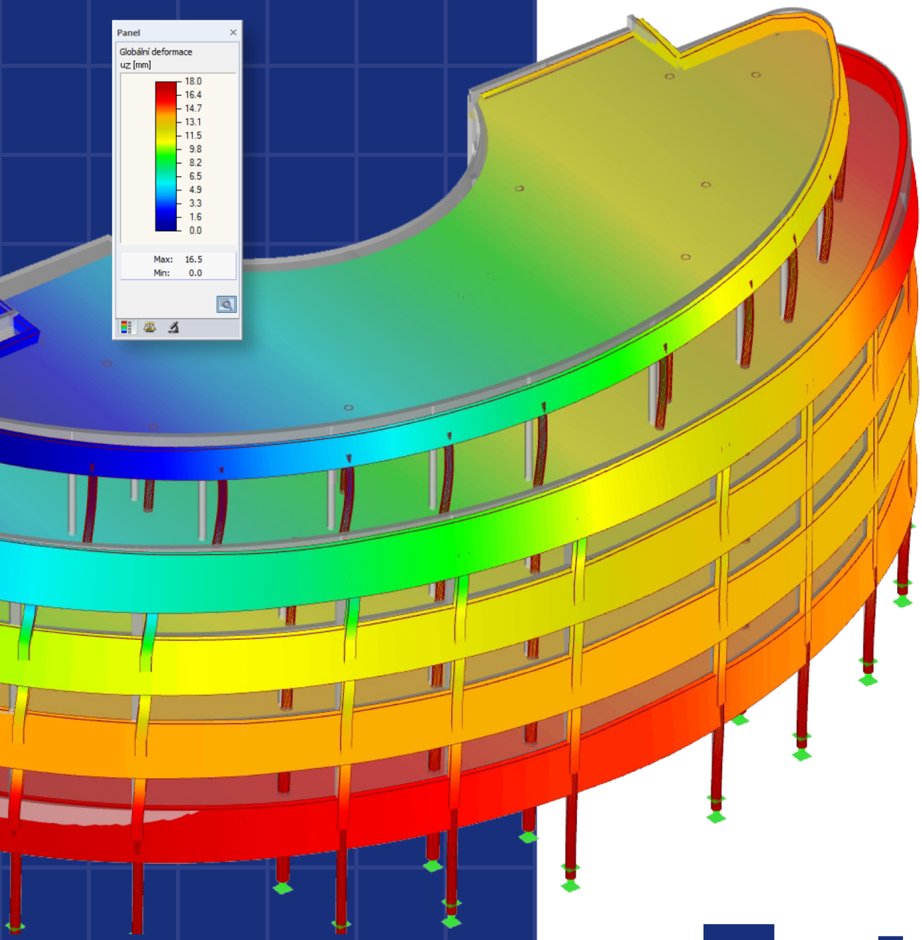
RFEM 5

Spatial Models Calculated According to Finite Element Method



Tutorial

Dlubal Software

Version

April 2018

Short Overview

Dlubal Software GmbH
Am Zellweg 2
93464 Tiefenbach
Germany

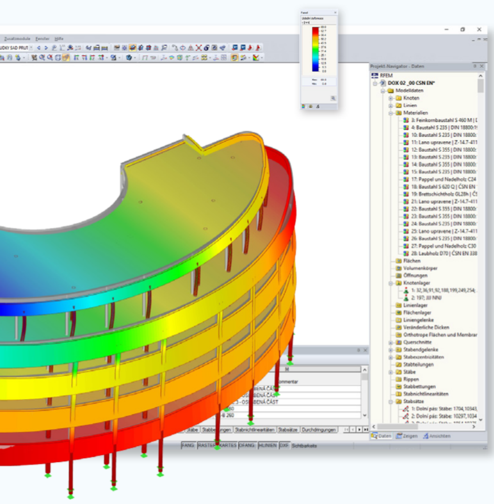
Telephone: +49 9673 9203-0
Fax: +49 9673 9203-51
E-mail: info@dlubal.com

Dlubal Software, Inc.
The Graham Building
30 South 15th Street
15th Floor
Philadelphia, PA 19102
USA

Phone: +1 267 702-2815
E-mail: info@dlubal.com

Using the Manual

The text of the tutorial shows the described buttons in square brackets, for example [OK]. In addition, they are pictured on the left. Expressions that appear in dialog boxes and menus are set in *italics*. The values and parameters required for the example are highlighted in **bold** characters.



Hint

In addition, you can also use the search function for the Knowledge Base and FAQs on our website to find a solution in the articles.

Topicality

The high quality standards placed on the software are guaranteed by a continuous development of the program versions. This may result in differences between program description and the current software version you are using. Thank you for your understanding that no claims can be derived from the figures and descriptions. We always try to adapt the documentation to the current state of the software.

Table of Contents

1 Introduction

In this tutorial, we would like to familiarize you with various features of RFEM. Like in any other software, there are several ways to reach your goal in RFEM. Depending on the situation and your personal preferences, it may be useful to approach something from a different angle. This tutorial is intended to encourage you to explore RFEM's features.

If you are a beginner, you should work on the simpler introductory example. It is available for download on our website at:
https://www.dlubal.com/en-US/downloads-and-information/examples-and-tutorials/introductory-examples-and-tutorials

The example shows a mixed construction with concrete and steel components. The examined load cases consist of self-weight, imposed load, snow, wind, and imperfection according to the first and second order analysis.

This tutorial's data can be entered, calculated, and evaluated with the demo version's restrictions - a maximum of two surfaces and twelve members. Please keep therefore in mind that the model meets the demands of realistic construction projects only to a limited degree. Rather, the features presented in the tutorial are meant to show you how to define model and load objects in various ways.

Since superimposing actions according to EN 1990 involves considerable effort, we will use the integrated load combination generator.

The 90-day trial version allows unrestricted work on the model. The demo version, however, does not allow you to save model data. Therefore, we recommend taking enough time to complete the tutorial (approx. two to three hours) so that you may calmly try out the features. You may also work on the model in the demo version intermittently, so long as RFEM is not closed.

Using two screens for the PDF and RFEM allows you to more easily enter the tutorial's data.

In the text, described buttons are given in square brackets, for example [Apply]. They are also depicted on the left. Terms in the dialog boxes, tables, and menus are marked in *italics*. Required input is written in **bold**.

You may look up the program's features in the RFEM manual, which can be found on our website at https://www.dlubal.com/en-US/downloads-and-information/documents/manuals .

2 System and Load

2.1 System Sketch

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

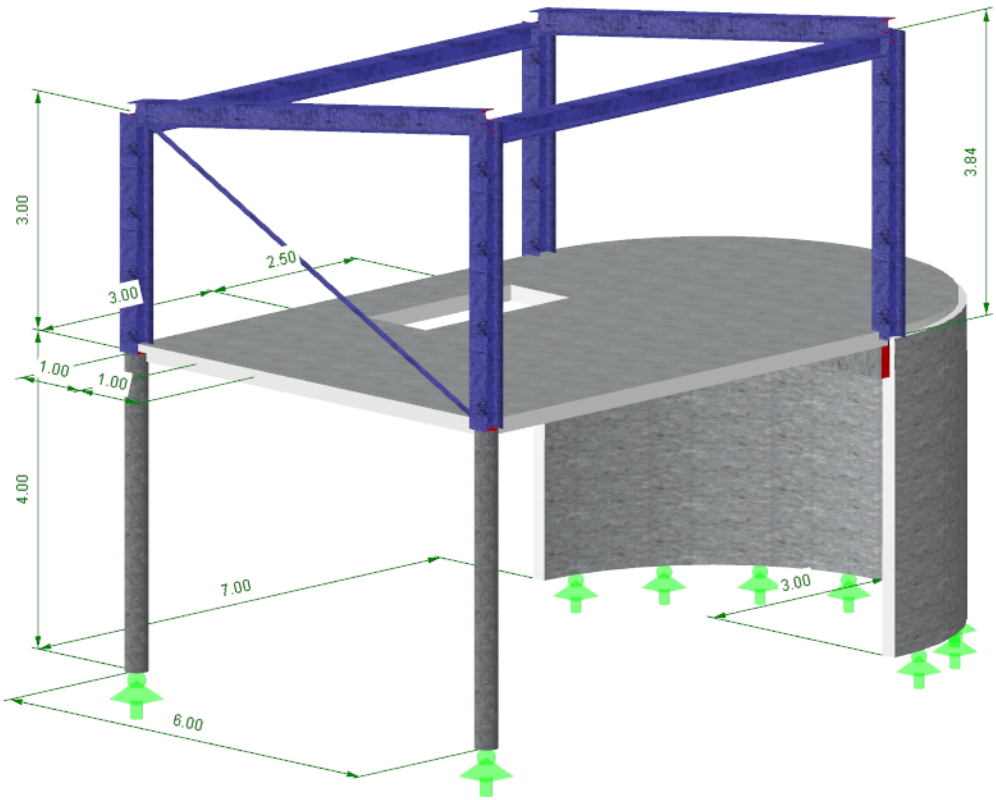


Figure 2.1 Static system

The reinforced concrete structure consists of a floor slab with a downstand beam, a semicircular shell, and two round columns. This section of the support structure is partially underground.

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

As mentioned before, this model depicts a more "theoretical" construction, managable with the demo version's 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.

Wall and floor thickness are uniformly 20 cm. Both concrete columns have a 30 cm diameter. The downstand beam has a width of 25 cm and a 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 profile L 80x8 with equal legs.

2.3 Load

Load case 1: self-weight and finishes

The self-weight of the model with its floor structure of 1.5 kN/m^2 is applied as loading. It is not necessary to determine the self-weight manually. RFEM will automatically calculate the weight based on the materials, surface thicknesses, and cross-sections.

In addition, earth pressure acts on the semicircular wall. The load ordinate at the bottom of the wall is determined as follows for a gravel backfill: q = 16.0 kN/m^3 · 4.0 m = 64 kN/m^2.

The roof load (roofing, sub- and supporting structure) is assumed to be 1.2 kN/m^2.

Load case 2: imposed load

The floor surface is a category C1 assembly room bearing an imposed load of 3.0 kN/m^2.

In addition, a vertically acting linear load of 5.0 kN/m affects the area around the opening. It represents loading due to a stair access.

Load case 3: snow

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

Load case 4: wind

In our example the wind load is only analyzed in Y-direction (inflow direction from low to high eave). It is applied according to EN 1991-1-4 for monopitch roofs and enclosed vertical walls. For the building, we apply wind zone 1 and terrain category III. Since the roof inclination is higher than 5°, positive and negative external pressure coefficients have to be taken into account. In this load case we will assume these coefficients to be positive.

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

Load case 5: imperfection

Imperfections must be considered according to, for example, Eurocode 3. Inclinations and precambers are managed in a separate load case. This allows us to assign specific partial safety factors when combining the load with other actions.

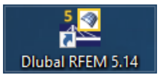
As with the wind load, we will analyze the imperfections only in Y-direction in the example.

According to EN 1993-1-1, Table 6.2, we must assume the buckling curve *b* (displacement in direction of y-axis) for the column cross-sections (HE-A 300). The inclinations φ_0 and precambers w_0 are determined according to EN 1993-1-1, section 5.3.2.

The imperfections for both reinforced concrete columns are applied according to EN 1992-1-1, section 5.2.

3 Creating the Model

3.1 Starting RFEM



We run RFEM using the task bar

Start → **All Programs** → **Dlubal** → **Dlubal RFEM 5.xx**

or by double-clicking the **Dlubal RFEM 5.xx** icon on our desktop.

3.2 Creating the Model

The RFEM work window opens showing us the dialog box below. We are prompted to define the general data of a new model.

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

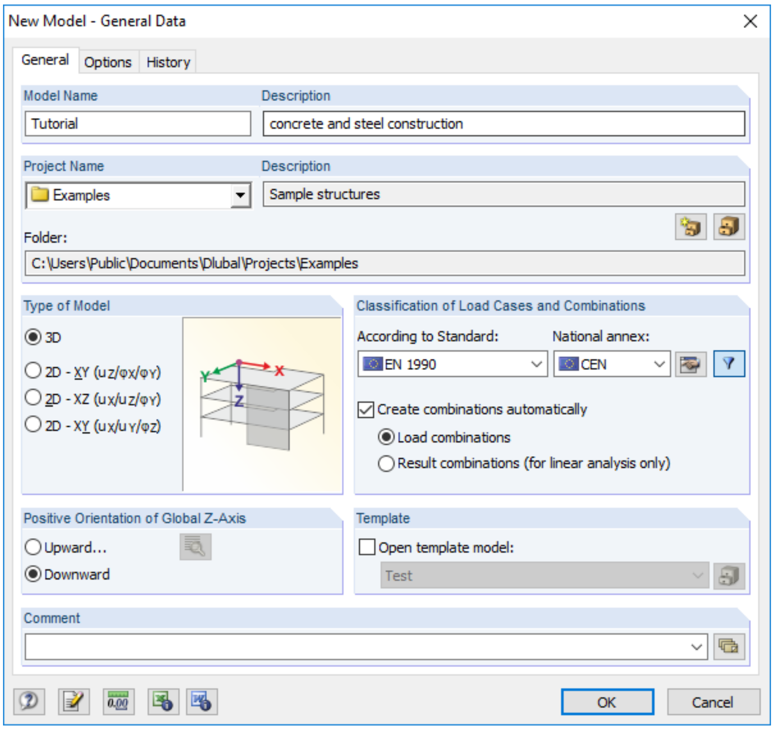
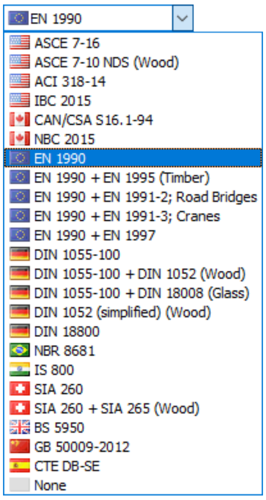


Figure 3.1 *New Model - General Data dialog box*

We enter **Tutorial** into the *Model Name* box and **concrete and steel construction** as the *Description*. The model name always has to be entered, since it is also the file name. However, a description does not have to be given.

For the *Project Name* box, we select **Examples** from the list if it is not already preset. The project *Description* and the corresponding *Folder* are displayed automatically.

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



The *Classification of Load Cases and Combinations* section requires some specifications: We select **EN 1990** from the *According to Standard* ist. We do not change the **CEN** setting in the *National Annex* box. These specifications are important when we combine actions with partial safety factors and combination coefficients conforming to standards.

We select the checkbox *Create combinations automatically*. We also want to superimpose the actions in **Load combinations**.

Now the model's general data is defined. We close the dialog box with the [OK] button.

The empty RFEM work window is displayed.

4 Model Data

4.1 Adjusting the Work Window and Grid

View

First, we maximize the work window using the button on the title bar. We see the coordinate axes with the global directions X, Y and Z displayed in the workspace.

To change the axes' position, we click the [Move, Zoom, Rotate] button in the toolbar above. The pointer turns into a hand. While holding down the left mouse button, we can now drag the work space anywhere.

We can also use the hand to rotate or zoom the view:

- Rotate: We keep the [Ctrl] key pressed down and drag with the pointer
- Zoom: We keep the [Shift] key pressed down and drag with the pointer

There are several ways to exit this feature:

- By clicking the button once more
- By pressing the [Esc] key
- By right-clicking into the workspace

Mouse functions

The mouse functions match the general standards of Windows programs: To select an object for further editing, simply click it with the **left** mouse button. Double-click the object when you want to open its dialog box for editing.

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

To change the displayed model's size, rotate the **wheel**. To move the model, keep the wheel button pressed down. If you keep the [Ctrl] key pressed at the same time, you can rotate the model. Rotating the model is also possible by moving the mouse while pressing the wheel button and the right mouse button at the same time. The pointer's form indicates the selected function.

Grid

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

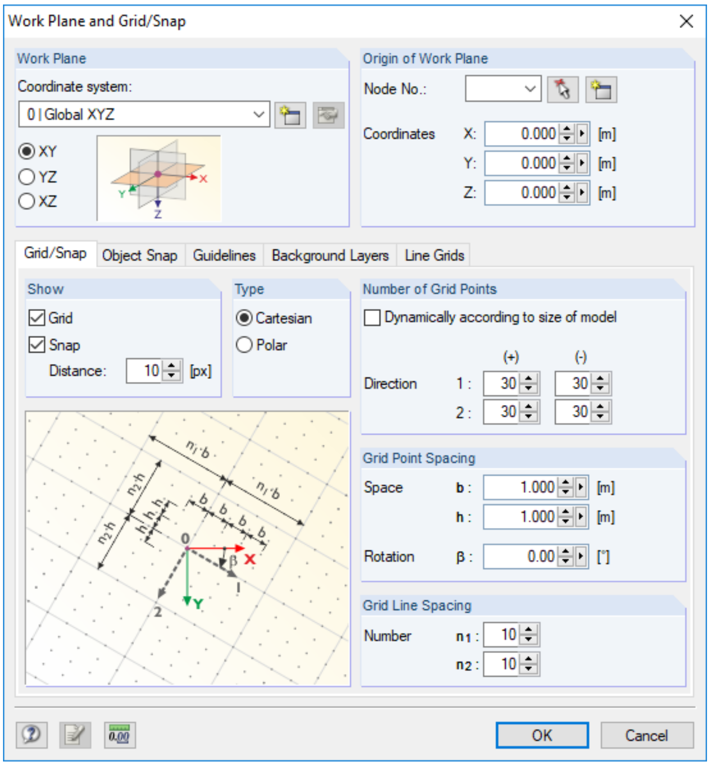


Figure 4.1 *Work Plane and Grid/Snap dialog box*

For entering data in grid points at a later point, it is important that the control fields *SNAP* and *GRID* on the status bar are activated. This makes the grid visible in the work space, and the points will be snapped to the grid when clicking.

Work plane

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

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

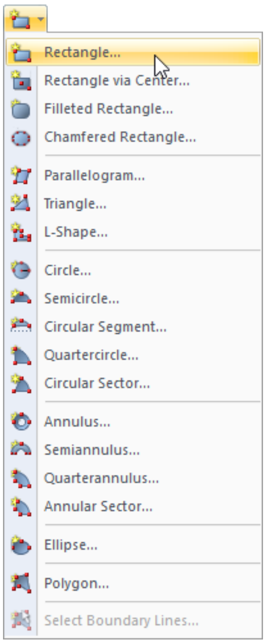
4.2 Creating Surfaces

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

4.2.1 Floor

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

4.2.1.1 Defining a Rectangular Surface



List of plane surfaces

Rectangular slabs are frequently used structural components. To quickly create them, use

Insert → Model Data → Surfaces → Plane → Graphically → Rectangle

or the corresponding list of plane surfaces. Clicking next to the button opens a menu offering a large selection of surface geometries.

The *Rectangle* options allows you to directly define the slab. Related nodes and lines will be created automatically.

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

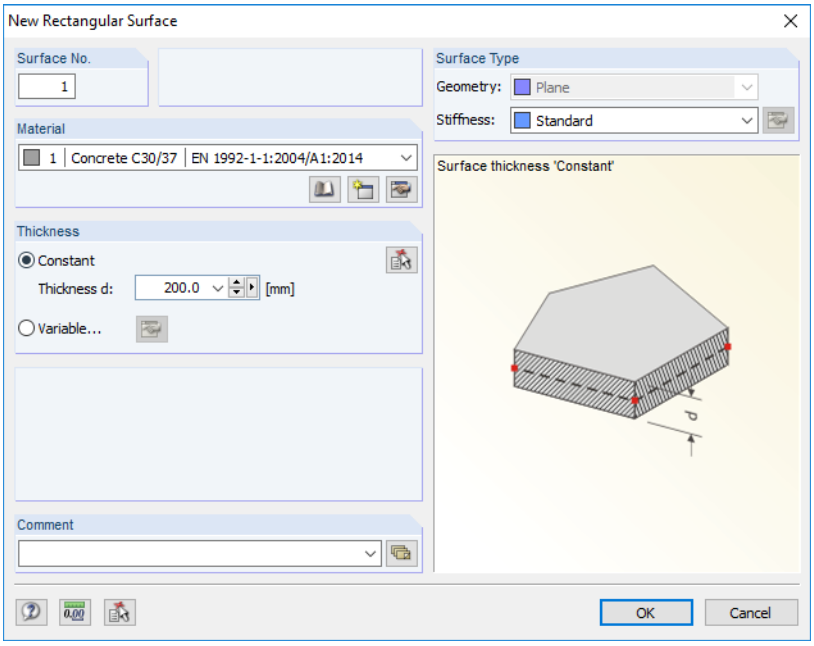


Figure 4.2 *New Rectangular Surface* dialog box

The new rectangular plate's *Surface No.* is specified as *1*. We do not change this number.

The default *Material* is *Concrete C30/37* according to EN 1992-1-1. If we want to use a different material, it can be selected using .

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

The *Stiffness* in the *Surface Type* section is suitably preset to *Standard*.

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

We can more easily define the surface by using the button shown on the left to set the view in Z-direction ("top view"). The input mode will not be affected.

To define the first corner, we click the **coordinate origin** (coordinates **0.000/0.000/0.000**). The current coordinates are displayed next to the reticle pointer.

We define the opposite corner by clicking the grid point with the X/Y/Z coordinates **7.000/6.000/0.000**.

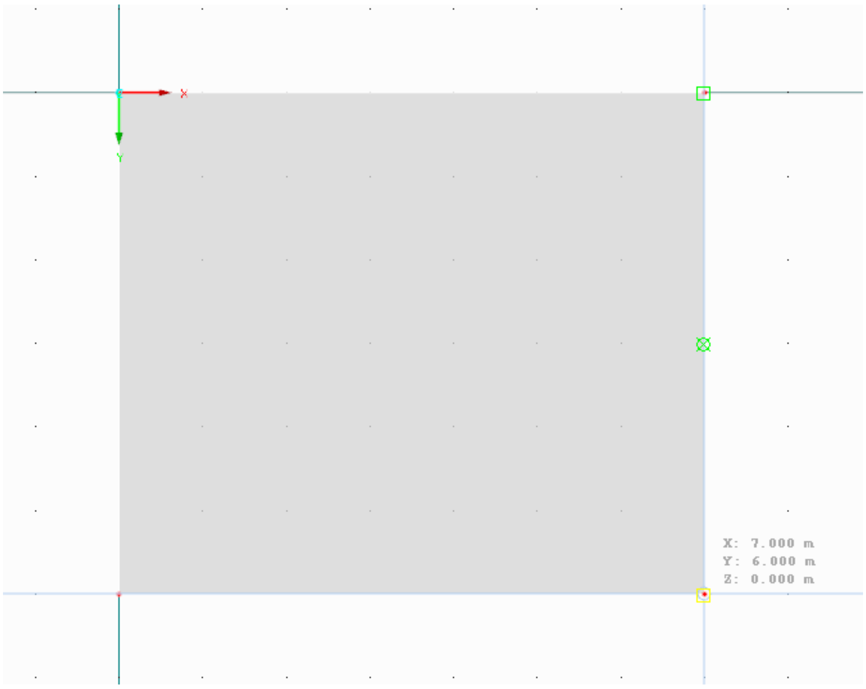


Figure 4.3 Rectangular surface 1

RFEM creates four nodes, four lines, and one surface.

As we do not want to create any more slabs, we quit the input mode either by pressing the [Esc] key or by right-clicking in a free area of the work window.

Showing the numbering

To quickly display the numbering of nodes, lines, and surfaces, right-click in a free area of the work window. This will result in the appearance of a shortcut menu with useful features. We select *Show Numbering*.

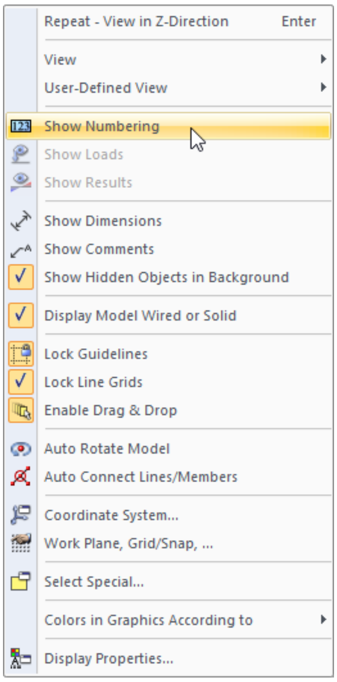


Figure 4.4 *Show Numbering* on the shortcut menu

To manage the numbering of objects in detail, use the *Display* tab in the navigator.

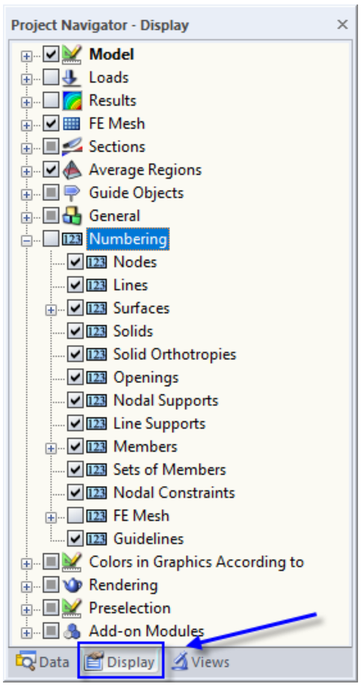


Figure 4.5 *Display* navigator for numbering

4.2.1.2 Creating an Arc

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

We click the arrow next to the list of lines in the toolbar to open the menu that provides tools for special line types. We select *Arc via Three Nodes*.

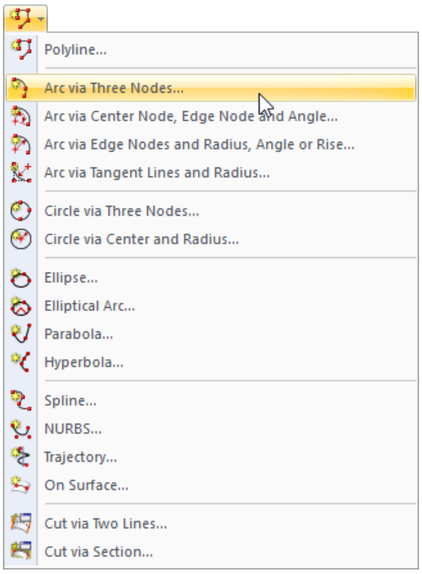


Figure 4.6 List of lines - *Arc via Three Nodes*

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.

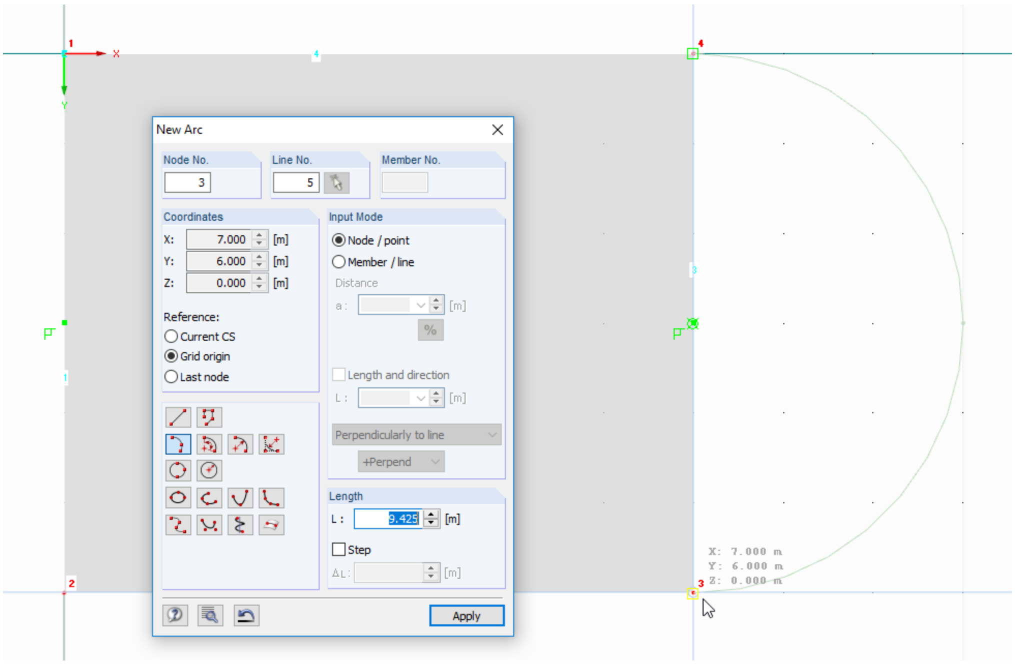


Figure 4.7 Defining a circular arc via three points

We quit the input mode using the [Esc] button.

4.2.1.3 Adjusting the Floor Surface

Due to the demo version's restriction to just two surfaces, we cannot define the semicircular surface as a new surface. Therefore, we extend the rectangular surface to a general plane surface that encloses the arc area.

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

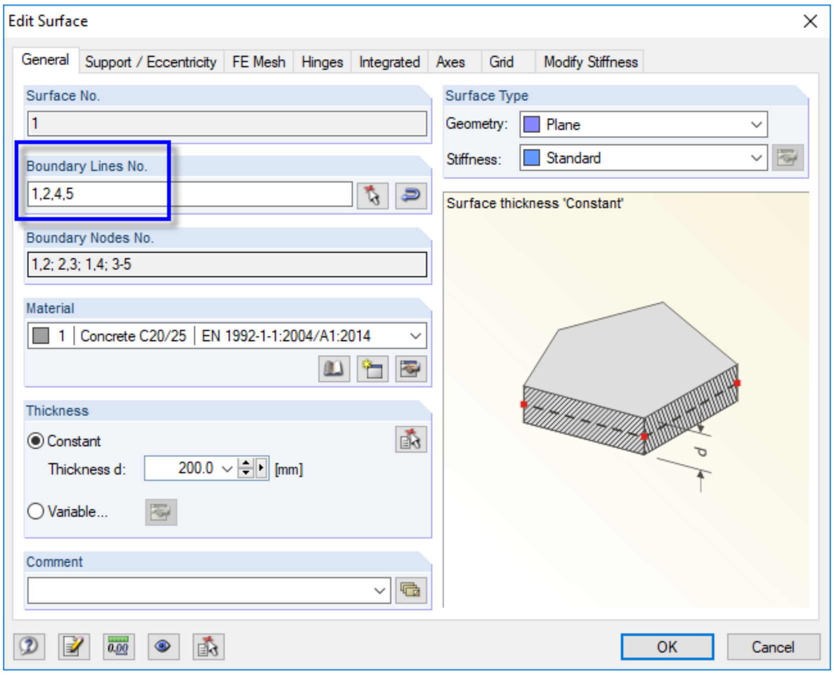
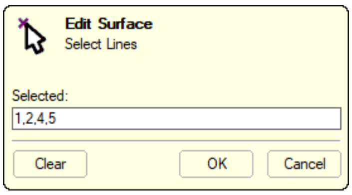


Figure 4.8 Modifying boundary lines



There are two input options:

- We can enter the numbers of the new border lines **1**, **2**, **4**, and **5** manually into the *Boundary Lines No.* box.
- The new boundary lines may also be selected graphically in the work window using . This requires us to [Clear] the preset list in the *Edit Surface* dialog box.

At this point the floor surface should look as in the figure below.

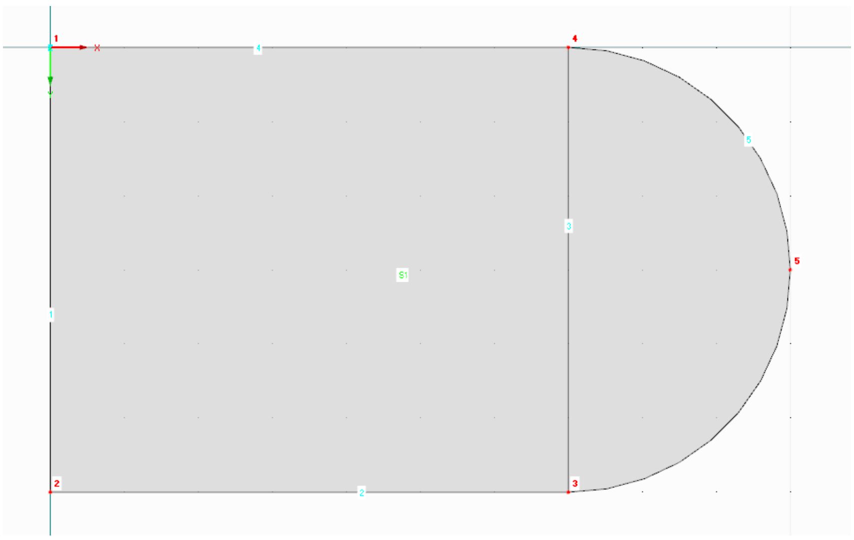


Figure 4.9 Floor slab

4.2.2 Wall

Copying the arc

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

We click arc line 5 once to select it. The line is now displayed in a different color. Yellow is the default selection color for black backgrounds.

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

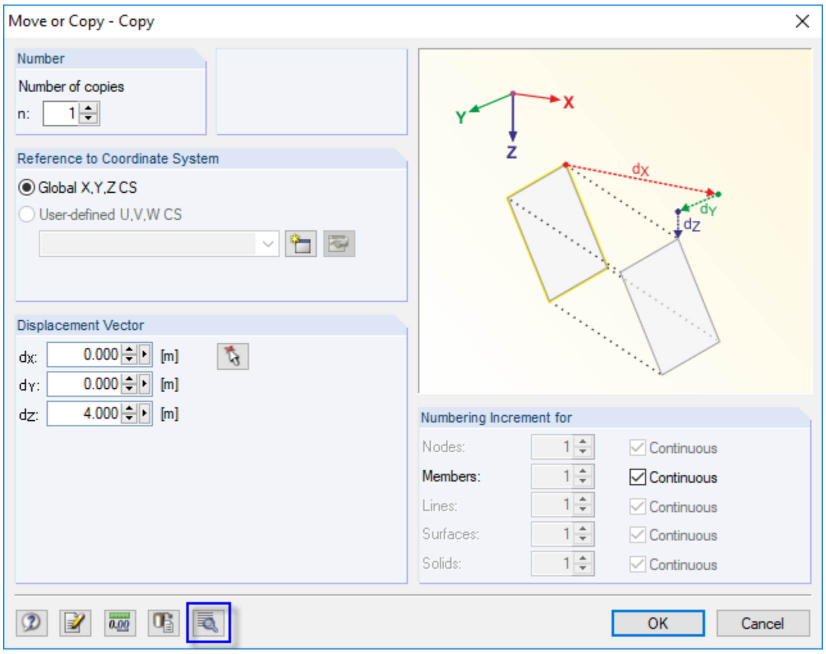


Figure 4.10 *Move or Copy dialog box*

We increase the *Number of copies* to **1**: This way the arc won't be moved but copied. Since the wall is 4 m high (system line), we enter the value **4.0** m as the *Displacement Vector* in *dZ*.

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

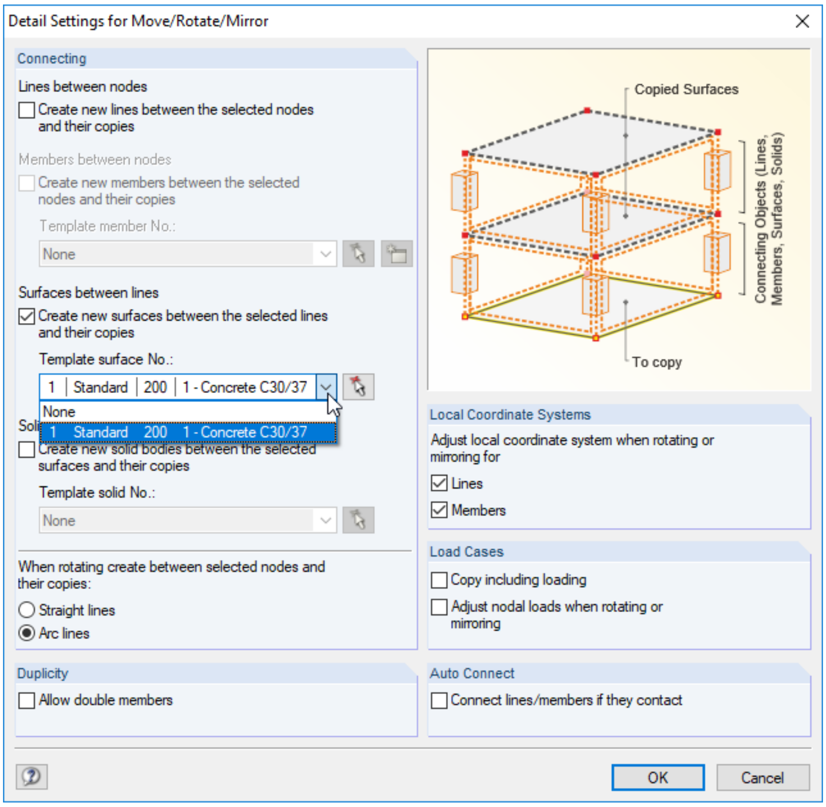


Figure 4.11 Detail Settings for Move/Rotate/Mirror dialog box

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

Create new surfaces between the selected lines and their copies

We select surface **1** from the list as the *Template surface*. This sets the floor slab's properties (material, thickness) as default for the new wall surface.

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

Switching to isometric view

We use the toolbar button on the left to switch to [Isometric View] to display the model in 3D.

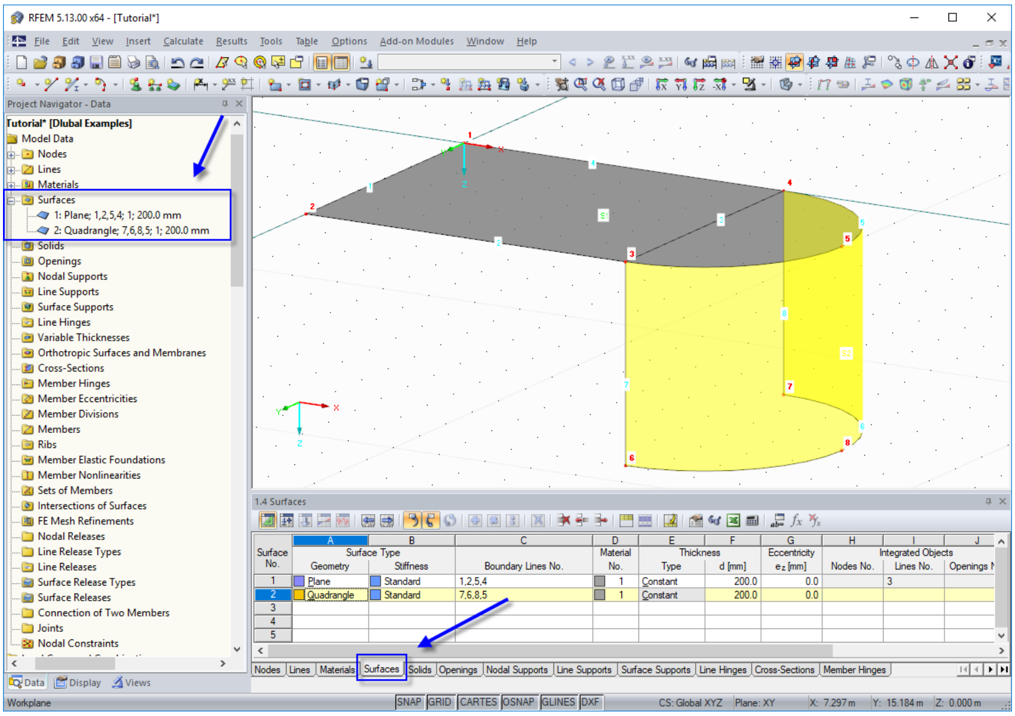


Figure 4.12 Model in isometric view with navigator and table entries

Checking data in the navigator and tables

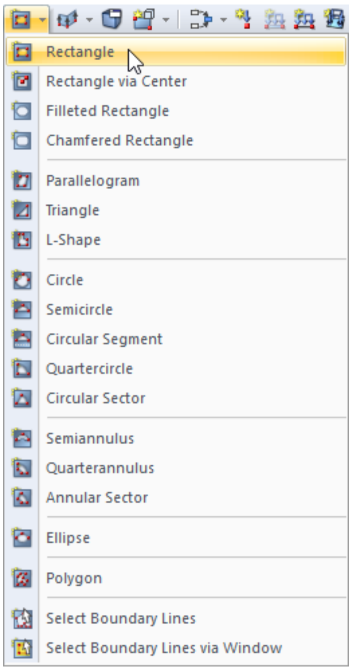
You can find all entered objects in the *Data* navigator's directory tree and in the table's tabs. You can open entries in the navigator by clicking the [+] sign (like in the Windows Explorer). To switch tables, click the individual tabs.

Both surfaces' input data can be found in numerical form in the navigator entry *Surfaces* and in Table *1.4 Surfaces* (see Figure 4.12). RFEM created the wall as a quadrangle surface, which is a shell that is bounded by four lines.

4.2.3 Opening

4.2.3.1 Creating an Opening

We now insert a rectangular opening into the floor slab. It is easier to input our data when we change back to [View in Z-direction].



List of openings

The opening can be applied directly, without a need to define lines first. We choose the *Rectangle* entry from the list of openings.

We set the first opening node to grid point **3.000/1.000/0.000**. We set the second node to grid point **5.000/2.000/0.000**.

The opening is too short that way. We have to adjust its length in the next step.

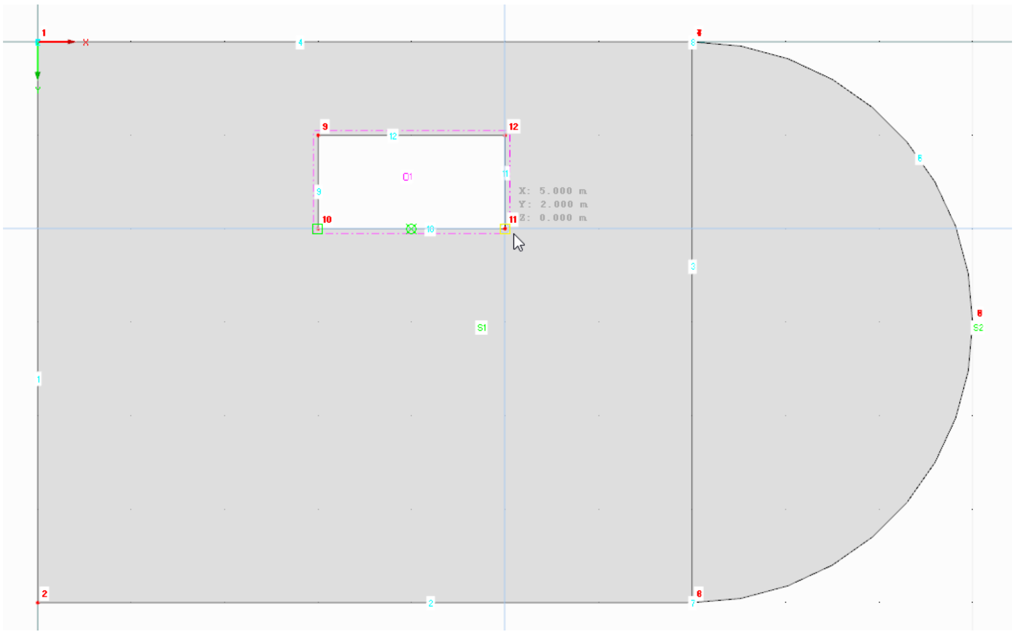


Figure 4.13 Defining the rectangular opening

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

4.2.3.2 Adjusting the Opening

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

We open the *Edit Node* dialog box by double-clicking one of these nodes.

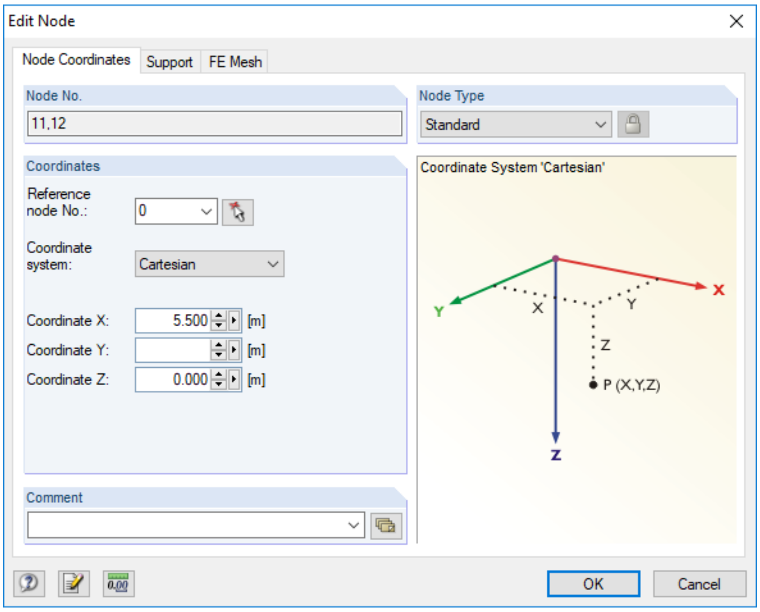
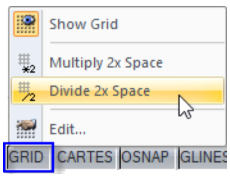


Figure 4.14 *Edit Nodes* dialog box



Both nodes are listed in the *Node No.* box. We correct the *Coordinate X* to **5.500** m and confirm with [OK]. Now the opening has the correct length.

