Beam on Elastic Foundation - BEB+

Contents

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.frilo.com in the Campus-download-section.

Application options

The *BEB+* application is suitable for the calculation of elastically supported beams and one-way slabs in accordance with the subgrade reaction modulus method.

You can optionally select whether foundations and stiffness should be constant, linear or erratic.

You can set the foundation to zero, either completely or in individual sections.

If you set the foundation to zero over the total beam, at least two supports are required. You can add further rigid or elastic supports.

Available standards

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

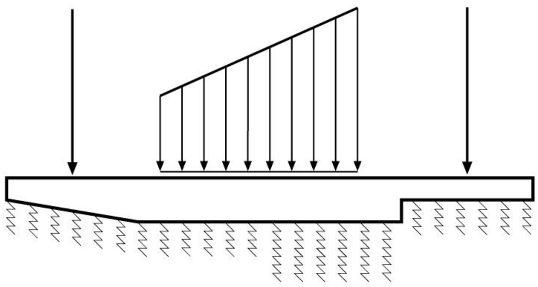
In addition, still DIN 1045 07-88 / DIN 1045-1 / DIN 1045-1/2008/ ÖNorm B4700

The standard for soil analysis is selected by the program to match the reinforced concrete standard.

Loading

- Uniformly distributed loads (UDL)
- Concentrated loads
- Concentrated moments
- Trapezoidal loads

In addition to this, you can calculate moving loads based on concentrated loads.



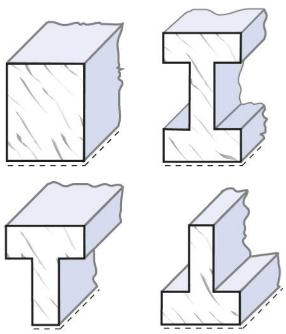
Joints

Bending joints can be defined at freely selectable points.

Cross-sections / Cross-sectional Jumps

- Rectangular cross-section
- T-beam with a slab on top and/or on bottom

You can divide the beam into individual segments with a cross-section at the beginning and one at the end of each beam segment.



Results / Design

- Internal forces
- Displacements
- Design
- Concrete stresses
- Reinforcing steel stress
- Crack width

- Connection of pressure and tension belt
- Shear force design with haunches
- Optional: Shear design for a slab
  The ‘Shear design of slab’ option is available to ensure a correct slab shear design for strips cut out of a slab even if the cross-section to be designed is that of a beam.

Results / geotechnical evidence

- Gaping joint
- Position stability
- Simplified verification

The program carries out the geotechnical verification for a shallow foundation. If the system does not allow this due to joints, supports, haunches or jumps in the cross-section, a simplified and more conservative verification can be carried out using the maximum bearing pressure ordinate from the determination of the internal forces. The ordinate is compared to the design value of the bearing pressure resistance from the foundation engineering standard used.

Additional option BEB-BEW

A reinforcement layout with reinforcement graphics for output in the document or in the large plan format can be purchased as an additional option BEB-BEW.

If this option is not licensed, it can be tested for 30 days as a demo version.

Basis of calculation

The calculation is based on the subgrade reaction modulus method and the displacement method.

A linear elastic behaviour of the soil is assumed, i.e. the subsidence at each point of the beam is proportional to the soil pressure at this point. The proportionality factor is the subgrade modulus (also: foundation modulus). It can be interpreted as a spring constant. The software can consider compression springs and tension springs. Because a tension spring does not comply with the actual soil behaviour, no foundation modulus should be specified for beam areas where the foundation is loaded by tension.

The total stiffness of the system is obtained by assembling the element stiffnesses of the individual beam segments and of the foundation.

The system unknowns in the set of equations are the displacements and translations at the nodes. The internal forces and the soil pressure are calculated from these entities.

Subgrade reaction modulus

You can take the subgrade reaction modulus C [kN/m³] either from general expert literature or from a soil expertise. It is typically in the order of 10,000 kN/m³.

Hahn gives the following guiding values for the subgrade reaction modulus C:

Wet clay soil:	20,000	...	30,000 kN/m³
Dry clay soil:	60,000	...	80,000 kN/m³
Fine gravelly sand soil:	80,000	...	100,000 kN/m³
Coarse gravelly sand soil:	150,000	...	200,000 kN/m³

In many cases, soil expertises specify the stiffness modulus E_s in *kN/m²* instead of the subgrade reaction modulus. The subgrade reaction modulus required by the software depends on the stiffness modulus.

According to reference to /2/, the following applies to rectangular surfaces:

$$C = \varsigma \cdot \frac{E_S}{(1-\nu_S{}^2)\cdot b}$$

ς is a coefficient that is determined by the relation $\frac{L}{b}$.

L/b	1.00	1.50	2.00	3.00	5.00	10.00	20.00	30.00	50.00
ς	1.05	0.87	0.78	0.66	0.54	0.45	0.39	0.33	0.30

The following applies in accordance with reference /2/, p. 283:	for sand/gravel:	ν_s	=	0.125	..	0.5
	for clay:	ν_s	=	0.2	..	0.4

The subgrade reaction modulus is only applicable if the member was divided into segments that are smaller than the elastic length L_e.

Elastic length: $L_e = \sqrt[4]{\frac{4 \cdot E \cdot J}{C \cdot b_{Beam}}}$ (*Note: L_e is calculated by the software*)

Input

You can enter values and define control parameters in the menu on the left screen section. The effect of the entered values is immediately shown in the graphical representation on the right screen section. Before entering any data, you can edit the dimensional units (cm, m ...) via the options
File ▸ Program settings.

Wizard

The Definition wizard is automatically launched when you start the software. You can disable the wizard in the settings menu. You can enter quickly the most important key figures of the structural system in the displayed window. These values can be edited subsequently in the parameter definition section or on the graphical user interface.

Input options in the three-dimensional GUI

A general description of the data-entry options in the GUI is given in the document "Basic Operating Instructions - PLUS".

- Clicking on a graphic element (beam, load, hinge...) opens the corresponding input dialog with the appropriate parameters.
- A click with the right mouse button on a graphic element opens the appropriate context menu.
- Sections can be divided/deleted.
- The dimensions can be edited directly.
- The texts at the top left of the graphics window can be clicked (text links).

