Version
October 2015

Program

RFEM 5

Spatial Models Calculated acc. to Finite Element Method

Introductory Example

Am Zellweg 2 D-93464 Tiefenbach

Tel.: +49 (0) 9673 9203-0
Fax: +49 (0) 9673 9203-51
E-mail: info@dlubal.com
Web: www.dlubal.com

Contents

1. Introduction

With the present introductory example we would like to make you acquainted with the most important features of RFEM. Often you have several options to achieve your targets. Depending on the situation and your preferences you can play with the software to learn more about the program's possibilities. With this simple example we want to encourage you to find out useful functions in RFEM.

We will model a floor slab supported by columns including two downstand beams. Then, we will design the structure according to linear-static and second-order analysis with regard to the following load cases: self-weight with finishes, imposed load and imperfection. With the features presented we want to show you how you can define model and load objects in various ways.

With the 30-day trial version, you can work on the model without any restriction. After that period, the demo mode will be applied. You can still enter the example and calculate it; saving data will not be possible, however.

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

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

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

The file **RFEM-Example-06.rf5** containing the model data of the following example can be found in the *Examples* project that has been created automatically during the installation. However, for the first steps with RFEM we recommend entering the model manually. If you have no time for it, you can also watch the videos on our website at www.dlubal.com/Videos-from-category-Videos-for-RFEM.aspx.

2. System and Loads

2.1 Sketch of System

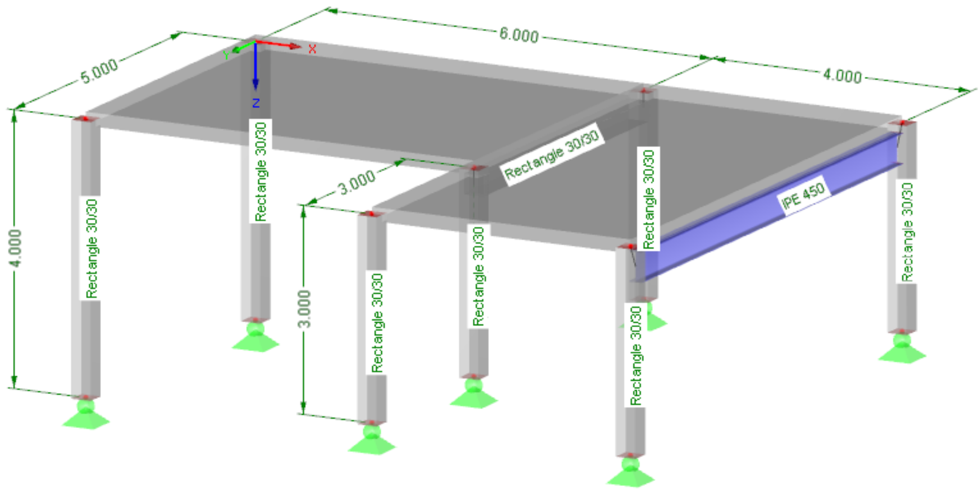


Figure 2.1: Structural system

The reinforced concrete floor consists of two continuous floor slabs with a downstand beam made of reinforced concrete and another one made of steel. The construction is supported by columns which are bending-resistant and integrated into the plate.

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

2.2 Materials, Thicknesses and Cross-Sections

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

The floor thickness is 20 cm. The concrete columns and the downstand beam consist of square cross-sections with a lateral lengths of 30 cm. For the steel beam we use an IPE 450 section.

2.3 Load

Load case 1: self-weight and finishes (permanent load)

As loads, the self-weight of the model including its floor structure of 0.75 kN/m² is applied. We do not need to determine the self-weight manually. RFEM calculates the weight automatically from the defined materials, surface thicknesses and cross-sections.

Load case 2: imposed load, area 1

The floor surface represents a domestic area of category A2 with an imposed load of 1.5 kN/m². The load is applied in two different load cases to cover the effects of continuity.

Load case 3: imposed load, area 2

The imposed load of 1.5 kN/m² is also applied to the second area. In addition, a vertically acting linear load of 5.0 kN/m is taken into account on the edge of the floor, representing a loading due to a balcony construction.

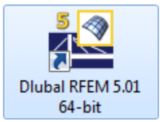
Load case 4: imperfections

Often imperfections must be considered, for example according to Eurocode 2. Inclinations and precambers are managed in a separate load case. So it is possible to assign specific partial safety factors when you combine this type of load with other actions.

The inclination is simplified for all columns by assuming $\varphi_0 = 1/200$ against direction Y. Precambers do not need to be considered according to Eurocode 2.

3. Creation of Model

3.1 Starting RFEM



To start RFEM in the taskbar, we

click **Start**, point to **All Programs** and **Dlubal**, and then we select **Dlubal RFEM 5.xx**

or we double-click the icon **Dlubal RFEM 5.xx** on the computer desktop.

3.2 Creating the Model

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

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

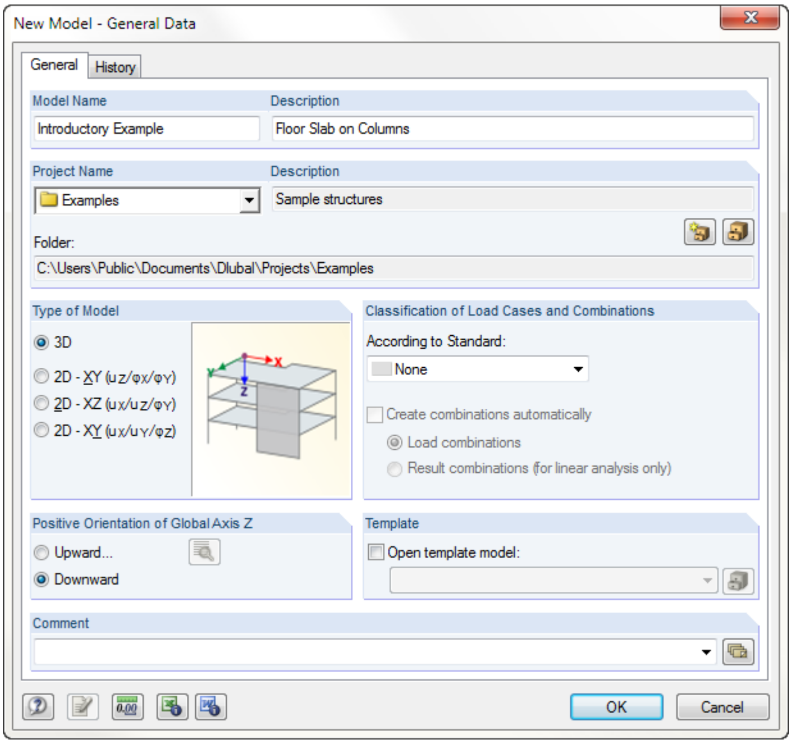


Figure 3.1: Dialog box *New Model - General Data*

We write **Introductory Example** in the *Model Name* box. To the right, we enter **Floor Slab on Columns** in the *Description* box. We always have to define a *Model Name* because it determines the name of the RFEM file. The *Description* box does not necessarily need to be filled in.

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

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

We check if the Standard option **None** is selected in the section *Classification of Load Cases and Combinations*. If not, we select this entry from the list.

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

4. Model Data

4.1 Adjusting Work Window and Grid

The empty work window of RFEM is displayed.

View

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

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

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

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

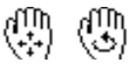
To exit the function, different ways are possible:

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

Mouse functions

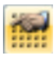
The mouse functions follow the general standards for Windows applications. To select an object for further editing, we click it once with the **left** mouse button. We double-click the object when we want to open its dialog box for editing.

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



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

Grid



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

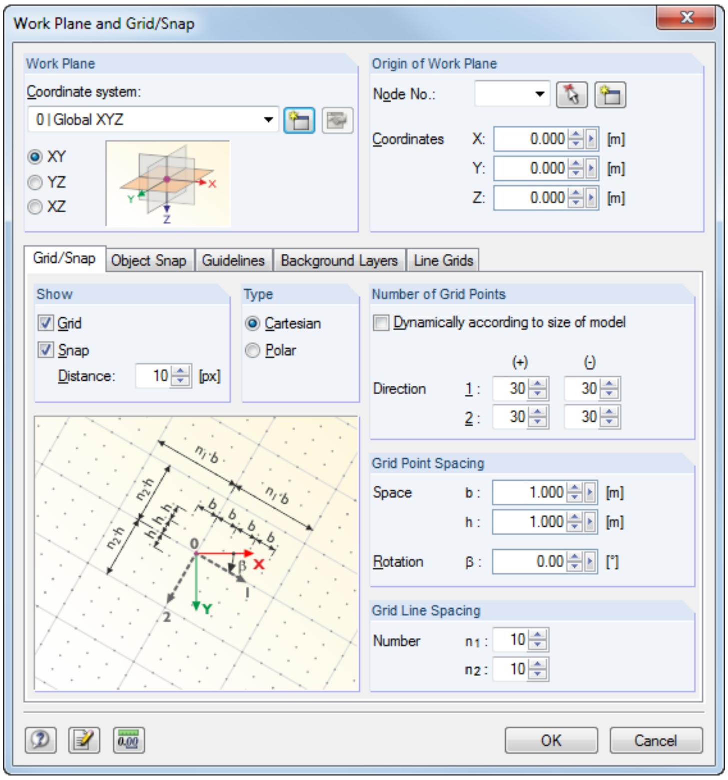


Figure 4.1: Dialog box *Work Plane and Grid/Snap*

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

Work plane

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

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

4.2 Creating Surfaces

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

We can define the ceiling as a continuous surface by means of outlines. But it is also possible to represent the floor by two rectangular surfaces which are rigidly connected in a common line. The second way of modeling makes it easier to apply loads to two areas.

Before we start creating the surfaces, we activate two useful functions. For this, we use the general *shortcut menu*. We right-click in an empty space of the work window to activate it.

Show Numbering

You can activate and deactivate functions by clicking within the shortcut menu. Active functions are marked by buttons highlighted in yellow. We activate the entry *Show Numbering*.

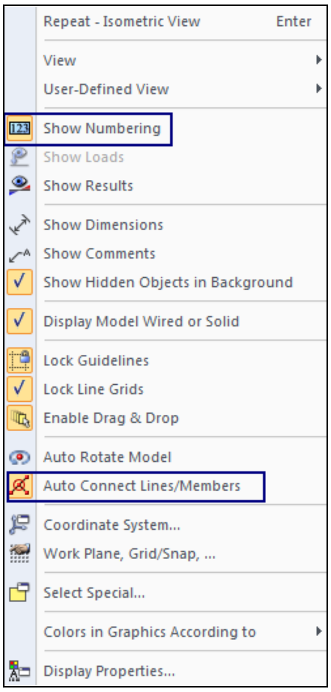
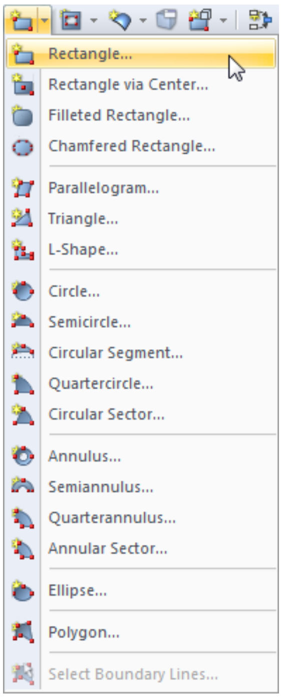


Figure 4.2: Show numbering in shortcut menu

Auto Connect Lines/Members

If the function *Auto Connect Lines/Members* is not active, we also activate it (please right-click again for the shortcut menu). It makes it easier to create the surfaces.

4.2.1 First Rectangular Surface



List button for plane surfaces

To create rectangular plates quickly,

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

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

With the command [Rectangular] we can define the plate directly. The related nodes and lines will be created automatically.

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

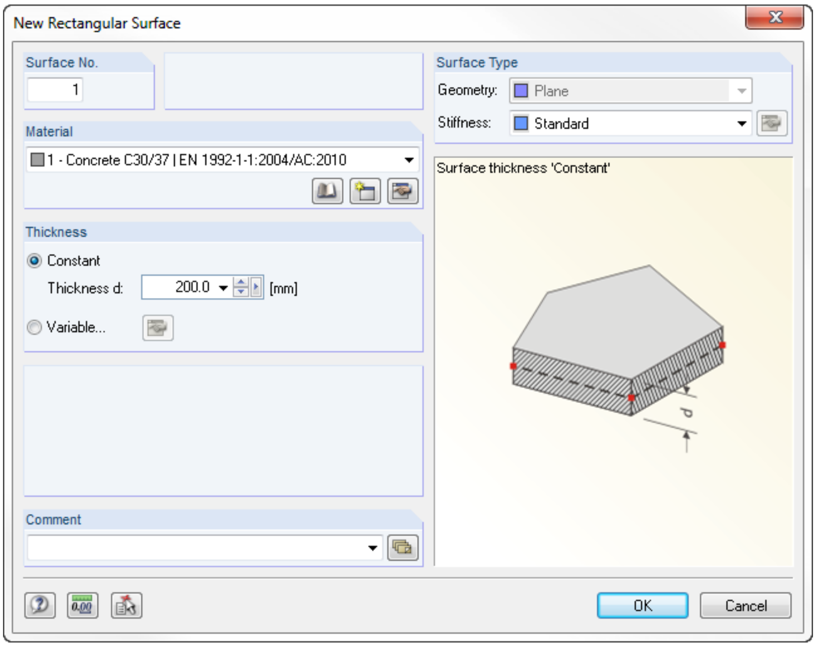


Figure 4.3: Dialog box *New Rectangular Surface*

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

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

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

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

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

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

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

Then, we define the opposite corner of the slab by clicking the grid point with the X/Y/Z coordinates **6.000/5.000/0.000**.

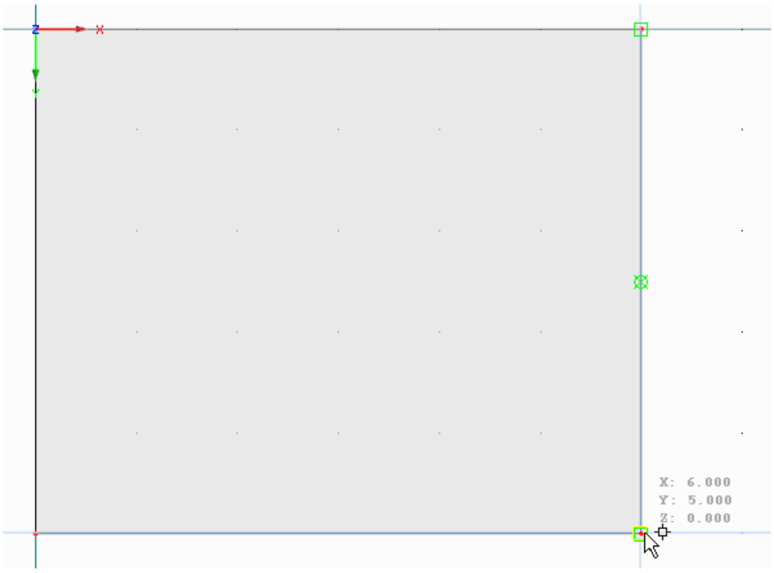


Figure 4.4: Rectangular surface 1

RFEM creates four nodes, four lines and one surface.

4.2.2 Second Rectangular Surface

As the function is still active, we can define the next surface immediately.

We click node **4** with the coordinates **6.000/0.000/0.000**, and then we select the grid point with the coordinates **10.000/8.000/0.000**.

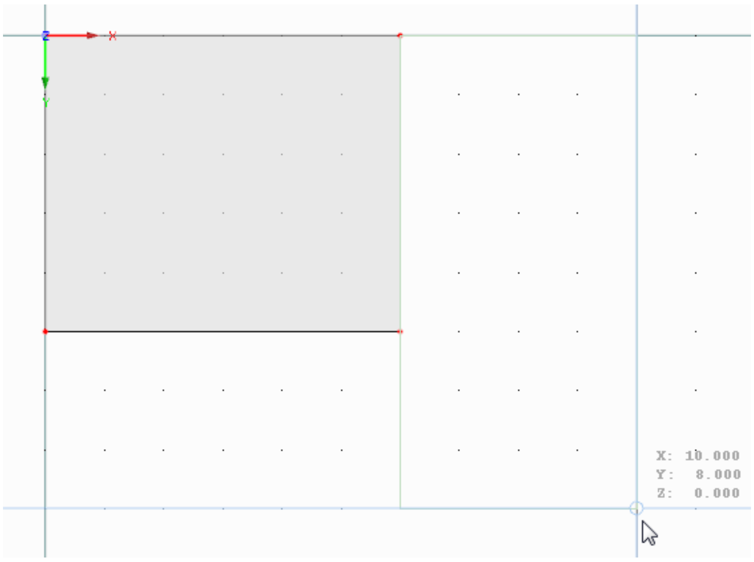


Figure 4.5: Rectangular surface 2

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

4.3 Creating Members

4.3.1 Downstand Beams

We specify member properties for the lines 3 and 7 to define two downstand beams.

4.3.1.1 Steel Girder

We open the dialog box *Edit Line* by double-clicking line 7.

We switch to the second tab *Member* where we select the check box for the option *Available*. The dialog box *New Member* appears.

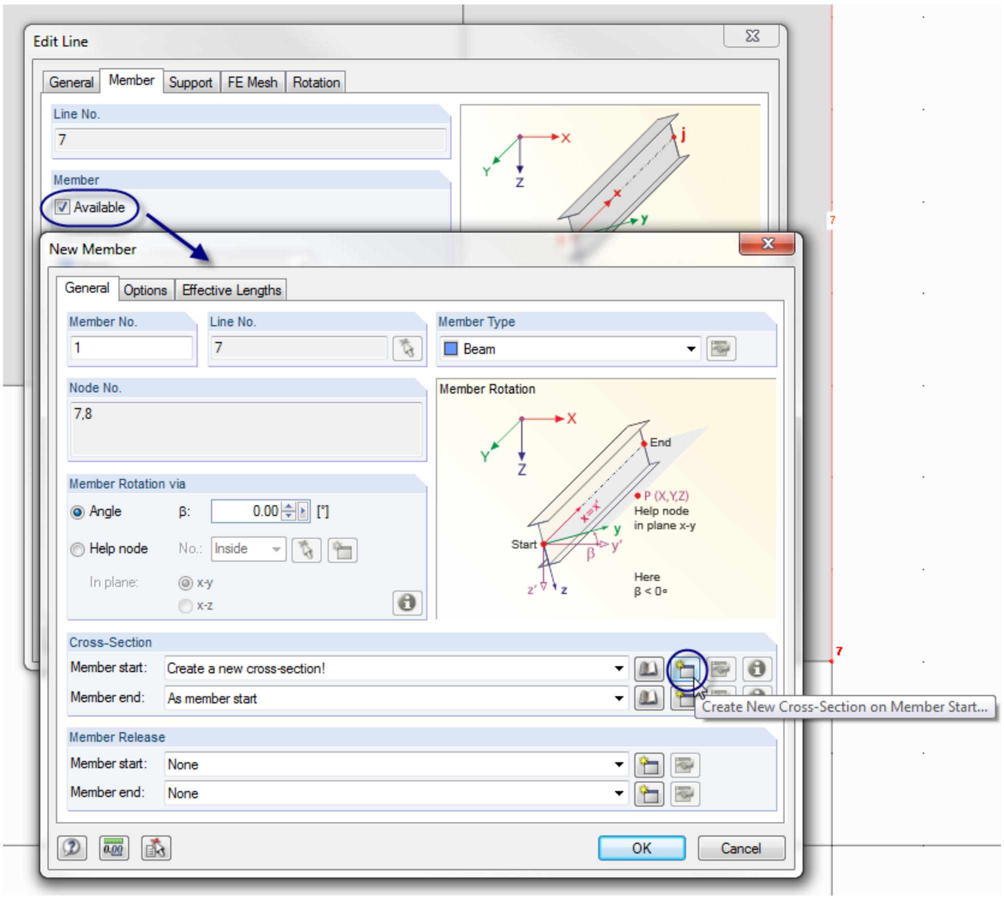


Figure 4.6: Dialog box *New Member*

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

The *New Cross-Section* dialog box appears. We click the [IPE] button in the upper part of the dialog box. The *Rolled Cross-Sections - I-Sections* dialog box opens where we can select the section **IPE 450** from the IPE cross-section table (see Figure 4.7).