Alternative: We could have set the opening without changing the coordinates by adjusting the grid. To do so, we would have had to reduce *Grid Point Spacing* to 50 cm in the *Work Plane and Grid/Snap* dialog box (see Figure 4.1). To change the grid spacing even quicker, use the [GRID] button's shortcut menu by right-clicking it (see figure to the left).

This completes the input of the surfaces.

4.3 Creating Concrete Members

4.3.1 Columns

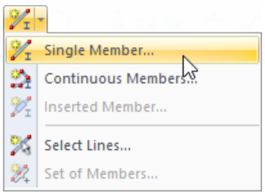
Member elements are tied to lines, which means a line is generated automatically when creating a member.

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 [Work Plane YZ], we click the second of the three plane buttons.

For further input we go back to the [Isometric View]. Now we can see that the input grid is spanned in the plane of the two columns (see Figure 4.18).

Defining a cross-section



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

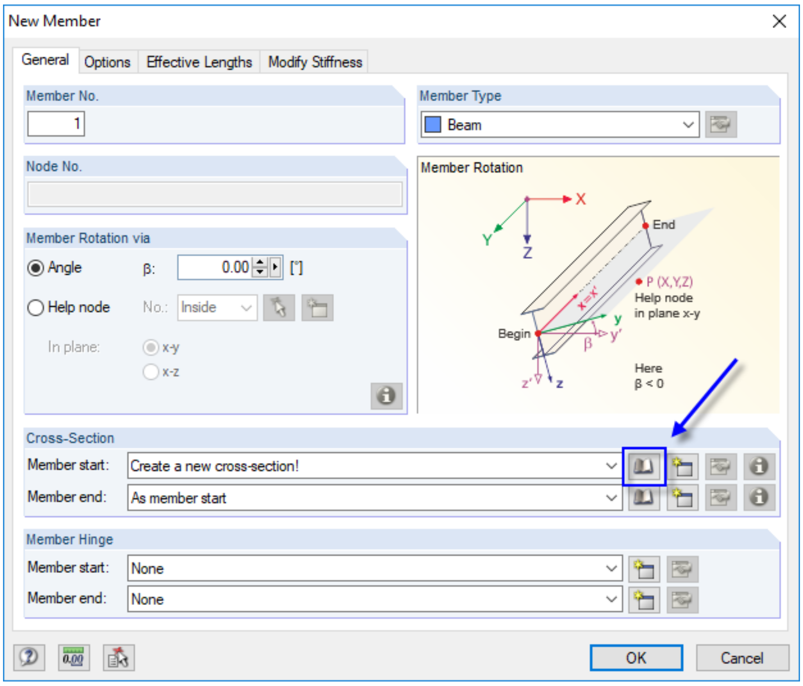


Figure 4.15 *New Member dialog box*

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 cross-section library opens.

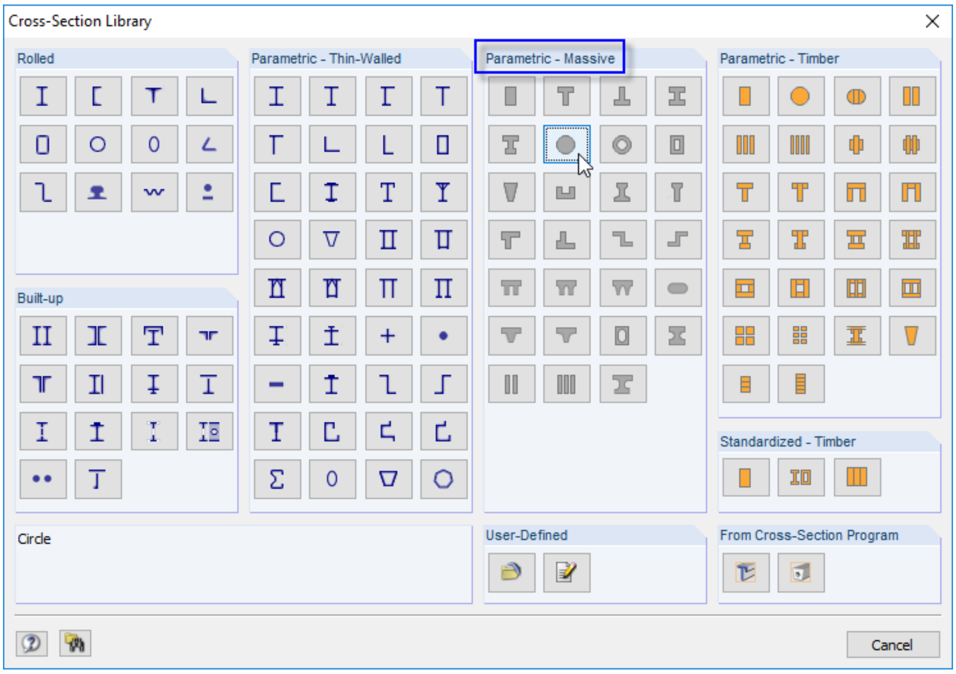


Figure 4.16 Cross-section library

We select the cross-section type *Circle* in the *Parametric - Massive* dialog section. This opens another dialog box.

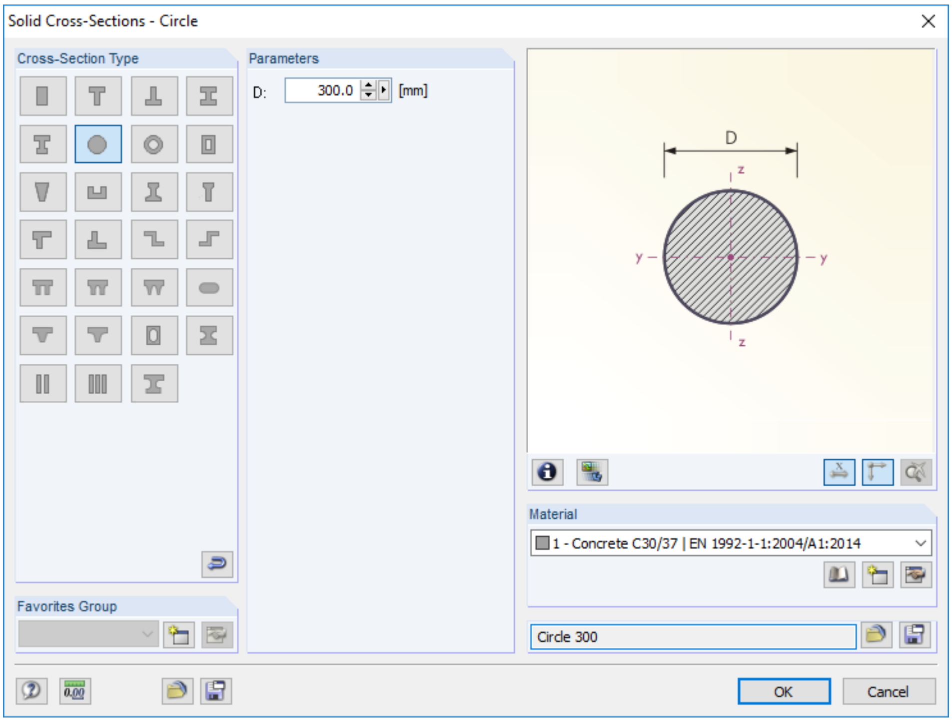


Figure 4.17 *Solid Cross-Sections - Circle* dialog box

We specify the column diameter *D* as **300** mm.

For massive cross-sections, RFEM normally defaults to *Concrete 30/37* as the *Material*.

To check the cross-section's properties, use the [Info] button.

We click [OK] to accept the cross-section's values and return to the initial *New Member* dialog box (see Figure 4.15). The *Member start* box now shows the new cross-section. We close the dialog box with [OK] to graphically set the columns.

Defining members graphically

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

We set the top end of the column at node **2**.

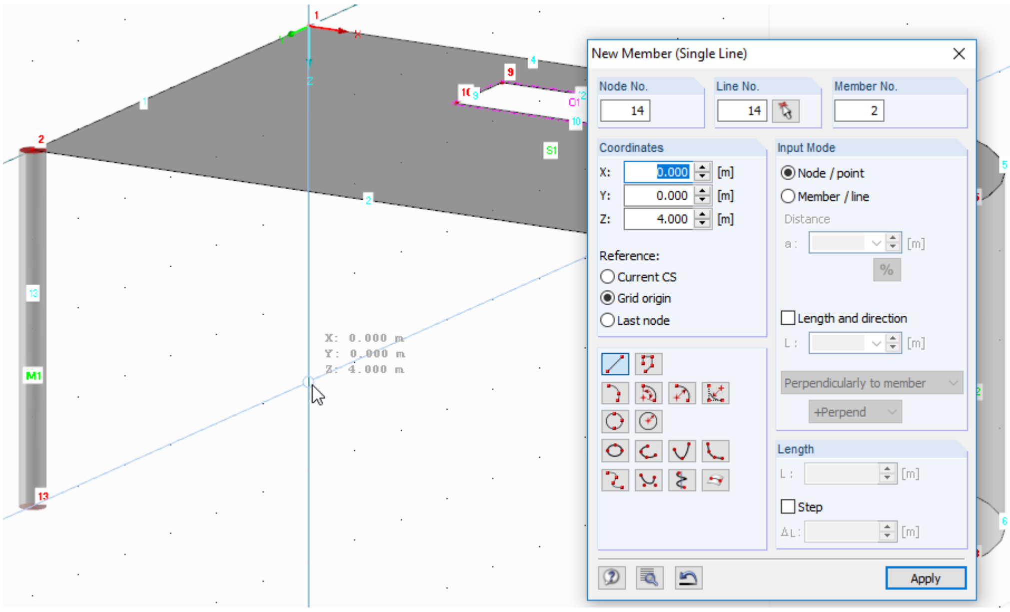


Figure 4.18 Defining the footing of the second column

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

We place the footing of the second column at the grid point **0.000/0.000/4.000** and the top in node **1**.

To quit the input mode, use the [Esc] key or right-click.

4.3.2 Rib

In the next step, we define the downstand beam below the floor.

Modifying line properties

We double-click line 3 to open the *Edit Line* dialog box. We go to the *Member* tab and select the *Available* option (see Figure 4.19).

The *New Member* dialog box opens again.

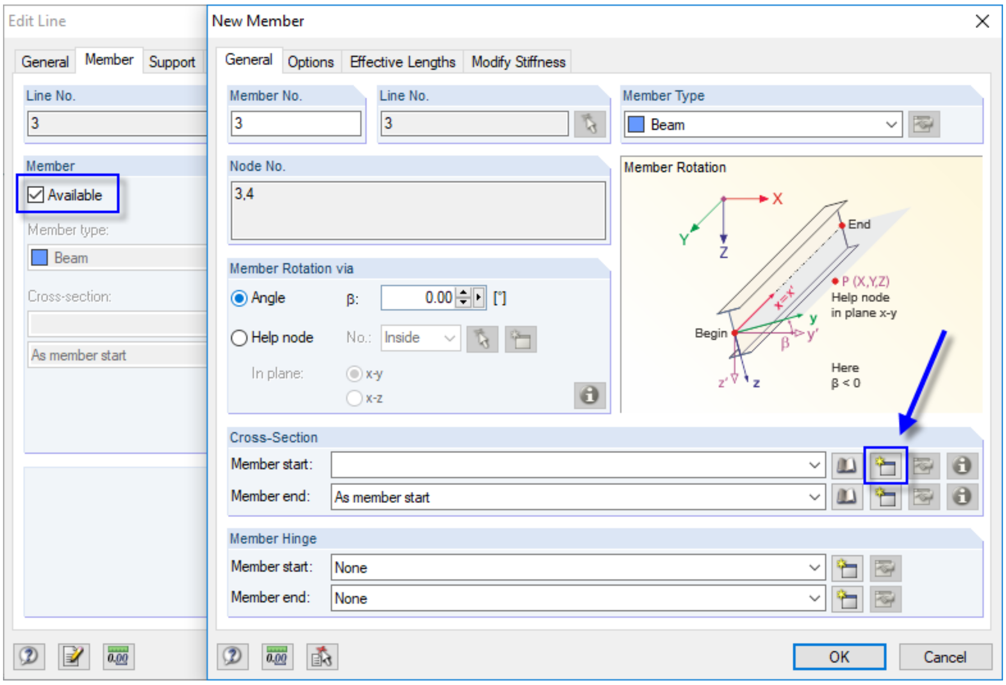


Figure 4.19 *New Member dialog box*

This time we use the button to define a cross-section at the *Member start*.

In the upper part of the *New Cross-Section* dialog box, we select the massive *REC* cross-section table (see Figure 4.20).

This opens the *Solid Cross-Sections - Rectangle* dialog box, where we define the width *b* as **250** mm and the height *h* as **400** mm.

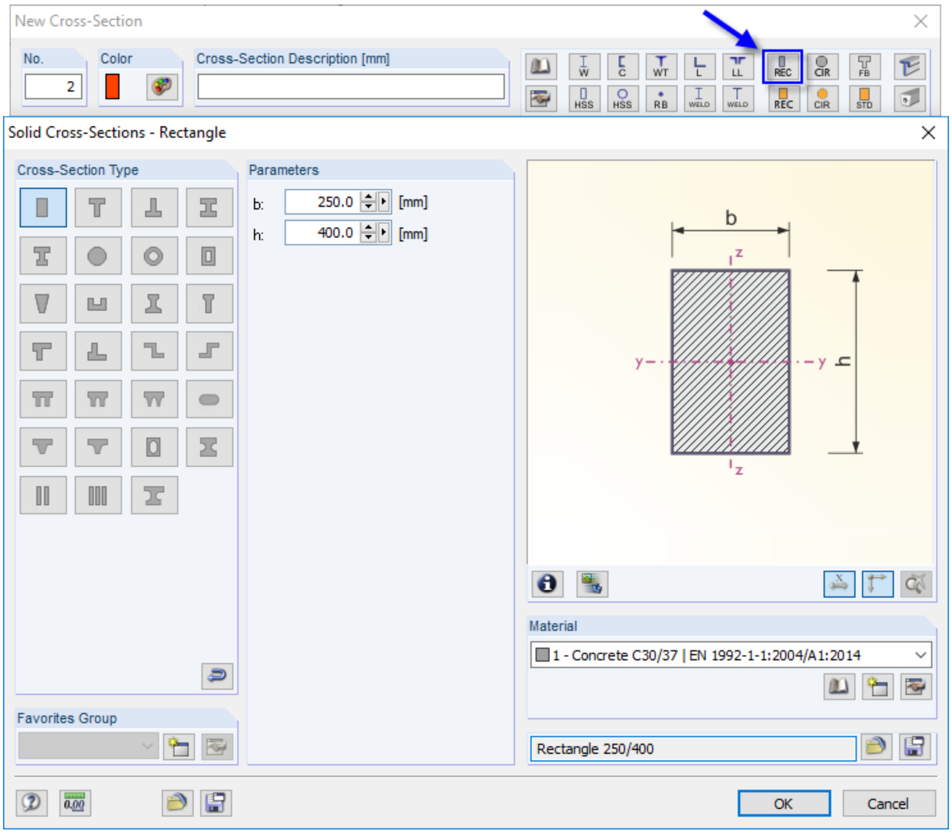


Figure 4.20 *Solid Cross-Sections - Rectangle* dialog box

We click [OK] to accept the cross-section's values into the *New Cross-Section* dialog box. We specify the material as *Concrete 30/37* again.

By clicking [OK], we return to the *New Member* dialog box from earlier. Now the *Member start* box shows the rectangular cross-section.

Defining a rib

In RFEM we can model a downstand beam as a rib member. To do so, we change the *Member Type* in the *New Member* dialog box: We select *Rib* from the list.

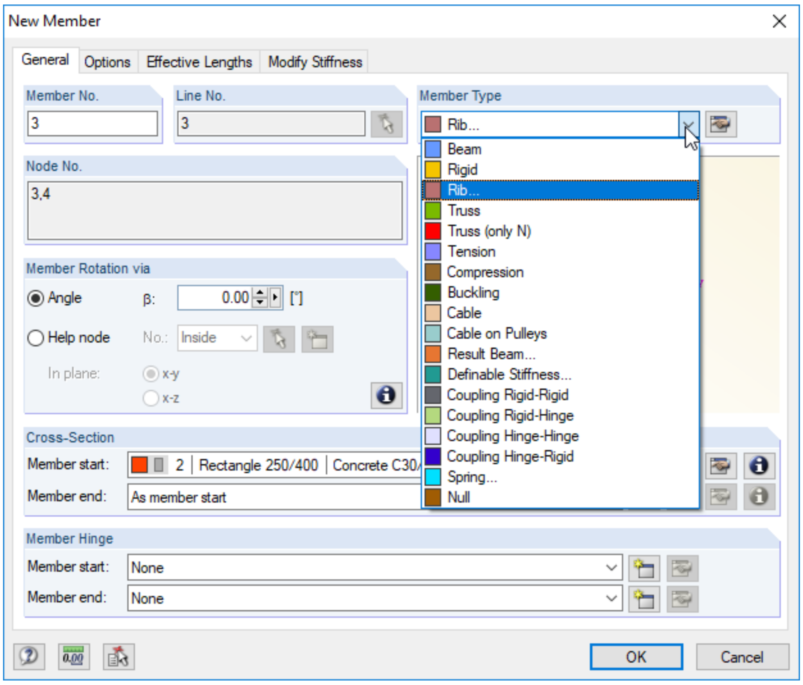


Figure 4.21 Changing the member type

With a click on [icon] we open the *New Rib* dialog box.

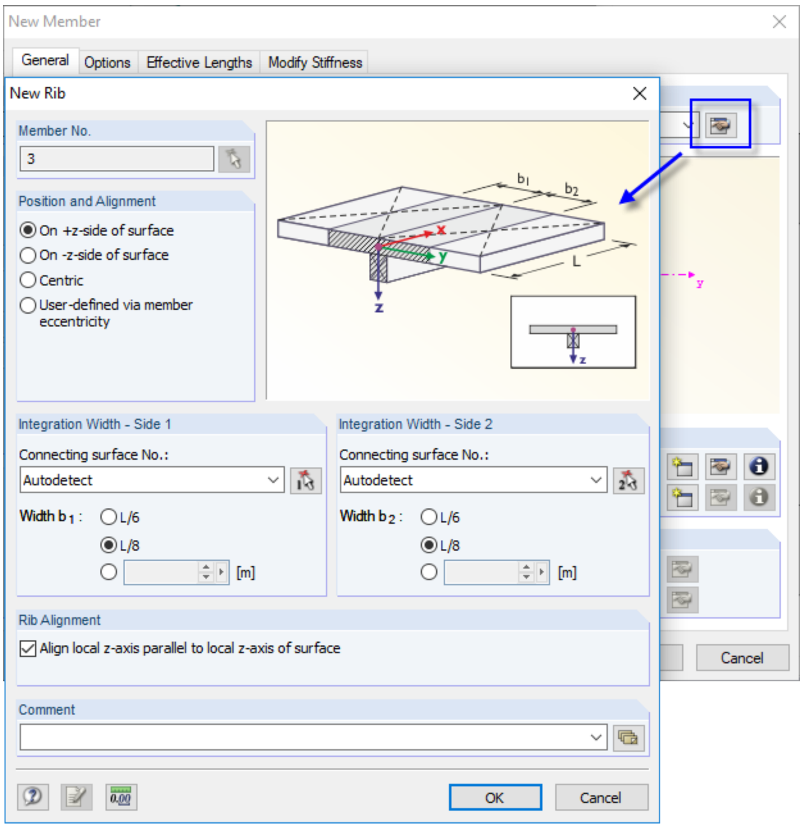


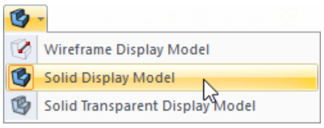
Figure 4.22 Defining the rib

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

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

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

Changing the display model



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

In the *Display* navigator, we select the *Surface* rendering option *Filled incl. thickness*.

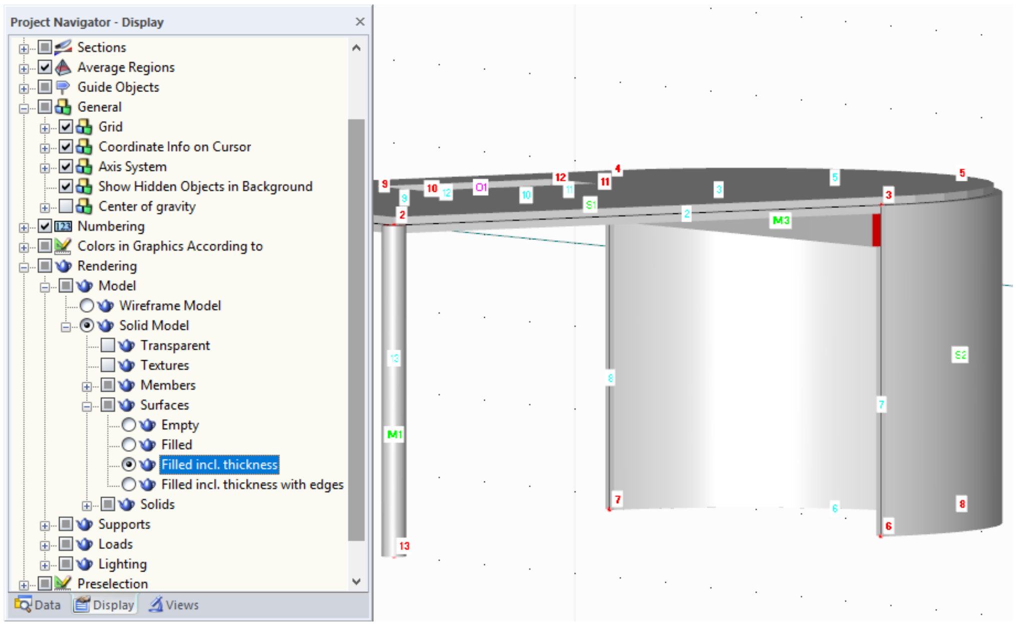
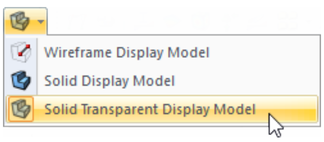


Figure 4.23 Displayed as solid model

To adjust the display, use the [Move, Zoom, Rotate] button (see Mouse functions in chapter 4.1). The pointer turns into a hand. The model can be rotated by pressing down the [Ctrl] key and dragging the pointer.



For further input we change the *Display Model* back to *Solid Transparent*. We also change the rendering for surfaces in the *Display* navigator back to *Filled* 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 column's bottom nodes by drawing a window across the area that includes nodes 13 and 14. Afterwards we click the [New Nodal Support] toolbar button to open the *New Nodal Support* dialog box.

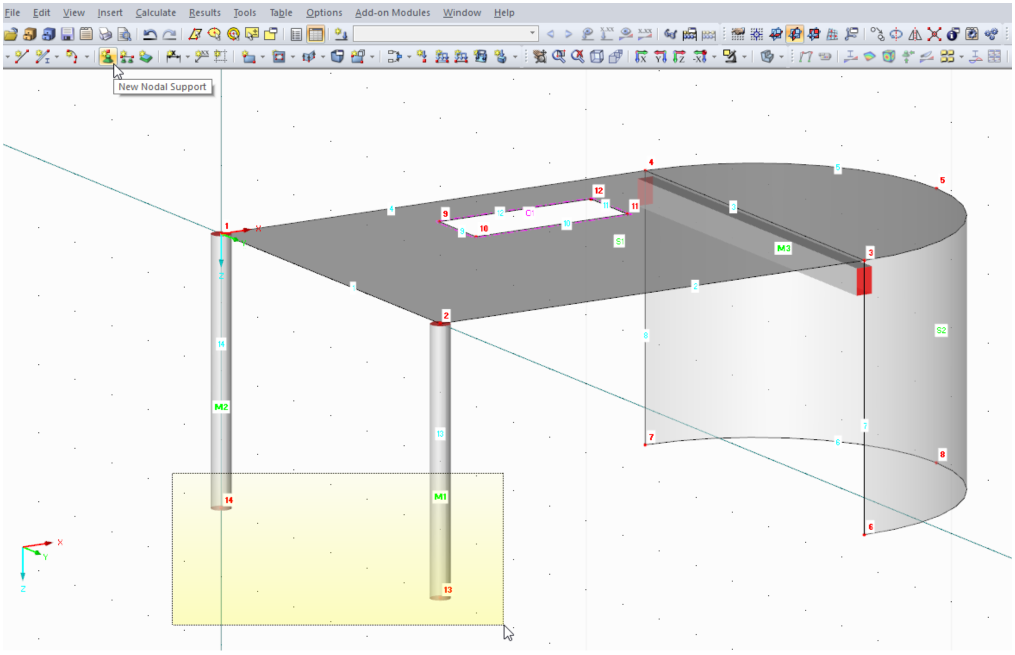


Figure 4.24 Selecting column nodes by drawing a window

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

We modify the support type since the default support type *1* results in restraint around the longitudinal member axis. For that we click  (see Figure 4.25).

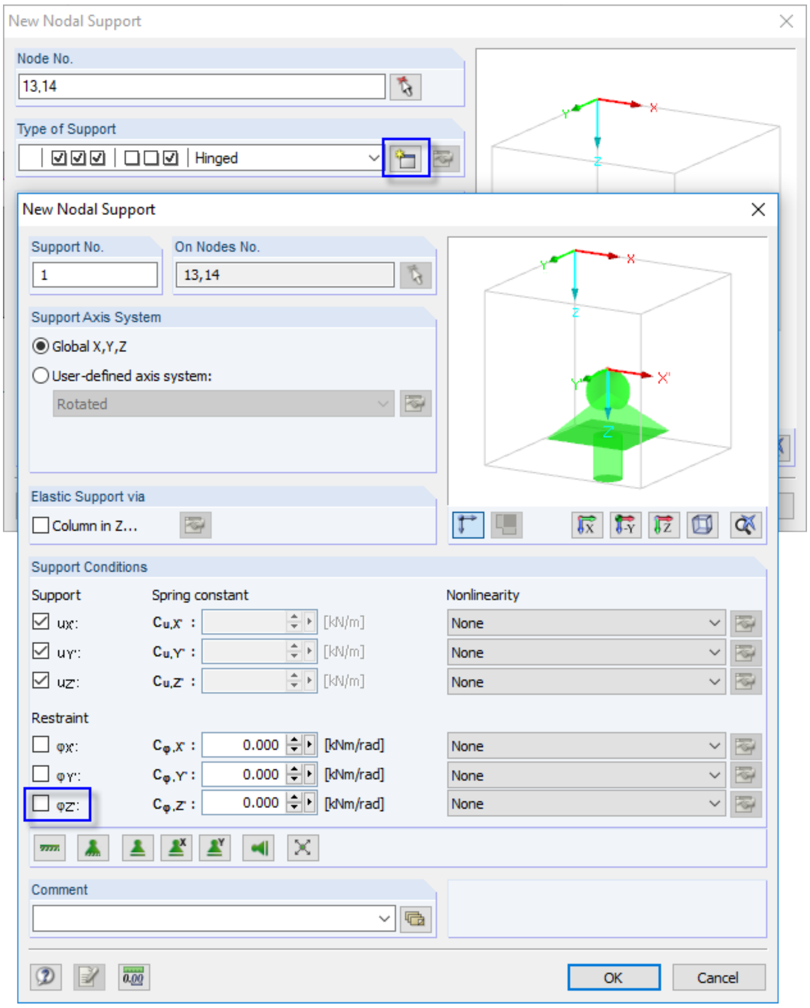


Figure 4.25 Defining degrees of freedom

In the second *New Nodal Support* dialog box, we clear the *Restraint* for rotation φZ'.

We confirm the dialog boxes with [OK] and can see support symbols displayed on the model.

4.4.2 Line Supports

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

We open the *New Line Support* dialog box by clicking the [New Line Support] button.

The *Hinged* option is the default *Type of support*. The first three selected check boxes indicate 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 around X, Y, and Z.

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

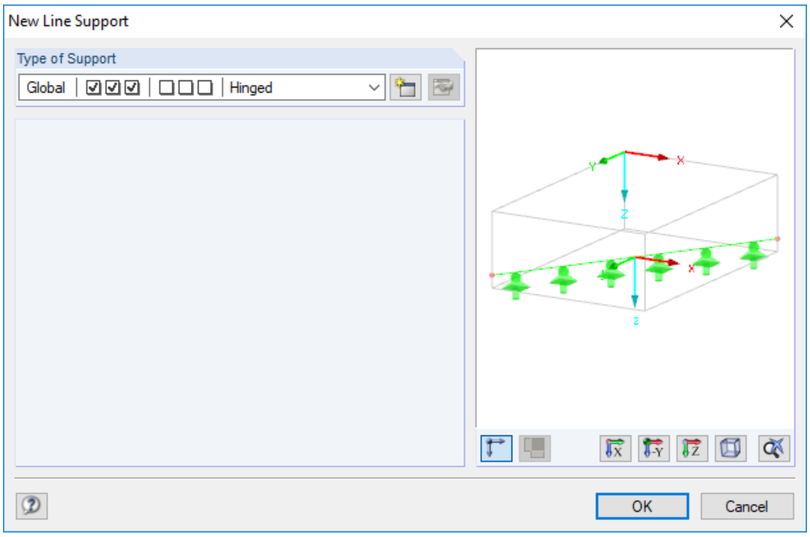


Figure 4.26 *New Line Support* dialog box

RFEM displays a support symbol next to the 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 place the support at the curved line 6.

This completes the input of the model's reinforced concrete construction.

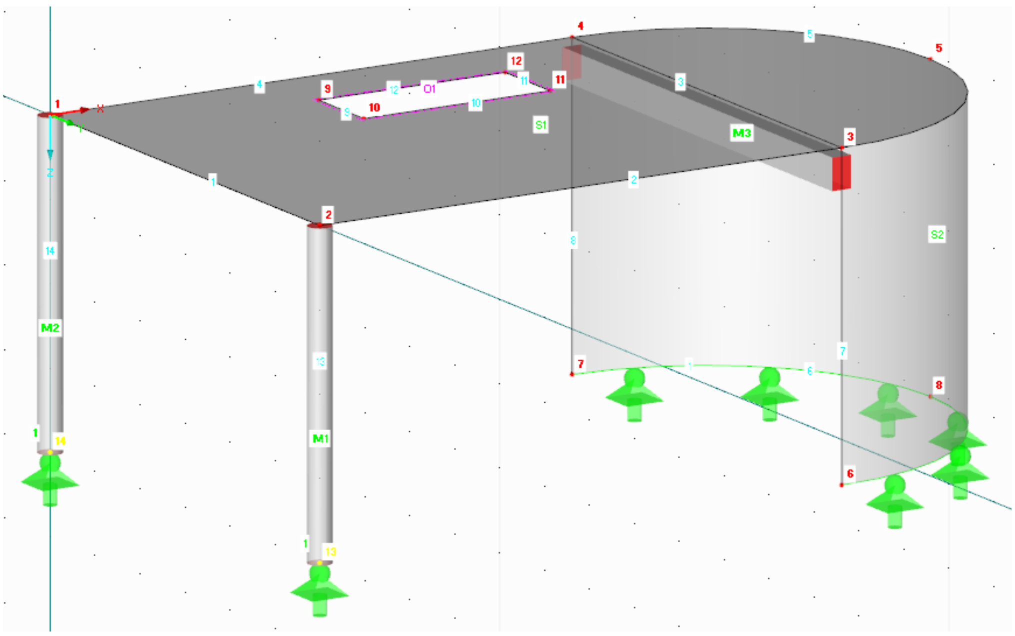


Figure 4.27 Reinforced concrete structure with supports

4.5 Creating Steel Members

4.5.1 Frame

To input the steel construction, we first have to define the frame that lies in the plane of the two columns. For that we create a partial view of the plane: The so-called *Visibility* feature allows us to work in a specific zone of the model without being distracted by objects that lie in a different plane.

Creating custom visibilities

To create visibilities, we use the *Views* tab in the navigator. A number of visibilities is already available. RFEM generated them based on input data.

To graphically extract a specific area from the model, use the [Visibility by Window] button: We activate the feature and use the mouse to draw a window from left to right that completely encompasses both column members.

When you draw the window from left to right, the visibility only includes objects that are completely contained within the window. When you draw the window from right to left, the partial view additionally includes objects that intersect with the window.

The rest of the model (floor, wall) is now displayed in a lower color intensity. These objects cannot be edited.

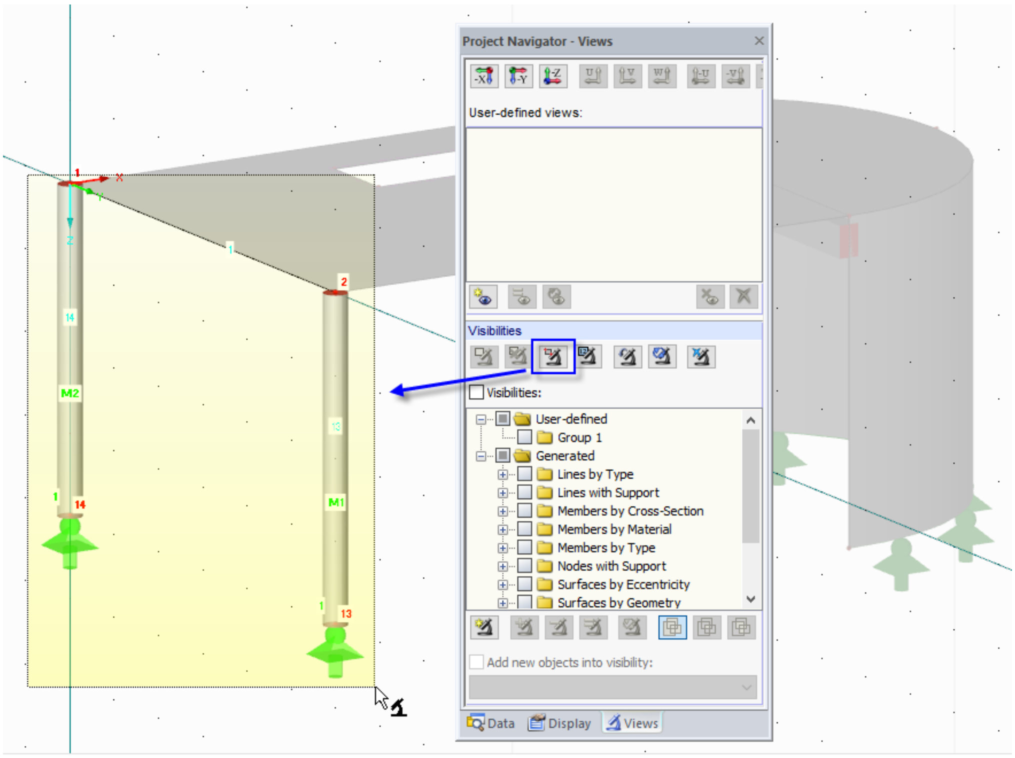
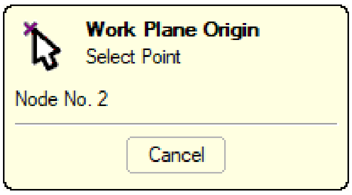


Figure 4.28 Creating a visibility by window

Changing the work plane's origin

Plane YZ is still set as the work plane, which is suitable for defining the frame of the monopitch roof. The work plane's origin suits our purpose as well. However, to demonstrate how to adjust the work plane, we will modify the position of the work plane's origin.

We click the [Set Origin] toolbar button. In the work window, we select node 2 to be the new origin of the work plane - the head of the front column. The grid's crosshair changes its position.



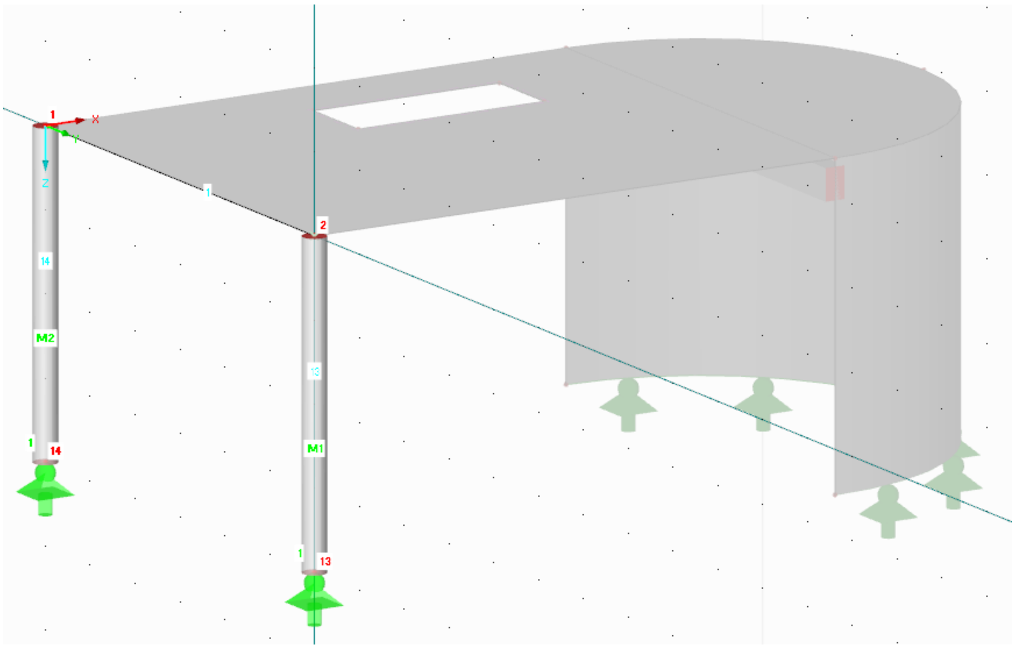
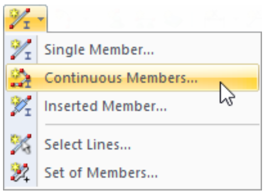


Figure 4.29 New origin in node 2



4.5.1.1 Defining Members Continuously

We want to create the monopitch roof frame as a polygonal chain. We click the [New Member] button and select *Continuous Members*.

The *New Member* dialog box opens and we check to see if the *Member Type* **Beam** is selected.

As shown in Figure 4.19, we create a cross-section for the *Member start* using [button]. In the *New Cross-Section* dialog box, we click the [HE-A] button on the right. If HE-A is not one of the buttons, click the [Import Cross-Section from Library] button and select *I-Sections*. Then, in the *Rolled Cross-Sections - I-sections* dialog box, we select the cross-section **HE A 300** among the HE A cross-sections. For rolled cross-sections, RFEM usually defaults to number 2 - *Steel S 235*.

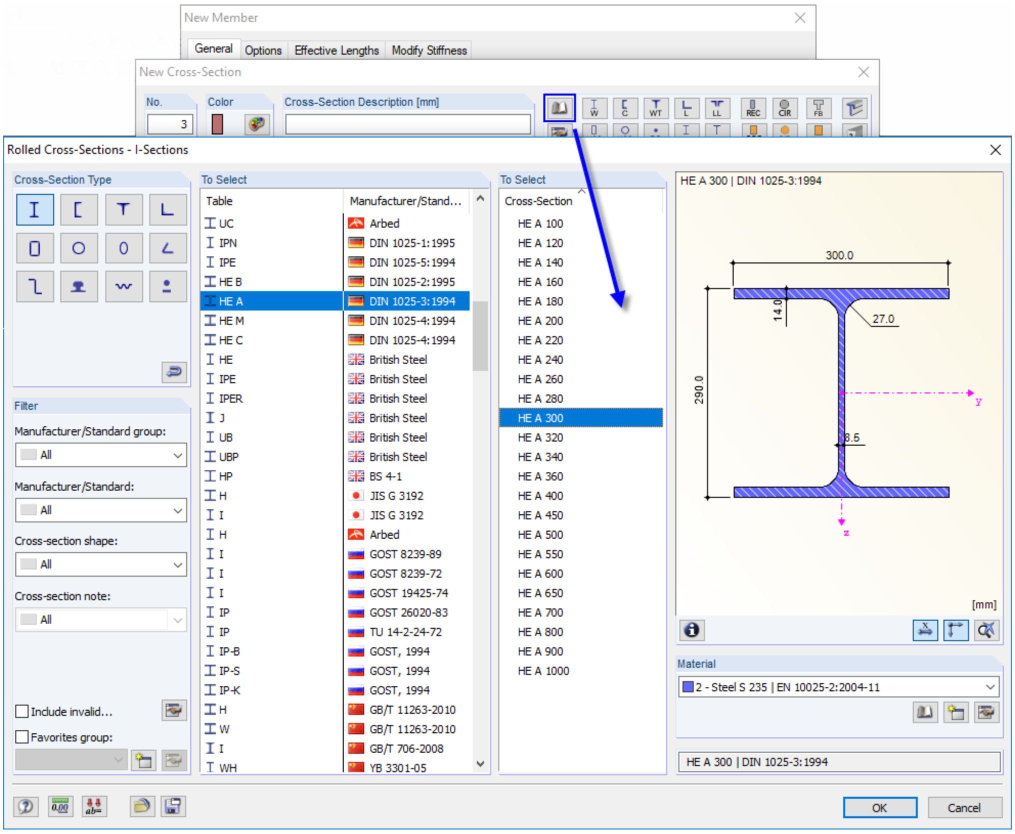


Figure 4.30 Selecting HE A 300 in the library

We confirm all dialog boxes with [OK].

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

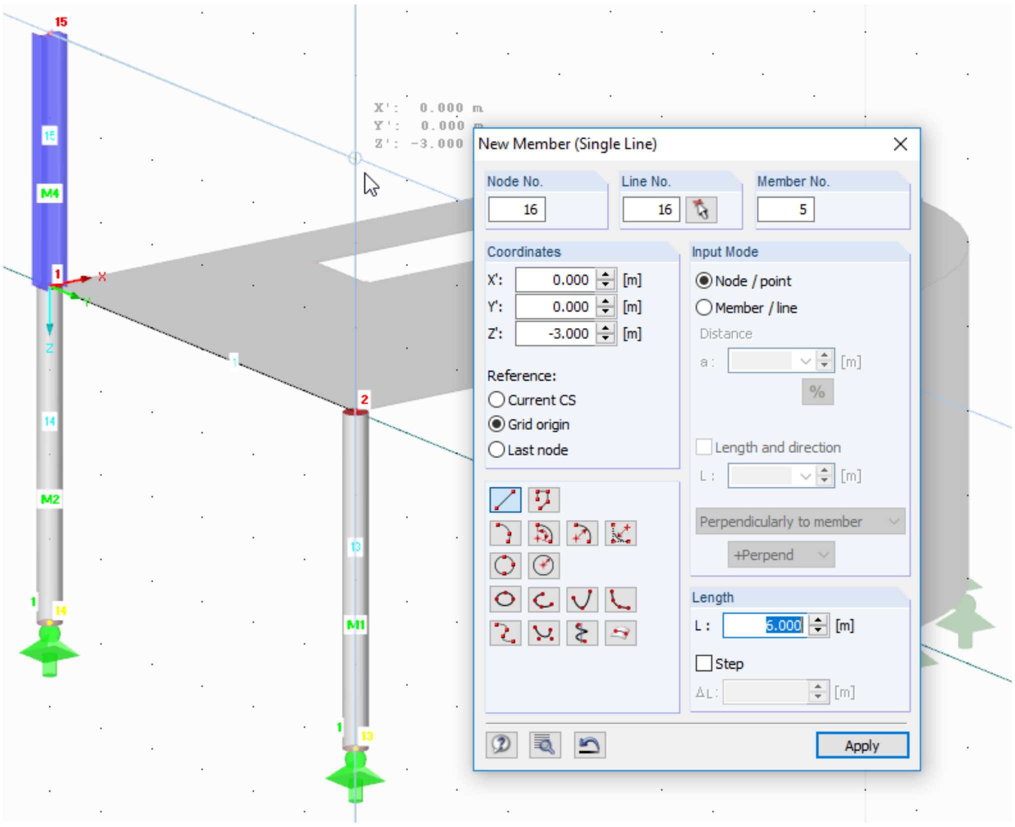


Figure 4.31 Defining members continuously

Once 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 floor nodes 1 and 2. Though this kind of restraint can hardly be built in reality, we forgo a modeling of hinge properties in our example and accept the simplification.

