

Walls by Finite Elements SCN

Contents

Application options	3
Basis of calculation	5
Input	6
Graphical input	6
Numeric input	6
DXF import	6
System and load input	6
Result sections	7
FE-mesh	8
Properties	8
Generation	8
Delete	8
Computation / Superposition...	9
Design: Settings	10
Slab design	10
Crack widths ...	11
Superposition	12
Results: Superpositions	13
Results: Settings	13
Grid	13
Scaling	14
Iso lines	14
Output & results	15
SCN output profile	19
Application-specific icons	21
Additional menus in SCN	22
Edit menu	22
Results menu	22
Options menu	23
Input menu	23
Graphic options menu	23
Tools menu	23
Graphical input	24
Three-dimensional construction graph	24
Literatur	24

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 in the Campus-download-section.

Note: This manual deals with the basic features of the SCN application.

The input is primarily made via the graphical input, which is described in the document:

[Graphical input.pdf](#) Graphical input for programs GEO, PLT, SCN and WL

Tip: Go back - e.g. after a link to another chapter / document - in the PDF with the key combination "ALT" + "left arrow key".

Application options

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

The [graphical input module](#) offers numerous new functions and options that provide for a quick and comfortable system generation and give a detailed system overview at the same time.

This application is particularly suitable for the structural calculation of walls and wall-type girders and their design in accordance with the BAUMANN method ([/4/](#), [/6/](#)).

Load-bearing structures are substantially considered as panels if the following relations exist between the typical system dimensions system length L and cross sectional height H:

- Cantilever: $L / H < 1$,
- Single-span girder: $L / H < 2$,
- Multi-span girder: $L / H < 2.5$.

The panel is a plane load-bearing structure to which the external forces apply in its plane and generate an evenly distributed membrane state of stress. Effects of actions perpendicular to this plane are excluded (flexural effects → slab).

As long as the states of equilibrium in the plane of the structure and those perpendicular to it may be considered in undeformed condition, the two bearing effects can be examined separately from each other. Problems of stability and bracing are not treated.

The restricting assumptions of the beam theory such as the Euler-Bernoulli hypothesis, the equality of the cross sectional shifts of all points in the same cross sectional plane and the disregard of the orthogonal axial stress σ_y perpendicular to the system axis can be dispensed with in the panel theory. According to the panel theory, shear stress does not only result from equilibrium considerations as in the beam theory but also from a fully valid stress/distortion relationship.

Examples for the application of panels are wall bracings in high-rise buildings, wall-type girder systems and corbels. In the current version, you cannot couple panels and single bars, which means that you must discretise bars as panel elements (drawback: great shear deformations).

Standards

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

still available:

- DIN 1045 / DIN 1045-1 (2001 + 2008)
- ÖNorm B 4700



Interfaces to CAD systems

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

Formwork drawings from CAD systems like 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.
- Sheet stresses are not available.

Basis of calculation

The calculation is performed in accordance with the finite elements method and based on the theory of elasticity. The design of the reinforced concrete panel is performed in accordance with the BAUMANN method for an orthogonal reinforcement mesh. The structural calculation of a reinforced concrete panel is based on the uncracked state I. For the elasticity values those of the concrete are assumed.

Approximation methods such as finite elements satisfy equilibrium conditions related to the finite element by calculating the arithmetic average of all stress states. Therefore, stress concentrations in the individual elements can only be represented incompletely with the usual methods of discretisation. A local refinement of the mesh would improve the results in these areas under normal conditions. In view of the discontinuous distribution of the reinforcement, this is however not imperative. In addition to this, the refinement of the mesh involves the risk of unfavourable element proportions that falsify the results for numerical reasons. It is the task of the engineer to find a compromise and evaluate the individual results.

The calculation method alone does not reveal how far the results are from reality. The only way to find this out is to estimate it by finer discretisation. We recommend you to select at least 10 elements between two points of discontinuity (bearings).

