

Reinforced Timber Beam HTV+

Table of contents

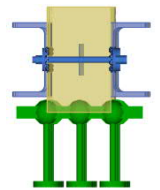
Application options	2
Calculation bases	3
Input - general operating instructions	4
Basic parameters	6
System	7
Spans/segments	9
Supports	10
Joints	11
Cross-section of timber core	11
Cross-section of reinforcement	11
Reinforcements left / right	12
Fasteners	12
Core failures	13
Sections	13
Lateral restraints (stability)	13
Loading	14
Design	16
Output	17
Output scope / calculation / results	17
Load transfer	18

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 HTV+ program calculates single and multi-span timber beams with variably definable reinforcements (timber or steel reinforced). The reinforcements can be stored and loaded separately from the timber core. Bolts, pins, threaded rods and connector joints can be selected as fasteners. Furthermore, systems with cantilevers can be defined.

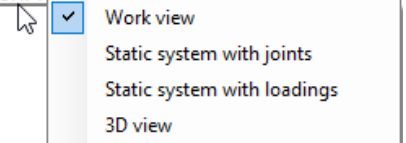


Cross-sections can be different. Spans can be divided into sections and joints are also possible. Superposition and design take place automatically.

The program is designed to work in a graphically interactive manner. Three different views of the system are offered for optimal input.

Timber beam, DIN EN 1995:2013, Softwood C24, CS

Graphic: Work view



Standards

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

Assistant

The Assistant can be used to make the entries necessary for a simple basic system. This basic system can then be easily modified and supplemented using graphical-interactive input.

Reinforcements/connections

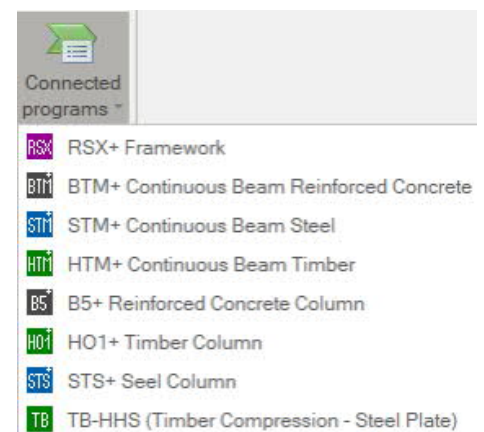
The lateral reinforcements can be connected to the timber core as a coupled system using a selection of fasteners. The timber core can be reinforced on one or both sides using a selection of steel sections or timber cross-sections. The supports of the different static systems can be freely selected and the loads can be set individually. The coupled system is calculated using a truss analysis and the loads are distributed across the fasteners.

Loads

Load types: uniformly distributed, trapezoidal, triangular, concentrated load and concentrated moment. Free selection of the load approach on the timber core or any reinforcement.

Interfaces to additional programs

- Direct transfer from the DLT+/HTM+ to the HTV+
- Spatial framework [RSX+](#) (alternative calculation)
- Multi-span beams [BTM+](#) / [STM+](#) / [HTM+](#) (alternative calculation)
- [Transfer](#) of the support loads to the column programs [B5+](#), [STS+](#) and [HO1+](#) as well as the toolbox Timber Compression Steel Plate [TB-HHS](#).



Calculation bases

The calculation method used is based on a strut-and-tie model in which the timber beam is flexibly coupled to the reinforcements at discrete points via the fasteners. This allows the loads and deformations of the timber beam and reinforcement to be calculated in each section and in each connecting element.

The actual support conditions and the actual load application can be taken into account precisely. This means that not only can the dimensions of the beams be determined, but the connecting elements can be used precisely in the places where the loads require it.

Based on the resulting internal forces, all timber cross-sections and fasteners are verified according to Eurocode 5 as well as the steel sections according to Eurocode 3.

Input - general operating instructions

Assistant

When you start the program, the [Assistant](#) window automatically appears.

The most important key data of the system can be quickly entered here, which can then be edited in the input area and/or in the interactive [graphic interface](#).

Self-defined items can also be imported here as templates. Saving as a template is done via **File** ▶ **Save as** ▶ mark the "Use as template" option.

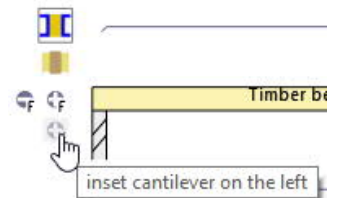
Entries in the Assistant:

- Type of timber
- Strength class
- Number of spans
- Span length
- Cross-section of timber core
- Reinforcement (section selection, arrangement on one/both sides)
- Fasteners
- Permanent line load
- Live 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 input

