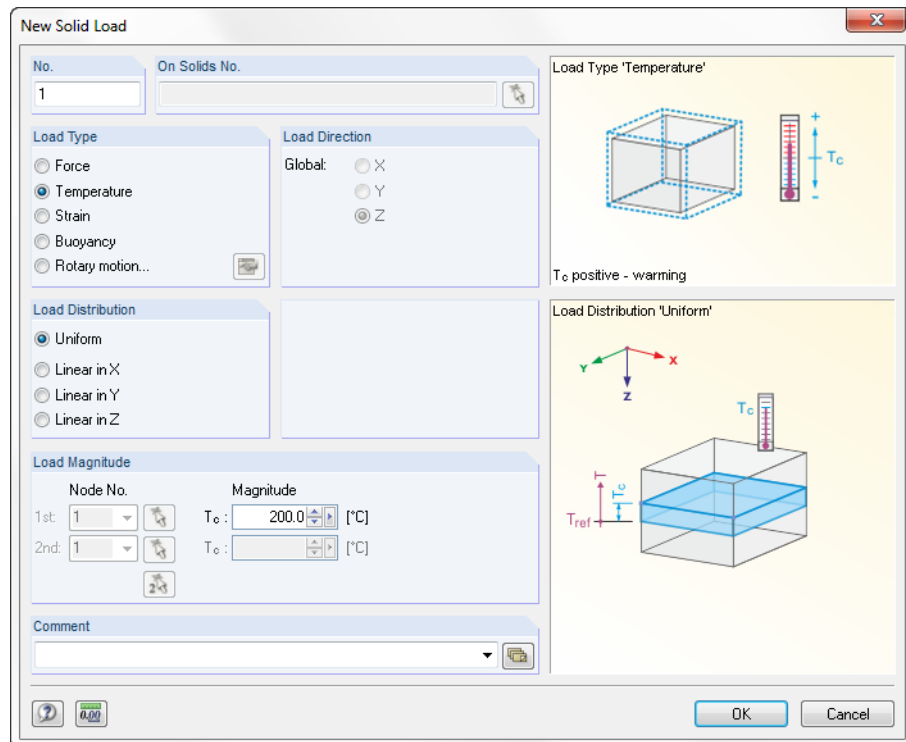
Solids can be loaded by the same loads as surfaces or member elements. Additionally, there are specific loads for solids: **Force, Temperature, Strain, Buoyancy** and **Rotary Motion**.

6.2.1 LC1: Self-Weight, Uniform Temperature

We define a new permanent load case **LC1**, taking into account the automatic self-weight.

Next we create a solid load by clicking the **[New Solid Load]** button.

The **New Solid Load** dialog box opens. We select the load type **Temperature** with a **Uniform** load distribution. The load magnitude is **200 °C**. We confirm the input by **[OK]**.

Dialog box *New Solid Load*

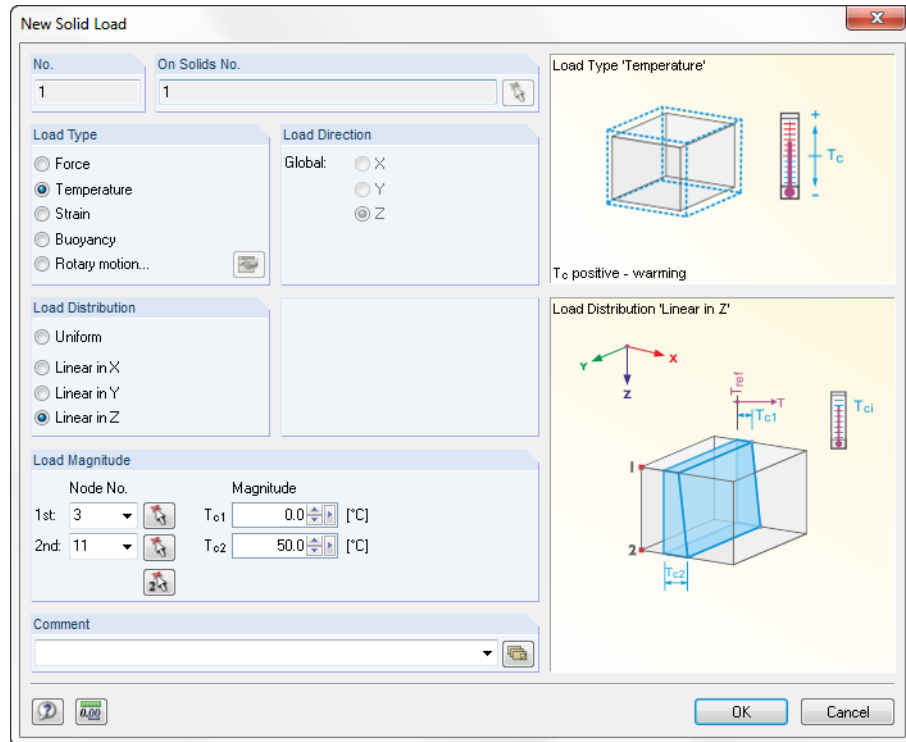
We click the two solids one after the other to allocate the temperature load.

6.2.2 LC2: Temperature Difference

We create the variable load case **LC2**.

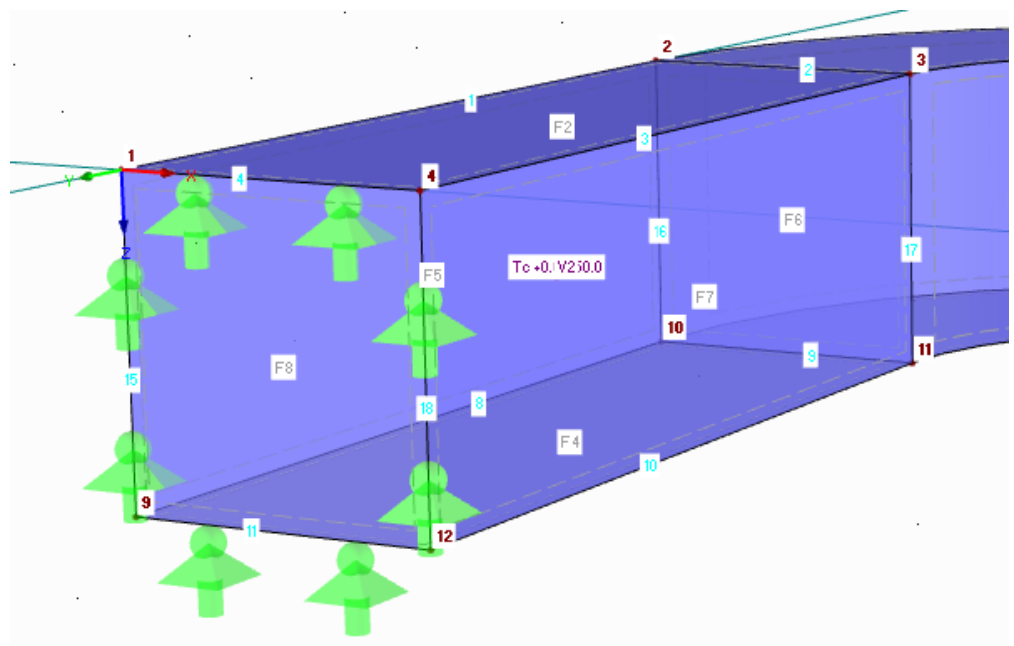


In this load case, we define a solid load again. We select the load type **Temperature** with a load distribution that is **Linear in Z**. The load magnitudes are between **0 °C** and **50 °C**. As reference nodes, we select two nodes that are part of the upper and lower surfaces, for example nodes 3 and 11.



Dialog box *New Solid Load*

We click **[OK]**. Then we select **Solid 1** to allocate the solid load.



