Slabs by Finite Elements PLT

Contents

Application options

The PLT application is based on graphic features and supports the calculation of plate load-bearing structures with complex bearing conditions or load arrangements that can hardly be handled using traditional approximation methods.

The Graphical input module offers numerous functions and options that provide for a quick and comfortable system generation and give a detailed system overview at the same time.

This application was particularly developed for the definition and design of reinforced concrete slabs with downstand beams and elastically bedded slabs.

If a design computation is not required for the defined system, you can use any orthotropic or isotropic material, → see basic parameters.

You can generate any slab outline, recess or wall shape via the Graphical input of polygon lines and arc elements.

Downstand beams always have a line shape.

Standards

- DIN EN 1992
- ÖNORM EN 1992
- NTC EN 1992
- BS EN 1992
- PN EN 1992
- EN 1992

Interfaces to CAD systems

You can import/export DXF files with auxiliary structures for instance.

Formwork drawings from CAD systems make Glaser (ISB-CAD) can be imported and edited. Formwork drawings from ALLPLAN can also be imported via the ASCII interface. The transfer of reinforcement calculation results to ISB-CAD or ALLPLAN-CAD is handled via direct interfaces.

ASCII interface

Interface for the export/import of system data.

Restrictions

- Only one material per slab is admitted.

- A linearly elastic calculation (state 1) is performed.

- For the deflection, a calculation in state II can be activated.

- Sheet stresses are not available.

- Separate slabs can not be calculated.

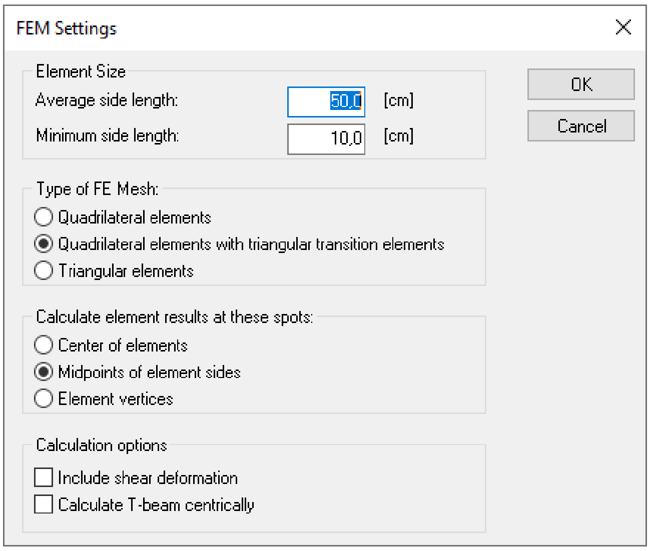
Basis of calculation

Mesh generator

The implemented mesh generator works according to the "Advancing Front Method". It is suitable for mesh generation based of two-dimensional surfaces of any shape.

You can generate meshes with triangular and squared elements as well as mixed meshes. First, define nodes along the default lines. After this, generate successively at several active fronts triangular and/or squared elements. During the generation of the elements, the quality of each newly generated element is examined and optimized.

FE section



Elements with 4 or 3 nodes are used for the calculation of the slab.

Hybrid elements are available for thin slabs, which are common in general building construction. The advantage of hybrid elements resides in the fact that the moments and shear forces can be calculated with a considerably higher accuracy.

In contrast to thin slabs, in the calculation of which the shear deformation can be neglected in accordance with Kirchhoff's theorem, it might be necessary to consider the shear deformation with thick slabs. To be able to do this, the application offers additional elements based on the discrete Kirchhoff-Mindlin method in the section FE mesh ‣ settings.

Where unbedded slabs are concerned, the ratio of the shortest span (l) between two bearings (wall/column) to the slab thickness (d) is often used to simplify the distinction between thin and thick slabs. According to this method, a plate is thick when

$l/d < 10$ is true.

The calculation of action-effects on the element could be performed at various positions:
- element centre,
- element corner points,
- mid-points of element edges.

T-beams

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

Design

The design of the reinforcement is performed in accordance with the Baumann method. A cracked slab element is used as a model. The direction of the cracks results from the condition that the deformation energy produced by the reaction forces must be a minimum. The design approach assumes an orthogonal mesh reinforcement in the first place.

Input

Graphical input

The PLT application offers a graphical user interface, i.e. elements such as the slab outline, load coordinates etc. are drawn with the help of the mouse on the basis of a DXF file, for instance, and only particular values, such as those of forces, have to be entered numerically in corresponding dialogs.

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

The "Graphical input" is an independent application module that is linked to the PLT application. The functions of the Graphical input module are described in a separate document Graphical input.pdf.

Numeric input

Of course, you can enter values and coordinates any time via numeric input fields if you want to make a precise numerical specification. How to do this is described in the document Graphical input.pdf.

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

DXF import

You can import geometrical data providing the basis for the system definition via the DXF interface. Glaser files (-isb CAD interface) and ALLPLAN CAD files (ASCII interface) can be processed directly.

System and load input

The system and load input functions are part of the "Graphical input" module and are described in detail in the document Graphical input.pdf.

The definition of a system starts with the input of the slab outline and the definition of the basic parameters.

The basic parameters include material, standard selection, slab thickness, ceiling top edge, floor height, concrete cover, torsional rigidity, bedding, tension spring exclusion and possibly durability.

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

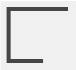
Result sections

Access via the main tree ▸ Result sections

This function allows you to define result sections. After the calculation, you can display the action-effects, the deformation behaviour, the base compression behaviour (for bedded slabs) as well as the behaviour of the values indicating the cross section of the longitudinal reinforcement.

Note: Finish the operations per right click and select "Exit".

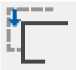
The section is defined by clicking on an edge / line.



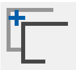
Enter a section as a polygon line. Define your polygonal section line with the help of the mouse or via the numeric input.

Result section defined via two points (line).

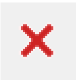
Edit the course of a section subsequently. Click on the corresponding section and drag the corner points to the desired target positions using the mouse.



Move a section. Click on the corresponding section and drag it to the desired target position with the help of the mouse.

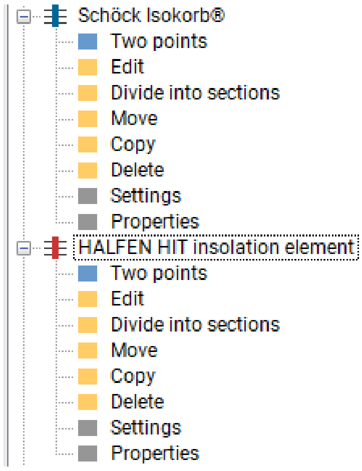


Copy a section. Click on the corresponding section and drag the copy with the help of the mouse to the desired target position.



Delete a section or several sections (one after the other).

