



**Dlubal Software** 

Version
April 2018

# **Short Overview**

1	Introduction	4
2	System and Load	5
3	Creating the Model	7
4	Model Data	9
5	Load	49
6	Combining Actions	69
7	Calculation	77
8	Results	81
9	Documentation	97
10	Outlook	105



**Dlubal Software GmbH** Am Zellweg 2 93464 Tiefenbach Germany

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

#### Dlubal Software, Inc.

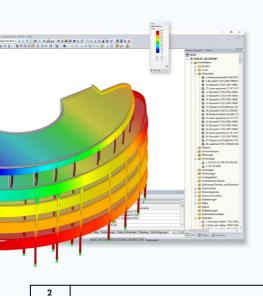
The Graham Building 30 South 15th Street 15th Floor Philadelphia, PA 19102 USA

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

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.

#### Using the Manual

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



### Hint

In addition, you can also use the search function for the Knowledge Base 2 and FAQs 2 on our website to find a solution in the articles.



#### Topicality

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

# **Table of Contents**

1	Introduction	4
2	System and Load	5
2.1	System Sketch	5
2.2	Materials, Thicknesses, and Cross-Sections	5
2.3	Load	6
3	Creating the Model	7
3.1	Starting RFEM	7
3.2	Creating the Model	7
4	Model Data	9
4.1	Adjusting the Work Window and Grid	9
4.2	Creating Surfaces	11
4.2.1	Floor	11
4.2.1.1	Defining a Rectangular Surface	11
4.2.1.2	Creating an Arc	14
4.2.1.3 4.2.2	Adjusting the Floor Surface	15
4.2.2 4.2.3	Wall Opening	16 19
4.2.3.1	Creating an Opening	19
4.2.3.2	Adjusting the Opening	20
4.3	Creating Concrete Members	21
4.3.1	Columns	21
4.3.2	Rib	24
4.4	Defining Supports	28
4.4.1	Nodal Supports	28
4.4.2	Line Supports	30
4.5	Creating Steel Members	31
4.5.1 4.5.1.1	Frame	31 33
4.5.1.1	Defining Members Continuously Chamfering Horizontal Beams	35 35
4.5.1.3	Connecting Beams with Hinges	36
4.5.1.4	Reversing Member Orientation	38
4.5.1.5	Copying a Frame	39
4.5.2	Purlins	40
4.5.2.1	Defining Members Individually	40
4.5.2.2	Connecting Members Eccentrically	42
4.5.3	Diagonal	44
4.5.3.1	Defining the Member	44
4.5.3.2	Rotating a Member	46
4.6	Checking the Input	48

5	Load	<b>49</b>
5.1	Load Case 1: Self-Weight	49
5.1.1	Self-Weight	50
5.1.2	Floor Structure	50
5.1.3	Earth Pressure	51
5.1.4	Roof Load	52
5.2	Load Case 2: Imposed Load	54
5.2.1	Floor Slab	55

Т

5.2.2	Edge of the Opening	56
5.3	Load Case 3: Snow	57
5.3.1	Roof	57
5.3.2	Floor	59
5.4	Load Case 4: Wind	61
5.4.1	Steel Construction Loads	61
5.4.2	Column Loads	63
5.5	Load Case 5: Imperfection	65
5.5.1	Steel Columns	66
5.5.2	Concrete Columns	67
5.6	Checking Load Cases	68

6	Combining Actions	<b>69</b>
6.1	Checking Actions	69
6.2	Defining Combination Expressions	70
6.3	Creating Action Combinations	73
6.4	Creating Load Combinations	73
6.5	Checking Result Combinations	76

7	Calculation	<b>1</b>
7.1	Checking Input Data	77
7.2	Creating the FE Mesh	78
7.3	Calculating the Model	80
8	Results	<b>81</b>

9	Documentation	<b>97</b>
0.5		/4
8.5	Creating a Section	94
8.4	Displaying Result Diagrams	93
8.3.3	Range of Values	91
8.3.2	Results on Objects	89
8.3.1	Custom Visibilities	86
8.3	Filtering Results	86
8.2	Results Tables	84
8.1	Graphical Results	81

9	Documentation		97
9.1	Creating a Printout Report		97
9.2	Adjusting the Printout Report		98
9.3	Printing Graphics in the Printout Report	1	00

10 Outlook 🔳 105

ĺ

# 1 Introduction

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



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

examples-and-tutorials 🛛

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

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

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

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

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

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

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



 $\mathbf{P}$ 

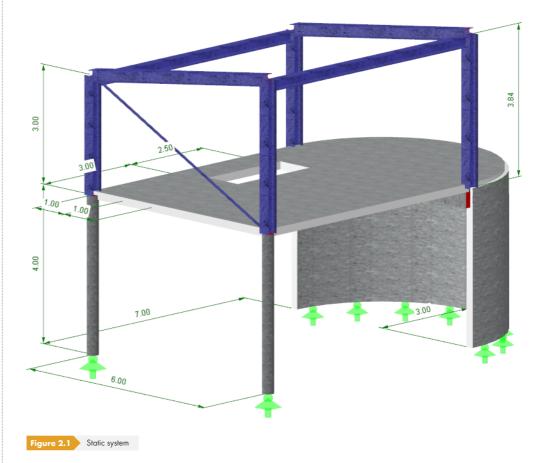
2.1

2

# 2 System and Load

# System Sketch

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



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

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

As mentioned before, this model depicts a more "theoretical" construction, managable with the demo version's maximum of two surfaces and twelve members.

2.2

# Materials, Thicknesses, and Cross-Sections

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

Dlubal

Wall and floor thickness are uniformly 20 cm. Both concrete columns have a 30 cm diameter. The downstand beam has a width of 25 cm and a height of 40 cm.

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

## Load

2.3

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

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

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

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

#### Load case 2: imposed load

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

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

#### Load case 3: snow

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

#### Load case 4: wind

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

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

#### Load case 5: imperfection

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

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

According to EN 1993-1-1, Table 6.2, we must assume the buckling curve b (displacement in direction of y-axis) for the column cross-sections (HE-A 300). The inclinations  $\varphi_0$  and precambers w<sub>0</sub> are determined according to EN 1993-1-1, section 5.3.2.

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

3

# **3** Creating the Model

3.1



3.2

# **Starting RFEM**

We run RFEM using the task bar

Start  $\rightarrow$  All Programs  $\rightarrow$  Dlubal  $\rightarrow$  Dlubal RFEM 5.xx

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

# Creating the Model

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

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

lew Model - General Data	:
General Options History	
Model Name Descr	iption
Tutorial conc	rete and steel construction
Project Name Descr	iption
Examples Samp	ole structures
Folder:	*3 <b>3</b>
C: \Users \Public \Documents \Dlubal \Projects	Examples
Type of Model	Classification of Load Cases and Combinations
<ul> <li>③ 3D</li> <li>○ 2D - XY (uz/φx/φy)</li> </ul>	According to Standard: National annex:
O 2D - XZ (ux/uz/φγ) O 2D - X <u>Y</u> (ux/uγ/φ2)	Create combinations automatically  Create combinations  Result combinations (for linear analysis only)
Positive Orientation of Global Z-Axis	Template
O Upward	Open template model: Test
Comment	
	~ ] <b>C</b>
2 📝 🚾 🐴 🖏	OK Cancel
igure 3.1 New Model - General Dai	ra dialoa box

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

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

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





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

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

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

The empty RFEM work window is displayed.



# 4 Model Data

4.1

# Adjusting the Work Window and Grid

#### View

-

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

đ

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

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

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

There are several ways to exit this feature:

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

#### **Mouse functions**

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

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

٢ 🖑

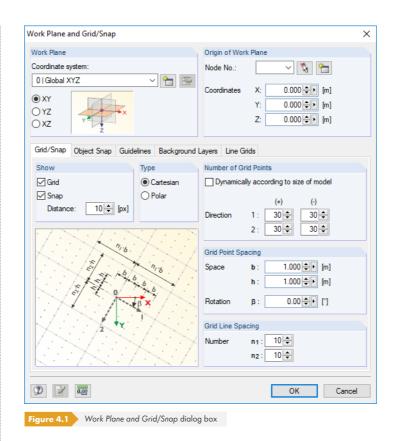
-

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

#### Grid

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

Δ



#### SNAP GRID

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

#### Work plane

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

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







# **Creating Surfaces**

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



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

#### 4.2.1.1 Defining a Rectangular Surface

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

#### Insert $\rightarrow$ Model Data $\rightarrow$ Surfaces $\rightarrow$ Plane $\rightarrow$ Graphically $\rightarrow$ Rectangle

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

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

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

New Rectangular Surface	×
Surface No.	Surface Type
1	Geometry: Plane
Material	Stiffness: Standard 🗸 🔄
□       Concrete C30/37       EN 1992-1-1:2004/A1:2014         □       □       □         □       □       □         □       □       □         □       □       □         □       □       □         □       □       □         □       □       □         □       □       □         □       □       □         □       □       □         □       □       □         □       ∨       □         □       ∨       □         □       ∨       □         □       ∨       □         □       ∨       □         □       □       □         □       ∨       □         □       ∨       □         □       ∨       □         □       ∨       □         □       □       □         □       □       □         □       □       □         □       □       □         □       □       □         □       □       □         □	Surface thickness 'Constant'
	OK Cancel

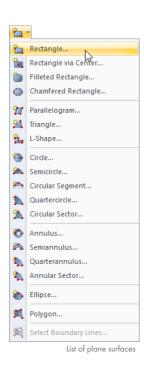
Figure 4.2 New Rectangular Surface dialog box

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

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

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

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



\*

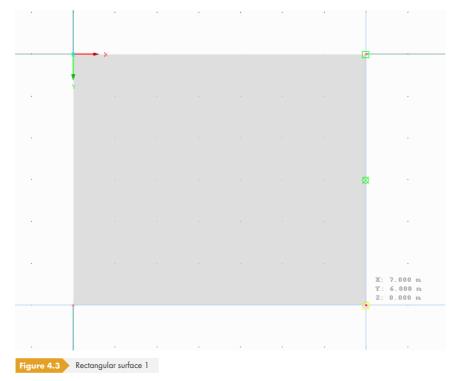
ÎZ

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

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

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

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



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

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

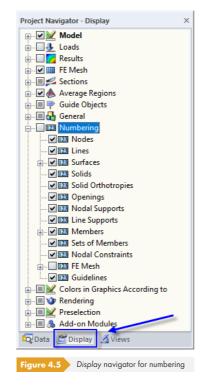
Y

#### Showing the numbering

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

	Repeat - View in Z-Direction	Enter
	View	۱.
	User-Defined View	Þ
123	Show Numbering	
P	Show Loads	
0	Show Results	
À	Show Dimensions	
~A	Show Comments	
√	Show Hidden Objects in Backgro	ound
<b>v</b>	Display Model Wired or Solid	
-	Lock Guidelines	
<b>/</b>	Lock Line Grids	
	Enable Drag & Drop	
•	Auto Rotate Model	
¢	Auto Connect Lines/Members	
Ģ	Coordinate System	
0	Work Plane, Grid/Snap,	
7	Select Special	
	Colors in Graphics According to	×
	Display Properties	

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





47-

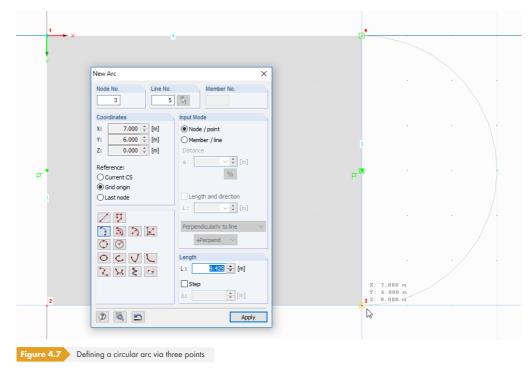
### 4.2.1.2 Creating an Arc

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

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

7	Polyline
}	Arc via Three Nodes
0	Arc via Center Node, Edge Node and Angle
2	Arc via Edge Nodes and Radius, Angle or Rise
¢	Arc via Tangent Lines and Radius
D	Circle via Three Nodes
9	Circle via Center and Radius
5	Ellipse
5	Elliptical Arc
J	Parabola
¢	Hyperbola
2	Spline
U	NURBS
Ş	Trajectory
b)	On Surface
9	Cut via Two Lines
7	Cut via Section

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



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



## 4.2.1.3 Adjusting the Floor Surface

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

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

dit Surfac	ce							2
General	Support / Eccentricity	FE Mesh	Hinges	Integrated	Axes	Grid	Modify Stiffness	
Surface	No.				Surfa	се Туре	3	
1					Geom	etry: [	Plane	~
Boundar	ry Lines No.				Stiffne	ess:	Standard	~ 🕾
1,2,4,5				\$ 2	Surfa	ice thick	kness 'Constant'	
	ry Nodes No. 1; 1,4; 3-5							
Material		1002 1 1.2	004/01-0	014				
	Concrete C20/25   EN	1332-1-1.2	004/A1.2	14 🔍				
Thicknes	SS					•		
Cons Thick		<b>√ ≑ ►</b> [mr	n]	Ň			o t	
🔿 Varia	ble						I	
Commen	nt							
				~				
2	/ 📖 💿 💦						ОК	Cancel
iqure 4	.8 Modifying bou	ndary line	s					

There are two input options:

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

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

	×		4			
Ļ						
						5 
1						
2				2	 	3
Figure 4.9	Floor	slab				

*گ	Edit Surface Select Lines		
Selecte 1,2,4,5		 	
Clei	ar	ок	Cancel



4.2.2 Wall

## Copying the arc

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

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

°}

We open the <i>N</i>	Nove or Cop	y dialog box	using the	toolbar butto	on shown on the left
----------------------	-------------	--------------	-----------	---------------	----------------------

Number		
Number of copies n: 1÷	Y X	
Reference to Coordinate System	Ž dv	
● Global X,Y,Z CS User-defined U,V,W CS ♥ ■ ♥		
Displacement Vector dx: 0.000 € [m] dy: 0.000 € [m]		
dz: 4.000 € [m]	Numbering Increment for	
	Nodes: 1 Continuous	
	Members:	
	Lines: 1 - Continuous	
	Surfaces: 1 Continuous	
	Solids: 1 🗧 Continuous	

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

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



Δ

Connecting	
ines between nodes	Copied Surfaces
Create new lines between the selected nodes and their copies	
vlembers between nodes	eige s
Create new members between the selected nodes and their copies	Contecting Objects (Lines
Template member No.:	1 de
None 🗸 🦄	Deectin
Surfaces between lines	We want the second seco
Create new surfaces between the selected lines and their copies	
Template surface No.:	То сору
1   Standard   200   1 - Concrete C30/37 📉 🏷	
None	Local Coordinate Systems
Coli 1 Standard 200 1 - Concrete C30/37 Create new solid bodies between the selected surfaces and their copies	Adjust local coordinate system when rotating or mirroring for
Template solid No.:	✓ Lines
None V	Members
	Load Cases
When rotating create between selected nodes and heir copies:	Copy including loading
Straight lines	Adjust nodal loads when rotating or
Arc lines	mirroring
Duplicity	Auto Connect
Allow double members	Connect lines/members if they contact
2	OK Cancel

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

Create new surfaces between the selected lines and their copies

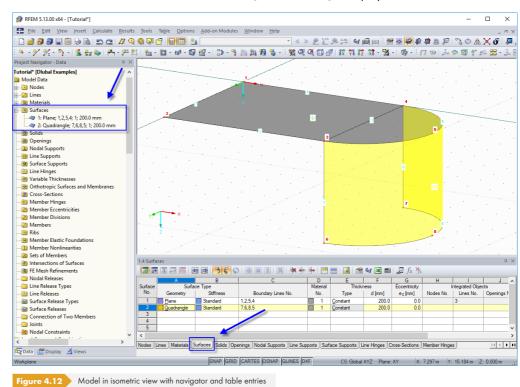
We select surface 1 from the list as the *Template surface*. This sets the floor slab's properties (material, thickness) as default for the new wall surface.

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



#### Switching to isometric view

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

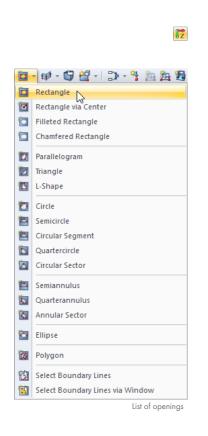


#### Checking data in the navigator and tables

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

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





## 4.2.3 Opening

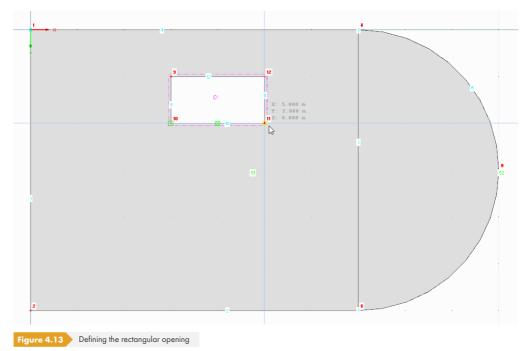
### 4.2.3.1 Creating an Opening

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

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

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

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



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



### 4.2.3.2 Adjusting the Opening

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

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

Node Coordinates Support FE Mesh	
Node No.	Node Type
11.12	Standard 🗸
Coordinates	Coordinate System 'Cartesian'
Reference node No.: 0 V	
Coordinate system: Cartesian V	x
Coordinate X: 5.500 🜩 [m]	Y X Y
Coordinate Y:	ż
Coordinate Z: 0.000 (m) [m]	• P (X,Y,Z)
	Z
Comment	
\[         \]     \[	à
2 2 000	OK Cancel

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

Alternative: We could have set the opening without changing the coordinates by adjusting the grid. To do so, we would have had to reduce *Grid Point Spacing* to 50 cm in the *Work Plane and Grid/Snap* dialog box (see Figure 4.1 2). To change the grid spacing even quicker, use the [GRID] button's shortcut menu by right-clicking it (see figure to the left).

This completes the input of the surfaces.



4.3

#### 

$\mathcal{V}_{\mathrm{I}}$	• <u>2 - 3 5 5 5 6 7</u>
1	Single Member
$^{\circ}$	Continuous Members.
У́г	Inserted Member
%	Select Lines
24	Set of Members

# **Creating Concrete Members**

#### 4.3.1 Columns

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

#### Changing the work plane

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

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

#### **Defining a cross-section**

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

ew Member		>
General Option	ns Effective Lengths Modify Stiffness	
Member No.		Member Type
1		Beam V
Node No.		Member Rotation
tode no.		
		× X End
Member Rotatio		Y + // Y
Angle	β: 0.00 <b>≑</b> ▶ [°]	• P (X,Y,Z)
Help node	No.: Inside 🗸 🏷 😁	Help node in plane x-y
In plane:	⊚ x-y	Begin B⊳y'
in piane.	• x-y • x-z	Here
	0	z' <sup>V</sup> 'z β<0
Cross-Section		
Member start:	Create a new cross-section!	- 🔟 🚡 🐷 🙆
Member end:	As member start	- 🔟 🔁 🗟
Member Hinge		
Member start:	None	~ 🔁 🖾
Member end:	None	
2 0.00	6	OK Cancel
	New Member dialog box	

It is not necessary to change the default settings. We only have to create a Cross-Section. To define the cross-section at the Member start, we click **(**). The cross-section library opens.



Δ

Rolled				Paramet	ric - Thin	-Walled		Paramet	tric - Mas	sive		Paramet	ric - Timb	er	
I	Γ	T	L	I	Τ	Γ	Т		T	T	I		•		
0	0	0	۷.	Т	L	L		T		$\bigcirc$				•	0
l	1	~	<b>1</b>	L	I	Т	T	$\square$		I		Т	Т		
				0	$\nabla$	Π	Π	T	L	L	r	T	T	Ξ	1
Built-up				Ĩ	Ŭ	Π	Π	π	TT	TT					
Π	I	T	76	Ŧ	Ť	+	•			D				Ι	
Т	I	Ŧ	T	-	İ	l	Г		000	T					
I	Ī	I	Iē	Τ	Ľ	Ľ,	Ľ					Standard	dized - Ti	imber	
••	1			Σ	0	$\nabla$	0						ID		
Circle								User-De	efined			From Cro	oss-Sect	ion Progr	am
								2				E	•		
2	28													С	ancel

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

Solid Cross-Sections - Circle		×
Cross-Section Type	Parameters	
ITII	D: 300.0 + [mm]	
T • •		
T L L J		у –
<b>T T O</b>		
		l z
11 111 1		
		0 🗮
9		□ 1 - Concrete C30/37   EN 1992-1-1:2004/A1:2014 ∨
Favorites Group		Circle 300
2 👼		OK Cancel
Figure 4.17 Solid Cross-Sec	tions - Circle dialog box	

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

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

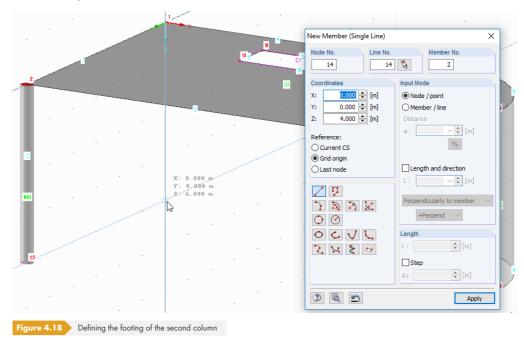
To check the cross-section's properties, use the [Info] button.

