Demoversion

RFEM 3D

Short Introduction
for a quick start

Ingenieur-Software
Dlubal

Ingenieur-Software Dlubal

Demoversion

RFEM 3D

Short Introduction for a quick start

Finite elements for three dimensional structures consisting of shell, slab and frame like members.

Am Zellweg 2, D-93464 Tiefenbach
Phone: 09673 / 9203-0
Fax: 09673 / 1770
E-Mail: info@dlubal.com
Internet: www.dlubal.de

Ingenieur-Software Dlubal

CONTENT

Ingenieur-Software Dlubal

Ingenieur-Software Dlubal

Dear RFEM User,

We thank you for your interest in RFEM of Ing.-Software Dlubal GmbH. You are about to get to know an extensive, powerful and above all user friendly and three dimensional finite element program.

With the help of this brochure you'll learn the basic functions of this program within a few hours. You can then extend your knowledge beyond the functions learned herein such that in a short space of time you'll be modelling and designing your own projects without any difficulties and be using the full extent of the RFEM performance spectrum for your daily work. We've decided to describe the functions in RFEM and use of the program by means of several relatively simple examples. The generation of examples has been sub-divided into several chapters in order to simplify restarting your tuition session and the subsequent location of working steps after an interruption. The functions concerning the viewing of the structure being modelled are listed in an extra chapter.

It should also be mentioned that all of the following illustrated explanations concerning appearance and functionality are related to the default settings of the operating system and the program RFEM 1.xx. In case of altered settings in your software the following statements made may not apply completely under certain conditions. Basic knowledge of Windows is presumed in order to operate the program and will not be explained in this brochure.

Your Team at Dlubal wishes you a lot of fun and success while using this software

System requirements

Minimum configuration:
300 MHz processor
64 MB RAM
400 MB free hard disk memory
Windows 98, Me, NT4.0 SP5 upwards, 2000 or XP
Graphic resolution 800x600

Recommended configuration:
800 MHz processor
256 MB RAM
500 MB free hard disk memory
Windows 2000 or XP
Graphic resolution 1024x768

Demo restrictions

The following restrictions are included in the demo version of RFEM
A maximum number of 2 surfaces and 12 elements can be calculated.
Saving of input and results is not possible
Printing of results is not possible.

Installation

Note: Administrator rights are required for the installation. Before starting the installation please close all other applications and carry out the following installation procedure.

1. Insert the RFEM-CD in your CD-ROM drive
2. The installation routine will start automatically
3. Follow the instructions of the installation menu and the setup assistant in order to install RFEM. When queried as to the location of the authorization any arbitrary path may be specified e.g. „C:\“ as no full version should be installed.

If the CD-ROM drive Auto play option is deactivated the installation will not start automatically. To install RFEM manually start the Setup.exe by double clicking with the left hand mouse key. You'll find this file in the CD-ROM directory in the RFEM CD.

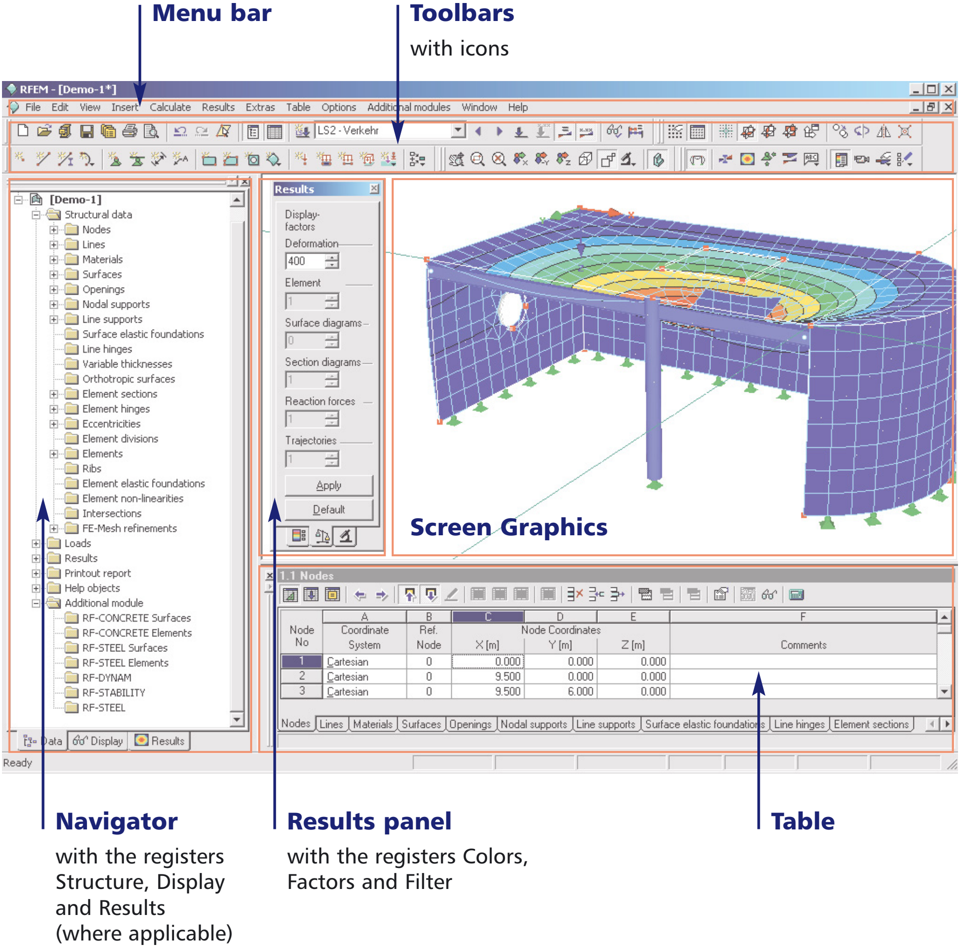
The RFEM Help system will be automatically installed with the program. These can be called upon by clicking on the menu item Help in RFEM. You'll find support for every theme related to RFEM in Help.

Starting RFEM

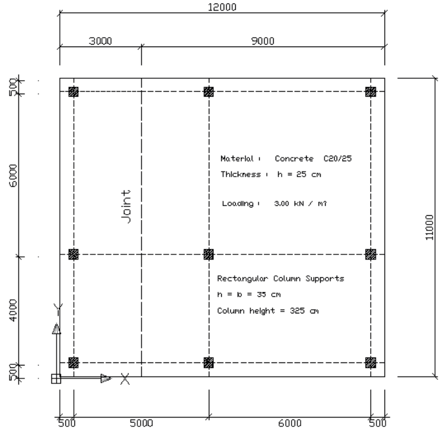
An RFEM icon will be created on the desktop during installation. The program can be started either with a double click on the icon or by activating the program RFEM in the Start menu under programs → Dlubal applications.

The RFEM user interface

In order to get to know the most common descriptions of parts of the user interface the most important elements of which will be briefly displayed.



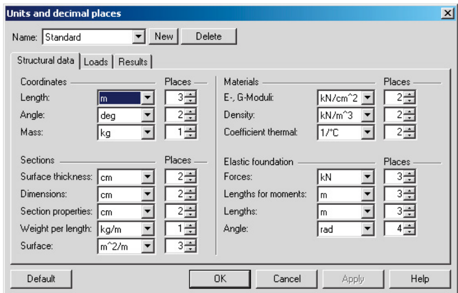
In the first example of this quick start manual a point supported flab slab will be generated and designed.



Create new position

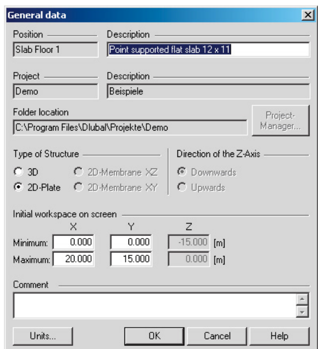
If RFEM is started for the first time or if a new position is created manually, a dialog opens in which details of the new position are entered. The position name which is also the file name, the description with which the position maybe found more quickly in the project manager and the basic settings such as structure type and Z-axis direction can be entered.The position can be assigned to particular projects, the organisation of which can be completed in the Project manager.

By clicking on the button „Units“ the units of the structural dimensions, loads and results can be customized as required. The defaults settings will be used here.



In order to create the position for the following modelling of the flat slab the following will be entered in the dialog:

1. Enter „Slab Floor 1“ as **position name.**
2. Enter „Point supported flat slab 12 x 11“ as the **description.**
3. Select „2D Plate“ as **structure type**.
4. By clicking on the **OK** button the position will be created.

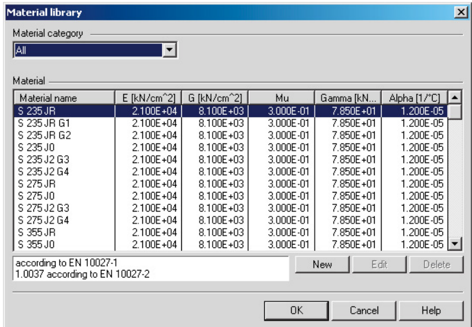


Application: Create New Position

Ingenieur-Software Dlubal

Inputting the floor slab

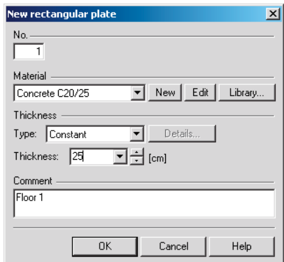
The input of the floor slab can be done most easily through the command **New planar surface**, for which an icon is available in the toolbars. In the dialog which appears the surface can be defined by material, thickness and surface type and a comment can also be added.



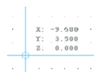
By clicking on the button **Library** you obtain access to an extensive material library, which can also be complemented by self defined materials.

Here are the following steps in defining the surface properties:

1. Enter a **thickness** of **25 cm**. This can be done either by direct input per keyboard or by clicking on the arrow to the right of the value. Should the thickness already be available somewhere in the structure then this can be selected from the list box. The list box drops open by clicking on the list box arrow.
2. Enter **„Floor 1“** as comment.
3. Confirm the input through **OK**.



On confirmation of the entries made the cursor transforms into cross-hairs. With the help of the mouse pointer cross-hairs the diagonally opposite corner nodes of the surface can be entered. This happens quite simply through a simple click on the desired points of the screen. The coordinates are displayed next to the cross-hairs as a form of control.



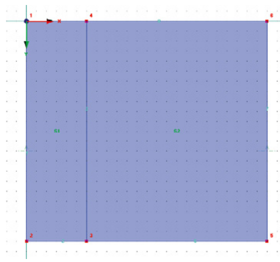
Creating the first surface portion

1. Move the cross-hairs to coordinate **0.00 / 0.00 / 0.00 m**.
2. Confirm the first point with a left click.
3. Move the cross-hairs to the coordinates **3.00 / 11.00 / 0.00 m**.
4. Confirm the second point with a left click.

Applied: Material Library Library... New Planar Surface 

After having entered both corner nodes the surface will be created with the previously defined properties. As the function **New planar surface** is still active the second surface portion can be generated immediately without having to recall the function. This is done in the following manner:

1. Move the cross-hairs to coordinate 3.00 / 0.00 / 0.00 m. This is node 4, a generated node in the first surface portion.
2. Confirm the first point of the second surface portion with a left click.
3. Move the cross-hairs to the coordinates 12.00 / 11.00 / 0.00m
4. Confirm the second point with a left click.
5. Exit the function **New planar surface** with a right click.



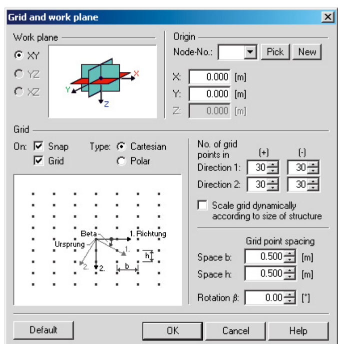
In order to obtain the best overview of the total structure it is recommendable to select the whole structure through the command Show whole structure. The respective icon is to be found

in the toolbar. With respect to this function and other viewing possibilities please see chapter 11 RFEM Functions.

Defining and placing columns

As the grid, on which the input is orientated, is currently set at 1.00m spacing while the columns are supposed to be positioned at a distance of 0.5m from the edges of the slab, it is at first necessary to adjust the grid as follows.

1. Click on the icon **Grid and Work plane** settings in the  toolbar in order to open the corresponding dialog window.
2. Set the value of both b and h to 0.5m in the fields Grid point spacings.
3. By clicking on OK the values will be accepted and the changed grid be assumed.



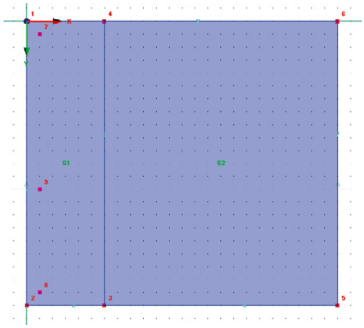
Application: Show whole structure  Grid and Work plane 

The first column should be placed at 0.5m from the upper and left hand edge of the slab. In order to place a support at these coordinates a node will be firstly placed in this position.

Procedure:

1. Click the icon **New node** in the toolbar. 
2. Move the cursor to the coordinates 0.50 / 0.50 / 0.00 m.
3. By a left click create a node at this point.

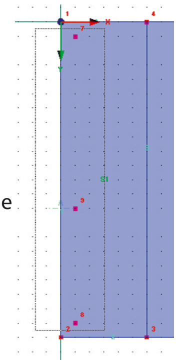
The node has now been created. As the function **New node** is still activated the following column support nodes which are positioned below the first node can be created immediately.



1. Move the cursor to the coordinates 0.50 / 6.50 / 0.00 m.
2. Create a node on this point with a left click.
3. Move the cross-hairs to the
coordinates 0.50 / 10.50 / 0.00m
4. Create a node on this point with a left click.
5. Exit the function **New node** with a right click.

The column row can now be copied using the function **Move / Copy**, so that the final column grid can be created. First of all the three support nodes are to be activated by dragging a window across the nodes such that all three nodes are marked but also ensuring that only those to be copied are selected. Then they can be copied by entering a specific displacement vector.

Note: The numbering of structural subcomponents can be switched on or off under the item **Numbering** which is to be found in the register **Display** in the **Navigator**.



The selection procedure is done as follows:

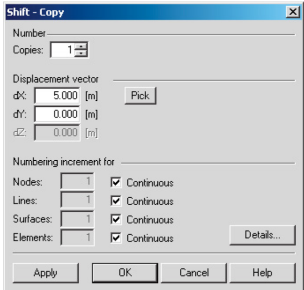
1. Move the mouse pointer to a position above and to the left of node 7 outside of the surface.
2. Hold down the left mouse key and keep held down.
3. Drag the mouse pointer to a position below and to the right of **node 9** (see figure to the right)
4. Release the mouse key.

The nodes or all structural components which lie within the marked zone are thus selected and can be subsequently commonly edited or used.

Application: Create node  Selection frame

The marked nodes numbered 7, 8 and 9 should firstly be copied and displaced by a distance of 5m in order to create the second row of columns.

1. Select the function Move/Copy by clicking on the respective icon in the toolbar.
2. Set the number of copies to „1“.
3. Enter the displacement vector of „5.00“ in the box denoted by dX.
4. The nodes are copied by clicking on OK.



The nodes have thus been copied. As the copied nodes are still marked as being active, these can be used to create the third row of columns.

1. Select the function Move/Copy by clicking on the respective icon in the toolbar.
2. Enter the displacement vector of „6.00“ in the box denoted by dX.
3. The nodes are copied by clicking on OK.

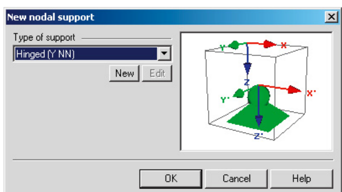
The thus created node should now be assigned with the required supports. This is done with the function **New nodal support**, for which an icon is available in the toolbar.

Initially the nodes to be supported must be selected:

1. Drag the selection frame over nodes 7-15 such that only these objects lie within the frame (left mouse key held down).
2. Release mouse key in order to select the nodes.

The type of support will be defined next:

1. Click on the icon **New nodal support** in the toolbar.
2. Click on the button New to define a new type of support.
3. Activate the button **Column in Z**. ☑ Column in Z Details...
4. Click on the button Details in order to define the column.
5. Enter the column dimensions with width x = 35cm, width y = 35cm.
6. Activate **Spring in Z**.
7. Enter a height of 325cm.
8. Through clicking three times on **OK** in the respective windows the columns will be created.

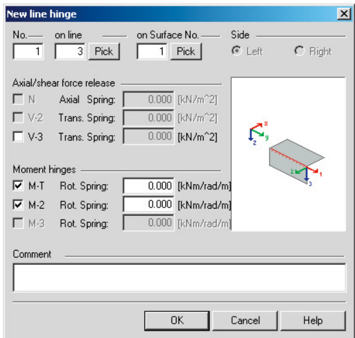


Application: Move / Copy Create nodal support

Hinge definition between the floor slabs

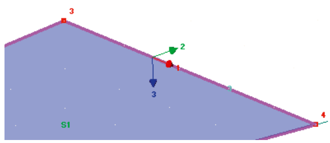
In order to model a joint between the individual floor slabs a hinge is to be inserted which prevents the transfer of moment from one slab to the other. Such a hinge element is known as a Line hinge in RFEM. In order to insert such a hinge the following steps are to be undertaken.

