Panels by Finite Elements SCN

FRILO Software GmbH

www.frilo.com

info@frilo.com

As of 03/08/2016
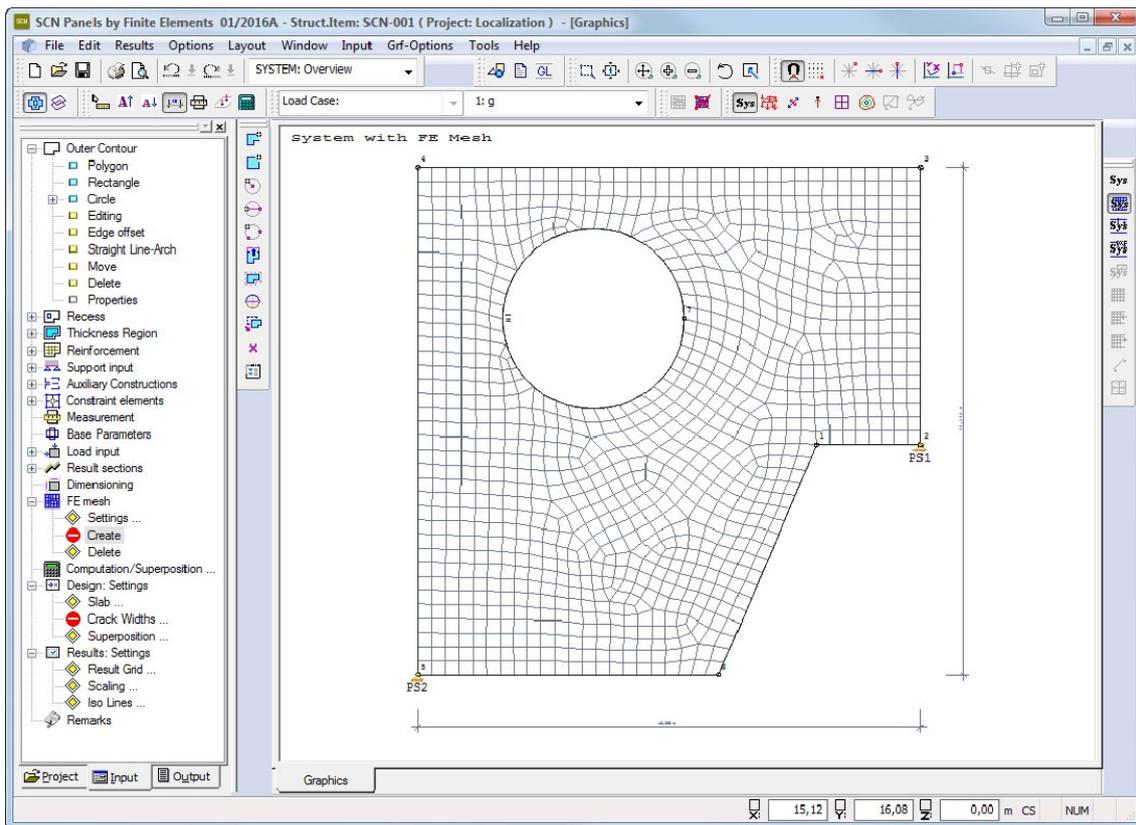
Panels by Finite Elements SCN

This manual deals with the basic features of the *SCN* application.

Contents

You will find the documentation of the graphical input in the following manual:

Graphical input.pdf   Graphical input for programs GEO, PLT, SC7, SCN and WL

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 $\rightarrow$ 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

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

still available:
- DIN 1045/1045-1, ÖNorm B4700

Standard
DIN EN 1992:2012
DIN 1045:1988
DIN 1045-1:2001
DIN 1045-1:2008
DIN EN 1992:2011
DIN EN 1992:2012
DIN EN 1992:2013
ÖNorm B 4700:2001
ÖNORM EN 1992:2007
ÖNORM EN 1992:2011
NTC EN 1992:2008
BS EN 1992:2004
BS EN 1992:2009
PN EN 1992:2010
EN 1992:2010

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.

- 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.