Schöck-Isokorb® / HALFEN HIT Iso-Element



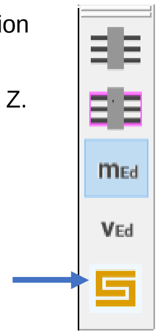
Determining a Schöck Isokorb or a HALFEN HIT Iso-element based on existing internal forces.

Similar to the definition of a result section, a line can be defined along which, due to the moments and shear forces present there, a suitable Isokorb is proposed.

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

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

From Release 2024-2:



Call up the results dialog for the Schöck Isokorb® via the Schöck button (Fig. right, below). Manual selection of products is also possible in the dialog.

Further functions: Isokorb with height offset type K-U and K-O as well as with free load-bearing effect type Z.

Optional neglect of small internal forces. Display of bar spacing for collision planning.

Websites

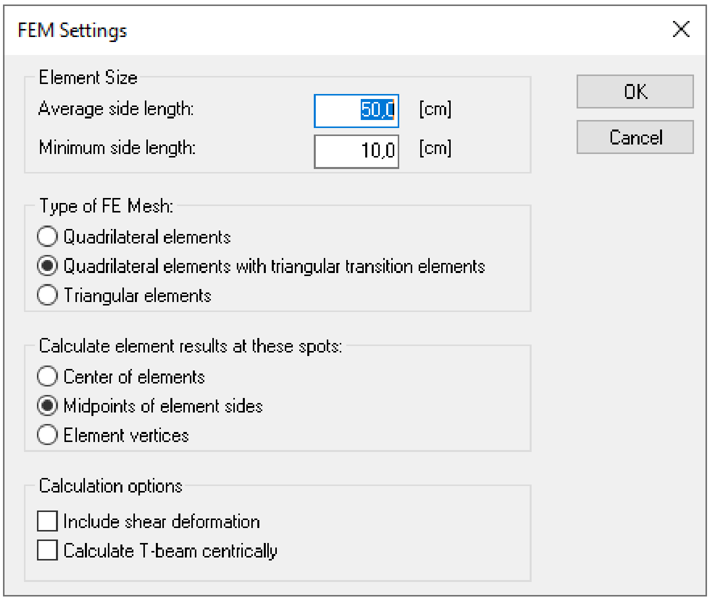
https://www.schoeck.de

https://www.leviat.com

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

FE mesh

See also → [Basis of calculation]



Properties

You can define various basic settings relevant for the generation of the FE mesh in this section:

Element Size	Specify the desired (average) element size (edge length) for the automatic mesh generation. If the mesh cannot be generated with this size it is reduced automatically. *Tip: You should always select the size for the FE mesh in such a manner that the deformation line comes close to reality, i.e. each field should at least have six elements.*
Minimum side length	You can define the minimum element side length. The side length used for mesh generation must not fall below this value. If smaller elements are required, the mesh generation is aborted and a corresponding message is displayed.
Type of FE Mesh	You can choose between rectangular elements, rectangular with triangular transition elements and triangular elements.
Element results ...	You can select the places where element results should be calculated.
Include shear deformation	In this section, you can change over from hybrid elements to elements based on the Kirchhoff-Mindlin theorem for the calculation, → see Basis of calculation. When the shear deformation should be considered, elements based on the Kirchhoff-Mindlin theorem are used instead of hybrid elements for the FE calculation. The following restrictions apply to these elements: 1. You cannot reduce the torsional rigidity of the slab, i.e. the reduction factor is "1.0" (full torsional rigidity) → see Basic parameters (Graphical input.pdf). 2. You cannot define an orthotropic material for the slab, → see Basic parameters (Graphical input.pdf). 3. You cannot use supporting direction areas.

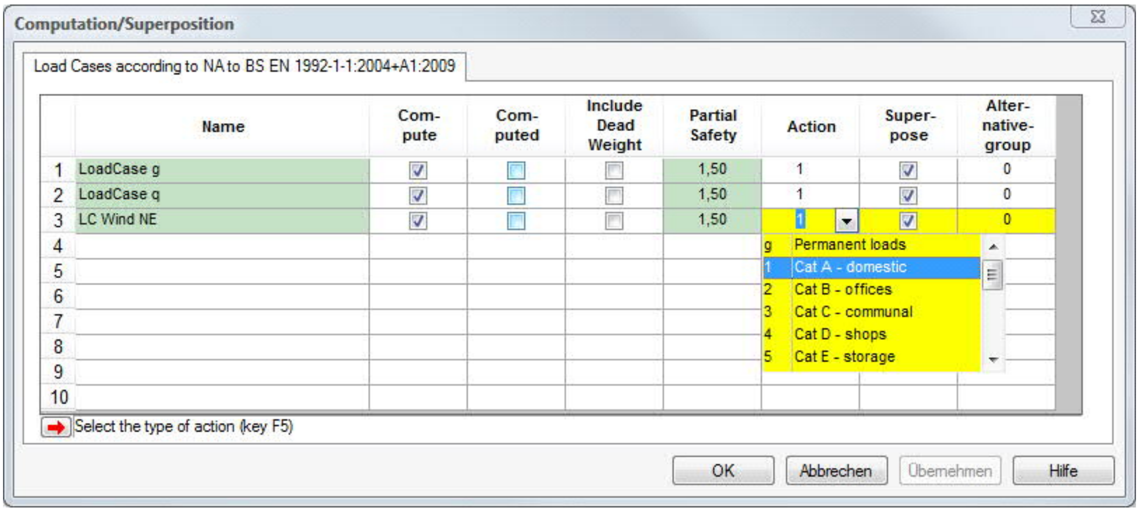
Create

This menu item launches the generation of the FE mesh based on the values and options set in the "FE mesh properties" dialog. Alternatively, you can click on the icon to generate the FE mesh.

Delete

This menu item allows you to delete an existing FE mesh. Alternatively, you can click on the icon to delete it.

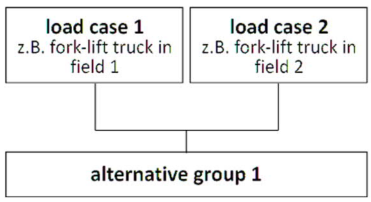
Computation / Superposition...



Compute	Tick/untick this option to select the load cases that should be included in the calculation.
Computed	You can see in this column whether a load case has already been calculated.
Incl. Dead Weight	Tick this option if the selfweight should be considered in the calculation.
Partial Safety	This section displays partial safety factors depending on the selected type of action (permanent or non-permanent).
Action	Select the desired type of action from the selection list.
Superposition	Tick the load cases that should be considered for the superposition.
Alternative group	Load cases of the same alternative group exclude each other. You can enter load cases that cannot occur simultaneously with the help of so-called alternative groups. Example: Wind from the left or the right, load position of a fork lift. Loads of the alternative group "0" may occur in combination with all other load cases. All load cases of an alternative group (marked with the same number) exclude each other. Obviously, only load cases from non-permanent actions can be members of alternative groups. The alternative groups are considered after the calculation in the course of the superposition of the results. Therefore, they can only be used for linear calculation (i.e. no tension spring exclusion).

