

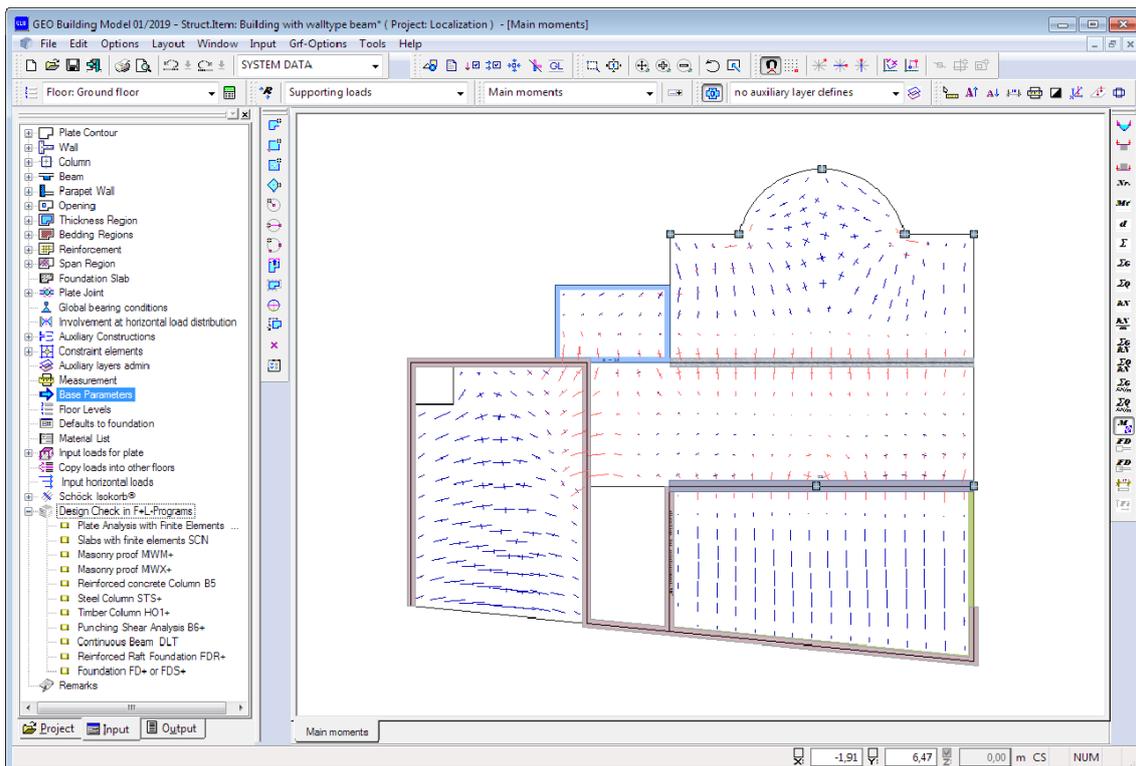
Graphical input

FRILO Software GmbH

www.friilo.com

info@friilo.com

As of 05/03/2019



Graphical input

Input module for the applications Building, PLT-Plates by finite elements, SCN-Panels/Sheets by finite elements and WL-Wind loads.

Contents

Input options	4
Numeric and graphical input of coordinates	4
Basic tips for the graphical input of data	4
How to access the individual input functions	5
General graphical input functions.....	7
Basic parameters	9
Definition of a foundation slab	11
Floor selection	11
Basic building data	12
Material selection (walls and columns)	13
Material list	14
Plate contour	14
Wall input	15
Wall properties	17
Input of columns	21
Column parameters	23
Bearing (SCN)	25
Upstand/downstand beam	26
Upstand/downstand beam parameters.....	28
Parapet	29
Opening	31
Areas - Regions	31
Thickness / bedding area	31
Reinforcement	33
Bearing direction area.....	34
Plate joint	35
Global bearing conditions	36
Loads	37
Load case.....	38
Concentrated load	39
Line load.....	40
Distributed load	42
Temperature load	44
Load transfer from building item (<i>Only available in the Building application</i>)	44
Basic features of the graphical input	45
Auxiliary structures	46
Auxiliary grid	47
Auxiliary layers	48

Constrained geometric elements.....	50
Capture function	51
Boolean operation	51
Numeric input of coordinates	52
System of coordinates.....	53
Input modes	55
Context-sensitive menu	56
Settings in the context-sensitive menu.....	57
Grid / Background mesh	57
View toolbar.....	58
Construction mode	58
Additional menu items	59
Graphical options	59
Tools.....	59
Three-dimensional representation (OpenGL).....	61
Automatic data backup	61

Software application with integrated graphical user interface:

[FRILO Building Model](#)

[PLT – Plates by finite elements](#)

[SCN – Panels by finite elements](#)

Basic Documentation – Overview

In addition to the individual program manuals, you will find basic explanations on the operation of the programs on our homepage www.friilo.com ▶ Support ▶ Articles/Information ▶ Basic operating instructions.

Input options

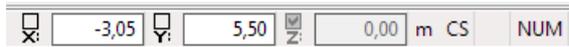
Important note:

The application module "Graphical input" is used in various applications (PLT, GEO, WL, SCN). We describe all graphical input 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 input", this application module allows you to enter a floor plan (outline, openings), walls, columns (bearings), upstand and downstand beams, parapets, thickness, bedding, reinforcement and bearing direction areas as well as loads either in graphical and/or numerical manner.

Numeric and graphical input of coordinates

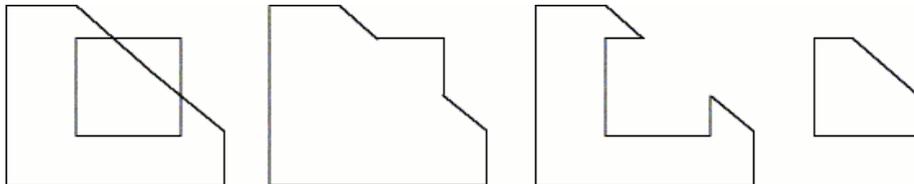
In general, you enter coordinates using the [numeric input of coordinates](#) or a DXF file. Alternatively you can enter them graphically using the mouse and a suitable [background grid](#). The cursor coordinates are continuously displayed on bottom right of your screen (either with relative or absolute reference).



Basic tips for the graphical input of data

- Boolean operations

The easiest way to produce more complex outlines is to link rectangles, polygons and circles by [Boolean operations](#).



- Auxiliary lines, background grid

You can accurately construct geometrical shapes with your mouse when using [auxiliary lines](#), [background grids](#) and the [capture functions](#).

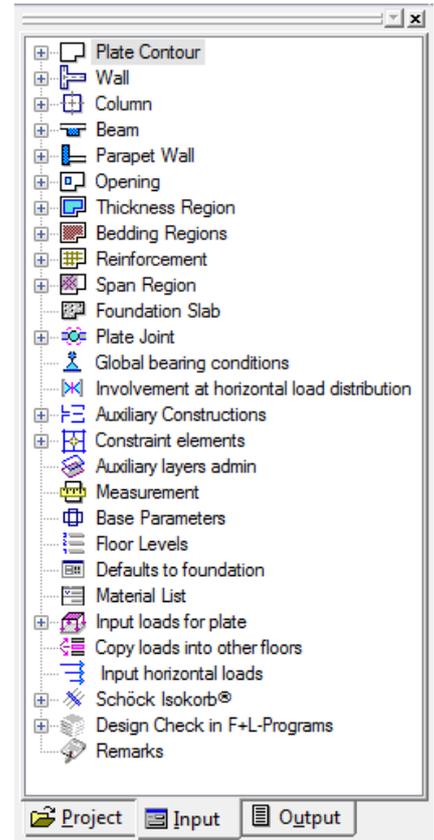
How to access the individual input functions

The menu items of the graphical input module are available either in the [context-sensitive](#) menu (right mouse button), the main tree or in the submenu of the Input item. You can select the individual input options with a simple mouse click (Plate contour, Wall...)

Alternatively, you can display the tool bar "Input objects" and click on the corresponding icon (display the tool bar via ▶ Options ▶ Customize toolbars ▶ Toolbars).

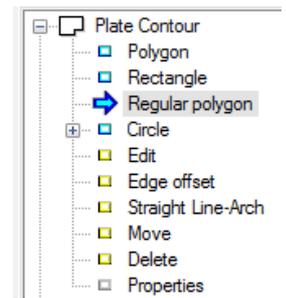
Tip: Move the cursor over an icon to display the tool tip describing its function.

	Input of the plate outline
	Input of walls
	Input of columns
	Input of upstand and downstand beams
	Input of parapets
	Input of openings
	Input of thickness regions
	Input of bedding regions
	Input of reinforcement regions
	Input of bearing direction regions (Span region)
	Input of a foundation slab (<i>only available in the Building application.</i>)
	Plate Joint . Defining pinned joints, expansion joints etc.
	Input of global bearing conditions
	Selection/deselection of the involvement at horizontal load distribution (<i>only in Building application</i>)
	Input of auxiliary constructions
	Definition of constraint elements
	Insert auxiliary layers (<i>not available in the SCN application</i>)
	Measurement of distances and angles inside the graphic
	Input of Base parameters
	Floor levels (<i>in GEO only</i>)
	Defaults to foundation (<i>in GEO</i>)
	Material list
	Input of loads
	Copying loads to other floors
	Input of horizontal loads (<i>in GEO</i>)
	Design check in FRILO-Programs. Depending on the program (GEO , PLT), individual components can be transferred via this interface to the corresponding FRILO design program.
	Remarks - Input/editing of brief notes which optionally can be printed along with the data.



Depending on the selected input function (plates contour, wall, column...) the relevant input options (polygon, rectangle...) are displayed in the form of icons and submenu items of the main tree (see illustration on the right).

Tip: Any functions of the toolbar are also accessible via menu items in the context-sensitive menu (right mouse button) or the submenu of the main tree. You can move the toolbars on the screen to any place or hide them at any time (reset via ▶ Options ▶ Customize toolbars).



General graphical input functions

Many operations such as the input of a polygonal chain, a rectangle, an arch structure or the deletion of objects are done in the same or a similar way. Therefore, we explain these "common" functions centrally in this document. Of course you can define these objects also by entering numerical coordinates - see the chapter [numeric input of coordinates](#).



Polygon

The function allows you to enter polygon lines. Position the individual polygon points with a mouse click or via the [coordinate input](#) in the graphic window. After having entered the final polygon point, click with the **right mouse button** and select in the [context-sensitive menu](#) "Exit". This option also completes the polygonal chain or other two-dimensional objects (e. g. plate contour or openings).



Rectangle

This function allows you to draw rectangles (plate contour, areas...). Click on the first corner point, zoom out a rectangle with the mouse and click on the second (diagonally opposite) corner point. Alternatively, you can easily define the object via [coordinates](#).



Regular Polygon

The number of corners can be selected as well as the reference point for the input (corner or edge center).



Circle - definition via the radius

First, define the centre point of the circle per mouse click or enter the coordinates. After this, enter the coordinates of a point on the circle perimeter or drag the mouse away from the centre point and click again to define the desired radius.



Circle - definition via the diameter

Enter the circle by clicking on an arbitrary point on the perimeter. After this, zoom out the circle with the mouse and define the second perimeter point by clicking once more. See also [coordinate input](#).



Circle - definition via three points

Similar to the definition of a circle via two points, you can exactly position and define a circle via three points.



Edit / Edit outline

Select the desired outline with a mouse click. The individual polygon points of the outline are displayed and you can move them by drag and drop with the mouse (tip: the cursor changes its look depending on whether you position it on a polygon point or a line).

The various [input modes](#) and the [numeric input of coordinates](#) are very helpful in this connection.



Interactive editing of an outline: the mouse cursor changes when you move it over an outline. You can set a new corner point by clicking on the outline and move this point by keeping the right mouse button pressed.

Any changes only become effective when you finish the process (perform a right mouse click and select "Exit" in the context-sensitive menu).



Edge offset: shift outline (enlarge/reduce)

This function allows you to shift all edges of an outline outwards or inwards by a specified dimension, i. e. the outlined surface becomes greater or smaller. Click on the outline and specify the desired dimension in the displayed dialog.



Straight Line-arc / Rise height (Convert line into arc/arc into line)

Allows you to convert a line into an arc and vice versa or change the curvature by specifying a rise size.

Converting a line into an arc:

Click on the line to be curved to display the dialog "Enter rise size". The maximum positive/negative rise size is displayed. Specify a value within these limits in the rise size input field or click on "Positive maximum (+)" or "Maximum (negative)" to set the maximum value.

You can see the positive/negative direction (referenced to a line from point 1 to point 2) in a sample graphic that is displayed.

Converting an arc into a line:

In order to convert an arc into a line, select the arc and specify the rise size "0" in the dialog.



Move outline

In order to move an outline click on it and drag it to the desired position using the mouse. Click again to drop it there and finish the moving operation.



Delete

Click on an object to delete it. If you want to delete several objects (e. g. several columns) at a time, use the selection modes [crossing or window](#). (You can select several objects by zooming out a rectangle that surrounds them →see the chapter "Input modes").

Note: You can restore unintentionally deleted objects with the help of the "Undo" function.