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

#### **Defining members graphically**

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

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



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

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

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



## 4.3.2 Rib

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

#### **Modifying line properties**

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

The New Member dialog box opens again.

eneral Member	Support	General	Ontione	Effective Lengths	Modify Stiffpoor			
Line No.	Support	Member I		Line No.	Modily Juintess	15	Member Type	2
Member		Node No. 3,4					Member Rotation	_
Beam Cross-section:		Member I	;	β: 0.00	•		Y Y Z Help node in plane x-y	
As member start		O <b>Help</b> i In pla		No.: Inside ∨ ⊚ x-y ○ x-z	N 1	0	Begin $\beta > y'$ z $\beta < 0$	/
		Cross-Se Member Member	start:	As member start				
		Member Member	start: 📘	lone			✓ 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	
0 📝 👼	]	2 0.00					ОК	Cance

Figure 4.19 New Member dialog box

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

In the upper part of the New Cross-Section dialog box, we select the massive REC cross-section table (see Figure 4.20 12).

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



New Cro	oss-Secti	on			>
No.	2 Colo	or Ø	Cross-	Section Description [mm]	
Solid Cro	ss-Sectio	ons - Rea	tangle		
Cross-Se	ection Typ	pe		Parameters	
	T	L	I	b: 250.0 ↔ [mm] h: 400.0 ↔ [mm]	b
T		$\bigcirc$		h: 400.0 主 [mm]	
$\square$	Ш	I			
T	L	L	r		у
π	T	77			
	000	T			
					Material
					1 - Concrete C30/37   EN 1992-1-1:2004/A1:2014
	-		ņ		
Favorites	s Group	~ •			Rectangle 250/400
2	.00	2			OK Cancel
Figure 4	4.20	Solid C	Cross-Sect	tions - Rectangle dialog box	

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

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

### **Defining a rib**

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



Δ

General Optio	ons Effective Lengths Modify Stiffness		
Member No.	Line No.	Member Type	
3	3	Rib	x 🕾
		Beam	
Node No.		Rigid	
3.4		Rib	
		Truss	
		Truss (only N)	
		Tension	
Member Rotati		Compression	,
Angle	β: 0.00 <b>≑ ▶</b> [°]	Buckling	
		Cable	
O Help node	No.: Inside 🗸 🐧 🛅	Cable on Pulleys Result Beam	
In plane:	() x-y	Definable Stiffness	
ni piano.	○ × J ○ x-z	Coupling Rigid-Rigid	
	0.*2		
Cross-Section	1	Coupling Hinge-Hinge	
Member start:	2 Rectangle 250/400 Concrete C	Coupling Hinge-Rigid	🤜 🔐
		Spring	
Member end:	As member start	Null	📨 d
Member Hinge			
Member start:	None	🔪 🛅 🗄	<u>s</u>
Member end:	None	V 🍋 🖡	
	*		
2 0.00	3	OK	Cancel

With a click on 💽 we open the New Rib dialog box.

lew Mem	nber			>
General	Options	Effective Lengths	Modify Stiffness	
On +2 On -2 Centri User-1	ind Alignm z-side of su	urface rface		y
	: OL/6 @L/8 O	: No.:	$\begin{tabular}{ c c c c c } \hline \label{eq:constraint} Integration Width - Side 2 \\ \hline \begin{tabular}{ c c c c c c c c c c c c c c c c c c c$	<b>•</b> 0
		parallel to local z-a:	kis of surface	Cancel
Comment	2 0.00		OK Cancel	

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

For the Integration Width we specify  $\mathbf{L/8}$  for both sides. RFEM will find the surfaces automatically.

Δ

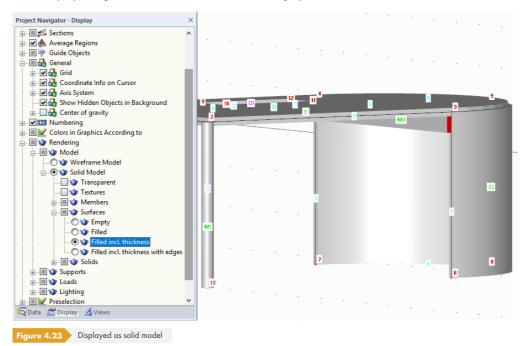
3	•
Ø	Wireframe Display Model
ø	Solid Display Model
9	Solid Transparent Display Model

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

## Changing the display model

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

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



関 🕅



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

For further input we change the Display Model back to Solid Transparent. We also change the rendering for surfaces in the Display navigator back to Filled to hide thicknesses.

4.4

3

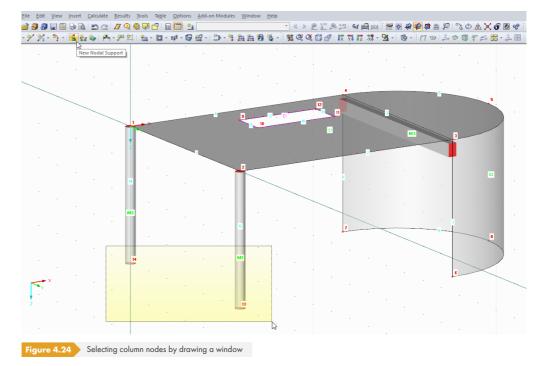
# **Defining Supports**

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

#### 4.4.1 Nodal Supports

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

We select the column's bottom nodes by drawing a window across the area that includes nodes 13 and 14. Afterwards we click the [New Nodal Support] toolbar button to open the New Nodal Support dialog box.



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

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



Node No.						
13,14			<b>\$</b>			
Type of Support					8	->
✓✓✓	Hinged	~	8			
New Nodal Su	upport					×
Support No.	On Nodes No			v		
1	13,14		1.		$\sim$	1
Support Axis	System					
Global X,Y						
	,- ied axis system:				► X'	
Rotated	,					
				Z		
					$\geq$	1
Elastic Suppo	rt via					
Column in			₹ [	TX TY	17 🗊	X
_					V-   U	•
Support Cond			Nonlinea	alle a		
Support	Spring constant	<b>↓</b> [kN/m]	None	nty	~	1
⊡ ux: ⊠uv:	Cu,Y :	↓ [kN/m]	None		Ť	
⊡ ur: ⊠ uz:	Cu,Y :	↓ [kN/m]	None		, v	
	00,2	Control 1	None			
Restraint	0	.000 💠 [kNm/rad]	None			pages.
φχ: φγ:	•	000 + [kNm/rad]			~	
	• • •	000 + [kNm/rad]			~	
Ll øz:	οφ.2		None		Ť	
<del>7777</del> .	<u>⊥</u> <u>⊥</u> <u>⊥</u> (	A X				
Comment						
		~	/ 🕞			
2	0.00			OK	Cano	

In the second New Nodal Support dialog box, we clear the Restraint for rotation  $\phi_{Z^{\prime}}$ 

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



#### 4.4.2 Line Supports

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

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

The Hinged option is the default Type of support. The first three selected check boxes indicate that a support is available in the directions X, Y, and Z. The final three boxes are not selected because the hinged support type has no restraint around X, Y, and Z.

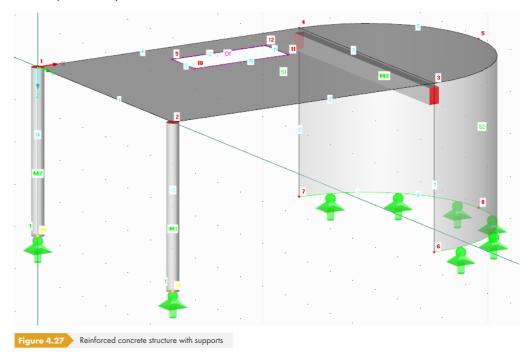
We confirm the dialog box with [OK] because the hinged support is suitable for our example.

Type of Support Global   및 및 및     Hinged	~ 🔁 🖾		
		2 2	<b>+ +</b>
	1.		ž 🗊 🏹
$\mathfrak{D}$		ОК	Cancel



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

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



4.5

N

# **Creating Steel Members**

#### 4.5.1 Frame

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

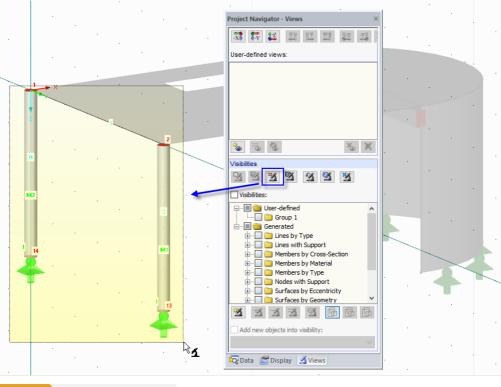
#### **Creating custom visibilities**

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

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

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

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





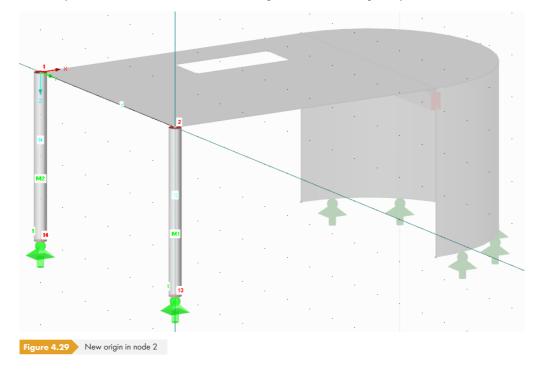


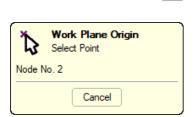
Δ

## Changing the work plane's origin

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

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





-¢-



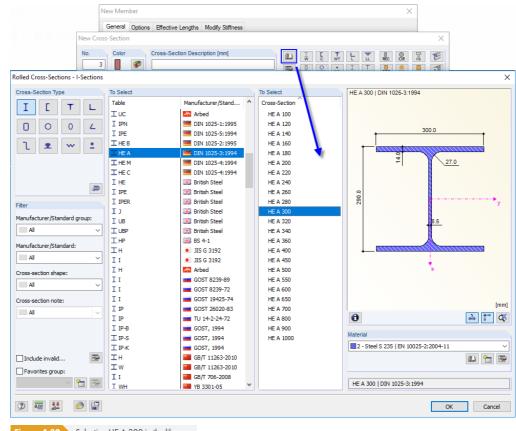
¥.	• * * * * * * * * * * * * *
Ŷ₁	Single Member
2	Continuous Members
💓	Inserted Member
%	Select Lines
22	Set of Members

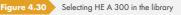
### 4.5.1.1 Defining Members Continuously

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

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

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





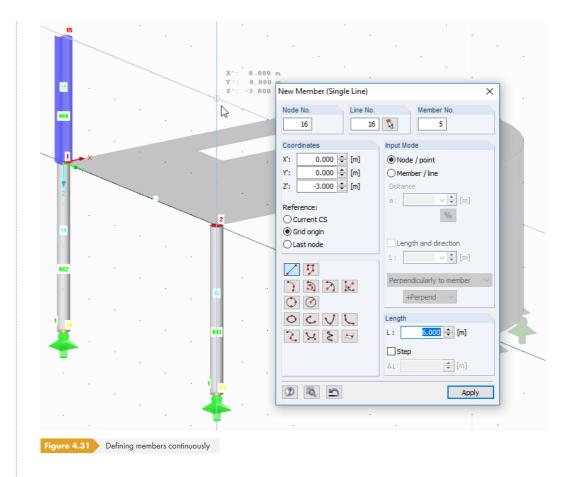
We confirm all dialog boxes with [OK].

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

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



4



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

Both columns in our model are connected rigidly to floor nodes 1 and 2. Though this kind of restraint can hardly be built in reality, we forgo a modeling of hinge properties in our example and accept the simplification.

Δ

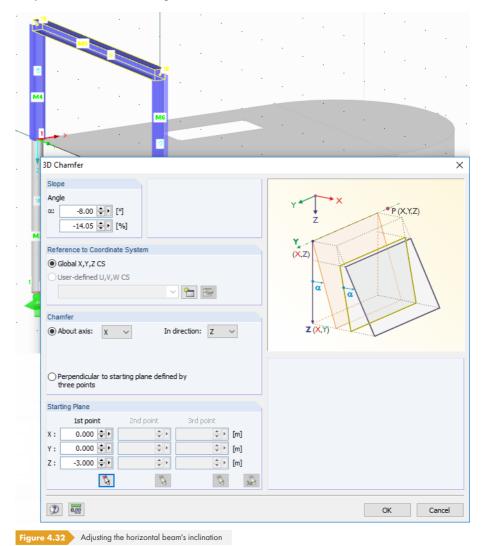
#### 4.5.1.2 Chamfering Horizontal Beams

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

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

#### Edit → Chamfer

to open the 3D Chamfer dialog box.



We want to modify the beam's inclination by -**8**° About axis **X**. We have to enter a negative value because the objects will be rotated counterclockwise about the X-axis. The chamfering is applied vertical *In direction* **Z**. We define the rotation axis' *1st point* using **S**. We select node *15* with the

coordinates 0.000/0.000/-3.000 and click [OK] to confirm the input.



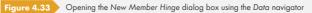
### 4.5.1.3 Connecting Beams with Hinges

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

#### **Defining hinges**

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

Project Navigator - Data ×		New Member Hinge					×
▶ RFEM       ▲         ► Model Data       ★         ★ Materials       ★ <t< th=""><th></th><th>Member Hinge No.</th><th></th><th>ľ</th><th>X Z V2 N2</th><th>MT.</th><th></th></t<>		Member Hinge No.		ľ	X Z V2 N2	MT.	
	Enter	Hinge Conditions       Hinge     Spring       ux     Cux       uy     Cuy       uy     Cuy       uz     Cuz	: [KN/m]	Noni Non Non	ie	> >	ছা ছা
Image: Second	Del	Hinge $\[ \] \varphi_X \qquad C_{\varphi_X} :$ $\[ \] \varphi_y \qquad C_{\varphi_Y} :$ $\[ \] \varphi_z \qquad C_{\varphi_Z} :$	: 0.000 (*) [d\m/rad] : 0.000 (*) [d\m/rad]	Non Non	e	~ ~ ~	19 19 19
Display Properties     Display Properties     Crine Refeases     Surface Release Types     Control Pattern     Surface Release Types     Display 2 Views		Comment		<ul> <li>G</li> </ul>	OK	Canc	cel



In the New Member Hinge dialog box, we have to select the displacements or rotations that are released at the member end. In our case they are the rotations  $\phi_y$  and  $\phi_{z_r}$  which means no bending moments can be transferred at the node.

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

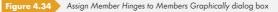
#### **Assigning hinges**

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

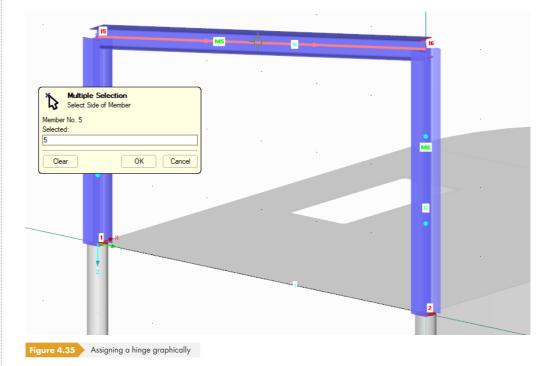
#### Insert $\rightarrow$ Model Data $\rightarrow$ Member Hinge $\rightarrow$ Assign to Members Graphically

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

Assign Member Hinges to Members Graphically	Х
Member Hinge	Ŋ
🗖 1   Local   🗆 🗆   🖸 🗹	
None	_
UN CONCENT	



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

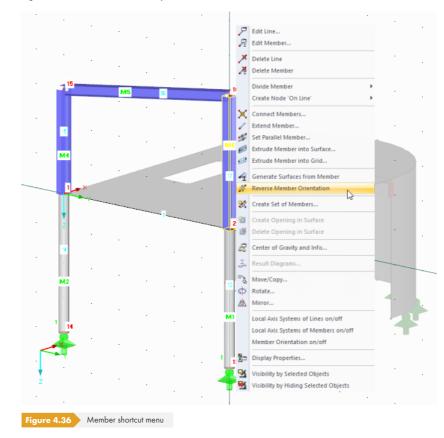




## 4.5.1.4 Reversing Member Orientation

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

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





## 4.5.1.5 Copying a Frame

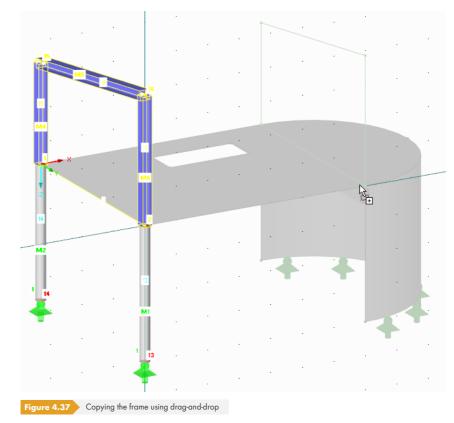
The second frame of the monopitch roof can be quickly created as a copy.

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

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

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

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



Nodes and lines are automatically merged with already defined objects.





$\mathcal{V}_{\mathbf{I}}$	• <u>3 - 12 b</u> b b A - 2
¥.	Single Member
$^{\circ}$	Continuous Members
Ŷ∕ı	Inserted Member
%	Select Lines
24	Set of Members

## 4.5.2 Purlins

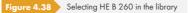
## 4.5.2.1 Defining Members Individually

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

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

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

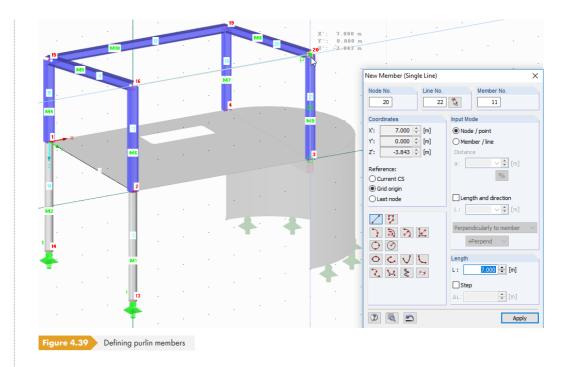
New Cross-	Section			×
No.	Color Cross-Section	on Description [mm]		
Rolled Cross-Sections - I-Sections				AND AND ARE AN
Cross-Section Type	To Select		To Select	HE B 260   DIN 1025-2:1995
ILTL	Table	Manufacturer/Stand ^	Cross-Section	
	I IPN	🙈 Arbed	HE B 100	
	I IPE	🙈 Arbed	HE B 120	200.0
	IHE	🙈 Arbed	HE B 140	260.0
1 🔹 🕶 🖆	THD	🙈 Arbed	HE B 160	+
	IHL	🔼 Arbed	HE B 180	
	ΙHP	🙈 Arbed	HE B 200	÷ <u>24.0</u>
	Ξw	🔼 Arbed	HE B 220	
	I WTM	🙈 Arbed	HE B 240	
Ð	Iuc	점 Arbed	HE B 260	0.
Filter	I IPN	DIN 1025-1:1995	HE B 280	560.0
	I IPE	DIN 1025-5:1994	HE B 300	
Manufacturer/Standard group:	THEB	🥅 DIN 1025-2:1995	HE B 320	10.0
All 🗸	I HE A	DIN 1025-3:1994	HE B 340	
Manufacturer/Standard:	THEM	DIN 1025-4:1994	HE B 360	
· · · · · · · · · · · · · · · · · · ·	THEC	DIN 1025-4:1994	HE B 400	
All ~	IHE	British Steel	HE B 450	
Cross-section shape:	I IPE	🔠 British Steel	HE B 500	z
All	I IPER	🔠 British Steel	HE B 550	
	I)	🔠 British Steel	HE B 600	
Cross-section note:	IUB	🔠 British Steel	HE B 650	[mm]
All	I UBP	🚟 British Steel	HE B 700	
	ΙHP	🚟 BS 4-1	HE B 800	ð 🏹 🕰
	IΗ	<ul> <li>JIS G 3192</li> </ul>	HE B 900	Material
	II	JIS G 3192	HE B 1000	
	IΗ	🙈 Arbed		2 - Steel S 235   EN 10025-2:2004-11
🗌 Include invalid 🛛 🖉	II	GOST 8239-89		🕰 🔁
Favorites group:	II	GOST 8239-72		
- · 🎦 🐼	II	GOST 19425-74		HE B 260   DIN 1025-2:1995
	T IP	GOST 26020-83		
D 000 110 000 000 000 000 000 0000 0000				OK Cancel



We confirm all dialog boxes with [OK].

