4 Diubal

Version October 2014

Program

RSTAB 8

Structural Analysis for General Frameworks

Introductory Example

All rights, including those of translations, are reserved.

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

© Dlubal Software GmbH Am Zellweg 2 D-93464 Tiefenbach

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



Contents

| | Contents | Page | | Contents | Page |
|-------|----------------------------------|------|-------|--|------|
| 1. | Introduction | 4 | 5.4.2 | Continuous Members with Horizontal Beams | 34 |
| 2. | System and Loads | 5 | 5.5 | Checking Load Cases | 36 |
| 2.1 | Sketch of System | 5 | 6. | Combination of Actions | 37 |
| 2.2 | Materials and Cross-sections | 5 | 6.1 | Checking the Actions | 37 |
| 2.3 | Load | 6 | 6.2 | Defining Combination Expressions | 38 |
| 3. | Creation of Model | 7 | 6.3 | Creating Action Combinations | 40 |
| 3.1 | Starting RSTAB | 7 | 6.4 | Creating Load Combinations | 40 |
| 3.2 | Creating the Model | 7 | 6.5 | Checking Result Combinations | 42 |
| 4. | Model Data | 8 | 7. | Calculation | 43 |
| 4.1 | Adjust Work Window and Grid | 8 | 7.1 | Checking Input Data | 43 |
| 4.2 | Creating Members | 10 | 7.2 | Calculate Model | 43 |
| 4.2.1 | Columns | 10 | 8. | Results | 44 |
| 4.2.2 | Horizontal Beams | 14 | 8.1 | Graphical Results | 44 |
| 4.3 | Connecting Members Eccentrically | 19 | 8.2 | Results Tables | 46 |
| 4.4 | Creating Sets of Members | 22 | 8.3 | Filter Results | 47 |
| 4.5 | Arranging Supports | 23 | 8.3.1 | Visibilities | 47 |
| 4.6 | Changing the Numbering | 24 | 8.3.2 | Results on Objects | 49 |
| 4.7 | Checking the Input | 25 | 8.4 | Display of Result Diagrams | 50 |
| 5. | Loads | 26 | 9. | Documentation | 51 |
| 5.1 | Load Case 1: Self-weight | 26 | 9.1 | Creation of Printout Report | 51 |
| 5.1.1 | Self-weight | 27 | 9.2 | Adjusting the Printout Report | 52 |
| 5.1.2 | Roof Structure | 27 | 9.3 | Inserting Graphics in Printout | 5- |
| 5.2 | Load Case 2: Snow | 28 | 210 | Report | 53 |
| 5.3 | Load Case 3: Wind | 30 | 10. | Outlook | 56 |
| 5.4 | Load Case 4: Imperfection | 32 | | | |
| 5.4.1 | Column Members | 33 | | | |



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.

5

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 crosssections 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/m^2 . For a frame distance of 5 m, the load is:

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

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 $s_k = 0.85 \text{ kN/m}^2$. 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/m² * 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 $q_p = 0.5 \text{ kN/m}^2$. So we get the following wind pressure on the upwind side:

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

For the downwind side we have the following suction load:

 $w_s = 0.5 * 0.5 \text{ kN/m}^2 * 5 \text{ m} = 1.25 \text{ kN/m}$

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:

 $\begin{array}{ll} \mbox{Inclination} & \phi_0 = 1/200 \\ \mbox{Precamber} & e_{0,d} = L/200 \end{array}$

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.

| Model Name | | Description | | |
|------------------------------|-------------|---------------|--|-----------------------|
| Introductory Example | | Hall frame | | |
| Project Name | | Description | | |
| 🚞 Examples | • | Sample Mod | lels | |
| Folder: | | | | * |
| C:\Users\Public\Documen | ts\Dlubal\f | Projects\Exam | ples | |
| Type of Model | | | Classification of Load Cases a | nd Combinations |
| 🔘 1D - in X | | | According to Standard: | National annex: |
| 2D - in XZ | ←→x | | EN 1990 | 👻 🔟 CEN 👻 💽 |
| 🔘 2D - in XY | + | | Create combinations automa | tically |
|) 3D | | | Load combinations | sticely |
| | | | Result combinations (for | linear analysis only) |
| Positive Orientation of Glob | al Axis Z | _ | Template | |
| Upward | | | Open template model: | |
| Downward | | | | |
| Comment | | | | |
| [| | | | - 6 |

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

đ

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

| ork Plane and Grid/Sha | p | _ |
|------------------------|---------------------|---|
| Vork Plane | | Origin of Work Plane |
| Coordinate system: | | Node No.: 🗾 🔻 🖍 🎮 |
| Global XYZ | - 🎦 🔤 | |
| | ★ x | Coordinates X: 0.000 ⊕ h [m] Y: ● h [m] Z: 0.000 ⊕ h [m] |
| Grid/Snap Object Snap | Guidelines Backgrou | nd Layers Line Grids |
| Show | Туре | Number of Grid Points |
| 🔽 Grid | O Cartesian | (+) (-) |
| 🔽 Snap | 🔘 Polar | Direction 1: 30 🖨 30 🖨 |
| | | Direction 2: 30 🚔 30 🚔 |
| | | Dynamically according to size of model |
| | | |
| | | Grid Point Spacing |
| | | Grid Point Spacing Space b: 1.000 구분 [m] |
| | β | Grid Point Spacing Space b: 1.000 ⊕ k Space h: 1.000 ⊕ k |
| | | Grid Point Spacing Space b: 1.000 ψ b [m] Space h: 1.000 ψ b [m] Botation β: 0.00 ψ b [m] |

Figure 4.1: Dialog box Work Plane and Grid/Snap

snapped on the grid when clicking.

SNAP GRID



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.

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

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

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

1. N

2

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.

| ew Member | | × |
|------------------------------|---------------------------------------|--|
| General Option | ns Effective Lengths Modify Stiffness | |
| Member No. | | Member Type |
| 1 | | Beam 👻 |
| Node No. | | Member Rotation |
| Member start: Member end: | | X End |
| Member Rotatio | on via | Z |
| Angle | β: 0.00 🗭 [°] | |
| Help node | No.: 🗾 🔻 🏠 🔭 | Start |
| In plane: | @ x-y | Here |
| | O X-Z | p vo- |
| Cross-Section | | |
| Member start: | Create a new cross-section! | |
| Member end: | As member start | Create New Cross-Section for Member Start. |
| Member Releas | se | |
| Member start: | None | ► ► |
| Member end: | None | Image: Image: Ima |
| 2 🚾 👔 | 8 | OK Cancel |

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**.

| New Cross-Se | ection | | | | 22 |
|--|--|-----------------------|---|---------------------------------------|--------------|
| No. | Color Cross-Section De | escription | | | |
| Rolled Cross-Sections - I-Sections | To Select | | To Select | HEA 300 | × |
| I [T L O O L L I V I Filer Manufacture / Standard court | Table I I I IPE I IPE I IPE I IPE I IPES I IPES I IPES I IPES I IPES I IPES I IPEA I IPEA I IPEA I HEA I HEB I HEB | Manufacturer/Standard | Cross-Section HE A 100 HE A 120 HE A 120 HE A 140 HE A 160 HE A 180 HE A 200 HE A 220 HE A 240 HE A 240 HE A 280 HE A 320 HE A 320 HE A 320 | 23000 | 300.0 8.5 |
| None Manufacturer/Standard: All Cross-section shape: All V Cross-section note: All V | I HSL I HP | | HE A 360 HE A 400 HE A 450 HE A 500 HE A 550 HE A 550 HE A 650 HE A 700 HE A 800 HE A 900 HE A 1000 | E Material 1 - Steel S 235 EN | [mm] |
| Include invalid Favorites group: | | | | ← 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.