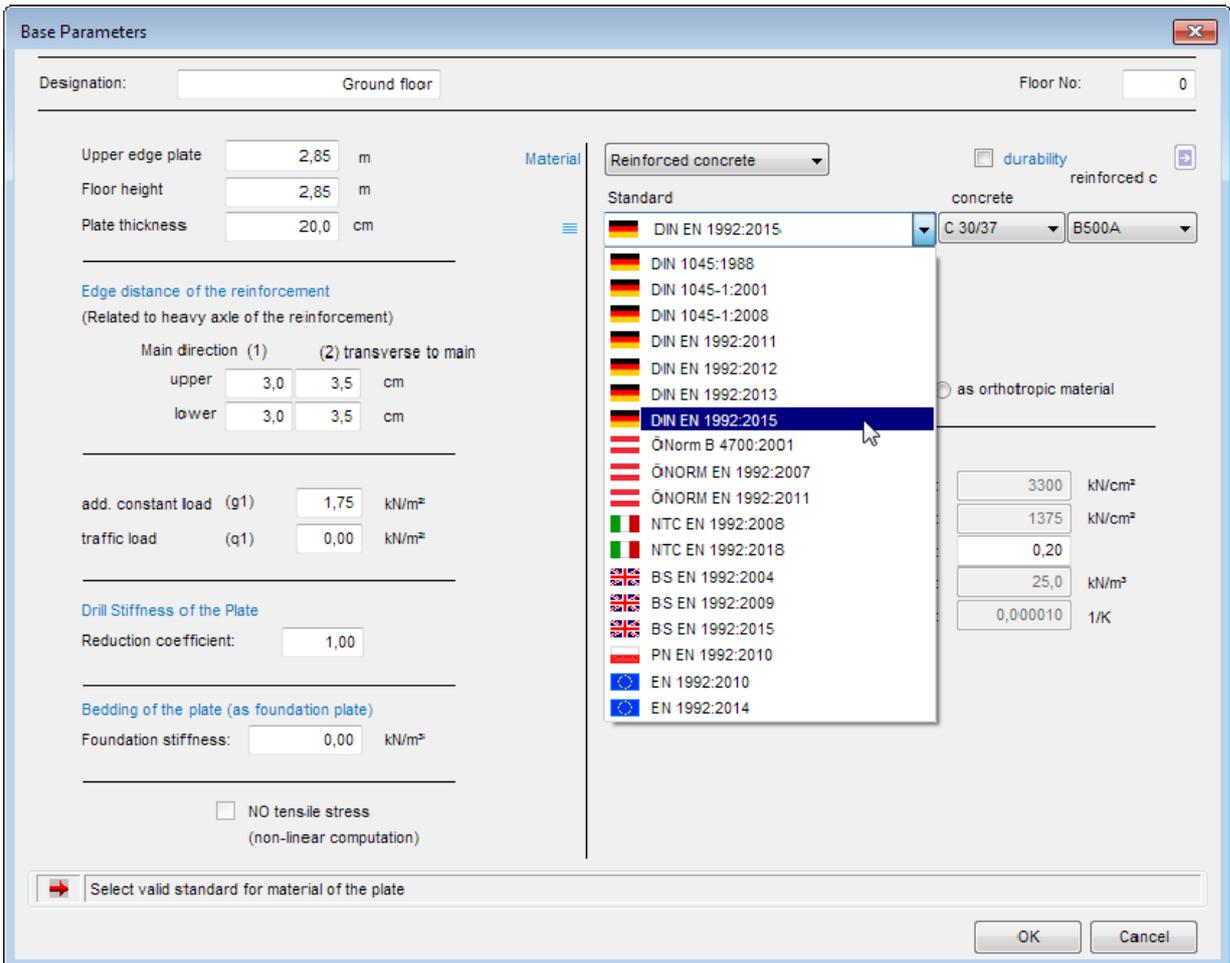
Properties

The parameters (properties) of the selected object (e. g. a column or a thickness area) are displayed in a window where they can be edited. Click first on this symbol and then on the corresponding object to display the pertaining input dialog.

Basic parameters

The corresponding item in the main tree accesses the basic parameters dialog.

Note: *This dialog is displayed automatically when you define the first plate contour. With applications that support several floors (e. g. Wind loads WL, Building), you can define the basic parameters for each floor individually.*



Designation	You can assign a name to each floor.
Floor no.	You can define a "start number" (default = 0) for the first floor to be entered (e. g. - 2). All additional floors are defined via the floor selection. The floor numbers are assigned by counting incrementally up (next floor above) or down (next floor below).
Note:	<i>Select a floor for editing or define a new floor via the floor selection, → see also the chapter Floor selection.</i>
Upper edge plate	The input of a ceiling top level is relevant in applications with automatic wind load generation (Wind loads WL, Building). You can define a height above sea level and reference a ceiling top level to this height via "Options – Basic building data ".
Floor height	The specification of a floor height is relevant in applications with automatic wind load generation (Wind loads WL, Building). In addition, this value is used for the automatic bearing rigidity computation of columns and walls in the PLT and Building applications.
Plate	Thickness of the plate in [cm].

Load g1/q1 In the Building application, you can additionally define a load that acts permanently upon the total surface (g1) and a non-permanent load (q1) in this window.

Material Currently, only reinforced concrete is available as material for the plate design. The material parameter section allows you to define the Poisson ratio.
In the PLT and Building applications, you can define also other isotropic or orthotropic materials. However, only an action-effect calculation is performed for these materials not a design calculation.

Isotropic material

Enter the modulus of elasticity and the Poisson ratio.

Orthotropic material

Enter the moduli of elasticity 1 / 2, the Poisson ratio and the angle.

Durability Tick this option to enable the button that accesses the durability dialog. Click on this button  to display the dialog. See document [Durability – Creep Coefficient and Shrinkage Strain](#).



Standard Select the desired standard from the list.

Torsional rigidity Reducing coefficient of the torsional rigidity. The coefficient is adjustable between "0.00 (no torsional rigidity) and "1.00" (full torsional rigidity).

Plate bedding Enter the subgrade modulus for the entire foundation plate.
Define areas with different beddings (elastic beddings) by entering [bedding areas](#).
Reference values for the subgrade modulus according to Lang ·Huder, Bodenmechanik und Grundbau, fourth edition (Springer-Verlag):

Note: The assumed values should be confirmed by a basement surveyer – see [bedding areas](#).

Tensioning spring exclusion PLT: This option is enabled only if subgrade modulus > 0.

You can select here whether the system should compute a bedded plate with exclusion of tensioning springs (non-linear computation) or without exclusion of tensioning springs (linear computation). Please note in this connection that the non-linear computation is only available for individual load cases not for application-side superposition. Therefore, the superpositions to be calculated must be set by the user if the tension spring was excluded (see the chapter "Wall properties - [Tension spring exclusion](#)). The result of the non-linear computation can be displayed as the results of the individual load cases, the results of the individual superpositions or the crucial values of all superpositions.

Definition of a foundation slab

(Only available in the Building application)

The menu item "Foundation Slab" and the corresponding icon  offer a comfortable way to define a floor slab. Both options display the "basic parameters" dialog. The name "Foundation slab" is assigned by default and the floor height is automatically set to the plate thickness. Enter the desired values and confirm them with OK. The outline (exterior outline and cut-outs) of the current floor is automatically copied as foundation slab to the bottom position (without columns and walls etc.)

Note: If you use thickness regions on the foundation slab, be shure that the floor high is even greater than the maximum thickness region.

The three-dimensional view (by clicking on ) allows you to check the building visually. Of course, you can apply changes to the foundation slab subsequently (e. g. enlarge outline) using the "[layer](#)" function.

Floor selection

Only available in the GEO - Building + WL-Wind Loads applications

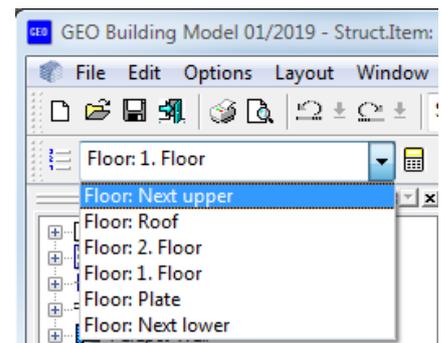
You can navigate through the floors of the building with the help of the "Floor selection".

See also [GEO](#) manual

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.

To generate an "empty" floor level (without copying the currently selected floor) select the menu item "[Floor levels](#)" in the main tree and select no when prompted whether to take over data. You can also delete floors via this dialog.



Basic building data

► Options ► Building Base Data

Standard for the load assumptions

See ill. on the right side.

EN 1991-1-4

You can find a description of the wind load parameters in accordance with Eurocode in the document "[WL Parameter EN.pdf](#)".

Terrain category:

Selection list for the effect of the terrain ruggedness

Standard case according to 10.3 (4):

- I: Open sea
 - North Sea islands
 - Plain land
- I/II: Baltic Sea islands
 - Coast and coastal areas up to 5 km inland
- II/III: Inland

A more refined assignment in accordance with Annex B without detailed examination of the change in ruggedness is possible when

- the building height < 50 m
- The distance to the next change in ruggedness > 3 km

Categories according to Annex B:

- I: Open sea
 - North Sea islands
 - Plain land
- II: Isolated buildings
 - Trees and hedges
- III: Forests
 - Outskirts
 - Industrial zones
- IV: Town with more than 15 % of buildings higher than 15 m

Hf User-defined increase factor for $h_{NN} > 1100$ m

Material selection (walls and columns)

You can access the material selection dialog via the input windows [Wall properties](#) and [Column parameters](#) or via the [Material list](#) by selecting "New material" in the material selection box.

Reinforced

concrete Selection of the concrete quality and the reinforcing steel.

Masonry Selection of the kind and type of masonry as well as of the strength class, the bulk density class and the mortar class. You can also freely enter the material parameters of the masonry by ticking the option "According to approval".

Steel You can select a steel type as material for columns in this section. The subsequently displayed window allows you to select the desired profile section.

Timber You can select a timber type as material for columns in this section.

Note:

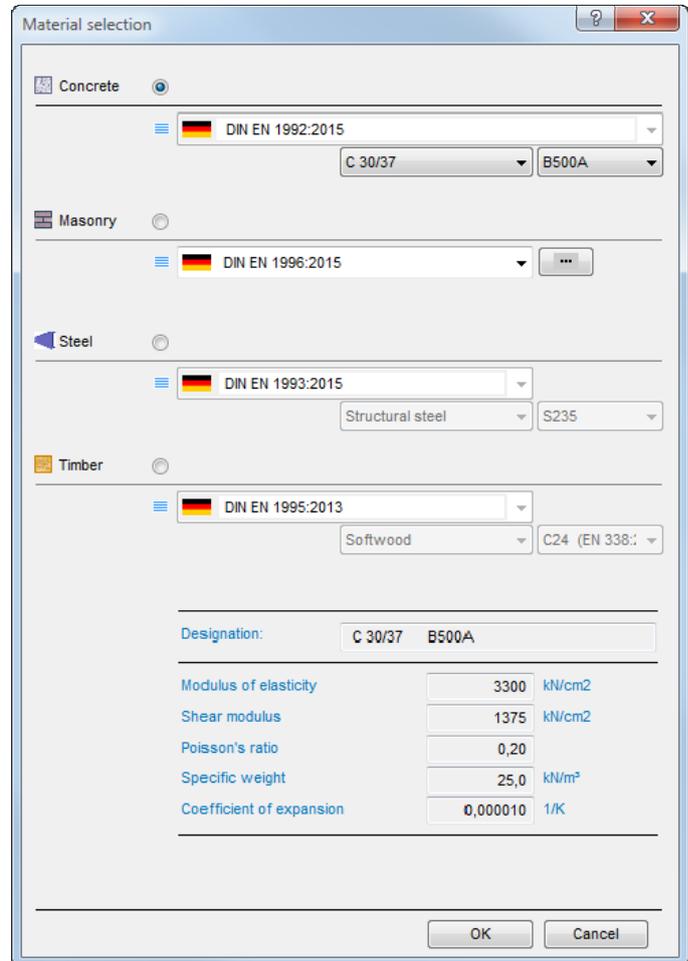
The [material list](#) gives a well-structured overview of the materials that you have entered and you can check and edit them also via this list.

Masonry

Type Selection list with the following items: brick, aerated cement block, lightweight concrete block, standard concrete block, breeze brick and Poroton block.

s The selection lists for the type, the strength class, the bulk density class and the mortar class are matched to the selected masonry type.

The values for σ_0 (Sig0) and the modulus of elasticity are displayed. If the option "According to approval" in the masonry type section is ticked, these values and the specific weight have to be entered (input fields are displayed then).



The dialog box 'Material selection' is shown with the following settings:

- Concrete:** Selected. Standard: DIN EN 1992:2015. Strength class: C 30/37. Rebar class: B500A.
- Masonry:** Not selected. Standard: DIN EN 1996:2015.
- Steel:** Not selected. Standard: DIN EN 1993:2015. Type: Structural steel. Profile: S235.
- Timber:** Not selected. Standard: DIN EN 1995:2013. Type: Softwood. Class: C24 (EN 338).

Designation: C 30/37 B500A

Modulus of elasticity	3300	kN/cm ²
Shear modulus	1375	kN/cm ²
Poisson's ratio	0,20	
Specific weight	25,0	kN/m ³
Coefficient of expansion	0,000010	1/K

Buttons: OK, Cancel

Material list

Access: Main tree ▶  Material list or via the [context menu](#)

The list shows all materials entered for this item (also those that are not used).

Select a material to display the corresponding details, edit them or delete the material.

Use

Displays where the material is used (foundation slab...).

Material data

Details such as the modulus of elasticity, the shear modulus, the Poisson coefficient, the specific weight and the thermal expansion coefficient.

New material

Accesses the [material selection](#) dialog where you can select a new material.

Delete

Materials that are not "in use" can be deleted.

- In use: All materials in use are marked with this symbol.

Import Materials

This button allows you to import the material list from another item. In the item selection window, select the appropriate item, click "Open" and apply.

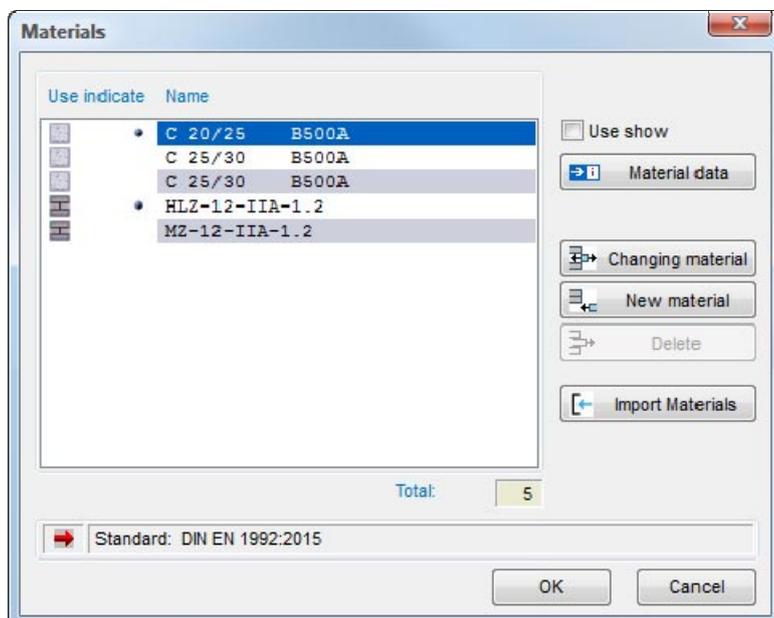


Plate contour

You can draw the outline of the plate/slab in this section. Various functions for the drawing of rectangles, polygon lines or circles are available for the input of the outline.

The icons shown on the right are displayed when you activate the contour input function (menu or Symbol ).

Input functions

The various input functions (polygon, rectangle, regular polygon, circle - definition via the radius...) are described in the chapter "Input options - [General graphical input functions](#).



Wall input

The wall-icons are displayed when you activate the wall input function (menu or Symbol ).



Wall on top of edge (outline)

To define a wall along an existing outline (edge of the plate or a opening), click on the wall input icon and subsequently on the corresponding outline. To finish the definition of the wall, click right (context-sensitive menu) and select "Exit". The [wall properties](#) window is displayed where you can enter further details such as the wall thickness etc.



Wall polygon

Input of a wall chain. Click left to define the start point of the line. Each additional mouse click defines a corner point of the wall chain. Finish the definition with a right click and select "Exit".

The [wall properties](#) input window is displayed – where you can enter details such as the position, the wall thickness, the material etc.

See also [Numeric input of coordinates](#).



Define a wall via two points

Enter the wall by defining the two limiting points.



Edit a wall

Click on the wall to be edited. The front and rear end of the wall become highlighted and you can drag them to another position with the mouse.



Edit the wall length

To lengthen or shorten a wall, click on the front or rear end of the wall. This edge of the wall becomes highlighted with a green cross, so that you can control the operation visually. Specify the value by which the wall should be lengthened or shortened (negative value) in the displayed dialog.