Solid load

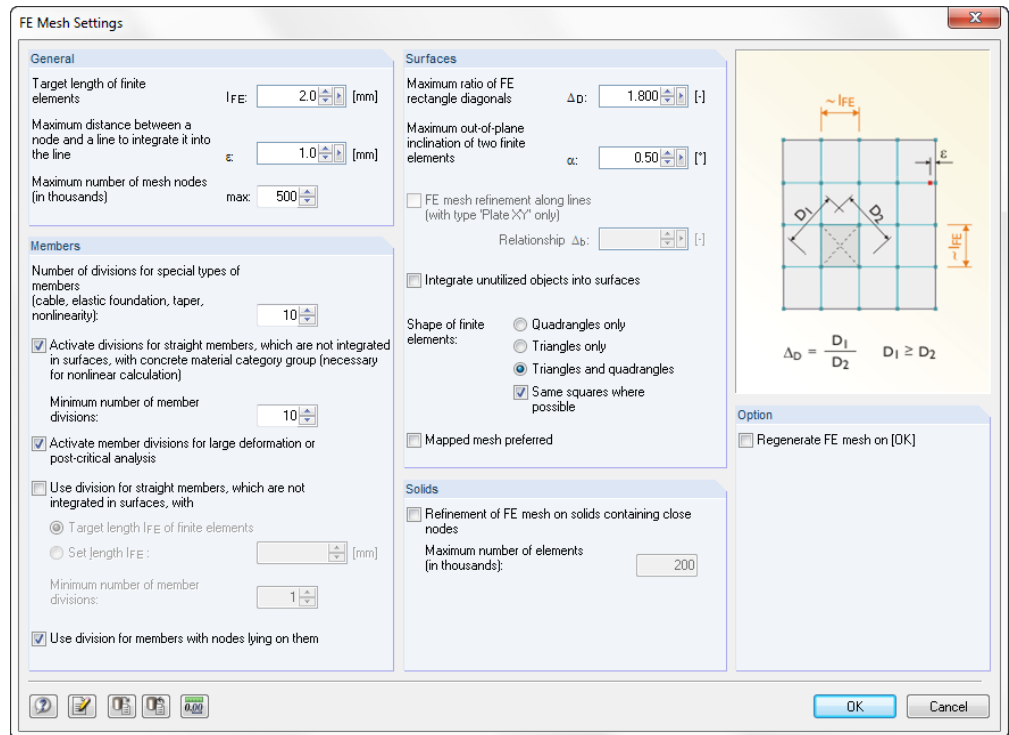
6.3 FE Mesh

By clicking menu

Calculate → **FE Mesh Settings**,

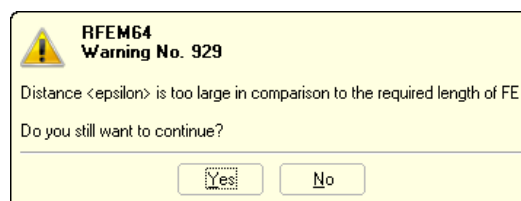
we open the **FE Mesh Settings** dialog box.

We set the target length to **2 mm** which is to be applied for the global mesh.



Dialog box *FE Mesh Settings*

If a warning appears, we confirm it with **[Yes]**.



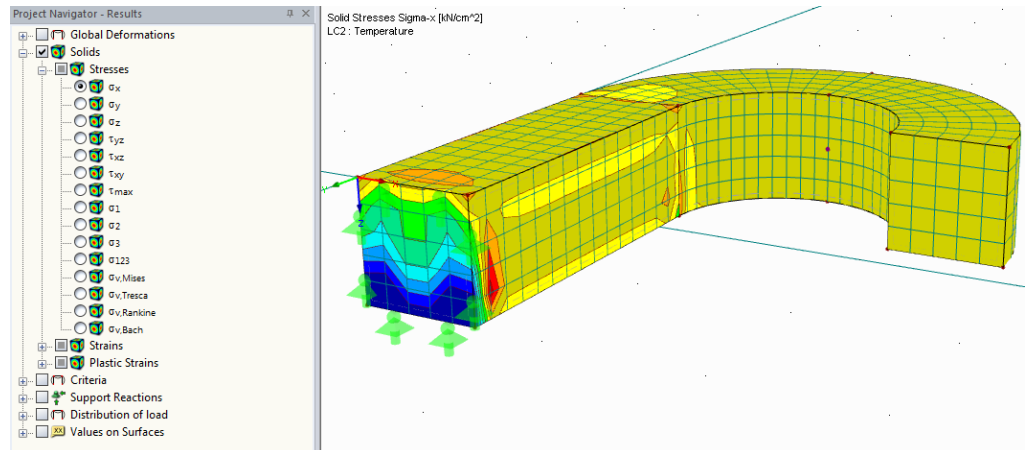
RFEM warning

6.4 Calculation and Results Evaluation



To start the calculation of all load cases, we click **[Calculate All]**.

After the calculation, RFEM shows the **Solids** entry in the **Results** navigator. It lists all calculated stresses that can be displayed graphically.



Results navigator: stresses and strains of solids

Unlike surface stresses, solid stresses cannot be described by simple equations. σ_x , σ_y , σ_z , τ_{xy} , τ_{yz} , and τ_{xz} are basic stresses that are determined directly by the solver. The principal stresses σ_1 , σ_2 , and σ_3 are eigenvalues of the stress tensor. They are determined from the basic stresses.

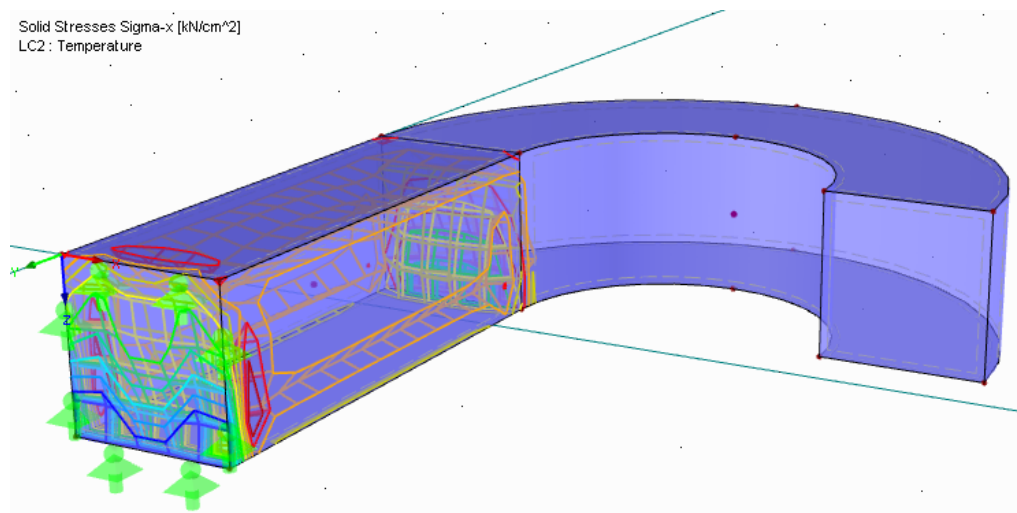
The stress $\tau_{\max}$ is determined according to the following formula:

$$\tau_{\max} = \frac{1}{2} \cdot (\sigma_1 - \sigma_3)$$

The equivalent stress according to **von Mises** is obtained from the following formula:

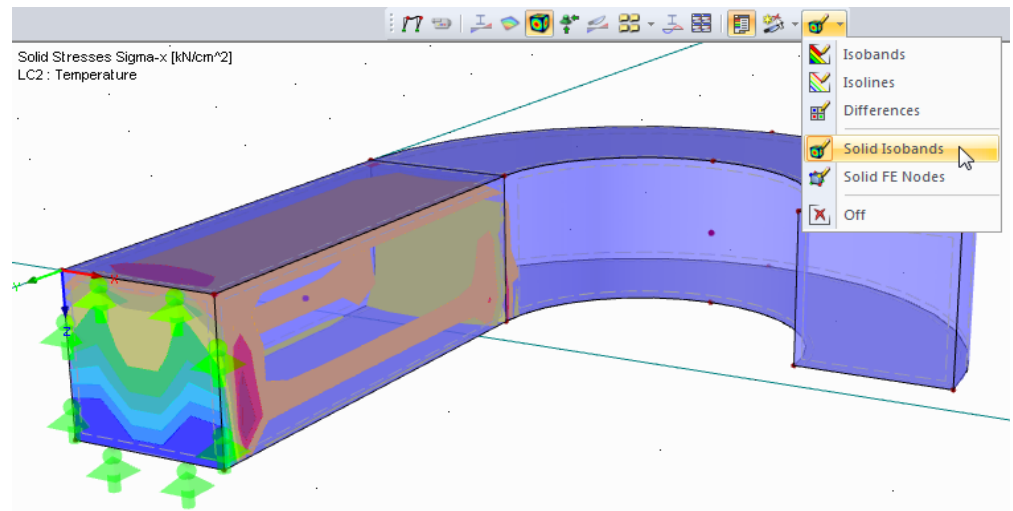
$$\sigma_v = \sqrt{\sigma_x^2 + \sigma_y^2 + \sigma_z^2 - \sigma_x \cdot \sigma_y - \sigma_x \cdot \sigma_z - \sigma_y \cdot \sigma_z + 3 \cdot (\tau_{xy}^2 + \tau_{xz}^2 + \tau_{yz}^2)}$$

There are many possibilities to display the results of the solids. The figure above shows the **Isobands** display option. The figure below shows the **Isolines** display option.



