Version October 2015 (US)

Program



Spatial Models Calculated acc. to Finite Element Method

Introductory Example

All rights, including those of translations, are reserved.

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

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

Tel.: (267) 702-2815 E-mail: info-us@dlubal.com Web: www.dlubal.com

Contents

| | Contents | Page | | Contents | Page |
|---------|--|---------|-------|--|------|
| 1. | Introduction | 4 | 5.2 | Load Case 2: Live Load, Area 1 | 28 |
| 2. | System and Loads | 5 | 5.3 | Load Case 3: Live Load, Area 2 | 30 |
| 2.1 | Sketch of System | 5 | 5.3.1 | Surface Load | 30 |
| 2.2 | Materials. Thicknesses and Cross- | - | 5.3.2 | Line Load | 31 |
| | Sections | 5 | 5.4 | Load Case 4: Wind | 32 |
| 2.3 | Load | 6 | 5.5 | Checking Load Cases | 34 |
| 3. | Creation of Model | 7 | 6. | Combination of Load Cases | 35 |
| 3.1 | Starting RFEM | 7 | 6.1 | Creating Load Combinations | 35 |
| 3.2 | Creating the Model | 7 | 6.2 | Creating Result Combinations | 38 |
| 4. | Model Data | 8 | 7. | Calculation | 39 |
| 4.1 | Adjusting Work Window and Grid | 8 | 7.1 | Checking Input Data | 39 |
| 4.2 | Creating Surfaces | 10 | 7.2 | Generating the FE Mesh | 40 |
| 4.2.1 | First Rectangular Surface | 11 | 7.3 | Calculating the Model | 40 |
| 4.2.2 | Second Rectangular Surface | 12 | 8. | Results | 41 |
| 4.3 | Creating Members | 13 | 8.1 | Graphical Results | 41 |
| 4.3.1 | Downstand Beams | 13 | 8.2 | Results Tables | 43 |
| 4.3.1.1 | Steel Girder | 13 | 8.3 | Filter Results | 45 |
| 4.3.1.2 | T-Beams | 15 | 8.3.1 | Visibilities | 45 |
| 4.3.2 | Columns | 17 | 8.3.2 | Results on Objects | 47 |
| 4.4 | Support Arrangement | 21 | 8.4 | Display of Result Diagrams | 48 |
| 4.5 | Connecting Member with Hinge and Eccentricity | d 23 | 9. | Documentation | 49 |
| 4.5.1 | Hinge | 23 | 9.1 | Creation of Printout Report | 49 |
| 4.5.2 | Member Eccentricity | 24 | 9.2 | Adjusting the Printout Report | 50 |
| 4.6 | Checking the Input | 25 | 9.3 | Inserting Graphics in Printout Report | 51 |
| 5. | Loads | 26 | 10 | Outlook | 54 |
| 5.1 | Load Case 1: Self-Weight and Finishes | 26 | 10. | Outook | 54 |
| 5.1.1 | Self-Weight | 27 | | | |
| 5.1.2 | Floor Structure | 27 | | | |
| | | | | | |



1. Introduction

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

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

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

5

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

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

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



2. System and Loads

2.1 Sketch of System

Figure 2.1: Structural system

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

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

2.2 Materials, Thicknesses and Cross-Sections

We use concrete f'c = 4000 psi and steel A992 as materials.

The floor thickness is 8 in. The concrete columns and the downstand beam consist of square cross-sections with a lateral lengths of 12 in. For the steel beam we use a W 18x50 section.



2.3 Load

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

As loads, the self-weight of the model including its floor structure of 15 psf is applied. We do not need to determine the self-weight manually. RFEM calculates the weight automatically from the defined materials, surface thicknesses and cross-sections.

Load case 2: live load, field 1

The floor surface represents a domestic area with a live load of 30 psf. The load is applied in two different load cases to cover the effects of continuity.

Load case 3: live load, field 2

The live load of 30 psf is also applied to the second field. In addition, a vertically acting linear load of 350 lbf/ft is taken into account on the edge of the floor, representing a loading due to a balcony construction.

Load case 4: wind in -Y

For the wind loads of the walls (not contained in our model), we have to allocate the surface loads to the lines of the floor and ceiling slabs. The forces acting on the floor shall not be taken into account in this example.

The line loads for the ceiling are:

 $\frac{1}{2} \cdot 10 \text{ psf} \cdot 13 \text{ ft} = 65 \text{ lbf/ft} \text{ (for line 4)}$

 $\frac{1}{2} \cdot 10 \text{ psf} \cdot 10 \text{ ft} = 50 \text{ lbf/ft} \text{ (for line 8)}$

 $\frac{1}{2} \cdot 8 \text{ psf} \cdot 13 \text{ ft} = 52 \text{ lbf/ft} \text{ (for line 2)}$

 $\frac{1}{2} \cdot 8 \text{ psf} \cdot 10 \text{ ft} = 40 \text{ lbf/ft} \text{ (for line 6)}$



3. Creation of Model

3.1 Starting RFEM

To start RFEM in the taskbar, we click **Start**, point to **All Programs** and **Dlubal**, and then we select **Dlubal RFEM 5.xx**

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

3.2 Creating the Model

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

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

| | New Mo | del - General Data | E × |
|--|----------------------|---|---|
| General Options History | | | |
| Model <u>N</u> ame | Description | | |
| Introductory Example | Floor Slab or | n Columns | |
| Project Name | Descripti <u>o</u> n | | |
| Dlubal Examples | Sample strue | ctures | |
| Folder: | | | 9 |
| C: \Users\Public\Documents\Dlubal\P | rojects\Exampl | es | |
| Type of Model | | Classification of Load Cases and Combinations | |
| ● 3D 2D - ½Y (uz/ex/ey) 2D - XZ (ux/uz/ey) 2D - XI (ux/uy/ez) | | According to Standard: None Create combinations automatically Load combinations Result combinations (for linear analysis only) | Ŷ |
| Positive Orientation of Global Z-Axis | | Template | |
| Upward Downward | | Open template model: US Template | Image: A start of the start of |
| Comment | | | v 🖻 |
| ۵ ک 🐱 | | ОК | Cancel |

Figure 3.1: Dialog box New Model - General Data

0

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

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

In the dialog section *Type of Model*, the **3D** option is preset. This setting enables spatial modeling. We check whether the *Positive Orientation of Global Axis Z* and the *Orientation of Local z-Axis* go **Upward.**

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

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





4. Model Data

4.1 Adjusting Work Window and Grid

The empty work window of RFEM is displayed.

