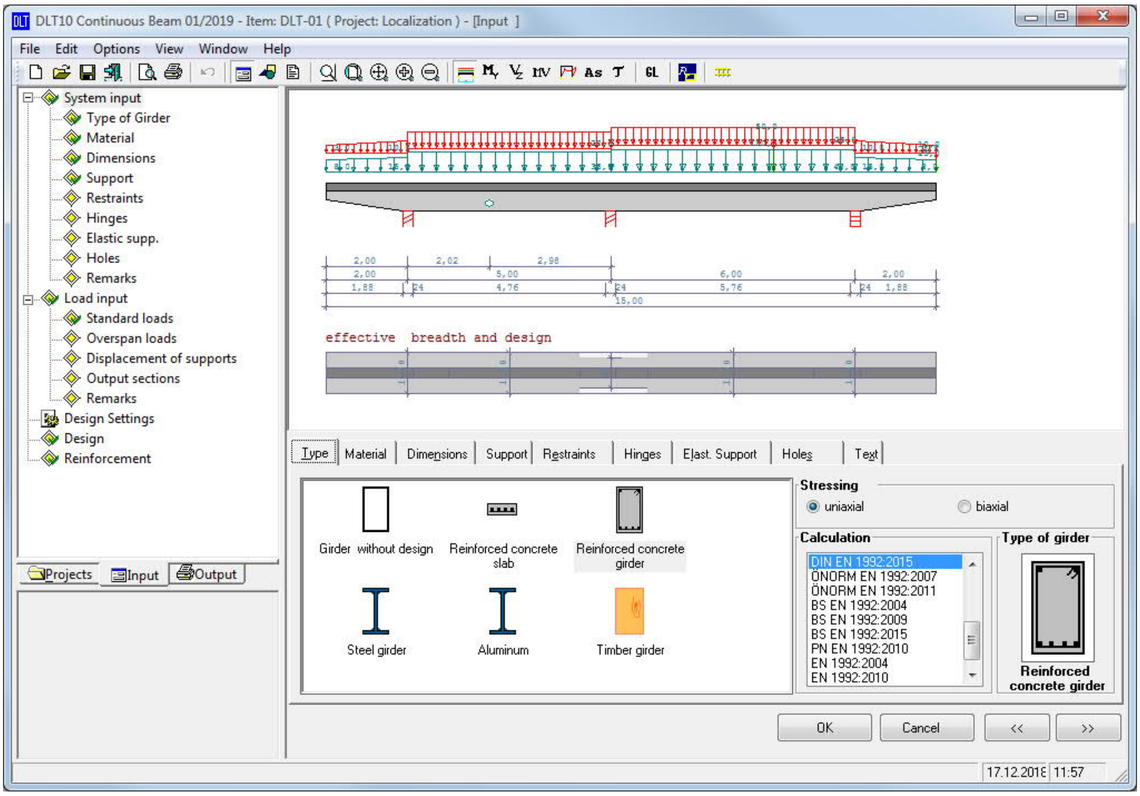
DLT - Continuous Beam

FRILO Software GmbH
www.frilo.com
info@frilo.com
As of 07/09/2021



DLT - Continuous Beam

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 ▸ Support ▸ Articles/Information ▸ Basic operating instructions.

Application options

The application is intended for the calculation of single-span and continuous beams with 12 spans maximum.

Standards

- EN 1992
- EN 1993
- EN 1995
- EN 1999

Note: *Listing of implemented Eurocodes and Annexes: see www.frilo.eu*
▸ Service ▸ Articles/Information.
The specified Eurocodes include the corresponding National Annex.

Available types of beams/girders

- Beam w/o design (free input of the modulus of elasticity)
- Reinforced concrete slab
- Reinforced concrete beam (biaxial available as optional feature)
- Steel girder (biaxial available as optional feature)
- Aluminium girder (biaxial available as optional feature)
- Timber beam (biaxial available as optional feature)

Optional application modules for DLT

The following optional features are not included within the scope of delivery of the DLT application. They are available as additional modules (*see current price list*).

- Biaxial effect of actions, steel, timber aluminium (DLT-HS2)
- Biaxial effect of actions, reinforced concrete (DLT-SB2)
- Reinforcement layout (DLT-BEW)

Design

- The application performs the design and/or stress analysis for the pre-selected cross section dimensions for the material types concrete, steel and timber.
- If the option "Beam w/o design" was selected, merely the action-effects and deflections are calculated in accordance with the preset rigidity.

Reinforced concrete:

- Automatic calculation of the effective slab width.
- Calculation of the deformations in state II for reinforced concrete cross sections with standards based on the partial safety concept.
- Crack width evidence (limiting diameter) and stress analysis.
- Consideration of the durability requirements.
- Calculation and consideration of the creep coefficient and the shrinkage strain in serviceability analyses.
- Analysis of the shear joint for slabs and T-beams.
- You can perform an analysis of the connection of the compression flange (shear analysis) for T-beams.

Timber:

- You can optionally consider shear deformations with timber beams.
- Proof of vibration
- Fire design

Steel and timber:

- Optimization of the dimensioning and design with steel girders and timber beams.

General:

- Cross sections constant or variable
- The rigidity within the span can be constant or variable.
- You can define hinges.

Load import

You can import support loads (from other Frilo-items) that you have calculated in other items (<F5> key).

Holes

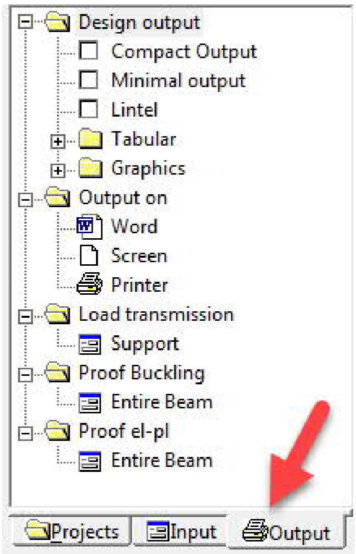
You can define round or rectangular holes for reinforced concrete beams, the calculation is performed in accordance with Booklet 399 DAfStb[1].

Support reactions

The support reactions are put out for the simple and/or γ-fold loads depending on the application control and the task. In addition, the support reactions are put out according to groups of actions.

Interfaces (Output register)

- The loads can be transferred to the column applications B5, HO1, HO1+, B9, B10 and STS.
- You can perform the torsional buckling resistance and elastic-plastic analyses per data transfer to the BT II (second-order bending torsion analysis) or ST7 application.



Restrictions

- Suspension reinforcement is not considered.
- The anchorage of longitudinal bars is not analyzed at haunches and jumps.

[1] German Association for Reinforced Concrete

Basis of calculation

The calculation is based on the displacement method.

Generation of the system geometry

Bars are generated from the entered data according to the following rules:

- at least two bars per span, at least 10 bars with reinforced concrete (due to the calculation of the deformation in state II).
- Nodes are generated at all supports, cross section changes and articulations.
- Haunches are divided into three sections minimum.

Definition of loads

You can define loads for each span (standard loads) or over several spans (multi-span loads).

Calculation of condition lines

Condition lines are calculated for all loads. The following locations are examined in this calculation:

- in each span at the 1/10 points
- at all changes in cross section
- to the left and right of cross sectional jumps
- to the left and right of concentrated loads and individual moments
- at the faces of supports
- to the left and right of hinges
- in the centre of bock-outs

Relevant action-effects

At each point, the greatest and smallest action-effect is sought after.

This is done as follows:

- The permanent load portions of all load cases are summarized.
- The variable load portions in load groups are summarized.
- The decisive action-effects of alternative groups are sought after.

Note: *The application described in this document is suitable for the handling of continuous beams and slabs. If the proportion of the span width to the beam height falls below 2, the system should be calculated as a flat plate and designed accordingly.*

System input

Settings - calculation options

The calculation options (menu item ▸ Options ▸ Settings DLT continuous beam) allow you to make various settings for the calculation. The options are described in the dialog and can be selected via individual tabs (General, Reinf. concrete, Steel, ...).

General

<u>Default loads:</u>

If you tick the default loads option, the load values Gri/Qre are set by default to the values of Gli/Qli when you edit the latter values and the left and right values had been equal to each other before.

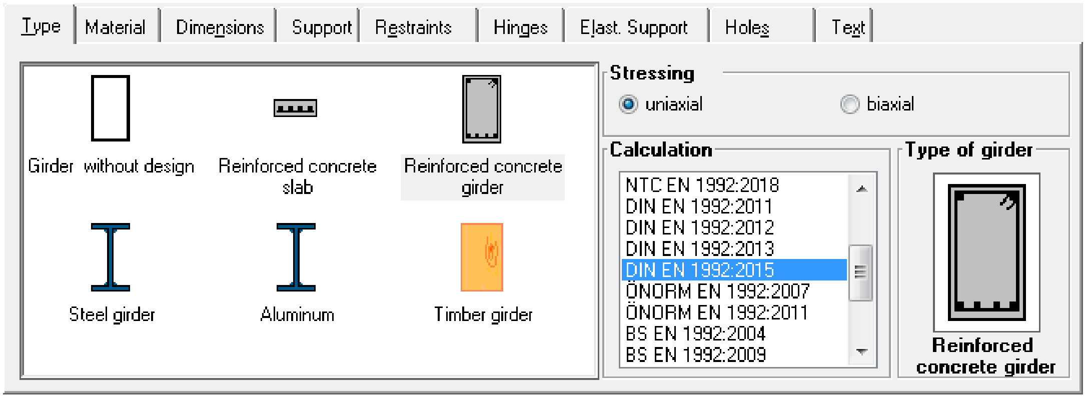
<u>System axis at tripart point of support:</u>

If the user checks this option, the system axis of the end supports runs through the tripart point instead of the centre point of the support.

Type of beam - standard selection - effect of actions

Start with your input on the type tab. Select the type of beam, the kind of effect of actions and the desired standard.

The options, selection lists and input tables available for the system input depend on the selected type of beam and standard.



Beam type	You can select among the options - beam w/o design, - reinforced concrete slab, - reinforced concrete beam, - steel girder, - aluminium girder and - timber beam.
Effect of actions:	You can select between uniaxial or biaxial effect of actions (except reinforced concrete slab).
Note:	*For the biaxial effect of actions, the corresponding optional module DLT-HS2 or DLT-SB2 must be available (extra charge, see current price list).*
Calculation:	Select the desired standard.

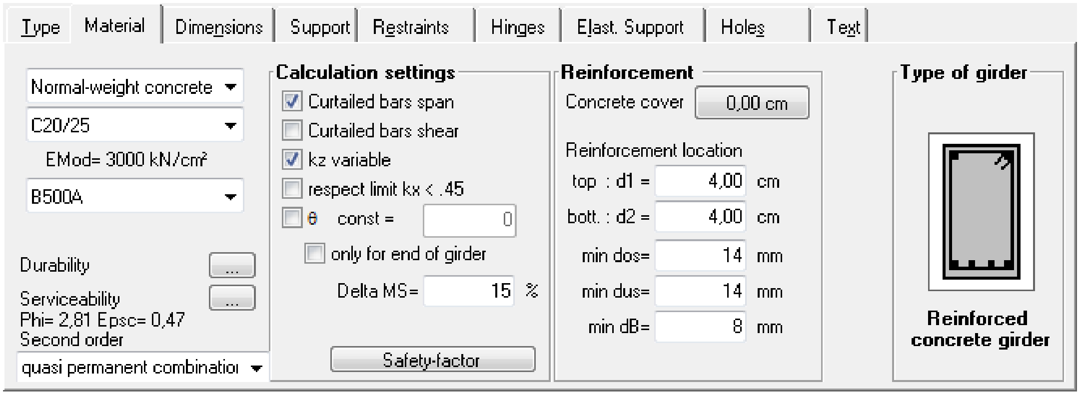
Material selection

Beam w/o design

You can freely define the modulus of elasticity and the shear modulus for the input of the beam.

Reinforced concrete slab and beam

The materials available in the selection lists depend on the selected standard (on the type tab).



Concrete

For standard concrete, the strength classes up to C100/115 and for lightweight concrete, the strength classes up to LC 60/66 are selectable. You must additionally specify the specific weight γ for lightweight concrete.

Reinforcing steel

The available reinforcing steels depend on the selected standard.

Durability

The button ... allows you to access the Durability dialog. Durability is ensured via compliance with the minimum strength of the concrete, the minimum concrete cover and the dimensional allowance as well as additional parameters that result from the requirement class such as the permissible crack width, for instance. It is important in this connection to assign the component to exposure classes.

Serviceability - creep coefficient and shrinkage strain φ, ε

The button ... allows you to access the input dialog for the creep coefficient and the shrinkage strain. These values are needed for the calculation of the deformation in state I and II.

Analysis of the deformations in state I and II

The longitudinal reinforcement (input via the menu item "Reinfocement") is considered by the application. If no reinforcement was defined, the required reinforcement is assumed in each section. If you have not ticked the option "Curtail bottom reinforcement bars", As = max As (max Mf) = constant is assumed.

The results are put out in the table "Serviceability - deflections":

Serviceability - Deflection (cm) φ= 3.16 εcs = 0.43 ‰

span	x	fEI	fEIφ	fEIφε	fEII	fEIIφ	fEIIφε	f
1	1.60	0.17	0.38	0.48	0.33	0.47	0.59	0.59
2	1.80	0.03	0.06	0.07	0.06	0.09	0.09	0.09

x	Location of the maximum deflection within the span
fEI	Deflection in state I
fEIφ	Deflection in state I if creep is considered
fEIφε	Deflection in state I if creep and shrinkage is considered
fEII	Deflection in state II
fEIIφ	Deflection in state II if creep is considered
fEIIφε	Deflection in state II if creep and shrinkage is considered
f	the decisive deflection value

The calculation of the deflection in state I takes the ideal rigidity of the uncracked cross section and the existing reinforcement into account.

After the calculation of the deflection in state I, the deflection in state II is calculated.

Therefore, each span is divided into 10 sections. The effective rigidity in state II is calculated for each of these sections. The calculation of the deflection is based on the average rigidity for the quasi-permanent combination of actions in accordance with EN 1992-1-1 7.4.3.

Distribution coefficient ξ:

$\xi = 1 - 1.0 \cdot 0.5 \cdot (Mcr / M)$ and

$EI = EI\_II \cdot \xi + (1 - \xi) \cdot EI\_I$

The variable actions are included in the quasi-permanent combination with their $\psi 2$-fold values.

Calculation defaults

Curtail bottom reinforcement bars

Ticking this option has two effects: In the analysis EN 1992-1-1, the reinforcement layout is based exclusively on this curtailment.

In the analysis Eurocde standards, the existing longitudinal reinforcement is included in the shear stress analysis. If the option is ticked, the arithmetically required longitudinal reinforcement is assumed at the point x. Otherwise, the longitudinal reinforcement that is required for the greatest midspan moment is considered. If you have entered a longitudinal reinforcement via the reinforcement module DLT-BEW, the actually existing reinforcement is considered at the point x.

Curtail shear reinforcement bars

If this option is ticked, the application prints the required stirrup reinforcement at intervals of ~ h.

Variable Kz

Tick this option to include a variable k_z value into the calculation. Otherwise, a constant k_z value of 0.875 is used.

Comply with limit kx < 0.45 / 0.50

Tick this option to limit the proportion x/h (compression zone height) for areas of plastic deformation EN 1992-1-1 (section 5.5, para. 4).

ϑ const =

You can enter a constant strut inclination. The limitation of the strut inclination stipulated in Paragraph 6.2.3, Sub-section 2 /Eq. 6.7. (EN 1992-1-1) should be observed.

Only for end supports

If you tick this option, ϑ const. applies only to end supports.

Delta MS

You can reduce the moments over continuous edges for slabs and beams.

When you set the reduction percentage in accordance with EN 1992-1-1 5.5, please note that a reduction is only permissible for beams and girders in typical building construction. You can set a reduction of up to 15 %/20 %.

The application will consider the selected reduction even if the limit specified by Eurocode is not complied with.

According to Eurocode, the permissible maximum redistribution percentage amounts to 15% / 20% for steel with standard ductility and to 30 % for steel with high ductility. The limitation of the redistribution percentage required by EN 1992-1-1 5.5 is considered by the application in this connection. In addition, the application ensures that the mid-span moment that corresponds to the reduced moment over continuous edges does not exceed the existing maximum moment.

Element ceiling

Only relevant for steel concrete slabs: Analysis of the steel-concrete interface

Partial safety coefficients for precast components according to Eurocode

You can define the partial safety coefficients γ_c that should apply to the concrete in the design according to Annex A.