Note: *In the current version you can only use isotropic materials for the calculation.*

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.
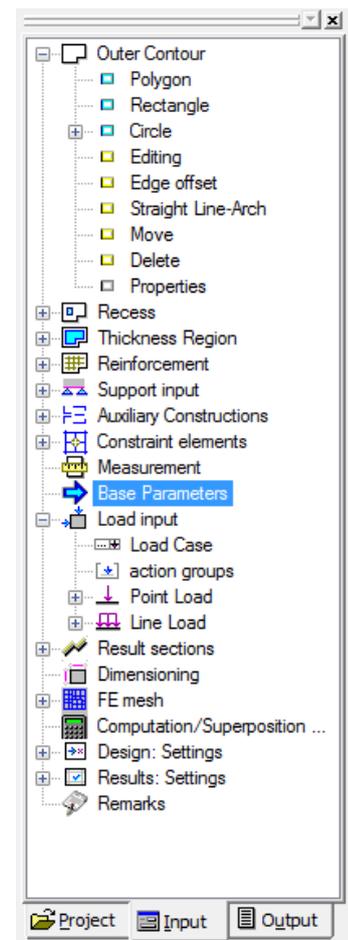
Result sections

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.

Enter a section as a polygon line. Define your polygonal section line with the help of the mouse or via the numeric input. Finish the operation per right click and select "Exit".

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. Finish the operation per right click and select "Exit".

Move a section. Click on the corresponding section and drag it to the desired target position with the help of the mouse . Finish the operation per right click and select "Exit".

Copy a section. Click on the corresponding section and drag the copy with the help of the mouse to the desired target position. Finish the operation per right click and select "Exit".
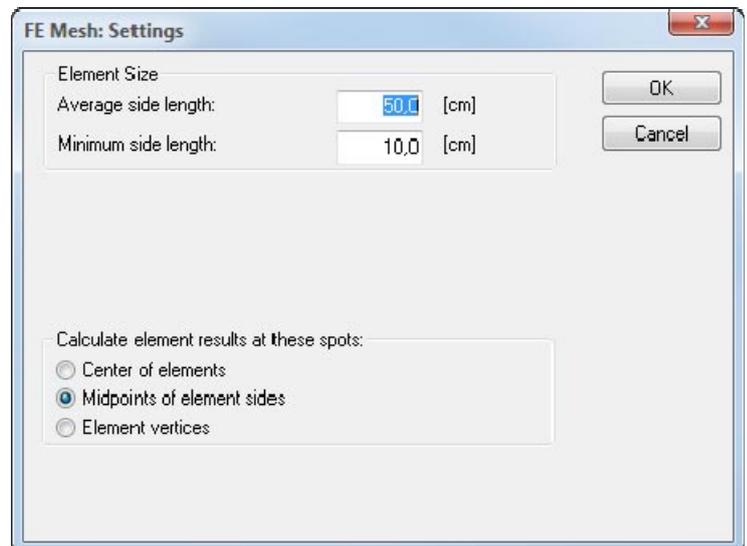
Delete a section or several sections (one after the other). Finish the deleting operation with a right click and select "Exit".

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 verices" 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...

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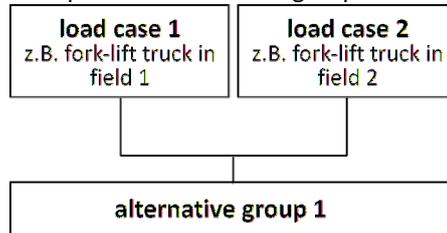
The load cases 1 and 2 are assigned to alternative group 1 because the fork lift is either in field 1 or in field 2.

EN 1992 1-1 (NA D/AT/GB), DIN 1045-1 : 2004/2008

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.
	Note: The current version does not include the non-linear calculation of panels. Therefore, the last column of the leading action is not enabled.

B4700 und EC2 Italien

The afore-mentioned statements apply analogously to the Austrian standard B4700 and the Italian EC2.

DIN 1045 7/88

|	
Permanent	Permanent load (e.g. selfweight)
Non-permanent	Non-permanent load (e.g. live load)
	The application searches the relevant load combination (e. g. g or g+p) for each design point.
	The application <u>cannot</u> consider a load per field automatically. If the live load should be assumed per field, a separate load case must be generated for each field (or a checkered distribution).
	If the live load is defined over the total surface, the field reinforcement must be increased by constructive measures in systems with multiple fields.
Factor	You can specify a multiplying factor for the respective load case in this column.