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

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



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



## 4.5.2.2 Connecting Members Eccentrically

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

### **Defining eccentricity**

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

Edit Member		×
General Options Effective L	engths Modify Stiffness	
Member No.		
Member Eccentricity		
None	~ 🛅 🖾	
New Member Eccentricity		×
Member Eccentricity No.		Absolute Offset
Absolute Offset	Relative Offset	Y X Ey
Reference system O Local x,y,z (e) Global X,Y,Z Member start i ei,X 0.0 (c) (mm) ei,Y 0.0 (c) (mm)	Cross-section alignment	z ez ez ez ez y y
ei,Z 0.0 + [mm]	Object: No. ● Member 6 ✓ 🖏 ○ Surface	Axial Offset
Member end j           ej,X         0.0 + [mm]           ej,Y         0.0 + [mm]           ej,Z         0.0 + [mm]	Axis offset O O O O O O Z Z Z	
Member hinge location at end node (if the hinge is set) Member start Member end	Axial offset from adjoining members at: Member start Member end	Comment
		OK Cancel

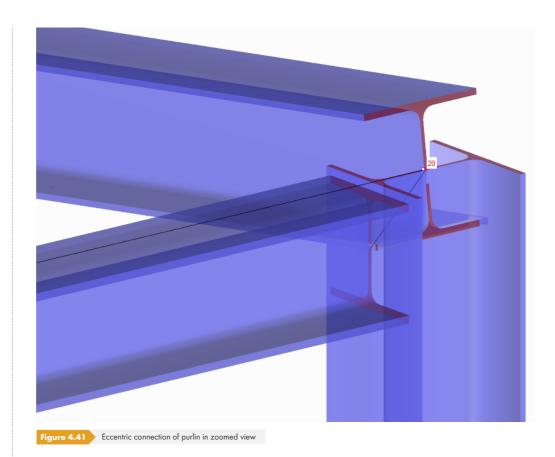
Figure 4.40 New Member dialog box, Options tab, New Member Eccentricity dialog box

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

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

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

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



Applying eccentricity to another member

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

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

	A	B	C	D	E	F	G	H		J	K	L	М	N
Member	Line		Cross-Se	ection No.	Membe	r Rotation	Hing	e No.	Eccentr.	Division	Taper	Length	Weight	
No.	No.	Member Type	Start	End	Туре	β["]	Start	End	No.	No.	Shape	L [m]	W [kg]	
1	13	Beam	1	0 1	Angle	0.00	0	0	0	0		4.000	706.86	Z
2	14	Beam	1	1	Angle	0.00	0	0	0	0		4.000	706.86	Z
3	3	Rib	2	2	Angle	0.00	0	0		0		6.000	1500.00	Y
4	15	Beam	<b>I</b> 3	<b>I</b> I 3	Angle	0.00	0	0	0	0		3.000	266.11	Z
5	16	Beam	<b>I</b> 3	<b>I</b> I 3	Angle	0.00	1	1	0	0		6.059	537.46	YZ
6	17	Beam	<b>I</b> 3	<b>I</b> 3	Angle	0.00	0	0	0	0		3.843	340.92	Z
7	19	Beam	<b>I</b> 3	<b>I</b> I 3	Angle	0.00	0	0	0	0		3.000	266.11	Z
8	20	Beam	<b>I</b> 3	<b>I</b> 3	Angle	0.00	1	1	0	0		6.059	537.46	YZ
9	21	Beam	<b>I</b> 3	<b>I</b> 3	Angle	0.00	0	0	0	0		3.843	340.92	Z
10	18	Beam	<b>I</b> 4	<b>I</b> 4	Angle	0.00	0	0	1 🗾	0		6.700	620.62	X
11	22	Beam	<b>I</b> 4	<b>I</b> I 4	Angle	0.00	0	0		0		6.700	620.62	X
12														
< .														

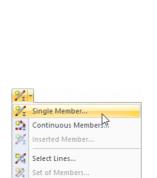
Figure 4.42 Assigning eccentricity in Table 1.17 Members

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

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

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





### 4.5.3 Diagonal

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

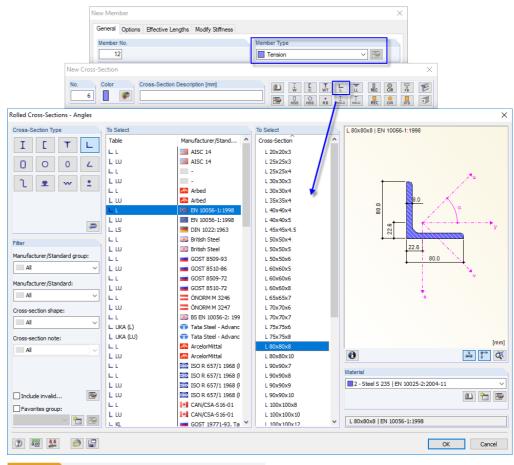
A tension member has the effect that the model is calculated non-linearly: In case of compression forces, this member is removed from the stiffness matrix (failure).

### 4.5.3.1 Defining the Member

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

We define a new cross-section for the Member start, which we import from the database using the section (see Figure 4.19 2).

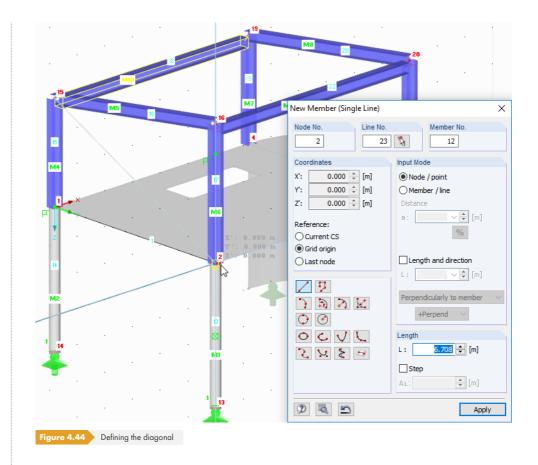
In the New Cross-Section dialog box, we click the [L] button, and in the subsequent Rolled Cross-Sections - Angles dialog box, we select the EN standard cross-section **L 80x80x8**. The default material is number 2 - Steel S 235 again.



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

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

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





## 4.5.3.2 Rotating a Member

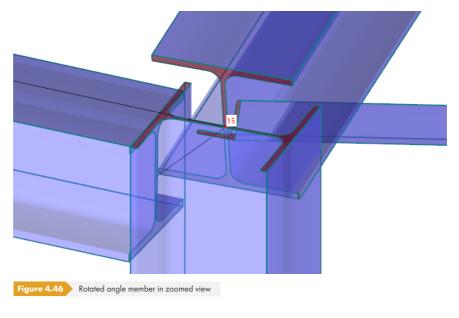
A tension member adds to the stiffness only via its cross-sectional area. Thus, from a structural point of view, the member's rotation is irrelevant. For the rendered view, however, we want to rotate the angle section.

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

General Ontion	ns Effective Lengths Modify St	
Uption	is Effective Lengths Modify St	timness
Member No.	Line No.	Member Type
12	23 🗸 🏷	Tension 🗸 🖾
Node No.		L 80x80x8
15,2		
Member Rotatio	n via	
Angle	β: -90.00 <b>≑</b> ▶ [°]	
O Help node	No.: Inside 🗸 🍾 🏠	
In plane:	() x-y	
	○ x-z	
		0
Cross-Section		
Member start:	L 80x80x8 L	Steel S 235
		- 📖 🛏 📨 🔒
Member end:		
Member end: Member Hinge Member start:		
Member Hinge		



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



### Undo/Redo

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

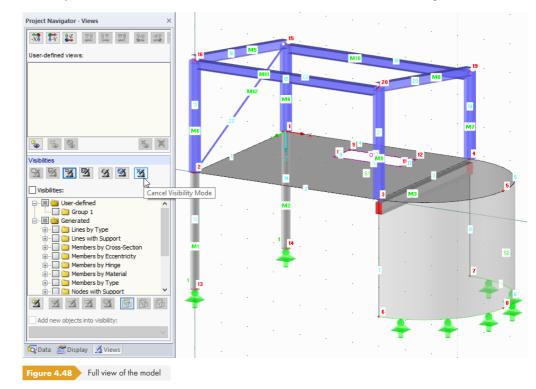
 File
 Edit
 View
 Insert
 Calculate
 Results
 Jools

 Image: Imag

#### **Canceling visibility mode**

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

The full spatial view can be set with the [Isometric View] button in the toolbar (see Figure 4.48 2).





2

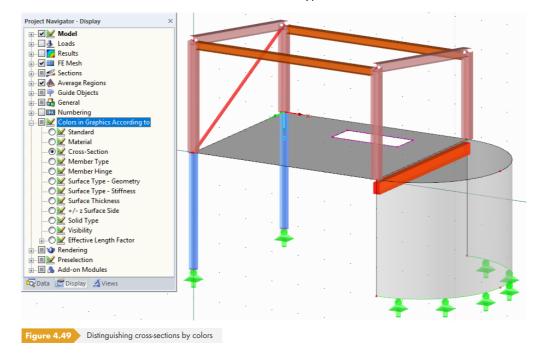
5



### **Adjusting color assignments**

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

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



From here on, the numbering of objects may be turned off. Do to so, right-click in an empty area of the work window. Clear Show Numbering in the shortcut menu (see Figure 4.4 12).

Afterwards we reset the colors to Standard.

## Checking the Input

### Checking the Data navigator and tables

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

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

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

### Saving data

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

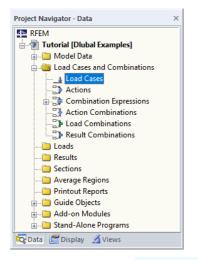
#### File → Save

or the corresponding toolbar button.

4.6

5

# 5 Load



5.1

B

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

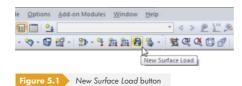
- Load Cases
- Actions
- Combination Expressions
- Action Combinations
- Load Combinations
- Result Combinations

Load cases are used to define the different types of loads such as self-weight, snow load, and wind load. Afterwards, they are assigned actions and superimposed with partial safety factors according to the combination expressions of the standard (see Chapter 6 🗷 ).

## Load Case 1: Self-Weight

The first load case contains the permanently acting loads from self-weight, floor structure, earth pressure, and roof construction (see Chapter  $2.3 \square$ ).

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



The Edit Load Cases and Combinations dialog box appears.

d Cases	Actions	Combination Expressions	Action Combinatio	ns Load Combinati	ons Result Combinations		
sting Load	I Cases		LC No.	Load Ca	se Description		To Solve
G LC1 Dead Load	1	Dead Lo		~			
				Calculation Parame	ters		
			Action Ca			EN 1990   CEN	
						· · ·	
	Self-Wei						
				r in direction:			
	x :	0.000 ≑ [-]					
	Y:	0.000 ≑ [-]					
	Z :	1.000 ≑ [-]					
			> Comment			~ 6	
	37	B Z X Z	×				
							ОК Са



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

### 5.1.1 Self-Weight

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

### 5.1.2 Floor Structure

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

New Surface Load			
No. On Surfaces No.			Load Type 'Force' Load Distribution 'Uniform'
Load Type	Load Direction		
Force     Temperature     Axial strain	Local related to true area:	⊖x ⊖y ⊖z	
O Precamber O Rotary motion  Load Distribution	Global related to true area:	O XL O YL ● ZL	
Uniform     Uinear     Uinear in X     Uinear in X     Uinear in Z	Global related to projected area:	○ XP ○ YP ○ ZP	
O Radial			Load Direction 'ZL'
Load Magnitude			Y
Node No.         Magn           1st:         1         3         p:	tude 1.50 ★ ▶ [kN/m <sup>2</sup> ] ★ ▶ [kN/m <sup>2</sup> ] ★ ▶ [kN/m <sup>2</sup> ]		ż
Comment		~	
2 <b></b> 4			OK Cancel

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

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



X\_XX

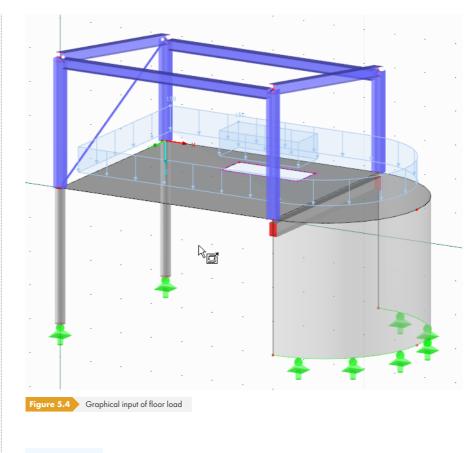
We can now graphically assign the load to the floor surface: Next to the pointer is a small load symbol, which disappears when it gets close to a surface. We apply the load to surface 1 with a click (see Figure  $5.4 \square$ ).

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

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

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

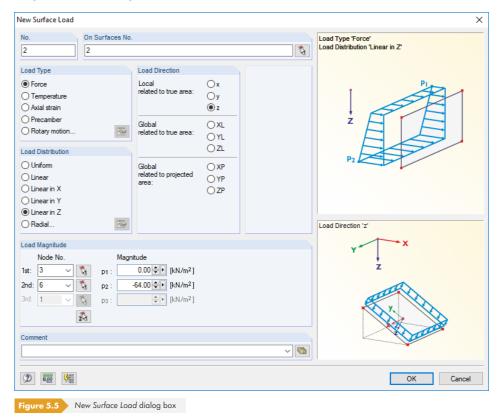




## 5.1.3 Earth Pressure

8

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

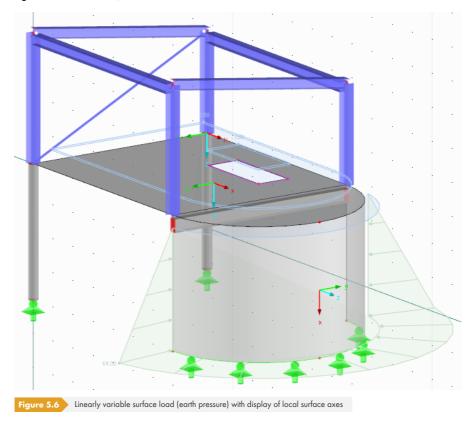




The load is specified as the load type Force with the Load Distribution **Linear in Z** perpendicular to the surface. Thus, we select **Local z** as the Load Direction.

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

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



## 5.1.4 Roof Load

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

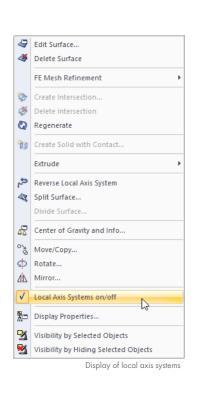
To open the generator dialog box, use

#### Tools $\rightarrow$ Generate Loads $\rightarrow$ From Area Loads on Members via Plane.

In the Convert Area Loads to Member Loads via Planes dialog box, we enter the following settings (see Figure 5.7 2):

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

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





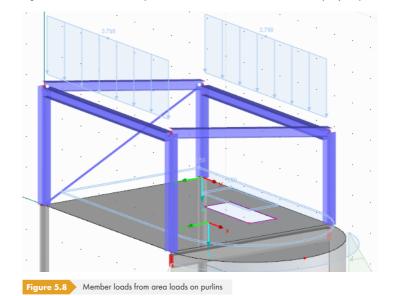
凑

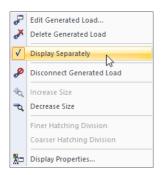
The roof's supporting structure (not displayed in the model) introduces the roof loads into the structural system along the purlins. This means: Both of the monopitch roof's horizontal beams do not participate in transferring loads from the roof loads and are thus excluded from the load generation. We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

Convert Area Loads t	o Member L	oads via Planes			×
Area Load Direction		Member Load Di	rection	Load Distribution Type	
Perpendicular to plane:	⊖z	Direction of gene loads:	erated member	○ Axes of angles ○ Constant	A THE TO
Global related to projected area:	○ XP ○ YP ○ ZP	<ul> <li>Global in X, Y</li> <li>Local in x, y,</li> </ul>	Z	Combined	A LITTLE
Global related to true area:	⊖xl ⊖yl ⊚zl	Area of Load Ap	lane		
Area Load Distribution	n	Area Load Magr	itude		
Uniform     Linear     Varying in direction     Z     Z	n:	Node No. 1: 2: 3: V	1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1	Magnitude           1.20         ↓ ▶         [kN/m²]           ↓ ▶         [kN/m²]           ↓ ▶         [kN/m²]	Area Load Direction 'ZL'
Boundary of the Area	Load Plane				
Corner nodes: 16,15,19,20		✓ <sup>3</sup> ×	Note: Each row in the denotes one pla	drop down list box ane.	
Remove Influence fro	m				
Single members:		\$	Members paralle	el to member:	A CONTRACTOR
Convert Loads to Mer	mbers No.				
10,11				]	
۵۵ 🔊		x.xx ▲			OK Cancel
Figure 5.7 Cor	overt Area Lo	ads to Member L	oads via Planes	dialog box	

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

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





This completes the input for the Dead Load load case.

# Load Case 2: Imposed Load

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

#### Insert $\rightarrow$ Loads $\rightarrow$ New Load Case

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

oad Cases	Actions Combination Expressions A	tion Combinations Load Combinations Result Combinations	
Existing Loa	d Cases	LC No. Load Case Description	To Solve
G LC1	Dead Load	2 Imposed load	
Qi C LC2	Imposed load		
		General Calculation Parameters	
		Action Category EN 1990   C	EN .
			_
		G Permanent 1.A	Ň.
		Gg Permanent/Imposed 1.B	-0
		P Prestress 2	
		QIA Imposed - Category A: domestic, residential areas 3.A	
		OIB Imposed - Category B: office areas 3.B	
		QIC Imposed - Category C: congregation areas         3.C           QID Imposed - Category D: shopping areas         3.D	-
		OLE Imposed - Category E: storage areas 3.E	
		QIF Imposed - Category F: traffic area - vehicle weight ≤ 30 kN 3.F	
		Imposed - Category G: traffic area - vehicle weight ≤ 160 kN 3.G	
		QIH Imposed - Category H: roofs 3.H	
		Qs         Snow (Finland, Iceland, Norway, Sweden)         4.A           Qs         Snow (H > 1000 m a.s.l.)         4.B	
		Qs Snow (H ≤ 1000 m a.s.l.) 4.C	
		Qw Wind 5	
		Qt Temperature (non fire) 6	
		A Accidental 7	
		AE Earthquake 8	
<		Comment	
<u>م</u>	🚱 🗞 💱 🕅 🖏		à
الت الت			
۵ 🤌			OK Can

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

To change the Action Category to **Q**<sub>i</sub> Imposed - Category C: congregation areas, use the action category list (see Chapter 2.3 🖻). This classification is important for the partial safety factors and combination coefficients of the load combinations.