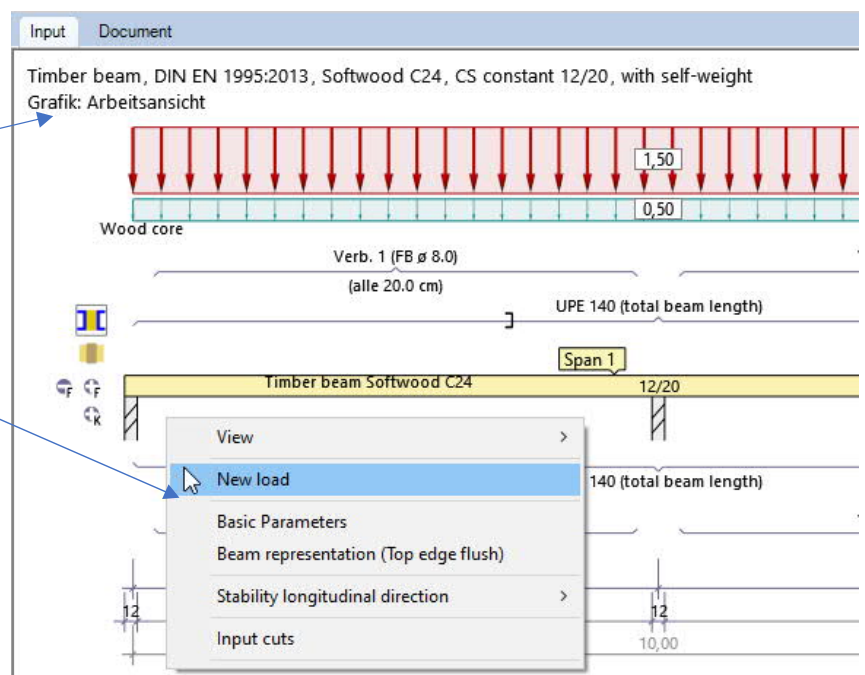
The graphical input is structured in such a way that all inputs can be accessed directly in the graphic window. For example, dimensions or load values can be clicked on and changed directly. Other entries are accessed via the general context menu (right click on an empty graphic area) or the [context menus](#) of the individual objects (span, bearing, load, etc.) or via the interactive texts at the top left. Spans and cantilevers can be added using the +/- symbols on the right and left.



See also "[interactive graphics](#)" in the operating basics.

Graphics: Here you can select three different views of the system for optimal input. Alternative selection via the System View icon in the top menu bar.

General context menu:



Interactive dimensional chains

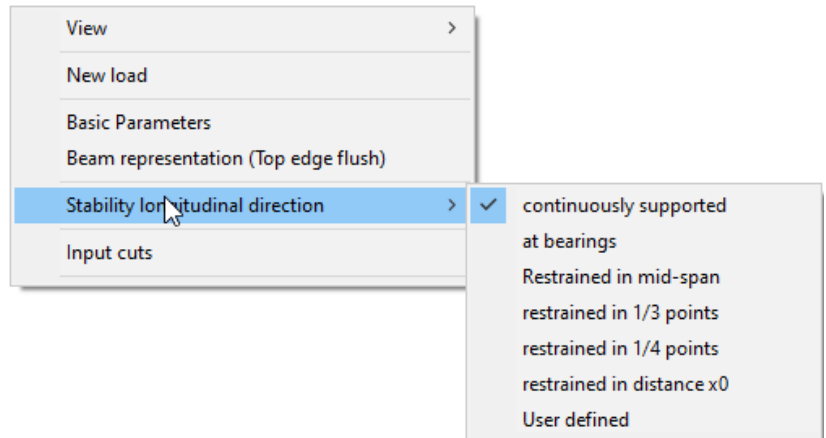
As in all Plus programs, the measurements are editable and can be changed directly in the graphics.

Tip: The span lengths can also be changed by moving a bearing. To do this, click on the bearing with the left mouse button; you can move the bearing while holding down the mouse button.

Context menu

There are appropriate context functions for each object (span, bearing, load, etc.). These functions are displayed via the right mouse button and, as the name suggests, are appropriate for the selected object.

A general context menu (fig. on the right) appears if no object is selected. Here you can find functions that have no representation of a graphical object, such as view functions, stability, sections, etc.



