

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	12
Loads	13
Design	15
Reinforcement Layout (optional add-on DLT-BEW)	17
Resisting tensile force coverage	18
Shear force coverage	19
Reinforcement graphic	20
Output	21
Output scope, calculation, results	21
Load transfer	22

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 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. 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

	DIN EN 1992:2012
	DIN EN 1992:2013
	DIN EN 1992:2015
	NTC EN 1992:2008
	ÖNORM EN 1992:2011
	BS EN 1992:2009
	BS EN 1992:2015
	PN EN 1992:2010
	EN 1992:2010
	EN 1992:2014

Wizard

You can enter the data necessary to define a simple basic system in a wizard. 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. 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 "[for each beam](#)" for the design and by defining a beam spacing.

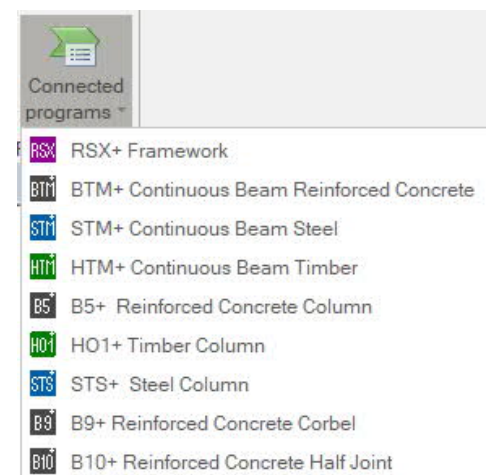
Interfaces to other programs

- 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

[Reinforcement Layout](#) is optionally available as DLT-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

Wizard

The [wizard](#) 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 wizard:

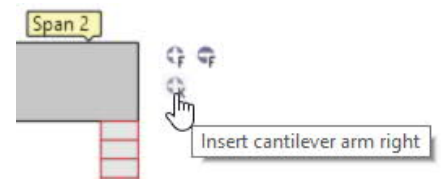
- Loading
- Concrete qualities and steel grades
- Precast component
- Specification of the modulus of elasticity
- Number of spans (or optionally, just a cantilever)
- Span length
- Cross-section
- Beam spacing
- Permanent line load
- Variable line load and type of action

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

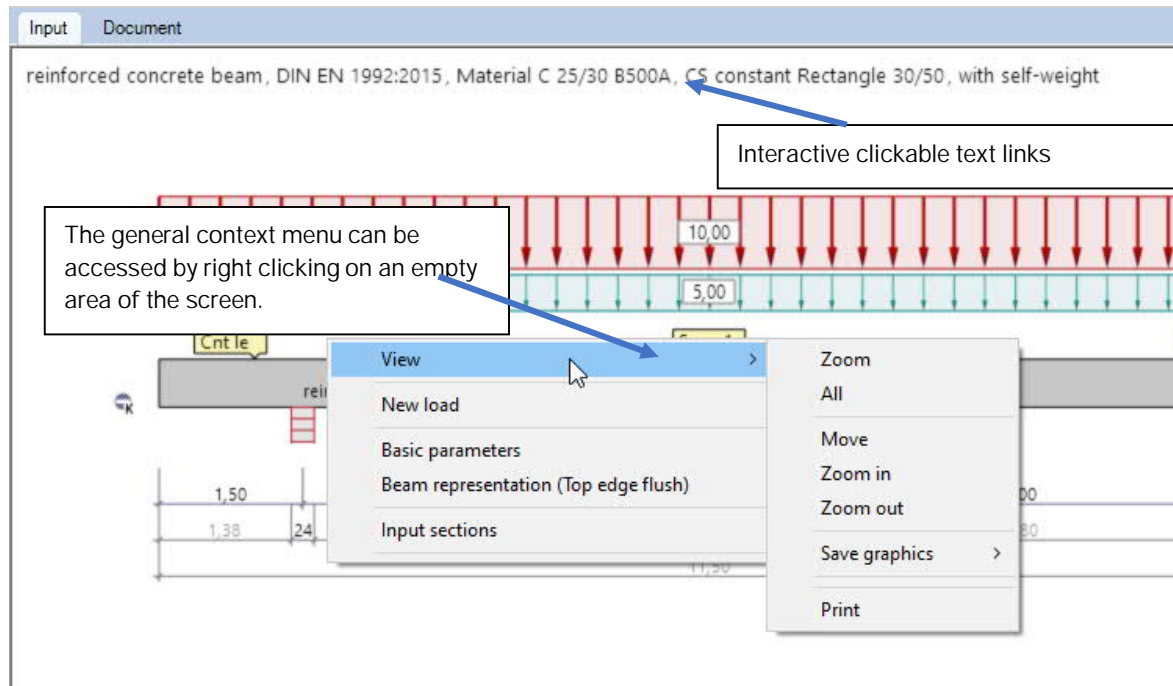
Graphical user interface (GUI)

