

# GEO – The Frilo Building Model

## Contents

Application options	2
Basis of calculation	4
Input	5
Graphical user interface	5
Numerical user interface	5
Importing data from CAD, PLT, WL, ASCII	5
Defining a building	6
Floors	7
Floor levels	7
Floor selection /copy	8
Copy loadcases to other floors	8
Foundation design - user-defined settings	9
Input loads for plate	10
Load transfer from building item	10
Horizontal loads – additional option GEO-HL and GEO-EB	11
Wind load parameters	13
Earthquake - additional option GEO-EB	14
Differences between DIN 4149 and DIN EN 1998-1	16
Involvement in the horizontal load distribution	18
Predefined superpositions of the horizontal actions	18
Calculation of loads	19
Output	20
Quantity survey – add-on module GEO-ME	21
Output Settings	22
Design check in Frilo application	25
Multi-part column	27
FE mesh	27
Additional menu items in GEO	28
Edit	28
Options	28
Grf-Options	29
Tools menu	29
Icons – functions - results	30
Result graphics	31
Result sections	32
Schöck Isokorb /HALFEN HIT Insolation element	32

Further information and descriptions are available in the relevant documentation [Graphical input](#) (description of the basic functions). In addition to the individual program manuals, you will find basic explanations on the operation of the programs in the document [Basic operating instructions-PLUS\\_eng.pdf](#).

## 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. (see also below: Importing CAD models with the FRILO BIM-Connector®)

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

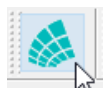
- The As-values (FE plate) or the reinforcement layout can be exported from the design applications to CAD programs (Allplot by Nemetschek AG, -isb cad- Glaser) to create reinforcement drawings quickly and easily.

### Interfaces

The components, including all loads and action groups, can be transferred to [FRILO design programs](#) (PLT slab program, B5+ column program, etc.).

SCIA The SCIA button can be used to automatically transfer the building to [SCIA Engineer](#) in SAF format. To do this, the SCIA program directory must be specified once.

CAD/BIM Any CAD models can be imported directly into the building model via the [FRILO BIM-Connector®](#).



## Additional options

### [GEO-HL](#): Horizontal load transfer

With the optional additional module GEO-HL, horizontal loads from wind, inclination and earthquakes can be determined. The wind loads can be calculated depending on the building geometry and the given wind parameters. Inclination loads are automatically generated from the calculated vertical loads.

The distribution of the loads on the stiffening components takes place in relation to the bending stiffnesses. A contribution to the horizontal load transfer can be defined both in groups (for example, reinforced concrete walls only) or for individual components.

### [GEO-EB](#): Earthquake

Determination of earthquake loads according to DIN EN 1998-1 and DIN 4149, 6.2.2 for Germany and ÖNORM B 1998-1: 2013-06/2017, 4.3.3.2 for Austria.

The determination of the earthquake loads takes place according to the simplified response spectrum method.

! This module requires GEO-HL

### [GEO-ME](#): Quantity survey

The quantity determination is controlled via the output profile. This allows you to define the table output individually for the individual materials and components as well as for individual floors or across the floors. Also, the export as an Excel folder or CSV file is possible.

## 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.

### Wall pillar

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.

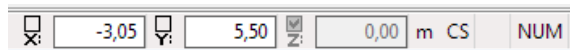
You can see the defined graphical objects immediately on the screen. The hide/display options for individual elements such as load arrangements provide for a well-structured overview of even highly complex systems.

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 from CAD, PLT, WL, ASCII

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.

Plates can be imported from PLT and the building system can be imported from the program WL - Windloads.

Allplan  
GLASER -isb cad-  
DXF  
FRILO ASCII  
Floor Data from PLT-item  
Load WL-Position  
Load transfer from building item

## 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, concentrated, line, area and temperature loads as well as load transfer from an other GEO-item.

### 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

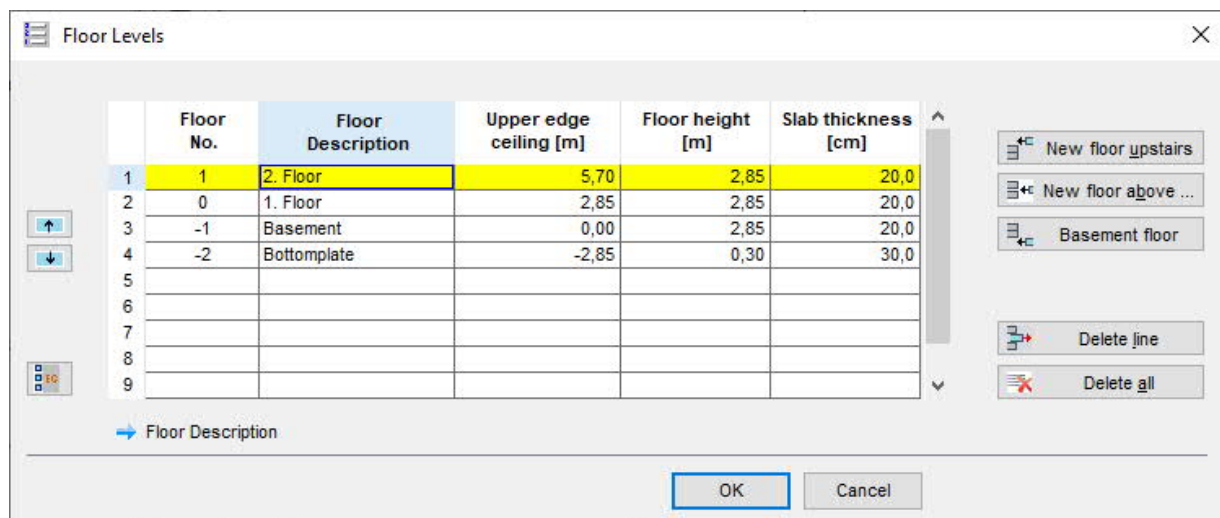
- define
- copy
- delete

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.

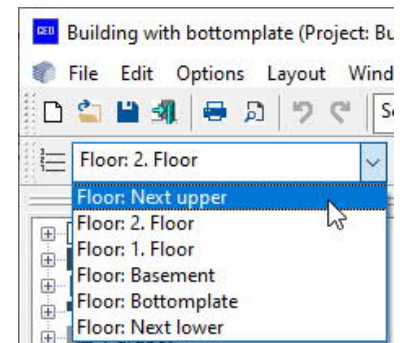
## Floor selection /copy

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

### Copy floor/generate new floor

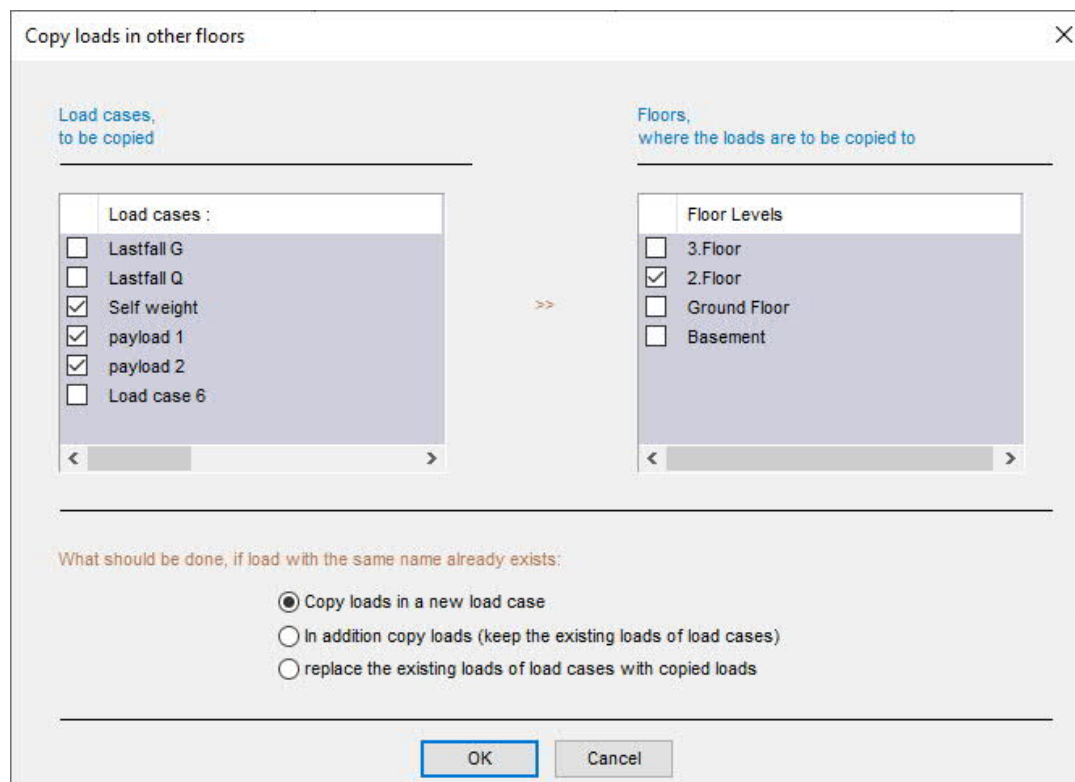
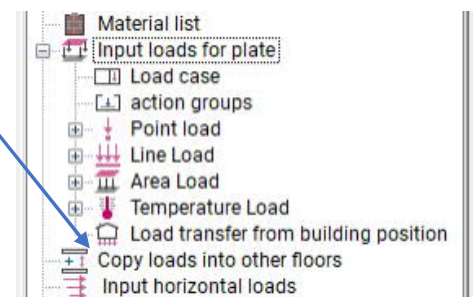
You can define additional floors via "Next upper floor" or "Next lower floor". This option copies the currently selected floor and displays the basic parameters dialog. You have the option to copy the loads too.