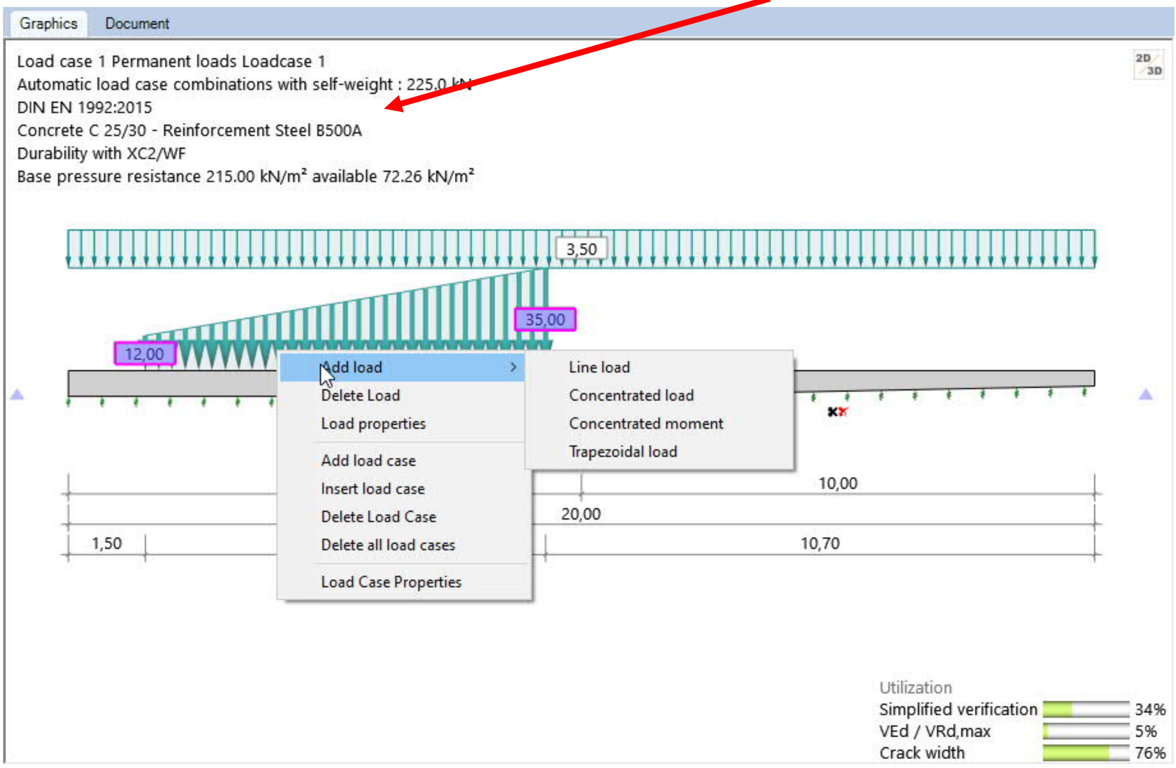
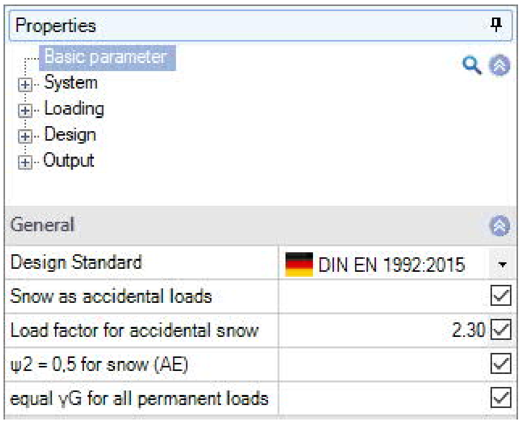


Fig.: the context menu of the load is displayed when right-clicking on the load.

Basic parameters



Design Standard

Select the desired 'reinforced concrete' standard.

Accidental snow load

When you check this option, snow loads are automatically included as accidental action in addition to the typical design situations. The user can either specify a freely selectable load factor for the accidental snow loads or have it determined automatically by the software.

Load factor for accidential snow

Instead of automatically setting the load factor, you can determine the load factor yourself by clicking this option. With this load factor, the snow load - in terms of its characteristic value - is considered an accidental action.

ψ2=0,5 for snow

Specifies whether in the design situation earthquake (AE) the combination coefficient ψ2 (Psi2) for the action snow should be raised to the value 0.5. (See introductory decrees of the german federal states, for example Baden-Württemberg).

Equal γG for all permanent loads

Specifies whether all permanent loads or load cases should be used with the same partial safety factor (γG, sup or γG, inf). Otherwise, all permanent loads or load cases are combined with γG, sup and γG, inf.

System

Concrete / In situ supplement / Reinforcing steel

The selection options for the concrete quality, in situ supplement and the reinforcing steel grade depend on the selected standard.

Stroke weight

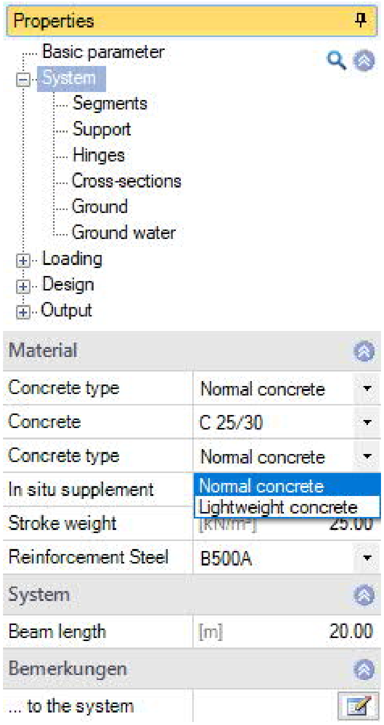
Unit weight of the concrete, which should be taken into account when calculating the dead load.

Beam length

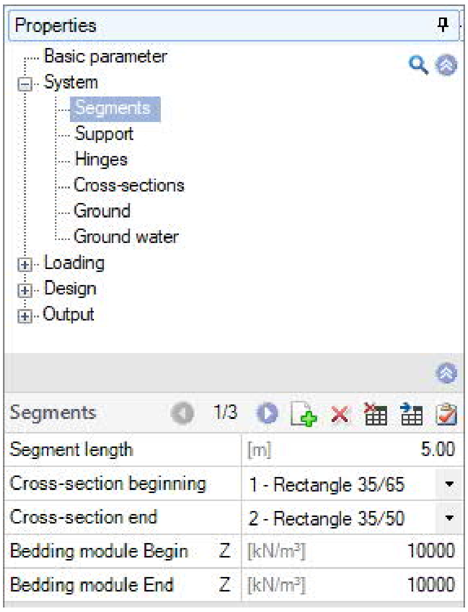
Total length of the beam.
Note: If you define several beam segments subsequently, the total length specified here remains constant.

Remarks

Click on the button to enter your own comments on the system.



Segments



You can divide the member into any number of segments. You should segment the member at cross-sectional jumps or discontinuities in the elastic foundation.

Note: The entered beam length remains constant, even if multiple segments are defined.

Enter the data of the first segment (< beam length) either in the corresponding data-entry mask or directly in the load case table, which you can display below the graphic by activating the Segments tab.

Segment tool bar:
see Definition via tables (Basic Operating Instructions).

To define an additional segment, insert a new row first by activating the button.

Tip: *A description is displayed in the status line each time you click into a specific data-entry field.*

Segment length

Define the length of the segment.

Note: If you change the length of a segment, this does not change the length of the beam. If only one segment was defined, its length is limited to the beam length.

Cross-section at front/rear end

Selection of a cross-section at the front end and at the rear end of the segment. Intermediate values are determined by interpolation. By activating the "Cross-sections" option, you can directly access the data-entry section for cross-sections.