In the current version, the application uses two-dimensional squared elements. These elements are very efficient in regard to the computing time. The accuracy of the results is very high due to the higher-order shift approach for $u(x,y)$ and $v(x,y)$. Considerable variations occur in the area of discontinuities, in combination with local stress variations in the proximity of bearings, for instance. Comparative calculations show that the deformation results of this type of element deviate from the more accurate results by 10 % maximum if discretisation is sufficiently fine.

With a finer subdivision, this element produces even more accurate results. The element's lengths relations should however not exceed the ratio of 2 to 1.

Mesh generator

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

First, nodes are generated along the default lines. After this, squared elements are successively generated at several active fronts. During the generation of the elements, the quality of each newly generated element is examined and optimised.

Design

The design of the reinforcement is performed in accordance with the Baumann method. A cracked panel 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 become a minimum. The design approach assumes an orthogonal mesh reinforcement in the first place.

Input

Graphical input

The SCN application offers a graphical user interface, i.e. elements such as the plate 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 SCN 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.

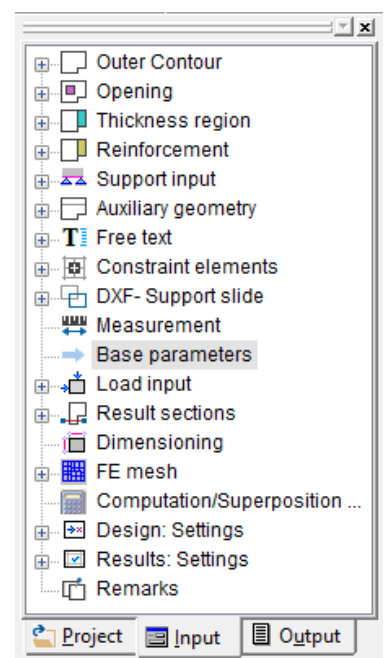
System and load input

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](#).

Material, standard selection, panel thickness, concrete coverage and details concerning the serviceability, if required.

Various drawing functions are available for the definition of an outline and block-outs 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 operation 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).

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.

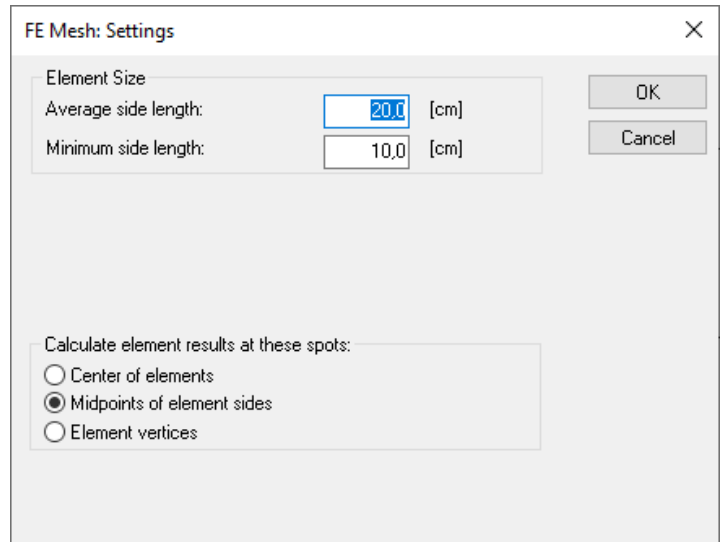
Element results... This option allows you to select the points at which element results should be calculated.

You can select among the following design points:


- centre points of the elements
- mid points of the element sides (default)
- element vertices (corner points of the elements)

Note: The option "element vertices" can produce unfavourable results in so-called points of singularity.

Note: In the current version of the application, the FE mesh also includes the position of all applying loads in the calculation. Therefore, the FE mesh which is generated without loads for verification purposes may differ from the FE mesh that is used for the calculation.



Generation

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.