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

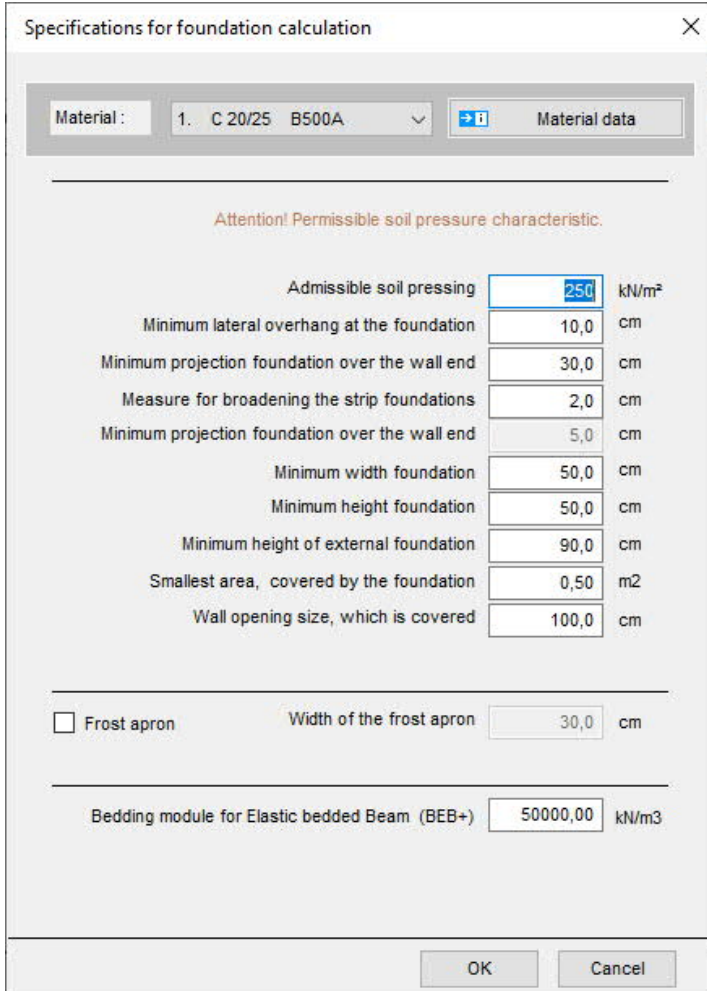
You can choose loadcases to copy them from the currently selected floor into other floors.





## Foundation design - user-defined settings

The menu item ► Defaults of foundation allows you to set parameters for the foundation design (permissible soil pressure...) and the Elastic Bedded Beam BEB+ (Bedding module).



Specifications for foundation calculation

Material: 1. C 20/25 B500A + i Material data

Attention! Permissible soil pressure characteristic.

Admissible soil pressing	250	kN/m <sup>2</sup>
Minimum lateral overhang at the foundation	10,0	cm
Minimum projection foundation over the wall end	30,0	cm
Measure for broadening the strip foundations	2,0	cm
Minimum projection foundation over the wall end	5,0	cm
Minimum width foundation	50,0	cm
Minimum height foundation	50,0	cm
Minimum height of external foundation	90,0	cm
Smallest area, covered by the foundation	0,50	m <sup>2</sup>
Wall opening size, which is covered	100,0	cm

☐ Frost apron      Width of the frost apron 30,0 cm

Bedding module for Elastic bedded Beam (BEB+) 50000,00 kN/m<sup>3</sup>

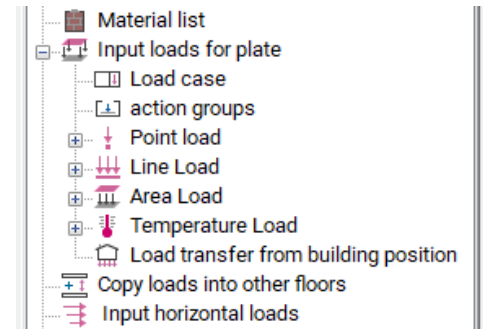
OK Cancel

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.

## Input loads for plate

The cross-program functions of the load input are described in the document "Graphical input" in the chapter [Loads](#). The Loads menu item includes the load case management, action groups, point, line, area and temperature loads.



## Load transfer from building item

You can access this functionality via the main tree:

*Input of loads for plate ▶ Load transfer from building item or via the menu item*

*File ▶ Import ▶ Load transfer from building item*

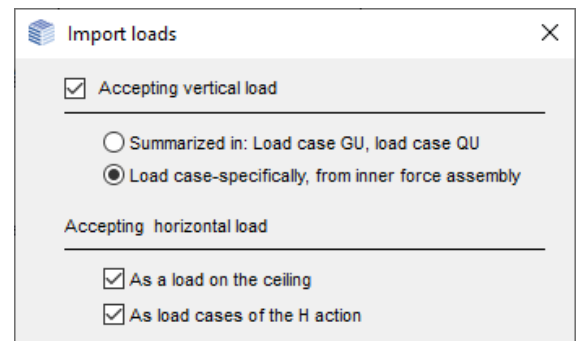
This function was developed for a situation close to practice when several sections of a building are built on a common plate (basement or underground garage ceiling).

You can import the load on the individually calculated building items into the currently active floor level (e. g. the common basement ceiling) of the currently accessed building item.

A window for project and item selection opens.

The load transfer is only available for items including calculated/stored results.

When importing, various options are available for vertical/horizontal load transfer.



## Horizontal loads – additional option GEO-HL and GEO-EB

With the option GEO-HL horizontal loads can be determined from wind, inclination - in addition to GEO-HL the additional option [Earthquake GEO-EB](#) can be purchased.

Access via the menu item ► Input horizontal loads

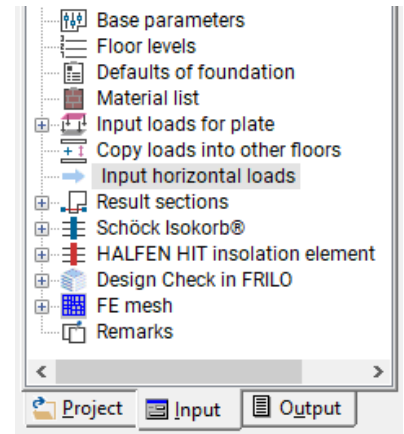
The horizontal loads are distributed on the bracing components of each floor based on their bending stiffness.

For separate walls or [wall pillars](#) of one wall, the stiffness of the weak axis at the walls can be switched off in the menu under [Options](#) ► H-Load distribution.

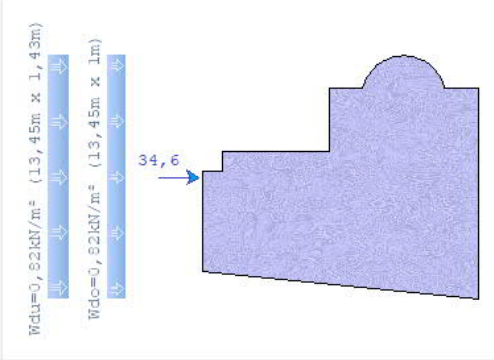
Shear field stiffness is not taken into account. You should note in this context that only non-sway buildings braced with walls can be calculated, because frame effects cannot be taken into account in the software application.

**Building joints:** If a continuous joint is planned in the building, the individual parts of the building should be calculated separately in different models. If a plate joint in a floor is entered as a complete joint, the user is informed of this requirement.

