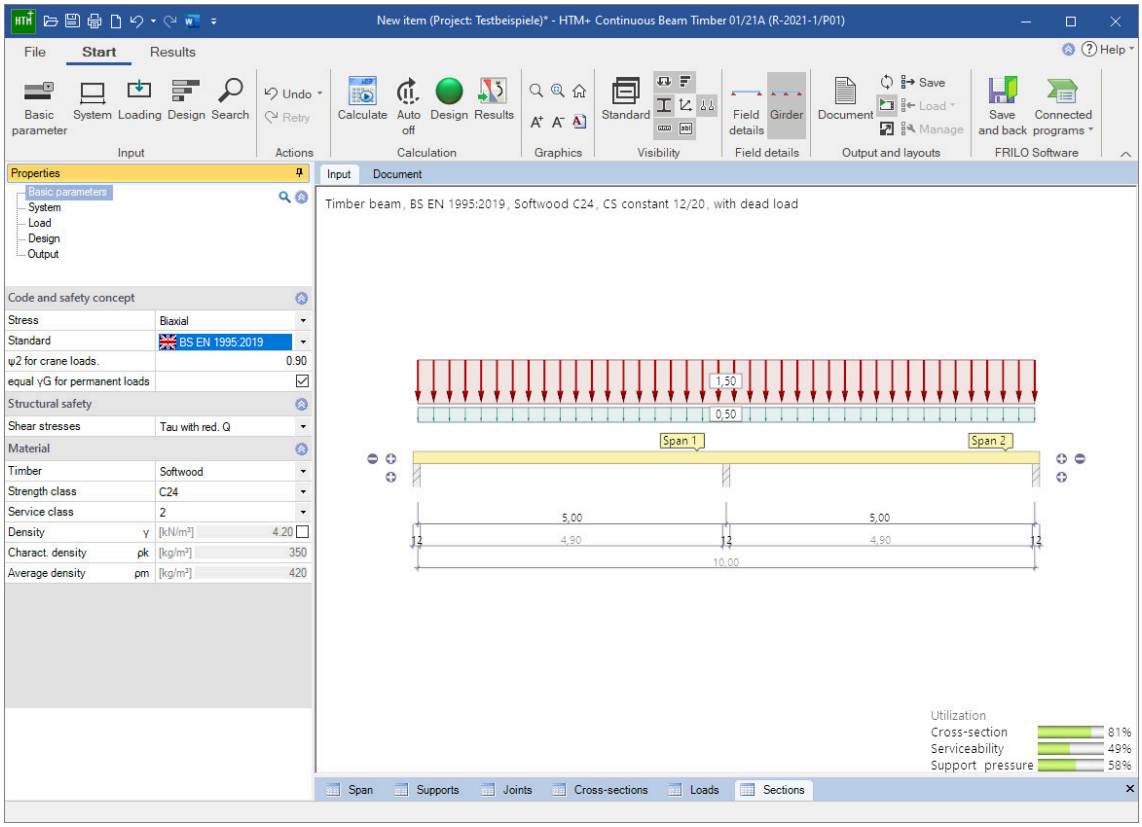
HTM+ Continuous Beam Timber

FRILO Software GmbH
www.frilo.com
info@frilo.com
As of 26/11/2020



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 ▸ Support ▸ Articles/Information ▸ Basic operating instructions.

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.

The superposition and the design are performed automatically.

The output is compact and can be configured in a detailed manner.

The program is designed for a graphically supported interactive workflow.

Available standards

- DIN EN 1995
- ÖNORM EN 1995
- EN 1995

Wizard

Simple basic system:
The data necessary to define a simple system can be entered with a wizard.

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-S 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.

Interfaces to other applications

- Three-dimensional frame RSX+
- Multi-span beam steel STM+

Add-on modules

- HTM-2 Biaxial
- HTM-S Stability

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.

Data entry via the wizard:

- Number of spans (or optionally, just a cantilever)
- Span length
- Cross section
- Permanent line load
- Variable line load and type of action

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 can be called up 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.

Interactive dimensional chains

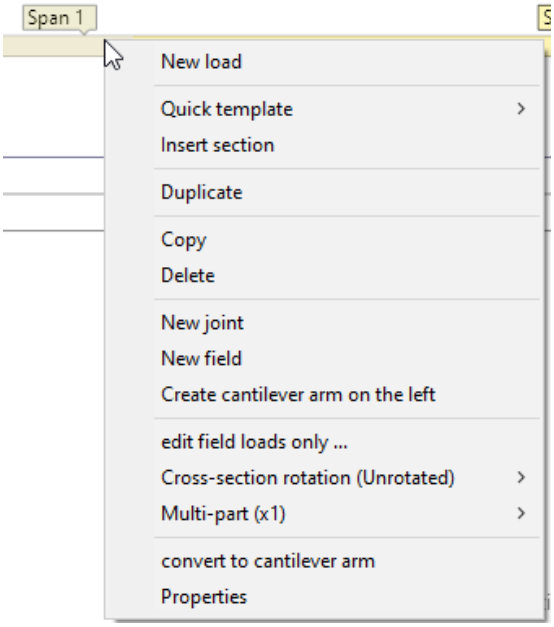
As in all Plus programs, the dimensional values are editable also in HTM+ and can be changed directly on the graphic screen.