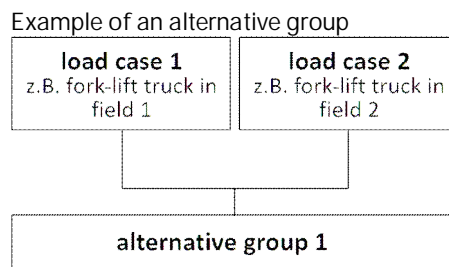
Delete

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

Computation / Superposition...

Load Cases according to DIN EN 1992-1-1/NA:2015-12								
	Name	Com- pute	Com- puted	Include Self- Weight	Partial Safety	Action	Super- pose	Alter- native- group
1	Lastfall G (Ge:0)	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	1,35	g	<input checked="" type="checkbox"/>	0
2	Lastfall Q (Ge:0)	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	1,50	1	<input checked="" type="checkbox"/>	0
3	Lastfall GU (Ge:0)	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	1,35	g	<input checked="" type="checkbox"/>	0

- 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. Self Weight** Tick this option if the deadweight should be considered in the calculation.
- Partial Safety** The partial safety factors are displayed here depending on the selected type of action (permanent or non-permanent).
- Action** Here you select the desired type of action.
- 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).



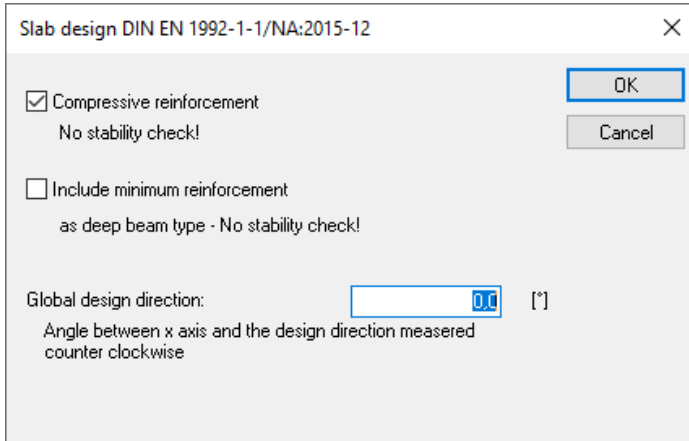
The load cases 1 and 2 are assigned to alternative group 1 because the fork lift is either in field 1 or in field 2.

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.

Slab design



Compressive reinforcement

Tick this option to consider possibly required compressive reinforcement when the compressive strength of the concrete is exceeded. This reinforcement might be required in load introduction areas or smaller bearing areas.

The compressive reinforcement is marked with an exclamation mark (!) in the output grid.

Minimum

Tick this option to include minimum reinforcement for wall-type reinforcement girders.

Note: This option does not refer to a minimum reinforcement ensuring stability. You must perform the stability analysis separately if required.

Design direction

The horizontal and vertical directions are set as default global design directions. You can however freely select the global design directions according to your requirements.

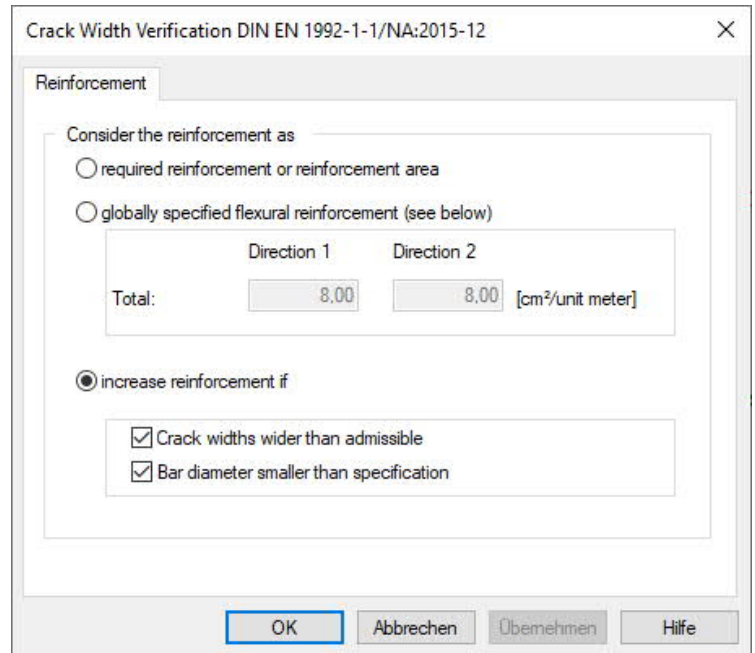
Crack widths ...

