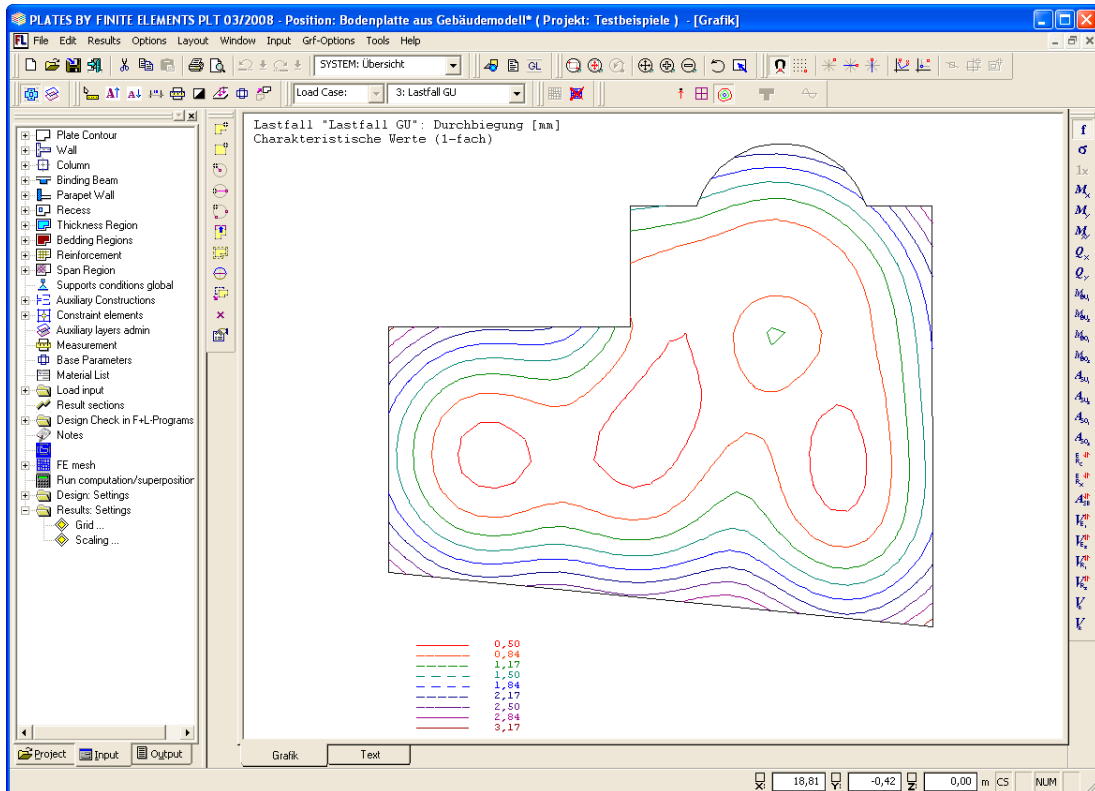


Slabs by finite elements PLT

User manual for F+L design calculation applications under Windows



© Friedrich + Lochner GmbH 2007

F+L on the web

www.frilo.de

E-mail: info@frilo.de

PLT Manual, revision 1/2007

F+L Windows Application: PLT - Slabs by Finite Elements

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

Table of contents

Application options	3
Basis of calculation	4
Input	5
Graphical input	5
Numeric input	5
DXF import	5
System and load input	5
Result sections	6
FE mesh	7
Properties	7
Generation	7
Delete	7
Calculation/superposition	8
Design: Settings	10
Design situation	10
Bending	11
Shear force	11
Crack widths	14
Results: Settings	15
Grid	15
Scaling	15
Output & results	16
PLT output profile	20
Design in F+L application	21
Application-specific icons	21
Additional menus in PLT	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	25

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 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 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, block-out or wall shape via the Graphical input of polygon lines and arc elements.

Downstand beams always have a line shape.

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 Nemetschek-Allplan can also be imported via the ASCII interface.