4.5.1.2 Chamfering Horizontal Beams

The monopitch roof has an inclination of 8°. Therefore, we have to adjust the horizontal beam.

We draw a selection window across member 5 that also comprises both end nodes. Then we select the menu option

Edit → Chamfer

to open the *3D Chamfer* dialog box.

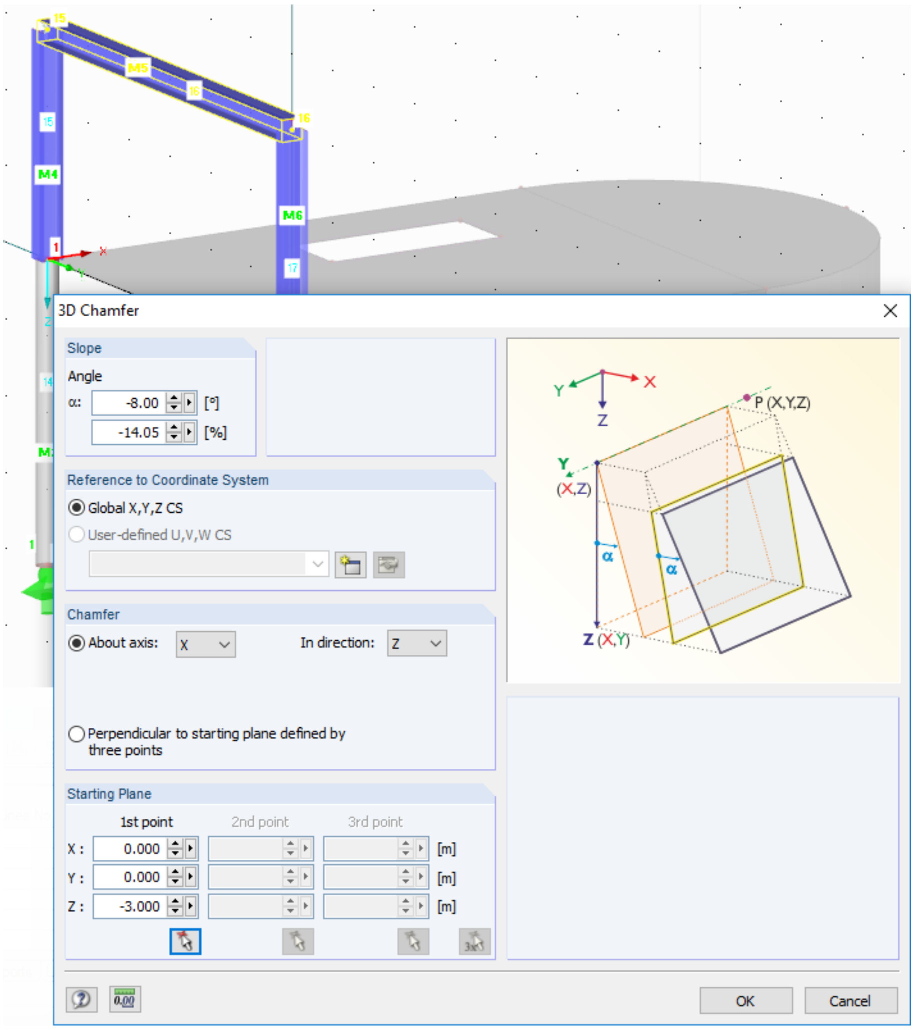


Figure 4.32 Adjusting the horizontal beam's inclination

We want to modify the beam's inclination by **−8°** *About axis* **X**. We have to enter a negative value because the objects will be rotated counterclockwise about the X-axis. The chamfering is applied vertical *In direction* **Z**. We define the rotation axis' *1st point* using . We select node *15* with the coordinates **0.000/0.000/-3.000** and click [OK] to confirm the input.

4.5.1.3 Connecting Beams with Hinges

The horizontal beam cannot transfer any bending moments into the columns because of its connection type.

Defining hinges

We use the *Data* navigator to define the type of hinge: We right-click *Member Hinges* and select *New Member Hinge* in the shortcut menu.

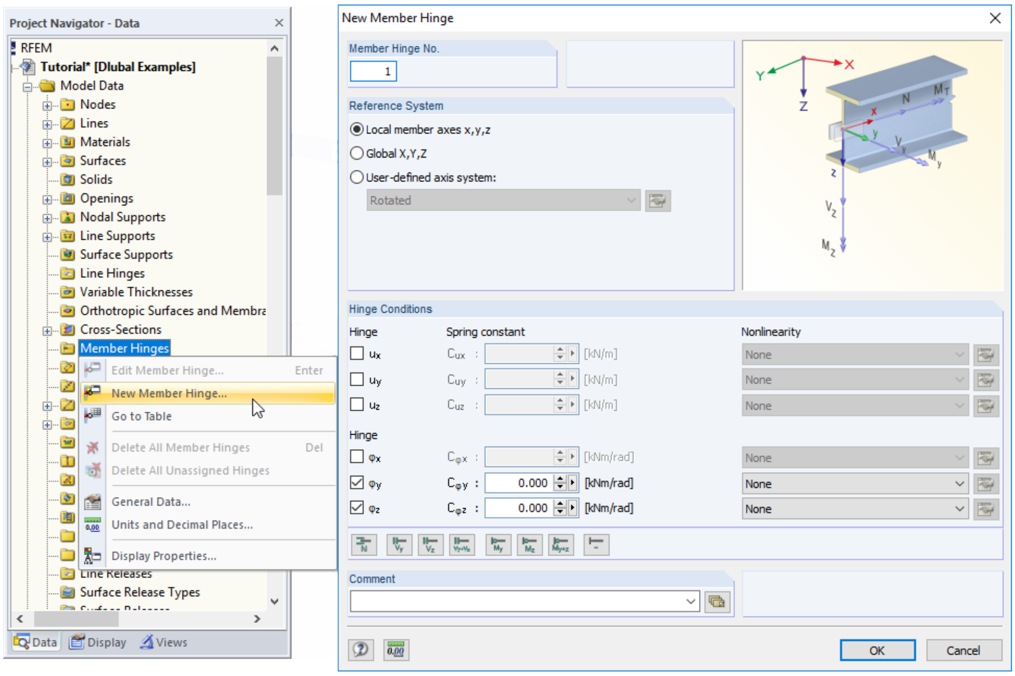


Figure 4.33 Opening the *New Member Hinge* dialog box using the *Data* navigator

In the *New Member Hinge* dialog box, we have to select the displacements or rotations that are released at the member end. In our case they are the rotations φ_y and φ_z, which means no bending moments can be transferred at the node.

Without changing anything, we close the dialog box by clicking [OK].

Assigning hinges

We could double-click the member to open the *Edit Member* dialog box and assign the hinges. However, we use a special feature available in the following menu:

Insert → Model Data → Member Hinge → Assign to Members Graphically

This opens the *Assign Member Hinges to Members Graphically* dialog box. In the list we select type 1, which we have just defined, and click [OK].

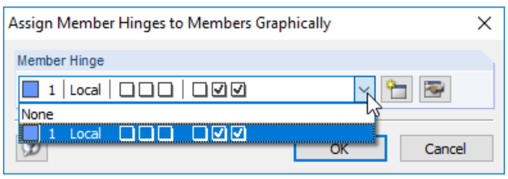


Figure 4.34 *Assign Member Hinges to Members Graphically* dialog box

In the work window we can see that RFEM has applied a one-third division to the members. By clicking the end of a member, we can graphically define the hinge at that member end. We click member *5* in its middle area to assign the hinge to both sides (see Figure 4.35).

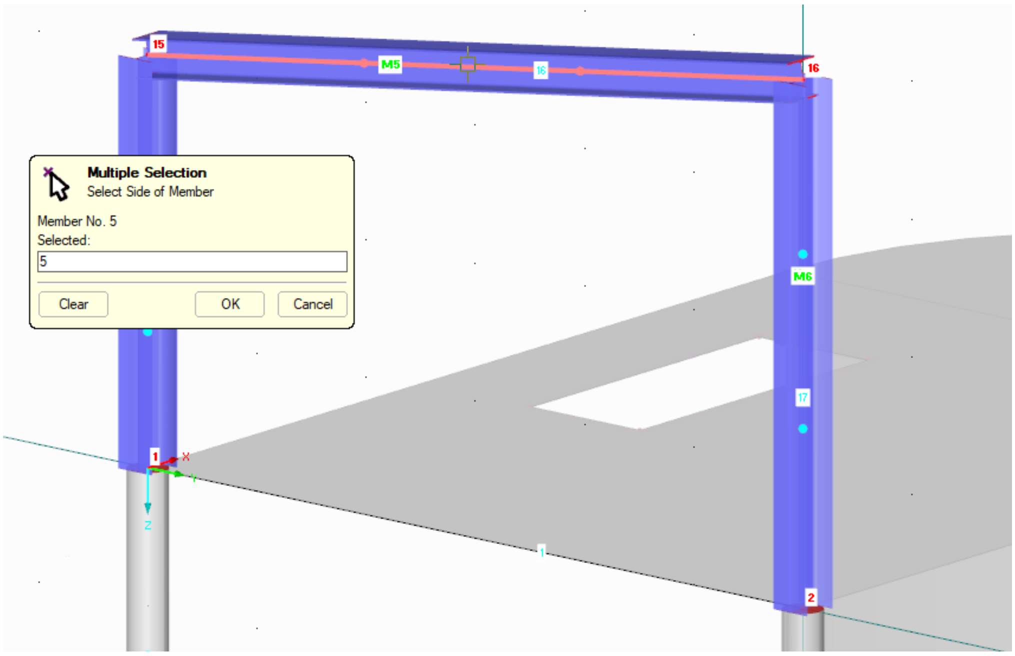


Figure 4.35 Assigning a hinge graphically

4.5.1.4 Reversing Member Orientation

For the graphical display of imperfections, it may be suitable if the member orientation of columns is directed from bottom to top. For that reason, we change the orientation of the right steel column, using a feature from the member shortcut menu.

We move the pointer near member 6, which displays the orientation arrow on the member. We right-click the member and open its shortcut menu. We select *Reverse Member Orientation*.

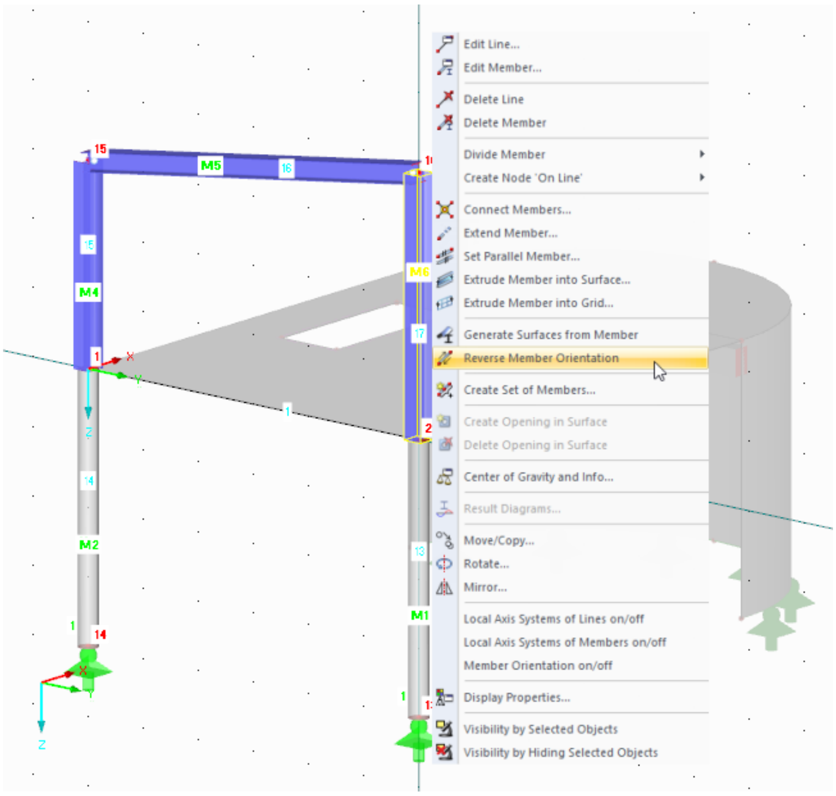
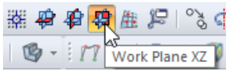


Figure 4.36 Member shortcut menu

4.5.1.5 Copying a Frame

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

We draw a selection window across the frame that encompasses members *4* to *6*. Please take care not to include any of the concrete columns! If necessary, rotate the model for a more favorable view or click the members one after the other while holding down the [Ctrl] key.



Before we create the copy, we set the work plane to [Work Plane XZ] so we can copy the structure from the frame.

We press the [Ctrl] key. Now we grab our selection near the foot point of the higher column (node 2) and drag it to the end of the arc at the upper wall. The [+] symbol next to the pointer indicates that the objects are being copied.

As soon as the grid point coordinates **7.000/6.000/0.000** are displayed in the status bar, we release the mouse button.

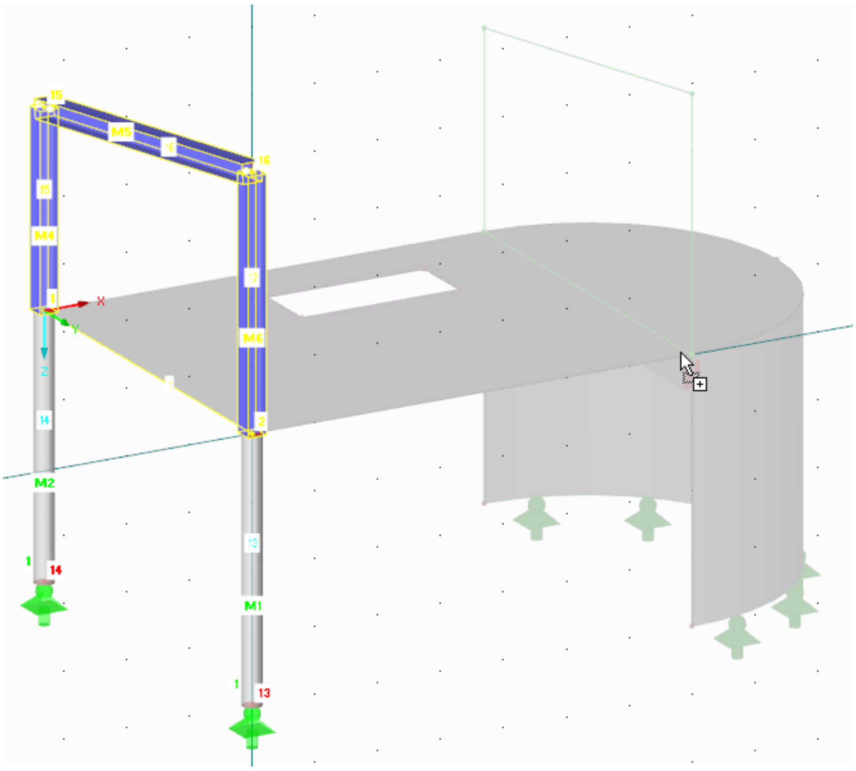
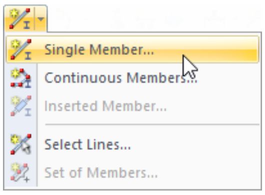


Figure 4.37 Copying the frame using drag-and-drop

Nodes and lines are automatically merged with already defined objects.

4.5.2 Purlins

4.5.2.1 Defining Members Individually



To define both purlins, we use the [New Member] button in the toolbar. We select *Single Member* and open the *New Member* dialog box.

Using , we define a cross-section for the *Member start* once more (see Figure 4.19).

In the *New Cross-Section* dialog box, we click the [HE-B] button at the top or select I-*Sections* from the library. In the *Rolled Cross-Sections - I-Sections* dialog box, we select the **HE B 260** cross-section from among the HE B cross-sections. Number 2 - *Steel S 235* is the default material again.

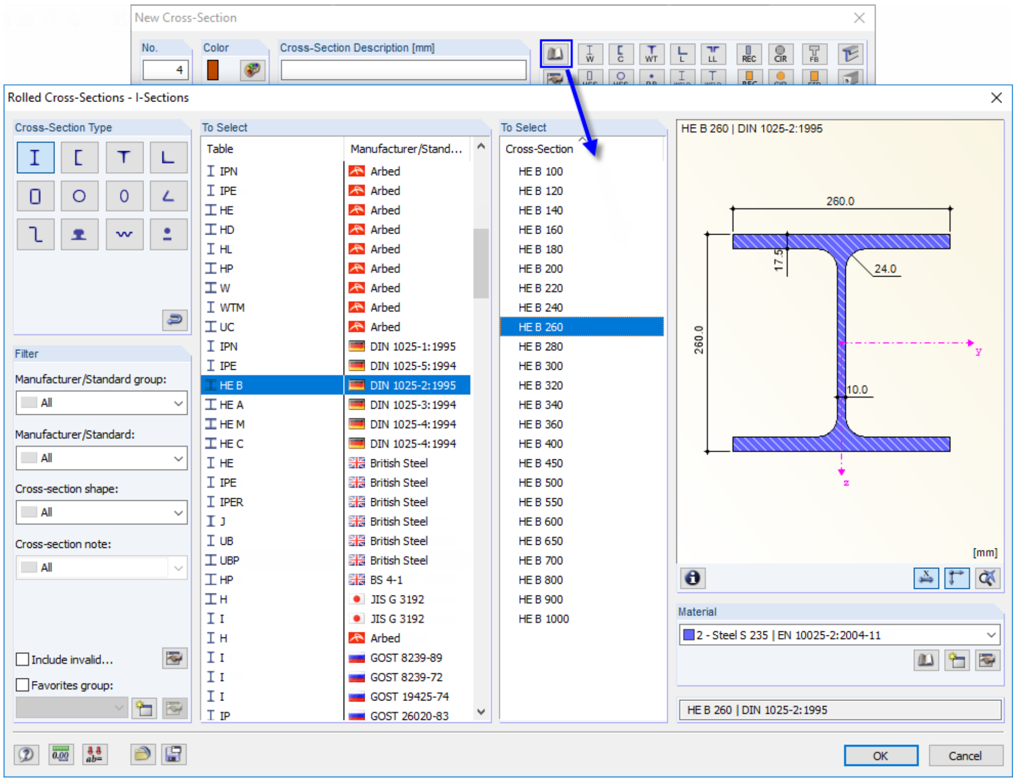


Figure 4.38 Selecting HE B 260 in the library

We confirm all dialog boxes with [OK].

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

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

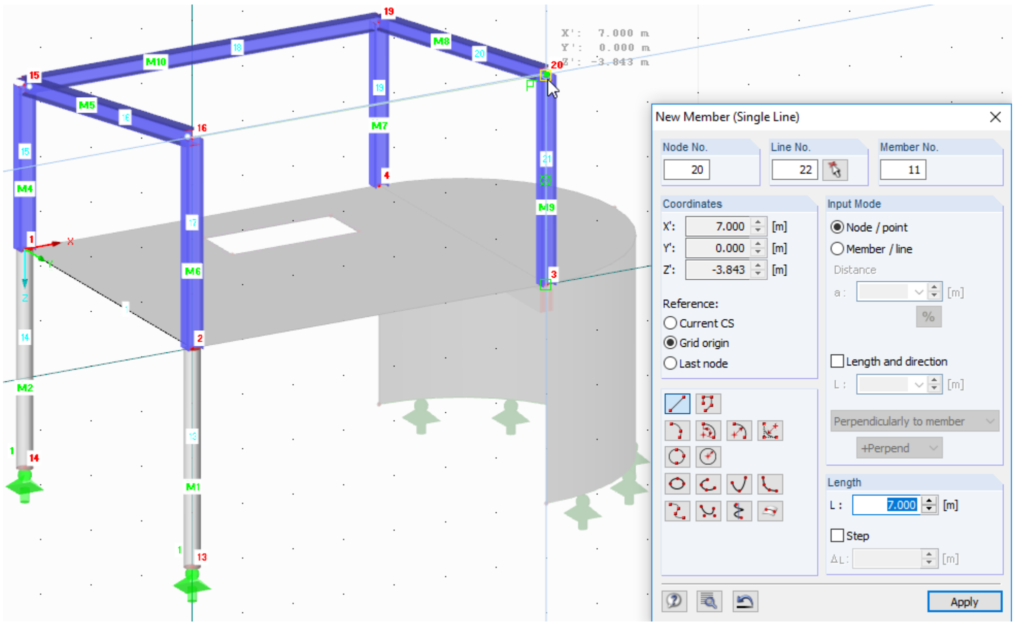


Figure 4.39 Defining purlin members

To quit the input mode, use the [Esc] key or right-click.

4.5.2.2 Connecting Members Eccentrically

We want to connect the purlins eccentrically to the frame columns. This shortens the system line by half of the cross-section height of the HE A 300 columns.

Defining eccentricity

We double-click the purlin at the high eave (member *11*). In the *Edit Member* dialog box, we go to the *Options* tab. Now we click in the *Member Eccentricity* section to open the *New Member Eccentricity* dialog box.

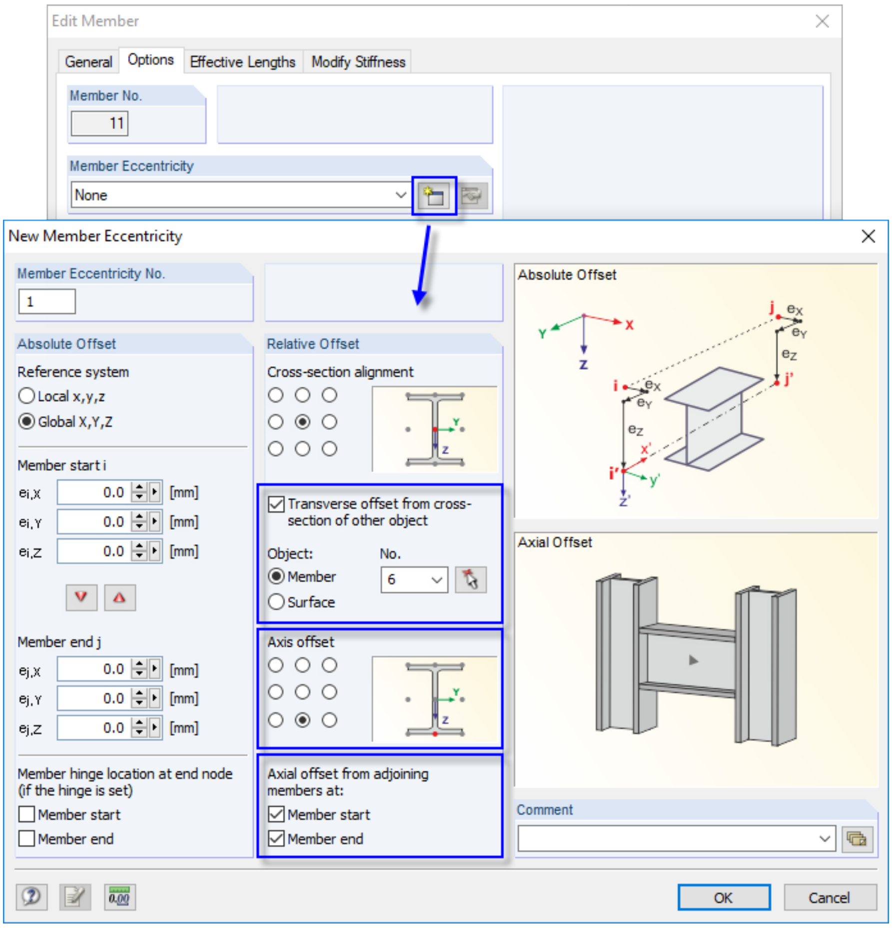


Figure 4.40 *New Member dialog box, Options tab, New Member Eccentricity dialog box*

We select *Transverse offset from cross-section of other object*. In our case, the *Object* is the column: Using we select **Member 6** graphically.

We define the *Axis offset* in the direction of the positive cross-section axis z.

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

After confirming all dialog boxes, we can check the result in magnified view (for example zooming by rotating the wheel, moving by holding down the wheel button, rotating by pressing down the wheel button and right mouse button at the same time).

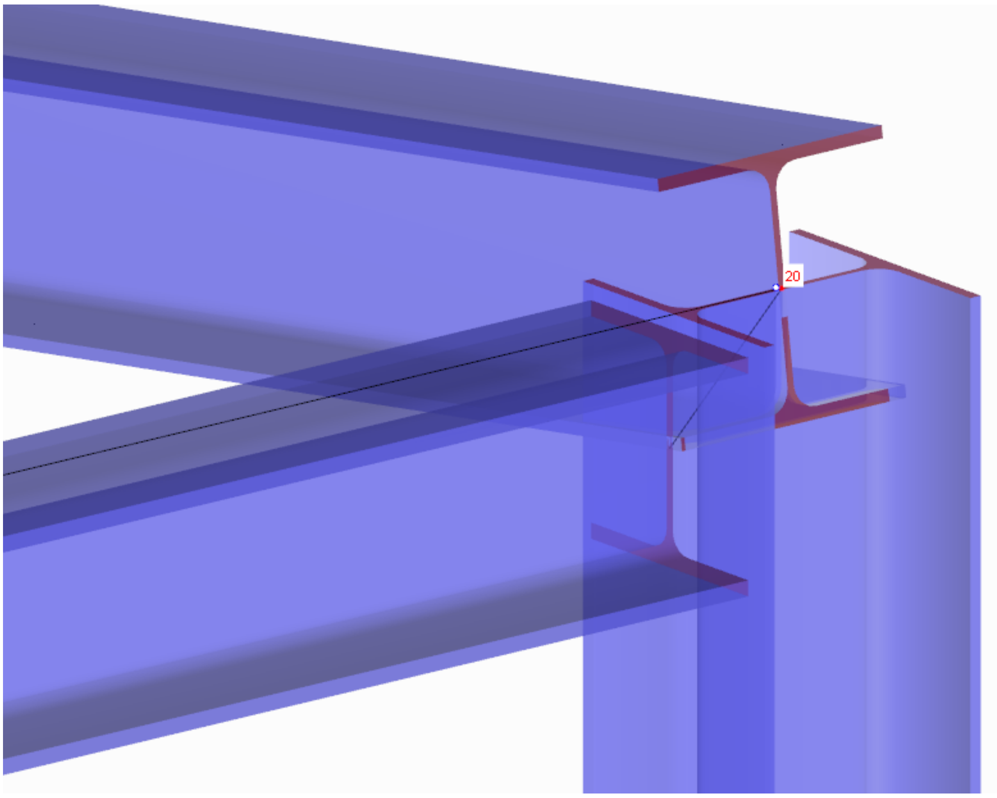
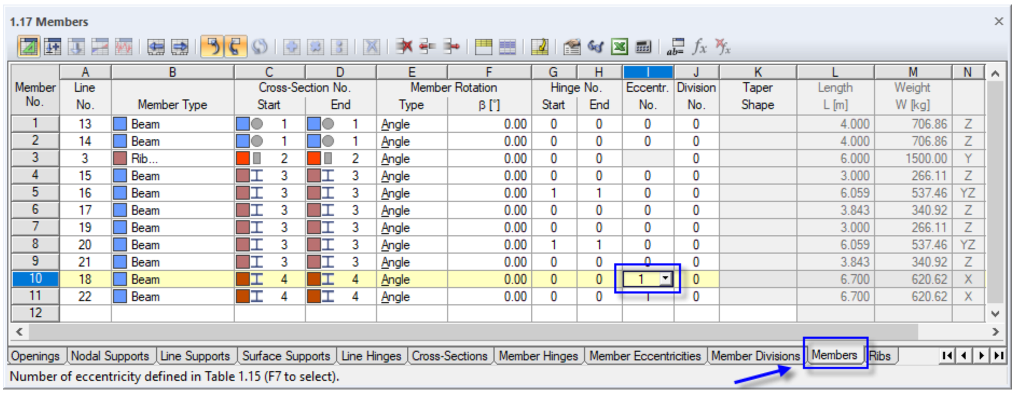


Figure 4.41 Eccentric connection of purlin in zoomed view

Applying eccentricity to another member

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

We browse to Table *1.17 Members*, which lists the member data of all previously defined members numerically. When we click row 10, we can see that the second purlin is highlighted in the work window in the selection color.



1.17 Members

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 β [°]	G Hinge No. Start	H Hinge 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	706.86	Z
2	14	Beam	1	1	Angle	0.00	0	0	0	0		4.000	706.86	Z
3	3	Rib...	2	2	Angle	0.00	0	0		0		6.000	1500.00	Y
4	15	Beam	3	3	Angle	0.00	0	0	0	0		3.000	266.11	Z
5	16	Beam	3	3	Angle	0.00	1	1	0	0		6.059	537.46	YZ
6	17	Beam	3	3	Angle	0.00	0	0	0	0		3.843	340.92	Z
7	19	Beam	3	3	Angle	0.00	0	0	0	0		3.000	266.11	Z
8	20	Beam	3	3	Angle	0.00	1	1	0	0		6.059	537.46	YZ
9	21	Beam	3	3	Angle	0.00	0	0	0	0		3.843	340.92	Z
10	18	Beam	4	4	Angle	0.00	0	0	1	0		6.700	620.62	X
11	22	Beam	4	4	Angle	0.00	0	0	1	0		6.700	620.62	X
12														

Openings | Nodal Supports | Line Supports | Surface Supports | Line Hinges | Cross-Sections | Member Hinges | Member Eccentricities | Member Divisions | Members | Ribs

Number of eccentricity defined in Table 1.15 (F7 to select).

Figure 4.42 Assigning eccentricity in Table *1.17 Members*

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

After leaving the table cell using either the [Tab] or [↵] key, the changes are displayed in the graphic.

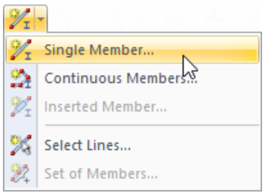
It would be possible to define another eccentricity for the horizontal beams in the same way. As these members are connected to the column webs, however, we neglect these additional moments in our example.

4.5.3 Diagonal

The final member that we insert is a diagonal for stiffening, only capable of transferring tensile forces. Generally, bracings are defined crosswise, but the demo version's calculation only allows for twelve 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 the Member



Using the [Single Member] button, we open the *New Member* dialog box again. We select **Tension** as the *Member Type*.

We define a new cross-section for the Member start, which we import from the database using (see Figure 4.19).

In the *New Cross-Section* dialog box, we click the [L] button, and in the subsequent *Rolled Cross-Sections - Angles* dialog box, we select the EN standard cross-section **L 80x80x8**. The default material is number 2 - *Steel S 235* again.

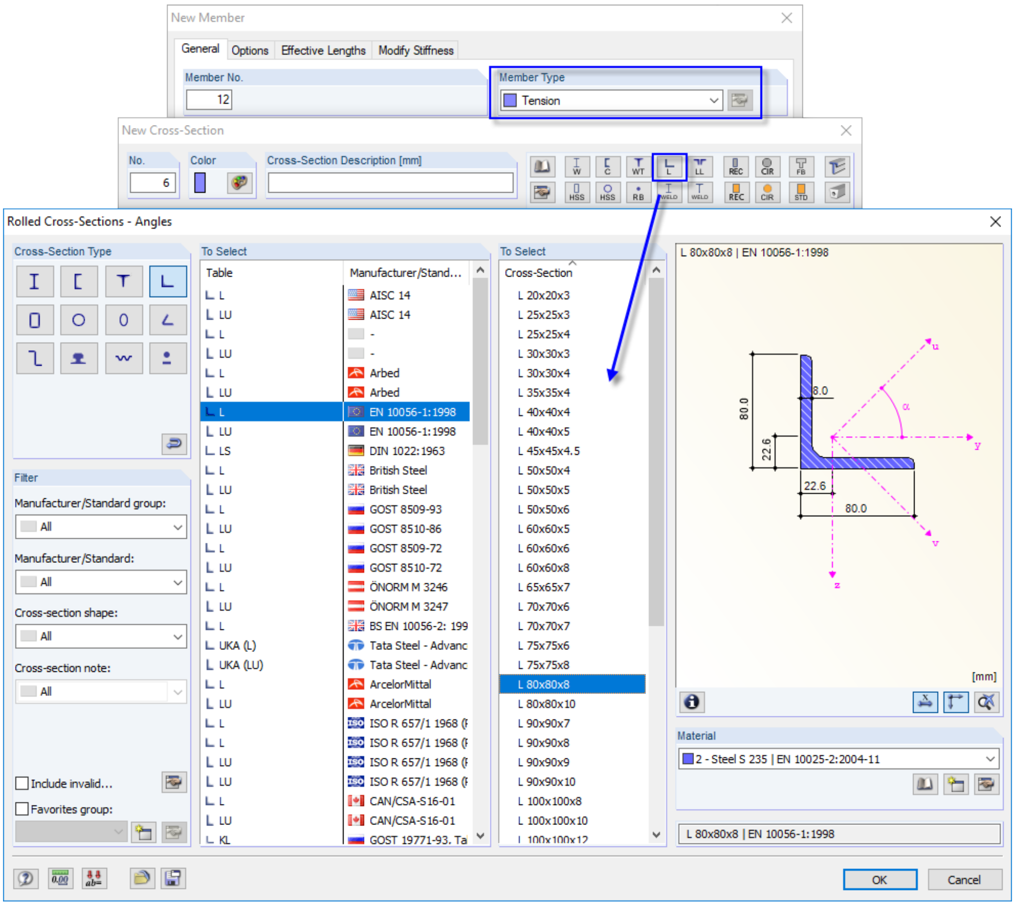


Figure 4.43 Defining tension member with cross-section L 80x80x8

We confirm all dialog boxes with [OK] and click nodes **15** and **2** one after the other to define the diagonal (see Figure 4.44).

To quit the input mode, use the [Esc] key or right-click.

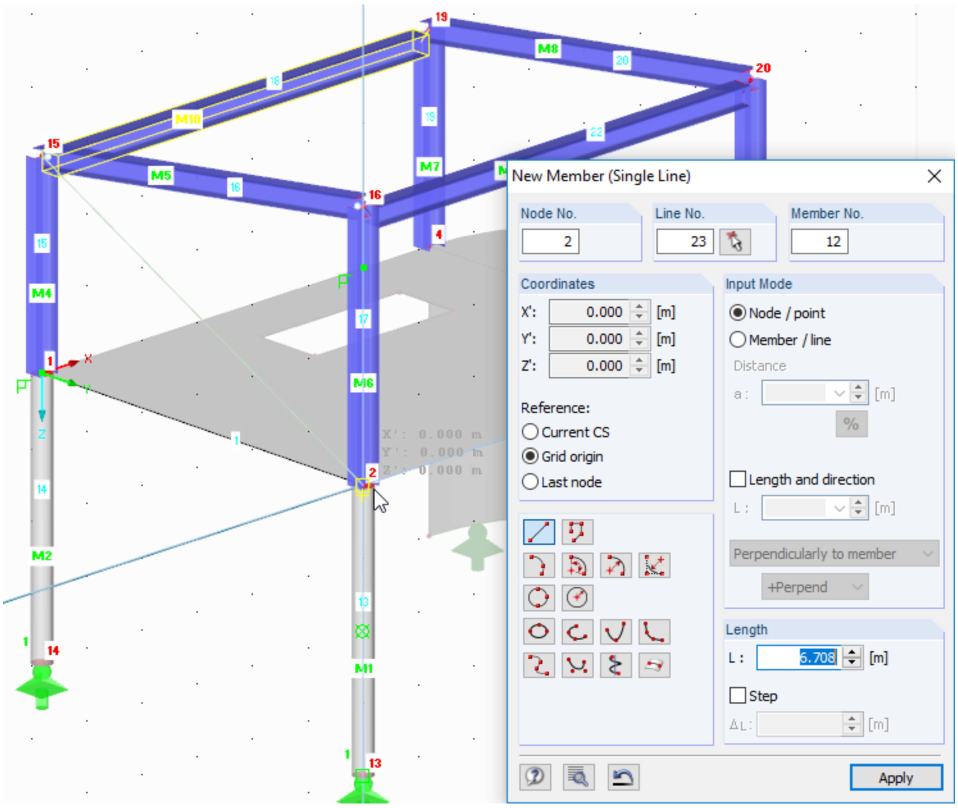


Figure 4.44 Defining the diagonal

4.5.3.2 Rotating a Member

A tension member adds to the stiffness only via its cross-sectional area. Thus, from a structural point of view, the member's rotation is irrelevant. For the rendered view, however, we want to rotate the angle section.

We double-click member 12 to open the *Edit Member* dialog box where we define a member rotation of **−90°**.

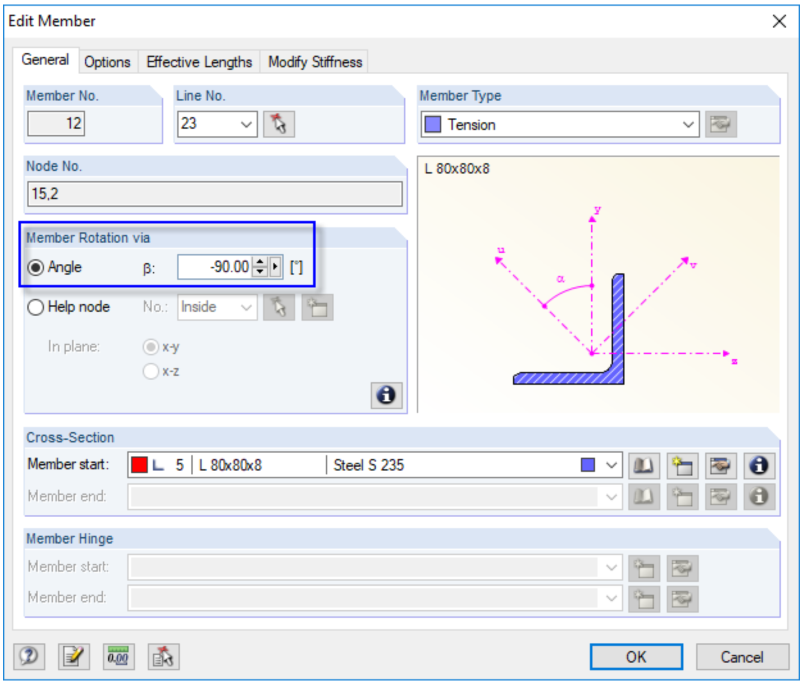


Figure 4.45 Defining member rotation

We can examine the result in magnified view using the zoom and move functions (see Mouse functions).

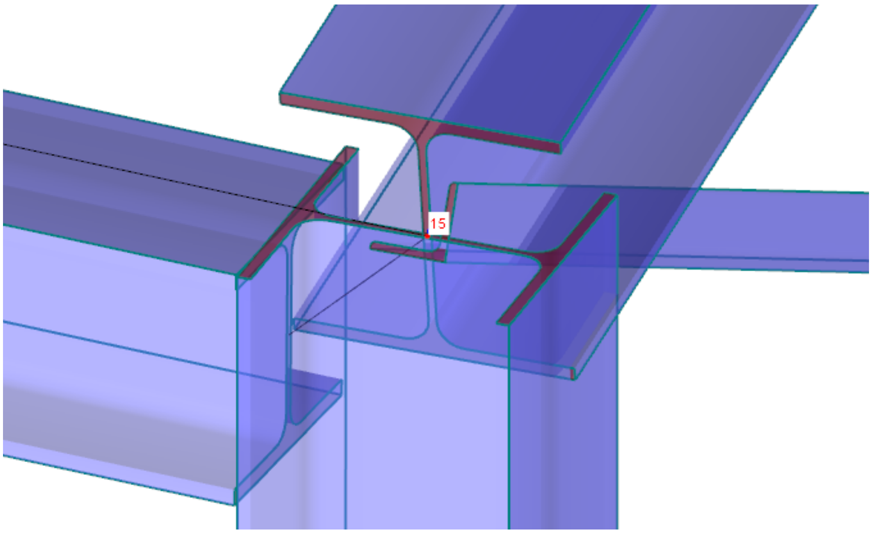


Figure 4.46 Rotated angle member in zoomed view

Undo/Redo

You may [Undo] the member rotation in this view to check the cross-section's initial position. With the default features *Undo* and *Redo*, known from Windows programs, data entered into RFEM may also be undone or redone.

Figure 4.47 *Undo* and *Redo* buttons

Canceling visibility mode

The transparent parts of the model may be reactivated in the *Views* navigator: To display all objects, click the [Cancel Visibility Mode] button.

The full spatial view can be set with the [Isometric View] button in the toolbar (see Figure 4.48).

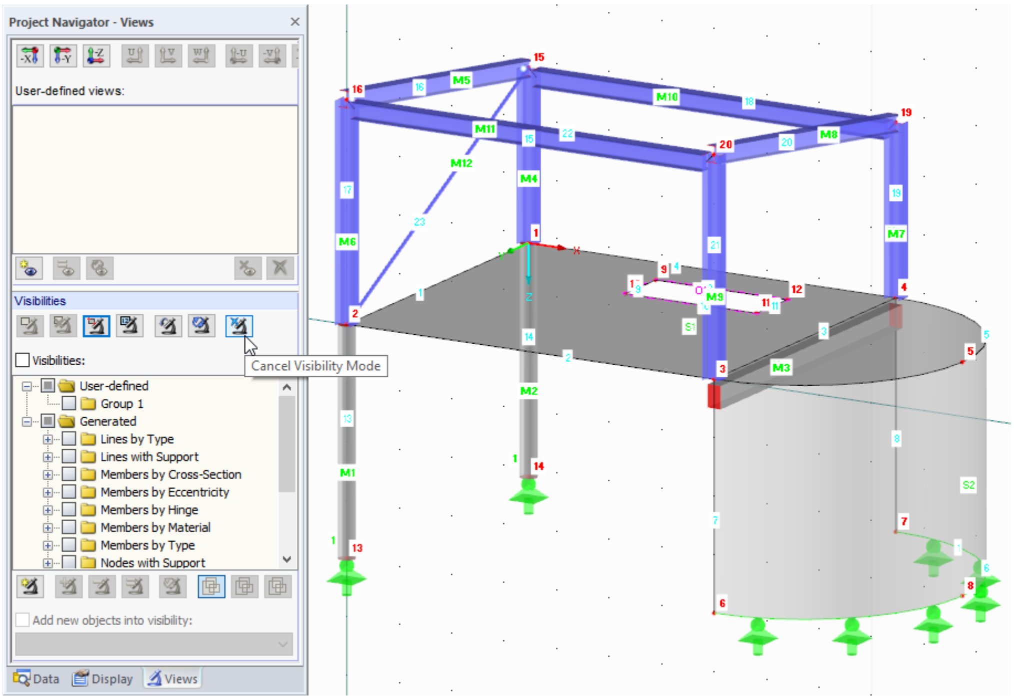


Figure 4.48 Full view of the model

Adjusting color assignments

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

We click through the menu items to change the display. The *Cross-Section* option, for example, allows for an immediate distinction between the different section types.

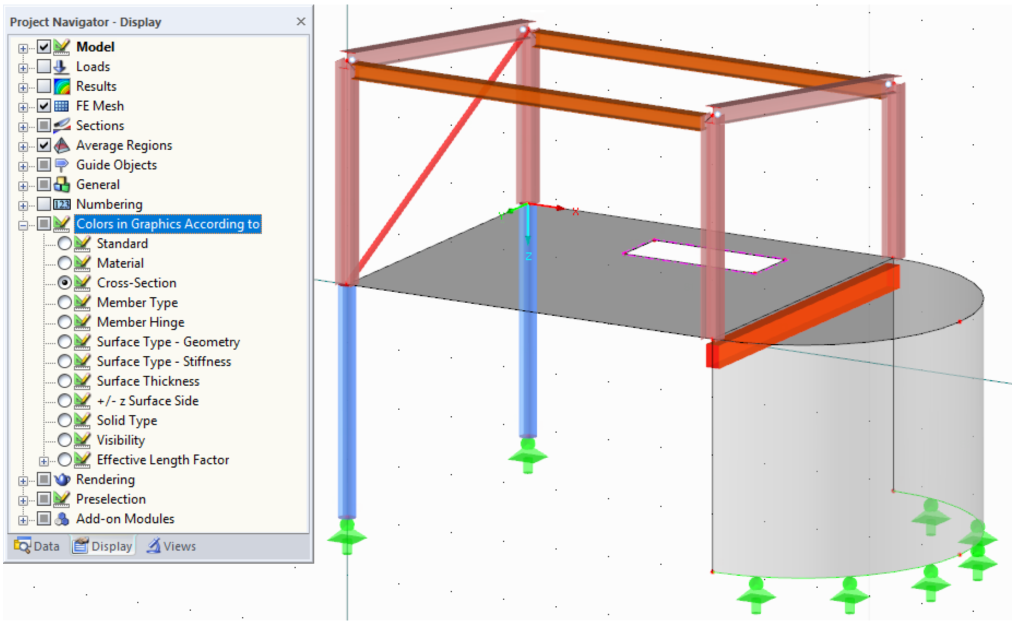


Figure 4.49 Distinguishing cross-sections by colors

From here on, the numbering of objects may be turned off. Do to so, right-click in an empty area of the work window. Clear *Show Numbering* in the shortcut menu (see Figure 4.4).

