

# Continuous Concrete Beam BTM+

## Table of contents

|                                                |    |
|------------------------------------------------|----|
| Application options                            | 2  |
| Data entry                                     | 3  |
| General operating instructions                 | 3  |
| Basic parameters                               | 5  |
| Structural System                              | 7  |
| Spans/segments                                 | 9  |
| Supports                                       | 10 |
| Joints                                         | 11 |
| Cross-sections                                 | 11 |
| Composite joints                               | 12 |
| Cut-outs (Holes)                               | 12 |
| Output sections                                | 13 |
| Loads                                          | 14 |
| Design                                         | 16 |
| Reinforcement Layout (optional add-on BTM-BEW) | 18 |
| Resisting tensile force coverage               | 19 |
| Shear force coverage                           | 20 |
| Reinforcement graphic                          | 21 |
| Output                                         | 22 |
| Output scope, calculation, results             | 22 |
| Load transfer                                  | 23 |

## Basic documentation - overview

In addition to the individual program manuals, you can find basic explanations on how to operate the programs on our homepage [www.frilo.eu](http://www.frilo.eu) in the Download area (Manuals).

*Tip: To go back in the PDF - e. g. to a link to another chapter/document - use the key combination "ALT" + arrow key "left".*

## Application options

The BTM+ program is used to calculate single-span and multi-span reinforced concrete beams. A cantilever beam can also be selected as a special case.

The cross-sections can differ from each other and can have a haunch. Single-sided slab beams are also possible. The spans can be divided into segments, also pinned joints can be defined.

The superposition and the design are performed automatically.

The data can be entered in tables or via the graphical user interface.

### Available standards

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



### Assistant (Wizard)

You can enter the data necessary to define a simple basic system in an assistant. Subsequently, this basic system can be easily modified and supplemented via the interactive Graphical User Interface (GUI).

### Supports

You can define supports in the z-direction as well as for torsion about the y-axis - with biaxial action also in Y-direction/around the Z-axis. In each case, you can optionally define rigid supports or enter a spring value. A column settlement can be pre-set for the individual supports. Alternatively, you can also have the spring values of a single column that you can define underneath and/or above the beam calculated by the program and then use them for the beam calculation.

### Loads

Load types: uniformly distributed, trapezoidal, triangular, concentrated loads and concentrated moment. Entered loads can be converted into area loads by activating the option "[per girder](#)" for the design and by defining a beam spacing.

### Interfaces to other programs

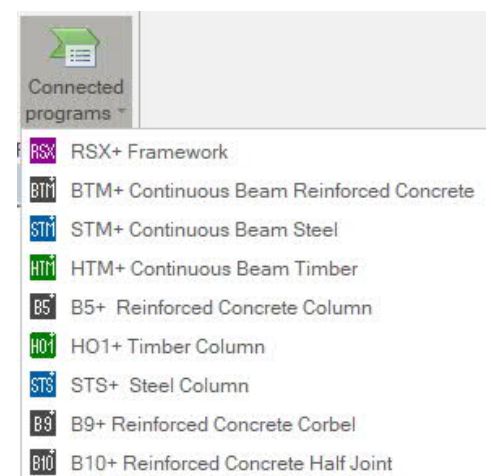
- Framework [RSX+](#) (alternative calculation)
- Continuous Beam Steel [STM+](#) (alternative calculation)
- Continuous Beam Timber [HTM+](#) (alternative calculation)
- [Transfer](#) of the support reactions to the Column programs [B5+](#), [STS+](#) and [HO1+](#) as well as to the Corbel programs [B9+](#) and [B10+](#).

### Add-on modules

BTM-2 Biaxial design of reinforced concrete beams. The calculation is based on one reinforcement placement per corner for the ULS and SLS.

BTM-BEW [Reinforcement Layout](#) is optionally available as BTM-BEW add-on.

The new reinforcement layout has been implemented in the program in a first preliminary version. Rectangular beams with or without cut-outs, as well as slabs, are available. Other cross-sections or beam types are constantly being added and further developed.



## Data entry

### General operating instructions

#### Assistant (Wizard)

The [assistant](#) is launched automatically when you start the program.

You can enter quickly the most important key figures of the structural system in the displayed window. These values can be edited subsequently in the input section or on the [Graphical User Interface \(GUI\)](#).

Self-defined items can be imported as templates. To save them as a template go to ▶ File ▶ Save as ▶ and select the option "Use as template".

Data entry via the assistant:

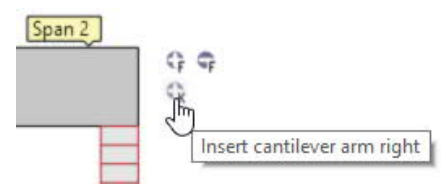
- Loading/action (uni- or [biaxial](#))
- Concrete qualities and steel grades
- Precast component
- Specification of the modulus of elasticity
- Number of spans (or optionally, just a cantilever)
- Span length
- Cross-section (for a rectangular cross-section, plate beams/one-sided plate beams can be defined in the cross-section table)
- Beam spacing
- Permanent line load
- Variable line load and type of action

*Tip: Via ▶ File ▶ Settings you can change some basic settings concerning the wizard, the colour scheme or the units of measurement.*

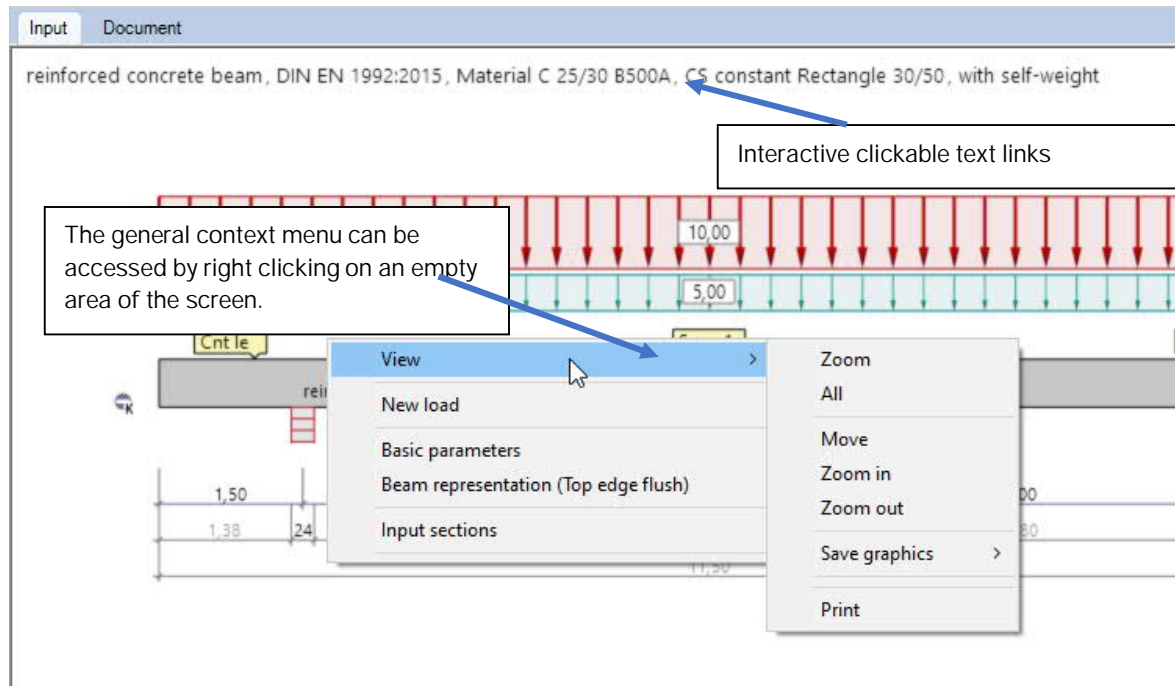
#### Graphical user interface (GUI)

The graphical user interface is structured in such a way that all entered data can be accessed directly in the graphics window. Dimensions or load values, for instance, can be directly clicked and changed. Other data-entry options are accessible via the general context menu (right click on an empty area in the GUI) or via the [context menus](#) of the individual objects (span, support, load ...) or via the interactive texts on top left. Spans and cantilevers can be added/deleted using the +/- icons on the right and left.