View

2

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

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

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

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

To exit the function, different ways are possible:

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

Mouse functions

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

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

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



Grid

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

| | Work Plane a | and Grid/Snap 🖙 💌 |
|--|------------------|---|
| Work Plane Coordinate system: I Giobal XYZ XYZ XYZ XZ XZ | - 1 | Origin of Work Plane Node No.: ▼ ▼ ● Coordinates X: 0.00 ÷ [ft] Y: 0.00 ÷ [ft] Z: 0.00 ÷ [ft] |
| Grid/Snap Object Snap Guide | lines Background | Layers Line Grids |
| Grid | Cartesian | Dynamically according to size of model |
| ♥ Snap Distance: 10 ♥ [px] | O Polar | Direction $1: 40 \div 40 \div 2: 40 \div 40 \div$ |
| | | Grid Point Spacing |
| 2 • 4 | I R v | Space b: 1.00 ⊕ i [ft] h: 1.00 ⊕ i [ft] |
| nen tan o | b.b.l | Rotation β: 0.00 (*) |
| 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 | nib | Grid Line Spacing |
| inib. | | Number n1: 10 ♠ n2: 10 ♠ |
| 2 | | OK Cancel |

Figure 4.1: Dialog box Work Plane and Grid/Snap

SNAP GRID

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

Work plane

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

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





4.2 Creating Surfaces

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

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

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

Show Numbering

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

| | Repeat - Isometric View Enter | |
|--------------|-----------------------------------|---|
| | View | Þ |
| | User-Defined View | Þ |
| 123 | Show Numbering | |
| P | Show Loads | |
| 2 | Show Results | |
| ¥, | Show Dimensions | |
| ∠^^ | Show Comments | |
| \checkmark | Show Hidden Objects in Background | |
| v | Display Model Wired or Solid | |
| † ° | Lock Guidelines | |
| \checkmark | Lock Line Grids | |
| | Enable Drag & Drop | |
| ۲ | Auto Rotate Model | |
| ø | Auto Connect Lines/Members | |
| æ | Coordinate System | |
| 1 | Work Plane, Grid/Snap, | |
| 2 | Select Special | |
| | Colors in Graphics According to | Þ |
| A D | Display Properties | |

Figure 4.2: Show numbering in shortcut menu

Auto Connect Lines/Members

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





List button for plane surfaces

4.2.1 First Rectangular Surface

To create rectangular plates quickly,

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

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

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

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

| | New Rectangu | lar Surfac | e | | e | × |
|-------------------------------------|--------------|-------------------------|----------------|------|-------|----|
| Surface No. | | Surface Ty Geometry: | rpe | | ~ | |
| Material | | Stiffness: | Standard | | ~ | 1 |
| 1 Concrete fc = 4000 psi ACI 318-14 | | Surface th | ickness 'Const | ant' | | |
| Thickness | | | | | | |
| Constant Thickness d: | ۵. | | | | | |
| Comment | v 🖻 | | | | | |
| | | | | ОК | Cance | el |

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

The *Material* is preset as *Concrete* f'c = 4000 psi according to ACI 318-14. When we want to use a different material, we can select another one using the [Material Library] button.

The Thickness of the surface is Constant. We leave the value d by 8 in.

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

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

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



*

Figure 4.3: Dialog box New Rectangular Surface



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

Then, we define the opposite corner of the slab by clicking the grid point with the X/Y/Z coordinates **20.00/-16.00/0.00**.

| | ľ | • | • | • | • | • | • | • | • | • | İ | • | • | • | • | | • | • | • | • | | | • | • |
|---|-------|-----|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|----------|-----|------|-----|
| | | • × | - | - | • | - | - | - | | | | - | - | - | - | - | - | - | - | | - | | • | • |
| | | | | | • | • | • | • | • | • | | | • | • | • | - | | | - | | • | | • | • |
| | | | | | • | • | • | | • | • | | | • | | | - | | | | | • | | • | • |
| | • • | | | | • | • | • | • | · | • | | | • | | | - | | | | | • | | | • |
| | | | | | | | | | • | | | | | | | | | | | | | | | • |
| | | | | | | | | | | | | | | | | | | | | | | | | • |
| | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | • |
| | 1 | | | | | | | | | | | | | | | | | | | | | | | • |
| | | | | | | | | | | | | | | | | | | | | | | | | • |
| | | | | | | • | | | • | | | | | | | | | | | | • | | | • |
| | | | | | • | • | • | • | • | • | | | • | | | | | | | | | | • | • |
| | | | | | • | | • | • | • | • | | | • | | | | | | | | X: Y: | -16 | .00 | ft |
| - | · · · | | | | | | | | | | 8 | | | | | | | | | F | 2: | 0. | . 00 | ft. |

Figure 4.4: Rectangular surface 1

RFEM creates four nodes, four lines and one surface.

4.2.2 Second Rectangular Surface

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

We click node **4** with the coordinates **20.00/0.00/0.00**, and then we select the grid point with the coordinates **33.00/-26.00/0.00**.

| | | · | • | | • | • | • | | | | • | | | | • | • | • | | | • | | | • | • | • | | • | | | • | |
|--|---|---|---|---|---|---|---|---|---|---|---|-----|---|---|---|---|---|---|------|---|---|-----|---|----|---|-----|---|----------|----------|--------------|-----|
| | - | | | | ÷ | | | | | | - | | | | | | ÷ | | | • | ÷ | | | | | | | | ÷ | | |
| | 1 | | : | ÷ | : | : | : | : | : | : | | · · | | | ÷ | ÷ | : | : | | ÷ | : | | | ÷. | : | : : | ÷ | 1 | : | : | · · |
| | | • | • | : | • | • | • | : | • | • | | • • | | · | • | • | : | • | · | : | • | • • | • | ł | • | | : | : | · | · | ••• |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | · | • • |
| | | | | | : | | | | | : | | • • | | | ÷ | ÷ | | | | ÷ | | | | ÷ | | | | : | : | ÷ | |
| | | ÷ | : | : | : | : | : | : | : | : | | • • | | : | ÷ | ÷ | : | : | : | ÷ | : | • • | ÷ | ÷ | : | | ÷ | : | : | ÷ | |
| | | ÷ | | : | : | : | : | : | : | : | | | | : | | : | : | : | | : | : | | : | : | : | | | : | : | ÷ | : : |
| | | | | | | • | • | | | | | | | | | | | | | | | | | | | | · | . | | • | |
| | | | | | | | | | | | | | | | | ÷ | | | | | | | | ÷ | | | | : | : | | |
| | 1 | | | • | • | • | | | • | • | - | · · | | | | • | • | • | 1 | ÷ | : | | ÷ | ĵ, | : | | ÷ | : | : | ÷ | · · |
| | | : | : | ÷ | ÷ | : | : | : | : | : | 1 | · · | : | : | ÷ | ÷ | : | : | : | ÷ | : | | ÷ | ÷ | : | | ÷ | : | ÷ | ÷ | · · |
| 21: 33:00:26: 32: 33:00:26: 32: 32: 33:00:26: 32: 32: 33:00:26: 32: 32: 33:00:26: 32: 32: 33:00:26: 32: 32: 32: 32: 32: 32: 32: 32: 32: 32 | | · | · | ÷ | · | ÷ | · | · | · | ÷ | | • • | • | · | · | · | · | · | · · | ÷ | : | • • | ÷ | ÷ | • | | ÷ | | ÷ | · | ••• |
| X: 133100161 | | • | • | | • | • | • | | | • | | | | | • | • | • | | | | | | | | | | | | | | |
| ±±±±±±±±±±±±±±±±±±±±±±±±±±±±±±±±±±±±±± | | : | : | : | : | : | : | | : | : | | | | : | : | : | : | : | | | | | | ÷ | | | | : | : | : | |
| $\chi_{1}^{2} = 26.00 \text{ ft}$ | | : | : | : | : | : | : | : | : | : | 1 | · · | : | : | : | : | : | : | : | ÷ | : | | ÷ | ÷ | : | | ÷ | X: Y: | •3 -2 | 3:00 6,00 | ft |

Figure 4.5: Rectangular surface 2

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



4.3 Creating Members

4.3.1 Downstand Beams

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

4.3.1.1 Steel Girder

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

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

| | Edit Line | E × |
|-----------------------------|--|--|
| General Member S | upport FE Mesh Rotation | |
| Line No. 7 Member Available | | Ž Ž X X X X X X X X X X X X X X X X X X |
| | New Me | ember 🕒 🔀 🗙 |
| General Option | ns Effective Lengths Modify Stiffness | |
| Member No. | Line No. | Member Type |
| 1 | 7 | Beam V |
| Node No. | | Member Rotation |
| 6,7 | | Z Y X End |
| Member Rotatio | n via β: 0.00 € ['] No: Inside V & M | P (X,Y,Z) Help node in plane x-y |
| In plane: | ● x-y ○ x-z | $z' \nabla z \beta < 0$ |
| Cross-Section | | |
| Member start: | Create a new cross-section! | |
| Member end: | As member start | Import from Cross-Section Library |
| Member Hinge | | |
| Member start: | None | * * |
| Member end: | None | |
| 2 0.00 | 6 | OK Cancel |

Figure 4.6: Dialog box New Member

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

The Cross-Section Library dialog box appears (see Figure 4.7).