DIN 1045-1 : 2004/2008

In accordance with DIN 1045-1, 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).

For the determination of the percentage of reinforcement you can either select the reinforcement required by the bending design or a default reinforcement to be specified. If the option "default flexural tension reinforcement" is ticked, always the higher reinforcement value of the statically required and the default reinforcement is used.

In addition, you can increase the flexural tension reinforcement until the crack width proof criteria are complied with via the option "Increase flexural tension reinforcement".



Note: Ticking the option "Increase reinforcement" may produce different results on the front and rear side of the panel in particular cases if the durability requirements to be satisfied differ for each side. In these cases, a separate output of the results for the front and rear side is available.

EN 1992 1-1 (NA D/AT/GB)

The options and settings described for DIN 1045-1 apply analogously to this standard.

Austrian standard B4700

This option is not enabled for this standard.

EC2 Italy

The options and settings described for DIN 1045-1 apply analogously to this standard.

DIN 1045 superseded

This option is not enabled for this standard.

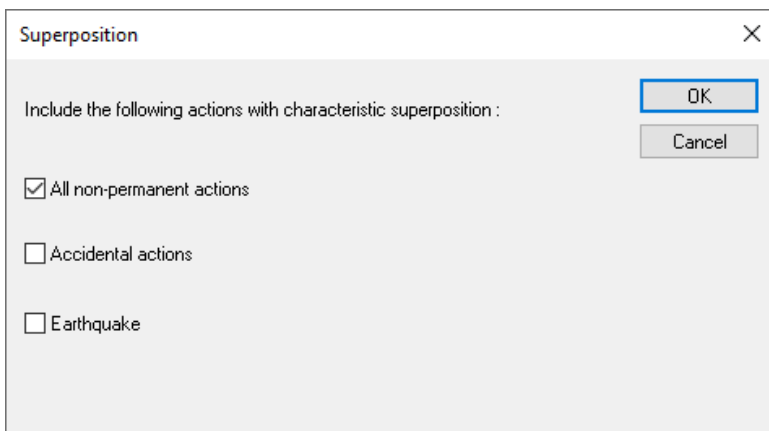
Superposition

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



Superposition

Include the following actions with characteristic superposition :

All non-permanent actions

Accidental actions

Earthquake

OK

Cancel

Results: Superpositions

Superpositions

All results of each required design situation can be displayed and put out.

If the nature of the effects of actions requires you to examine more than one design situation, such as the "permanent/transient" and the "accidental" situations in the ultimate limit state (ULS) and the "quasi-permanent" situation in the serviceability limit state (SLT), the application generates a decisive superposition of all design situations involved.

In addition to the design situation prescribed by the standards a so-called characteristic superposition is performed. It consists in a simple superposition without partial safety and combination coefficients.