Example of an alternative group (see ill.)

The load cases 1 and 2 are assigned to alternative group 1 because the fork lift is either in field 1 or in field 2.



Note: *A subsequent superposition of the individual results cannot be performed for non-linear calculations (tension spring exclusion for bedded slabs or walls). Therefore, the leading action must be determined in advance. The leading action column (last column) is only enabled for non-linear calculations, the value “foundation stiffnes” therefore must be > 0.*

Start the calculation by confirming this window with OK.

Design: Settings

The section Design: Settings offers various options and settings depending on the selected standard (Bending, Shear force, Servicability, Superposition)..

Bending ...

This section allows you to select for slabs and beams individually.

Note instructions in the dialog box.

Design orientation

As global design directions, the horizontal and vertical directions are predefined.

If required, the global design directions can also be defined by the user.

Minimum reinforcement

Optional output of the minimum reinforcement to ensure the ductile component behaviour as per EN 1992-1-1, 9.3.1.1 . Tick this option if the required reinforcement should be included.

Global preset reinforcement

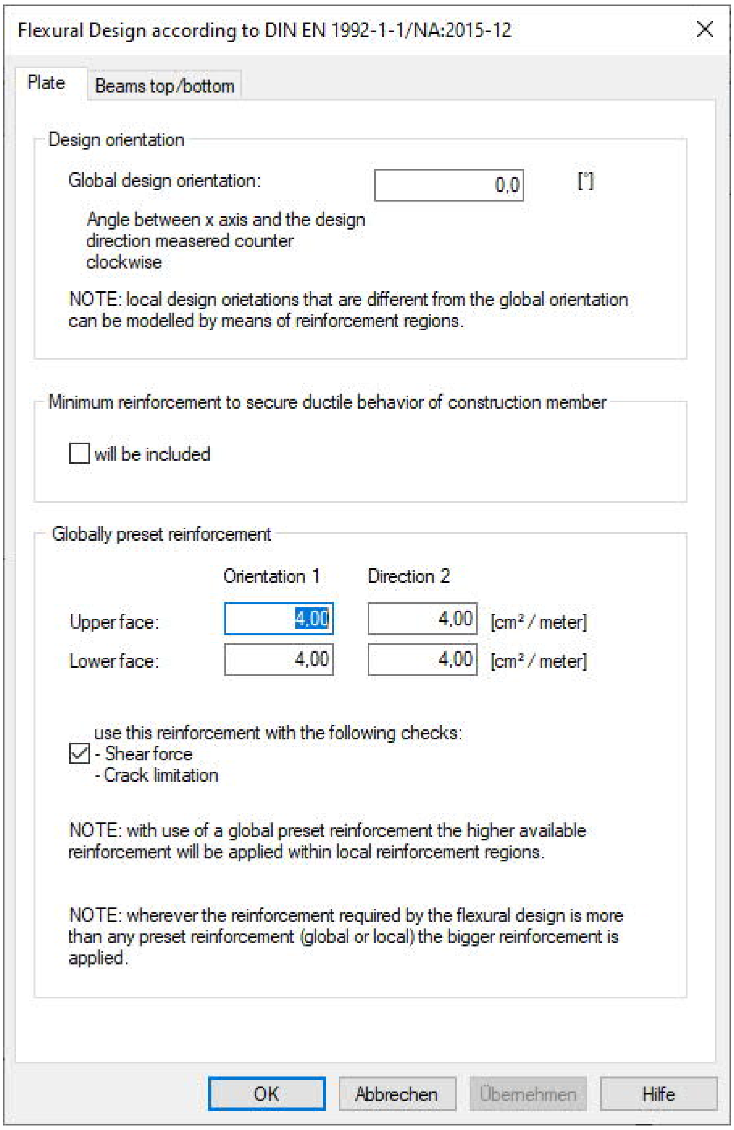
According to EN 1992-1-1, the design value of the shear force capacity of the concrete $V_{Rd, ct}$ is calculated as a function of the percentage of reinforcement of the flexural tension reinforcement (para. 10.3.3, equation 70).

Default for the determination of the percentage of reinforcement is the reinforcement required by the bending design.

You can also specify a reinforcement (upper/lower, direction 1/2): In this case the program applies always the higher reinforcement value of the statically required and the default reinforcement.

Note:

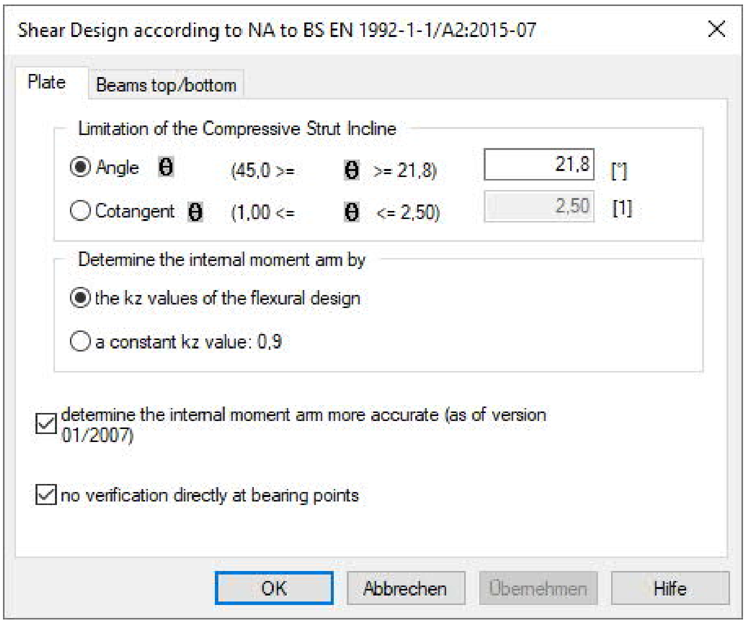
If local reinforcement areas have been defined the greater reinforcement is applied in these areas.



Shear force ...

Limitation of the strut inclination

For the calculation of the shear force reinforcement, the application assesses the minimum strut inclination depending on the shear force acting at the respective point of design. The user can can provide for a steeper angle via the option "limitation of the strut inclination".



Calculation of the internal lever arm with...

You can select via this option whether the internal lever arm assessed for the bending design calculation or $0.9 \cdot d$ should be used for the shear force design calculation.

More accurate calculation of the internal lever arm and the concrete cover

When this option is ticked, the settings made in the Durability section concerning the concrete cover and the bar diameter are considered for the calculation of the internal lever arm.