Reinforcement

Input of the concrete cover, the reinforcement location, the minimum diameter of the longitudinal reinforcement (min dos/dus), the minimum diameter of the stirrup reinforcement (min dB).

The value entered for the concrete cover is required for the reinforcement layout.

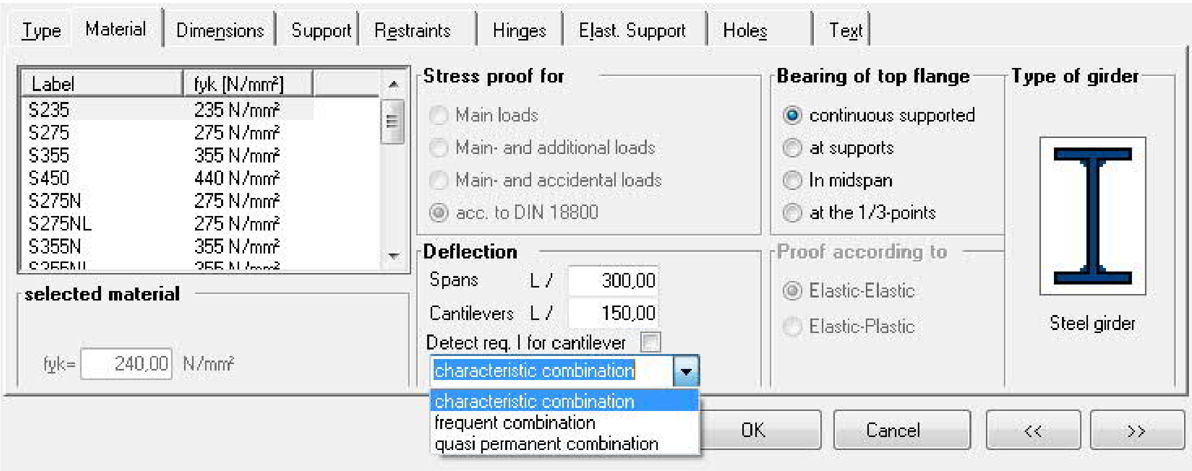
The reinforcement location should be referenced to the centre of gravity of the laid-in reinforcement.

The minimum diameter is used for the proposed reinforcement. You should select this diameter always in view of the problems to be expected to avoid nonsensical reinforcement layouts.

Steel girder

Select the material – f_{yk} is displayed. When you tick the option "Free", you can enter the value of f_{yk} manually.

The design is performed in accordance with EN 1993.



Coefficients for permanent and variable loads

If you select a National Annex according to EN 1993, the partial safety coefficients of the related EN 1990 are used in the design.

Deflection

The permissible deflections are set by default to L/300 (span centre) for the span and L/150 (span end) for the cantilevers. Please note that changes of the rigidity in the span have an effect on the deflection of the cantilevers!

Calculate req. I for cantilever

When this option is ticked the required I is considered with negative deflection. Particularly with "short" cantilevers, the condition $f_K<L_K/150$ might produce nonsensical cross sections.

Top flange support
For the design in accordance with EN 1993, the buckling resistance analysis is automatically performed for the following supports: fork support, fork support + discrete translational spring in the centre of the span (point of application at the top flange), fork support + discrete translational spring in the tripart points (point of application at the top flange).

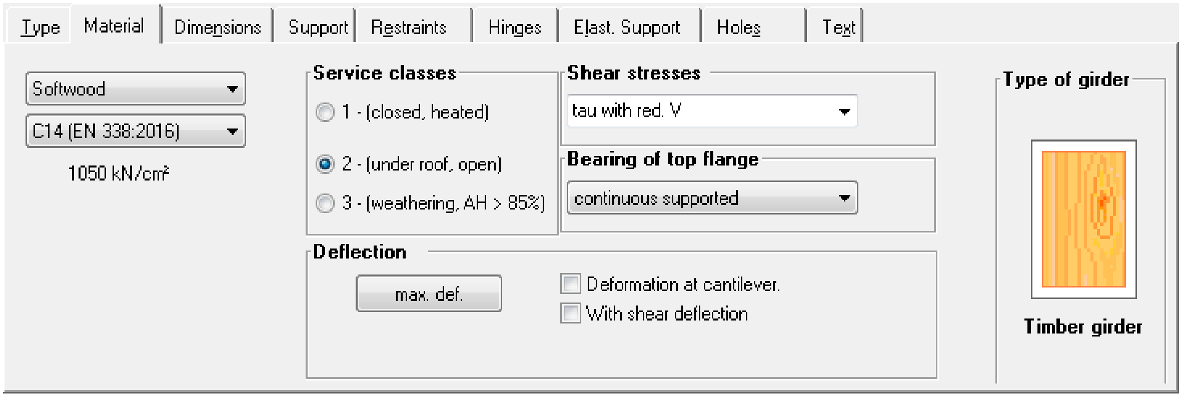
Analysis
Design according to EN 1993-1-1:
The cross sections are assigned to the cross section classes and the verifications are performed according to the equations 6.1 and 6.2 of EN 1993-1-1.

Aluminium girder

The calculation is performed only according to DIN /DIN EN 1999 standard. The input fields are similar to those of the material input for steel described above.

Timber beam

You can select among the materials softwood, hardwood and laminated wood (glulam) for timber beams.



Service classes

According to EN 1995-1- 2.3.1.3

Shear stresses

With concentrated loads applying close to the support, you can optionally perform the shear stress analysis with reduced shear forces acc. to DIN EN 1995-1-1 / NA 2010-12 6.1.7 (NA 5) at a distance of *h* in front of the front face of the support.

Tau with red. V = reduced shear force

Tau with unreduced V = full shear force

Tau with red. V (FE support)

With continuous beams without hinges, the permissible increase of the bending stresses above the inner supports is considered.

In the shear stress analysis according to DIN EN 1995-1-1 / NA 2010-12, a potential increase of the permissible shear stresses is considered for cross sections that are farer than 150 cm away from a timber front (DIN EN 1995-1-1/NA: 2010-12 NDP 6.1.7(2)).

Support of top flange

The buckling resistance analysis is automatically performed for the following supports: fork support, fork support + discrete translational spring in the centre of the span (point of application at the top flange), fork support + discrete translational spring in the tripart, 1/4, 1/5 and 1/6 points (point of application at the top flange).

Deflection

max. def.

Enter the permissible deflections in this section, if you want to use other values than those specified by the design standard.

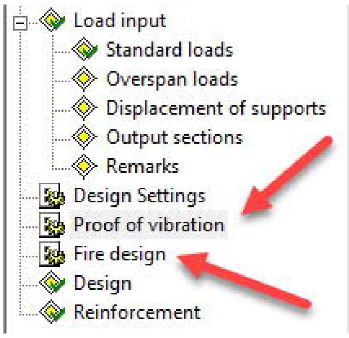
Deformation Eta,f at cantilever

When you tick this option, the utilization resulting from the existing and permissible deflection is also verified at defined cantilevers, if applicable. Otherwise, the cantilevers are ignored in connection with the permissible deflection.

With shear deflection / with creepdeflection

Tick this options, if you want to take shear-/creepdeflection into account.

Fire design



Input of fire resistance duration and charring rates (for the entire beam).

To apply the fire design to the output, activate the corresponding option in the output profile.

- See also „Fire protection analysis timber“ (EN1995-1-2).

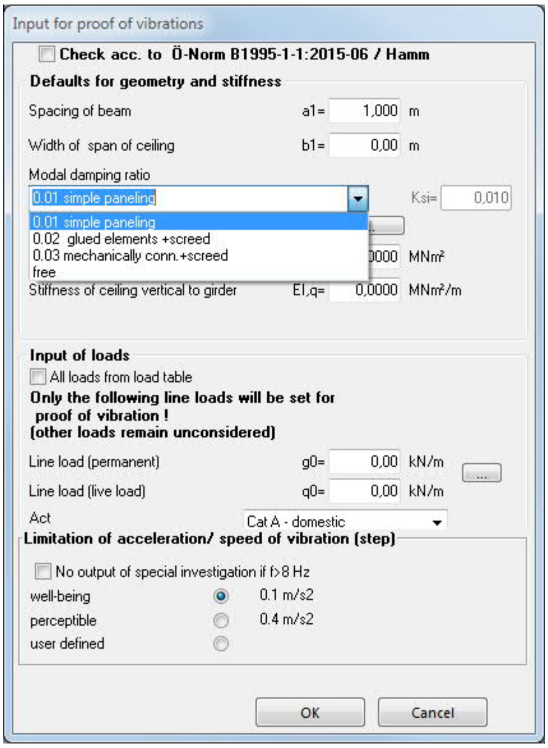
Proof of vibration (EN 1995-1-1 7.3)

(Verification of vibration restistance)

Verification according to EN 1995-1-1 7.3 (as well as ÖNORM B 1995-1-1: 2015-06)

Call the dialog via the item "Proof of vibration".

To apply the proof of vibration to the output, activate the corresponding option in the output profile.



Presetting of the modal damping ratio *Ksi*:
According to "Bauen mit Holz" 11/2001, p. 29 "Deckenschwingungen" by Mohr, there are several options for the modal damping ratio:

0.01 for timber floors with simple planking

0.02 for timber floors with glued board stack elements with additional superstructures such as screed flooring

0.03 for timber floors with mechanically connected board stack elements with additional superstructures such as screed flooring

Select "free" to enter *Ksi* yourself.

With the exception of Ksi and a1, all other parameters are optional.

Additional rigidities from the ceiling structure can be specified in layers via a separate dialog (button ...).

Load specifications

The loads are taken over from the load table by default. Alternatively, you can enter the loads yourself (remove the check mark from the option to show the input fields for the line loads).
With the button ... you can use area loads instead of line loads, which are then converted by the program into line loads.
Act. Selection of the action group.

Limitation of the acceleration / speed of vibration (EN 1995)

f eigenfrequency

f > 8Hz: In this case, the following requirements should be met for apartment ceilings.

- Limitation of the deflection $\frac{w}{F} \leq a \;\; mm/KN$

- Limitation of the vibration velocity v due to the unit impulse $v \leq \beta^{(f1 \cdot \zeta - 1)} \;\; m/(Ns^2)$

f ≤ 8Hz: In this case, a special examination should be carried out for apartment ceilings.
Here, two additional proofs are performed, which correspond to the approach of / 1 /.
/1/ Blaß, H. J. Erläuterungen zu DIN 1052-2004-08 , Bruderverlag März 2005

- Limitation of the vibration velocity v due to kicking $v \leq 6 \cdot \beta^{(f1 \cdot \zeta - 1)} \;\; m/(Ns^2)$

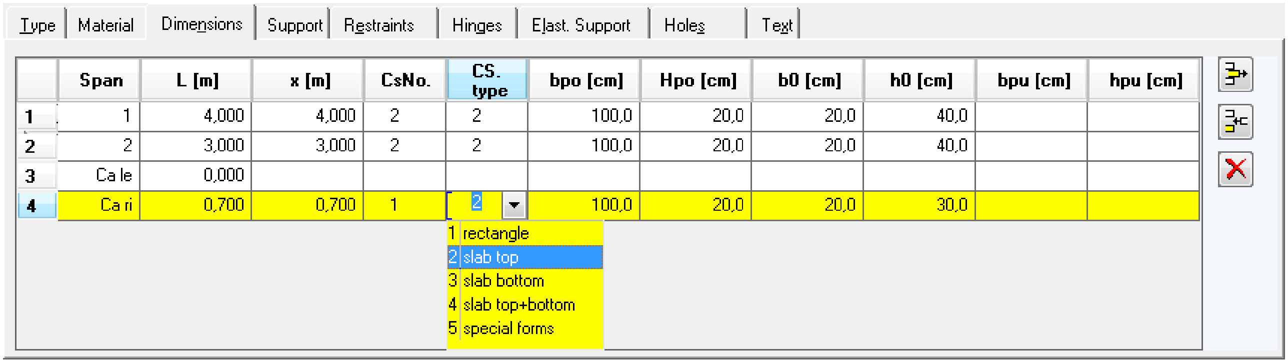
- Limitation of acceleration $a_{vert} \leq 0{,}1 \;\; m/s^2 - 0{,}4 \;\; m/s^2$

These additional proofs can optionally be switched off for systems with f> 8 Hz.
Example (in german) on www.frilo.eu: Schwingung nach DIN EN 1995.

Dimensions

You can enter the lengths and cross sections of the individual spans via this table.

The available input columns depend on the selected type of beam.



Span — Consecutive numbering of spans. The span number is consecutively incremented by the application, cantilevers are designated according to their position "Ca le" = left cantilever and "Ca ri" = right cantilever.
When you delete or insert table rows the spans are renumbered automatically.

L[m] — Input of the span length / cantilever length in [m]

x[m] — Specification of several cross sections in a span.
Input of the point(s) x for Cross sectional changes/jumps.
Set x = L (default) for cross sections that are constant over the total span.
Reference point for x: x is defined by the distance to the left support - if a left cantilever was defined, the value of x specifies the distance to the first (left) support.

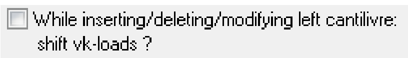
CsNo — Cross section number. Select "0" in the drop-down list to generate a new cross section number.

Cross section selection: select the number of a cross section that you have already defined.

<F5> key: Define or edit cross sections. Press the key to display the corresponding dialog.
Reinforced concrete: The <F5> key allows you to access a dialog for the input of the dimensions of the cast-in-place complement.
See the chapter Analysis of working joints on T-beams.

CS.type bpo, hpo, b0, h0, bpu, hpu → see Cross sections for reinforced concrete beams

Shift FE loads (VK-loads)



FE loads – see multiple span loads - always refer to the FE of the beam. In case of a beam without cantilever, the value entered in the FE column refers to the left support. If you introduce subsequently a left cantilever, the cantilever end becomes the FE of the beam, i.e. the load is displaced to the left by the length of the cantilever.

When you tick the option, the first support continues to be the reference point of loads that you have already entered.

This applies analogously to the deflection of the left cantilever.

Entering, assigning and editing cross sections

You can define, assign or edit cross sections via the “CsNo” column (cross section number). The system assigns automatically a consecutive cross section number to each newly defined cross section. When defining new spans/areas, you can assign an existing cross section to it by selecting its number in this column.

Several cross sections can be assigned to one span via the definition of discontinuity points x →, see "cross sectional areas - multiple cross sections per span."

For a constant cross section over a span, set x = L.

Defining a cross section

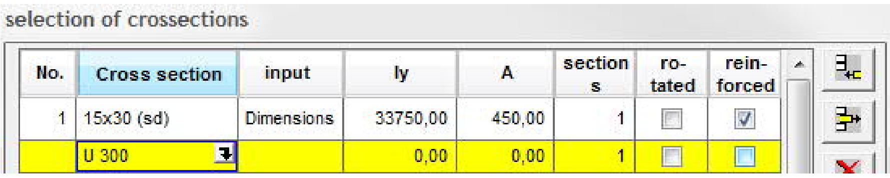
To define a new cross section, enter the value "0" in the column "CsNo" (in the dimension dialog) and press the return key

or

press the <F5> when the cursor is positioned in the "CsNo" column

The cross section selection dialog is displayed subsequently. With reinforced concrete slabs, the corresponding input fields (width b0, height h0) are displayed.

If you have selected the type reinforced concrete beam, an "adjusted" input dialog is displayed - see the chapter Cross sections for reinforced concrete beams



No. — The consecutive cross section number that is automatically assigned by the application.

Cross section — Press the <F5> key or click on . The dialog "Select/edit cross section" is displayed and you can select a cross section from the profile file or define a cross section via its dimensions or its I, A ,W-values.

Input — Help information, STDAT= cross section from the F+L profile file, dimension = define a cross section via its dimensions, IAW = define a cross section via its IAW values.

Iy, A — Help information

multipart — L-, U- and double-T-profile sections can be arranged next to each other in simple, double or triple configuration (specify the number of profile sections in the column "multipart").

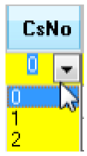
The rigidities are linearly added up in the calculation. The influence of profile sections welded together is not considered. Two sections next to each other are treated in principle as if 2 bars with a section each would lie next to each other and bear half of the load each. Where unsymmetrical profile sections are concerned, I_{yz} = 0 is assumed in addition.

Rotated — You can use the entered section either in its standard position or rotated by 90°.
Unticked option = standard position: the web is in the z axis.
Ticked option = rotated by 90°

Note: *Unsymmetrical cross sections (e.g. angle profile sections) are introduced into the system with the axes parallel to the profile legs and not with their main inertia axes.*

Reinforced — You can reinforce rectangular timber cross sections with steel profile sections, → see the chapter "Cross sections for timber beams - steel reinforcement".

