

**Version**  
**July 2013**

**Program**

# **RFEM 5**

**Spatial Models Calculated acc. to  
Finite Element Method**

## **Tutorial**

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](mailto:info@dlubal.com)  
Web: [www.dlubal.com](http://www.dlubal.com)



# Contents

Contents		Page	Contents		Page
<b>1.</b>	<b>Introduction</b>	<b>5</b>	4.5.3	Diagonal	42
<b>2.</b>	<b>System and Loads</b>	<b>6</b>	4.5.3.1	Defining Member	42
<b>2.1</b>	<b>Sketch of System</b>	<b>6</b>	4.5.3.2	Rotating Member	43
<b>2.2</b>	<b>Materials, Thicknesses and Cross-sections</b>	<b>6</b>	<b>4.6</b>	<b>Checking Input</b>	<b>45</b>
<b>2.3</b>	<b>Load</b>	<b>7</b>	<b>5.</b>	<b>Loads</b>	<b>46</b>
<b>3.</b>	<b>Creating Model</b>	<b>8</b>	<b>5.1</b>	<b>Load Case 1: Self-weight</b>	<b>46</b>
<b>3.1</b>	<b>Starting RFEM</b>	<b>8</b>	5.1.1	Self-weight	47
<b>3.2</b>	<b>Creating Model File</b>	<b>8</b>	5.1.2	Floor Structure	47
<b>4.</b>	<b>Model Data</b>	<b>9</b>	5.1.3	Earth Pressure	48
<b>4.1</b>	<b>Adjusting Work Window and Grid</b>	<b>9</b>	5.1.4	Roof Load	49
<b>4.2</b>	<b>Creating Surfaces</b>	<b>11</b>	<b>5.2</b>	<b>Load Case 2: Live Load</b>	<b>51</b>
4.2.1	Floor	11	5.2.1	Floor Slab	52
4.2.1.1	Defining Rectangular Surface	11	5.2.2	Edge of Opening	53
4.2.1.2	Creating Arc	14	<b>5.3</b>	<b>Load Case 3: Snow</b>	<b>54</b>
4.2.1.3	Adjusting Floor Surface	15	5.3.1	Roof	54
4.2.2	Wall	16	5.3.2	Floor	55
4.2.3	Opening	19	<b>5.4</b>	<b>Load Case 4: Wind</b>	<b>57</b>
4.2.3.1	Creating Opening	19	5.4.1	Steel Construction Loads	58
4.2.3.2	Adjusting Opening	20	5.4.2	Column Loads	59
<b>4.3</b>	<b>Creating Concrete Members</b>	<b>21</b>	<b>5.5</b>	<b>Load Case 5: Imperfection</b>	<b>62</b>
4.3.1	Columns	21	5.5.1	Steel Columns	63
4.3.2	Rib	24	5.5.2	Concrete Columns	64
<b>4.4</b>	<b>Defining Supports</b>	<b>28</b>	<b>5.6</b>	<b>Checking Load Cases</b>	<b>65</b>
4.4.1	Nodal Supports	28	<b>6.</b>	<b>Combination of Actions</b>	<b>66</b>
4.4.2	Line Supports	30	<b>6.1</b>	<b>Checking Actions</b>	<b>66</b>
<b>4.5</b>	<b>Creating Steel Members</b>	<b>31</b>	<b>6.2</b>	<b>Defining Combination Expressions</b>	<b>67</b>
4.5.1	Frame	31	<b>6.3</b>	<b>Creating Action Combinations</b>	<b>70</b>
4.5.1.1	Defining Members Continuously	32	<b>6.4</b>	<b>Creating Load Combinations</b>	<b>70</b>
4.5.1.2	Shearing Horizontal Beams	34	<b>6.5</b>	<b>Checking Result Combinations</b>	<b>73</b>
4.5.1.3	Connecting Beams with Releases	35	<b>7.</b>	<b>Calculation</b>	<b>74</b>
4.5.1.4	Reverse Member Orientation	36	<b>7.1</b>	<b>Checking Input Data</b>	<b>74</b>
4.5.1.5	Copying Frame	37	<b>7.2</b>	<b>Generating FE Mesh</b>	<b>75</b>
4.5.2	Purlins	38	<b>7.3</b>	<b>Calculating Model</b>	<b>77</b>
4.5.2.1	Defining Members Individually	38	<b>8.</b>	<b>Results</b>	<b>78</b>
4.5.2.2	Connecting Members Eccentrically	40	<b>8.1</b>	<b>Graphical Results</b>	<b>78</b>

# Contents

Contents		Page	Contents		Page
<b>8.2</b>	<b>Results Tables</b>	<b>81</b>	<b>9.1</b>	<b>Creating Printout Report</b>	<b>94</b>
<b>8.3</b>	<b>Filtering Results</b>	<b>83</b>	<b>9.2</b>	<b>Adjusting the Printout Report</b>	<b>95</b>
8.3.1	Visibilities	83	<b>9.3</b>	<b>Inserting Graphics in Printout Report</b>	<b>97</b>
8.3.2	Results on Objects	86	<b>10.</b>	<b>Outlook</b>	<b>102</b>
8.3.3	Range of Values	88			
<b>8.4</b>	<b>Display of Result Diagrams</b>	<b>90</b>			
<b>8.5</b>	<b>Creating Section</b>	<b>91</b>			
<b>9.</b>	<b>Documentation</b>	<b>94</b>			

# 1. Introduction

With this tutorial we would like to make you acquainted with various features of RFEM. Often you will have several options to achieve your targets. Depending on the situation and your preferences, different ways may be useful. We would like to invite you to play with the software to learn more about the program's possibilities. With this example we want to encourage you to find out useful functions in RFEM.

The following example represents a mixed construction built of concrete and steel elements. We want to calculate the model for the load cases self-weight, live load, snow, wind and imperfections according to linear static and nonlinear second-order analysis.

You can enter, calculate and evaluate the example of this tutorial also with demo restrictions - maximum 2 surfaces and 12 members. Therefore, you may understand that the model meets the demands of realistic construction projects only to some extent. With the features presented we just want to show you how you can define model and load objects in various ways.

As superimposing actions according to EN 1990 involves considerable time and effort, we will use the generator for load combinations already integrated in RFEM 5.



With the 30-day trial version, you can work on the model without any restriction. After that period, the demo mode will be applied so that saving data is no longer possible. In this case, you should allow for enough time (approximately two to three hours) to enter the data and try the functions without stress. You can also interrupt your work on the model in the demo version as long as you do not close RFEM: When you take a break, do not shut down your computer but use the standby mode.



It is easier to enter data if you use two screens, or you may print the description to avoid switching between the displays of PDF file and RFEM input.



The text of the manual shows the described **buttons** in square brackets, for example [Apply]. At the same time, they are pictured on the left. In addition, **expressions** used in dialog boxes, tables and menus are set in *italics* to clarify the explanations. Input required is written in **bold** letters.

You can look up the description of program functions in the RFEM manual that you can download on the Dlubal website at [www.dlubal.com/Downloading-Manuals.aspx](http://www.dlubal.com/Downloading-Manuals.aspx).

## 2. System and Loads

### 2.1 Sketch of System

Our example is a reinforced concrete structure to which a steel construction is attached.

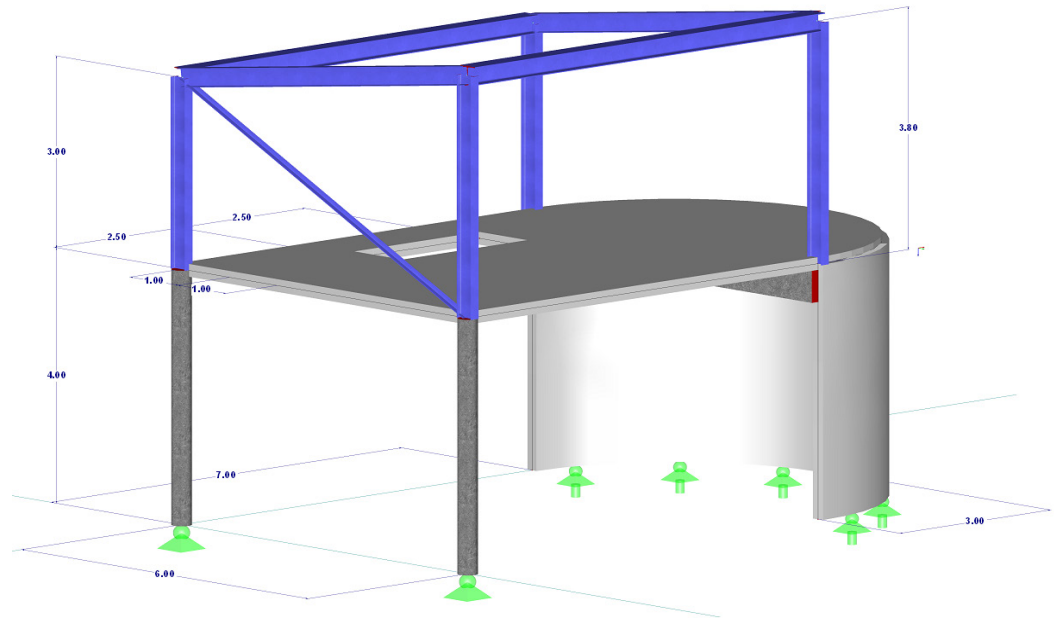


Figure 2.1: Structural system

The reinforced concrete structure is a substructure consisting of a floor slab with downstand beam, a semicircular shell and two round columns. The structural system is built partially in the earth.

The steel frame is a monopitch roof construction stiffened with a diagonal.

As mentioned above, the model represents a rather "abstract" structure that can be designed also with the demo version whose functions are restricted to a maximum of two surfaces and twelve members.

### 2.2 Materials, Thicknesses and Cross-sections

We use concrete C30/37 and steel S 235 as materials.

The thickness of walls and floors is 20 cm each. Both concrete columns have a diameter of 30 cm. The beam has a width of 25 cm and height of 40 cm.

We use HE-A 300 cross-sections for the left and right steel frames of the monopitch roof structure. Both purlins are defined with HE-B 260 cross-sections. The bracing diagonal consists of an angle iron L 80x8 with equal legs.

### 2.3 Load

#### Load case 1: self-weight and finishes

In the first load case, the self-weight of the model including its floor structure of  $1.5 \text{ kN/m}^2$  is applied. We do not need to determine the self-weight manually. RFEM calculates the weight automatically from the defined materials, surface thicknesses and cross-sections.

Earth pressure is acting additionally on the semicircular wall. The load ordinate at the bottom of the wall is determined for a gravel backfill as follows:  $q = 16.0 \text{ kN/m}^3 * 4.0 \text{ m} = 64 \text{ kN/m}^2$ .

The roof load (roofing, supporting structure) is assumed with  $1.2 \text{ kN/m}^2$ .

#### Load case 2: live load

The area of the floor surface is used as an assembly room of category A1 bearing an imposed load of  $3.0 \text{ kN/m}^2$ .

In addition, a vertically acting linear load of  $5.0 \text{ kN/m}$  is taken into account around the opening, representing a loading due to a stair access.

#### Load case 3: snow

The snow load is applied according to EN 1993-1-3 for the snow load zone 2 in Germany and for an altitude of 500 m.

#### Load case 4: wind

In our example we analyze wind loading only in direction Y (wind direction: from low to high eaves). It is applied according to EN 1991-1-4 for monopitch roofs and enclosed vertical walls. Furthermore, we apply wind zone 1 and terrain category III for the building. As the roof inclination is higher than  $5^\circ$ , we need to take into account positive and negative external pressure coefficients. In this load case we will assume the positive coefficients.

The reinforced concrete area is exposed to wind loads only partially. For the column at the low eaves side we assume a trapezoidal equivalent load with ordinates  $0.5 \text{ kN/m}$  and  $2.0 \text{ kN/m}$ . For the column at the high eaves side we apply a uniform equivalent load of  $1.5 \text{ kN/m}$ .

#### Load case 5: imperfection

Often imperfections must be considered, for example according to Eurocode 3. Inclinations and precambers are managed in a separate load case. So it is possible to assign specific partial safety factors when you combine this type of load with other actions.

In our example we will analyze imperfections only in direction Y.

For the column cross-sections (HE-A 300) we assume the buckling curve  $b$  (displacement in direction of y-axis) according to EN 1993-1-1, table 6.2. The inclinations  $\varphi_0$  and precambers  $e_{0,d}$  are determined according to EN 1993-1-1, section 5.3.2.

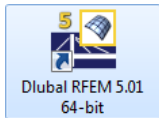
We apply imperfections for both reinforced concrete columns according to EN 1992-1-1, clause 5.2.

## 3. Creating Model

### 3.1 Starting RFEM

To start RFEM in the taskbar, we

click **Start**, point to **All Programs** and **Dlubal**, and then we select **Dlubal RFEM 5.01** or we double-click the icon **Dlubal RFEM 5.01** on the computer desktop.



### 3.2 Creating Model File

The RFEM work window opens showing us the dialog box below. We are asked to enter the basic data for the new model.

If RFEM already displays a model, we close it by clicking **Close** on the **File** menu. Then, we open the *General Data* dialog box by clicking **New** on the **File** menu.

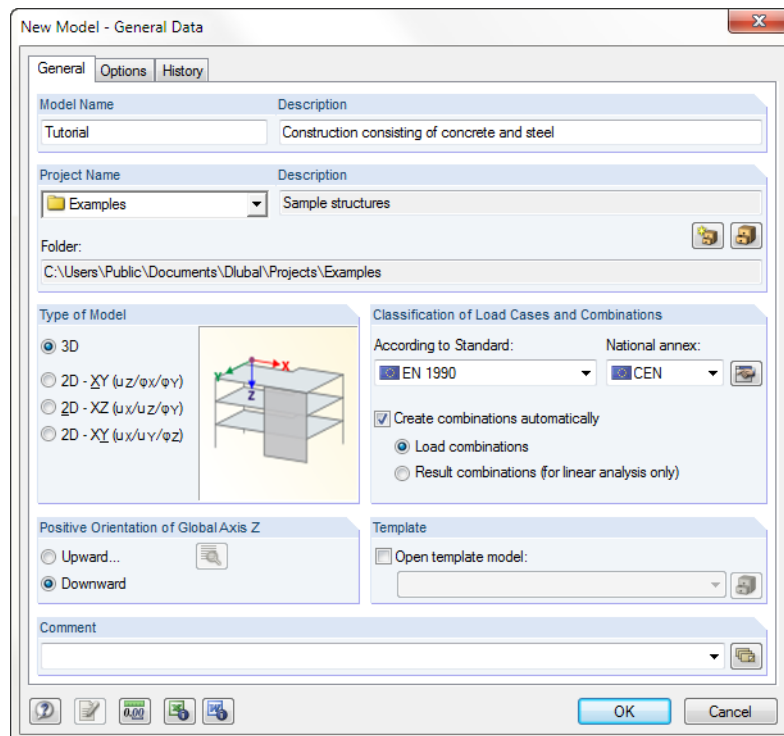
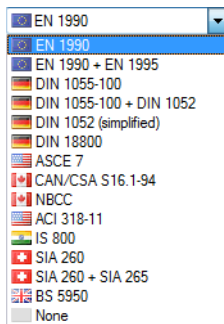


Figure 3.1: Dialog box *New Model - General Data*

We write **Tutorial** into the input field *Model Name*. To the right, we enter **Construction consisting of concrete and steel** as *Description*. We always have to define a *Model Name* because it determines the name of the RFEM file. The *Description* field does not necessarily need to be filled in.

In the input field *Project Name*, we select **Examples** from the list if not already set by default. The project *Description* and the corresponding *Folder* are displayed automatically.

In the dialog section *Type of Model*, the **3D** option is preset. This setting enables a spatial modeling. We also keep the default setting **Downward** for *The Positive Orientation of Global Axis Z*.



The dialog section *Classification of Load Cases and Combinations* requires some settings: we select the entry **EN 1990** from the list *According to Standard*. We don't change the setting **CEN** in the field *National annex* to the right. These specifications are important when we combine actions with partial safety factors and combination coefficients conforming to standards.

Then, we select the check box *Create combinations automatically*. We want to superimpose the actions in **Load combinations**.

Now, the general data for the model is defined. We close the dialog box by clicking the [OK] button.

The empty work window of RFEM is displayed.

## 4. Model Data

### 4.1 Adjusting Work Window and Grid

#### View

First, we click the [Maximize] button on the title bar to enlarge the work window. We see the axes of coordinates with the global directions X, Y and Z displayed in the workspace.

To change the position of the axes of coordinates, we click the button [Move, Zoom, Rotate] in the toolbar above. The pointer turns into a hand. Now, we can position the workspace according to our preferences by moving the pointer and holding the left mouse button down.

Furthermore, we can use the hand to zoom or rotate the view:

- Zoom: We move the pointer and hold the [Shift] key down.
- Rotation: We move the pointer and hold the [Ctrl] key down.

To exit the function, different ways are possible:

- We click the button once again.
- We press the [Esc] key on the keyboard.
- We right-click into the workspace.

#### Mouse functions

The mouse functions follow the general standards for Windows applications. To select an object for further editing, we click it once with the **left** mouse button. We double-click the object when we want to open its dialog box for editing.

When we click an object with the **right** mouse button, its shortcut menu appears showing us object-related commands and functions.

To change the size of the displayed model, we use the **wheel button** of the mouse. By holding down the wheel button we can shift the model directly. When we press the [Ctrl] key additionally, we can rotate the structure. Rotating the structure is also possible by using the wheel button and holding down the right mouse button at the same time. The pointer symbols shown on the left show the selected function.



**Grid**

The grid forms the background of the workspace. In the dialog box *Work Plane and Grid/Snap*, we can adjust the spacing of grid points. To open the dialog box, we use the button [Settings of Work Plane].

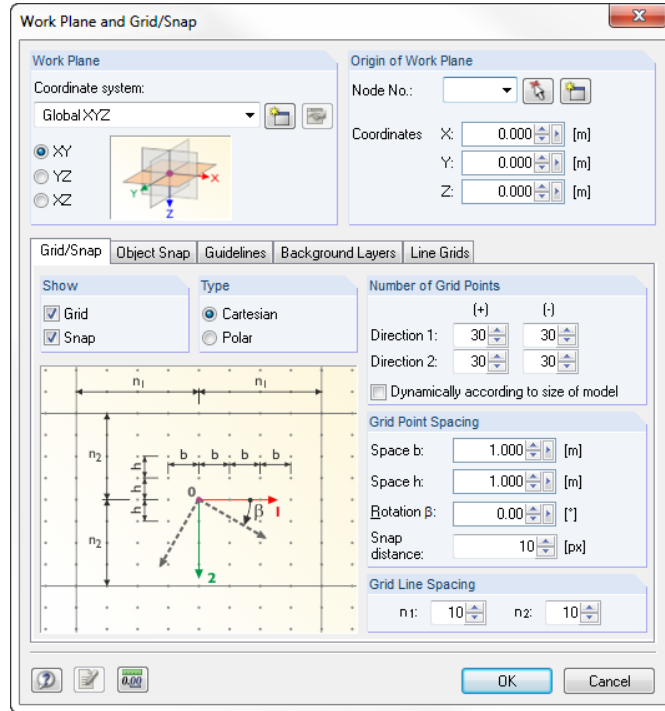


Figure 4.1: Dialog box *Work Plane and Grid/Snap*

SNAP GRID

Later, for entering data in grid points, it is important that the control fields *SNAP* and *GRID* in the status bar are set active. In this way, the grid becomes visible and the points will be snapped on the grid when clicking.

**Work plane**

The XY plane is set as work plane by default. With this setting all graphically entered objects will be generated in the horizontal plane. The plane has no significance for the data input in dialog boxes or tables.

The default settings are appropriate for our example. We close the dialog box with the [OK] button and start with the model input.

## 4.2 Creating Surfaces

It would be possible to define corner nodes in the graphic or table first to connect them with lines which we can use to create surfaces. But in our example we use the direct graphical input of lines and surfaces.

### 4.2.1 Floor

The floor surface consists of a rectangular and a semicircular surface.

#### 4.2.1.1 Defining Rectangular Surface

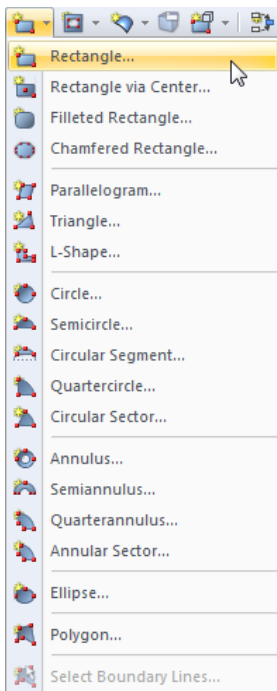
Rectangular slabs are frequently used structural components. To create rectangular plates quickly,

we click **Model Data** on the **Insert** menu, then we point to **Surfaces, Plane** and **Graphically** and select **Rectangle**,

or we use the corresponding list button for the selection of plane surfaces. We click the arrow button [▼] to open a pull-down menu offering a large selection of surface geometries.

With the menu item [Rectangle] we can define the plate directly. Related nodes and lines will be created automatically.

After selecting this function, the dialog box *New Rectangular Surface* opens.



List button for plane surfaces

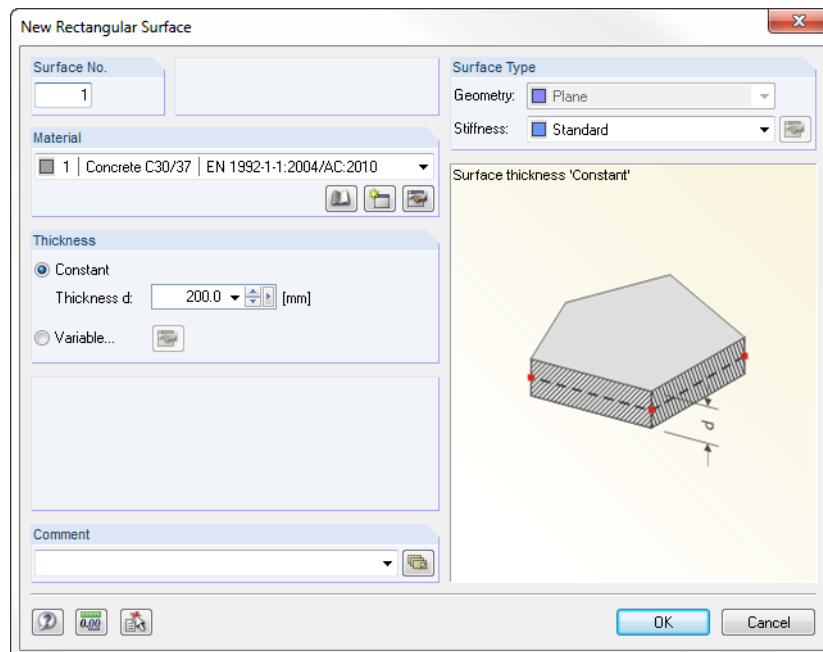


Figure 4.2: Dialog box *New Rectangular Surface*

The *Surface No.* of the new rectangular plate is specified with 1. It is not necessary to change this number.

The *Material* is preset with *Concrete C30/37* according to EN 1992-1-1. When we want to use a different material, we can select another one by means of the [Material Library] button.

The *Thickness* of the surface is *Constant*. We increase the value  $d$  to **200** mm, either by using the spin box or by direct input.

In the dialog section *Surface Type* the *Stiffness* is preset appropriately with *Standard*.

We close the dialog box with the [OK] button and start the graphical input of the slab.





We can make the surface definition easier when we set the view in Z-direction (top view) by using the button shown on the left. The input mode will not be affected.

To define the first corner, we click with the left mouse button into the **coordinate origin** (coordinates X/Y/Z **0.000/0.000/0.000**). The current pointer coordinates are displayed next to the reticle.

Then, we define the opposite corner of the slab by clicking the grid point with the X/Y/Z coordinates **7.000/6.000/0.000**.

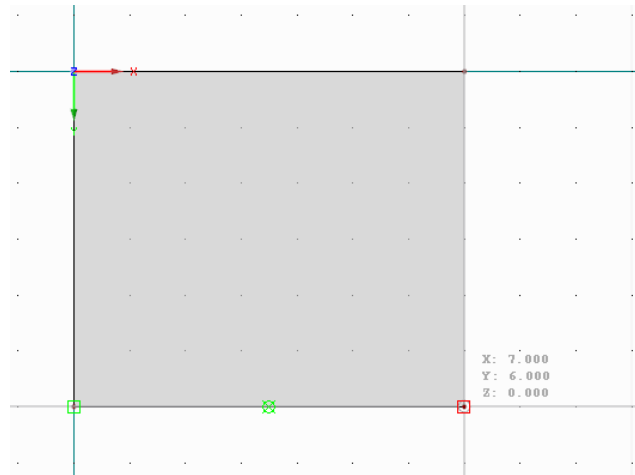


Figure 4.3: Rectangular surface 1

RFEM creates four nodes, four lines and one surface.

As we don't want to create any more plates, we quit the input mode by pressing the [Esc] key. We can also use the right mouse button to right-click in an empty area of the work window.



### Show numbering

If we want to display the numbering of nodes, lines and surfaces, we right-click into an empty space of the work window. A shortcut menu with useful functions appears. We activate the *Numbering*.

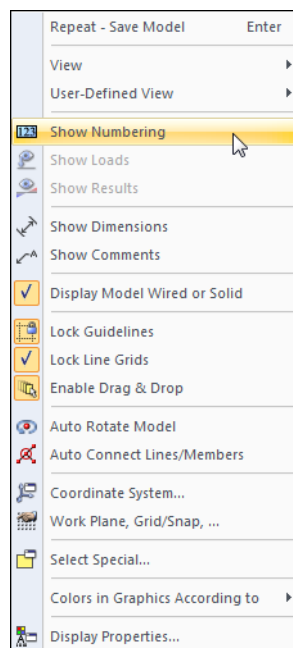


Figure 4.4: Show numbering in shortcut menu

We can use the *Display* tab in the navigator to control the numbering of objects in detail.

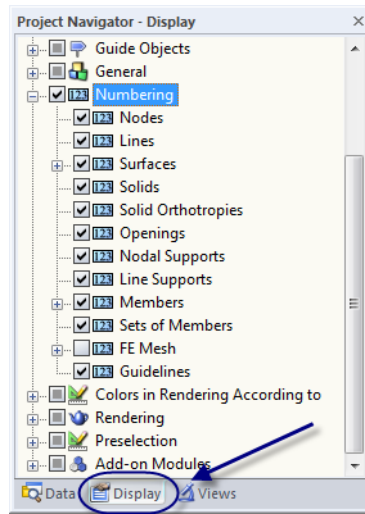


Figure 4.5: *Display* navigator for numbering

### 4.2.1.2 Creating Arc

Now, we need to define an area that is limited by a circular arc.



We click the arrow [▼] of the list button for lines available in the toolbar to open the pull-down menu offering tools for special line types. We select the entry *Arc via Three Nodes*.

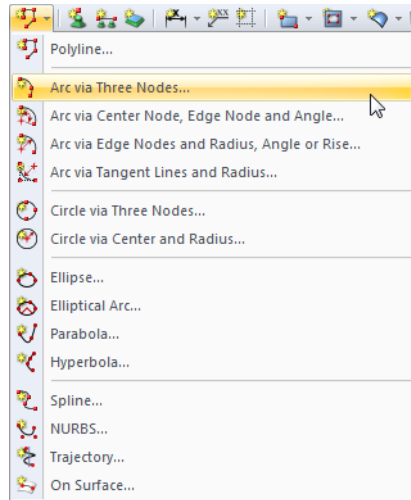


Figure 4.6: List button for lines - *Arc via Three Nodes*

In the work window, we click the following nodes one after the other: node **4**, the grid point with the coordinates **10.000/3.000/0.000** and node **3**. After clicking the last node, the arc will be created as line 5.

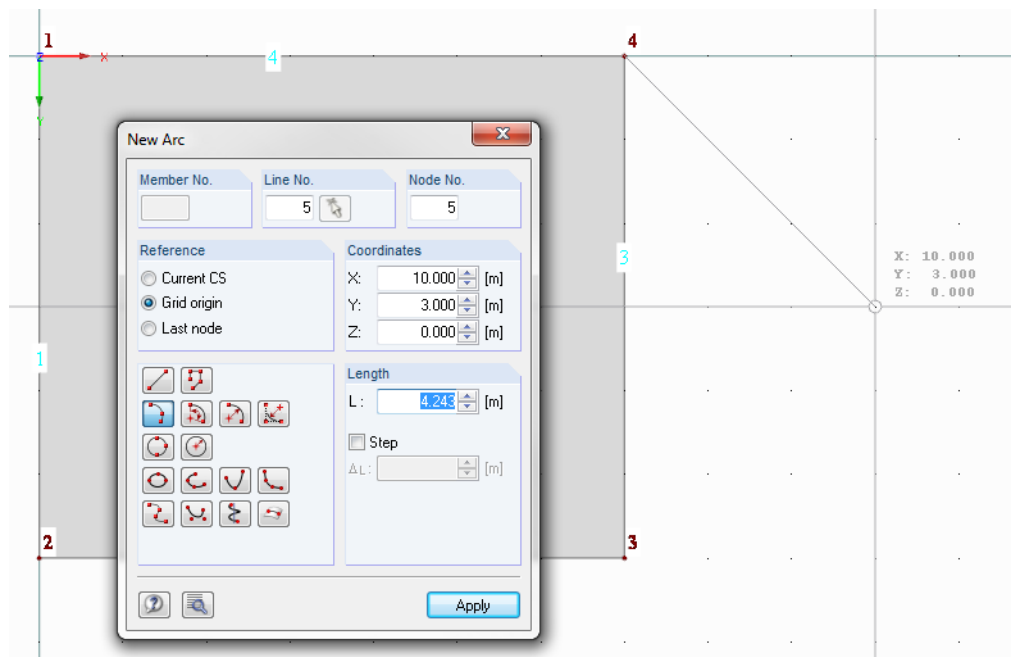


Figure 4.7: Defining a circular arc with three points

To quit the input mode, use the [Esc] button.

### 4.2.1.3 Adjusting Floor Surface

As the demo version allows for the definition of only two surfaces, we cannot define the semi-circular surface as a new surface. Therefore, we extend the rectangular surface to a general plane surface enclosing the arc area.

We double-click surface 1 in the work window to open the dialog box *Edit Surface*.

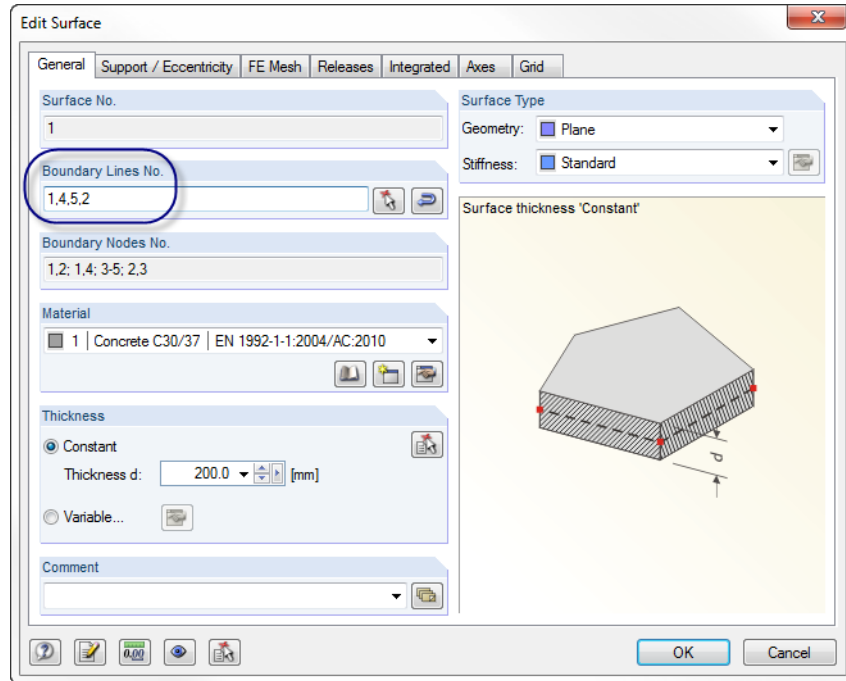


Figure 4.8: Modifying boundary lines

We have two input options:

- In the input field *Boundary Lines No.*, we can enter the numbers of the new border lines **1**, **2**, **4** and **5** manually.
- We can use the [Select] button shown on the left to select the new boundary lines graphically in the work window. But first it is necessary to empty the preset list in the selection window *Edit Surface* by clicking the button [Clear].

Now, the floor surface looks like the figure below.

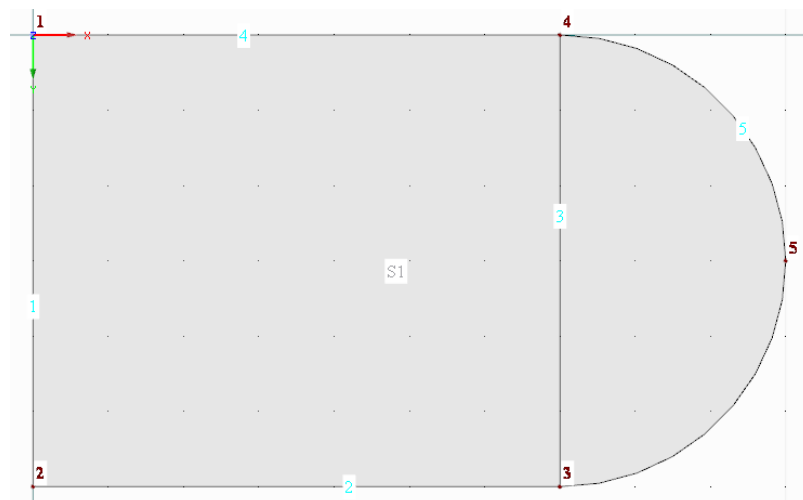
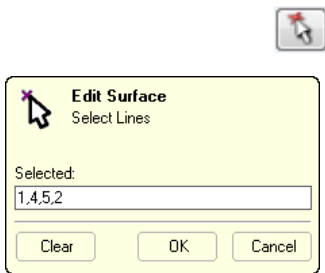


Figure 4.9: Floor slab

## 4.2.2 Wall

### Copy the arc

The most comfortable way to create a curved surface is copying the circular arc by specifying particular settings for the copy process.

We click arc line 5 with the left mouse button (single click) to select it. The line is now displayed with a different color. Yellow is preset as selection color for black backgrounds.

We use the toolbar button shown on the left to open the dialog box *Move or Copy*.

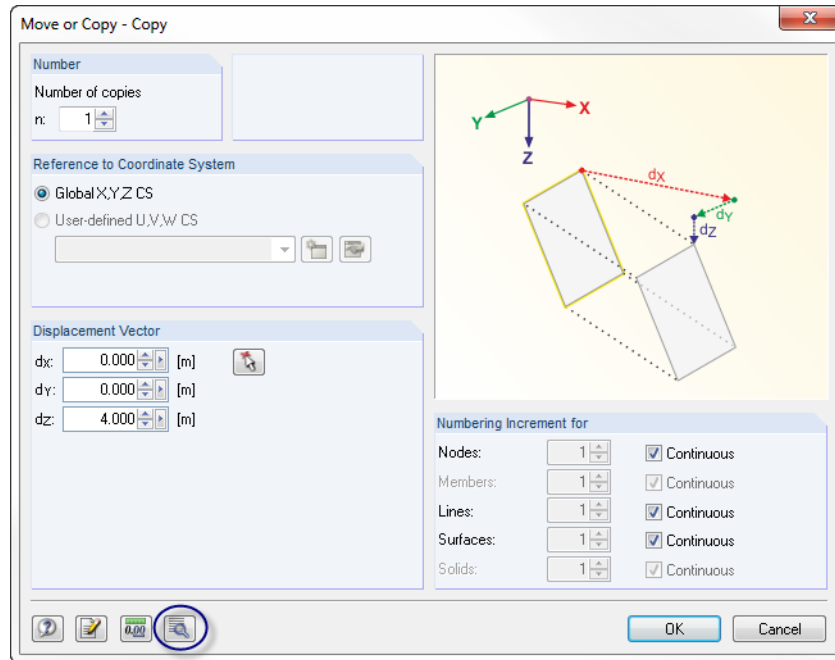


Figure 4.10: Dialog box *Move or Copy*

We increase the *Number of copies* to **1**: With this setting the arc won't be moved but copied. As the wall is 4 m high (system line), we enter the value **4.0** m for the *Displacement Vector* in  $d_z$ .

Now, we click the [Details] button to specify more settings.



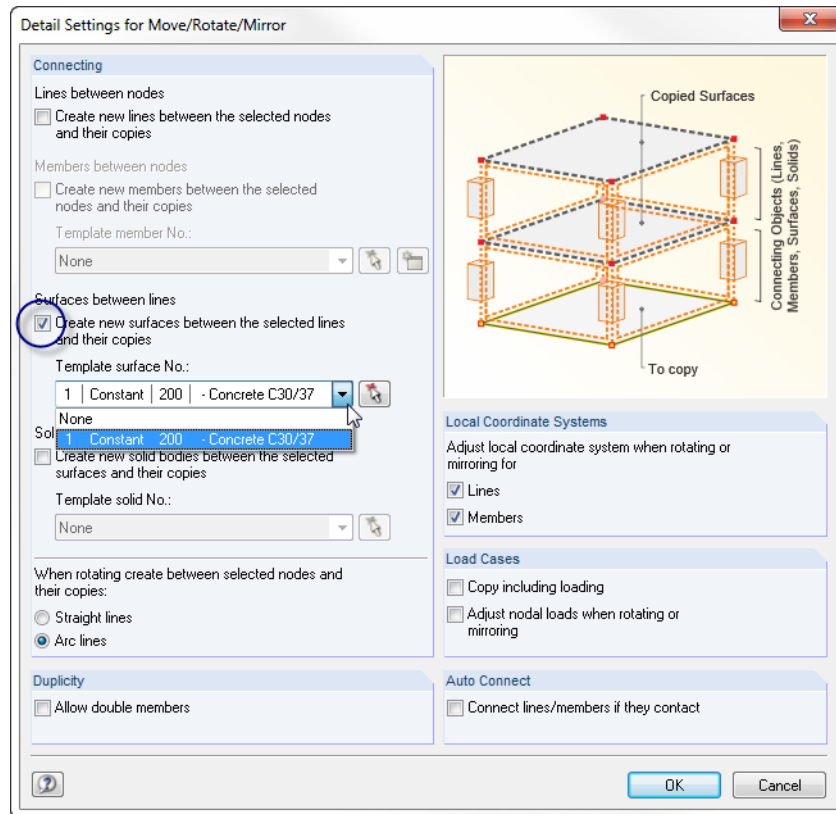


Figure 4.11: Dialog box *Detail Settings for Move/Rotate/Mirror*

In the dialog section *Connecting*, we select the check box of the following option:

- Create new surfaces between the selected lines and their copies

Then, we select surface **1** from the list to define it as *Template surface*. In this way, the properties of the floor slab (material, thickness) are preset for the new wall surface.

We close both dialog boxes by clicking the [OK] button.

Set the isometric view

We use the toolbar button shown on the left to set the [Isometric View] because we want to display the model in a graphical 3D representation.

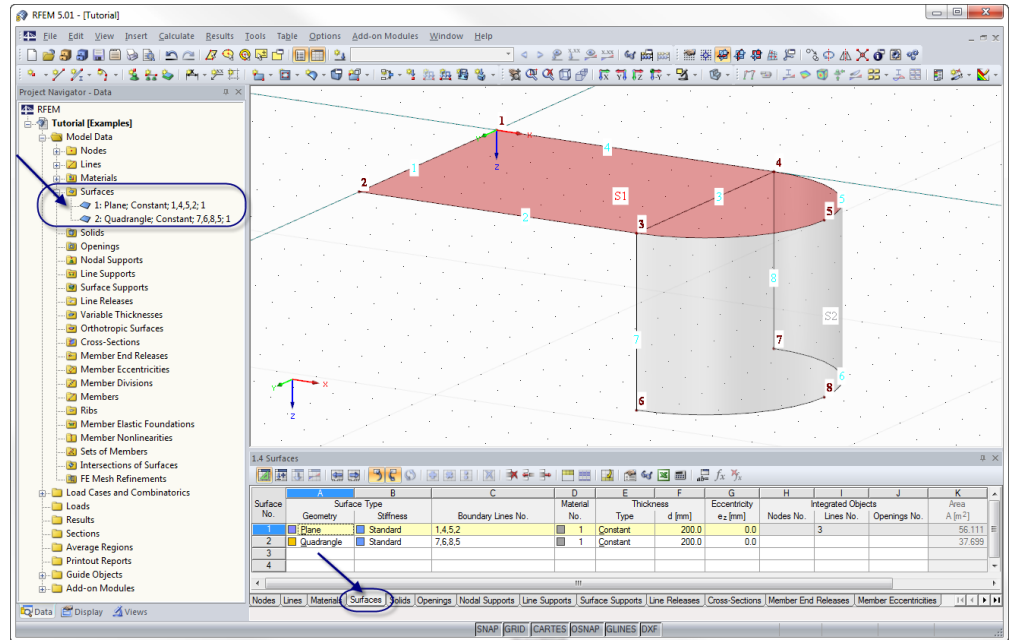


Figure 4.12: Model in isometric view with navigator and table entries

Checking data in navigator and tables

All entered objects can be found in the directory tree of the *Data* navigator and in the tabs of the table. The entries in the navigator can be opened by clicking the [+]  
sign (like in Windows Explorer). To switch between the tables, we click the individual table tabs.

In the navigator entry *Surfaces* and in table 1.4 *Surfaces*, we see the input data of both surfaces in numerical form (see figure above). RFEM created the wall as a quadrangle surface, i.e. a shell that is limited by four lines.

### 4.2.3 Opening

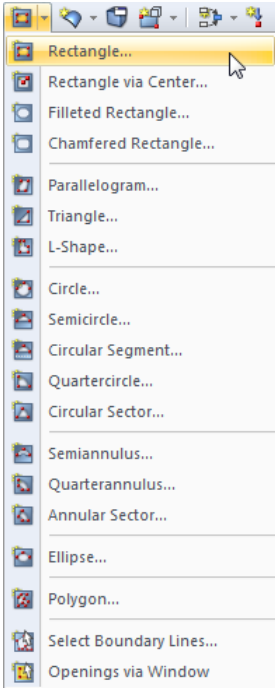
#### 4.2.3.1 Creating Opening

Now, we insert a rectangular opening into the floor slab. The data input is easier when we reset the [View in Z direction].

We can apply the opening directly, which means without defining lines first. We use the list button for openings available in the toolbar and select the entry *Rectangle*.

We set the first opening node in grid point **3.000/1.000/0.000**. The second node is defined in grid point **5.000/2.000/0.000**.

The opening is too small. We will adjust its length in the next step.



List button for openings

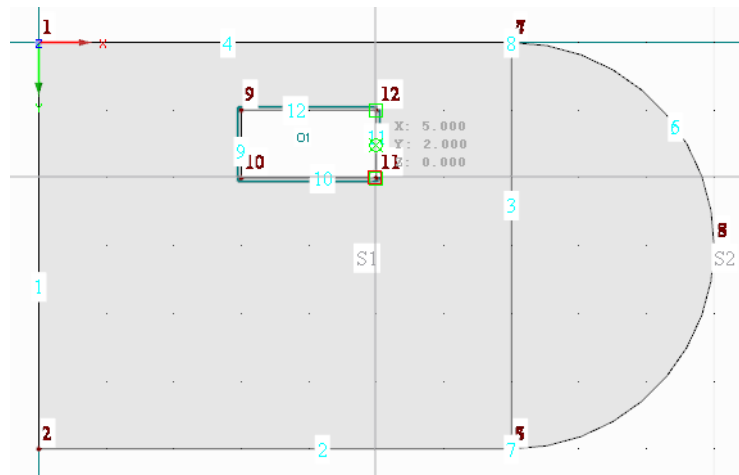


Figure 4.13: Defining the rectangular opening

We close the input mode with the [Esc] button or with a right-click into the empty workspace.