1. Select the menu item Insert from the menu bar
2. Select the menu item Structural data from the menu Insert.
3. Select the menu item 1.9 Line hinges from the menu Structural data.
4. Select the menu item Dialog from the menu Line hinges.
5. Enter the number „3“ in the box denoted on line No. or select the line graphically via **Pick**.
6. Enter the number „1“ in the box denoted on surface No. or select the surface graphically via **Pick**.
7. Activate the moment hinges M-T and M-2 by setting a tick in the respective check boxes.
8. By clicking on Ok the hinge will be created.



Here follows a detailed description of the function in order to explain the exact effect of a Line hinge:

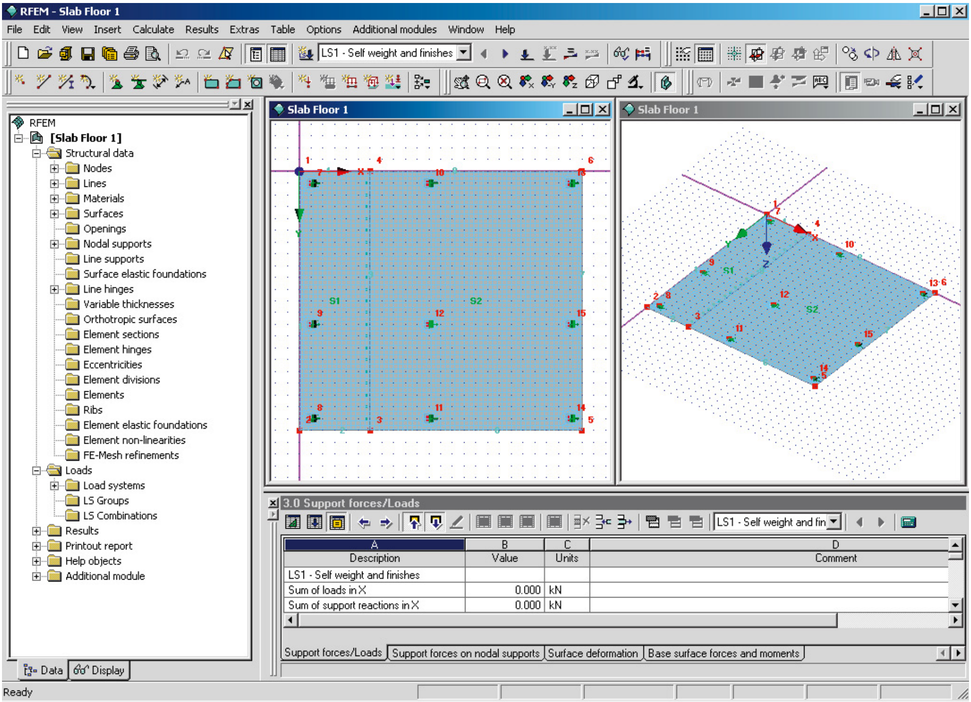
Required for **the definition of a line hinge** is the line number upon which the hinge is to act, the surface number to which the hinge is related and the type of hinge required. With this a local line hinge coordinate system is generated. This system adopts the direction 1 as the direction 1 of the line. The 2nd direction of the local line hinge coordinate system lies directly in plane of the surface to which the line hinge refers. The 3rd direction lies thus perpendicular to the plane of the surface. The hinge types refer to this coordinate system.



As the structure in this example has been defined as a 2D-Structure, not all hinge types are at your disposal.

Application: Define and set line hinge

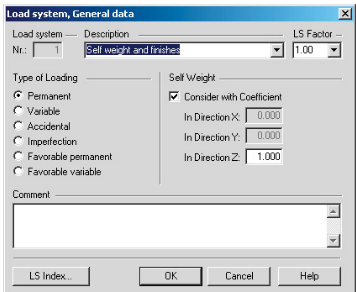
The input of the structure is now complete. The result of the modelling is displayed in the picture below in two different perspectives. In the left hand window the structure is viewed form above in the Z-direction whereas the second window displays an isometric view.



Load application

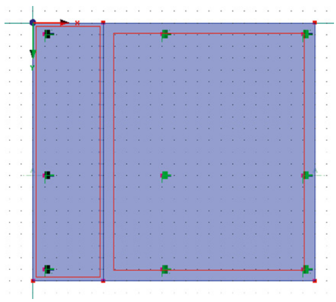
The loads will be applied very simply in this example. The actions are represented firstly by the dead weight of the slab and a permanent, uniformly distributed load of 3 kN/m² over the total surface area. An icon for the input of a uniform load over the full extent of the surface is available in the toolbar.
The load generation in this example is completed in the following manner:

1. Click on the icon New surface load in the toolbar. As no load case has been laid on up till now a dialog named New load case will open automatically, in which the load case settings for the current load case can be activated in which the following load input should apply.
2. Selection of the description „Dead weight and finishes" from the predefined list in order to describe the load case. This is done by clicking on the list box arrow next to the field denoted by description and by selecting the corresponding entry from the list. As this is the first load case to be generated the self weight will always be automatically considered and the type of loading will be defined as „permanent".



3. By clicking on OK the load case will be created and the surface load dialog will then open.
4. A load magnitude of „3.00" kN/m² is entered in the respective input box of the surface load dialog.
5. By clicking on Ok the surface load is generated and the surfaces to be loaded can now by selected with the mouse pointer.
6. Click on surface No. 1 loads this surface
7. Click on surface No. 2 loads this surface
8. Click on right hand mouse key exist the surface load function.

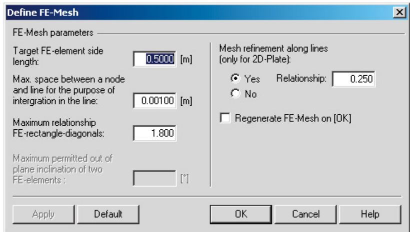
The input of loading is now complete and the structure should be displayed in the graphics similarly to the picture shown to the right.



Application: Create load case  Apply loading 

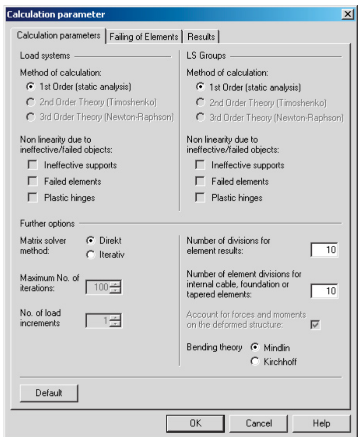
Calculation and Results

Before the calculation can be started the Finite-Element-Mesh must be generated. The mesh generation takes place automatically on starting the calculation. It is, however, recommended to check the FE-Mesh settings before starting the calculation. The FE-Mesh will be generated here in order to check the FE-Mesh settings, without starting the calculation:



1. Open the menu Calculate in the Menu bar.
2. Select the menu item Define FE-Mesh. Critical above all is the entry in the field **Target FE Element side length**. The element size should be adjusted to suit the structure size and shape and the support and loading conditions. For this example the settings are ok.
3. By clicking on OK the window will close.
4. Open the menu Calculate in the menu bar.
5. Select the menu item Generate FE-Mesh from the menu and the FE-Mesh will be calculated, generated and displayed on the structure.

The same procedure applies for the calculation of the structure. Firstly the calculation parameters are verified in order to subsequently start the calculation:

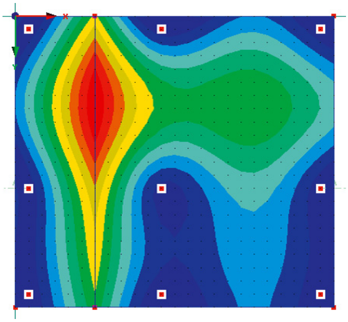


1. Open the menu Calculate in the menu bar.
2. Select the menu item Calculation parameters for the menu. The default settings are displayed here and can be adjusted as required.
   Remarks on the definitions: The direct matrix solver is optimal for small to normal sized structures, whereas the iterative method is preferable for large structures. The Mindlin method is optimized for thicker plates and the Kirchhoff theory for thin plates.
3. The window closes by clicking on OK
4. Open the menu Calculate in the menu bar.

5a. Selecting the menu item Calculate all, results for all load cases will be calculated.

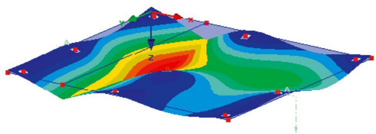
5b. By selecting the menu item To calculate, single or several load cases can be selected for calculation, should it be wished that not all results be calculated.

Application: Define and generate FE-Mesh, Calculation parameters, Calculate

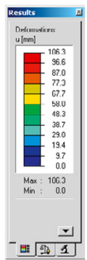


Directly after the calculation the deformation of the total system will be displayed. In order to also visualize the deformation on the structure the view should be switched to isometric. This can be done by clicking on the icon Isometric view. See also **Chapter 11 RFEM functions** regarding the selection of viewing options

The effect of the line hinge is clearly identifiable from the deformed figure. Surface results directly above the columns are not shown as these positions aren't governing for the design and the display of singularities are avoided

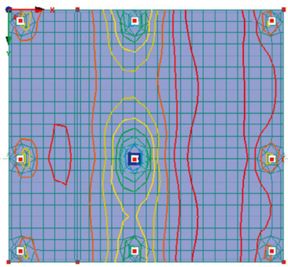


Should results be displayed in the graphics window the display of those results shown can be controlled in the results panel. This can be switched on or off by clicking on the **Results-control panel** icon in the toolbar.



All available results are assigned with uniform spacing to the maximum color palette and are displayed under the register **color spectrum** of the results panel. The assignment can be customized by the user with respect to Value ranges, intermediate values, colors and number of colors through a double click on the color or value spectrum.

The deformation size display factors can be individually adjusted under the register Factors.



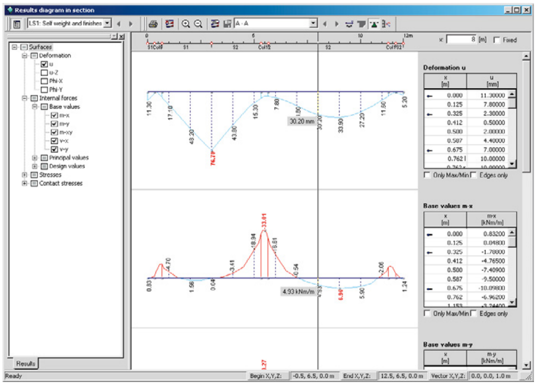
Should only certain parts of the structure be of interest regarding the results then those elements for which the results should be displayed can be selected under the register Filter. In this example the results display can be restricted to the left hand portion of the slab by entering the number „1“ in the respective box and then clicking the button **Apply**. By selecting the entry **All** from the list the results on all structural components will be displayed once more.

With the help of the menu Type of display for surface results diagram, (RFEM → suface!!!) for which an icon is available in the toolbar, the user can decide if the results should be displayed in the form of **isobands** or **isolines** or indeed if **No results display** is desired at all.

In order to observe the deformation, forces and moments more closely a section through the slab will be inserted at the desired location. For this the view in Z will be selected which is achieved by clicking the icon View in Z-direction.

In order to set a section across the slab through a row of columns the following steps are necessary:

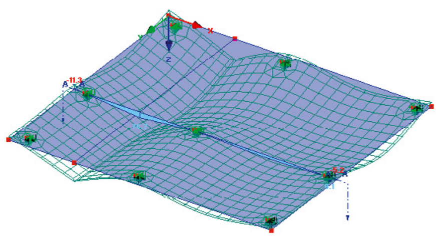
1. Click on the icon Sections in the toolbar.
2. Select the menu item **Define section graphically and results diagrams** from the menu.
3. Move the mouse pointer to the coordinates X = -0.50, Y = 6.50, Z = 0.00 m and set the start point of the section with a mouse click.
4. Move the mouse pointer to the coordinates X = 12.50, Y = 6.50, Z = 0.00 m and set the end point of the section with a mouse click.
5. Click on the icon Name and save section in the window which now opens and save the section by input of a section description (e.g. „**A**“) and a click on OK.



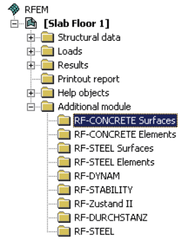
The selected types of results diagram from the sections navigator will then be displayed one above the other. The results diagram display can be individually adjusted in the dialog **Results diagram settings** which can be open via the icon **Settings**.

With the icon **Show results over columns** the result values above the columns can be switched on or off.

Once the section has been saved then the results in the section can also be displayed in the results display on the RFEM screen graphics.
The section can be edited and its position adjusted at will.



The additional module RF-Concrete Surfaces



With the additional module **RF-Concrete Surfaces** the slab can be designed to DIN 1045-88, DIN 1045-01 or EC2. The module is started by double clicking on the **module RF-Concrete Surfaces** in the directory **Additional Modules** under the **register Data** in the navigator. (See picture, right)

A module window opens in which the input, serving as the basis for design, is entered.

General data is entered in table 1.1. The relevant design code and load cases to be considered for the design are entered.

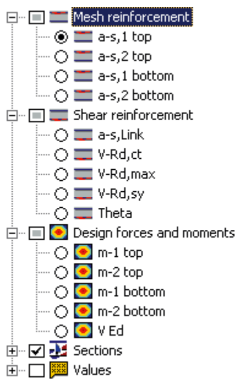
Further Input windows can be found by clicking on the **Forward button**, or by clicking on the corresponding icon in the module navigator.

In the following window, Table 1.2 the material and section data (e.g. Thickness) can be edited, which means that these can deviate from the model data in RFEM. In this example the data from RFEM will be accepted.

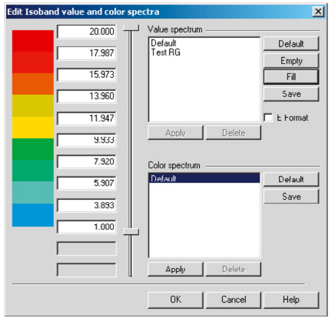
Four registers are available in Module **RF-Concrete Surfaces**, table 1.3 serving the input of reinforcement details in which the layout is to be stipulated. In this case all predefined settings will be used.

A click on the calculation starts the calculation. On completion the numerical results are automatically displayed in the module window. The results can also be displayed graphically in the RFEM graphics window by clicking on the button **Graphic**.

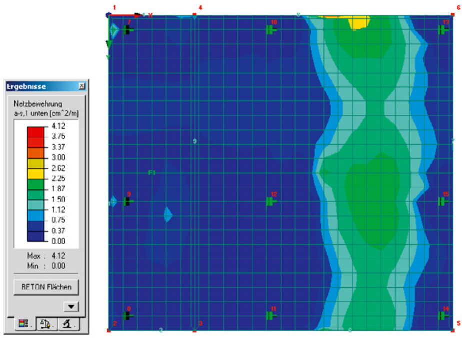
An additional entry „RF-Concrete Surfaces“ now appears in the list box of available load cases on the RFEM screen and should this be activated then the results display of design results can be controlled via the navigator under the register **Results**. (See picture, right)



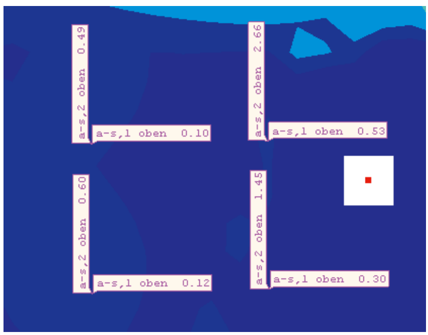
By adjusting the boundary values of the color spectrum the reinforcement area required can be adapted to available reinforcing meshes in order to display solely those areas in the graphics window where additional reinforcing bars are required. Editing the properties of the color spectrum is enabled by double clicking on the color or value spectrum.



The graphics can be configured impressively to requirements due to the possibility of displaying numerical results alongside the graphical results. It is possible to display numerically several values simultaneously by activating the display of values in the navigator under the register Results and selecting the option Target. The value bubbles can be switched on or off through a right hand click on the bubbles in order to restrict the numerical output to certain areas.



It's possible to display existing value groups or create new groups by activating the option Groups. These can be aligned to suit their respective directional effect as can be seen in the picture, right.



Application: RF-Concrete Surfaces

The Printout report

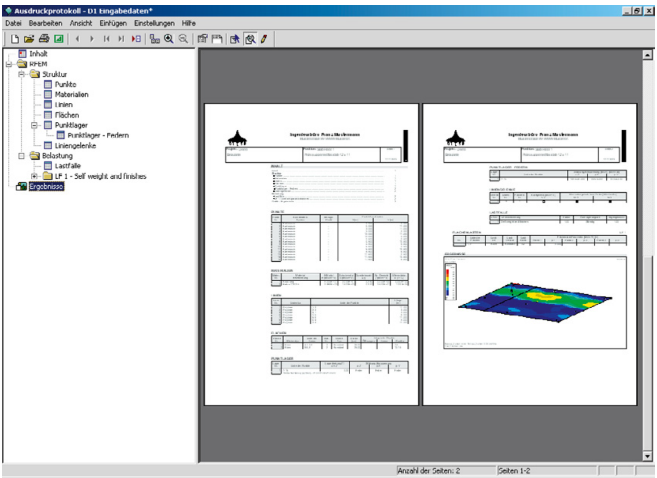
The clearest documentation of input data and results is possible by means of the printout report. Graphics can also be printed to the printout report via the **Print function** for which an icon is available in the toolbar and the option in **printout report** within the print dialog.