For rolled cross-sections RFEM automatically sets the *Material* number *2 - Steel 235*.

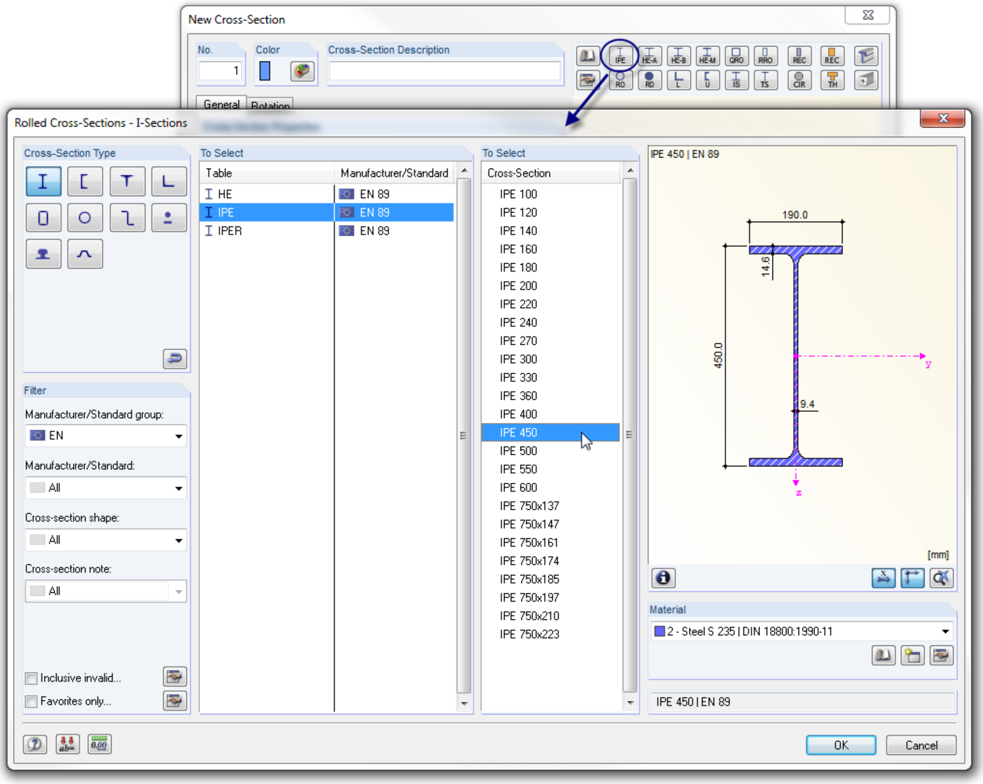


Figure 4.7: Selecting the cross-section IPE 450

We click [OK] to import the cross-section values to the *New Cross-Section* dialog box.

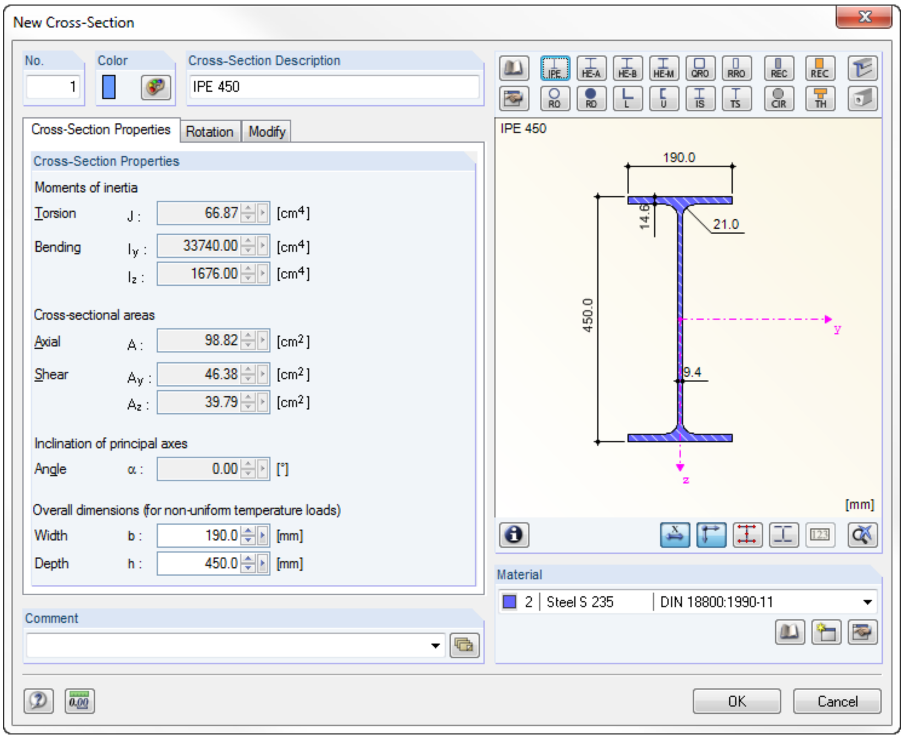


Figure 4.8: Dialog box *New Cross-Section* with cross-section properties

We click [OK] and return to the initial *New Member* dialog box. Now the *Member start* box shows the new cross-section. We close the dialog box with [OK]. We also close the *Edit Line* dialog box with the [OK] button. The steel girder is now displayed on the edge of the floor.

4.3.1.2 T-Beams

We define the downstand beam below the ceiling in the same way: We double-click line *3* to open the *Edit Line* dialog box. In the *Member* tab, we select the option *Available* (see Figure 4.6).

Definition of cross-section

The *New Member* dialog box opens. To define the cross-section at the *Member start*, we click the [New] button again (see Figure 4.6).

In the upper part of the *New Cross-Section* dialog box, we select the massive **REC** cross-section table. The *Solid Cross-Sections - Rectangle* dialog box opens where we define the width *b* and the depth *h* as **300** mm.

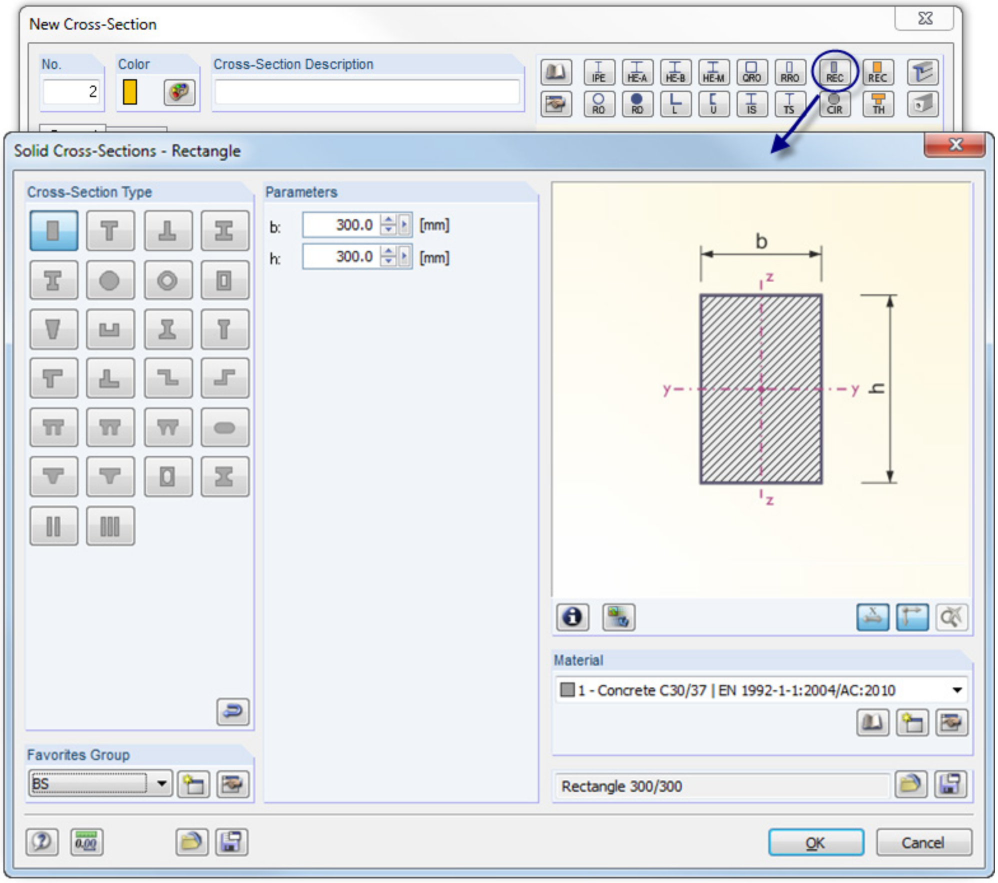


Figure 4.9: Dialog box *Solid Cross-Sections - Rectangle*

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

For solid cross-sections RFEM automatically sets the *Material* number *1 - Concrete C30/37*.

We click [OK] to import the cross-section values to the *New Cross-Section* dialog box.

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

Definition of rib

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

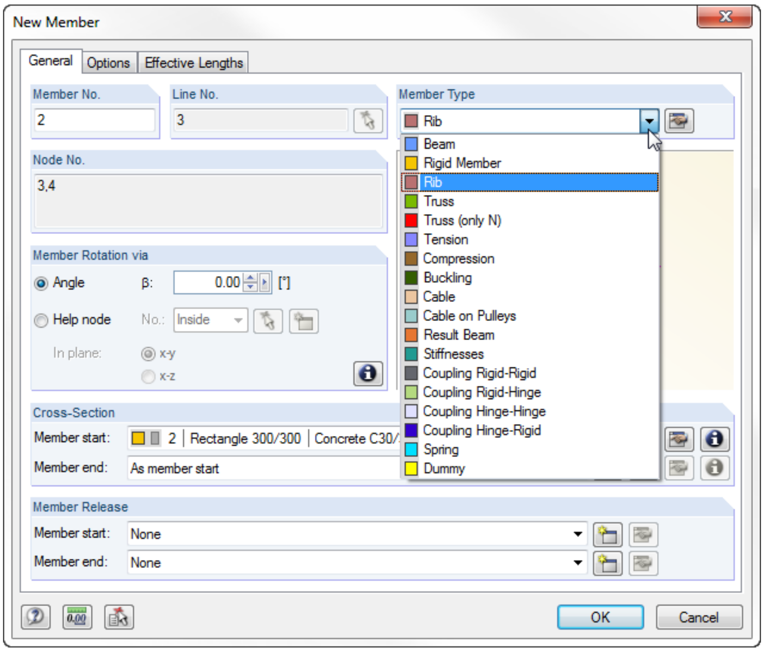


Figure 4.10: Changing the member type

Then, we click the [Edit] button to the right of the list box to open the *New Rib* dialog box.

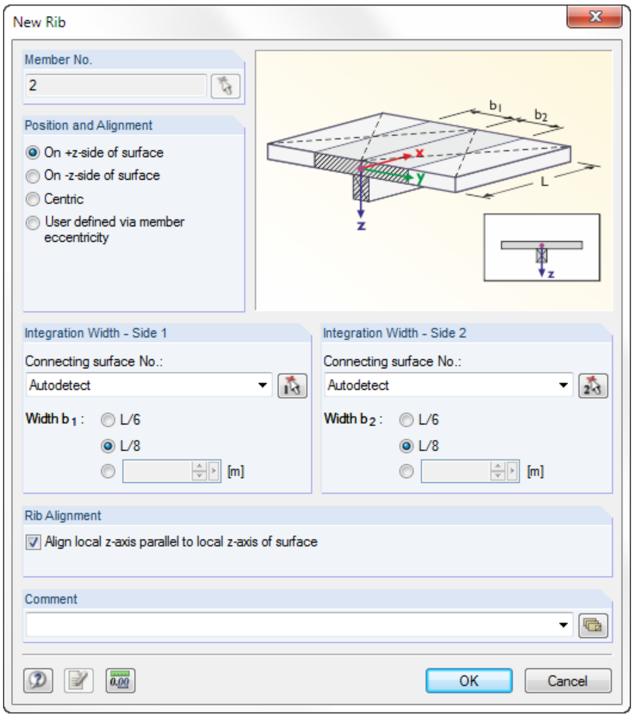


Figure 4.11: Defining the rib

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

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

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

Changing the view

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

To adjust the display, we use the [Move, Zoom, Rotate] button (see "mouse functions", page 8). The pointer turns into a hand. When we hold down the [Ctrl] key additionally, we can rotate the model by moving the pointer.

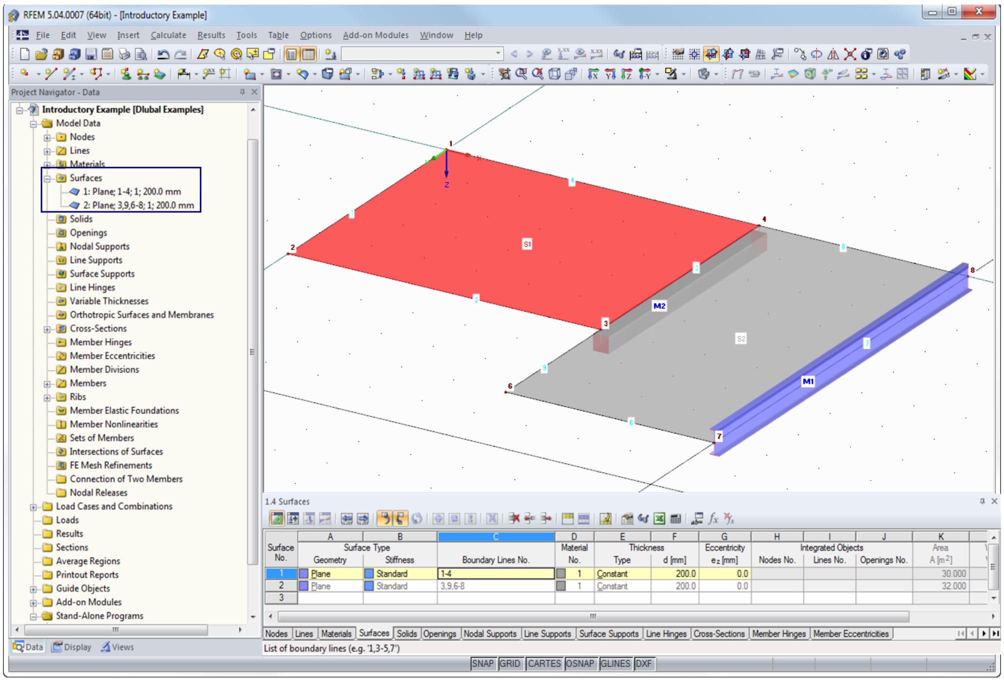


Figure 4.12: Model in isometric view with navigator and table entries

Checking data in navigator and tables

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

For example, in the navigator entry *Surfaces* and in Table 1.4 *Surfaces*, we see the input data of both surfaces in numerical form (see figure above).

4.3.2 Columns

The most comfortable way to create columns is copying the floor nodes downward by specifying particular settings for the copy process.

Node selection

First, we select the nodes that we want to copy. To open the corresponding dialog box,

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

or we use the toolbar button shown on the left.

The *Special Selection* dialog box has *Nodes* as the category by default (see Figure 4.13**Fehler! Verweisquelle konnte nicht gefunden werden.**).

As we want to select *All* nodes, we can confirm that dialog box without making any additional changes by clicking [OK].

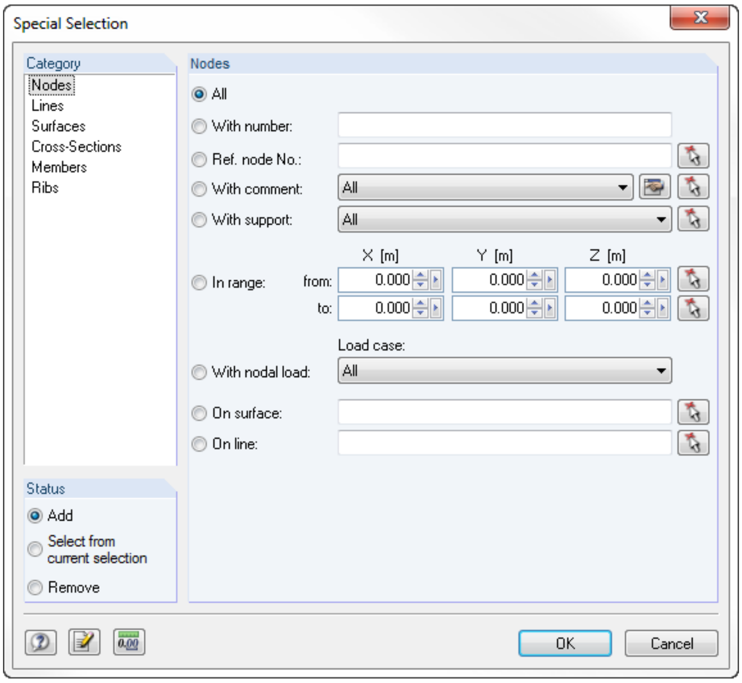


Figure 4.13: Dialog box *Special Selection*

The selected nodes are now displayed with a different color. Yellow is the default selection color for black backgrounds. (If, in addition, a surface is selected, it can be removed from the selection by holding the [Ctrl] key and clicking the surface.)

Copying nodes

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

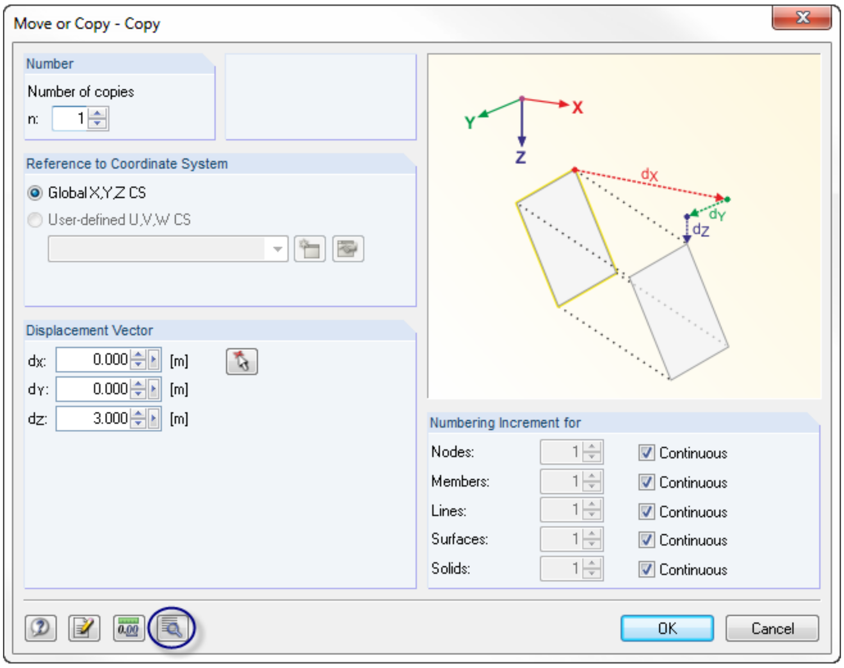


Figure 4.14: Dialog box *Move or Copy*

We increase the *Number of copies* to **1**: With this setting the nodes won't be moved but copied. As the columns are 3 m high, we enter the value **3.0** m for the *Displacement Vector* in d_Z.