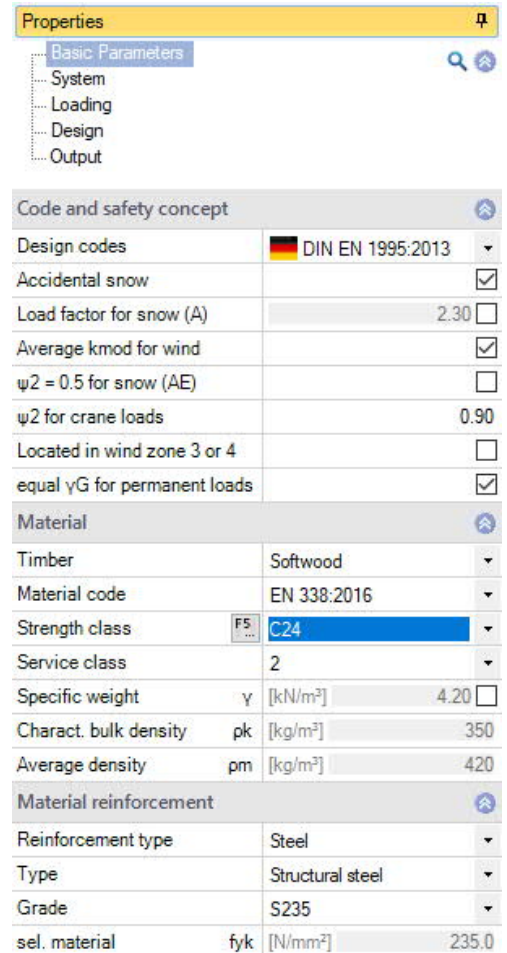
Interactive texts

The texts displayed in the graphics at the top left are, as in all PLUS programs, interactive and can be clicked on. This means that dialogs can be accessed in the graphics that would otherwise only be accessible via the left menu. See also [Operating basics](#).

Basic parameters

Standard and safety concept

Design codes/Standard	Definition of the design standard with national annex.
Accidental snow	If the option is checked, the snow loads are considered as accidental actions in addition to the usual design situations.
Load factor for snow (A)	This factor is used to calculate the accidental snow load based on its characteristic value. It can be freely specified (check option to enter the value) or determined automatically by the program.
Average kmod...	If the option is checked, the modification coefficient kmod for wind is set as the average for the short and very short load duration classes.
$\psi_2 = 0,5$ for snow (AE)	If the option is selected, the combination factor ψ_2 for the snow action is increased to the value 0,5 in the earthquake (AE) design situation. (See introductory decrees of the federal states, e.g. Baden-Württemberg).
ψ_2 for crane loads	Defines the combination factor ψ_2 for crane loads (ratio of permanent component to total crane load).
Wind zone location 3 or 4	Check this option if the building location is in wind zone 3 or 4. In this case, the action 'snow' is not considered as an accompanying action to the main action 'wind'.
equal γ_G ...	If the 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 of each other with their lower and upper partial safety factors.



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

- Code and safety concept:**
 - Design codes: DIN EN 1995:2013
 - Accidental snow:
 - Load factor for snow (A): 2.30
 - Average kmod for wind:
 - $\psi_2 = 0.5$ for snow (AE):
 - ψ_2 for crane loads: 0.90
 - Located in wind zone 3 or 4:
 - equal γ_G for permanent loads:
- Material:**
 - Timber: Softwood
 - Material code: EN 338:2016
 - Strength class: **C24** (with 'F5...' key)
 - Service class: 2
 - Specific weight: γ [kN/m³] 4.20
 - Charact. bulk density: ρ_k [kg/m³] 350
 - Average density: ρ_m [kg/m³] 420
- Material reinforcement:**
 - Reinforcement type: Steel
 - Type: Structural steel
 - Grade: S235
 - sel. material: f_{yk} [N/mm²] 235.0

Material

Selection and entry of the timber type/material standard/material values for softwood, hardwood or glued laminated timber of the timber core. User-defined values can also be defined for the strength class using the F5 key. Furthermore, the usage class and the specific weight can be set. This is determined automatically depending on the selected timber strength, but can also be defined yourself.

You can choose between steel and timber for reinforcement. For the definition of the reinforcement cross-sections (U/L profiles and flat steel or rectangle for timber), see "[Cross-section of reinforcement](#)".

System

You can enter spans/sections/cantilevers/supports/joints etc. directly in the graphics using the context menu (right mouse button) or via a table (tabs under the graphics).

[Spans/sections](#) to the table

System axis final support The system axis of the final supports can be at the third point or in the center of the support.

Beam spacing The beam spacing is taken into account if the influence width is to be taken into account (see under ["per beam" in the load table](#)).

[Supports](#) to the table

Identical support geometry
Checking this option sets the width/depth of all supports to be the same.

Equal Kc90
Checking this option displays a line for entering a common kc90 value. Otherwise, this value can be defined (differently) for each support directly in the table.

Kc90 Lateral pressure coefficient kc90 according to Chapter 6.1.5 for the verification of the support pressure. Press the F5 key for a selection dialog (Fig. right).