<u>08</u>



## 5.2.1 Floor Slab

We choose a new way to enter the surface load: We select floor surface 1 by clicking it. When we now open the familiar dialog box using the [New Surface Load] button, the surface number is already filled in.

lew Surface Load				
No.	On Surfaces No.			Load Type 'Force' Load Distribution 'Uniform'
Load Type		Load Direction		
<ul> <li>Force</li> <li>Temperature</li> <li>Axial strain</li> </ul>		Local related to true area:	⊖x ⊖y ⊖z	
Precamber     Rotary motion Load Distribution	2	Global related to true area:	O XL O YL ⊚ ZL	
Uniform     Uniform     Unear     Unear in X     Unear in Y     Unear in Z		Global related to projected area:	○ XP ○ YP ○ ZP	
O Radial	1			Load Direction 'ZL'
Load Magnitude				X
Node No.           1st:         3 ~           2nd:         6 ~           3rd:         1 ~	Magn           국         p:           국         p:           국         p:           국         p:	3.00 ★ [kN/m²]           ★ [kN/m²]           ★ [kN/m²]		
Comment				

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

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



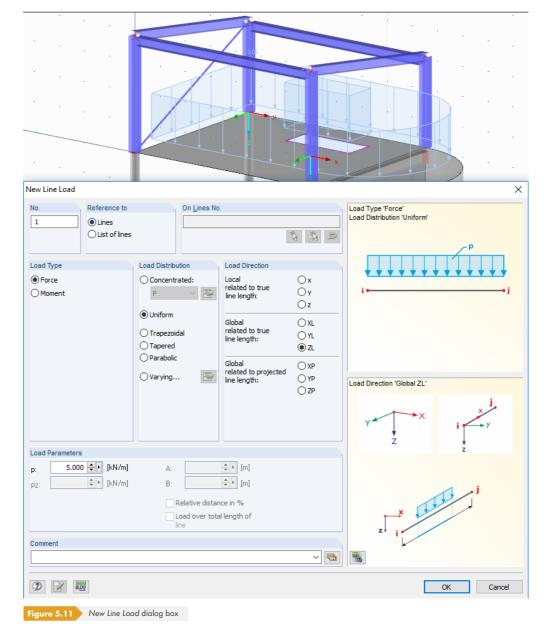
٦

Load

## 5.2.2 Edge of the Opening

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

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



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



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

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

Æ

<u>în</u>

5

#### Load Case 3: Snow 5.3 Once again, we create a [New Load Case] to enter the snow load. Edit Load Cases and Combinations × Load Cases Actions Combination Expressions Action Combinations Load Combinations Result Combinations Existing Load Cases LC No. Load Case Description To Solve G LC1 Dead Load 3 Snow Qi C LC2 Imposed load General Calculation Parar 0= 10 Action Category EN 1990 | CEN Qs Snow (H ≤ 1000 m a.s.l.) G Permanent Gq Permanent/Imposed 1.A 1.B 2 3.A Perstress Imposed - Category A: domestic, residential areas Imposed - Category B: office areas Imposed - Category B: ongregation areas Imposed - Category C: congregation areas Imposed - Category C: storage areas Imposed - Category E: storage areas Imposed - Category C: taffic area - vehicle weight ≤ 30 kN Imposed - Category C: taffic area - vehicle weight ≤ 160 kN Imposed - Vehicle weight ≤ 160 kN Prestres 3.B 3.C 3.D 3.E 3.F 3.G 3.H 4.A Os Snow (H > 1000 m a.s.l.) 4.B Qw Wind Qt Temperature (non fire) A Accidental AE Earthquake ~ 🖻 1 🔁 🕪 📞 🛛 🛪 🖧 × OK Cancel ٦ 🖻

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

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

We specify the Action Category as Qs Snow ( $H \leq 1000 \text{ m a.s.l.}$ ).

#### Roof 5.3.1

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

#### Tools $\rightarrow$ Generate Loads $\rightarrow$ From Snow Loads $\rightarrow$ Flat/Monopitch Roof.

In the Generate Snow Loads - Flat/Monopitch Roof dialog box, we enter the following data (see Figure 5.13 (2):

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

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

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

We want to create Member loads again but the two monopitch roof beams do not contribute to the load transfer (the snow loads are introduced into the structural system by the roof's supporting structure via purlins). We use the [Select Parallel Member] button in the Remove Influence from section to graphically select one of the horizontal beams (member 8 or member 5).

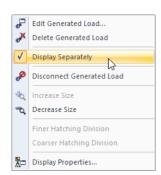




Generate Snow Loads - Flat/Monopitch Roof		×
Snow Load Parameters According to Standard: EN 1991-1-3 National Annex: Zone number Z: 2	Additional Snow Loads         Drifted snow load acc. to 5.3,4(3)         Snow overhang         Snowguard / obstade	Flat/Monopitch Roof
Altitude A : 500.000 (m)	Coefficients	
load sk: 1.60 ↔ [kl/m²] Topography type:	Exposure     Ce:     1.000 +       Thermal coefficient     Ct:     1.000 +	
Roof Geometry	Position of Drifted Snow Load	111111111111111111111111111111111111111
Node No. Node A: 16 $\textcircled{\ }$ $\textcircled{\ }$ B: 15 $\textcircled{\ }$ C: 19 $\textcircled{\ }$ D: 20 $\textcircled{\ }$ Load Case to Generate $\fbox{\ }$ LC s1: $\fbox{\ }$ LC3 $\checkmark$ $\textcircled{\ }$ $\textcircled{\ }$ $\textcircled{\ }$ $\textcircled{\ }$ Create Load Type $\textcircled{\ }$ Member loads $\bigcirc$ Surface loads	Drifted snow load via µ2 instead µ1 Node A: B: C: D: D: D: Load Case for Drifted Snow Load LC s2: Load Distribution Type Axes of angles  Combined Constant	A D C
Remove Influence from		Inclination α : 8.0 [°]
Single members:	Members parallel to member: 8	Generate Snow Loads on Members No.
D 100 100 100 100 100 100 100 100 100 10		OK Cancel
Figure 5.13 Generate Snow Loads - Flat	/Monopitch Roof dialog box	

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

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



## 5.3.2 Floor

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

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

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

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

The load is projected in the XY Plane.

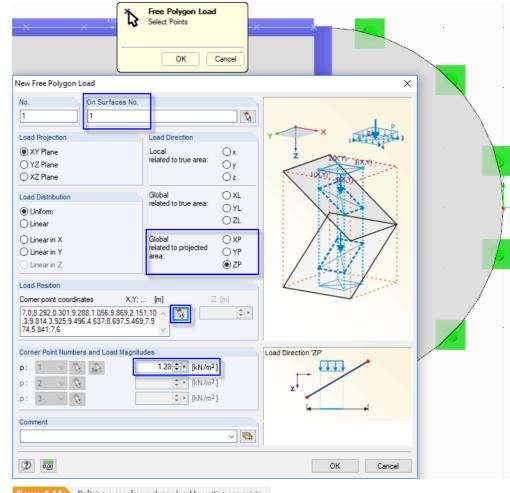


Figure 5.14 Defining a new free polygon load by setting arc points

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

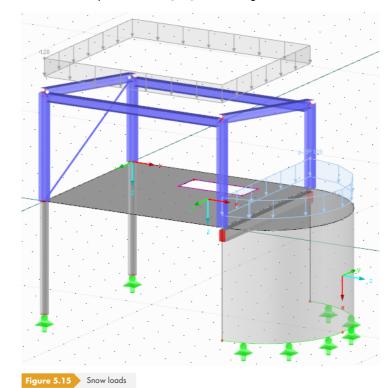
In the Corner Point Numbers and Load Magnitudes section, we enter a value of  $1.28 \text{ kN/m}^2$  — this value was designated as a roof snow load by the generator (see text after Figure 5.13  $\square$ ).





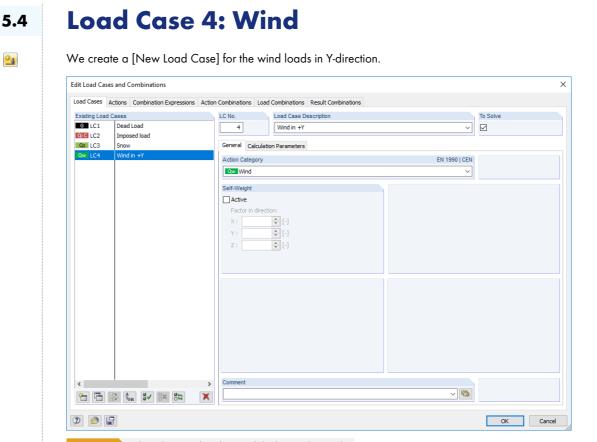
After confirming with [OK], the load is placed on the semicircular surface.

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









Edit Load Cases and Combinations dialog box, Load Cases tab

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

#### **Steel Construction Loads** 5.4.1

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

#### Tools $\rightarrow$ Generate Loads $\rightarrow$ From Wind Loads $\rightarrow$ Vertical Walls with Roof.

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

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

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

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

In the Load Cases to Generate section, we deactivate the load case w-. As described in Chapter 2.3 2, only positive external pressure factors are examined. Load generation of LC w+ is to occur for LC4.

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



滿

enerate Wind Loads	<ul> <li>Vertical Walls with Re</li> </ul>	oof		
Velocity Pressure			Vertical Walls with Roof	
According to National Annex: Wind zone: Terrain category:	EN 1991-1-4 V DIN V 1 V Category III V	Structure height h: 8.000 (m) Altitude A: 500 (m) Fundamental wind velocity		
Simplified case for s Lack of correlation a	tructures with h ≤ 25 m acc. to 7.2.2(3)	vb,0: 22.50 + [m/s]		
Base Geometry Node N Node I: J: L: L:	1 % (A) 4 % 3 % 2 %	Node No.         Node No.           A :         15         E :         Node No.           B :         19         F :         Node No.           D :         16         Image: Second Sec		C C C
Load Cases to General		Set Wind on Side  A - B C - D D - A  Internal Pressure Consider only increasing loads Cpi:  C		70
Create Load Type Member loads		Load Distribution Type O Axes of angles O Combined		
O Surface loads		◯ Constant		
Remove Influence from Single members:	73	Members parallel to member: 8	Generate Wind Loads on Members No. 3,4,6,7,9-11	
2 000 🖻 🖡			OK	Cancel

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

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

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

## 5.4.2 Column Loads

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

### Defining a uniform member load

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

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

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

o.	Reference to		On Member	's No.		Load Type 'Force'	
1	Members		1			Load Distribution 'Uniform'	
	<ul> <li>List of memb</li> <li>Sets of mem</li> </ul>				۵ 🖗 🧳		<u>∠</u> P
oad Type		Load Distrib	ution	Load Direction			ΠΊΠΠ
Force		O Concent	rated:	Local	○×		
Moment		Ρ	~ 🖉	related to true member length:	⊖y ⊖u ⊖z ⊖v	i•	•
) Temperature		Uniform		Global	OXL		
Axial strain		O Trapezoi	dal	related to true	⊙ xL ⊚ YL		
Axial displace	ement	O Tapered		member length:	⊙ zL		
Precamber		O Parabolio	:	Global	. O XP		
) Initial prestre	ss	O Varying.	1	related to projected member length:	H OXP		
) End prestres	s			member lengut.	OZ₽	Load Direction 'Global YL'	
Extra:							
Displacemen	t 🗸 🖾					Y X	i X
ad Parameters	8					2	* z
1.50	0 💠 [kN/m]	A:		<b>≑</b> ▶ [m]			
2:	<b>≑</b> ▶ [kN/m]	B:		<b>≑</b> ▶ [m]			. i. –
	<b>≑</b> ▶ [kN/m]		Relative dista	nce in %			<del>\$</del> /.7
2:			Load over tot	al length of		× 4	
2·			member			z	
omment							
					~ 🖻	<b>*</b>	
	19/11					Г	OK Cance

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

#### Defining a trapezoidal member load

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

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

As before, we define the Load Direction in **YL** globally. The Load Distribution, however, is specified as **Trapezoidal**. This enables two Load Parameters: We enter a value of **0.5** kN/m for member start p 1 and **3** kN/m for member end p2. We defined the columns from bottom to top; thus the member start is at the column base.

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

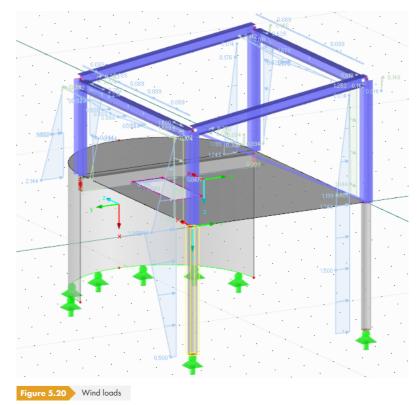
20



	Defense to		On Member	- 11-			
No. 2	Reference to		2	S NO.		Load Type 'Force' Load Distribution 'Trapez	zoidaľ
2	List of mem	bare	2	Г	C & Z		
	O Sets of men				S 🗿		
and Trees		Load Distrib		Lood Discotion		P1	1
Load Type ● Force				Load Direction	0		* * * * * * * * *
Force Moment		O Concent		related to true	Ox Oy Ou	i e	ti
0		P	V 100	member length:			
<ul> <li>Temperatur</li> </ul>	e	OUniform					
🔿 Axial strain		Trapezoi	dal	Global related to true	⊖ xL		
🔿 Axial displac	cement	O Tapered		member length:	⊚ YL ⊖ ZL		
O Precamber		O Parabolio			-	-	
0		- 	House .	Global related to projected	() XP		
<ul> <li>Initial prest</li> <li>End prestre</li> </ul>		O Varying.		member length:	O Y₽	Load Direction 'Global YI	Ľ
	55				⊖ ZP		
O Extra:							×
Displaceme	ent 🛛 🖂 🐼					Y	i y
						ž	
oad Paramete	rs						ż
p1: 0.5	i00 争 [kN/m]	A:		<b>≑</b> ▶ [m]			
p2: 3.0	00 💠 [kN/m]	B:		<b>≑</b> ▶ [m]			
p1:			Relative dista			y _	5/ /
p2:			Load over tot member	ai length of			
Comment		1.00				Zt i	_ <b>x</b>
comment					~ 🖻	<b>a</b> .	
					~ <u>~</u>		
2							0%
	0.00 <mark>%</mark> E						OK Cancel

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

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



**Dlubal** 

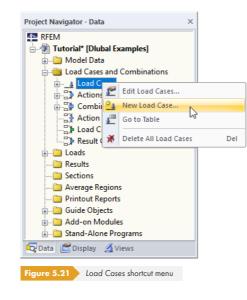
I

## 5.5

## Load Case 5: Imperfection

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

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



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

ad Cases 🛛 🗚	Actions Combination Expressions Ac	on Combinations Load Combinations Result Combinations	
kisting Load	Cases	LC No. Load Case Description	To Solve
GLC1	Dead Load Imposed load	5 Imperfection towards +Y	~ V
s LC3	Snow	General Calculation Parameters	
w LC4	Wind in +Y Imperfection towards +Y	Action Category	EN 1990   CEN
		Imp Imperfection	~
		Self-Weight	
		Active	
		X:	
		Y:	
		Z :	
		Comment	
5 6 8	Se S√ S× S≥ 7		

Then we confirm the dialog box with [OK].





### 5.5.1 Steel Columns

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

ew Imperfection					
1	Reference to Members List of members Sets of members	On Members No.		=0°	α×0°
Direction local axis: () y	According to Standard EN 1993-1-1: 2005-07	(Eurocode 3)	~	X	Y
⊖z	Parameters			*	± *v
Principal Ou axis: Ov	Reference:	<ul> <li>Relative</li> <li>Absolute</li> </ul>		δ	*
	Inclination 1/φ <sub>0</sub> :	200.00 (-)	<b>ð</b>	j •	
	Precamber L/eg : Precamber activity	250.00 + [-] EN 1993-1-1 (5.8)	0		eo _
Calcu	criterion: late Value of Inclination - I				X
Demment Basic Struc D 0.00 Num one I Redu	ation Parameters         c value $\Phi_0$ : 1 /         ture height       h :         ber of columns in row       m :         uction factor $\alpha_h$ :         uction factor $\alpha_m$ :         hation $\Phi : 1 /$	200.00 € [·] 4.000 € [·] 1 € 1.000 [·] 1.000 [·]	Inclination according to EN 1993-1-1: 2005-07 $\Phi = \Phi_{o} \cdot \alpha_{h}$ $\alpha_{h} = \frac{2}{\sqrt{h}}$ $\alpha_{m} = \sqrt{0.5}$	(Eurocode 3) $\cdot \alpha_{m}$ $\geq \frac{2}{3} \leq 1,0$	
D	<u>0.00</u>			ОК	Cancel

Figure 5.23 New Imperfection dialog box

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

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

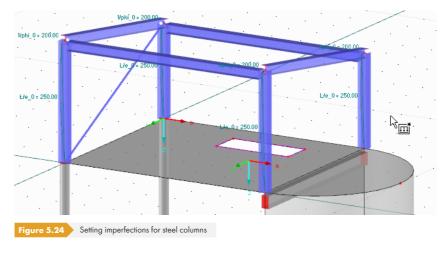
To enter the Inclination  $1/\varphi_0$ , we use B which opens the Calculate Value of Inclination dialog box. We change the Structure height to **4** m. To return to the previous dialog box, we click [OK].

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

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

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

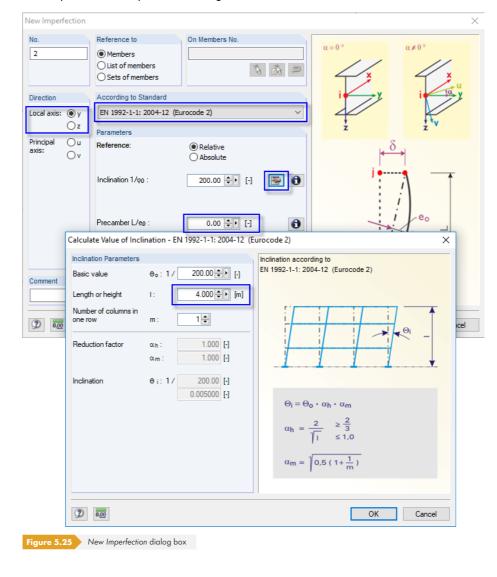
To quit the function, right-click or use [Esc].



5.5.2 Concrete Columns

10 -

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



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



As before, we use  $\mathbb{B}$  to define the Inclination  $1/\varphi_0$ : In the Calculate Value of Inclination dialog box, we change the Length or height to **4** m.

As precambers do not have to be taken into consideration according to Eurocode 2, we specify the Precamber  $L/e_0$  to be **O**.

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

## **Checking Load Cases**

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

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

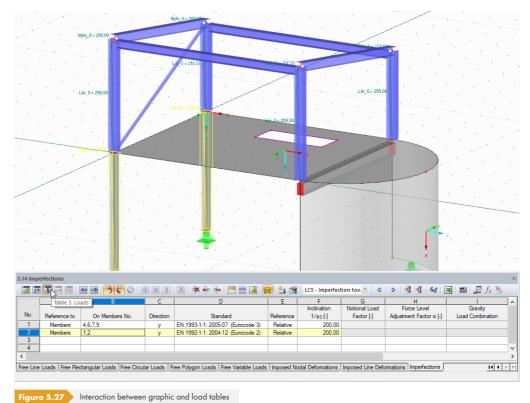
LC4 - Wind in +Y	🖸 😒 🛃 🔁 🖉 🔁 🔁
- 邱 - 🗊 🔐 - 🖄 - 🎙 월 월 월	🍄 - 🕅 Previous Load Case, 🕅 🔽
Figure 5.26 Browsing the load cases	

√₫

5.6

The loads' graphical input is also reflected in both the Data navigator tree and the tables.

We can access the load data in Table 3. Loads, which can be selected on the table toolbar with the button shown on the left. Graphic and tables are interactive once again: For example, to find an imperfection in the table, go to Table 3.14 *Imperfections* and select the load in the work window. The pointer jumps to the corresponding table row.



Dlubal

# **6** Combining Actions

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

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

6.1

4

## **Checking Actions**

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

Load cases, actions, and combinations are managed in the *Edit Load* Cases and Combinations dialog box (see Figure 5.22 🗷) and in the number 2 tables. The latter are accessible by clicking the button depicted on the left. Table 2.1 Load Cases shows our five load cases with the defined action categories in an overview.