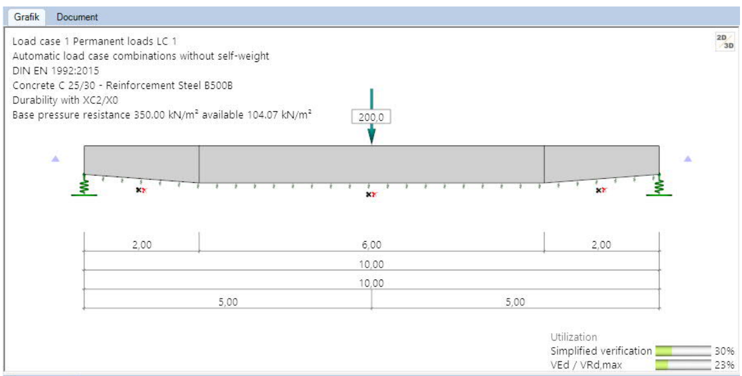
Subgrade reaction modulus at the beginning/end of the cross-section

In many cases, soil expertises specify the stiffness modulus *Es* in kN/m² instead of the subgrade reaction modulus. The subgrade reaction modulus required by the software depends on the stiffness modulus. See also "Basis of calculation":

Note: If you set the foundation to zero over the total beam, at least two supports are required.

Fig.:

Example for the definition of segments and cross-sections.

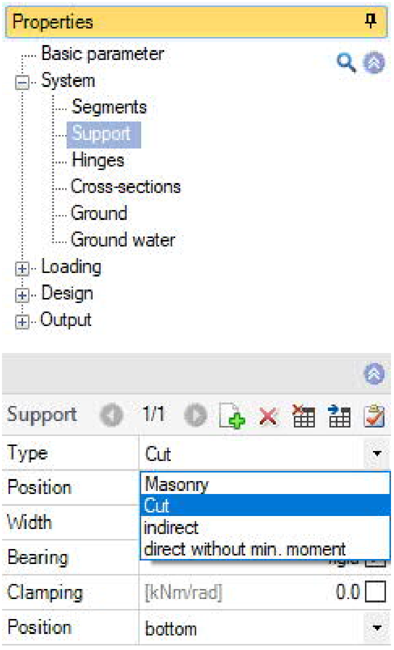


	Segment length [m]	CS beginning	End cross-section	Bedding begin Z [kN/m³]	End bedding Z [kN/m³]
1	2.00	2 - Rectangle 35/50	1 - Rectangle 35/65	10000	10000
2	6.00	1 - Rectangle 35/65	5 - Rectangle 35/65	10000	10000
3	2.00	5 - Rectangle 35/65	2 - Rectangle 35/50	10000	10000

Supports

Type

Selection of the type of support: masonry, cutting edge, indirect, direct without minimum moment.



Position

Distance to the front end of the beam.

Bearing/Rigid

Check this option for a non-sway support in the z-direction.

Spring / Clamping

An elastic support is defined by specifying a corresponding spring stiffness.

The dimension of the spring stiffness is [force/length].

You can calculate the spring stiffness by applying a unit force to the load-bearing component. The spring stiffness results from the following equation:

$$C = \frac{\text{unit force}}{\text{deformation}}$$

Position

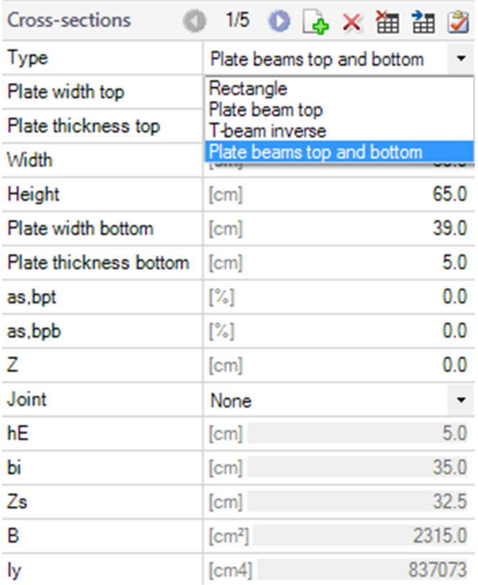
Clamping at the bottom, top or both.

Hinges

Moment hinges are defined via the position of the hinge in relation to the beam front end.

Hinges at a distance *a* < beam height to the front end or rear end of the beam are not recommended.

Cross-sections



Type

- Rectangle
- T-beam with a slab on top and/or on bottom

Plate width on top/on bottom

See graph: b_{pt} or b_{pb}.

The slab width must always exceed the web width by at least 4 cm.

Plate thickness

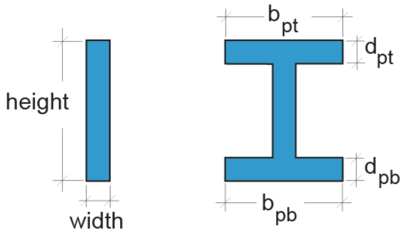
See graph: d_{pt} or d_{pb}.

Width / Height

See graph: b_{pt} or d_{pb}.

as,bpt / as,bpb

Define the amount of the bending reinforcement that is not in the web area but in the plate area. This percentage is considered for the check of the lateral connection reinforcement of the T-bar.



Z

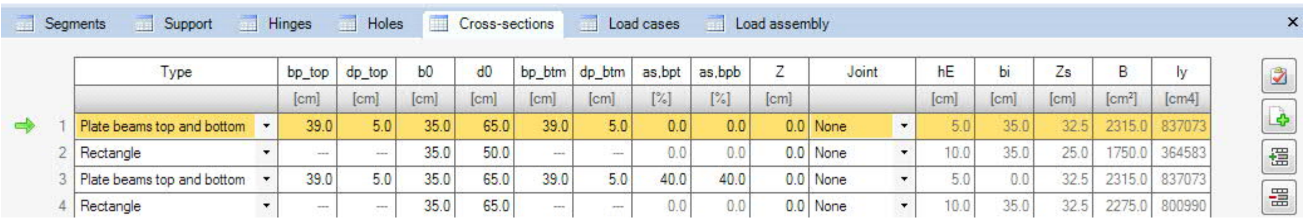
Define a height offset for cross-section jumps at the top edge of the beam.

Joint

Define whether this cross-section has an in-situ concrete supplement in which you adjust the roughness.

hE / bi

Height or width of the joint of the in-situ concrete supplement.



