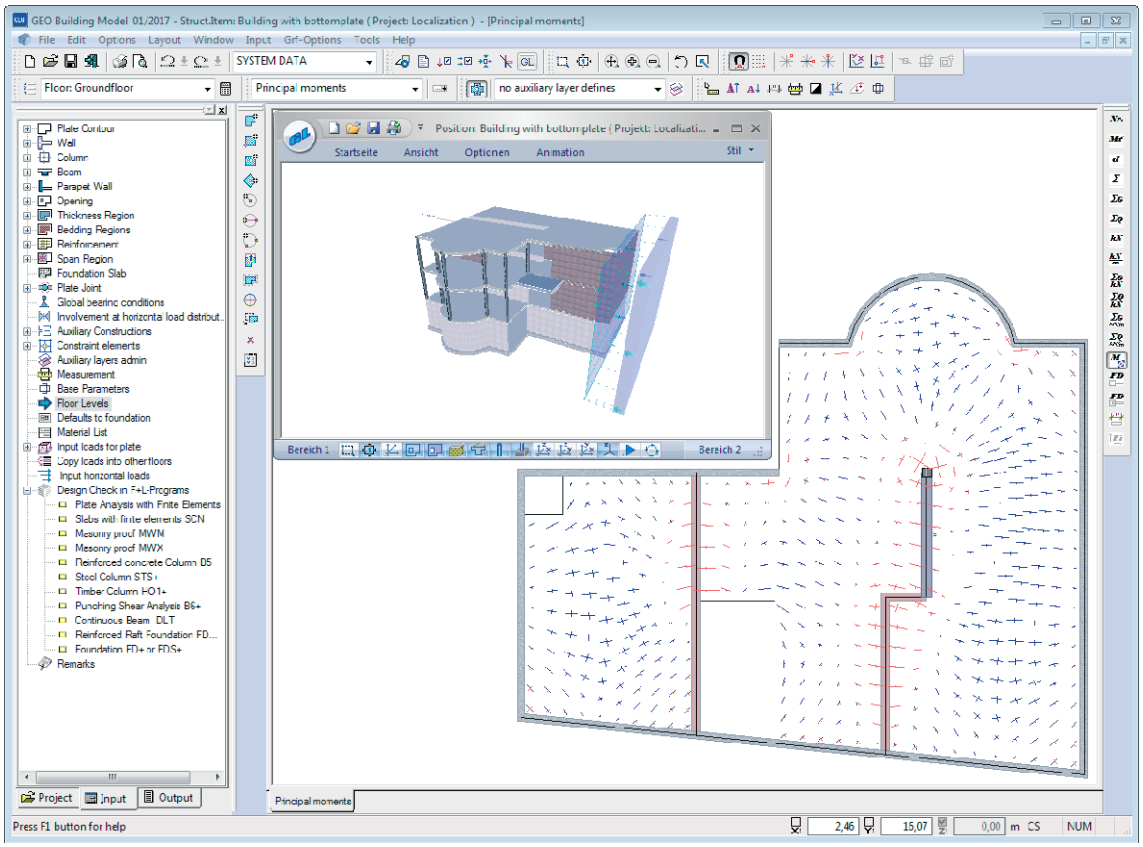
GEO – The Frilo Building Model

FRILO Software GmbH
www.frilo.com
info@frilo.com
As of 07.06.2017



GEO – The Frilo Building Model

Contents

Further information and descriptions are available in the relevant documentations:

Graphical input	Description of the basic functions
FDC – Basic Operating Instructions	General instructions for the manipulation of the user interface
FDC – Menu items	General description of the typical menu items of Frilo software applications
FDC – Output and printing	Output and printing
FDC - Import and export	Interfaces to other applications (ASCII, RTF, DXF …)
FCC	Frilo.Control.Center - the easy-to-use administration module for projects and items
FDD	Frilo.Document.Designer - document management based on PDF
Frilo.System.Next	Installation, configuration, network, database

Application options

Typical buildings in solid construction are considered as complete load-bearing structures. You can enter data via a graphical user interface, which you might know from the slab application PLT. You can import DXF files from CAD applications for each floor and use them as floor plans.

The data are recorded for each floor with all load-bearing components:

- Floor slabs with different areas (supporting direction, thickness, foundation and reinforcement areas)
- Walls (material, wall thickness)
- Columns
- Beams
- Parapets

A chain of walls with several sections of the same material is considered as a uniform wall pillar, unless otherwise defined by the user.

General

The software application assumes a "standard floor" with all typical constructive properties. After the definition of the floor, the user can copy it to create the next one. Optionally, the user can define a different floor plan for each floor or import a floor plan from a CAD application.

The details of each floor can be edited.

The foundations can be dimensioned by specifying the permissible soil pressure and the minimum dimensions and projections.

Benefits

Pre-design

- Fast calculation of loads for the foundations
- Well-structured representation of the load transfer, even in complex buildings
- Inclusion of the calculation of stiffeners for horizontal loads
- Pre-design of the load-bearing components (beams, walls, columns and foundations) and determination of the cross-sections in the first design phase

Final design

- Representation of the loads suitable for review, loads are shown separately for each floor and component as G, P and full loads
- Direct transfer of data to the software applications for the design of the individual components

Reinforcement drawings

- The As-values (FE plate) or the reinforcement layout can be exported from the design applications to CAD programs (Allplot by Nemetschek AG, -isb cad- Glaser) to create reinforcement drawings quickly and easily.

Standards

Reinforced concrete:

- DIN 1045 / DIN 1045-1
- DIN EN 1992
- ÖNORM B 4700
- ÖNORM EN 1992
- UNI EN 1992
- BS EN 1992
- PN EN 1992

Wind loads:

- New wind load standards DIN 1055-4 and EN 1991-1-4 (DIN, ÖNORM, BS)

Materials for steel columns as per:

- DIN 18800
- DIN EN 1993
- ÖNORM EN 1993
- BS EN 1993

Basis of calculation

The bearing loads are calculated for each floor with the help of a FEM plate calculation and transferred to the floor below.

Basic structural deformations are not taken into account.

The base of a wall pillar is in a constant stress state due to vertical loads. Therefore, a constant linear load caused by these vertical loads is transferred to the next floor at the base of a wall pillar. This assumption is allowed if the centroid of all loads on a floor slab is approximately identical to the centroid of the transferred loads.

A first-order analysis is performed for state I.

As the calculation handles the individual floors from top to bottom, floor slabs suspended to the floor slab above cannot be considered.

Wall-type beams over several floors cannot be taken into account either.

A bracing calculation is only available for non-sway systems, because the horizontal loads on each floor level are split up according to the bending stiffness of the bracing components (see also chapter Horizontal loads.

Load determination for load-transferring components

The loads applying to the transferring components are determined with the help of the finite elements method and not via load influence areas.

It should be noted in this connection that the typical particularities of this method at singularity points (e. g. high shear force jumps at re-entrant corners) will occur also in this calculation. Comparative calculations with 3D-FE-applications show that normally the calculated loads differ less than 5 % from the accurate three-dimensional calculation under normal conditions.

Input

Graphical user interface

The GEO application offers a graphical user interface, i.e. elements such as slab outlines, load coordinates etc. are drawn with the help of the mouse on the screen. Under normal conditions, you only need to enter numerical values of forces, etc. in corresponding dialogs.