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



Input horizontal loads



x: -10,49 y: -1,34

Load case building

Nr.	Load case	Active	AG	Alt. Grp.
1	Wind Wx	<input checked="" type="checkbox"/>	9	1
2	Wind Wy	<input checked="" type="checkbox"/>	9	1
3				
4				
5				
6				
7				
8				
9				
10				
11				
12				
13				
14				
15				
16				

New

Copy

Apposed

Delete line

Wind

Wind (+/-)e (10%)

Inclination

Earthquake

Generate Wind

Horizontal loads

Nr.	Designation floors	Upper edge slab [m]	Floor height [m]	Wx [kN]	y [m]	Wy [kN]	x [m]
1	2. Floor	5,70	2,85	34,60	5,21	0,00	0,00
2	1. Floor	2,85	2,85	40,66	5,21	0,00	0,00
3	Basement	0,00	2,85	40,67	5,20	0,00	0,00
4	Bottomplate	-2,85	0,30	22,56	5,20	0,00	0,00
5							
6							
7							
8							

Equal input in all floors

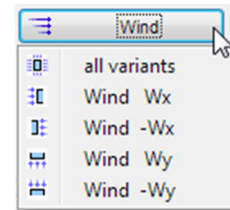
→ Floor description

OK Cancel

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".

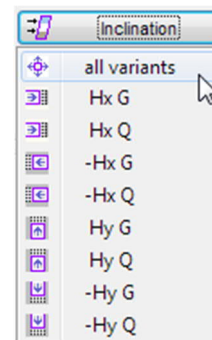
*Note concerning automatic generation:*

*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*

Wind (+/-)e (10%): Like "Wind", but not centered, but with an eccentricity of +/- 10%.

**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 (Button "New"). 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

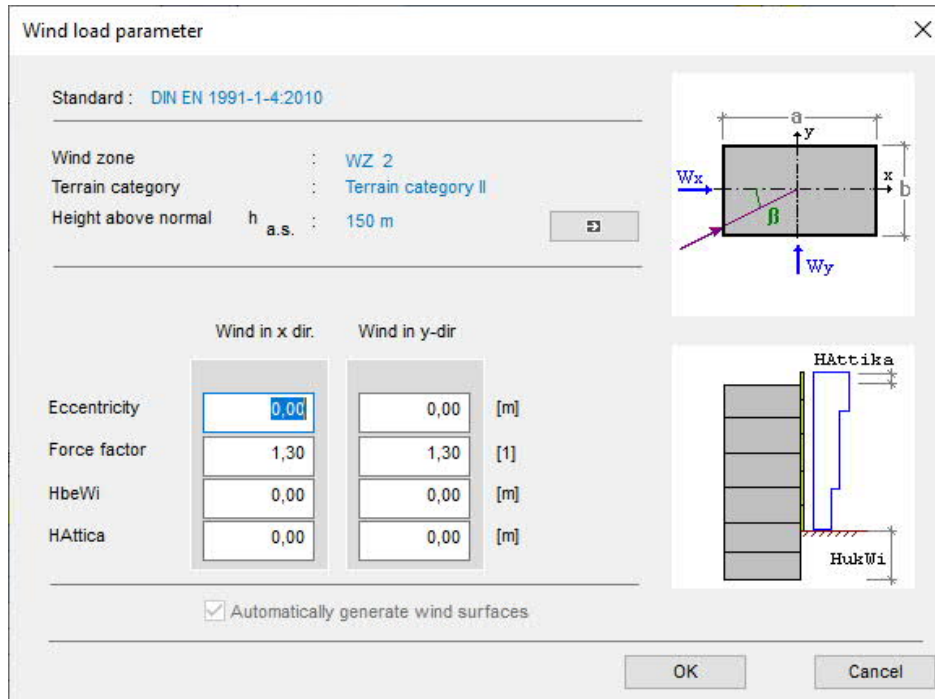
*Tip: You can print the graph of this dialog via the context menu options (right mouse button).*

### 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.



Wind load parameter

Standard : [DIN EN 1991-1-4:2010](#)

Wind zone : WZ 2

Terrain category : [Terrain category II](#)

Height above normal h a.s. : 150 m

Wind in x dir. Wind in y-dir

Eccentricity 0,00

Force factor 1,30

HbeWi 0,00

HAttika 0,00


☒ Automatically generate wind surfaces

OK Cancel

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.
- HbeWi** 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 - additional option GEO-EB

► Earthquake is available as additional option GEO-EB.

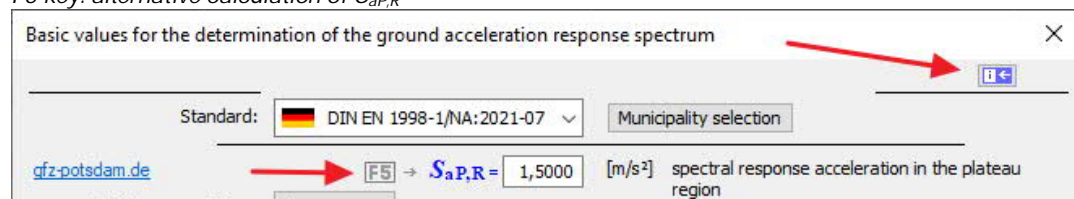
! This module requires the additional option [GEO-HL](#) Horizontal loads.

Determination of earthquake loads according to DIN EN 1998-1 and DIN 4149, 6.2.2 for Germany and ÖNORM B 1998-1: 2013-06/2017, 4.3.3.2 for Austria

The determination of the earthquake loads takes place according to the simplified response spectrum method.

Access: ► Input horizontal loads ► Earthquake button

*Tip:* Note the *i* button in the earthquake dialog, which can be used to display additional information.  
F5 key: alternative calculation of  $S_{aP,R}$



Standard selection: DIN 4149 is set as the standard.

*Note:* Although DIN EN 1998-1 (as of January 2023) has not yet been introduced, we have implemented this option in order to be able to carry out comparative calculations.  
See also differences between DIN 4149 and DIN EN 1998-1.

### 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,  $EI$  the average stiffness and  $\alpha_1$  the vibration coefficient, which depends on the number of floors.

Under normal conditions, the first equation produces slightly lower basic vibration times. The maximum equivalent earthquake forces result from the  $T_1$  value, which lies within the plateau of the response spectrum.  $T_1$  can optionally be entered as a user value.

### 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. This can optionally be set in the program (option "no rising gradient of the response spectrum up to  $T_b$ ").

### 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  $\lambda$  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  $M_T$  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 ( $0.05 \cdot L$  or  $0.05 \cdot B$ )

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



## Load cases in the FRILO building model

You can handle earthquake loads in the window "[Input 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.

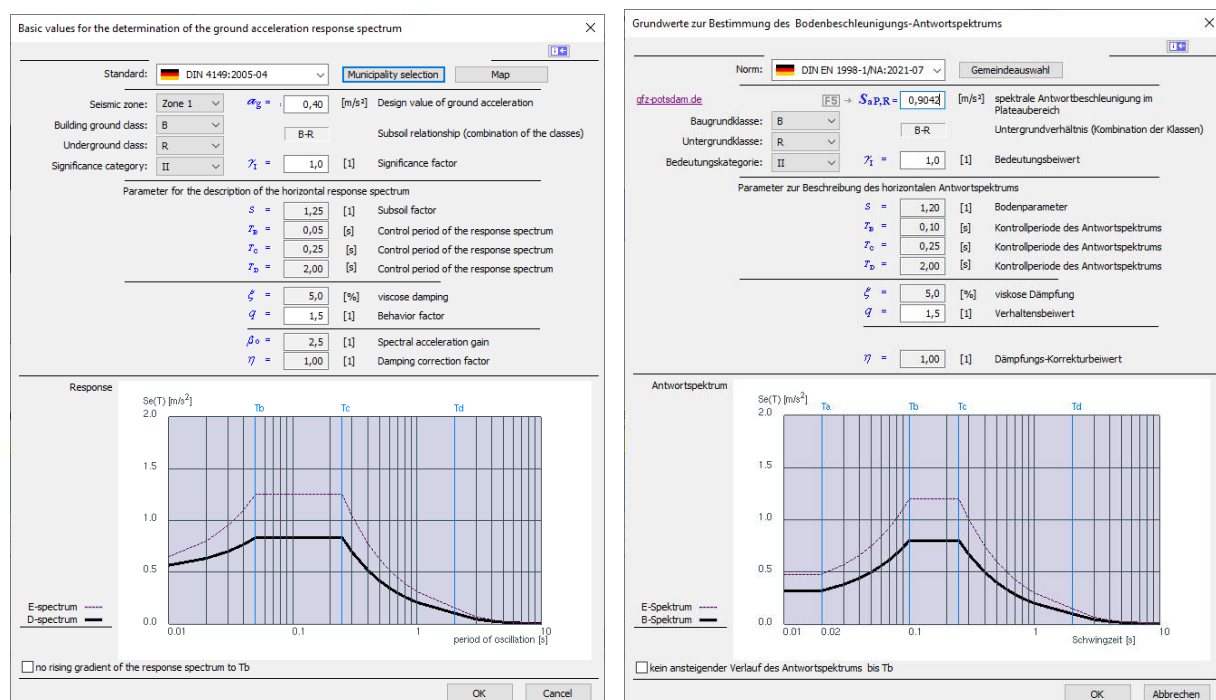
## Verifications

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.