The transfer of reinforcement calculation results to ISB-CAD or Nemetschek-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

Mesh generator

The implemented mesh generator works according to the "Advancing Front Method". It is suitable for mesh generation based on 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 plates, 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 properties](#).

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

$$l/d < 10 \text{ is true.}$$

With bedded slabs, the ratio of the elastic length (e) to the plate thickness (d) is considered. According to this method, a plate is thick when

$$e/d < 10 \text{ is true,}$$

$$\text{with } e = \sqrt[4]{\frac{4 \cdot D}{C}}, \quad D = \frac{E \cdot d^3}{12 \cdot (1 - \nu^2)}$$

E - modulus of elasticity, C - subgrade modulus

Note in this connection that the typical element length must not exceed the elastic length.

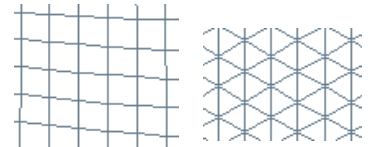
The calculation of action-effects on the element could be performed at various positions: element centre, element corner points, mid-points of element edges.

Type of FE Mesh:

- Quadrilateral elements
- Quadrilateral elements with triangular transition elements
- Triangular elements

Calculate element results at these spots:

- Center of elements
- Midpoints of element sides
- Element vertices



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 Nemetschek 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.

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".



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 line. 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".



Move a section line. 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 dimensions 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 edge length You can define the minimum element edge length. The edge 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.


Consider 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:

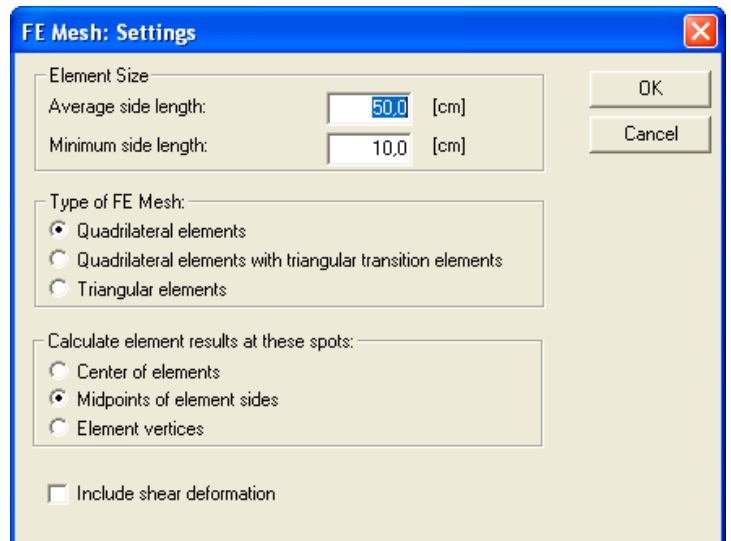
1. You cannot reduce the torsional rigidity of the slab, i.e. the reduction factor is "1.0" (full torsional rigidity) → see [Basic parameters](#) (Graphical input.pdf).
2. You cannot define an orthotropic material for the slab, → see [Basic parameters](#) (Graphical input.pdf).
3. You cannot use supporting direction areas.

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.



Calculation/superposition...

Calculate Tick/untick this option to select the load cases that should be included in the calculation.

Calculated You can see in this column whether a load case has already been calculated.

Incl. selfweight Tick this option if the selfweight 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.

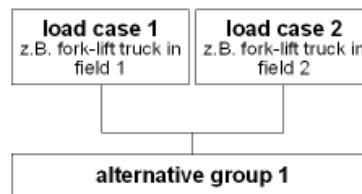
Loads of the alternative group "0" may occur in combination with all other load cases.

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.

DIN 1045-1

Load Cases according to DIN 1045-1 (2001)									
	Name	Com- pute	Com- puted	Include Dead Weight	Partial Safety	Action	Super- pose	Alter- native- group	Domi- nant Action
1	Lastfall G	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	1,35	g	<input checked="" type="checkbox"/>	0	<input type="checkbox"/>
2	Lastfall P	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	1,50	1	<input checked="" type="checkbox"/>	0	<input type="checkbox"/>

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.