The calculation is launched when you confirm your settings in this window with OK.

Design: Settings

The section Design - Settings offers various options and settings depending on the selected standard.

Slab design
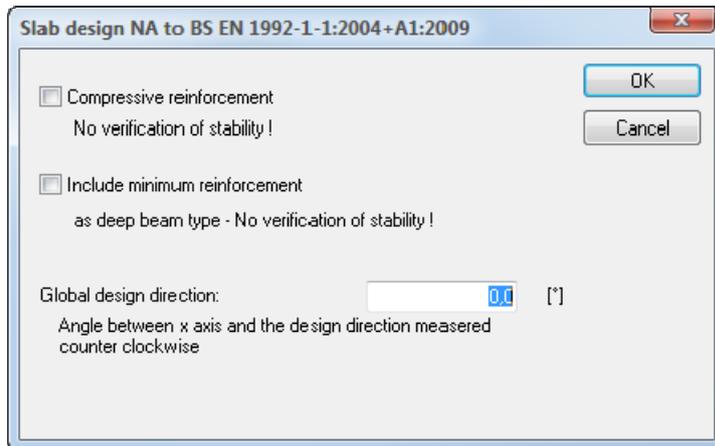
Compressive reinforcement	Tick this option to consider possibly required compressive reinforcement when the compressive strength of the concrete is exceeded. This reinforcement might be required in load introduction areas or smaller bearing areas.
	The compressive reinforcement is marked with an exclamation 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".
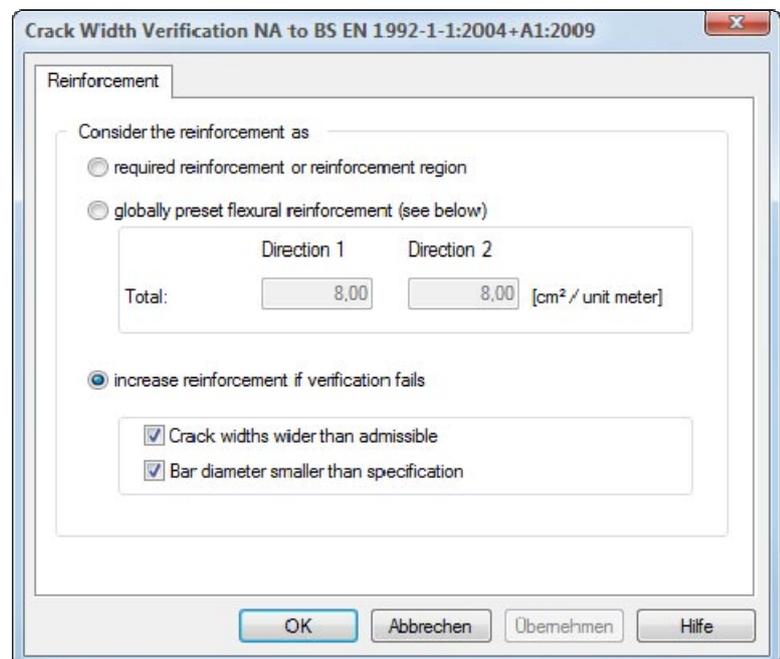
Note: Ticking the option "Increase reinforcement" may produce different results on the front and rear side of the panel in particular cases if the durability requirements to be satisfied differ for each side. In these cases, a separate output of the results for the front and rear side is available.

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
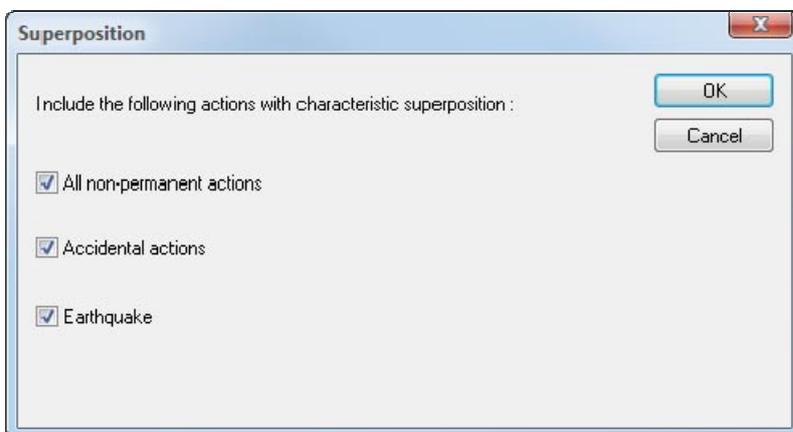
Results: Superpositions

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.