The printout report can be opened at any stage by clicking on the icon **Printout report**.

RFEM provides the possibility to create and organize several printout reports. You can create a short form for a paper printout, a long form for the transfer in file format and a printout report for the design.

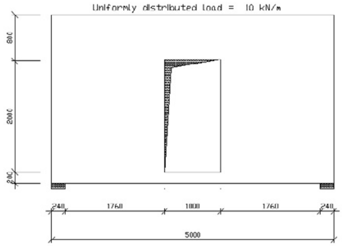
The output is structured in tree like fashion in the printout report navigator and is thus easy and quick to use.

As the output of results in the printout report isn't possible in the demo version of RFEM only the structure will be shown where graphic printout are made. The results of all results, be they numerical or graphical, can be printed either directly to the printer, in the printout report or to the clipboard in the fully licensed version of RFEM.



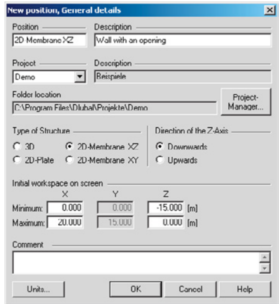
Application: Print Printout report

Create new position



Create a new position with the following details:

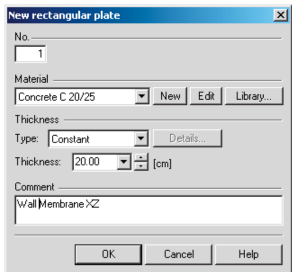
1. Create new position through a click on the icon New position.
2. Position name: „2D Membrane-XZ".
3. Description: „Wall with an opening"
4. Type of structure: „2D Membrane XZ".
5. Create new position with OK.



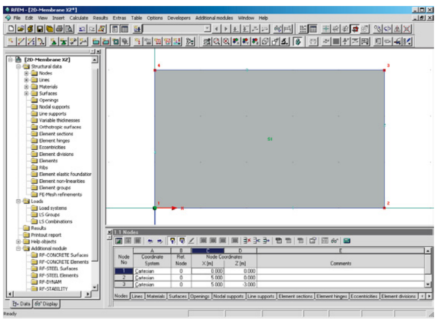
Create Wall

Create a surface in the following manner:

1. Click on the icon **New planar surface**
2. Material: „Concrete C20/25"
3. Type: „Constant"
4. Thickness: „20.00 cm".
5. Comment: „Wall"
6. Confirm surface properties with OK.
7. Move mouse pointer to the coordinates 0.00 / 0.00 / 0.00m and with a left hand click set the first corner node.
8. Move mouse pointer to the coordinates 5.00 / 0.00 / -3.00m and with a left hand click set the second corner node and with this the surface is created.
9. Exit the function with a right hand click.



Through the command **Show whole structure** the structure will be zoomed in the graphic window.



Application: New 2D Membrane structure

Insert opening

A window opening is to be inserted into the wall. The clear dimensions of the opening are B/H = 1.00 m / 2.00 m. The lower edge of the opening lies at a level 0.20 m above the lower edge of the wall and is position in the middle of the wall horizontally.

In order to enter the coordinates quickly and easily it's better to adjust the spacing of grid points on which the mouse pointer orientates to a sensible value.
To edit the grid and work plane please proceed as follows:

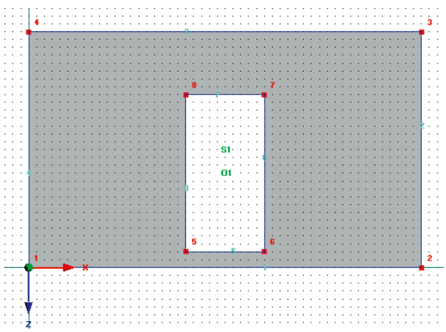
1. Select he function Work plane and grid by clicking on the respective icon in the toolbar.
2. Set the spacing between grid points to o.10 m.
3. Activate the check box for **Scale grid dynamically according to size of structure**.
4. The input is confirmed on OK.

The opening which should be created is bordered by boundary lines. These are created using the function New line. The corresponding icon is to be found in the toolbar.

The input of the border lines for the window opening is done as follows:
1. Select the function New line by clicking on the respective icon in the toolbar.
2. Click on the starting point of the boundary line. This node has the coordinates 2.00 / 0.00 / -0.20 m.
3. Click on the other corner points of the opening. The nodes have the following coordinates 3.00 / 0.00 / -0.20 ; 3.00 / 0.00 / -2.20 : and 2.00 / 0.00 / -2.20 m.
4. Click on the first node to have been set and thus close the boundary line.
5. Exit the line function with right hand mouse click.

As these nodes just created are closed and lie within the surface, an opening can be created. This is done using the Opening function, for which an icon is available in the toolbar.

To insert the opening, the following steps are required:
1. Click on the respective icon in the toolbar.
2. Aim for one of the boundary lines with the cross-hairs (all such lines just created will turn yellow as only these form a continuous boundary).
3. With a left hand click create the opening.
4. Exit the function with a right hand click.



Application: Grid and Work plane Create line Insert Opening

Edit Wall

The wall must still be edited, in order to make a change in thickness.
In order to get to know another method of editing a structure in RFEM the wall will be edited using the table. The table can be found by default at the bottom right area of the RFEM user-interface. (See Chapter 3 „Starting and operating RFEM“) and can be switched on or off via the icon provided in the toolbar.

Should structural components be currently edited in the table the corresponding row will be colored-in and in the graphic screen image the structural elements will be shown as being selected which can be detected by the change in color. In this manner good control possibilities are provided as to whether the correct element is being edited and where this is positioned in the structure.

Surfaces are stored in the table numbered 1.4 and are also to be edited there. In the table you can move form register to register either via the arrow icons F2 or F3 key or by directly clicking on the respective register name under the table. Here you get to the desired table by clicking on the register Surfaces.
The wall is the only surface, that with the number „1“.

To change the wall thickness, please proceed as follows:

1. Open the Table 1.4 Surface.
2. Set the wall thickness to 25 cm. This happens by entering the value directly using the keyboards into column E.
3. Confirm the input in the table cell by pressing the return key.

1.4 Surfaces

	A	B	C	D	E	F	G	H	I
Surface no	Surface type	List of boundary lines	Material No.	Thickness Type	Thickness [cm]	Integrated objects Openings	Lines	Nodes	Comment
1	Plane	1-4	1	Uniform	25.00	1			Wall Membrane XZ
2									
3									
4									
5									
6									

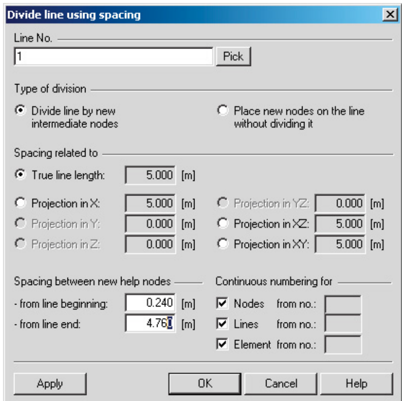
Nodes | Lines | Materials | Surfaces | Openings | Nodal supports | Line supports | Element sections | Element hinges | Eccentricities | Element divisions | Elements | Element elastic foundations

Please enter the thickness of the surface!

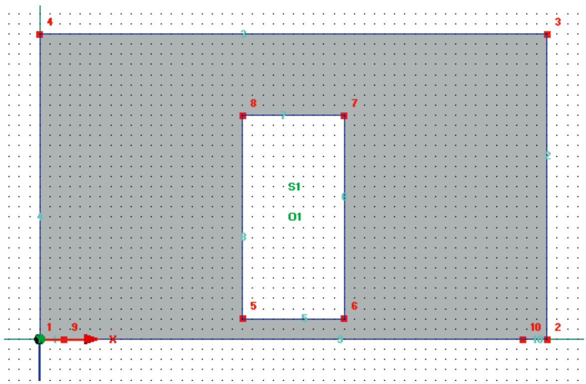
Application: Working in the table

Create Supports

The wall is to be supported over a length of 24 cm on each end of the bottom edge. In order to model this line support with the corresponding length is to be set at each position. As line supports always support whole lines and not just part of them it's necessary to divide up the line which forms the bottom edge of the wall. This is done by the following steps:



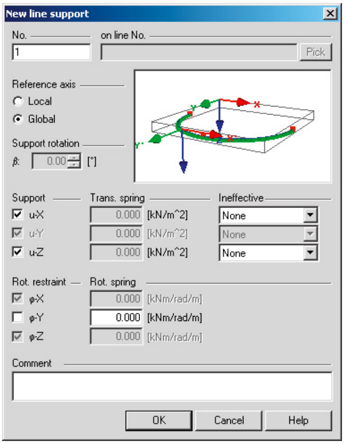
1. Move the **mouse pointer** directly over the line to be divided. This will change color correspondingly as soon as the pointer lies directly above it.
2. A **right hand click** will open the Line context menu.
3. Select the menu item **Divide lines** from the menu.
4. Select the menu item **Space** from the **Divide lines menu**.
5. Enter the value „0.240“ m in the field **Spacing between new node from line beginning**.
6. Confirm the first division through OK.
7. By clicking on a space next to the structure remove the marking from the line.
8. Repeat steps 1-4 on line 9.
9. Enter the value „0.240“ m in the field **Spacing between new node from line end**.
10. Confirm the first division through OK.
11. By clicking on a space next to the structure remove the marking from the line.



Application: Divide lines

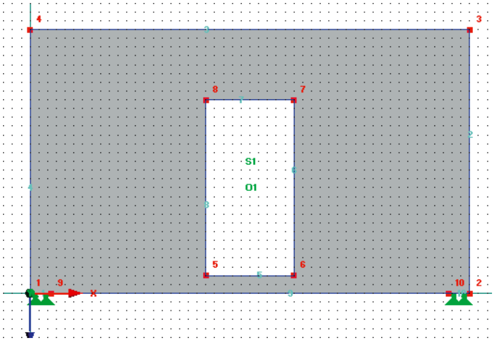
The line support will now be placed on the lines just created, each with a length of 24.00 cm. The left hand line viewed is to be held in the Z and X directions, while the other line is to be held in the Z- direction only. **The support on the left hand line is to be created in the following manner:**

1. Click on the icon New line support in the toolbar. 
2. By clicking on the button New define a new type of support.
3. Activate supports u-X and u-Z. Deactivate the rotational restraint about Y.
4. Complete the definition by clicking OK.
5. By clicking on OK select the support definition just created.
6. Move the mouse pointer on line 1 (short left hand line).
7. Place the support with a left hand mouse click.
8. Exit the function with a right hand click.



Creation of the right hand support is done in a similar manner:

1. Click on the icon **New line support** in the toolbar. 
2. By clicking on the button New define a new type of support.
3. Activate **Supports u-Z**. Deactivate the support u-X and rotational restraint about Y.
4. Complete the definition by clicking OK.
5. By clicking on OK select the support definition just created.
6. Move the mouse pointer on line 10 (short right hand line).
7. Place the support with a left hand mouse click.
8. Exit the function with a right hand click.

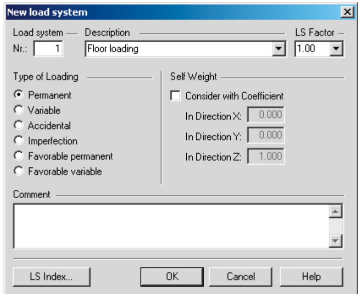


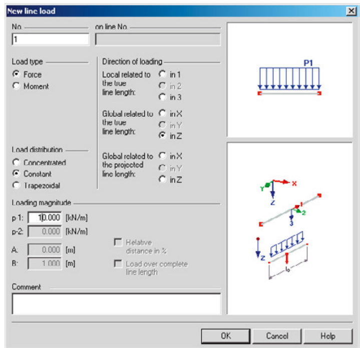
Application: Create line support 

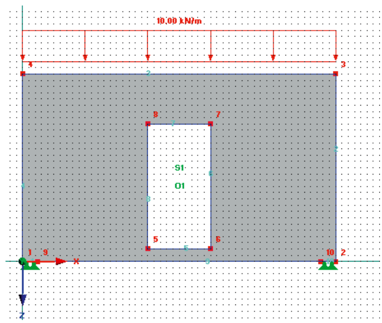
Apply loading

The wall is to be loaded through a uniform load on the upper edge of the wall. This load of 10.00 kN / m will be applied to the line which borders the wall at the top. Dead weight of the wall isn't to be considered.

1. Click on the icon **New line load** in the toolbar.
2. Enter the description **„Floor loading“** in the window **New load case**.
3. Create the load case on **OK**.
4. Enter a load magnitude **p-1** of **10.00 kN / m** in the window **New line load**.
5. Complete the load definition through **OK**.
6. Move mouse pointer on to line number 3 (upper line)
7. With a **left hand click** place the line load on the line.
8. Exit the line load function with a **right hand click**.





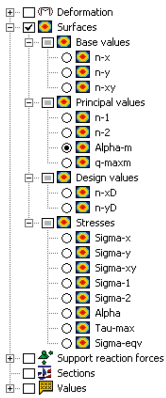


Calculation and results

First of the **plausibility check** should be carried out by clicking on the corresponding icon in the toolbar. The plausibility check is a tool which checks the whole structure. In this manner incomplete or missing input regarding element definition or errors in the geometry can be detected and reported. The plausibility check should always be carried out before starting calculation and also from time to time during the input of data in order to help create a correct set of data.

In order to check the FE-Mesh this must firstly be generated under the command **Generate FE-Mesh** in the Menu Calculate. It can be seen that the default element size of 0.50 m is too large; therefore the mesh will be refined.

1. Open the menu **Calculate**.
2. Select the menu item **Define FE-Mesh**.
3. Enter the value **0.10 m** in the box denoted **Target FE-element side length**.
4. Confirm the input with OK.

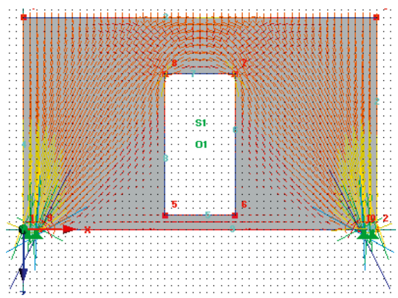


The wall results can now be calculated by carrying out the following steps:

1. Open menu **Calculate**.
2. Select the menu item **Calculate all** form the menu.

The results output can be regulated via the register **Results** in the navigator.

Selected here is the output of the principal membrane forces including their directional trajectories, done by activating the output of Alpha-m in the results navigator in the menu tree Surfaces.
Here the compression and tension zone can be clearly identified (Tensile zones are marked in red).
Singularities, which arise at the supports, are also clearly recognisable (Blue trajectories).

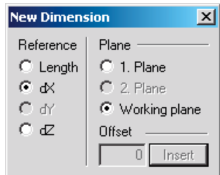


Application: Plausibility check   Define FE-Mesh   Results display

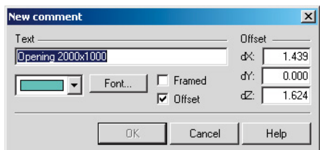
Dimensions and comments

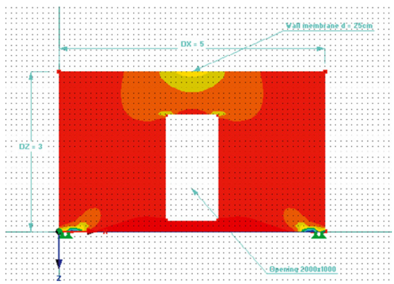
In order to configure the structure in a more informative manner it is often sensible to include dimensions. Insertion of dimensions is done via the **New dimension** icon in the toolbar. A window named **New dimension** will open in which the dimension reference direction and plane are entered. The possibility exists to account for an offset regarding the spacing between dimensioning lines and the structure.

In the section Plane, the user can decide in which plane the dimensions should be displayed. In the input box offset the distance between the dimensioning line and the line measured can be stipulated. After having marked the start and end nodes of the lien to be dimensioned the dimensioned can then be generated by clicking on **Insert**. Alternatively, the position of the dimension line can be determined by a third click of the mouse. The function can be exited with a right hand click of the mouse. Should any of the nodes be moved at a later stage then the dimension will change automatically.

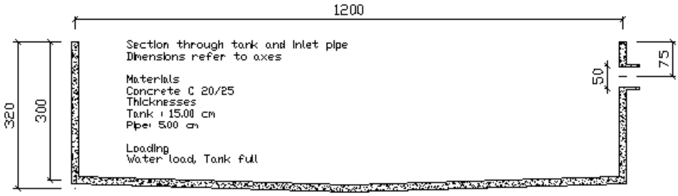


Comments can also be inserted into the graphics with a function sharing the same name. After clicking on the icon **New comment** the dialog named New comment will appear. In order to insert the comment it is necessary that a node is clicked on with the mouse. The comment is entered in the section denoted by Text. Color and font for the text can also be defined. Under **Offset** the distance between the node and the comment text can be stipulated. After having marked a node, the comment can be inserted on **OK**.The position of the new comment can also be determined by a second mouse click. The function is exited by a **click** on the right hand mouse button





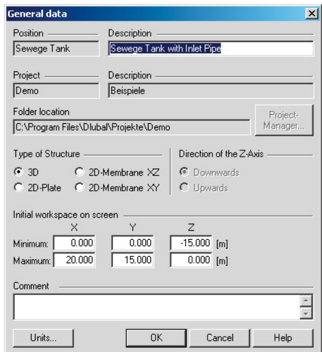
Application: Dimensions Comments



Create new position

Create a new position with the following details:

1. Create new position through a click on the icon New position.
2. Position name: „Sedimentation Tank“.
3. Description: „ Sedimentation Tank with Inlet Pipe“
4. Type of structure: „3D“.
5. Create new position with OK.