No verification directly on bearing points

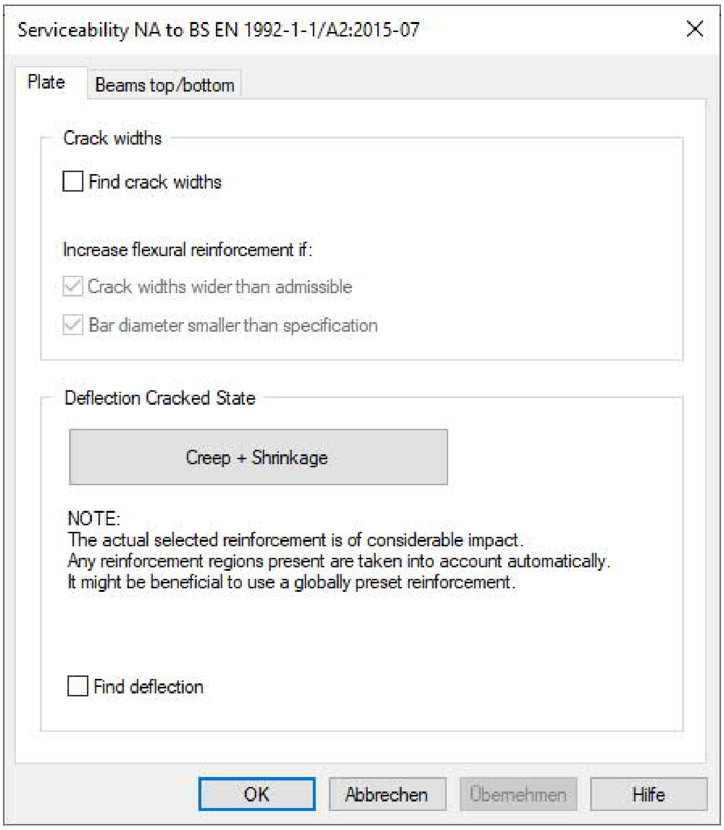
When you tick this option and you have selected the "mid-point of element sides" as a design point for instance, the design point on the wall axis is not considered in the shear force design calculation at walls.

Beams top/bottom

The description of the options on the slab (plate) tab applies analogously to the shear force design of downstand beams.

Serviceability ...

To unlock this menu item, the "durybility" option in the dialog to be "[base parameters]" must be set.



Crack widths

In accordance with EN 1992-1-1, the calculation of the existing crack width and/or the permissible limit diameter of the longitudinal reinforcement depends on the percentage of the flexural tension reinforcement (para. 11.2.3 and 11.2.4) – see bending.

Increase the flexural reinforcement

With this option the program increases the flexural reinforcement as long as the crack width verifications are met.

Deflection state II

By setting the option "Find deflection" the verification of the deflection state II is activated.

The deflection in state II is calculated for the quasi-permanent combination.

Here then a precise calculation of deformations by numerical integration of the curvatures is performed.

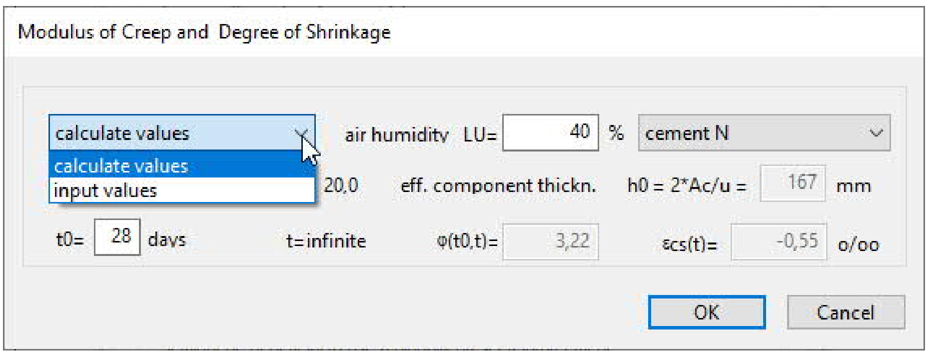
The basis for this are the Mk-lines (with consideration of the cracking and the involvement of the concrete under tension for a particular cross-section).

Since the calculation depends on the existing reinforcement, it is appropriate to take into account the planned reinforcement by the entry of a global or local reinforcement.

Creep + Shrinkage

Über diesen Button können die Einflüsse aus Kriechen und Schwinden in einem Dialog optional vorgegeben oder vom Programm berechnet werden.

Options for the influences of creep and shrinkage: definition of the values or calculation by program.



See document Durability - Creep Coefficient and Shrinkage Strain.pdf.

Superposition ...

In the standardized design situations of the ultimate limit state it has been defined which action groups in which situation has to be considered

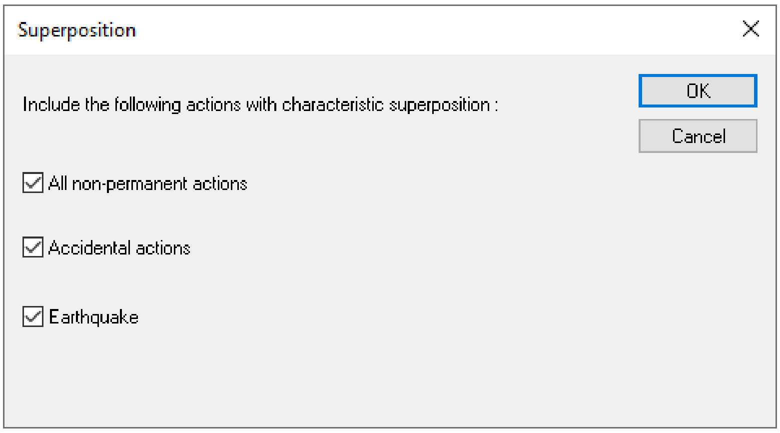
The so called “characteristic” superposition does not correspond to the standardized design situations, but is a simple superposition without partial safety- and combination coefficients.

Thus for the characteristic superposition we provide the following options, so you can define which action types should be taken into account. In this way you can compare the influences of different actions.