Leading action The leading action is considered in the linear calculation for each place and each action-effect. A value of ψ (psi) = 1.0 is assumed. If there are several non-permanent load cases with different action groups, the other variable actions are multiplied with the corresponding Ψ_0 factor.

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.

B4700 Austria and EC2 Italy

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

DIN 1045

Load Cases according to DIN 1045 (7.88)

	Name	Com- pute	Com- puted	Include Dead Weight	Perma- nent	Non- Perma- nent	Factor	Super- pose	Alter- native- group
1	Lastfall G	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	1,00	<input checked="" type="checkbox"/>	0
2	Lastfall P	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	1,00	<input checked="" type="checkbox"/>	0

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 cannot 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.

Design situation

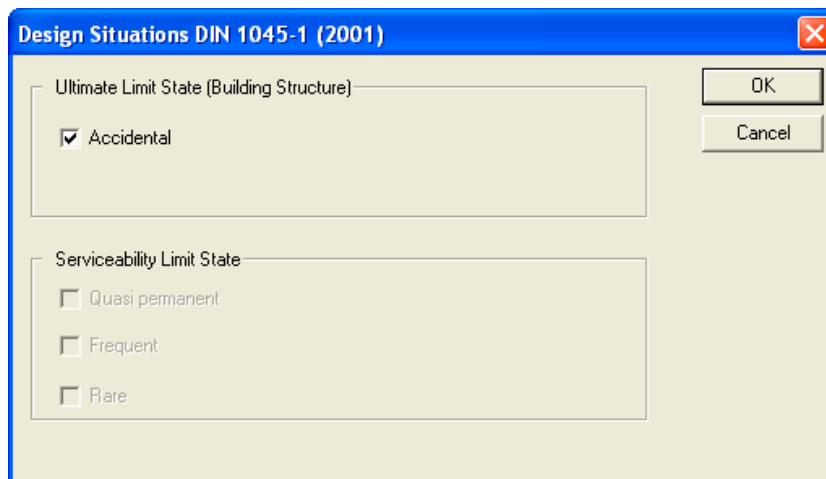
DIN 1045 superseded

Disabled: You cannot apply any settings.

DIN 1045-1

In this section, you can define the design situations that should be examined in addition to the permanent and transient design situations.

In the ultimate limit state, it is the accidental design situation. A prerequisite for the selection of this option is that you have defined a load case as accidental action when determining the [types of effects](#).



For the serviceability limit state, you can select among quasi-permanent, frequent and non-frequent design situations. The design situation that should be selected for the crack proof depends on the requirement class applying to the respective component. The requirement class is determined by the exposition classes selected for the slab.

Un-prestressed reinforced concrete components should comply with the requirement classes E or F in accordance with DIN 1045-1, 11.2.1. The quasi-permanent design situation is relevant for the crack width proof in connection with these requirement classes.

More stringent limitations of the crack width may be required for components with special requirements (e. g. components impermeable to water). These limitations must be proven in addition, if necessary.

The prerequisite for a selection concerning the serviceability limit state is the enabling of the "durability" option in the [basic parameters](#) dialog.

Austrian standard B4700

Disabled: You cannot apply any settings.

EC2 Italy

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

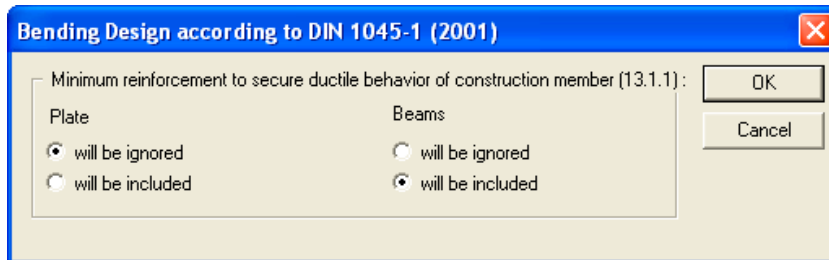
Bending ...