Generating the tank

The editing of structures in RFEM, using the tables will be done more intensively in this example. By default the table is positioned directly beneath the graphic portion of the screen. Please read more in Chapter 3 Starting and operating RFEM.

You navigate through the table by means of the arrow buttons or by clicking directly on the respective register names beneath the table.

Structural and loading data can both be entered in the table and results may be viewed. The individual table may be activated simply by clicking on the respective icons, which themselves become active when input is present in the respective tables.

Application: Navigate through the table

Setting nodes

Table 1.1 is obtained by clicking directly on the register tab or by moving through the tables with the arrow buttons

By a mouse click in the first column and row of the table the input of nodes can begin. The coordinate system, to which the respective node is related, can be defined in the first column, Column A. The Cartesian system, which is the default setting, is to be retained. By clicking on the **Return key** the entry will be confirm and the next cell will be automatically selected. In the second column of the **Nodes table** the reference node is selected, to which the coordinate refer. The default value of „0", which refers to the origin, should be confirmed with the **return key**. In the following coordinate fields the node coordinates X=0.0 m, Y=6.0 m, Z=-3.0 m are to be entered. **The node input can be done briefly as follows:**

1. Select **table 1.1 Nodes**.
2. Click on **Row 1**, **Column A** with the mouse.
3. Confirm the coordinate system **Cartesian** with the **return key**.
4. Confirm the **reference node „0"** (Coordinate system origin) with the **return key**.
5. Confirm the **X-coordinate „0.00"** with the **return key**.
6. Enter the **Y-coordinate „6.00"** and confirm with the **return key**.
7. Enter the **Z-coordinate „-3.00"** and confirm with the **return key**.

On confirmation of the Z-coordinate the node is defined. The X-coordinate input field of the next node becomes active and the input of columns A and B automatically retained as these settings are rarely changed. **The generation of the remaining nodes can be complete through direct coordinate input as follows:**

8. Confirm the **X-coordinate „0.00"** of node 2 with the **return key**.
9. Enter the **Y-coordinate „6.00"** and confirm with the **return key**.
10. Enter the **Z-coordinate „0.00"** and confirm with the **return key**.
11. Confirm the **X-coordinate „0.00"** of node 3 with the **return key**.
12. Enter the **Y-coordinate „6.00"** and confirm with the **return key**.
13. Enter the **Z-coordinate „0.00"** and confirm with the **return key**.
14. Enter the **Z-coordinate „0.20"** and confirm with the **return key**.

The input of boundary line for the sewage tank is now complete.

Application: Enter nodes in the table Nodes

Table 1.1 disposes now of the following entries:

1.1 Nodes

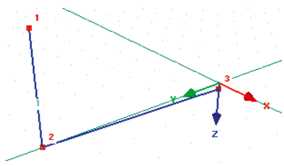
Node No	A Coordinate System	B Ref. Node	C Node Coordinates X [m]	D Y [m]	E Z [m]	F Comments
1	Cartesian	0	0.000	6.000	-3.000	
2	Cartesian	0	0.000	6.000	0.000	
3	Cartesian	0	0.000	0.000	0.200	
4	Cartesian	0				
5						
6						

Nodes | Lines | Materials | Surfaces | Openings | Nodal supports | Line supports | Surface elastic foundations | Line hinges | Element sections

Please enter the coordinate!

Line definition

The definition of lines is done in Table 1.2 of the structural data tabular input. This is activated by clicking either on the register Lines beneath or by using the arrow buttons. As the line is defined by the recently entered nodes numbered 1, 2 and 3 (as seen in the picture above) then the text „1-3" should thus be entered in column B List of nodes per keyboard and confirmed with the return key. The length of the line will be automatically determined in the program and entered in the table in column C (see picture below). After completing the input the line is automatically generated in the graphics screen. (See picture, right)



1.2 Lines

Line no.	A Line type	B List of nodes	C Length [m]	D Comment
1	Polyline	1-3	9.003	
2				
3				
4				
5				
6				

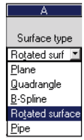
Nodes | Lines | Materials | Surfaces | Openings | Nodal supports | Line supports | Surface elastic foundations | Line hinges | Element sections

Application: Define line in the table Lines

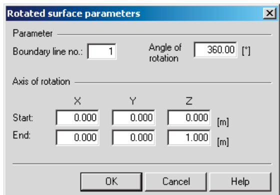
Create Rotated Surface

As the sewage tank is symmetric, the generation, of which, is very simple. The boundary line has already been created. Only the rotation must still be defined. This is done as follows:

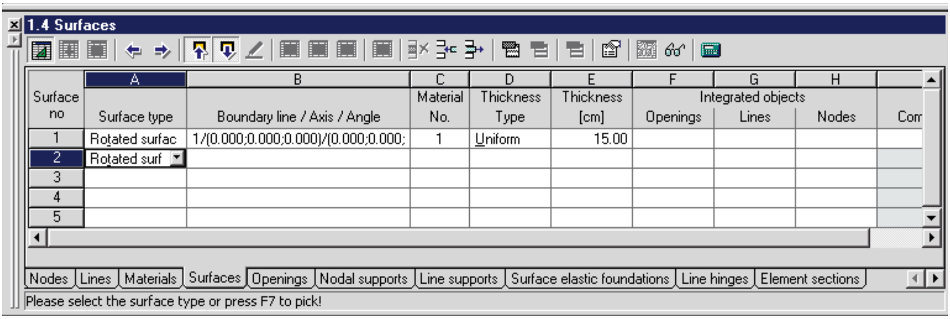
1. Activate the table 1.4 by clicking on the register Surfaces or by using the arrow buttons.
2. Select the surface type Rotated surface from the list box, which opens up after clicking on the list box button.
3. Confirm the selection with the return key.
4. The field boundary line/ Axis/ Angle will be activated automatically. The window in which the input of the rotation parameters is made opens by clicking on the (…) button.



5. Enter the line number „1“ in the input box Boundary line no.
6. A mouse click in the corresponding box enables input of the angle of rotation.
7. Enter the angle of rotation of „360°“.
8. As the axis of rotation is vertical this can be clearly defined through the points 0 / 0 / 0 and 0 / 0 / 1 such that only the value of „1“ m for the Z-ordinate of the 2nd point must be entered.
9. By clicking on OK the input parameters will be entered in the table and confirmed with the return key.
10. In the column Material the material 1 - Concrete C20/25 will be selected and confirmed by return.
11. The thickness type „Uniform“ is confirmed by return.
12. The value of „15.00“ cm is to be entered in the column Thickness and confirmed with the return key.

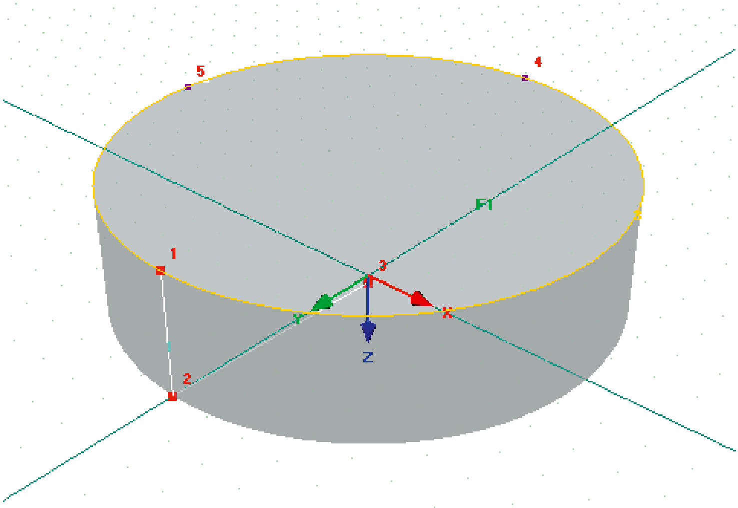


Application: Generate a rotated surface in the table Surfaces



Surface no	A Surface type	B Boundary line / Axis / Angle	C Material No.	D Thickness Type	E Thickness [cm]	F Integrated objects Openings	G Lines	H Nodes	Com
1	Rotated surfac	1/(0.000;0.000;0.000)/(0.000;0.000;	1	Uniform	15.00				
2	Rotated surf								
3									
4									
5									

The input of the rotated surface data is now complete. The input is clearly identifiable in the picture above. After the successful input of the rotated surface data the generated surface will be shown on the screen, such that for a solid model the structure should appear as displayed in the picture below.



Regarding the set up of the graphical output, in this case the solid model view of surfaces, see **Chapter 11 RFEM functions** in the section **the register Display in the navigator**.

Generating the inlet pipe

Setting nodes

In order to generate the nodes, which define the end points of the middle axis of the inlet pipe, the following steps should be undertaken.

1. Select the **Table 1.1 Nodes**.
2. Click with the mouse in the first available cell **of column A**, **row 6** in this case.
3. Confirm **Cartesian** coordinate system through the **return key**.
4. Confirm **reference node „0"** through the return key.
5. Enter the **X-coordinate „0.000"** of **node 6** and confirm with **return**.
6. Enter the **Y-coordinate „-6.500"** with the keyboard and confirm with **return**.
7. Enter the **Z-coordinate „-2.250"** with the keyboard and confirm with **return**.
8. Enter the **X-coordinate „0.000"** of **node 7** and confirm with **return**.
9. Enter the **Y-coordinate „-5.500"** with the keyboard and confirm with **return**.
10. Enter the **Z-coordinate „-2.250"** with the keyboard and confirm with **return**.

Define line

The middle line of the inlet pipe is to be generated with the help of the two nodes just created.

1. Select the table **1.2 Lines**.
2. Click with the mouse in the first available cell of **column A**, **row 4** in this case.
3. Select the line type **Polyline** from the list box and confirm with **return**.
4. Enter **„6-7"** in the column **List of nodes** and confirm with **return**.

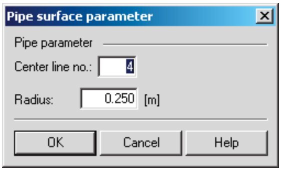
Generate Pipe

Through the line just generated we now have the centre line of the inlet pipe.
In order to generate the pipe itself, please proceed as follows:

1. Select the table 1.4 Surfaces.
2. Click with the mouse in the first available cell of column A, row 2 in this case.
3. Select surface type „Pipe" from the list box and confirm with return.
4. The field Center line/ Radius will be activated automatically. The window in which the input of the pipe surface parameters is made opens by clicking on the (…) button.

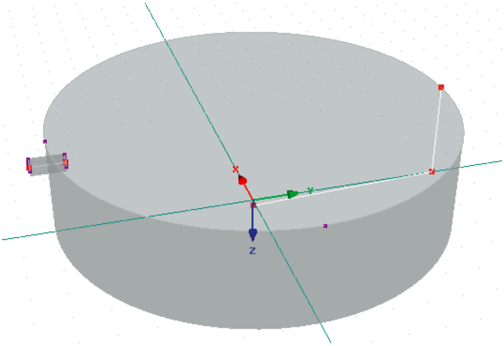
Application: Generate a pipe in the table Surfaces

5. Enter a value of „4“ in the box marked Center line no.
6. The input of the radius is enabled by clicking in the corresponding box.
7. Enter a value of „0.25“ m.
8. Clicking on OK confirms the input which is then adopted by the table and confirmed through OK.



9. In the column Material the Material 1-Concrete C20/25 is to be selected and confirmed through return.
10. Confirm the thickness type „Uniform“ with return.
11. Enter a value of „5.00“ for the thickness and confirm with return.

The input of the pipe data is therefore complete and the pipe surface will be displayed on screen. The structure appears as shown here.



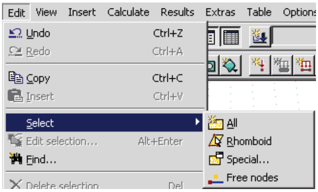
Surface intersections

In order to restrict the pipe to the outside of the tank and create an opening in the tank wall the surfaces must be intersected. This means the intersection line between both surfaces will be calculated in the program and if closed surface types are present the respective surfaces will be subdivided in to partial surfaces.

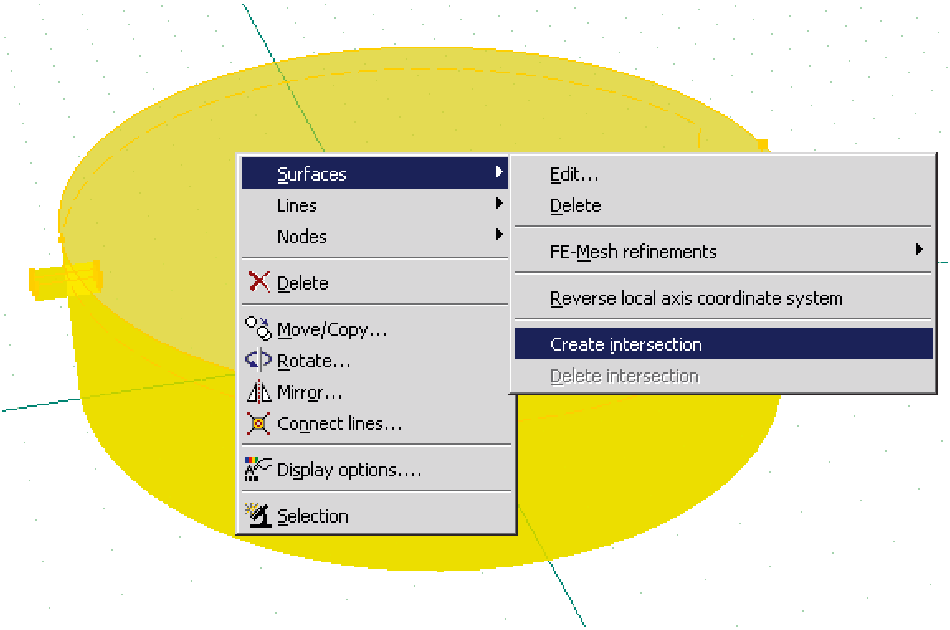
In order to generate the intersection, it's necessary to mark the intersecting elements. As the surfaces to intersect are also the only surfaces in the structure they can be quickly marked through the command Select all from the menu as follows:

1. Select the menu item Edit form the menu bar.
2. Choose the menu item Select.
3. From the sub menu click on the menu item All.

Hereby all elements of the structure are marked and are shown as being selected in the graphic screen due to the color change.



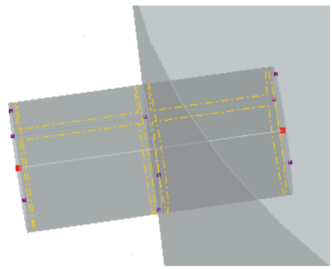
Application: Select all



In order to generate the intersection of the two surfaces, please proceed as follows:

1. Move the pointer on to a part of the marked structure.
2. Click on the structure with the right hand mouse button in order to open the context menu.
3. Open the Surface context menu by clicking on the menu item Surface.
4. Select by mouse click the menu item Generate Intersection in the surface context menu.

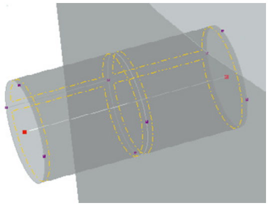
The program now calculates the intersection line and sub-divides the surfaces if necessary. The successful subdivision of the surfaces can be recognised by the pre-selection (Mouse pointer above the surface), as can be see in the picture to the right.



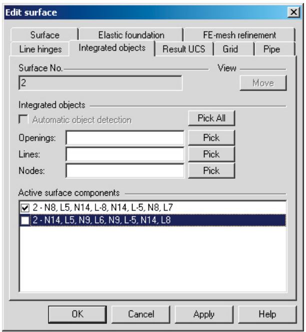
Removing surface components

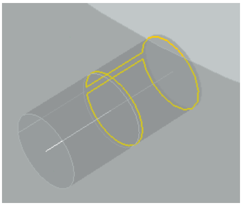
The surface components created through the intersection (Pipe inside the tank, Pipe outside the tank, Tank surface within the pipe diameter and tank surface outside of the pipe diameter) can be deactivated. This means that the surface components are removed from the calculation and can therefore neither contribute to loading, be it external loading or self weight, nor to structural stiffness and resulting distribution of forces and moments.

The portion of the pipe, which lies within the tank, must be deactivated at first.

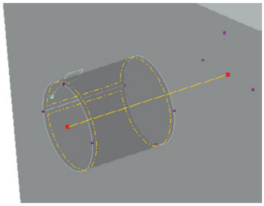


1. Move the pointer over the surface, such that this is pre-selected, which can be recognised by the dashed marking line (see picture, right).
2. The surface context menu is now opened by a right hand mouse click while the portion is pre-selected.
3. Select the menu item Edit Surface in order to open the window of the same name. All surface properties can be edited in this window.
4. Select, by mouse click, the register Integrated objects within the Edit surface window.
5. The available surface components are listed in the lower part of the Integrated objects register. The active surface components can be recognised by a check button (see picture, right)
6. Is a surface portion marked by mouse click then this will be highlighted in the screen graphics.
7. In order to deactivate the pipe surface within the tank, the tick in the corresponding check box is to be removed by mouse click.
8. The deactivation of the surface component is confirmed with OK.