Afterwards we reset the colors to *Standard*.

4.6 Checking the Input

Checking the *Data* navigator and tables

As mentioned, RFEM provides various ways to enter model data. The graphical input is reflected in both the *Data* navigator tree and the tables. To display or hide navigator and tables, use **View → Navigator/Table** or the corresponding toolbar buttons.

The various types of objects are categorized in the tables' tabs. Graphics and tables are interactive: To find an object in the table, such as a member, go to Table *1.17* and click the member in the work window. The corresponding table row is now highlighted in color (see Figure 4.42).

This is a quick way to check the model's numerical data.

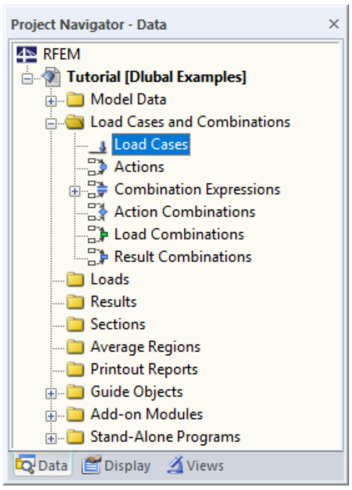
Saving data

The input of model data is now complete. To save our file, we use

File → Save

or the corresponding toolbar button.

5 Load



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

- Load Cases
- Actions
- Combination Expressions
- Action Combinations
- Load Combinations
- Result Combinations

Load cases are used to define the different types of loads such as self-weight, snow load, and wind load. Afterwards, they are assigned actions and superimposed with partial safety factors according to the combination expressions of the standard (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 construction (see Chapter 2.3).

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

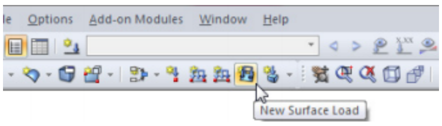


Figure 5.1 *New Surface Load* button

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

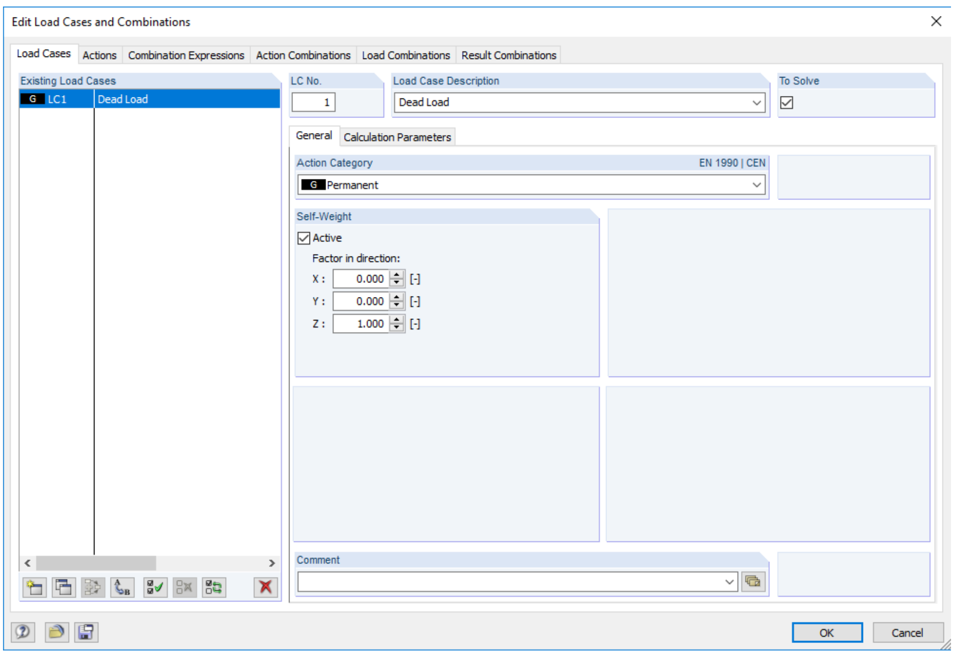
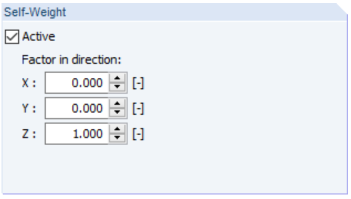


Figure 5.2 *Edit Load Cases and Combinations* dialog box, *Load Cases* and *General* tabs

Load case number *1* with the action category *Permanent* is preset. Additionally, we enter the *Load Case Description* **Dead Load**.

5.1.1 Self-Weight



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

5.1.2 Floor Structure

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

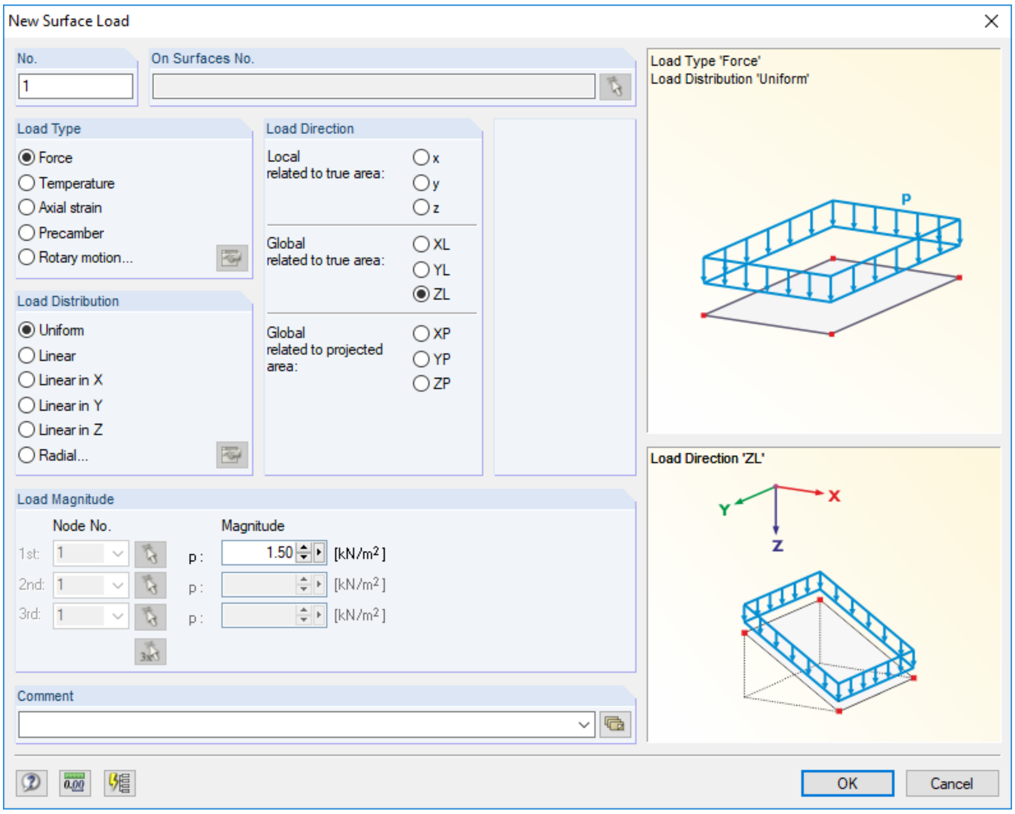
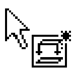


Figure 5.3 *New Surface Load dialog box*

The floor structure acts as the load type *Force*, the load distribution is *Uniform*. We leave these default settings along with the load direction *Global ZL*.

In the *Load Magnitude* box, we enter a value of **1.5** kN/m^2 (see Chapter 2.3) and confirm with [OK].



We can now graphically assign the load to the floor surface: Next to the pointer is a small load symbol, which disappears when it gets close to a surface. We apply the load to surface **1** with a click (see Figure 5.4).

The surface load is not applied in the opening. The area without any load is marked accordingly.

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

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

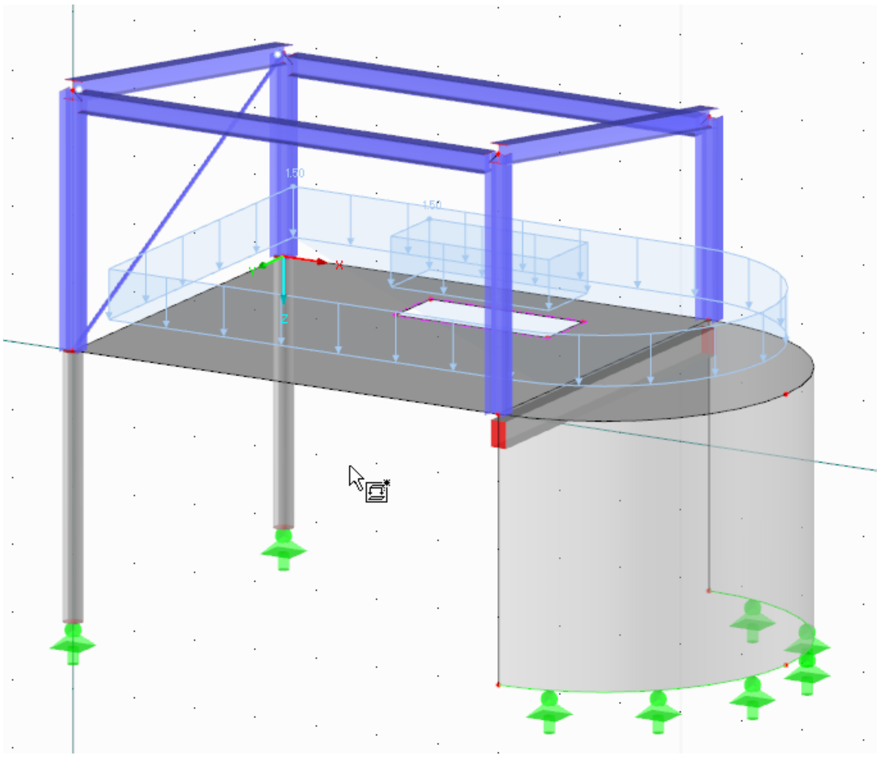


Figure 5.4 Graphical input of floor load

5.1.3 Earth Pressure

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

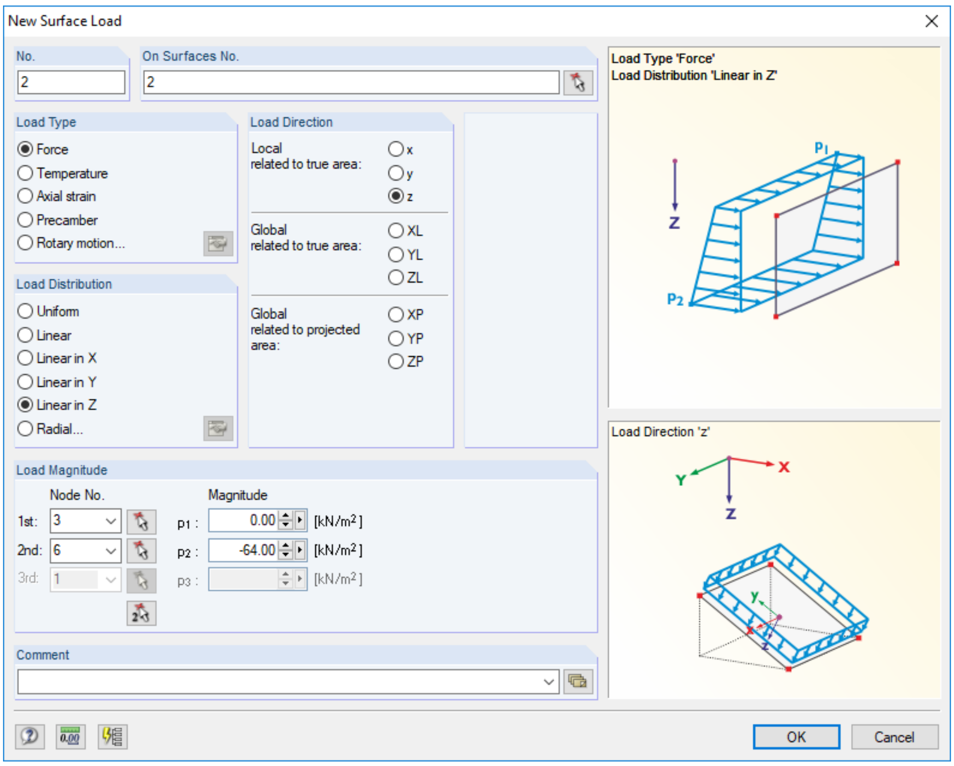
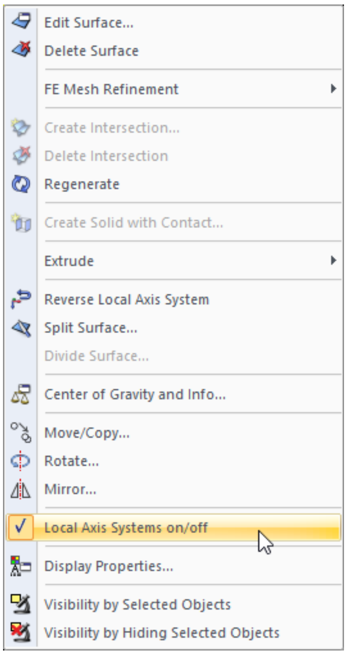


Figure 5.5 New Surface Load dialog box

The load is specified as the load type *Force* with the *Load Distribution* **Linear in Z** perpendicular to the surface. Thus, we select **Local z** as the *Load Direction*.

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



Display of local axis systems

After clicking [OK], the linear surface load is displayed on the model, increasing downwards and acting perpendicular to the shell. We use the shortcut menu shown on the left (opens when we right-click the surface) to show the local surface axes.

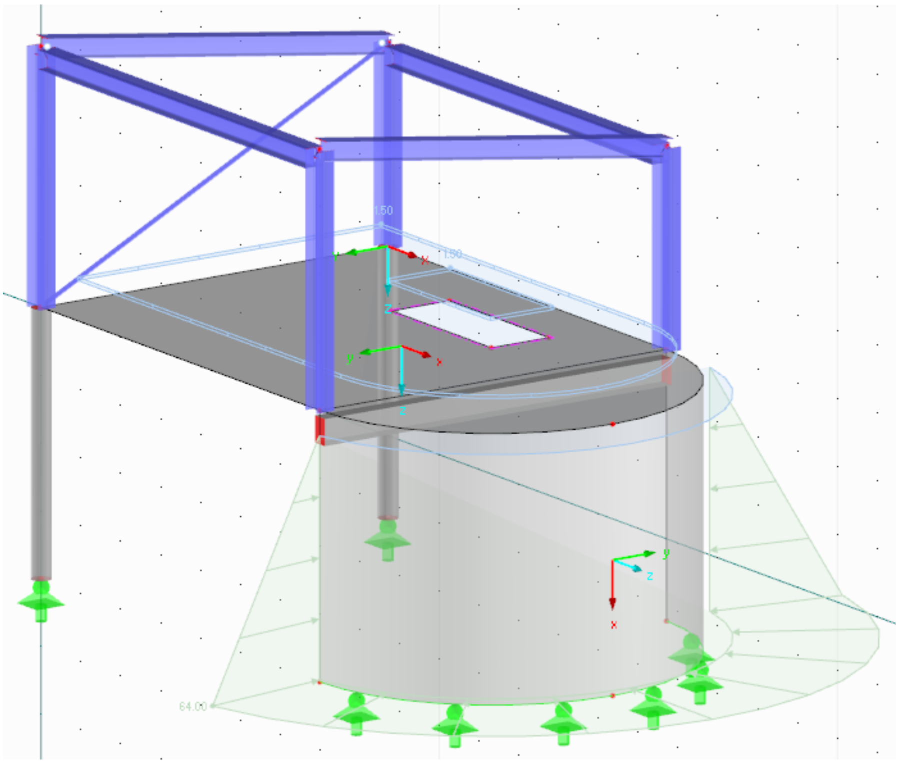


Figure 5.6 Linearly variable surface load (earth pressure) with display of local surface axes

5.1.4 Roof Load

The load due to roof finishes (roofing, supporting structure) also acts as a permanent load. For applying loads to the steel construction that act on surfaces, RFEM provides a tool that is able to convert area loads into member loads.

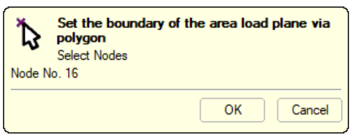
To open the generator dialog box, use

Tools → Generate Loads → From Area Loads on Members via Plane.

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

The roof structure's *Area Load Direction* is *Global related to true Area* **ZL** with an *Area Load Magnitude* of **1.2** kN/m^2 (see Chapter 2.3).

We define the area load's plane graphically using : In the work window, we click the four corner nodes **16**, **15**, **19**, and **20** of the roof area one by one. Afterwards, we click [OK] to close the selection window shown on the left.



The roof's supporting structure (not displayed in the model) introduces the roof loads into the structural system along the purlins. This means: Both of the monopitch roof's horizontal beams do not participate in transferring loads from the roof loads and are thus excluded from the load generation. We use the [Select Parallel Member] button in the *Remove Influence from section* to graphically select one of the horizontal beams in the working window (member **8** or member **5**). After clicking [OK] in the selection window, the generator dialog box should look as follows.

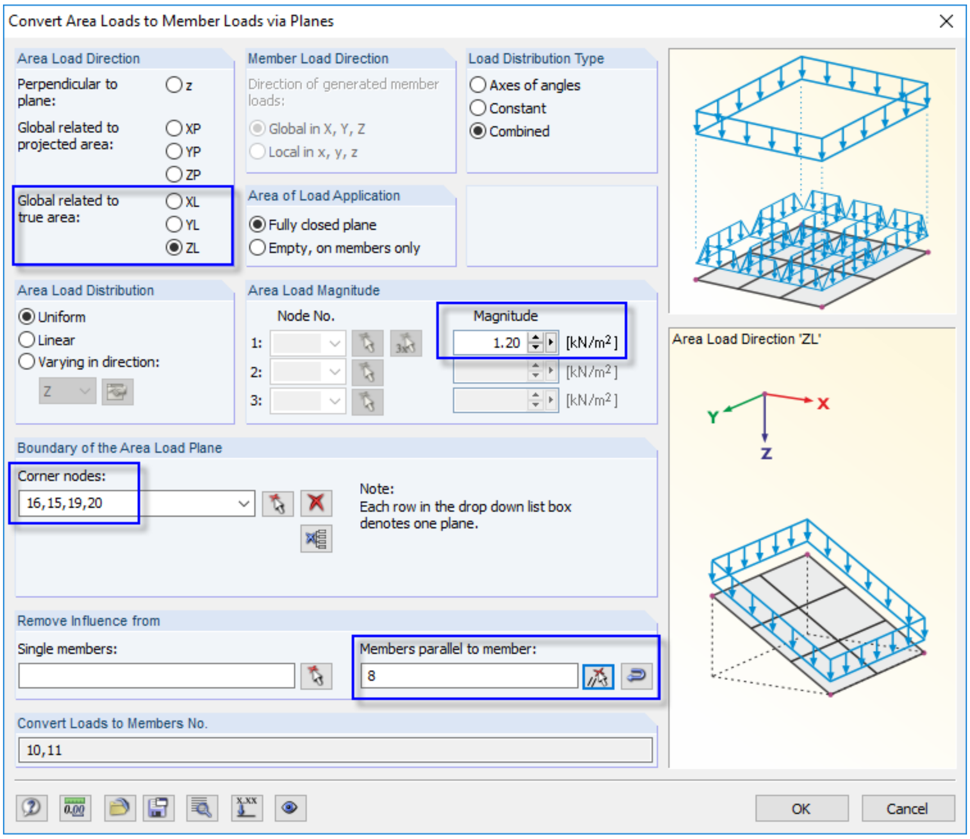
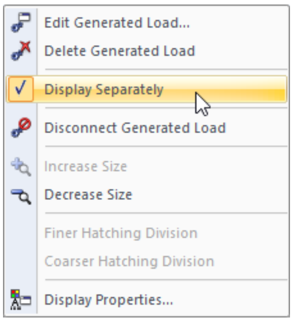


Figure 5.7 *Convert Area Loads to Member Loads via Planes* dialog box

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



The load is represented as a roof area load. To display the generated loads that act on both purlins, right-click the load and open the shortcut menu. Select the *Display Separately* option.

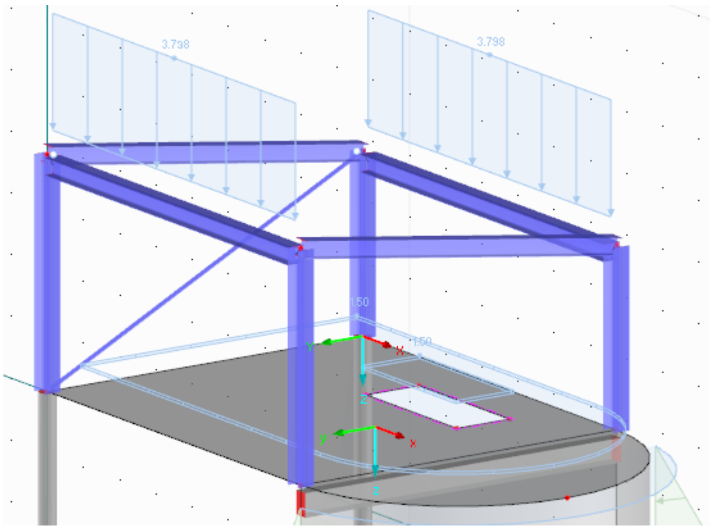


Figure 5.8 Member loads from area loads on purlins

This completes the input for the *Dead Load* load case.

5.2 Load Case 2: Imposed Load

Before entering the imposed loads, we create a new load case. Do do so, we can use the menu item

Insert → Loads → New Load Case

or the corresponding toolbar button (to the left of the load case list).

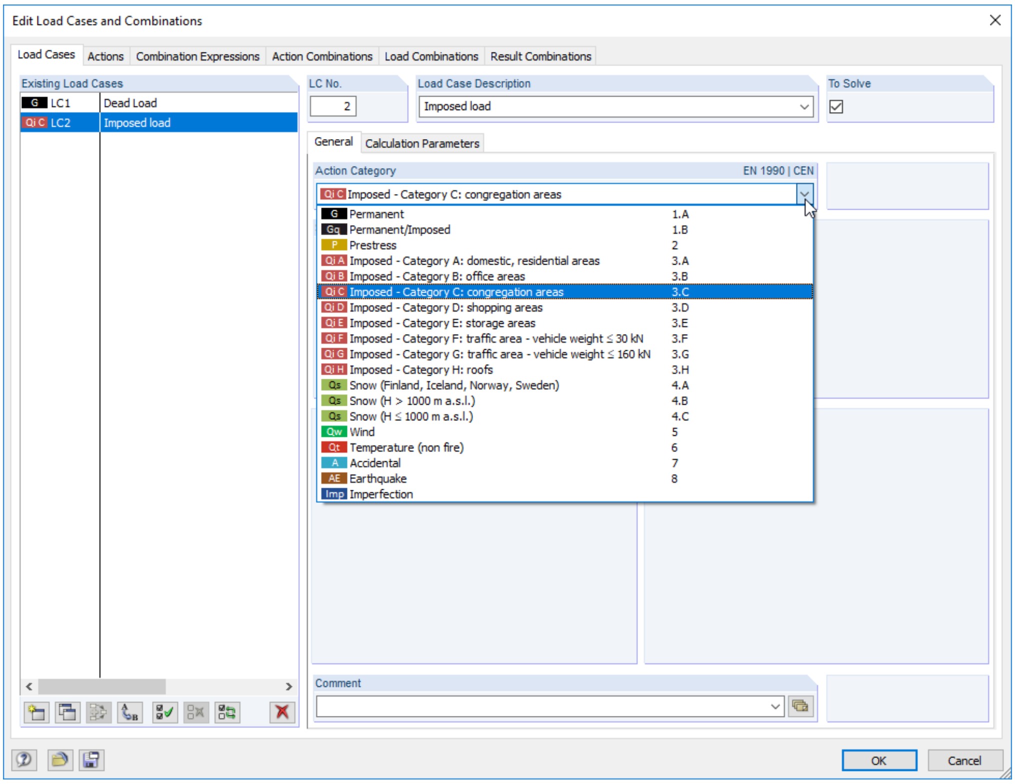


Figure 5.9 *Edit Load Cases and Combinations* dialog box

We enter **Imposed load** as the *Load Case Description* or select it from the list.

To change the *Action Category* to **Qi Imposed - Category C: congregation areas**, use the action category list (see Chapter 2.3). 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 way to enter the surface load: We select floor surface *1* by clicking it. When we now open the familiar dialog box using the [New Surface Load] button, the surface number is already filled in.

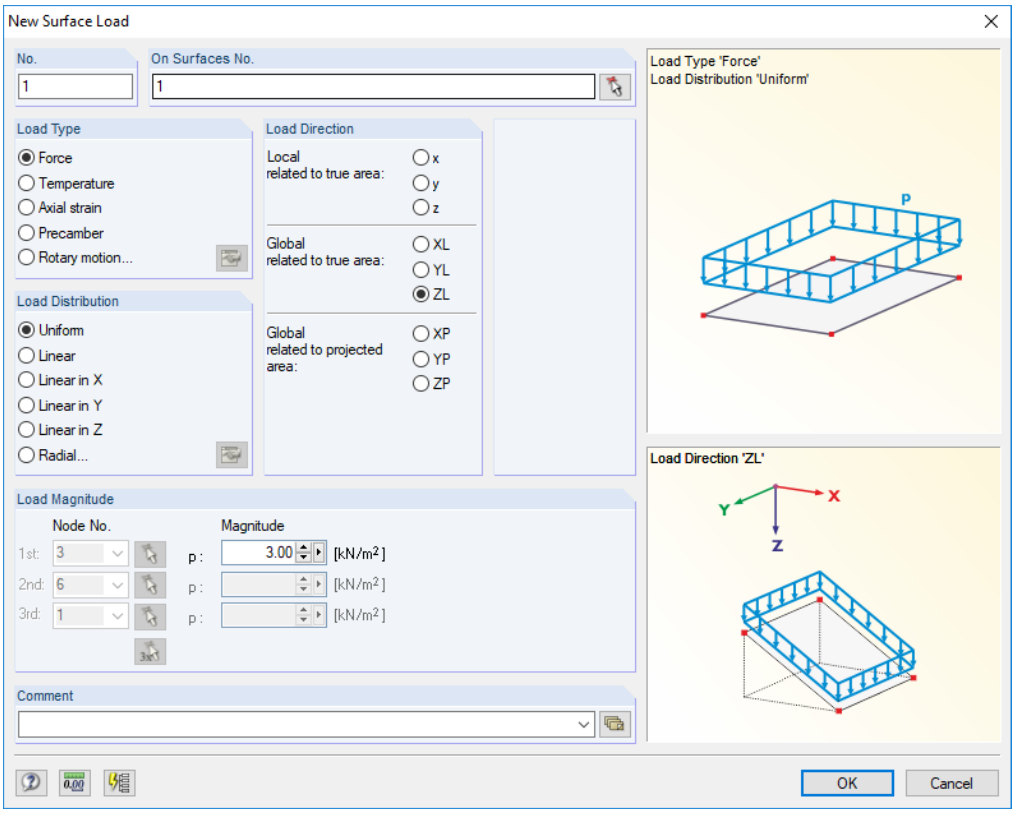


Figure 5.10 *New Surface Load* dialog box

The imposed load acts as the load type *Force*. We specify the load distribution to be *Uniform* and select *Global ZL* as the load direction.

As the *Load Magnitude*, we enter a value of **3** kN/m^2 (see Chapter 2.3). Then we confirm the dialog box with [OK].

5.2.2 Edge of the Opening

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

We open the *New Line Load* dialog box with the [New Line Load] toolbar button (right next to the [New Surface Load] button).

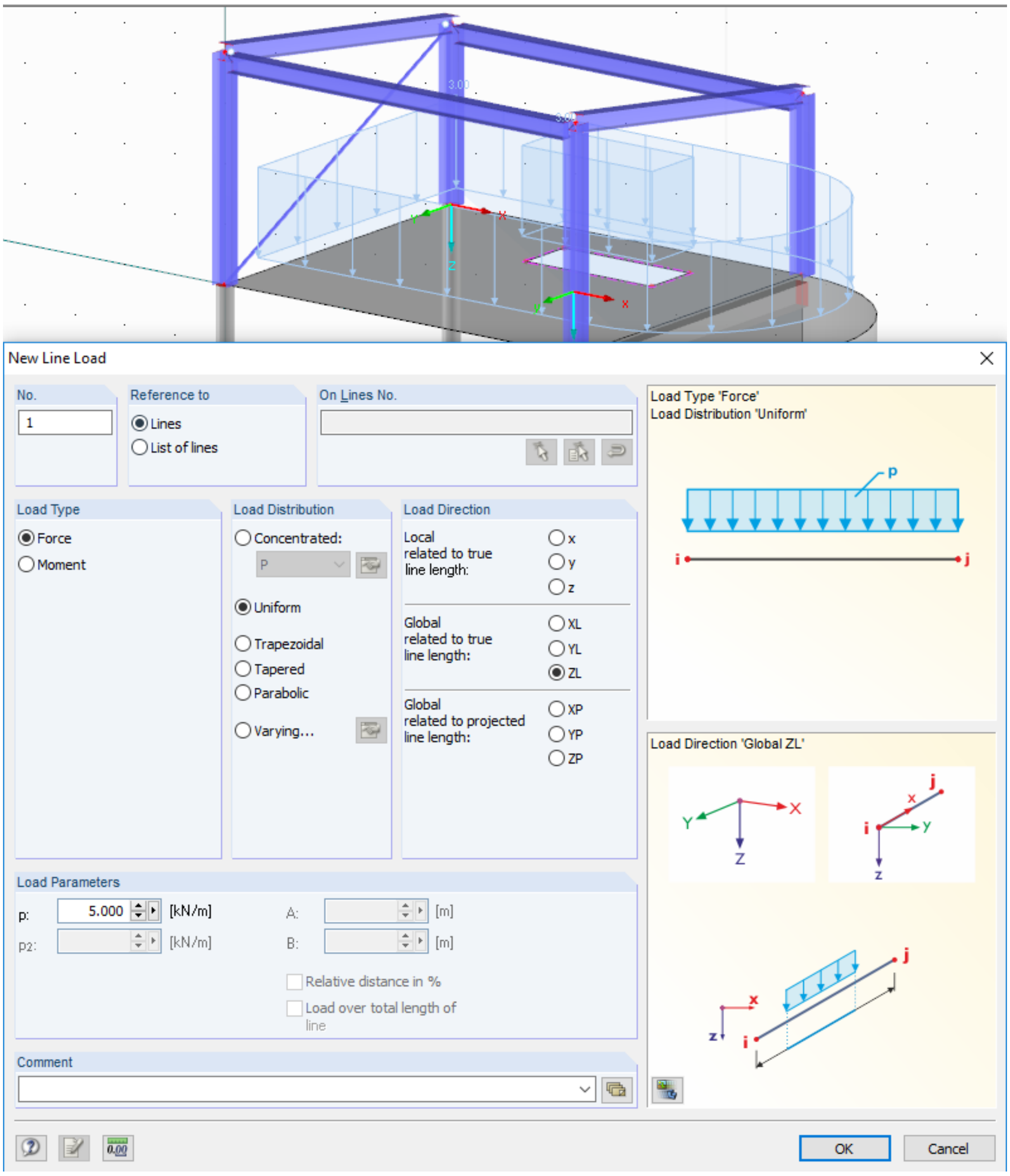


Figure 5.11 *New Line Load* dialog box

The line load with the load type *Force* and a *Uniform* load distribution acts in the load direction *ZL*.

We enter **5** kN/m into the *Load Parameters* box. We confirm with [OK] and click line **11** at the opening's edge (check via display in the status bar).

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

5.3 Load Case 3: Snow

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

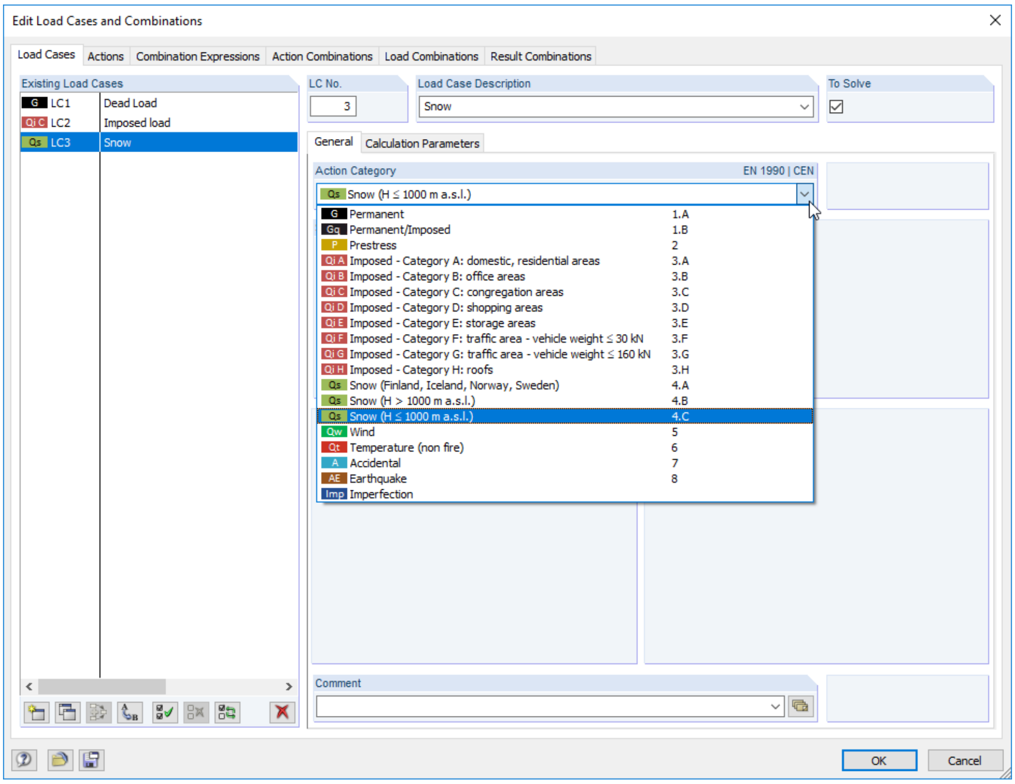


Figure 5.12 *Edit Load Cases and Combinations dialog box, Load Cases tab*

We enter **Snow** in the *Load Case Description* or select it from the list.

We specify the *Action Category* as **Qs Snow (H ≤ 1000 m a.s.l.)**.

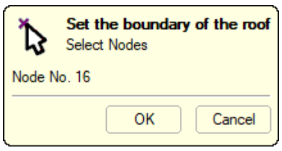
5.3.1 Roof

To enter the monopitch roof's snow load, we use a load generator again. To access this function, use

Tools → Generate Loads → From Snow Loads → Flat/Monopitch Roof.

In the *Generate Snow Loads - Flat/Monopitch Roof* dialog box, we enter the following data (see Figure 5.13):

Snow Load Parameters according to *EN 1991-1-1-3* with national annex *DIN* apply. We select *Zone number* **2** and change the *Altitude* to **500** m (see Chapter 2.3).



We define the *Roof Geometry* graphically using by clicking the four corner nodes **16**, **15**, **19**, and **20** of the roof area one by one (see selection dialog box on the left).

We check if the newly created **LC3** is selected in the *Load Case to Generate* section.

We want to create *Member loads* again but the two monopitch roof beams do not contribute 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 Parallel Member] button in the *Remove Influence from* section to graphically select one of the horizontal beams (member **8** or member **5**).

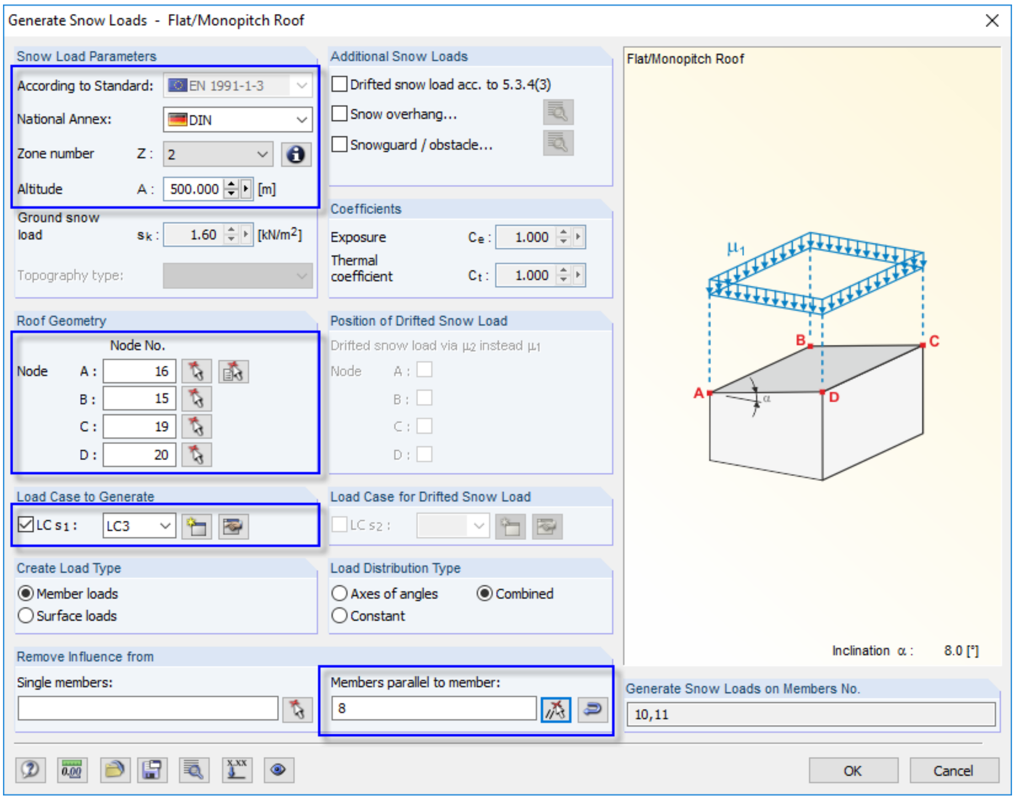
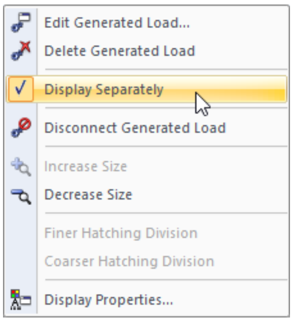


Figure 5.13 *Generate Snow Loads - Flat/Monopitch Roof* dialog box

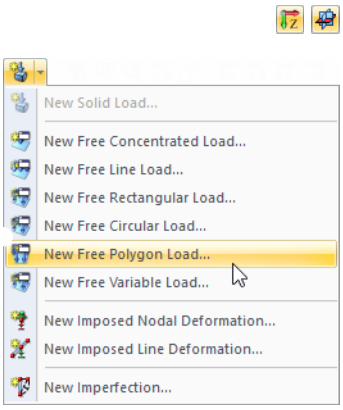


We confirm with [OK]. An *Info* dialog box appears with information about the conversion of area load values to member loads. We confirm this dialog box as well. The load is represented as a roof area load with a value of 1.28 kN/m^2.

To display the generated loads that act on both purlins, we can use the *Display Separately* option again, which is available in the load shortcut menu. This renders both member loads visible, with 4.02 kN/m each.

5.3.2 Floor

The snow load also acts on the semicircular area of the floor surface, which is outside. Since only part of surface 1 bears a load, the *New Surface Load* feature cannot be used. In the full version, the ceiling would be divided into two surfaces to easily place a surface load on the semicircular surface. Since the demo version allows for just two surfaces to be used in a model, we choose a somewhat more complex way.



List of loads

First we set the View in [Z]-direction. For the new work plane we choose plane [XY].

We define the snow load as a *Free Polygon Load*. The corresponding feature can be found in the [New Load] toolbar button's list of loads (to the right of the [New Surface Load] button).

In the *New Free Polygon Load* dialog box (see Figure 5.14), we specify the load to only act *On Surfaces No.* **1** and be *Globally related to projected area* **ZP**. Snow loads must be related to the base area, not the true area (like self-weights). For horizontal surfaces, this makes no difference, of course.

The load is projected in the *XY Plane*.

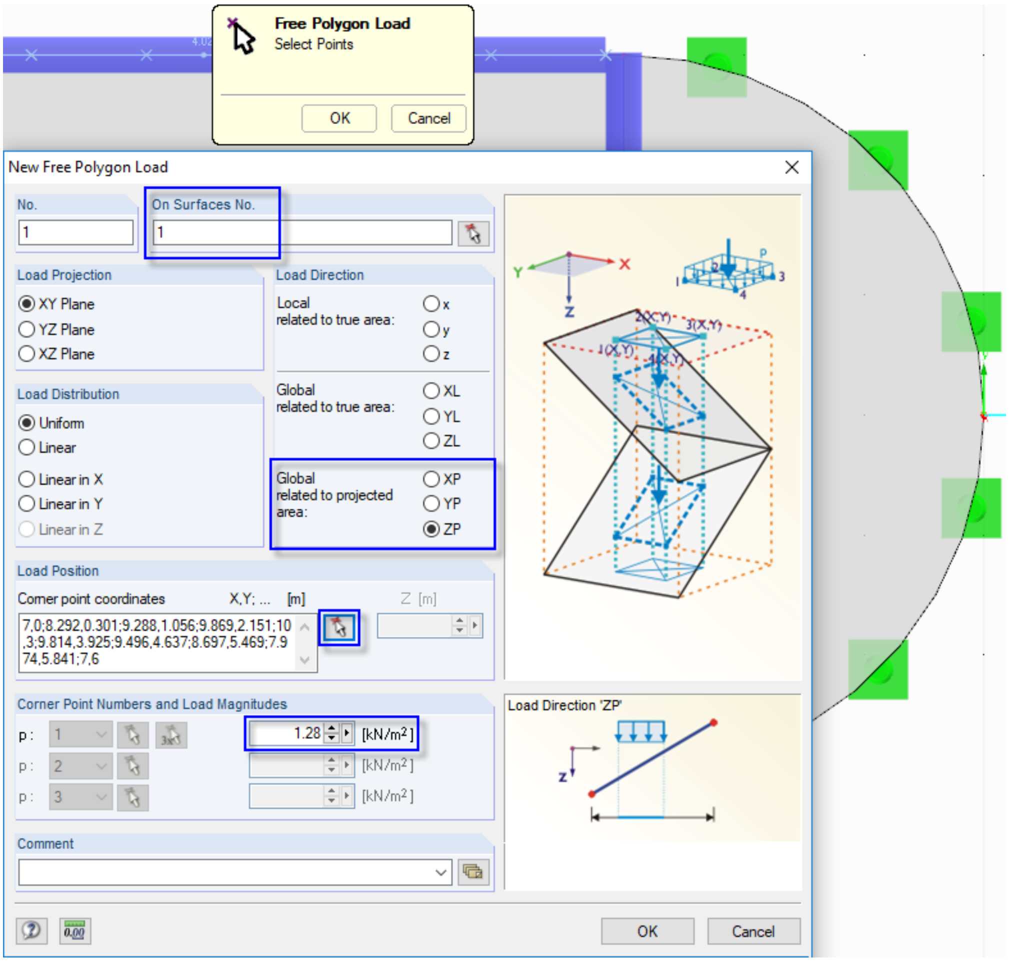


Figure 5.14 Defining a new free polygon load by setting arc points

We define the *Load Position* in the work window using [button] : We start at arc node **4** at the top and then use the reticle pointer to click any points on the arc line to approximate the semicircular surface with a polygonal chain. Once we have reached the arc end at node **3**, we close the small yellow dialog box with [OK].

In the *Corner Point Numbers and Load Magnitudes* section, we enter a value of **1.28** kN/m^2 — this value was designated as a roof snow load by the generator (see text after Figure 5.13).

After confirming with [OK], the load is placed on the semicircular surface.

We close the input mode with [Esc] or with a right click and switch back to the [Isometric View].