You can see the defined graphical objects immediately on the screen. The hide/display options for individual elements such as load arrangements provide for a well-structured overview of even highly complex systems.

The graphical user interface itself is an independent application module, which is integrated with the GEO application.

The functions of the graphical user interface module are described separately in the document Graphical User Interface.pdf.

Numerical user interface

You can define values of course via numerical input fields each time a precise numerical specification is needed or preferred.

Note: *Direct help and support referring to the current input operation is given in the form of a short comment in the status line on bottom left of the screen.*

Importing data

You can import geometrical data providing the basis for the system definition via the DXF interface. Files of the -isb cad- application by Glaser can be processed directly. From Allplan, data of PLT Plates by Finite Elements can be imported into the building model too. A prerequisite to this is that components have been used in Allplan.

Defining a building

The definition of a building starts with the floor slab of the first floor. It is entered by drawing or defining the outline and the basic parameters.

The basic parameters are as follows: Material, slab thickness, top edge of floor slab, floor height, additional permanent load, live load, concrete cover.

Various drawing functions are available for the definition of outlines, recesses, walls, columns, loads, auxiliary lines etc. They are accessible via icons that can be activated per mouse click. There are icons for the input of lines, rectangles, polygons and circles. The definition of these outlines, i.e. the input of decisive coordinates, lengths and radii is done per mouse click under normal conditions. You can however always enter individual or all coordinates numerically via the keyboard.

Basic building data

In the menu ▸ Options ▸ Basic building data, you can define the elevation of the top edge of the floor slab of the selected floor. After this, you can select the desired standard for the load assumption. In the wind load section, you can select the wind load zone and the terrain profile.

Definition of the loads

In the Loads menu, you can find the load case management, action groups as well as concentrated, line, area and temperature loads. The load definition functions are part of the graphical user interface module.

Multi-storage buildings

We recommend using the copying and editing functions for the efficient input of several floors.

Floors

Floor levels

Access via the  button or the "floor levels" menu item in the main tree.

In the "floor levels" dialog, the user can

- define

- copy

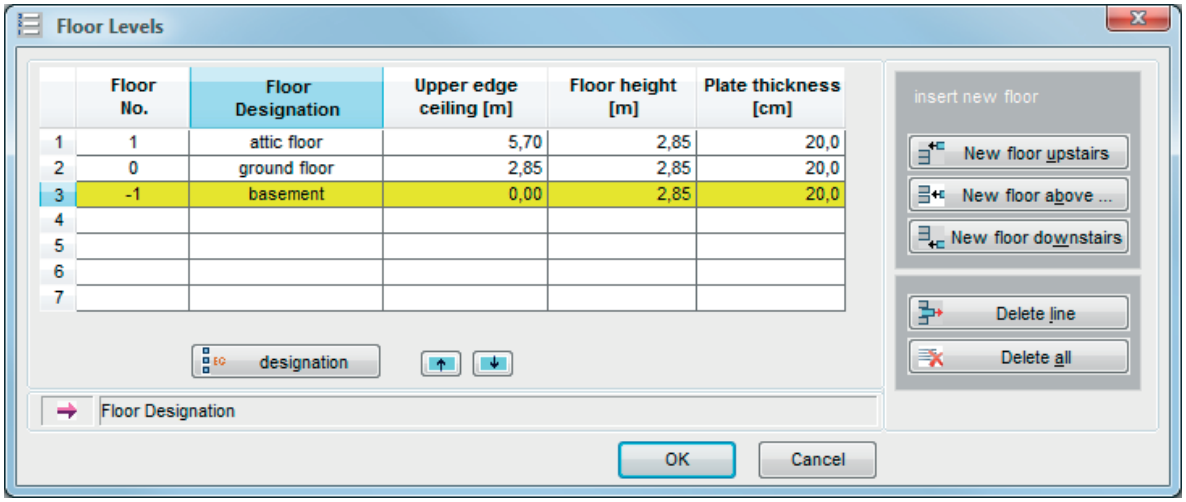
- delete

floor levels and edit the floor height, the floor slab thickness and the floor slab top edge.

The user can assign a name to each floor in the "Name" column. Alternatively, you can assign names automatically (first basement floor, ground floor, first floor, etc) via the "Name" button.

The floor number in the first column is generated automatically irrespective of the name. The user can edit the floor numbers too.

The buttons in the section "Add new floor" allows the user to add additional floors above or below the currently active floor (the currently active floor is that of the currently active row in the floor table). When adding a new floor you are prompted whether to copy the data of the floor above or underneath. If you want to import a new floor plan from a CAD application, select No in the prompt and activate the Import function to load the new floor plan.



Deleting floors

Activating the "Delete row" button, deletes the contents of the currently active row (the currently active floor).

Activating the "Delete all" button deletes all rows (floors).

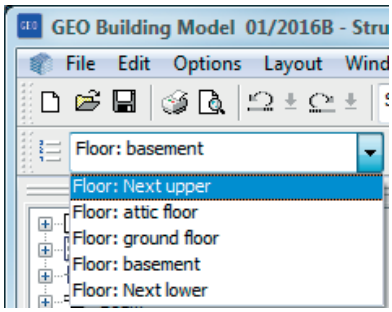
Copying floors

See the chapter Floor selection.

Moving floors

 With these buttons you can move the active floor up or down.

Floor selection

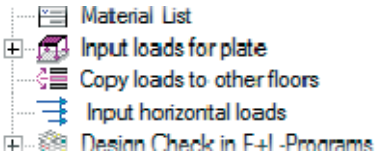


You can navigate through the floors of the building by using the floor selection function.

Copy floor/generate new floor

You can define additional floors via "Next upper floor" or "Next lower floor". This option copies the currently selected floor and displays the basic parameters dialog. You have the option to copy the loads too.
To generate a new floor (which is not a copy of the currently active one), activate the menu item "floor levels". The submenu options allow you to create and delete floors.

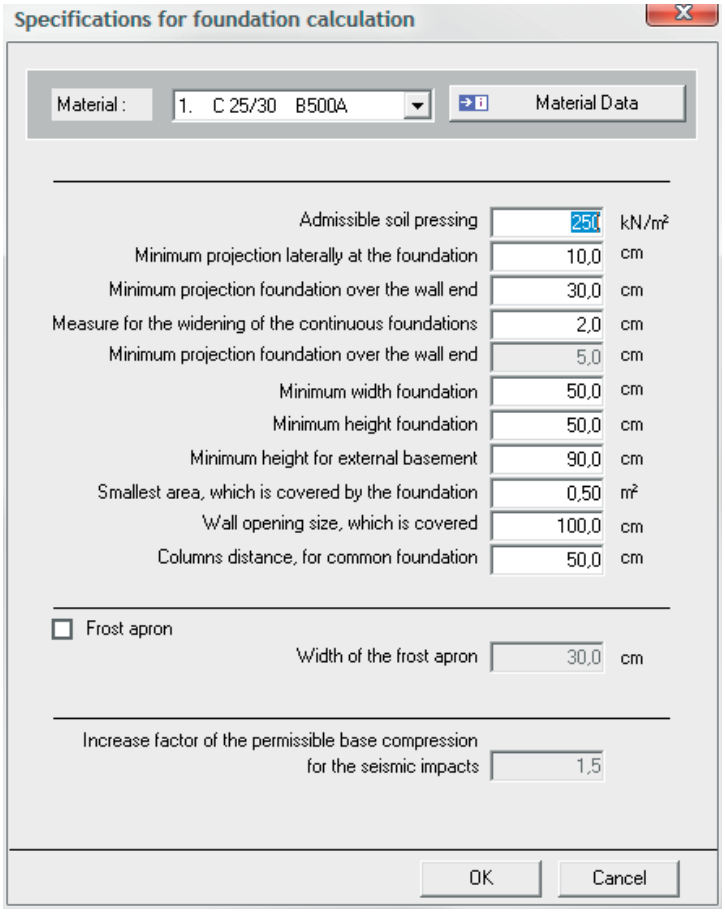
Copy loadcases to other floors



You can choose loadcases to copy them from the currently selected floor into other floors.