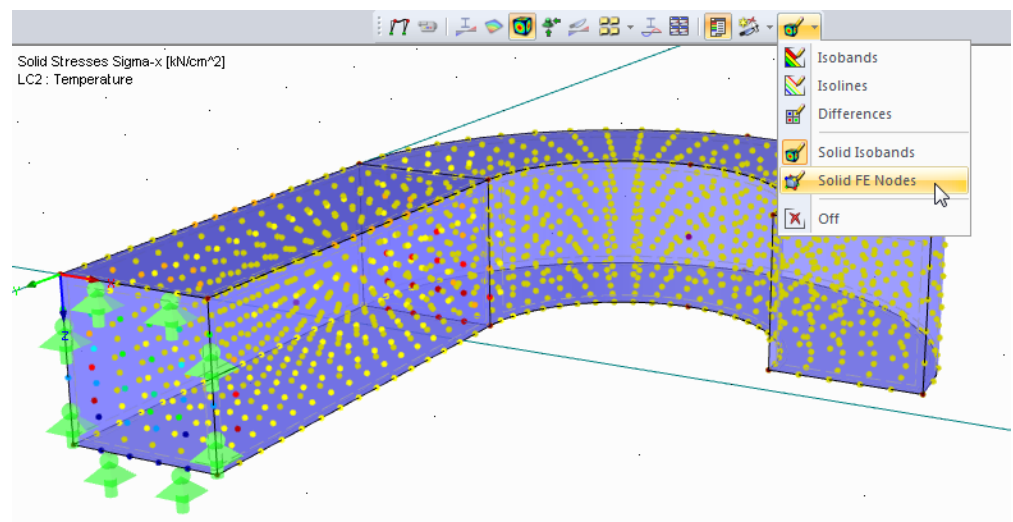
Isolines of stresses

With the **Solid Isobands** display option, the results look as follows:



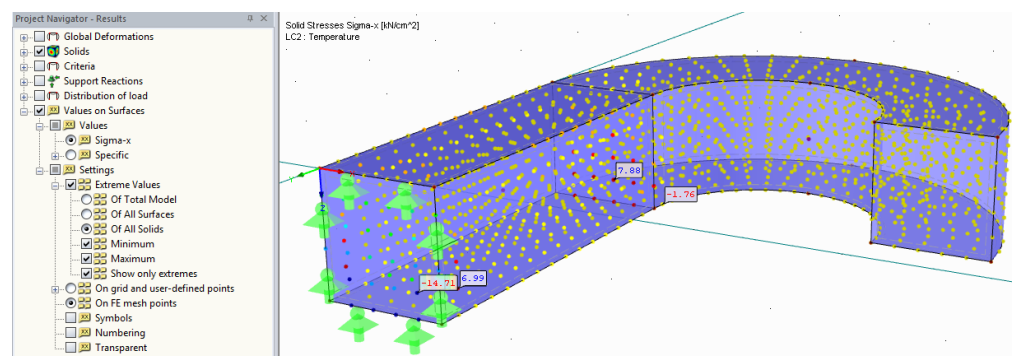
Solid isobands

The next figure shows the results in the **Solid FE Nodes**.



Solid FE nodes

There is also the possibility to display the result values of solids.



Extreme values of solids

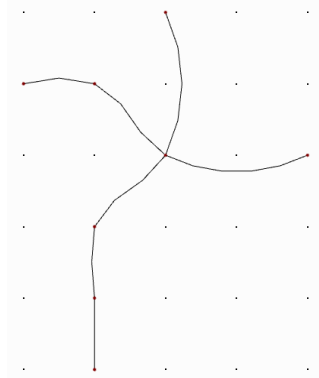
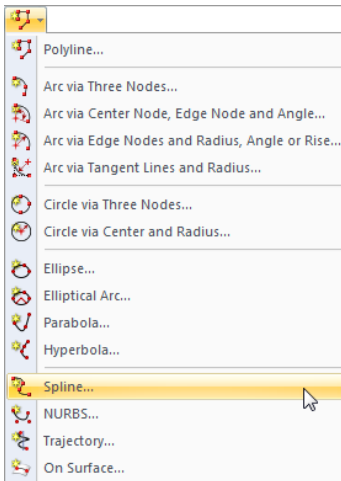
7. Special Topics

7.1 Intersections

At first, we create a new model with the name **Intersections**.

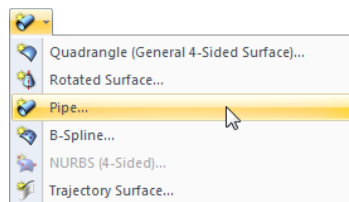
7.1.1 Pipes

We draw two crossing lines by using the line type **Spline**. The result should look like this:



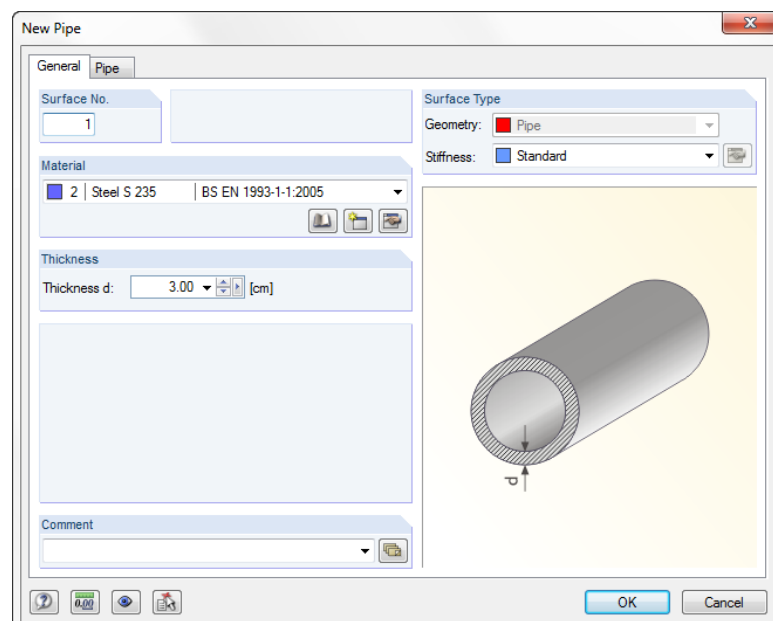
Crossing splines

From these lines, we create two pipes by clicking the **[New Pipe Surface]** list button.