- All non-permanent actions

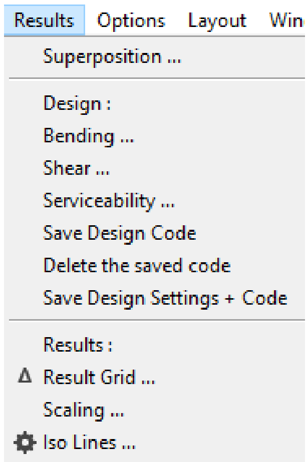
- Accidental actions

- Earthquake



Results: Settings

Grid



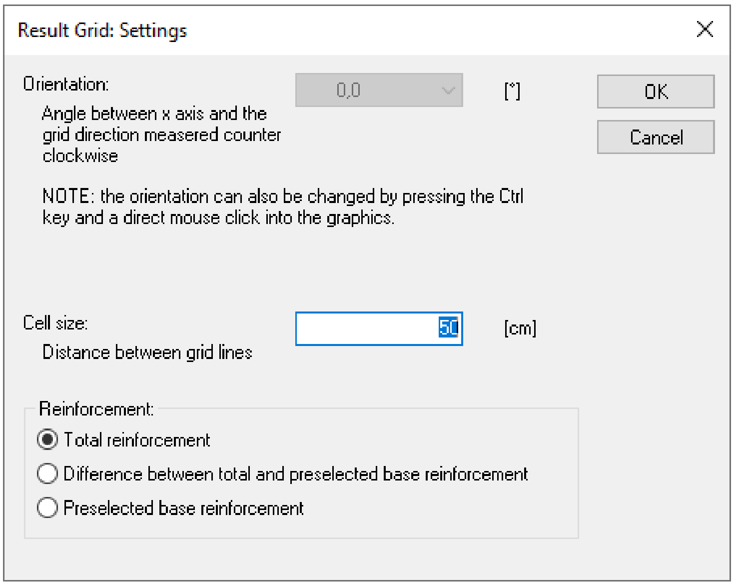
"Results - Grid" menu

Orientation

The orientation (angle) of the output grid depends on the value set for the reinforcement area. If several areas with different orientations are defined or the defined reinforcement areas do not cover the entire slab, you can switch over between the different angles in this section. The grid shows only the areas with results for the corresponding angle. Areas for which no rotated reinforcement area was defined are consequently shown when you select the orientation 0 [°].

Cell size

You can define the spacing of grid lines in this section.



Reinforcement

You can select whether the reinforcement areas to be put out should include the total reinforcement, the difference between the total and the default reinforcement or merely the default reinforcement.

The application analyzes the results of the FE-elements included in the area of the result grid and shows the relevant results in the grid.

It may happen that results are shown in the area of a recess, for instance, when small recesses have been defined . This effect is due to the regular layout of the result grid. The displayed results refer to FE-elements that border the recess, i.e. the reinforcement put out in this section must be inserted at the edge of the recess, for instance. In some cases, you can improve the representation by modifying the size of the result grid.

Scaling

"Results - Scaling" menu

In this section, you can define the scaling factors for the representation of the deformation results as a three-dimensional mesh (for individual load cases) or the wall results (for printing and/or display on the screen).

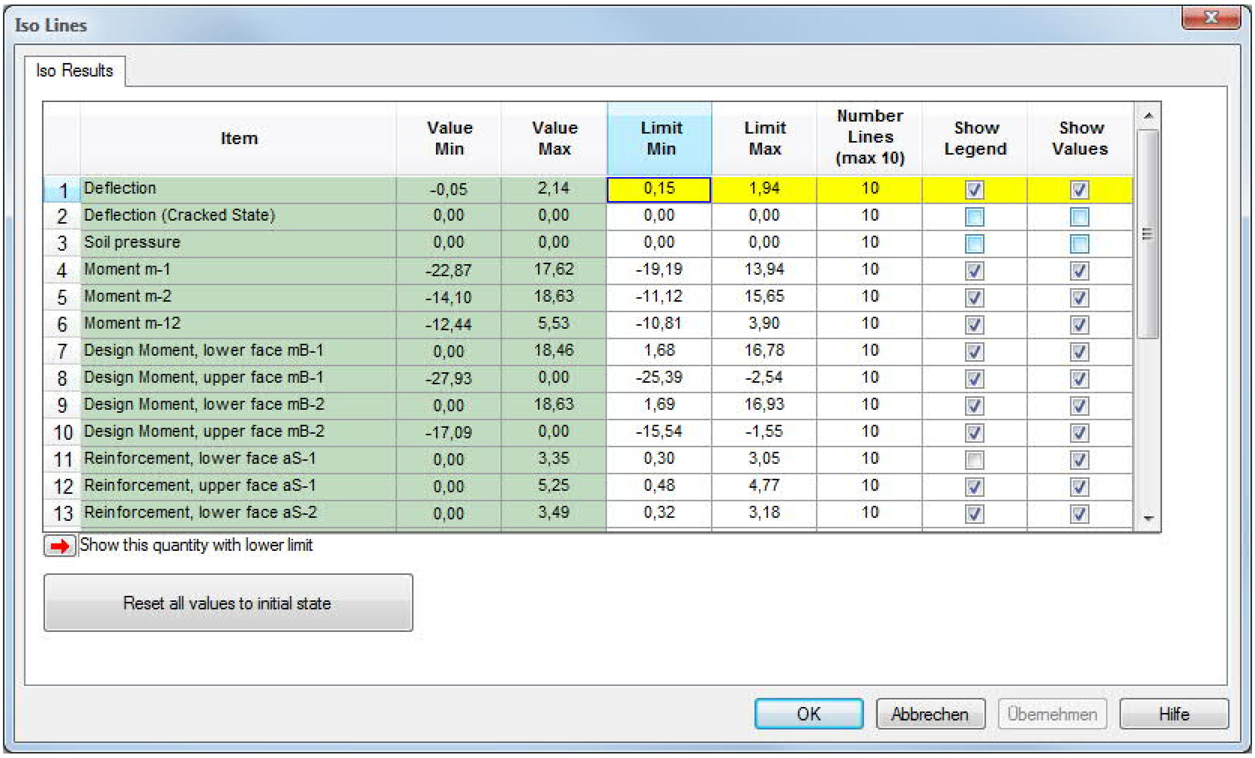
Iso lines

Various settings are available for the individual result parameters:

- lower and upper limits and

- number of subdivisions

The representations of legends and line labels can be switched on and off.



	Item	Value Min	Value Max	Limit Min	Limit Max	Number Lines (max 10)	Show Legend	Show Values
1	Deflection	-0,05	2,14	0,15	1,94	10	☑	☑
2	Deflection (Cracked State)	0,00	0,00	0,00	0,00	10	☐	☐
3	Soil pressure	0,00	0,00	0,00	0,00	10	☐	☐
4	Moment m-1	-22,87	17,62	-19,19	13,94	10	☑	☑
5	Moment m-2	-14,10	18,63	-11,12	15,65	10	☑	☑
6	Moment m-12	-12,44	5,53	-10,81	3,90	10	☑	☑
7	Design Moment, lower face mB-1	0,00	18,46	1,68	16,78	10	☑	☑
8	Design Moment, upper face mB-1	-27,93	0,00	-25,39	-2,54	10	☑	☑
9	Design Moment, lower face mB-2	0,00	18,63	1,69	16,93	10	☑	☑
10	Design Moment, upper face mB-2	-17,09	0,00	-15,54	-1,55	10	☑	☑
11	Reinforcement, lower face aS-1	0,00	3,35	0,30	3,05	10	☐	☑
12	Reinforcement, upper face aS-1	0,00	5,25	0,48	4,77	10	☑	☑
13	Reinforcement, lower face aS-2	0,00	3,49	0,32	3,18	10	☑	☑

After the computation the setting options are accessible via

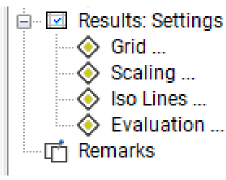
1. the menu bar ▸ Results ▸ Contour lines

2. the main menu ▸ Results: Settings ▸ Contour lines

The results can be displayed as isolines/contour lines via the symbol .