Assigning an existing cross section



Click on the arrow of the drop-down list in the column "CsNo" to display the existing cross section numbers and select a number.

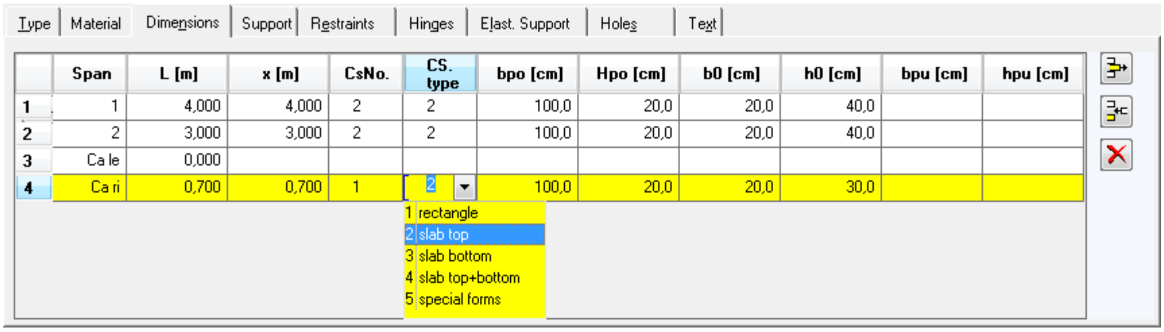
Modifying a cross sections

To edit an existing cross section, position the cursor in the "CsNo" column and press the <F5> key (exception: reinforced concrete slab - edit the dimensions in the columns b0 and h0 in this case). The list of available cross sections is displayed. Click on the cross section to be edited and press <F5> again to access the dialog Select/edit cross section. Apply your changes subsequently.

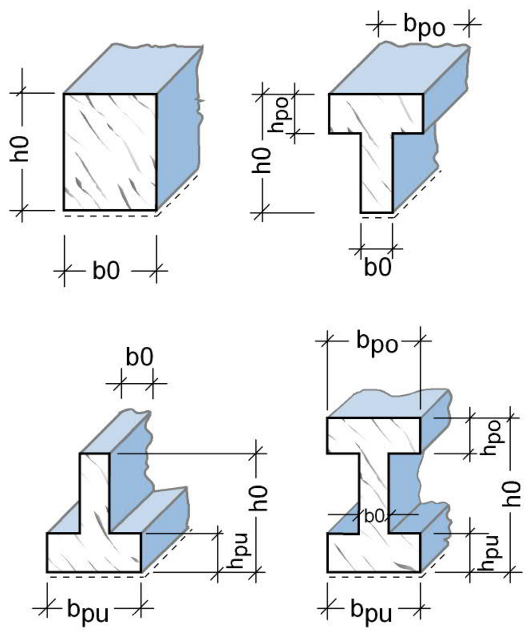
Cross sections for reinforced concrete beams

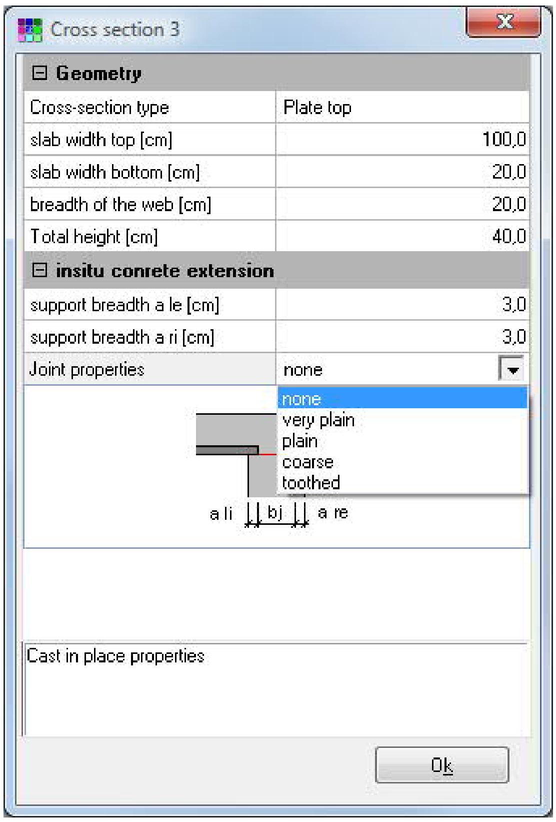
Select in the column "Cross sec." the desired cross section type or the special shapes displayed in a separate window.

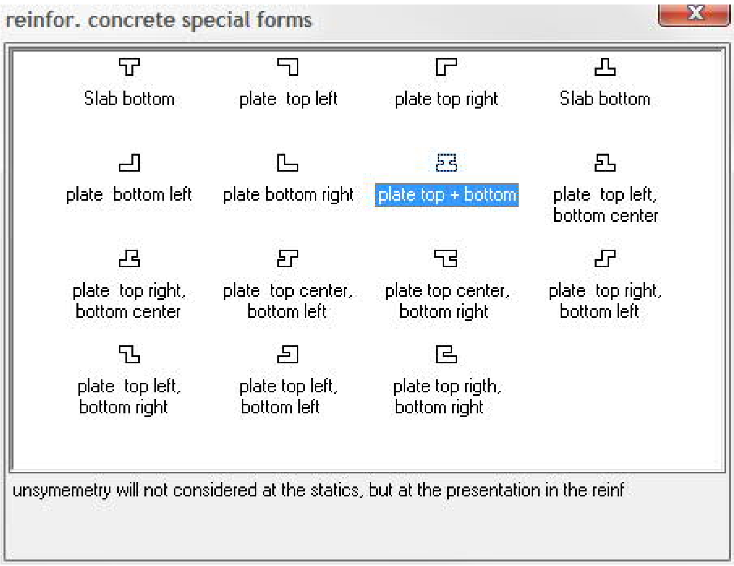
Specify the dimensions of each cross section in the corresponding columns (b_{po}, h_{po}, b_o, h_o, b_{pu}, h_{pu}).



By pressing the F5 key in the column CsNO (Crosssection Number) a dialog shows up in which you can define the shear connection (none, very plain - toothed).





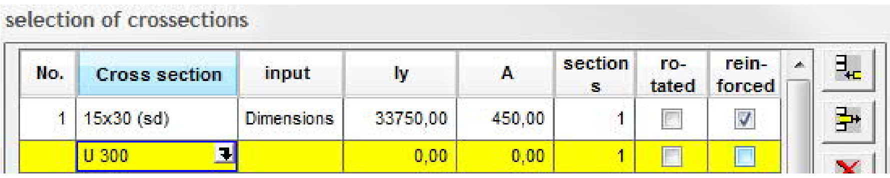


Ill.: selection of special shapes

Cross sections for timber beams - steel reinforcement

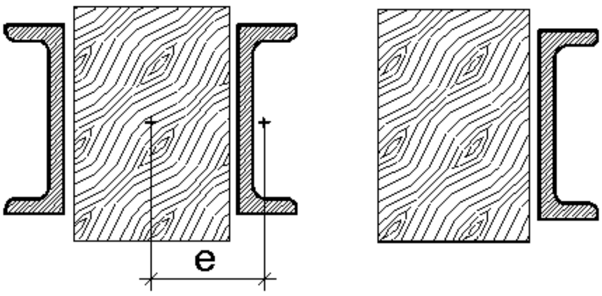
You can reinforce rectangular timber cross sections with steel sections.

To do this, tick the option "Reinforced" in the dialog "Cross section selection" (to tick/untick activate the space key or click with the mouse) A new cross section line is displayed where you can select or define the desired steel reinforcement section.


selection of crossections

No.	Cross section	input	Iy	A	sections	ro-tated	rein-forced
1	15x30 (sd)	Dimensions	33750,00	450,00	1	☐	☑
	U 300		0,00	0,00	1	☐	☐

The rigidities are linearly added up for the calculation of the action-effects, whereby St37 is assumed for the steel section.



In case of biaxial effect of actions, the effective rigidity for an effect of actions around the z-axis is assumed:

$$EI_{eff} = \sum EI + A_i \cdot e^2$$

You can connect the sections at one or both sides. The influence of reinforcement sections connected only on one side on the supporting behaviour (torsion) is not considered.

For the stress analysis, the action-effects are distributed according to the rigidities. For steel reinforcement the analysis of the connecting means is included.

Due to the shrinkage properties of timber, additional stresses can be produced in the steel section that are not considered by the application.

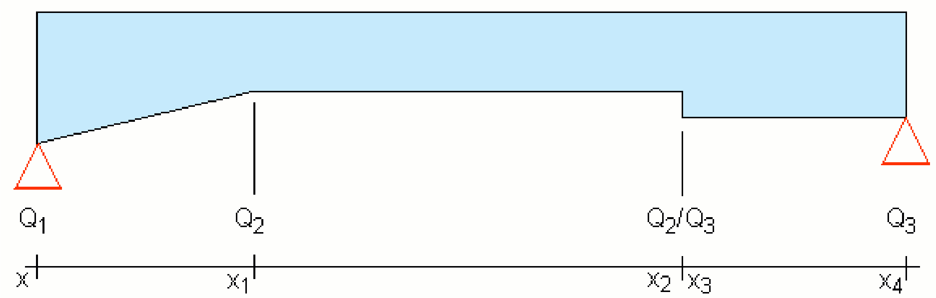
In the graphical representations of the action-effects, the total moment is shown without its distribution on the individual elements.

In the relevant specialist literature, only the simply supported reinforced beam is represented even though this construction is also frequently used with continuous beam. If you reinforce the inner span of a multi-span beam, for instance, you should comply with the following limiting condition:

A small area without reinforcement should be defined at the beginning and the end of the span unless the steel section is connected beyond the support line in a force-closed manner.

Cross sectional areas - several cross sections per span

If the cross section changes within a span, the individual cross section areas should be defined in the input table by their distance x to the left support.



x Distance of the change in cross section to the left support in [m].

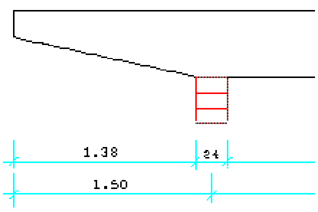
<u>Left cantilever:</u> The first (left) support is considered to be the reference point for the left cantilever (x), → *see the example below*.

When you enter a value x < L (span length), a new line is displayed for this span that allows you to define additional changes in cross section. The input for this span is terminated when you enter a value for x that is equal to L.

Enter changes in cross section continuously from the left to the right support.

For cross sectional jumps, enter separate cross sections to the left and the right face of the section, →*see the illustration above*.

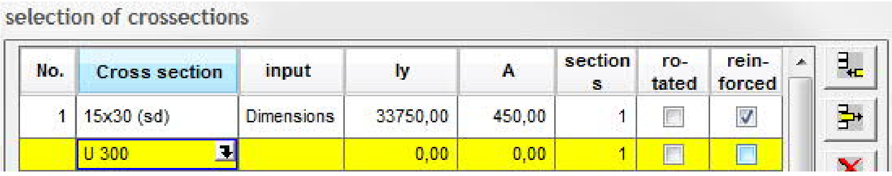
CsNo Specify always "1" for the first cross section number in the first span.



	span	L [m]	x [m]	CsNo	Crosssec.	bpo [cm]	Hpo [cm]	b0 [cm]	h0 [cm]
1	1	5,000	0,000	1	1			30,0	50,0
2			5,000	1	1			30,0	50,0
3	Ca li	1,500	0,000	1	1			30,0	50,0
4			1,500	2	1			30,0	30,0
5	Ca ri	0,000							

The dialog "Select/edit cross section"

You can access the dialog "Select/edit cross section" by pressing the <F5> key when the cursor is positioned in the column "Cross section" in the cross section selection dialog.



Ill.: the dialog for the cross section selection of wooden beams (with reinforcement)

Depending on the selected material, the following options are available to define a cross section:

- Frilo profile file
- Dimensions
- Structural values – I, A, W
- FRILO cross section applications Q1, Q2, Q3

See also the document Select - edit cross section_eng.pdf

Support

No — Consecutive support number

Type — For reinforced concrete or timber you define the type of support by entering or selecting of the corresponding number.

Reinforced concrete:

1	Knife
2	Brickwork wall
3	Concrete, direct (with minimum moment over continuous edges)
4	Concrete, indirect
5	Concrete, direct (without minimum moment over continuous edges)

Timber:

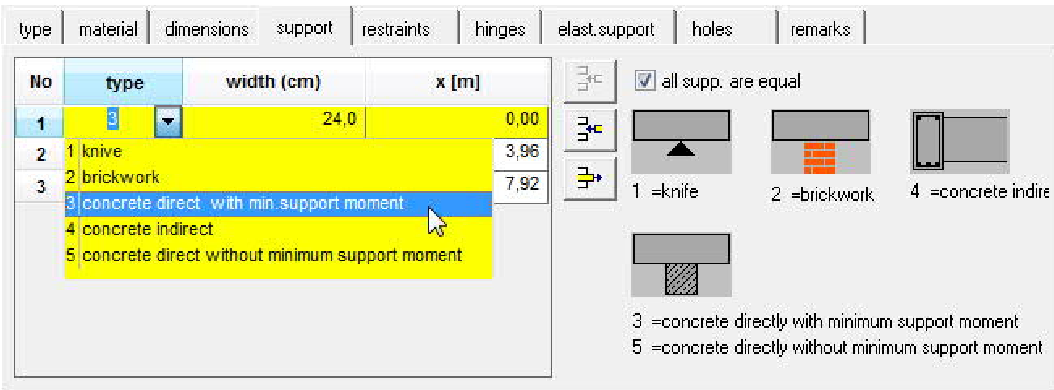
1 = direct,

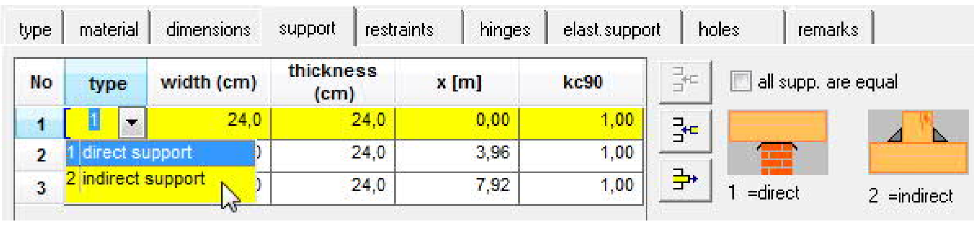
2 = indirect

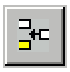
Width — Enter the width of the support in [cm] into this column.

Thickness — The support depth in [cm] (only with timber)

x — The distance of the support from the front edge of the beam is shown in column x. You can change the span width by editing this value. The entered standard loads are automatically converted to beam-related loads and remain therefore stationary.





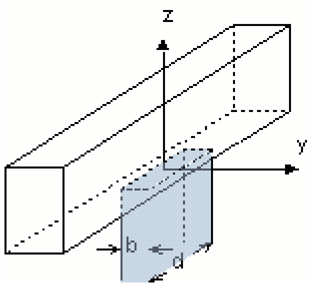


You can generate a three-span beam out of a two-span beam, for instance, by inserting a support.

If all supports are the same you can save input work by ticking the option "All supports are equal".