Foundation design - user-defined settings

The menu item ▸ Foundation design settings allows you to set parameters for the foundation design (permissible soil pressure...).



The specified soil pressure and the calculated loads are used to determine the required dimensions of the foundation. If these dimensions are smaller than the specified minimum dimensions, the minimum values are used.

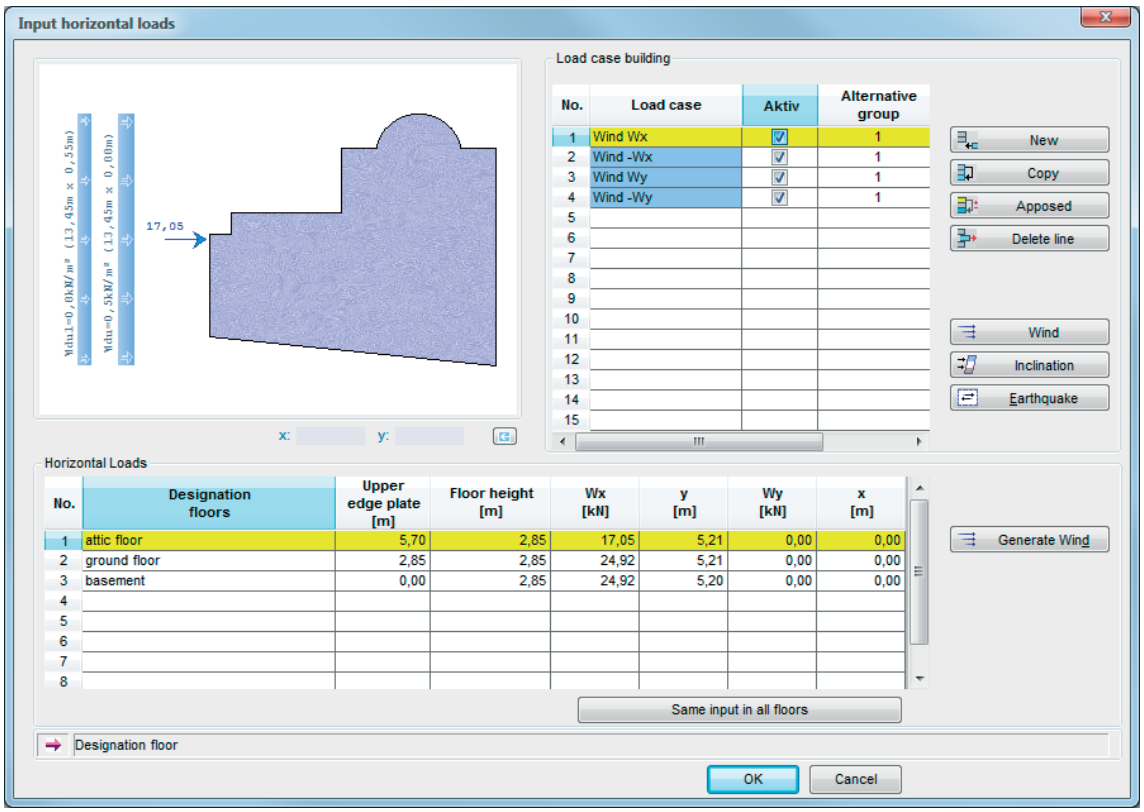
The actual foundation design is done in the respective foundation design software as with other components.

Horizontal loads

Access via the menu item ▸ Input of horizontal loads

The horizontal loads are distributed on the bracing components of each floor based on their bending stiffness. Shear field stiffness is not taken into account. You should note in this context that only non-sway buildings braced with walls can be calculated, because frame effects cannot be taken into account in the software application.

You can enter horizontal loads manually or generate wind loads and loads due to inclination automatically.



If the "Active" box is checked, the corresponding load case is taken into account in the calculation of the building bracing.

Alternative group: See the chapter Load case. With loads due to inclination, alternative grouping also applies to the permanent loads.

Wind

With this button the standard loadcases for Wind can be generated.
According to the base parameters, the building geometry and wind load parameters of the load case, the wind load is thereby generated automatically.
With every change of the relevant wind data automatic adjustments (new generation) of the wind load are carried out in the defined standard load cases.

All variants: With this option, several load cases with the same wind load parameters for the corresponding wind direction are applied. If necessary, individual wind load parameters for these load cases can be defined by use of the function "Generate Wind".

Note concerning automatic generation:
This automatic was introduced with version 3/2015. Former items are checked. If wind loads are defined and a mismatch is detected, the user will get a notification an an automatic adjustment is possible.

If the user rejects this proposal, the load is re-classified to a custom horizontal load. Eg. load case "Wind Wx" is then named "Wind Wx_UserD". This user defined load case is not subject to the automatic adjustment and remains unchanged as a general load case of the horizontal loads. New items can be opened from former programversions

Inclination — This button allows you to calculate inclination loads in dependence of vertical loads in accordance with DIN 1045-1, 7.2 or EN 1992, 5.2. The vertical loads are determined in a first calculation run. Subsequently, the horizontal loads from inclination are calculated in a second run.

Earthquake — This option accesses a dialog for the definition of the Basic parameters for the determination of the soil acceleration response spectrum.

Individual input of horizontal loads

You can generate any number of additional horizontal load cases. In this user-defined load cases, you can freely select the direction of wind loads. For "inclined" wind loads, an x- and y-component must be specified separately, however.

Same input for all floors: Activate this button to assign the values of the currently active row to all other floors.