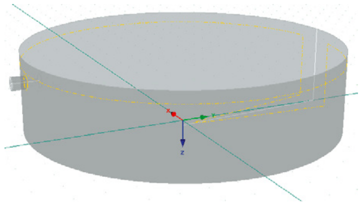




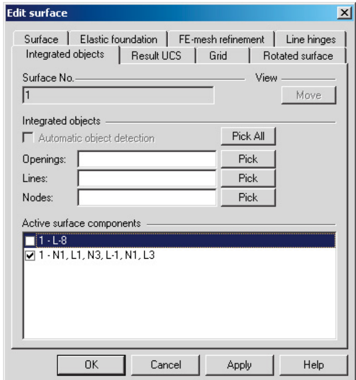
The deactivation is to be checked graphically, as deactivated components are blended out.



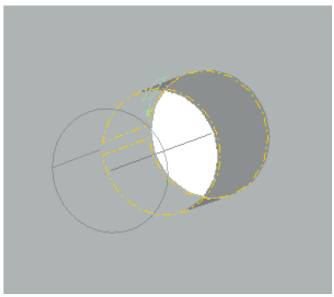
The deactivation of the tank surface within the pipe diameter is done analogously to the steps previously described.



1. Move the pointer over the tank surface, such that this is pre-selected, which can be recognised by the dashed marking line (see picture, right).
2. The surface context menu is now opened by a right hand mouse click while the portion is pre-selected.
3. Select the menu item Edit Surface in order to open the window of the same name. All surface properties can be edited in this window.
4. Select, by mouse click, the register Integrated objects within the Edit surface window.
5. By clicking on the corresponding row the surface component within the pipe is to be marked so that this is highlighted graphically in the screen graphics.
6. In order to deactivate the tank surface within the pipe, the tick in the corresponding check box is to be removed by mouse click.
7. The deactivation of the surface component is confirmed with OK.



The now completed deactivation is to be checked in the screen graphics, as deactivated surfaces are blended out. By selecting a certain viewing perspective (sees Chapter 11, RFEM functions, section Views) is the opening, created by the deactivation of the surface components, clearly visible. (See picture, right). This opening will be automatically adjusted by a change in geometry, e.g. pipe diameter.

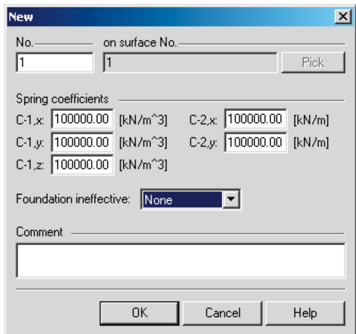


Application: Deactivate surface components generated through intersection

Generate elastic foundation

The tank is to be bedded elastically in order to model the soil contact.
The procedure in creating an elastic foundation is done as follows.

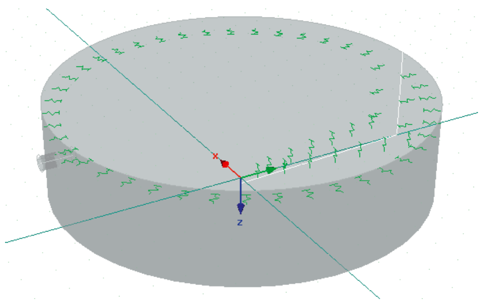
1. Pre-select the tank surface by holding the mouse pointer over the surface which results in the dashed line being highlighted.
2. Open the surface context menu with a right click on the surface.
3. Select the menu item Edit surface.
4. Select the register Elastic foundation in the window.
5. Activate the check button for the elastic foundation.
6. Open the window New Elastic Foundation with a mouse click on the button New.
7. From published tables for the soil group GE the value 100000 kN/m³ is read in. The value „100000" kN/m³ is to be entered in the Spring coefficient box C-1, z.
8. Enter the values 100000 kN/m³ in the plane of the base slab(C-1, x and C-1, y), i.e. friction.
9. The values C-2, x and C-2, y represent the shear resistance of the soil in both surface directions. These will be set to „10000" kN/m
10. The elastic foundation will be placed by clicking the OK button.



The criterion for foundation ineffectiveness can be selected from the list box of the same name.

For more on elastic foundations, see RFEM Manual-Help Chapter 3 in the Section Elastic Foundations.

With this input the structural model is now complete. Now the loading must be applied.

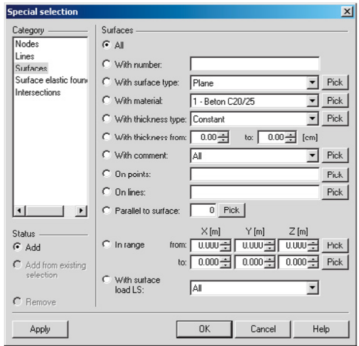


Application: Generate elastic foundation

Load application

The tank is to be loaded by a hydrostatic load. The water load is to be applied to both the inlet pipe as well as the tank wall. In order to have to define the load just once, both surfaces must be selected and the load applied to both simultaneously. Please proceed as follows in order to select both surfaces.

1. Open the Edit menu by a **click** on **Edit** in the menu bar.
2. Open the sub menu **Select**.
3. Select the menu item **Special**.



In the window which now opens all structural components may be selected according to type or geometry.

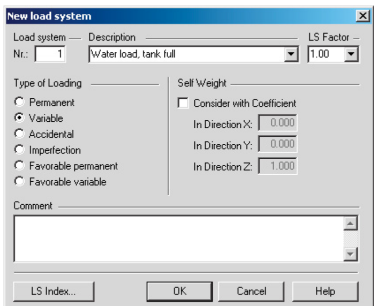
4. Select the category **Surfaces**.
5. Select the option **All** in the surface selection.
6. Confirm selection by click on **OK**.

The selection of both surfaces can be recognised by the yellow coloring. Loading can now be applied to both of these surfaces. **The following steps must now be taken.**

1. Click on the **New surface load** icon in the toolbar. 

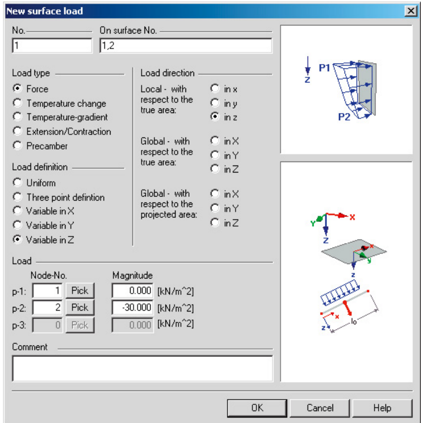
In order generate the loading a load case must always be available, but, as no load case has been created till now the window **New load case** will open automatically in order to allow a load case to be defined.

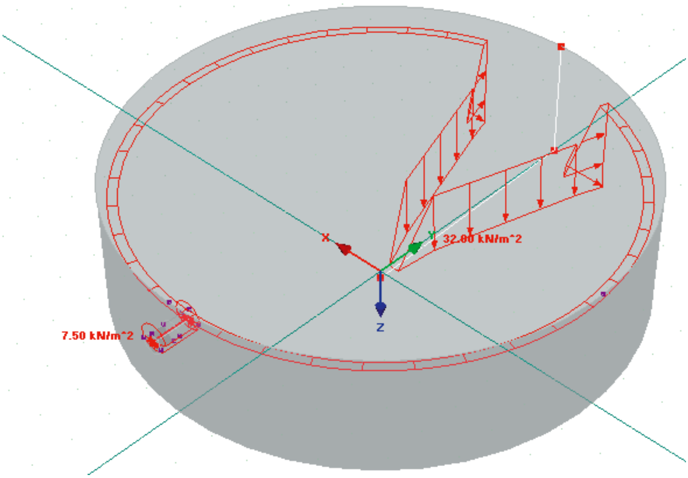
2. Enter **„Water load, Tank full"** as description.
3. Select loading type **Variable**.
4. Disregard **self weight** by deactivating the check button named Consider with coefficient.
5. By clicking on **OK** the load case will be laid on, which means that all following load input refers to this load case only.



The window **New surface load** now opens. In the dialog the surfaces to load have already been entered. The following loading properties have yet to be determined.

6. Set loading type to Force.
7. Set Load direction to Local with respect to the true area in z.
8. Set Load definition to Variable in Z in order to allow the hydrostatic load to be applied.
9. For the load magnitude p-1 enter the node number „1“.
10. Enter a load magnitude of „0.00“ kN/m^2 for load p-1.
11. For the load magnitude p-1 enter the node number „2“.
12. Enter a load magnitude of „-30.00“ kN/m^2 for load p-2.
13. With a click on the OK button the load will be applied and display on the screen graphics.



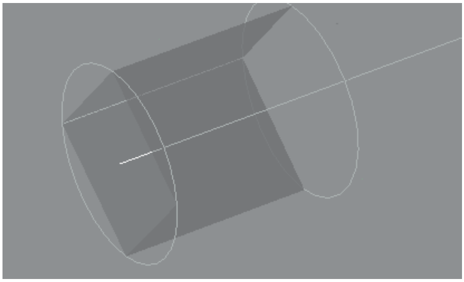


Application: Entering a variable surface load.

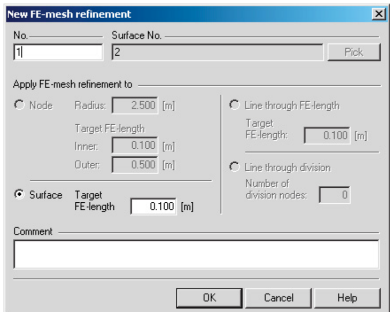
Calculation and results

Before calculation the FE-Mesh should always be checked in order to adjust it to suit the structure if necessary. The FE-Mesh can be generated and display through selection of the menu item **Generate FE-Mesh** in the Menu Calculate.

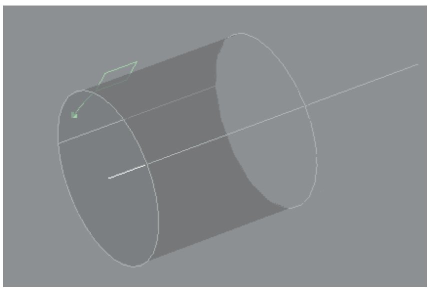
It should be noted that the approximation for inlet pipe is poor with the default FE-Element length of 0.50 m. (see picture, right) To avoid placing a much too fine a mesh over the complete structure, which leads to very long calculation duration and in other parts of the structure achieves no noticeable advantage, only the element size of the inlet pipe is to be changed.



1. Open the menu **Insert** from the menu bar.
2. Open the sub menu **Structural data**.
3. Select the menu item **1.21 FE-Mesh refinements**.
4. Select the input option **Dialog**.
5. Apply FE-Mesh refinement to **Surface** by activating the corresponding button.
6. Enter surface number **„2“** in the respective box or select the pipe in the graphics by clicking on it.
7. Enter the target FE-Length in the corresponding box. **„0.10 m“** in this case.
8. By a click on the **OK button** the mesh refinement is set.



After regenerating the mesh the refinement can be verified. It can be clearly seen that the pipe is now much better modelled and that the surrounding mesh of the tank surface adjusts automatically in the transition zone. (See picture, right)

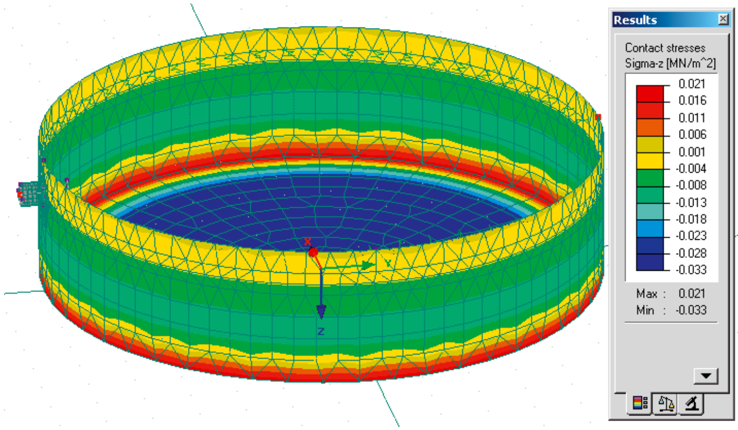


Application: Generate FE-Mesh refinement

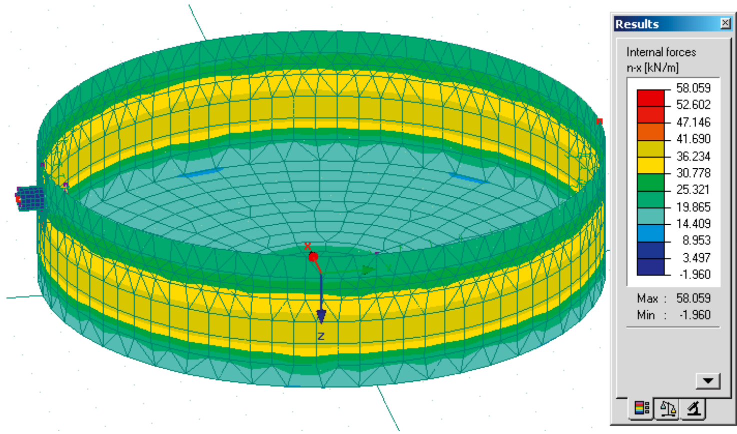
In order to view the results, the structure must first be calculated. This is done with the  command Calculate all in the menu Calculate of the menu bar or by activating the Results icon by mouse click. If no results are available for the current load case then these will be calculated and subsequently displayed on screen.

Displayed here are the contact stresses, i.e. the stresses in the elastic foundation, and membrane forces n-x, i.e. tangential forces.

Contact stresses

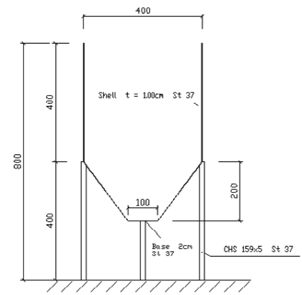


Axial Membrane forces



The Structure

The structure is already available on the CD as are the other structures. In this chapter the modelling of the available structure should become understood.
Please open the file **Silo.rfe**. The complete steel silo is shown in the screen graphics.
As RFEM provides many opportunities in entering data, as seen in the previous examples, only the generated results will be shown. The manner in which the user arrives at this point is unimportant.



Generate the boundary line of the silo shell

A **polyline** with the nodes 0.00/-2.00/-8.00 m, 0.00/-2.00/-4.00 and 0.00/-0.50/-2.00m is to be generated. This polyline will form the rotated shell at a later stage.

Generate column line

Generate a **line** with the end node coordinates 0.00/-2.00/-4.00m and 0.00/-2.00/0.00m.

Generate silo shell

Generate a **rotated surface** of steel grade St 37 with the thickness 1.00 cm, the boundary line number „1“, an angle of rotation of 360° and the z-axis as axis of rotation.

Generate silo base

Generate a plane surface of steel grade St 37, thickness 2cm and line 3 as boundary line.

Define element properties and generate columns

A new element will be generated from a circular hollow section RO 159x5 of steel grade St 37by clicking on the icon New element and subsequently assigned to line no. 2.

Create column

By selecting the element and the command Rotate e.g. by clicking on the corresponding icon in the toolbar the three remaining columns will be generated. The number of copies is to be set to 3 and the angle of rotation is 90° about the z-axis.

Define column support

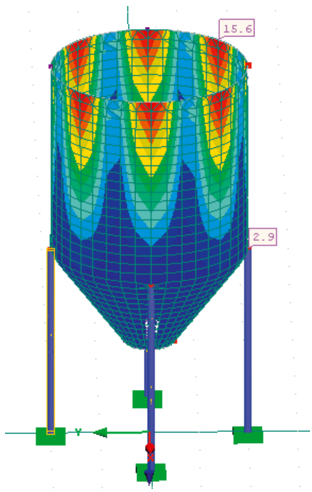
The nodes of the column bases (nodes 4, 10, 12 and 14) will be assigned to nodal supports with the characteristic „Rigid“

Apply loads

A load case is to be generated which contains the effects of the dead weight of the structure as well as a hydrostatic load on both surfaces. This load is to be applied as a surface load in the local z-direction of the surface. Set the load type to **variable** and the load magnitude is to be set such that the load varies from 0.00kN/m², at the top edge, to 60.00kN/m², at the base.
For loading in local directions it is important that both surfaces have the same orientation (in this case, outwards).
The local coordinate system of the surface should be checked by activating the display of the system via the navigator, register **Display** (see also chapter 11) under **Structure → Surfaces → → local surface coordinate system**. The direction reversal of the coordinate system is achieved with a right hand mouse click on the respective surface and selecting the command **Reverse local surface coordinate system**.

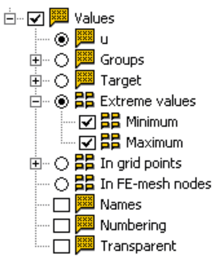
Calculation and results

In order that the FE-Mesh fit better to the structure the target FE-Element length is to be reduced to 0.20 m, before the results are calculated. After calculation the deformed shape will be automatically displayed.