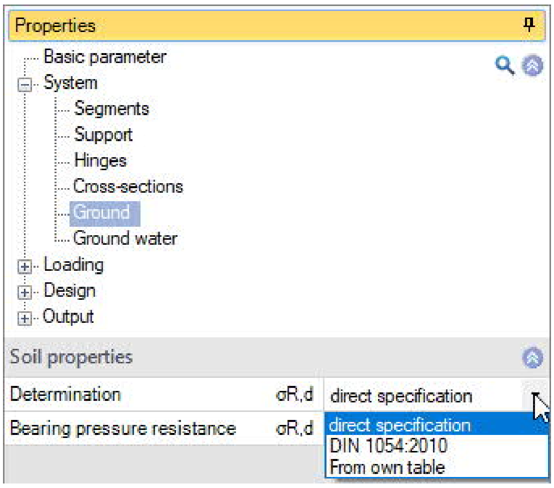
	Type	bp_top	dp_top	b0	d0	bp_btm	dp_btm	as,bpt	as,bpb	Z	Joint	hE	bi	Zs	B	Iy
		[cm]	[cm]	[cm]	[cm]	[cm]	[cm]	[%]	[%]	[cm]		[cm]	[cm]	[cm]	[cm²]	[cm4]
1	Plate beams top and bottom	39.0	5.0	35.0	65.0	39.0	5.0	0.0	0.0	0.0	None	5.0	35.0	32.5	2315.0	837073
2	Rectangle	---	---	35.0	50.0	---	---	0.0	0.0	0.0	None	10.0	35.0	25.0	1750.0	364583
3	Plate beams top and bottom	39.0	5.0	35.0	65.0	39.0	5.0	40.0	40.0	0.0	None	5.0	0.0	32.5	2315.0	837073
4	Rectangle	---	---	35.0	65.0	---	---	0.0	0.0	0.0	None	10.0	35.0	32.5	2275.0	800990

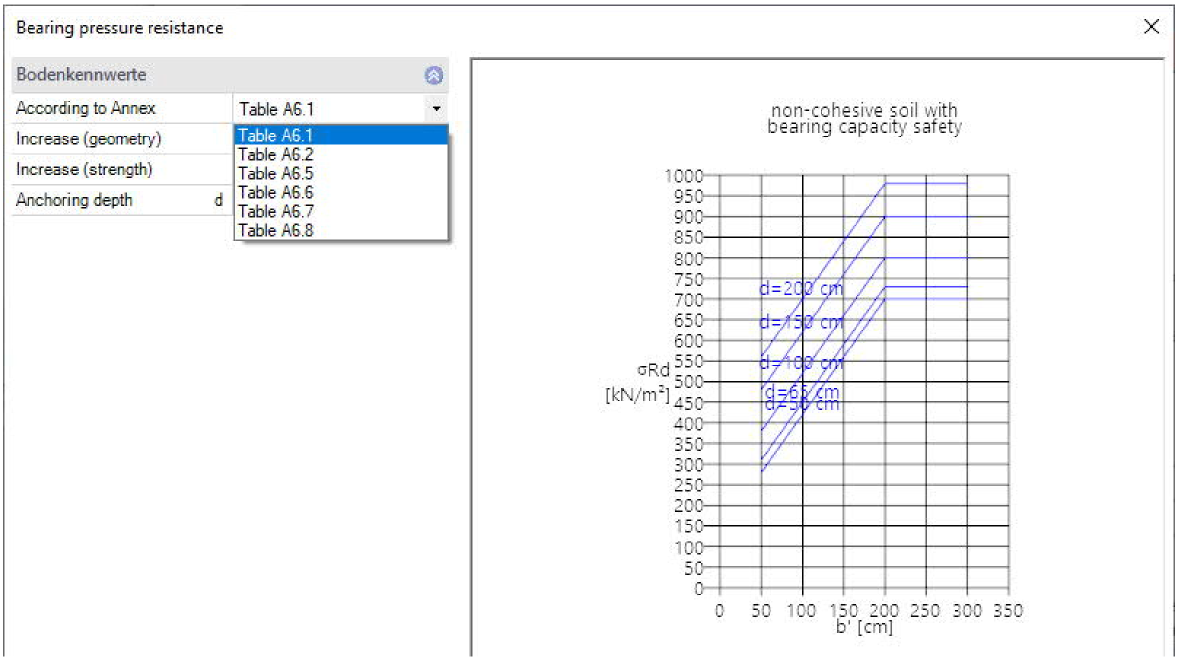
Fig.: Cross-section dimensions / designations

Ground

Soil properties

Determination σR,d — Specify, whether the design value of bearing pressure resistance should be entered directly, or is taken from the standardized table (DIN 1054) or an own defined table. The corresponding table dialogs are opened automatically.





Bearing pressure resistance — In the case of a direct specification, enter the design value of the bearing pressure resistance σR,d for the permanent design situation DS-P. For design situations DS-A, DS-E and DS-T, the design value is increased according to the ratio of the partial safety factors of the bearing resistance.

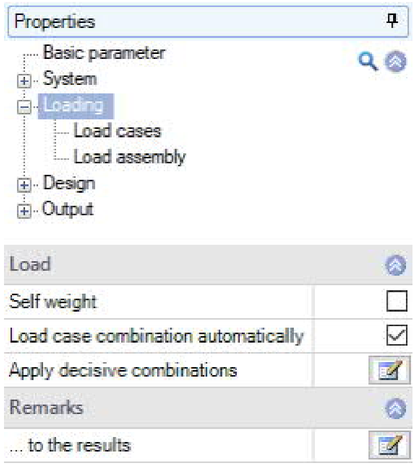
For example, 1.4/1.2 = about 116% or 1.4/1.3 = about 107%

Ground water

Ground water existing — Check this option to show the groundwater level input field. You can define negative values for groundwater below the foundation level.

Loading

You can optionally take self-weight into account.



Automatic load case combination:

Is enabled as a standard.

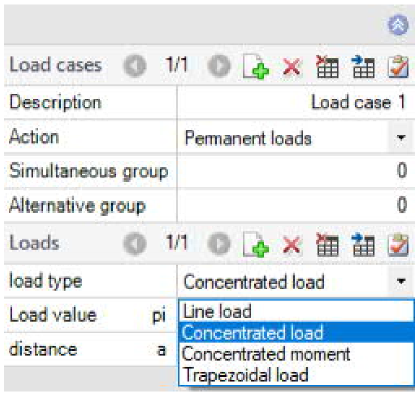
If the automatic load case combination is disabled, only user-defined Load case combinations are considered.

The positions of the moving loads are combined together with the load cases if the automatic load case combination is enabled. If the automatic load case combination is disabled, each position of the moving loads acts as an additional user-defined load case combination.

Apply decisive combinations:

By clicking on the button, the decisive combinations are adopted as the specified load case combinations - the automatic load case combination is then deactivated (can also be activated again).

Load cases



Enter the data of the first load case either in the corresponding data-entry mask or directly in the load case table, which you can display below the graphic by activating the 'load cases' tab.

Load case toolbar: 

see Data-entry via tables (Basic Operating Instructions).

To add load cases, always set up a new load case first by activating the  button (a data-entry mask for the new load case is displayed each time).

Tip: *A description is displayed in the status line each time you click into a data-entry field.*

Description

You can enter a name for the load case.

Actions

Actions based on EN 1990:2010 Table A.1.1 - Recommendations for numerical values for combination coefficients in building construction.

Simultaneous group

Loads of an action group can be defined as "always acting simultaneously" by assigning them to concurrency groups.

