Dlubal

Version
October 2014

Program

RSTAB 8

Structural Analysis for
General Frameworks

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 this introductory example, we would like to make you acquainted with the most important features of RSTAB. 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 possibilities of the program. With this simple example we want to encourage you to find out useful functions in RSTAB.

We will model a planar two-hinged frame and the loads. Then we design the model according to nonlinear second-order analysis with regard to the following load cases: self-weight, snow, wind and imperfection.

As superimposing actions according to EN 1990 involves considerable time and effort, we will use the generator for load combinations already integrated in RSTAB 8.

With the 30-day trial version, you can work on the model without any restriction. Even after the expiry of the 30-day period, you can enter an example and calculate it; however, saving your results is no longer possible.

It is easier to enter data if you use two screens, or you may print the description to avoid switching between the displays of PDF file and RSTAB 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 RSTAB manual that you can download on the Dlubal website at http://www.dlubal.com/Downloading-Manuals.aspx.

The file **RSTAB-Example-06.rs8** containing all 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 RSTAB we recommend you to enter the model manually. If you do not have the time for it, you can watch the videos on our website at www.dlubal.com/videos-from-category-videos-for-rstab.aspx.

2. System and Loads

2.1 Sketch of System

In our example we analyze the steel frame of a hall that is reinforced by haunches.

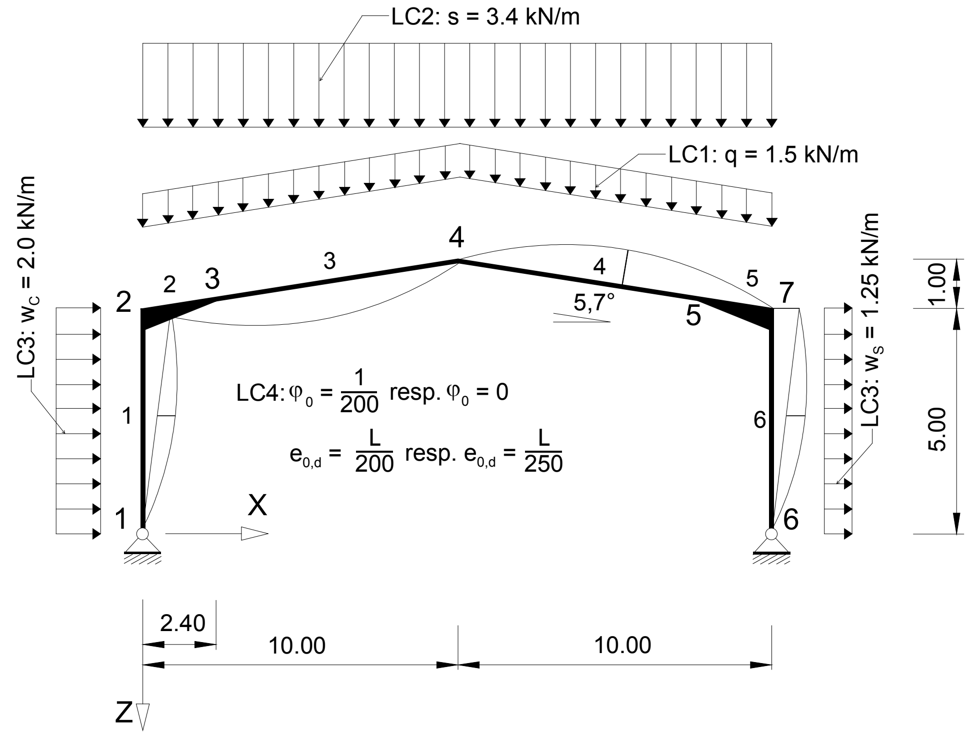


Figure 2.1: System and loading

2.2 Materials and Cross-sections

We use the standard steel S 235.

Both columns consist of cross-sections of the type HE-A 300. The horizontal beams are cross-sections of the type IPE 360.

The taper is made of a metal sheet reinforcement with a thickness of 10 mm, reaching a height of 300 mm on the column connection.

2.3 Load

Load case 1: self-weight and finishes

As load, the framework's self-weight (steel cross-sections) is to be considered that RSTAB determines automatically from the cross-section properties.

In addition, the hall's roof structure is applied with 0.3 kN/m². For a frame distance of 5 m, the load is:

$q = 0.3\ kN/m^2 * 5\ m = 1.5\ kN/m$

There are no wall loads transferred by the columns.

Load case 2: snow

For the hall we apply snow load zone 2 in Germany and an altitude of 200 m. According to EN 1991-1-3, we have to apply the characteristic value $s_k = 0.85\ kN/m^2$. Together with the shape coefficient $\alpha = 0.8$ for the roof inclination of 5.7° we get a snow load of:

$q = 0.8 * 0.85\ kN/m^2 * 5\ m = 3.4\ kN/m$

Load case 3: wind

To simplify our example, we apply the wind loads of wind zone 1 and terrain category III on the columns only, neglecting the roof areas. The wind loads are determined according to EN 1991-1-4 and the national annex (NA) for Germany with the simplified velocity pressure $q_p = 0.5\ kN/m^2$. So we get the following wind pressure on the upwind side:

$w_D = 0.8 * 0.5\ kN/m^2 * 5\ m = 2.0\ kN/m$

For the downwind side we have the following suction load:

$w_S = 0.5 * 0.5\ kN/m^2 * 5\ m = 1.25\ kN/m$

Load case 4: imperfection

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

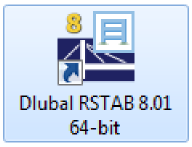
For the column cross-sections (HE A 300) we assume the buckling curve *c* (displacement in direction of z-axis) according to EN 1993-1-1, Table 6.2. The inclinations φ_0 and precambers $e_{0,d}$ are determined according to EN 1993-1-1, Clause 5.3.2 as follows:

Inclination $\varphi_0 = 1/200$
Precamber $e_{0,d} = L/200$

As the buckling curve *b* is valid for IPE 360 cross-sections, we can apply a smaller precamber of $e_{0,d} = L/250$ for the horizontal beams. For the beam, no inclination is assigned.

3. Creation of Model

3.1 Starting RSTAB



To start RSTAB in the taskbar, we

click **Start**, point to **All Programs** and **Dlubal**, and then we select **Dlubal RSTAB 8.xx**,

or we double-click the icon **Dlubal RSTAB 8.xx** on the computer desktop.

3.2 Creating the Model

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

If RSTAB 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.

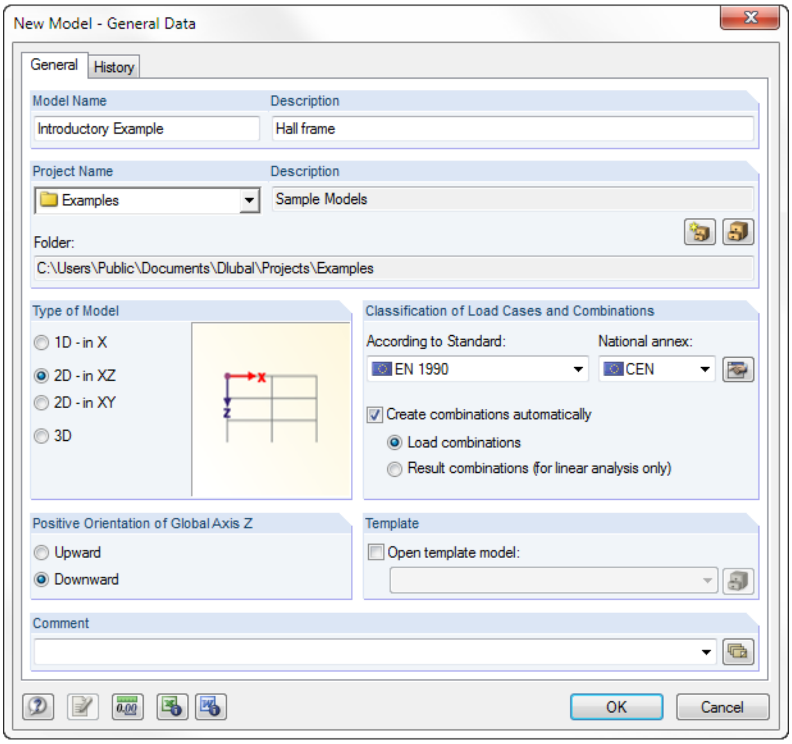


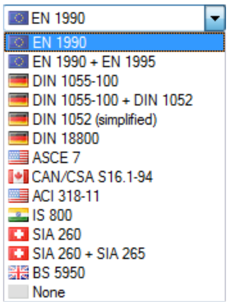
Figure 3.1: Dialog box *New Model - General Data*

We write **Introductory Example** into the input field *Model Name*. To the right, we enter **Hall frame** as *Description*. We always have to define a *Model Name* because it determines the name of the RSTAB file. The *Description* field does not necessarily need to be filled in.

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

In the dialog section *Type of Model*, we select the option **2D - in XZ** because we want to model a planar framework. It is always possible to extend the model to a 3D framework structure later.

We keep the default setting **Downward** for *The Positive Orientation of Global Axis Z*.



In the dialog section *Classification of Load Cases and Combinations*, we select the entry **EN 1990** from the list *According to Standard*. We don't change the setting **CEN** in the field *National annex* to the right. This specification is important when we combine actions with partial safety factors and combination coefficients conforming to standards.

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

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

The empty work window of RSTAB is displayed.

4. Model Data

4.1 Adjust Work Window and Grid

View

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

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

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.

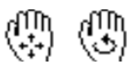
To exit the function, different ways are possible:

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

Mouse functions

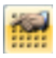
The mouse functions follow the general standards for Windows applications. To select an object for further editing, we click it once with the **left** mouse button. We double-click the object when we want to open its dialog box for editing.

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



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

Grid



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

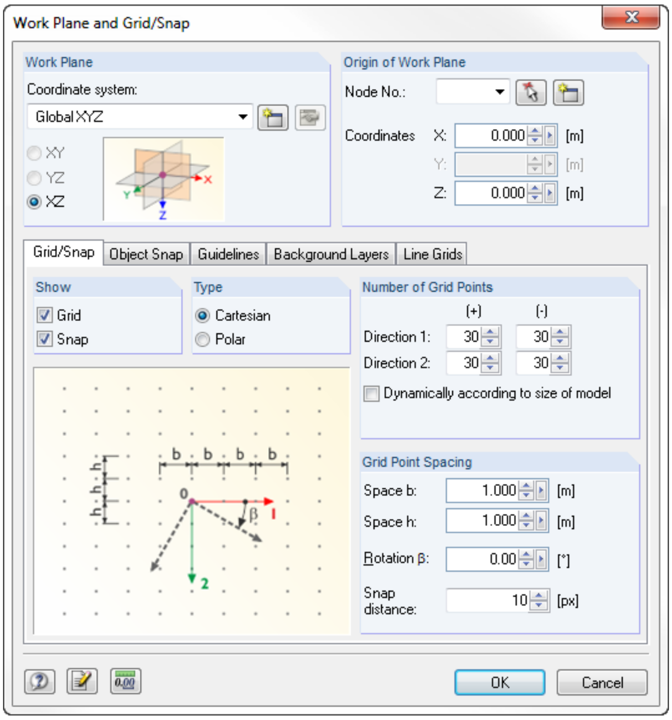
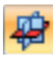


Figure 4.1: Dialog box *Work Plane and Grid/Snap*

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

Work plane



The XZ plane is preset as work plane. With this setting all graphically entered objects will be generated in the vertical plane. The plane has no significance for the data input in dialog boxes or tables.

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

4.2 Creating Members

It would be possible to define the nodes graphically or in the table to connect them with members subsequently. But in our example we use the direct graphical input of members.

4.2.1 Columns

Before we can set a member in the work window, we have to define its properties. To open the corresponding dialog box,

we click **Model Data** on the **Insert** menu, then we point to **1.7 Members** and **Graphically**, and then we select **Single**,

or we use the corresponding button in the toolbar, which is the quicker way of selecting the function.

The dialog box *New Member* opens.

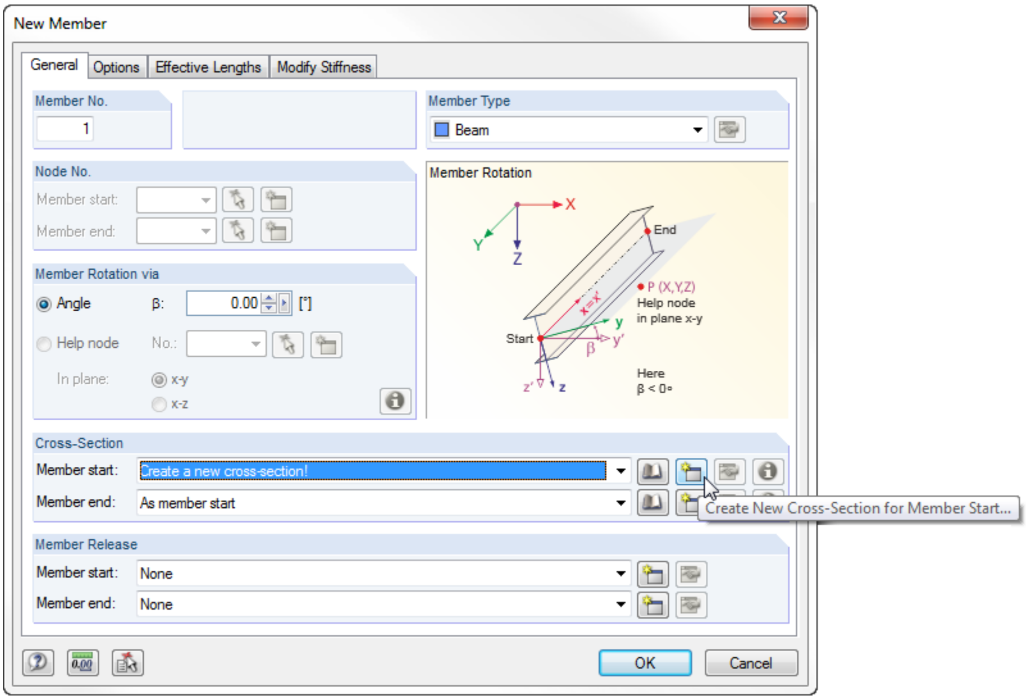


Figure 4.2: Dialog box *New Member*

The *Member No.* **1** and the *Member Type* **Beam** are already preset. It is not necessary to change these settings.

Defining a cross-section

In the dialog section *Cross-Section*, we click the button [New] to assign a stiffness for material and cross-section to the *Member start*. The dialog box *New Cross-Section* appears (see Figure 4.3 in background).

As the columns consist of rolled cross-sections, we click the button [HE-A] in the dialog box. The dialog box *Rolled Cross-Sections - I-Sections* appears where we select the cross-section **HE A 300**.

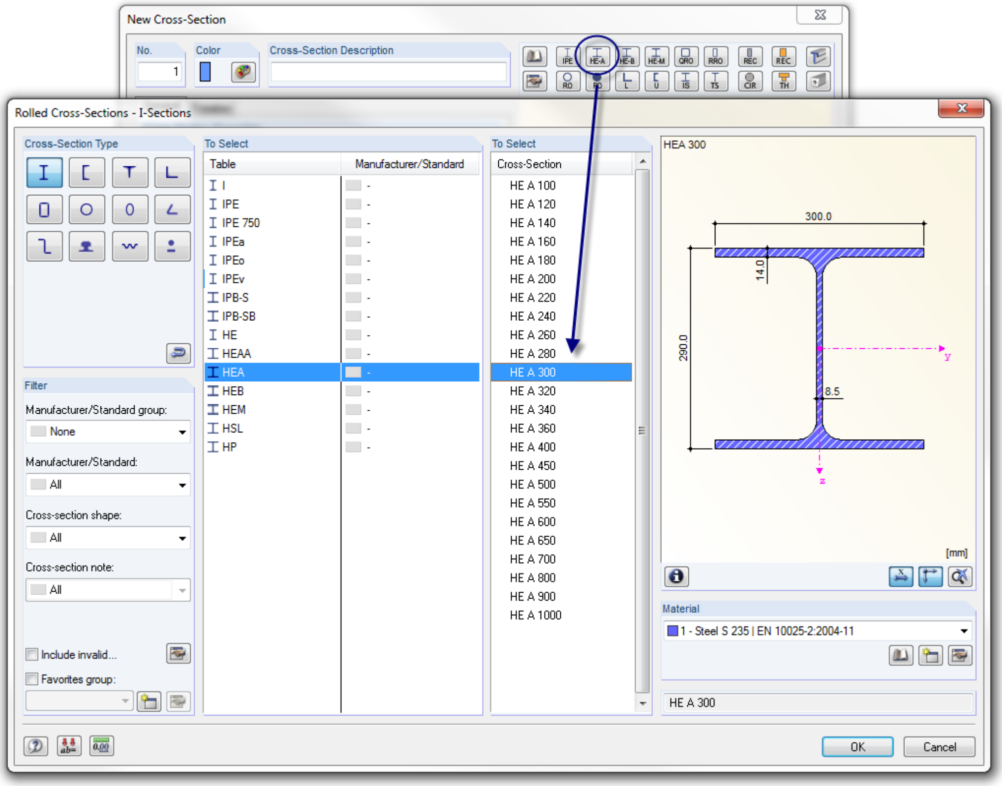


Figure 4.3: Selecting HE A 300 in the library

To check the properties of the cross-section, we can use the [Info] button.

We close the dialog box *Rolled Cross-Sections - I-Sections* to import the cross-section properties to the dialog box *New Cross-Section*. As our framework is a 2D model, only relevant input fields are filled with data in this dialog box.

For rolled cross-sections RSTAB presets the *Material* with number *1 - Steel S 235* according to EN 10025-2. When we want to use a different material, we can select another one by means of the [Material Library] button.

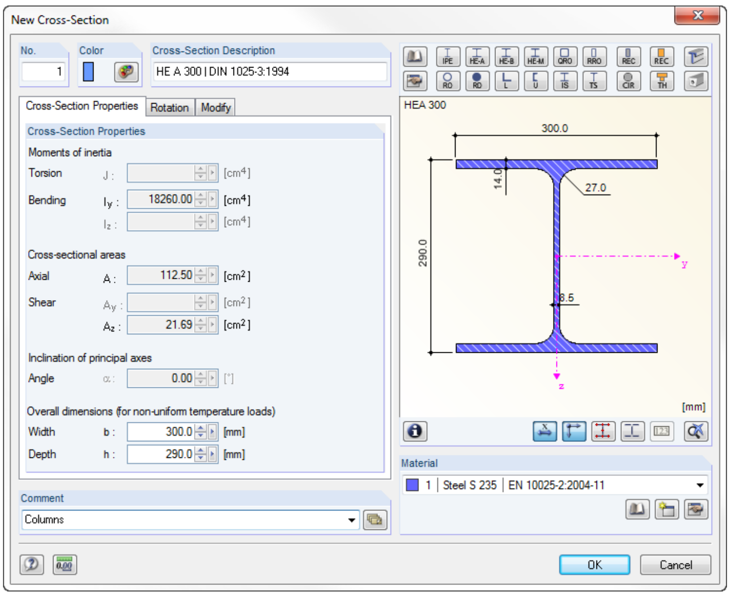


Figure 4.4: Dialog box *New Cross-Section*

In the *Comment* field, we can enter **Columns** to specify the cross-section.

We close the dialog box by clicking [OK] and return to the initial dialog box *New Member*.

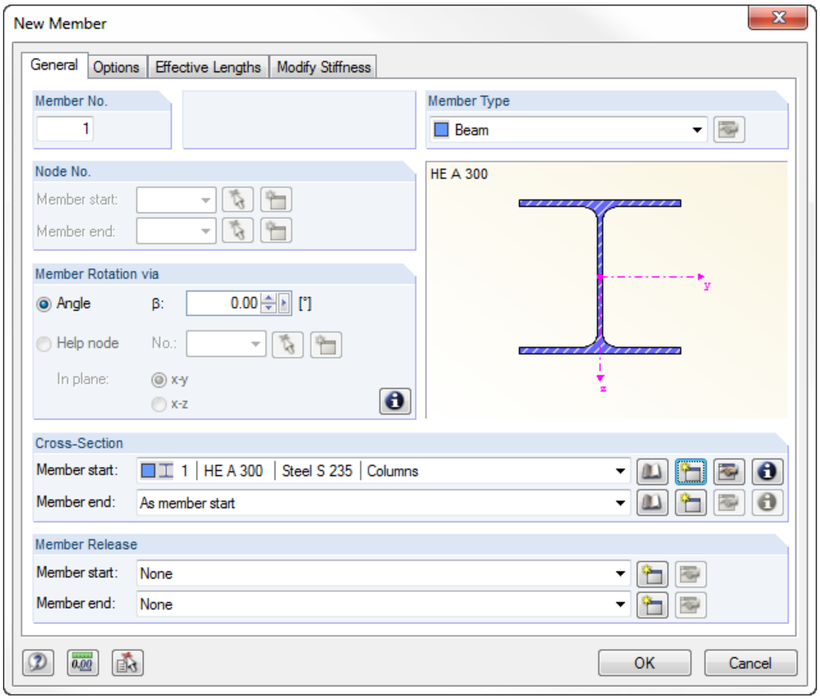


Figure 4.5: Dialog box *New Member*

When we have checked the input fields, we close the dialog box with the [OK] button. Now we can set the column members.

Placing members

Members can be set directly by clicking grid points or nodes. If the start or end points of the member do not lie within the grid that has been set, we can enter the coordinates in the floating dialog box *New Member* manually. Please take care not to move with the pointer outside the dialog window, otherwise its coordinates are considered again for the input. Furthermore, we can switch between the input fields by using the mouse or the [Tab] key on our keyboard. Instead of clicking the [Apply] button used to set a node, we can also use the [Enter] key.

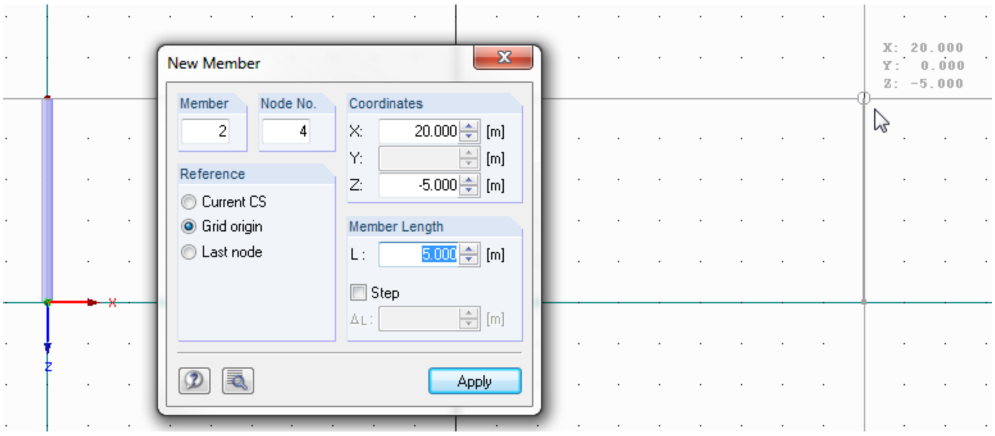


Figure 4.6: Defining columns graphically

We define the support node of the first column by a single click with the left mouse button into the **coordinate origin** (X/Y/Z coordinates **0.000/0.000/0.000**). The current pointer coordinates are displayed next to the reticle.