0



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

| Member No. | | Member Type | |
|----------------|-----------------------------------|-------------|-----------|
| 1 | | Beam | ▼ 🔤 |
| Node No. | | HE A 300 | |
| Member start: | - 🖏 🎦 | | |
| Member end: | - 😵 🎦 | | N I |
| Member Potatio | n via | | |
| | B: 0.00 € 1°1 | | У |
| U Vilgio | p. 0.00 v 11 | | |
| Help node | No.: 🚽 🗞 音 | | |
| In plane: | (ii) x-y | | 4 |
| | 🔘 x-z | | - |
| Cross-Section | | | |
| Member start: | 1 HE A 300 Steel S 235 Colu | mns | - 🔊 🎦 💽 🤅 |
| Member end: | As member start | | - 🔟 🎦 🖻 🕻 |
| Member Releas | e | | |
| Member start: | None | | - 🎦 🖾 |
| Member end: | None | | - 🍋 🔤 |

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.

| | New Member | | | | • | • | • | • | • | | X: 20. Y: 0. Z: -5. | 000 000 000 |
|-------|--|---------------|------|---|---|---|---|---|---|---|---------------------------|-------------------|
| | 2 Node No. | X: 20.000 | [m] | | | | | | | | ₽. | |
| | Reference Current CS | Z: -5.000 | [m] | • | | | | | | | | |
| | Grid origin Last node | Member Length | (m) | | | • | | • | | | | |
| × | - | Step | [m] | - | | | | | | | | |
| · · · | | | pply | • | • | • | • | • | • | • | | • • |
| | | | | | | | | | | | | |

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**). 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**.

4 Model Data

đ



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*.

| | Repeat Save Model | Enter |
|------------|------------------------------|-------|
| | View | ÷ |
| | User-Defined View | • |
| 123 | Show Numbering | |
| P | Show Loads | |
| 9 | Show Results | |
| ¥. | Show Dimensions | |
| ∠^^ | Show Comments | |
| v | Display Model Wired or Solid | |
| † 1 | Lock Guidelines | |
| | Enable Drag & Drop | |
| ۲ | Auto Rotate Model | |
| ø | Auto Connect Members | |
| p | Coordinate System | |
| 1 | Work Plane, Grid/Snap, | |
| ď | Select Special | |
| | Colors in Graphics According | to 🕨 |
| X = | Display Properties | |

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.

| Project Navigator - Display | × |
|----------------------------------|---|
| 🖃 🗹 💓 Model | |
| 🗹 🔛 Nodes | |
| 🖶 🗹 💓 Nodal Supports | |
| 🖶 🗹 🔛 Members | |
| 🕀 🗹 💓 Sets of Members | |
| 🕀 🗹 💓 Tendons | |
| 🗄 🗆 🕒 Loads | |
| 🖶 🗆 🚾 Results | |
| 🚋 🗐 💎 Guide Objects | |
| 🛓 🗐 🔂 General | |
| | |
| V 123 Nodes | |
| | |
| 🗀 🗹 💷 Members | |
| Member Numbering | |
| Material Numbering | |
| Cross-Section Numbering | |
| Member End Releases | |
| | |
| ✓ III Guidelines | |
| Colors in Rendering According to | |
| 🖶 🔲 🍑 Rendering | |
| Preselection | |
| 🗄 🗐 🚴 Add-on Modules | |
| 🔯 Data [Display 🕺 Views | |

Figure 4.8: *Display* navigator for numbering



4.2.2 Horizontal Beams

Placing members

2

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.

| Uptio | ns Effective Lengths Modify Stiffness | | | | |
|----------------|---|-------------|-----------|-----------------|-----------------------|
| Member No. | | Member Type | | | |
| 3 | | E Beam | - | 2 | |
| Node No. | | HE A 300 | | | |
| Member start: | 4 👻 🖏 🚰 | | | | |
| Member end: | 4 🔻 🏷 🛅 | | l l | | |
| Member Rotatio | on via | | ·····• | | |
| Angle | β: 0.00 🖈 [°] | | | | |
| Help node | No.: 4 👻 🏷 🛅 | | | | |
| In plane: | (i) x-y | | * | | |
| | 🔿 x-z | | | | |
| Cross-Section | | | | | |
| Member start: | 1 HE A 300 Steel S 235 Column | 5 | 🔊 🎦 🖉 | 2 📵 📔 | |
| Member end: | As member start | | ▼ 🛍 웥 Cre | ate New Cross-S | ection for Member Sta |
| Member Releas | se | | _ | | |
| Member start: | None | | - 🎦 📨 | | |
| Member end: | None | | - 🚑 📼 | | |

Figure 4.9: Dialog box New Member

| New Cros | -Section | | | | | |
|--|--|---|--|---|-----------------|------|
| No. | Color Cross-Sect | on Description | | | REC DE | |
| Rolled Cross-Sections - I-Section | Transfer. | | | | | × |
| Cross-Section Type I I | To Select Table I I I PE I IPE I IPE 750 I IPEa I IPEo I IPES I IPES I IPESB I IPESB I HEA I HEA I HEA I HEB I HEB I HEM I HSL I HP | Manufacturer/Standard Manufacturer/Standard | To Select Cross-Section IPE 80 IPE 100 IPE 120 IPE 120 IPE 140 IPE 160 IPE 200 IPE 200 IPE 200 IPE 200 IPE 200 IPE 200 IPE 400 IPE 400 IPE 450 IPE 500 IPE 550 IPE 550 | 1PE 360 | | -•y |
| Cross-section shape: All Cross-section note: All Cross-section note: Include invalid Favorites group: The section of the section o | | | IPE 600 | Material 1 - Steel S 235 EN IPE 360 | 10025-2:2004-11 | [mm] |

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.

| | | | | | | 5 | | | | | | |
|-------|-----|--|---|--|-------|---|---|---|--|---|---|-------|
| | | | 3 | | | 5 | _ | 4 | | | - | |
| 2 | | | | | | | | | | | | |
| | | | | | | | | | | | | |
| | | | | | | | | | | | | |
| 1 | | | | | | | | | | | | 2 |
| | | | | | | | | | | | | |
| 1 | - × | | | | _ | | | | | _ | _ | 3 |
| 2 | | | | | | | | | | | | |

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 <u>right</u> mouse button. The member context menu opens where we select **Divide Member** by **Distance**.