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

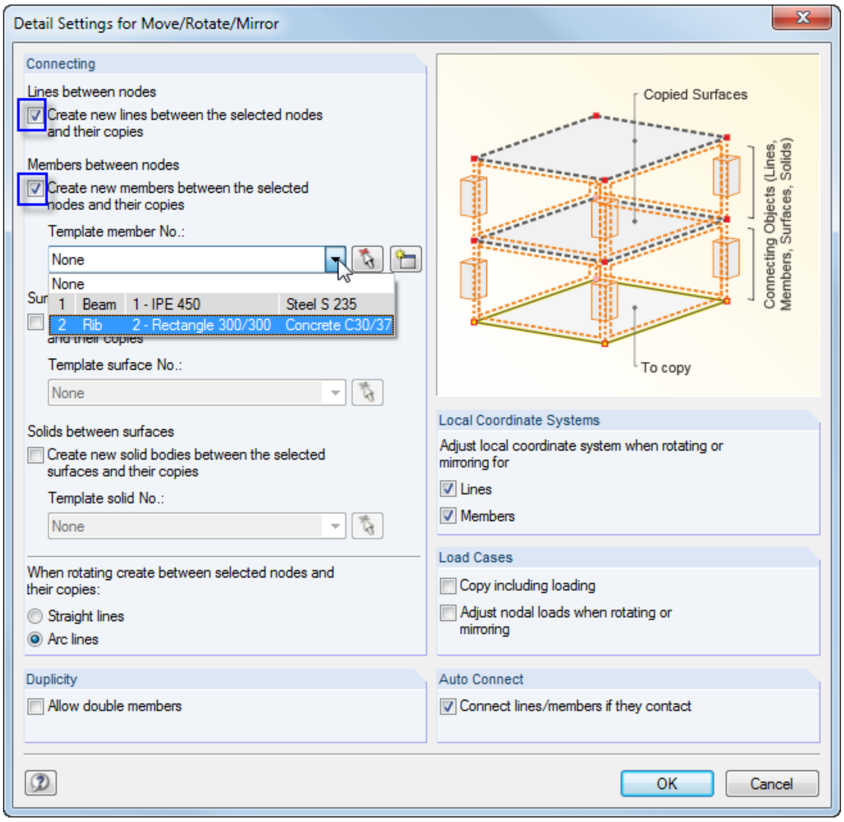


Figure 4.15: Dialog box *Detail Settings for Move/Rotate/Mirror*

In the dialog section *Connecting*, we select the check boxes for the following options:

☑ Create new lines between the selected nodes and their copies

☑ Create new members between the selected nodes and their copies

Then, we select member **2** from the list to define it as the *Template member*. Thus, the properties of the T-beam (member type, cross-section, material) are automatically set for the new columns.

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

Editing surfaces

Because we defined the template member as a *Rib* with integration widths, we now have to adjust the member type. We choose another way for the selection of columns.

First, we set the view in the [Y] direction by using the button shown on the left.

Now, we use the pointer to draw a window from the right to the left across the footing nodes of the columns. In this way, we select all objects that are completely or only partially contained in the window, so our columns are selected as well. (When we draw the window from the left to the right, we select only those objects that are completely contained in the window).

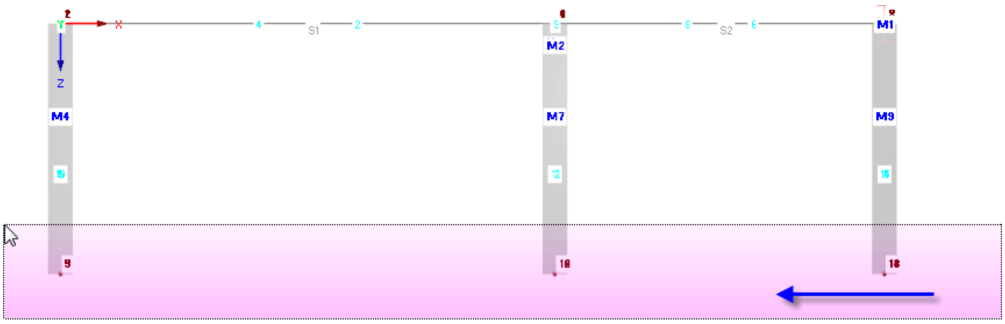


Figure 4.16: Selecting with window

Now, we double-click one of the selected columns. The *Edit Member* dialog box appears. The numbers of the selected members are shown in the *Member No.* box.

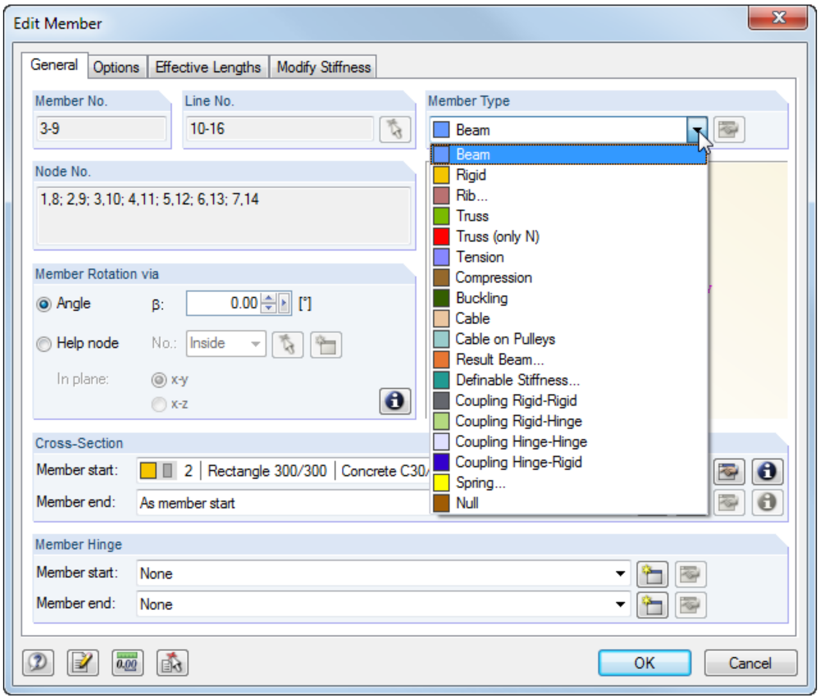


Figure 4.17: Adjusting the member type

We change the member type to **Beam** and close the dialog box with the [OK] button.

Again, we set the [Isometric View] to display our model completely.

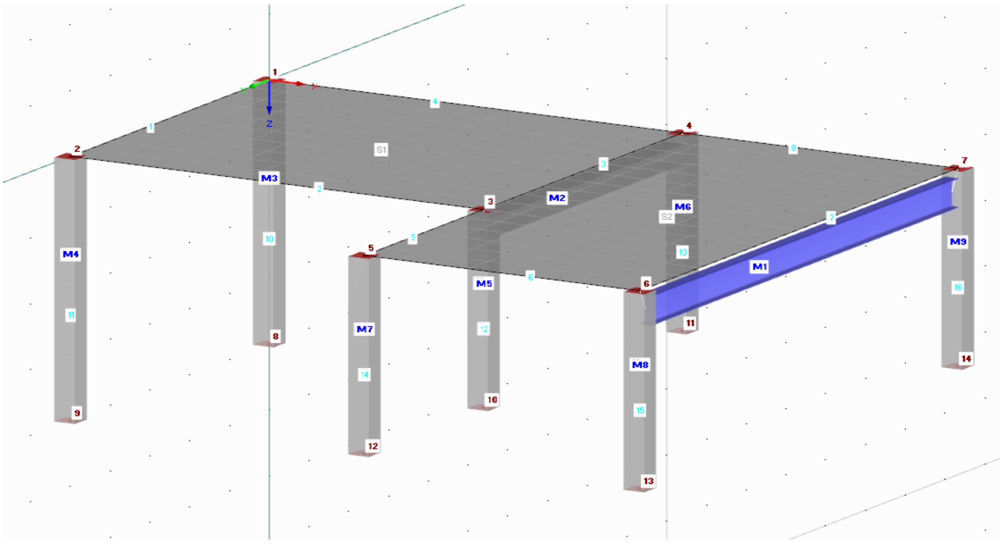


Figure 4.18: Full isometric view

4.4 Support Arrangement

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

Assigning nodal supports

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

The foot nodes and the columns remain selected as long as we do not click in the work window. If necessary, we select those objects again by window selection (see Figure 4.16).

Now, we double-click one of the selected foot nodes. Watching the status bar in the bottom left corner we can check if the pointer is placed on the relevant node.

The dialog box *Edit Node* opens.

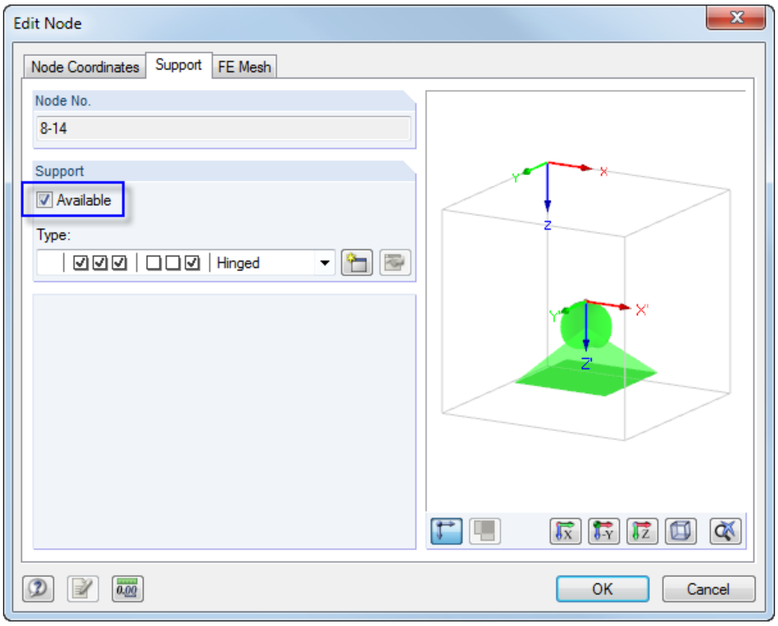


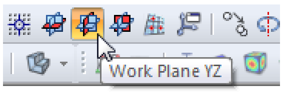
Figure 4.19: Dialog box *Edit Node*, tab *Support*

In the *Support* tab, we select the check box *Available*. With this setting we assign the *Hinged* support type to the selected nodes.

After clicking the [OK] button we can see the support symbols displayed in the model.

Changing the work plane

We want to correct the length of the two columns on the left to 4 m. Therefore, we shift the work plane from the horizontal to the vertical plane.



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

The grid is now displayed within the plane of the left columns. This setting allows us to define lines graphically or to displace nodes in this work plane.

Adjusting support nodes

We cancel the selection of nodes by clicking with the left mouse button into an "empty" space of the work window.

Now, we shift node *9* with the mouse by **1 m** to the grid point below. Please take care to select the node and not the member. Again, we can check the node numbers and the coordinates of the pointer in the status bar.

We repeat the same step for node *8*.

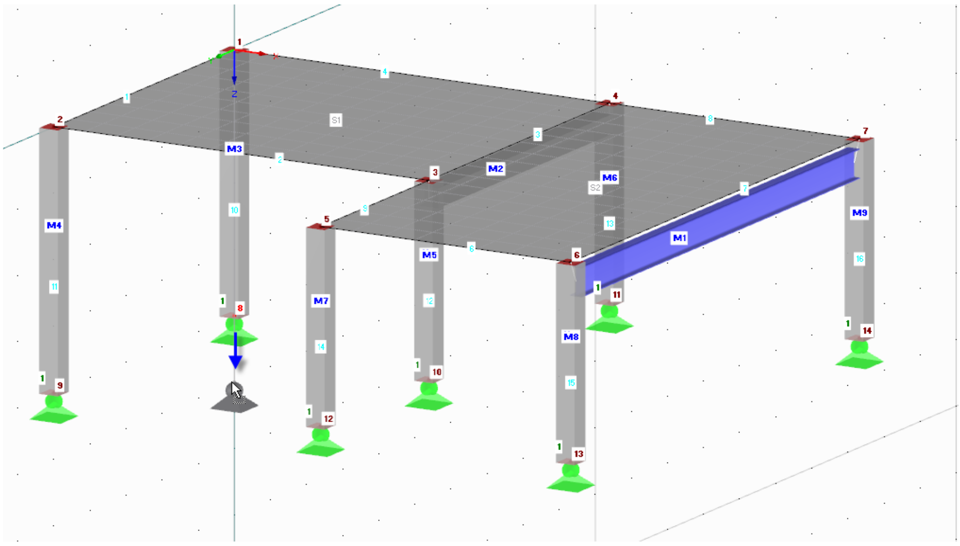


Figure 4.20: Shifting support node

Alternatively, it would be possible to double-click one of the nodes and to enter the correct Z-coordinate in the *Edit Node* dialog box.

4.5 Connecting Member with Hinge and Eccentricity

4.5.1 Hinge

The steel girder cannot transfer any bending moments to the columns because of its connection. Therefore, we have to assign hinges to both sides of the member.

We double-click member 1 to open the *Edit Member* dialog box.

In the *Member Hinge* dialog section, we click the [New] button to define a hinge type for the *Member start* (see also Figure 4.23).

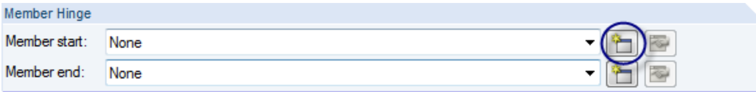


Figure 4.21: Dialog box *Edit Member*, dialog section *Member Hinge*

The *New Member Hinge* dialog box appears in which the displacements or rotations can be selected that are released at the member end. In our example, we select the check boxes for the rotations φ_y and φ_z. Thus, no bending moments can be transferred at the node.

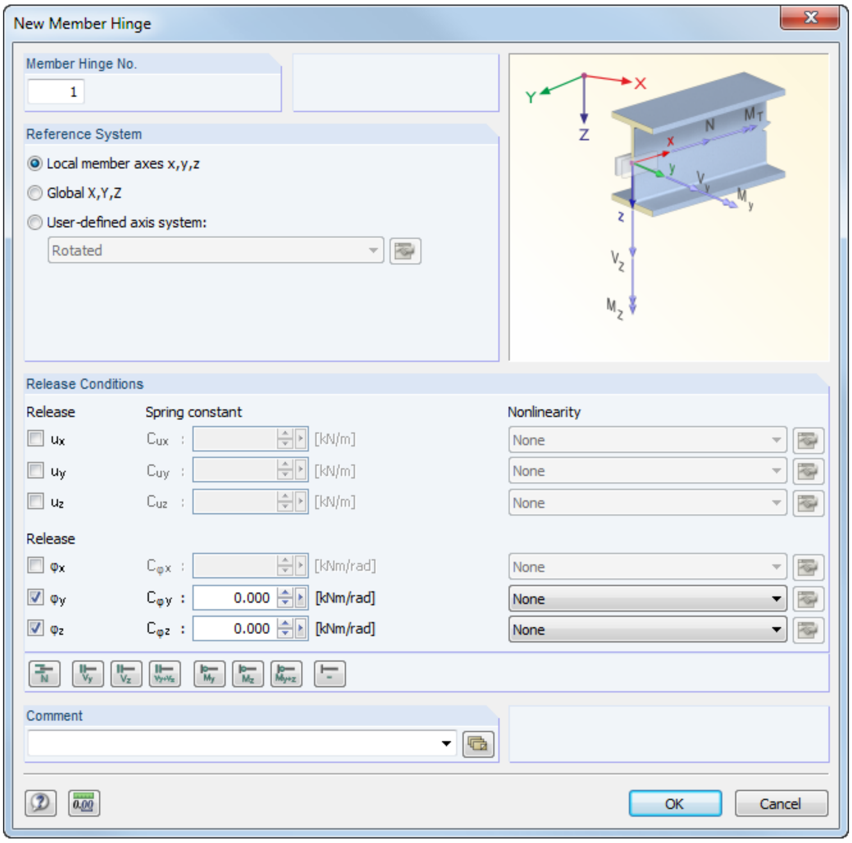


Figure 4.22: Dialog box *New Member Hinge*

We confirm the default settings and close the dialog box by clicking the [OK] button.

In the *Edit Member* dialog box we see that hinge 1 is now entered for the *Member start*. We define the same hinge type for the *Member end* by using the list (see Figure 4.23).

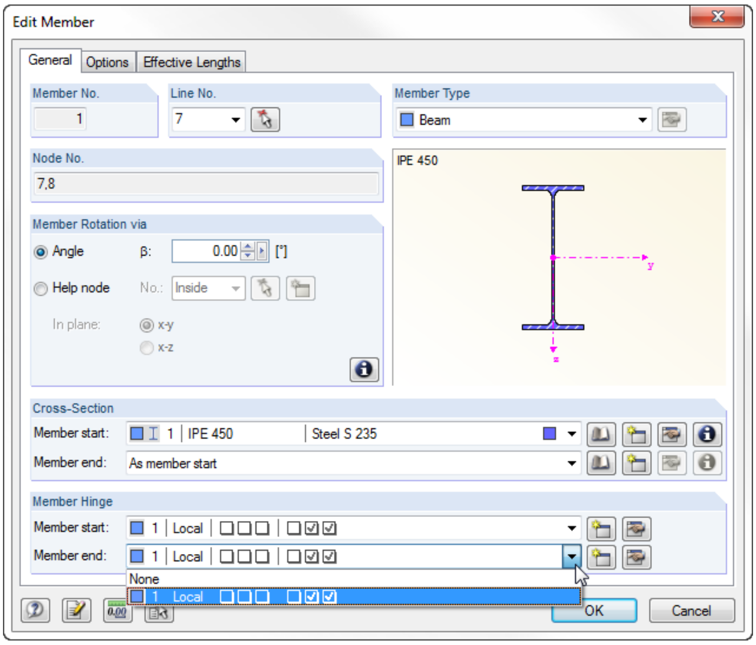


Figure 4.23: Assigning hinges in the *Edit Member* dialog box

4.5.2 Member Eccentricity

We want to connect the steel girder eccentrically below the floor slab.

In the *Edit Member* dialog box, we switch to the *Options* tab. In the *Member Eccentricity* section, we click the [New] button to open the *New Member Eccentricity* dialog box.

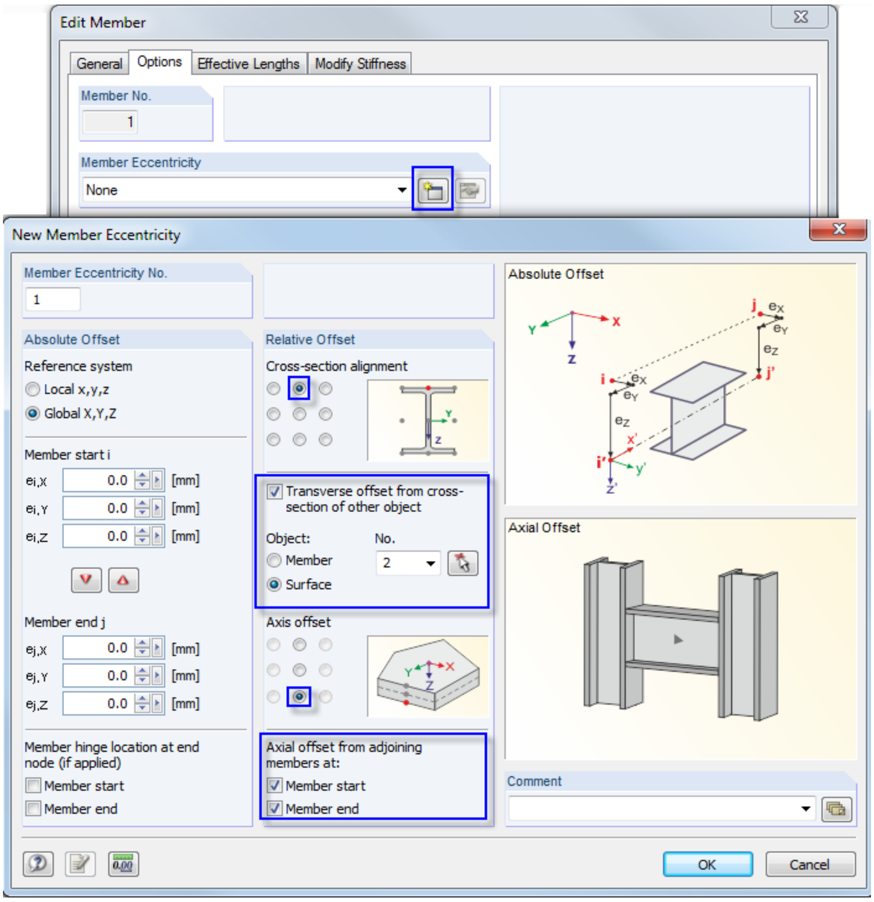


Figure 4.24: Dialog box *New Member Eccentricity*

We select the option *Transverse offset from cross-section of other object*. In our example, the object is the floor slab: We use the [Pick] function to define **Surface 2** graphically.