2	C 🖂 🖾 🛛 📾 🔜 🔁 🕄 🖓	) 🔮 🥵 🕻	I I 🐹 I 🗱 🖶 🖶 I 🛄 🧱 I 🎊 I	6 <b>4</b> 🛛 🕱	📰 ab=	$f_x \not f_x$			
Load	Table 2. Load Cases and Combinatio	ns B	C EN 1990   CEN	D Se	E f-Weight - F	F actor in Direc	G   ction	Н	
Case	Description	To Solve	Action Category	Active	X	Y	Z	Comment	
LC1	Dead Load	<b>V</b>	G Permanent		0.000	0.000	1.000		
LC2	Imposed load	<b>V</b>	QiA Imposed - Category A: domestic, resider						
LC3	Snow		Qs Snow (H <le>1000 m a.s.l.)</le>						
LC4	Wind in +Y	<b>V</b>	Qw Wind						
LC5	Imperfection towards +Y	<b>J</b>	Imp Imperfection						
10		A.r. C. I.:	ations   Load Combinations   Result Combination						_

Figure 6.1 Table 2.1 Load Cases

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

		S 🗟 🗟 🕺 🔺 🕶 🖿 🖪		,		<u></u>		
	Α	В	C	D	E	F	G	
	Action	EN 1990   CEN		Load	I Cases in A	ction		
Action	Description	Action Category	Acting	LC.1	LC.2	LC.3	Comment	
A1	Permanent	G Permanent		LC1				
A2	Imposed	QiA Imposed - Category A: domestic, resider		LC2				
A3	Snow	Qs Snow (H ≤ 1000 m a.s.l.)		LC3				
A4	Wind	Qw Wind		LC4				
A5								

Figure 6.2 Table 2.2 Actions

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



## 6.2

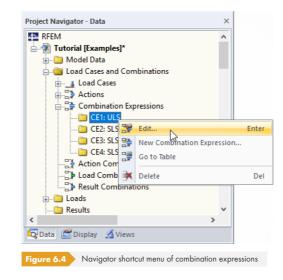
# **Defining Combination Expressions**

In accordance with EN 1990, we have to combine the actions to verify the ultimate limit states and serviceability limit states according to certain rules. Table 2.3 Combination Expressions manages, which limit states are set to be analyzed.

	A	В	С	D	E	_ ^
Combin.	Combination Expression		EN 1990   CEN		Consider	
Express.	Description	Use	Design Situation	Favorable G Actions	Imperfection LCs	E
CE1	ULS	<ul><li>✓</li></ul>	STR ULS (STR/GEO) - Permanent / transient - Eq. 6.10			
CE2	SLS		S Ch SLS - Characteristic			
CE3	SLS		S Fr SLS - Frequent			
CE4	SLS		S Op SLS - Quasi-permanent			~
<						>
Load Cas	es Actions Combination Expressions	Action Com	oinations Load Combinations Result Combinations			

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

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



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

oud cuses	Actions Combination Expressions /	Action Combinations L	oad Combinations Result Combinations				
Existing Comb	bination Expressions	CE No.	Combination Expression Description			Use	
STR CE1	ULS	1	ULS		~		
S Ch CE2 S Fr CE3	SLS	Ceneral Date	Ice - Leading Variable Actions Options for Comb				
Qp CE4	SLS						
		Design Situatio		EN 1	1990   CEN		
		SIR ULS (SI	R/GEO) - Permanent / transient - Eq. 6.10		~ 🕄		
		Settings					
		Consider:					
		Favorable p	ermanent actions				
		Imperfectio					
		Differen	tly for each combination expression				
			/simultaneously acting load cases				
		Differen	tly for each combination expression				
		Reduce numbe by:	r of generated combinations				
			umber of load cases				
		Examining r					
			ading variable actions	Result Combinations			
		<u> </u>		Generate additional (result envelopes)	lly Either/Or	result combination	
				Generate additionally a separate Either/Or result combinati for each combination expression			ibinatio
				Generated Load Comb	inations		
			Senerated Combinations	Method of analysis:	Second-r	order analysis (P-Delti	.a)
		First number of Load combinati	-				
		Result combination					
		Result combina					
<		> Comment					
🖕 🗞	<b>≝</b> √ ×	Imperfection	only with wind		~ 🖻		E
	-					OK	Can

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

LC	Load Case Description	Use	Only wit	h Load C	ases Ne	ver with Load Case	s	
LC5	Imperfection towards +Y		LC4					
					1			
				Select	Load Cases			
					G LC1	Dead Load		
					Qi A LC2	Imposed load		
					Qs LC3	Snow Wind in +Y		
					Qw LC4	wind in +r		
ptions								
	nperfection load cases as alternative							
	ove co-existence of the same load o	combination wi	ith and					
with	out imperfection							
Rem	ove all load combinations without im	perfection						
	er common assignment of imperfection			<b>3</b> 1	8 <b>2</b>  X	All	~	
	er common assignment of imperfectio	5115			04 0^	All		
7 Subi	ect to specific load cases							-1
							OK	Close

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



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

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

In the Settings section of the initial dialog box (Figure 6.5 2), we can reduce the number of generated combinations by Selecting leading variable actions. Selecting this option adds a new tab to the dialog box.

1	ULS	~			
eneral Re	educe - Leading Variable A	tions Options for Combinat	ions		
elect Lead	ng Variable Actions				
	Action	EN 1990   CEN		Leading	
Action	Description	Action Category	Load Cases in Action	Actions	
Action	Description				
Action A2	Imposed	Qi A Imposed - Category /	LC2	<b>V</b>	
				<u> </u>	
A2	Imposed	QiA Imposed - Category /			

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

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

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

2.3 Comb	bination Expression	5					×
2	3 🖂 🌆 🛛 😂	🛃 <mark> 🎖</mark> 🕲	😳 🥽 🕄 🚿 🎐	( 🔤 🕪   💾 📰 📝	📸 😽 🗷 🖩	$f_x \xrightarrow{m} f_x \xrightarrow{m} f_x$	
	E	F	G	Н		J	
Combin.	Consider			Reduce number due to		Generated	
Express.	Imperfection LCs	Ex/Inclusive LCs	Restriction of Load Case	Examining Results	Leading Actions	Action Combinations	
CE1	<b>V</b>				V	AC1 AC13 (13/13)	
CE2	Image: A start and a start						
CE3	<b>V</b>						
CE4	V						<b>~</b>
<							>
Load Cas	ses Actions Combin	ation Expressions 🕅	ction Combinations Load	Combinations Result Co	mbinations		

Figure 6.8 Table 2.3 Combination Expressions, Generated Action Combinations column

6.3

## **Creating Action Combinations**

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

	A	B	С	D	E	F	G	H		J	K
Action	Action Combination		EN 1990   CEN		tion.1		tion.2		ction.3		tion.4
ombin.	Description	Use	Design Situation	Factor	No.	Factor	No.	Factor	No.	Factor	No.
AC1	1.35G	Image: A start and a start	STR ULS (STR/GEO) - Permanent / transie	1.35	G A1						
AC2	1.35G + 1.50QiA	Image: A start and a start	STR ULS (STR/GEO) - Permanent / transie	1.35	G A1	1.50	QiA A2				
AC3	1.35G + 1.50QiA + 0.75Qs	✓	STR ULS (STR/GEO) - Permanent / transie	1.35	G A1	1.50	QiA A2	0.75	Qs A3		
AC4	1.35G + 1.50QiA + 0.75Qs + 0.90Qw	<ul> <li>✓</li> </ul>	STR ULS (STR/GEO) - Permanent / transie	1.35	G A1	1.50	QiA A2	0.75	Qs A3	0.90	Qw A4
AC5	1.35G + 1.50QiA + 0.90Qw	✓	STR ULS (STR/GEO) - Permanent / transie	1.35	G A1	1.50	QiA A2	0.90	Qw A4		
IC6	1.35G + 1.50Qs		STR ULS (STR/GEO) - Permanent / transie	1.35	G A1	1.50	Qs A3				
AC7	1.35G + 1.05QiA + 1.50Qs		STR ULS (STR/GEO) - Permanent / transie	1.35	G A1	1.05	QiA A2	1.50	Os A3		
AC8	1.35G + 1.05QiA + 1.50Qs + 0.90Qw		STR ULS (STR/GEO) - Permanent / transie	1.35	G A1	1.05	QiA A2	1.50	Os A3	0.90	Qw A4
C9	1.35G + 1.50Qs + 0.90Qw		STR ULS (STR/GEO) - Permanent / transie	1.35	G A1	1.50	Qs A3	0.90	Qw A4		
C10	1.35G + 1.50Qw	Image: A start and a start	STR ULS (STR/GEO) - Permanent / transie	1.35	G A1	1.50	Qw A4				
	1.35G + 1.05QiA + 1.50Qw	Image: A start of the start	STR ULS (STR/GEO) - Permanent / transie	1 35	G A1	1.05	QiA A2	1.50	Qw A4		
C11	1.30G + 1.00QIA + 1.00QW										Qw A4

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

6.4

### **Creating Load Combinations**

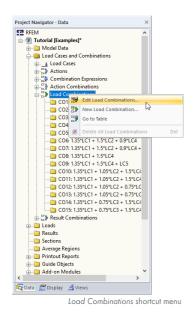
When we browse to Table 2.5 Load Combinations, 15 load combinations are automatically generated from the nine relevant action combinations (see Figure 6.9 2).

	A	В	C	D	E	F	G	H		J	K
Load Combin.		Load Combination			LC.1		LC.2		LC.3		LC.4
	00	Description	To Solve	Factor	No.	Factor	No.	Factor	No.	Factor	No.
CO1	STR	1.35*LC1	<ul><li>✓</li></ul>	1.35	G LC1						
CO2	STR	1.35*LC1 + 1.5*LC2	✓	1.35	G LC1	1.50	Qi A LC2				
CO3	STR	1.35*LC1 + 1.5*LC2 + 0.75*LC3	Image: A start and a start	1.35	G LC1	1.50	Qi A LC2	0.75	Qs LC3		
CO4	STR	1.35*LC1 + 1.5*LC2 + 0.75*LC3 + 0.9*	Image: A start and a start	1.35	G LC1	1.50	Qi A LC2	0.75	Qs LC3	0.90	Qw LC4
CO5	STR	1.35*LC1 + 1.5*LC2 + 0.75*LC3 + 0.9*	✓	1.35	G LC1	1.50	Qi A LC2	0.75	Qs LC3	0.90	Qw LC4
CO6	STR	1.35*LC1 + 1.5*LC2 + 0.9*LC4	✓	1.35	G LC1	1.50	Qi A LC2	0.90	Qw LC4		
C07	STR	1.35*LC1 + 1.5*LC2 + 0.9*LC4 + LC5	✓	1.35	G LC1	1.50	Qi A LC2	0.90	Qw LC4	1.00	Imp LC5
CO8	STR	1.35*LC1 + 1.5*LC4	<b>V</b>	1.35	G LC1	1.50	Qw LC4				
CO9	STR	1.35*LC1 + 1.5*LC4 + LC5	✓	1.35	G LC1	1.50	Qw LC4	1.00	Imp LC5		
CO10	STR	1.35*LC1 + 1.05*LC2 + 1.5*LC4	✓	1.35	G LC1	1.05	Qi A LC2	1.50	Qw LC4		
CO11	STR	1.35*LC1 + 1.05*LC2 + 1.5*LC4 + LC5	Image: A start and a start	1.35	G LC1	1.05	Qi A LC2	1.50	Qw LC4	1.00	Imp LC5
CO12	STR	1.35*LC1 + 1.05*LC2 + 0.75*LC3 + 1.5	<b>V</b>	1.35	G LC1	1.05	Qi A LC2	0.75	Qs LC3	1.50	Qw LC4
CO13	STR	1.35*LC1 + 1.05*LC2 + 0.75*LC3 + 1.5	<b>V</b>	1.35	G LC1	1.05	Qi A LC2	0.75	Qs LC3	1.50	Qw LC4
CO14	STR	1.35*LC1 + 0.75*LC3 + 1.5*LC4	<b>V</b>	1.35	G LC1	0.75	Qs LC3	1.50	Qw LC4		
CO15	STR	1.35*LC1 + 0.75*LC3 + 1.5*LC4 + LC5	<b>V</b>	1.35	G LC1	0.75	Qs LC3	1.50	Qw LC4	1.00	Imp LC5
<											

Figure 6.10 Table 2.5 Load Combinations

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

As required, imperfections can only be found in combination with wind actions Q<sub>w</sub>.



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

Edit Load Cases and Combinations					×
Load Cases Actions Combination Expression	Action Combinations	oad Combinations Result Combinations			
Existing Load Combinations	CO No.	Load Combination Description		To Solve	
STR CO1 1.35*LC1	7	STR 1.35%C1 + 1.5%C2 + 0.9%C4 + LC5	4		
STR CO2 1.35*LC1 + 1.5*LC2				_	
STR CO3 1.35*LC1 + 1.5*LC2 + 0.75*	LC3 General Calo	ulation Parameters			
STR CO4 1.35*LC1 + 1.5*LC2 + 0.75*	Load Cases in	Load Combination CO7			
STR CO5 1.35*LC1 + 1.5*LC2 + 0.75*	INO. Fdu		Action	Leading 🤉	
STR CO6 1.35%LC1 + 1.5%LC2 + 0.9%L		350 G LC1 - Dead Load	G A1 - Permanent		1.35
STR C07 1.35 <sup>4</sup> C1 + 1.5 <sup>4</sup> C2 + 0.9 <sup>4</sup>		.500 @iA LC2 - Imposed load 900 Qw LC4 - Wind in +Y	QiA A2 - Imposed Qw A4 - Wind		1.50 1.50 0.60
STR CO8 1.35*LC1 + 1.5*LC4		.000 Imp LC5 - Imperfection towards +Y	Qw A4 - Wind		1.50 0.60
STR CO9 1.35*LC1 + 1.5*LC4 + LC5					
STR         CO10         1.35*LC1 + 1.05*LC2 + 1.5*           STR         CO11         1.35*LC1 + 1.05*LC2 + 1.5*					
STR CO12 1.35°LC1 + 1.05°LC2 + 1.5° STR CO12 1.35°LC1 + 1.05°LC2 + 0.75					
STR CO12 1.35°C1 + 1.05°C2 + 0.75 STR CO13 1.35°C1 + 1.05°C2 + 0.75					
STR CO14 1.35°LC1 + 1.05°LC2 + 0.75°LC3 + 1.5°					
STR C014 1.35*LC1 + 0.75*LC3 + 1.5*					
				1	
<	> Comment				
All (15)	/ ×		~ 🔁		🕾 🔍
۵ 🖹				OK	Cancel
Figure 6.11 Edit Load Case	es and Combinatio	ns dialog box, Load Combinations tab			

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

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

	Safety Factors Combination		nsequenc	es Class			
artia	al Safety Factors for Static Ed	quilibrium		Basic	Design Situation		
ctio	n Category			Combination	Accidental	Earthquake	
A.	Permanent Actions	Unfavorable	γG,sup∶	1.10 ≑	1.00 🌲	1.00 ÷	
		Favorable	γG,inf∶	0.90 🌲	1.00 🌲	1.00 ≑	
В	Permanent / Imposed	Unfavorable	γG,Q:	1.10 🜩	1.00 🜲	1.00 ‡	
	Prestress	Unfavorable	γP,sup∶	1.10 ‡	1.00 🌲	1.00 ‡	
		Favorable	γP,inf∶	0.90 ‡	1.00 ‡	1.00 ‡	
	Variable Actions	Unfavorable	<b>γ</b> Ω:	1.50 ‡	1.00 🔹	1.00 ≑	
	Accidental Actions		γA:		1.00 ‡		
	Seismic Actions		<u>γ1</u> :			1.00 🚖	
artia	al Safety Factors for Ultimate	Limit State			Design Situation		
	al Safety Factors for Ultimate n Category	Limit State		Basic Combination	Design Situation Accidental	Earthquake	
ctio	-	Limit State Unfavorable	γG,sup :	Basic	-		
ctio	n Category		γG,sup: γG,inf:	Basic Combination	Accidental	Earthquake	
	n Category	Unfavorable	γiG,inf∶	Basic Combination	Accidental	Earthquake	
.A .B	n Category Permanent Actions	Unfavorable Favorable	γiG,inf∶	Basic Combination	Accidental	Earthquake	
.ctio .A .B	n Category Permanent Actions Permanent / Imposed	Unfavorable Favorable	γG,inf: γG,Q: γP:	Basic Combination 1.35 ‡ 1.00 ‡ 1.35 ‡	Accidental	Earthquake	
.A .B	n Category Permanent Actions Permanent / Imposed Prestress	Unfavorable Favorable Unfavorable	γG,inf: γG,Q: γP:	Basic Combination 1.35 ÷ 1.00 ÷ 1.35 ÷ 1.00 ÷	Accidental	Eathquake	
.A .B	n Category Permanent Actions Permanent / Imposed Prestress Variable Actions	Unfavorable Favorable Unfavorable	ΥG,inf: ΥG,Q: ΥΡ: ΥQ:	Basic Combination 1.35 ÷ 1.00 ÷ 1.35 ÷ 1.00 ÷	Accidental	Eathquake	
.A .B 	n Category Permanent Actions Permanent / Imposed Prestress Variable Actions Accidental Actions	Unfavorable Favorable Unfavorable	Υ G,inf : Υ G,Q : Υ Ρ : Υ Q : Υ Α :	Basic Combination 1.35 ÷ 1.00 ÷ 1.35 ÷ 1.00 ÷	Accidental	Eathquake 1.00 ¢ 1.00 ¢ 1.00 ¢ 1.00 ¢	

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

xisting Load C		CO No. Load Combination Description	To Solve
TR CO1	1.35%C1		
TR CO2	1.35%C1 + 1.5%C2	7 1.35*LC1 + 1.5*LC2 + 0.9*	LC4 + LC5
TR CO3	1.35 LC1 + 1.5 LC2	General Calculation Parameters	
TR CO4	1.35 <sup>a</sup> C1 + 1.5 <sup>a</sup> C2 + 0.75 <sup>a</sup> C3 + 0.		
TR CO5	1.35 <sup>a</sup> C1 + 1.5 <sup>a</sup> C2 + 0.75 <sup>a</sup> C3 + 0.	Method of Analysis	Options
TR CO6	1.35 <sup>°</sup> LC1 + 1.5 <sup>°</sup> LC2 + 0.9 <sup>°</sup> LC4	○ Geometrically linear analysis	Modify loading by factor:
TR CO7	1.35*LC1 + 1.5*LC2 + 0.9*LC4 + LC	Second-order analysis (P-Delta / P-delta)	Divide results by loading factor
TR CO8	1.35*LC1 + 1.5*LC4	O Large deformation analysis	Activate stiffness factors of:
TR CO9	1.35*LC1 + 1.5*LC4 + LC5	O Postcritical analysis	Materials (partial factor YM)
TR CO10	1.35°LC1 + 1.05°LC2 + 1.5°LC4		Cross-sections (factor for J, I <sub>V</sub> , I <sub>z</sub> , A, A <sub>V</sub> , A <sub>z</sub> )
TR CO11	1.35*LC1 + 1.05*LC2 + 1.5*LC4 + L0		Members (definition type)
TR CO12	1.35*LC1 + 1.05*LC2 + 0.75*LC3 +	Nonlinear algebraic equations:	
TR CO13	1.35*LC1 + 1.05*LC2 + 0.75*LC3 +	Newton-Raphson	Surfaces (definition type)
TR CO14	1.35 <sup>1</sup> C1 + 0.75 <sup>1</sup> C3 + 1.5 <sup>1</sup> C4	Newton-Raphson combined with Picard	Activate special settings in tab:
TR CO15	1.35*LC1 + 0.75*LC3 + 1.5*LC4 + L0	Picard	Modify stiffness
		O Newton-Raphson with constant stiffness matrix	Extra options
		O Modified Newton-Raphson	Deactivate
		O Dynamic relaxation	Consider favorable effects due to tension of members
		Incrementally Increasing Loading	Refer internal forces to deformed
		Activate	structure for:
		Initial load factor kp :	✓ Normal forces N
		Load factor increment $\Delta_k$ :	Shear forces ⊻ <sub>V</sub> and V <sub>z</sub>
		Refinement of the last load	Moments My, Mz and MT
		increment: 10 🗘	Try to calculate kinematic mechanism (add low stiffness in first iteration)
		$\hfill Stopping condition for: u \hfill \lor$	Apply separate number of load increments for this load combination:
		Node No: Any 🗸 🗘 🗭 [mm	Save the results of all load increments
	>	Use initial load (not increasing):	Deactivate nonlinearities for this load combination
	6B All (15) V		S
<b>)</b>			OK Can