Figure 4.12: Context menu of member

4 Model Data



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*.

| Member No. | | | _ | |
|------------------|------------|--------|------------|--|
| 3 | | | 3 | X |
| Distance Relativ | re to | | | Z |
| True | | Length | | |
| member leng | jth: | 10.050 | [m] | |
| Projection in | ο Y· | 10.000 | [m] | New Node |
| direction | ● <u>∩</u> | 0.000 | [m] [m] | |
| | 07. | 1.000 | [11] [1 | and the second sec |
| | U 2. | 1.000 | լոյ | |
| Projection into |) YZ: | 1.000 | [m] | Distance Between New Node and |
| plane |) XZ: | 10.050 | [m] | Member start: 2.400 - [m] 24.000 - [%] |
| | O XY: | 10.000 | [m] | Member end: 7.600 km [m] 76.000 km [%] |
| | | | | Numbering Starts with |
| | | | | No. |
| | | | | Node: 0 🚽 🗹 Continuous |
| | | | | Member: 🛛 🕞 📝 Continuous |

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*.

| Member No. | | Member Type |
|---------------------------|-------------------------|---------------------------------|
| 3 | | Beam 🔻 🖾 |
| Node No. | | IPE 360 |
| Member start: | 2 🗸 🏹 🎦 | |
| Member end: | 6 🚽 🚯 🛅 | |
| Member Rotatio | n via | |
| Angle | в: 0.00 🖘 [°] | У |
| | | |
| Help node | No.: 4 👻 🍾 🛅 | |
| In plane: | @ x-y | ÷ |
| | 🔘 x-z | |
| Cross-Section | | |
| Member start: | I 2 IPE 360 Steel S 235 | 🗖 🗕 🔁 💽 👩 |
| Member end: | As member start | ▼ Import from Cross-Section Lib |
| Member Releas | se | |
| Member start: | None | - 🔁 🗃 |
| Member end: | None | |

Figure 4.14: Dialog box Edit Member

4 Model Data



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

| Cross-Section Library | | | X |
|-------------------------------------|---------------------------------------|----------------------|----------------------------|
| Rolled | Parametric - Thin-Walled | Parametric - Massive | Parametric - Timber |
| ILTL | ΙΙΤΓΤ | | |
| 004 | TLL | | |
| 1 • • • | LITT | | ΤΤΠΠ |
| | Ο Π Π | T L I J | ΤΙΠΠ |
| Built-up | ΠΠΠ | | |
| II I T T | Į I + • | | |
| TIIII | - 1 l J | | |
| I I I-Sections | with Cut I-Sections, Flat Bars or T-S | Sections | Standardized - Timber |
| ••] | ΣΟ | | |
| ICU - ICO - IBU - IBO - SFBo - SFBu | - ICTo - ICTu | User-Defined | From Cross-Section Program |
| | | | |
| (2) (3) | | | Cancel |

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.

Ŧ



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.

| General Option | ns Effective Lengths Modify Stiffness | | |
|----------------|---|--------------------|---------|
| Member No. | | Member Type | |
| 3 | | Beam | ▼ 🐼 |
| Node No. | | IBU IPE 360-300/10 | |
| Member start: | 2 🗸 🏹 🎦 | IBU IPE 360-1/10 | |
| Member end: | 6 🔫 🍢 🎦 | | |
| Member Rotatio | n via | | |
| Angle | β: 0.00 👘 [°] | | * |
| Help node | No.: 4 🚽 🕵 🐃 | | |
| In plane: | | | |
| in plane. | ⊙ x-z (ð |] | Ļ |
| Cross-Section | | | |
| Member start: | ■ ∓ 3 IBU IPE 360-300/10 Steel S 23 | 15 | 🔊 🎦 💽 🖯 |
| Member end: | ■ ∓ 4 IBU IPE 360-1/10 Steel S 23 | 15 | 🔊 🎦 🐼 🖯 |
| Member Releas | e | | |
| Member start: | None | | - 🎦 📨 |
| Member end: | None | | - 🎦 📨 |

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**.

| Cross-Section | | | | | | |
|---------------|------------|--------------------|-------------|---------|-----|-------|
| Member start: | ∎∓4 | IBU IPE 360-1/10 | Steel S 235 | | | 🔊 🔁 🔁 |
| Member end: | ∎∓3 | BU IPE 360-300/10 | Steel S 235 | | 🗖 🗖 | 🔊 🔁 🔁 |
| | As memb | er start | | | 4 | 6 |
| | | HE A 300 | Steel S 235 | Columns | | |
| | I 2 | IPE 360 | Steel S 235 | | | |
| | | IBU IPE 360-300/10 | Steel S 235 | | | |
| | 🗖 ∓ 4 | IBU IPE 360-1/10 | Steel S 235 | | | |

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

2

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*.

| Edit Member General Options Effective | Lengths Modify Stiffness | 2 |
|---|--|---|
| Member No. | | |
| None | | |
| New Member Eccentricity | | Alkaplita Offect |
| 1 | * | |
| Absolute Offset Reference system | Relative Automatically Offset Cross-section alignment | |
| Local x,y,z Global X,Y,Z | | i · · · · · · · · · · · · · · · · · · · |
| Member start i ei, x 0.0 h [mm] ei, y (mm] | Transverse offset from cross- section of other object | i' y' |
| ei,Z 0.0 (mm) | No. Member: 1 🗸 | Axial Offset |
| Member end j ej,x 0.0 ⊕ ≥ [mm] ej,y ↓ [mm] ei,z 0.0 ⊕ ≥ [mm] | Axis offset | |
| | Axial offset from adjoining members at: | Comment |
| | Member end | |
| | | OK Cancel |

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_{iz} 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

1

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

F

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.

| No. | Description Horizontal beam left | |
|--|-------------------------------------|--|
| Type Continu Group of Members N | ous members of members | |
| 3,5 | | |
| Comment | _ (6) | |

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



6

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.

| 5 | |
|--|---|
| 47_ | |
| 6 | 4 |
| · · · Set of Members · · · · · · · · · · · · · · · · · · · | |
| · · · · · · · · · · · · · · · · · · · | |
| Clear OK Cancel 2 | |
| | |
| | 3 |

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.

Figure 4.23: Dialog box New 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.

| New Nodal Support | | × |
|-------------------|-------|-----------|
| Node No. 1,3 | 1 | |
| Type of Support | - 🛅 🖻 | * |
| | | |
| | | |
| | | |
| | | |
| | | |
| | | OK Cancel |



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.



8



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.

| 1st Priority | 2nd Priori | ity | 3rd Priority | |
|---------------------|------------|-------------|---------------------|--|
| Coordinate on axis: | Coordina | te on axis: | Coordinate on axis: | |
| ۵X | ΟX | | ΟX | |
| ΟY | ΟY | | Y | |
| ⊚ Z | © Z | | ΟZ | |
| Direction: | Direction | | Direction: | |
| Positive | 🔘 Positi | ve | Positive | |
| 🔊 Negative | Negal | tive | Negative | |
| Selected Objects | | | | |
| Torenumber Fin | st number | Increment | | |
| V Nodes | 1 🚔 | 1 🚔 | | |
| Members | 1 🖨 | 1 🚔 | | |
| | | | | |
| | | | | |
| | | | | |
| | | | | |

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.

| dit Load Case | es and Co | ombinations | | | | |
|---------------|-----------|-------------------------|----------|---|-------------------------------------|-----------|
| Load Cases | Actions | Combination Expressions | Action | Combinations Lo | ad Combinations Result Combinations | |
| Existing Loa | d Cases | | | LC No. | Load Case Description | Solve |
| G LC1 | Self-w | /eight | ^ | 1 | Self-weight | ▼ |
| G LC1 | Sef-w | eght | E | 1 General Calcu Action Type Self-Weight V Active Factor in dir X: 0.0 Y: Z: 1.0 | Istion Parameters | |
| | | B ZV IX II | X | | | |
| ۵ (ک | | | | | | OK Cancel |

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



Project Navigator - Data

🗉 📄 Model Data

Doads

Results

Printout Reports
Guide Objects

🖶 🛅 Add-on Modules

Load Ca

introductory Example [Examples]

🗄 📹 Load Cases and Combinations

E Combination Expressions

Action Combinations

Result Combinations Super Combinations



Self-Weight Active

X:

7.