## Differences between DIN 4149 and DIN EN 1998-1



The earthquake zone known from DIN 4149, from which the design value of the ground acceleration  $a_g$  resulted, no longer exists in DIN EN 1998-1.

The coefficient of the spectral acceleration  $\beta_0$  is also no longer available.

You still have to choose the building ground class yourself, the subsoil class can be entered directly or set via the municipality selection.

The seismic hazard map replaces the old zone maps. Here, the seismic hazard is shown spatially continuously with a grid of 0.1° from geographic longitude/latitude.



For the map display, the mean value of the amplitudes for the periods  $T = 0.1 \text{ s}$ ,  $0.15 \text{ s}$  and  $0.2 \text{ s}$  was calculated for each grid point. This value is called the "plateau spectral response acceleration"  $S_{aP,R}$ .

In the "normal case" a verification limit of  $S_{aP,R} \leq 0.6 \text{ m/s}^2$  applies. This boundary is shown as a **red contour** on the map (Note: the former Zone 0 is close to this contour).

DIN EN 1998-1: the value  $S_{aP,R}$

The value for the spectral response acceleration in the plateau area  $S_{aP,R}$  can be determined on the GFZ Potsdam website. To do this, click on the [gfz-potsdam.de](https://gfz-potsdam.de) link.

Here you can search for municipality names.

Tip 1: If several communities with the same name are found, you can move the mouse cursor over the entries in a selection list - the corresponding location is shown on the map as a blue dot.

A click on "Search" then displays the values and a diagram.

Tip 2: When moving the mouse over the map, the coordinates are displayed (use the zoom function). You can enter these coordinates, for example, in our alternative calculation dialog - see alternative calculation of  $S_{aP,R}$  via the F5 key.

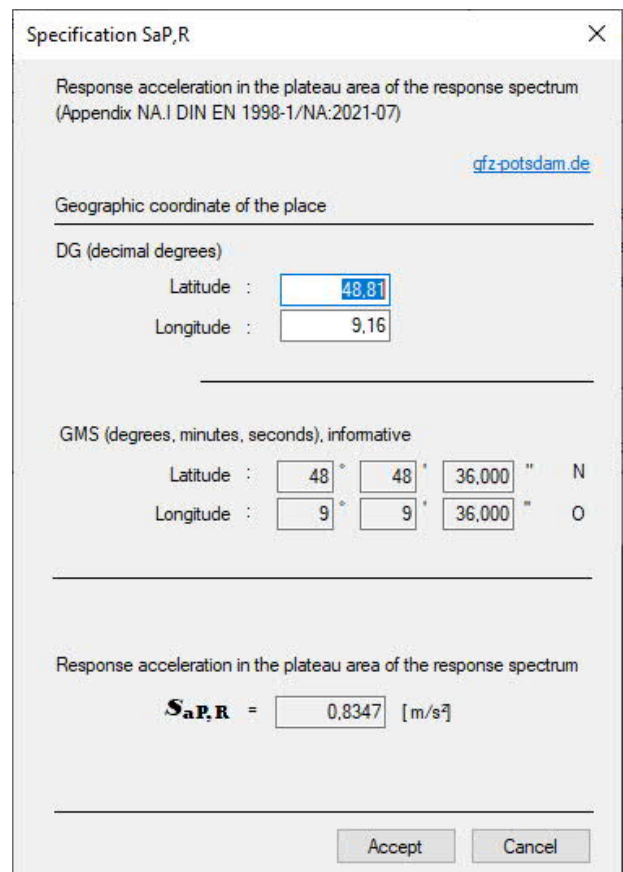
#### Alternative calculation of $S_{aP,R}$ via F5 key

For the  $0.1^\circ$  grid, there are table values in the informative digital appendix NA.J to DIN EN 1998-1, which are available as \*.csv files (download from [gfz-potsdam.de](https://gfz-potsdam.de)). According to the standard, the spectral accelerations between the calculated support points may be linearly interpolated.

In the program, you can therefore use the F5 key in the  $S_{aP,R}$  input field to call up the "Specification  $S_{aP,R}$ " dialog and enter the longitude and latitude here. The program then calculates the interpolated value for the mean return period of 475 years (file R475a\_mean.csv), which you can accept.

The coordinates must be entered in decimal format. For your information, the traditional degrees/minutes/seconds format is also shown.

*Tip: The coordinates are displayed in Google Maps if you click on the desired location in the map display.*



**Specification  $S_{aP,R}$**

Response acceleration in the plateau area of the response spectrum  
(Appendix NA.I DIN EN 1998-1/NA:2021-07)

[gfz-potsdam.de](https://gfz-potsdam.de)

Geographic coordinate of the place

DG (decimal degrees)

Latitude :

Longitude :

GMS (degrees, minutes, seconds), informative

Latitude :  °  '  " N

Longitude :  °  '  " O

Response acceleration in the plateau area of the response spectrum

**$S_{aP,R}$**  =  [m/s<sup>2</sup>]

Accept Cancel

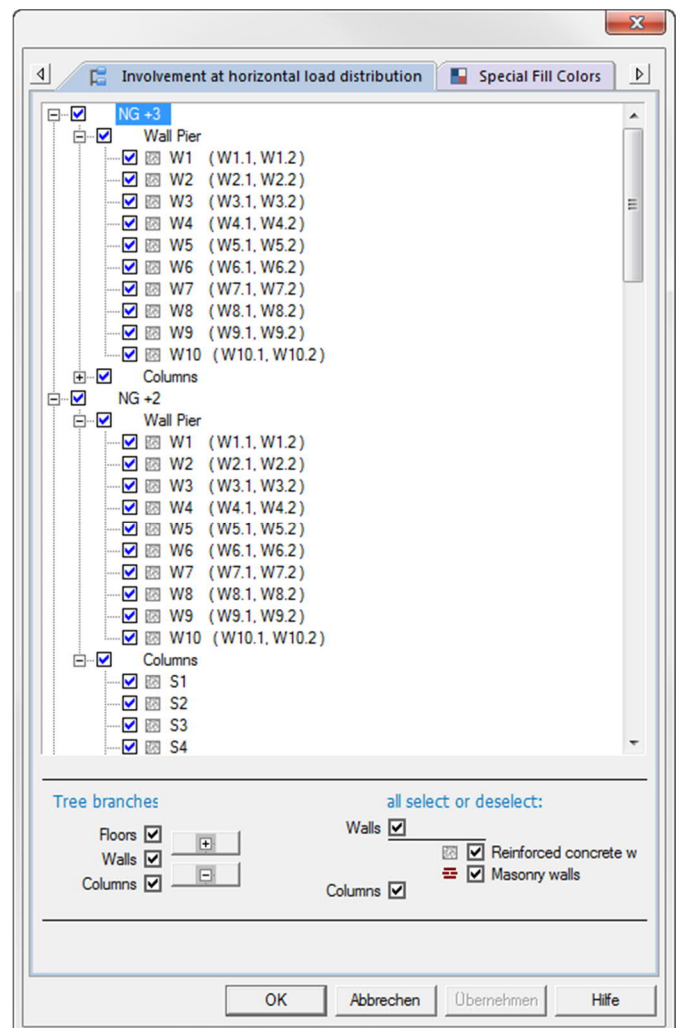
## Involvement in the horizontal load distribution

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

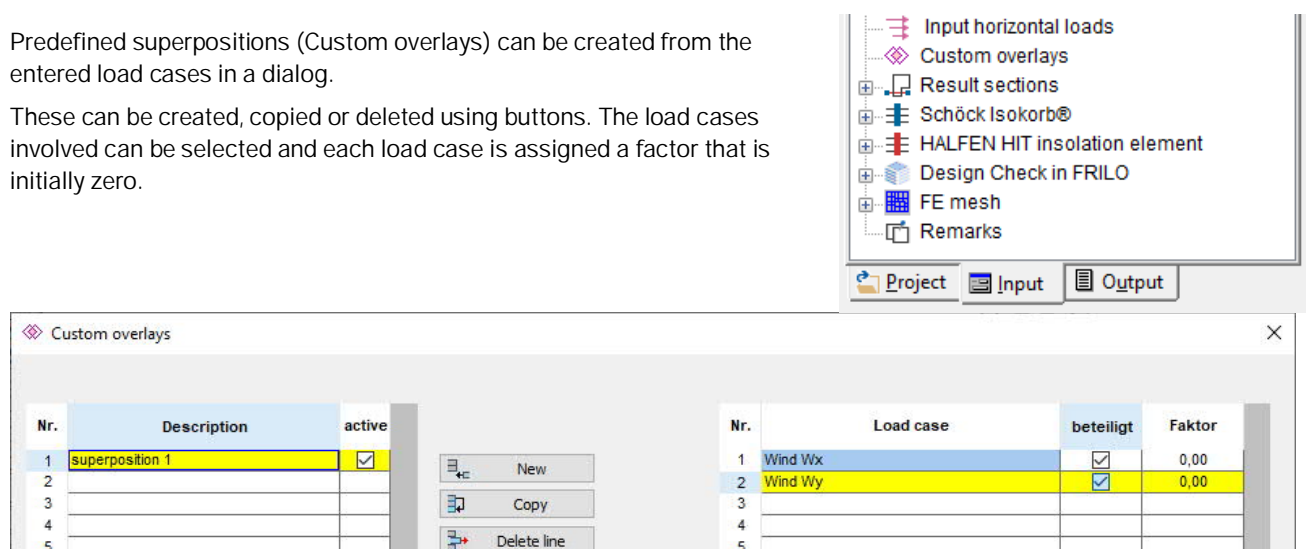
The fill colors can be individually defined for walls and columns which do not participate in the horizontal load transfer.