The top end of the column is set into the grid point **0.000/0.000/-5.000**.

We continue our input and define member 2 by clicking the grid points **20.000/0.000/0.000** for the footing of the column and **20.000/0.000/-5.000** for the head.

When the grid points lie outside the workspace, we can zoom out the view by using the wheel button of the mouse. It is also possible to shift the view slightly to the left by holding down the wheel button (see also chapter 4.1, page 8). The command for placing members will not be affected.

For now we stop setting other members. We quit the input mode by pressing the [Esc] key. We can also use the right mouse button to right-click in an empty area of the work window.

Show numbering

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

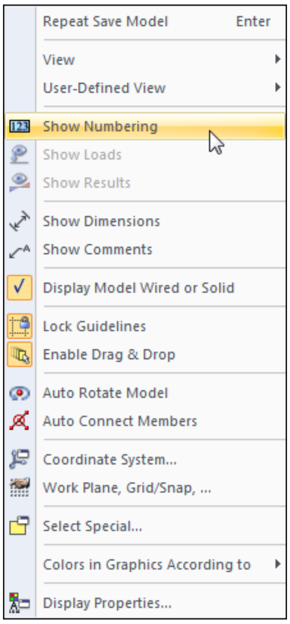


Figure 4.7: Show numbering in context menu

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

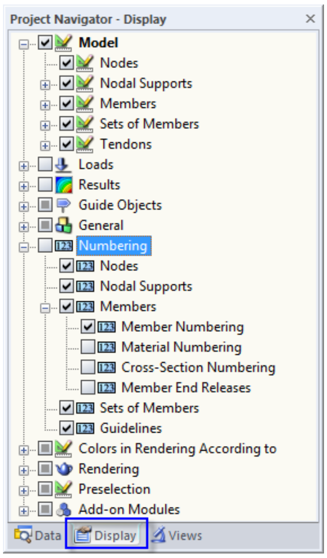


Figure 4.8: *Display* navigator for numbering

4.2.2 Horizontal Beams

Placing members

As the horizontal beams are connected to each other, we can define them as a polygonal chain. To open the corresponding dialog box,

> we click **Model Data** on the **Insert** menu, then we point to **1.7 Members** and **Graphically**, and then we select **Continuous**,

or we use the corresponding button in the toolbar shown on the left.

The dialog box *New Member* opens again. We create a [New] cross-section as cross-section number **2** and select an **IPE 360** section from the IPE cross-section table.

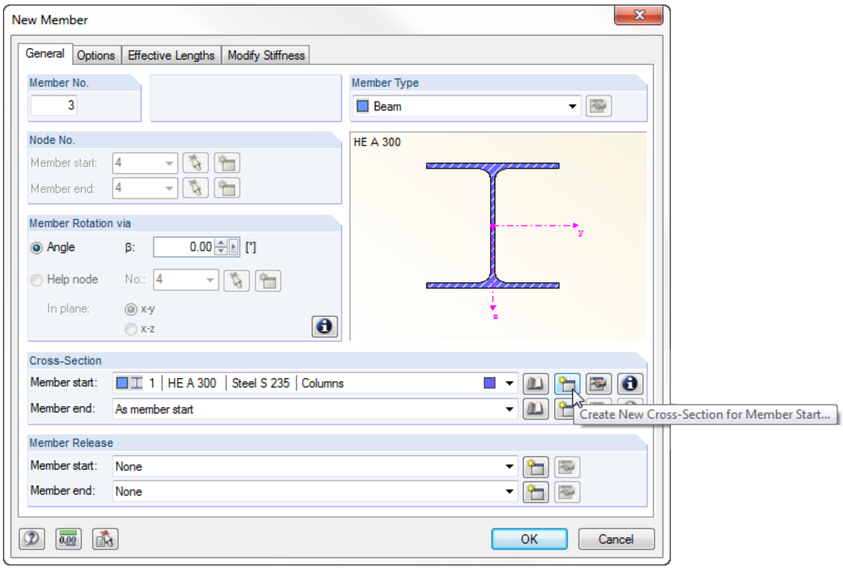


Figure 4.9: Dialog box *New Member*

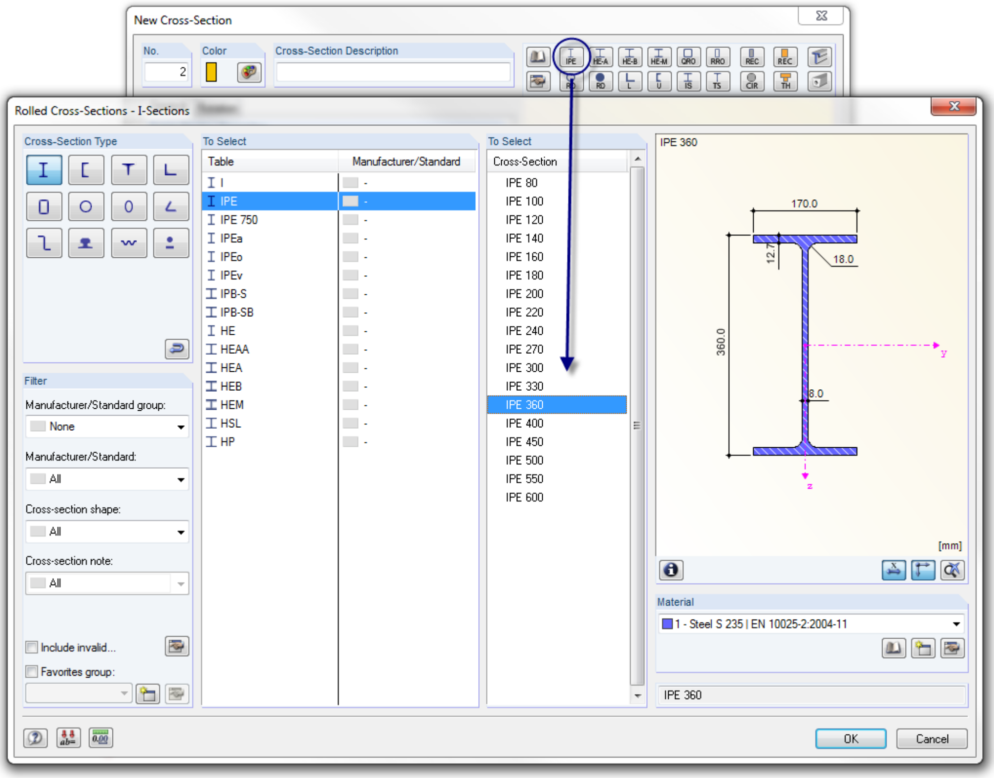


Figure 4.10: Selecting IPE 360 in the library

When we have confirmed all dialog boxes, we set the first horizontal beam by clicking the following points: We start with node **2** (0.000/0.000/-5.000) that is already defined. Then, we set the member end into the grid point **10.000/0.000/-6.000** (node of ridge). Finally, we click node **4** (**20.000/0.000/-5.000**) to finish the input.

By clicking these three points RSTAB creates the members 3 and 4. We quit the input mode by pressing the [Esc] button twice. We can also right-click two times into in an empty space of the work window.

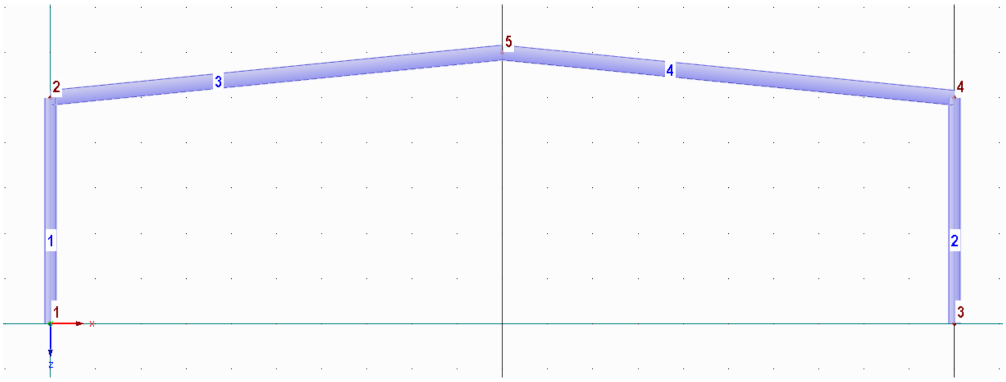


Figure 4.11: Horizontal beams 3 and 4

Dividing members

The entered model does not yet correspond completely to the specified system. The reinforcements for the frame joints are still missing.

First, we divide the horizontal beams by intermediate nodes to define the reinforcement zones. We click the left beam with the right mouse button. The member context menu opens where we select **Divide Member** by **Distance**.

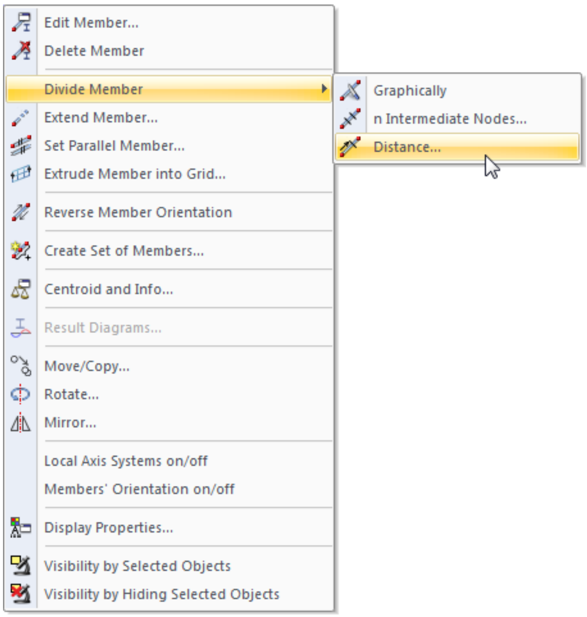


Figure 4.12: Context menu of member

In the dialog box *Divide Member Using Distance*, we change the reference of the distance to **Projection in direction X**. Then, we enter the value **2.40** m (length of taper) into the input field *Distance Between New Node and Member start*.

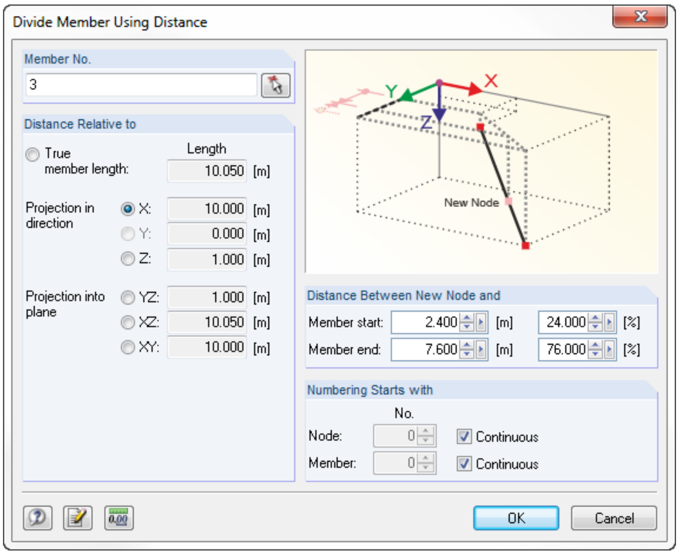


Figure 4.13: Dialog box *Divide Member Using Distance*

When confirming with [OK], RSTAB divides the left horizontal beam. Member 5 is created.

Now, we divide the right beam in the same way. This time we define the distance of **2.40** m for the *Member end*.

Defining reinforcements for horizontal beams

We create the beam reinforcement with a gusset. We define two new cross-sections and assign them to the members. In this way, we can model a taper: The cross-section depth of the member is changing linearly along the member length.

Horizontal beam - left

We double-click member *3* to open the dialog box *Edit Member*.

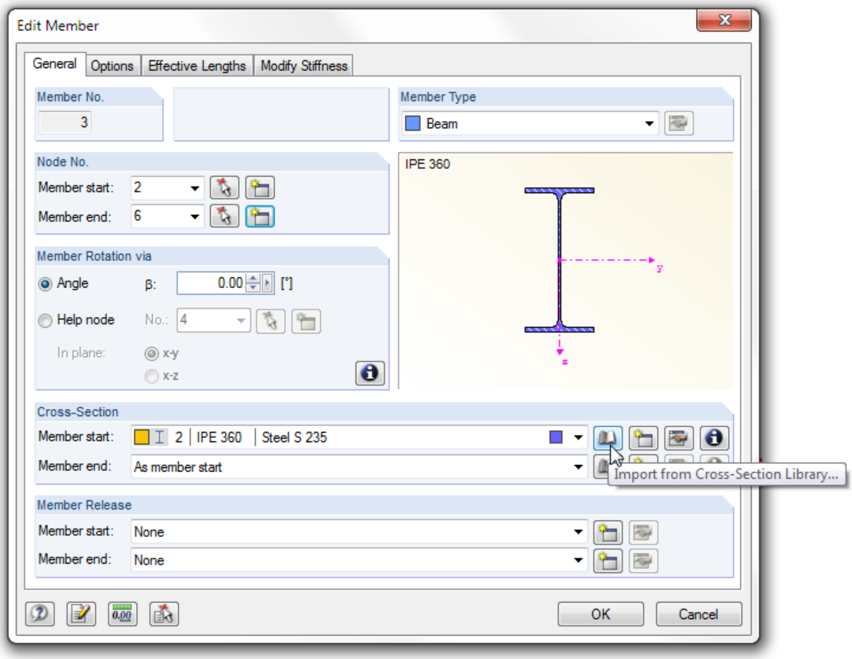


Figure 4.14: Dialog box *Edit Member*

In the dialog section *Cross-Section*, we select a new cross-section for the *Member start* from the [Library].

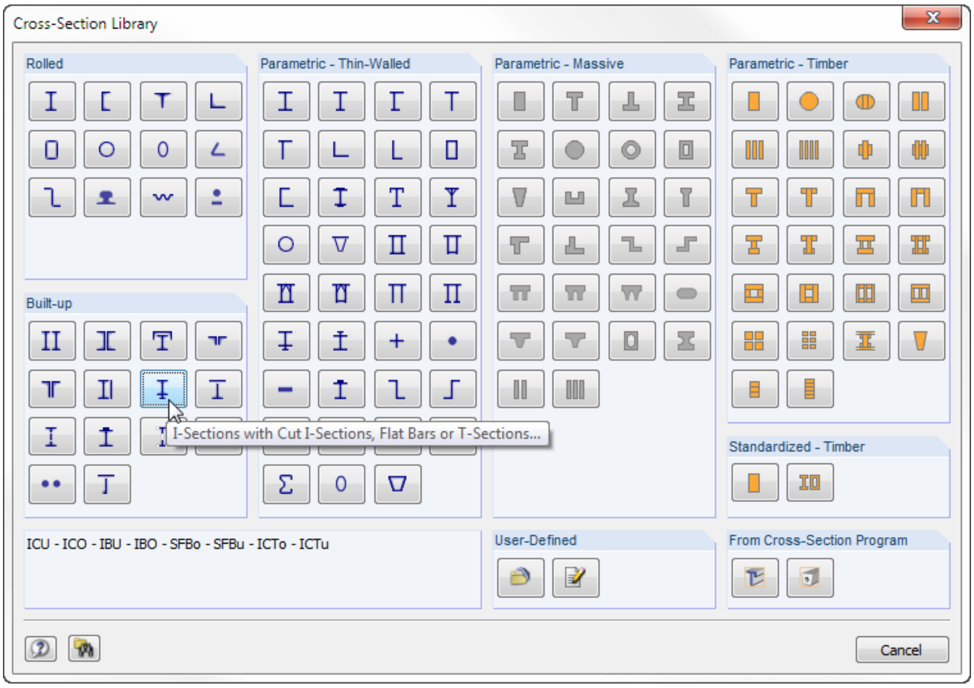


Figure 4.15: Cross-Section Library

In the *Built-up* dialog section, we click the button [I-Sections with Cut I-Sections, Flat Bars or T-Sections].

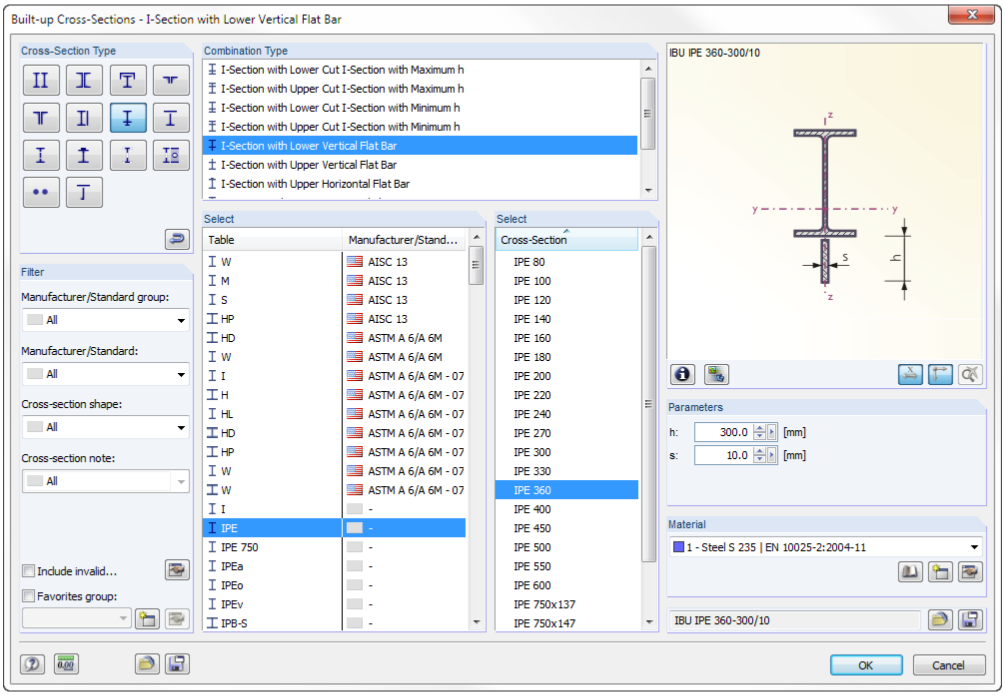


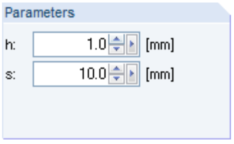
Figure 4.16: Dialog box *Built-up Cross-Sections - I with Lower Vertical Flat Bar*

We select the *Combination Type* **I-Section with Lower Vertical Flat Bar**. Then, we set the *Table* **IPE** and select the *Cross-Section* **IPE 360**.

For the reinforcing sheet we enter the following parameters on the right:

- Depth *h*: **300** mm
- Thickness *s*: **10** mm

After clicking the [OK] button we can see the cross-section properties displayed in the dialog box *New Cross-Section*. We also confirm the input in this dialog box and return to the initial dialog box *Edit Member*.



We repeat the procedure for the *Member end*: We click the [Library] button, select the option [I-Sections with Cut I-Sections, Flat Bars or T-Sections] again and define the combined cross-section *I-Section with Lower Vertical Flat Bar* as shown in Figure 4.16. This time, however, we enter **1** mm for the depth *h*. RSTAB is able to calculate the taper by interpolating the cross-section properties between both end cross-sections.

After confirming the input with the [OK] button the dialog box *Edit Member* looks as follows.

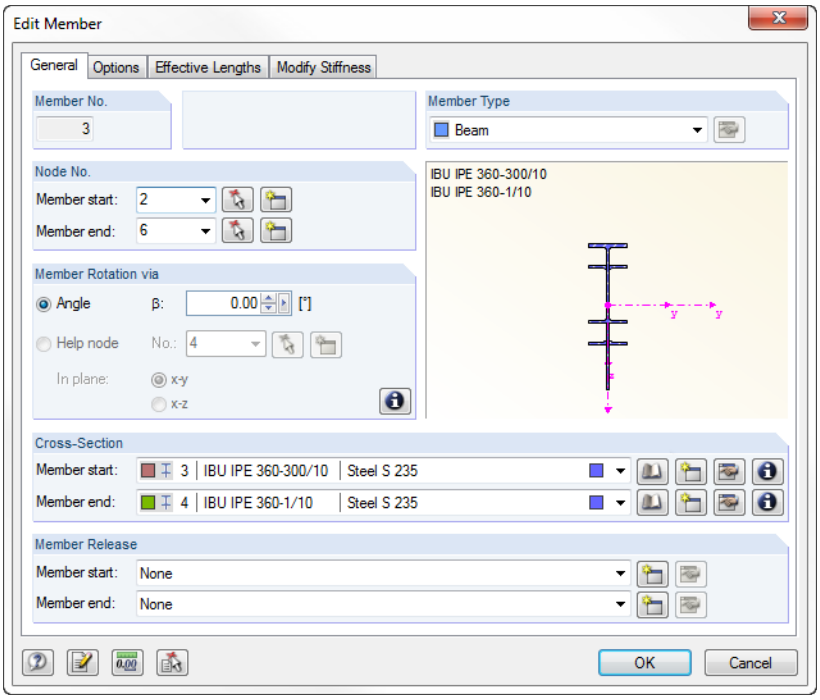


Figure 4.17: Dialog box *Edit Member*

We confirm the dialog box, and then we see the gusset displayed as a taper on the model. The cross-section's inclination in the rendered view (slightly curved) results from shifting the centroid that is always located on the system line.

Horizontal beam - right

Now, we double-click member 6 to assign another taper. As the cross-sections are already defined, we can select them from the list in the dialog box *Edit Member*. We set cross-section **4** for the *Member start*, for the *Member end* we set cross-section **3**.

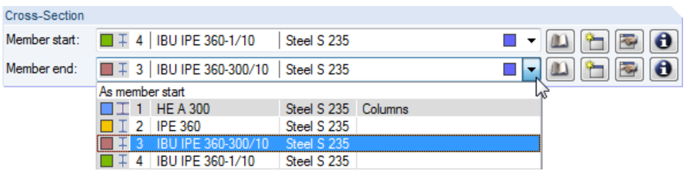


Figure 4.18: Selecting cross-sections from the list

We confirm the dialog box with the [OK] button, and then we click in an empty space of the work window to cancel the selection of member 6.

4.3 Connecting Members Eccentrically

We connect the tapered members eccentrically to the columns in order to determine the additional moments due to eccentrical load introduction. Therefore, we shorten the system line by half of the columns' cross-section height.