I

| | Cross-Sec | tion Library | E X |
|----------|----------------------------------|----------------------|----------------------------|
| Rolled | Parametric - Thin-Walled | Parametric - Massive | Parametric - Timber |
| ILTL | IIIT | ITII | |
| 004 | T L L D | | III III 0 00 |
| l 🔹 🕶 🖆 | | | ттпп |
| | Ο Δ Π Π | TLIJ | тттт |
| Built-up | ΠΠΠ | π π • | |
| II I T T | Ξ , Ξ , + • | | II II V |
| TIII | - 1 l J | 00 000 | |
| IIII | TCCC | | Standardized - Timber |
| •• T | ΣΟΦΟ | | I 0 |
| | | User-Defined | From Cross-Section Program |
| | | | |
| D 🕅 | | | Cancel |

Figure 4.7: Cross-section Library

We click the button for I sections in the upper left part of the dialog box. The *Rolled Cross-Sections - I-Sections* dialog box opens where we can select the section **W 18x50** from the W cross-section table (see Figure 4.8).

For rolled cross-sections, RFEM automatically sets the Material number 2 - Steel A992.



Figure 4.8: Dialog box Rolled Cross-Sections - I-Sections

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



4.3.1.2 T-Beams

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

Definition of cross-section

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

In the *Parametric - Massive* section of the library, we select the **Rectangle** button. The *Solid Cross-Sections - Rectangle* dialog box opens where we define the width *b* and the depth *h* as **12** in.



Figure 4.9: Dialog box Solid Cross-Sections - Rectangle



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

For solid cross-sections RFEM automatically sets the Material number 1 - Concrete f'c = 4000 psi.

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



Definition of rib

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

| | New Member | 🖙 🛛 🗙 |
|---------------------------|--|-------------------------------------|
| General Option | s Effective Lengths Modify Stiffness | |
| Member No. | Line No. Member Ty | pe |
| 2 | 3 Beam | ▼ 🗟 |
| | Beam | |
| Node No. | Rigid | |
| 3,4 | | |
| | Truss | (only N) |
| | Tensia | n l |
| Member Rotatio | n via Compr | ession , |
| Angle | β: 0.00 € [°] Bucklin | ng |
| | | en Bulleun |
| Help node | No.: Inside V R H | Beam |
| In plane: | x-y Defina | ble Stiffness |
| | 🔿 x-z 🕤 🗖 Couplin | ng Rigid-Rigid |
| | Couplin | ng Rigid-Hinge |
| Cross-Section | | ng Hinge-Hinge |
| Member start: | 2 Rectangle 12/12 Concrete f'c = 40 Spring | ig hinge-higid 🔯 📵 |
| Member end: | As member start | 2 0 |
| Member Hinge | | |
| Member start: | None | Y 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 |
| Member end: | None | * |
| 2 0.00 | ġ | OK Cancel |

Figure 4.10: Changing the member type

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

| | New | / Rib | 🖻 🗙 |
|---|---------------|--|--------|
| Member No. 2 Position and Alignment On ±z side of surface On jzside of surface Centric User defined via member eccentricity | | | |
| Integration Width - Side 1 Connecting surface No.: Autodetect Width b ₁ : OL/6 ©L/8 OC C Ift] | ✓ 13 | Integration Width - Side 2 Cognecting surface No.: Autodetect Width b ₂ : OL/6 © [_/2] O | × 23 |
| Bib Alignment ✔ Align local z-axis parallel to local z-axis <u>Comment</u> | is of surface | 3 | v @ |
| | | Oł | Cancel |

Figure 4.11: Defining the rib

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

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

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





Changing the view



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

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



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

Checking data in navigator and tables

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

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

4.3.2 Columns

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

Node selection

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

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

or we use the toolbar button shown on the left.

The Special Selection dialog box has Nodes as the category by default (see Figure 4.13).

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



| | Special Selection | E |
|-------------------------------|-----------------------|-----------------------------|
| odel Data | | |
| Category | Nodes | |
| Activate Category | All With number: | A |
| ✓ Nodes | | |
| | | |
| Materials | | |
| Surfaces | Ref. node No.: | |
| Cross-Sections | | * |
| Members | | E. |
| Ribs | | |
| | With comment: | |
| | All | |
| | | · 🖼 3 |
| | | |
| | With support: | |
| | All | v 15 |
| | | |
| | | |
| | With mesh refinement: | |
| | All | ✓ [™] ₂ |
| | | |
| | Co line: | |
| | On line: | |
| | | To a |
| | | |
| | On surface: | |
| N 1 | | |
| Status | | 3 |
| Add | () <u>O</u> r | |
| Select from current selection | On solid: | |
| Remove from current selection | | * |
| | | 73 ~ |
| | | |

Figure 4.13: Dialog box Special Selection

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

Copying nodes

6°

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

| Move or C | Сору - Сору 🖙 💌 |
|--|--|
| Number Number of copies n: 1÷ | Z Y |
| Reference to Coordinate System | X dy |
| Global X.Y.Z CS User-defined U.V.W CS Displacement Vector dx: 0.00 + [t] | d2 ^{dy} |
| dy: 0.00 € ▶ [ft] dz: -10.00 € ▶ [ft] | Numbering Incompatifier |
| | Nodes: 1 Image: Continuous Members: 1 Image: Continuous Lines: 1 Image: Continuous Surfaces: 1 Image: Continuous Solids: 1 Image: Continuous |
| | OK Cancel |

Figure 4.14: Dialog box Move or Copy

We increase the *Number of copies* to **1**: With this setting the nodes won't be moved but copied. As the columns are 10 ft high, we enter the value **-10.0** ft for the *Displacement Vector* in d_z . a

ı Dlubal

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

| Detail Settings for M | love/Rotate/Mirror 🖙 🗙 |
|--|--|
| Connecting Lines between nodes Vireate new lines between the selected nodes | Copied Surfaces |
| Members between nodes Verate new members between the selected hodes and their copies Template member No.: None Sur 1 Beam 1 - W 18x50 Steel A992 2 Rib 2 - Rectangle 12/12 Concrete f'c = 4000 ps artig their copies Template surface No.: | Lo cobh |
| None Image: Constraint of the selected surfaces Create new solid bodies between the selected surfaces and their copies Template solid No.: None | Local Coordinate Systems Adjust local coordinate system when rotating or mirroring for ✓ Lines ✓ Members |
| When rotating create between selected nodes and their copies: Straight lines Arc lines | Load Cases Copy including loading Agjust nodal loads when rotating or mirroring |
| Duplicity Allo <u>w</u> double members | Auto Connect Connect lines/members if they contact |
| ٦ | OK Cancel |

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

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

- ☑ Create new lines between the selected nodes and their copies
- $\ensuremath{\ensuremath{\boxtimes}}$ Create new members between the selected nodes and their copies