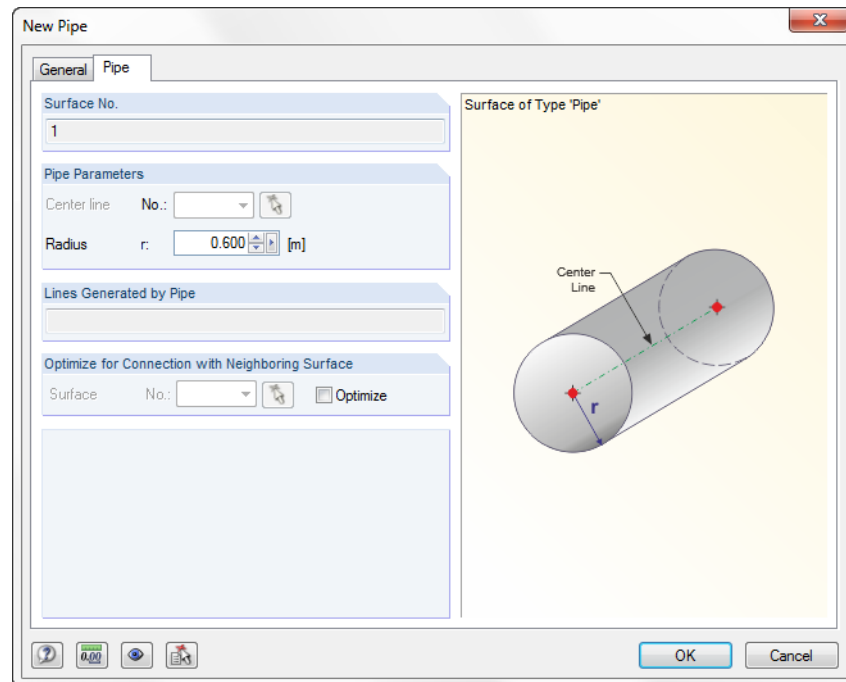
List button for quadrangles, rotated surfaces, pipes etc.

In the **New Pipe** dialog box, we set **Steel S 235** and enter the value **3 cm** for the thickness d .



Dialog box **New Pipe**, tab **General**

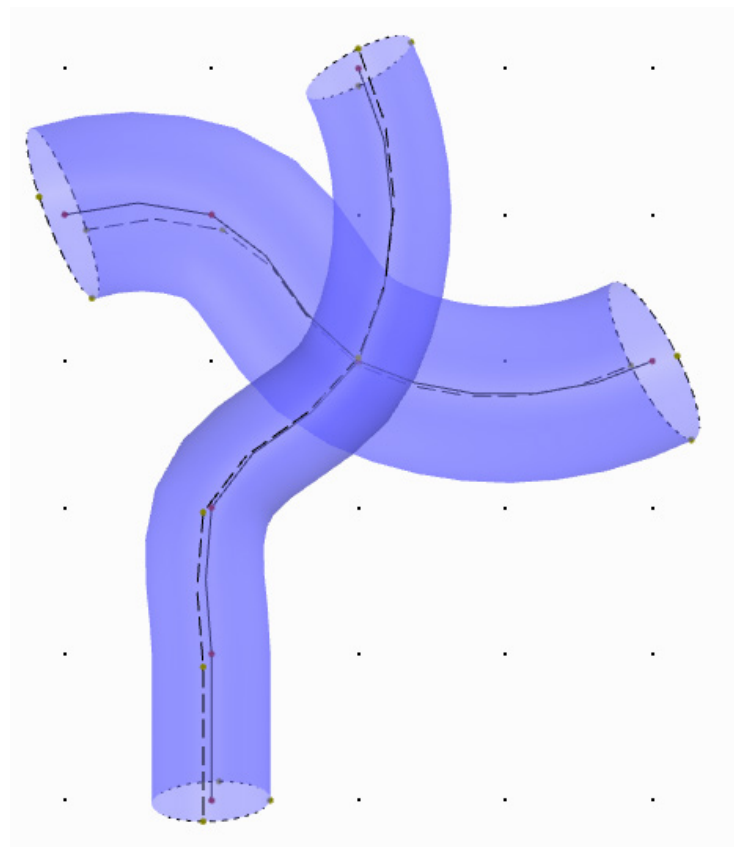
In the **Pipe** tab, we enter the radius **0.6 m**. Then we close the dialog box by clicking **[OK]**.



Dialog box *New Pipe*, tab *Pipe*

We click the "horizontal" line to allocate the pipe surface.

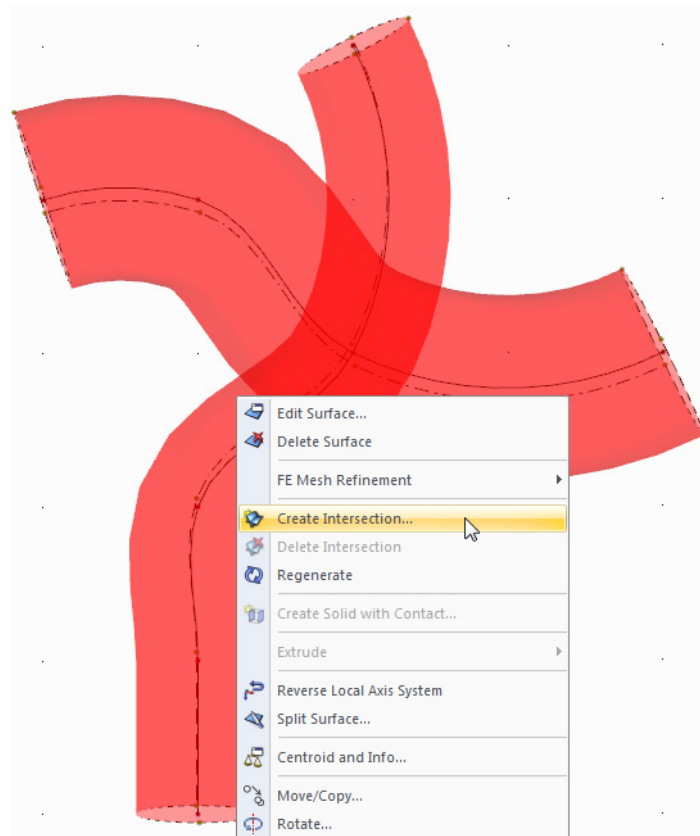
Then we open the **New Pipe** dialog box again. This time, we set the radius to **0.4 m**. After **[OK]**, we click the "vertical" line.



Pipes with different diameters

7.1.2 Intersections

Now we select the two pipes and right-click them. On the context menu, we select the option **Create Intersection**.

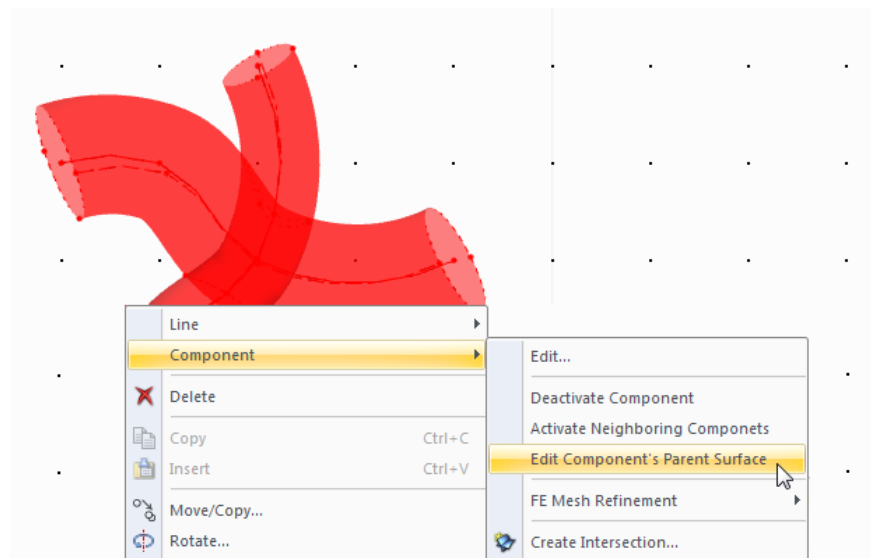


Surfaces context menu

RFEM creates the intersection.