Taper - left

We double-click the left taper (member 3). In the dialog box *Edit Member*, we change to the dialog tab *Options*. In the dialog section *Member Eccentricity*, we click the [New] button to open the dialog box *New Member Eccentricity*.

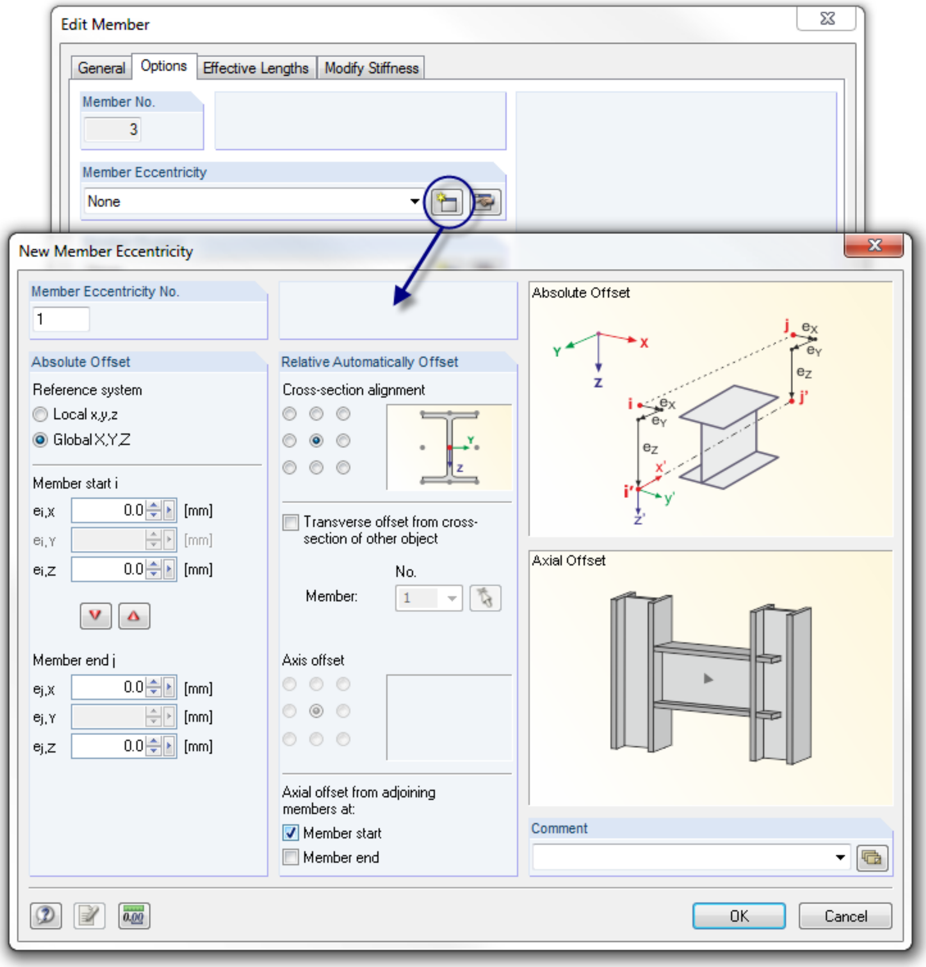


Figure 4.19: Dialog box *New Member Eccentricity*

In the dialog section *Axial offset from adjoining members*, we tick the check box for **Member start**.

In the dialog section *Absolute Offset*, we could define an eccentricity $e_{i,Z}$ to compensate the difference of the centroidal distances. In this way, it would be possible to repair the mentioned "slight curve" displayed in the rendering. But we do without this definition because this eccentricity has almost no influence on the results.


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

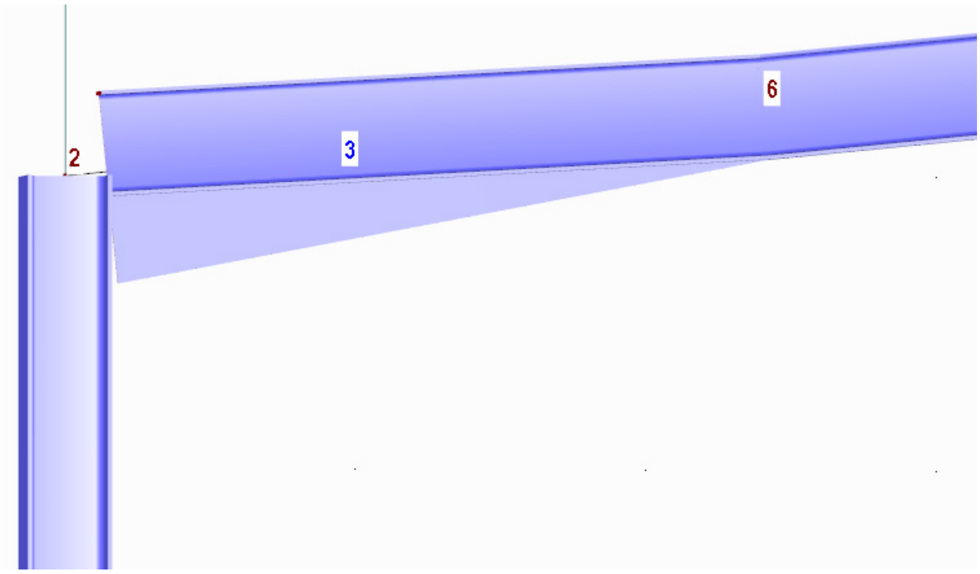


Figure 4.20: Eccentric connection in zoomed view

Taper - right

The eccentricity of the second tape is defined in the same way: We double-click member *6* and create a [New] member eccentricity.

This time we tick the check box for **Member end** in the dialog section *Axial offset from adjoining members*.

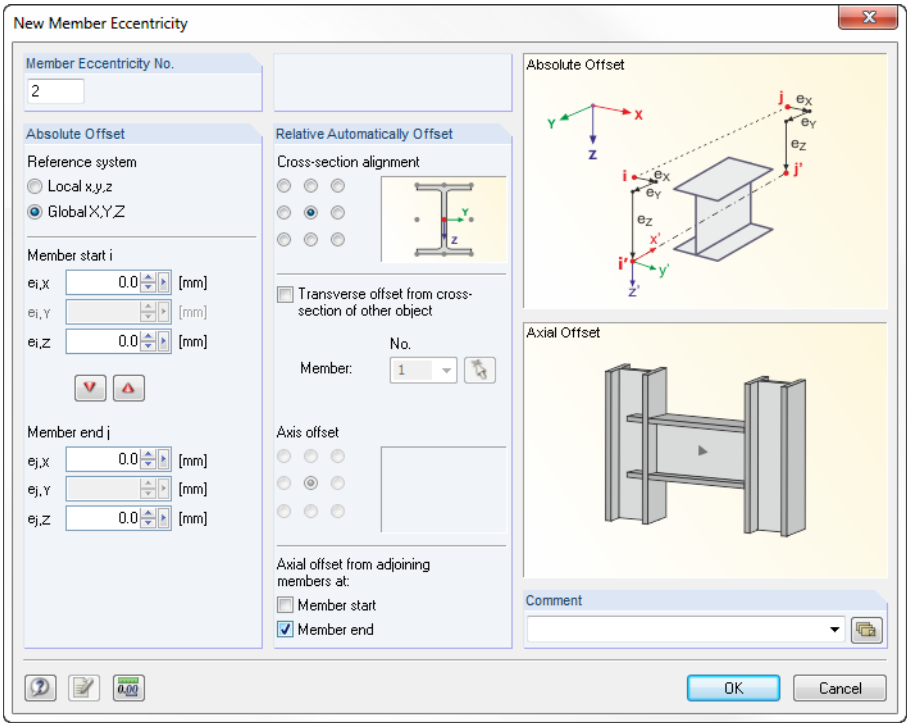


Figure 4.21: Dialog box *New Member Eccentricity*

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

Changing the view

We use the button [Isometric View] in the toolbar to set the spatial full view of the model.

Adjusting the color assignment

The *Display* navigator provides an option to display *Colors According to* particular criteria.

The default setting is the display of material colors. With the option *Cross-Sections* we can distinguish the different cross-section types at a glance.

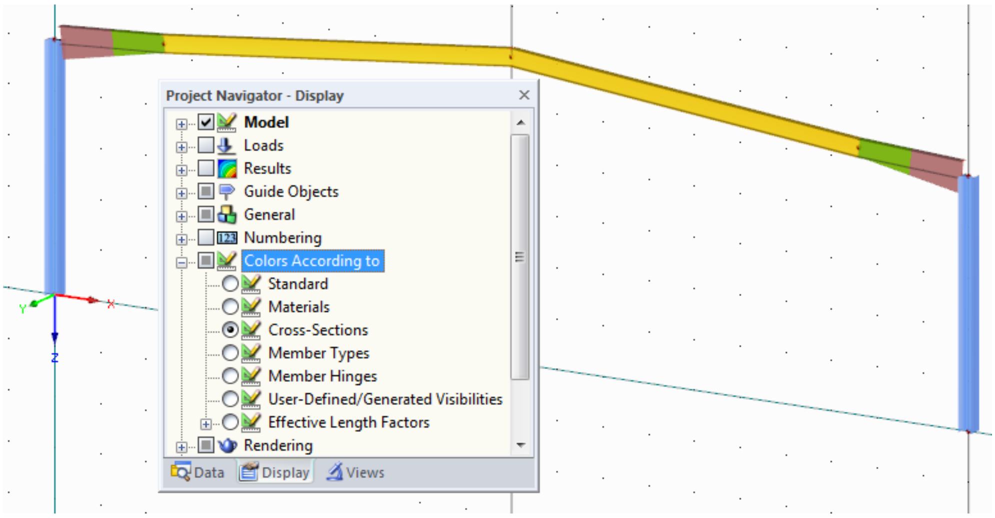


Figure 4.22: Distinguishing cross-sections by colors

RSTAB assigns the taper colors for each start and end cross-section to the center of the member.

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

4.4 Creating Sets of Members

Members can be combined in sets of members. In this way, it is easier to apply loads and evaluate results. RSTAB distinguishes between *continuous members* with continuously connected members and *groups of members* with arbitrarily arranged members.

We create two continuous members, each from the horizontal beams of one roof side. To open the dialog box below, we select **Model Data** on the **Insert** menu, point to **1.11 Sets of Members** and click **Dialog Box**.

In the *Description* field, we enter **Horizontal beam left** and select **Continuous members** in the *Type* dialog section. We use the [↖] button to select the horizontal beams **3** and **5** one after the other by mouse-click in the graphical work window. After confirming the selection by clicking the [OK] button in the *Set of Members* window, the dialog box looks as follows.

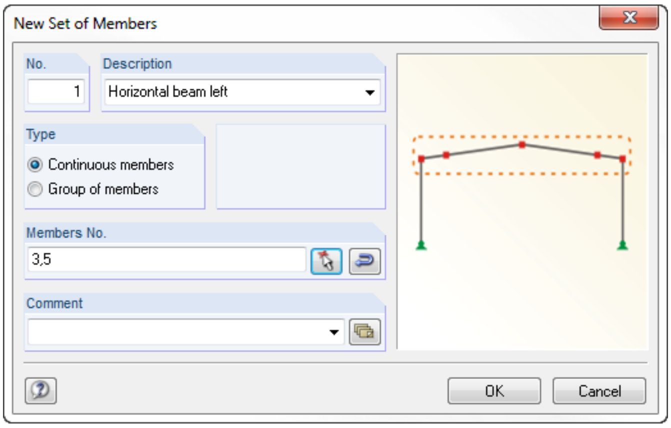


Figure 4.23: Dialog box *New Set of Members*

To complete the definition of the set of members, we click [OK].

We define the right set of members of horizontal beams graphically by means of the button [New Set of Members] (second row of toolbar, sixth button).

The window *Set of Members* appears.

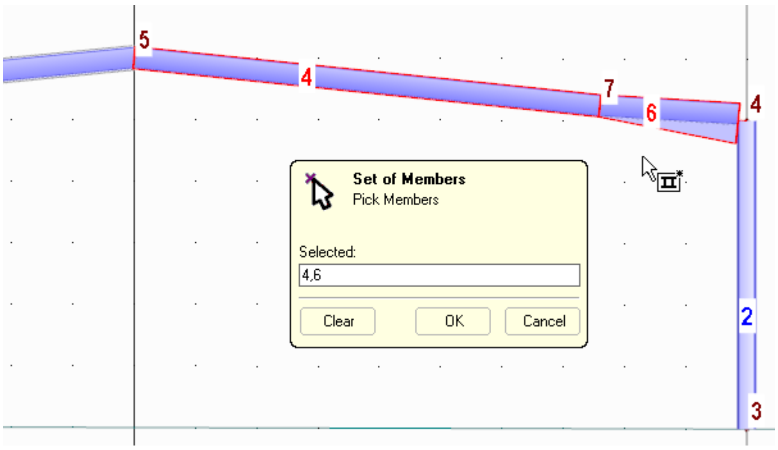


Figure 4.24: Window *Set of Members*

We click the two horizontal beams **4** and **6**. After [OK], the dialog box *New Set of Members* appears where we enter the *Description* **Horizontal beams right**. The *Type* **Continuous members** is preset.

We click [OK] to define the second set of members.

4.5 Arranging Supports

The model is still without supports. Therefore, we select the nodes **1** and **3** by holding the left mouse button down and drawing a window across both nodes. The selected nodes are highlighted in a different color.

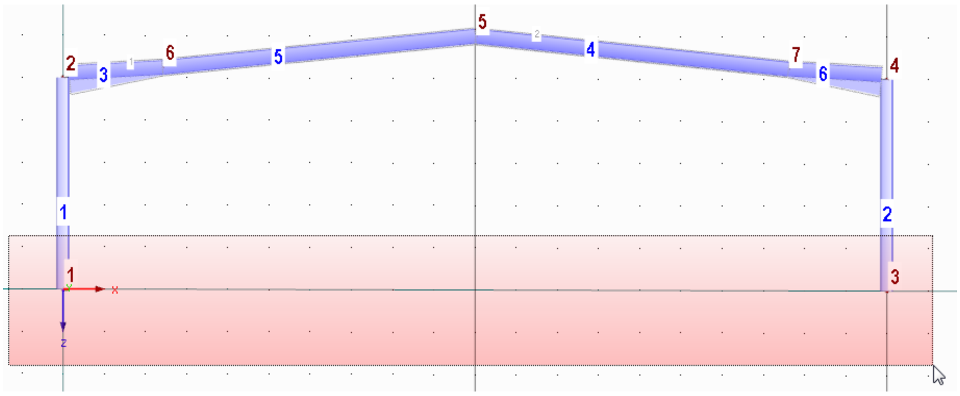


Figure 4.25: Selecting nodes of column bases with window

Tips for selection:

- If you pull up the window from the left to the right, the partial view contains only objects that are completely within this window. If you pull up the window from the right to the left, the partial view additionally contains those objects that are cut by the window.
- The selection is acting "alternatively": When you click an object (node, member, load), the selection of an already selected object will be canceled. Only the new object is selected. If you want to add the object to an existing selection, hold down the [Shift] key when clicking.

Now, we click the button [New Nodal Support] to open the dialog box *New Nodal Support*.

The *Node No.* **1** and **3** as well as the **Hinged** *Type of Support* are preset.

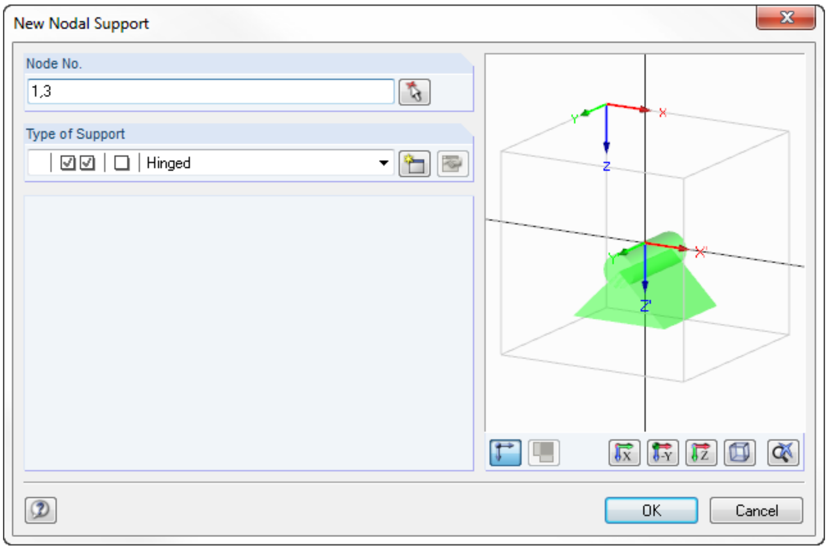


Figure 4.26: *New Nodal Support*

We could use the [New] button to create a user-defined support type, but we accept the hinged support. The three check boxes indicate the direction of the column supported in X and Z. A restraint about Y is not defined.

We confirm the dialog box with [OK]. Now, the input of model data is complete.

4.6 Changing the Numbering

The numbers of members and nodes got mixed up a bit because of the member divisions. It has no significance for the calculation. However, a clearly arranged numbering makes the input and evaluation of data easier.

RSTAB is able to correct irregular numberings automatically. First, we select all objects by drawing a window across the entire structure.

Then, we point to **Renumber** on the **Tools** menu and select **Automatically**. The following dialog box opens where we specify the priorities for the numbering directions.

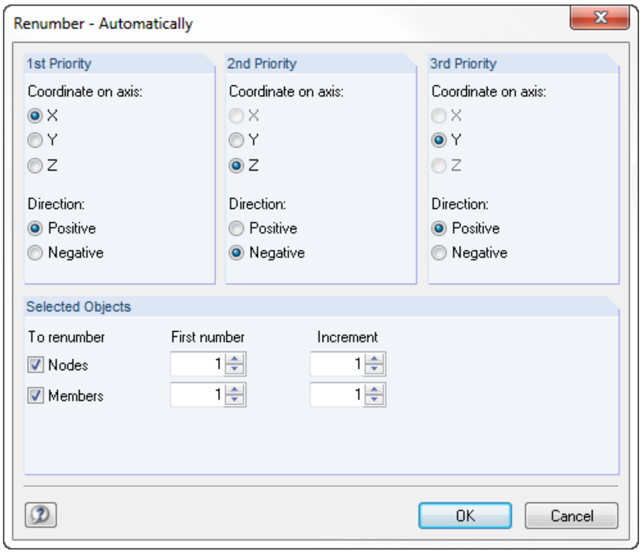


Figure 4.27: Dialog box *Renumber - Automatically*

First, we want to number the nodes and members according to X-coordinates, in ascending order and in direction of the positive axis X.

The Y-axis is of no importance for our 2D example. Therefore, **Axis Z** is the second priority. In addition, we change the *Direction* of the numbering to **Negative**. Thus, RSTAB will number the support nodes first and then the nodes above (they lie in negative direction Z).

After clicking the [OK] button the numbers of nodes and members are changed according to the specifications.

4.7 Checking the Input

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.

We can display and hide navigator and tables by selecting **Navigator** or **Table** on the **View** menu. We can also use the corresponding toolbar buttons.

In the tables, structural objects are organized in numerous tabs. Graphics and tables are interactive: To find an object in the table, for example a member, we set table 1.7 *Members* and select the member in the work window by clicking. We see that the corresponding table row is highlighted.

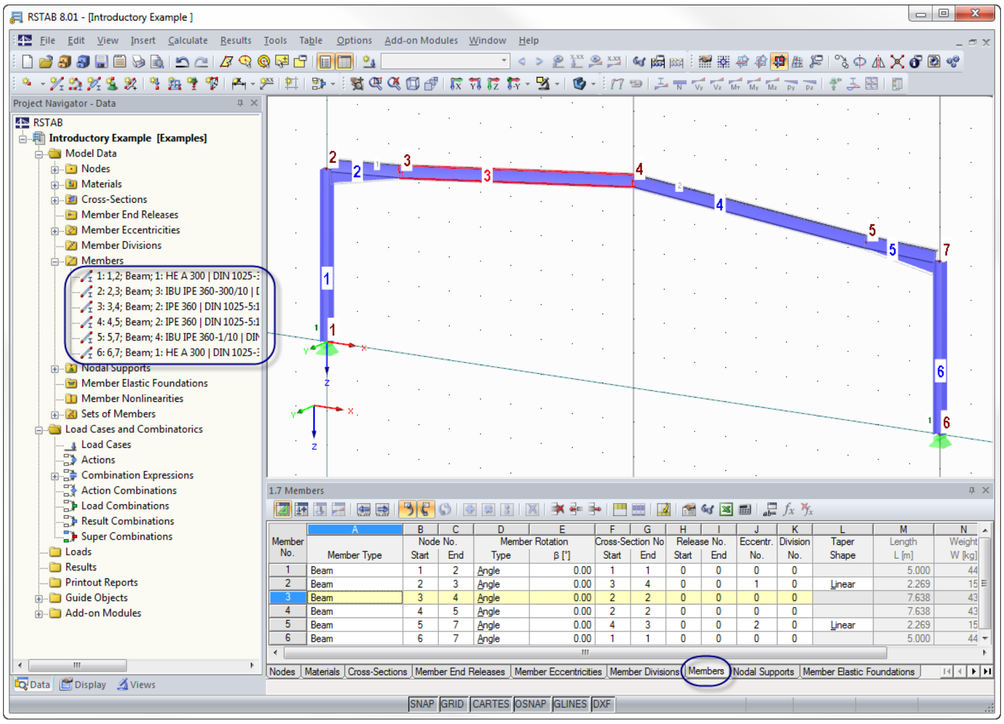
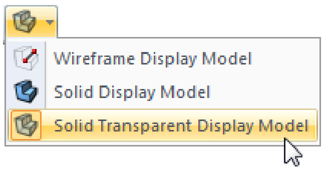


Figure 4.28: Model in isometric view with navigator and table entries

Changing the model display



The frame is displayed as transparent model by default. We can use the list button shown on the left to set other options for visualization:

- Wireframe display model
- Solid display model
- Solid transparent display model

When complex structural systems are modeled, the wireframe setting allows for a clear overview.

Saving data

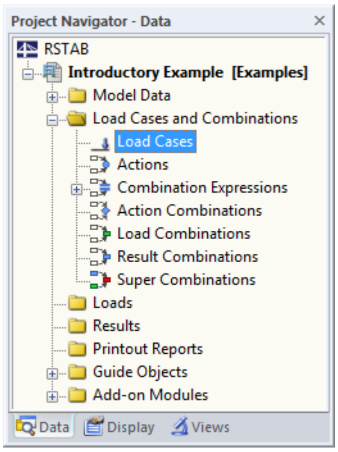
Finally, the input of model data is complete. To save our file,

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

or use the toolbar button shown on the left.

Saving data in the demo version is not possible.

5. Loads



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

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

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

5.1 Load Case 1: Self-weight

The first load case contains the permanently acting loads from self-weight and roof structure.