### 4.2.3.2 Adjusting Opening

The length of the opening is 2.50 m. We select nodes 11 and 12 one after the other by holding down the [Ctrl] key when clicking.

We open the dialog box *Edit Node* with a double click on one of these nodes.

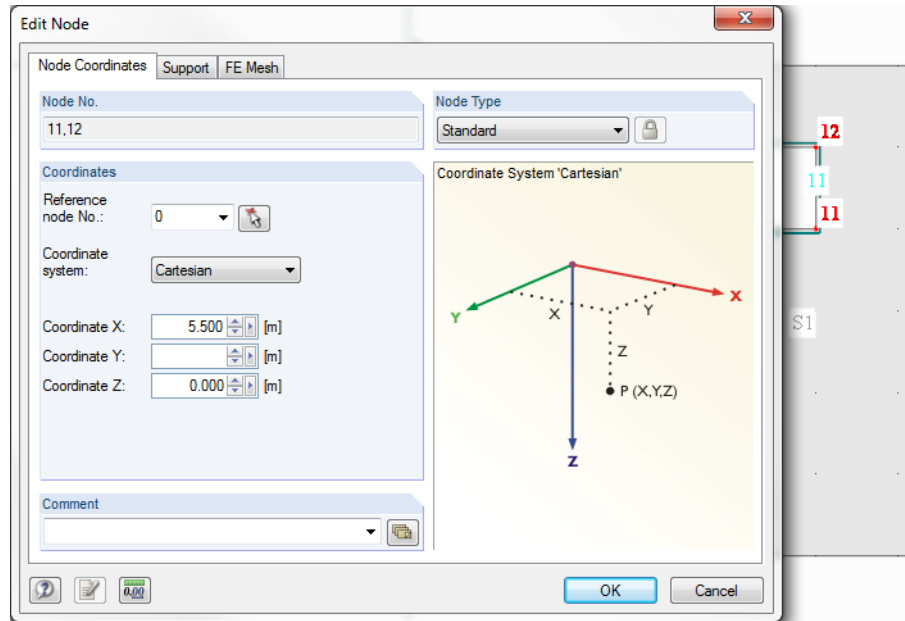
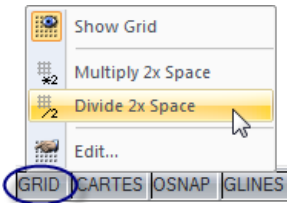


Figure 4.14: Dialog box *Edit Node*

Both nodes are listed in the input field *Node No.* We correct the *Coordinate X* by entering **5.500** m, and then we confirm the input with [OK]. Now, the opening has an appropriate length.

Alternative: It is also possible to apply the opening instantly without modifying coordinates when we use an adjusted grid: Before, we have to open the dialog box *Work Plane and Grid/Snap* (see Figure 4.1, page 10) where we reduce the *Grid Point Spacing* to 50 cm. We can also use the shortcut menu of the GRID button available in the status bar to modify the grid spacing quickly. Just right-click the button shown on the left.



Now, the input of surfaces is complete.

## 4.3 Creating Concrete Members

### 4.3.1 Columns

Member elements depend on lines: By creating a member a line is generated automatically.

#### Changing the work plane

We want to define the columns graphically, so we need to shift the work plane from the horizontal to the vertical plane. To set the [Work Plane YZ], we click the second of the three plane buttons in the toolbar.



We reset the [Isometric View]. Now, we can see that the input grid is spanned in the plane of the two columns (see Figure 4.18).

#### Definition of cross-section

We click the list button [New Members Single] to open the dialog box *New Member*.

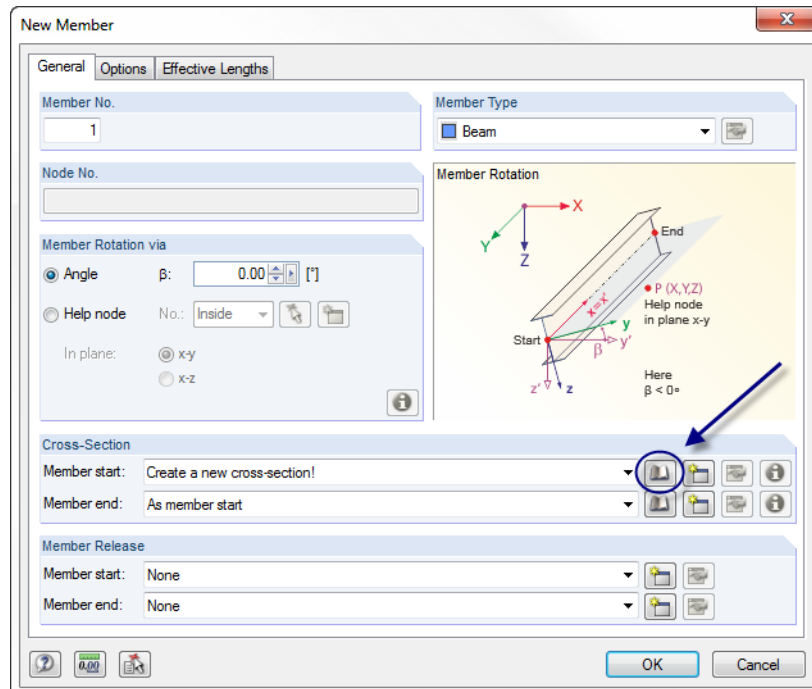
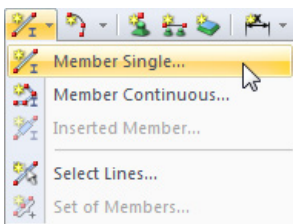


Figure 4.15: Dialog box *New Member*

It is not necessary to change the default settings. We only have to create a *Cross-Section*. To define the cross-section at the *Member start*, we click the [Library] button. The cross-section data-base appears.

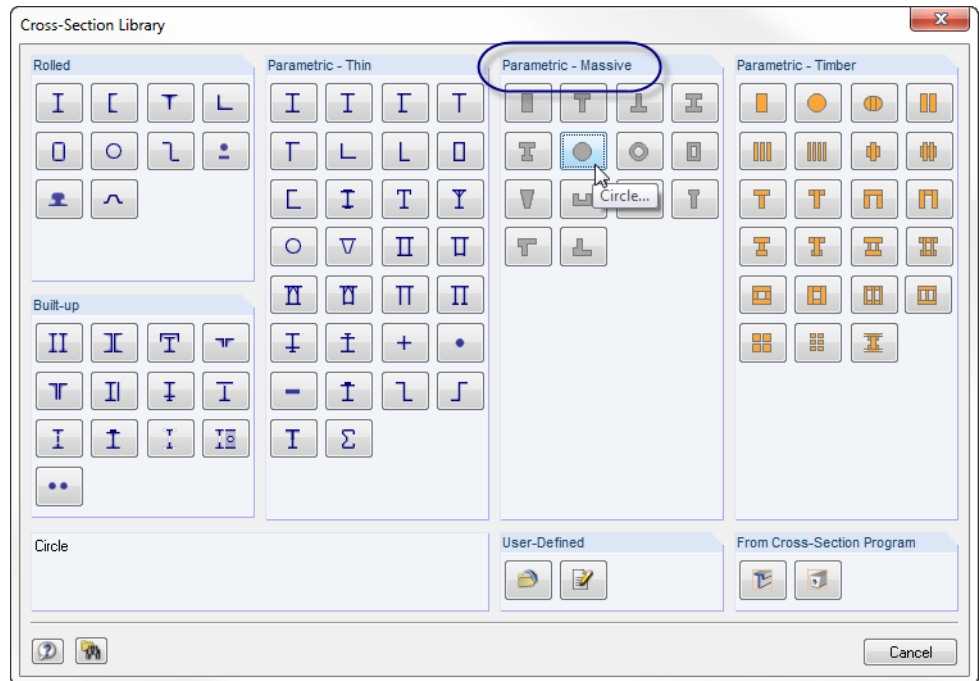


Figure 4.16: Cross-Section Library

In the dialog section *Parametric - Massive*, we select the cross-section type *Circle*. Another dialog box appears.

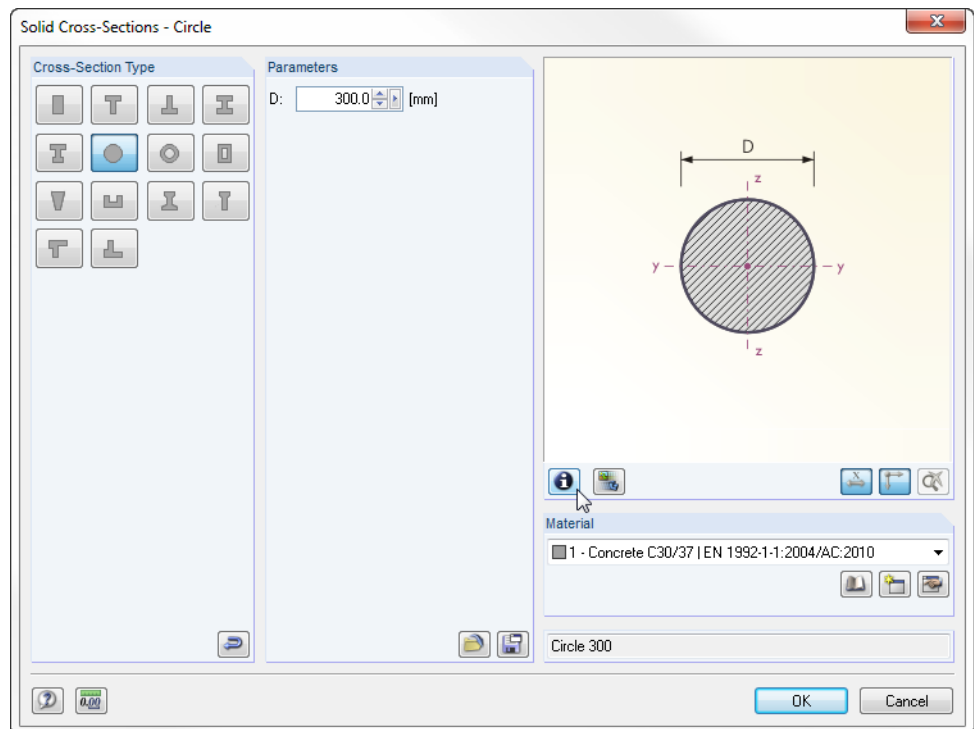


Figure 4.17: Dialog box *Solid Cross-Sections - Circle*

We define the column diameter  $D$  with **300** mm.

For solid cross-sections RFEM presets number *1 - Concrete C30/37* as *Material*.

We can use the [Info] button to check the properties of the cross-section.



We click [OK] to import the cross-section values and return to the initial *New Member* dialog box. Now the input field *Member start* shows the new cross-section. We close the dialog box with [OK] to set the columns graphically.

### Graphical definition of members

We define the footing of the front column by clicking the grid point **0.000/6.000/4.000**.

The top end of the column is set into node **2**.

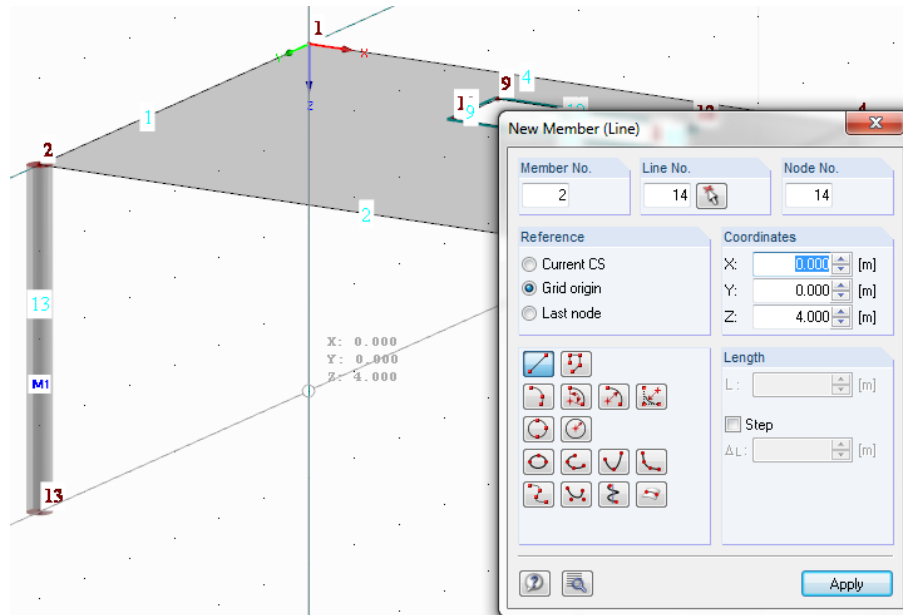


Figure 4.18: Defining the footing of the second column

The input command *Define member* is still active. Therefore, we can continue with the definition of the rear column.

We place the footing of the second column into the grid point **0.000/0.000/4.000**. The column's top end is then defined in the grid zero point which is node **1**.

To quit the input mode, we use the [Esc] key. We can also right-click into the empty work window.

### 4.3.2 Rib

In the next step, we enter the downstand beam below the ceiling.

#### Modification of line properties

We double-click line 3 to open the dialog box *Edit Line*. We change to the second tab *Member* where we select the check box of the option *Available* (see Figure 4.19).

The dialog box *New Member* opens again.

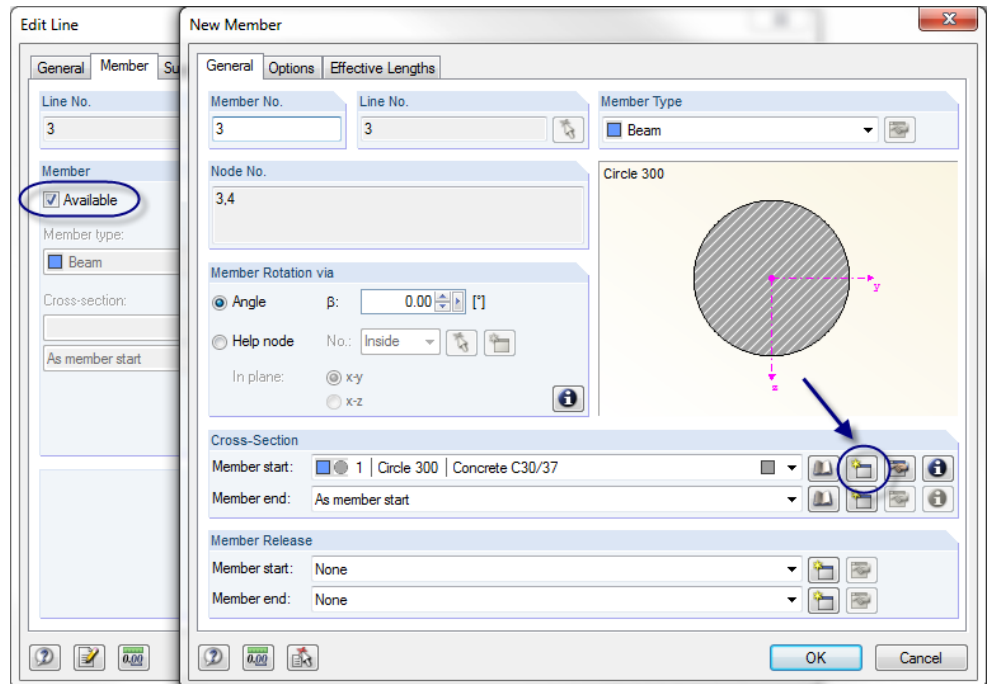


Figure 4.19: Dialog box *New Member*



This time we click the [New] button to define a cross-section at the *Member start*.

In the upper part of the dialog box *New Cross-Section*, we select the massive *REC* cross-section table. The dialog box *Solid Cross-Sections - Rectangle* opens where we define the width *b* with **250 mm** and the depth *h* with **400 mm**.

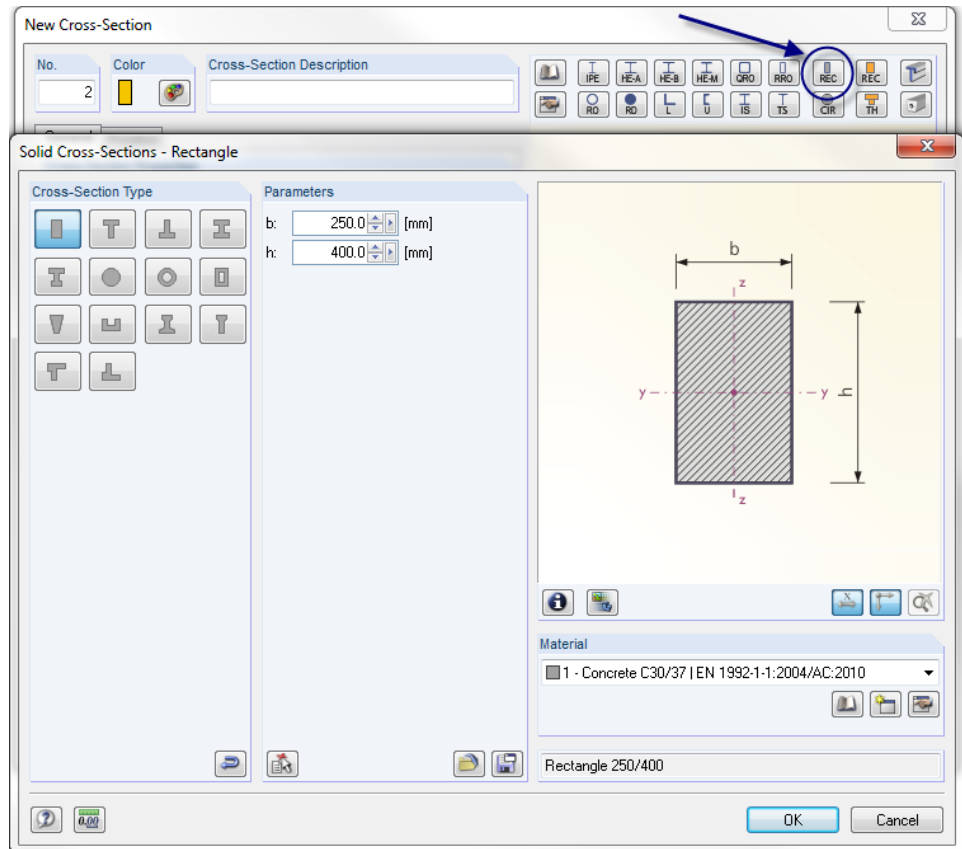


Figure 4.20: Dialog box *Solid Cross-Sections - Rectangle*

We click [OK] to import the cross-section values to the dialog box *New Cross-Section*. Again, material number *1 - Concrete 30/37* is preset.

We click [OK] and return to the initial dialog box *New Member*. Now the input field *Member start* shows the rectangular cross-section.

### Definition of rib

In RFEM we can model a downstand beam with the member type *Rib*. We just change the *Member Type* in the dialog box *New Member*. We select the entry *Rib* from the list.

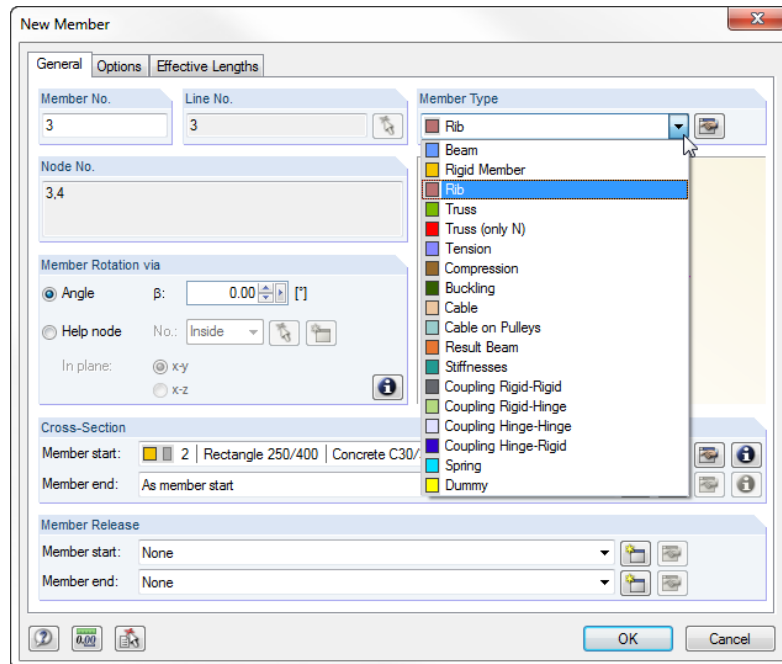


Figure 4.21: Changing the member type

With a click on the button [Edit Member Type] we open the dialog box *New Rib*.

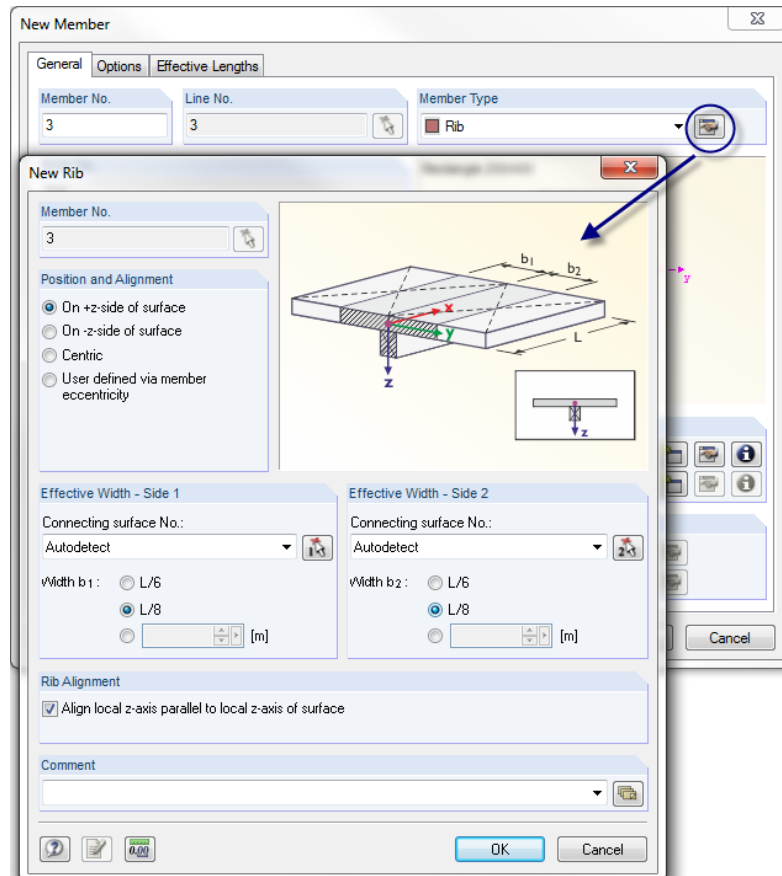


Figure 4.22: Defining the rib

We define the *Position and Alignment* of the Rib **On +z-side of surface**. This is the bottom side of the floor slab.

For the *Effective Width* we specify **L/8** for both sides. RFEM will find the surfaces automatically.

We close all dialog boxes with the [OK] button and check the result in the work window.

### Display options

RFEM displays the rib as a member that is eccentrically arranged. As the transparent rendering model does not show surface thicknesses, we set the *Solid Display Model* by means of the list button shown on the left. This display mode helps us to check the placement of the rib.

In addition, we set the rendering option *Filled incl. thickness* available in the *Display* navigator.

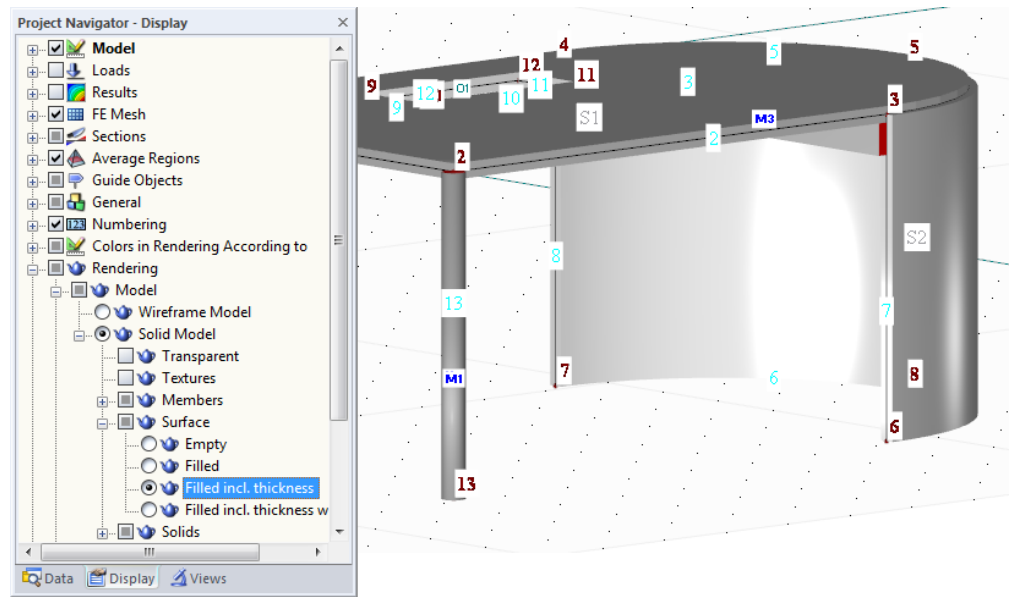
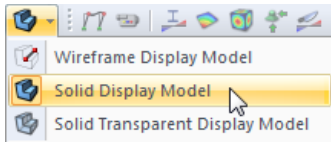
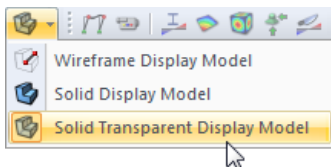


Figure 4.23: Graphical representation of solid model



To adjust the display, we use the button [Move, Zoom, Rotate] (see "mouse functions" on page 9). The pointer turns into a hand. When we hold down the [Ctrl] key additionally, we can rotate the model by moving the pointer.



For the following input we reset the mode of the *Solid Transparent Display Model*. We also reset the rendering for surfaces to *Filled* provided in the *Display* navigator in order to hide thicknesses.

## 4.4 Defining Supports

The model is still without supports. In RFEM we can assign supports to nodes, lines, members and surfaces.

### 4.4.1 Nodal Supports

The columns are supported on their footing in all directions but without restraint.



We select the bottom nodes of the columns by drawing a window across the area including the nodes X and Y. Then, we click the toolbar button [New Nodal Support] to open the dialog box *New Nodal Support*.

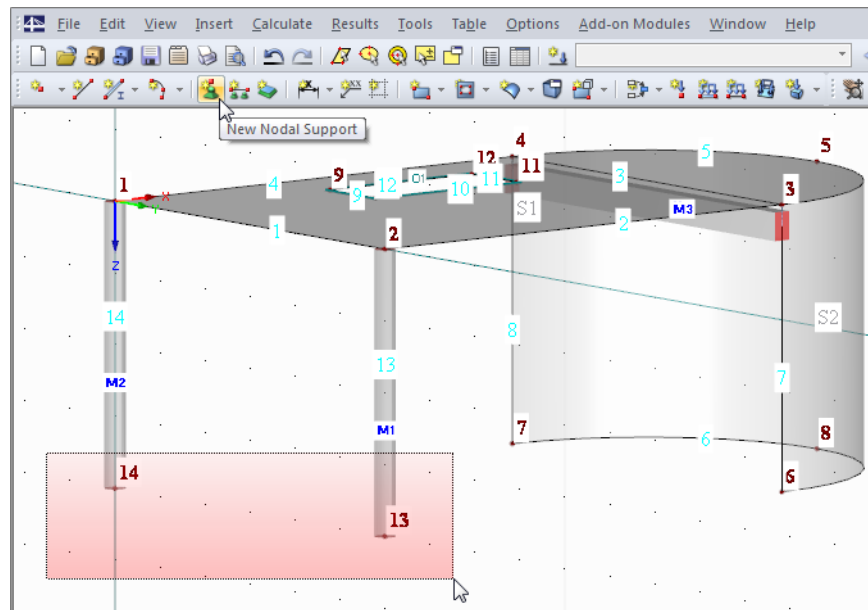


Figure 4.24: Selecting column nodes with window

Both node numbers 13 and 14 are shown in the field *Node No.*



We modify the type of the support because the preset support type 1 results in a restraint about the longitudinal member axis. With a click on the button [New] (see Figure 4.25) we open another dialog box.

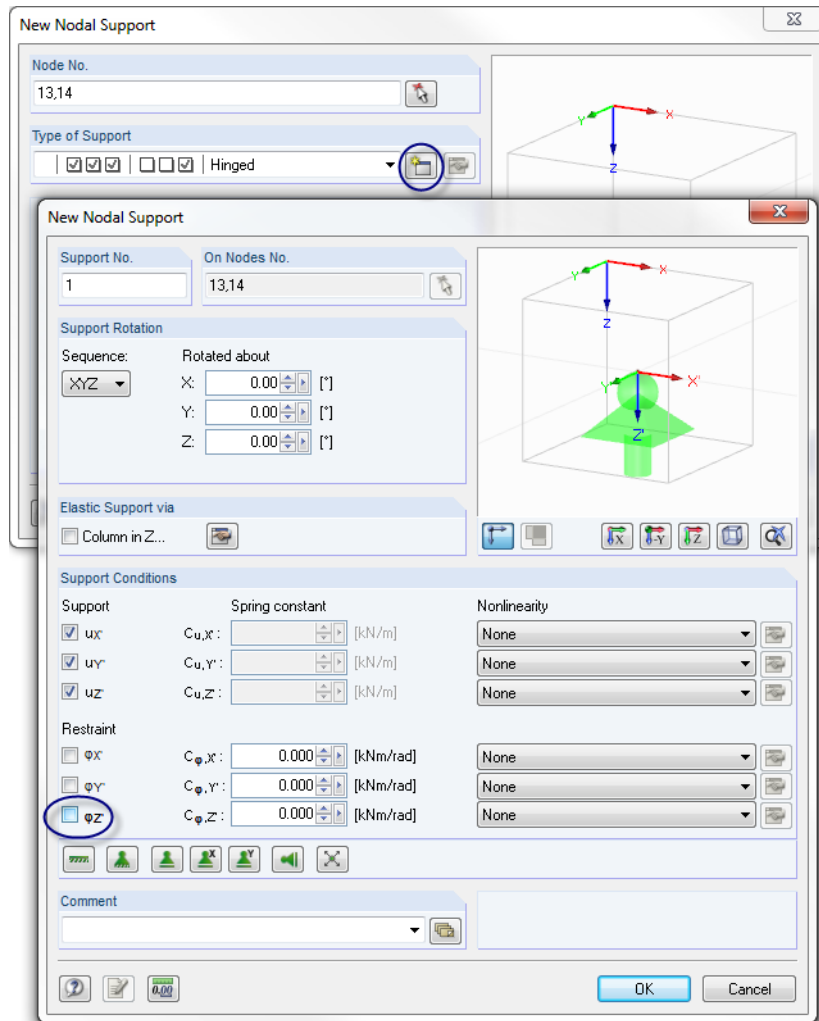


Figure 4.25: Defining degrees of freedom

In the second dialog box *New Nodal Support*, we remove the check mark of the *Restraint* for rotation  $\phi_z$ .

We confirm the dialog boxes with [OK]. Now, we see support symbols displayed on the model.

### 4.4.2 Line Supports

The bottom line of the curved wall is supported as well. This time, we choose another way to enter support data. First, we define the support properties. Then, we assign them to the object graphically.



With a click on the button [New Line Support] we open the dialog box *New Line Support*.

The option *Hinged* is preset as *Type of Support*. The first three selected check boxes are indicating that a support is available in the directions X, Y and Z. The final three boxes are not selected because the hinged support type has no restraint about X, Y and Z.

We confirm the dialog box with [OK] because the hinged support is adequate for our example.

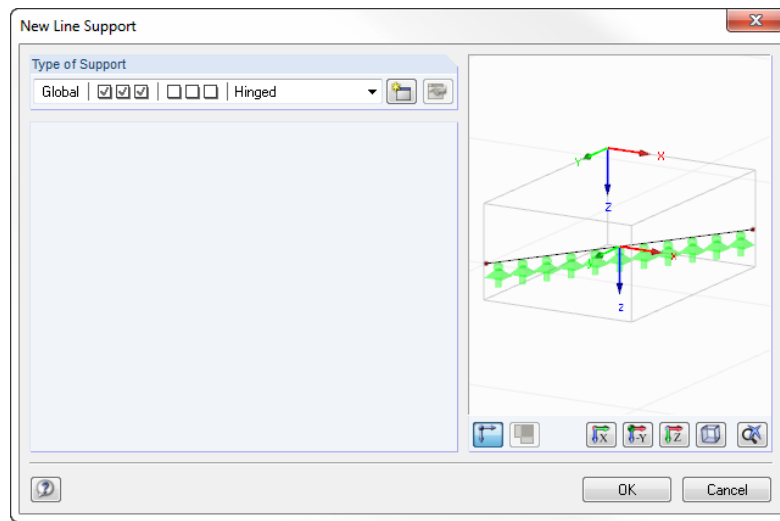


Figure 4.26: Dialog box *New Line Support*



RFEM displays a support symbol next to the mouse pointer. It becomes a reticle as soon as we approach a line. The number of the corresponding line is displayed in the status bar. We set the support on the curved line 6.

Now, the input of the model's reinforced concrete construction is complete.

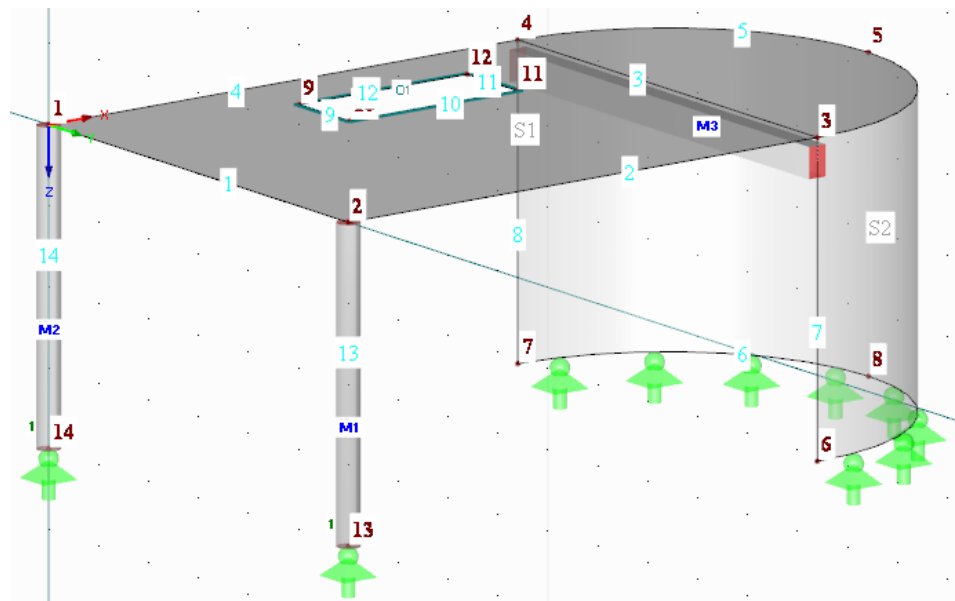


Figure 4.27: Reinforced concrete structure with supports

## 4.5 Creating Steel Members

### 4.5.1 Frame

Now we enter the steel construction. First, we define the frame that lies in the plane of the two columns. It is helpful to use a so-called visibility for this plane: This type of partial view allows us to work in a particular zone of the model and we are not disturbed by objects that lie within another plane.

#### Creation of visibilities

We set the *Views* tab in the navigator. A number of visibilities is already available. They were *Generated* by RFEM based on the data we entered.



The button [Visibility by Window] makes it possible to abstract a specific zone from the model graphically: We activate the function and draw a window from the left to the right, completely enclosing both column members.



Please note: If you pull up the window from the left to the right, the visibility contains only objects that are completely within this window. If you pull up the window from the right to the left, the visibility additionally contains those objects that are cut by the window.

Now we can see that the rest of the model (floor, wall) is displayed with a lower color intensity. The corresponding objects cannot be edited.

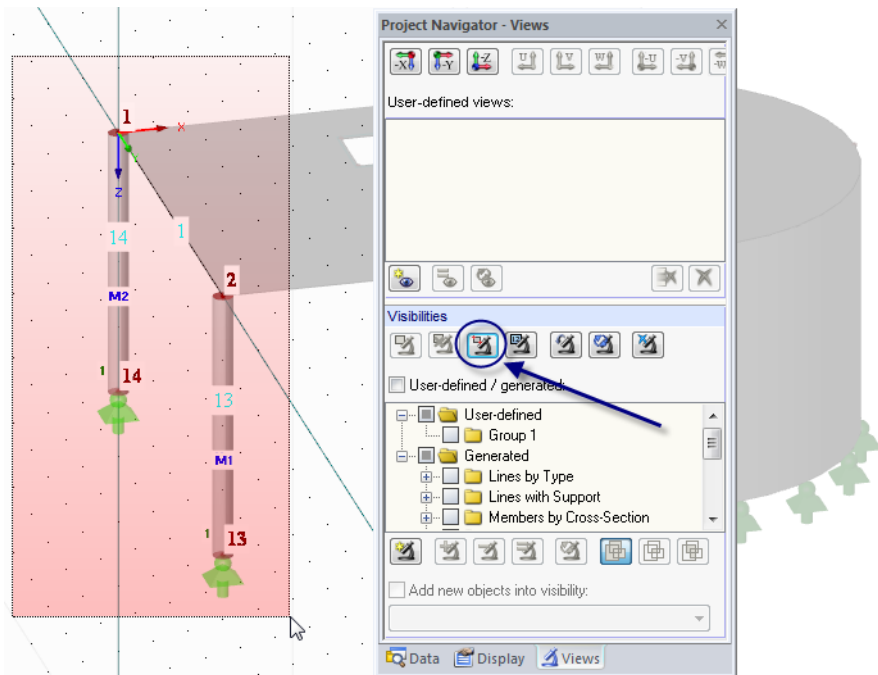
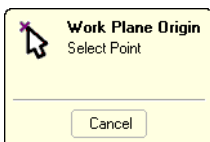


Figure 4.28: Creating a visibility by window

#### Changing the origin of the work plane

Plane YZ is still set as the work plane, which is appropriate for defining the frame of the mono-pitch roof. Also the origin of the work plane is convenient for our purpose. However, we want to show you how you can adjust the work plane. Therefore, we modify the position of the work plane origin.

We click the toolbar button [Set Origin]. Then, in the work window, we select node 2, which is the head of the front column, to be the new origin of the work plane. The reticle of the grid changes its position.



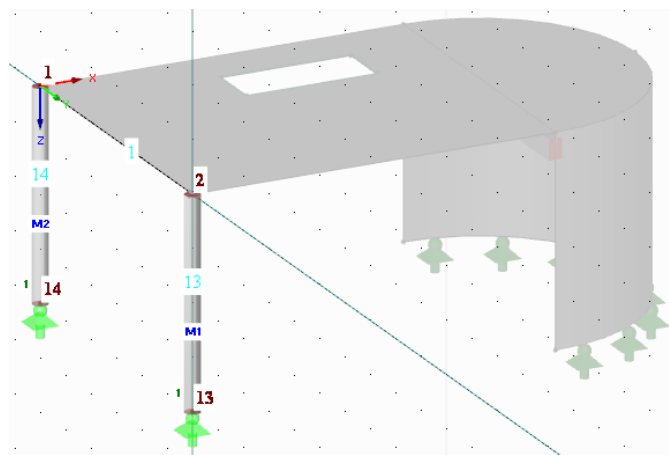


Figure 4.29: New origin in node 2

### 4.5.1.1 Defining Members Continuously

We want to create the monopitch roof frame as a polygonal chain. We open the list button [New Member] and select the menu item *Member Continuous*.

The dialog box *New Member* opens. First, we change the *Member Type* to **Beam**.

As shown in Figure 4.19 on page 24, we create a cross-section for the *Member start* by using the [New] button. The dialog box *New Cross-Section* opens where we click the button [HE-A] at its top. Then, in the dialog box *Rolled Cross-Sections - I-sections*, we select the cross-section **HE A 300** from the HE A cross-section table. For rolled cross-sections RFEM presets number **2 - Steel S 235** as *Material*.

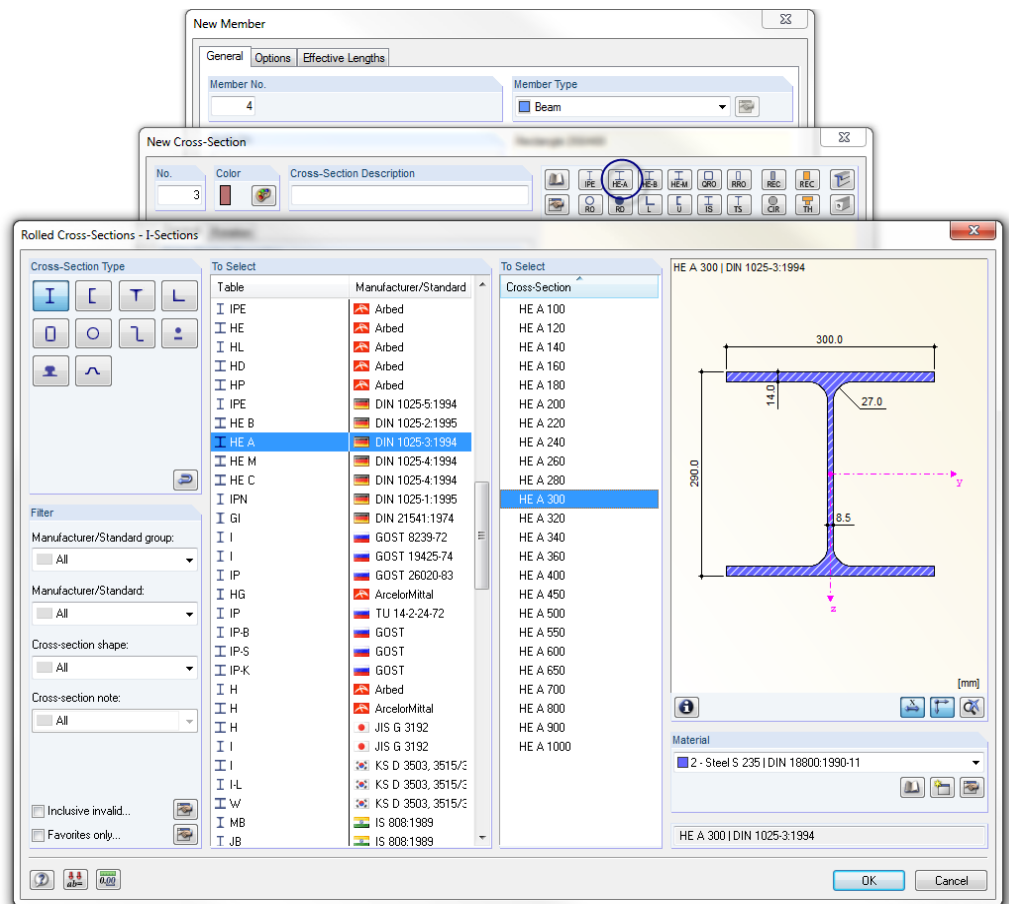
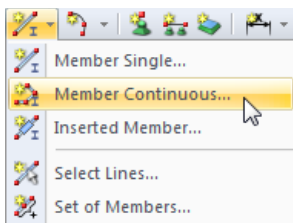


Figure 4.30: Selecting HE A 300 in the library

We confirm all dialog boxes with [OK].