Moreover, you can move supports or loads that do not extend over the full length of the beam with the mouse or by entering a coordinate value.



See also [Graphical User Interface \(GUI\)](#) in the Basic Operating Instructions.



### Interactive dimensional chains

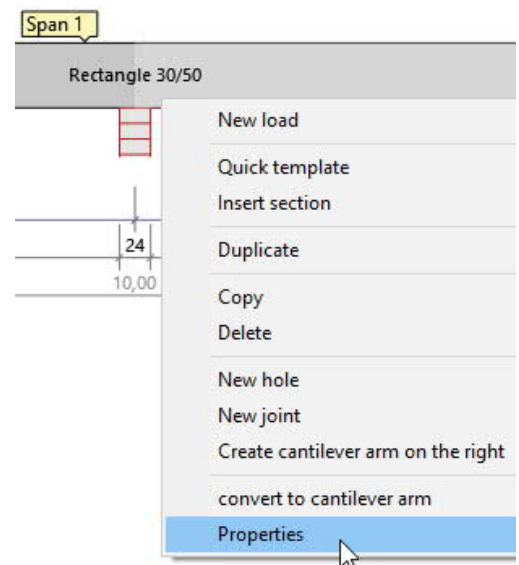
As in all Plus programs, the dimensional values are editable also in BTM+ and can be changed directly on the graphic screen.

*Tip: You can change the span length also by moving a support. To do this, click on the support using the left mouse button and move the support while holding down the mouse button.*

### Context menu

For each object (span, support, load, etc.) the appropriate context functions are available (figure on the right, context menu of a span). These functions can be displayed via the right mouse button and are, hence the name, matched to the selected object.

A general context menu is displayed when no object is selected. In this menu, you will find functions such as view functions, stability or sections that are not relating to a particular graphical object.



### Interactive texts

The texts displayed in the top left section of the GUI are interactive as in all PLUS programs and can be clicked on. This allows you to display dialogs in the GUI, which are otherwise only accessible via the menu on the left. See also the Basic [Operating Instructions](#).

## Basic parameters

### Standard and safety concept

|                          |                                                                                                                                                                                                                                                                         |
|--------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| Beam type                | Reinforced Concrete Beam or Reinforced Concrete Slab                                                                                                                                                                                                                    |
| Action effect            | Uni- or biaxial ( <a href="#">add-on</a> BTM-2)                                                                                                                                                                                                                         |
| Norm (Standard)          | Definition of the design standard and its national annex.                                                                                                                                                                                                               |
| accidental snow          | When you check this option, snow loads are considered as accidental action in addition to the common design situations.                                                                                                                                                 |
| Load factor for snow (A) | This factor is used to take the accidental snow load related to its characteristic value into account. You can freely specify it (check option to enter the value) or have it automatically calculated by the program.                                                  |
| $\psi_2$ for crane loads | determines the combination coefficient $\psi_2$ for crane loads (ratio of permanent share to total crane load).                                                                                                                                                         |
| Located in wind zone     | Check this option, if the building is located in wind zone 3 or 4. In this case, 'snow' is not considered as an accompanying action to 'wind', which is the leading action.                                                                                             |
| Equal $\gamma_G$ ...     | If this option is checked, all permanent loads or load cases are applied together with the same partial safety factor ( $\gamma_{G,sup}$ or $\gamma_{G,inf}$ ), otherwise permanent loads are combined independently with their lower and upper partial safety factors. |

### Material

Depending on the selected standard, the corresponding material parameters, such as concrete quality or steel grade, are available for selection.

If you check the precast component option, a dialog box for entering parameters/characteristic values specific to precast components is displayed.

If required, you can specify a user-defined modulus of elasticity, to take a known deviation due to the aggregates used into account, for example.

#### User-defined concrete

Press the F5 key or the "F5" button in the "Concrete quality" input field to open a dialog via "New". The default values correspond to the currently selected quality. These values can now be adjusted and given your own short name.

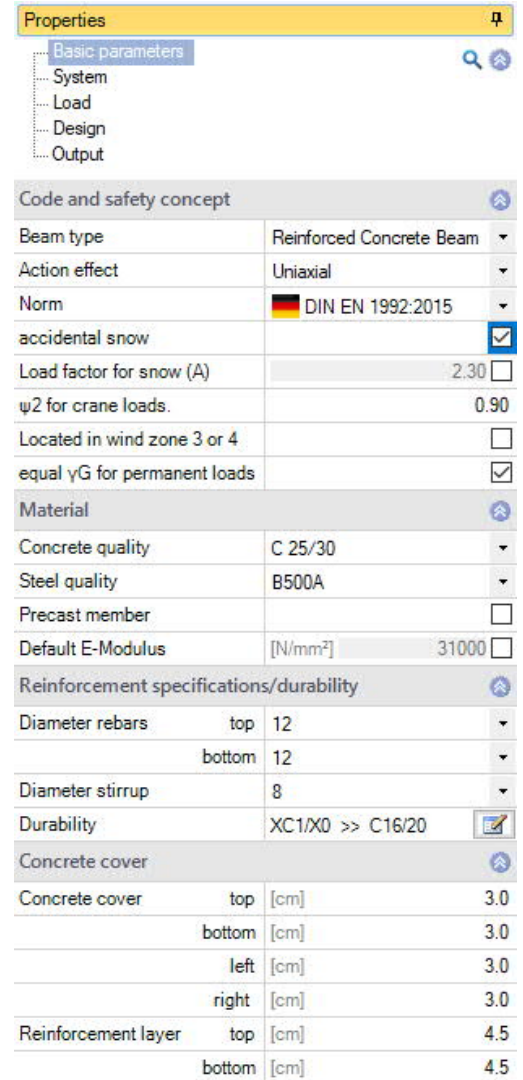
Three variants are available for determining the concrete parameters:

According to EC2 formulas: Normal and lightweight concretes with any  $f_{ck}$  value.

According to EC2 Table 3.1: standardized normal and lightweight concretes.

Free input: all concrete parameters are freely definable

See also [manual B2+](#).



The screenshot shows the 'Properties' dialog box with the following settings:

- Basic parameters:** System, Load, Design, Output.
- Code and safety concept:**
  - Beam type: Reinforced Concrete Beam
  - Action effect: Uniaxial
  - Norm: DIN EN 1992:2015
  - accidental snow:
  - Load factor for snow (A): 2.30
  - $\psi_2$  for crane loads: 0.90
  - Located in wind zone 3 or 4:
  - equal  $\gamma_G$  for permanent loads:
- Material:**
  - Concrete quality: C 25/30
  - Steel quality: B500A
  - Precast member:
  - Default E-Modulus: [N/mm<sup>2</sup>] 31000
- Reinforcement specifications/durability:**
  - Diameter rebars: top 12, bottom 12
  - Diameter stirrup: 8
  - Durability: XC1/X0 >> C16/20
- Concrete cover:**
  - Concrete cover: top [cm] 3.0, bottom [cm] 3.0, left [cm] 3.0, right [cm] 3.0
  - Reinforcement layer: top [cm] 4.5, bottom [cm] 4.5

### Specifications for reinforcement/durability

|                    |                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                              |
|--------------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| Diameter bar steel | Selection of bar steel diameters for the upper/lower reinforcement and the stirrups (6 mm–40 mm)                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                             |
| Durability         | <p>Accesses the dialog for <a href="#">Durability, Creep Coefficient and Shrinkage Strain</a></p> <p>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.</p> <p>The values for the creep coefficient and the shrinkage strain are needed for the calculation of the deformation in state I and state II.</p> |

### Concrete cover

The concrete cover can be defined differently on four sides. These input values are required for the reinforcement layout.

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

## Structural System

You can enter spans/sections/cantilevers/supports/joints etc. directly in the graphical user interface via the context menu or via a table (tabs below the graphic).

To the table → [Spans/Segments](#)

**System axis end supports** The system axis of the end supports can run through the tripart point or the centre of the support.