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

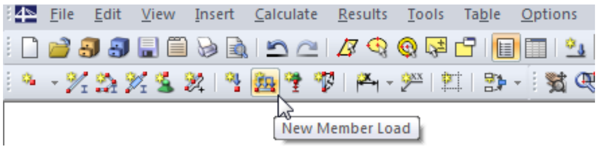


Figure 5.1: Button *New Member Load*

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

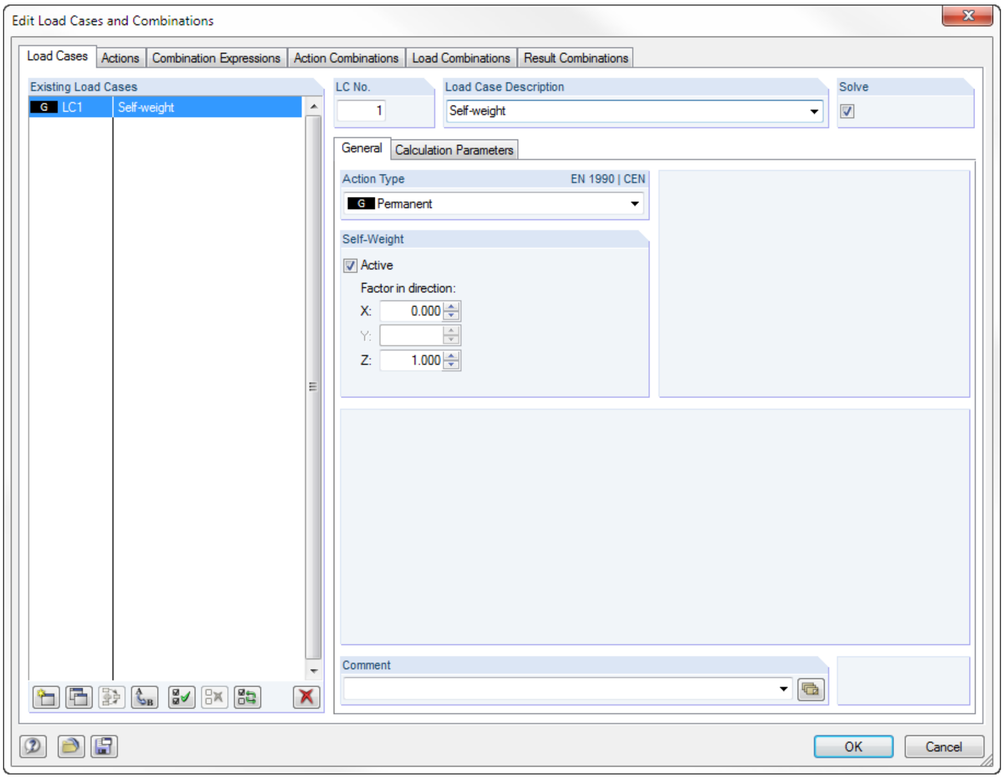
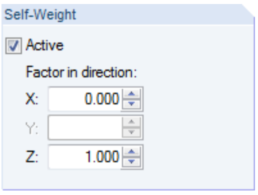


Figure 5.2: Dialog box *Edit Load Cases and Combinations*, tab *Load Cases*

Load case no. *1* is preset with the action type *Permanent*. For the *Load Case Description* we enter **Self-weight**, or we choose the entry from the list.

5.1.1 Self-weight



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

5.1.2 Roof Structure

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

The self-weight of the roof structure acts as load type *Force*. The load distribution is *Uniform*. We accept these presettings as well as the load direction in *Global Z* and the *True member length* as reference length.

In the dialog section *Load Parameters*, we enter the value **1.5** kN/m (see chapter 2.3, page 6). Then, we close the dialog box by clicking [OK].

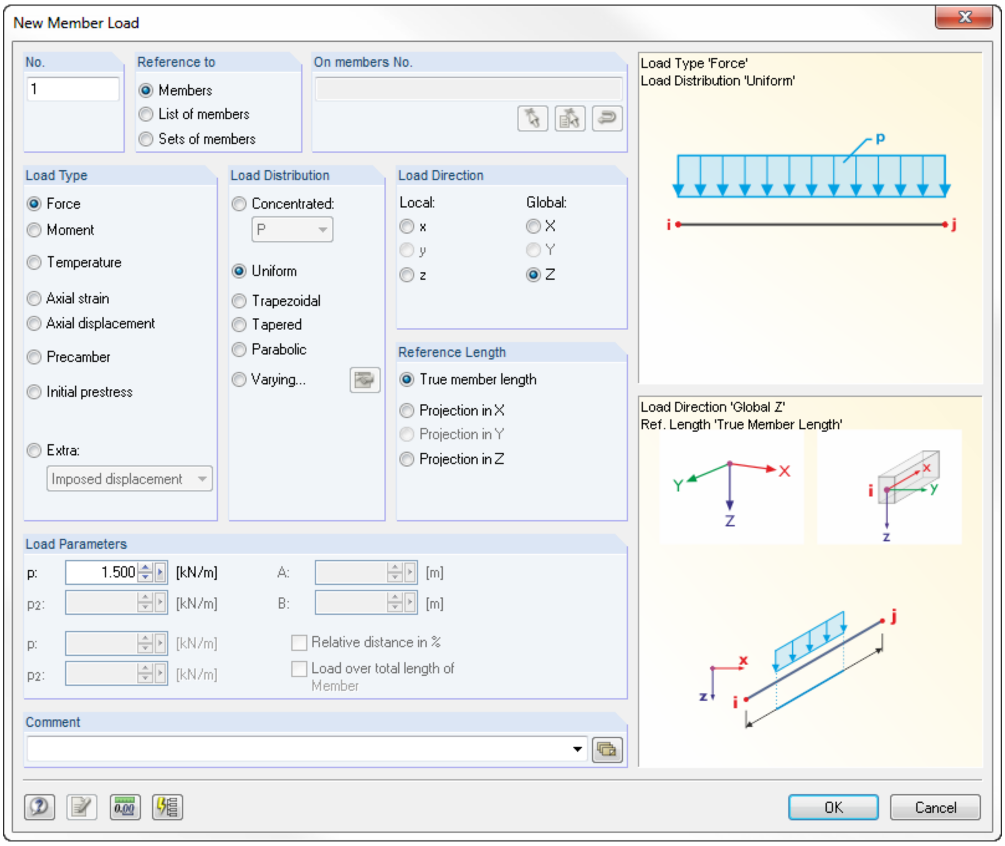
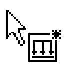


Figure 5.3: Dialog box *New Member Load*



Now, we can assign the load graphically to the horizontal beams: 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 member. We click the members **2**, **3**, **4** and **5** one after the other to put the loads on the horizontal beams (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 into the empty work window.

The input of the load case *Self-weight* is complete.

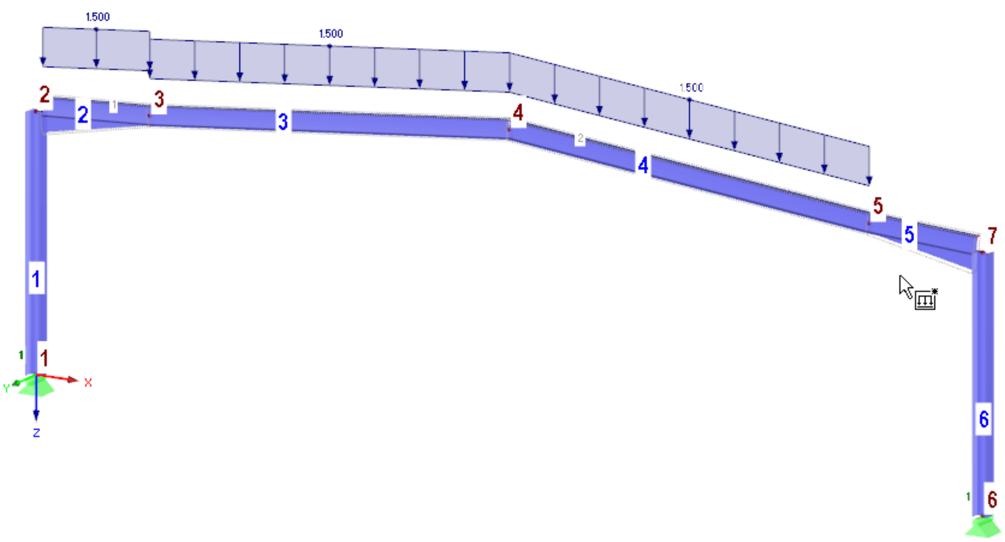


Figure 5.4: Assigning roof loads graphically

5.2 Load Case 2: Snow

Before we enter the snow load, we create a new load case. To open the corresponding dialog box,

we point to **Load Cases and Combinations** on the **Insert** menu and select **Load Case**,

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

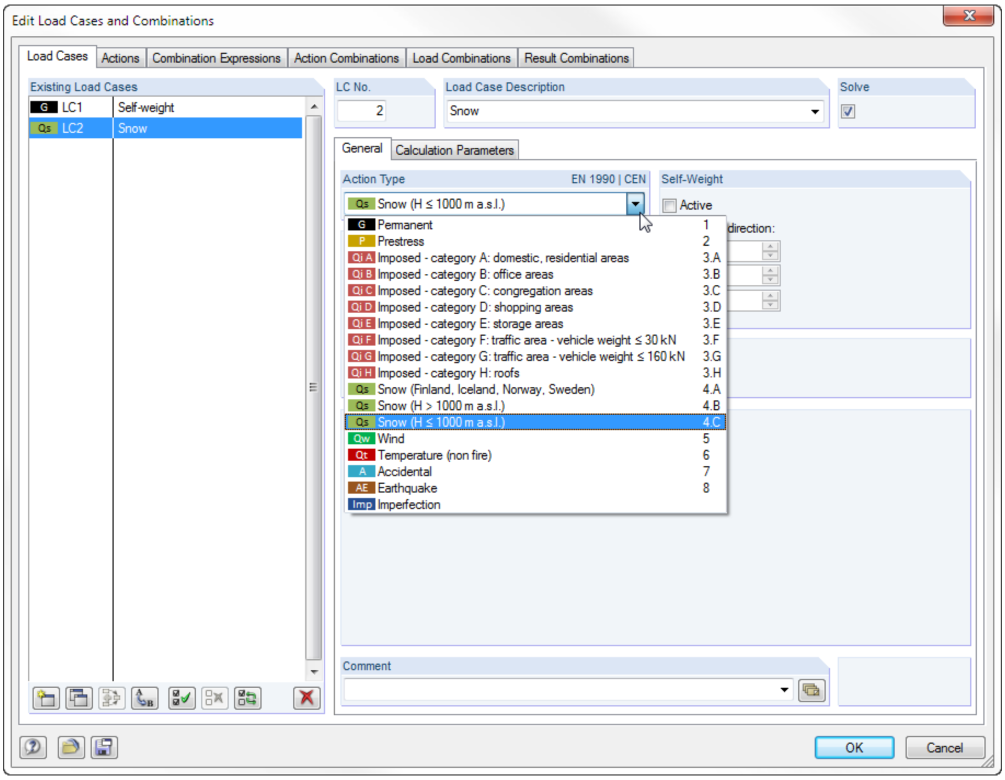


Figure 5.5: Dialog box *Edit Load Cases and Combinations*, tab *Load Cases*

As *Load Case Description* we select the entry **Snow** from the list.

We set the *Action Type* to **Q_s Snow (H ≤ 1000 m a.s.l.)** and confirm the input with [OK].

We choose a new way to enter the member loads: We select all horizontal frame beams (members 2 to 5) by drawing a selection window from the left to the right over this area. Now, when we open the dialog box by means of the button [New Member Load], we can see that the numbers of the selected members are already entered.

The load type is preset to *Force*. The load distribution is set to *Uniform* with the load direction *Global Z*. In contrast to dead loads which refer to the true area, snow loads must refer to the base area, and therefore we change the *Reference Length* to **Projection in Z**.

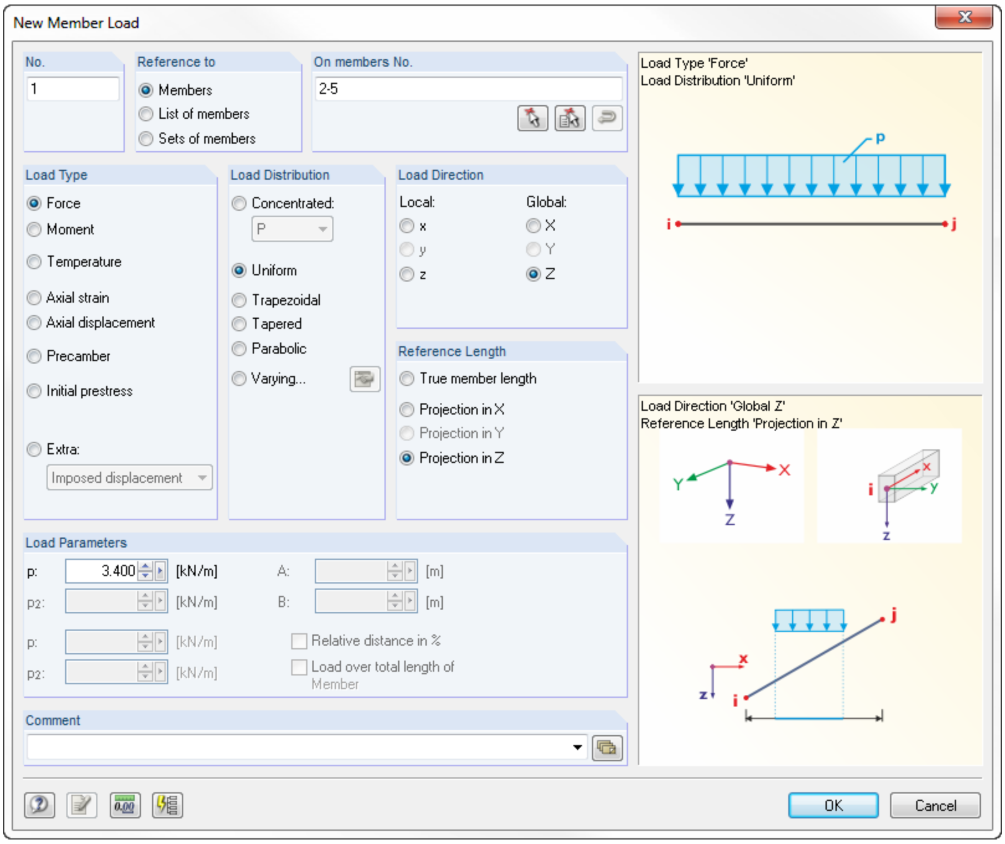


Figure 5.6: Dialog box *New Member Load*

In the dialog section *Load Parameters*, we enter the value **3.4** kN/m (see chapter 2.3, page 6). Then, we close the dialog box by clicking [OK].

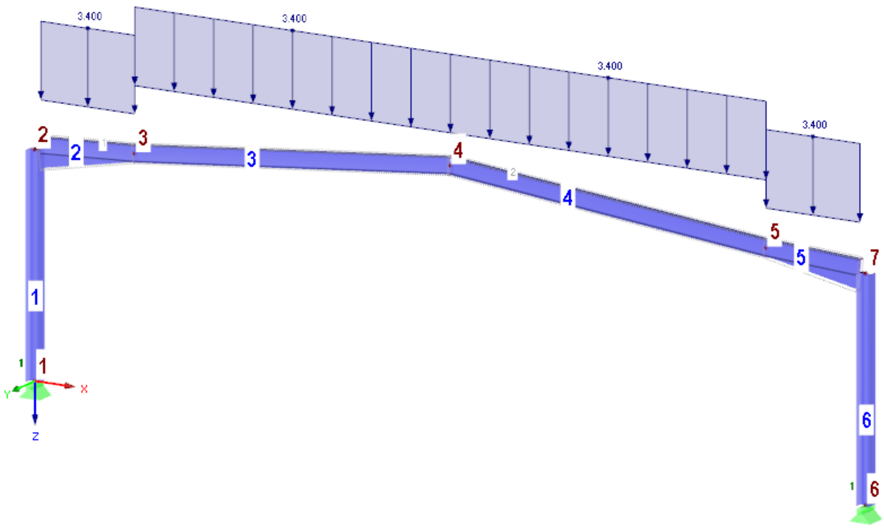


Figure 5.7: Snow load

5.3 Load Case 3: Wind

We put the wind load for wind zone 1 and terrain category III only on the two columns to simplify our example. The roof areas are neglected.

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

In the dialog field *Load Case Description*, we select **Wind in +X** from the list. The *Action Type* changes automatically to **Q_w Wind**.

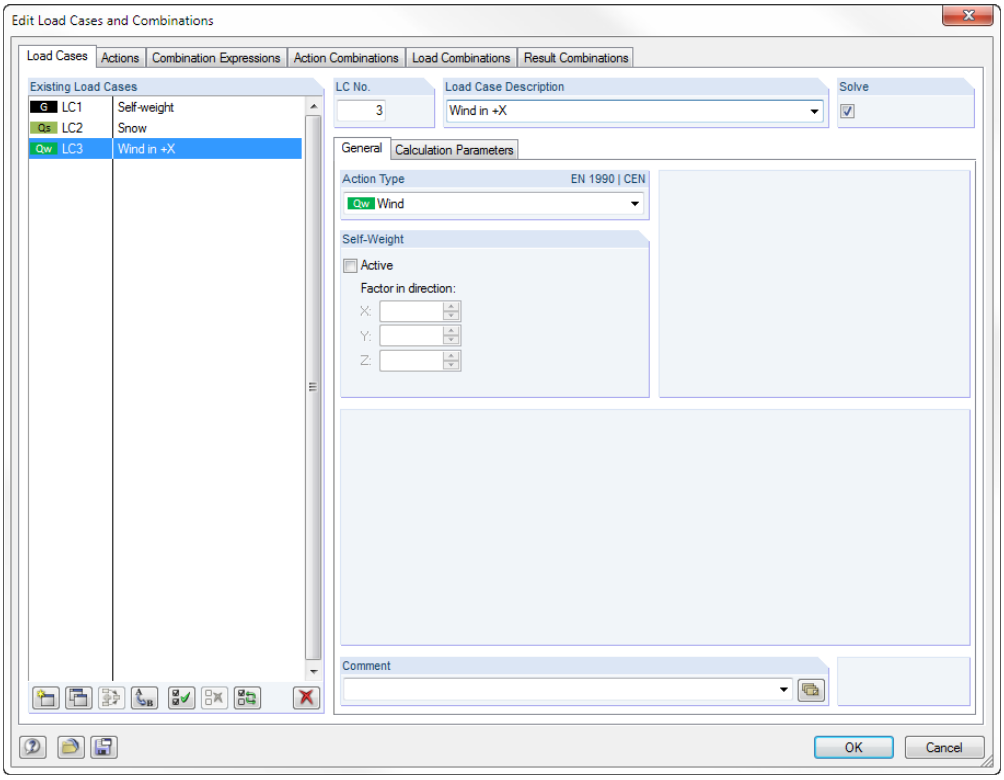


Figure 5.8: Dialog box *Edit Load Cases and Combinations*

We click [OK], and then we select the two column members *1* and *6* by mouse-click. When clicking we hold down the [Ctrl] key, as common for Windows applications, to do a multiple selection.

With a click on the button [New Member Load] we open the dialog box shown in Figure 5.9.

The load is acting as a *Force*. The load diagram is *Uniform*. We set the load direction to *Global* **X**. Again we select the **True member length** as the *Reference Length*.

In the dialog section *Load Parameters*, we enter **2.0** kN/m. This is the wind load component that acts on the left column (see chapter 2.3, page 6). Later, we will adjust the wind suction value on the right column.

We click [OK]. The member loads are displayed on the columns.

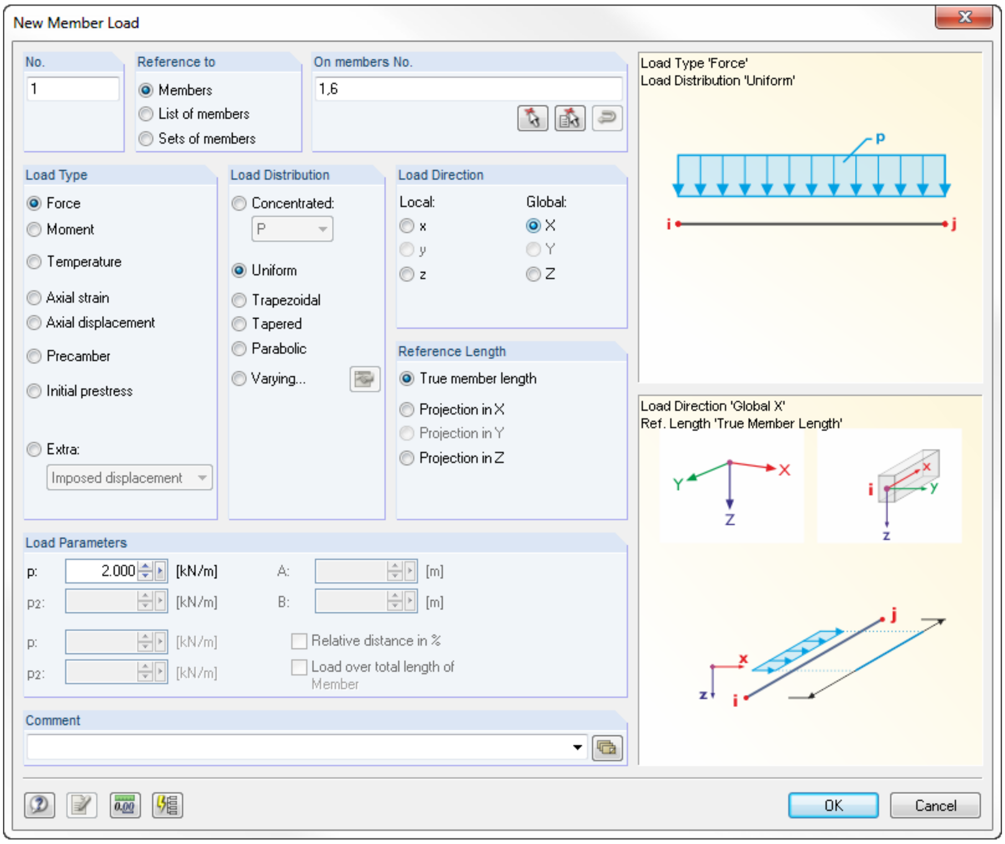


Figure 5.9: Dialog box *New Member Load*

Now, we have to correct the wind load of the right column. We double-click the load in the work window.

The dialog box *Edit Member Load* opens and we change the load value to **1.25** kN/m.

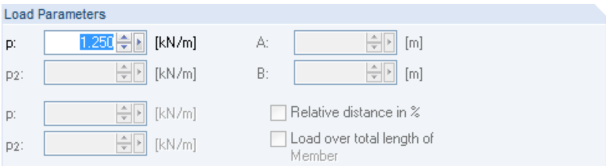


Figure 5.10: Adjusting the load value in the dialog box *Edit Member Load*

We click [OK]. The wind loads are displayed as follows.

