Continuous Beam Steel STM+

Table of 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.

Tip: Go back - e.g. after a link to another chapter / document - in the PDF with the key combination "ALT" + "left arrow key".

Application options

The STM+ program calculates single and multiple span steel beams.

A cantilever beam is selectable as a special case. Cross-sections can be defined with a haunch, with hinges or with a reinforcement, also multi-part cross-sections are possible. The user can optionally perform either an elastic or plastic design of the cross-section in accordance with EN 1993. Overlay and dimensioning are carried out automatically.

The output is compact and can be configured in small parts.

The program is designed for a graphically interactive way of working.

Norms

- DIN EN 1993
- ÖNORM EN 1993
- BS EN 1993
- PN EN 1993
- EN 1993

Assistant

The entries required for a simple system can be made with an “Assistant”.
This basic system can then be modified and supplemented very easily with the graphical-interactive input.

Supports/holders

You can enter supports in the Z direction (and in the case of two-axis loading in the Y direction) as well as for the rotation around the y axis. There is always the option of rigid mounting or entering a spring value. A support depression can be specified for the individual supports. Alternatively, the spring values can also be calculated by the program from a column that can be defined below and/or above the beam and then adopted for the beam calculation.
For the proof of stability, the brackets can be defined on the cross-section. A distinction is made between the position of the retainer in the longitudinal direction of the carrier and the position on the cross section.

Loads

Load types: constant, trapezoidal, triangular, single load and single moment.

Interfaces to further programs

- The 2nd Theory of Torsion-Bending BTII (entire structural system)
- Spatial Framework RSX+ (entire structural system)
- Continuous Beam Timber HTM+ or Concrete BTM+ (entire structural system)
- Forwarding of the support reactions to the programs Reinforced Concrete Column B5+, Single-Span Steel Column STS+ and Timber Column HO1+.

Additional options

STM-2 2-axis
With the help of this add-on, the user can define biaxial loading and perform the corresponding design. Each load can apply at an angle between 0° and 90°. The loads are automatically separated into a horizontal and a vertical load share. The user can define elastic supports, restraints in upright members and end restraints as well as column settlement in each direction separately. Joints function as bending joints about the y-axis as well as about the z-axis.

Quick start and graphical input

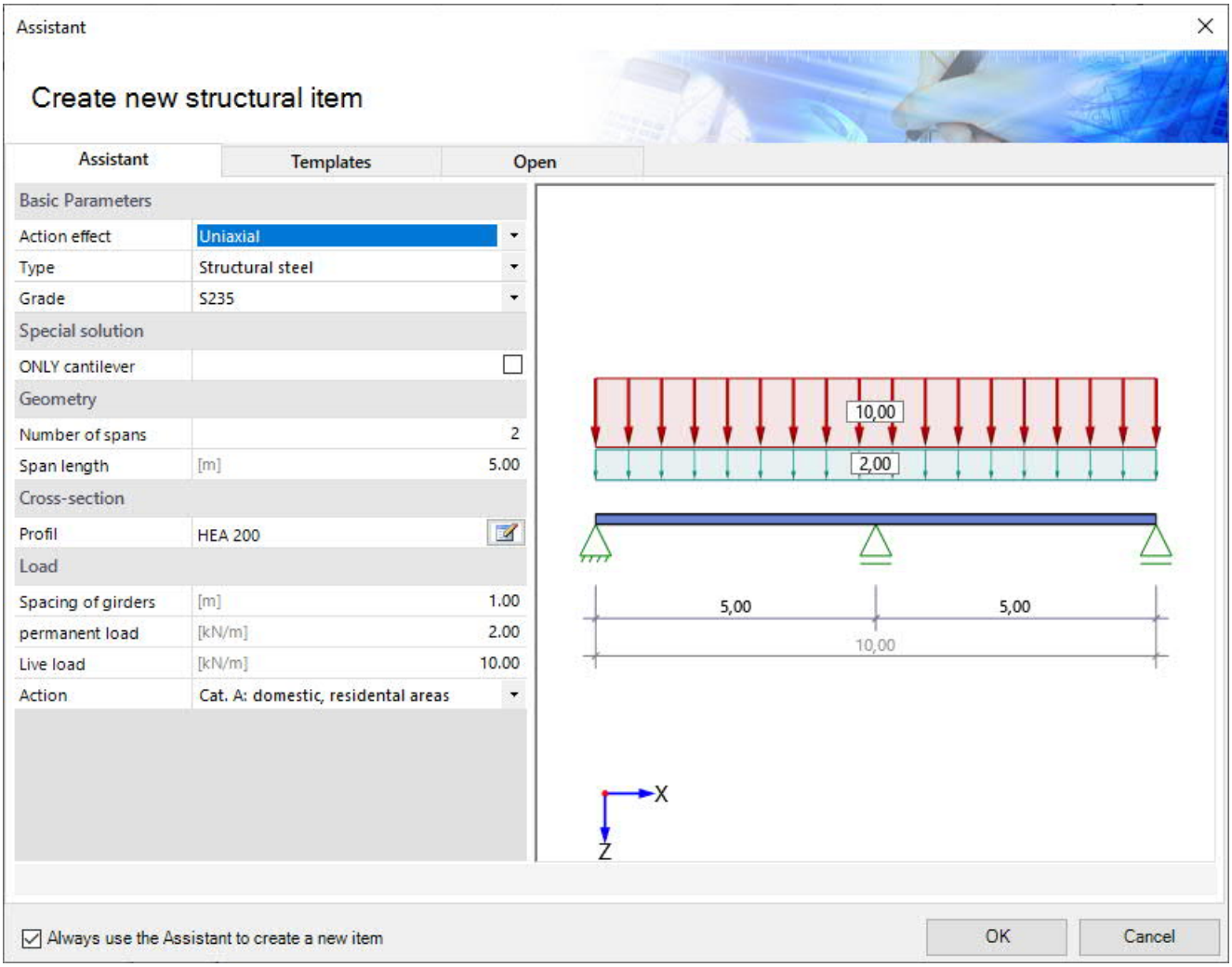
The Assistant

The Assistant is displayed by default when a new item is created - it can also be switched off if necessary (option in the lower window area).