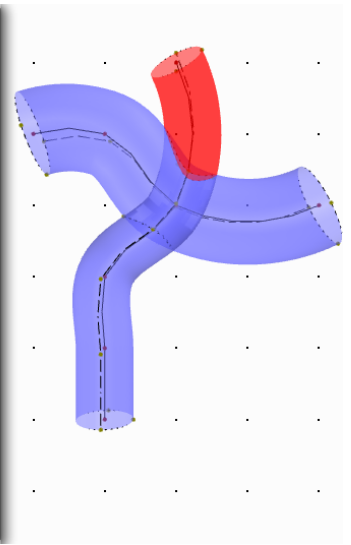
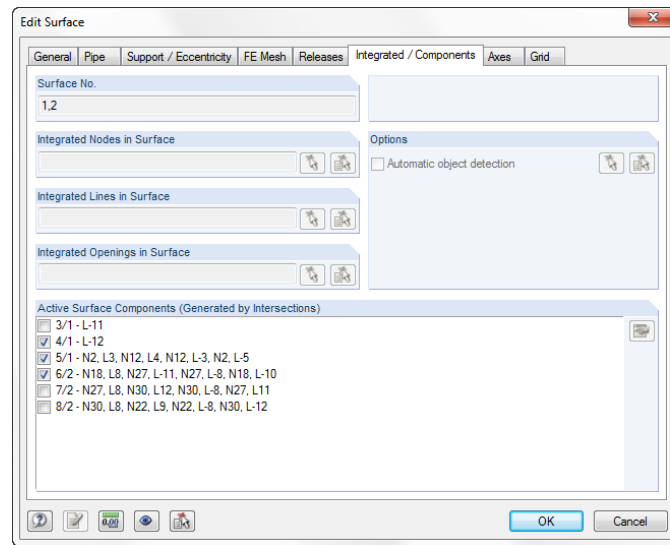
We select the model again and then right-click it. On the context menu, we click

Component → **Edit Component's Parent Surface**.



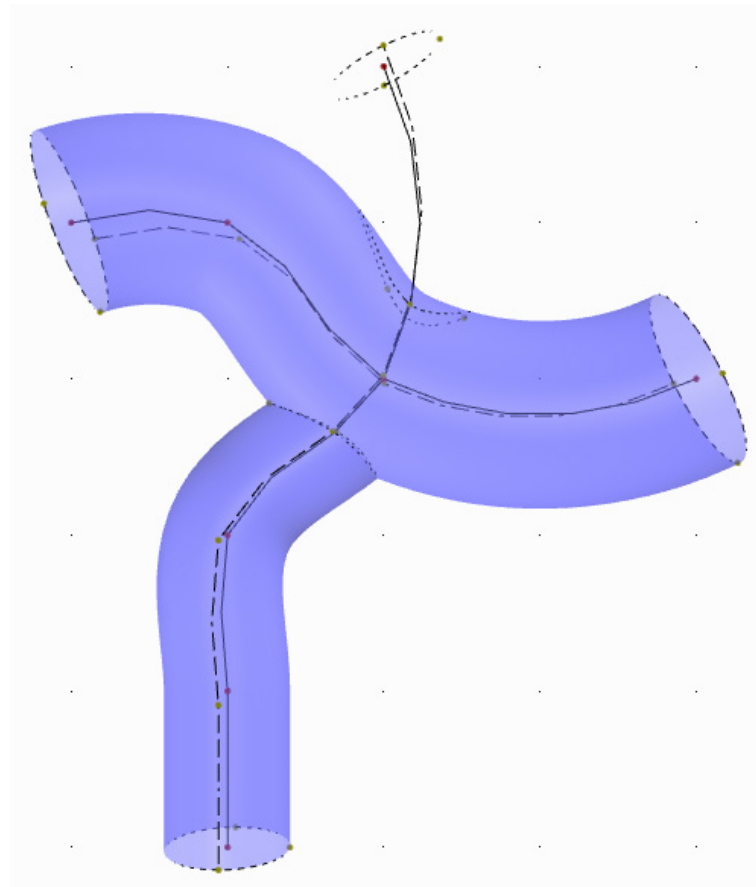
Component context menu

In the **Integrated / Components** tab of the **Edit Surface** dialog box, we find a list with all the surface components that were created by the intersection.



Dialog box *Edit Surface*, tab *Integrated/Components*

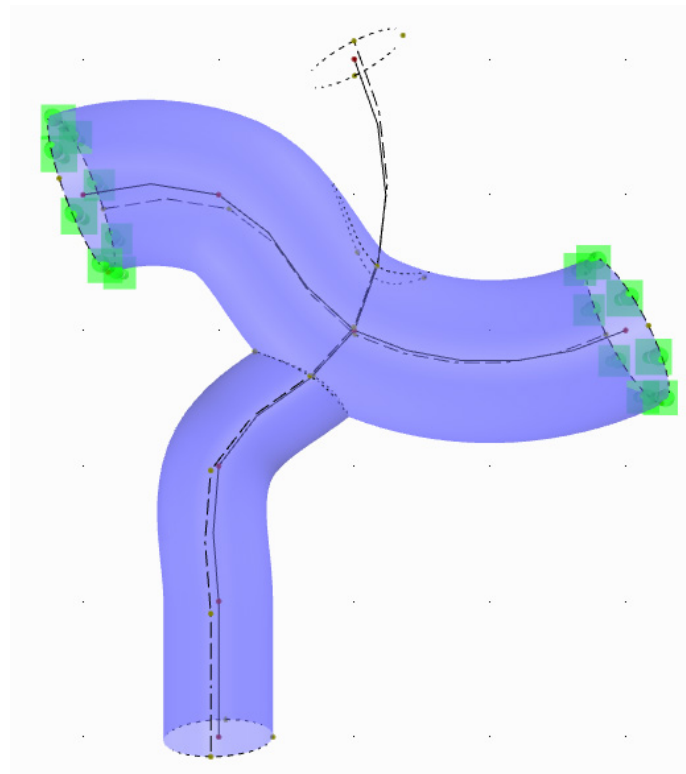
We deactivate the components that we do not need (see figure above) to obtain the following model:



Pipe model with deactivated surface components

7.1.3 Supports

We assign hinged line supports to the model as follows:



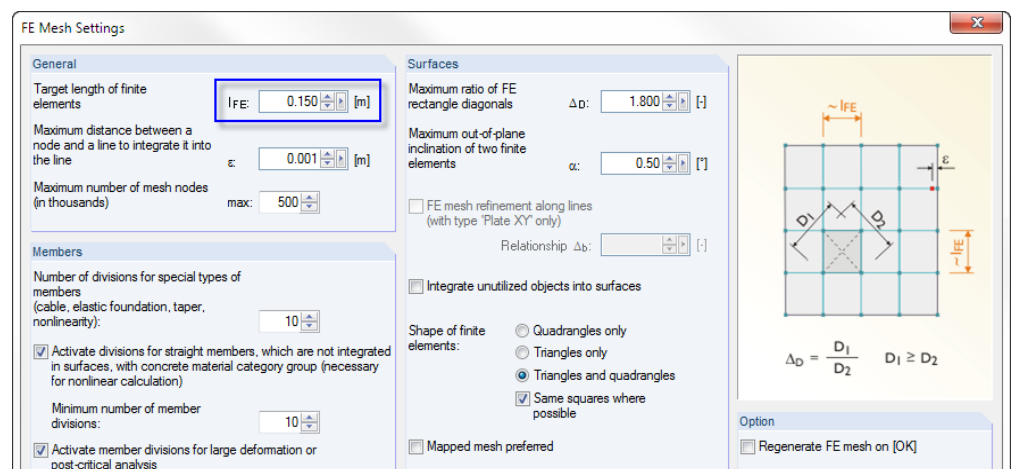
Line supports of horizontal pipe

7.1.4 LC1: Self-Weight

We create LC1 where we take into account the automatic self-weight.