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

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

Editing surfaces

VI

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

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

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



Figure 4.16: Selecting with window

4 Model Data



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

| | Edit Me | ember | 🖙 🗙 |
|---|---------------------------------------|---|--------|
| General Option | ns Effective Lengths Modify Stiffness | | |
| Member <u>N</u> o. | Line No. | Member Type | |
| 3-9 | 5,10-15 | Rib | ✓ |
| Ngde No. 1.8: 2.9: 3.10: Member Rotatio ● Angle → Help node | 4,11; 5,12; 6,13; 7,14 | Beam Rigid Rib Truss Truss Truss Truss Compression Buckling Cable Cable on Pulleys | , |
| In plane: Cross-Section | © KY | Result Beam Definable Stiffness Coupling Rigid-Rigid Coupling Rigid-Hinge Coupling Hinge-Hinge | |
| Member start: | 2 Rectangle 12/12 Concrete f'c = | | 🔤 🕤 |
| Member end: | As member start | Nul | 20 |
| Member Hinge | | | |
| Mem <u>b</u> er start: | None | Y 1 Sec. Sec. | 2 |
| Member en <u>d</u> : | None | v 🎦 🛛 | 2 |
| 2 | 2 | ОК | Cancel |

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

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



Figure 4.17: Adjusting the member type



4.4 Support Arrangement

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

Assigning nodal supports

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

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

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

The *Edit Node* dialog box opens.

| | Edit Node | 🖙 🗙 |
|--|--------------|----------|
| Node Coordinates Supp | ort FE Mesh | |
| Node No. 8-14 Support Available Type: VVV | | Z Y X |
| | [1] | K Cancel |

Figure 4.19: Dialog box Edit Node, tab Support

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

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

Changing the work plane

We want to correct the length of the two columns on the left to 13 ft. Therefore, we shift the work plane from the horizontal to the vertical plane.

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

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





Adjusting support nodes

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

Now, we shift node 9 with the mouse by **3 ft** to the grid points below. Please take care to select the node and not the member. Again, we can check the node numbers and the coordinates of the pointer in the status bar.

We repeat the same step for node 8.



Figure 4.20: Shifting support node

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



4.5 Connecting Member with Hinge and Eccentricity

4.5.1 Hinge

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

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

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

| Member Hinge | | |
|---------------|--------|--|
| Member start: | None | |
| Member end: | None 🗸 | |

Figure 4.21: Dialog box Edit Member, dialog section Member Release

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

| | New Member H | inge 🖙 | × |
|---------------------|--|---------------------------------------|-----|
| Member Hinge | ₹ <u>N</u> o. | Z Y | |
| Reference Sy | stem | X | |
| Local mem | ber axes x,y,z | MT | |
| O Global X,Y | ,Z | A A A A A A A A A A A A A A A A A A A | |
| O User-defin | ed axis system: | | |
| Rotated | With the second s | V _z M _z | |
| Release Cond | litions | | |
| R <u>e</u> lease | Spring constant | Noninearity | - |
| | | None | |
| uy | | None | |
| L uz | Cuz : | None 🗸 | No. |
| Rele <u>a</u> se | | | |
| 🔲 φχ | C _{@X} : [lbfft/rad] | None 🗸 | 1 |
| 🖌 Фу | C _{φy} : 0.0 争 [bfft/rad] | None v | 1 |
| 🖌 🍳 | C _{φz} : 0.0 ♥ [lbfft/rad] | None 🗸 | 2 |
| <mark>⊒</mark> N Uy | II- Vz II- Vyvvz My Mz I0- My Z Myvz - | | |
| Comment | ~][| a | |
| 2 0.00 | | OK Cance | I |

Figure 4.22: Dialog box New Member Hinge

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

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



4 Model Data

| | Edit M | ember 🖙 | 2 |
|----------------------|---------------------------------------|--|---|
| General Option | ns Effective Lengths Modify Stiffness | | |
| Member <u>N</u> o. | Line No. | Member Type | |
| 1 | 7 🗸 🏅 | 🗖 Beam 🗸 🐷 | |
| N <u>o</u> de No. | | W 18x50 | |
| 6,7 | | | |
| Member Rotatio | n via | | |
| Angle | β: 0.00 €► [°] | », | |
| ◯ <u>H</u> elp node | No.: Inside 🗸 🏷 🛅 | | |
| In plane: | (iii) x-y | | |
| | _ x- <u>z</u> | ļ į | |
| | 0 | | |
| Cross-Section | | | |
| Member start: | I 1 W 18x50 Steel A992 | 🗖 🗸 🔝 🔁 🖉 🤅 |) |
| Member end: | As member start | - 🛍 🔁 🕾 🤅 |) |
| Member Hinge | | | |
| Member start: | 1 Local | | |
| Member en <u>d</u> : | | The second seco | |
| | None | | |
| 2 2 | 2 B3 | OK Cance | |

Figure 4.23: Assigning hinges in the Edit Member dialog box

4.5.2 Member Eccentricity

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

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



Figure 4.24: Dialog box New Member Eccentricity



1

4 Model Data



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

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

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

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



Figure 4.25: Steel girder with release and eccentricity

4.6 Checking the Input

Checking Data navigator and tables

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

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

We can check the entered numerical data quickly.

Saving data

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

we select Save on the File menu

or use the toolbar button shown on the left.



品



5. Loads

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

5.1 Load Case 1: Self-Weight and Finishes

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

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



Figure 5.1: Button New Surface Load

The dialog box Edit Load Cases and Combinations appears.

| | Ed | it Load Cases and Combinations | E 2 |
|--|--------------------------------------|--|------------------|
| Load Cases Load Combinations Result Combinations | | | |
| Existing Load Cases G LC1 Self-weight and finishes | LC <u>N</u> o. | Load Case Description Self-weight and finishes V | To <u>S</u> olve |
| | General Calculati | on Parameters | |
| | Action Category | | |
| | G Permanent | • | |
| | ✓ <u>A</u> ctive | | |
| | Factor in direct | ion: | |
| | <u>X</u> : 0.000 <u>Y</u> : 0.000 | | |
| | <u>Z</u> : -1.000 | ÷ [-] | |
| | | | |
| | | | |
| | | | |
| | | | |
| | | | |
| | | | |
| | | | |
| | | | |
| | Comment | | |
| | | v 🔄 | |
| | L | | |
| 9 | | | OK Cancel |

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

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



5.1.1 Self-Weight

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

5.1.2 Floor Structure

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

| | | | New S | Surface Load | E | × |
|---|-----------------|---|----------------------|--------------|--|---|
| No. | On Surfaces No. | | | E. | Load Type 'Force' Load Distribution 'Uniform' | |
| Load Type | | Load Direction | | | | |
| Force Temperature Axial strain | | Local related to true area: | ⊖x ⊖y ⊖z | | | |
| Precamber Rotary motion Load Distribution | P | Global related to true area: | ⊖XL ⊖YL ⊚ZL | | Barris | |
| Uniform Linear Linear in X Linear in Y Linear in Y Linear in Z | | Global related to projected area: | ○ XP ○ YP ○ ZP | | | |
| O Radial | 1 | | | | Load Direction 'ZL' | |
| Load Magnitude | | | | | t t t t t t t t t t t t t t t t t t t | |
| Node No. 1st: 1 v 2nd: 1 v 3rd: 1 v | Magn ∛ p: | tude 15.000 (psf) (psf) (psf) (psf) | | | ×× | |
| Comment | | | | | | |
| | | | | Y (G | | |
| Ø | | | | | OK Cancel | |

Figure 5.3: Dialog box New Surface Load

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

In the *Load Magnitude* dialog section, we enter a value of -**15 psf** (see Chapter 2.3, page 6). Then, we close the dialog box by clicking [OK].