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

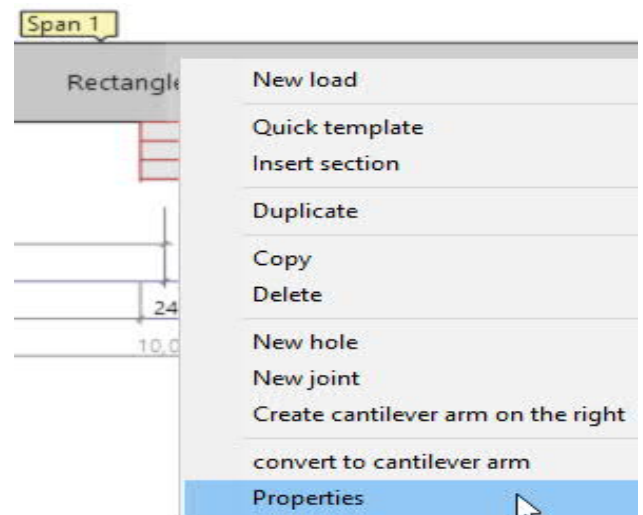
As in all Plus programs, the dimensional values are editable also in BTM+ and can be changed directly on the graphic screen.

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.



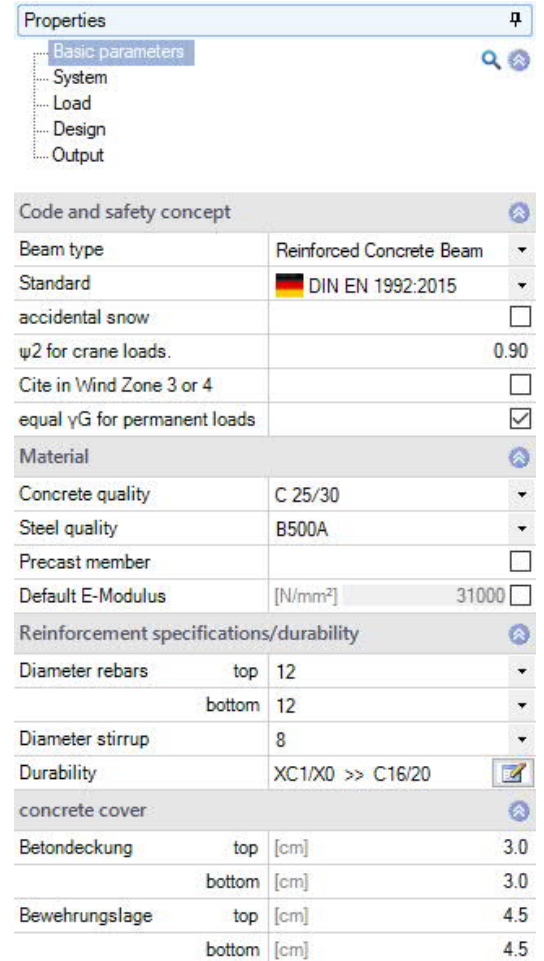
Interactive texts



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
Standard	Definition of the design standard and its national annex.
Snow as accidental load	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.
ψ_2 for crane loads	determines the combination coefficient ψ_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 γ_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.