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).
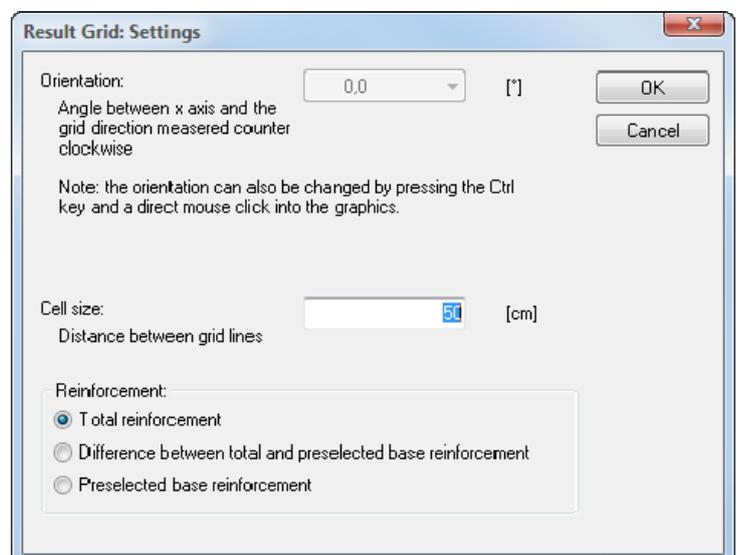
Iso lines

Various settings are available for the individual result parameters:

- lower and upper limits and

- number of subdivisions
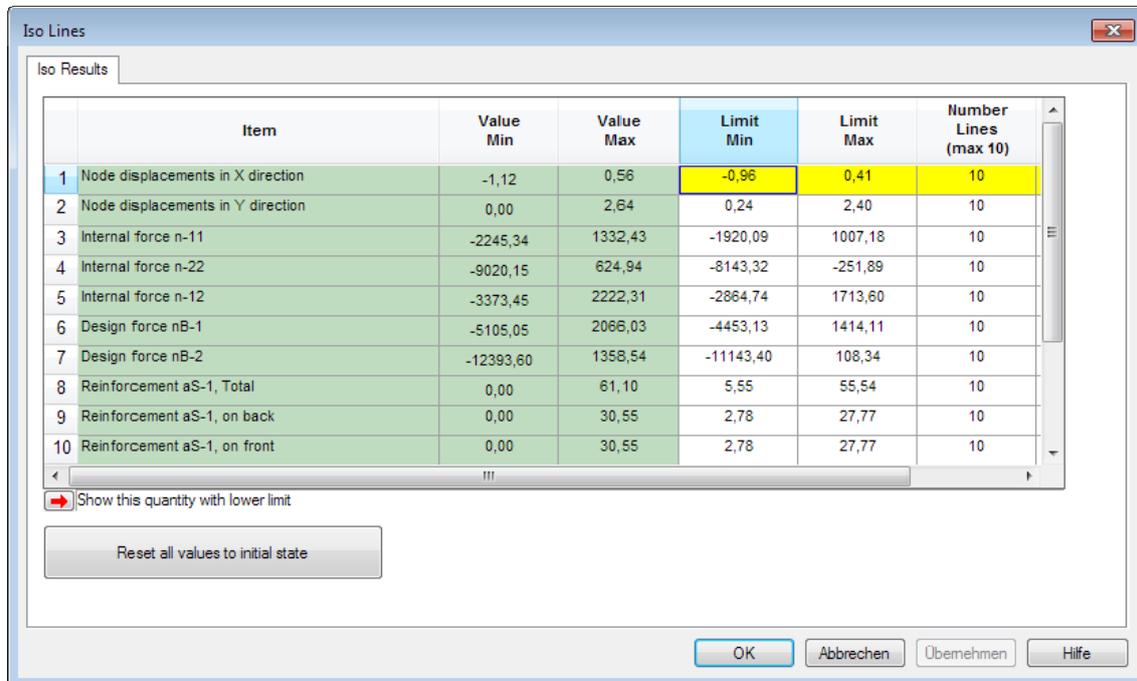
After the computation the setting options are accessible via

1. the menu bar ▸ Results ▸ Contour lines

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

3. the toolbar for the iso results.