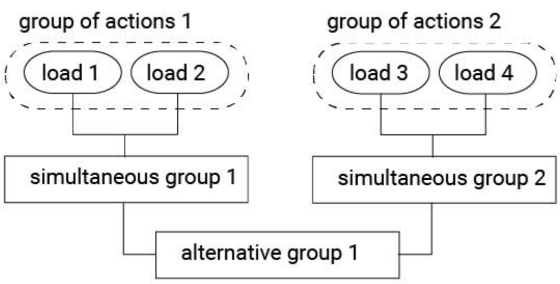


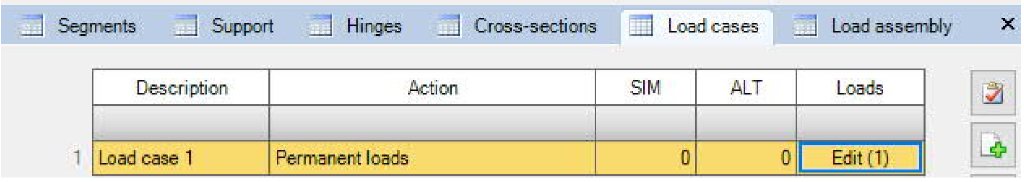
Fig.: *Example to demonstrate the effect of alternative and concurrency groups*

Alternative group

Different variable load cases with similar actions can be combined to an alternative load case group by assigning an alternative group number to them. Only the decisive load case of this alternative load case group is invoked in the superposition.

Loads

Selection of the load type (line, concentrated, trapezoidal load and concentrated moment) and entry of the associated load parameters. Call also via the "Edit" button under the "Load cases" tab. See the following chapter Edit loads.

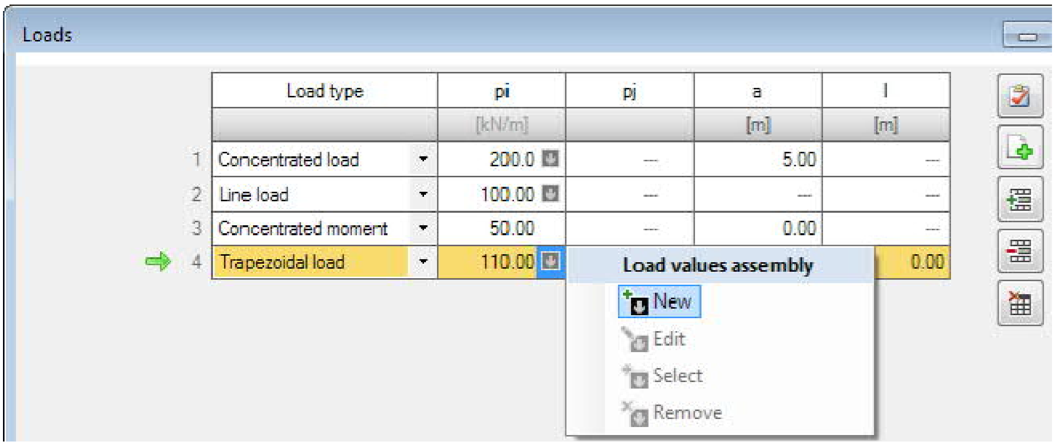


Edit loads

Access the load table via the “Edit” button below 'load cases'.

The following load types are available for selection (schematic drawing on the right):

- Line load over the entire beam
- Concentrated load (lifting loads shall be entered with a negative value)
- Concentrated moment (anticlockwise moments shall be entered with a negative value)
- Trapezoidal load



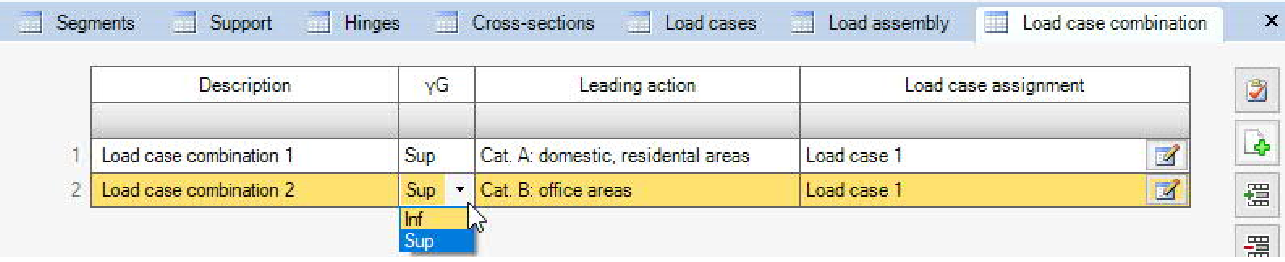
pi / pj Load value pi in kN/m (for a trapezoidal load, pi=left and pj=right load value)

a distance ‘a’ from the left beam end (front edge)

l load length (of a trapezoidal load)

By clicking on the arrow icon you can access a load value summary - see the description of the LOAD+ application.

Load case combinations



If the automatic generation of load case combinations is disabled, the specified load case combinations are generated.

For each load case, you can define the following parameters

- the name
- the upper and lower partial safety factors for permanent loads
- the leading action and, of course,
- the assignment of the individual load cases

The software automatically varies the load case combination for all limit states and design situations.

The results are always the maximum values of all load case combinations.

If you have defined additional moving loads, their load positions act alternatively to all individual load case combinations.

Load assembly settings (moving loads)

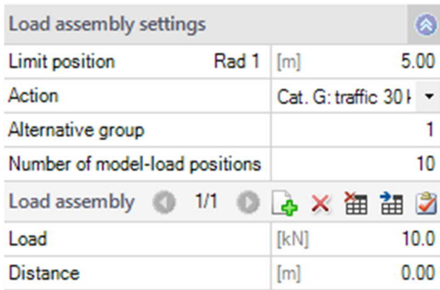
You can define moving loads based on concentrated loads.

Moving loads are defined by specifying the amount of each concentrated load and the distance of this load to the left beam end.

The moving loads must be fully on the beam in the initial position already.

This means that you cannot push moving loads from outside onto the beam.

In the frame of the calculation, the moving loads are pushed over the beam so far that the rightmost load reaches the beam end.



Limit position of wheel 1

This option refers to the distance, the moving loads travel from their defined position via several load positions to the right.

Action

The selected action has an influence on the factorisation of the individual load positions in the load case combinations, which are generated from the load cases and the alternating load positions of the moving loads.

Alternative group

Select an alternative group for the alternating load positions of the moving loads.
If a load case is assigned to the same alternative group than the moving loads, this load case is never superimposed together with any of the load positions.

Number of model-load positions

Define the number of loads positions the moving loads should assume on their way to the limit position.

Load

Define the magnitude of this moving load.

Distance

Define the distance of this load in the first load position, measured from the left front edge of the beam. The load moves in additional load positions to the right.

The individual positions of the moving loads are treated like load cases by assigning an alternative group number to them. They are considered for the global result of the calculation. If pre-defined load case combinations are used, the positions of the moving loads act alternatively to the load case combinations.