## Predefined superpositions of the horizontal actions


Predefined superpositions (Custom overlays) can be created from the entered load cases in a dialog.

These can be created, copied or deleted using buttons. The load cases involved can be selected and each load case is assigned a factor that is initially zero.



## Calculation of loads



Use the "Calculate" button  or ► Edit ► Load calculation 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

Via the menu [Options](#) ► FE mesh properties you can 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](#).

### Select plan format ...

Selection of a [plan format](#) (title block, A4 - A0, user-defined).

See [output & results in program PLT](#).

### 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.

Word

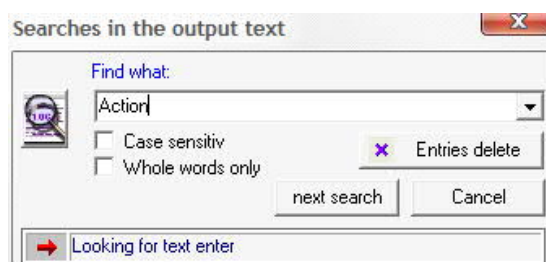
Output to MS-Word, if this software is installed on the computer.

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.



CSV file

CSV files can be saved, for example to import into an Excel spreadsheet.

XLSX file

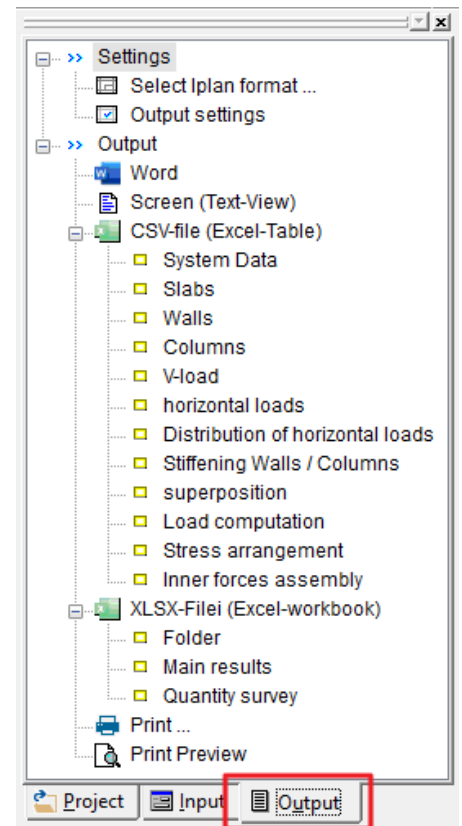
Output of main results and/or quantity survey in an Excel file.

Print all

Starts the [output on the printer](#).

Print Preview

Displays a preview of the printed page on the screen.



### SYSTEM DATA

Number of floors	: 5
Admissible soil pressing	: 250
<b>FLOOR</b>	<b>Search</b> F3
Floor	<input checked="" type="checkbox"/> Output settings
Label	<input type="checkbox"/> Print Preview
RF +2	<input type="checkbox"/> Print... Strg+P
Roof	<input type="checkbox"/> Print All
2. Flc	<input type="checkbox"/> Export to CSV-file (Excel)
1. Flc	
Plate	

### Results graphics

The functions for displaying the results graphs can be found in the chapter ["Results Graphics"](#).

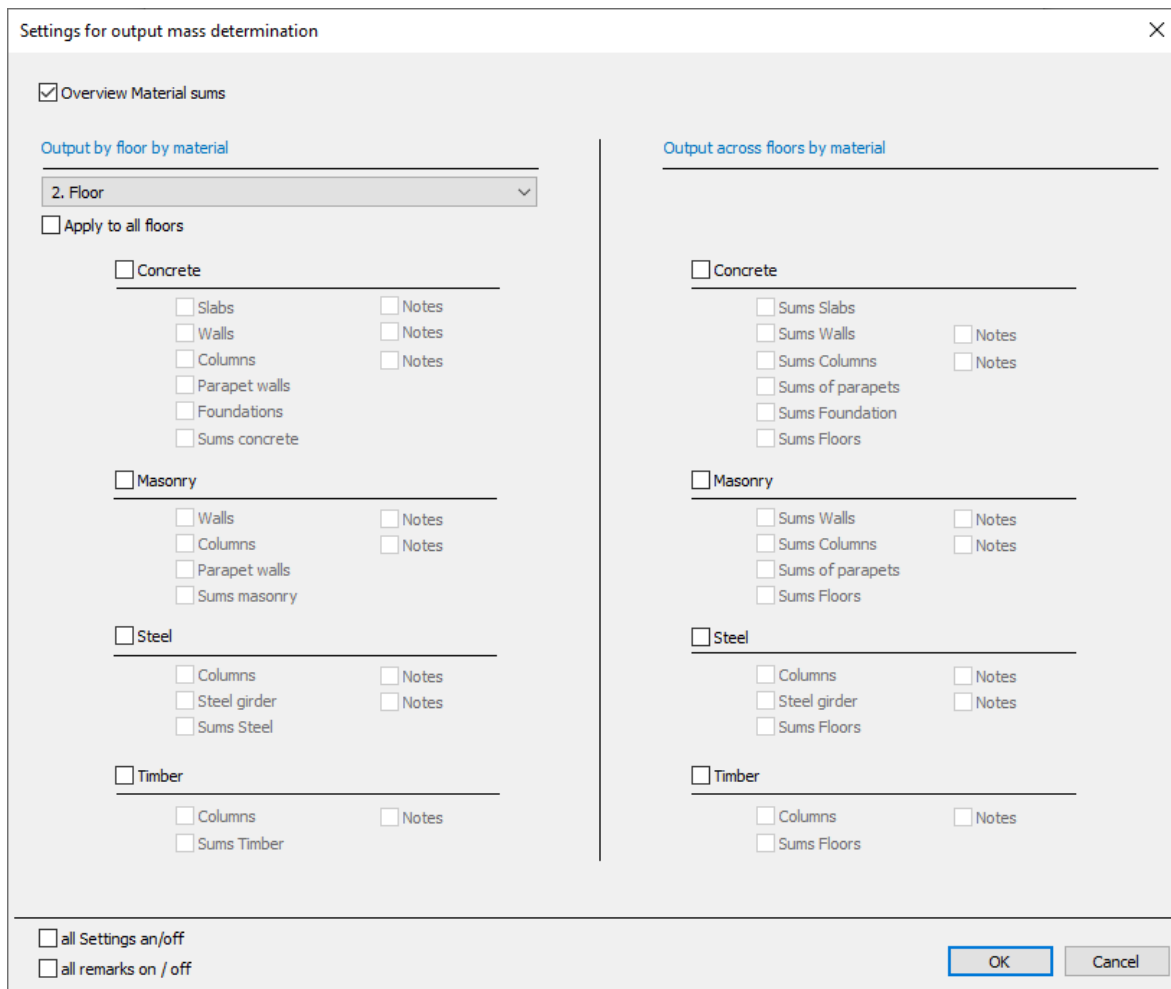
## Quantity survey – add-on module GEO-ME

If the GEO-ME add-on module is licensed, the quantities for the individual components can be determined separately by material and used as a basis for the tender, for example.

The calculation is possible for individual floors or across all floors. The quantities can also be exported to an Excel spreadsheet or a CSV file.

You can control the quantity survey via the [output settings](#).

►"Output" tab ►Output settings ►"Quantity surveying" button. In the dialog, select the storeys, materials and component types for which you want to determine the masses.