Then, we define the *Cross-section alignment* as well as the *Axis offset* as shown in Figure 4.24. Please watch out the local axis system on the picture

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

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

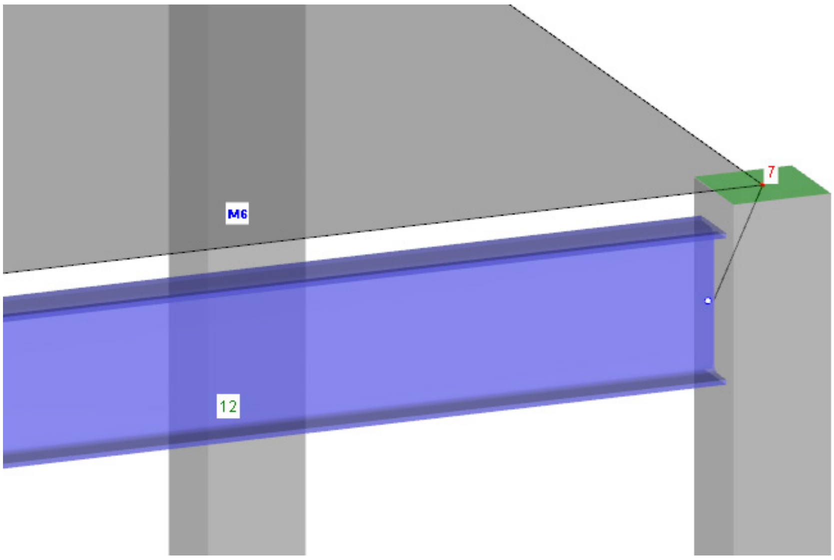


Figure 4.25: Steel girder with release and eccentricity

4.6 Checking the Input

Checking *Data* navigator and tables

The graphical input is reflected in both the *Data* navigator tree and the tables. We can display and hide the navigator and tables by selecting **Navigator** or **Table** on the **View** menu. We can also use the corresponding toolbar buttons.

In the tables, structural objects are organized in numerous tabs. Graphics and tables are interactive: To find an object in the table, for example a surface, we set Table 1.4 *Surfaces* and select the surface in the work window by clicking it. We see that the corresponding table row is highlighted (see Figure 4.12, page 17).

We can check the entered numerical data quickly.

Saving data

Finally, the entry of model data is complete. To save our file,

we select **Save** on the **File** menu

or use the toolbar button shown on the left.

5. Loads

First, loads such as self-weight, imposed or wind loads are described in different load cases. In the next step, we superimpose the load cases with partial safety factors according to specific combination rules (see Chapter 6).

5.1 Load Case 1: Self-Weight and Finishes

The first load case contains the permanently acting loads from self-weight and floor structure (see Chapter 2.3, page 6).

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

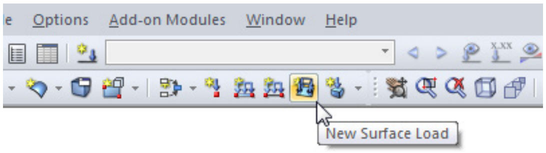


Figure 5.1: Button *New Surface Load*

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

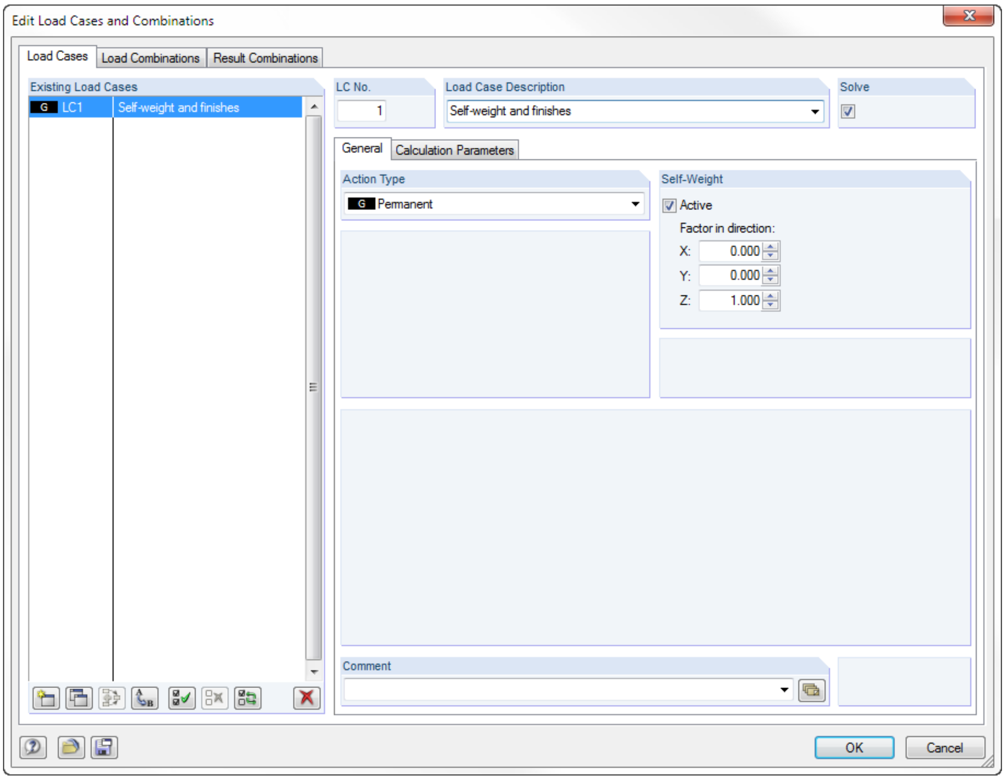
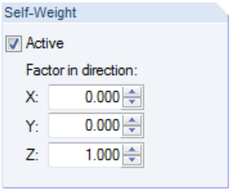


Figure 5.2: Dialog box *Edit Load Cases and Combinations*, tabs *Load Cases* and *General*

Load case No. *1* is preset with the action type *G Permanent*. In addition, we enter the *Load Case Description* **Self-weight and finishes**.

5.1.1 Self-weight



The *Self-Weight* of surfaces and members in the *Z*-direction is automatically taken into account when the *Active* check box is selected and the factor is specified as *-1.000* by default.

5.1.2 Floor Structure

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

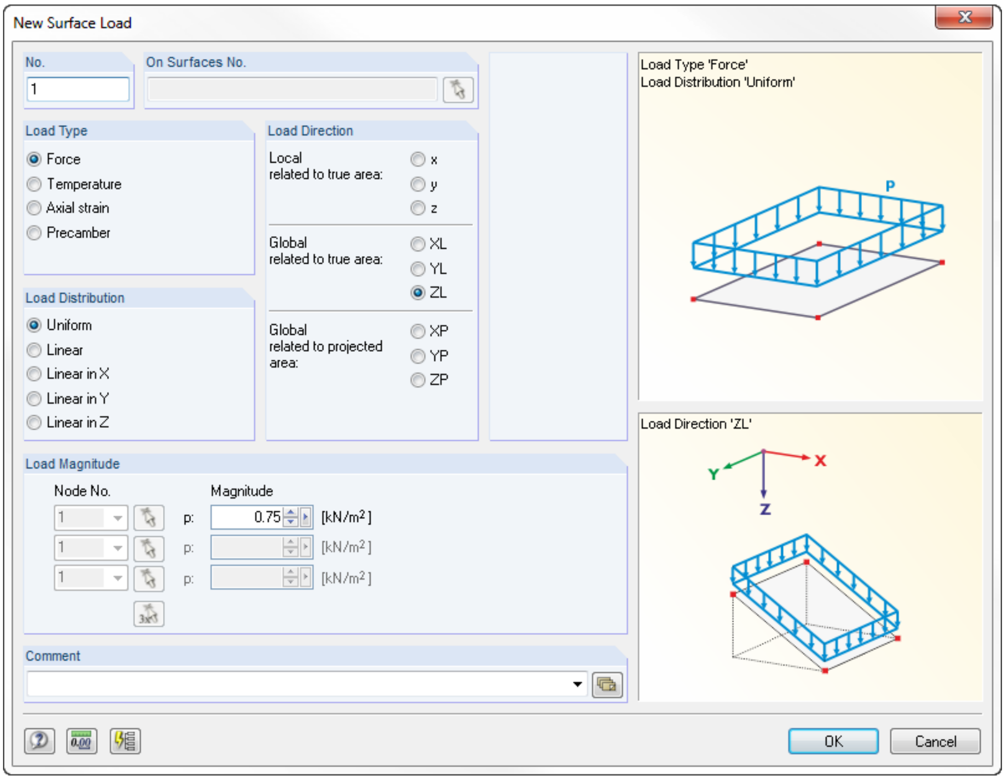


Figure 5.3: Dialog box *New Surface Load*

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

In the *Load Magnitude* dialog section, we enter a value of **0.75** kN/m^2 (see Chapter 2.3, page 6). Then, we close the dialog box by clicking [OK].

Now, we can assign the load graphically to the floor surface: We can see that a small load symbol has appeared next to the pointer. This symbol disappears as soon as we move the pointer across a surface. We apply the load by clicking the surfaces **1** and **2** one after the other (see Figure 5.4).

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

To quit the input mode, we use the [Esc] key. We can also right-click in the empty work window. The input for the load case *Self-weight and finishes* is complete.

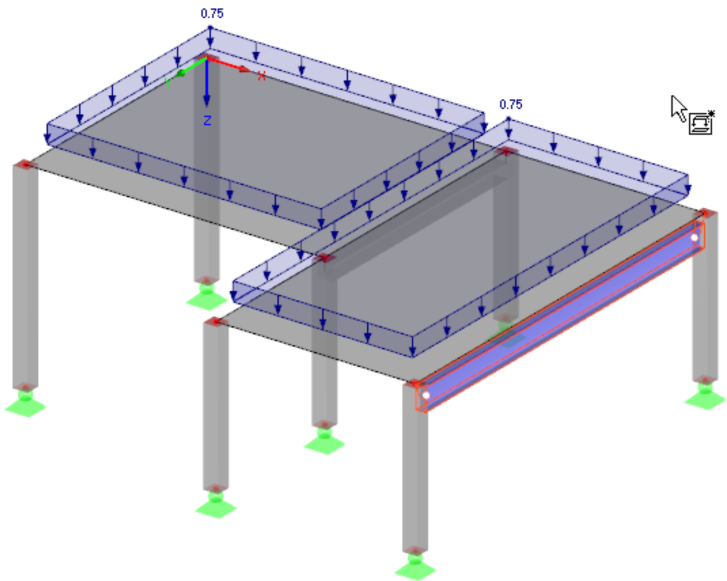


Figure 5.4: Graphical input of floor load

5.2 Load Case 2: Imposed Load, Area 1

We divide the imposed load of the floor into two different load cases because of the effects of continuity. To create a new load case,

we point to **Loads** on the **Insert** menu and select **New Load Case**

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

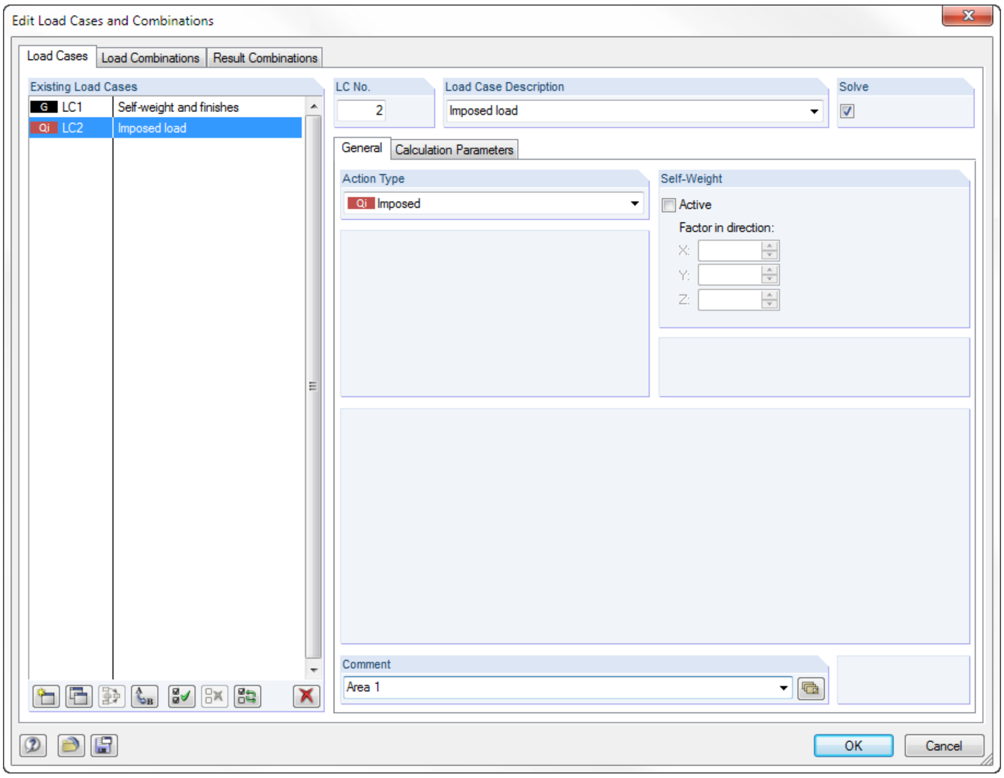


Figure 5.5: Dialog box *Edit Load Cases and Combinations*, tab *Load Cases*

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

The *Action Type* is set automatically to **Q_i Imposed**. This classification is important for the partial safety factors and combination coefficients of the load combinations.

In the *Comment* box, we can enter **Area 1** to describe the load case in detail.

After confirming, we enter the surface load in a new entry method: First, we select floor surface *1* by clicking. Now, when we open the dialog box by means of the [New Surface Load] button, we can see that the number of the surface is already entered.

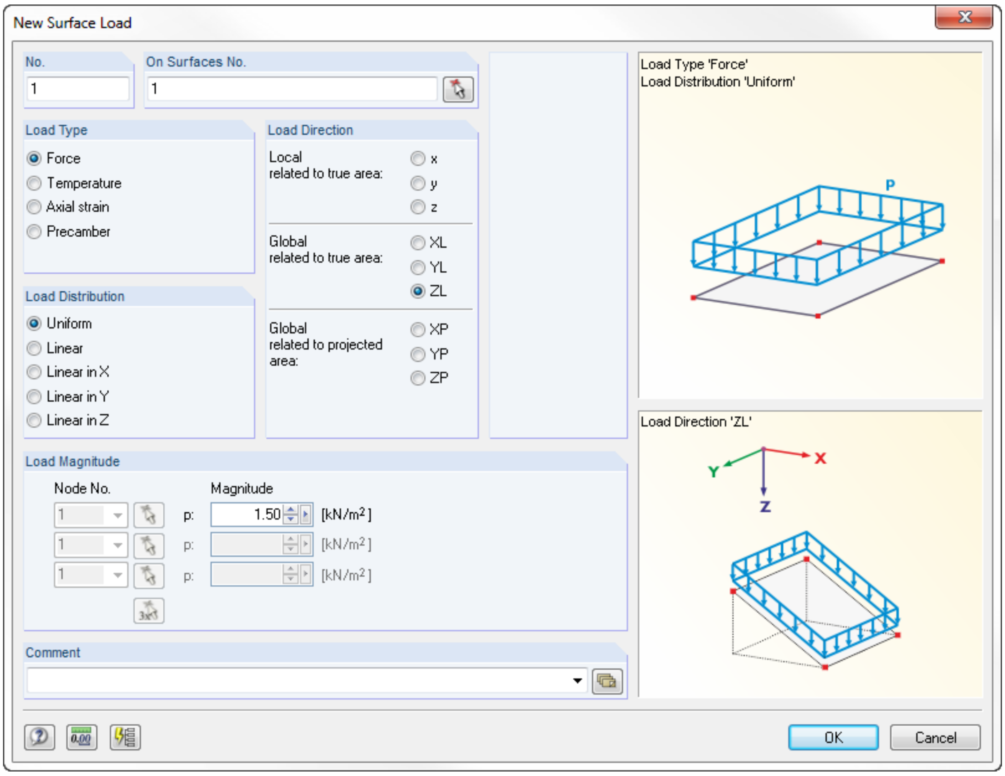


Figure 5.6: Dialog box *New Surface Load*

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

In the *Load Magnitude* dialog section, we enter a value of **1.5** kN/m^2 (see Chapter 2.3, page 6). Then, we close the dialog box by clicking [OK].

The surface load is displayed in the left area of the floor.

5.3 Load Case 3: Imposed Load, Area 2

We create a [New Load Case] to enter the imposed load of the right area.

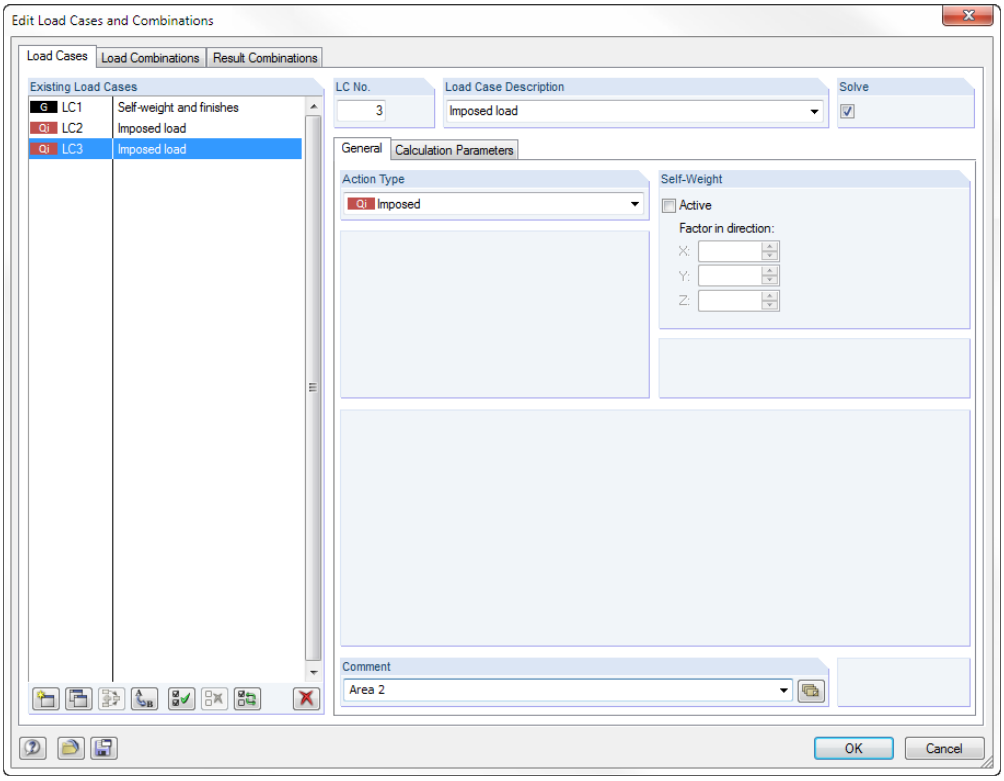


Figure 5.7: Dialog box *Edit Load Cases and Combinations*, tab *Load Cases*

Again, we enter **Imposed load** for the *Load Case Description*. In the *Comment* box, we enter **Area 2**. Then we close the dialog box with [OK].

5.3.1 Surface Load

This time we select floor surface *2* and open the dialog box *New Surface Load* with the [New Surface Load] button.

In addition to surface 2, we can see that the parameters of the recent entry are automatically set (load type *Force*, load distribution *Uniform*, load direction *Global ZL*, *Load Magnitude* **1.5** kN/m^2 – see Figure 5.6). We can confirm the dialog box without making any changes.

The surface load is displayed in the right area of the floor (see Figure 5.8).

5.3.2 Line Load

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

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

The line load as load type *Force* with a *Uniform* load distribution is acting in the *ZL* load direction. In the *Load Parameters* dialog section, we enter **5** kN/m (see Chapter 2.3, page 6).