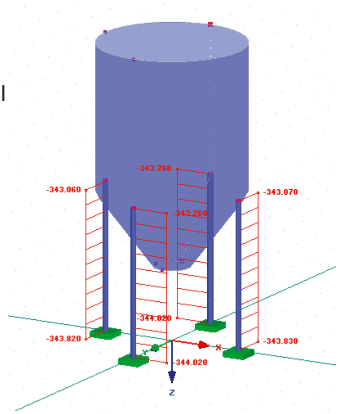
Numerical output on surfaces

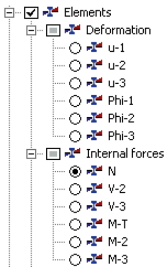
It is sometimes sensible to display numerical results alongside the graphical display or even on their own. Shown in the picture to the right is the deformation with numerical display of the max/min values. The output of numerical results on screen graphics is controlled in the Results register in the navigator (see below, right). It's possible to show values orientated in their direction of effect, e.g. reinforcement values in the direction of reinforcement, principal stresses in the respective directions in order to allow an immediate optical connection.



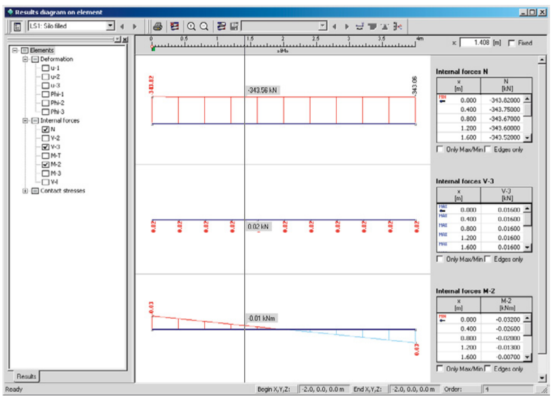
Results diagrams on Elements

The silo structure is a mixed structure consisting of 2D surface elements and 1D line elements. The results of the 1D line elements can also be displayed in the screen graphics. The control of the element results display is done also in the navigator under the register **Results**. (see picture, right) The display can be selected to show element results only or also in combination with surface results. In the picture to the right are axial forces in the columns together with the n-y surface results.



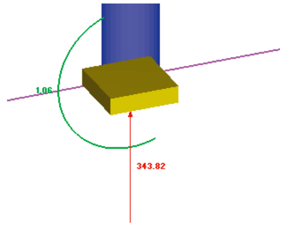


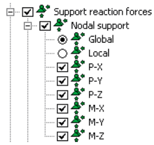
The results of individual elements can also be displayed and evaluated in a separate window. For this, one single element or several elements must be selected and then select the menu item **Results diagrams on selected elements** in the results menu. A window opens in which the deformation, forces, moments and contact stresses (where applicable) can be displayed. Here, a silo column has been selected and the axial and shear forces and moment activated.



Support reactions

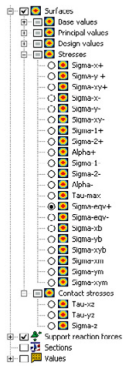
The support reactions can either be displayed on their own or together with other results, as for the elements. The control is done in the navigator in the register **Results** (see picture below, right). In the picture is a silo column support with its reactions.

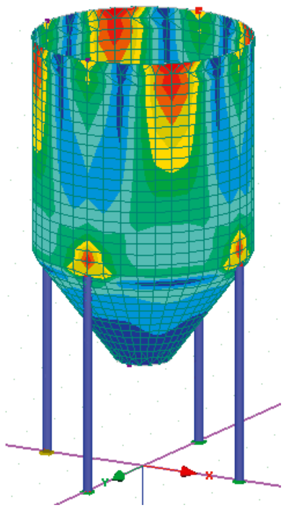


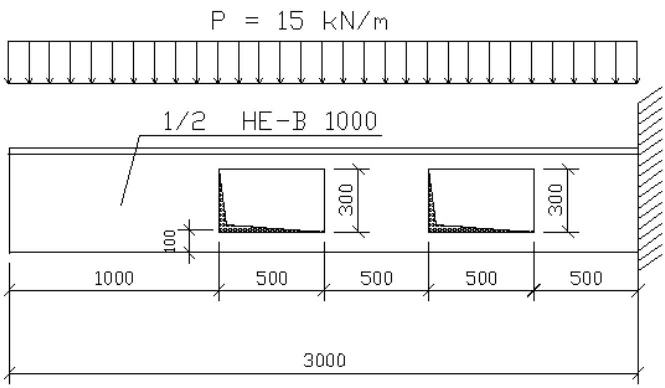


Additional Module RF-Steel

With the additional modules **RF-Steel Elements** and **RF-Steel Surfaces** comfortable complete stress analyses for element and surfaces can be carried out (see Chapter 8 also). Furthermore all stresses can be displayed for surfaces. This can be selected under **Surfaces → Stresses** in the register Results in the navigator (see picture, right) Here, the equivalent stresses Sigma-eqv+ are displayed. The „+“ means that the stress refers to the positive side of the surface, i.e. the side to which the local surface coordinate system z-axis points. Clear to see are the stress peaks above the columns. These are critical points and can be reinforced in the model with additional stiffeners, causing a reduction in stress. Furthermore, it's possible to display stresses separately according to their origin. Displayed could be membrane stresses due to axial forces in the shell, bending stresses in the shell and shear stress resulting from shear force in the shell. Membrane stresses are labelled through the added m in the stress designation, bending stresses through the added b. Shear stress due to shear force is the stress designated Tau-max.







Create new position

Create a new position with the following details:

1. Lay on new position with a click on the button New position.
2. Position name: „T-Beam"
3. Description: „ Calculation comparison Elements-Surfaces"
4. Type of structure :"3D"
5. With (OK) create the new position.

Generate Elements

2 Elements will be generated through the following steps:

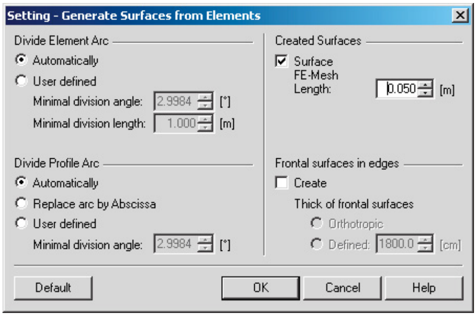
1. Create a line with the end nodes 0 / 0 / 0 and 3 / 0 / 0.
2. Define an element on this line with an 1/2 HE-B 1000 section. This section is to be found in the section library under the rolled T-sections
3. Mark element and create a copy displaced by 2m in the Y-direction

Convert element to surfaces

The element lying on the x-axis should now be converted into surface elements. RFEM provides an automatic function for conversions. This can be called upon via the element context menu or from the menu. Through the menu item **Extras → Generate surfaces from elements → → Options**, the user can define the parameters with which the conversion should take place. During conversion the surface elements, into which the element is dissected, are to be created with an FE-Mesh refinement of 5cm side length.

Application: New element

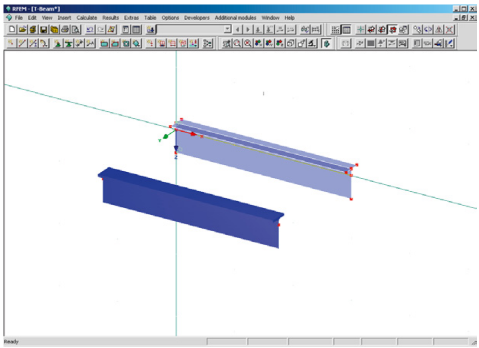
1. Open menu item **Extras → Generate Surfaces from Elements → Options**. 
2. Activate the **check button** for a target FE-length and set the target length to 0.050m.



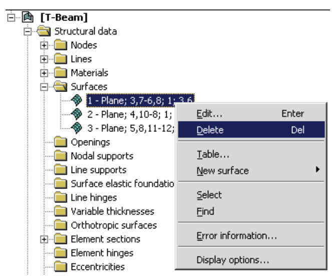
Now, all element conversions with these predefined settings will be undertaken until the settings are re-edited. The conversion is done as follows:

1. Right click on the element, which lies on the x-axis (Element no. 1)
2. Select the menu item **Generate surfaces from elements**. 

Through this command the element will be converted into individual surface elements. As the dummy element is no longer required it may be removed. The line which formed originally the longitudinal axis of the element is no longer required either and can also be deleted. It should appear roughly like the following picture.



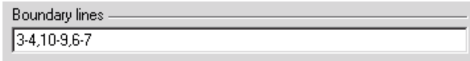
The structure consists of a 1D line element and three surfaces as the upper flange in converted into two surfaces. As RFEM demo restriction this structure can't be calculated with the demo version. The demo restriction prevents the start of the calculation for structures with more than 12 elements or two surfaces. In order to calculate the structure in the demo version we need to use a couple of small tricks. The two surfaces of the upper flange can be reduced to just one surface. The following steps are necessary:



1. Open the navigator branch **Structure → Surfaces** in the Data register.
2. Right click on surface no.1
3. Select the menu item **Delete** in order to remove the surface (see picture).

Application: Generate Surfaces from Elements - Options  - Generate 

4. Right click on surface no. 2.
5. Select the menu item Edit in order to change the boundary lines.
6. Delete the existing numbers of the boundary lines and replace through the line list 3-4, 10-9, 6-7. Please note that continuous line numbering is entered by a dash and this also includes descending number series.

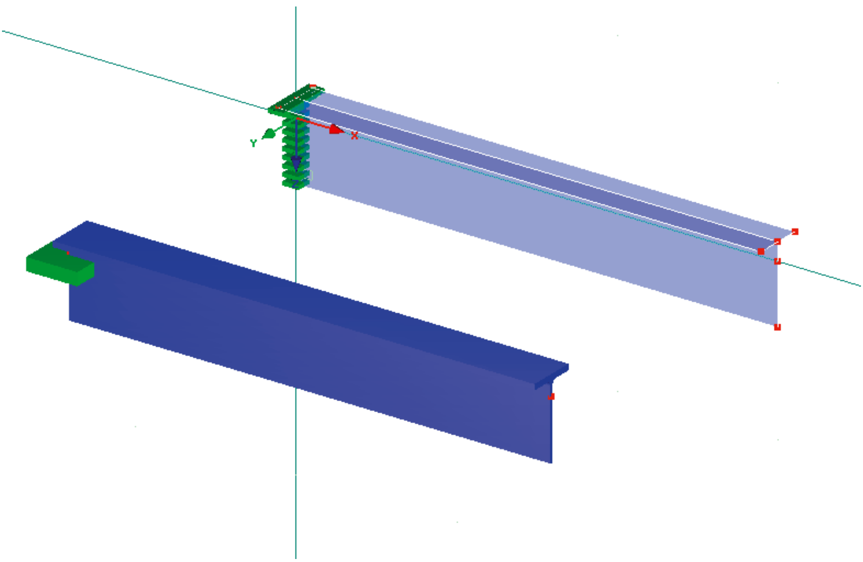


Through these steps the surfaces with the numbers 1 and 2 have been melted into the changed surface no. 2 and the structure can now be calculated with the demo version of RFEM. The FE-Mesh refinement, created by the initial conversion, has been retained.

Place supports

Both beams are to be rigidly supported at X=0.00 m. Nodal and linear supports are to be set.

1. Define a **nodal support** of type **Rigid** on node 3 (Coordinates 0 / 2 / 0). 
2. Define a **line support** of type **Rigid** on lines 3 and 4 (Flange) and on line 5 (Web). 



Apply loading

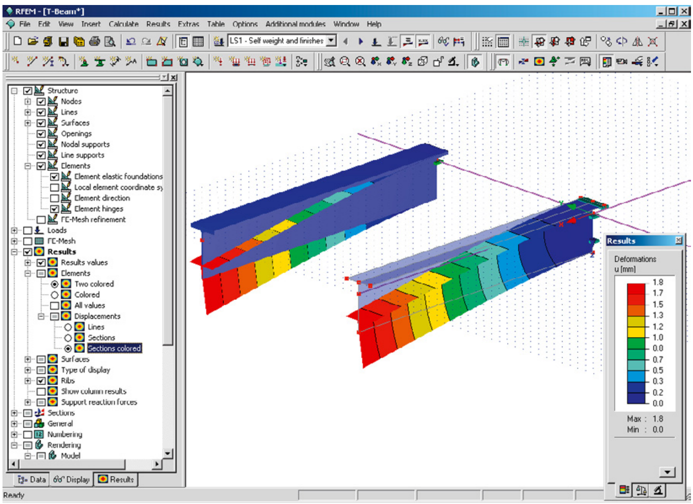
The self weight of the beams is to be considered as well as a uniformly distributed load of 15kN/m.

1. Create load case with the description „Self weight and finishes“, Type of loading „permanent“ and self weight with factor 1.00 in the Z-direction.
2. Place an element load of 15kN/m in the Z-direction on element no.2.
3. Place a line load of 15kN/m in the Z-direction on line no. 8 (Upper edge of the web).

Comparison of results

As the structure, loading and FE-Mesh have all been determined the results of the structure, consisting of the two beams, can now be calculated. By selecting the menu item Calculate → Calculate all the structure will be calculated and the deformation displayed. In order to compare the deformation of both beams more easily, the display of 1D line elements deformation can be done with cross section and in color.
This form of display can be activated in the Display register through the option **Results → Elements → Deformation → Sections colored**.

It can now be seen that the results determined by modelling of a 1D line element are identical to those of the FE Model.

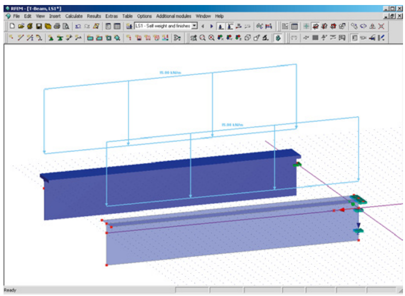


Generate opening in the web

The greatest advantage of being able to convert elements in to surfaces and therefore compute according to finite element methods is that there are practically no limits to further editing possibilities. Connection details, stiffeners and openings can be modelled and calculated. In this case two new web openings are to be inserted.

In order to simplify the editing and creation of the openings, please proceed as follows:

1. In order to have the structure only without results or FE-mesh, the mesh is to be deleted through the menu item Calculate ŕ Delete FE-Mesh.
2. As the openings dispose of dimensions of 50cm x 30cm the spacing between grid points, on which the input cross-hairs are orientated, will be reduced to 0.10 m. The corresponding dialog is opened via the button Grid and Work plane settings.
3. In order to edit the web the XZ work plane will be activated with the icon Work plane XZ.
4. The structure should be now zoomed and rotated such that graphical input on the plane of the web becomes simple (see picture)



Now, the boundary lines of the opening can be set.

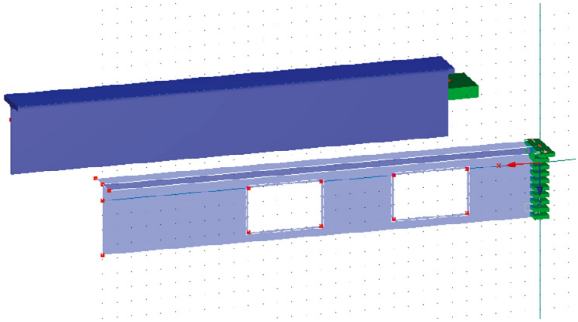
1. Select the function New line by clicking on the icon of the same name in the toolbar.
2. Click on the following coordinates in order to create the boundary lines of the first opening; **0.50 / 0.00 / 0.30 ; 0.50 / 0.00 / 0.00 ; 1.00 / 0.00 / 0.00 ; 1.00 / 0.00 / 0.30 ; 0.50 / 0.00 / 0.30**.
3. Right hand mouse click in order to exit the multiple line function. The line function is still active.
4. Click on the following coordinates in order to create the boundary lines of the second opening: **1.50 / 0.00 / 0.30; 1.50 / 0.00 / 0.00; 2.00 / 0.00 / 0.00; 2.00 / 0.00 / 0.30; 1.50 / 0.00 / 0.30**.
5. Doubleclick with right mouse button in order to exit the line function.

The openings can now be generated simply as the lines just created lie exactly in the plane of the web and therefore automatically integrated in the surface.

1. Activate the function **New opening** with a click on the respective icon in the toolbar.
2. Click on the respective boundary lines of the openings to be generated.
3. Right hand mouse click exits the function.

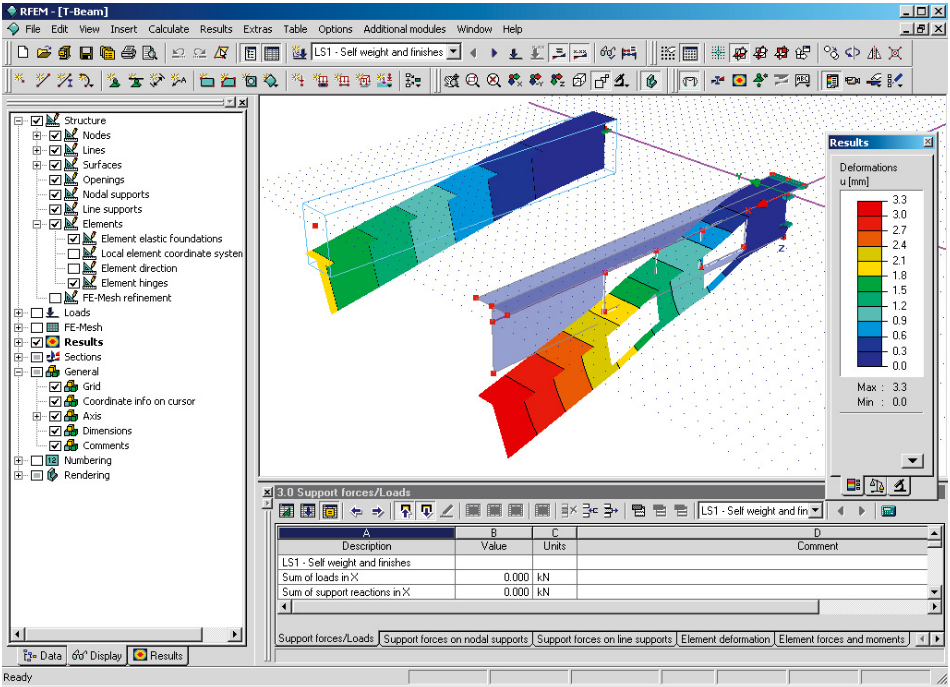
Application: Calculate - Delete FE-Mesh and activate Work plane XZ

The openings will be displayed immediately. The material lying within the opening has been removed and no FE-Mesh will be generated here.