Horizontal loads

Wx — resulting horizontal force in x-direction

y — distance of the resultant to zero in y-direction

Wy — resulting horizontal force in x-direction

x — distance of the resultant to zero in x-direction

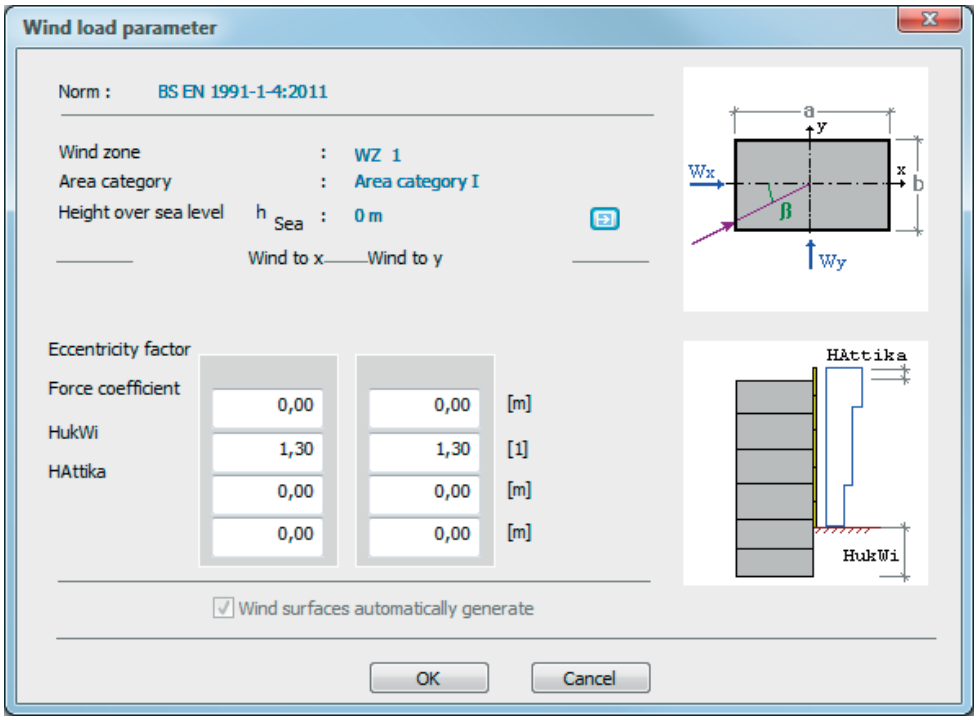
Tip: You can print the graph of this dialog via the context menu options (right mouse button).

Generate Wind

This button allows you to access the Wind load parameters dialog.

Wind load parameters

In this section, the user can generate wind loads automatically.



The wind loads are generated with consideration to the defined geometry and the border conditions specified in this dialog (Wind to x / y).

Eccentricity The user can define an eccentricity for the wind load resultant. If the eccentricity factor is set to "0", the resultant is assumed to apply in the centroid of the floor.

Force coefficient A force coefficient of 1.30 for pressure and suction is set by default. The setting can be edited.

HukWi The height of the building section below ground level is specified in this section. No wind loads are generated for this area.

HAttika The height of the roof parapet can be specified, if it should be added to the wind action area on the top floor.

The building basic parameters (standard, MSL ...) are also shown in this dialog.
The button provides access to the basic building parameters dialog.

You can find a description of the wind loads parameters in accordance with Eurocode in the document "WL Parameter eng.pdf".

Need for calculation according to theory II. Order

In compliance with defined criteria of bracing effects from theory II. order can be neglected.

Earthquake

Access: ▸ Input of horizontal loads ▸ Earthquake button

General

The calculation is based on the simplified response spectrum method as per DIN 4149, chap. 6.2.2 or EN 1998, 4.3.3.2. According to this method, the entire system can be reduced to two plain systems that can be examined independently of each other. It is sufficient to consider the first eigenform (basic vibration) for each system in each direction in the examination.

A prerequisite for the application of the simplified method is a system with regular floor plans and elevations according to DIN 4149 and EN 1998.

The regularity of the elevation is the most important condition. This means among other criteria that the supporting walls run continuously from the foundations to the top edge of the building. Therefore, no vertical loads on components are generated by the horizontal earthquake loading.

The calculation in the software is based on the assumption that horizontal loads are transferred exclusively via the walls, frame supporting effects are not taken into account. Deformations by shear are not considered either.

The requirement for a regular floor plan is less compelling and may be disregarded to a certain extent if the prerequisites for a simplified verification of the effects by torsion in accordance with DIN 4119, 6.2.2.4.2 (2) or EN 1998, 4.3.3.1 are satisfied.

Calculation of the participating masses

The calculation of the participating masses is based on DIN 4249:2005, 5.5 or EN 1998, 3.2.4(2) and 4.2.4. You should note in this connection that the relevant DIN standards require a factor of $\psi_2 = 0.5$ for snow loads in Baden-Württemberg. A corresponding option is available in the software. Prerequisite for the enabling of this option is a load case with the action type snow applying to any of the floors.

DIN EN 1998 prescribes a factor of $\psi_2 = 0.5$ for snow loads in general.

Calculation of basic vibration times

The determination of the basic vibration times T_1 is based on the energy method (Rayleigh quotient with a uniform distribution of masses). In literature, various formulas are recommended for this calculation.

The software uses the equation $T_1 = 2 \cdot \pi \cdot H^2 \cdot \sqrt{\frac{0.6 \cdot m_1}{h \cdot E \cdot I \cdot 8}}$ developed by Küttler, "Erdbebensicherheit von Bauwerken nach DIN 4149" (2005 edition). Alternatively, the equation $T_1 = \frac{2 \cdot \pi \cdot H^2}{\alpha_1^2} \cdot \sqrt{\frac{m_1}{h \cdot E \cdot I}}$ developed by Müller/Keintzel, "Erdbebensicherung von Hochbauten" can be used. H is the height of the building, h the height of the floor, m the average floor mass, El the average stiffness and α_1 the vibration coefficient, which depends on the number of floors.

Under normal conditions, the first equation produces slightly lower basic vibration times and consequently, slightly greater equivalent earthquake forces.

Calculation of the spectral value

In the next step, the spectral values are assessed from the elastic response spectra, which depend on the subsoil. The results depend on the respective basic vibration times.

See also DIN 4149:2005, 5.4.2, and/or EN 1998, 3.2.2.2.

You should note in this connection that taking the foremost rise in the response spectrum into account might produce too favourable results in the calculation of the equivalent earthquake forces due to the averaging of the masses and stiffnesses. Therefore, it is recommended using the plateau value in this area too to be on the safe side. A corresponding option is available in the software.

Acceleration design values

To obtain the design values of the acceleration S_d, it is necessary to divide the calculated spectral values by the respective behaviour coefficient q, which depends on the ductility class and the slenderness of the wall.

Calculation of the horizontal equivalent earthquake forces

The total earthquake force F_b is calculated with the help of the equation $F_b = S_d \cdot M \cdot \lambda$, whereby M is the total mass of the building and λ the correction factor for the effective modal mass in the first eigenform.

The distribution of the total earthquake force over the individual floors is based on a linearly rising horizontal shift. The horizontal force applying at the height z results from the following equation

$$F_i = F_b \cdot \frac{z_i \cdot m_i}{\sum z_j \cdot m_j} .$$

Consideration of torsional effects

In the Frilo building model, the calculation of torsion is based on DIN 4149, 6.2.2.4.2, Para. (11) and/or EN 1998, 4.3.3.2.4.

The horizontal loads applying actually at the mass centroid M are assumed to apply at a distance max e or min e from the stiffness centroid. The torsional moment MT produced this way must be distributed over the individual walls. The more unfavourable of both values must be used for each wall.

For earthquake excitation in y-direction, the following torsional moments are obtained:

$M_i = F_i \cdot \max e = F_i \cdot (e_0 + e_1 + e_2)$

$M_i = F_i \cdot \min e = F_i \cdot (0.5 \cdot e_0 - e_1)$

e_0 actual eccentricity (distance between mass centroid and stiffness centroid)

e_1 random eccentricity due to the fact that the distribution of masses is unsure (0.05 · L or 0.05 · B)

e_2 additional eccentricity (decoupling of the bending and torsional vibrations)

Load cases in the FRILO building model

You can handle earthquake loads in the window "Definition of horizontal loads".