Line-arc (rise size)

This option allows you to convert a straight into a curved wall or change the radius of the curvature by specifying a rise size.

Click on the wall to be curved to display the dialog for the input of the rise size. The maximum positive/negative rise size is displayed. Specify a value within these limits in the rise size input field or click on "Positive maximum" or "Negative maximum" to set the maximum value.

You can see the positive/negative direction (positive = towards the wall i. e. from point 1 to point 2, on the left) in a sample graphic that is displayed.



Opening (cut a wall open)

Click on the front or rear edge of the wall and specify the distance from this edge to the cut-out. The corresponding edge of the wall becomes highlighted with a green cross to provide for visual control of the operation. Specify the distance of the cut-out to the front edge of the wall (L_a) and the width of the opening (L_o) in the displayed dialog.



Section a wall

This function allows you to section a wall into several wall segments. This might be useful if you do not want to distribute the load evenly over the total wall length but limit concentrated loads to individual wall sections.





Combine walls

This function allows you to combine several wall segments to a single wall, i. e. to undo the effects of the wall sectioning.



Move a wall

This function allows you to perform a parallel shift of a wall. Click on the wall to be moved and subsequently on the target position.



Copy a wall

Click on the wall to be copied, drag it to the target position and drop it there by clicking again.



Delete a wall segment

In order to delete an individual wall segment, click on it.



Wall properties

Click on a wall. The [wall properties](#) input window is displayed where you can view/edit details such as the position, the wall thickness, the material etc.



Separate wall group

This function allows you to separate a wall from an intersecting wall in order to assign different materials to each of these walls, for instance.



Combine wall group

This function allows you to combine two separated intersecting walls to a single wall group. To be able to do this, the walls to be combined must be of the same material.



Delete a wall group

Click on the desired wall group to delete it.



Wall group properties (edit)

A wall group is composed of several joined walls. Click on the wall group to edit it (the cursor is shown as a square). The [wall column properties](#) dialog is displayed.

Wall properties

This dialog is automatically displayed when you [enter a wall](#) or activate the properties function for a selected wall.

Wall location

You can place a wall to the left, to the right or centrally on top of the entered line. The options left and right refer to the entering direction of the line.

Important note:

Define exterior walls always in line with wall axis of the outline. Otherwise, the elements might be too small when the FE mesh is generated.

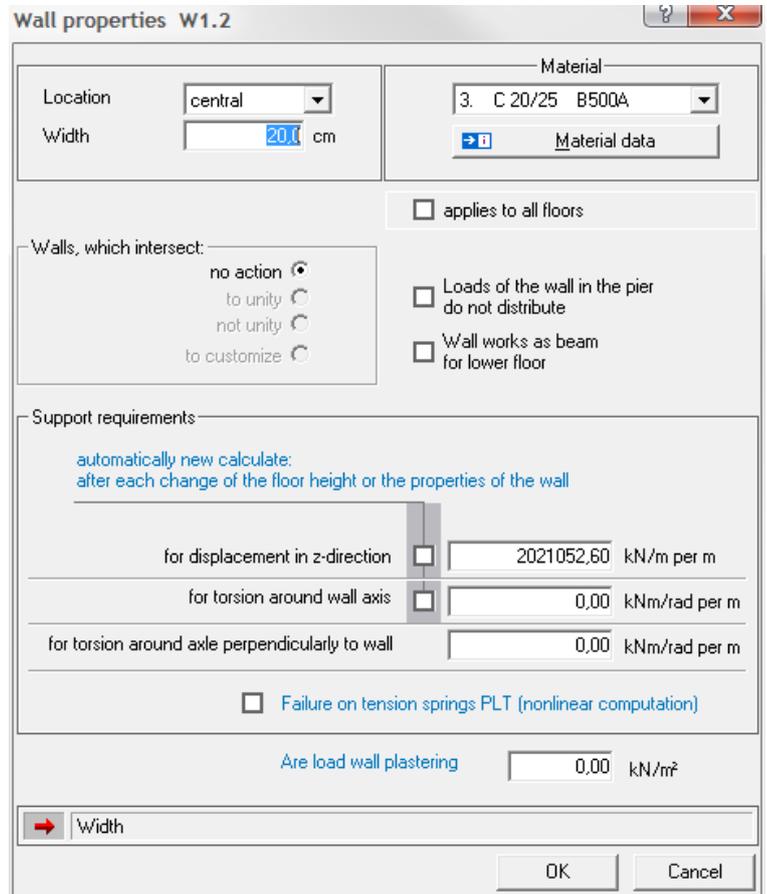
Wall thickness

Enter the wall thickness in cm.

Material

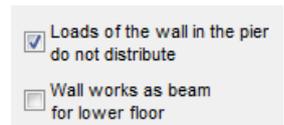
Select the material. When selecting "New material" the [material selection dialog](#) is displayed. See also [Material list](#).

You can display the material details such as the modulus of elasticity, the shear modulus, the Poisson coefficient, the specific weight and the thermal expansion coefficient via the material parameters button.



Options *These options are only enabled in the Building application!*

You can select whether the load on this wall should be distributed over the wall column and whether the wall of the floor underneath should act as an upstand beam.



e.g.: No distribution of loads of the wall over wall group

A wall group consists of single walls 1.1 and 1.2; if the option is tagged, the impact of a single load on wall 1.1 only concerns the wall itself. The load will not be distributed uniformly over both walls, as it will be with the option untagged.

e.g: Wall works as beam fo lower floor

This option causes, that a wall will be taken into account as well as a line load in the active floor and as an upper beam in the lower floor.

Applies to all floors:

This option is only enabled when more than one floor was defined. When defining multi-storey buildings, you can easily transfer (copy) the current wall to all existing floors via this option.

Intersecting walls:

You can optionally combine walls at intersections.

Walls, which intersect:

no action

to unity

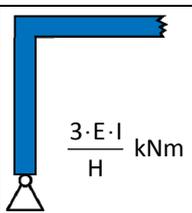
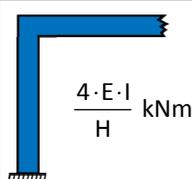
not unity

to customize

Bearing conditions

Enter -1 to provide for rigid bearing,
 0 to allow free torsion or displacement,
 >0 to provide for elastic bearing and/or torsion.

In order to consider the resilience of a bearing, you can define springs e.g.:

Translational spring:	$E \cdot A / H$ [kN/m]	per linear metre of bearing length.	Parameters
Torsional spring:		for articulated wall base,	E = modulus of elasticity of the wall I = moment of inertia of the wall A = surface of the wall H = wall height
		for restrained wall base per linear meter of the bearing length.	

Specify springs for walls always per linear metre of the wall length.

Recalculate automatically

Tick this option if the application should recalculate the bearing each time the floor height or the properties of this component are edited. See also [Global bearing conditions](#).

F5 key: Calculation of the spring rigidity.

When the cursor is positioned in the input field of the bearing conditions, the application calculates the corresponding spring rigidity when you press the **F5 key**. The calculation of the spring rigidity is based on the previously defined floor height, the specified wall thickness as well as the modulus of elasticity of the selected material.

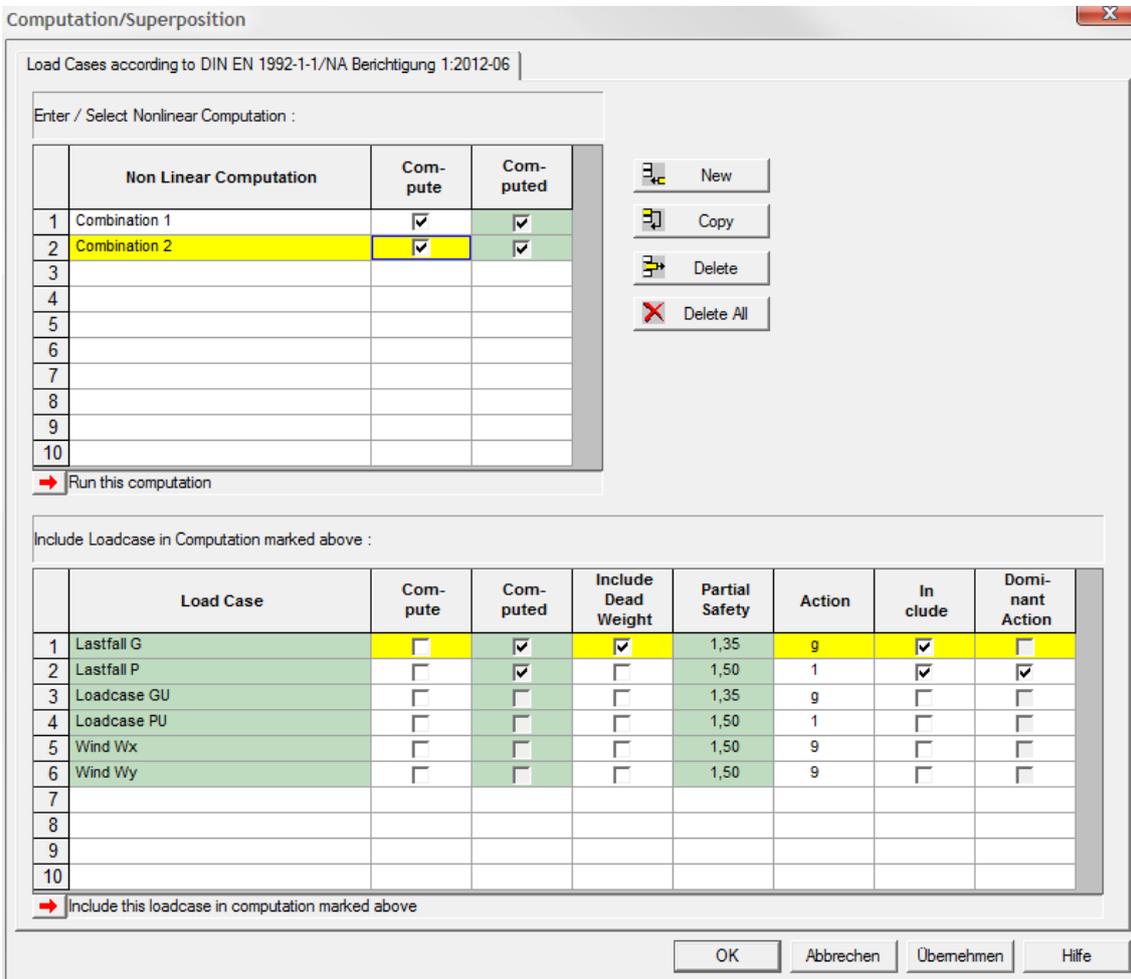
Tension spring exclusion

Optional exclusion of the tension springs for walls and columns in computations in the [PLT application](#).

You can define whether a bearing should be able to transmit tensile forces or not via the wall or column property "tension spring exclusion".

If the option "tension spring exclusion" is selected, a non-linear equilibrium iteration is performed, i. e. the calculation of the system is repeated as many times as necessary to achieve an equilibrium without tension springs. The results of the individual load cases cannot be superposed subsequently (the superposition principle cannot be applied subsequently) because, unlike to the linear computation, the equilibrium is calculated on the deformed system. In contrast to the linear computation, the application cannot assess the most unfavourable non-permanent impact (main impact) for each case during superposition when the calculation is based on design values according to DIN 1055-100 / EN 1990. Therefore, this impact must be given by the user.

PLT allows the definition of several load cases in the calculation with excluded tension spring. A non-linear calculation is performed for these load cases. Clicking to "Run Computation/Superposition" in PLT displays a corresponding dialog if tension spring exclusion was defined.



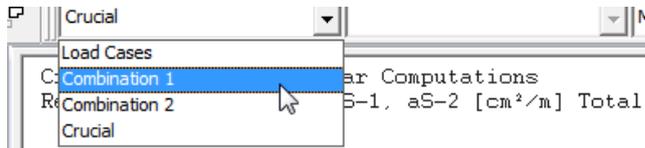
This dialog allows you to define several load combinations. First, enter first a name for the combination in the upper table (e. g. combination 1). After this, select the load case that should be assigned to combination 1 in the column "Add". Finally, select the leading action for this combination and activate the "Calculate" option in the upper table (non-linear calculation). The "New" button allows you to define additional load combinations.

Tip: You must define several combinations to calculate the same load combination with different leading actions. Just activate the "Copy" button and change the leading action.

If the results of the individual load cases should be available after the calculation too, check the corresponding checkboxes in the "Calculate" column in the lower table.

After the calculation, the following results are available:

- Load cases
- Combinations
- Crucial



The option "Crucial" displays the decisive value for the respective design point resulting from the calculated combination. The option "Combinations" allows the user to display the results of individual combinations. The desired combinations are selected from the displayed list. In addition, the user can display results of individual load cases. If there is no result available, a corresponding note is displayed, indicating that the calculation has not been performed yet.

Distributed load wall plaster *(only available in the Building application)*

The application calculates the weight of the plaster (symmetrically) for both sides of the wall from the specified distributed load [kN/m²] and adds the result to the selfweight of the wall.

Input of columns

The icons shown on the right are displayed when you activate the column input function  =>



Input of an individual column

Click on the desired position of the column. The [column parameters](#) dialog is displayed.



Input of several columns

This option allows you to enter several similar columns one after the other. Click successively on the desired column positions. Finish the operation via "Exit" (in the [context-sensitive menu](#)). The [column parameters](#) dialog is displayed.



Number columns

This function allows you to renumber the columns you have entered. Select the columns that should be renumbered (zoom out a selection area with the mouse or select the columns per mouse click). The current column numbers are displayed. Click right and select "Exit" in the context-sensitive menu. Specify a new start number and the desired increment in the displayed dialog and confirm your settings with OK. To check the new numbers, select the columns once more (as described above), check the displayed numbers and terminate the operation by clicking right and selecting "Cancel".

Note: In order to renumber all columns zoom out the selection area around the whole system.

You can select several columns also by clicking successively on them while pressing the control key (ctrl).



Move a column

Click on the column to be moved and subsequently on the target position or enter the target coordinates [numerically](#).

Finish the operation via "Exit" (in the [context-sensitive menu](#)).



Copy a column

Click on the column to be copied and subsequently on the target position of the copied column or enter the target coordinates [numerically](#).

Finish the operation via "Exit" (in the [context-sensitive menu](#)).



Delete a column/several columns

Select this option and click subsequently on the columns to be deleted. You can restore unintentionally deleted columns with the undo function.

Note: In order to delete several columns you can also use the functions ["Crossing"](#) or ["Window"](#) or the F4 key.



Edit a column

Click on the column to be edited and enter your changes in the [column parameters](#) dialog.



Enter columns via the auxiliary grid

To do this, an [auxiliary grid](#) must have been generated before. Columns are placed on all intersecting points of the auxiliary grid. Enter the column details in the displayed [dialog](#).





Design a column *(only available in the Building application)*

Click on the column to be designed in the Building application to launch the column design application B5. The geometrical and load data are transferred automatically to the design application. You can still apply changes in the column design application. Geometrical changes can be retransferred to the Building application.