Back in the work window, we define the three frame members in one go by clicking the following nodes and grid points:

- Node 1
- Grid point **0.000/-6.000/-3.000** (grid origin has been modified)
- Grid point **0.000/0.000/-3.000** (roof inclination will be adjusted later)
- Node 2

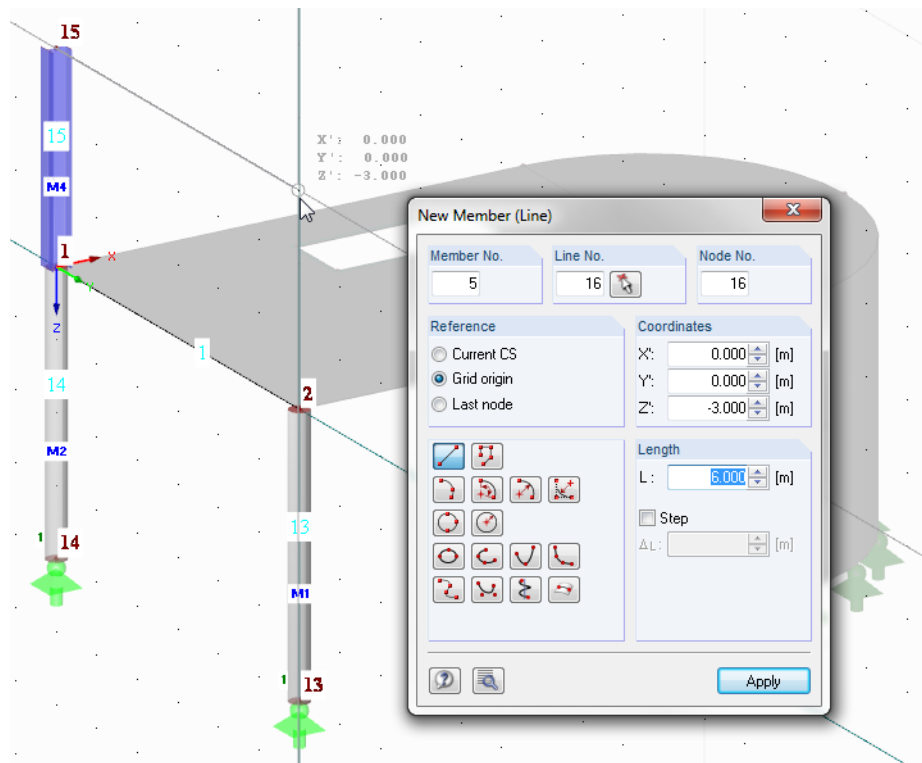


Figure 4.31: Defining members continuously

When the last node is defined, we right-click twice into the empty work window to quit the input mode.

Both columns in our model are connected rigidly to the floor nodes 1 and 2. Though this kind of restraint can hardly be built in reality, we do without a modeling of release properties in our example and accept this simplification.

### 4.5.1.2 Shearing Horizontal Beams

The monopitch roof has an inclination of  $8^\circ$ . That's why we have to adjust the horizontal beam. We draw a selection window across member 5 comprising both end nodes. Then, in the toolbar we select **Shear** on the **Edit** menu to open the dialog box *3D Shearing*.

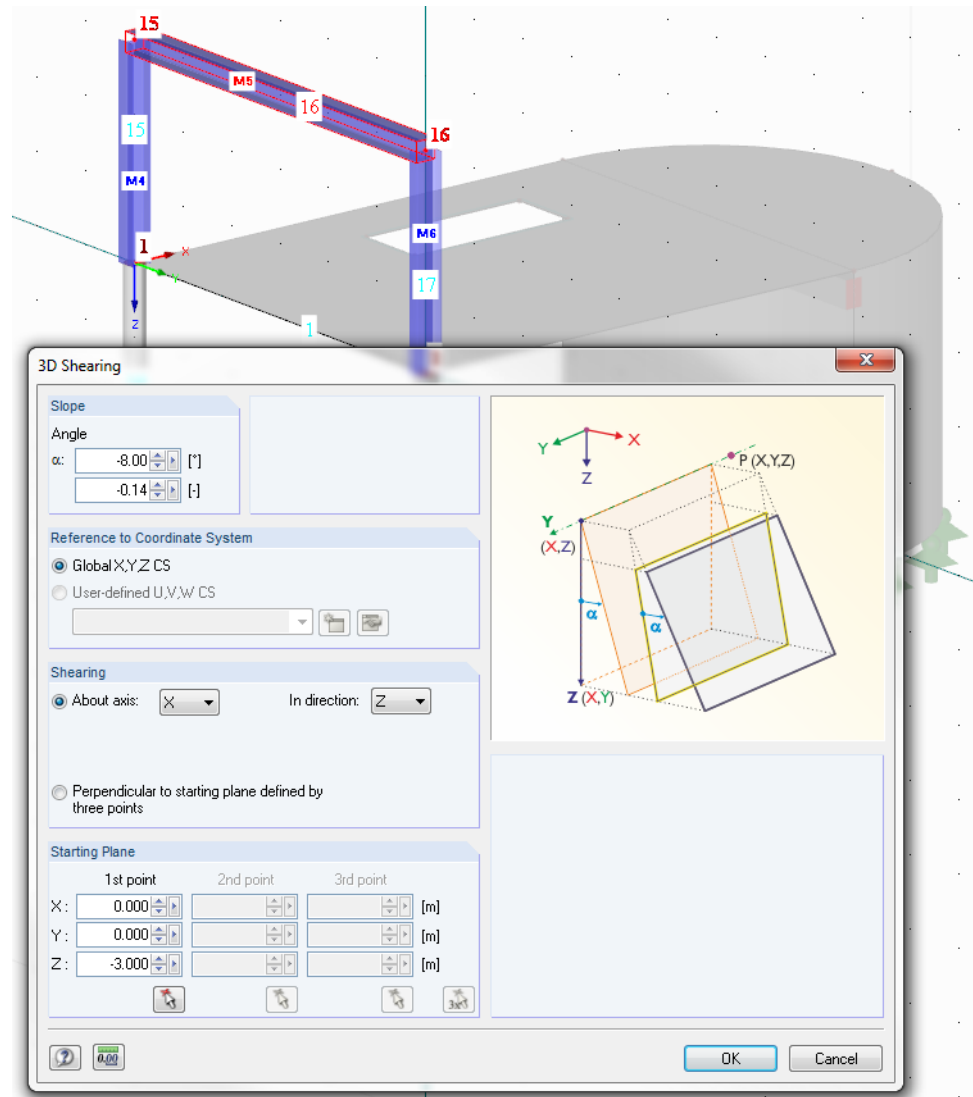


Figure 4.32: Adjusting the inclination of the horizontal beam

We want to modify the inclination of the beam by  $-8^\circ$  About axis **X**. We have to enter a negative value because the objects will be rotated counterclockwise about the axis **X**. The shearing is applied in vertical *direction Z*. Finally, we define the *1st point* of the rotation axis by using the [Select] button. We select node 15 having the coordinates **0.000/0.000/-3.000** and confirm the input with [OK].

### 4.5.1.3 Connecting Beams with Releases

#### Release definition

The horizontal beam cannot transfer any bending moments into the columns because of its connection type. Therefore, we define a release that we assign to both sides of the beam later.

This time we use the *Data* navigator: We right-click the entry *Member End Releases* and select *New Member End Release* in the shortcut menu.

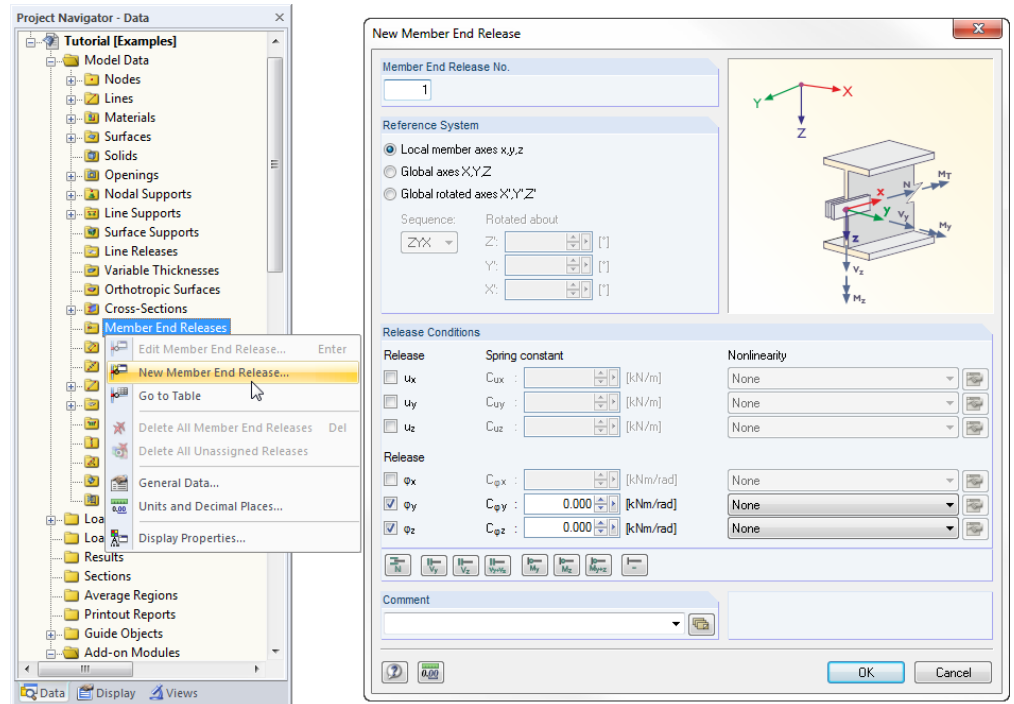


Figure 4.33: Opening the dialog box *New Member End Release* in the *Data* navigator

In the dialog box *New Member End Release*, displacements or rotations can be selected that are released at the member end. In our example, those are the rotations  $\varphi_y$  and  $\varphi_z$ . Thus, no bending moments can be transferred at the node.

We close the dialog box by clicking the [OK] button without modifying any data.

#### Release assignment

It would be possible to double click the member to open the dialog box *Edit Member* and assign the releases. However, we use a special function that is available in the following menu:

On the **Insert** menu, we select **Model Data**, point to **Member Releases** and select **Assign to Members Graphically**.

The dialog box *Assign Member End Releases to Members Graphically* appears. We open the list and select release type 1 that we have just defined. Then, we click [OK].

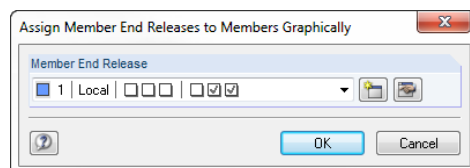


Figure 4.34: Dialog box *Assign Member End Releases to Members Graphically*

We see in the work window that RFEM has applied a one-third division to the members. By clicking the end of a member we can define the release graphically at this member end. Now we click member 5 in its middle area to assign the release to both sides.

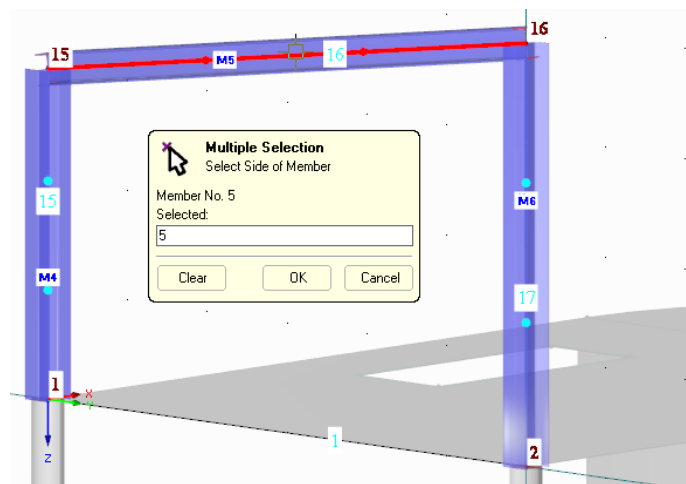


Figure 4.35: Assigning release graphically

#### 4.5.1.4 Reverse Member Orientation

For the graphical representation of imperfections it may be comfortable for us when the member orientation of columns is directed from the bottom to the top. Therefore, we change the orientation of the right steel column, using a function of the member shortcut menu.

Moving the pointer near member 6, we can see the orientation arrow appearing on the member. We right-click the member and open its shortcut menu where we select the menu item *Reverse Member Orientation*.

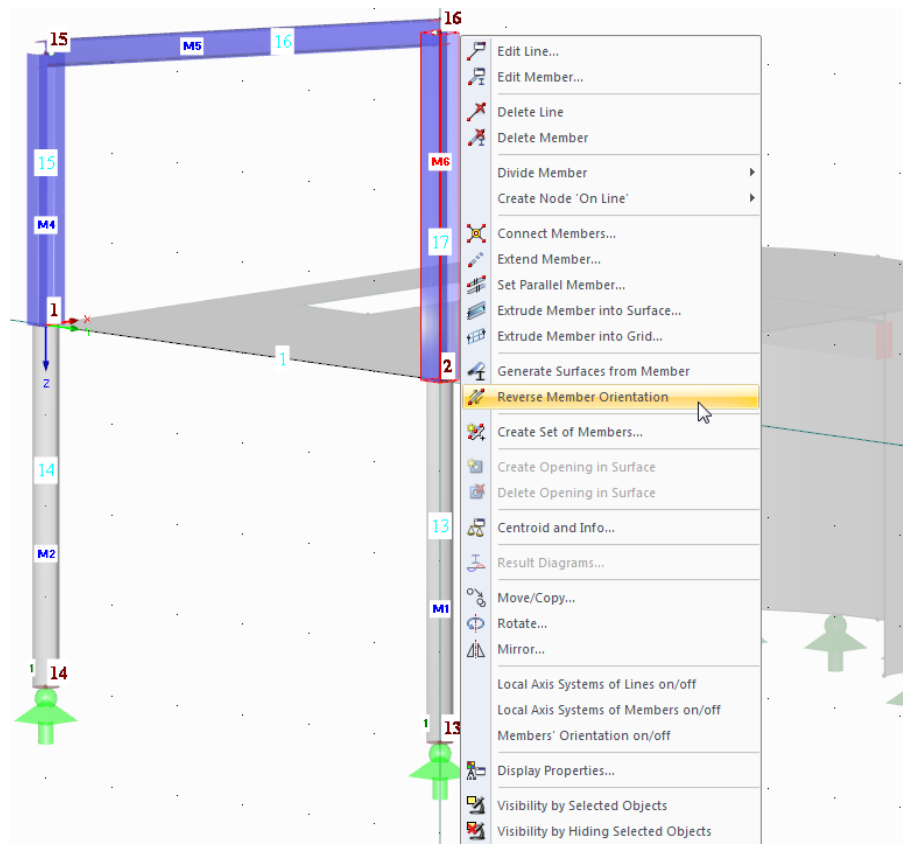


Figure 4.36: Shortcut menu of member

#### 4.5.1.5 Copying Frame

The second frame of the monopitch roof can be created very quickly as a copy.

We draw a selection window across the frame, enclosing members 4 to 6. Please take care not to include any of the concrete columns! If necessary, you can rotate the model to set a more favorable view. We can also click the members one after the other by holding down the [Ctrl] key.



Before we create the copy, we set the [Work Plane XZ] so that we are able to copy the structure out of the frame plane.

We press the [Ctrl] key. Now, we grab the selection near the foot point of the higher column (node 2) and shift it to the arc end at the wall top. The [+] symbol next to the mouse pointer indicates that the objects are being copied. As soon as the coordinates of the grid point **7.000/6.000/0.000** are displayed in the status bar, we release the mouse button.

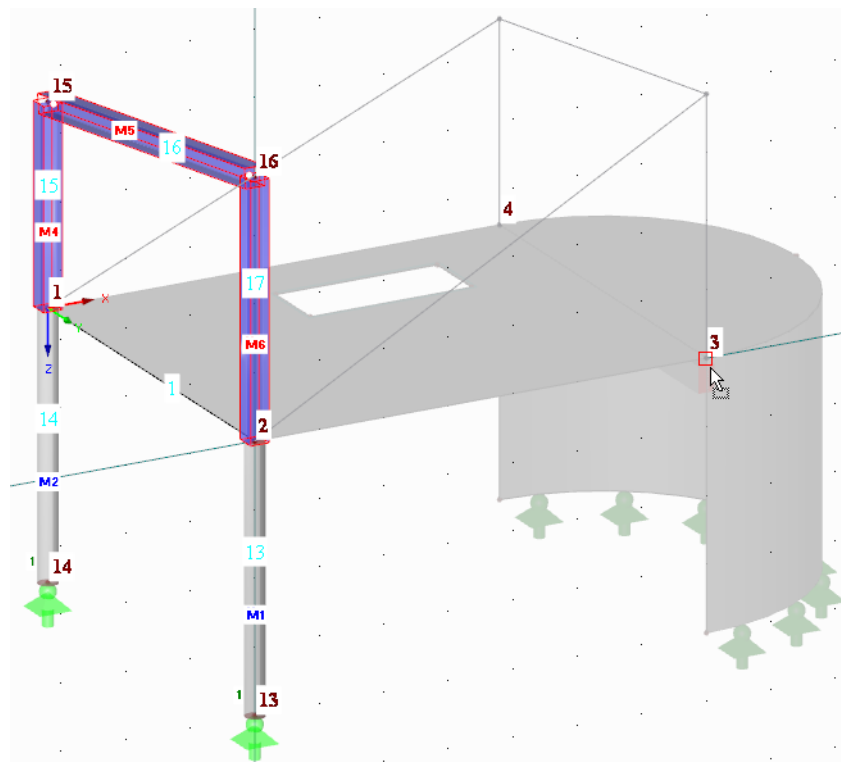
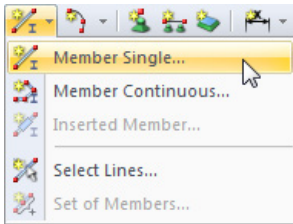


Figure 4.37: Copying frame by drag-and-drop

Nodes and lines are merged automatically with already defined objects.

## 4.5.2 Purlins

### 4.5.2.1 Defining Members Individually



Again, we use the list button [New Member] for the definition of both purlins. We select the option *Member Single* and open the dialog box *New Member*.

We define a cross-section for the *Member start* by using the [New] button again (see Figure 4.19, page 24).

In the dialog box *New Cross-Section*, we click the button [HE-B] at the box top. The dialog box *Rolled Cross-Sections - I-Sections* opens where we select the cross-section **HE B 260** from the HE B cross-section row (see figure below). Again, number **2 - Steel S 235** is preset.

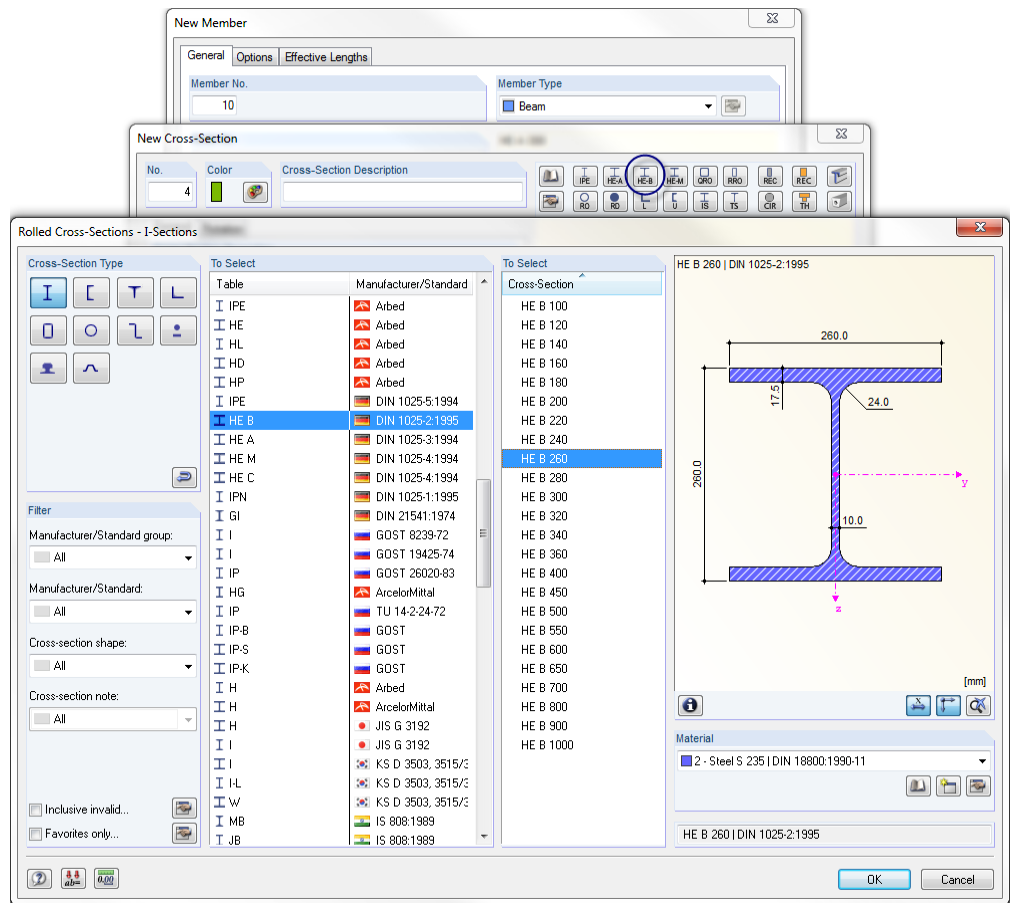


Figure 4.38: Selecting HE B 260 in the library

We confirm all dialog boxes with [OK].

We define the purlin at the lower eaves by clicking nodes **15** and **19** one after the other.

Then we click nodes **16** and **20** to create the second purlin.

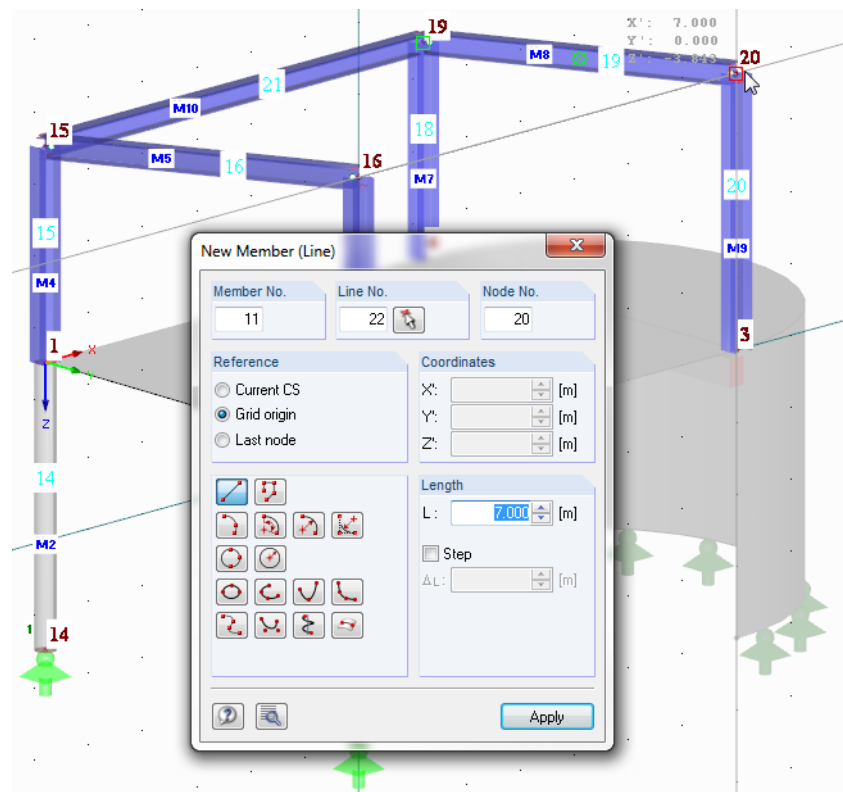


Figure 4.39: Defining purlin members

To quit the input mode, we use the [Esc] key or right-click into the empty work window.

### 4.5.2.2 Connecting Members Eccentrically

We want to connect the purlins eccentrically to the frame columns. Thus, we shorten the system line by half of the cross-section height of the HE A 300 columns.

#### Definition of eccentricity

We double-click the purlin at the high eaves (member 11). In the dialog box *Edit Member*, we change to the dialog tab *Options*. In the dialog section *Member Eccentricity*, we click the [New] button to open the dialog box *New Member Eccentricity*.

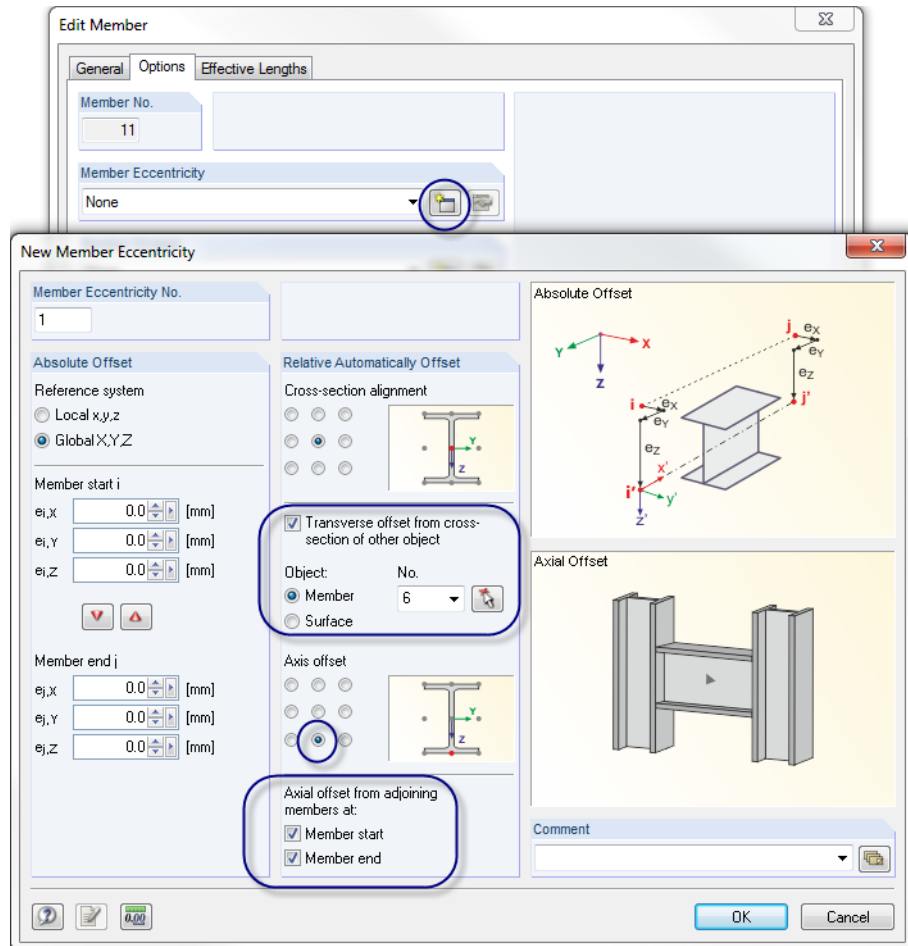


Figure 4.40: Dialog box *New Member*, tab *Options* and dialog box *New Member Eccentricity*



We select the option *Transverse offset from cross-section of other object*. In our example, the *Object* is the column: We use the [Select] function to select **Member 6** graphically.

Then, we define the *Axis offset* in direction of the positive cross-section axis *z*.

Finally, in the dialog section *Axial offset from adjoining members*, we select the check boxes for **Member start** and **Member end** to arrange the offset on both sides.



After confirming all dialog boxes we can check the result with a maximized view (for example zooming by rolling the wheel button, moving by holding down the wheel button, rotating by holding down the wheel button and keeping the right mouse button pressed).

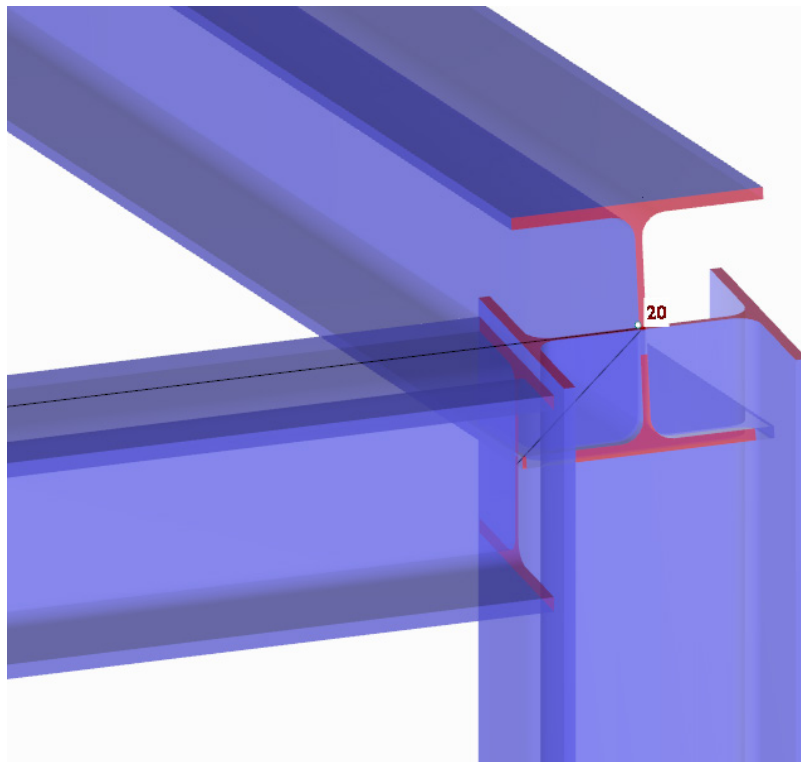


Figure 4.41: Eccentric connection of purlin in zoomed view

### Apply eccentricity to another member

To transfer the eccentricity to the second purlin, we use the input tables.

We set table 1.17 *Members* listing numerically the member data of all members that we have defined so far. When we click into table row 10, we see that the second purlin is highlighted in the work window and displayed in the selection color.

Member No.	A Line No.	B Member Type	C Cross-Section No. Start	D Cross-Section No. End	E Member Rotation Type	F Member Rotation $\beta$ [°]	G Release No. Start	H Release No. End	I Eccentr. No.	J Division No.	K Taper Shape	L Length L [m]	M Weight W [kg]	N
1	13	Beam	1	1	Angle	0.00	0	0	0	0		4.000	706.9	Z
2	14	Beam	1	1	Angle	0.00	0	0	0	0		4.000	706.9	Z
3	3	Rib	2	2	Angle	0.00	0	0		0		6.000	1500.0	Y
4	15	Beam	3	3	Angle	0.00	0	0	0	0		3.000	266.1	Z
5	16	Beam	3	3	Angle	0.00	1	1	0	0		6.059	537.5	YZ
6	17	Beam	3	3	Angle	0.00	0	0	0	0		3.843	340.9	Z
7	18	Beam	3	3	Angle	0.00	0	0	0	0		3.000	266.1	Z
8	19	Beam	3	3	Angle	0.00	1	1	0	0		6.059	537.5	YZ
9	20	Beam	3	3	Angle	0.00	0	0	0	0		3.843	340.9	Z
10	21	Beam	4	4	Angle	0.00	0	0	1	0		6.700	620.6	X
11	22	Beam	4	4	Angle	0.00	0	0	1	0		6.700	620.6	X
12														

Figure 4.42: Assigning eccentricity in table 1.17 *Members*

We place the pointer into column *I* and enter **1**, which is the number of the eccentricity that we have just defined. We can also select it from the list.

After leaving the table cell with the [Tab] or [-] key, we see the modification displayed in the graphic.

In the same way, it would be possible to define another eccentricity for the horizontal beams. However, as these members are connected to the column webs, we want to neglect those additional moments in our example.

### 4.5.3 Diagonal

The final member that we insert is a diagonal for stiffening. It can only transfer tensile forces. Generally, bracings are defined crosswise but the calculation in the demo version only allows for 12 members. A tension member has the effect that the model is calculated non-linearly. In case of compression forces, this member is removed from the stiffness matrix (failure).

#### 4.5.3.1 Defining Member

With the button [Member Single] we open again the dialog box *New Member* where we select the entry **Tension** from the *Member Type* list.

We define a new cross-section for the *Member start* by using the button [New] (see Figure 4.19, page 24) that opens the cross-section database.

In the dialog box *New Cross-Section*, we click the [L] button. The dialog box *Rolled Cross-Sections - L-Sections* appears where we select cross-section **L 80x8** from the cross-section table L. The material **2 - Steel S 235** is preset again.

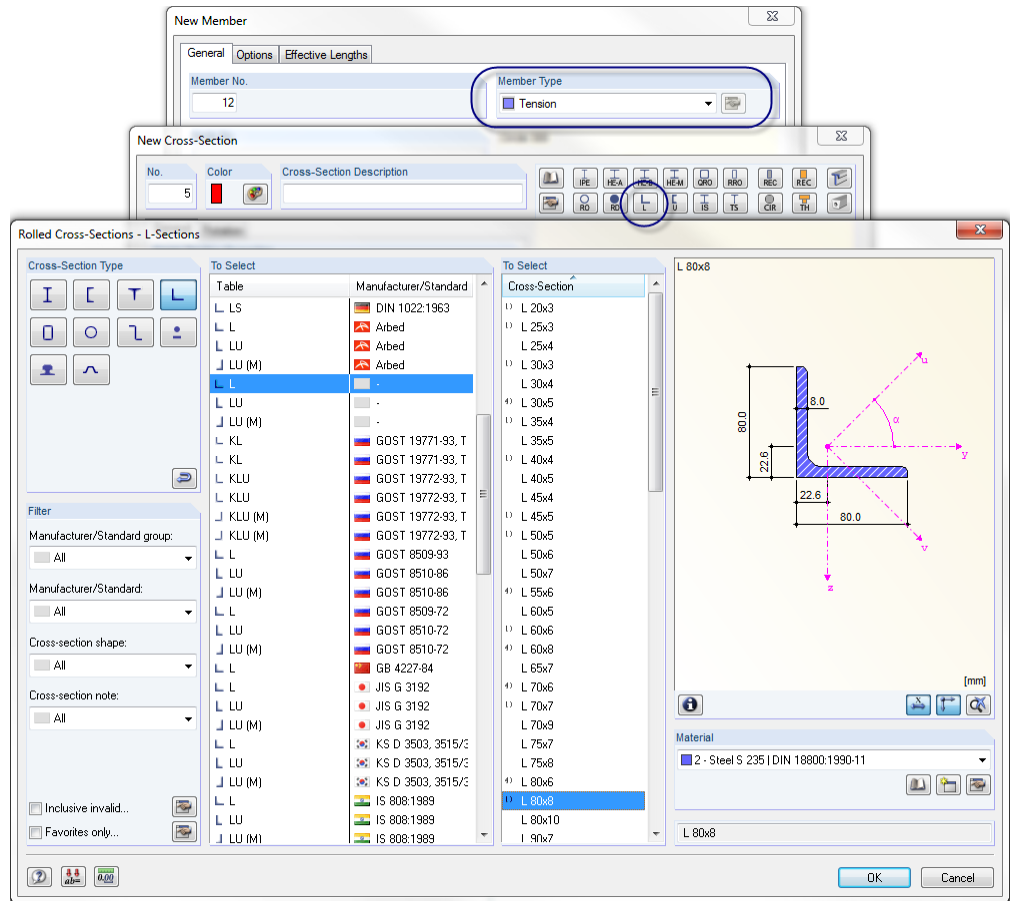
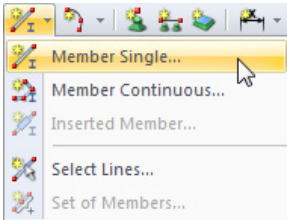


Figure 4.43: Defining tension member with cross-section L 80x8

We confirm all dialog boxes with [OK], and then we click the nodes **15** and **2** one after the other to define the diagonal (see figure below).

To quit the input mode, we use the [Esc] key. We can also right-click into the empty work window.

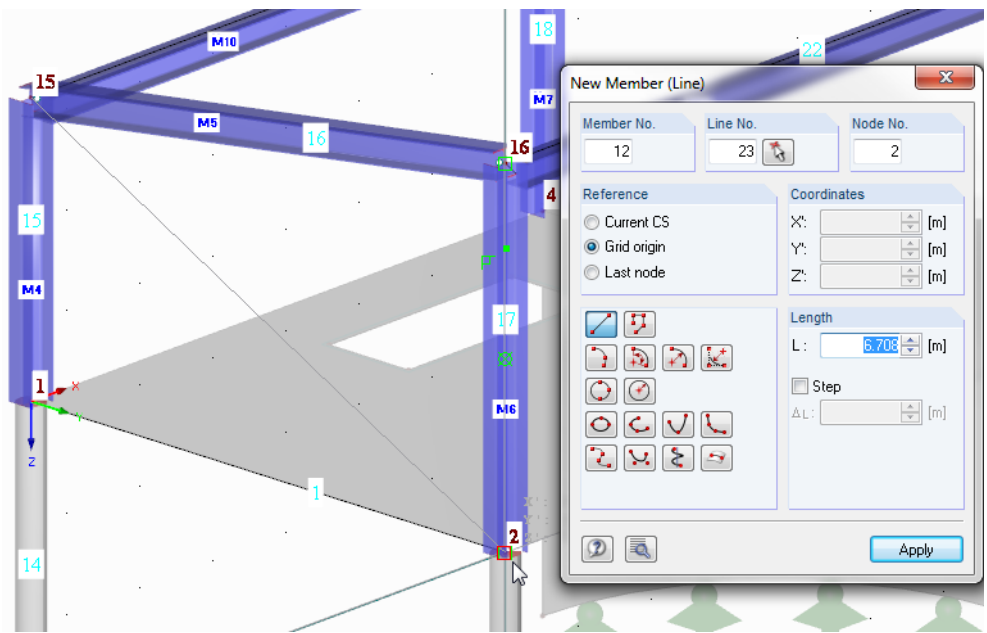


Figure 4.44: Defining the diagonal

### 4.5.3.2 Rotating Member

A tension member adds to stiffness only by its cross-sectional area. Thus, seen from a structural point of view, the rotation of the member is not irrelevant. For the rendered view, however, we want to rotate the angle section.

We double-click member 12 to open the dialog box *Edit Member* where we define a member rotation of  $-90^\circ$ .

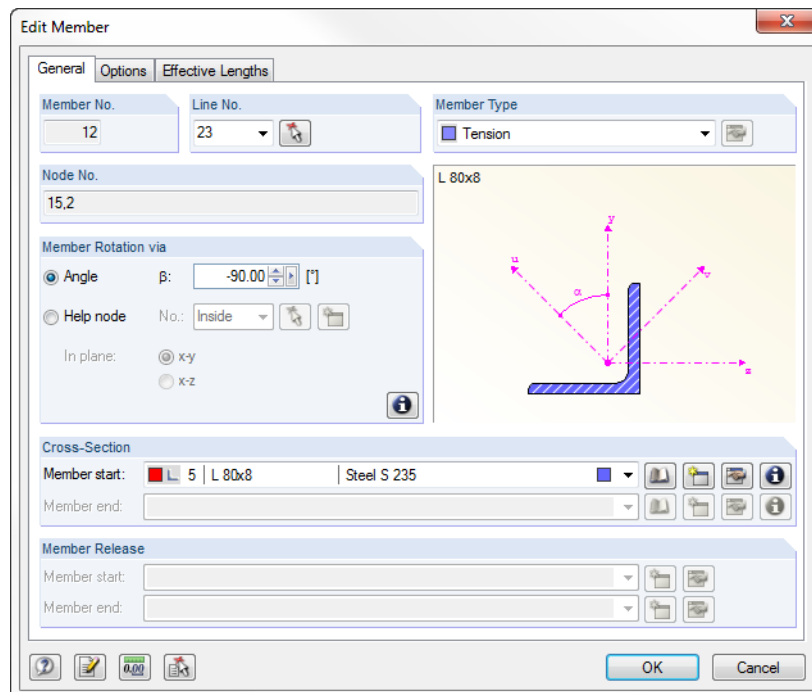


Figure 4.45: Defining a member rotation



Again, we can check the result in a maximized view by using the zoom and moving function (see page 9).

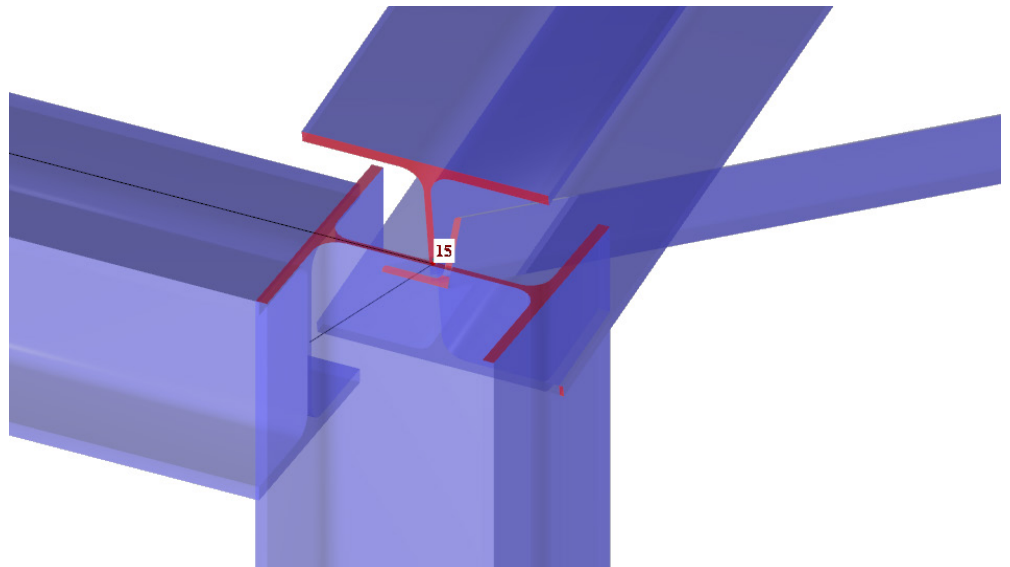


Figure 4.46: Rotated angle member in zoomed view

### Undo/restore



If you want, you can [Undo] the member rotation in this view in order to check the initial position of the cross-section. With the default functions *Undo* and *Redo* that you already know from Windows applications we can undo or restore input data in RFEM, too.

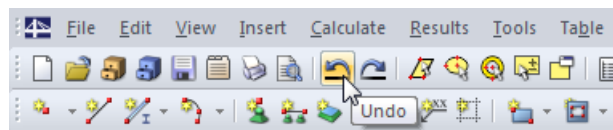


Figure 4.47: Buttons *Undo* and *Redo*

### Cancel visibility mode



The parts of the model displayed in RFEM as transparent objects can be reactivated in the *Views* navigator: With a click on the button [Cancel Visibility Mode] all objects are fully displayed again. With the toolbar button [Isometric View] we can reset the spatial full view.