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



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

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

| Self-Weig | ght |
|-----------|-----------------|
| Active | • |
| Facto | r in direction: |
| X: | 0.000 ≑ |
| Y. | 0.000 🚖 |

Z

-1.000 ≑



Figure 5.4: Graphical input of floor load

5.2 Load Case 2: Live Load, Area 1

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

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

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

| | | ł | Edit Load Cases and Combinations | | |
|---------------|----------------------------------|-----------------|----------------------------------|------------|--------|
| Load Cases | Load Combinations Result Combina | tions | | | |
| Existing Load | Cases | LC No. | Load Case Description | To Solve | |
| G LC1 | Self-weight and finishes | 2 | Live Load, Area 1 | v v | |
| Qi LC2 | Live Load, Area 1 | | | | |
| | | General Calcu | lation Parameters | | |
| | | Action Category | / | | |
| | | Qi Imposed | | × | |
| | | Self-Weight | | | |
| | | <u>A</u> ctive | | | |
| | | Factor in din | ection: | | |
| | | A : | | | |
| | | 7: | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| < | | > Comment | | | |
| | | X | | ¥ 12 | |
| | | | | 01 | Caraci |
| | | | | OK | Cancel |
| | | | | | |

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

For the *LC Description* we enter **Live Load, Area 1**.



The Action Type is set automatically to **Q**_i **Imposed**. This classification is important for the load factors of the load combinations.

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

| | | | New S | Surface Load | e | × |
|---|--|--|---|--------------|--|----|
| <u>N</u> o. | On <u>S</u> urfaces No. 1 | | | \$ | Load Type 'Force' Load Distribution 'Uniform' | |
| Load Type Eorce Iemperature Aydal strain Precamber Rotary motion Load Distribution Unform Unear Uinear in X Uinear in Y | 8 | Load Direction Local related to true area: Global related to true area: Global related to projected area: | ○ x ○ y ○ z ○ XL ○ YL @ ZL ○ XP ○ YP ○ ZP | | | |
| ◯ Linear in <u>Z</u> ◯ Ra <u>d</u> ial | 2 | | | | Load Direction 'ZL' | _ |
| Load Magnitude Node No. 1st: 1 V T 2nd: 1 V T 3rd: 1 V T 3rd: 2 V 3 Comment | <u>Magn</u> ↓ p: 3 ↓ p: 4 ↓ p: 5 ↓ | tude 0.000 ↓ [psf] ↓ [psf] ↓ [psf] | | v @ | | |
| Ø | | | | | ОК Салсе | əl |

Figure 5.6: Dialog box New Surface Load

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

In the *Load Magnitude* dialog section, we enter a value of **-30 psf** (see Chapter 2.3, page 6). Then, we close the dialog box by clicking [OK].

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

B



5.3 Load Case 3: Live Load, Area 2

We create a [New Load Case] to enter the live load of the right field.

| Load Cases | and Combinations | | | | |
|-----------------|-------------------------------------|-----------------|-----------------------|----------------------|-------|
| oad Cases L | and Combinations Result Combination | ins | | | |
| Existing Load C | ases | LC No. | Load Case Description | | Solve |
| G LC1 | Self-weight and finishes | <u> </u> | Live Load, Area 2 | • | |
| Qi LC2 | Live Load, Area 1 | Ganaral | Line Brenn terr | | |
| QI LCS | Live Load, Alea 2 | Clerieral Calcu | liation Parameters | | |
| | | Action Type | | Self-Weight | |
| | | Qi Imposed | • | Active | |
| | | | | Factor in direction: | |
| | | | | | |
| | | | | Z: | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | | | | |
| | | Comment | | | |
| <u>-</u> | | K | | - |] |
| | | | | | |

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

We enter **Live Load**, **Area 2** in the *Load Case Description* text box. Then we close the dialog box with [OK].

5.3.1 Surface Load

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

In addition to surface 2, we can see that the parameters of the recent entry are automatically set (load type *Force*, load distribution *Uniform*, load direction *Global ZL*, *Load Magnitude* **-30 psf** – see Figure 5.6). We can confirm the dialog box without making any changes.

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



Œ

μЦ



5.3.2 Line Load

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

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

The line load as load type *Force* with a *Uniform* load distribution is acting in the *ZL* load direction. In the *Load Parameters* dialog section, we enter -**350 lbf/ft** (see Chapter 2.3, page 6).



Figure 5.8: Dialog box New Line Load

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

γЩ

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



5.4 Load Case 4: Wind

In the last load case we define the wind load which is acting to the walls (not represented in our model).

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



Figure 5.9: Shortcut menu Load Cases

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

| | Ec | lit Load Cases and Combinations | E 🗙 | |
|---|-----------------|---|-----------|--|
| Load Cases Load Combinations Result Combinations | | | | |
| Existing Load Cases Self-weight and finishes Gi LC1 Self-weight and finishes Gi LC2 Live Load, Area 1 Gi LC3 Live Load Area 2 | LC No. | LC No. Load Case Description 4 Wind in -Y V | | |
| Ow LC4 | Action Category | UUD I Parl ameter s | | |
| | Comment | v 🕲 | | |
| | | | OK Cancel | |

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

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

5 Loads

| \angle | \searrow |
|----------|------------|
| | |
| DI | ubal |

| 321 | | - |
|-----|---|-----|
| X | | CL. |
| | 2 | ~~~ |

R ₽ We click the [New Line Load] toolbar button again to open the New Line Load dialog box.

| | | New Line Load | d | E 🔀 |
|---|---|--|---|-------------|
| No. Reference 1 | f lines | No. | Load Type 'Force' Load Distribution 'Uniform | P |
| ● <u>Force</u> ○ <u>Moment</u> | Concentrateg: P Vinform Trapezoidal Tapered Parabolic Varying | Local related to true line length: Global related to true line length: Global related to projected line length: C |) x) y) z) XL) YL) ZL) XP) YP) ZP Load Direction 'Global YL' Z Y | j i i |
| Load Parameters p:65.0 € ↓ [lbf p2: ↓ ↓ [lbf Comment | f/ft] A: [[/ft] B: [[Relative dir [Load over 1 [line | <pre> (t) (t) (t) (t) (t) tance in % total length of </pre> | z j | Ì |
| | | | [| OK Cancel |

Figure 5.11: Dialog box New Linie Load

We define the *Load Direction* as **YL** and enter the force of -**65 lbf/ft** (see Chapter 2.3, page 6). After [OK], we click the lines **2**, **4**, **6**, and **8** to apply the effective wind load on the ceiling.





To adjust the loads on lines **2,6** and **8** according to the values listed in Chapter 2.3 on page 6, we double-click each load and change the force.



Figure 5.13: Graphical input of wind loads

Changing the model display

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

5.5 Checking Load Cases

All four load cases have been completely entered. It is recommended to [Save] the model now. We can check each load case quickly in the graphics: The buttons [◀] and [▶] in the toolbar allow us to select previous and subsequent load cases.

| Ta <u>b</u> le | <u>O</u> ptions | <u>A</u> dd-on Modules | <u>W</u> indow | <u>H</u> elp | | |
|----------------|-----------------|------------------------|----------------|--------------|---|---|
| - 1 | 🔲 🖄 | LC3 - Live Load, Are | a 2 | | 💽 扵 🎅 🎬 🙈 📨 🛛 🐼 🛤 🗄 🎬 🐺 | 2 |
| • | 🗟 - 🔇 | 웹 - 원 - 일 | <u>b 54</u> 16 | 🍄 - | 🕂 🕱 Previous Load Case, 🤃 🏹 🕈 🕇 💆 - 🗌 🧭 | Ŧ |

Figure 5.14: Browsing the load cases



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

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







6. Combination of Load Cases

According to ASCE 7, we have to combine the load cases with factors. The *Action Type* specified before, when we created the load cases, makes generating combinations easier (see Figure 5.10, page 32). In this way, we can control the load factors and when combinations are created.