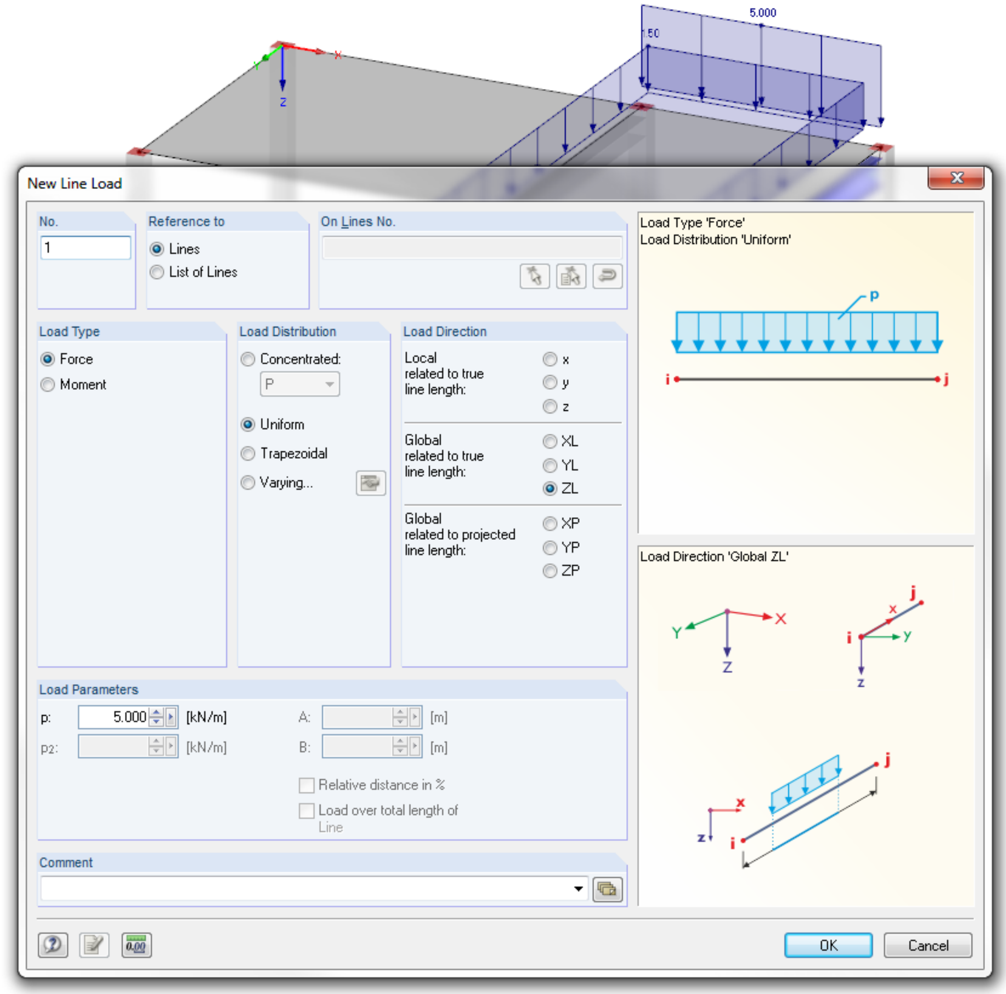
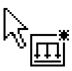


Figure 5.8: Dialog box *New Line Load*



After clicking the [OK] button we click line **8** at the floor's rear edge (check by status bar).

We close the input mode with the [Esc] button or by right-clicking in the empty workspace. Then, we reset the [Isometric View].

5.4 Load Case 4: Imperfections

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

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

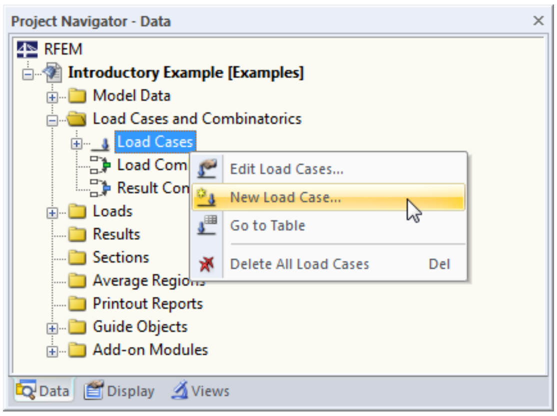


Figure 5.9: Shortcut menu *Load Cases*

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

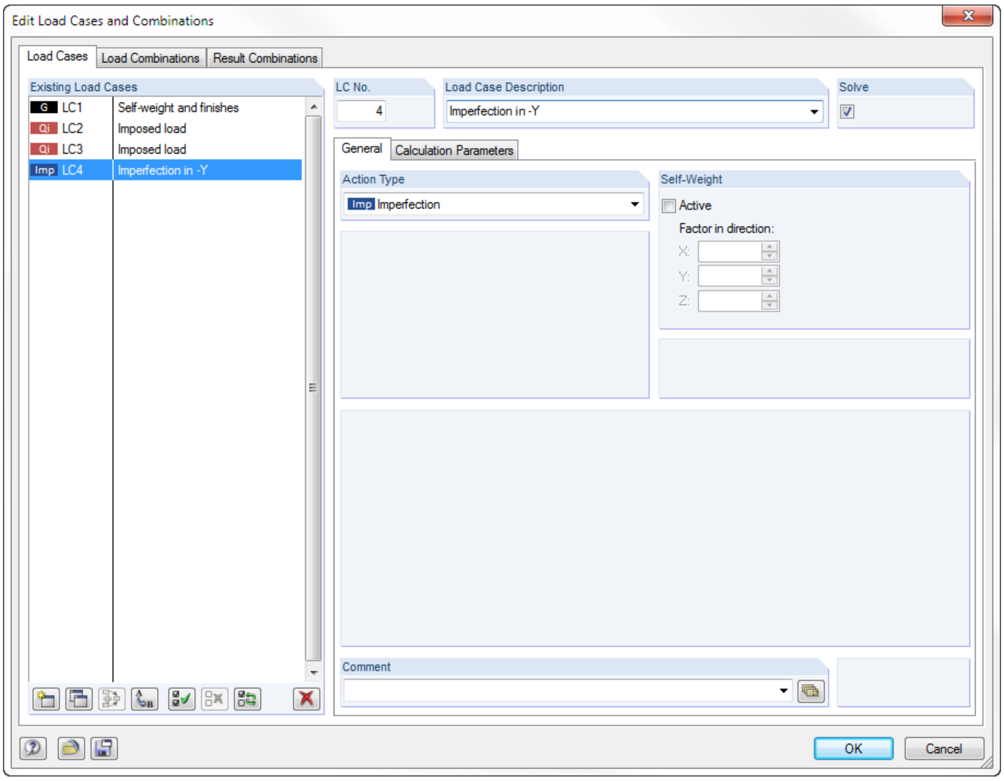
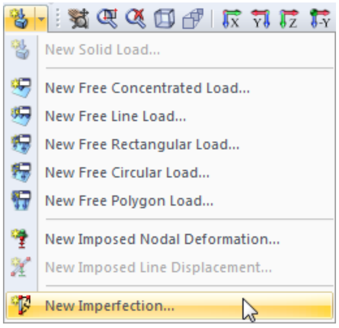


Figure 5.10: Dialog box *Edit Load Cases and Combinations*, tab *Load Cases*

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



List button for loads

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

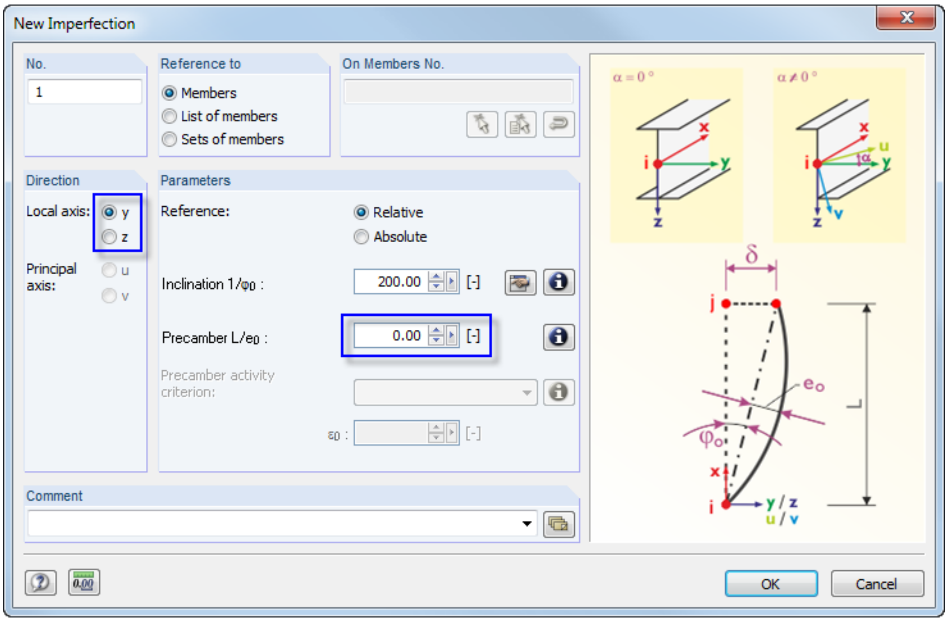
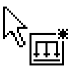


Figure 5.11: Dialog box *New Imperfection*

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

We set the *Precamber L/e₀* to **0.00** and confirm the dialog box by clicking the [OK] button.

We can assign the imperfection easily by using a selection window. First, we put the model in a more appropriate position: We click the [Move, Zoom, Rotate] button and incline the model a little bit backwards by holding down the left mouse button and keeping the [Ctrl] key additionally pressed. We stop changing the view with the [Esc] button or a right-click in the window without canceling the function "Select Members for Imperfections".



Then, we draw a selection window from the right to the left. We have to take care that we catch each column with the window, but the steel girder must lie outside the selection zone.

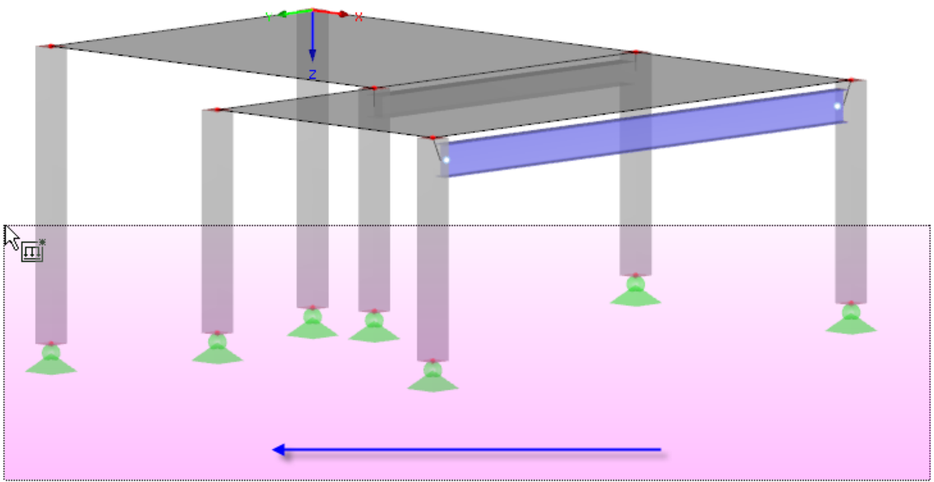


Figure 5.12: Selecting columns for imperfections

When the second corner of the window is set, RFEM assigns the imperfections.

We quit the function with the [Esc] key or a right-click. Finally, we reset the [Isometric View].

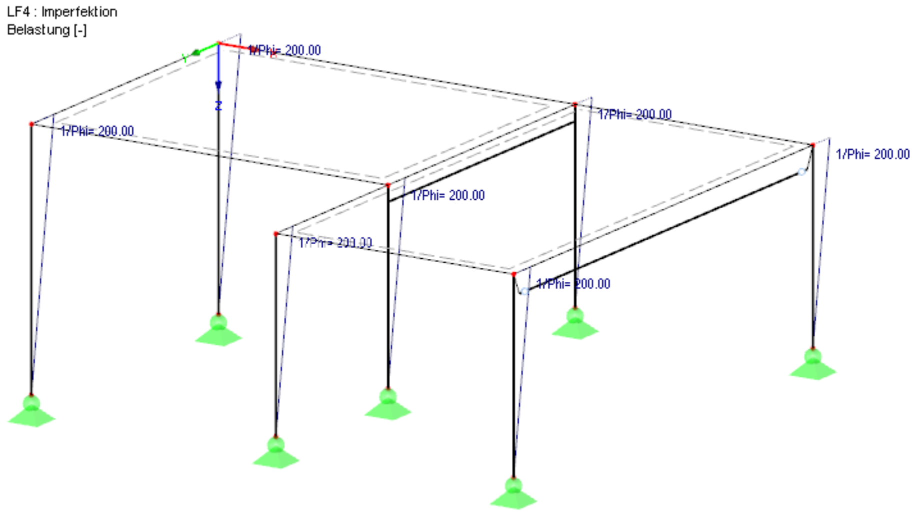
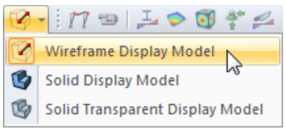


Figure 5.13: Imperfections shown in line model

Changing the model display



The figure above shows the structure as *Wireframe Display Model*. We can set this display option with the toolbar button shown on the left. In this way, the imperfections are no longer overlapped by rendered columns.

5.5 Checking Load Cases

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

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

Figure 5.14: Browsing the load cases

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

Again, the graphic and tables are interactive: To find a load in the table, for example an imperfection, we set Table 3.14 *Imperfections*, and then we select the load in the work window. We see that the pointer jumps into the corresponding row of the table.

6. Combination of Load Cases

According to EN 1990, we have to combine the load cases with factors. The *Action Type* specified before, when we created the load cases, makes generating combinations easier (see Figure 5.10, page 32). In this way, we can control the partial safety factors and combination coefficients when combinations are created.

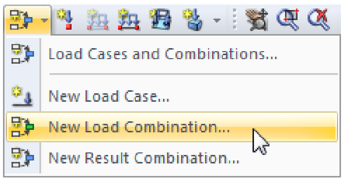
6.1 Creating Load Combinations

With our four load cases we create the following load combinations:

- 1.35*LC1 + 1.5*LC2 + 1.0*LC4 Imposed load in area 1
- 1.35*LC1 + 1.5*LC3 + 1.0*LC4 Imposed load in area 2
- 1.35*LC1 + 1.5*LC2 + 1.5*LC3 + 1.0*LC4 Full load

We calculate the model according to nonlinear second-order analysis.

Creating CO1



We open the menu for [Load Cases and Combinations] and create a *New Load Combination*. The *Edit Load Cases and Combinations* dialog box appears again.

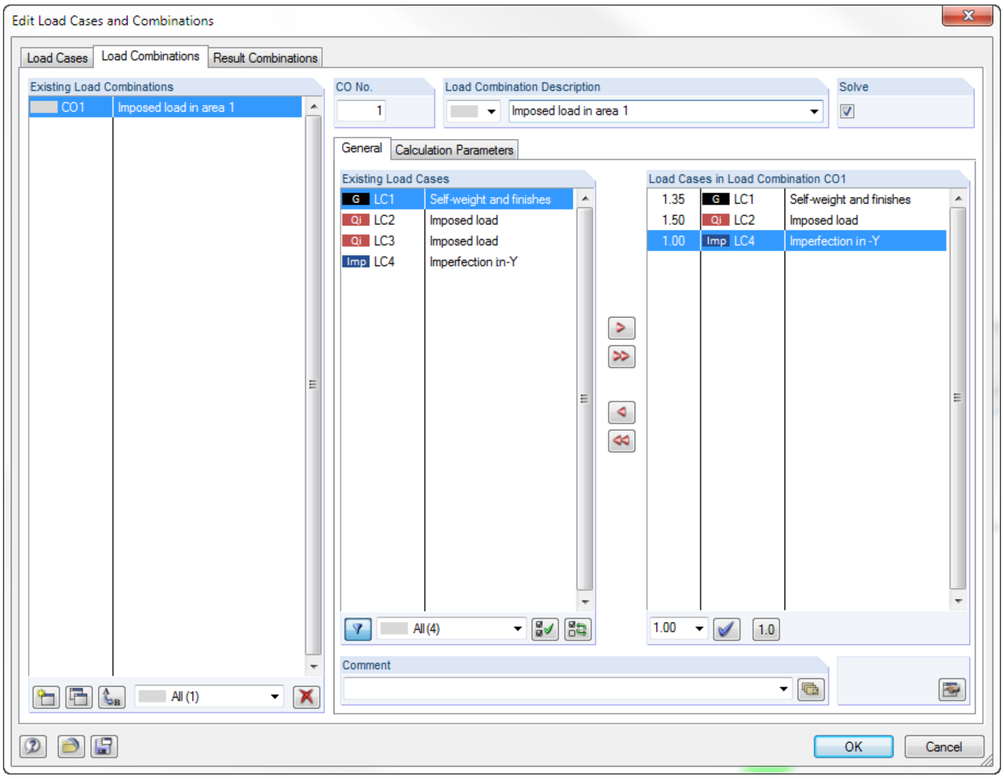


Figure 6.1: Dialog box *Edit Load Cases and Combinations*, tab *Load Combinations*

We enter **Imposed load in area 1** for the *Load Combination Description*.

Below, in the list *Existing Load Cases*, we click **LC1**. Then, we use the [▶] button to transfer the load case to the list *Load Cases in Load Combination CO1* on the right. We do the same with **LC2** and **LC4**.

In the tab *Calculation Parameters*, we check if the *Method of Analysis* is set according to *Second-order analysis* (see the following picture).

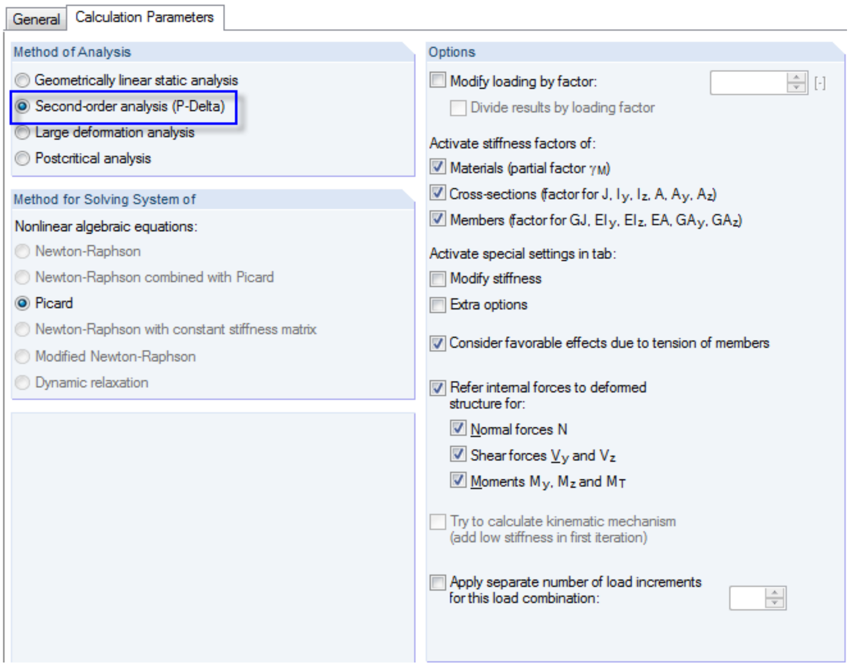


Figure 6.2: Tab *Calculation Parameters*

After clicking [OK] all loads contained in the load combination are shown in the model. The factors of the load cases have been considered for the values.