**Settings for output mass determination**

☒ Overview Material sums

**Output by floor by material**

2. Floor

☐ Apply to all floors

☐ **Concrete**

☐ Slabs  
☐ Walls  
☐ Columns  
☐ Parapet walls  
☐ Foundations  
☐ Sums concrete

☐ Notes  
☐ Notes  
☐ Notes

☐ **Masonry**

☐ Walls  
☐ Columns  
☐ Parapet walls  
☐ Sums masonry

☐ Notes  
☐ Notes

☐ **Steel**

☐ Columns  
☐ Steel girder  
☐ Sums Steel

☐ Notes  
☐ Notes

☐ **Timber**

☐ Columns  
☐ Sums Timber

☐ Notes

**Output across floors by material**

☐ **Concrete**

☐ Sums Slabs  
☐ Sums Walls  
☐ Sums Columns  
☐ Sums of parapets  
☐ Sums Foundation  
☐ Sums Floors

☐ Notes  
☐ Notes

☐ **Masonry**

☐ Sums Walls  
☐ Sums Columns  
☐ Sums of parapets  
☐ Sums Floors

☐ Notes  
☐ Notes

☐ **Steel**

☐ Columns  
☐ Steel girder  
☐ Sums Floors

☐ Notes  
☐ Notes

☐ **Timber**

☐ Columns  
☐ Sums Floors

☐ Notes


☐ all Settings on/off  
☐ all remarks on / off

## 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 "equal settings in all floors" applies the settings for this floor to all other floors.

 With the "+ E" buttons advanced detail options can be selected.

Use the buttons "General", "Loads horizontal", etc. to call up the output options for the respective data as separate dialogs.

In this way, you can set your output scope in detail and individually.

*Note: for quantity surveying, the additional option [GEO-ME](#) must be licensed.*

Output settings

System data and load per floor

Floor	a	b	c	d	e	f	g	h	i	j	k	l
1 NG +3	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
2 NG +2	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
3 NG +1	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
4 NG (Normalgeschoß)	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
5												

equal setting in all floors

Load computation per floor

Floor	a	b	c	d	e	f	g	h	i	j	k	l	m
1 NG +3	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
2 NG +2	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
3 NG +1	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
4 NG (Normalgeschoß)	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
5													

Graphic load/ Scale / Plan format

equal setting in all floors

General >>    Instability values >>    Foundation >>

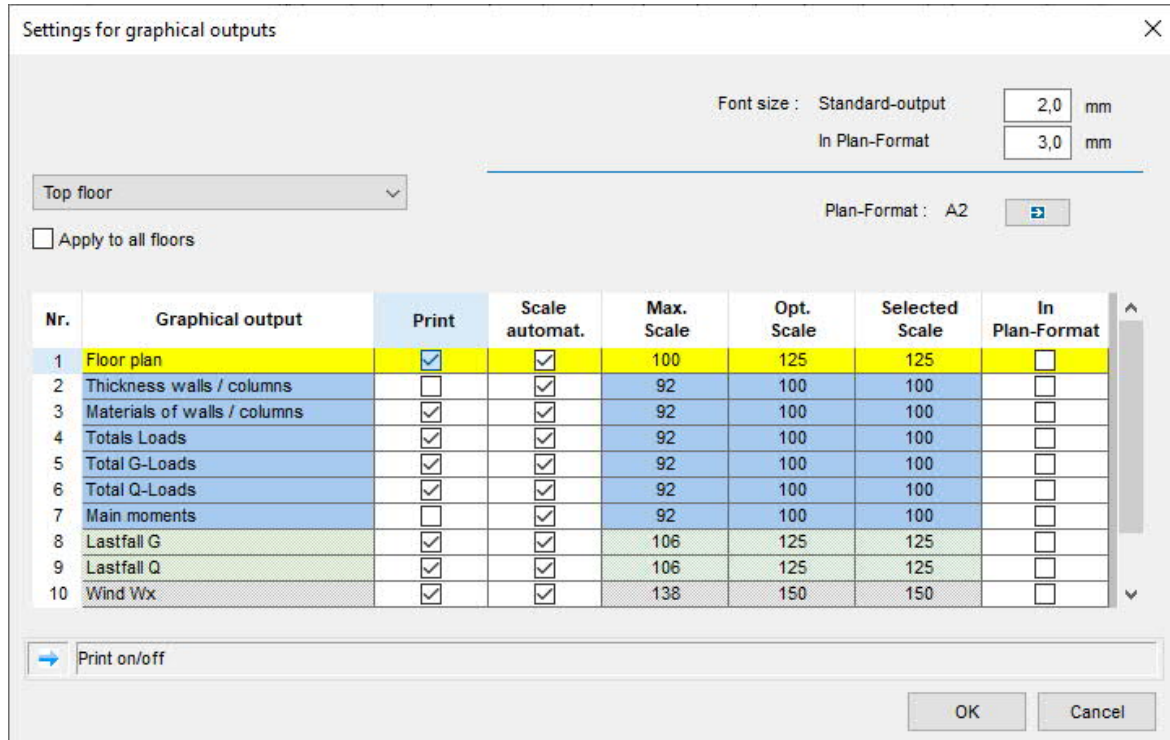
Loading horizontal >>    Quantity surveying >>    Earthquake >>

Plate contour / Recesses

OK    Cancel

Use the "Graphic load / Scale / Plan format" button to open the dialog for selecting the graphic output.

Here you select which graphics should be output (check print). The scale is set automatically by default, but you can optionally set it yourself (column "Selected scale").



Settings for graphical outputs

Font size : Standard-output 2,0 mm  
In Plan-Format 3,0 mm

Top floor ▼

☐ Apply to all floors

Plan-Format : A2

Nr.	Graphical output	Print	Scale automat.	Max. Scale	Opt. Scale	Selected Scale	In Plan-Format
1	Floor plan	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	100	125	125	<input type="checkbox"/>
2	Thickness walls / columns	<input type="checkbox"/>	<input checked="" type="checkbox"/>	92	100	100	<input type="checkbox"/>
3	Materials of walls / columns	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	92	100	100	<input type="checkbox"/>
4	Totals Loads	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	92	100	100	<input type="checkbox"/>
5	Total G-Loads	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	92	100	100	<input type="checkbox"/>
6	Total Q-Loads	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	92	100	100	<input type="checkbox"/>
7	Main moments	<input type="checkbox"/>	<input checked="" type="checkbox"/>	92	100	100	<input type="checkbox"/>
8	Lastfall G	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	106	125	125	<input type="checkbox"/>
9	Lastfall Q	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	106	125	125	<input type="checkbox"/>
10	Wind Wx	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	138	150	150	<input type="checkbox"/>

Print on/off

OK Cancel

The settings can apply to each floor or, optionally, to all floors. To output a graphic in plan format, select this option - see also settings in [settings for plan format](#).

## Instability values

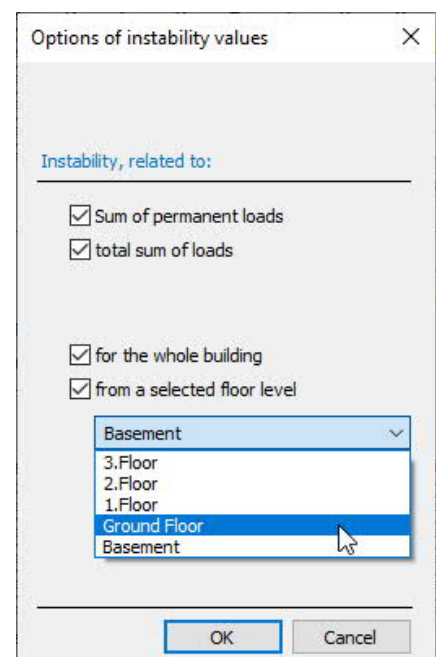
In a dialog you can choose whether the instability calculation should refer to the entire building (related to the lowest floor) or to the floors from a selected floor (or to both). This selection is useful, for example, for buildings with different levels of softness/stiffness.

The verifications are carried out for both rotation and translation.

*Note on the output:* When verifying the translation, the formula from the standard was adapted so that the verification is calculated against the same limit value as for the rotation.

## Varying bending stiffness over the building height

The decisive bending and torsional stiffnesses of the stiffening components combined to form the overall member are initially those of the lowest floor. In the case of bending stiffnesses that change over the height of the building, an equivalent stiffness is determined with the same top deformation as a result of the unit load in the respective direction (FaxX, FakY). Jumps in torsional stiffness cannot be taken into account. When determining the stiffness, state I is assumed.