Note: *The settings apply to all load cases and load case combinations. Depending on the location, the selected section cannot be displayed in combination with particular load cases and combinations.*

Evaluation

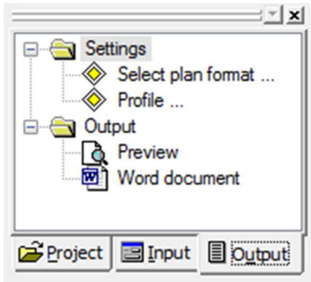


Here you can define whether the node results are smoothed along the sections. For internal forces, you can also select whether the smoothing is to take place with an extreme value or an average value. Furthermore, the decimal places for the support forces and the grid results of the reinforcement can be defined.

Output & results

The "Output" tab offers the following settings and output options:

Select a plan format...



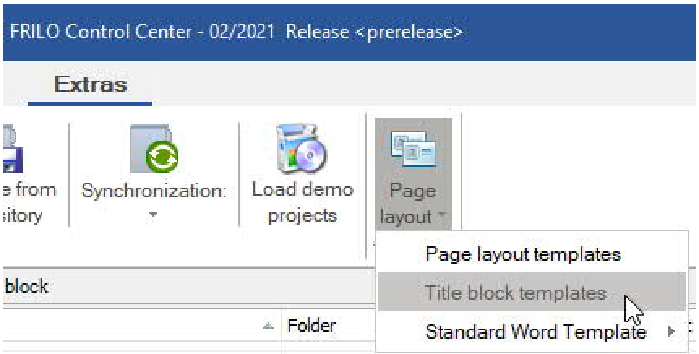
Selection of a plan format (A4 – A0, userdefined).

To display/print the graphics of a chosen plan format click the “Plans” tab in the print preview. When using the Frilo.Document.Designer these plans are appended to the end of the static document.

Note: *The register "plans" is only displayed when at least one image is selected (for printing) in the output profile and the option "In Plan Format" is set.*

Note: *The printer should be able to handle the chosen large-format.*

You can define special headers for the planformat (Frilo Control Center FCC ▸ Extras ▸ [Page Layout](#) – Title block template.



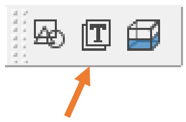
Example: define a Title block

A detailed example of defining a title block can be found in the document [PLT - Define Title Block](#).

Profile...	This option allows you to set up an output profile. You can select which data/graphs should be put out. See the chapter [Output profile](#).
Preview	You can check the pages on the screen prior to printing.
Word	You can export your results into word files (MS Word must be installed on this computer).

Output on the screen

Click on the Text-icon to display data (system data, results) in a text window in the form of tables.



Note: *the font size can be selected via the "Font size" tab of the output profile.*

Printing of the displayed graph (exclusively)

The functions "Zoom" or "Full screen" define a section of the graphic that is to be printed.

Activate the "Print" icon () on the tool bar or the menu item File ▸ Print... to print your selected graph.

Note: *The font size on the screen corresponds to the printed font size.*

Tip: *You can copy the selected graph to the clipboard using the shortcut "Ctrl + C" and paste it into any document.*

Deflections

The deflection of the plate will be calculatetd under the assumption of state 1 and the cracked state II.

To calculate the deflection in cracked state II the durability must be switched on in the dialog base parameters.

Individual load cases and superposition

Partial safety coefficients:

- Superposition results are put out γ-fold according to the combination rules of EN 1990. Bearing reactions, deflections and base compressions of foundation slabs can also be displayed as simple (characteristic) values in addition.
- The results of the individual load cases are are put out as simple (characteristic) values (with tension spring exclusion, however γ-fold).
- For all design results, the design values are used of course.

MIN/MAX superposition of bearing forces

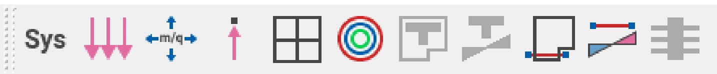
In a so-called MIN/MAX superposition, the greatest absolute positive value and the greatest absolute negative value are calculated.

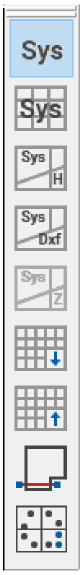
With respect to the bearing forces, the MIN value corresponds to the greatest lifting force. If there is no lifting force, the MIN value consists merely of the positive permanent action value. The permanent action must always be included as a positive value in the MIN value.

According to the relevant standard, you may assume a γ - value of 1.00 instead of 1.35 for a favourable permanent action. The permanent load is considered to be favourable because it counteracts lifting forces and therefore a γ - value = 1.00 may be assigned to it.

Make sure in this case that you do not mix up the gamma-fold forces with the simple ones.

Display on the screen





The icons shown above allow you to display the system, the loads and the results on the screen. If you select the display of results and you have not performed a calculation yet, the application prompts you whether to start the calculation now.

Attention: *When launching the calculation via this icon it refers only to the currently active load case. In order to calculate several load cases or a superposition, you must select the load cases or superposition to be calculated via the menu item "Calculation/superposition ".*

System. Various views, selection with the icon bar on the right side – see tooltips.

See also: Auxiliary structures, DXF-Import, Constrained elements, Bending-Preset reinforcement, Result sections .

Show loads

This option allows you to view the defined loads of the currently active load case.

Attention: Inactive load cases for which the option "display" was ticked in the load input table are also shown.

You can view point-, line-, area-, or temperature loads separately (see load toolbar).

Principal internal reactions (toolbar M / Q):
- moments
- shear forces

These options are available only for individual load cases.

Node results. The toolbar for the representation of the node results is displayed (deformations, shifts, superposition values, base compressions, bearing forces...)
→ see the paragraph "Node results" below.

Results in the output grid. The toolbar for the representation of the results in the selected output grid is displayed (deformations, shifts, superposition values, base compressions, bearing forces...)
→ see the following paragraph "Results in the output grid" and the chapter Results menu.

Results as isolines. The toolbar for the representation of the results of individual load cases in the form of isolines is displayed (moments, shear forces, deformations, reinforcement, $V_{Ed}/V_{Rd\text{-}ct}$) → see the paragraph "Isolines" below.

Upstand/downstand beam results (floorplan)

Upstand/downstand beam results in a separate graphic window.

Section results (floorplan)

Section results in a separate graphic window.

Show thermal insulation element results.