Restraints

You must make the following inputs to define real restraints:



Support: Number of the support

Type: Base of the column

- Hinged: enter "0"
- Restrained: enter "1"

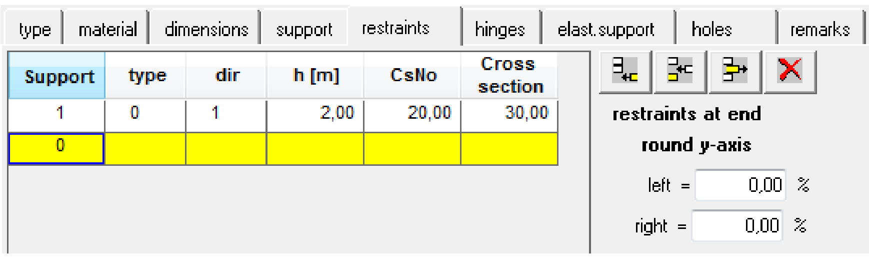
Dir: Direction of restraint:

- Bottom "1" (around the y-axis)
- Top "2"
- With biaxial effect of actions
- Front "3" (around the z-axis)
- Rear "4"

h: Height of the restrained column in [m]

b: Dimension of the column transverse to the beam axis [cm]

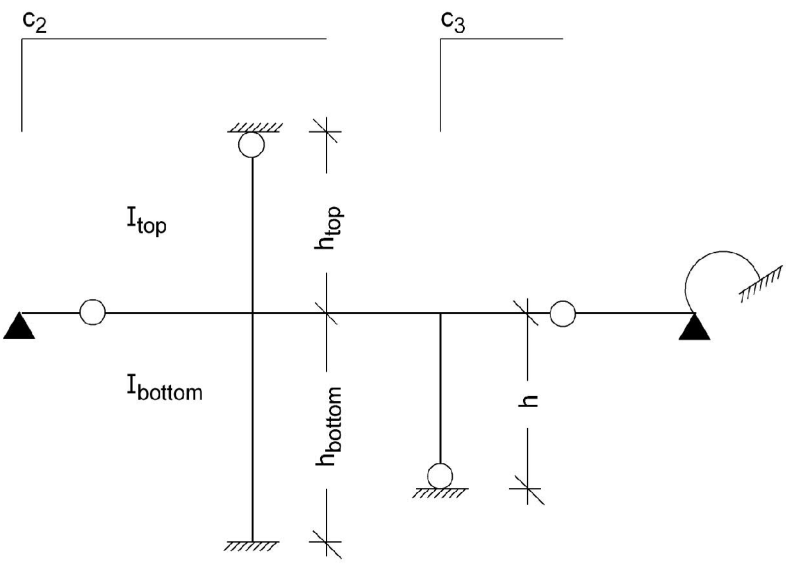
d: Dimension of the column in direction of the beam axis [cm]



If a column is added to the top an the bottom of a support, these columns should be entered one after the other with the same support number. The torsional springs are added up.

$$c_3 = \frac{3 \cdot E \cdot I}{h}$$

$$c_2 = \frac{3 \cdot E \cdot I_{top}}{h_{top}} + \frac{4 \cdot E \cdot I_{bottom}}{h_{bottom}}$$



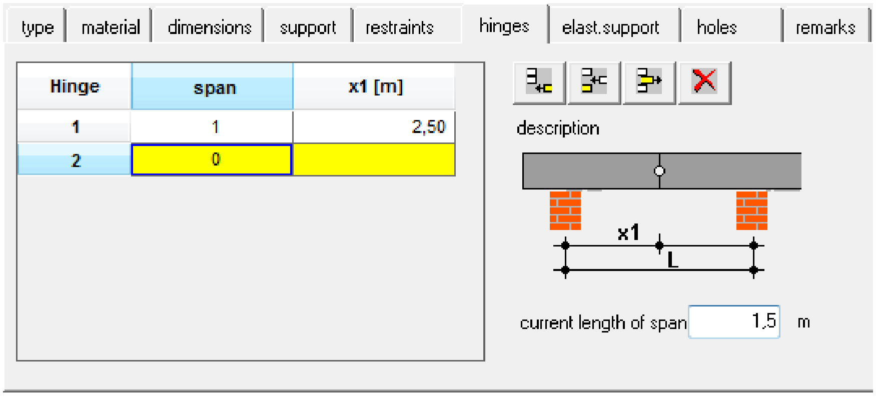
Restrained end supports

You can specify percentage restraints for end supports. They are treated by the application as torsional springs. You cannot define real restraints for the end supports in this case!

Percentage restraints represent an approximation solution. A precise description of the limit conditions requires the definition of torsional springs!

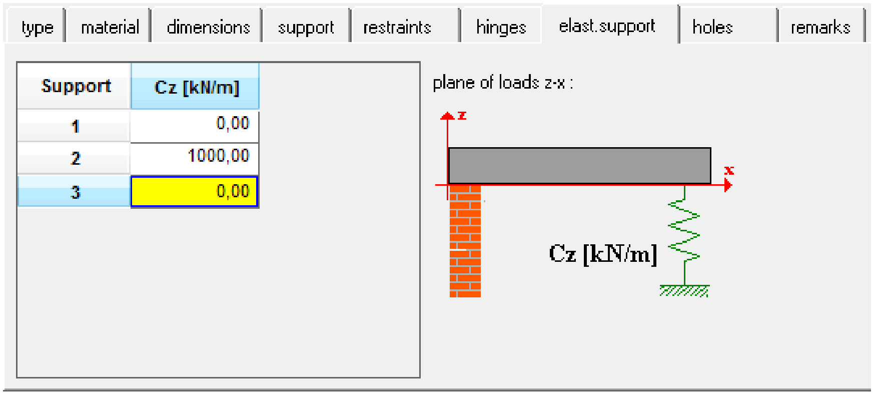
Hinges

Define bending hinges via the input of a span number and the distance "x1" to the left support.



Elastic support

You can calculate the spring rigidities by applying a unit force to the load-support component.



The spring rigidity results from the following equation: $C = \frac{\text{unitload}}{\text{deformation}}$

The following equation applies to a single bar: $C = E \cdot \frac{A}{h}$

Holes for reinforced concrete beams

You can define round and rectangular holes for reinforced concrete beams.

VK/FE	Reference point (Front edge) for the distance (see illustrations in the dialog) 1: left support in the span 2: front edge of the total beam
Reference	Reference point for the block-out (see illustrations in the dialog) 1: axis of the block-out 2: front edge of the block out
Span	If several spans have been defined, specify the no. of the span to which the block-out refers.
Type	1: rectangular block-out (additional dimensions: Dist., be, h, L) 2: circular block-out (additional dimensions: Dist., bo, D)
Dist	Distance either to the left support or the front edge of the beam in [m] depending on the defined front edge setting
Low edg.	Distance of the bottom edge of the beam to that of the block-out.
h/D	Height or diameter of the block-out
L	Length of the block-out (with rectangles). The length of the block out should be smaller than 1/3 of the span length.

If concentrated loads apply in the area of the block out, you must check the load distribution manually with slender remaining cross sections (beam support effect).

The analysis is performed according to Booklet 399 DAfStb[1]. The same analysis is used for round and rectangular holes.

The load case combinations for max/min M and max/min Q in the centre of the span are examined. The shear forces are distributed to the top and bottom flange according to the proportion of the rigidities. The following limit conditions apply:

For positive moments 70 % minimum are assigned to the top flange and 90 % maximum to the shear force. If the dimensions of the bottom flange are smaller than 8 cm, the full shear force is assigned to the top flange and the bottom flange acts only as a tension chord.

These conditions apply vice versa to negative moments.

Due to the shear forces, a bending analysis with tension/pressure and a shear analysis should be performed for the flanges. The assumed reinforcement location is d1 for the top flange and d2 for the bottom flange.

In addition, suspension reinforcement is required to the left and right of the block-out.

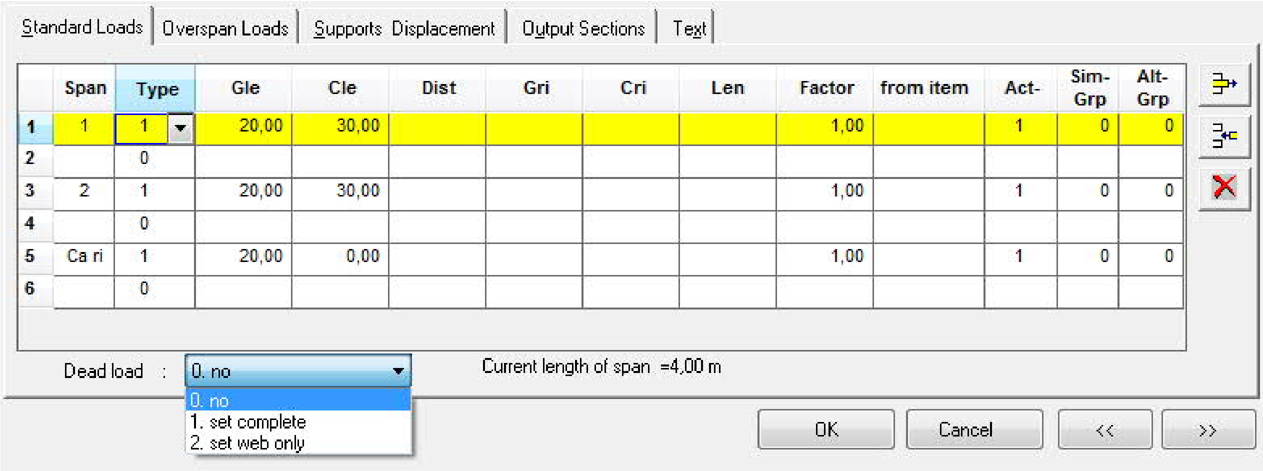
If the dimensions of the block comply with the condition L and h < d0/10, no analysis is performed.

Holes that begin or end in the area Dist. < d0 or Dist. < 0.10 · span length cannot be calculated using the method according to Booklet 399.

Loads

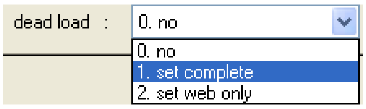
Select the option Load input in the main tree.

You can access the input tables of the various load types (standard, multi-span,...) by clicking on the corresponding tab.



Text You can enter additional text concerning the load in this section. These texts are put out together with the load data.

Self-weight

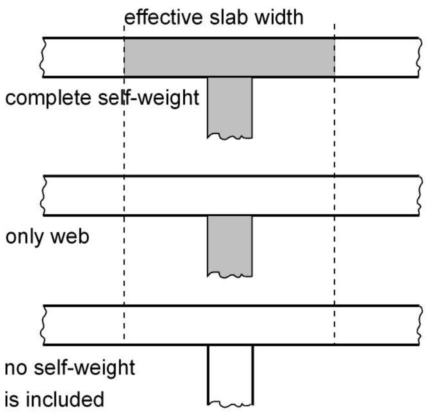


You can select among the following options:

No No self-weight is included

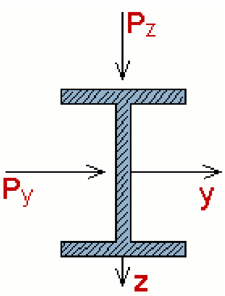
Complete The self-weight of the effective slab width and the web is completely included.

Only web Only the self-weight of the web is included



The loads are assumed to apply at the top edge of the beam. If applicable, you must perform additional analyses for appended loads.

Loads are defined as positive when they act in their normal position as shown for the cross section.



In case of biaxial effect of actions, you can define an angle for each load that determines the direction of action. The application calculates the load portions qz and qy.

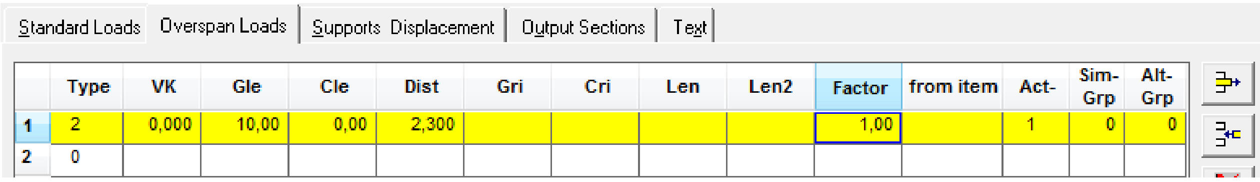
The following applies to the angle φ: φ = 0° load in z-direction (vertical)

φ = 90° load in y-direction (horizontal)

Standard and multi-span loads

Standard loads are entered spanwise. Six load types are distinguished. For multi-span loads, 10 load types (1..6 and 11..16) are available.

The loads are divided into g- and q-portions.



Ill.: input table for multi-span(Overspan) loads

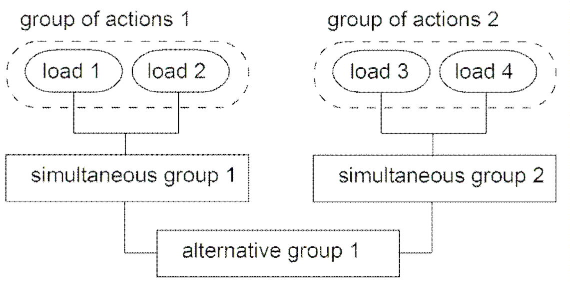
Type	Load type. Loads are entered by specifying their load type and the g- and q-portions → see the chapter Load types .
VK	(Front edge) This column is only displayed with multi-span loads. You can displace the reference point for the input of the loads by entering a value into this column. If a left cantilever was defined, VK=0 refers to the left beam front edge (beam end). If there is no left cantilever, VK=0 refers to the support centre. *Tip: You should enter test values and check their effect in the graphical representation to improve your understanding.*
Gle	Load ordinate for g left [kN , kN/m]
Qle	Load ordinate for q left [kN , kN/m]
Dist	With standard loads: Distance a from the left support [m]. If loads apply to the left cantilever, a refers to the distance to the left cantilever end. With multi-span loads: Distance a to the front edge + FE [m] → see the chapter Load types
Gri	Load ordinate for g2 right [kN , kN/m]
Qri	Load ordinate for q right [kN , kN/m]
Len	Load length b [m] → see the chapter Load types.
Len2	Load length c [m] → see the chapter Load types.
Factor	Factor for this load (e.g. for the affected widths)
Angle	Application angle (this column is only displayed with biaxial effects of actions) Input: 0° = vertical, 90° = horizontal.
From item	Indication of the load source (load transfer); it is included in the print-out. You can enter six characters maximum.
Ac-grp	Group of actions. Select a group from the list (living rooms, offices ...). You can also define new groups of actions via the menu option ▸ Edit ▸ Action groups. → See the chapter Groups of actions → See also the chapter "Design defaults - interpretation of the combination coefficients". Depending on the selected action group, corresponding Ψ coefficients are taken into account for the variable loads, if several different action groups have been selected.

Sim-grp Simultaneous group.
You can combine the loads that apply simultaneously in a simultaneous group (value > 0).
0 No combination, the loads apply separately.
1,2,... Loads that are assigned to the simultaneous group 1, for instance, apply simultaneously (one load case). This group is suitable for a load train, for instance, or negative and positive loads that apply simultaneously.

Alt-grp Alternative group.
Loads in an alternative group exclude each other, e.g. wind loads from different directions.
0 means: "this load is not in any alternative group)

See also Load groups

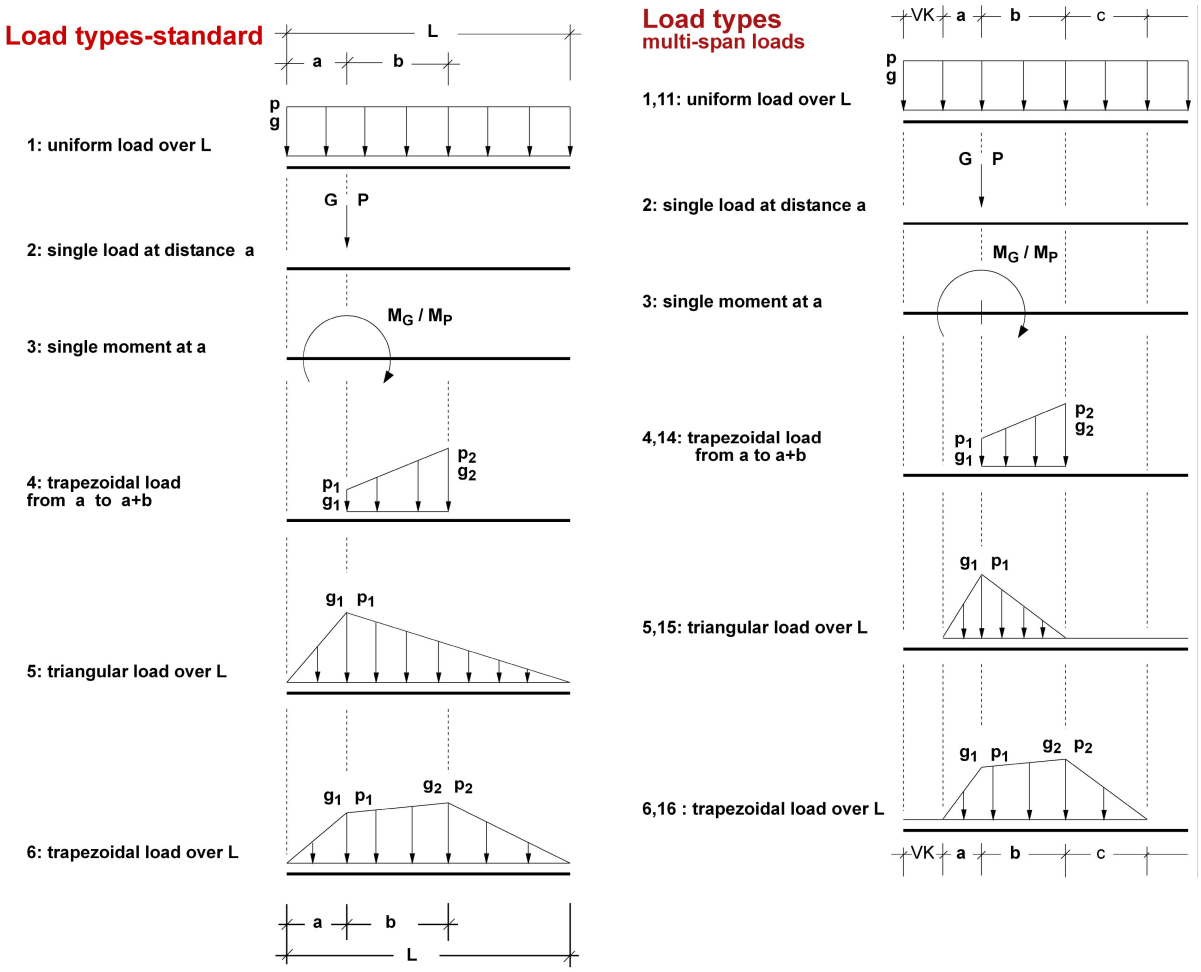
Fig .: Example of the functioning of alternative and simultaneous groups.