**Spacing of girders** The beam spacing is considered if the affected width is taken into account in the calculation. In the [Load Table](#) you can consider the beam spacing for each load individually by ticking the option “per girder”.

To the table → [Supports](#)

**Identical support geometry**  
Check this option to define the same width and depth for all supports.

To the table → [Joints](#)

To the table → [Cross-sections](#)

To the table → [Bonded \(composite\) joints](#)

To the table → [Holes](#) (cut outs)

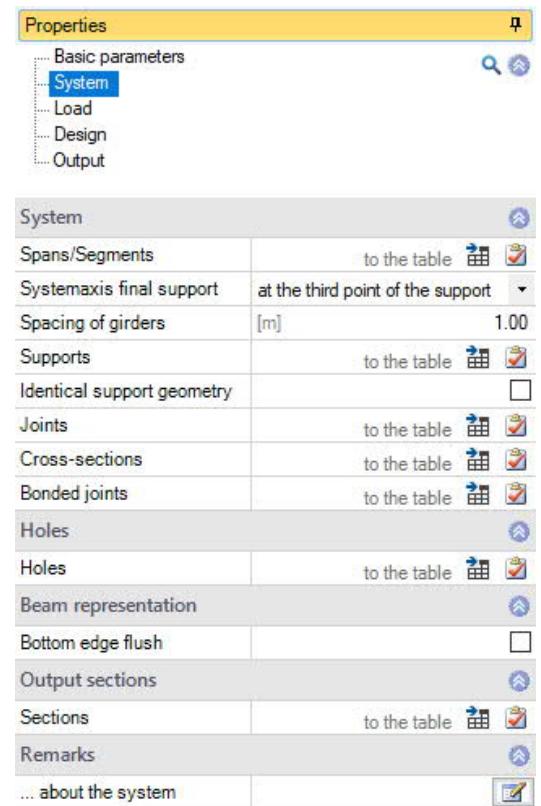
**Beam representation**

For different cross-sections, you can select between a flush bottom edge or a flush top edge in the graphical representation. This setting has no influence on the calculation and is intended only for the graphical representation.

To the table → [Output sections](#)

**Remarks on the system**

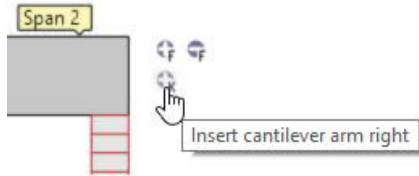
You can enter comments in a [Remarks Editor](#), which are then shown in the output at the relevant position.



## Data-entry options in the GUI

### Spans and cantilevers

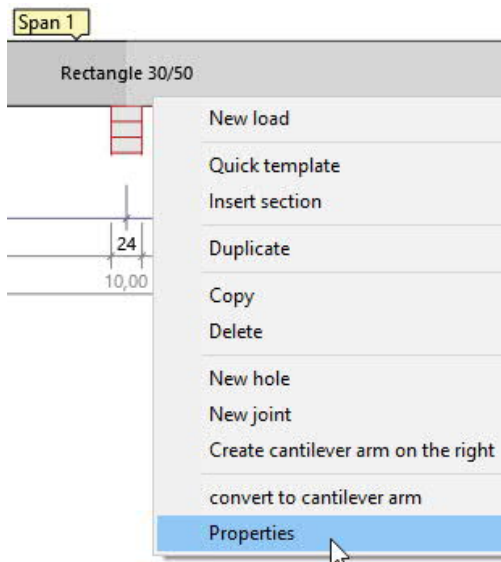
Spans and cantilevers can be inserted/removed directly in the graphics window using the +/- icons.



A right-click on the span in the graphical user interface displays context menu functions similar to those described for the data-entry via tables ([Spans/Segments](#)). You can edit the parameters for this span via the menu option "Properties", for instance.

### Additional data-entry functions in the GUI

Joints or segments can also be inserted/deleted via the context menu and various other functions as well as some quick templates are available.



*Note: The other objects in the graphic (supports, loads, etc.) also have their own context menus that allow you to quickly find the desired function.*

For data entry in the GUI in the PLUS programs, see also the [Basic Operating Instructions PLUS](#).



## Spans/segments

### Entry of spans/segments in a table

To enter data in the table, click on the tab "Spans/segments" below the graphic screen. Use the buttons on the right of the table [to add or delete table rows](#).

| Span | Span length<br>[m] | Segment | Segment length<br>[m] | CS-No.                                                           | CS-type   | bt<br>[cm] | ht<br>[cm] | b0<br>[cm] | h0<br>[cm] | bb<br>[cm] | hb<br>[cm] | Haunch                   |
|------|--------------------|---------|-----------------------|------------------------------------------------------------------|-----------|------------|------------|------------|------------|------------|------------|--------------------------|
| 1    | Crit le            | 1       | 1.50                  | 1. PT 60/20/30/50                                                | Plate top | 60.0       | 20.0       | 30.0       | 50.0       | ---        | ---        | <input type="checkbox"/> |
| 2    | Span 1             | 1       | 5.00                  | 1. PT 60/20/30/50                                                | Plate top | 60.0       | 20.0       | 30.0       | 50.0       | ---        | ---        | <input type="checkbox"/> |
| 3    | Span 2             | 1       | 5.00                  | 0 - new cross-section<br>1. PT 60/20/30/50<br>2. Rectangle 30/70 | Plate top | 60.0       | 20.0       | 30.0       | 50.0       | ---        | ---        | <input type="checkbox"/> |
| 4    | Span 3             |         | ---                   |                                                                  | ---       | ---        | ---        | ---        | ---        | ---        | ---        | <input type="checkbox"/> |

**Span** Automatic assignment of designations to the individual spans/cantilevers.

**Span length** Entry of the lengths of the individual spans/cantilevers.

**Segment** Consecutive numbering of the segments in each span

**Segment length** a member can be divided into several segments. If you enter a segment length that is smaller than the span length, a new row is automatically inserted for another segment and the remaining span length is registered automatically as the length of the new segment. You can further divide this segment in the same way.

*Tip: You can make the division into segments also in the graphical user interface via the [Context Menu](#).*

**CS No.** Each cross-section is designated with a consecutive number followed by the cross-sectional dimensions. To define a (new) cross-section, just click in the selection list on „0 – new cross section“.

In a separate dialog, you can then [define a new cross-section](#).

To select an existing cross-section, just click in the corresponding span on the selection list and select the cross-section in question.

In addition to rectangular cross-sections, you can also define t-beams (also one-sided).

bt/bb = slab width on top/bottom, ht/hb = slab thickness on top/bottom, b0 = width/width of the web, h0 - height/height of the web.

| CS-No.                |
|-----------------------|
| 1. Rectangle 30/50    |
| 0 - new cross-section |
| 1. Rectangle 30/50    |
| 2. Rectangle 30/70    |

**Haunch** Check this option to define a haunch for a span/cantilever/segment. An additional row is automatically inserted below the corresponding span/segment where you can specify the cross-section at the right end.

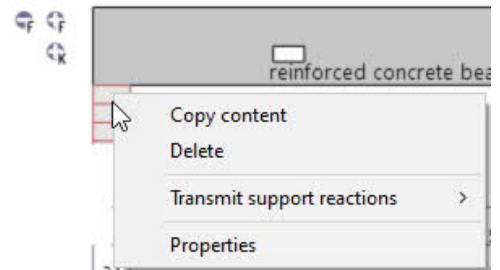
*Tip: Editing in the GUI*

You can also click on a span on the graphic screen to access editing functions such as Split segment or Edit cross-section at front/rear end.

## Supports

### Graphical user interface/editing of supports

The support properties are called up by double-clicking on the support or by right-clicking and selecting the appropriate option in the context menu. It is also possible to delete supports or transfer the properties of the current support to another support using the "Copy contents" function. Alternatively, you can use the data-entry functions of the table (tab "Supports" see "Data-entry via tables" below).



### Load transfer:


The support reactions can be transferred to the Column programs B5+/STS+/HO1+ for further calculation. Right-click on the support ▶ Transfer support reactions ▶ select the program. See also "[Output](#)."