[Joints](#) to the table

[CS Timber core](#) to the table

[CS Reinforcement](#) to the table

[Reinforcement left/right](#) to the table

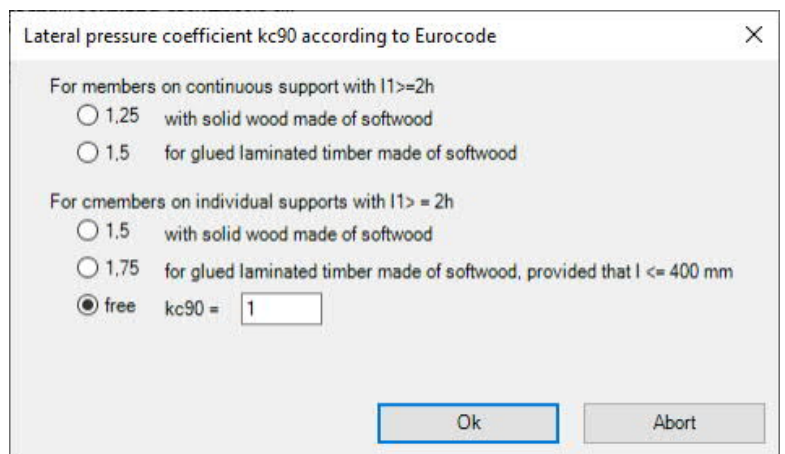
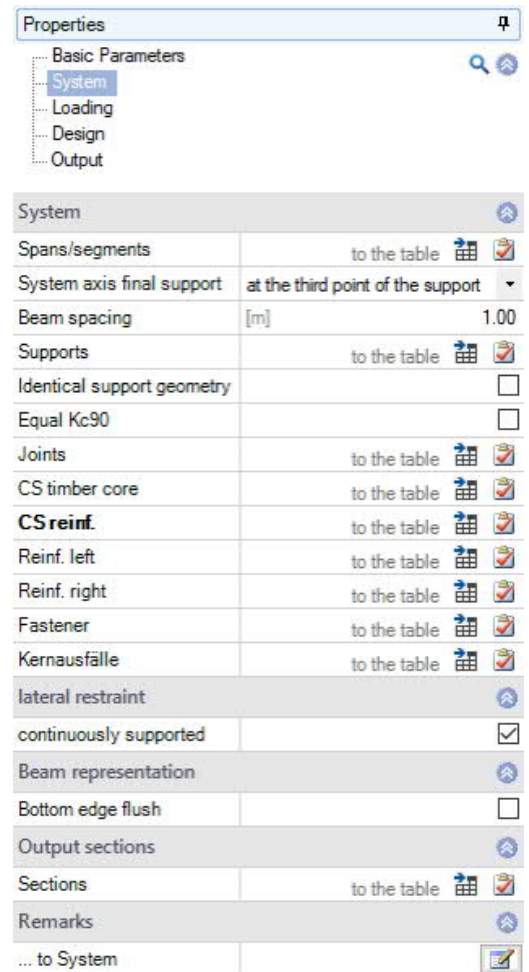
[Fasteners](#) to the table

[Core failures](#) to the table

Lateral restraint
Restrains on the cross-section for verification of stability – see [lateral restraints](#).

Beam representation
For different cross-sections, the graphical representation can be selected between a flush bottom or top edge. This setting has no influence on the calculation and is only used for graphical representation.

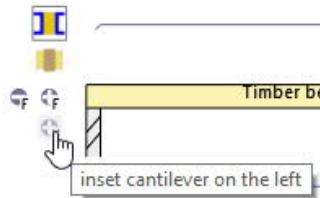
[Sections](#) to the table



Input options in the graphics

Spans and cantilevers

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



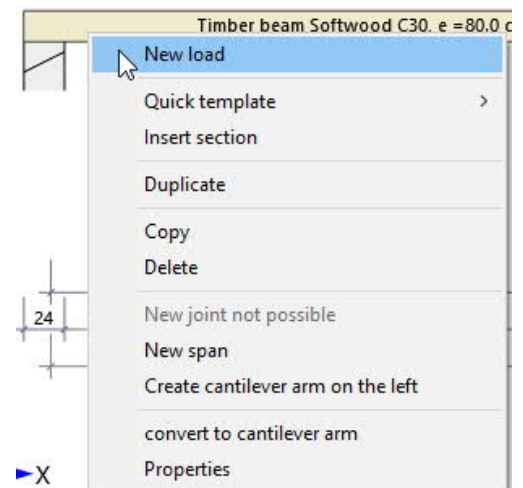
After right-clicking on the span in the graphics (context menu), functions as described for tabular input (below) are available. For example, the parameters for this span can be edited via "Properties".

Additional input functions in the graphics

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

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

For graphical input in the PLUS programs.
See also [Basic Operating instructions PLUS](#).



Spans/segments

Tabular entry of spans/segments

For tabular input, click on the "Spans/segments" tab below the graphics. You can [add or delete input lines](#) using the buttons to the right of the table.

Span	Span length [m]	Segment	Section length [m]	CS-No.	
1	Cnt. le	1.50	1	1.50	1. 12/20
2	Span 1	5.00	1	5.00	1. 12/20
3	Span 2	5.00	1	5.00	1. 12/20
4	Span 3	0.00		--	--
5	Cnt. ri	1.50	1	1.50	1. 12/20


Span Automatic designation of the individual spans/cantilevers. Activate a line by clicking.