Attention: If current load calculation results are not available for the building, the load calculation starts automatically when you click on the column.



Punching shear analysis

Click on the column to be designed to launch the punching shear application B6. The geometrical and load data are transferred automatically to the design application. The type of column (interior, edge or corner column) is assessed and shown in an intermediate dialog. You can still edit and change the column type in this dialog and change or add column details (e.g. small openings) in the punching shear application.

Attention: If current load calculation results are not available for the building, the load calculation starts automatically when you click on the column.

Column parameters

After the [input of a column](#), the dialog for the input of the column parameters is displayed.

Column form

You can select whether the column should have a rectangular or a circular cross section and whether it should be solid (default) or hollow (tick the option). The displayed input fields depend on the selected column type.

The input fields for the inner dimensions ("bi", "di") of the hollow column are only enabled if the option "Hollow column" was ticked.

The following applies to rectangular columns:

b = dimension in direction of the x-axis

d = dimension in direction of the y-axis

Torsion/Rotation angle:

You can enter rectangular columns by defining and angle to the x-axis. The specified value refers to the edges that run in direction of the x-axis.

Material

Select the corresponding material.

When selecting "New material" the [material selection dialog](#) is displayed.

Participation in the horizontal load transfer:

If this option is ticked, horizontal loads are considered for the load transfer.

Applies to all floors:

If this option is ticked, the entered details apply to all floors.

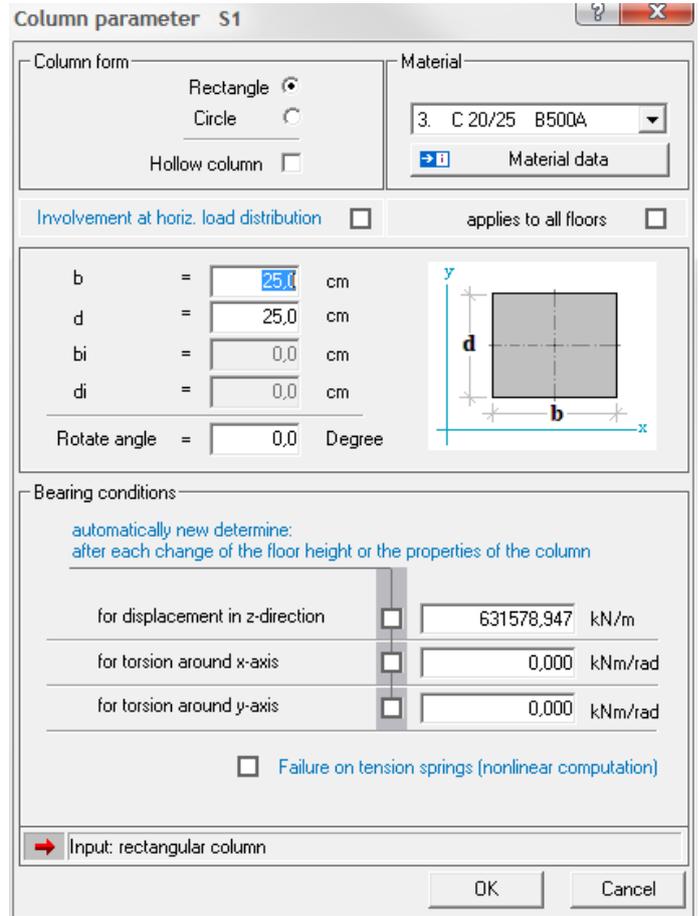
Bearing conditions

Shifting = enter the translational rigidity

Torsion = rotational rigidity

Enter

- 1 to provide for rigid bearing,
- 0 to allow free torsion or displacement,
- >0 to provide for elastic bearing and/or torsion.



In order to consider the resilience of a bearing, you can define springs:

e. g.: Translational spring: $E \cdot A/H$ [kN/m]

Torsional spring: $3 \cdot E \cdot I/H$ [kNm] for an articulated

$4 \cdot E \cdot I/H$ [kNm] for a restrained base of the column

E = modulus of elasticity of the column I = moment of inertia of the column

A = surface of the column H = column height

F5 key

When the cursor is positioned in the input field of the bearing conditions, the application calculates the corresponding spring rigidity when you press the F5 key. The calculation of the spring rigidity is based on the previously defined floor height, the specified column dimensions as well as the modulus of elasticity of the selected material.

You can define the bearing conditions also under [global bearing conditions](#) .

Tension spring exclusion

Optional exclusion of the tension springs for walls and columns in computations in the PLT application. → See description in the chapter [wall properties](#).

Recalculate automatically

Tick this option if the application should recalculate the bearing each time the floor height or the properties of this component are edited.

If you define bearing conditions under [global bearing conditions](#) the related options of the single components will be ticked automatically.

Bearing (SCN)

The icons shown on the right are displayed when you select the input of bearings in the [SCN application](#). =>

Note: You can use the suitable [input modes](#) or the [numeric input of coordinates](#) to define the bearing precisely.

Input of an individual point bearing



Click on the desired position of the bearing. The "boundary conditions" dialog is displayed.

Input of several point bearings



This option allows you to enter several similar bearings one after the other. Click successively on the desired positions of the bearings.
Finish the operation via "Exit" (in the [context-sensitive menu](#)). The "boundary conditions" dialog is displayed.

Line bearing



Click on the front end position of the bearing and then on the rear end position. The "boundary conditions" dialog is displayed.



Edits the start and end points of a line support



Moves a line support



Copies a line support

Delete a bearing



Click on the desired bearing to delete it. You can restore unintentionally deleted bearings with the undo function.

Edit a bearing



Click on the desired bearing to display the boundary conditions dialog.



Upstand/downstand beam

The icons shown on the right are displayed when you select the input of

an upstand or downstand beam (icon ). =>

Note: You can use the suitable [input modes](#) or the [numeric input of coordinates](#) to define the beam accurately.



Input of a downstand beam on top of an edge or an auxiliary line

To define a downstand beam along an existing outline (edge of the plate or a opening) or auxiliary line click on the icon to define a downstand beam and subsequently on the corresponding outline(s). To complete the selection click right ([context-sensitive menu](#)) and select "Exit". The [upstand/downstand beam parameters](#) window is displayed and you can enter the beam dimensions.



Input of an upstand/downstand beam as polygonal chain

A left click positions the front end of the upstand/downstand beam, each subsequent click defines an additional corner point. After having defined the last point, perform a right click and select "Exit" in the displayed context-sensitive menu.

The [upstand/downstand beam parameters](#) dialog is displayed.



Define a downstand beam via two points

Perform a left click to define the front end of the upstand/downstand beam and a second left click to define the rear end.

The [upstand/downstand beam parameters](#) dialog is displayed.



Edit an axis

Click on the downstand beam to be edited. The front and rear end of the downstand beam become highlighted and you can drag them to another position with the mouse. Finish this operation with a [right click](#).



Edit the position

To lengthen or shorten a downstand beam, click on the front or rear end of the beam. This edge of the beam becomes highlighted with a green cross providing for visual control of the operation. Specify the value by which the beam should be lengthened or shortened (negative value) in the displayed dialog.



Rise size

This option allows you to convert a straight downstand beam into a curved one or change the curvature of a beam by specifying a rise size.

Click on the downstand beam to be curved to display the dialog for the input of the rise size. The maximum positive/negative rise size is displayed. Specify a value within these limits in the rise size input field or click on "Positive maximum" or "Negative maximum" to set the maximum value.

You can see the positive/negative direction (positive = towards the beam i. e. from point 1 to point 2, on the left) in a sample graphic that is displayed.



Number downstand beams

Click successively on all downstand beams to be renumbered. Finish the selection process by clicking right and selecting "Exit". Specify subsequently a start number and the increment for the renumbering process.

**Move a downstand beam**

This function allows you to perform a parallel shift of a downstand beam. First click on the downstand beam to be moved and then on the target position of the beam.

**Copy a downstand beam**

This function allows you to copy a downstand beam. First click to the downstand beam to be copied and then on the target position of the copied beam.

**Delete an upstand/downstand beam**

Click to the beam to delete it. You can restore unintentionally deleted beams with the undo function.

**Edit an upstand/downstand beam**

Click on the desired beam to display the [upstand/downstand beam parameters](#) dialog.

**Input on top of horizontal lines of the auxiliary grid**

To do this, an [auxiliary grid](#) must have been generated before.

A downstand beam is positioned on each horizontal line of the auxiliary grid. Enter the downstand beam dimensions in the displayed [dialog](#).

**Input on top of vertical lines of the auxiliary grid**

To do this, an [auxiliary grid](#) must have been generated before.

A downstand beam is positioned on each vertical line of the auxiliary grid. Enter the downstand beam dimensions in the displayed [dialog](#).

**Input on top of vertical and horizontal lines of the auxiliary grid**

To do this, an [auxiliary grid](#) must have been generated before.

A downstand beam is positioned on each vertical and horizontal line of the auxiliary grid. Enter the downstand beam dimensions in the displayed [dialog](#).

**Calculation as a continuous beam** *(only available in the Building application)*

First click on the downstand beam to be designed. When doing so, you can combine several downstand beam sections in one line to a continuous beams. To complete the selection click right ([context-sensitive menu](#)) and select "Exit". The geometrical and load data are automatically transferred to the design application. You can still apply changes in the continuous beam application DLT10.

Attention: If current load calculation results are not available for the building, the load calculation starts automatically when you finish the selection.

Upstand/downstand beam parameters

Enter the parameters of the upstand or downstand beam in the dialog that is displayed after the input of a [upstand / downstand beam](#).

- Type** Select upstand or downstand beam.
- Material** This option displays the material defined in the basic parameters of the plate, because the definition of an upstand or downstand beam currently requires that the plate and the beams are of the same material.
- b0, d0, bm, dp** The meaning of the dimension codes is revealed in the illustration below.
- It** You can reduce the torsional rigidity via this factor (> 0).
For the calculation of the beam's torsional rigidity, the rib portion is assumed over the total height.
Note: You can calculate with 0.3 (torsion) for standard downstand beams in building construction.
For the calculation of the flexural rigidity, the total cross section without reduction is considered.
- Ib** Factor for the increase of the flexural rigidity.

T-beams are considered by adding the rigidity terms along the beam axis. Since the plate elements do not include normal forces, the gravity axis of the beam elements is assumed to lie in the plate plane.

Applies to all floors:

This option is only enabled if more than one floor was generated.

When defining multi-storey buildings, you can easily transfer (copy) the current upstand/downstand beam to all currently existing floors.

Parapet

Access via ▶ Parapet in the main tree or the icon .

The following functions are subsequently available via the displayed tool bar:



Parapet on top of edge

Click on the plate edges to which a parapet should be attached and enter the required details in the displayed dialog.



Enter a parapet as polygonal chain

A left click positions the front end of the parapet, each subsequent mouse click defines an additional corner point. After having defined the last point, perform a right click and select "Exit" in the displayed [context-sensitive menu](#).

The dialog to edit the parapet is displayed.



Define a parapet via two points

Perform a left click to define the front end of the parapet and a second left click to define the rear end.

The dialog to edit the parapet is displayed.



Edit the axis of the parapet

Click on the parapet to be edited. The front and rear end of the parapet become highlighted and you can drag them to another position using the mouse. Define the new coordinates per mouse click. Finish this operation with a [right click](#).



Edit the position

To lengthen or shorten a parapet, click on its front or rear edge. The corresponding edge point is shown as a green cross providing for visual control of the operation. Specify the value by which the parapet should be lengthened or shortened (negative value) in the displayed dialog.



Rise size

This option allows you to convert a straight parapet into a curved one or change the radius of the curvature of a parapet by specifying a rise size.

Click on the parapet to be curved to display the dialog to enter the rise size. The maximum positive/negative rise size is displayed. Specify a value within these limits in the rise size input field or click on "Positive maximum" or "Negative maximum" to set the maximum value.

You can see the positive/negative direction (positive = from point 1 to point 2) in a sample graphic that is displayed.



Number parapets

Click successively on all parapets to be numbered. Finish the selection process by clicking right and selecting "Exit". Specify subsequently a start number and the increment for the numbering process.



Move a parapet

This function allows you to perform a parallel shift of a parapet. Click on the parapet to be moved and subsequently on the target position of the parapet.



Copy a parapet

This function allows you to copy a parapet. Click on the parapet to be copied and subsequently on the target position of the copied parapet.



Delete

Click on a parapet to delete it. You can restore unintentionally deleted parapets with the undo function.



Properties

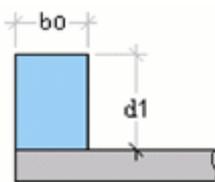
Click on the desired parapet to display the parapet parameters dialog.



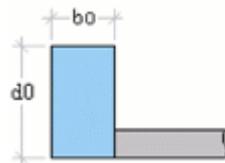
Interface to the Frilo application DLT.

Type of parapet

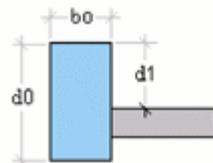
Top-attached



Front-attached flush to the lower edge



Front-attached with top and bottom projection



Bearing / non-bearing

The parapet is integrated as a bearing or a non-bearing element. Only top-attached parapets could be defined as bearing ones.

d0 / d1 / b0

Dimensions of the parapet → see drawing above or in the dialog.

It

You can reduce the torsional rigidity via this factor (> 0).

Ib

Factor for the increase of the flexural rigidity.

Material

To select a material, click on the corresponding entry in the selection list.

Material parameters: The material parameters are displayed in a separate window. See also the [material list](#).

Opening

You can draw openings into the plate in this section. Various functions for the drawing of rectangles, polygon lines or circles are available for the input of the outline. You can generate any outline by combining these elements via [Boolean operations](#).

The corresponding icons are displayed when you select the icon  to enter openings.

Input functions

The individual general input functions (polygon, rectangle, circle - definition via the radius...) are described in the chapter Input options - [General graphical input functions](#).



Areas - Regions

Thickness / bedding area

After having defined an area, enter the subgrade modulus in the displayed dialog.

The toolbar shown on the right is displayed when you select the input of thickness areas (icon ) or bedding areas (icon ) . =>

Input functions

The individual general input functions (polygon, rectangle, circle - definition via the radius...) are described in the chapter Input options - [General graphical input functions](#).



On contour

Define the area over an existing closed contour - analogous as described under [distributed load](#).



Delete an area

Select this icon and click on an area to delete it. You can restore unintentionally deleted areas with the undo function.



Edit properties

In order to change the thickness or the subgrade modulus for a defined area, click first on this icon and then on the desired area. The corresponding dialog is displayed subsequently.



Bedding areas (elastic bedding)

Areas with different (elastic) bedding can be defined via the function "Bedding Regions". A bedding of the entire plate can be defined via the menu item "[Basic parameters](#)" (only available in the Building and PLT applications). An elastic bedding should always be defined via one of these two input options.

In case of multiple definitions or overlapping, the most recent input data are used.

You need not define additional bearings for foundation slabs. We recommend to use upstand beams to provide for the wall rigidity of rising reinforced walls.