When considering the torsional moments, two load cases each are generated in the Frilo building model for earthquake excitation in the x- and y-direction. Because earthquakes act similar to wind in positive as well as negative direction, the button "opposing" provides for the generation of four additional load cases.

The generated load cases are automatically assigned to an alternative group for each direction.

Verifications

In the Frilo building model, the distribution of the horizontal earthquake loads over the bracing components is calculated. The different load cases are handled separately. The actual component verification is done in the corresponding design application as usual. The combination of actions is also done in the design applications.

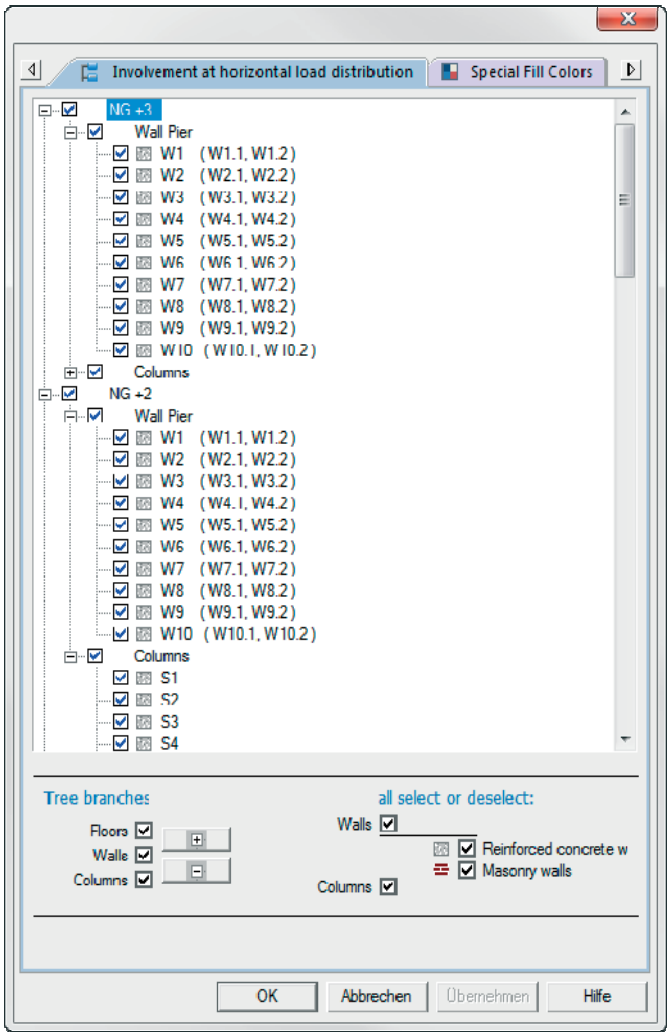
Output profile

The output profile in the output section offers a dedicated "earthquake" button, which displays a separate dialog.

Involvement in the horizontal load transfer

You can select which vertical components should be used for the horizontal load transfer over several floors. Entire component groups such as all columns or all masonry walls or individual components can be excluded from the horizontal load transfer in this section.

Alternatively, you can exclude the components from the load transfer in the properties dialog of the individual component.



Fill Colors

The fill colors can be individually defined for walls and columns which do not participate in the horizontal load transfer.

Calculation of loads

Use the "Calculate" button  to start the load calculation of the building.
The progress of the calculation is displayed in progress bar diagrams.
Status / OK messages are then displayed for the generation of the FE network and the calculation. Exit the dialog with "OK".

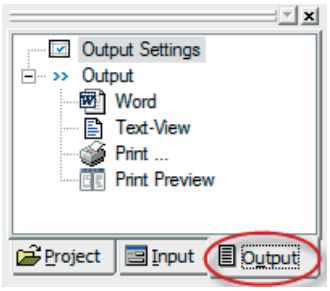
FE element size

This option allows you to specify an average edge length and a minimum edge length in [cm] for the FE elements. See also Tools ▸ Adjustment Geometry.

Output

The output tab provides access to the output options.

System data, results and graphics can be put out on the screen or printer.

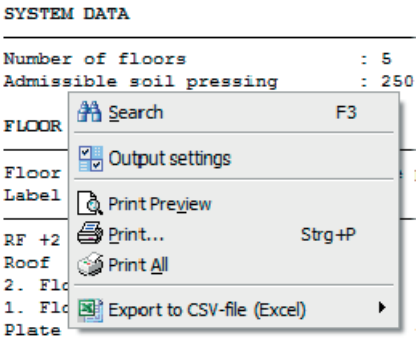


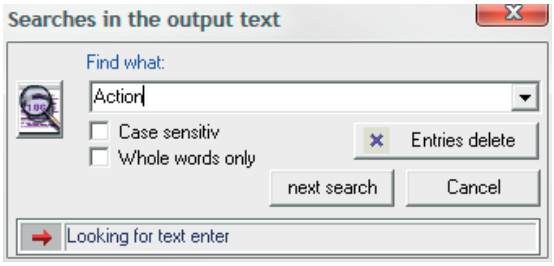
Output settings — The dialog offers comprehensive options for the control of the output scope. Check or uncheck the desired output options for the system, the load cases, the results and superpositions.

Text-View — Displays the values in a text window on the screen.

You can save the contents of the text window in a csv file via the context menu. csv files can be imported into Microsoft Excel for instance.

Activating the F3 key displays a dialog for searching the output text.





Word — Output to MS-Word, if this software is installed on the computer.

Print all — Starts the output on the printer.

Print Preview — Displays a preview of the printed page on the screen.

Results graphics

The functions for displaying the results graphs can be found in the chapter "Results Graphics".