Calculation

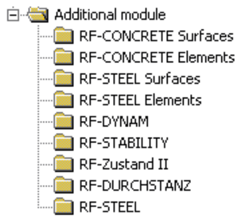
The only load case LS 1 will be calculated and the deformation results display by clicking on the menu item **Calculate → Calculate all**. Clear to see is the influence of the openings, in particular the shear deformation in the area around the openings. For a more optimal overview the element display has been reduced to contours and the surface display deactivated. This is done in the **navigator** in the register **Display**.



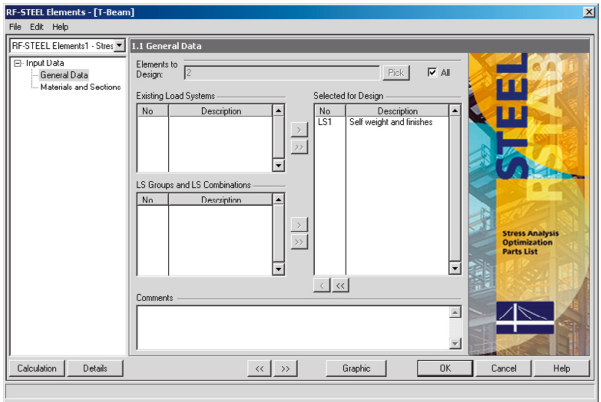
Additional module RF-Steel Elements

Comfortable and quick stress analyses can be carried out with the additional module RF-Steel Elements.

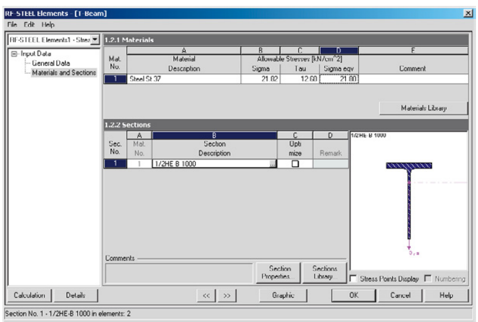
The module can be started in the navigator in the register Data in the folder additional modules or in the menu under the menu item Additional modules → Design.



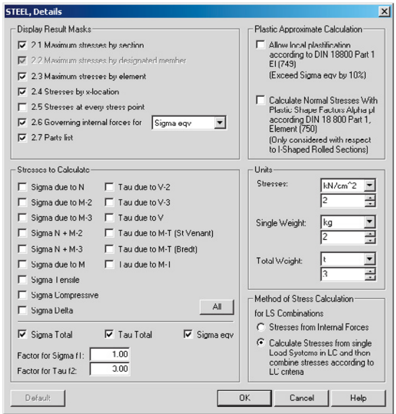
The module starts on Page 1.1. Selected in this page are the load cases, load groups and combinations which should be considered in the design. By default, all elements are activated. In this case the only load case LS 1 will be selected though a double click.



Allowable stresses and sections to be designed are stipulated on page 1.2.1 and 1.2.2 which are both to be found in one single window. The possibility of optimization should be mentioned here in particular. The optimization allows the selection of the section, from the respective section table, which displays the highest utilization ratio (less than or equal to 1.0). Export of the optimized section to RFEM is possible enabling a new calculation with the optimized sections. In this case, optimization will not be carried out.

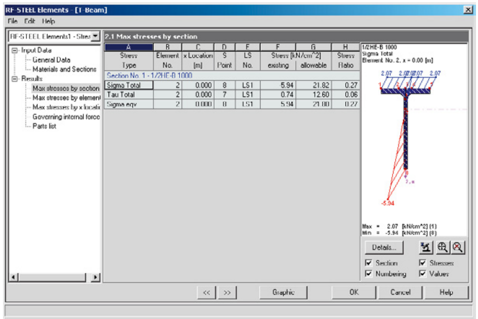


By clicking on the button Details you open a window where various settings, to which RF Steel Elements calculate, can be determined. For example, the stresses can be determined individually according to their origins, stress range delta Sigma, required for fatigue analysis, can be determined, the units, the plastic capacity, the type of calculation and the extent of output are all selectable. In this instance the default setting will be accepted with a click on OK.



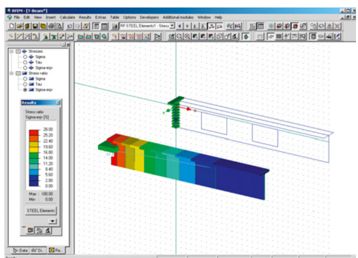
The **calculation** is started with a click on the button Calculate in the module window.

In the first output page, **Page 2.1**, the stresses are displayed by section. In this case it can be recognised that the stress ratio or utilization due to the given loading lies at 27%.

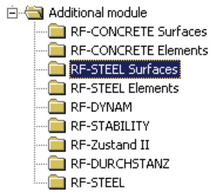


In table 2.7 a parts list is given, which provides a good oversight over the actual mass of steel required.

With a click on the button Graphic the results from RF Steel Elements will be displayed in the screen graphics.



Additional Module RF-Steel Surfaces

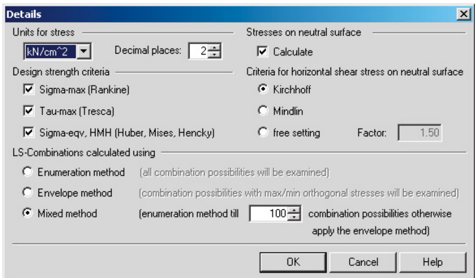


Comfortable and quick stress analyses for surfaces can be carried out with the additional module RF-Steel Surfaces.

The module can be started in the navigator in the register **Data** in the folder additional modules or in the menu under the menu item **Additional modules → Design**.

The module starts on Page 1.1. Selected in this page are the load cases, load groups and combinations which should be considered in the design. By default, all elements are activated. In this case the only load case LS 1 will be selected though a double click.

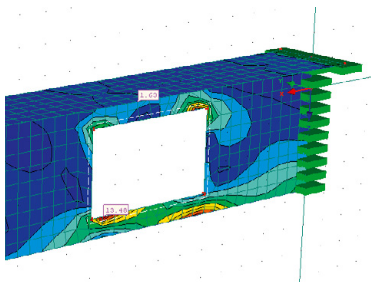
By clicking on the button **Details** you open a window where various settings, to which RF-Steel Elements calculate, can be determined. For example, the units, the type of calculation and the extent of output are all selectable.



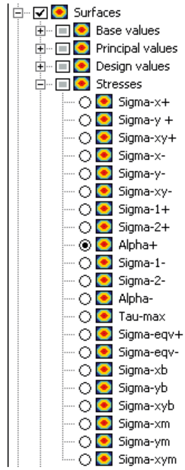
In page 1.2 the material data and surface properties can be checked and edited.
In this instance no changes shall be made.

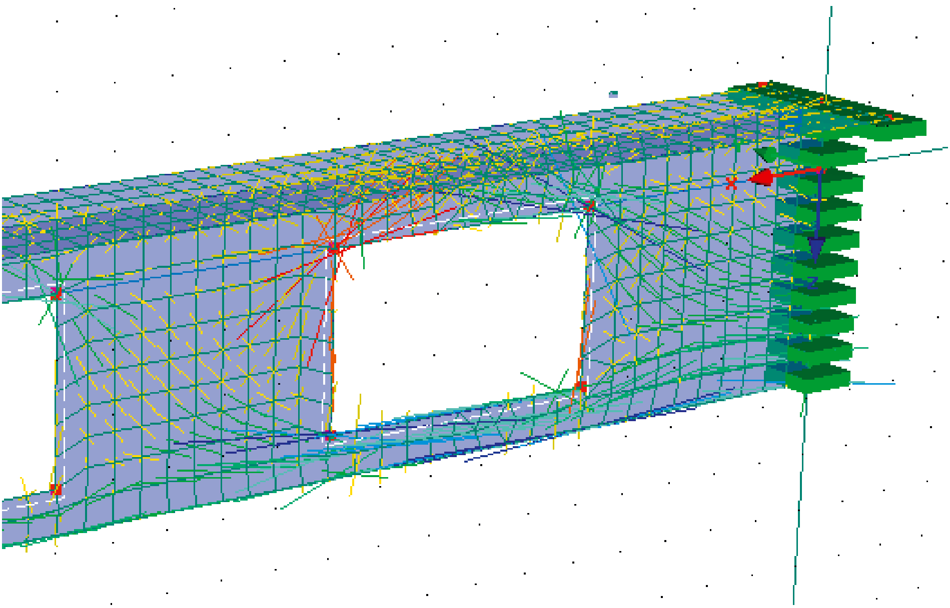
With a click on **Calculation** the design will be carried out and the results displayed in pages 2.1 and 2.2.

It can be seen in page 2.1 that surface no. 3, the web surface is utilized to 62%. Though a click on Graphic, the results will be displayed graphically and it can be recognised that the relatively high stress ratio is located in the region around the opening.



The display of stress trajectories can be very interesting from a point of view of structure analysis and stress distribution through a member. These trajectories show magnitude and direction of the stresses graphically. This results display option can be found in the **results navigator** of the load case, load group or combination under the branch **Surfaces → Stresses**.



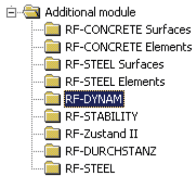


Open Position

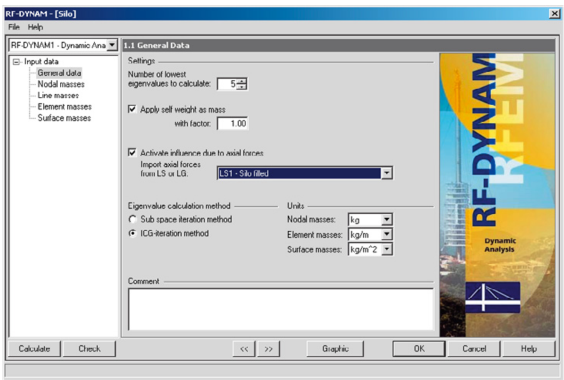
In order to carry out a dynamical analysis it is not necessary to create a new position. The silo, described in chapter 7 of this brochure and to be found on the RFEM CD, will be considered. Please open the file already created by you or the corresponding example file Demo-Silo.rfe which was installed with RFEM.

RF-DYNAM

The module can be started from the folder Additional modules in the register **Data** or in the menu under the menu item **Additional modules → Dynamic**.



The module begins with page 1.1 in which the general data for the dynamic calculation can be determined.



Enter here the number of lowest eigenvalues to be calculated. **RF_DYNAM** calculates the **lowest eigenvalues** of a structure. Set this value to **5**.

The **self-weight** is to be considered as a mass acting with a factor of 1.00, as given by default.

Activate the influence of axial forces:
The geometrical stiffness matrix can be affected by the inclusion of axial forces of a load case or load system from RFEM. Tensile forces in an element or surface tend to increase the natural frequencies. Pre-stressing forces can thus be taken into account. No axial forces are to be considered here.

Eigenvalue solver methods
Two powerful equation solvers are available for the determination of the eigenvalues.

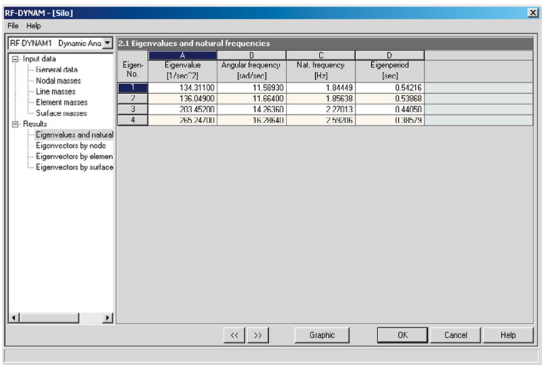
Sub-Space Iteration method: In the first phase of the calculation Cholesky-decomposition is applied. The algorithm for sparse matrices is used here, which is the same as that used for the direct method solver. This is followed by iteration of the sub space, which ends as soon as the required accuracy or maximum number of iterations has been achieved. All Eigenvalues required are solved simultaneously. The method is particularly suitable for all practice related problems, above all for small to medium sized structures. This method is quicker than the vector method if enough RAM is available. For larger structure (in particular for those with a large band width) and insufficient RAM a regular swapping of data takes place resulting in an extremely reduced effectiveness of this method. It can also occur that for large structure the virtual memory of 2 GB is exceeded meaning that no solution is possible with this method.

ICG Iteration Method (Incomplete Conjugate Gradient): This is simple algorithm (vector iteration method), which has little demand on the memory. This results in a large number of iterations. In contrast to the Sub-space iteration method the band width has no influence. Iterations converge quicker, should the membrane component govern. Otherwise (Bending governs) the method converges very slowly. The Eigenvalues are solved one by one and hence the time required for computation is proportional to the number of required Eigenvalues.

In this case the ICG-Iteration method is to be selected.

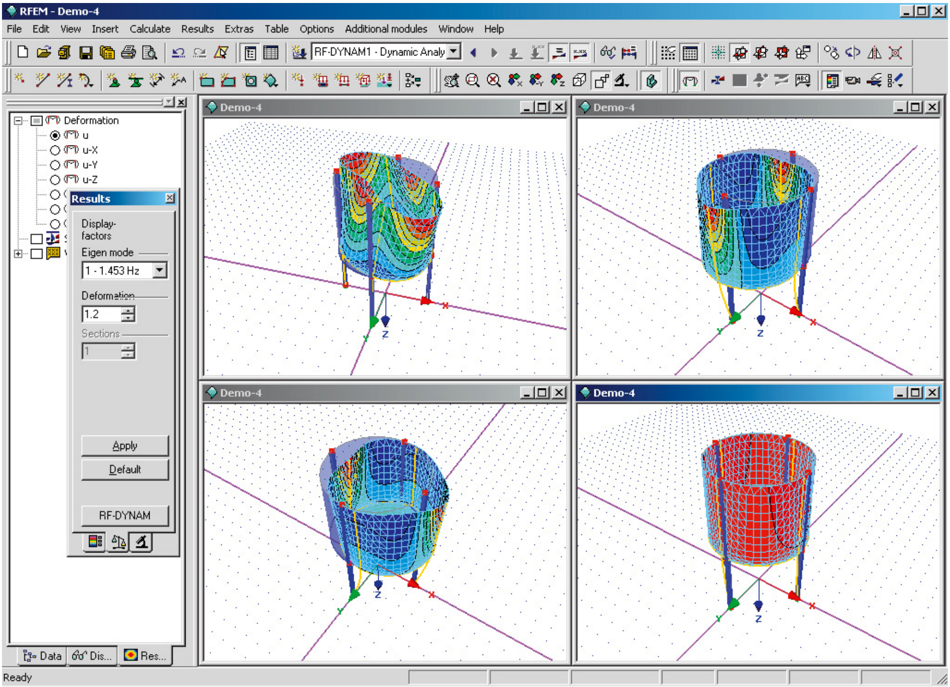
In addition or as an alternative to considering self-weight, it's possible to define **additional nodal, line, element or surface masses on pages 1.2 - 1.5**. We will not avail of this in the example.

By clicking on the button **Calculate**, the dynamic analysis is begun. After successful calculation the results of the Eigenvalues and natural frequencies will be displayed on page 2.1. On the following pages, a more detailed listing of results is provided, whereby sorting of results is possible according to object or Eigenvalue.



With a click on the button Graphic, the RF-DYNAM results will be displayed in the RFEM screen graphics.

In the following picture several Eigenmodes are displayed simultaneously. The Eigenmode can be selected in the register **Factors** in the results panel.



A powerful tool in demonstrating the oscillations is provided by **Animation**. In the animation the deformation will be displayed from the largest positive to largest negative deformation and back again in animated form.

With a click on the button **RF-DYNAM** you arrive back in the module window once again.