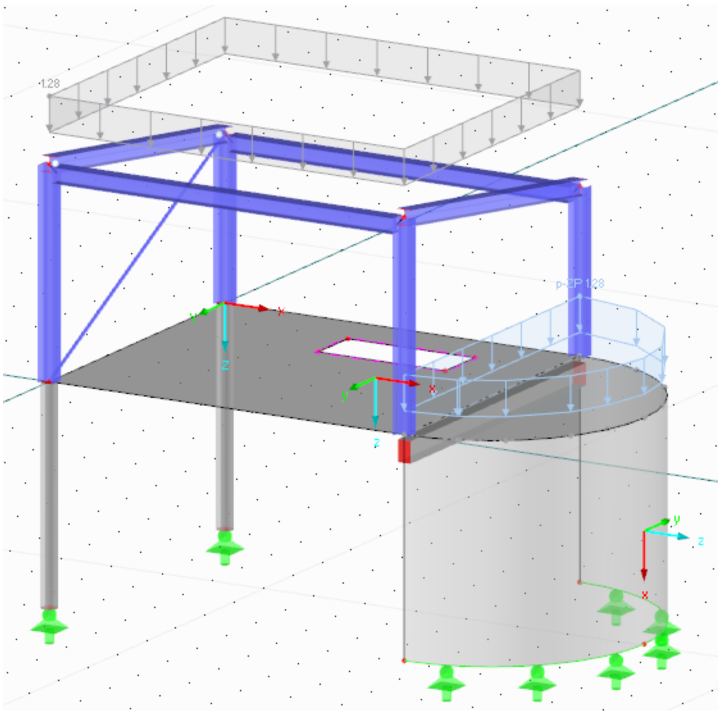


Figure 5.15 Snow loads

5.4 Load Case 4: Wind

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

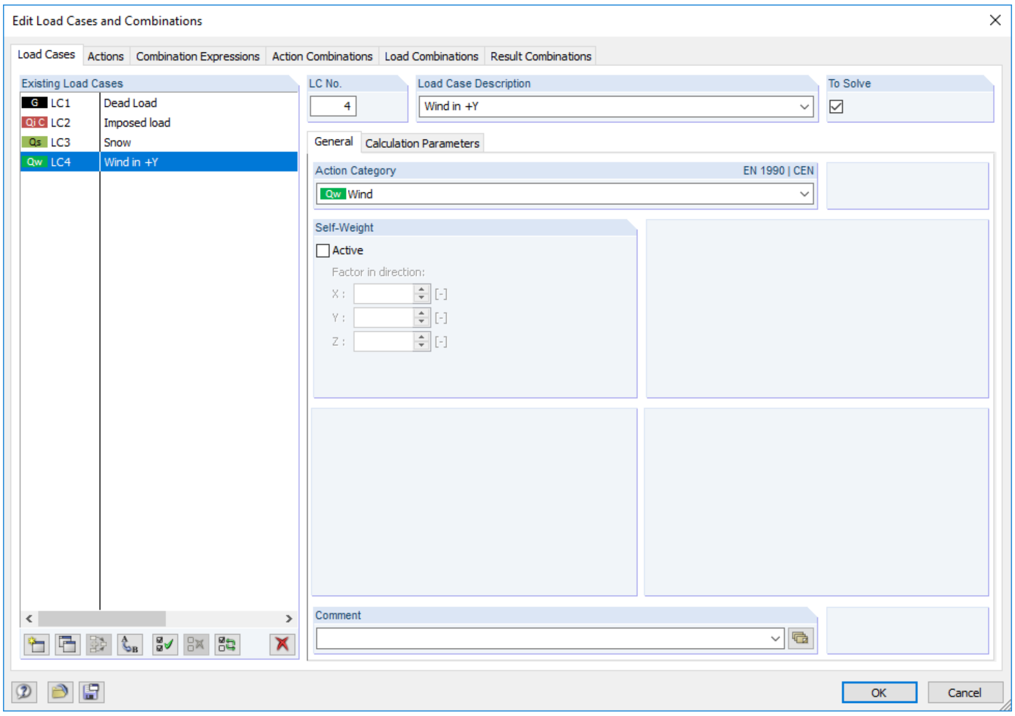


Figure 5.16 *Edit Load Cases and Combinations* dialog box, *Load Cases* tab

We select **Wind in +Y** from the list as the *Load Case Description*. The *Action Category* automatically changes to **Q_w Wind**.

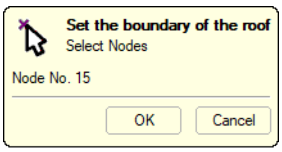
5.4.1 Steel Construction Loads

We once more use a load generator to enter the wind load that acts on the walls that are closed on all sides and on the monopitch roof. To access this function, we use

Tools → Generate Loads → From Wind Loads → Vertical Walls with Roof.

In the *Generate Wind Loads - Vertical Walls with Roof* dialog box, we enter the following data:

Velocity Pressure is determined according to *EN 1991-1-4* with the national annex *DIN*. We specify the *Wind zone* **1** and *Terrain category* **III** (see Chapter 2.3). The buttons make the selection easier. We change the *Structure height* to **8** m and the *Altitude* to **500** m.



We define the *Base Geometry* using by clicking floor nodes **1**, **4**, **3**, and **2** (observe order according to dialog box sketch). We also use to define the roof geometry by clicking roof nodes **15**, **19**, **20**, and **16**.

We check whether the inflow direction **A - B** is set in the *Set Wind on Side* section.

In the *Load Cases to Generate* section, we deactivate the load case *w-*. As described in Chapter 2.3, only positive external pressure factors are examined. Load generation of *LC w+* is to occur for **LC4**.

As before, *Member loads* are to be created while the monopitch roof's horizontal beams do not contribute to the load transfer. Using [Select Parallel Member] in the *Remove Influence from* section, we once again select one of the horizontal beams graphically (member **8** or member **5**). Diagonal member *12* is automatically excluded from the load transfer.

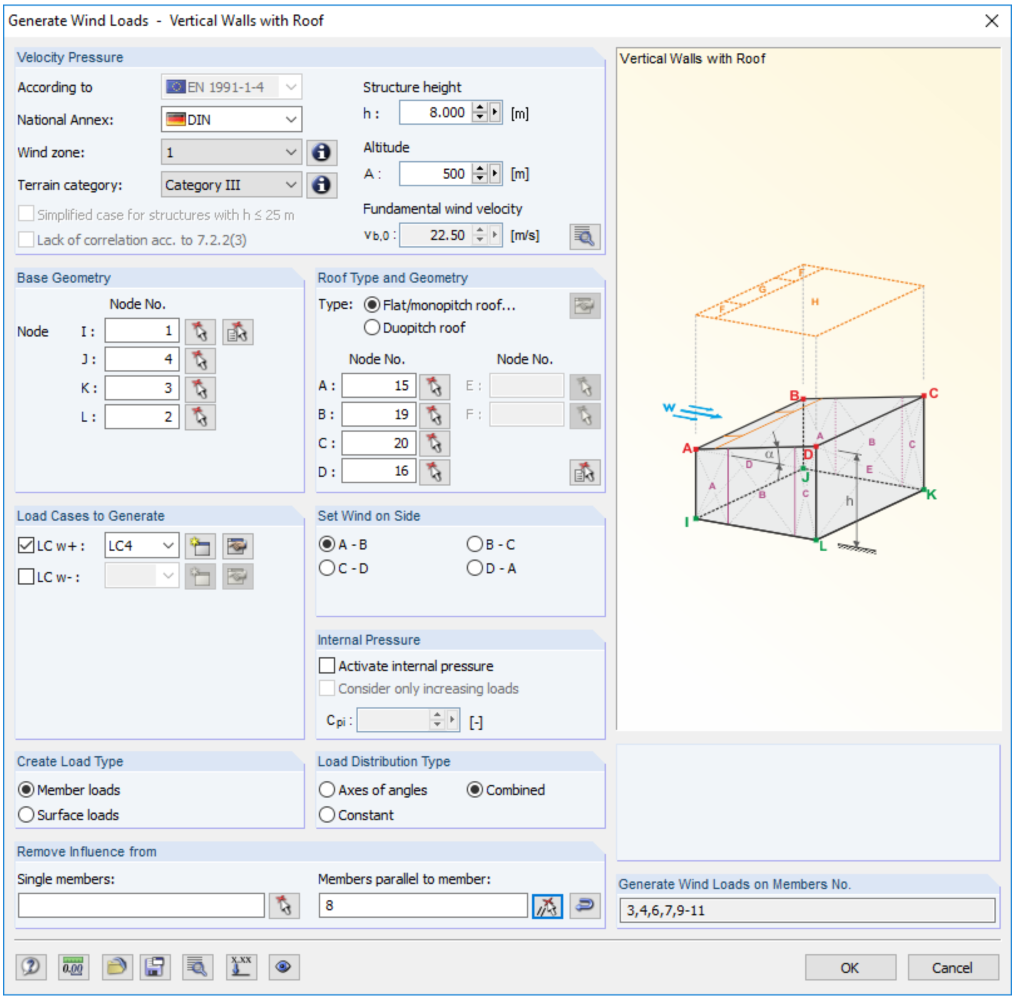


Figure 5.17 *Generate Wind Loads - Vertical Walls with Roof* dialog box

After clicking [OK], a dialog box appears with information about the generation data, which we confirm as well.

Now the wind loads are displayed as surface loads on the model. We use *Display Separately* in the load shortcut menu to display the corresponding member loads.

5.4.2 Column Loads

We define the loads on the lower part of the structure manually.

Defining a uniform member load

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

We select column member *1* per mouse click and use the [New Member Load] button to open the dialog box for entering the wind load.

The *Load Direction* is globally related to the true member length in **YL**. To the column, we assign a wind load component of **1.5** kN/m. We enter this value as a *Load Parameter*.

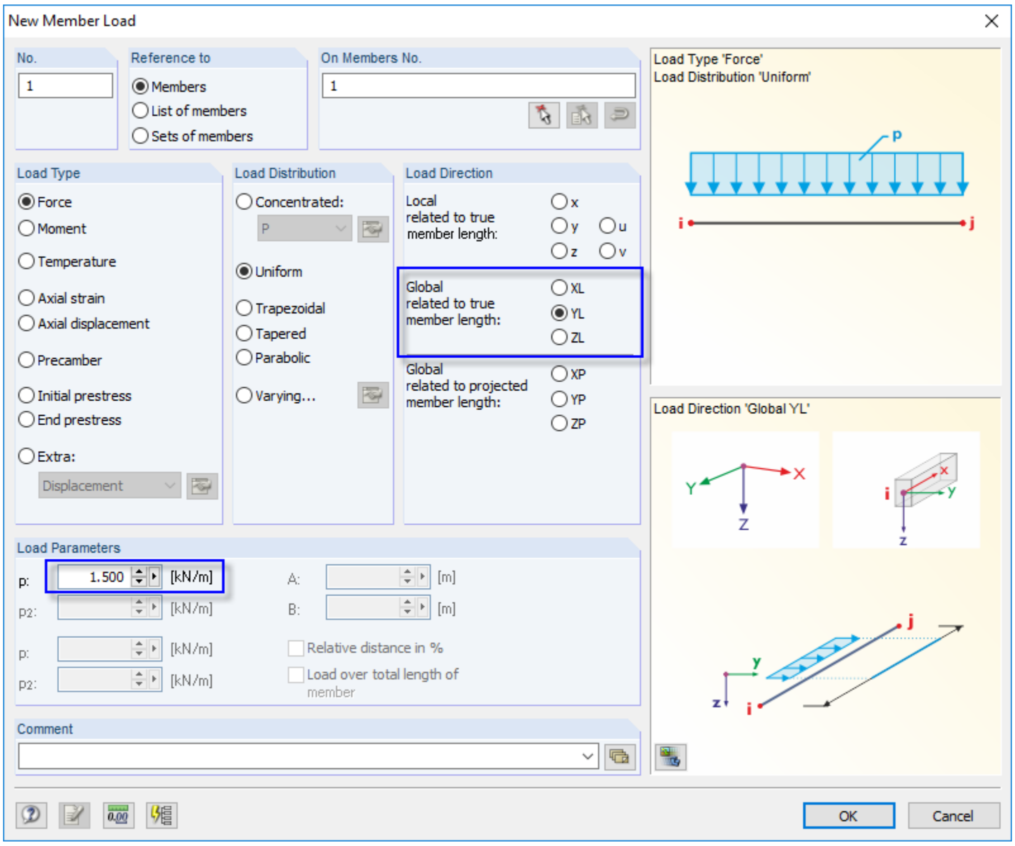


Figure 5.18 Defining wind suction as uniform member load

We click [OK]. Now the load is represented on the column.

Defining a trapezoidal member load

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

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

As before, we define the *Load Direction* in **YL** globally. The *Load Distribution*, however, is specified as **Trapezoidal**. This enables two *Load Parameters*: We enter a value of **0.5** kN/m for member start *p1* and **3** kN/m for member end *p2*. We defined the columns from bottom to top; thus the member start is at the column base.

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

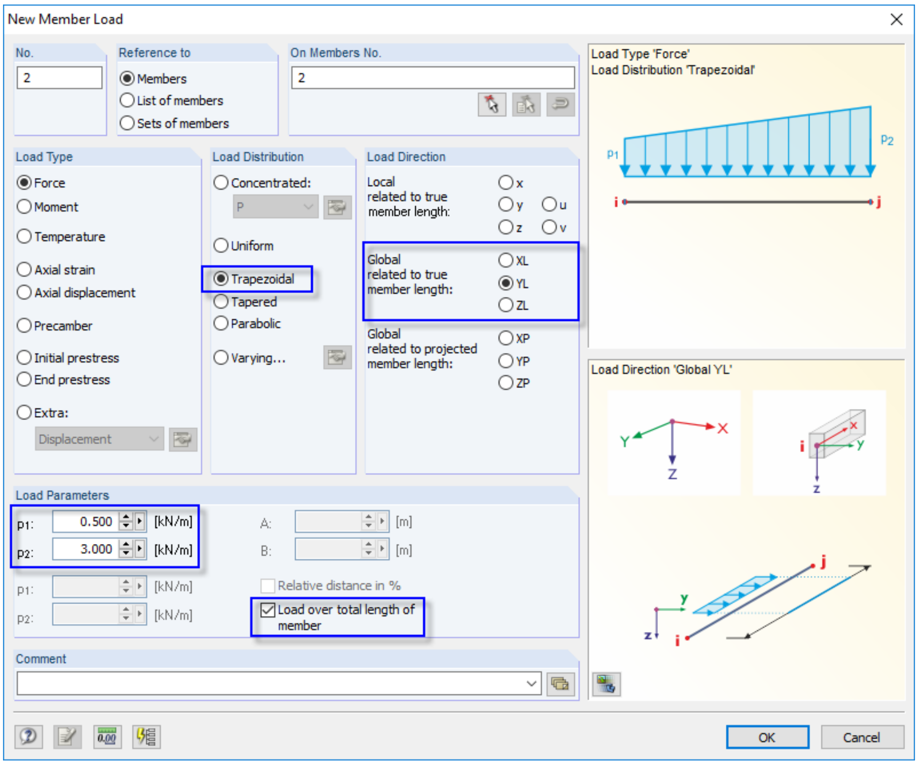


Figure 5.19 Defining wind pressure as trapezoidal member load

We click [OK]. Now the member load is represented on the second column (see Figure 5.20).

The graphic that shows the generated and manually defined wind loads should now look like the following figure.

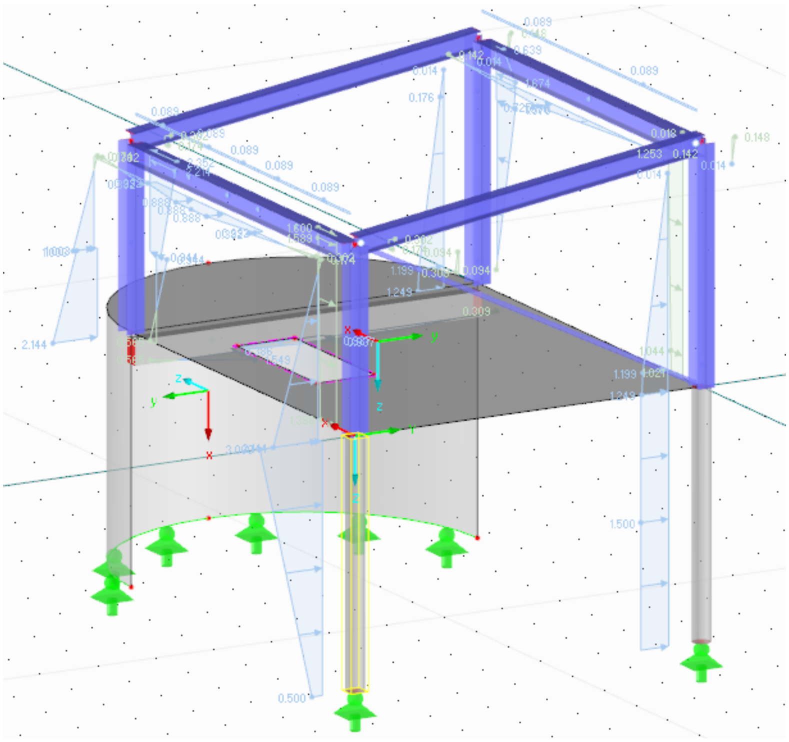


Figure 5.20 Wind loads

5.5 Load Case 5: Imperfection

In the final load case, we define imperfections for the columns that are stressed by normal forces.

This time we use the *Data* navigator to create a new load case: Right-click *Load Cases* and select *New Load Case* in the shortcut menu.

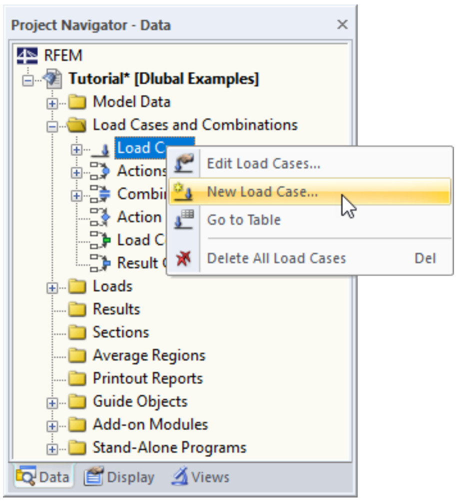


Figure 5.21 *Load Cases shortcut menu*

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

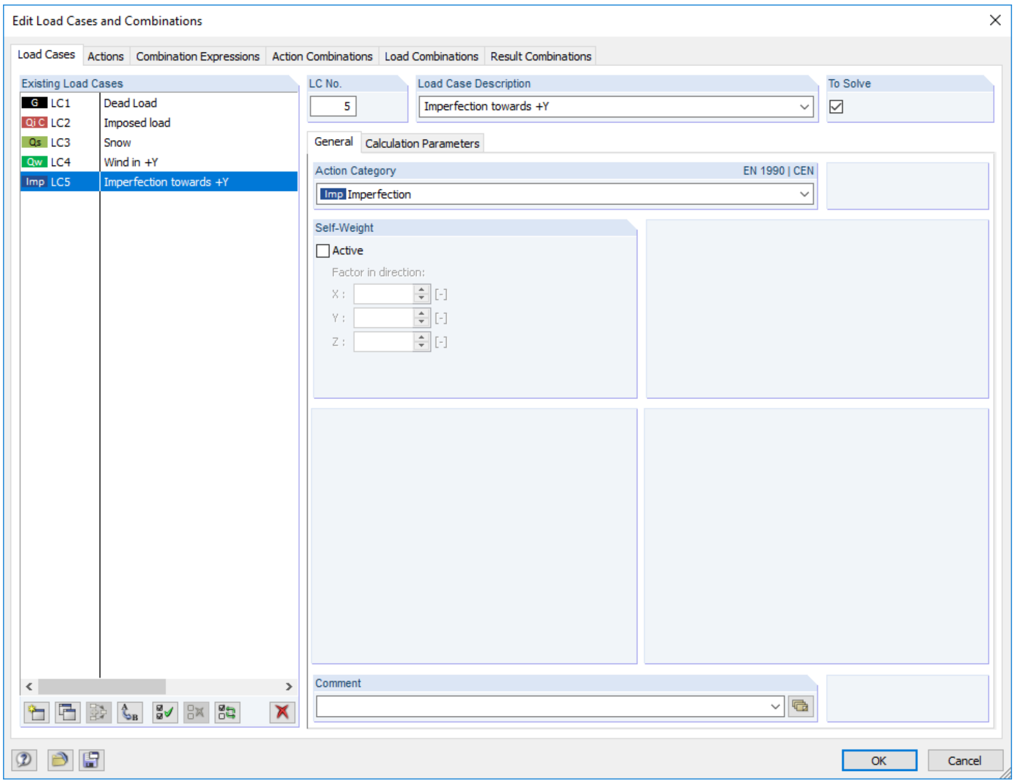
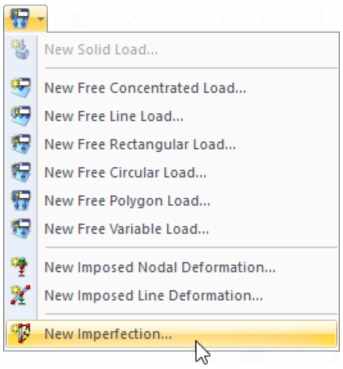


Figure 5.22 *Edit Load Cases and Combinations dialog box, Load Cases tab*

Then we confirm the dialog box with [OK].

5.5.1 Steel Columns



List of loads

We use the arrow next to the [New Free Polygon Load] toolbar button to select *New Imperfection* from the list to open the following dialog box.

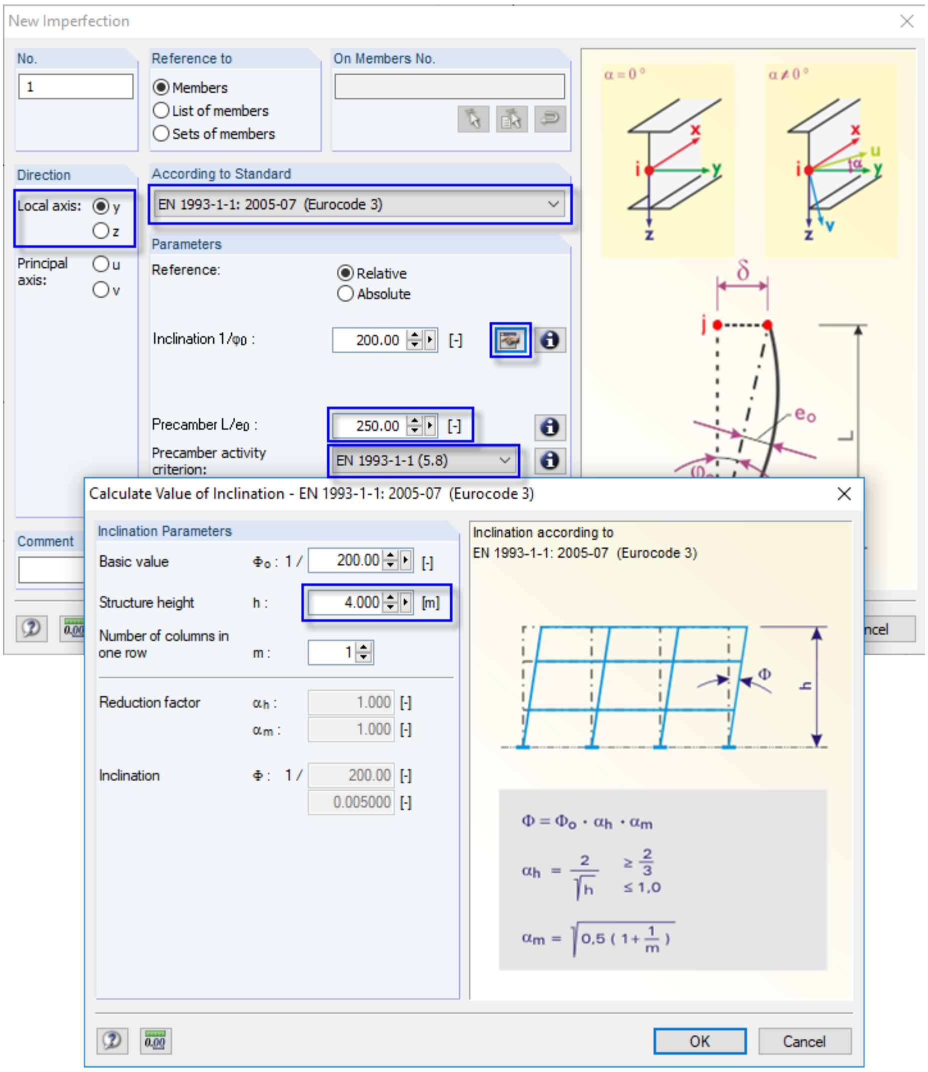


Figure 5.23 *New Imperfection* dialog box

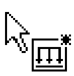
We define the imperfection in the *Direction* of local axis **y**. This is the direction of the 'weak' member axis, which is parallel to the global y-axis in our example.

EN 1993-1-1: 2005-07 is the relevant *Standard* for the steel columns' imperfection. If this is not preset, we select the correct list entry.

To enter the *Inclination* $1/\varphi_0$, we use [button] which opens the *Calculate Value of Inclination* dialog box. We change the *Structure height* to **4** m. To return to the previous dialog box, we click [OK].

For the buckling curve *b* of HE A 300 profiles, we specify a *Precamber* L/e_0 of **250** according to EN 1993-1-1, Table 5.1 (see Chapter 2.3).

We change the *Precamber activity criterion* to **EN 1993-1-1 (5.8)**.



We confirm the dialog box with [OK]. After that, we click the four steel columns with the member numbers **6**, **4**, **9**, and **7** to assign the imperfections.

To quit the function, right-click or use [Esc].

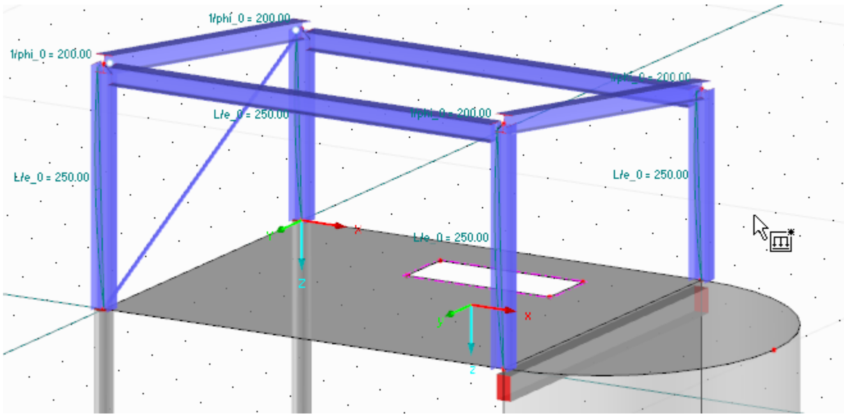


Figure 5.24 Setting imperfections for steel columns

5.5.2 Concrete Columns

We reopen the *New Imperfection* dialog box to define the concrete columns' inclination.

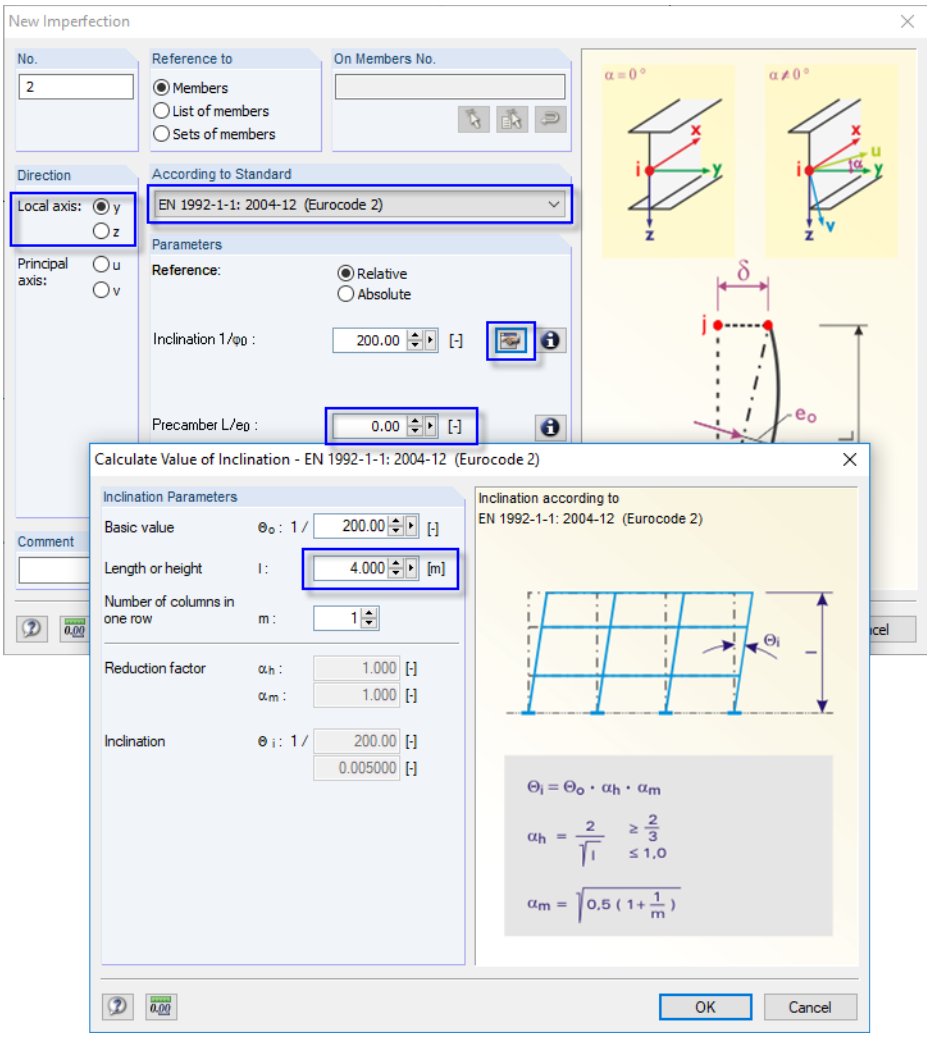


Figure 5.25 *New Imperfection* dialog box

We select **EN 1992-1-1: 2004-12** as the *Standard*.

As before, we use to define the *Inclination 1/φ0*: In the *Calculate Value of Inclination* dialog box, we change the *Length or height* to **4** m.

As precambers do not have to be taken into consideration according to Eurocode 2, we specify the *Precamber L/e0* to be **0**.

After confirming the dialog box, we click the two concrete columns with member numbers **1** and **2** to assign the imperfections.

5.6 Checking Load Cases

The input of all five load cases is now complete. We recommend to [Save] the entered data again.

We can check each load case quickly in the graphics: The and buttons in the toolbar allow us to browse the load cases.

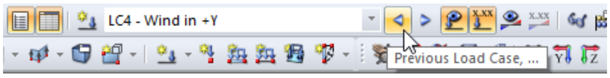


Figure 5.26 Browsing the load cases

The loads' 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 selected on the table toolbar with the button shown on the left. Graphic and tables are interactive once again: For example, to find an imperfection in the table, go to Table 3.14 *Imperfections* and select the load in the work window. The pointer jumps to the corresponding table row.

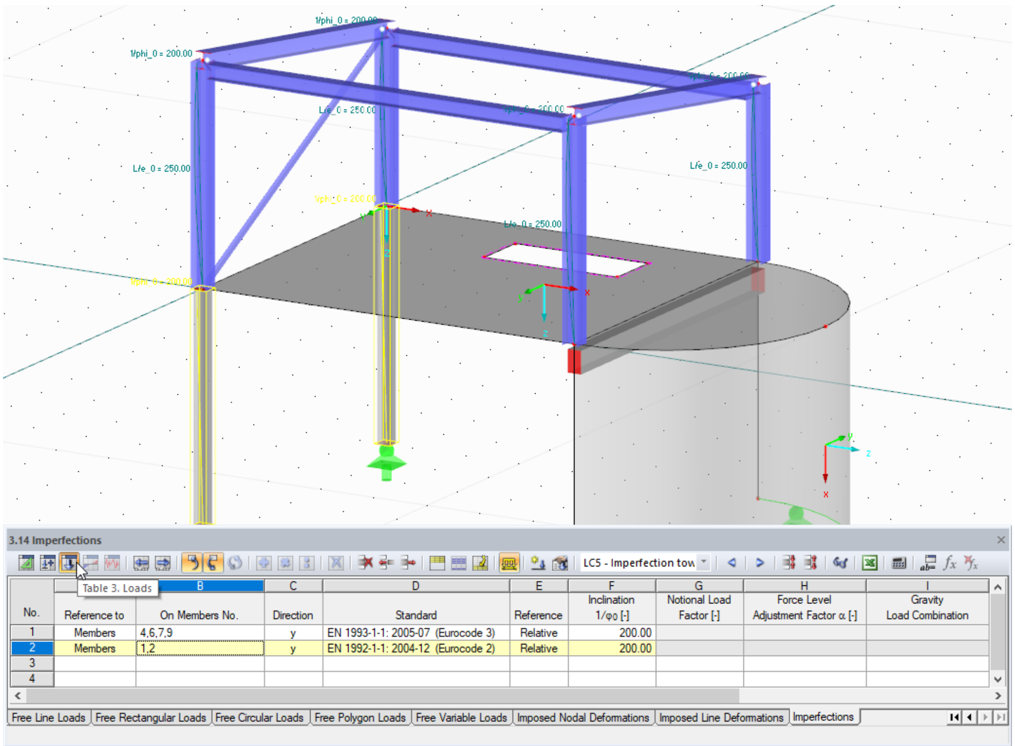


Figure 5.27 Interaction between graphic and load tables

6 Combining Actions

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

The *Action Category* defined for the load cases (see Figure 5.22) determines the way load cases are combined in different design situations.

6.1 Checking Actions

The load cases must be assigned to *Actions*, which are subsequently superimposed in accordance with regulations. Actions represent independent factors of influence that arise from different origins. The correlation existing between them may be neglected due to the structural system's reliability.

Load cases, actions, and combinations are managed in the *Edit Load Cases and Combinations* dialog box (see Figure 5.22) and in the number 2 tables. The latter are accessible by clicking the button depicted on the left. Table 2.*1 Load Cases* shows our five load cases with the defined action categories in an overview.

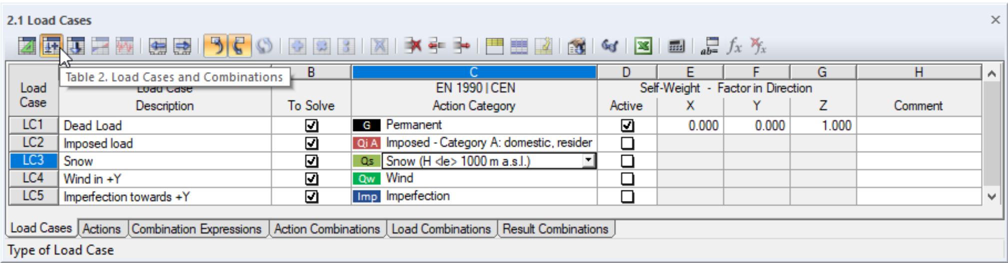

2.1 Load Cases

Table 2. Load Cases and Combinations

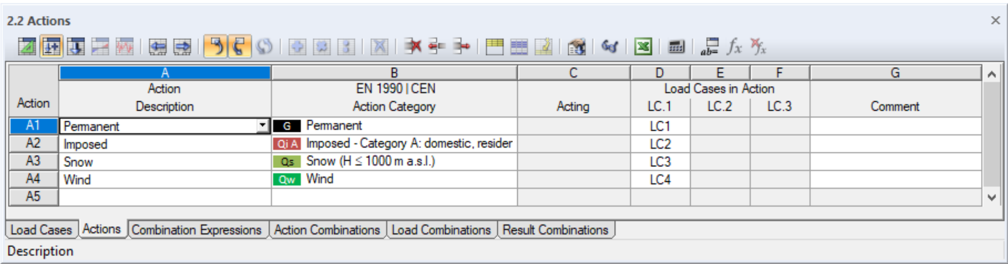
Load Case	Load Case Description	To Solve	EN 1990 | CEN Action Category	Self-Weight - Factor in Direction Active	X	Y	Z	Comment
LC1	Dead Load	☑	G Permanent	☑	0.000	0.000	1.000	
LC2	Imposed load	☑	Qi A Imposed - Category A: domestic, resider	☐				
LC3	Snow	☑	Qs Snow (H <le> 1000 m a.s.l.)	☐				
LC4	Wind in +Y	☑	Qw Wind	☐				
LC5	Imperfection towards +Y	☑	Imp Imperfection	☐				

Load Cases | Actions | Combination Expressions | Action Combinations | Load Combinations | Result Combinations

Type of Load Case

Figure 6.1 Table 2.1 *Load Cases*

The next table 2.2 *Actions* shows, which load cases are contained in the individual actions. In our example, each load case is assigned to a different action. However, had we defined several wind load cases for the different directions, for example, they would all be listed in the *Wind* action.


2.2 Actions

Action	Action Description	EN 1990 | CEN Action Category	Acting	Load Cases in Action LC.1	LC.2	LC.3	Comment
A1	Permanent	G Permanent		LC1			
A2	Imposed	Qi A Imposed - Category A: domestic, resider		LC2			
A3	Snow	Qs Snow (H ≤ 1000 m a.s.l.)		LC3			
A4	Wind	Qw Wind		LC4			
A5							

Load Cases | Actions | Combination Expressions | Action Combinations | Load Combinations | Result Combinations

Description

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 to verify the ultimate limit states and serviceability limit states according to certain rules. Table 2.3 *Combination Expressions* manages, which limit states are set to be analyzed.

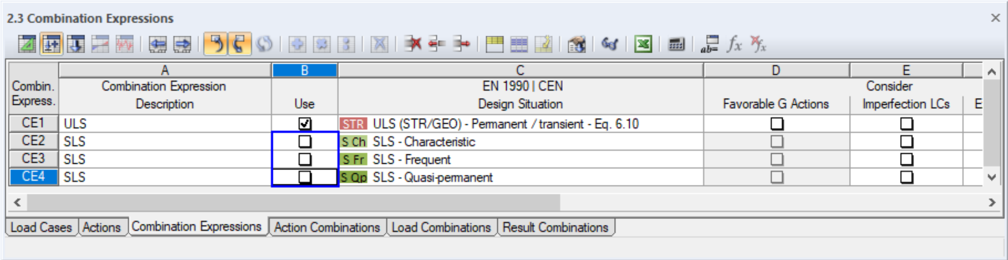


Figure 6.3 Table 2.3 *Combination Expressions*

For our example, only the ultimate limit state (*ULS*) is relevant. For that reason, we clear the serviceability limit state (*SLS*) check boxes in the *Use* column of the combination expressions.

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

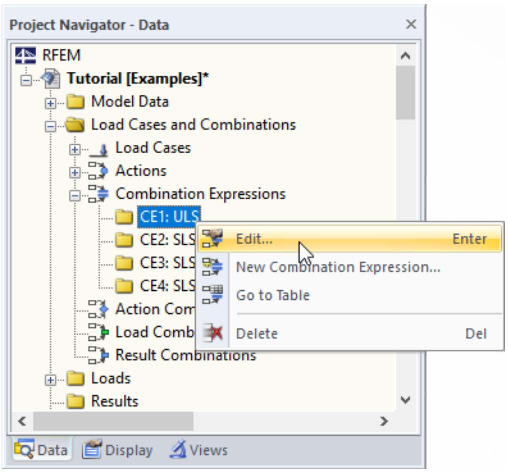


Figure 6.4 Navigator shortcut menu of combination expressions

Using [i] in the dialog box (see Figure 6.5), we can obtain information about the combination expression for the STR/GEO design situation.

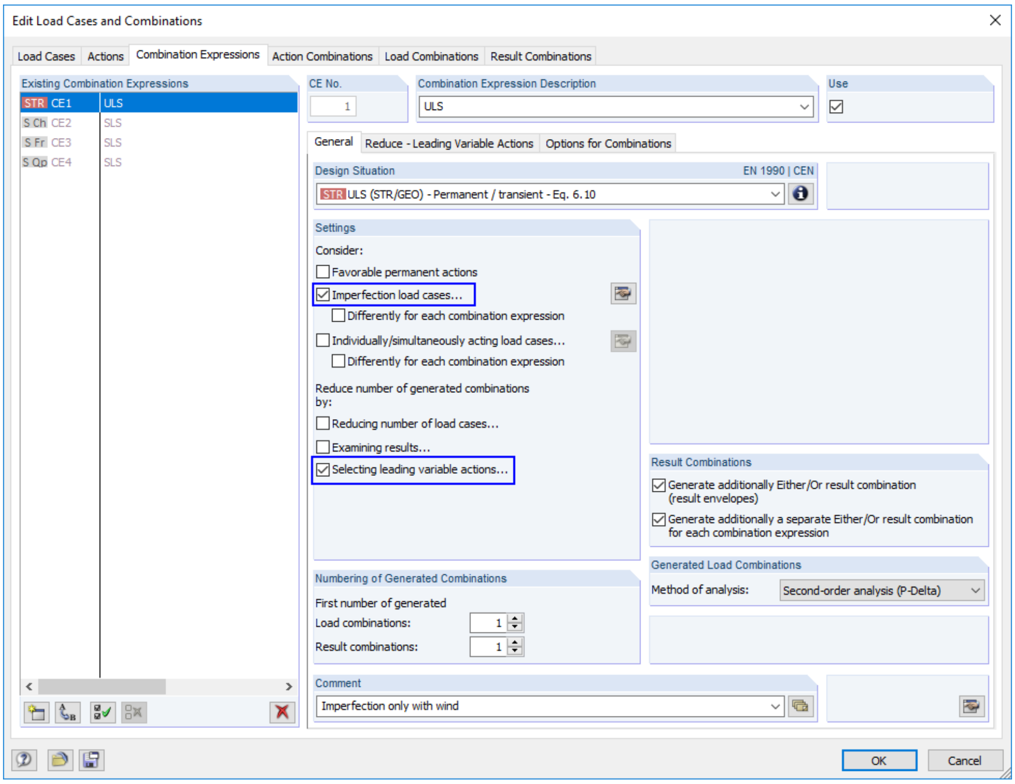


Figure 6.5 *Edit Load Cases and Combinations* dialog box, *Combination Expressions* tab

In the *Settings* section, we select **Imperfection load cases** to *Consider* the imperfections when the combinations are generated. The following dialog box opens when selecting it.

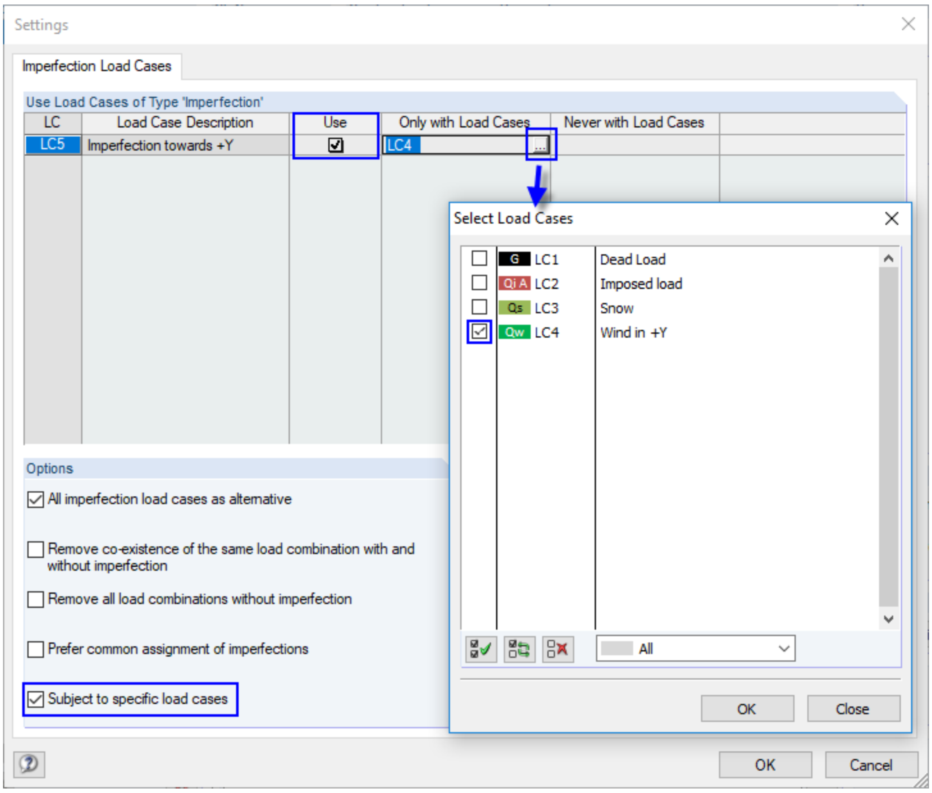


Figure 6.6 *Settings* dialog box for imperfection load cases.