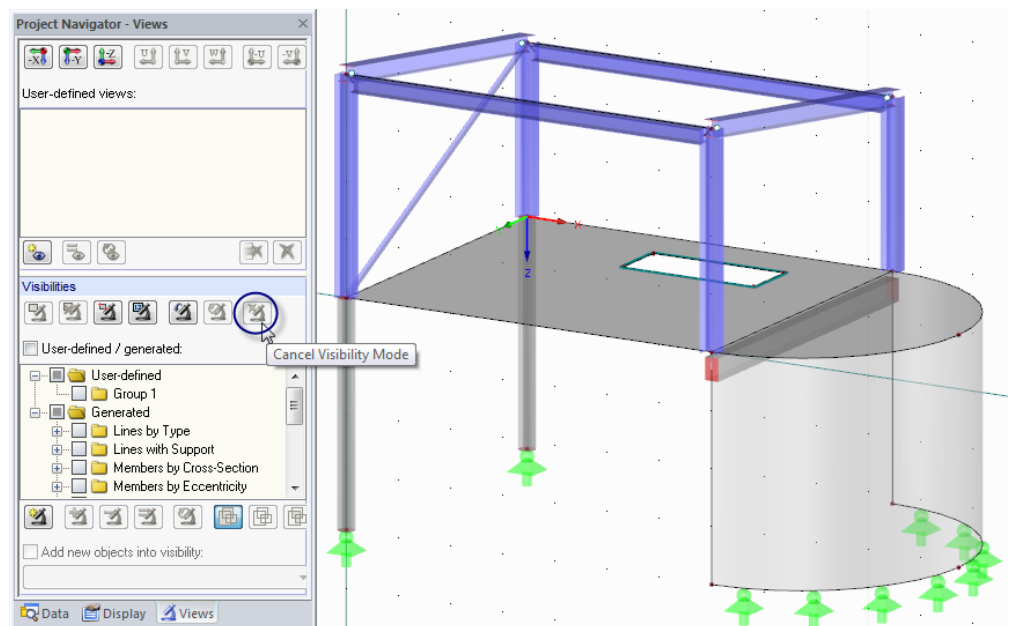


Figure 4.48: Full view of model

### Adjusting the color assignment

The *Display* navigator provides an option to display *Colors in Rendering According to* particular criteria. The default setting is the display of material colors.

You can click through the menu items to see how the display changes. With the option *Cross-Sections* for example we can distinguish different cross-section types at a glance.

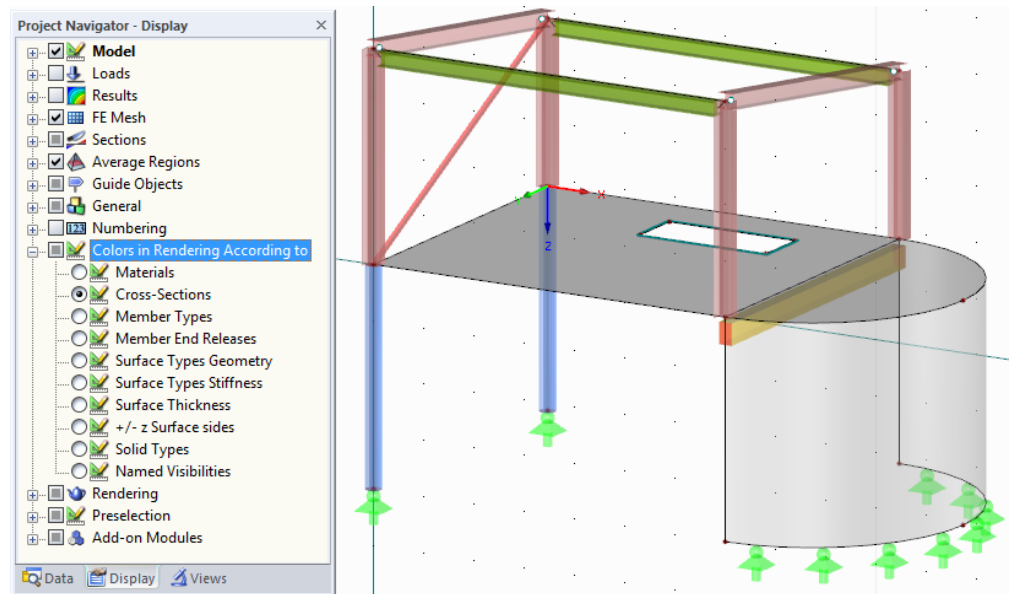


Figure 4.49: Distinguishing cross-sections by colors

For the following input we reset the option *Materials*.

## 4.6 Checking Input

### Checking *Data* navigator and tables



As mentioned before, RFEM offers us various possibilities to enter model data. The graphical input is reflected in both the *Data* navigator tree and the tables. We can display and hide navigator and tables by selecting **Navigator** or **Table** on the **View** menu. We can also use the corresponding toolbar buttons.

In the tables, the model objects are organized in numerous tabs. Graphics and tables are interactive: To find an object in the table, for example a member, we set table 1.17 *Members* and select the member in the work window by clicking. We see that the corresponding table row is highlighted (see Figure 4.42, page 41).

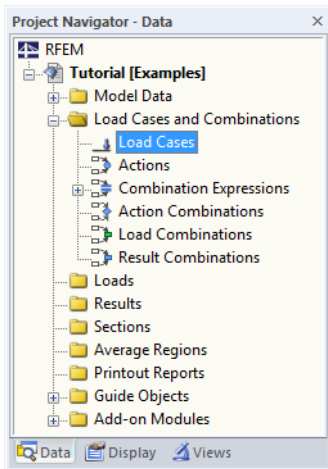
We can check the numerical data of our input quickly.

### Saving data



Finally, the input of model data is complete. To save our file, we select **Save** on the **File** menu or use the toolbar button shown on the left.

## 5. Loads



The *Data* navigator contains the following entries in the folder *Load Cases and Combinations*:

- Load cases
- Actions
- Combination expressions
- Action combinations
- Load combinations
- Result combinations

We define the actual loading like self-weight, snow and wind load in load cases. Then, load cases are organized in actions and superimposed with partial safety factors according to the standard's combination expressions (see chapter 6).

### 5.1 Load Case 1: Self-weight

The first load case contains the permanently acting loads from self-weight, floor structure, earth pressure and roof finishes (see chapter 2.3, page 7).

We use the button [New Surface Load] to create a load case.

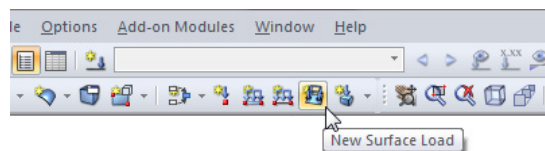


Figure 5.1: Button *New Surface Load*

The dialog box *Edit Load Cases and Combinations* appears.

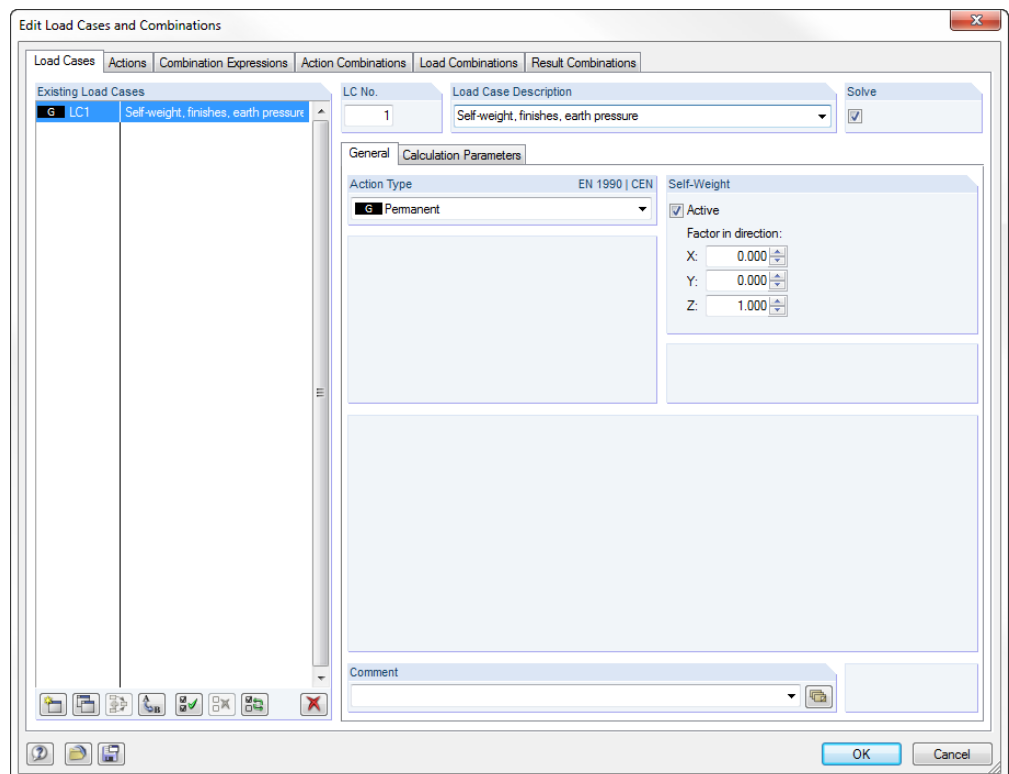
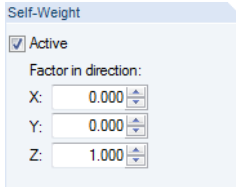


Figure 5.2: Dialog box *Edit Load Cases and Combinations*, tabs *Load Cases* and *General*

Load case no. 1 is preset with the action type *Permanent*. In addition, we enter the *Load Case Description* **Self-weight, finishes, earth pressure**.

### 5.1.1 Self-weight

The *Self-Weight* of surfaces and members in direction Z is automatically taken into account when the factor *Active* is specified with *1.000* as already preset.



### 5.1.2 Floor Structure

We confirm the input by clicking the [OK] button. The dialog box *New Surface Load* appears.

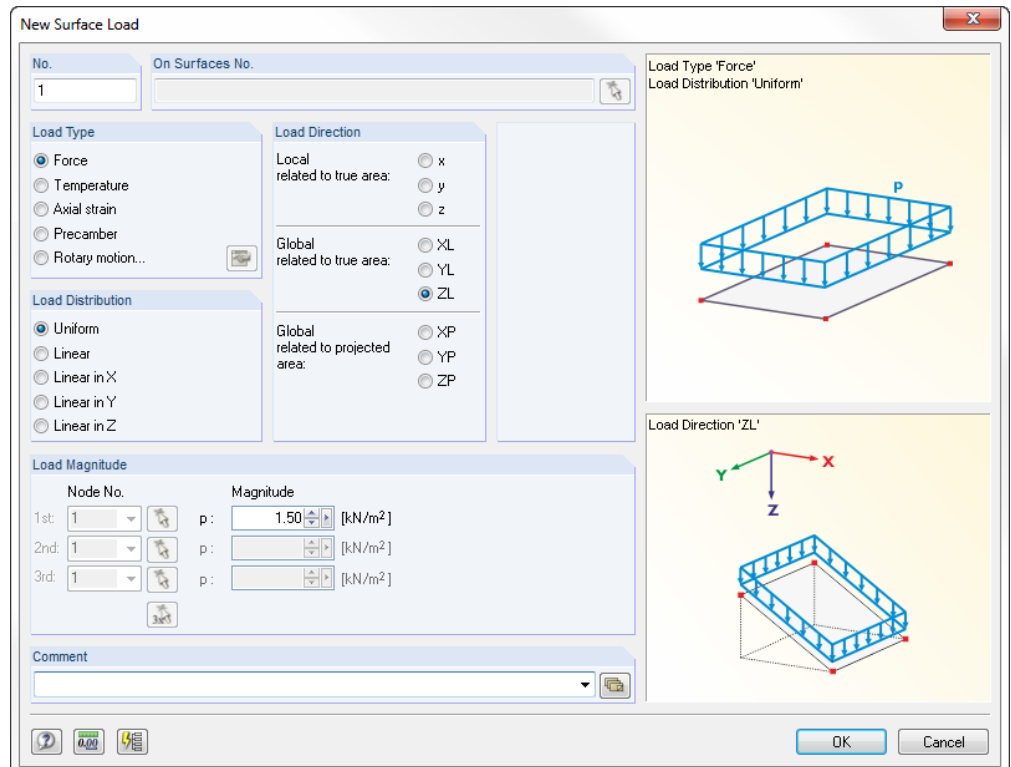


Figure 5.3: Dialog box *New Surface Load*

The floor structure is acting as load type *Force*, the load distribution is *Uniform*. We accept these presettings as well as the setting *ZL* for *Global* in the dialog section *Load Direction*.

In the dialog section *Load Magnitude*, we enter a value of **1.5** kN/m<sup>2</sup> (see chapter 2.3, page 7). Then, we close the dialog box by clicking [OK].

Now, we can assign the load graphically to the floor surface: We can see that a small load symbol has appeared next to the pointer. This symbol disappears as soon as we move the pointer across a surface. We apply the load to the floor with a click on surface **1** (see Figure 5.4).

The surface load is not applied to the opening. We can see the non-load bearing area identified by the load application symbol.

We can hide and display the load values with the toolbar button [Show Load Values].

To quit the input mode, we use the [Esc] key. We can also right-click into the empty work window.



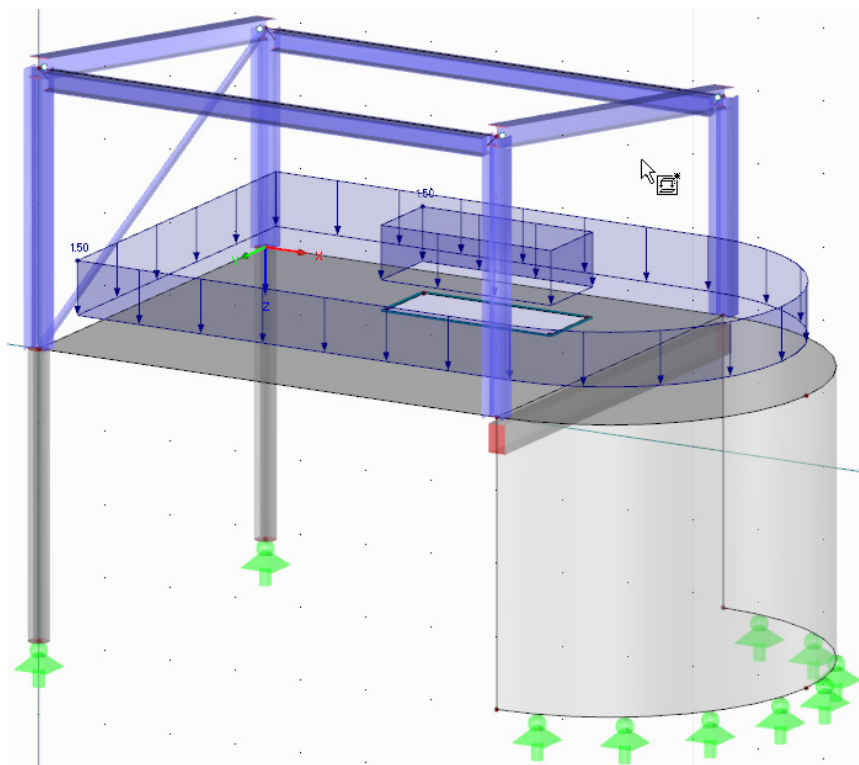


Figure 5.4: Graphical input of floor load

### 5.1.3 Earth Pressure



The earth pressure stressing the wall is represented by a linearly variable load acting perpendicular to the surface. This time, we select the curved surface **2** first, and then we open the corresponding dialog box for load input by using the button [New Surface Load].

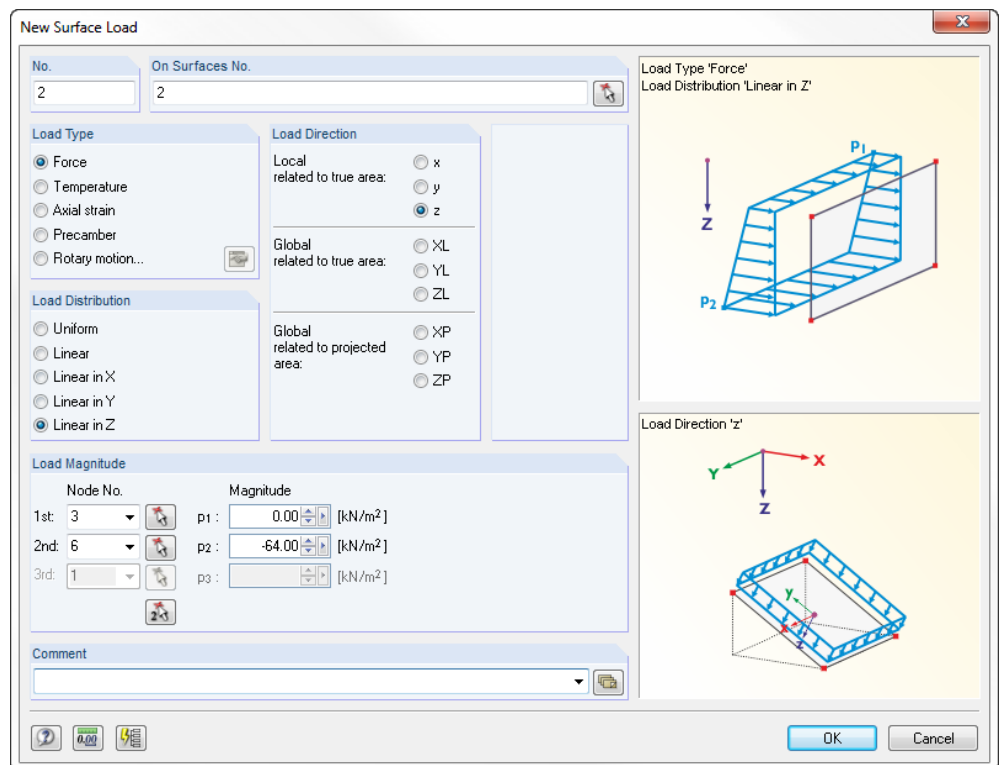
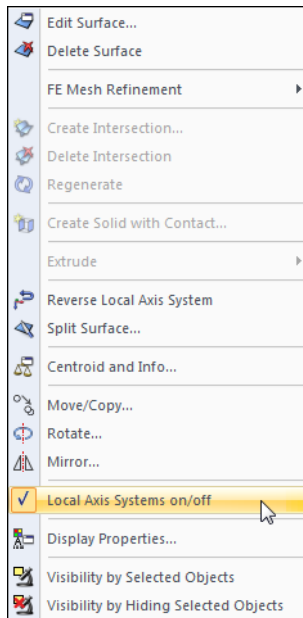


Figure 5.5: Dialog box *New Surface Load*

The load is set as load type *Force* with the *Load Distribution* **Linear In Z**. Thus, we select **Local z** as *Load Direction*.

To enter the *Load Magnitude*, we use the [Select] button to select significant locations on the model to which we assign load ordinates: We click *Node No. 3* and enter the *Magnitude* **0** kN/m<sup>2</sup>. Then, we click the [Select] button again to select *Node No. 6* to which we assign the *Magnitude* **-64** kN/m<sup>2</sup> (see chapter 2.3, page 7). We enter the load with a negative number because the local z-axis of the surface is directed to the outside.

After confirming the dialog data by clicking [OK], we see the linear surface load displayed in the model, increasing downwards and acting perpendicularly on the shell. We use the shortcut menu shown on the left (appears when we right-click the surface) to show the local surface axes.



Display of local axis systems

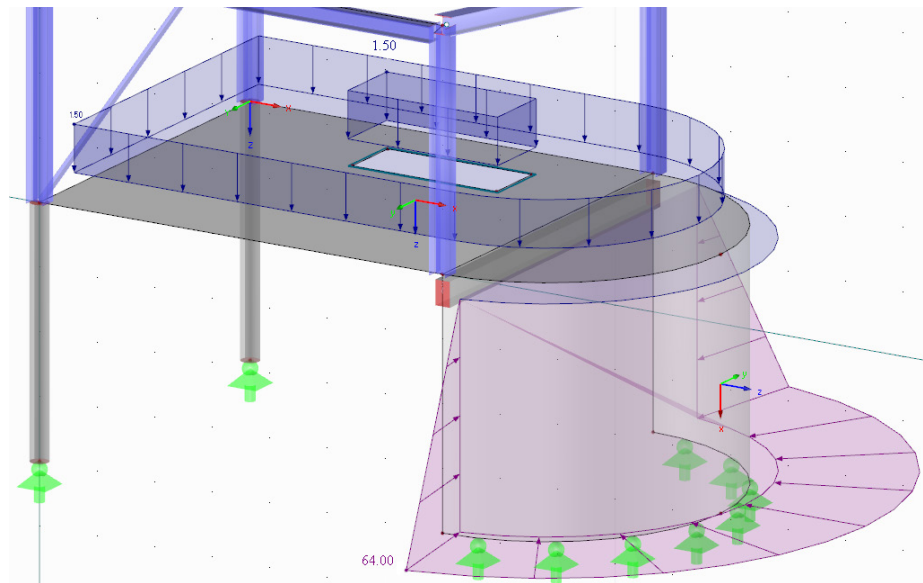


Figure 5.6: Linearly variable surface load (soil pressure) with display of local surface axes

### 5.1.4 Roof Load

The loading due to roof finishes (roofing, supporting structure) is acting as permanent load, too. For applying loads acting upon surfaces to the steel construction, RFEM offers us a tool that is able to convert area loads into member loads.

To open the generator dialog box,

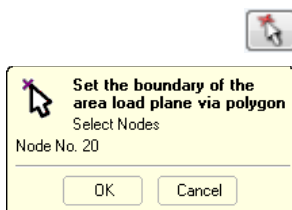
we point to **Generate Loads** on the **Tools** menu, and then we select **From Area Loads via Plane**.

In the dialog box *Convert Area Loads to Member Loads via Planes*, we specify the following settings (see Figure 5.7):

The *Area Load Direction* of the roof structure is *Global related to the true area ZL* with the *Area Load Magnitude* **1.2** kN/m<sup>2</sup> (see chapter 2.3, page 7).

Then, we define the plane of the area load graphically by means of the [Select] button: In the work window, we click the four corner nodes of the roof area **16, 15, 19** and **20** one after the other. Finally, we close the selection window with [OK].

The roof's supporting structure introduces the roof loading (not displayed in the model) into the structural system along the purlins. This means: Both horizontal beams of the monopitch roof do not participate in transferring loads from the roof loading. Thus, they must be excluded from the load generation. We use the [Select] button shown on the left, available in the dialog section *Remove Influence from*, to select one of the horizontal beams graphically in the work window (member **8** or **5**). We click [OK] in the selection window. After that, the generator dialog box should look as follows.



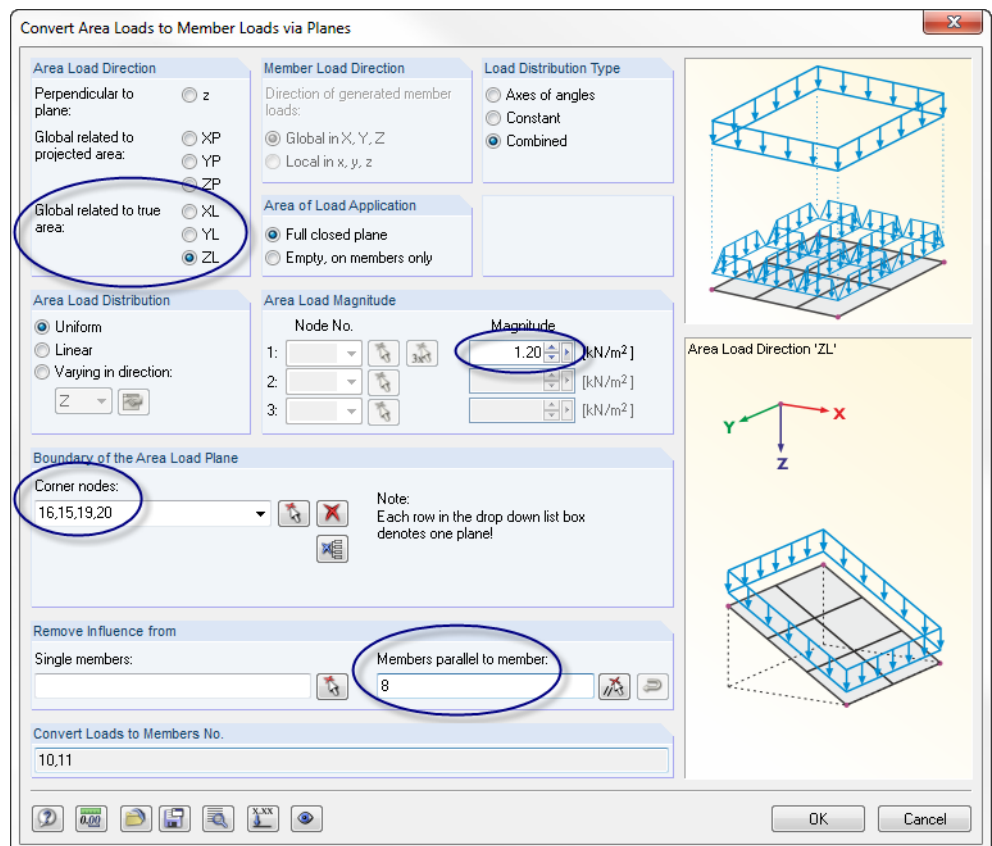


Figure 5.7: Dialog box *Convert Area Loads to Member Loads via Planes*

We confirm the dialog settings with [OK]. An *Info* dialog box appears showing us information about the conversion of area load values into member loads. We confirm this dialog data as well.

The loading is represented as roof area load. To display the generated loads acting on both purlins, we right-click this load and open the shortcut menu where we select the option *Disconnect Generated Load*.

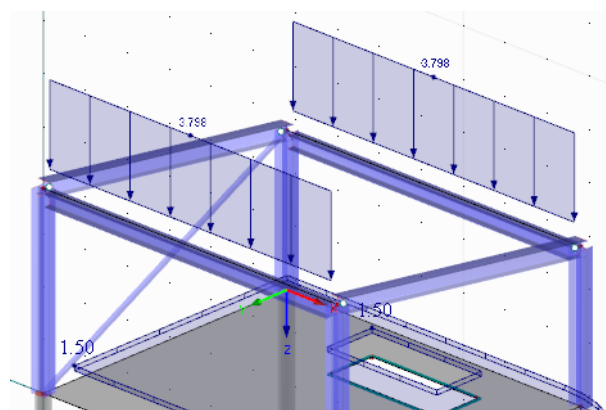
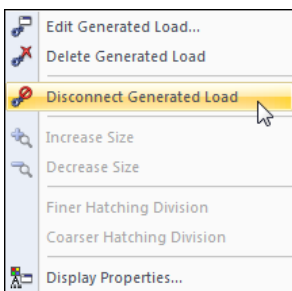


Figure 5.8: Member loads from area loads on purlins



However, we [Undo] this specification step: The input parameters entered in the generator dialog box get lost for disconnected loads. It would no longer be possible to adjust for example the load magnitude in case of subsequent modifications.

Now, the input for the load case *Self-weight* is complete.

## 5.2 Load Case 2: Live Load



Before we enter live loads, we create a new load case. To open the corresponding dialog box, we point to **Load Cases and Combinations** on the **Insert** menu and select **Load Case**, or we use the corresponding button in the toolbar (to the left of the load case list).

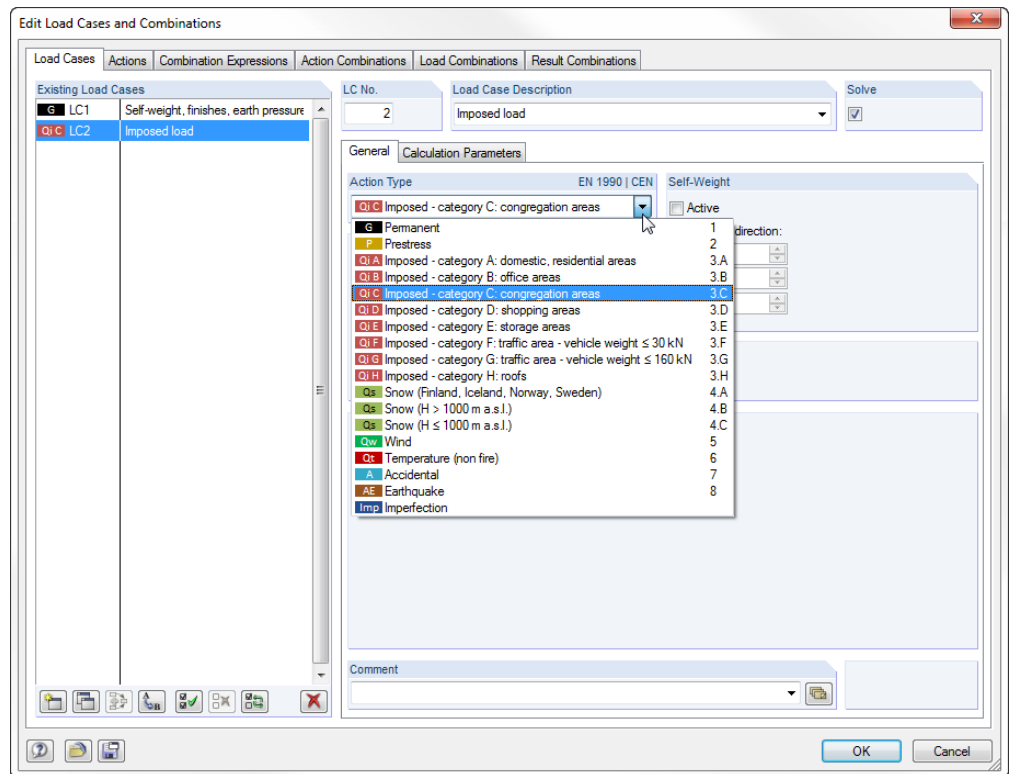


Figure 5.9: Dialog box *Edit Load Cases and Combinations*, tab *Load Cases*

For the *Load Case Description* we enter **Imposed load**, or we choose the entry from the list.

We change the *Action Type* to **Q<sub>i</sub> C Imposed - category C: congregation areas** (see chapter 2.3, page 7) by using the selection list. This classification is important for the partial safety factors and combination coefficients of the load combinations.

### 5.2.1 Floor Slab



We choose a new input option to enter the surface load: First, we select floor surface 1 with a mouse click. Now, when we open the already familiar dialog box by means of the button [New Surface Load], we can see that the number of the surface is already entered.

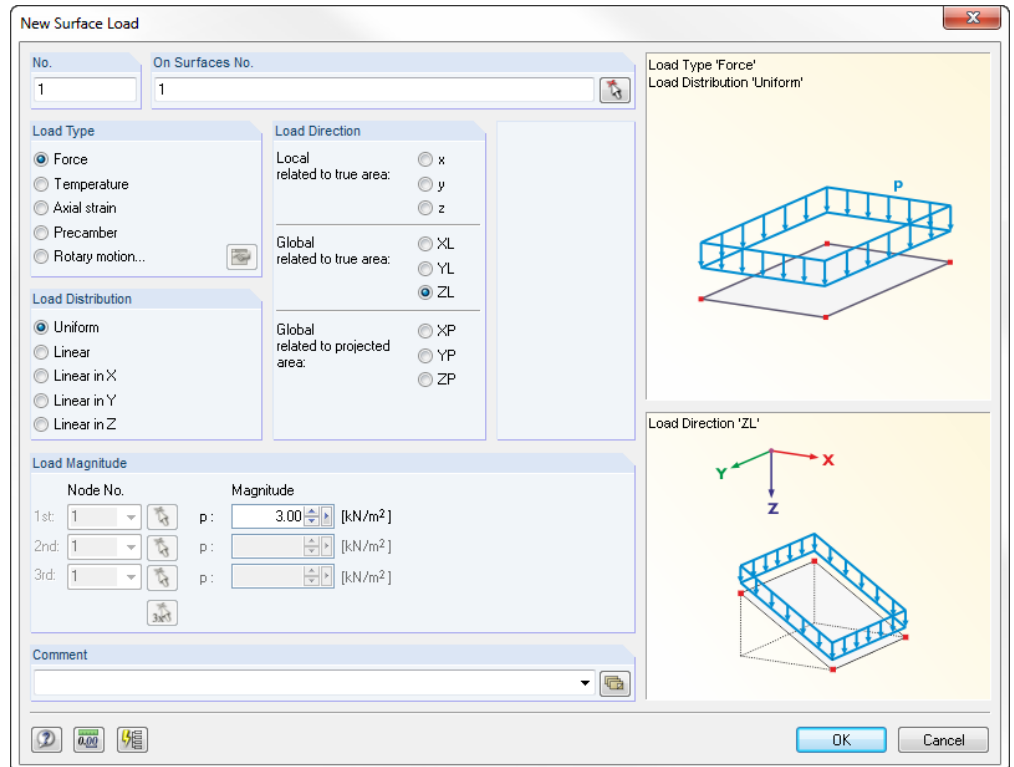


Figure 5.10: Dialog box *New Surface Load*

The imposed load is acting as load type *Force*, the load distribution is *Uniform*. We accept these presettings as well as the setting *ZL* for *Global* in the dialog section *Load Direction*.

In the dialog section *Load Magnitude*, we enter a value of  $3 \text{ kN/m}^2$  (see chapter 2.3, page 7). Then, we close the dialog box by clicking [OK].

### 5.2.2 Edge of Opening



It is easier to apply the linear load to the edge of the opening when we maximize the display of this area by using the *Zoom* function or the wheel button.



With the toolbar button [New Line Load] to the left of the button [New Surface Load] we open the dialog box *New Line Load*.

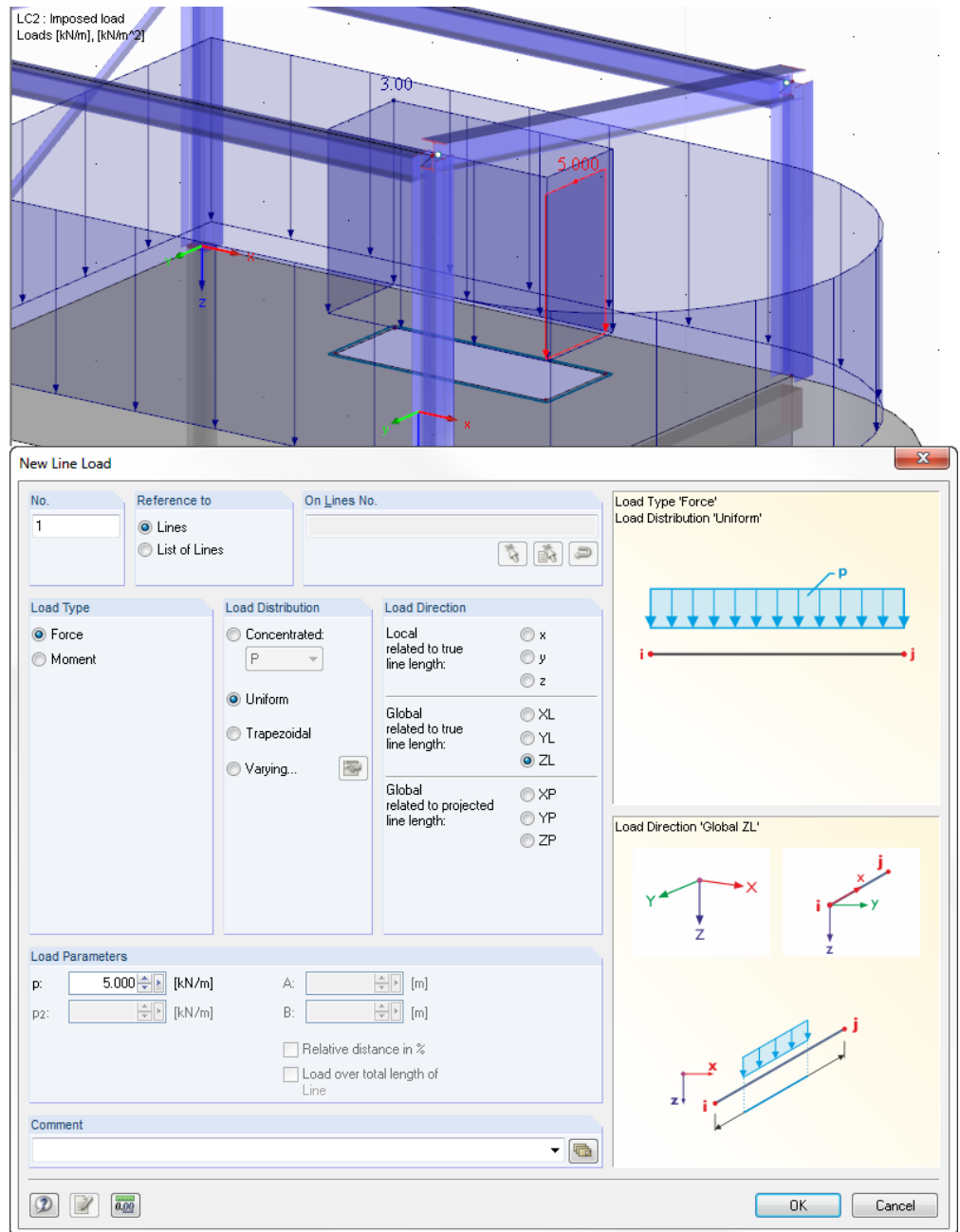


Figure 5.11: Dialog box *New Line Load*

The line load as load type *Force* with a *Uniform* load distribution is acting in the load direction *ZL*.



In the dialog section *Load Parameters*, we enter **5** kN/m. After clicking the [Ok] button we click line **11** at the opening's edge (check by display in status bar).

To quit the input mode, we use the [Esc] key or right-click into the empty work window.

## 5.3 Load Case 3: Snow

Again, we create a [New Load Case] to enter the snow load.

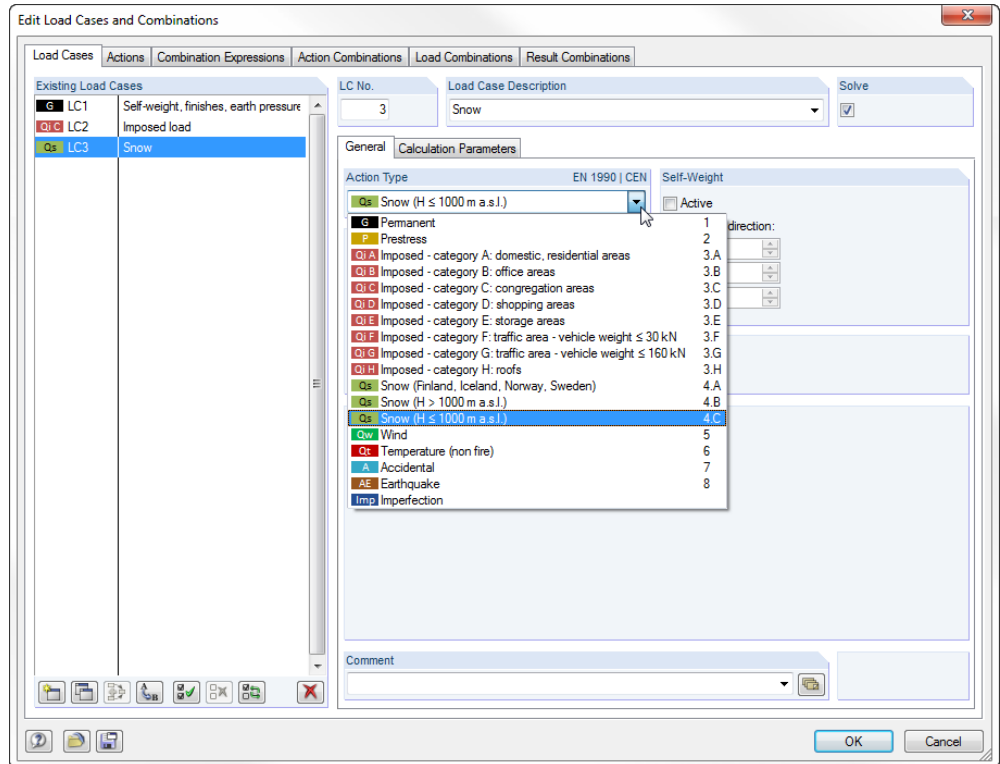


Figure 5.12: Dialog box *Edit Load Cases and Combinations*, tab *Load Cases*

For the *Load Case Description* we enter **Snow**, or we choose the entry from the list.

We set the *Action Type* to **Qs Snow (H ≤ 1000 m a.s.l.)**.

### 5.3.1 Roof

To enter the snow load for the monopitch roof, we use a load generator again. To open the corresponding dialog box,

we point to **Generate Loads** on the **Tools** menu, then we select **From Snow Loads** and **Flat/Monopitch Roof**.

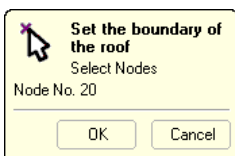
The dialog box *Generate Snow Loads - Flat/Monopitch Roof* opens where we define the following input (see Figure 5.13).

We select the *Snow Load Parameters* according to the national annex for Germany for snow load *Zone number 2*. Furthermore, we change the value of the *Altitude* to **500 m** (see chapter 2.3, page 7).

Then, we define the *Roof Geometry* graphically by using the [Select] button: We click the four corner nodes of the roof area **16, 15, 19** and **20** one after the other (see selection window shown on the left).

We check if the new **LC3** is set in the dialog section *Load Case to Generate*.

Again, we want to create *Member loads*, but both monopitch roof beams do not make a contribution to the load transfer (the snow loads are introduced into the structural system by the roof's supporting structure via purlins). We use the [Select] button shown on the left, available in the dialog section *Remove Influence from*, to select one of the horizontal beams graphically (member **8** or **5**).



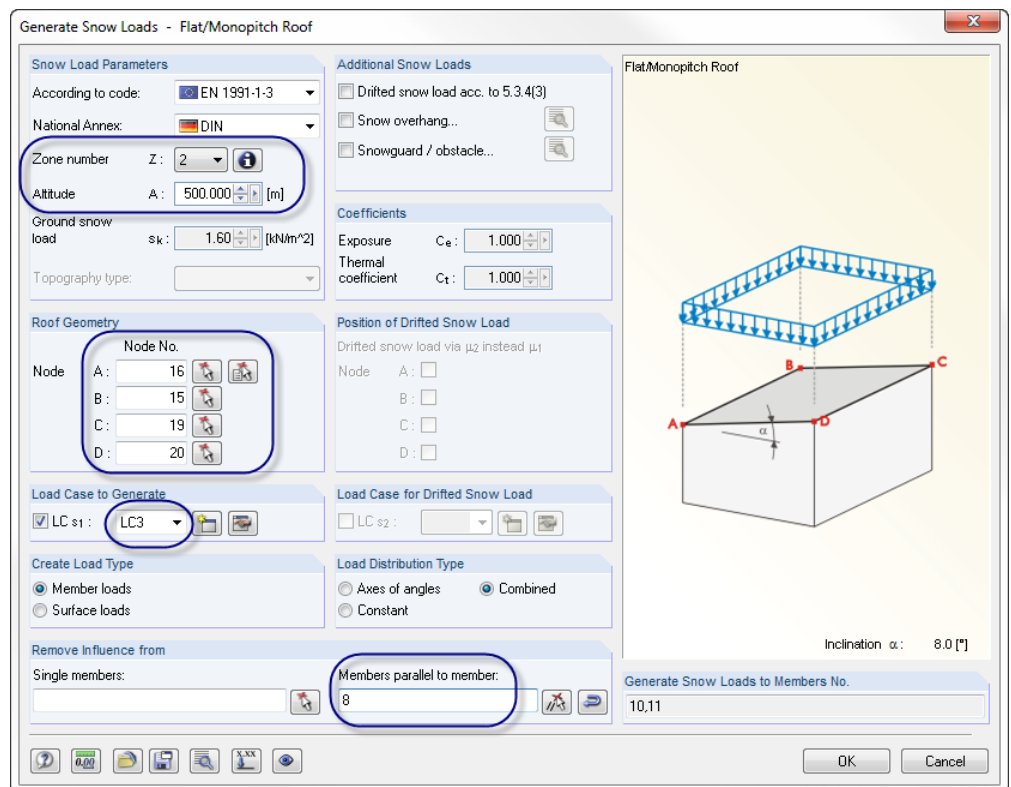
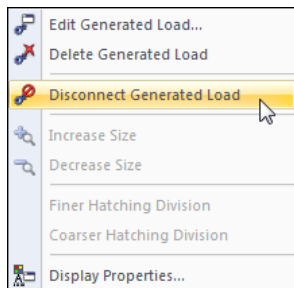


Figure 5.13: Dialog box *Generate Snow Loads - Flat/Monopitch Roof*



We confirm the dialog settings with [OK]. An *Info* dialog box appears showing us information about the conversion of area load values into member loads. We confirm this dialog data as well. The load is represented as roof area load with a value of 1.282 kN/m<sup>2</sup>.