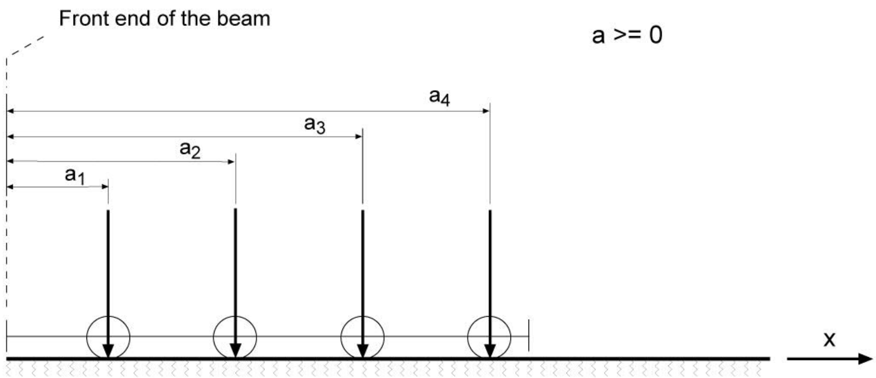
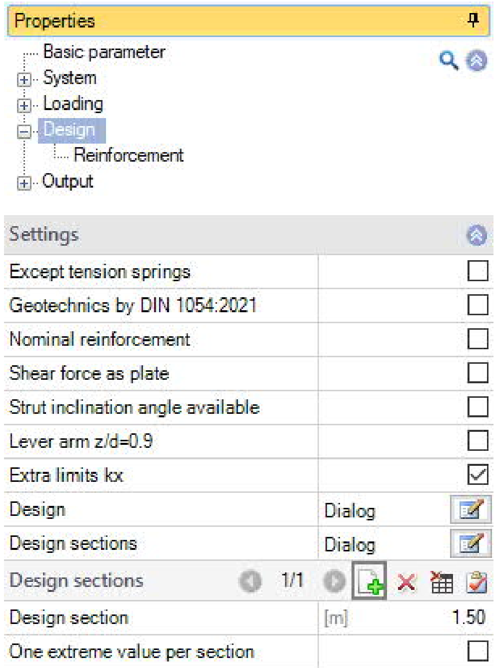


Fig.: Moving loads in their initial position

Design

Settings



Except tension springs In case of tension spring failure, the program checks whether there are tension springs and eliminates them for each load combination. This process is repeated until all tension springs have failed. If the system becomes kinematic, the results are discarded and an error message is displayed.

Geotechnics by.. If the option is activated, the bedded beam is verified as a flat foundation in accordance with the applicable standard. If this is not easily possible due to cross-section jumps, haunches, supports and joints you can deactivate this option. In this way a simplified proof (safe side) of the maximum base pressure ordinate can be carried out .

Nominal reinforcement Reinforcement of ductility according to DIN EN 1992-1-1 9.2.1.1 (1)

Shear force as plate if this option is checked, the shear resistance verification is performed for a plate (instead of a beam), also if a beam cross-section was defined.

Strut inclination angle.. Activate this option to show the input field for the strut inclination angle.

Lever arm kz=0.9 this option allows you to define whether the inner lever arm for the shear force design should be calculated with the constant kz-value of 0.9 or with kz-values from the bending design.

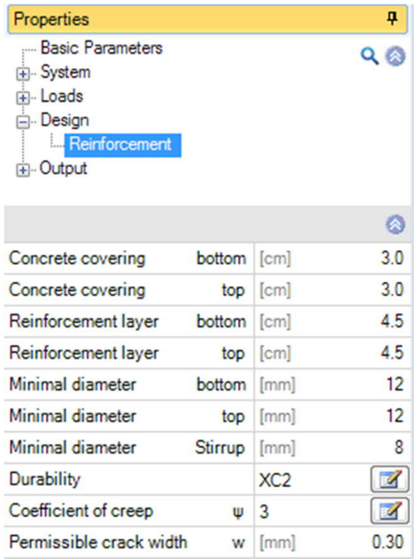
Extra limits kx if you disable this option, the compression zone height is only limited by the yield strength of the steel. In linear elastic calculations of continuous beams, the compression zone height should be limited if no constructive measures are undertaken. Compliance with this criterion is achieved by modifying accordingly the limit steel strain that requires the calculation of compression reinforcement.

Design Call up of the Designdialog.

Design sections activating the button allows you to access the dialog for the definition of design cross-sections in a table. The defined design cross-sections are included in the text and the graphic output in addition to the extreme values of the defined beam segments calculated by the software. The buttons allow you to add or delete table rows (tooltips are displayed when you hover the mouse over the buttons).

One extreme value per section: If this option is activated, only the decisive extreme value is shown in the bar for each section. Otherwise all extreme values are shown.

Reinforcement



Concrete covering The concrete cover refers to the lateral force reinforcement enclosing the bending reinforcement and has an influence on the transverse force design.

Reinforcement layer Center of gravity of the reinforcement.

Minimal diameter Defines a minimum diameter used in the calculation of the concrete cover and the center of gravity of the reinforcement.
In the reinforcement dialog, an attempt is then made to create reinforcement from this minimum diameter.

Durability activating the button displays the Durability dialog. When you confirm your settings in this dialog with OK, the concrete cover, reinforcement layers and their diameter are checked and adjusted accordingly.

Creep factor ψ accesses the dialog for the creep factor and the shrinkage strain.

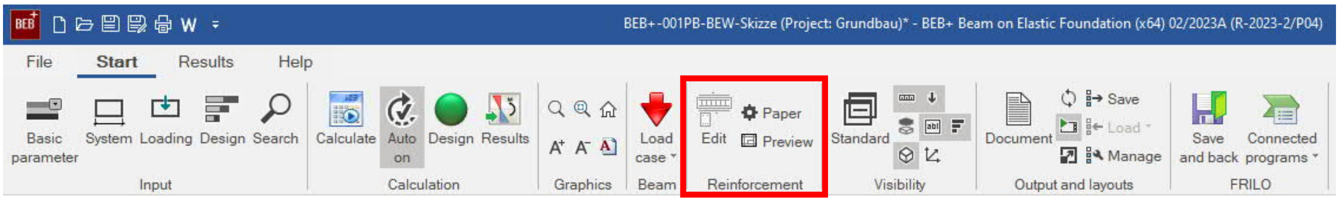
Permissible crack width Here you define the permissible crack width for the verification in the serviceability limit state. Alternatively, you can call up the durability dialog which, after exiting with OK, sets the permissible crack width if the current value is too large. In order to verify the crack width, open the reinforcement dialog and design with rather smaller bar diameters. The crack width check is not activated by default in the output profile.

Reinforcement dialog / Design

General

The BEB-BEW reinforcement layout can be licensed as an additional option. The new reinforcement layout is available in a first pre-release version.