In the Assistant, the entries required for a simple system can be made in an interface:

- Action effect unaxial or - with aditional option STM-2 – biaxial
- Steel type and grade
- Number of spans (or optionally only cantilever)
- Span length
- Cross-section
- Permanent line load
- Variable line load and type of action

This basic system can then be modified and supplemented very easily with the graphical-interactive input.



Graphic input

The graphic 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 made through the context menus of the individual objects (field, support, load...) or through the interactive texts at the top left. Fields and cantilevers can be added using the symbols on the right and left.

It is also possible to 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 operating principles.

Visibility

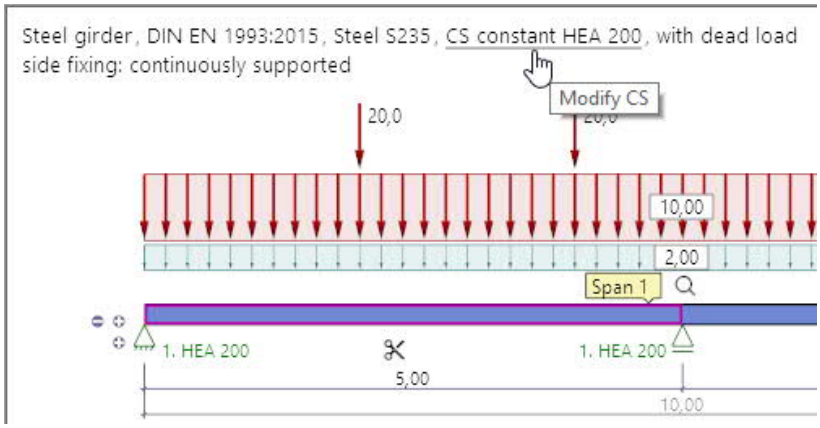
Individual objects can be shown and hidden in the graphics window as required. The individual switches can be found in the menu ribbon under "Visibilities". You can show or hide the load, cross section, dimensional chains, load, coordinate system, interactive text links and sections.

After the calculation, the workload is displayed in the lower right corner of the graphic window and offers a good overview of the economic efficiency of the system entered.

Interactive text links

The texts displayed at the top left are interactive and can be clicked on. These "text links" are used to call up dialogues that have no graphic representation.

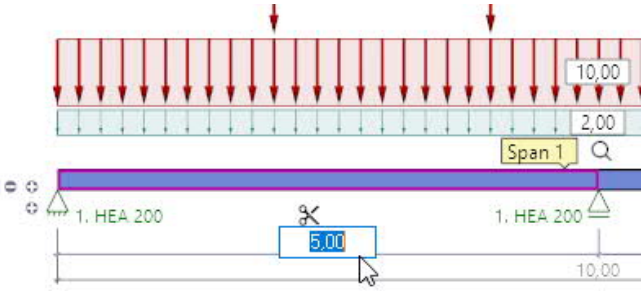
Clicking on the text section with the standard starts the basic settings dialog, clicking on the material leads to the material properties. If you want to modify the cross-section, click on the cross-section to open the cross-section selection. By clicking on “with dead weight”, the consideration of the dead weight can be controlled.



Interactive dimensional chains

As in all Plus Programs, the dimensions can also be edited in STM+ and can be changed directly in the graphic.

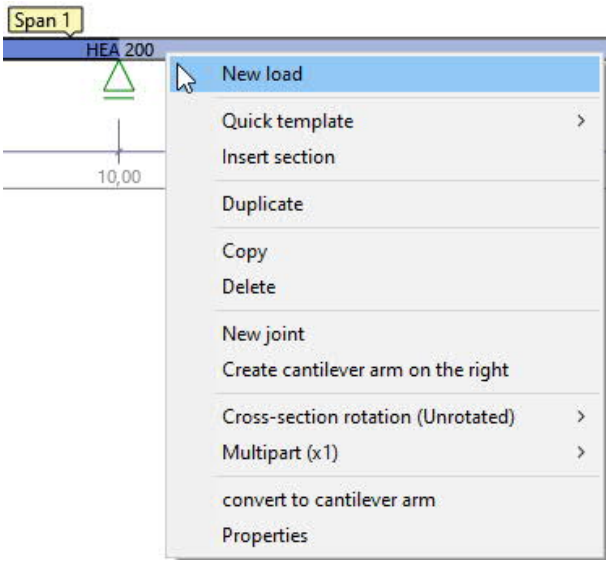
Tip: the span lengths can also be changed by moving a support. To do this, click the support with the left mouse button, hold the mouse button and move the support.



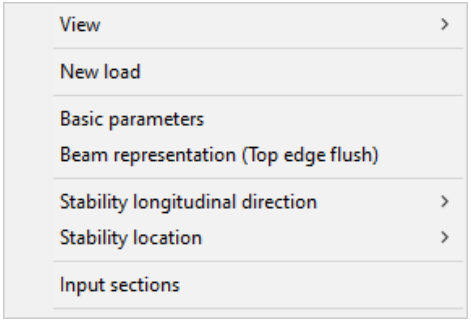
Context menu

There are appropriate context functions for each object (field, support, load, etc.). These functions are displayed with the right mouse button and, as the name suggests, they match the selected object.

A general context menu appears when no object is selected. Here you can find also functions that do not represent a graphic object (View, New load ...).