Factor in direction 0.000 \$

1.000 ≑



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].

| 10. | Reference to | On men | nbers No. | | Load Type 'Force' Load Distribution 'Uniform' | |
|-----------------------------------|---|--|--|--------------------|---|----------|
| | List of men Sets of me | nbers mbers | | s à 7 | | ∼ P |
| oad Type | | Load Distribution | Load Direction | Global: | | |
| Moment | | P - | © × | © X | i• | • |
|) Temperatu | ure | Uniform | © z | © Z | | |
| O Axial strain O Axial displation | n acement | Trapezoidal Tapered | | | | |
| Precamber | a. | O Parabolic | Reference Ler | ngth Ier length | 1 | |
| Extra: | displacement 🔻 | | Projection i Projection i Projection i | nX nY nZ | Load Direction 'Global Z' Ref. Length 'True Member Length' | i y |
| oad Paramet | ters | | | | | z |
| : 1)2: | [.500 € [kN/m] | A:B: | (m) | | | _i |
| 02: | [kN/m] | Relative Load ov Member | edistance in % er total length of | | × | A |
| Comment | | | | - | Zt i | |
| | | | | | | |

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.



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).

| Edit Load Cases and Combinations | | × |
|--|--|-----------|
| Load Cases Actions Combination Expressions | Action Combinations Load Combinations Result Combinations | |
| Existing Load Cases | LC No. Load Case Description | Solve |
| G LC1 Self-weight | ▲ 2 Snow ▼ | |
| Ca LC2 Snow | General Calculation Parameters Action Type EN 1990 CEN Self-Weight Imposed - category A: domestic, residential areas 3.A Imposed - category B: office areas 3.B Imposed - category C: congregation areas 3.C Imposed - category D: shopping areas 3.D Imposed - category | |
| | | |
| | C | OK Cancel |

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].

5 Loads

μų.



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**.

| 0. | Reference to | On | members | No. | | Load Type 'Force' |
|--|---|---|------------------------------------|--|--------------------------------|---|
| 1 | Members List of members Sets of members | 2- bers hbers | 5 | | s (*) | Load Distribution 'Uniform' |
| oad Type | | Load Distribution | | Load Direction | | |
| Force Moment Temperature Axial strain Axial displacen Precamber Initial prestress Extra: Imposed displaced | nent acement 💌 | Concentratec P Uniform Trapezoidal Tapered Parabolic Varying | | Local: x y z Reference Length True member let Projection in X Projection in Z | Global: X Y Z ngth | i Load Direction 'Global Z' Reference Length 'Projection in Z' |
| oad Parameters | | | | | | Z |
| 3.400 3.22 22 22 22 22 22 22 22 22 22 22 22 22 | kN/m kN/m kN/m kN/m kN/m kN/m | A: B: Re Loa Me | lative dista ad over to mber | ★ [m] ★ [m] ★ [m] ance in % (al length of | • | z, i i i i i i i i i i i i i i i i i i i |

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].





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**.

| ad Cases | Actions | Combination Expressions | Action C | Combinations | Load Combin | ations | Result Combinations | | |
|-------------|------------|-------------------------|----------|--------------|-----------------|---------|---------------------|---|-------|
| kisting Loa | d Cases | | | LC No. | Load | Case De | scription | | Solve |
| G LC1 | Self-v | veight | * | 3 | Wind | in +X | | • | |
| Ds LC2 | Snow | 1 | _ [] [] | | | | | | |
| tw LC3 | Wind | in +X | | General | alculation Para | ameters | | | |
| | | | | Action Type | • | | EN 1990 CEN | | |
| | | | | Qw Wind | 1 | | - | | |
| | | | | Self-Weight | t | | | | |
| | | | | Active | | | | | |
| | | | | Factor in | direction: | | | | |
| | | | | ×: | * * | | | | |
| | | | | Y: | | | | | |
| | | | | Z: | × | | | | |
| | | | ≡ | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | * | Comment | | | | | |
| - | | | X | | | | | - | |
| رلت رب | | | | | | | | | |
| | - - | | | | | | | _ | |

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.





| 0. | Reference to | | On membe | rs No. | | Load Type 'Force' | |
|-----------------|--------------|-----------------------------|----------------|---------------------------------|------------|---------------------------------|-----|
| 1 | Members | | 1,6 | | | Load Distribution 'Uniform' | |
| | List of men | nbers | | | 🖏 🔊 | | |
| | Sets of me | mbers | | | | | P |
| oad Type | | Load Distrib | ution | Load Direction | | | |
| Force | | 🔘 Concenti | ated: | Local: | Global: | | |
| Moment | | P | - | © × | X | i• | +j |
|) Temperatur | в | I Iniform | | O y | OY OZ | | |
| Axial strain | | C Trapezoi | dal | 2 | <u> </u> | | |
| Axial displace | ement | Tapezor | | | | | |
|) Precamber | | Parabolic | | Reference Lengt | th | | |
| | | 🔘 Varying | | True member | length | | |
| Initial prestre | 325 | | | Projection in 2 | × | Load Direction 'Global X' | |
| | | | | Projection in ' | Y | Ref. Length 'True Member Length | |
| Imposed di | splacement 👻 | | | Projection in 2 | Z | X | ×× |
| <u> </u> | | | | | | | i y |
| oad Paramete | rs | | | | | 2 | ž |
| : 2.0 | 000 🔃 [kN/m] | A: | | > [m] | | | |
| 2: | € [kN/m] | B: | | <u>^</u> ▶ [m] | | | |
| c . | ↓ [kN/m] | |] Relative dis | tance in % | | B | |
| 2: | € [kN/m] | | Load over I | otal length of | | × ~/ | |
| | | | member | | | zt i | |
| omment | | | | | - 6 | | |
| | | | | | - 1 | | |

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.

| Load | Parameters | | |
|------|----------------|-------------------------------------|--|
| p: | 1.250 🕀 [kN/m] | A: [m] | |
| p2: | kN/m] | B: [m] | |
| p: | kN/m] | Relative distance in % | |
| p2: | kN/m] | Load over total length of Member | |

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







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**.