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.



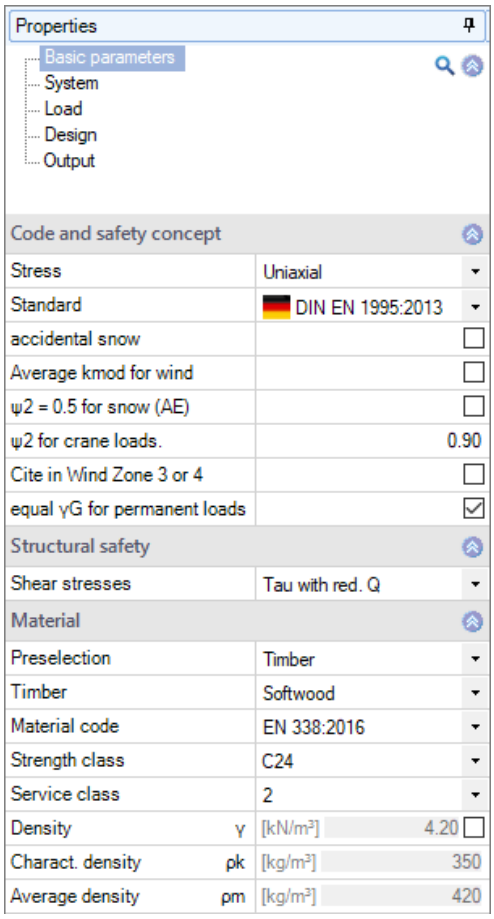
Interactive texts

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.

Basic parameters

Standard and safety concept

Effects of actions	uniaxial or biaxial (HTM-2 add-on).
Standard	definition of the design standard and its national annex.
ψ2 for crane loads	determines the combination coefficient ψ2 for crane loads (ratio of permanent share to total crane load)
Snow as accidental load	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.
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.



Structural safety

Shear stresses	option to include shear force in the calculation of the shear stress. See also the chapter Design.

Material

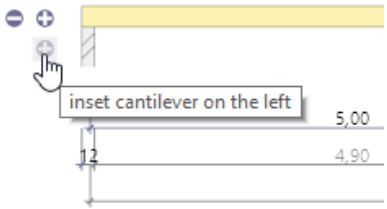
Input of the material values for softwood, hardwood or glulam.

Structural system

Spans

In the graphic

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



After right-clicking on the span in the graphic, functions as described below for the data-entry via tables are available, e. g. the values of this span can be entered via "Properties".

Moreover, you can also insert/delete joints and various quick templates are available.

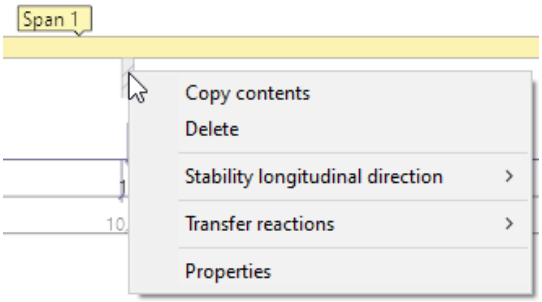
Data-entry via tables

To enter data via tables, click on the "Span" tab below the graphic screen. Use the buttons on the right of the table to add or delete table rows.

Span	select span or cantilever
Span length	
Cross-sections	allows you to define a new cross section or select an existing one.
Span loads	the table for entering the span loads is called up. See the chapter Loads for further information.
Sections	here you can divide a member into several sections. Enter the beginning and the end, the corresponding cross-sections, the rotation (unrotated, 90°, 180°, 270°). In the "multi-part" field, you can specify the number of profile sections next to each other. Two profile sections next to each other are treated in principle as if two members would lie next to each other and bear half of the load each.