Context menu „Span“

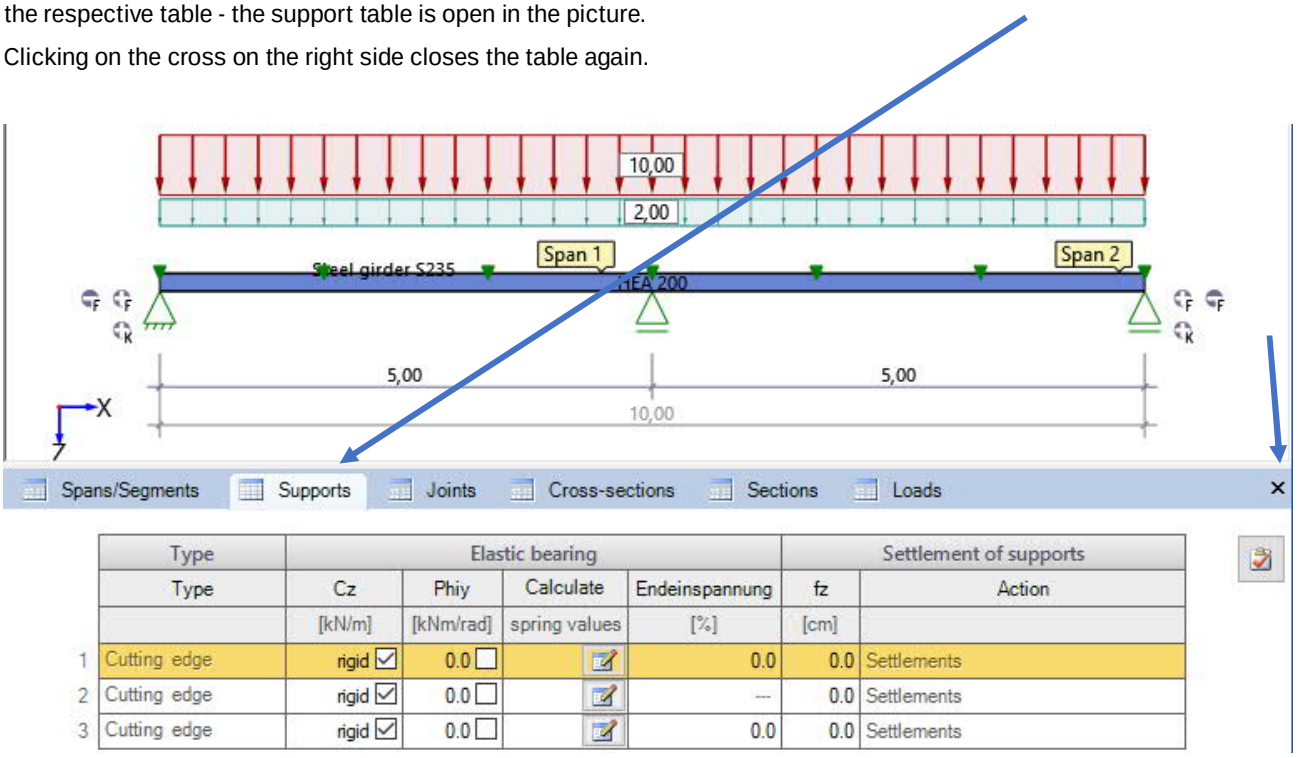


Context menu „General“

Tables

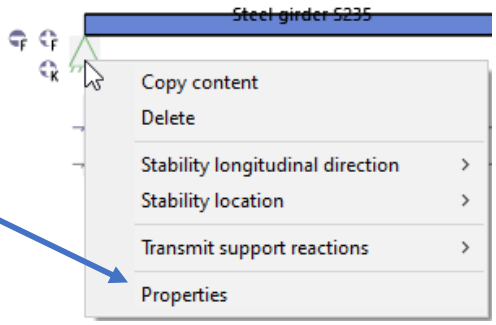
Several tabs are visible on the left in the lower area of the graphics window. A click on one of these tabs opens the respective table - the support table is open in the picture.

Clicking on the cross on the right side closes the table again.



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

All input values of the table can also be found in the context menu of the respective object under “Properties“.



Please also read the table entry in the operating principles.

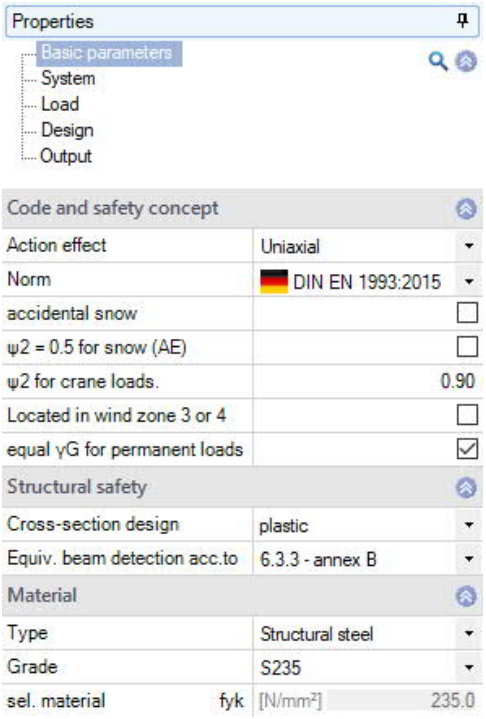
„Properties“ menu

As an alternative to the pure interactive input in the graphics window, all input parameters can be reached in the left menu.

This properties menu bar can be expanded and collapsed as required - see operating instructions.
Tip: individual parameters can be found quickly using the search magnifier.