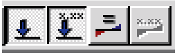
Application: Animation 

Open position

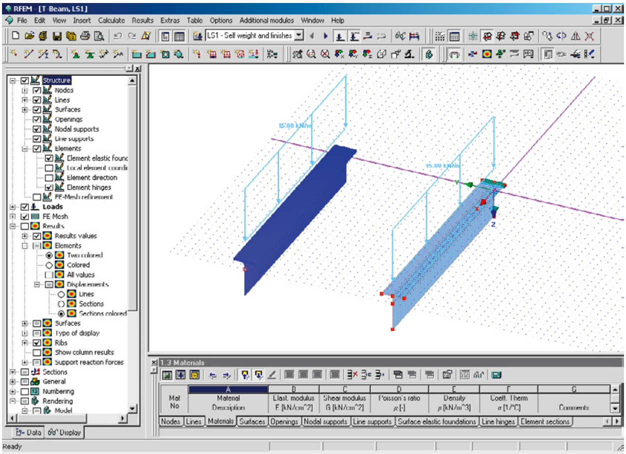
No new position is needed for the stability analysis. The T-beam, created in chapter 8 and also available on the RFEM CD, will be used here.
Please open the file either created by you or the corresponding example file Dem-T-Beam which was installed with RFEM.

Edit Structure

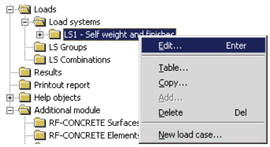
As only the beam consisting of surface elements and openings is to be considered here, please delete the beam defined as an element together with its loading. This can be done as follows:



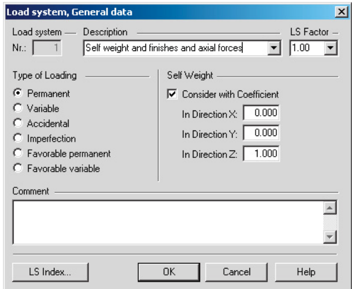
1. Display the loaded structure but without results.
2. Zoom and rotate the structure such that both beams lie next to each other, as seen in the picture.
3. Drag a selection window over the element, its support and loading by holding the left hand mouse key down, such that all are selected, as can be seen form the picture, right).
4. Press the Delete key on the keyboard in order to delete all selected objects.



Now an axial force is to be applied. Please proceed as follows:



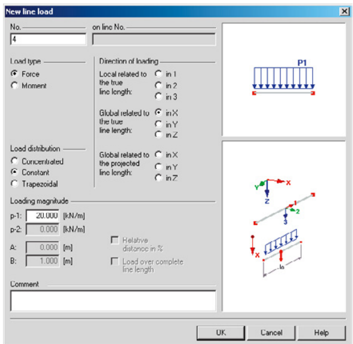
1. Click in the navigator in the register Data with the right hand mouse button the entry Loads → Load cases → LC 1 - Self weight and finishes.
2. Select the menu item Edit with the left hand mouse button.
3. Complete the load case description with the term Axial force.
4. Accept the changes by clicking on OK.

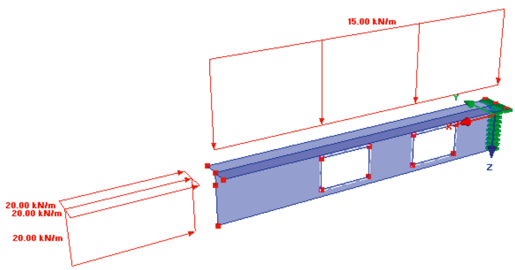


The axial force will be applied as follows:

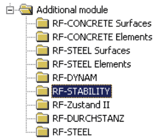
1. Select the function New line load from the toolbar.
2. Set the load direction to Global in x with respect to the true line length.
3. Click on the end lines of the beam in order to load it. These are the lines numbered 6,9,11.

The structure is now additionally loaded by an axial force.





RF-STABILITY



The module is started in the register Data in the folder Additional modules or under the menu item **Additional modules → Stability**. The module begins with page 1.1 in which general details for the stability analysis can be entered. Enter „6" for the **number of the lowest buckling modes** to be determined. Select that axial forces from **load case 1 - Self weight and finishes** be taken into account for the analysis.

The **utilization of favourable effects due to tensile forces** and the **effect of axial forces due to initial pre-stressing loads** remain unconsidered. Please read the RF-Stability manual or the corresponding page in the Help system for further details.

The **sub-space iteration method** should be selected for the method of determining the Eigenvalues as this results in a much quicker computation time. For more info please read chapter 9 of the brochure or the RF-Stability manual. By clicking on the button **Calculate** the stability design calculation will start. After a successful analysis the critical load factors will be displayed on page 2.1.

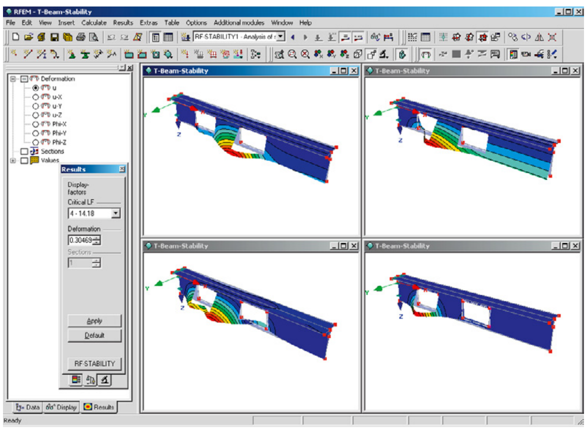
2.1 Critical Load Factors

	A	B
Shape No	Critical Load Factor	Magnification Factor
1	6.17336	1.19330
2	6.78723	1.17279
3	13.61880	1.07925
4	14.17710	1.07589

In this case the lowest critical load factor lies above 6. This means that for the current load case no global instability will occur. On the following pages the effective lengths and buckling modes are tabularized numerically, whereby sorting of results according to objects or buckling mode is possible.

By clicking on the button **Graphic** the results will be displayed in the RFEM screen graphics.

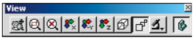
Here the four most critical buckling modes are displayed simultaneously. The buckling mode can be selected in the results panel in the register Factors. The buckling mode number and buckling load factor are shown in the corres-ponding list boxes.



With the **Animation** function you also have a very good possibility to display the local buckling modes. During animation the deformation display wanders from the maximum positive deformation to the largest negative deformation and back again.

With a click on the button **RF-Stability** you will arrive back in the module window once again.

Views



By means of the **viewing** tools it is possible to display the structure in such a way that editing or displaying results is optimal.
The toolbars can be found in the uppermost part of the RFEM window directly underneath the Menu bar, but can be also pulled in to the screen as a window. Here the available functions will be demonstrated briefly.

Move

Activate the function **Move, Zoom, Rotate** in the toolbar. The mouse pointer turns into a hand with an inscribed cross. Click on the screen with the left mouse button and hold down. The structure can now be moved. The function is exited by clicking the right mouse button.

Zoom

Activate the function **Move, Zoom, Rotate** in the toolbar. The mouse pointer turns into a hand with an inscribed cross. With the **(Shift)** - key held down click on the screen with the left mouse button. The cross inscribed in the hand transforms into a magnifying glass. By dragging the mouse you can zoom toward or away from the structure. Exit the function with a click on the right mouse button or through **(Esc)**. With a mouse wheel no buttons need to be held in order to utilize the zoom function. Turning the mouse wheel with an active **Move, Zoom, Rotate** function zooms toward or away from the point on which the mouse pointer lies.

Rotate

Activate the function **Move, Zoom, Rotate** in the toolbar. The mouse pointer turns into a hand with an inscribed cross. . With the **(Ctrl)** - key held down click on the screen with the left mouse button. The cross inscribed in the hand transforms into a rotating arrow. Rotate the structure through mouse movements. Exit the function with the right mouse button or through **(Esc)**.

Zoom with window

The function Zoom with window enables you to zoom in on a visible portion of the screen. The mouse pointer turns into a magnifying glass. Position the magnifying glass over one corner of the region considered, and, by clicking and holding down the left mouse button, drag a window across the region to be zoomed. Should the region encompassed by the window be that required, then the region will be zoomed by releasing the mouse button and the function exited.

Show whole structure

With the function **Show whole structure** the structure will be centred and zoomed in upon such that the zoom factor is a maximum and the structure being displayed is completely visible.

View in X direction

With the function View in X direction the perspective changes such that the observers view is orientated towards the center of the structure in the positive x direction. The whole structure is contained in the window.

View in the Reverse Y direction

With the function View in Reverse Y direction the perspective changes such that the observers view is orientated towards the center of the structure in the negative y direction. The whole structure is contained in the window.

View in Z direction

With the function View in X direction the perspective changes such that the observers view is orientated towards the center of the structure in the positive z direction. The whole structure is contained in the window.

Isometric view

With the function Isometric view the structure is displayed in 3D and maximized and centred on screen.

Perspective

The perspective view is active by default in RFEM. With this function the engineer achieves a realistic view of the structure and can more or less look into the structure and achieve an optimized view of the structural layout and its individual components. By switching the function off real elevation and plan views can be generated.

Cut and Selection

With the aid of the Cut and Selection function individual or several members can be selected from the complete structure and then either be displayed or blended out in isolation, i.e. without the remainder of the structure.

Rendering Mode

With the command Display solid / line model you can switch between types of display. The type of rendering can be determined more exactly under the register Display in the navigator. Several options are available for the type of display for elements and surfaces. In order to create a realistic impression the lighting can be influenced via local and global lights which can be switched on or off, as required.

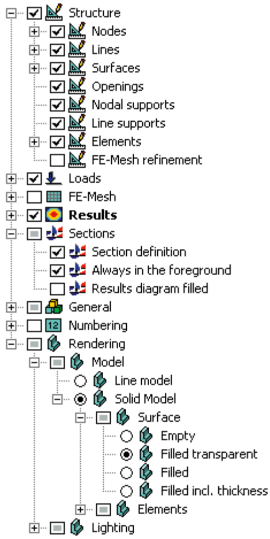
The register Display in the Navigator

The settings for display can be determined in the register Display. To stipulate is what is to be displayed and how.

All that is needed is the activation, deactivation or selection of display options.

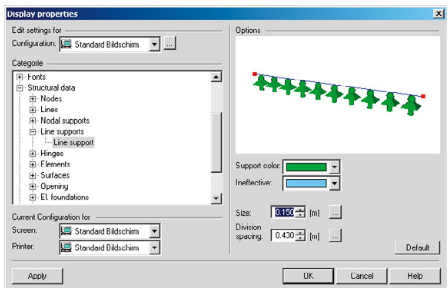
The individual navigations zones are structured in a tree like manner. Individual structural elements (nodes, lines, surfaces etc.) can be activated and deactivated or complete tree zones (e.g. Loads) can be included in or removed from the display.

The type of structural display regarding rendering is also determined in this tree structure, as can be seen in the picture to the right. There are four display options to choose from, concerning the display of surfaces or the whole structure can be displayed as a line model which reduces the demands on the computer.



Display properties

Through the display properties colors, shapes, sizes and fonts for the graphical output on screen and printer can be stipulated. Editing these properties is made possible by opening the menu Options from the menu bar, then selecting the menu item Display options and the command Edit from the drop down menu, or open the context menu with a right click on a structural element (e.g. a support) and select the menu item Display options. In the picture to the right the window containing display options for line supports is shown.



...learn even more about RFEM

With this you have just successfully modelled and calculated your first structures in RFEM. In the process you were able to experience, at first hand, how easy and quickly RFEM is to learn. The functions introduced here are only a small part of RFEM, however. You can learn more about RFEM by means of a test licence or through an RFEM training seminar. With a test licence you can install the complete RFEM family in your office for 6 weeks and test under real conditions without binding, restrictions or fees. In an RFEM basic training seminar you learn further RFEM functions and receive answers to your specific questions. Is RFEM the program that you've been looking for, for so long? Then simply consult us or fill out the form on the following page and return it to us by fax.

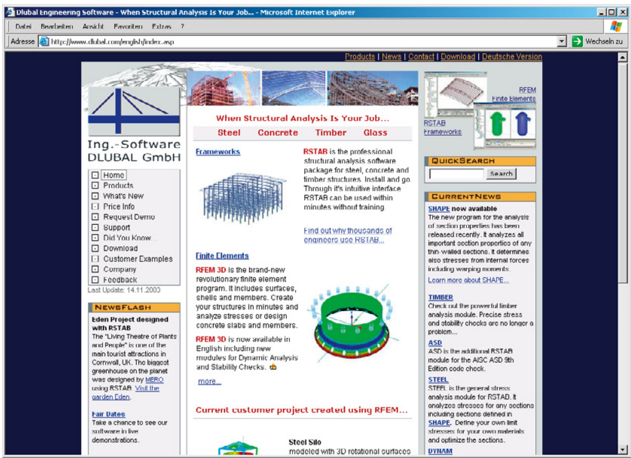
Finally we would like to thank you for you interest in RFEM. We hope you enjoyed the short introduction and that it helped in getting to know RFEM better.

Yours Ing.-Software Dlubal

www.dlubal.de
…always well informed.

Subjects of interest concerning statics, all details and news on our practise orientated statics programs can be found on our home page www.dlubal.com

Rummage through Questions and Answer, read about new products, download manuals and demo versions or view short videos of our software solutions.



Notes:

TELEFAX REPLY

Ingenieur-Software
Dlubal

...the quickest way to a new generation of statics software.

Fax: ++49-9673-1770

☐ I would like to test RFEM 6 weeks for free. Please send me the Test request form (please state fax number)

☐ I'm interested in an RFEM basic training seminar. Please inform me of the next seminar dates.

☐ Please call me, as I ☐ have further questions on RFEM
☐ wish to order RFEM.

I can be best contacted on:

. . . .

Date Time Telephone number

☐ I'm interested in other Dlubal products. Please send me a product overview with price lists.

☐ Miscellaneous remarks, questions and requests.

. . . .

. . . .

Sender:

Company: . . . .

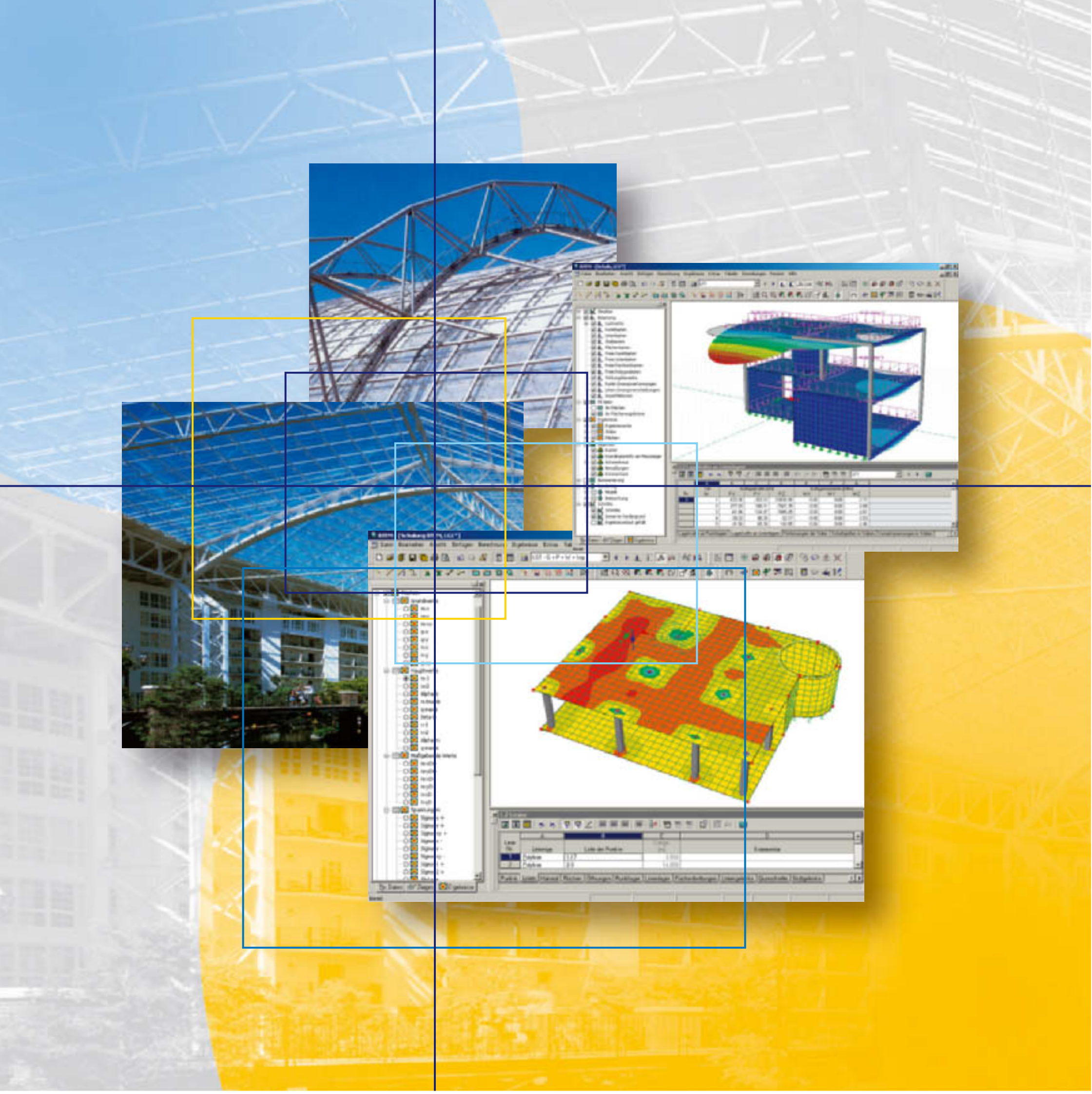
Surname: . . . . Name: . . . .

Street: . . . .

Zip/Post Code: . . . . City: . . . .

Tel.: . . . . Fax: . . . .

Email: . . . .



Structural design, made easy...

www.dlubal.de

Ingenieur-Software

Dlubal