Properties			
Basic parameters			
System			
Load			
Design			
Output			
Code and safety concept			
Beam type		Reinforced Concrete Beam	
Standard		 DIN EN 1992:2015	
accidental snow			<input type="checkbox"/>
ψ_2 for crane loads.			0.90
Cite in Wind Zone 3 or 4			<input type="checkbox"/>
equal γ_G for permanent loads			<input checked="" type="checkbox"/>
Material			
Concrete quality		C 25/30	
Steel quality		B500A	
Precast member			<input type="checkbox"/>
Default E-Modulus	[N/mm ²]	31000	<input type="checkbox"/>
Reinforcement specifications/durability			
Diameter rebars	top	12	
	bottom	12	
Diameter stirrup		8	
Durability		XC1/X0 >> C16/20	
concrete cover			
Betondeckung	top	[cm]	3.0
	bottom	[cm]	3.0
Bewehrungslage	top	[cm]	4.5
	bottom	[cm]	4.5

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.

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 Durability, Creep Coefficient and Shrinkage Strain</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 values entered for the concrete cover 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.

Beam spacing 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 "for each beam".

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 → [Composite joints](#)

To the table → [Holes](#) (cut outs)

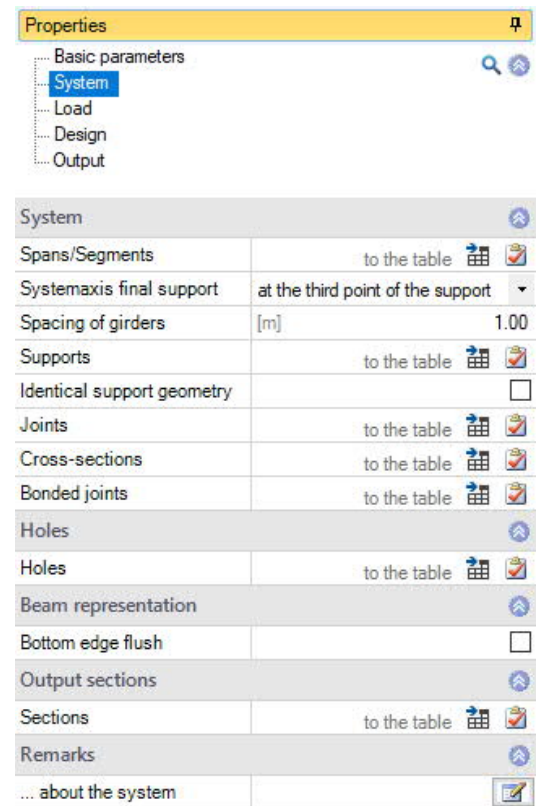
Beam representation

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

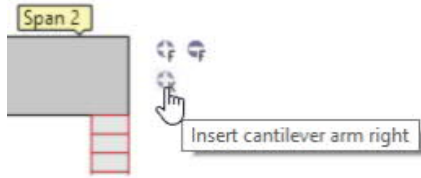
You can enter comments in a [Remarks Editor](#), which are then shown in the output at the relevant position.



Data-entry options in the GUI

Spans and cantilevers

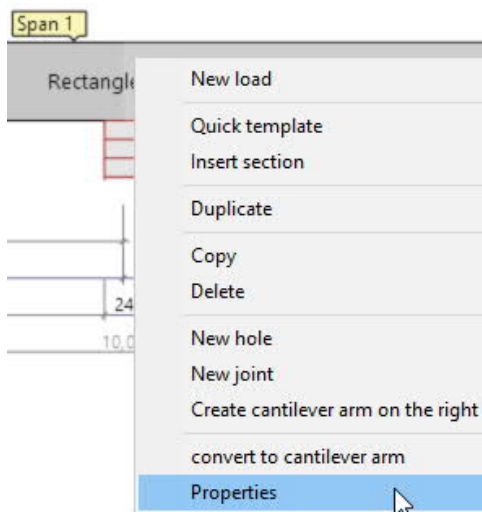
Spans and cantilevers can be inserted/removed directly in the graphics window using the +/- icons.