Node results

The following node results can be displayed via these icons:

- Show deformations.
  This options allows you to view the node shifts in the form of a deformation picture. It is only available for individual load cases.
- Deformations state II.
  Node shifts.
- Deflections
  Node shifts numerically.
- Deflections state II.
  Only for quasi permanent combinations.
- Soil pressure.
  This option allows you to view the base compressions with bedded slabs. To be able to do this, you must have previously defined a base slab with elastic bedding (Basic parameters menu).
- Support reactions.
  This options allows you to view the vertical support reactions.
- Show fixed-end moments around the local x / y - axis.
  This option allows you to view fixed-end moments parallel / vertical to the wall axis (with walls) or around the local x / y - axis (with columns). To be able to do this, you must have previously defined a restraint or spring rigidity in the corresponding direction in the bearing conditions section.
- Show vertical bearing forces as curves.
- Show vertical bearing forces as rectangular curves.
- Show vertical bearing forces as nodes.
- Show the position of the resultant of the wall support forces (as a point).

Tip: Hover your mouse pointer over an icon to see the explanatory tooltip.

Results in the output grid

- Moments. View the moments m_x, m_y and m_{xy} (torsional moment) of the slab.
- Shear forces.View the shear forces q_{xz} and q_{yz} of the slab.

Note: Hereafter in a non-rotated system of coordinates, the labeled direction “1” corresponds to the x-direction and direction “2” to the y-direction.

- Design moments for the upper/lower reinforcement (m_{B-1} and m_{B-2}).
- Lower/upper reinforcement (a_{s-1} and a_{s-2})
- Shear reinforcement:
  This option allows you to display the basic value of the shear stress in the top line. If a shear reinforcement is required, the line in the middle displays the strut inclination angle and the bottom line the required shear reinforcement. An asterisk (*) is displayed in areas where a shear proof cannot be produced. This means that no design can take place at this point. This is usually caused by input errors in the geometry or the FE mesh setting. These points should be urgently checked and reworked. A shear verification must be carried out at all necessary points required by structural engineering.
- VEd / VRd
- VEd
- Resistance VRd,ct
- Resistance VRd,max
- Lower/upper crack width w_{1} / w_2
- Lower/upper limit bar diameters
- Reinforcement: select total/difference.
  Click on this button to access the result grid dialog.
  Attention: If the application cannot find any points with action- or design-effects for a particular grid cell, this cell is not shown. This provides for an easier distinction between cells in which the design orientation does not correspond to the grid orientation ("-") and those for which no results are available.
  Missing cells do mainly occur when you have defined the action-effects in the element centres, for instance (there are considerably fewer element centres than element edge mid-points), or the average element size is too large compared to the cell size.
- Show design points.

Tip: Hover your mouse pointer over an icon to see the explanatory tooltip.

Isolines

This icons allow you to display the results in the form of isolines. Have a look on the tooltips.

PLT output profile

Output tab >>Settings >>Profile ...

See also [Output & Results]

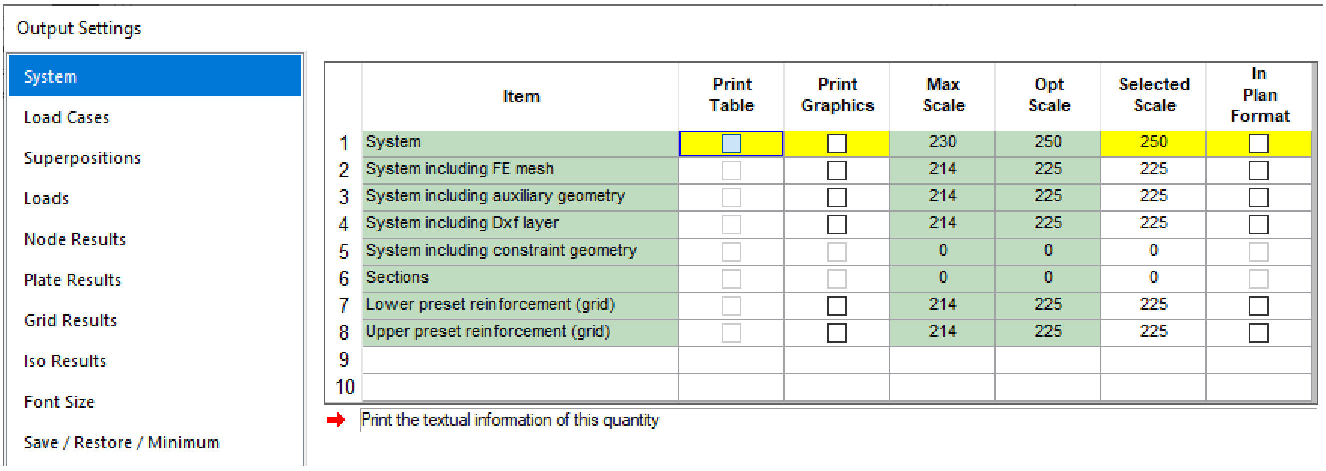
Output profile

Here you can specify the scope of the output in detail, e.g. for

System — This tab allows you to select whether the system data should be put out in the form of a table (Print text option) and/or a graph (Print graph option) and whether the output should be handled via the standard or the graphic printer.

If you specify in the column "Selected scale" a scale that is greater than the maximum scale and the graphic printer is not accessible, the data are automatically distributed over several pages and put out on the standard printer.

Attention: The displayed scales always refer to the standard printer. If you want to put out your data on the graphic printer you should perform a test print to check the selected scale.



	Item	Print Table	Print Graphics	Max Scale	Opt Scale	Selected Scale	In Plan Format
1	System	☐	☐	230	250	250	☐
2	System including FE mesh	☐	☐	214	225	225	☐
3	System including auxiliary geometry	☐	☐	214	225	225	☐
4	System including Dxf layer	☐	☐	214	225	225	☐
5	System including constraint geometry	☐	☐	0	0	0	☐
6	Sections	☐	☐	0	0	0	☐
7	Lower preset reinforcement (grid)	☐	☐	214	225	225	☐
8	Upper preset reinforcement (grid)	☐	☐	214	225	225	☐
9							
10							

Load cases — This tab allows you to select the load cases including the results that should be printed.

Results — The different result tabs allow you to select among various load and result representations for the output.

Node, slab, grid or iso results.

Upst./downst. beams — This tab allows you to select the upstand/downstand beams and the effects that should be put out.

Font size — This tab allows you to select the font size in the printed graph.

Save as — You can save the output profile also as a standard template for new items. This output profile is used if no item-specific output profile is available (normally with new items.) This option provides for the availability of default settings when defining new items.

Output options that are greyed out are disabled (because no output data are available).

Save your setting by clicking OK.

Design check in FRILO application

The "Design check in FRILO" menu in the main tree provides access to the applications for the design of individual components or the production of individual proofs, if these applications are included in your software package.