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

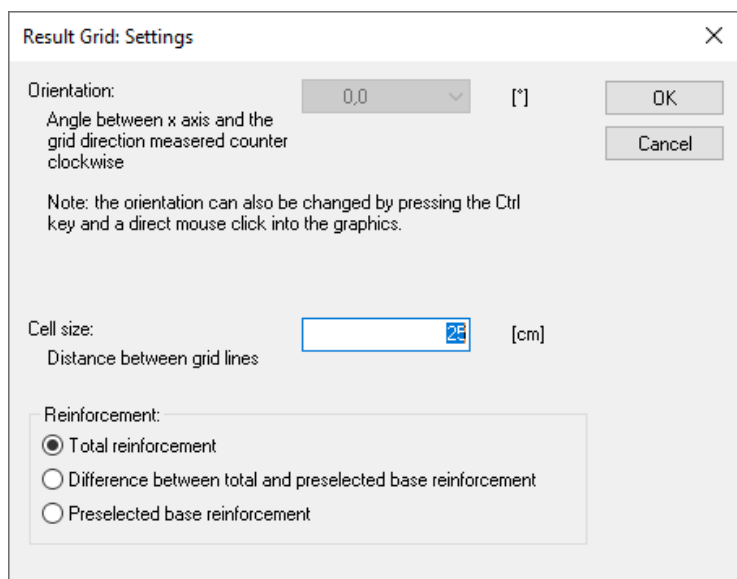
You can define the spacing of grid lines in this section.

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 block-out, for instance, when small block-outs 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 block-out, i.e. the reinforcement put out in this section must be inserted at the edge of the block-out, for instance. In some cases, you can improve the representation by modifying the size of the result grid.



Result Grid: Settings

Orientation: 0,0 [°] OK Cancel

Angle between x axis and the grid direction measured counter clockwise

Note: the orientation can also be changed by pressing the Ctrl key and a direct mouse click into the graphics.

Cell size: [] [cm]

Distance between grid lines

Reinforcement:

- Total reinforcement
- Difference between total and preselected base reinforcement
- Preselected base reinforcement

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

Iso Lines X

Iso Results

	Item	Value Min	Value Max	Limit Min	Limit Max	Number Lines (max 10)	Show Legend	Show Values
1	Node displacements in X direction	0,00	0,05	0,00	0,05	10	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
2	Node displacements in Y direction	-0,00	0,08	0,01	0,08	10	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
3	Internal force n-11	-285,77	116,72	-249,18	80,13	10	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
4	Internal force n-22	-1169,26	328,99	-1033,05	192,79	10	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
5	Internal force n-12	-263,36	135,32	-227,11	99,08	10	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
6	Design force nB-1	-358,24	210,11	-306,57	158,44	10	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
7	Design force nB-2	-1234,04	332,60	-1091,62	190,18	10	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
8	Reinforcement aS-1, Total	0,00	6,21	0,56	5,65	10	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
9	Reinforcement aS-1, on back	0,00	3,11	0,28	2,82	10	<input type="checkbox"/>	<input type="checkbox"/>
10	Reinforcement aS-1, on front	0,00	3,11	0,28	2,82	10	<input type="checkbox"/>	<input type="checkbox"/>
11	Reinforcement aS-2, Total	0,00	9,84	0,89	8,94	10	<input type="checkbox"/>	<input checked="" type="checkbox"/>
12	Reinforcement aS-2, on back	0,00	4,92	0,45	4,47	10	<input type="checkbox"/>	<input type="checkbox"/>
13	Reinforcement aS-2, on front	0,00	4,92	0,45	4,47	10	<input type="checkbox"/>	<input type="checkbox"/>

➔ Show this quantity with lower limit

Reset all values to initial state

After the computation the setting options are accessible via

1. the menu bar ▶ Results ▶ Contour lines
2. the left menu ▶ Results: Settings ▶ Contour lines

The results can be displayed as isolines/contour lines using 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.

Output & results

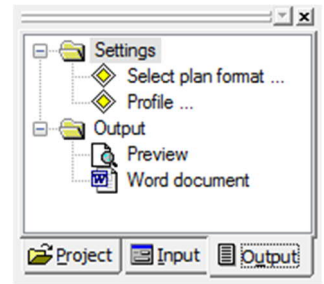
The "Output" tab offers the following settings and output options:

Select a plan format...