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 [m]	Segment	Segment length [m]	CS-No.	CS-type	bt [cm]	ht [cm]	b0 [cm]	h0 [cm]	bb [cm]	hb [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	Plate top	60.0	20.0	30.0	50.0	---	---	<input type="checkbox"/>
4	Span 3		---	1. PT 60/20/30/50 2. Rectangle 30/70	---	---	---	---	---	---	---	<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](#).

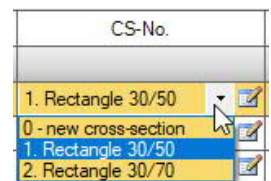
CS No. Each cross-section is designated with a consecutive number followed by the cross-sectional dimensions. To define a (new) cross-section, just click in the selection list on „0 – new cross section“.

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.

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.



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

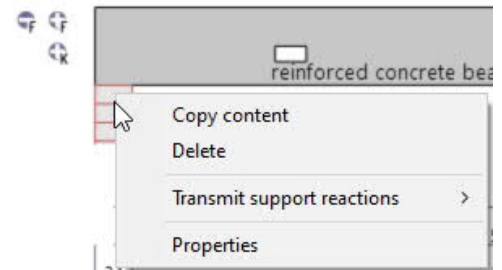
You can also click on a span on the graphic screen to access editing functions such as Split segment or Edit cross-section at front/rear end.

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.

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

	Type	Width [cm]	Cz [kN/m]	Phi _y [kNm/rad]	Calculate (spring values)	fz [cm]	Action
1	Masonry	24.0	rigid	0.0		0.0	Settlements
2	Cutting edge	24.0	rigid	0.0		0.0	Settlements
3	Concrete directly with minimum support moment	24.0	rigid	0.0		0.0	Settlements

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.

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

Cross-sections

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

No	Type	bt [cm]	ht [cm]	b0 [cm]	h0 [cm]	bb [cm]	hb [cm]
1	Plate top	60.0	20.0	30.0	50.0	---	---
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, you can define a cross-section and also name and save it (management of cross-sections).

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

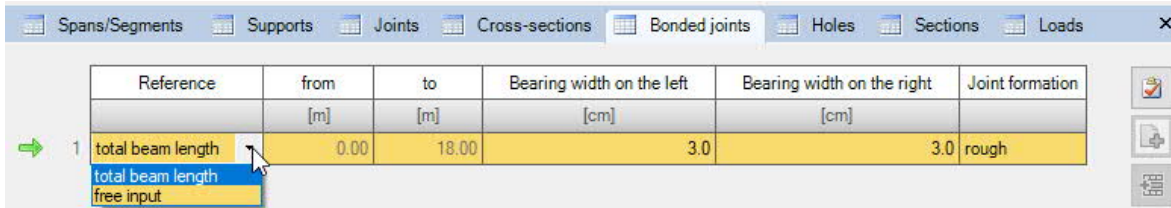
h0 Height of a rectangular section or height of the cross-section of a T-beam

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.



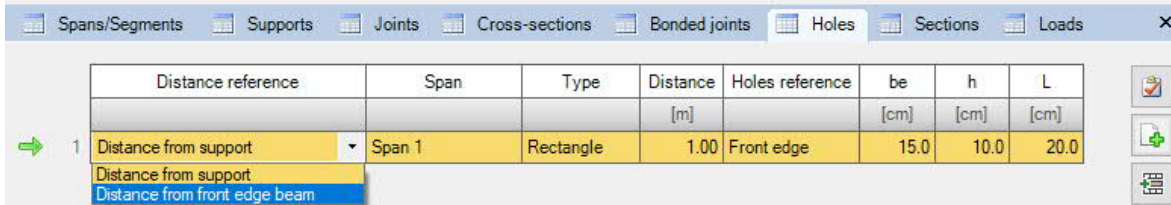
The specifications "from"/"to" refer to the left front end of the beam.

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.