### Properties

#### Type and dimensions

You can select the type of support and enter the width of the support.

- Cutting edge
- Masonry
- Concrete, direct (with minimum moment over continuous edges)
- Concrete, indirect
- Concrete, direct (without minimum moment over continuous edges)

| Properties of the support [1] |             | ?     | ×                                                                                     |
|-------------------------------|-------------|-------|---------------------------------------------------------------------------------------|
| Type and dimensions           |             |       |                                                                                       |
| Type                          | Masonry     |       |                                                                                       |
| Width                         | [cm]        | 24.0  |                                                                                       |
| Elastic bearing               |             |       |                                                                                       |
| Cz                            |             | rigid | <input checked="" type="checkbox"/>                                                   |
| Phiy                          | [kNm/rad]   | 0.0   | <input type="checkbox"/>                                                              |
| Calculate spring values       |             |       |  |
| Settlement of supports        |             |       |                                                                                       |
| fz                            | [cm]        | 0.0   |                                                                                       |
| Action                        | Settlements |       |                                                                                       |

#### Elastic support

You can define supports in the z-direction as well as for torsion about the y-axis. In each case, you can optionally define rigid supports or enter a spring value.

### Calculating spring values:

Alternatively, you can also have the spring values of a single column that can be defined underneath and/or above the beam calculated by the program and then use them for the beam calculation. To do this, click on the "Calculate spring values" button. In a separate dialog, select the corresponding options for the calculation (coil spring/torsion spring) and enter the parameters (pinned/restrained support, column height). You can edit the spring values (C, Phi) as desired.

Endeinspannung      Clamping at end supports can be entered as a percentage.

#### Column settlement

You can specify column settlements (fz/fy) at the individual supports or select an action.

### Data-entry via tables/editing of the supports

If you do not use the graphical user interface (see above), click on the "Support" tab below the graphic to open the table and enter the parameters there.

| Type                                                  |       | Elastic bearing                           |                              |               |                | Settlement of supports |             |
|-------------------------------------------------------|-------|-------------------------------------------|------------------------------|---------------|----------------|------------------------|-------------|
| Type                                                  | Width | Cz                                        | Phiy                         | Calculate     | Endeinspannung | fz                     | Action      |
|                                                       | [cm]  | [kN/m]                                    | [kNm/rad]                    | spring values | [%]            | [cm]                   |             |
| 1 Masonry                                             | 24.0  | rigid <input checked="" type="checkbox"/> | 0.0 <input type="checkbox"/> |               | 0.0            | 0.0                    | Settlements |
| 2 Cutting edge                                        | 24.0  | rigid <input checked="" type="checkbox"/> | 0.0 <input type="checkbox"/> |               | ---            | 0.0                    | Settlements |
| 3 Masonry                                             | 24.0  | rigid <input checked="" type="checkbox"/> | 0.0 <input type="checkbox"/> |               | 0.0            | 0.0                    | Settlements |
| Concrete directly with minimum support moment         |       |                                           |                              |               |                |                        |             |
| Concrete indirect                                     |       |                                           |                              |               |                |                        |             |
| Concrete directly without a minimum supporting moment |       |                                           |                              |               |                |                        |             |

### Joints

In the context menu of a span on the graphic screen you can select the option "New joint". The joint is shown as a small circle in the span and you can now enter the distance to the support in the dimensional chain also shown in the graphic.

Alternatively, you can click on the plus button in the Joints table to add a new row for the joint and enter the distance X1 to the front end of the left span.

| Span     | x1   |
|----------|------|
|          | [m]  |
| 1 Span 1 | 2.50 |
| 2 Span 1 | 0.00 |
| Span 1   |      |
| Span 2   |      |
| Span 3   |      |

### Cross-sections

You can define multiple different cross-sections in the table on the tab "Cross-sections" below the graphic.

| No  | Type      | bt   | ht   | b0   | h0   | bb   | hb   |
|-----|-----------|------|------|------|------|------|------|
|     |           | [cm] | [cm] | [cm] | [cm] | [cm] | [cm] |
| 1 1 | Plate top | 60.0 | 20.0 | 30.0 | 50.0 | ---  | ---  |
| 2 2 | Rectangle | ---  | ---  | 30.0 | 70.0 | ---  | ---  |

To define a new cross-section (new row), first click on the plus button.

Type Select the type of cross-section. By pressing the Edit button, the cross-section management is called up. Here you can define a cross-section, also name and save it (management of cross-sections). In addition to rectangular, top/bottom/both sides, single-sided slab beams can also be selected.

bt Width of the slab on top

ht Height of the slab on top

b0 Width of a rectangular section or width of the web of a T-beam

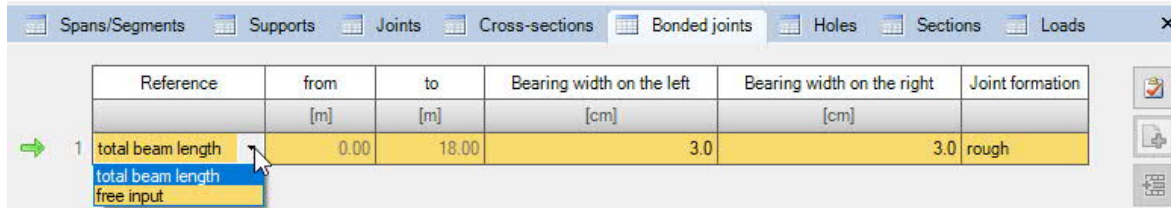
h0 Height of a rectangular section or height of the cross-section of a T-beam

bb Width of the slab on bottom

hb Height of the slab on bottom

## Composite joints

Entry of composite joints over the total length of the beam or by user-defined specifications.



| Reference | from | to    | Bearing width on the left | Bearing width on the right | Joint formation |
|-----------|------|-------|---------------------------|----------------------------|-----------------|
|           | [m]  | [m]   | [cm]                      | [cm]                       |                 |
| 1         | 0.00 | 18.00 | 3.0                       | 3.0                        | rough           |

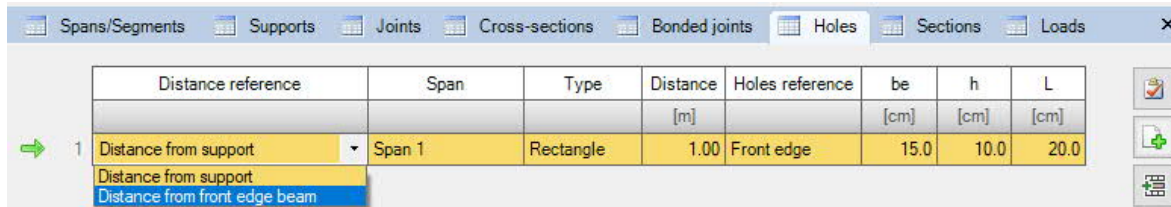
The specifications "from"/"to" refer to the left front end of the beam.

For the joint design, the options none, very smooth, smooth, rough and interlocked are available.

## Cut-outs (Holes)

You can specify the dimensions for round and rectangular cut-outs.

When positioning the cut-out you can select between the distance to the left support or to the (left) front edge of the beam.



| Distance reference | Span   | Type      | Distance | Holes reference | be   | h    | L    |
|--------------------|--------|-----------|----------|-----------------|------|------|------|
|                    |        |           | [m]      |                 | [cm] | [cm] | [cm] |
| 1                  | Span 1 | Rectangle | 1.00     | Front edge      | 15.0 | 10.0 | 20.0 |

The cut-out can be referenced to the (left) front edge or the central axis of the cut-out.

be Distance of the lower cut-out edge to the bottom edge of the beam

h Height of a rectangular or diameter of a circular cutout

L Length of a rectangular cut-out

The verification of the cut-out is performed according to booklet 399 of the DAfStb in combination with the recommendations of Leonhardt (lectures on solid construction, 3rd part, chapter 9.12) and engineering assumptions. The same verification is performed for circular and rectangular cut-outs.

The load combinations for max/min M and max V in the center of the cut-out are considered. The shear forces are distributed to the upper and lower chord in the ratio of the stiffnesses of the uncracked concrete cross-section.