6.1 Creating Load Combinations

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

- 1.2*LC1 + 1.6*LC2 live load in area 1
 1.2*LC1 + 1.6*LC3 live load in area 2
 1.2*LC1 + 1.6*LC2 + 1.6*LC3 live load in both fields
- 1.2*LC1 + 1.6*LC2 + 1.6*LC3
 - 1.2*LC1 + 1.0*LC2 + 1.0*LC3 + 1.0*LC4 Full load

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

Creating CO1

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

| | | Edit Load Cases and Comb | pinations | | | | | |
|---|----------------|---------------------------------|-----------|------------|--------|-------------|------------------|--------|
| Load Cases Load Combinations Result Combinations | ; | | | | | | | |
| Existing Load Combinations CO1 live load in area 1 | CO <u>N</u> o. | Load Combination Description | | | | v | To <u>S</u> olve | |
| | General Calo | elation Parameter | | | | | | |
| | | | | | | | | |
| | G LC1 | Self-weight and finishes | | Load Cases | G LC1 | Self-weight | and finishes | |
| | Qi LC2 | Live Load, Area 1 | | 1.60 | Qi LC2 | Live Load, | Area 2 | |
| | Qi LC3 | Live Load, Area 2 Wind in -Y | | | | | | |
| | 201 | | | | | | | |
| | | | | | | | | |
| | | | > | | | | | |
| | | | >> | | | | | |
| | | | | | | | | |
| | | | 4 | | | | | |
| | | | 44 | | | | | |
| | | | ~~ | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | 1 1 | 1.60 | . 10 | | | |
| | | w (4) V 😡 🖂 | | 1.00 🖤 | 1.0 | | | |
| < > | Comment | | | | | | | |
| All (1) 🗸 🗙 | | | | | | × @ | | 2 |
| 2 🔊 🖫 | | | | | | | ОК | Cancel |
| | | | | | | | | |

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

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



In the tab *Calculation Parameters*, we check if the *Method of Analysis* is set according to *Second-order analysis* (see Figure 6.2).



>

6 Combination of Load Cases



Figure 6.2: Tab Calculation Parameters

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



Figure 6.3: Loads of load combination CO1

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



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

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

Creating CO3

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

| Project Navigator - Data | | × | | | | | | |
|--------------------------|------------------------------|-----|--|--|--|--|--|--|
| RFEM | | | | | | | | |
| Model Data | | | | | | | | |
| Load Cases and Combi | I had Cases and Combinations | | | | | | | |
| Load Cases | | | | | | | | |
| LC1: Self-weight | t and finishes | | | | | | | |
| 🗀 LC2: Live Load, / | Area 1 | | | | | | | |
| 🗀 LC3: Live Load, / | Area 2 | | | | | | | |
| LC4: Wind in -Y | | | | | | | | |
| 🖨 📑 Load Combinati | Edit Load Combinations | | | | | | | |
| — 🚞 CO1: live loi 📑 | Edit Load Combinations | | | | | | | |
| 🔤 CO2: live lo: 🛃 | New Load Combination | | | | | | | |
| 🔤 🖓 Result Combina 🔐 | Go to Table | | | | | | | |
| 🗄 🖷 🛄 Loads 🛛 🖉 | Delete All Load Combinations | Dal | | | | | | |
| Results | Delete Air Load Combinations | | | | | | | |
| Sections | | | | | | | | |
| Average Regions | | | | | | | | |
| Printout Reports | | | | | | | | |
| Guide Objects | | | | | | | | |
| Add-on Modules | | | | | | | | |
| | | | | | | | | |
| 🔽 Data 🖆 Display 🔏 Views | | | | | | | | |

Figure 6.4: Creating COs using the navigator shortcut menu

We enter **live load in both fields** for the *Load Combination Description*.

Now we select the *Existing Load Cases* LC1, LC2 and LC3 simultaneously by holding the [Ctrl] key. We use the [▶] button again to transfer them to the list on the right. The factors are set automatically.

Creating CO4

To create the last load combination, we use the Table 2.5 *Load Combinations*. In column B we enter the description *Full Load* and we define the different load factors and load cases in column D through K.

| 2.5 Load | .5 Load Combinations × | | | | | | × | | | | | |
|----------|---|--------------------------------------|---------------------|--------|-------|--------|--------|--------|--------|--------|--------|---|
| | 🔟 🔢 🖂 🕺 🤐 🛃 🕒 🚱 😥 😥 🕄 🛛 🔺 🗣 🏴 🗮 🔛 🥵 🕼 🖬 💭 🛱 🏂 | | | | | | | | | | | |
| | Α | В | С | D | E | F | G | H | | J | K | |
| Load | | Load Combination | | | LC.1 | | LC.2 | | LC.3 | | LC.4 | |
| Combin. | DS | Description | To Solve | Factor | No. | Factor | No. | Factor | No. | Factor | No. | |
| CO1 | | live load in area 1 | V | 1.20 | G LC1 | 1.60 | Qi LC2 | | | | | |
| CO2 | | live load in area 2 | V | 1.20 | G LC1 | 1.60 | Qi LC3 | | | | |] |
| CO3 | | live load in both fields | ✓ | 1.20 | G LC1 | 1.60 | Qi LC2 | 1.60 | Qi LC3 | | | |
| _CO4 🤇 | | Full load | ✓ | 1.20 | G LC1 | 1.00 | Qi LC2 | 1.00 | Qi LC3 | 1.00 | Qw LC4 | |
| CO5 | | | | | | | | | | | | ~ |
| | | | | | | | | | | | | |
| < | | | | | | | | | | | | > |
| Load Cas | es L | oad Combinations Result Combinations | | | | | | | | | | |
| <u>`</u> | | ^ | / | | | | | | | | | |

Figure 6.5: Table 2.5 Load Combinations

To create navigato Project Navi

2

20 男 20 - 25 - 25 - 45

Load Cases and Combinations...

💁 New Load Case...

New Load Combination...

New Result Combination...



6.2 Creating Result Combinations

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

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

| | Ec | dit Load Cases and Combi | nation | S | | | | 🖻 🗙 |
|--|------------------|---|------------|--------------------------|--------------|------------------|-----------|--------|
| Load Cases Load Combinations Result Combinations |] | | | | | | | |
| Existing Result Combinations | RC No. | Result Combination Description | | | | Solve | | |
| RC1 Governing Result Combination | 1 | Governing Result C | ombinatio | n | * | ✓ | | |
| | | | | | | | | |
| | General Calcula | tion Parameters | | | | | | |
| | Existing Loading | | | Loading in Result Combin | ation RC | | | |
| | C01 | ive load in area 1 | | Factor No. | Descrip | tion | Criterion | Group |
| | CO2 | ive load in area 2 ive load in both fields | | 1.00 CO2 | live load in | area 1 area 2 | Permanent | |
| | CO4 | Full load | | 1.00 CO3 | live load in | both fiel | Permanent | 1 |
| | | | | 1.00 CO4 | Full load | | Permanent | 1 |
| | | | | | | | | |
| | | | | | | | | |
| | | | ▶ | | | | | |
| | | | (P | | | | | |
| | | | \bigcirc | | | | | |
| | | | | | | | | |
| | | | 4 | | | | | |
| | | | 4 | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | Y CO Loa | d Combinations (4 ∨) | | 1.00 🗸 🖌 1.0 | Permanent | ~ | 1 | ✓ |
| | Comment | | | | Variable | | | |
| | Semilon | | | | | | | |
| All (1) 🗸 🗙 | | | | | V 12 | | | -63 |
| | | | | | | | | |
| ۵ 🖻 😭 | | | | | | | ОК | Cancel |