Supports

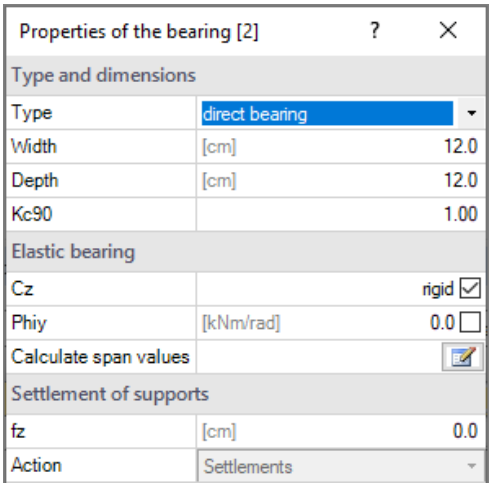
The bearing properties are called up by double-clicking on the support or by right-clicking and selecting the appropriate option in the context menu. In this section, it is also possible to delete supports or transfer the properties of the support to another support using the "Copy contents" function.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



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.

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’.



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

- 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.

Position at the cross section

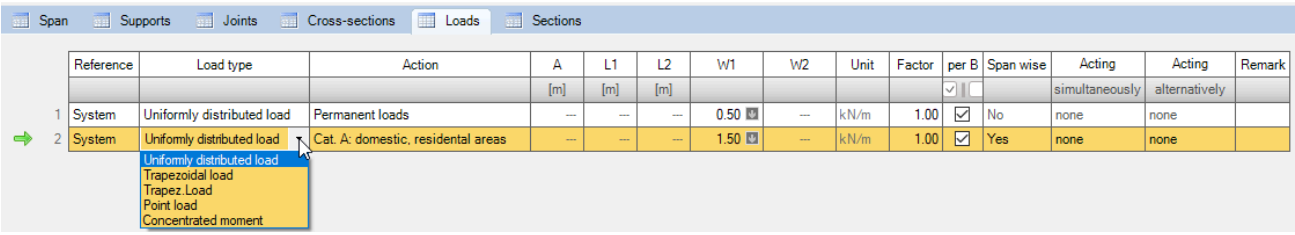
In addition to the shear centre, the lower and upper flange are available for selection here.

Cross-sections

You can define multiple cross-sections here.

Loads

Select first whether self-weight should be included in the calculation or not.



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 or area loads as well as concentrated moments.

Action — select an action from a list.

Full length — you can define whether uniformly distributed loads, trapezoidal loads or triangular loads shall apply over the entire beam or span length or only in a partial area. If "No" (partial areas) is selected, the data-entry fields for the distance (A) from the left beam or span end (depending on the selected reference), length 1 and length 2 are enabled. Check the loads entered in the graphic.

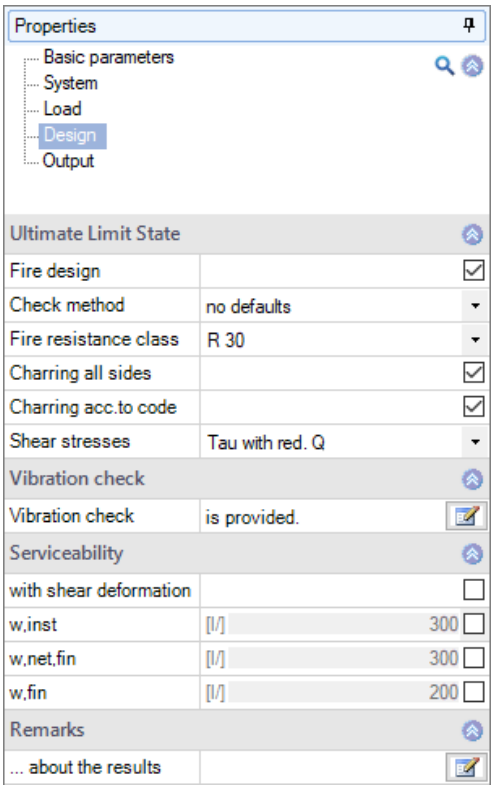
In general, the distance refers either to the left-hand edge of the structural system or to the left-hand end of the span depending on which of both was selected as a reference.

UDL	for loads applying over the full length only the load value W1 can be entered here. Ootherwise, the distance A of the front end of the load and the length L1 of the load can be defined additionally.
Trapezoidal load	for loads applying over the full length, the load values W1 and W2 can be entered as start value and end value, otherwise the distance A of the beginning of the load and the length of the load L1 can be defined additionally.
Triangular load	for loads applying over the full length, the maximum load value W1 and the length L1 up to the maximum value can be entered here, otherwise the distance of the beginning of the load A and the length L2 up to the end of the load can also be defined.

W1/W2	entry of the load values. By clicking on the arrow icon  you can access a load value compilation.
Factor	multiplication factor for the load ordinates
Spanwise	you can choose whether loads that are entered over several spans shall apply span by span or only in combination.
Concurrency	the loads of a concurrency group always apply simultaneously.
Alternative	only one of the loads of an alternative group is used. A load over several spans is considered to be one single load and is not alternatively applied span by span. Whether or not a span-by-span load approach is used depends on the entry in the ‘Alternative’ column.
Text	allows you to enter comments on the system. They are included in the output.