Output Settings

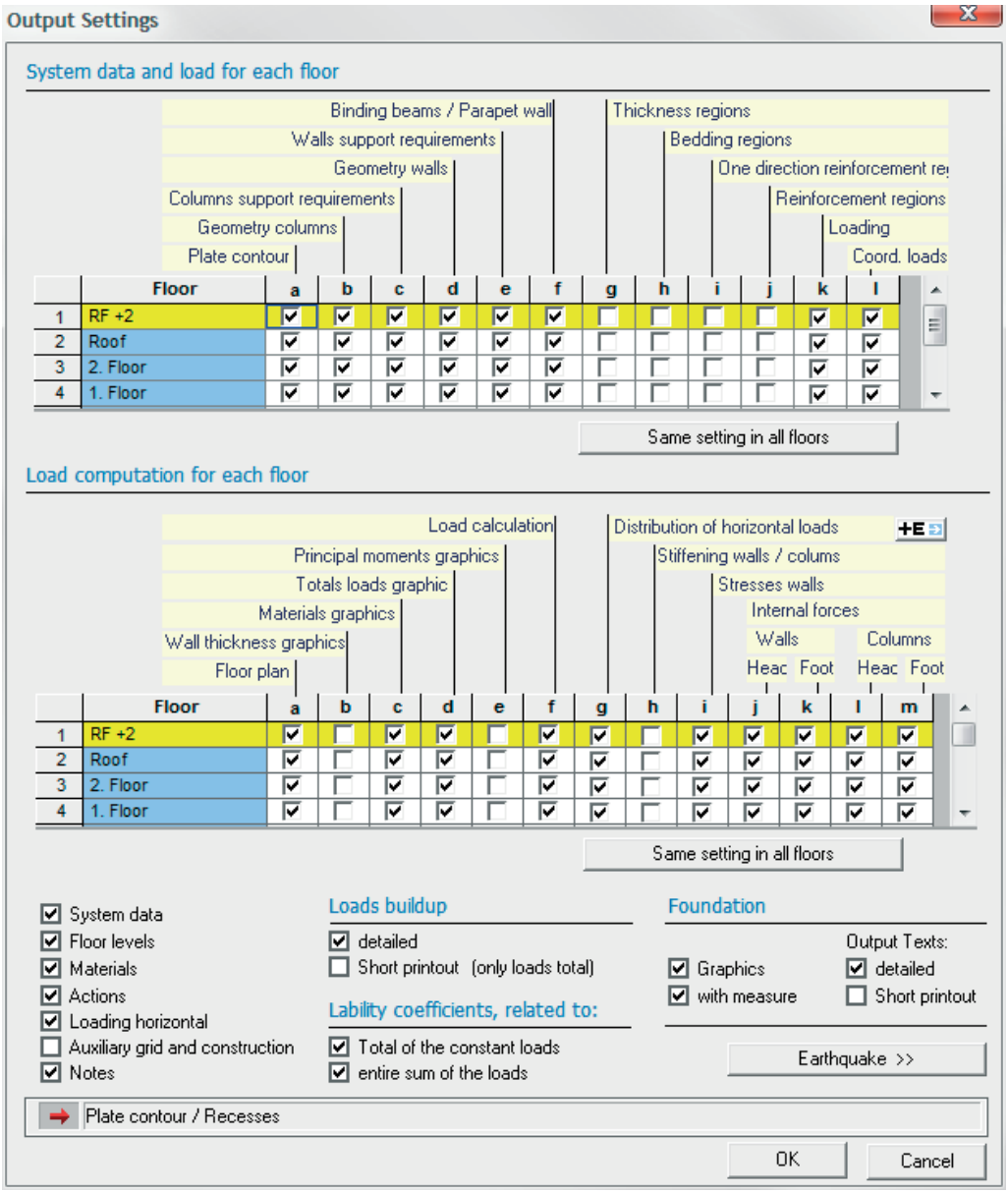
The output settings dialog is accessible via the Output tab in the main menu.

You can define the scope of the output via the available options in this section.

Check system and load data, graphics and load combinations for each floor as desired. Activating the option "Apply to all floors" applies the settings for this floor to all other floors.

You can add material data, comments, horizontal loads and auxiliary grids or structures to the output profile if required. The load combinations can be put out as a summary (only the totals of the loads) or in a more detailed report.

Graphics with or without dimensions, and texts, either in summarized or detailed form, can be selected and added to the output scope for the foundation.

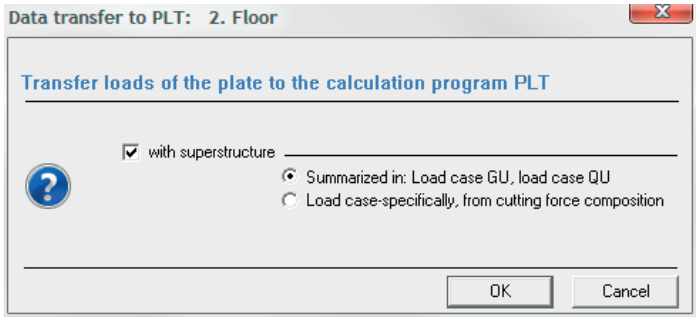


Design check in Frilo application

The "Design check in Frilo programs" option in the main menu provides access to the software applications for the design of individual components or particular verifications, if these applications are included in your software package.

Double-click on the desired application and select the corresponding component (click on the component(s), the cursor is shown as a square). Depending on the application, you must finish your selection with a right click and select "Exit" in the context-sensitive menu. Subsequently, the corresponding application is launched you can perform your design calculation there.

Plates by Finite Elements PLT	The currently active floor is transferred to the PLT application (see Floor selection). The live loads from the superstructure can optionally be transferred in a load case-specific manner (just check the corresponding option).
Panels by Finite Elements	Select a single of several connected walls and complete your selection with the help of the "Exit" option in the context menu. A dialog to select the loads (total G, P and load case-specific loads from stresses at the wall base) that should be transferred to the panel application SCN is displayed. Click OK to confirm your selection. The panel application is launched.
Masonry Verification MW and Masonry Design MWX	When selecting this option, the floor plans of all masonry walls are shown with green lines. Select a wall in the currently active floor with the help of the squared cursor (only walls made of masonry are available for selection).
Reinforced Concrete Column B5	launches the data transfer to the B5 application. You can also transfer multipart reinforced concrete columns.
Steel Column STS+	Select a steel column.
Timber Column HO1+	Select a timber column.
Punching Shear Analysis B6+	Select a column.
Continuous Beam DLT10	Select one or several connected contiguous beams per mouse click. Finish your selection with a right click and "Exit" (the application detects automatically when all possible contiguous objects have been selected). An intermediate dialog allows you to check the selected objects and the load cases. Confirm your selection with OK to launch the Continuous Beam application.



	Indirect support (with downstand beams and supporting parapets) is considered. At the point of the indirect support, a support with a minimum spring stiffness in vertical direction is defined on the beam.
Foundation FD+/FDS+	
Reinforced raft foundation FDR+	Select the component that should be transferred to a foundation application. You can select the loads to be transferred (actions from max./min. superpositions or loads applying at the wall base) in the displayed dialog.

Multi-part column

▸ Design check in F+L programs ▸ Reinforced concrete column B5

If the software application detects the multipart column in the defined building, a dialog is displayed allowing the user to select the column sections to be transferred.

Select per mouse click a column on a particular floor that should be transferred to the design application B5.

The software checks whether there are other columns underneath or above the selected one.

The following criteria are taken into consideration in this verification:

1. Same material for all column sections (reinforced concrete)
2. The centroidal axes of the columns above or below run in the area of the cross-sectional dimensions of the selected column.
3. Maximum number of sections: 10
4. Priority in the composition of multipart columns have the columns above the selected one.

First, suitable columns are searched in the floors above (max. 10).

If the maximum number of allowed sections is not reached, the software also searches in the floors underneath for suitable columns. The search is aborted when the maximum number of sections (10) is attained.

In the "Multi-part column" dialog, the currently active floor is marked with a dot. The user selects the column sections to be transferred in the displayed graphic per mouse click.

Columns that have been selected for transfer are marked yellow.

Columns that should not be transferred (disabled) are marked blue.

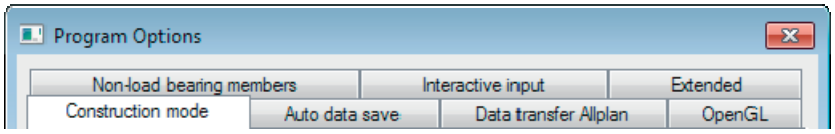
Additional menu items in GEO

This section describes the submenu items and functions that are not handled in the general document Menu Items.