| Load Cases | s and Complinations | | | | |
|--------------|---------------------------------|--------|----------------------|----------------------------------|----------|
| ad Cases | Actions Combination Expressions | Action | Combinations Load | Combinations Result Combinations | |
| cisting Load | Cases | | LC No. | Load Case Description | Solve |
| G LC1 | Self-weight | - | 4 | Imperfection towards +X | - ▼ |
| Ds LC2 | Snow | | Gammal C. L. Lui | D | |
| mp LC4 | Imperfection towards +X | - 11 | Certeral Calculation | on Parameters | |
| | | | Action Type | EN 1990 CEN | |
| | | | Imp Imperfection | · · · · · · | |
| | | | Self-Weight | | |
| | | | Active | | |
| | | | Factor in direction | on: | |
| | | | ×: [| A V | |
| | | | Y: | | |
| | | | Z: | <u>↓</u> | |
| | | E | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | - | Comment | | |
| <u>-</u> | 🚰 📞 🖉 🗮 🛱 | × | | | ▼ 1 |
| | | | | | |
| | - | | | | OK Canor |

Figure 5.13: Dialog box Edit Load Cases and Combinations

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

P



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.

| No. | | Reference to | On members No. | | |
|----------------------|------------|---|---|----------------|----------------------------|
| 1 | | Members List of members Sets of members | () () () () () () () () () () () () () (| | α≠0 ⁻ x u |
| Direction | | Parameters | | | |
| Local (axis: (|) y O z | Reference: | ● Relative ◯ Absolute | ž | ž 'v |
| Principal (axis: |) u v | Inclination 1/φ ₀ : | 200.00 🔃 💽 🕄 | ⁸ → | |
| | | Precamber L/ep : | 200.00 🛬 [·] | | <i>!</i>] |
| | | Precamber activity criterion: | EN 1993-1-1 (5.8) 🔹 🕄 | | eo |
| | | 8 | 0: | φ. 1 | |
| Comment | | | | * / | ►y/z |
| | | | - | | u/v |
| (a) [| ו | | | | |

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* $1/\varphi_0$ 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.



9



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.

| Category | Sets of Members | |
|---|------------------------|--|
| Nodes Cross-Sections | All | |
| Member Eccentricities Members | O With number: | |
| Nodal Supports | With description: | Without |
| Sets of Members | With type: | Continuous Members 🗾 🚺 |
| | With comment: | All 🔹 🐼 |
| | With member: | |
| | With length: from: | 0.000 🔃 🐧 to: 0.000 🔃 [m] |
| | In range: from: to: | X [m] Y [m] Z [m] 0.000 ⊕ k ⊕ * 0.000 ⊕ k 0.000 ⊕ k ⊕ * 0.000 ⊕ k 0.000 ⊕ k ⊕ * * |
| Status | | |
| Add Select from current selection | | |
| Hemove from current selection | | |

Figure 5.16: Dialog box Special Selection

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

| ew Imperfe | ection | | | | × |
|----------------------|-------------------|--|--|-------------|-----------|
| No. 2 | | Reference to Members List of members Sets of members | On sets of members No. | α=0° | |
| Direction Local (|) y | Parameters Reference: | Relative | | 2 V |
| Principal (axis: | ● z ○ u ○ v | Inclination 1/φ ₀ : Precamber L/e ₀ : | Absolute 0.00 ★ [-] 250.00 ★ [-] | j • | |
| | | Precamber activity criterion: | EN 1993-1-1 (5.8) - (1) | φ. 1 × 1 | |
| Comment | | | - 🗟 | i 🌽 | y/z |
| D | | | | | OK Cancel |

Figure 5.17: Dialog box New Imperfection

| or the horizontal beams we |
|---|
| ich side. This time we select |
| selecting continuous mem |
| we select Select on the I |
| we use the toolbar button |

RSTAB Introductory Example $\ensuremath{\mathbb{S}}$ 2014 Diubal Software GmbH



The numbers of both sets of members are preset.

We set the *Inclination* $1/\varphi_0$ to **0** and change the value of the *Precamber* L/e_0 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*.

| 200.1 | | U/ 250.00 1/P的可见 |
|---|---|------------------|
| dit Imperfection | | × |
| No. 2 | Reference to On sets of members No. Members 2 List of members 🔊 Sets of members | $a = 0^{\circ}$ |
| Direction Local y axis: © z Principal u axis: v | Parameters Reference: | |
| Comment | | |

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.





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.

| ons | <u>A</u> dd-on Modules | <u>W</u> indow <u>H</u> | elp |
|----------|------------------------|-------------------------|--|
| <u>*</u> | LC3 - Wind in +X | - 🗸 | > 😢 💯 🔍 🕪 📾 📾 🖩 🎬 聯 🌚 🛱 🎘 😕 🔿 🛝 |
| đ (| R 🔍 🗊 🗗 I 🕅 | YI 🛱 井 | Previous Load Case, P J N Vy Vz Mr My Mz Py Pz V J II II |

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



Ļ



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.

| 2.1 Load | Cases | 5 | ◎ 🕄 🗙 🗣 🔤 🛄 | | 8 61 1 | | $ab=f_X$ | ∛ x | × |
|----------|----------------------------|---------------|-------------------------------------|--------------|--------------------|------------------|----------|------------|---|
| Load | Table 2. Load Cases and Co | ombinatorics | C EN 1990 CEN | D Self-v | E veight - Fact | F or in Direc | G | Н | - |
| Case | Description | To Solve | Action Category | Active | X | Y | Z | Comment | |
| LC1 | Self-weight | 2 | G Permanent | V | 0.000 | | 1.000 | | Ε |
| LC2 | Snow | 2 | Qs Snow (H <le> 1000 m a.s.l.)</le> | | | | | | |
| LC3 | Wind in +X | V | Qw Wind | | | | | | |
| LC4 | Imperfection towards +X | 2 | Imp Imperfection | | | | | | - |
| Load Cas | ses Actions Combination Ex | pressions Act | ion Combinations Load Combinations | s Result C | ombinations | | | | |
| Type of | Load Case | | | | | | | | |

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*.

| 2.2 Actio | ns | | | | | | | × |
|-----------|--------------------------|-------------------------------------|----------------|-------------|------------|-------|------------------------|---|
| | 1 🛛 🖂 🖷 📾 🍤 | 🧲 🛇 💀 🕮 🕃 🕅 達 e | | 1 📫 🚮 | 69 2 | s 📰 . | $ab=f_x \mathcal{Y}_x$ | |
| | A | В | С | D | E | F | G | |
| | Action | EN 1990 CEN | | Load | Cases in A | ction | | |
| Action | Description | Action Category | Acting | LC.1 | LC.2 | LC.3 | Comment | |
| A1 | Permanent 🗾 | G Permanent | | LC1 | | | | Ξ |
| A2 | Snow | Qs Snow (H ≤ 1000 m a.s.l.) | | LC2 | | | | |
| A3 | Wind | Qw Wind | | LC3 | | | | |
| A4 | | | | | | | | - |
| Load Cas | es Actions Combination E | expressions Action Combinations Loa | d Combinations | Result Corr | binations | | | |
| Descripti | on | | | | | | | |

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.