The following boundary conditions apply:

For positive moments, the top chord is assigned at least 70% and a maximum of 90% of the shear force. If the dimensions of the bottom chord are smaller than 8 cm, the full shear force is assigned to the top chord and the bottom chord acts only as a tension chord. For negative moments, these conditions apply accordingly in reverse.

As a result of the shear forces, the chords must be verified for bending with tension/compression and for shear. The reinforcement layer is  $d1$  for the top chord and  $d2$  for the bottom chord. The acting bending tensile or bending compressive force is applied at the center of gravity of the respective chord. In addition, there is a required suspension reinforcement to the left and right of the cut-out.

The following restrictions and design notes must be observed:

Cut-outs whose edge distance to the support is smaller than the beam height cannot be verified with the procedure according to booklet 399. From an engineering point of view, cut-outs should not begin or end at a distance of  $< 0.10 \cdot \text{span length}$ .

In addition, according to booklet 399, the clear distance between the cut-outs must be greater than twice the static effective height of the component so that the interference areas of the truss models do not overlap.

Within the program, the minimum chord height was set to 2.5 times the reinforcement layer in order to have a sufficient lever arm for the design.

Furthermore, due to the weakening of the cross-section, the length of a cut-out should be limited to a maximum of 1/3 of the span length.

In the case of concentrated loads directly above the cut-out or at a distance  $\leq$  beam height, the load distribution must be checked subsequently in the case of slender residual cross sections (beam support effect).

Subsequently, it must also be checked whether the anchorage lengths of the additional longitudinal reinforcement are complied with.

Due to the minor effect on the load-bearing behavior, no verification is performed for cut-outs with dimensions  $L$  and  $h < \text{beam height} / 10$ .

## Output sections

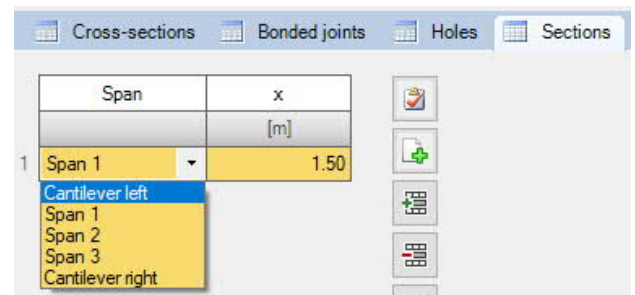
The program performs the design at all marked points. You can define sections to obtain additional output data.

Access via the tab "Sections" on bottom of the graphic screen.

Click on the plus button to display a new table row. Then, select Span/Cantilever and define the distance  $x$  from the left front end of the span/support [m].

You can display or hide the sections in the [result graphics](#).

*Tip: You can position sections freely in the graphic with the mouse by clicking on a free space and selecting the Option "Input sections" in the displayed context menu.*



## Loads

Select in the left menu under Loads whether self-weight should be included in the calculation or not, whereby only the web can be taken into account. This applies to all types of slab beams.

The girder spacing is considered if the affected width is taken into account in the calculation. In the Load Table, you can consider the girder spacing for each load individually by checking the option "per girder".

The tab "Loads" displays the load table where you can enter additional parameters.

| Reference | Load type                  | Action                              | D   | L1  | L2  | W1    | W2 | Unit | Factor | per girder                          | Span wise | Acting         | Acting        | Designation |
|-----------|----------------------------|-------------------------------------|-----|-----|-----|-------|----|------|--------|-------------------------------------|-----------|----------------|---------------|-------------|
|           |                            |                                     | [m] | [m] | [m] |       |    |      |        | <input checked="" type="checkbox"/> |           | simultaneously | alternatively |             |
| System    | Uniformly distributed load | Permanent loads                     | --  | --  | --  | 5.00  | -- | kN/m | 1.00   | <input checked="" type="checkbox"/> | No        | none           | none          |             |
| System    | Uniformly distributed load | Cat. A: domestic, residential areas | --  | --  | --  | 10.00 | -- | kN/m | 1.00   | <input checked="" type="checkbox"/> | Yes       | none           | none          |             |

**Reference** Allows you to select whether distance "A" refers to the left end of the beam (system) or to the left end of the respective span.

**Load type** Selection of the load type: uniformly distributed, trapezoidal, triangular, concentrated loads as well as concentrated moments.

**Action** Select an action from a list. Using the left menu, you can define [user-defined actions](#) and edit them.

**D, L1 / L2** A is the distance of the front end of the load application area to the selected reference point (see above, left beam end or left cantilever end or left span end). L1 is the length of the trapezoidal load application area. For triangular loads, the left and right lengths of the two triangular sections are specified via L1 and L2.

**W1 / W2** Enter the load value (W1) or, in the case of a trapezoidal load, also the second load value W2.

By clicking on the arrow icon you can access a [load value compilation](#).

**Note:** Check the loads entered in the graphic. Tip: Hover with the mouse over a load value to display details.

**Unit** Line load (kN/m) or area load (kN/m<sup>2</sup>), see the column "per beam".

**Factor** Multiplication factor for the load ordinates

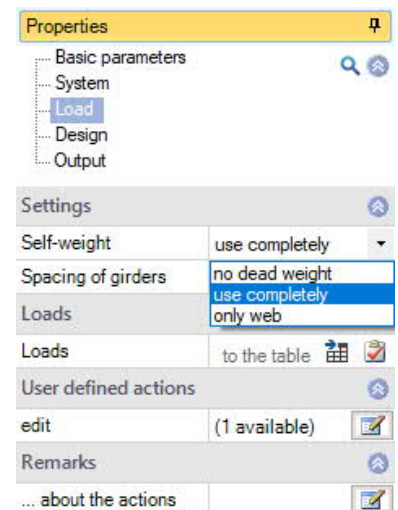
**Per girder (beam)** By default, the option "per girder" is checked, i. e. the entered load (W1/W2) is applied to this beam - without taking the girder spacing into account (for line loads, column unit = kN/m).

If you uncheck this option (remove the tick), the girder spacing is taken into account. The load ordinates are linked to this distance and the load values are calculated with the affected width (for line loads, column unit = kN/m<sup>2</sup>) in this case.

**L-Shape** For biaxial effect of actions ([additional option BTM-2](#)): 0 = vertical load, 90 = horizontal load.

**Spanwise** You can choose whether loads that are entered over several spans shall apply span by span or only in combination.