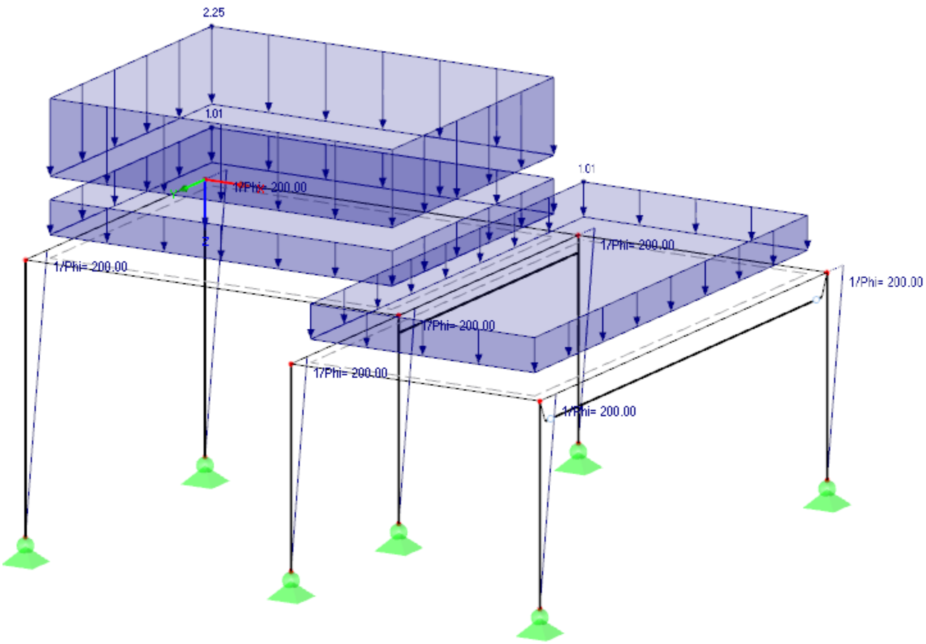
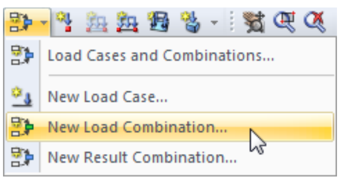


Figure 6.3: Loads of load combination CO1

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



Creating CO2

We create the second load combination in the same way: We create a [New Load Combination], but this time we enter **Imposed load in area 2** for the *Load Combination Description*.

The load cases which are relevant for this load combination are **LC1**, **LC3** and **LC4**. Again, we use the [▶] button to select them.

Creating CO3

To create the last load combination, we choose another way of creation: We right-click the navigator entry *Load Combinations* and select *New Load Combination* in the shortcut menu.

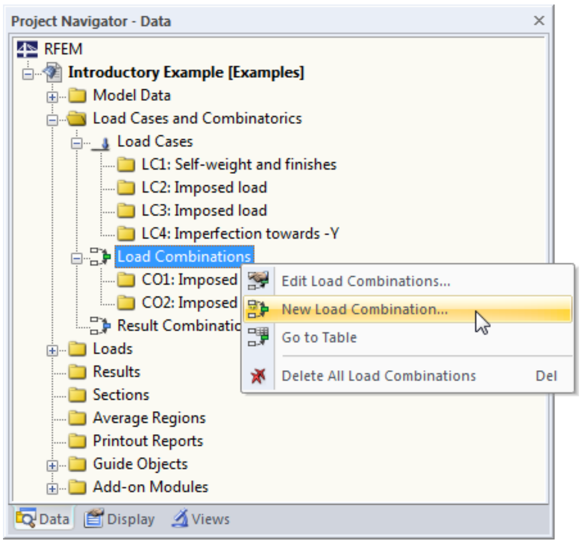


Figure 6.4: Creating COs using the navigator shortcut menu

We enter **Full load** for the *Load Combination Description*. With the [Add All Load Cases] button we can transfer all four load cases together to the list on the right.

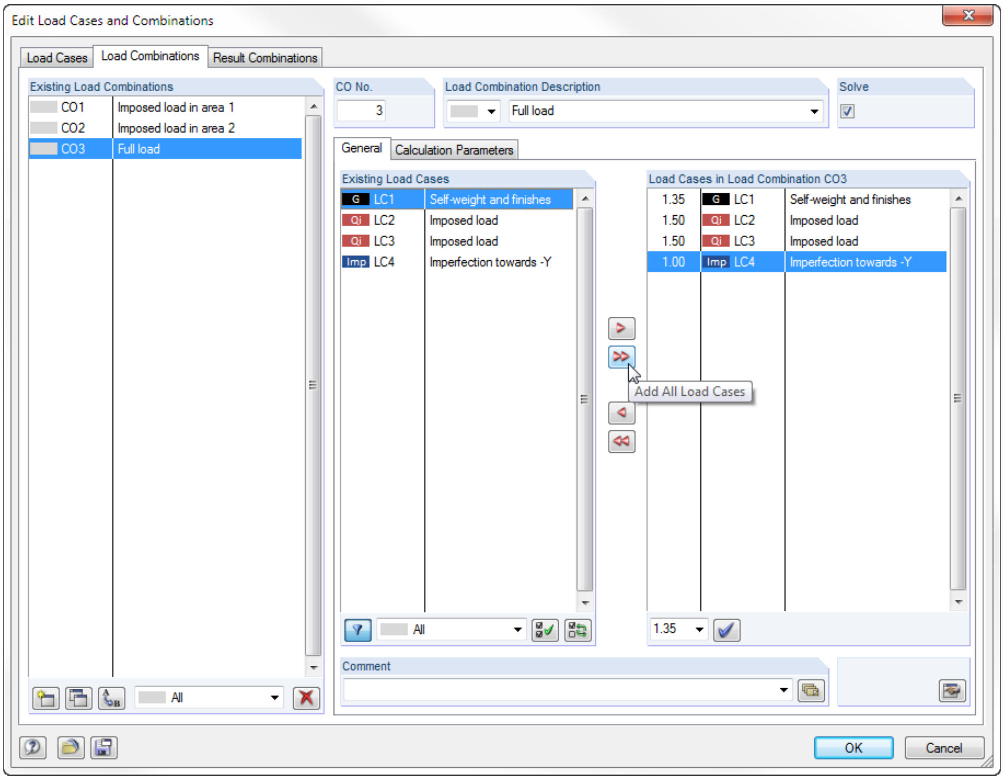
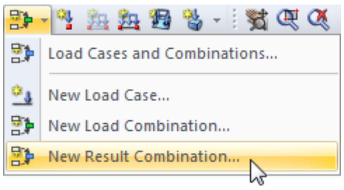


Figure 6.5: Dialog box *Edit Load Cases and Combinations*, tab *Load Combinations*

As the load cases LC2 and LC3 are assigned to the action type *Imposed*, they are both applied with the partial safety factor 1.5. In the case of different categories one load case would be the leading action, the other one would be the secondary load with reduced factor.

6.2 Creating Result Combinations

From the results of the three load combinations we create an envelope containing the positive and negative extreme values.



In the menu for [Load Cases and Combinations], we select *New Result Combination*. We see the *Edit Load Cases and Combinations* dialog box which is already familiar to us.

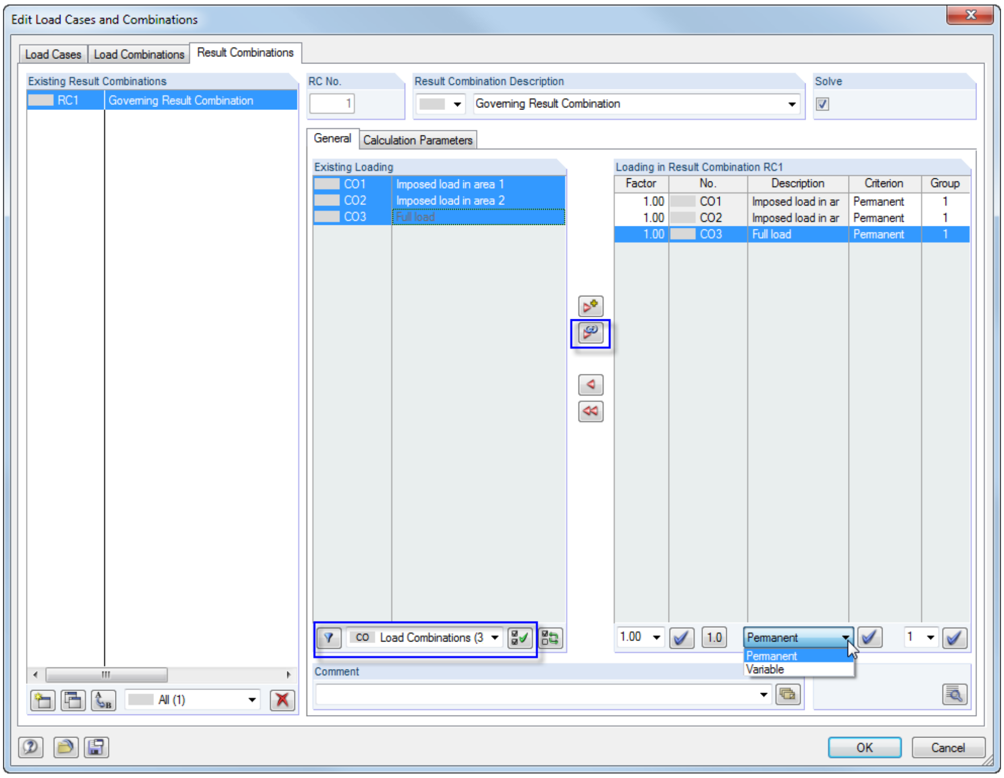


Figure 6.6: Dialog box *Edit Load Cases and Combinations*, tab *Result Combinations*

We choose **Governing Result Combination** from the *Result Combination Description* list.

To display the load combinations in the dialog section *Existing Loading*, we select *CO Load Combinations* from the list below the load table on the left. Then, we select all three load combinations by clicking the [Select All Listed Loading] button.

The selection box below the load table on the right indicates the superposition factor which is preset to 1.00. The setting conforms to our intention to determine the extreme values of this load combination. We change the superposition rule to **Permanent** in the list for all load combinations. Thus, RFEM always takes into account at least one of the actions.

We use the [Add Selected with 'or'] button to transfer the three load combinations to the list on the right. The value 1 below the final column tells us that all entries belong to the same group: They won't be treated as additive but alternatively acting.

Now, the superposition criteria is completely defined. We click [OK] and [Save] the entry.

7. Calculation

7.1 Checking Input Data

Before we calculate our structure, we want RFEM to check our structural and load data. To open the corresponding dialog box,

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

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

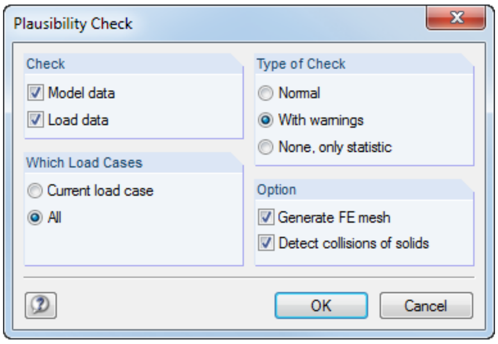


Figure 7.1: Dialog box *Plausibility Check*

If no error is detected after clicking [OK], the following message is displayed. In addition, a short summary of structural and load data is shown.

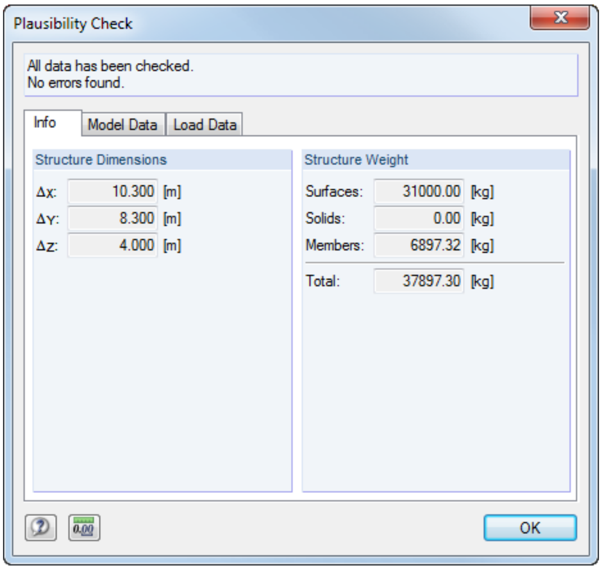


Figure 7.2: Result of plausibility check

We find more tools for checking the structural and load data by selecting

Model Check on the **Tools** menu.

7.2 Generating the FE Mesh

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

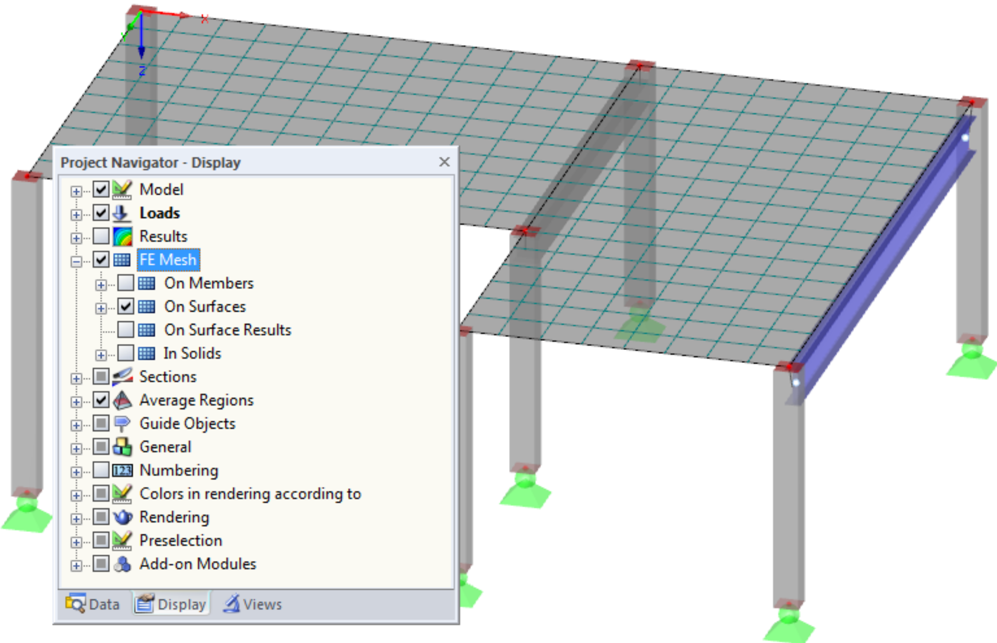


Figure 7.3: Model with generated FE mesh

7.3 Calculating the Model

To start the calculation,

we select **Calculate All** on the **Calculate** menu

or we use the toolbar button shown on the left.

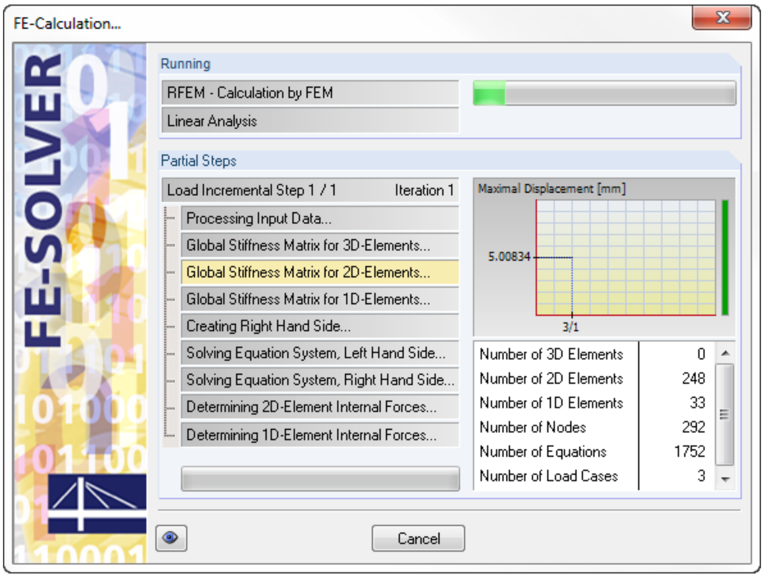


Figure 7.4: Calculation process

8. Results

8.1 Graphical Results

As soon as the calculation is finished, RFEM displays the deformations of the load case currently set. The last load setting was RC1, so now we see the maximum and minimum results of this result combination.

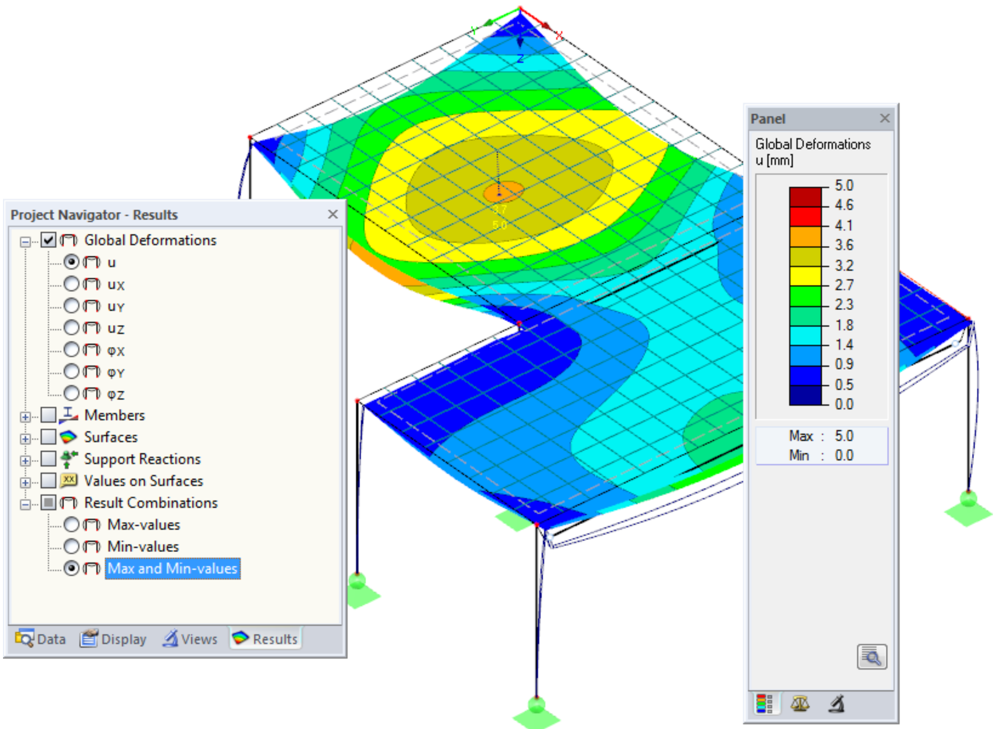


Figure 8.1: Graphic of max/min deformations for result combination RC1

Selecting load cases and load combinations

We can use the toolbar buttons [◀] and [▶] (to the right of the load case list) to switch between the results of load cases, load combinations and result combinations. We already know the buttons from checking the load cases. It is also possible to select the loads in the list.

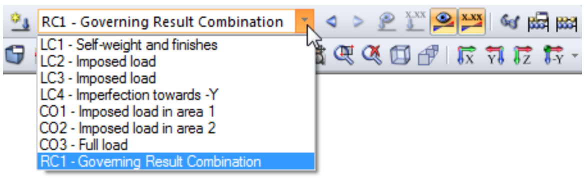


Figure 8.2: Load case list in the toolbar

Selecting results in the navigator

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

The check boxes for the individual results categories (for example *Global Deformations*, *Members*, *Surfaces*, *Support Reactions*) determine which deformations or internal forces are shown. Within these categories are even more individual types of results that we can select for display.