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

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



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

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

۶

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

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



60

7. Calculation

7.1 Checking Input Data

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

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

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

| Plausibi | lity Check 🖙 💌 |
|---|---|
| Check | Type of Check O Normal With warnings |
| Which Load Cases © Current load case All | None, gnly statistic Option ✓ Generate FE mesh ✓ Detect collisions of solids |
| ٢ | OK Cancel |

Figure 7.1: Dialog box Plausibility Check

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

| | | Plau | sibil | ity Check | : | | ē | × |
|----------------------|---------------------------|-----------|-------|-------------|---------|-----|----|---|
| All data No error | has been che rs found. | cked. | | | | | | |
| Info | Model Data | Load Data | | | | | | |
| Struct | ure Dimension | S | | Structure V | Veight | | | |
| Δχ: | 34.00 | [ft] | | Surfaces: | 61942.6 | [b] | | |
| Δγ: | 27.00 | [ft] | | Solids: | 0.0 | [b] | | |
| Δ z : | 10.00 | [ft] | | Members: | 13394.4 | [b] | | |
| | | | | Total: | 75337.0 | [b] | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| ٢ | 0.00 | | | | | | ОК | |

Figure 7.2: Result of plausibility check

We find more tools for checking the structural and load data by selecting Model Check on the Tools menu.



7.2 Generating the FE Mesh

As we have selected the option *Generate FE mesh* in the *Plausibility Check* dialog box (see Figure 7.1), we have automatically generated a mesh with the standard mesh size of 1 ft. (We can modify the default mesh size by selecting *FE Mesh Settings* on the *Calculate* menu.)



Figure 7.3: Model with generated FE mesh

7.3 Calculating the Model



To start the calculation,

we select Calculate All on the Calculate menu

or we use the toolbar button shown on the left.



Figure 7.4: Calculation process

8. Results

8.1 Graphical Results

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



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

Selecting load cases and load combinations

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



Figure 8.2: Load case list in the toolbar

Selecting results in the navigator

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



•

M



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

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



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

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

Display of values

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



| 8 Results | | | | | | | |
|---|----------|--------------------|----------------|--------|--------|-------|---|
| | | | | | | | Dlubal |
| Project Navigator - Results | × -5.708 | +0.319 | 0.513 | 0.641 | 0.657 | 0.636 | Panel Basic Internal Forces |
| ⊕ ✓ ✓ ✓ ✓ ⊕ ← ● ✓ ✓ ✓ ⊕ ↓ ✓ ✓ ✓ ✓ ⊕ ↓ ✓ ✓ ✓ ✓ Ø ↓ ✓ ✓ ✓ ✓ Ø ↓ ✓ ✓ ✓ ✓ Ø ↓ ✓ ✓ ✓ ✓ ✓ Ø ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓ | 1.788 | 1.935 | 1.832 3.004 | 2.677 | 2, 415 | 1.505 | 5.064 3.283 1.503 -0.278 -2.059 |
| | 4.677 | 4.229 | 3.750 | 3.321 | 2.966 | 2.683 | 3.840 -5.621 - 7.401 - 9.182 - 10.963 - 12.744 |
| On grid and user-defined points | 5.042 | <mark>4.531</mark> | 4.032 | 3.566 | 3.189 | 2.880 | Max : 5.064 Min : -14.524 |
| - 🦰 Numbering | 4.810 | <mark>4.348</mark> | 3.862 | 3. 423 | 3.060 | 2.771 | |
| | 3.996 | 3.686 | 3.263 | 2.900 | 2.611 | 2.383 | |

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

8.2 Results Tables

🛱 Data 🖀 Display 🔏 Views 🗢 Results

We can also evaluate results in tables.

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

| A | В | C | D | | |
|---|-----------|------|--|--|--|
| Description | Value | Unit | Comment | | |
| CO1 - live load in field 1 | | | 1 | | |
| Sum of loads in X | 0.000 | kip | | | |
| Sum of support forces in X | 0.000 | kip | | | |
| Sum of loads in Y | 0.000 | kip | | | |
| Sum of support forces in Y | 0.000 | kip | | | |
| Sum of loads in Z | -119.398 | kip | | | |
| Sum of support forces in Z | -119.398 | kip | Deviation: 0.00 % | | |
| Maximum displacement in X-direction | 0.050 | in | Member No. 2, x: 11.000000 ft | | |
| Maximum displacement in Y-direction | -0.021 | in | Member No. 4, x: 4.000000 ft | | |
| Maximum displacement in Z-direction | -0.221 | in | FE Node No. 162 (X: 9.000000, Y: -8.000000, Z: 0.000000 ft) | | |
| Maximum vectorial displacement | 0.224 | in | FE Node No. 162 (X: 9.000000, Y: -8.000000, Z: 0.000000 ft) | | |
| Maximum rotation about X-axis | -1.7 | mrad | FE Node No. 735 (X: 0.000000, Y: -14.000000, Z: 0.000000 ft) | | |
| Maximum rotation about Y-axis | | mrad | FE Node No. 741 (X: 2.000000, Y: 0.000000, Z: 0.000000 ft) | | |
| Maximum rotation about Z-axis | -0.2 | mrad | Member No. 5, x: 0.000000 ft | | |
| Method of analysis | 2nd Order | | Second order analysis (Nonlinear) | | |

Figure 8.5: Table 4.0 Results - Summary

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



📕 🚳 💰



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

< >

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



Results Navigator

ſ₽

Filter Results 8.3

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

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

8.3.1 Visibilities

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

Results display for concrete columns

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

- Members sorted by type:
- Members sorted by cross-section: 2 - Rectangle 12/12 ٠

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



Figure 8.7: Moments M_y of concrete columns in scaled representation

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

Adjusting the scaling factor

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



Results display of floor slab

In the same way, we can filter surface results in the *View* navigator. We deactivate the options *Members by Type* and *Members by Cross-Section* and select *Surfaces by Thickness* where we select the entry *8 in*.



Figure 8.8: Shear forces of floor

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



8.3.2 Results on Objects

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

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

| Visibilities | |
|----------------------------------|---|
| y y y y y y | |
| Visibilities: | |
| 🖃 🗐 🔄 User-defined | |
| 🛄 🛄 🛅 Group 1 | |
| 🖮 🔳 🚞 Generated | |
| 🖶 🖳 🦲 Lines by Type | |
| 🖶 🔲 🚞 Members by Cross-Section | - |
| <u>a</u> a a a a <u>b</u> e | |
| Add new objects into visibility: | |
| | |

Figure 8.9: Resetting the overall view in Views navigator



1

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

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



Figure 8.10: Shear force diagram of left surface

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





Shortcut menu Member

Ð

8.4 Display of Result Diagrams

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

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

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



Figure 8.11: Display of result diagrams of downstand beam

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

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

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

We quit the function Result Diagrams by closing the window.