Options of instability values

Instability, related to:

☒ Sum of permanent loads  
☒ total sum of loads

☒ for the whole building  
☒ from a selected floor level

Basement ▼  
3.Floor  
2.Floor  
1.Floor  
Ground Floor  
Basement

OK Cancel



### Select layout format (also Select plan format)

The following settings and output options are available here:

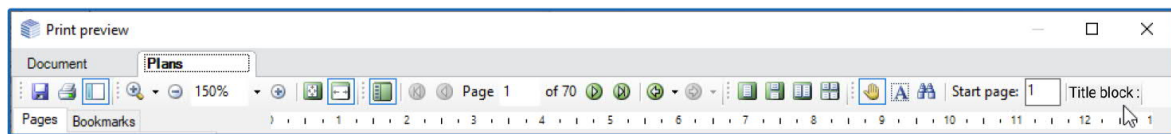
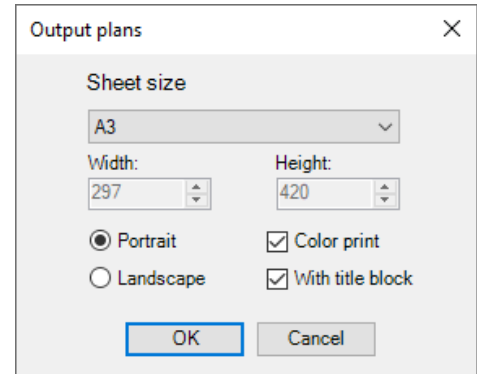
Choice of a plan format (A4 - A0, user-defined).



Unter Schriftgrößen können Sie die Größen in mm für Standard- und Planausgabe individuell einstellen.

Font size you can set the sizes in mm for standard and plan output individually.

The graphics of a selected plan format are displayed in the [print preview](#) via the "Plans" tab and can also be printed out here. In the overall statics document of the [FRILO Document Designer](#), these are attached at the end of the statics document.



**Tip:** The "Plans" tab is only displayed if at least one graphic is selected for printing in the output profile and the "In plan format" option is set.

**Note:** In this case, the entire document should be printed out on a printer that can print both formats / paper sizes, otherwise the plan format may be printed out truncated. You can also print the pages with the plan formats separately on a suitable printer.

For the graphics in plan format, in the FRILO Control Center [FCC](#) ▶ Extras ▶ Page layout ▶ Title block templates, title blocks can be defined, which can also be stored on a project-specific basis (as a project property - see also the example [PLT - Define title block](#)).

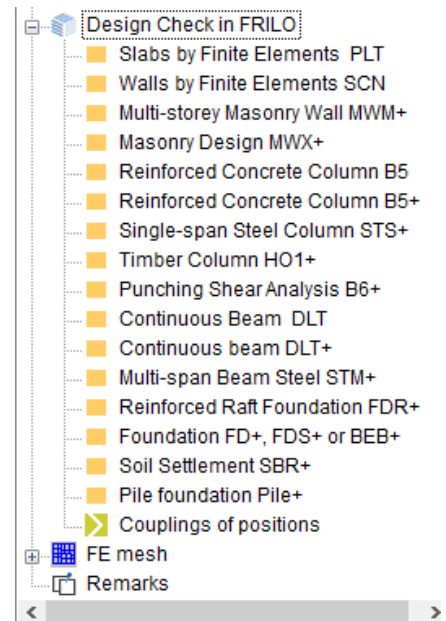


## Design check in Frilo application

The "Design check in FRILO" 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.

Interface to [SCIA Engineer](#) – see [Application options](#).



### Slabs 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).

### Walls by Finite Elements SCN

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 MWM+ and

### Masonry Design MWX+

Select a wall in the currently active floor with the help of the squared cursor (only walls made of masonry are available for selection). In an intermediate dialog you can select the loads to be transferred to the design program (total G,P or load case specific).

### Reinforced Concrete Column B5/B5+

launches the data transfer to the B5/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, a wall end or an inner wall corner for the transfer to B6+. In the case of a base slab, the supports that stand on the slab are displayed after selecting this function.

The determined loads are displayed before the handover. The loads can be adjusted here if necessary.

When the wall end and wall corner are transferred, the load introduction length is also displayed here and can be adjusted.

For an explanation, additional information about the determined load can be viewed via the info button.



### Continuous Beam DLT/DLT+

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.

	Optional: Determination of the relevant panel width according to EN 1992-1-1 point 5.3.2.1 for the design.
	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.
Multi-span Beam Steel STM+	Select a steel beam to transfer to the STM+ design program.
Foundation FD+/FDS+/FDR+/BEB+	Select the component that should be transferred to a foundation application respectively BEB+. You can select the loads to be transferred in the displayed dialog.
Soil Settlement calculation SBR+	After clicking, the loads and dimensions of all foundations are transferred to the soil settlement program. In an pre-dialog, all foundations are listed with their column or wall affiliation before the data is transferred.
Pile Foundation Pile+	Select a reinforced concrete column to transfer to the Pile+ design program. The return data from Pile+ also includes a proposal for the vertical spring stiffness of the column, material or cross-section. This suggestion can optionally be confirmed/accepted.

### Couplings / Link to the item of the design program

If a component is transferred from GEO to a design program for dimensioning, the item of the design program in the building model is recorded as a coupled (linked) item. When the design program is called up again with this component, this item is suggested for the start of the design program. Couplings that are no longer relevant can also be removed again via the submenü ► *Couplings of positions*. The settings for couplings can be controlled under the menu item ► [Options](#) ► *Coupling items*.

## Multi-part column

### ► Design check in FRILO ► Reinforced concrete column B5/[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).

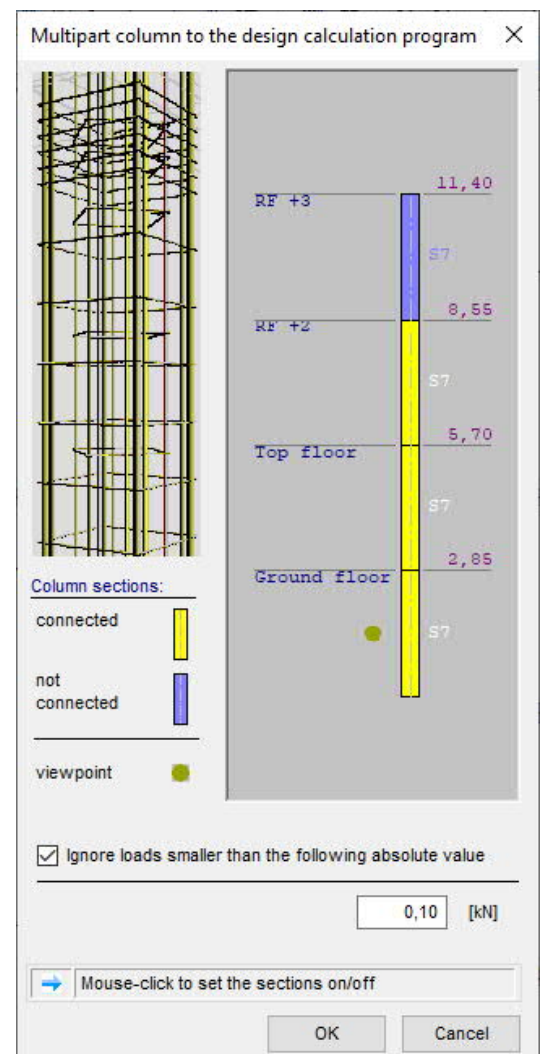
If the maximum number of allowed sections is not reached, the software also searches in the floors underneath for suitable columns. The search is aborted when the maximum number of sections (10) is attained.

In the "Multi-part column" dialog, the currently active floor is marked with a dot.

Select the column sections that are not to be transferred in the graphic shown with a mouse click:

these columns are then displayed in "blue" (deactivated).

The columns that are to be transferred are shown in "yellow".



## FE mesh

You can use the "Generate" submenu item to generate an FE mesh for the slab of the current floor level ([similar to PLT](#)) - this way you can identify and correct any problems before the [transfer to PLT](#).

The values/[options](#) set via the "FE mesh: Properties" dialog are used here.

## Additional menu items in GEO

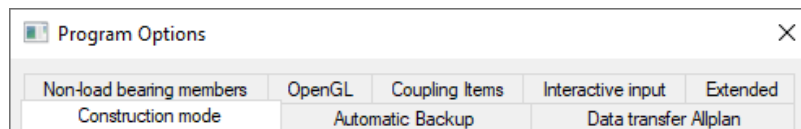
This section describes the submenu items and functions that are not handled in the general document [Menu Items](#).

### Edit

Undo/Redo	previous definitions and changes can be undone
Load calculation building	→ see chapter <a href="#">Load calculation</a>

### Options

#### Settings – GEO Building model



<a href="#">Construction mode</a> ,	Useful functions for graphical input (zoom, full screen, orthogonal, horizontal, vertical, exit, cancel) are combined in a movable symbol window and are therefore more readily available than with the standard symbols above.
Non-load bearing members	Selection of colors and line types for the representation of non-structural components.
<a href="#">Automatic Backup</a> ,	Optional activation of a periodic data backup in order to restore the last saved position after program interruptions.
<a href="#">Coupling Items</a>	The connection / coupling to items in the design programs can be switched on or off for each program.
<a href="#">Interactive input</a>	Background grid, 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 ...".
OpenGL, Extended	The hardware acceleration for the 3D display is switched on by default. The last window layout (size / division) can be saved by default and appears when the program is started again. The menu item "Customize toolbar" is hidden by default, but can be made visible here.
<a href="#">Colors</a>	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.
<a href="#">Output setting</a>	You can define the output scope via the available options in this section.
<a href="#">Basic building parameters</a>	Height above sea level MSL, standard for the load assumptions, wind load.