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



6.5

## **Checking Result Combinations**

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

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

			oad Combinations Result Combinati						
isting Result C		RC No.	Result Combination Description				To So	ive	
R RC1	ULS (STR/GEO) - Permanent / tra	nsient 1	STR V ULS (STR/GEO) - F	ermanen	t / transient	- Eq. 6.10	~ 🗹		
		General Calo	ulation Parameters						
		Cuic							
		Existing Loadir	Dead Load		Factor	Result Combina No.	Description	Criterion	Grou
		QIC LC2	Imposed load			STR CO1	1.35*LC1	Permanent	1
		Qs LC3	Snow			STR CO2	1.35*LC1 + 1.5*LC		1
		Qw LC4	Wind in +Y			STR CO3	1.35*LC1 + 1.5*LC		1
		Imp LC5	Imperfection towards +Y			STR CO4	1.35*LC1 + 1.5*LC		l i
		STR CO1	1.35*LC1			STR CO5	1.35*LC1 + 1.5*LC		1
		STR CO2	1.35*LC1 + 1.5*LC2			STR CO6	1.35*LC1 + 1.5*LC		l i
		STR CO3	1.35*LC1 + 1.5*LC2 + 0.75*LC3	D.		STR CO7	1.35*LC1 + 1.5*LC	Permanent	1
		STR CO4	1.35*LC1 + 1.5*LC2 + 0.75*LC3 +	$\rhd^\oplus$	1.00	STR CO8	1.35*LC1 + 1.5*LC	Permanent	1
		STR CO5	1.35*LC1 + 1.5*LC2 + 0.75*LC3 +	P		STR CO9	1.35*LC1 + 1.5*LC	Permanent	1
		STR CO6	1.35*LC1 + 1.5*LC2 + 0.9*LC4		1.00	STR CO10	1.35*LC1 + 1.05*L	Permanent	1
		STR CO7	1.35*LC1 + 1.5*LC2 + 0.9*LC4 + L			STR CO11	1.35*LC1 + 1.05*L	Permanent	1
		STR CO8	1.35*LC1 + 1.5*LC4	$\triangleleft$	1.00	STR CO12	1.35*LC1 + 1.05*L	Permanent	1
		STR CO9	1.35*LC1 + 1.5*LC4 + LC5		1.00	STR CO13	1.35*LC1 + 1.05*L	Permanent	1
		STR CO10	1.35*LC1 + 1.05*LC2 + 1.5*LC4	$\triangleleft  $	1.00	STR CO14	1.35*LC1 + 0.75*L	Permanent	1
		STR CO11	1.35*LC1 + 1.05*LC2 + 1.5*LC4 +		1.00	STR CO15	1.35*LC1 + 0.75*L	Permanent	1
		STR CO12	1.35*LC1 + 1.05*LC2 + 0.75*LC3						
		STR CO13	1.35*LC1 + 1.05*LC2 + 0.75*LC3						
		STR CO14	1.35*LC1 + 0.75*LC3 + 1.5*LC4						
		STR CO15	1.35*LC1 + 0.75*LC3 + 1.5*LC4 +						
					4.00	1 11	-	4	
		A	ll (20) 🗸 🖉		1.00 ~	1.0	Permanent		~ Ø
		O							
		> Comment							
E &	All (1)	×					~ 🖻		2
	· · · · · · · ·								
۱								ОК	Cano

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

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

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

# 7 Calculation



# 7.1 Checking Input Data

60

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

#### Tools $\rightarrow$ Plausibility Check

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

f Check mal h wamings ne, only statistic
h warnings
nerate FE mesh rect collisions of solids
OK Cancel

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

Info	Model Data	Load Data	
Struc	ture Dimensions	;	Structure Weight
Δχ: Δγ: [ ΔΖ: [	10.150 6.300 7.987	[mm]	Surfaces:         45619.70         [kg]           Solids:         0.00         [kg]           Members:         6508.71         [kg]           Total:         52128.40         [kg]
D	0.00		ОК

We can find more tools for checking the data by selecting

#### $\mathbf{Tools} \longrightarrow \mathbf{Model} \ \mathbf{Check}$

which can also be applied to our model.

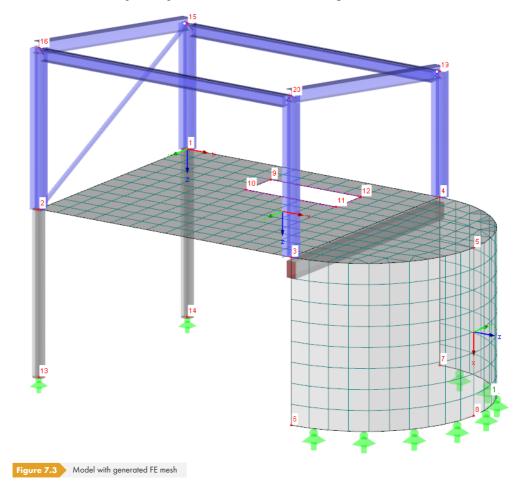


7.2

### **Creating the FE Mesh**

#### **Generating the FE mesh**

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



### **Creating FE mesh refinement**

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

We double-click node **3** to open the *Edit* Node dialog box. We go to the *FE* Mesh tab and select the **Available** box (see Figure  $7.4 \square$ ).

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

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

A spherical refinement area is displayed on the selected node.

Node Coordinates Support FE Mesh	New FE Mesh Refinement	
Node No.	No. Node No.	
FE Mesh Refinement	FE Mesh Refinement Applied to	
☑ Available Type:	Node - circular     Node - rectangular     Line - FE-length     Line - division     Line - gradually     Surface     Solid	
	Parameters           Radius:         2.500 ♀ ▶ [m]           Target FE length           - Inner:         0.100 ♀ ▶ [m]           - Outer:         0.500 ♀ ▶ [m]	
	Comment	OK Cancel

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

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

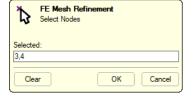
Edit FE Mesh Refinement	>
No.         Node No.           1         3,4           FE Mesh Refinement Applied to	
Node - circular   Node - rectangular   Line - FE-length   Line - division   Line - gradually   Surface   Solid     Parameters   Radius: 2.500 Imiliary   Target FE length   - Inner: 0.100 Imiliary   Outer: 0.500 Imiliary   [m]	
Comment	
2 2	OK Cancel

We close all dialog boxes with [OK].

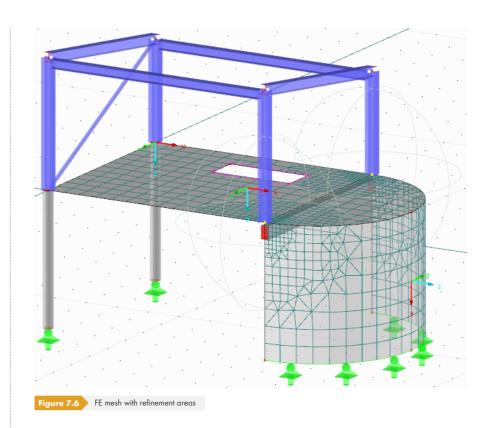
We allow the mesh to be generated again using

#### Calculate → Generate FE Mesh.

Afterwards we check the refinement areas.







### 7.3

**83** 

## **Calculating the Model**

To start the calculation, use the menu item

#### $\textbf{Calculate} \longrightarrow \textbf{Calculate} \textbf{All}$

or the corresponding toolbar button.

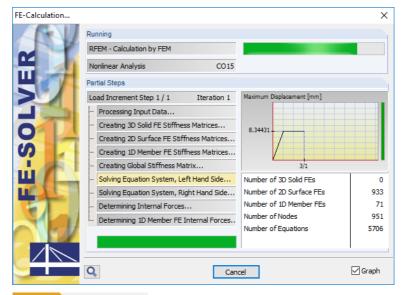


Figure 7.7 Calculation process

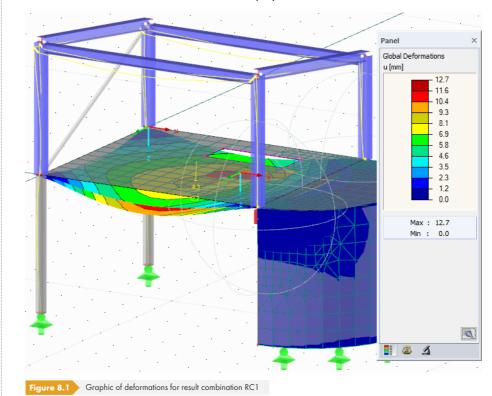
Dlubal

8

# 8 **Results**

## **Graphical Results**

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



### Selecting load cases and load combinations

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

	1 + 1.5°LC2 + 0.75°LC3 1 + 1.5°LC2 + 0.75°LC3 + 0.9°LC4 1 + 1.5°LC2 + 0.75°LC3 + 0.9°LC4 + LC5 1 + 1.5°LC2 + 0.9°LC4
CO10 - 1.35°L CO11 - 1.35°L CO12 - 1.35°L CO13 - 1.35°L CO13 - 1.35°L CO15 - 1.35°L CO15 - 1.35°L	1 + 1.5'LC4 1 + 1.5'LC4 + LC5 C1 + 1.05'LC2 + 1.5'LC4 C1 + 1.05'LC2 + 1.5'LC4 + LC5 C1 + 1.05'LC2 + 0.75'LC3 + 1.5'LC4 C1 + 1.05'LC2 + 0.75'LC3 + 1.5'LC4 + LC5 C1 + 0.75'LC3 + 1.5'LC4 + LC5 R/GEO) - Permanent / transient - Eq. 6.10

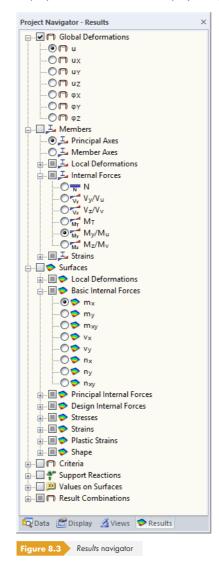


8.1



#### Selecting results in the navigator

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

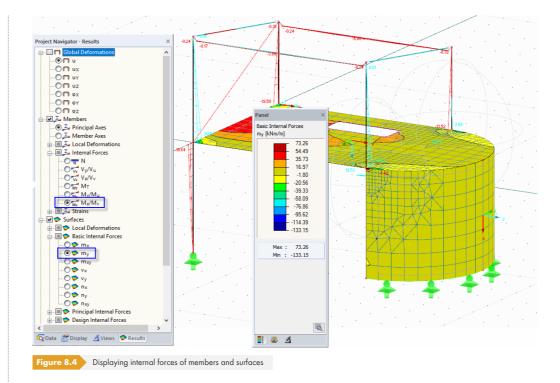


There are check boxes next to every result category (for example Global Deformations, Members, Surfaces, Support Reactions). When we select one of the boxes, the corresponding deformation or internal force is displayed. Next to the entries listed within the categories are more check boxes used to adjust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

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

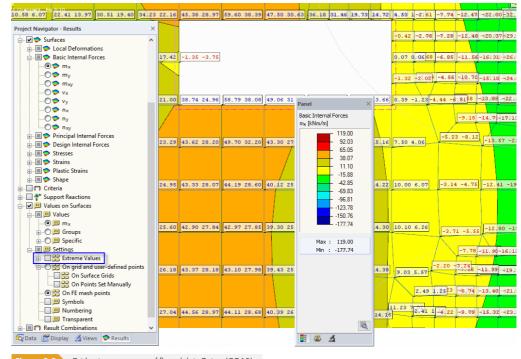
Figure 8.4  $\square$  shows the internal forces of members  $M_z$  and the internal forces of surfaces  $m_y$ , which were calculated for CO13. For displaying internal forces, we recommend to use the wire-frame model, which can be selected with the button shown on the left.





### **Displaying values**

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







8.2

-

## **Results Tables**

We can view the results in the tables as well.

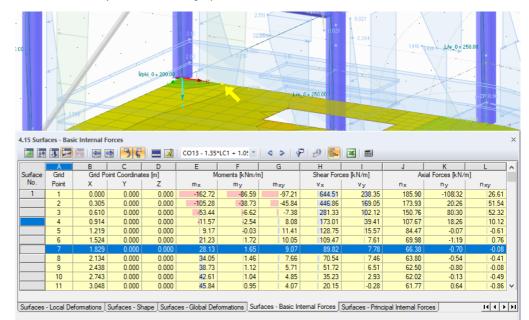
#### **Displaying the tables**

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

A	В	C	D
Table 4. Results Scription	Value	Unit	Comment
□ CO13 - 1.35*LC1 + 1.05*LC2 + 0.75*LC3 + 1.5	5*LC4 + LC5		
<ul> <li>Sum of loads in X</li> </ul>	-1036.79	kN	
Sum of support forces in X	-1036.79	kN	Deviation: 0.00 %
Sum of loads in Y	41.29	kN	
<ul> <li>Sum of support forces in Y</li> </ul>	41.29		Deviation: 0.00 %
<ul> <li>Sum of loads in Z</li> </ul>	1109.74		
<ul> <li>Sum of support forces in Z</li> </ul>	1109.74	kN	Deviation: 0.00 %
<ul> <li>Maximum displacement in X-direction</li> </ul>		mm	Member No. 2, x: 2.800 m
<ul> <li>Maximum displacement in Y-direction</li> </ul>		mm	Member No. 10, x: 2.680 m
Maximum displacement in Z-direction	11.4	mm	FE Mesh Node No. 437 (X: 3.000, Y: 2.500, Z: 0.000 m)
<ul> <li>Maximum vectorial displacement</li> </ul>		mm	FE Mesh Node No. 437 (X: 3.000, Y: 2.500, Z: 0.000 m)
Maximum rotation about X-axis		mrad	FE Mesh Node No. 96 (X: 0.500, Y: 0.000, Z: 0.000 m)
Maximum rotation about Y-axis	-3.7	mrad	FE Mesh Node No. 945 (X: 1.000, Y: 6.000, Z: 0.000 m)
Maximum rotation about Z-axis	-1.4	mrad	Member No. 10, x: 6.700 m

Figure 8.6 Table 4.0 Results - Summary

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



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

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

 Image: Solid Transparent Display Model

 Image: Solid Transparent Display Model

⊿ ■ Dlubal

### Adjusting the results grid

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

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

General	Support / Eccentricit	y FE Mesh	Hinges	Integrated	Axes	Grid	Modify \$	Stiffnes	s			
Surface	No.				Grid	or Resu	lt Values					
1								_	4 b +	h ha		
Grid Type	e					•	!		1 1			
● Carte ○ Polar		Customiz	e			2	: :	:	•••	:	:	
Grid Para	ameters						• •	•	• •	•	•	
Grid poin (+) 1: 40 2: 24	÷ 0÷	🗹 Adapt gri	d distance	35	4 4 4 7 7		· · ·	•	· · ·	•	•	
Grid dista	0.250 + [m]			"								
h :	0.250 🜩 [m]	β: 0.	00 <b>≑</b> ►	"	Grid (	-					id Axis 1	1
Numberir	na					linate (m			Coordin			12
incremen					X: Y: Z:	0.00	0 \$ F 0 \$ F 0 \$ F	~	X: Y: Z:	0.000	) 💠 🕨	1 AN
2		\$						Г	OK		Car	ncel

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

[	A	В	C	D	E	F	G	H Í		J	K	L
Surface	Grid	Grid Poi	nt Coordinat	tes [m]	Mo	ments [kNm/m	1	Shear Forc	es [kN/m]	Axi	ial Forces [kN/m	1
No.	Point	X	Y	Z	mx	my	mxy	Vx	vy	nx	ny	nxy
1	1	0.000	0.000	0.000	-162.72	-86.59	-97.21	644.51	238.35	185.90	-108.32	26.6
	2	0.250	0.000	0.000	-115.61	-47.34	-55.08	482.40	181.51	176.08	-2.85	47.00
	3	0.500	0.000	0.000	-68.49	-8.09	-12.94	320.28	124.67	166.26	102.61	67.50
	4	0.750	0.000	0.000	-34.15	4.74	-0.26	231.44	73.24	130.91	51.72	32.88
	5	1.000	0.000	0.000	0.18	1.39	12.42	142.59	21.80	95.56	0.83	-1.7
	6	1.250	0.000	0.000	10.44	0.16	11.27	126.80	14.70	82.91	-0/20	-0.45
	7	1.500	0.000	0.000	20.69	1.72	10.13	111.01	7.59	70.26	-1.23	0.83
	8	1.750	0.000	0.000	26.34	1.67	9.33	94.90	7.74	67.31	-0.83	0.14
	9	2.000	0.000	0.000	32.00	1.61	8.52	78.78	7.88	64.37	-0.42	-0.56
	10	2.250	0.000	0.000	35.83	1.33	6.92	63.35	7.10	63.30	-0.64	-0.29
	11	2.500	0.000	0.000	39.67	1.05	5.31	47.92	6.31	62.24	-0.86	-0.0
	12	2.750	0.000	0.000	42.69	1.04	4.84	34.87	2.84	62.01	-0.11	-0.50
	13	3.000	0.000	0.000	45.71	1.04	4.37	21.83	-0.64	61.79	0.63	-0.99
	14	3.250	0.000	0.000	46.40	0.57	2.84	13.07	1.23	61.71	0.68	-0.33



Figure 8.9 Result values of surface 1 with refined grid



### 8.3

## **Filtering Results**

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

### 8.3.1 Custom Visibilities

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

### Displaying results for the concrete columns

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

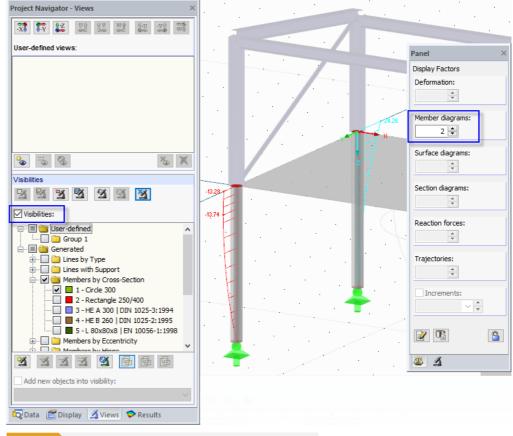


Figure 8.10 Moments M<sub>z</sub> of the concrete columns in scaled representation (CO13)

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

### Adjusting the scaling factor

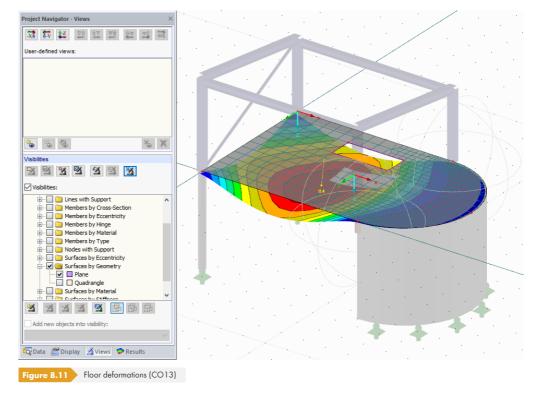
To easily read the diagram of internal forces on the rendered model, we scale the data display in the control tab of the panel: We change the *Member diagrams* coefficient to **2** (see Figure 8.10 ...).

8

#### Displaying results for the floor slab

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

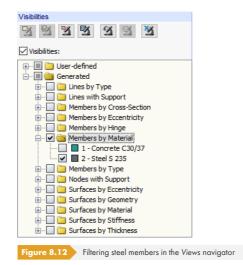
When we change the result type with the [Deformation] button in the toolbar, we get the following display.



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

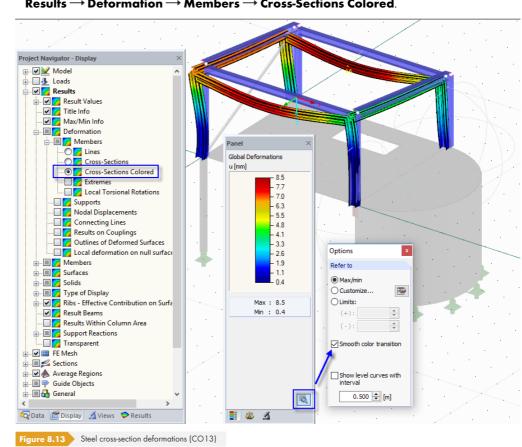
#### Displaying the steel construction's deformations

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



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





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



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

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

8

### 8.3.2 Results on Objects

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

First we clear Visibilities in the Views navigator.

roject Navigator - Views	×	
A P P P P P P P P P	<del>.</del>	
User-defined views:		
	$\times$	
Visibilities		
Visibilities:		
Add new objects into visibility:		
😨 Data 🖀 Display 🏾 🔏 Views 🗢 Results	~	

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

Project Navigator - Results	×	
Members		
🖲 🚣 Principal Axes		
O 🗾 Member Axes		
🗄 🗉 📕 🚣 Local Deformations		
Internal Forces		
••••••••••••••••••••••••••••••••••••••		
·····O vy Vy/Vu		
Ovz Vz/Vv		
····O MT MT		
My/Mu Mz/Mv		
time → Mz Miz/Miv		
Strains		
in in it is a strains		
. Support Reactions		
Distribution of load		
Values on Surfaces		
🛱 Data 🕤 Display 🔏 Views 📚 Results		
Figure 8.15 Displaying member axial forces N	l in tl	ne Results naviga

#### **Special selection**

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

Edit  $\rightarrow$  Select  $\rightarrow$  Special

or the corresponding toolbar button.