**Simultaneously** The loads of a simultaneous group always apply simultaneously.





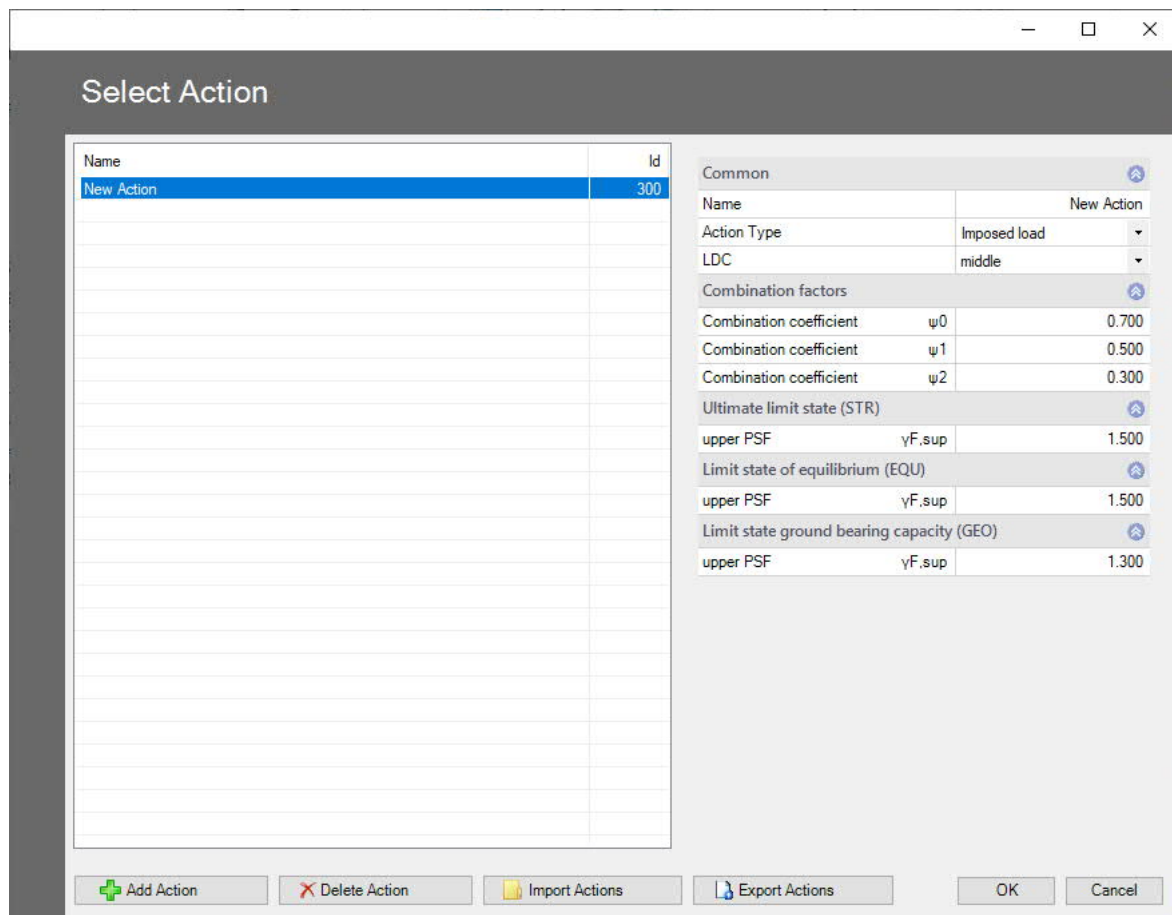
**Alternatively** The loads of an alternative group always act individually and are not superimposed. You should note in this context that if "spanwise" is defined in addition, each individual span with this load is already considered alternatively. If the load is defined as not acting "spanwise", the entire load is set acting alternatively to another load of the same alternative group.

**Note** *Simultaneous/Alternative Groups: Select "New group" to create a group with a consecutive number (Sim1, Sim2 etc.). You can enter via the left menu tree comments to the actions that are included in the output.*

**Designation** Allows you to enter comments on the loads. They are included in the output.

### User-defined actions

In the left menu under Load, you have the possibility to define and save your own actions as "[User-defined actions](#)". Click on the edit button to access the following dialog. PSF = partial safety factor.



### Defining a new action

By activating "Add action" you can generate a new list entry. In the right column, you can name the action (if the name exists already, the program automatically adds an extension (\*)). Change the parameter settings as desired and confirm your changes by clicking OK. You can edit/change defined actions at a later time, of course. To do this, select the corresponding row in the list.

### Export/Import

You can save the list of the actions in a file in the format \*.act and import this list in other programs.

## Design

### Moment

**Span reinforcement staggered** This option forms the basis of the reinforcement layout (add-on with separate licence [BTM-BEW](#)). 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 BTM-BEW, the actually existing reinforcement is considered at the point x.

**Minimum reinforcement** When you check this option, a minimum reinforcement as per EN 1992-1-1 is considered.

**Moment redistribution up to** You can reduce the moments over continuous edges for slabs and beams.

**Comply with limit  $k_x < 0.45$**  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).

**Hor. branch stress line** The horizontal section of the stress-strain diagram as per EN 1992-1-1, figure 3.8 is considered.

### Shear force

**const  $\vartheta$**  Independent of the state of effects of actions, the strut inclination angle can be set to "const  $\vartheta$ ". 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.

**const  $\vartheta$  only at the end support** The constant strut inclination angle is only set for the end support.

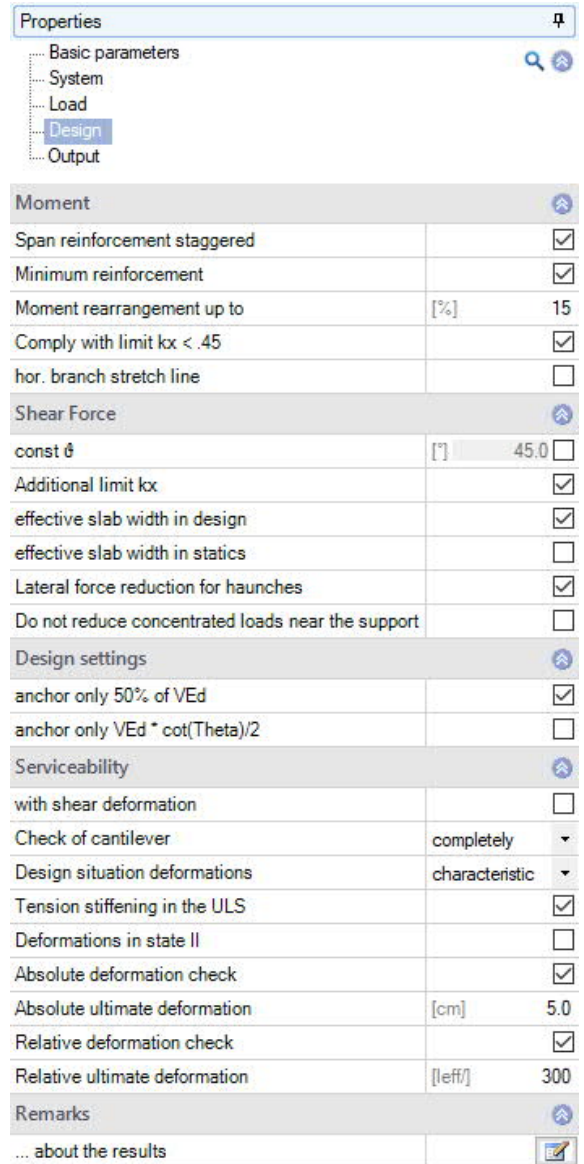
**Additional limitation of  $k_x$**  If you disable this option, the compression zone height is only limited by the yield strength of the steel. In linear elastic calculations of continuous beams, the compression zone height should be limited if no constructive measures are undertaken. Compliance with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

**Effective slab width in ...** The setting of this option defines whether the effective slab width of T-beams in accordance with EN 1992-1-1 (5.3.2.1) is considered in the design and/or the determination of the internal forces.

**Minimum reinforcement  $Q_k$**  Minimum reinforcement according to EN 1992-1-1 is taken into account.

**Lateral force reduction for haunches** When this option is ticked, the shear force is reduced for haunches in accordance with EN 1992-1-1, 6.2.1.

**Concentrated loads near supports ...** Disables the reduction of concentrated loads near supports for reinforced concrete.





## Design settings

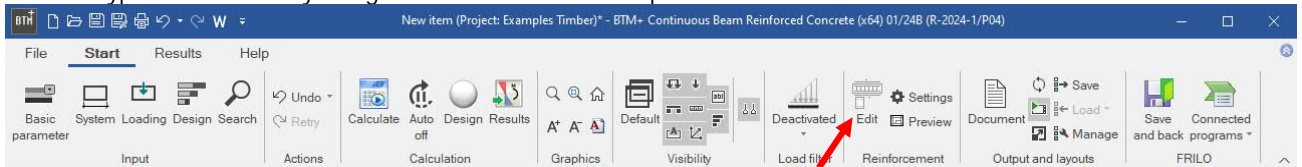
|                         |                                                                                                                           |
|-------------------------|---------------------------------------------------------------------------------------------------------------------------|
| Only 50 % of VEd ...    | Anchoring at the end support only 50 % of VEd on front edge of the end support.                                           |
| Only VEd* cot Theta/2.. | Anchoring at end support of VEd at the front edge of the end support * cot Theta/2. see DIN EN 1992-1-1, Chapter 9.2.1.4. |

## Serviceability

|                                      |                                                                                                                                                                                                           |
|--------------------------------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| With shear deformation               | You can optionally take deformation by shear into account.                                                                                                                                                |
| Deformations design situation        | You can select the design situation the verifications in the serviceability limit state should be based on – characteristic, frequent and quasi-permanent.                                                |
| Creep design situation               | Selection of the design situation that should be used for the calculation of creep deformations.                                                                                                          |
| Tension stiffening SLS               | When this option is checked, the stiffening effect of uncracked concrete areas between the calculated main cracks is considered in the calculation of the deformations in the serviceability limit state. |
| Deformation in state II              | Check this option if the deformations in state II should be calculated.                                                                                                                                   |
| Verification of absolute deformation | When you check this option, the serviceability verification is performed with consideration of the deformation difference in relation to the undeformed system.                                           |
| Absolute limit deformation           | The permitted maximum absolute deformation of the structural system.                                                                                                                                      |
| Verification of relative deformation | When you check this option, the serviceability verification is performed with regard to the effective lengths, which are determined by the inflection points (moment passage) of the bending line.        |
| Relative limit deformation           | The permitted maximum relative deformation of the structural system.                                                                                                                                      |

## Reinforcement Layout (optional add-on BTM-BEW)

The BTM-BEW Reinforcement Layout is available as [optional add-on](#) with an additional licence. Rectangular beams with or without cut-outs, as well as slabs and T-beams are available. Other cross-sections or beam types are constantly being added and further developed.