Basic parameters

Code and safety ceoncept



Action effect	Uni- or biaxial
Norm/Standard	Selection of the standard. See also application options.
Aciddental snow	Here you define whether, in addition to the usual design situations, the snow loads should also be automatically applied as an accidental action. The load factor for the accidental snow loads can be freely specified or automatically determined by the program.
ψ2=0,5 for snow	Specifies whether the combination coefficient ψ2 for the action of snow should be increased to a value of 0.5 in the earthquake (AE) design situation. See introductory decrees of the federal states, e.g. Baden-Württemberg.
ψ2 for crane loads	Combination coefficient ψ2 for crane loads (= ratio of permanent share to total crane load).
Located in windzone	Indicates whether the building is located in wind zone 3 or 4. In this case, the “snow” action does not need to be included as an accompanying action to the “wind” lead action.
equal γG ...	If the option is selected, all permanent loads or load cases are applied together with the same partial safety factor (γG,sup oder γG,inf). Otherwise all permanent loads or load cases are combined with γG,sup and γG,inf.

Structural safety

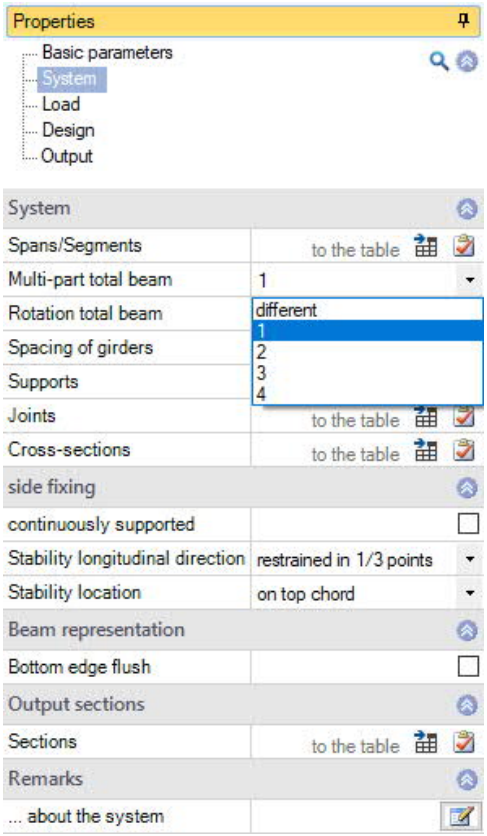
Cross-sections design	This takes place elastically according to equation 6.1 or plastically according to equation 6.2.
Equiv. beam detection acc. to	This is done according to 6.3.3 (annex A or B) or 6.3.4

Material

Type	Choice of standardized steel type or user-defined input.
Grade	Choice of standardized steel quality or user-defined input (characteristic values).
sel. material	Display for information.

System

Spans/Segments, supports, joints and cross-sections are entered via the tables (tabs under the graphics window) - see also tables.

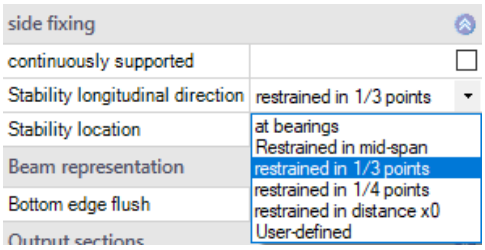


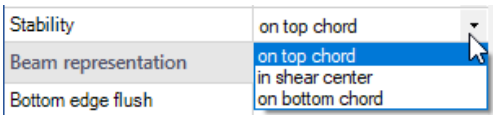
Multi-part total beam	Define multiple parts for the entire beam. In the Spans/Segment-table you can define multiple parts for each segment.
Rotation total beam	Here you can define the rotation (90°/180°/270°) of the entire beam. In the Spans/Segment-table you can define the rotation of each segment.
Spacing of girders	In the load table, the beam spacing for each load can be taken into account individually by ticking the 'per beam' column.

Side fixing

Continuously supported:

Deactivate this option to show the selection options for the position of the side fixing (lateral bracket) in the longitudinal direction or the location (on top chord/shear center/bottom chord) - see the following figure as well as chapter “Side fixing”.

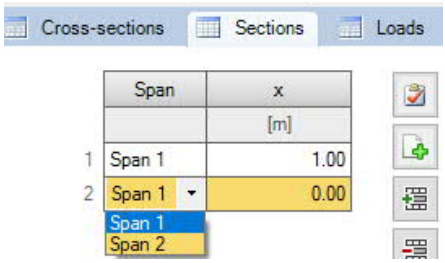




Output sections

You can define output sections in a table (sections).

See also Output and results.



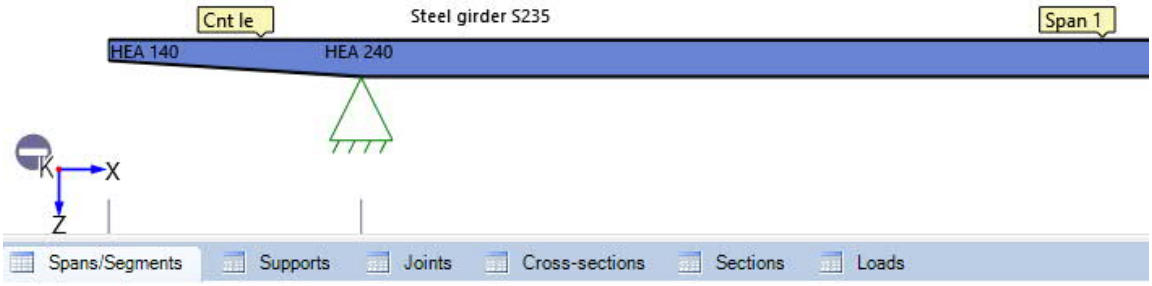
Remarks

Input of comment text on the system via the comment editor.

Spans / segments / haunches

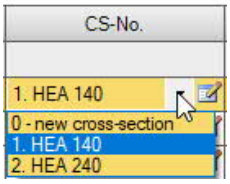
Entry of spans/segments in a table

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.	Rotation	Multipart	Haunch
1	Cnt le	1.50	1	1.50	1. HEA 140	Unrotated	1	☑
2		---		---	2. HEA 240	---	1	☑
3	Span 1	5.00	1	5.00	2. HEA 240	Unrotated	1	☐
4	Span 2	5.00	1	5.00	2. HEA 240	Unrotated	1	☐

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 (for example “1. HEA 140”).