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

uk 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

Output sections

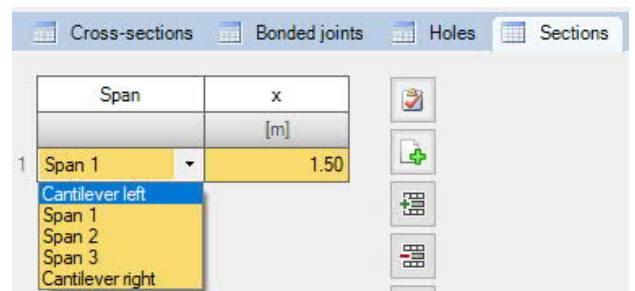
The program performs the design at all marked points. You can define sections to obtain additional output data.

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.

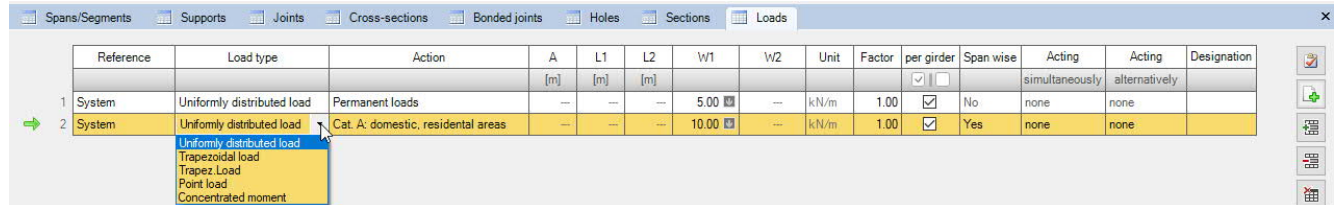


Loads

Select in the left menu under Loads whether self-weight should be included in the calculation or not.

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 checking the option "for each beam".

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



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.

A, 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 - see the description of the LOAD+ program.

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/m²), see the column "per beam".

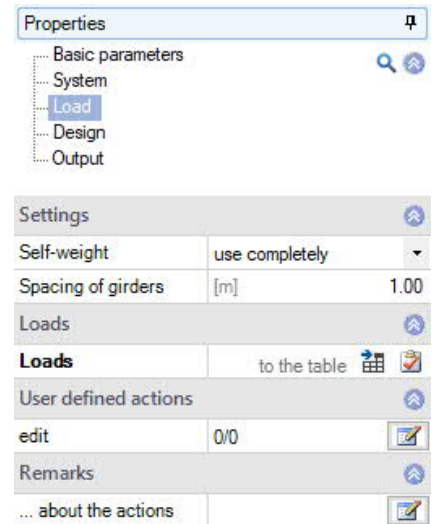
Factor Multiplication factor for the load ordinates

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 beam spacing into account (for line loads, column unit = kN/m). If you uncheck this option (remove the tick), the beam 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/m²) in this case.

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.

Alternative 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.



Design

Moment

Span reinforcement staggered This option forms the basis of the reinforcement layout (add-on with separate licence DLT-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 DLT-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 ϑ Independent of the state of effects of actions, the strut inclination angle can be set to "const ϑ ". 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 ϑ 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.

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.

The screenshot shows the 'Properties' dialog box with the 'Design' tab selected. The 'Design' category is highlighted in blue. The 'Moment' section includes: 'Span reinforcement staggered' (checked), 'Minimum reinforcement' (checked), 'Moment rearrangement up to' (set to 15%), 'Comply with limit $k_x < .45$ ' (checked), and 'hor. branch stretch line' (unchecked). The 'Shear Force' section includes: 'const ϑ ' (set to 45.0), 'Additional limit k_x ' (checked), 'effective slab width in design' (checked), 'effective slab width in statics' (unchecked), 'Lateral force reduction for haunches' (checked), and 'Do not reduce concentrated loads near the support' (unchecked). The 'Design settings' section includes: 'anchor only 50% of VEd' (checked) and 'anchor only VEd * cot(Theta)/2' (unchecked). The 'Serviceability' section includes: 'with shear deformation' (unchecked), 'Check of cantilever' (set to 'completely'), 'Design situation deformations' (set to 'characteristic'), 'Tension stiffening in the ULS' (checked), 'Deformations in state II' (unchecked), 'Absolute deformation check' (checked), 'Absolute ultimate deformation' (set to 5.0 cm), 'Relative deformation check' (checked), and 'Relative ultimate deformation' (set to 300). The 'Remarks' section has a button for '... about the results'.