To display the generated loads acting on both purlins, we can use the option *Disconnect Generated Load* which is available in the load shortcut menu. In this way, we can make visible both member loads of 4.023 kN/m each.

Separating loads should be undone immediately by using the [Undo] function so that the input is kept in the generator dialog box.

### 5.3.2 Floor

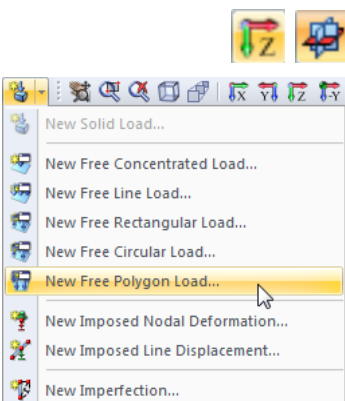
Snow loading also acts on the semicircular area of the floor surface. As surface 1 is stressed only partially by snow, we cannot use the function *New Surface Load*. In the full and trial versions of RFEM, it would be advisable to subdivide the floor into two surfaces in order to simply set a surface load on the semicircular surface. As the demo version allows only for two surfaces used in the model, we choose a more complex input option.

First, we set the [View in Z direction]. Then, we select plane [XY] as our new work plane.

We define the snow load as *Free Polygon Load*. We find the corresponding function in the list of the toolbar button [New Load] (to the right of the button [Surface Load]).

The dialog box *New Free Polygon Load* opens (see Figure 5.14) where we define the load to be acting *On Surfaces No. 1* and *Global related to projected area ZP*. In contrast to dead loads (like self-weight) which refer to the true area, snow loads must be related to the base area (this difference is not significant for horizontal surfaces).

The load is projected in the *XY Plane*.



List button for loads

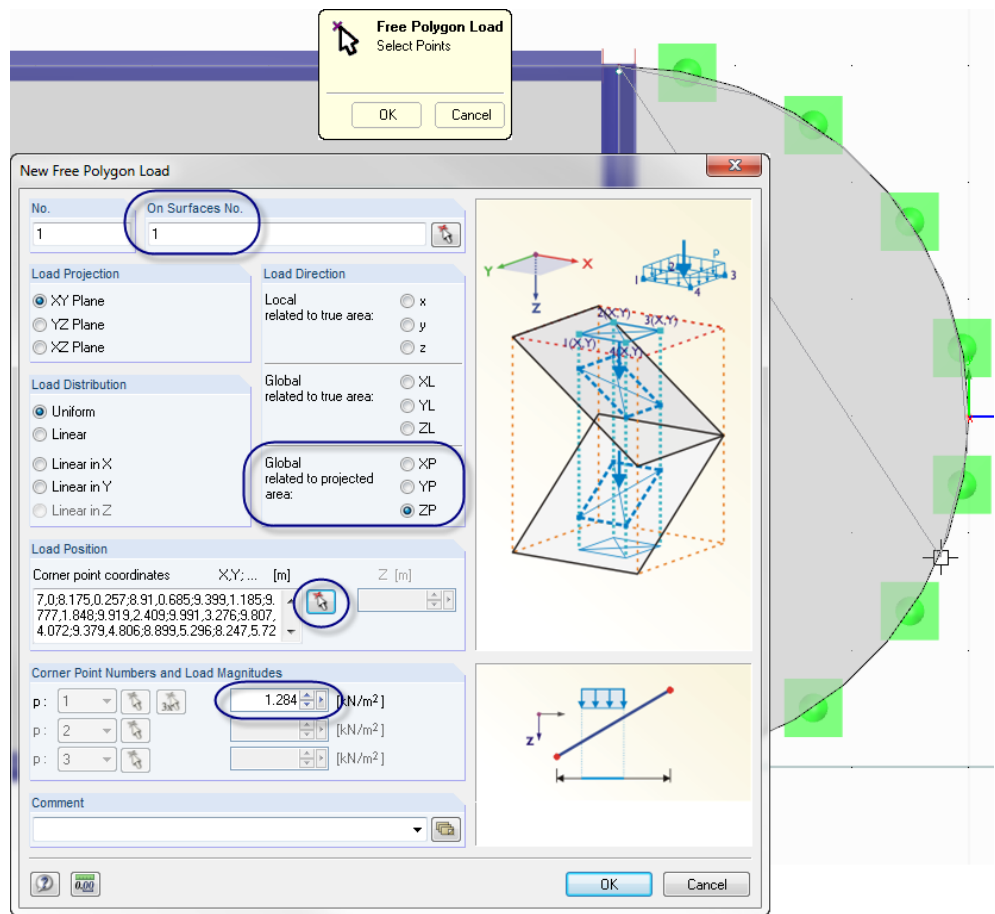


Figure 5.14: Defining new free polygon load by clicking arc points



We define the *Load Position* in the work window by using the [Select] button: We start at arc node **4** at the top of the arc, and then we use the reticle cursor to click any points on the arc line one after the other so that we approach the semicircular surface with a polygonal chain. As soon as we reach the arc end at node **3**, we close the yellow dialog box with [OK].



In the dialog section *Corner Point Numbers and Load Magnitudes*, we enter the value **1.284** kN/m<sup>2</sup> which was given as roof snow load by the generator.

We click [OK]. RFEM puts the load on the semicircular surface. We close the input mode with the [Esc] button or a right-click into the empty workspace. Then, we change to the [Isometric View].

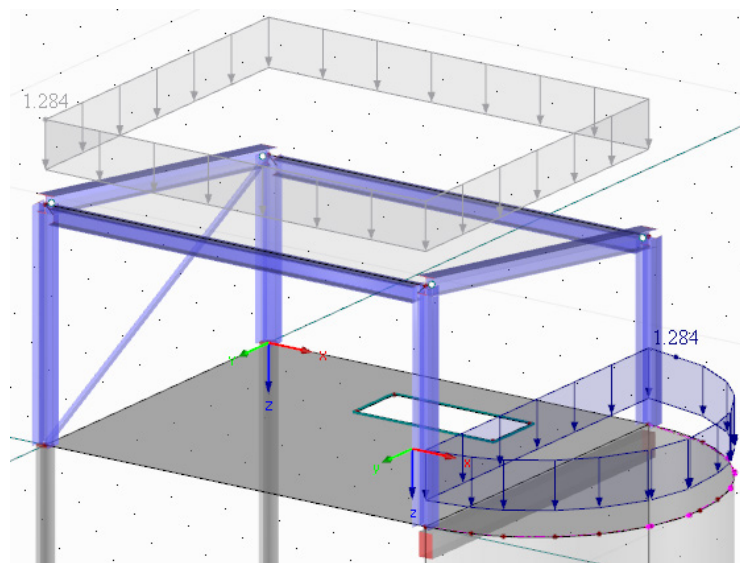
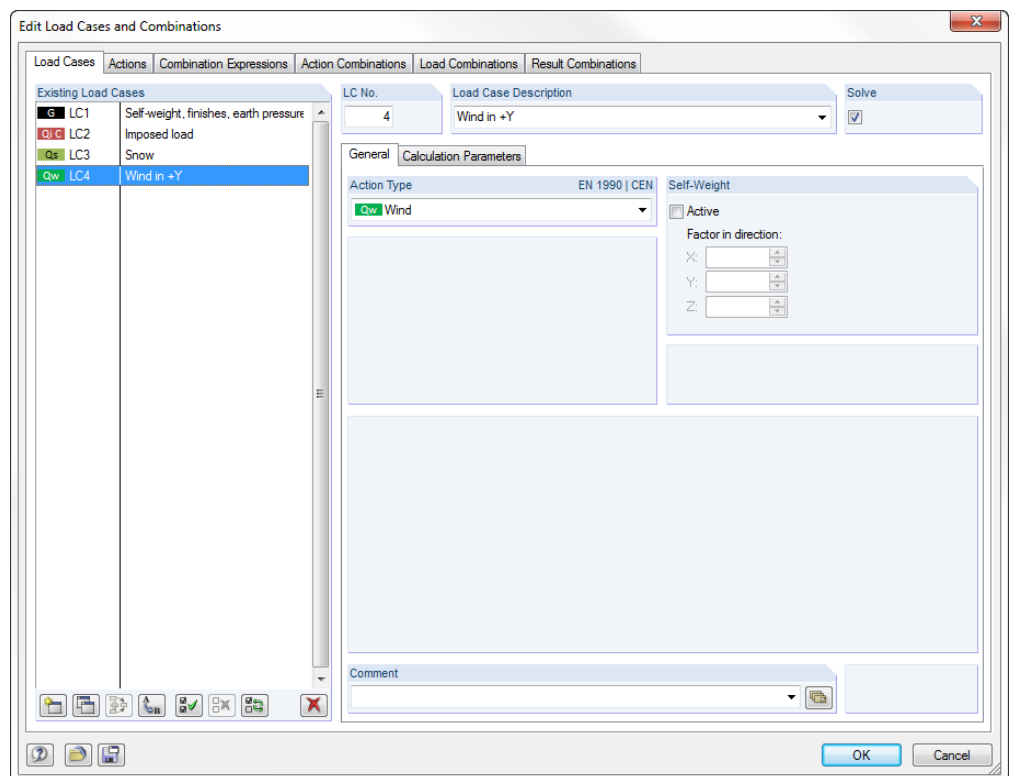


Figure 5.15: Snow loads

## 5.4 Load Case 4: Wind

We create a [New Load Case] for the wind loads in direction Y.

Figure 5.16: Dialog box *Edit Load Cases and Combinations*, tab *Load Cases*

In the dialog field *Load Case Description*, we select **Wind in +Y** from the list. The *Action Type* changes automatically to **Q<sub>w</sub> Wind**.

### 5.4.1 Steel Construction Loads

Again, we use a load generator to enter the wind load applied to the walls that are closed on all sides and to the monopitch roof. To access the corresponding function,

we point to **Generate Loads** on the **Tools** menu, and then we select **From Wind Loads** and click **Vertical Walls with Roof**.

In the dialog box *Generate Wind Loads - Vertical Walls with Roof*, we specify the following settings (see Figure 5.17):

The *Velocity Pressure* is defined according to the national annex of Germany for *Wind zone 1* and *Terrain category III* (see chapter 2.3, page 7). The [Info] buttons facilitate the assignment. Finally, we change the value of the *Structure height* to **8 m**.

Furthermore, we define the *Base Geometry* with the [Pick] button, selecting floor nodes **1, 4, 3** and **2** (please pay attention to the order shown in the dialog sketch). For the roof geometry we use the [Pick] function again, clicking the roof nodes **15, 19, 20** and **16**.

We check if wind direction **A - B** is set in the dialog section *Set Wind on Side*.

In the dialog section *Load Case to Generate*, we deactivate both load cases *w-* and *w+*. As described in chapter 2.3, page 7, we want to analyze only the positive external pressure coefficients. The load of *LC w+* will be generated for **LC4**.

Again, we want to create *Member loads*, but the monopitch roof beams do not make a contribution to the load transfer. We use the [Pick Member] button shown on the left, available in the dialog section *Remove Influence from*, to select one of the horizontal beams graphically (member **8** or **5**). The diagonal member **12** is automatically excluded from the load transfer.

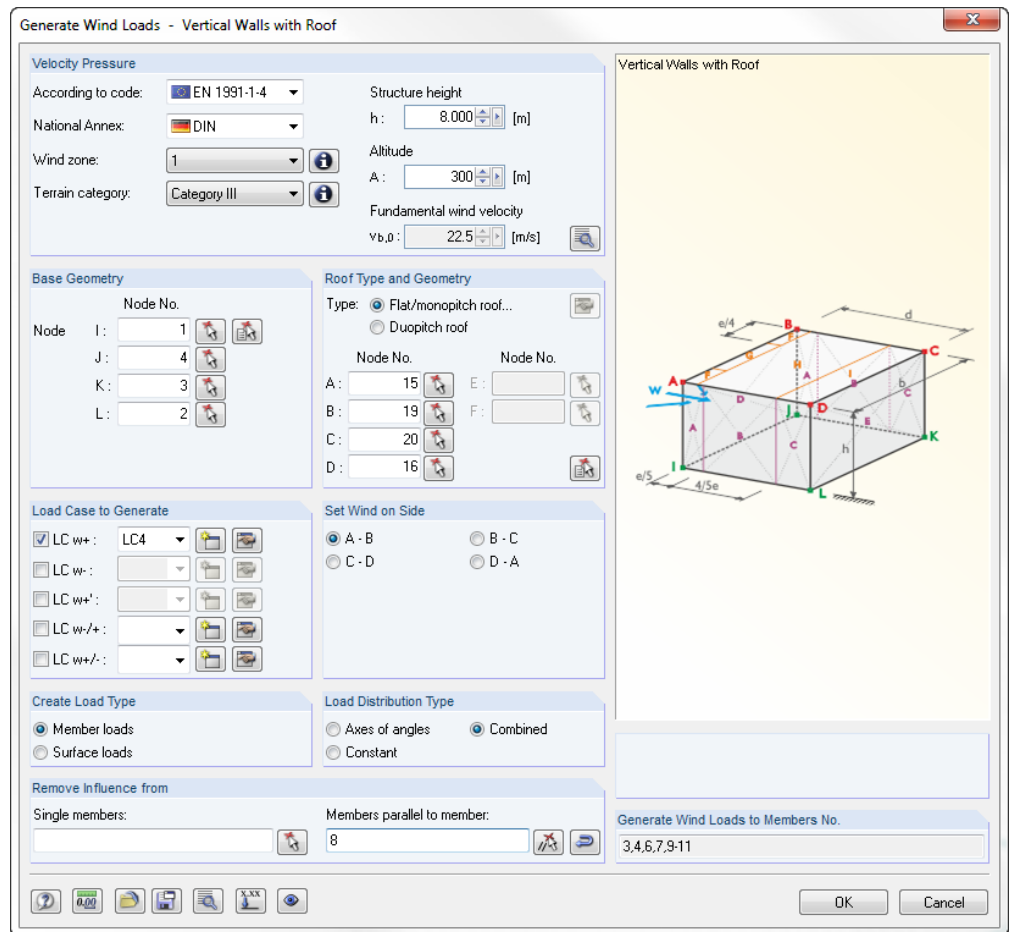
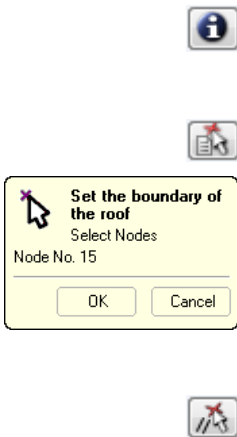


Figure 5.17: Dialog box *Generate Wind Loads - Vertical Walls with Roof*

After clicking [OK] a dialog box appears showing us information about the generation data. We click [OK] to confirm the dialog box. Now, we can see the wind loads displayed as surface loads on the model.



In addition, we can use the option *Disconnect Generated Loads* available in the load shortcut menu to make the member loads visible. However, we undo this operation immediately.

### 5.4.2 Column Loads

The loads applied to the lower part of the structure will be defined manually.

#### Defining a uniform member load

The wind suction acts on the column at the high eaves side with a constant value.



We select column member 1 with a mouse click and open the dialog box shown in Figure 5.18 with the button [New Member Load].

The *Load Direction* is globally related to the true member length in **YL**. The wind load component apportioned to the column is **1.5 kN/m**. We enter the value as *Load Parameter*.

We click [OK]. Now we see the member load represented on the column.

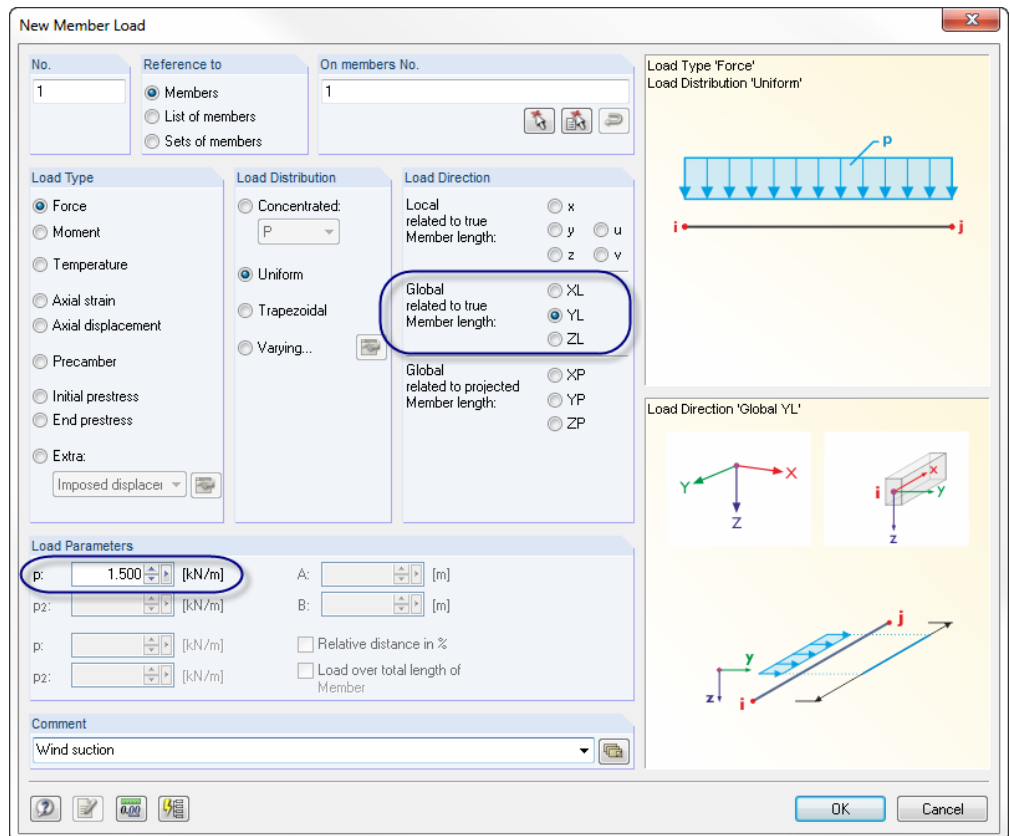


Figure 5.18: Defining wind suction as uniform member load

### Defining a trapezoidal member load

Due to a backfill set in a certain zone, the low eaves side reveals an asymmetrical load application area for wind pressure. Thus, the load distribution on the column is variable.



We select column member 2 and use the button [New Member Load] to open the dialog box *New Member Load* again.

Again, the *Load Direction* is defined globally in **YL**, but the *Load Distribution* is **Trapezoidal**. With this setting two *Load Parameters* become accessible: We enter **0.5** kN/m for the member start  $p_1$  and **3** kN/m for the member end  $p_2$ . (We defined the columns from bottom to top; thus the member start is equal to the column base).

As the *Load* acts over *total length of Member*, we select the corresponding check box.

We click [OK]. Now we see the member load represented on the second column.

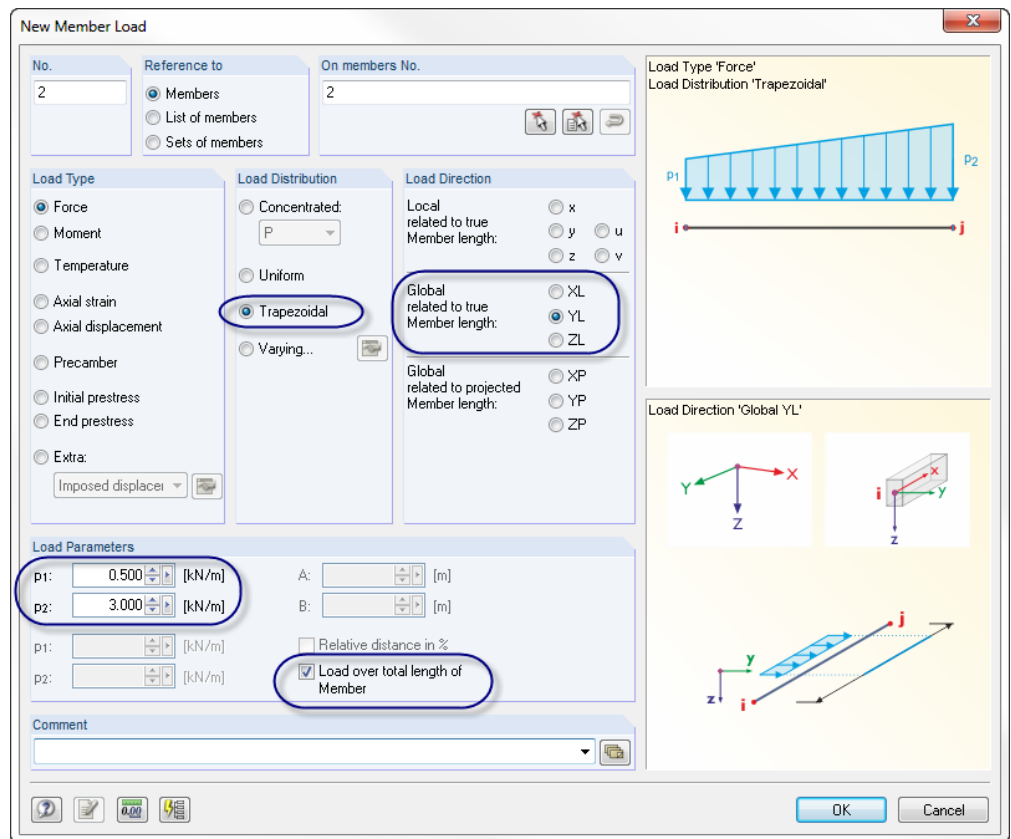


Figure 5.19: Defining wind pressure as trapezoidal member load

Now, the RFEM graphic showing the generated and manually defined wind loads should look like the figure below.

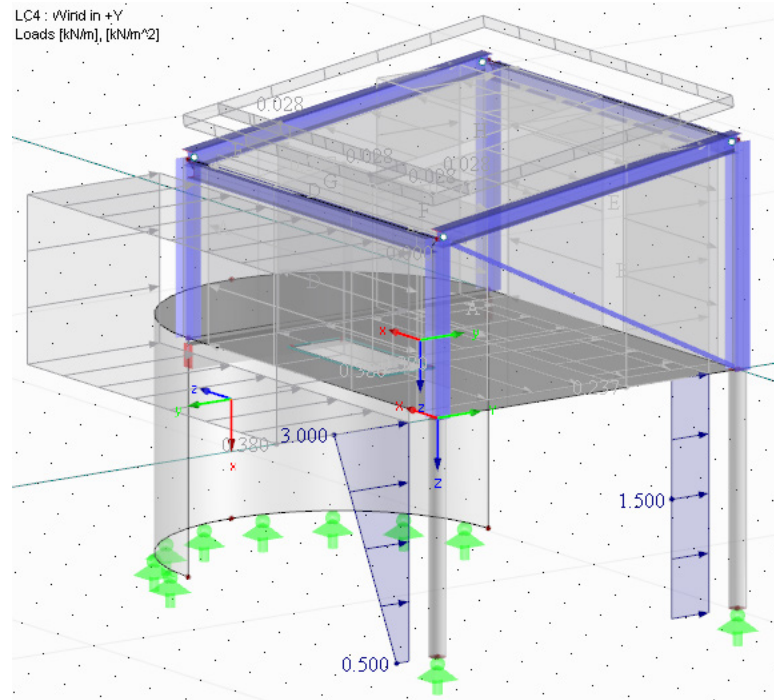


Figure 5.20: Wind loads

## 5.5 Load Case 5: Imperfection

In the final load case we define imperfections for the columns that are stressed by axial force.

This time, we use the *Data* navigator to create a new load case: We right-click the entry *Load Cases* to open the shortcut menu, and then we select *New Load Case*.

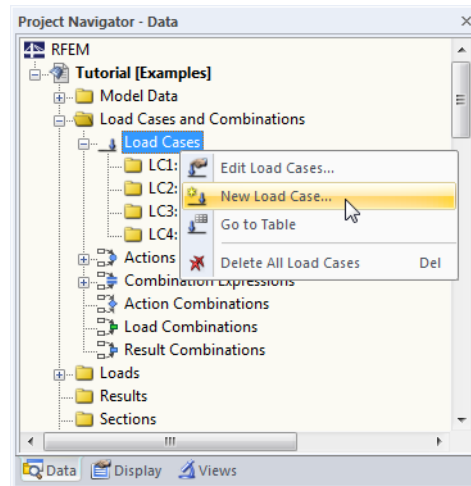


Figure 5.21: Shortcut menu *Load Cases*

We choose **Imperfection towards +Y** from the *Load Case Description* list. The *Action Type* changes automatically to **Imp Imperfection**.

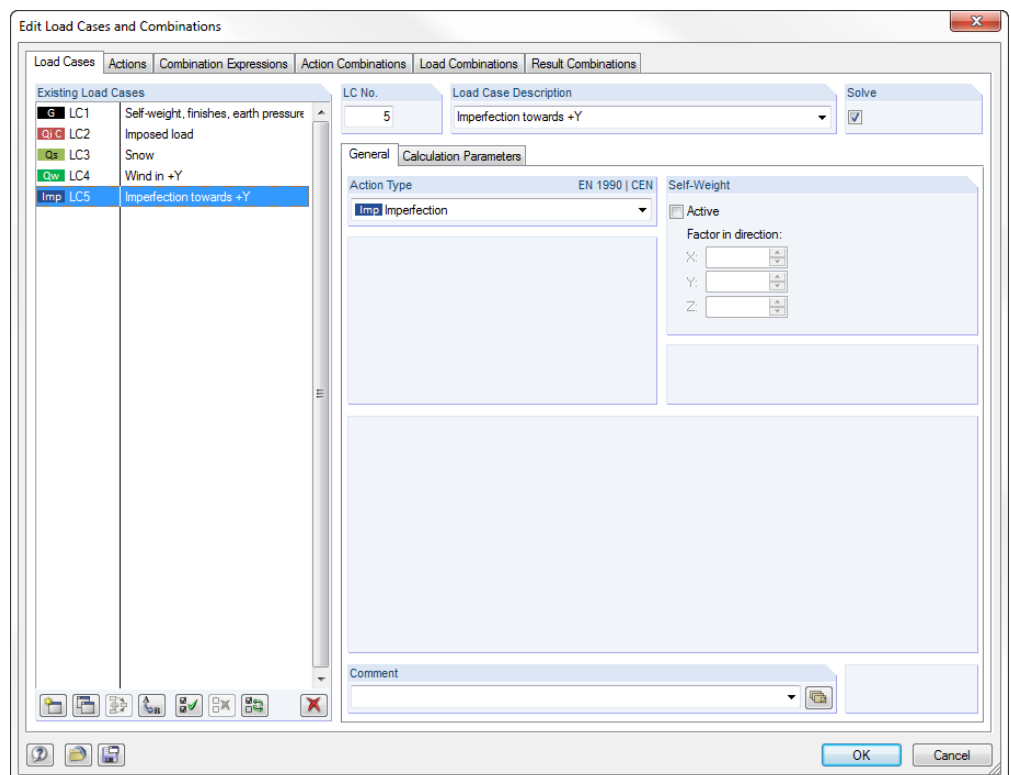
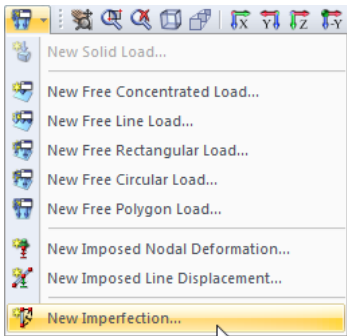


Figure 5.22: Dialog box *Edit Load Cases and Combinations*, tab *Load Cases*

We close the dialog box by clicking the [OK] button.

### 5.5.1 Steel Columns

We click the toolbar button [New Polygon Load] to open its list menu where we select the entry *New Imperfection*. The following dialog box opens.



List button for loads

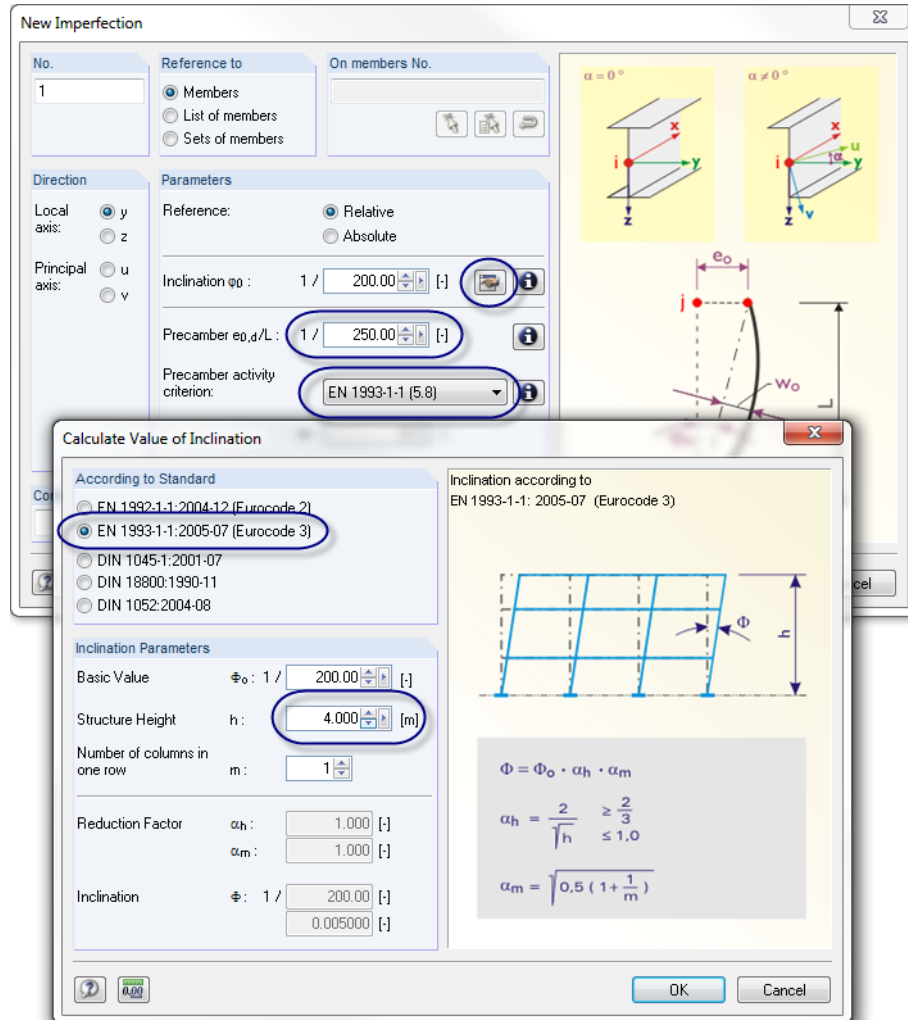


Figure 5.23: Dialog box *New Imperfection*

We want to apply the imperfection in *Direction* of the column axes **y**, which is the direction of the 'weak' member axis that is parallel aligned with the global axis Y in our example.



To enter the *Inclination*  $\varphi_0$ , we use the [Edit] button that opens the dialog box *Calculate Value of Inclination*. We set *Standard* **EN 1993-1-1** and change the *Structure Height* to **4 m**. We click [OK] and return to the initial dialog box.

According to EN 1993-1-1, table 5.1, we have to apply a *Pecamber*  $e_{0,d}/L$  of **1/250** (see chapter 2.3, page 7) for the buckling curve b of cross-sections HE A 300.

Then, we change the *activity criterion* for the precamber to **EN 1993-1-1 (5.8)** and confirm the dialog box with [OK].



Finally, we click the four steel columns with the member numbers **6, 4, 9** and **7** to assign the imperfections.

We quit the function with the [Esc] key or a right-click.

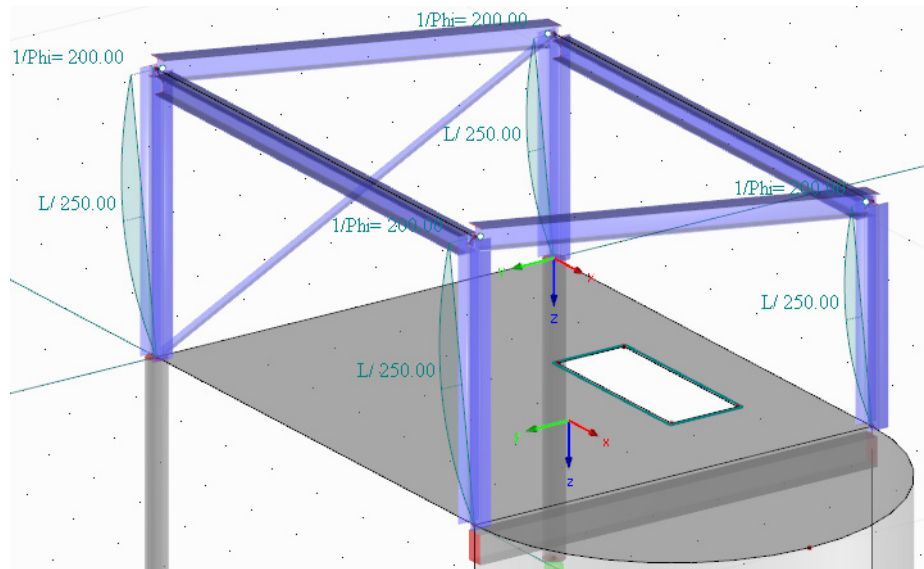


Figure 5.24: Assigning imperfections to steel columns

### 5.5.2 Concrete Columns



Once again, we open the dialog box *New Imperfection* to define the inclination of the concrete columns.

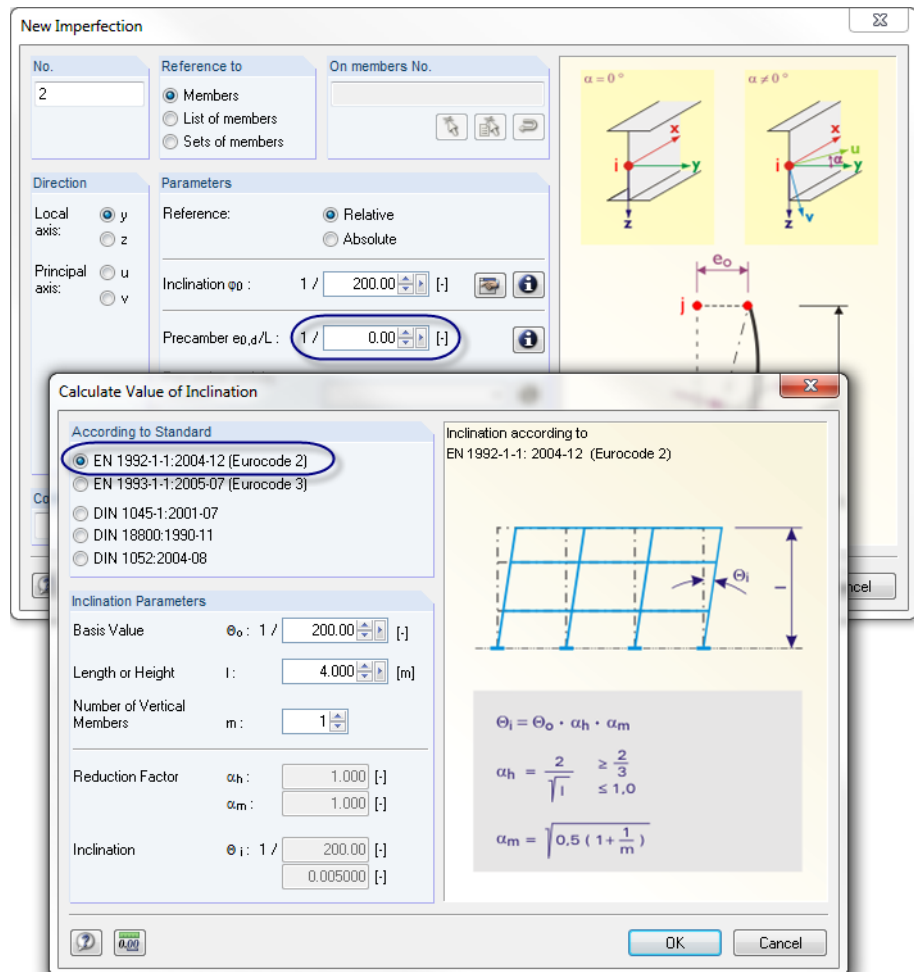


Figure 5.25: Dialog box *New Imperfection*



Again, we define the *Inclination*  $\varphi_0$  with the [Edit] button: In the dialog box *Calculate Value of Inclination*, we set *Standard EN 1992-1-1*.

As we do not have to consider precambers in accordance with Eurocode 2, we specify the *Precamber*  $e_{0,d}/L$  with zero (1/0.00) in the initial dialog box.



We check if settings are defined as shown in the figure above. Then, we confirm the dialog box and click both concrete columns with the member numbers **1** and **2** to assign the imperfections.

## 5.6 Checking Load Cases

All five load cases have been completely entered. It is recommended to [Save] the input now.

We can check each load case quickly in the graphics: The buttons [◀] and [▶] in the toolbar allow us to select previous and subsequent load cases.

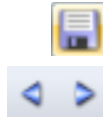


Figure 5.26: Browsing the load cases

The loading's graphical input is also reflected in both the *Data* navigator tree and the tables. We can access the load data in table 3. *Loads* which can be set with the button shown on the left.



Again, graphic and tables are interactive: To find a load in the table, for example an imperfection, we set table 3.13 *Imperfections*, and then we select the load in the work window. We see that the pointer jumps into the corresponding row of the table (see the following figure).



LC5: Imperfection towards +Y  
Loads [-]

3.13 Imperfections

No.	Reference to	On Members No.	Direction	Reference	Inclination 1/φ <sub>0</sub> [°]	Precamber l/w <sub>0</sub> [°]	Activity Criterion	Apply w <sub>0</sub> from s <sub>0</sub> [°]	Comment
1	Members	4,6,7,9	y	Relative	200.00	250.00	EN 1993-1-1 (5.8)		
2	Members	1,2	y	Relative	200.00	0.00			
3									
4									
5									

Free Circular Loads | Free Polygon Loads | Imposed Nodal Deformations | Imposed Line Displacements | Imperfections

Figure 5.27: Interaction between graphic and load tables

# 6. Combination of Actions

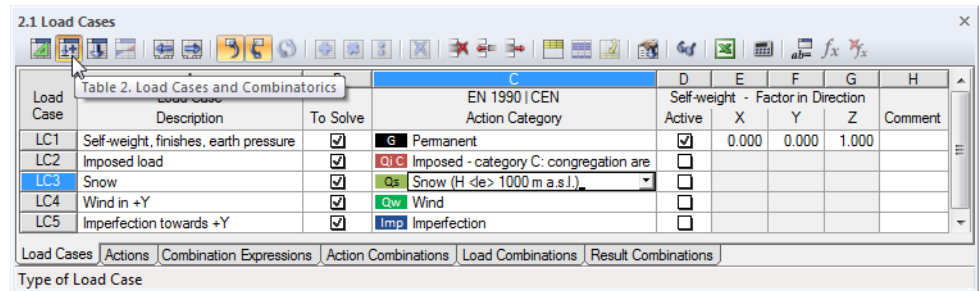
We combine the load cases according to EN 1990. We take advantage of the generator integrated in the program to superimpose the actions with the required partial safety factors and combination coefficients. The relevant conditions have already been created when the model was defined in the dialog box *General Data* where we have selected the option *Create combinations automatically* (see Figure 3.1, page 8).

The *Action Type* defined for the load cases (see Figure 5.22, page 62) determines the way how load cases are combined in different design situations.

## 6.1 Checking Actions

The load cases must be assigned to *Actions* which will be superimposed in accordance with regulations. Actions represent independent influence values that arise from different origins. The correlation existing between them may be neglected with regard to the reliability of the structural system.

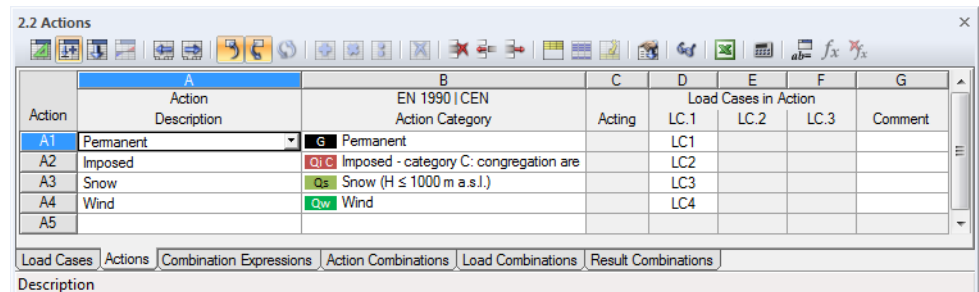
Load cases, actions and combinations are managed in the dialog box *Edit Load Cases and Combinations* (see Figure 5.22, page 62) as well as in tables number 2. We can access these tables by clicking the table button shown on the left. Table 2.1 *Load Cases* shows us the five load cases with the selected action categories in a clear overview.

Load Case	Description	To Solve	Action Category	Self-weight	Factor in Direction			
				Active	X	Y	Z	Comment
LC1	Self-weight, finishes, earth pressure	<input checked="" type="checkbox"/>	G Permanent	<input checked="" type="checkbox"/>	0.000	0.000	1.000	
LC2	Imposed load	<input checked="" type="checkbox"/>	Q <sub>i</sub> C Imposed - category C: congregation are	<input checked="" type="checkbox"/>				
LC3	Snow	<input checked="" type="checkbox"/>	Q <sub>s</sub> Snow (H <= 1000 m a.s.l.)	<input type="checkbox"/>				
LC4	Wind in +Y	<input checked="" type="checkbox"/>	Q <sub>w</sub> Wind	<input type="checkbox"/>				
LC5	Imperfection towards +Y	<input checked="" type="checkbox"/>	Imp Imperfection	<input type="checkbox"/>				

Figure 6.1: Table 2.1 *Load Cases*

The subsequent table 2.2 *Actions* shows us the load cases that are contained in the individual actions. Each load case of our example is assigned to another action. However, if we had defined several wind load cases for different directions, they all would be listed in the action *Wind*.



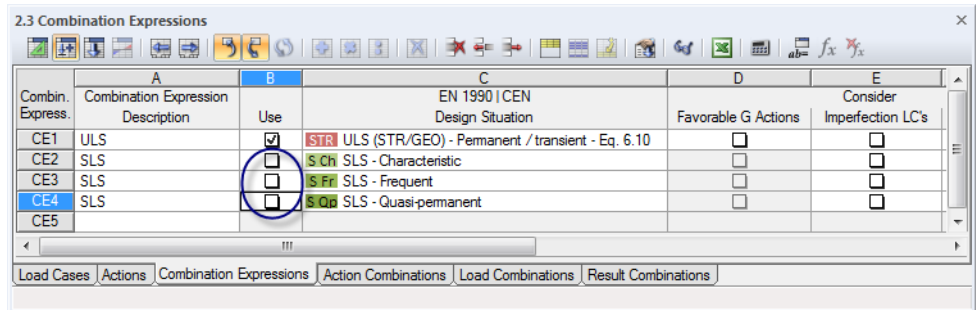
Action	Description	Action Category	Acting	LC.1	LC.2	LC.3	Comment
A1	Permanent	G Permanent	<input checked="" type="checkbox"/>	LC1			
A2	Imposed	Q <sub>i</sub> C Imposed - category C: congregation are	<input checked="" type="checkbox"/>	LC2			
A3	Snow	Q <sub>s</sub> Snow (H ≤ 1000 m a.s.l.)	<input checked="" type="checkbox"/>	LC3			
A4	Wind	Q <sub>w</sub> Wind	<input checked="" type="checkbox"/>	LC4			
A5			<input type="checkbox"/>				

Figure 6.2: Table 2.2 *Actions*

The imperfections are missing in this table because they do not represent "real" actions.

## 6.2 Defining Combination Expressions

In accordance with EN 1990, we have to combine the actions for the ultimate and the serviceability limit state design according to certain rules. Table 2.3 *Combination Expressions* shows us which limit states are set to be analyzed.