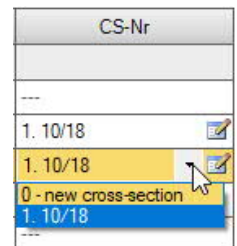
Span length Enter the length of the individual spans/cantilevers.

Segment Consecutive section numbering per span.

Segment length A bar can be divided into several sections. As soon as you enter a section length smaller than the span length, a new line is automatically inserted for the following section with an automatically adjusted remaining length. This section can also be divided in the same way.

Tip: You can also do the division directly in the graphics using the context menu.

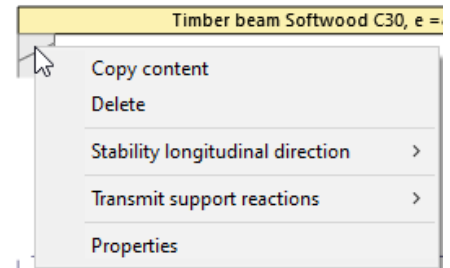
CS no. Each cross-section is designated with a serial number followed by the cross-section dimensions. To define a (new) cross-section, click on the edit button . Here you can [define a new timber cross-section](#) in a separate dialog. To select an existing cross-section, simply click in the span and open the selection list.



Supports

Graphical input/editing of supports

The support properties are accessed by double-clicking on the support or by right-clicking and selecting the appropriate option in the context menu. Here it is also possible to delete supports or transfer the properties of the bearing to another support using the "Copy contents" function. The lateral restrains for verification of stability can also be defined here. Alternatively, you can also use the entry via the table ("Support" tab, see Tabular entry below).



Load transfer:

The support loads can be transferred to the support programs B5+ / STS+ / HO1+ as well as to TB-HHS for further calculation: right-click on the support ▶ Forward support forces ▶ Program. See also "Output".

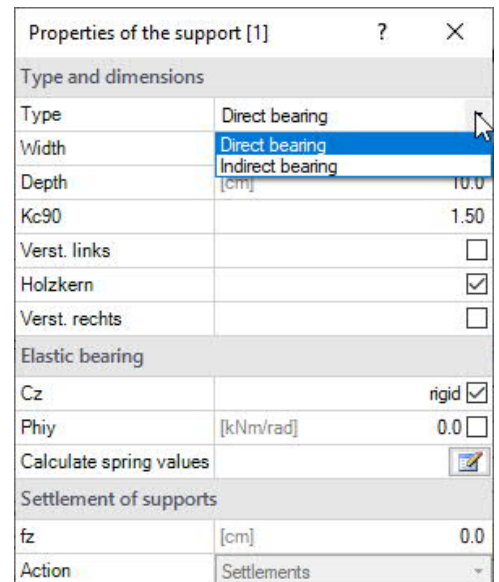
Bearings can be entered in the Z direction as well as for rotation around the y axis. There is the option of rigid bearing or entering a spring value. Options can be used to define which of the 3 members (reinforcement left/right, timber core) this bearing works for (standard: all 3).

In addition, a settlement of support f_z can be specified at the individual bearings.

Kc90 Lateral pressure coefficient. See explanation under [System](#).

Calculate spring values

Alternatively, the spring values can also be calculated by the program from a support that can be defined below and/or above the beam and then adopted 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 (travel/torsion spring) and enter the parameters. The spring values to be adopted (C, Phi) can also be edited if necessary.



Tabular entry/editing of supports

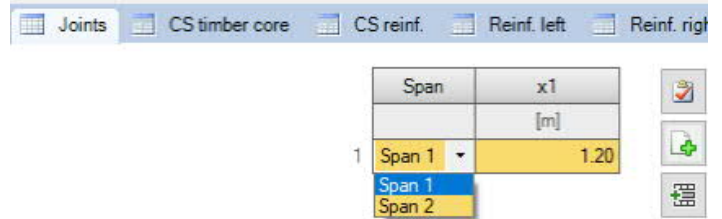
Unless you are using graphical input (see above), click the Supports tab below the graphics to open the table and enter the parameters there.

Type and dimensions								Elastic bearing			Settlement of supports	
Type	Width	Depth	Kc90	Reinf. left	Timber core	Reinf. right	Cz	Phiy	Calculate	fz	Action	
	[cm]	[cm]		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	[kN/m]	[kNm/rad]	spring values	[cm]		
1 Direct bearing	12.0	12.0	1.00	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	rigid <input checked="" type="checkbox"/>	0.0 <input type="checkbox"/>		0.0	Settlements	
2 Direct bearing	12.0	12.0	1.00	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	rigid <input checked="" type="checkbox"/>	0.0 <input type="checkbox"/>		0.0	Settlements	
3 Indirect bearing	12.0	12.0	1.00	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	rigid <input checked="" type="checkbox"/>	0.0 <input type="checkbox"/>		0.0	Settlements	

Joins

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.