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

Category			Members	
Activate	Category	All		
	Nodes		With length:	
	Lines	ā	from: 主 🖏 to: 主 [mm] 🖏	
	Materials			
	Surfaces			
	Solids		With beta:	
	Surface Supports		- from: 🔹 🔹 to: 🔹 🔹	
	Cross-Sections			
2	Members			
	FE Mesh Refinements		With help node:	
				1 and 1
			On nodes:	
				1 all
			Parallel to member:	
			1	S.
			Same as member:	
				*
Ø / D.	- 9te			1 and 1
₩ .	A 69			
			In range:	
Status		~	X [mm] Y [mm] Z [mm]	
Add		Or	from:	1 B
O Selec	t from current selection	O And		10
O Perma	ve from current selection		to:	19
	ive from current selection			

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

### Showing axial forces of columns

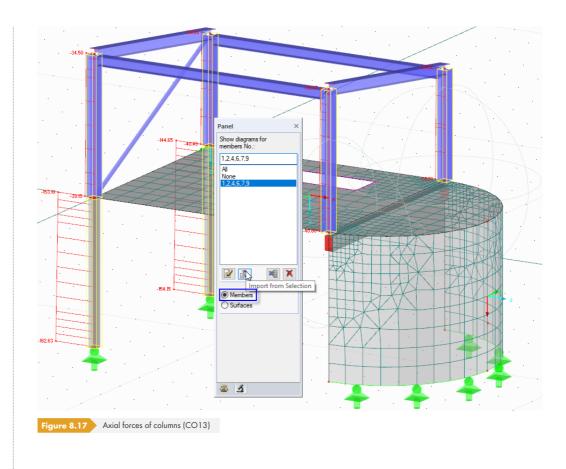
In the panel, we go to the filter tab and select Members.

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



G

8





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

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

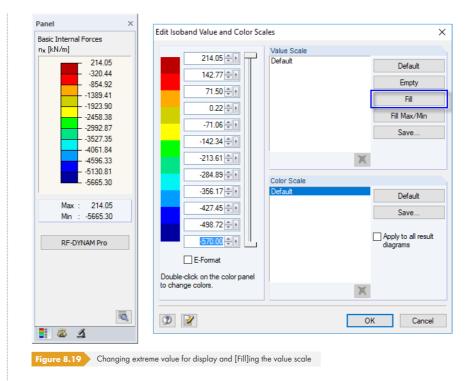
Project Navigator - Results	×
🖅 🔲 🗇 Global Deformations	~
+ Jan Members	
🖶 🔲 🐤 Local Deformations	
🖃 🗐 🗇 Basic Internal Forces	
🔿 🤝 my	
🔿 🤝 m <sub>xy</sub>	
🔿 🗢 vx	
🔿 🗢 vy	
(0 🗢 nx	
🔿 🐤 ny	
🕀 🔲 🗢 Principal Internal Forces	$\checkmark$
🛱 Data 🖀 Display 🔏 Views 🗢 Results	
Figure 8.18 Displaying basic internal forces	s n <sub>x</sub>

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

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

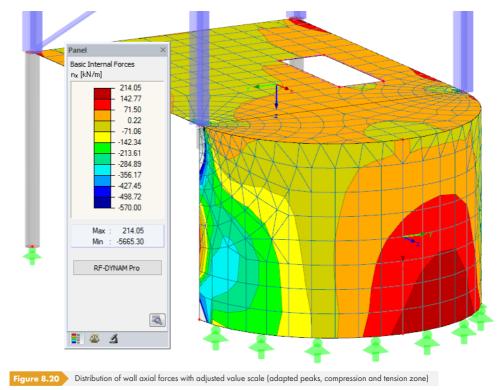


Dlubal



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

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



∠ <u>►</u> Dlubal

8.4

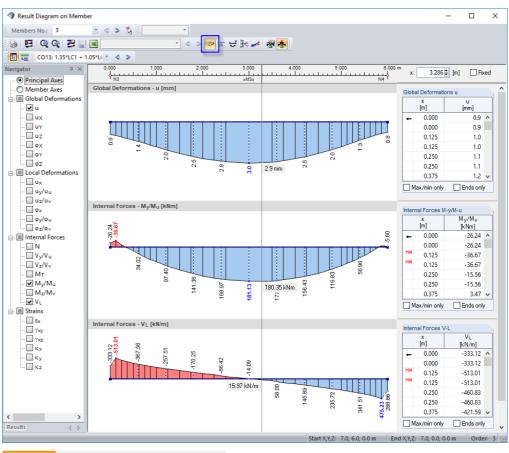
### **Displaying Result Diagrams**

We can also evaluate the results in a diagram that is available for lines, members, line supports, and sections (see Chapter 8.5 2). We can use this feature to take a closer look at the result diagram of the downstand beam.

We hide the surface results. After that, we go to the panel's *Filter* tab and reselect the **All** option for member diagrams (see Figure 8.17 12).

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

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





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

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

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

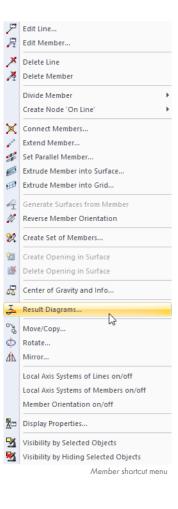
We close the Result Diagrams feature by closing the window.

### 3

9

Q Q

SX.



8.5

## **Creating a Section**

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

We create a new section with

#### Insert — Section

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

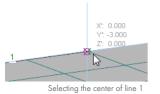
4 RFEM		
🖮 衝 Tutorial (Exam	ples]	
🛓 📄 Model Data		
🗄 🗀 Load Cases	and Combinations	
🗄 🗀 Loads		
🗄 🗀 Results		
Sections	1	
📄 Averag	Edit Section	Enter
- 📄 Printol 🌮	New Section	
🛓 🗀 Guide (	Find Section	
🖶 🚞 Add-oi		
🗄 🗀 Stand 🍏	Delete All Sections	Del
🗟 Data 🛛 🖀 Displ	General Data	
0,00	Units and Decimal Place	·s
	Display Properties	

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

We define the section's edge points graphically using  $\boxed{3}$ : We first click the midpoint of line 1 (global coordinates: 0.000/3.000/0.000), then arc node 5.

No. Section Name   I Center     Section Through   Image: Solid     Solid     Solid     Solid     Solid     Image: Solid     Sol	Vew Section			>
Section Inrough       Section Trough            Surface          O line No.:          Solid          On line No.:          Edge Points of Section         XA:          0.000 ÷ [m]         YA:          3.000 ÷ [m]         ZA:          0.000 ÷ [m]         ZA:          0.000 ÷ [m]         Section on Surface - Projection Direction         YA:          0.000 ÷ [m]         Section on Surface - Projection Direction         YA:          0.000 ÷ [m]         Section on Surface - Projection Direction         YA:          0.000 ÷ [m]         Section on Surface - Projection Direction         YA:          0.000 ÷ [m]         YA:          0.000 ÷ [m]				
Code Fonds of Section         XA:       0.000 ÷ [m]         YA:       3.000 ÷ [m]         YA:       3.000 ÷ [m]         ZA:       0.000 ÷ [m]         Section on Surface - Projection Direction         OX       • Vector         OY       aY:         0.000 ÷ [m]         ZZ:       1.000 ÷ [m]         Show Result Diagram in         Plane:       Local in +z         On surfaces No.       Show values on isolines         On surfaces No.       Show values on isolines	Surface	Via Plane	Y Z X	
O X       ● Vector       ax:       0.000 ♀ • [m]         O Y       a Y:       0.000 ♀ • [m]         O Z       a Z:       1.000 ♀ • [m]         Show Result Diagram in       Options         Plane:       Local in +z       Show values on isolines         O Show walues on isolines       Show values on isolines	X <sub>A</sub> : 0.000 <b>↓</b> Y <sub>A</sub> : 3.000 <b>↓</b>	[m] ×B: 10.000 € [m] [m] YB: 3.000 € [m]		
Plane:     Local in +z     Save section       Show values on isolines     Show values on isolines	OX OY	● Vector ax: 0.000 ÷ [m] ay: 0.000 ÷ [m]		
	Plane: Local in	→ Save section Save section Show values on isolines	x	
OK Cancel	2		ОК Са	incel

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



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

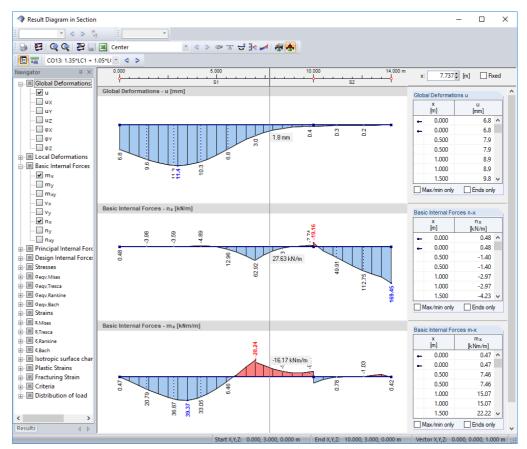


Figure 8.24 Section display in the Result Diagram window

x

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

We deactivate the surface results to display only the diagrams of the section (see Figure 8.26 12).

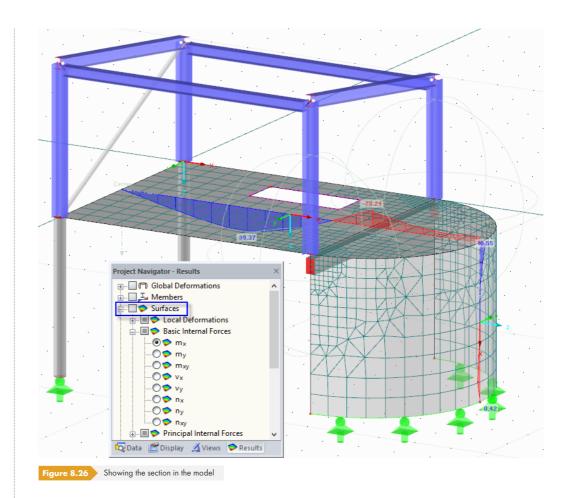
Using the filled display option for sections available in the *Display* navigator, we can highlight the moment diagram in the model.

Project Navigator - Display	×	
🕀 🗹 💓 Model	^	
🛓 🖳 🖶 Loads		
🗄 🗹 🌠 Results		
👳 🗹 🎟 FE Mesh		
🚊 🗉 🗾 🚅 Sections		
🗹 🚅 Descriptions		
🔤 🚅 Draw in Foreground		
🗹 🚅 Result Diagrams Filled		
🗹 🚅 Hatching		
All Values		
🚋 🗹 📣 Average Regions		
🚋 🗐 軯 Guide Objects		
🚋 🗐 🛃 General		
🗄 🗆 🔲 🔝 Numbering	~	
Data 🖆 Display 🔏 Views 🗢 Results		
Figure 8.25 Display options for sections in t	he Disp	olay navigator

Dlubal

٦

Results





# 9 Documentation

9.1

## Creating a Printout Report

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

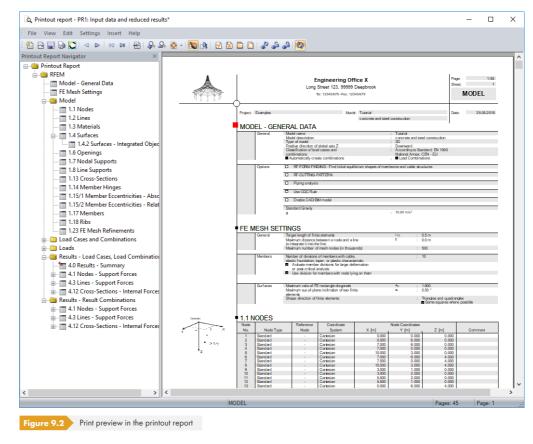
We open the printout report using

#### File → Open Printout Report

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

No. Description Input data and reduced results Printout Report Template	1
Printout Report Template	
1 - Input data and reduced results	× 🔁 🖻
ОК	Cancel

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





9.2

## **Adjusting the Printout Report**

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

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

Printout Report Navigator	×	
🖃 🛁 Printout Report		
🗄 📲 💼 RFEM		
🔤 📶 Model - General Data		
FE Mesh Settings		
🗉 👝 Model		
🖃 🗀 Load Cases and Combinations		
🚛 💼 Loads		
🖃 🚞 Results - Load Cases, Load Comb	inations	
4.0 Results - Summary		
+ 4.3 Lines - Support Forces		
	Forces	
Results - Result Combinations		
🗊 🖬 4.12 Cross-Sections - Inter	-	
	Remo	ve from Printout Report
	Start v	vith New Page
	Select	ion
	Prope	rties
Figure 9.3 Cross-Sections - Internal Forces	hortout	