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

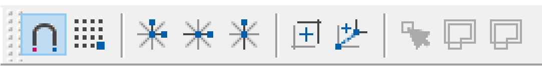
Punching shear B6+	Select a column that should be calculated in the B6+ application.
Continuous beam DLT/DLT+	Select continuous beams per mouse click. Finish your selection with a right click and "Exit" (the application detects automatically when all continuous objects have been selected). An intermediate dialog allows you to check the selected objects and the load cases. Confirm your selection with OK to launch the continuous beam application. Indirect support (with downstand beams and bearing parapets) is considered. Instead of the beam, a bearing with a minimum torsional rigidity is defined in vertical direction at the point of indirect support.
Multispan steel beam STM+	A steel beam can be designed with STM+.
Pile foundation Pile+	Bearing forces on columns can be transferred to Pfahl+ on a load case basis. Click on the desired column.

Application-specific icons

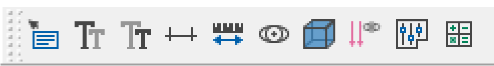
Icons of the load input section

Icons for the various input modes

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

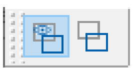


View toolbar



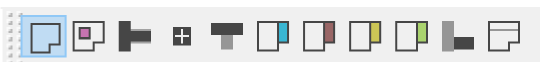
Icons for the display of results and output options

Icons for auxiliary slides



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

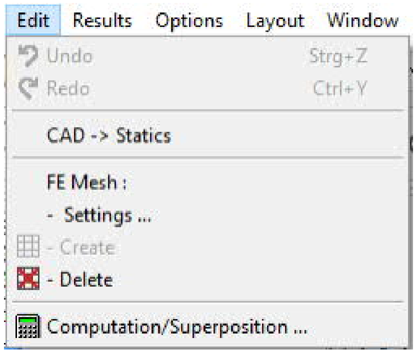
Icons of the Graphical input module



Can be viewed if necessary (off by default).

Additional menus in PLT

Edit menu



CAD → Statics

This option allows you to convert a CAD model into a structural one via geometric alignment (e.g. matching of the slab outline to the wall axis) after the import of the CAD data ▸ File ▸ Import.

FE Mesh: Settings

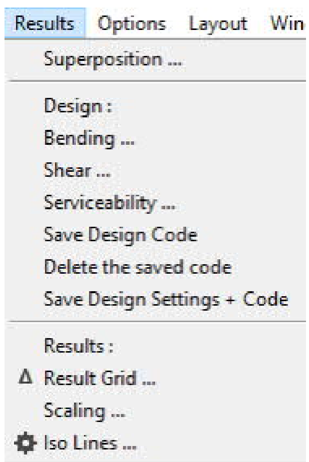
Create

Delete

See the chapter FE-mesh

Camputation/Superposition See the chapter Calculation/superposition...

Results menu



Superposition In the standardized design situations in the ultimate limit state, it is specified which types of action are to be taken into account in which situation.
See chapter Dimensioning: Settings.

Design

Bending... Options for displaying the minimum reinforcement to ensure ductile component behavior.

Shear... See chapter Design Settings shear ....

Serviceability... See chapter Design Settings Serviceability...

Save Design... You can save the selected standard and the settings as default for new items.

Results

Result Grid... This menu item allows you to define a grid for the results and select options for the output of the reinforcement.

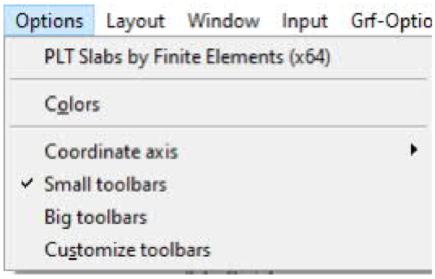
See also the chapter Results: Settings.

Scaling This menu item allows you to define scaling factors for the representation of the deformation results and/or the wall results (for printing and/or representation on the screen). See also the chapter Results: Settings.

Iso Lines... Configuration of the display of Iso Lines.

See also Results & output options.

Options menu



Settings - Slabs by Finite Elements

Various settings, e.g.:

[Construction mode](#),
[Auto data save](#),
[Interactive input](#) (background grid, axes of coordinates),

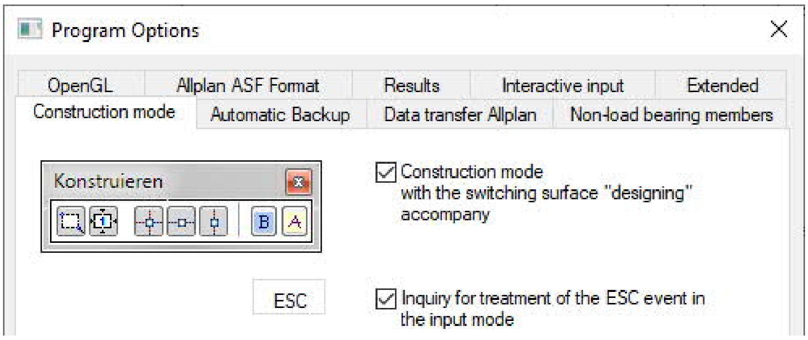
Data transfer from Allplan:

You can import data of partial drawings from Nemetschek CAD directly into the graph (via the shortcut CTRL-T). In order to display an additional dialog with a list of slides, tick the corresponding option.

Allplan ASF-files

You can set export options (ALLPLAN version) for the output file to be imported from Allplan.

Results



With this option, the results of a load case superposition can be restored after opening an item, if the results of the individual load cases are available.

Non loadbearing members

The graphic representation - fill color, contour color, line type - of non-load-bearing components can be set separately for walls, columns and parapets.

Colors

See document [menu items](#) ▸ Color settings

Coordinate axis

Representation options for the axes of coordinates.

Extended ▸ Customize Toolbars

This rarely used function is normaly hidden from the options menu by default but can be displayed by ticking this option. This allows the symbols and toolbars to be configured individually.

Input menu

The functions of this menu are also accessible via the icons of the toolbar
→ see [Graphical input](#).

Graphic options menu

Hide/display auxiliary slides: The menu item allows you to display or hide an imported auxiliary slide (from DXF...).

Tools menu

See chapter “Additional menu items” in the manual “[Graphical input](#)”.

Graphical input

The functions of the application-integrated Graphical input module are described in the document "Graphical input.pdf".

Important note:
The application module “Graphical input” is used in various applications (PLT, Building, WL, SCN). We describe all Graphical input functions in the document Graphical input.pdf, 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 in graphic mode a floor plan (outline, recesses), walls, columns (bearings), upstand and downstand beams, parapets, thickness, bedding, reinforcement and supporting direction areas as well as loads.

Three-dimensional construction graph

Access via the icon .

The three-dimensional construction graph shows a rendered representation of the system that provides for excellent visual control.

The system is shown in a perspective projection seen from a virtual camera position.

You can rotate the system using the arrow keys or the mouse while keeping the mouse button pressed. Please note that you merely change the camera position not the system when you move or zoom the representation.

You can also launch animations such as rotation or camera flight.

Further information: → see description in the document 3D-Construction Graphics