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.

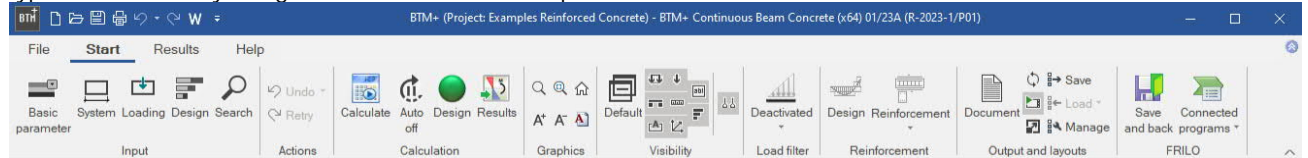
Serviceability

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.

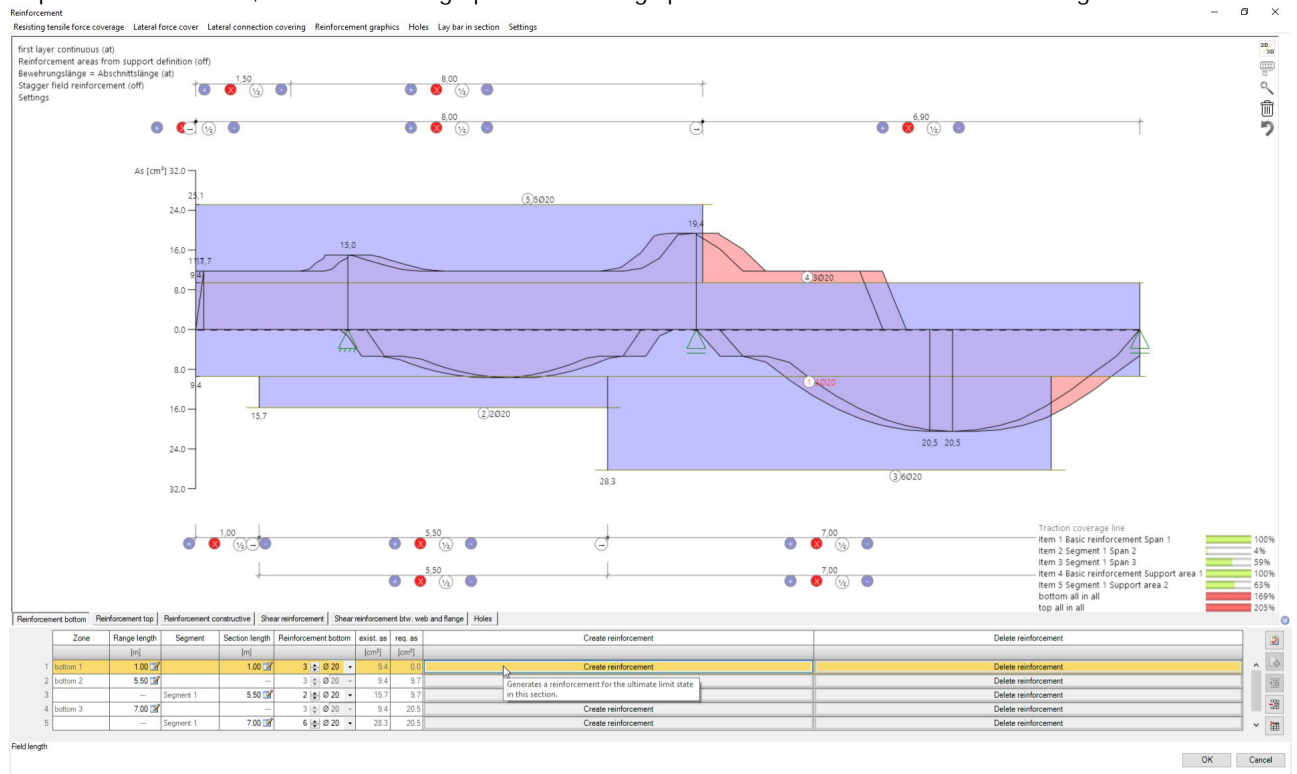
Reinforcement Layout (optional add-on DLT-BEW)

The DLT-BEW Reinforcement Layout is available as optional add-on with an additional licence.

