HTM+ Continuous Beam Timber

Contents

Basic Documentation – Overview

In addition to the individual program manuals, you will find basic explanations on the operation of the programs on our homepage www.frilo.com in the Campus-download-section.

Application options

The HTM+ program is suitable for the calculation of single-span and multi-span timber beams.

A cantilever beam can also be selected as a special case.

Cross-sections can be different, with haunch, multi-part, with reinforcement. Spans can be divided into sections, joints are also possible.

The superposition and the design are performed automatically.

The program is designed for a graphically supported interactive workflow.

Available standards

- DIN EN 1995
- ÖNORM EN 1995
- NTC EN 1995
- BS EN 1995
- PN EN 1995
- EN 1995

Assistant

The data necessary to define a simple basic system can be entered via the assistant. Subsequently, you can easily modify and supplement this basic system via the interactive graphic user interface.

Supports/fixed restraints

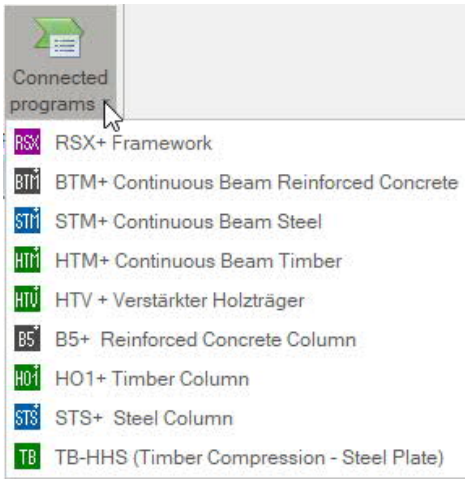
You can define supports in the z-direction and for the torsion about the y-axis (with the HTM-2 add-on, you can also define supports for biaxial loading in the y-direction). In each case, you can optionally define rigid supports or the enter a spring value. A column settlement can be pre-set for the individual supports. Alternatively, also the spring values of a single column that can be defined underneath and/or above the beam can be calculated by the program and then be used for the beam calculation.

For the stability verification (HTM-S add-on), the fixed restraints can be defined on the cross-section. A distinction is made between the position of the fixed restraints in the longitudinal direction of the beam and their position on the cross-section.

Loads

Load types: uniformly distributed, trapezoidal, triangular, concentrated loads and concentrated moments.

Loads can be converted into area loads via the design specification "per beam" and the definition of a beam spacing.



Interfaces to other applications

- Three-dimensional frame RSX+ (alternative calculation)
- Multi-span beam BTM+ / STM+ / HTM+ / HNV+ (alternative calculation)
- Forwarding of the bearing loads to the support programs B5+, STS+ and HO1+ as well as TB-Timber Compression, Steel Plate TB-HHS.

Add-on modules

HTM-2 Biaxial

HTM-S Stability

Data entry - general operating instructions

Assistant

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.

Self-defined items can also be imported as templates. Saving as a template is done via ▸ File ▸ Save as via ▸ the option “Use as Template”.

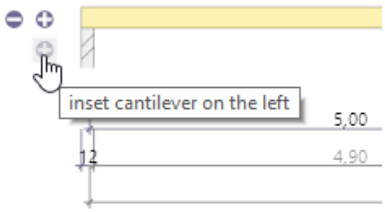
Data entry via the assistant:

- Loading uniaxial or biaxial
- Type of timber
- Material standard
- Strenght class
- 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: *You can use ▸ File ▸ Settings to change some basic settings for the assistant, color display or units of measurement.*

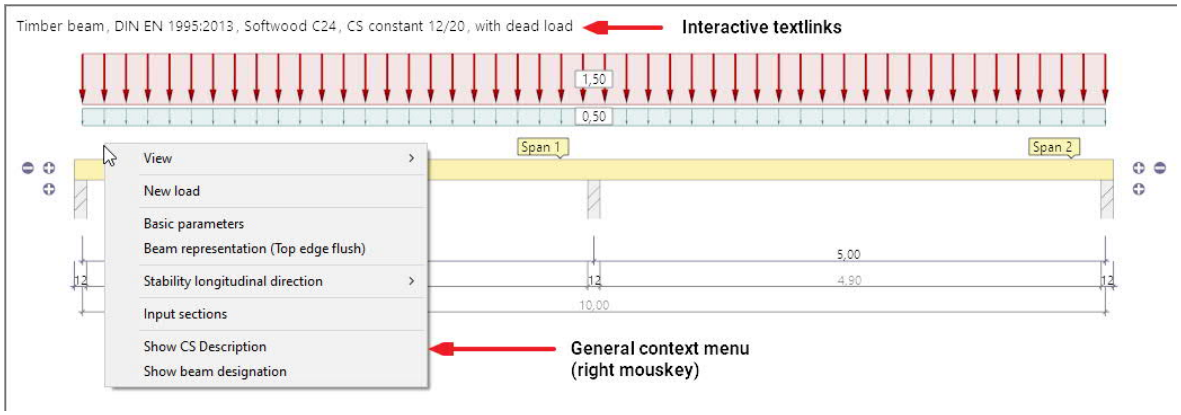
Graphical user interface

The graphical user interface is structured in such a way that all entered data can be accessed directly in the graphics window. For example, dimensions or load values can be directly clicked and changed. Other inputs acan be called up via the general context menu (right-click on an empty graphic area) or via the context menus of the individual objects (span, support, load...) or via the interactive texts in the upper left corner. Spans and cantilevers can be added 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 "interactive graphics" in the basic operating instructions.



Interactive dimensional chains

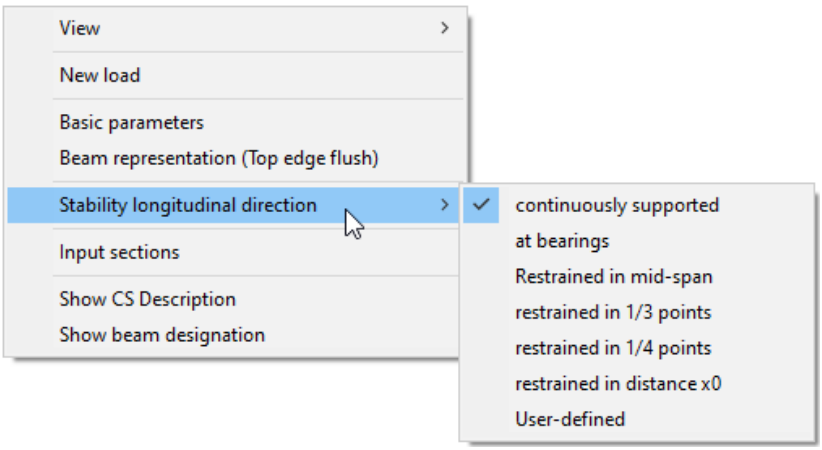
As in all Plus programs, the dimensional values are editable also in HTM+ and can be changed directly on the graphic screen.