[FE mesh properties](#)

FE-element size settings.

See also [PLT-FE network properties](#).

Tip: a fine-meshed FE network means longer computing times. In phases with frequent calculation runs, it makes sense to temporarily set a coarser network for very large buildings in order to keep the calculation time short. The computer capacity can also reach its limits if the network settings are too fine. If the element results are only calculated for the center points of the elements, this also saves computing time. The network settings are also transferred when the data is transferred to PLT.

## H-load distribution

GEO: For single walls or for wall pillars that only consist of wall segments of one alignment, you can optionally neglect the stiffness about the weak axis for the stiffening calculation.

The reference of the H-force can be made either to the shear center or the center of mass.

The H-load distribution can optionally be based on wall surfaces instead of stiffness.

## SCIA Engineer

The program directory in which SCIA Engineer is installed is queried when the SCIA interface is called up for the first time and can be viewed or changed here.

## Grf-Options

These functions are generally explained in the "[Graphical Input](#)" document.

## Tools menu

These functions are generally explained in the "[Graphical Input](#)" document.

## 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](#)

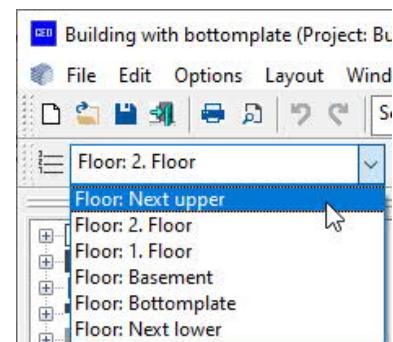
### Input modes

#### 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](#)

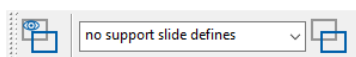


### View toolbar

► see [Graphical input.pdf](#)



### Auxiliary layers



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

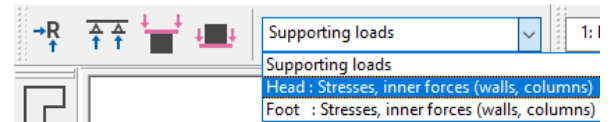
## Result graphics



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);

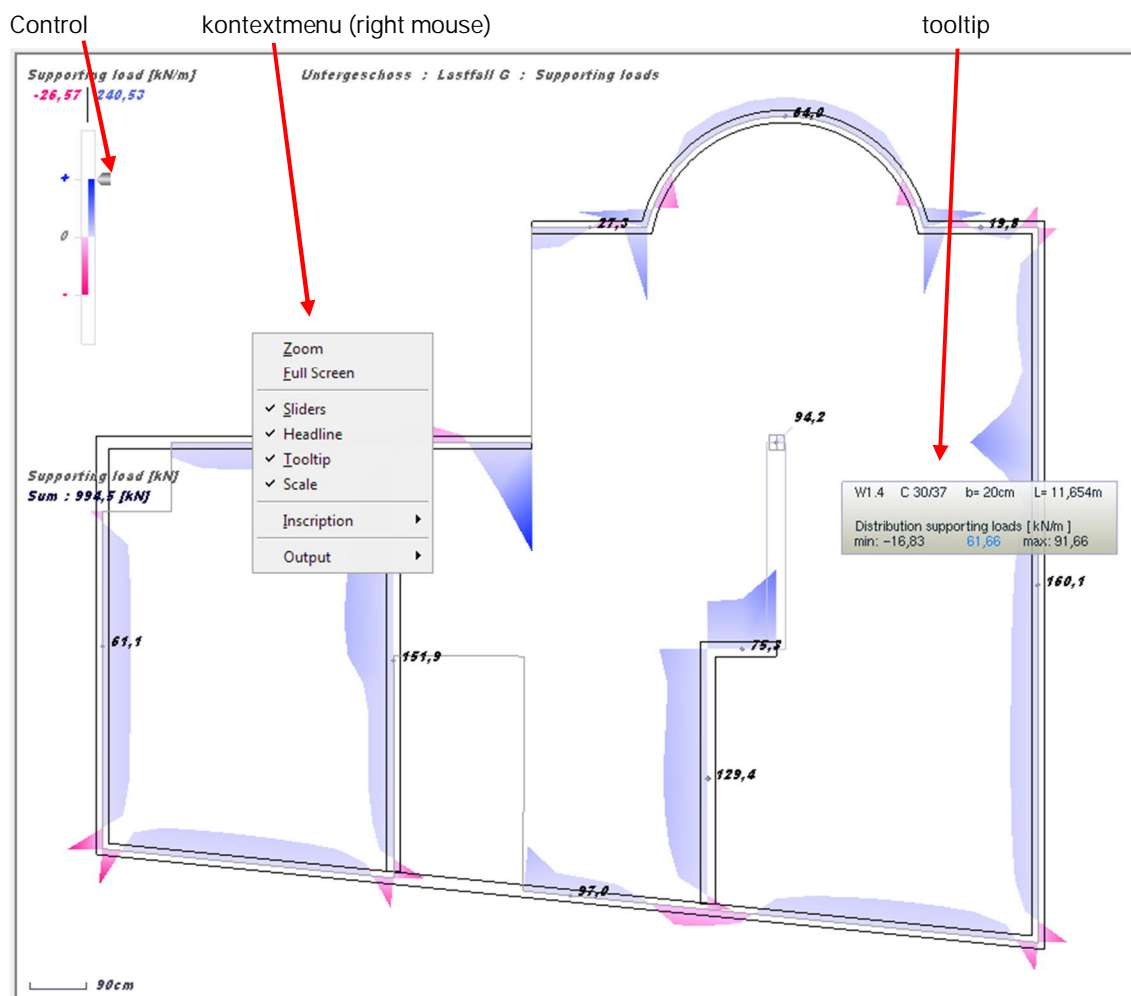


This is a load-specific presentation of the results.

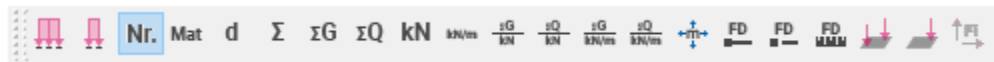
The respective load case is selected via the list selection of the load cases (to the right of the "R" selection box).

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, Load transfer from superstructure, 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, foundation loads, eccentricities of the equivalent earthquake force.

## Result sections

Call via main selection ► Result sections

This functionality makes the result sections from imported PLT items appear visible in GEO and enables the processing and preparation of the result sections for the transfer of a storey level to PLT.

Description: See [PLT - Result Sections](#).

## Schöck Isokorb /HALFEN HIT Insolation element

Determine a Schöck Isokorb or a HALFEN HIT Insulation element based on existing internal forces.

See document [Plt-waermedaemmelemente.pdf](#) auf [www.friilo.eu](http://www.friilo.eu) (in german)

Similar to the definition of a result section, a line can be defined along which a suitable element is suggested based on the moments and shear forces.

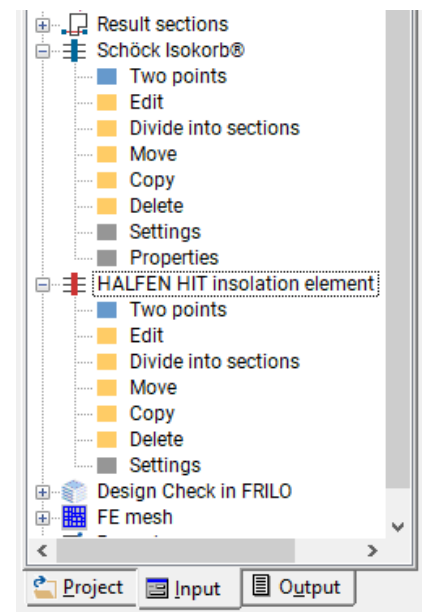
The labeling and the optical representation of the element in the graphic can be selected via the settings.

From Release 2022-2 2 a new version of the Schöck Isokorb® program option is integrated. The internal forces determined in PLT along the defined connection line are evaluated via a Schöck web service. The web service is based on the Schöck Scalix® design software and uses the input parameters to return an economical Isokorb® installation to PLT.

### Websites

<https://www.schoeck.de>

<https://www.leviat.com>



**Note:** When entering via two points, the direction of entry is important to define which side of the line the balcony is on.