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 and T-beams are available. Other cross-sections or beam types are constantly being added and further developed.



Click on the "Reinforcement" 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.



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

In release 2023-1, only rectangular beams and T-beams with and without cut-outs as well as slabs are available for selection.


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.

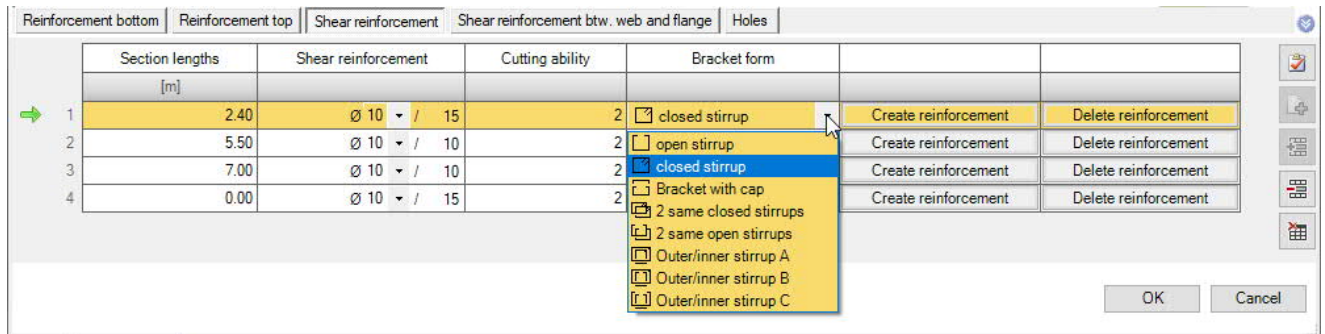
Stagger span reinforcement

The span reinforcement can be staggered in the automatic generation process, see also [Design](#).



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 [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

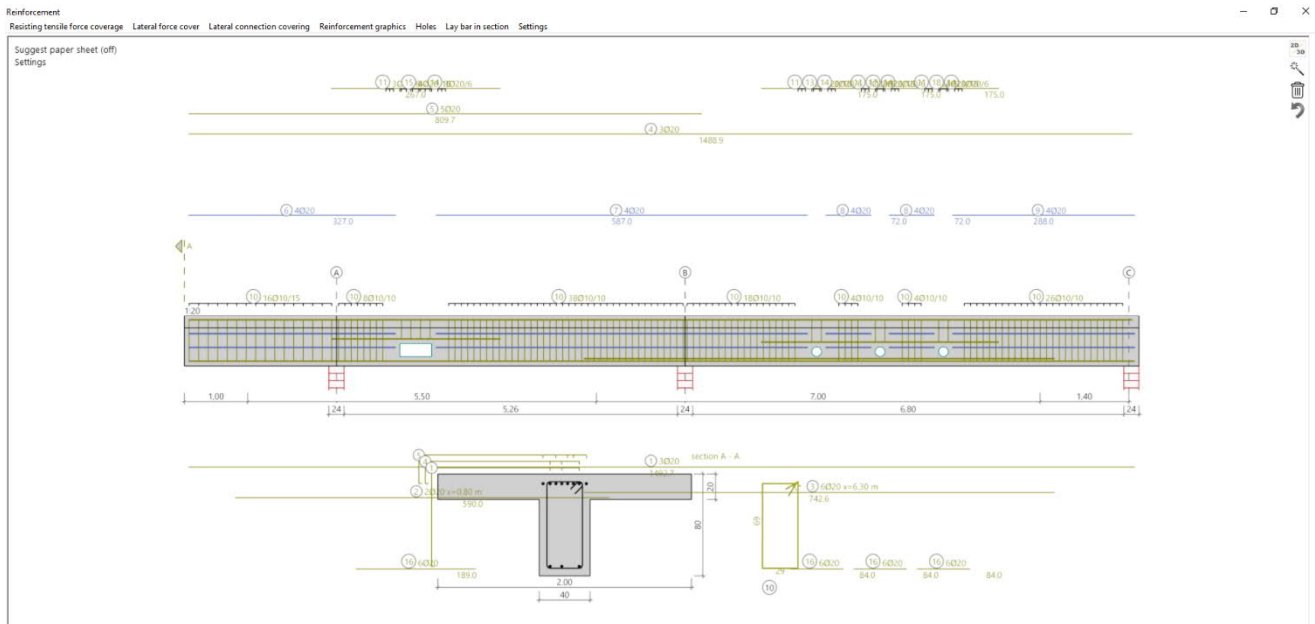
Double-shear and quadruple-shear stirrups are available as reinforcement