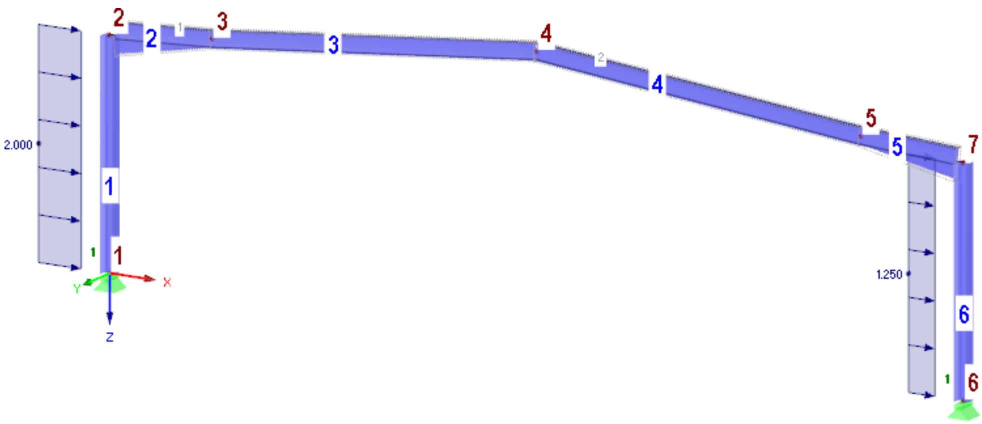


Figure 5.11: Wind loads for compression and suction

5.4 Load Case 4: Imperfection

In the final load case we define imperfections for the columns and horizontal beams.

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

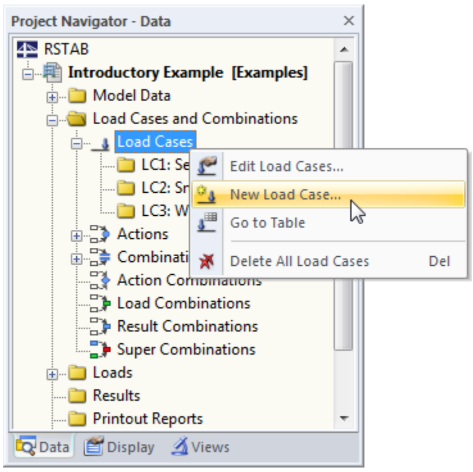


Figure 5.12: Context menu *Load Cases*

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

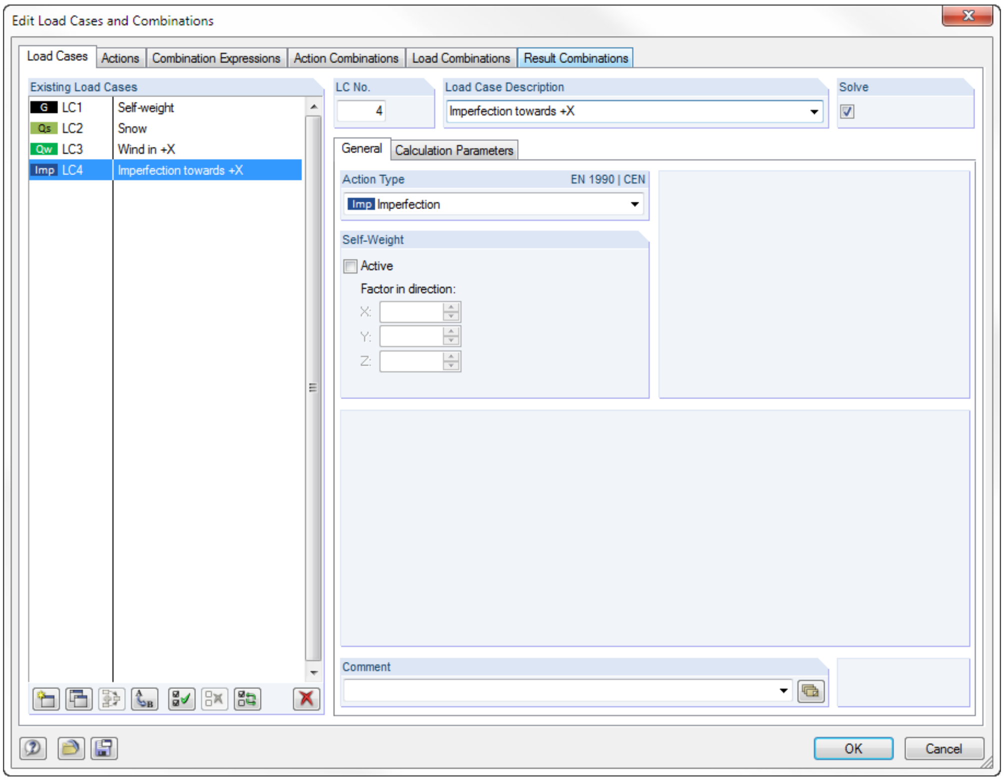


Figure 5.13: Dialog box *Edit Load Cases and Combinations*

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

5.4.1 Column Members

Member 6 is still selected. We cancel the selection by clicking with the left mouse button into an "empty" space of the work window.

Now we use the button [New Imperfection] to open the dialog box *New Imperfection*.

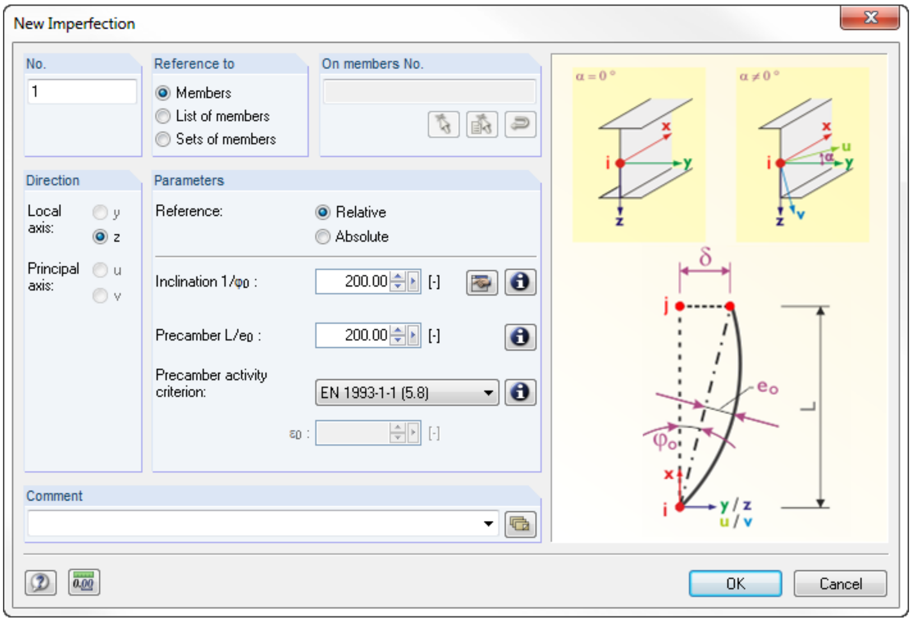
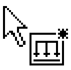


Figure 5.14: Dialog box *New Imperfection*

In our planar structural system, we can apply the imperfection only in *Direction* of the member axis *z*.

We do not change the preset *Inclination 1/φ_0* of *200*, but we modify the value of the *Precamber L/e_0* to **200** (see chapter 2.3, page 6).

Then, we set the *activity criterion* for the precamber by selecting **EN 1993-1-1 (5.8)** from the list.



We confirm the input with [OK] and click the column members **1** and **6** to assign the imperfections.

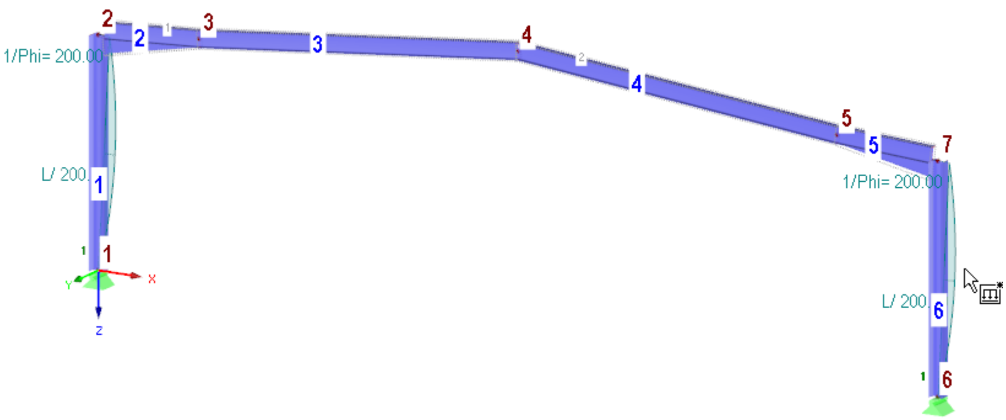


Figure 5.15: Assigning imperfections to columns

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

5.4.2 Continuous Members with Horizontal Beams

For the horizontal beams we must apply a "continuous" imperfection across both members on each side. This time we select the objects first. Then we assign the imperfections.

As selecting continuous members graphically in the rendering is not that easy,

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

or we use the toolbar button shown on the left.

In the dialog box *Special Selection*, we set the category *Sets of Members* on the left. As we want to select *All* sets of members, we can confirm the dialog box directly with the [OK] button.

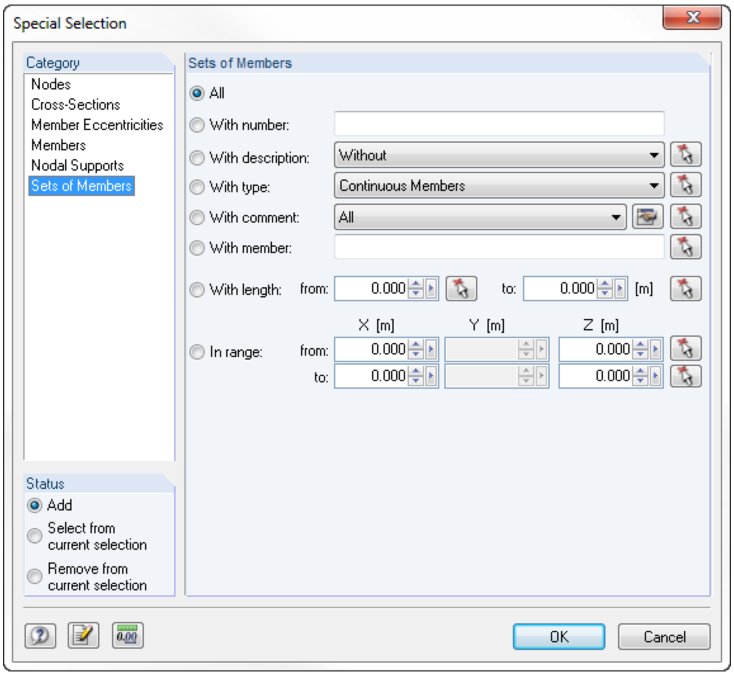


Figure 5.16: Dialog box *Special Selection*

Again we use the button [New Imperfection] to open the dialog box *New Imperfection*.

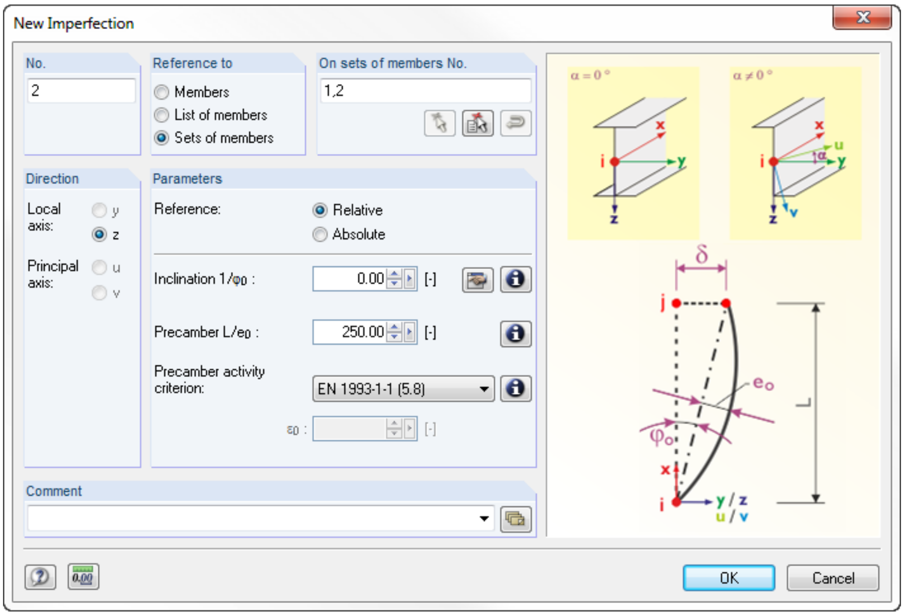


Figure 5.17: Dialog box *New Imperfection*

The numbers of both sets of members are preset.

We set the *Inclination 1/φ₀* to **0** and change the value of the *Precamber L/e₀* to **250** (see chapter 2.3, page 6).

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

The direction of inclination and precamber on the right beam does not yet fulfill the specifications (see Figure 2.1, page 5). We double-click this imperfection to open the dialog box *Edit Imperfection*.

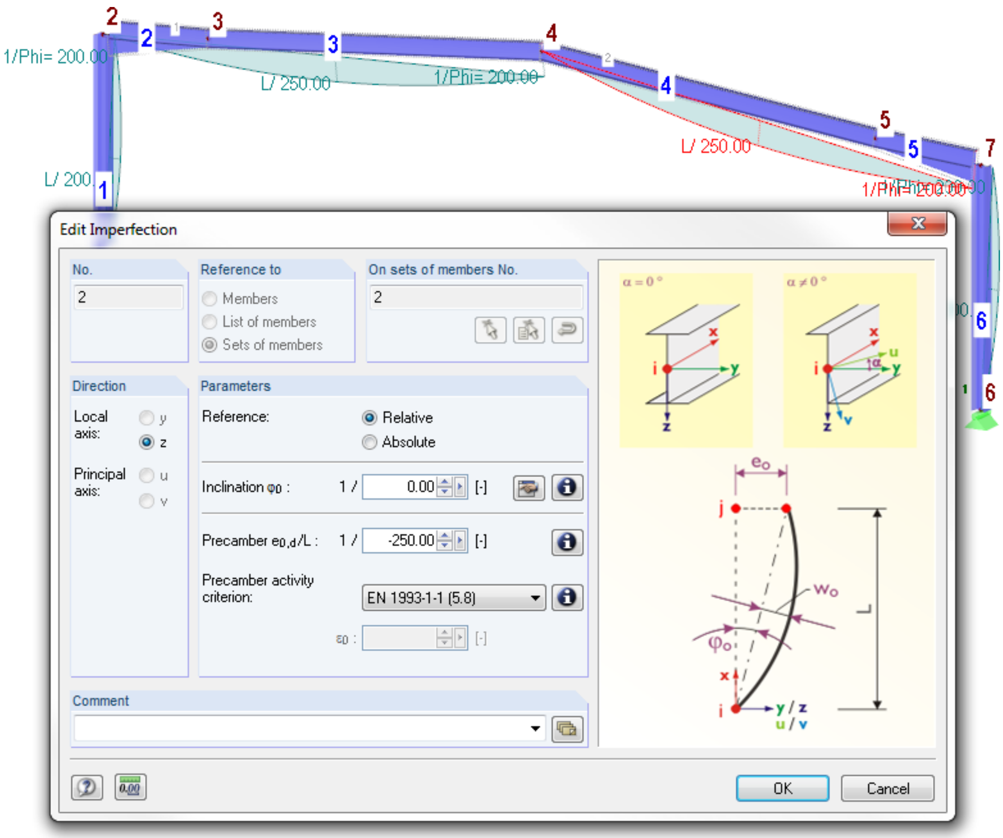


Figure 5.18: Adjusting the orientation of inclination and precamber

We put a negative sign [**-**] in front of the value for precamber. Consequently, the imperfection is acting in direction of the negative member axis z.

When we click [OK] we can see that the imperfections are displayed correctly.

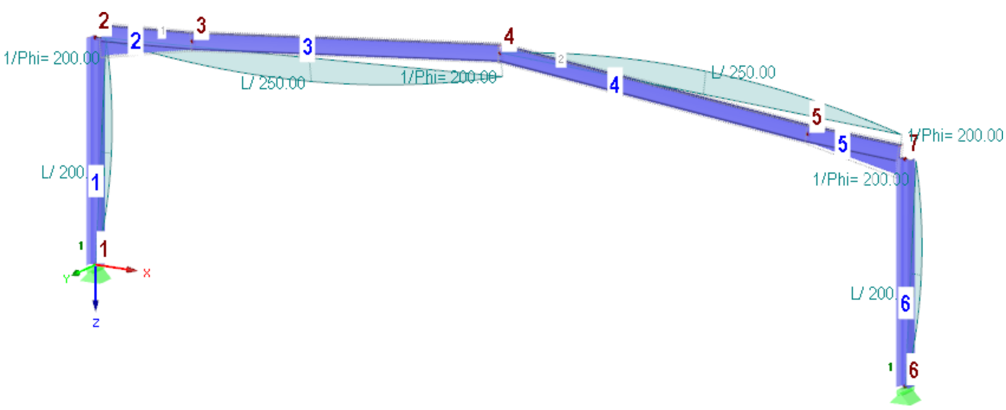
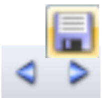


Figure 5.19: Imperfections for columns and horizontal beams

5.5 Checking Load Cases



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

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

Figure 5.20: Browsing the load cases

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

Again, graphic and tables are interactive: To find a load in the table, for example a member load, we set table 3.2 *Member Loads*, and then we select the load in the work window. We see that the pointer jumps into the corresponding row of the table.

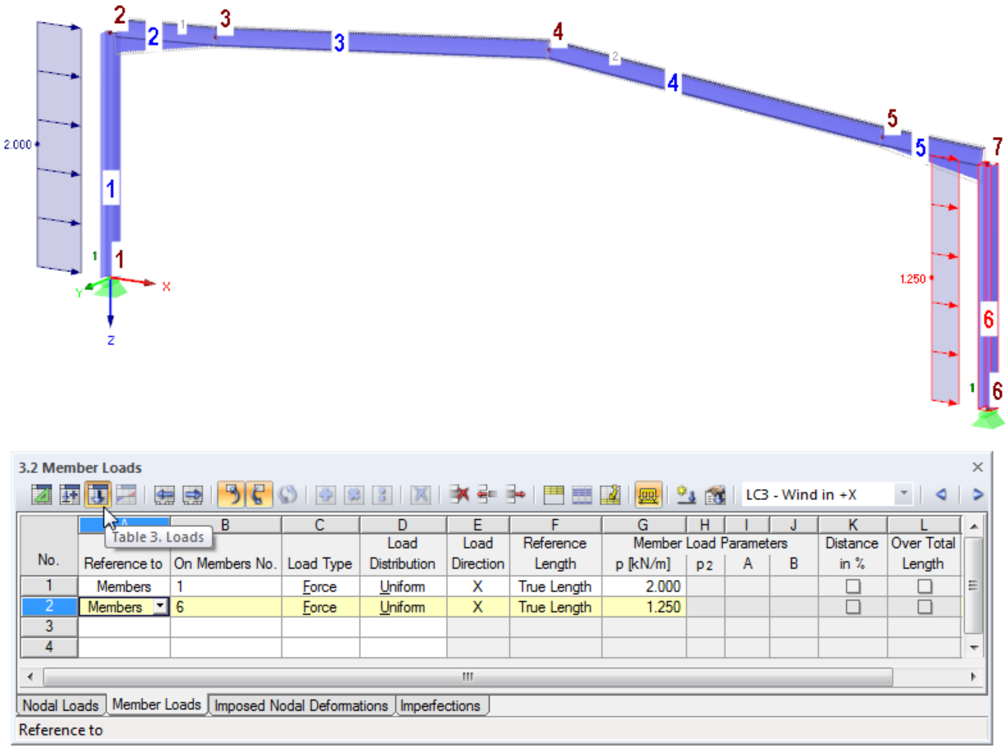


Figure 5.21: Interaction between graphic and load tables

6. Combination of Actions

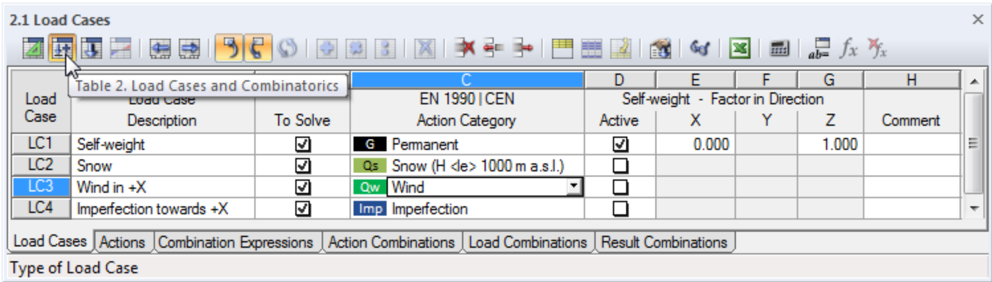
We combine the load cases according to EN 1990. We take advantage of the generator integrated in the program to superimpose the actions with the required partial safety factors and combination coefficients. The relevant conditions have already been created when the model was defined in the dialog box *General Data* where we have chosen the option *Create combinations automatically* (see Figure 3.1, page 7).

The *Action Type* defined for the load cases (see Figure 5.13, page 32) determines the way how load cases are combined in different design situations.

6.1 Checking the Actions

The load cases must be assigned to *Actions* which will be superimposed in accordance with regulations. Actions represent independent influence values that arise from different origins.

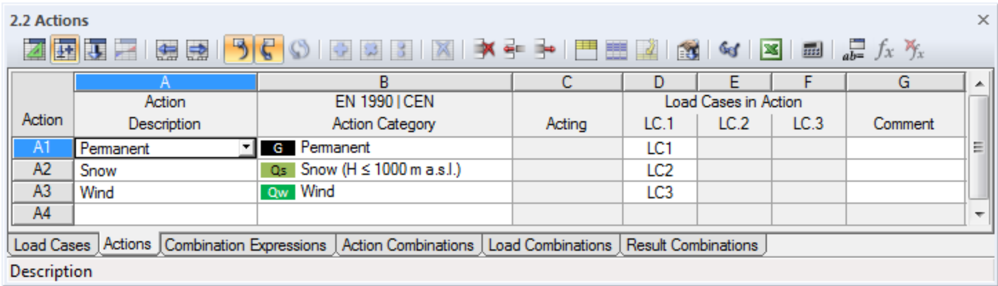
The load cases, actions and combinations are managed in the dialog box *Edit Load Cases and Combinations* (see Figure 5.13, page 32) as well as in tables number *2*. We can access these tables by clicking the table button shown on the left. Table 2.1 *Load Cases* shows us the four load cases with the selected action categories in a clear overview.