Plan format: A0 – A3

To display/print the graphics of a chosen plan format click the "Plans" tab in the print preview.

Note: When using the [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.

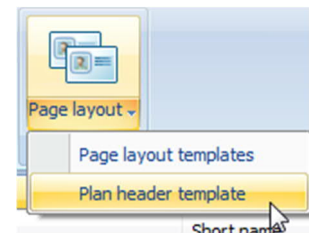
Note: In this case, the entire document should be printed on a printer that can print both formats/paper sizes, otherwise the plan format may be printed cropped. You can print the pages with the plan formats separately on a suitable printer.

You can define special headers for the planformat (FRILO Control Center FCC ▶ Extras ▶ Page Layout – Plan header template).

Define title block/plan header example

A detailed example of how to define a plan header can be found in the document

[Defining the title block for the program PLT.](#)

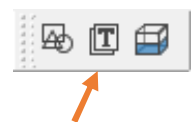


- | | |
|------------|--|
| Profile... | This option allows you to set up an output profile. You can select which data/graphs should be put out
- see the chapter Output profile . |
| Preview | You can check the pages on the screen prior to printing. |
| Word | You can export your results into word files (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: In this window, you can define the font sizes for printing separately (select the font size via the "Font size" tab of the output profile).



Printing of the displayed graph (exclusively)

Click on the "Graph" tab. Define the graph or the section of the graph to be printed via the functions "Zoom" or "Full screen". 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.

Individual load cases and superpositions

Partial safety coefficients:

- Superposition results are put out γ -fold in accordance with the combination rules and the applicable design situation. Bearing reactions and deformations can be displayed as characteristic values in addition.
- The results of the individual load cases are put out as characteristic values (in the case of tension spring exclusion, however γ -fold).
- For all design results, the design values of the action-effects are used of course.

Design situation

All superposition results of each required design situation can be displayed and put out separately.

In addition to this, a characteristic superposition without partial safety and combination coefficients is always performed for the action-effects, bearing reactions and deformations. The characteristic superposition does not include any design results. Via the main menu item "Design: Options ▶ Superposition...", you can select the effects (non-permanent, accidental or earthquake) that should be considered for the characteristic superposition.

If the nature of the effects of actions requires you to examine more than one design situation, such as the "permanent/transient" and the "accidental" situations in the ultimate limit state (ULS) and the "quasi-permanent" situation in the serviceability limit state (SLT), the application generates a decisive superposition of all design situations involved.

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 gamma 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 gamma 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](#)".

Display system data



Selection of different system displays. A toolbar with various display options is displayed. The respective tool tips inform about the function of the individual symbols.

See also: [Auxiliary construction](#), [DXF import](#), [constrained geometry](#), [sections](#).

Display loads



Display 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.



Display point loads only



Display line loads only



Display all loads

Display main action-effects



Display main axial forces.



Via the toolbar that appears you can

- display the course of the main moments.
- Display the course of the main transverse forces.

It is only available for individual load cases.



Display node results. The tool bar for the representation of the node results is displayed (Deformed system, Shifts, Bearing reactions) → see the paragraph "Node results" below.



Display results in the output grid. The tool bar for the representation of the results in the selected output grid is displayed → see the paragraph "Results in the output grid" below and the chapter [Results menu](#).



Display results as isolines. The tool bar for the representation of the results of individual load cases in the form of isolines is displayed (axial forces, design forces, reinforcement) → see the paragraph "Isolines" below.



Display section results (in the ground plan). This option allows you to display action-effects and design results along the previously defined sections.



Display [section results](#) graphically in a separate graphic window. This option allows you to display action-effects and design results along the previously defined sections.

Node results

The following node results can be displayed via these icons:



Display the deformed system. This options allows you to display the node shifts in the form of a deformation picture. It is only available for individual load cases.