Reference values for the subgrade modulus according to Lang ·Huder, Bodenmechanik und Grundbau, fourth edition (Springer-Verlag):

Soil type	Subgrade modulus k_s in kN/m^3 approx.
Peat, humus	5000 - 20000
Clay, soft	20000 - 40000
Clay, plastic	30000 - 60000
Clay, harsh	50000 - 90000
Clay, very harsh	100000 - 120000
Clay, sandy	80000 - 100000
Sand, loose	10000 - 30000
Sand, compact	80000 - 100000
Gravel, fine with sand	100000 - 120000
Gravel, medium with sand	120000 - 150000
Gravel, coarse with sand	180000 - 240000
Gravel, compacted	200000 - 300000

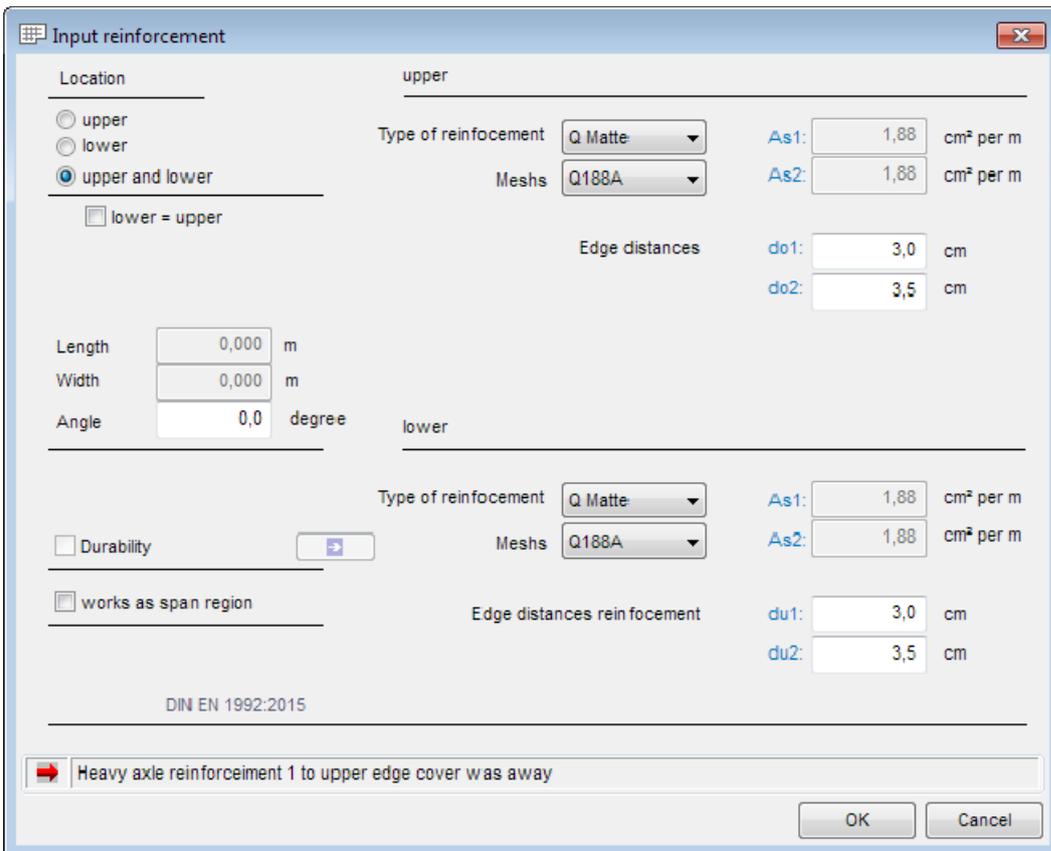
Note: The subgrade reaction moduli in traffic route engineering are determined by plate load tests using a plate with a diameter of 762 mm and loads between 0.07 and 3 NM/m^2 . These moduli are not suitable for the design of foundation slabs in most cases. Compared to values determined by soil expertise, the listed values are approximately 10 times too high. Therefore, the assumed values should be confirmed by a soil expertise.

Reinforcement

You can define areas with a selected basic reinforcement with the help of reinforcement areas. The basic reinforcement is neither considered in the strength computation nor for the design. It is only relevant for the representation of the results and allows you to show the difference between the basic and the required reinforcement.

A change of the basic reinforcement does therefore not require a new calculation or design, merely the new difference is displayed.

After the input of a reinforcement area, the dialog "Default reinforcement details" is displayed. You can enter the reinforcement location (top/bottom), the angle, the distance to the edges and the type of reinforcement (mesh reinforcement...).



You can define a reinforcement area for the basic reinforcement of the lower or the upper reinforcement or for both.

If you tick the option "lower = upper", the parameters for the upper reinforcement are transferred to the lower reinforcement.

When ticking the option "Acts as bearing direction area", the entered reinforcement area is simultaneously defined as bearing direction area, see also the chapter "[Bearing direction area](#)".

Recommendation: You should define supporting direction areas always in combination with reinforcement areas, because this allows you to adjust the main direction of the reinforcement to the supporting direction.

The toolbar shown on the right is displayed when you select the input of reinforcement areas (icon



Input functions

The individual general input functions (polygon, rectangle, circle - definition via the radius...) are described in the chapter "Input options - [General graphical input functions](#).



On contour

Define the area over an existing closed contour - analogous as described under [distributed load](#).



Wall reinforcement area

Click on the wall or a wall area to access the input dialog. After having confirmed your settings with OK, the defined reinforcement area is shown graphically. You can subsequently click on additional wall areas.



Column reinforcement area

Click on a column to access the input dialog. After having confirmed your settings with OK, the defined reinforcement area is shown graphically. You can subsequently click on additional columns.



Delete a reinforcement area

Select this icon and click on an area to delete it, the cursor is shown as a square. You can restore unintentionally deleted areas with the undo function.



Properties

In order to change the parameters for a defined area, click first on this icon and then on the desired area or wall or column. The corresponding input dialog is displayed subsequently.



Bearing direction area

You can access the input functions for the bearing direction area via the icon  or, alternatively, via the context-sensitive menu or the main tree.

Define an area (see also [Input options](#)). A dialog for the input of the angle [in °] to the x-axis (positive = anticlockwise) is displayed subsequently.

Via bearing direction areas, you can define ceiling areas with uniaxial load transfer (e. g. when using semi-finished components). An angle of 0° means a load transfer in direction of the x-axis, an angle of 90° means a load transfer in direction of the y-axis, angles in-between mean a corresponding rotation of the bearing direction.

Input functions

The individual general input functions (polygon, rectangle, circle - definition via the radius...) are described in the chapter "Input options - [General graphical input functions](#).



Properties

Click on the outline of the bearing direction area to display its properties (angle in degrees) and edit them if necessary.

Plate joint

You can define pinned joints e.g. for height offsets in the ceiling, expansion joints etc.

Click on the menu item "Plate joint" ().

The corresponding toolbar is displayed in addition to the menu options.



Two points

You can define the front end and rear end of the pinned joint. Select subsequently the type of joint in the displayed dialog.

Shear force joint:

This type of joint allows vertical displacement. Therefore, no shear force is transmitted.

It does not allow torsion, however.

Moment joint:

This type of joint allows torsion and, consequently, moments are not transmitted.

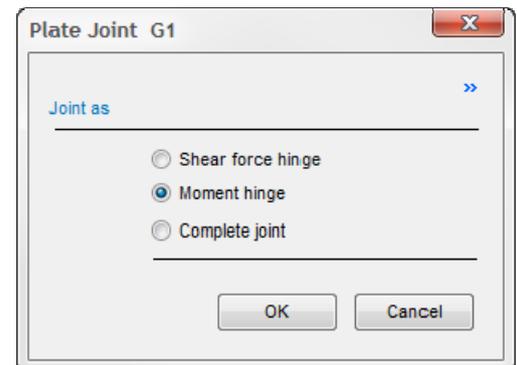
Example: height offset in the ceiling without rigid connection or use of Schöck ISO-Korb® reinforcing cages (Q-type).

Force and moment joint:

This type of joint allows torsion as well as vertical displacement, i.e. neither moments nor shear force are transmitted.

Example: expansion joints

Important note: you cannot define a joint on a wall - a wall prevents vertical displacement. You have to define a moment joint.



The functions of the items "Edit" to "Properties" are similar to those described in the [Line load](#) menu, for instance.

Global bearing conditions

The menu item "Global bearing conditions" allows you to define whether the bearing conditions should apply to the current floor or the entire building (only building model). The input is done separately for walls and columns. You can select whether the spring rigidities should be calculated on the basis of the specified geometrical and material parameters or the vertical structural components should be considered as rigid bearings. If you select the option "No action", the settings in the [Wall properties](#) section apply.

The option "Wall base restrained" has an effect on the calculation of the torsional springs (You can find more information in the chapter [Wall properties](#)).

Global bearing conditions

Setting the support conditions

for current floor: 1. Floor

for all floors

Columns:

Compute the spring stiffness

rigid support

no action

automatic adjustment of stiffness on change of floor height and the properties of the column

for displacement in z-direktion

for torsion around x-axis

for torsion around y-axis

Column base clamped

Walls:

Compute the spring stiffness

rigid support

no action

automatic adjustment of stiffness on change of floor height and the properties of the wall

for displacement in z-direktion

for torsion around wall axis

for torsion around axle perpendicularly to wal

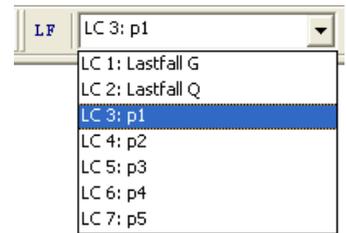
Wall foot clamped

the setting is not valid for the walls with the vertical stiffnes = 0 (basic walls)

Loads

Before entering a load, make sure that you have selected the desired load case because the newly entered load is always assigned to the currently selected load case.

If you have not yet defined a load case, the application displays automatically the load case table.



You can select a load case in the corresponding selection list (click on ). The results of the selected load case are displayed in text and graphic mode.

You can access the individual input options via the main tree or the input menu item as well as via the load case toolbar.

Input and editing of [concentrated loads](#).

Input and editing of [line loads](#).

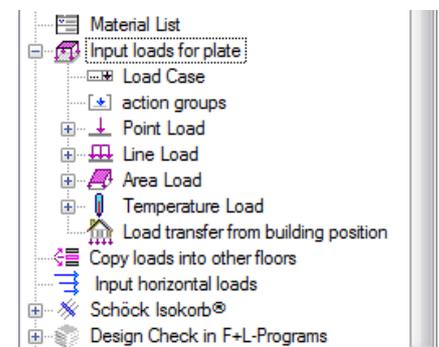
Input and editing of [distributed loads](#).

Input and editing of [temperature loads](#).

Load transfer from [building item](#) (only available in the Building application).

[Copy loads/loadcases](#) to other floors (Building only)

Input and editing of [horizontal loads](#) (Building only).



Load case

You must define load cases in this section before entering a load.

When entering a load, you should always check the currently active load case because the entered load is always assigned to this load case.

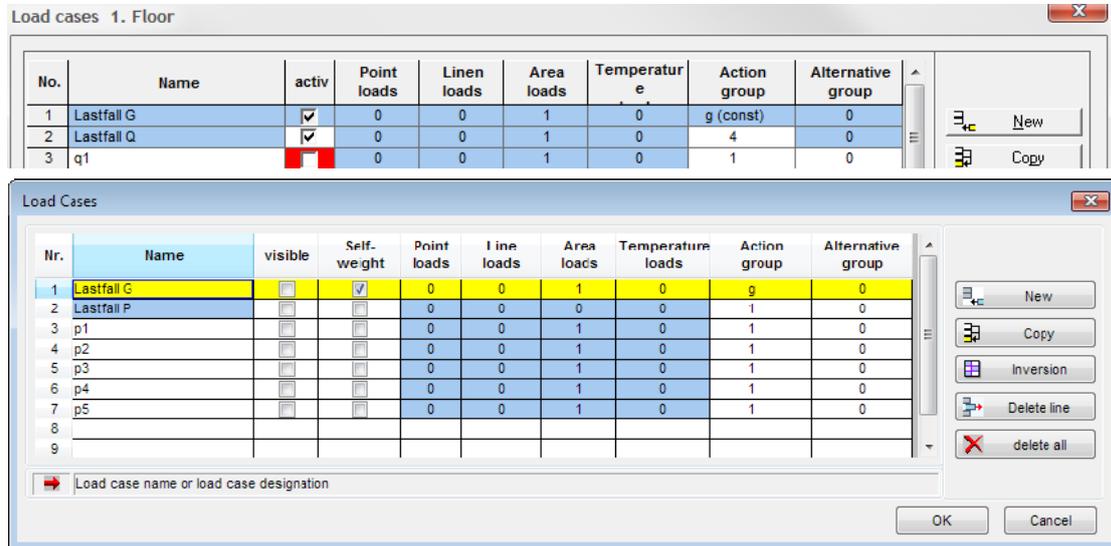


Illustration: Load case dialog in the Building and the PLT application

Options (application-specific)

- Active** The load case is only considered in the calculation if this option is ticked otherwise it is not.
- Visible** If this option is ticked, this load case is always shown on the graphic screen, otherwise only the active load case is shown.
- Selfweight** If this option is ticked, the selfweight is considered in the calculation of the load case (plate and upstand/downstand beams). In the building application, the selfweight loads are always assessed by the application.
- Action group** Select the desired action group in this section. The menu item "Action groups" (main tree) allows you to freely define new action groups (Icon "New") in a separate dialog and also display their application in the load cases (icon "Application").
- Alternative group** You can combine load cases that exclude each other into groups i.e. these load cases cannot act simultaneously. All load cases marked with a zero (default) can act in combination with all other load cases. All load cases marked with the same positive number exclude each other. If the results interfere, the most unfavourable load case of those excluding each other is taken into consideration.
- New** A new line is added for the input of a new load case.
- Copy** This function allows you to copy one or several load cases into a new load case. You can specify an additional factor that is multiplied with the load cases to be copied.
- Inversion** Generation of inverted load surfaces.
This function allows you to generate a distributed load acting upon the surface that remained after having defined other load surfaces. Click on this button to display the dialog "Reference load case selection" and select one or several load cases (with distributed loads). Confirm your selection with OK and enter in the subsequently displayed dialog a distributed load for the remaining surface. The new distributed load is assigned to the currently active load case.

Delete line The currently selected line is deleted.

Delete all All load cases are deleted.

The columns Concentrated loads, Line loads... show the number of loads of this type that have already been entered for information.

Concentrated load

The load that you enter is assigned to the currently active (selected) load case →, see also the chapter [Loads](#).



Input of a concentrated load

Click on the desired location of the new load, the cursor is shown in form of a cross providing for accurate placement. The concentrated load dialog is displayed subsequently.



Input of several concentrated loads with the same load values

Click successively on the desired locations of the loads, the cursor is shown as a cross providing for accurate placement.

Finish the operation with ▶ a right click and select ▶ "Exit" ([context-sensitive menu](#)).

The concentrated load dialog is displayed subsequently and you can enter the load values.



Move a concentrated load

To move a concentrated load click on it, drag it to the desired position with the mouse and click again to drop it there and finish the moving operation.