Note: *The specifications for actions groups, concurrent groups and alternative groups refer only to the variable loads (Q). Permanent-loads (G) always apply.*

Load types

The illustrations below make the input options for standard and multi-span loads clear.



Multi-span loads

Multi-span loads are defined via 10 different load types (1...6 and 11...16) that are divided into g- and q-portions. The reference point is the beam front edge (left beam end).

If the load type < 10, the q-portions of the loads are included spanwise.

With the load types 11, 14, 15, 16, the q-portions are not distributed over the spans but considered to apply as a whole. Load type 11 for instance allows you define snow loads applying to the whole system.

Column lowering

You can specify a displacement in [cm] for each column.

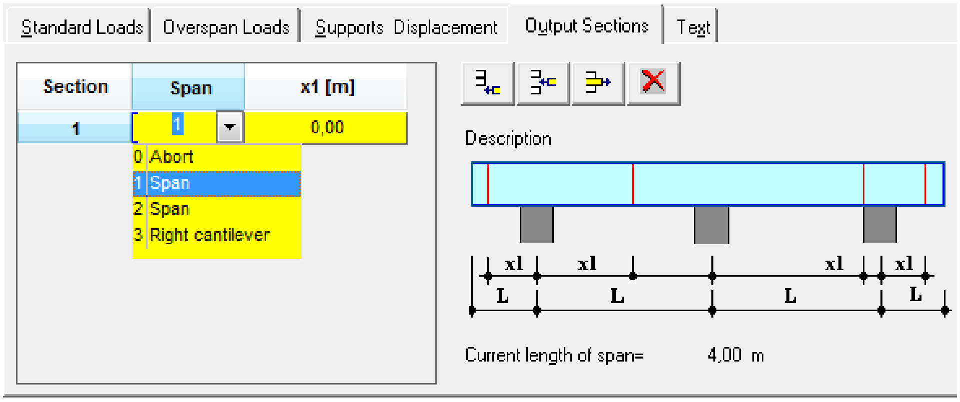
If the option "biaxial" is available, you can enter the column lowering separately for the y- and the z-direction. In the basic version, you can do this only for the z-direction.

Cross section optimization with steel and timber is not available for the column lowering load case.

Support:	Number of the support
fz:	Column lowering in z-direction [cm]
fy:	Column lowering in y-direction [cm] *(only available with biaxial effect of actions)*
Act grp:	Assignment to a group of actions.

Output sections

The application performs the design at all marked points. You can define sections to obtain additional output data.

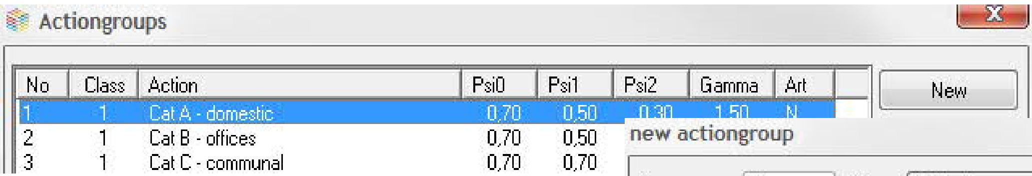


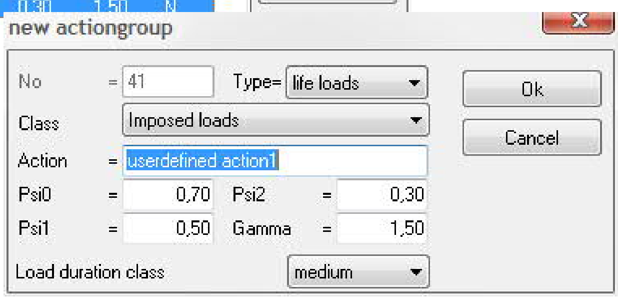
Define sections by entering or selecting a span number and the distance "x1" to the left support [m].

With left cantilevers, the value refers to the distance to the first (left) support.

Groups of actions

You can define groups of actions for the load assignment via the menu item ▸ Edit ▸ Action groups or the load input table by pressing the <F5> key in the column "Act grp".





New — Input dialog for the definition of a new group of actions.

Edit — To edit of a selected (user-defined) group of actions.

Delete — To delete the selected group of actions.

You cannot delete or edit the default groups of actions. You cannot delete user-defined groups of actions either after their definition has been confirmed with OK.

Psi0 to Psi2 — are the combination coefficients.

Art — N stands for normal actions, A for accidental actions.

Gamma — is the partial safety coefficient

A partial safety coefficient gamma is assigned to the loads for the strength analysis:

GammaG for the permanent portion,

GammaQ for the variable portion of the load and

GammaP for the pre-stressing.

The material is considered via a material-specific coefficient (GammaM).

Gammas is set = 1.0 for the serviceability analyses.

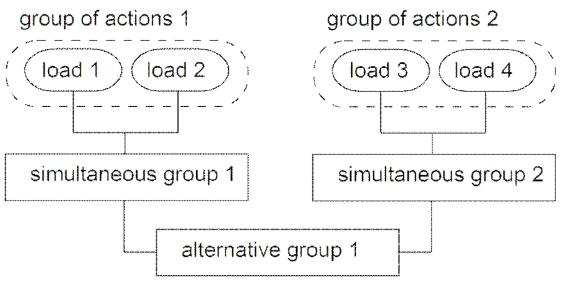
Load groups

The load grouping has only an effect on the variable loads.

Permanent loads are always considered.

You can assume loads from a single or several groups of actions as excluding each other (alternative) or applying together (combined). This method corresponds to the typical superposition load case.

Example of groups of actions and load groups in a single item



The loads 1 and 2 are assigned to the group of actions 1. Correspondingly, the loads 3 and 4 are assigned to the group of actions 2.

Load 1 and 2 are assumed to be wind loads in one direction that always apply together.

The loads 3 and 4 are wind loads in the opposite direction.

As wind can only act in one or the other direction, the both simultaneous groups 1 and 2 are assigned to the alternative group 1.

The effect is that either the simultaneous group 1 or 2 or none of both is considered depending on whether the loads become decisive for the design or not.

Biaxial effect of actions

Optional feature for
- steel and timber girders: DLT-HS2;
- reinforced concrete beams: DLT-SB2.

The optional module is not included within the scope of delivery of the DLT application and must be purchased separately.

You can define an angle for each load that determines the direction of application.

The following applies to the angle φ: $\varphi = 0°$ load in z-direction (vertical)

$\varphi = 90°$ load in y-direction (horizontal)

For all other angles, the effect of actions is internally spit into the components q_y and q_z.

Lowering of columns can be entered for each direction separately.

The structural system is assumed to be identical in both axes directions, i.e. the number of springs and the locations of the supports must be the same in both directions of the effect of action. If a support is missing, you can include a very small support spring in this direction, if required.

Hinges act as bending hinges around the y-axis as well as the z-axis.

You can define elastic supports, restraints in uprights and end restraints in each direction separately.

The minimum output of the results for the marked sections includes the maximum/minimum moments My with the pertaining variables Mz, Vz and Vy. My and Vz refer to the vertical and Mz and Vy to the horizontal effects of actions.

The specified maximum field moments may differ slightly from the real maximum values because they are not calculated explicitly. The action-effects at the n-part points as well as all load and cross section jumps are calculated instead.

Steel girder

The stresses and the utilization ratio Eta (η) are shown for all marked sections. The deflections are put out separately for each direction plus the resultant for each span. The specified utilization ratio refers to the resultant deflection.

Timber beam

The bending and shear stresses are calculated for all marked section and printed out in separate tables. In addition, the utilization ratio Eta (η) is indicated. The reduction of the concentrated loads near the support is not considered with a biaxial effect of actions.

For beams with constant cross sections, you can use the same design window as with uniaxial effects of actions. With multi-part cross sections, the rigidities and resistance values are linearly added up as with uniaxial effects of actions.

Reinforced concrete beam

The design of the biaxial reinforced concrete beam is performed analogously to the first-order analysis of a reinforced concrete column without consideration of the undesired eccentricity *ea*. A reduction of the moments over continuous edges is not considered. The longitudinal reinforcement is concentrated in the corners in this case. Compliance with the rules for minimum field moments and minimum moments over continuous edges is not checked.

Load import

Load import from other items

Important note! Only the forces to be transferred are filed/saved (but not the moments).

In the load input table, you can access these values for uniformly distributed loads (load type 1) and concentrated loads (load type 2): press the <F5> key when the cursor is positioned in the input field for loads (eg. column Gle). If results are available (after a calculation run has been performed), a selection box with the computed items is displayed.

If a biaxial calculation was performed for the selected item, you are prompted whether to import the support reactions for the y- or z-direction.

After having selected the item, a selection box with the transferrable loads of all supports is displayed.

When you select a row including "G" and "Pmax" these values are transferred to the input table. With concentrated loads, you must specify the distance of the load ("Dist" column) in addition.

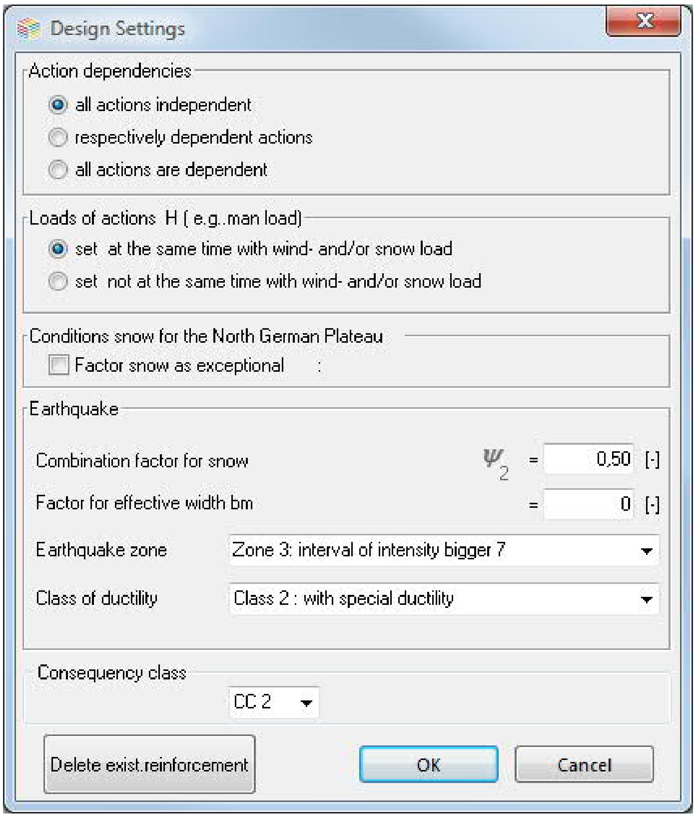
If the load value of "Pmin" is negative and its amount greater than "G", you must also transfer this load in a second step (lifting support analysis).

Calculation and design

Design defaults - Interpretation of the combination coefficients

‣ Main tree ‣ Design settings

According to the explanatory notes to the load standard by Professor Grünberg, there are different ways to interpret the relation between imposed and live loads depending on the source of the action. You can select the appropriate interpretation via the main tree item "Design defaults".



All imposed and live loads are independent

This option corresponds to the traditional concept that all variable loads can be freely combined with other loads whereby the guiding action and ψ factors are considered.

Imposed loads/live loads are dependent each

When you select this option, all imposed loads together and, analogously, all live loads together are considered as a single action each. This means that the loads of each category are additively (w/o combination coefficient) superimposed and the greatest ψ-factor of the relevant category is used when combining them with other variable loads.

All imposed and live loads are dependent

When you select this option, all imposed and live loads together are considered as a single action. This means that all loads are additively (w/o combination coefficient) superposed and the greatest ψ-factor of both categories is used when combining them with other variable loads.

Design defaults for actions of category H

You can select whether actions of category H should apply simultaneously with other variable actions.

Alternatively, you can make sure that actions of category H do not apply simultaneously with wind or snow loads.

This option has the following background.

EN 1991-1-1 (Amendment 1 of September 2009) item 3.3.2 stipulates that imposed loads on roofs need not be considered in combination with simultaneously applying snow and/or wind loads.

Combination coefficients for snow

According to the model list of technical construction regulations MLTB 09/2009, snow loads should be multiplied by the combination coefficient Ψ2 = 0.5 as specified by DIN 4249 equation (12) (in derogation of DIN 1055-100) in the calculation of the effective masses for earthquake loads. The user can select the combination coefficient at this point of the calculation.

Factor for the effectively involved width bm

In accordance with DIN 4149 (2005-04) 8.3.6.2 (3), the effectively involved width of beams should be assumed in relation to the connected columns and the slab thickness. The bm factor limits the effectively involved slab width to the web thickness plus factor *bm* times the slab thickness.

Earthquake zone as per DIN 4149 (2005-04)

The earthquake zone is included in the output text for information.

Ductility class as per DIN 4149 (2005-04)

The ductility class is used to check the reliability of the reinforcing steel.

Superposition

A g load case is generated from the defined loads and a q load case for each variable load in addition. Loads that were defined as being concurrent are summarized to one single load case.

Status lines are determined for the generated load cases and the loads are superimposed as simple or γ-fold loads. If several independent actions apply, the decisive action is determined for each internal force and each design value.

Superposition follows the rules of EN 1990. The fundamental combination rules are used for the superposition. If accidental actions have been defined, the combination rules of EN 1990 for the accidental design situation are used. If several accidental loads have been specified, only the decisive accidental load is taken into account. Exception: several accidental loads are member of the same concurrent group.

According to DIN EN 1991/NA, the value of the characteristic snow load may be increased by a factor in the accidental design situation, if the building is located in the Northern Lowlands of Germany, for instance. The option "Factor for accidental snow load" in the design defaults provides for this.

Reinforced concrete design

Bending design as per Eurocode EN 1992

Verification of the load-bearing capacity

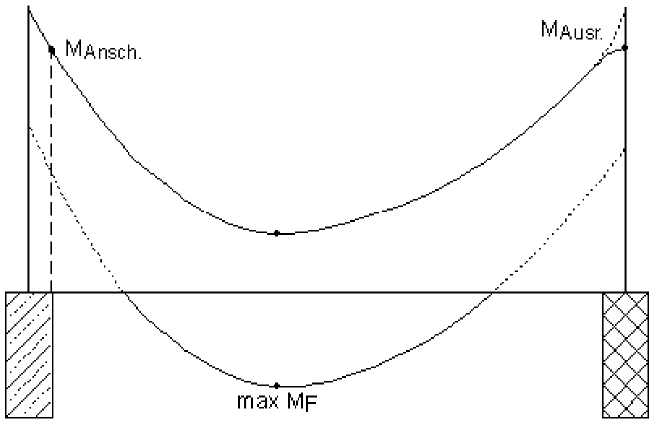
The internal forces for the verification in the ultimate limit state are determined in accordance with DIN EN 1992-1. The superposition is based on the specifications of Eurocode EN 1990. The design is performed in accordance with EN 1992-1-1, para. 6.

The bottom reinforcement is calculated for the maximum midspan moments. If the maximum midspan moment is negative, the required upper reinforcement is put out in addition.

The design is also performed at cross-section jumps and previously defined sections. The maximum positive moments are included in this calculation. If no positive moments result, the negative moments with greatest values are included

If there are also negative moments in a design section, the upper reinforcement is calculated too.