Display bearing forces in x-direction (horizontal bearing forces).



Display bearing forces in y-direction (vertical bearing forces).



Display bearing forces as curves.



Display bearing forces as rectangular curves.



Display bearing forces as nodes.

Results in the output grid

In an unrotated system of coordinates, direction 1 corresponds to the x-direction and direction 2 to the y-direction.



Display panel forces: n_{11} , n_{22} and n_{12} (shear force).



Display panel stresses: σ_{11} , σ_{22} and σ_{12} .



Display design forces for the reinforcement: $n_{B,1}$ and $n_{B,2}$.



Display concrete utilisation in per cent.



Display reinforcement: $a_{s,1}$ / $a_{s,2}$.



Display crack widths: $w_{k,1}$ / $w_{k,2}$.



Display limit diameter of the reinforcement: $d_{s,1}$ / $d_{s,2}$.



Reinforcement: select total/difference.

Click on this button to access the [result grid](#) dialog.

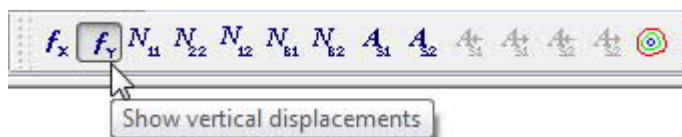
Attention: If the application cannot find any points with action- or design-effects for a particular grid cell, this cell is not shown. This provides for an easier distinction between cells in which the design orientation does not correspond to the grid orientation ("-") and those for which no results 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.



Display design points.

Isolines



These icons allow you to display the results in the form of isolines. For detailed info please have a look on the tooltips.

SCN output profile

Output tab >>Settings >>Profile ...

See also [Output & Results](#)

Output profile

Here you can define 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.

	Item	Print Text	Print Graph	Max Scale	Opt Scale	Selected Scale	In Plan Format
1	System	<input checked="" type="checkbox"/>	<input type="checkbox"/>	30	33	33	<input type="checkbox"/>
2	Show system including FE mesh	<input type="checkbox"/>	<input checked="" type="checkbox"/>	15	20	20	<input checked="" type="checkbox"/>
3	Show system including auxiliary geometry	<input type="checkbox"/>	<input type="checkbox"/>	26	33	33	<input type="checkbox"/>
4	Show system including Dxf layer	<input type="checkbox"/>	<input type="checkbox"/>	26	33	33	<input type="checkbox"/>
5	Show system including constraint geometr	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0	<input type="checkbox"/>
6	Result Sections	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0	<input type="checkbox"/>
7	Preset reinforcement, Total (Grid)	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0	<input type="checkbox"/>
8	Preset reinforcement, on back (Grid)	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0	<input type="checkbox"/>
9	Preset reinforcement, on front (Grid)	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0	<input type="checkbox"/>
10							

Load cases

This tab allows you to select the load cases the results of which you want to put out.

In the first line "Load case overviews" you can select whether an overview of action-effects, load data and specific result summaries (if available) should be put out together with each load case.

The column "Calculated" shows for each load case whether the calculation has been performed yet.

Superpositions

On this tab you can select the superpositions the results of which you want to put out. You can select the superposition in accordance with the required design situations and include the characteristic and also the decisive superposition if required.

The Superpositions tab is not available if no superposition results have been generated.

In the first line "Superposition overviews" you can select whether an overview of the loads and effects involved should be put out together with each superposition.

Loads

Output options for the individual load types.

...Results

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

Node, slab, grid or iso results.

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).

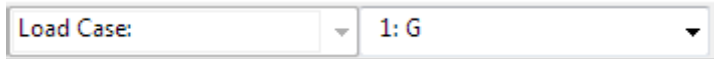
You can select the individual menu options by clicking on the corresponding tab.

Save your setting by clicking OK.