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. 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 load cases, settings or visibility that are not relating to a particular graphical object, such as view functions, stability, input sections, etc..

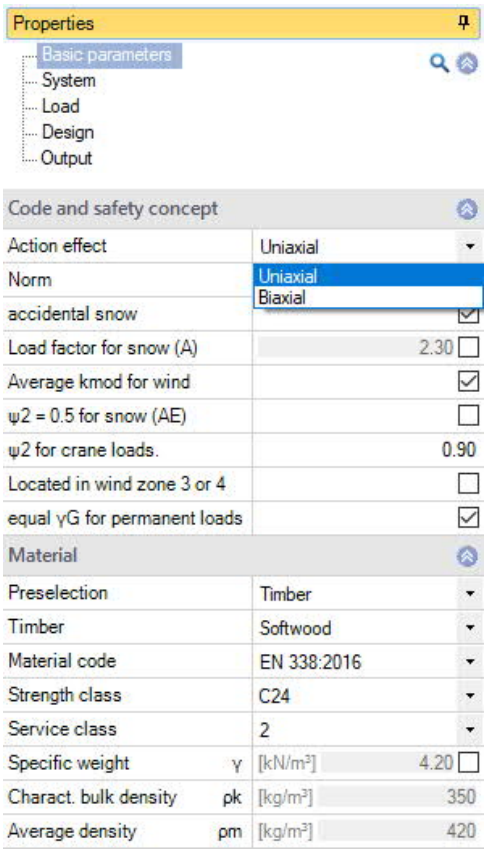


Interactive textlinks

The texts displayed in the top left section of the graphic screen are interactive as in all PLUS programs and can be clicked on. This allows you to display dialogs in the graphic, which are otherwise only accessible via the menu on the left. See also "interactive graphics" in the basic operating instructions.

Basic parameters

Standard and safety concept



Action effect — uniaxial or biaxial (HTM-2 add-on).

Standard (Norm) — definition of the design standard and its national annex.

accidental snow — when you check this option, snow loads are included as accidental action in addition to the typical design situations.

Load factor for snow (A) — this factor is used to determine the accidental snow load related to its characteristic value. This factor can be freely specified (mark the option to enter the value) or automatically determined by the program.

Average kmod for wind — If the option is selected, the modification coefficient kmod for wind is used as the mean value for the classes of the load duration short and very short.

ψ2 = 0.5 for snow (AE) — If the option is selected, the combination coefficient ψ2 for the action of snow is raised to a value of 0.5 in the earthquake (AE) design situation.
(See introductory decrees of the federal states (germany), e.g. Baden-Württemberg).

ψ2 for crane loads — determines the combination coefficient ψ2 for crane loads (ratio of permanent share to total crane load).

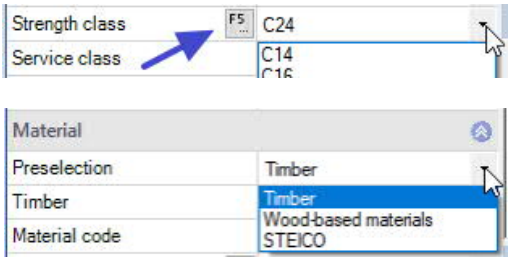
Located in wind zone 3 or 4 — Select this option if the building is located in wind zone 3 or 4. In this case the action 'snow' is not considered as an accompanying action to the leading action 'wind'.

Equal γG for permanent loads — if this option is checked, all permanent loads or load cases are applied together with the same partial safety factor (γG,sup or γG,inf), otherwise permanent loads are combined independently with their lower and upper partial safety factors.

Material

Input of the material values for softwood, hardwood or glulam. For the strength class user-defined values can be defined using the F5 key. Furthermore, the service class and the density can be set. The density is determined automatically depending on the selected wood strength, but can also be freely defined.

With DIN EN 1995: 2013 you first select timber / wood-based materials / STEICO via a preselection. Laminated veneer lumber (LVL22 C - LCL 80 P) and "STEICO" (LVL R, LVL RL, LVL Rs and LVL X) are available.



Structural system

You can enter spans / segments / cantilever arms / supports / joints etc. directly in the graphic using the context menu or using a table (tabs below the graphic).

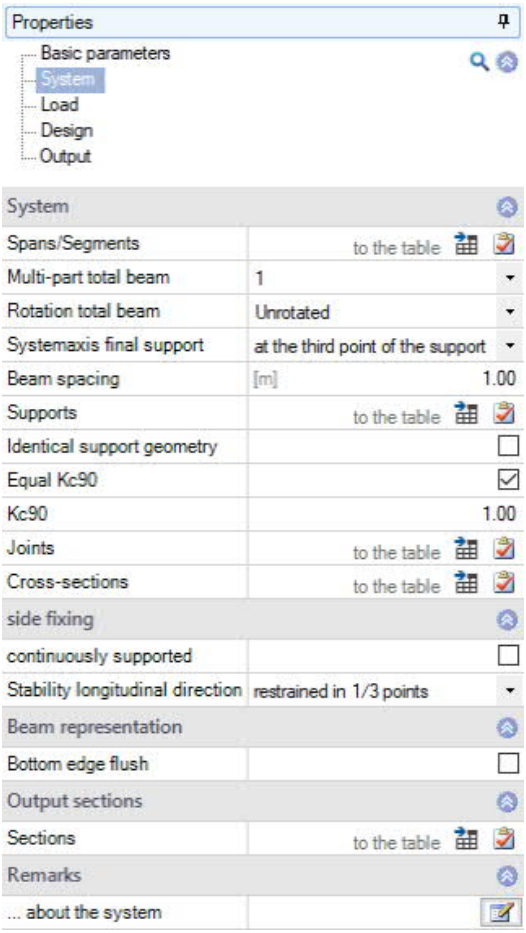


Table → Spans / Segments

Multipart total beam — here the entire beam can be defined in several parts (up to 4 parts). In the Spans/Segments table, however, the individual segments can be defined differently - in this case "different" is displayed here.