Copy a concentrated load

To copy a concentrated load, click on it, drag it to the desired position with the mouse and click again to drop it there and finish the copying operation. You can edit the load values via "Properties" (see below).



Delete a concentrated load

Select this option and click subsequently on the concentrated load to be deleted. You can restore unintentionally deleted loads with the undo function.



Edit a concentrated load

Click on the concentrated load to be edited. The cursor is shown as a square. The concentrated load dialog is displayed subsequently.



Enter concentrated loads via the auxiliary grid

This option allows you to place an individual load on each intersecting point of the auxiliary grid. You can define the load values in the subsequently displayed dialog.

Input dialog for concentrated loads

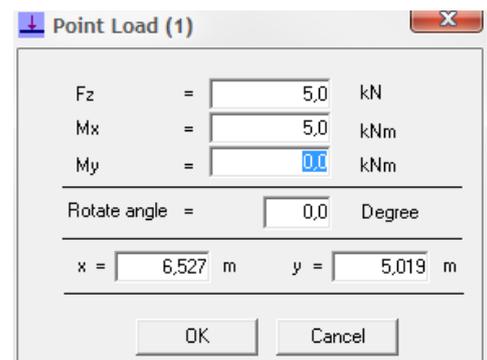
Fz Value for the concentrated load

Mx Torque around the x-axis

My Torque around the y-axis

Angle of rotation Angle for Mx/My

x= / y= You can enter the coordinates also numerically via the x and y input fields or edit them via these fields.



Fz	=	<input type="text" value="5,0"/>	kN
Mx	=	<input type="text" value="5,0"/>	kNm
My	=	<input type="text" value="0,0"/>	kNm
Rotate angle	=	<input type="text" value="0,0"/>	Degree
x =		<input type="text" value="6,527"/>	m
y =		<input type="text" value="5,019"/>	m

OK Cancel

Line load

The load that you enter is assigned to the currently active (selected) load case →, see also the chapter [Loads](#).



Load on outline edge (line or arc segment)

Define a line load on an outline edge (plate outline or opening) by clicking on the desired line or arc segment. The line load dialog is displayed subsequently and you can enter the values.



Enter a polygonal chain

After completion of the input process (right click), the line load dialog is displayed.



Enter a line load

Place the cursor shown as a cross on the start point of the line load and click left, move the cursor to the end point of the line load and click again. The line load dialog is displayed subsequently.



Edit a line load

The function allows you to edit the end point coordinates of a line. Click on the line to be edited and drag the end point(s) to the desired position with the help of the highlighted drag points. Finish the operation with ▶ a right click and select ▶ "Exit" ([context-sensitive menu](#)). You can edit the load values via "Properties" (see below).



Transform a straight line load into a curved one

After clicking on this icon select the line load to be transformed. Drag the highlighted drag points to the new target positions until the curve has the desired shape. Click again to fix the new shape. Finish the operation with ▶ a right click and select ▶ "Exit" ([context-sensitive menu](#)).



Move a line load

To move a line load click on it, drag it to the desired position with the mouse and click again to drop it there and finish the copying operation.



Copy a line load

To copy a line load, click on it, drag the copy to the desired position with the mouse and Click again to drop it there and finish the copying operation. You can edit the load values via "Properties" (see below).



Delete

Delete a line load by clicking on it. You can restore unintentionally deleted loads with the undo function.



Properties

This function allows you to edit the load values or coordinates of a line load. Click on the line load to be edited. The cursor is shown as a square. The line load dialog is displayed subsequently.

Line load input dialog

"Constant load" option: You must disable this option to be able to define different load values for $q1/q2$, $m1/m2$.

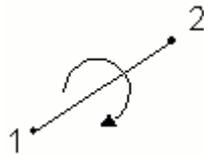
qz1 Load value for the start point of the line

qz2 Load value for the end point of the line

m1 Torque at the start point of the line

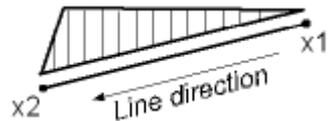
m2 Torque at the end point of the line

The orientation of the sectional torques is assigned in accordance with the corkscrew rule (right hand), i. e. the orientation of a positive torque runs in clockwise around the x-axis from point 1 to point 2.



x1/2, y1/2 You can edit the coordinates of a line load also in numeric mode.

Attention: Lines that have been entered from the right to the left, have their start point on the right ($x1$).



Distributed load

The load that you enter is assigned to the currently active (selected) load case →, see also the chapter [Loads](#).

The input of rectangular or polygonal surfaces is done as described in the chapter [Plate contour](#).



Polygonal distributed load q_z

This function allows you to enter a polygonal surface. Position the individual polygon points per mouse click on the graphic screen. After having entered the final polygon point, click with the **right mouse button** and select in the [context-sensitive menu](#) "Exit" to complete the polygonal chain.

Select in the subsequently displayed window whether the distributed load is a constant (default) or variable load. Enter the load value q_z directly for constant distributed loads.

If you have selected a variable distributed load, you are prompted to enter three reference points for the load ordinates (normally, three corner points of the outline). Enter the load ordinates q_{z1} , q_{z2} , and q_{z3} for these points in the subsequently displayed dialog. You can edit reference points that you have positioned per mouse click also numerically via the input fields $x_{1,2,3}$ and $y_{1,2,3}$.



Lay a distributed load q_z over an existing surface

In order to lay a distributed load over an existing surface (e. g. plate outline or an area), click on this icon and subsequently on the respective surface outline and specify the load value in the displayed dialog.



Rectangular distributed load q_z

This function allows you to enter a rectangular distributed load. Click on the first corner point, zoom out a rectangle with the mouse and click on the second (diagonally opposite) corner point.

Select in the subsequently displayed window whether the distributed load is a constant (default) or variable load. Enter the load value q_z directly for constant distributed loads.

If you have selected a variable distributed load, you are prompted to enter three reference points for the load ordinates (normally, three corner points of the outline). Enter the load ordinates q_{z1} , q_{z2} , and q_{z3} for these points in the subsequently displayed dialog. You can edit reference points that you have positioned per mouse click also numerically via the input fields $x_{1,2,3}$ and $y_{1,2,3}$.



Edit the outline of a distributed load

Click on the outline to highlight the corners with small squared drag points. You can drag these points to new target positions with the mouse. Finish the operation with a right click and select "Exit" from the context-sensitive menu.



Shift the outlines of a distributed load

This function allows you to shift all edges of a surface outline inwards or outwards by a specified dimension.



Rise height (convert line into arc/arc into line)

Allows you to convert a line into an arc and vice versa or change the curvature by specifying a rise size.

Converting a line into an arc:

Click on the line to be curved to display the dialog "Enter rise size". The maximum positive/negative rise size is displayed. Specify a value within these limits in the rise size input field or click on "Positive maximum (+)" or "Maximum (negative)" to set the maximum value.

You can see the positive/negative direction (referenced to a line from point 1 to point 2) in a sample graphic that is displayed.

Converting an arc into a line:

In order to convert an arc into a line, select the arc and specify the rise size "0" in the dialog.



Move the outline of a distributed load

This function allows you to move the complete outline of a distributed load by clicking on the load outline and dragging it with the help of the mouse to the desired position. Click again to drop it there and finish the moving operation.



Copy the outline of a distributed load

This function allows you to copy a distributed load by clicking on the outline of the load and dragging the copy to the target position. Click again to drop it there and finish the copying operation.



Delete a distributed load

Select this function and click on the outline of the distributed load to delete it. You can restore unintentionally deleted loads with the undo function.



Properties

This function allows you to display and edit load values and coordinates. Click on the outline of the respective load to display the corresponding dialog.

Temperature load

The load that you enter is assigned to the currently active (selected) load case.
→, see also the chapter [Loads](#).

The input of temperature loads is similar to that of distributed loads.
→ see the chapter [Distributed load](#).

Enter the temperatures for the top and bottom face of the plate in °C.

-  Polygonal constant temperature load
-  Temperature load over an existing outline
-  Rectangular constant temperature load
-  Edit the outline of a temperature load
-  Shift the outline of a temperature load
-  Rise height (convert line into arc/arc into line)
-  Move the outline of a temperature load
-  Copy the outline of a temperature load
-  Delete a temperature load
-  Properties

Load transfer from building item *(Only available in the Building application)*

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.

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

Basic features of the graphical input

In addition to the input via the mouse, you can enter lines, surfaces, etc. of course also via the [numeric input of coordinates](#) or by alternating both input modes.

Input of lines

Clicking left with the mouse (do not hold the button) on the graphic screen defines the coordinates of the line's start point. Alternatively, you can enter the coordinates numerically. Move the cursor with the help of the mouse to the line's end point and click again or enter the coordinates of the second point into numeric input fields. Click right and select "Exit".

Note: When defining a line via the option "Two points" you need not finish the operation via "Exit".

Tip: The selection of a suitable [input mode](#) facilitates the input of vertical and horizontal lines.

Polygon lines / surfaces

Start to enter a polygonal chain (e. g. the plate outline) like a single line. Move the mouse successively to the coordinate points of the polygonal chain and click left each time you want to define a point. After having defined the last point, perform a **right click** and select "Exit" in the displayed [context-sensitive menu](#). When entering a closed-up surface, select the option "Close" or "Exit" in the context-sensitive menu to link the last polygon point to the first one.

Rectangles and circles / selection of objects

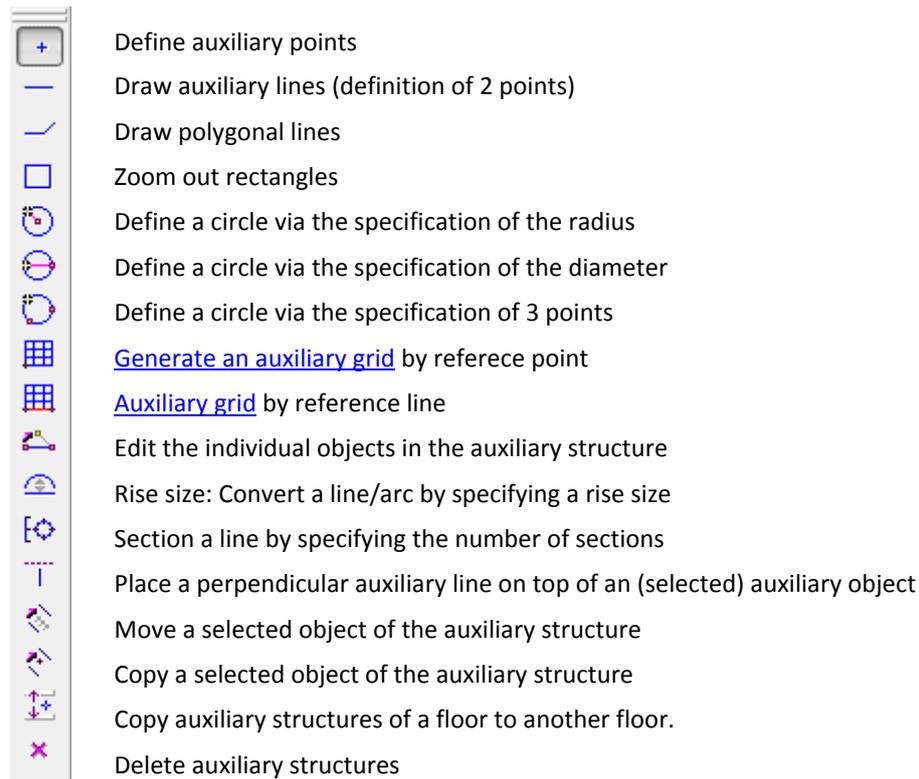
The input of symmetrical surfaces (rectangle, circle) and the selection objects within a particular area, both require the zooming out of a suitable selection area with the help of the mouse. To do this, click on a point that should become the corner point of a rectangle or the centre point of a circle and drag the cursor away from this point to enlarge the area. When the area has the required size, click again to fix its coordinates and size. Any corresponding notes are displayed in the comment line on bottom of the screen.

Auxiliary structures

The "Auxiliary structures" item (right click - context-sensitive menu) or alternatively, the main tree or the menu item "Input" allows you to enter auxiliary lines that facilitate the definition of a complex system. The individual points of an auxiliary structure can be "[captured](#)" during the construction of the system. You can select the individual functions for the input of auxiliary points, lines, rectangles, or circles via the displayed toolbar (see illustration below).

The input is similar to the [input](#) of outlines, columns etc.

See also [Basic features of the graphical input](#).



In order to **edit** an object of a defined auxiliary structure you must select it per mouse click. In the editing mode, you can move individual points of a polygon line, for instance. The **move** function allows you to move an entire object. When using the **section** function you can specify the number of equal parts a line or circular arc should be sectioned into. You can also optically section the lines of an outline, for instance. The **perpendicular** function allows you to define a line perpendicular to the selected line. When copying selected auxiliary structures to other floors, you can define to which floor the structure should be copied in an intermediate dialog.

The function **delete** allows you to delete individual points, lines and circles of the auxiliary structure by clicking on them.

To finish the operation, click right and select "Exit" in the displayed [context-sensitive menu](#).

Auxiliary grid

You can access this function via the "[Auxiliary structure](#)" item (right click - context-sensitive menu) or, alternatively, via the main tree or the input menu item. Click then on the icon  respectively  to activate the function.

First, define the first reference point respectively the reference line (per mouse or numeric input). The dialog displayed subsequently allows you to enter the corresponding parameters.

Spacing

- | | |
|------------|---|
| Regular | This option provides for a regular spacing of the grid lines based on the extent defined in the "grid size" section and the number of spaces defined in the "grid pitch" section. |
| Individual | This options allows you to define the pitch of the grid individually in the "spacing" section. |

Reference point (PS)

The x- and y-coordinates of the previously selected reference point are shown in this section. The reference point is always on bottom left of the grid.

Rotational angle

Enter an angle for the grid – this will be visually displayed for inspection.

Projection

You can define a projection of the grid lines in vertical and horizontal direction in this section.

Naming of axes

You can define axis names in this section. When doing so, you can select for each axis direction whether the axis should be named numerically or alphabetically. The naming of axis always starts of top left of the grid.

Grid size and grid pitch

You can define the size of the grid as well as the grid pitch in each direction.

Spacing (x- or y-direction)

If the option "Individual" was selected, you can define the grid pitch manually in this section.

The button "Applies to additional spaces" provides for the adjustment of all additional spaces to the last entry in the table (if "Individual" was selected).

Auxiliary layers

You can access this function via the item "Auxiliary layers management (icon ) or the main tree (or the input menu item).

Auxiliary layers can be very helpful when entering outlines, walls, etc. in graphical mode.

Insertion of auxiliary layers

You can import DXF files, floors of the currently active item and auxiliary layers saved as *.aux files.