Load Case	A: Load Case Description	B: To Solve	C: EN 1990 | CEN Action Category	D: Self-weight Active	E: Factor in Direction X	F: Y	G: Z	H: Comment
LC1	Self-weight	☑	G Permanent	☑	0.000		1.000	
LC2	Snow	☑	Qs Snow (H <le> 1000 m a.s.l.)	☐				
LC3	Wind in +X	☑	Qw Wind	☐				
LC4	Imperfection towards +X	☑	Imp Imperfection	☐				

Figure 6.1: Table 2.1 *Load Cases*

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



Action	A: Action Description	B: EN 1990 | CEN Action Category	C: Acting	D: Load Cases in Action LC.1	E: LC.2	F: LC.3	G: Comment
A1	Permanent	G Permanent		LC1			
A2	Snow	Qs Snow (H ≤ 1000 m a.s.l.)		LC2			
A3	Wind	Qw Wind		LC3			
A4							

Figure 6.2: Table 2.2 *Actions*

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

6.2 Defining Combination Expressions

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

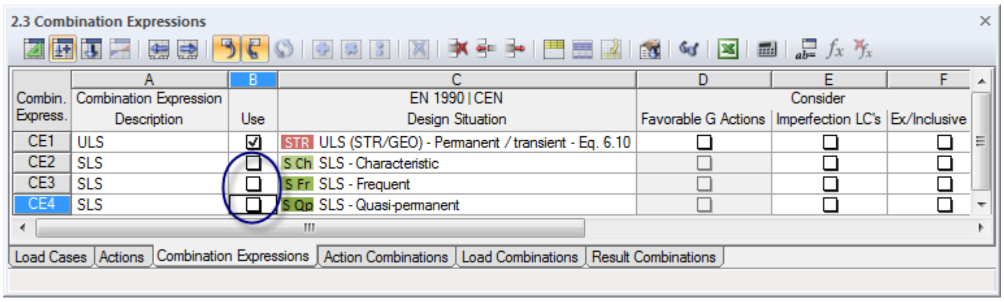


Figure 6.3: Table 2.3 *Combination Expressions*

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

We right-click table entry **CE1** (in the first table column). The context menu opens where we select the entry *Edit via Dialog Box*.

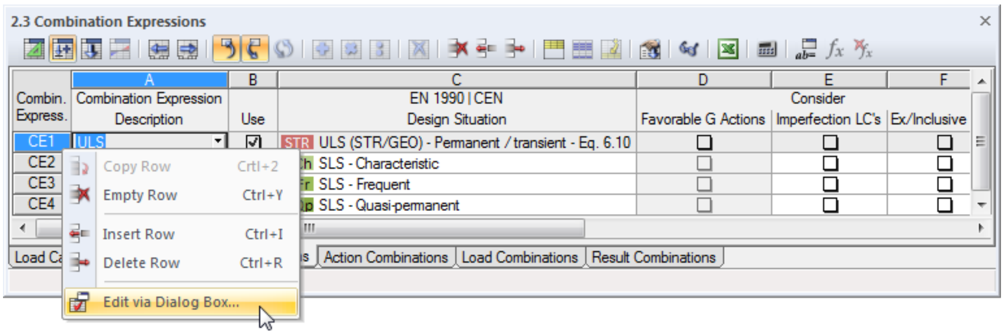


Figure 6.4: Context menu of table row

The dialog box *Edit Load Cases and Combinations* appears (see figure below). In the dialog tab *Combination Expressions*, we can adjust the settings according to which the combinations are created.

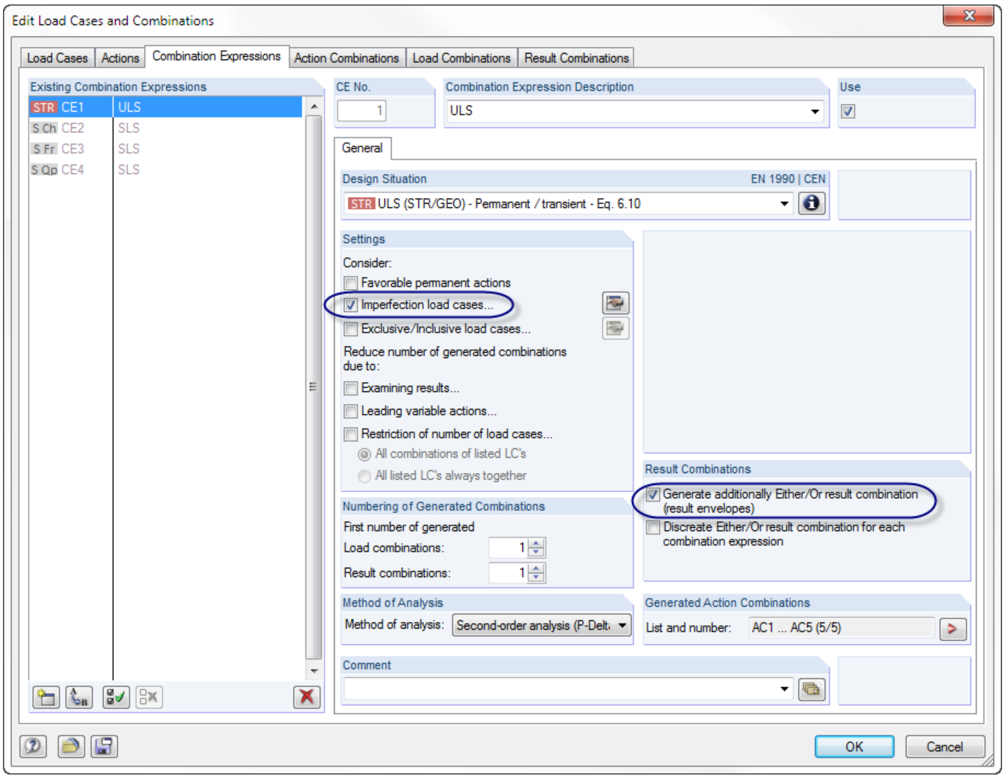


Figure 6.5: Dialog box *Edit Load Cases and Combinations*, tab *Combination Expressions*

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

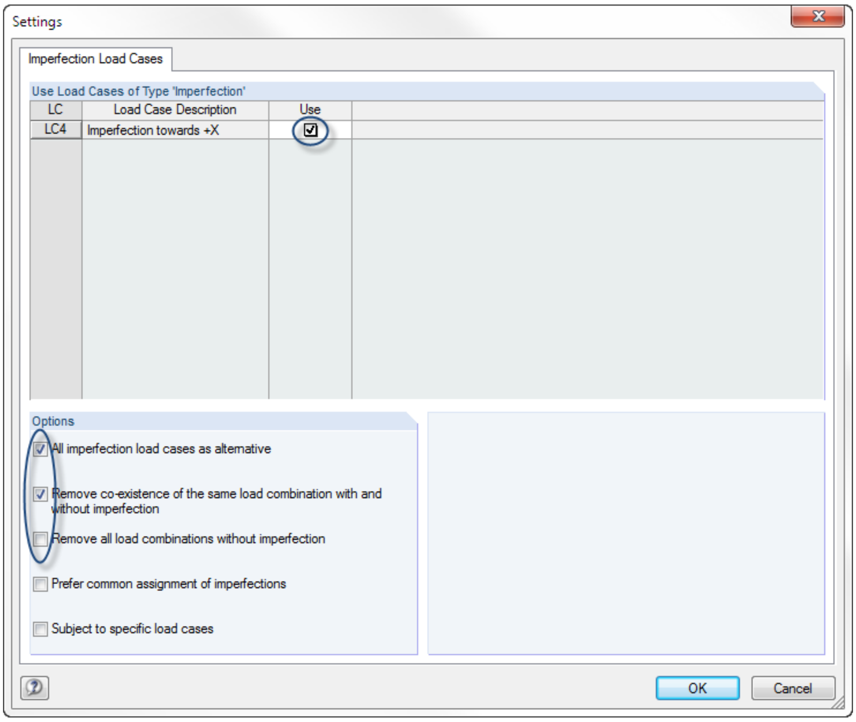


Figure 6.6: Dialog box *Settings* for imperfection load cases

We set the check marks as shown in the figure above. Then, we confirm the data with [OK].

In the dialog box *Edit Load Cases and Combinations* (see Figure 6.5), we make sure that the option **Generate additionally Either/Or result combination** is checked as well. This combination provides the extreme values from the results of all load combinations (envelope).

Then we confirm the dialog box with [OK].

6.3 Creating Action Combinations

When we move on to the next table 2.4 *Action Combinations*, RSTAB creates five combinations which are listed and sorted by 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: Generated Load Combinations
AC1	1.35G	☑	STR ULS (STR/GEO) - Permanent	1.35	G A1					CO1 (1/5)
AC2	1.35G + 1.50Qs	☑	STR ULS (STR/GEO) - Permanent	1.35	G A1	1.50	Qs A2			CO2 (1/5)
AC3	1.35G + 1.50Qs + 0.90Qw	☑	STR ULS (STR/GEO) - Permanent	1.35	G A1	1.50	Qs A2	0.90	Qw A3	CO3 (1/5)
AC4	1.35G + 1.50Qw	☑	STR ULS (STR/GEO) - Permanent	1.35	G A1	1.50	Qw A3			CO4 (1/5)
AC5	1.35G + 0.75Qs + 1.50Qw	☑	STR ULS (STR/GEO) - Permanent	1.35	G A1	0.75	Qs A2	1.50	Qw A3	CO5 (1/5)

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

Active

Figure 6.7: Table 2.4 *Action Combinations*

This overview corresponds to the presentation of actions described in the standards. Looking at the *Use* column we can see which action combinations will be considered for the generation of load combinations.

6.4 Creating Load Combinations

Five load combinations are automatically created from the five action combinations. The result is listed in the subsequent table 2.5 *Load Combinations*.

2.5 Load Combinations

Load Combin.	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	1.35*LC1 + LC4	☑	1.35	G LC1	1.00	Imp LC4				
CO2	1.35*LC1 + 1.5*LC2 + LC4	☑	1.35	G LC1	1.50	Qs LC2	1.00	Imp LC4		
CO3	1.35*LC1 + 1.5*LC2 + 0.9*LC3 + LC4	☑	1.35	G LC1	1.50	Qs LC2	0.90	Qw LC3	1.00	Imp LC4
CO4	1.35*LC1 + 1.5*LC3 + LC4	☑	1.35	G LC1	1.50	Qw LC3	1.00	Imp LC4		
CO5	1.35*LC1 + 0.75*LC2 + 1.5*LC3 + LC4	☑	1.35	G LC1	0.75	Qs LC2	1.50	Qw LC3	1.00	Imp LC4

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

Figure 6.8: Table 2.5 *Load Combinations*

Table columns D to K inform us about the load cases including partial safety and combination factors that are taken into account for each load combination.

The imperfections are contained in all combinations as specified before (see Figure 6.6).

To open the dialog box *Edit Load Cases and Combinations*, we use the context menu of the table row (see Figure 6.4, page 38), or

we point to **Load Cases and Combinations** on the **Edit** menu, and then we click **Load Combinations**,

to look at the created load combinations.

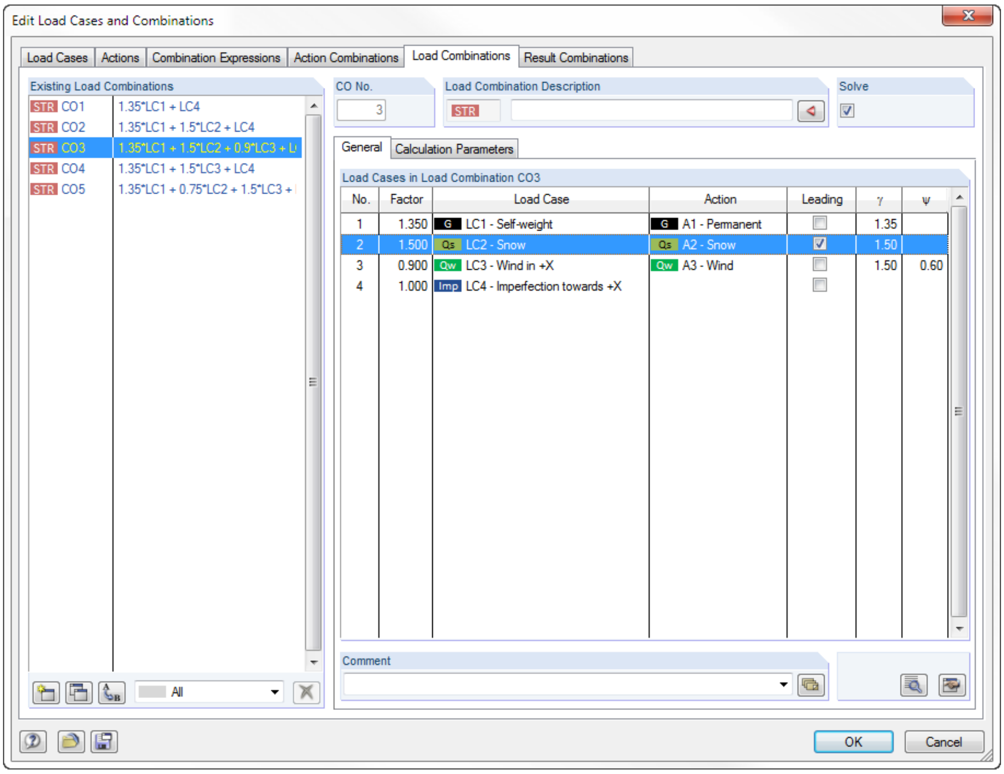


Figure 6.9: Dialog box *Edit Load Cases and Combinations*, tab *Load Combinations*

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

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

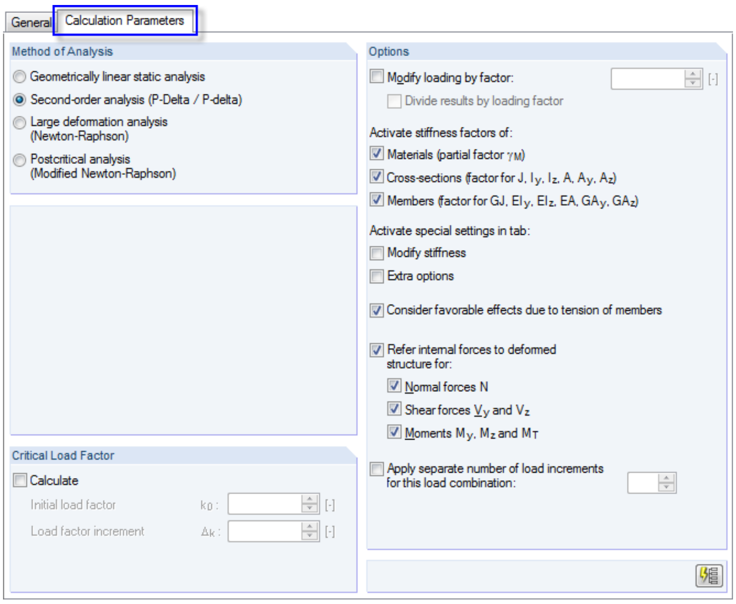


Figure 6.10: Checking the *Calculation Parameters* of a load combination

Basically, load combinations are analyzed non-linearly according to the *Method of Analysis* for *Second-order analysis*. Thus, it is possible to take into account influences from structural deformation resulting in an increase of internal forces.

6.5 Checking Result Combinations

When we defined the combination expressions, we activated the option *Generate additionally Either/Or result combination* (see Figure 6.5, page 39) giving us information about the extreme values of all load combinations.

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

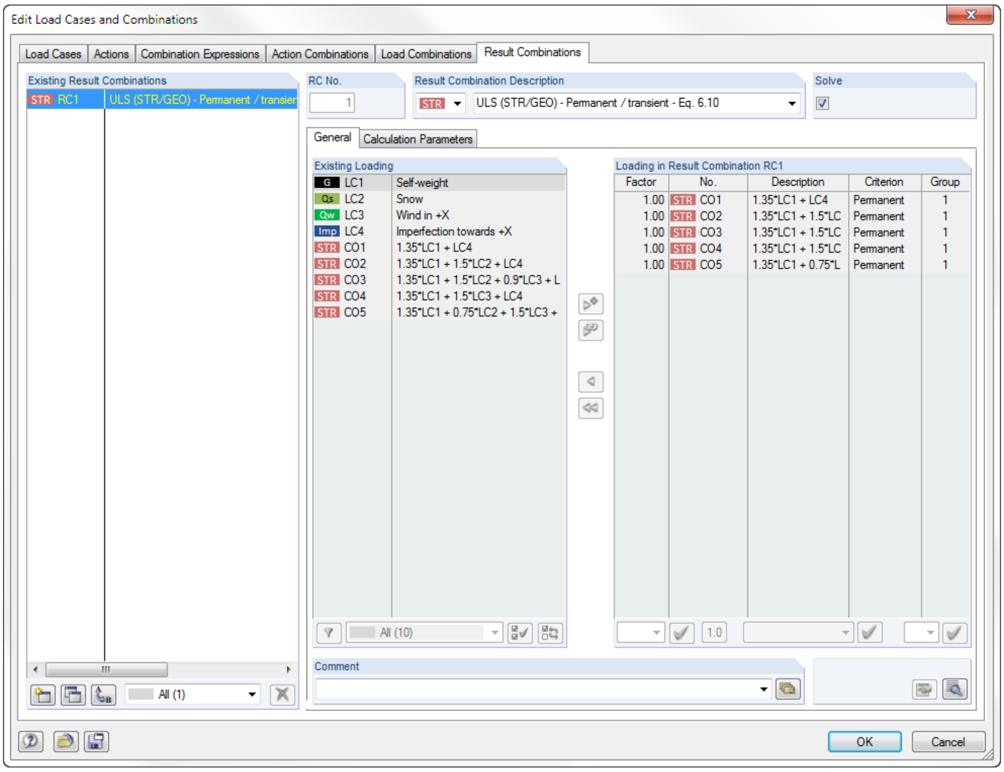


Figure 6.11: Dialog box *Edit Load Cases and Combinations*, tab *Result Combinations*

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

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

7. Calculation

7.1 Checking Input Data

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

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

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

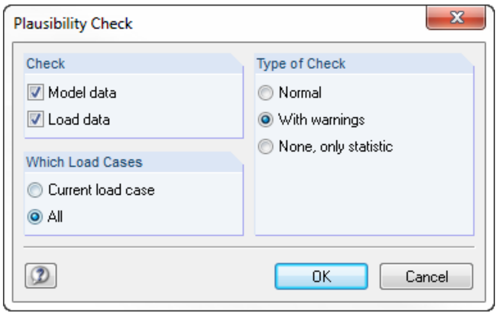


Figure 7.1: Dialog box *Plausibility Check*

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

7.2 Calculate Model

To start the calculation,

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

or use the toolbar button shown on the left.

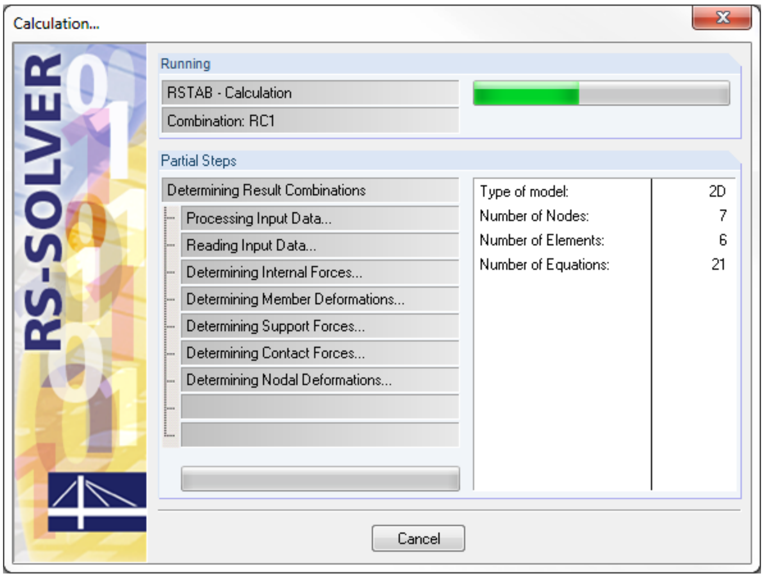


Figure 7.2: Calculation process

8. Results

8.1 Graphical Results

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

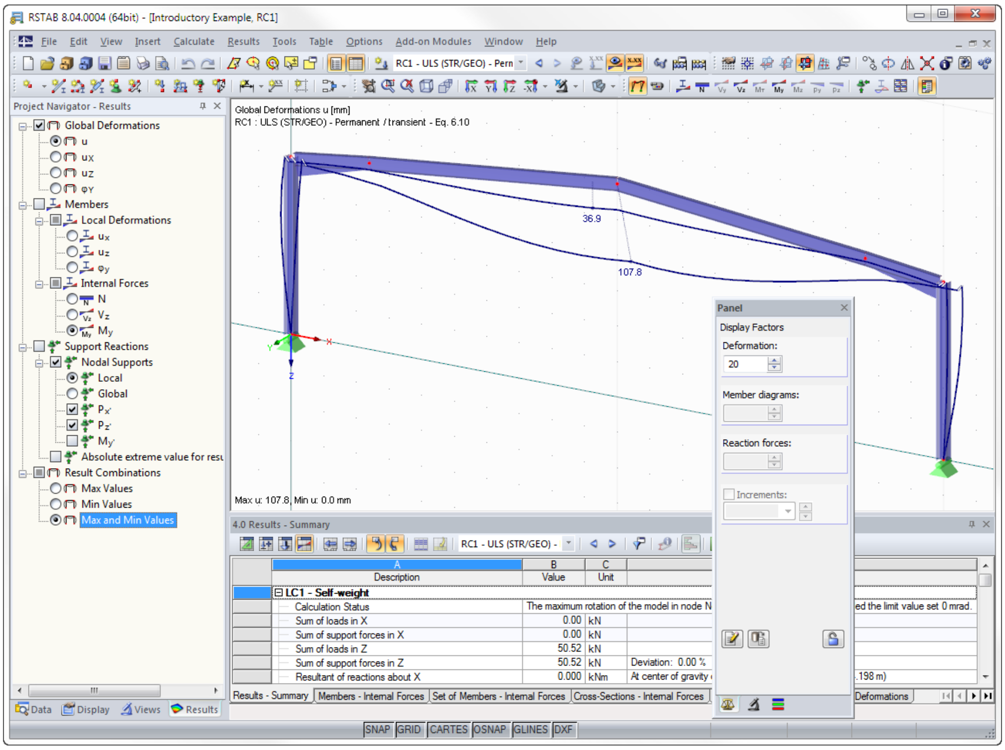


Figure 8.1: Graphic of deformations for result combination RC1

Selecting load cases and load combinations

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

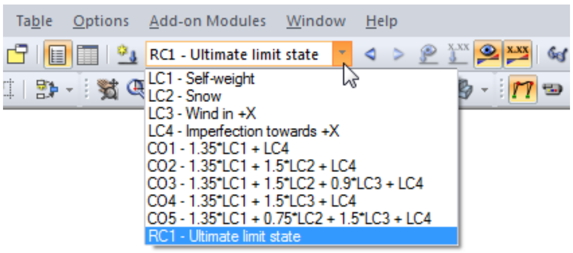


Figure 8.2: Load case list in the toolbar

Selecting results in the navigator

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

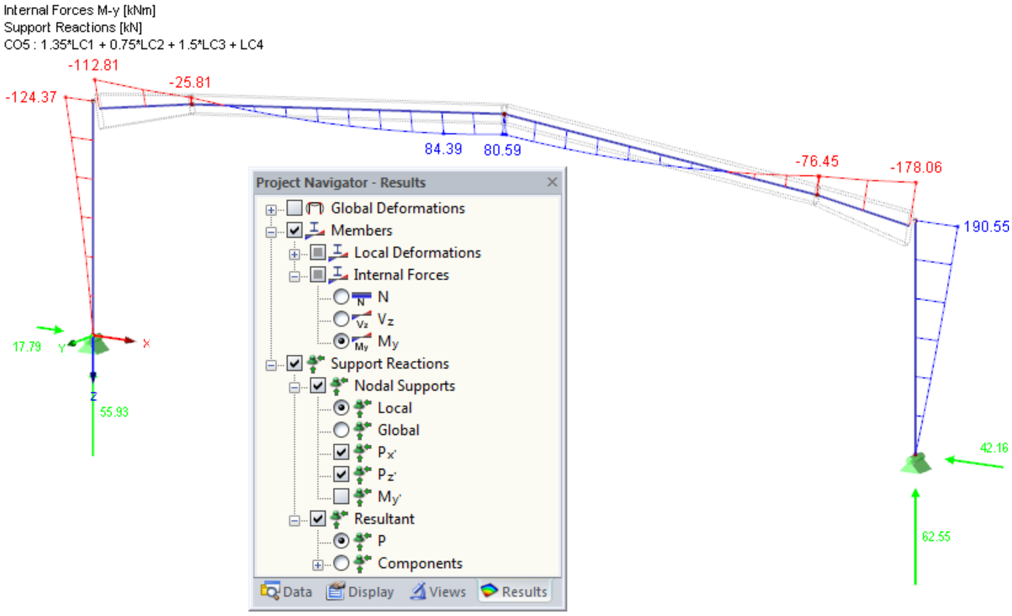
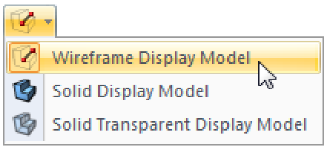


Figure 8.3: *Results* navigator

Now, we can browse the single load cases. By selecting the different results categories we see deformations, internal forces and support reactions displayed in RSTAB.