Click on the "Edit" button to access the reinforcement layout dialog. In addition to the structurally required reinforcement, a reinforcement graphic with editing options can also be created in this dialog. With the additional option BTM-2, biaxial reinforcement per corner is also possible.

The 'Reinforcement' dialog box displays a cross-section of a beam with reinforcement layout. The vertical axis represents the depth in cm, ranging from 8.0 to -8.0. The horizontal axis shows reinforcement segments with lengths and bar counts. A table at the bottom provides detailed data for each reinforcement zone.

| Zone | Range length [m] | Segment   | Section length [m] | Reinforcement bottom | exist. as [cm <sup>2</sup> ] | req. as [cm <sup>2</sup> ] | Create reinforcement | Delete reinforcement |
|------|------------------|-----------|--------------------|----------------------|------------------------------|----------------------------|----------------------|----------------------|
| 1    | bottom 1         |           | --                 | 2   Ø 12             | 2.3                          | 2.8                        | Create reinforcement | Delete reinforcement |
| 2    |                  | Segment 1 | 5.00               | 2   Ø 12             | 4.5                          | 2.8                        | Create reinforcement | Delete reinforcement |
| 3    | bottom 2         |           | --                 | 2   Ø 12             | 2.3                          | 2.8                        | Create reinforcement | Delete reinforcement |
| 4    |                  | Segment 1 | 5.00               | 2   Ø 12             | 4.5                          | 2.8                        | Create reinforcement | Delete reinforcement |

The reinforcement layout is generated on the basis of the tensile force or shear force coverage diagram. For T-beams, a required shear and, if necessary, shoulder shear coverage is also taken into account. Moreover, you can reinforce cut-outs existing in the structural system. Optionally, you can lay the required reinforcement in the section.

With slabs, only the resisting tensile force coverage diagram is evaluated. Reinforcement mats can also be laid. In the current version, you cannot apply reinforcement to slabs with required shear force reinforcement.

## Limitations


Beams with a variable cross-section, such as haunches or systems with cross-sectional discontinuities cannot be reinforced at present. However, you can define the structurally required reinforcement by activating the "Design" button.

A possibly required suspension reinforcement is not considered in the program and needs to be verified separately.

The anchorage of longitudinal bars is not verified at haunches and discontinuities.

## Resisting tensile force coverage

The diagram of the resisting tensile force coverage and the shear force coverage is shown as a design line (with offset dimension, moments rounding etc.).

By activating the magic wand  reinforcement can be generated automatically based on the set diameters. In the lower table, you can enter the lower, upper and constructive reinforcement separately. You can operate the graphic fully interactively and it offers options for customising every single item.

In the input table, specify the number and diameter of the reinforcement bars for the individual layers.

In the first column of the lower or upper reinforcement, the basic reinforcement is defined. It is laid across the entire structural system. If the first layer should not be continuous, you can also switch the automatic laying off via the [interactive text](#) "first layer continuous". Additional reinforcement can be entered in different segments for each span. The selection of the segments is either guided by the spans or the supports. You can also freely define segments.

After each entry, a small line is drawn in the resisting tensile force diagram showing how much  $A_s$  is covered.

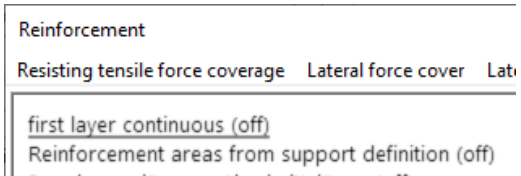
The reinforcement displayed in the graphical user interface can also be increased or decreased interactively by dragging and dropping specific elements. A double click displays an editing menu in which the selected bar can be customised individually.

If cut-outs exist, you can reinforce and customise them individually via the "Holes" (cut-outs) tab.

The function "Lay bars in the section" can be used to adjust the reinforcement at any point in the structural system. Individual bars can be modified directly in the GUI. After confirming the dialog, the resulting reinforcement layer is transferred to the program and adopted automatically.

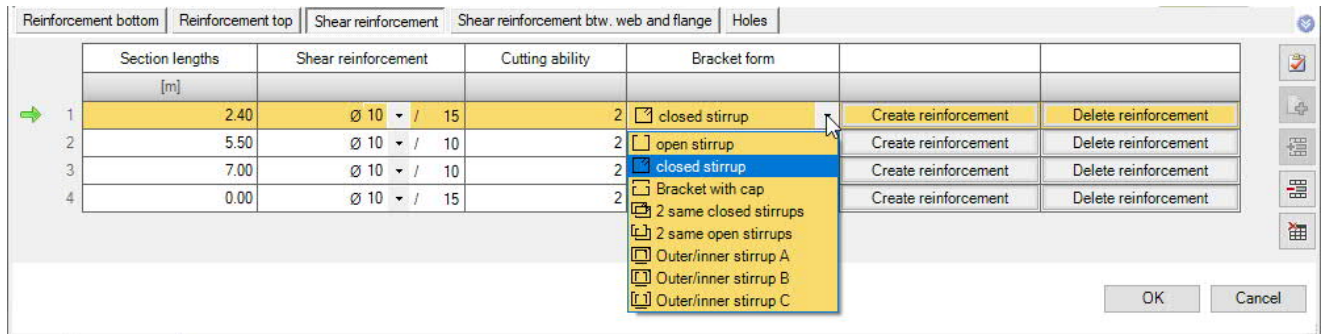
## Options

Clicking on an [interactive text](#) switches the corresponding option on or off or opens another context menu.

|                                               |                                                                                                                                                            |                                                                                       |
|-----------------------------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------|---------------------------------------------------------------------------------------|
| First layer continuous                        | The reinforcement bars of the first layer are laid across all spans.                                                                                       |  |
| Reinforcement regions from support definition | The reinforcement regions at the bottom of the spans and at the top above the columns can be generated automatically above the supports or defined freely. |                                                                                       |
| Reinforcement length = segment length         | Optionally, the segment reinforcement can be set to the same length as the basic reinforcement.                                                            |                                                                                       |
| Stagger span reinforcement                    | The span reinforcement can be staggered in the automatic generation process, see also <a href="#">Design</a> .                                             |                                                                                       |

## Shear force coverage

You can freely enter shear reinforcement across any number of segments. The pre-defined segments are based on the spans existing in the system.



|   | Section lengths<br>[m] | Shear reinforcement | Cutting ability | Bracket form     |                      |                      |
|---|------------------------|---------------------|-----------------|------------------|----------------------|----------------------|
| 1 | 2.40                   | Ø 10 / 15           | 2               | closed stirrup   | Create reinforcement | Delete reinforcement |
| 2 | 5.50                   | Ø 10 / 10           | 2               | open stirrup     | Create reinforcement | Delete reinforcement |
| 3 | 7.00                   | Ø 10 / 10           | 2               | closed stirrup   | Create reinforcement | Delete reinforcement |
| 4 | 0.00                   | Ø 10 / 15           | 2               | Bracket with cap | Create reinforcement | Delete reinforcement |