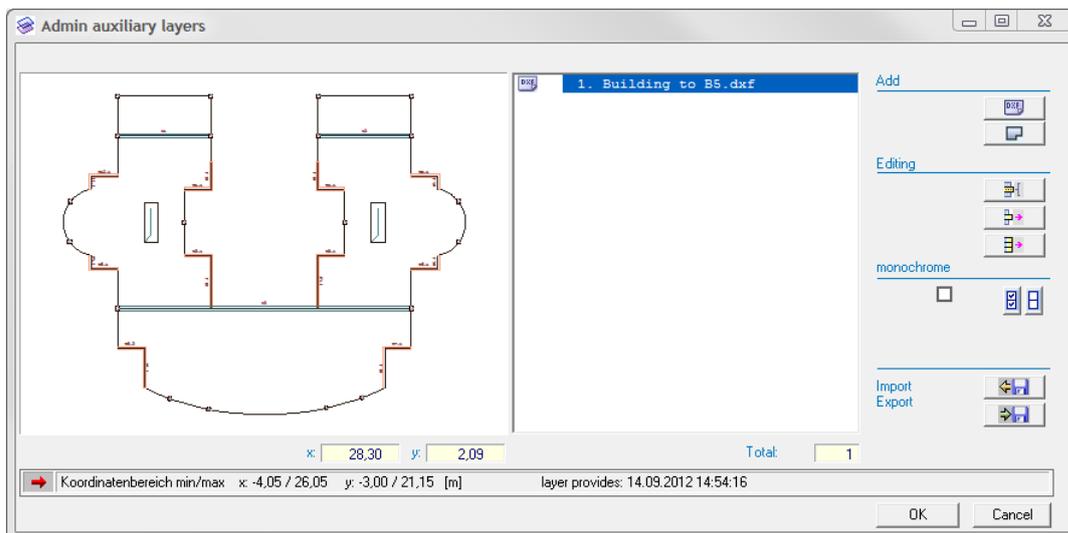
Insertion of an auxiliary layer via the selection of a DXF file.



Generation of an auxiliary layer on the basis of the entered floor data by selecting individual or all (*only available in the Building application*) floors of the currently active item.



Insertion (import) of an auxiliary layer by selection of an AUX file. These files are ASCII files and you can generate them also via the export function of the PLT application, for instance.



The individual auxiliary layers that you have inserted are shown in a list in the right window section. Click on an entry in this list to select an auxiliary layer. A preview of the layer is displayed in the left window section. When you confirm your selection with OK, the layer is shown in the currently active item.

Display and hide an active auxiliary layer

You can easily switch an auxiliary layer on or off by clicking on the icon  (in the upper toolbar).



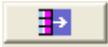
Editing



The properties of the selected auxiliary layer can be edited.



Delete the selected auxiliary layer



Delete all auxiliary layers in the list.

Assign a colour to an auxiliary layer

You can colour auxiliary layers for better distinction.

Tick this option to colour the selected auxiliary layer. To change the colour, click on the selection list box and select a colour per mouse click on the corresponding square or via the "Other" icon.

You can assign a particular colour to **all** layers simultaneously by activating the Select all button on top right or reset the original colour by activating the Deselect button on top right.

The lower colour bar shows the default colour. You can change it to the currently selected colour by clicking on the  icon. The selected colour is then transferred to the default colour bar. Click OK to save the new default colour. You can also assign the colour shown in the default colour bar to the currently active auxiliary layer by clicking on the  icon.



Representation

You can zoom out a section with the mouse and display it in the left window section, a right click returns you to the full screen. The x/y coordinates of the mouse are displayed and help you to control your operation.

Export to AUX file

The export icon allows you to save all auxiliary layers in the list as AUX files and make them available in other applications (e. g. PLT) or new items. AUX is an ASCII format and you can display the content of this file in any text editor.

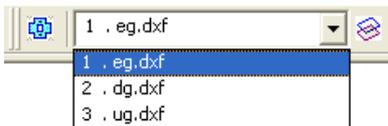
Move, rotate, align, scale auxiliary layers

Via the [context-sensitive menu](#) (right click) or the menu item ▶ Tools ▶ Auxiliary layer, you can move, rotate, align and scale a layer to a selected target position with the help of the mouse or the numeric input of coordinates.

Tip: The activation of the [capture function](#) will facilitate the accurate positioning during the input.

Quick selection of auxiliary layers

In order to select an available auxiliary layer, click on the corresponding list selection box in the toolbar.



Constrained geometric elements

You can enable the input function for constrained points via the option "Constrained elements" in the main tree or by activating the  icon. These points will be considered in the mesh generation and the mesh nodes are assigned to the entered points, lines, circles, etc. during the generation process.



Input of individual constrained points

This option allows you to place the mesh nodes on the points that you enter.

Click successively on the individual positions where constrained points should be set. Finish the input with a right click ([context-sensitive menu](#)) and select ▶ Exit.



Input of an individual line



Input of a polygonal line



Input of a rectangle



Input of a circle via centre point and radius



Input of a circle via perimeter point and diameter

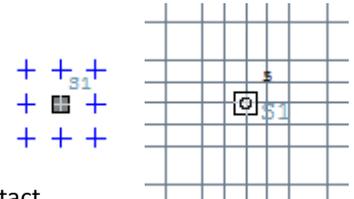


Input of a circle via three perimeter points



Constrained mesh column

After the selection of one or several columns, eight constrained points are positioned automatically around the column at a distance four times as long as that from the gravity point of the column to the outer column edge. This is due to the fact that the automatic mesh generation process produces two elements between two points (in this case the centre of the column and a constrained point) that are relevant for the mesh generation. Therefore, the contact surface outline of the column runs through the centre of the elements or the mid-points of the sides of the adjacent elements.



Constrained mesh wall

After the selection of a wall, the system automatically positions constrained lines (distance from the wall axis: $2 * \text{wall thickness}$). Therefore, the contact surface outline of the wall runs through the centre axis of the first element.

Finish the selection process via the context-sensitive menu.



Edit constrained points

Select the constrained geometric object. You can edit the coordinates of the object using the [numeric input of coordinates](#) or the mouse.



Move a constrained geometric object (using the [numeric input of coordinates](#) or the mouse)



Copy a constrained geometric object or several objects

Finish the operation via the [context-sensitive menu](#) or the ESC key.



Delete constrained points/lines

Click successively on the objects to be deleted. Finish the deleting operation with a right click ([context-sensitive menu](#)) and select ▶ Exit. You can restore unintentionally deleted objects with the undo function.

Capture function



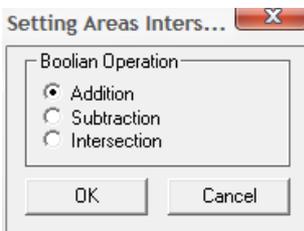
Clicking on this icon enables or disables the capture function.

Capture functions facilitate accurate positioning during the input. Move the cursor towards an existing line or a corner point of an outline, for instance. When the cursor comes close to the line it is "captured". It changes its look into a coloured square indicating the size of the "capture zone". The capture of intersection and limit points of lines is shown by a coloured circle. When you click in capture mode to position an object, it is aligned accurately to this point.

In cases of doubt (if several potential capture points are available, for instance), you should check the correct position with the help of the zoom function.

Boolean operation

► Tools ► Boolean operation

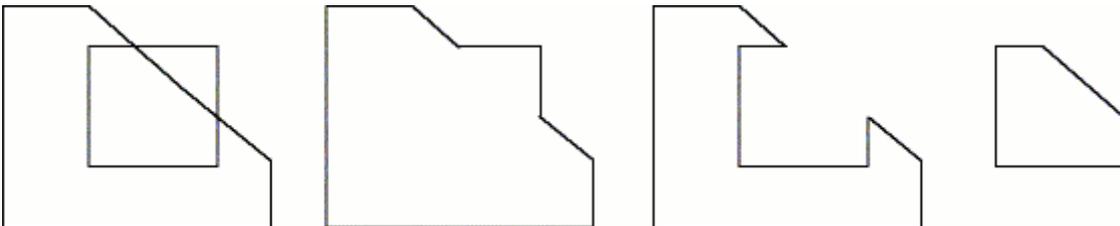


You can combine individual graphical objects (polygonal chain, rectangle, circle) that intersect into a single outline. The type of combination is defined via Boolean operations. The default operation is the addition.

The selected operation remains active during the drawing process until another operation is selected.

Example:

The following illustrations show examples of the combination of two graphical elements (from left to right). First, an outline was drawn using the [polygon function](#). After this, a rectangle was projected over the slanted polygon edge. The second illustration shows the result of an addition, the third that of a subtraction and the fourth that of an intersection.



Numeric input of coordinates



You can enter coordinates of polygon points, corner point, etc, numerically via the input fields **x:** and **y:** in the footer. The input field **z:** is only relevant for the definition of three-dimensional objects (three-dimensional bar structure...) in some applications.

Click into the x-field and enter the x-coordinate, press return and enter the y-coordinate (into the y-field). Press return, the cursor jumps to the z-field. Press return again to show the coordinate on the graphic screen. When entering several points (polygonal chain or rectangle outline...), you can alternate between the input per mouse click and the numeric input at any time. Enter for instance the first point of an outline numerically, zoom out a rectangle with the mouse and click left to define the diagonally opposite corner point of the rectangle.

Fix a coordinate



The check boxes above the coordinate codes (x, y, z) allow you to **fix** a coordinate. This function facilitates the numeric input of several columns with the same x-coordinate because you need not enter this value for each column individually.

Example of a numeric input

Select "Plate contour - Rectangle" in the graphical input section. Enter the number 0 as x-coordinate. (You need not click explicitly into the **x:** field, it is enabled automatically.) Press <Return> and enter the number 0 into the y-field. Continue to press <Return> until the entered point is shown on the graphic screen. Enter now the value 10 (the **x:** field is automatically enabled), press <Return>. Enter 10 also into the y-field and press <Return> successively until the input of the outline is finished and the graphic screen shows a square that has an edge length of 10 m and the lower left corner of which is positioned on (0/0).

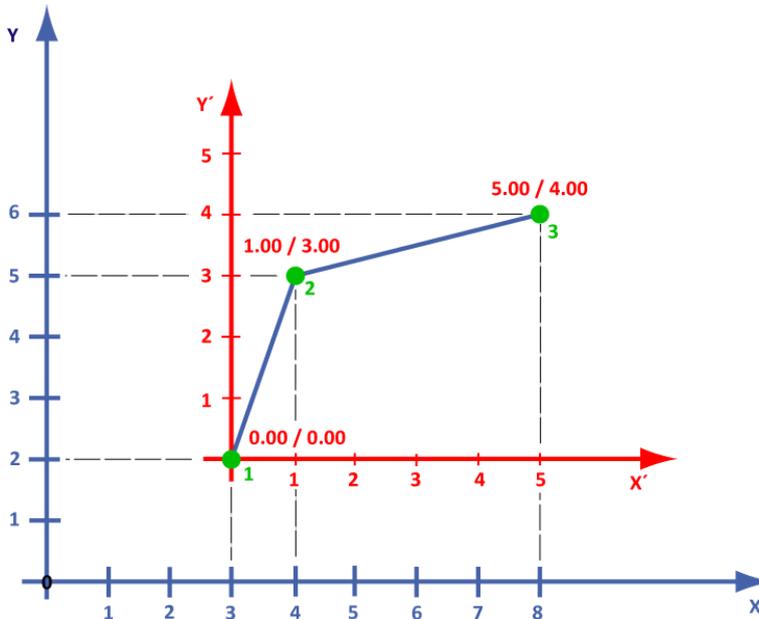
System of coordinates

The input and display of coordinates can be performed in a global or local coordinate system. The zero point of the global coordinate system is fixed, the zero point of the local coordinate system can be defined as useful.

Defining a local coordinate system makes the coordinate input easier because the coordinates can be defined relative to the local zero point (such as the lower left corner of a building).

Explanation of the terms **global**, **local** or **absolute** and **relative** on the basis of an example.

Polygon with three points (1, 2 and 3, green colored):



Global / local

You can describe the polygon shown above with the coordinates of the global (X/Y, blue) or the local (X'/Y', red) system. The zero point of the local system has the global coordinates (3.00/2.00) in the example.

global			local		
Point	x	y	Point	x'	y'
1	3.00	2.00	1	0.00	0.00
2	4.00	5.00	2	1.00	3.00
3	8.00	6.00	3	5.00	4.00

Relative

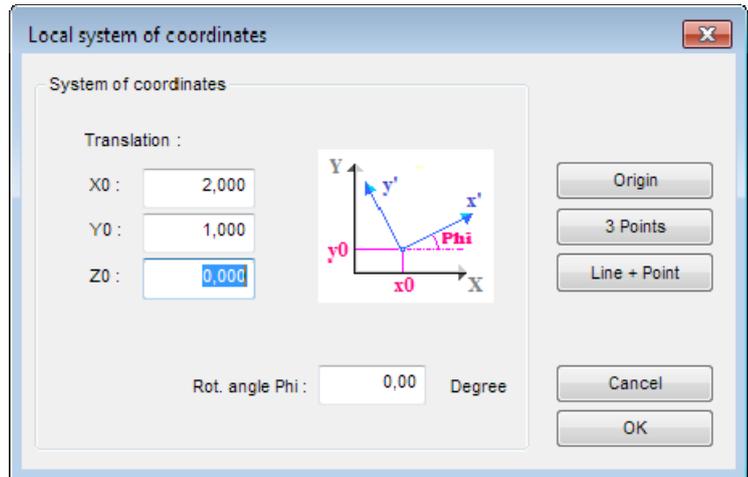
In this mode, the coordinates of the first point are entered in reference to the selected system of coordinates, all additional points are referenced to the respective previous point.

Setting the local system of coordinates

You can define the "zero point" via the dialog

"Local system of coordinates (click on ).

You can define the local system of coordinates either interactively (per mouse clicks on the graphic screen) or numerically, via input of the corresponding values (X_0 , Y_0 , Z_0 , angle of rotation).



Numeric input

The fields X_0 , Y_0 and Z_0 determine the zero point of the local system of coordinates in relation to the global system.

Interactive input

- Zero point** Click on the "Origin" button and subsequently the desired position of this point on the graphic screen.
- 3 points** Click on this button to define the system of coordinates interactively via three points (per mouse clicks on the graphic screen). This mode allows you to rotate and move the system of coordinates, which is particularly useful when defining a rectangular outline that is not parallel to the direction of the global axes. Click first on the point that should be the zero point of the local system, then on a point that should be on the x-axis and finally on the third point at either side of the x-axis to determine the orientation of the y-axis.
- Line + point** This function allows you to define the system of coordinates via a line and a point.

Hint: The background mesh is always transformed in relation to the local system of coordinates.

Input modes



- | | | |
|---|-----------------|---|
|  | Object capture | switches the capture function on and off. |
|  | Background mesh | switches the background grid on and off. |

Input of horizontal/vertical lines

- | | | |
|---|------------|---|
|  | Orthogonal | you can enter horizontal and vertical lines only. |
|  | Horizontal | you can enter horizontal lines only. |
|  | Vertical | you can enter vertical lines only. |

The icons are designed as on/off switches. Clicking on an icon switches a function on, clicking again on this icon switches this function off.