| 2.3 Comb | pination Expressions | | | | | | × |
|----------|-------------------------|----------|--|---------------------|-------------------|--------------|---|
| 2 | 1 📰 🖂 । 📾 📑 📑 | 3 6 | 🛇 😔 📖 🐹 🌬 🔤 💾 🧱 🞑 | 武 🐼 🗷 🖩 | $ab = f_x - f_x$ | | |
| | A | В | С | D | E | F | |
| Combin. | Combination Expression | | EN 1990 CEN | | Consider | | |
| Express. | Description | Use | Design Situation | Favorable G Actions | Imperfection LC's | Ex/Inclusive | 1 |
| CE1 | ULS | 2 | STR ULS (STR/GEO) - Permanent / transient - Eq. 6.10 | | | | Ξ |
| CE2 | SLS | | S Ch SLS - Characteristic | | | | |
| CE3 | SLS | | S Fr SLS - Frequent | | | | |
| CE4 | SLS | | S Qp SLS - Quasi-permanent | | | | Ŧ |
| • | | \sim | | | | | • |
| Load Cas | ses Actions Combination | n Expres | sions Action Combinations Load Combinations Result (| Combinations | | | |

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*.

| 2.3 Com | bination Expressions | | | | | | | × |
|----------|------------------------|----------|---|--------------------|---------|-------------------|--------------|---|
| 2 | 3 🗔 🖂 🤐 😁 🦰 | 3 6 0 | 😔 🗟 🔣 💌 達 🔤 🛄 | 2 📸 😽 | × . | $ab = f_x - f_x$ | | |
| | A | B | С | D | | E | F | |
| Combin. | Combination Expression | | EN 1990 CEN | | | Consider | | |
| Express. | Description | Use | Design Situation | Favorable G | Actions | Imperfection LC's | Ex/Inclusive |) |
| CE1 | IULS 🔹 | V S | R ULS (STR/GEO) - Permanent / transient - Eq. (| 6.10 | | | | Ξ |
| CE2 | Copy Row | CrtI+2 | h SLS - Characteristic | | | | | |
| CE3 | Trunks Dave | Challery | r SLS - Frequent | | | | | |
| CE4 | | Cui+i | D SLS - Quasi-permanent | | | | | Ŧ |
| • | Insert Row | Ctrl+I | | | | | | Þ |
| Load Ca | 📄 Delete Row | Ctrl+R | s_Action Combinations Load Combinations Re | esult Combinations | J | | | |
| | 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.

| oad Cases | Actions | Combination Expressions | Action | Combinations Load | Combinations | Result Combinations | | |
|---------------|------------|-------------------------|----------|-------------------------------------|--------------------|-------------------------------------|---|-------------------|
| Existing Corr | bination E | xpressions | | CE No. | Combination E | xpression Description | n | Use |
| STR CE1 | ULS | | A | 1 | ULS | | • | V |
| S Ch CE2 | SLS | | | | | | , , | |
| S Fr CE3 | SLS | | | General | | | | |
| S Qp CE4 | SLS | | | Design Situation | | | EN 1990 CEN | |
| | | | | STR ULS (STR | GEO) - Permane | nt / transient - Eq. 6.1 | 10 🗸 🔒 | |
| | | | | 010 010 (0110 | acoy romana | and a definition of the Ltd. of the | | |
| | | | | Settings | | | | |
| | | | | Consider: | | | | |
| | | | | Favorable per | nanent actions | | | |
| | | | | Imperfection lo | ad cases | * | | |
| | | | | Exclusive/Incl | usive load cases | | | |
| | | | | Beduce number of | f generated com | hinations | | |
| | | | | due to: | r generated con | ion ration io | | |
| | | | Ξ | Examining resu | ilts | | | |
| | | | | Leading variab | le actions | | | |
| | | | | Restriction of r | number of load c | ases | | |
| | | | | All combination | tions of listed LC | s | | |
| | | | | All listed LC | 's always togethe | er | Result Combinations | |
| | | | | | | | Generate additionally Either/Or | esult combination |
| | | | | Numbering of Ger | erated Combina | tions | (result envelopes) | |
| | | | | First number of ge | nerated | | Discreate Either/Or result combination expression | nation for each |
| | | | | Load combination | s: | 1 | | |
| | | | | Result combinatio | ns: | 1 ≑ | | |
| | | | | Method of Analys | s | | Generated Action Combinations | |
| | | | | Method of analysi | s: Second-orde | er analysis (P-Delt: 🔻 | List and number: AC1 AC5 /5 | (5) |
| | | | | | | | as and number. No I No J (a | ~ |
| | | | - | Comment | | | | |
| | 1 | | _ | | | | - 6 | |

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.

| ttings | | | × |
|-----------|---|---------------------------|---|
| Imperfect | tion Load Cases | | |
| Use Loa | ad Cases of Type 'Imperfection' | | |
| LC | Load Case Description | Use | |
| LC4 | Imperfection towards +X | | |
| | | | |
| | | | |
| | | | |
| | | | |
| | | | |
| | | | |
| | | | |
| | | | |
| | | | |
| | | | |
| | | | |
| Ontions | | | |
| A | for the second | | |
| V All im | iperfection load cases as alternation | ve | |
| | | d a surplication with and | |
| vitho | ove co-existence of the same load out imperfection | combination with and | |
| | فريطة بريمية تحتطيهم أحجا الحجري | in a fastian | |
| | ove all load combinations without | Impenection | |
| Profe | or common projectment of importer | tions | |
| Field | a common assignment of imperied | 4015 | |
| | | | |
| C Subi | act to enacific load cases | | |
| Subje | ect to specific load cases | | |
| Subje | ect to specific load cases | | |

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 | n Combinations | C S |) o s 3 x 3 e 3 | • | | 1 | d 🛛 | | f_x | fx |
|------------|--------------------------|--------|-----------------------------------|-----------|-------------|---------|--------|--------|--------|-------------------|
| | A | В | С | D | E | F | G | н | | J |
| Action | Action Combination | | EN 1990 CEN | Acti | ion.1 | Ac | tion.2 | Ac | tion.3 | Generated |
| Combin. | Description | Use | Design Situation | Factor | No. | Factor | No. | Factor | No. | Load Combinations |
| AC1 | 1.35G | V | STR ULS (STR/GEO) - Permanent | 1.35 | G A1 | | | | | CO1 (1/5) |
| AC2 | 1.35G + 1.50Qs | 1 | STR ULS (STR/GEO) - Permanent | 1.35 | G A1 | 1.50 | Qs A2 | | | CO2 (1/5) |
| AC3 | 1.35G + 1.50Qs + 0.90Qw | 1 | 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 | 1 | STR ULS (STR/GEO) - Permanent | 1.35 | G A1 | 1.50 | Qw A3 | | | CO4 (1/5) |
| AC5 | 1.35G + 0.75Qs + 1.50Qw | 1 | STR ULS (STR/GEO) - Permanent | 1.35 | G A1 | 0.75 | Qs A2 | 1.50 | Qw A3 | CO5 (1/5) |
| • | | | | | | | | | | · · |
| Load Cas | es Actions Combination E | xpress | ions Action Combinations Load Con | nbination | s Result (| Combina | tions | | | |
| 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*.