For LC4 to be taken into account, we check the box in the *Use* column.

In the *Options* section, we activate the **Subject to specific load cases** function. We click into the *Only with Load Cases* box. A button appears at its end. It opens the *Select Load Cases* dialog box, where we select **LC4 Wind in +Y**. This way, RFEM will consider imperfections only in combinations that include wind load cases.

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

In the *Settings* section of the initial dialog box (Figure 6.5), we can reduce the number of generated combinations by **Selecting leading variable actions**. Selecting this option adds a new tab to the dialog box.

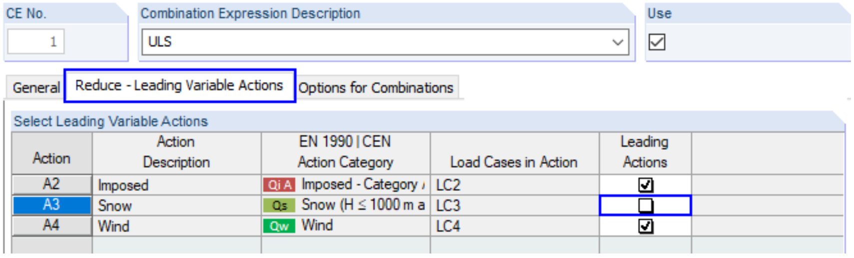


Figure 6.7 Selecting leading actions in the *Reduce - Leading Variable Actions* tab

We clear Action **A3** in the *Reduce - Leading Variable Actions* tab, because the *Snow* load case is only meant to be superimposed as an accompanying action. This reduces the number of generated combinations.

Before we confirm the *Edit Load Cases and Combinations* dialog box, we check if the **Generate additionally Either/Or result combination** option is selected in the *General* tab. This result combination provides the extreme values of all load combinations (result envelopes).

After clicking [OK], we proceed to the next Table 2.4 *Action Combinations*. This generates the action combinations. If we return to Table 2.3 *Combination Expressions*, we can find 13 *Generated Action Combinations* in column J.

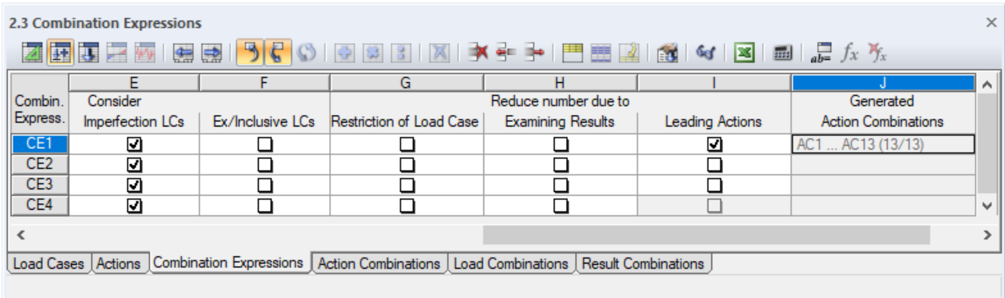
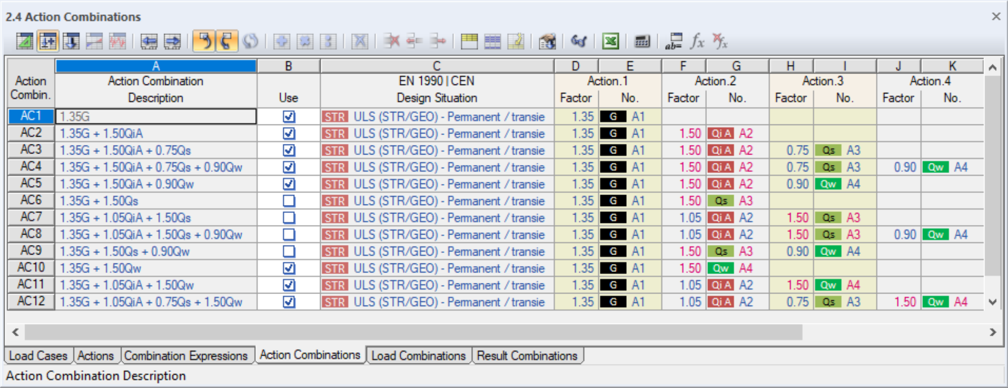


Figure 6.8 Table 2.3 *Combination Expressions, Generated Action Combinations* column

6.3 Creating Action Combinations

RFEM creates 13 action combinations (see Figure 6.8). They are listed in Table 2.4 *Action Combinations* according to actions.



2.4 Action Combinations

Action Combin.	A: Action Combination Description	B: Use	C: EN 1990 | CEN Design Situation	D: Action.1 Factor	E: Action.1 No.	F: Action.2 Factor	G: Action.2 No.	H: Action.3 Factor	I: Action.3 No.	J: Action.4 Factor	K: Action.4 No.
AC1	1.35G	☑	STR ULS (STR/GEO) - Permanent / transie	1.35	G A1						
AC2	1.35G + 1.50QiA	☑	STR ULS (STR/GEO) - Permanent / transie	1.35	G A1	1.50	QiA A2				
AC3	1.35G + 1.50QiA + 0.75Qs	☑	STR ULS (STR/GEO) - Permanent / transie	1.35	G A1	1.50	QiA A2	0.75	Qs A3		
AC4	1.35G + 1.50QiA + 0.75Qs + 0.90Qw	☑	STR ULS (STR/GEO) - Permanent / transie	1.35	G A1	1.50	QiA A2	0.75	Qs A3	0.90	Qw A4
AC5	1.35G + 1.50QiA + 0.90Qw	☑	STR ULS (STR/GEO) - Permanent / transie	1.35	G A1	1.50	QiA A2	0.90	Qw A4		
AC6	1.35G + 1.50Qs	☐	STR ULS (STR/GEO) - Permanent / transie	1.35	G A1	1.50	Qs A3				
AC7	1.35G + 1.05QiA + 1.50Qs	☐	STR ULS (STR/GEO) - Permanent / transie	1.35	G A1	1.05	QiA A2	1.50	Qs A3		
AC8	1.35G + 1.05QiA + 1.50Qs + 0.90Qw	☐	STR ULS (STR/GEO) - Permanent / transie	1.35	G A1	1.05	QiA A2	1.50	Qs A3	0.90	Qw A4
AC9	1.35G + 1.50Qs + 0.90Qw	☐	STR ULS (STR/GEO) - Permanent / transie	1.35	G A1	1.50	Qs A3	0.90	Qw A4		
AC10	1.35G + 1.50Qw	☑	STR ULS (STR/GEO) - Permanent / transie	1.35	G A1	1.50	Qw A4				
AC11	1.35G + 1.05QiA + 1.50Qw	☑	STR ULS (STR/GEO) - Permanent / transie	1.35	G A1	1.05	QiA A2	1.50	Qw A4		
AC12	1.35G + 1.05QiA + 0.75Qs + 1.50Qw	☑	STR ULS (STR/GEO) - Permanent / transie	1.35	G A1	1.05	QiA A2	0.75	Qs A3	1.50	Qw A4

Load Cases | Actions | Combination Expressions | Action Combinations | Load Combinations | Result Combinations

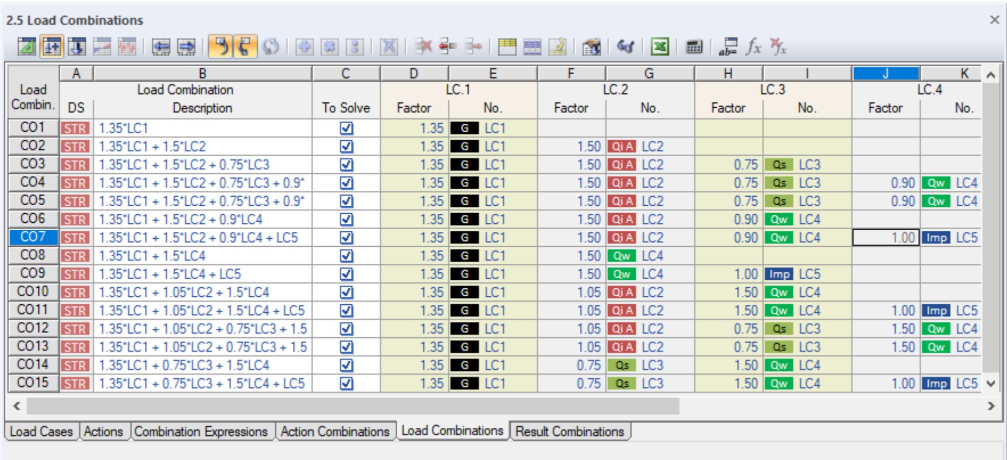
Action Combination Description

Figure 6.9 Table 2.4 *Action Combinations*

This overview corresponds to the presentation of actions described in the standards. The *Use* column determines, which action combinations are considered for the generation of load combinations. Since we specified the Q_s (Snow) action to be just an accompanying action, its action combinations in which the action Q_s is the leading action are not selected.

6.4 Creating Load Combinations

When we browse to Table 2.5 *Load Combinations*, 15 load combinations are automatically generated from the nine relevant action combinations (see Figure 6.9).



2.5 Load Combinations

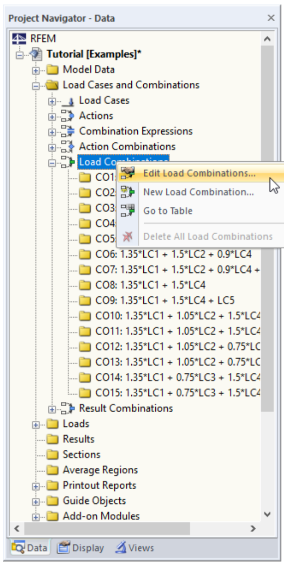
Load Combin.	A: DS	B: Load Combination Description	C: To Solve	D: LC.1 Factor	E: LC.1 No.	F: LC.2 Factor	G: LC.2 No.	H: LC.3 Factor	I: LC.3 No.	J: LC.4 Factor	K: LC.4 No.
CO1	STR	1.35*LC1	☑	1.35	G LC1						
CO2	STR	1.35*LC1 + 1.5*LC2	☑	1.35	G LC1	1.50	QiA LC2				
CO3	STR	1.35*LC1 + 1.5*LC2 + 0.75*LC3	☑	1.35	G LC1	1.50	QiA LC2	0.75	Qs LC3		
CO4	STR	1.35*LC1 + 1.5*LC2 + 0.75*LC3 + 0.9*	☑	1.35	G LC1	1.50	QiA LC2	0.75	Qs LC3	0.90	Qw LC4
CO5	STR	1.35*LC1 + 1.5*LC2 + 0.75*LC3 + 0.9*	☑	1.35	G LC1	1.50	QiA LC2	0.75	Qs LC3	0.90	Qw LC4
CO6	STR	1.35*LC1 + 1.5*LC2 + 0.9*LC4	☑	1.35	G LC1	1.50	QiA LC2	0.90	Qw LC4		
CO7	STR	1.35*LC1 + 1.5*LC2 + 0.9*LC4 + LC5	☑	1.35	G LC1	1.50	QiA LC2	0.90	Qw LC4	1.00	Imp LC5
CO8	STR	1.35*LC1 + 1.5*LC4	☑	1.35	G LC1	1.50	Qw LC4				
CO9	STR	1.35*LC1 + 1.5*LC4 + LC5	☑	1.35	G LC1	1.50	Qw LC4	1.00	Imp LC5		
CO10	STR	1.35*LC1 + 1.05*LC2 + 1.5*LC4	☑	1.35	G LC1	1.05	QiA LC2	1.50	Qw LC4		
CO11	STR	1.35*LC1 + 1.05*LC2 + 1.5*LC4 + LC5	☑	1.35	G LC1	1.05	QiA LC2	1.50	Qw LC4	1.00	Imp LC5
CO12	STR	1.35*LC1 + 1.05*LC2 + 0.75*LC3 + 1.5	☑	1.35	G LC1	1.05	QiA LC2	0.75	Qs LC3	1.50	Qw LC4
CO13	STR	1.35*LC1 + 1.05*LC2 + 0.75*LC3 + 1.5	☑	1.35	G LC1	1.05	QiA LC2	0.75	Qs LC3	1.50	Qw LC4
CO14	STR	1.35*LC1 + 0.75*LC3 + 1.5*LC4	☑	1.35	G LC1	0.75	Qs LC3	1.50	Qw LC4		
CO15	STR	1.35*LC1 + 0.75*LC3 + 1.5*LC4 + LC5	☑	1.35	G LC1	0.75	Qs LC3	1.50	Qw LC4	1.00	Imp LC5

Load Cases | Actions | Combination Expressions | Action Combinations | Load Combinations | Result Combinations

Figure 6.10 Table 2.5 *Load Combinations*

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

As required, imperfections can only be found in combination with wind actions Q_w.



Load Combinations shortcut menu

Using the navigator shortcut menu shown on the left, we reopen *Edit Load Cases and Combinations* to view the generated load combinations in the dialog box.

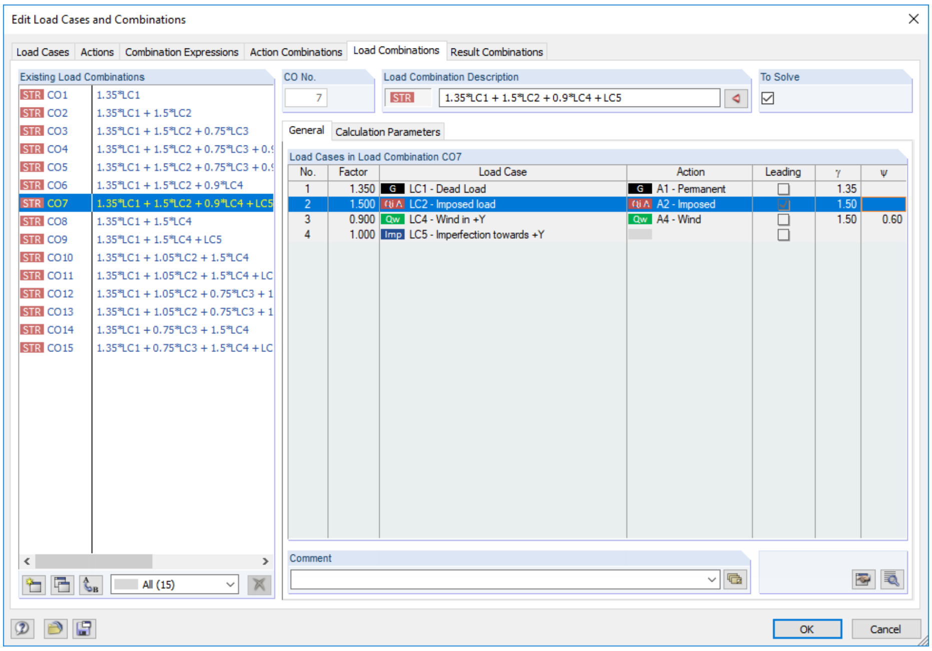


Figure 6.11 *Edit Load Cases and Combinations* dialog box, *Load Combinations* tab

When browsing the list of *Existing Load Combinations*, the load cases with partial safety factors and combination coefficients are shown in the right section. Load cases that act *Leading* in a combination are marked accordingly.

The partial safety factors and combination coefficients are available using the [Details] button.

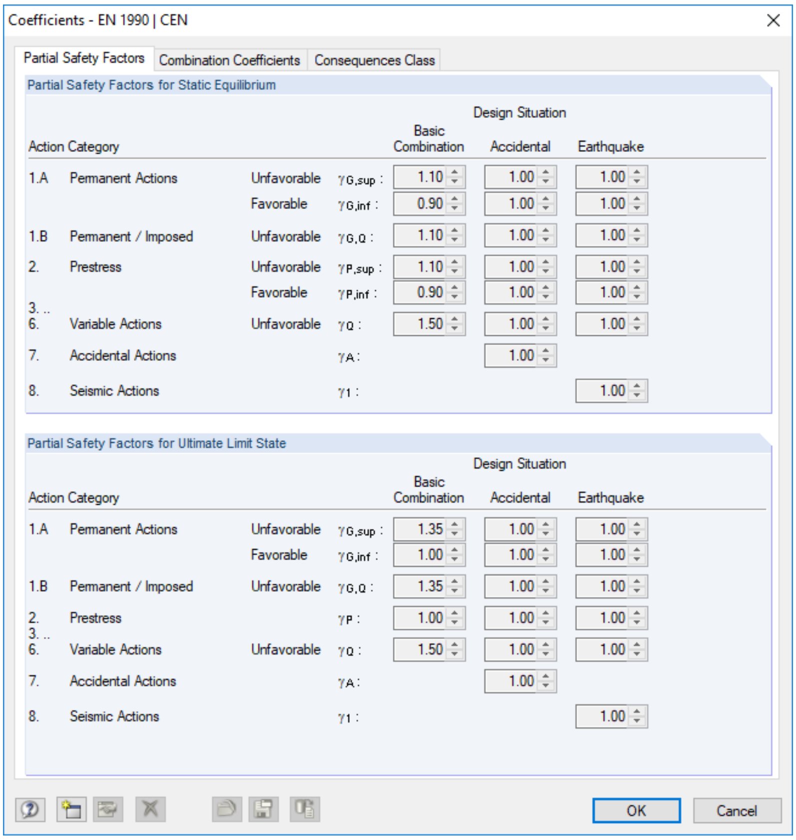


Figure 6.12 Coefficients dialog box, *Partial Safety Factors* tab

We can check the specifications RFEM uses to calculate the individual load combinations in the *Calculation Parameters* tab.

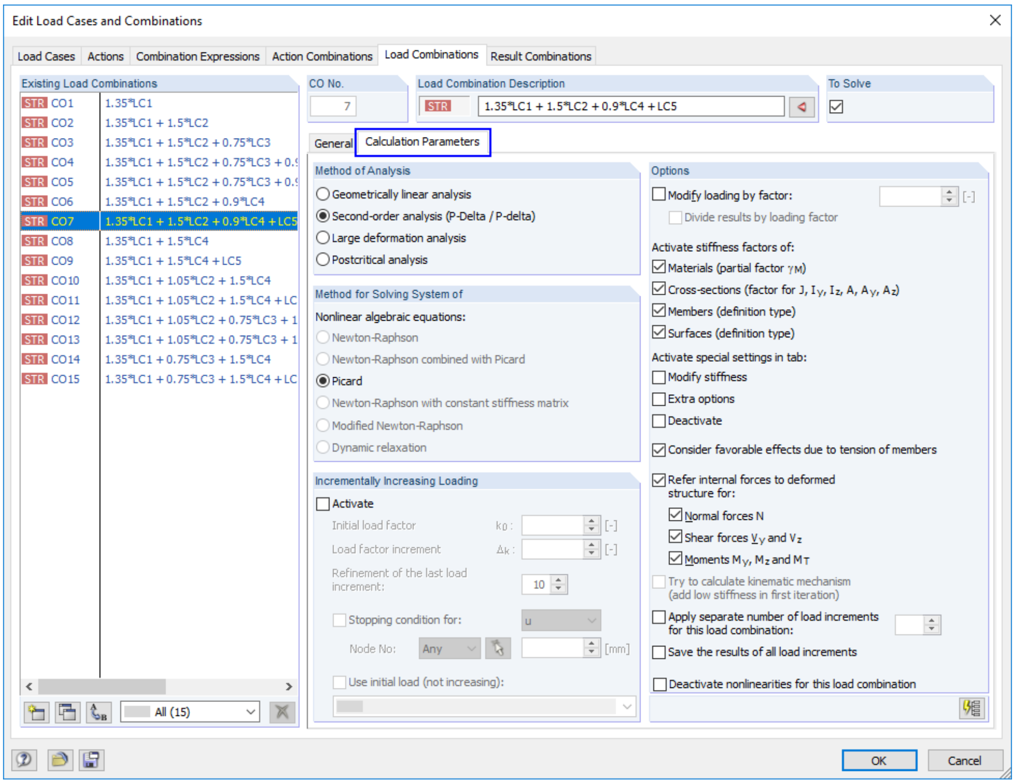


Figure 6.13 Checking the *Calculation Parameters* of a load combination

Generally, load combinations are analyzed non-linearly according to the *second-order analysis*.

6.5 Checking Result Combinations

When defining the combination expressions, we have the *Generate additionally Either/Or result combination* option active (see Figure 6.5). This allows for the extreme values of every load combination to be determined.

RFEM generates a results envelope from the load combinations. The conditions for superimposition are available in the last tab of the *Edit Load Cases and Combinations* dialog box and in Table 2.6 *Result Combinations*.

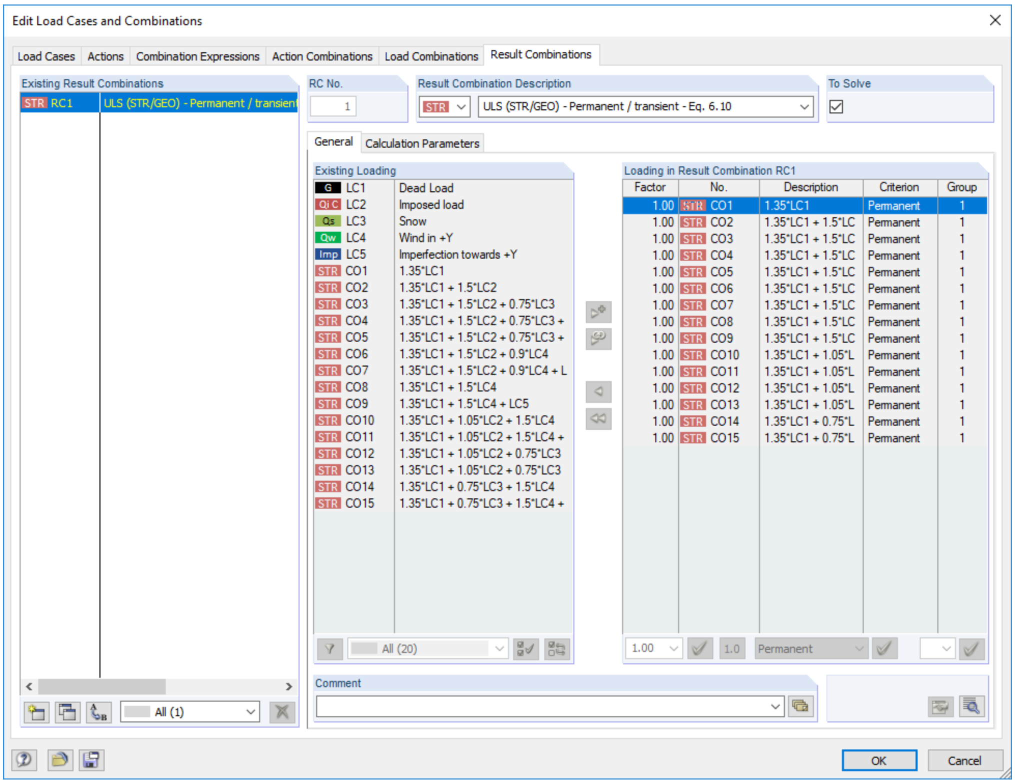
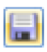


Figure 6.14 *Edit Load Cases and Combinations* dialog box, *Result Combinations* tab

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



The combination criteria are now completely defined. We can [Save] the current data again.

7 Calculation

7.1 Checking Input Data

Before we calculate our structure, we want RFEM to check our input. We open the menu

Tools → Plausibility Check

and define the following settings in the *Plausibility Check* dialog box.

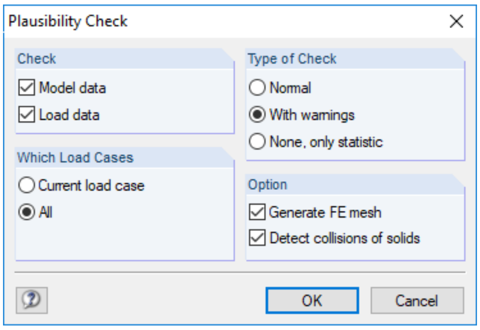


Figure 7.1 *Plausibility Check* dialog box

If no inconsistencies are found after clicking [OK], a corresponding message is displayed with a summary of the model and load data.

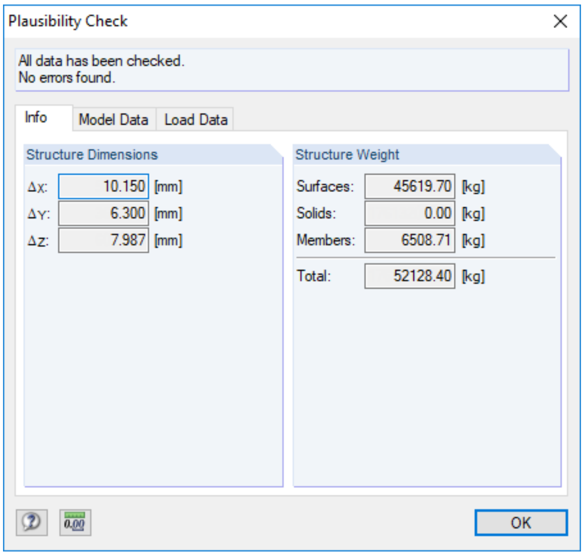


Figure 7.2 Result of plausibility check

We can find more tools for checking the data by selecting

Tools → Model Check

which can also be applied to our model.

7.2 Creating the FE Mesh

Generating the FE mesh

Since the *Generate FE mesh* option was selected in the *Plausibility Check* dialog box (see Figure 7.1), a mesh with a standard mesh size of 50 cm was generated automatically. The default mesh size can be changed using **Calculate → FE Mesh Settings**.

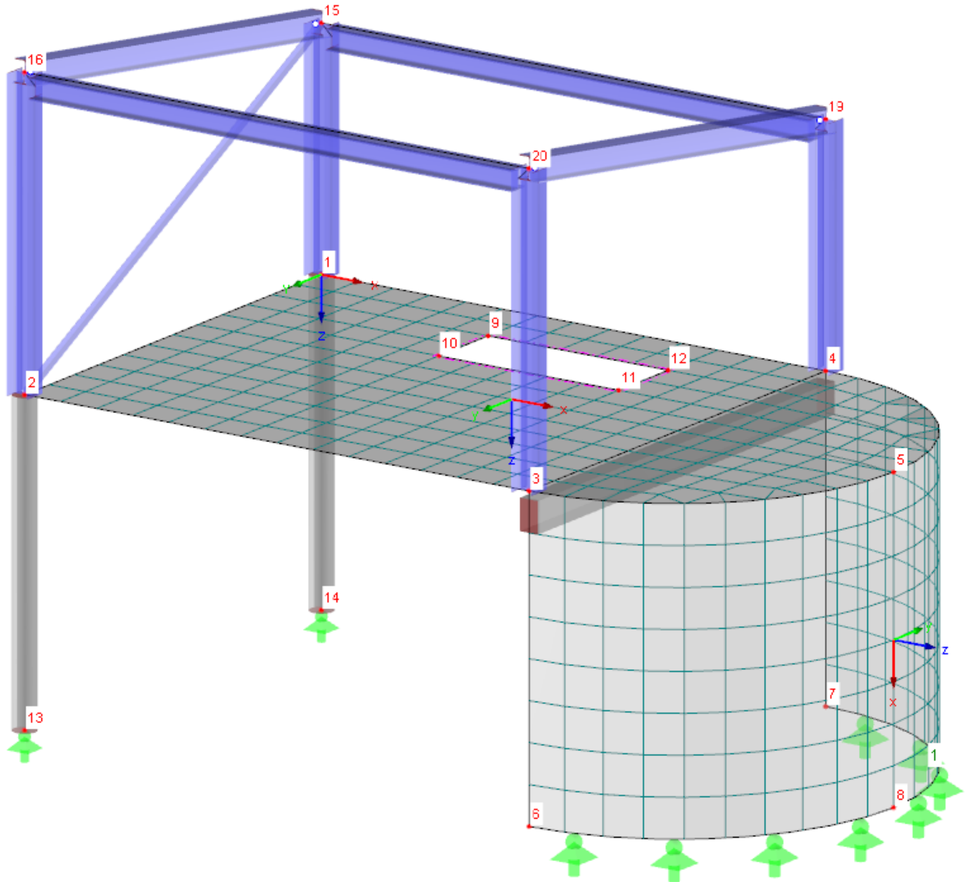


Figure 7.3 Model with generated FE mesh

Creating 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 *Edit Node* dialog box. We go to the *FE Mesh* tab and select the **Available** box (see Figure 7.4).

Since no FE mesh refinement type has been defined yet, the *New FE Mesh Refinement* dialog box opens automatically.

The *Node - circular* default setting and the suggested *Parameters* can remain as they are. After confirming both dialog boxes with [OK], the FE mesh is deleted.

A spherical refinement area is displayed on the selected node.

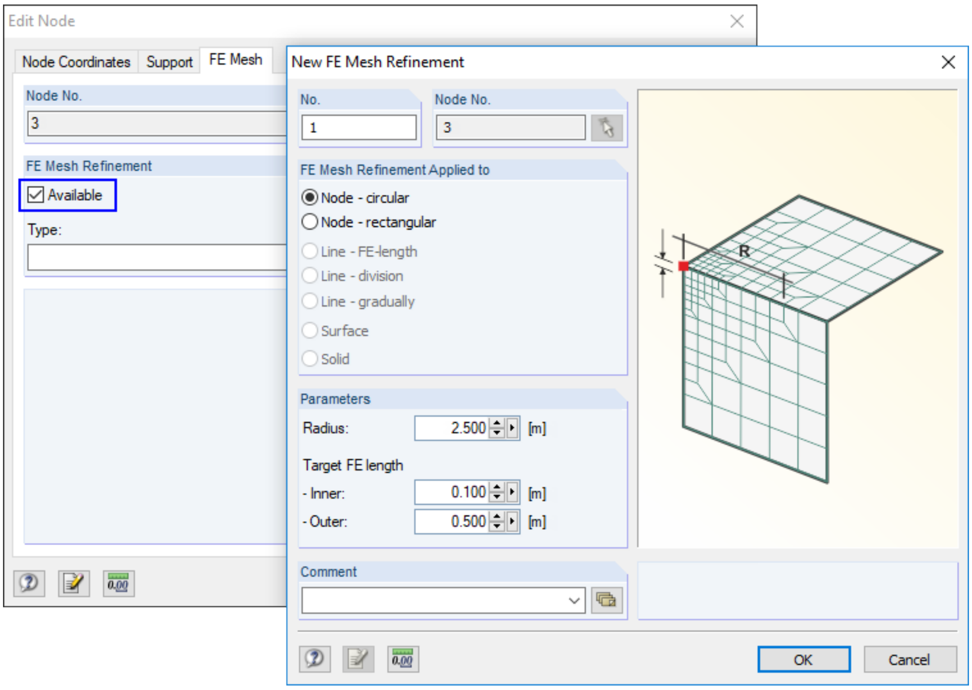
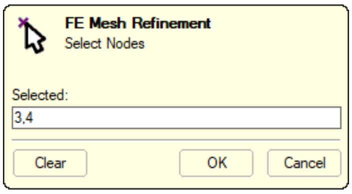


Figure 7.4 *Edit Node* and *New FE Mesh Refinement* dialog boxes

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



After clicking the button, we select the rib's second node graphically in the work window.

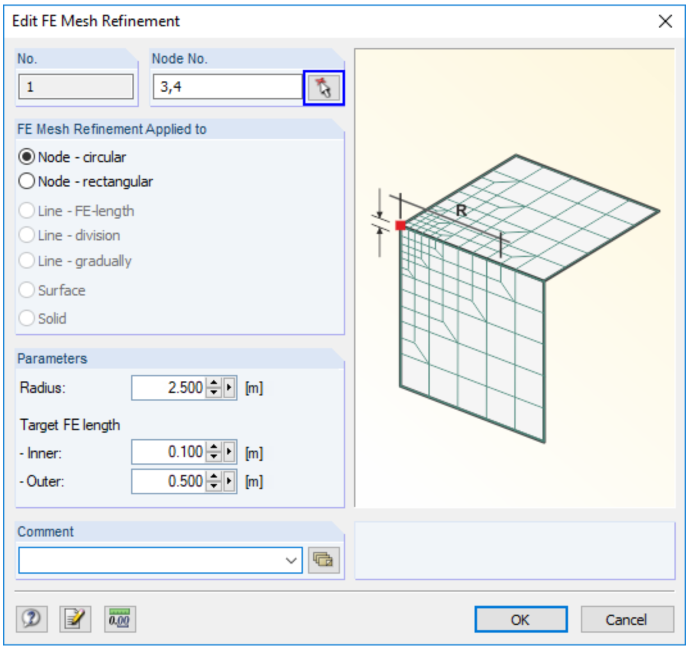


Figure 7.5 Defining the second node graphically using the button

We close all dialog boxes with [OK].

We allow the mesh to be generated again using

Calculate → Generate FE Mesh.

Afterwards we check the refinement areas.

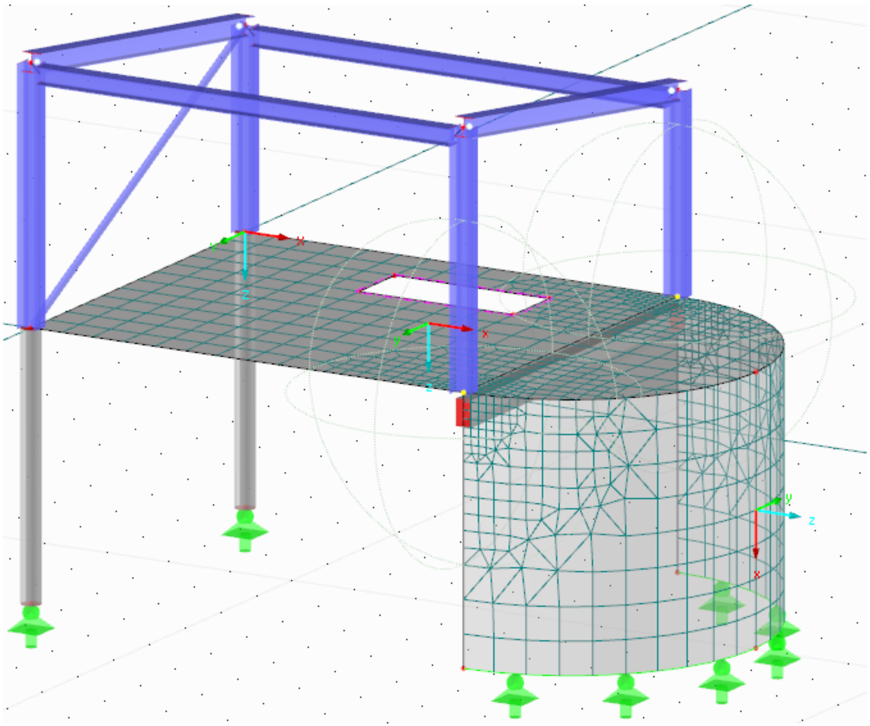


Figure 7.6 FE mesh with refinement areas

7.3 Calculating the Model

To start the calculation, use the menu item

Calculate → Calculate All

or the corresponding toolbar button.

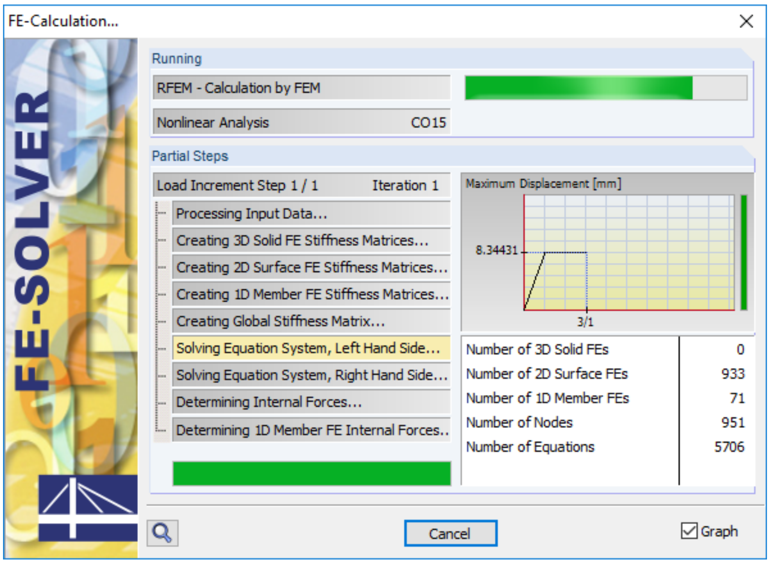


Figure 7.7 Calculation process

8 Results

8.1 Graphical Results

As soon as the calculation is finished, RFEM displays the deformations of the current load case.

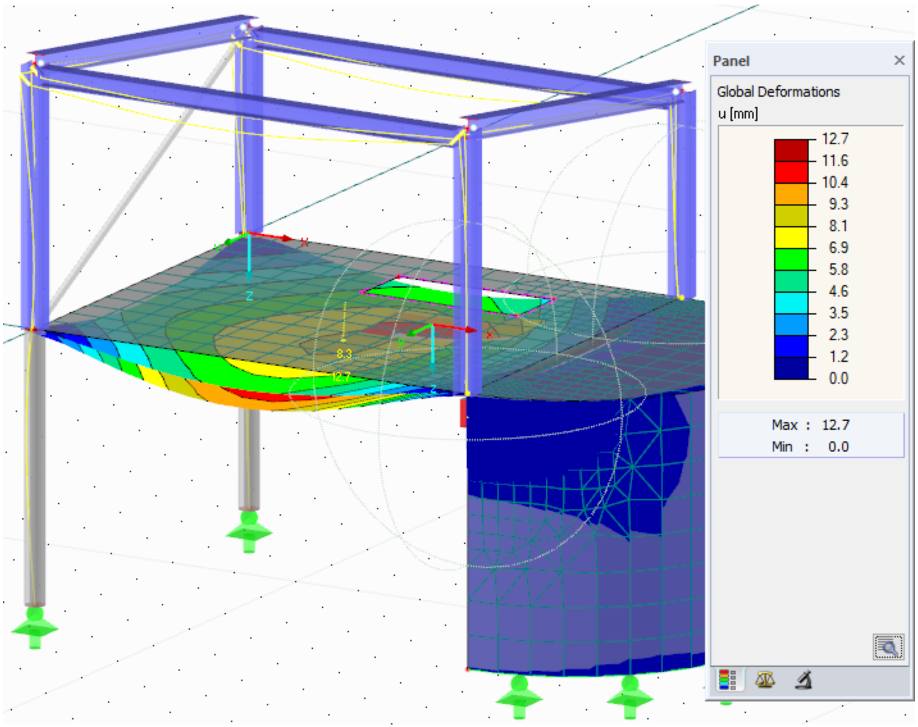


Figure 8.1 Graphic of deformations for result combination RC1

Selecting load cases and load combinations

As we know from checking the load cases, we can use ◁ and ▷ in the toolbar (to the right of the load case list) to switch between the results of load cases, load combinations, and result combinations. You may also select specific load cases or combinations from the list.

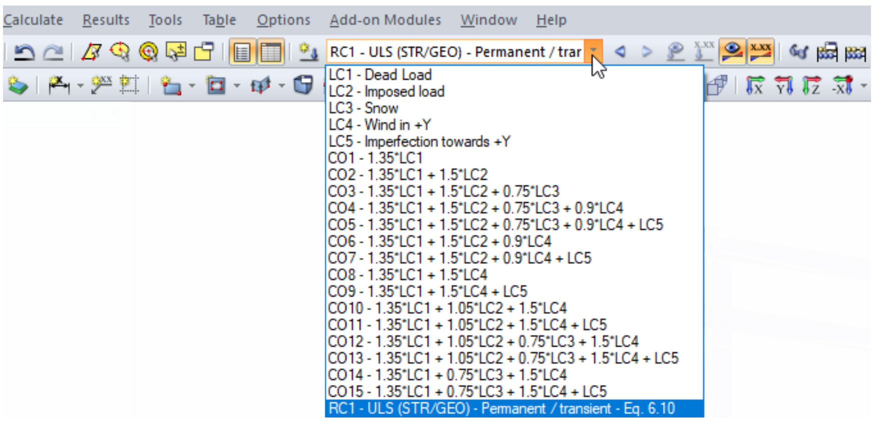


Figure 8.2 List of load cases in the toolbar

Selecting results in the navigator

A new, fourth navigator has appeared that manages the various result types for the graphical display. We only have access to this *Results* navigator when the results display is active. The results can be displayed and hidden in the *Display* navigator or by using the [Show Results] button.

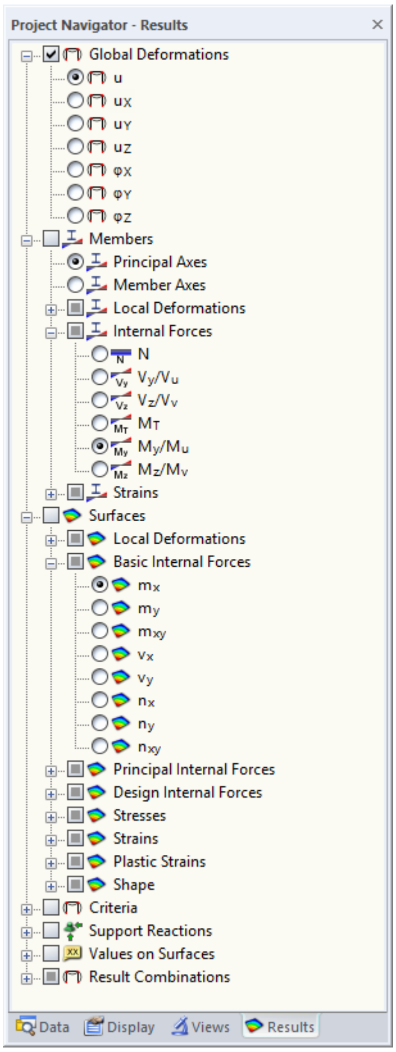


Figure 8.3 *Results navigator*

There are check boxes next to every result category (for example *Global Deformations*, *Members*, *Surfaces*, *Support Reactions*). When we select one of the boxes, the corresponding deformation or internal force is displayed. Next to the entries listed within the categories are more check boxes used to adjust the desired type of result.

Now we can browse through the individual load cases and load combinations. The different result categories allow us to display deformations, internal forces of members and surfaces, and stresses or support forces.

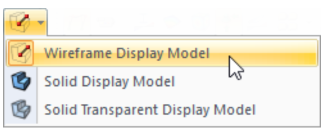


Figure 8.4 shows the internal forces of members M_z and the internal forces of surfaces m_y, which were calculated for CO13. For displaying internal forces, we recommend to use the wire-frame model, which can be selected with the button shown on the left.

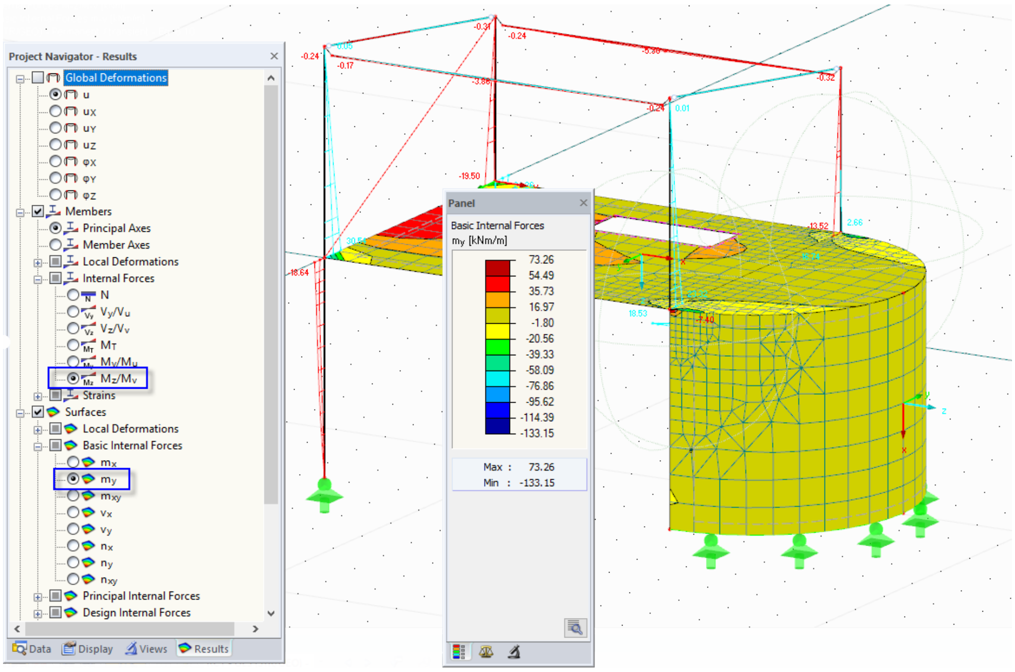


Figure 8.4 Displaying internal forces of members and surfaces

Displaying values

The control panel's color scale informs us about the assignment of color ranges. The result values for certain locations may also be displayed by checking the **Values on Surfaces** option in the *Results* navigator. To show all values of the FE mesh, we have to clear the *Extreme Values* checkbox.