System of coordinates

- | | | |
|--|----------|---|
|  | Local | switches the local system of coordinates on and off. |
|  | Relative | switches the relative system of coordinates on and off. |

Selection modes

Three modes are available to select one or several objects at a time. The selection of the suitable mode allows you to select quickly and reliably individual elements or a group of objects for an editing operation.

- | | | |
|---|---------------|---|
|  | Capture small | This mode is enabled by default if you have not selected any other mode. The cursor is shown as a capture rectangle. This mode is particular suitable for the selection of individual elements and (intersecting) points per mouse click. |
|  | Crossing | In the crossing mode, you select objects by zooming out an area with the help of the mouse. All elements that are included partially or totally in this area or intersect/cross this area are selected. |
|  | Window | In the window mode, you select objects by zooming out an area with the help of the mouse. Only elements that are totally included in this area are selected. |

Multiple selection via the Ctrl key

In order to edit the properties of several similar objects (several walls, columns, downstand beams...) you can select these objects by clicking on them successively while keeping the Ctrl key pressed. When you release the key the properties dialog for the respective object type (e. g. column) is displayed. You can edit the parameters for all selected objects in this section. This function of the Ctrl key is also available for the selection modes crossing and window. You can also copy or move several objects simultaneously with the help of the multiple selection mode.

Context-sensitive menu

You can display the context-sensitive menu(right mouse button).

The options of the context-sensitive menu depend on the currently active input function.

All input functions (Plate contour, wall, column,... loads) are also available per right click. A description of these functions is given in the corresponding chapters.

Consecutive input several points

Depending on the currently active input mode, the following functions are additionally available per right click:

Exit

This option allows you to finish operations the completion of which cannot be detected by the application such as the input of polygon lines.

Cancel

This option allows you to abort the current input operation.

Undo

If you have entered an erroneous polygon point during the input of a polygon chain for instance, this options returns you to the previously defined point.

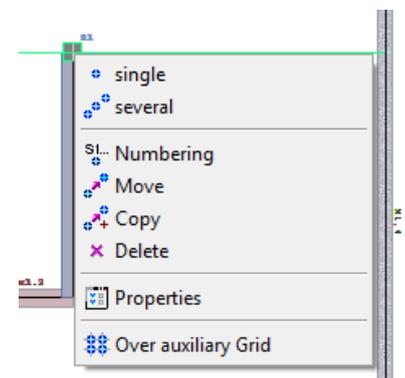
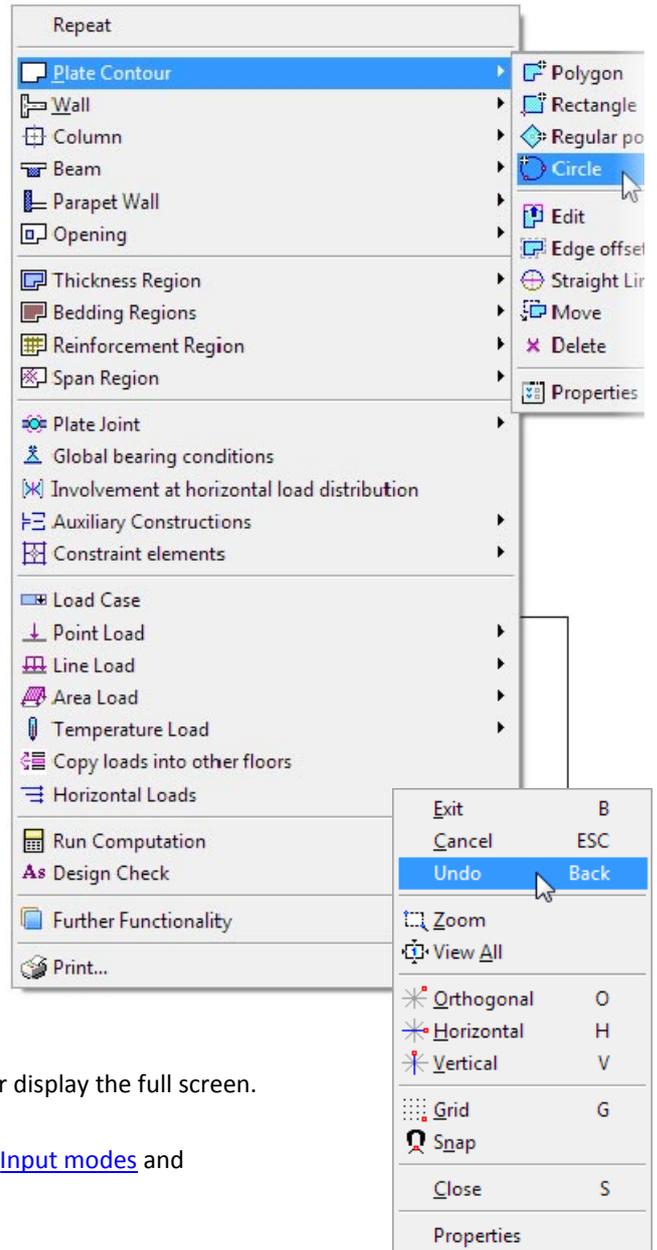
Zoom/View all

This option allows you to zoom a section of the screen or display the full screen.

Further options (Orthogonal...) are described in chapter [Input modes](#) and [Grid / background mesh](#).

Object-related context menu

The optionally activatable object-related context menu is activated via the "Show values"  symbol. With the object-related context menu activated, you can directly select the appropriate context menu for objects such as column, wall, loads, etc. - right-click on it. When moving over the objects (column, wall, etc.) with the mouse they are highlighted in color.



Settings in the context-sensitive menu

These options are accessible via the [context-sensitive menu](#) (right click) ▶ Further Functionalty ▶ Settings.

General

In this section, the coordinates of the current screen size and the displayed window section are shown. You can edit these coordinates in order to define the section to be shown numerically instead of using the zoom function.

The following options are available for the display of the system of coordinates.

- The axes of coordinates are not shown
- The global axes passing though the zero point are shown
- The global axes are shown on bottom left with direction indication.

Grid / Background mesh

The background mesh is accessible via the [context-sensitive menu](#) (right click) ▶ Further Functionalty ▶ Settings ▶ Grid.

The background mesh facilitates the graphical input. Due to the [capture function](#), you can align the objects that you enter exactly to the intersection points of the grid (with the exception of numerically entered points and those resulting from the object capture such as end points of existing lines).

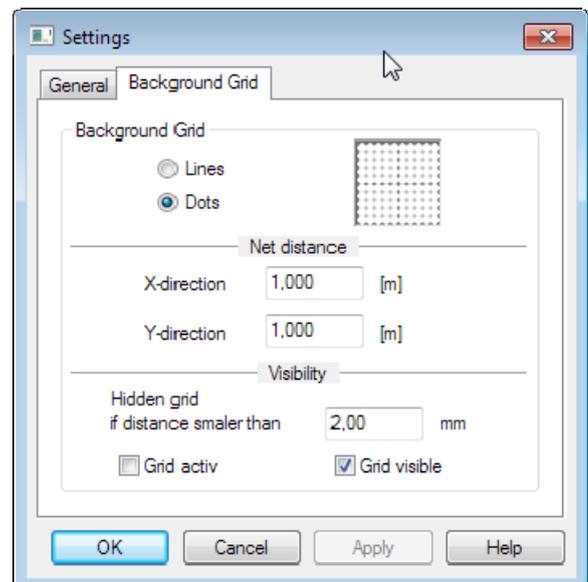
You can select whether the mesh is shown as a line or point grid.

The spacing of the grid lines is specified in [m].

Hidden grid Depending on the size of the representation, it might be useful to hide grid lines/points (we recommend values between 2 and 5 mm).

Grid active The ticking of this option enables the capture function. Untick this option to position a point close to a grid line for instance.

Grid visible You can show/hide the mesh by ticking this option, the capture function is still active then.



Note that the background mesh points are always captured when you move the mouse while the background grid is enabled. As a consequence, only coordinates on the grid are shown in the display of the numerical coordinates.

View toolbar



Move the mousecursor over an icon to show the function of the icon.

Functions:

- Display values when the cursor is positioned on an object (column...).
In addition, the object-related context menu is activated, e.g. you can right-click directly on the appropriate context menu for objects such as column, wall, loads, etc.
- Increase the font size of texts displayed on the graphic screen.
- Reduce the font size of texts displayed on the graphic screen.
- Show/hide exterior dimensions.
- Measure distances and angles. Click successively on the two end points of a distance to be measured. The measurement result is displayed subsequently.
- Visibility: show/hide options - define in the pertaining dialog which graphical elements (outlines, walls, columns, loads, texts, etc.) should be shown or hidden.
- 3D-view of the loads
- Visibility: show/hide the loads of the [selected load case](#).
- Access the [basic parameters](#) dialog.

Construction mode

Construction toolbar

The toolbar is context-sensitive and therefore displayed automatically (you can optionally hide it:

► Options ► Settings ► Construction mode.

It provides for quick changeover between the modes orthogonal, horizontal and vertical when entering polygon lines for instance.



The functions from left to right:

Zoom, full screen, orthogonal, horizontal, vertical, exit, cancel.

ESC event in input mode

This option (► Options ► Settings) was implemented to ensure a uniform handling of ESC commands when using FRILO applications and ALLPLAN.

You can select whether pressing the ESC key should

- abort the input of data without saving the entered data
- abort the input of data while saving the entered data

If this option is ticked and you press the ESC key during the input of a polygonal line for instance, a displayed dialog prompts you whether to save the entered polygon line or cancel it. If you tick the option "Do not ask this question again" your selection (yes/no) is applied to all subsequent ESC commands. To display the dialog again for each activation of the key, untick the corresponding option again.

Additional menu items

The graphical input module adds special menu items providing for the graphical input to the menu items of the main application to which it is linked.

The following menu items refer to functions of the graphical input:

Graphical options

You can display or hide the following views:

- Show/hide auxiliary layers An imported auxiliary layer (DXF ...) can be shown or hidden.
- Show/hide FE-critical points If FE critical locations are detected in the system (e.g., wall end not set exactly), these locations will be marked with a colored circle. The display of these circles can be switched off with this option if required.

Tools

[Boolean operation](#)

The combination of surfaces via Boolean operations is a user-friendly feature for the graphical input of complex outlines → see the chapter "[Basic features of the graphical input](#)".

[Measurement](#)

Round coordinates

Measurement of distances and angles.

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

Point Modify

You can move a common point (identical coordinates) of several objects to a new position using the mouse or the numeric input of coordinates. Click on the desired point and enter subsequently the target coordinates or click on the respective target position.

Mirror

Setting:

Click this option, if you want the mirrored objects/floor to be reproduced (like a copy-function).

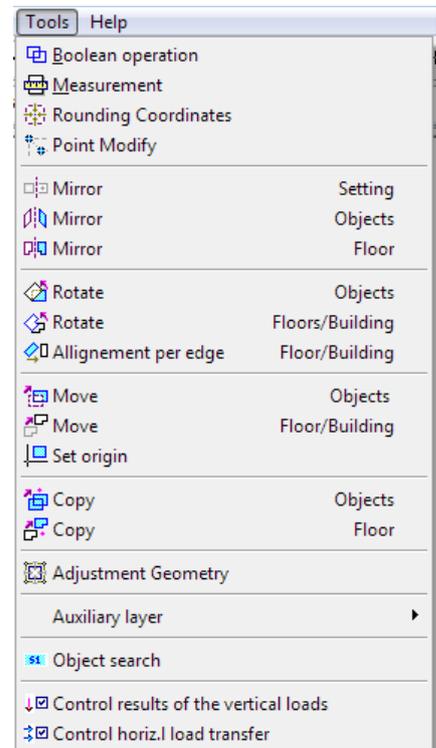
Objects:

Like in CAD tools, you can mirror one or several objects. Any type of object (columns, walls, beams, etc.) or a combination of different types of objects can be selected and (re)produced as a mirror. For reproducing the object select the option under „Mirror - Setting“.

Select the objects and finish the selection process via the context-sensitive menu ▶ Exit. Define subsequently the axis (select 2 points of the axis by mouseclicks or numerical input) along which the mirrored object should be (re)produced.

Floor/complete:

If you want to mirror the whole floor, use this function (as described under „Objects“ - yet the selection of objects is not applicable).



Rotate	<p><u>Objects:</u> Select the objects as described for the mirror 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 "numeric input".</p> <p><u>Floors/Building/complete:</u> Rotate all geometric objects of the currently selected floor including loads. Select the centre of rotation per mouse click. Perform the rotation via a mouse movement or the numeric input of the number of degrees (a corresponding input field is displayed on the bottom of the screen).</p>
Alignement per edge	<p>As an alternative to rotation, the plan view can be rotated with respect to a selected contour edge (click on an edge) or an auxiliary line so that this edge is directed "automatically" either parallel to the X or Y axis, depending on which direction the smaller angular difference is.</p>
Move	<p><u>Objects:</u> Select the objects as described for the mirror function. Move the objects via drag and drop or numeric input.</p> <p><u>Floor/Building/complete</u> Move a floor/building (interactively). You can move an entire floor/building by defining first the start point for the moving operation and then the target point to which the start point should be moved per mouse click.</p> <p>You can also move by entering numerical values (x-/y-coordinates). In order to move an object by one metre in the x-direction for instance, enter X=1 and Y=0.</p>
Set origin	<p>As an alternative to moving the floor/building, the origin of the coordinate system can be moved.</p>
Copy objects/Floor	<p>Select the objects as described for the mirror function. Position the copied object via drag and drop or numeric input. When copying an entire floor click in the graphics window and then move the copy of the entire floor to the desired position.</p>
Geometric alignment	<p>This function provides for geometric alignment of the plate if problems occur during the generation of the FE mesh. The alignment ensures that the FE mesh can be generated.</p>
Auxiliary layer	<p><u>Move/rotate/align/scale:</u> You can move/rotate/align by edge or scale a displayed auxiliary layer. The cursor changes its look into a square. Click on an arbitrary (distinctive) point of the displayed auxiliary layer. You should enable the capture function for this operation. You can implement the movement of the layer per mouse click on the target (zero) point of via numeric input of coordinates.</p>
Object search	<p>Would you like to know where the column "S5" or the wall "W1.2" is located? In an input box underneath the graphic dialog (signed with an „X“) you can enter the name of an object the object an begins to blink.</p>

control of ...

Only GEO

Interactive control dialog: results of the computation of the vertical/horizontal loads. Problem spots are indicated by an oval mark highlighted. Right-click on this marker to bring up the context menu:

- load differences,
- load composition and
- Settings

When clicking on load differences, a description of the cause of the difference is displayed with the exact name of the component where the difference occurs. By clicking on Settings, the percentage deviation considered critical can be set.

Three-dimensional representation (OpenGL)

This function provides for a rendered (three-dimensional) representation of the system.

You can access the three-dimensional mode by clicking on the icon .

Automatic data backup

! Only available in the Building + PLT applications

▶ Options ▶ Settings - APPLICATION NAME ▶ Automatic data backup

When selecting the option "Automatic background data backup", the data that you enter are saved at pre-defined intervals. The most recently saved version of your item is made available after a system crash. This option is particularly helpful when entering complex items.

Interval: Specifies the period between automatic backup runs.