Rotation total beam — as with multi-part design, the cross-section rotation can be set for the entire beam or under Spans/Segments for individual segments.

System axis end support — the system axis of the end supports can be in the third point or in the middle of the support.

Beam spacing — the beam spacing is taken into account if calculation should include the affected width (see "per beam" in the load table).

Table → Supports

Identical support geometry

Ticking this option equates the width/depth of all supports.

Equal Kc90

By ticking this option, a line for entering a common kc90 value is displayed. Otherwise, this value can be defined directly in the table (differently) for each support.

KC90 Lateral pressure coefficient kc90 according to Chapter 6.1.5 for the verification of the bearing pressure. Press the F5 key for a selection dialog (Fig. Right).

Table → Joints

Table → Cross sections

Explanation for the → Lateral restraints (stability analysis / HTM-S add-on)

Beam representation

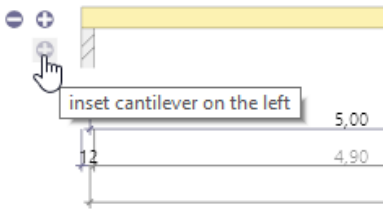
In the case of different cross-sections, you can select between flush lower or upper edge for the graphic representation. This setting has no influence on the calculation and is only used for graphic representation.

Table → Output sections

Input options in the graphic

Spans and cantilevers

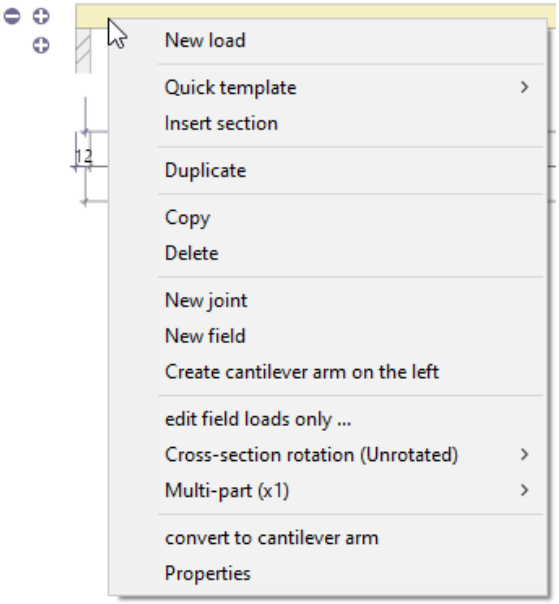
Spans and cantilevers can be inserted/removed directly in the graphics window using the +/- icons.



After right-clicking on the span in the graphic (context menu), functions are available as they are also described for the tabular entry (below). For example, the parameters for this span can be edited via "Properties".

Further input functions in the graphic

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



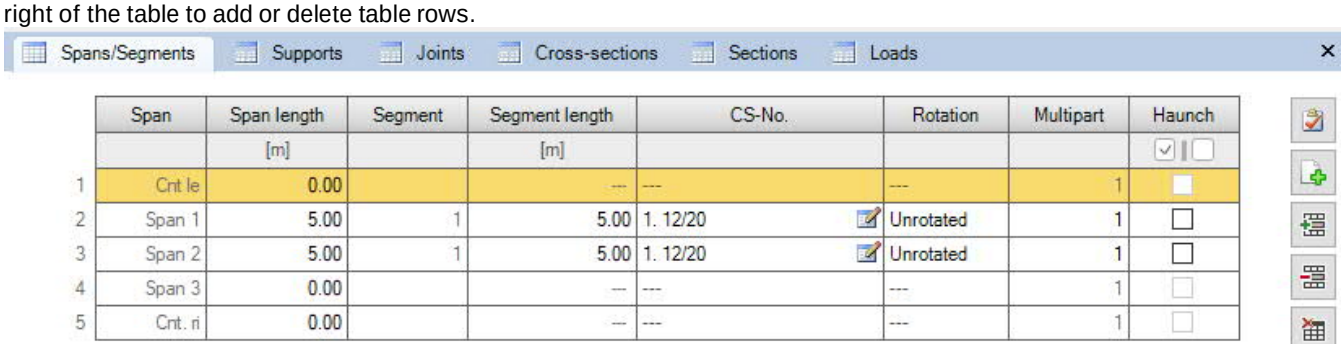
Note: The other objects in the graphic (supports, loads, etc.) also have their own context menus, which you can use to quickly access the desired function.

For graphical input in the PLUS programs, see also the Basic operating instructions-PLUS.

Spans / Segments

Input of spans/segments via table

To enter data via tables, click on the "Span/Segments" tab 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.	Rotation	Multipart	Haunch
1	Cnt le	0.00		---	---	---	1	☐
2	Span 1	5.00	1	5.00	1. 12/20	Unrotated	1	☐
3	Span 2	5.00	1	5.00	1. 12/20	Unrotated	1	☐
4	Span 3	0.00		---	---	---	1	☐
5	Cnt. ri	0.00		---	---	---	1	☐

Span — automatic designation of the individual spans/cantilever arms. Activate a line by clicking.

Span length — enter the length of the individual spans/cantilever arms.

Segment — Consecutive segment numbering per span.

Segment length — A span can be divided into several segments. As soon as you enter a section length smaller than the span length, a new line is automatically inserted for the following segment with an automatically adjusted remaining length. This segment can also be subdivided again in the same way.
Tip: You can also make the division directly in the graphic using the context menu.

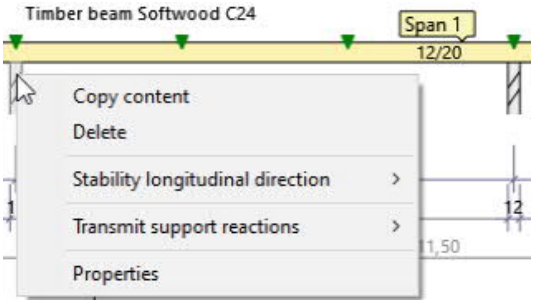
CS-No. — Each cross-section is identified with a serial number, followed by the cross-section dimensions. To define a (new) cross section, click the edit button. Here you can define a new cross-section in a separate dialog, whereby reinforcements timber/steel or timber/timber are also possible.
To select an existing cross-section, simply click in the field and open the selection list.