Tip: *A mouse click into the head of a tick options column either ticks or unticks all options at a time.
A mouse click into the head of the "Max scale" or "Opt scale" column transfers the maximum respectively the optimum scales to the selected ones.*

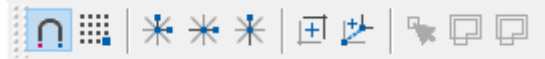
Application-specific icons

[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](#)



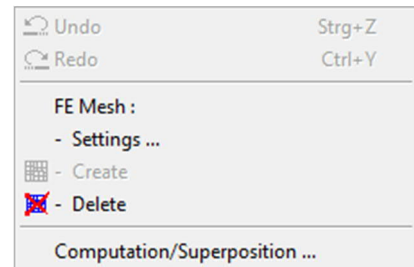
Can be displayed if required (switched off by default).

To do this, tick the Option - Settings - Extended - "Customise toolbar...", only then can the toolbars be switched on/off under the menu item Options).

Additional menus in SCN

Edit menu

FE mesh: Settings
 Create
 Delete
 See the chapter [FE-mesh](#)



Computation/Superposition See the chapter [Computation/Superposition...](#)

Results menu

Superposition... The standardised design situations in the ultimate limit state specify which types of action are to be taken into account in which situation. See chapter [Design: Settings](#).

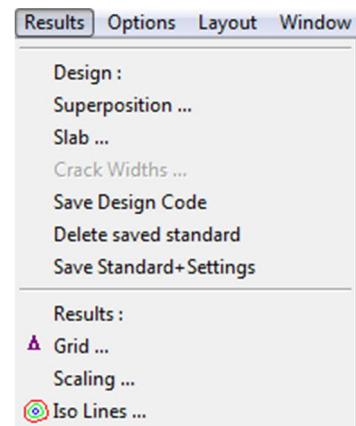
Slab... Options for reinforcement and design direction

Save Design Code Save the selected standard and/or settings as the default for new items.

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 - Walls 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

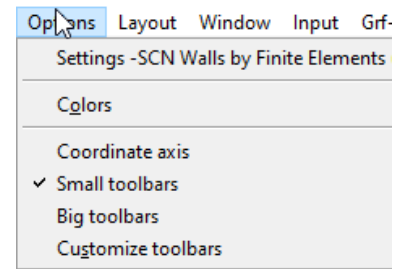
Select this option if you want to „always restore the results of a loadcase superposition when opening a project item, if the results of the individual loadcases are available“.

Non load bearing members

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

Extended ▶ Customize toolbars

This rarely used function is hidden from the options menu by default and can be shown here. This allows you to configure the toolbars and icons individually.



[Colors](#)

Setting colors for graphics.

Axes of coordinates

Representation options for the axes of coordinates.

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, block-outs), 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 info: see description in document [3D-construction-graphic.pdf](#)

Literatur

- /1/ R.H. GALLAGHER: *Finite-Element-Analysis*. Berlin, Heidelberg, New York (Springer) 1976. (S. 261 ff. über Rechteckelemente)
- /2/ H.R. SCHWARZ: *Methode der finiten Elemente*. Stuttgart (Teubner) 1984.
- /3/ DIN 1045 Beton und Stahlbeton, Bemessung und Ausführung.
- /4/ F. LEONHARDT: *Vorlesungen über Massivbau*. Teil II: Sonderfälle im Stahlbetonbau (F. Leonhardt und E. Mönning). Berlin, Heidelberg, New York (Springer) 1975.
- /5/ H. KUPFER, R. MANG: „Rißfeld und Druckfeld in Stahlbetonscheiben“, in: *Fortschritte im konstruktiven Ingenieurbau. Gallus Rehm zum 60. Geburtstag*. Berlin (Ernst & Sohn) 1984.
- /6/ TH. BAUMANN: „Zur Frage der Netzbewehrung in Flächentragwerken“, in: *Der Bauingenieur* 47.1972, Heft 10, S. 367 - 377.