| Z 11 | 🗷 🖂 I 🤮 🖷 🔁 🕲 I 🛛 | 0 😫 3 | X | × == >> | | 1 📝 🚮 | હ્ય | × . | ab= fx 🎽 | f _x | |
|---------|-------------------------------------|--|------------|-----------|----------|------------|-----------|---------|----------|----------------|---|
| | В | С | D | E | F | G | Н | | J | K | |
| Load | Load Combination | | l | .C.1 | l | C.2 | L | .C.3 | | LC.4 | |
| ombin. | Description | To Solve | Factor | No. | Factor | No. | Factor | No. | Factor | No. | |
| CO1 | 1.35*LC1 + LC4 | V | 1.35 | G LC1 | 1.00 | Imp LC4 | | | | | |
| CO2 | 1.35*LC1 + 1.5*LC2 + LC4 | Image: A start and a start | 1.35 | G LC1 | 1.50 | Qs LC2 | 1.00 | Imp LC4 | | | |
| CO3 | 1.35*LC1 + 1.5*LC2 + 0.9*LC3 + LC4 | Image: A start and a start | 1.35 | G LC1 | 1.50 | Qs LC2 | 0.90 | Qw LC3 | 1.00 | Imp LC4 | |
| CO4 | 1.35*LC1 + 1.5*LC3 + LC4 | Image: A start and a start | 1.35 | G LC1 | 1.50 | Qw LC3 | 1.00 | Imp LC4 | | | |
| CO5 | 1.35*LC1 + 0.75*LC2 + 1.5*LC3 + LC4 | Image: A start and a start | 1.35 | G LC1 | 0.75 | Qs LC2 | 1.50 | Qw LC3 | 1.00 | Imp LC4 | |
| | | | | | | | | | | | |
| oad Cas | es Actions Combination Expressions | Action Co | mbinations | Load Comb | inations | Result Com | binations |] | | | - |

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.

| Existing Load | Combinations | CO No | uoris coe | Load Combination Description | | So | lve | |
|---------------|-----------------------------------|-------|------------|-----------------------------------|------------------|---------|------|------|
| STR CO1 | 1.35*LC1 + LC4 | | 3 | STR | | | | |
| STR CO2 | 1.35*LC1 + 1.5*LC2 + LC4 | | | | | | | |
| STR CO3 | 1.35°LC1 + 1.5°LC2 + 0.9°LC3 + L/ | Gener | al Calcul | ation Parameters | | | | |
| STR CO4 | 1.35*LC1 + 1.5*LC3 + LC4 | Load | Cases in L | oad Combination CO3 | | | | |
| STR CO5 | 1.35*LC1 + 0.75*LC2 + 1.5*LC3 + | No. | Factor | Load Case | Action | Leading | γ | Ψ |
| | | 1 | 1.350 | G LC1 - Self-weight | G A1 - Permanent | | 1.35 | |
| | | 2 | 1.500 | Qs LC2 - Snow | Qs A2 - Snow | | 1.50 | |
| | | 3 | 0.900 | Qw LC3 - Wind in +X | Qw A3 - Wind | | 1.50 | 0.60 |
| | | 4 | 1.000 | Imp LC4 - Imperfection towards +X | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | = | | | | | | | |
| | 1 | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | 1 | | | | | |
| | - | Comm | ent | | | | | |
| | - - | | | | | | (| 3 |

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.

| General Calculation Parameters | |
|--|--|
| Method of Analysis | Options |
| Critical Load Factor Critical Load Factor Critical Load Factor Calculate | Mgdfy loading by factor: Mgdfy loading by factor: Ovide results by loading factor Activate stiffness factors of: Materials (partial factor <i>γw</i>) Cross-sections (factor for J, Iy, Iz, A, Ay, Az) Members (factor for GJ, Ely, Elz, EA, GAy, GAz) Activate special settings in tab: Modify stiffness Extra options Consider favorable effects due to tension of members Refer internal forces to deformed structure for: Momal forces N Shear forces V, and Vz Moments My, Mz and MT Apply separate number of load increments for this load combination: |
| Load factor increment Δ_k : | |
| | |

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*.

| ad Cases | Actions Combination I | Expressions Act | ion Combinations | oad Combinations | Result Combination | ons | | | | | | |
|--------------|-----------------------|-------------------|------------------|--------------------|--------------------|-----------------|--------------|----------------|------------|----------|-----------|-------|
| kisting Resu | t Combinations | | RC No. | Result Combi | nation Description | | | | | Solve | | |
| TR RC1 | ULS (STR/GEO) - P | ermanent / transi | er 1 | STR - | ULS (STR/GEO) - | Permanen | t / transien | - Eq. 6.10 | - | V | | |
| | | | | | | | | | | | | |
| | | | General Calc | ulation Parameters | | | | | | | | |
| | | | Existing Loadin | ng | | | Loading in | Result Combina | tion RC1 | | | |
| | | | G LC1 | Self-weight | | | Factor | No. | Descrip | tion | Criterion | Group |
| | | | Qs LC2 | Snow | | | 1.00 | STR CO1 | 1.35*LC1 + | LC4 | Permanent | 1 |
| | | | QW LC3 | Wind in +X | | | 1.00 | STR CO2 | 1.35"LC1 + | 1.5*LC | Permanent | 1 |
| | | | STPL CO1 | 1 35*LC1 + LC4 | aius +A | | 1.00 | STR CO3 | 1.35°LCT + | 1.5 LC | Permanent | |
| | 1 | | STR CO2 | 1.35°LC1 + 1.5°L | C2 + I C4 | | 1.00 | STR CO5 | 1.35101+ | 0.75*1 | Permanent | 1 |
| | | | STR CO3 | 1.35*LC1 + 1.5*L | C2 + 0.9*LC3 + L | | 1.00 | 005 | 1.55 ECT + | 0.75 L | remarient | |
| | 1 | | STR CO4 | 1.35*LC1 + 1.5*L | C3 + LC4 | | | | | | | |
| | | | STR CO5 | 1.35*LC1 + 0.75* | LC2 + 1.5*LC3 + | \square^{q_2} | | | | | | |
| | | | | | | P | | | | | | |
| | | | | | | | | | | | | |
| | 1 | | | | | | | | | | | |
| | 1 | | | | | \triangleleft | | | | | | |
| | | | | | | | | | | | | |
| | 1 | | | | | 99 | | | | | | |
| | 1 | | | | | | | | | | | |
| | | | | | | | | | | | | |
| | 1 | | | | | | | | | | | |
| | 1 | | | | | | | | | | | |
| | | | | | | | | | | | | |
| | 1 | | | | | | | | | | | |
| | 1 | | | | | | | | | | | |
| | | | | | | | | | | | | |
| | 1 | | | | | | | | | | | |
| | | | | | | | | | | | | |
| | | | | | | | | | | | | |
| | 1 | | | | | | | | | | | |
| | | | | wi (10) | · 87 04 | | Ľ | | | | | |
| | | | Comment | | | | | | | | | |
| | | | Common | | | | | | | | | |
| 5 (6) (| Св All (1) | - 🗙 | | | | | | | - 🖻 | | | |
| | | | | | | | | | | | | |

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.

60

Calculation 7.