Rotation	You can define the rotation for each segment (under “System” you can define the rotation for the entire beam).
Multipart	You can define multiple parts for each segment (under “System” you can define multiple parts for the entire beam).
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 right click on a span on the graphic screen to access editing functions such as Split segment.

Insert field / cantilever, split bars

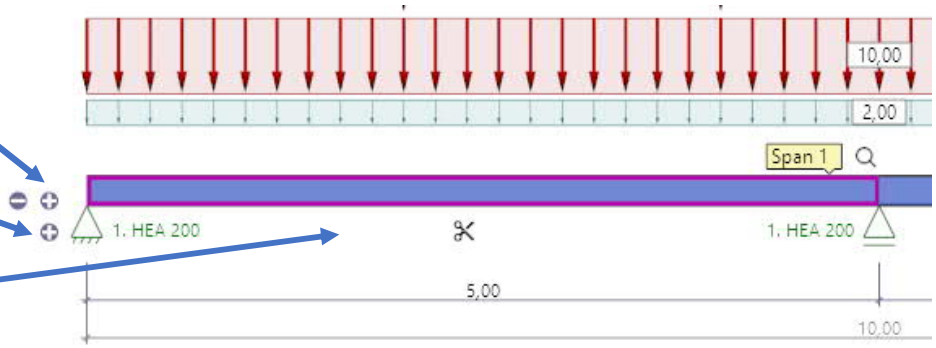
You can also insert new fields, cantilevers or split bars into bar sections directly via the graphic. The symbols on the carrier are intended for this.

Insert field

Delete field

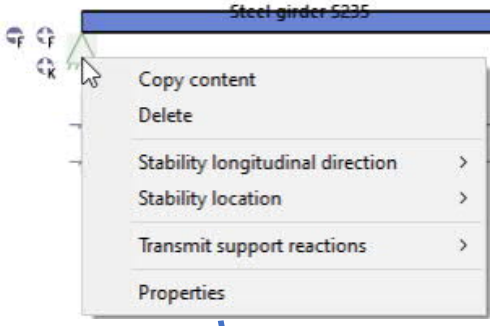
Insert cantilever

Divide the rod into sections (The scissors symbol appears after clicking/marking the field)



Support

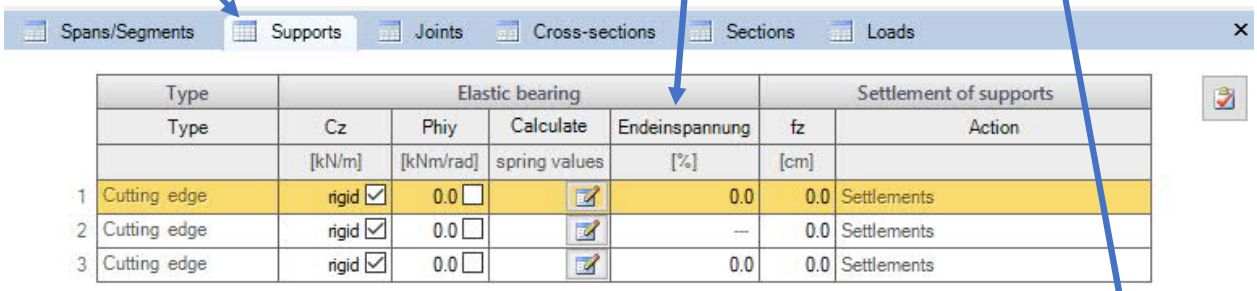
The support properties are called up by double-clicking on the support or by right-clicking and making the appropriate selection in the context menu. Here it is also possible to delete supports or to transfer the properties of the support to another support using the “Copy contents” function. In addition, the side fixing for the stability verification can be defined here.



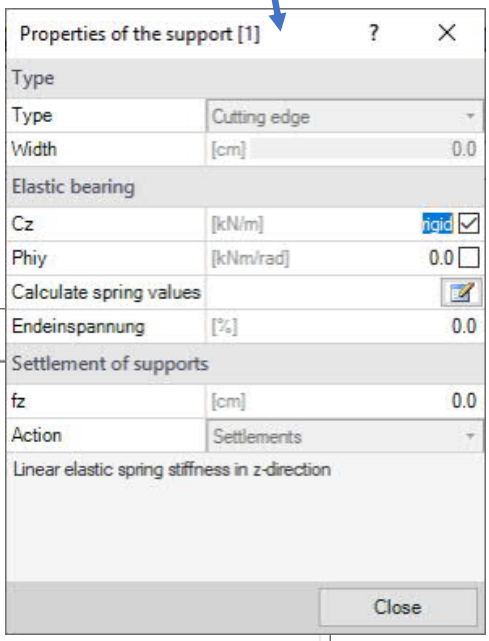
You can enter supports in the Z direction (and in the case of biaxial loading in the Y direction) as well as for the rotation around the y axis. There is always the option of rigid mounting or entering a spring value.

A lowering/settlement of the supports can also be specified for the individual supports.

The "Supports" tab below the graphic can be used to enter percentage restraints on end supports.