DIN 1045 superseded

Disabled: You cannot apply any settings.

DIN 1045-1

This section allows you to select for slabs and beams individually whether the minimum reinforcement to ensure the ductile component behaviour as per DIN 1045-1, 13.1.1 should be included in the output of the required reinforcement or not.



Austrian standard B4700

Disabled: You cannot apply any settings.

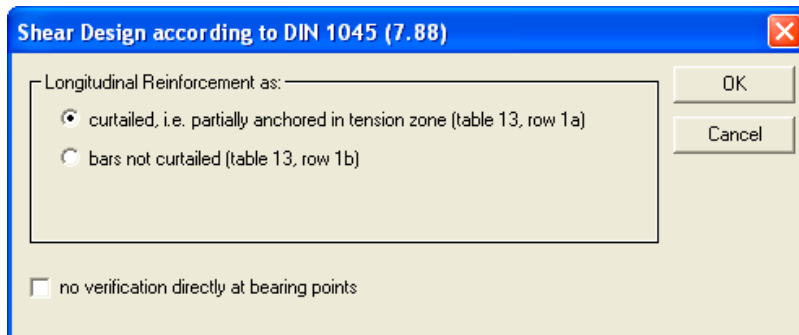
EC2 Italy

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

Shear force...

DIN 1045 7/88

You can select in this section whether the shear force proof should be established for the graded or non-graded flexural tension reinforcement.



No proof immediately 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.

DIN 1045-1

Consideration of the flexural tension reinforcement with...

According to DIN 1045-1, 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).

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.

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 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 proof immediately 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.

Upstand/downstand beams

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

Shear Design according to DIN 1045-1 (2001)

Plate | Beams

Assume the Bending Reinforcement as

- required flexural reinforcement or reinforcement region
- globally preset flexural tensile reinforcement (see below)

	Direction 1	Direction 2	
Upper	4,00	4,00	[cm ² / Meter]
Lower	4,00	4,00	[cm ² / Meter]

Limitation of the Compressive Strut Incline

- Angle θ (59.9 \geq θ \geq 18.4) [°]
- Cotangent θ (0,58 \leq cot θ \leq 3,00) [1]

Determine the internal moment arm by

- the k_z values of the bending design
- a constant k_z value: 0,9

determine the internal moment arm more accurate (as of version 01./2007)

no verification directly at bearing points

OK Abbrechen Übernehmen Hilfe

Shear Design according to DIN 1045-1 (2001)

Plate | Beams

Assume the Bending Reinforcement as

- required flexural tensile reinforcement
- preset flexural tensile reinforcement (see below)

Upper	4,00	[cm ²]
Lower	4,00	[cm ²]

Limitation of the Compressive Strut Incline

- Angle θ (59.9 \geq θ \geq 18.4) [°]
- Cotangent θ (0,58 \leq cot θ \leq 3,00) [1]

determine the internal moment arm more accurate (as of version 01./2007)

no verification directly at bearing points

OK Abbrechen Übernehmen Hilfe

Austrian standard B4700

Consideration of the flexural tension reinforcement with...

According to Ö-Norm B4700, the design value of the shear force capacity of the concrete VRd1 is calculated as a function of the percentage of reinforcement of the flexural tension reinforcement (para. 3.4.4.4, equation 39).

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.

- Limitation of the strut inclination
- Calculation of the internal lever arm with...
- More accurate calculation of the internal lever arm and the concrete cover
- No proof immediately on bearing points

As described for DIN 1045-1.

Field reinforcement grading

You can select whether and how the field reinforcement should be graded.