Combin. Express.	A Combination Expression Description	B Use	C EN 1990   CEN Design Situation	D Favorable G Actions	E Consider Imperfection LC's
CE1	ULS	<input checked="" type="checkbox"/>	STR ULS (STR/GEO) - Permanent / transient - Eq. 6.10 GEO ULS (STR/GEO) - Permanent / transient - Eq. 6.10	<input type="checkbox"/>	<input type="checkbox"/>
CE2	SLS	<input type="checkbox"/>	Ch SLS - Characteristic	<input type="checkbox"/>	<input type="checkbox"/>
CE3	SLS	<input type="checkbox"/>	Fr SLS - Frequent	<input type="checkbox"/>	<input type="checkbox"/>
CE4	SLS	<input type="checkbox"/>	Qu SLS - Quasi-permanent	<input type="checkbox"/>	<input type="checkbox"/>
CE5					

Figure 6.3: Table 2.3 *Combination Expressions*

Only the ultimate limit state (ULS) is relevant for our example. Therefore, we remove the three check marks in the table column *Use* for the combination rules of the serviceability limit states (SLS).

Now, we use the navigator shortcut menu to open the dialog box *Edit Load Cases and Combinations*. We want to *Edit* the parameters of the combination expression *CE1*.

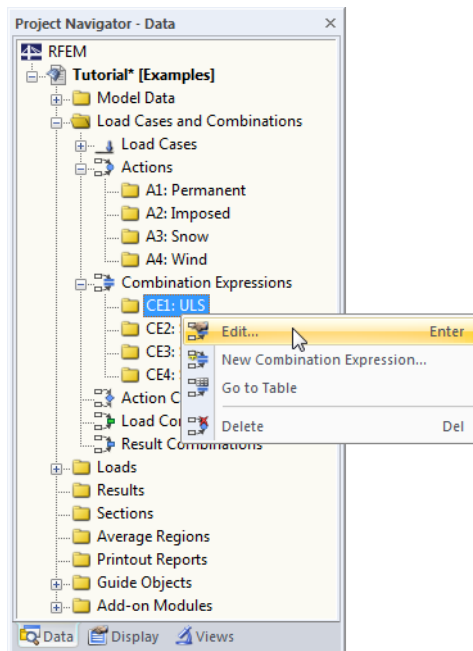


Figure 6.4: Navigator shortcut menu of combination expressions



To read describing comments informing us about the combination expression for the limit states STR and GEO, we use the [Info] button available in this dialog box (see figure below).

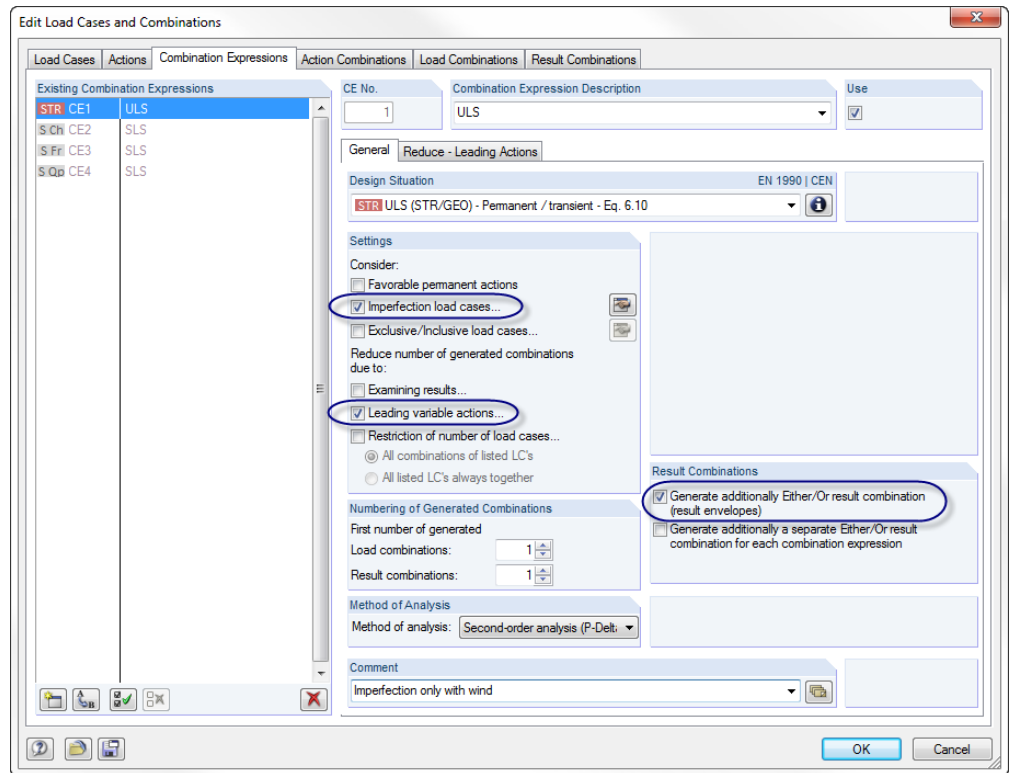


Figure 6.5: Dialog box *Edit Load Cases and Combinations*, tab *Combination Expressions*

In the dialog section *Settings*, we activate the option **Imperfection load cases** to *Consider* the imperfections for the generation of combinations. When the check box is ticked, the following dialog box opens.

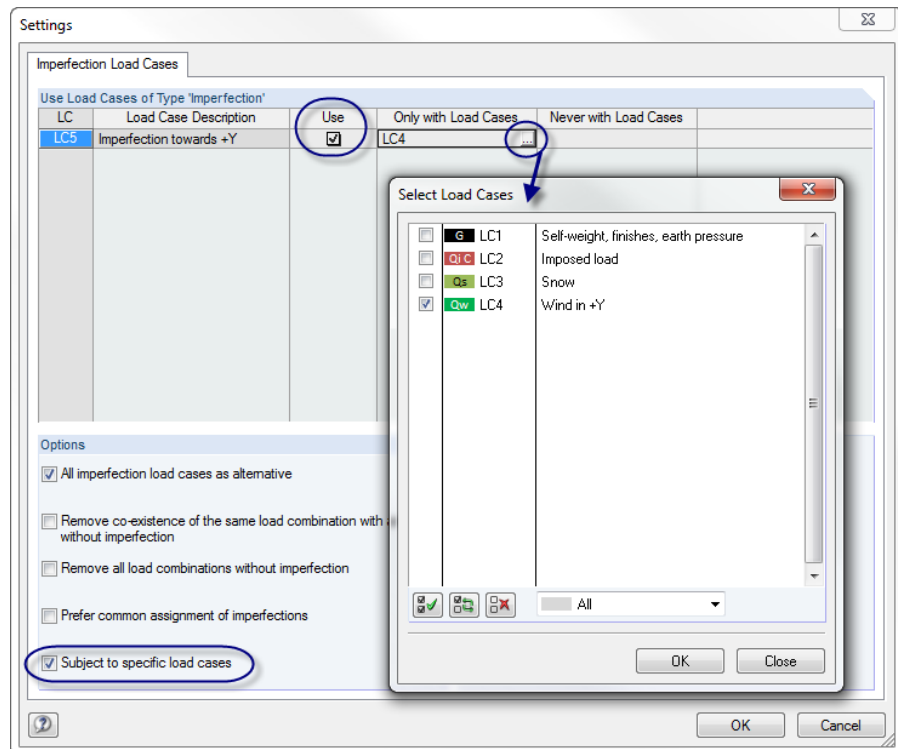


Figure 6.6: Dialog box *Settings* for imperfection load cases

We tick the check box in the table column *Use* so that LC4 can be taken into account.

Then, we activate the function **Subject to specific load cases** in the dialog section *Options*. Two more table columns will be enabled where we click into the field *Only with Load Cases*. A button [...] appears at the field end which we can use to access the dialog box *Select Load Cases*. We select **LC4 Wind in +Y**. In this way, RFEM will consider imperfections only in combinations which include wind load cases.

We confirm the dialog boxes shown in Figure 6.6 with [OK].

In the dialog section *Settings* of the initial dialog box, we *Reduce number of generated combinations* by taking into account only **Leading variable actions**. When we tick the option, another tab is enabled in the dialog box.

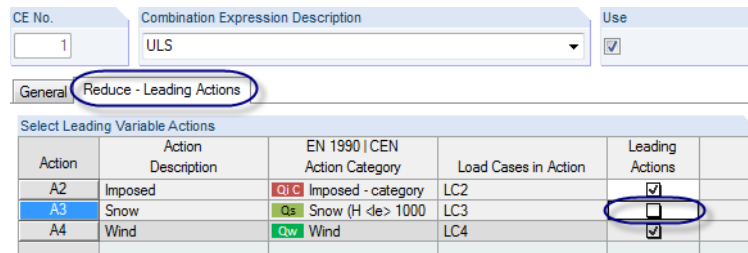
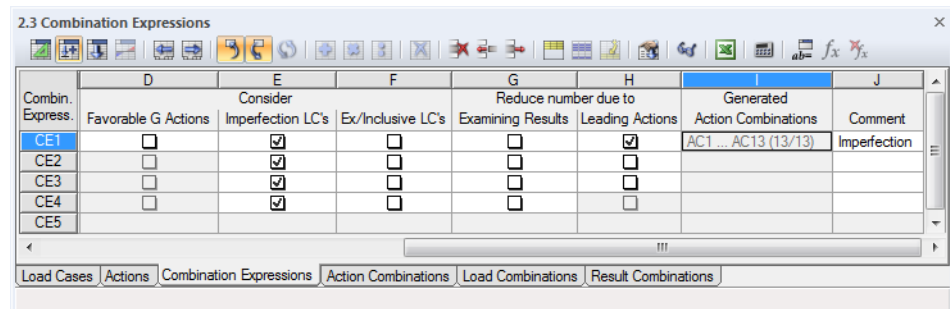


Figure 6.7: Defining leading actions in dialog tab *Reduce - Leading Actions*

In the tab *Reduce - Leading Actions*, we clear the check box for action **A3** because we want to superimpose the *Snow* load case as secondary variable load only. Thus, the number of generated combinations will be reduced.

Before we close the dialog box *Edit Load Cases and Combinations* with [OK], we make sure that the option **Generate additionally Either/Or result combination** in the *General* tab is checked as well. This result combination provides the extreme values from the results of all load combinations (envelope).

RFEM creates the action combinations. When we click table tab 2.4 *Action Combinations* and return to table 2.3, the column *Generated Action Combinations* informs us that 13 combinations were created.



Combin. Express.	D Favorable G Actions	E Consider Imperfection LCs	F Ex/Inclusive LCs	G Reduce number due to Examining Results	H Leading Actions	J Generated Action Combinations	Comment
CE1	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	AC1 ... AC13 (13/13)	Imperfection
CE2	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>		
CE3	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>		
CE4	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>		
CE5	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>		

Figure 6.8: Table 2.3 *Combination Expressions*, table column *Generated Action Combinations*

### 6.3 Creating Action Combinations

RFEM creates automatically 13 action combinations (see Figure 6.8) which are listed and sorted by actions in table 2.4 *Action Combinations*.

Action Combin.	Action Combination Description	Use	EN 1990   CEN Design Situation	Action.1 Factor	Action.1 No.	Action.2 Factor	Action.2 No.	Action.3 Factor	Action.3 No.	Action.4 Factor	Action.4 No.
AC1	1.35G	<input checked="" type="checkbox"/>	STR ULS (STR/G)	1.35	G A1						
AC2	1.35G + 1.50QiC	<input checked="" type="checkbox"/>	STR ULS (STR/G)	1.35	G A1	1.50	Qi C A2				
AC3	1.35G + 1.50QiC + 0.75Qs	<input checked="" type="checkbox"/>	STR ULS (STR/G)	1.35	G A1	1.50	Qi C A2	0.75	Qs A3		
AC4	1.35G + 1.50QiC + 0.75Qs + 0.90Qw	<input checked="" type="checkbox"/>	STR ULS (STR/G)	1.35	G A1	1.50	Qi C A2	0.75	Qs A3	0.90	Qw A4
AC5	1.35G + 1.50QiC + 0.90Qw	<input checked="" type="checkbox"/>	STR ULS (STR/G)	1.35	G A1	1.50	Qi C A2	0.90	Qw A4		
AC6	1.35G + 1.50Qs	<input type="checkbox"/>	STR ULS (STR/G)	1.35	G A1	1.50	Qs A3				
AC7	1.35G + 1.05QiC + 1.50Qs	<input type="checkbox"/>	STR ULS (STR/G)	1.35	G A1	1.05	Qi C A2	1.50	Qs A3		
AC8	1.35G + 1.05QiC + 1.50Qs + 0.90Qw	<input type="checkbox"/>	STR ULS (STR/G)	1.35	G A1	1.05	Qi C A2	1.50	Qs A3	0.90	Qw A4
AC9	1.35G + 1.50Qs + 0.90Qw	<input type="checkbox"/>	STR ULS (STR/G)	1.35	G A1	1.50	Qs A3	0.90	Qw A4		
AC10	1.35G + 1.50Qw	<input checked="" type="checkbox"/>	STR ULS (STR/G)	1.35	G A1	1.50	Qw A4				
AC11	1.35G + 1.05QiC + 1.50Qw	<input checked="" type="checkbox"/>	STR ULS (STR/G)	1.35	G A1	1.05	Qi C A2	1.50	Qw A4		
AC12	1.35G + 1.05QiC + 0.75Qs + 1.50Qw	<input checked="" type="checkbox"/>	STR ULS (STR/G)	1.35	G A1	1.05	Qi C A2	0.75	Qs A3	1.50	Qw A4
AC13	1.35G + 0.75Qs + 1.50Qw	<input checked="" type="checkbox"/>	STR ULS (STR/G)	1.35	G A1	0.75	Qs A3	1.50	Qw A4		

Figure 6.9: Table 2.4 *Action Combinations*

This overview corresponds to the presentation of actions described in the standards. By selecting the check box in the *Use* column we can define the action combinations which will be considered for the generation of load combinations. Because we have applied action *Qs* (snow) to be an accompanying variable action only, we can now see that corresponding action combinations where action *Qs* is the leading one are disabled.

### 6.4 Creating Load Combinations

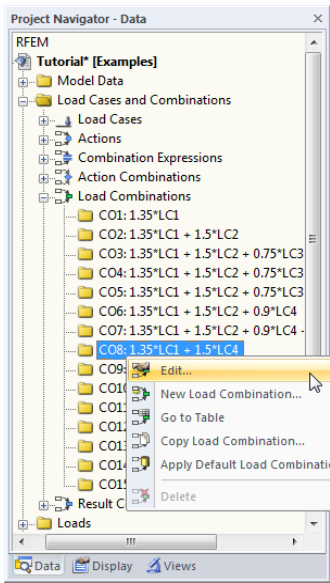
15 load combinations are automatically created from the nine relevant action combinations (see Figure 6.9). The result is listed in the subsequent table 2.5 *Load Combinations*.

Load Combin.	Load Combination Description	To Solve	Factor	LC.1 No.	LC.2 Factor	LC.2 No.	LC.3 Factor	LC.3 No.	LC.4 Factor	LC.4 No.	LC.5 Factor	LC.5 No.
CO1	1.35*LC1	<input checked="" type="checkbox"/>	1.35	G LC1								
CO2	1.35*LC1 + 1.5*LC2	<input checked="" type="checkbox"/>	1.35	G LC1	1.50	Qi C LC2						
CO3	1.35*LC1 + 1.5*LC2 + 0.75*LC3	<input checked="" type="checkbox"/>	1.35	G LC1	1.50	Qi C LC2	0.75	Qs LC3				
CO4	1.35*LC1 + 1.5*LC2 + 0.75*LC3 + 0.9*	<input checked="" type="checkbox"/>	1.35	G LC1	1.50	Qi C LC2	0.75	Qs LC3	0.90	Qw LC4		
CO5	1.35*LC1 + 1.5*LC2 + 0.75*LC3 + 0.9*	<input checked="" type="checkbox"/>	1.35	G LC1	1.50	Qi C LC2	0.75	Qs LC3	0.90	Qw LC4	1.00	Imp LC5
CO6	1.35*LC1 + 1.5*LC2 + 0.9*LC4	<input checked="" type="checkbox"/>	1.35	G LC1	1.50	Qi C LC2	0.90	Qw LC4				
CO7	1.35*LC1 + 1.5*LC2 + 0.9*LC4 + LC5	<input checked="" type="checkbox"/>	1.35	G LC1	1.50	Qi C LC2	0.90	Qw LC4	1.00	Imp LC5		
CO8	1.35*LC1 + 1.5*LC4	<input checked="" type="checkbox"/>	1.35	G LC1	1.50	Qw LC4						
CO9	1.35*LC1 + 1.5*LC4 + LC5	<input checked="" type="checkbox"/>	1.35	G LC1	1.50	Qw LC4	1.00	Imp LC5				
CO10	1.35*LC1 + 1.05*LC2 + 1.5*LC4	<input checked="" type="checkbox"/>	1.35	G LC1	1.05	Qi C LC2	1.50	Qw LC4				
CO11	1.35*LC1 + 1.05*LC2 + 1.5*LC4 + LC5	<input checked="" type="checkbox"/>	1.35	G LC1	1.05	Qi C LC2	1.50	Qw LC4	1.00	Imp LC5		
CO12	1.35*LC1 + 1.05*LC2 + 0.75*LC3 + 1.5	<input checked="" type="checkbox"/>	1.35	G LC1	1.05	Qi C LC2	0.75	Qs LC3	1.50	Qw LC4		
CO13	1.35*LC1 + 1.05*LC2 + 0.75*LC3 + 1.5	<input checked="" type="checkbox"/>	1.35	G LC1	1.05	Qi C LC2	0.75	Qs LC3	1.50	Qw LC4	1.00	Imp LC5
CO14	1.35*LC1 + 0.75*LC3 + 1.5*LC4	<input checked="" type="checkbox"/>	1.35	G LC1	0.75	Qs LC3	1.50	Qw LC4				
CO15	1.35*LC1 + 0.75*LC3 + 1.5*LC4 + LC5	<input checked="" type="checkbox"/>	1.35	G LC1	0.75	Qs LC3	1.50	Qw LC4	1.00	Imp LC5		

Figure 6.10: Table 2.5 *Load Combinations*

Table columns D to M inform us about load cases including respective partial safety and combination factors.

Imperfections are contained, according to our specifications, only in combinations with wind actions *Qw*.



Shortcut menu *Load Combinations*

We use the navigator shortcut menu shown on the left to open the dialog box *Edit Load Cases and Combinations* where we want to look at the created load combinations.

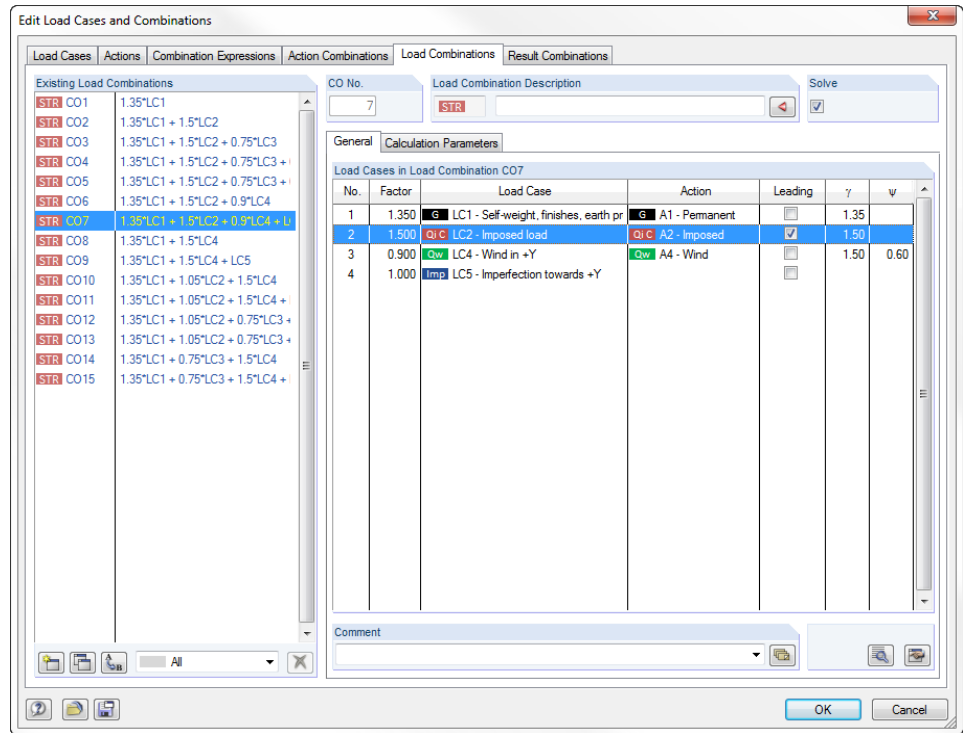


Figure 6.11: Dialog box *Edit Load Cases and Combinations*, tab *Load Combinations*

When we select the *Existing Load Combinations* one after the other in the list, we can see all load cases together with the respective partial safety and combination factors displayed in the dialog section to the right. Load cases which act as *Leading* within the combination are identified by a check mark.

To access the partial safety and combination factors, we use the [Details] button.

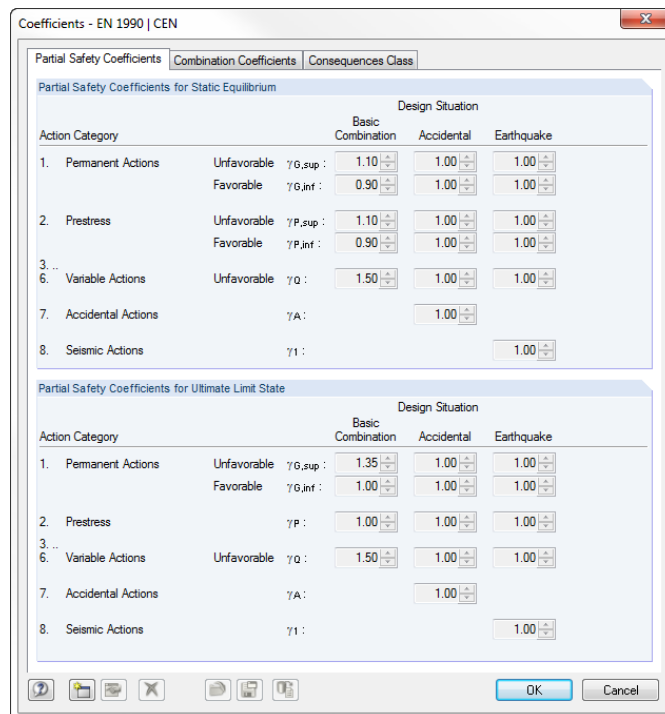


Figure 6.12: Dialog box *Coefficients*, tab *Partial Safety Coefficients*

Furthermore, we can use the *Calculation Parameters* tab to check the specifications applied by RFEM for the calculation of different load combinations.

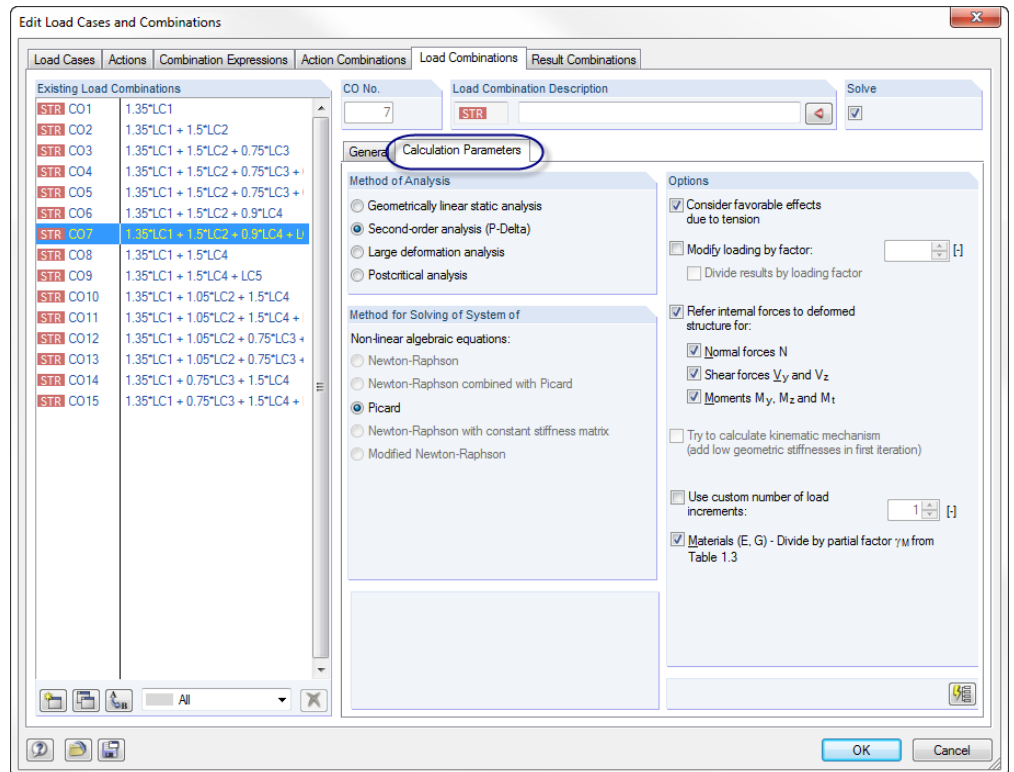


Figure 6.13: Checking the *Calculation Parameters* of a load combination

Basically, load combinations are analyzed non-linearly according to the *Method of Analysis for Second-order analysis*.

## 6.5 Checking Result Combinations

When we defined the combination expressions, we activated the option *Generate additionally Either/Or result combination* (see Figure 6.5, page 68) giving us information about the extreme values of all load combinations.

RFEM generates a results envelope from the load combinations. The definition criterion can be checked in the final tab of the dialog box *Edit Load Cases and Combinations* as well as in table 2.6 *Result Combinations*.

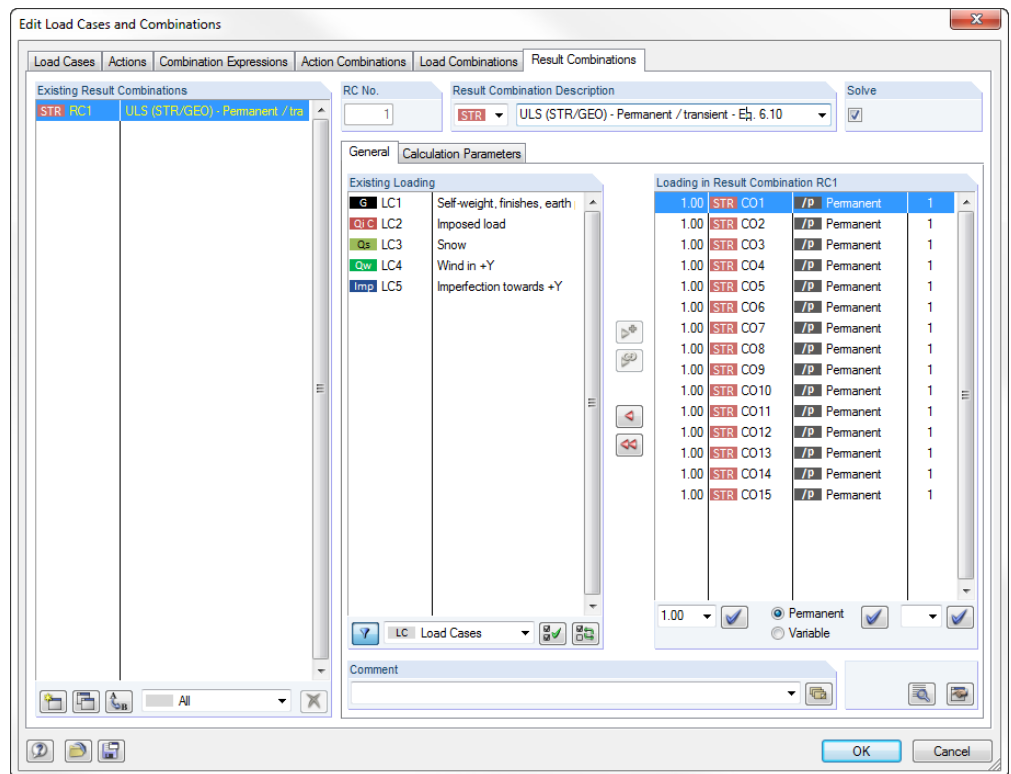


Figure 6.14: Dialog box *Edit Load Cases and Combinations*, tab *Result Combinations*

All load combinations are superimposed with factor *1.00* and the criterion *Permanent*. They all are assigned to group *1*, which means that they act alternatively.



Now, the superposition criteria is completely defined. We can save the input with the [Save] button.

# 7. Calculation

## 7.1 Checking Input Data



Before we calculate the model, we want RFEM to check our input. To open the corresponding dialog box,

we select **Plausibility Check** on the **Tools** menu.

The dialog box *Plausibility Check* opens where we define the following settings.

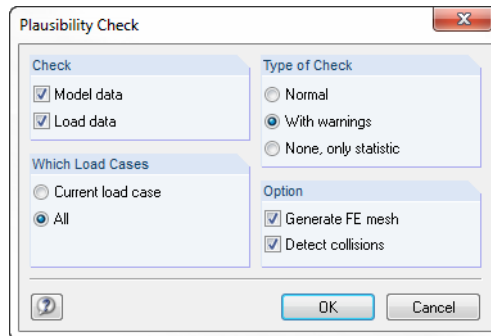


Figure 7.1: Dialog box *Plausibility Check*

If no error is detected after clicking [OK], a corresponding message is displayed, including summary of model and load data.

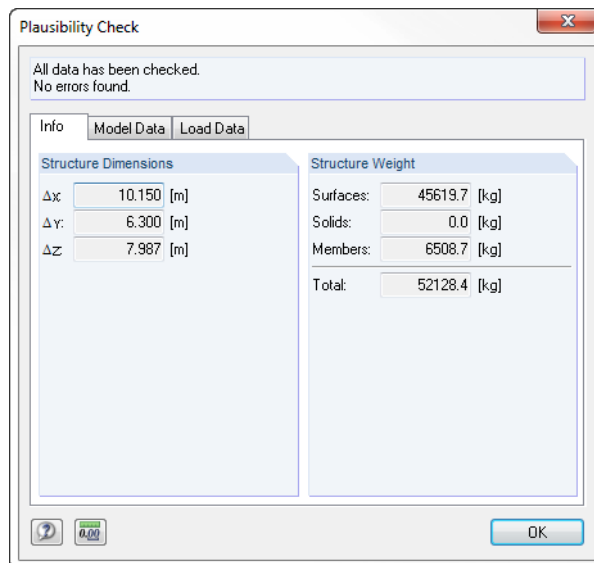


Figure 7.2: Result of plausibility check

We find more tools for checking the input by selecting

**Model Check** on the **Tools** menu.

## 7.2 Generating FE Mesh

### Generate the FE mesh

As we have selected the option *Generate FE mesh* in the dialog box *Plausibility Check* (see Figure 7.1), we have automatically generated a mesh with the standard mesh size of 50 cm. (We can modify the preset mesh size by selecting *FE Mesh Settings* on the *Calculate* menu.)

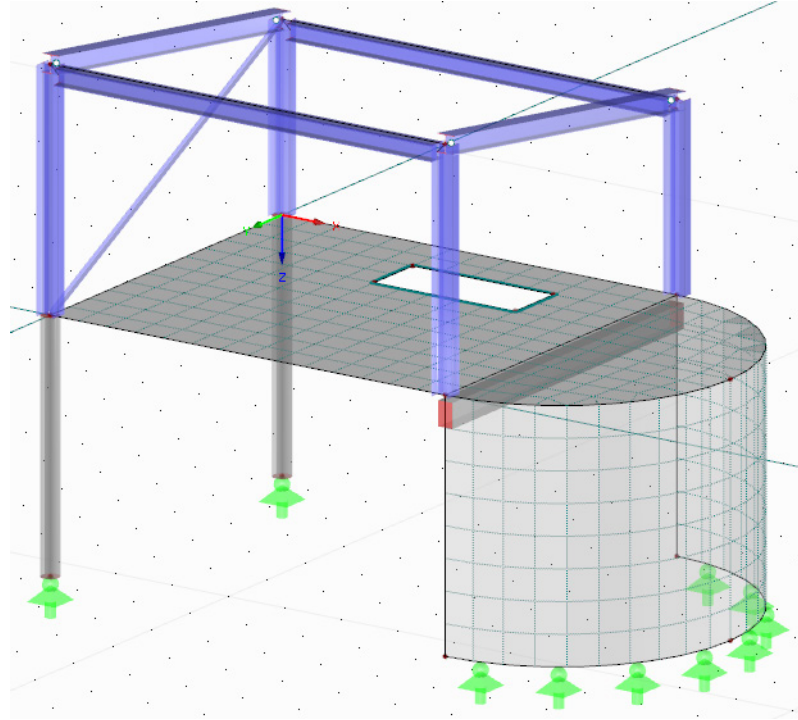


Figure 7.3: Model with generated FE mesh

### Create a FE mesh refinement

We define refinement areas for both ends of the downstand beam to generate a finer FE mesh.

We double-click node **3** to open the dialog box *Edit Node*. We change to the tab *FE Mesh* where we select the check box of the option **Available** (see Figure 7.4).

Since we have not yet defined any type of FE mesh refinement, the dialog box *New FE-Mesh Refinement* opens automatically.

It is not necessary to modify the default settings *Node - circular* as well as the proposed *Parameters*. The FE mesh will be deleted after confirming both dialog boxes with [OK].

A refinement area represented as spherical form is displayed on the selected node.

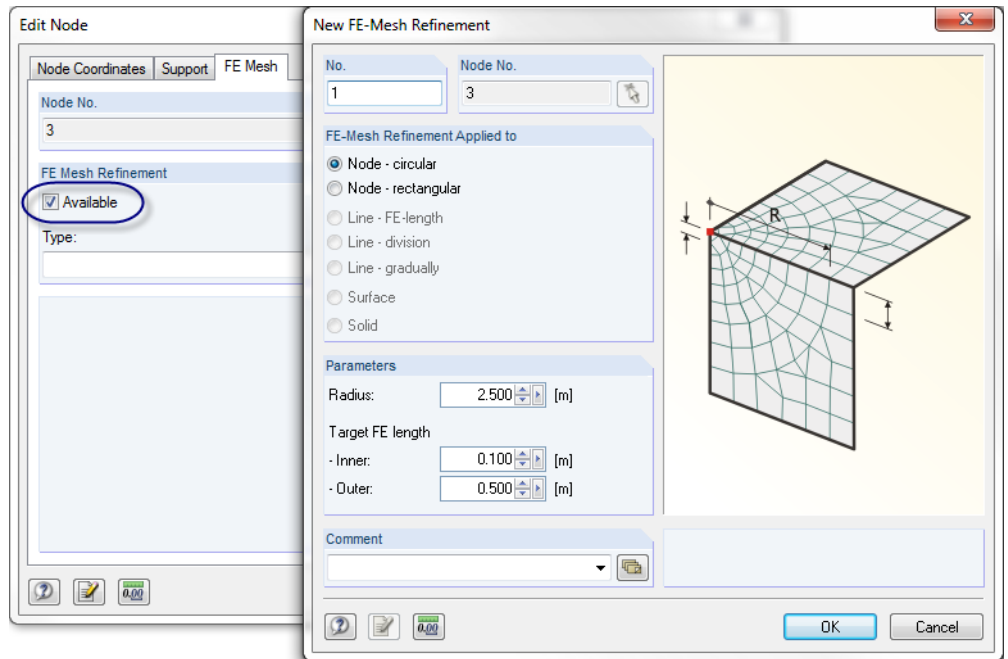


Figure 7.4: Dialog boxes *Edit Node* and *New FE-Mesh Refinement*

Now, we have to transfer the FE mesh refinement to the second end node of the downstand beam. In the *Data* navigator, we double-click entry 1 listed below the *FE Mesh Refinements* to access the dialog box *Edit FE-Mesh Refinement*.

We click the [Select] button in the dialog section *Node No.* to select the second node of the rib graphically in the work window.

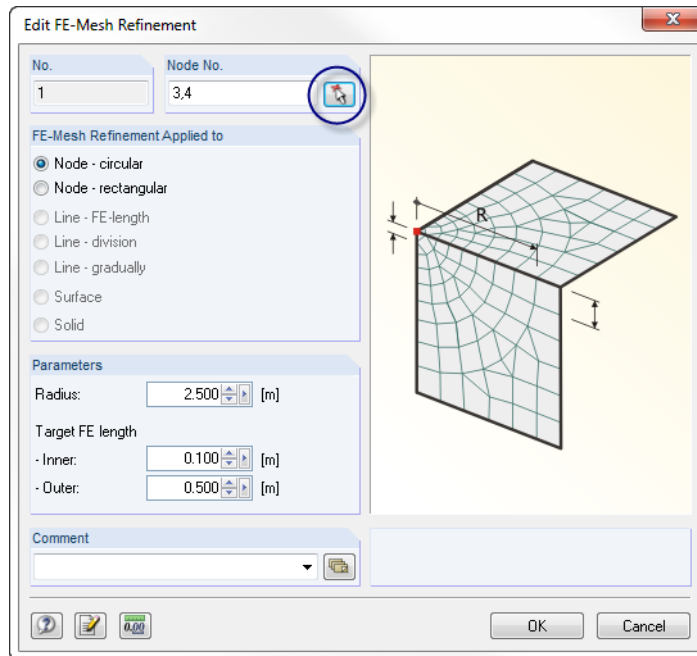
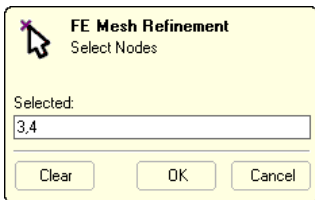


Figure 7.5: Defining the second node graphically by using the [Select] function

We close all dialog boxes with [OK].

We create the mesh once again by selecting **Generate FE Mesh** in the **Calculate** menu. Then, we check the refinement areas.

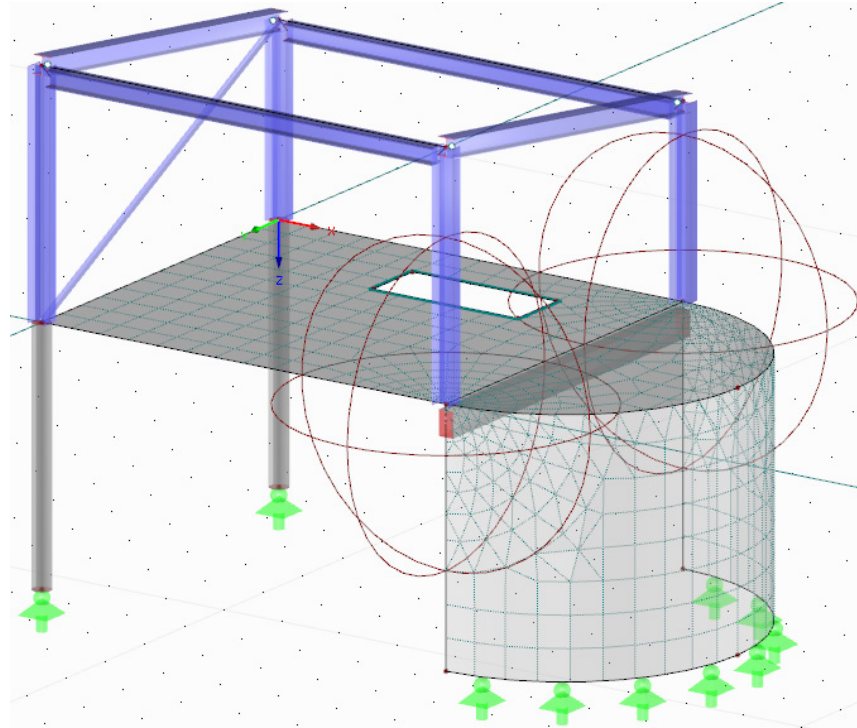


Figure 7.6: FE mesh with refinement areas

## 7.3 Calculating Model



To start the calculation, we select **Calculate All** on the **Calculate** menu or use the toolbar button shown on the left.

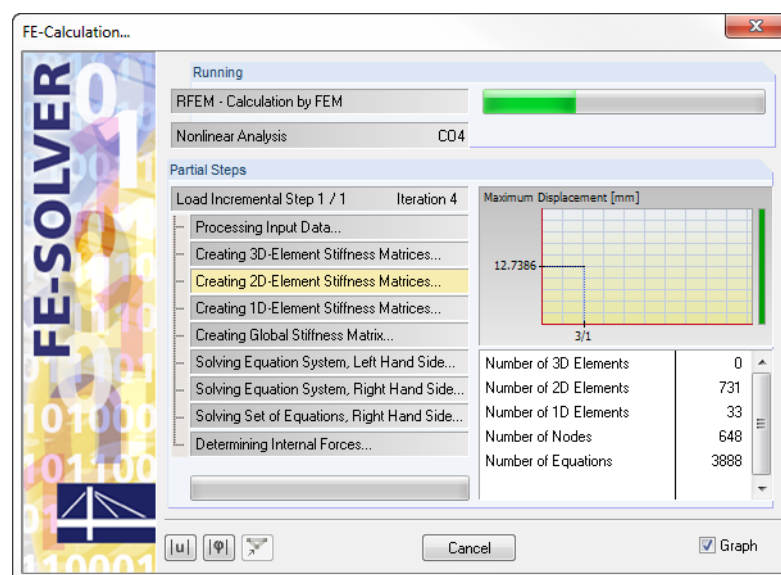


Figure 7.7: Calculation process

# 8. Results

## 8.1 Graphical Results



As soon as the calculation is complete, RFEM shows the deformations of the active load case.

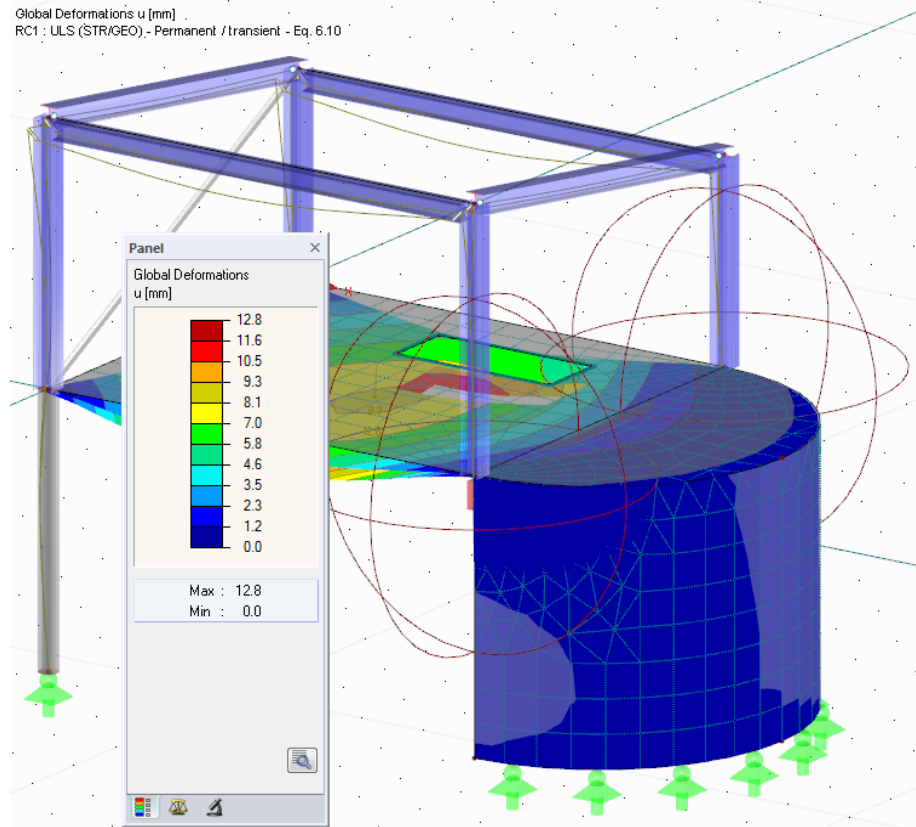


Figure 8.1: Graphical representation of deformations for result combination RC1

### Selecting load cases and load combinations



We can use the toolbar buttons [◀] and [▶] (to the right of the load case list) to change among the results of load cases, load combinations and result combinations. We already know those buttons from checking the load cases. It is also possible to select a specific load case or combination in the list.