In the figure above, we see the member internal forces M_y and the support reactions $P_{x'}$ and $P_{z'}$ calculated for CO5. To display the forces, it is recommended to use the wireframe model. We can set this display option with the button shown on the left.

Show deformations as animation

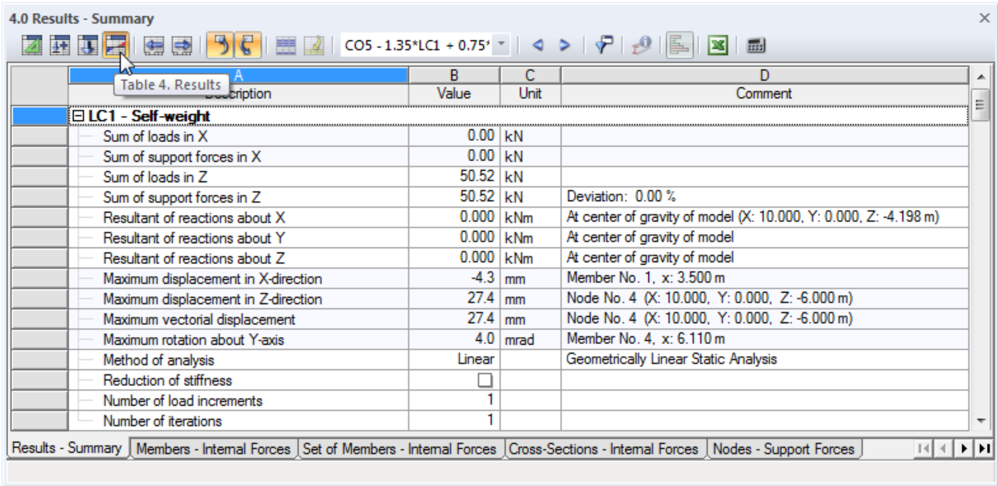
The deformation of the model cannot be represented only in its final state but as motion sequence. First, in the *Results* navigator, we set the results category **Global Deformations**. We can also use the toolbar button shown on the left.

Then, we click the button [Animation] to display the deformation process for all steps between initial state and final deformation. To close the animated view, we click the button again.

8.2 Results Tables

We can evaluate results also in tables.

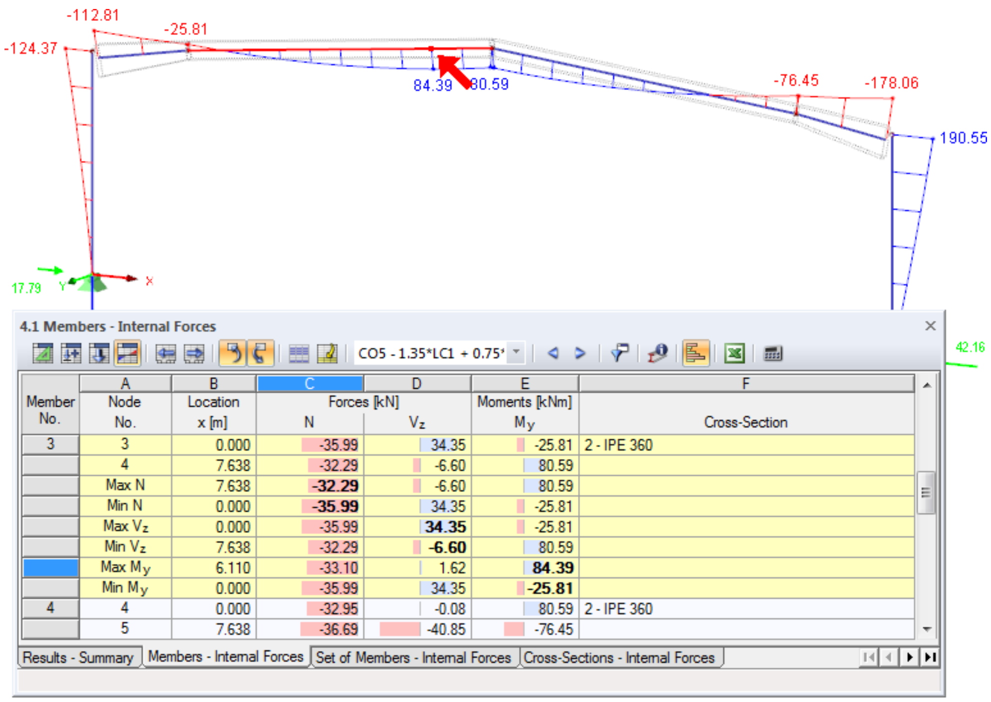
The results tables are displayed automatically after the calculation. There are different tables for the results, too. Table 4.0 *Results - Summary* offers us a summary of the calculation process, sorted by load cases and load combinations.



A Description	B Value	C Unit	D Comment
LC1 - Self-weight			
Sum of loads in X	0.00	kN	
Sum of support forces in X	0.00	kN	
Sum of loads in Z	50.52	kN	
Sum of support forces in Z	50.52	kN	Deviation: 0.00 %
Resultant of reactions about X	0.000	kNm	At center of gravity of model (X: 10.000, Y: 0.000, Z: -4.198 m)
Resultant of reactions about Y	0.000	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	-4.3	mm	Member No. 1, x: 3.500 m
Maximum displacement in Z-direction	27.4	mm	Node No. 4 (X: 10.000, Y: 0.000, Z: -6.000 m)
Maximum vectorial displacement	27.4	mm	Node No. 4 (X: 10.000, Y: 0.000, Z: -6.000 m)
Maximum rotation about Y-axis	4.0	mrad	Member No. 4, x: 6.110 m
Method of analysis	Linear		Geometrically Linear Static Analysis
Reduction of stiffness	☐		
Number of load increments	1		
Number of iterations	1		

Figure 8.4: Table 4.0 *Results - Summary*

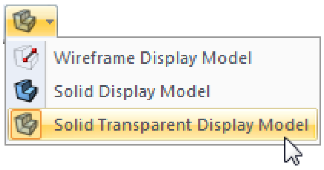
To select other tables, we click their table tabs. To find specific results in the table, for example the internal forces of a horizontal beam, we set table 4.1 *Members - Internal Forces*. When we click the member in the graphic, we can see that the pointer jumps to the corresponding internal forces in the table. The current member location x, that means the position of the pointer in the table row, is indicated by a marking arrow in the graphic.



Member No.	A Node No.	B Location x [m]	C Forces [kN] N	D Forces [kN] V_z	E Moments [kNm] M_y	F Cross-Section
3	3	0.000	-35.99	34.35	-25.81	2 - IPE 360
	4	7.638	-32.29	-6.60	80.59	
	Max N	7.638	**-32.29**	-6.60	80.59	
	Min N	0.000	**-35.99**	34.35	-25.81	
	Max V_z	0.000	-35.99	**34.35**	-25.81	
	Min V_z	7.638	-32.29	**-6.60**	80.59	
	Max M_y	6.110	-33.10	1.62	**84.39**	
	Min M_y	0.000	-35.99	34.35	**-25.81**	
4	4	0.000	-32.95	-0.08	80.59	2 - IPE 360
	5	7.638	-36.69	-40.85	-76.45	

Figure 8.5: Internal forces in table 4.1 and indication of current location x on the model

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



8.3 Filter Results

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

8.3.1 Visibilities

Partial views and cutouts can be used as so-called *Visibilities* in order to evaluate results. In our example, we set the [Solid Transparent Display Model] and display the internal forces M_y again.

Show results of columns

We click the tab *Views* in the navigator. We select *Members by Cross-Section* listed under the visibilities that RSTAB has *generated* from the entered data. In addition, we tick the entry **HE A 300**.

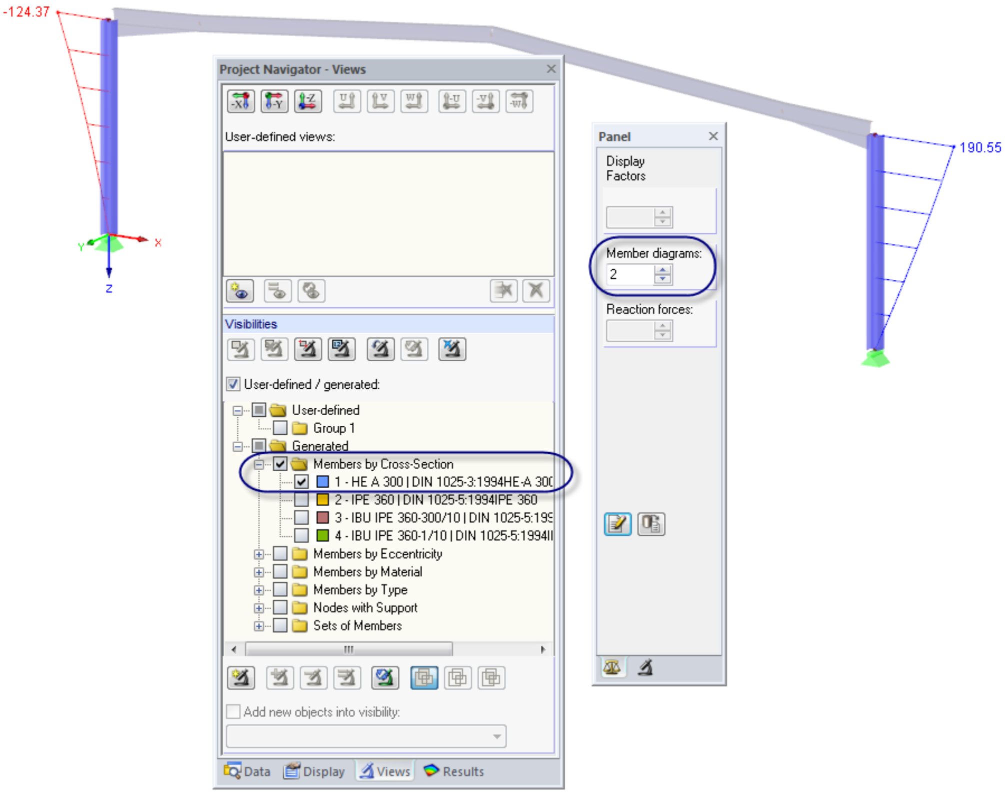


Figure 8.6: Moments M_y of columns in scaled representation

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

Adjusting the scaling factor

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

Show results of a continuous member

In the same way, we can filter the results of the continuous members in the *View* navigator. We clear the check box for *Members by Cross-Section* and select the entry *Sets of Members* where we tick the set of members **No. 2**.

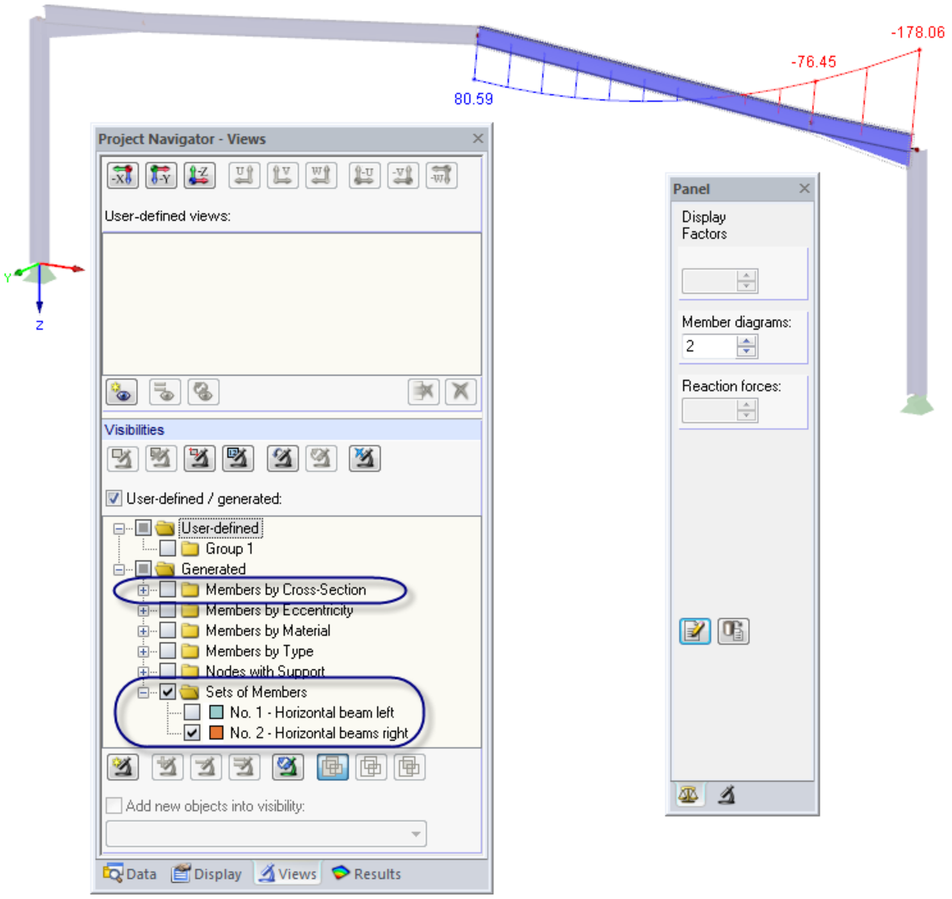
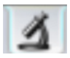


Figure 8.7: Moments M_y of right continuous member

As already described, we can change the display of result categories in the *Results* navigator (see Figure 8.3, page 45). The figure above shows the moment diagram of the right horizontal beam determined for CO5.

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 to display their results exclusively. In contrast to the visibility function, the model will be displayed completely in the graphic.

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

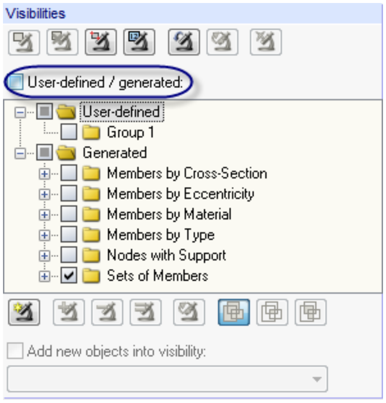


Figure 8.8: Resetting the overall view in the *Views* navigator

We select member *6* (right column) by mouse click. Then, in the panel, we change to the filter tab.

We click the button [Import from Selection] and see that the number of the selected member has been entered into the input field above. Now, the graphic shows only the results of the right column.

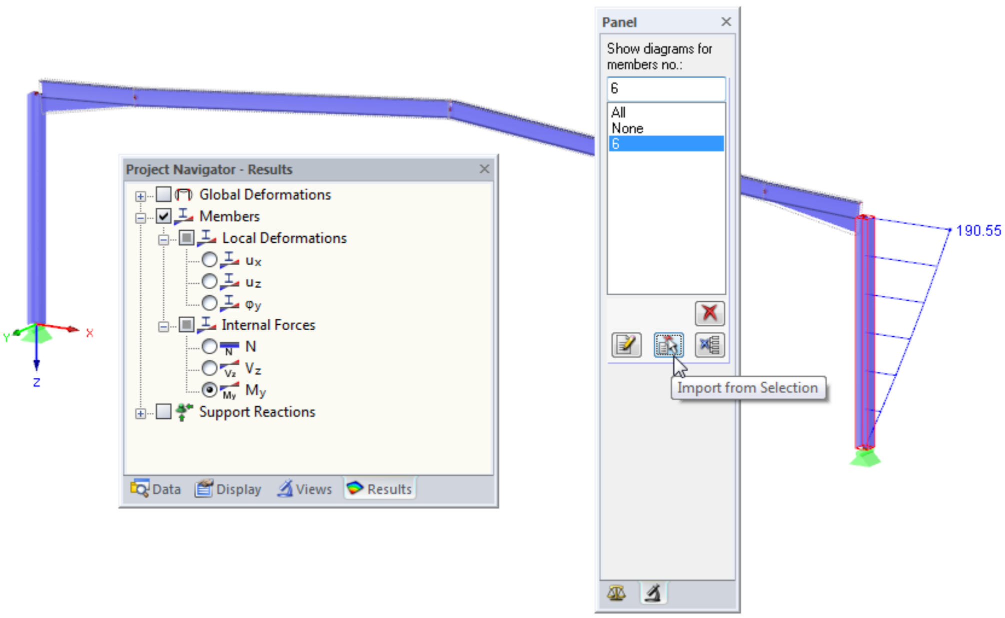
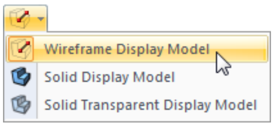


Figure 8.9: Displaying the member moments of the right column

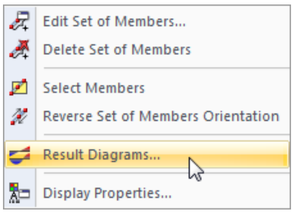
We use the panel option *All* to reset the full display of results.

8.4 Display of Result Diagrams



Another way to evaluate member results is the diagram. Now, we use this function to look at the result diagram of the left continuous member.

The graphical selection of continuous members is easier when the [Wireframe Display Model] is set.



Context menu for set of members

We right-click the set of members *1* and select the entry *Result Diagrams* in the context menu.

A new window opens displaying the result diagrams of the left horizontal beam.

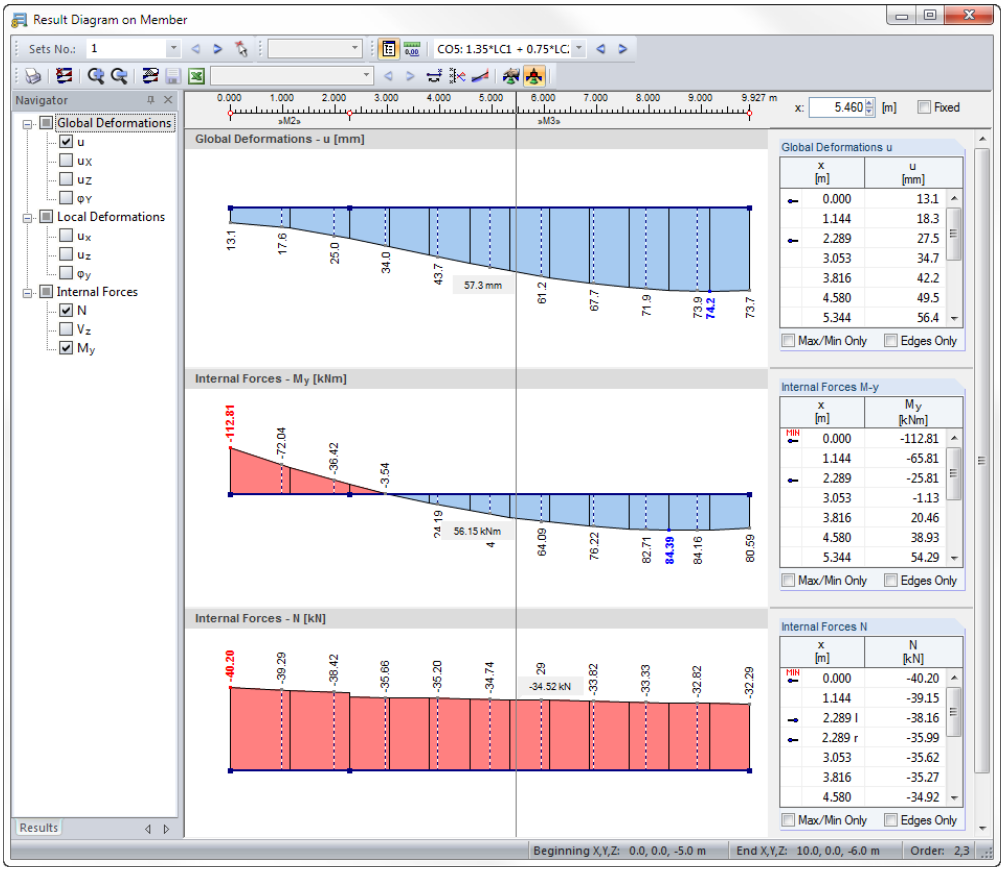
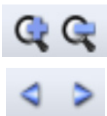


Figure 8.10: Result diagrams of left horizontal beam

In the navigator, we tick the check boxes for the global deformations *u* and the internal forces *N* and M_y.