7.1.5 FE-Mesh Settings

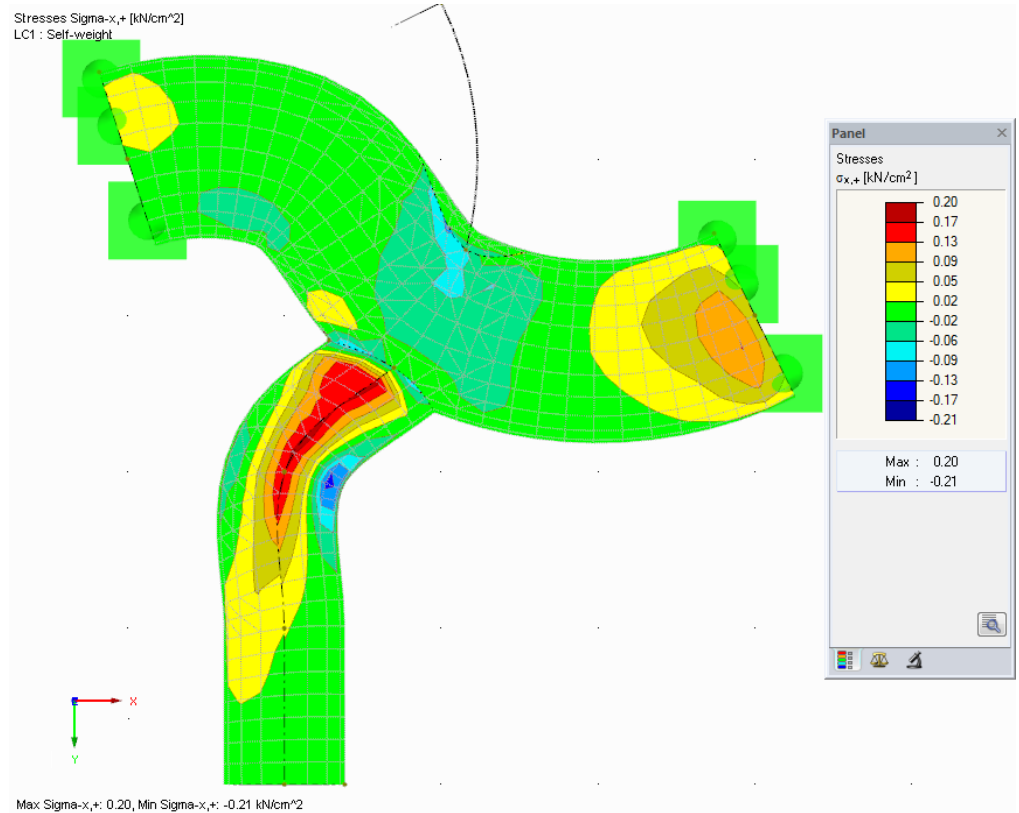
In the dialog box **FE Mesh Settings**, we set the target length of the finite elements to **0.15 m**.



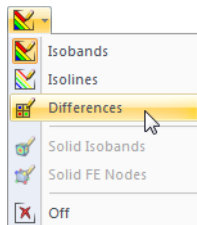
Dialog box **FE Mesh Settings**

7.1.6 Results

We calculate load case LC1. Then we display the surface stresses $\sigma_{x,+}$, for example.

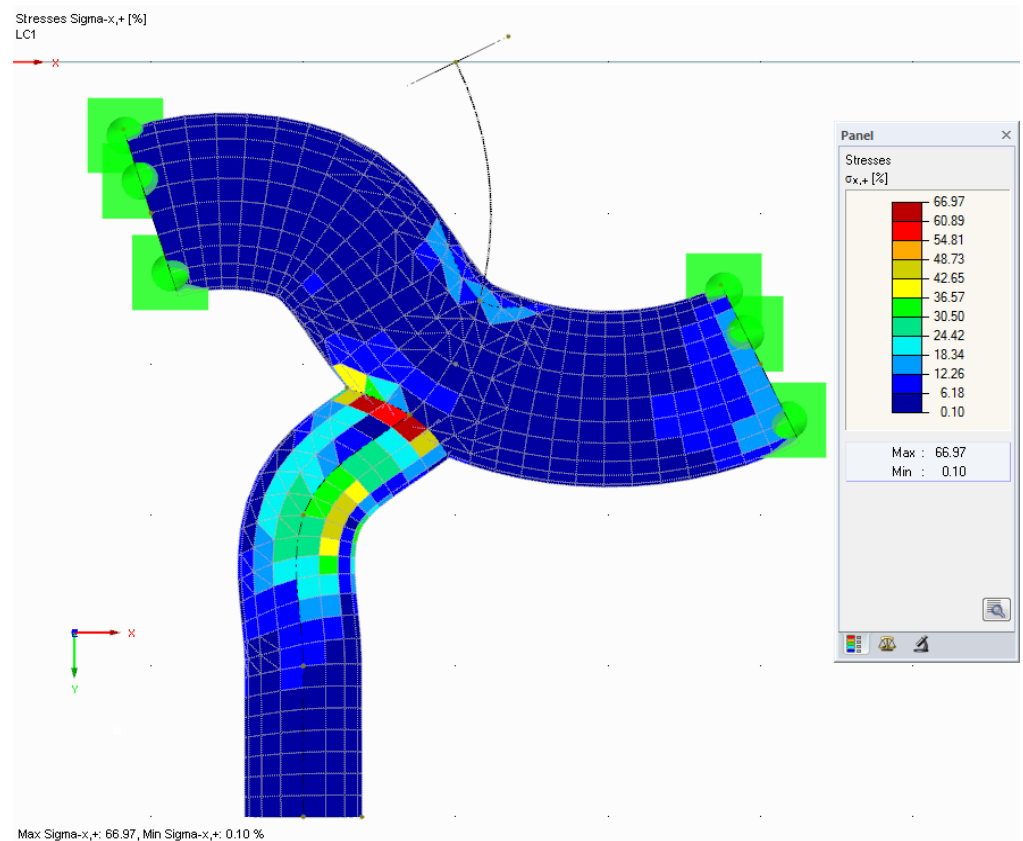


Surface stresses $\sigma_{x,+}$



Is the FE mesh fine enough? There is a useful function to answer that question: On the **[Results as Isobands]** button menu, we select the **Differences** option which is only available for surface stresses and deformations. Thus, the differences of the finite element results are shown in percent.

The figure below shows that the differences significantly increase in the connection zone. We ought to refine the FE mesh in those areas.

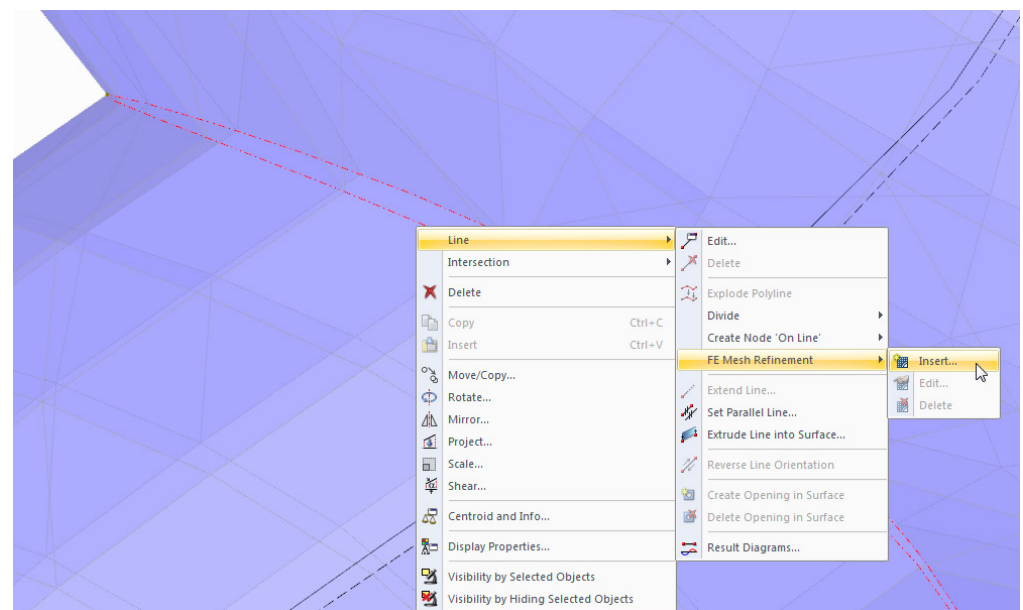


Differences of stresses $\sigma_{x,+}$

7.1.7 FE Mesh Refinement

Line

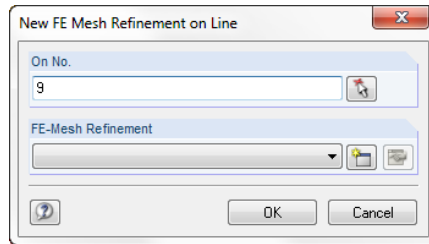
We define a FE mesh refinement for the connection line of the two pipes by using the context menu of that line.