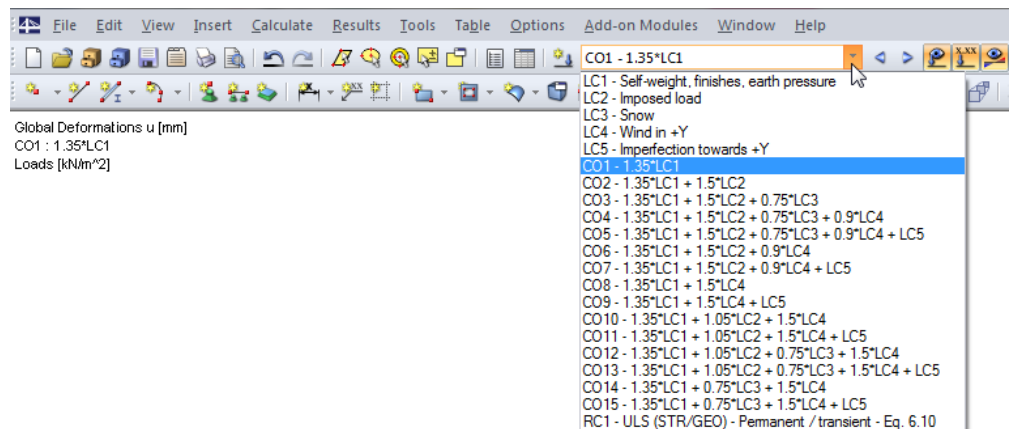


Figure 8.2: Load case list in the toolbar

### Selecting results in the navigator



A new navigator has appeared, managing all result types for the graphical display. We can access the *Results* navigator when the results display is active. We can switch the results display on and off in the *Display* navigator, but we can also use the toolbar button [Show Results] shown on the left.

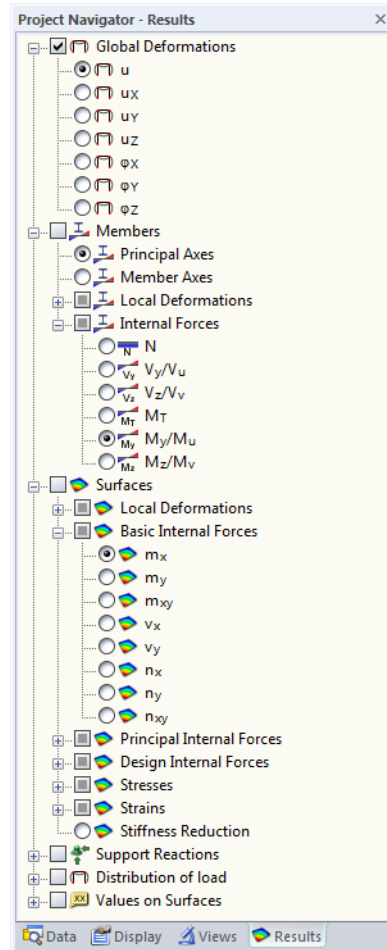
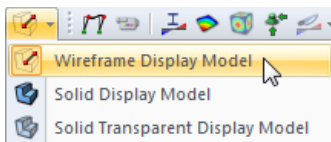


Figure 8.3: Results navigator

We can see check boxes placed in front of each result category (for example *Global Deformations*, *Members*, *Surfaces*, *Support Reactions*). When we select a box, we see the corresponding deformation or internal force displayed in RFEM. In front of the entries listed in the categories we see even more check boxes by which we can set the type of results to be displayed.

Finally, we can browse the single load cases and load combinations. The variety of result categories allows us to display deformations, internal forces of members and surfaces as well as stresses or support forces.

The figure below shows the member internal forces  $M_z$  and the surface internal forces  $m_y$  calculated for CO13. To display the forces, it is recommended to use the wire-frame model. We can set this display option with the button shown on the left.



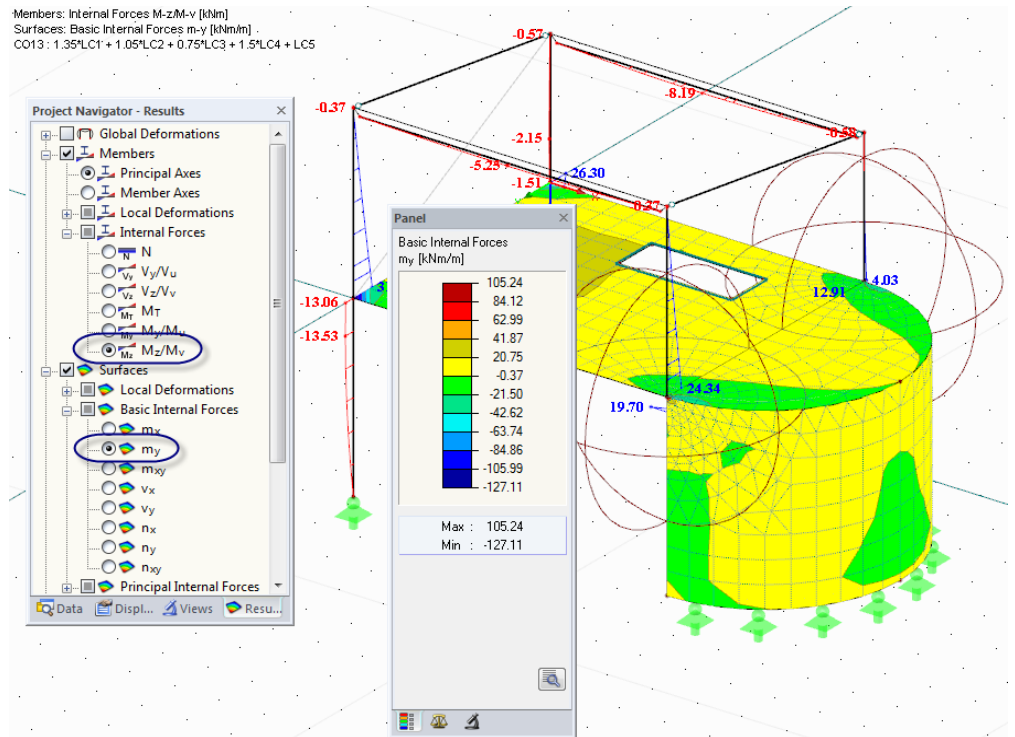


Figure 8.4: Representation of internal forces of members and surfaces (CO13)

### Display of values

The color scale in the control panel informs us about the assignment of color ranges. Moreover, we can switch on the result values for particular locations by ticking the option **Values on Surfaces** in the *Results* navigator. To display all values of the FE mesh nodes or grid points, we deactivate the option *Extreme Values* additionally.

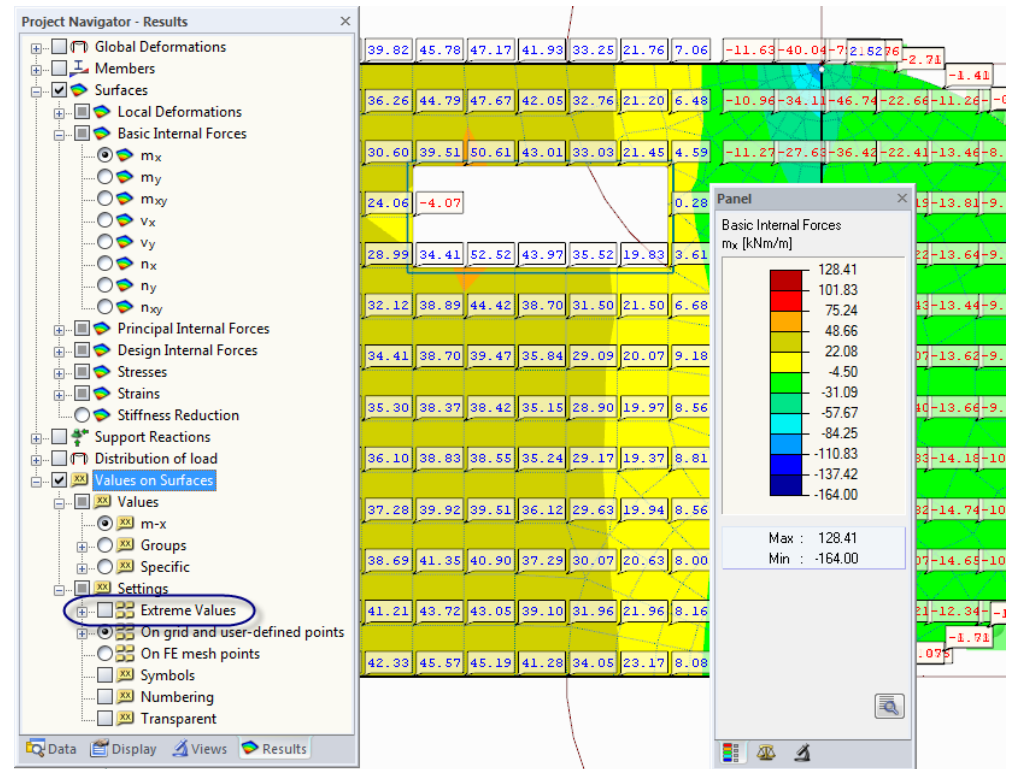


Figure 8.5: Grid point moments  $m_x$  of floor slab in Z view (CO13)

## 8.2 Results Tables

We can evaluate results also in the tables.

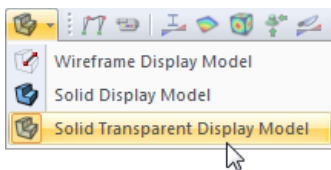
### Table display

The results tables are displayed automatically as soon as the structure is calculated. Like for the numerical input we see various tables with results. Table 4.0 *Summary* offers us a summary of the calculation process, sorted by load cases and combinations.

	B	C	D
	Value	Unit	Comment
<b>LC1 - Self-weight, finishes, earth pressure</b>			
Sum of loads in X	-768.00	kN	
Sum of support forces in X	-768.00	kN	Deviation: 0.00 %
Sum of loads in Y	0.00	kN	
Sum of support forces in Y	0.00	kN	
Sum of loads in Z	652.10	kN	
Sum of support forces in Z	652.10	kN	Deviation: 0.00 %
Resultant of reactions about X	-0.167	kNm	At center of gravity of model (X:6.078, Y:3.045, Z:0.587 m)
Resultant of reactions about Y	-1350.130	kNm	At center of gravity of model
Resultant of reactions about Z	-34.190	kNm	At center of gravity of model
Maximum displacement in X-direction	-0.7	mm	Member No. 1, x: 2.400 m
Maximum displacement in Y-direction	1.0	mm	Member No. 5, x: 3.029 m
Maximum displacement in Z-direction	6.1	mm	FE Node No. 28 (X: 3.000, Y: 2.500, Z: 0.000 m)
Maximum vectorial displacement	6.1	mm	FE Node No. 28 (X: 3.000, Y: 2.500, Z: 0.000 m)

Figure 8.6: Table 4.0 Summary

To select other tables, we click their table tabs. To find specific results in the table, for example the internal forces of the floor slab, we set table 4.14 *Surfaces - Basic Internal Forces*. Now, we select the surface in the graphic (setting the *Solid Transparent Display Model* makes our selection easier), and we see that RFEM jumps to the surface's basic internal forces in the table. The current grid point, that means the position of the pointer in the table row, is indicated by a marking arrow in the graphic.



Surface No.	A Grid Point	B C D Grid Point Coordinates [m]			E F G Moments [kNm/m]			H I Shear Forces [kN/m]		J K L Axial Forces [kN/m]		
		X	Y	Z	m <sub>x</sub>	m <sub>y</sub>	m <sub>xy</sub>	v <sub>x</sub>	v <sub>y</sub>	n <sub>x</sub>	n <sub>y</sub>	n <sub>xy</sub>
1	1	0.000	0.000	0.000	-164.00	-79.00	-93.60	654.65	214.45	182.90	-112.50	24.16
	2	0.500	0.000	0.000	-67.85	17.24	-9.84	327.45	135.96	166.53	103.74	67.47
	3	1.000	0.000	0.000	0.67	-1.39	13.02	147.17	20.97	95.80	0.93	-1.66
	4	1.500	0.000	0.000	20.95	1.85	10.39	113.81	7.69	70.49	-1.19	0.86
	5	2.000	0.000	0.000	32.13	1.70	8.73	80.50	8.06	64.61	-0.41	-0.55
	6	2.500	0.000	0.000	39.82	1.15	5.49	49.35	5.48	62.56	-0.91	0.00
	7	3.000	0.000	0.000	45.78	0.96	4.32	21.77	0.71	61.85	0.60	-0.39
	8	3.500	0.000	0.000	47.17	0.18	1.45	5.41	2.14	61.58	1.03	0.08
	9	4.000	0.000	0.000	41.93	0.48	1.34	-6.54	0.71	62.66	-1.16	-0.62
	10	4.500	0.000	0.000	33.25	-0.07	0.84	-16.50	1.63	68.28	0.87	0.67
	11	5.000	0.000	0.000	21.76	-0.12	-0.07	-19.73	1.36	75.77	-1.12	0.54

Figure 8.7: Surface internal forces in table 4.14 and marker of current grid point in the model



Like the browsing function in the main toolbar we can use the buttons [◀] and [▶] to select the load cases in the table. We can also use the list in the table toolbar to set a particular load case.

### Adjust the results grid

The surface results shown in the tables are listed in the grid points that are defined for the surface. The results grid, like the FE mesh, has a default mesh width of 50 cm.

To refine the grid for the result values of surface 1, we double-click the floor slab in the work window (or the relevant entry in the *Data navigator*). Then, in the dialog box *Edit Surface*, we change settings in the *Grid* tab: We select **25 cm** for both *b* and *h* as new distance of the grid points.

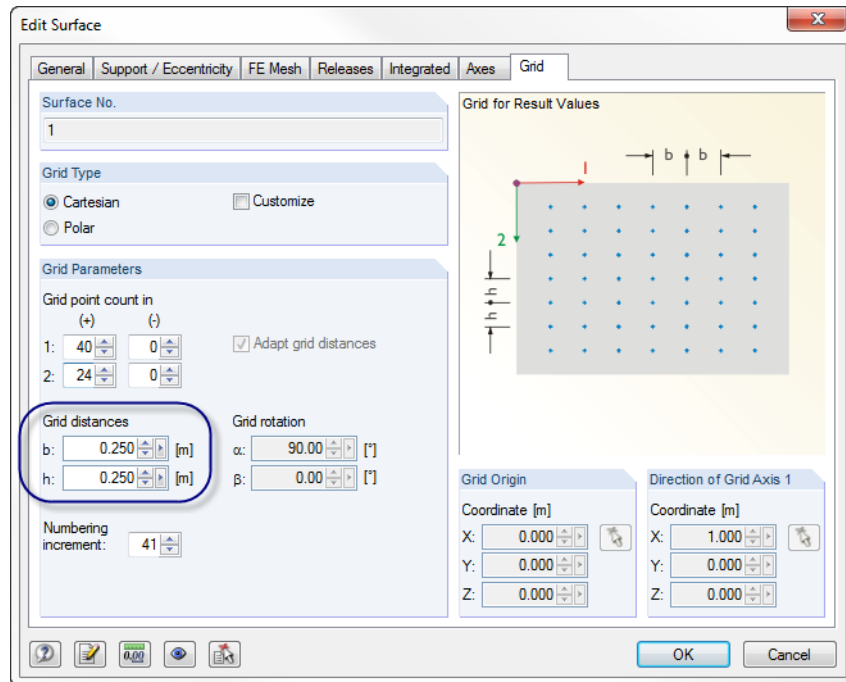


Figure 8.8: Changing the grid for result values

When we click [OK], RFEM updates the result values in the table. A recalculation is not required because the grid point results are determined from the values available in the FE nodes.

4.14 Surfaces - Basic Internal Forces

CO13 - 1.35\*LC1 + 1.0!

Surface No.	Grid Point	Grid Point Coordinates [m]			Moments [kNm/m]			Shear Forces [kN/m]		Axial Forces [kN/m]		
		X	Y	Z	$m_x$	$m_y$	$m_{xy}$	$v_x$	$v_y$	$n_x$	$n_y$	$n_{xy}$
1	1	0.000	0.000	0.000	-164.00	-79.00	-93.60	654.65	214.45	182.90	-112.50	24.16
	2	0.250	0.000	0.000	-115.92	-43.12	-51.72	491.05	175.21	174.71	-4.38	45.81
	3	0.500	0.000	0.000	-67.85	-7.24	-9.84	327.45	135.96	166.53	103.74	67.47
	4	0.750	0.000	0.000	-33.59	-4.32	1.59	237.31	78.47	131.17	52.33	32.91
	5	1.000	0.000	0.000	0.67	-1.39	13.02	147.17	20.97	95.80	0.93	-1.66
	6	1.250	0.000	0.000	10.81	0.23	11.71	130.49	14.33	83.14	-0.13	-0.40
	7	1.500	0.000	0.000	20.95	1.85	10.39	113.81	7.69	70.49	-1.19	0.86
	8	1.750	0.000	0.000	26.54	1.78	9.56	97.15	7.88	67.55	-0.80	0.16
	9	2.000	0.000	0.000	32.13	1.70	8.73	80.50	8.06	64.61	-0.41	-0.55
	10	2.250	0.000	0.000	35.98	1.43	7.11	64.92	6.77	63.59	-0.66	-0.28
	11	2.500	0.000	0.000	39.82	1.15	5.49	49.35	5.48	62.56	-0.91	0.00
	12	2.750	0.000	0.000	42.80	1.06	4.90	35.56	3.09	62.21	-0.15	-0.19
	13	3.000	0.000	0.000	45.78	0.96	4.32	21.77	0.71	61.85	0.60	-0.39

Figure 8.9: Result values of surface 1 with refined grid

## 8.3 Filtering Results

RFEM offers us different ways and tools by which we can represent and evaluate results in clearly-structured overviews. We can use these tools also for our example.

### 8.3.1 Visibilities

We have already worked with visibilities when we entered the steel frame (see chapter 4.5.1, page 31). Those visibility functions are also useful when evaluating the results.

#### Show results for concrete columns

We set the *Views* tab in the navigator. We select *Members by Cross-Section* listed under the visibilities that RFEM has *Generated* from the entered data. In addition, we select the check box for *Circle 300* and activate the option *User-defined / generated* on top of the navigator.

Visibility mode - generated  
 Internal Forces M-zM-v [kNm]  
 CO13 : 1.35\*LC1 + 1.05\*LC2 + 0.75\*LC3 + 1.5\*LC4 + LC5

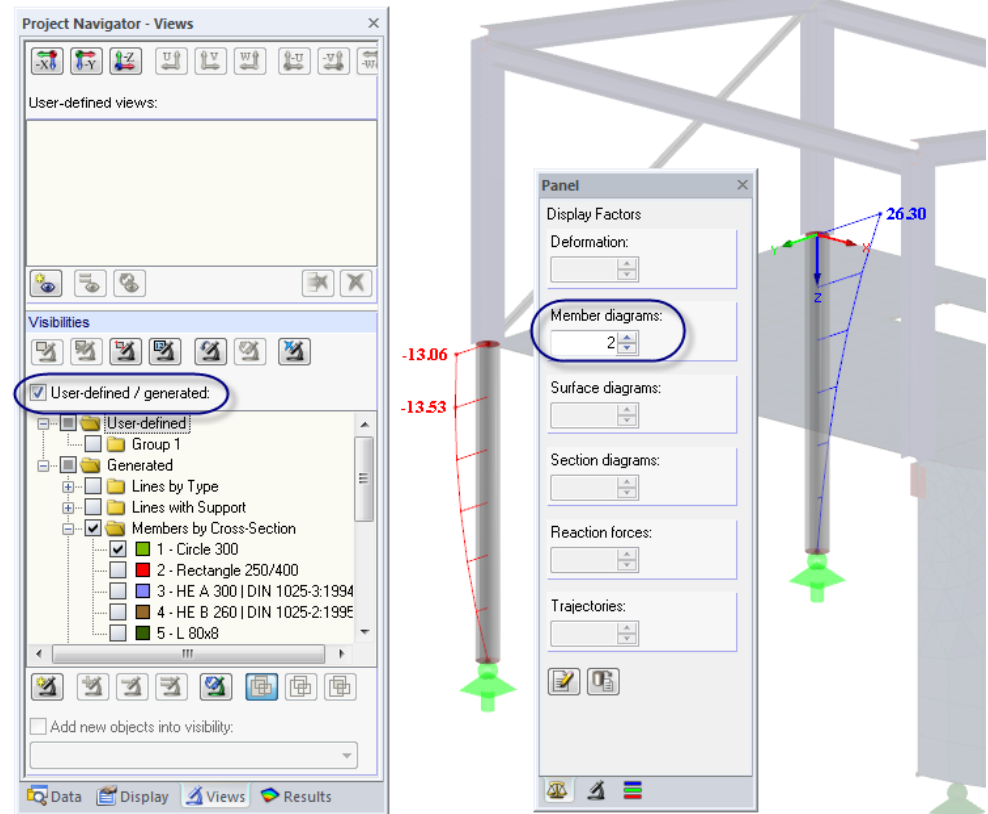


Figure 8.10: Moments  $M_z$  of concrete columns in scaled representation

The display shows the concrete columns including results. The remaining model is displayed only in gray and without results.

#### Adjusting the scaling factor

In order to check the diagram of internal forces on the rendered model without difficulty, we scale the data display in the control tab of the panel. We change the factor for *Member diagrams* to **2** (see figure above).

**Show results for floor slab**

In the same way, we can filter also surface results in the *Views* navigator. We clear the check box for *Members by Cross-Section* and select *Surfaces by Geometry* where we select the check box of the entry *Plane*.



Now, when we change the result type with the button [Deformation] available in the toolbar, we see the following RFEM display.

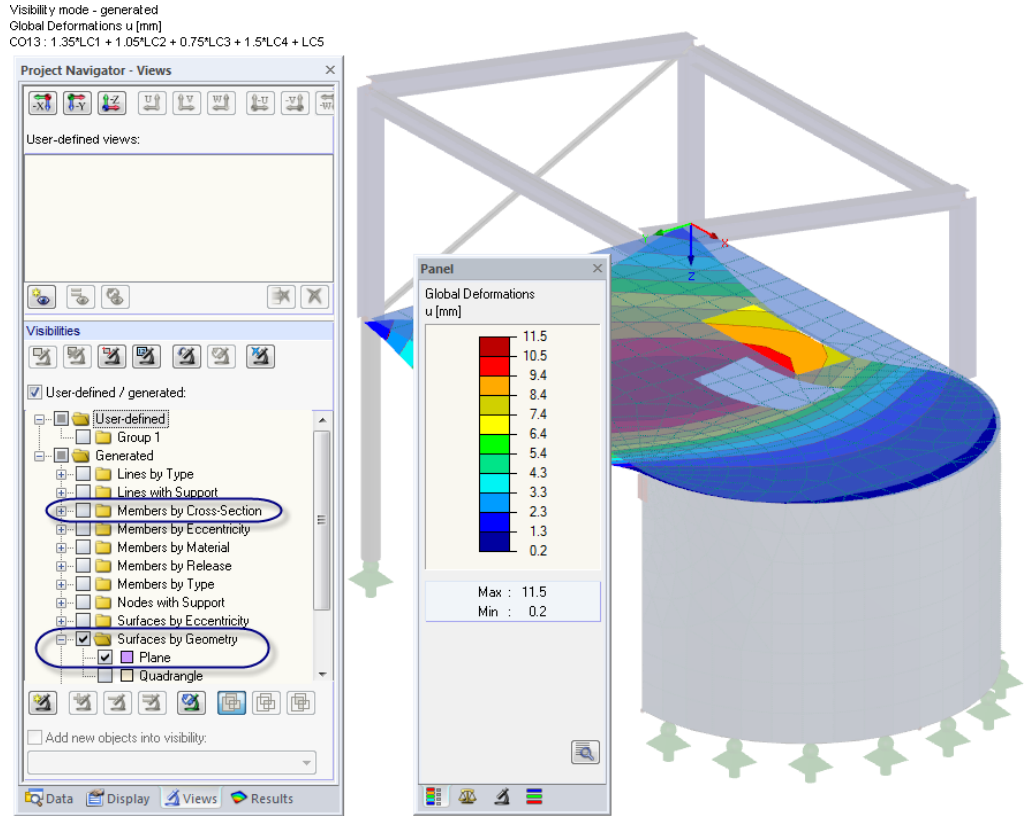


Figure 8.11: Deformations of floor

As already described, we can change the display of result types (deformations, internal forces, stresses etc.) in the *Results* navigator (see Figure 8.3, page 79).

### Show deformations of steel construction

Now, in the *Views* navigator, we clear the check box of *Surfaces by Geometry*, and then we select *Members by Material* where we select the check box for *Steel S 235*.

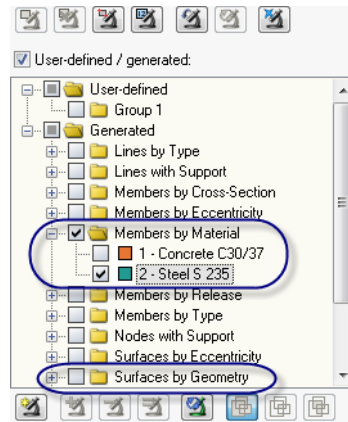


Figure 8.12: Filtering steel members in Views navigator

The graphic shows us the steel construction's deformations displayed as lines.

It is also possible to display the deformations of cross-sections: We switch to the *Display* navigator where we

select **Results, Deformation, Members** and **Cross-sections Colored**.

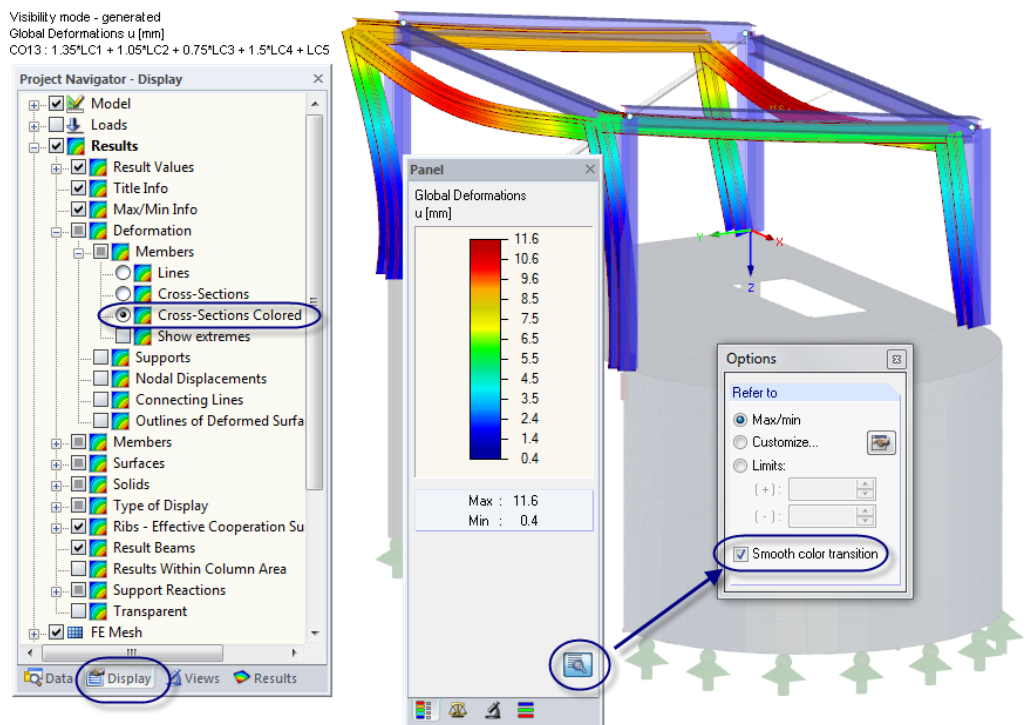


Figure 8.13: Deformations of steel cross-sections

With the panel button [Options] we can additionally activate the option *Smooth color transition*.

RFEM does not display any deformations of the L cross-section, for example in CO13. We have defined this member to be a tension member. However, the deformation shows us that compression forces are occurring. They result in failure of the diagonals, which means that this member is not effective within the system.



### 8.3.2 Results on Objects



Another possibility to filter results is using the filter tab of the control panel where we can specify numbers of particular members or surfaces to display their results exclusively. In contrast to the visibility function, the model will be displayed completely in the graphic.

First, we deactivate the option *User-defined / generated* in the *Views* navigator.

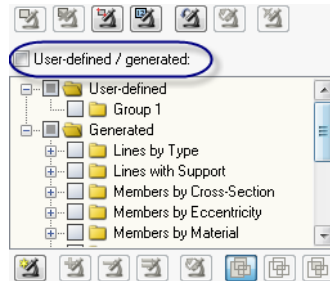


Figure 8.14: Resetting the overall view in the *Views* navigator

Then, in the *Results* navigator, we select the axial forces  $N$  of members. We can deactivate the results of *Surfaces*.

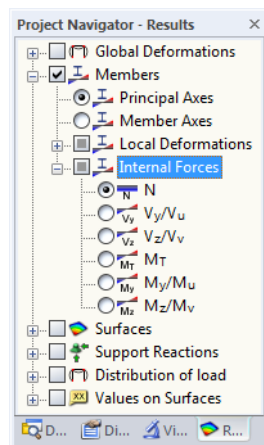


Figure 8.15: Display settings for member axial forces  $N$

### Special selection

To display only the axial forces of columns, we select all members in vertical position. RFEM offers special selection options available in a dialog box. To open the dialog box,



we select **Select** on the **Edit** menu, and then we click **Special**,

or we use the toolbar button shown on the left.

We select the category *Members* and the option *Parallel to member*.

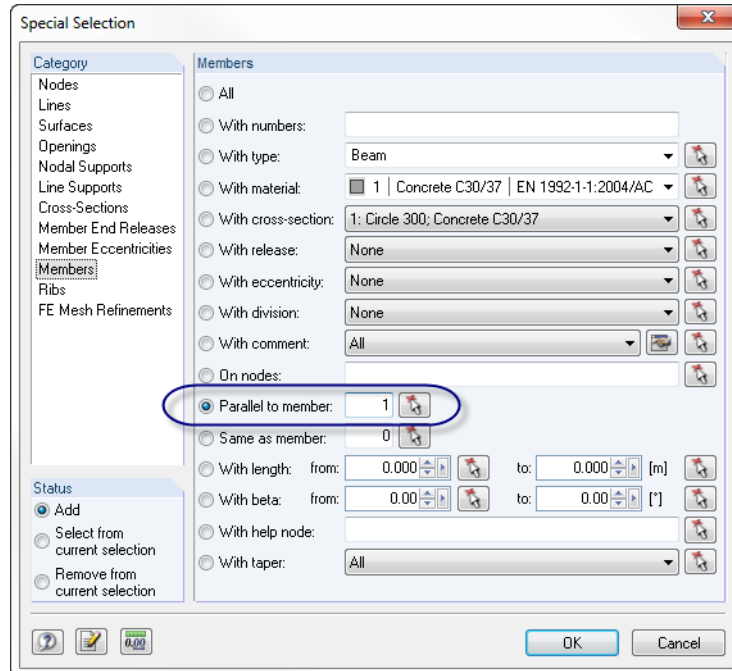


Figure 8.16: Selecting parallel members



Then, we use the [Select] function to click one of the column members in the work window. After closing the dialog boxes with [OK], all members in vertical position are highlighted.

### Show axial forces of columns

In the panel, we change to the filter tab where we activate the setting *Members*.

We click the button [Import from Selection] and see that the numbers of all column members have been entered into the input field above. The axial forces of the rib as well as of the horizontal beams and purlin members disappear in the graphic.

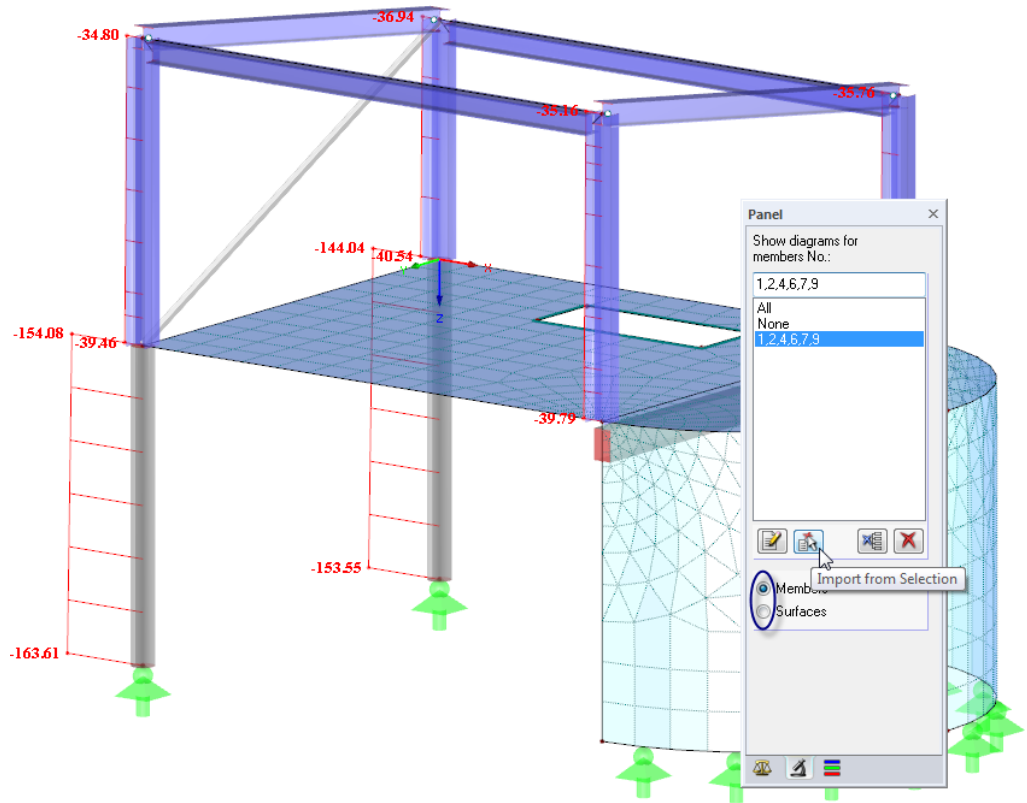


Figure 8.17: Axial forces of members

### 8.3.3 Range of Values

With the color scale tab of the panel we can filter results by result values.

First, in the *Results* navigator, we remove the check mark of the member results. Then, we set the basic internal forces  $n_x$  of surfaces for display. In this case, the forces are axial forces acting in direction of the local surface axis  $x$ . The axis  $x$  of the curved wall surface points downwards.

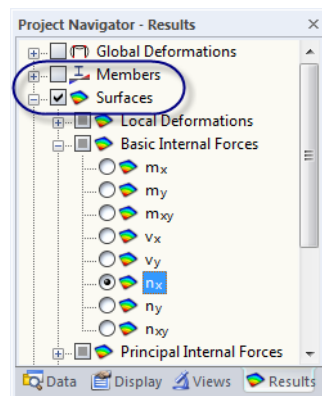


Figure 8.18: Setting basic internal forces  $n_x$  for display

Looking at the wall surface, we can see high compressive forces occurring near the upper arc end nodes. In CO13 for example, they are introduced by both steel columns and across the rib. The high values are indicating singularity effects.

To cut these result peaks for the evaluation, we switch into the color scale tab of the panel. We double-click the color spectrum (not the values!) and open the dialog box *Edit Isoband Value and Color Spectra*.

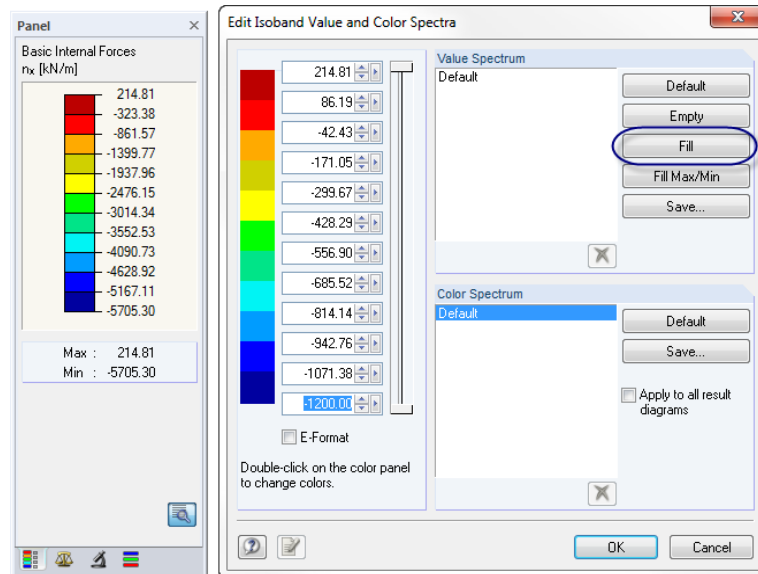


Figure 8.19: Changing extreme value for display and [Fill]ing the value spectrum

We reduce the extreme value of the compression forces as shown in the figure above, for example for CO13 (-1200 kN/m instead of -5705.30 kN/m). Finally, we subdivide the value spectrum into equal ranges between top and bottom limit value by clicking the button [Fill].

After clicking [OK] the force distribution is more differentiated. Now, the zone where lifting forces are occurring is clearly visible. Locations of singularities not covered by the color scale are represented without color.

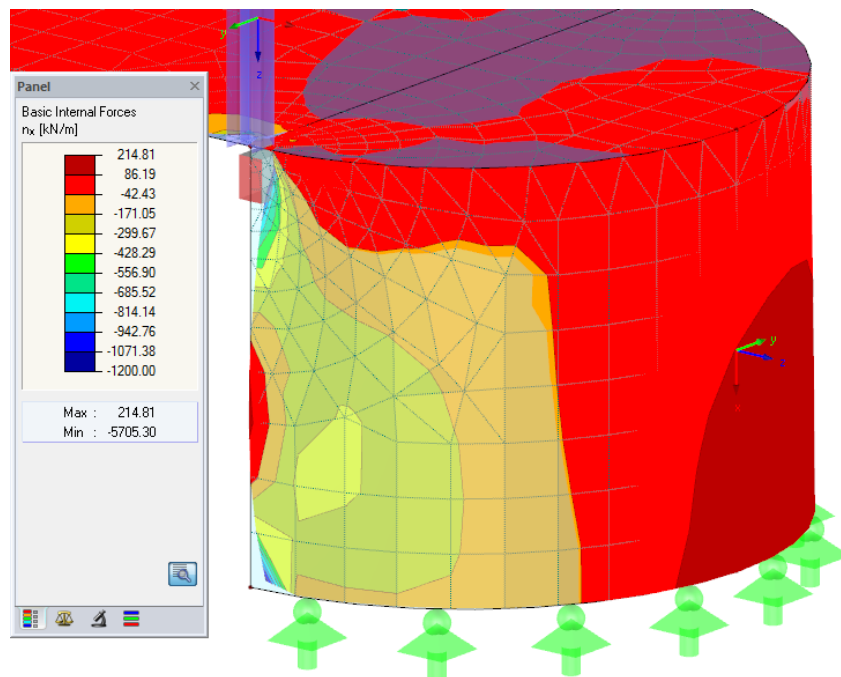


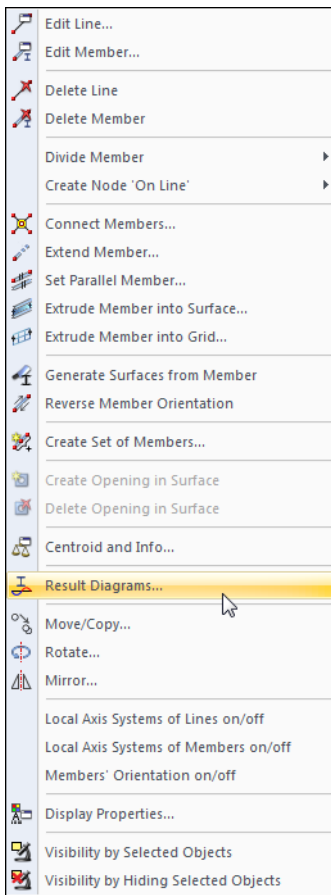
Figure 8.20: Distribution of wall axial forces with adjusted value spectrum (adapted peaks, compression and tension zone)

## 8.4 Display of Result Diagrams

We can evaluate results also in a diagram available for lines, members, line supports and sections (see chapter 8.5). Now, we use this function to look at the result diagram of the downstand beam.

We right-click member 3 (when we have problems we can switch off the surface results) and select the option *Result Diagrams*.

A new window opens displaying the result diagrams of the rib member.



Shortcut menu *Member*

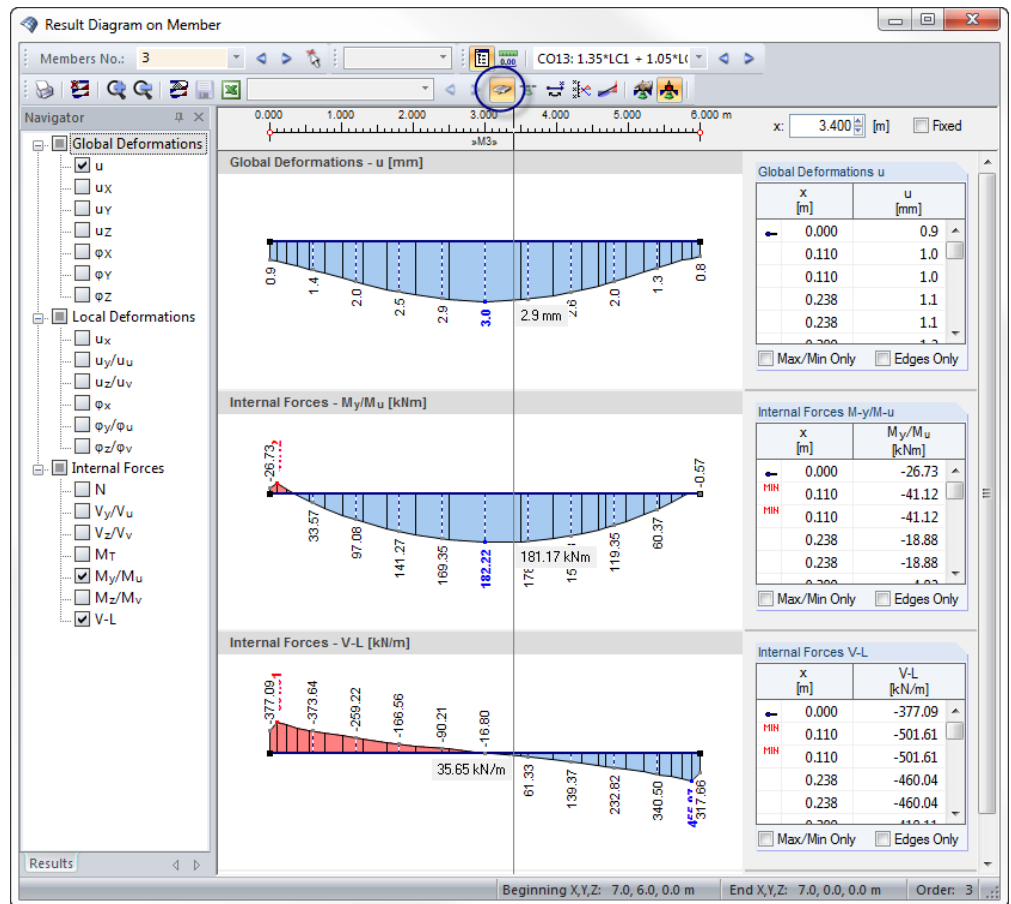


Figure 8.21: Display of result diagrams of downstand beam

In the navigator, we select the check boxes for the global deformations  $u$  and the internal forces  $M_y$  and  $V-L$ . The last option represents the longitudinal shear force between surface and member. These forces are displayed when the button [Results with Ribs Component] is set active in the toolbar. When we click the button to turn it on and off, we can clearly see the difference between pure member internal forces and rib internal forces with integration components from the surfaces.