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

The buttons [◀] and [▶] for load case selection are also available in the result diagram window. But we can also use the list to set the results of a load case. The same buttons are provided for the *Sets* of members selection in the upper left corner of the window: We click them to switch between both continuous members.

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

9. Documentation

9.1 Creation of Printout Report

It is not recommended to send input and output data directly to the printer. Therefore, RSTAB prepares data in a print preview, we call it "printout report". 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 sample for the new printout report.

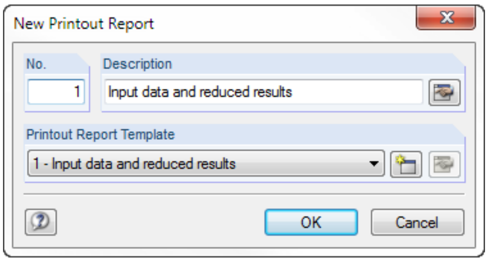


Figure 9.1: Dialog box *New Printout Report*

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

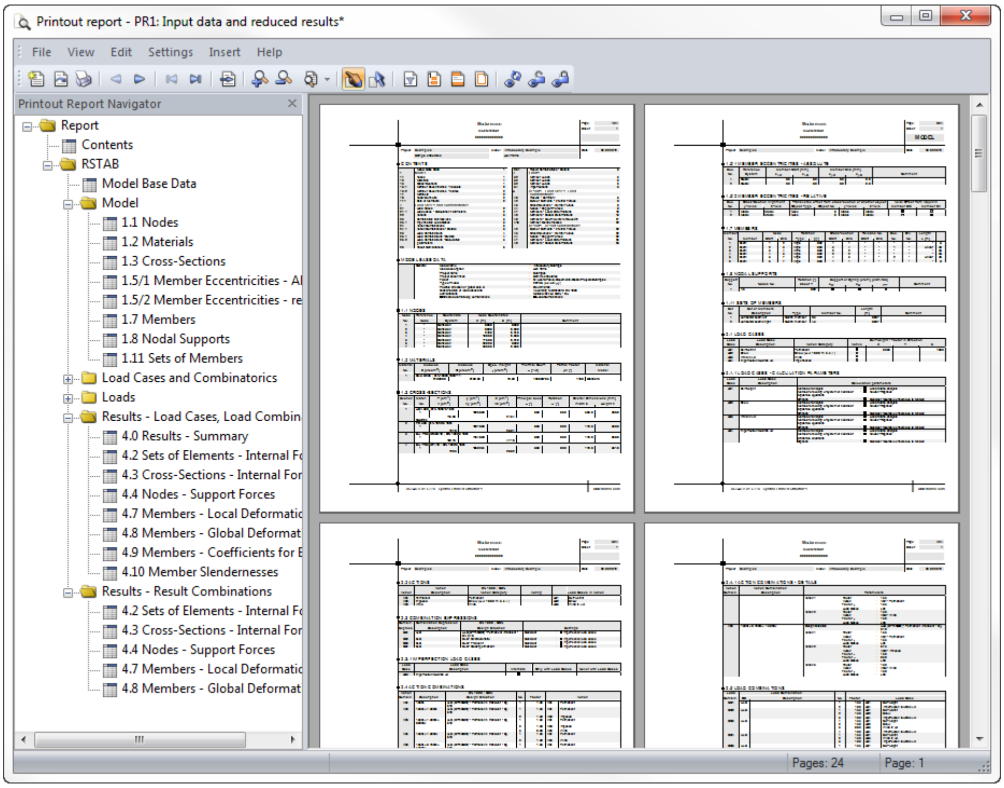


Figure 9.2: Print preview in printout report

9.2 Adjusting the Printout Report

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

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

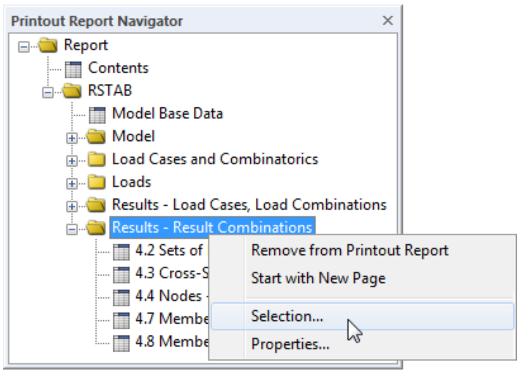


Figure 9.3: Context menu *Results - Result Combinations*

In the dialog box *Printout Report Selection*, we make sure that table row *4.1 Members - Internal Forces* is checked. Then we place the pointer into the table cell to the right. The button [...] becomes available which opens the dialog box *Details - Internal Forces by Member*.

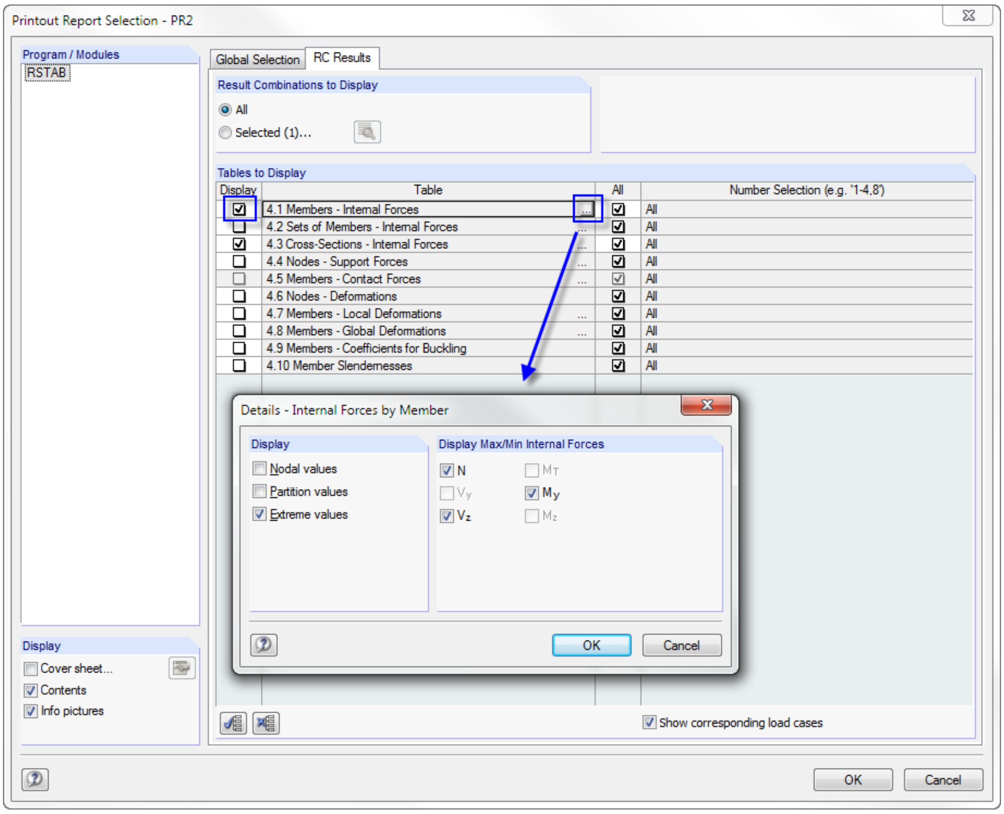
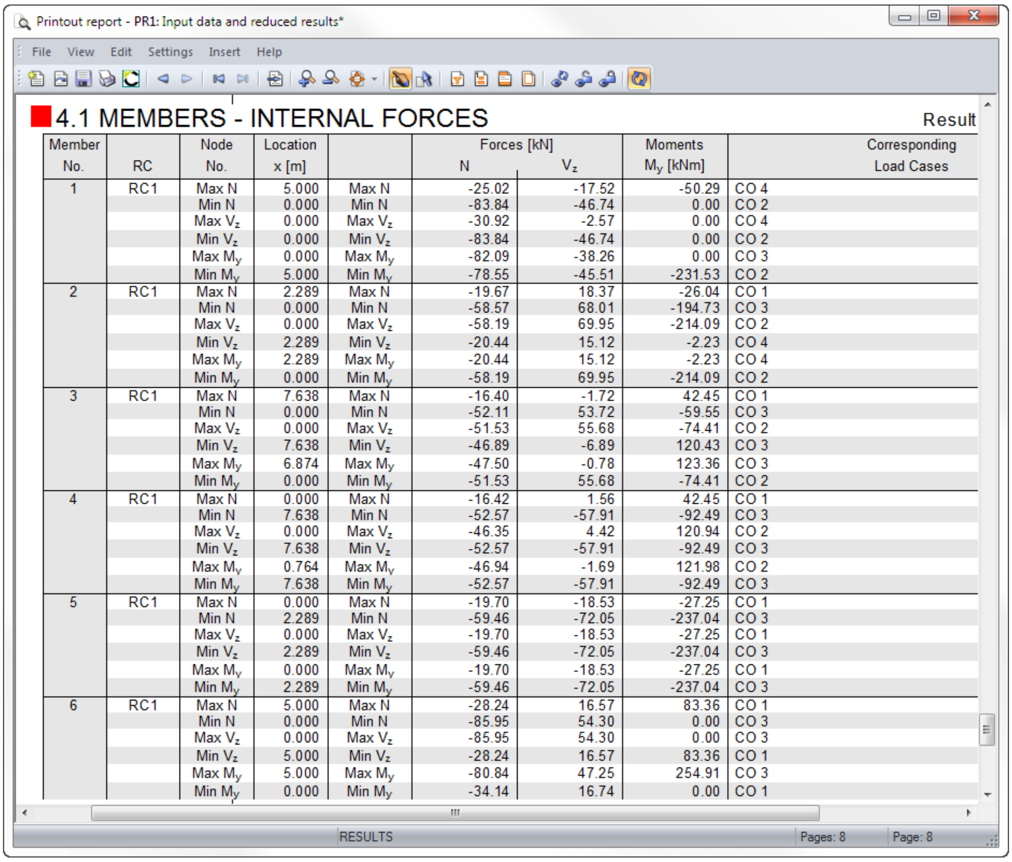


Figure 9.4: Integration of member internal forces in *Printout Report Selection*

We check if only the **Extreme values** of the internal forces are activated for the printout.

After confirming the dialog boxes, RSTAB adjusts the output of internal forces according to our settings.



Printout report - PR1: Input data and reduced results*

File View Edit Settings Insert Help

4.1 MEMBERS - INTERNAL FORCES — Result

Member No.	RC	Node No.	Location x [m]		Forces [kN] N	V_z	Moments M_y [kNm]	Corresponding Load Cases
1	RC1	Max N	5.000	Max N	-25.02	-17.52	-50.29	CO 4
		Min N	0.000	Min N	-83.84	-46.74	0.00	CO 2
		Max V_z	0.000	Max V_z	-30.92	-2.57	0.00	CO 4
		Min V_z	0.000	Min V_z	-83.84	-46.74	0.00	CO 2
		Max M_y	0.000	Max M_y	-82.09	-38.26	0.00	CO 3
		Min M_y	5.000	Min M_y	-78.55	-45.51	-231.53	CO 2
2	RC1	Max N	2.289	Max N	-19.67	18.37	-26.04	CO 1
		Min N	0.000	Min N	-58.57	68.01	-194.73	CO 3
		Max V_z	0.000	Max V_z	-58.19	69.95	-214.09	CO 2
		Min V_z	2.289	Min V_z	-20.44	15.12	-2.23	CO 4
		Max M_y	2.289	Max M_y	-20.44	15.12	-2.23	CO 4
		Min M_y	0.000	Min M_y	-58.19	69.95	-214.09	CO 2
3	RC1	Max N	7.638	Max N	-16.40	-1.72	42.45	CO 1
		Min N	0.000	Min N	-52.11	53.72	-59.55	CO 3
		Max V_z	0.000	Max V_z	-51.53	55.68	-74.41	CO 2
		Min V_z	7.638	Min V_z	-46.89	-6.89	120.43	CO 3
		Max M_y	6.874	Max M_y	-47.50	-0.78	123.36	CO 3
		Min M_y	0.000	Min M_y	-51.53	55.68	-74.41	CO 2
4	RC1	Max N	0.000	Max N	-16.42	1.56	42.45	CO 1
		Min N	7.638	Min N	-52.57	-57.91	-92.49	CO 3
		Max V_z	0.000	Max V_z	-46.35	4.42	120.94	CO 2
		Min V_z	7.638	Min V_z	-52.57	-57.91	-92.49	CO 3
		Max M_y	0.764	Max M_y	-46.94	-1.69	121.98	CO 2
		Min M_y	7.638	Min M_y	-52.57	-57.91	-92.49	CO 3
5	RC1	Max N	0.000	Max N	-19.70	-18.53	-27.25	CO 1
		Min N	2.289	Min N	-59.46	-72.05	-237.04	CO 3
		Max V_z	0.000	Max V_z	-19.70	-18.53	-27.25	CO 1
		Min V_z	2.289	Min V_z	-59.46	-72.05	-237.04	CO 3
		Max M_y	0.000	Max M_y	-19.70	-18.53	-27.25	CO 1
		Min M_y	2.289	Min M_y	-59.46	-72.05	-237.04	CO 3
6	RC1	Max N	5.000	Max N	-28.24	16.57	83.36	CO 1
		Min N	0.000	Min N	-85.95	54.30	0.00	CO 3
		Max V_z	0.000	Max V_z	-85.95	54.30	0.00	CO 3
		Min V_z	5.000	Min V_z	-28.24	16.57	83.36	CO 1
		Max M_y	5.000	Max M_y	-80.84	47.25	254.91	CO 3
		Min M_y	0.000	Min M_y	-34.14	16.74	0.00	CO 1

RESULTS Pages: 8 Page: 8

Figure 9.5: Extreme values of member internal forces in printout report

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

To change the position of a chapter within the printout report, we move it to the new navigator position by using the drag-and-drop function. When we want to delete a chapter, we use the context 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 graphics of internal forces

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 RSTAB.

In the work window, we set the *Results* and select the moments M_y for **CO5**. Then, we put the graphic in an appropriate position.

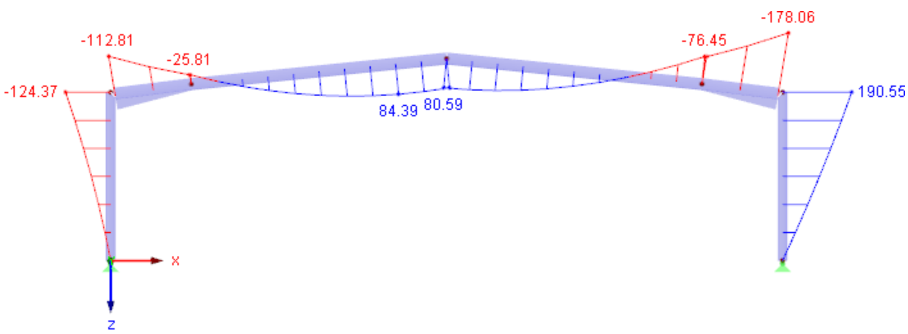


Figure 9.6: Moment diagram of CO5

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 dialog box *Graphic Printout*. It is not necessary to change the default settings in the tab *Options*.

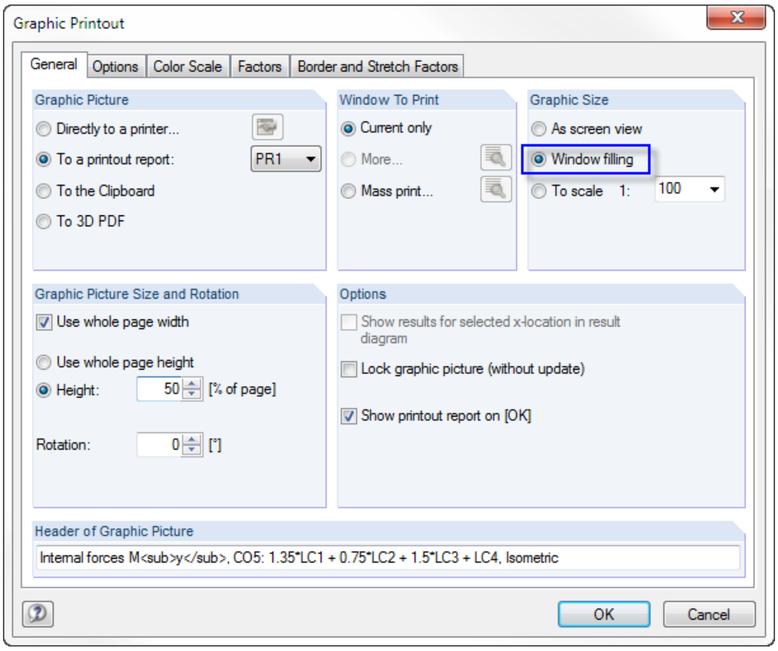


Figure 9.7: Dialog box *Graphic Printout*

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

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

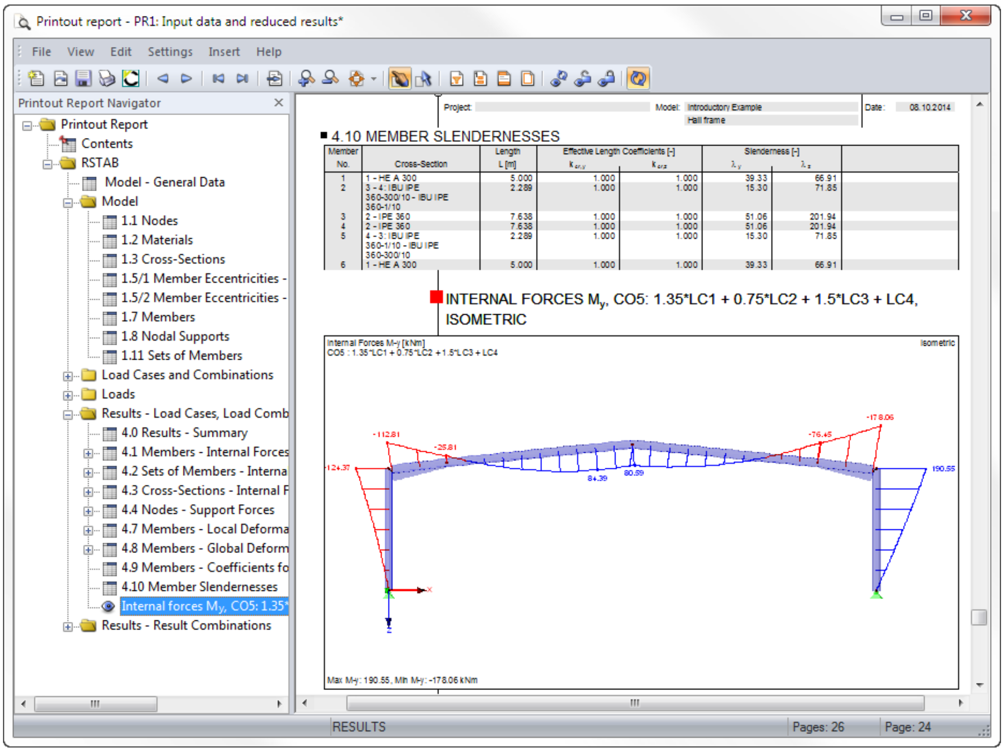


Figure 9.8: Graphical moment diagram in printout report

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 RSTAB makes it possible to put out report data as 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 facilitating the navigation in the digital document.

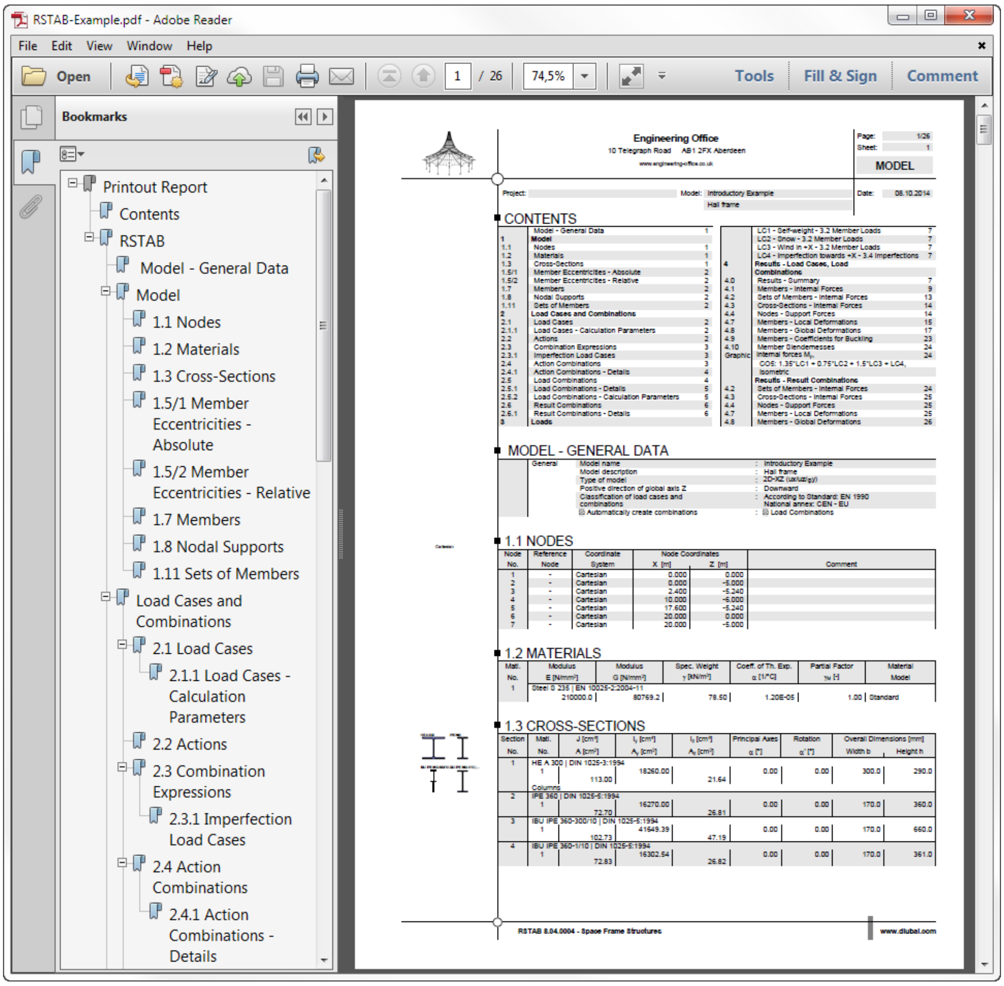


Figure 9.9: Printout report as PDF file with bookmarks

10. Outlook

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

With the **Help** menu or the [F1] key it is possible to open the program's online help system where you can search for particular terms like in the manual. The help system is based on the RSTAB 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 carried out in the add-on modules for the design results (for example STEEL, STEEL EC3, RSBUCK). In this way, you will be able to perform the design, getting an insight into the functionality of the add-on modules. Then, you can also evaluate the design results in the RSTAB work window.