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

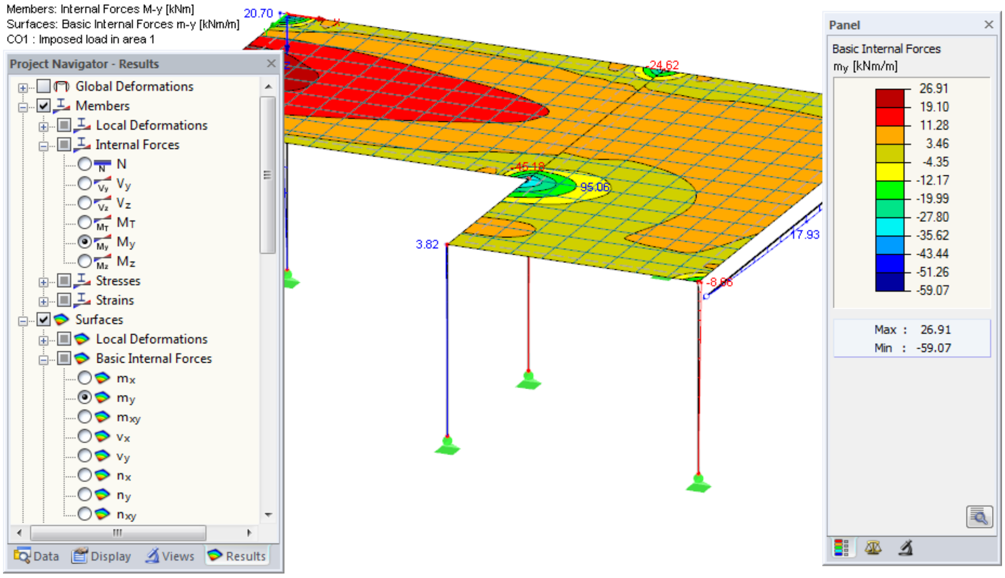
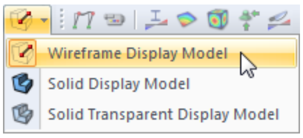


Figure 8.3: Setting internal forces of members and surfaces in *Results* navigator



In the figure above, we see the member internal forces M_y and the surface internal forces m_y calculated for CO1. To display the forces, it is recommended to use the wire-frame model. We can set this display option with the button shown on the left.

Display of values

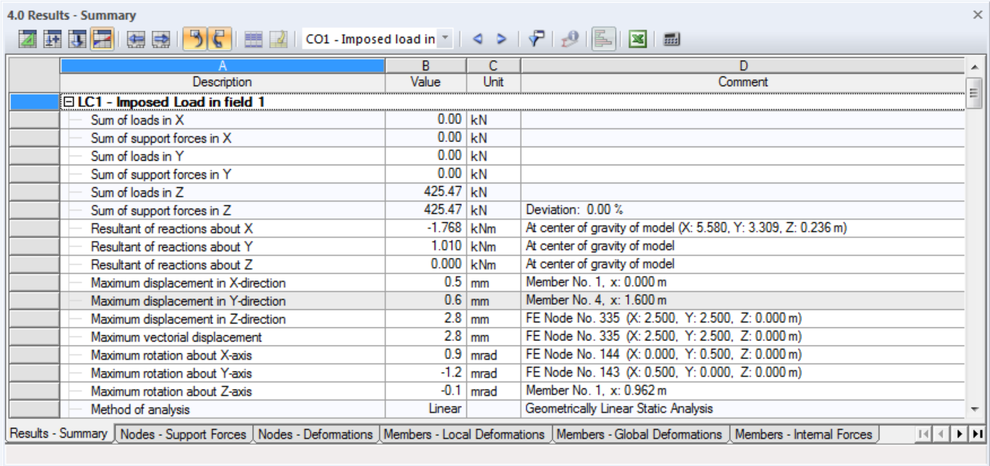
The color scale in the control panel shows us the color range. We can switch on the result values by selecting the option **Values on Surfaces** in the *Results* navigator. To display all values of the FE mesh nodes or grid points, we clear the selection for *Extreme Values* additionally.

Figure 8.4: Grid point moments m_x of floor slab in Z-view (CO1)

8.2 Results Tables

We can also evaluate results in tables.

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



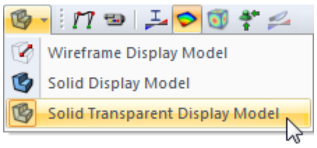
4.0 Results - Summary

CO1 - Imposed load in

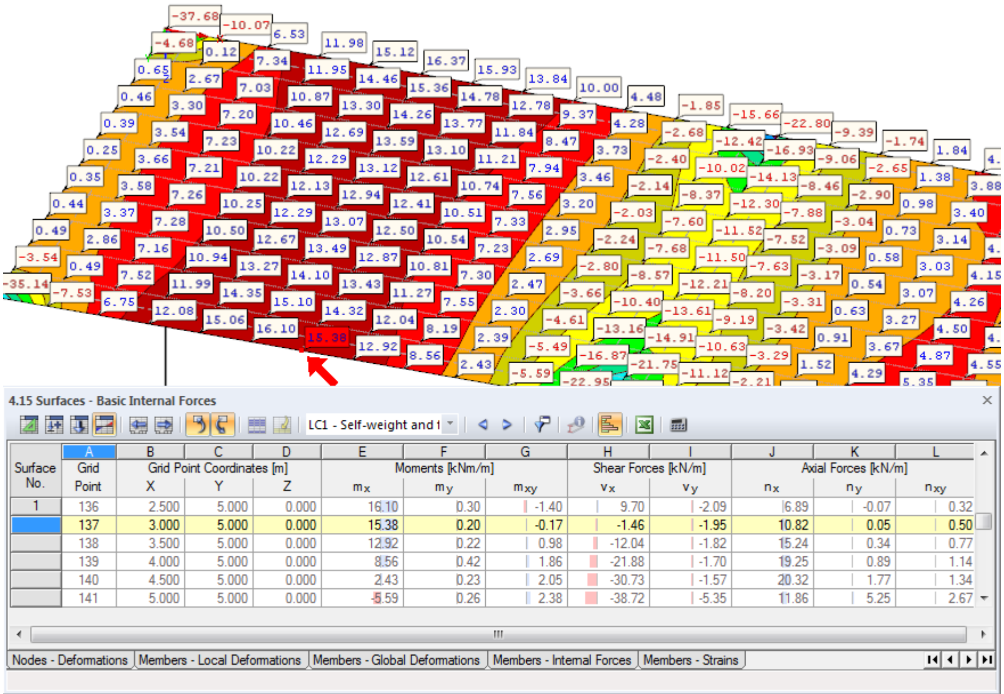
Description	Value	Unit	Comment
LC1 - Imposed Load in field 1			
Sum of loads in X	0.00	kN	
Sum of support forces in X	0.00	kN	
Sum of loads in Y	0.00	kN	
Sum of support forces in Y	0.00	kN	
Sum of loads in Z	425.47	kN	
Sum of support forces in Z	425.47	kN	Deviation: 0.00 %
Resultant of reactions about X	-1.768	kNm	At center of gravity of model (X: 5.580, Y: 3.309, Z: 0.236 m)
Resultant of reactions about Y	1.010	kNm	At center of gravity of model
Resultant of reactions about Z	0.000	kNm	At center of gravity of model
Maximum displacement in X-direction	0.5	mm	Member No. 1, x: 0.000 m
Maximum displacement in Y-direction	0.6	mm	Member No. 4, x: 1.600 m
Maximum displacement in Z-direction	2.8	mm	FE Node No. 335 (X: 2.500, Y: 2.500, Z: 0.000 m)
Maximum vectorial displacement	2.8	mm	FE Node No. 335 (X: 2.500, Y: 2.500, Z: 0.000 m)
Maximum rotation about X-axis	0.9	mrad	FE Node No. 144 (X: 0.000, Y: 0.500, Z: 0.000 m)
Maximum rotation about Y-axis	-1.2	mrad	FE Node No. 143 (X: 0.500, Y: 0.000, Z: 0.000 m)
Maximum rotation about Z-axis	-0.1	mrad	Member No. 1, x: 0.962 m
Method of analysis	Linear		Geometrically Linear Static Analysis

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

Figure 8.5: Table 4.0 *Results - Summary*



To select other tables, we click the corresponding table tabs. To find specific results in the table, for example the internal forces of floor surface 1, we set Table 4.15 *Surfaces - Basic Internal Forces*. Now, we select the surface in the graphic (the transparent model representation makes the selection easier) and we see that RFEM jumps to the surface's basic internal forces in the table. The current grid point, that means the position of the pointer in the table row, is indicated by an arrow in the graphic.



4.15 Surfaces - Basic Internal Forces

LC1 - Self-weight and 1

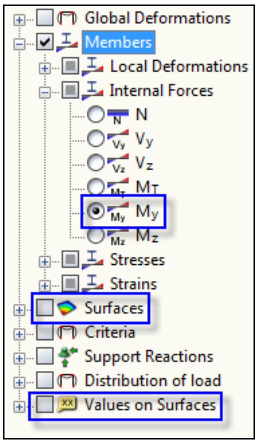
Surface No.	Grid Point	Grid Point Coordinates [m] X	Y	Z	Moments [kNm/m] m_x	m_y	m_{xy}	Shear Forces [kN/m] v_x	v_y	Axial Forces [kN/m] n_x	n_y	n_{xy}
1	136	2.500	5.000	0.000	16.10	0.30	-1.40	9.70	-2.09	6.89	-0.07	0.32
	137	3.000	5.000	0.000	15.38	0.20	-0.17	-1.46	-1.95	10.82	0.05	0.50
	138	3.500	5.000	0.000	12.92	0.22	0.98	-12.04	-1.82	15.24	0.34	0.77
	139	4.000	5.000	0.000	8.56	0.42	1.86	-21.88	-1.70	19.25	0.89	1.14
	140	4.500	5.000	0.000	2.43	0.23	2.05	-30.73	-1.57	20.32	1.77	1.34
	141	5.000	5.000	0.000	-5.59	0.26	2.38	-38.72	-5.35	11.86	5.25	2.67

Nodes - Deformations | Members - Local Deformations | Members - Global Deformations | Members - Internal Forces | Members - Strains

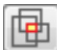
Figure 8.6: Surface internal forces in Table 4.15 and marker of current grid point in the model

Like the browsing function in the main toolbar, we can use the arrow buttons [◀] and [▶] to select a load case or combination in the table. We can also use the list in the table toolbar.

8.3 Filter Results



Results Navigator



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

We display the member internal force $\mathbf{M_y}$ in the *Results* navigator. We deactivate the display of the internal forces in surfaces as well as the values on surfaces (see figure on the left).

8.3.1 Visibilities

Partial views and sections can be used as so-called *Visibilities* in order to evaluate results.

Results display for concrete columns

We click the tab *Views* in the navigator. We select the following entries listed under *Generated*:

- Members sorted by type: *Beam*
- Members sorted by cross-section: *2 - Rectangle 300/300*

In addition, we create the intersection of both options with the [Show Intersection] button.

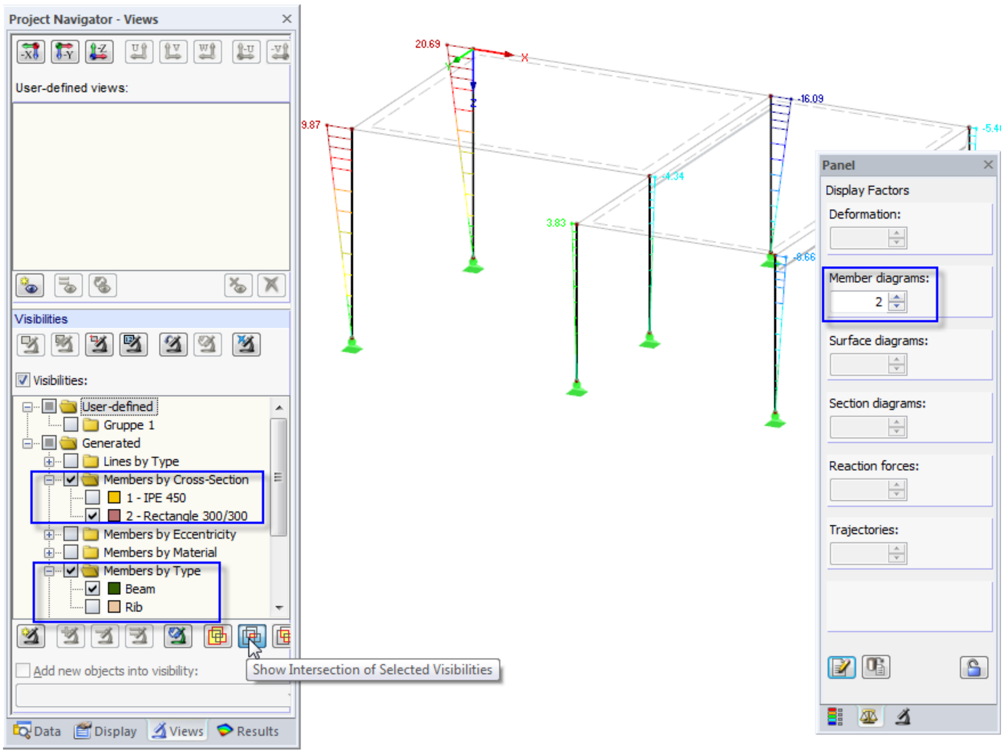


Figure 8.7: Moments M_y of concrete columns in scaled representation

The display shows the concrete columns including results. The remaining model is displayed lighter and without results.

Adjusting the scaling factor

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

Results display of floor slab

In the same way, we can filter surface results in the *View* navigator. We deactivate the options *Members by Type* and *Members by Cross-Section* and select *Surfaces by Thickness* where we select the entry *200 mm*.

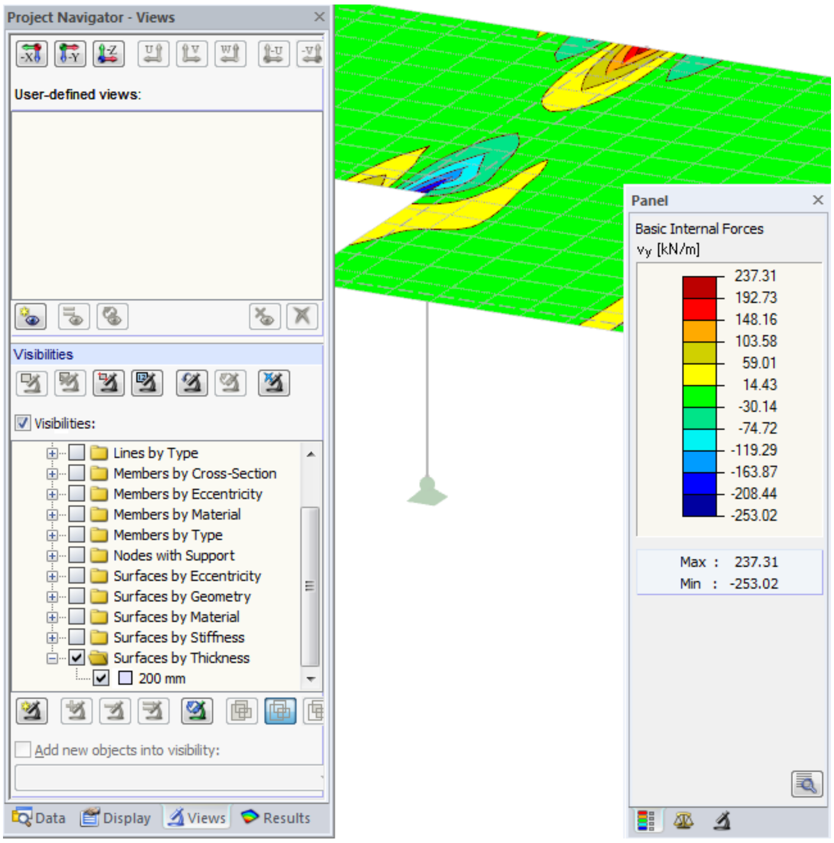


Figure 8.8: Shear forces of floor

As already described, we can change the display of result types in the *Results* navigator (see Figure 8.3, page 42). The figure above shows the distribution of the shear forces v_y for CO1.

8.3.2 Results on Objects

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

First, we clear the selection for *Visibilities* in the *Views* navigator.

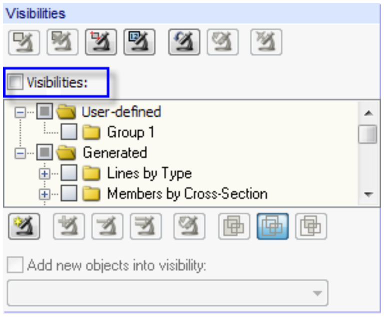


Figure 8.9: Resetting the overall view in *Views* navigator

We select surface 1 with one click. Then, in the panel, we change to the filter tab and check if *Surfaces* is selected.

We click the [Import from Selection] button and see that the number of the selected surface has been entered into the box above. Now, the graphic shows only the results of the left surface.

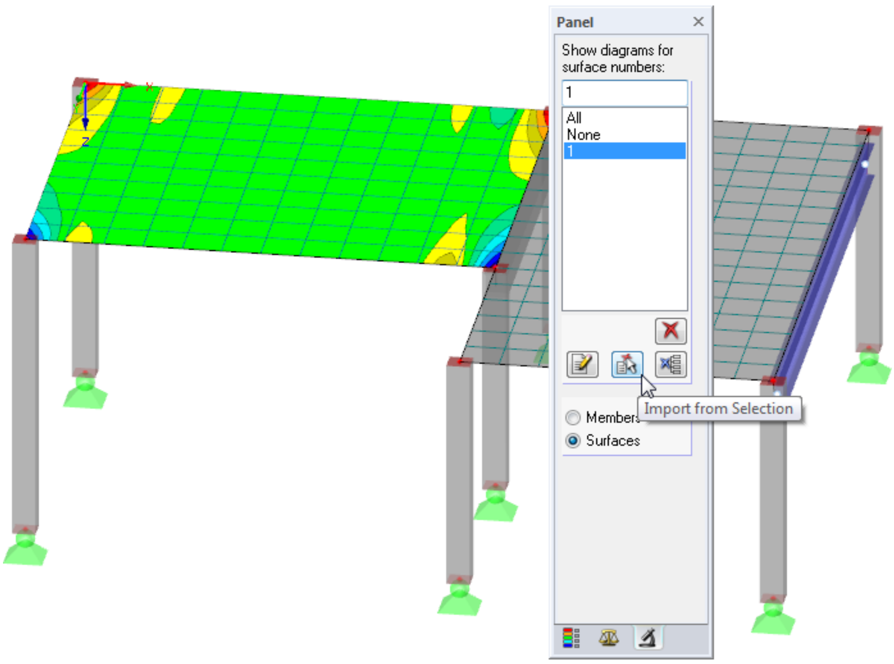


Figure 8.10: Shear force diagram of left surface

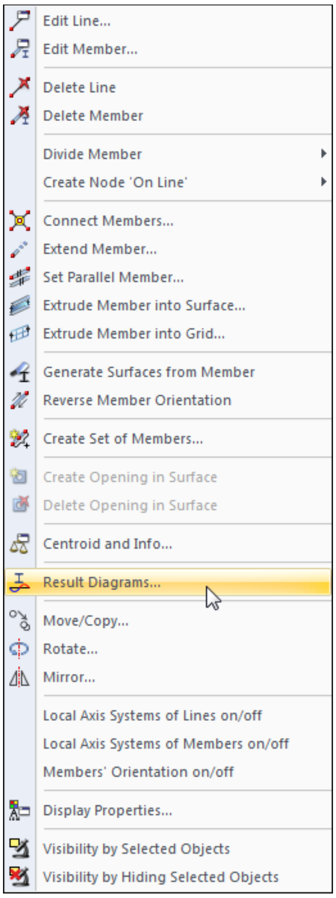
We use the panel option *All* to reset the full display of results.

8.4 Display of Result Diagrams

We can evaluate results also in a diagram available for lines, members, line supports and sections. Now, we use this function to look at the result diagram of the T-beam.

We right-click member *2* (when we have problems we can switch off the surface results) and select the option *Result Diagrams*.