The punching negative moments are not verified separately because the upper reinforcement layer should be graded in line with the resisting tensile force diagram in these cases. A design for the negative midspan moment with the smallest value is not helpful in the example shown.



Biaxial bending design

Both the biaxial design of reinforced concrete and that of steel, timber and aluminium are available as add-on modules.

In the biaxial bending design, a rectangular cross section with longitudinal reinforcement concentrated in the corners is used. If a T-beam was defined, the internal forces are calculated with the help of the beam stiffness values and the design is performed on a rectangular cross section. This rectangular cross section comprises the web and the corresponding slab part immediately above or underneath the web in this case.

The rules for minimum span moments and minimum moments over continuous edges are not considered in the biaxial bending design. Moments over continuous edges are not reduced.

Biaxial shear design

In the biaxial shear design, the shear verifications are performed separately for each direction. We recommend multiplying the greater result value referring to the stirrups by 1.4 and assign the result to the stirrup reinforcement.

In the design dialog, the greater of both values is offered without consideration of the factor 1.4.

Redistribution in accordance with Eurocode EN 1992-1-1 para. 5.5

In accordance with EN 1992-1-1 para. 5.5, the permissible redistribution of moments for high-ductile steel is limited to 30 % for concretes of strength classes <= C50/60 and to 20 % for concretes of higher quality and lightweight concretes. Moreover, the standard makes sure that the span moment associated to the reduced moment over continuous edges does not exceed the maximum existing span moment or the maximum resisting span moment, if a higher reinforcement was selected.

If steel with normal ductility is used, EN 1992-1-1 specifies a limit of 20 %. For high-ductile steel, the limit is 30 %.

In addition, the permissible redistribution is limited by the referenced compression zone height x_d/d after the redistribution.

Redistribution in accordance with para. 5.6.3 - Verification of the rotation capacity

If standard concrete and high-ductile reinforcing steel have been selected, a simplified verification of the plastic rotation capacity is performed.

Only the inner columns are considered. The exterior columns are not examined, even if the end span moment results from a restraint and not from a cantilever

Initial values are:

- moment over continuous edges
- associated span moments on the left and right with consideration to the combination coefficients
- maximum span moments on the left and right
- existing span moments on the left and right
- existing support reinforcement on the left and right
- resisting span moments on the left and right with existing As

The moment over continuous edges should be reduced to the preset size Delta M, which is normally 30 %.

The following limits should be observed in this connection:

- The existing bottom reinforcement is able to bear the associated span moment. If no reinforcement was defined, the maximum span moment represents this limit.
- The permissible plastic rotation as per EN 1992-1-1, 11.5.1 is complied with.
- The limitation of the compression zone height as per EN 1992-1-1, 5.5 is complied with.

The existing rotation is calculated by integration of the curvatures in the adjacent spans.

The verification is based on iteration. The reinforcement above the column is increased until the verification condition is satisfied.

If the existing reinforcement has caused the increase of the span moment (existing bottom reinforcement is greater than the required span reinforcement), the shear forces must be adjusted accordingly. The shear force acting on the examined column is not reduced. However, the shear forces acting on the adjacent columns are increased, if necessary.

Serviceability verifications

1. Limitation of the crack width in accordance with EN 1992-1-1, 7.3

   The requirements referring to the width of the cracks are determined by the requirement class resulting from the exposure class as per EN 1992-1-1, 7.1N. Reinforced concrete components are verified under the quasi-permanent load combination. The verification is performed in accordance with EN 1992-1-1, 7.3.

   If reinforcement was defined, the actually existing reinforcement is considered in the verification, otherwise the statically required reinforcement.

2. Deformation verifications

   In addition to the calculation of deformations in state I with simple loads and gross concrete cross-section, deformations in state I and II can be calculated in accordance with EN 1992-1-1. In the latter case, the calculation takes the quasi-permanent load combination and the actually existing reinforcement into account. Therefore, considerably lower deformations result in state I than in the verification mentioned first. The influence of creep and shrinkage is taken into account via the creep and shrinkage parameters defined in the materials section. The resulting deflection values are put out separately.

   Subsequently, the deformations in state II are calculated by iteration using the effective deformations. To be able to do this, each span is subdivided into 10 elements in the system modelling process. In this process, again creep and shrinkage are taken into account in the calculation.

Anchorage of the end supports

Verification of the anchorage of the end supports

with $V_{ed,VK}$ at the front edge (VK) of the support

with $V_{ed,x}$ at the cross-section decisive for the shear force verification

and F_1: portions of concentrated loads applying close to the support in the front edge area VK < a < x with

$F_1 = F_1 \cdot a_1/z_1 + F_2 \cdot a_2/z_1 + F_3$ .... as described in booklet 430 by the German Committee for Reinforced Concrete DAfStB

The verification of the anchorage at the end support can optionally be performed with 50% and 0.5 · cotTheta or 50 % or 100 % of the shear force at the front edge of the end supports. The relevant options are available in the menu ▸ Options ▸ DLT settings ▸ Reinforced concrete:

1. 50 % option and 0.5 · cotTheta (Default)

$F_{sd} = \max \{ V_{ed,VK} / 2 \ , V_{ed,VK} \cdot \text{CotTheta} / 2 \}$

2. 50 % option

$F_{sd} = \max \{ V_{ed,VK} / 2 \ , V_{ed,x} \cdot \text{CotTheta} / 2 + F_1 \}$

3. (no option checked):

$F_{sd} = \max \{ V_{ed,VK} \ , V_{ed,x} \cdot \text{CotTheta} / 2 + F_1 \}$

Shear design in accordance with EN 1992

The verification of the shear force bearing capacity is based on Eurocode EN 1992-1-1, 6.2. The shear force is calculated in accordance with para. 6.2.1.

The analysis of the shear resistance is based on a truss model with compressive concrete struts and steel ties (stirrups). The minimum stirrup requirements result from the flattest possible strut inclination. This depends among other factors from the loading on the cross section in relation to the crack friction force of the concrete and the longitudinal stresses in the cross section. A flatter inclination increases the strut force, which is limited by the concrete class and the smallest width of the cross section. Moreover, the forces in the tension chord increase too, causing an increase of the offset dimension.

When entering the material parameters, the user can select whether the strut inclination should be variable or constant in the verification.
With constant inclination, the inclination angle is freely selectable according to EN 1992-1-1, 6.2.3 (6.7N).

With direct support, the verification is performed at a distance *d* from the edge of the support in accordance with EN 1992-1-1, 6.2.

Concentrated loads applying at the distance $a \leq 2.5 \cdot d$ from the support edge are reduced with direct support in accordance with EN 1992-1-1 6.2.2 (6).

In shear force areas, where the limit $V_{Rd,max}$ is exceeded at the face of the support, a corresponding note is put out.

For the design, the calculated required longitudinal reinforcement is taken as existing reinforcement. If the option Curtail span reinforcement is not checked in the material section, the longitudinal reinforcement determined by the maximum midspan moment is considered constant over the total length.

If the user has defined longitudinal reinforcement with the help of the reinforcement module, the actually existing longitudinal reinforcement is used.

In the area of haunches, shear force is increased or decreased in accordance with EN 1992-1-1, 6.2.1.

A redistribution of the shear forces as a result of a plastic calculation is taken into account if it has an unfavourable effect.

The user can select whether the inner lever arm should be calculated with a variable z value determined in the bending design or a constant value $z = 0.9\,d$. In addition, the software application takes the limit $z \leq d - 2\,c_{nom}$ into account. c_{nom} depends on the concrete cover and with beams, 6 mm are added for stirrups.

The inclination of the strut should be limited.

Shear force transmission in joints of reinforced concrete slabs

The verifications are performed in accordance with Eurocode EN 1992-1-1, 6.2.5.

The roughness grades "smooth" and "rough" are automatically taken into account, if the option reinforced concrete slab was checked in the menu Material/Element floor. The roughness grades "very smooth" and "interlocked" are not taken into account in the verification. A more accurate verification is available in the B2 application for the design of reinforced concrete.

The verification examines the conditions at the edge of the support.

Verification of construction joints of tee beams

If the cross section type "Tee beam" was selected for the reinforced concrete beam (input dialog "Dimensions"), an additional dialog is available per mouse click in the "CsNo" column. A symmetrical T-beam with schematic representation of the pre-cast components is displayed. In the verification of the construction joint as per Eurocode EN 1992-2-2, the joint width is calculated as follows:

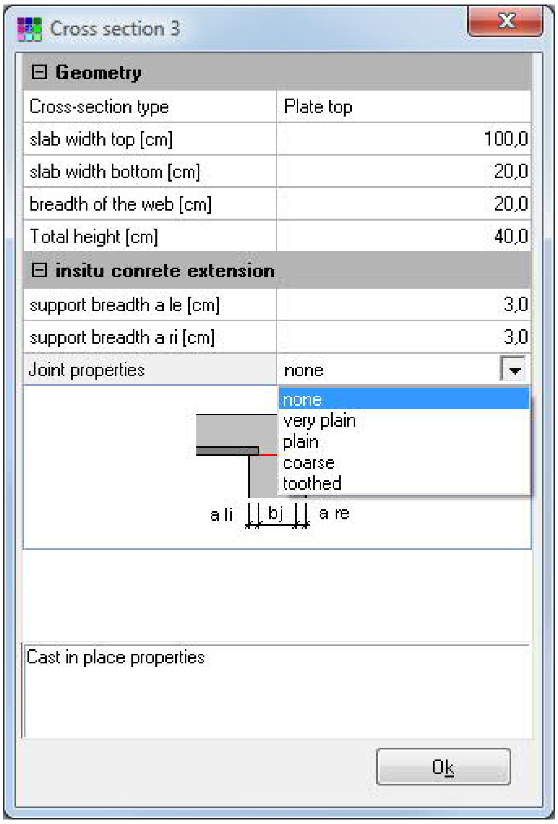
$bj = b0 - a_{le} - a_{ri}$

The L-beam is not represented explicitly. For the verification, the T-beam is used and the corresponding support length is set to 0.

The support length of the pre-cast components is taken into account in the calculation of the greatest stirrup length.

For the representation of the reinforcement layout in the section, an element thickness of 4 cm is assumed.

At the bottom of the T-beam, the support length is normally set to 0.



Connection of compression flanges in accordance with Eurocode

Shear joint verification

Supporting slab of the T-beams under compression should be connected to the web in a shear-proof manner. The shear-proof connection is ensured by struts and ties.

The struts and ties are shown in the model below. In the verification, the strut bearing capacity of the concrete and the tie bearing capacity of the transverse reinforcement are checked.

In accordance with EN 1992-1-1, evidence is to be established that the longitudinal shear force V_{Ed} does not exceed the resisting bearing capacities $V_{Rd,max}$ und $V_{Rd,sy}$.

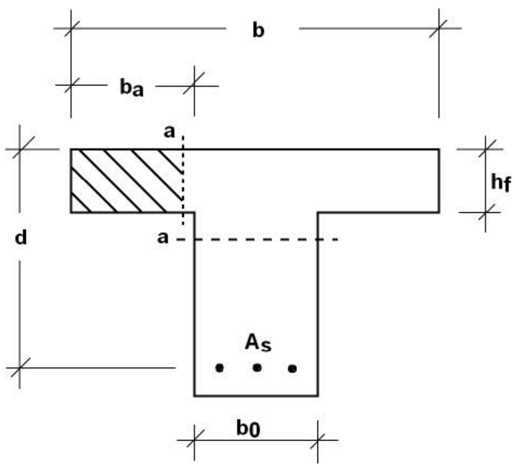
$$V_{Ed} \leq V_{Rd,max}$$
$$V_{Ed} \leq V_{Rd,sy}$$
$$V_{Ed} = \Delta F_d$$

The software application sets V_{Ed} equal to $V_{Rd,sy}$. After rearrangement of the equation, the required reinforcement results.

ΔF_d is the difference of the longitudinal forces acting in a flange section on one side over the length a_v. The length a_v describes a section where the longitudinal shear force is assumed as being constant. The length of the section should not be greater than half of the distance of the moment zero point to the maximum moment value. The software application calculates the average moment zero point from the individual load combinations in accordance with the moment limiting line.

If higher concentrated loads apply, the section length should be limited by their shear force jumps in addition. This is not implemented in the software.

The DLT application calculates the longitudinal force difference ΔF_d to provide for the situation where the forces of a compression flange ΔF_{cd} are needed, only compression flanges are handled in this software application.



$$\Delta F_{cd} = \frac{M_{Ed}}{z} \cdot \frac{F_{ca}}{F_{cd}}$$

M_{Ed} design moment

z internal lever arm

F_{ca} concrete compression force in the adjacent flange

F_{cd} total concrete compression force

The location of the zero line of the slab expansion is $x \leq h_f$

$$\Delta F_{cd} = \frac{M_{Ed}}{z} \cdot \frac{A_{sa}}{A_s} = \frac{M_{Ed}}{z} \cdot \frac{b_a}{b}$$

Shear and transverse bending

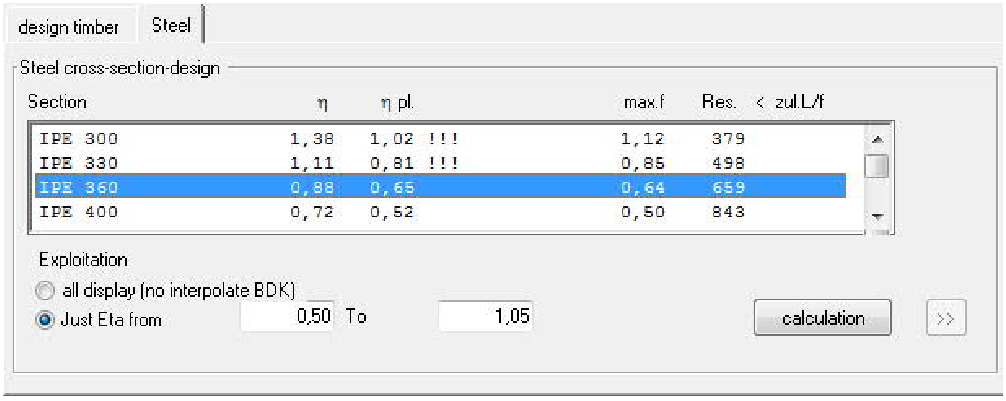
EN 1992-1-1, 6.2.5 (5) describes how to proceed if shear and transverse bending occur simultaneously.

Steel design

The "Design" option (in the main tree) displays a list for single-span or multi-span beams showing the utilization ratio ETA (η) for the profile sections of the selected series.

Prerequisites for the availability of this window:

- multi-span beams must have a constant cross section
- the profile section was selected from the FRILO profile selection file (the ARBED series is not enabled in this selection).
- no elastic supports
- no load case "settlement of supports"
- if the user defines a range via the option "Eta from/to", the resistance against lateral torsional buckling is verified if corresponding supports have been defined. If the user checks the "Display all" option, the entire profile section series is designed without lateral torsional buckling verification in order to save computing time.



If η > *1*, a warning in the form of three exclamation marks is displayed (!!!).

The following values are displayed in addition:

max. f maximum deflection for the selected profile section

L / f quotient length/maximum deflection

If L/f exceeds the permissible maximum length, a corresponding note is displayed.

Steel design results

For steel beams, the maximum moments, the associated axial, shear and comparison stresses are indicated for each span. In addition, the highest utilization eta (η) and the required I are put out.

The property values are specified at all decisive points from the front edge to the rear edge of the beam:

The total loading M and Q is specified for multi-piece cross sections. For the stress calculation of the individual profile section, these values are divided by the number of shapes.

In order to control the deflection the cantilevers, the inner spans must be reinforced under certain conditions. The pertaining options are only available if the cross section is constant. With variable cross sections, deflection control of the cantilevers can be achieved by reinforcing the spans and/or the cantilevers.

If biaxial loading applies, the deflections are specified for both directions (y/z).

When the cross-section dimensions of the profile section are known (e.g. from the Frilo profile selection file), the edge stresses, the highest shear stress and the comparison stress are verified in each cross section. The cross-sectional properties of most of the common rolled shapes are stored in the Frilo profile selection file.