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



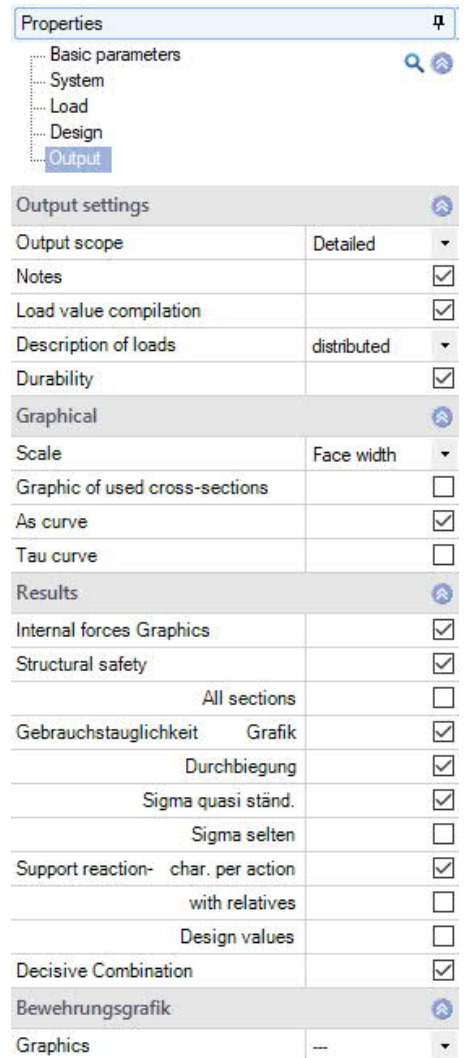
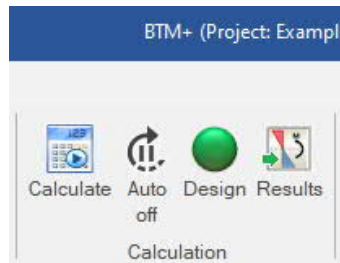
The tab "Reinforcement graphic" displays a graphical representation of the defined reinforcement. The graphical representation can be inserted in the structural analysis document or saved in a drawing format with a title block. For this purpose, the program offers individual setting options for the scale 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.

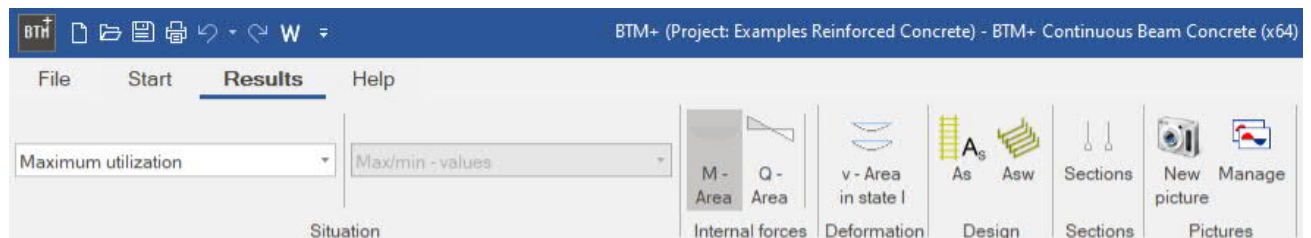
Note: When the filter is set, unselected loads (grey) can be added to the current selection (coloured) by holding down the CTRL key and clicking on the load.

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](#).