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



Shortcut menu *Member*

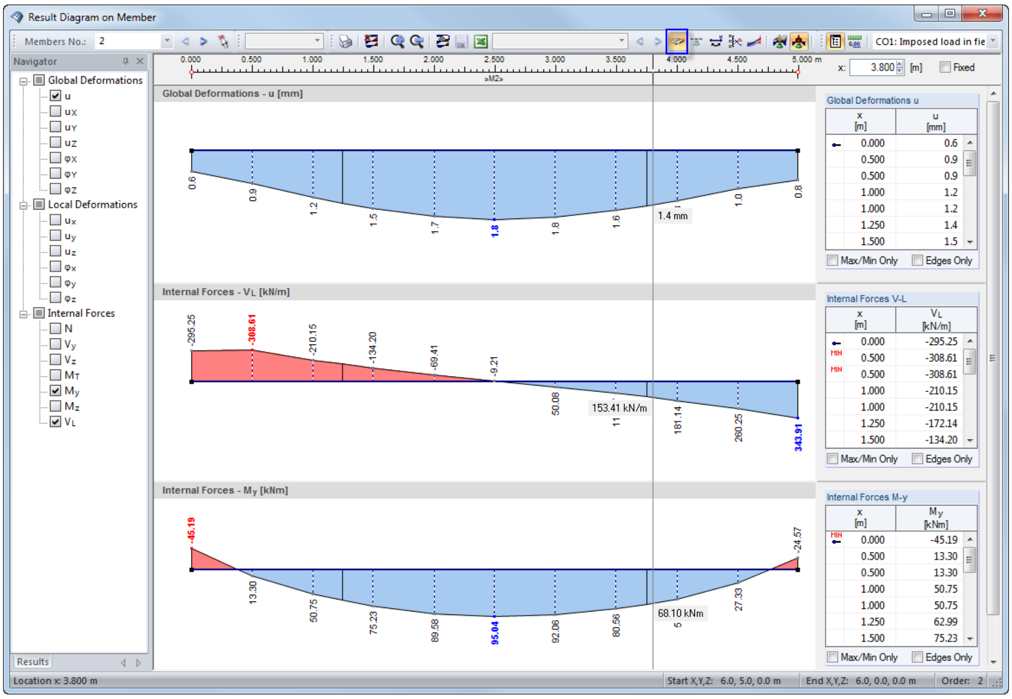


Figure 8.11: Display of result diagrams of downstand beam

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

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

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

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

9. Documentation

9.1 Creation of Printout Report

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

To open the printout report,

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

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

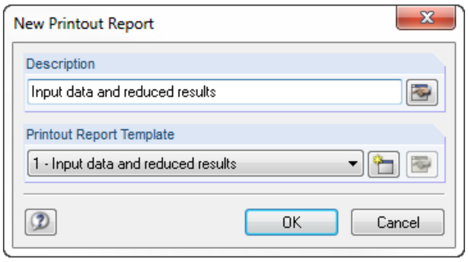


Figure 9.1: Dialog box *New Printout Report*

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

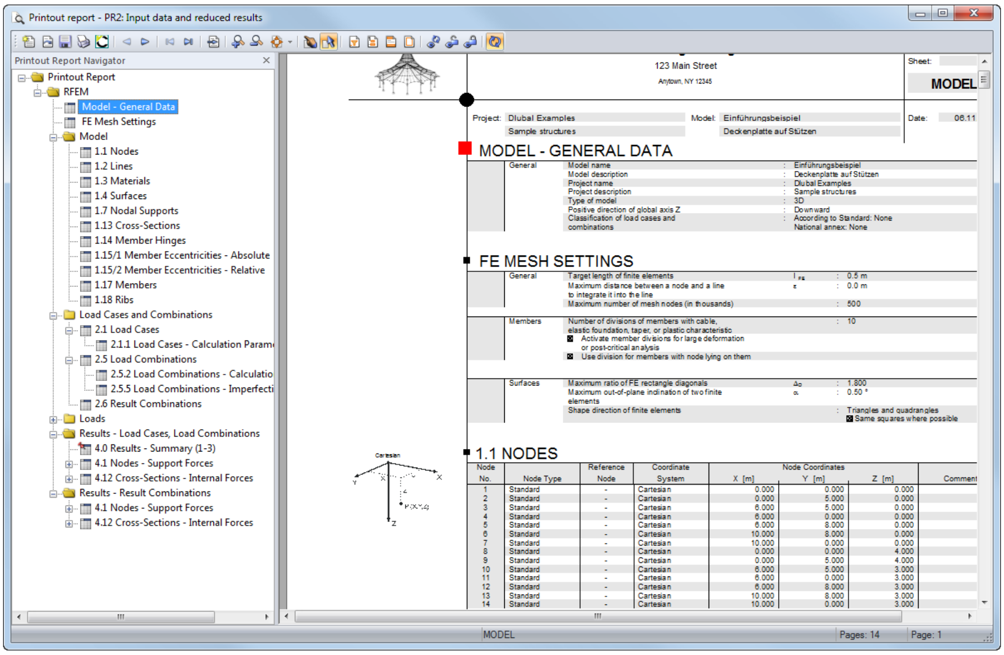


Figure 9.2: Print preview in printout report

9.2 Adjusting the Printout Report

Also the printout report has a navigator which lists the selected chapters. By right-clicking a navigator entry we can see the corresponding contents in the window to the right.

The default contents can be specified in detail. Now, we adjust the output of the member internal forces: Under *Results - Result Combinations*, we right-click *Cross-Sections - Internal Forces*, and then we click *Selection*.

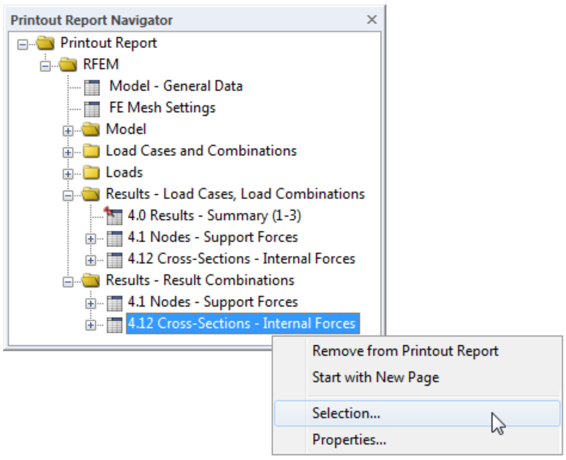


Figure 9.3: Shortcut menu *Cross-Section - Internal Forces*

A dialog box appears, offering detailed selection options for RC results of members.

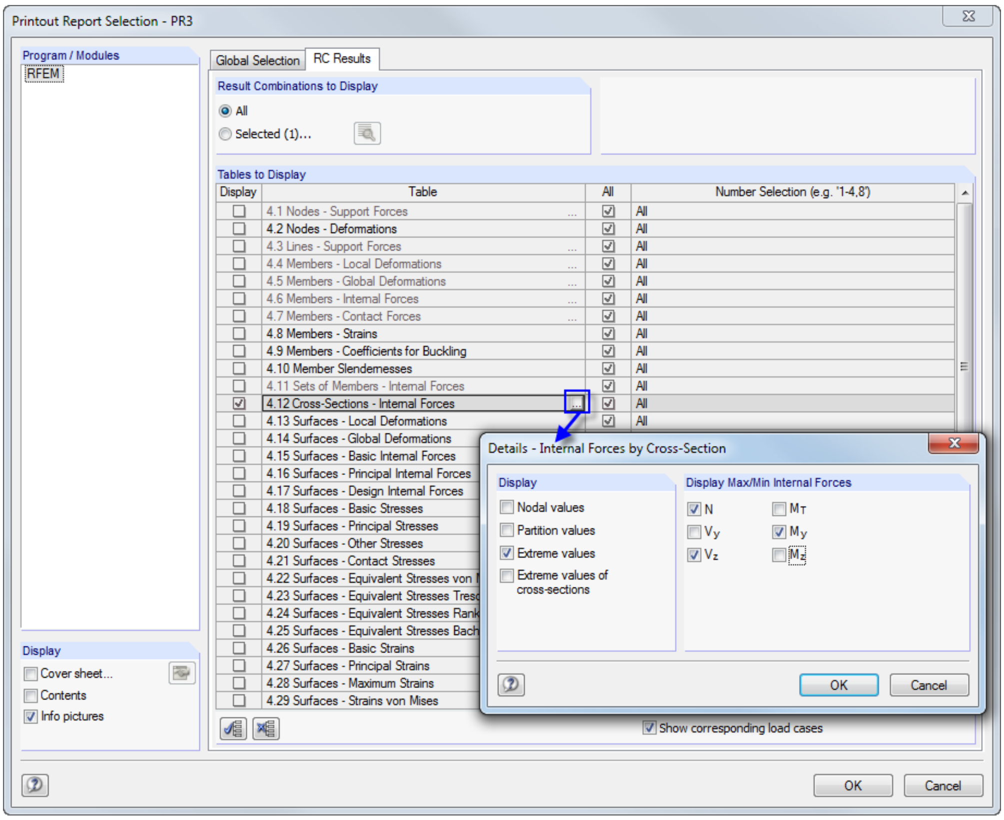


Figure 9.4: Reducing output of internal forces by means of *Printout Report Selection*

We place the pointer in table cell *4.12 Cross Sections - Internal Forces*. The [...] button becomes active which opens the dialog box *Details - Internal Forces by Cross-Section*. Now, we reduce the output to the **Extreme values** of the internal forces **N**, **V**$_z$ and **M**$_y$.

After confirming the dialog box we see that the table of internal forces has been updated in the printout report. We can adjust the remaining chapters for the printout in the same way.

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

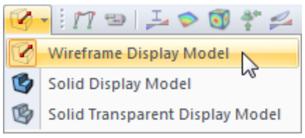
9.3 Inserting Graphics in Printout Report

Often, we integrate graphics in the printout to illustrate the documentation.

Printing deformation graphics

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

In the work window, we set the *Deformation* of **CO1 - Imposed load in area 1** and put the graphic in an appropriate position.



As deformations can be displayed more clearly as *Wireframe Display Model*, we set the corresponding display option.

Unless not already set, we change the display to *All* surfaces in the filter tab of the panel.

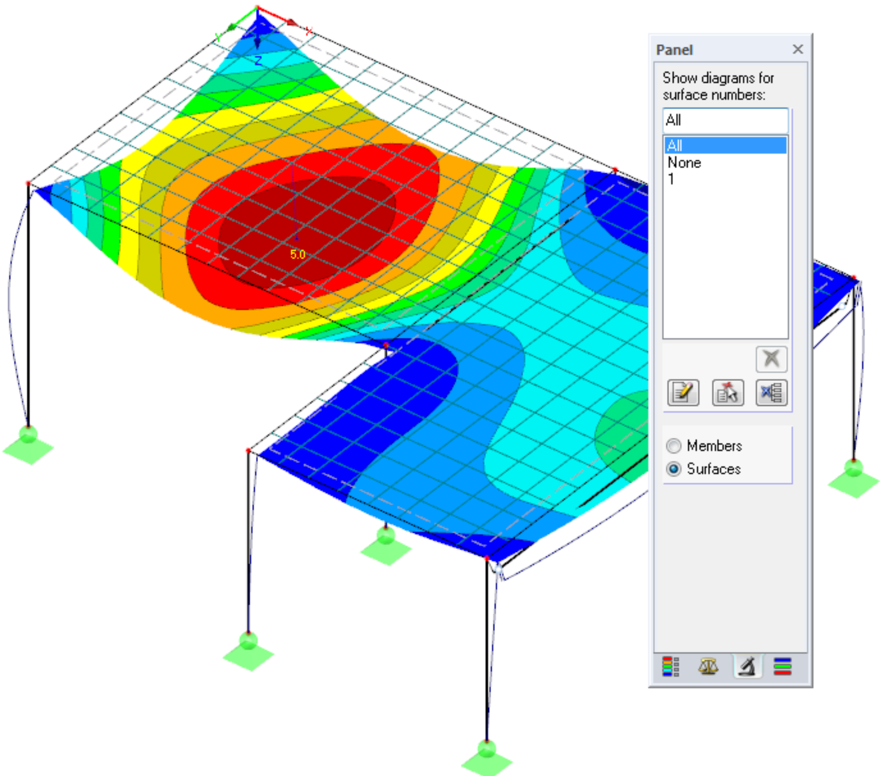


Figure 9.5: Deformations of CO1

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

We select **Print Graphic** on the **File** menu

or use the toolbar button shown on the left.

We set the following print parameters in the *Graphic Printout* dialog box. It is not necessary to change the default settings in the *Options* and *Color Spectrum* tabs.

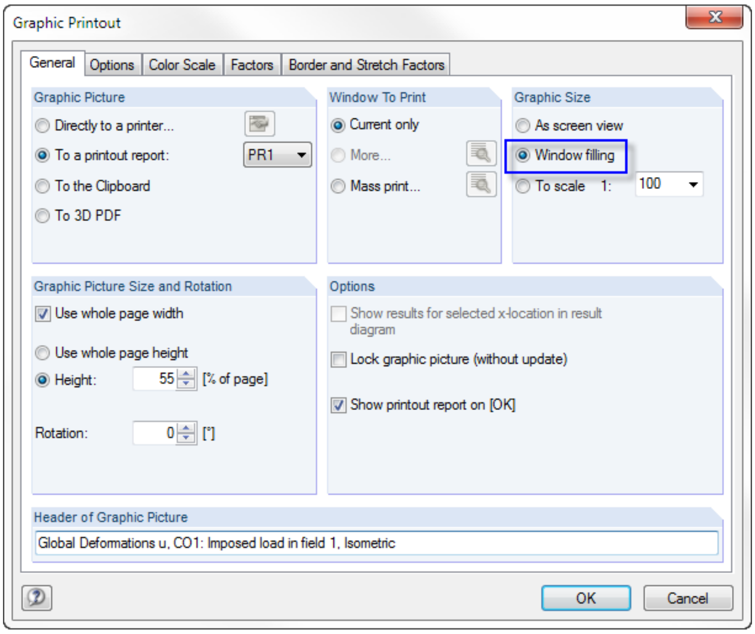


Figure 9.6: Dialog box *Graphic Printout*

We click [OK] to print the deformation graphic to the printout report.

The graphic appears at the end of chapter *Results - Load Cases, Load Combinations*.

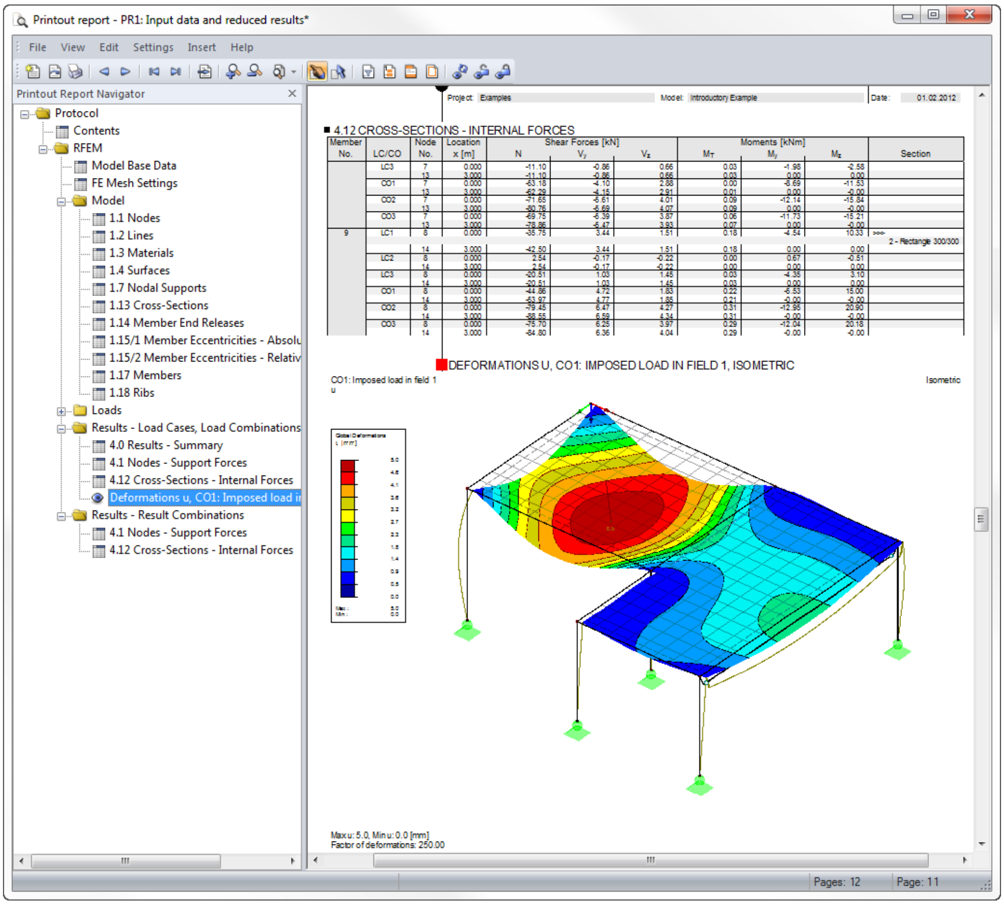


Figure 9.7: Deformation graphic in printout report

Printing the printout report

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

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

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

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

By clicking the [Save] button we create a PDF file with bookmarks. They facilitate navigating in the digital document.

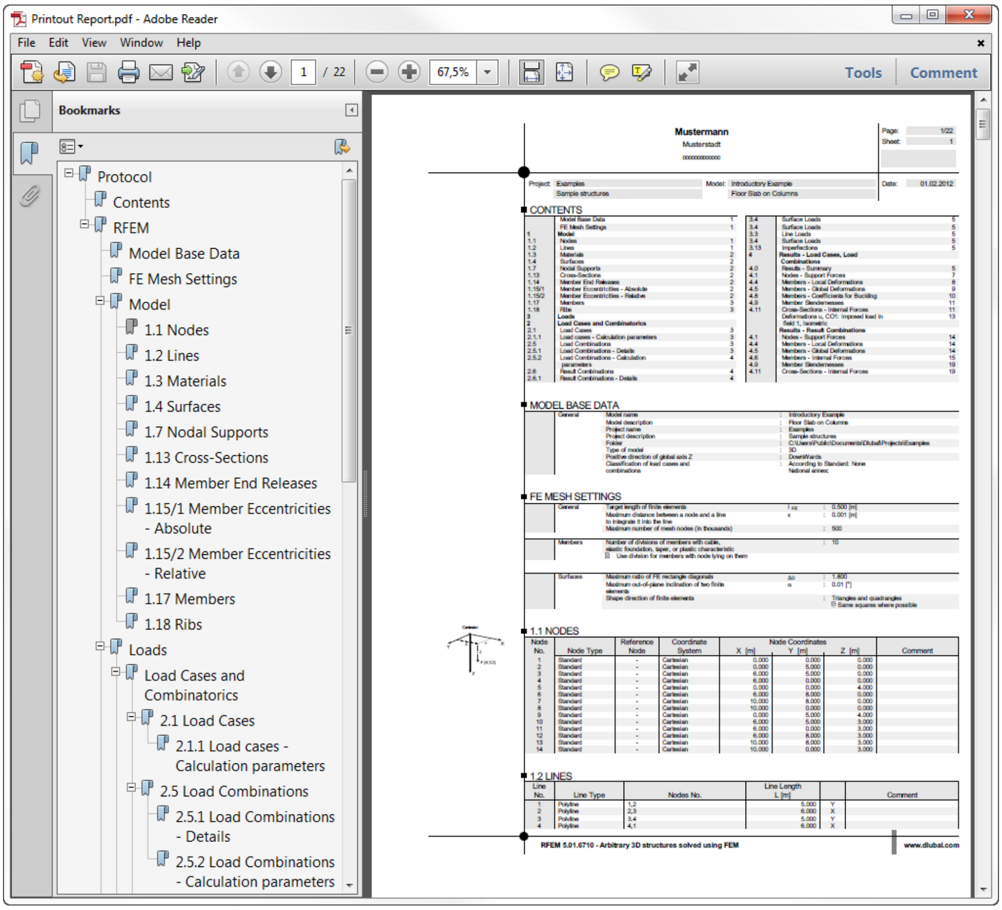


Figure 9.8: Printout report as PDF file with bookmarks

10. Outlook

Now, we have reached the end of the introductory example. We hope that this short introduction helps you to get started with RFEM and makes you curious to discover more of the program functions. You can find a detailed program description in the RFEM manual that you can download on our website at www.dlubal.com/downloading-manuals.aspx. On this download page, you can also find a tutorial example describing more comprehensive program functions.

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

Finally, if you have any questions, you are welcome to use our free e-mail hotline or to have a look at the FAQ page at www.dlubal.com or on our DLUBAL blogs at www.dlubal.com/blog.

Note: This example can be continued in the add-on modules, for example for steel and reinforced concrete design (RF-STEEL Members, RF-CONCRETE Surfaces/ Members etc.). In this way, you will be able to perform further design, getting an insight into the functionality of the add-on modules. Then, you can also evaluate the design results in the RFEM work window.