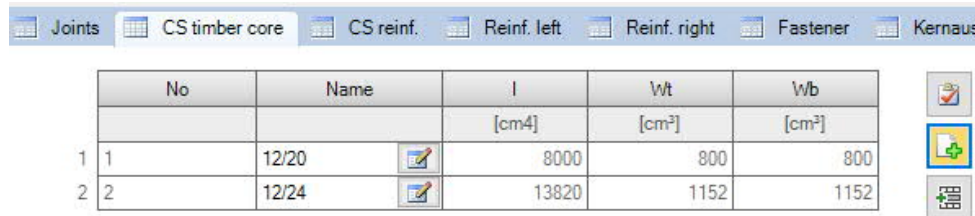
In the joint table, first click on the plus button to add a new row for the joint, then select the desired span number and enter the distance X1 to the start of the left span.



Cross-section of timber core

Click on the "CS timber core" tab below the graphics.

Here you can define several cross-sections (name, width and height).



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

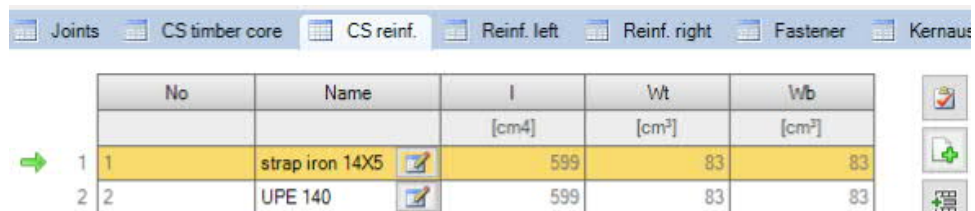
Then call up the cross-section dialog using the edit button ("Name" column).

Cross-section of reinforcement

In the "CS Reinf." tab you define the cross-sections for the reinforcement. Use the edit button to open the steel construction cross-section selection, where you select channel and L sections as well as flat steel and the desired section series. You can also define your own cross-sections here.

Different reinforcement cross-sections are also possible.

See document [Cross-Section Selection Plus](#).



Reinforcements left / right


Different reinforcement cross-sections can be selected for the entire beam length or for sections on the left and right. A 90° rotation can be selected for flat steel.

Reference	from [m]	to [m]	Reinforcement cross-section	Rotation
1 total beam length	0.00	13.00	1. UPE 80	---

Fasteners

The fasteners can be selected for the entire length of the beam or for sections.

Reference	from [m]	to [m]	Fastener	Distance A1 [cm]
1 free input	0.00	6.50	FB ø 8.0	20.0
2 free input	6.50	13.00	FB ø 10.0	20.0

An input dialog is called up using the edit button . Here you select the type.

- bolts,
- pins,
- threaded rod or
- connector joints.

Depending on the selected type, the appropriate input parameters are then displayed.

Verbindungsmittel		?	×
Fastener type			
Type		Fit bolt	▼
Fastener selection		Bolt	▼
		Fit bolt	▼
		Threaded rod	▼
		Connectors	▼
Strength class		4.6	▼
Washer	acc.to	EN ISO 7094:2000-12	▼
Fastener properties			
Diameter	d [mm]	20.0	□
Tensile strength	fuk [N/mm ²]	400.00	□
Asp	[cm ²]	2.5	□
Yield moment	MyRk [Nmm]	289641	□
Washer diameter	dsa [mm]	72.0	□
Washer diameter	dsi [mm]	22.0	□
Surmount/sinking	uv [mm]	0.0	□

Core failures

Input of partial core failures in the system. A partial failure of the timber beam can be simulated in user-defined areas. The area can be defined several times in freely selectable lengths per beam. The system must be load-bearing via the reinforcements.

Distance reference	Span	Distance [m]	Bezug der Öffnung	L [cm]
1 Distance from support	Span 1	0.10	Front edge	5.0

Distance reference The distance can be defined to the left support or to the front edge of the beam.

Span Selection of the field for the core failure

Distance/reference of the opening

Axis-related: Distance to the center of the core failure in relation to the respective field

Front edge: Distance of the start of the core failure in relation to the respective field

L Length of the core failure

Sections

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

Span	x [m]
1 Span 1	0.00

Graphical input

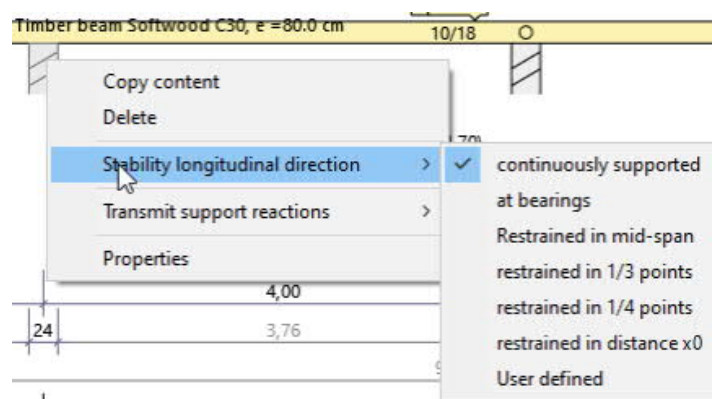
You can also create multiple sections (mouse clicks) directly in the graphic using the context menu (right mouse button, enter sections). The position can be moved appropriately using the mouse. Right-click to confirm/finish your entry. The sections are shown as symbols in the graphics and can also be moved later.