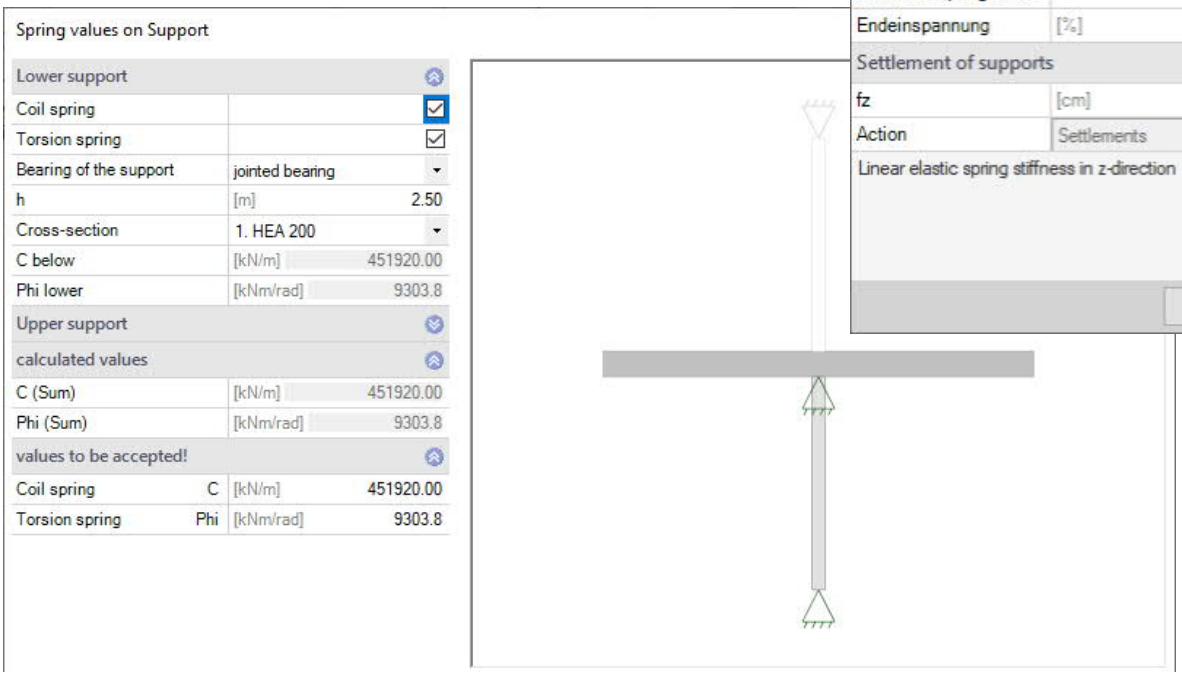


	Type	Elastic bearing				Settlement of supports	
	Type	Cz	Phiy	Calculate	Endeinspannung	fz	Action
		[kN/m]	[kNm/rad]	spring values	[%]	[cm]	
1	Cutting edge	rigid ☑	0.0 ☐		0.0	0.0	Settlements
2	Cutting edge	rigid ☑	0.0 ☐		---	0.0	Settlements
3	Cutting edge	rigid ☑	0.0 ☐		0.0	0.0	Settlements



Calculate spring values

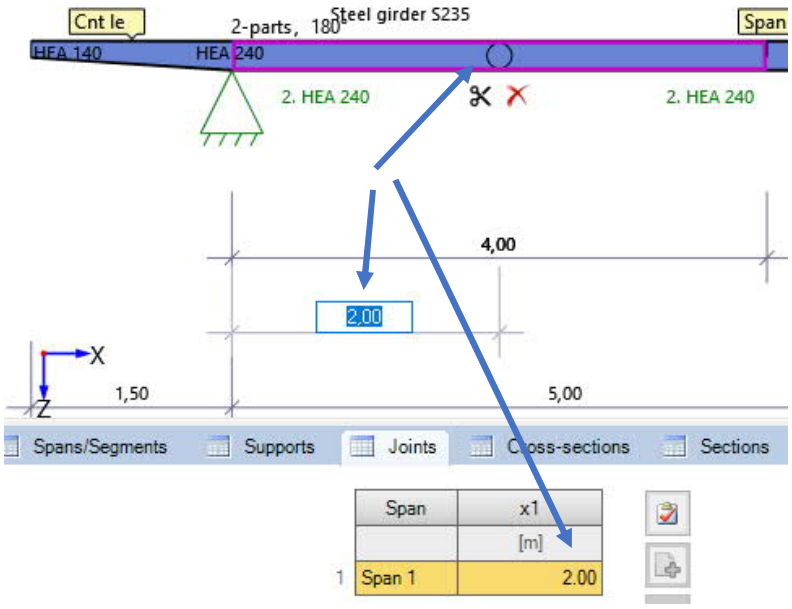
Alternatively, the spring values can also be calculated by the program from a column that can be defined below and/or above the beam and then adopted for the beam calculation.



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.