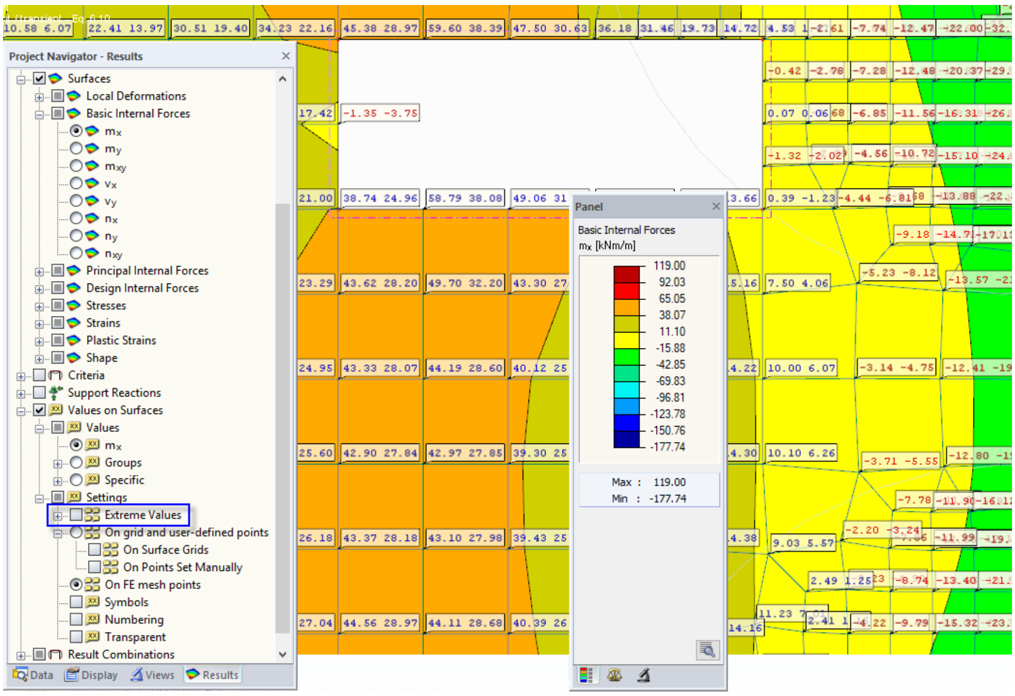


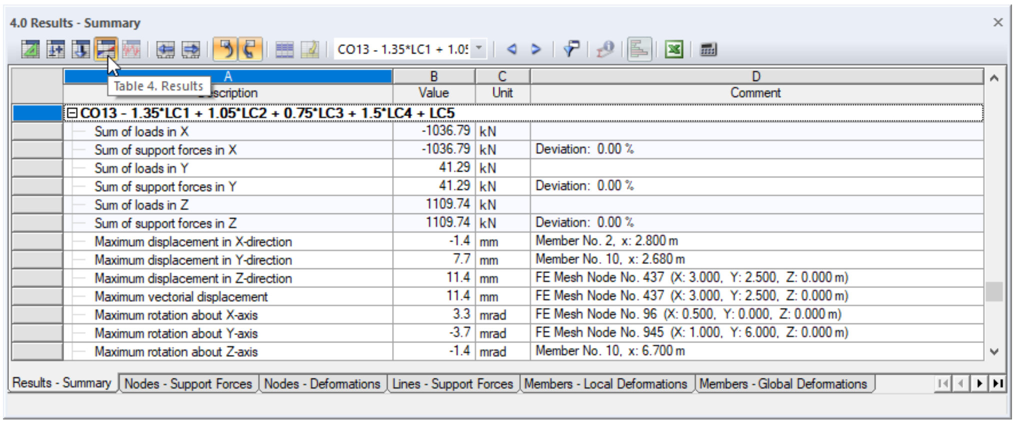
Figure 8.5 Grid point moments m_x of floor slab in Z-view (CO13)

8.2 Results Tables

We can view the results in the tables as well.

Displaying the tables

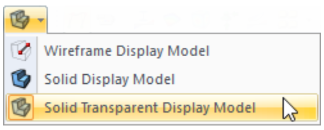
The results tables are automatically displayed after calculation. Table *4.0 Results - Summary* offers a summary of the calculation process, sorted by load cases and load combinations.



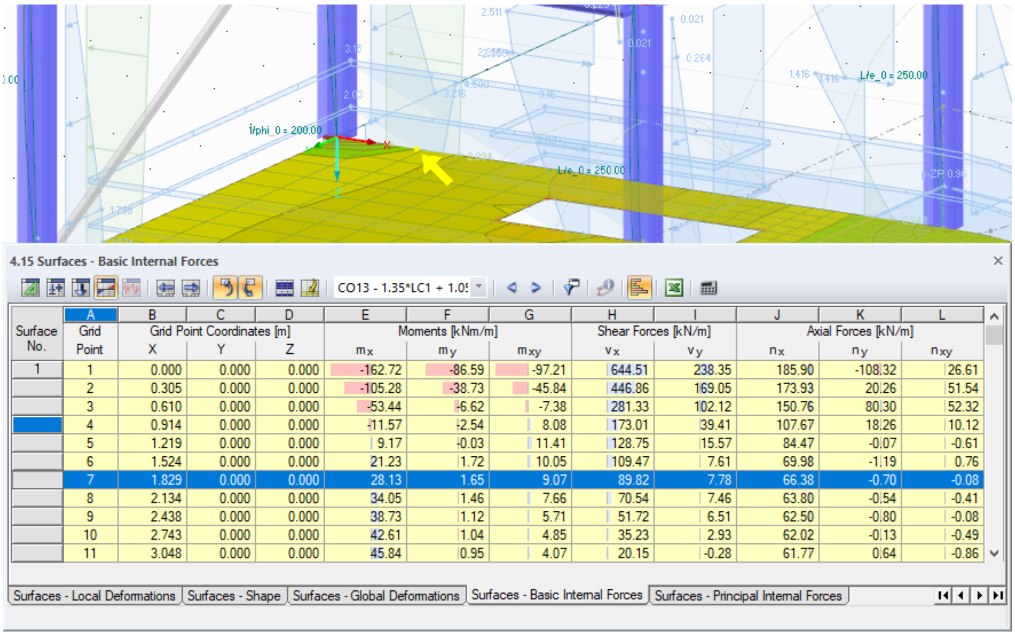
4.0 Results - Summary

A Description	B Value	C Unit	D Comment
CO13 - 1.35*LC1 + 1.05*LC2 + 0.75*LC3 + 1.5*LC4 + LC5			
Sum of loads in X	-1036.79	kN	
Sum of support forces in X	-1036.79	kN	Deviation: 0.00 %
Sum of loads in Y	41.29	kN	
Sum of support forces in Y	41.29	kN	Deviation: 0.00 %
Sum of loads in Z	1109.74	kN	
Sum of support forces in Z	1109.74	kN	Deviation: 0.00 %
Maximum displacement in X-direction	-1.4	mm	Member No. 2, x: 2.800 m
Maximum displacement in Y-direction	7.7	mm	Member No. 10, x: 2.680 m
Maximum displacement in Z-direction	11.4	mm	FE Mesh Node No. 437 (X: 3.000, Y: 2.500, Z: 0.000 m)
Maximum vectorial displacement	11.4	mm	FE Mesh Node No. 437 (X: 3.000, Y: 2.500, Z: 0.000 m)
Maximum rotation about X-axis	3.3	mrad	FE Mesh Node No. 96 (X: 0.500, Y: 0.000, Z: 0.000 m)
Maximum rotation about Y-axis	-3.7	mrad	FE Mesh Node No. 945 (X: 1.000, Y: 6.000, Z: 0.000 m)
Maximum rotation about Z-axis	-1.4	mrad	Member No. 10, x: 6.700 m

Figure 8.6 Table *4.0 Results - Summary*



Use the tabs to view the other tables. To find the internal forces of the floor slabs in the table, for example, go to Table *4.15 Surfaces - Basic Internal Forces*. If the surface is now selected by mouse click (The *Solid Transparent Display Model* makes the selection easier), the program jumps to the surface's basic internal forces in the table. The current grid point, i.e. the pointer's position in the table row, is indicated by an arrow in the graphic.



4.15 Surfaces - Basic Internal Forces

Surface No.	A Grid Point	B Grid Point Coordinates [m] X	C Y	D Z	E Moments [kNm/m] m_x	F m_y	G m_{xy}	H Shear Forces [kN/m] v_x	I v_y	J Axial Forces [kN/m] n_x	K n_y	L n_{xy}
1	1	0.000	0.000	0.000	-162.72	-86.59	-97.21	644.51	238.35	185.90	-108.32	26.61
	2	0.305	0.000	0.000	-105.28	-38.73	-45.84	446.86	169.05	173.93	20.26	51.54
	3	0.610	0.000	0.000	-53.44	-6.62	-7.38	281.33	102.12	150.76	80.30	52.32
	4	0.914	0.000	0.000	-11.57	-2.54	8.08	173.01	39.41	107.67	18.26	10.12
	5	1.219	0.000	0.000	9.17	-0.03	11.41	128.75	15.57	84.47	-0.07	-0.61
	6	1.524	0.000	0.000	21.23	1.72	10.05	109.47	7.61	69.98	-1.19	0.76
	7	1.829	0.000	0.000	28.13	1.65	9.07	89.82	7.78	66.38	-0.70	-0.08
	8	2.134	0.000	0.000	34.05	1.46	7.66	70.54	7.46	63.80	-0.54	-0.41
	9	2.438	0.000	0.000	38.73	1.12	5.71	51.72	6.51	62.50	-0.80	-0.08
	10	2.743	0.000	0.000	42.61	1.04	4.85	35.23	2.93	62.02	-0.13	-0.49
	11	3.048	0.000	0.000	45.84	0.95	4.07	20.15	-0.28	61.77	0.64	-0.86

Figure 8.7 Surface internal forces in Table 4.15 and marker of current grid point in the model

As seen in the graphic, we can use the list in the toolbar to select a specific load case or ◀ and ▶ to browse the load cases.

Adjusting 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, using the *Edit Surface* dialog box, we make modifications in the *Grid* tab: We select **25** cm for both *b* and *h* as the new distance between grid points.

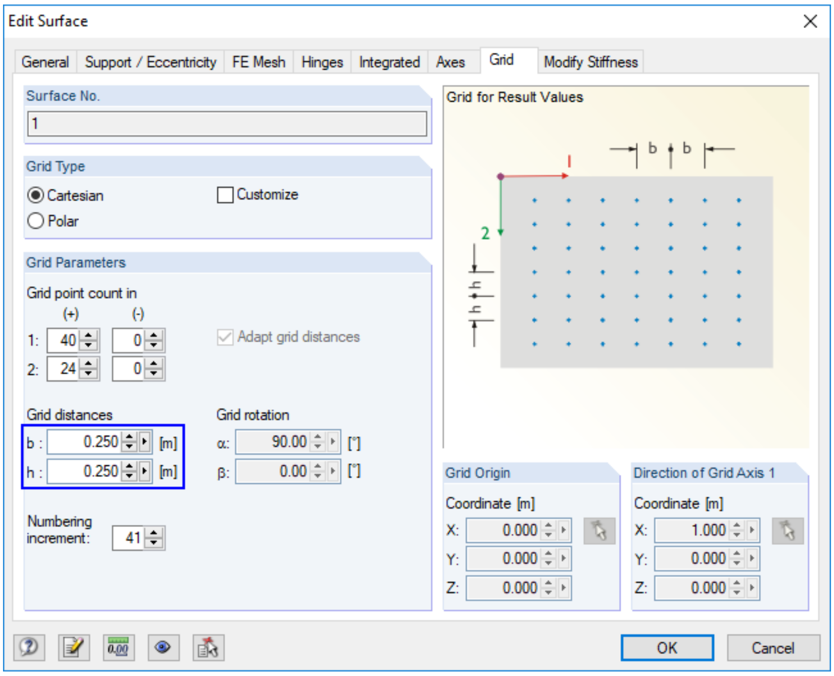
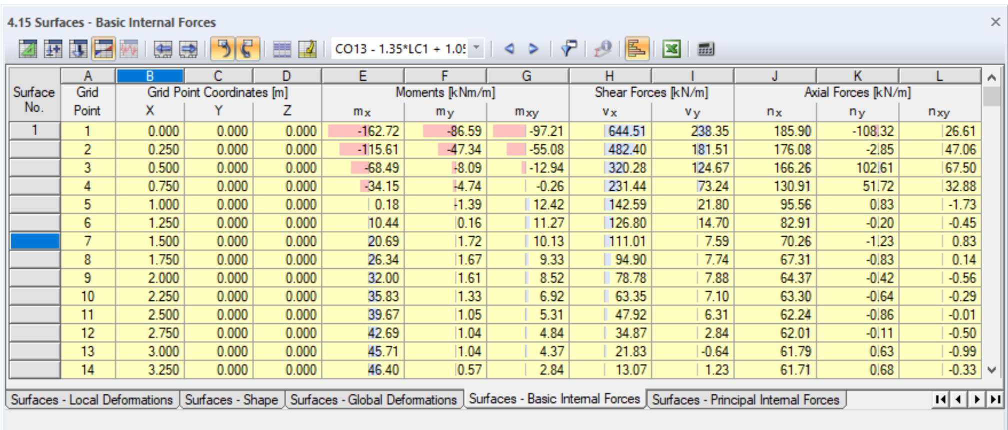


Figure 8.8 Changing the grid for result values

After clicking [OK], the result values are updated in the table. A recalculation is not required because the grid point results are determined from the values available in the FE nodes.



4.15 Surfaces - Basic Internal Forces

CO13 - 1.35*LC1 + 1.0!

Surface No.	A Grid Point	B Grid Point Coordinates [m] X	C Y	D Z	E Moments [kNm/m] mx	F my	G mxy	H Shear Forces [kN/m] vx	I vy	J Axial Forces [kN/m] nx	K ny	L nxy
1	1	0.000	0.000	0.000	-162.72	-86.59	-97.21	644.51	238.35	185.90	-108.32	26.61
	2	0.250	0.000	0.000	-115.61	-47.34	-55.08	482.40	181.51	176.08	-2.85	47.06
	3	0.500	0.000	0.000	-68.49	-8.09	-12.94	320.28	124.67	166.26	102.61	67.50
	4	0.750	0.000	0.000	-34.15	-4.74	-0.26	231.44	73.24	130.91	51.72	32.88
	5	1.000	0.000	0.000	0.18	-1.39	12.42	142.59	21.80	95.56	0.83	-1.73
	6	1.250	0.000	0.000	10.44	0.16	11.27	126.80	14.70	82.91	-0.20	-0.45
	7	1.500	0.000	0.000	20.69	1.72	10.13	111.01	7.59	70.26	-1.23	0.83
	8	1.750	0.000	0.000	26.34	1.67	9.33	94.90	7.74	67.31	-0.83	0.14
	9	2.000	0.000	0.000	32.00	1.61	8.52	78.78	7.88	64.37	-0.42	-0.56
	10	2.250	0.000	0.000	35.83	1.33	6.92	63.35	7.10	63.30	-0.64	-0.29
	11	2.500	0.000	0.000	39.67	1.05	5.31	47.92	6.31	62.24	-0.86	-0.01
	12	2.750	0.000	0.000	42.69	1.04	4.84	34.87	2.84	62.01	-0.11	-0.50
	13	3.000	0.000	0.000	45.71	1.04	4.37	21.83	-0.64	61.79	0.63	-0.99
	14	3.250	0.000	0.000	46.40	0.57	2.84	13.07	1.23	61.71	0.68	-0.33

Surfaces - Local Deformations | Surfaces - Shape | Surfaces - Global Deformations | Surfaces - Basic Internal Forces | Surfaces - Principal Internal Forces

Figure 8.9 Result values of surface 1 with refined grid

8.3 Filtering Results

RFEM provides different tools to display and evaluate results in a clear and structured way. We can use these tools for our example as well.

8.3.1 Custom Visibilities

We already used visibilities when we input the steel frame (see Chapter 4.5.1). This feature is also suitable for evaluating the results.

Displaying results for the concrete columns

We go to the *Views* tab in the navigator. Among the *Visibilities* RFEM generated automatically, we select *Members by Cross-Section* and click the *Circle 300* entry.

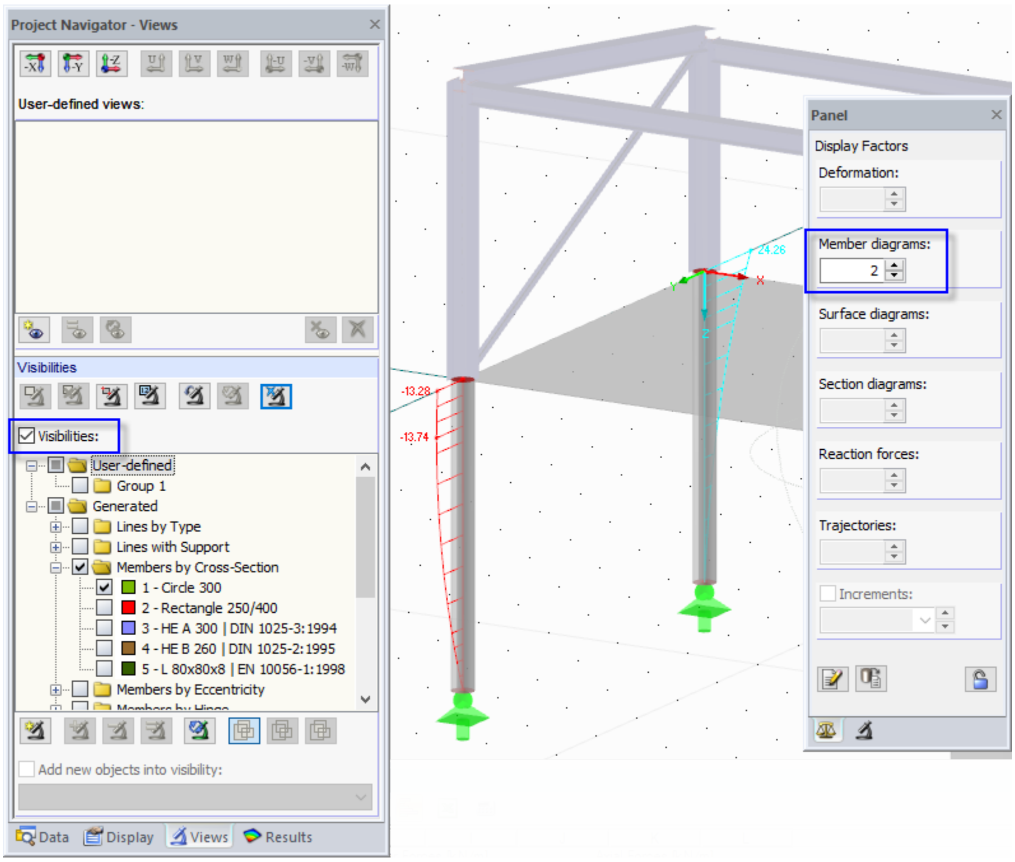


Figure 8.10 Moments M_z of the concrete columns in scaled representation (CO13)

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

Adjusting the scaling factor

To easily read the diagram of internal forces on the rendered model, we scale the data display in the control tab of the panel: We change the *Member diagrams* coefficient to **2** (see Figure 8.10).

Displaying results for the floor slab

In the same way, we can filter by surface result in the *Views* tab. We clear *Members by Cross-Section* and select *Surfaces by Geometry* instead. We select *Plane*.

When we change the result type with the [Deformation] button in the toolbar, we get the following display.

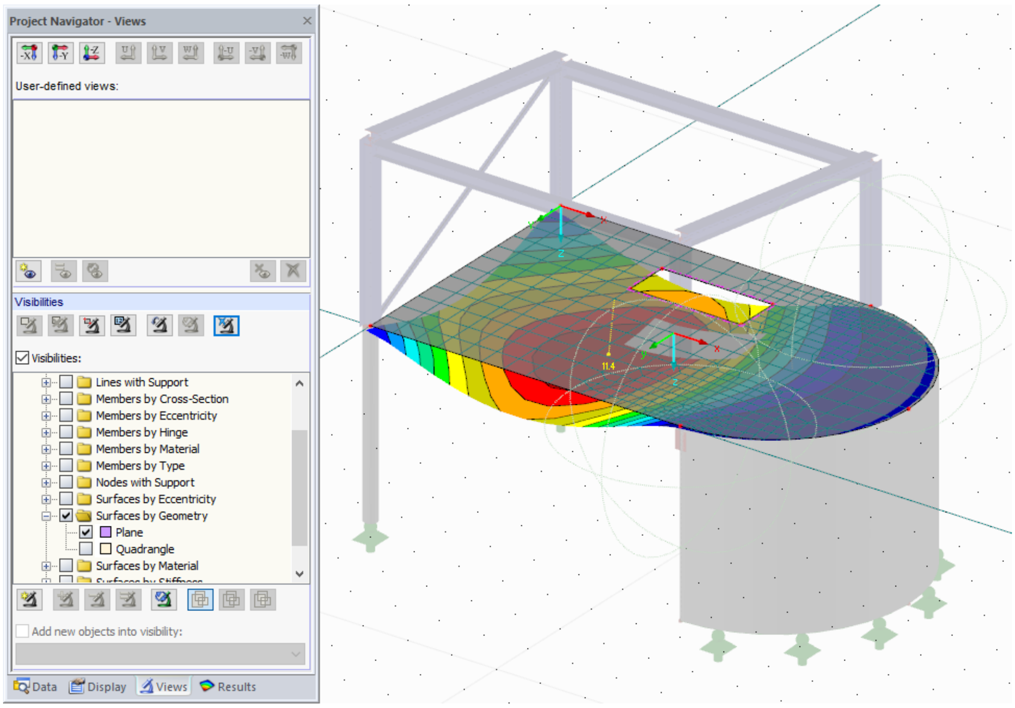


Figure 8.11 Floor deformations (CO13)

As described earlier, we can change the display of result types (deformations, internal forces, stresses, and so on) in the *Results* navigator (see Figure 8.3).

Displaying the steel construction's deformations

We clear *Surfaces by Geometry* in the views navigator and select *Members by Material*. We select *Steel S 235*.

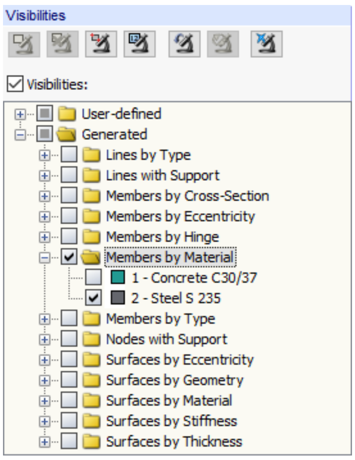


Figure 8.12 Filtering steel members in the *Views* navigator

Now the graphic displays the steel construction's deformations as lines.

It is also possible to display the cross-sections' deformations. For that, we switch to the *Display* navigator and activate

Results → Deformation → Members → Cross-Sections Colored.

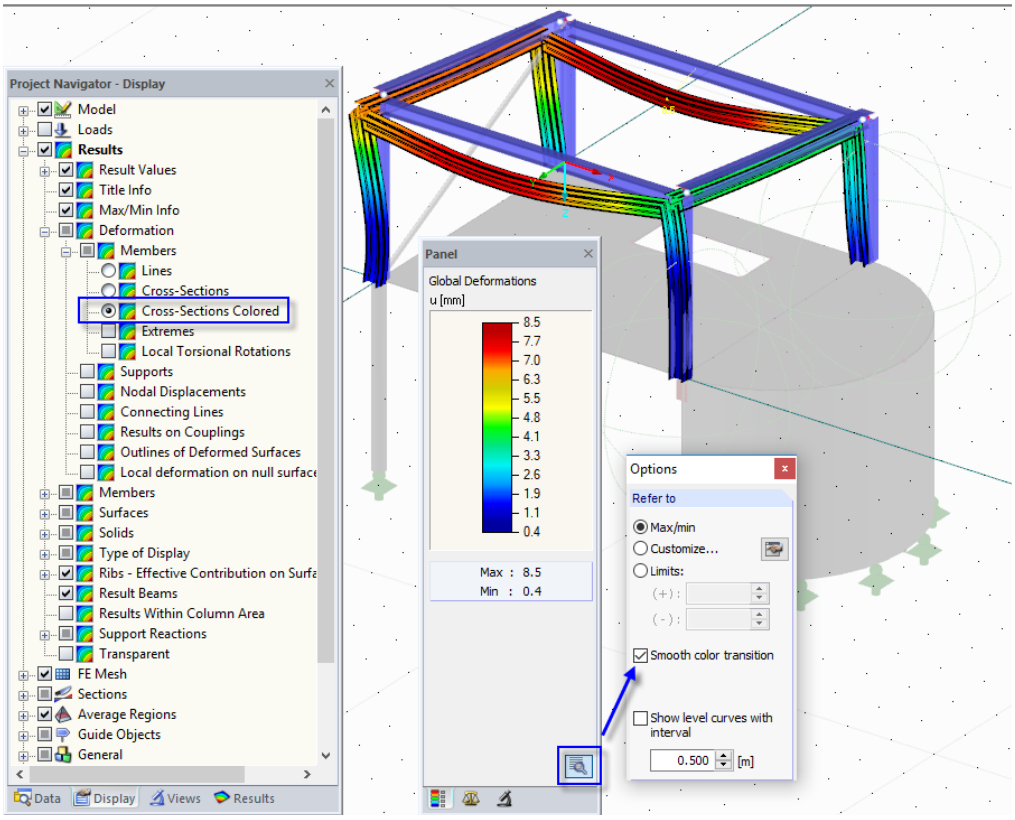


Figure 8.13 Steel cross-section deformations (CO13)

Using , *Smooth color transition* can be activated in the panel.

In CO13 for example, L-profile deformations are not displayed. We have defined this member to be a tension member. However, the deformation shows us that compression forces are occurring. They lead to the failure of the diagonal, rendering the member ineffective in the system.

8.3.2 Results on Objects

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

First we clear *Visibilities* in the *Views* navigator.

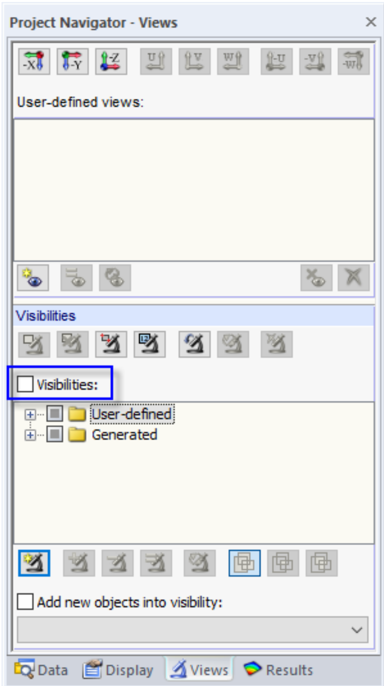


Figure 8.14 Resetting to overall view in the *Views* navigator

In the results navigator, we set the members' axial forces *N* to be displayed. If results for *Surfaces* are displayed as well, we deactivate them.

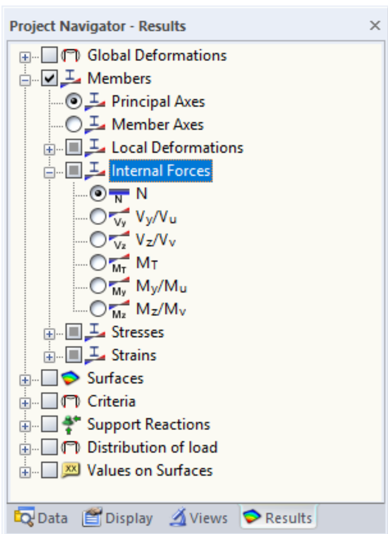


Figure 8.15 Displaying member axial forces *N* in the *Results* navigator

Special selection

To display only the axial forces of columns, we select all members in vertical position. To do so, we use the *Special Selection* dialog box

Edit → Select → Special

or the corresponding toolbar button.

We go to the *Members* category and select the *Parallel to member checkbox*.

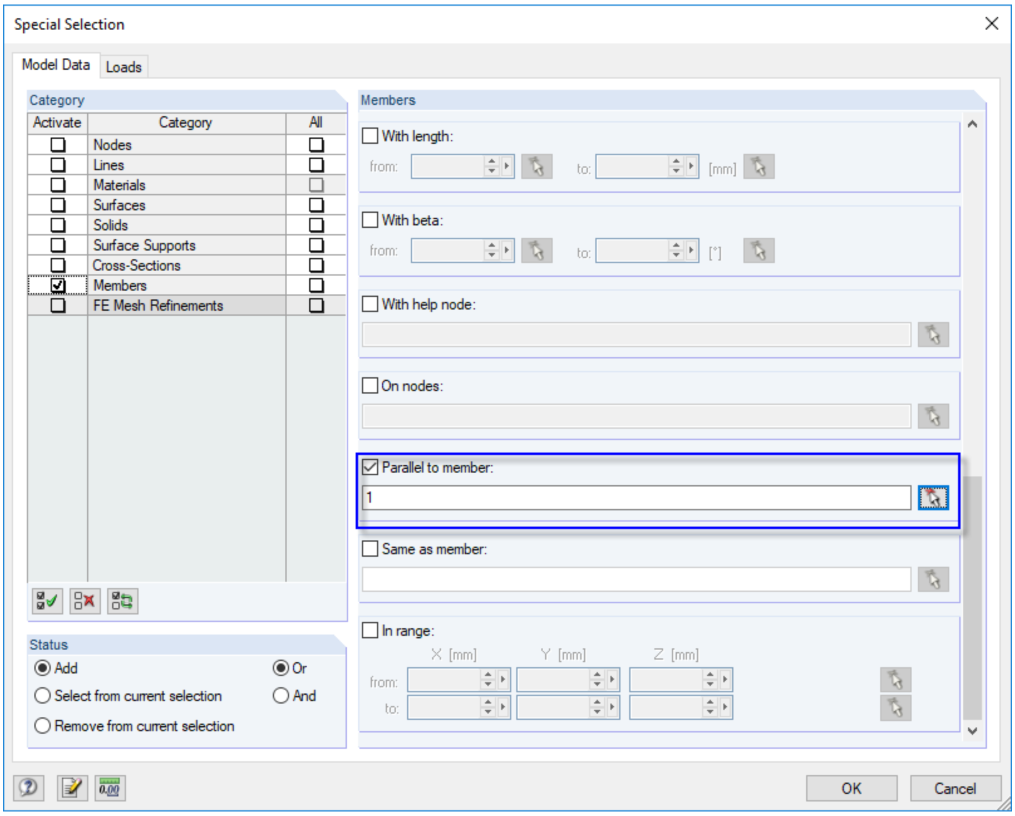


Figure 8.16 Selecting parallel members

Using , we select one of the column members in the work window. After closing the dialog boxes with [OK], all members in vertical position are selected.

Showing axial forces of columns

In the panel, we go to the filter tab and select *Members*.

We click , which enters the column members' numbers into the box above. The axial forces of the rib, horizontal beams, and purlin members disappear in the graphic.

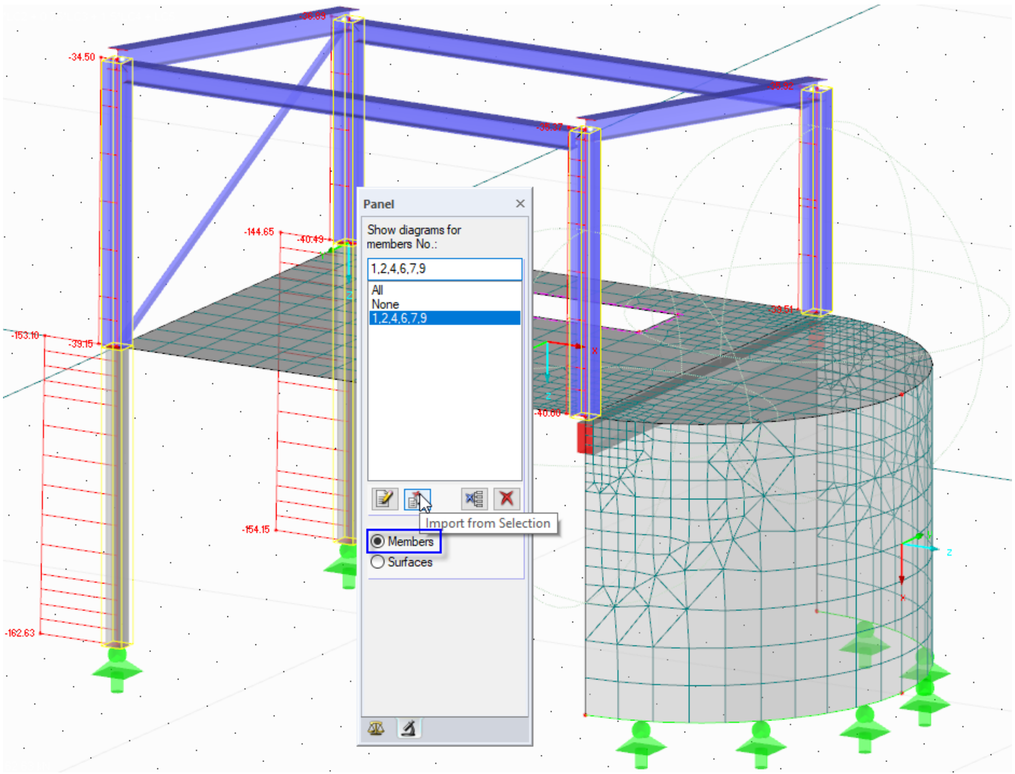


Figure 8.17 Axial forces of columns (CO13)

8.3.3 Range of Values

Using the panel's color scale tab, we can filter by result values.

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

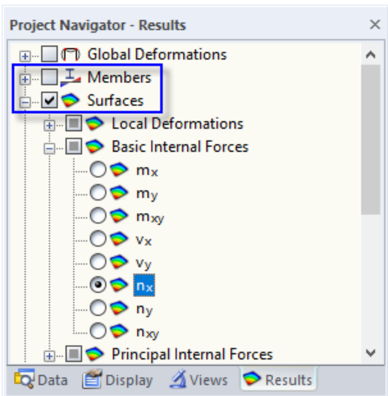


Figure 8.18 Displaying basic internal forces n_x

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. These are singularity effects.

To cut the peak values for the evaluation, we switch into the panel's color scale tab. We double-click the color scale, which opens the *Edit Isoband Value and Color Scales* dialog box.

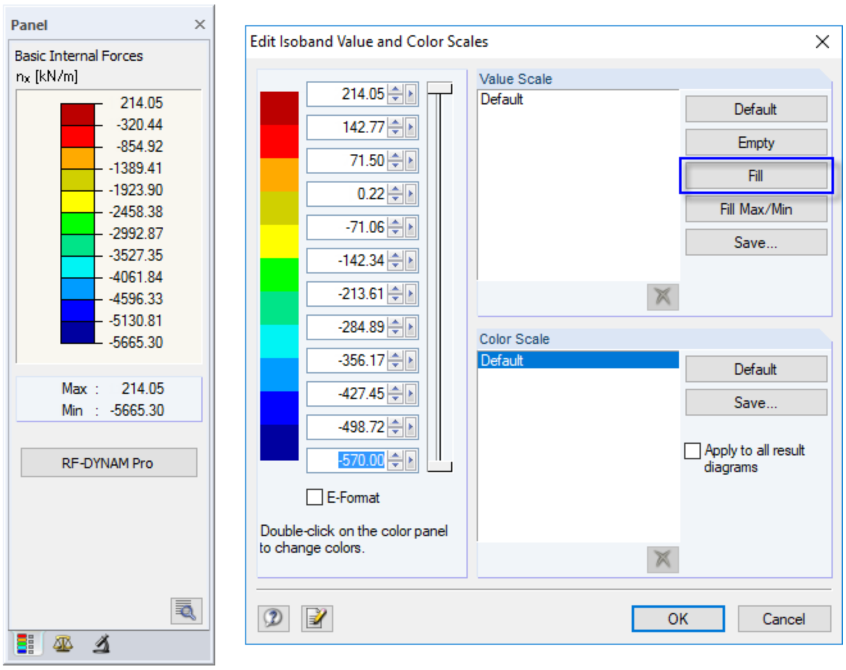


Figure 8.19 Changing extreme value for display and [Fill]ing the value scale

We reduce the compression forces' extreme value as shown in the figure above, for example **-570** kN/m instead of -5665.30 kN/m for CO13. After that we subdivide the spectrum into equal sections between the top and bottom limit values by clicking the [Fill] button.

After confirming with [OK], the force distribution is more differentiated. The zone where lifting forces occur is now clearly visible. Locations of singularities not covered by the color scale are represented without color.

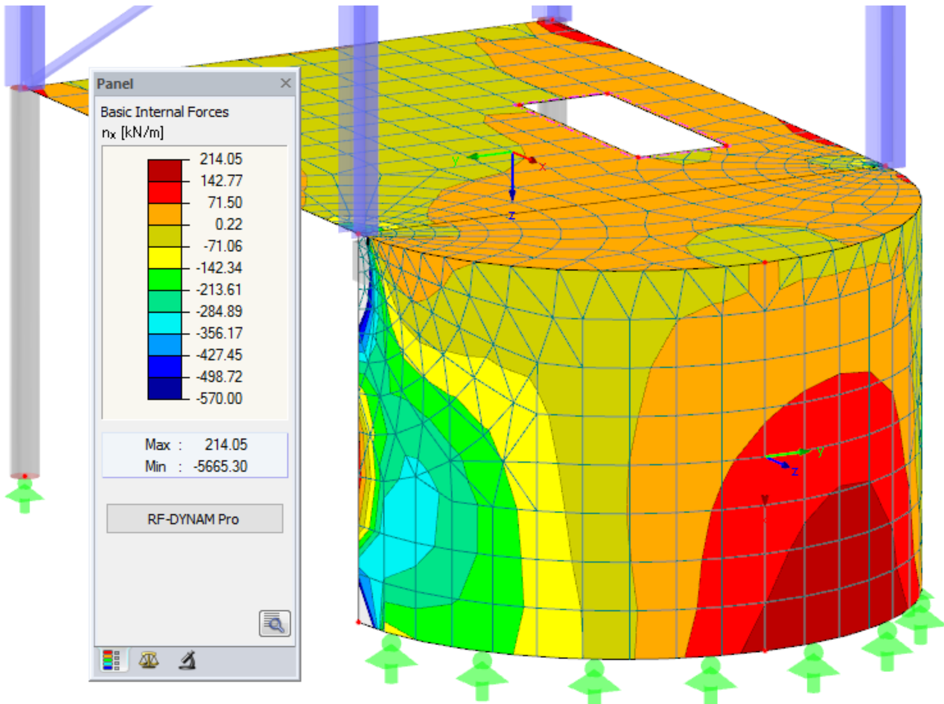


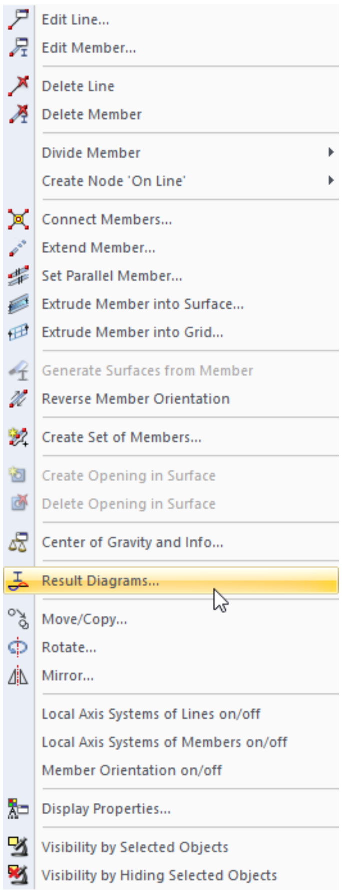
Figure 8.20 Distribution of wall axial forces with adjusted value scale (adapted peaks, compression and tension zone)

8.4 Displaying Result Diagrams

We can also evaluate the results in a diagram that is available for lines, members, line supports, and sections (see Chapter 8.5). We can use this feature to take a closer look at the result diagram of the downstand beam.

We hide the surface results. After that, we go to the panel's *Filter* tab and reselect the **All** option for member diagrams (see Figure 8.17).

We right-click member 3 and select *Result Diagrams* on the shortcut menu.



Member shortcut menu

A new window opens showing the rib member's result diagrams.

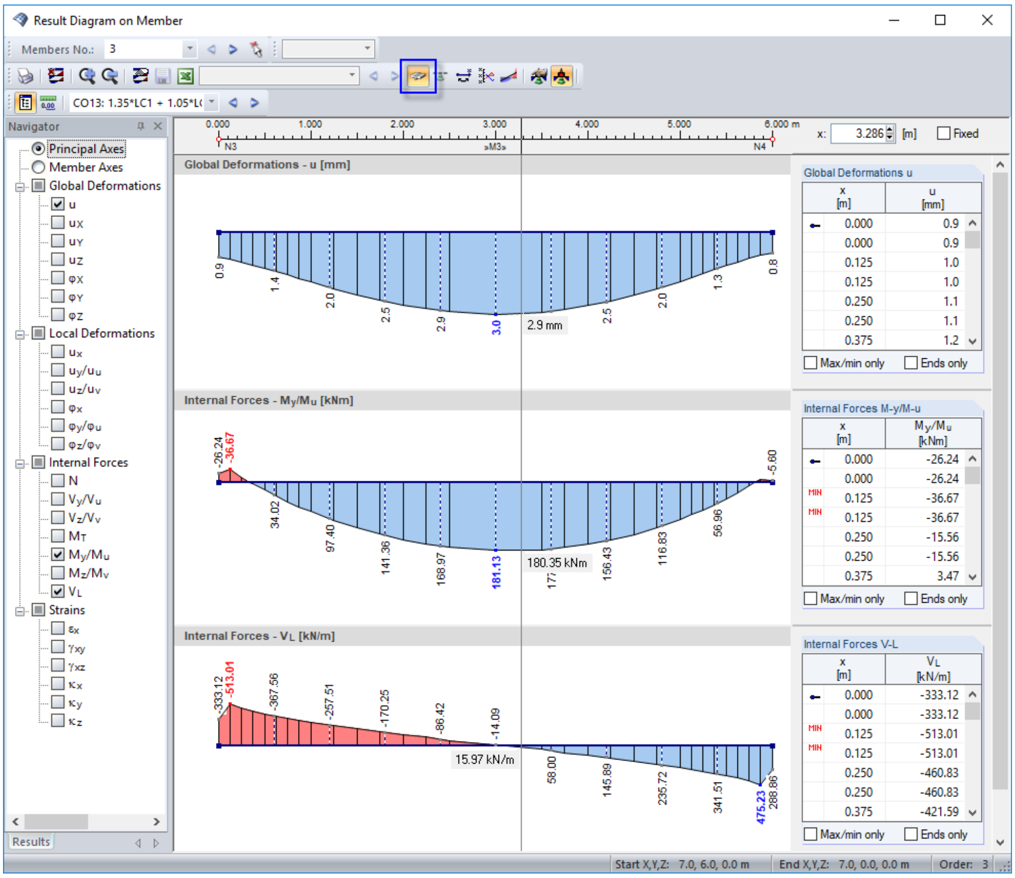


Figure 8.21 Showing result diagram of downstand beam

We select the global deformations *u* and the internal forces M_y and V_L in the navigator. The last option represents the longitudinal shear force between surface and member. These forces are displayed when the [Results with Ribs Component] button in the toolbar is active. When we click the button to turn it on and off, we can clearly see the difference between just the member internal forces and the rib internal forces with integration components from the surfaces.

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

The and load case selection buttons are also available in the result diagram window. We can also use the list to set the results of a load case.

We close the *Result Diagrams* feature by closing the window.

8.5 Creating a Section

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

We create a new section with

Insert → Section

or by using the section shortcut menu in the *Data* navigator.