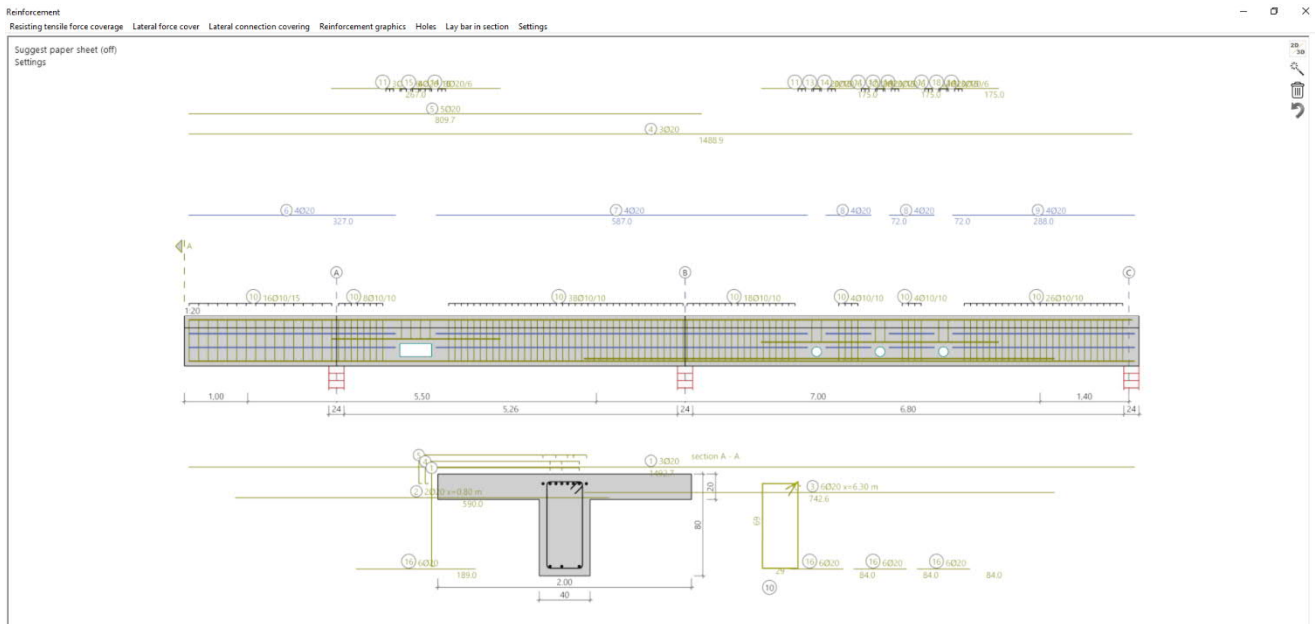
Double-shear and quadruple-shear stirrups are available as reinforcement

To define the stirrup reinforcement, enter the diameter  $\varnothing$ , the stirrup spacing  $e$  as well as the type of stirrup (single-, double-, quadruple-shear, etc) for each segment.

### Stirrup shape (Bracket form)

In this column, you can define the shape of the stirrup via a selection list.

## Reinforcement graphic



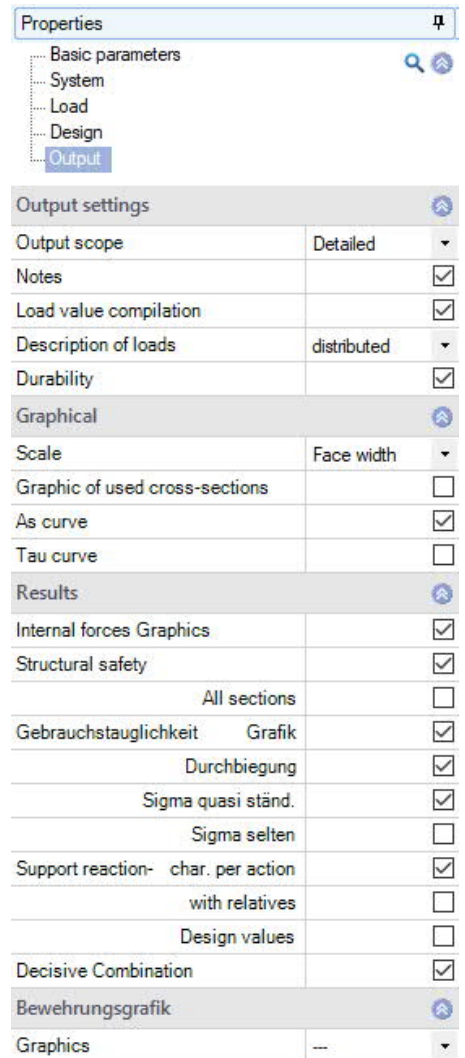
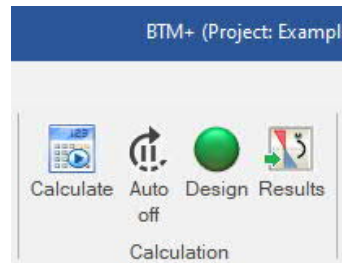
The tab "Reinforcement graphic" displays a graphical representation of the defined reinforcement. The graphical representation can be inserted in the structural analysis document or saved in a drawing format with a title block. For this purpose, the program offers individual setting options for the scale of the graphic, the font size and the dimensional chains. The area for the title block stored in the project is already indicated. The reinforcement elements can be freely moved and individually arranged in the same way as in the segments.

## Output

### Output scope, calculation, results

Before starting the output, click on the calculation icon if the option automatic calculation after each entry is switched off ("Auto off"/"Auto on" icon).

After the calculation, the loading is displayed in the bottom right-hand corner of the GUI and provides a good overview of the economic efficiency of the defined structural system.



### Visibility

In the upper toolbar, you can display or hide particular elements in the graphical representation by clicking the corresponding button.

### Load filter

The Load filter button can be used to filter loads according to actions and grouping (Alternative/Concurrent). The selection is then highlighted visually/by colour in the graphic. This allows you to easily check the loads and edit them immediately in the GUI. The option "Deactivate" disables the filter.

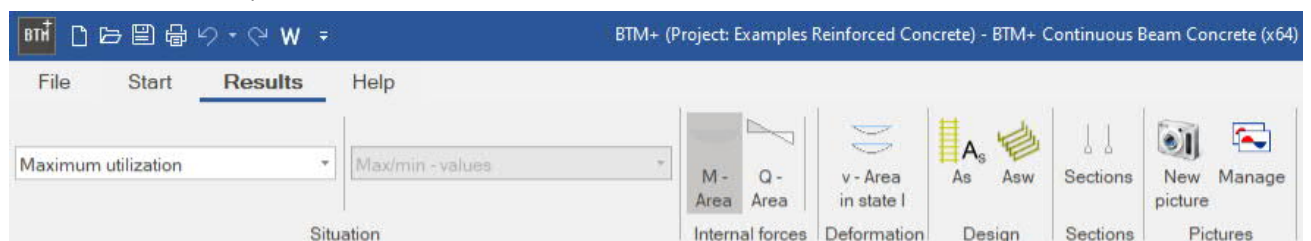
*Note: When the filter is set, unselected loads (grey) can be added to the current selection (coloured) by holding down the CTRL key and clicking on the load.*

### Result options

Via the "Results" tab, you can display the different result graphics.

You can display and hide the defined [output sections](#).

Via the camera icon you can take a snapshot of the displayed graphic and name it. Use the right-hand symbol "Manage" to display the list of snapshots, which can also be deleted from the list. These snapshots are automatically taken over into the output document.



### Output scope

By checking the desired options, you can determine the scope of data to be put out.

### Output as a PDF document

Via the [Document](#) tab, you can display the document in PDF and print it.

See also [Output and printing.pdf](#)

## Load transfer

To [transfer](#) the support reactions to the column programs [B5+](#), [STS+](#), [HO1+](#) or to the corbel programs [B9+](#) and [B10+](#) right-click on the respective support and select the corresponding program from the context menu list. See also the section Interfaces to other Programs in the chapter [Application options](#).