9. Documentation

9.1 Creation of Printout Report

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

To open the printout report,

we select Open Printout Report on the File menu

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

| New Printout | Report 🖙 💌 |
|---|------------|
| No. Description Input data and reduced res | ults |
| Printout Report Template 1 - Input data and reduced results | v 🔁 👺 |
| Ø | OK Cancel |

Figure 9.1: Dialog box New Printout Report

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

| Â | Printout repo | rt - PR1: Ir | put data ar | nd reduced re | esults* | | ♦ ⊑ | |
|---|---------------|---|------------------------|---|---|-------------------|---------------------------------------|-------------------|
| File View Edit Settings Insert Help | | | | | | | | |
| 🏝 🗟 📓 🥪 💟 🤜 🗖 🛤 🛤 🖓 🖧 췋 | - 🔊 💦 🖻 | 1 🗈 🗅 🖸 | 🔗 🔑 🔑 | | | | | |
| Printout Report Navigator | × | | | | | | | |
| 🖃 📹 Printout Report | | | | | | | | |
| RFEM | | | | | | | | |
| | | 1 | | | | | | 1 |
| FE Mesh Settings | | | | | Musterma | inn | | |
| 🖶 🛅 Model | The second | | | | Musterstad | t | | |
| 1.1 Nodes | The second | | | | 000000000000 | 0 | | |
| 1.2 Lines | | -0 | | | | | | |
| 1.3 Materials | | Project: | | | Mod | el: Model | | |
| 🛅 1.4 Surfaces | | | | | | | | |
| 🛅 1.7 Nodal Supports | | МО | DEL - GE | | ATA | | | |
| 🛅 1.13 Cross-Sections | | | General | Modelname | | | : Model | |
| 🛅 1.14 Member Hinges | | | | Type of model Positive direction of | folobal axis Z | | : 3D : Upward | |
| - 1.15/1 Member Eccentricities - Absolute | | | | Classification of loa | ad cases and | | : According to Sta National anney: | and ard: None |
| | | | | 001101101010 | | | | |
| - 1.17 Members | | | | | | | | |
| 1.18 Ribs | | PFE | MESH SE | TINGS | | | | |
| Load Cases and Combinations | | General Target length of finite elements I FE Maximum dictance between a node and a line z | | | | l re E | 1.0 ft 0.0 ft | |
| 🖮 🛄 Loads | | to integrate it into the line | | | | 500 | | |
| Results - Load Cases, Load Combinations | | | | waxmon nomber | or mean nodes (in tho | usanus) | | |
| | | | Members | Number of division elastic foundation, | s of members with cal taper, or plastic chara | ble, cteristic | | 10 |
| 4.1 Nodes - Support Forces | | | | Activate memb or post-oritical | er divisions for large o analysis | leformation | | |
| 4.12 Cross-Sections - Internal Forces | | | | Use division for | r members with node | lying on them | | |
| Kesults - Result Combinations | | | | | | | | |
| 4.1 Nodes - Support Porces | | | Surfaces | Maximum ratio of F Maximum out-of-pl | E rectangle diagonals ane inclination of two f | i finite | Δο : | 1.80 0.50 ° |
| H- 4.12 Cross-Sections - Internal Forces | | | | elements Share direction of | finite elements | | | Trian cles and ou |
| | | | | onape areation of | | | | Same squares |
| | | • 1 1 | NODES | | | | | |
| | | Node | | Reference | Coordinate | | Node Coordinates | |
| | P (X, Y, Z) | No. | Node Type | Node | System | X [ft] | Y [ft] | Z [ft] |
| | .Y 7 | 2 | Standard | | Cartesian | 0.00 | -16.00 | 0.0 |
| | × | 3 | Stand ard Stand ard | - | Cartesian Cartesian | 20.00 | -16.00 | 0.0 |
| | × × | 5 | Stand ard | - | Cartesian | 20.00 | -28.00 | 0.0 |
| | | 7 | Stand ard | - | Cartesian | 33.00 | -28.00 | 0.0 |
| | | 8 | Stand ard Stand ard | - | Cartesian Cartesian | 0.00 | -16.00 | -10.0 |
| | | 10 | Standard | - | Cartesian | 20.00 | -16.00 | -10.0 |
| | < | | | | | | | |
| | 110 | DEI | | | | | Pages: 12 | Page: 1 |

Figure 9.2: Print preview in printout report





9.2 Adjusting the Printout Report

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

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

| Printout Report Navigator | × | | | | | |
|---|------------|--|--|--|--|--|
| ⊡ Printout Report | | | | | | |
| RFEM | | | | | | |
| Model - General Data | | | | | | |
| FE Mesh Settings | | | | | | |
| 🖅 🛅 Model | | | | | | |
| 🛓 🗀 Load Cases and Com | binations | | | | | |
| 🚛 🛅 Loads | | | | | | |
| 🖃 🚞 Results - Load Cases, Load Combinations | | | | | | |
| | | | | | | |
| 🗄 🛅 4.1 Nodes - Support Forces | | | | | | |
| 4.12 Cross-Sections - Internal Forces | | | | | | |
| 🖮 🖮 Results - Result Combinations | | | | | | |
| 4.1 Nodes - Support Forces | | | | | | |
| Hand 4.12 Cross-Sections - Internal Forces | | | | | | |
| Remove from Printout Re | | | | | | |
| Start with New Page | | | | | | |
| | Selection | | | | | |
| | Properties | | | | | |

Figure 9.3: Shortcut menu Cross-Section - Internal Forces

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

| | Printout Report Selection - PR1 | Ē |
|--|--|----------|
| Program | Giobal Selection RC Results | |
| RFEM | Result Combinations to Display | |
| | | |
| | O Selected (1) | |
| | | |
| | Tables to Display | |
| | Display Table All Number Selection (e.g. 1 | 1-4,8') |
| | 4.1 Nodes - Support Forces V Al | |
| | 4.3 Lines - Support Forces V All | |
| | 4.4 Members - Local Deformations V All | |
| | 4.5 Members - Global Deformations V All | |
| | 4.6 Members - Internal Forces V All | |
| | 4.8 Members - Strains ✓ All | |
| | □ 4.9 Members - Coefficients for Buckling ☑ All | |
| | 4.10 Member Slendemesses ✓ All All | |
| | 4.11 Sets of Members - Internal Forces | |
| | 4.13 Surfaces - Local Deformations | |
| | 4.14 Surfaces - Global Deformations I All | |
| | Details - Internal Forces by Cross-Section 🛛 🗁 🔀 | |
| | Display Display M Min Internal Forces | |
| | Nodal values | |
| | Partition values Vy V My | |
| | ✓ Extreme values ✓ V ₂ ✓ M ₂ | |
| | Extreme values of | |
| | gross-sections | |
| | | |
| Display | | |
| Cover sheet | | |
| Contents | OK Cancel | |
| | ponding load cases | |
| oppercase unes | | |
| ۲ | | OK Canad |
| and the second s | | Cancel |

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

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

....



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

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

9.3 Inserting Graphics in Printout Report

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

Printing deformation graphics

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

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

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

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





x



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

| Graphic Printout | | | | | | | | |
|---|--|-----------------------------|--|--|--|--|--|--|
| General Options Color Scale Factors Borde | er and Stretch Factors | | | | | | | |
| Graphic Picture | Window To Print | Graphic Scale | | | | | | |
| O Directly to a printer | Current only | ◯ As screen view | | | | | | |
| To a printout report: PR1 | <u>M</u> ore | Window filling | | | | | | |
| ○ To the Clipboard | Mass print | ◯ To scale <u>1</u> : 100 ∨ | | | | | | |
| ○ To <u>3</u> D PDF | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| Graphic Picture Size and Rotation | Options | | | | | | | |
| Use full page <u>w</u> idth | Show results for selected <u>x</u> -location in result | | | | | | | |
| O Use full page height | Lock graphic picture (without update) | | | | | | | |
| ● Height: 55 + [% of page] | | our update) | | | | | | |
| | Show printout report on [0 | K] | | | | | | |
| Rotation: 0 🜩 [°] | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| Header of Graphic Picture | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| D | | OK Cancel | | | | | | |
| | | | | | | | | |

Figure 9.6: Dialog box Graphic Printout

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

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



Figure 9.7: Deformation graphic in printout report



Printing the printout report

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

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

we select Export to PDF on the File menu.

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

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





Save

B



10. Outlook

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

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

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

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