Rotation — The cross-section of each section can be rotated (0°/90°/180°/270°). A rotation of the entire beam can be entered in the menu tree on the left.

Multipart — The cross-section of each section can be defined in multiple parts (maximum of 4 cross sections next to each other). To define the entire beam in multiple sections (all segments are then defined with the same number of cross sections), enter this in the menu tree on the left. In principle, two sections next to each other are treated as if there were 2 members with one section each next to each other, each of which is subjected to half the load.

Haunch — Tick this option to define a haunch for a span / cantilever / segment. An additional line is automatically inserted under the corresponding span / segment in which you can specify the cross-section at the right end.

Supports



Graphical input / editing of the supports

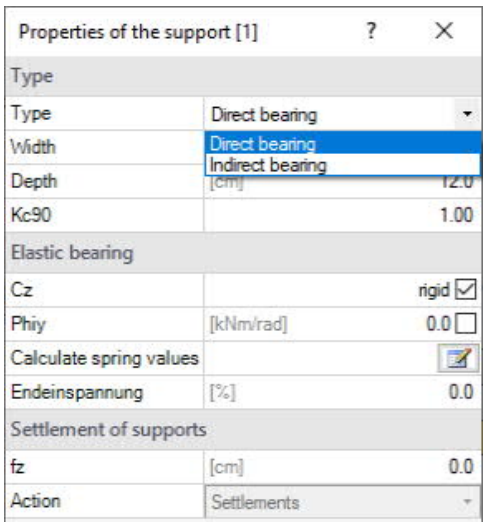
The bearing properties are called up by double-clicking on the support or by right-clicking and selecting the appropriate option in the context menu. In this section, it is also possible to delete supports or transfer the properties of the support to another support using the "Copy contents" function. Furthermore, the lateral restraints for the stability analysis (HTM-S add-on) can be defined here. Alternatively, you can also use the table ("Support" tab, see table entry below).

Transfer support reactions

The support loads can be transfered to the support programs B5 +, STS + and HO1 + as well as TB-HHS for further calculation: Right-click on the support ▸ Transmit support reactions ▸ choose the program. See also "Output".

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

Moreover, a column settlement can be pre-set for the individual supports.



Kc90 Lateral pressure coefficient. See explanation under System.

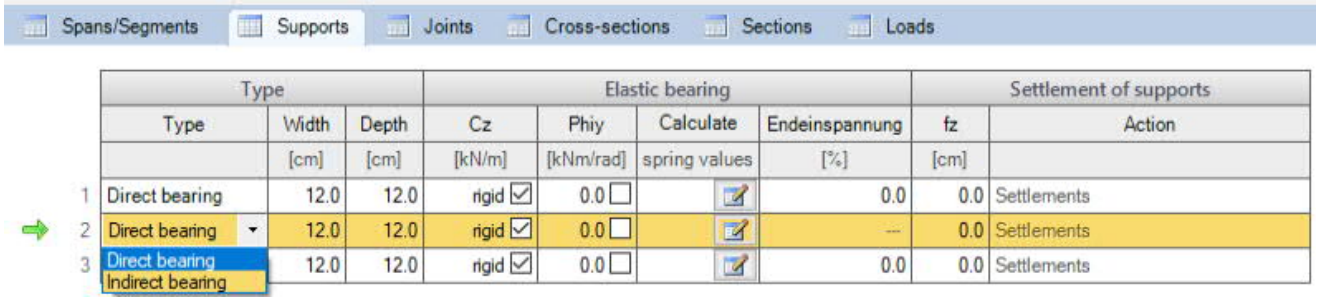
Calculate spring values

Alternatively, also the spring values of a single column that can be defined underneath and/or above the beam can be calculated by the program and then be used for the beam calculation (click on the button 'Calculate spring values'). In a separate dialog, tick the appropriate options (Coil spring/Torsion spring) for the calculation and enter the parameters. The spring values to be accepted (C, Phi) can also be edited if necessary.

End clamping Clamping at end supports can be entered as a percentage.

Input via table / editing of the supports

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

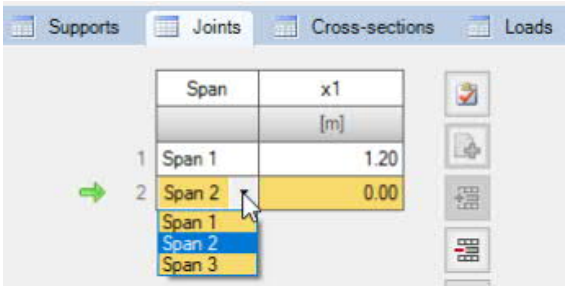


	Type			Elastic bearing				Settlement of supports	
	Type	Width [cm]	Depth [cm]	Cz [kN/m]	Phiy [kNm/rad]	Calculate spring values	Endeinspannung [%]	fz [cm]	Action
1	Direct bearing	12.0	12.0	rigid ☑	0.0 ☐		0.0	0.0	Settlements
2	Direct bearing	12.0	12.0	rigid ☑	0.0 ☐		---	0.0	Settlements
3	Direct bearing / Indirect bearing	12.0	12.0	rigid ☑	0.0 ☐		0.0	0.0	Settlements

Joints

In the graphic, you can select “new joint” in the context menu of a span. The joint is displayed as a small circle in the span and you can now enter the distance to the support in the dimension also shown in the graphic.

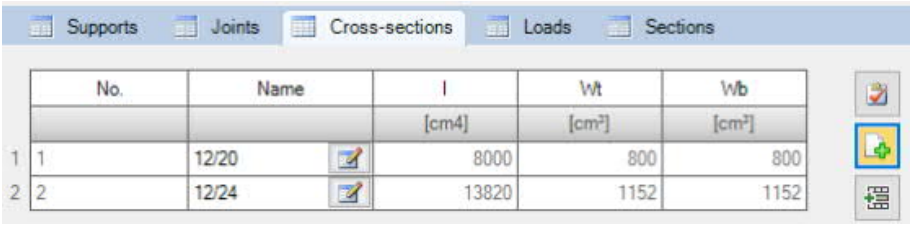
In the joint table, first click on the plus button to add a new line for the joint, then select the desired span number and enter the distance X1 to the left span start.