The sections can be shown/hidden in the [result graphics](#).

Lateral restraints (stability)

The restraints on the cross-section for the stability analysis can be defined using the context menu (click on a support with the right mouse button).

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



Position in the longitudinal direction

- only at the bearings

- additionally in mid span

- in the third or quarter points of the spans

- In distance X0 -

A uniform distance can be specified here.

- User-defined

Free definition of the restraints. Using the "Intermediate bearing" tab that appears, enter the distances between the restraints and the left end of the beam or cantilever.

Loading

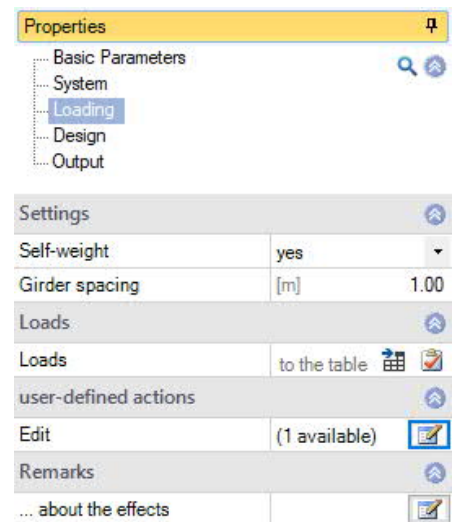
In the left menu under "Loading", select whether you want to calculate with or without your self-weight.

The beam spacing is taken into account if the influence width is to be taken into account (see "per beam" below).

Position	Reference	Load type	Action	D	L1	L2	V1	V2	Unit	Factor	per beam	Span by span	Simul	Acting	Description
				[m]	[m]	[m]					<input type="checkbox"/>		taneous	alternatively	
Timber core	System	Uniformly distributed load	Permanent loads	---	---	---	0.50	---	kN/m	1.00	<input checked="" type="checkbox"/>	No	none	none	
Timber core	System	Uniformly distributed load	Cat. A: domestic, residential areas	---	---	---	1.50	---	kN/m	1.00	<input checked="" type="checkbox"/>	Yes	none	none	
Reinf. left															
Timber core															
Reinf. right															

The load table is displayed in the "Loads" tab, in which you enter the other parameters.

- Position** Selection of whether the load is based on the timber core or the reinforcement on the left/right.
- Relation** Here you select whether the distance A refers to the left end of the beam (system) or to the left end of the respective span. For reinforcements, the respective reinforcement on the left or right is selected if there are several reinforcements on each side.
- Load type** Selection of the load type: uniformly distributed, trapezoidal, triangular, concentrated load and concentrated moment.
- Action** Selection of the action from a list. You can also create/edit [user-defined actions](#) yourself in the left menu.



- D, L1 / L2** D is the distance from the start of the load to the selected reference (see above, left final beam for system or left final cantilever or span or final reinforcement). L1 is the load length of a trapezoidal load. For triangular loads, the right and left lengths of the two triangle sections are specified via L1 and L2.

- V1 / V2** Enter the load value (V1) or, for trapezoidal loads, also the second load value V2. A [load value compilation](#) can be called up using the "arrow symbol"

Note: Check the loads entered in the graphics. Tip: Hover 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 values defined under V1 or V2.
- per beam** By default, the option "per beam" is selected, i.e. the entered load (V1 / V2) is applied to this beam - without taking the beam spacing into account (for line loads, unit column = kN/m). If this option is deactivated (no checkmark), the beam spacing is taken into account. The load coordinates are linked to this distance and the load values are then calculated using the influence width (for line loads, unit column = kN/m²).

- Span wise** Here you select whether loads that are entered across several spans should be applied by the program on a span by span basis or should only be taken into account in combination.

- Simultaneous** Loads of a simultaneous group are always assessed combined.
- Alternative** Loads of an alternative group are always applied individually and are not overlaid. It should be noted that with the additional selection "Span wise", each individual span with this load is already considered alternatively. If the load is defined as "not span wise", the entire load is set as an alternative to another load from the same alternative group.