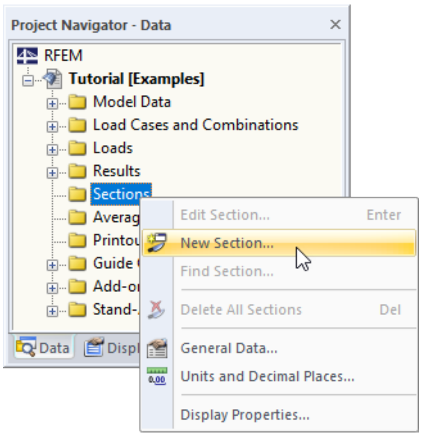
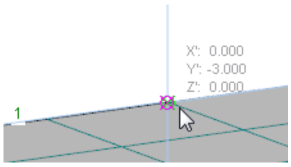


Figure 8.22 *Sections* shortcut menu in the *Data* navigator

In the *New Section* dialog box, we enter the *Section Name* **Center** because we want to define the section along the plate center.

We define the section's edge points graphically using : We first click the midpoint of line 1 (global coordinates: 0.000/3.000/0.000), then arc node 5.



Selecting the center of line 1

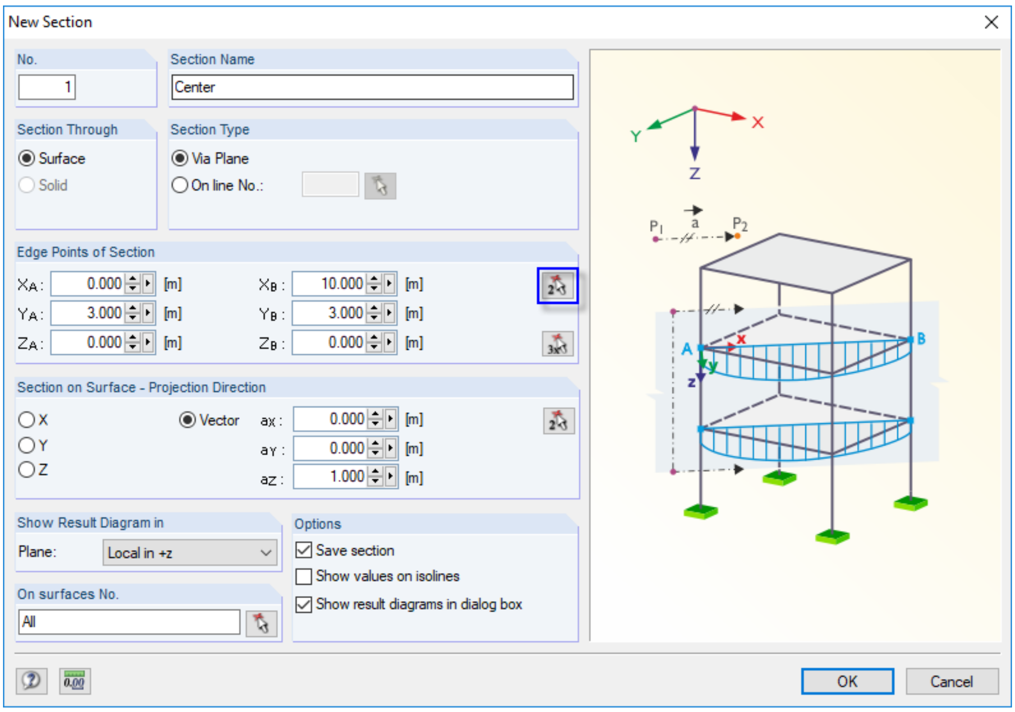


Figure 8.23 Defining the section

We leave the remaining default settings as they are and confirm the dialog box with [OK]

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

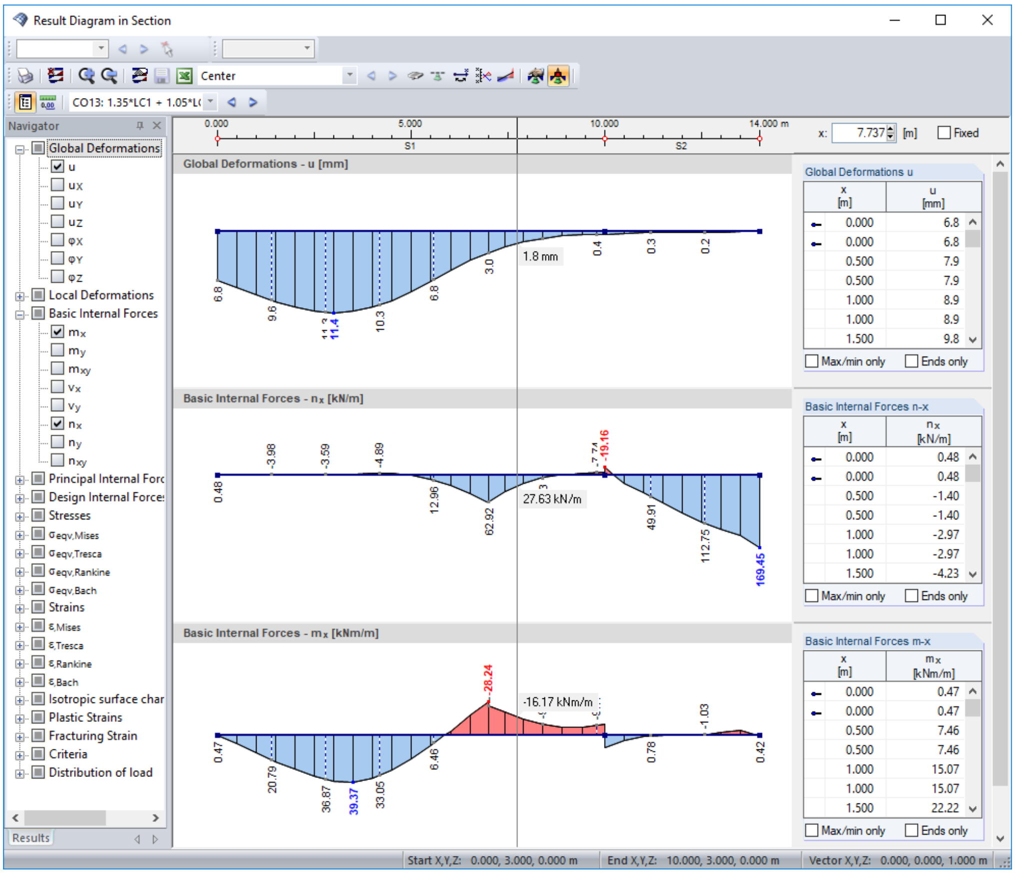


Figure 8.24 Section display in the *Result Diagram* window

After we [Close] the *Result Diagram* window, we can see that the section is displayed in the RFEM work window as well. In it, we specify the basic internal forces m_x.

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

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

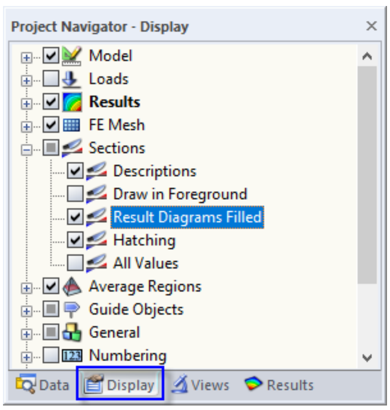


Figure 8.25 Display options for sections in the *Display* navigator

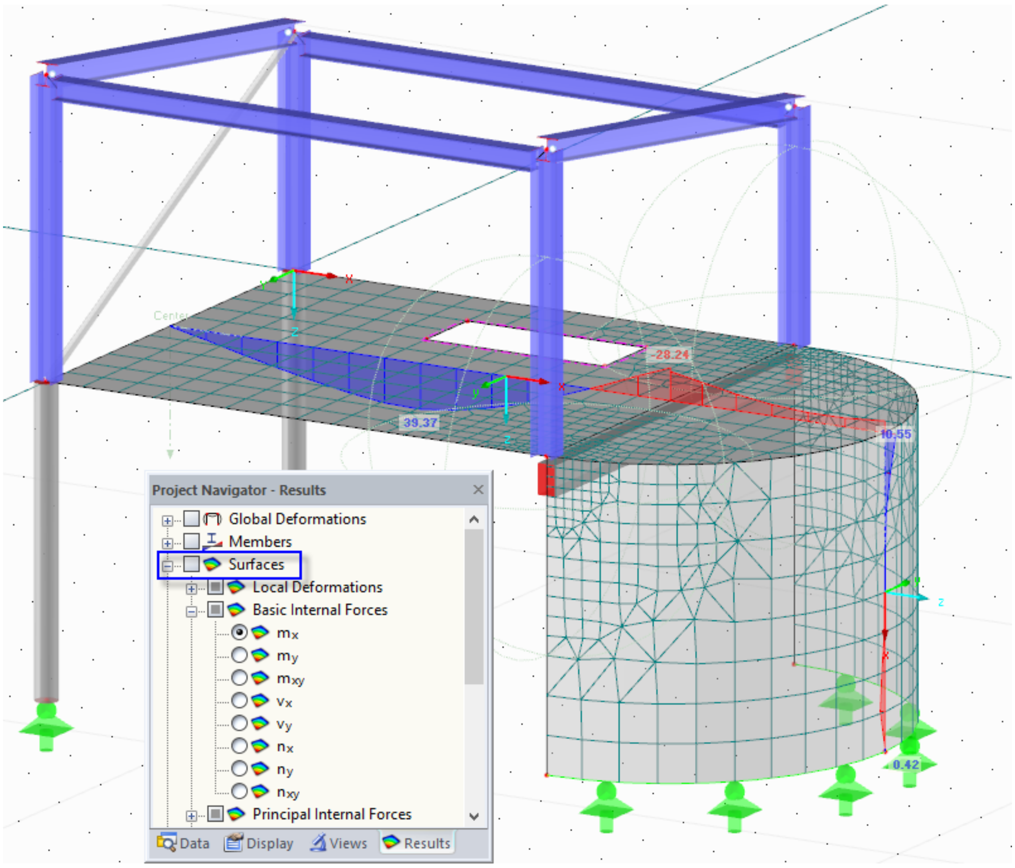


Figure 8.26 Showing the section in the model

9 Documentation

9.1 Creating a Printout Report

We do not recommend to print the complex results of an FE calculation directly. For that reason, RFEM first generates a print preview - the "printout report". It is used to determine the data we want to include in the printout. Moreover, it is possible to add graphics, descriptions and scans.

We open the printout report using

File → Open Printout Report

or the corresponding button. A dialog box appears where we can select a *Template* for the new printout report.

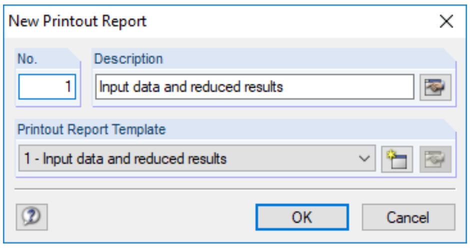


Figure 9.1 *New Printout Report* dialog box

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

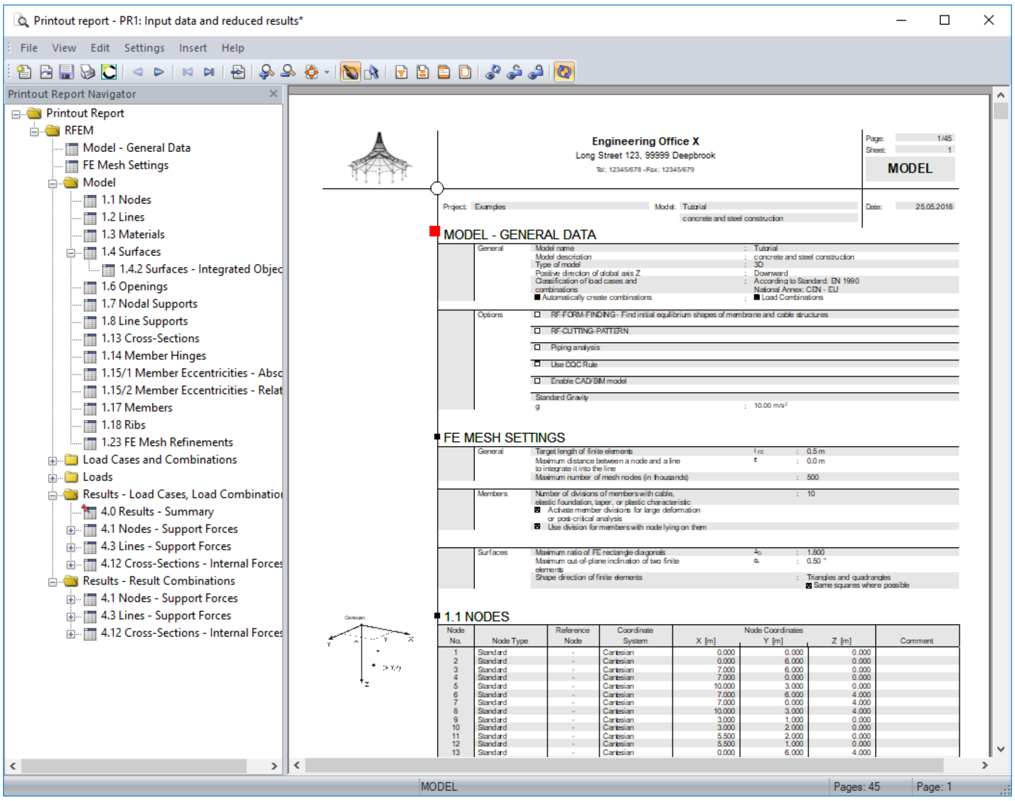


Figure 9.2 Print preview in the printout report

9.2 Adjusting the Printout Report

The printout report has a navigator similar to RFEM's that lists the selected chapters. When we click a navigator entry, the content of the corresponding chapter is displayed on the right.

The default content can be managed in detail. We adjust the output of the member internal forces: We right-click *Cross-Sections - Internal Forces* in *Results - Result Combinations* and click *Selection* on the shortcut menu.

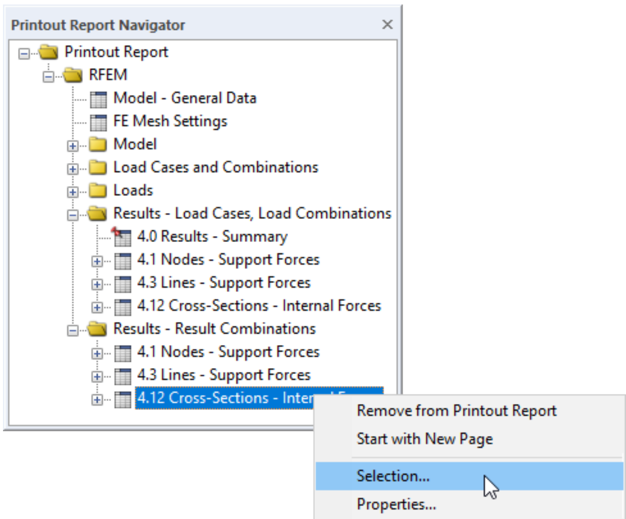


Figure 9.3 *Cross-Sections - Internal Forces* shortcut menu

A new dialog box appears, offering detailed selection options for RC results of members (see Figure 9.4).

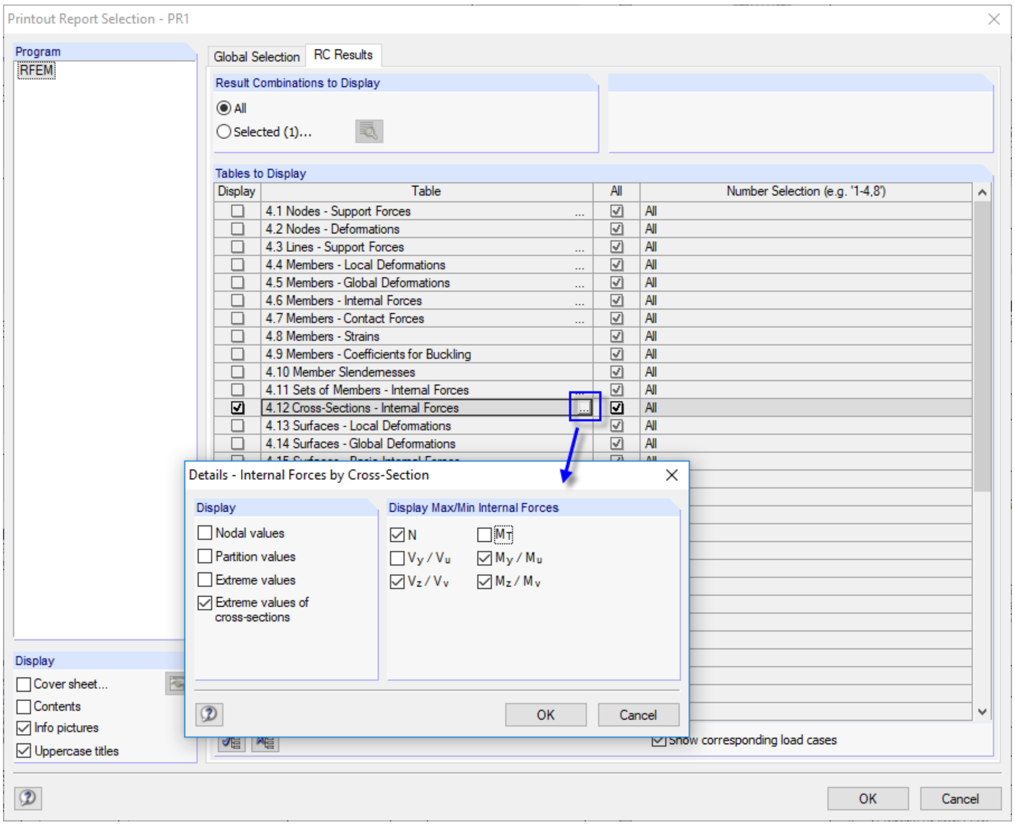
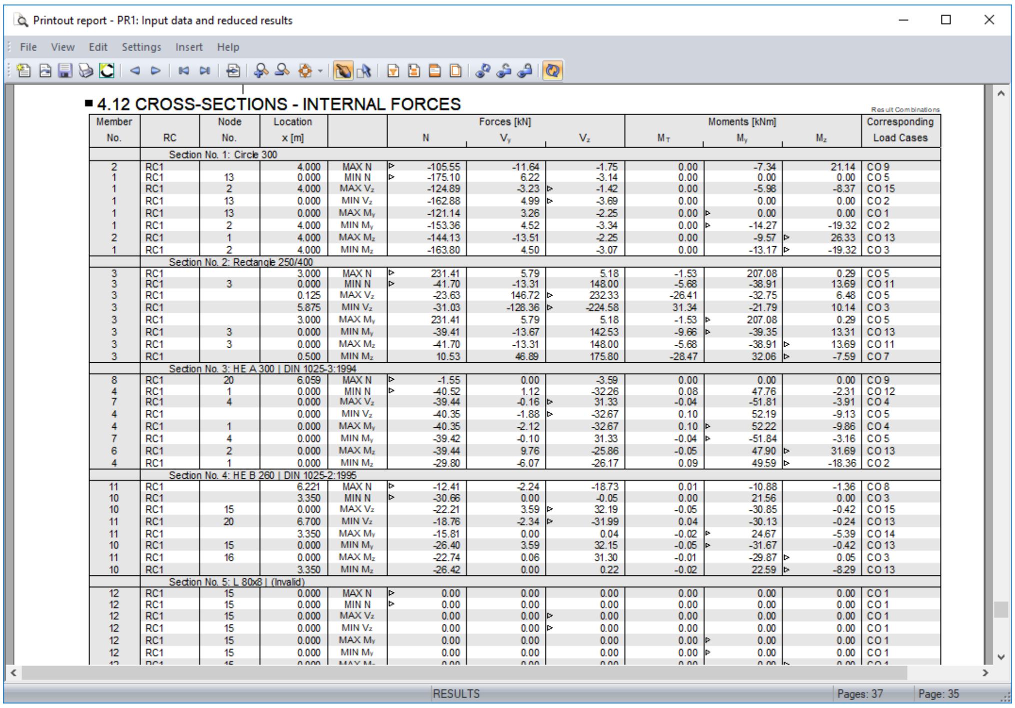


Figure 9.4 Reducing output of internal forces with *Printout Report Selection*

We click into the *Cross-Sections - Internal Forces* row. The [...] button appears, which we use to open the *Details - Internal Forces by Cross-Section* dialog box. We reduce the output to the **Extreme values of cross-sections N**, **V_z**, **M_y**, and **M_z**.

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



■ 4.12 CROSS-SECTIONS - INTERNAL FORCES

Result Combinations

Member No.	RC	Node No.	Location x [m]		Forces [kN] N	V_y	V_z	Moments [kNm] M_T	M_y	M_z	Corresponding Load Cases
	Section No. 1: Circle 300										
2	RC1		4.000	MAX N	-105.55	-11.64	-1.75	0.00	-7.34	21.14	CO 9
1	RC1	13	0.000	MIN N	-175.10	6.22	-3.14	0.00	0.00	0.00	CO 5
1	RC1	2	4.000	MAX V_z	-124.89	-3.23	-1.42	0.00	-5.98	-8.37	CO 15
1	RC1	13	0.000	MIN V_z	-162.88	4.99	-3.69	0.00	0.00	0.00	CO 2
1	RC1	13	0.000	MAX M_y	-121.14	3.26	-2.25	0.00	0.00	0.00	CO 1
1	RC1	2	4.000	MIN M_y	-153.36	4.52	-3.34	0.00	-14.27	-19.32	CO 2
2	RC1	1	4.000	MAX M_z	-144.13	-13.51	-2.25	0.00	-9.57	26.33	CO 13
1	RC1	2	4.000	MIN M_z	-163.80	4.50	-3.07	0.00	-13.17	-19.32	CO 3
	Section No. 2: Rectangle 250/400										
3	RC1		3.000	MAX N	231.41	5.79	5.18	-1.53	207.08	0.29	CO 5
3	RC1	3	0.000	MIN N	-41.70	-13.31	148.00	-5.68	-38.91	13.69	CO 11
3	RC1		0.125	MAX V_z	-23.63	146.72	232.33	-26.41	-32.75	6.48	CO 5
3	RC1		5.875	MIN V_z	-31.03	-128.36	-224.58	31.34	-21.79	10.14	CO 3
3	RC1		3.000	MAX M_y	231.41	5.79	5.18	-1.53	207.08	0.29	CO 5
3	RC1	3	0.000	MIN M_y	-39.41	-13.67	142.53	-9.66	-39.35	13.31	CO 13
3	RC1	3	0.000	MAX M_z	-41.70	-13.31	148.00	-5.68	-38.91	13.69	CO 11
3	RC1		0.500	MIN M_z	10.53	46.89	175.80	-28.47	32.06	-7.59	CO 7
	Section No. 3: HE A 300 | DIN 1025-3:1994										
8	RC1	20	6.059	MAX N	-1.55	0.00	-3.59	0.00	0.00	0.00	CO 9
4	RC1	1	0.000	MIN N	-40.52	1.12	-32.26	0.08	47.76	-2.31	CO 12
7	RC1	4	0.000	MAX V_z	-39.44	-0.16	31.33	-0.04	-51.81	-3.91	CO 4
4	RC1		0.000	MIN V_z	-40.35	-1.88	-32.67	0.10	52.19	-9.13	CO 5
4	RC1	1	0.000	MAX M_y	-40.35	-2.12	-32.67	0.10	52.22	-9.86	CO 4
7	RC1	4	0.000	MIN M_y	-39.42	-0.10	31.33	-0.04	-51.84	-3.16	CO 5
6	RC1	2	0.000	MAX M_z	-39.44	9.76	-25.86	-0.05	47.90	31.69	CO 13
4	RC1	1	0.000	MIN M_z	-29.80	-6.07	-26.17	0.09	49.59	-18.36	CO 2
	Section No. 4: HE B 260 | DIN 1025-2:1995										
11	RC1		6.221	MAX N	-12.41	-2.24	-18.73	0.01	-10.88	-1.36	CO 8
10	RC1		3.350	MIN N	-30.66	0.00	-0.05	0.00	21.56	0.00	CO 3
10	RC1	15	0.000	MAX V_z	-22.21	3.59	32.19	-0.05	-30.85	-0.42	CO 15
11	RC1	20	6.700	MIN V_z	-18.76	-2.34	-31.99	0.04	-30.13	-0.24	CO 13
11	RC1		3.350	MAX M_y	-15.81	0.00	0.04	-0.02	24.67	-5.39	CO 14
10	RC1	15	0.000	MIN M_y	-26.40	3.59	32.15	-0.05	-31.67	-0.42	CO 13
11	RC1	16	0.000	MAX M_z	-22.74	0.06	31.30	-0.01	-29.87	0.05	CO 3
10	RC1		3.350	MIN M_z	-26.42	0.00	0.22	-0.02	22.59	-8.29	CO 13
	Section No. 5: L 80x8 | (Invalid)										
12	RC1	15	0.000	MAX N	0.00	0.00	0.00	0.00	0.00	0.00	CO 1
12	RC1	15	0.000	MIN N	0.00	0.00	0.00	0.00	0.00	0.00	CO 1
12	RC1	15	0.000	MAX V_z	0.00	0.00	0.00	0.00	0.00	0.00	CO 1
12	RC1	15	0.000	MIN V_z	0.00	0.00	0.00	0.00	0.00	0.00	CO 1
12	RC1	15	0.000	MAX M_y	0.00	0.00	0.00	0.00	0.00	0.00	CO 1
12	RC1	15	0.000	MIN M_y	0.00	0.00	0.00	0.00	0.00	0.00	CO 1

Figure 9.5 Extreme values of cross-section internal forces N, V_z, M_y, and M_z for RC1 in the printout report

In the same way, we can adjust any other chapter for the printout.

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

9.3 Printing Graphics in the Printout Report

Generally, graphics that illustrate the documentation are included in the printout.

Printing deformation graphics

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

In the work window, we set the *Deformations* to be of **RC1 - ULS (STR/GEO)** and position the graphic accordingly. We hide the *Sections* in the *Results* navigator.

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

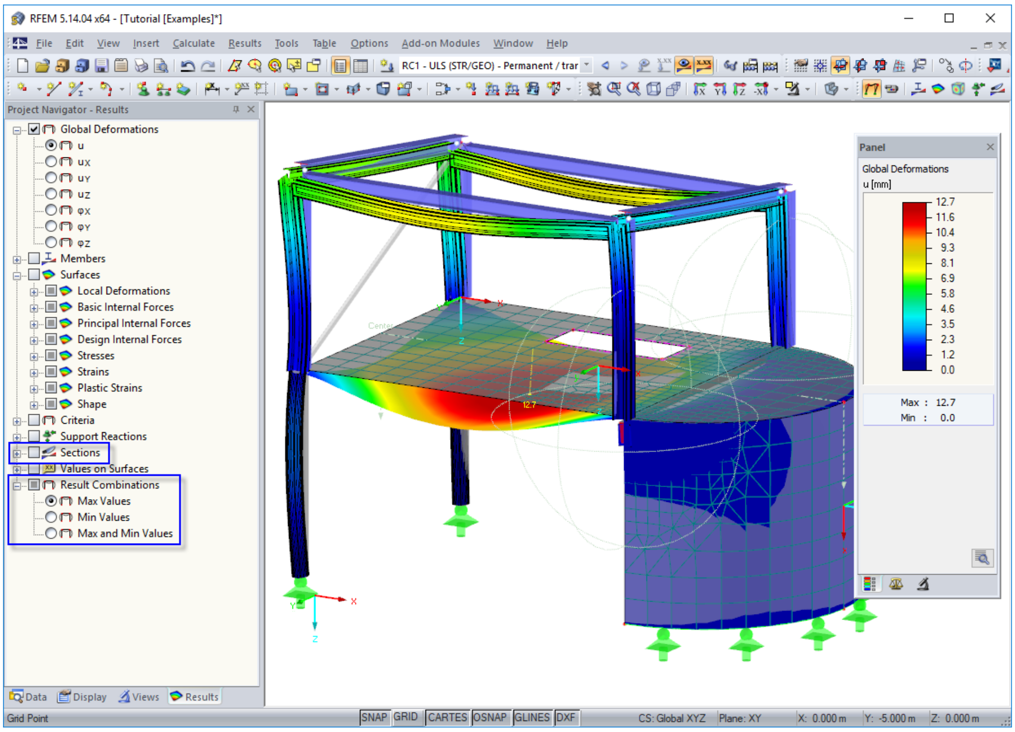


Figure 9.6 Displaying the maximum deformation results of RC1

We now transfer the graphic to the printout report using

File → Print Graphic

or the corresponding toolbar button.

The *Graphic Printout* dialog box appears (see Figure 9.7).

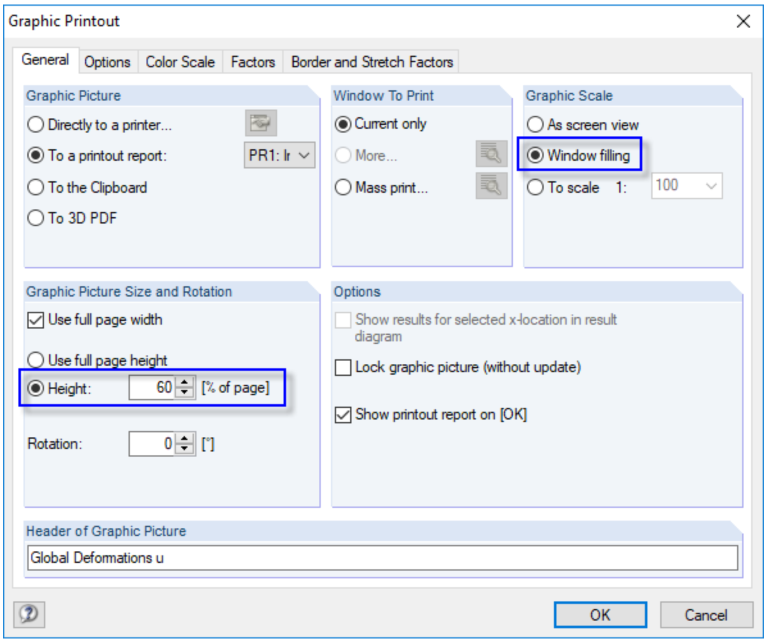


Figure 9.7 *Graphic Printout* dialog box

We set up the print parameters as shown in Figure 9.7. We do not need to change the default settings in the rest of the tabs.

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

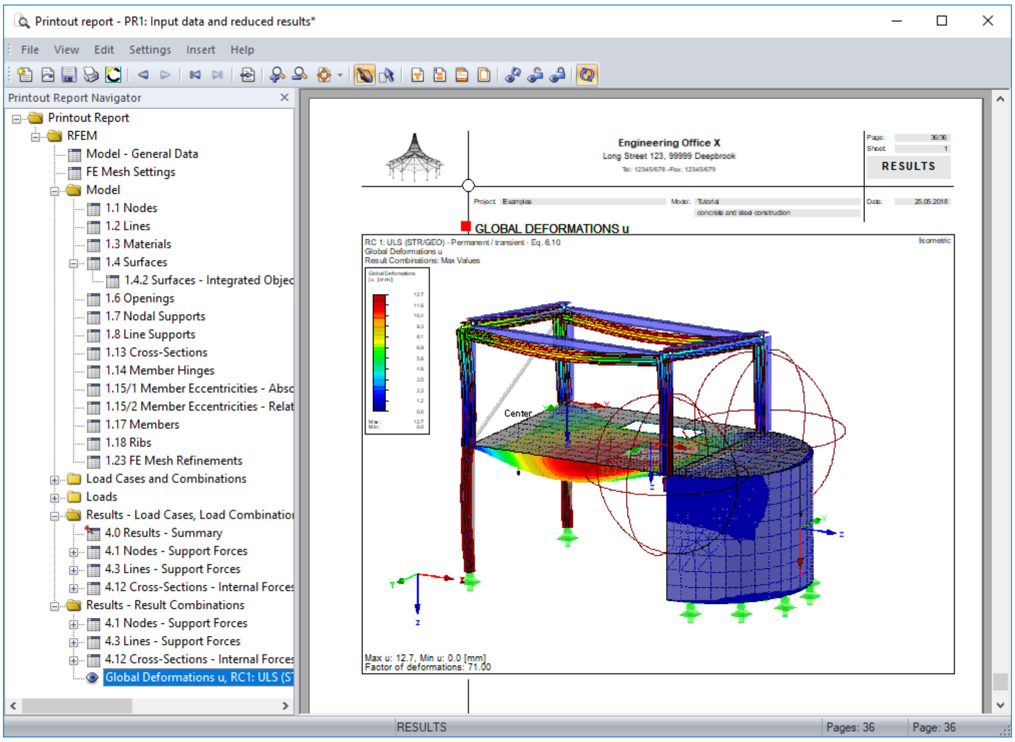


Figure 9.8 Deformation graphic in the printout report

Printing result diagrams

Finally, we want to document the distribution of internal forces in a steel purlin. Again, we close the printout report with [X].

In the RFEM work window, we right-click member *11* (purlin on high eave). In the member shortcut menu (see Chapter 8.4), we select the *Result Diagrams* option to access the result diagram.

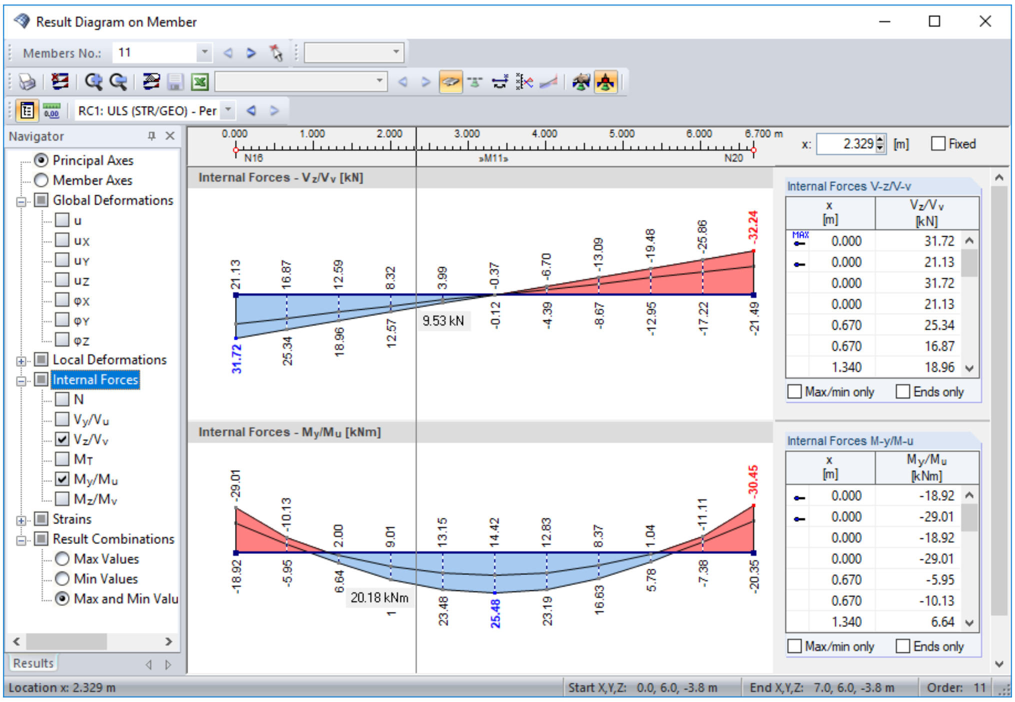


Figure 9.9 Shear force and moment diagram of purlin

The window shows the result diagrams of *RC1*. For the printout, we select only the internal forces $\mathbf{V_z/V_v}$ and $\mathbf{M_y/M_u}$. The result diagram shows the *Max and Min Values*.

Using the print button, we open the *Graphic Printout* dialog box. We can keep the default settings in the *General* tab. In the *Options* tab we change a few things.

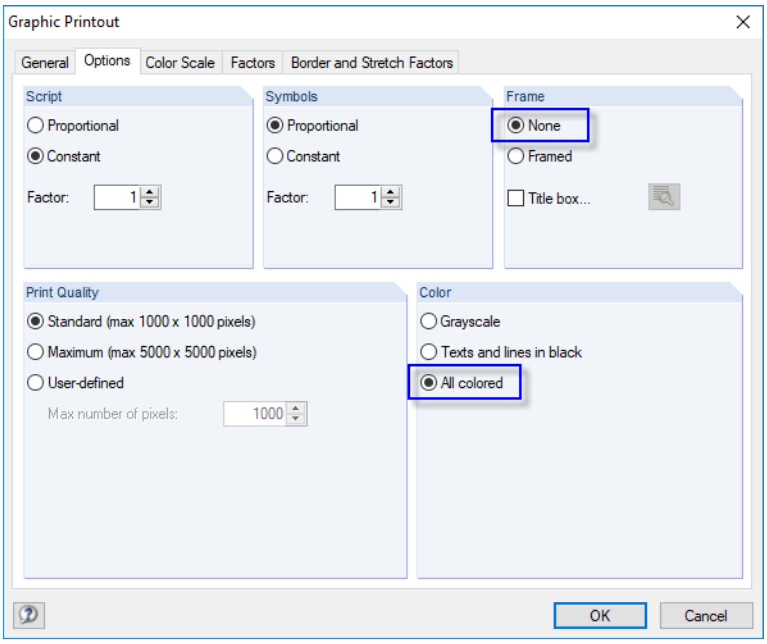


Figure 9.10 *Graphic Printout dialog box, Options ab*

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

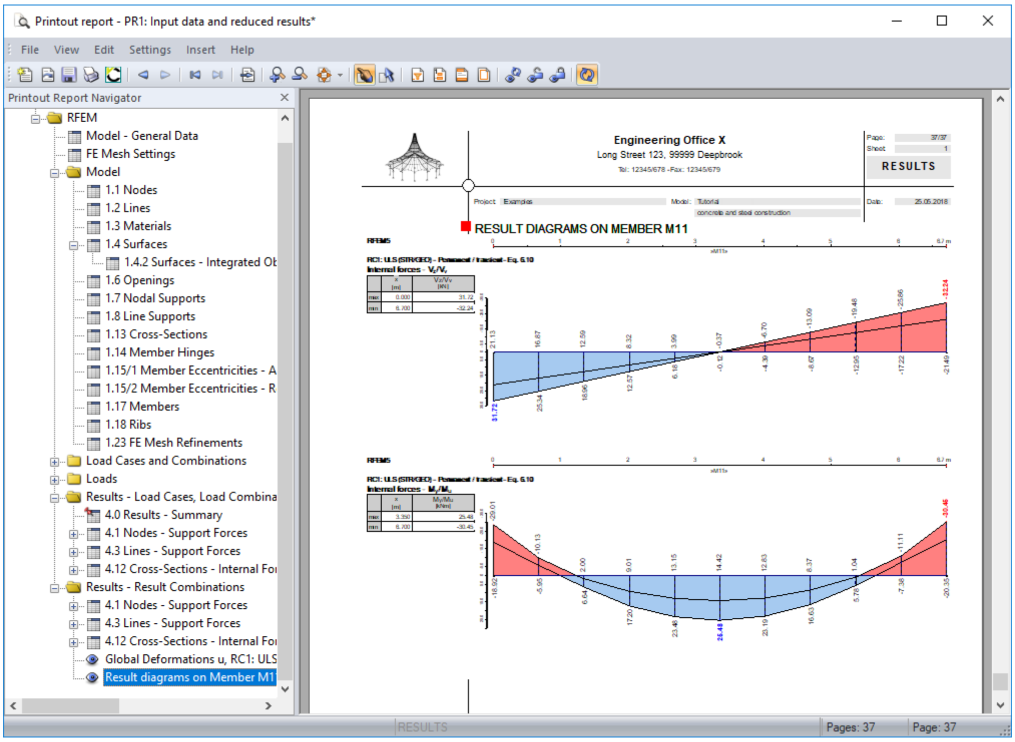


Figure 9.11 Diagrams of internal forces of purlin in the printout report

Changing the language of the printout report

The printout report's language in independent of that of the RFEM user interface. Thus, we are able to create a German printout report in the English version, for example. For that, we use the printout report menu

Settings → Language.

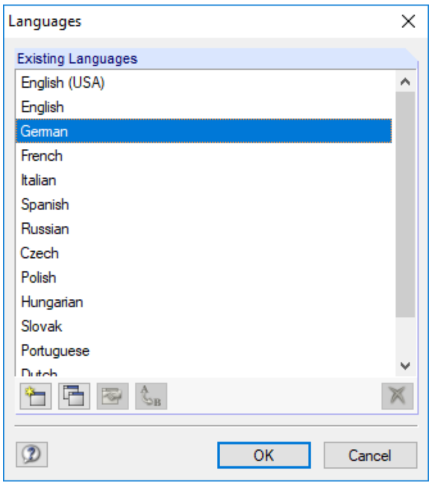


Figure 9.12 Switching languages in the printout report

In the *Languages* dialog box, we select **German** (or a different language) as the new language. We can check the changes in the print preview after clicking [OK].

Custom entries such as load case descriptions or comments are not 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 integrated PDF printer also allows the report data to be exported as a PDF file. We use this feature through the printout report menu

File → Export to PDF.

The Windows dialog box *Save As* opens where we specify the location and the file name.

Save

The [Save] button creates a PDF file with bookmarks to simplify navigation in the digital document.

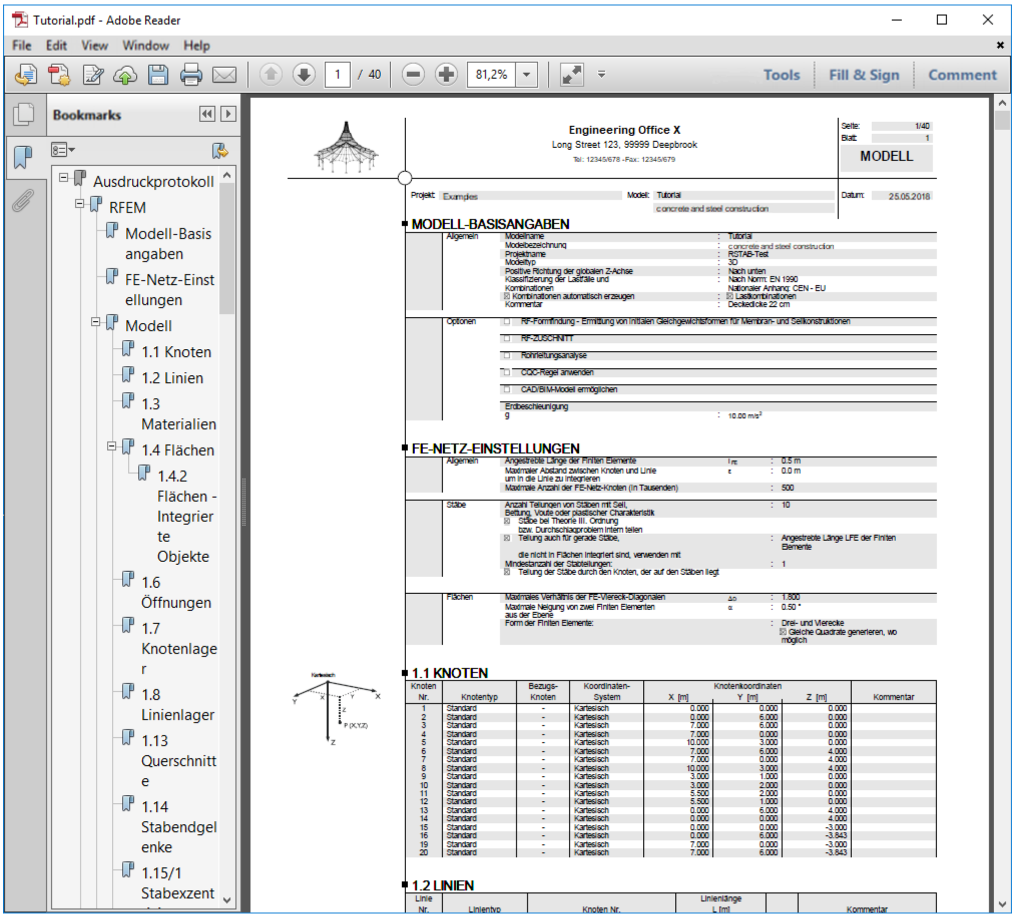


Figure 9.13 German printout report as PDF file with bookmarks

10 Outlook

We have now reached the end of our example. We hope that this tutorial helps you get started with RFEM and makes you curious about the rest of the program's features. You can find a detailed program description in the RFEM manual, which you can download on our website at https://www.dlubal.com/en-US/downloads-and-information/documents/manuals.

You can access the program's online help with the **Help** menu or using [F1] and search for certain terminology, much like in the manual. The online help is based on the RFEM manual.

Finally, if you have any questions, you are welcome to use our free e-mail hotline. You can also browse through the FAQ or Knowledge Base pages on our website.

You may also use this example in the add-on modules for steel and reinforced concrete design (for example RF-STEEL Members, RF-CONCRETE Surfaces/Members, or RF-STABILITY). This will provide an insight into the functionality of the design modules. The designs may also be evaluated in the RFEM work window.