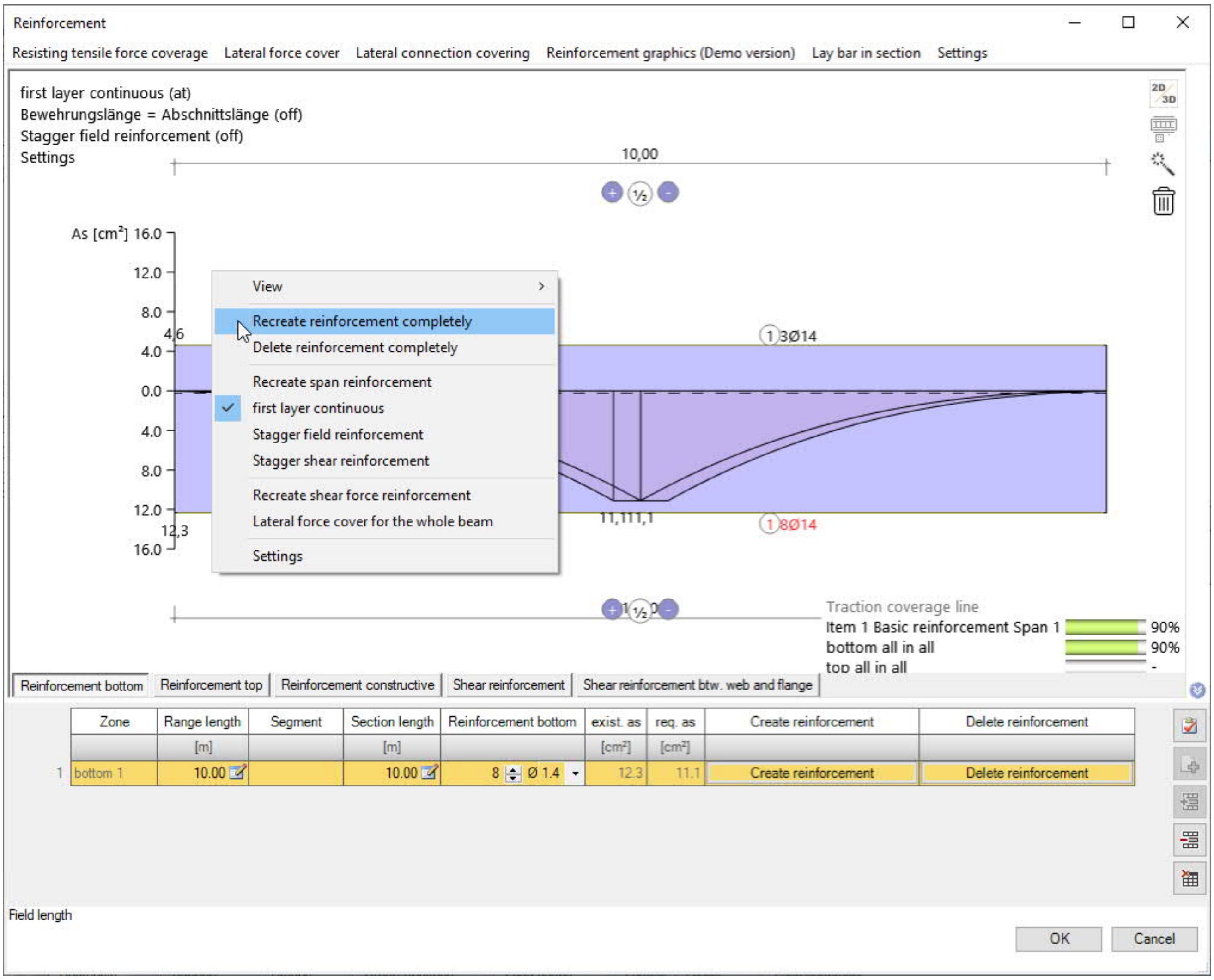
To open the reinforcement dialog, click with the mouse on the "Edit" symbol in the ribbon bar.



Here the item can be covered with reinforcement according to the tensile force and lateral force cover line.

If no reinforcement has been defined yet, the program will suggest a reinforcement.

The reinforcement dialog opens with the tabs for the tensile force coverage line, lateral force, serviceability, reinforcement graphic (additional option BEB-BEW), lay bar in section and the settings.



You can edit interactively in the graphic - right-click to bring up the pop-up menu.

You can use the editing symbols (+/-, ½) to add/delete support and span reinforcement, split or delete a section, and change the section length in the dimension chain directly.

If you hover the mouse over a symbol or the reinforcement information, the corresponding information/tips will be displayed as a tool tip.

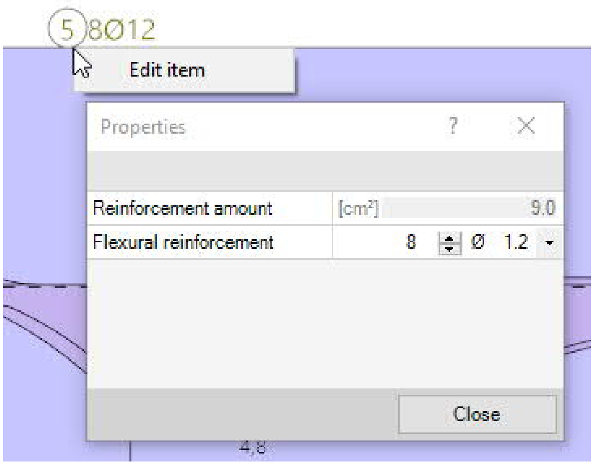
Resisting tensile force coverage

The tensile force coverage line is constructed with an offset and a reinforcement proposal is made if no reinforcement has been defined yet.

The items can be added, deleted and edited via the context menu (right-click directly on the displayed position, see image on the right).

The number of bars in the items can be changed by moving the mouse vertically. Keep the mouse button pressed directly on the item and move it.

Further options are available via the context menu (right mouse button).



The support reinforcement above the end support is inserted into the span for at least 25% of the span length and should be at least 25% of the span reinforcement.

The support reinforcement above the central supports is inserted into the fields at 25% of the field lengths.

The support reinforcement is constructed at least long enough to cover the envelope, the offset dimension and the anchorage.

The field reinforcement is 3 cm shorter at the end support than the beam or the plate. It is guided and anchored at least over the calculated support line. In the dimensional chains, span and support reinforcement can be specified using the plus symbols. The ½ icon divides a section. The red cross icon deletes a section.

It is also possible to activate table input. To do this, click on the small double arrow at the bottom right of the dialog or activate it via "Settings".

Lateral force coverage

By clicking on “Lateral force cover”, the lateral force coverage line is displayed.

The maximum of the shear force curves, which is decisive for the design, is at the distance of the static useful height *d* to the cut of the support. If it is an indirect support, the decisive maximum is located directly at the cut of the support. Usually, the design is calculated for this maximum and this reinforcement is then constantly laid in the entire component. Feasibility is simple and may not be economical depending on labor and material costs.

If the design is taken on the entire component and the reinforcement is inserted in a staggered manner, this is referred to as shear force coverage. The demand for steel decreases. The amount of work and the complexity of the execution increase.

Cutting in the shear force

The shear force coverage line is cut in by the program. To do this, the program limits the shear force between the associated design sections of a support of the design sections. This is followed alternately in the field direction by application areas and incision areas. Together, these always have the length of the associated static useful height and are arranged in such a way that the application and incision are the same area.