Line context menu

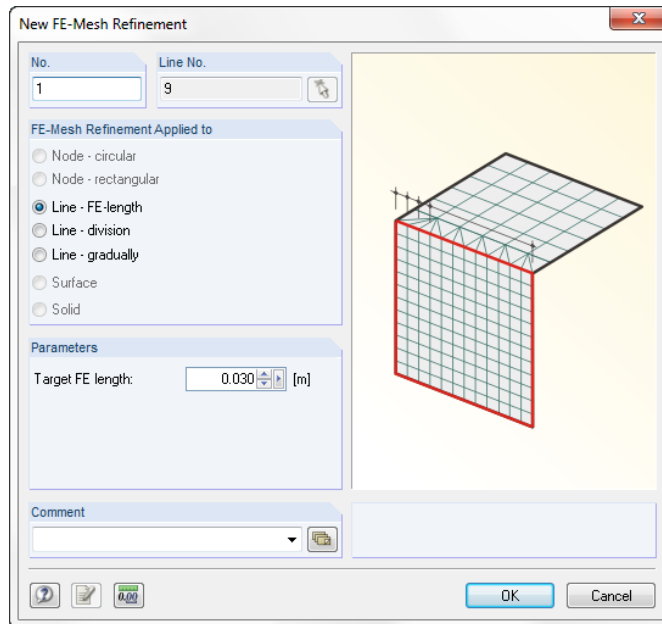


In the **New FE Mesh Refinement on Line** dialog box, we click **[New FE Mesh Refinement]**.



Dialog box *New FE Mesh Refinement on Line*

We select the **Line - FE-Length** option and then set the target FE length to **0.030 m**.



Dialog box *New FE Mesh Refinement*

We click **[OK]** and calculate LC1 again.

Stresses Sigma-x,+: [%]
LC1 : Self-weight

Panel

Stresses
 $\sigma_{x,+}$ [%]

53.55
48.69
43.83
38.97
34.11
29.25
24.39
19.53
14.67
9.80
4.94
0.08

Max : 53.55
Min : 0.08

Max Sigma-x,+: 53.55, Min Sigma-x,+: 0.08 %

217

7.2 Singularities

7.2.1 Theory

What are singularities?

The phenomenon of the so-called singularities occurs in certain critical parts of an FE model. Singularities are the locations of discontinuity in the distribution of internal forces, whereby a smaller size of the finite elements increases the internal forces. If the mesh width approaches zero, the internal force becomes (at least theoretically) infinite at the location of singularities.

In the reality, singularities are of no importance, since the specific boundary conditions which are responsible in the first place do not exist in the real model. Therefore, a support is always a (small) surface in reality. Single loads also need a surface that they can be applied on. In reality, there are no "rigid" supports, too. Furthermore, the material is not ideally elastic, but yielding.

Where do singularities occur?

Singularities occur in case of:

- Node supports
- Line supports
- Single loads
- "Pointed" corners

How can we avoid singularities in the FE model?

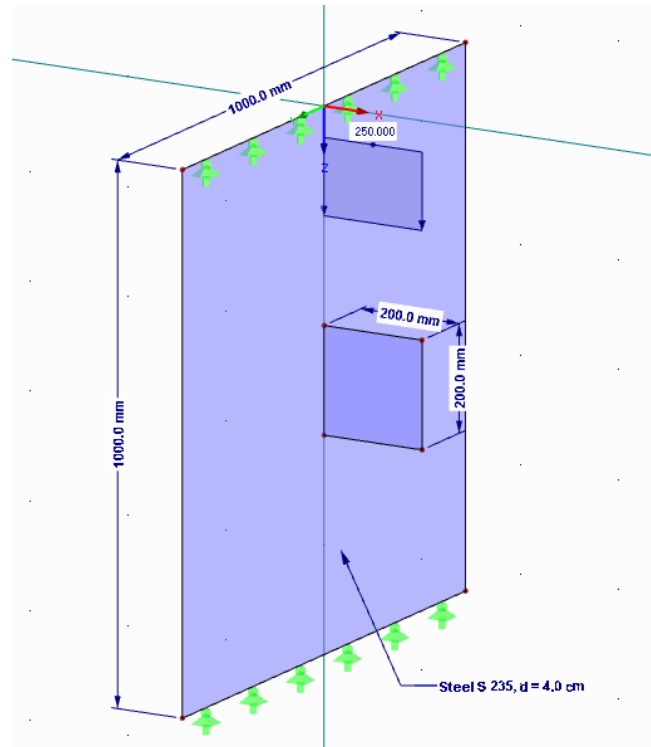
- Model single loads as surfaces loads
- Define support conditions with failure in case of tension or as springs
- Model node and line supports as surface supports ("column or wall")
- Smooth "pointed" corners by arcs or straight lines

What can we do with unavoidable singularities?

- Allocate mesh refinements to restrict singularities to specific areas
- Ignore if the singularity is within the geometrical dimensions of adjacent component
- Use filled opening

7.2.2 Example

First, we create a new model that we name **Lug panel**. Then we define the model as follows:

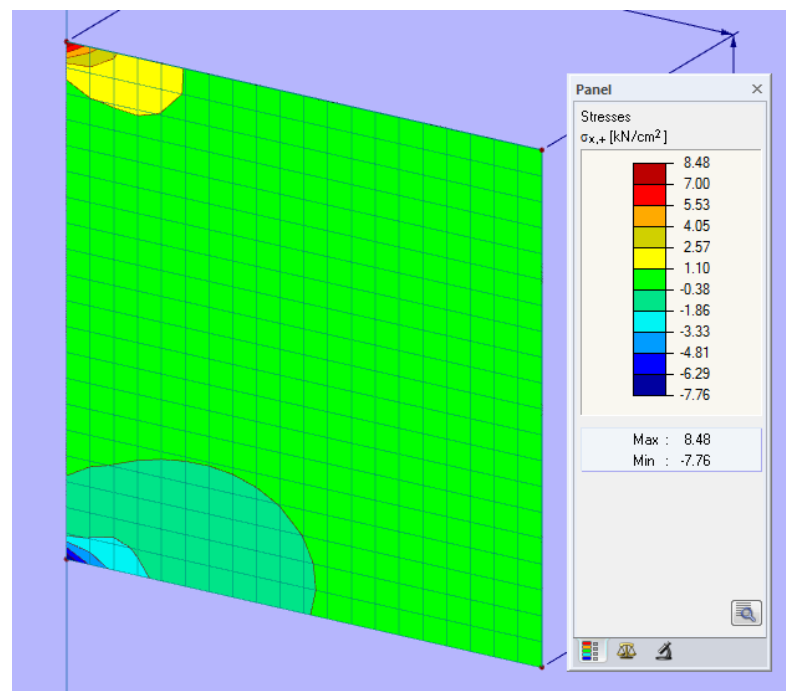


Model and line load

Load case LC1 contains the self-weight and a line load of **250 kN/m** (see figure above).

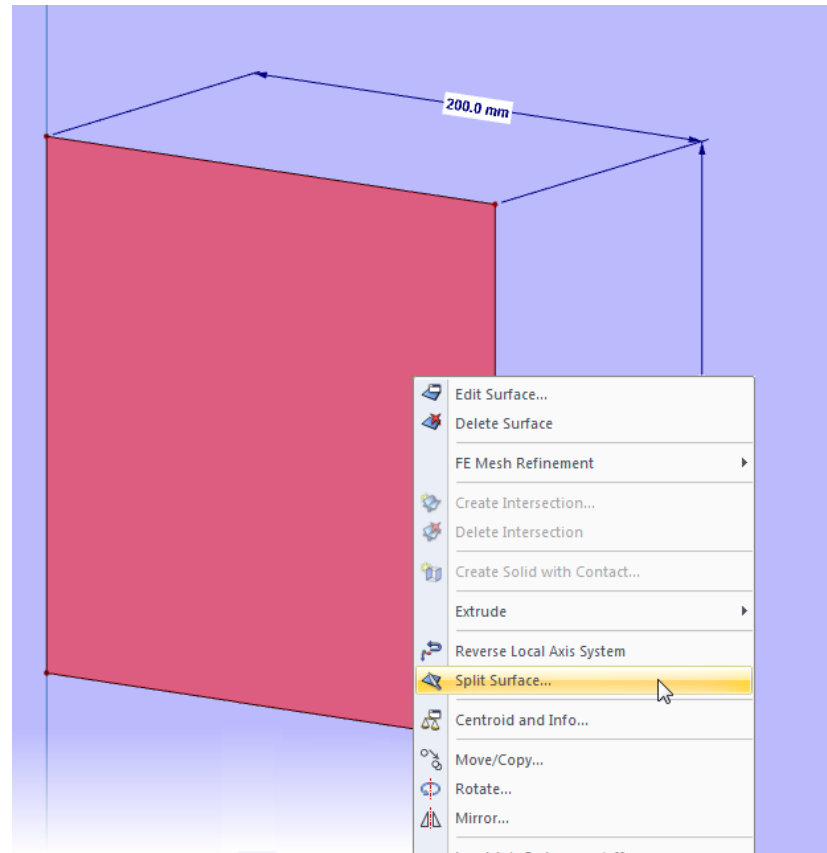
We set the target length of the FE mesh to **0.01 m**.