Edit

Undo/Redo	previous definitions and changes can be undone
Load calculation building	→ see chapter Load calculation

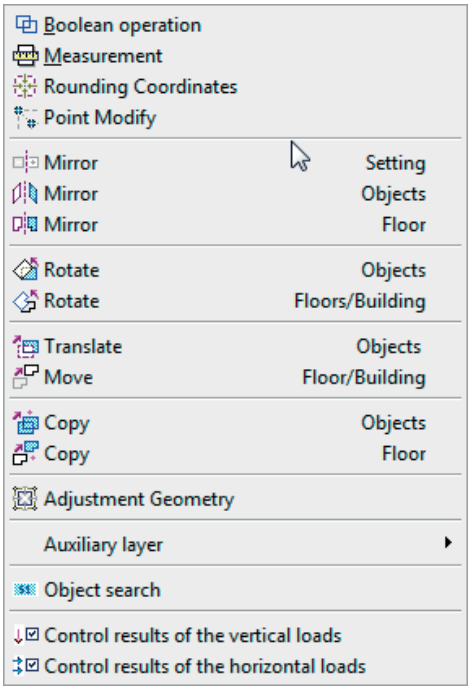
Options

Settings – GEO Building model	Construction mode, Automatic backup, Interactive definition (background grit, axes of coordinates)  - Data transfer Allplan: You can import data of partial drawings from ALLPLAN-CAD directly into the graphic (via the shortcut CTRL-T). In order to display an additional dialog with a list of layers, tick the option “Dialogue for the data transfer ...”. - Non load bearing members: Selection of colors and linetypes for the representation of non-bearing components. - Self-explanatory options: Hardware acceleration, layout ...
Colors	Settings for the use of colors for the individual components separately selectable for screen and printer. Default: FRILO pre-defined color sets. Save and load your own color settings.
Output setting	You can define the output scope via the available options in this section.
Basic building parameters	Height above sea level MSL, standard for the load assumptions, wind load.
Settings load calculation	Load calculation behaviour, FE-element size settings

Grf-Options

Visability of teh auxiliary layer	The menu item allows you to display or hide an imported auxiliary layer (from DXF...).
FE-critical points	If FE-critical points (e.g. wall end inaccurately defined) have been detected in the system, they are marked with a coloured circle. The user can hide the circles with the help of this options.

Tools menu



Boolean operation
The combination of surfaces via Boolean operations is a user-friendly feature for the graphical definition of complex outlines → see chapter "Fundamentals of the Graphical User Interface".

Measure
Measurement of distances and angles.

Round coordinates
You can adjust the rounding accuracy [cm] for coordinates and select whether the set rounding accuracy should apply to all floors or only to the currently selected floor.

Modify point
You can move a common point (identical coordinates) of several objects to a new position with the help of the mouse or by specifying the coordinates numerically. Click on the desired point and enter subsequently the target coordinates or click on the respective target position.

Mirror

Setting:
Option that allows you to preserve the original object/floor after mirroring (similar to copying).

Objects:
Like in CAD tools, you can mirror one or several objects. You can select several objects independently of the object type (columns, beams, walls etc.) and mirror them along a definable axis. The original object is preserved after the reproduction if the corresponding option (Mirroring - Settings) was checked.
Select the desired objects and complete your selection via the item ▸ "Exit" in the context menu. Define subsequently the axis (selection/definition of two points) along which the mirrored object should be reproduced.

Floor:
Activate this option to mirror the entire floor. Mirroring is done as described for the mirroring of objects, you need not select any objects, however.

Rotate

Objects:
Select the objects as described for the mirroring function. Select the centre of rotation. Rotate the object either via drag and drop with the mouse or by entering the angle of rotation into an input field that is displayed to the left of the "Numerical User Interface".

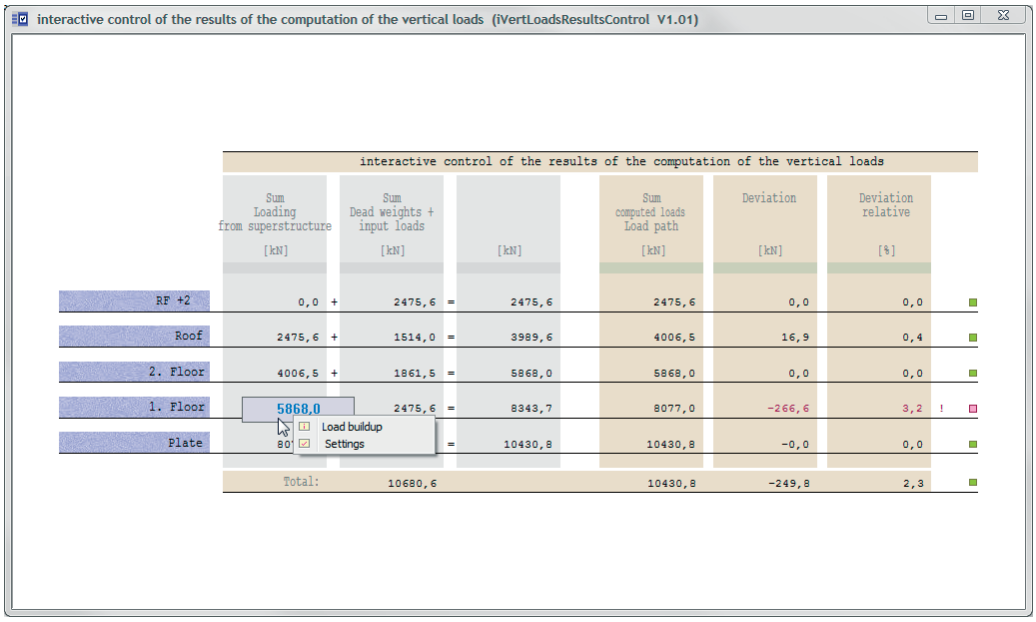
Floor:
Rotate all geometric objects of the currently selected floor including loads. Select the centre of rotation per mouse click. Perform the rotation via drag and drop with the mouse or by specifying the number of degrees numerically (a corresponding input field is displayed on the bottom of the screen).

Move

Objects:
Select the objects as described for the mirroring function. Move the objects via drag and drop or by specifying numerical values in the numerical user interface.

Floor
Moves an entire floor (horizontal). If you want to move a floor level up or down ▸ see chapter floor levels.

Copy	Select the objects/floor as described for the mirroring function. Position the copied object via drag and drop or by specifying numerical values in the numerical user interface.
Geometric alignment	This function provides for the geometric alignment of the plate if problems occur with the generation of the FE mesh. The alignment ensures that the FE mesh can be generated.
Move/rotate auxiliary layer:	You can move, rotate or scale a displayed auxiliary layer. Move: The cursor is shown as a square. Click on an any (distinctive) point of the displayed auxiliary layer. You should enable the capture function for this operation. Drag the layer with the help of the mouse. You can move the layer per mouse click on the target (zero) point or the numerical specification of coordinates. Rotate: Click to the point of rotation and rotate the layer with the help of the mouse or by specifying the angle of rotation numerically.
Object search	You would like to know the location of column S5 or wall W1.2. Enter the corresponding name or code in the field at the bottom of the window. The desired object flashes on the screen (exit with ESC).
Check vertical load transfer	Well-structured overview of the results of the vertical load transfer. Problem points are indicated by an oval marking. Click with the right mouse button on this marking to display the context menu. - Load differences, - Load combinations and - Settings Clicking to load differences displays a description of the cause of the difference with exact specification of the component in question. Via the Settings options, the user can set the deviation in per cent that should be considered as critical.



Icons – functions - results

In addition to the standard symbols, additional symbols / toolbars are available for the specific GEO functions.