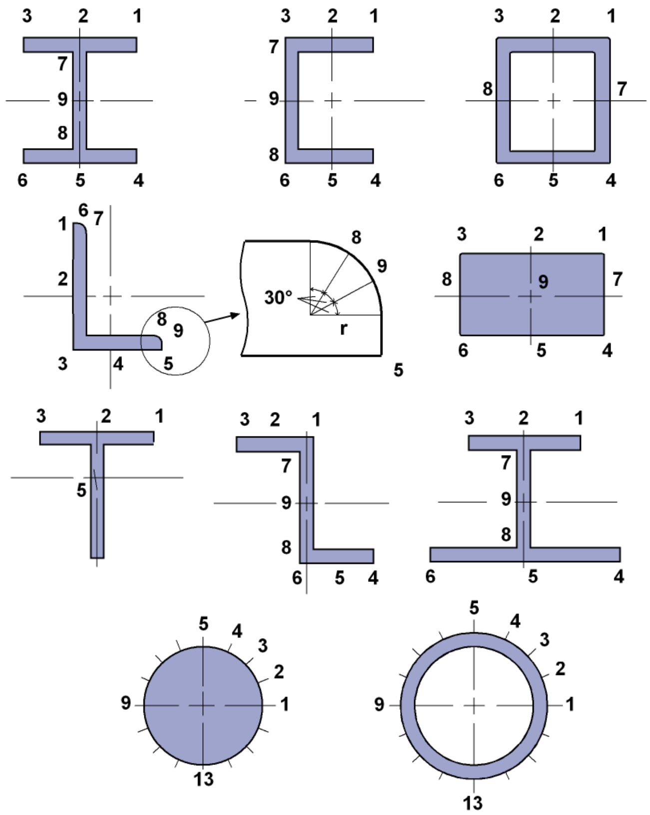


Figure: Profile section overview (Frilo profile selection file) / output points for stresses

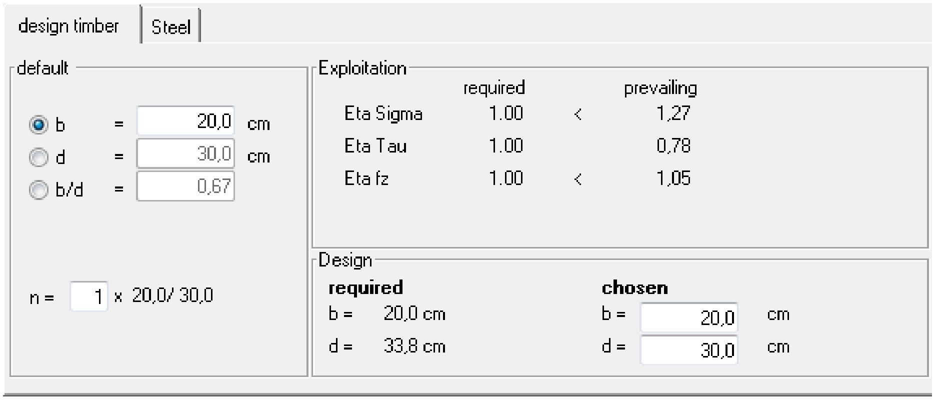
If you use the cross sectional properties from the Frilo profile selection file, all parameters for the accurate stress resistance verification are available.

Timber design

The user can vary the cross section of timber as with steel according to the requirements of the design.

Prerequisites:
- multi-span beams must have a constant cross section
- no elastic supports
- no load case "settlement of supports"

The cross section optimization window specifies the utilization ratio eta (separately for σ_B, τ and deflection f) for the selected shape.



Cross section optimization

In order to achieve the required properties, adjust the dimensions as follows:

1. Via the pre-setting options you can decide which of the values you like to use:
   b or *d* or the ratio *b/d*.
2. Multi-piece cross sections can be defined in the "Multi-piece" section. You can adjust the pitch between 1 and 3 via the arrow keys <↑↓>.
3. Depending on your settings, sufficiently great dimensions are displayed in the "Required" section.
4. Finally important are the values specified in the "Selected" input fields. You can increase/decrease these values via the <↑↓ > keys. The corresponding eta values are adjusted accordingly.

Timber design results

With timber beams, the maximum moments and axial stresses are specified on top/bottom for each span. The same results are put out at cross section jumps and changes in the cross section.

At the supports, the highest moment over continuous edges is calculated and the axial stresses on top/bottom are specified.

The following values are put out in addition:

- transversal forces on the left and right of the column with associated shear stresses
- maximum deflection
- *preset I* and *req. I* in accordance with limits of the deflection

For the deflection control of the cantilevers, the inner spans must be reinforced under certain conditions. The pertaining options are only available if the cross section is constant.

The bearing stresses are calculated in accordance with EN 1995 and put out with a corresponding note, if a support width was defined.

If biaxial loading applies, the existing deflection w_y, w_z, as well as the decisive deformation w and the permissible deformation perm. w are put out.

Reinforced profiles are marked accordingly in the output.

Timber design as per Eurocode EN 1995

General

In the design as per EN 1995, the permanent, transient, accidental, quasi-permanent and characteristic design situations are considered. For multi-span beams, the partial safety factor for the permanent actions varies in each span.

In the ultimate limit state, the following design situations are examined in accordance with EN 1990:

- Permanent and transient design situation as per para. 6.4.3.2
- Accidental design situation as per para. 6.4.3.3
- Actions due to earthquake as per para 6.4.3.4

In the serviceability limit state, the following superpositions are performed as per para 6.5.3:

- Characteristic combination
- Quasi-permanent combination

Moisture action is considered via the usage class. The usage period determines the modification factor k_{mod} as per EN 1995-1-1, table 3.1.

Usage class 1	closed and heated buildings, humidity < 65 %, equilibrium moisture content < 12 %
Usage class 2	open building with roofing, humidity < 85 %, equilibrium moisture content < 20 %
Usage class 3	building exposed to weathering, humidity > 65 %, equilibrium moisture content > 20 %

For load case combinations with actions of various load action periods, the shortest action period is decisive.

Verifications in the ultimate limit state

Redistribution in accordance with EN 1995-1-1, para. 6.1.6

The verifications are based on the equations 6.11 and 6.12.

For rectangular cross sections of solid wood and glued laminated timber, k_{red} = 0.7, otherwise k_{red} = 1.0.

In accordance with EN 1995, equations 3.1 and 3.2, the characteristic strength of solid timber beams may be increased by h ≤ 150 mm and of glued laminated timber by h ≤ 600 mm.

Redistribution in accordance with EN 1995-1-1, para. 6.1.7

The verification of the resistance to uniaxial shear is based on equation 6.13 and to biaxial shear on DIN EN 195-1-1, equation NA.54.

According to EN 1995, the strength values for timber and derived timber materials are taken from other standards (EN 338 for solid timber, EN 1994 for laminated timber). The strength values in the specified tables refer to the uncracked condition. Therefore, EN 1995-1-1 proposes the consideration of an effective width in the shear stress verification as per equation 6.13a. This produces increased shear utilization in the software application. The handling of the crack coefficient k_{cr} depends strongly on the selected National Annex and differing utilizations may result in the shear stress resistance verification.

If uniaxial loading applies almost exclusively to the biaxial continuous beam system, the uniaxial and biaxial utilization may differ considerably from each other due to the squaring.

In accordance with DIN EN 1995-1-1, NDP for 6.1.7 (2), the design value of the shearing strength is increased by 30 % in areas that have a minimum distance from the front face of the timber of at least 1.50 m. This applies only to coniferous timber.

Concentrated loads in the proximity of the support can be disregarded according to EN 1995-1-1, para. 6.1.7 (3).

Bearing stress in accordance with EN 1995-1-1, para. 6.1.5

The verifications are based on the equations 6.3 and 6.4.

The verification is included in the output of the results, when the user has defined a corresponding support or confirmed the default values in the menu ▸ System settings ▸ Supports. In accordance with para. 6.1.5 (1), the width of the effective area under lateral pressure is increased automatically by 2 times 30 mm for intermediate supports and 30 mm for end supports. Lateral pressure coefficients for bearing pressure l ≤ 400 mm are not implemented in the current version of the software application.

The lateral pressure coefficient can be edited by the user by pressing F5-key in kc90 column of the support definition table.

Verifications with the help of the equivalent bar method

Lateral torsional buckling of flexural members in accordance with EN 1995-1-1, para. 6.3.3

In the menu ▸ System settings ▸ Material ▸ Top flange support, the user can select the spacing of the supports against tilting of the beam. The selected spacing is used as effective length l_{eff} of the equivalent member.

Axial force cannot be defined in DLT and is not taken into account in the stability verifications.

The verifications are based on the equations 6.30 to 6.35.

Verifications in the serviceability limit state

Vibration resistance verification in accordance with Eurocode EN 01/01/1995 para. 7.3

According to para. 7.3.3. Residential floors, the vibration resistance verification is performed as follows:

- Calculation of mass, stiffness and resonance frequency
- Calculation and limitation of the deflection caused by a concentrated load as per equation 7.3.
- Limitation of the vibration velocity due to unit impulse as per equation 7.9 and due to footfall in accordance with "Mohr - bauen mit Holz" [11/2001, p. 29].
- Limitation of the acceleration in accordance with "Mohr - bauen mit holz" [11/2001 p.29]

Deformations as per EN 1995-1-1, para. 2.2.3

Averaging of the deformation coefficient k_{def} is not allowed, but a load portion is considered for each participating action. The deformations are calculated with the help of averaged stiffness values E_{mean} and G_{mean}.

The deformation portions are indicated in the output as follows:

$w_{g,inst}$	characteristic deformation by permanent loads
$w_{g,fin}$	deformation by permanent loads with influence of creep
$w_{q,inst}$	deformation by variable loads, infrequent situation
$w_{g,fin}$	deformation by variable loads with influence of creep

Verifications in accordance with EN 1995-1-1, para. 7.2

After the separate calculations of the permanent and variable deformation portions in a characteristic or quasi-permanent design situation, the resulting deformations are compared to the limits w_{inst} and w_{fin}, each time with consideration of the creep portion and without creep.

Recommendations for the limiting values of the deflection in the initial and final state are given in table 7.2. The user can display and edit the deflection limits via the "perm. w" button. For cantilevers, the permissible values are doubled.

Calculation, results

For span moments, the location of the maximum moment (x_0) with the moment (M_f) and the associated moments and shear forces at x=0 (M_{le}, V_{le}) and x=L (M_{ri}, V_{ri}) are put out for each span.

The maximum span moments are moments with a positive sign. If the span moments are negative, select the output option "minimum internal forces" or change the direction of action of the load.

For the moments over continuous edges, the moments on the left (M_{le}) and right (M_{ri}) of the column, the associated shear forces (V_{le}, V_{ri}) and the maximum and minimum bearing forces (max V, min V) are put out.

For the full load and self-weight load cases, only the moments over continuous edges and the corresponding shear and bearing forces are put out.

The moment limiting line shows the maximum and minimum moment behaviour in the tenpart points.

In the bearing forces table, the maximum and minimum bearing forces and those from permanent and variable loads as well as from full load are represented.

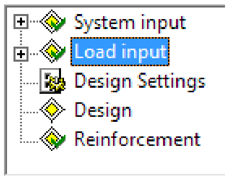
The maximum and minimum deflections are put out for each span.

If applicable, also end moments of the vertical supporting members are put out.

DLT-BEW - Reinforcement layout for DLT (optional)

DLT BEW is optionally available as an additional module - see price list.

The reinforcement drawing is generated via the menu item "Design" (main tree).



Design

Access the input dialog "Resisting tensile and shear force coverage":

Double-click on the menu item "Design" in the main tree

or select ▸ Edit ▸ Design

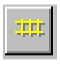
Accessing the reinforcement layout

▸ Edit ▸ Reinforcement...

or

double-click on the menu item "Reinforcement" in the main tree

or

click on the icon  in the tool bar.

→ See the chapter [Operating the reinforcement layout]

Application options of the reinforcement layout

You can develop the reinforcement layout of continuous beams interactively on the screen. The development is based on the resisting tensile force diagram and the shear distribution diagram.

With slabs, only the diagram of the resisting tensile force for round bar reinforcement is analyzed. Fabric reinforcement is not available yet.

Restrictions

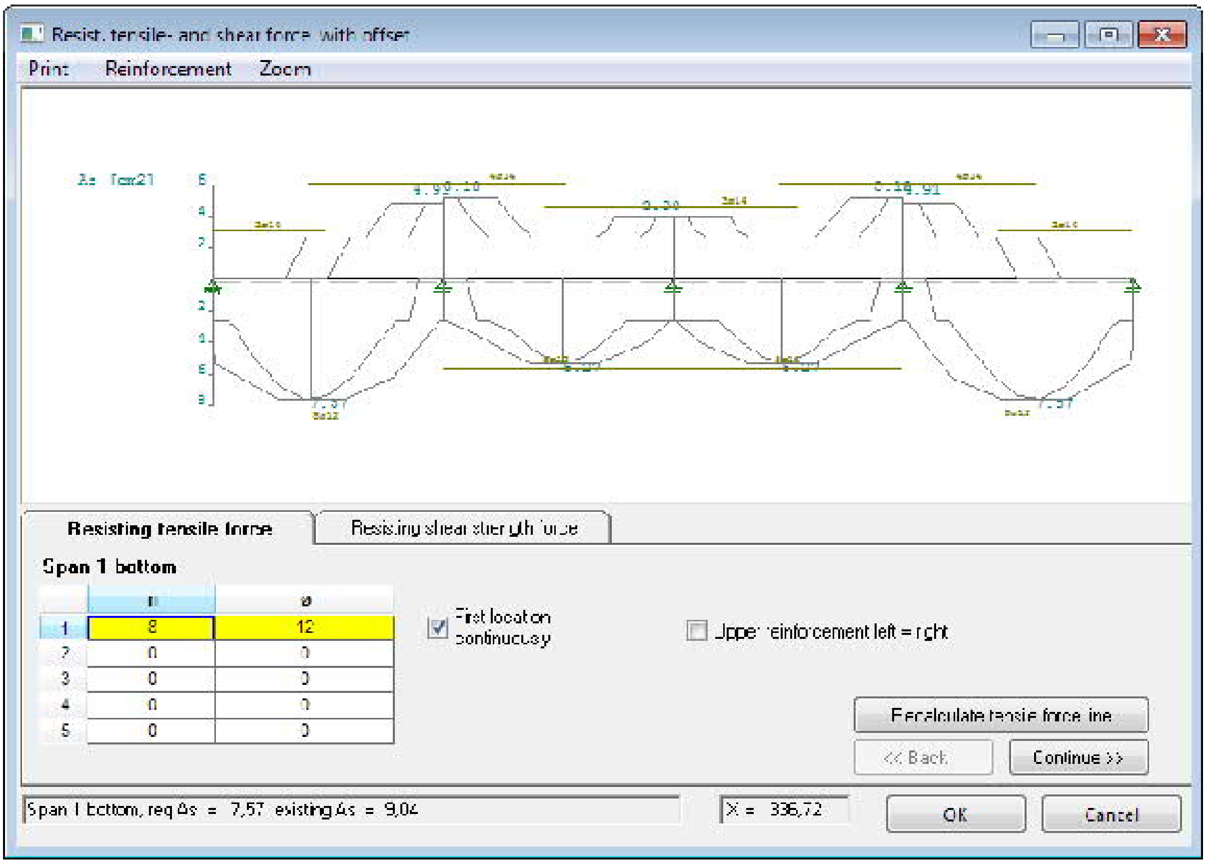
Suspension reinforcement is not considered.

The anchorage of longitudinal bars is not analyzed at haunches and jumps.

Resisting tensile and shear force coverage

Menu items

Print	The option allows you to print the currently displayed graphic or both graphics (resisting tensile and shear force coverage).
Reinforcement	The submenu items of this option allow you to regenerate either the complete reinforcement or only the span reinforcement or only the shear reinforcement.
Zoom	This option allows you to display a section of the graphic zoomed out with the mouse. "Previous" displays the section previously zoomed out, "Restore" displays the entire graphic again.



Resisting tensile force coverage

The resisting tensile force diagram is shown with an offset of v = 1.0 h on the screen. The required reinforcement is shown in the span and at the columns.

Select the reinforcement successively for all spans and columns. The current values of the “required As” and the “existing As” are shown for each definition step together with the span or column number.

In the input table, specify the number and diameter of the reinforcement bars for the individual locations. The buttons "Back" and "Continue" allow you to toggle between the individual span and column entries.

You can define up to 5 entries for each span or column. The first entry is always interpreted as the hanger bar. You should always enter two bars!

After each input, a small line is drawn in the resisting tensile force diagram showing how much As is covered.

n	Quantity
∅	Diameter

You must define 2 hanger bars minimum to the left and right of end supports.