To adjust the size of the displayed result diagrams, we use the buttons [+ ] and [-].

The buttons [◀] and [▶] for load case selection are also available in the result diagram window. But we can also use the list to set the results of a load case.

We quit the function *Result Diagrams* by closing the window.



## 8.5 Creating Section

The evaluation is made easier by user-defined sections defined as planes slicing through the model. The navigator manages all sections as independent objects.

To create a new section,

we select **Section** on the **Insert** menu

or we use the section shortcut menu in the *Data* navigator.

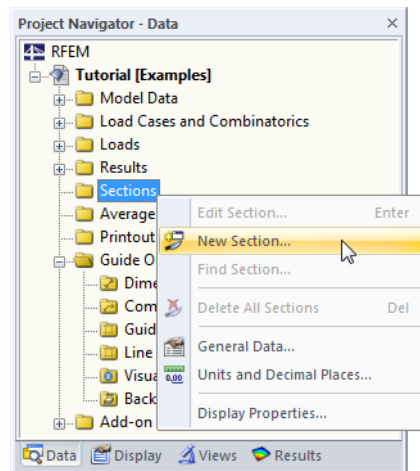
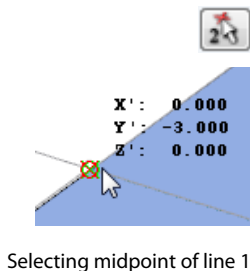


Figure 8.22: Shortcut menu *Sections* in *Data* navigator

The dialog box *New Section* opens where we enter the *Section Name* **Center** because we want to define the section along the plate center.

Then, we define the edge points of the section graphically by using the [Select] function: We click the midpoint of line 1 (global coordinates 0.000/3.000/0.000), and then we select arc node 5.



Selecting midpoint of line 1

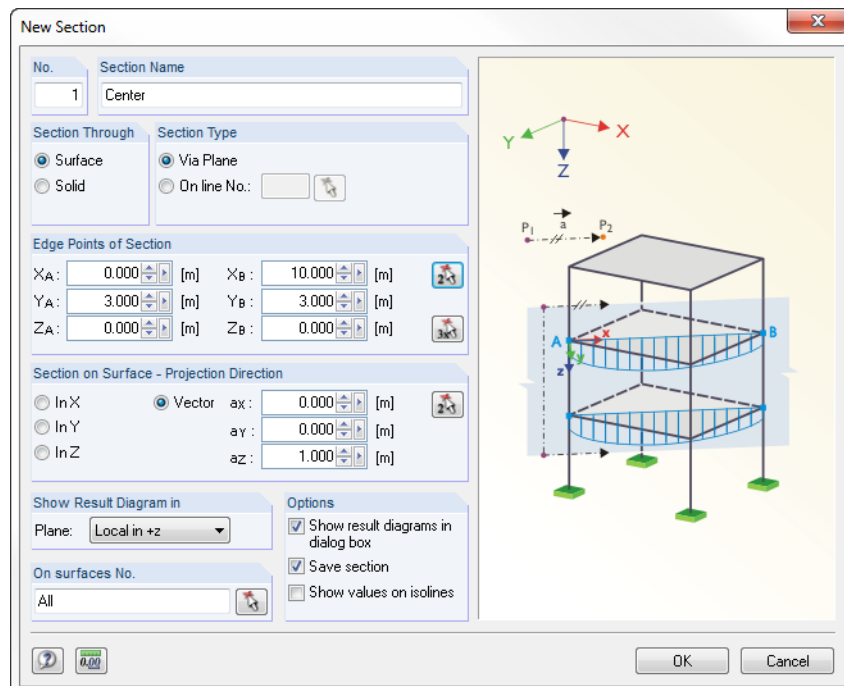


Figure 8.23: Defining the section

We accept the remaining presettings and confirm the dialog box with [OK].

The familiar *Result Diagram* window appears. In the navigator, we select the check boxes for the global deformations  $u$  and the basic internal forces  $m_x$  and  $n_x$ . The results of the surfaces  $S1$  and  $S2$  caught by the section are displayed continuously on a line.

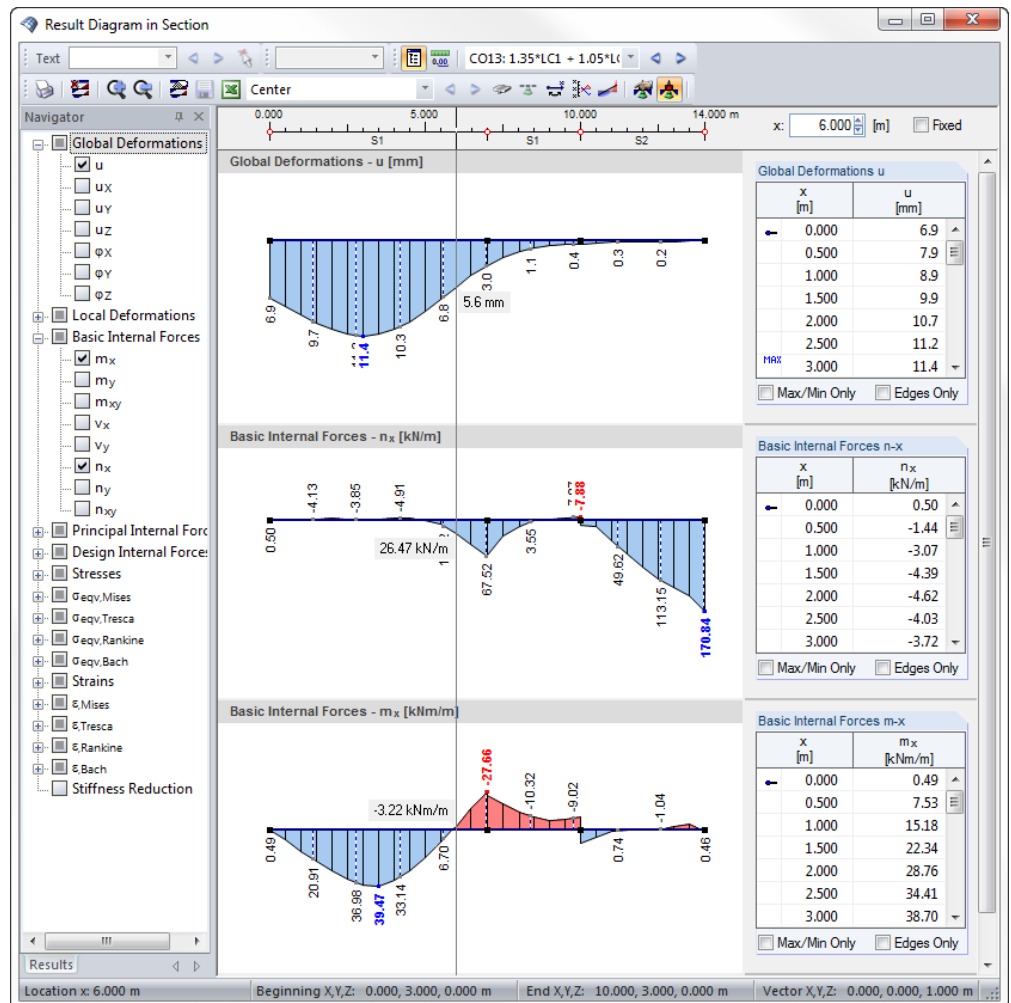


Figure 8.24: Section display in *Result Diagram* window



We [Close] the *Result Diagram* window. Now, we see the section as well displayed in the RFEM work window where we set the basic internal forces  $m_x$ .

We switch off the surface results to display only the diagrams of the section (see Figure 8.26).

With the filled display option for sections available in the *Display* navigator we can highlight the moment diagram in the model.

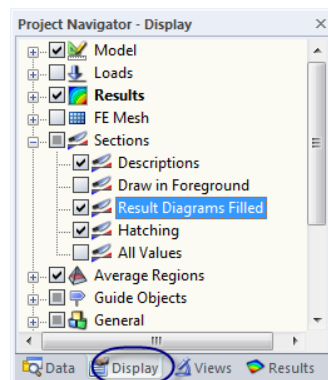


Figure 8.25: Display options for sections in *Display* navigator

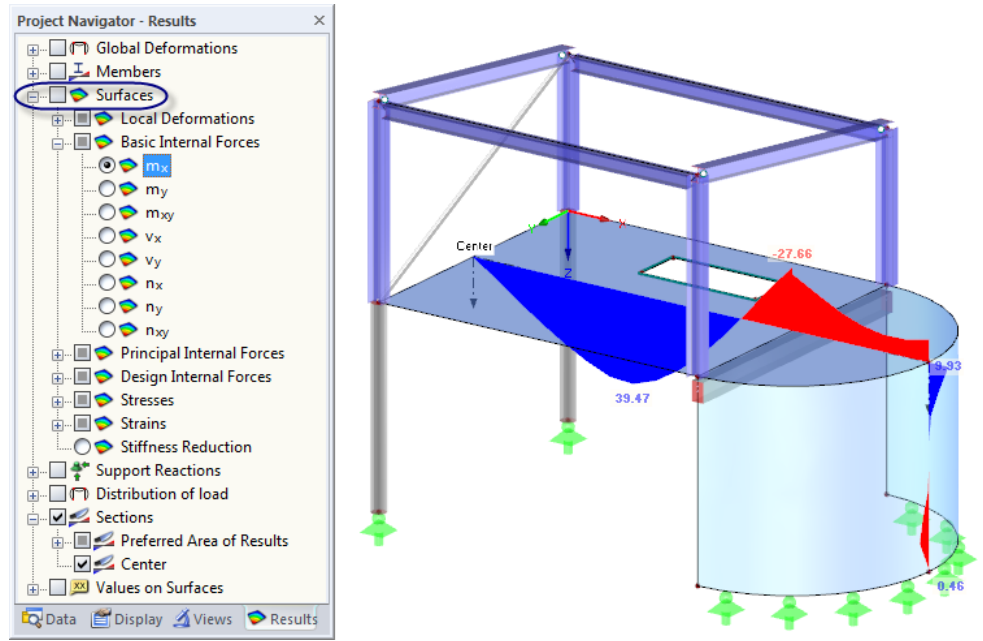


Figure 8.26: Showing the section in the model

# 9. Documentation

## 9.1 Creating Printout Report

It is not recommended to send the complex results output of an FE calculation directly to the printer. Therefore, RFEM generates a print preview first, which is called "printout report" containing input and results data. We use this report to determine the data that we want to include in the printout. Moreover, we can add graphics, descriptions or scans.



To open the printout report,

we select **Open Printout Report** on the **File** menu

or we use the button shown on the left. A dialog box appears where we can specify a *Template* as sample for the new printout report.

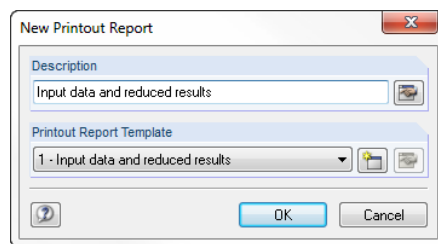


Figure 9.1: Dialog box *New Printout Report*

We accept template 1 - *Input data and reduced results* and generate the print preview with [OK].

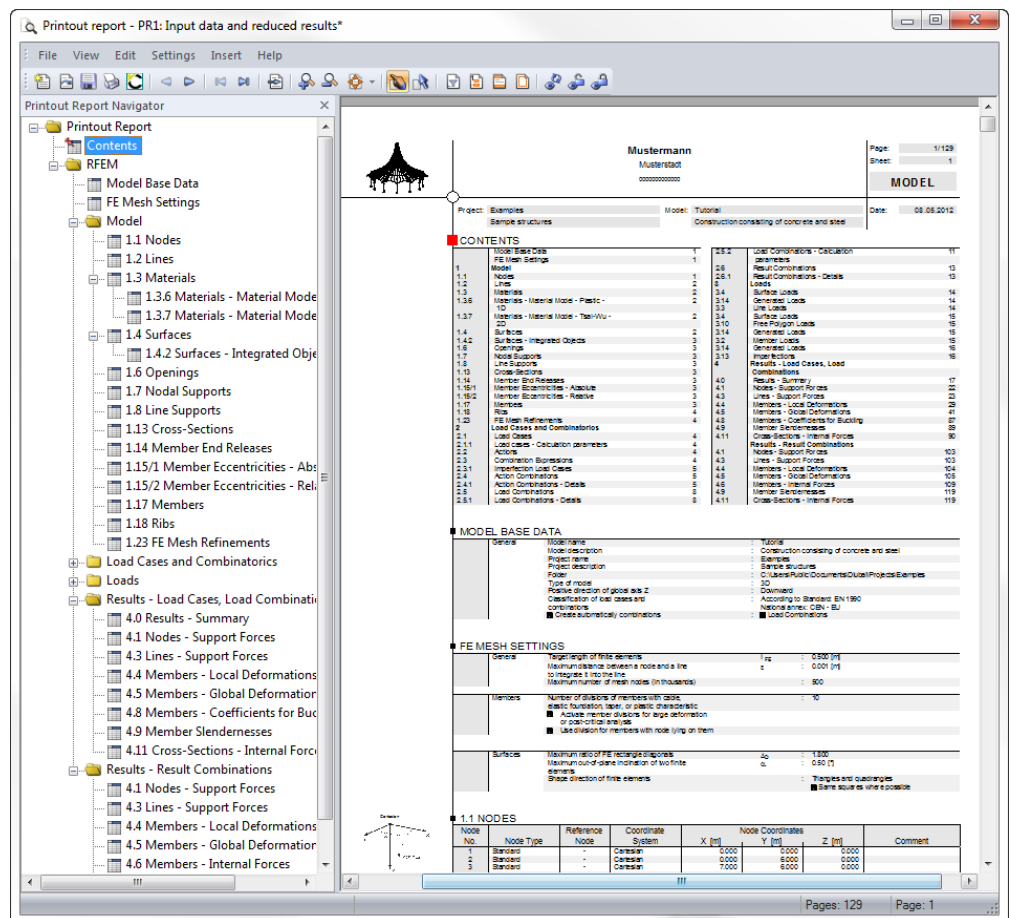


Figure 9.2: Print preview in printout report

## 9.2 Adjusting the Printout Report

The printout report has a navigator, too, listing the selected chapters. By clicking a navigator entry we can see its contents displayed in the window to the right.

The preset contents can be specified in detail. To adjust the output of internal forces, we right-click chapter *Results - Result Combinations*. Then we click *Selection* in the shortcut menu.

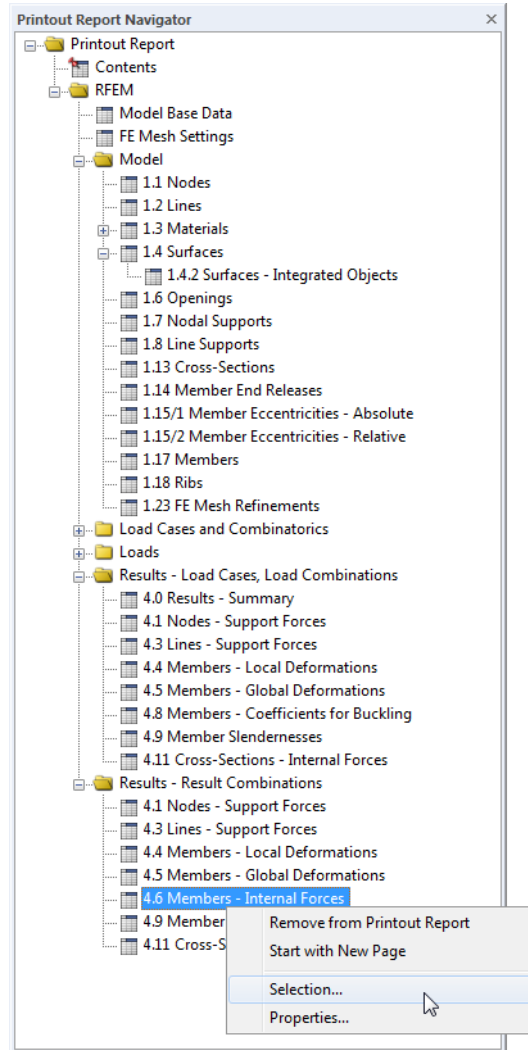


Figure 9.3: Shortcut menu *Members - Internal Forces*

A dialog box appears, offering detailed selection options for RC results of members (see figure below).

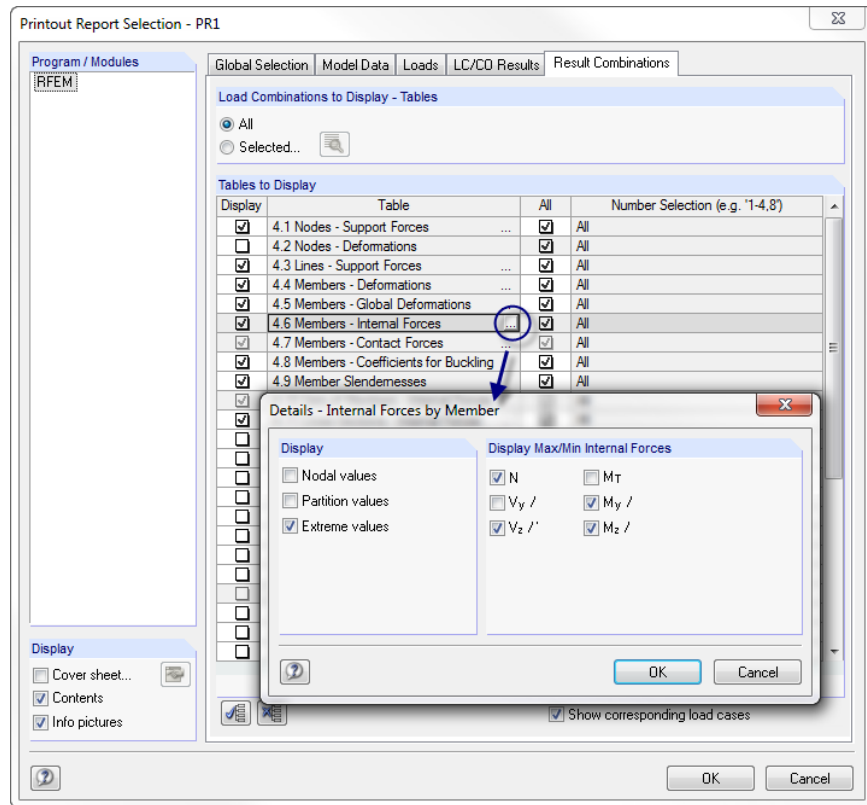


Figure 9.4: Reducing output of internal forces by means of *Printout Report Selection*

We place the pointer in table cell *4.6 Members - Internal Forces* (see figure above). The button [...] becomes active which opens the dialog box *Details - Internal Forces by Member*. Now, we reduce the output to the **Extreme values** of the member internal forces **N**, **V<sub>z</sub>**, **M<sub>y</sub>** and **M<sub>z</sub>**.

After confirming the dialog boxes, RFEM adjusts the output of internal forces accordingly.

Printout report - PR1: Input data and reduced results\*

4.6 MEMBERS - INTERNAL FORCES

Member No.	RC	Node No.	Location x [m]	Forces [kN]			Moments [kNm]				
				N	V <sub>y</sub>	V <sub>z</sub>	M <sub>T</sub>	M <sub>y</sub>	M <sub>z</sub>		
1	RC1	Max N	4.000	Max N	-111.56	3.04	-2.24	0.00	-9.38	-12.75	CO1
		13	0.000	Min N	-175.05	6.24	-3.39	0.00	0.00	0.00	CO5
		Max V <sub>z</sub>	4.000	Max V <sub>z</sub>	-124.83	-2.56	-1.61	0.00	-6.79	-8.42	CO15
		13	0.000	Min V <sub>z</sub>	-162.84	5.00	-3.89	0.00	0.00	0.00	CO2
		13	0.000	Max M <sub>y</sub>	-121.09	3.27	-2.41	0.00	0.00	0.00	CO1
		Min M <sub>y</sub>	4.000	Min M <sub>y</sub>	-153.33	4.53	-3.52	0.00	-15.04	-19.35	CO2
		13	0.000	Max M <sub>z</sub>	-121.09	3.27	-2.41	0.00	0.00	0.00	CO1
		Min M <sub>z</sub>	4.000	Min M <sub>z</sub>	-163.73	4.51	-3.28	0.00	-14.09	-19.35	CO3
2	RC1	Max N	4.000	Max N	-105.51	-11.61	-1.95	-0.00	-8.15	21.11	CO9
		14	0.000	Min N	-166.72	-5.52	-3.13	0.00	0.00	0.00	CO3
		Max V <sub>z</sub>	4.000	Max V <sub>z</sub>	-115.98	-11.57	-1.75	-0.00	-7.36	21.01	CO14
		14	0.000	Min V <sub>z</sub>	-156.68	-4.91	-3.35	0.00	0.00	0.00	CO7
		14	0.000	Max M <sub>y</sub>	-116.05	-3.60	-2.06	0.00	0.00	0.00	CO1
		Min M <sub>y</sub>	4.000	Min M <sub>y</sub>	-146.17	-9.96	-3.04	-0.00	-12.96	25.59	CO7
		Max M <sub>z</sub>	4.000	Max M <sub>z</sub>	-144.04	-12.77	-2.53	-0.00	-10.77	26.30	CO13
		14	0.000	Min M <sub>z</sub>	-116.05	-3.60	-2.06	0.00	0.00	0.00	CO1
3	RC1	Max N	3.000	Max N	226.45	6.04	5.20	-1.63	203.00	2.25	CO5
		3	0.000	Min N	-60.90	82.66	168.56	4.64	-24.80	20.01	CO5
		Max V <sub>z</sub>	0.110	Max V <sub>z</sub>	-15.74	76.91	222.30	-37.11	-37.85	5.55	CO5
		Min V <sub>z</sub>	5.890	Min V <sub>z</sub>	-23.72	-59.53	-214.33	41.52	-26.83	7.85	CO3
		Max M <sub>y</sub>	3.000	Max M <sub>y</sub>	226.45	6.04	5.20	-1.63	203.00	2.25	CO5
		Min M <sub>y</sub>	0.110	Min M <sub>y</sub>	-13.18	77.26	202.92	-33.23	-41.12	4.96	CO13
		3	0.000	Max M <sub>z</sub>	-60.90	82.66	168.56	4.64	-24.80	20.01	CO5
		Min M <sub>z</sub>	0.566	Min M <sub>z</sub>	16.26	43.31	167.46	-28.21	39.19	-6.56	CO7
4	RC1	Max N	3.000	Max N	-25.93	-4.03	-19.21	0.00	-25.80	0.00	CO1
		1	0.000	Min N	-40.54	1.13	-31.11	0.08	45.39	-2.27	CO12
		Max V <sub>z</sub>	3.000	Max V <sub>z</sub>	-25.59	0.02	-16.70	-0.09	-26.60	-0.55	CO8
		1	0.000	Min V <sub>z</sub>	-40.35	-2.11	-31.33	0.09	49.61	-9.82	CO4
		1	0.000	Max M <sub>y</sub>	-40.35	-2.11	-31.33	0.09	49.61	-9.82	CO4
		Min M <sub>y</sub>	3.000	Min M <sub>y</sub>	-36.75	-3.42	-27.61	-0.07	-38.92	-0.34	CO5
		1	0.000	Max M <sub>z</sub>	-40.36	2.64	-27.12	0.04	34.56	2.87	CO15
		1	0.000	Min M <sub>z</sub>	-29.79	-6.06	-24.90	0.08	47.30	-18.31	CO2

Figure 9.5: Extreme values of member internal forces N, V<sub>z</sub>, M<sub>y</sub> and M<sub>z</sub> in printout report

In the same way, we can adjust all remaining chapters for the printout.

To change the position of a chapter within the printout report, we move it to the new position by using the drag-and-drop function. When we want to delete a chapter, we use the shortcut menu (see Figure 9.3) or the [Del] key on the keyboard.

### 9.3 Inserting Graphics in Printout Report

Generally, graphics illustrating the documentation are included in the printout.

#### Printing deformation graphics

We close the printout report with the [X] button. The program asks us *Do you want to save the printout report?* We confirm this query and return to the work window of RFEM.

In the work window, we set the *Deformations* of **RC1 - Ultimate limit state**. Furthermore, we deactivate the *Sections* in the *Results* navigator.

RFEM provides two results for each result combination, the maximum and minimum extreme values. Both values are displayed at the same time in the graphic. For our printed graphic we reset the display to show only the *Max-values*.

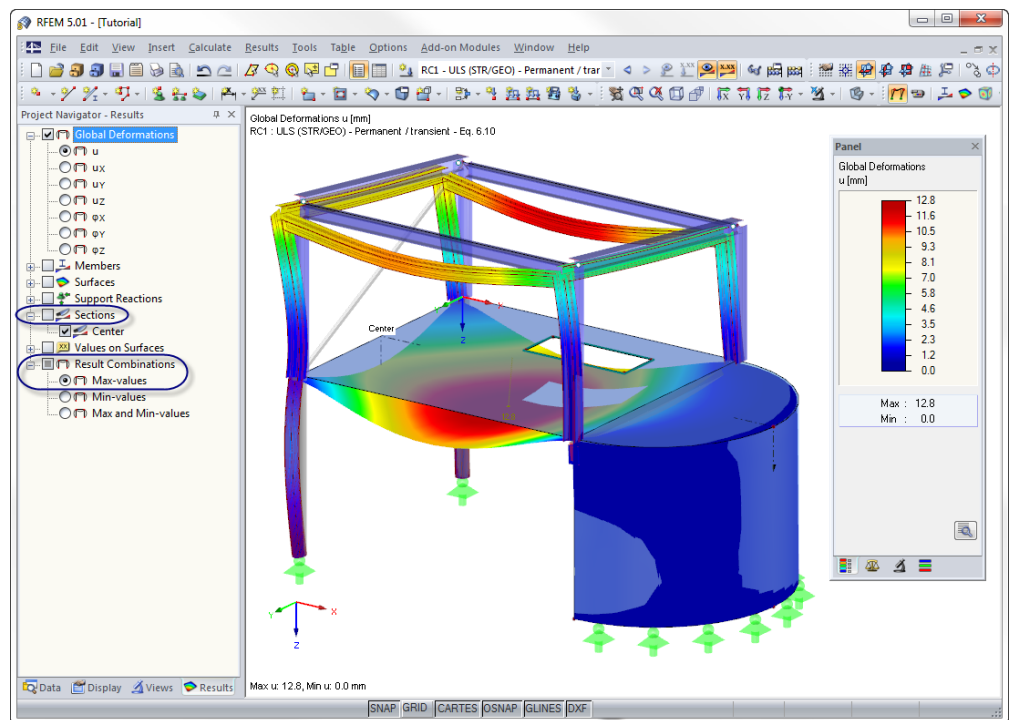


Figure 9.6: Showing the maximum deformation results of RC1

Now, we transfer this graphical representation to the printout report.

We select **Print Graphic** on the **File** menu or we use the toolbar button shown on the left.

The dialog box *Graphic Printout* appears (see figure below).



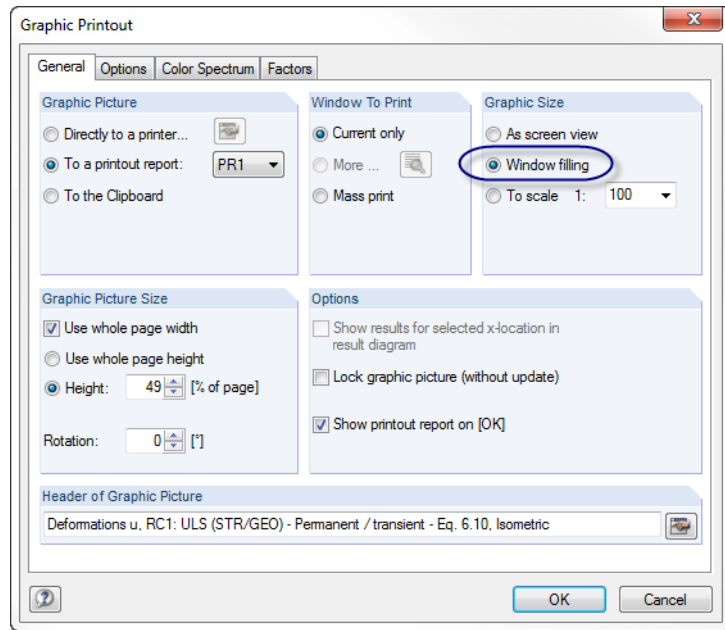


Figure 9.7: Dialog box *Graphic Printout*

We set the print parameters as shown in Figure 9.7. We do not need to modify the default settings of the dialog tabs *Options* and *Color Spectrum*.

We click [OK] to print the deformation graphic into the printout report. The graphic appears at the end of chapter *Results - Load Cases, Load Combinations*.

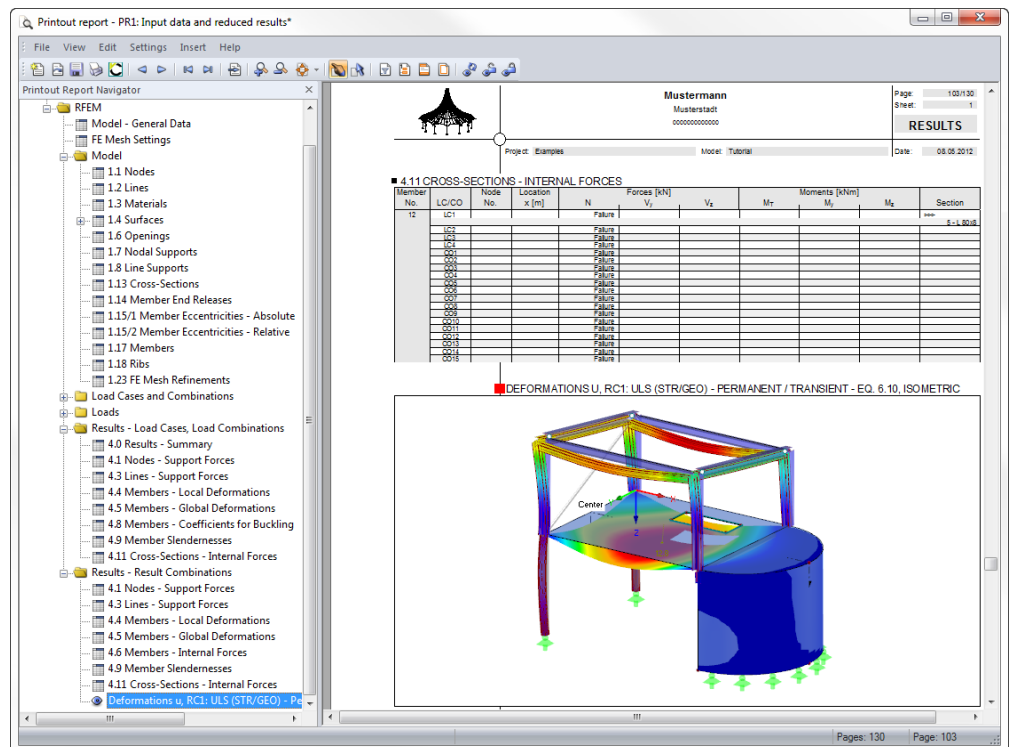


Figure 9.8: Deformation graphic in printout report



### Printing result diagrams

Finally, we want to document the distribution of internal forces available in a steel purlin. We close the printout report again by using the button [X].

Back in the RFEM work window, we right-click member 11 (purlin on high eaves). The member's shortcut menu opens (see page 90) where we select the option *Result Diagrams* to access the result diagram.

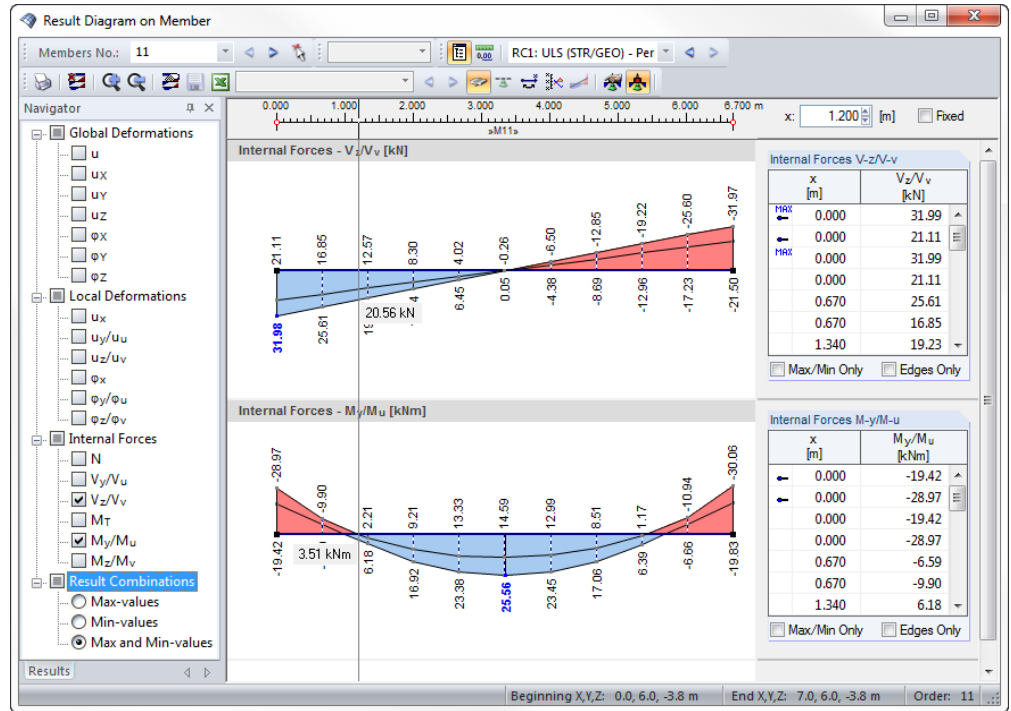


Figure 9.9: Shear force and moment diagram of purlin

The window shows the result diagrams of *RC1*. For our printout we select only the check boxes of the internal forces **V<sub>z</sub>** and **M<sub>y</sub>**. The result diagram shows us the *Max and Min Values*.



With the [Print] button we open the dialog box *Graphic Printout*. We can keep the default settings of the tab *General*. In the *Options* tab, we change some settings.

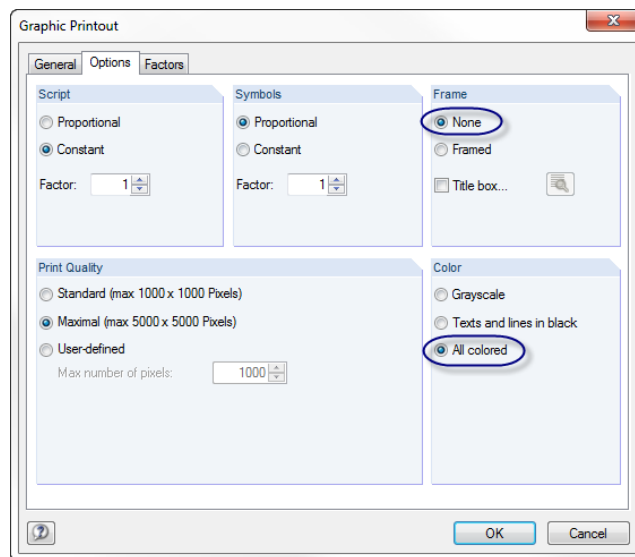


Figure 9.10: Dialog box *Graphic Printout*, tab *Options*

We transfer the graphic to the printout report with [OK].

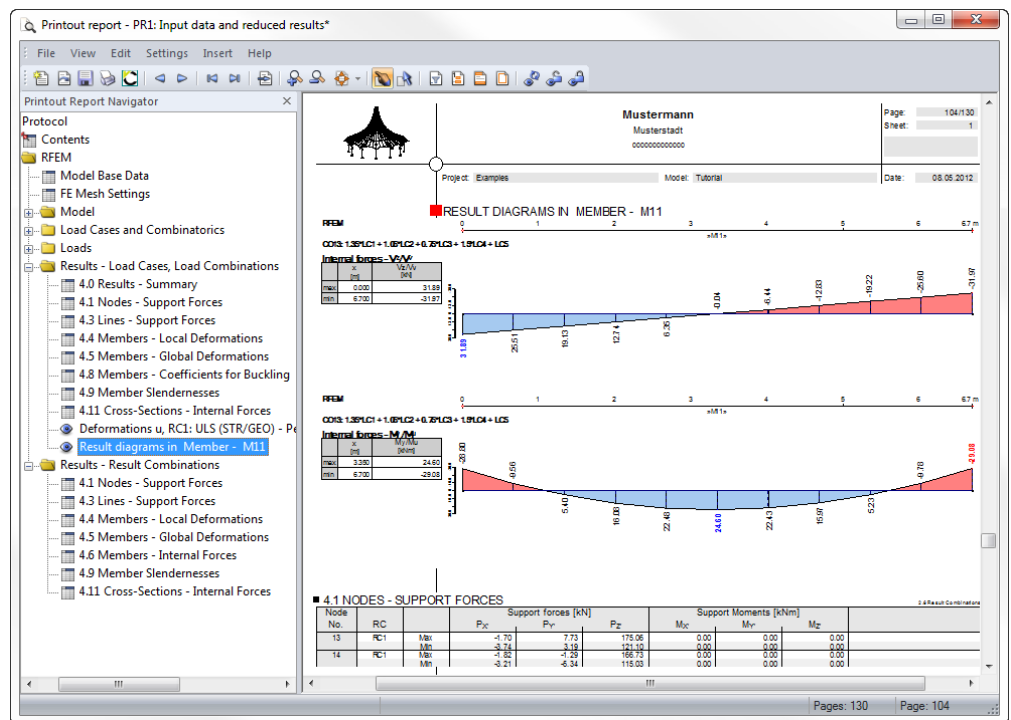


Figure 9.11: Diagrams of internal forces for purlin in printout report

### Changing the language in the printout report

The language in the printout report can be selected independently of the language that is set for the RFEM user interface. Thus, we are able to create for example a German printout report in the English version. To activate the function,

we select **Language** on the **Settings** menu of the printout report.

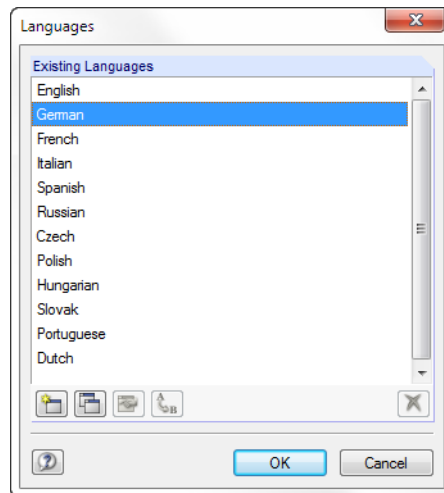


Figure 9.12: Switching languages in printout report

In the dialog box *Languages*, we set **German** (or another language) as the new language. We can check the corresponding modifications in the print preview when we click [OK].

User-defined entries such as load case descriptions or comments won't be translated.

### Printing the printout report

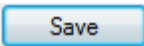


When the printout report is completely prepared, we can send it to the printer by using the [Print] button.

The PDF print device integrated in RFEM makes it possible to put out report data as a PDF file. To activate this function,

we select **Export to PDF** on the **File** menu.

In the Windows dialog box *Save As*, we enter file name and storage location.



By clicking the [Save] button we create a PDF file with bookmarks facilitating the navigation in the digital document.

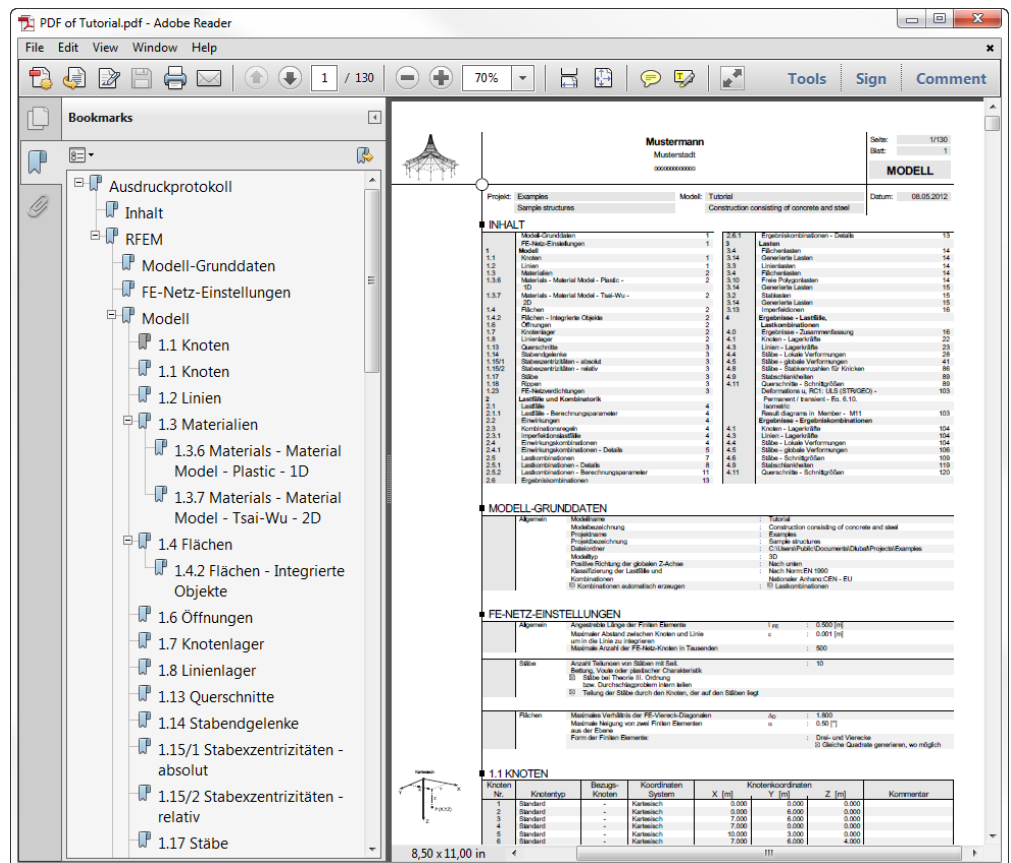


Figure 9.13: German printout report as PDF file with bookmarks

## 10. Outlook

Now, we have reached the end of our example. We hope that this tutorial helps you to get started with RFEM and makes you curious to discover more of the program functions. You find the detail program description in the RFEM manual available as download on our website at [www.dlubal.com/downloading-manuals.aspx](http://www.dlubal.com/downloading-manuals.aspx).

With the **Help** menu or the [F1] key it is possible to open the program's online help system where you can search for particular terms like in the manual. The help system is based on the RFEM manual.

Finally, if you have any questions, you are welcome to use our free fax and e-mail hotline or to have a look at the FAQ page at [www.dlubal.com](http://www.dlubal.com).



Note: This example can be carried out with the demo versions of the add-on modules, for example for steel and reinforced concrete design (RF-STEEL Members, RF-CONCRETE Surfaces/ Members, RF-STABILITY etc.). To comply with the programs' demo restrictions, we suggest to replace objects: For example in RF-STEEL EC3, you can replace the beam by a cross-section IPE 300. In this way, you will be able to perform the design, getting an insight into the functionality of the add-on modules. Then, you can evaluate the design results in the RFEM work window.