Upstand/downstand beams

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

EC2 Italy

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

The screenshot shows a software dialog box with a blue title bar and a close button (X) in the top right corner. The dialog is titled "Plate" and "Beams" are visible in the top left. The main content area is divided into several sections:

- Consider the bending reinforcement as:**
 - required flexural reinforcement or reinforcement
 - globally preset flexural tensile reinforcement (s)
- Reinforcement values:** A table with columns "Direction 1" and "Direction 2", and rows "Upper" and "Lower". Each cell contains a text input field with the value "4,00" and a unit label "[cm² / Meter]".
- Limitation of the Compressive Strut Incline:**
 - Angle β (21.8 <= β <= 68.2) [45,0] [°]
 - Tangent β (0,40 <= tan β <= 2,50) [1,00] [1]
- Determine the internal moment arm by:**
 - the kz values of the bending design
 - a constant kz value: 0,9
- determine the internal moment arm more accurate (as of version 01/2007)
- no verification directly at bearing points
- Field reinforcement is:**
 - curtailed: less than 50 % continues end to end
 - curtailed: more than 50 % continues end to end
 - not curtailed: 100 % continues end to end

At the bottom of the dialog, there are four buttons: "OK", "Abbrechen", "Übernehmen", and "Hilfe".

Crack widths...

DIN 1045 superseded

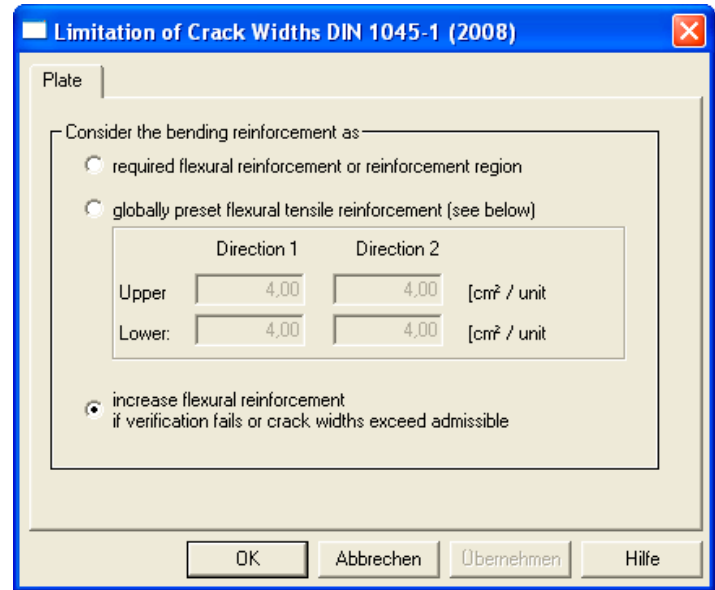
This option is not enabled for this standard.

DIN 1045-1

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".



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.

Results: Settings

Grid

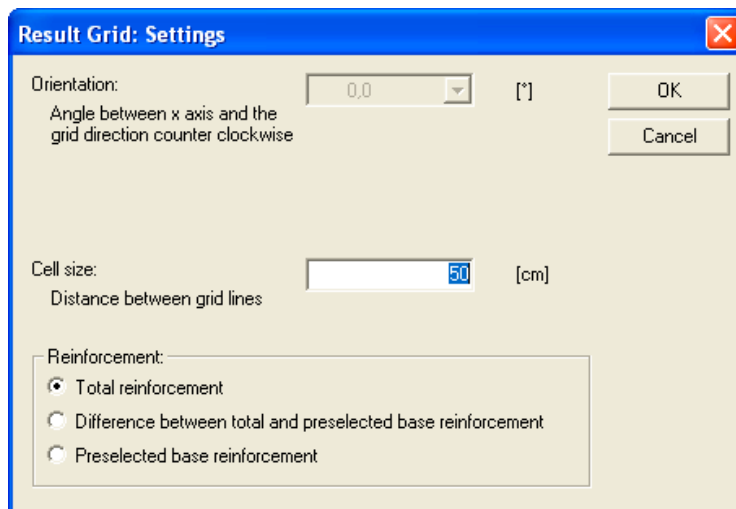
"Results - Grid" menu

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).

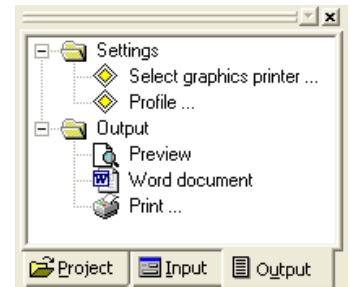
Output & results

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

Select graphic printer...