Cross-sections

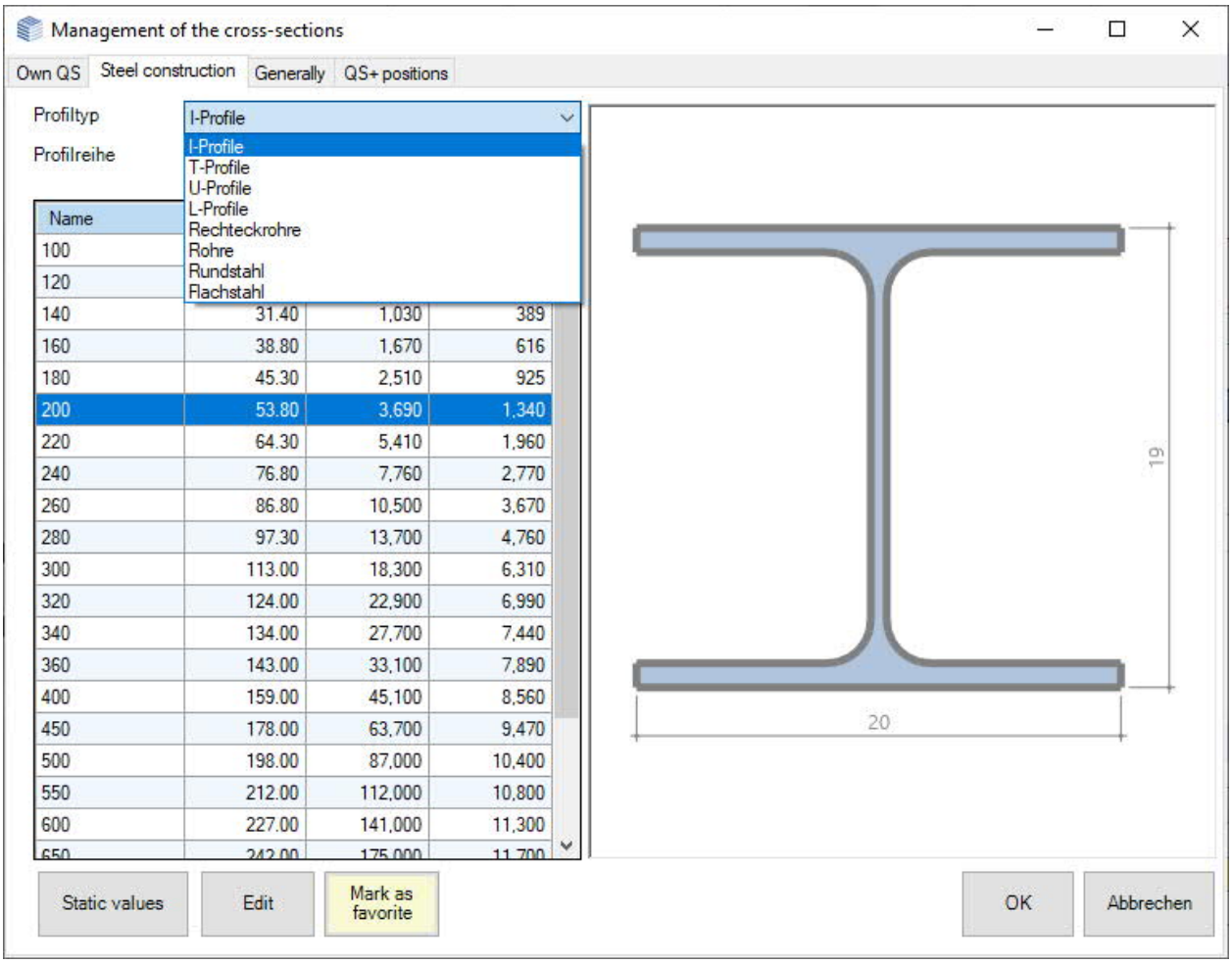
An extensive selection of profiles is available in the program.

Each profile can be used as the basis for processing. The “Edit” function opens a dialog for adapting the geometric values of a profile. User-defined profiles can also be created. If this self-defined profile is also to be used outside for other programs or systems, the profile can be saved as “Own Profile” (you will then find the selection of your own cross-sections under the "Own CS" tab“.

For quick access to preferred profiles, the favorites are stored. Each profile can be added to the list of favorites using the context menu. This list is saved in the personal settings on this workstation.

All static values of a profile can be displayed.

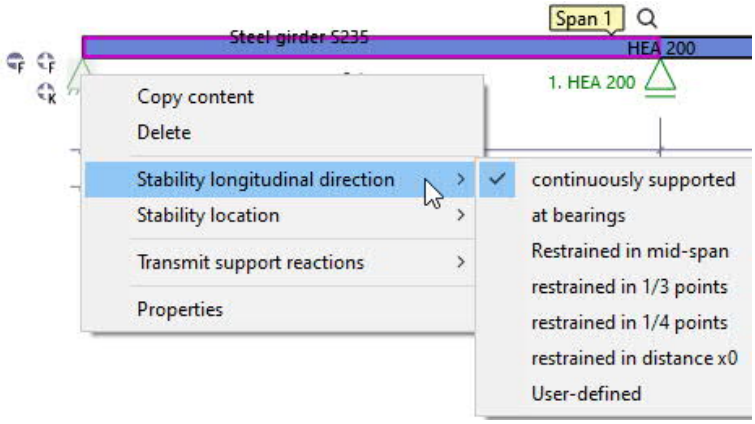
See also document Cross-Section Selection-PLUS.



QS+ positions: This tab can be used to import user-defined cross-sections from the General QS+ steel cross-sections program.

Side fixing / Stability

You can also use the context menu to define the holders on the cross-section for the stability analysis. A distinction is made between the position of the fixation in the longitudinal direction of the girder (Stability longitudinal direction) and the position on the cross section (Stability location).



See also System ▸ side fixing.

Stability in the longitudinal direction of the girder

A distinction can be made here between continuous mounting, mounting only on the supports, additionally in the center of the field or in the third or quarter points of the fields.

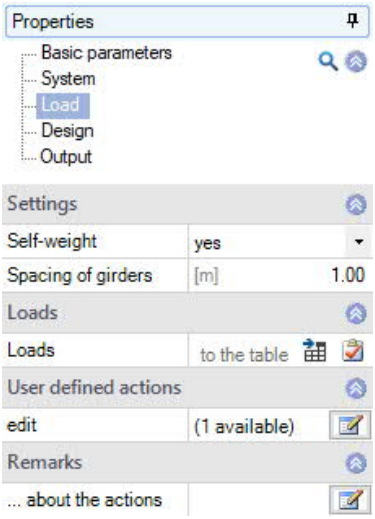
In addition, an even distance can be specified using the “At distance X0” option. With “User-defined” the holders can be completely freely defined.

Stability location / position on the cross-section

In addition to the shear center, the lower and upper chords can be selected.

Loads

The load parameters are entered via the load table (tab below the graphics window) - see also tables.