Design

Structural safety

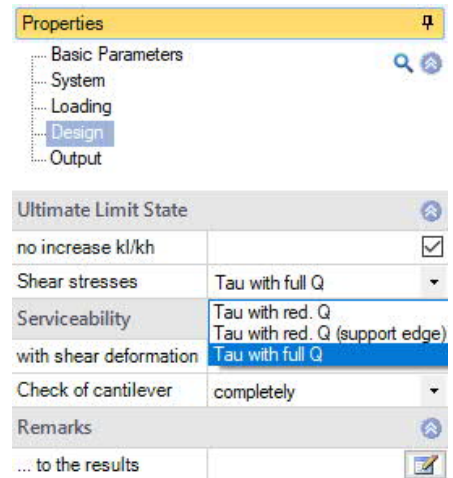
No increase kl/kh Option for deactivating the material-related height coefficient k_h and the length coefficient k_l

Approach of the shear force when calculating the shear stress

- Tau (τ) with red. S = reduced shear force
- Tau (τ) with red. S (anch. beaming)
- Tau (τ) with full S = full shear force

Serviceability

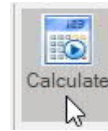
The shear deformation can optionally be taken into account.



Output

Output scope / calculation / results

Before output, click on the calculate icon if automatic calculation is switched off after each input. After the calculation has been carried out, the utilization is displayed at the bottom right of the graphic window and provides a good overview of the economic efficiency of the system entered.



Output scope

By clicking on the various output options you determine the scope of the output.

Visibility

The individual representations in the graphics can be switched on or off in the upper toolbar.

Load filter

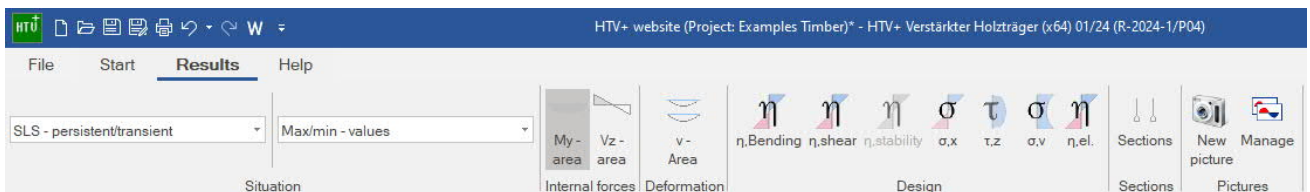
Using the "Load filter" button, loads can be filtered according to actions and groupings (alternative/combined). The selection is then highlighted visually/colorfully in the graphics. This means that loads can be checked clearly and edited straight away in the graphics. The filter can be canceled again using "Deactivate".

Note: If the selection is set, unselected loads (gray) can be added with just one click by holding down the CTRL key and clicking on the load of the current selection (colored).

Results

You can view the result graphics using the "Results" tab.

The defined [output sections](#) can be shown and hidden.

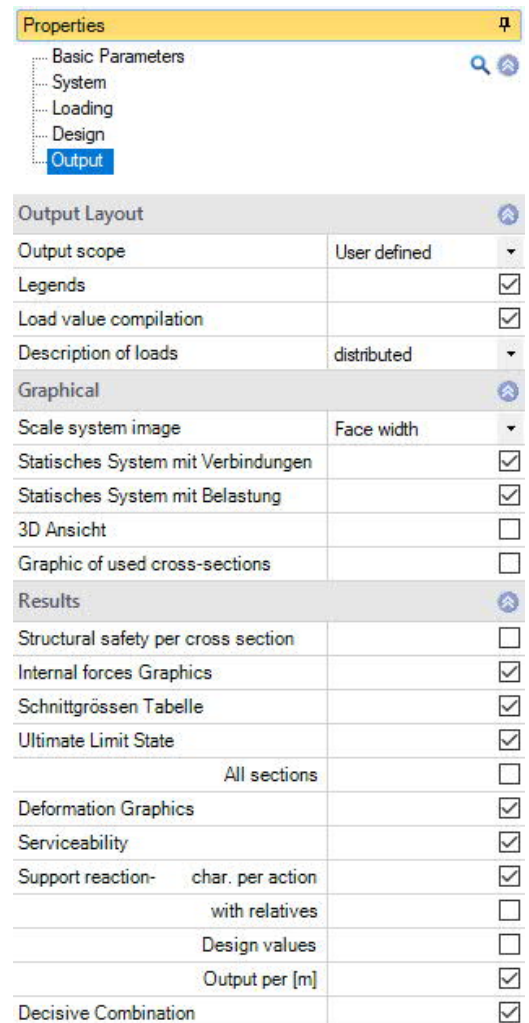


You can use the camera icon to take a snapshot of the displayed graphics and name it. Use the symbol on the right to display the list of recordings, which can also be deleted here. These images are automatically included in the output.

Output as a PDF document

Using the "[Document](#)" tab, the output document is displayed in PDF format and can be printed.

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



Load transfer

To transfer the support loads to the column programs [B5+](#), [STS+](#), [HO1+](#) and [TB-HHS](#), right-click on the respective support and select the corresponding program. See also connected programs under "[Application options](#)".