A graphic printer is a supplementary printer in addition to the standard printer. A large-format printer for drawing formats such as DIN A3 or a plotter are reasonable.

The graphic printer does not replace the standard printer. Therefore you can only select this option when a second printer is connected either directly or via a network to your computer. In general, the standard printer handles standard formats such as DIN A4 and is suitable for the output of texts whereas the graphic printer should be able to handle large formats such as DIN A3 or should be a plotter. You can optionally direct particular graphs to the graphic printer.



- 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).
- Print...** This function allows you to transmit the results and data defined in the output profile to the printer.


Output on the screen

Click on the "Text" tab (below the input area) 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).

Reinforcement steel:	BSt 500 S(A)	
Reinforcement layer, top face:	d1 = 3,0	d2 = 3,5 [cm]
Reinforcement layer, bottom face:	d1 = 3,0	d2 = 3,5 [cm]
DESIGN: Settings		
Design standard:	DIN EN 1992 1-1	

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 superposition

Partial safety coefficients:

- Superposition results are put out **γ -fold** according to the combination rules of DIN 1055-100. 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 put out as **simple** (characteristic) values (with tension spring exclusion, however γ -fold).
- For all [design results](#), the design values are used of course.



Show 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 paragraph "Results in the output grid" below and the chapter [Results menu](#).



Show 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-ct}) → see the paragraph "Isolines" below.



Show upstand/downstand beam results in a separate graphic window. This option allows you to view the action-effects and design results of the upstand/downstand beams.



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

Node results



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.



Show simple superposition values.



Show node shifts. This options allows you to view the node shifts numerically.



Show base compression. 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).



Show bearing forces. This options allows you to view the vertical bearing forces.



Show fixed-end moments around the local x-axis. This option allows you to view fixed-end moments parallel to the wall axis (with walls) or around the local x-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 fixed-end moments around the local y-axis. This option allows you to view fixed-end moments perpendicular to the wall axis (with walls) or around the local 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.

Results in the output grid



Show moments. This option allows you to view the moments m_x , m_y and m_{xy} (torsional moment) of the slab.



Show shear forces. This option allows you to view the shear forces q_{xz} and q_{yz} of the slab.



Show design moments for the lower reinforcement. This option allows you to view the design moments of the lower reinforcement m_{B-1} and m_{B-2} . In a non-rotated system of coordinates, direction 1 corresponds to the x-direction and direction 2 to the y-direction.



Show design moments for the upper reinforcement. This option allows you to view the design moments of the upper reinforcement m_{B-1} and m_{B-2} . In a non-rotated system of coordinates, direction 1 corresponds to the x-direction and direction 2 to the y-direction.



Show lower reinforcement. This option allows you to view the lower reinforcement a_{s-1} and a_{s-2} . In a non-rotated system of coordinates, direction 1 corresponds to the x-direction and direction 2 to the y-direction.



Show upper reinforcement. This option allows you to view the upper reinforcement a_{s-1} and a_{s-2} . In a non-rotated system of coordinates, direction 1 corresponds to the x-direction and direction 2 to the y-direction.



Show 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. The user must produce a punching proof in this case.



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



This icons allow you to display the results in the form of isolines.

PLT 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.

Beams		Result Sections		Font Size		Save	
System	Load Cases	Loads	Summary of results	Node Results	Plate Results	Grid Results	Iso Results
	Item	Print Load Cases	Print Super-position	Max Scale	Opt Scale	Selected Scale	Graphics Printer
1	Deformed System	<input type="checkbox"/>	<input type="checkbox"/>	98	100	75	<input type="checkbox"/>
2	Support reactions (nodes), Characteristic values	<input type="checkbox"/>	<input checked="" type="checkbox"/>	88	100	125	<input type="checkbox"/>

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

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.

Design in F+L application

The "Design in F+L application" 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

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.