Cross-sections

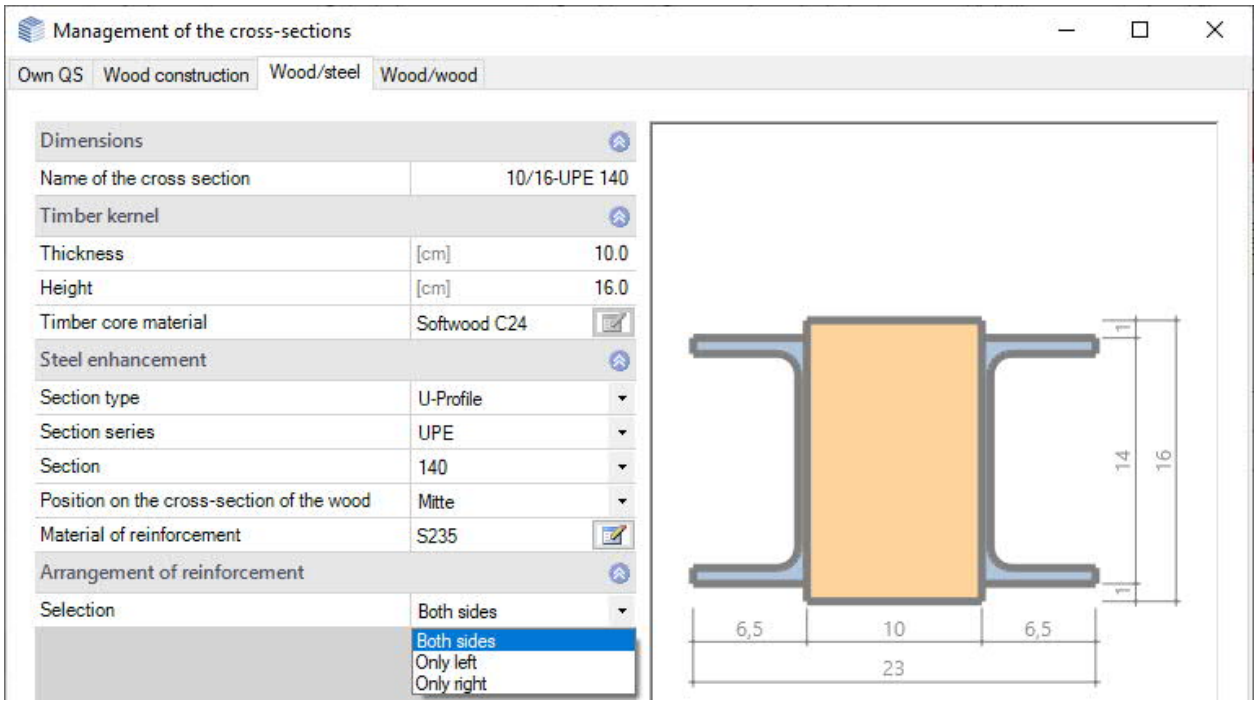
Click on the "Cross-Sections" tab under the graphic.

You can define multiple cross-sections here. Cross-sections that are reinforced one-sided/both-sided can also be selected (tabs timber/steel, timber/wood).



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

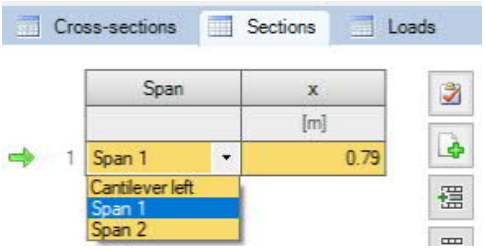
Then call up the cross-section dialog via the edit button  ("Name" column). Various profiles and materials can be selected for steel reinforcement, whereby the reinforcement can be arranged at the top/middle/bottom (Position on the cross section...).



Further operating instructions for the cross-section dialog can be found in the document Select - edit cross section - PLUS.

Sections

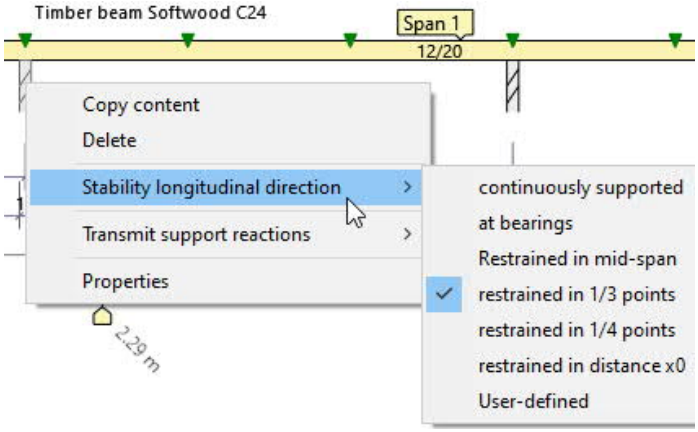
User-defined output sections can be defined here. A table of characteristic internal forces max/min is automatically generated for each action group.



Lateral restraints (stability)

Using the context menu, you can also define the restraints on the cross-section for the stability analysis (HTM-S add-on).

Instead of a continuous restraint, the position of restraints in the longitudinal direction of the beam and on the cross-section can be defined.