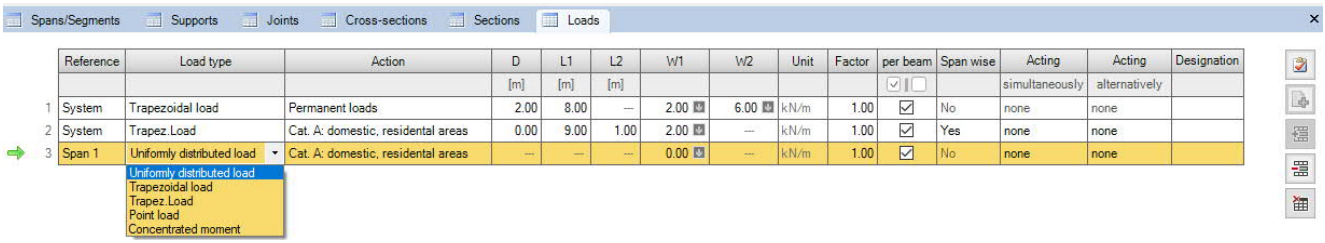
Self-weight	Selection of whether to calculate with or without self-weight.
Spacing of girders	see System.

User defined actions

In addition to the selectable actions in the load table, user-defined actions can also be defined and named here, which are then available for selection in the load table.

See actions user defined.

Load-table



Reference	Choose whether the load entry relates to the entire system or just to a single field. In general, when referring to the system, the distance (column A) refers to the left edge of the system and when referring to the field to the beginning of the field on the left.
Load type	Selection: uniform load, trapezoidal load, triangular load, single load, single moment.
Action	Selection of the type of action from a list. See also User defined actions.

Columns A, L1 / L2 and W1 / W2 are used depending on the type of load.

To check your entries, check the load representation in the graphic and pay attention to the selected reference point (see "Reference" above).

D	Distance between the start of the load and the reference point.
L1 / L2	Length of the load (L1) or, in the case of a triangular load, the left (L1) and right (L2) load section.
W1 / W2	Load value W1 or with trapezoidal load start (W1) and end value (W2). You can call up a load value compilation using the arrow symbols.
Unit	Display of the unit. Note: the units can be changed via File - Settings.
Factor	Optional input of a load factor.
per beam	If this option is checked (default), the beam spacing is taken into account individually for each load (automatic conversion to area loads).
Span wise	Here it is defined whether loads that are entered over several spans are to be applied span wise by the program or are only taken into account in combination.

Acting simultaneously	Here you can define (several) groups (Sim 1, Sim 2, etc.) and assign them to the loads. Loads of a group are always applied together.
Acting alternatively	Here you can define (several) groups (Alt 1, Alt 2, etc.) and assign them to the loads. Only one of the loads of an alternative group is ever applied. A load over several spans is considered to be one load and is not used as an alternative span wise.
Designation	Optional input of a designation text.

Design

Structural safety — as described under basic parameters.

Serviceability

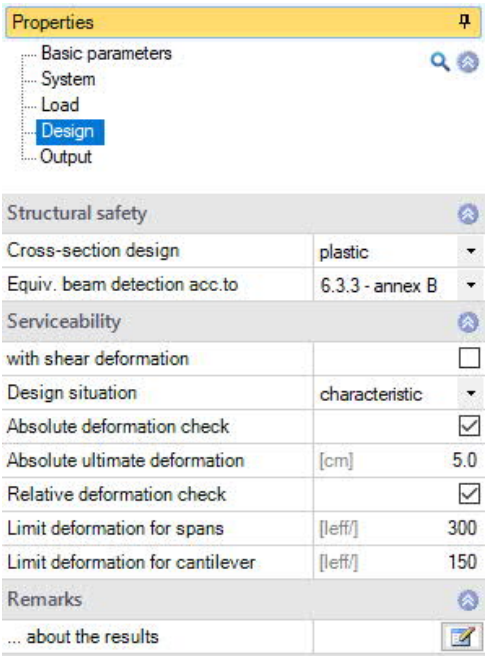
With shear deformation — When calculating the deformations, the shear deformation is taken into account.

Design situation — Design situation on which the verifications in the serviceability limit state are to be based: characteristic, frequent, quasi-permanent.

Absolute deformation check — Provides proof of serviceability for use with the deformation difference to the undeformed system.

Absolute ultimate deformation — Shows the maximum permitted absolute deformation of the system.

Relative deformation check — Performs the proof of serviceability based on lengths that are determined by the support points.



Remarks

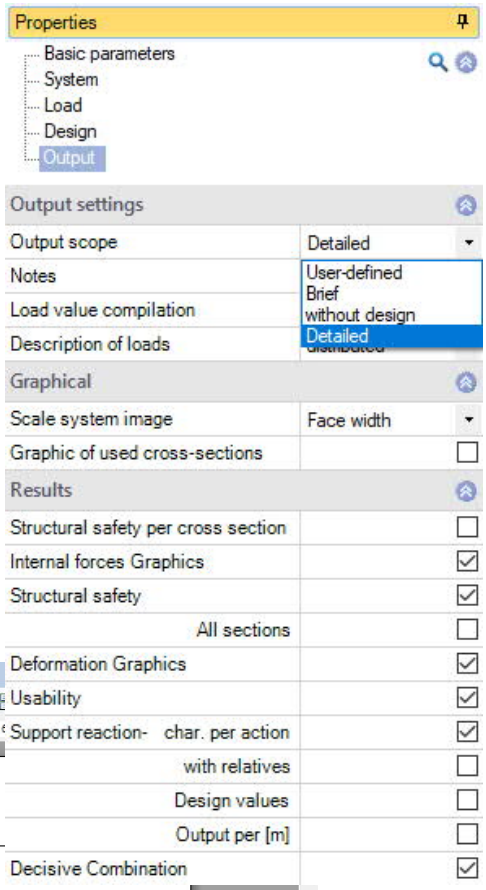
Calling up the remarks editor.

Output and results

Use the "Document" tab to switch to the display of the output.

The output scope (output profile) can be selected.

Output sections can be defined under System.



Calculation

You start the calculation via the "Calculate" symbol in the top menu bar. Result graphics can also be displayed here and the evidence can be displayed.

See also: Output and printing