After the calculation, we display the stresses $\sigma_{x,+}$ of the lug panel only. The singularities are distinctly shown in the two corner points of the connection line.



Stresses $\sigma_{x,+}$ in lug panel showing singularities

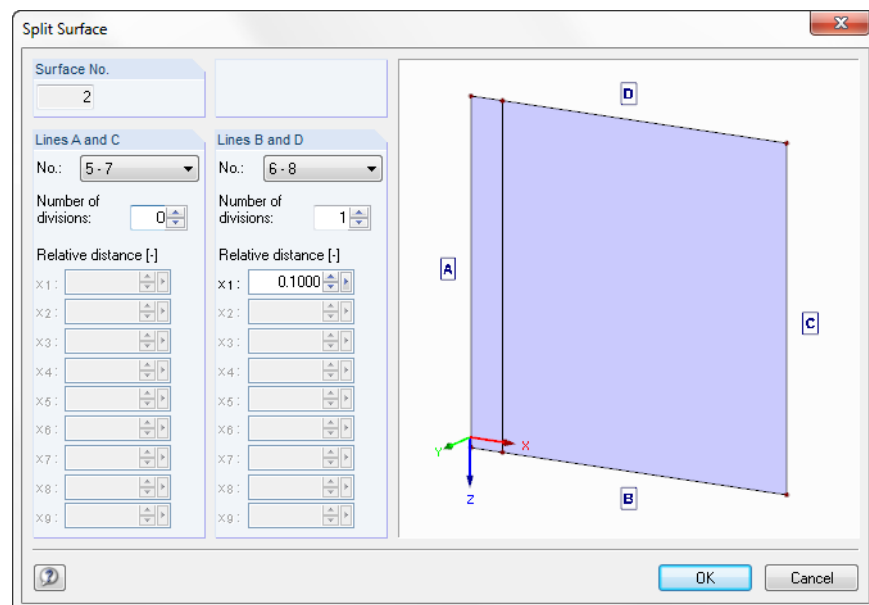
We right-click the surface of the lug panel. On the context menu, we select **Split Surface**.



Surface context menu

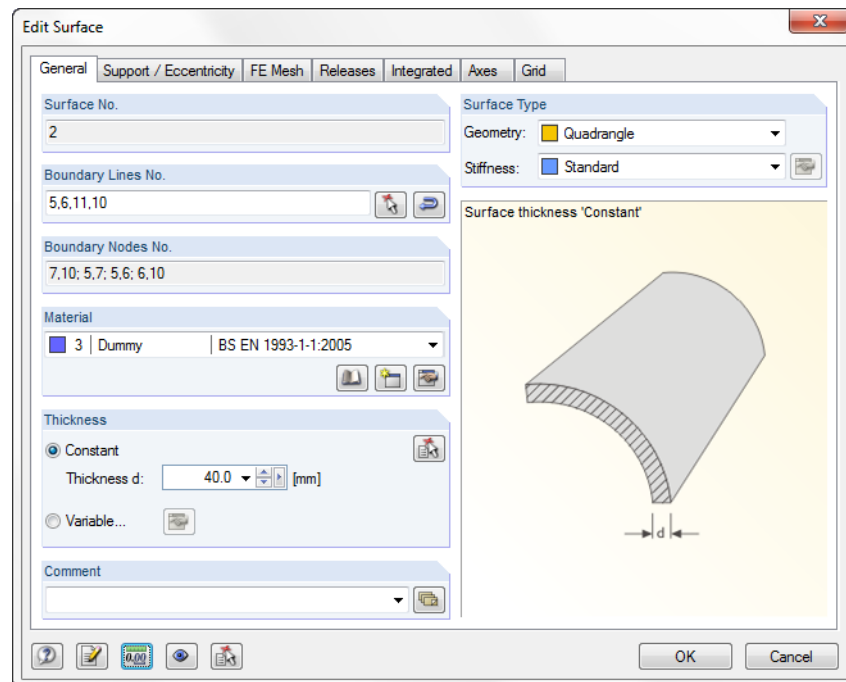
2 cm are exactly half of the surface thickness. This means that any results closer than 2 cm to the connection line are irrelevant. They can be ignored. Our surface is 200 mm long. Therefore, the relative distance of a division must be 0.1 to obtain those 2 cm.

We specify the dialog box as shown in the figure below and then click **[OK]**.



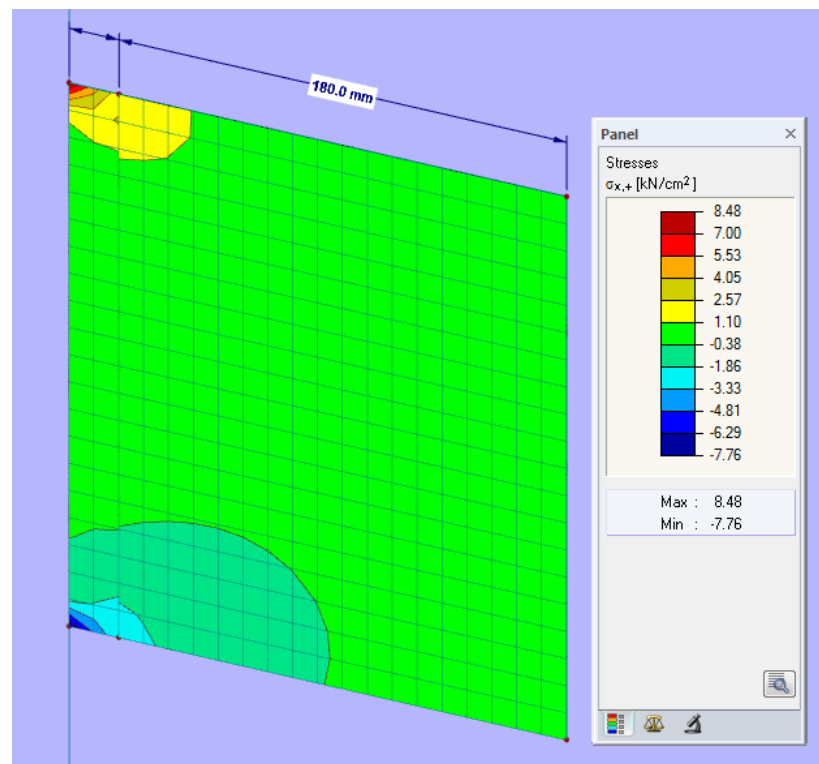
Dialog box *Split Surface*

We assign a new material to the small new surface. The material has the same properties as the basic material Steel S 235, but we name it "**Dummy**". Thus, we can select it easily.



Dialog box *Edit Surface*

We calculate the model again. The stresses $\sigma_{x,+}$ look similar to the previous model with only one panel surface.



Stresses $\sigma_{x,+}$ in lug panel (two surfaces)

Visibilities

☒ User-defined / generated:

- User-defined
 - Groupe 1
 - Generated
 - Lines by Type
 - Lines with Support
 - Surfaces by Eccentricity
 - Surfaces by Geometry
 - Surfaces by Material
 - ☒ 1 - Steel S 235
 - 3 - Dummy, Steel S 235**
 - Surfaces by Stiffness
 - Surfaces by Thickness

☐ Add new objects into visibility:

Data
 Display
 Views
 Results

222