Application-specific icons

[Icons of the Graphical input module](#)

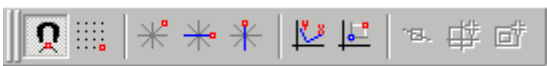


[Icons of the load input section](#)



[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...)

Additional menus in PLT

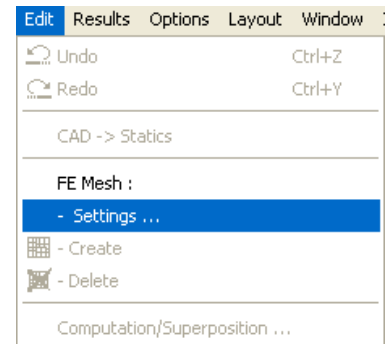
Edit menu

CAD → structural analysis

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: Properties
Generate
Delete

See the chapter [FE-mesh](#)



Calculation/superposition See the chapter [Calculation/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](#).

Shear force design...

A dialog allows you to select whether the longitudinal reinforcement should be considered or not and if so, whether the required or the default longitudinal reinforcement should be taken into account. The dialog offers also additional options for the shear force design calculation.

See also the chapter Design Settings [Shear force ...](#).

See also [Results & output options](#).

Options menu

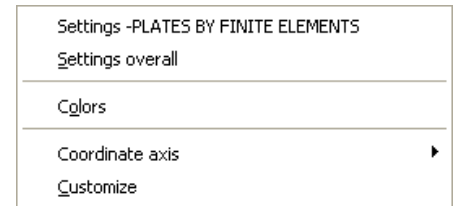
Settings - Slabs by finite elements

[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.

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

[Boolean operation](#)

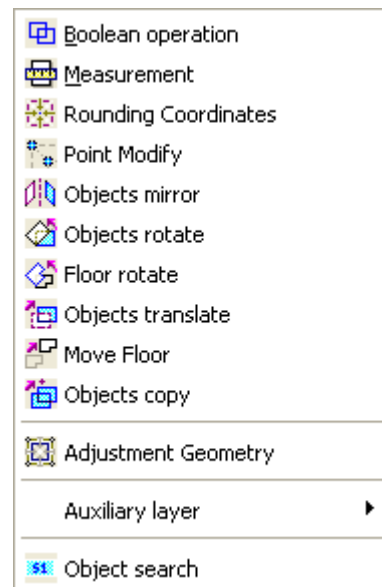
The combination of surfaces via Boolean operations is a user-friendly feature for the Graphical input of complex outlines → see the chapter "Basic features of the Graphical input".

[Measurement](#)

Measurement of distances and angles.

Round coordinates

The entered coordinates are rounded according to the specified accuracy value [in cm].



Edit point

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

Mirror objects

Like in CAD tools, you can mirror one or several objects. Any type of object (columns, walls, beams, etc.) or a combination of different types of objects can be selected and reproduced as a mirror.

[Select the objects](#) and finish the selection process via the context-sensitive menu ▶ Exit. Define subsequently the axis (as

	a line) along which the mirrored object should be reproduced. After completion of the axis input, you can see the original and the reproduced object on the graphic screen.
Rotate objects	Select the objects as described for the mirror function. Select the centre of rotation. Rotate the object either via drag and drop with the mouse or by entering the angle of rotation into an input field that is displayed to the left of the " numeric input ". To rotate the system together with all loads you must first display all loads/load cases.
Move objects	Select the objects as described for the mirror function. Move the objects via drag and drop or the numeric input.
Move floor	Move an entire floor.
Copy objects	Select the objects as described for the mirror function. Position the copied object via drag and drop or numeric input.
Geometric alignment	This function provides for geometric alignment of the slab if problems occur during the generation of the FE mesh.
Move auxiliary slide	You can move a displayed auxiliary slide. The cursor is shown as a square. Click on an arbitrary (distinctive) point of the displayed auxiliary slide. You should enable the capture function for this operation. You can implement the movement of the slide per mouse click on the target (zero) point or via numeric input of coordinates .

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, SC7). 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 SC7).

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.