Options

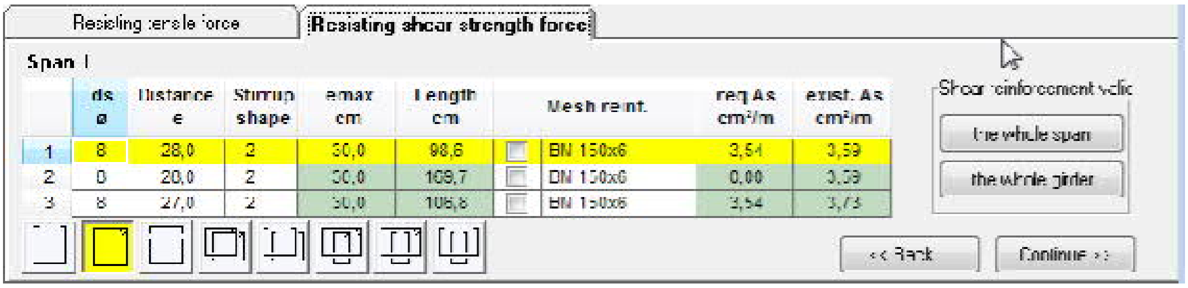
The options may facilitate the input in some cases.

First location continuously	The reinforcement bars of the first location continue over all spans.
Upper reinforcement left=right	
	When you enter the left supporting reinforcement, for instance, and tick this option, the value is automatically set also for the right supporting reinforcement.
Recalculate tensile force line	Activate this option to recalculate the resisting tensile force diagram.

Resisting shear force coverage

The shear reinforcement is defined for each span separately. The span is subdivided into three sections (left column, span, right column).

The buttons "Back" and "Continue" allow you to toggle between the individual span entries.



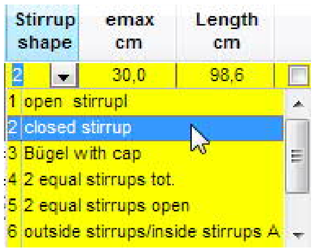
Double-shear and quadruple-shear stirrups and stirrup and fabric meshes are available as reinforcement

To define the stirrup reinforcement, enter the diameter ds ∅, the stirrup spacing e as well as the type of stirrup (single-, double-, quadruple-shear, etc) for each area.

Stirrup shape

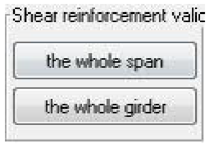
In this column, you can define the shape of the stirrup via a selection list (click on ▼ to expand it).

Alternatively, you can click on one of the stirrup icons below the table to select the stirrup shape for the currently selected row or enter the corresponding code directly (e.g. 3 for stirrup with cap).

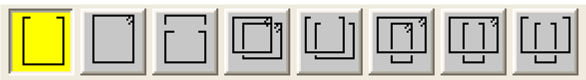


Resisting shear force coverage applies to span/beam

By clicking on one of the two right icons you can quickly define that the resisting shear force coverage specified in the current row applies to the entire span or beam (the specified values are copied to the corresponding fields in the other rows).



Selection of graphic elements



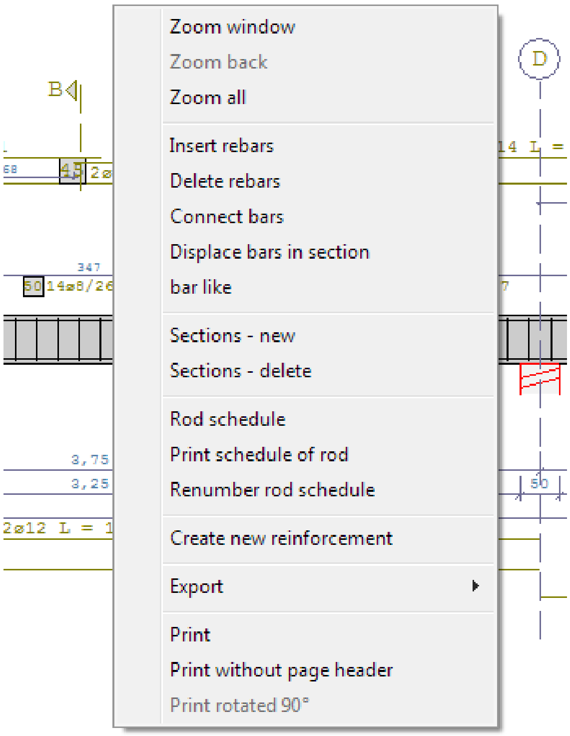
You can also click on an area in the graphic (e.g. span 2) to display the relevant input fields.

Representation of As

The required and existing As are shown

Operating options of the reinforcement layout

- The menu is accessible via the right mouse button. Click right on the graphic interface to display the context-sensitive menu (see right illustration).
- Double-click on a bar to display the input dialog for the bar properties of the relevant bar.
- Double-click on a stirrup to display a dialog to edit the stirrup geometry.



Bar properties

Options can be ticked on the on the left and right. You can edit the diameter and the bar length. When editing the bar length, specify which point of the bar should be kept constant, the left end (standard), the right end or the centre. If the left end of the bar should be kept constant, a modification of the length is applied to the right end.

You can displace a selected bar. Enter a value with a negative sign to displace the bar to the left by the specified value (in [cm], otherwise, the bar is displaced to the right.

Stirrup geometry

You can edit the stirrup section lengths in this dialog.

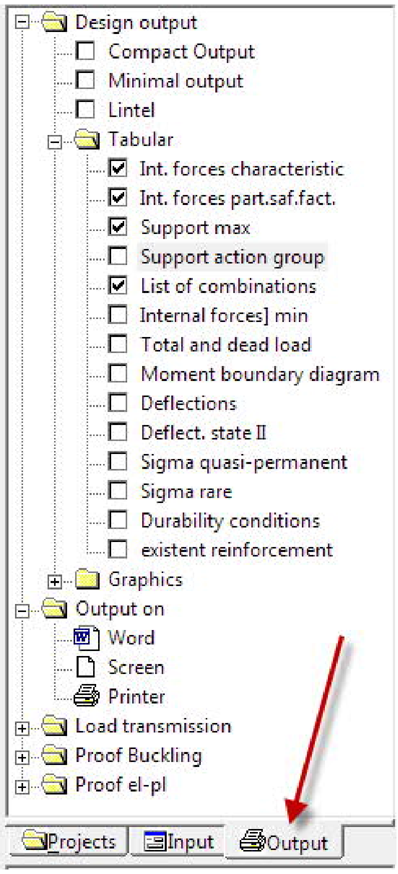
Context-sensitive menu

Zoom	This option allows you to increase or reduce the size of the graphical reinforcement representation.
Connect bars	You can unite two bars lying next to each other. The diameter of the thicker bar is adopted for the new bar. Click successively on both bars. Tip: Enlarge the screen section with the help of the zoom function for this operation.
Bars like...	This function allows you to transfer the properties of a reinforcement bar to another one per mouse click. Click first on the bar with the properties you like to transfer and subsequently on the bar the properties should be transferred to. Check the properties per double-click on the latter bar.
Sections - new	This option allows you to generate an additional section.
Sections - delete	Click on a section figure to delete it.
Rod schedule	This option allows you to generate a list including the item, the quantity (no), the diameter (D), the length and the weight per piece (G[kg]) as well as the total weight. You can print this list via the option "Print schedule of rod".
Print schedule of rod	This option allows you to print the table generated via the option "Rod schedule".
Create new reinforcement	This options allows you to regenerate the reinforcement.
Export	You can export the reinforcement graphic to a DXF or WMF file.
Print	Output of the graphical representation on a printer.
Print w/o header	Output of the graphical representation on a printer without the default header.

Output

Output of the system data, results and graphical representations on the screen or printer.

To display the options of the output menu, click on the output tab in the main tree.



Output profile

You can select the tables and graphics that you want to put out by ticking the corresponding options. The output profile helps to reduce the output scope to the requirements of the user.

Output in Word

This option launches the text editor MS Word if it is installed on the respective computer and imports the output data from an RTF file. You can adjust the layout of your data via the formatting functions of Word.

Output on the screen

You can check the output data in a text window before printing. To do this double-click on the option "Screen".

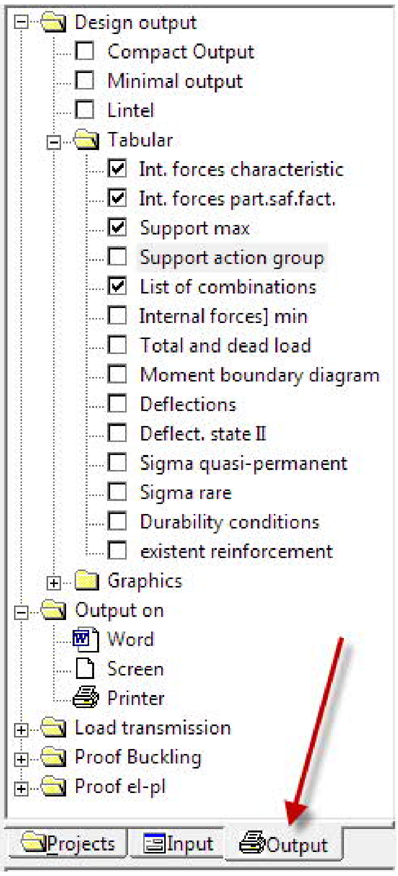
You can select among various graphic representations such as the moment and shear force diagram, deflections, the As line, the resisting shear force coverage etc. via the tool bar.

Output on the printer

A double-click on the option printer starts the output on the default printer.

Tip: *A page view is available via >> File >> Print preview.*

Output profile



The option "output design" (in the main tree) allows you to determine the scope of the output.

Tick the options you want to include.

Tip: You can save the current output profile by clicking right on the option "output profile" and activating the button "Save output profile".

Compact Output — This option allows you to put out the loads, action-effects and support forces in a condensed form. The design results are put out in the "normal" tables.

Lintel — This option allows you to put out a simple reinforced concrete beam in a condensed form on a DIN A4 sheet including the reinforcement layout and the steel shapes. The reinforcement proposition is based on the minimum diameter that was defined in the material section. A change of the minimum diameter becomes effective only after you have deliberately regenerated the reinforcement (see the chapter Operating the reinforcement layout)

Tabular outputs

Action-effects — Maximum moments and shear forces, optionally minimum moments and shear forces.

Full load and self-weight — Moments over continuous edges with relevant shear and support forces.

Moment boundary diagr. — Maximum and minimum moment behaviour in the tenpart points [kNm].

Deflections — The elastic maximum/minimum deflections are shown in [cm] for each span including the distance x in [cm] to the left support or the first support if a left cantilever was defined.

Fire design — see chapter Timber beam

Proof of vibration — see chapter Timber beam

Deflect. state II — Output of the deformations in state II
→ see also Material selection)

Exis. reinforcement — Output of the existing reinforcement.

Output of shear stresses

You can put out the shear stresses for slabs even if shear reinforcement is not required.

Output of the shear reinforcement

Output of the shear stresses with slabs.

Graphics

System graphic — Graphical representation of the system and the loads including dimensions.

M-diagram	Graphical representation of the moment diagram including the maximum values [kNm].
V-diagram	Graphical representation of the shear force diagram including the maximum values [kNm].
Deflections	Limiting line of the deflections.
As-diagram	Graphical representation of the system including As values [cm^2].
Tau diagram	Shear stresses limiting line.

With biaxial effect of actions:

Action-effects	max M_y and maxM_z including the associated action-effects
max/min My	max/min M_y, incl. (M_z, V_z, V_y) [kNm, kN]
max/min Mz	max/min M_z, incl. (M_y, V_z, V_y) [kNm, kN]
max/min Vz	max/min Q_z, incl. (M_y, M_z, V_y) [kNm, kN]
max/min Vy	max/min Q_y, incl. (M_y, M_z, V_z) [kNm, kN]
Full load/self-weight	M_y, M_z, V_y, V_z
Deflections	f_z, f_y, f_{res}, in [cm], utilization ratio.
Market points	Maximum values at the cross sections and shear force jumps

Output scope of tables

Input data	System, loads, comments (texts concerning the system and the loads).
Structural calc. results	Action-effects, support forces, table of the end moments of upright members, if applicable, all data ticked in the output profile.
Design results	For reinforced concrete: span, supporting and shear reinforcement.
	For steel: stress analysis, deflections.
	For timber: stress analysis, deflections, support stresses.

Output of results

For the span moments, the location of the maximum moment in combination with the relevant moment is put out for each span including the associated moments and shear forces at x=0 and x=L.

The maximum span moments are moments with a positive sign.

For the moments over continuous edges the moment to the left and the right of the column, the corresponding shear forces and the maximum and minimum support force are put out.

For the load cases full load and self-weight only the moments over continuous edges and the corresponding shear and support forces are put out.

The moment limiting line reflects the maximum and minimum of the moment behaviour in the tenpart points.

For beams with connected uprights, a table including the maximum/minimum face moments and the associated N is put out.

If uprights are connected to the top and bottom of a support, the entire support force for both uprights is put out as the normal force. If required, it must be split subsequently by the user while taking account of the constructive limiting conditions.

A table is put out for the support forces that includes a list of the forces to be transferred.

The maximum and minimum deflections are put out for each span.

Column load transfer

The calculated support forces and head moments can directly be transferred to the applications

- B5 - Reinforced Concrete Column and

- HO1/HO1+ - Timber Column,

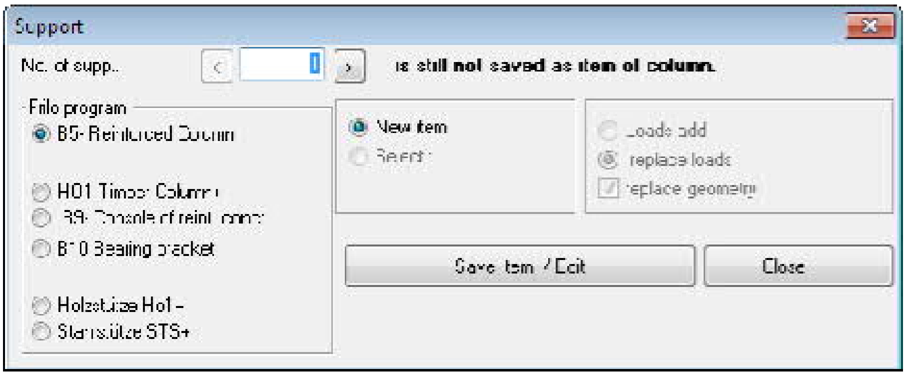
- B9 - Reinforced concrete corbel of column or

- B10 - Beam with dapped end

- STS+ Single-span Steel Column

to perform an analysis:

An intermediate dialog prompts you to select the desired support number. Via the option "Edit/save item", a subordinated application with the corresponding data and dimensions of the selected member and the calculated head loads is launched.



Lateral torsional buckling analysis

The lateral torsional buckling analysis is performed directly for double-T cross sections. For the analysis, the spans are cut out and the load cases max Mf, min Ma and min Mb are examined by the calculation module of the application BTII Lateral Torsional Buckling. The top flange is supported in the middle of the span or in the tripart points depending on the specifications in the material section. The analysis is only performed for beams exposed to uniaxial effects of actions.

For beams exposed to biaxial effects of actions and beams with other cross sections and more complex supports, you can transfer the system and loading data via this menu item to the BTII application.

El-pl analysis

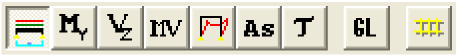
For steel girders, a general stress analysis according to EN 1993-1-1 is performed.

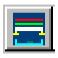
In addition, you can benefit from the enhanced analysis options of the ST7 application for the design. In this case, all cross sections and all decisive action-effects in the relevant sections are transferred to the application ST7 Structural Safety Analysis for Steel. You can check them with various analysis formulas in this application. See ST7 documentation.

Application-specific icons

In addition to the standard icons, each application offers application-specific functions via additional icons and tool bars.

In these applications, the following icons are available in addition to the standard ones:





System view: displays the entered system on the screen.

Moment diagram [kNm].

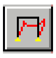
Shear force diagram [kN].

Representation of the moments and shear force behaviour together on the screen (EN2).

Shear force behaviour in the vertical plane with standards based on the partial safety concept.

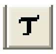
Moment behaviour in the horizontal plane.

Shear force behaviour in the horizontal plane with standards based on the partial safety concept.



Deformations.

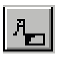
As diagram for reinforced concrete beams. The resisting tensile force diagram is shown with an offset of v=1.0 h on the screen The required reinforcement is shown in the span and at the columns. A third of the maximum field reinforcement is applied to the end supports and a fourth to the inner columns.



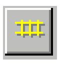
Tau, resisting shear force coverage

The 3D construction graphic (OpenGL) provides for a rendered (three-dimensional) representation of the system (illuminated volume model), that is suitable for verification purposes. You can easily check the locations of cross sections.

Simple reinforced concrete beam, put out as a lintel in the reinforcement drawing.



Holes



Reinforcement layout