Reinforcement selection for shear force

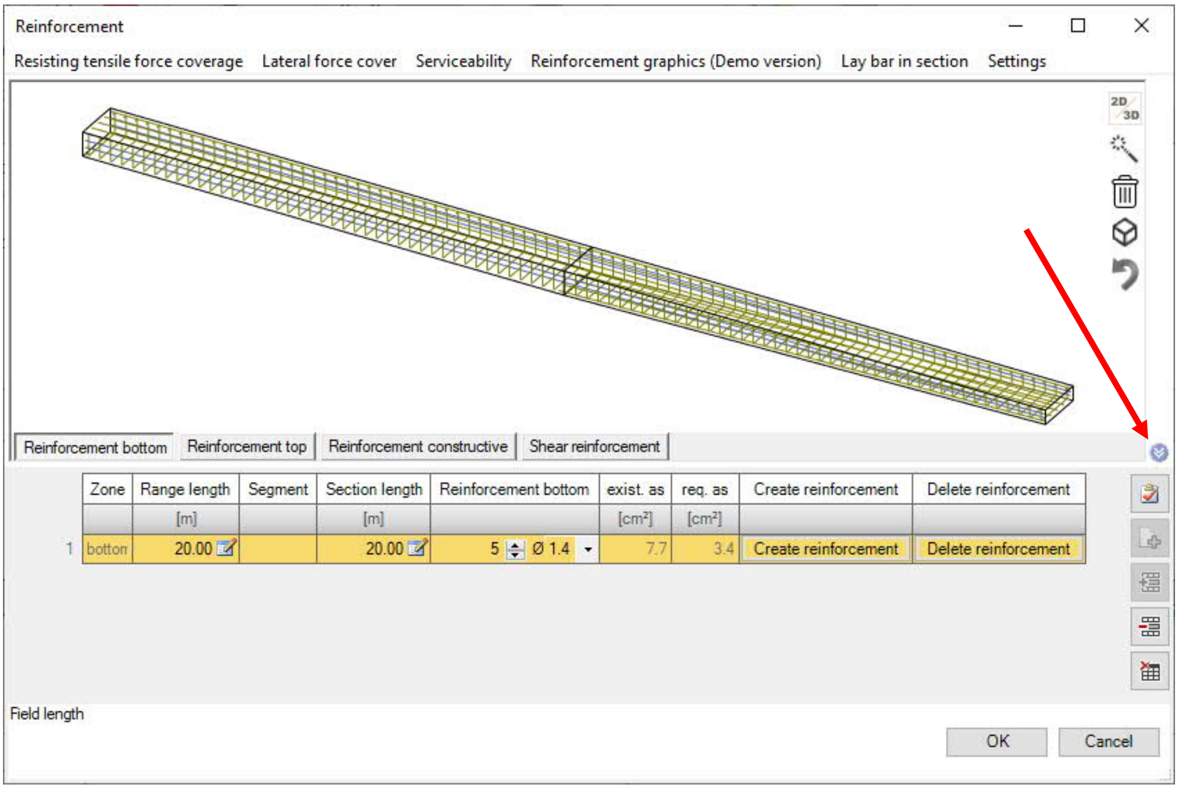
In a later version it will be possible to stagger or constant the shear reinforcement in a bay or section. With a constant shear force reinforcement in the section, any grading can still be achieved if the section is divided into further sections.

If the option for staggered shear reinforcement is selected, the program calculates three reinforcement areas per section. It is first examined how wide the central area can be at most so that the reinforcement currently selected there covers the required reinforcement. Then the lengths of the left and right areas of the section result.

Utilization factors of the selected reinforcement are displayed for all areas at the bottom right of the graphic. The stirrup spacing can be adjusted by holding down the left mouse button and moving the mouse vertically. If the maximum or minimum bar spacing is exceeded, the position is shown in red. If other properties of the stirrup, such as diameter or stirrup shape, are to be changed, this is offered by right-clicking on the position in the context menu under the "Edit" item.

Tables

It is also possible to activate/deactivate the table input. To do this, click on the small double arrow at the bottom right of the dialog.



3D graphics

To view the 3D graph, click the 2D/3D icon at the top right. The concrete can be hidden and displayed.

Output

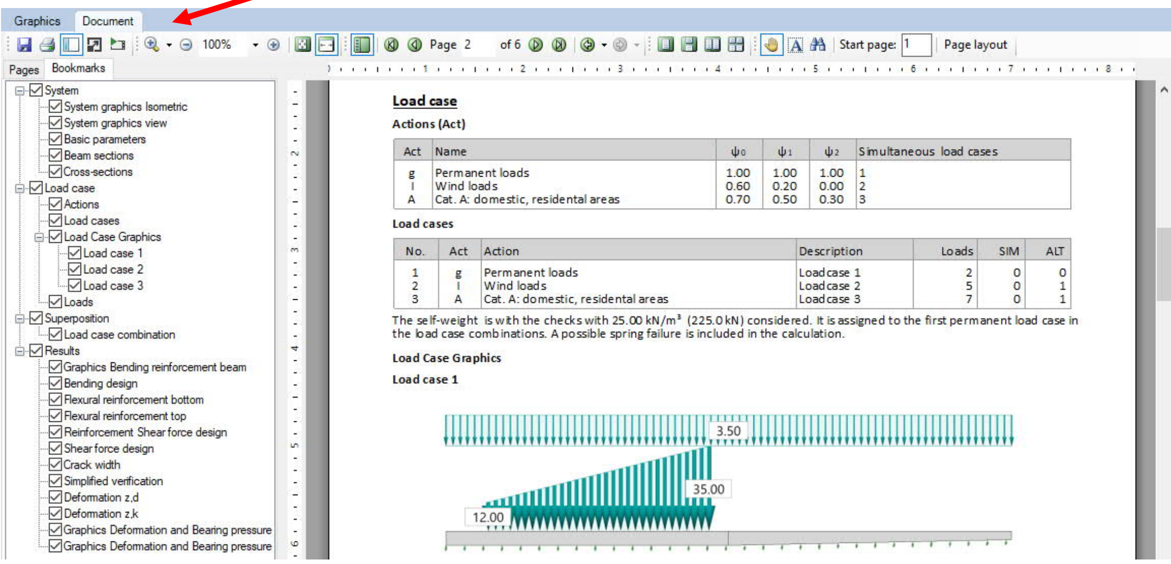
The output scope can differ for the text and the graphics.

You can define the scope of the output either by selecting one of the pre-defined scopes, ‘brief’, ‘standard’ and ‘detailed’ or by checking the individual options.

The font sizes in the graphical representation are customizable.

To display the output, klick on the 'Document' tab.

See also Output and Printing.pdf

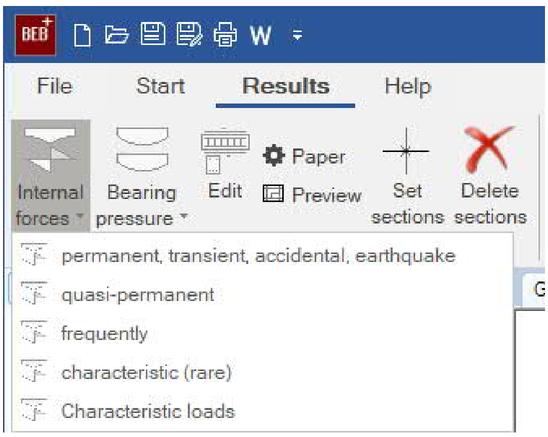


Results / additional buttons

You can display the result graphs (internal forces, bearing pressure, reinforcement).

You can also set sections in the graphic - to finish this action click on the "Set sections" symbol again.

Description of the functions of the different buttons are displayed in tooltips when you place the move cursor onto a button.



Reference literature

/1/ *Beton-Kalender 1980*, Part II, page 592.

/2/ Hahn, J.: *Durchlaufträger, Rahmen, Platten und Balken auf elastischer Bettung*. (Werner) Düsseldorf 1970