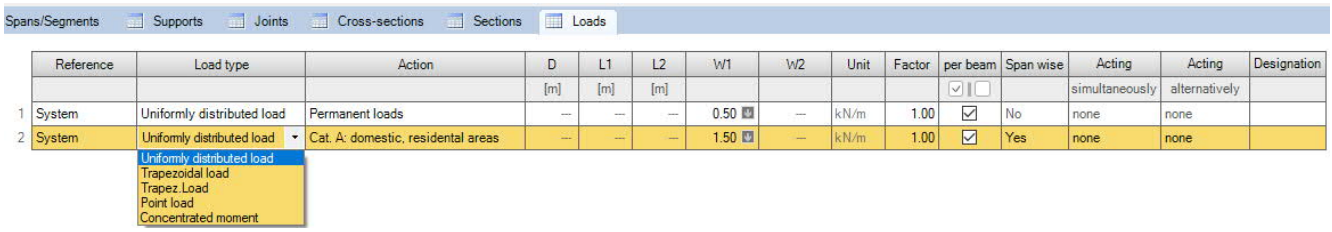


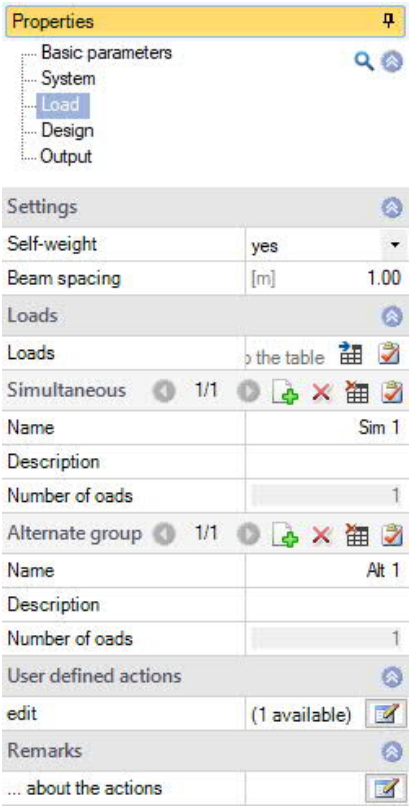
Position in length direction

- at bearings (only on the supports)
- additionally, in the centre of the span
- in the third or quarter points of the spans
- at a distance of X0 - a uniform spacing can be specified here.
- user-defined free definition of restraints. Use the "Intermediate support" tab to specify the distances of the restraints to the left-hand end of the beam or cantilever.

Loads

Select first whether self-weight should be included in the calculation or not. The beam spacing is taken into account if calculation should include the affected width (per beam). Via the 'Loads' tab, the load table is displayed.





Reference — allows you to select whether distance A refers to the left end of the beam (structural system) or to the left end of the respective span.

Load type — selection of the load type: uniformly distributed, trapezoidal, triangular, concentrated load or concentrated moments.

Action — select an action from a list.
In the left menu you can also create / edit your own user-defined actions.

D, L1 / L2 — D is the distance from the start of the load to the selected reference (see above, left end of the beam for the system or left end of the cantilever arm or field).
L1 is the load length of a trapezoidal load. In the case of triangular loads, the right and left lengths of the two triangular sections are specified via L1 and L2.

W1 / W2 — entry of the load value (W1) or, in the case of a trapezoidal load, also the second load value W2.
A load value compilation can be called up using the "arrow symbol" .

Note: — *check the loads in the graphic. Tip: Move the mouse pointer over a load value to view details.*

Unit — line load (kN/m) or area load (kN/m²) - see column "per beam".

Factor — multiplication factor for the load ordinates

per beam — by default, the option "per beam" is selected, i.e. the entered load (W1 / W2) is applied to this bar (for line loads, column unit = kN/m) without taking the bar spacing into account.
If this option is deactivated (no tick), the beam spacing is taken into account. The load ordinates are linked with this spacing and the load values are then calculated with the affected width (for line loads, column unit = kN/m²).

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

Acting simultaneously — the loads of a simultaneous group always apply simultaneously.

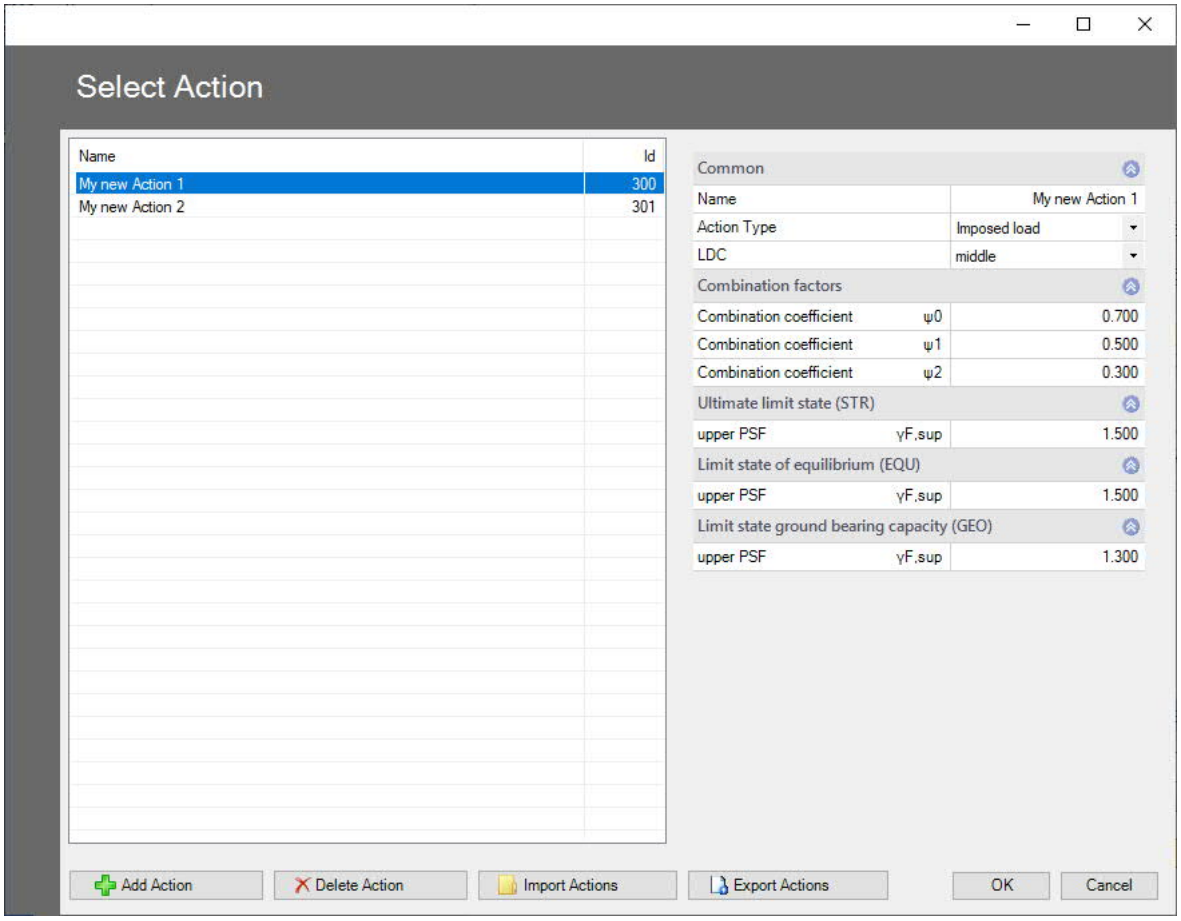
Acting alternatively — loads of an alternative group are always applied individually and are not superimposed. It should be noted that with the additional selection "span-wise", each individual span with this load is already considered as an alternative. If the load is defined as "not span-wise", the entire load is set as an alternative to another load in the same alternative group.