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

Biological Selected (1)         Tables to Display         Display         141 Nodes - Support Forces         142 Nodes - Deformations         143 Lines - Support Forces         144 Members - Local Deformations         145 Members - Incent Deformations         144 Members - Local Deformations         145 Members - Contact Forces         147 Members - Contact Forces         148 Members - Incent Forces         149 Members - Strains         141 Notes - Strains         141 Notes - Strains         143 Members - Contact Forces         143 Members - Contact Forces         141 Numbers Stendermesses         141 Notes - Strains         141 Starfaces - Colobal Deformations         141 Starfaces - Colobal Deformations         141 Starfaces - Colobal Deformations         141 Starfaces - Local Deformations         141 Starfaces - Colobal Deformation	RFEM	Result C	combinations to Display	/							
Tables to Display       Table       All       Number Selection (e.g. '14.8)         Daplay       1       Number Selection (e.g. '14.8)       All         4.1       Number Selection (e.g. '14.8)       All         4.2       Nodes - Deformations       Q       All         4.3       Lines - Support Forces       Q       All         4.3       Lines - Support Forces       Q       All         4.4       Members - Local Deformations       Q       All         4.5       Members - Strains       Q       All         4.6       Members - Strains       Q       All         4.10       Members - Strains       Q       All         4.11       Suffaces - Local Deformations       Q       All         4.11       Suffaces - Local Deformations       Q       All         4.11       Suffaces - Local Deformations       Q       All         4.12       Suffaces - Local Deformations       Q       All         4.13       Suffaces - Local Deformations       Q       All         4.14       Suffaces - Local Deformations       Q       All         4.14       Suffaces - Local Deformations       Q       All         Data Lis Concess - Section <t< td=""><td></td><td>All</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></t<>		All									
Display       Table       All       Number Selection (e.g. '14.8')         4.1 Nodes - Support Forces        All         4.2 Nodes - Deformations        All         4.3 Lines - Support Forces        All         4.4 Members - Local Deformations        All         4.4.5 Members - Calabed Deformations        All         4.5 Members - Colab Deformations        All         4.5 Members - Contact Forces        All         4.6 Members - Strains        All         4.10 Members Bendemesses        All         4.11 Sets of Members - Internal Forces        All         4.113 Surfaces - Global Deformations        All         4.12 Coses-Sections        All         14.12 Set forces by Cross-Section        All         15.15 Linetral Forces         All         16.25		⊖ Selec	cted (1)	5							
<ul> <li>4.1 Nodes - Support Forces</li> <li>4.2 Nodes - Deformations</li> <li>4.1 Members - Support Forces</li> <li>4.4 Members - Coal Deformations</li> <li>4.1 Members - Strains</li> <li>4.1 Members - Strains</li> <li>4.1 Members - Strains</li> <li>4.1 Members - Strains</li> <li>4.1 Staff of Buckking</li> <li>4.1 Staff of Buckking</li> <li>4.1 Staff of Strains</li> <li>4.1 Staff</li></ul>		Tables to	o Display								
<ul> <li>4.2 Nodes - Deformations</li> <li>4.3 Lines - Support Forces</li> <li>4.4 Members - Local Deformations</li> <li>4.4 Members - Global Deformations</li> <li>4.4 Members - Strains</li> <li>4.6 Members - Ortact Forces</li> <li>4.11</li> <li>4.7 Members - Strains</li> <li>4.11 Staff of Buckling</li> <li>4.11 Staff of Buckling</li> <li>4.13 Surfaces - Local Deformations</li> <li>4.13 Surfaces - Local Deformations</li> <li>4.13 Surfaces - Local Deformations</li> <li>4.13 Surfaces - Calol Deformations</li> <li>4.14 Surfaces - Calol Deformations</li> <li>4.15 Surfaces - Calol Deformations</li> <li>4.14 Surfaces - Calol Deformations</li> <li>4.14 Surfaces - Calol Deformations</li> <li>4.15 Surfaces - Calol Deformations</li> <li>4.14 Surfaces - Calol Deformations</li> <li>4.15 Surfaces - Calol Deformations</li> <li>4.14 Surfaces - Calol Deformations</li> <li>4.15 Surfaces - Calol Deformations</li> <li>4.15 Surfaces - Calol Deformations</li> <li>4.16 Surfaces - Local Deformations</li> <li>4.17 Surfaces - Calol Deformations</li> <li>4.18 Surfaces - Calol Deformations</li> <li>4.19 Surfaces - Calol Deformations</li> <li>4.10 Member - Section</li> <li>5.10 Member - Section</li> <li>1.10 Member - Section</li> <li></li></ul>		Display		Table		1	All		Number S	election (e.g. '1-4,8')	
0       4.3 Lines - Support Forces        V       All         1       4.4 Members - Local Deformations        V       All         1       4.5 Members - Iocal Deformations        V       All         1       4.6 Members - Internal Forces        V       All         1       4.6 Members - Internal Forces        V       All         1       4.8 Members - Strains       V       All         1       4.9 Members - Strains       V       All         1       4.9 Members - Internal Forces       V       All         1       4.10 Members - Internal Forces       V       All         1       4.11 Star of Members - Internal Forces       V       All         1       4.12 Cross-Sections - Internal Forces       V       All         1       4.13 Surfaces - Local Deformations       V       All         1       4.14 Surfaces - Global Deformations       V       All         1       4.14 Surfaces - Global Deformations       V       All         1       4.14 Surfaces - Global Deformations       V       All         1       4.15 Surfaces - Local Deformations       V       All         1 <t< td=""><td></td><td></td><td>4.1 Nodes - Support</td><td>Forces</td><td></td><td> [</td><td>✓</td><td>All</td><td></td><td></td><td></td></t<>			4.1 Nodes - Support	Forces		[	✓	All			
<ul> <li>4.4 Members - Local Deformations</li> <li>4.5 Members - Global Deformations</li> <li>7. All</li> <li>4.5 Members - Internal Forces</li> <li>7. All</li> <li>4.7 Members - Strains</li> <li>7. All</li> <li>4.8 Members - Strains</li> <li>7. All</li> <li>4.9 Members - Cofficients for Buckling</li> <li>7. All</li> <li>4.10 Member - Strains</li> <li>7. All</li> <li>4.11 Sets of Members - Internal Forces</li> <li>7. All</li> <li>4.12 Cross-Sections - Internal Forces</li> <li>7. All</li> <li>8</li></ul>			4.2 Nodes - Deformation	tions			✓	All			
All   4.5 Members - Global Deformations   4.6 Members - Internal Forces   All   4.7 Members - Contract Forces   All   4.8 Members - Strains   All   4.9 Members - Coefficients for Buckling   All   4.11 Sets of Members - Internal Forces   All   4.13 Surfaces - Global Deformations   All   4.13 Surfaces - Global Deformations   All   4.14 Surfaces - Global Deformations   All   4.13 Surfaces - Global Deformations   All   4.14 Surfaces - Global Deformations   All   4.13 Surfaces - Global Deformations   All   4.14 Surfaces - Global Deformations   All   4.13 Surfaces - Global Deformations   All   4.14 Surfaces - Global Deformations   All   4.13 Surfaces - Global Deformations   All   4.14 Surfaces - Global Deformations   All   14.14 Surfaces - Global Deformations   All   14.15 Surfaces - Global Deformations   All   14.14 Surfaces - Global Deformations   All   15 Sufface   Pattion values   Pattion values   Vy/Vu   My / Mu   Extreme values of cross-sections   Contents   Info pictures											
All   4.5 Members - Internal Forces   4.8 Members - Strains   4.9 Members - Strains   4.9 Members - Strains   4.10 Members - Internal Forces   4.11 Sets of Members - Internal Forces   4.12 Cross-Sections - Internal Forces   4.13 Surfaces - Local Defomations   4.14 Surfaces - Colab Deformations   4.14 Surfaces - Colab Deformations   4.15 Extreme values   Nodal values   Nodal values   Nodal values   Vy / Vu   Vy / Vu   Vy / Vu   Mu											
All   4.7 Members - Contact Forces   4.8 Members - Cofficients for Buckling   4.10 Members - Cofficients for Buckling   4.11 Sets of Members - Internal Forces   4.11 Sets of Members - Internal Forces   4.13 Surfaces - Local Deformations   4.14 Surfaces - Global Deformations   4.15 Conserved   All   4.13 Surfaces - Local Deformations   4.14 Surfaces - Global Deformations   All   Details - Internal Forces by Cross-Section   N   Display   Display Max/Min Internal Forces   N Mathematic Surfaces   Vy / Vu   Vy / Vu   Vy / Vu   Mathematic Surfaces   Vy / Vu   Mathematic Surfaces   Osplay   Contents   Vitor protections   Mathematic Surfaces   Mathematic Surfaces   Vy / Vu   Mathematic Surfaces   Mathematic Surfaces   Vy / Vu   Mathematic Surfaces   Mathematic Surfaces   Vy / Vu   Mathematic Surfaces   Vy / Vu   Mathematic Surfaces   Mathematic Surfaces   Mathematic Surfaces   Mathematic Surfaces   Mathe											
All         4.8 Members - Strains         4.9 Members - Coefficients for Buckling         All         4.10 Member Strains         4.11 Sets of Members - Internal Forces         All         4.13 Surface - Coclo Deformations         4.14 Surface - Global Deformations         All         4.14 Surface - Global Deformations         All         Details - Internal Forces by Cross-Section         X         Display         Display         Display Max/Min Internal Forces         Nodal values         N         Pattion values         Vy / Vu       My / Mu         Extreme values of cross-sections         Corterts         Info pictures											
A:9 Members - Coefficients for Buckling       V       All         4:10 Members - Internal Forces       V       All         4:11 Sets of Members - Internal Forces       V       All         4:13 Surfaces - Local Deformations       V       All         4:14 Surfaces - Global Deformations       V       All         0:14 Surfaces - Local Deformations       V       All         0:14 Surfaces - Clobal Deformations       V       All         0:14 Surfaces - Clobal Deformations       V       All         0:15 Play       Display Max/Min Internal Forces       N         Display       Vy / Vu       My / Mu         Extreme values of cross-sections       Vz / Vy       Mz / Mu         OK       Cancel       Cancel											
110 Member Stendemesses       ✓       All         4.11 Sets of Members - Internal Forces       ✓       All         4.13 Surfaces - Local Deformations       ✓       All         14.13 Varfaces - Colobal Deformations       ✓       All         0       4.14 Surfaces - Global Deformations       ✓       All         0       0       1.13 Surfaces - Global Deformations       ✓       All         0       0       1.14 Surfaces - Global Deformations       ✓       All         0       0       1.14 Surfaces - Global Deformations       ✓       All         0       0       1.14 Surfaces - Global Deformations       ✓       All         0       0       1.14 Surfaces - Global Deformations       ✓       All         0       0       1.14 Surfaces - Global Deformations       ✓       All         0       0       1.16 Surfaces - Global Deformations       ✓       All         0       Nodal values       N       Mm       Mm         Partition values       Vy / Vu       My / Mu       Mu       Mu         Extreme values of cross-sections       ✓       Mu       ✓       Mu         Contents       ✓       OK       Cancel       ✓       ✓											
All         Details - Internal Forces         Nodal values         N         Patition values         Vy / Vu         My / Mu         Extreme values of cross-sections         Cortents         Info pictures             OK					Ig						
Image: Sections - Internal Forces       Image: Sections - Internal Forces         Image: All Surfaces - Local Deformations       Image: All Surfaces - Global Deformations         Image: All Surfaces - Global Deformations       Image: All Surfaces - Global Deformations         Image: All Surfaces - Global Deformations       Image: All Surfaces - Global Deformations         Image: All Surfaces - Global Deformations       Image: All Surfaces - Global Deformations         Image: All Surfaces - Global Deformations       Image: All Surfaces - Global Deformations         Image: All Surfaces - Global Deformations       Image: All Surfaces - Global Deformations         Image: All Surfaces - Global Deformations       Image: All Surfaces - Global Deformations         Image: All Surfaces - Global Deformations       Image: All Surfaces - Global Deformations         Image: All Surfaces - Global Deformations       Image: All Surfaces - Global Deformations         Image: All Surfaces - Global Deformations       Image: All Surfaces - Global Deformations         Image: All Surfaces - Global Deformations       Image: All Surfaces - Global Deformations         Image: All Surfaces - Global Deformations       Image: All Surfaces - Global Deformations         Image: All Surfaces - Global Deformations       Image: All Surfaces - Global Deformations         Image: All Surfaces - Global Deformations       Image: All Surfaces - Global Deformations         Image: All Surfaces - Global Deformat											
All         4.13 Surfaces - Local Deformations         All         4.14 Surfaces - Global Deformations         Details - Internal Forces by Cross-Section         Display         Display Max/Min Internal Forces         Pattion values         Pattion values         Vy / Vu       My / Mu         Extreme values of cross-sections         Cortents         Info pictures					s is						
All         Details - Internal Forces by Cross-Section         Display         Display         Display Max/Min Internal Forces         Nodal values         Partition values         Dy/Vu         Max/Min Internal Forces         Owner walkes         Vy/Vu         Max/Min Internal Forces         Owner walkes         Vy/Vu         War/Mu         Extreme values of cross-sections         Contents         Info pictures											
Details - Internal Forces by Cross-Section       X         Display       Display Max/Min Internal Forces         N       MT         Pattion values       Vy / Vu         Display       Vy / Vu         Display Max/Min Internal Forces         Oracle View       Vy / Vu         Display Max/Min Internal Forces         Pattion values       Vy / Vu         Display Max/Min Internal Forces         Display Max/Min Internal Forces         Pattion values         Vy / Vu       My / Mu         Extreme values of cross-sections         Ororss-sections         OK       Cancel											
Details - Internal Forces by Cross-Section × Display Display Max/Min Internal Forces Nodal values N M MT Patition values Vy / Vu My / Mu Extreme values Vy / Vu My / Mu Extreme values of cross-sections Vy / Vu Mz / Mv Model values of cross-sections Vy / Vu Mz / Mv Model values Vy / Vu Mz / Mv Mz / Mv / Mv Mz / Mv / Mv Mz / Mv /					/						
Indal values       N       Imm         Imm       Pattion values       Vy/Vu       My/Mu         Imm       Imm       Imm         Imm       Imm       Imm <t< td=""><td>[</td><td></td><td></td><td></td><td>*</td><td></td><td></td><td></td><td></td><td></td><td></td></t<>	[				*						
Image: Partition values       Image: Wy/Vu       Image: Wy/Vu <td></td> <td>Display</td> <td></td> <td>Display Max/N</td> <td>In Internal Forces</td> <td></td> <td></td> <td></td> <td>_</td> <td></td> <td></td>		Display		Display Max/N	In Internal Forces				_		
Image: Partition values       Image: Wy/Vu       Image: Wy/Vu <td></td> <td>Nodal va</td> <td>alues</td> <td></td> <td>MR</td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td>		Nodal va	alues		MR						
Betreme values       □ y/v o □ My/M2         Betreme values of cross-sections       □ Vz / Vv □ Mz / Mv         Cover sheet       □         Contents       □         Info pictures       OK Cancel											
Isplay     Image: Contents     Image: C											
Cross-sections		Extreme	values	Vz/Vv	Mz/Mv						
Display  Cover sheet  Contents  More sheet											
Cover sheet     Image: Cover sheet       Contents     Image: Cover sheet       If opictures     Image: Cover sheet		cross-se	ctions								
Cover sheet     Image: Cover sheet       Contents     Image: Cover sheet       If opictures     Image: Cover sheet											
Contents Diffo pictures OK Cancel	Display										
Contents Diffo pictures OK Cancel	Cover sheet			J							
Info pictures		~									
KI Show corresponding load cases		2			OK		Car	ncel	_		
✓ Uppercase titles									w corresponding	load cases	
	✓ Uppercase titles										

⊿<u>⊳</u> Dlubal We click into the Cross-Sections - Internal Forces row. The Details - Internal Forces by Cross-Section dialog box. We reduce the output to the **Extreme values** of cross-sections N, V<sub>z</sub>, M<sub>y</sub>, and M<sub>z</sub>.

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

		tings Inse													
8	<u>&gt; الکا (</u>		¤	è 🕹 🎸 -	8	🛛 🖹 🛛		ଛି 🔶 👶 🙋	2						
	4 12	CROSS	SECTIC	DNS - INT		FOR	CES								
	Member		Node	Location			020	Forces [kN]			Moments	[kNm]		Result Combine Correspondi	
	No.	RC	No.	x [m]		N		V <sub>v</sub> .	Vz	Μτ	. M <sub>v</sub>	[]	Mz	Load Case	-
			n No. 1: Circle					• •	•2					2000 0000	
	2	RC1		4.000	MAX N	₽ .	105.55	-11.64	-1.75	0.0	0	-7.34	21.14	CO 9	_
	1	RC1	13	0.000	MIN N	₽ -	175.10	6.22	-3.14	0.0		0.00	0.00	CO 5	_
	1	RC1	2	4.000	MAX Vz		124.89	-3.23 Þ	-1.42	0.0		-5.98	-8.37	CO 15	
	1	RC1	13	0.000	MIN Vz		162.88	4.99 ⊳	-3.69	0.0		0.00	0.00	CO 2	_
	1	RC1 RC1	13	0.000	MAX My MIN My		121.14	3.26	-2.25	0.0		0.00	0.00	CO1 CO2	
	2	RC1	1	4.000	MAX Mz		144.13	-13.51	-2.25	0.0		-9.57 >	26.33		
	1	RC1	2	4.000	MIN Mz		163.80	4.50	-2.23	0.0		13.17	-19.32		
			n No. 2: Rectar						-0.07	0.0					-
	3	RC1		3.000	MAX N	Þ	231.41	5.79	5.18	-1.5	3 2	07.08	0.29	CO 5	
		RC1	3	0.000	MIN N		-41.70	-13.31	148.00	-5.6		38.91	13.69		
	3	RC1		0.125	MAX Vz		-23.63	146.72 >	232.33	-26.4		32.75	6.48		_
	3	RC1 RC1		5.875 3.000	MIN Vz MAX My		-31.03 231.41	-128.36 ► 5.79	-224.58 5.18	31.3		21.79 07.08	10.14	CO3 CO5	
	3	RC1	3	0.000	MAX My MIN My		-39.41	-13.67	5.18	-1.5		39.35	13.31		
	3	RC1	3	0.000	MAX M <sub>z</sub>		-41.70	-13.07	148.00	-5.6		38.91 >	13.69		
	3	RC1	5	0.500	MIN Mz		10.53	46.89	175.80	-28.4		32.06	-7.59		
			No 3: HEA:	300 I DIN 1025-			10.00	40.00	110.00	-20.4		02.00 p	-1.55	001	_
	8	RC1	20	6.059	MAX N	Þ	-1.55	0.00	-3.59	0.0		0.00	0.00	CO9	_
	4	RC1	1	0.000	MIN N	Þ	-40.52	1.12 -0.16 ⊳	-32.26	0.0		47.76	-2.31	CO 12	_
	7	RC1	4	0.000	MAX Vz		-39.44		31.33	-0.0		51.81	-3.91	CO 4	
	4	RC1	1	0.000	MIN Vz		-40.35	-1.88 >	-32.67 -32.67	0.1		52.19 52.22	-9.13		
	7	RC1 RC1	4	0.000	MAX My MIN My		-40.35	-2.12	-32.67	0.1		52.22	-9.86 -3.16	CO4 CO5	
	6	RC1	2	0.000	MAX M <sub>z</sub>		-39.42	9.76	-25.86	-0.0		47.90 ⊳		CO 13	
	4	RC1	1	0.000	MIN M <sub>2</sub>		-29.80	-6.07	-26.17	-0.0		49.59	-18.36		
	-			260   DIN 1025-			-20.00	-0.07	-20.11	0.0		10.00	-10.00	001	_
	11	RC1		6.221	MAX N		-12.41	-2.24	-18.73	0.0		10.88	-1.36		_
	10	RC1		3.350	MIN N		-30.66	0.00	-0.05	0.0		21.56	0.00		
	10	RC1	15	0.000	MAX Vz		-22.21	3.59 ⊳	32.19	-0.0		30.85	-0.42	CO 15	
	11	RC1	20	6.700	MIN Vz		-18.76	-2.34 >	-31.99	0.0		30.13	-0.24		
	11 10	RC1 RC1	15	3.350	MAX My MIN My		-15.81 -26.40	0.00	0.04 32.15	-0.0		24.67 31.67	-5.39	CO 14 CO 13	
	10	RC1	15	0.000	MAX M <sub>2</sub>		-26.40	0.06	32.15	-0.0		29.87	-0.42	CO13 CO3	
	10	RC1	10	3.350	MIN Mz		-26.42	0.00	0.22	-0.0		22.59	-8.29		
	10		n No. 5: L 80x8		11114 1112		-20.42	0.00	0.22	-0.0	<u> </u>	~~~~ V	-029	0013	_
	12	RC1	15	0.000	MAX N	⊳	0.00	0.00	0.00	0.0		0.00	0.00	CO1	
	12	RC1	15	0.000	MIN N	Þ	0.00	0.00	0.00	0.0		0.00	0.00		
	12	RC1	15	0.000	MAX Vz		0.00	0.00 Þ	0.00	0.0		0.00		CO 1	
	12	RC1	15	0.000	MIN Vz	1	0.00	0.00 Þ	0.00	0.0		0.00	0.00	CO1	
	12	RC1	15	0.000	MAX My		0.00	0.00	0.00		9 0	0.00	0.00	CO1	
	12	RC1	15	0.000	MIN My		0.00	0.00	0.00	0.0	0 0	0.00	0.00	CO1	
					and y al.										

Figure 9.5 Extreme values of cross-section internal forces N, Vz, My, and Mz for RC1 in the printout report

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

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



9.3

x

17

## **Printing Graphics in the Printout Report**

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

### **Printing deformation graphics**

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

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

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

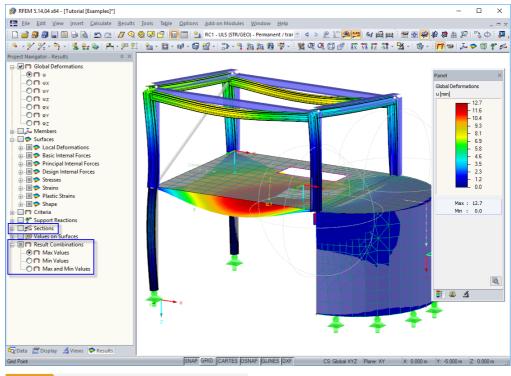


Figure 9.6 Displaying the maximum deformation results of RC1

We now transfer the graphic to the printout report using

#### File $\rightarrow$ Print Graphic

or the corresponding toolbar button.

The Graphic Printout dialog box appears (see Figure 9.7 2).

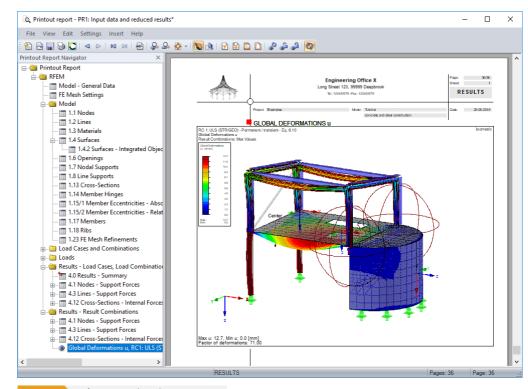




-								
General	Options	Color Scale	Factors	Borde	r and Stretch Factors			
Graphic	Picture				Window To Print		Graphic Scale	
ODirec	tly to a pri	inter	1		Current only		O As screen view	
🖲 To a	printout re	eport:	PR1: I	$r \sim$	More		Window filling	
⊖ To th	ne Clipboa	rd			Mass print	2	○ To scale 1: 100	$\sim$
_ To 3	D PDF						_	
Graphic	Picture Si	ze and Rotatio	n		Options			
Use 🗸	full page v	vidth			Show results for se diagram	elected x	location in result	
O Use	full page h	eight			Lock graphic pictu	ire (with	out update)	
Heig	ht:	60 ≑ [% (	of page]					
	_				Show printout repo	ort on [O	K]	
Rotation	: Е	0 ≑ [°]						
	of Graphic							
Global	Deformatio	ins u						
<b>A</b>								
2							OK (	Cancel

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

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







#### **Printing result diagrams**

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

In the RFEM work window, we right-click member 11 (purlin on high eave). In the member shortcut menu (see Chapter 8.4 12), we select the *Result Diagrams* option to access the result diagram.

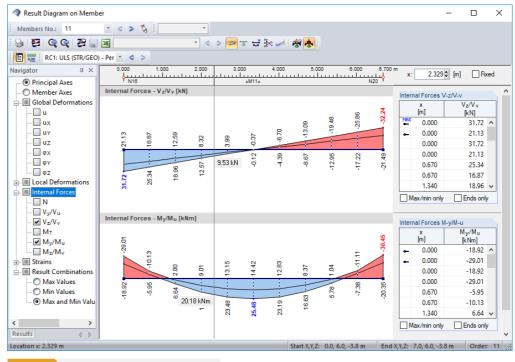


Figure 9.9 Shear force and moment diagram of purlin

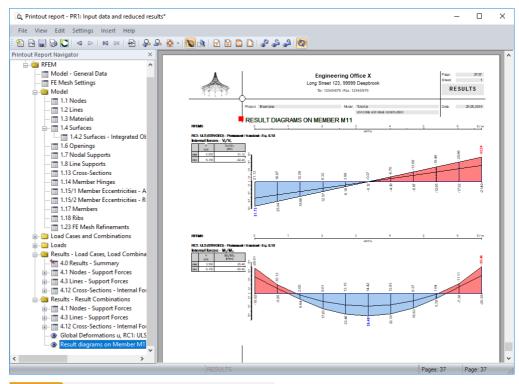
The window shows the result diagrams of RC1. For the printout, we select only the internal forces  $V_z/V_v$  and  $M_y/M_u$ . The result diagram shows the Max and Min Values.

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

raphic Printout		
General Options Color Scale Fa	ctors Border and Stretch Factors	3
Script	Symbols	Frame
O Proportional	Proportional	None
<ul> <li>Constant</li> </ul>	◯ Constant	O Framed
Factor: 1 🜲	Factor:	Title box
Print Quality	Color	
Standard (max 1000 x 1000 pixels)	s) 🔾 G	irayscale
O Maximum (max 5000 x 5000 pixel	s) O Te	exts and lines in black
O User-defined	() A	Il colored
Max number of pixels:	1000 ≑	
D		OK Cancel





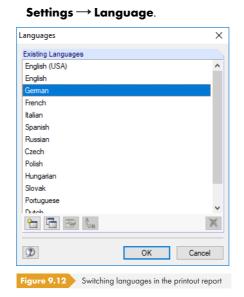


We transfer the graphic to the printout report with [OK].

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

#### Changing the language of the printout report

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



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

Custom entries such as load case descriptions or comments are not translated.



#### Printing the printout report

Save

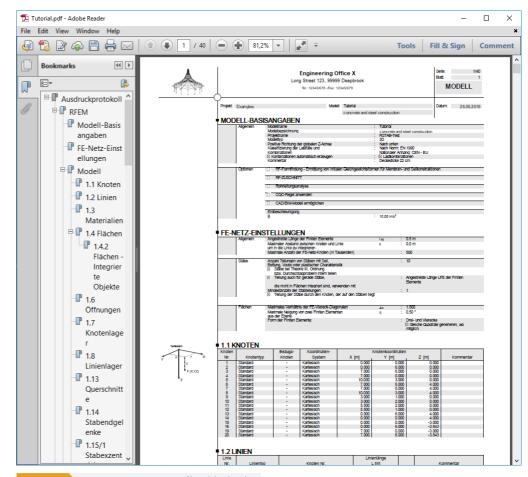
When the printout report is completely prepared, we can send it to the printer by using the [Print] button.

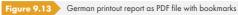
The integrated PDF printer also allows the report data to be exported as a PDF file. We use this feature through the printout report menu

#### File $\rightarrow$ Export to PDF.

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

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







# 10 Outlook

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

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

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

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