Design

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



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.

Shear stresses

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

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

Oscillation/Vibration analysis

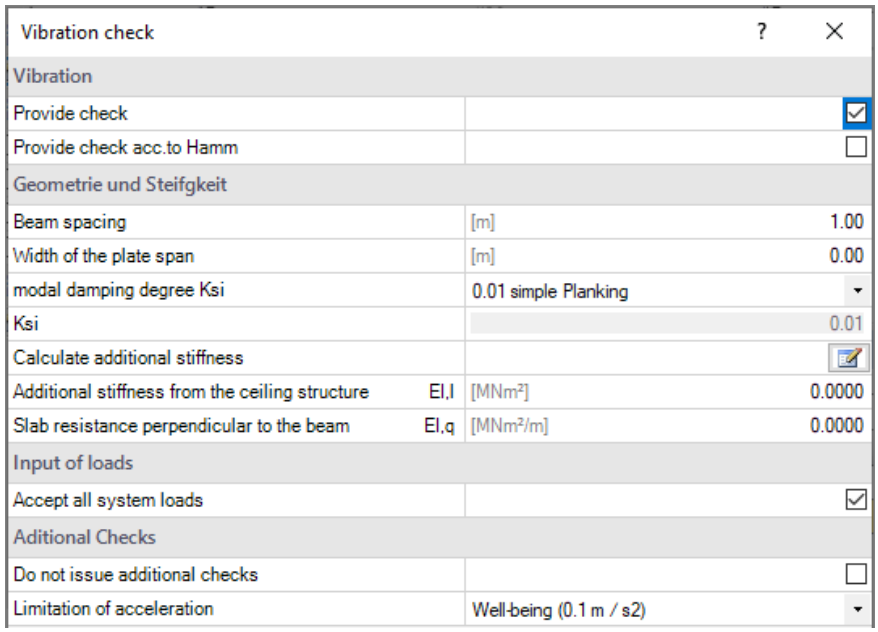
You can select the oscillation analysis on the "Design" tab.

 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
- kstrut — system factor

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.

Limitation of acceleration for EN 1995

Options: comfortable, tangible, user-defined.

Note:

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

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

Output

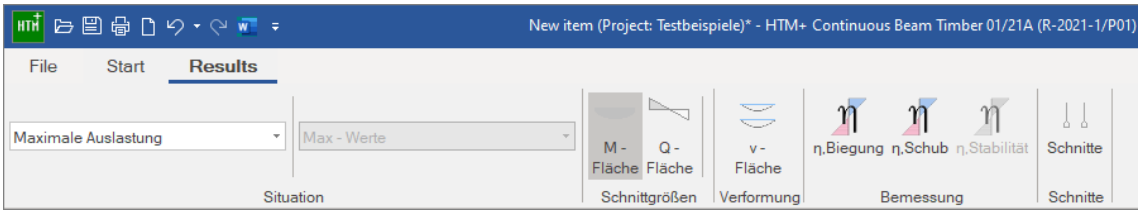
Before starting the output, click on the calculation icon if the option ‘automatic calculation after each input’ is switched off ("Auto off"/"Auto on" icon).

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.

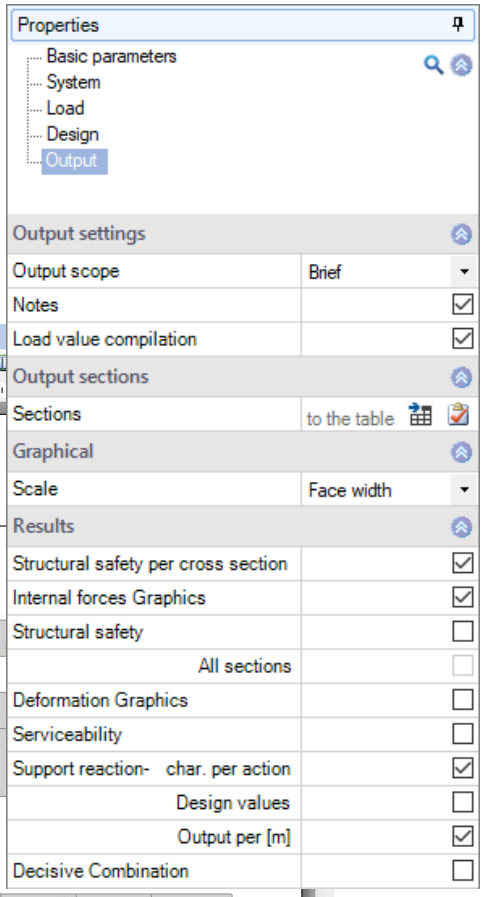
Result options



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

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.pdf