Note — *simultaneous- / alternative groups: Select "New group" to create a group with a consecutive index (Sim 1, Sim 2, etc.). You can also give descriptions to the groups in the menu tree on the left, which will then appear in the printout.*

Designation — allows you to enter brief comments on the load. They are included in the output.

User defined actions

In the left menu under “Load” you have the option to define and save your own actions (“user-defined actions”). Click the Edit button to bring up the following dialog.



Add a new action

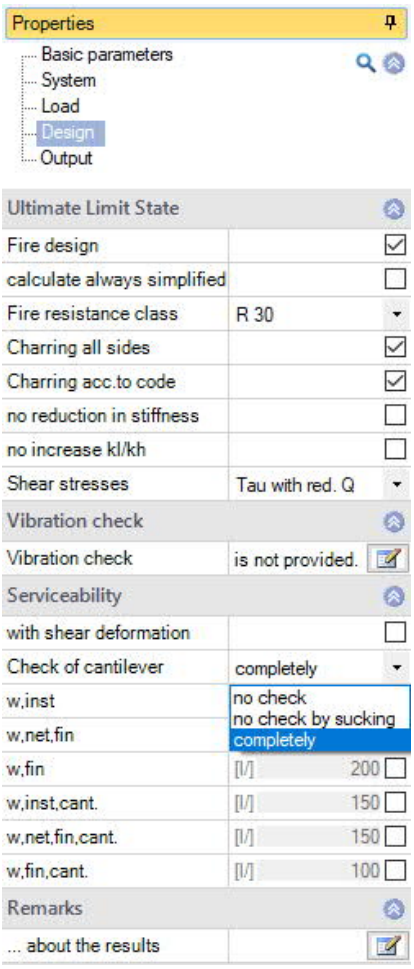
You can generate a new entry in the list using "Add action". Name the new action in the right-hand area (if the name already exists, the program will automatically add an extension (*)). Change the parameters as desired and confirm at the end of all entries with the OK button. A defined action can of course also be changed / edited later - please mark the corresponding line in the list on the left.

Export/Import

You can export the list of actions as an * .act file and import it again into other programs.

Design

In this section, you can optionally enable fire design and the oscillation analysis and also control the shear stress analysis.



Ultimate Limit State

Fire design

Optional verification under fire exposure. The corresponding data-entry fields ‘Verification method’, ‘Fire resistance class’, ‘Charring all sides’ and ‘Charring rates’ are displayed. If the "Calculate always simplified" option is selected, the method with reduced cross-sections is always used for the verification procedure for the fire design.

No reduction in stiffness

Depending on the selected standard (NA), the modulus of elasticity of compression-loaded components is reduced by means of *kdef* in stability verifications if the proportion of permanent loads in the total load is large. This leads to smaller (less favorable) buckling coefficients *kc*.

No increase kl/kh

Specifies whether there should be no increase with the factor kl/kh.

Shear stresses

Consideration of the shear force when calculating the shear stress

Tau with red. Q = reduced shear force

- Tau with red. Q (support front edge)
- Tau with full Q = full shear force

Vibration check

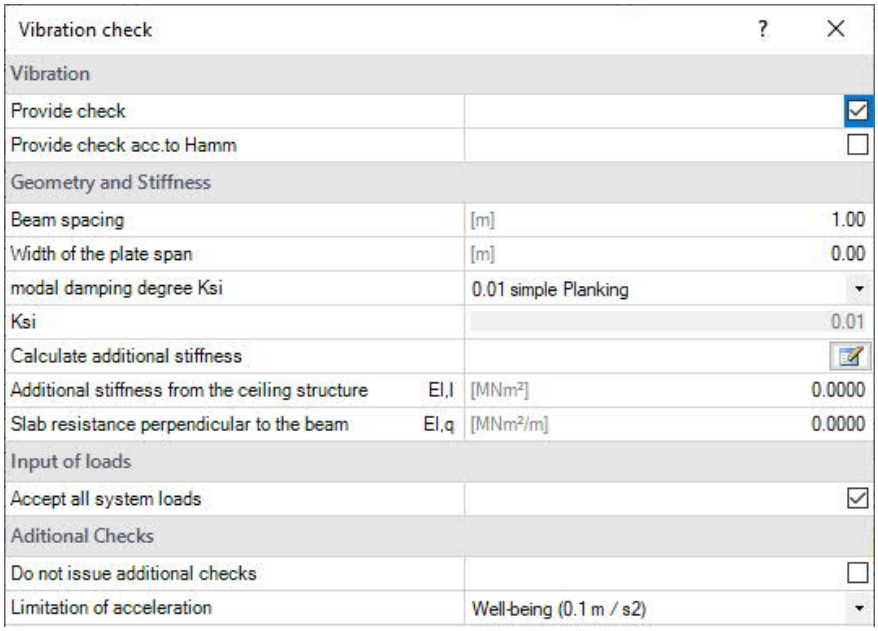
You can select the oscillation analysis on the "Design" tab.

this button allows you to access the dialog for the oscillation analysis.

Check the desired option for the oscillation analysis.

Analysis in accordance with Hamm:

Hamm, P.; Richter, A.: Bemessungs- und Konstruktionsregeln zum Schwingungsnachweis von Holzdecken. Symposia on timber construction 2009. Leinfelden-Echterdingen.



Geometry and stiffness

- Beam spacing — beam spacing for area loads
- Width of the ceiling span
- Modal damping ratio Ksi
- Additional stiffnesses — in a separate dialog, you can define the additional stiffnesses
- EI,l — displays the additional stiffness from the ceiling structure
- EI,q — displays the ceiling stiffness perpendicular to the beam

Load specifications

You can accept the system loads or optionally enter the loads manually (uncheck the box).

g0 permanent area load

q0 variable area load

You can select an action group in the selection list on the right.