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.*
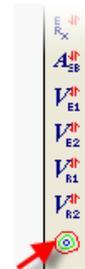
Output & results

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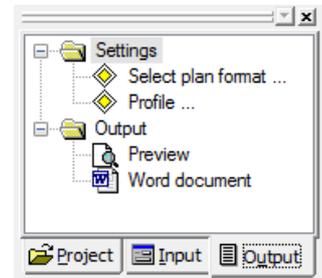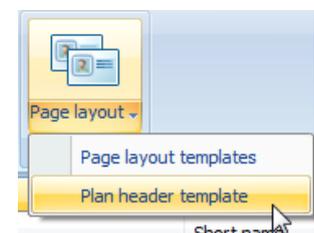
Profile...	This option allows you to set up an output profile. You can select which data/graphs should be put out - see the chapter Output profile.
Preview	You can check the pages on the screen prior to printing.
Word	You can export your results into word files (MS Word must be installed on this computer).

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

- Display system with dimensioning and ground plan
- Display default reinforcement

  Note: This function is only available after the calculation.

- Display sections This option allows you to display the location of the defined sections.

Display loads

This option allows you to 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

N_x Display main axial forces. This option allows you to display the behaviour of the main axial 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

f →▸ ⩫ �糸 ⩁ ⊔ ⁖⁖

f 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.

N σ N_B % A_s ✗ ✎ A_s A_s ✗ ✗ ✎ ✎ $\mathbf{\mathring{A}}$

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

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

N_B Display design forces for the reinforcement: n_{B-1} and n_{B-2} .

%   Display concrete utilisation in per cent.

A_s 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}.

$\mathbf{\mathring{A}}$ 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.

Isolines

f_x f_y N_{11} N_{22} N_{12} N_{B1} N_{B2} A_{s1} A_{s2} A_s A_s A_s A_s ◎

Show vertical displacements
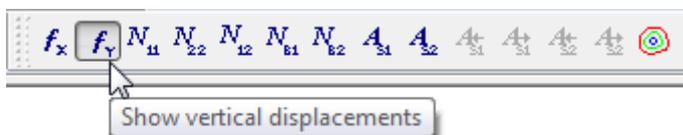
This icons allow you to display the results in the form of isolines. For detailed infos please have a look on the tooltips.

SCN output profile

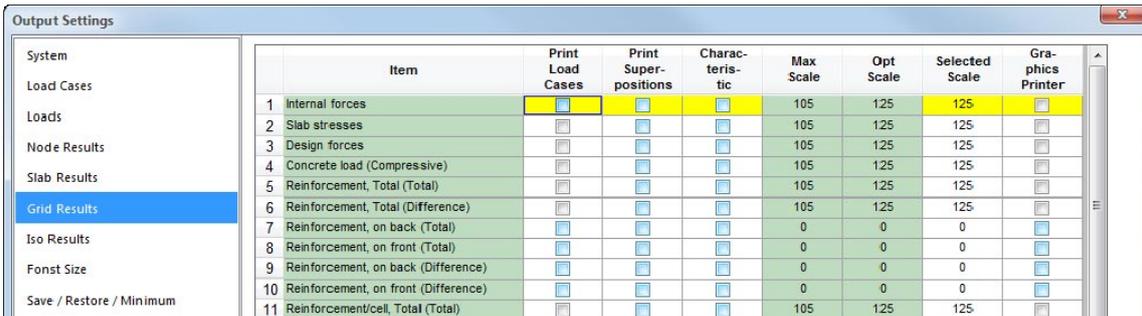Output tab >>Settings >>Profile ...

See also Output & Results

Output profile

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.

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   In version 04/2006 and later versions, 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

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

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…

See also Results & output options.

Options menu

Settings - Panels 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".

Settings overall   Various individual options.

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 <u>all</u> 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 GL.

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.

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. Mönnig). 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.