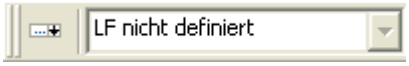
Tip: *The functions of the buttons are shown in a tooltip when you point with the mouse cursor to the button for a short time.*

Graphical User Interface

▸ see Graphical input, Input options

Load cases

▸ see Graphical input, chapter Loads



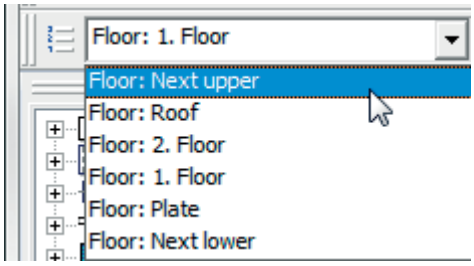
Load calculation

▸ see the chapter Load calculation

The calculation symbol appears in the colors yellow = not yet calculated or green = already calculated.

Floor selection

▸ see the chapter Floor selection



Input modes

Icons for the various input modes

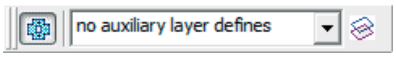
Capture function, background grid, line input, coordinate system, selection mode

▸ See also Graphical input, chapter Basic features of the graphical input

View toolbar

▸ see Graphical input.pdf

Auxiliary layers

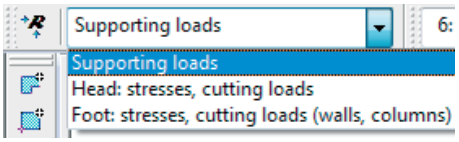


Hide/display an auxiliary layer, selection list to enable a particular layer, auxiliary layers management (import/export...).

Result graphics

The results of the load calculation can be displayed using the symbol "R" (results / summary / rendering).

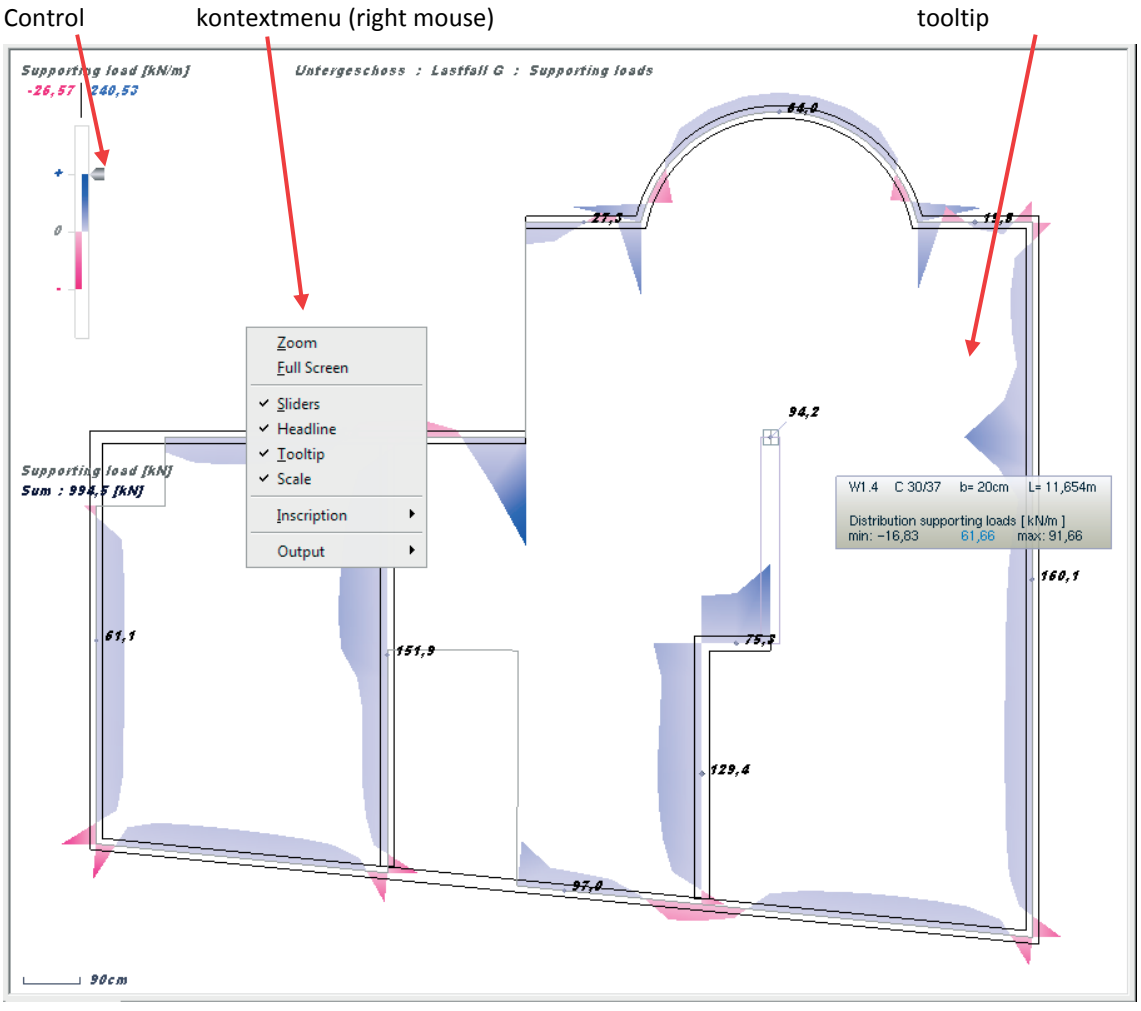
The following views of the results can be selected:
- bearing forces (with the bearing forces along the wall axes);
- stresses + cutting forces on the head (walls and columns);
- tensions + cutting forces on the foot (walls and columns);



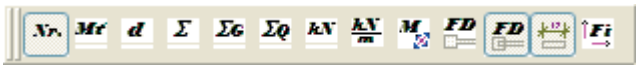
This is a load-specific presentation of the results.
The respective load case is selected via the list selection of the load cases (to the right of the "R" selection box).

For each type of representation of the results, special interactive control elements (slidable controls) are available in the graphic field at the top left (position movable) in the graphics window.

Tipp: *Move the mouse cursor over the individual graphics elements to display the respective data in tooltip form.*



Buttons for the display options in the graphics.



Display/hide of support reactions/stresses, component numbers and dimensions of vertical components, totals of loads (wall base, column base), totals of G-loads, totals of Q-loads, total loads in [kN], total loads in [kNm], principal moments, foundation design, individual foundations, foundation dimensions, eccentricities of the equivalent earthquake force.

Graphical user interface

The functions of the graphical user interface module are described separately in the document Graphical input.pdf.

Note: *The "Graphical User Interface" module is used in various applications (PLT, GEO, WL, SCN). We describe all graphical user interface functions in this document, particular functions may however not be available in some applications (e. g. there is no floor selection option in PLT and SCN).*

Depending on the application that you use in combination with the "Graphical User Interface", this application module allows you to enter a floor plan (outline, recesses), walls, columns (supports), upstand and downstand beams, parapets, thickness, foundation, reinforcement and supporting direction areas as well as loads either in graphical and/or numerical mode.

Schöck Isokorb

See document PLT-Schöck-Isokorb.pdf auf www.frilo.de (in german)