Additional checks

Do not issue additional checks — special examinations at frequencies greater than 8 Hertz are not issued.

Limitation of acceleration — predefined limit values or self-defined limit value (for EN 1995)

Notes:

f resonance frequency

f > 8 Hz: in this case, the following requirements should be complied with for residential ceilings:

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

- limitation of the speed of oscillation v caused by the unit pulse $v \leq \beta^{(f1 \cdot \zeta - 1)}$ m/(Ns²)

f ≤ 8 Hz: in this case, a separate examination should be carried out for residential ceilings.

In this connection, two additional verifications are performed that correspond to the approach described in reference /1/.

/1/ Blaß, H. J. Erläuterungen zu DIN 1052-2004-08 , Bruderverlag March 2005

- limitation of the speed of oscillation v caused by footfall $v \leq 6 \cdot \beta^{(f1 \cdot \zeta - 1)}$ m/(Ns²)

- limitation of the acceleration $a_{vert} \leq 0{,}1 \text{ m/s}^2 - 0{,}4 \text{ m/s}^2$

Serviceability

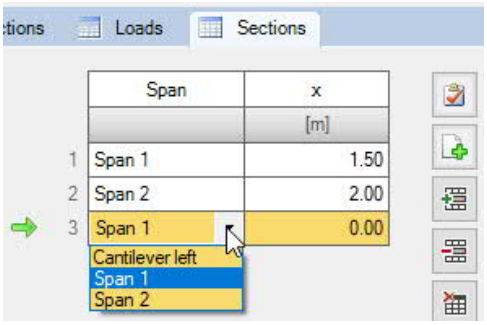
You can optionally take deformation by shear into account.

Furthermore, data-entry fields for the limit values (LV) of the deformations are available in this section:

w,inst,cantil	LV of the elastic deflection of a cantilever beam
w,net,fin,cantil	LV of the summarized elastic deflection and creep deformation of a single cantilever
w,fin,cantil	LV of the final deformation of a cantilever
w,inst	LV of the elastic deflection of a single-span girder
w,net,fin	LV of the summarized elastic deflection and creep deformation of a single-span girder
w,fin	LV of the final deformation of a single-span girder

Sections

You can use the "Sections" tab to define output sections by specifying the span and a distance X to the start of the span on the left (click + button for a new entry).

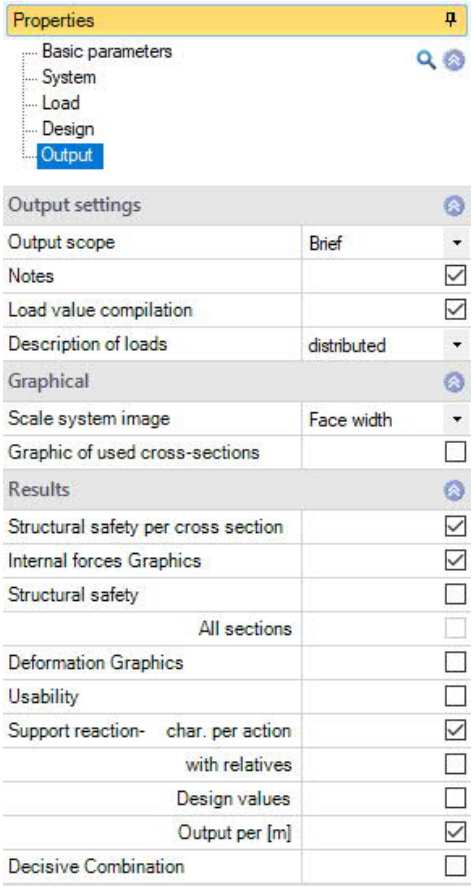


Graphic input

You can also create several sections (mouse clicks) directly in the graphic using the context menu (right mouse button, input sections). The position can be shifted appropriately with the mouse. Confirm / finish the entry with a right click.
The sections are shown as symbols in the graphic and can also be moved later.
The sections can be shown / hidden in the result graphics.

Output

Scope of output / Calculation / Results



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

After the calculation, the load is displayed in the bottom right-hand corner of the graphics window and provides a good overview of the efficiency of the structural system entered.

Display/hide

In the upper toolbar, the individual representations in the graphic can be switched on or off.

Load filter

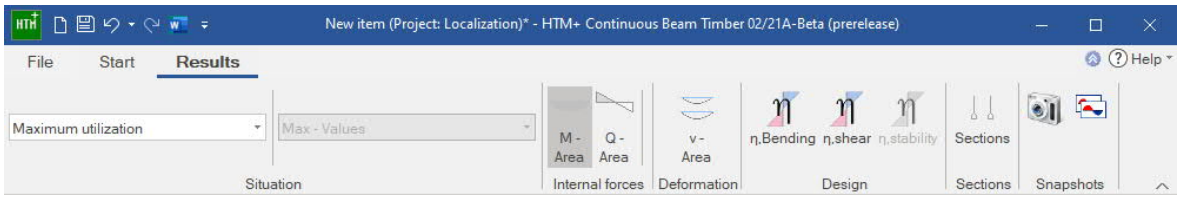
The Load filter button can be used to filter loads according to actions and groupings (alternative / simultaneous). The selection is then highlighted visually / in color in the graphic. In this way loads can be checked clearly and edited in the graphic. The filter is canceled again with "Deactivate".

Results

Via the ‘Results’ tab, you can display the different result graphs.

The defined output sections can be shown and hidden.

You can use the camera symbol to take a snapshot of the displayed graphic and give it a name. Use the right icon to display the list of recordings that can also be deleted here. These images are automatically included in the output.



Output scope

By checking the desired options, you can determine the scope of data to be put out. You can also define Output sections.

Output as a PDF document

Via the Document tab, you can displays the document in PDF and print it.

See also Output and printing

Transfer of support reactions

The support loads can be forwarded to the support programs B5+, STS+ and HO1+ as well as TB-Timber Compression, Steel Plate TB-HHS for further calculation:

Right-click on the support ▸ Transmit support reactions ▸ choose the program.

See also connected programs under "Application options".