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.

| Plausibility Check | × |
|---|---|
| Check | Type of Check Normal With warnings None, only statistic |
| Which Load Cases Current load case All | |
| 2 | OK Cancel |

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.

Calculate Model 7.2

To start the calculation,

we select Calculate All on the Calculate menu

or use the toolbar button shown on the left.

| Calculation | | | × |
|-------------|---|---|--------------------|
| VER | Running RSTAB - Calculation Combination: RC1 | - | |
| RS-SOL | Partial Steps Determining Result Combinations Processing Input Data Reading Input Data Determining Internal Forces Determining Member Deformations Determining Support Forces Determining Contact Forces Determining Nodal Deformations | Type of model: Number of Nodes: Number of Elements: Number of Equations: | 2D 7 6 21 |
| | Cancel | | |

Figure 7.2: Calculation process

M

4 >



Results 8.

Graphical Results 8.1

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

We can use the toolbar buttons $[\blacktriangleleft]$ and $[\blacktriangleright]$ (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 load cases and load combinations



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 | В | C | D |
|---|--------|------|---|
| Table 4. Results | Value | Unit | 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.



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.

4 >



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 My 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.







1

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.



 Wireframe Display Model

 Solid Display Model

 Solid Transparent Display Model

| 2 | Edit Set of Members |
|------------|------------------------------------|
| Ä | Delete Set of Members |
| 1 | Select Members |
| X | Reverse Set of Members Orientation |
| 4 | Result Diagrams |
| ⊼ ⊐ | Display Properties |

Context menu for set of members

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.

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 .



x

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.

| New Printou | t Report | × |
|------------------------------|---|--------|
| No. | Description Input data and reduced results | |
| Printout Rep 1 - Input da | oort Template ata and reduced results | • 🔁 🖻 |
| | ОК | Cancel |

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*.

| ogram / Modules | Global Selection RC Results | |
|-------------------------------|---|---|
| STAB | Desuit Combinations to Display | _ |
| | | |
| | | |
| | Selected (1) | |
| | Tables to Directory | |
| | Tables to Unsplay Display Table All Number Selection (e.g. '1-4.8') | _ |
| | M 4.1 Members - Internal Forces | _ |
| | 4.2 Sets of Members - Internal Forces All | |
| | 4.3 Cross-Sections - Internal Forces All | |
| | 4.4 Nodes - Support Forces | |
| | 4.5 Members - Contact Forces | _ |
| | 4.0 Nodes - Deformations | - |
| | 4.8 Members - Global Deformations Q All | _ |
| | 4.9 Members - Coefficients for Buckling All | |
| | 4.10 Member Slendemesses All | |
| | | |
| | Details - Internal Forces by Member | |
| | | |
| | Display Display Max/Min Internal Forces | |
| | N NT | |
| | Partition values Vy VMy | |
| | Vz Mz | |
| | | |
| | | |
| | | |
| | | |
| | | |
| splay | OK Cancel | |
| Cover sheet | | |
| | | |
| Contents | | |
| Contents | | |
|] Contents] Info pictures | ✓ ✓ ✓ ✓ ✓ Show corresponding load cases | |

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.

| File View Fo | tit Settin | as Insert | Help | | | | | | | |
|--------------|-------------|----------------------|----------|--------------------|--------|---------|----------------------|------|---|---------------|
| | | | | | | | 2 | | | |
| | | NA | | » 😵 i 🐚 | | | | | | |
| 4.1 M | EMB | ERS - I | INTERI | NAL FC | RCES | | | | | Resu |
| Member | | Node | Location | | Forces | [kN] | Moments | | | Corresponding |
| No. | RC | No. | x [m] | | Ν. | Vz | M _v [kNm] | | | Load Cases |
| 1 | RC1 | Max N | 5 0 0 0 | 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 Vz | -83.84 | -46.74 | 0.00 | CO 2 | | |
| | | Max My | 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 My | 6.874 | Max My | -47.50 | -0.78 | 123.36 | CO 3 | | |
| | | Min My | 0.000 | Min My | -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 | | |
| | | Ivlax Vz | 0.000 | Max Vz | -46.35 | 4.42 | 120.94 | 002 | | |
| | | IVIIN V _z | 7.638 | Min V _z | -52.57 | -57.91 | -92.49 | 003 | | |
| | | Max M _v | 0.764 | Max M _v | -46.94 | -1.69 | 121.98 | 002 | | |
| | D 04 | IVIIN My | 7.638 | Min My | -52.57 | -57.91 | -92.49 | 003 | | |
| 5 | RC1 | Max N Min N | 0.000 | Max N Min N | -19.70 | -18.53 | -27.25 | 001 | | |
| | | Max V | 2.269 | Max V | -59.46 | -72.05 | -237.04 | 003 | | |
| | | Min V | 2 289 | Min V | -15.70 | - 10.55 | -21.20 | 003 | | |
| | | Max M | 0.000 | Mox M | -05.40 | -12.05 | -237.04 | CO 1 | | |
| | | Min M | 2 289 | Min M | -15.70 | - 10.55 | -21.20 | 003 | | |
| 6 | PC1 | Max N | 2.209 | Max N | -05.40 | -12.05 | -231.04 | CO 1 | | |
| 0 | RUT | Min N | 0.000 | Min N | -20.24 | 54 30 | 03.30 | 003 | | |
| | | Max V- | 0.000 | Max V- | -85.95 | 54.30 | 0.00 | CO 3 | | |
| | | Min V- | 5.000 | Min V- | -28.24 | 16.57 | 83.36 | CO 1 | | |
| | | Max M | 5,000 | Max M. | -80.84 | 47.25 | 254 91 | 003 | | |
| | | Min Mu | 0.000 | Min My | -34.14 | 16 74 | 0.00 | CO 1 | | |
| | | | 0.000 | with why | -34.14 | 10.74 | 0.00 | 001 | _ | |

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.









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*.

| Graphic Printout | | × | | | | | |
|--|--|---------------------|--|--|--|--|--|
| General Options Color Scale Factors Borde | er and Stretch Factors | | | | | | |
| Graphic Picture | Window To Print | Graphic Size | | | | | |
| O Directly to a printer | Current only | As screen view | | | | | |
| To a printout report: | O More | Window filling | | | | | |
| To the Clipboard | 🔘 Mass print | ○ To scale 1: 100 ▼ | | | | | |
| © To 3D PDF | | | | | | | |
| | | | | | | | |
| Graphic Picture Size and Rotation | Options | | | | | | |
| Use whole page width | Show results for selected x-location in result | | | | | | |
| 🔿 l ha whale anna haisht | diagram | | | | | | |
| Height: 50 1% of page] | Lock graphic picture (without update) | | | | | | |
| | Show printout report on [OK] | | | | | | |
| Rotation: 0 (* [°] | | | | | | | |
| | | | | | | | |
| | | | | | | | |
| Header of Graphic Picture | | | | | | | |
| Internal forces M _y , CO5: 1.35*LC1 + 0.75*LC2 + 1.5*LC3 + LC4, Isometric | | | | | | | |
| | | | | | | | |
| OK Cancel | | | | | | | |

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.





Save

D

RSTAB Introductory Example © 2014 Dlubal Software GmbH



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.

